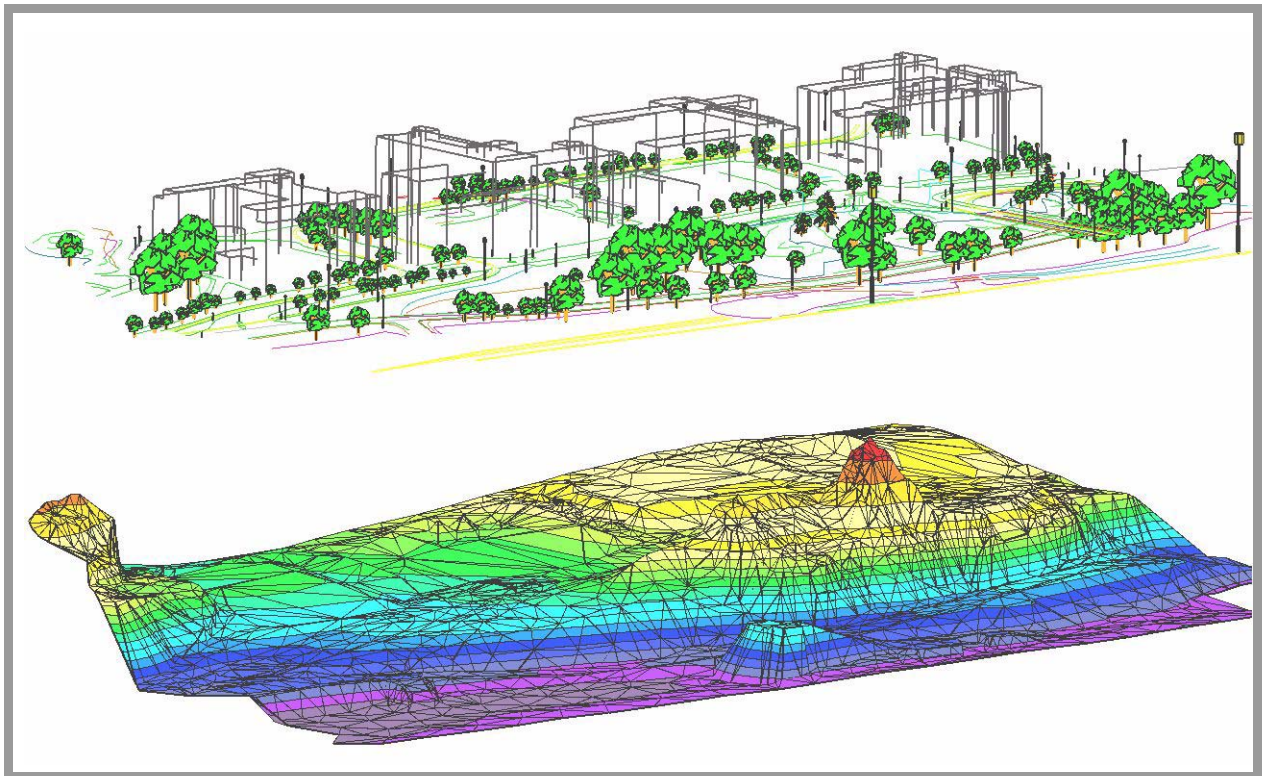


# *TerraModeler User's Guide*

---

--- for the 32-bit version of TerraModeler ---



## Trademarks

MicroStation®, MDL® and MicroStation stylized "M" are registered trademarks of Bentley Systems, Incorporated. Bentley Map PowerView, MicroStation PowerDraft, and MicroStation GeoOutlook are trademarks of Bentley Systems, Incorporated.

TerraBore, TerraGas, TerraLink, TerraMatch, TerraModeler, TerraPark, TerraPhoto, TerraPipe, TerraScan, TerraSlave, TerraStereo, TerraStreet, and TerraSurvey are trademarks of Terrasolid Limited.

Windows is a trademark of Microsoft Corporation.

Acrobat Reader is a trademark of Adobe Systems Incorporated.

Intergraph Raster File Formats - Copyright - 1994 Intergraph Corporation. Used with permission.

## Copyright

© 1995-2016 Arttu Soininen, Terrasolid. All rights reserved.

## Acknowledgements

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

# Table of Contents

## Getting Started

<b>1 Documentation</b>	9
About the documentation	9
Accessing the documentation online	9
Document conventions	9
MicroStation documentation	9
<b>2 Introduction to TerraModeler</b>	10
Introduction	10
Terra application family	11
<b>3 Installation</b>	12
Hardware and software requirements	12
Installation media	12
Installation from zip file	13
Installation from CD/USB-Stick	13
<b>4 Starting TerraModeler</b>	15
Start TerraModeler	15
Unload TerraModeler	16

## Tutorial

<b>5 Tutorial</b>	18
TerraModeler interface	18
Opening an example design file	18
Starting TerraModeler	18
Creating a surface model	19
Viewing surface statistics	20
Viewing triangulation	20
Placing elements on surface elevation	21
Calculating slopes	21
Creating a design surface	23
Computing excavation quantities	24
Displaying a colored grid	26
Drawing a profile	27

## Tool Reference

<b>6 General Tools</b>	29
General tool box	29
Settings	30
Define Coordinate Setup	45
Set Scale	47
Surfaces	47
Save Surfaces	48
About TerraModeler	48
Help On TerraModeler	48

<b>7 Creating Surface Models</b>	49
Create Surfaces tool box	49
Triangulate Survey	50
Triangulate View	52
Triangulate Elements	53
Insert Random Elements	56
Insert Breakline Element	57
Triangulate Multiple Sources	58
Rebuild Models	60
<b>8 Drawing Elements</b>	61
Draw using Surface tool box	61
View Elevation	62
Compare Elevations	63
View Slope	63
Drop Element On Surface	64
Calculate Slope	66
Fill Area With Cells	67
Drawing Utilities tool box	69
Measure Slope	70
Label Elevation	71
Label Area	71
Place Sloped Line	72
Force Downstream Flow	72
Set Linear Elevation	73
Thin Linear Element	74
Densify Linear Element	74
Connect Linear Elements	75
Copy Linear Element	76
Alignment Offset	78
Validate Linear Elements	79
Fix Touching Elements	82
Read Piles From File	83
<b>9 Editing Surface Models</b>	85
Edit Point tool box	85
Construct Breakline	86
Insert Point Along Breakline	87
Insert Point	88
Move Point	89
Remove Point	90
Exclude Triangle	91
Edit Area tool box	93
Flatten Area	94
Move Area	95
Drop Area	96
Exclude Area	97
Insert Hole Element	98
Remove Area	99
<b>10 Generating Displays</b>	100
Display Surface tool box	100
Display Contours	103
Display Triangles	110



Display Grid .....	112
Display Elevation Texts.....	113
Display Slopes .....	116
Display Shaded Surface .....	118
Update Displays.....	120
Erase Display .....	120
Display Boundary .....	121
Label Peaks and Pits .....	122
Display Regions tool box .....	123
Display Drainage .....	124
Display Themes .....	126
Display Region Triangles .....	128
Display Raster Triangles .....	129
<b>11 Drawing Profiles.....</b>	<b>130</b>
Profiles tool box .....	130
Draw Section View .....	131
Draw Profile .....	133
Draw Alignment Sections.....	137
Update Profile .....	140
Setup Automatic Sections.....	141
Project Into Profile.....	142
Project From Profile .....	143
Label Profile Elevations .....	144
Output Profile Elevations .....	146
Output Section Elevations .....	148
<b>12 Computing Quantities.....</b>	<b>151</b>
Quantity tool box .....	151
Compute Quantity.....	152
Compute Prismoidal Quantity .....	156
Compute Alignment Quantity .....	157
Define Section Templates.....	160
Compute Section Quantity.....	162
Compute Tunnel Volume .....	166
Draw Surface Intersection .....	168
Compute Area.....	169
<b>13 Using Domains.....</b>	<b>170</b>
Domains tool box .....	170
Define Domains .....	171
Domain View .....	173
Set Area Domain .....	173
Set Domain Between Lines .....	174
Set Triangle Domain.....	175
Display Domain Triangles.....	176
<b>14 Placing Single Display Elements .....</b>	<b>177</b>
Display Single tool box.....	177
Place Contour .....	178
Place Elevation Text.....	179
Place Slope Arrow .....	180
Modify Contour Symbolology.....	181

<b>15 Using Regions</b>	182
Regions tool box	182
Regions	183
Create Region	186
Delete Region	188
Edit Region Info	189
Move Region Center	190
Update Regions	190
<b>16 Managing Surfaces</b>	191
File pulldown menu	192
File / New surface	193
File / Open surface	193
File / Close surface	194
File / Save surface	195
File / Open reference	196
File / Import	196
File / Export	196
File / Restore from file	197
File / User settings	197
File / Exit	197
Edit pulldown menu	198
Edit / Modify settings	199
Edit / Profile settings	199
Edit / Exclude long triangles	200
Edit / Modify elevations	201
Edit / Thin	202
Edit / Compress	203
Edit / Add inferred points	204
Edit / Delete surface	205
Utility pulldown menu	206
Utility / View statistics	207
Utility / Copy surface	208
Utility / Merge surfaces	209
Utility / Subtract surfaces	211
Utility / Statistical model	212
Utility / Output control report	214
Utility / Produce contours	216
Utility / Produce lattice models	219
Utility / Produce triangles	223
Utility / Lattice database	226
Tools pulldown menu	227
View pulldown menu	228
View / Fields	229
View / Sort by	229
Help pulldown menu	230
Help / Contents	230
<b>17 Importing and Exporting Data</b>	231
Import / XYZ text file	232
Import / XYZ binary file	234
Import / Lattice file	235
Import / Triangles	236

Export / Xyz text file .....	237
Export / Triangle text file .....	238
Export / Lattice file.....	239
Export / Raster image .....	242
Export / Graphical elements .....	245
<b>18 Lattice Database .....</b>	<b>246</b>
Lattice database / Draw boundaries .....	247
Viewing lattice database elevation .....	247
Lattice database / Draw contours.....	248
Lattice database / Import fence contents .....	249
<b>19 Flood Mapping .....</b>	<b>250</b>
Flood Mapping tool box.....	250
Import River Sections .....	251
Edit River Sections .....	252
Export River Sections .....	259
Input Water Elevations .....	260
Draw River Section Boundaries .....	261
Display Flood .....	262
<b>20 Lifting 2D Contours .....</b>	<b>264</b>
2D Contours tool box.....	264
Lift 2D Contours.....	265
Set Contour Elevation.....	267
Check Linear Elevations.....	268

## Programming Interface

<b>21 MDL Public Functions.....</b>	<b>270</b>
Calling Method .....	271
Function Prototypes .....	272
Triangulate Survey .....	281
<b>22 TIN File Format Specification .....</b>	<b>283</b>
File Organization .....	283
C Structure Definitions.....	286

## Additional Information

<b>23 Installation Directories .....</b>	<b>288</b>
<b>24 Configuration Variables .....</b>	<b>289</b>

# Getting Started

# 1 Documentation

---

## About the documentation

This document serves as a user's guide for two 32-bit versions of TerraModeler. The entry-level version, TerraModeler Lite, is functionally a subset of the full version, TerraModeler. Tools available in TerraModeler and TerraModeler Lite work identically in the two versions. Tools that are not available in TerraModeler Lite are marked as "*Not Lite*" in the documentation.

This User's Guide is divided into several parts:

- **Getting Started** - general information about TerraModeler and instructions on how to install and run the application.
- **Tutorial** - lessons in order to introduce the basic concepts and some of the most common tools in the application.
- **Tool Reference** - detailed descriptions of all the tools and menu commands in TerraModeler.
- **Programming Interface** - a list of public functions as well as a description of file formats.
- **Additional Information** - information about the installation configuration.

## Accessing the documentation online



The documentation is accessible as an Acrobat Reader PDF document which serves the role of online help. Accessing the electronic format of the documentation has the following advantages:

- You can conduct automated searches for keywords in topic names or body text.
- You can click hypertext to "jump" to related topics.

## Document conventions

The following conventions and symbols appear in this guide:

- **Data click** - mouse click with the data mouse button, usually the left button on a right-hand mouse.
- **Reset click** - mouse click with the reset mouse button, usually the right button on a right-hand mouse.
- **< >** - angle brackets used for keyboard keys, for example, <Return>.
- **Key in** - input a command in the key-in line of MicroStation and then press <Return>.
- **OR** - alternate procedures or steps in a procedure.
- **C:/TERRA** - paths to directories of files on a hard disk are written with capital letters.
- Icons used to specify special information:

Icon:	Appears next to:
	Notes and Hints
	Procedures

- When no distinction between MicroStation versions is necessary, this document refers to the CAD environment simply as "MicroStation".

## MicroStation documentation

This document is written under the assumption that the reader knows how to use basic MicroStation features. You should refer to the printed documentation or on-line help of MicroStation whenever you need information about using the CAD environment.

## 2 Introduction to TerraModeler

---

### Introduction

TerraModeler is a terrain modeling application built on top of MicroStation. 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, or laser points loaded in TerraScan or stored in LAS or TerraScan binary formats.

Theoretically, TerraModeler can handle an unlimited number of different surfaces in the same design file. In practice, however, the number and size of loaded models as well as the performance of TerraModeler and MicroStation 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 visualisation 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 laser points because it is tightly integrated with TerraScan. Laser data 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 laser data. In addition, laser points 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 laser points organized in a TerraScan project. Additionally, breaklines can be included in the contour line/lattice model/triangle creation.

## Terra application family

TerraModeler is just one in a full family of civil engineering applications. Almost all Terra applications are tightly integrated with MicroStation presenting an easy-to-use graphical interface to the user.

**TerraBore** is a solution for reading in, editing, storing and displaying bore hole data. You can triangulate soil layers with the help of TerraModeler.

**TerraMatch** fixes mismatches between laser points from different data strips automatically. It can be used for the calibration of a laser scanner system and for fixing project data.

**TerraPhoto** rectifies digital photographs taken during laser scanning survey campaigns and produces orthorectified images.

**TerraPipe** is used for designing underground pipes. It gives you powerful tools for designing networks of drainage, sewer, potable water, or irrigation pipes.

**TerraScan** processes laser scanning data. It reads in laser points from text or binary files and lets you view the point cloud three dimensionally, classify the data, and create vector data based on the points.

**TerraSlave** is a stand-alone application that processes TerraScan macros. It enables distributed processing and scheduling tasks to gain optimal time and working performance.

**TerraStereo** is a stand-alone application for viewing very large point clouds in mono and stereo mode. It utilizes advanced point rendering techniques and the graphics card memory in order to display huge amounts of points.

**TerraStreet** is an application for street design. It includes all the terrain modeling capabilities of TerraModeler. The street design process starts with the creation of horizontal and vertical geometries for street alignments.

**TerraSurvey** reads in traditional survey data and creates a three dimensional survey drawing. The application recognizes a number of survey data formats automatically.



## 3 Installation

---

### Hardware and software requirements

TerraModeler is built on top of MicroStation. You must have a computer system capable of running this CAD software.

To run TerraModeler, you must have the following:

- Pentium or higher processor
- Windows 8, 7, Vista, XP, or 2000 (64-bit version recommended)
- mouse
- 1024\*768 resolution display or better
- 1 GB RAM or better
- MicroStation V8, MicroStation V8i (Select Series 2 or higher) or Map PowerView. Check Terrasolid's web pages for a more detailed overview of compatible MicroStation versions.

The installation of TerraModeler requires about 5 MB of free hard disk space.

### Installation media

TerraModeler may be delivered on a CD/USB-Stick or as a zip file.

A **zip package** only contains the actual software - it does not include the PDF User's Guide.

A **Terra Installation CD/USB-Stick** includes the software and the online documentation. When you install from the CD/USB-Stick, the software and the documentation are copied to your hard disk. The CD/USB-Stick may include versions for multiple environments. You should locate the directory which corresponds to your operating system and MicroStation version.

Directory on CD/USB	For operating system	For MicroStation
\setup\eng	Windows	V8 or V8i

## Installation from zip file

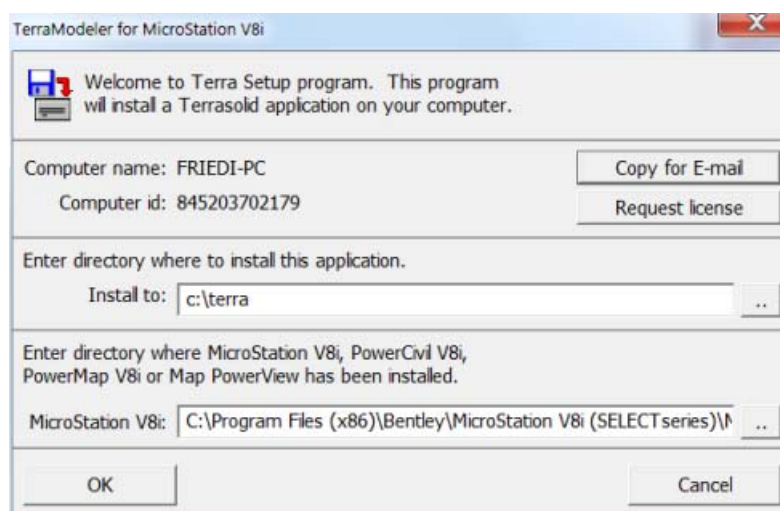
➤ **To install TerraModeler from a zip file:**

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 MicroStation has been installed. It automatically searches all local hard disks to find the MicroStation directory.

The installation dialog opens:



3. Enter the directory where to install TerraModeler.  
The default path is C:\TERRA. You may change this to another location. The specified directory is created automatically if it does not exist.
4. Check the MicroStation directory. Replace the path if the correct location was not found automatically.
5. Click OK to start the installation.

When the installation is finished, a message is displayed.

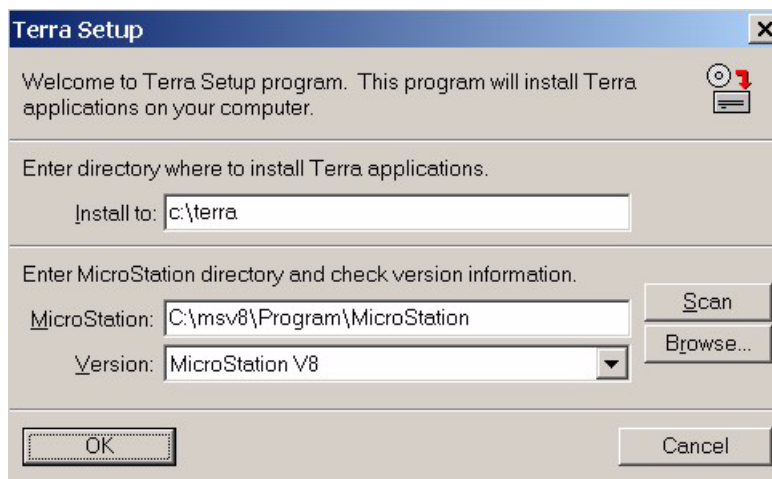
✍ See chapters **Installation Directories** on page 288 and **Configuration Variables** on page 289 for more information.

## Installation from CD/USB-Stick

➤ **To install TerraModeler from CD/USB-Stick:**

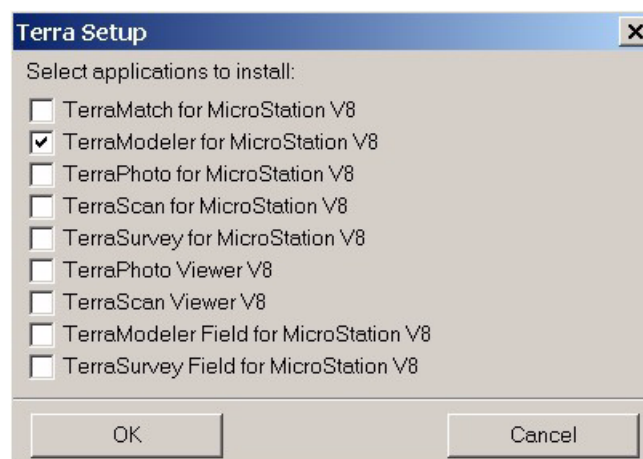
1. Insert the **Terra Installation CD/USB-Stick**.
2. Locate the correct directory which corresponds to your computer configuration.
3. Start **SETUP.EXE** from that directory.

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



4. Enter the directory where to install the application(s).  
The default path is C:\TERRA. You can change this to another location. The specified directory is created automatically, if it does not exist.
5. Check the **MicroStation** directory. Replace the path if the correct location was not found automatically.  
Alternatively, you can use the **Scan** button to automatically search the hard disk for the MicroStation installation or you can use the **Browse** button to locate the MicroStation executable yourself.
6. Check the MicroStation version information in the **Version** field. Select the correct version if it was not detected automatically.
7. Click OK to continue.

This opens another **Terra Setup** dialog:



8. Check **TerraModeler for MicroStation** item in the dialog.  
You may select other applications as well for which you have installation files.
9. Click OK to start the installation.  
A message is displayed when the installation is finished.

✍ See chapters **Installation Directories** on page 288 and **Configuration Variables** on page 289 for more information.

## 4 Starting TerraModeler

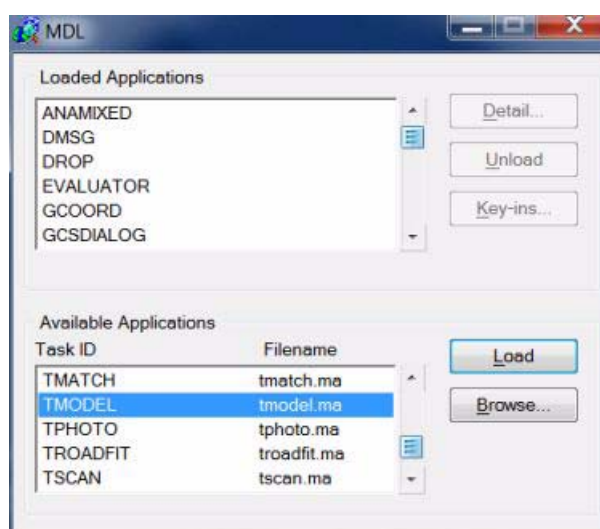
### Start TerraModeler

TerraModeler is an MDL application that runs on top of MicroStation.

➤ **To start TerraModeler:**

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

The **MDL** dialog opens:



2. In the **Available Applications** list, select **TMODEL**.
3. Click on the **Load** button.

OR

1. Key in MDL LOAD TMODEL.

The user settings determine what menus and tool boxes are opened during startup. See **Operation category** in TerraModeler **Settings** for more information. By default, TerraModeler adds an **Applications** menu to MicroStation's menu bar and opens the TerraModeler **Main** tool box:



- ✍ The **Available Applications** list shows all MDL applications that MicroStation is able to locate. MicroStation searches for MDL applications in the directories listed in the MS\_MDLAPPS configuration variable. If MicroStation can not find TMODEL.MA, you should check the value assigned to this configuration variable. Make sure the directory path of the TMODEL.MA file is included in the variable. To view configuration variables, select **Configuration** command from the **Workspace** menu in MicroStation. See also Sections **Installation Directories** on page 288 and **Configuration Variables** on page 289 for more information.

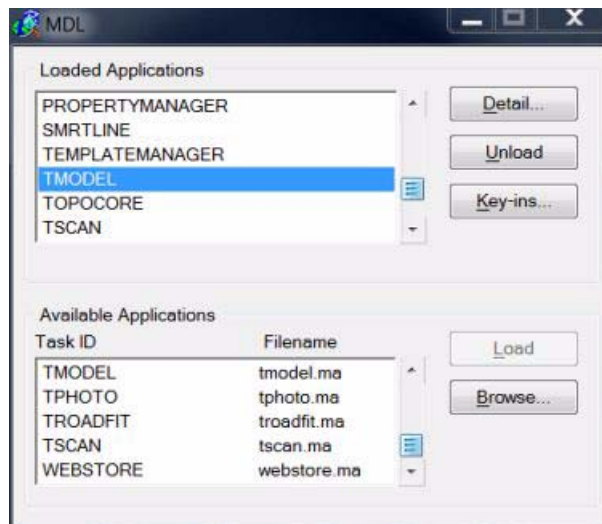
# Unload TerraModeler

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

➤ **To unload TerraModeler:**

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

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.

OR

1. Close TerraModeler's **Main** tool box.

This unloads the application, if the corresponding user setting is on. See **Operation category** in TerraModeler **Settings** for more information.

# Tutorial

## 5 Tutorial

This tutorial introduces some basic concepts and terminology used by TerraModeler. With the help of an example 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 MicroStation.

### TerraModeler interface

The user interface of TerraModeler is very similar to that of MicroStation. When you load TerraModeler, it opens its **Main** tool box. All TerraModeler tools are located in this tool box, which is actually a parent of 13 children tool boxes. These children tool boxes can be dragged out from the Main tool box. 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 fields 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 design file that is installed with 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** menu 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:\TERRA, you can find the design file in C:\TERRA\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. You can rotate this view or zoom in to see smaller details.

### Starting TerraModeler

Start TerraModeler as described in Section **Starting TerraModeler** on page 15.



## 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	<i>Triangulate Survey</i> after reading in the survey data with TerraSurvey.
Design file elements	<i>Triangulate View</i> or <i>Triangulate Elements</i> .
XYZ text file	<i>Surfaces</i> tool opens a window for managing surfaces. The <b>File</b> pulldown menu in this window has a command for importing xyz text files.

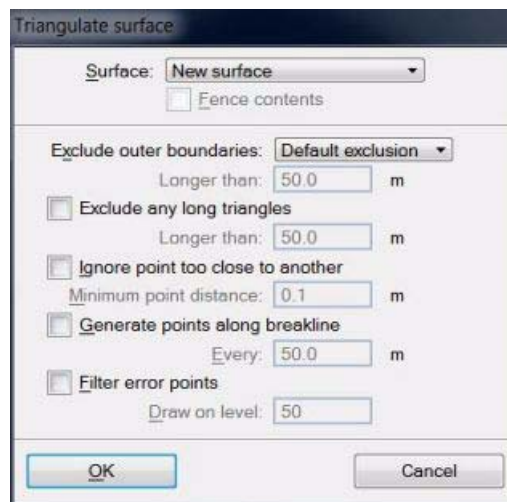
### ➤ To create a surface model in the example design file:

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.

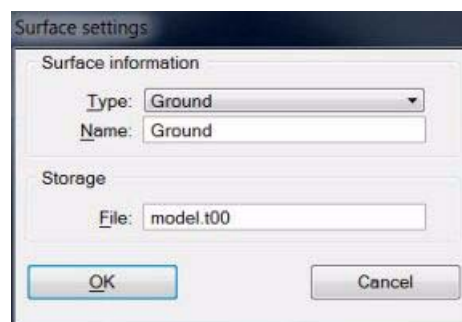
2. Select *Triangulate View* tool from the **Create Surfaces** tool box.
3. Identify view 2.

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.



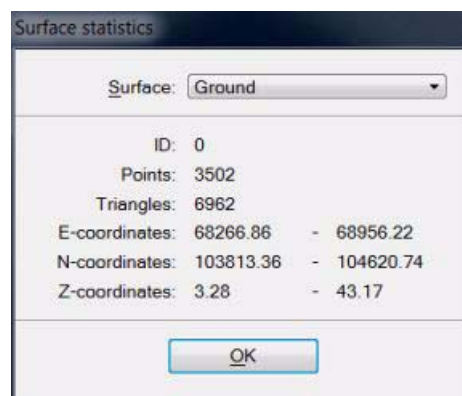
### To view surface statistics:

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

The **Surfaces** window opens showing a list of loaded surfaces. Select *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** tool box.

This tool can be used to excluding or including triangles. In TerraModeler's 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** tool box.

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:

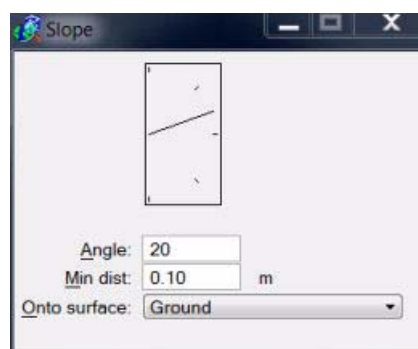
1. Bring view 4 in front of all other views.

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

### ➤ To calculate a slope upwards to the ground surface:

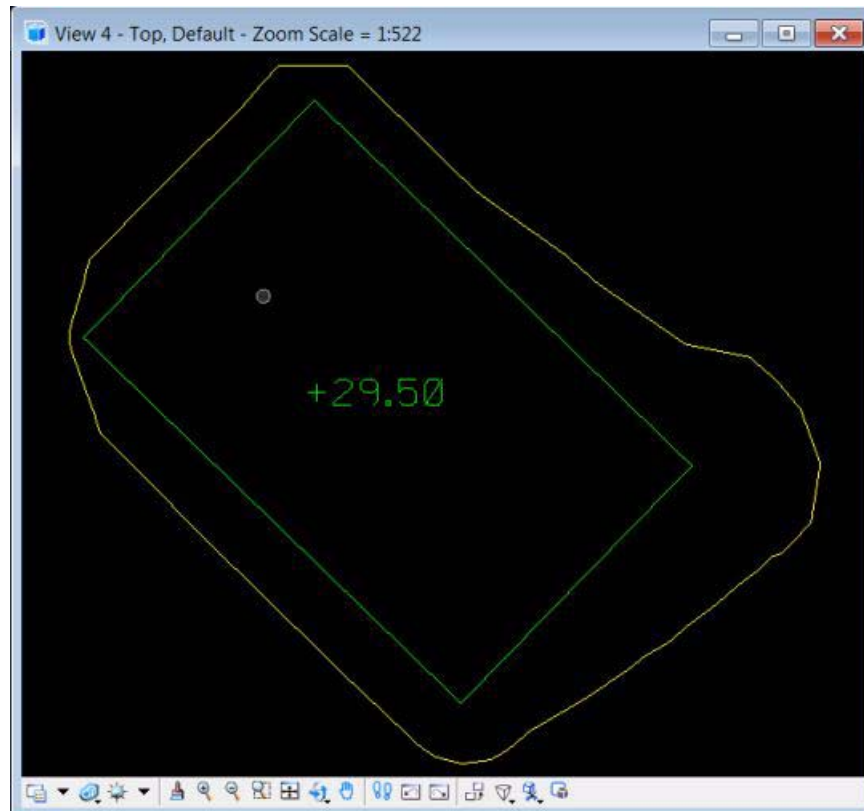
1. Select yellow as the active color.
2. Select the *Calculate Slope* tool.

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.



## 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.



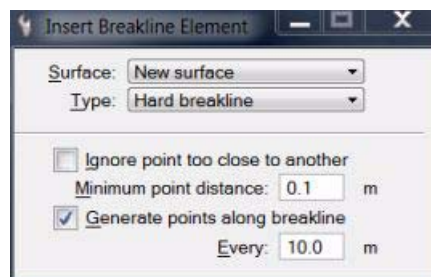
### To create a design surface:

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** tool box.

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



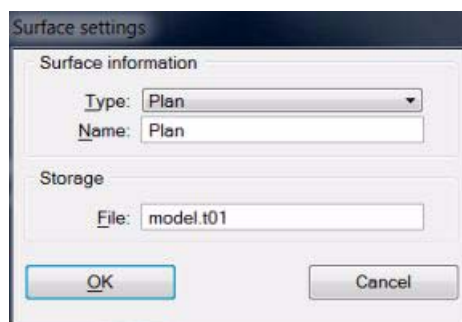
3. Select **New surface** as the **Surface** to insert points into and **Type** as **Hard breakline**.
4. Switch **Generate points along breakline** on and enter 10.0 into the **Every** field.

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 in turn 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.

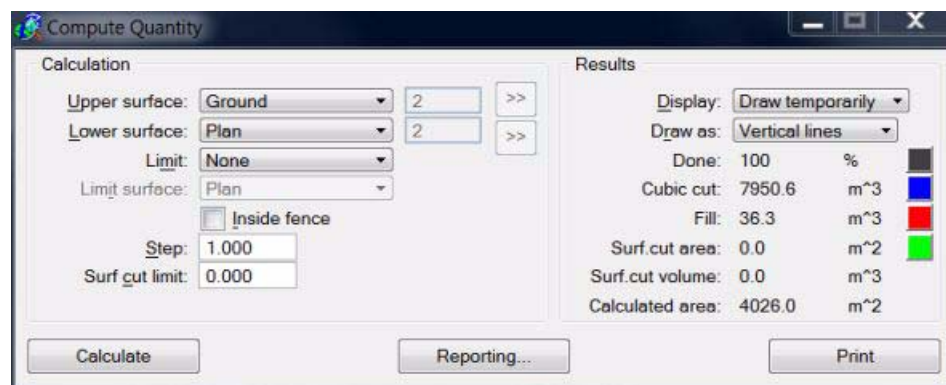
You are ready to compute excavation quantities using the *Compute Quantity* tool. It computes the volume between two surfaces and produces results summing up both, cut and fill volumes. The calculation is based on a grid method.



### To compute excavation quantities:

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

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** (MicroStation V8i) is selected as the **Display** option and **Vertical lines** as **Draw as** option.
6. Click on the **Calculate** button to start computation.

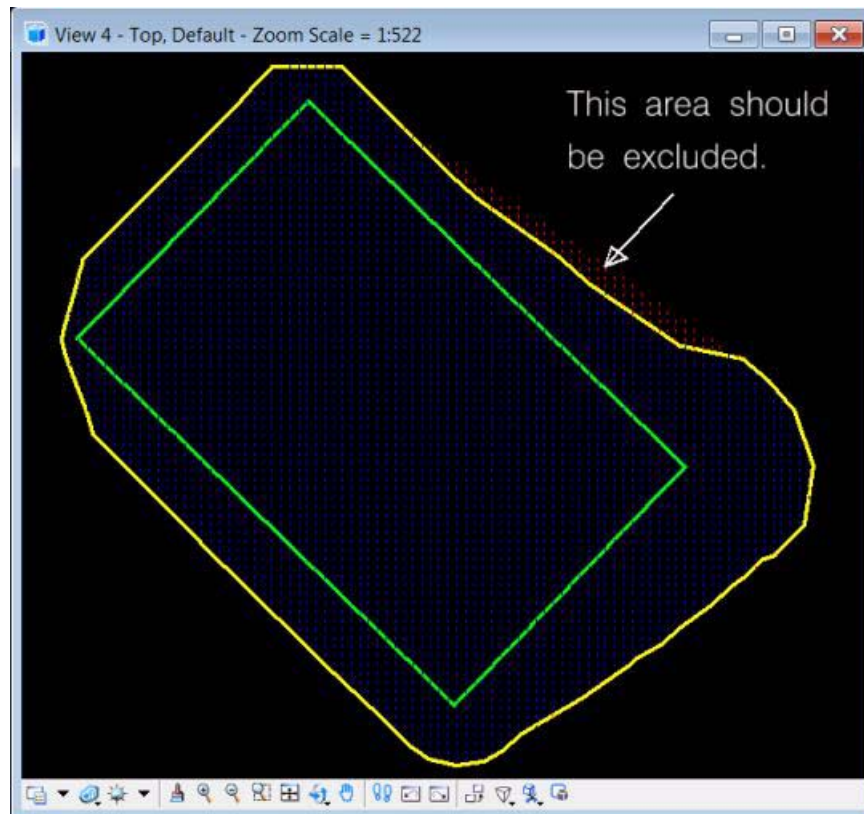
The application starts calculating the volume between the two surfaces. It displays the progress of the calculation as vertical linear elements. The color of each line indicates if that grid position results in cut or fill volume.

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** tool box.

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

2. Select **Edit / Exclude long triangles**.

The **Exclude long triangles** dialog opens:



3. Enter 12.0 as the **Maximum length** for outer boundary triangles.
4. Click OK.

➤ **To exclude triangles manually:**

1. Select **Exclude Triangle** tool from the **Edit Point** tool box.
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



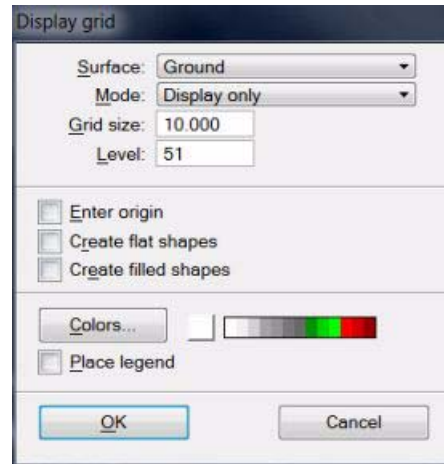
A surface model is often used for visualization purposes. TerraModeler is able to display a surface model with several display method.



**To display a colored grid:**

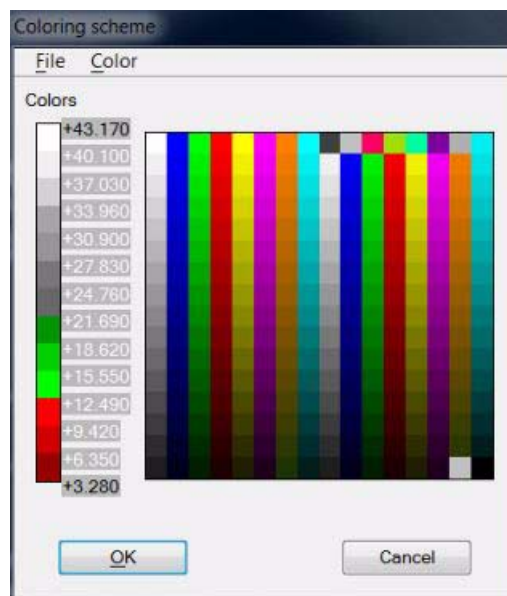
1. Select *Display Grid* tool from the **Display Surface** tool box.

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:\TERRA, you can find this file in C:\TERRA\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:

**Draw profile**

Name  
 Name: Tutorial example  
 Layout: No layout

Scale  
 Horizontal 1: 500  
 Vertical 1: 100  
 Label: Upper corner

Range  
 Color: 16  
 Start: 15.00 m  
 End: 45.00 m  
 Step: 5.00 m

Stations  
 Start station: 0.000  
 End station: 291.727  
 Increasing  
☐ Draw only partial alignment  
 From station: 0.000  
 To station: 0.000

**Profile labels**

Stations  
☒ Label stations  
 Step: 10.0 m

Label surface elevations  
☐ At fixed intervals  
 Step: 10.00 m  
☐ Draw vertical line  
☒ At locations : Peaks and pits  
☐ Label station of each location  
☐ Draw vertical line

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.








# Tool Reference

## 6 General Tools

### General tool box

The tools in the **General** tool box 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.

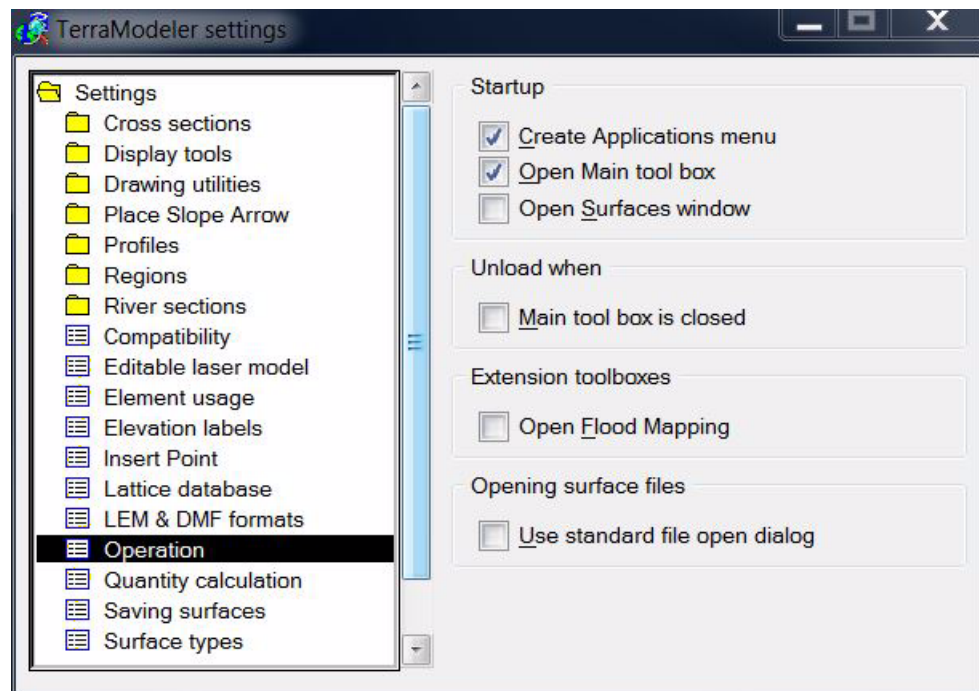


To:	Use:
Change user settings	 <i>Settings</i>
Define coordinate range and resolution	 <i>Define Coordinate Setup</i>
Set intended plotting scale / rescale elements	 <i>Set Scale</i>
Manage surfaces	 <i>Surfaces</i>
Save all modified surface models	 <i>Save Surfaces</i>
View information about TerraModeler	 <i>About TerraModeler</i>
View on-line help	 <i>Help On TerraModeler</i>

## Settings



*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 to the right of the category list.

### 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 MicroStation line styles.
Weight	Defines the line weight of range lines. Uses MicroStation 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 <b>Left</b> , on the <b>Right</b> , or on <b>Both</b> 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 MicroStation.
Size	Text size of elevation labels. Given in millimeters on paper.
Direction	Text rotation: <b>horizontal</b> or <b>vertical</b> .
Digits	Number of full-number digits in elevation labels.
Decimals	Number of decimals in elevation labels.

Setting:	Effect:
Color	Text and note line color for binding elevation labels. Uses the active color table in MicroStation.

### 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 MicroStation.
Size	Text size of offset labels. Given in millimeters on paper.
Color	Color of offset labels. Uses the active color table in MicroStation.
Direction	Text rotation: <b>horizontal</b> or <b>vertical</b> .
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: <ul style="list-style-type: none"> <li>• <b>First one at the bottom</b> - station values increase upwards.</li> <li>• <b>First one at the top</b> - station values increase downwards.</li> </ul>
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: <ul style="list-style-type: none"> <li>• <b>Fixed</b> - every color occupies the same height in the bar. You can define the height of one color as millimeters on paper.</li> <li>• <b>Relative</b> - height of a color is relative to its elevation or slope interval. You can define the height of the whole bar.</li> </ul>
Height	Height of one color in the color bar or of the whole color bar. This depends on the setting for <b>Scale</b> 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 <b>Border</b> color and line weight.
Draw boundary lines	If on, each color of the color bar is surrounded by a boundary line using the given <b>Boundary</b> color and line weight.
Symbology	Color and line weight of boundary labels of color schemes. Uses MicroStation colors and line weights.

Setting:	Effect:
Font	Font type of color scheme labels. Uses font types available in MicroStation.
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 MicroStation.
Size	Text size of contour labels. Given in millimeters on paper.
Style	Line style of contour line labels. Uses MicroStation line styles.
Weight	Line weight of contour line labels. Uses MicroStation 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 cycles	Number of color cycles. Use zero to create a gray scale display showing triangle slope only.
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 MicroStation.
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: <b>Above line</b> , <b>Extending from line</b> , or <b>Below line</b> .
Font	Font type of elevation label text elements. Uses font types available in MicroStation.
Size	Text size of elevation label texts. Given in millimeters on paper.



Setting:	Effect:
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.

### 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 MicroStation.
Size	Text size of slope measurement labels. Given in millimeters on paper.
Color	Color of slope measurement labels. Uses the active color table in MicroStation.
Style	Line style of slope measurement labels. Uses MicroStation line styles.
Weight	Line weight of slope measurement labels. Uses MicroStation line weights.

### Place Slope Arrow / Arrows category

**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: <b>Arrow start</b> or <b>Arrow center</b> .
Arrowheads	Number of arrowheads in a slope arrow: <ul style="list-style-type: none"> <li>• <b>One</b> - all arrows have one arrowhead.</li> <li>• <b>One to three</b> - number of arrowheads depends on the slope.</li> </ul>
Length	Two fields specifying the maximum and the minimum length of a slope arrow. Given in millimeters on paper.
Slope	Slope values at which the arrow length is equal to: <ul style="list-style-type: none"> <li>- maximum length</li> <li>- two thirds of maximum length</li> <li>- one third of maximum length</li> <li>- minimum length</li> </ul>

### Place Slope Arrow / Labels category

**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 text label is placed along a slope arrow.
Level	Level number in the MicroStation design file on which the arrow label is placed.
Font	Font type of slope arrow labels. Uses font types available in MicroStation.
Size	Text size of slope arrow labels. Given in millimeters on paper.
Accuracy	Number of decimals of the slope arrow label.

Setting:	Effect:
Symbology	Color and line weight of slope arrow labels. Uses MicroStation 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 MicroStation line styles.
Weight	Line weight of grid lines. Uses MicroStation 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 MicroStation.
Size	Text size of elevation grid labels in profiles. Given in millimeters on paper.
On side	Defines the location of grid labels: on the <b>Left</b> , on the <b>Right</b> , or on <b>Both</b> 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 MicroStation.
Size	Text size of profile labels. Given in millimeters on paper.
Color	Text color of station labels. Uses the active color table in MicroStation.
Direction	Text rotation of station labels: <b>horizontal</b> or <b>vertical</b> .
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	<p>Defines the type of information displayed in the bottom row:</p> <ul style="list-style-type: none"> <li>• <b>Surface elevations</b> - elevations of surfaces of the given <b>Surface</b> type</li> <li>• <b>Surface difference - Difference</b> between surfaces of the two given surface types</li> <li>• <b>Pipe</b> - space reservation for information of a given <b>Pipe type</b>. The content is filled by TerraPipe.</li> <li>• <b>Pipe stations</b> - space reservation for pipe stationing of a given <b>Pipe type</b>. The content is filled by TerraPipe.</li> <li>• <b>Stationing</b> - stations along the alignment element of the profile</li> <li>• <b>Other</b> - space reservation for any other content that can be added manually.</li> </ul>
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: <ul style="list-style-type: none"> <li>• <b>Active level</b> - draws are components on the active level in the design file.</li> <li>• <b>Level settings</b> - you can assign levels for the different components of the profile cell.</li> </ul>

## 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 MicroStation.
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 in MicroStation.
Text weight	Line weight of bottom row titles. Uses MicroStation line weights.
Font	Font type of bottom row titles. Uses font types available in MicroStation.
Size	Text size of bottom row titles. Given in millimeters on paper.

## Regions / Region design category

**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 MicroStation 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

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

Setting:	Effect:
Level usage	Levels used for region components by defining the MicroStation level numbers.

## Regions / Region types category

**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.
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

**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.

## River sections / Coefficients category

**Coefficients** category in **River sections** folder defines default friction coefficients for river sections.

Setting:	Effect:
Left bank	Default friction coefficient for the left bank.
Channel	Default friction coefficient for the channel.
Right bank	Default friction coefficient for the right bank.

## River sections / Eeby files category

**Eeby files** category in **River sections** folder defines a setting for handling codes in Eeby files.

Setting:	Effect:
First letters	Defines points that are ignored when reading in Eeby files.

## River sections / Symbology category

**Symbology** category in **River sections** folder defines the levels and the symbology of river cross section cells.

Setting:	Effect:
Surveyed section group	Level, color, line style, and line weight of the surveyed part of the river section. Uses MicroStation symbology.
Extended section group	Level, color, line style, and line weight of the part of the river section which has been derived by extending a section based on a ground surface. Uses MicroStation symbology.
Labels group	Level and symbology settings for river section labels. Uses MicroStation symbology and font types. The size of a label is given in millimeters on paper.

## Compatibility category

**Compatibility** category defines how triangles are displayed in preview mode using the *Display Triangles* tool. This applies only for MicroStation versions earlier than V8i.

Setting:	Effect:
Draw filled shapes in fast mode	If on, the setting effects the preview mode of displaying triangles ( <i>not MicroStation V8i</i> ).

## Editable laser model category

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

Setting:	Effect:
Naming	Naming method for new surface models: <ul style="list-style-type: none"> <li><b>Ask name</b> - user types a name when a new model is created.</li> <li><b>Automatic</b> - the new model gets the name given in the <b>Name</b> field automatically.</li> </ul>
Exclude	Method of exclusion of outer boundary triangles for new surface models: <ul style="list-style-type: none"> <li><b>No exclusion</b> - no triangles are excluded.</li> <li><b>Default exclusion</b> - the software computes a reasonable triangle length for exclusion from the surface's point density.</li> <li><b>By key-in length</b> - triangles up to the length given in the <b>Longer than</b> field are excluded.</li> </ul>
Exclude any long triangles	If on, any triangles with edge lengths longer than the value given in the following <b>Longer than</b> 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 <b>Radius</b> 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.

## 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 **Export / Graphical elements** command.

Setting:	Effect:
Justify	Location of the placement point (= point of elevation measurement): <ul style="list-style-type: none"> <li>• <b>Left, Center, Right   Top, Middle, Bottom</b> - placement point location relative to the label text box.</li> <li>• <b>Decimal point</b> - placement point is located at the decimal point.</li> </ul>
Offset dx	Offset between the placement point and the label in x (left-right) direction. Given in millimeters on paper. Only active if <b>Justify</b> is not set to <b>Decimal point</b> .
Offset dy	Offset between the placement point and the label in y (up-down) direction. Given in millimeters on paper. Only active if <b>Justify</b> is not set to <b>Decimal point</b> .
Font	Font type of elevation label text elements. Uses font types available in MicroStation.
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: <ul style="list-style-type: none"> <li>• <b>None</b> - no point marker is drawn.</li> <li>• <b>Character</b> - a given character is drawn as point marker.</li> <li>• <b>Zero length line</b> - a line element of zero length (= point) is drawn as point marker.</li> <li>• <b>Library cell</b> - a cell element from the active cell library is drawn as point marker.</li> </ul>
Font	Font type of the point marker character. Uses font types available in MicroStation. Only active if <b>Point marker</b> is set to <b>Character</b> .
Size	Text size of point marker character. Given in millimeters on paper. Only active if <b>Point marker</b> is set to <b>Character</b> .



Setting:	Effect:
Character	Character used as point marker symbol. Only active if <b>Point marker</b> is set to <b>Character</b> .
Weight	Weight of the point marker. Uses MicroStation line weights. Only active if <b>Point marker</b> is set to <b>Zero length line</b> .
Cell	Name of the cell used as point marker. Only active if <b>Point marker</b> is set to <b>Library cell</b> .
Size	Cell size give in millimeters on paper. Only active if <b>Point marker</b> is set to <b>Library cell</b> .

### Insert Point category

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

Setting:	Effect:
Draw permanently	If on, the text element is written to the design file. If off, the text element is displayed temporarily.

### Lattice database category

**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 <b>Directory</b> are drawn in profiles and cross sections.
Color	Color of the lattice database files. Uses the active MicroStation color table. Only active if <b>Draw in profiles</b> is switched on.
Weight	Line weight of the lattice database files. Uses MicroStation line weights. Only active if <b>Draw in profiles</b> is switched on.
Style	Line style of lattice data base files: <ul style="list-style-type: none"> <li>• <b>Standard</b> - uses MicroStation line styles.</li> <li>• <b>Custom</b> - uses user-defined line styles.</li> </ul> Only active if <b>Draw in profiles</b> is switched on.
Style name	Name of a user-defined line style. The <b>Select</b> button opens a list of custom styles. Only active if <b>Style</b> is set to <b>Custom</b> .
Style scale	Scale of a user-defined line style. Expressed as relation between the original custom style and a scale factor. Only active if <b>Style</b> is set to <b>Custom</b> .
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 <b>Active design</b> file, <b>Reference files</b> , or <b>Both</b> .
Water bodies	Level in the design file(s) where water polygons are located.
Islands	Level in the design file(s) where island polygons are located.
Row order	Order of rows in LEM files: <b>Bottom first</b> or <b>Top first</b> .

## Operation category

**Operation** category defines what menus the application opens at startup and how the application can be closed.

Setting:	Effect:
Create Applications Menu	If on, TerraModeler creates an <b>Applications</b> pulldown menu in MicroStation's menu bar at startup. This menu contains items for opening TerraModeler tool boxes.
Open Main tool box	If on, the application opens its <b>Main</b> tool box at startup.
Open Surfaces window	If on, the application opens the <b>Surfaces</b> window at startup.
Main tool box is closed	If on, the application is unloaded when the <b>Main</b> tool box is closed.
Open Flood Mapping	If on, the application opens the <b>Flood Mapping</b> extension tool box at startup.
Use standard file open dialog	If on, the application uses a standard Windows dialog for opening surface files.

## 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 design 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*, **Utility / Produce contours**, **Utility / Produce lattice models**, and **Utility / Produce triangles**.

The settings in **Triangulate Survey** category determine what applications can be called to enquire 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: <ul style="list-style-type: none"> <li>• <b>Active feature list</b> - call TerraSurvey to enquire suitability of an element based on the current feature list.</li> <li>• <b>Creation time settings</b> - 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.</li> </ul>
PRO600	If on, call Leica's PRO600.
DigiCad	If on, call GeoCad's DigiCad.
DigiCad - Load if not loaded	If on, load DIGICAD.MA if necessary. Only active if <b>DigiCad</b> is switched on.
Espa	If on, call ESPA System's Espa software.
Scan reference files	If on, the application checks reference files attached to the active design 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 MicroStation element.

Setting:	Effect:
Apply to snapped points	If on, the elevation value derived from the surface given in the <i>View Elevation</i> 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.

## Define Coordinate Setup



*Define Coordinate Setup* tool sets up the coordinate system that TerraModeler uses for surface model points. It determines the coordinate range inside which all points must be located and the resolution to which coordinate values are rounded. The coordinate setup is stored into the active design file and is used by TerraScan, TerraPhoto, TerraModeler, and TerraMatch.

TerraModeler uses signed 32 bit integer values for storing coordinates of points. 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 a coordinate setup:

1. Select the *Define Coordinate Setup* tool.

This opens the **Define Coordinate Setup** dialog:

Units and resolution		
Master unit:	m	
Resolution:	1000	per m

Origin		
Easting:	2500000.000	
Northing:	6700000.000	
Elevation:	-0.000	

Coordinate range		
Eastings:	+352516	.. +4647484
Northings:	+4552516	.. +8847484
Elevations:	-2147484	.. +2147484

OK Cancel

2. Define settings and click OK.

This modifies the coordinate system used by all Terra Applications in the active design file.

## MicroStation SE and MicroStation J

Each design file contains a definition of a 32 bit integer coordinate system which MicroStation uses internally for vector elements. All applications share the same coordinate setup with MicroStation. When you change the coordinate setup with *Define Coordinate Setup*, it changes the design file coordinate system.

✍ Since Terra Applications' version 009.00x these MicroStation versions are no longer supported.

## MicroStation V8

MicroStation V8 uses 64 bit values for storing vector elements. TerraModeler uses a coordinate setup which is separate from the design file coordinate system. The application uses a default coordinate setup of 100 integer steps for each master unit. You can use *Define Coordinate Setup* tool to change the coordinate setup which the application stores in the design file but it does not affect MicroStation itself or the vector elements.

## Set Scale



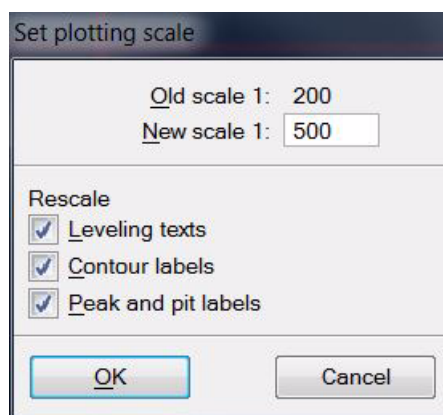
*Set Scale* tool sets the intended plotting scale of the design and rescales elements. Plotting scale affects the size of text elements that TerraModeler creates.



**To set the intended plotting scale:**

1. Select the *Set Scale* tool.

The **Set plotting scale** dialog opens:



2. Enter value for the new scale.
3. Select elements to be rescaled.
4. Click OK.

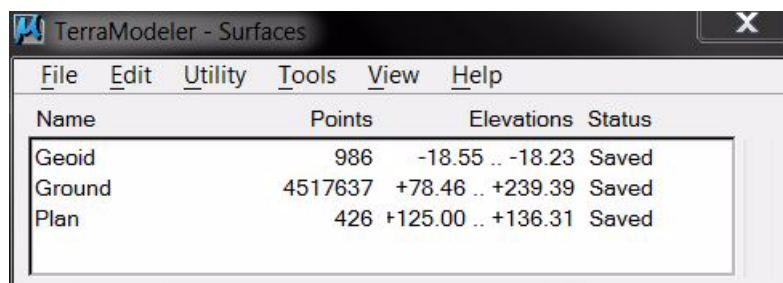
The selected element types are rescaled.

## 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** on page 191 for a complete description of the menu commands.

## 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 a category for controlling when surface models are saved automatically. See **Saving surfaces category** on page 42 for more information.

## 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:

License information

**TerraModeler for MicroStation**  
**Version 013.005**

Number:

User name:

Computer name:

Computer ID:

Check sum:

Code:

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:  
[www.terrasolid.com/registration.php](http://www.terrasolid.com/registration.php).

## Help On TerraModeler



*Help On TerraModeler* tool launches Acrobat Reader for accessing this Users' Guide in PDF format. The PDF must be stored in the */docs* folder of your Terra Software installation directory.

If you installed TerraModeler in the default directory C:\TERRA, the Users' Guide must be stored in C:\TERRA\DOCS\TMODEL.PDF.

The PDF has hypertext links built in, so you can jump between topics by clicking on the topic names highlighted in green color.



Accessing the PDF also requires that you have the Acrobat Reader installed on your computer. The software looks for a file named ACRORD32.EXE. If the file can not be found, you are asked to locate the file on the hard disk manually.










## 7 Creating Surface Models

### Create Surfaces tool box

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



TerraModeler surface models are stored as triangulated surface files. A description of the file format can be found in Chapter **TIN File Format Specification** on page 283.

To:	Use:
Triangulate survey elements	 <i>Triangulate Survey</i>
Triangulate elements visible in a view	 <i>Triangulate View</i>
Triangulate elements by type and level	 <i>Triangulate Elements</i>
Insert elements as random points	 <i>Insert Random Elements</i>
Insert elements as breaklines	 <i>Insert Breakline Element</i>
Triangulate breaklines and/or laser points	 <i>Triangulate Multiple Sources</i>
Rebuild models	 <i>Rebuild Models</i>

## Triangulate Survey



*Triangulate Survey* tool creates a surface model from survey data that has been drawn into the design 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 by Terrasolid.
- **PRO600** - a mapping application by Leica.
- **DigiCad** - a map digitizing application by GeoCad.
- **Espa** - a mapping application 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** on page 281.

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:

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.

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

The image shows a 'Surface settings' dialog box. It contains two main sections: 'Surface information' and 'Storage'. In the 'Surface information' section, there is a 'Type' dropdown menu set to 'Plan' and a 'Name' text field containing 'Plan 1'. In the 'Storage' section, there is a 'File' text field containing 'plan1.tin'. At the bottom of the dialog, there are two buttons: 'OK' and 'Cancel'.

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.
7. Click OK.

TerraModeler starts scanning the design file for survey elements.

Setting:	Effect:
Surface	Determines how the surface is created: <ul style="list-style-type: none"> <li>• <b>New surface</b> - a new surface is created.</li> <li>• <b>“surface name”</b> - new features are added to an existing surface model that is loaded in TerraModeler.</li> </ul>
Inside fence only	If on, only elements inside a fence are triangulated.
Exclude outer boundaries	Exclusion method of outer boundary triangles of the surface model: <ul style="list-style-type: none"> <li>• <b>No exclusion</b> - no triangles are excluded.</li> <li>• <b>Default exclusion</b> - the software computes a reasonable triangle length for exclusion from the surface’s point density.</li> <li>• <b>By key-in length</b> - triangles with edge lengths given in the subsequent <b>Longer than</b> field are excluded.</li> </ul>
Exclude any long triangles	If on, any triangles with edge lengths longer than the value given in the subsequent <b>Longer than</b> 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 <b>Minimum point distance</b> , 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 <b>Every</b> 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 <b>Draw on level</b> field.

## 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.

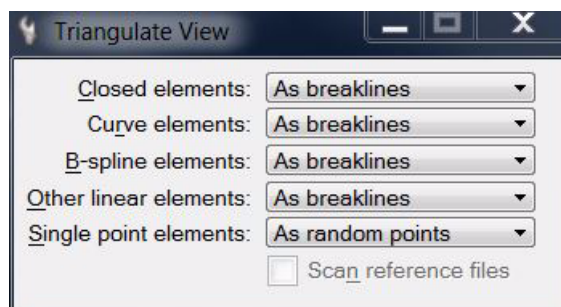
You can use this tool to create a new surface model or to add points to an existing model loaded in TerraModeler.



### 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. Continue as described for **Triangulate Survey** on page 50.

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

Setting:	Effect:
Closed elements	How to use closed elements: <b>As breaklines</b> , <b>As 3D holes</b> or <b>As 2D holes</b> .
Curve elements	How to use curve elements: <b>As breaklines</b> or <b>As contours</b> .
B-spline elements	How to use B-splines: <b>As breaklines</b> or <b>As contours</b> .
Other linear elements	Other linear elements are always used as breaklines.
Single point elements	Single point elements are always used as random points.
Scan reference files	If on, elements in reference files that are attached to the active design file are used for surface creation as well.

## Triangulate Elements



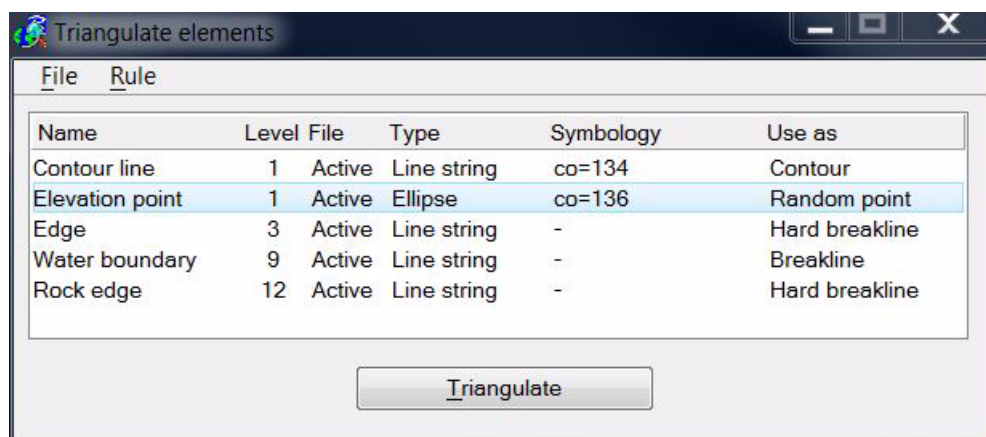
*Triangulate Elements* tool creates a surface model from graphical elements filtered by level, element type, and symbology. You can save the filtering rules to a text file for later use.

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. Continue as described for **Triangulate Survey** on page 50.

## Filtering rules

The **Triangulate elements** window lists filtering rules for graphical elements in a design 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 design file.

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

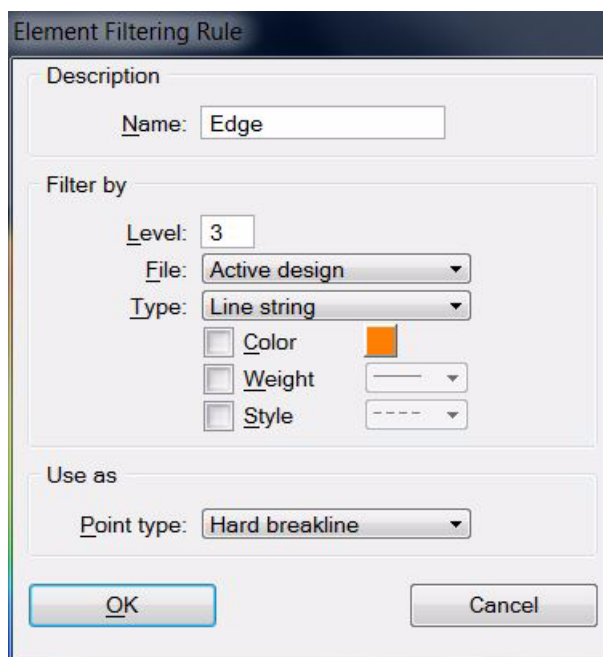
To open a set of filtering rules from a file, select **Load** from the **File** pulldown menu.

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.

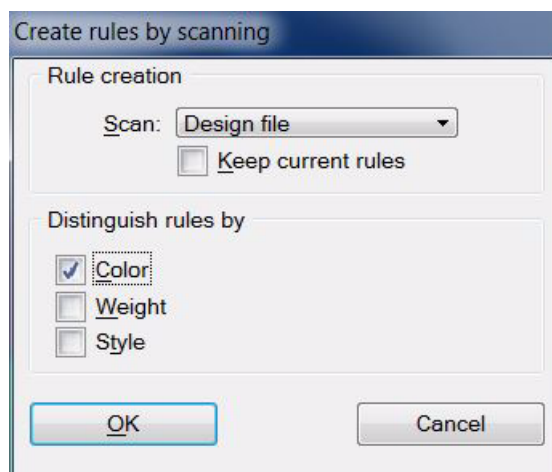
Setting:	Effect:
Name	Description of the feature depicted by the elements.
Level	The rule applies to elements on this design file level.
Type	Element type the rule applies to.
Color	If on, the rule applies only to elements with a given line <b>Color</b> .
Weight	If on, the rule applies only to elements with a given line <b>Weight</b> .
Style	If on, the rule applies only to elements with a given line <b>Style</b> .
Point type	Defines how the element vertices are used as points in the surface model: <b>Random point</b> , <b>Hard breakline</b> , <b>Soft breakline</b> , <b>Guiding breakline</b> , <b>Contour</b> , <b>2D hole</b> , <b>3D hole</b> , <b>2D boundary</b> , or <b>3D boundary</b> .

 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 design 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.

The software creates the rules automatically.

Setting:	Effect:
Scan	Defines where the software scans elements to create the rules: <ul style="list-style-type: none"> <li>• <b>Design file</b> - in the active design file</li> <li>• <b>Design and references</b> - in the active design file and in reference files attached to the active design file</li> <li>• <b>View contents</b> - in a selected view. Only elements visible in the selected view are used for rule creation.</li> </ul>
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.

## Insert Random Elements



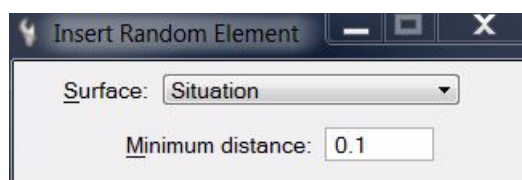
*Insert Random Elements* tool adds graphical elements as random points to a surface model. You can use this tool to create a surface model or to add points to an existing model loaded in TerraModeler.



### 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.
4. Accept the element.

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 element(s).

If a new surface is created, the **Triangulate surface** dialog opens. Continue as described for **Triangulate Survey** on page 50.

TerraModeler inserts the element(s) to the surface.

Setting:	Effect:
Surface	Surface to which points are inserted: <ul style="list-style-type: none"> <li>• <b>New surface</b> - a new surface is created.</li> <li>• <b>“surface name”</b> - points are added to an existing surface model that is loaded in TerraModeler.</li> </ul>
Minimum distance	Thins the model by leaving out unnecessary points. If the distance between two points is less than <b>Minimum distance</b> , one of the points is left out.



## Insert Breakline Element

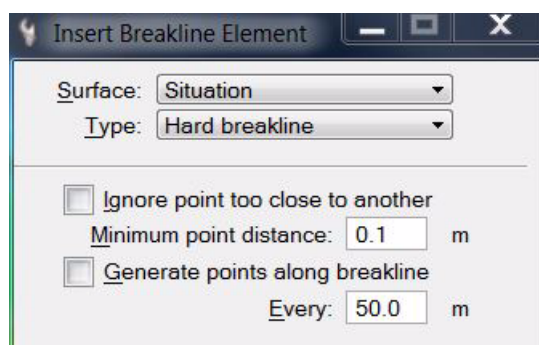


*Insert Breakline Element* tool adds graphical elements as breakline points to a surface model. You can use this tool to create a surface model or to add points to an existing model loaded in TerraModeler.

➤ **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.
5. Accept the element.

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 element(s).

If a new surface is created, the **Triangulate surface** dialog opens. Continue as described for **Triangulate Survey** on page 50.

TerraModeler inserts the element(s) to the surface.

Setting:	Effect:
Surface	Surface to which breakline points are inserted: <ul style="list-style-type: none"> <li>• <b>New surface</b> - a new surface is created.</li> <li>• <b>“surface name”</b> - points are added to an existing surface model that is loaded in TerraModeler.</li> </ul>
Type	Defines how the element vertices are used as points in the surface model: <b>Random point</b> , <b>Hard breakline</b> , <b>Soft breakline</b> , <b>Guiding breakline</b> , <b>Contour</b> , <b>2D hole</b> , <b>3D hole</b> , <b>2D boundary</b> , or <b>3D boundary</b> .
Ignore point too close to another	If on, filter out unnecessary points. If the distance between two points is less than <b>Minimum point distance</b> , 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 <b>Every</b> field.

## Triangulate Multiple Sources



*Triangulate Multiple Sources* tool creates a new surface model from breakline elements and/or laser 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 loaded points in TerraScan.

The breaklines can be defined as vector elements filtered by rules. See **Filtering rules** on page 53 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**.

Laser data can be used from laser points loaded into 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:

**Triangulate Multiple Sources**

**Surface model**

Create As: Rebuildable model

Type: Ground

Name: Ground

File name: ground.tin

**Data sources**

☒ Vector elements  
Rule file: R:\Data\model\_filter\_rules.txt Browse...

☐ Survey elements

☒ Laser points in TerraScan  
Classes: 2

☒ Make this 'Editable model'

☐ Run preprocessing macro Browse...  
Macro:

☐ Laser points from binary file  
File: R:\Data\laser\ground.bin Browse...

OK Cancel

2. Define settings for the surface model creation.
3. Select data sources and define settings.
4. Click OK.

This opens the **Triangulate surface** dialog. Continue as described for **Triangulate Survey** on page 50.

Setting:	Effect:
Create As	Defines surface model rebuild options: <ul style="list-style-type: none"> <li>• <b>Normal model</b> - the model can not be rebuild.</li> <li>• <b>Rebuildable model</b> - the model can be rebuild after changes to the source data using the <i>Rebuild Models</i> tool.</li> </ul>
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 <b>Rule file</b> .
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 <b>Classes</b> 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 <b>Laser points in TerraScan</b> is switched on.
Run preprocessing macro	If on, the given TerraScan <b>Macro</b> is applied to the loaded points before the surface model is created. Only active if <b>Laser points in TerraScan</b> is switched on.
Laser points from binary file	If on, laser points from the given TerraScan binary <b>File</b> stored on a hard disk are included in the model.

## 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.

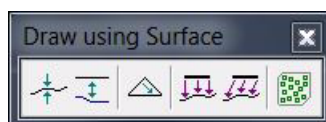
If there are one or more rebuildable models loaded in TerraModeler, the rebuilding process starts immediately for these models.







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

## 8 Drawing Elements

### Draw using Surface tool box

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



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

## View Elevation



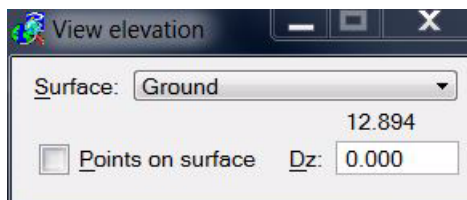
*View Elevation* tool shows the elevation of a surface model at the mouse pointer location. It also supports the placement of 3D vector elements by deriving surface elevations for the elements.



### 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 **Points on surface** lock on.

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 MicroStation 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.

## 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.

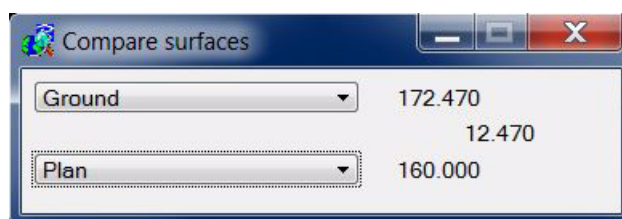
The tool requires at least two surfaces loaded in TerraModeler.



**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.

## View Slope



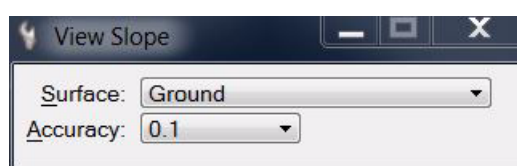
*View Slope* tool shows the slope gradient and the slope direction of a surface model triangle at the mouse pointer location.



**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 MicroStation.

Setting:	Effect:
Surface	Name of the surface for slope gradient and direction display.
Accuracy	Number of decimals for the slope gradient display.

## Drop Element On Surface



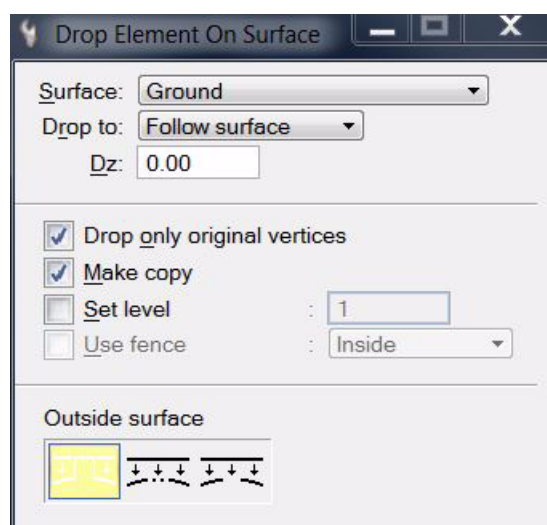
*Drop Element On Surface* tool drops elements to follow a surface model. The XY shape of an element remains unchanged but its Z shape is forced to follow the elevation of a surface model.

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.
4. 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: <ul style="list-style-type: none"> <li>• <b>Follow surface</b> - each vertex gets the elevation value of the surface at its location.</li> <li>• <b>Highest elevation</b> - all vertices get the elevation value of the highest vertex.</li> <li>• <b>Lowest elevation</b> - all vertices get the elevation value of the lowest vertex.</li> </ul>
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.



Setting:	Effect:
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 <b>Drop to</b> is set to <b>Follow surface</b> .
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	If on, a MicroStation fence is used to determine what elements are effected. MicroStation fence modes are used: <ul style="list-style-type: none"> <li>• <b>Inside</b> - only elements that are fully inside a fence are dropped.</li> <li>• <b>Overlap</b> - elements that overlap with a fence are dropped.</li> <li>• <b>Clip</b> - the part of an element that is overlapped by a fence is dropped, other parts are not effected.</li> <li>• <b>Void</b> - only elements that are fully outside a fence are dropped.</li> <li>• <b>Void overlap</b> - elements that do not overlap with a fence are dropped.</li> <li>• <b>Void clip</b> - the part of an element that is not overlapped by a fence is dropped, overlapped parts are not effected.</li> </ul>
Outside surface	Defines the way an element is drawn outside the surface model area. The methods are: <ul style="list-style-type: none"> <li>• Outside part remains unchanged in elevation but is drawn with active color, line style, and line width.</li> <li>• Outside part gets interpolated elevation values and is drawn with active color, line style, and line weight.</li> <li>• Outside part is not drawn and thus, resulting in a gap.</li> </ul>



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.

## Calculate Slope



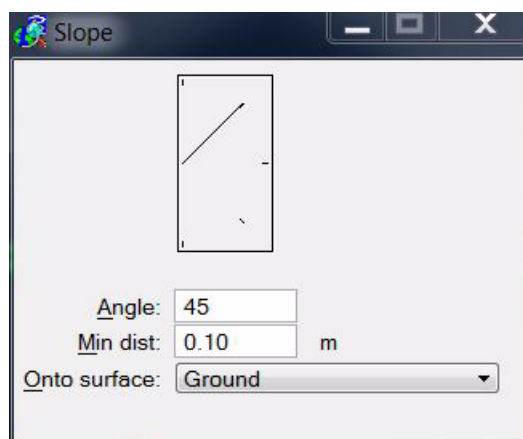
*Calculate Slope* tool calculates a slope from a graphical element onto a surface model. The starting element is either the top or the bottom edge of a slope. The tool calculates the other edge of the slope and draws it as a new line string element. The new element follows the elevation of a selected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

The starting element has to be linear. Valid MicroStation 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 and a minimum distance value.
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 degrees. 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.
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.

## Fill Area With Cells



*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 closed 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 closed 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: <ul style="list-style-type: none"> <li>• <b>Normal cell</b> - cells from a MicroStation cell library are used.</li> <li>• <b>RPC cell</b> - RPC cells are used.</li> </ul>
Cell name	Name of a two -or three-dimensional cell. This is only active if <b>Cell type</b> is set to <b>Normal cell</b> .
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 <b>Cell type</b> is set to <b>RPC cell</b> .
Level	Level on which cells are placed.

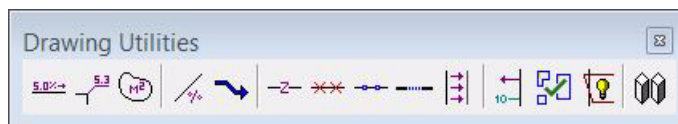
Setting:	Effect:
Area borders	Defines how cells are placed close to the area boundaries: <ul style="list-style-type: none"> <li>• <b>Cells completely within borders</b> - cells have to be completely within the area boundaries.</li> <li>• <b>Cells may overlap borders</b> - cells may partially overlap area boundaries.</li> </ul>
Bottom as	Name of a surface model for the bottom elevation of cells. If set to <b>Fixed elevation</b> , all cells are placed at the given <b>Elevation</b> .
Top as	Name of a surface model for the top elevation of cells. If set to <b>Fixed height</b> , all cells are drawn with the given <b>Height</b> .
Radius as	Defines the width of cells: <ul style="list-style-type: none"> <li>• <b>Constant</b> - use the given <b>Radius</b> for all cells.</li> <li>• <b>Ratio from height</b> - calculates the width as a direct ratio from the cell height defined in the cell library.</li> </ul>
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 Terrasolid software. They are purchased by Archvision ([www.archvision.com](http://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.

## Drawing Utilities tool box

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



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

## 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:

$$\text{Slope} = 100 * (\text{End } z - \text{Start } z) / \text{Horizontal length of element}$$

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.

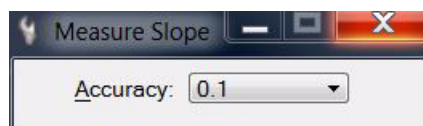
The element to be measured has to be linear. Valid MicroStation element types include lines, line strings, curves, and arcs.

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. Select the number of decimals for slope gradient display in the **Accuracy** list.
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 MicroStation status bar.

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

## 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 design 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 MicroStation.



### 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.

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

## 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 design file.

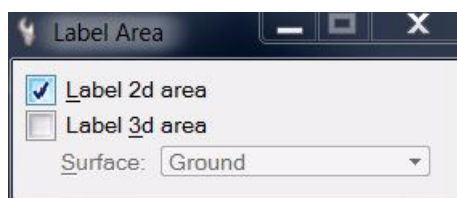
The tool places the area size value and possibly some prefix and/or suffix. The settings for the tool are defined in *Drawing utilities / Label Area category* of the TerraModeler **Settings**. The symbology of the label is further determined by the active symbology settings in MicroStation.



### To create area labels:

1. Select the *Label Area* tool.

This opens the **Label Area** dialog:



2. Select the area type for labeling.
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 design file. You can continue with step 2 or 3.

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 <b>Surface</b> .

## 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.

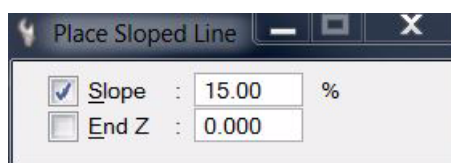
The slope is expressed as a percentage by the following mathematical expression:

$$\text{Slope} = 100 * \text{Vertical distance} / \text{Horizontal distance}$$

### ➤ 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. Enter the start point of the line with a data click.
4. Enter the end point of the line data click.

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

## 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.
3. Accept the element selection.

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).

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



## Set Linear Elevation



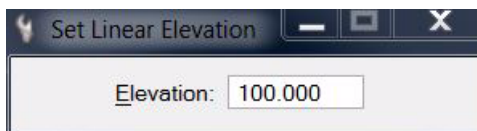
*Set Linear Elevation* tool adjusts a linear element to a given elevation. It sets a constant elevation for each vertex of the element.



**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.
4. Accept the element.

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).

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

## 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.
4. Accept the element.

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).

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

## Densify Linear Element



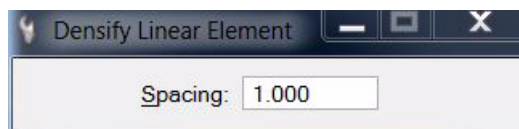
*Densify Linear Element* tool adds intermediate vertices to linear elements. A spacing value determines the desired distance between consecutive vertices. The tool adds vertices if the distance between two consecutive vertices in the original element is longer than  $1.333 * Spacing$ .



**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).

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

## 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 MicroStation level using the active symbology settings.

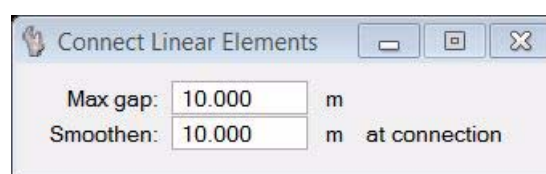
The tool can be used, for example, to create a continuous line from separate line elements that were extracted automatically, such as paint markings extracted from mobile laser scanner data in TerraScan.



### To connect linear elements:

1. Select line elements using the MicroStation *Selection* tool.
2. Select the *Connect Linear Elements* tool.

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



3. Define settings.
4. Place a data click inside a MicroStation view in order to accept the line string creation.

This draws the new line string into the design 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.
Smoother	Determines the level of smoothing in XY and Z applied to the new line string. The larger the value, the more smoothing is applied.



Each data click inside a MicroStation 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 MicroStation.

## Copy Linear Element



*Copy Linear Element* tool creates a parameterized 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:

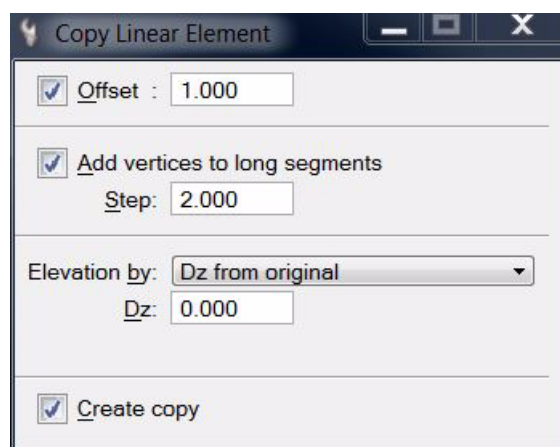
- no change in elevation
- set elevation to a fixed value
- use a given elevation difference from the original element
- use a given elevation difference or a gradient from another linear element



### 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, 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.

Setting:	Effect:
Elevation by	<p>Defines the method of elevation value derivation for the new element:</p> <ul style="list-style-type: none"> <li>• <b>Keyin value</b> - constant elevation given as a numerical value in the <b>Elevation</b> field.</li> <li>• <b>Elevation point</b> - constant elevation given by a data click.</li> <li>• <b>Dz from original</b> - elevation difference from the original element given in the <b>Dz</b> field.</li> <li>• <b>Dz from another element</b> - elevation difference from another element given in the <b>Dz</b> field.</li> <li>• <b>Gradient from another element</b> - gradient percentage from another element given in the <b>Gradient</b> field.</li> </ul>
Projection	<p>Defines how the elevation of the new element is derived from another element:</p> <ul style="list-style-type: none"> <li>• <b>Shortest distance</b> - each vertex is compared against the closest position on the other element.</li> <li>• <b>Perpendicular</b> - each vertex is projected perpendicularly to the other element and the elevation is derived from the projected location.</li> </ul> <p>Only active if <b>Elevation</b> by is set to <b>Dz from another element</b> or <b>Gradient from another element</b>.</p>
Create copy	<p>If on, a copy of the original element is created. The original element remains unchanged.</p>

## 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.

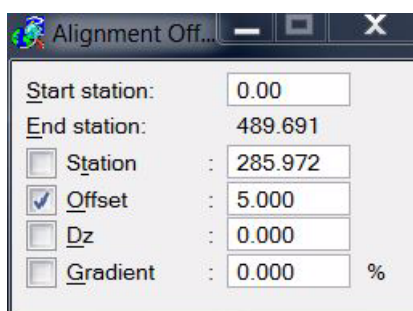
*Alignment Offset* tool opens a dialog which dynamically displays station and offset values at the mouse pointer location. You can lock data points by station, offset, elevation difference, or gradient from the selected alignment.



### 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 MicroStation 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.
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.



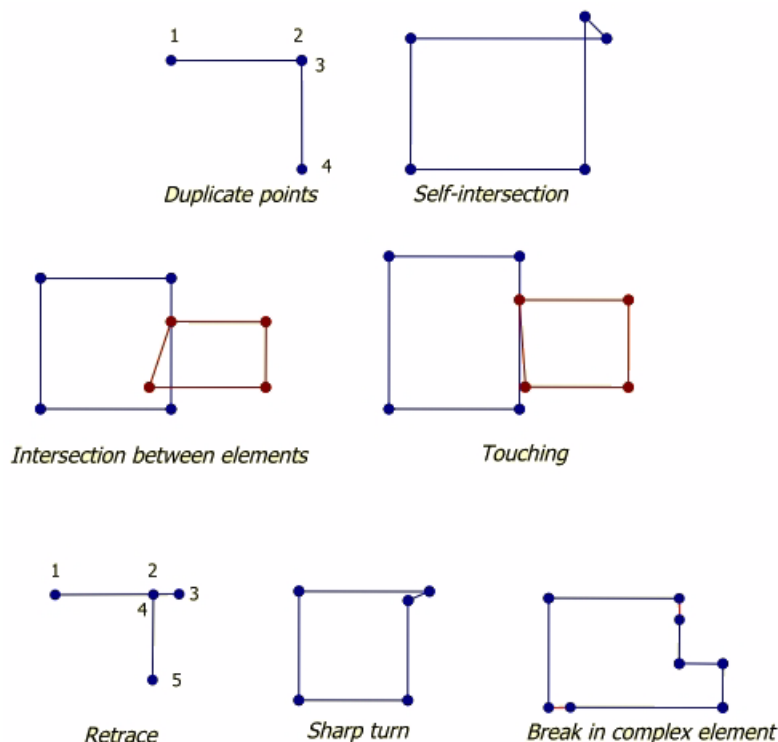
Make sure to turn off **Alignment Offset** locks if you do not need it any longer. As it effects all data points, it may interfere with your normal work. The lock is turned off when the **Alignment Offset** dialog is closed.

## 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 using the *MicroStation Selection* tool.
2. Select the *Validate Linear Elements* tool.

This opens the **Validate linear elements processing** dialog:

Validate linear elements processing

☒ Duplicate point
☐ Sharp 3D turn

☒ Self-intersection
☐ Sharp z turn

☒ Intersection between elements
☒ Sharp xy turn

☒ Touching
☒ Break in complex element

☒ Retrace

Stroking: 0.0010

Touching: 0.0100
☒ Exclude crossings

Angle: 145.00

z-Angle: 60.00

OK

Cancel

3. Select which geometrical flaws you want to validate.

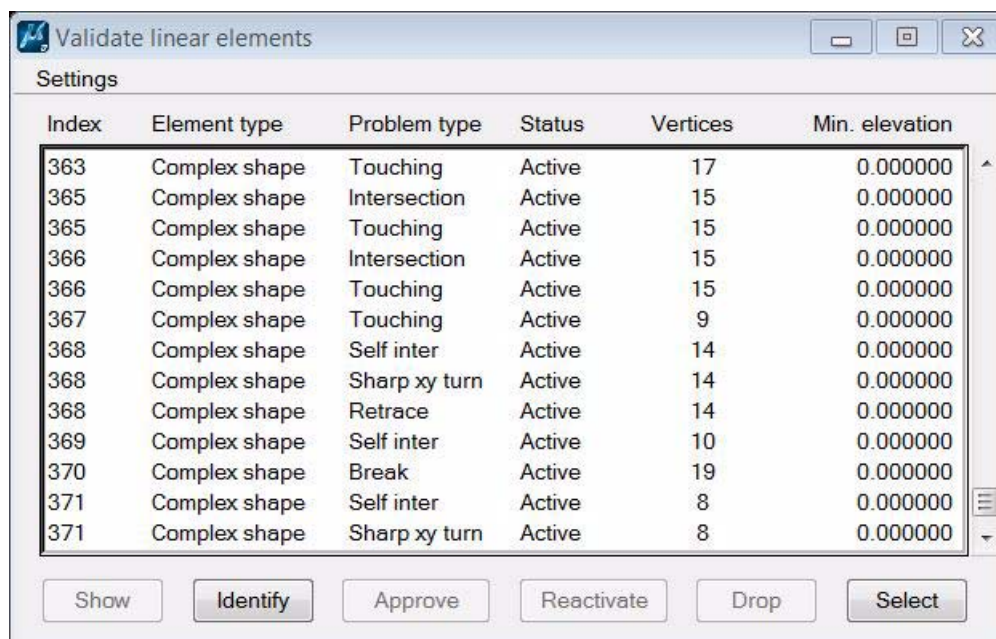
4. If applicable, define additional settings.
5. Click OK.

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 <b>Angle</b> . Values can range from 0 to 180 degree.
Exclude crossings	

### Validate linear elements

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. MicroStation 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** - MicroStation 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**



## Buttons

**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 MicroStation 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 MicroStation 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.

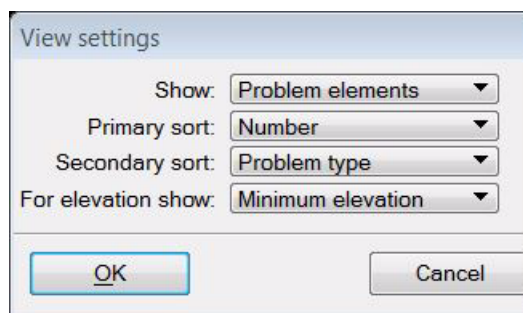
**Drop** - drops a complex element into its single elements. Dropping a complex element is necessary to fix a gap (break). The button is only available if an element with problem type **Break** and status **Active** is selected. The status of the dropped element is changed to **Dropped**.

**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.

## Settings pulldown menu

**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.

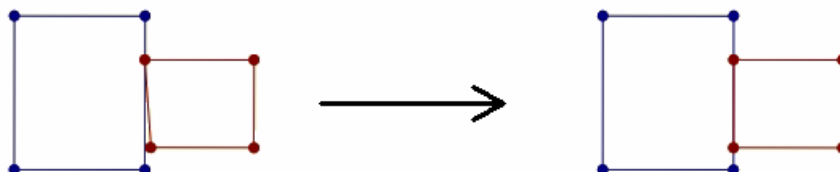


Setting:	Effect:
Show	Determines what elements are displayed in the list: <b>All elements</b> or <b>Problem elements</b> .
Primary sort	The elements are sorted primarily according to the given attribute.
Secondary sort	The elements are sorted secondarily according to the given attribute.
For elevation show	Determines what vertex elevation value is shown in the last column of the list: <b>Minimum elevation</b> , <b>Maximum elevation</b> , or <b>Elevation change</b> .

## 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.



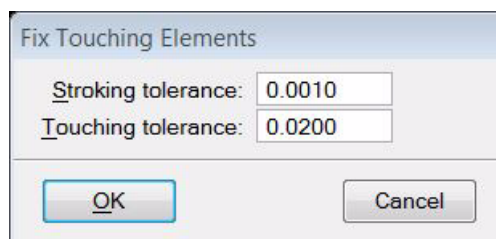
You may run the *Validate Linear Elements* tool with validation method Touching on in order to find out if there are such issues in a set of vector elements.



### To fix touching elements:

1. Select elements using the MicroStation *Selection* tool.
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 MicroStation 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.

## Read Piles From File



*Read Piles From File* tool imports and draws pile locations into the design 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 cylindric cell element in the design 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 **Done**.

This opens the **File format for import** dialog:

File format for import

Delimiter: Space ☐ Non-continuous point number breaks string ☐ Invalid row breaks string

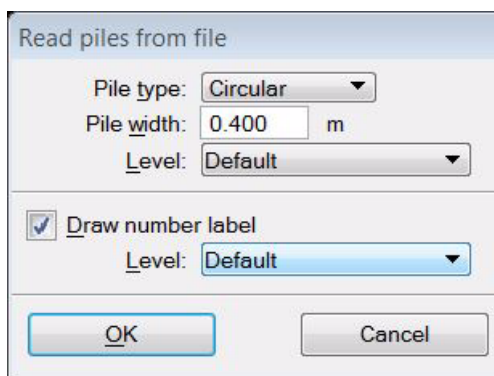
Number	Easting	Northing	Top Z	Length	Ignore	Ignore	Ignore
#ID	X	Y	TopZ	Length			
P10	485938.42	6902520.98	82.12	5.42			
P11	485940.01	6902523.82	82.10	5.49			
P12	485935.58	6902515.71	82.11	5.19			
P13	485937.04	6902518.42	82.11	4.98			
P23	486988.75	6903664.53	81.47	5.86			
P24	486992.21	6903668.18	81.47	5.97			
P25	486989.45	6903663.82	81.48	5.16			

OK Cancel

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 design file. The elements (and labels) are drawn on the given level(s) using the active symbology of MicroStation.







Setting:	Effect:
Pile type	Shape of the pile: <b>Circular</b> or <b>Rectangular</b> .
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	Design file level on which the pile elements are drawn.
Draw number label	If on, the number attribute is drawn on the given <b>Level</b> . This requires a number attribute for each pile in the text file.

## 9 Editing Surface Models

### Edit Point tool box

The tools in the **Edit Point** tool box are used to modify a surface model on point-by-point basis.



To:	Use:
Construct / remove a breakline between points	 <i>Construct Breakline</i>
Insert point along a breakline	 <i>Insert Point Along Breakline</i>
Insert a point to the model	 <i>Insert Point</i>
Move a point in the model	 <i>Move Point</i>
Remove a point from the model	 <i>Remove Point</i>
Exclude/include triangles	 <i>Exclude Triangle</i>

## 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.

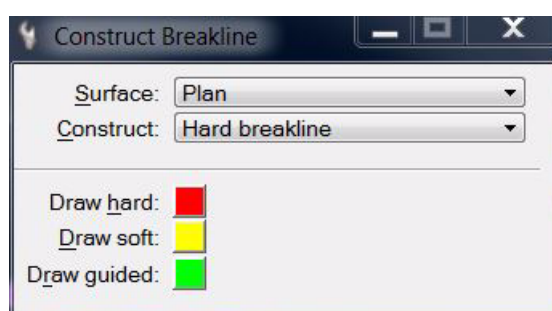
The tool prompts you to identify two points in a surface model. A breakline is constructed, if the two points are not already connected by a breakline of the same type. If the two points are connected by a breakline of the same type, this tool removes the existing breakline.



### To construct or remove breaklines:

1. Select the *Construct Breakline* tool.

The **Construct Breakline** dialog opens:



2. Select a view for breakline display.

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: <b>Hard breakline</b> , <b>Soft breakline</b> , or <b>Guided breakline</b>
Draw hard	Preview color for hard breaklines.
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.

## 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.

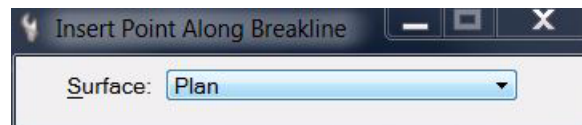
The elevation along the breakline does not change as the elevation of the new point is interpolated from the two end points of the breakline.



### 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. Enter a new point. 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.

## 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.

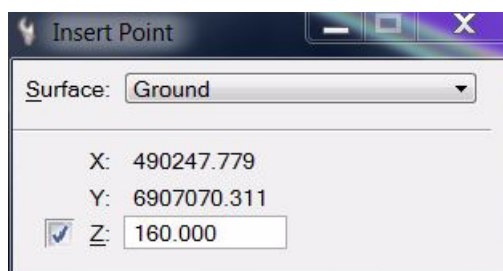
In addition, the tool can add a leveling text for the new point and optional, a point marker. The settings for the label are defined in *Elevation labels category* and *Insert Point category* of the TerraModeler **Settings**. The symbology of the label is further determined by the active symbology settings in MicroStation.



### 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 **Z** on and enter an elevation value in order to use a key-in elevation for new points.

OR

Switch **Z** off in order to use the data point elevation for new points.

4. Move the mouse pointer inside a view.

The XY coordinates 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.



## 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.

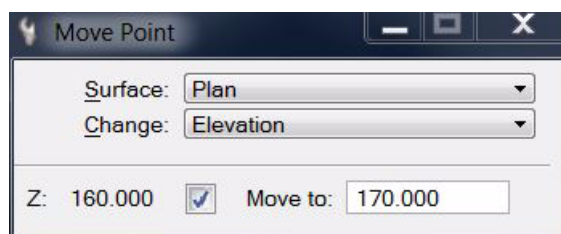
The tool opens a dialog which shows dynamically the current coordinate values of the point to be moved and the new coordinate values close to the mouse pointer location to which the point is moved.



### 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: <ul style="list-style-type: none"> <li>• <b>Elevation</b> - the Z coordinate of the point is modified.</li> <li>• <b>Xy location</b> - the XY coordinates of the point are modified.</li> <li>• <b>Xyz location</b> - the XYZ coordinates of the point are modified.</li> </ul>
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 <b>Change</b> is set to <b>Elevation</b> .

## Remove Point



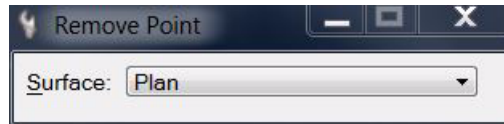
*Remove Point* tool deletes a point from a surface model. It automatically reorganizes the triangulation at the modified location.



**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.

## 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.

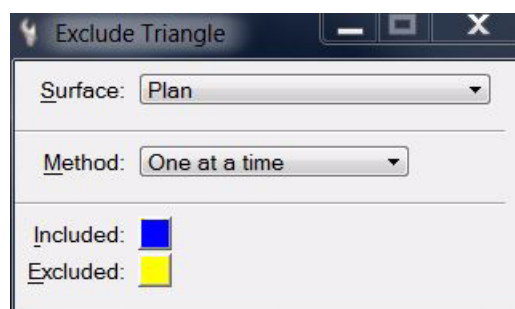
There are several methods of changing the status of a triangle with the tool:

- **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 mouse 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 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.

## Edit Area tool box

The tools in the **Edit Area** tool box are used to modify surface model points or triangles inside or outside an area. The area is defined by a MicroStation fence or a shape element. Valid element types include shapes, complex shapes, and ellipses.



To:	Use:
Flatten points to a fixed elevation	 <i>Flatten Area</i>
Raise / lower points by a vertical distance	 <i>Move Area</i>
Drop / lift points to follow another surface	 <i>Drop Area</i>
Exclude triangles	 <i>Exclude Area</i>
Insert elements as internal hole boundaries	 <i>Insert Hole Element</i>
Remove points from the model	 <i>Remove Area</i>

 These tools consider 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.

## 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.

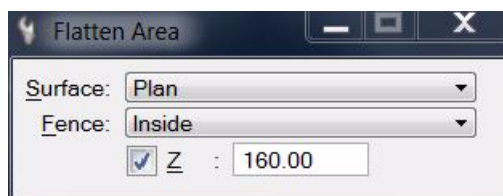
You can enter the new elevation as a key-in value or as a data point.

The fence can be defined by a MicroStation fence or by a closed 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 **Flatten Area** 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: <ul style="list-style-type: none"> <li>• <b>Inside</b> - points inside the fence are modified in elevation.</li> <li>• <b>Outside</b> - points outside the fence are modified in elevation.</li> </ul>
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. Set the active depth of a top view to the desired elevation.
2. Place a shape element at the desired elevation to define the area.
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.

## 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.

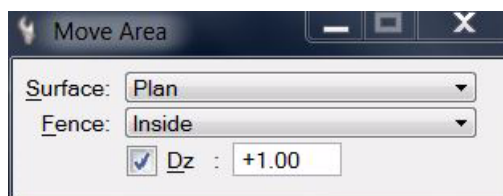
The fence can be defined by a MicroStation fence or by a closed shape element. Valid element types include shapes, complex shapes, and ellipses.



### 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 **Move Area** 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: <ul style="list-style-type: none"> <li>• <b>Inside</b> - points inside the fence are modified in elevation.</li> <li>• <b>Outside</b> - points outside the fence are modified in elevation.</li> </ul>
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 moved by the distance defined with the last data click.

## 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:

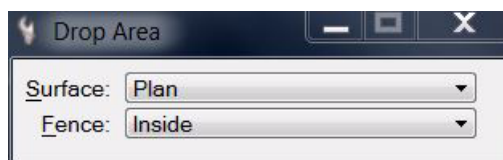
- **Down** - a point in the active surface is modified if one of the selected surfaces is located below that point's elevation. The elevation of the point is set to the highest elevation of the surfaces below the point.
- **Up** - a point in the active surface is modified if one of the selected surfaces 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 MicroStation fence or by a closed shape element. Valid element types include shapes, complex shapes, and ellipses.

### ➤ To drop area points:

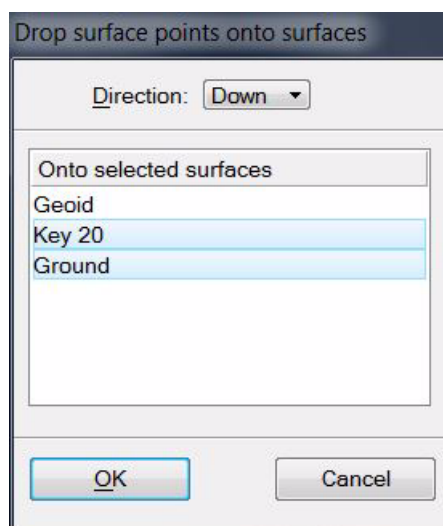
1. Place a fence or a shape element to define the area. (Optional) Select the shape.
2. Select the *Drop Area* tool.

The **Drop Area** 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:



6. Select a **Direction** and surface(s) to which points are dropped.
7. 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.

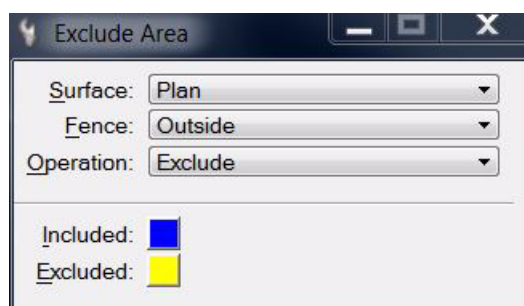
The fence can be defined by a MicroStation fence or by a closed shape element. Valid element types include shapes, complex shapes, and ellipses.



### To include/exclude triangles inside an area:

1. Place a fence or a shape element to define the area. (Optional) Select the shape.
2. Select the *Exclude Area* tool.

The **Exclude Area** 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: <ul style="list-style-type: none"> <li>• <b>Inside</b> - status of triangles inside the fence is changed.</li> <li>• <b>Outside</b> - status of triangles outside the fence is changed.</li> </ul>
Operation	Status to which the triangles are changed: <b>Exclude</b> or <b>Include</b> .
Included	Highlight color for included triangles.
Excluded	Highlight color for excluded triangles.



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.

## 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.

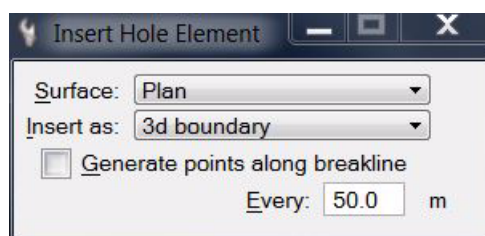
The boundary element has to be a closed linear element. Valid element types include shapes, complex shapes, and ellipses.

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.
4. Accept the element with a 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).

TerraModeler inserts the element(s) to the surface and creates void area(s) inside.

Setting:	Effect:
Surface	Name of the effected surface model.
Insert as	Set the boundary type: <ul style="list-style-type: none"> <li>• <b>2d boundary</b> - use only the two dimensional shape of the element.</li> <li>• <b>3d boundary</b> - insert vertices using the elevation of the element.</li> </ul>
Generate points along breakline	If on, additional intermediate vertices are added to the boundary element at the given <b>Every</b> distance. This is only active if <b>Insert as</b> is set to <b>3d boundary</b> .



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.

## 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 MicroStation fence or by a closed 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 **Remove Area** 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.

## 10 Generating Displays

### Display Surface tool box











The tools in the **Display Surface** tool box 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 design file or as temporary elements which are not saved with the design 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:
Display contours	 <i>Display Contours</i>
Display colored triangles	 <i>Display Triangles</i>
Display colored grid	 <i>Display Grid</i>
Display elevation as grid spaced texts	 <i>Display Elevation Texts</i>
Display grid spaced slope arrows	 <i>Display Slopes</i>
Display coloring by elevation and sun angle	 <i>Display Shaded Surface</i>
Update displays after surface modification	 <i>Update Displays</i>
Erase contours, triangles or grid	 <i>Erase Display</i>
Display surface model boundary	 <i>Display Boundary</i>
Label local minimum and maximum points	 <i>Label Peaks and Pits</i>

## Display modes

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

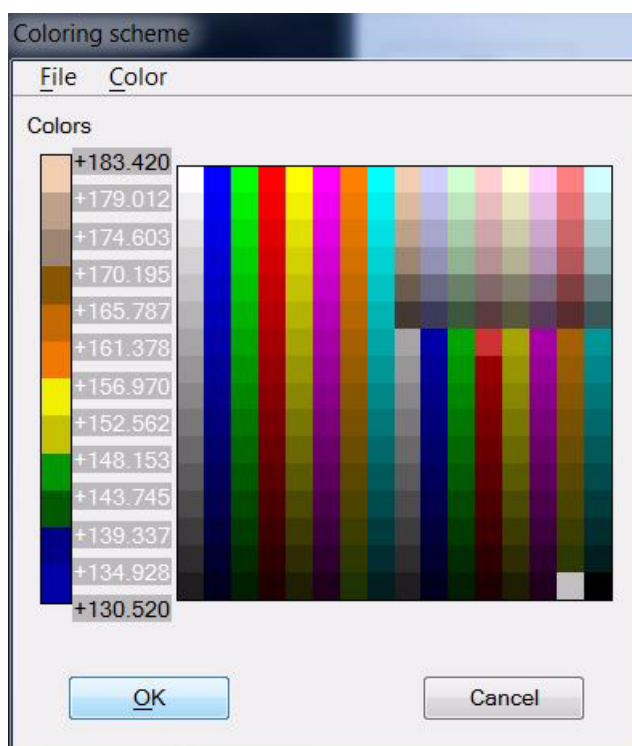
**Write to file** mode adds the created elements to the design file. This has the disadvantage of increasing the design file size quite substantially with some display methods. Very large surface models may result in design files which are very cumbersome for processing with MicroStation tools or for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

**Display only** mode computes the display elements and stores them in the RAM. Elements are drawn when a view is updated. This mode is supported only by tools which may require longer calculations to create the display. These include *Display Contours* and *Display Grid* tools.

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

## Creating a coloring 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 MicroStation.



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 a dialog where you can enter an 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 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 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** on page 216 for more information.

Contours can be drawn as:

- **Curves** - smooth elements for which MicroStation 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 larger 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 design file.

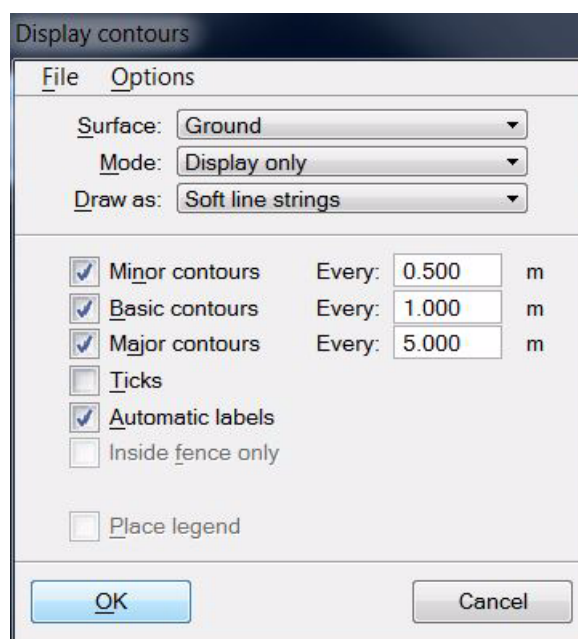
In addition to the contour lines, the software can create labels and ticks for the contours. Further, depressions can be displayed using different symbology settings than for hills. Labels, ticks, and depression line symbology are available in display modes **Write to file** and **Display only**, but 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.

The contours for the selected surface model are displayed.

Setting:	Effect:
Surface	Name of the effected surface model.
Mode	Display mode for contour lines: <ul style="list-style-type: none"> <li>• <b>Display only</b> - elements are stored in RAM and redrawn whenever a view is updated.</li> <li>• <b>Write to file</b> - elements are written and stored in the design file.</li> <li>• <b>Preview</b> - elements are recalculated and redrawn whenever the view is updated.</li> </ul>
Draw as	Element type of the contour lines: <ul style="list-style-type: none"> <li>• <b>Curves</b> - smooth elements for which MicroStation computes the curvature, may overlap each other, compact elements.</li> <li>• <b>Line strings</b> - sharp elements, suitable for preview.</li> <li>• <b>Soft line strings</b> - smooth elements for which TerraModeler computes smoothening, best quality, large elements.</li> </ul>
Minor contours	If on, minor contours are drawn at <b>Every</b> distance.
Basic contours	If on, basic contours are drawn at <b>Every</b> distance.
Major contours	If on, major contours are drawn at <b>Every</b> distance.
Ticks	If on, ticks are drawn along contour lines. Not available if <b>Mode</b> is set to <b>Preview</b> .
Automatic labels	If on, labels are drawn along contour lines. Not available if <b>Mode</b> is set to <b>Preview</b> .
Inside fence only	If on, contours are displayed only inside a MicroStation fence. This requires a MicroStation fence drawn in the design file. Not available if <b>Mode</b> is set to <b>Preview</b> .
Write attributes	If on, the software writes the contour elements in a way that enables the update of the contours using the <i>Update Displays</i> tool and the removal of contours using the <i>Erase Display</i> tool. This is only active if <b>Inside fence only</b> is switched on.
Place legend	If on, a legend for contour line symbology is placed. This is only active if <b>Color by</b> is set to <b>Elevation</b> in the <b>Contour options</b> dialog.



## 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.

Contour option:	Effect:
Color by	Contour line coloring method: <ul style="list-style-type: none"> <li>• <b>Contour type</b> - different colors according to the type of the contour line.</li> <li>• <b>Elevation</b> - a coloring scheme is used for coloring contour lines according to their elevation value.</li> </ul>
Colors	Color scheme for coloring contours by their elevation value. The color scheme is defined using the <b>Define</b> button. See <a href="#">Creating a coloring scheme of discrete colors</a> for information about how to create a coloring scheme. Only active if <b>Color by</b> is set to <b>Elevation</b> .
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.

Contour option:	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 <b>Draw contours</b> option is switched on. See <i>Define Domains</i> 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 <b>Minimum area</b> . Separate settings for <b>Hills</b> and <b>Depressions</b> . 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 design 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 <b>Thin by removing points if</b> 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 <b>Thin by removing points if</b> 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 <b>Thin by removing points if</b> 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.

Contour label option:	Effect:
Label	Defines the contour line type(s) for labeling: <b>All contours</b> , <b>Major and basic contours</b> , or <b>Major contours only</b> .
Position	Location where label text elements are placed along a contour line: <ul style="list-style-type: none"> <li>• <b>Above contour</b> - above the contour line at specified intervals.</li> <li>• <b>On contour</b> - overlapping the contour line at specified intervals.</li> <li>• <b>Below contour</b> - below the contour line at specified intervals.</li> <li>• <b>At contour start</b> - at the start point of a contour chain.</li> </ul>
Rotation	Rotation of the label text element: <ul style="list-style-type: none"> <li>• <b>Uphill</b> - label is readable when looking uphill.</li> <li>• <b>Downhill</b> - label is readable when looking downhill.</li> <li>• <b>North up</b> - label is readable when looking in north direction.</li> </ul>
Every	Distance between consecutive contour labels. Given in meters along the contour.

Contour label option:	Effect:
Cut underlying contour	If on, the contour line is cut under the label text element. The line under a label text is drawn as a MicroStation 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: <ul style="list-style-type: none"> <li>• <b>Contour level and color</b> - a label is drawn using level and color of its contour line type (major, basic, or minor).</li> <li>• <b>Own settings</b> - all labels are drawn using the same level and color assigned to labels.</li> </ul>
Level	Level and color of contour labels. This is only active if <b>Use</b> is set to <b>Own settings</b> .
Style	Line style of contour labels.
Weight	Line weight of contour labels.
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.

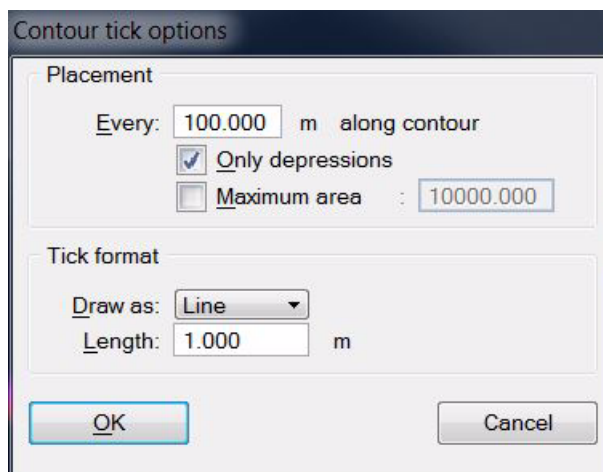
## Contour tick options

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.

Contour tick option:	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: <b>Line</b> or <b>Triangle</b> .
Length	Length of the tick line or triangle.

## Display Triangles

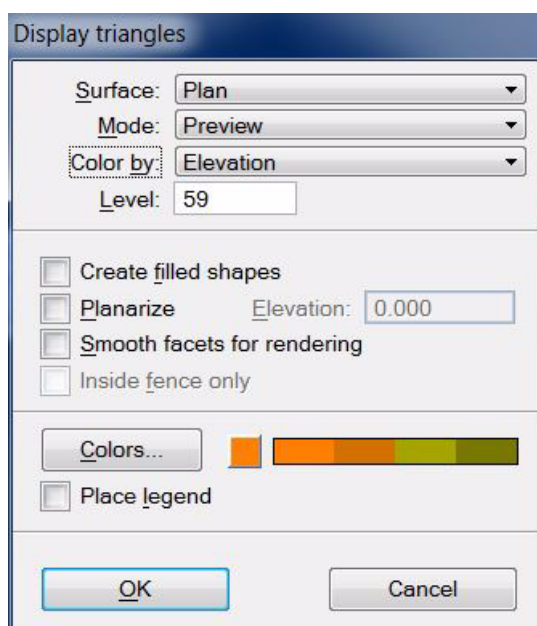


*Display Triangles* tool draws colored triangles for a surface model. You can create a coloring scheme where the color changes according to the surface elevation or according to the slope gradient of the triangles.

### ➤ 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: <ul style="list-style-type: none"> <li>• <b>Write to file</b> - elements are written and stored in the design file.</li> <li>• <b>Preview</b> - elements are recalculated and redrawn whenever the view is updated.</li> </ul>
Color by	Attribute used for coloring: <ul style="list-style-type: none"> <li>• <b>Elevation</b> - elevation value of the triangle determines its color.</li> <li>• <b>Slope</b> - slope gradient of the triangle determines its color.</li> </ul>
Level	Number of the level in the design 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 <b>Elevation</b> .
Smooth facets for rendering	If on, triangles are displayed with a smoother change in coloring if MicroStation rendering is applied ( <i>Not MicroStation V8i</i> ).

Setting:	Effect:
Inside fence only	If on, triangles are drawn if their center point is located inside a fence. This is only active if <b>Mode</b> is set to <b>Write to file</b> and if a MicroStation fence has been drawn before starting the tool.
Colors	Opens the <b>Color scheme</b> dialog. See <b>Creating a coloring scheme of discrete colors</b> for more information.
Color button	Opens the MicroStation color table for single color selection.
Place legend	If on, a legend for the triangle colors can be placed in the design file. This is only active if a color scheme is defined.



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

## Display Grid



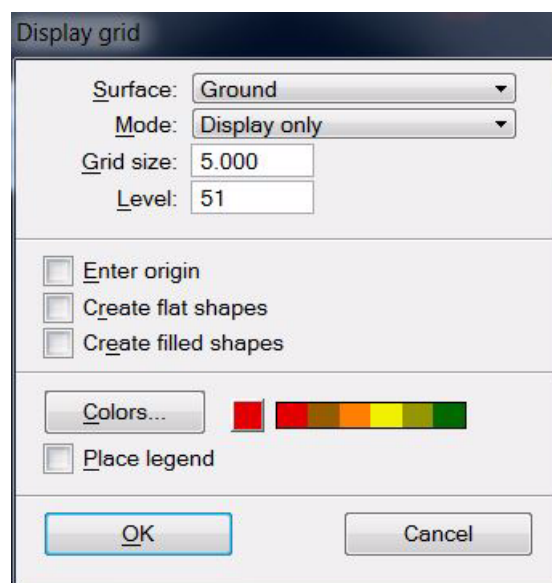
*Display Grid* tool generates a grid presentation of a surface model. You can create a coloring scheme where the color of the grid squares changes according to the surface elevation.



### 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 point.  
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 point.

Setting:	Effect:
Surface	Name of the effected surface model.
Mode	Display mode for the grid: <ul style="list-style-type: none"> <li>• <b>Display only</b> - elements are stored in RAM and redrawn whenever a view is updated.</li> <li>• <b>Write to file</b> - elements are written and stored in the design file.</li> </ul>
Grid size	Size of rectangular squares in the grid.
Level	Number of the level in the design 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 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 <b>Color scheme</b> dialog. See <b>Creating a coloring scheme of discrete colors</b> for more information.
Color button	Opens the MicroStation color table for single color selection.
Place legend	If on, a legend for the triangle colors can be placed in the design file. This is only active if a color scheme is defined.



## Display Elevation Texts



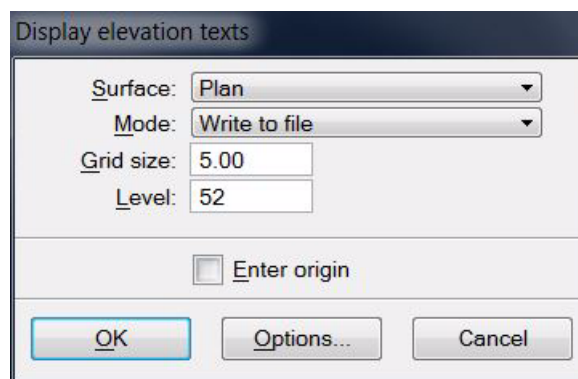
*Display Elevation Texts* tool draws a grid of labels which display the surface elevation. The exact location of the elevation can be marked by a point marker or by the decimal point of the elevation text.



### 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 point.

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: <ul style="list-style-type: none"> <li>• <b>Write to file</b> - elements are written and stored in the design file.</li> </ul>
Grid size	Distance between elevation text locations.
Level	Number of the level in the design 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.
Options	Opens the <b>Elevation text settings</b> dialog. See <b>Elevation text settings</b> on page 114 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): <ul style="list-style-type: none"> <li>• <b>Left, Center, Right   Top, Middle, Bottom</b> - placement point location relative to the label text box.</li> <li>• <b>Decimal point</b> - placement point is located on the decimal point.</li> </ul>
Offset dx	Offset between the placement point and the label in x (left-right) direction. Given in millimeters on paper. Not active if <b>Justify</b> is set to <b>Decimal point</b> .
Offset dy	Offset between the placement point and the label in y (up-down) direction. Given in millimeters on paper. Not active if <b>Justify</b> is set to <b>Decimal point</b> .
Symbology	Color and line weight of the elevation text. Uses the MicroStation color table and line weights.
Font	Font type of elevation texts. Uses font types available in MicroStation.
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: <ul style="list-style-type: none"> <li>• <b>None</b> - no point marker is drawn.</li> <li>• <b>Character</b> - a given character is drawn as point marker.</li> <li>• <b>Zero length line</b> - a line element of zero length (= point) is drawn as point marker.</li> </ul>
Symbology	Color and line weight of the point marker. Uses the MicroStation color table and line weights. This is only active if <b>Point marker</b> is set to <b>Character</b> or <b>Zero length line</b> .

Setting:	Effect:
Font	Font type of the point marker character. Uses font types available in MicroStation. This is only active if <b>Point marker</b> is set to <b>Character</b> .
Size	Text size of the point marker character. Given in millimeters on paper. This is only active if <b>Point marker</b> is set to <b>Character</b> .
Character	Character used as point marker symbol. This is only active if <b>Point marker</b> is set to <b>Character</b> .
Weight	Weight of the point marker zero length line. Uses MicroStation line weights. This is only active if <b>Point marker</b> is set to <b>Zero length line</b> .

## 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 point.

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: <ul style="list-style-type: none"> <li>• <b>Write to file</b> - elements are written and stored in the design file.</li> </ul>
Grid size	Distance between slope arrow locations.
Level	Number of the level in the design file on which triangles are drawn.
Color	Color of the arrows. Uses the MicroStation color table.
Weight	Line weight of the arrows. Uses MicroStation line weights.
Style	Line style of the arrows. Uses MicroStation 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 <b>Slope settings</b> dialog. See <b>Slope settings</b> on page 117 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.

**Slope settings**

**Arrow symbols**

Place by: **Arrow start**

Arrowheads: **One to three**

Length	Slope
5.000	1000.00 %
3.333	10.00 %
1.667	5.00 %
0.000	0.00 %

**Slope labels**

☐ Write slope label

Level: **53**

Font: **0**

Size: **2.0** mm

Accuracy: **0.1**

Color: **0**

Weight: **—**

**OK** **Cancel**

Setting:	Effect:
Place by	Placement point (= point of elevation measurement) of a slope arrow: <b>Arrow start</b> or <b>Arrow center</b> .
Arrowheads	Number of arrowheads of a slope arrow: <ul style="list-style-type: none"> <li>• <b>One</b> - all arrows have one arrowhead.</li> <li>• <b>One to three</b> - number of arrowheads depends on the slope.</li> </ul>
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: <ul style="list-style-type: none"> <li>- maximum length</li> <li>- two thirds of maximum length</li> <li>- one third of maximum length</li> <li>- minimum length</li> </ul>
Write slope label	If on, a text label showing the slope gradient is placed along a slope arrow.
Level	Level number in the MicroStation design file on which the label is placed.
Font	Font type of slope arrow labels. Uses font types available in MicroStation.
Size	Text size of slope arrow labels. Given in millimeters on paper.
Accuracy	Number of decimals in the slope arrow label.
Color	Color of slope arrow labels. Uses the active color table of MicroStation.
Weight	Line weight of slope arrow labels. Uses MicroStation line weights.

## 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 MicroStation views, it is not written into the design 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.

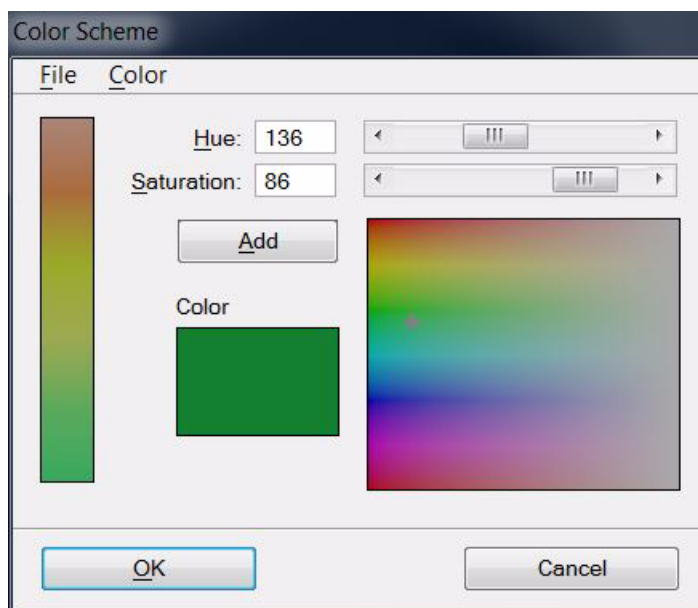
Setting:	Effect:
Surface	Name of the effected surface model.
Sun azimuth	Direction from which the sun illuminates the model. Zero is north and angles increase clockwise.
Sun angle	Sun angle above the horizon.
Color scheme	Color scheme to use: <ul style="list-style-type: none"> <li>• <b>Default</b> - default color scheme containing magenta, red, yellow, green, cyan, and blue.</li> <li>• <b>Selected colors</b> - use the <b>Define</b> button to create a customized color scheme. See <b>Define a customized color scheme</b> on page 119 for more information.</li> </ul>
Color cycles	Number of color cycles. Use zero to create a gray scale display showing triangle slope only.
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 hues of the color cycle from top to bottom and color saturations 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.

## 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.

## Erase Display

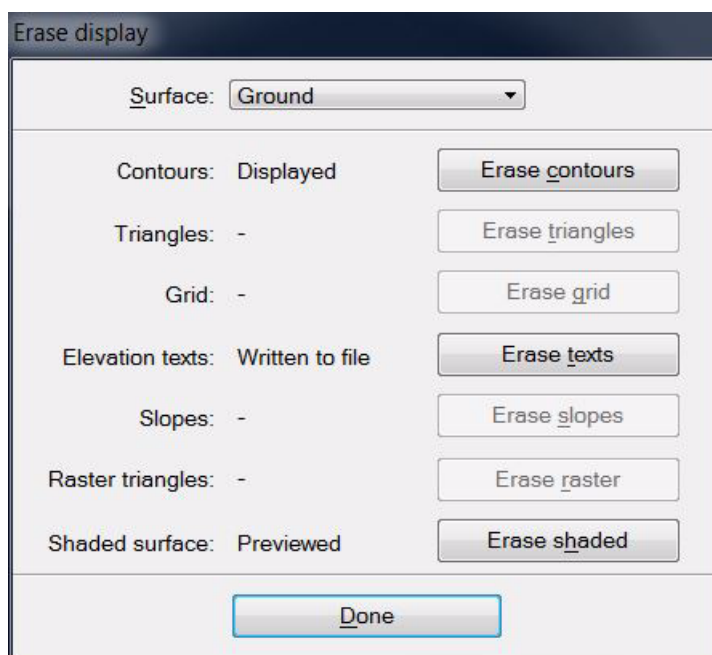


*Erase Display* tool removes previously displayed contours, triangles, a grid, elevation texts, slope arrows, raster triangles, or shaded surfaces.

The tool opens a window which shows the status of the display methods for the selected surface. In the example below, for the surface named *Ground*, contours are displayed in **Display only** mode, elevation texts are drawn in **Write to file** mode, and a shaded surface is displayed as well. Triangles, a grid, slope arrows, or raster triangles are not displayed.

### ➤ To erase display methods:

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



3. Select the **Surface** for which to remove a display method.
4. Use the **Erase** button next to a display method to erase the respective display.
5. Click **Done** in order to close the **Erase display** dialog.



## Display Boundary

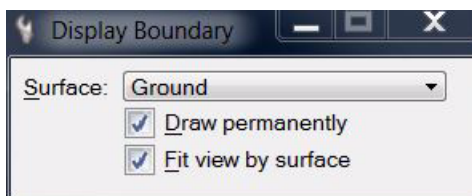


*Display Boundary* tool draws a boundary shape of a surface model. This can be used to outline the area covered by the surface model.



### 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 design 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 design 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.

## 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 design file. The label contains a text showing the elevation value and a point marker placed at the location of the elevation point.

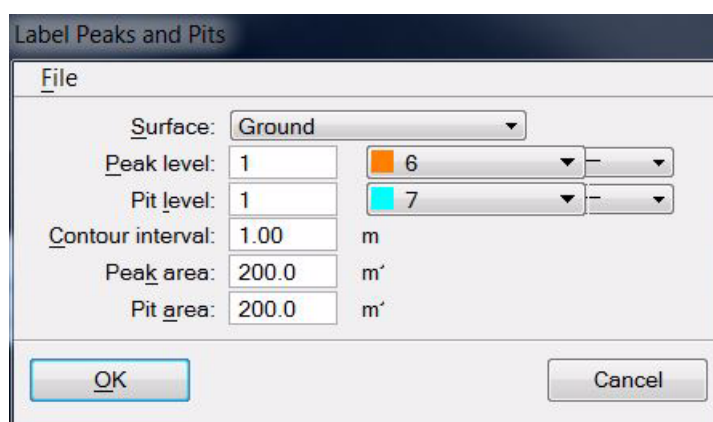
The layout of the label is determined by the settings in *Elevation labels category* of the TerraModeler **Settings**.

From the dialog of the tool, a peaks and pits settings file can be saved for being used in automatic contour production. See **Utility / Produce contours** on page 216 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 design 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.

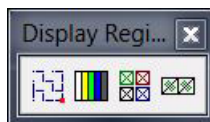
The software searches for peaks and pits in the surface model and writes the labels into the design file.

Setting:	Effect:
Surface	Name of the effected surface model.
Peak level	Level, color, and line weight of labels for peaks. Uses the active MicroStation color table and MicroStation line weights.
Pit level	Level, color, and line weight of labels for pits. Uses the active MicroStation color table and MicroStation 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.





## Display Regions tool box

*Not Lite*

The tools in the **Display Regions** tool box 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:
Display rain water flow on surface	 <i>Display Drainage</i>
Display theme polygons colored by elevation, slope, or domain	 <i>Display Themes</i>
Display triangles colored by region boundaries	 <i>Display Region Triangles</i>
Display triangles with raster materials	 <i>Display Raster Triangles</i>

## 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 neighbouring cells. If a lower elevation is found, the rain drop travels to the neighbouring 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.

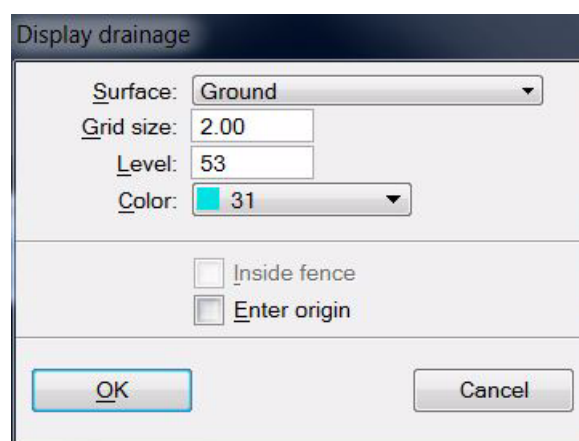
*Display Drainage* tool creates design file elements which are not linked to a surface model. Thus, TerraModeler is not able to update or erase elements created by this tool.



### 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.

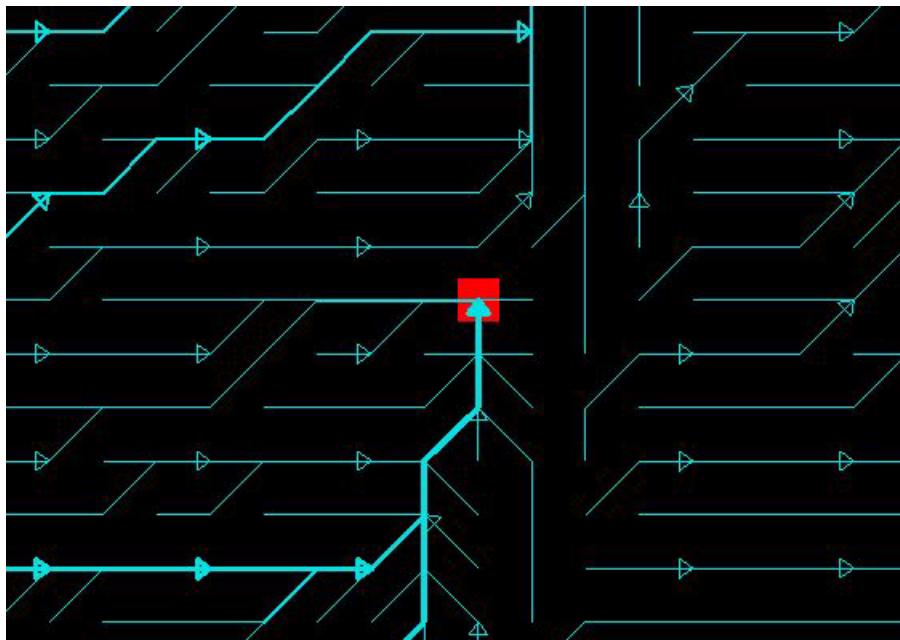
This computes the drainage water flow and displays the results temporarily in all open views. The display is updated every time when MicroStation updates the view contents.

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

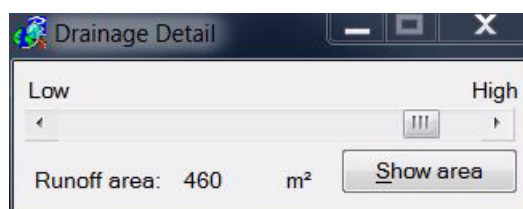
Setting:	Effect:
Surface	Name of the effected surface model.
Grid size	Size of the grid cells for drainage computation.
Level	Number of the level in the design file on which the drainage network is drawn if it is drawn permanently into the design file.
Color	Color of the lines and arrows the represent the water flow. Uses the MicroStation color table.
Inside fence	If on, the drainage system is only displayed inside a fence. This is only active if a MicroStation 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.

### ➤ 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 design file using the active symbology.

When you close the **Drainage Detail** dialog, the software asks you if you want to draw the drainage display permanently. Confirm with **Yes** in order to draw the drainage network into the design file or with **No** in order to remove the temporary display of the drainage network.

## Display Themes

Not Lite



*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 design file size compared with the triangle visualization approach which is advantageous especially in dynamic views.

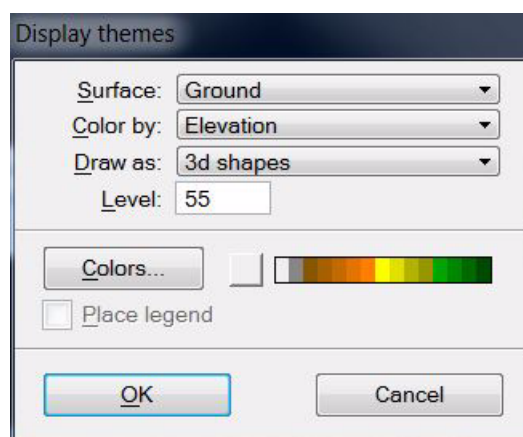
*Display Themes* tool creates design file elements which are not linked to a surface model. Thus, TerraModeler is not able to update or erase elements created by this tool.



### 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 design 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: <ul style="list-style-type: none"> <li>• <b>Elevation</b> - each theme area represents an elevation range.</li> <li>• <b>Slope</b> - each theme area represent a slope range.</li> <li>• <b>Domain</b> - creates theme polygons from neighbouring triangles assigned to the same domain. See <i>Define Domains</i> tool for more information.</li> </ul>
Draw as	Defines the shape type of theme areas: <ul style="list-style-type: none"> <li>• <b>3d shapes</b> - the shapes follow the surface model elevation.</li> <li>• <b>2d shapes</b> - the shapes are plane and drawn at zero elevation.</li> </ul>
Level	Number of the level in the design file on which the theme area shapes are drawn.
Colors	Opens the <b>Color scheme</b> dialog. See <i>Creating a coloring scheme of discrete colors</i> for more information.
Color button	Opens the MicroStation color table for single color selection.

Setting:	Effect:
Place legend	If on, a legend for the theme colors can be placed in the design file. This is only active if a color scheme is defined.

## Display Region Triangles

Not Lite



*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.

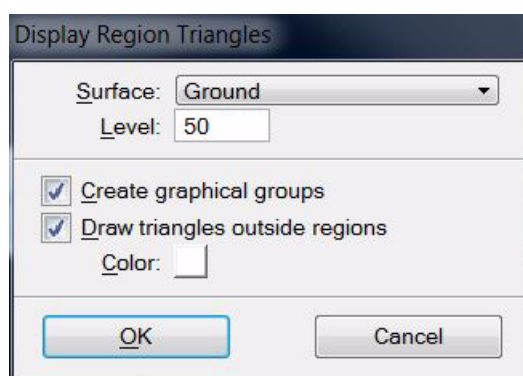
*Display Region Triangles* tool creates design file elements which are not linked to a surface model. Thus, TerraModeler is not able to update or erase elements created by this tool.



### To display region triangles:

1. Use MicroStation *Selection* tool to 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 design file.

Setting:	Effect:
Surface	Name of the effected surface model.
Level	Number of the level in the design file on which region triangles are drawn.
Create graphical groups	If on, the triangles inside each region form one graphical group.
Draw triangles outside regions	If on, triangles which are outside all selected boundary shapes are drawn using the given <b>Color</b> .



## 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.

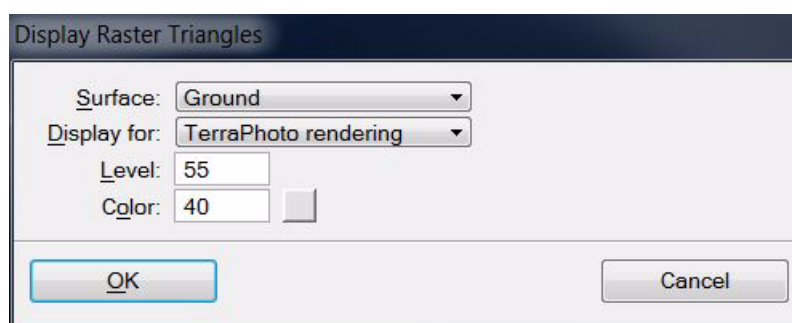
The raster triangles are not drawn permanently into the design file and they remain linked to the surface model. This allows TerraModeler to update the display after modifications and to erase the display using the *Erase Display* tool.



### 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.
Display for	Rendering method that uses the raster triangles: <ul style="list-style-type: none"> <li>• <b>TerraPhoto rendering</b> - raster files attached in TerraPhoto are draped on the triangles if a view is rendered with TerraPhoto rendering tools.</li> </ul>
Level	Number of the level in the design file on which the raster triangles are drawn.
Color	Color of the raster triangles. Uses the active MicroStation color table.











## 11 Drawing Profiles

# Profiles tool box

The tools in the **Profiles** tool box 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 design 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:
Draw cross section view	 <i>Draw Section View</i>
Draw profile along an alignment	 <i>Draw Profile</i>
Draw cross sections along an alignment	 <i>Draw Alignment Sections</i>
Update profile or cross section	 <i>Update Profile</i>
Setup views to automatically display surface sections	 <i>Setup Automatic Sections</i>
Project 3d elements into a profile	 <i>Project Into Profile</i>
Project elements from a profile to 3d	 <i>Project From Profile</i>
Label profile with an element's elevations	 <i>Label Profile Elevations</i>
Output surface elevations along an alignment	 <i>Output Profile Elevations</i>
Output surface elevations for cross sections	 <i>Output Section Elevations</i>

## 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.

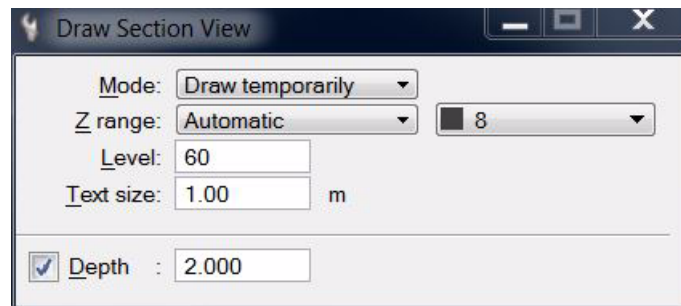
The tool draws a line for each surface at the section's location if the surface type is selected to be drawn in profiles. The symbology of the line is related to the surface type and determined by settings in **Surface types category** in TerraModeler **Settings**. Optionally, the elevation range can be drawn including grid lines and labels.



### To draw a section view:

1. Select the *Draw Section View* tool.

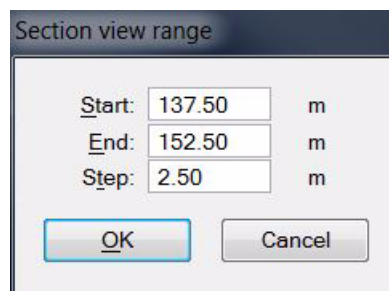
This opens the **Draw Section View** dialog:



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:



7. 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: <ul style="list-style-type: none"> <li>• <b>Draw temporarily</b> - (<i>not MicroStation V8i</i>) the display is erased on view update.</li> <li>• <b>Write to file</b> - drawn as permanent elements.</li> </ul>
Z range	Defines how the elevation range is set: <ul style="list-style-type: none"> <li>• <b>None</b> - the elevation range is not drawn.</li> <li>• <b>Automatic</b> - TerraModeler decides the elevation range based on surface elevations.</li> <li>• <b>Prompt for</b> - the elevation range can be defined in <b>Section view range</b> dialog.</li> </ul>
Level	Design 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.

## 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 design 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 design 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.

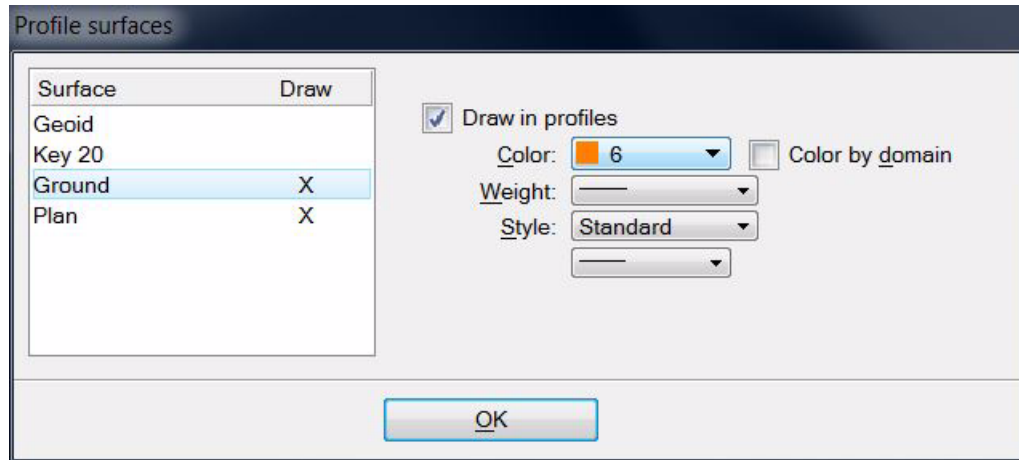
The profile outline is displayed at the mouse pointer position.

8. Define the location of the profile cell in a design file top view with a data click.  
This draws the profile into the design 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: <ul style="list-style-type: none"> <li>• <b>No layout</b> - no data row is drawn below the profile.</li> <li>• <b>“layout name”</b> - uses the selected layout scheme which has been defined in <b>Profiles / Layouts category</b> of TerraModeler <b>Settings</b>.</li> </ul>
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: <b>Along axis</b> or <b>Upper corner</b> .
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: <ul style="list-style-type: none"> <li>• <b>Increasing</b> - station values are increasing from start to end station.</li> <li>• <b>Decreasing</b> - station values are decreasing from start to end station.</li> </ul>
Draw only partial alignment	If on, the profile is only drawn for a part of the alignment defined by <b>From station</b> and <b>To station</b> .

## 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.

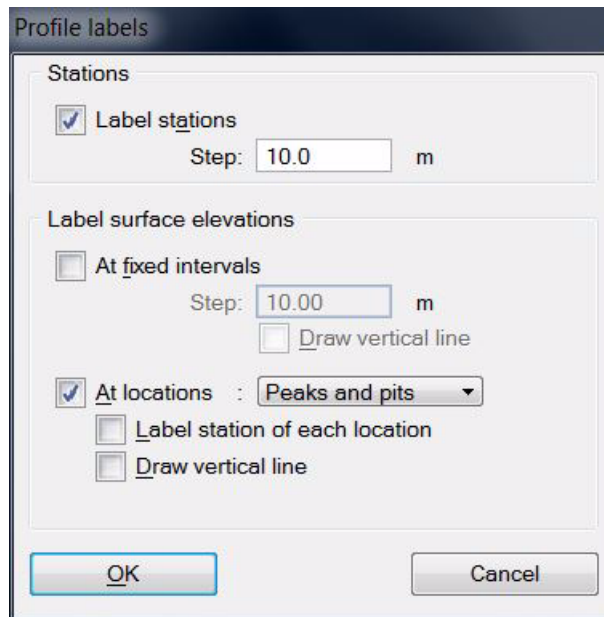


Setting:	Effect:
Draw in profiles	If on, the selected surface in the list on the left side of the dialog is drawn in the profile.
Color	Line color for the selected surface. Uses the active MicroStation color table.
Color by domain	If on, the color of domain(s) is used. See <i>Define Domains</i> tool for more information.
Weight	Line weight for the selected surface. Uses MicroStation line weights.
Style	Line style for the selected surface: <ul style="list-style-type: none"> <li><b>Standard</b> - uses MicroStation standard line styles.</li> <li><b>Custom</b> - the user can define an own line style based on MicroStation line styles and a scale.</li> </ul>
Style name	Name of a custom line style. Use the <b>Select</b> button to select a MicroStation line style definition. This is only active if <b>Style</b> is set to <b>Custom</b> .
Style scale	Scale of a custom line style. This is only active if <b>Style</b> is set to <b>Custom</b> .

 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 <b>Steps</b> .
Draw vertical line	If on, a vertical line is drawn at the location of an elevation label. This is only active if <b>At fixed intervals</b> is switched on.
At locations	If on, elevation labels are drawn at locations where there is a change in the surface model: <ul style="list-style-type: none"> <li>• <b>All triangle edges</b> - locations where the alignment intersects a triangle edge.</li> <li>• <b>Breakline edges</b> - locations where the alignment intersects a breakline edge.</li> <li>• <b>Slope changes</b> - locations where the slope changes more than the given <b>Change limit</b> value.</li> <li>• <b>Peaks and pits</b> - local maximum and minimum elevations.</li> </ul>
Label station of each location	If on, a station label is placed at the location of an elevation label. This is only active if <b>At locations</b> 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 <b>At locations</b> is switched on.



## Draw Alignment Sections



*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 design 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 design file on the active level or on several levels that are defined in **Profiles / Levels category** of the TerraModeler **Settings**.



### 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. Click OK in the **Draw sections along alignment** dialog.

The cross section outlines are displayed at the mouse pointer position.

8. Define the location of the group of profile cells in a design file top view with a data click.  
This draws the cross sections into the design 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: <ul style="list-style-type: none"> <li>• <b>Increasing</b> - stations values are increasing from start to end station.</li> <li>• <b>Decreasing</b> - station values are decreasing from start to end station</li> </ul>
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.
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: <ul style="list-style-type: none"> <li>• <b>Stays constant</b> - the range stays the same as for the first cross section.</li> <li>• <b>Follows 3d alignment</b> - the range is adjusted to the elevation of the alignment at the cross section location.</li> <li>• <b>Follows surface</b> - the range is adjusted to the elevation of the selected <b>Surface</b> at the cross section location.</li> </ul>
From	Lowest elevation range value for the first cross section. The <b>1. z</b> 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.
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 <b>Step</b> 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 <b>Step</b> field.
At fixed intervals	If on, surface elevations labels at fixed <b>Steps</b> 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 <b>At fixed intervals</b> is switched on.
At locations	If on, elevation labels are drawn at locations where there is a change in the surface model: <ul style="list-style-type: none"> <li>• <b>All triangle edges</b> - locations where the alignment intersects a triangle edge.</li> <li>• <b>Breakline edges</b> - locations where the alignment intersects a breakline edge.</li> <li>• <b>Slope changes</b> - locations where the slope changes more than a given <b>Change limit</b>.</li> <li>• <b>Peaks and pits</b> - local maximum and minimum elevations.</li> </ul>
Label offset of each location	If on, an offset label is placed at the location of an elevation label. This is only active if <b>At locations</b> 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 <b>At locations</b> is switched on.

## Update Profile



*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 design file. Once the alignment element is deleted, the update 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.

## Project Into Profile



*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 design file. Once the alignment element is deleted, the projection of element into a profile is no longer possible.

## Project From Profile



*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.

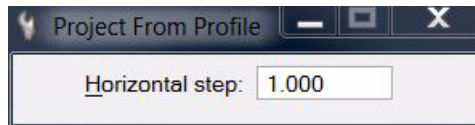
The tool creates a new 3D complex chain running along the profile's alignment. The symbology of the new element is determined by the active symbology in MicroStation.



### 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 design file. Once the alignment element is deleted, the projection of element from a profile is no longer possible.

## Label Profile Elevations



*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.

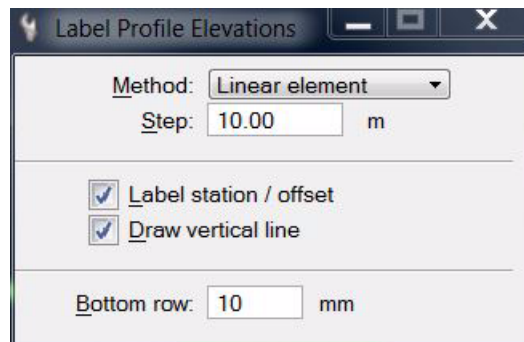
This tool can be used, for example, with an element representing a vertical alignment that has been designed in the profile. The element has to be linear. It should not extend beyond the profile's left or right margin. The tool translates the y-coordinates of the element into elevation values and writes 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 MicroStation.

### ➤ 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 labelled. 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 design file.

You can continue with step 5.

Setting:	Effect:
Method	Labeling method: <ul style="list-style-type: none"> <li>• <b>Linear element</b> - labels elevations of a linear element at fixed Steps.</li> <li>• <b>Single location</b> - labels the elevation of a single data point.</li> </ul>
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 <b>Profiles / Layouts category</b> of the TerraModeler <b>Settings</b> .
Draw vertical line	If on, a vertical line is drawn at each elevation label location.



Setting:	Effect:
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 design 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.

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:
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: <ul style="list-style-type: none"> <li><b>Increasing</b> - station values are increasing from start to end station.</li> <li><b>Decreasing</b> - station values are decreasing from start to end station.</li> </ul>
Output at fixed intervals	If on, output surface elevations at a fixed interval. The <b>Step</b> field value determines the size of the interval.

Setting:	Effect:
At locations	<p>If on, output surface elevations at locations where there is a change in the surface model:</p> <ul style="list-style-type: none"> <li>• <b>All triangle edges</b> - locations where the alignment intersects a triangle edge.</li> <li>• <b>Breakline edges</b> - locations where the alignment intersects a breakline edge.</li> <li>• <b>Slope changes</b> - locations where the slope changes more than a given limit.</li> <li>• <b>Peaks and pits</b> - local maximum and minimum elevations.</li> </ul>

## Profile Output format

The **Profile output format** dialog provides some choices for formatting the output of the *Output Profile Elevations* tool.

Profile output format

**Write starting rows**

☒ Header : [TerraModeler profile]

☒ Surface name

☐ Alignment name

**Write point fields**

☒ Station Delimiter: Space

☐ Easting

☐ Northing

☒ Elevation

☒ Outside : OUTSIDE

**Holes in surface between edges**

☒ Write row : HOLE

OK Cancel

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.

Setting:	Effect:
End station	End station value defined by the length of the alignment element: <ul style="list-style-type: none"> <li>• <b>Increasing</b> - station values are increasing from start to end station.</li> <li>• <b>Decreasing</b> - station values are decreasing from start to end station.</li> </ul>
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 <b>Step</b> 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: <ul style="list-style-type: none"> <li>• <b>All triangle edges</b> - locations where the alignment intersects a triangle edge.</li> <li>• <b>Breakline edges</b> - locations where the alignment intersects a breakline edge.</li> <li>• <b>Slope changes</b> - locations where the slope changes more than a given limit.</li> <li>• <b>Peaks and pits</b> - local maximum and minimum elevations.</li> </ul>

## Section output format

The **Section output format** dialog provides some choices for formatting the output of the *Output Section Elevations* tool.

Section output format

**Write starting rows**

☒ Header : [TerraModeler sections]

☐ Alignment name

☐ Surface count

☒ Section start : SECTION

☒ Surface start : SURFACE

**Write point fields**

☒ Offset Delimiter: Space

☒ Easting

☒ Northing

☒ Elevation

☒ Outside : INVALID

**Holes in surface between edges**

☒ Write row : OUTSIDE

OK Cancel

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.

# 12 Computing Quantities









## Quantity tool box

The **Quantity** tool box 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 MicroStation fence.

*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:
Compute quantities using grid method	 <i>Compute Quantity</i>
Compute quantities using prismoidal method	 <i>Compute Prismoidal Quantity</i>
Compute quantities along an alignment	 <i>Compute Alignment Quantity</i>
Define section templates for trenches	 <i>Define Section Templates</i>
Compute trench quantities using section templates	 <i>Compute Section Quantity</i>
Compute tunnel volume from section elements	 <i>Compute Tunnel Volume</i>
Draw intersection of two surfaces	 <i>Draw Surface Intersection</i>
Compute surface area	 <i>Compute Area</i>

## 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.

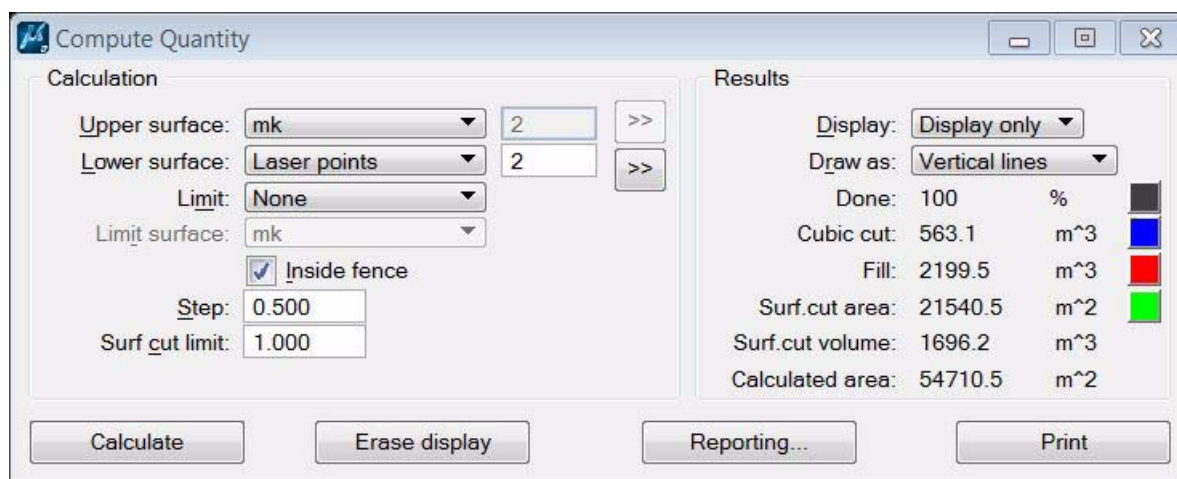
The surfaces for volume calculation can be defined by surface models loaded in TerraModeler or by laser points loaded in TerraScan.

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 **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 **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 design 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 <b>Laser points</b> is selected, the field for entering class number(s) becomes active. Use the >> button in order to open the <b>Select classes</b> dialog for class selection. This requires that respective laser points are loaded in TerraScan.



Setting:	Effect:
Lower surface	Name of the lower surface model for the quantity calculation. If <b>Laser points</b> is selected, the field for entering class number(s) becomes active. Use the >> button in order to open the <b>Select classes</b> dialog for class selection. This requires respective laser points are loaded in TerraScan.
Limit	Limiting surface to restrict the calculation area: <ul style="list-style-type: none"> <li>• <b>None</b> - no limiting surface used.</li> <li>• <b>Area</b> - calculate only within the <b>Limit surface</b> area.</li> <li>• <b>Lower limit</b> - calculate only above <b>Limit surface</b>.</li> <li>• <b>Upper limit</b> - calculate only below <b>Limit surface</b>.</li> </ul>
Limit surface	Surface to limit the calculation area. This is not active if <b>Limit</b> is set to <b>None</b> .
Inside fence	If on, the calculation takes place only inside a fence or a selected shape element. This is only active if a MicroStation fence has been placed or a shape selected.
Step	Defines the grid size for the calculation. A small step produces more accurate results 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 design file: <ul style="list-style-type: none"> <li>• <b>No display</b> - nothing is drawn into the design file.</li> <li>• <b>Display only</b> - draws temporary vertical line elements.</li> <li>• <b>Write to file</b> - draws permanent elements on the active level.</li> </ul>
Draw as	Elements used to present the results graphically: <ul style="list-style-type: none"> <li>• <b>Vertical lines</b> - a vertical line is drawn at the center of each calculated grid cell. This is the only available option if <b>Display</b> is set to <b>Display only</b>.</li> <li>• <b>Horizontal lines</b> - a horizontal lines is drawn for each calculated grid cell row.</li> <li>• <b>Filled shapes</b> - a shape is drawn for each calculated grid cell row.</li> <li>• <b>Pilars</b> - 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.</li> </ul>
Color fields next to calculation result lines	Defines the display color for drawing the results in the design file. Uses the active color table in MicroStation.

## 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.

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.

Setting:	Effect:
Report subtotals by regions	If on, the quantity calculation results are reported for each region separately.
Regions	Method of how regions are defined: <ul style="list-style-type: none"> <li>• <b>Coordinate blocks</b> - regions are defined by squared blocks with an edge length of the given <b>Block size</b> value.</li> <li>• <b>Selected shapes</b> - regions are defined by selected shape elements</li> </ul>
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. This is only active if <b>Regions</b> is set to <b>Selected shapes</b> .
Hilite problems	If on, region shapes without a text element inside are highlighted.
Variables	Determines if a variable text file is written and saved: <ul style="list-style-type: none"> <li>• <b>Do not write</b> - no text file is written.</li> <li>• <b>Write text file</b> - a text file written and saved at the given <b>File</b> location.</li> </ul>
Metadata	The button opens the <b>Quantity metadata</b> 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: <ul style="list-style-type: none"> <li>• <b>Elevation</b> - quantities are reported for defined elevation ranges.</li> <li>• <b>Depth</b> - quantities are reported for defined depth ranges.</li> </ul>
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: <b>Regions as rows</b> or <b>Subtotals in rows</b> .

## 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.

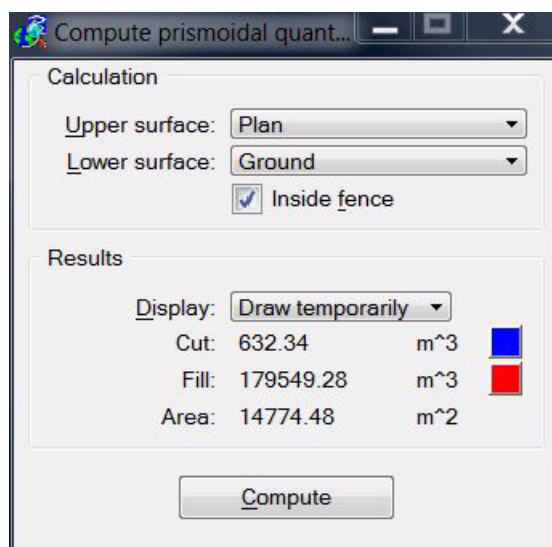
You can use a fence or a selected shape element to limit the area for calculation.



### 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 design 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. This is only active if a MicroStation fence is placed or a shape is selected.
Display	Display method for presenting the calculation results graphically in the design file: <ul style="list-style-type: none"> <li>• <b>No display</b> - nothing is drawn into the design file.</li> <li>• <b>Draw temporarily</b> - (not MicroStation V8i) draws triangles as temporary elements.</li> <li>• <b>Write to file</b> - draws triangles as permanent elements into the design file on the active level.</li> </ul>
Color fields next to calculation result lines	Defines the display color for drawing the results in the design file. Uses the active color table in MicroStation.

## 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 laser points loaded in TerraScan.

The results of the computation are shown in the tool's dialog. They can be presented graphically in the design 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.
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 design file. If **Print to file** was selected in the first dialog, a window opens to define the location for storing the text file on a hard disk.

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: <ul style="list-style-type: none"> <li>• <b>Increasing</b> - station values are increasing from start to end station.</li> <li>• <b>Decreasing</b> - station values are decreasing from start to end station</li> </ul>
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 design file. The scale is divided into sections of the length defined by the <b>Interval</b> value. For each section, the rounded calculated quantity value is displayed.

Setting:	Effect:
Upper surface	Name of the upper surface model for quantity calculation. The use of <b>Laser points</b> 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 <b>Laser points</b> requires that the points which define the surface are loaded into TerraScan.

Setting:	Effect:
Limit	Limiting surface to restrict the calculation area: <ul style="list-style-type: none"> <li>• <b>None</b> - no limiting surface used.</li> <li>• <b>Area</b> - calculate only within the <b>Limit surface</b> area.</li> <li>• <b>Lower limit</b> - calculate only above <b>Limit surface</b>.</li> <li>• <b>Upper limit</b> - calculate only below <b>Limit surface</b>.</li> </ul>
Limit surface	Surface to limit the calculation area. This is not active if <b>Limit</b> is set to <b>None</b> .
Step	Defines the grid size for the calculation. A small step produces more accurate results 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 design file: <ul style="list-style-type: none"> <li>• <b>No display</b> - nothing is drawn into the design file.</li> <li>• <b>Draw temporarily</b> - (not MicroStation V8i) draws temporary elements.</li> <li>• <b>Write to file</b> - draws permanent elements into the design file on the active level.</li> </ul>
Draw as	Elements used to present the results graphically: <ul style="list-style-type: none"> <li>• <b>Vertical lines</b> - a vertical line is drawn at the center of each calculated grid cell.</li> <li>• <b>Horizontal lines</b> - a horizontal lines is drawn for each calculated grid cell row.</li> <li>• <b>Filled shapes</b> - a shape is drawn for each calculated grid cell row.</li> <li>• <b>Pilars</b> - 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.</li> </ul>
Color fields next to calculation result lines	Defines the display color for drawing the results in the design file. Uses the active color table in MicroStation.

✍ 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.



## 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.

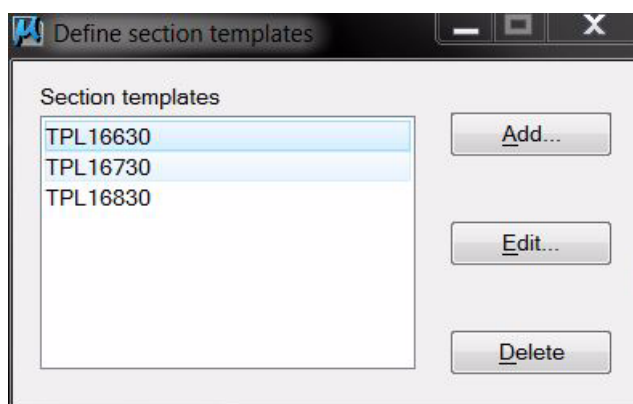
TerraModeler's cross sections templates are meant to be used for computing trench excavation quantities with the *Compute Section Quantity* tool. The cross section templates are automatically saved into a file using the same directory and name as the active design file. The file extension is \*.KDB. For example, if the active design 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.

The **Define section templates** dialog opens:

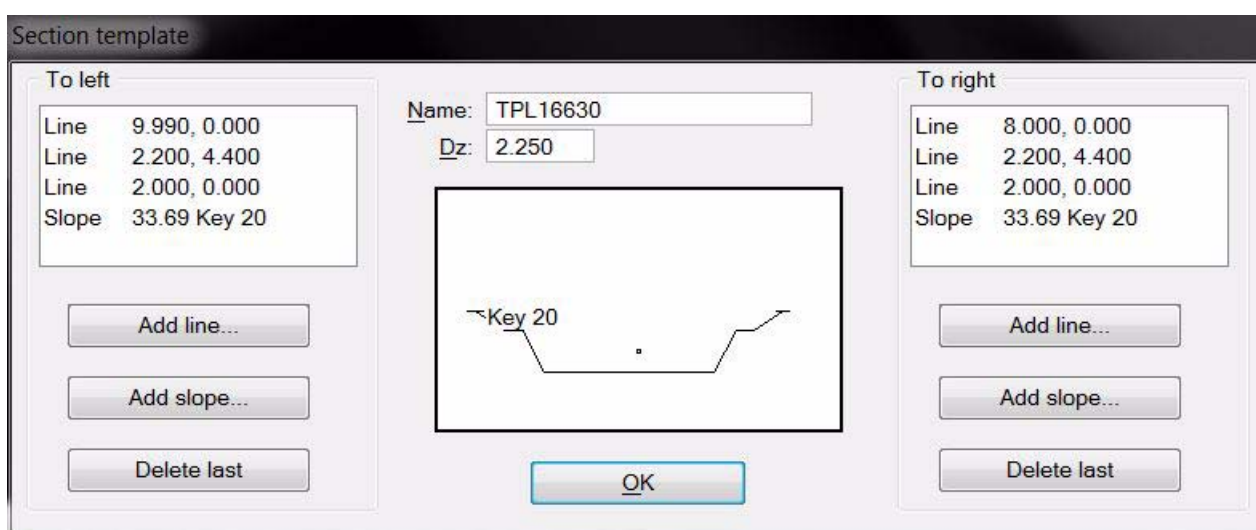


2. Use the **Add** button to add new template definitions to the list.

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.
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.

## 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.

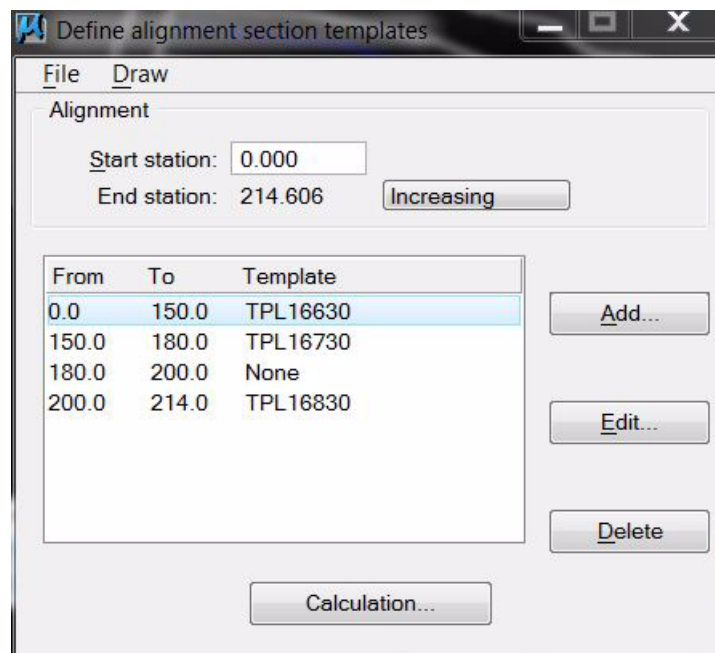
The results of the computation are shown in the tool's dialog. They can be presented graphically in the design file and a text report can be stored as well.



### 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: <ul style="list-style-type: none"> <li><b>Increasing</b> - station values are increasing from start to end station.</li> <li><b>Decreasing</b> - station values are decreasing from start to end station</li> </ul>

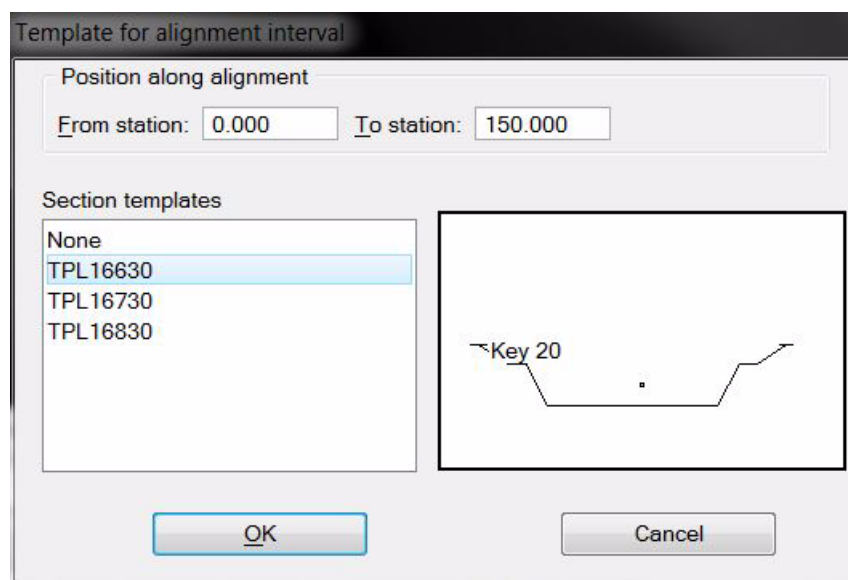
To:	Choose button/menu command:
Add a new station range to the list.	Add
Modify the template for a selected station range.	Edit
Delete a selected station range from the list.	Delete
Start the quantity calculation based on the template list.	Calculation
Load a previously saved template list file from 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 <i>Draw Alignment Sections</i> tool.	Draw / In sections

- Use the **Add** button to add a new template to the list.

OR

- Use the **Edit** button to edit a selected template.

This opens the **Template for alignment interval** dialog:



- Define **From station** and **To station** values in order to define the station range for using this template.
- Select a section template form the list of **Section templates** that is used for the given section range.
- 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 for computation 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.

- Click on the **Calculation** button in the **Define alignment section templates** dialog.

This closes the list and opens the **Compute alignment quantities** dialog:

**Compute alignment quantities**

**Calculation**

From station: 0.00  
To station: 210.00  
Step: 2.00 m  
Calculate: Entire trench  
Upper surface: Ground  
Lower surface:

**Report results by interval**

☐ Print to file  
☒ Draw in design  
Interval: 50.00 m

**Results**

Draw: Temporarily  
Done: 100 %  
Cut volume: 8179.1 m³

Calculate

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 design file. If **Print to file** was selected in the dialog, a window 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: <ul style="list-style-type: none"> <li><b>Entire trench</b> - the quantity of excavation is computed for the area enclosed by a surface model and the trench boundaries defined by the section templates.</li> <li><b>Layer</b> - the quantities of excavation is computed for the area between two surface models and the trench boundaries defined by the section templates.</li> </ul>
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 <b>Calculate</b> is set to <b>Layer</b> .
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 design file. The scale is divided into sections of the length defined by the <b>Interval</b> value. For each section, the rounded calculated cut volume value is displayed.

Setting:	Effect:
Draw	Display method for presenting the calculation results graphically in the design file: <ul style="list-style-type: none"><li>• <b>Temporarily</b> - (<i>not MicroStation V8i</i>) draws temporary elements.</li><li>• <b>Permanently</b> - draws permanent elements into the design file on the active level.</li></ul>
Color fields next to calculation result lines	Defines the display color for drawing the results in the design file. Uses the active color table in MicroStation.

## 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 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.

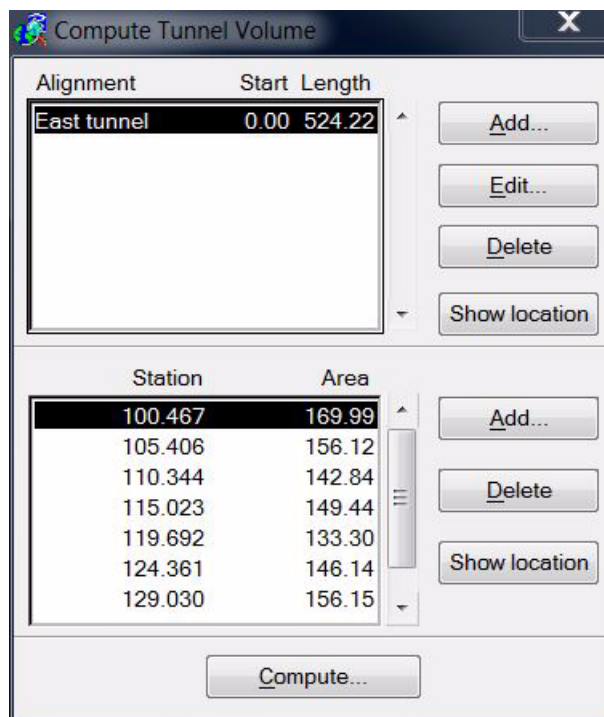
The volume can be computed for the whole tunnel area defined by the cross sections. Upper and lower elevation values can be defined for limiting the calculation to the area between the limit(s) and the cross section boundaries. The limits are given in absolute elevation values.

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. Use the MicroStation *Selection* tool to 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. Use the MicroStation *Selection* tool to 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: <ul style="list-style-type: none"> <li>• <b>One volume</b> - calculates one volume for the complete tunnel area defined by the cross sections.</li> <li>• <b>Multiple volumes</b> - calculates volumes for multiple elevation steps within the tunnel and the cross sections.</li> </ul>
Upper limit	If on, the volume is calculated from the tunnel floor up to the given elevation value. This is only active if <b>Report</b> is set to <b>One volumes</b> .
Lower limit	If on, the volume is calculated from the given elevation value up to the tunnel roof. This is only active if <b>Report</b> is set to <b>One volumes</b> .
Move	Limit that is moved to define the elevation steps for multiple volume calculation: <b>Upper limit</b> or <b>Lower limit</b> . This is only active if <b>Report</b> is set to <b>Multiple volumes</b> .
First	First elevation value included in volume calculation. This is only active if <b>Report</b> is set to <b>Multiple volumes</b> .
Last	Last elevation value included in volume calculation. This is only active if <b>Report</b> is set to <b>Multiple volumes</b> .
Step	Size of an elevation interval for volume calculation. This is only active if <b>Report</b> is set to <b>Multiple volumes</b> .

## 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.

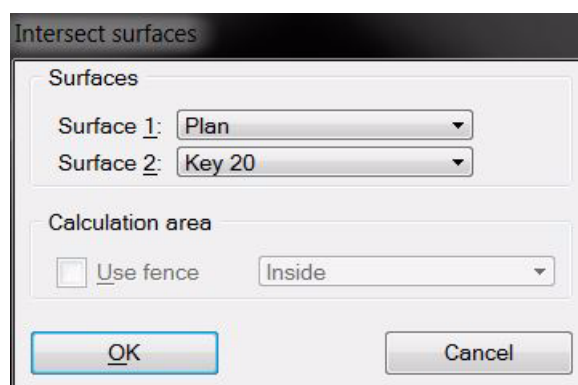
When calculating the line of intersection, TerraModeler compares individual triangles from the two surfaces with each other. If the planes of two triangles intersect each other, the intersection is drawn as a line segment. Thus, the intersection line might be represented by quite complex line strings 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 design file using active level and symbology settings in MicroStation.

Setting:	Effect:
Use fence	If on, the intersection is calculated for an area that is defined by a MicroStation fence: <b>Inside</b> or <b>Outside</b> the fence. This is only active if a MicroStation fence is drawn.



## 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 design file.

Setting:	Effect:
Surface	Name of the surface model for area calculation.
Compute	Area to compute: <ul style="list-style-type: none"> <li>• <b>Whole surface</b> - whole surface model.</li> <li>• <b>Fence contents</b> - only inside a MicroStation fence.</li> <li>• <b>Selected shapes</b> - only inside selected shape element(s).</li> </ul>
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 design file ( <i>not MicroStation V8i</i> ).
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 <b>Compute</b> is set to <b>Selected shapes</b> .

## 13 Using Domains

### Domains tool box

*Not Lite*

The tools in the **Domain** tool box 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 design file level, it is easier to do further manipulation with MicroStation tools, such as:

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

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*.

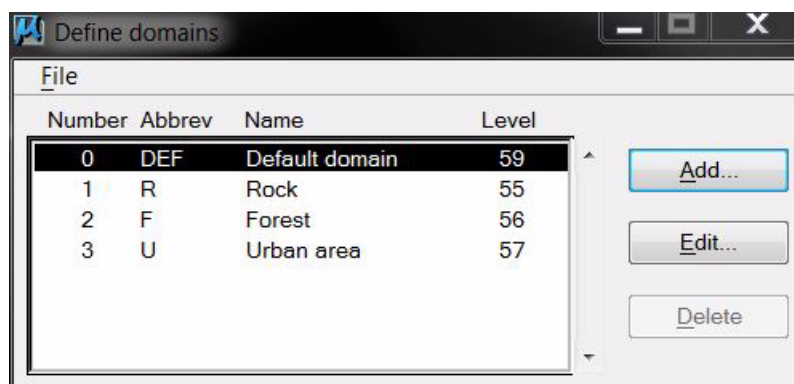
To:	Use:
Define domains	 <i>Define Domains</i>
Assign a view for dynamic domain display	 <i>Domain View</i>
Set domain of triangles inside an area	 <i>Set Area Domain</i>
Set domain of triangles between two linear elements	 <i>Set Domain Between Lines</i>
Set domain of triangles	 <i>Set Triangle Domain</i>
Display triangles using domain coloring	 <i>Display Domain Triangles</i>

## Define Domains

Not Lite



The *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 design file, TerraModeler automatically loads the domain definition file that was used last with this design 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.

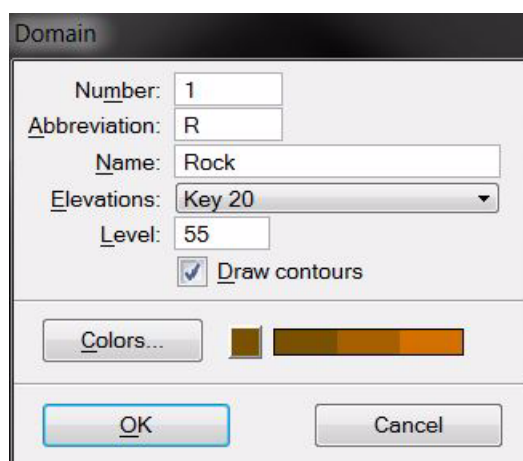
### ➤ To add or edit a domain:

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 coloring scheme.
Level	Level number for displaying triangles in the design file.
Draw contours	If on, contour lines are drawn for this domain. This effects the display of contour lines with <i>Display Contours</i> tool.
Colors	Opens the <b>Color scheme</b> dialog which is used to define a coloring scheme. See <b>Creating a coloring scheme of discrete colors</b> for more information.
Color button	Opens the active MicroStation color table for single color selection.

## Domain View

Not Lite



*Domain View* lets you select a view which displays domain assignments of the surface model you are working on. Whenever MicroStation 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



*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.

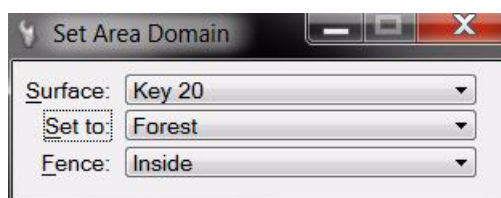
The area can be defined by a MicroStation fence or by selected shape element(s).



### 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 mouse 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: <b>Inside</b> or <b>Outside</b> .

## Set Domain Between Lines

Not Lite



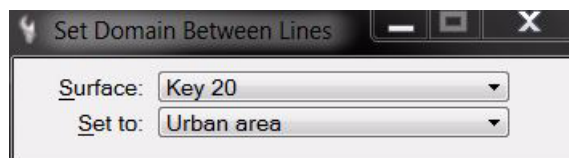
*Set Domain Between Lines* tool assigns a domain to all triangles between two linear elements. This tool is designed to be used with linear elements that are close to parallel to each other. TerraModeler uses the center point of a triangle to decide if the triangle is between the two linear elements.



### 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.
4. Identify the second linear element.
5. Accept the two highlighted linear elements.

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 assiged to the triangles.

## Set Triangle Domain

Not Lite

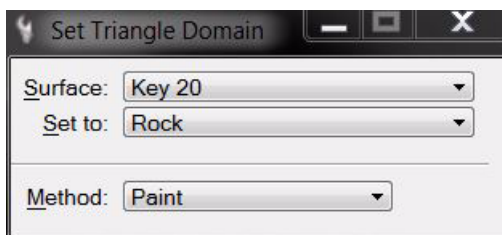


*Set Triangle Domain* tool assigns a domain to individual triangles. The tool can assign a domain to single triangles or to multiple triangles using a paint method.



**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 MicroStation view.

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

5. To set the domain of a single triangle, enter 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, enter another data point.

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: <ul style="list-style-type: none"> <li>• <b>One at a time</b> - each triangle is defined by a data click in order to assign the domain.</li> <li>• <b>Paint</b> - all triangles the mouse pointer passes over between two data clicks are assigned to the domain.</li> </ul>

## Display Domain Triangles

Not Lite



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

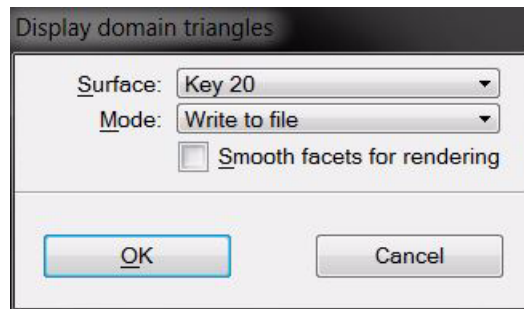
If there are areas of the surface model where no specific domain is assigned, the display options of the default domain are used.



### 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: <ul style="list-style-type: none"> <li>• <b>Preview</b> - displays triangles in preview mode.</li> <li>• <b>Write to file</b> - draws triangles permanently into the design file.</li> </ul>
Smooth facets for rendering	If on, triangles are displayed with a smoother change in coloring if MicroStation rendering is applied ( <i>Not MicroStation V8i</i> ).



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.

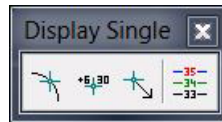






# 14 Placing Single Display Elements

## Display Single tool box

*Not Lite*

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



To:	Use:
Place a contour at a single elevation	 <i>Place Contour</i>
Place an elevation text	 <i>Place Elevation Text</i>
Place a slope arrow	 <i>Place Slope Arrow</i>
Modify symbology of contours at given intervals	 <i>Modify Contour Symbology</i>

## 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 MicroStation.

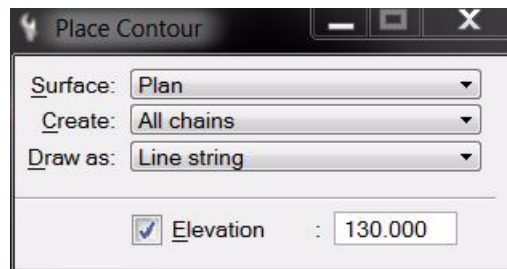
You can place the contour at a precise elevation by locking the elevation to a key-in value.



### 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 design 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: <ul style="list-style-type: none"> <li>• <b>Only one</b> - only the contour line at the mouse pointer location is drawn.</li> <li>• <b>All chains</b> - all contour lines at the elevation of the mouse pointer location are drawn.</li> </ul>
Draw as	Element type of the contour lines: <ul style="list-style-type: none"> <li>• <b>Curves</b> - smooth elements for which MicroStation computes the curvature, compact elements.</li> <li>• <b>Line strings</b> - sharp elements, suitable for preview.</li> </ul>
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.

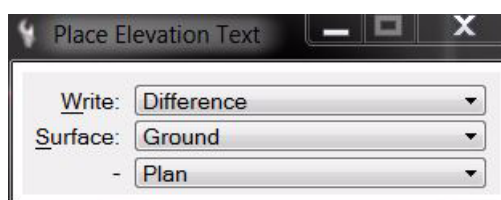
Text font, size, and numeric format are determined by leveling point settings in **Elevation labels category** of the TerraModeler **Settings**. The settings also define the location of the label relative to the exact elevation point and if a point marker is set at the location of the elevation point.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



### 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 design file. You can continue with steps 2 or 3.

Setting:	Effect:
Write	Defines the type of the label: <ul style="list-style-type: none"> <li>• <b>Elevation</b> - the elevation value of a surface model is drawn.</li> <li>• <b>Difference</b> - the elevation difference between two surface models is drawn.</li> </ul>
Surface	Surface model for which a label is placed. If <b>Write</b> is set to <b>Difference</b> , the surface is used as lower surface for the difference calculation.
-	If <b>Write</b> is set to <b>Difference</b> , 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.

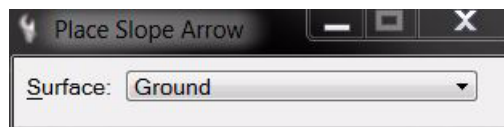
The shape of the slope arrows can be defined in **Place Slope Arrow / Arrows category** of the TerraModeler settings. It is further determined by the active symbology settings in MicroStation. In addition, a label showing the gradient of the slope can be placed for each slope arrow. The label settings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

## Modify Contour Symbology

Not Lite



*Modify Contour Symbology* tool modifies the symbology of contour lines. It can also move contour lines to another level in the design 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.

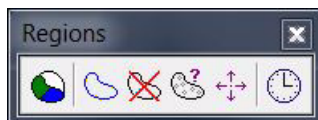
Setting:	Effect:
Surface	Name of the effected surface model.
Interval	Interval of contour lines that are modified.
Set contour level	If on, the contour lines are moved to the given MicroStation <b>Level</b> .
Set contour symbology	If on, the color, style, and weight of the contour lines are modified to the given <b>Symbology</b> . Uses the active color table of MicroStation, and MicroStation line styles and weights.
Set label level	If on, the contour labels are moved to the given MicroStation <b>Level</b> .
Set label symbology	If on, the color, style, and weight of the contour labels are modified to the given <b>Symbology</b> . Uses the active color table of MicroStation, and MicroStation line styles and weights.

# 15 Using Regions

## Regions tool box

*Not Lite*

The tools in the **Regions** tool box are used to create, manage, delete, and update regions.









Regions are another method to divide an area into thematical 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 thematical 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 tool box**, region tools do not rely on surface models or relate to the triangle structure of surfaces.

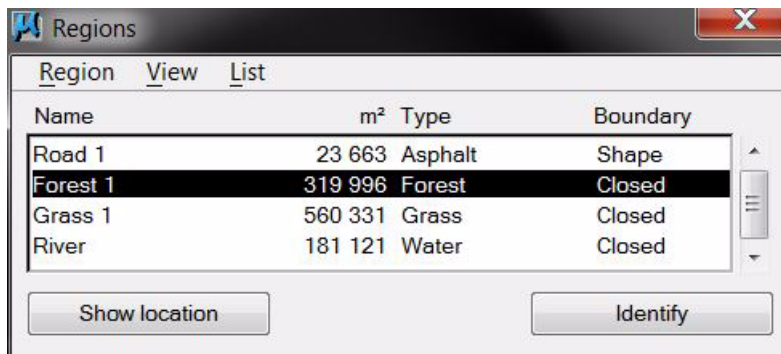
To:	Use:
Open regions window	 <i>Regions</i>
Create a new region definition	 <i>Create Region</i>
Delete existing region definition	 <i>Delete Region</i>
Edit region information	 <i>Edit Region Info</i>
Move region center point and title	 <i>Move Region Center</i>
Update region definitions (modified boundaries)	 <i>Update Regions</i>

## Regions

Not Lite



*Regions* tool opens the **Regions** window. The window contains a list of regions that are currently defined in the design 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	<b>Region / Create</b>
Modify the information for a region	<b>Region / Edit information</b>
Move the center point of a region	<b>Region / Move center point</b>
Assign a new boundary to a region	<b>Region / Assign new boundary</b>
Delete a region	<b>Region / Delete</b>
Modify the size of the window	<b>View/Small dialog</b> <b>View/Medium dialog</b> <b>View/Large dialog</b>
Define fields that are displayed in the list	<b>View / Fields</b>
Sort the list of regions	<b>View / Sort by</b>
Create output files for the list of regions	<b>List / Regions</b>

### 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 MicroStation shape element.

#### ➤ To assign a new boundary to a region:

1. Place the new boundary as MicroStation 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 design 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.

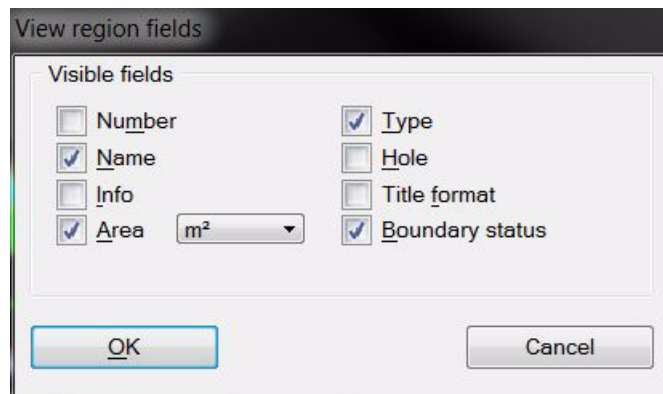
3. Click **Yes** in the **Alert** dialog.

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 MicroStation cell element into the design file. The list can include a color field and/or an area pattern field which illustrate the symbology of the different regions.
- **Text file** - a simple text file is created where fields are delimited by spaces.



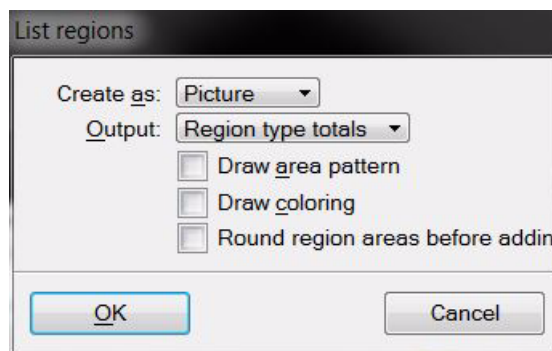
- **Table file** - a text file is created where fields are delimited by a tab.

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

3. If **Create as** is set to **Text file** or **Table file**, a standard 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: <ul style="list-style-type: none"> <li>• <b>Picture</b> - draws the list of regions as a cell element into the design file.</li> <li>• <b>Text file</b> - saves a space-delimited text file.</li> <li>• <b>Table file</b> - saves a tab-delimited text file.</li> </ul>
Output	Defines how regions of the same type are included in the output: <ul style="list-style-type: none"> <li>• <b>Each region</b> - the output includes a line for each region.</li> <li>• <b>Region type totals</b> - the output includes a line for each region type, the areas of regions of the same type are summarized.</li> </ul>
Draw area pattern	If on, the list output includes an area pattern field for each region (type). This is only active if <b>Create as</b> is set to <b>Picture</b> .
Draw coloring	If on, the list output includes a color field for each region (type). This is only active if <b>Create as</b> is set to <b>Picture</b> .
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 <b>Output</b> is set to <b>Region type totals</b> .

## Create Region

Not Lite



*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, *Set Linear Elevation* tool or *Drop Element On Surface* tool in TerraModeler, or the *Drape Linear Element* tool together with laser 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 design 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 design 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 <b>Regions / Region types category</b> for instruction how to define region types.
Title	Title format for labeling the new region. See <b>Regions / Title formats category</b> for instruction how to define region titles.

Setting:	Effect:
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 <i>Update Regions</i> 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 surrounded by another region, the option is switched on automatically. The name of the surrounding region is displayed next to the <b>Select</b> button. If the region should not form a hole, click on the <b>Select</b> button and select <b>Not a hole</b> in the <b>Select parent region</b> dialog.



Create region tool does not draw the region shape or any other display into the design file. Use *Update Regions* tool in order to display the region.

## Delete Region

*Not Lite*



*Delete Region* tool deletes a region definition. The region must be drawn in the design 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 design file.



### To delete a region:

1. Select the *Delete Regions* 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 via a MicroStation linkage. If a region definition is deleted, all display elements are deleted as well.

## Edit Region Info

Not Lite



*Edit Region Info* tool enables the modification of thematical 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 <b>Regions / Region types category</b> for instruction how to define region types.
Title	Title format for labeling the region. See <b>Regions / Title formats category</b> for instruction how to define region titles.
Name	Free text that is used a name for the region.
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 shrinked 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 <i>Update Regions</i> 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 automaticall. The name of the surrounding area is displayed next to the <b>Select</b> button. If the region should not form a hole, click on the <b>Select</b> button and select <b>Not a hole</b> in the <b>Select parent region</b> dialog.

## Move Region Center

Not lite



*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.

## Update Regions

Not lite



*Update Regions* tool redraws the regions in the design file. If new region definitions were created or existing regions were updated, the tool draws all display elements into the design 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**.

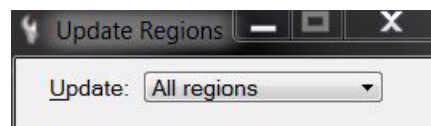
The different region elements can be drawn on several MicroStation levels which supports their modification with MicroStation tools. The levels are defined in **Regions / Region levels category** of the TerraModeler **Settings**.



### 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.

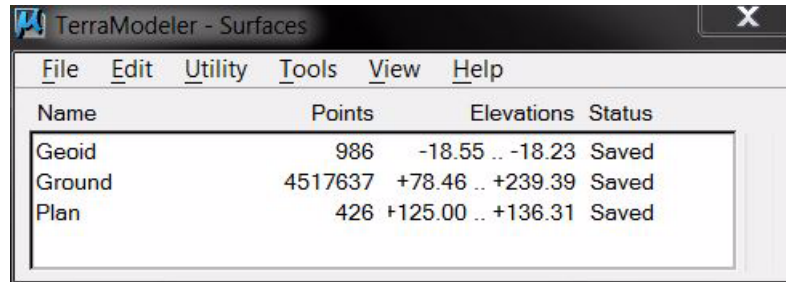


If a region can not be drawn, an error message is displayed. This can be caused, for example if region boundaries are not closed or if they are deleted. There are some tolerances related to gaps in the boundary line network which are defined in **Regions / Region design category** of the TerraModeler **Settings**.

## 16 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 tool boxes that are part of the TerraModeler **Main** tool box.

There are three methods included in the **Surfaces** window that produce surface representation files in batch processes:

- **Produce contours** - creates contour design 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 design files that stores the triangle shapes.

The batch processes can include data from several sources, such as laser data from TerraScan projects, survey elements from TerraSurvey or other applications, and breaklines filtered by breakline rules.

## File pulldown menu

Menu commands from **File** pulldown menu in the **Surfaces** window are used to open, save, close, and restore surfaces.

To:	Choose menu command:
Create a new surface file	<b>File / New surface</b>
Open a surface file stored on a hard disk	<b>File / Open surface</b>
Close a surface file	<b>File / Close surface</b>
Save a surface file on a hard disk	<b>File / Save surface</b>
Open a reference surface	<b>File / Open reference</b>
Import surface files	<b>File / Import</b>
Export to surface files	<b>File / Export</b>
Restore a surface after modifications	<b>File / Restore from file</b>
Open the TerraModeler <b>Settings</b> dialog	<b>File / User settings</b>
Unload TerraModeler	<b>File / Exit</b>



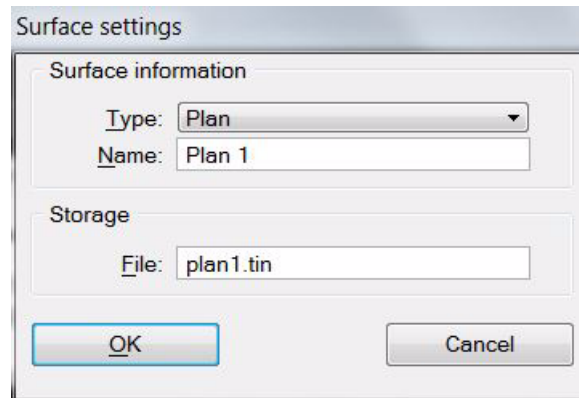
## File / New surface

**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 tool box**.

## 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.

The **Open surface** dialog opens, a standard TerraModeler or MicroStation dialog for opening files.

2. 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.

## File / Close surface

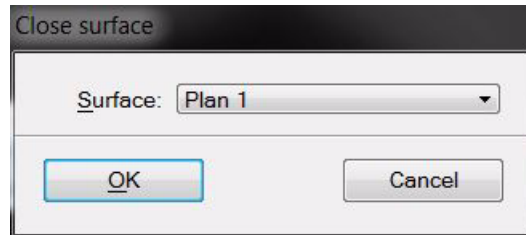
**Close surface** command removes a surface model from the list of loaded surfaces. It does not delete the surface model file from the hard disk.



### To close a 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.

Alternatively, all loaded surface models can be closed at once with the **Close all** command from the **File** pulldown menu.

## File / 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 design 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** on page 283 for more information about the surface model file format.

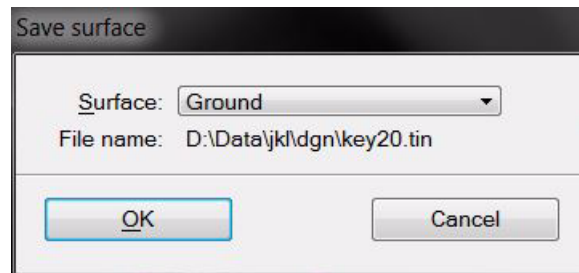
You can save surface models to files using one of the three options in the **File** menu of the **Surfaces** window:

- **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:



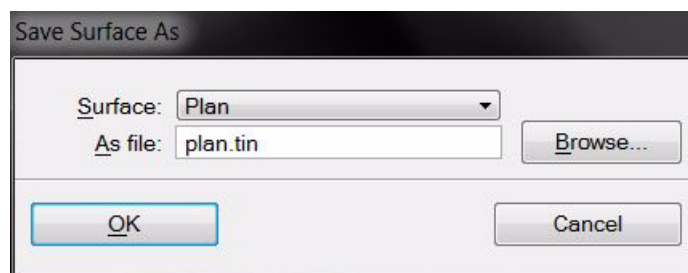
2. Select a **Surface** to save.
3. Click OK.

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 design file is saved or after a model is created. This is defined in **Saving surfaces category** of the TerraModeler settings.

## File / 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 be 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 TerraModeler or MicroStation 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.

## File / 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** on page 231.

## File / 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** on page 231.

## File / Restore from file

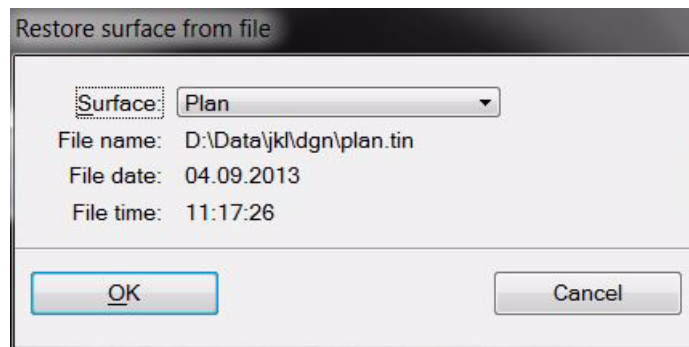
**Restore from file** command reloads the surface model file from the hard disk. This can be used after modifying a surface in order to restore the previously saved situation, for example if it turns out that the modifications were a mistake.

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.

## File / User settings

**User settings** command opens the TerraModeler **Settings** window. It performs the same action as the *Settings* tool in TerraModeler **General tool box**.

## File / Exit

**Exit** command closes TerraModeler, it unloads the application from MicroStation.

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.

## 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:	Choose menu command:
Modify surface information	<b>Edit / Modify settings</b>
Modify display settings for surfaces in profiles	<b>Edit / Profile settings</b>
Exclude long triangles from a surface	<b>Edit / Exclude long triangles</b>
Change elevations of a surface by a mathematical equation	<b>Edit / Modify elevations</b>
Remove unnecessary points from a surface	<b>Edit / Thin</b>
Remove previously deleted points and triangles from a surface	<b>Edit / Compress</b>
Add intermediate points to a surface	<b>Edit / Add inferred points</b>
Delete a surface file from the hard disk	<b>Edit / Delete surface</b>

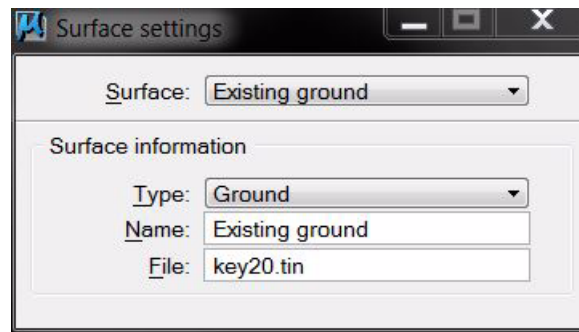
## Edit / Modify settings

**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** 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, 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.

## Edit / 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.

## Edit / Exclude long triangles

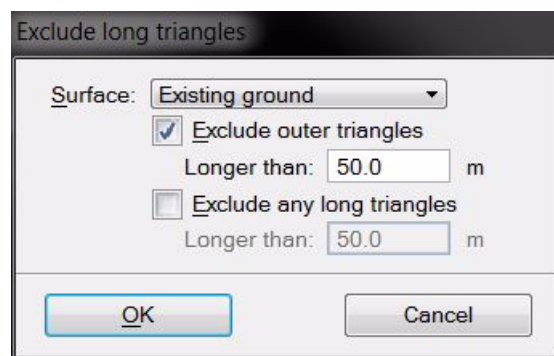
**Exclude long triangles** command excludes narrow, long triangles from a surface model. There can be different exclusion settings for triangles at the outer boundary of a surface model or for any long triangles.

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 **Edit / 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** 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 <b>Longer than</b> the given value are excluded.
Exclude any long triangles	If on, any triangles of the surface model with an edge length <b>Longer than</b> 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.



## Edit / Modify elevations

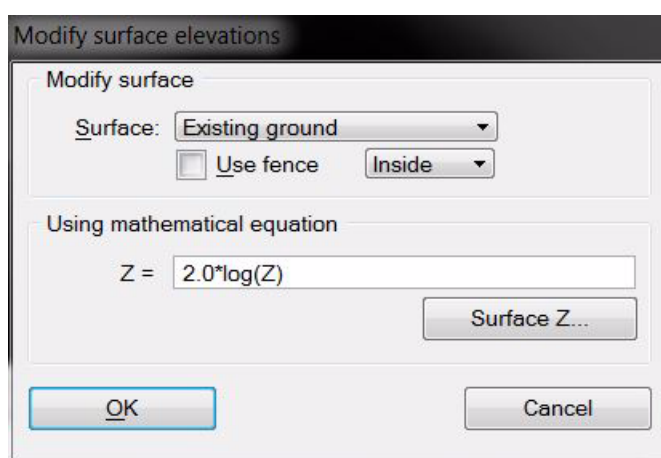
**Modify elevations** command changes the elevation values of a surface model by using a mathematical equation. The process computes a new elevation value for each point of a surface model.

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** 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 MicroStation method 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 <b>Inside</b> or <b>Outside</b> a fence are effected. This is only active if a MicroStation fence was drawn before the command is selected.
Z =	Mathematical expression that is applied to surface model points in order to calculated new elevation values.
Surface Z	The button opens the <b>Append surface</b> dialog which lets you select a surface model. The elevation values of this surface are then used in the mathematical expression.

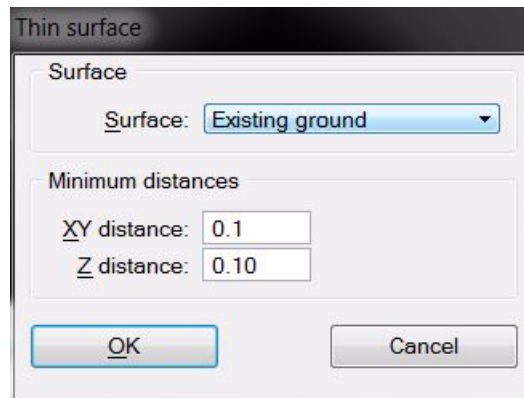
## Edit / Thin

**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 **Edit / Compress** command in order to delete the inactive data completely from a surface model.

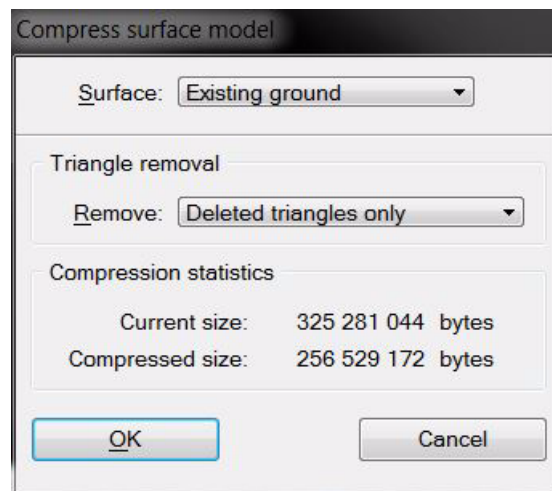
## Edit / Compress

**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**.

## Edit / Add inferred points

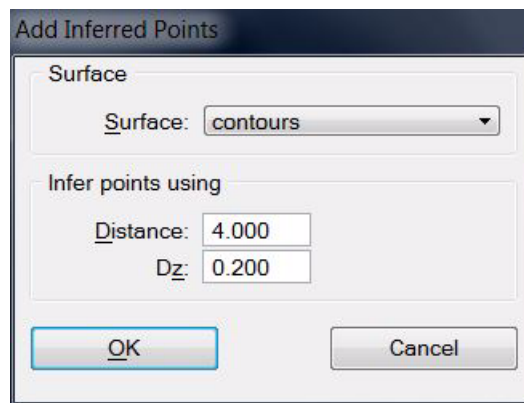
**Add inferred points** command inserts points on hill tops and in depressions in a surface model that is created from *contour point* types. The added points result in a rounder shape for hill tops and depressions.

The points are added as points of type *inferred points* to the surface model. See **Point Types** on page 285 for more information about surface model point types. The surface model can be created, for example, from contour line elements drawn in the design 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.

## Edit / 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.

## 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:	Choose menu command:
View surface statistics	<b>Utility / View statistics</b>
Create a copy of a surface	<b>Utility / Copy surface</b>
Create a combined surface from two surfaces	<b>Utility / Merge surfaces</b>
Subtract elevations of two surfaces	<b>Utility / Subtract surfaces</b>
Create a statistical grid surface model	<b>Utility / Statistical model</b>
Check surface model elevations against ground control points	<b>Utility / Output control report</b>
Create contour lines in batch mode	<b>Utility / Produce contours</b>
Create lattice files in batch mode	<b>Utility / Produce lattice models</b>
Create triangles in batch mode	<b>Utility / Produce triangles</b>
Work with lattice databases	<b>Utility / Lattice database</b>

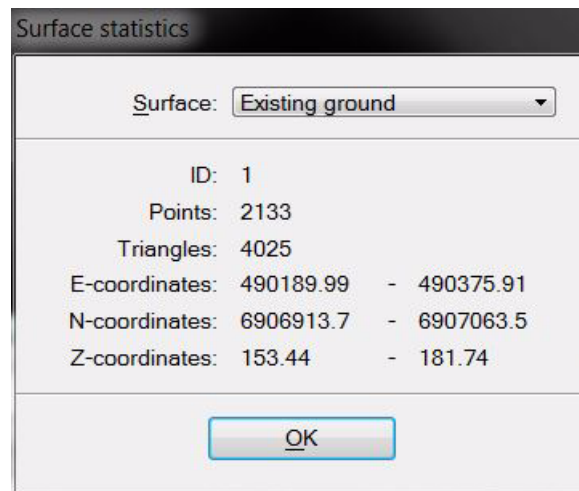
## Utility / View statistics

**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.

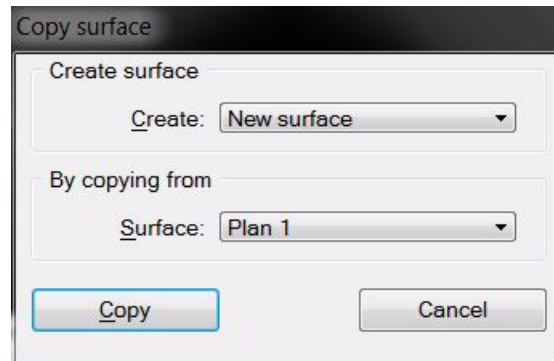
## Utility / Copy surface

**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:**

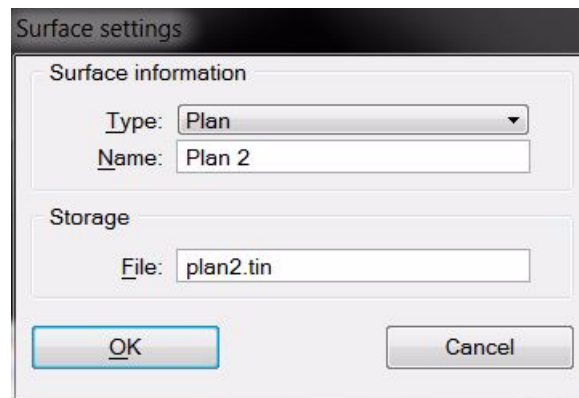
1. (Optional) Select a surface model from the list in the **Surfaces** window.
2. Select **Copy surface** command from the **Utility** pulldown menu.

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.



## Utility / Merge surfaces

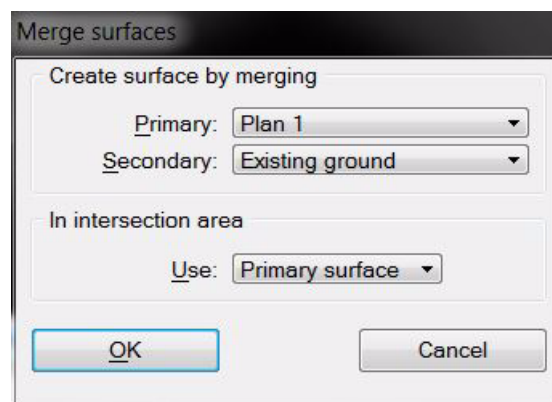
**Merge surfaces** command combines two surfaces. It creates a new surface model in the RAM and as surface model file on a hard disk. The new surface model file is saved at the same location where the original surface models are stored.

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:

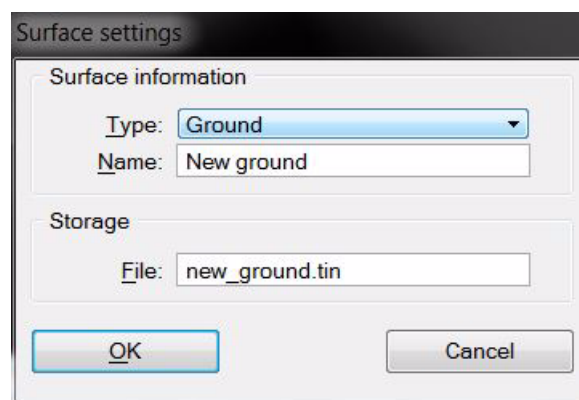
1. Select **Merge surfaces** command from the **Utility** pulldown menu.

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.

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.

Setting:	Effect:
Primary	All the information from the primary surface is copied to the new surface.

Setting:	Effect:
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: <ul style="list-style-type: none"><li>• <b>Primary surface</b> - use the primary surface.</li><li>• <b>Highest surface</b> - use the higher of the two surfaces.</li><li>• <b>Lowest surface</b> - use the lower of the two surfaces.</li></ul>

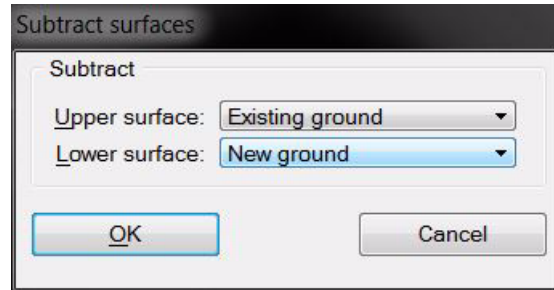
## Utility / Subtract surfaces

**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 *File / Save surface* command in order to save the new surface model.

➤ **To subtract two surface models:**

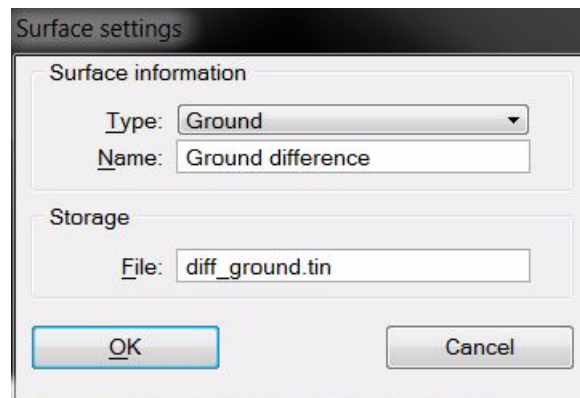
1. Select **Subtract surfaces** command from the **Utility** pulldown menu.

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.

## Utility / Statistical model

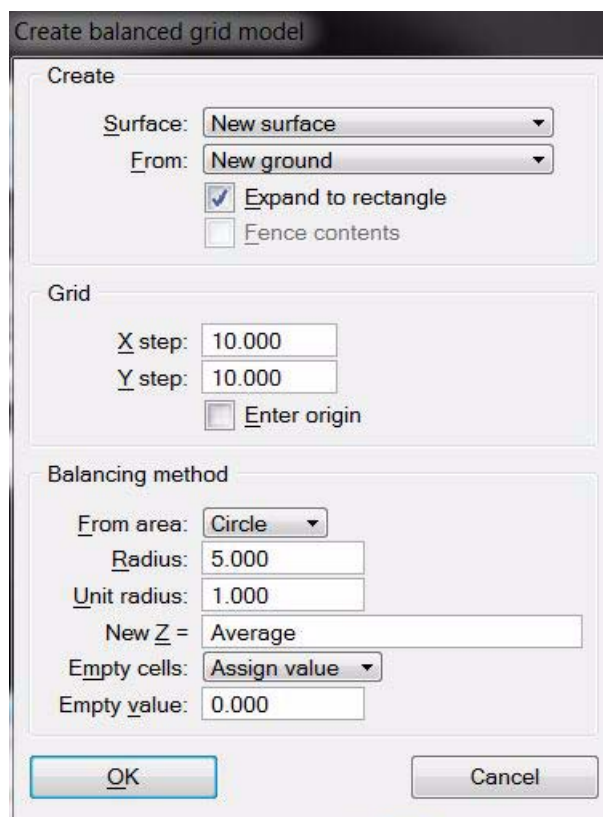
**Statistical model** command calculates statistical values based on an existing surface model. It creates a new grid model in the RAM and as surface model file on a hard disk. The new grid model file is saved at the same location where the original surface model is stored.

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.

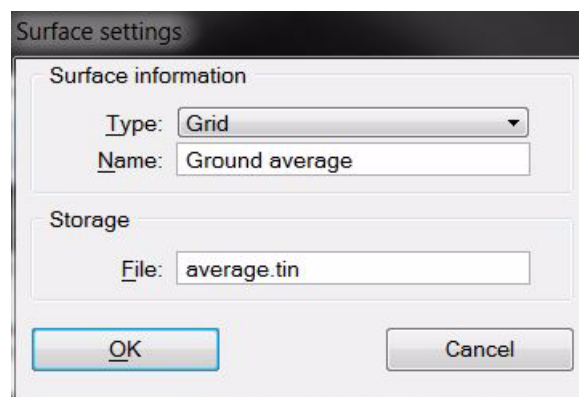
This opens the **Create balanced grid model** dialog:



The 'Create balanced grid model' dialog box is divided into three main sections: 'Create', 'Grid', and 'Balancing method'.  
 - **Create section:** Includes a 'Surface' dropdown menu set to 'New surface', a 'From' dropdown menu set to 'New ground', a checked checkbox for 'Expand to rectangle', and an unchecked checkbox for 'Fence contents'.  
 - **Grid section:** Includes input fields for 'X step' and 'Y step', both set to '10.000', and an unchecked checkbox for 'Enter origin'.  
 - **Balancing method section:** Includes a 'From area' dropdown menu set to 'Circle', input fields for 'Radius' (5.000) and 'Unit radius' (1.000), a 'New Z =' text field set to 'Average', an 'Empty cells' dropdown menu set to 'Assign value', and an 'Empty value' input field set to '0.000'.  
 At the bottom are 'OK' and 'Cancel' buttons.

2. Define settings and click OK.

This opens the **Surface settings** dialog:



The 'Surface settings' dialog box has two main sections: 'Surface information' and 'Storage'.  
 - **Surface information section:** Includes a 'Type' dropdown menu set to 'Grid' and a 'Name' text field set to 'Ground average'.  
 - **Storage section:** Includes a 'File' text field set to 'average.tin'.  
 At the bottom are 'OK' and 'Cancel' buttons.

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.

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 MicroStation 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.
From area	Shape of the calculation area around a grid point: <ul style="list-style-type: none"> <li>• <b>Circle</b> - circle with a given radius.</li> <li>• <b>Square</b> - square with <b>Radius</b> value equal to half the side length.</li> </ul>
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: <ul style="list-style-type: none"> <li>• <b>Count</b> - the number of points.</li> <li>• <b>Average</b> - the average of elevation values.</li> <li>• <b>Sum</b> - the sum of elevation values.</li> <li>• <b>Gravsum</b> - the gravitational sum of elevation values.</li> <li>• <b>Minimum</b> - smallest elevation value.</li> <li>• <b>Maximum</b> - highest elevation value.</li> </ul>
Empty cells	Determines how cells without a calculated value are handled in the grid model: <ul style="list-style-type: none"> <li>• <b>Assign value</b> - a constant value is assigned.</li> <li>• <b>Skip</b> - empty cells are skipped in the model.</li> </ul>
Empty value	Constant value that is assigned to cells without a calculated value in the grid model. This is only active if <b>Empty cells</b> is set to <b>Assign value</b> .

## Utility / 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:

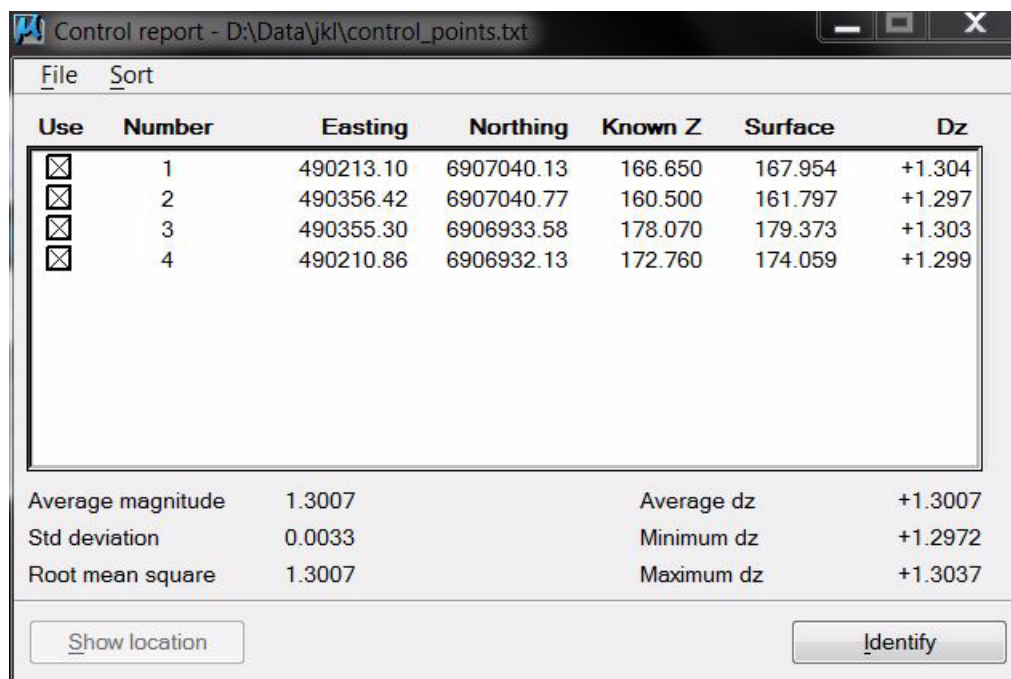
2. Select the text file that stores the control points.
3. Define settings and click OK.

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 computing the terrain slope so that small triangles do not exceed the <b>Max slope</b> 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 screenshot shows a window titled "Control report - D:\Data\jkl\control\_points.txt". It contains a table with 7 columns: Use, Number, Easting, Northing, Known Z, Surface, and Dz. There are 4 rows of data, all with checked boxes in the "Use" column. Below the table, there is a summary section with four rows: Average magnitude (1.3007), Std deviation (0.0033), Root mean square (1.3007), and Average dz (+1.3007). To the right of these are Average dz (+1.3007), Minimum dz (+1.2972), and Maximum dz (+1.3037). At the bottom, there are two buttons: "Show location" and "Identify".

Use	Number	Easting	Northing	Known Z	Surface	Dz
<input checked="" type="checkbox"/>	1	490213.10	6907040.13	166.650	167.954	+1.304
<input checked="" type="checkbox"/>	2	490356.42	6907040.77	160.500	161.797	+1.297
<input checked="" type="checkbox"/>	3	490355.30	6906933.58	178.070	179.373	+1.303
<input checked="" type="checkbox"/>	4	490210.86	6906932.13	172.760	174.059	+1.299

Average magnitude	1.3007	Average dz	+1.3007
Std deviation	0.0033	Minimum dz	+1.2972
Root mean square	1.3007	Maximum dz	+1.3037

Show location Identify

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 respective 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 MicroStation 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 MicroStation 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 **Edit / Modify elevations** command in the **Surfaces** window to apply the elevation correction to the surface model.

## Utility / Produce contours

*Not Lite*

**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 design files, vector elements created with TerraSurvey or any other application enabled in **Triangulate Survey category** in the TerraModeler **Settings**.
- **Breakline elements** - in reference design files, vector elements properly organized by level and symbology, breaklines filtered by rules using the *Triangulate Elements* tool.

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 design files. A seed file is used as starting point for creating the contour design 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 MicroStation 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* tool, *Contour label options* tool, and *Contour tick options* tool.
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 MicroStation 3D design 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.
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 design 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 design 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:

**Produce contours**

**Model settings**

Model buffer: 200.000 m around tile

☐ Save each model

**Contours and labels for peaks and pits**

☒ Draw contours

Settings: R:\Data\lattice\_contours\dgn\contour\_label\_ticks.set

☒ Label peaks and pits

Settings: R:\Data\lattice\_contours\dgn\peaks\_pits.set

**Design files to create**

Seed file: R:\Data\lattice\_contours\out\_contours\contours\_template.dgn

Directory: R:\Data\lattice\_contours\out\_contours

Name prefix: c

**Model data sources**

☒ Laser points

Project: R:\Data\lattice\_contours\laser\_contours\training.prj

Classes: 8

☐ Survey elements

☒ Vector elements

Rule file: R:\Data\lattice\_contours\dgn\breaklines.txt

11. Define settings and click OK.

This opens the **Triangulate surface** dialog.

12. Define settings for surface creation. See **Triangulate Survey** on page 50 for a description of the settings in the dialog.
13. Click OK.

The software starts the contour line production. First, it closes the original design file from which the process was started. Next, it opens the seed file, computes the contours for the first shape, writes them into the design file, and closes the design file. Then the process continues with the next shape. After all contour design files are created, the original design 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.
Save each model	If on, a surface model file is saved for each shape.
Draw contours	If on, contours are drawn in the design files. The given <b>Settings</b> are used for contour computation and display.
Label peaks and pits	If on, labels for peaks and pits are drawn in the design file. The given <b>Settings</b> are used for label display.
Seed file	Location of the design file that serves as seed file for contour line production.

Setting:	Effect:
Directory	Location on a hard disk where the contour design files are stored.
Name prefix	Text string that is added at the beginning of the file name of contour design files. If text elements are selected for each shape, the contour design file names are a combination of the prefix and the selected text element. If no text elements are selected, the contour design files are named with the prefix and an automatically increasing number.
Laser points	If on, laser points from the given <b>Project</b> and <b>Class</b> are used for surface model creation.
Survey elements	If on, feature coded elements created by TerraSurvey or any other application enabled in <b>Triangulate Survey category</b> in the TerraModeler <b>Settings</b> are used for surface model creation.
Vector elements	If on, vector elements filtered as breaklines according to the given <b>Rule file</b> are used for surface model creation.

## Utility / Produce lattice models

*Not Lite*

**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 laser data 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 design files, vector elements created with TerraSurvey or any other application enabled in **Triangulate Survey category** in the TerraModeler **Settings**.
- **Breakline elements** - in reference design files, vector elements properly organized by level and symbology, breaklines filtered by rules using the *Triangulate Elements* tool.

See **Export / Lattice file** on page 239 for a description of how to create a lattice model file from a surface model.



### **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 MicroStation 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:

Produce lattice models

**Model settings**  
Model buffer: 200.000 m around tile  
☐ Save each model

**Model data sources**  
☒ Laser points  
Project: R:\Data\lattice\_contours\laser\_contours\training.prj Browse...  
Classes: 2  
☐ Survey elements  
☒ Vector elements  
Rule file: R:\Data\lattice\_contours\breakline\_rules.bt Browse...

**Lattice files**  
Grid spacing: 1.000 m  
File format: Shaded GeoTIFF  
Values at: Cell center  
Range: Uniform 100.000 - 200.000  
Sun azimuth: 45.0  
Sun angle: 90.0 deg above horizon  
Color scheme: Default Define...  
Color cycles: 1

**Lattice file naming**  
Directory: R:\Data\lattice\_contours\out\_lattice Browse...  
Name prefix: dem\_  
Extension: tif

OK Cancel

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.
Save each model	If on, a surface model file is saved for each shape.
Laser points	If on, laser points from the given <b>Project</b> and <b>Class</b> are used as data source.
Survey elements	If on, feature coded elements created by TerraSurvey or any other application enabled in <b>Triangulate Survey category</b> in the TerraModeler <b>Settings</b> . are used as data source.
Vector elements	If on, vector elements filtered as breaklines according to the given <b>Rule file</b> are used as data source.
Grid spacing	Constant distance between points in the grid structure of the lattice model files.

Setting:	Effect:
File format	<p>Format of the output files. Available formats are:</p> <ul style="list-style-type: none"> <li>• ArcInfo Grid, Intergraph GRD</li> <li>• GeoTIFF formats, Shaded GeoTIFF</li> <li>• Japanese DMF and LEM</li> <li>• Raw image formats</li> <li>• Surfer ASCII and binary</li> <li>• XYZ text</li> </ul> <p>Many of the additional settings for the lattice files depend on the selected format for the output files.</p>
Values at	<p>Defines the exact location of the lattice point coordinates:</p> <ul style="list-style-type: none"> <li>• <b>Cell center</b> - the coordinates of the grid cell center are stored.</li> <li>• <b>Cell corner</b> - the coordinates of the grid cell corner are stored.</li> </ul> <p>For some output file formats only one option is available.</p>
Outside Z	<p>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.</p>
Z unit	<p>Defines the elevation value unit for several raster formats.</p>
Create TFW files	<p>If on, external georeferencing files are created for <b>GeoTIFFs</b>.</p>
Write coordinate block	<p>If on, a coordinate block file is written for <b>Intergraph GRD</b> files.</p>
Byte order	<p>Defines the byte order for <b>Raw integer</b> image formats related to the processor type: <b>Intel</b> or <b>Motorola</b>.</p>
Range	<p>Determines how the color scheme is fitted to the elevation values for <b>Shaded GeoTIFF</b> files:</p> <ul style="list-style-type: none"> <li>• <b>Uniform</b> - the range should include all elevation values of the area for which GeoTIFFs are produced in order to ensure seamless coloring over all images.</li> <li>• <b>Fit to tile</b> - the colors are fitted to elevation values of each tile individually.</li> </ul>
Sun azimuth	<p>Direction of the light for <b>Shaded GeoTIFFs</b>. The direction from the north is defined with zero and angle values increase clockwise.</p>
Sun angle	<p>Height of the light source above the horizon.</p>
Color scheme	<p>Color scheme to use for <b>Shaded GeoTIFFs</b>:</p> <ul style="list-style-type: none"> <li>• <b>Default</b> - default color scheme containing magenta, red, yellow, green, cyan, blue, and purple.</li> <li>• <b>Selected colors</b> - use <b>Define</b> button to create a customized color scheme. See <b>Define a customized color scheme</b> on page 119 for more information.</li> </ul>
Color cycles	<p>Number of color cycles used for <b>Shaded GeoTIFFs</b>. Use zero to create a gray scale display showing triangle slope only.</p>
Outside points	<p>Defines for <b>Xyz text</b> files how to handle grid points outside the lattice model area where no elevation value can be derived from the source data:</p> <ul style="list-style-type: none"> <li>• <b>Skip</b> - outside grid points are not written into the text file.</li> <li>• <b>Output</b> - outside grid points are written into the text file using the constant value given in the <b>Outside Z</b> field.</li> </ul>

Setting:	Effect:
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 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.

## Utility / Produce triangles

*Not Lite*

**Produce triangles** command creates triangles in an automated process. This is useful for triangle design 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 design files, vector elements created with TerraSurvey or any other application enabled in **Triangulate Survey category** in the TerraModeler **Settings**.
- **Breakline elements** - in reference design files, vector elements properly organized by level and symbology, breaklines filtered by rules using the *Triangulate Elements* tool.

The automatic process writes the triangles as shape elements into multiple design files. A seed file is used as starting point for creating the triangle design files.



### **To produce triangle design 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 MicroStation and TerraScan tools.
3. Create a seed file for storing the triangles.

The seed file is usually an empty MicroStation 3D design 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.

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 design 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 design 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:



**Produce triangles**

**Model settings**  
 Model buffer: 200.000 m around tile  
☐ Save each model

**Settings for drawing triangles**  
 Draw triangles: Center inside tile  
 Color by: Elevation - Uniform  
 Range: 100.000 - 200.000  
 Level: 59

**Design files to create**  
 Seed file: R:\Data\example\seed.dgn   
 Directory: R:\Data\example   
 Name prefix: triangles\_

**Model data sources**  
☒ Laser points  
 Project: R:\Data\example\laser\demo.prj   
 Classes: 2  
☐ Survey elements  
☒ Vector elements  
 Rule file: R:\Data\example\breakline\_rules.txt

9. Define settings and click OK.

This opens the **Triangulate surface** dialog.

10. Define settings for surface creation. See **Triangulate Survey** on page 50 for a description of the settings in the dialog.
11. Click OK.

The software starts the triangle production. First, it closes the original design file from which the process was started. Next, it opens the seed file, computes the triangles for the first shape, writes them into the design file, and closes the design file. Then the process continues with the next shape. After all triangle design file are created, the original design 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.
Save each model	If on, a surface model file is saved for each shape.



Setting:	Effect:
Draw triangles	Defines how triangles are drawn on tile boundaries: <ul style="list-style-type: none"> <li>• <b>All</b> - all triangles are drawn.</li> <li>• <b>Center inside tile</b> - triangles are drawn completely if the center point is inside the tile.</li> <li>• <b>Clipped to tile boundary</b> - triangles are clipped to the tile boundaries. This results in rectangles along tile boundaries..</li> </ul>
Color by	Defines the coloring mode: <ul style="list-style-type: none"> <li>• <b>Elevation - Uniform</b> - triangles are colored by elevation value. The colors are fitted to the given <b>Range</b> of elevation values.</li> <li>• <b>Elevation - Fit to tile</b> - triangles are colored by elevation value. The colors are fitted to elevation values of each tile individually.</li> <li>• <b>Slope</b> - triangles are colored by slope gradient.</li> </ul>
Range	Elevation range for all triangle design 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 design files. This is only active if <b>Color by</b> is set to <b>Elevation - Uniform</b> .
Level	Level on which the triangles are drawn into the design file.
Colors	Opens the <b>Color scheme</b> dialog for the definition of a color scheme for triangle coloring. See <b>Creating a coloring scheme of discrete colors</b> for more information.
Color button	Opens the MicroStation color table for single color selection.
Seed file	Location of the design file that serves as seed file for triangle production.
Directory	Location on a hard disk where the triangle design files are stored.
Name prefix	Text string that is added at the beginning of the file name of triangle design files. If text elements are selected for each shape, the triangle design file names are a combination of the prefix and the selected text element. If no text elements are selected, the triangle design files are named with the prefix and an automatically increasing number.
Laser points	If on, laser points from the given <b>Project</b> and <b>Class</b> are used as data source.
Survey elements	If on, feature coded elements created by TerraSurvey or any other application enabled in <b>Triangulate Survey category</b> in the TerraModeler <b>Settings</b> are used as data source.
Vector elements	If on, vector elements filtered as breaklines according to the given <b>Rule file</b> are used as data source.

## Utility / Lattice database


*Not Lite*

**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** on page 246.

## Tools pulldown menu

Menu commands from **Tools** pulldown menu in **Surfaces** window can be used to open tool boxes of TerraModeler. The tool boxes are the same as in the **Main** tool box of the application. In addition, there is an **Extension** category with two tool boxes that are not part of the **Main** tool box: Flood Mapping and Lifting 2D Contours .

To:	Choose menu command:
Open the <b>Main</b> tool box	Tools / Main
Open the <i>General tool box</i>	Tools / General
Open the <i>Create Surfaces tool box</i>	Tools / Create Surfaces
Open the <i>Draw using Surface tool box</i>	Tools / Draw using Surface
Open the <i>Drawing Utilities tool box</i>	Tools / Drawing Utilities
Open the <i>Edit Point tool box</i>	Tools / Edit Point
Open the <i>Edit Area tool box</i>	Tools / Edit Area
Open the <i>Display Surface tool box</i>	Tools / Display Surface
Open the <i>Display Regions tool box</i>	Tools / Display Regions
Open the <i>Profiles tool box</i>	Tools / Profiles
Open the <i>Quantity tool box</i>	Tools / Quantity
Open the <i>Domains tool box</i>	Tools / Domains
Open the <i>Flood Mapping tool box</i>	Tools / Extensions / Flood Mapping
Open the <i>2D Contours tool box</i>	Tools / Extensions / 2D Contours

 In TerraModeler Lite, only tool boxes that are available in the Lite version are included in the **Tools** pulldown menu.

## View pulldown menu

Menu commands from **View** pulldown menu in **Surfaces** window are used to change the appearance of the **Surfaces** window.

To:	Choose menu command:
Display a minimized <b>Surface</b> window	<b>View / Minimal dialog</b>
Display a small size <b>Surfaces</b> window	<b>View / Small dialog</b>
Display a medium size <b>Surfaces</b> window	<b>View / Medium dialog</b>
Display a large size <b>Surfaces</b> window	<b>View / Large dialog</b>
Change the display of fields in the <b>Surfaces</b> window	<b>View / Fields</b>
Sort the list of surfaces in the <b>Surfaces</b> window	<b>View / Sort by</b>

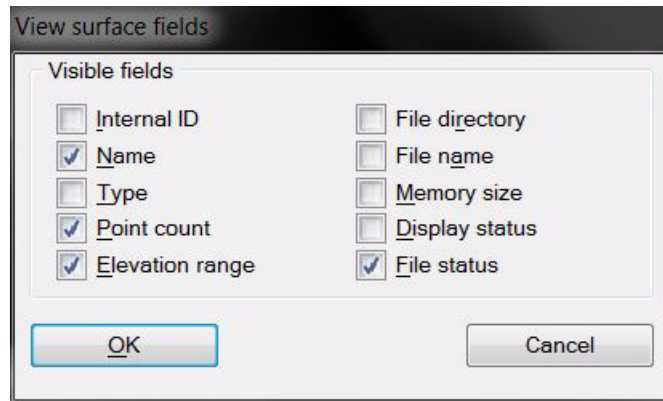
## View / Fields

**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. Set desired fields on and click OK.

## View / Sort by

**Sort by** command sorts the list of loaded surface models according to the selected attribute. The submenu 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 design 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.

## Help pulldown menu

Menu commands from **Help** pulldown menu in **Surfaces** window are used to open this Users' Guide and to display license information for TerraModeler.

To:	Choose menu command:
Open the TerraModeler Users' Guide	<b>Help / Contents</b>
Display the license information dialog	<b>Help / About TerraModeler</b>

### Help / Contents

**Contents** command launches Acrobat Reader for accessing this Users' Guide in PDF format. It performs the same action as the *Help On TerraModeler* tool.

# 17 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** windows.

For data type:	Use for import:	Use for export:
Xyz text file	<b>Import / XYZ text file</b>	<b>Export / Xyz text file</b>
Xyz binary file (LAS, BIN, FBI)	<b>Import / XYZ binary file</b>	-
Disimp grid file	<b>Import / Lattice file</b>	<b>Export / Lattice file</b>
Intergraph grid file (GRD)	<b>Import / Lattice file</b>	<b>Export / Lattice file</b>
Ordnance Survey NTF grid file	<b>Import / Lattice file</b>	-
Triangles in design file	<b>Import / Triangles</b>	<i>Display Triangles</i> tool
Triangle text file	-	<b>Export / Triangle text file</b>
ArcInfo grid file (GRD)	-	<b>Export / Lattice file</b>
GeoTIFF grid file	-	<b>Export / Lattice file</b>
Japanese grid file	-	<b>Export / Lattice file</b>
Raw binary grid file	-	<b>Export / Lattice file</b>
Surfer grid file	-	<b>Export / Lattice file</b>
Xyz text grid file	<b>Import / XYZ text file</b>	<b>Export / Lattice file</b>
Elevation-colored raster image	-	<b>Export / Raster image</b>
Graphical elements	<i>Triangulate Elements</i> tool	<b>Export / Graphical elements</b>
SiteWorks surface model	<b>File / Open surface</b>	-

## Import / XYZ text file

**Xyz text file** command is used to create a new surface model or to add points to an existing model from a text file. The text file stores 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.

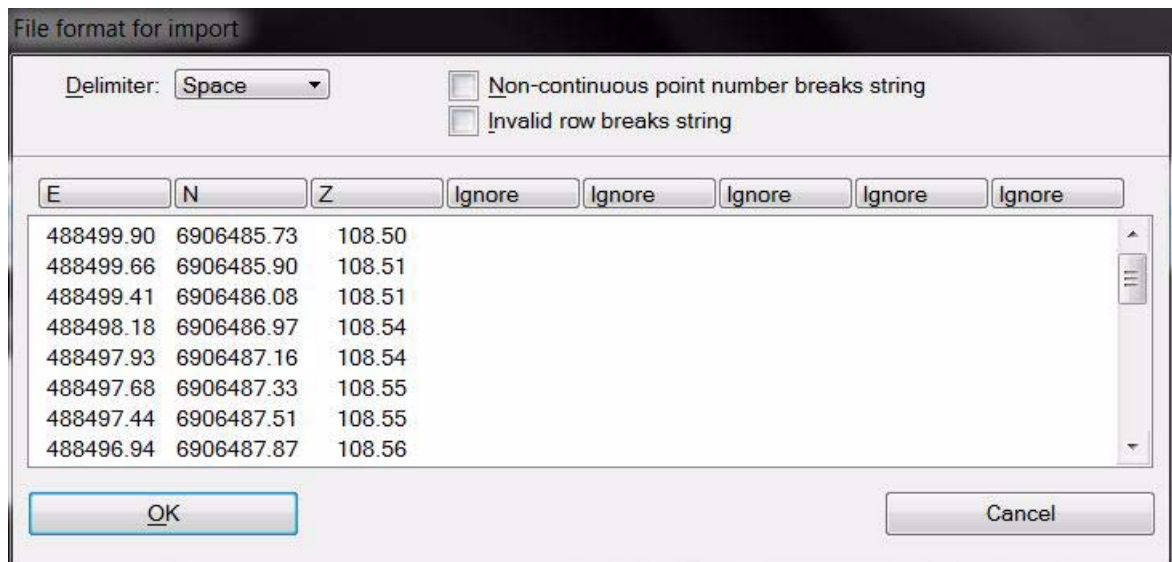
### ➤ To import points from a text file

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.

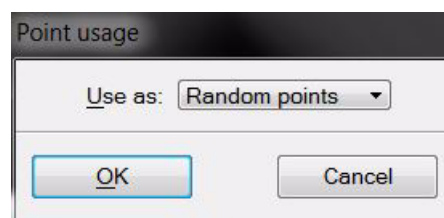
This opens the **File format for import** dialog:



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. Continue as described for **Triangulate Survey** on page 50.

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: <b>Space</b> , <b>Comma</b> , or <b>Semicolon</b> .
Non-continuous point number breaks string	If on, breaklines are splitted into separate chains if the point number increment is bigger than 1.
Invalid row breaks the string	If on, breaklines are splitted 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: <ul style="list-style-type: none"> <li>• <b>Ignore</b> - the field is ignored during import.</li> <li>• <b>E, N, Z</b> - easting, northing, elevation coordinates.</li> <li>• <b>Code</b> - text string. Defines, for example, the type of a breakline element.</li> <li>• <b>Number</b> - point number.</li> <li>• <b>String</b> - chain number. Defines, for example, contineous breakline elements.</li> <li>• <b>Pen up</b> - marks the end of a chain if it is not 0.</li> </ul>

## 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 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** on page 285 for the code definitions.

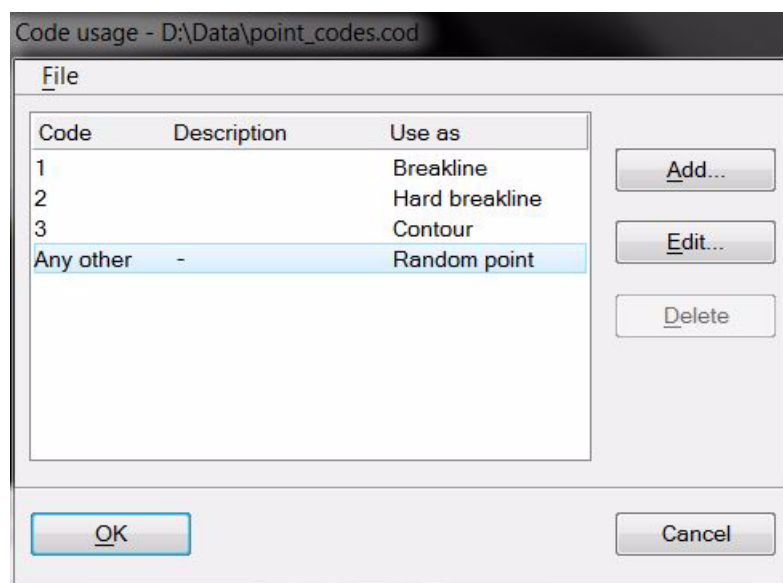
### ➤ To import points from a binary file

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. Continue as described for **Triangulate Survey** on page 50.

A new surface model is created from the xyz binary file or the points are added to an existing surface model.

## Import / Lattice file

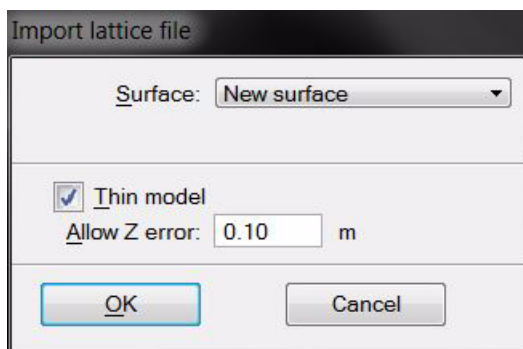
*Not Lite*

**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** on page 193.  
A new surface model is created from the lattice file.

Setting:	Effect:
Surface	Surface to be created from the lattice file. Always <b>New surface</b> .
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.
Allow Z error	Tolerance for leaving out points from the original lattice file. The software creates the final surface model file within the given tolerance value. This is only active if <b>Thin model</b> 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.

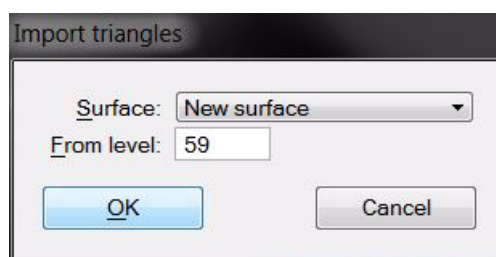
## Import / Triangles

**Triangles** command creates a new surface model from triangle elements drawn in the design file. This can be used, for example, to import a triangulated surface model created by another application.

➤ **To import triangles from the design 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. Continue as described for **File / New surface** on page 193.

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 <b>New surface</b> .
From level	Level in the design file on which the triangle elements are located.

## 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, 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 <b>Format</b> is set to <b>Tekla</b> .
Codes	Feature codes for different surface point types in the output file. This is only active if <b>Format</b> is not set to <b>XYZ Text</b> .

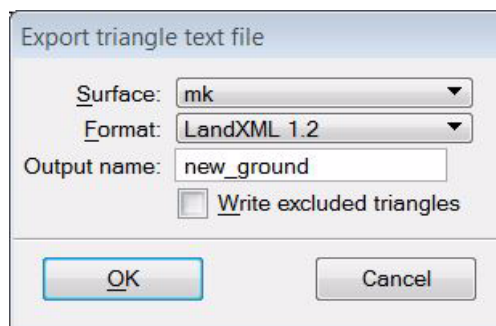
## Export / Triangle text file

**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 Point & edges text file, LandXML 1.0/1.2, 4ce DOT, GMS, SMS / WMS, ICC, Moss triangulation, 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.
ID	Surface identifier for <b>GMS</b> triangle files.
Output name	Name of the surface model written into <b>LandXML 1.0</b> and <b>LandXML 1.2</b> files.
Write excluded triangles	If on, excluded triangles are written into <b>LandXML 1.2</b> 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 <b>WorldToolKit NFF</b> files.
Output textures	If on, domain-related texture information is written into <b>WorldToolKit NFF</b> files.

## Export / Lattice file

*Not Lite*

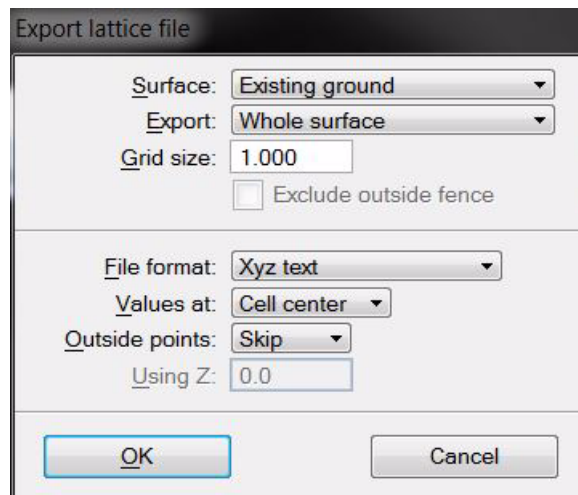
**Lattice file** command creates a lattice model file from a surface model. Lattice models are representations of a surface using a grid structure of points. The lattice model files include regularly distributed points at constant intervals in both, X and Y directions. Common storage formats are text 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 design 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.

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.

Setting:	Effect:
Export	<p>Area covered by the output file:</p> <ul style="list-style-type: none"> <li>• <b>Whole surface</b> - the complete surface model is exported.</li> <li>• <b>Rectangle</b> - the surface model area for export is defined by a orthonormal rectangle.</li> <li>• <b>Rotated rectangle</b> - the surface model area for export is defined by a rotated rectangle. This is only active if <b>File format</b> is set to <b>Intergraph GRD</b>.</li> <li>• <b>Selected rectangle(s)</b> - the area defined by selected rectangle(s) is exported. This is only active if at least one rectangular shape is selected.</li> </ul>
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 fence are treated as outside points. This is only active if a MicroStation fence is drawn or if a shape element is selected.
File format	<p>Format of the output file. Available formats are:</p> <ul style="list-style-type: none"> <li>• ArcInfo Grid, Intergraph GRD</li> <li>• Disimp raster format</li> <li>• GeoTIFF formats, Shaded GeoTIFF</li> <li>• Japanese DMF and LEM</li> <li>• Raw image formats</li> <li>• Surfer ASCII and binary</li> <li>• XYZ text</li> </ul> <p>Many of the additional settings for the lattice files depend on the selected format for the output files.</p>
Values at	<p>Defines the exact location of the lattice point coordinates:</p> <ul style="list-style-type: none"> <li>• <b>Cell center</b> - the coordinates of the grid cell center are stored.</li> <li>• <b>Cell corner</b> - the coordinates of the grid cell corner are stored.</li> </ul> <p>For some output file formats only one option is available.</p>
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 <b>GeoTIFFs</b> .
Write coordinate block	If on, positional information is written into the structure of <b>Intergraph GRD</b> files.
Byte order	Defines the byte order for <b>Raw</b> image formats related to the processor type: <b>Intel</b> or <b>Motorola</b> .
Outside points	<p>Defines for <b>XYZ text</b> files how to handle grid points outside the lattice model area where no elevation value can be derived from the source data:</p> <ul style="list-style-type: none"> <li>• <b>Skip</b> - outside grid points are not written into the text file.</li> <li>• <b>Output</b> - outside grid points are written into the text file using the constant value given in the <b>Using Z</b> field.</li> </ul>



Setting:	Effect:
File naming	<p>Defines the file naming method if several output files are created:</p> <ul style="list-style-type: none"> <li>• <b>Enter name for each</b> - the software asks you to enter a name for each lattice file that is created.</li> <li>• <b>Selected text elements</b> - the software uses a text element placed and selected inside the rectangle(s) as file name.</li> </ul> <p>This is only active if <b>Export</b> is set to <b>Selected rectangle(s)</b>.</p>
Directory	<p>Directory for storing the output file(s). This is only active if <b>File naming</b> is set to <b>Selected text elements</b>.</p>
Extension	<p>File extension for the output file(s). This is only active if <b>File naming</b> is set to <b>Selected text elements</b>.</p>

## Export / Raster image

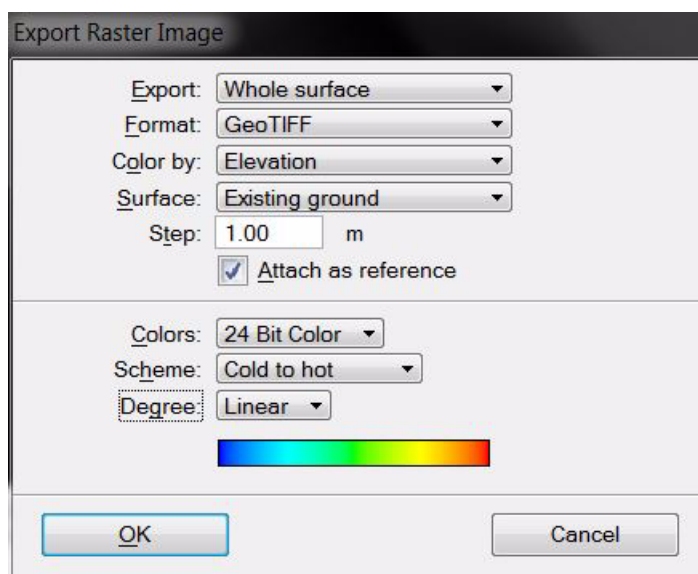
Not Lite

**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 (grey-scale).

### ➤ To export a raster image:

1. (Optional) Draw and select a rectangle that defines the area of the output file.
1. (Optional) Select a surface model from the list in the **Surfaces** window.
2. Select **Raster image** command from the **File / Export** menu.

This opens the **Export Raster Image** dialog:



3. Define settings and click OK.

TerraModeler computes the pixel colors and then opens another **Export Raster Image** dialog, a standard dialog for saving files.

4. Enter a name for the output file and click OK.

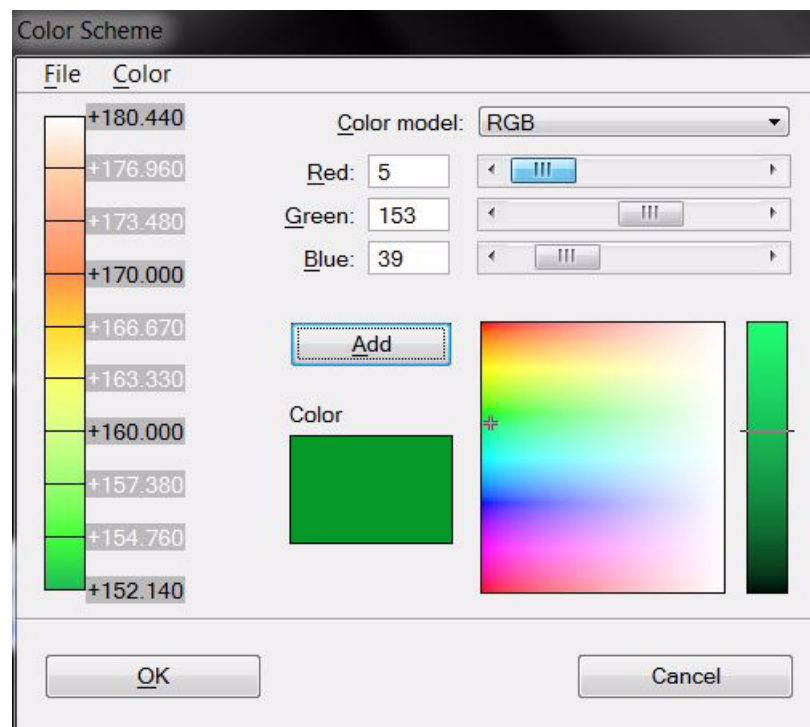
A raster image is created.

Setting:	Effect:
Export	Area covered by the output file: <ul style="list-style-type: none"> <li>• <b>Whole surface</b> - the complete surface model is exported.</li> <li>• <b>Selected rectangle</b> - the area defined by a selected rectangle is exported. This is only active if a rectangular shape is selected.</li> </ul>
Format	File format of the output file: <b>Windows BMP</b> or <b>GeoTIFF</b> .
Color by	Attribute used for coloring the output file: <ul style="list-style-type: none"> <li>• <b>Elevation</b> - coloring based on the elevation values of a surface model.</li> <li>• <b>Vertical difference</b> - coloring based on the vertical difference between two surface models.</li> <li>• <b>Shortest distance</b> - coloring based on the shortest distance between two surface models.</li> <li>• <b>Shaded surface</b> - coloring based on lightning conditions.</li> </ul>

Setting:	Effect:
Surface(s)	Name of the surface(s) to be used for output file creation. If color by is set to <b>Vertical difference</b> or <b>Shortest distance</b> , 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 MicroStation ( <i>not MicroStation V8i</i> ).
Colors	Color depth of the output raster image: <ul style="list-style-type: none"> <li>• <b>24 Bit Color</b> - true-color image.</li> <li>• <b>256 Colors</b> - 256 colors image.</li> <li>• <b>Grey scale</b> - 8 bit grey scale image.</li> </ul>
Scheme	Type of coloring scheme: <ul style="list-style-type: none"> <li>• <b>Cold to hot</b> - varies from blue color hues for low values to red color hues for high values.</li> <li>• <b>Hot to cold</b> - varies from red color hues for low values to blue color hues for high values.</li> <li>• <b>Selected colors</b> - you can define an own coloring scheme. Click on the <b>Define</b> button in order to define the <b>Color scheme for raster image files</b>.</li> </ul>
Degree	Determines how the color hues are distributed in <b>Cold to hot</b> and <b>Hot to cold</b> color schemes: <ul style="list-style-type: none"> <li>• <b>Hot, Warm</b> - color hues are shifted to the red-yellow color range.</li> <li>• <b>Linear</b> - color hues are linearly distributed.</li> <li>• <b>Cool, Cold</b> - color hues are shifted to the blue-green color range.</li> </ul>

### 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.

➤ **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.

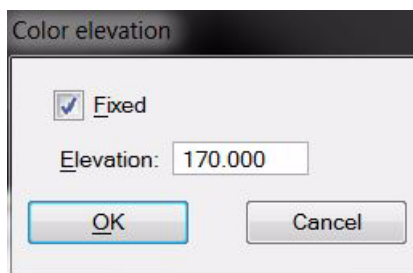
3. Click **Add** in order to add the selected color to the color scheme.

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.

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.

## Export / Graphical elements

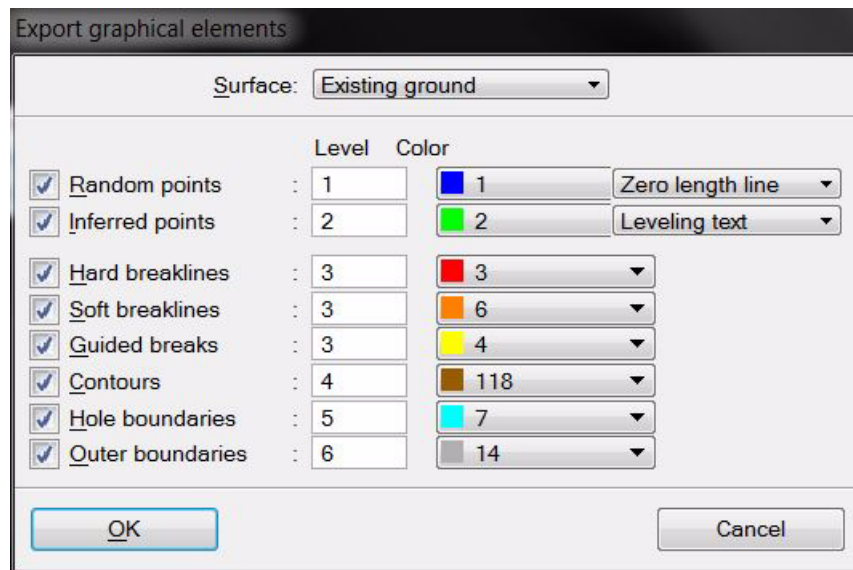
**Graphical elements** command draws different types of surface model points and lines on given levels into the design file. Random and inferred points can be drawn as leveling text elements, as zero length lines (points), or as X character text elements. Breaklines, contours, hole boundaries, and outer 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 design 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 design 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 design file.

The command may prove 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 design file.
2. Switch all other levels off in a view.
3. Move the graphical elements using MicroStation tools.
4. Delete the old surface model.
5. Use **Triangulate View** tool to create a new surface model from the shifted graphical elements.

## 18 Lattice Database

*Not Lite*

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 design 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** - have to be orthonormal.

To:	Choose menu command:
Draw boundaries for the lattice database area	<b>Lattice database / Draw boundaries</b>
View lattice database elevation values	<i>View Elevation</i> tool
Draw contours for lattice database files into the design file	<b>Lattice database / Draw contours</b>
Triangulate lattice database points	<b>Lattice database / Import fence contents</b>

## Lattice database / Draw boundaries

*Not Lite*

**Draw boundaries** command draws the boundaries of all lattice files in the lattice file directory into the design 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 MicroStation. The name of each lattice file is placed as text element at the center of the corresponding rectangle using active text setting in MicroStation.



### To draw lattice file boundaries into the design 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 design file showing the area covered by each file. An information dialog is displayed that shows for how many lattice files the boundaries are drawn.

## Viewing lattice database elevation

*Not Lite*

*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.

## Lattice database / Draw contours

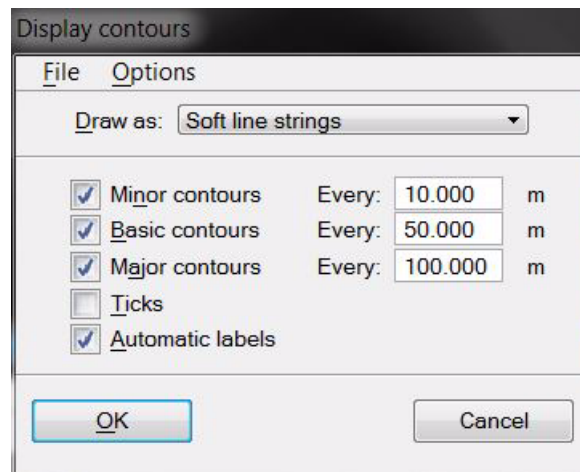
*Not Lite*

**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 design file.



## Lattice database / Import fence contents

Not Lite

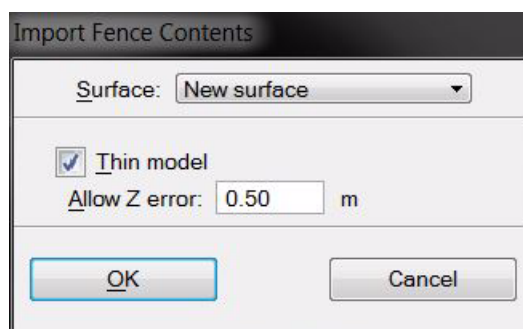
**Import fence contents** command provides an option for creating a triangulated surface model from lattice database points inside a fence. The fence may overlap several individual lattice files. A triangulated surface model enables the addition of new elevation data, such as random points or breaklines. 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 MicroStation fence.
2. Select **Import fence contents** command from the **Utility / Lattice database** pulldown menu.

This opens the **Import Fence Contents** dialog:



3. Define settings and click OK.

This opens the **Surface settings** dialog. Continue as described for **File / New surface** on page 193.

The software processes all lattice files inside the fence and creates a triangulated surface model from the points.

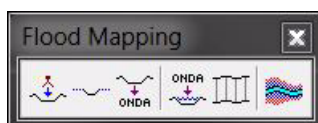
Setting:	Effect:
Surface	Surface to be created from the lattice file. Always <b>New surface</b> .
Thin model	If on, the software leaves out less relevant points from the lattice file in order to reduce the file size of the surface model file.
Allow Z error	Tolerance for leaving out points from the original lattice file. The software creates the final surface model file within the given tolerance value. This is only active if <b>Thin model</b> is switched on.







# 19 Flood Mapping

## Flood Mapping tool box

Not Lite

The **Flood Mapping** tool box is used to generate validated river cross sections from Eeby survey files or from cross section elements drawn in the design file. There are also tools to output the river cross sections to calculation software, and display the results of flood calculations. The **Flood Mapping** tool box belongs to the TerraModeler **Extensions** and can be opened from the **Tools pulldown menu** in the **Surfaces** window.



To:	Use:
Import surveyed river sections	 <i>Import River Sections</i>
Modify or extend river sections	 <i>Edit River Sections</i>
Output sections to calculation software	 <i>Export River Sections</i>
Import resulting water levels	 <i>Input Water Elevations</i>
Display the area covered by sections	 <i>Draw River Section Boundaries</i>
Display flood extents	 <i>Display Flood</i>

## Import River Sections

*Not Lite*



*Import River Sections* tool reads in cross section survey files and draws these into the design file. The input survey files have to be in Eeby format.

TerraModeler displays the imported river sections in temporary mode and opens the **River cross sections** dialog. The dialog offers a number of tools for validating and modifying section information. After modifications, the sections can be drawn permanently into the design file.

The river cross sections can be drawn as cell elements into the design file. Each cell contains a line element representing the cross section and a text element as section label. The label includes the section name and description.

The settings for the import of river cross section files and the symbology of the section display can be defined in **River sections / Coefficients category**, **River sections / Eeby files category**, and **River sections / Symbology category** of the TerraModeler **Settings**.



### To import river sections:

1. Select the *Import River Sections* tool.

This opens the **Import river cross sections** dialog, a standard dialog for opening files.

2. Add the desired file(s) to the list of files to process and click **Done**.

TerraModeler reads in the selected files, draws the cross sections in temporary mode and opens the **River cross sections** dialog.

3. Validate and modify the cross section information using the commands and buttons in the **River cross sections** dialog. See Section **Edit River Sections** on page 252 for detailed information.

When you close the dialog, another dialog appears which lets you confirm whether you want to draw the cross sections permanently into the design file or not.

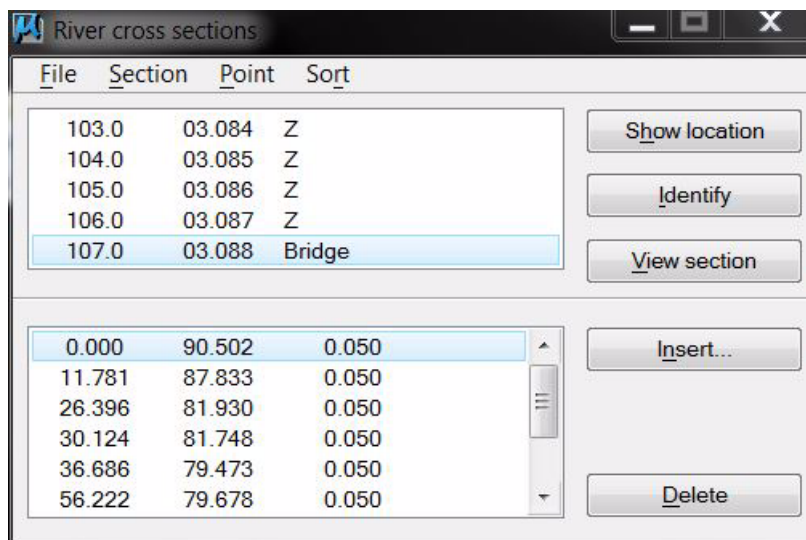
## Edit River Sections

Not Lite



*Edit River Sections* tool lets you modify river cross sections which were imported and drawn into the design file. The tool can also be used to create new river cross sections from section lines drawn into the design file by other means.

When river cross sections are imported or modified, the **River cross sections** dialog is opened. The dialog provides commands and buttons for working on the cross sections.



The upper list of the dialog shows the river cross sections with their station number, name, and description. In the lower list the vertices of the selected river cross section are listed. For each vertex, the xy distance from the section's origin point, the elevation, a label, and the friction coefficient of the channel are shown.

With the **Show location** button, a selected section from the list can be highlighted in a design file view. Select a section from the list, click on the **Show location** button and move the mouse pointer inside a view. The origin point of the section is highlighted. With a data click, the origin point is centered in the view.

The **Identify** button selects a section in the list by clicking close to the section's location in a view. Click on the **Identify** button and move the mouse pointer inside a view. The section closest to the mouse pointer is highlighted. Enter a data click in order to select the section in the list.

The **View section** button can be used to display a river cross section in a section view. Select a section from the list, click on the **View section** button, and click inside a view. TerraModeler rotates the view and adjusts the display depth in order to show the river section in the view.

To:	Choose menu command:
Create a formatted output file of a selected river cross section	<b>File / Output to Onda</b>
Close the <b>River cross sections</b> dialog	<b>File / Close</b>
Edit a selected river cross section	<b>Section / Edit</b>
Move the origin of a river cross section	<b>Section / Move xy origin</b>
Change the direction of a river cross section	<b>Section / Rotate</b>
Extend a river cross section from its end points	<b>Section / Extend</b>
Create a new river cross section from a linear element	<b>Section / Create from element</b>
Modify the start station number of an alignment	<b>Section / Set stations</b>
Delete a selected river cross section	<b>Section / Delete</b>
Edit a selected river cross section point	<b>Point / Edit selected</b>

To:	Choose menu command:
Delete a selected river cross section point	<b>Point / Delete selected</b>
Insert a river cross section point	<b>Insert button</b>
Delete a river cross section point	<b>Delete button</b>
Sort the list of river cross sections by station number	<b>Sort / Increasing station</b> <b>Sort / Decreasing station</b>

➤ **To modify a single river cross section:**

1. Select the *Edit River Sections* tool.
2. Identify the river cross section to edit with a data click.

This opens the **River cross sections** dialog displaying the selected section.

3. Modify the cross section information using the tools in the dialog.
4. Close the dialog when you are done.

➤ **To modify several river cross sections:**

1. Place a fence around the river cross sections you want to modify.
2. Select the *Edit River Sections* tool.
3. Accept the fence contents with a data click.

This opens the **River cross sections** dialog displaying the sections found within the fence.

4. Modify the cross sections using the tools in the dialog.
5. Close the window when you are done.

## File / Output to Onda

**Output to Onda** command creates an ONDA output file of a selected river cross section.

➤ **To output a river cross section to ONDA:**

1. Select the river cross section from the list.
2. Select **Output to Onda** command from the **File** menu.

This opens the **Output to ONDA** dialog, a standard dialog for saving the output file.

3. Define a location and a name for the output file.
4. Click OK.

This writes the selected river cross section into a formatted text file.

- ✍ Use the *Export River Sections* tool in order to export multiple river sections into an ONDA output file.

## Section / Edit

**Edit** command lets you modify the informational and locational attributes of a selected river cross section.

➤ **To edit the attributes of a river cross section:**

1. Select the river cross section in the list.
2. Select **Edit** command from the **Section** menu.

This opens the **River cross section** dialog:

**River cross section**

**Section information**

Station: 107.000  
Name: 03.088  
Description: Bridge

**Location**

Bearing: 43.487  
Origin X: 487287.708  
Y: 6904518.825  
Z: 90.502

OK Cancel

3. Modify the values for the attributes that need to be changed.
4. Click OK.

Setting:	Effect:
Station	Station number of the river cross section.
Name	Free text string used as name attribute of the river cross section.
Description	Free descriptive text string.
Bearing	Direction angle of the river cross section. North direction is zero and angles increase clockwise.
Origin X, Y, Z	Easting, northing, and elevation coordinates of the origin point of the river cross section.

### Section / Move xy origin

**Move xy origin** command can be used to correct locational errors in the xy coordinates of a river cross section. A cross section is moved by its origin point.

➤ **To move a river cross section in xy:**

1. Select a river cross section in the list.
2. Select **Move xy origin** command from the **Section** menu.
3. Move the mouse pointer inside a top view.

The new location of the section is displayed dynamically at the mouse pointer location.

4. Enter a data click in order to define the new origin point of the river vsection.

### Section / Rotate

**Rotate** command is used to modify the direction of a river cross section. The direction of a section is defined as angle clockwise off from north direction around the origin point.

Alternatively, the angle can be modified by adjusting its value in the **River cross section** dialog.

➤ **To modify a river cross section direction:**

1. Select a river cross section in the list.
2. Select **Rotate** command from the **Section** menu.
3. Move the mouse pointer inside a top view.

The new location of the section is displayed dynamically at the mouse pointer location.

4. Enter a data click in order to define the new section location.

OR

2. Select **Edit** command from the **Section** menu.

This opens the **River cross section** dialog.

3. Enter a new **Bearing** angle and click OK.

The river cross section is drawn in the new direction starting from its origin point.

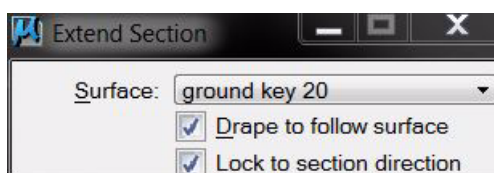
## Section / Extend

**Extend** command can be used to extend a river cross section based on a surface model of the ground. The command lets you insert points to either end of the section so that it accurately follows the ground surface elevations.

### ➤ To extend a river cross section:

1. Select a river cross section in the list.
2. Select **Extend** command from the **Section** menu.

This opens the **Extend Section** dialog:



3. Select a **Surface** model.
4. Define additional settings.
5. Enter a new end point for the river section.

You can continue with step 4 or click reset when you are done.

Setting:	Effect:
Surface	Surface model used to adjust the elevations of the new section points.
Drape to follow surface	If on, the section line between the new end point and the previous end point of the section is draped to the surface model elevations.
Lock to section direction	If on, the cross section can be extended only in the direction of the section.

- ✍ Points that are added as extension to a river cross section get the label “X” in the list of river cross section points.

## Section / Create from element

**Create from element** command creates a new river cross section from a linear element that represents the section. This can be used, for example, if no river cross section survey file is available or if not all cross sections are included in the survey file.

### ➤ To create a new river cross section from a line element in the design file

1. Draw a cross section of a river as linear element into the design file.
2. Select the *Edit River Sections* tool.

This opens the **River cross section** dialog.

3. In the dialog, select **Create from element** command from the **Section** menu.
4. Identify the linear element that represents the cross section.

This opens the **River cross section** dialog:

5. Define **Station**, **Name**, and **Description** for the cross section.
6. (Optional) Correct the values for the **Bearing** angle and the coordinates of the **Origin**.
7. Click OK.

You can continue with step 4.

The new cross section is added to the list in the **River cross sections** dialog and drawn into the design file. The cross section is drawn as cell element using level and symbology settings as defined in **River sections / Symbology category** of the TerraModeler **Settings**.

## Section / Set stations

**Set stations** command sets the station number for the river cross sections according to distances along an alignment element. The alignment can be any linear element, for example, a river center line. The stations of the cross sections are calculated from the start point of the alignment. For example, if the first cross section crosses the alignment at a distance of 25 m from the start point, the station is set to 25.000.

### ➤ To set stations of river cross sections along an alignment element:

1. Select the alignment element.
2. Select **Set stations** command from the **Section** menu.

This opens the **Set section stations** dialog:

3. Define the station value at the **Alignment start** point.
4. Click OK.

This calculates the station values for the river cross sections that cross the alignment. The station values are calculated as the distance along the alignment from the alignment's start point to the cross section plus the alignment's start point station value.



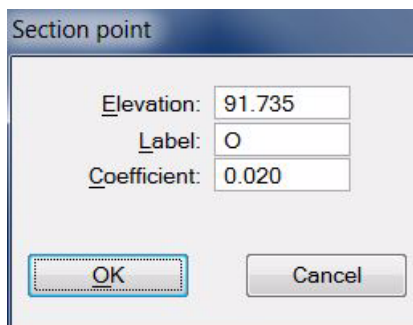
## Point / Edit selected

**Edit selected** command lets you modify the attribute information of a river cross section point.

### ➤ To edit a river cross section point:

1. Select a river section and a river section point in the lists.
2. Select **Edit selected** command from the **Point** menu.

This opens the **Section point** dialog:



3. Define settings and click OK.

The attributes of the selected section point are changed.

Setting:	Effect:
Elevation	Elevation value of the river cross section point.
Label	Label of the river cross section point.
Coefficient	Friction coefficient of the river cross section point.

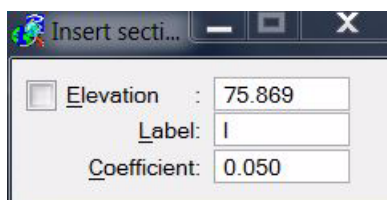
## Insert

**Insert** button can be used to add intermediate river cross section points along the section line. The xy locations of the new points are defined by a data click. The elevation can be fixed to a given value or adjusted to a surface model.

### ➤ To insert a new river cross section point:

1. Select a river cross section in the list.
2. Click on the **Insert** button.

This opens the **Insert section point** dialog:



3. Define settings.
4. Move the mouse pointer inside a view.

A preview of the inserted point is displayed at the mouse pointer location.

5. Define the location of the new section point with a data click.

You can continue with inserting new points for the selected cross section.

Setting:	Effect:
Elevation	If on, the elevation value of the inserted river cross section point is fixed to the given elevation.
Label	Label of the river cross section point.
Coefficient	Friction coefficient of the river cross section point.

-  You can use the *View Elevation* tool with the **Points on surface** lock in order to insert new section points that are adjusted in elevation to a surface model.

## Delete

**Delete** button can be used to remove river cross section points from the section line using the mouse.


### To delete a river cross section point:

1. Select a river cross section in the list.
2. Click on the **Delete** button.
3. Move the mouse pointer inside a view.

The cross section point closest to the mouse pointer location is highlighted.

4. Define the section point to be deleted with a data click.

This removes the point from the section line.

-  Another option to delete cross section points is the **Delete selected** command from the **Point** pulldown menu. The command lets you delete a section point that is selected in the list.

## Export River Sections

*Not Lite*



*Export River Sections* tool creates an ONDA output file of river sections. You can output all cross sections drawn to the design file or use a fence to output only cross sections inside the fence area.



### To output river sections to ONDA:

1. (Optional) Place a fence around the river sections you want to output.
2. Select the *Export River Sections* tool.
3. (Optional) Accept the fence contents with a data click.

This opens the **Output to ONDA** dialog, a standard dialog for saving the output file.

4. Define a location and a name for the output file.
5. Click OK.

This writes the river cross sections into a formatted text file.

## Input Water Elevations

Not Lite



*Input Water Elevations* tool defines water elevation values for river cross sections. It can read in water elevation data produced by ONDA. The data basically contains water elevation values for the river cross sections used in the calculation. TerraModeler tries to find a matching cross section in the design file for each elevation value. Alternatively, the water elevation values can be defined manually.

The tool draws a line string along the river cross section at the water elevation. The line strings can be used, for example, to create a surface model of the water elevations which might represent a certain flood level. If the elevation value is 0.0, no water elevation line is drawn for this section.



### To define water elevation values for river cross sections:

1. (Optional) Place a fence around the river sections for which to define water elevations.
2. Select the *Input Water Elevations* tool.
3. (Optional) Accept the fence contents with a data click.

This opens the **Input Water Elevations** dialog:

Section	Description	Water
03.084	Z	95.20
03.086	Z	95.25
03.085	Z	95.18
03.087	Z	95.15
03.088	Bridge	95.20

Water:

OK Cancel

4. Select **Import results** command from the **File** pulldown menu.

This opens the **Import water elevation results** dialog, a standard dialog for opening files.

5. Locate and select the ONDA file and click **Open**.

TerraModeler reads in the elevation data.

OR

4. Select a river cross section in the list.
5. Define an elevation value in the **Water** field.
6. Click the <Tab> key in order to apply the value for this section and to jump to the next section.

When the **Input Water Elevations** dialog is closed with OK, the software draws the water elevation line strings along the respective river cross sections using active level and symbology settings.

You can use *Triangulate View*, *Triangulate Elements*, or *Insert Random Elements* tool to create a surface model of the water elevations.



Default exclusion logic does not work very well when creating a surface model from river cross sections. The **Exclude outer boundaries** field in the **Triangulate surface** dialog controls the exclusion logic. For river sections, you should select **By key-in length** option and enter a **Maximum length** value, such as 300.0 meters.

## Draw River Section Boundaries

*Not Lite*



*Draw River Section Boundaries* tool generates a 3D shape element around the area covered by river sections. The shape is drawn using active level and symbology settings in MicroStation.



### **To draw a shape around the river sections:**

1. Select the **Draw River Section Boundaries** tool.

The boundary shape is drawn and an information dialog is displayed that shows the number of river cross sections enclosed by the shape.

## Display Flood

Not Lite



*Display Flood* tool computes and displays the extent of flooding. It can compare several flood models (5 year, 10 year, 25 year, ...) with a ground surface model and display each using a specific color.

The calculation is based on a grid method. You can control the accuracy and speed of calculation by defining the grid size.



### To display flood extents:

1. Place a fence or select a polygon around the area to compute.
2. Select the *Display Flood* tool.

This opens the **Display flood** dialog:

Surface	Draw	Clr	Wgt	Sty
Flood	X	39	2	0

3. Select a ground **Surface** model that is used as reference for the flood calculation.
4. Define other settings and click OK.

TerraModeler compares each selected flood surface model with the ground surface model and draws the calculated grid according to the display settings.

Setting:	Effect:
Surface	Name of the surface model to be used as reference for calculating the flood extend.
Mode	Display method for presenting the calculation results graphically in the design file: <ul style="list-style-type: none"> <li>• <b>Draw temporarily</b> - (not MicroStation V8i) draws temporary elements.</li> <li>• <b>Write to file</b> - draws permanent elements on the active level.</li> </ul>
Draw as	Elements that are drawn into the design file: <b>Lines</b> or <b>Filled shapes</b>
Step size	Defines the grid size for the calculation. A small step produces more accurate results but the calculation takes longer.
Draw	If on, the selected flood surface model in the list on the left side of the dialog is displayed.
Color	Color of the selected flood surface. Uses the active MicroStation color table.




Setting:	Effect:
Weight	Line weight of the selected flood surface. Uses MicroStation line weights.
Style	Line style of the selected flood surface. Uses MicroStation line styles.

# 20 Lifting 2D Contours

## 2D Contours tool box

The **2D Contours** tool box is used to lift 2D contour elements into 3D or to validate elevations of linear elements. The **2D Contours** tool box belongs to the TerraModeler **Extensions** and can be opened from the **Tools pulldown menu** in the **Surfaces** window.



To:	Use:
Lift 2D contours using contour labeling	 <i>Lift 2D Contours</i>
Set elevation of a contour element	 <i>Set Contour Elevation</i>
Validate elevations of linear elements	 <i>Check Linear Elevations</i>



## 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.

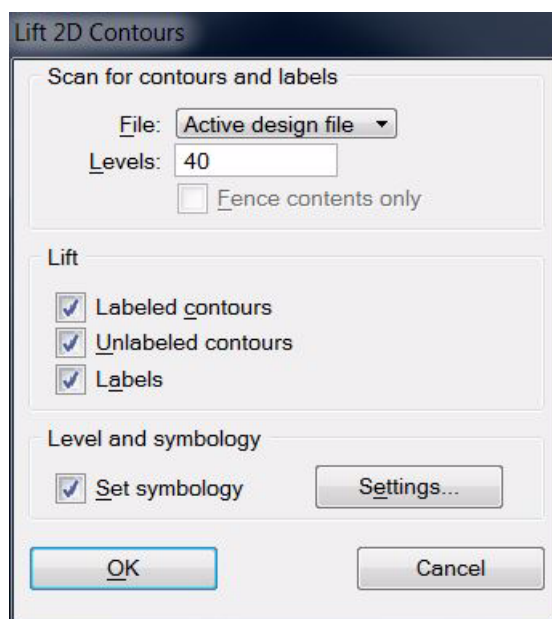
The tool scans the given levels in either the active design file or in attached reference files for text elements and linear elements. It tries to find a matching contour elevation label for each contour chain. A match is found, if the start or the end point of a contour chain is close to an elevation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



### To lift 2D contours into 3D:

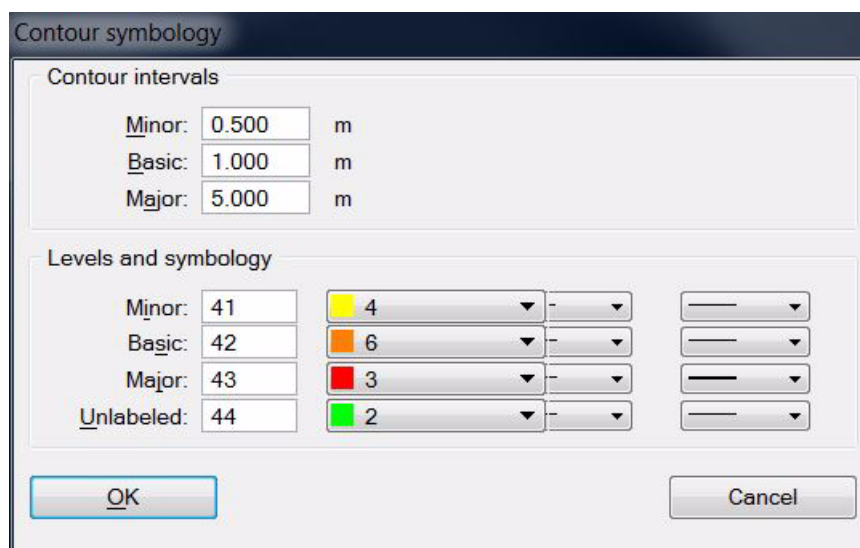
1. Select the *Lift 2D Contours* tool.

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 design file using the given contour symbology.

Setting:	Effect:
File	Files from which to search for text and linear elements: <ul style="list-style-type: none"> <li>• <b>Active design file</b> - scan the active design file.</li> <li>• <b>Reference files</b> - scan all reference files with <b>Locate</b> setting on in MicroStation Reference file manager.</li> </ul>
Levels	List of levels from which to scan for elements. For example: <ul style="list-style-type: none"> <li>• 61 - scan level 61.</li> <li>• 15,21-24 - scan levels 15, 21, 22, 23 and 24.</li> </ul>
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 the <b>Settings</b> button.

## Set Contour Elevation

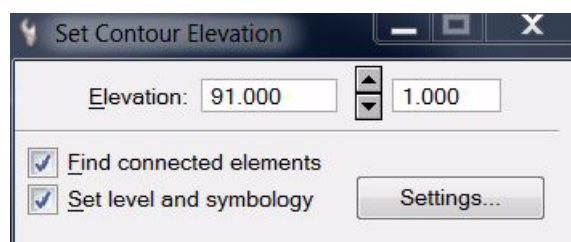


*Set Contour Elevation* tool lifts a linear element to a given elevation. It is designed to be used for lifting single two-dimensional contour elements to a correct elevation. It can be applied to line, shape, and text elements. The original elements are moved to their new elevation.

➤ **To set elevation of contour element(s):**

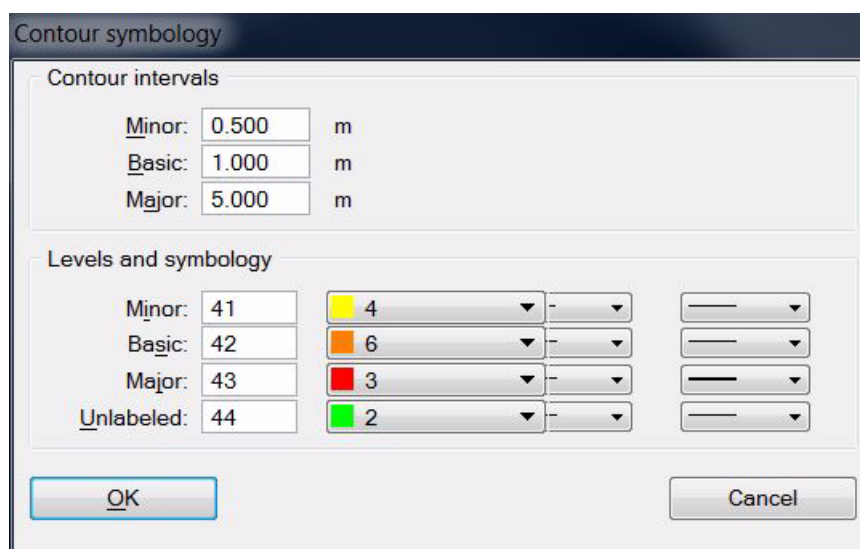
1. Select the *Set Contour Elevation* tool.

The **Set Contour Elevation** dialog opens:



2. Set an elevation value in the **Elevation** field.
3. Define additional settings. 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.
7. Accept the element.

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 <b>Settings</b> button.

## Check Linear Elevations



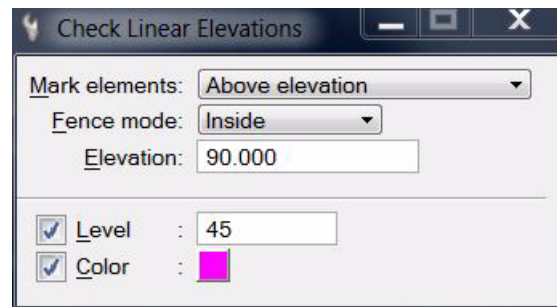
*Check Linear Elevations* tool validates the elevations of linear elements defined by a fence. It marks invalid elements by modifying level and color of the elements.



### To check elevations of linear elements:

1. Place a fence to define the area to process.
2. Select the *Check Linear Elevations* tool.

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



3. Define settings.
4. Accept the fence contents 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: <ul style="list-style-type: none"> <li>• <b>Above elevation</b> - mark elements which extend above a given elevation.</li> <li>• <b>Below elevation</b> - mark elements which extend below a given elevation.</li> <li>• <b>With non-uniform elevations</b> - mark elements which have vertices at more than one elevation.</li> </ul>
Fence mode	Determines what elements are validated. MicroStation fence modes are used: <ul style="list-style-type: none"> <li>• <b>Inside</b> - only elements that are fully inside a fence.</li> <li>• <b>Overlap</b> - elements that overlap with a fence.</li> <li>• <b>Clip</b> - the part of an element that is overlapped by a fence, other parts are not validated.</li> <li>• <b>Void</b> - only elements that are fully outside a fence.</li> <li>• <b>Void overlap</b> - elements that do not overlap with a fence .</li> <li>• <b>Void clip</b> - the part of an element that is not overlapped by a fence, overlapped parts are not validated.</li> </ul>
Elevation	Defines the elevation limit for valid elements. This is only active if <b>Mark elements</b> is set to <b>Above elevation</b> or <b>Below elevation</b> .
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.
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 in MicroStation.

# Programming Interface

## 21 MDL Public Functions

---

TerraModeler has a number of functions that can be called by other MDL applications. A third party can write an application which obtains elevation values from a TerraModeler surface model or which drives operations that TerraModeler performs.

TerraModeler publishes public functions with `mdlCEXpression_symbolPublish()`. This makes the functions available for any MDL applications.

Many Terra applications use this method of interaction between different modules. For example, TerraStreet calls `FnSurfaceZ()` when labeling a profile with surface elevations.



## Function Prototypes

```

/*-----
   Open a surface model with file location and name FileName.
*/

int      FnModelOpenSurface( char *FileName) ;

/*-----
   Close all open surface models.
*/

int      FnModelCloseSurfaces( void) ;

/*-----
   Return number of triangulated surface models.
*/

int      FnSurfaceCount( void) ;

/*-----
   Fill structure with information about triangulated surfaces.
   S points to a table of SurfInfo structures. Calling application
   should have allocated it to hold at least FnSurfaceCount()
   items.

   Return number of items filled in the table (=FnSurfaceCount()).
*/

int      FnSurfaceList( SurfInfo *S) ;

typedef struct {
    int      Id ;                /* Id */
    int      DrawInProfile ;     /* Draw in profiles? */
    char      Name[32] ;         /* Name */
    int      Type ;              /* Type id */
    char      Reserved[76] ;     /* For future development - ignore now */
} SurfInfo ;

/*-----
   Copy name of surface with id I into Str.

   Return 1 if id I was valid.
   Return 0 if id was invalid.
*/

int      FnSurfaceName( char *Str, int I) ;

/*-----
   Return profile color of surface with id I.
   Return 0 if id was invalid.
*/

int      FnSurfaceColor( int I) ;

/*-----
   Compute bounding cube for surface with id I.
   Mn is set to be the minimum x,y,z of the surface model.
   Mx is set to be the maximum x,y,z of the surface model.

   Return 1 if successful.
   Return 0 if failed.
*/

int      FnSurfaceMinMax( Dpoint3d *Mn, Dpoint3d *Mx, int I) ;

```



```

/*-----
Calculate elevation Pt->z at location Pt->x,Pt->y from surface
with id I.

The elevation can not be calculated if:
- the surface id is invalid
- the surface is empty
- the point is outside surface
- the point is inside an excluded triangle

Return 1 if successful.
Return 0 if the elevation could not be calculated.
*/

int      FnSurfaceZ( Dpoint3d *Pt, int I) ;

/*-----
Calculate elevation for an array of x,y points from surface
with id I.

Tbl is a pointer to an array of x,y points.
Cnt is the number of points in the array.

The calling application should initialize the z values
of the points to an invalid value such as -999999.
If the elevation of a x,y point can not be calculated,
the z value will remain unchanged.

Return 1 if successful.
Return 0 if the surface is invalid.
*/

int      FnSurfaceArrayZ( Dpoint3d *Tbl, int Cnt, int I) ;

/*-----
Drape an array of points over surface model with id I.

Tbl is a pointer to an array of x,y points.
Cnt is the number of points in the array.
I is the id of the surface model.

After the routine:
*Pnt points to a table of generated points.
*Val points to a table of characters which indicate
what point intervals are valid. Val[0] indicates
if point interval Pnt[0] - Pnt[1] is valid.

Return the number of points generated.
Return 0 if failed.
*/

int      FnDrapeArray( Dpoint3d **Pnt, char **Val, const Dpoint3d *Tbl, int Cnt, int I) ;

/*-----
Free the results from a call to FnDrapeArray().

Pnt points to a table of generated points.
Val points to a table of characters.

Return 0.
*/

int      FnDrapeFree( Dpoint3d *Pnt, char *Val) ;

```

```

/*-----
Drape descriptor Dp over surface model with id I.
FileNr is the file number of descriptor Dp.
Method determines what is generated for places where
the original descriptor is outside the surface.
If Method is
0 : Outside part will remain unchanged.
1 : Outside part will be drawn with active symbology.
2 : Outside part will not be drawn, thus resulting in a gap.

Create new descriptor as *Pp (Dp is not changed in the process).

Return 1 if successful.
Return 0 if failed.
*/
int      FnDrapeDscr( MSElementDescr **Pp, MSElementDescr *Dp,
                    int FileNr, int Method, int I) ;

/*-----
Has surface with id I been triangulated?
*/

int      FnModelExists( int I) ;

/*-----
Fill an option button with surface names. This function lets
an external application create an option button for selecting
a surface.

If P == 0, list all surfaces.
If P == 1, list all triangulated surfaces.
*/

int      FnSurfaceOption( RawItemHdr *Raw, int P) ;

/*-----
Fill option button with surface type names.

Return 1 always.
*/

int      FnSurfTypeOption( RawHdr *Raw) ;

/*-----
Create a new empty surface model into which points can be added
using FnAddPoints() or FnInsertPoints().

Type is the surface type as defined in Modeler user settings.
Calling application should use zero (normally ground) or let
the user select type from an option button (use FnSurfTypeOption).

Name is an optional name for the new surface. If Name == NULL,
surface type name is retrieved for the surface model name.

Return ID of the new surface (0,1,...).
Return -1 if failed (out of memory).
*/

int      FnCreateSurface( int Type, char *Name) ;

```

```

/*-----
Add random points to surface with id I. This routine can
be used to insert points to an existing surface or to create
a new surface. TerraModeler will always triangulate
the surface during this function.

Tbl points to an array of Cnt points.
Cnt is the number of points in Tbl.
If AskSet is true, ask triangulation settings.

In a result of this call,

Return true if points were inserted.
Return 0 if unsuccessful.
*/

int      FnInsertPoints( int I, int AskSet, Dpoint3d *Tbl, int Cnt) ;

/*-----
Add points of type Type to surface with id I.
TerraModeler will add these points to its point list
but will not immediately triangulate the new points.

This routine can be used to insert points to an existing
surface or to create a new surface.

Tbl points to an array of Cnt points.
Cnt is the number of points in Tbl.
Type is the point type:
    1 = random
    2 = breakline
    3 = contour
   10 = external 3D boundary
   11 = external 2D boundary
   20 = internal 3D boundary
   21 = internal 2D boundary

Return true if points were inserted.
Return 0 if unsuccessful.
*/

int      FnAddPoints( int I, Dpoint3d *Tbl, int Cnt, int Type) ;

/*-----
Triangulate surface with id I. The calling application will
typically call this function after calling FnAddPoints()
a number of times.

If AskSet is true, ask triangulation settings.

Return 1 if triangulation was completed.
Return 0 if unsuccessful.
*/

int      FnAddDone( int I, int AskSet) ;

/*-----
Retriangulate surface with id I after calls to FnAddPoints().

Return 1 if triangulation was completed.
Return 0 if unsuccessful.
*/

int      FnAddRetriangulate( int I) ;

```

```

/*-----
Add a linear element representing surface elevations to a
profile descriptor. This routine can be called by an
application creating a profile in which surface(s) are
drawn as line string type elements.

This routine uses surface's profile color and line style
when adding elements to the descriptor.

Dp is descriptor where elements will be added.
L defines the profile settings and the alignment.

Return 1 if drawn.
Return 0 if failed.
Return -1 if no need to draw:
    - surface is empty or
    - 'Draw in profile' setting is off
*/

int      FnProfileDscr( MSElementDescr *Dp, FnProf *L) ;

typedef struct {
    int      SurfId ;      /* Surface id or -3=all surfaces */
    int      Lvl ;         /* Level to draw on */
    Dscr      *Align ;     /* 2D alignment */
    double    AlignLen ;   /* Length of the alignment */
    double    ExaggZ ;     /* Horizontal scale / Vertical */
    double    RangeStart ; /* Elevation range from */
    double    RangeEnd ;   /* Elevation range to */
    Dpoint3d   Org ;       /* Lower left corner of range*/
    int      Scale ;       /* Horizontal scale */
} FnProf ;

/*-----
Find out grid elevation at location Pt->x, Pt->y.
This function searches the lattice database for a grid
file which covers the given location and reads the closest
elevation value.

Return 1 if Pt->z was successfully set.
Return 0 if point was outside grid data files.
Return -1 if no grid files were found.
*/

int      FnGridZ( Dpoint3d *Pt) ;

```

```

/*-----
Compute lattice elevation values from surface SurfId.
The calling application should:
1. Fill in values in TinLtc structure
2. If integer (uor) elevation values are needed:
   * allocate memory as T->ElevInt (CntX*CntY*sizeof(int))
   * initialize every value to an invalid value (such as 0x80000000)
3. If floating point (master unit (m)) elevation values are needed:
   * allocate memory as T->ElevInt (CntX*CntY*sizeof(double))
   * initialize every value to an invalid value (such as -999999.0)
4. If elevations from excluded triangles are needed:
   * allocate memory as T->Valid (CntX*CntY*sizeof(char))

This routine sets only those elevation values for which an elevation
from SurfId can be calculated. Lattice points outside the surface will
not be affected.

If the calling application provider T->Valid as non-NULL, TerraModeler
will calculate elevations from excluded triangles as well and fill
in T->Valid[] to indicate which point are:

    0 outside surface
    1 inside an excluded triangle
    2 inside a valid triangle

Return 1 on success.
Return 0 if SurfId is invalid.
*/

int FnComputeTinLattice( TinLtc *T, int SurfId) ;

typedef struct {
    Dpoint3d    Org ;        /* Origin point (lower left) */
    double      StpX ;        /* X interval */
    double      StpY ;        /* Y interval */
    int         CntX ;        /* Column count */
    int         CntY ;        /* Row count */
    int         *ElevInt ;    /* Elevation as integer values (uor) */
    double      *ElevDbl ;    /* Elevation as double values (master units) */
    char        *Valid ;     /* Validity information */
} TinLtc ;

/*-----
Calculate elevation Pt->z at location Pt->x,Pt->y from
a surface of type Type. This routine will loop thru all
surfaces to find one of matching type where location Pt
is valid.

Return 1 if successful.
Return 0 if the elevation could not be calculated.
*/

int      FnSurfaceTypeZ( Dpoint3d *P, int Type) ;

```

```

/*-----
    Compute elevation for each P->x,P->y point in table Tbl.
    Use first triangulated surface of type Type to compute
    the elevations.

    Tbl is a pointer to an array of x,y points.
    Cnt is the number of points in the array.

    The calling application should initialize the z values
    of the points to an invalid value such as -999999.
    If the elevation of a x,y point can not be calculated,
    the z value will remain unchanged.

    Return 1 if a surface of Type exists.
    Return 0 if no surface.
*/

int      FnSurfTypeArrayZ( Dpoint3d *Tbl, int Cnt, int Type) ;

/*-----
    Open window for specifying how to display surfaces in profiles.

    Return 0 always.
*/

int      FnAskProfSurfaces( void) ;

/*-----
    Compute slope end points starting from master alignment points
    Vrt[] using normal direction as defined by Nrm[].x and Nrm[].y.
    Find solution from first surface of type Typ where Prj[] is
    valid.

    Make sure slope end points are at least Dst from master points.
    Store end points in Vrt[]. If slope can not be computed, compute
    vertex by offseting master point by at least Dst.

    Return number of successfully computed points.
    Return -1 if no surface of type Typ.
*/
int      FnSlopeVertices( Dp3d *Vrt, Dp3d *Nrm, int Cnt, double Rup,
                          double Rdn, int Type, double Dst) ;

/*-----
    Display shaded surface of surface Idv in views Vwb.
    Vwb specifies views as a bit mask (bit 0 for view 1, ...).

    This routine does not update views.

    Return 1 on success.
    Return 0 if failed (I out of range).
*/

int      FnDisplayShaded( int Idv, int Vwb) ;

```

```

/*-----
Display contours for surface with id SurfId.
Ap points to contour action settings.
Sp points to contour settings. Sp may be NULL.
Lp points to label settings. Lp may be NULL.
Tp points to tick settings. Tp may be NULL.

Return 1 on success.
Return 0 if failed (no surface or invalid Ap).
*/

int FnDisplayContours( int SurfId, ContAct *Ap, ContSet *Sp, ContLbl *Lp, ContTck *Tp);

// Contour action for FnDisplayContours()

typedef struct {
    int      Mode ;           // 0=display only, 1=write to file, 2=preview
    int      Type ;           // 0=curves, 1=line strings, 2=soft line strings
    int      DrawTicks ;      // Draw contour ticks?
    int      DrawLabels;      // Draw contour labels?
    int      Draw1 ;          // Draw minor contours?
    int      Draw2 ;          // Draw basic contours?
    int      Draw3 ;          // Draw major contours?
    int      Reserved1 ;
    double   Every1 ;         // Minor contour interval (master unit)
    double   Every2 ;         // Basic contour interval (master unit)
    double   Every3 ;         // Major contour interval (master unit)
    char     Reserved2[80];
} ContAct ;

// Contour settings for FnDisplayContours()

typedef struct {
    int      ColorBy ;        // Color by 0=contour type, 1=elevation
    int      Lvl1 ;           // Minor contour level
    int      Lvl2 ;           // Basic contour level
    int      Lvl3 ;           // Major contour level
    int      Clr1 ;           // Minor contour color 0-255
    int      Clr2 ;           // Basic contour color 0-255
    int      Clr3 ;           // Major contour color 0-255
    int      Sty1 ;           // Minor contour style
    int      Sty2 ;           // Basic contour style
    int      Sty3 ;           // Major contour style
    int      Wgt1 ;           // Minor contour weight
    int      Wgt2 ;           // Basic contour weight
    int      Wgt3 ;           // Major contour weight
    int      Reserved1 ;
    int      ClrCnt ;         // Number of scheme colors
    int      Clrs[127];       // Scheme colors max 127
    long     ClrElev[128];    // Color elevations
    char     ClrFix[128];     // Fixed flags for elevations
    int      Domains ;        // 0=all domains, 1=only selected domains
    int      ElevLimit ;      // 0=all elevations, 1=only ElevMin-ElevMax
    double   ElevMin ;        // - lowest elevation to draw
    double   ElevMax ;        // - highest elevation to draw
    int      Smoothing ;      // Smoothen contours?
    int      AreaSkip ;       // Skip small closed contours?
    double   AreaMin ;        // - min area (sq master units)
    int      Complex ;        // Build complex chains?
    int      Thin ;           // Thin contours?
    double   ThinDst ;        // - distance (master units)
    double   ThinDev ;        // - deviation (master units)
    double   ThinAng ;        // - turn angle (degrees)
    char     Reserved2[80] ;
} ContSet ;

```

```
// Contour label settings for FnDisplayContours()

typedef struct {
    int      LabelsOn ;      // 0=major, 1=major&basic, 2=all
    int      Pos ;           // 0=above, 1=on, 2=below, 3=start point
    int      UpHill ;        // 0=label downhill, 1=uphill
    int      Cut ;           // Cut underlying contour? (if Pos == 1)
    double   Every ;         // Label interval (master units)

    int      Sty ;           // Style
    int      Wgt ;           // Weight
    int      OwnSymb ;        // 0=contour level+color, 1=own level+color
    int      Lvl ;           // - own level
    int      Clr ;           // - own color

    char     Prefix[12];     // Prefix text
    char     Suffix[12];     // Suffix text
    int      Decim ;         // Number of decimals 0,1,2 or 3
    int      Plus ;          // Display plus?
    int      Minus ;         // Display minus?
    char     Reserved1[80] ;
} ContLbl ;

// Contour tick settings for FnDisplayContours()

typedef struct {
    double   Every ;         // Tick interval (master unit)
    int      OnlyDepr ;      // Only depressions?
    int      Triangle ;      // 0=draw line, 1=draw triangle
    int      MaxUse ;        // 0=all, 1=only closed contours < MaxArea
    double   Length ;        // Symbol length (master unit)
    double   MaxArea ;       // Maximum area (sq master unit)
} ContTck ;
```



## Triangulate Survey

TerraModeler's *Triangulate Survey* tool creates a surface model from survey elements. TerraModeler does not understand any details about survey elements or survey feature coding. TerraSurvey is responsible for deciding what elements and what feature codes can be used as valid information for the surface to be created.

TerraModeler scans the design file and calls a function in TerraSurvey for each element to find out if that element can be used in triangulation.

A similar calling sequence has been implemented for DigiCad, PRO600, and Espa applications. The example code below shows how an application should publish the functions for being called by TerraModeler.

```
#define PRO600_SYMBSET 250 /* Any value of your choice */
char *ProSym = NULL ;
ProSym = mdlCEExpression_initializeSet( PRO600_SYMBSET, 0, 1);
mdlCEExpression_symbolPublish( ProSym, "Pro600StartTriMap", SYMBOL_CLASS_FUNCTION,
                              (CType *) &intType, Pro600StartTriMap);
mdlCEExpression_symbolPublish( ProSym, "Pro600AcceptTriMap", SYMBOL_CLASS_FUNCTION,
                              (CType *) &intType, Pro600AcceptTriMap);

/*-----
   PRO600 should use this function to add grid points
   to the surface model. This function enables PRO600 to add
   points which do not have corresponding design file elements.

   Triangulate Survey command has been started for surface
   with id I.

   PRO600 can make calls to FnAddPoints() to add points
   to the surface. PRO600 should not call FnAddDone() from
   within this routine.

   Return 1 if Pro600AcceptTriMap should be called while scanning.
   Return 0 if Pro600AcceptTriMap does not need to be called.
*/

int      Pro600StartTriMap( int I) ;

/*-----
   TerraModeler calls this function for every design file
   descriptor if Pro600StartTriMap() returned 1.

   Pro600 should return one of the following values:
       0 : element is invalid for surface type Type or element is not
          a PRO600 element
       1 : random point element
       2 : breakline element
      10 : external 3D boundary
      11 : external 2D boundary
      20 : internal 3D boundary
      21 : internal 2D boundary
*/

int      Pro600AcceptTriMap( MSElementDescr *Dp, int FileNr, int Type) ;
```

```

/*-----
This function enables DigiCad to add points which do not
have corresponding design file elements.

Triangulate Survey command has been started for surface
with id value of I.

DigiCad can make calls to FnAddPoints() to add points
to the surface. DigiCad should not call FnAddDone() from
within this routine.

Return 1 if DigiAcceptTriMap should be called while scanning.
Return 0 if DigiAcceptTriMap does not need to be called.
*/

int      DigiStartTriMap( int I) ;

/*-----
TerraModeler calls this function for every design file
descriptor if DigiStartTriMap() returned 1.

DigiCad can set the exact point in *Xyz (for leveling text
elements where the position of the decimal point is not accurate).

DigiCad should return one of the following values:
    0 : element is invalid for surface type T or element is not
        a DigiCad element
    1 : random point element
    2 : breakline element
    10 : external 3D boundary
    11 : external 2D boundary
    20 : internal 3D boundary
    21 : internal 2D boundary
*/

int      DigiAcceptTriMap( Dpoint3d *Xyz, MSElementDescr *Dp, int FileNr, int T) ;

```

## 22 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

Data Type	Item	Size	Required
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 neighbour relation of the triangles.

Data Type	Item	Size
UINT[3]	Vertex	12 bytes
UINT[3]	Neighbour	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.
- **Neighbour** - neighbouring triangles in clockwise order. Value 0 means no neighbouring triangle. Value 1 refers to first triangle record. Neighbour[0] is the triangle on the other side of edge Vertex[0] - Vertex[1]. Neighbour[1] is the triangle on the other side of edge Vertex[1] - Vertex[2]. Neighbour[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:

$$\begin{aligned}\text{WorldX} &= \text{XOrigin} + \text{X} / \text{CoordinateResolution} \\ \text{WorldY} &= \text{YOrigin} + \text{Y} / \text{CoordinateResolution} \\ \text{WorldZ} &= \text{ZOrigin} + \text{Z} / \text{CoordinateResolution}\end{aligned}$$

## 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    Neighbour[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
```

# **Additional Information**

## 23 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:\TERRA.

c:\terra	directory where TerraModeler is installed
 config	for configuration files
 tmodel.cfg	defines environment variables
 docs	for documentation and online help
 tmodel.pdf	documentation in PDF format
example	for example files used in the Tutorial
 tmodel.dgn	example design files
 tmodel.clr	example coloring scheme
 license	for user license files
 tmodel.lic	user license
 ma	for application files
 tmodel.ma	application
 tmodel.dll	library
 seed	for MicroStation seed files
 tmodel	for user preferences and settings



## 24 Configuration Variables

---

MicroStation is 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 MicroStation's CONFIG\APPL subdirectory.

For example, C:\...\MICROSTATION\CONFIG\APPL\TERRA.CFG may contain:

```
#-----  
#  
# 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 may contain:

```
#-----  
#  
# 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, MicroStation automatically includes these settings as configuration variables. You can use MicroStation's **Configuration** command from **Workspace** menu to check the values for these variables. In case these variables have not been defined correctly, you should define them manually.