anadelta
SOFTWARE

TESSERA 4

The complete design environment for
road planning & **construction**

Table of Contents

Chapter 1 General 9

Chapter 2 Technical Support 12

Chapter 3 Program Requirements 14

1 Software Requirements ... 14
2 Computer System Requirements ... 15
3 Setup Wizard ... 16
4 Dongle Insertion ... 17

Chapter 4 Features 20

Chapter 5 Start-up Guide 24

1 Program Start ... 24
2 Calculation of Curved Perimeter ... 25
3 Oasis Calculation ... 26
4 Terrain Validation ... 27
5 Calculation of Triangles ... 28
6 Calculation of Contour Lines ... 29
7 Insertion of Breaklines ... 30
8 Road Insertion ... 31
9 Road Calculation - PI Data ... 32
10 Widenings ... 33
11 Stationing ... 34
12 Profile Update ... 35
13 Tangent Curve Creation ... 36
14 Cross Sections Update ... 37
15 Typical Tasks Import ... 38
16 Typical Cross Sections Import ... 39
17 Batch Cross Section Calculation ... 40
18 Typical Cross Sections Editing ... 41
19 Mass Table ... 42
20 Export Files to DXF ... 43

Chapter 6 A First Look 46

1 Creating a New File ... 46

2 Opening an Existing File 47
3 Working with Multiple Files 48
4 Workspaces of the Program 49
5 Getting Familiar with the Enviornment of the Program 50
6 Managing the Program Window 55
7 The Dialogue Boxes 56
8 The Windows 59
9 Error - Confirmation Messages 61
10 Executing Commands 62
11 Using the Shortcut Menus 63
12 Undo and Redo of Commands 64
13 Specifying Points 65
14 Objects 67
15 Object Properties 69
16 Drawing View Options 73
17 Modifying the Program Options 76
18 Project Management 84
19 Saving a File 88
20 How to Select and Save Files 89
21 Closing a File 90
22 To Exit Tessera 91

Chapter 7 Horizontal Alignment 94

1 The Horizontal Alignment Environment 94
Workspace 94
Workspace Modes 95
Categories of Objects 95
Horizontal Alignment Display Options 96
The Snap Feature 100
Selection of Objects 101
Main Object Properties 101
Project Information 103
Quick Find 104
Moving Project Elements 105
Colour Zones 105
2 Topographical Background - Workspace - Terrain 108
Creating the Terrain Model 108
Terrain Management - Properties 108
Terrain Options 110
Terrain Points 111
Creating and Editing Perimeters 115
Creating and Editing Oases 118
Inserting Breaklines 119
Terrain Validation 121
Triangulation and Contour Line Calculation 123

Display of Terrain Elements 126
Terrain Model Export 130
Terrain Model Import 131
Calculating Slope Areas 132
Surface Calculation - Volumes 134
Adjusting the Elevations 135
Slope Analysis 135

3 Road Design 137

Introduction 137
Creating a New Road 137
Road Properties 139
PI Properties 141
Editing PIs 142
Editing Road Stations 145
Editing PI Data 145
Types of Transition Curve 146
Superelevation Diagram 151
Displaying the Superelevation Diagram 159
Setting the Widths 160
Calculation of a Road 161
Widenings 162
Stationing 167
Creating Service Roads 172
Editing Diagrams 173
Terrain Sampling 175
Horizontal Alignment Statistics 176
Structures 177
Intersections 181

4 Printing 191

Preferences Setting 191
Selecting Terrain Objects 194
Road Objects Selection 196
Inserting and Editing Pages 198
Preview 201
Printing a Page 202
Export to DXF File 203

Chapter 8 Profile 206

1 Creating a Road Project 206
2 Workspace 209
3 Profile Workspace Modes 211
4 Profile Display Options 213
5 Bruckner Mode 219
6 Surface Diagram Mode 220
7 Terrain Profile Line 221
8 Terrain Profile Editing 226
9 Road Properties 230
10 Creating and Inserting a Tangent Curve 231
11 Editing a Tangent Curve 234

12 Road Profile Calculation Data ... 235
13 Mandatory Points ... 236
14 Profile Data ... 238
15 Printing ... 239
16 Export to DXF File ... 244
17 Export to ASCII File ... 245
18 Printing the Superelevation Diagram ... 246
19 Bruckner Diagram ... 247

Chapter 9 Cross Sections 252

1 Introduction ... 252
2 Creating a Project ... 256
3 The Workspace ... 259
4 Road Properties ... 262
5 Cross Sections Management ... 263
6 Typical Tasks ... 266
Creation - Management of Typical Tasks ... 266
Families ... 267
Families - Tasks ... 269
Mass Table Columns ... 271
Order of Columns and Tasks ... 273
7 Creation - Automatic Insertion of Lines ... 275
8 Importing a Line from a Text File ... 278
9 Editing Lines ... 281
10 Editing Line Points ... 285
11 Creation - Management of Diagrams ... 288
12 Cross Sections Calculation ... 291
13 Quantities - Areas ... 297
14 Sideslopes Coating ... 300
15 Drawings ... 301
16 Walls ... 304
17 Benches ... 307
18 Batch actions ... 309
Batch Cross Section Calculation ... 310
Batch Drawing Assignment ... 313
Batch Bench Assignment ... 315
Batch Wall Assignment ... 315
Soil Improvement Insertion ... 316
Guardrail management ... 318
Batch Line Management ... 319
Batch Cross Sections Management ... 323
Batch Cross Section Import from another Project ... 324
Batch Cropping of Lines ... 325
Natural Ground from Cross Slopes ... 326

19 Creating Diagram from Point 328
20 Excavations 330
21 Mass Table 333
22 Mass Table Creation 336
23 Mass Table Printing - Export 340
24 Dimensioning Points - Roadlines 343
25 Print Setup 344
26 Cross Sections Printing 347
27 Data Export 348

Chapter 10 Composite Cross Sections 354

1 Horizontal Alignment 354
2 Profile 357
3 Cross Sections 358

Chapter 11 Typical Cross Sections 362

1 Overview 362
2 Operations 363
3 Management 365
4 Main Parts 367
5 Tests 369
6 Creation and Insertion of Levels 370
7 Setting the Branches 372
8 Sideslopes 374
9 Setting the Sideslopes 375
10 Pavings 377
11 Paving Sideslope Coating 379
12 Outer Gutter Filling of Fixed Width 381
13 Guardrails Support 382
14 Insertion - Editing of Lines 384
15 Calculating the Position of a Line Point 387
16 Selected Drawings 391
17 Topsoil 392

Chapter 12 Drawing 396

1 Lines 396
2 Curves 398
3 Clothoids 401
4 Texts 404
5 Images 406

6 Object Properties 407
7 Layers 408
8 Object Import 412
9 Object Editing 413
10 Editing Lines 418
11 Data Export 421

Chapter 1

1 General

Following the release of the successful software package *Anadelta Software, Anadelta* develops its series of programs for road design projects and studies, by introducing the *Anadelta Tessera. Anadelta Tessera* is a continually developing specialized application, offering automated design and layout to simplify and speed up the Road Design procedure. This sophisticated software replaces the *Anadelta Road Design* and takes advantage of all elements of an existing road design that were included in the aforementioned program.

Anadelta Tessera is the development of a new generation series of programs of *Anadelta Software.* In this series, emphasis was put on speed, user-friendly operation and accuracy, thanks to the new autonomous integrated graphics environment of *Anadelta Software.*

This manual covers every basic feature of *Anadelta Tessera,* providing examples where appropriate. Figures, which are true copies of the screen, are given in every chapter, enabling the user to always perform the correct operations.

Our company has made every effort so that the contents of this manual are aligned with the program. However, in a continually developing program, whose purpose is to respond quickly to the ever-broadenings needs and demands of the customers, this is not always possible.

Chapter 2

2 Technical Support

According to its standard policy, *Anadelta Software* offers you access to new versions and any other aid that will help you learn how to use the program correctly. Especially for *Anadelta Tessera,* you can download the latest version by visiting the updated site of our company, in the electronic address www.anadelta.com.

We hope that the continuous and essential technical support offered by our company will cover any gaps that may exist in the manual, especially regarding the new features of the program. Our permanent goal is a substantial cooperation with all our long-term customers and with everyone who chooses our programs for the first time. We welcome your comments and suggestions both for the manual and for the program. We thank you all for your choice and trust.

Chapter 3

3 Program Requirements

3.1 Software Requirements

The software runs in Windows XP or Vista environment without needing any additional design program.
It includes its own design environment and supports all basic operations of a CAD (Computer Aided Design) program.

It communicates with other CAD programs via DXF files.

Finally, the program can print to all printers and Plotters.

3.2 Computer System Requirements

Minimum requirements:
Pentium 4 class processor
512 MB of RAM
200 MB free disk space
CD-ROM driver for the initial installation
USB port for the protection dongle

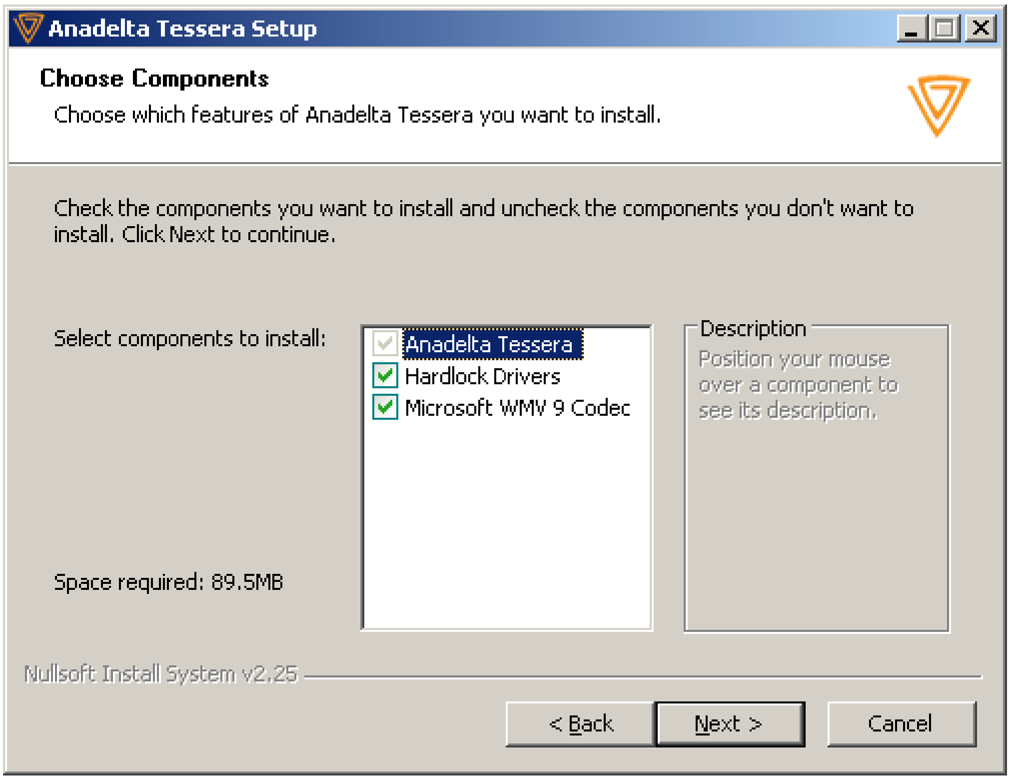
Recommended:
Video card capable of 3D accelerated graphics is needed for the photorealistic 3D mode and video production
1 GB of RAM

3.3 Setup Wizard

You can install the program easily, by following the instructions of the setup wizard.

How to install Anadelta Tessera

1. Insert the program CD into the CD-ROM drive.
2. If the setup screen appears, proceed to the next step. Otherwise, in order for the installation of the program to begin, go to the Start menu, select My Computer, right-click on the drive where you have inserted the installation CD, select Open and run the program.
3. When the initial screen appears, left-click on the Next button to proceed. If you want to stop the procedure, left-click on the *Cancel* button.
4. In the step *Choose Components,* in addition to the main program that will be installed, you must also select whether to install the Hardlock Drivers and the WMV 9 program, which enables you to see the auxiliary videos that are included in the *Help* menu of the program. Once you have completed this step, press the *Next* button to proceed to the next step. If you wish to return to the previous screen to modify your selections, press the *Previous* button.
5. Specify the program installation path and press *Next*.
6. Press the *Install* button to start the installation of the program.



Once the installation is completed, *Anadelta Tessera* informs you accordingly, showing the relevant message.

3.4 Dongle Insertion

After the installation and in order for the program to operate, you must insert the dongle into the USB port of your computer. The dongle is included in the original software package together with the installation CD.

Chapter 4

4 Features

When the program starts, you have two main options. You can either create a new file and start importing data into the empty file, or open an existing *Tessera file* (.ADF extension) or an *Anadelta* ROAD CONSTRUCTION file (.HOR extension). If you open an existing file, all the selections and settings that you have specified are restored, because such information is saved as part of the file. When you start a new file, you must adjust certain settings that will help you during the file processing. You can modify these setting any time you want to.

Project Elements Import

Tessera contains commands that create many different types of object, in order to facilitate you during the import and processing of the performance data. The elements for the study of the terrain, the road layout, the profile and the cross-section of a road can be imported either directly into the program or collectively through ASCII files, or, for most elements, from DXF files.

Error Undo

If you make a mistake while creating, processing or viewing the study, in most, if not in all cases, you can undo it. You can undo either your last action or a number of prior operations. If you accidentally undo an action, you can restore it.

Accurate Design

The points are determined by specifying their X and Y coordinates (type them or select them with the mouse). *Tessera* uses a Cartesian coordinate system, both for the *Horizontal Alignment* workspace and for the *Profile* and *Cross-Sections* workspaces.
You may even select the crosshair to snap points of objects. By selecting the snap feature, the crosshair goes from line to line and/or from point to points. In this way, you can get accurate results without entering coordinates.

Object Properties

There are certain properties that are related to all objects created by *Tessera.* For most objects, you can specify the type of the drawing line (e.g. continuous, dashed or axial), the scale of the line type, the line thickness and the colour of lines and texts.

Object Processing

Once you have entered the data of the design, you can modify the objects you have created. *Tessera* offers various processing methods that minimize the working time required.
You can change the properties of the objects and pan or rotate the objects within the drawing area. You can even delete or shift individual points of objects or/and add points to objects. Finally, you have the option to delete or more objects.

Viewing of Objects

The two coordinate axes (an horizontal and a vertical axis) show you at any time the exact position and the dimensions of your project at real scale.
Tessera enables you to change the project magnification on your screen. You can zoom in for detailed works or zoom out the drawing image that appears in the drawing area in order to see a larger part of the drawing. These operations are being carried out easily and quickly, either by using the mouse wheel or by pressing the <+> or <->

keys respectively.

To see another part of your drawing, you can pan the drawing without changing the magnification. You can use the pan feature by using the mouse (click and hold the middle or the right button), the scroll bars or the left, right, up and down arrows of the keyboard.
All of the above will not affect the true dimensions of the drawing, they will only change the way in which the drawing is displayed on the screen.

Calculations

The program carries out automatically the road calculations in the Horizontal Alignment, Profile and Cross-Sections once you set the parameters and update from the other workspaces.
Also, the user can always intervene and process the results of the Horizontal Alignment, the Profile and the Cross-Sections.
In order to communicate with other programs, you can export this data to an ASCII file or to a DXF file.

Printing

You can print easily and quickly, directly from the program, without the need of any other program. The procedure is simple. First, you specify the properties of the printer and the dimensions of the page and next you select the scale and the items to be printed.
By using the *Print* command and before you proceed to the actual printing, you can preview on the screen the image of the drawing to be printed.

Chapter 5

5 Start-up Guide

5.1 Program Start

Before starting a new study, you must create a new project.

To create a new file

From the *File* menu, select *New Project.* This command creates a new file and takes you to the *Horizontal Alignment* workspace. An empty window will appear on the screen, since you haven't inserted any elements yet.

To import terrain points from an ASCII file

1. From the *File* menu, select the *Import from ASCII* submenu and run the *Terrain Points* command.
2. From the dialogue box that appears on the screen select the *Projects* subdirectory and then choose one of the examples contained in this folder, e.g. Demo 1.GRD.
3. When you click *Open*, a dialogue box appears. Click *OK* in the dialogue box to import the terrain points.
4. If you wish to bring the terrain to the screen extents, go to the *Display* menu and select *Zoom Extents*.

5.2 Calculation of Curved Perimeter

Next, you must insert a perimeter, which will demarcate the area in which the triangles will be created. A perimeter is a zigzag line whose peaks must coincide with terrain points. Therefore, you cannot insert a perimeter unless you have inserted terrain points.

How to insert a curved perimeter

From the *Terrain* menu, run the *Calculate curved perimeter* command. This command creates a closed curved polygonal line that includes all the points of the current terrain.

How to automatically adjust a perimeter

1. Select the perimeter you wish to adjust.
2. Right click on the screen to display the shortcut menu. To add or delete points, select *Adjust perimeter Expand* or *Shrink respectively*.
3. Tick the *General* checkbox if you wish to add/delete points along the perimeter, or tick the *Local* checkbox if you wish to add/delete points at a single part of the perimeter. If you select *Local,* you must specify the width of the sides between which the boundary will be adjusted. The default value is 7 but you can change it. Next, you must specify the perimeter part that will be adjusted. To do this, move the mouse pointer over this part to highlight it.
4. To proceed to the adjustment, roll the mouse wheel down. The effect is cumulative, namely more points are added or deleted as long as you keep rolling the wheel. The effect on the boundary can be seen immediately on the screen. Keep rolling the wheel until you are satisfied with the result. If you wish to undo the expansion/ shrinkage, roll the mouse wheel up.
5. When you finish, click *OK.* This closes the dialogue box, takes you back to the program window and applies the adjustment.

5.3 Oasis Calculation

To insert an oasis, you must have at least one perimeter. The oasis is used for the demarcation of an area in which you will create triangles (e.g. wherever there is a settlement or a lake).

How to insert an oasis

From the *Terrain* menu, run the *Insert Oasis* command. This command creates an open polygonal line that enables you to include all the points that will not be used for the calculation of the triangles.

The oasis processing features are the same as the ones of the perimeter. More specifically, you can:

1. Define circularly the terrain points that will compose the oasis (the mouse snaps automatically the terrain points).
2. Right click to finish the procedure, when you have completed the entry of points.

5.4 Terrain Validation

Before calculating the triangles, you must carry out terrain validation. This stage (before creating triangles) enables the detailed validation of the points. The terrain validation is performed automatically when you request triangulation, but it can also be carried out at any given time, upon request by the user.

How to carry out terrain validation

1. From the *Terrain* menu, run the *Terrain Validation* command. The program will start checking the terrain elements. A bar appears at the bottom right corner of the drawing area, showing you the validation progress.
2. When validation is completed, a dialogue box appears on the screen. In this dialogue box, you can detect and correct any errors of the terrain model. Usually, most errors refer to the terrain geometry.

One of the buttons in this window can be used to show you the wrong terrain point, which is highlighted on the screen.

5.5 Calculation of Triangles

The next step for the creation of the terrain model is the calculation of triangles. First, however, you must specify the settings of the digital terrain model by clicking *Terrain* Options from the *Terrain* menu.

Calculation of triangles

To calculate the triangles, run the *Calculate Triangles* command from the *Terrain* menu. The program will carry out a *Terrain Validation* and if it doesn't find any significant problems, it will start creating the terrain model based on the inserted points, perimeters and oases. In this procedure, the terrain points are interconnected with straight lines in order to form triangles. To create the terrain model, the program takes into account the perimeter and any oases that you may have inserted.

Editing of Triangles

You can intervene and edit the terrain model. If you run the *Triangles Editing* command from the *Terrain* menu, you will notice that the triangles which are located under the mouse pointer turn green. Move the pointer to the desired triangles and click to change the diagonal of the quadrilateral. In this way, you can visually edit any anomaly in the digital terrain model. The changes you are making are shown immediately on the screen. The review and editing is facilitated by the various ways of presentation that the program offers for the terrain model.

5.6 Calculation of Contour Lines

The next step of the terrain processing is the calculation of the contour lines.

The horizontal equivalent of the contour lines for the current terrain is defined from the dialogue box *Terrain Options* of the *Terrain* menu. More specifically, in the group *Calculate Contour Lines* of the *General* tab, type in the text box Insert a contour line every, the distance between two contour lines. From the same group, you can specify the elevation difference between two contour lines in the *Main contour lines* every text field.

How to calculate the contour lines

From the *Terrain* menu, run the *Calculate Contour Lines* command. The program starts to create the contour lines according to the specified settings.
You can activate or deactivate the display of the contour lines by selecting the *Options* command from the *Display* menu.
At this point, while viewing the contour lines, it is possible to edit the terrain model in order to get a better view of the relief. If you wish the contour lines to reflect the results of the editing, you must repeat the calculation of the contour lines.
You can select the way in which the contour lines will appear on the screen (zigzag or curved).

5.7 Insertion of Breaklines

The purpose of the Breaklines is the demarcation of the triangles. These are also zigzag lines, whose peaks must coincide with terrain points. During the triangulation, the program creates the triangles in a way that their sides coincide with the Breaklines and never intersect them.

Insert Breaklines with the mouse

1. From the *Terrain* menu, select *Insert Breaklines*. The pointer turns into a crosshair and snaps only terrain points.
2. Define successively the terrain points that will comprise the Breakline. If you wish, the program can include automatically the points that lie between the previous point and the point that you define. Thus, when the crosshair snaps a terrain point, the screen will display the points that will be included in the breakline.
3. When you have finished the insertion of points, right click to end the procedure.

5.8 Road Insertion

In order to insert a new road, first you must specify the coordinates of its PIs and then the geometric characteristics of each PI.

How to insert the PIs of a road

1. From the *Road Design* menu, select the *Insert road* command. The highlighted square in the middle of the mouse cross shows you the spot where the PI will be placed.
2. By using the mouse, define in ascending order the position of the PI of the road's tangent curve, from the beginning towards the end of the road.
3. When you have finished the insertion of PIs, right click to end the procedure.

How to define the road preferences

1. Select the desired road. From the *Road Design* menu, select *Road Preferences*.
2. From the dialogue box of the *Road Preferences* command, activate a tab and proceed to the desired settings.
3. When you have finished selecting the preferences, click *OK* to close the dialogue box.

You can make settings referring to:

- The elements of the road PIs (*General, PI* and *Slopes* tabs).
- The road widths (*Widths* tab).
- The necessary settings for the calculation of the superelevations diagram (*Slopes* and *Diagram* tabs).
- The stationing preferences (*Stationing* and *Terrain Sampling*).

5.9 Road Calculation - PI Data

To calculate the road, you must go to the *Road Design* menu and run the *Road Calculation* command.
After you have inserted all the PIs of the tangent curve of a road in order to enable the calculation of the road, the next step is to specify the data of each PI.

How to insert the data of a road's PI

1. Select the road you wish to process.
2. Select the PI you wish to process.
3. Right click on the selected PI to display the shortcut menu and choose *Properties*.
4. From the *Properties* window select the value of the *PI Data*. Click the button with the three dots that is displayed when selecting the box.
5. In the resulting window you can modify the data of the relevant PI.
6. Click *OK* to confirm the changes or click *Cancel* to restore the original settings.

5.10 Widenings

The widening of a pavement is the increase or decrease of the pavement semi-width either on its left or on its right side. By using the widenings you can modify the roadlines of the pavement.

Insertion and editing of pavement widening

Select the road you wish to insert for widening. From the *Road Design* menu, select *Widenings* and then select *Pavement Widening*.

The program creates a movable drawing of the widening, which is identified by the position of a yellow triangle. By using the mouse, you can also specify the roadside that will be widened. Select the appropriate position and left click to insert the widening.

Next, select the widening and from the shortcut menu select *Properties*.
In the fields of this window you can edit the value of the chainage where the widening will be inserted, and set the width of the widening, which can be either positive for moving away from the road axis or negative for moving towards the road axis. In the other fields, you can set the length of the widening, as well as the first and second curvature radii respectively.

5.11 Stationing

Stationing can take place after the calculation of the road. Stationing is the calculation and insertion of stations according to the preferences that you specify by using the *Road Preferences* command from the *Stationing* tab. By default, the program inserts stations at the beginning and at the end of the road, as well as at the beginning and at the end of every curve (stations TS and ST). You may, however, insert stations at the other characteristic points of the curves, such as at the beginning and at the end of the circular arcs (stations CS and SC), at the centre of the circular arc (stations CC) and at the vertical from circle's centre to the curve's centre (stations CP and PC).

The first station of the road is the POB station, while the last station is the POE station. A composite name is assigned to the characteristic stations of each curve. This name is comprised of one letter (TS, CP, CS, CC, SC, PC, ST) and one number, which is identified with the numbering of the respective PI (e.g. TS1, CP1 for PI1). The names of the remaining, non-characteristic stations derive from a serial numbering, starting from the number that you have assigned.

Automatic insertion of stations

From the *Road Design* menu, select *Stationing.* The program will calculate and insert the stations according to the aforementioned preferences.

Insertion of a new station

To insert a new station, go to the *Road Design* menu, select *Stationing* and then select *New Station.* You will see a green line, which is vertical to the axis of the selected road. Every time you move the mouse, this line also moves to the position of the mouse pointer. Select the appropriate position and click the left mouse button to insert the station. To specify the accurate chainage and the name of the station that you have inserted, select the station (by right-clicking over it) and then select *Properties.* Repeat the procedures for all stations you wish to add.

Delete a station

Select the station you wish to delete, right-click to display the shortcut menu and then select *Delete station*.

You can also delete all stations of a road. To do that, select the road, go to the *Road Design* menu, then to the *Stationing* submenu and then select *Delete stations*.

5.12 Profile Update

The communication between the three workspaces and the transfer of data from one workspace to the other is performed from the *Project management* dialogue box. If you wish to request the creation of the road profile and cross-sections based on the road data, you must proceed to the relevant updates. When updating from *Horizontal Alignment* to *Profile,* the program will transfer the following data:

- *Terrain Update:* the chainage and the name of every cross-section, as well as the elevation of the natural ground at the axis of the cross-section.
- *Update Diagrams:* transfers the diagrams of the left and right superelevations, as well as the diagram of tangent curves.
- *Update road profile:* the road profile (if any) is copied as a road profile tangent.

When update is completed, the profile will include the cross-sections of the project and their chainages, as well as the elevation of the natural ground.

5.13 Tangent Curve Creation

The next step in the design of the *Profile* is the insertion and processing of the road tangent, in order to determine the elevation of the road profile.

You can create one or more tangent curves by defining its PIs.

How to create a tangent curve with the mouse

1. From the *Profile* menu, select *Insert Tangent Curve*.
2. By using the mouse, define the positions of the *Tangent Curve* Pis within the drawing area.
3. When you have finished, press *Esc* or right click at any part of the screen to end the procedure.

At this stage, you can see on your screen the tangent, which is displayed in red, as well as the road profile elevations. You can also see the road profile, if the program could estimate it with the current values of the rounding radii of the tangent PIs, which have a default value. If you wish to change the original default value of the rounding radii, go to the *Profile* menu, select *Road Properties* and change the value in the *PI initial radius* text field.

5.14 Cross Sections Update

You can update the cross sections from the *Horizontal Alignment* workspace. More specifically:

1. From the *Project* menu, run the *Project Management* command.
2. For a quick update, from the *Update* group, select the item *From Horizontal Alignment to Cross Sections*.
3. In the box that appears below, you can select the items that you wish to update.
4. Click *Update*.
5. Click *OK* to close the dialogue box.

After you have completed the update of the Cross Sections from the Horizontal Alignment, you must update the *Cross Sections from the Profile,* in order to have the roadway elevation for every cross section. To do that, go to the *File* menu and run the *Update from Profile* command.

When updating from the Horizontal Alignment to Cross Sections, the program will transfer the following data:

- *Terrain Update:* the chainage and the name of every cross section, as well as the line of the natural ground on both sides of the cross section axis. The points at the line of the natural ground, as they result from the intersection of the profile axis with the triangles of the terrain model. To set the length of the natural ground line, go to the *Terrain Sampling* tab, select the *Road Preferences* command and enter in the appropriate fields of the dialogue box the values for the left and right sides. If there is no terrain model, you can alternatively request calculation of cross sections. To do that, from the *Cross section elevation calculation way,* go to the *Calculations* tab, select the *General Preferences* command and in the dialogue box activate the option *From stations zone.* In this case, the program creates a natural ground by projecting on the cross section axis the points contained in a zone around every cross section. You can set the length of the zone in the relevant field of the *Road Preferences* command, in the *Terrain Sampling* tab.
- *Update diagrams:* the diagrams that will be transferred are those of the superelevations, the semi-width, the paveway and the roadway for the left and the right sides, as well as the diagram of the tangents, the diagrams of the road that already exists on the left and on the right or the generic use diagrams that have been created.
- *Cross Sections Synchronization:* the positions of the cross sections are correlated based on the chainage or the name. The cross section correlation matters only in case you have already worked inside the *Cross Sections* workspace and want to update the road cross sections, not to delete them or create them from the beginning.

When you have finished processing the road in the *Profile* and *Cross Sections* workspaces, you must update the *Horizontal Alignment* in order to insert the road profile and to draw the roadway and sideslope lines along the road.

5.15 Typical Tasks Import

The lines comprising each one of the cross sections that are included in the program, are grouped in families. This sorting out of the lines has a qualitative meaning. It allows every line, depending on its family, to *locate cross sections* and to calculate areas, by using every time the proper lines. It also serves to determine how the lines will appear both on the screen and on the printouts, as well as to the absolute parametric drawing up of the mass and paving tables.

This standardisation of the lines (tasks) and the correspondance of the calculated quantities to the mass table columns are referred to shortly as *Typical Tasks.* The determination of the typical tasks is one of the main operations of the program and must be done before starting to draw the cross sections.

From the *Parameters* menu, run the *Typical Tasks* command to open the window in which you will specify the typical tasks.

How to import typical tasks from another file

1. From the *File* menu, select the *Import* submenu and run the *Typ. Tasks* command.
2. The *Open* dialogue box appears on the screen. From the *File Type* field, select the type of file (*.ATE or *.ADF), then select the desired file and click *Open*.

Before you run this command, you must be certain that you do not need the current typical tasks that you are using for your project. It is advisable to export them first in a file and then to replace them, because the import of a typical tasks file deletes the current typical tasks and replaces them with the ones contained in the file.

5.16 Typical Cross Sections Import

To import typical cross sections from an external file into the current project, go to *Import* submenu of the *File* menu and run the *Typ. Cross Sections* command. You can import typical cross sections either from another project of *Anadelta Tessera* (*.ADF), or from a cross sections text file.

How to import typical cross sections from another file

1. From the *File* menu, select the *Import* submenu and run the *Typ. Cross Sections* command.
2. The *Open* dialogue box appears on the screen. From the *File Type* field, select the type of file (*.ATD), then select the desired file and click *Open*.

5.17 Batch Cross Section Calculation

For the program to calculate cross sections, the user or the program must first create the main parts of the lines, then extend or remove parts of these lines so as to create between them closed surfaces, and finally measure the area of these surfaces. The calculation of a cross section and the setting of the calculation parameters are being carried through the *Cross Sections Data* management window. This operation can be carried out collectively for all cross sections of the project.

The actions referring to the calculation of a cross section can be carried out collectively for all cross sections of the project by running the *Calculation* command from the *Batch Actions* menu. The purpose of this command is the parameterisation of batch calculations in the cross sections of the current road of a project. The calculations may also include settings of the parameters of the branches, the topsoil, the paving, the sideslopes or/and all of the above.

In all cases, you must click the *Calculation* button for the program to start the batch calculation. When calculation ends, the program informs you on the results by displaying the relevant message in the message window at the bottom of the program window. Any problems that may have occurred, appear in detail inside the window. Click the *Exit* button to close the dialogue box and return to the workspace of the program.

5.18 Typical Cross Sections Editing

To edit a cross section, you must set the gradient of the sideslopes, both for the fillings and for the cuttings. You can set the formation of the sideslopes in the *Sideslopes* tab.

At the bottom of this tab, you will see all the existing sideslope formations for the current typical cross section. Every typical cross section must have at least two sideslope formations, of which one will correspond to the filling and the other to the cutting. To create a new formation, select one of the formations that are listed in the table (e.g. Filling 1:3, 1:2, 2:3) and click the button that corresponds to the creation of a new set of sideslopes. A new formation, which is a duplicate of the one that you selected, will be created on the screen (e.g. New Filling 1:3, 1:2, 2:3). To change the name of the formation, just activate its field and type the new name. When you have finished, press *Enter* to confirm the name. To delete a formation from the table, select the desired formation and click the respective button.
Now that you have created the new sideslope gradients, you can go to the *Cross Sections* workspace and apply these sideslopes to the cross sections that do not have an intersection with the natural ground.

In the *Defaults* group, you can set the sideslope formations that you wish to apply by default to the current typical cross section, both for the left and the right part, as well as for the cutting and the filling. The formations are selected from the respective drop-down lists of this group. The selected sideslope formation is applied instantly and appears in the workspace of the typical cross section. The default sideslope formations are also applied during the cross sections calculation, unless otherwise selected in the *Cross Sections Data* window during the calculation of a particular cross section or in the dialogue box of the *Calculation* command in the *Batch Actions* menu during batch calculation.

5.19 Mass Table

Run the *Print mass table* command from the *Print* menu to display the window in which you will create the mass table.
The dialogue box that appears, has four buttons. Use these buttons to specify certain information that you wish to appear on the mass table. These buttons are *Table Titles, Fixed Columns, Fonts* and *Page Layout*.

- To set the general title of the table, click the *Table Titles* button and type the title in the *Main Title of Mass Table* text field. In the two groups that appear, enter the left and right titles respectively.
- You can change the names of the titles and specify titles for the axis data and the mass haul in the window that appears when you click *Fixed Columns*.
- You can select the fonts that will be used when printing the mass table, as well as the font size and style.
- To set the layout preferences, click *Page Layout*.

5.20 Export Files to DXF

You can export the contents of the *Horizontal Alignment* in .DXF format, to process them in another drawing program.

How to export the Horizontal Alignment data to a DXF file

1. From the *File* menu, select *Export to DXF*.
2. Enter the name of the file in the dialogue box of the *File Name* field.
3. In the *File Type* field, select the type of the file according to the desired version, 14 or 2000.
4. Click *Save* to finish the creation of the file.

- You can export the data of the *Profile* in DXF format, provided that you have activated the *Profile Print Preferences* from the *Print* menu. The export procedure is the same that applies to the *Horizontal Alignment*.
- You can also export the *Cross Sections* data in DXF format. From the dialogue box of the *Typical Tasks* command of the *Parameters* menu, you can select the format and the appearance of the lines, as well as whether some lines will be exported to the file with a layer. The format and the number of all other elements that will be exported, are also selected from the dialogue box of the *Print Preferences* command of the *Print* menu. Type the layer names of every category of elements in the respective *DXF layer* text box.

How to export the Cross Sections data to a DXF file

1. From the *File* menu, select *Export to DXF*.
2. Select the desired cross sections from the dialogue box that appears on the screen. From the *Selected Cross Sections* and *Cross Section Selection* groups, you select the cross sections that will be printed.
3. Click the *Export to DXF* button. Enter the name of the file in the dialogue box of the *File Name* field.
4. In the *File Type* field, select the type of the file according to the desired version, 14 or 2000.
5. Finally, click *Save* to finish the creation of the file.

Chapter 6

6 A First Look

6.1 Creating a New File

If you want to start a new study but you do not have files of the *Anadelta Tessera* software package (files with the .ADF extension), you must create a new file to save the data of your study.

To Create a New File

From the *File* menu, select the *New Project* command or press the *Ctrl+N* keys or click  the button on the main toolbar line. This command creates a new file and transfers you to the *Horizontal Alignment* workspace. An empty window will appear on the screen, since you have not imported any elements yet. The *Tessera* files have an .ADF extension.

You can also create files of the *Profile* and *Cross Sections* workspaces.

6.2 Opening an Existing File

The *Tessera* files have an .ADF extension. To open an existing file, start the program and select *Open* from the *File* menu. You can also open the files by "dragging" them from Windows Explorer and dropping them anywhere inside the *Tessera* window. If the file is not a *Tessera* file or a *Road Design* file, the program shows a warning message, informing you that you cannot open the file.

You can also start *Tessera* and open the file by double-clicking a file from Windows Explorer. If *Tessera* is already running, the file opens in a new window.

If, for any reason, the program does not start when you double-click a *Tessera* file, you can restore the registration by clicking *ADF files registration* in the *Environment* tab that you will find in the dialogue box of the *General Preferences* command of the *Tools* menu.

To open a new project

1. Start the program.
2. From the *File* menu, select *Open*, or press the *Ctrl+O* keys, or click the  button on the main toolbar.
3. From the *Open* dialogue box, go to the subdirectory that contains the file, select the file and click the *Open* button. The files that you can open in this way are *Anadelta Tessera* files (.ADF extension) and *Horizontal Alignment* files (.HOR extension) of the *Anadelta Road Design* program.

You can type the name of the file in the *File Name* field and then click *Open* or you can double-click the file name in the directory of files.

Opening a Project Folder

During the installation of the program the Projects folder is created. This is the folder where you can save your files. If you cannot find this folder, go to the *Tools* menu and click *Open Project File* to go directly to this folder.

6.3 Working with Multiple Files

You can open simultaneously more than one files. To go from one file to another, left-click the button of the respective file on the task bar or press *Alt+Tab*. If, however, you go from one window to another when a command is running, then the command is cancelled. You cannot go from window to window when the program executes certain operations, e.g. calculation of the triangles. This feature enables you to copy or transfer objects from and to different files.

6.4 Workspaces of the Program

The program has three workspaces: *Horizontal Alignmet, Profile* and *Cross Sections,* where you can process the horizontal alignment, the profile and the cross sections of your project's roads respectively. For every project there is an *Horizontal Alignment* workspace, where you can view and edit all roads contained in your project. On the contrary, the same project has as many workspaces for editing the profile or the cross sections, as the roads that you are processing.

The *Horizontal Alignment* exists by default when you create a new file. The *Profile* and *Cross Sections* workspaces are created automatically, either when you request the update of the profile or the cross sections of a road from the *Horizontal Alignment,* or when you create a profile or cross sections file respectively.

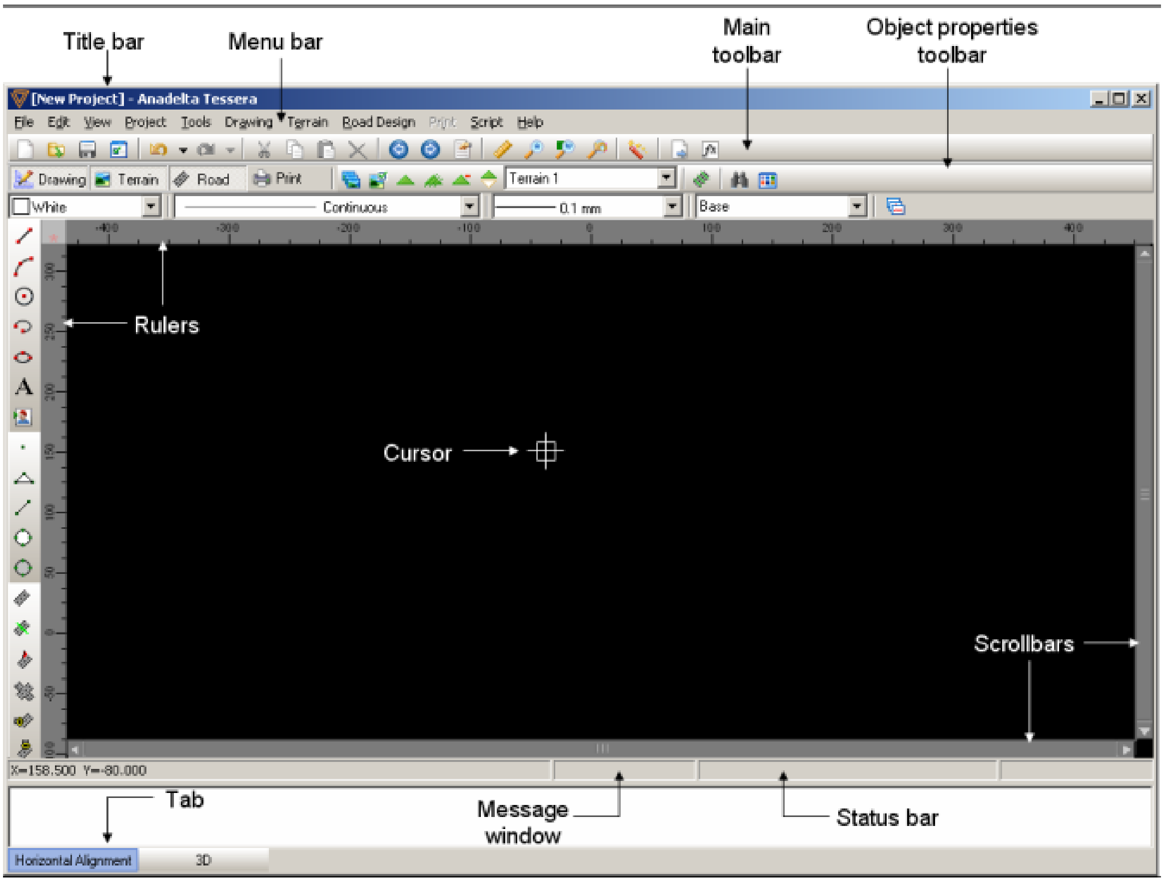
This allocation facilitates the user to enter, edit and process the data of the design.

You can shift among the workspaces by pressing successively the *Ctrl+Tab*, buttons until you are transferred to the desired workspace. Another way is to left-click one of the tabs at the bottom of the program window. You can also go to a workspace by selecting it from the *Display* menu.

If you don't want the program to display a workspace, go to the subject workspace and run the *Close* window command from the *File* menu, or go to the *Project Management* dialogue box and disable the relevant check box from the *Show/Hide* group.

6.5 Getting Familiar with the Enviornment of the Program

Upon program start, the *Anadelta Tessera* window opens. This window is your workspace. It contains objects that you can use to create and process your study. The following sections describe in detail one by one the objects contained in the *Anadelta Tessera* window.



Title Bar

This is the bar on the top of the window. On the left of this bar you can see the file path and name. On the right, there are three buttons: 

1. The minimize button (left button) minimizes the *Tessera* window.
2. The middle button has a double function. If the window occupies only part of the screen, then this button maximizes the window so as to occupy the whole screen. If the window is already maximized, then this button restores the window so as to occupy only part of the screen. The functions of this button can also be performed by double-clicking the title bar.
3. The exit button (right button) closes the program.

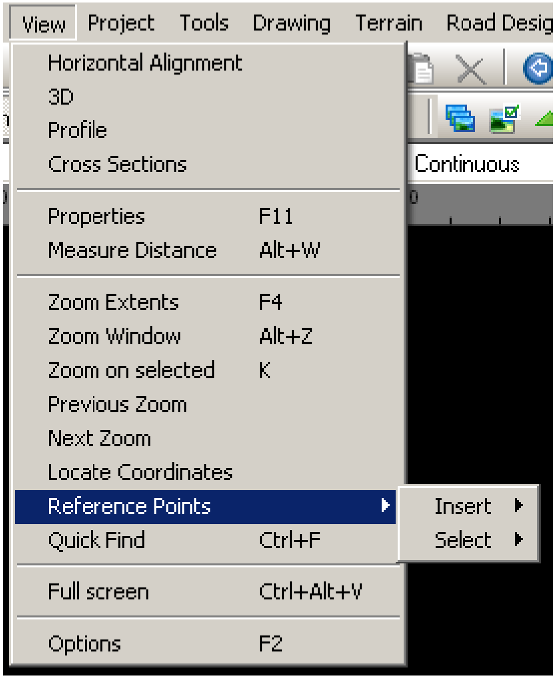
If you right-click on the bar, you will display the shortcut menu with the functions that can be performed on the program window.

Menu Bar

This is the bar with the program menus. From the menu bar you can access most of the commands of the program.

Command Bar

The command menus are lists containing the *Tessera* commands and are displayed in the form of frames one below the other. The usual form of a *Tessera* menu is shown in the Figure:



Most of the bar menus have submenus. The submenus are displayed when you activate a menu on the bar. The availability of a submenu is indicated by an arrow.

To activate a menu, left-click on the menu to display it. Next, to activate a menu command, place the pointer over the command to highlight it and then click the left mouse button.

To activate the menu bar from the keyboard, press the *Alt* key. To select the desired menu, use the left and right arrow keys. Once you've selected the desired menu, press *Enter* to display it. Use the up and down arrow keys to scroll through the commands and submenus until you highlight the desired one. Next, press *Enter* again to execute the command or to display the submenu.

The number and content of the menus shown on the program bar vary according to the current workspace. For example, the *Terrain* menu is shown in the *Horizontal Alignment* workspace, while the *Profile* menu is shown in the *Profile* workspace. Also, the *Print* menu, which is common to both workspaces, displays different commands. When a menu or a menu command is not available, the name of the command appears faded.

Sometimes the icon of the button that corresponds to a command may be displayed on the left of the particular command. You will find this icon on one of the toolbars of the window. On the right, you will find keyboard shortcuts, if available, namely the keys that you must press to execute the command from the keyboard, without using the menus.

Toolbars

Toolbars contain buttons that you can activate by clicking them with the left mouse button. These buttons are on top, on the right and below the drawing area. Each button corresponds to a different command (the commands that are used most frequently). The command is activated when you click the corresponding button (left-click on the button). Each workspace has a different toolbar.

Main Toolbar

Contains the most common commands that the program uses, such as the basic operations *New, Open* and *Save.* It also contains the copy, cut, paste, undo and redo commands.

The main toolbar and the commands it contains are the same for all workspaces.

Balloon Help

The Balloon Help shows you a short description of every button. To see the Balloon Help, move the mouse pointer over the desired button and leave it there for a few moments.

Drop-Down Lists of Options

An arrow pointing downwards on the left border of the field indicates the presence of a drop-down list. The arrow is either displayed permanently or when the field is enabled. When the list is inactive, the option shown in the field is the current one.

To display the options list, left-click on the arrow and the list will drop down. To select an option, place the pointer over the option to highlight it and click the left mouse button or press *Enter*. The three dots next to an option mean that when you select this option, the respective dialogue box will appear, enabling you to create a new list option.

If there is not enough space in the list to show all of the available selections, you can gain access to all options in the box by moving the *scroll bar* up and down in the options list.
To move the scroll bar, either left-click the two arrows of the scroll bar or drag and drop the adjustment button of the scroll bar. You can also use the up and down keys to scroll through the options.

Drawing Area

The black part of the screen is the drawing area and corresponds to a drawing paper of infinite size. The size of the drawing area varies according to the size of the *Tessera* window. You cannot access the drawing area when a window or a dialogue box is open.

The part of the project that will be displayed in the drawing area is not standard, but it can be dynamically defined by the user.

Crosshair and Bounding Box

When the mouse pointer is inside the drawing area, it has the form of a crosshair. The crosshair is comprised of two intersected lines with a small box at their intersection point. The crosshair shows the position of the mouse pointer inside the drawing area and enables you to place points, as well as to select and draw objects. When a command requires from you to define one or more points, the bounding box turns red and follows the crosshair.

Scroll Bars

On the right and bottom border of the drawing area there are two bars. Use these bars to move towards both directions (horizontally and vertically) inside your drawing. Each bar has a button, whose size is proportional to the magnification percentage on the drawing screen, always with respect to the limits of your drawing. To move from one part of your project to another, simply use the mouse to drag the button of the bar and drop it at the desired point.

Coordinate Axes

The coordinate axes are on the left and upper border of the drawing area. By default, the horizontal axis is the X axis and the vertical axis is the Y axis. The step between the indications of the axis changes dynamically according to the magnification of the screen.

Status Bar

The status bar is located at the bottom of the screen and is comprised of four parts:

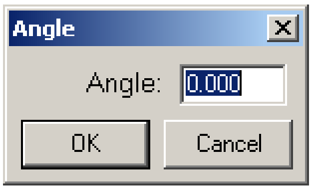
- The first part shows the coordinates of the current position of the crosshair.
- The second part has a button that displays the *Input Form,* which is described below.
- Every time you execute a command, the third part shows instructions on how to execute it, while when you don't execute any command, it shows the identity of the object under the crosshair, depending on the active statuses.
- In the *Profile* and *Cross Sections* workspaces, the fourth part shows information on the current point of the selected line.

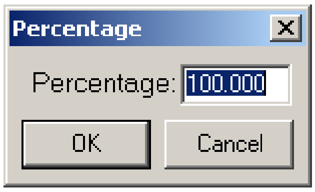
Input Form

Many commands require from you to specify a numerical quantity (coordinates, angle, percentage). *Tessera* enables you to input the quantity in two ways. You can either use the mouse to select a point on the drawing area or you can type the quantity in the relevant fields of the dialogue box of the *Input Form* command.

When the input form is available in the current command, the middle part shows the relevant button.

The *Input Form* is a dialogue box, whose form depends on the command that is being executed. Some of these forms are shown in the figure:





How to display the input form

1. Execute the desired command until the program prompts you to enter the relevant quantity.
2. Press *Enter*, or use the left mouse button to click the *Input Form* button on the status bar.
3. Type the quantity in the respective field/s.
4. Click *OK* to exit the form.

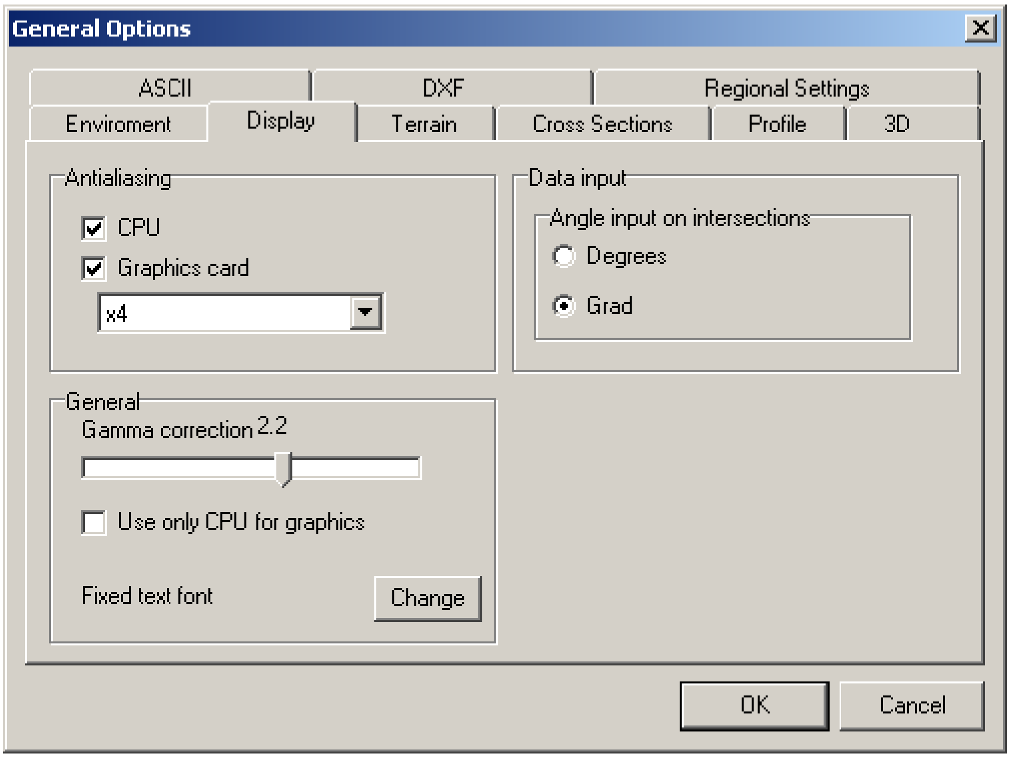
The *Input Form* is a dialogue box.

6.6 Managing the Program Window

The window of the program has the same features as any other window of *Windows.* You can *maximize* it, so as to occupy the full screen, *minimize* it, so as to appear as a button on the Windows task bar, or close it. You can do all of the above by clicking each time the appropriate button on the title bar. You can also restore the window, so as to occupy only part of the screen. In this case, you can also move the window anywhere on the screen, by using the drag-and-drop feature after you have positioned the mouse pointer anywhere on the title bar. You can even change the window size by moving the pointer at the borders of the window (the pointer changes into one of the forms shown below) ↔ ⤡ and by using the drag-and-drop feature.

6.7 The Dialogue Boxes

The dialogue boxes enable you to set the program parameters. They appear when you execute certain commands. A dialogue box is comprised of a box that has many objects, such as buttons, text fields, option fields, which correspond to settings and are grouped in groups and tabs.



You can select and set any item with the mouse or the keyboard. Use the *Tab* or the *Shift+Tab* keys to go successively from one field or button to the next or to the previous one, respectively. The item surrounded by a dashed line is the item you are currently using.

When a dialogue box is open, you don't have access to the items of the program's main screen. To perform any action on the main screen, first you must close the dialogue box and return to the main screen.

To maximize, minimize or move a dialogue box, proceed as you would do in the case of a program window.

Most dialogue boxes offer you at all times two main possibilities. You can either close the dialogue box and cancel the settings by clicking *Cancel,* or you can close the box and accept the changes by clicking *OK*.

Some dialogue boxes have only one button, the Close button. In this case, to exit the dialogue box you must first accept the changes that you have made.

The exit button on the title bar and the *Alt+F4* or *Esc* keys close the dialogue box, but they do not ensure that your settings will be stored this depends on the particular dialogue box.

Tab

When a dialogue box includes many options, for your own convenience, these options are usually grouped in tabs. When you select a tab, all options in this tab will be displayed in the dialogue box. The title of the tab appears on the top of the tab.

To move between tabs, use the left mouse button to click the title and activate the tab. To do the same with the keyboard, activate any tab and then select the desired tab with the left and right arrow buttons.

Group

The options – settings that you can make are listed in groups in every tab. To distinguish them visually, the groups are surrounded by a frame on the top of which they bear a distinctive title.

Buttons

The buttons in a dialogue box usually display a new dialogue box or carry out an operation. 

. To activate a button, simply left-click the button (*button press*) or activate the button and press *Enter*.

Slide Bars

These are comprised of a longitudinal bar with an adjustment button. Use the slide bars to set the amount of a quantity.

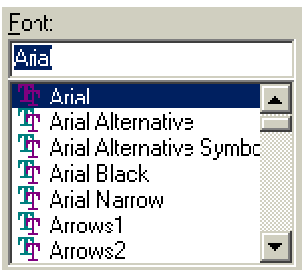
To set the amount of a quantity, drag the button of the bar until the quantity reaches the desired amount and then release the left mouse button. You can do the same with the up, down, right and left arrow keys when the bar is active.

Text Fields

The text fields are rectangular boxes, usually white. In a text field you can type text or numbers, depending on the kind of the setting. For your guidance, next to the fields (on the right or on the left) a description of the quantity you define as well as the unit used, are indicated.

To type inside a text field, first you must activate it. You can activate the field by using the keyboard or the left mouse button. In all cases, you must first wait for the cursor to appear (a flashing line inside the field) or for the existing text to be highlighted.

Options Lists



When there is a number of options in a field, usually the field offers a list of the available options. You can also type the option in the current field. Activate the field with the mouse or the keyboard and then type the option. If you type it correctly, then the option will appear highlighted in the options list. You may even select the option from the list, by using the up and down buttons.

Preview Fields

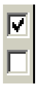
Sometimes the options lists are linked to certain preview fields. You cannot intervene in the preview fields you cannot even select them with the mouse. They only show you the result that you will get if you accept the settings that you have made.

Drop-Down Lists

When the list field is active, you can use the up and down arrows to scroll through the options without displaying the list.

Check boxes

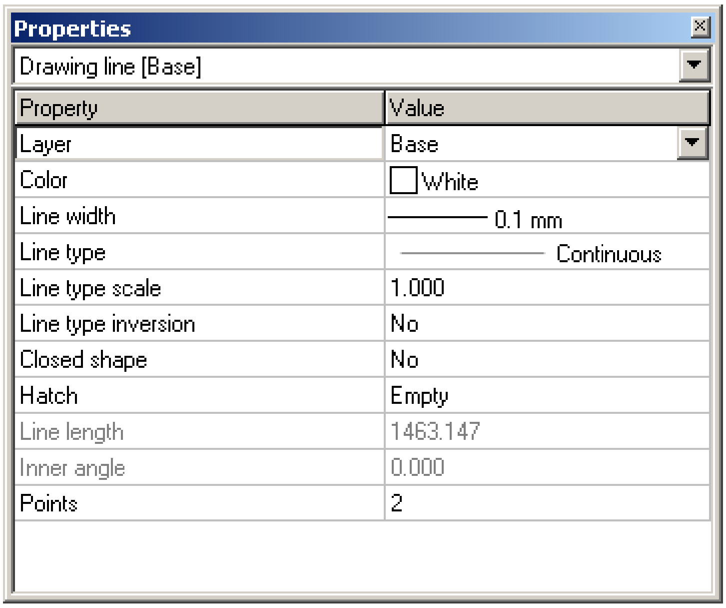
Use them to choose one of the available options or to select whether a setting shall apply or not. They are displayed as a circle or a box.

 A white box corresponds to every option. If the box is empty (inactive) then the setting shall not apply. If the box is enabled (ticked), the setting shall apply. You cannot check the disabled options, which appear gray instead of black. To activate or deactivate a check box, left-click inside the box or select the check box and press the *Spacebar* key.

 If it is mandatory to choose one setting out of various available settings, you will find these settings in the same group of a list. Next to the settings there is a circular check box. If the circle is empty, then the corresponding setting shall not apply; the opposite occurs when the circle is active (there is a dot inside the circle). Use the up and down arrow keys of the keyboard to select (check) the desired setting from the current group of settings.

6.8 The Windows

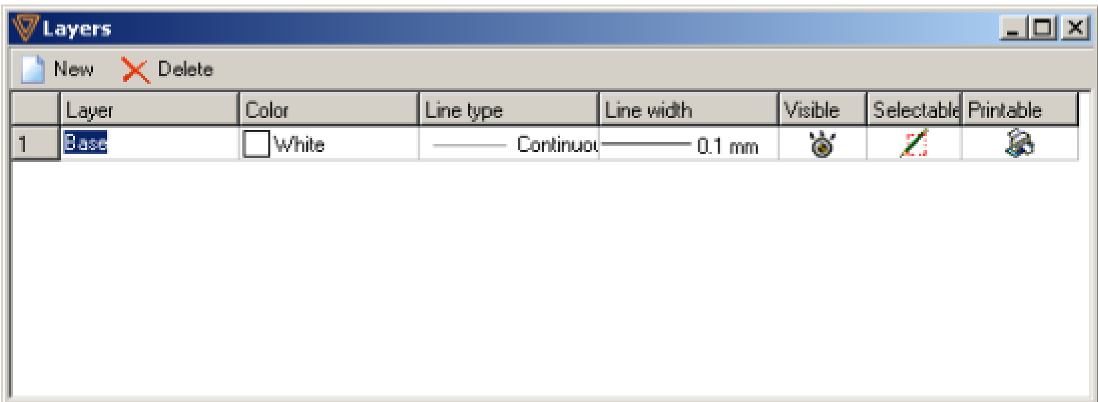
The execution of multiple commands results in the display of one or more windows.



The window is a screen area that you can move, open and close, just like you would do in the case of a program window. When you open more than one window, you cannot work in all windows at the same time, you can only work in the current window. The program windows usually contain lists of fields with data. In most windows you can change the content of the fields. In the case of a text field, left-click to activate the field and then type a new value or text, just like you would do in a dialogue box. There are fields where you do not have to type anything. These fields have lists of preset values. You can choose the desired value from the relevant list. A button with three dots (...) on the right of an active field means that when you press this button a dialogue box shall appear.

Management Windows

These are windows that carry out a large number of operations. These windows offer additional features and possibilities, but you can manage them just like the common windows.



You can change collectively certain fields that belong to the same group by selecting them all together and then adjusting the property. The settings you make for one field shall apply to the next fields. There are also standard fields, in which usually an icon is displayed. To change the property that the field controls, use the mouse to double-click the field.

The management fields may also contain buttons, which have the same function as the buttons of the program main screen toolbars.

6.9 Error - Confirmation Messages

When an operation is being carried out, *Tessera* communicates with you by displaying various messages on the screen. These messages are actually special dialogue boxes, but they do not have the same form as the common dialogue boxes.

Error – Warning Messages

These messages appear either when an error has occurred and the program cannot continue executing the command, or to inform you that a procedure has finished.
These dialogue boxes have only one button that closes the box after you have read the message.

Confirmation Messages

These are displayed when you ask the program to carry out an irreversible operation. In this case, the program will show you a message, asking you to confirm the operation (*Yes* or *OK* buttons) or to cancel the operation (*Cancel* or *No* buttons) by clicking the relevant buttons.

6.10 Executing Commands

In *Tessera,* there are three ways to select a command:

1. You can use the mouse to choose a command from a *drop-down menu* or a shortcut menu.
2. You can use the mouse to click a button on the toolbars.
3. You can press the *auxiliary keys* that correspond to the particular command.

To stop the execution of a command, press the *Esc* key or click the right mouse button.

6.11 Using the Shortcut Menus

When you click with the right mouse button in the drawing area, shortcut menus appear on the screen. These menus enable you to select very quickly certain operations. With certain keyboards you can also display the shortcut menus by pressing the relevant key. The commands contained in the shortcut menus vary, meaning that the displayed menu and its options depend on whether an item has been selected and on the nature of this item.

6.12 Undo and Redo of Commands

If you execute a command by mistake, in most cases you can undo it. You can undo either only your last action or various previous operations. You can also redo certain actions that you have undone accidentally.

How to Undo a Command

From the *Edit* menu, click *Undo*, or press the *Ctrl+Z* keys, or click the button  on the main toolbar. If desired, you can display the drop-down list of this button. The list contains the commands that you have executed after you last started the program. Select the commands you wish to undo.

How to Restore a Command

From the *Edit* menu click *Redo*, or press the *Shift+Ctrl+Z* keys, or click the button  on the main toolbar. If desired, you can display the drop-down list of this button. The list contains the commands that you have undone. Select the commands you wish to restore.

6.13 Specifying Points

When you execute a large number of commands, e.g. to import lines, you must specify points. In this case, the form of the crosshair changes and the message *Specify point* is shown on the status bar. This means that you must specify the *X, Y* coordinates for every point by using one of the available methods offered by the program. You can use the mouse to specify a point by left-clicking inside the drawing area at the desired point or you can input the absolute coordinates in the *Input form*.

Coordinates System

Tessera uses a Cartesian coordinate system. This system has two axes, *x* and *y*. When you input coordinates values, you specify the distance (in units) and the direction (+ or -) of a point along the X,Y axes with respect to the start of the axis (0,0). By default, X is the horizontal axis and Y is the vertical axis. When you create boundaries, you specify the chainage and the distance from the road axis.

To View the Coordinates

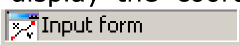
The absolute coordinates for the current position of the crosshair are displayed on the left of the *Status Bar.* The coordinates follow the crosshair and change accordingly.

For most lines you can view the coordinates of their points from the *Properties window,* either directly or by using the Points command in the box of the *Properties command*.

Input of Absolute Coordinates

The input of the absolute coordinates is very simple.

To input the absolute coordinates directly:

1. Run the desired command until the program prompts you to specify one or more points.
2. Press *Enter* to display the coordinates *Input form* or click the relevant button on the status bar 

.
3. Type the *x* and *y* coordinates in the respective fields.
4. Click *OK* to exit the Input form.

If you wish to specify further points, repeat the procedure.

Accurate Design

Tessera enables you to snap objects for easy and accurate design. By using these tools, you can design with accuracy, without having to input coordinates or make difficult calculations. You can also use the search tools to find instantly information on the drawing and its objects.

Grid

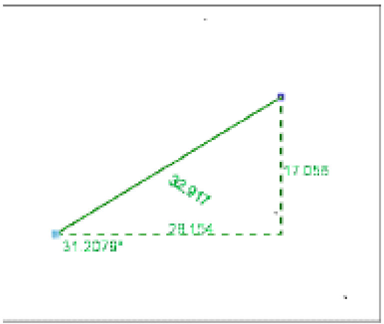
The grid is a squared set of lines that covers the whole drawing area. Its use is similar to the use of a transparent graph paper over your paper drawing. The grid enables you to align objects and to visually estimate the real distances. When you work in the *Profile* and *Cross Sections* workspaces, the grid will not be printed on paper. The size of the grid is adjusted automatically when you zoom in or out, depending on the scale.

Snap Feature

The snap feature is a quick way to locate or specify, depending on the current operation, the exact position of a point without knowing its exact coordinates. Therefore, this method is very accurate and convenient.

You can use the snap feature anytime *Tessera* asks you to specify a point or a size which can be specified by selecting a point on the screen, e.g. the rotation angle of an object. You can enable one or more snap types at the same time. The snap feature remains enabled until you disable it.

Distance and Angle Measurement



You can view the following information between two specified points.

1. The distance between the two points.
2. The angle of the straight part with respect to the horizontal axis, measured both counterclockwise and clockwise.
3. The distances on the X and Y axes (Dx and Dy respectively).

This information will help you verify the accuracy of your data.

You can also determine the number of the information that you will obtain. Namely, you can either perform a simple measurement of the distance and view the distance and the values of the *Dx* and *Dy* on the third section of the status bar, or you can perform a *full* measurement. In this case, the values shown on the status bar are the same as the values of the simple measurement. The values are also displayed on the screen, together with the value of the angle with respect to the horizontal axis.

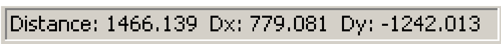
To select the type of measurement

1. From the *Tools* menu, select the *General Options* command.
2. In the resulting dialogue box, select the *Display* tab.
3. In the *Measure Distance* group, select *Full* or *Simple*.
4. Click *OK* to close the dialogue box.

To measure a distance

1. From the *Display* menu select *Measure Distance*, or press the *Alt+W* keys, or click the button  on the main toolbar.
2. Set the first and the second point of the distance you wish to measure.

Tessera will display the information, as described above



6.14 Objects

Selecting Objects

Before editing the objects in your file, you must create a set of selected objects. This set may be comprised of only one object or it can be a set of multiple objects: for example all objects of the same color or all objects of the same group or category. These sets must be created *before running an editing command*. You can select objects in one of the following ways.

1. Use the mouse to select the objects and then run an editing command.
2. Run a command that creates a set of objects. Then run an editing command.

Whatever the method, when you select an editing command, *Tessera* will search for objects. There are additional commands that enable you to select drawing elements.

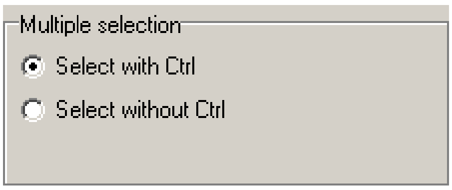
Selecting Objects with the Mouse

The easiest way to create a set of selected objects is to use the mouse.

How to select an object with the mouse

1. Activate the mode of the object you wish to select (e.g. *Drawing* mode in the *Horizontal Alignment* workspace).
2. Position the crosshair with the bounding box in such a way, so as to display the object category on the status bar. You can generally do that if you position the mouse pointer over the object (over a part of the object or over the line connecting parts of the object), e.g. to select a *Road* you must position the mouse pointer over a *PI*, to select a *Frame to Print* position the mouse pointer over a point within the frame.
3. Click the left mouse button.

The selected object will be highlighted on the screen.



By default, if you select another object with the same procedure, this object will be added to the existing set. If you want to add an object that you select to the existing set, go to the *Tools* menu → *General Options* → *Environment* → *Multiple selection* and activate *Select with Ctrl*. When you select an object by using this option, the existing selection set is cleared and only the new object is selected. If you select an object by pressing and holding down the *Ctrl* key, this object is added to the existing set. On the contrary, if you activate *Select without Ctrl,* the multiple selection of objects is performed without pressing any key, only by clicking the left mouse button.

Selection Box

You can select objects by enclosing them in a selection box. A *selection box* is a rectangular field that you can form within the drawing area, by defining its two opposite corners. The *order* in which you will define the corners is important, because it determines which objects will be selected. If you create the selection box from left to right, the selected objects will be those having all their points inside the box. If you create the selection box from right to left, the selected objects will be those having at least one point inside this box.

Deselect All

To deselect all of the selected objects, go to the *Edit* menu and select *Deselect all*.

To deselect all of the selected objects from the *Edit* menu or from the shortcut menu that appears when you right-click in the drawing area, click *Deselect All* or press the *Shift+Ctrl+N* keys.

If *Select with Ctrl* is enabled in the dialogue box of the *General Options* command, you can also deselect all objects by left-clicking on an empty spot in the drawing area.

Editing Points of Objects

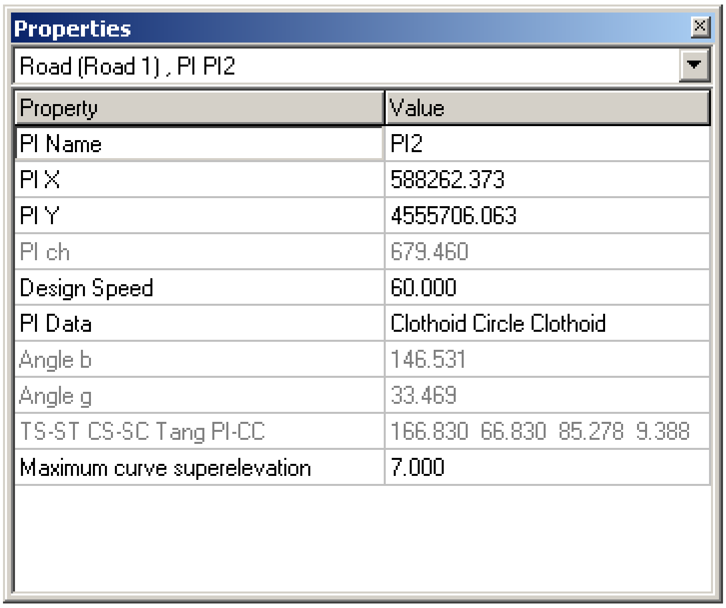
When selected, the object is highlighted and its points are indicated by a box. For example, when you select a line, a box appears at every end of the lines where the line points are.

If you want to use the points in order to edit an object, select the point that enables you to run the desired editing command. To select the point of an object, simply click the left mouse with the crosshair over the desired point. The current point is highlighted and is filled with a red color. To move between the points of an object, use the left and right arrow keys while holding down the *Ctrl* key, or use the buttons   on the main toolbar.

You can edit points with the mouse in order to combine main editing commands, such as *Move*, *Zoom*, etc. To activate a selected point, click again the left mouse button with the crosshair over the point.

6.15 Object Properties

Every object that you create in *Tessera* has a set of properties. These properties are displayed in the *Properties* window. The *Horizontal Alignment* workspace gives you easy access to the main properties of the objects. You can also access the properties from the *Object properties* toolbar. The best way, however, to modify the properties of a set of objects, is to use the properties window.



To View Object Properties

Go to the window of the Properties command to view and change the properties of one or more objects.

To view the properties of one or more objects

1. In the drawing area, select one or more objects whose properties you want to view.

2. From the *Display* menu, select *Properties* or press the *F11* key or click the button  on the main toolbar.

You can also execute this command from the shortcut menu that appears when you right-click on the screen after you have selected an object.

The window lists all properties of the selected objects. It is comprised of two columns. The *Property* column shows the name of the property, while the *Value* column shows the current value of the property. Here you can change any modifiable property.

You can leave the properties window open while you are working. When you select an object, the *Properties* window will display the properties of the selected object. When you select objects of another category, the window will display the common properties of the selected objects. If no object is selected, the window displays the main properties, namely the layer, the color, the line type, the line width and the line type scale. If no object is selected and you select a new value from this window, then you specify the current settings for the new *drawing* elements.

To Edit Object Properties

You can change the properties of an object from the window of the *Properties* command. To change the values in this window, select the desired object and do one of the following:

1. Type a new value.
2. Select a value from a drop-down list.
3. Change the property through a dialogue box or a window.

To change the properties of one or more objects

1. From the *Display* menu, select the *Properties* command.
2. Select the object/s you wish to edit. In the window of the *Properties* command you can see all properties of the selected object or the common properties in the case of multiple selected objects.
3. Select the property you wish to change and type a new value, or select a value from a drop-down list. In some cases, when you select a property value, the button , will be displayed. If you click this button, you will be transferred to a dialogue box, where you can make the desired changes and then apply them by clicking the *OK* or *Apply* button of the box.

Main Object Properties

The objects that you create in *Tessera* have certain main properties, such as *color, line type, line type scale* and *line width*. These properties enable you to group together elements of your design, so as to facilitate the visual identification of these items, both on screen and on the printout. This makes the processing of the study easier and improves the appearance and quality of the end drawings.

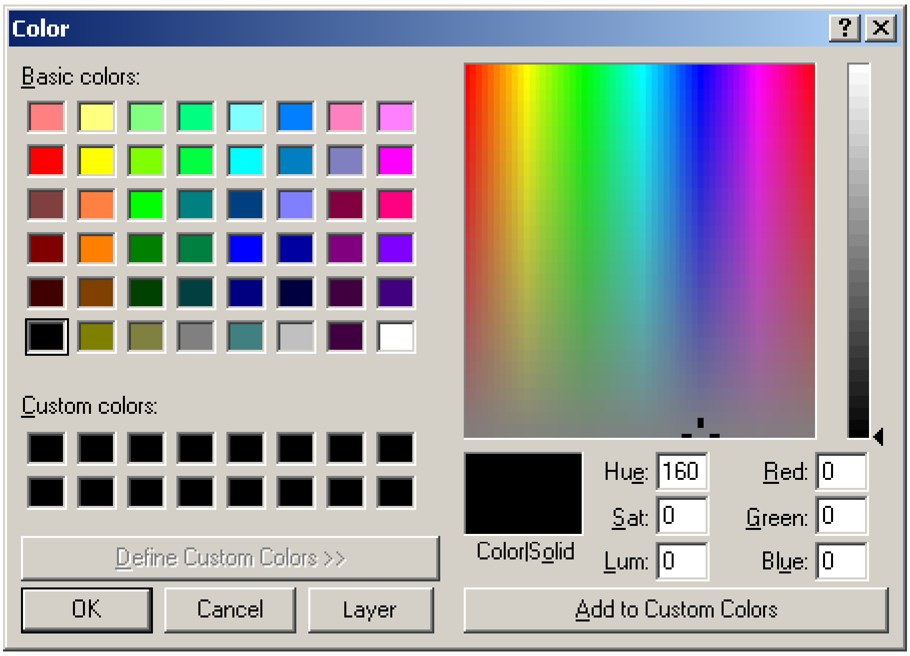
Color

You can define a color separately for every object or generally for every category. Every color is characterized either by its parameters or by a predefined name. A set of objects can have the same color. You can use colors to distinguish instantly the objects on the screen. The program provides the main colors with default parameter values and a default name, but you can also define any other color you like.

To select a color for an object or a category, you can choose from the list one of the default colors or you can define a new color by clicking *Other*. The default colors are:

BLACK
GREY
BLUE
DARK GREY
GREEN
LIGHT BLUE
CYAN
LIGHT GREEN
RED
LIGHT CYAN
MAGENTA
LIGHT RED
BROWN
LIGHT MAGENTA
YELLOW

To Define a New Color

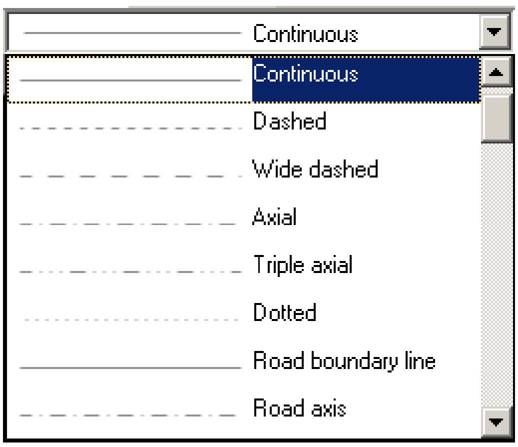


1. From the drop-down list of the *Color* property, which is displayed in the dialogue box of the *Properties* command, click *Other*.
2. In the dialogue box of the *Color* command, select a basic color or define a new color.
3. To define a new color, click the *Define custom colors* button. To define the details of the new color either fill-in the six fields on the bottom right part of the box or select the color from the palette on the top left part of the screen and adjust the brightness by using the slide bar on the right of the palette.
4. When you have finished, click the *Add to custom colors* button and then click *OK*.

Line Types and Line Type Scale

The line type is a repeated pattern of dashes, dots and spaces. The name and the definition of the line type describe a specific sequence of dashes and dots as well as the relevant sizes of the dashes and dots.

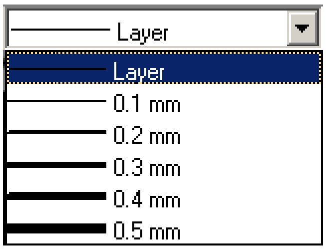
There are fifty-two line types in *Anadelta Tessera* and you can select the desired one from the drop-down list.



The line type scale is user-defined. The lower the value you set for the scale, the more the repetitions of the pattern per drawing unit. The default value is 1.0. The relative size of the dashes and dots is always proportional to the line type scale and their size on paper is always fixed, regardless of the print scale. Their on-screen size is always identical to their size on paper, according to the drawing scale.

Line Widths

There are six line widths in *Anadelta Tessera*.



This property adds width to the lines of your objects, both on screen and on the printout. With this item you can create thin or thicker lines, so as to distinguish them visually and to read your drawing easier, both on the screen and on the printed drawing.
The objects are printed according to their assigned value. The line width values are in millimeters and always fixed. The line width of the drawings you export to other file formats can be maintained. The line width also remains unchanged when you copy objects from one file to another.

6.16 Drawing View Options

Tessera provides many options that can help you control how your drawing will be displayed in the drawing area. The percentage of the drawing inside the drawing area is the zoom, while the point in the center of the drawing area is referred to as *center of view*. A particular zoom and observation point give you the view of your drawing. While you're processing your project, you can change the view and go quickly from one area of your drawing to another. You can zoom in or out the image of your drawing that is displayed inside the drawing area or to pan it, to save a particular view and to recall it again in the future. The absolute sizes of the drawing will not be affected when you change the zoom or the center of view, only the way in which the drawing is displayed within the drawing area will change.

Using the Zoom and Pan Commands

The easiest way to change the view is to use one of the many zoom features of *Tessera* in order to increase or decrease the scale of the drawing.

Real-Time Zoom and Pan

The *Zoom* and *Pan* commands of *Tessera* enable you to watch in real time how the view changes. With the *Zoom* command, you can zoom in or out on the drawing by rolling the mouse wheel.

How to Zoom in real time

Roll the mouse wheel. To increase the image scale (*zoom in*), roll the mouse wheel up and to decrease the scale (*zoom out*) roll the mouse wheel down. You can do the same by pressing the + and - keys respectively.

With the *Pan* command, you can change the center of view.

How to Pan in real time

1. Place the crosshair inside the drawing area.
2. Click the mouse wheel.
3. Hold the mouse wheel down and move the pointer (within the limits of the drawing area), until you view the desired part of your drawing. You can do the same by pressing the up, down, right and left arrow keys.

The Zoom Window Command

You can *zoom* in very quickly on an area of your drawing if you define its borders.

The area that you define is centered in the new view. The size of the sides of the defined window is proportional to the size of the drawing area.

To zoom in on an area by defining its borders

1. From the *Display* menu, run the *Zoom Window* command, or press the *Alt+Z* key, or click the button  on the main toolbar. The mouse pointer turns into a window.
2. Set one corner of the area by clicking the left mouse button.
3. Set the opposite corner.

You can also run this command from the shortcut menu that appears when you right-click in the drawing area 

.

To Display the Previous View

When you're working on small parts of your drawing, you will have to *zoom out* frequently to view your whole project. The *Previous Zoom* command restores quickly the previous view.

Tessera can save an indefinite number of previous views. The *Previous Zoom* command restores only the zoom scale and the center of view and has no effect on the settings or properties of the objects.

To return to the previous view

From the *Display* menu, run the *Previous Zoom* command, or click the button  on the main toolbar.

To display the next view, run the *Next Zoom* command from the *Display* menu.

Centering the View to a Point

You can set a specific point in your drawing as the center of view. The *Locate Coordinates* command moves the view to a specific point in the center of the screen, without changing the zoom.

To center the drawing within the drawing area

1. From the *Display* menu, run the *Locate Coordinates* command.
2. In the dialogue box that appears, type the coordinates of the point you wish to become the new center of view.
3. Click *OK*.

The view is shifted and the point that you have specified will be in the center of the drawing area.

Zoom on Selected Objects

In order to view on screen the desired objects, run the *Zoom on selected* command.

The *Zoom on selected* command changes the view so as to include all the user-selected objects, displaying them with the highest zoom, in order to fit in the drawing area.

To display all the selected objects

1. Select the objects you wish to be displayed inside the drawing area.
2. From the *Display* menu run the *Zoom on selected* command or press the *K* key.

Zoom on Project's Limits

To adjust the view on your project's limits, select the *Zoom extent* command.

The view that you get with the *Zoom extent* command includes all the objects of your project with the highest zoom, so as to fit in the drawing area.

To display the whole project

From the *Display* menu, run the *Zoom extent* command, or press the *F4* key, or click the button  on the main toolbar.

Saved Views

You can save a view to recall it at a future time. *Tessera* can save up to four (4) different views.

Saving a Reference Point

When you save a view, the following data are also saved automatically:

1. The center of view.
2. The zoom coefficient.

To save a reference point

1. From the *Display* menu, select the *Reference points* submenu.
2. From the *Reference points* submenu, select *Insert* and then choose one of the four available points.

Restoring a Reference Point

You can recall a saved view any time you need to use it.

To recall a saved reference point

1. From the *Display* menu, select the *Reference points* submenu.
2. From the *Reference points* submenu, select the *Select* submenu and then choose one of the saved views.

6.17 Modifying the Program Options

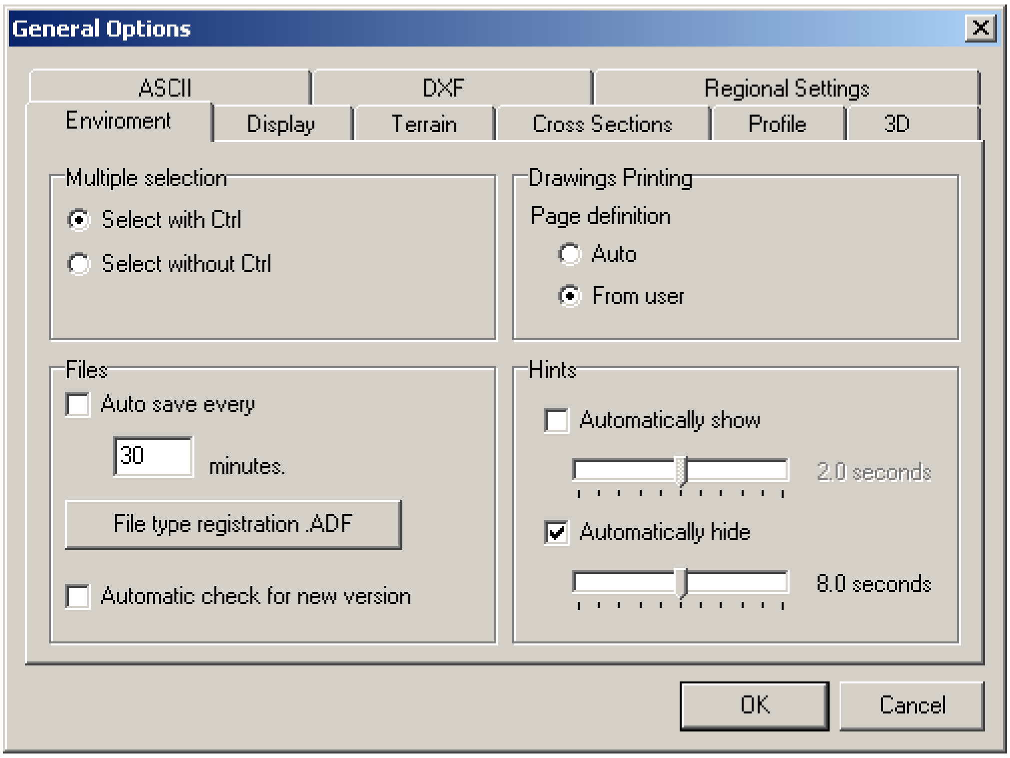
You can modify many settings of the *Tessera* window from the dialogue box of the *General Options* command of the *Tools* menu. For example, you can specify the object selection method. Experiment with the available options until you create an environment that enables you to work with the maximum speed and accuracy. The settings that you can select in the tabs below can be applied to any file, even to a new project.

To change the options of Tessera

1. From the *Tools* menu, select the *General Options* command.
2. From the dialogue box of the *General Options* command, choose a tab and select the settings you wish to change.
3. When you have made your options, click *OK*.

Environment Options

The *Environment* tab that you can find in the dialogue box of the *General Options* command, controls the object selection method and the printing parameters.

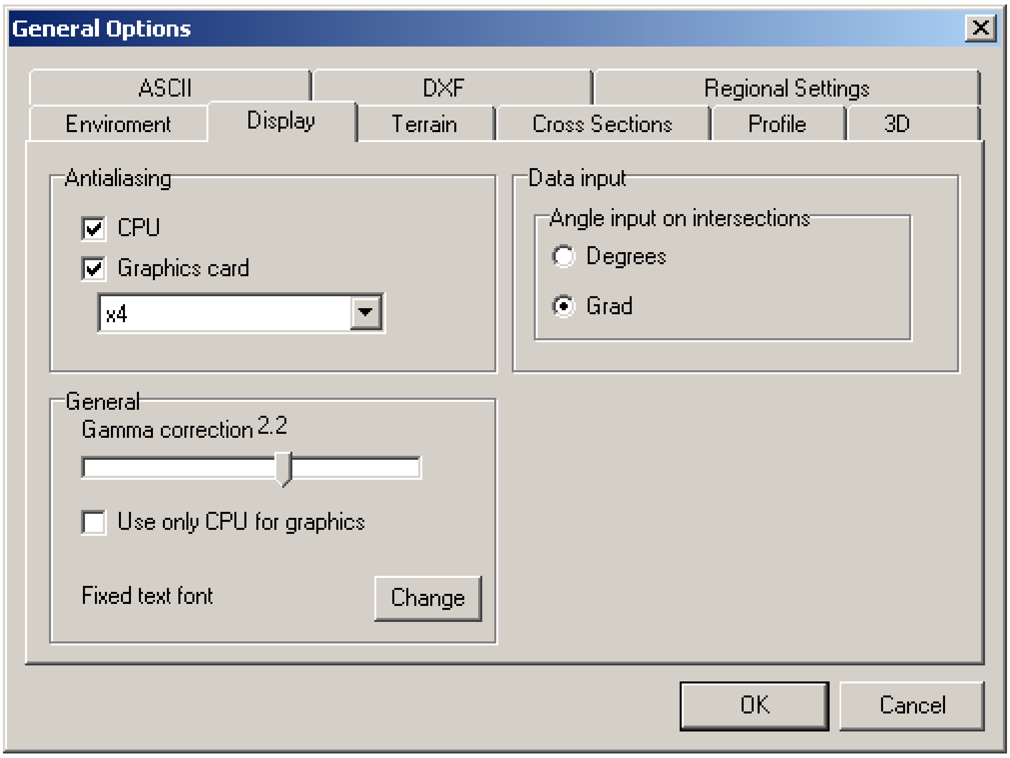


You can make the following settings:

1. Method of multiple object selection.
2. Printing page definition.
3. Registration of .ADF files with *Tessera*.

Display Options

The *Display* tab in the dialogue box of the *General Options* command controls mainly the way in which the main items of the program are displayed. For example, you can modify the type and size of the fonts that will be used when viewing fixed texts.



You can make the following settings:

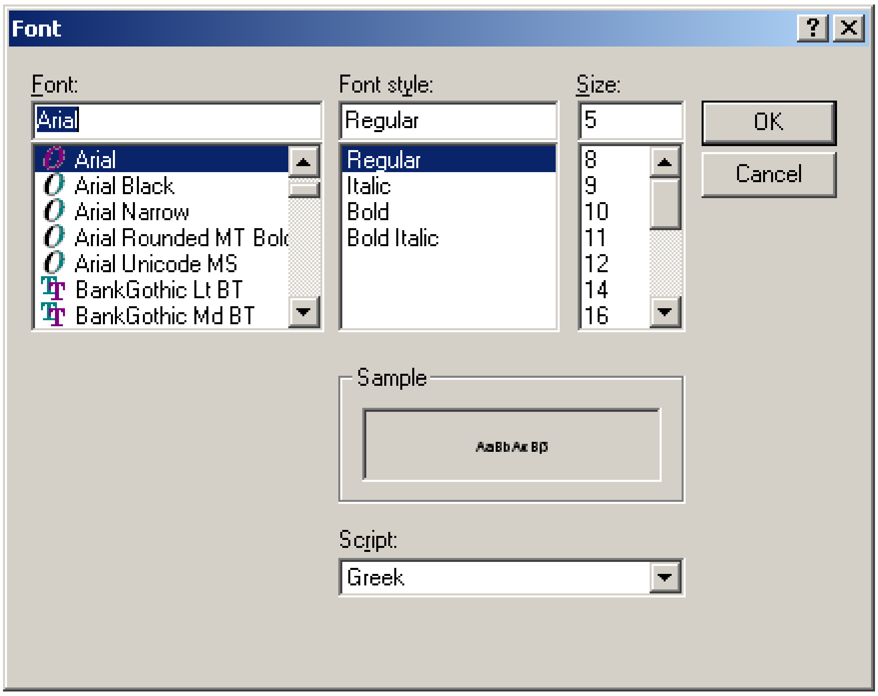
1. Antialiasing of figures.
2. Define the number of data shown when measuring the distance.
3. Select the fixed text font.
4. Filling the *Select Objects/Zoom* quadrilateral (window).

Text Font

You can define a font and a font size for the on-screen texts. Select the font from the *Font* dialogue box that appears when you click the button 

, in the *Display* tab of the *General Options* dialogue box.

To change the font or the font size – To use bold or italics in selected text or numbers

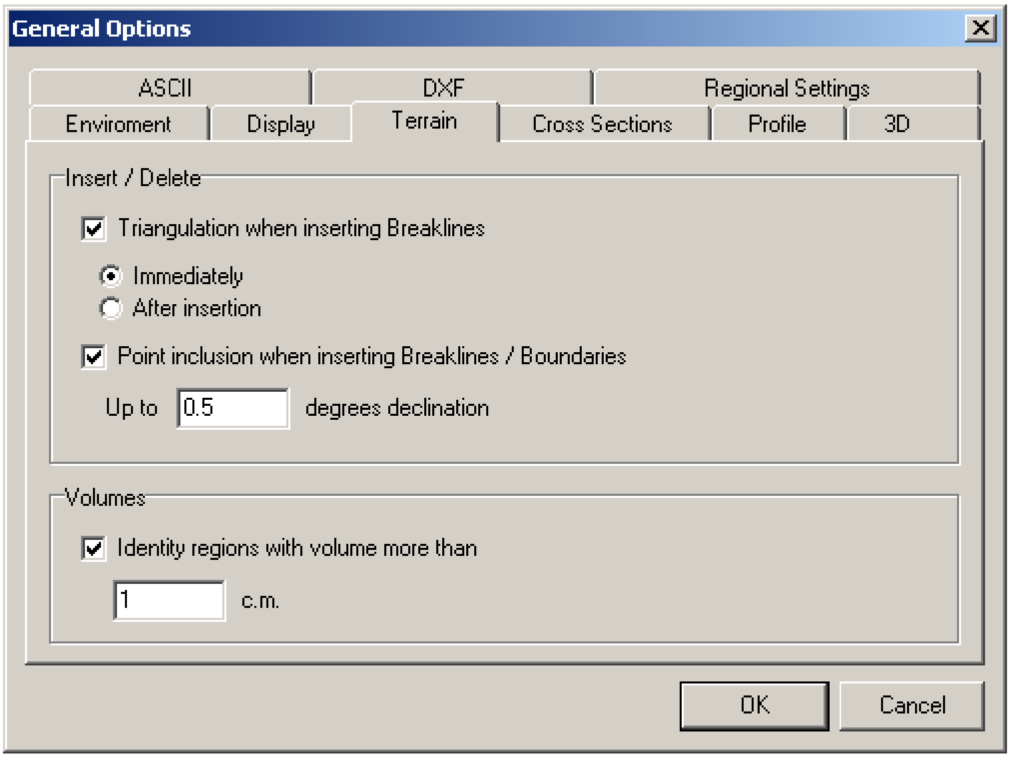


1. Select the desired font in the *Font* dialogue box of the *Font list,* and then select the desired Script from the *Script* list.
2. From the *Size* list, select the desired font size.
3. From the *Style* list, select the desired style.

The *Sample* preview field shows how the text will be displayed according to your settings.

Terrain Options

The *Terrain* tab in the dialogue box of the *General Options* command contains settings for the terrain.

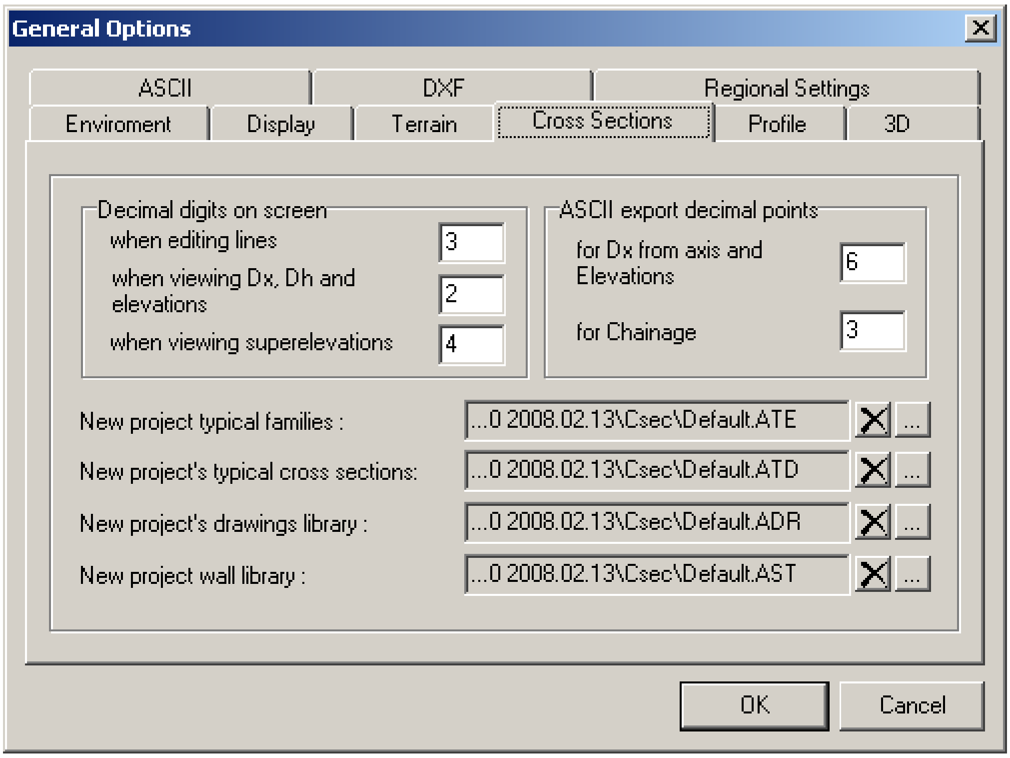


The settings included in this tab are:

- Instant triangulation when inserting Breaklines.
- Definition of the variation of the points that will be included automatically when inserting Breaklines or Boundaries.
- Definition of the highest value in m3 for the identification of the volumetric regions.

Cross section Options

The *Cross Section* dialogue box of the *General Options* command contains the Cross Sections settings.



The settings that you can make in the above tab are:

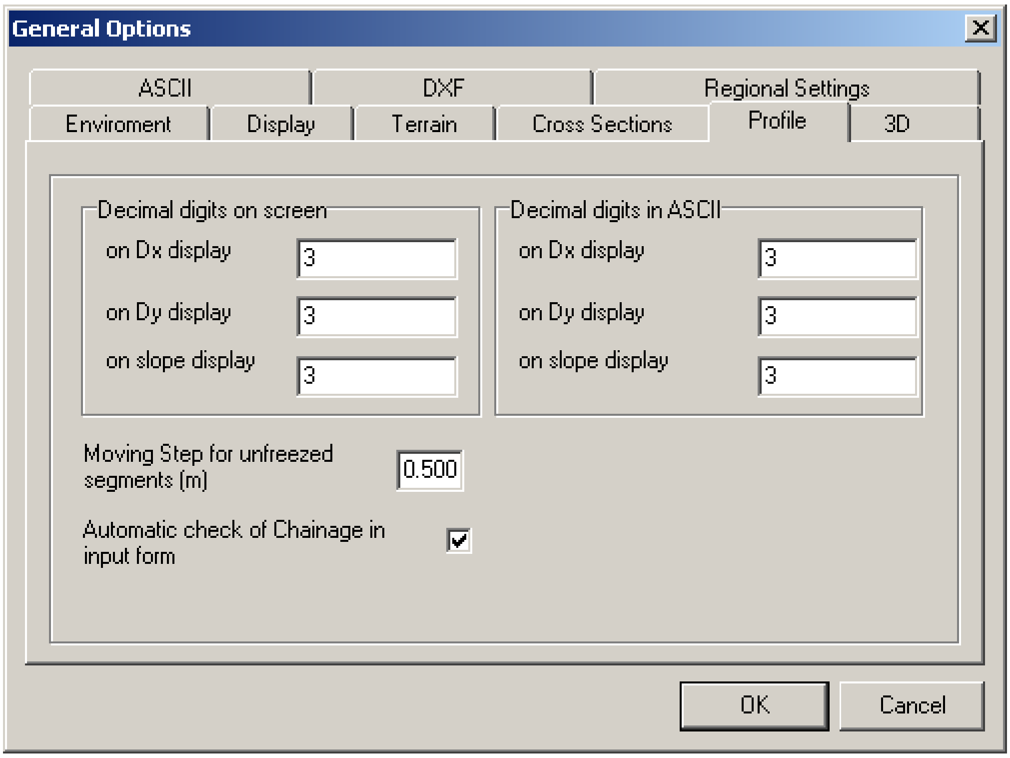
Set the decimal digits that appear on screen:
- when editing lines.
- when viewing Dx, Dh and elevations.
- when viewing superelevations.

Set the decimal digits for export to Ascii files:
- for Dx from axis and elevations.
- for chainage.

Select the files from which the following are obtained:
- the typical families for a new project.
- the typical cross sections for a new project.
- the drawings library for a new project.
- the walls library for a new project.

Profile Options

The *Profile* tab that appears in the dialogue box of the *General Options* command contains the Profile settings.



The settings you can make in this tab are:

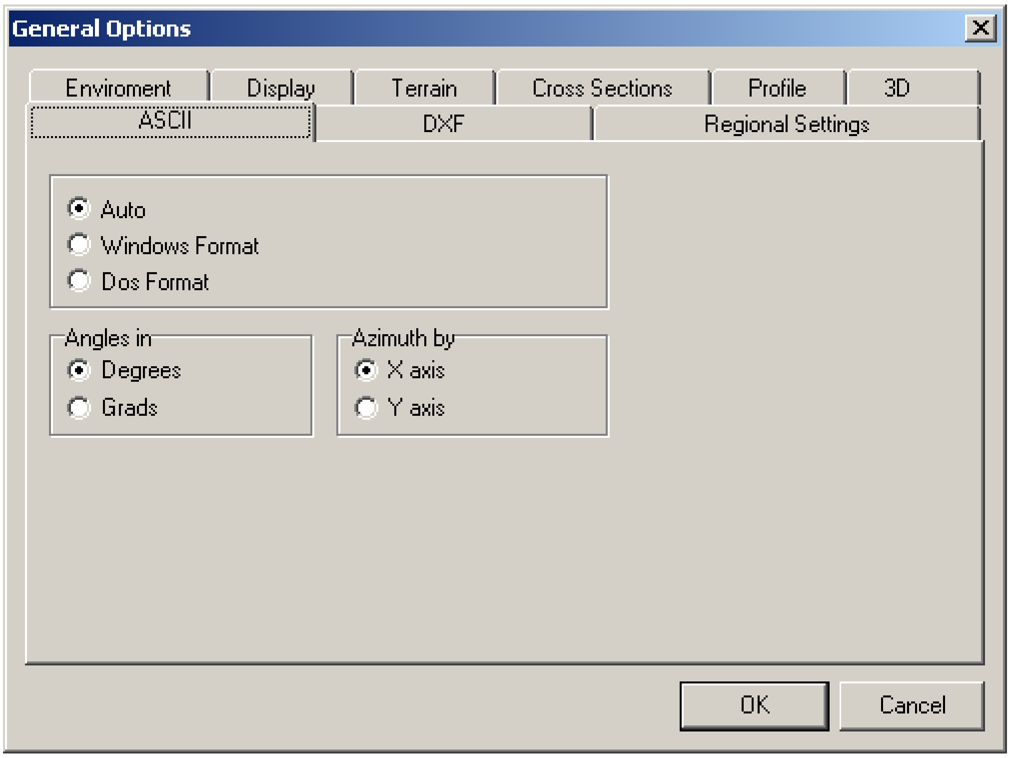
Set the decimal digits that are displayed on screen:
- on Dx display.
- on Dy display.
- on slope display.

Set the decimal digits for export to Ascii files:
- on Dx.
- on Dy.
- on slopes.

- Set the moving step for unfreezed segments.
- Enable the automatic check when inputting data based on chainage.

Ascii Options

The *Ascii* tab in dialogue box of the *General Options* command contains settings for data export to Ascii files.

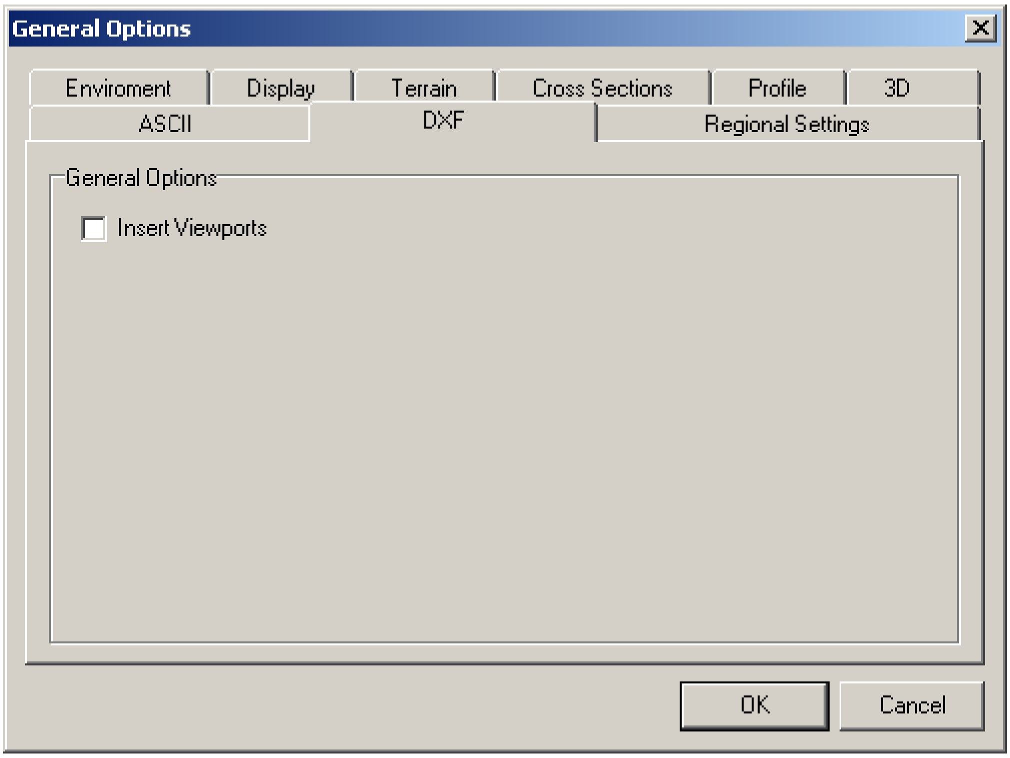


The settings you can make in this tab are:

- Selection of the text format (Windows / Dos) for the import or export of text files (For compatibility with older versions).
- Selection of the way the angles will be displayed.
- Selection of the way the angles will be displayed with respect to the x or y axes.

DXF Options

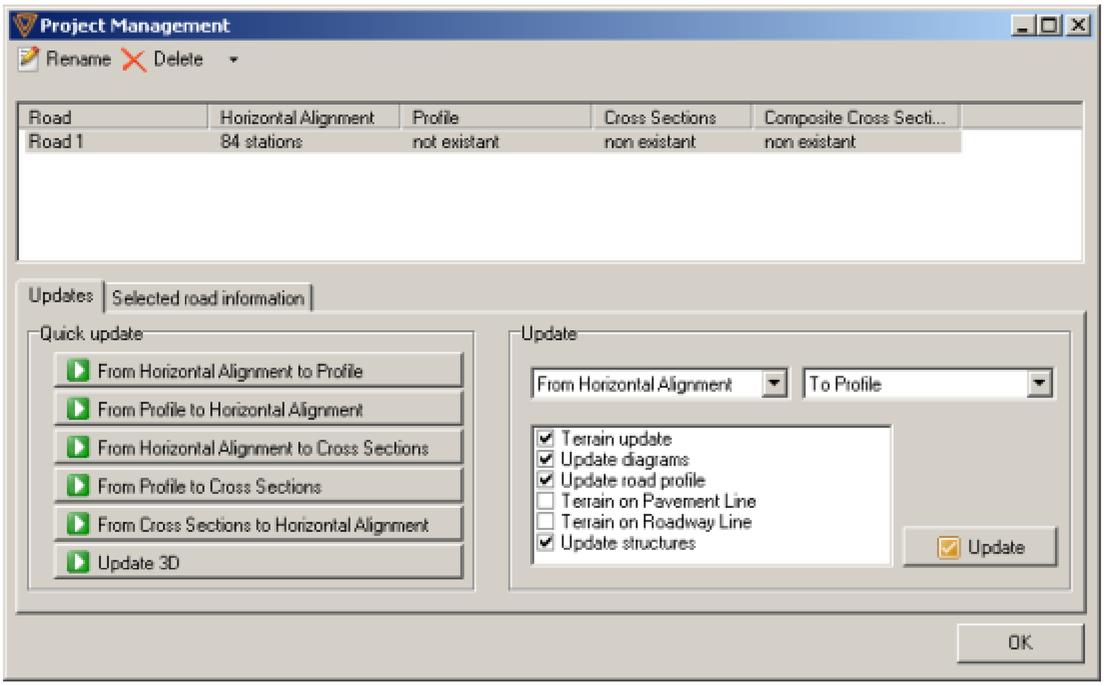
The *Dxf* tab in the dialogue box of the *General Options* command contains settings for data import from Dxf files.



Select *Import Viewports* when a file is not imported correctly.

6.18 Project Management

The communication between the three workspaces and the transfer of data from one workspace to another is done from the *Project Management* dialogue box.



To display the box, click the button on the main toolbar or execute the *Manage* command from the menu *Project*. In the dialogue box you can request the update of one workspace from another (e.g. *from Horizontal Alignment to Profile*). To update, first select the desired road from the list on the top part of the box. Next, if you wish to update with the main options, click the corresponding button of the *Quick Update* group in the *Updates* tab. This deletes the old data and creates all data from the beginning. Or, in the *Update* group, select from the two drop-down lists the workspaces where the updates will take place. All the updates that are to take place will be displayed in the checklist under the drop-down lists. You can uncheck any update you do not want to take place. In all cases, click the *Update* button, to carry out the update.

Selected Road Information

Once you have selected the road as described above and gone to the *Selected road information* tab, this dialogue box informs you on the updates that must take place so that the road is synchronized in all three workspaces.

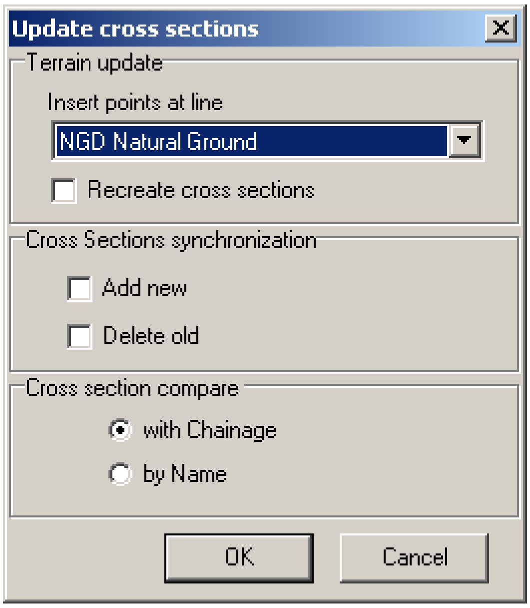
Update Cross Sections from Horizontal Alignment

If, from the drop-down lists of the Update group, you select to update from *Horizontal Alignment* to *Cross Sections*, the resulting options are:

- *Terrain Update:* Only if this command is active, the *Terrain* is updated. In case this update has already been performed once and you wish to update again, click the *Update* button. In the resulting dialogue box you can specify the update as well as the line to which the terrain points will be transferred.
- *Update Diagrams:* Select this command if you modify any of the diagrams in the *Horizontal Alignment* mode and you wish to update the corresponding ones in the *Cross Sections*.
- *Update Cross Sections:* Select this option to update the cross sections. A dialogue

box appears in which you can specify the actions to be performed.

The following dialogue box includes all the above windows that appear for the *Updates* (*Terrain Update, Update Cross Sections*). Namely, if you select only the *Terrain Update*, the window appearing includes the *Terrain Update* and *Cross section compare* groups, if you select only the *Update Cross Section*, the window includes the *Cross Section Synchronization* and *Cross section compare* groups. If you select to update all fields (*Terrain Update, Update Diagrams, Update Cross Sections*) the following window appears.



In this window, you can select from the drop-down list of the *Terrain Update* group the line onto which you want to insert the terrain points. With the *Recreate cross sections* option the program deletes any existing cross sections and updates again the terrain.
In case new cross sections have been inserted in the *Horizontal Alignment* mode, you can select the *Add new* field in the *Cross section synchronization*, in order to add these new cross sections to the *Cross sections mode*, without affecting the existing cross sections. In the opposite case, namely if you delete certain cross sections from the *Horizontal Alignment*, then in the aforementioned group you must select the *Delete old* field, in order to delete these cross sections without affecting the remaining ones.
Finally, in the *Cross section compare* group, you can select the way in which cross sections will be updated, i.e. either based on the *Chainage* or the *Name*.

Update Cross Sections from Profile

If, in the *Update* group, you select from the drop-down lists to update *from Profile to Cross Sections*, the resulting options are:

- *Update Road Profile:* This option transfers the road profile elevations to the *Cross sections* mode.
- *Create Cross Sections:* Deletes any existing cross sections and creates new ones, where the terrain is an horizontal straight section that crosses the axis and has an elevation equal to the *Terrain Profile*.
- *Update Diagrams:* Select this command if you modify any of the diagrams in the Profile mode and you wish to update the corresponding ones in the *Cross Sections*.

Update Profile from Horizontal Alignment

If, in the *Update* group, you select from the drop-down lists to update *from the*

Horizontal Alignment to Profile, the resulting options are:

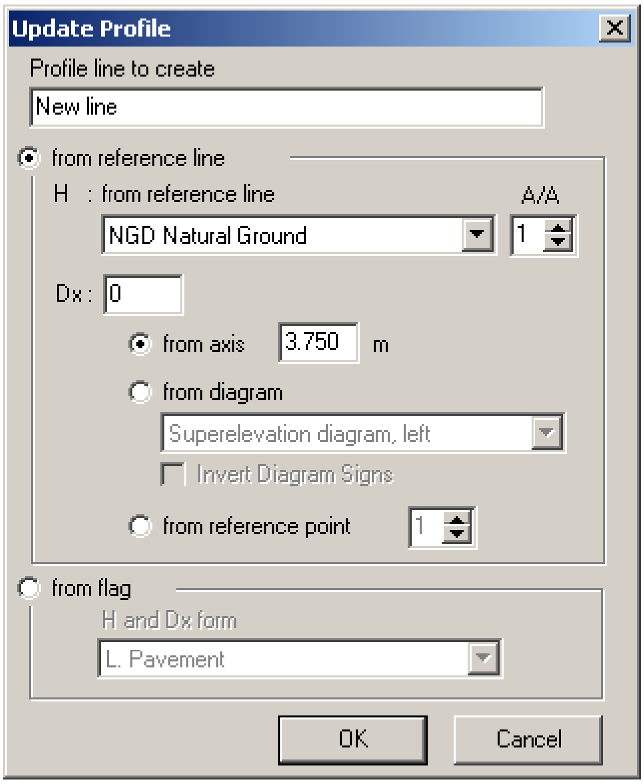
- *Terrain Update:* Only if this command is enabled, the *Terrain* is updated.
- *Update Diagrams:* Select this command if you modify any of the diagrams in the *Horizontal Alignment* mode and you wish to update the corresponding ones in the *Cross Sections*.
- *Update Road Profile:* In case you have deleted the road profile from the Profile, you can transfer it from the *Horizontal Alignment* but only if you first update the *Horizontal Alignment from the Profile*.
- *Terrain on Pavement line:* Provided that the widths of the road design have been defined correctly (Road Design → Road Preferences → Widths tab), you can transfer the *Pavement* lines to the *Profile* by selecting this command.
- *Terrain on Roadway line:* Here too, you must have defined correctly the widths in order to be able to transfer the *Roadway* lines to the *Profile* mode.
- *Update structures:* This command updates the structures in the *Profile*.

Update Profile from Cross sections

If, in the *Update* group, you select from the drop-down lists to update *from Cross Sections to Profile*, the resulting options are:

- *Terrain Update:* This command updates the *Terrain*.
- *Update Diagrams:* Select this command if you modify any of the diagrams in the *Cross section* mode and you wish to update the corresponding ones in the *Profile*.
- *Remote Update:* Use this command to display in the *Profile* elevations of characteristic *Cross sections* points (e.g. ditch deep point, pavement edge, etc.).

When you select the *Remote Update* command the following dialogue box appears:



In this window, type in the *Profile line to create* field the desired name of the new line that will be inserted into the profile.

- If the *from reference line* group is activated, the available fields will be:
  - *H: from reference line:* From the resulting drop-down list select the line whose elevations you wish to transfer to the profile diagram.

- *A/A:* In case there are more than one lines of the same group, select which line will be the reference line based on their order of import.
- *From axis:* Specify the distance from the road axis where the elevations of the reference line points will be obtained.
- *From diagram:* Select the diagram that will define the distance where the elevations will be obtained.
  Invert diagram signs: When diagrams referring to the left part of the cross section are going to be used, you must activate this field so that their values can be assigned to the left part of the cross section.
- *From reference point:* Depending on the selected reference line, you can specify which point of this line will transfer its elevation to the *Profile*.
- *Dx:* If you have selected *From diagram* or *From Reference Point*, you can define here an extra offset to the distance of the point whose elevation will be transferred to the *Profile*.

- From the *H and Dx from* the drop-down list of the from flag group, you can select the cross section point whose elevation will appear in the *Profile*.

Update Horizontal Alignment from Profile

If, in the *Update* group, you select from the drop-down lists to update *from Profile to Horizontal Alignment*, the resulting options are:

- *Update Road Profile:* This option transfers the PIs of the tangent curve to the *Horizontal Alignment*.
- *Update structures:* This option updates the structures in the *Horizontal Alignment*.

Update Horizontal Alignment from Cross sections

If, in the *Update* group, you select from the drop-down lists to update *from Cross Sections to Horizontal Alignment*, the resulting options are:

- *Update roadlines:* This update transfers the roadlines, e.g. the *Pavement* roadlines and the *Roadway* roadlines, to the *Horizontal Alignment* mode. A main prerequisite for this update is the correct placement of the flags in the *Cross sections*.

6.19 Saving a File

While working on a file, you must save it at regular intervals. If you want to create a new copy of your file without altering the original file, you can save it under a different name.

To save a file:

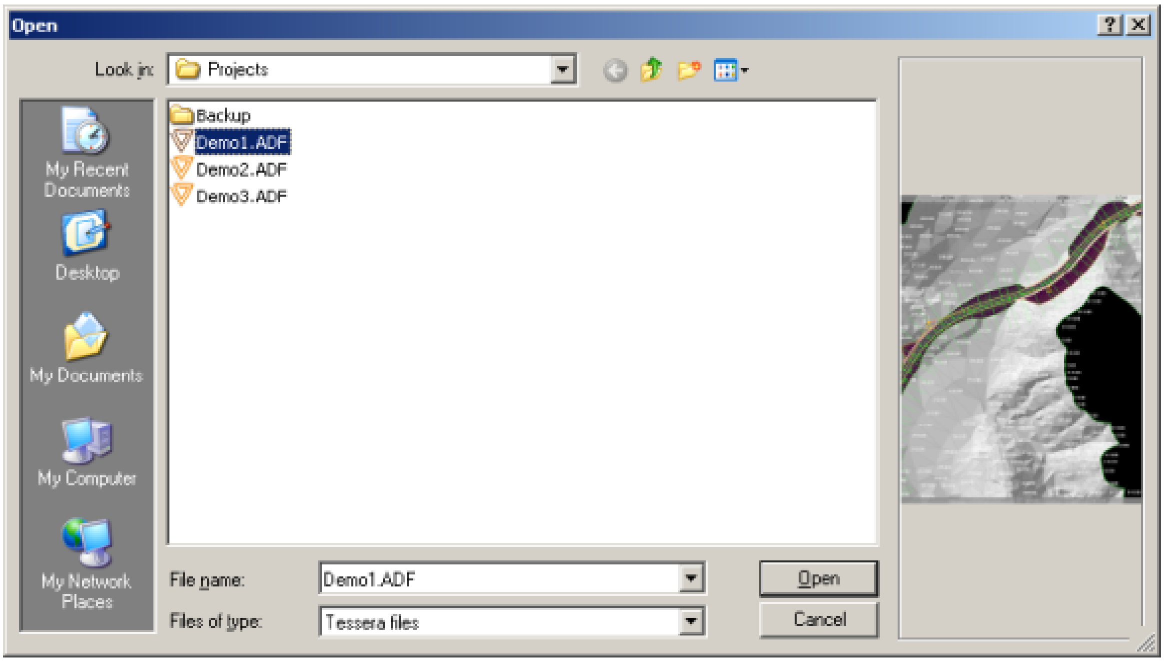
1. From the *File* menu, select the *Save* command, or press the *Ctrl+S* keys, or click the button  on the main toolbar . If you have already saved and named your file, *Tessera* automatically saves the changes you have made and you can continue editing the file. If this is the first time you save the file, the *Save as* dialogue box appears on the screen.
2. In the *File Name* field of the *Save as* dialogue box, type the new name of the file (the file extension is not required).
3. Click the *Save* button.

Tessera files have the .ADF extension.

6.20 How to Select and Save Files

The program will frequently ask you to give the path and the name of a file. This enables the program to read the file and use its data, e.g. when you open a program file or when inputting data.

You can do this from the *Open* dialogue box.



In this box, first you must go to the location (drive, folder, subfolder) where your file is. To do this, use the buttons and lists of the box.

To open a file, type the full path and name of the file in the *File* name field on the bottom part of the box and click the *Open* button. You can also open a file by double-clicking its name in the *Files* field or by clicking once and then clicking the *Open* button.
The files field, apart from the files of the current subfolder, also contains the subfolders included in the current subfolder. To open a subfolder, select it and press *Enter* or double-click with the mouse.
In the *File Name* field, you can enter the full name of the file or you can activate the drop-down list of the field. The list includes the files you have opened recently, since you last started the program. You can navigate inside this window by using the *PgDn* and *PgUp* arrows or the *Home* and *End* keys to go to the beginning or to the end of the window, respectively. Next press *Enter* to select the desired file and then press *Enter* again to open it. Of course, you can open the file far more quickly by double-clicking it. To cancel the selection, simply click the *Cancel* button or press the *Esc* key.
The *Save as* dialogue box has a similar function. To save a file, you must click the *Save* button.

6.21 Closing a File

The *Close* command closes the current file. If you have saved the changes you have made recently to the file, you can close *Tessera* without saving the file again. If you have not saved the changes, *Tessera* prompts you to do so, by displaying the relevant confirmation message.

1. On the Windows taskbar, left-click on the button that corresponds to the desired file, in order to display the file.
2. From the *File* menu, select the *Close* command or press the *Ctrl+F4* keys.

6.22 To Exit Tessera

The *Exit* command closes all open files and the program. If you have saved the changes you have made recently to all open files, you can close *Tessera* without saving the files again. You can also double-click the exit button on the title bar. If you have not saved the changes, *Tessera* prompts you to do so, by displaying the relevant confirmation message.

From the *File* menu, select *Exit* or press the *Alt+F4* keys.

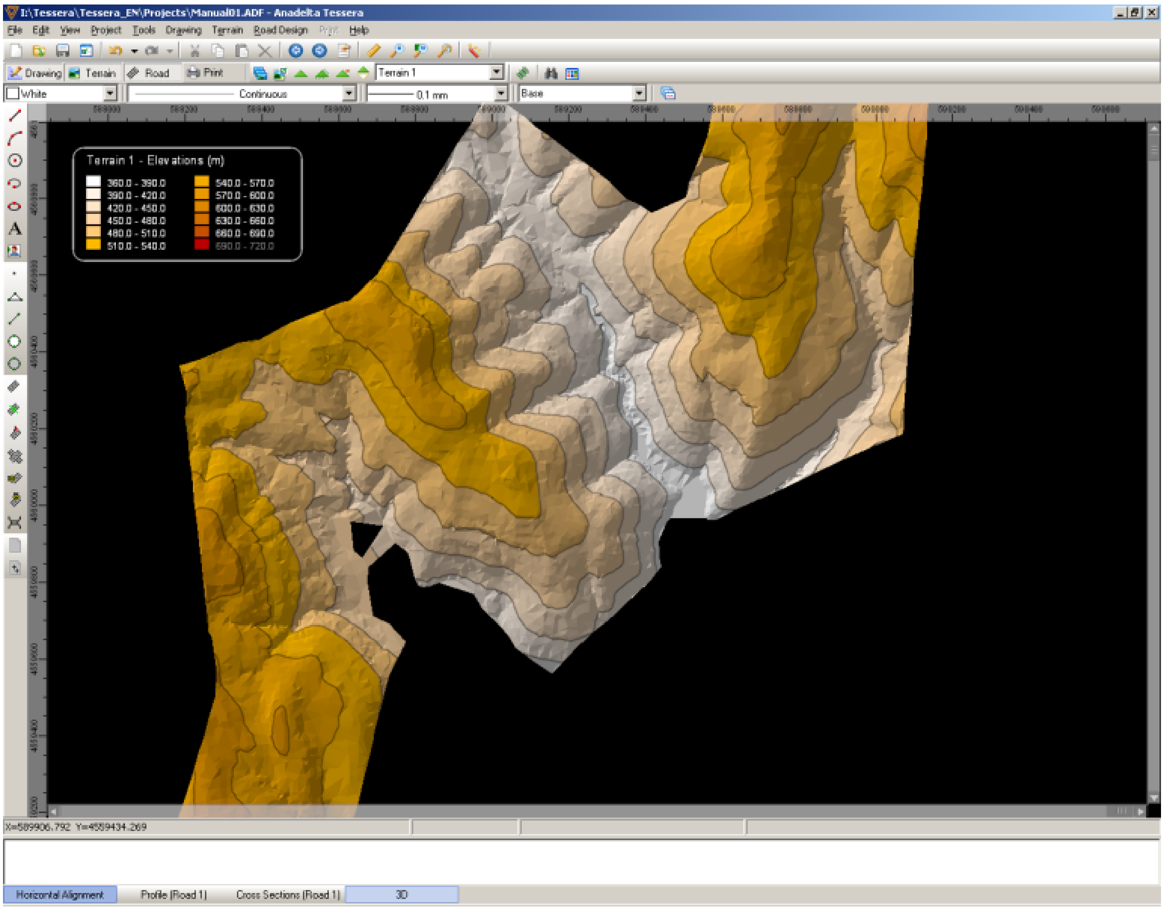
Chapter 7

7 Horizontal Alignment

7.1 The Horizontal Alignment Environment

7.1.1 Workspace

Every time you create a new file or open an existing one, you are transferred automatically to the *Horizontal Alignment* workspace. If you are in another workspace, you can return to the *Horizontal Alignment* by clicking the *Horizontal Alignment* tab at the bottom of the screen or by executing the *Horizontal Alignment* command from the *Display* menu.



At the top of the screen, you can see the familiar menu bar where you can find the *File, Edit, View, Project, Tools, Print* and *Help* menus, which contain commands that are common to all workspaces, as well as special commands that are displayed or can be used only in the *Horizontal Alignment* workspace. There are also three menus, *Drawing, Terrain,* and *Road Design,* which are displayed only when you are in this workspace. Below the menu bar, you will find the main toolbar and below the latter, you will find the *State* and *Horizontal Alignment* toolbars, which contain buttons that carry out basic operations. Right below these toolbars there is the *Object Properties* toolbar, where you set the properties of the objects, such as the color, the line type and the line width, the layer to which an existing drawing object belongs, the current terrain, as well as the properties that will be assigned to every new object that is being created. This toolbar does not have any buttons, only drop-down lists.

On the left part of the program's window, you will find the *Element Drawing* toolbar, whose functions refer to the creation of *Horizontal Alignment* objects. This toolbar offers access to the most common commands for the creation of elements referring to the design, terrain, road and printing pages.
If desired, you can maximize the drawing area so as to occupy the full screen of your computer. You can do this either by executing the *Full screen* command from the *Display* menu, or by pressing the *Ctrl+Alt+V* keys.

7.1.2 Workspace Modes

There are four modes in the *Horizontal Alignment* workspace. This allocation facilitates the user to insert, edit and process the elements of the design.

- *Drawing* Mode:
  In this mode you can enter and edit any of the drawing objects, such as lines, circles, ellipses, arcs, texts, etc. In this mode only the drawing objects can be selected on the drawing screen and you can execute only commands referring to such objects.
- *Terrain* Mode:
  In this mode you can insert - edit the terrain elements of your project and create the terrain model.
- *Road* Mode:
  In this mode you can insert - edit the roads of your project, while you can also perform the road stationing and the formation of the roadlines.
- *Print* Mode:
  Here you can specify the format, the elements that will be displayed and the part of your design that will be printed to the printer or the Plotter.

Mode Toolbar

Each of the four buttons on this toolbar corresponds to a mode. Every time you click one of these four buttons, a mode is activated and the other three modes are deactivated. It is possible to activate more than one mode.

How to activate multiple modes

Press and hold the *Shift* key and click the button that corresponds to the mode you wish to activate, maintaining active the already selected mode.

By default, the first three modes are always active every time you create a new *Tessera* file.

7.1.3 Categories of Objects

Every object you create belongs to a category of the *Horizontal Alignment* workspace. Most objects, even if they have a different shape, may belong to the same category. For example, a quadrilateral and a straight part are both drawing lines.
The object categories of the *Horizontal Alignment* workspace are described below. To view the identity of an object, either select the object and run the *Properties* command, or drag the crosshair over the object (the corresponding drawing mode must be active) and read its identity in the third part of the status bar.

Drawing Elements

1. *Drawing Line:* An open or closed zigzag line, comprised of one or more straight parts.
2. *Circle:* A circle.
3. *Circular Arc:* The part of a circle.

4. *Ellipse:* An ellipse.
5. *Elliptical Arc:* The part of an ellipse.
6. *Text:* A line of alphanumeric characters.
7. *Clothoid:* A line in clothoid shape.
8. *Parabola:* A line in parabolic shape.

The drawing elements are created by defining their main points, e.g. the centre and a point of a circle's circumference. These points are an integral part of the drawing lines and you can view and modify them at any time while editing your design.

Rendering/Terrain Elements

1. *Terrain Point:* A point whose position in space is defined by three numbers, the coordinates X,Y,Z.
2. *Triangle:* A triangular flat surface.
3. *Breakline:* A line on which the triangles always abut and they never intersect it.
4. *Perimeter:* A closed line demarcating an area that contains triangles.
5. *Oasis:* A closed line without any triangles.
6. *Contour lines:* Lines on which every point has the same elevation.

Road Elements

1. *Road:* It consists of PIs.
2. *PIs:* The points that define the sides of the road's tangent curve. The points of the PIs (e.g. radius, length of clothoid) determine the position and the form of the road axis. When you select a road, you actually select the closest PI.
3. *Roadway Lines:* These are lines, along the road axis, belonging to the roadway surface of the current road.
4. *Sideslope Lines:* These are straight lines, parallel to the road axis, belonging to the surface of the sideslopes on both sides of the current road.

You cannot create the *Roadway Lines* and the *Sideslope Lines* from the *Horizontal Alignment;* they are created automatically during the calculation of the road and the update from the *Cross Sections* workspace. You can, however, select them and set some of their main properties, display them on screen, use the snap feature on them and print them as part of your final drawing.

Print Elements

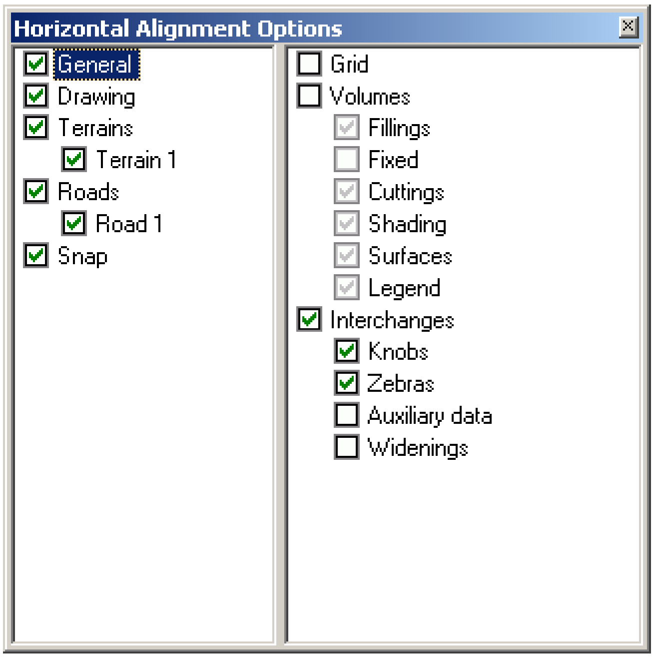
Page: A rectangular surface whose dimensions correspond to the dimensions of the paper page that you have set, always according to the print scale. The part of the design that is included in the selected page is the one that will be printed when you request a printout.

7.1.4 Horizontal Alignment Display Options

You can choose which object categories or which parts of the objects (e.g. line points) will be displayed on screen.

How to define how the objects are displayed

1. From the *Display menu,* run the *Options* command, or press the *F2* key, or from the *Horizontal Alignment* toolbar click the button . The *Horizontal Alignment Options* window appears on the screen.

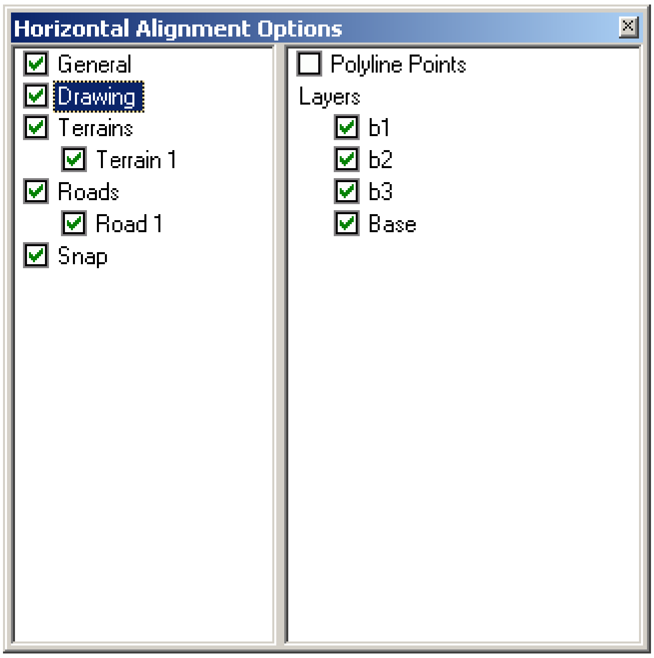


The window consists of two parts. The object categories are displayed on the left part.

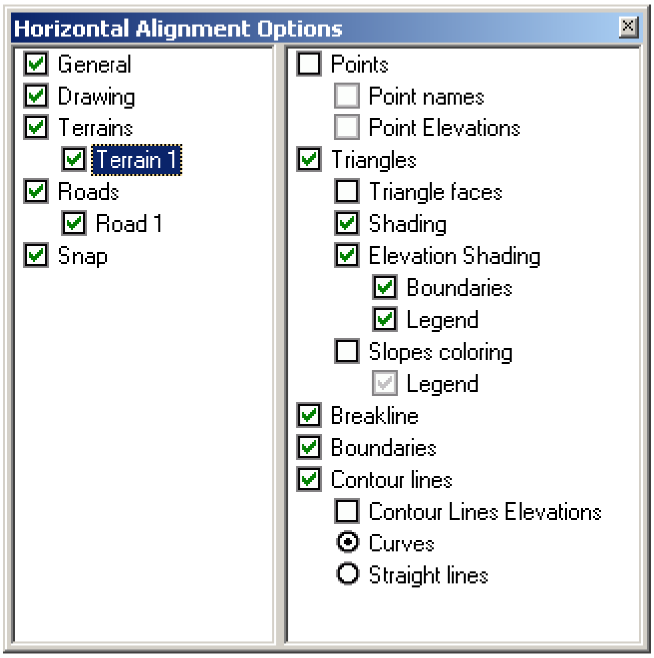
2. Select an object category.
3. If you want none of the elements of the category to be displayed, uncheck the box next to the category name on the left part of the window. If you want only some of the elements of a category to be displayed, select them from the right part of the window by ticking the corresponding check box, which is displayed next to their name.
4. To exit the window, click the exit button  on the title bar of the window.

When the *General* option is ticked, the available options are:

- *Grid:* Shows or hides the grid.
- *Volumes:* This option opens the legend with the volumetric differences (fillings, cuttings, unalterables).
  - *Fillings:* Shows or hides the fillings.
  - *Unalterable:* Shows or hides the unalterables.
  - *Cuttings:* Shows or hides the cuttings.
  - *Shading:* Select this field to render the regions with hatch instead of shading.
  - *Surfaces:* Select this option to hide from the legend the surfaces that cover the fillings, the cuttings and the unalterables.
  - *Legend:* Shows or hides the Volumes legend.



When the *Drawing* option is ticked, the available options are:
- *Polyline Points:* Shows or hides the polyline points.
- *Layers:* This option includes all the available drawing layers that you can enable or disable.



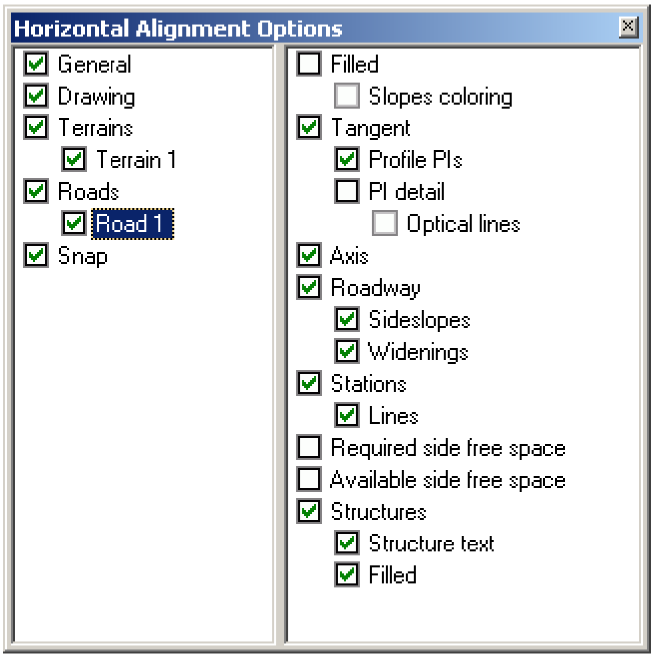
When the *Terrain* option is ticked, the available options are:
- *Points:* Shows or hides the Terrain points.
  - *Point names:* Shows the names of the Terrain points.
  - *Point elevations:* Shows the elevations of the Terrain points.

When both of the above options are ticked, the names of the points are displayed above the bordeline and the elevations are displayed below the borderline.
- *Triangles:* Once the triangulation is performed, the resulting triangles are displayed.
  - *Triangle faces:* Shows the faces of the triangles.
  - *Shading:* This option shows the morphology of the terrain.
  - *Elevation shading:* Shows the relief in a coloured presentation according to the

elevation of points.
- *Boundaries:* Shows or hides the boundaries of the elevational zones.
- *Legend:* Shows or hides the Elevation Shading legend.
- *Slopes coloring:* Shows the terrain slopes in a coloured presentation.
  - *Legend:* Shows or hides the Slopes Coloring legend.

Note: The *Elevation Shading* cannot be displayed when the *Slopes Coloring* is displayed and vice versa.

- *Breakline:* Shows the breaklines.
- *Boundaries:* Shows the boundaries that have been drawn (Oases, Perimeters).
- *Contour lines:* You can display the contour lines once their calculation has been carried out.
  - *Contour Lines Elevations:* Shows the elevations of the contour lines.
  - With the *Curve* or *Straight Lines* option, you can define the way in which the contour lines will be rendered.



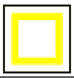
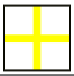
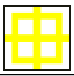
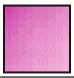
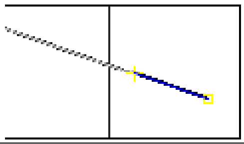
When the *Road* option is ticked, the available options are:
- *Filled:* This option draws the roadway.
  - *Slopes coloring:* Shows the roadway slopes.
- *Tangent:* Shows or hides the road's tangent curve.
  - *Profile PIs:* Once the Profile is updated from the Horizontal Alignment, you can display the PIs of the profile.
  - *PI Detail:* Shows the centre of the circular arc of every PI, as well as the radii that correspond to the characteristic stations.
    - *Optical lines:* When a PI of the road is selected, this option displays the optical lines that define the surrounding visibility.
- *Axis:* Shows or hides the road's axis.
- *Roadway:* Shows or hides the roadway.
  - *Sideslopes:* Once the Horizontal Alignment is updated from the Cross Sections, you can display the sideslopes (fillings and cuttings).
  - *Widenings:* Shows any widenings that may have been drawn.
- *Stations:* Once the stationing is performed, you can display the stations.
  - *Lines:* Shows the lines that define the width of each cross section.
- *Required side free space:* Once the visibility calculations are performed, you can display the *Required Side Free Space*.
- *Available side free space:* This option displays the *Available Side Free Space*.

7.1.5 The Snap Feature

You can only snap items shown on the screen. You can not use the snap feature on drawing elements that belong to hidden or non-selectable layers.

The snap feature uses visual assistance to help you distinguish the snap type and use the snap feature more effectively. The visual assistance is comprised of the following items:

1. *Notations:* They indicate the type of snap by displaying the corresponding symbol at the point of snap.

On line point, terrain point, station or PI:	
On line:	
On grid:	
On line intersection:	
On extension:	

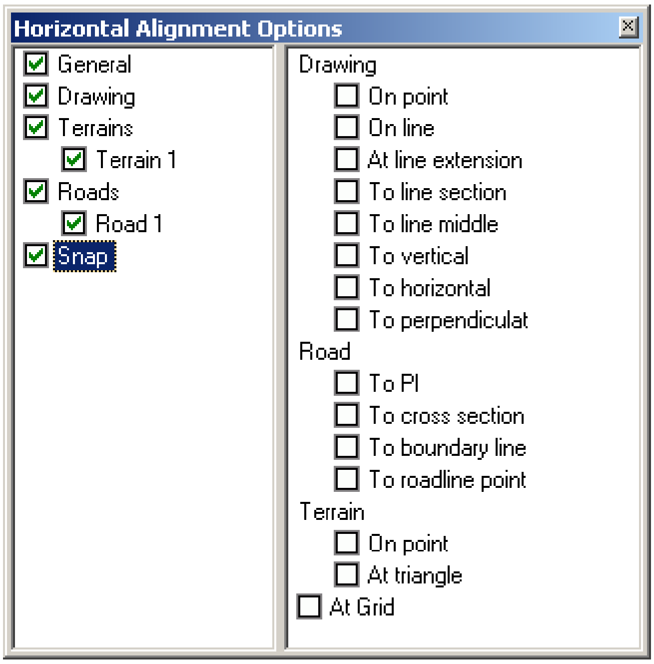
2. *Magnet:* Moves the pointer automatically to lock it on a snap point, when the pointer is close to this point.

With the snap feature you have the following possibilities:

1. Snap *drawing* elements:
- On Line point
- On Line
2. *On Extension:* to snap the extension of a line, place the crosshair over a point of the line and then move it to the desired side. Note that in order to use this type of snap, you must also select the snap on point.
3. *On Line section:* if the lines are not intersected, you must also select both the snap on point and the snap on extension.
4. Snap *terrain* elements:
- On terrain point
- On triangle line
5. Snap on *grid*
6. Snap *road* elements:
- On road PI
- On road cross section

How to activate and set the snap feature

1. From the *Display* menu run the *Options* command, or press the *F2* key, or from the *Horizontal Alignment* toolbar click the button . The *Horizontal Alignment Options* dialogue box appears on the screen.
2. From the left part of the window select *Snap*.
3. Select the type/s of snap you want to use, by ticking the relevant check box, next to the name of the snap on the right part of the window.
4. To close the window, click the button .



When the *Snap* option is enabled, you can see all the fields that you can snap:

- *Drawing:* You can select one or more of the above available fields, in order to use the snap feature in the drawing mode.
- *Road:* Select the fields you wish to snap.
- *Terrain:* Select the desired type of snap.
- *On grid:* When this field is selected and the grid is enabled, you can use the snap feature on any point of the grid.

If certain options are disabled, this is because from the *Display* tab you have chosen not to display them. To activate them, go to this tab and choose to display them.

7.1.6 Selection of Objects

To select the objects, the corresponding mode must be activated. Therefore, for the drawing elements you must activate the *Drawing* mode, for the road elements you must activate the *Road* mode and for the print elements you must activate the *Print* mode.

Very often, especially when multiple modes are simultaneously active, it is difficult to select an object that is too close to or on top of another. In such cases, move the crosshair over the point where the desired object is and click the right mouse button. A shortcut menu will appear on the screen. This shortcut menu contains the *Object selection...* commands, which are as many as the selectable objects. Run the command that corresponds to the object you want to select.

7.1.7 Main Object Properties

With *Tessera* you can edit the main properties of the drawing objects in two ways: either from the *Object* properties toolbar or from the *Properties* command window.

The Object Properties Toolbar

You can use the lists of the *Object* properties toolbar to view directly or to change the main properties of a drawing object, such as the *Line type*, the *Line width* and the *Color*. The *Object* properties toolbar includes all the commands that you need in order to view and change the main properties of an object. When you select an object, its properties are displayed automatically in the drop-down lists of the toolbar.

Color Editing

From the *Color* field (the first field), you can view and change the color of the selected object.
The drop-down menu includes the *Layer* colors (only for the drawing elements), the fifteen default colors, and all the colours you have defined from the dialogue box of the *Color* command. If the desired color is not on the list, select the *Other* option and define a new color from the dialogue box of the *Color* command.

When no objects are selected, the *current color* is displayed. When you create new drawing objects, these objects will appear in the current colour.

When multiple objects are selected, if all the drawing objects have the color property and are of the same color, then this color is displayed, otherwise the field is blank. If only drawing objects are selected and their colour is the one of the layer, the *Layer* option is displayed in the field. The *Layer* value means that the color of the object is the same as the color you have defined for the layer to which the object belongs.

To change the color of an object

1. Select the objects whose color you wish to change.
2. From the *Object* properties toolbar, activate the color selection drop-down list.
3. Select a color.
4. If you do not see the desired color, select *Other*.
5. In the *Color* command dialogue box, select the desired color.
6. Click *OK*.

Tessera will change the color of the selected objects into the color that you just selected.

Line Type Editing

From the *Line type* field, you can view and change the line type of a selected object, and set the current line type.

The drop-down list includes the *Layer* line type (only for the drawing elements) and the default line types.

When no objects are selected, the current line type is displayed. The new drawing objects that will be created, will have the current line type.

When multiple objects are selected, if all the drawing objects have the line type property and the same line type, then this type is displayed, otherwise the field is blank. If only drawing objects of different layers are selected and they all have the layer line type, then the *Layer* value appears in the field. The *Layer* value means that the object's line type is identical to the one you have defined for the layer to which the object belongs.

To change the line type of an object

1. Select the objects whose line type you wish to change.
2. From the *Object* properties toolbar, activate the line type selection drop-down list.
3. Select a *Line type*.

Tessera will change the line type of the selected objects into the line type you just selected.

Line Width Editing

From the *Line width* field, you can view and change the line width of a selected object, and set the current line width.

The drop-down list includes the *Layer* line width (only for the drawing elements) and the five default line widths.

When no objects are selected, the current line width is displayed. The new drawing objects that will be created will have the current line width.

When multiple objects are selected, if all the drawing objects have the line width property and the same line width, then this value is displayed, otherwise the field is blank. If only drawing objects of different layers are selected and they all have the line width of the layer, then the *Layer* value is shown in the field. The *Layer* value indicates that the line width of the object is the same as the one you have defined for the layer to which the object belongs.

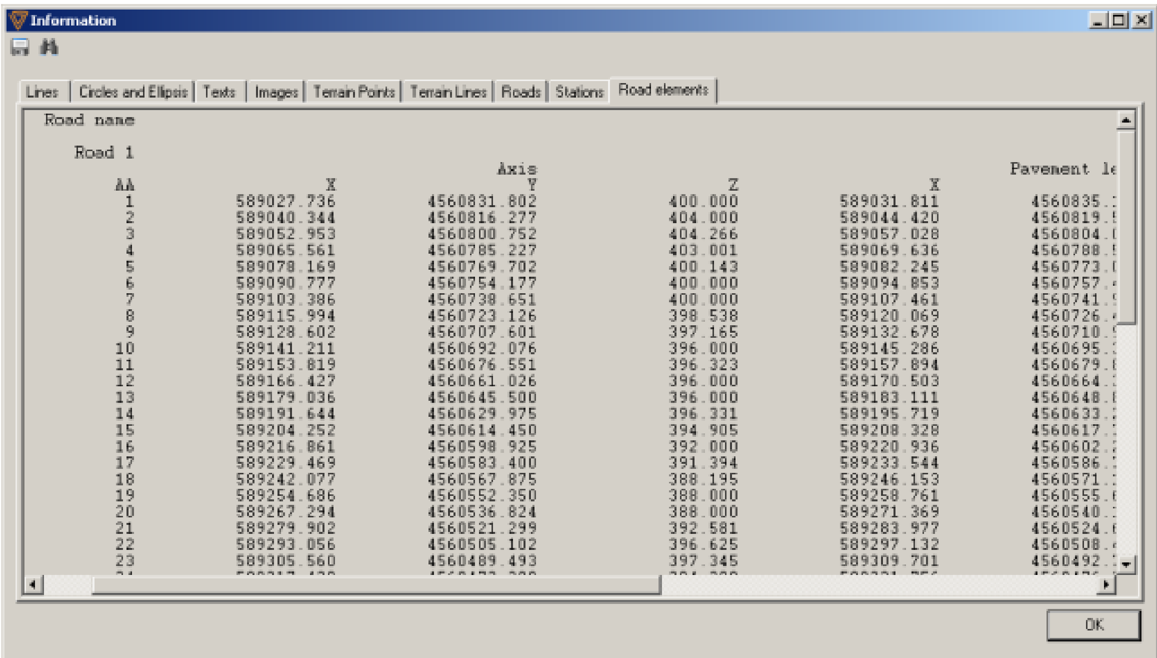
To change the line width of an object

1. Select the objects whose line width you want to change.
2. From the *Object* properties toolbar, activate the line width selection drop-down list.
3. Select a line width.

Tessera will change the line width of the selected objects into the line width you have selected.

7.1.8 Project Information

You can view the data of all objects of the project in the dialogue box of the *Project data information* command of the *Tools* menu. This command opens the following dialogue box:



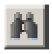
This box shows all information about the objects of all project categories of the

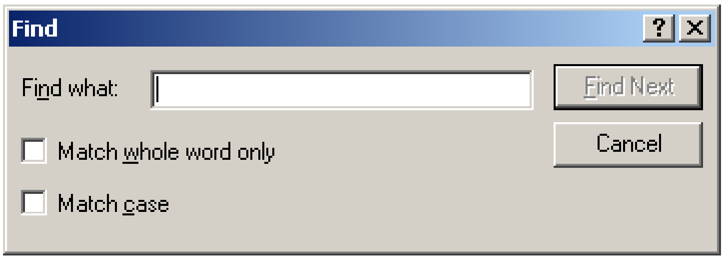
Horizontal Alignment workspace. Each tab of the dialogue box corresponds to an object type.

You can use the dialogue box of the *Find* command to find easily any desired object:

Tessera searches the text you have typed in the field throughout the selected tab. If desired, you can request from the program to find whole words only.

How to search a text in the Information dialogue box

1. Go to the desired tab (e.g. Terrain Points) and click the button  on the top of the dialogue box. The *Find* dialogue box will appear on the screen.

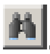


2. In the *Find what* text field, type the text you want to find. If you want to find whole words only, then you must tick the *Match whole word only* check box.
3. Click the *Find Next* button. *Tessera* will select the first text that matches the criteria you have specified. If you want to continue the search to find the next text, click the same button again. Otherwise click the *Cancel* button to close the dialogue box.

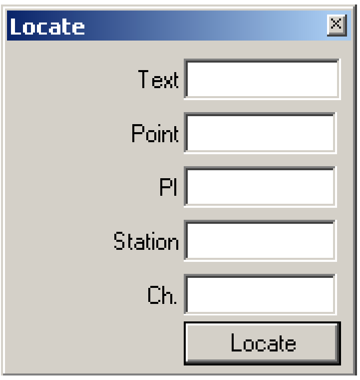
When the search is over, Tessera will display the relevant message.

If desired, you can export to a text file the information you see in a tab. Simply go to the tab whose data you want to save and click the button  at the top left of the dialogue box. The *Save as* dialogue box will appear on the screen. In this box you must specify the path and the name of the file that will be created. When you click the *Save* button, *Tessera* will create a text file that will contain the tab data and will have the .txt extension. You can set the character coding from the *ASCII* tab that you will find in the dialogue box of the *General Options* command of the *Tools* menu.

7.1.9 Quick Find

You can also use the *Quick Find* command to locate the desired objects within the drawing area. You can run this command from the *Display* menu by clicking the button .

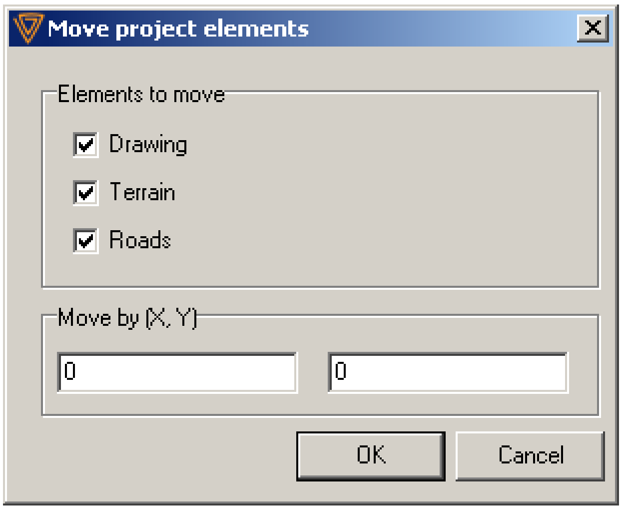
This part allows you to move among the project's elements (*Text, Point, PI, Station section and Ch*).



From here you can go to any element by simply entering its name. If, for example, you want to go to point 389 (i. e. to make it current), all you have to do is type this number in the relevant field and click the *Locate* button or press the *Enter* key regardless of the mode you are in.

7.1.10 Moving Project Elements

In the *Horizontal Alignment* mode, you can move certain elements of the project by running the *Move* command of the *Tools* menu. This command opens the following window:

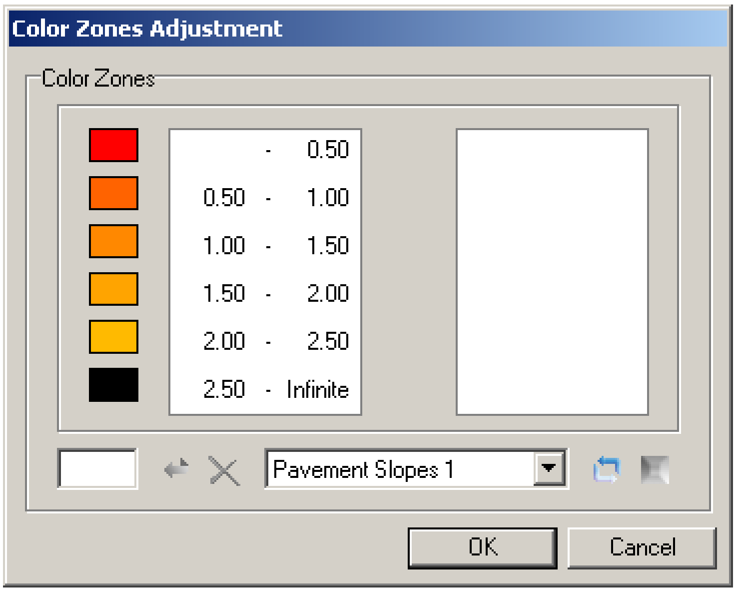


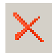
Select the element you wish to move (*Drawing, Terrain, Roads*) from the window.
In the *Move by (X, Y)* field, type in meters the desired distance by which the move will take place on the *X* and *Y* axes.

7.1.11 Colour Zones

Color Zones Adjustment

In the *Horizontal Alignment* mode you can set the colour zone of the road and terrain slopes by running the *Color Zones Adjustment* command of the *Tools* menu. When you run this command, the following window appears on the screen:



- To enter a new *Color Zone*, type in the field  the number that will represent the initial value of the *Zone* and then click the button  to register the value or press the *Enter* key.
- To delete a *Colour Zone* you must first select the *Zone/s* that you don't need by left-clicking the colored boxes. By selecting even one *Zone,* the button  is enabled. To delete the zone, click this button or press the *Delete* key.
- If you want to reverse the colours of the *Zones* (e.g. in the above figure, if you want black to become the first color and red the last color), click the button .
- If you want to place a specific colour in a *Zone,* right-click on any box to open the color palette as shown in the figure below:

Select the desired color and click *OK* to confirm.

- To create a color shading based on specific colors (e.g. shades of red) you must first select the boxes that will be the standard colors (e.g. red will be the first color and white will be the last one) and then click the button .

7.2 Topographical Background - Workspace - Terrain

7.2.1 Creating the Terrain Model

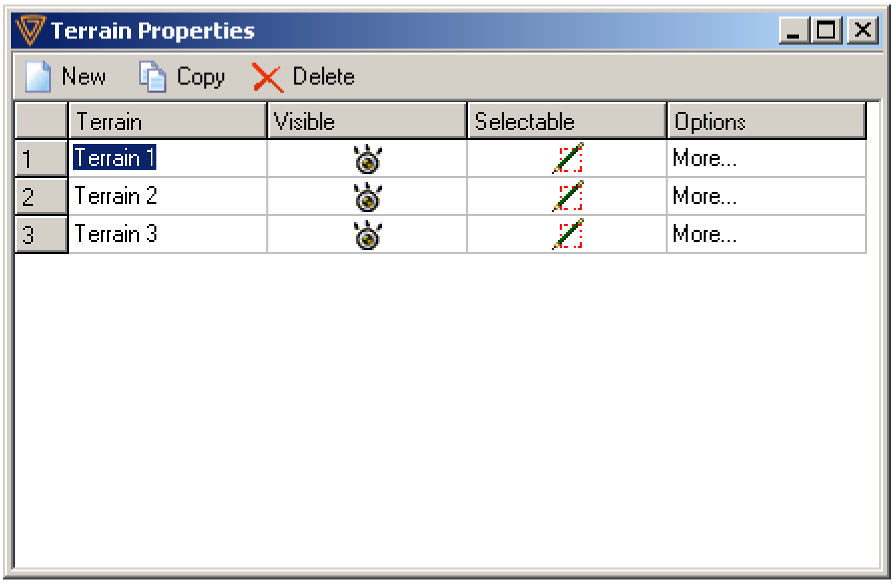
The terrain model consists of points, triangles, contour lines, breaklines, perimeters and oases. In the *Horizontal Alignment* workspace of *Anadelta Tessera* you can import or create the terrain model elements and edit them so as to optimize them in various ways. You can create the terrain model in the following ways:

1. Directly from the *Horizontal Alignment* workspace.
2. Import from text files (*ASCII*).
3. Import from *DXF* files.

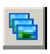
7.2.2 Terrain Management - Properties

The *Professional* version supports the simultaneous existence of multiple terrain models. That means that it is possible to create more than one terrain in the same file. It is also possible to calculate the volumetric difference of the two terrains.

Terrain management is performed from the *Terrain Properties* management window. Here you can run all main commands referring to terrains, e.g. to create or delete a terrain.



How to open and close the Terrain Properties management window

1. Click the button  on the *Horizontal Alignment* toolbar. The management window appears on the screen.
2. Proceed to the desired modifications.
3. When you have finished, click the exit button to close the management window and return to the main screen.

At first, the management window contains only one terrain, which cannot be deleted. You can create new terrains and name them as you wish.

How to create a new terrain

1. Open the *Terrain Properties* management window, as described above.
2. To create a new terrain whose elements will be inserted by you, click the *New*

button 

.

A new terrain will be created. Its name will be comprised of the word *Terrain* and a serial number, e.g. *Terrain 1, Terrain 2,* etc. Namely, every terrain has a unique name. Once the terrain has been created, you can change its name.

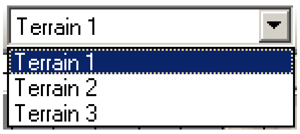
To change the name of a terrain

1. Open the *Terrain Properties* management window.
2. Left-click on the terrain name to activate the relevant field.
3. Type in a new name.
4. When you have finished, click the exit button to close the management window and return to the main screen.

All operations regarding the terrain (e.g. Point import) take place in the current terrain. A drop-down list in the *Object Properties* toolbar allows you to define the current terrain.

How to make a terrain current

Select the current terrain from the *Terrain* drop-down list of the *Object Properties* toolbar.



The program allows you to create a true copy of the terrain you are editing.

How to copy a terrain

1. Open the *Terrain Properties* management window, as described above.
2. To create a new terrain, which will be a true copy of an existing one, left-click on the terrain name to activate the relevant field and then click the *Copy* button 

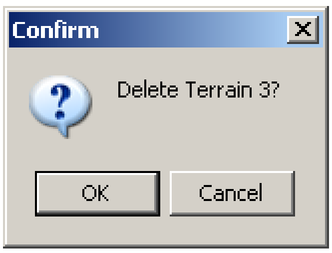
.

You can also delete a terrain.

How to delete a terrain

1. Open the *Terrain Properties* management window, as described above.
2. To delete a terrain, left-click on the terrain name to activate the relevant field and then click the *Delete* key 

.
3. If you wish to delete the terrain, click *OK* on the confirmation message that will appear.



Very often, while you are working, you may choose to make certain terrains invisible, or while editing a terrain you may choose the other terrains to be displayed without being editable. You cannot edit or select the elements of a non-selectable terrain, but the elements are visible when the terrain is visible. It is possible to use the snap feature on objects of a non-selectable terrain. The terrains that are not visible will not be displayed on the screen, but you can still print them.

How to make a terrain visible or invisible/ selectable or non-selectable.

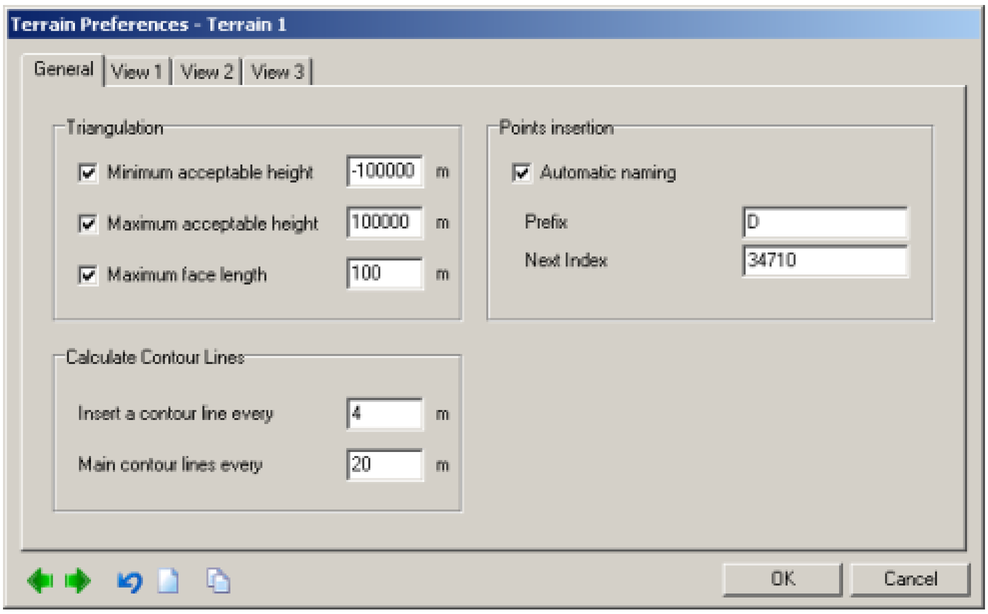
1. Open the *Terrain Properties* management window.
2. Select the fields of the terrains you want to make visible or invisible.
3. Double-click on the Visible field to change the icon so that it corresponds to the desired state. When a terrain is visible, then the icon looks like while when it is not visible the icon looks like .
4. Select the fields of the terrains you want to make selectable or non-selectable.
5. Double-click on the *Selectable* field to change the icon so that it corresponds to the desired state. When a terrain is selectable, then the icon looks like while when it is non-selectable it looks like .
6. When you have finished, click the exit button to close the management window and return to the main screen.

7.2.3 Terrain Options

Every terrain has its own elements, i.e. points, lines and triangles, as well as its own settings – options and properties, regarding the calculation of the triangles and contour lines, the insertion of points and the way the terrain is displayed on the screen.

How to change the options of a terrain

1. From the *Terrain* menu, select the *Terrain Properties* command, or from the *Terrain Properties* management window, enable one of the fields of the *Options* column and click the button with the three dots .
2. Use the *Previous Terrain* or *Next Terrain* buttons , at the bottom right corner of the dialogue box to go to the terrain whose settings you want to change. Make sure that the name of the terrain appears on the title bar of the dialogue box.



3. Go to the desired tab and make your settings.
4. If you wish to restore the settings that applied before making your changes, click the *Restore* from *Horizontal Alignment* button at the bottom right corner of the dialogue box. If you wish to restore the default settings that the program applies to every new terrain, click the *Set default settings* button . Finally, for the *Professional* version, if you want the current terrain settings to be copied to the other terrains, click the *Copy to all terrains* button. .
5. To exit the dialogue box click *OK* to confirm the changes you have made or click *Cancel* to return to the main screen.

7.2.4 Terrain Points

Terrain points appear on the screen as grey dots. You can create them one by one or import multiple terrain points from text or *.DXF* files. Terrain points have four properties: their *x, y, z* coordinates and their names.

They can be imported in the same file and in three ways. Namely, some points can be imported manually and some from a text file. They can also be imported in more than one phases, i.e. you can import points twice from a text file.

The terrain points are the base for the creation of the terrain model. All other elements (triangles, breaklines, etc.) refer to them and they cannot exist without them. The program has certain restrictions regarding the points. For example, it is not possible for two points to have the same group of three coordinates or to be less than one centimetre away from each other.

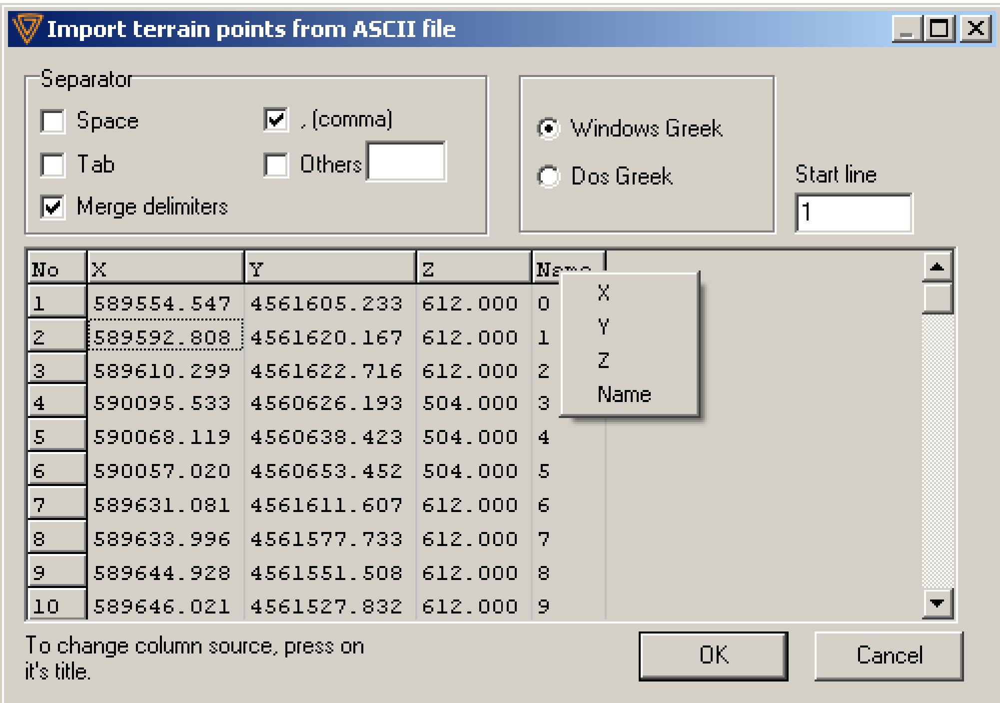
Importing Terrain Point from ASCII Files

The import of terrain points from text files is the most common, since the files that derive from the various recording instruments are in such format. Every line of such a file must correspond to a point and include the values of the four elements of the points. The elements of every point must appear in the same order from line to line.

How to import terrain points from a text file

1. From the *File* menu, select the *Import from ASCII* submenu and then run the *Terrain Points* command. This command will be displayed on the menu only if you first select the creation of a new file or open an existing one. In the resulting dialogue box you can select a file of terrain points (usually these files have the .XYZ and .GRD extensions) containing the X and Y coordinates, the elevation of the points and their names, which can be their serial number or a description (eg. S5).
2. From the *Browse* field find the subfolder where your file is located. If your file has a different extension than the ones stated above, select *All files* from the *File Type* drop-down list to display it. Select the desired file and click *Open*. Note that your file must not be open in another program, otherwise an error message may appear.

3. The program will try to sort the file's data in four columns. At the bottom of the dialogue box, you can view the form of part of the file namely the order in which the X, Y coordinates, the terrain elevations and the names of the points have been imported. In the *Separator* group you must select the symbol/s that will be used for the separation of the point elements into columns, by ticking the relevant check boxes. The most common separators are the space, the comma and the Tab symbols. If another symbol is used in your file, then tick the *Other* check box and type the symbol in the text field. In case there are two or more separating characters in the line (usually spaces) to delimit a column, tick the *Merge delimiters* check box, so that the separation in columns can be carried out correctly. By default, the program reads the data of the first column as the X coordinates of the points, the data of the second column as the Y coordinates, the data of the third column as Z elevations of the points and the data of the last column as their names. If the order of the elements in your file is different, then you must select the correct order from the menu that is displayed when you left-click on the column title. You must also specify the format of your created file. If your file has been created in *DOS* environment, tick the *DOS (OEM)* check box. In the *Start line* field, type the number of the line from the beginning of the file, to indicate the point from which the program will start to import data. In this way you can omit lines that contain titles or/and comments, or even the import of the first points that are in the beginning of the ASCII file.



4. Confirm by clicking *OK*. The program reads the data and imports it to the file. If a problem occurs during the file transfer, the program will display the relevant error message on the screen.

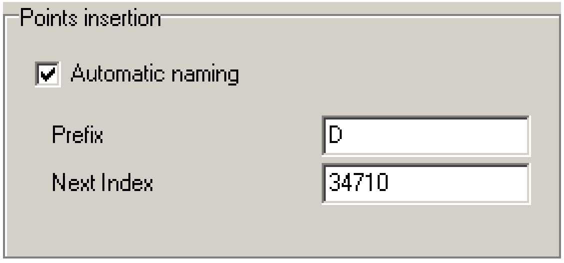
The points will appear on the screen, according to the way you have selected. If you want to check the points for possible errors (e.g. double points) you must run the *Terrain Validation* command to check and correct them.

Creating Terrain Points

Although the insertion of terrain points from *Anadelta Tessera* is not very common, it is very useful in case you want to add some points.

How to create terrain points

1. From the *Terrain* menu, run the Insert point command, or click the button on the *Elements Drawing* toolbar.
2. Use the mouse to set the position of the terrain point.



As regards the names of the points that will be inserted, you can either request from the program to name them automatically when they are inserted, or you can insert then without a name and name them afterwards, from the *Properties* command window. You can do this from the dialogue box of the *Terrain Options* command. If you want the program to automatically name the new inserted points, you must tick the *Automatic naming* check box in the *General* tab of the *Point insertion* group. The name given by the program is comprised of the prefix that you have typed in the *Prefix* text field and of a serial number.

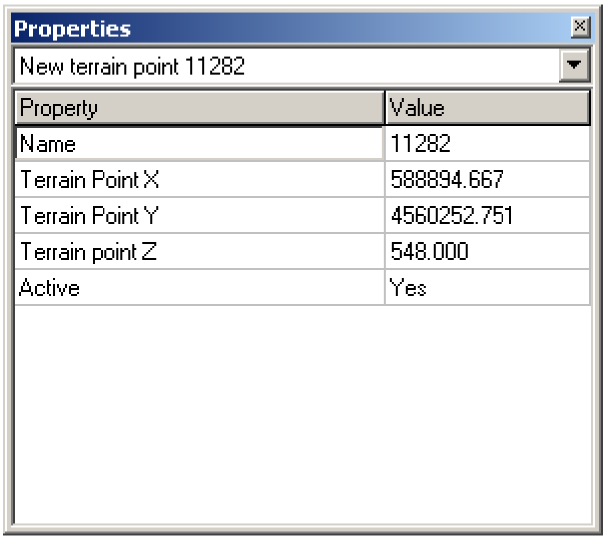
Editing Terrain Points

You can edit the points you have inserted, so as to give the model exactly the form that you wish. Therefore, you can edit the coordinates of the points or delete the points that you do not need.

Moving and Renaming Terrain Points

You can move the Terrain Points of your file.

How to move or rename the selected Terrain Points



1. With the mouse pointer over the desired *Terrain Point*, click the left mouse button to select the point.
2. Open the *Properties* command window. To change the name of the point, type the new name in the *Name* field. In the *Terrain Point X*, *Terrain Point Y* and *Terrain Point Z* fields, type the coordinates and the new elevation.

To move a point faster, select it, activate it and move it to its new position by using the mouse or the *Input Form* command.

Deleting Terrain Points

You can delete one or more Terrain Points, if desired.

How to delete the selected Terrain Points

1. Select the *Terrain Points* you wish to delete.
2. From the shortcut menu that is displayed when you right-click in the drawing area, run the *Delete Objects* command, or from the *Edit* menu run the *Delete* command, or press the *Del* key, or from the main toolbar click the button .

If desired, you can delete all terrain points of your file.

How to delete all the Terrain Points

From the *File* menu, select the *Delete all* submenu and then select the *Terrain Points* submenu and run the *All* command.

How to delete all the Terrain Points inside or outside a region

1. Select the boundary (perimeter or oasis) inside or outside of which you want to delete points.
2. From the *File* menu, select the *Delete all* submenu and then select the *Terrain Points* submenu and run the *Inside selected boundary* or *Outside selected boundary* command, respectively.

Deactivating Terrain Points

In many cases, there are points you want to exclude from the triangulation without deleting them, e.g. stops. The inclusion of a point in the Terrain Model is a property defined exclusively by the user. You can select not to include a point in the Terrain Model by deactivating it.

How to deactivate a point

1. Select the *Terrain Point* you wish to deactivate.
2. Open the *Properties* command window. From the drop-down list of the *Activated* field, select *No*. To activate a point, select *Yes*.

To view and activate all the deactivated points, run the *Terrain Validation* command from the *Terrain* menu.

Adding Points– Updating the Terrain Point Elevation from the Existing Model

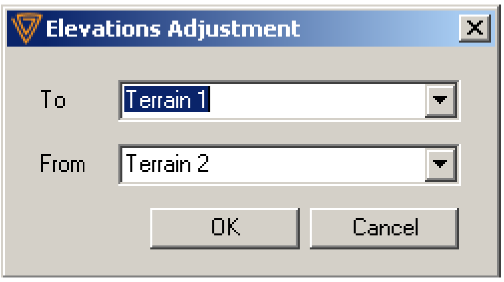
Every time you insert a new point, this has always zero elevation. You can update the elevation of the point from the existing terrain model.

How to update the elevation of a Terrain Point

1. Select the *Terrain Point* you wish to update.
2. From the shortcut menu that is displayed when you right-click in the drawing area, run the *Adjust Elevation* command.

Updating the Terrain Point Elevation from another Terrain

You can adjust the elevations of a terrain based on another terrain. From the *Terrain* menu, run the *Adjust Elevations* command. The following dialogue box appears on the screen.



From the drop-down list of the *To* field, select the terrain that will be updated and from the drop-down list of the *From* field select the terrain on which the update will be based. Next, click the *OK* button to apply the update.

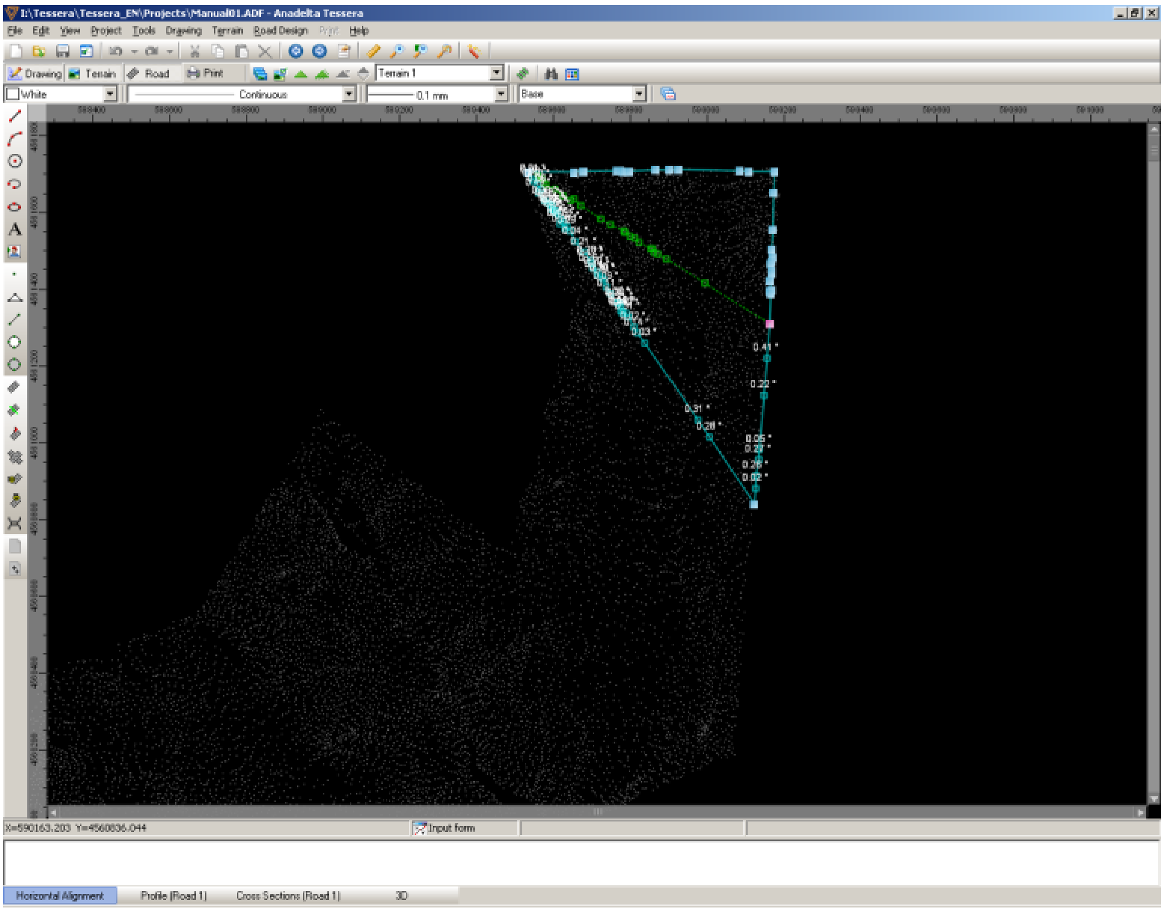
7.2.5 Creating and Editing Perimeters

For the correct triangulation, before the insertion of the triangles you must insert a perimeter, which demarcates the area in which the triangles will be created. A perimeter is a zigzag line whose peaks must coincide with terrain points. Therefore, you cannot insert a perimeter unless you have first inserted terrain points.

A perimeter can be inserted in three ways:

1. It can be created by the user, who must define its points one by one.
2. It can be calculated automatically by the program.
3. It can be imported from a text or DXF file.

In all cases, once it is created, the perimeter can be modified so as to describe better the terrain boundaries. Perimeters are shown on screen in green.



You can insert more than one perimeters. Two perimeters cannot be intersected, but one perimeter may be enclosed in another with an oasis between the two perimeters.

Automatic Calculation of the Perimeter

To save time, you can request from the program to automatically calculate the external perimeter. After this calculation, you can adjust the perimeter by adding or deleting points.

How to insert automatically a curved perimeter

From the *Terrain* menu, run the *Calculate curved perimeter* command. This results in a closed curved tangent line that includes all points of the current terrain.

Inserting a Perimeter

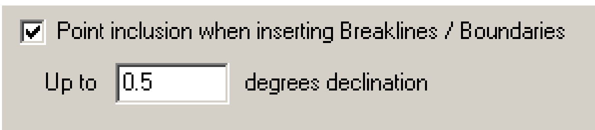
To insert a new perimeter, you must set the terrain points that define it. The points of a perimeter must coincide with the terrain points. This method serves mainly for the insertion of an internal perimeter.

How to set the points of a perimeter

1. From the *Terrain* menu, run the *Insert perimeter* command, or from the *Elements Drawing* toolbar, click the relevant button. You will see the highlighted box in the middle of the mouse cross, showing you where the perimeter point will be placed.
2. Define circularly the terrain points that will compose the perimeter (the mouse snaps automatically the terrain points).
3. Once you have finished inserting of the points, click the right mouse button to complete the procedure.

<u>Automatic Point Inclusion</u>

If you wish, you can request from the program to automatically include points that are located between the point you define and its previous point. To do this, go to dialogue box of the *General Options* command of the Tools menu and in the *Insert/Delete* group of the *Terrain* tab tick the Point inclusion when inserting *Breaklines/Boundaries* check box. The text field below the check box shows the maximum variation for the automatic inclusion of a point.



Import of Perimeters from External Files

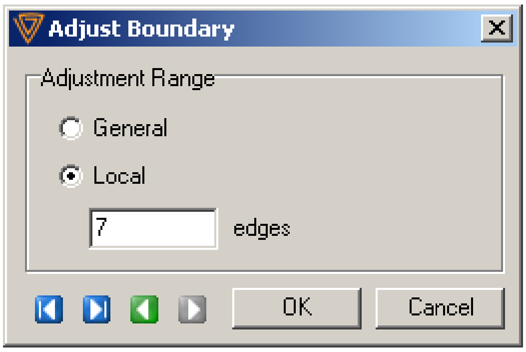
Perimeters can be imported either from ASCII text files or when importing other elements (e.g. drawing elements), which are all saved in a DXF file.

Automatic Addition or Deletion of Perimeter Points

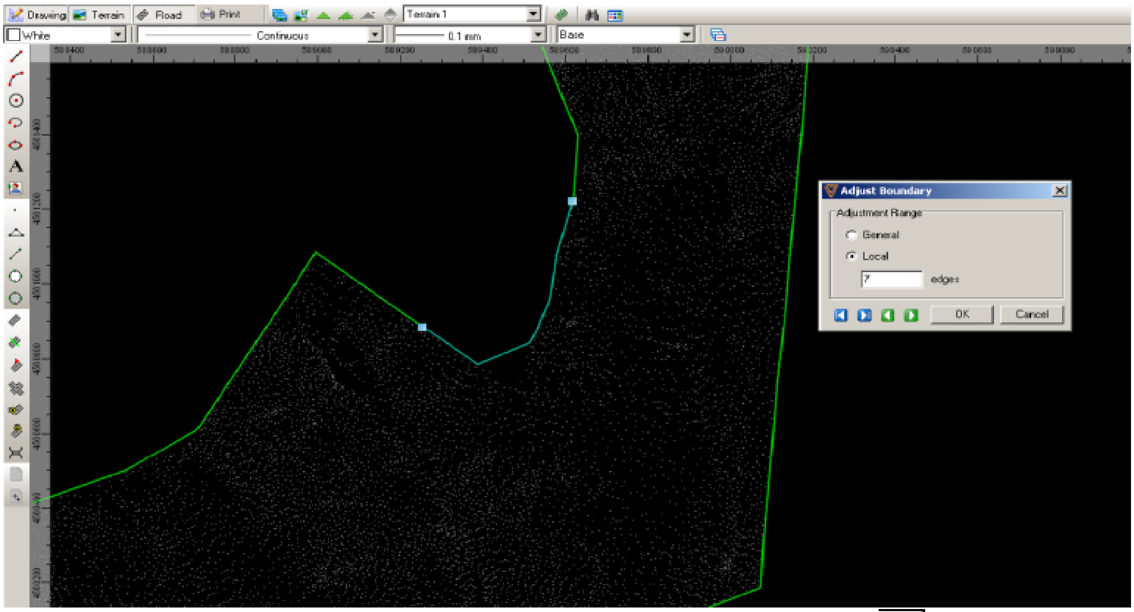
To adjust a perimeter faster, you can use the automatic perimeter adjustment.

How to automatically adjust a perimeter

1. Select the perimeter you wish to adjust.
2. From the shortcut menu that is displayed when you right-click on the screen, run the *Adjust perimeter Expand* or *Shrink* command to delete or add points, respectively. This command opens the following dialogue box.



3. If you want to add/delete points along the whole length of the perimeter, activate the *General* selection field. If you want to add/delete points to only one part of the perimeter, select *Local*. If you select *Local*, you must specify the width of the sides between which the boundary will be adjusted. The default value is 7. If you want to change this value, click the button to decrease it or the button to increase it. Next, you must define the perimeter part that can be adjusted by moving the mouse pointer over it to highlight it.



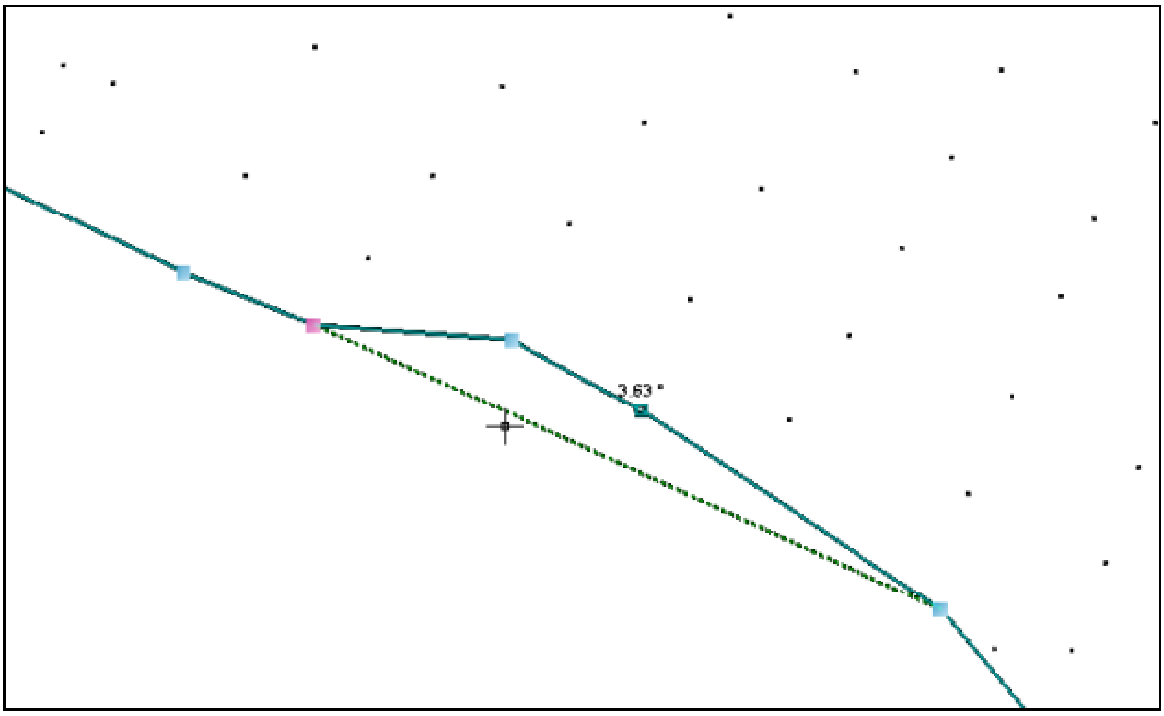
4. For the adjustment to take place, click the *Expand/Shrink* button , at the bottom of the dialogue box or roll the mouse wheel down. The effect is cumulative, namely more points are added or deleted as you keep rolling the wheel. The effect on the boundary can be viewed immediately on the screen. Keep clicking the button until you are satisfied with the result. If you want to undo the expansion/shrinkage, click the *Reset* button or roll the mouse wheel up.
5. When you have finished, click *OK* to apply the adjustment. This closes the dialogue box and returns you to the main screen.

Adding Points to a Perimeter

How to add points to a perimeter

1. Select a point of the perimeter.
2. From the shortcut menu that is displayed when you right-click on the screen, run the *Expand to the beginning* or *Expand to the end* command, depending on what the side you wish to insert the point (the points of the boundaries have a

counterclockwise direction). This command adds a new point to the line on screen, before or after the point you originally selected. The original point is highlighted in red, and you can place the new point by moving the mouse pointer as shown in the figure. The points on which the new point intersects are displayed in yellow, while the points that are to be added are displayed in green. The original form of the boundary is displayed as a dotted line.



If you wish, you can request from the program to automatically include points that are located between the point that you define and the previous point.

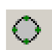
Deleting a Perimeter Point and a Terrain Point

The *Point removal* command deletes the selected PI of the perimeter/oasis and joins the previous PI with the next one. The program does not delete the terrain point. You can also run this command by pressing the *Shift+Delete* keys. If you have already created triangles and then you delete a perimeter point, the triangles will remain as they were and the terrain validation will report the errors. Therefore, you must also make the relevant corrections on the terrain model.
The *Point removal* command deletes the selected PI of the perimeter/oasis and the corresponding terrain point.

Deleting a Perimeter

You can delete one or more perimeters of the file. The procedure is similar to the one for the *Terrain Points*.

7.2.6 Creating and Editing Oases

To insert an oasis you must have at least one perimeter. The oasis demarcates an area that will be free of triangles (e.g. wherever there is a housing settlement or a lake). To insert an oasis, follow the same procedure that applies to the insertion of a perimeter, only in this case you must click the button . The editing options for the oases are the same as the ones for the perimeters.

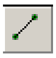
7.2.7 Inserting Breaklines

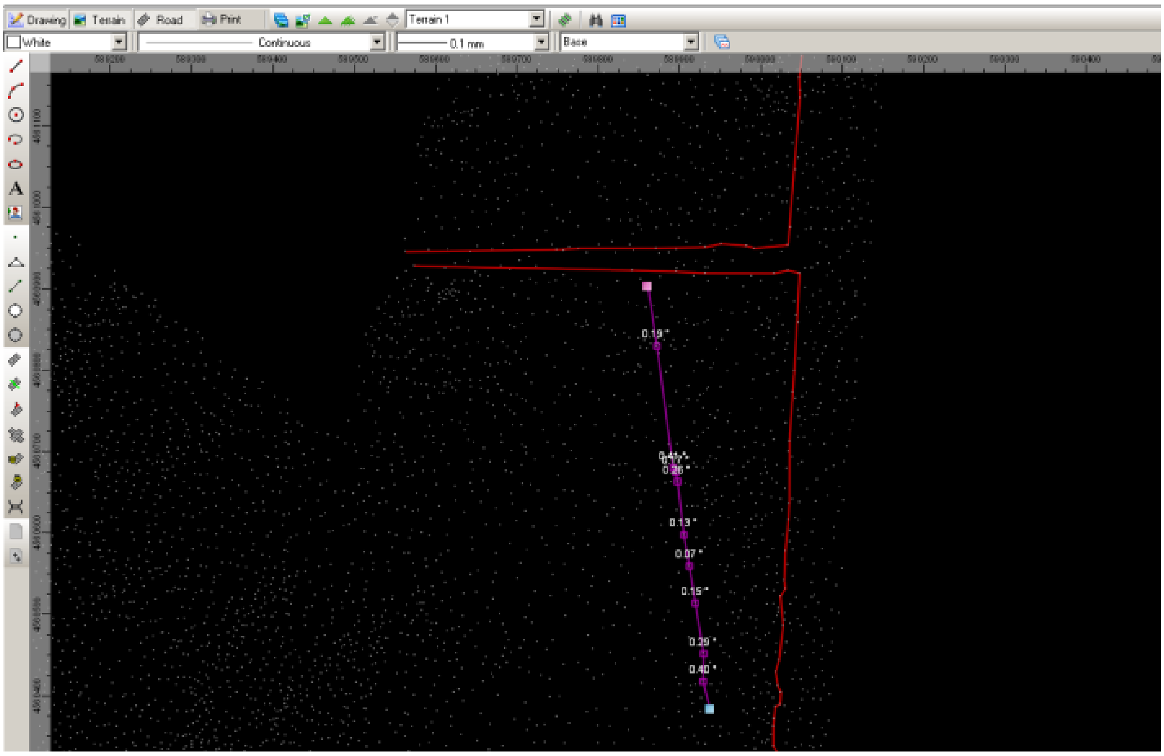
The purpose of the Breaklines is the demarcation of the triangles. Breaklines are also zigzag lines, whose peaks must coincide with terrain points. As a result, to insert them you must have first inserted terrain points. During the triangulation, the program creates the triangles in such a way, so that their sides always coincide with the breaklines and never intersect them. Thus, it is possible to simulate lines such as the tops and feet of sideslopes, the roadlines of existing roads, etc.

Breakline Insertion

One way to insert Breaklines is to define their PIs on screen by using the mouse.

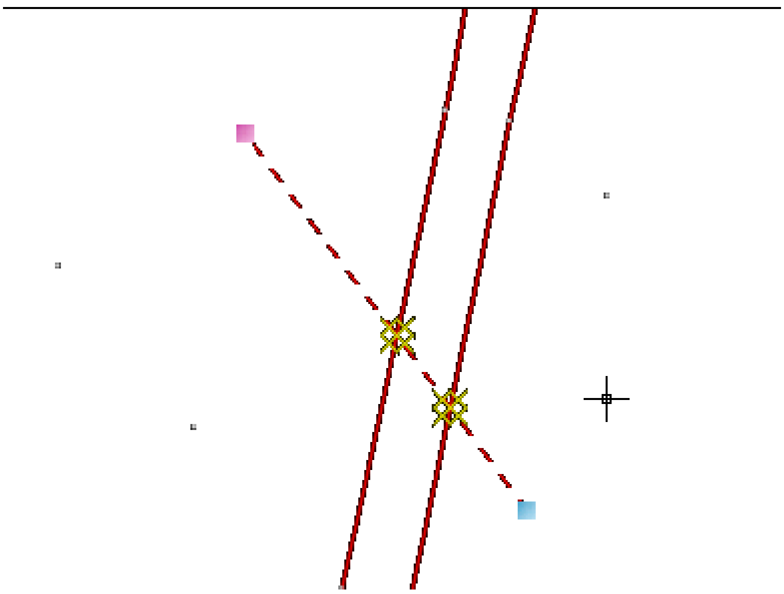
Inserting Breakline with the mouse

1. From the *Terrain* menu, run the Insert *Breakline* command, or from the *Elements Drawing* toolbar, click the  button. The pointer turns into a crosshair and snaps terrain points only.
2. Define successively the terrain points that will comprise the breakline. If desired, the program can automatically include points that are located between the point that you define and the previous point. Thus, when the crosshair approaches a terrain point, the screen will also display the points that will be included into the line.



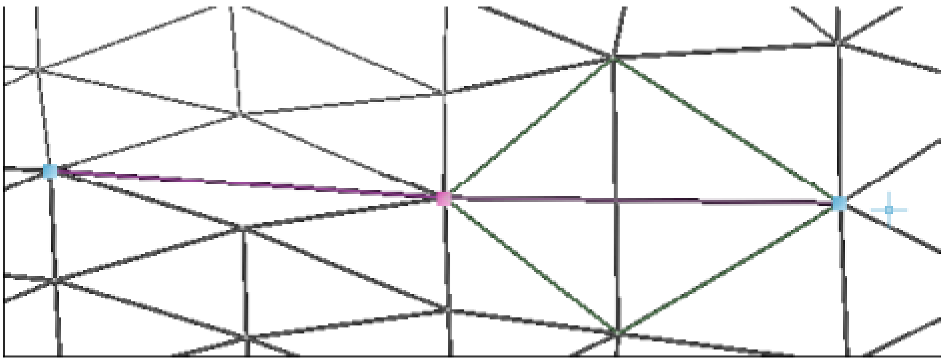
3. When you have finished inserting points, click the right mouse button to complete the procedure.

A breakline may intersect with other terrain lines (Perimeters, Oases, Breaklines), only on terrain points. In this case, the program does not allow PI insertion, while the points where the BKL is intersected are displayed in yellow.

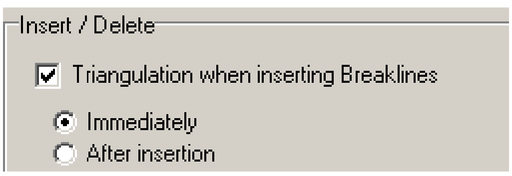


Also, a breakline cannot refer to disabled points. For this reason, when you approach the mouse pointer to such points, a yellow circle appears around the point.

You can also insert breaklines after the creation of triangles. In this case you can choose the triangulation to be carried out immediately upon breakline insertion, so that the triangles that are affected by the inserted breakline are immediately reformed.



To do this, go to the dialogue box of the *General Options* command of the *Tools* menu and in the *Import/Delete* group of the *Terrain* tab, tick the *Triangulation when inserting Breaklines* check box.



Conversion of Drawing Lines to Breaklines

Another way to create a breakline is the conversion of a drawing line. Select the line to be converted and from the shortcut menu that appears when you click the right mouse button select the *Conversion* submenu and then run the *Convert to Breakline* command. For the conversion to take place, all line points must coincide with terrain points. The properties of the new breakline are the current ones. The drawing line remains almost unchanged.

Editing Breaklines

You can insert additional points to a breakline, delete a point from a breakline or delete the whole breakline.

Breaking and Merging Breaklines

For easier management of the lines, you can break a breakline in two or more breaklines, or you can merge multiple breaklines together into a single breakline.

Merging Breaklines

To merge two breaklines, select them both and then run the Merge lines command from the menu that is displayed when you right-click on the screen. For the merging to take place, the last PI of one breakline must be the first PI of the next breakline.

Breaking Breaklines:

If desired, you can break a brakline into two or more parts.

To break a breakline in two

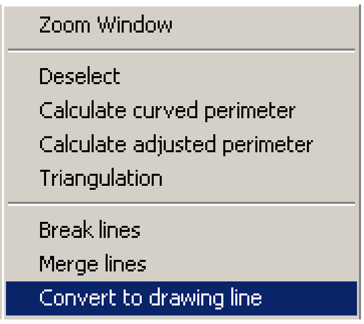
1. Select the breakline you wish to break. Select the PI where breaking will take place.
2. From the shortcut menu that is displayed when you right-click in the drawing area, run the *Break at point* command.

If desired, you can break a line into many parts; each of these parts will be comprised of only two PIs.

To break a line in multiple parts

1. Select the breakline you wish to break.
2. From the shortcut menu that is displayed when you right-click in the drawing area, select the *Break lines* command.

Converting Breaklines to Drawing Lines



To convert a breakline into a drawing line, select the desired breakline and from the menu that is displayed when you right-click on the screen run the *Convert to drawing line* command. The properties of the new line are the current ones.

7.2.8 Terrain Validation

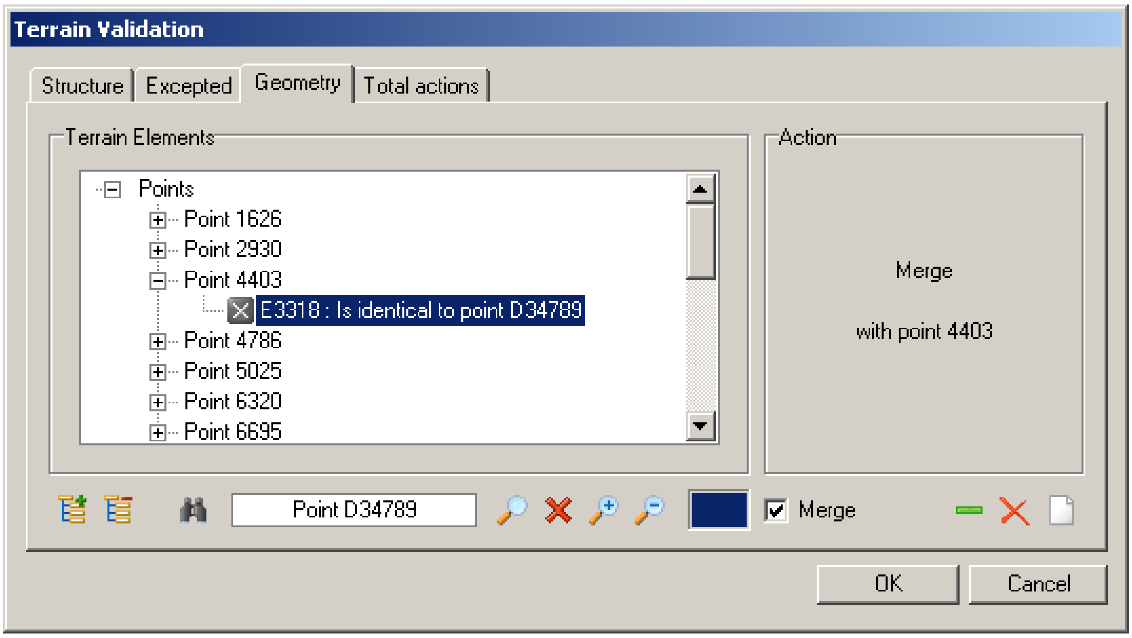
The terrain validation must be performed before the triangulation. This stage (before creating triangles) aims at the detailed validation of the points. Terrain validation is performed automatically when you request triangulation, but it can also be carried out at any given time, upon the user's request.

How to perform terrain validation

1. From the *Terrain* menu, run the *Terrain Validation* command, or press the *Ctrl+G*

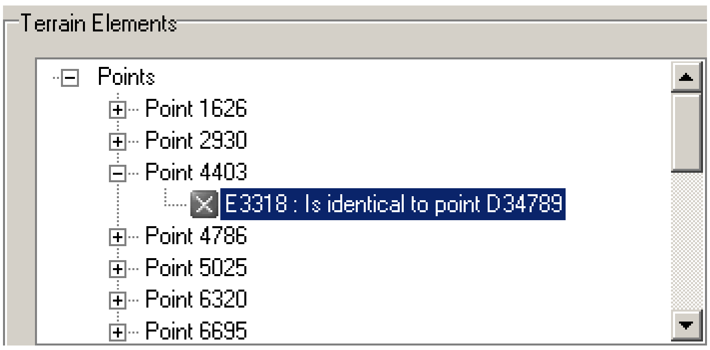
keys, or from the *Horizontal Alignment* toolbar click the  button. The program will start validating the terrain elements. A bar appears at the bottom right of the drawing area, showing you the validation progress.

2. When validation is over, a dialogue box opens on the screen. In this box you can find and correct any errors of the terrain model. In this phase (the terrain model has not yet been created), most errors refer to the terrain geometry.



The dialogue box has four tabs. The first two tabs (*Structure, Excepted*) display all kinds of problems. In the *Excepted* tab you can see the points that were exempted either by the user or by the program, according to the terrain model settings. Finally, in the *Total actions* tab, you can see the actions that will be carried out to correct the errors that have been detected during the terrain validation. The first three tabs consist of two groups.

In the *Terrain Elements* group, at the top left of the window, you can see the tree-structure graph with the terrain elements where a problem was detected.



Left-click on the + symbol next to each element category or double-click on the category, to view the relevant elements. Left-click on the + symbol next to each element or double-click on the element to view the problems of this element. You can select an element by left-clicking on it. The name of the selected element appears in the *Selected Element* box right below.

Below this group there are the buttons with the relevant actions. If you click the button, the problems of all elements and all categories will be displayed, while if you click the button, the problems will shrink.

The button transfers the drawing view to the drawing area where the selected element is. This element appears highlighted on the screen. With the and buttons you can change accordingly the view in the drawing area.

In the *Energy* group, which is located at the right part of the window, you can define the actions that will be carried out to solve the problems that were detected. The appearance of the group depends on the current tab. You can delete, activate or deactivate the selected element, or you can select no to proceed to any action at all. If an option is not available for the particular element, either this option will not be displayed at all or it will be inactive.

At the bottom of the window you will find the buttons; use these buttons to carry out an action for all elements of the selected category.

Finally, by ticking the *Merge* check box, the double points (points whose x, y, z coordinates are the same) are deleted.

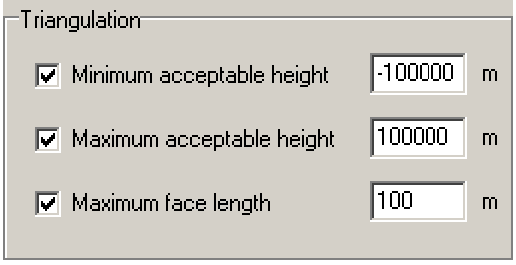
7.2.9 Triangulation and Contour Line Calculation

The next step in the creation of the terrain model is the triangulation and the calculation of contour lines. Prior to this, however, you must specify the settings of the digital terrain model by clicking the *Terrain Options* command from the *Terrain* menu.

Triangulation

Triangulation Settings

To select the triangulation settings, run the *Terrain Preferences* command and then go to the *Triangulation* group of the *General* tab.

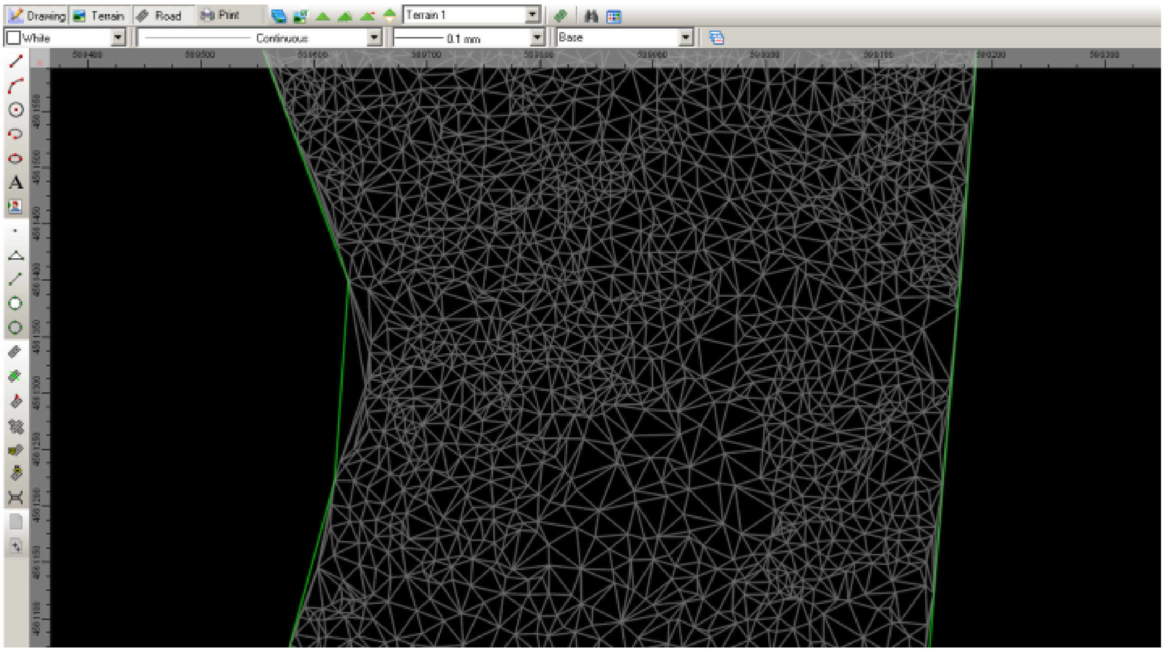


When the *Minimum acceptable height* check box is ticked, *Tessera* ignores points whose elevations are lower than the value typed in the corresponding field. When the *Maximum acceptable height* check box is ticked, the program ignores the points whose elevations are higher than the value of the field. When the *Maximum face length* check box is ticked, the program will not create triangles with faces larger than the value indicated in the corresponding field.

Triangulation

For the triangulation to take place, run the *Triangulation* command from the *Terrain* menu, or press the *Ctrl+B* keys, or from the *Horizontal Alignment* toolbar click the  button. When the *Terrain Validation* is completed, if there is no significant problem, the program will start creating the terrain model based on the inserted points, perimeters, oases and breaklines. During this process, the terrain points are joined together with straight lines to create triangles. When creating the terrain model, the program takes into account the breaklines, the perimeter, as well as any oases that you may have inserted.

When the calculation is over, if you have selected to view the triangles, and specifically only the faces of the triangles, you will see the following screen.



An empty area (as shown in the figure) means that the distance between the points of that area was higher than the one you had specified in the digital terrain model, and as a result the points were not joined to create triangles. These areas are the "natural oases", whose maximum face is the one you specified in the *Triangulation* dialogue box.

Insertion of Triangles

You can also create triangles manually. To do this, select the *Insert Triangles* command from the *Terrain* menu and then select one of the three secondary options, which are:

- *Three Points:* With this option you can create triangles using any terrain point.
- *Repeat Last Point:* When you run this command, the first point of every triangle you create will be the last point of the previous triangle.
- *Point and face:* To draw triangles with this command, you must define only one point/face of the triangle, since its two other points are defined automatically by the existing adjacent triangles.

<u>Editing of Triangles</u>

You can intervene and edit the terrain model. If you run the *Triangles Editing* command from the *Terrain* menu, you will notice that the triangles under the mouse pointer are turning green. Move the pointer to the desired triangles and click the left mouse button to change the diagonal of the quadrilateral. In this way, you can visually correct any anomaly in the digital terrain model. The changes you are making appear immediately on the screen. The review and editing process is facilitated by the various methods that the program offers for the presentation of the terrain model.

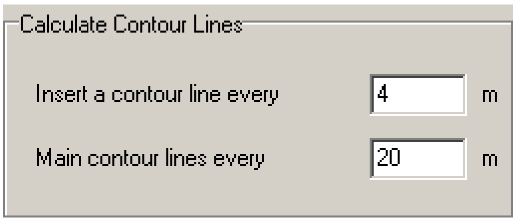
<u>Scanning Triangulation for Holes</u>

If certain parts of the terrain model that you have created, have holes, you can request from the program not only to scan for these holes, but also to create triangles in these parts without affecting the remaining terrain model. To do this, select the *Scan Triangulation for Holes* command from the *Terrain* menu and then select one of the following available options:

- *Insert Drawing Lines:* When you select this option, the program marks the holes with drawing lines.
- *Insert Perimeters:* With this option, the program inserts perimeters instead of drawing lines in the holes.
- *Triangulation:* With the triangulation option, triangles are created in the places of the holes, without affecting the remaining terrain model.

Calculation of Contour Lines

The next step in the presentation of the terrain is the calculation of the contour lines.

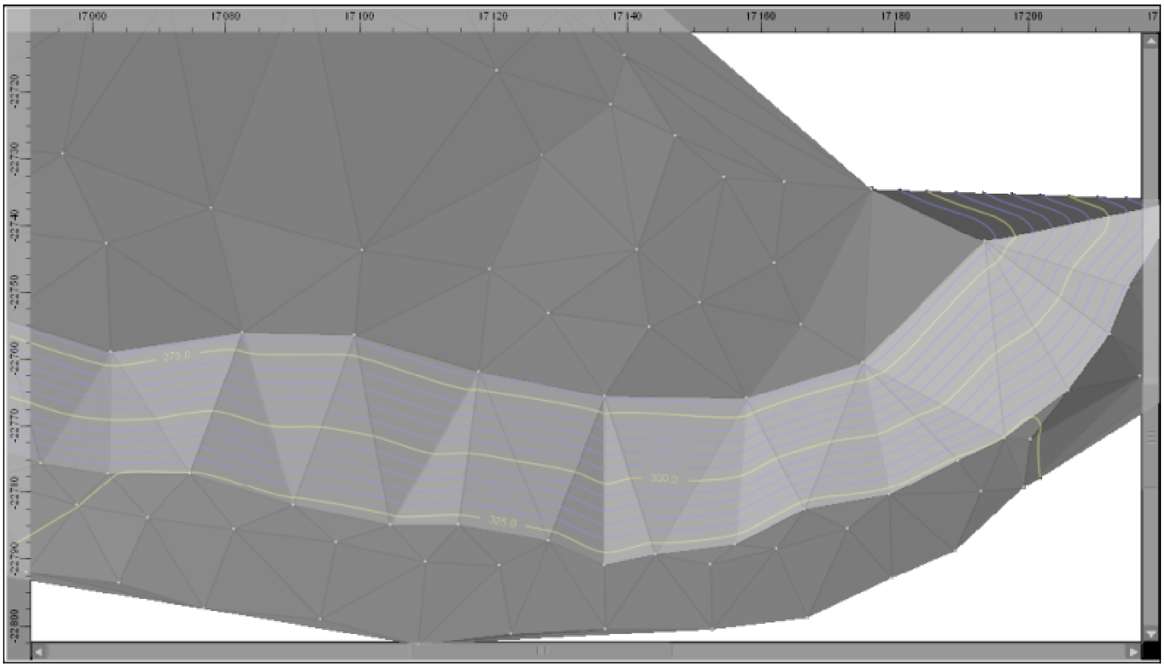


The horizontal equivalent of contour lines for the current terrain is defined from the dialogue box of the *Terrain Preferences* command. More specifically, go to the *Calculate Contour Lines* group of the *General* group and in the *Insert a contour line every* text field type the distance between two contour lines. In the *Main contour lines every* text field of the same group, you can specify the elevation difference between two main contour lines.

How to calculate the contour lines

From the *Terrain* menu, run the *Calculate Contour Lines* command, or from the *Horizontal Alignment* toolbar, click the  button.

When this operation is over, you can view the contour lines that were drawn by selecting the *View contour lines* option.



At this point, while viewing the contour lines, you can edit the terrain model in order to get a better view of the relief. To view the editing results on the contour lines, you must repeat the contour line calculation.

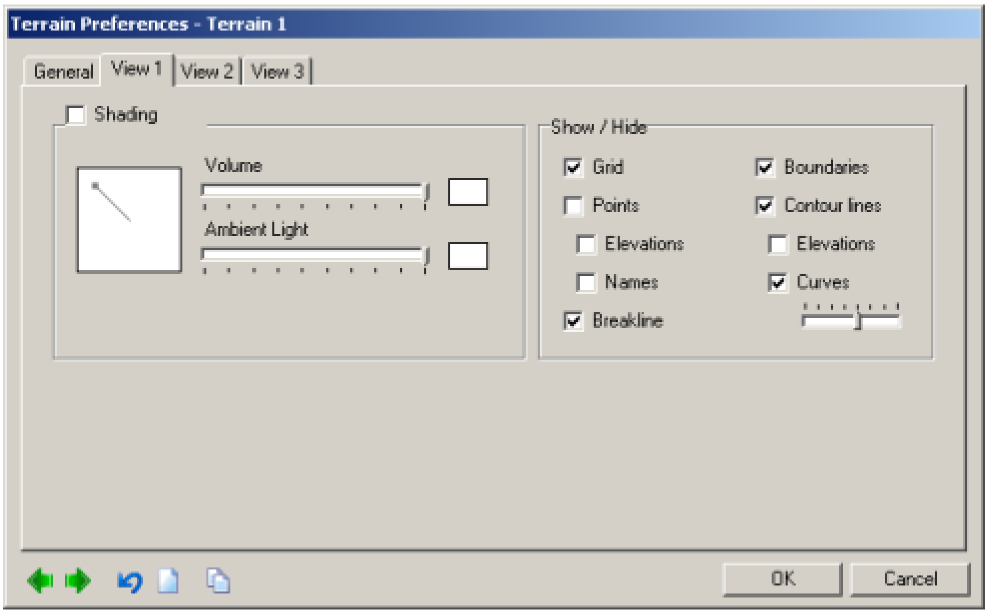
You can choose, from the Display options (*Curved or Straight lines* fields), how the contour lines will appear on the screen (zigzag or curved).

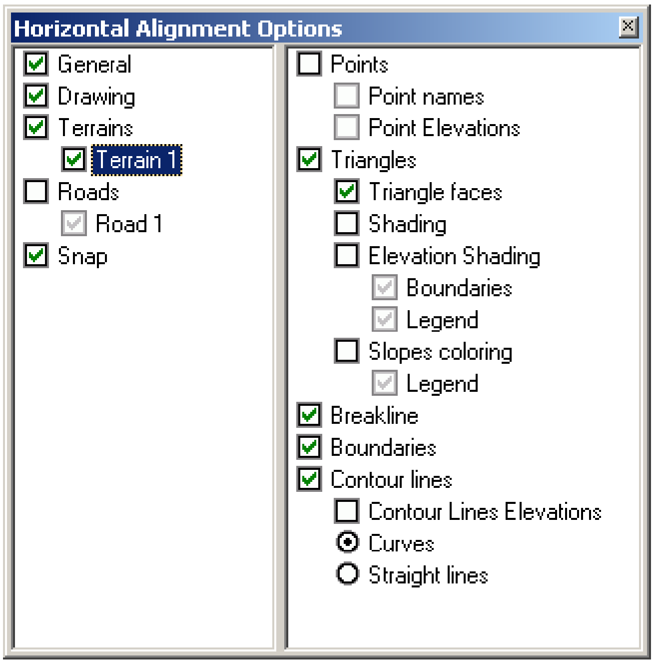
7.2.10 Display of Terrain Elements

The program enables you to control how the terrain elements will be displayed, both on screen and on the printout.

You can select which elements of a terrain will appear on screen and how they will appear, from the dialogue box of the *Display Options* command and from the *Terrain Preferences* dialogue box, and more specifically from the *View 1, View 2* and *View 3* tabs. Here you can define which of the terrain elements (and of the remaining project as well) will be visible in order to facilitate your work, as well as how they will be displayed. More specifically, the following apply to every terrain element:

1. *Points:* Points appear on the screen as gray dots. If desired, you can display the name and the elevation of the point. The name of the point appears at the top right of the point, while the elevation appears at the bottom right of the point. Enable this view either from the *Terrain Preferences* dialogue box, or from the *Options* command dialogue box. In all cases, select the desired terrain and tick the *Points* (to view the points), *Names* (to view the names) and *Elevations* (to view the elevations) check boxes.





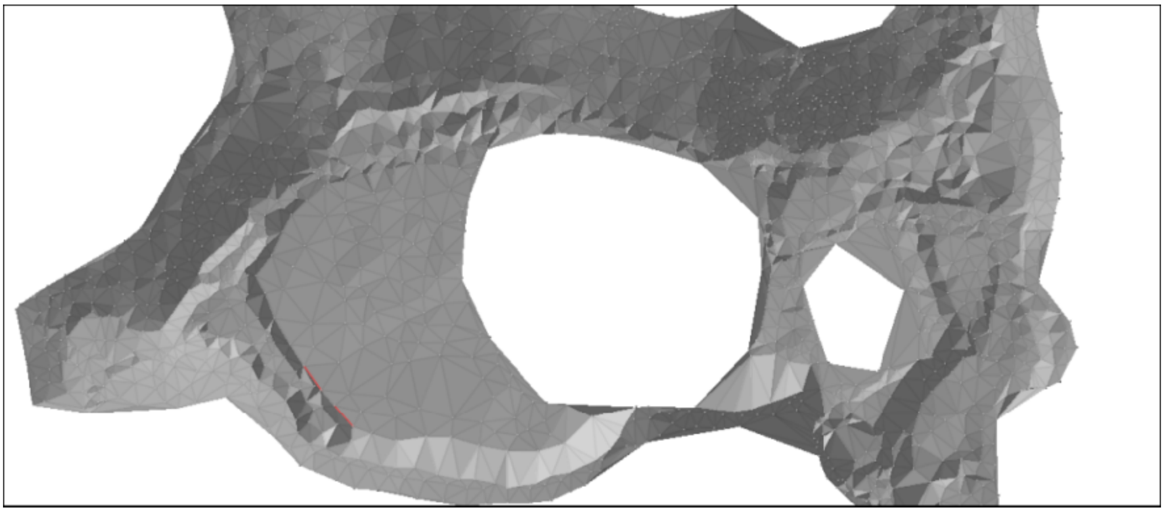
2. *Breakline and Boundaries (perimeters and oases):* Breaklines appear on screen as continuous red lines, while perimeters and oases appear as continuous green lines. You can activate this view either from the *Terrain Preferences* dialogue box or from the dialogue box of the *Options* command. In all cases, select the desired terrain and tick the *Breakline* check box (to view the breaklines) and the *Boundaries* check box (to view the perimeters – oases).
3. *Contour lines:* They are displayed on screen as continuous lines (curved or zigzag), the main ones in yellow and the intermediate ones in purple. If desired, the contour line elevations can be displayed through the lines. You can activate the display of the contour lines either from the *Terrain Preferences* dialogue box, or from the dialogue box of the *Options* command. In all cases, select the desired terrain and tick the *Contour lines* (to view the contour lines), the *Contour Lines Elevations* (to view their elevations) and the *Curves* check boxes (to view the contour lines as curves).

You can set the curvature factor, i.e. how rounded the curves will be, by moving the slider located in the *Terrain Preferences* dialogue box, right below the *Curves* check box. To decrease the curvature move the slider to the left, while to increase the curvature move the slider to the right.

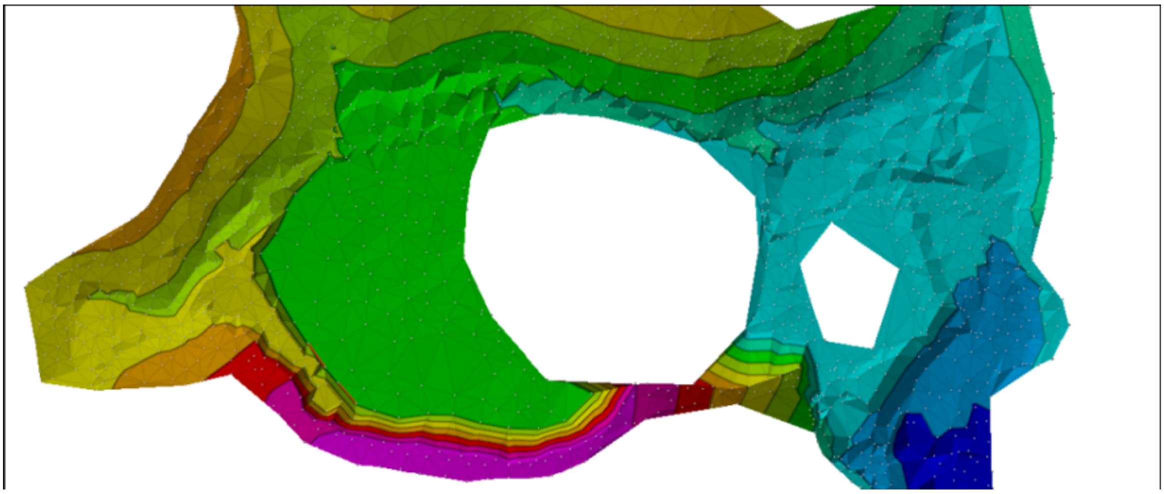
Display of Triangles

Tessera offers you four possible views of the triangles and you can combine them to achieve the best results.

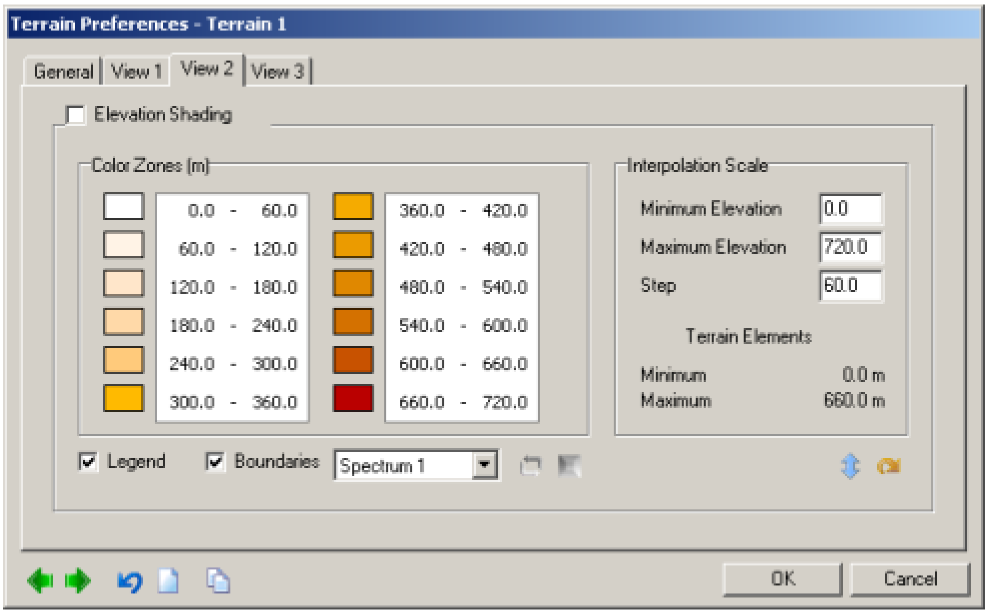
1. *General:* Regardless of how the triangles will be displayed, you can choose to view or hide them from the dialogue box of the *Options* command. Select the desired terrain and tick/untick the *Triangles* check box, to show or hide the triangles, respectively.
2. *Faces:* You can chose to view the triangle faces either from the *Terrain Preferences* dialogue box or from the dialogue box of the *Options* command. In all cases, select the desired terrain and tick the *Grid* or *Faces* check box, respectively. The figure shows an example of the way the triangles are displayed when you're using this method.



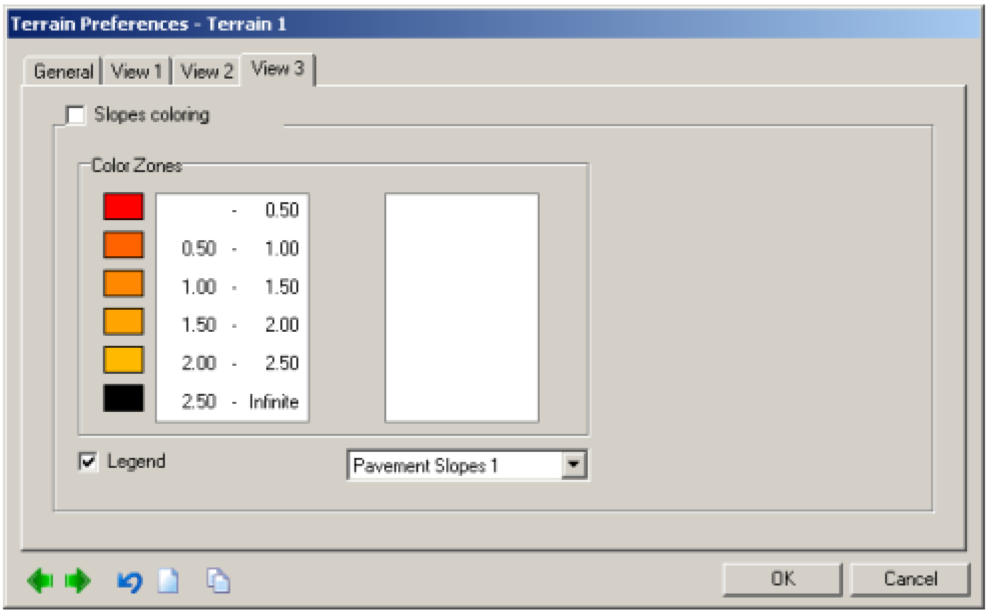
3. *Shading:* The figure shows an example of triangles with shading. You can activate the shading from the same dialogue boxes as in the case of the faces, only now you must tick the *Shading* check box of the *Shading* group in the *View 1* tab of the *Terrain Preferences* dialogue box. Use the sliders to set the brightness from the *Volume* field, as well as the color of light from the *Ambient Light* field. To set the angle of the light, left-click in the white box.



4. *Elevation Shading:* The figure shows an example of triangles with elevation shading. You can activate this view from the same dialogue boxes as in the case of the faces, only now you must tick the *Elevation Shading* check box in the *View 2* tab of the *Terrain Preferences* dialogue box. If you want to also view a legend with the elevation zones in the drawing area, tick the *Legend* check box. Likewise, to view a separating line on the color zones boundary, tick the Boundaries check box. Regarding the elevation definition of the zones, these can be defined either automatically by the program when you click the button or you can define them by filling in the *Minimum elevation* and *Step* fields and clicking the button. To set the colors, there are six color combinations stored in the program memory and you can choose the desired one from the drop-down list. When you select the *Personal 1* and *Personal 2* combinations, you can create your own color combinations. To select the color of a zone, click on the zone's colored box and select a color from the resulting dialogue box. To automatically fill the boxes with interpolations of two boxes, select the desired two boxes by click the right mouse button and then click the button. If you want the color zones to be reversed in terms of elevation, click the button.



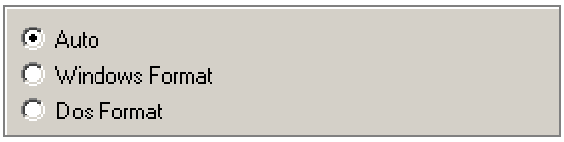
5. *Slopes coloring: Tessera* also provides you the possibility to color the triangles according to their slopes. You can activate this view from the same dialogue boxes as in the case of the faces, only now you must tick the *Slopes coloring* check box in the *View 3* tab of the *Terrain Preferences* dialogue box. The parameters that apply to the zone coloring are the same as in the case of the elevation shading.



7.2.11 Terrain Model Export

Exporting Terrain Points to a Text File

You can create a file that will contain the terrain points of the current terrain. The format of such files is similar to the format of the files that you use in order to import terrain points. You can set the character coding from the *ASCII* tab that you will find in the dialogue box of the *General Options* command of the *Tools* menu.



To export terrain points to ASCII files

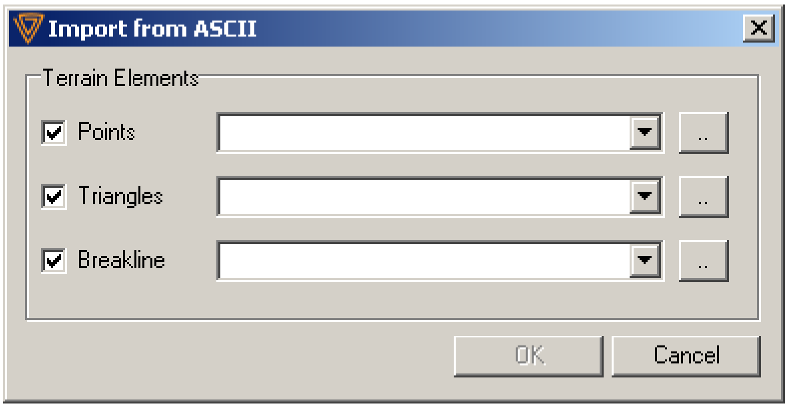
1. From the *File* menu, select the *Export to ASCII* submenu and run the *Terrain Points* command.
2. Specify the path and the name of the file where the points will be saved. Terrain points will be exported to a file with the .GRD extension.
3. Click *OK* to create the file.

Exporting the Whole Terrain Model to Text Files

You can export the terrain model you have created in the *Anadelta Road Design* to text files (ASCII). The software's export files are compatible with the RDDSS (Road Design Digital Submission Specifications).

How to export the terrain model to ASCII files

1. From the *File* menu, select the *Export to ASCII* submenu and run the *Terrain Points* command. The terrain points are exported to three files, one for each category of elements. The terrain points will be exported to a GRD file, the triangles to a TRI file and the breaklines to a BKL file. You can export RLN files by running the *Drawing Elements* command of the same submenu.



2. Tick the check boxes for every category of terrain points you wish to export. In the text field, type the file name and the drive path where your files will be created. If desired, you can specify the path for each category by clicking the button next to every text field.
3. Click *OK* to create the files. If the terrain points don't have a name, the program will ask you whether you wish the names to be added automatically, showing you the relevant confirmation message. Click *OK* to proceed.

7.2.12 Terrain Model Import

Opening an Older Version File (Files from Anadelta ROAD DESIGN 3.xx)

In *Anadelta Tessera* you can edit files of the *Horizontal Alignment* mode of the *Anadelta Road Design 3.xx* software pack. You can open these files with the *Open* command. The program recognizes fully the terrain elements contained in such files, enabling you to continue processing your work.

Importing the Whole Terrain Model from Text Files

You can import a terrain model from text files (ASCII) by running the *Terrain Points* command from the *Import from ASCII* submenu of the *File* menu. The procedure is the same as for the export of the terrain model. The terrain elements are imported in the current terrain.

Importing Terrain Points per Cross Section

The program can add points to a terrain based on the data resulting from the cross sections. More specifically, the *Terrain Points per cross section* files command of the *Import from ASCII* submenu of the *File* menu, imports additional points from a terrain file per cross section (*.ACS). Such files are also created by the *Cross Sections* program (*Version 3.xx*), as well as from the *Cross Section* workspace → *File* menu → *Export to ASCII* submenu → *Lines* command. To execute this command, you must have selected a calculated road where stationing has been performed.

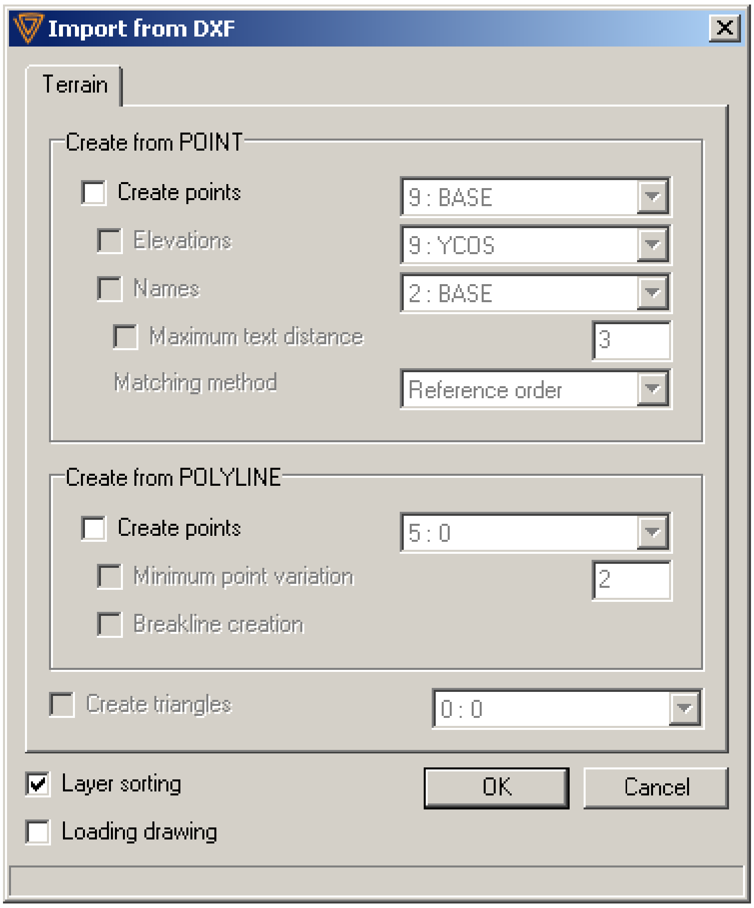
Importing the Whole Terrain Model from a DXF file

The last type of files from which you can import data are files with a .DXF extension. These files guarantee compatibility when working with other drawing packages (e.g. AutoCAD), from which the DXF files are exported.

How to import the terrain model from DXF files

1. From the *File* menu, select *Import from Dxf* and then run the *Import Terrain from DXF* command.

2. The program reads the file and displays the dialogue box shown below. In the Terrain tab select whether the points will be imported from *Point* or *Polyline*. When you tick the *Create points* check box in the *Create from POINT* group or in the *Create from POLYLINE* group, the options of each group become available.



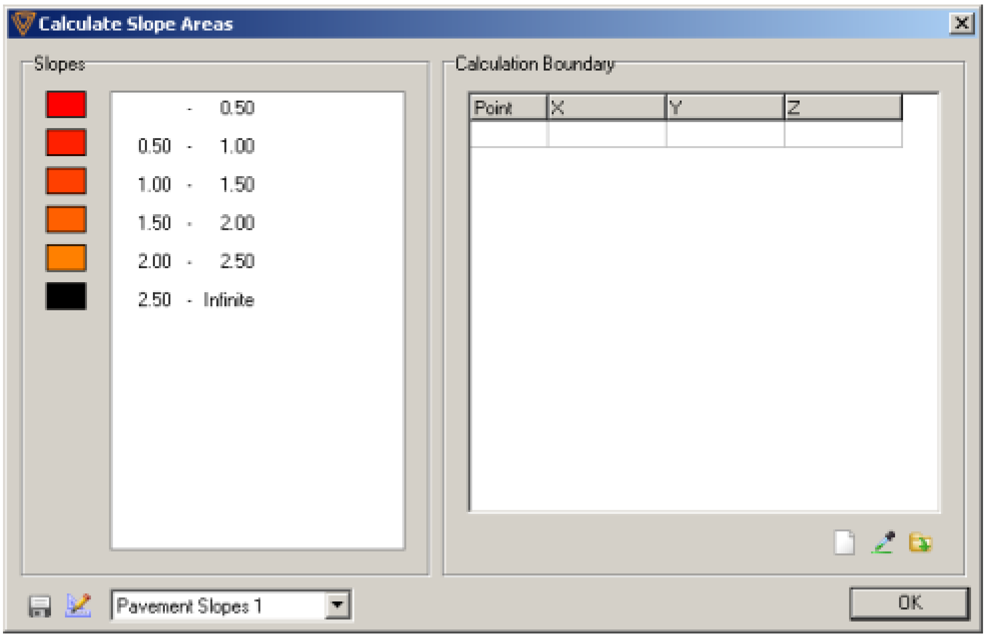
3. Tick the check boxes for every category of terrain points you wish to import and from the drop-down lists select the layer where these terrain points are. When the *Layer sorting* box is ticked, the program will sort out the layers according to the objects that are located in every layer.
4. If, in addition to the terrain points, you wish to import all other data of the file in the form of drawing, you must also select the *Loading drawing* option.
5. Click *OK* for the file to be read. The result of this import will be displayed immediately on the screen. The elements are imported in the current terrain.

If you want all data of the Dxf file (and the terrain points) to be imported in the form of drawing, you must select *Import from Dxf* from the *File* menu and then run the *Import Drawing from Dxf* command.

7.2.13 Calculating Slope Areas

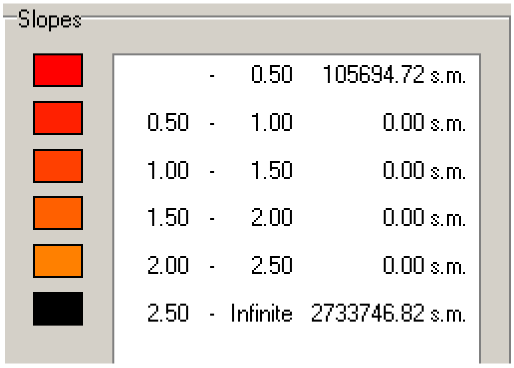
Area Calculation

With the *Calculate Slope Areas* command of the *File* menu, you can calculate the horizontal areas in groups that are determined according to the terrain slope. This calculation can be performed either for the whole area of the terrain or for a specific part. This command results in the following window:



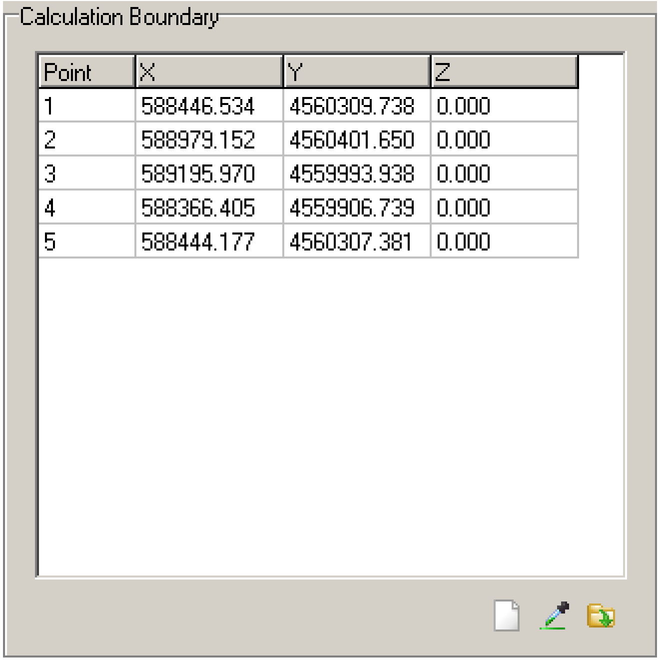
If you want the calculation to be applied to the whole terrain, select from the 

drop-down list the desired scale according to which the areas will be displayed and then click the  button, so that the surface areas can be calculated in m^2, as shown in the figure below:



To calculate the area of a particular surface, go to the *Horizontal Alignment* mode and circle with a drawing line all the terrain points you wish to be taken into account during the calculation.

Next, click the  button to move the coordinates of the selected line to the left part of the window, as shown below:



Point	X	Y	Z
1	588446.534	4560309.738	0.000
2	588979.152	4560401.650	0.000
3	589195.970	4559993.938	0.000
4	588366.405	4559906.739	0.000
5	588444.177	4560307.381	0.000

To save the results of the area measurement in text files, click the button, while to open an existing text file click the button.

To delete the data from the *Calculation Boundary* window, click the button.

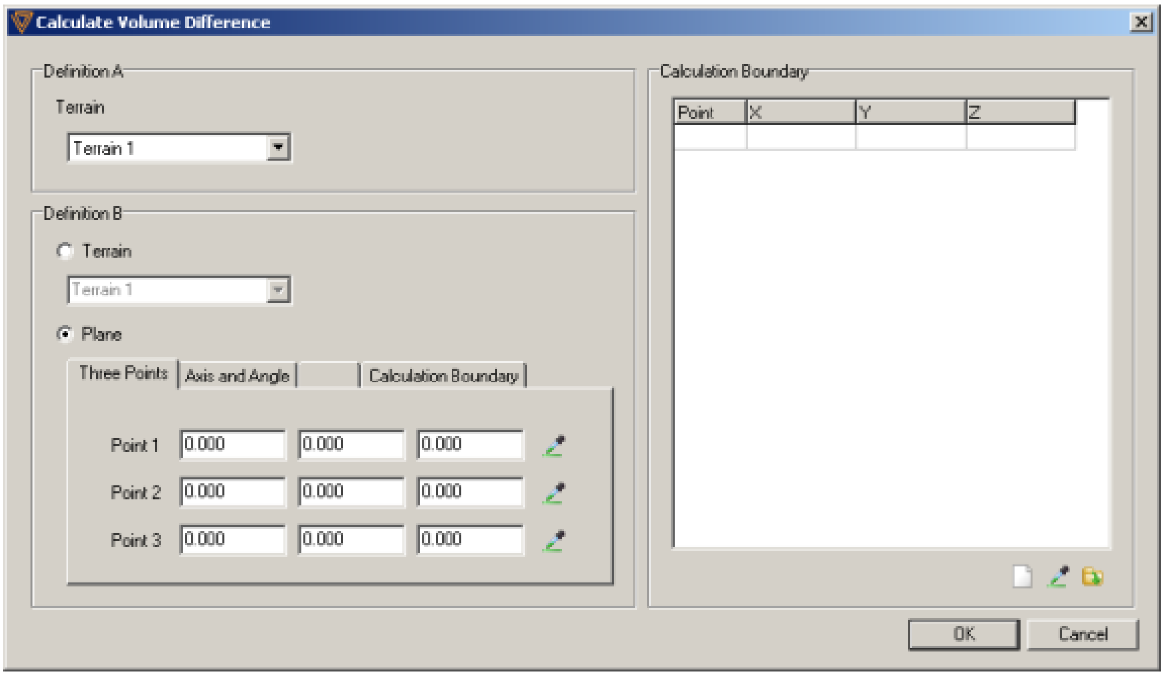
7.2.14 Surface Calculation - Volumes

Volumes

With the *Calculate Volumes Difference* command of the *Terrain* menu, you can execute volumes calculations. The calculation is performed between one terrain that you select from the drop-down dist of the *Definition A* group and a terrain or a plane that you select from the *Definition B* group.

To select a terrain, tick the *Terrain* check box and select the terrain from the drop-down list. To select a plane, tick the *Plane* check box and select the plane from the tabs right below. Each tab corresponds to a plane definition method. If the plane is parallel to the XY plane, select the respective tab and type the elevation of the plane in the Z text field. If the plane is defined by three points, go to the respective tab and type the coordinates of each point. If the plane is defined by two points and by its angle with respect to the horizontal, display the *Axis and angle* tab and type the coordinates of the points and the angle. If you select a terrain point on the screen and click the button, the coordinates of this point are copied.

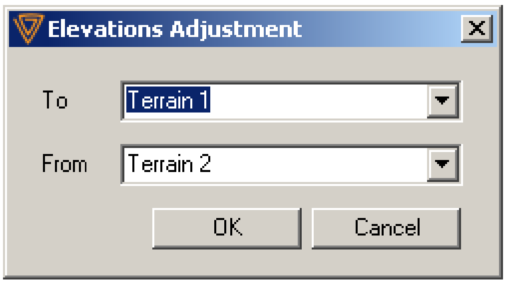
In all cases, when you click the *OK* button, the relevant calculations are performed. In the drawing area, the cuttings areas are displayed in red, the fillings in blue and the part of the terrain that remains unchanged is displayed in gray. From the *General* category of the *Options* command dialogue box, you can define how the volumes differences will be displayed.



7.2.15 Adjusting the Elevations

From the *Road Design* menu of the *Horizontal Alignment* mode, select the *Adjust Elevations* command. With this command you can adjust the elevations of a terrain to another terrain.
To execute this command, follow the steps below:

- Import a terrain (File Import from ASCII Terrain Points).
- Create a new *Terrain* (from the *Horizontal Alignment* toolbar).
- In the *Object Properties* toolbar, activate the second terrain that you created and then import the desired terrain in the same way as described above.
- When both terrains have been imported from the *Road Design* menu, click the *Adjust Elevations* command. The following window will appear on screen:



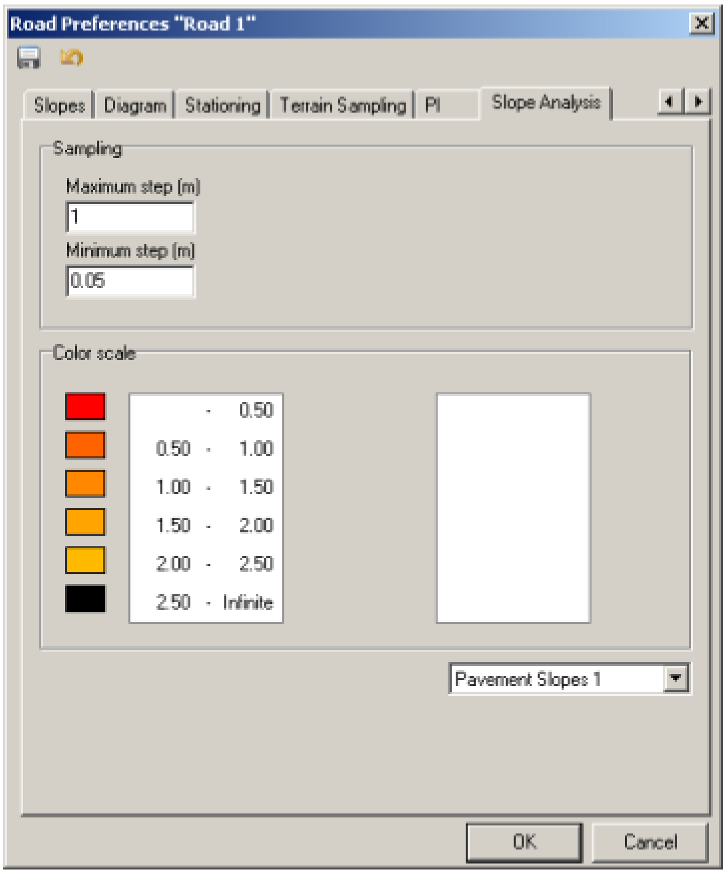
- In the *To* field select the terrain that will get the elevations, while in the *From* field define the terrain from which the elevations will be obtained.
- Click *OK* to close the window and to perform the adjustment of elevations.

7.2.16 Slope Analysis

With slope analysis you can control the rainwater run-off, provided that the horizontal and vertical road design are completed.

Selecting the Slope Scale

To select the desired scale, go to the *Road Design* menu, select the *Road Preferences* command and define the slope scale in the *Slope Analysis* tab. The following window appears on the screen:



In the *Sampling* group you can define the *Maximum* and the *Minimum steps* according to which the program searches the slope differences in the parts of the road. The bigger these steps are, the quicker the analysis, but with less accuracy.

In the *Color scale* group, you can select the desired scale from the 

drop-down list.

7.3 Road Design

7.3.1 Introduction

This chapter explains all operations that you can perform from the *Road* mode of the *Horizontal Alignment* workspace or that are directly related to the design of a road. The data of the tangent curve are entered graphically, showing also the distance of the PI to be inserted from the previous one, or numerically (by typing the coordinates of the PIs). They can also be imported from a text file or you can convert a drawing line to a tangent curve.

Before the road is calculated, you must check all properties of the road, the properties of the PIs and the general preferences. You can select from numerous sequences of transition curves and turns for every PI and for two successive PIs. For every PI or couple of PIs, the program suggests the appropriate values for the length or the parameter of the clothoid or the tangent offset.

You can also intervene graphically to the axis before or after the cross section calculation or even after the stationing. Every time you move an object, the program checks whether there is an acceptable solution. You can insert the cross sections numerically, but it is also easy to insert graphically the cross sections, as well as widenings and shoulders. This insertion is even easier when you use the snap feature on terrain point, drawing, etc.

After the calculation of the axis, you can view and edit the superelevation diagram by applying different rules and different parameters. After the statioting, you can immediately update and view the Profile workspace with the profile of the natural terrain, as well as the *Cross Section* workspace with the natural terrain per cross section. In the final stage, when you update from the *Profile* and *Cross Sections* workspaces, you can view in the *Horizontal Alignment* workspace data referring to the road profile, the roadlines and the sideslopes of the road.

7.3.2 Creating a New Road

If you have opened a HOR file, *Tessera* recognizes automatically the roads contained in the file and there is no need to recalculate them. In this case, you can also see the sideslope and roadway lines, if any.

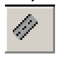
A road in *Tessera* is comprised of PIs. The procedure that you should follow to insert and calculate a new road has seven stages:

1. Definition of the names and coordinates of the road PIs .
2. Determination of the curve type at each road PI and of the corresponding values (e. g. Radius).
3. Determination of the road’s specifications regarding its widths and the superelevation diagram.
4. Calculation of the road and PI data editing in case an error occurs during the calculation of the road.
5. Definition of widenings and medians.
6. Creation of cross sections.
7. Update of the other two workspaces.

Inserting a Road

To insert a new road, you must first specify the coordinates of its PIs and then the geometric characteristics of every PI.

How to insert the PIs of a road

1. From the *Road Design* menu, run the *Insert road* command, or from the *Elements Drawing* toolbar, click the  button. The highlighted square in the middle of the mouse cross shows you the place where the PI will be inserted.
2. Use the mouse to specify in ascending order the position of the PIs of the road's tangent curve, or use the *Input Form* dialogue box or the dialogue box of the snap feature for more accuracy, working from the beginning to the end of the road.
3. When you have finished importing PIs, click the right mouse button to end the procedure.

If you want to set the road PIs with higher accuracy, you can do this from the window of the *Road PIs* command.

You can also create a tangent curve from a drawing line. Simply select the drawing line and from the menu that will be displayed if you right-click in the drawing area, select the *Convert* submenu and run the *To road* command. A new road will be created, whose PIs will be the points of the drawing line. The order of the PIs is the same as the order in which the points of the drawing line were inserted.

Importing a Road from a Text File

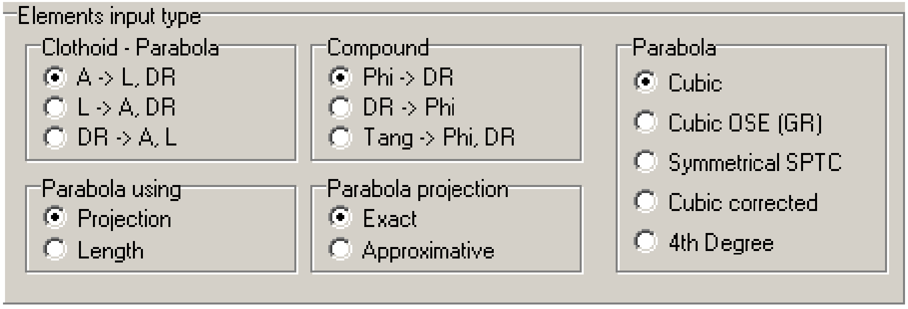
You can import the tangent curve of a road from text files (*ASCII*). To do this, select the *Import from ASCII* submenu of the *File* menu and run the *New road tangent curve* command. A dialogue box will appear on the screen, where you can select a tangent curve file (such files have usually the .XYV extension), containing all data required for the calculation.

Each line of the file corresponds to a PI. A comma separates the data on every line. The first data is the code that corresponds to the type of turn. The codes of the transitions are listed in the following table.

Code	**Type of turn**
1	Clothoid – Circle - Clothoid
2	Parabola – Circle - Parabola
3	Compound
4	Clothoid – Circle – Egg-shaped – Circle –Clothoid
5	Clothoid – Circle – Clothoid (two PIs) / Hairpin bend
6	Clothoid – Circle – Circle – Clothoid
10	Auxilliary PI (Used only when the previous PI is 4 or 5 or 6)

The codes from 1 to 3 refer only to the PI of the particular line. The codes from 4 up to 6 refer to the PI of the particular line and to the next one, which is always indicated by the code 10.

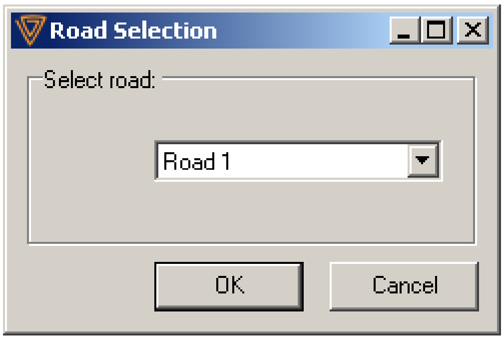
The other data that you must input is the PI name and the X and Y coordinates. Then, and according to the type of the PI, you must input the necessary elements and the method that you have selected for the calculation of the PIs. To do this, go to the *Project* menu → *General Preferences* command → *Calculations* tab → *Elements input type* group.



For the clothoid and the parabola (codes 1 and 2), first you must input the parameter or the length of the clothoid or the leading tangent offset (depending on the selected calculation method), and then the radius and the corresponding value at the entrance. For the compound, you must input the angle of the leading curve or the leading tangent offset or the tangent length following this order, and then the radius of the trailing curve, the radius of the main circular arc, the radius of the exit arc and finally the angle of the exit arc or the trailing tangent offset or the tangent length for the exit. For the double PIs, the values are the leading element and follow the radius of the first circular arc, the radius of the second circular arc and the trailing element. For the auxiliary PIs (code 10) no additional data is needed, since all the necessary data have been input for the previous PI.

Importing a Road from a Project

To transfer a calculated road from an existing file to the current project, you must go to the *File* menu and run the *Load Road* from *Project* command. The following window will appear on the screen:



Here, you can select the road you wish to import. Click the *OK* button to view the calculated road, which maintains not only its PI data but also its sideslopes.

Terrain Profile

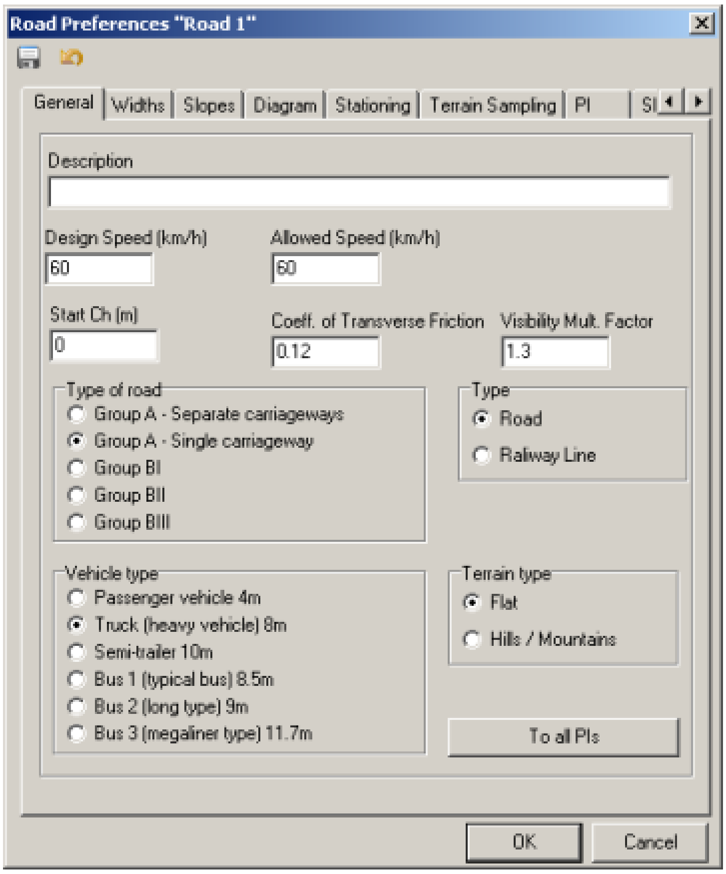
When the road design is performed, you can view what the terrain profile will be, without having to update the *Profile* workspace. In this way, you can view the slopes of the terrain at the point where the road has been designed. Thus, you can detect possible problems (big terrain slopes) and correct them by moving the road design. To do this, select the *Terrain Profile* option from the *Road Design* menu. A window will appear on the screen, showing the designed terrain profile.

7.3.3 Road Properties

The specifications of a road depend on many factors. The specifications that must be adhered to are usually those stipulated by the relevant regulations or by the supervising authority or, finally, by the project's needs. In any case, the careful setting of the road's specifications is a main prerequisite when designing a road, in order for the road to have the desired layout.

How to set the specifications of a road in Tessera

1. Select the desired road and from the *Road Design* menu select the *Road Preferences* command.
2. From the dialogue box of the *Road Preferences* command, select a tab and proceed to the desired settings.



3. When you have finished setting the preferences, click *OK* to close the dialogue box.

You can make settings about:

- The elements of the road PIs (*General, PI* and *Slopes* tabs).
- The road widths (*Widths* tab).
- The necessary parameters for the calculation of the superelevation diagram (*Slopes* and *Diagram* tabs).
- The stationing preferences (*Stationing* and *Terrain Sampling* tabs).

Chainage

The chainage of the road axis is performed according to the order in which the PIs were inserted, starting from the first PI.

<u>Starting Chainage</u>

The initial value of the starting chainage is zero. If your design concerns only a section of a road, obviously the *Starting Chainage* is different. This is defined from the dialogue box of the *Preferences* command.

How to define the Starting Chainage

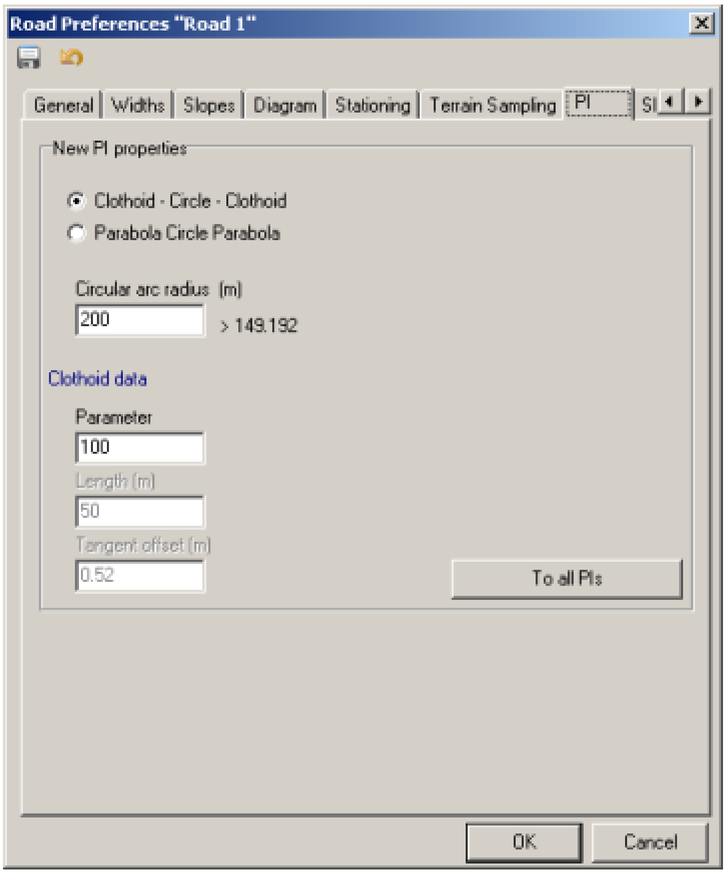
1. From the *Road Design* menu, run the *Road Preferences* command.
2. In the displayed dialogue box, select the *General* tab.
3. In the *Start Ch* text field, type the *Staring Chainage* of the road.
4. Click the *OK* button.

<u>Reverse Chainage</u>

Chainage is performed according to the order in which the PIs were inserted, i.e. zero chainage coincides with the first road PI that you inserted and is independent of the PI names. To reverse the chainage, select the road, right-click in the drawing area, and from the displayed menu run the *Reverse road* command. The program will rename the PIs and update the stations of the horizontal alignment based on the new chainage. In addition, the widenings will be transferred to the opposite side of the road. In order for the profile and the cross sections to comply, you must proceed to the relevant update.

7.3.4 PI Properties

The properties of the PIs are defined from the *Road Design* menu, by selecting the *Road Preferences* command. Set the properties in the corresponding fields of the *General* and *PI* tabs, in the dialogue box shown below:



In the General tab, you can set all the parameters required for your project. The required data is the road(s) design speed *V(km/h)*, the coefficient of transverse friction f and the maximum curve superelevation *q*. Type these values in the *Design speed* (km/h) and *Coefficient of transverse friction* fields of the *General* tab respectively, and in the *Maximum curve superelevation* field of the *Limit Values* group in the *Slopes* tab.

To set the type and the data of the calculated curve at every new PI, go to the *PI* tab. In this window and only in the Professional version, you can select the new PIs to be of the *Clothoid –Circle – Clothoid* or *Cubic Parabola* type, by ticking the relevant check box. In the *Standard* version, the new PIs are bound to be of the *Clothoid – Circle – Clothoid* type.

$$R = \frac{V^2}{127(q+f)}$$

In the *Circular arc radius* text field, enter the curvature radius *R*. The minimum suggested value for the curvature radius, which is displayed next to the field, derives from the above formula and depends on the design speed, the coefficient of transverse friction and the maximum curve superelevation that you have entered in the *General* tab. You have the option to change the minimum radius, if desired.

In the last three fields, you can set the parameter A of the curve, the length of the clothoid L as well as the offset ε, by using either the clothoid or the parabola (*Professional* version) as transition curve. According to literature, when the transition curve is a clothoid, these three values are related with the following formulas:

$$L = \frac{A^2}{R} \quad \varepsilon = \frac{L^2}{24R}$$

If you change, for example, the length of the clothoid, then the offset and the parameter A will also change. Depending on the calculation method you have selected from the *Project* menu *General Preferences* *Calculations* tab *Elements input type* group, only one of the three text fields is active.

When the transition curve is a parabola, the formulas applied for the calculation of the values will change according to the type of the parabola, which is also selected from the *Elements input type* group of the *Calculations* tab in the dialogue box of the *Project* menu.

When you have made all the desired settings as described above, and if these are the settings that you usually work with, it is recommended to click the *OK* button. The *To all PIs* button replaces all the values of the existing PIs of the road with the ones you have entered in the dialogue box.

7.3.5 Editing PIs

Adding PIs to a Road

You can add as many PIs as you want to an existing road.

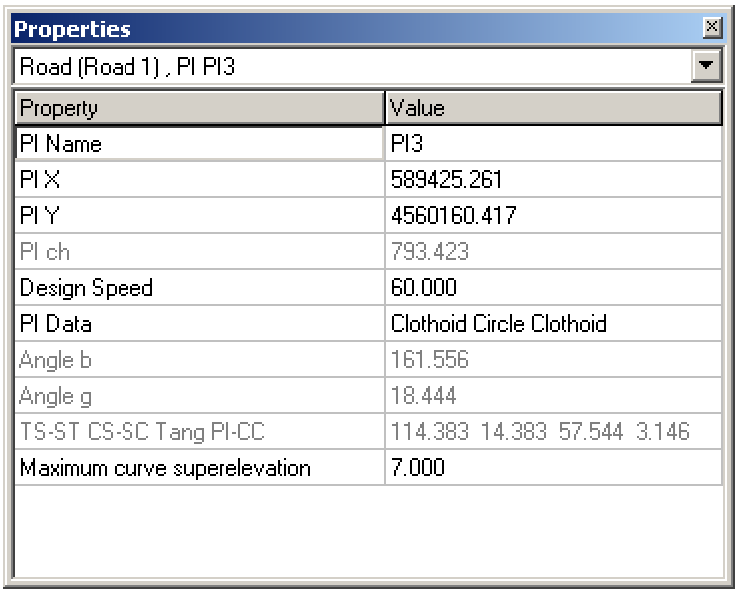
How to add a PI to a road

1. Select the road you wish to edit by placing the mouse pointer over a PI and clicking the left mouse button.
2. From the shortcut menu, which is displayed if you right-click in the drawing, select *Extend towards start* or *Extend towards end*, depending on the direction towards which you wish to insert the new PI.
3. Specify the position of the new PI. If the new PI is inserted between two existing PIs, the program displays a confirmation message, and you must select whether the PIs that are next to the new one will be renamed.

Moving PIs on a Road

You can easily move the PIs with the mouse. Left-click to select and highlight the desired PI and then simply drag the PI to the desired place. When you drag the PI, the L1 and L2 distances between the PI you are dragging and its two adjacent ones (previous and next) appear on the left of the status bar.

You can also change the position of the PI from the window of the Properties command.



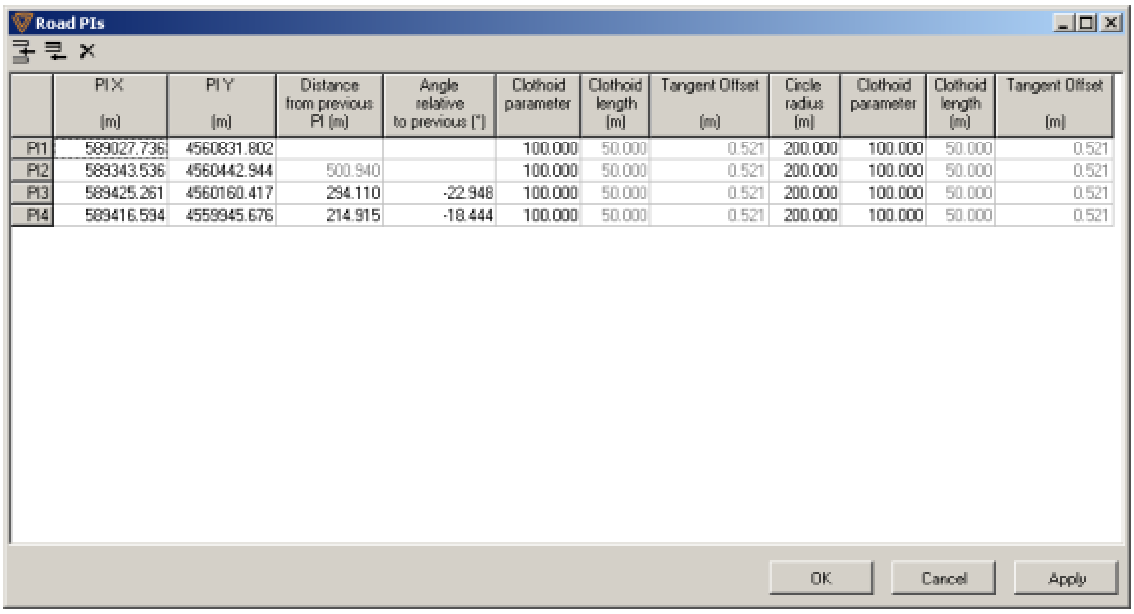
When this window is open and you select the desired PI, you can define the new position of the PI by typing the X and Y coordinates in the *PI X* and *PI Y* text fields.

Batch Moving of Road PIs

From the *Road PIs* command window you can carry out a batch editing of the PIs positions.

How to carry out a batch move of the road PIs

1. Select the desired road and then run the *Road PIs* command from the *Road Design* menu.
2. The program displays the following window, which is comprised of twelve columns that correspond to the serial number of the PI, the X and Y coordinates of each PI, the distance from the previous PI, the angle from the previous section, the leading clothoid parameter, the leading clothoid length, the leading tangent offset, the circle radius, the trailing clothoid parameter, the trailing clothoid length and the trailing tangent offset. Each line of the window corresponds to a PI.



	PI X (m)	PI Y (m)	Distance from previous PI (m)	Angle relative to previous (°)	Clothoid parameter	Clothoid length (m)	Tangent Offset (m)	Circle radius (m)	Clothoid parameter	Clothoid length (m)	Tangent Offset (m)
PI1	589027.736	4560831.802			100.000	50.000	0.521	200.000	100.000	50.000	0.521
PI2	589343.536	4560442.944	500.940		100.000	50.000	0.521	200.000	100.000	50.000	0.521
PI3	589425.261	4560160.417	294.110	-22.948	100.000	50.000	0.521	200.000	100.000	50.000	0.521
PI4	589416.594	4559945.676	214.915	-18.444	100.000	50.000	0.521	200.000	100.000	50.000	0.521

3. Change the values in the desired text fields by typing the new coordinates and then press the *Enter* key to save the value.
4. Click the *OK* button to confirm the changes or click *Cancel* to restore the original settings.

Renaming Road PIs

The program renames the PIs of a road in ascending order and according to the order in which the PIs were inserted. To change the name of a PI, run the *Properties* command and in the *PI name* field change the name of the PI.

Deleting a Road PI

You can also delete a road PI.

How to delete a road PI

1. To select the road PI you wish to delete, place the mouse pointer over the PI and click the left mouse button.
2. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Delete PI* command.

Graphical Change of the Radius

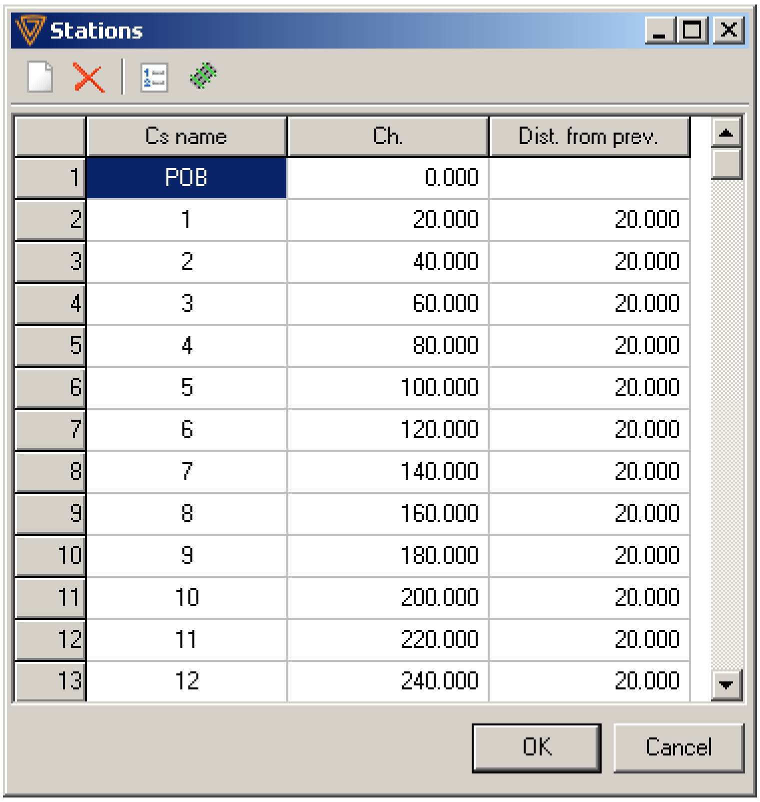
You can move graphically the radius of a road PI.

How to move graphically a radius

1. Select the PI whose radius you wish to change.
2. You will notice that a yellow box appears at the point that corresponds to the center of the circular arc.
3. If you place the mouse pointer there, the color changes.
4. With the mouse pointer in the square, click the left mouse button to lengthen or shorten the radius of the selected PI’s circular arc.

7.3.6 Editing Road Stations

From the *Road Design* menu, run the *Road Stations Table* command. In the displayed window you can proceed to the batch editing of the stations positions.



	Cs name	Ch.	Dist. from prev.
1	POB	0.000	
2	1	20.000	20.000
3	2	40.000	20.000
4	3	60.000	20.000
5	4	80.000	20.000
6	5	100.000	20.000
7	6	120.000	20.000
8	7	140.000	20.000
9	8	160.000	20.000
10	9	180.000	20.000
11	10	200.000	20.000
12	11	220.000	20.000
13	12	240.000	20.000

The above window has three columns: Station *name, Chainage* and *Distance from previous station*. There are also three buttons:

: Insert new station.

: Delete selected station.

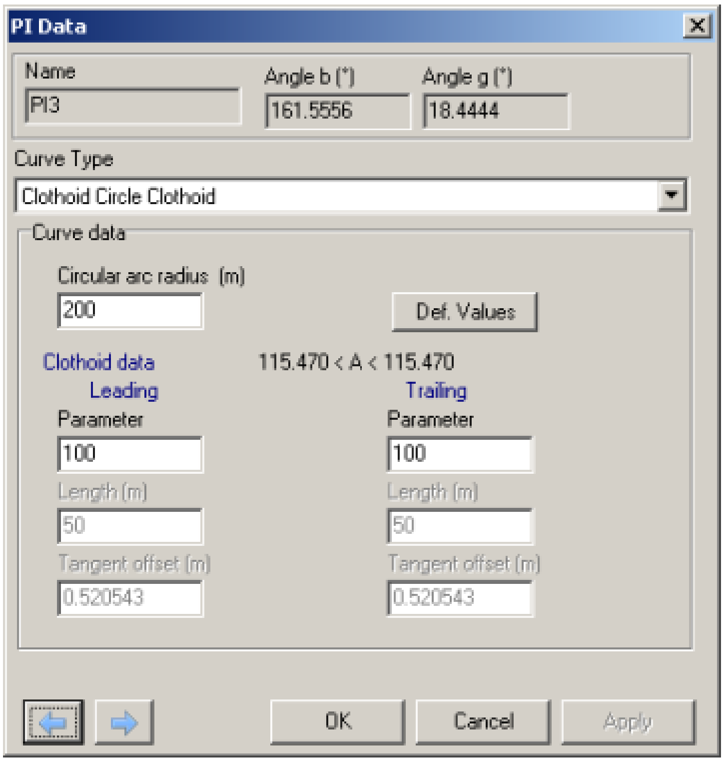
: Numbering of next station based on the name of the selected one.

7.3.7 Editing PI Data

Once you have imported all the PIs of a road's tangent curve, the next step for the calculation of the road is to define the data of every PI.

How to input the data of a road's PI

1. Display the window of the *Properties* command.
2. Select the road you wish to edit.
3. Select the PI you wish to edit.
4. In the *Properties* command window, select the value of the *PI Data* property. Click the three-dots button , appearing when you select the box.
5. The following dialogue box appears on screen:



Follow the same procedure to change the data of a road's PI, regardless of whether the road is calculated or not.

From the *Curve Type* field, select the desired type of curve for the PI. The parameter fields change based on the curve type.

7.3.8 Types of Transition Curve

Anadelta Tessera offers various types of transition curves, both in the *Standard* and in the *Professional* version.

How to change the type of transition curve

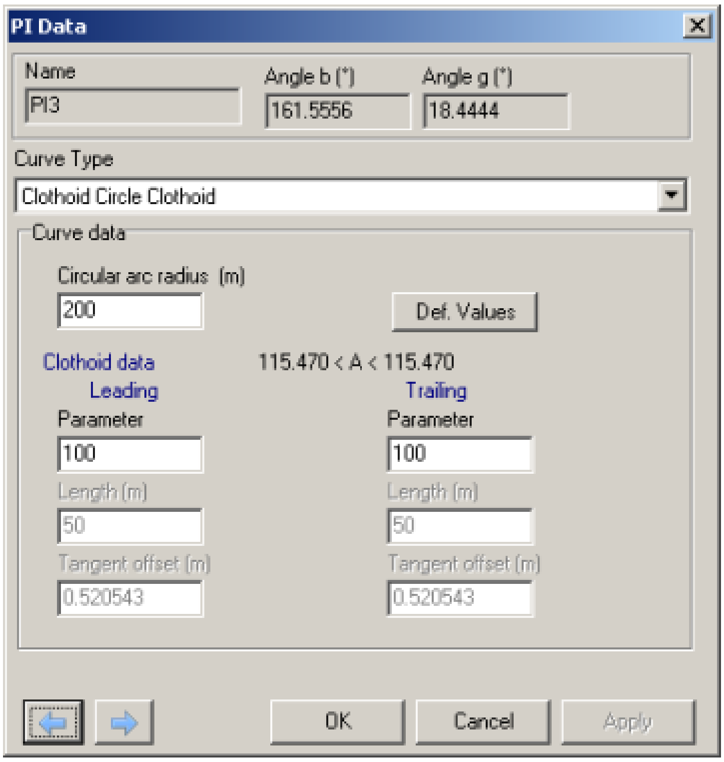
1. Display the *Properties* command window.
2. Select the road you wish to edit.
3. Select the PI you wish to edit.
4. In the *Properties* command window, select the value of the *PI Data* property. Click the three-dots button, appearing when you select the box.

Transition Curves in the Standard Version

There are three available types of transition curves in the *Standard* version: the simple clothoid, the compound and the clothoid circle clothoid (two PIs).

<u>Simple Clothoid</u>

The most common type of curve is the clothoid. When you select *Clothoid Circle Clothoid* from the drop-down list of the *Curve Type* field, the dialogue box of the *PI Data* command opens as follows:



In the *Circular arc radius* text field, you can edit in meters the radius of the circular arc of the turn. The radius *R* can even get a zero value if the road axis follows a zigzag line (in case you have to depict an existing state). If the values in the fields are such, that it is impossible to calculate the curve, then the fields appear in red. Enter the data for the leading transition curve in the text fields of the left column and the data for the trailing transition curve in the text fields of the left column, since you have the option to create asymmetrical turns. Depending on how the parameter, the length and the offset derive, according to your selection in the *Calculations* tab of the dialogue box of the *General Preferences* command, only one of the three text fields is active. Also, the *Clothoid data* field displays the minimum and maximum values you have set for the clothoid (*A,L,ε*). The clothoid values derive from the aesthetic convention *1/3<A/R<1m*, and by applying the relation *A2=L*R* the result is *R/9<R<L or R/3<A<R.*

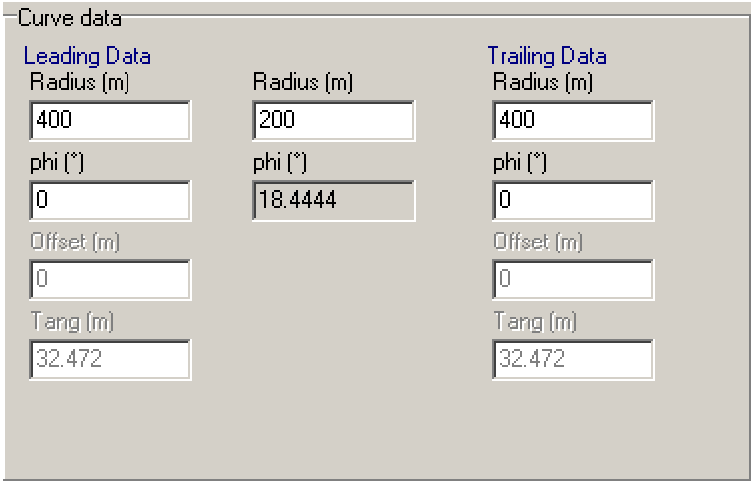
When the curvature radius *R* changes, the values of the clothoid length also change, while a change in the length of the clothoid also changes the value of the offset ε. Any change of the parameters affects the value of the *A/R* ratio (=$\sqrt{L/R}$). The limits of this ratio depend on the PI angle, therefore the maximum value derives from the *CS-SC>=0.15TS-CS* convention, while the minimum value derives from the *TS-CS>=0.4E E* convention.

The *A/R* ratio must be within the range of the suggested values. The minimum value of this ratio ensures the minimum clothoid length for adjusting the cross slope or the superelevation. It also ensures the minimum length imposed by reasons of dynamics. The minimum value of the ratio is replaced by the minimum value of the aesthetic convention (1/3) for angles wider than 182 degrees, while the maximum value is replaced by the maximum limit of the aesthetic convention for angles narrower than 137 degrees. The minimum value can be skipped for high curvature radii where the use of clothoid is not required. The maximum value can also be skipped for road design with high clothoid parameter values without an intermediate circular arc. When the curvature radius R changes, the suggested limits of the selected parameter also change accordingly.

When you click the *Def. Values* button, the program enters in the fields of the selected parameter the minimum value that has been calculated based on the radius, for the entrance and the exit. You can choose whether the field will be filled with the minimum or average or maximum value of the parameter from the menu that is displayed if you click the right mouse button.

Compound Curve

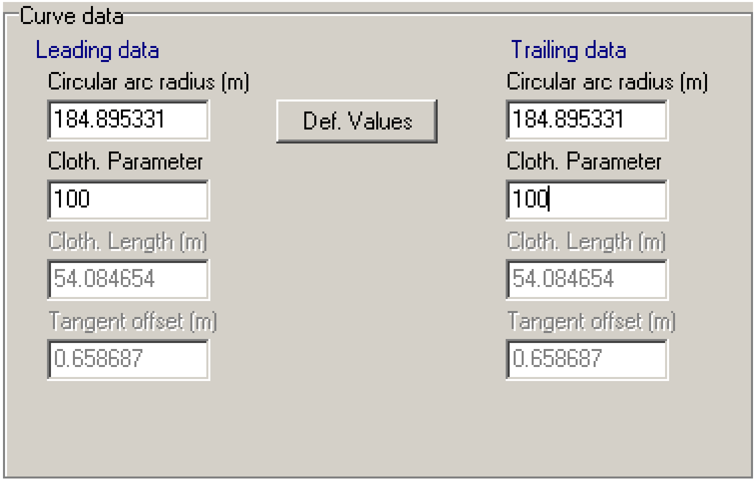
The next transition curve is the compound. When you select the *Compound* option from the drop-down list of the *Curve Type* field, the *Curve parameters* group of the *PI Data* command is displayed as shown in the figure:



The group has three columns with text fields. In the *Radius (m)* text field of the middle column, you can edit in meters the radius of the turn's circular arc. Enter the data for the leading transition curve in the text fields of the left column (*Leading Data*) and the data for the trailing transition curve in the text fields of the right column (*Trailing Data*). Here, in the *Radius (m)* text field, you can edit in meters the radius of the circular arc at the entrance or exit of the turn. According to the calculation method you have selected for the compound curves in the dialogue box of the *General Preferences* command, only one of the other three text fields is active: the central angle φ(°), the *Offset* (m), namely the distance of the central circular arc from the side of the tangent curve, or the length of the *tangent (m)*. The other two fields are automatically calculated by the program and their data are displayed in the box.

Clothoid – Circle – Clothoid (2 PIs)

If you want to design a hairpin bend between two PIs, you must select the *Clothoid circle clothoid (2 PIs)* from the drop-down list of the *Curve Type* field. The *Curve data* group in the dialogue box of the *PI Data* command box is displayed as shown in the figure:



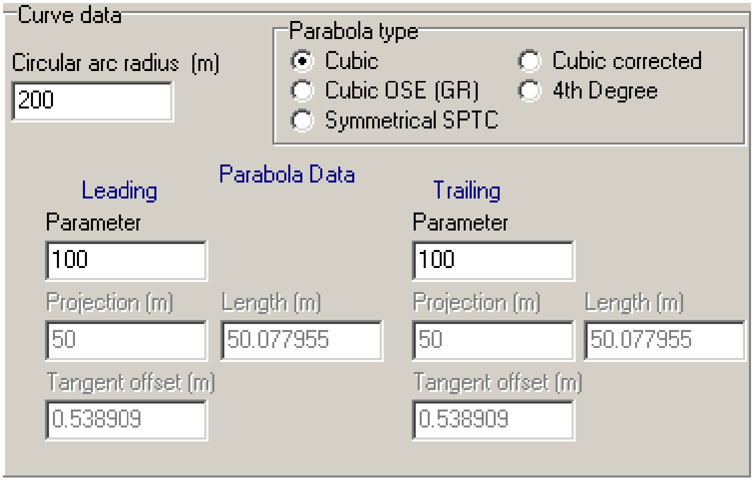
The group has two columns with text fields. Enter the data for the leading transition curve in the text fields of the left column (*Leading Data*) and the data for the trailing transition curve in the text fields of the right column (*Trailing Data*). In the *Circular arc radius (m)* text field of each column, you can edit in meters the radius of the circular arc of the turn. It is known that, depending on the position of the road PIs and the value of the offset, there is only one value for the radius that abuts on both PIs. When you click the *Def. Values* button, the program calculates this value according to the current value of the leading and trailing offset and enters the result in the relevant fields. According to the calculation method you have selected for the compound curves in the dialogue box of the General Preferences command, only one of the other three text fields is active: *Cloth. Parameter, Cloth. Length (m)* or *Tangent offset (m)*. The other two fields are automatically calculated by the program and their data are displayed in the box.

Transition Curves in the Professional Version

The *Professional* version offers three additional types of transition curves: cubic parabola, transition curve of two adjacent PIs with two circular arcs, and egg-shaped curve.

<u>Cubic Parabola</u>

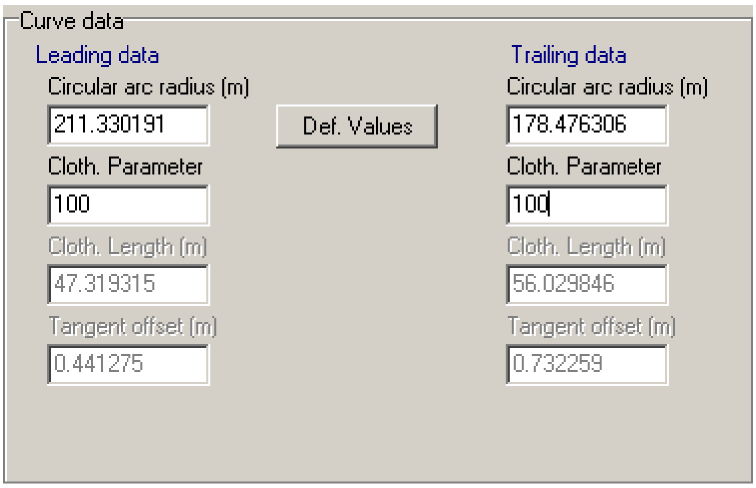
By selecting the *Cubic parabola* from the drop-down list of the *Curve Type* field, the dialogue box of the *PI Data* command is displayed as shown in the figure:



In the *Circular arc radius* text field, you can edit in meters the radius of the circular arc of the turn. In the text fields below, enter the data for the leading and trailing transition curves in the left and right columns, respectively. According to the calculation method you have selected for the length and the offset in the dialogue box of the *General Preferences* command, only one of the three text fields is active. Also, in the *Parabola Type* group, you can select one of the available cubic parabola types, which can be different from one PI to another.

<u>Clothoid – Circle – Circle - Clothoid</u>

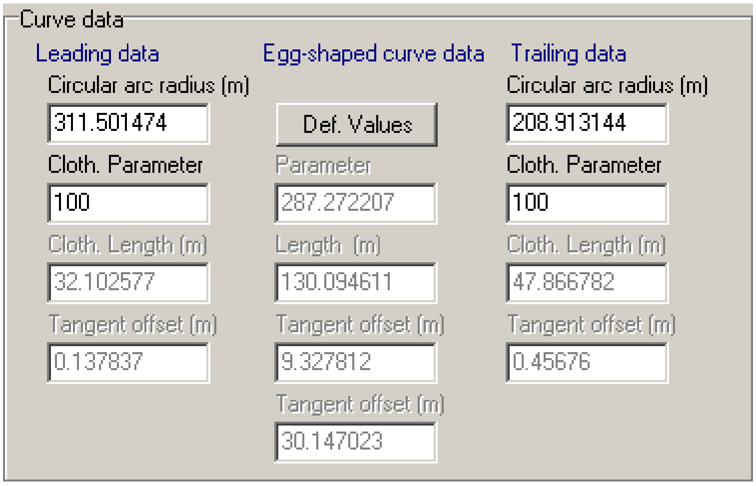
If you want to design a hairpin bend between two PIs with two different curvature radii, you must select the *Clothoid circle circle clothoid (2 PIs)* from the drop-down list of the *Curve Type* field. The *Curve data* group in the dialogue box of the *PI Data* command is displayed as shown in the figure:



The group has two columns with text fields. Enter the data for the leading transition curve in the text fields of the left column (*Leading Data*) and the data for the trailing transition curve in the text fields of the right column (*Trailing Data*). In the *Circular arc radius (m)* text field of each column, you can edit in meters the radius of the circular arc. It is known that, according to the position of the road PIs and given the value of a radius and the value of the leading and trailing offsets, there is only one value for the second radius that abuts on both PIs. Therefore, when you change the value of the leading radius, the trailing radius also changes automatically. If you change the leading offset, the leading radius will change accordingly. When you click the *Def. Values* button, the program suggests certain values for both radii, according to the angles of the two PIs and the values of the leading and trailing offsets. The suggested values are shown in the corresponding fields. Right below and depending on the calculation method you have selected for the clothoid in the dialogue box of the *General Preferences* command, only one of the other three text fields i.e *Parameter, Length(m)* or *Offset(m)*, is active. The other two fields are automatically calculated by the program and their data are displayed in the box.

<u>Egg-shaped Curve</u>

By selecting the *Clothoid circle egg-shaped circle clothoid (2 PIs)* from the drop-down list of the *Curve Type,* the *Curve data* group in the dialogue box of the *PI Data* command is displayed as follows:

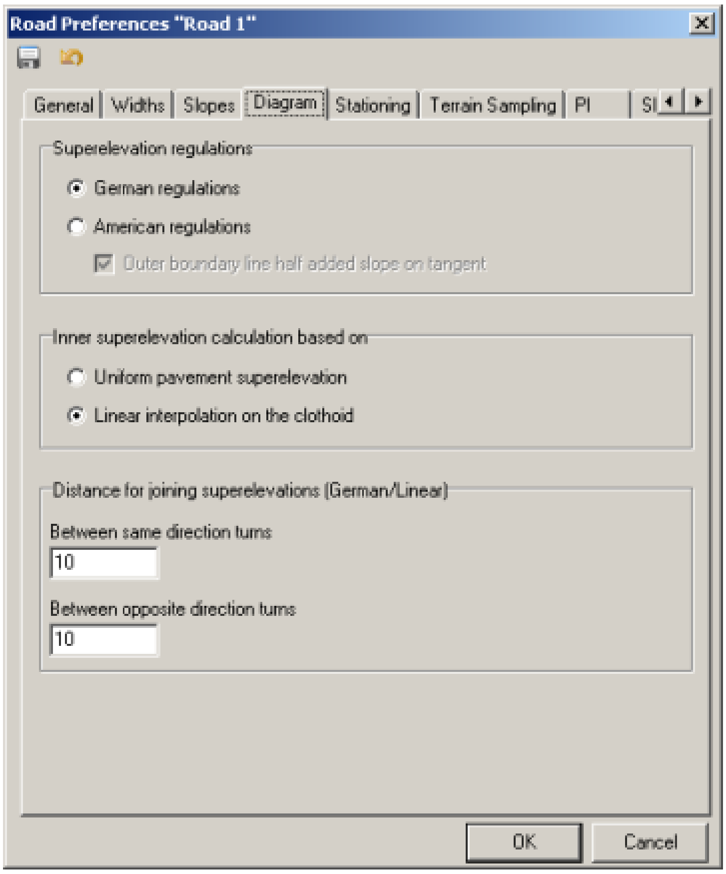


The group has three columns with text fields. In the *Circular arc radius (m)* field of each column, you can edit in meters the radius of the circular arc of the turn. By clicking the *Def. Values* button, the program calculates the radii of the circular arcs according to the angles of the two PIs and the values of the leading and trailing offsets, and enters the results directly in the corresponding fields. Enter the data for the leading transition curve in the text fields of the left column (*Leading Data*) and the data for the trailing transition curve in the text fields of the right column (*Trailing Data*). Depending on the calculation method you have selected in the *Elements input type* of the *Calculations* tab, in the dialogue box of the *General Preferences* command of the *File* menu, only one of the other three text fields i.e *Parameter, Length(m)* or *Offset(m)* is active. The other two fields are automatically calculated by the program and their data is displayed in the box.

7.3.9 Superelevation Diagram

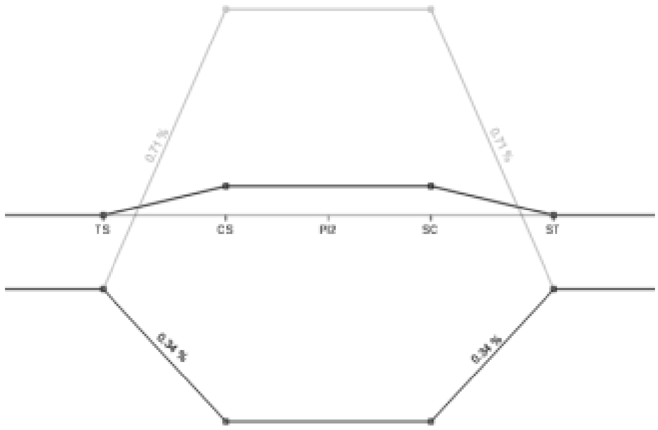
Methods for the Calculation of the Superelevation Diagram

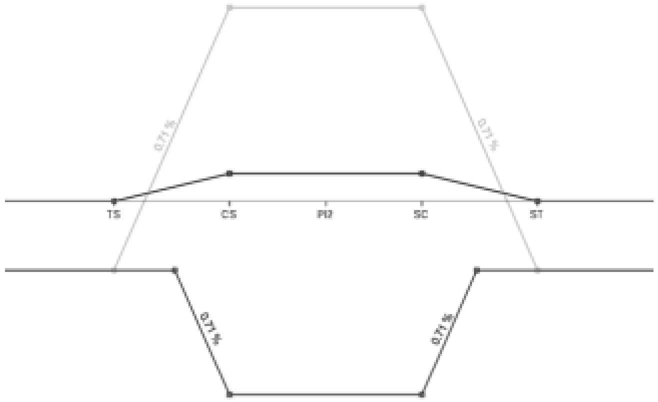
To specify the preferences that will apply for the calculation of the superelevation diagram, run the *Road Preferences* command from the *Road Design* menu and then go to the *Diagram* tab. The following dialogue box will open:



In this dialogue box, first select the method for the calculation of the superelevations. All of the following shall apply to curves of the clothoid-circular arch-clothoid type. In the case of the simple circular arc, the program ignores your selection of the superelavation calculation method. Even in the case of double PIs, these are considered as a single curve for the purposes of the superelevation diagram.

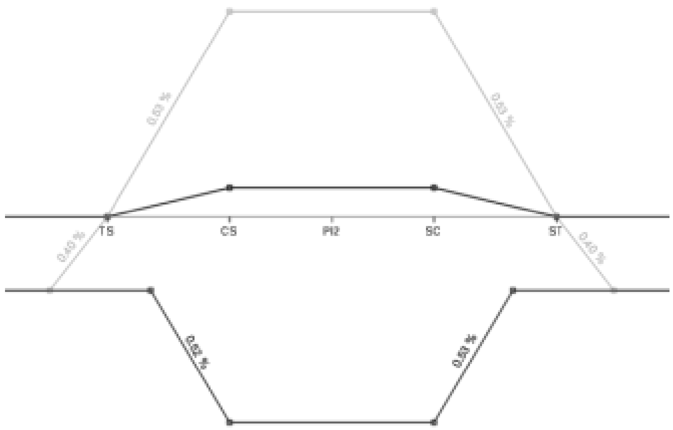
When you select the *German Preferences* option, the program assumes that the superelevations change (from the dual-grade station to the single-grade one) from the beginning of the curve, namely from the position where TS station will be inserted afterwards. It also assumes that the end of the curve (TS station) is the restoration point of the superelevations in the dual-grade station state. The start and end points of the circular arc (CS and SC) are identified with the start and end points of the single-grade station. The figure shows two examples of superelevation diagrams that have been calculated in this way.

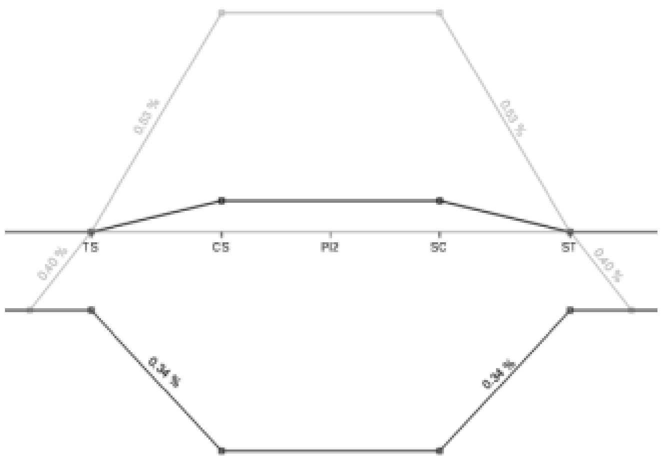




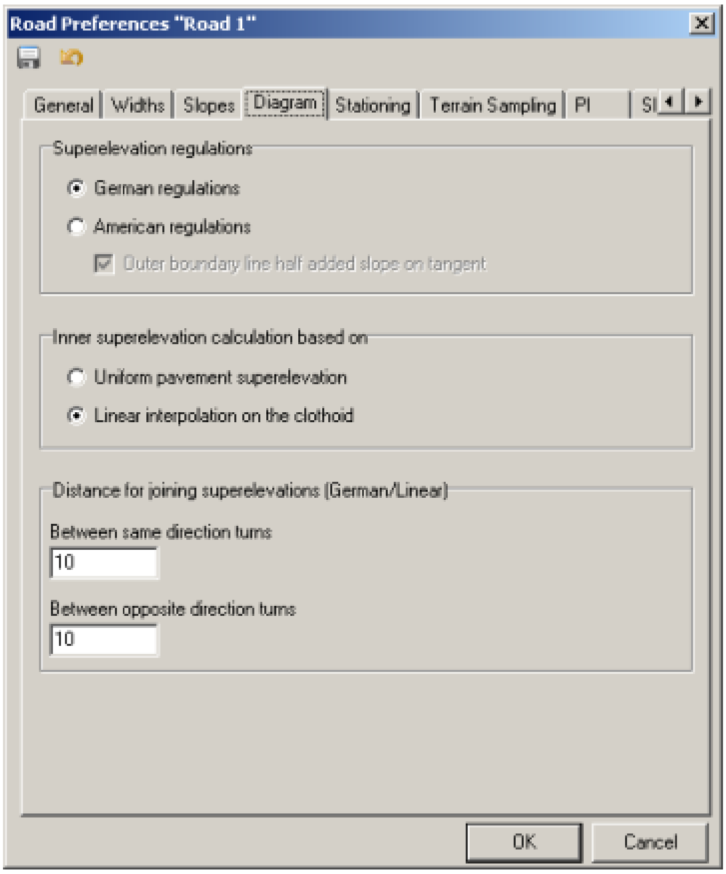
The first example has been calculated based on uniform pavement superelevation, which will be analyzed hereinafter, while the second one was based on linear interpolation on the clothoid. Apart from the two lines of the superelevation diagram, the figure also shows the curvature diagram, with the characteristic positions TS, CS, SC and ST.

When you select the *American Preferences* option, the program assumes that the superelevation of the external side at point SC (beginning of curve) and at point ST (end of curve) has already changed to zero value. The start and end points of the circular arc (TS and CS) are identified, as stated above, with the start and end points of the single-grade station. The points where the superelevation variation begins and ends (on tangents) derive from linear calculation. The figure shows two examples of superelevation diagrams that have been calculated in this way.





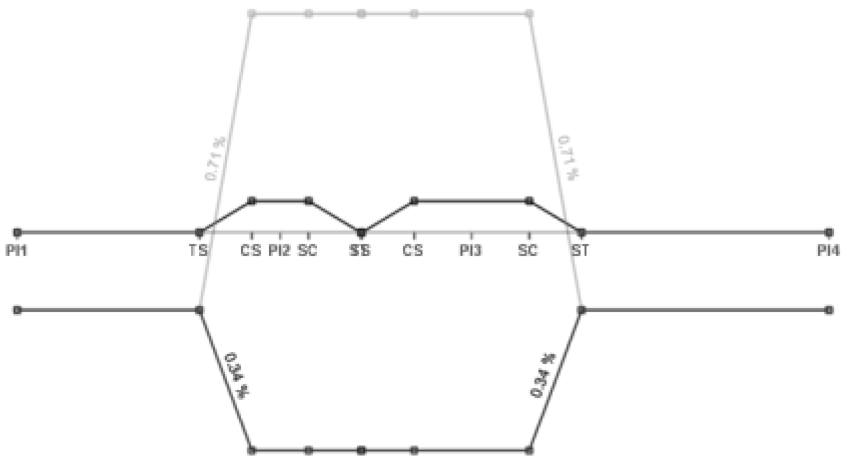
From the dialogue box shown below,



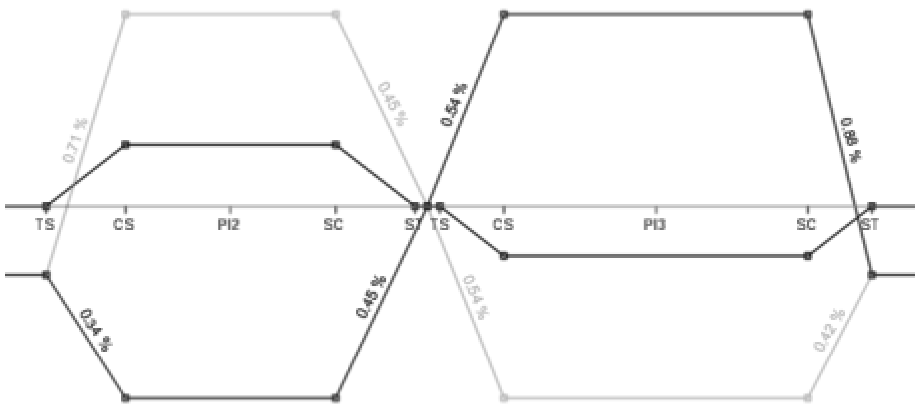
you can also select the superelevation calculation method inside the curve. If you select *Uniform pavement superelevation,* the superelevation of the inner side of the road will maintain the value it has on the tangents, until the superelevation of the outer side assumes the value that corresponds to the state of the single-grade station of uniform superelevation. The state of uniform superelevation is maintained until it assumes its maximum value at the CS point; this value remains the same until the SC point. From SC to ST, the procedure is reversed.

If you select *Linear interpolation on the clothoid,* the superelevation of the road's inner side changes linearly (from the value on tangent to the maximum value) and regardless of the change of the superelevation on the outer side. The above figures show the two alternative solutions of superelevation diagrams (inner roadline) for each of the first two calculation methods.

The *Distance for joining superelevations* matters only when the *German Preferences* and *Linear interpolation on the clothoid* check boxes are ticked. When the turns are of the same direction and the distance between the end of the trailing transition curve (ST) and the beginning of the leading transition curve of the next turn (TS) is shorter than the distance you have typed in the *Between same direction* turns text field, then the left and right superelevations will remain unchanged between the two turns.

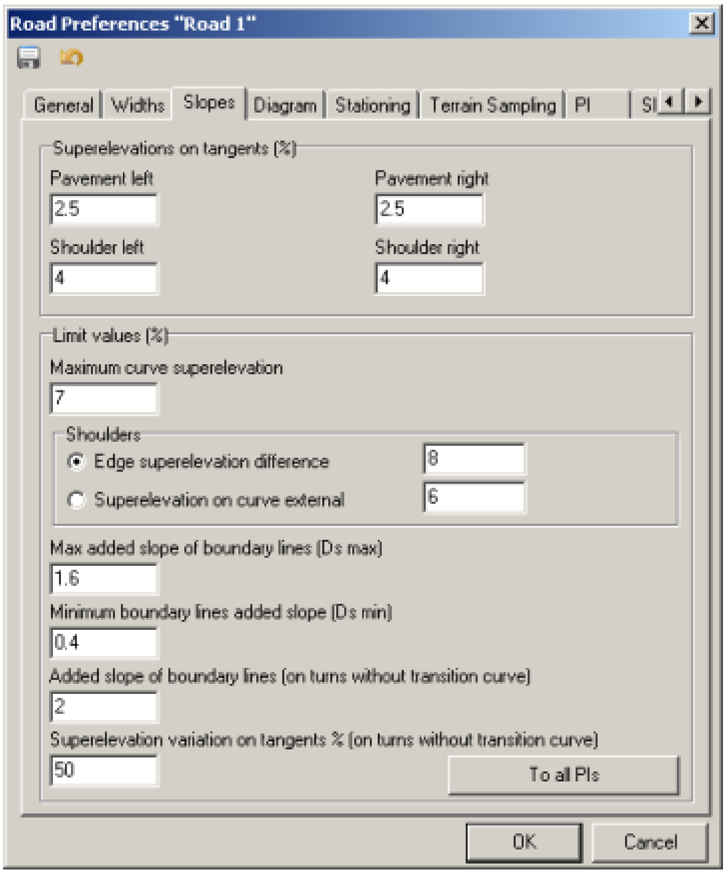


In the case of two consecutive opposite direction turns, if the distance between the end of the trailing transition curve (ST) and the beginning of the leading transition curve of the next turn (TS) is shorter than the distance you have typed in the *Between opposite direction turns* text field, then at the exit of the first turn the left and right superelevations are restored gradually to the 0% value, and not to the values of the normal cross section (-2%, 2% or -2.5%, 2.5%), while the zero point of the superelevations is not ST, but the center of the TS-ST distance, where the superelevations of the next turn begin.



Superelevation Diagram Values

To set the values of the superelevation diagram, namely the superelevation values on tangents and turns, go to the *Road Design* menu, run the *Road Preferences* command and make your settings in the dialogue box of the *Slopes* tab.



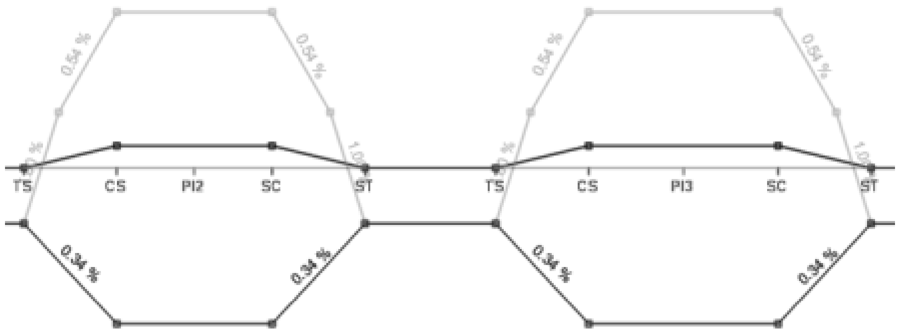
Enter the values of the superelevations on a road's tangents in the *Superelevations on tangents (%)* group fields. In the *Pavement left* and *Pavement right* text fields, enter the left and right superelevations of the pavement, respectively. Enter the left and right superelevations of the shoulder in the *Shoulder left* and *Shoulder right* text fields, respectively.

In the *Maximum curve superelevation* field, enter the value of the maximum superelevation along the circular arc. This setting shall apply to all PIs and to update them you must click the *To all PIs* button. If you want the superelevations on tangents to apply also to the turn, then the value that you will enter in the field must be negative to the value that you entered for the tangent (*e.g. -2% or -2.5%*). If you want to change the maximum superelevation at a particular turn, you can do this from the window of the *Properties* command.

In the *Shoulders* group, select how the program will calculate the shoulders. Detailed explanations on the fields are provided in another chapter, where the calculation of shoulder elevations is analysed.

In the *Max added slope of boundary lines (Ds max)* field, enter the maximum variation rate of the outer roadline (maximum longitudinal slope) between the value of the normal cross section (*-2% or -2.5%*) and the maximum value. This is required in the case of short clothoid lengths, in order to avoid breaks at the roadlines.

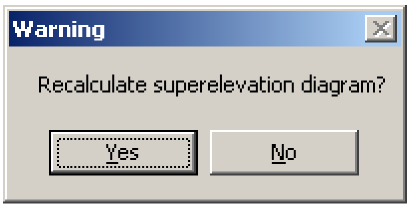
The *Minimum boundary lines added slope (Ds min)* field affects the elevation diagram that will derive, by determining the minimum variation rate of the outer roadline (minimum longitudinal superelevation) between the value of the normal cross section (-*2% or -2.5%*) and the positive value (*2% or 2.5%*). This is required in the case of long clothoid lengths, in order to avoid superelevation values near 0% for the road section that will be calculated. The figure shows how this value affects the superelevation diagram for both calculation methods, when the value in the relevant field is 1%.



The *Added slope of boundary lines (on turns without transition curve)* field matters only when the clothoid length is zero and the value specifies the variation rate of the outer roadline (longitudinal slope) from the value of the normal cross section (*-2% or -2.5%*) to the maximum value. This is required for the calculation of the necessary length for the cross slope variation. This length is calculated in conjunction with the value of the next field.

The *Superelevation variation on tangents % (on turns without transition curve)* matters only if the clothoid length is zero and determines the percentage of the attenuation length (%) on the tangent with respect to the total attenuation length that is required. The total superelevation attenuation length is the length required for the cross slope variation from the position of the normal cross section to the position of the maximum superelevation.

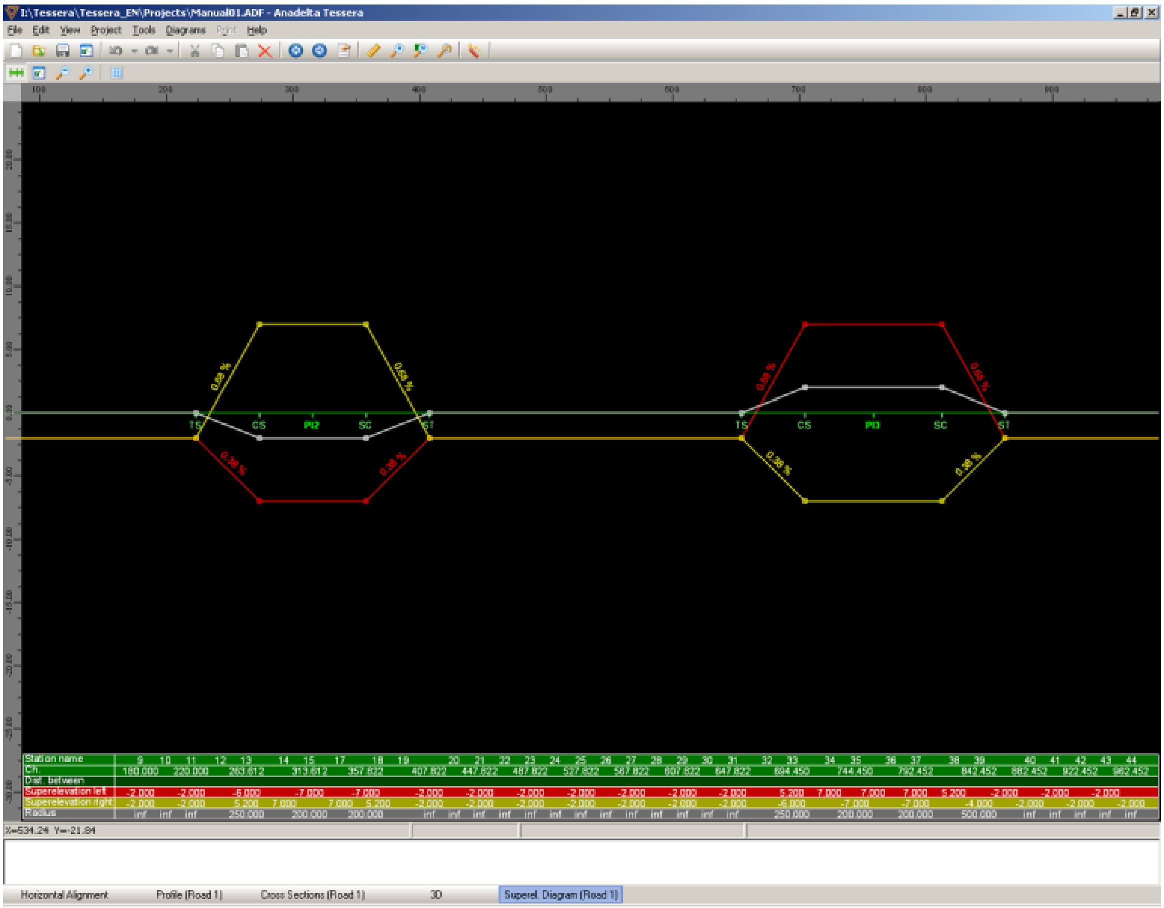
When a PI element that affects the superelevation diagram changes, then, when you click *OK* to close the dialogue box of the *Road Preferences* command, the following confirmation message will appear on screen,



prompting you to confirm the recalculation of the superelevation diagram. If you select *No*, in order for the roadlines to be recalculated, go to the *Road Design* menu, select the *Recalculate* submenu and run the *Superelevations* command.

7.3.10 Displaying the Superelevation Diagram

To view the superelevation diagram for the current road, run the *Superelevation Diagram* command of the *Road Design* menu. The following tab appears:

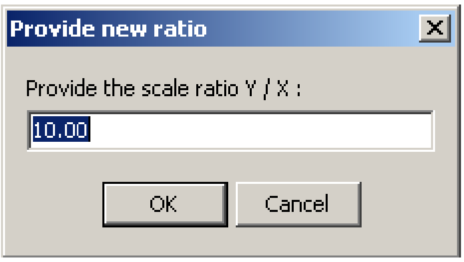


The three lines correspond to the curvature diagram (grey line), the superelevation of the left roadline (red line) and the superelevation of the right roadline (yellow line). Of course, in order that these lines are displayed, the calculation of the road must have already been performed.

As you will see, this tab has a menu entitled *Diagrams* and a toolbar (apart from the main one) with the operations that you can carry out from here. The only purpose of this tab is the graphic presentation of the superelevation diagram of the selected road, and therefore you cannot edit the diagram directly with the mouse, you can only change the preferences.

If you wish to change the values of the diagram, you must either run the *Edit Diagrams* command of the *Diagrams* menu, or change the values of the superelevation diagram settings from the *Diagram* and *Slopes* tab of the *Road Properties* dialogue box, which are displayed if you click the  button.

If you want to increase the horizontal axis scale with respect to the vertical V/X, run the *Stationing* command of the *Diagrams* menu or click the  button or press the *F5* key. If you want to reduce the horizontal axis scale with respect to the vertical V/X, run the *Spacing* command, of the *Diagrams* menu or click the  button or press the *F6* key.

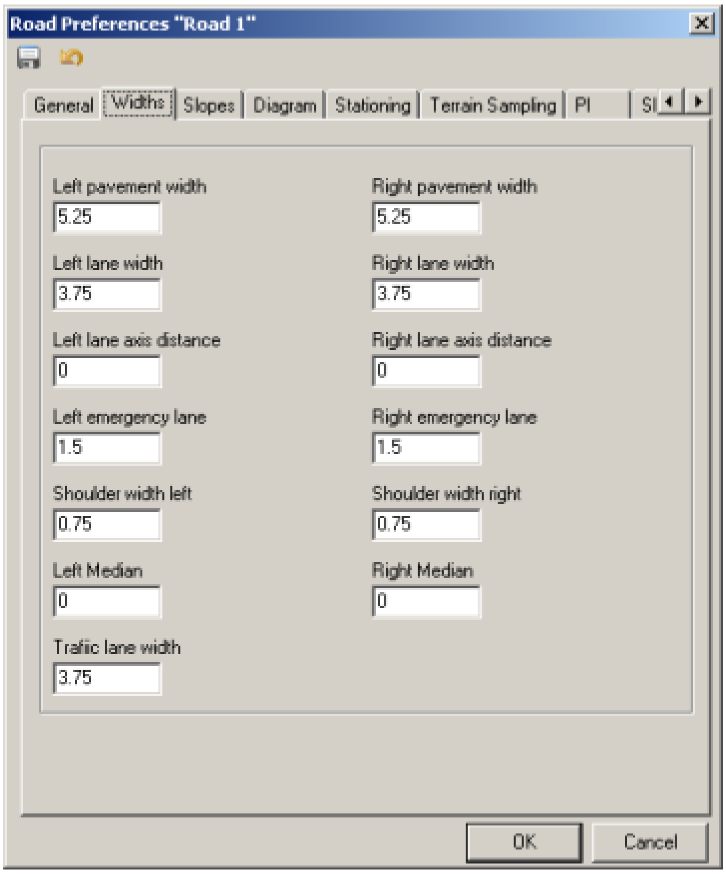


If you want to define the exact ratio of the V/X scales, run the *Scale X/Y* command of the *Diagrams* menu. The dialogue box shown in the figure will appear on screen; type the value of this ratio in the text field of this box.

If you don't want to view the axis and the curvature diagram, click the  button. To view them again, simply click the same button again.

7.3.11 Setting the Widths

You can set the widths of the various elements of a roadway from the dialogue box of the Widths tab, which is displayed if you run the *Road Preferences* command of the *Road Design* menu.

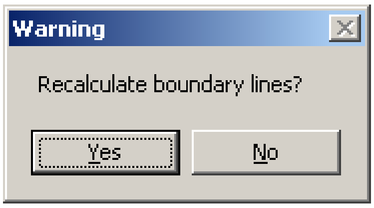


The tab has two groups of text fields. Enter the widths for the left and right part of the road in the text fields of the left and right column, respectively. The widths that you must enter are explained below in the order they appear:

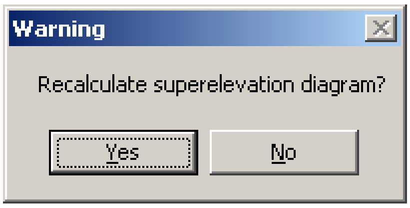
- the pavement width, namely the distance from the road axis to the end of the course layer,
- the lane width (a), which is used in the calculation of the Ds slopes of the superelevation diagram,
- the distance of the semiaxis (auxiliary axis) from the cross section axis (mostly double branch cross sections),
- the emergency lane width,
- the shoulder width, namely the distance between the end of the course and the end

of the shoulder's surface,
- the median width, namely the distance between the road axis and the end of the median.

Based on this data, during the calculation of the road the program will calculate the boundary lines, namely the pavement line, the shoulder line, the lines of the auxiliary axes and the median lines. These widths refer to the beginning of the road and they are fixed and equal to the values entered in the corresponding fields. If you want to enter a variation of the widths between two chainages, you can do this after the road has been calculated, by inserting the appropriate widening type.



If you modify any of the widths after the road has been calculated, click the *OK* button to close the dialogue box of the *Road Preferences* command. A confirmation message will appear on the screen, prompting you to confirm the recalculation of the boundary lines.



If you modify the lane width, a message will appear, prompting you to confirm the recalculation of the superelevation diagram. If you select *No* and you want the boundary lines to be recalculated, select the *Recalculate* submenu from the *Road Design* menu and run the *Roadlines* command.

7.3.12 Calculation of a Road

If the data you have entered for the PIs are not acceptable, the program will display the corresponding error messages, in the message window at the bottom of the screen.

For example, if the trailing clothoid of a PI (PI1) and the leading clothoid of the next PI (PI2) overlap, it is evident that the road cannot be created and the corresponding message is displayed. The program will give you the Tang + Tang? total value and the PI1PI2 distance. The error is that this value is higher than the PI1PI2 distance, therefore the difference of these two values gives the value of the error, namely how many metres are missing. You can solve this problem by modifying the curvature radius or the clothoid length (leading or trailing accordingly). Even if the leading clothoid and the trailing clothoid of a PI overlap, a message will appear, informing you at which PI this overlap occurs. You can deal with this problem either by increasing the PI angle or by decreasing the length of the clothoid (leading or trailing accordingly).

After the calculation of the road, you can edit the PIs, as previously described. When an element of the PI that affects the layout geometry changes, then the relevant part of the curve is recalculated automatically.

If the move is interrupted at a point where calculation is not possible, an error message will appear in the message window at the bottom of the screen, explaining why there is no solution.

The calculation of the road includes the calculation of the axis curve, the calculation of the roadway – pavement roadlines and the calculation of the superelevation diagram. Both during the calculation of the roadlines and during the superelevation diagram, the program takes into account various elements, which are analyzed in separate paragraphs.

UndoRoad Calculation

If you wish, you can undo the road calculation, in order to edit or change its elements. Undoing the calculation does not affect the values you have defined until now for each PI. Simply run the Undo *Road Calculation* command from the *Road Design* menu. This is useful for inserting and editing road elements without direct control by the program.

7.3.13 Widenings

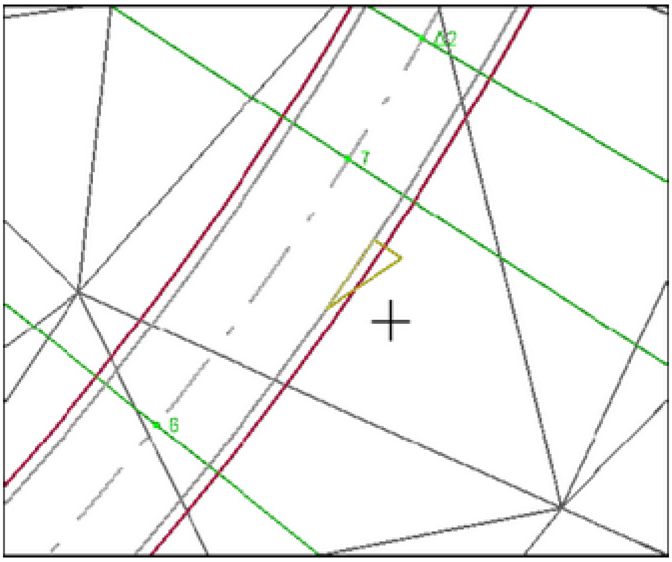
Inserting and Editing Pavement Widenings

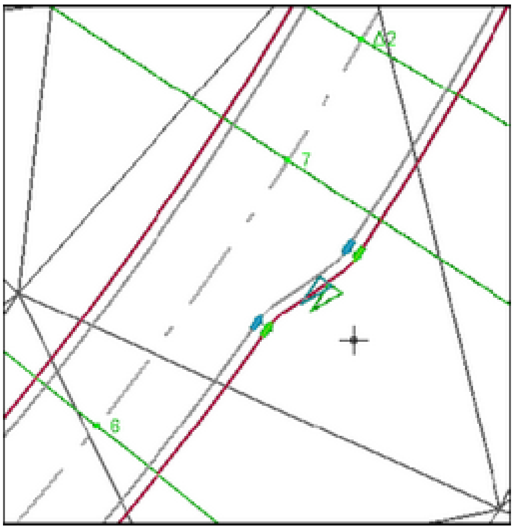
The concept of pavement widening in *Tessera* means to increase or decrease the semiwidth of the pavement on its left or right side. You can use the widenings to configure the roadlines of the pavement. The program creates automatically two types of roadway widenings. The first type refers to the roadway widening for the creation of additional traffic lanes, while the second type refers to the automatic widening of the pavement on curves.

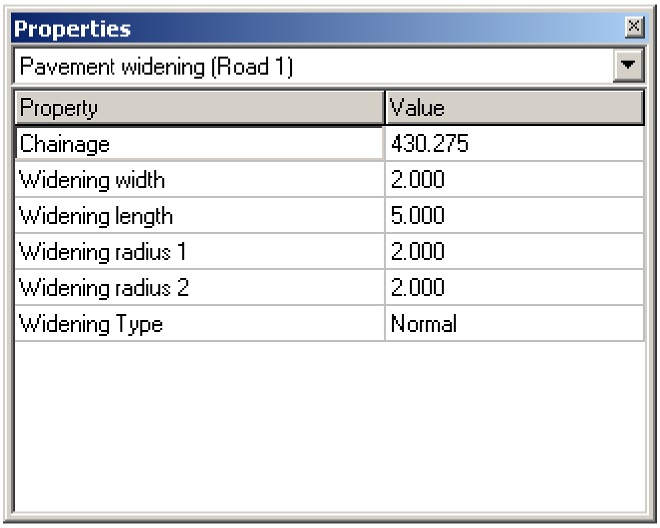
Inserting and Editing a Pavement Widening

Select the desired road where the widening will be inserted and then, from the *Road Design* menu, select *Widenings* and then *Pavement Widening*.

The program creates a movable drawing of the widening with a yellow triangle. The widening is defined by the position of the mouse pointer. With the mouse you can also specify the road side where the widening will take place. Once you have selected the appropriate position, click the left mouse button to insert the widening. Note in the figure below that the new widening configures the roadlines of both the paveway and the pavement.

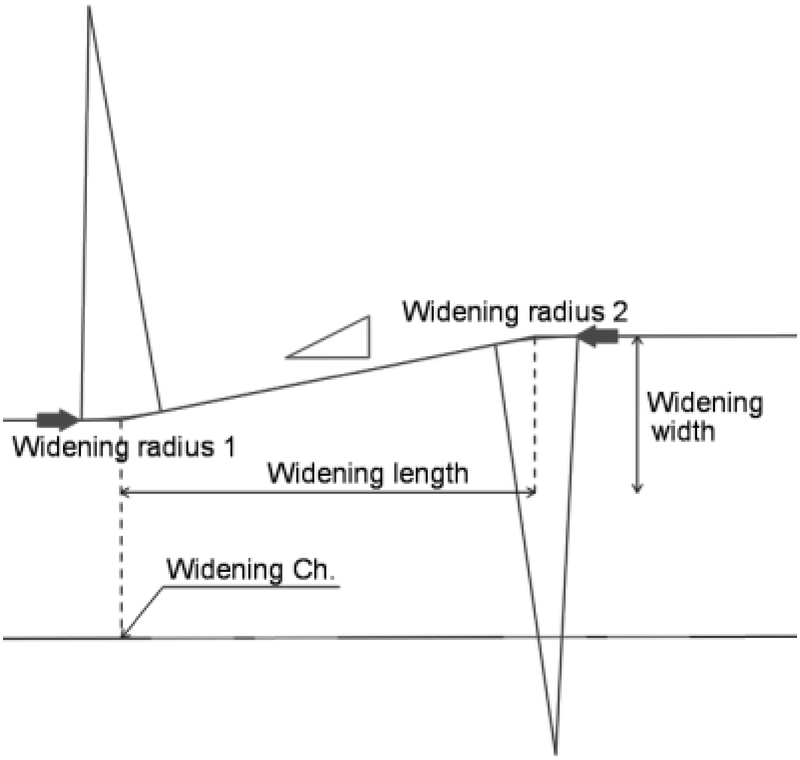






Next, run the *Properties* command, to open the window shown in the figure. Immediately after the insertion, the new widening remains selected. If you have somehow unselected the widening, you can select it again by clicking the mouse with the pointer inside the corresponding auxiliary triangle (the white one).

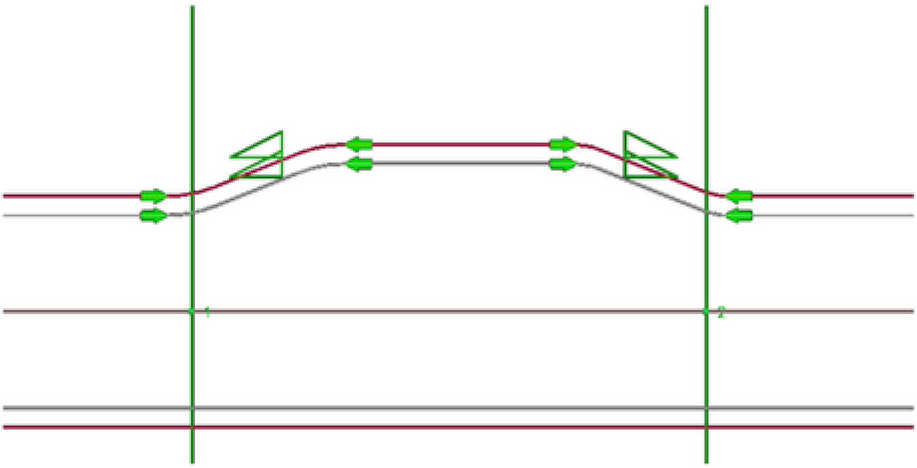
In the first field of the window, edit the value of the chainage where the widening will be inserted. In the second field, enter the widening width, which can be positive for moving away from the road axis, or negative for moving towards the road axis. In the next three fields, enter the widening length, as well as the first and second curvature radii respectively, as shown in the figure.



In the last field, enter the type of widening, which concerns mainly the form of the rounding arcs that are being used.

When you close the window, the program will insert the widening according to the data you have entered, from the point you have specified until the end of the road. Upon insertion of a pavement widening, a respective shoulder modification is also inserted with the same characteristics. Namely, a change in the road's width (widening) also affects (modifies) the shoulder (pavement roadlines), while this does not occur when inserting shoulder widenings, a procedure described in a following paragraph.

Next, you can insert another widening, with negative width value (using the same principle as described above) at another position after the first widening, in order to create a road section like the one shown in the figure:



In this example, the value entered for the first widening is +2 and for the second widening is -2.

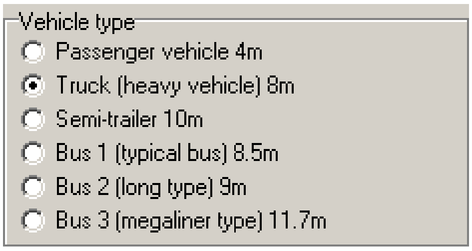
Note that it is possible to insert additional widenings between the aforementioned two ones. If the widening that you are inserting overlaps with an already existing one, or if the values you have entered are not compatible, then the program will inform you about this error and will not take into account the widening when configuring the roadlines, while the arrows indicating the beginning and the end of the widening as well as the corresponding triangle will turn yellow.

The procedure for editing widenings is the same that applies to the editing of all elements (Cross sections, PIs, points, perimeters, etc.). Namely, you must select the desired element and click the right mouse button. To select a widening, left-click inside the triangle of the desired widening and then right-click to display the widening editing menu. Select *Properties* to display the dialogue box and enter the new data. Any editing action on a widening applies also to the respective shoulder.

To delete a widening, select it and then right-click in the drawing area; from the displayed menu select *Delete object "Widening.... (Road…)"*. If the respective shoulder widening is also selected, then in the shortcut menu, the *Delete* command will be displayed and you can use it to delete both widenings.

<u>Inserting and Editing a Pavement Widening on Curve</u>

If you wish to insert a widening on a curve, select the road PI where the widening will be inserted and from the *Road Design* menu select the *Widenings* submenu and then run the *New Widening on Curve* command. The program will calculate the required widening based on the vehicle type that you have defined in the *Road Design* menu → *Road preferences* command → *General tab* → *Vehicle type* group.



The procedure for editing a curve widening is the same that applies to the editing of all elements. Left-click inside the triangle of the widening and then right-click to display the widening editing menu. Select *Properties* and enter the new data.

Inserting and Editing a Shoulder Widening

The concept of shoulder widening in *Tessera* means to increase or decrease the semiwidth of the roadway on its left or right side. Use the shoulder widenings to configure the roadlines of the pavement. As mentioned in the previous paragraph, when inserting a pavement widening, the shoulder widening is inserted automatically. However, it is possible to insert shoulder widenings separately.

To insert the particular shoulder widenings (without changing the pavement width), from the *Road Design* menu select *Widenings* and then run the *New Shoulder Widening* command. The insertion procedure is the same as the one described in the previous paragraph for the pavement widenings.

The procedure followed for editing shoulder widenings is similar to the one for editing widenings, regardless of whether or not the shoulder has resulted from the insertion of a corresponding pavement widening. In case that the shoulder corresponds to (i.e. it has the same starting chainage) a pavement widening, every editing action on the particular widening will also affect the parameters of the shoulder. Therefore, you must edit shoulder widenings only after you have edited the corresponding pavement widenings. When you change the chainage position of a shoulder widening that corresponds to a pavement widening, the shoulder will no longer correspond to this particular widening.

Everything we have mentioned previously for the pavement widenings also applies when you insert or edit a shoulder that overlaps with an adjacent one. To delete the shoulder widening, select *Delete object "Shoulder widening (Road…)"* from the shortcut menu.

Inserting and Editing Medians

The concept of the median in *Tessera* means to create a zone in the middle of the pavement. This zone is defined by two roadlines, the left and the right median line. Note that, during the creation of the road's cross sections, the *Horizontal Alignment* workspace supplies the *Cross Sections* workspace with the diagrams of the left and right medians. These diagrams can be used for determining the inner roadlines of the pavement (in the case of double branch cross section) or/ and for inserting a pavement encasing material based on the diagram (e.g. kerb, N. Jersey).

<u>Inserting a Median</u>

Set the widths for the left and right median in the same way you set the left and right semiwidth of the road, separately for every road of the project. You can set the widths of the median from the dialogue box that is displayed in the *Widths* tab of the *Road Preferences* command of the *Road Design* menu. In all cases, enter the widths of the left and right median in the relevant fields of the dialogue box that is displayed on the screen.
In this way, a median is inserted in the center of the roadway. The width of the median is the same from the beginning to the end of the road. To view the roadlines of the medians, you must have selected the display of the road's boundary lines.

<u>Median Widening</u>

In this paragraph you will see how to modify the medians of a road. To add a median widening, activate the *Road* mode and select the desired road. From the *Road Design* menu, select the *Widenings* submenu and then run the *New Median Widening* command.

The program will make a moveable yellow draft drawing of the median, at the chainage you have specified with the mouse. With the mouse you can also select the side of the road where the widening will take place.

Next, select the widening and run the *Properties* command. In the first field of the window, edit the value of the chainage where the widening will be inserted. In the second field enter the value of the widening. This can be positive for moving away from the road axis or negative for moving towards the road axis. In the next three fields, enter the widening length, the first curvature radius, and the second curvature radius, respectively. In the last field enter the type of the widening, which is mainly the type of the rounding arcs that are being used.

When you close the window, the program will insert the widening according to the data you have entered, and will configure the corresponding median boundary line from the position you have specified until the end of the road.

<u>Editing a Median</u>

The procedure for editing medians is the same that applies to the editing of all elements (Cross sections, PIs, points, perimeters, etc.). Left-click inside the triangle of the desired median and then right-click to display the relevant menu. Select *Properties* and enter the new data.

Deleting a Median Widening

To delete the median widening, click the right mouse button and from the displayed menu select the *Delete object "Median widening (Road…)"* command.

7.3.14 Stationing

When the calculation of the road is completed, stationing can take place. Stationing is the calculation of the position of stations on the road axis. The stations are inserted according to the preferences defined in the *Stationing* tab of the *Road preferences* command of the *Road design* menu. By default, the program places stations at the beginning and at the end of the road, as well as at the beginning and at the end of every curve (stations TS and ST). If you wish, you can insert stations at the other main points of the curves, such as at the beginning and the end of the circular arcs (stations CS and SC), at the center of the circular arc (station D) as well as at the points where the verticals from the center of the circle to the tangent intersect with the road axis (stations CP and PP, approximately in the middle of the transition curves).

The name of the first station is POB, while the name of the last station is POE. The cardinal cross sections of every curve get a composite name composed of one letter (TS, CP, CS, CC, SC, PP, ST) and one number, which is identified with the numbering of the respective PI (e.g. CS1, CP1 for PI 1). The names of the other, non-cardinal cross-sections derive from serial numbering, starting with the number you have assigned.

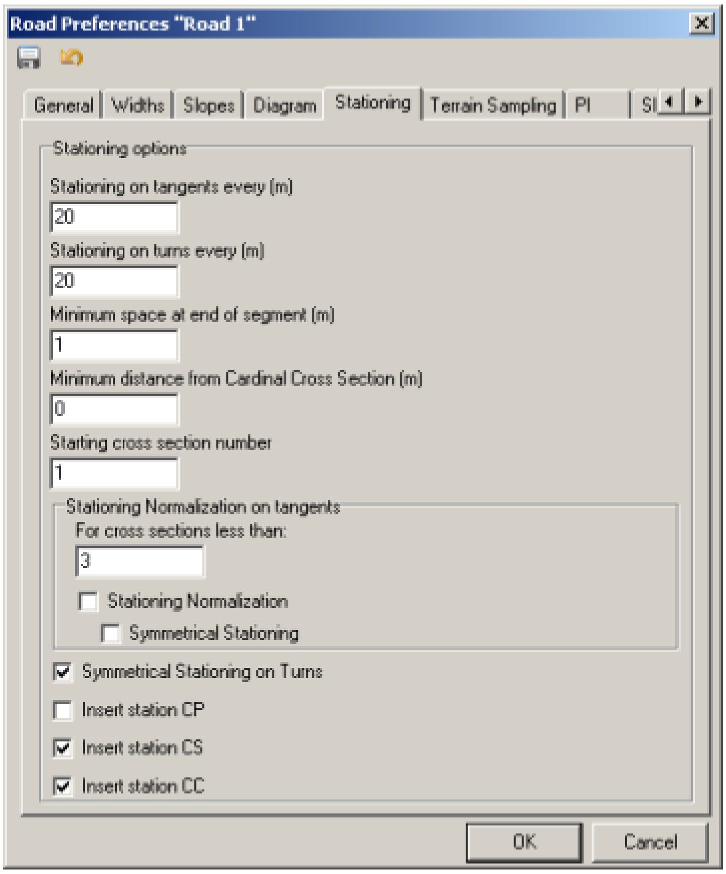
For the tangents, the program starts after the last station of the previous curve (e.g. ST1) and continues to insert stations at the distance that you have predefined in meters (e.g. d=20) up to the last station before the next curve (e.g. TS2). The distance of the last station from the beginning of the next curve will be greater than the *Minimum space* at end of *segment* that you have defined in the *Stationing* tab.

For the curves, the program starts after the first cardinal station (e.g. TS1) and continues to place stations at the distance that you have predefined in meters (e.g. d=10) up to the last station before the middle of the curve (e.g. CC1). The distance of the last station from the middle of the curve will be greater that the *Minimum space* at end of *segment* that you have defined in the *Stationing* tab.

If desired, stations that follow after the middle of the curve can be inserted symmetrically with respect to the previous ones. The same procedure is actually repeated, starting from the end of the curve and towards the middle. To do this, *the Symmetrical stationing on turns* field must be selected. The cardinal stations CP, PP, CS and SC are interpolated at the corresponding positions without affecting the distances between the non-cardinal stations.

Stationing Options for the Current Road

To set the stationing options, go to the *Road Design* menu, run the *Road Preferences* command and select the options in the *Stationing* tab. This command results in the dialogue box shown in the figure. Here you can select the stationing options.



You can enter the distance between the stations on tangents and the corresponding distance on turns in the *Stationing on tangents every (m)* and *Stationing on turns every (m)* fields.

In the *Minimum* space at the end of segment field, you can enter the minimum distance that the last station must have from the end of each segment. A segment is any tangent or turn. If the *Symmetrical stationing on turns* has been selected, a segment is the half of the curve. In the *Min. Distance from Cardinal Cross Section* field enter the minimum distance that a cross section must have in order to be inserted before or after a cardinal cross section.

In the *Stationing normalization on tangents* group, you can optimize the positions of the cross sections on tangents. This applies to tangents having fewer cross sections than the number you have entered in the *For cross sections less than:* field. In this case, the program will insert the last cross section of the Tangent in the middle of the distance between the penultimate cross section and the next cardinal cross section. If you also activate the *Symmetrical stationing* field, then the position of first and last station of the tangent will be inserted at an equal distance from the beginning and the end of the tangent.

In the last four fields you can select, if desired, the symmetrical stationing on the turns and the characteristic points where stations will be inserted, by ticking the relevant check boxes.

To confirm the parameters you have entered, click the *OK* button. If you change the stationing preferences after the program has already inserted stations, when you close the dialogue box, the program will display a confirmation message for the recalculation and the re-insertion of the stations of the selected road.

Automatic Insertion of Stations

From the *Road Design* menu and the *Stationing* submenu, run the *Stationing* command or, on the *Horizontal Alignment* toolbar click the button, or press the *<Alt>*+*<U>* keys, to proceed to the calculation and insertion of stations according to the aforementioned options.

Inserting a New Station

To insert a station, from the *Road Design* menu, run the *New Station* command or, on the *Elements Drawing* toolbar click the button. You will notice a green line, which is vertical to the axis of the selected road. Every time you use the mouse, this line moves to the position where the mouse pointer is. Select the desired position and insert the station by clicking the left mouse button. To specify the exact chainage and the name of the inserted station, select the station (by left-clicking on it) and run the *Properties* command, to display the relevant dialogue box. Repeat the process for all stations you wish to insert.

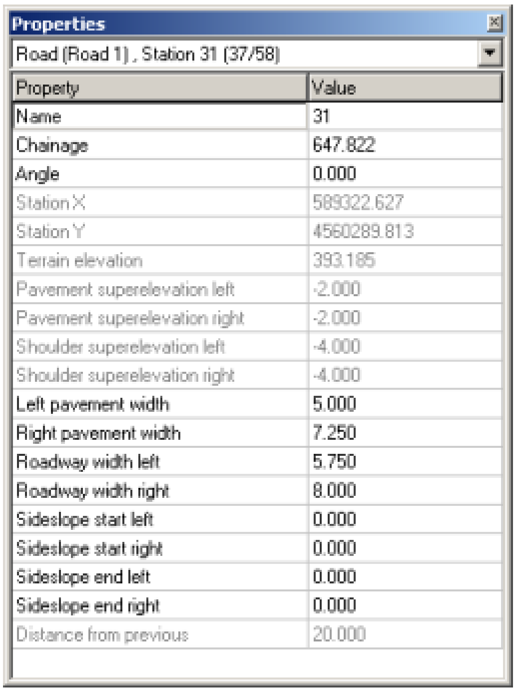
Deleting a Station

To delete a station, select the station you wish to delete, click the right mouse button and select the appropriate command (Delete cross section) from the displayed menu.

You can also delete all stations of a road. To do this, select the road and from the *Stationing* submenu of the *Road Design* menu run the *Delete stations* command. You can also carry out the same operation if you select the *Delete* submenu from the *File* menu and then run the *Stations* command.

Editing Stations

To edit a station, select with the mouse the road where the station is and then select the station.
You can move between stations with the <Ctrl>+< → > keys for the next station and the <Ctrl>+< ← > for the previous station. Right click on the screen to display the shortcut menu. From there, run the Properties command to display the dialogue box shown in the figure:



Properties

Road (Road 1) , Station 31 (37/58)

Property	Value
Name	31
Chainage	647.822
Angle	0.000
Station X	589322.627
Station Y	4560289.813
Terrain elevation	393.185
Pavement superelevation left	-2.000
Pavement superelevation right	-2.000
Shoulder superelevation left	-4.000
Shoulder superelevation right	-4.000
Left pavement width	5.000
Right pavement width	7.250
Roadway width left	5.750
Roadway width right	8.000
Sideslope start left	0.000
Sideslope start right	0.000
Sideslope end left	0.000
Sideslope end right	0.000
Distance from previous	20.000

The first three fields are the name of the station, its chainage and its angle with respect to the vertical to the road axis.

If you change the *Name* of a non-cardinal station, the program will prompt you to confirm the name (code) changes for all of the next non-cardinal stations. If you change the *Chainage* of the station, the station moves, while at the same time the terrain elevations and the values mentioned below are updated. If you change the Angle, you can rotate the level of the station with respect to the vertical to the road axis.

The *Station X, Station Y, Terrain elevation* and *Road profile elevation* fields are only for information purposes and their values are updated automatically after each change of the station chainage. The Terrain elevation field is updated only when the *Triangulation* has already been performed. The *Road profile elevation* field is updated only when the road profile has already been calculated in the *Profile* and *Update road profile* from *Profile* to *Horizontal Alignment.*

The *Pavement superelevation left, Pavement superelevation right, Shoulder superelevation left* and *Shoulder superelevation right* fields are also for information purposes and their values are automatically updated every time you change the superelevation diagram.

The next four fields refer to the pavement and roadway left and right semiwidths. These fields are updated automatically by the program every time you insert, edit or delete a pavement or shoulder widening, and every time the road is calculated. This automated process is referred to as initial update.
They are also updated every time the Roadlines are updated from *Cross Sections* to *Horizontal Alignment,* in order to reflect the real widths, as they have been calculated in the *Cross sections* after applying the *Typical cross section* or after a change in the coordinates has taken place at Cross section level. It is also possible for the user to change their values directly. However, any future change in the widenings will restore the initial update of the widths. You can achieve the same result by runing the *Recalculate Roadlines* command from the *Road Calculations* submenu of the *Road design* menu. Any update or change of the widths has a direct effect on the layout of

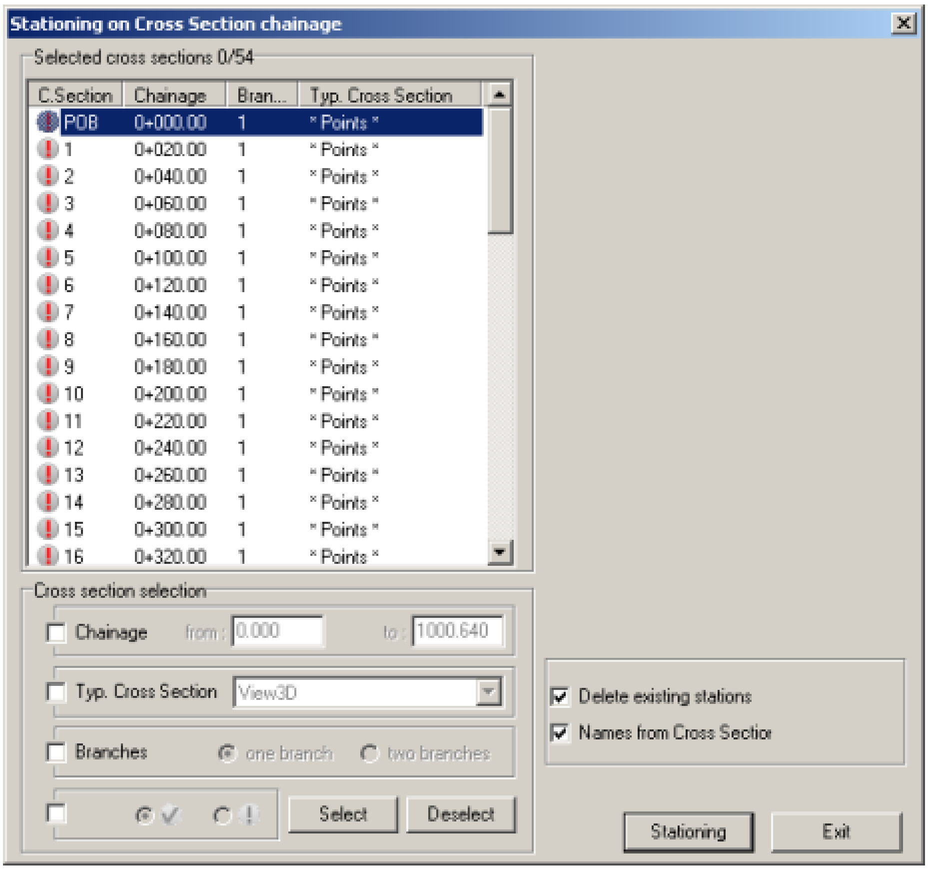
the roadlines, the pavement and the roadway.

The *Sideslope start left*, *Sideslope start right*, *Sideslope end left* and *Sideslope end right* fields are updated automatically by the program when the *Horizontal Alignment* is updated from the *Cross Sections* workspace. If the value of any of these fields changes, then the corresponding roadline also changes automatically.

Finally, at the bottom of the dialogue box you can see the distance of the current station from the previous one.

Stationing on Cross Section Chainage

In case you have a complete project (Horizontal Alignment, Profile, Cross Sections) and you wish to change the horizontal alignment and to maintain the already existing stations, go to the *Road Design* menu and from the *Cross Section Stationing* submenu run the *Stationing on Cross Section Chainage* command. This command results in the following window,



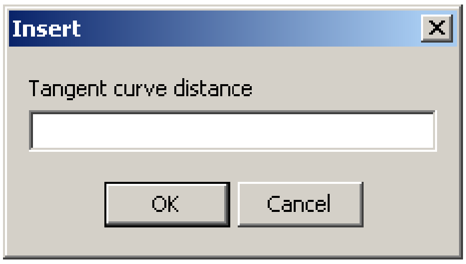
where you can select the cross sections you want to transfer to Horizontal Alignment. Moreover, if the *Delete existing stations* and *Names from Cross Sections* check boxes are ticked, you can delete the older stations of the layout and transfer the names of the cross sections to the Horizontal Alignment, so that the two workspaces are synchronized. Once you have made the desired settings, click the*Stationing* button to perform the road design stationing.

Cross Section Terrain

After the stationing, you can see what the terrain will be like in every cross section without having to update the Cross Sections workspace. To do this, select the *Cross section window* command from the *Road Design* menu. A window will appear on the screen where the terrain of the selected cross section will be drawn. To move to the next or previous cross section, you must click the relevant button at the top of the window.

7.3.15 Creating Service Roads

The *Professional* version features the automatic creation of the axes of service roads parallel to the main artery. To use this command, select the road where the main artery is. From the *Road Design* menu, run the *New Road by Copying Tangent Curve* command. The following dialogue box appears on screen:

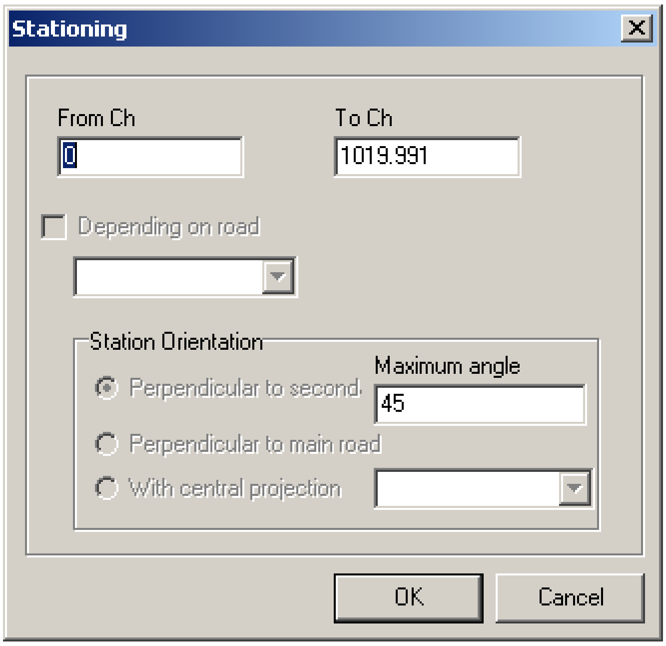


In the *Tangent curve distance* field, type the distance of the service road from the axis of the artery. The value of the distance is positive if you want the new road to be inserted to the right of the artery and negative if the road is inserted to the left. When you click *OK*, the program will create the axis of the new road in parallel to the artery. The radius of each PI of the new road will be decreased or increased according to the distance of the two roads.

Stationing Based on another Road

The calculation and insertion of stations (e.g. road 2) can be performed by taking into account the stations of another road (e.g. road 1). The positions of the stations of road 2 will result from the intersection of the extension of the main road stations with the axis of road 2. Also, the names of the stations of road 2 will be the same as the names of the stations of road 1.

This operation is performed from the *Road Design* menu, where you must select the *Stationing* submenu and then run the *Stationing (Partial/Dependent)* command. In the resulting dialogue box,



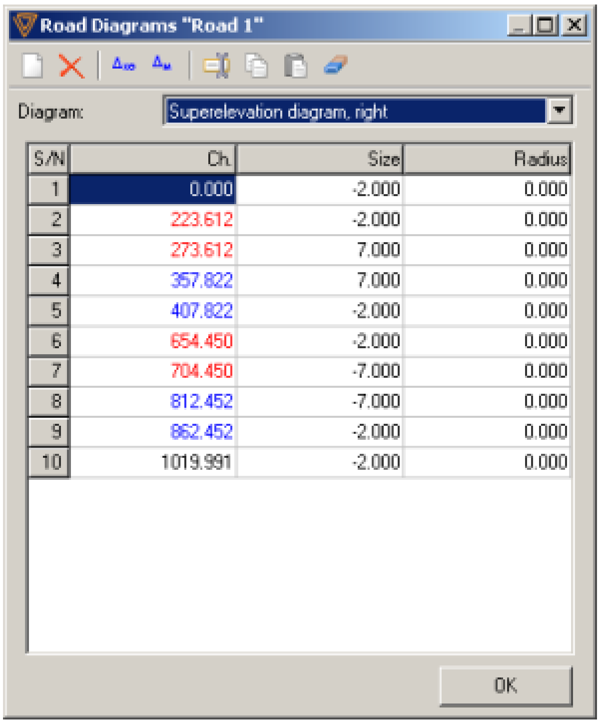
type the chainage of the section where partial stationing will take place in the *From Ch. and To Ch.* fields. For the stationing to take place as described above, tick the *Depending on road* check box and from the drop-down list, select the road based on which the stationing will be performed. In the case of service roads, in the *Station Orientation* group, tick the *Perpendicular to main road* option. Finally, press *OK* for the stationing to take place.

For more information regarding the composite cross sections and the partial/dependent stationing, refer to the chapter Composite Cross Sections - Horizontal Alignment.

7.3.16 Editing Diagrams

During the calculation of the road, apart from the calculation of the axis curve, the program calculates automatically the diagrams of the road, based on the data you have entered and the settings you have made.

You can view, edit and export these diagrams in text file format or you can load diagrams from text files. To view the diagrams, run the *Edit diagrams* command from the *Road Design* menu. When you run this command and a road has been selected, the following dialogue box appears on screen:

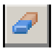


Everything described in this paragraph applies to all three workspaces: *Horizontal Alignment, Profile* and *Cross sections*.

From the *Diagram* drop-down list, select the desired diagram of the selected road. The diagram is displayed in the area right below, consisting of a table with three columns and variable number of lines. When importing – editing a diagram, you must keep in mind that you are actually importing or editing the breaking points (PIs) of the diagram. Each line corresponds to a breaking point (both the end and the beginning). The titles of the columns are shown in gray in the first line. Their names are *S/N, Ch., Size* and *Radius.* The program fills in the first column automatically with the serial number of the breaking point; it is displayed in gray and the user cannot intervene. In the second column you must always enter the *Chainage,* in the third column you must enter the value of size represented by the diagram, and in the fourth column you must enter the curvature radius for the particular PI of the diagram (only for diagrams of pavement, roadway and profile width).

If you click the  button, the diagram will be deleted. The program will display a warning message, prompting you to confirm the deletion. If you want to delete the diagram, click *Yes*.

To change the name of a diagram, click the  button. Enter the new name of the diagram in the corresponding text field of the dialogue box and click the *OK* button.

To delete all data, click the  button. The program will delete all data of the selected diagram.

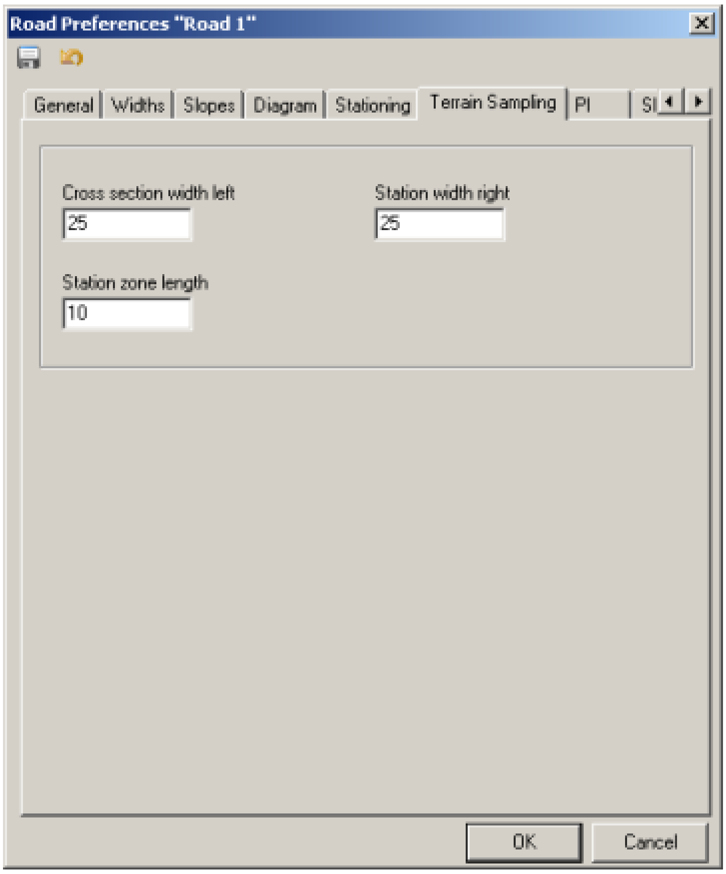
If you click the  button, a new diagram will be created. This button displays a drop-down list with the available types of diagrams. Select the type of the diagram you wish to create. The creation feature is only in the *Cross Sections* workspace and not in the *Horizontal Alignment.* If you select *New generic use diagram,* the program will display a dialogue box where you must enter the name of the new diagram in the relevant field.

After the diagram has been created, you can enter the data in the relevant fields. To select a field, left-click on it and then start typing. Press the *Enter* key to confirm the data you have entered and to move to the next field on the right. You can also move between the fields by using the up, down, right and left arrow keys. If you are at the end of a line, you can press the *Enter* key to move to the first field of the next line.

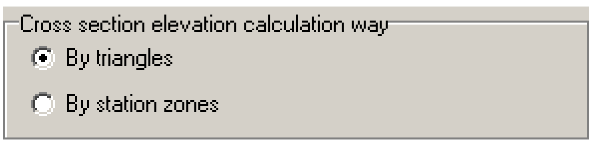
To insert a new line at the end of the table, simply select the cell that corresponds to the last column and line and press *Enter*. If you want to insert a new line between two existing lines, select the immediately next line and press the *Ctrl+I* keys. The new line will be inserted right above the selected one. If you wish to delete a line, simply select the line and press the *Ctrl+D* keys.

7.3.17 Terrain Sampling

To define the terrain sampling width for the cross sections of road, go to the *Road Design* menu, run the *Road Preferences* command and open the *Terrain Sampling* tab.



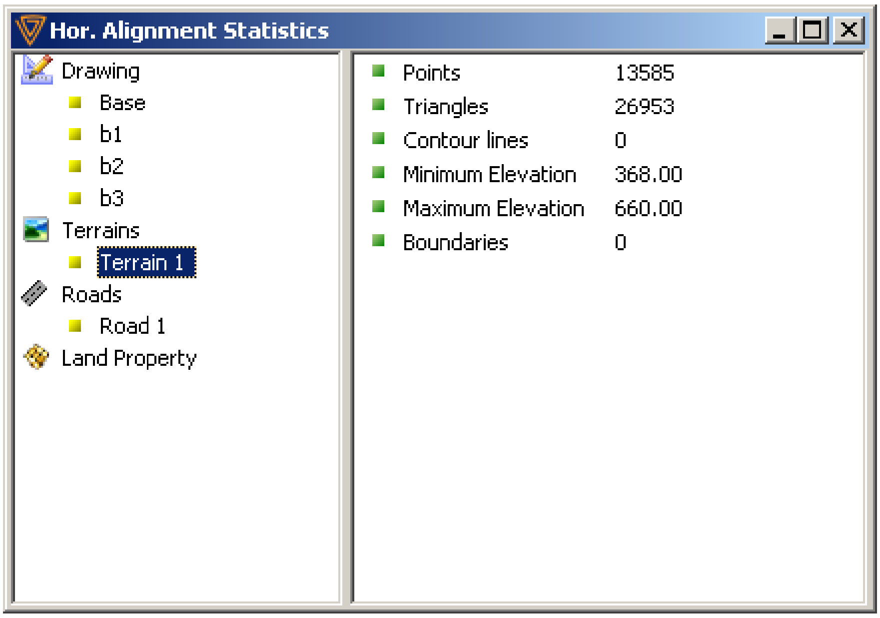
In the *Cross section width left* and *right* fields, enter the desired terrain width of every cross section.
You can use the *Station zone length* field to set the zone length when you don't have a terrain model but only points above or close to the cross sections level. To change the terrain sampling way, go to *Project* menu → *General Preferences* command → *Calculations* tab → *Cross section elevations calculation way* group → *By station zones*.



In this case, the program creates a natural terrain, displaying on the cross section axis, the points contained in a zone around every station.

7.3.18 Horizontal Alignment Statistics

If you select the *Horizontal Alignment Statistics* command from the *Tools* menu, the following window will appear on the screen:



Here you can view all elements and data used in each mode of the horizontal alignment. More specifically, you can view:

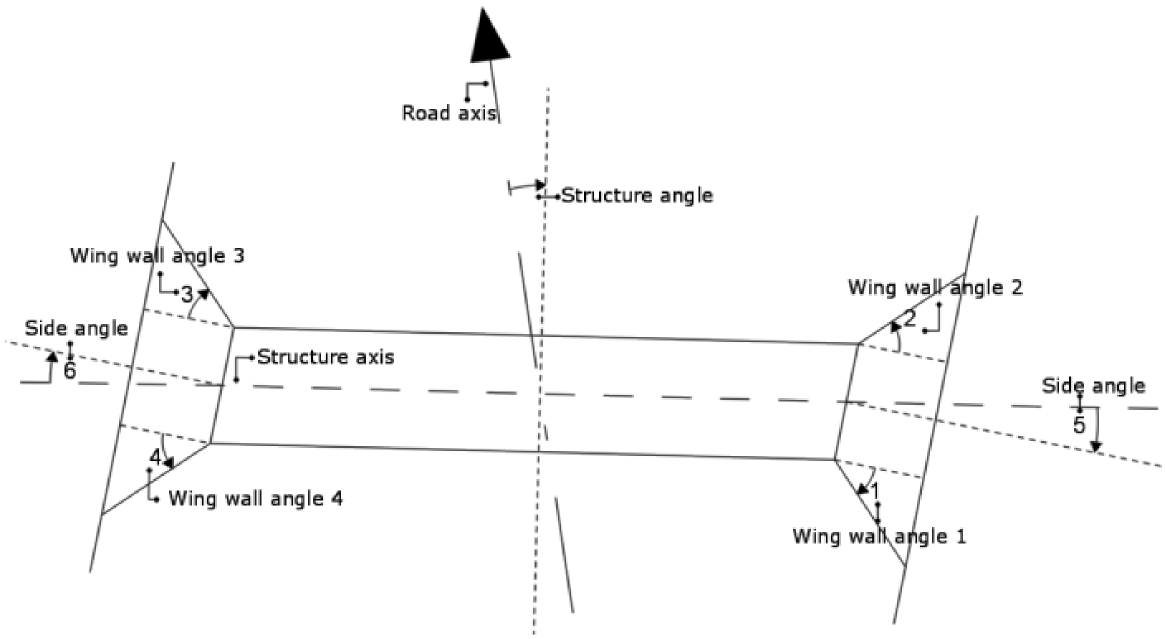
- the number of elements existing in the drawing.
- the terrains with the points, boundaries, triangles, etc.
- the roads with their elements (PIs, cross sections, etc.).

7.3.19 Structures

Overview

Tessera makes the calculation and management of the structures a simple procedure. It is better to insert the *Structures* from the *Horizontal Alignment* after having completed the *Road Design* at all three levels (Horizontal Alignment, Profile, Cross sections). Then the program will automatically insert the *Structures* and will configure their faces and sideslopes. You can also insert a *Structure* at the *Cross section* level before the design is completed, as you can also insert them directly from the *Profile.* In this case, the calculation is performed gradually. In all cases, the calculation is an easy procedure, with successive corrections and repetitions. The program can accommodate a number of values, enabling the complete parameterization of the structures.
The following drawing shows all angles that are required for the implementation of a structure, namely:

- *Angles of 1st, 2nd, 3rd and 4th wing wall*: the angles between the wing walls and the verticals to the sides of the structure.
- *Side Angle*: the angle between the vertical to the side of the structure and its axis.
- *Structure Angle*: this is determined by the vertical to the structure axis and the road axis.



Inserting a new structure

To insert a new structure, select *Structures* from the *Road Design* menu and then run the *New Structure* command. You will notice a blue rectangle, which is vertical to the axis of the selected road and follows the mouse pointer. Select the appropriate chainage and then insert the structure by clicking the left mouse button. To specify the exact chainage and the name of the selected structure, run the *Properties* command from the shortcut menu. Repeat the procedure for all the structures you wish to insert.

Deleting a structure

To delete a structure, select the desired structure, click the right mouse button to display the shortcut menu and run the *Delete object "structure"* command.

You can also delete multiple structures of a drawing. To do this, select the road and from the *File* menu select the *Delete* command and then *Structures*.

Resetting a structure

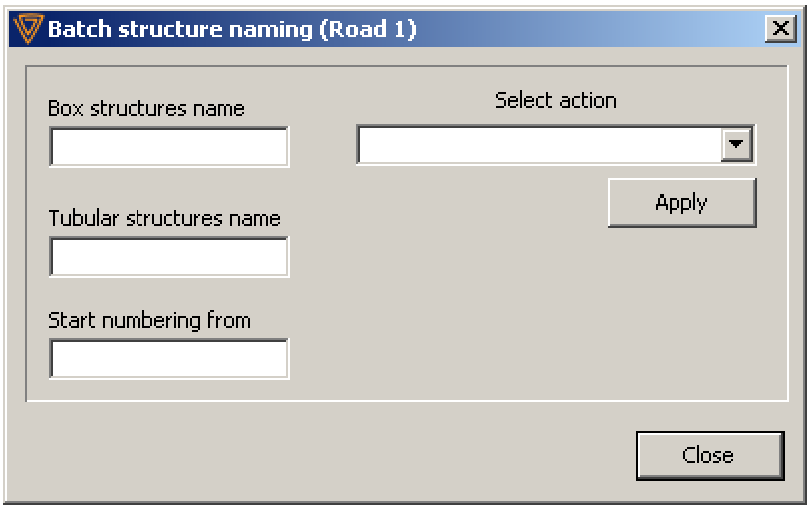
This operation configures the cross section of the structure, so that the left and right ends of the bottom slab of the structure pass through the intersection points of the road's sideslopes with the natural terrain. Namely, it is the automatic calculation of the structure that leads to the cancelation of any changes (elevations and/or slope) affecting the placement of the structure. To reset a structure, select *Structures* from the *Road Design* menu and then select *Reset structure*; or, from the shortcut menu select *Reset structure*. In case you have more than one structures and you wish to reset them all, select *Structures* from the *Road Design* menu and then select *Reset all.* You can also *Reset* the structures from the *Structure cross section* window of the shortcut menu.
Every time the Chainage or the angle of a structure changes, the structure is reset automatically.

Adjusting sideslopes to structures

To run this command, go to the *Road Design* menu, select *Structures* and then select *Adapt sideslopes to structures.* If, for any reason, the sideslopes cannot be adjusted according to the structure, you can achieve that by running this particular command.

Batch Structure Naming

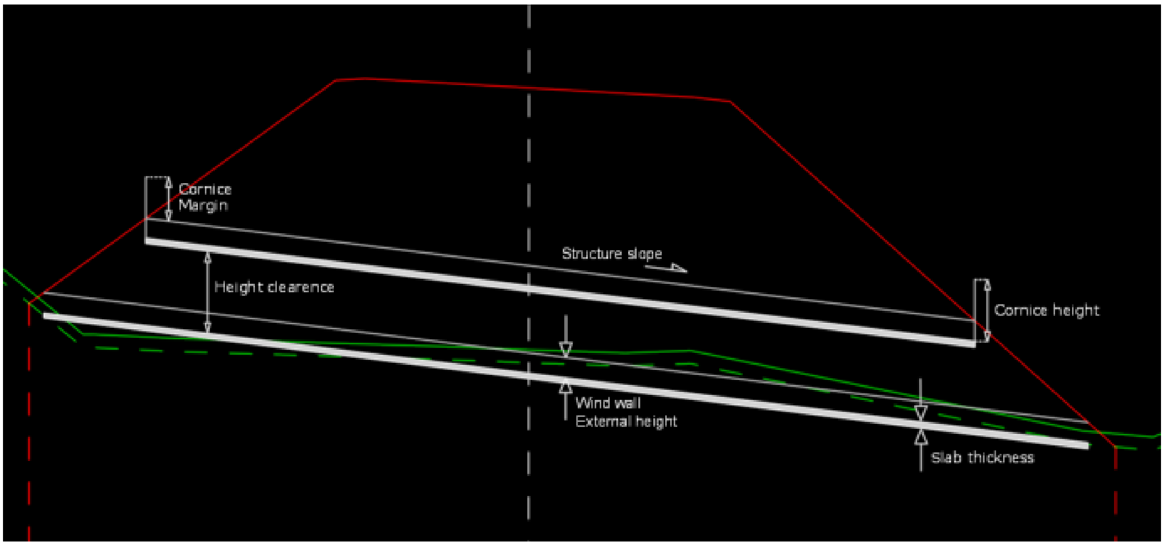
Run this command to name simultaneously all structures of your project, regardless of their type (Box or Tubular structure). To do this, select *Structures* from the *Road Design* menu and then select *Batch Structure Naming*. The following window will appear on the screen:



In the *Box structures name* and *Tubular structures name* fields, type the name you wish to assign to all structures. In the *Start numbering from* field, type the first number for the numbering of the structures. In the *Select action* field, select one of the available actions, which refer either to the naming and the numbering of the structures separately for each type, or to the naming and the numbering regardless of the structure type.
Click the *Apply* button to apply the settings you have made, or click the *Close* button to close the window and return to the horizontal alignment workspace.

Structure cross section

In the window that is displayed when you run the *Structure cross section* command from the shortcut menu, you can see a virtual cross section to which the structure you have inserted in the horizontal alignment is adjusted. In the following figure you can see some of the values of the structure cross section that you can edit.

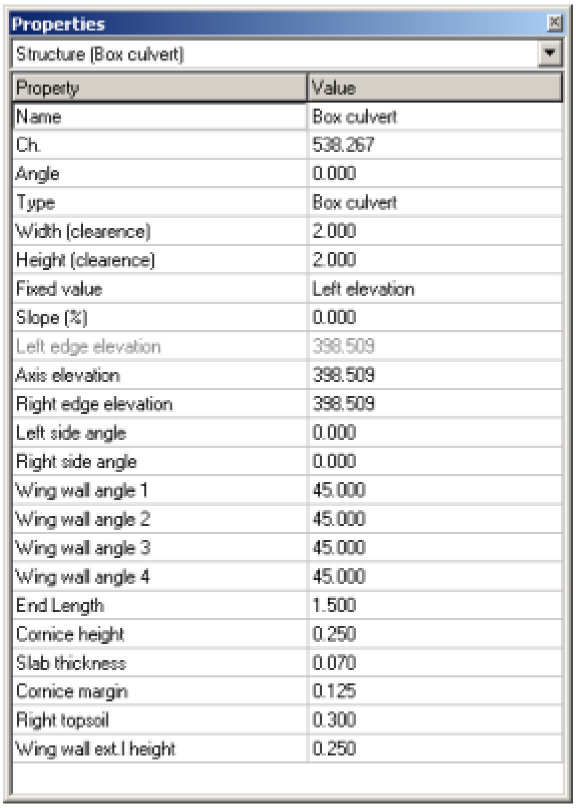


The other values that you can edit are the following:

Fixed value: Select which of the available options will become the fixed value. The fixed value can be the left or the right elevation, the axis elevation and the slope. The field of the element that gives the fixed value becomes inactive. Then, any change in one of the other three fields leads automatically to the change of the remaining two fields.
Slope: Define the desired slope of the structure.
Left and right edge elevation: In these fields you can change the elevations of the left and the right edge of the structure, respectively.
Axis elevation: In this field you can change the elevation that the structure will have, with respect to the axis.

Structure Properties

After the insertion of a structure, you can define the properties of this structure. Select the structure, right-click to display the menu and run the *Properties* command. The following window will appear on the screen:



The settings you can make are the following:

Name: Here you can change the name of the structure.
Ch.: Specify the exact position of the structure.
Angle: In this field you can set the desired angle of the vertical to the structure's axis with respect to the road's axis.
Type: From the drop-down list of this field you can select between the two available types of structures (Box and Tubular culvert).
Width (clearance): Set the width.
Height (clearance): Set the desired height of the structure.
Left and right side angle: Rotate the structure's entrance and exit.
Wing wall angle 1, 2, 3, 4: In these fields you can specify the angle of the wing walls.
End length: On the right and left of each wing wall there is an extra wall, parallel to the construction roadline that is represented by a line at the end of the sideslope. In this field you can specify the length of this wall.
Left and right grate width: Specify the desired grate width on the left and the right side of the structure. This option is displayed in the properties window when one or both sides of the structure are in a cutting.

All other properties of the structures have already been described in the *Structure Cross Section* paragraph.

Update structures

To update the *Profile* with the structures you have created in the *Horizontal Alignment,* select the *Management* command from the *Project* menu or click the relevant icon. Next, in the *Update* group, tick the *Update structures* check box to update both the Chainage and the elevation of the structure. If you go to the Profile workspace, you will notice that there are three symbols referring to the structure; the middle one shows the elevation on the axis while the other two symbols show the elevations of the left and right edge.
If you are not satisfied with the result, you can edit certain properties of the structure

in the Profile mode. To view the properties of the structure, select the structure by clicking the left mouse button, and then run the Properties command from the shortcut menu. To apply the changes you have made in the Horizontal alignment, select *Update structures* from *Profile* to *Horizontal Alignment*.

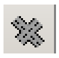
Exporting Structures Data to an ASCII file:

1. From the *File* menu, select the *Export to ASCII* submenu and run the *Structures Data* command.
2. Specify the path and the name for the file you wish to save. The structure data will be exported to a file with the .CSV extension.
3. Click *OK* to export the data.

To export the data you must go to the Horizontal Alignment workspace.

7.3.20 Intersections

Creating an Intersection

To insert an intersection, select the two roads where the intersection will be created (for multiple selection use the ctrl key) and from the menu that is displayed if you click the right mouse button, select the *New intersection* command. Alternatively, from the *Elements Drawing toolbar*, click the  button. In the case of a crossing, if you select the first method for creating an intersection, the first selected road will be the main road, while if you select the second method, the main road will be the one with the widest roadway. If both roads have the same roadway width, then the main road will be the one with the widest pavement.
A rhombus or a triangle will appear on the screen, at the point where the two roads intersect; this is the symbol of the intersection and you can also use it to select the intersection. You can also view the configuration of the roadlines and medians of the intersection that have been calculated according to the predefined *Intersection Preferences (Default)* of the program.

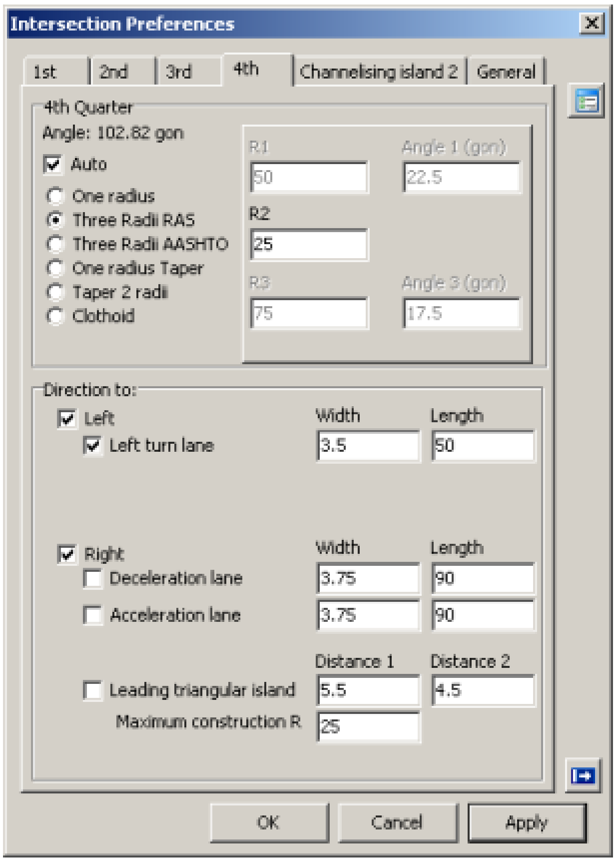
Intersection Preferences (Default)

By default, the preferences for the intersections are defined by using the *Intersection Preferences (Default)* command of the *Intersections* submenu in the *Road Design* menu. A dialogue box with four tabs will appear on the screen. In the *Quarter tab*, you can set the preferences for the transition of an intersection's quarters (radii, offsets, etc.) and for the configuration of the necessary traffic lanes. In the *Large Channelising Island* and *Small Channelising Island* tabs, enter the settings for the construction of the channelising islands. Finally, in the *General* tab, enter the preferences for the median and the widenings of the main road.

Intersection preferences

An intersection can have the shape of a crossing (cross) or a T shape. Apart from the relevant configuration, the shape is symbolized by the corresponding selection symbol, which can be a rhombus or a triangle respectively. These symbols appear in yellow for all intersections, except for the symbol of the selected intersection that appears in blue. When you select an intersection, you also select the current quarter, which appears in white. A direction arrow of the same color corresponds to the current quarter.
In case that the intersection has the shape of a crossing, all its quarters are active, while when the intersection has a T shape, only two of its quarters are active. The inactive quarters have only the *Move to:* group.
To define the preferences of an intersection, select the intersection by clicking its symbol and from the menu that is displayed if you click the right mouse button, select

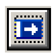
the *Intersection preferences* command. The following window will appear on the screen:

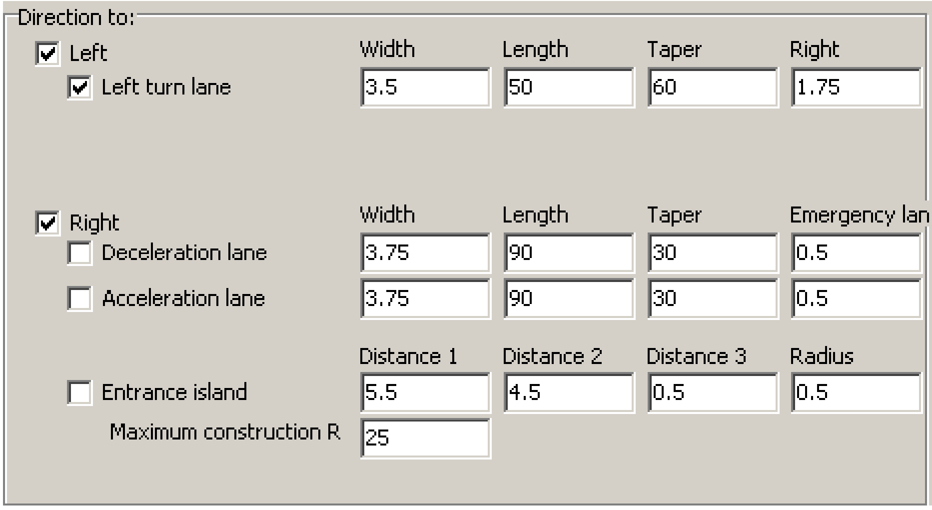


The Intersection preferences window has seven tabs. The first four tabs refer to the settings of the quarters, the next two tabs refer to the channelising islands and the last tab features the general settings of the intersection. In this window, where the 4 th quarter is the current one, you can define the preferences for the type of the quarter transition (radii) as well as some other parameters referring to traffic (extra lanes).

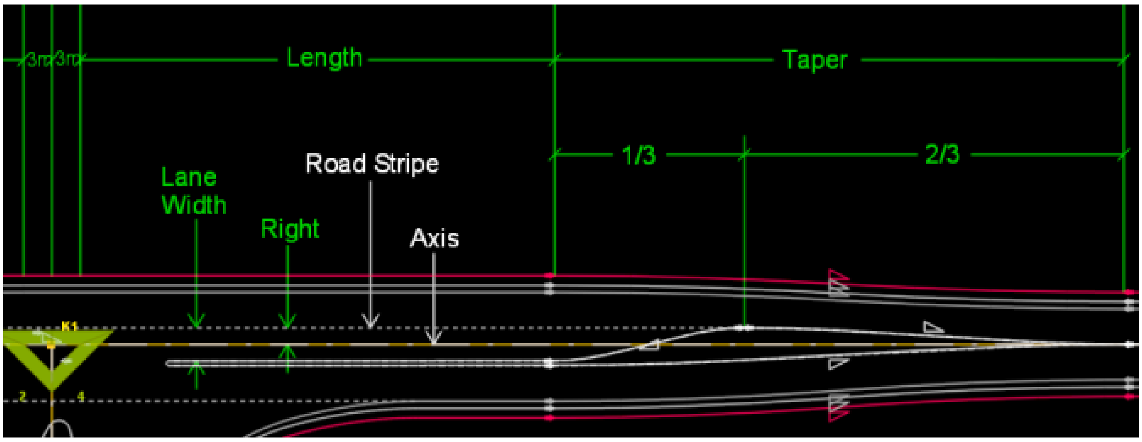
In the 4^{th} *Quarter* group, you can view the angle formed by the two axes (in degrees) and select one of the six available transition ways:

- One radius: only one circular arc
- Three radii RAS: three circular arches with 2:1:3 radius ratios according to the RAS-K-1 regulations. When the radius of the middle circular arc changes, the other two radii change automatically, provided that the Auto check box has been ticked.
- Three radii AASHTO: three circular arcs with leading and trailing offsets on both sides, according to the AASHTO regulations.
- One radius Taper: a circular arc between 2 Tapers. The leading and trailing Tapers are characterized by offsets and L:T ratios (Longitudinal to Transverse, length to offset), according to the AASHTO regulations.
- Taper 2 radii: two circular arcs after a Taper, according to the RAS-K-1 regulations, when there is no deceleration lane. The Taper is characterized by an offset and a L: T ratio and is always formed on the side of the superior road.
- Clothoid: a circular arc between two clothoids, a leading and a trailing one, with the relevant parameters.

In the Direction to: group you can define the desired movements of vehicles, as well as whether there will be a left turn, deceleration and acceleration lanes and a triangular island. This group has further settings, that you can view by clicking the  button as shown below:

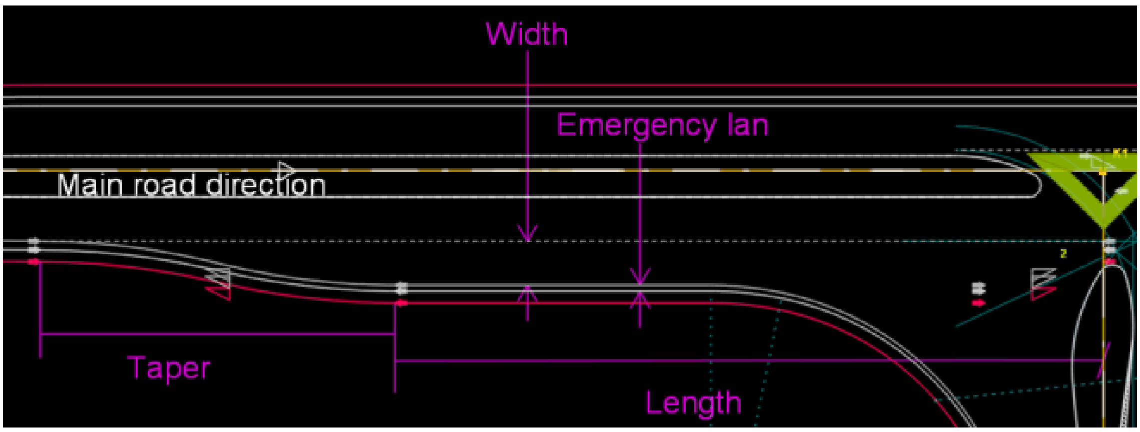


The *Left turn lane* data are shown in the following figure:



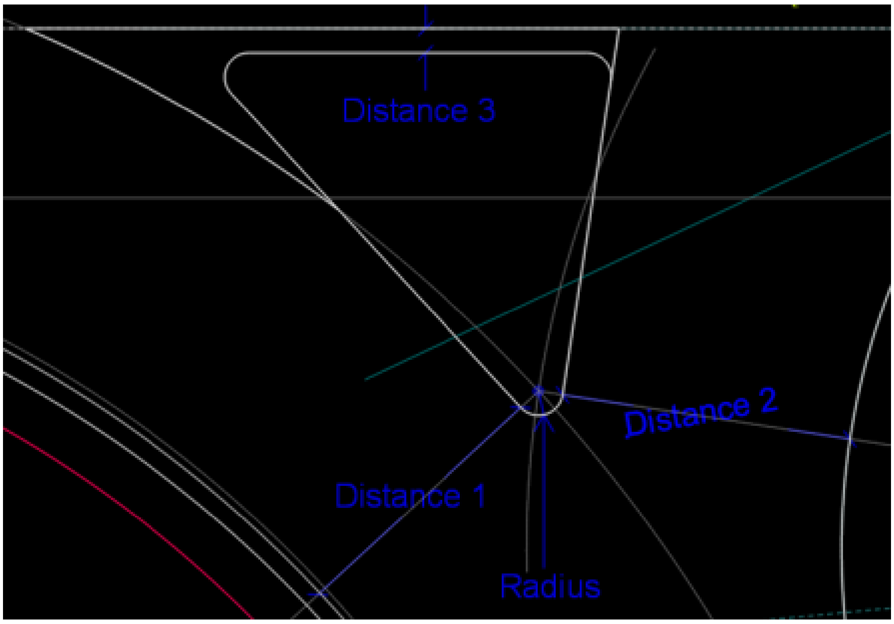
In the *Right* field, enter the width of the section of the left turn lane that will be on the right of the road axis, according to the direction of the lane. The minimum value that this field can accept is 0 and the maximum value is the sum of the width of the left turn lane and the minimum width of the median or the hatch. In this way, the widenings of both sides of the main road that are required for the configuration of the left turn lane with the median or hatch are set indirectly (*General* tab next).

The data of the *Deceleration lane* are shown in the following figure:



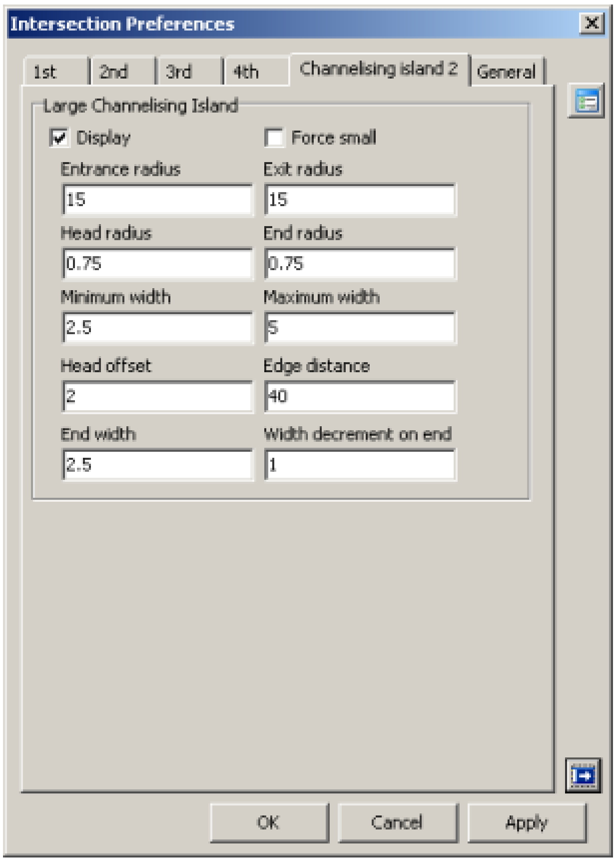
In the *Emergency lane width* field, enter the distance of the marginal strip from the edge of the paveway. This width is applied to the whole length of the deceleration lane and can differ from the width of the main road's *Emergency lane.* The update takes place automatically in the transition section by using a different width for the guidance strip with respect to the width of the corresponding paveway widening.

The data you define for the *Triangular Island* are shown in the following figure:



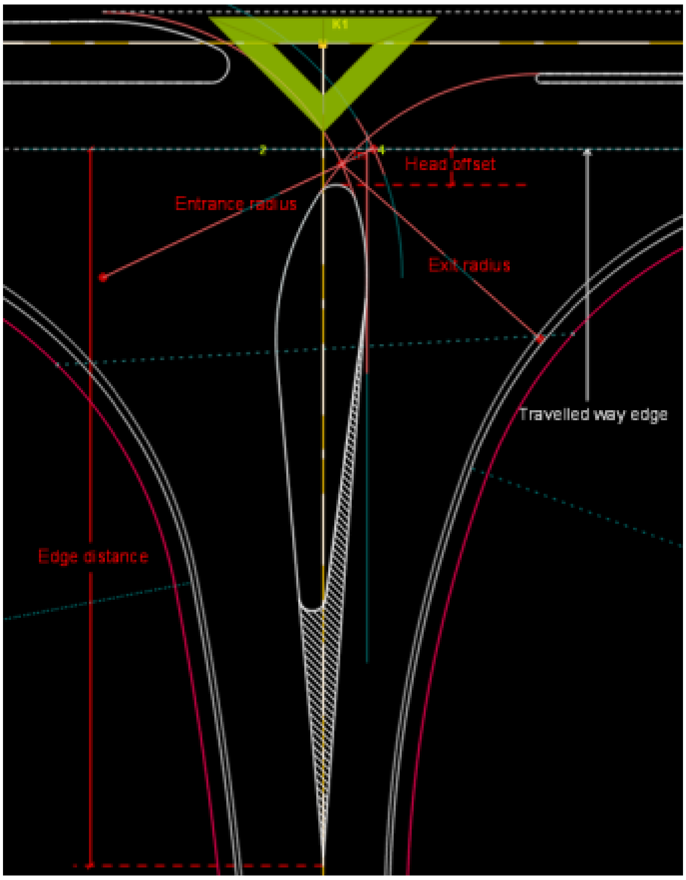
In this figure, the gray indicators show you how the triangular island was constructed. To view these indicators, you must have enabled the *Auxiliary data* field of the *Intersections* group in the *General* category of the *Horizontal Alignment Options*. *Distance 1, Distance 2* and *Radius* define the start point for the construction of the triangular island. *Distance 1* is the distance between the triangular island that will be constructed and the gray auxiliary line near the marginal strip. The user must then select the appropriate transition so that the marginal strip (or the edge of the paveway) snap on the aforementioned indicator.

The *Channelising island 2* is displayed as follows:

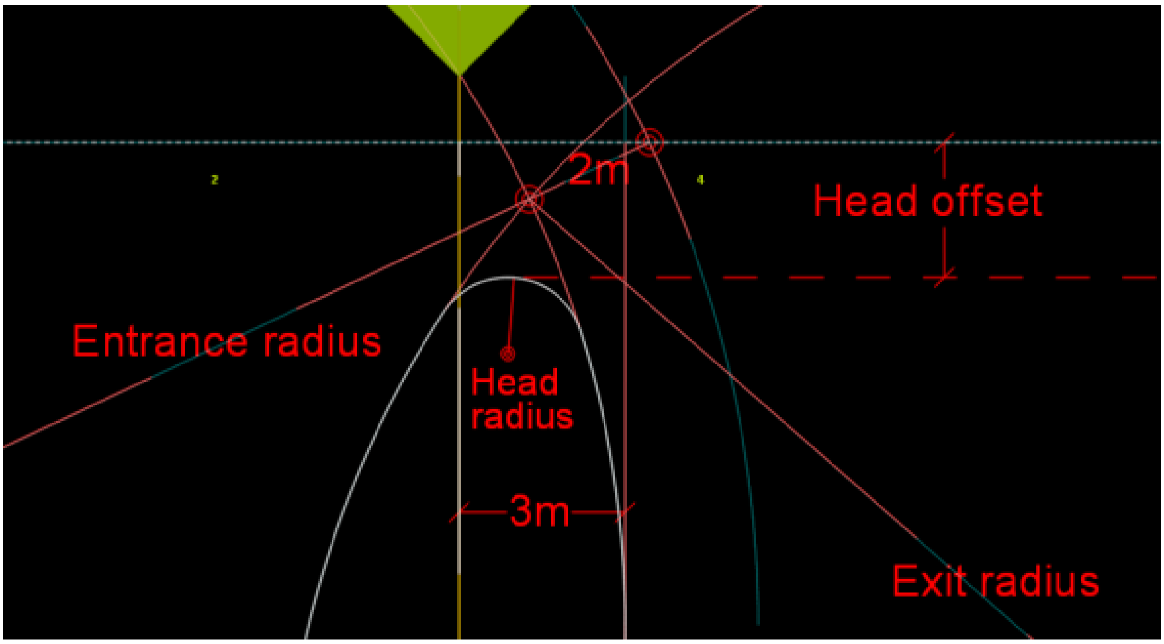


In the *Display* field you can define, if desired, whether or not there will be a channelising island at the intersection. You can also tick the Force small field if you want to insert a small channelising island when there is a left turn lane, which, depending on the way the channelising islands are created by the program, would automatically lead to the insertion of the big one.
In this tab you can also enter all the desired parameters for the channelising island, either big or small.

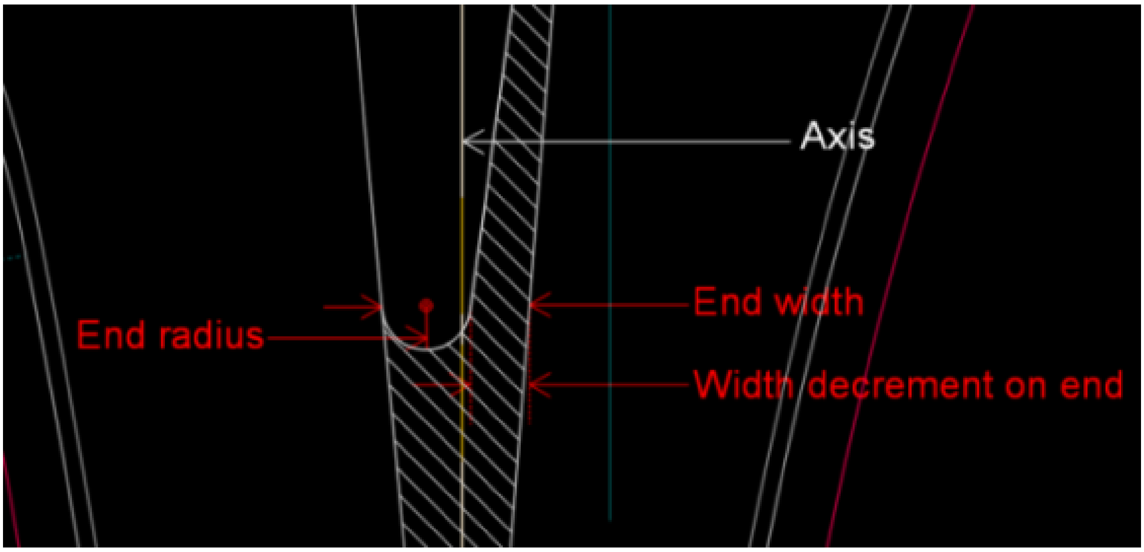
In the following figure you can view the basic elements that contribute to the calculation of the big channelising island:



First the channelising island is defined between the two arcs formed by the entrance and exit tracks of the superior road. The radii that are used for the tracks are entered in the *Entrance radius* and *Exit radius* fields, respectively. The program adjusts these radii by one meter, showing you the relevant message, so that the width of the resulting channelising island is between the limits of the *Minimum width* and *Maximum width* fields.

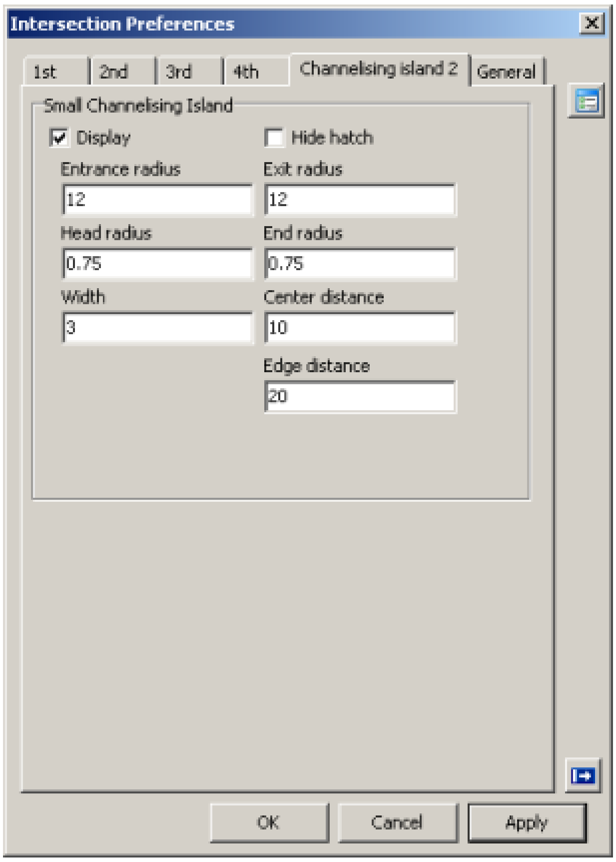


The arc with *Head radius* rounds up the front part of the channelising island, as shown in the detail of the above figure. The head radius is also adjusted by the program, to give the *Head offset* from the limit of the passing lanes of the superior road. In case of modification, the corresponding message is also displayed. Then the channelising island is limited by two tangent lines, to the two original arcs. The tangents start from the point that you define in the *Edge distance*. This distance is calculated by the limit of the passing lanes of the superior road.



The end of the channelising island is defined by first applying the *End width* between the two tangents. Next, the channelising island is further limited by applying the *Width decrement* on end by designing a parallel to the tangent, on the side of the entrance lane. The channelising island closes at the back section by applying an arc with *End radius.* The elements related to the calculation of the channelising island end are shown in the detail of the above figure.

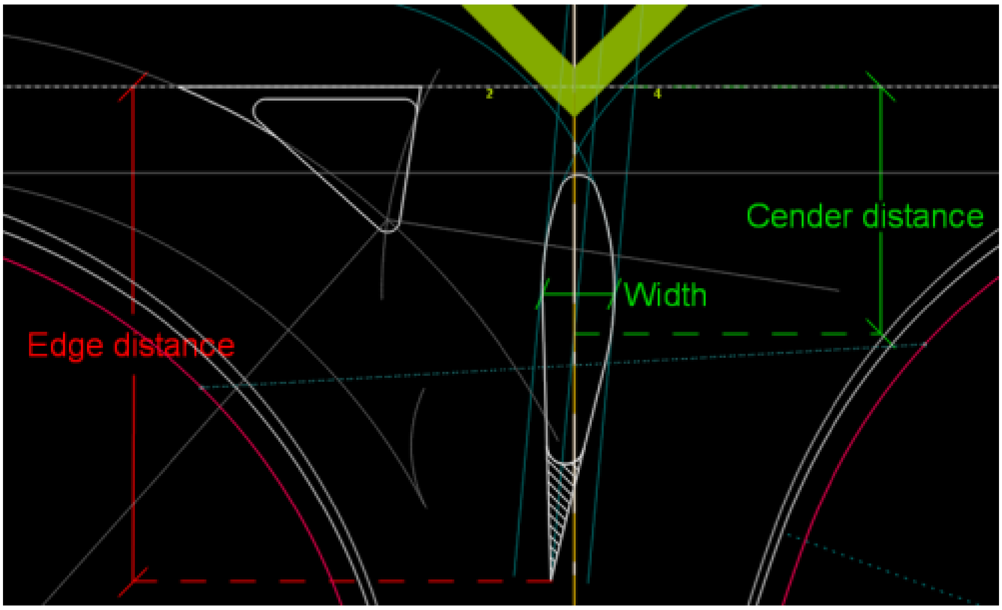
The program inserts automatically a small channelising island if there is no left turn lane on the main road. The *Channelising island 2* tab appears as shown in the following dialogue window,



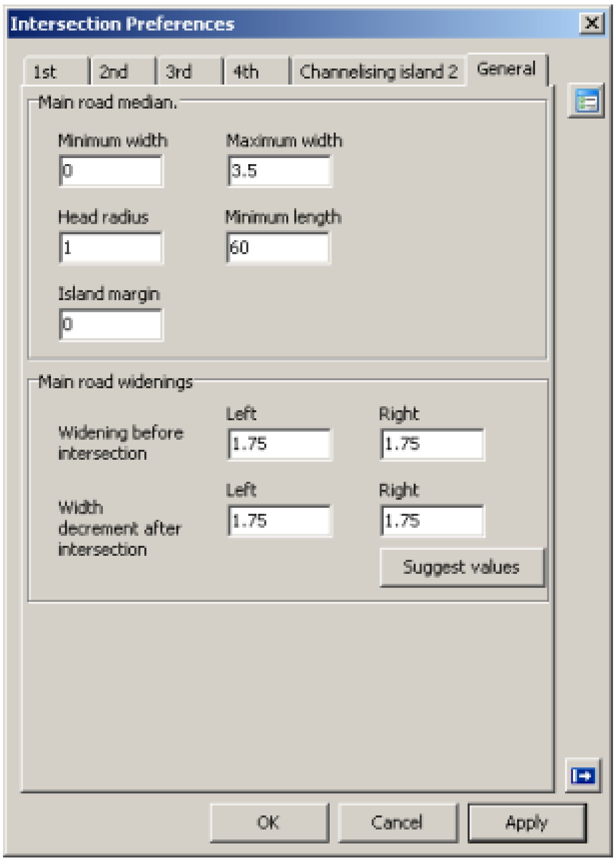
which, apart from certain characteristics of the big channelising island, also contains the extra options *Width* and *Center distance*.

The channelising island is first defined between the two arcs that are formed by the entrance and exit tracks of the superior road. The radii used for the tracks are defined in the *Entrance radius* and *Exit radius* fields, respectively. These arcs are tangent to two parallel lines, whose in-between distance is equal to the *Width.* These two lines are parallel to a straight line that derives from the curve of the secondary road's axis by 6 degrees around a point whose distance from the boundary of the passing lanes of the superior road is the *Center distance.*

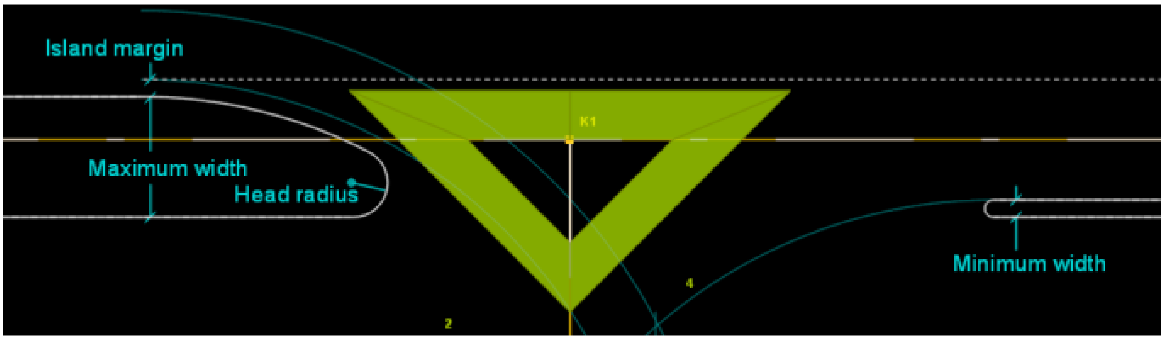
The arc with *Head radius* rounds up the front part of the channelising island. Then, the channelising island is limited by two lines which are tangential to the two original arcs. The *Road distance* determines the point from which these straight lines begin. This distance is calculated by the boundary of the passing lanes of the superior road. The end of the channelising island is defined by applying an arc with *End radius*. The aforementioned values are shown in the figure below:

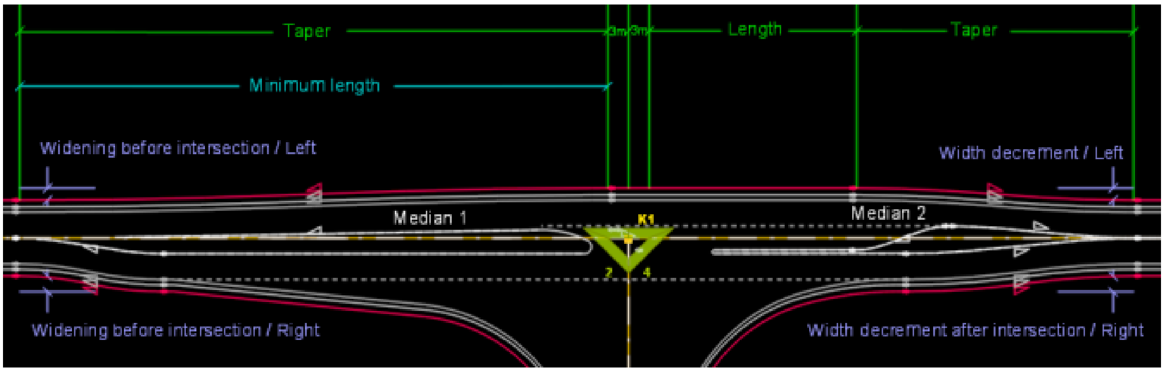


In the *General tab,* you can set the general preferences for the medians and the widenings of the main road. The tab is displayed as shown below:



In the *Main road median* group, you can enter the minimum and the maximum width, the head radius, the minimum length as well as the margin of the island. These values are displayed in the following figures:





The lengths of the transitions before and after the intersection are the same. They differ only if the value of the *Minimum length,* which refers to the length of island 1, is higher than the value of the *Transition* of the *Left turn lane.* In this case, the value of the length of island 1 becomes higher than the value of the transition length of island 2 and equal to the *Minimum length.*
In cases such as when you wish to edit the width of the traffic lanes or the *Right field* of the left turn lanes (2nd or / and 3rd quarter) for the correct and automatic setting of the widenings, which are automatically inserted before and after the intersection, press the *Suggest values* button in the *General* tab.

7.4 Printing

7.4.1 Preferences Setting

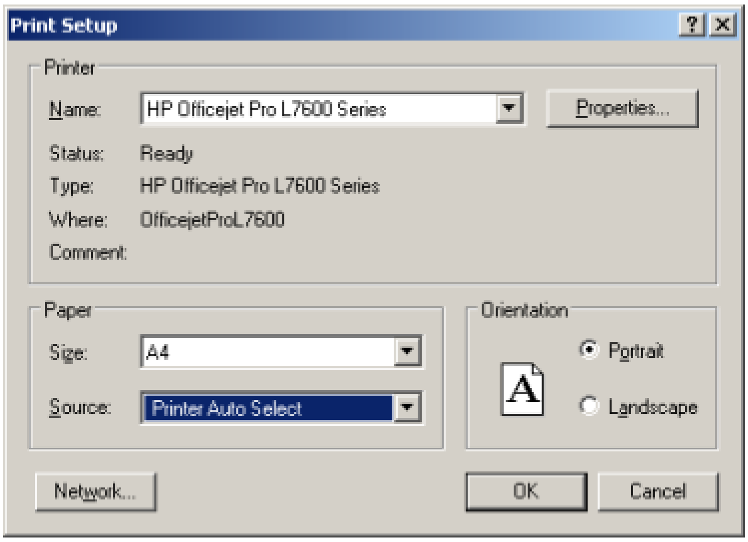
The *Print* menu contains the settings for the appearance of the *Horizontal Alignment* printouts.

Page Setup

First you must set the preferences for the paper, such as the feed tray, the orientation and the paper size.

How to setup the print page

1. From the *Print* menu, select the *Page Setup* command.
2. From the *Print Setup* dialogue box, in the *Name* field of the *Print* group select the desired printer by choosing from the dropdown list one of the available printers of the system. The other fields of the group show information about the selected printer, such as the status, the model, the port or the location of the printer, as well as any additional user-defined information – comment.

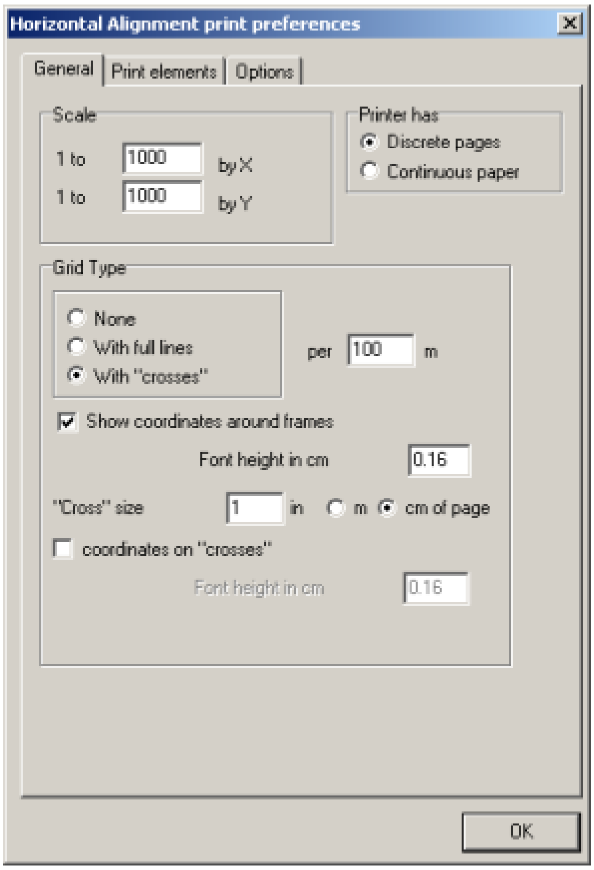


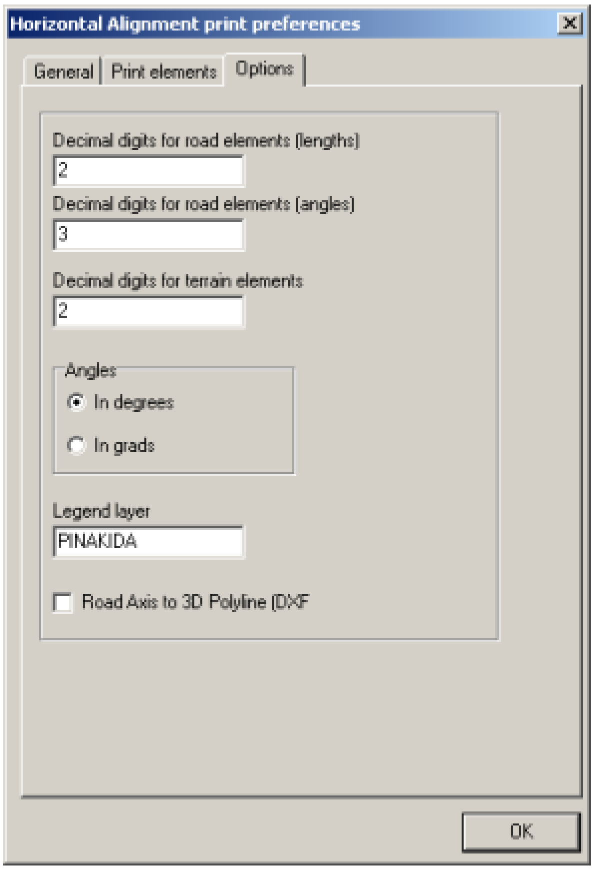
3. Click the *Properties …* button to change the Windows printer options for the selected printer. A dialogue box opens, enabling you to adjust settings that are not controlled by *Tessera*.
4. In the *Size* field of the *Paper* group, select from the dropdown list one of the available paper sizes. From the *Source* field, select one of the available paper feed trays.
5. In the *Orientation* group, select the orientation of the text on the paper for printers that support landscape orientation (Horizontal) or portrait orientation (Vertical). You can change the orientation to achieve a text rotation of 0 or 90 degrees by selecting *Vertical* or *Horizontal* option, respectively. The document icon shows the orientation of the paper on the selected printer. The letter icon shows the orientation of the text on the page.
6. *Vertical:* Orientates and draws so that the short end of the document represents the top part of the page.
7. *Horizontal:* Orientates and draws so that the long end of the document represents the top part of the page.

If you change the current printer, then the settings that are supported by the new printer will be kept, while for the other settings the default settings of the new printer shall apply.

Print Settings

This paragraph explains the dialogue box of the *Print Preferences* command, where you can define the print settings (data, options) for the whole file. The print preferences are common to every road contained in the file, as well as to the background (terrain-rendering) and the drawing objects. The dialogue box has three tabs. The *General* tab contains settings referring to the scale, the grid type and the type of paper that the printer uses. In the *Print elements* tab you can choose whether the drawing objects will be printed or not. Finally, in the *Options* tab, you can make settings referring to the decimals that will be used for the numbers that will appear in the printout.





Setting the Print Scale

Before inputting the pages to be printed, you must set the scale.

To set the scale

1. From the *Print* menu, select the *Print Preferences* command.
2. Type the scale in the text field of the *Scale* group of the *General* tab, e.g. type 1000 in the field if you want to print on a 1:1000 scale.

The scale setting affects the way that many objects appear on the screen, mainly the height of letters in all texts and the dimensions of the print frames. Therefore, you must set the scale as soon as you start to draw your design.

Selecting Drawing Objects for Printing

In the *Print elements* tab you will find the *Drawing* check box. Tick this box to print the drawing objects. Note that only the drawing elements that belong to printable parts will be printed.

Printing the Grid and Coordinates on the Print Page

You can choose to print the grid and its coordinates at the margins of the page and at its points. You have three options:

- *None:* The grid will not appear at all.
- *With full lines:* the grid will appear in the form of straight lines, which are parallel to the coordinates axes and extend up to the margins of the print page.
- *With "crosses":* the grid appears in the form of crosses, equally spaced one from another.

To select how the grid will be printed

1. From the *Print* menu, select the *Print Preferences* command.
2. In the *Grid Type* group of the *General* tab, select one of the available options.

If you choose the grid to appear on the printout, in the per text field you must type the actual distance (meters) between the lines or crosses.

If you have chosen to print the grid, you can also select the coordinates of the grid to appear at the margins of the print page.

To select how the grid coordinates will be printed

1. From the *Print* menu, select the *Print Preferences* command.
2. In the *Grid Type* group of the *General* tab, tick the *Show coordinates around frames* check box and in the *Font height in cm* field, type the height of the letters in centimeters.

If the selected form of the grid is the cross, then you must set the size of the cross and select the option that shows the coordinates on every cross.

How to define the form of grid crosses

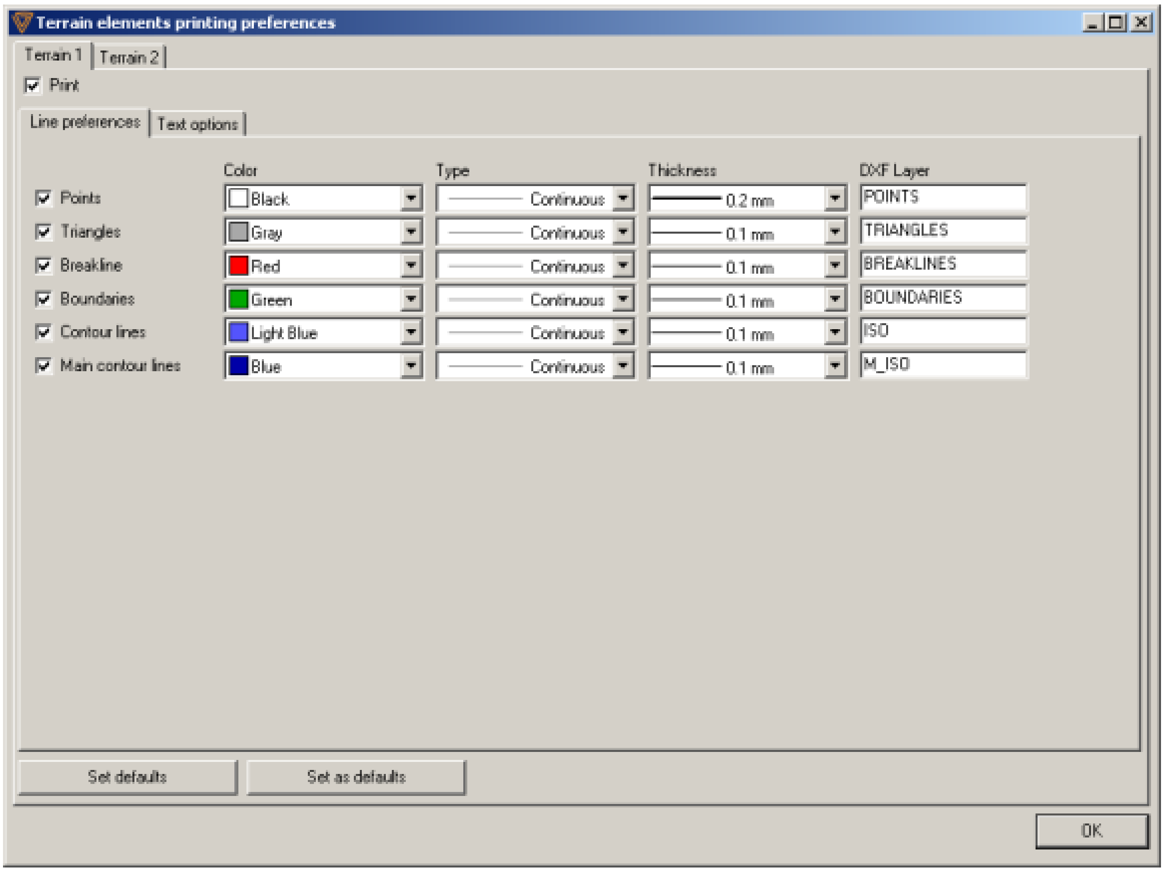
1. From the *Print* menu, select the *Print Preferences* command.
2. In the *Grid Type* group of the *General* tab, type the dimensions of the cross in the *"Cross" Size* field. If the dimensions you type is the actual distance, tick the *m* check box, while if it is in centimeters of printable paper tick the *cm* of page check box.
3. If you want the coordinates of the center of each cross to be printed next to the crosses, tick the coordinates at *"crosses"* check box and in the *Font height in cm* field type the height of the letters in centimeters.

Setting Decimal Points for the Print Elements

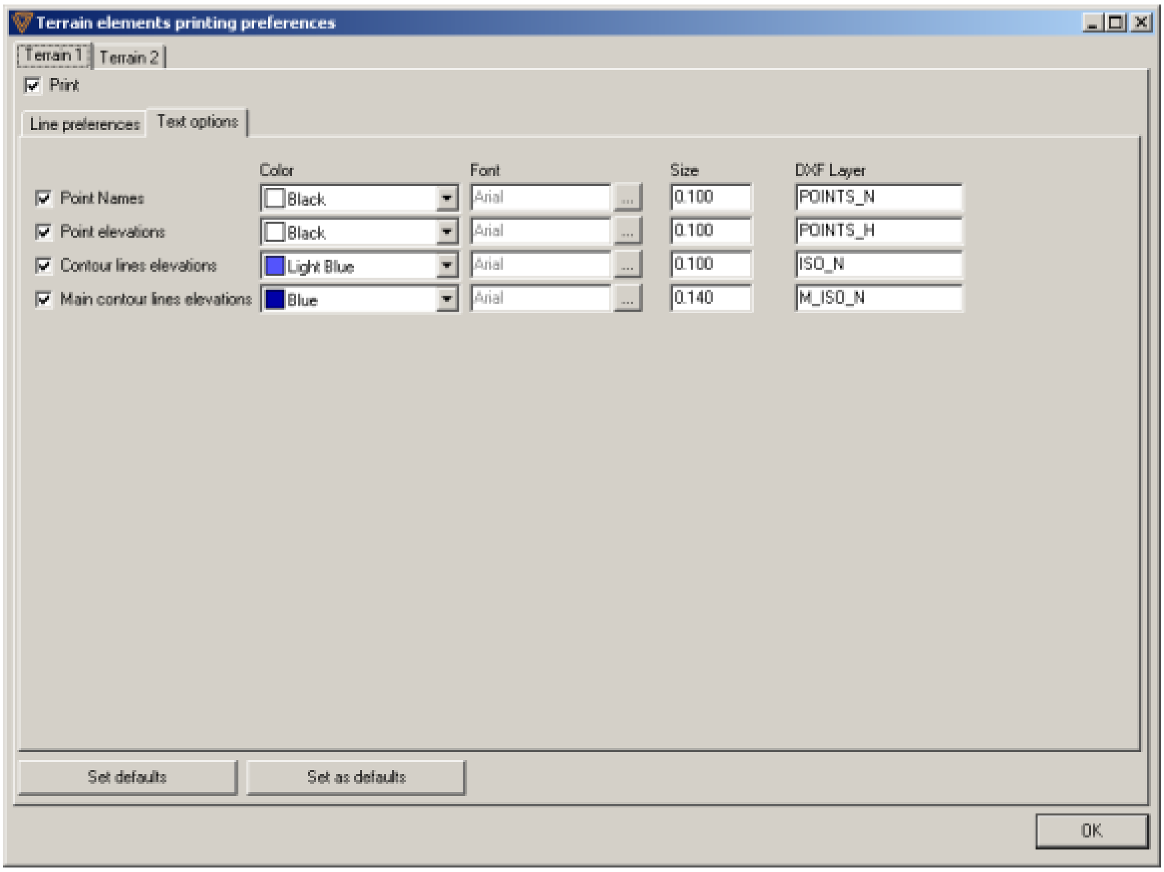
In the *Options* tab, set the number of the decimal points for the lengths of both the road and the terrain. You can also set the angle measurement unit.

7.4.2 Selecting Terrain Objects

In the dialogue box of the *Terrain elements printing preferences* command of the *Print* menu, you can specify which of the terrain objects will be printed and how. The dialogue box is divided in as many tabs as the terrains in the file (*Professional version*). Each one of the terrain tabs has two tabs, the *Lines preferences* tab for the lines



and the *Text options* tab for the texts,

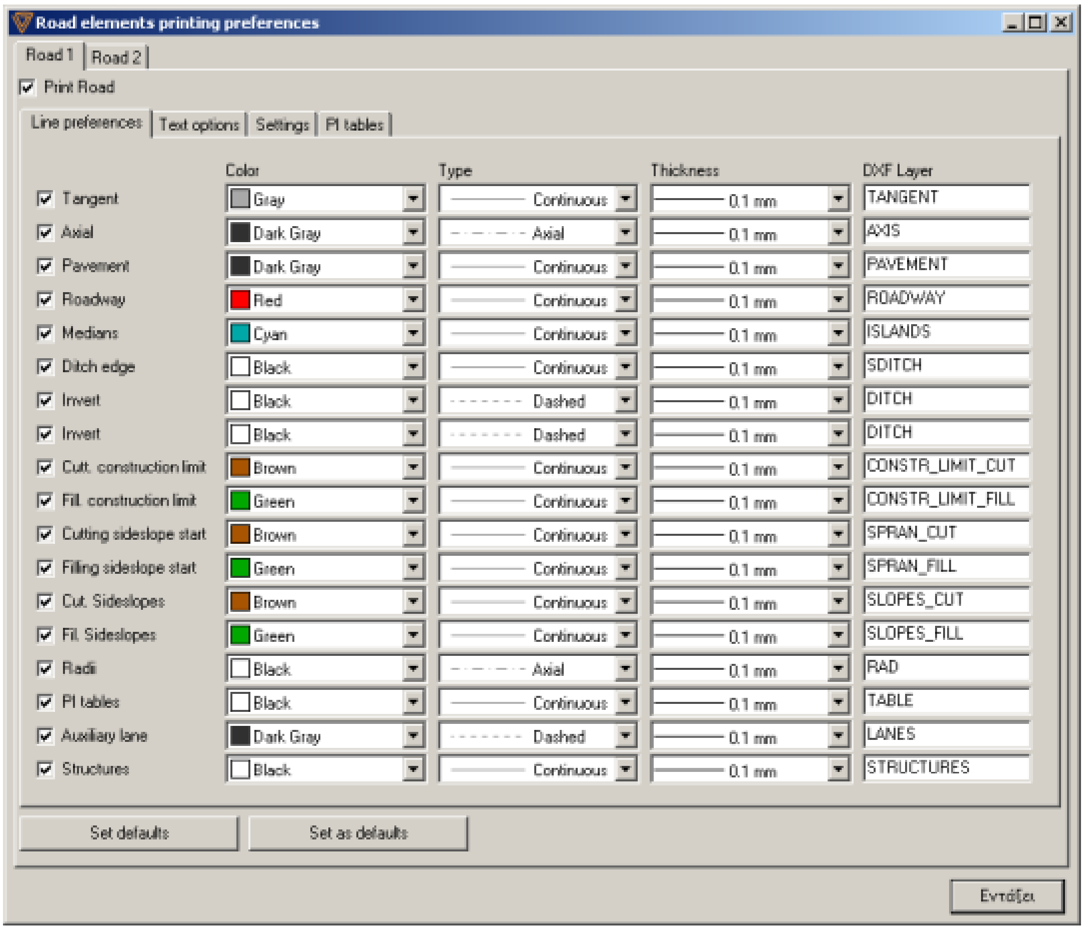


If there is a terrain that you don't want to print, untick the *Print* check box in the tab of the relevant terrain. Each tab has check boxes for each one of the terrain object categories that are available in *Tessera*. If you don't want to print one or more of these terrain objects, untick the relevant check boxes. You can make settings for the objects that will be printed from the relevant fields that are on the same line with the element. For the lines you can change the color, the type of line and the width, always in the terms of printing. The last column refers to the export to DXF format.

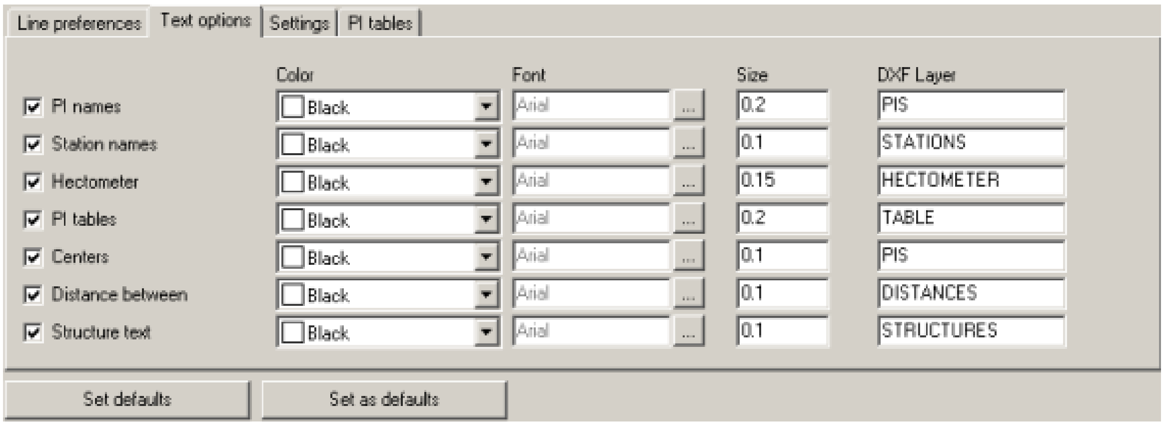
You can also change the color of the texts. To change the text font, select the *Font* field and click the button with the three dots to open a font dialogue box. In this box, you can choose a new font or whether the characters will appear bold or italic. To change the font height, simply type the new height in the *Size* field, in centimeters of printing paper. Here too, the last column refers to the export to DXF format.

7.4.3 Road Objects Selection

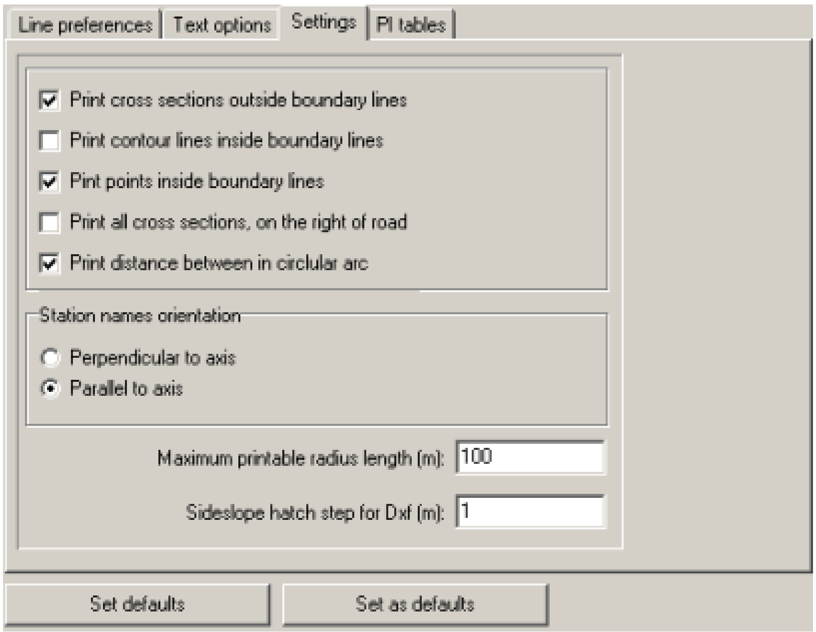
In the *Road elements printing preferences* command dialogue box of the *Print* menu, specify which road objects will be printed and in what form. The box is divided in as many tabs as the roads in the file (*Professional version*). Each road tab has four sub-tabs, the *Lines preferences* for the lines,



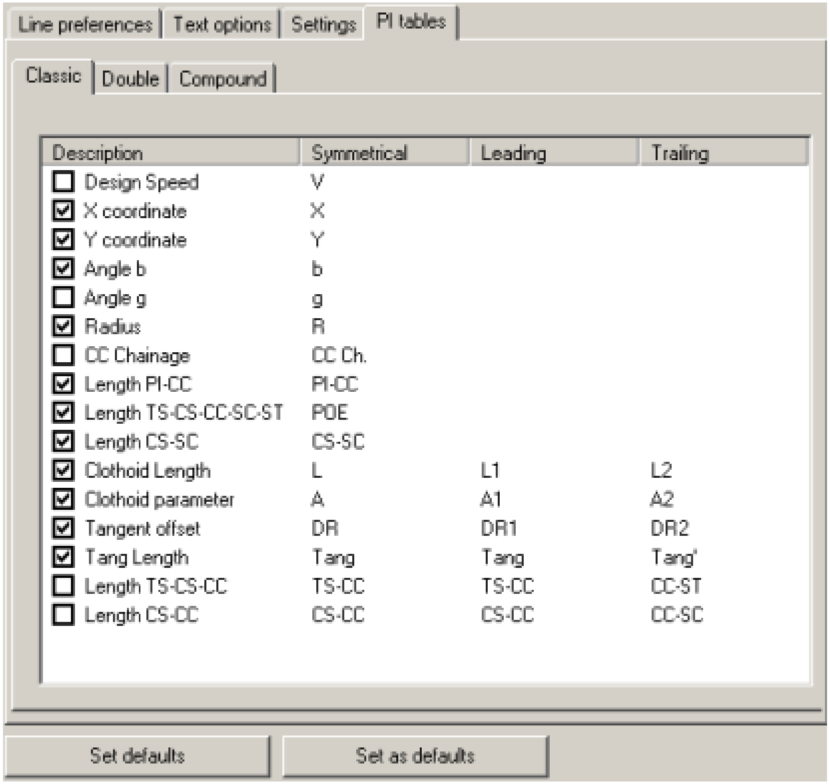
the *Text options* for the texts



the *Settings* for printing the road,



the *PI Tables* for setting the elements that will be printed in the tables of the road PIs.



If there is a road you don't want to print, untick the *Print road* check box in the relevant tab of the *Road elements printing preferences* window. Each tab has check boxes for each one of the road object categories that are available in *Tessera*. If you don't want to print one or more of these road objects, untick the relevant check box/ es. In the Color, Type, Size and DXF Layer fields you can make settings for the objects that you have selected to print.

7.4.4 Inserting and Editing Pages

The printing process is based on the use of print pages. Namely, you must specify an area that will contain the elements that will be printed. The dimensions of the print page are proportional to the paper page you have defined in the *Page Setup* dialogue box and to the print scale. In the dialogue boxes of the *Preferences, Terrain printing preferences* and *Road printing preferences* commands you can select the elements that will be displayed in this area.

Inserting Print Pages

You can insert a print page very easily, by using the mouse.

To insert a print page

1. From the *Print* menu, run the *New Page* command or, on the *Elements Drawing* toolbar, click the button.
2. Position the page by defining its center, either by clicking the left mouse button or by entering the center's coordinates with the assistance of the *Input Form* command.

The page number is written at the top of the print page, according to the order the page was inserted.

It is possible to insert a new page, which is conjoined to another existing page, so that they coincide on one of their sides.

How to insert a print page next to an already existing one

1. From the *Print* menu, run the *New conjoined page* command, or on the *Elements Drawing* toolbar, click the  button.
2. Joint the page with another page by placing the pointer into an already existing page, on the side you wish to join the pages. The page will be placed next to the existent one.
3. Left click to finalize the make the position of the page.

Editing Print Pages

You can change the position of a print page, or rotate the page, in order to get the position in the drawing. You can also delete a print page.

<u>Moving and Rotating Print Pages with the Mouse</u>

You can move or rotate a print page by using the mouse.

To move a print page with the mouse

1. Select the page by left clicking in it.
2. Left click again on the selected page.
3. Move the page to its new position by defining its center with the left mouse button. You can also move the page by entering the coordinates of its center with the assistance of the *Input Form* command.

To rotate a print page with the mouse

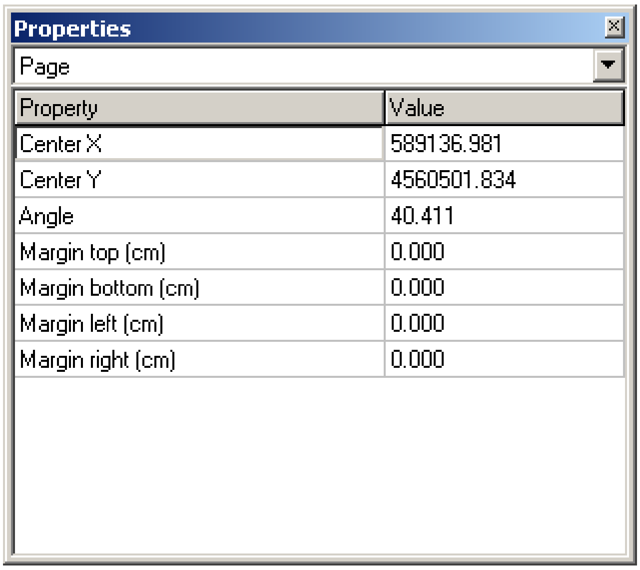
1. Select the page by left clicking in it.
2. Having selected the page, position the crosshair on the rotation control point (top right of each page) to highlight it and to make it appear in white color. Click again the left mouse button.
3. Rotate the page by moving the crosshair in circular direction. When the page reaches the desired position, click again the left mouse button to finalize (confirm) the position. You can also rotate the page by using the *Input Form* command.

<u>Moving and Rotating Print Pages</u>

You can also perform the actions of the previous paragraph from the window of the *Properties* command.

To rotate and move a print page

1. Select the page by left clicking in it.
2. Having selected the page, run the *Properties* command from the shortcut menu, which is displayed if you right click in the drawing area. The following dialogue box will appear on the screen:



3. Enter the new values of the X and Y coordinates of the page center in the respective value fields of the *Center X* and *Center Y* properties.
4. In the value field of the *Angle* property enter the new value of the rotation angle of the print page with respect to the horizontal direction.
5. In the value fields of the *Margin top (cm), Margin bottom (cm), Margin left (cm)* and *Margin right (cm)* properties type the desired margins of the page.
6. In all cases, click the exit button ☒, at the top left of the window, to close the *Properties* command window.

Deleting a Print Page

If you want to delete a print page, the program enables you to do so as follows:

To delete a print page.

1. Select the page by left clicking in it.
2. Having selected the page, run the *Delete Object "Page"* command, from the shortcut command which is displayed if you right click in the drawing area, or on the main toolbar click the button.

The other pages will be renumbered, so that there is no gap in their numbering.

PI Tables

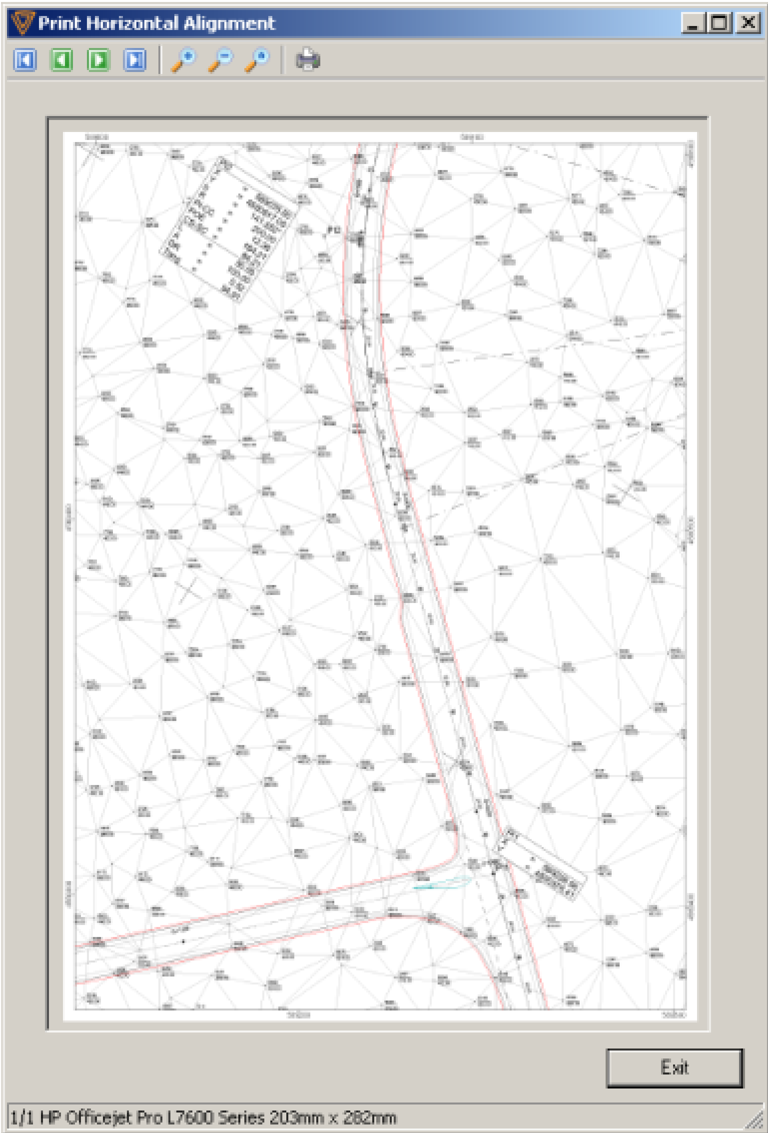
When the *Print* mode is activated, an outline appears near every PI, indicating the position where a small table with the PI's data will be printed. The data that will be printed in this table, are selected from the dialogue box of the *Road Print Preferences*. You can move and rotate these tables exactly as you would do for the print pages. You can also request from the program to rotate them, so that they become parallel to the selected print page. Simply select the print page and from the shortcut menu that is displayed if you right click in the drawing area, run the *PI Table Orientation* command. This command rotates only the tables whose PIs are within the margins of the selected page.
If you have moved or rotated some tables, run the *Reset PI Tables* command from the *Print* menu. The tables will be restored to their original positions and they will be rotated, either to become parallel to the print page to which they belong or, if there is no print page, to become vertical.

7.4.5 Preview

When you have finished making settings for the print preferences and the print pages, according to those described in this chapter, you can first display the print preview and then print your drawing.

To preview a print page

1. From the *Print* menu, run the *Print* command. The *Print Horizontal Alignment* window will appear on the screen.



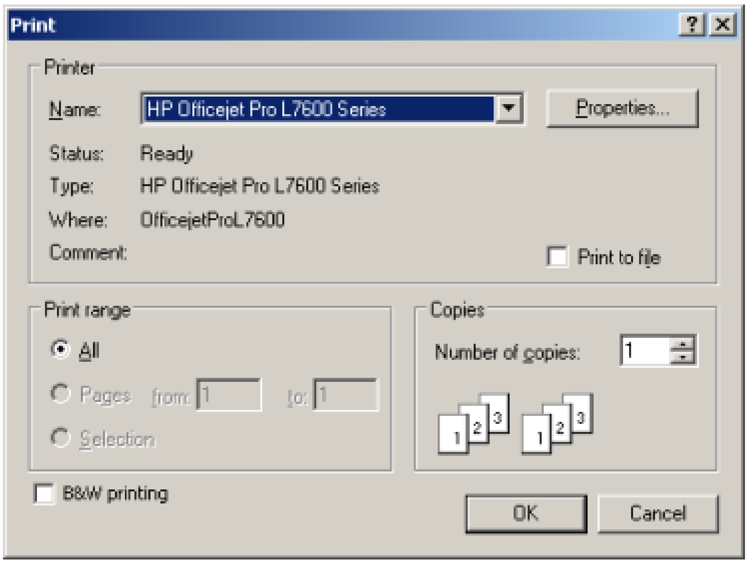
2. Click the maximize button on the title bar to view the window on full screen. This window is comprised of the following:

- *Preview:* the bottom part of the window where the image of the whole or part of the selected page is displayed.
- *Toolbar:* this is located at the top of the window. It carries the buttons that control the transition from page to page and the modification of the size of the page's viewable area. More specifically, the buttons on the toolbar are the following in order of appearance:

- *First Page:* Left click on the , button to view the first page.
- *Previous Page:* Click on the , button to view the previous page.
- *Next Page:* Click the , button to view the next page.
- *Last Page:* Left click the , button to view the last page.
- *Zoom In:* Click the , button to zoom in on the page. This feature does not affect the print size.
- *Zoom Out:* Click the , button to zoom out on the page. The zoom feature does not affect the print size.
- *Full Page:* Click the , button to view the whole page.
- *Print Page:* Click the , button to set the options before printing and to print the pages.
- *Page Elements:* This section is located below the preview section and shows information regarding the page that appears in the preview page, as well as information on the selected printer and the useful dimensions of the paper.

7.4.6 Printing a Page

You can print a page by running the *Print* command of the *Print* menu. This command results in the dialogue box shown in the figure:



How to print the Horizontal Alignment

1. From the *Print* menu, run the *Print* command. The *Print Horizontal Alignment* window will appear on the screen.
2. Click the button.
3. In the *Print* dialogue box, make the desired settings.
4. Press the *OK* button to start printing.

In this dialogue box, you can also select whether the drawing will be printed in black and white by ticking the *B & W printing* check box, at the bottom left of the dialogue box, or to print to a file, by ticking the *Print to file* check box. If you wish, you can print only part of the print pages in more than one copy.

Printing all pages, selected pages or a range of pages

1. From the *Print* menu, run the *Print* command. The *Print Horizontal Alignment* window will appear on the screen.
2. Click the  button.
3. In the *Print* dialogue box, which is below the *Print range* field, enter the part of the document you wish to print.

- To print all pages of the file, select the *All* check box.
- To print a range of pages, select the *Pages* check box and enter the range of pages you wish to include in the printout, by typing the number of the first page in the *From:* field and the number of the final page in the *To:* field.

To print two or more copies simultaneously

1. Click the *Print* button.
2. In the *Copies* field, enter the number of the copies you wish to print.

To print a copy of the whole document, tick the *Collate* check box before printing the first page of the next copy. If you prefer to print all the copies of the first page and then print all the copies of the next pages, untick the check box.

Alternatively, you can print only one print page, the current one. To do this, select the page and then run the *Print* command from the shortcut menu that is displayed if you right click anywhere in the drawing area. The next steps are identical to those described above.

7.4.7 Export to DXF File

You can export the contents of the horizontal alignment to .DXF format, so as to process them in another drawing program.

How to export data to a DXF file

1. From the *File* menu, select the *Export to DXF* command.
2. In the displayed dialogue box, enter the name of the file in the *File Name* field.
3. In the *File Type* field, enter the type of the file, depending on the desired version, *14 or 2000*.
4. In all cases, click the *Save* button to finish the creation of the file.

In the dialogue boxes of the *Print Preferences, Terrain Printing Preferences* and *Road Printing Preferences* commands you can set the form of the elements and how they will be displayed, as well as which elements will be exported to your file and to which layer. Therefore, it is advisable to preview first the whole or part of the design. Before exporting the file you must also have set the correct scale based on which the size of the data font is determined. Throughout the program, the size of the elements (names, terrain point elevations, PI names, cross sections and other texts) are in cm. In the DXF file this value is expressed in “universal” m. The conversion is performed with the use of the drawing scale.

Chapter 8

8 Profile

8.1 Creating a Road Project

With *Anadelta Tessera* you can start creating a new *Profile* project in two ways:

1. You can request the automatic creation of the Profile based on the elements that you have inserted in the *Horizontal Alignment* or in the *Cross Sections* workspaces.
2. You can create directly a new Profile project that you can process only when working in the *Profile* workspace and then, if necessary, to create the *Cross Sections* of the same road.

Creating a New Profile Project

This procedure is useful in case you wish to use the program only for creating a single profile, and also for making measurements by using cross sections.

In the *Profile,* a terrain line is comprised of points and a set of coordinates (Chainage, H), and each set corresponds to the chainage of a station and to the elevation of the terrain on the axis where the station is. The new project that you are creating is empty, namely it doesn't contain any stations or lines. All the elements of the profile (stations, terrain elevations, etc., apart from the road profile) must be created by the user, as described in the next paragraph.

How to create a new Profile file

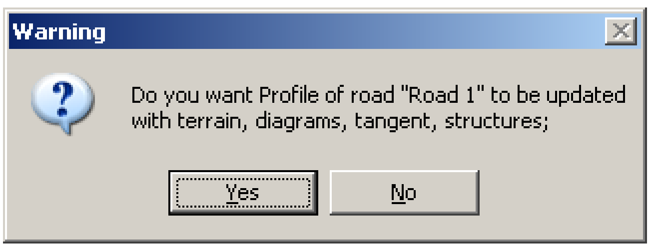
From the *File* menu, select the *New Project...* submenu and run the *Profile* command.

Creating or Updating a Profile from the Horizontal Alignment or the Cross Sections workspace

This procedure is performed after you have completed the horizontal design of the road in the *Horizontal Alignment* workspace or after you have created the *Cross Sections* independently and you wish to create and edit the *Profile.* In the first case, the cross sections that will comprise the *Profile* are those that have derived from the road stationing, while in the second case the cross sections are the ones of the Cross Sections workspace.

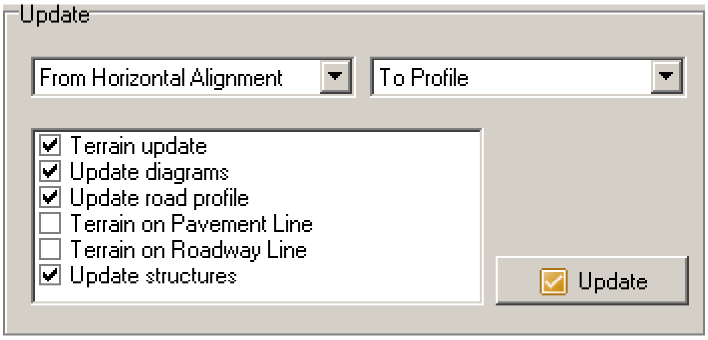
How to create the Profile from the Horizontal Alignment

1. From the *Horizontal Alignment* workspace, select the road whose *Profile* you wish to create or update.
2. From the *Project* menu, run the *Manage* command, or click the  button on the main toolbar. In the *Quick Update* group, click the *From Horizontal Alignment to Profile* button. The following confirmation message will appear on the screen:



3. Click *Yes* to continue. The program will create the *Profile.* A message appears at the end of the procedure, informing you that the procedure has been completed. To run

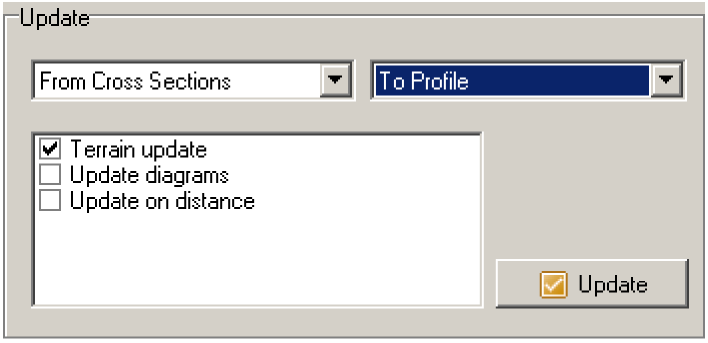
the command, you must have performed stationing on the selected road. When this type of update takes place, the main terrain line and its points are created. The left and right superelevation diagrams, as well as the tangent diagram are also transferred. Finally, if there is a road profile diagram in the *Horizontal Alignment,* then this diagram is copied to the main tangent curve of the *Profile.*
Alternatively, you can fully define which elements of the *Profile* will be updated. To do this, use again the *Manage* command. From the *Update* group of the *Updates* tab in the command's dialogue box, select *From Horizontal Alignment* in the first drop-down list and *To Profile* in the second drop-down list. Next, select which updates will take place by ticking or unticking the relevant check boxes and then click the *Update* button.



The program will create the *Profile* workspace, if there is no profile for the selected road; next, in *Profile* workspace, the program will create the *Terrain* line if you have requested it or by default if this is the first time this workplace is being created. If certain cross sections of the road in the *Horizontal Alignment* workspace are outside the terrain model, these cross sections get the elevation of the previous cross section and the program shows you a warning message, informing you about the inability to find natural ground for these cross sections. If your project does not have a terrain model or points, the *Terrain Line* will be created anyway, but all elevations of its cross sections will be zero. You can use the aforementioned commands to update again the *Profile,* after having edited the road design in the *Horizontal Alignment* workspace.

How to create the Profile from the Cross Sections

1. Go to *Cross Sections* workspace that corresponds to the road whose *Profile* you wish to create or update.
2. From the *Project* menu, run the *Manage* command, or click the  button on the main toolbar. From the *Update* group of the *Updates* tab in the command's dialogue box, select *From Cross Sections* in the first drop-down list and *To Profile* in the second drop-down list. Next, select which updates will take place by ticking or unticking the relevant check boxes and then click the Update button. The program will create the *Profile* workspace, if there is no profile for the selected road; next, in *Profile* workspace, the program will create the *Terrain* line if you have requested it or by default if this is the first time this workplace is being created. The diagrams of the Cross Sections workspace, as well as any new line that originates from selected points of the Cross Sections, will also be transferred, if you have requested so by making the relevant selections. You can use the aforementioned procedure to update again the *Profile,* after having edited the *Cross Sections* workspace.

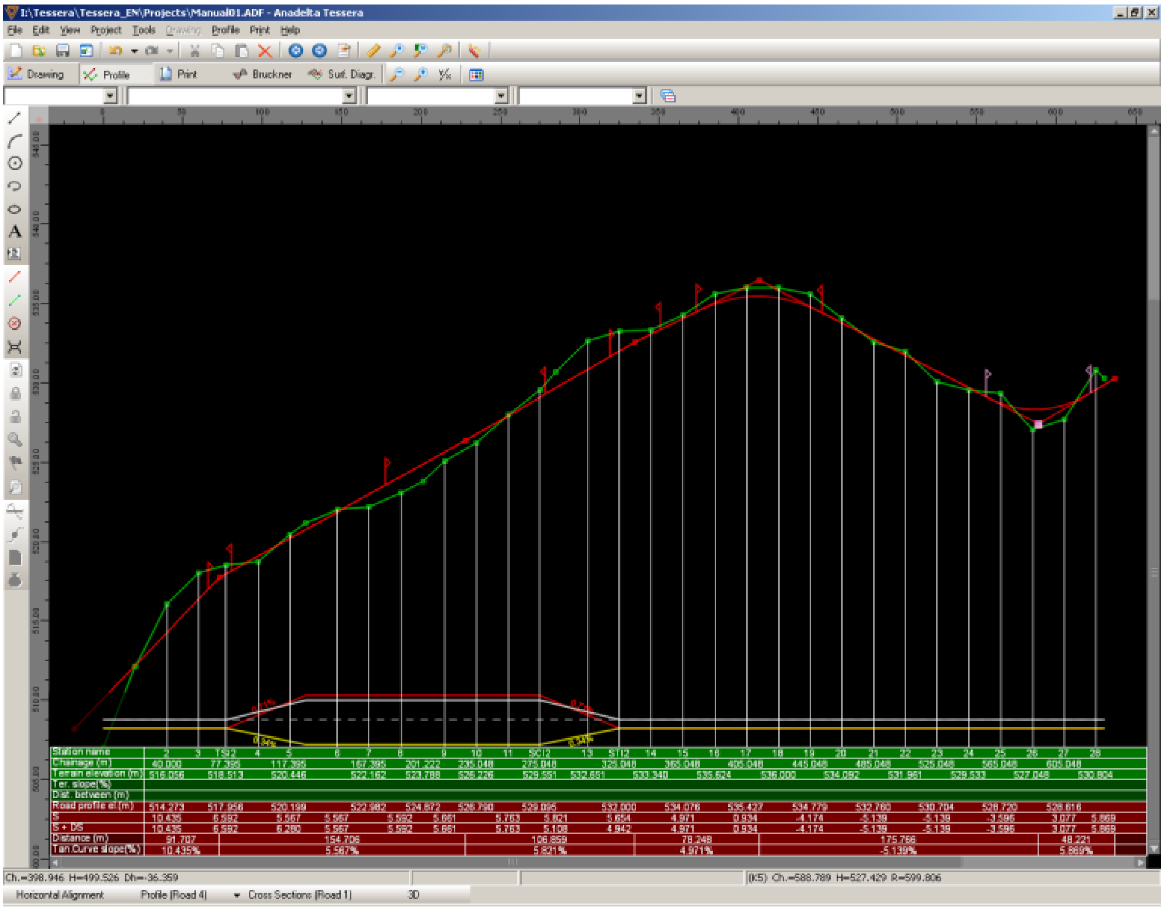

Update
From Cross Sections
To Profile
Terrain update
Update diagrams
Update on distance
Update

8.2 Workspace

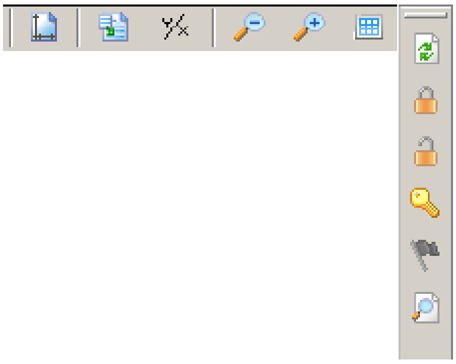
Every time you open or create a *Profile* project, you are transferred to the *Profile* workspace. The biggest part of the screen in this workspace is occupied by the graphic representation of the profile. At the bottom of the screen, you can view (if any) the superelevation and tangent diagrams.

To go to the Profile workspace

From the *View* menu run the *Profile* command, or select the *Profile* tab at the bottom of the main screen. The program transfers you to the *Profile* workspace.



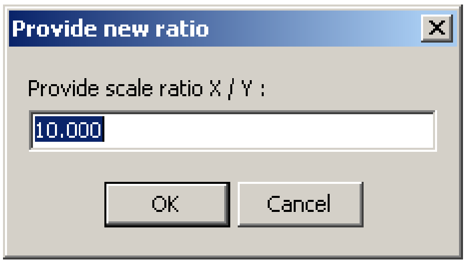
Similarly to the *Horizontal Alignment* workspace, at the top of the screen you will find the familiar menu bar, with the *File, Edit, View, Project, Tools, Print* and *Help* menus. A new menu has also been added to the *Profile.* Below the menu bar, you will find the main toolbar and below the latter you will find the *View* toolbar, which contains buttons that carry out operations relevant to the elements that are displayed on the screen, especially in the drawing area.

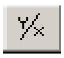


On the left of the screen, you can see the *Element Drawing* toolbar that contains various operations, which have been categorized in four groups. From now on, the first group will be referred to as *Drawing,* the second as *Edit,* the third as *Print* and the fourth as *Bruckner*.

The status bar at the bottom of the screen is comprised of four sections. The first section shows the coordinates of the current position of the crosshair (chainage and absolute elevation), as well as the elevation discrepancy between the station and the terrain elevation at the particular chainage. The second section displays the *Input Form,* if available. Next to it you can see the type of the line, e.g. Natural Ground, Tangent Curve, etc., on which the crosshair is located. The fourth and last section shows information about the selected station or PI of the profile.

The profile, the surface diagram and the Bruckner diagram are usually displayed in skewed scale (different scales for the X and Y axes). You can define the scale ratio for the display of the profile on screen from the dialogue box of the *Scale X/Y* command of the *View* menu. Type the ratio in the dialogue box and then click the *OK* button.



You can increase this ratio by 20%, if you run the *Zoom out* command (<*F5*>) from the *View* menu, or decrease the ratio by using the *Zoom In* command (<*F6*>). You will have the same result if you click the  buttons for zooming out and the  buttons for zooming in. You will find these buttons on the *View* toolbar. If you want the profile zoom ratio on the screen to be the same as the one you have set for printing, press on the same toolbar the  button.

8.3 Profile Workspace Modes

There are five modes in the *Profile* workspace. This allocation facilitates the user to insert, edit and process the elements of the design. By default, the *Profile* mode is always active every time you create a new *Tessera* file.

Profile Mode

In this mode you can create, edit and print the profile of a road.

In the *Profile* mode you can create two types of lines: the terrain lines and the tangent curves. A terrain line consists of one or more points that are visible to the user. Every point of this line has a set of coordinates x and h, that corresponds to the distance of the station from the beginning of the road (Chainage) and to the elevation of the terrain on the station. The terrain line points are displayed on the screen and they are contained in a small box in order to be more distinguishable. You can insert more than one natural ground lines. The program considers the first inserted line as the main line. The terrain lines have always points at the same chainages. Tangent curves are also comprised of one or more points that are visible to the user. Every point of a tangent curve has a set of coordinates (x, h) that correspond to the chainage and the elevation. A rounding radius corresponds to every PI. By default, the value of the first and last PI's radius is zero. Here too, you can insert more than one tangent curves, but the first tangent curve is always considered as the main or current one. The graphical representation of the lines is displayed in the drawing area. You can move left, right, up and down within the drawing area by using the respective arrow keys or the mouse.

The current point is the selected point, which appears highlighted in pink on the screen. The relevant information on the current point is displayed at the bottom right of the screen, in the last section of the status bar. In case that the current point belongs to a tangent curve, the following information is displayed in the order described below: the PI serial number from the beginning of the line, the distance of the point from the beginning, namely from its Chainage (X), its absolute elevation (H) and the rounding radius (R). Regarding the natural terrain, the following information is displayed in the order described below: the station name, the station chainage, the absolute elevation (H) and the elevation difference between the terrain and the current tangent curve (Dh).

At the bottom of the drawing area and below the profile graph, you can see the data axes (or tables) of the various lines whose names are displayed on the left. The available data tables are: *Station name, Chainage, Terrain elevation, Natural terrain slope, Distance between successive cross sections, Road profile elevation S, Roadline slope S+DS, Distance between tangent curve points, Tangent curve slope, Oblique slope left, Oblique slope right, Level discrepancy, Superelevation right, Superelevation left* and *Display data.* The first five tables, which refer to the *Natural Ground,* are shown in green and they are displayed when there is even a single terrain line. The next five tables are shown in red and they are displayed when there is even a single tangent curve. The last five tables are displayed in gray when there is both a terrain line and a tangent curve.

You can choose which tables will be displayed on screen, as well as the order in which they will be displayed. To move a table, select it by clicking the left mouse button and move it to the desired position. More specifically:

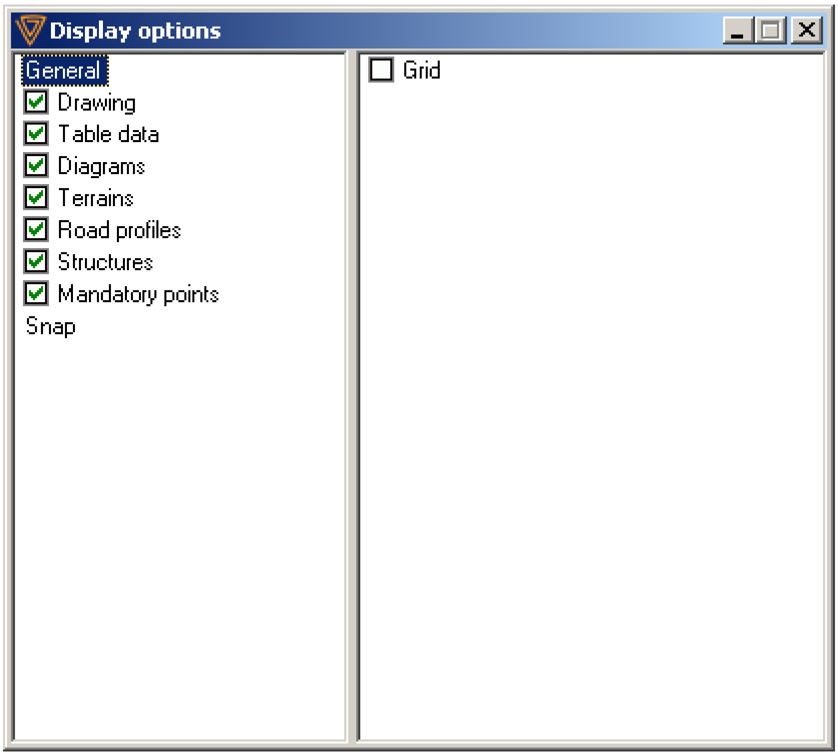
1. Click the left mouse button to select the data axis or table you wish to move. The color of the table changes automatically from green (in the case of a Natural ground table) to purple.

2. Left-click on the selected table to drag–and-drop it to the desired position.

To select which axis data will be displayed from the *View* menu, run the *Display Options* command, or press the *F2* key, or click the  button on the *View* toolbar.

8.4 Profile Display Options

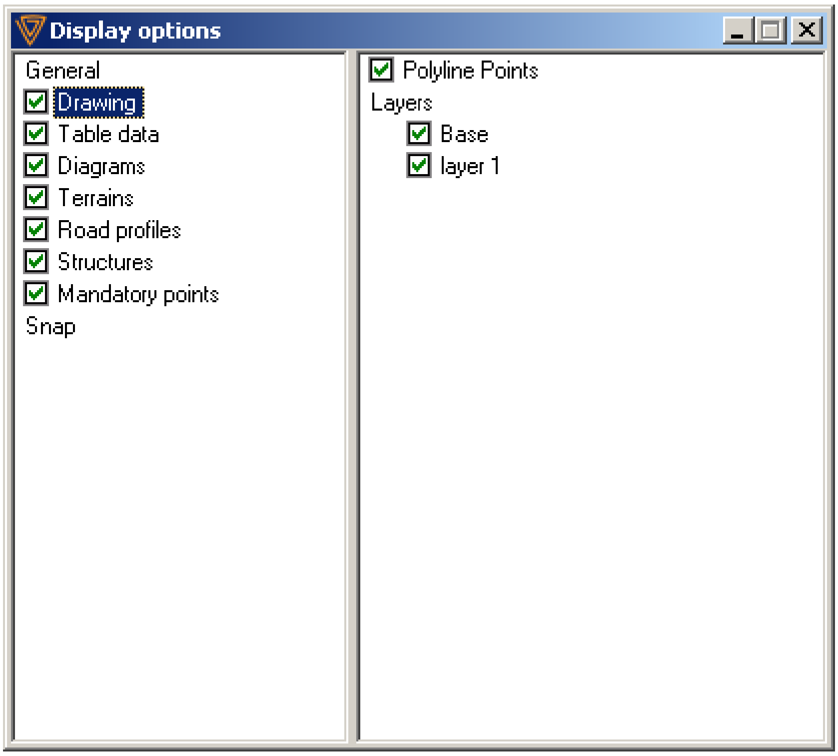
1. From the *View* menu, run the *Display Options* command, or press the *F2* key, or click the button on the *View* toolbar. The following dialogue box appears on screen:



2. Select one of the item categories that are displayed on the left part of the window.
3. If you want none of the items of the category to be displayed, untick the box next to the name of the category. If you want only some of the items of a category to be displayed, select them from the right part of the window by ticking the corresponding check box, which is displayed next to their name.
4. To exit the window, click the exit button on the title bar of the window.

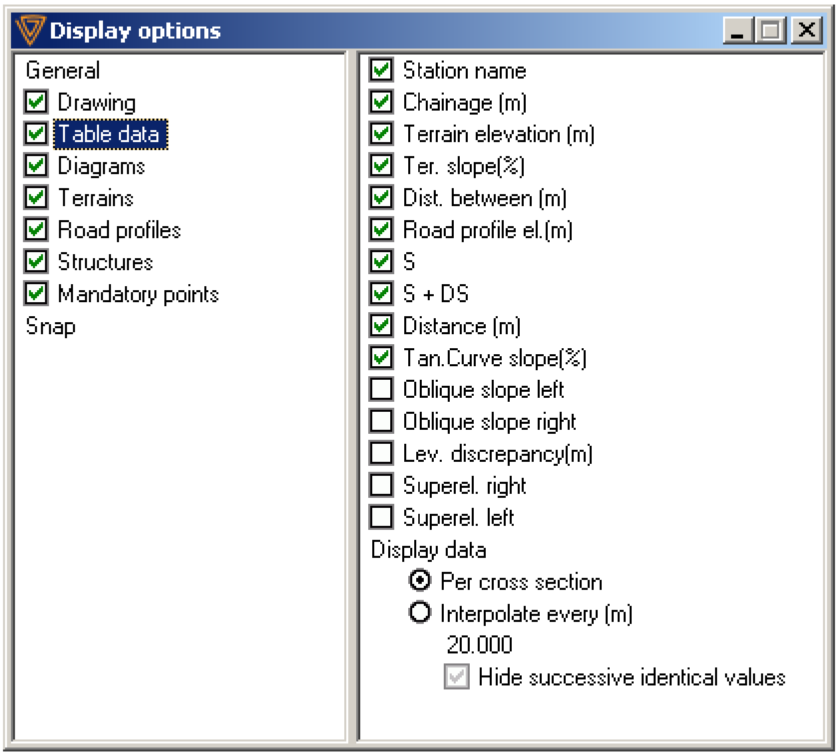
When the *General* option is ticked, the only available option is:

- *Grid:* Select this field to display the grid.



When the *Drawing* option is ticked, the available options are the same as in the *Horizontal Alignment*, namely:

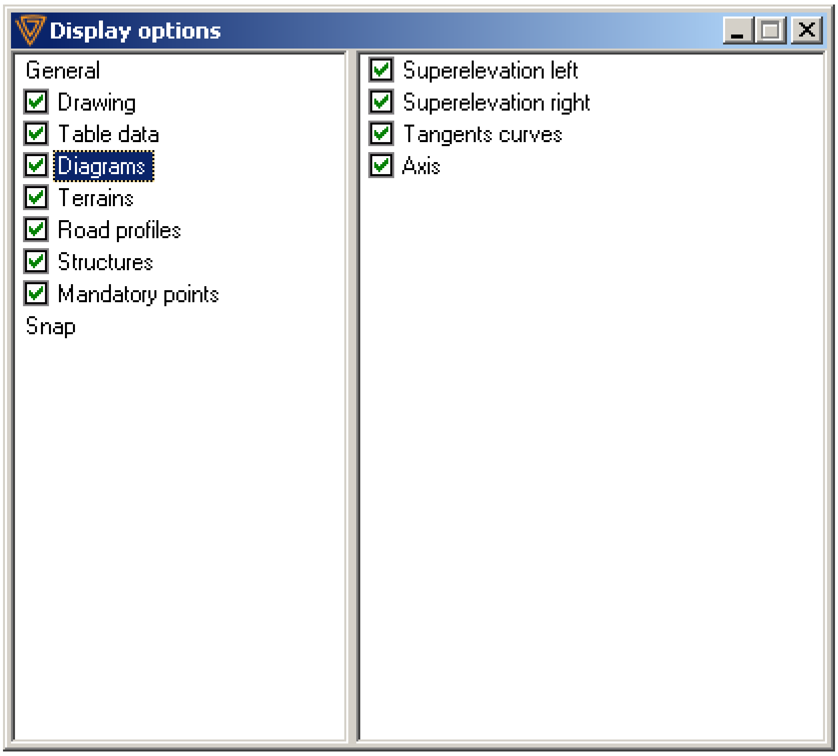
- *Polyline Points:* Shows or hides the polyline points.
- *Layers:* In this field you can see all the available layers of the drawing and you can choose to show them or hide them.



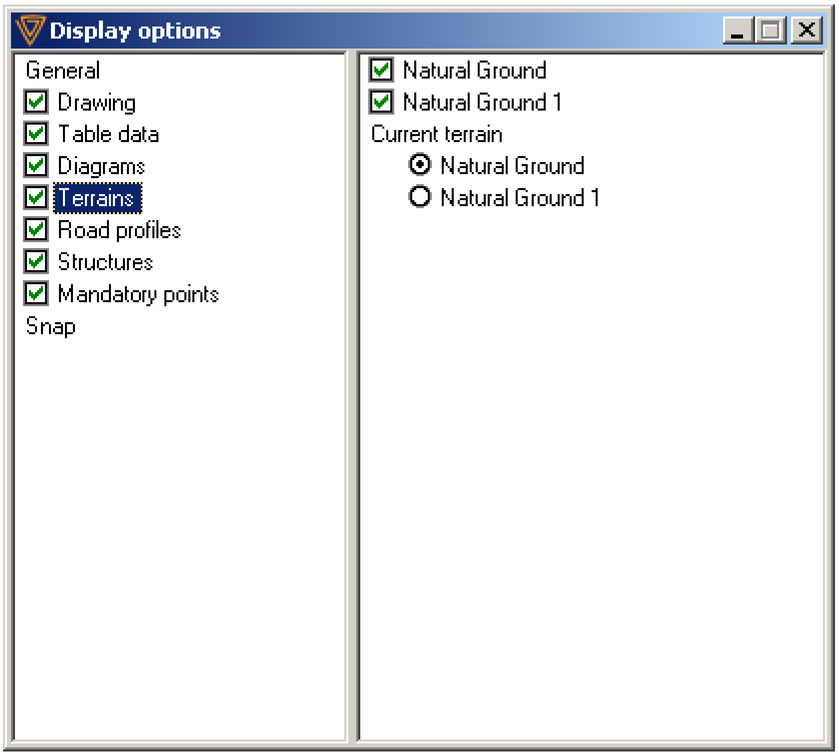
When the *Table* data option is ticked, the available options are:

- *Station name:* Shows or hides the station names axis.
- *Chainage:* This axis shows the Chainage per point.

- *Terrain Elevation:* The terrain elevation per point.
- *Terrain slope %:* The terrain slope between successive cross sections.
- *Distance between (m):* The distance between successive cross sections.
- *Road profile elevation (m):* Shows the road profile elevation per point.
- *S:* The longitudinal slope of the road per point.
- *S+DS:* The longitudinal slope of the roadlines per point.
- *Distance:* The distances between successive points of the tangent curve.
- *Tangent curve slope:* The slopes between successive points of the tangent curve.
- *Oblique slope left:* The oblique left slope per point.
- *Oblique slope right:* The oblique right slope per point.
- *Elevation discrepancy:* The elevation discrepancy between road profile and terrain.
- *Superelevation right:* The values of the right superelevations per point.
- *Superelevation left:* The values of the let superelevations per point.
- *Display data:* This option refers to all of the aforementioned data that is shown per point. You can choose to display this data:
  - *Per cross section:* The data is shown per cross section chainage.
  - *Interpolate every (m):* The data is shown per fixed distance, which is set in the immediately next field.
    - *Hide successive identical values:* In the case of interpolation, select it to hide the similar values for successive points.

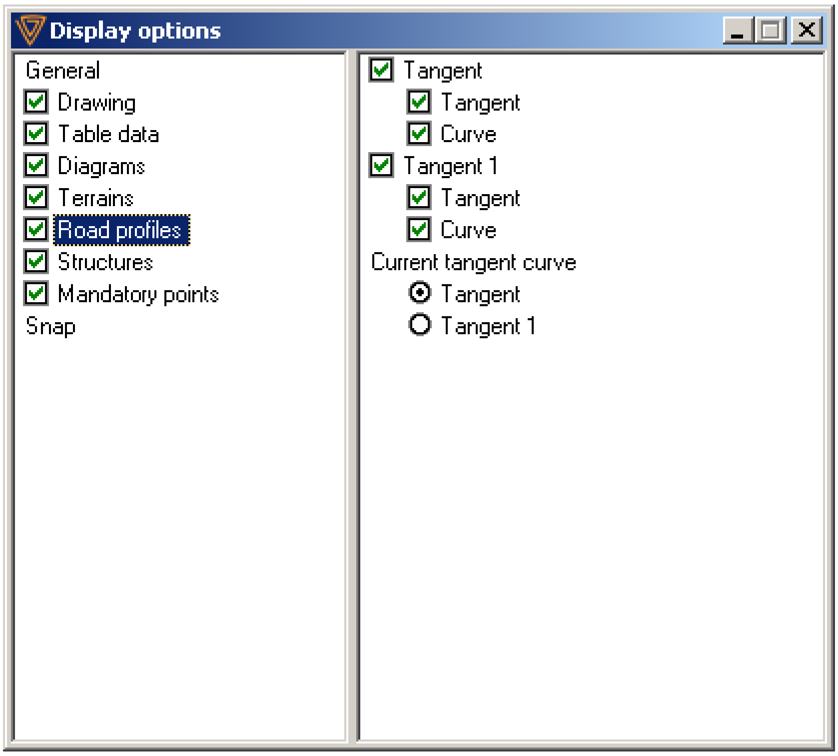


When the *Diagrams* option is ticked, the available options are:
- *Superelevation left:* Shows or hides the left roadline of the superelevation diagram.
- *Superelevation right:* Shows or hides the right roadline of the superelevation diagram.
- *Tangent curves:* Shows or hides the tangent curve diagram.
- *Axis:* Shows or hides the axis of the superelevation diagram.



When the *Terrains* option is ticked, the available options are:

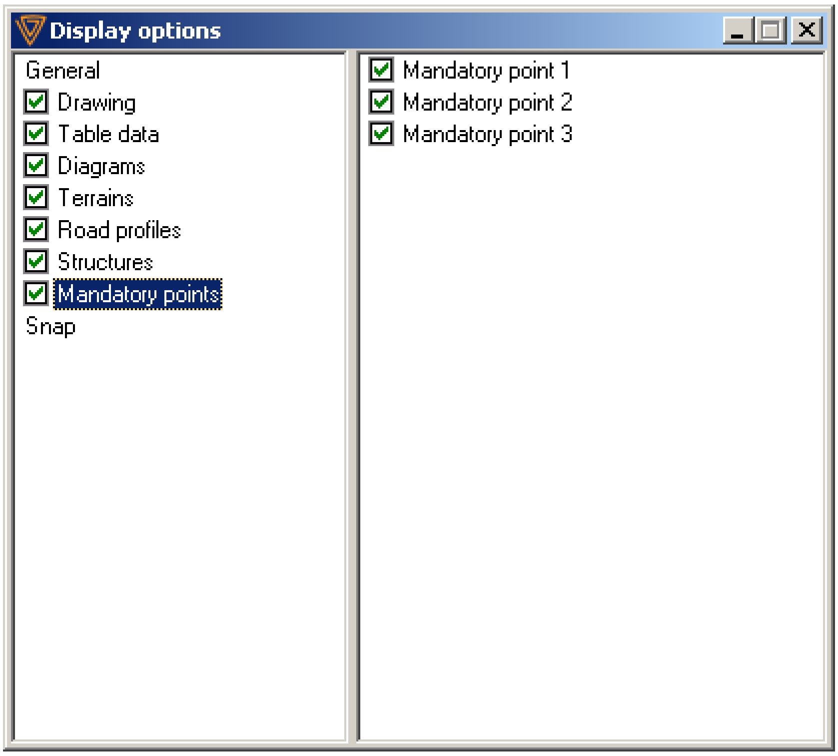
- *Natural Ground:* Shows or hides the Natural ground line.
- *Current terrain:* Select the terrain whose data will appear on the screen (e.g. in the data tables below the drawing). In the window shown in the figure above, both grounds are selected but the current terrain is Natural Ground 1.



When the *Road profiles* option is ticked, the available options are:

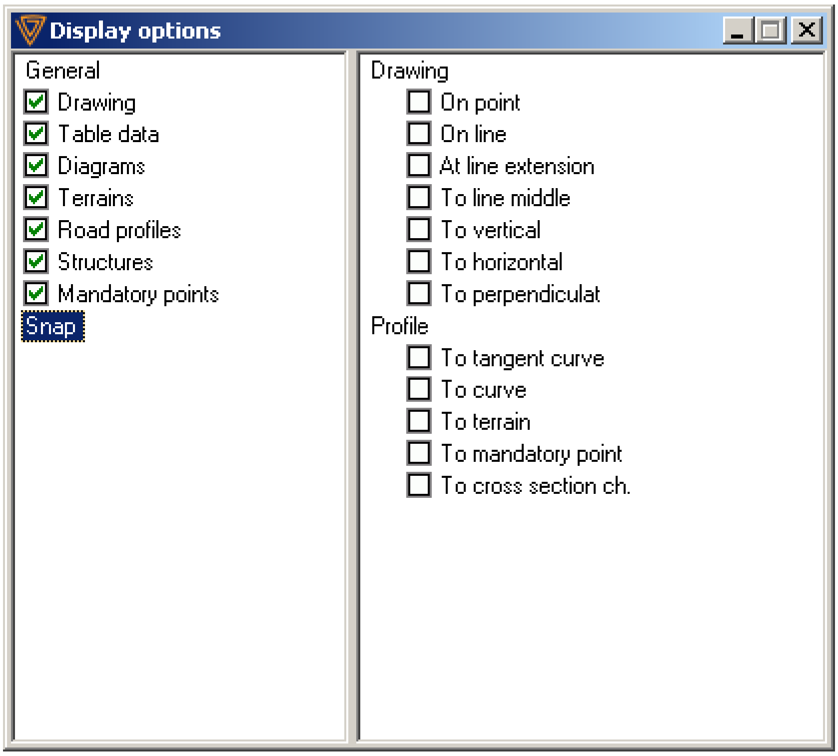
- *Tangent:* Shows or hides the tangent curve and road profile set.
  - *Tangent:* Shows or hides the tangent curve of the road profile.

- *Curve:* Shows or hides the road profile curve.
- *Current tangent curve:* Select the tangent curve whose data will appear on the screen (e.g. in the data tables below the drawing). The current tangent curve will be the one that will share its data with the Horizontal Alignment and the Cross Sections during the update procedures.



When the *Mandatory points* option is ticked, the only available option is:

- *Mandatory point ...:* The names of all mandatory points that have been inserted appear on the right part of the window. From there, you can show or hide the desired points.

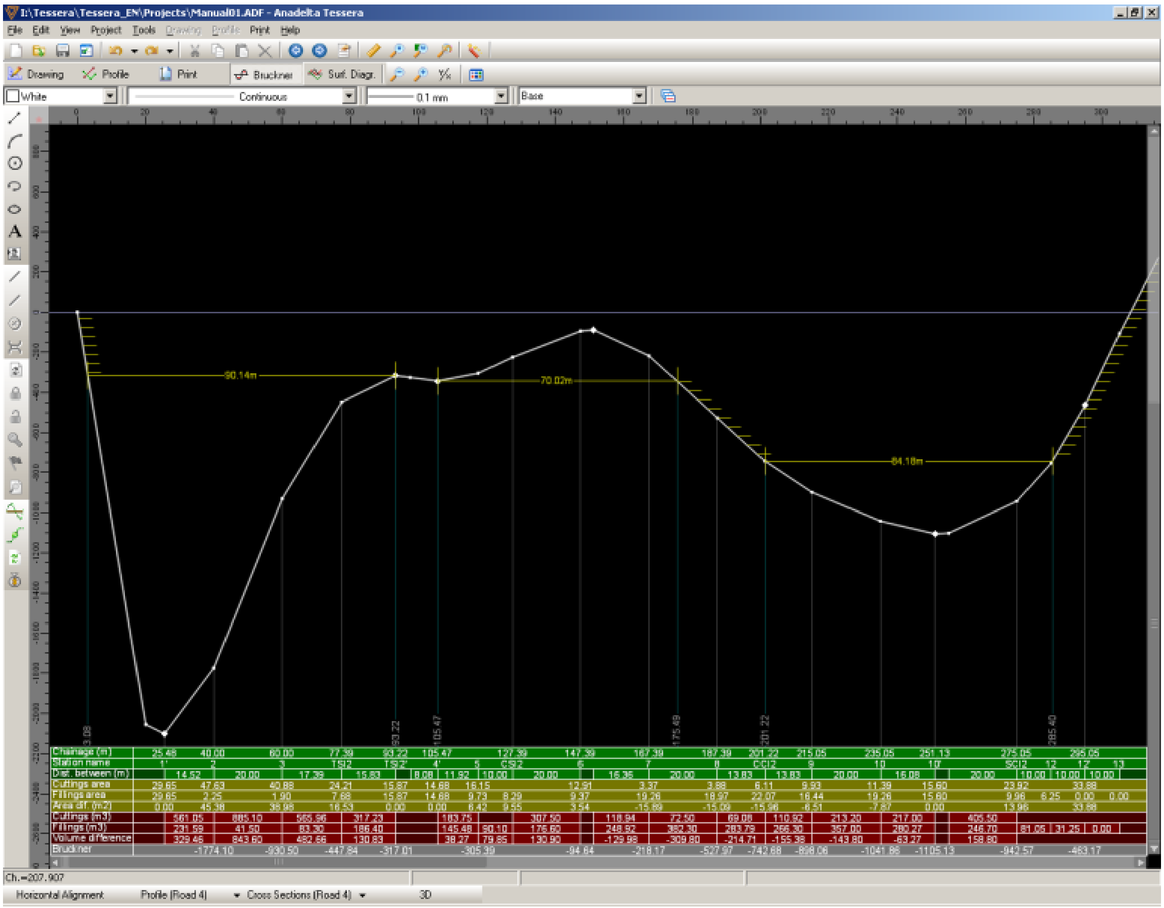


When the *Snap* category is ticked, all fields that correspond to objects that can be snapped with the mouse pointer appear on the right part of the window.

8.5 Bruckner Mode

In this mode you can edit the data of the mass haul diagram (Bruckner) and insert the haul lines. To go to this mode, click the relevant 

button as shown in the figure below. The drawing area displays the *Bruckner* diagram, as this derives from the cross section data.



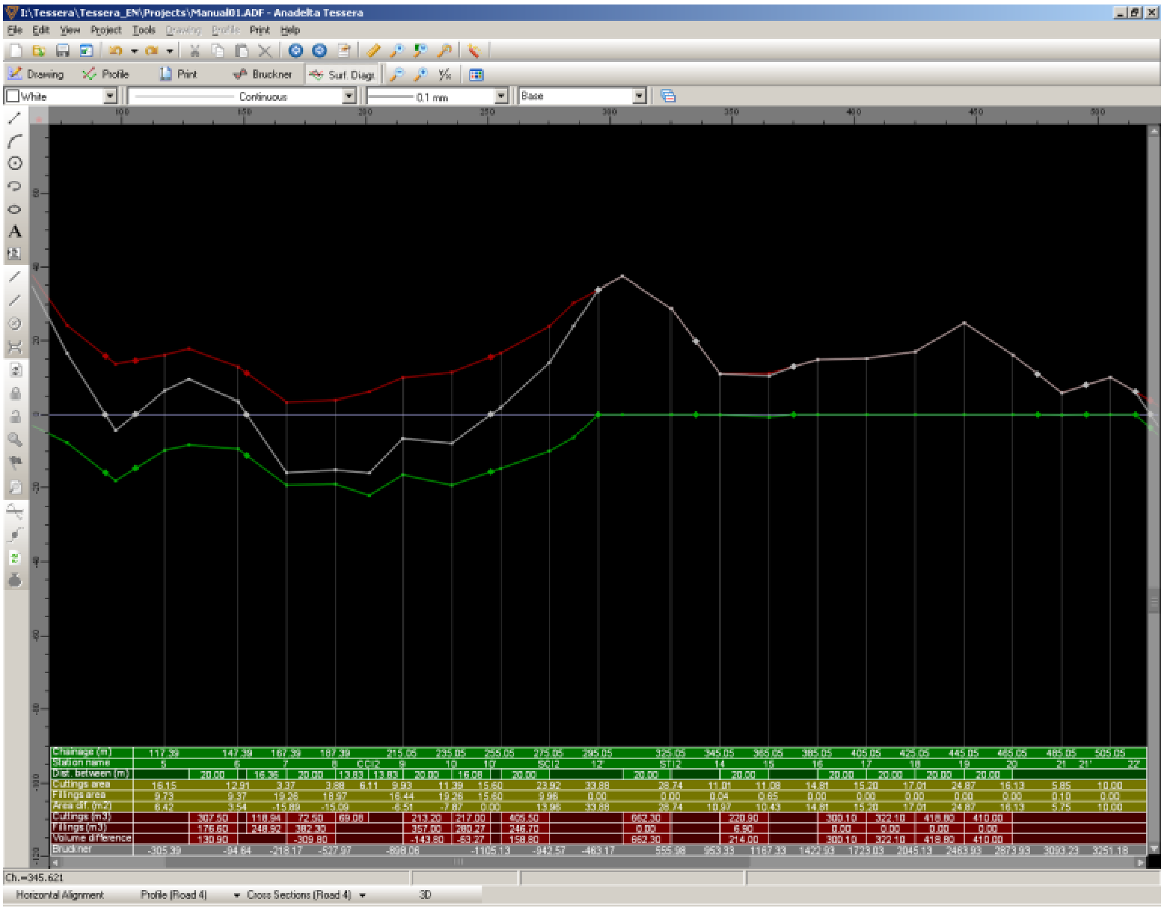
In the *Bruckner* mode, you can see the line of the algebraic sum of the cubes. Every point of this line has a set of coordinates (x, h) that corresponds to the Chainage and to the algebraic sum of the available masses (cuttings – fillings) from the beginning of the road. The cross section points can be seen on the screen and they are surrounded by a small square frame (true cross sections) or a rhombus (interscalar cross sections, inserted automatically into the surface diagram). The haul lines on the other hand, are horizontal lines that intersect with the cube line at two points at minimum. You can insert more than one haul line. The graphical representation of the lines is displayed in the drawing area. You can move left, right, up and down within the drawing area by using the respective arrow keys or the mouse. To select a haul line, left-click on it. The color of the line changes from yellow to blue.

At the bottom of the drawing area and below the surface graph, you can see the data axes or tables of the various lines whose names are displayed on the left. The tables are: *Station chainage, Station name, Distance between successive cross sections, Cuttings area, Fillings area, Self-Distributions (m2), Cuttings (m3), Fillings (m3), Volume difference* and *Bruckner*. The first three tables are green, the next three tables are yellow, the last three axes are red and the last one is gray.

8.6 Surface Diagram Mode

In this mode, the drawing area displays the road surface diagram, as this derives from the measurement of the cross section areas. To go to this mode, click the relevant

, button as shown in the figure below.



In the *Surf. Diag.* Mode, you cannot create or edit any of the cross sections that you see. In the drawing area there are three lines: the cuttings line, the fillings line and the line of the sums. The cuttings line, which is displayed in red color, is comprised of sets of coordinates (x, y), which represent the station chainage and the area of the station cutting (in m2). Respectively, the fillings line is displayed in green color and its ordinate corresponds to the area of the station filling (in m2). The aforementioned two lines represent the areas before deducting self-distributions. By default, the area of the cuttings is positive and the area of the filling is negative. The line of the sums on the surface diagram after the deduction of self-distributions is displayed in gray color and corresponds to the algebraic sum of the aforementioned two lines.

The line points are displayed on the screen and they are contained in a small box to be more distinguishable. At the points where, either the line of the sums intersects the horizontal axis, or the line of the fillings or the line of the cuttings becomes zero (shifting from a cross section that lies entirely in a cutting or filling to a mixed cross section and vice versa), the program inserts automatically an interscalar cross section, which does not correspond to any of the cross sections that are displayed in the *Cross Sections* workspace. This serves for the more accurate calculation of the mass haul diagram (Bruckner). All lines have points at the same places.
The axes or tables at the bottom of the drawing area are the same as those that are displayed in the *Bruckner* mode.

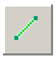
8.7 Terrain Profile Line

If you have created a profile project without having the *Horizontal Alignment,* or if you wish to insert a second terrain line which will correspond, e.g. to the terrain profile below a roadline, then you must insert the line points in one of the ways that are described below.

Creating a Terrain Line with the Mouse

You can create a terrain line directly in the *Anadelta Tessera* environment by setting its points one by one with the mouse.

To create a terrain line with the mouse

1. From the *Profile* menu, run the *Insert Terrain* command, or from the *Edit* toolbar, click the  button (green).
2. Use the mouse to define the positions of the terrain line points within the drawing area. To define a profile more accurately, use the *Input Form* dialogue box. If the line that you are inserting is the first one, then the axes referring to this line and its points (namely the cross sections of the road) are also displayed. In the case of secondary terrain lines, then you're actually defining only the elevations, since the cross sections and their in-between distances have already been set when you inserted the first terrain line.
3. When you have finished, press *Esc* or right-click anywhere on the screen to complete the insertion of points.

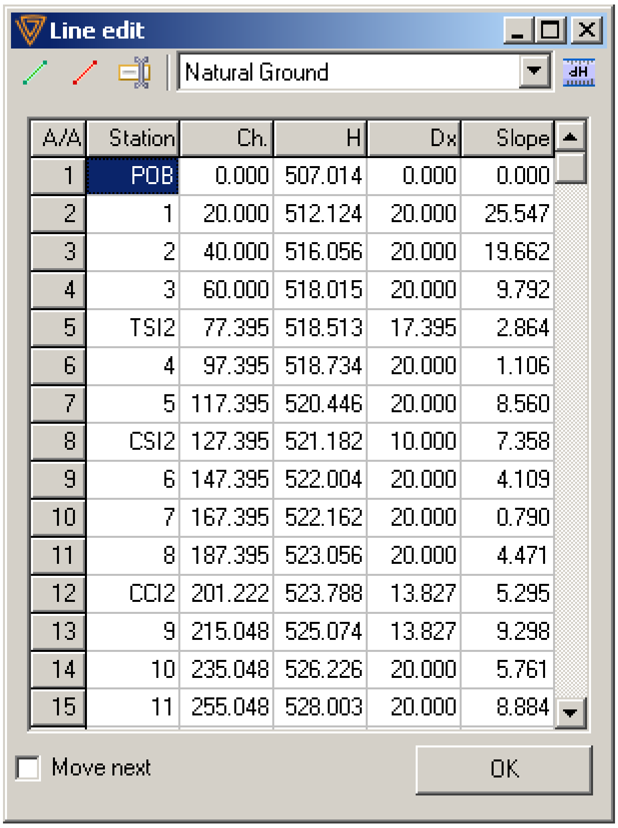
The profile and the relevant data on the axes are displayed on the screen

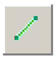
Creating a Terrain Line by Typing Absolute or Relative Coordinates

You can insert a line by typing its data on a list.

How to create a Terrain line from the window of the Line edit command

1. From the *Profile* menu, run the *Line edit* command. The following window appears on the screen:



2. Click the  (button (green) at the top left of the window. A dialogue box appears on the screen, where you must type a name for the new curve.

3. When you have finished typing, click the *OK* button to return to the dialogue box and start typing the points.

The main section of the dialogue box is comprised of a table with six columns and a variable number of rows. Each row corresponds to a cross section. The titles of the columns are shown in gray on the top. The columns are: *A/A, Station, Ch., H, Dx* and *Slope%.* The program fills in the A/A column automatically with the serial number of the cross section and the user cannot intervene. In the other columns you can enter the cross section data. If you haven't inserted any other terrain line yet, then only the first row of the table will be filled. If there is already a natural ground, then this is considered as the main one and the data in the Station and Ch. columns will be copied to the new ground. In this case, you cannot edit the data of the row in the *Station, Ch.* and *Dx* columns. In other words, the Chainages of the new line are always the same as the ones of the main line and you can only intervene to the elevation of the line at the station.

In the *Station* column, enter the name of the station you wish to be displayed at the profile axis. The positions of the points can be defined in one of the following ways:

1. If you know the chainage and the elevation of the points that will comprise the line, type them in the *Ch*, and *H* columns respectively.
2. If you know the distance of a point from its previous point, type it in the *Dx* column.

3. If you know the slope of a point with respect to its previous point, type it in the *Slope%* column.

When you change the data of one cell, the other cells of the row are updated automatically.

To select a field, left-click in it and then start typing. If you wish to restore the data of the cell, press the *Esc* key. Press the *Enter* key to confirm the data you have entered and to move to the next field on the right. You can also move between the fields by using the up, down, right and left arrow keys. If you are at the end of a row, you can press the *Enter* key to go to the first field of the next row.

Regarding the main line of the terrain, if you wish to insert a new station at the end of the table, go to the sixth column of the last station and press the *Enter* key. If you want to insert a new station between two existing stations, select the immediate next station and press the *Ctrl+I* keys. The new station is inserted automatically in the middle of the distance between the selected station and its previous one. If you wish to delete a station, simply select the desired station and press the *Ctrl+D* keys.

Importing Terrain Lines from a Text File (.txt)

If the points of a terrain line are available in digital form (ASCII text files) and you wish to import all of them to your project along the road axis or parallel to the road, you can use the specialized operation of batch point insertion. This method is very useful when you wish to transfer a profile from one file to another.

How to import the Natural Ground from an ASCII file

1. From the *File* menu, select the *Import from ASCII* submenu and then run the *Terrain Profile* command. The *Open* dialogue box appears on the screen.
2. Select the desired file and click *Open*.

The text files that the program searches to find have the .txt extension and must be comprised of at least three columns.

Line "Natural Ground"
Starting chainage : 0.000

PI	Chainage	Elevation	Tang. Curve Elev.	Elev. diff.
POB	0.000	507.014	510.099	3.085
1	20.000	512.124	512.186	0.063
2	40.000	516.056	514.273	-1.783
3	60.000	518.015	516.360	-1.654
TSI2	77.395	518.513	517.956	-0.557
4	97.395	518.734	519.085	0.351
5	117.395	520.446	520.199	-0.248
CSI2	127.395	521.182	520.755	-0.427
6	147.395	522.004	521.869	-0.135
7	167.395	522.162	522.982	0.821
8	187.395	523.056	524.097	1.041
CCI2	201.222	523.788	524.872	1.084
9	215.048	525.074	525.653	0.579
10	235.048	526.226	526.790	0.564
11	255.048	528.003	527.937	-0.065
SCI2	275.048	529.551	529.095	-0.456
12	285.048	530.691	529.677	-1.014
13	305.048	532.651	530.841	-1.810
STI2	325.048	533.257	532.000	-1.257
14	345.048	533.340	533.078	-0.262
15	365.048	534.288	534.076	-0.212
16	385.048	535.624	534.985	-0.640
17	405.048	536.000	535.427	-0.573
18	425.048	536.000	535.358	-0.642
19	445.048	535.611	534.779	-0.832
20	465.048	534.092	533.788	-0.304
21	485.048	532.573	532.760	0.187
22	505.048	531.961	531.732	-0.229

These columns are the station name, the station Chainage and the station Elevation. The first rows of the file may contain comments on the starting chainage and the name of the line, but the program will ignore them. The rows containing the data are separated by a row that must contain at least three consecutive dashes (___). In case that the terrain line that you wish to import is the main one, the program will read the contents of the second column. In the opposite case, the program shall ignore them and the points of the new line will have the same distances as the ones of the main line files, while their elevations will be those that the program has read from the file. Obviously, the file must have as many stations as the already existing Natural Ground.

If the file does not comply with the aforementioned requirements, then the program will not be able to read the file and will display the relevant error message.

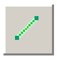
Copying a Terrain Line

You can create a copy of a terrain line with elevation offset.

How to copy a terrain line

1. Select the desired terrain line and from the *Profile* menu run the *New terrain with Offset* command, or from the menu which is displayed if your right-click anywhere in the drawing area, run the *Copy with offset* command.
2. The program will prompt you to enter the elevation of the first station of the new line, showing you the message *Enter profile point on the Status bar*. Enter the new point either by clicking the left mouse button in the drawing area or by using the *Input Form* command.

The profile of the new curve will be displayed on the screen. The other points of the curve will be inserted with respect to the position of the first station.

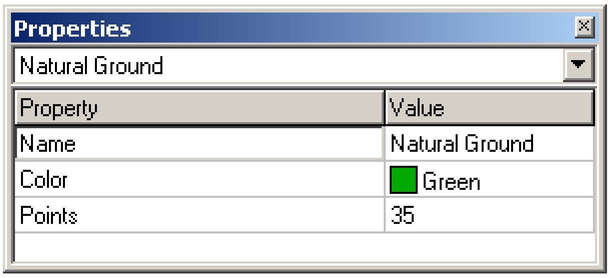
You can also copy the main line of the natural ground from the window of the *Line edit* command. Just click the  button. A dialogue box appears on the screen, where you must type a name for the new curve. Click the *OK* button. The program will create a new terrain line, which will be a true copy of the main line, without any elevation offset. Next, you can proceed to the offset by clicking the dH button at the top of the window, and type the value in the *Move line by Dh,* as described in the following section.

8.8 Terrain Profile Editing

You can edit the Terrain lines that you have created, so as to give them the desired form.

Changing the Name of a Terrain Line

To change the name of a terrain line, from the *Profile* menu run the *Line edit* command and then click the *Name Change* button. In the resulting dialogue box, type the new name. Next, click the *OK* button.



You can also edit the name of a line from the window of the *Properties* command, which is displayed if you click the right mouse button after you have selected the line you wish to rename. Simply run the command to open the window and in the field that corresponds to the *Name* command, type the desired name.

Moving a Terrain Line

You can move a terrain line with respect to the elevation.

To move a line with the mouse

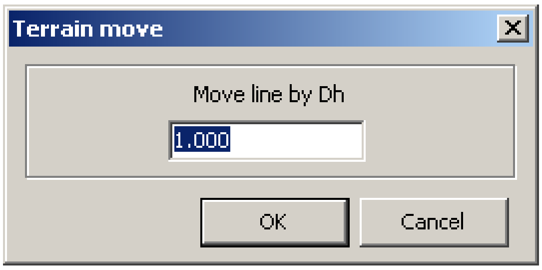
1. Select the line you wish to move.
2. From the menu that is displayed when you right-click in the drawing area, run the *Move* command.
3. The program will prompt you to set a reference point, by showing you the relevant message in the third section of the status bar. Set a reference point either by clicking the left mouse button in the drawing area or by using the *Input Form* command.
4. In the next step, the program will prompt you to set an end point, by showing you the relevant message in the third section of the status bar. Proceed as described above. The line will be transferred to the position you have specified.

This type of transfer has the meaning of vertical shift. Therefore, although you can specify both points with a different chainage, only their elevation discrepancy is taken into account.

You can also move the line from the window of the *Line edit* command.

How to move a curve from the window of the Line edit command

1. From the *Profile* menu, run the *Line edit* command or select a terrain line and then, from the menu which is displayed if your right-click anywhere in the drawing area, run the Line edit command.
2. From the drop-down list at the top of the displayed window, select the curve (if not selected) you wish to move. The data of the selected curve are shown in the table in the middle of the box.
3. To move the curve, click the  button on the right of the drop-down list. The following dialogue box appears on the screen:



4. In the *Move line by Dh* text field, type the distance at which you wish to move the curve, entering a positive sign for moving the line upwards and a negative sign for moving the line downwards.
5. Click the *OK* button. The curve will be moved within the drawing area and the table data will be updated with the new values.

You can also move only part of a terrain line by elevation or by the X axis (in the case of the main curve).

Inserting a Point on a Natural Ground Line

You can insert one or more points on a Natural Ground line.

To insert a point on a terrain line with the mouse

1. With the mouse pointer over a point of the terrain line, select the *Terrain* line by clicking the left mouse button.
2. Select the point next to the point you wish to insert.
3. From the menu that is displayed if you right-click in the drawing area, run the *Extend to left* or *Extend to right* command, depending on which direction you wish to insert the point.
4. The program will ask you to specify the position of the new point, by showing you the relevant message on the *Status* Bar. Set the point either by clicking the left mouse button on the screen or by using the *Input Form* command.

The point will be inserted at the position you have specified. Every time you insert a point on a line, a point with the same chainage will also be inserted on every terrain line of the project, while its elevation will be calculated through a linear interpolation from the neighboring points of the lines. Inserting a new point actually means inserting a new station into the road. The new station will be named after its serial number, namely, if the terrain lines have 25 stations, the name of the new station will be 26. Of course, you can rename the station, if desired.

You can also insert a point on the main natural ground from the window of the *Line edit* command.

Moving a Terrain Line Point

You can move a terrain line point easily and quickly by using the mouse.

How to move a terrain line point

1. With the mouse pointer over a point of the terrain line, select the line you wish to edit by clicking the left mouse button.
2. Highlight the point you wish to move.
3. Specify the new position of the point. You can move the point vertically or horizontally. Regarding the horizontal offset, this is only possible up to the positions of the two neighboring stations.

At the same time, the points of the other curves on the same station will be moved horizontally, in order to get the same chainage, but they will maintain their H elevation.

You can do the same from the dialogue box of the *Line edit* command.

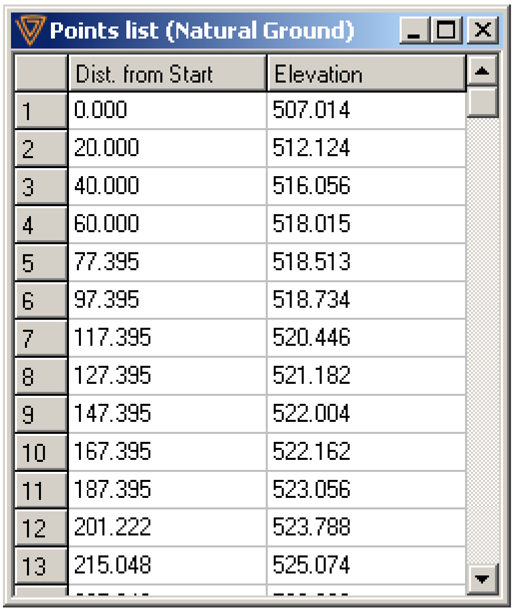
How to move a terrain line point from the window of the Line edit command

1. From the *Profile* menu run the *Line edit* command, or select a terrain line and then, from the menu which is displayed if your right-click anywhere in the drawing area, run the *Line edit* command.
2. From the drop-down list at the top of the window, select the terrain line (if not selected) you wish to move. The data of the selected line are shown in the table that is displayed in the middle of the box.
3. If the selected line is not the main one, you can intervene only in the *H* and *Slope* columns, namely you can move a point only by elevation. When the *Move next* check box is ticked, the points at the far right of the point that you're moving are also moved by the same distance. If the terrain line is the main one, you can move the points horizontally, by editing the data in the *Ch,* and *Dx* columns.

Batch Editing of Terrain Line Points

You can carry out a batch editing of the points positions either from the window of the *Points* command in the *Properties* or from the window of the *Line edit* command.

How to carry out batch editing of the profile points



	Dist. from Start	Elevation
1	0.000	507.014
2	20.000	512.124
3	40.000	516.056
4	60.000	518.015
5	77.395	518.513
6	97.395	518.734
7	117.395	520.446
8	127.395	521.182
9	147.395	522.004
10	167.395	522.162
11	187.395	523.056
12	201.222	523.788
13	215.048	525.074

1. Select the profile you wish to edit by left-clicking on it with the mouse when the pointer is on one of the profile points.
2. Display the *Properties* command window. In the value field of the *Points* property, click the button with the three dots the right of the field. The window of the *Points list* command appears on the screen. This window has two columns, which correspond to the *Distance from Start* and to the *Elevation*. Each line of the window corresponds to a line point.
3. Change the values in the desired fields by typing the new coordinates.
4. Every time you change a value, press *Enter* to confirm the new value.

Deleting a Terrain Line

You can delete very easily any terrain line you don't want to be included in your project any longer.

How to delete the selected terrain line

1. With the mouse pointer on a point of the terrain line, select the line you wish to delete by clicking the left mouse button.
2. From the *Edit* menu, run the *Delete* command, or, from the menu that is displayed if

you right-click on the screen, run the *Delete Objects* command, or press the *Del* key. If the line you selected to delete is the main one, this action also deletes all lines of the Natural Ground.

Deleting a Terrain Line Point

You can delete one or more points of a terrain line.

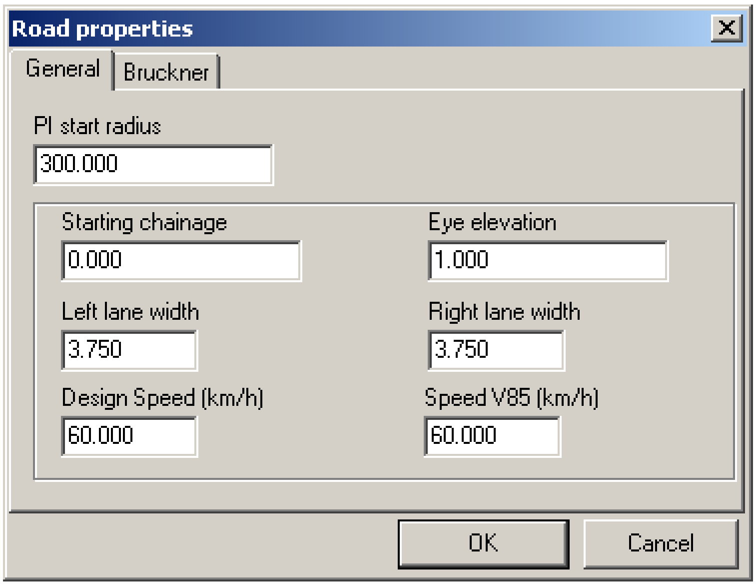
How to delete a terrain line point

1. With the mouse pointer on a point of the terrain line, select the line you wish to edit by clicking the left mouse button.
2. Select the point you wish to delete.
3. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Delete Point* command.

This action also deletes the points of every terrain line that are at the same chainage.
You can also delete a point from the dialogue box of the Line edit command, if you select the line that corresponds to the station to be deleted and then press the *Ctrl+D* keys.

8.9 Road Properties

If you run the *Road preferences* command from the *Profile* menu, the following dialogue box will appear on the screen:



The box has two tabs, where you can make various settings for the road and the Bruckner diagram.

The *General* tab offers the following settings:

- *PI start radius:* Enter the value of the PI radius that the program will give to every new tangent PI that you insert.
- *Starting chainage:* In this field you can enter the starting chainage of the road.
- *Eye elevation:* In this field you can enter the value of the elevation for the calculation of the minimum radius, in order to ensure the necessary sight distance.
- *Left and right lane width:* Enter the road's traffic width for the calculation of the Ds.
- *Design Speed:* Enter the road's design speed.
- *Speed* V^{85}*:* Enter the working V^{85} speed.

When you have finished your settings, click the *OK* button to save the changes and return to the *Profile* workspace.

8.10 Creating and Inserting a Tangent Curve

The next step in the design of the *Profile* is the insertion and editing of the road's tangent curve. The end purpose is to determine the elevation of the terrain road profile. You can insert the tangent curve PIs and their data in three ways.

1. Batch import of lines from text files (.txt)
2. Insertion of PIs within the *Anadelta Tessera* environment.
3. Conversion of a natural ground line to tangent curve.

Importing a Tangent Curve Line from a Text File (.txt)

If the tangent curve PIs and their data are available in digital form (ASCII text files) and you wish to import them into your design, you can do this anytime, either during the first stage of your design or while editing your design, and for as many times as you wish. This method is very useful when you wish to import into the program tangent curves from other files.

How to import a tangent curve from an ASCII file

1. From the *File* menu, select the *Import from ASCII* submenu and then run the *Tangent curve* command. The *Open* dialogue box appears on the screen.
2. Select the desired file and click *Open.* The text files that the program searches to find have the .txt extension and must be comprised of at least three columns. These columns are the tangent curve *Name*, its *Chainage* and its *Elevation.* The ASCII file may contain the PI rounding radii and you can import them into a column, which must be the next column after the elevation. The first lines of the file may contain comments, but the program will ignore them. The rows containing the data are separated by a row, which must contain at least three consecutive dashes (___).

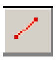
```
Line  "Tangent"
Starting chainage : 0.000
        PI      Chainage     Elevation        Radius             T             d        Slope%
------------------------------------------------------------------------------------------------
        K0       -18.497       508.169         0.000         0.000         0.000         0.000
        K1        73.210       517.739       300.000         7.302         0.089        10.435
        K2       227.916       526.351     39766.435        50.481         0.032         5.567
        K3       334.775       532.571      3779.774        16.064         0.034         5.821
        K4       413.023       536.461       783.629        39.612         1.001         4.971
        K5       588.789       527.429       599.806        33.013         0.909        -5.139
        K6       637.010       530.259         0.000         0.000         0.000         5.869
```

If the file does not comply with the aforementioned requirements, then the program will not be able to read the file and will display the relevant error message. After the file has been read, a new tangent curve will appear in the drawing area.

Creating a Tangent Curve

You can create one or more tangent curves by specifying the tangent curve PIs.

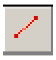
How to create a tangent curve with the mouse

1. From the *Profile* menu, run the *New Tangent Curve* command, or from the *Edit* toolbar, click the  button (red).
2. Use the mouse to define the positions of the tangent curve PIs within the drawing area. To define a PI more accurately, you can use the *Input Form* dialogue box.
3. When you have finished, press *Esc* or right-click anywhere on the screen to complete the procedure.

In this phase, you can see on your screen the tangent curve, which is displayed in red, as well as the elevations of the road profile per cross section. You can also see the road profile, if it can be calculated with the current values of the PI rounding radii. The next step is to edit the default value of the rounding radii of the tangent curve PIs. If you wish to change the default value that the program assigns to the rounding radii, go to the *Profile* menu, run the *Road properties* command and change the value in the *PI initial values* text field.

You can also insert a tangent curve from the window of the *Line edit* command.

How to create a tangent curve from the window of the Line edit command

1. From the *Profile* menu, run the *Line edit* command.
2. Click the  button (red) at the top left of the window. A dialogue box appears on the screen, where you must type a name for the new tangent curve.
3. When you have finished typing, click the *OK* button to return to the previous window and start typing the points. You will notice that the tangent curve that you have created appears highlighted in the drop-down list.

When a tangent curve is selected, the main section of the dialogue box is comprised of a table with six columns and a variable number of rows. Each row corresponds to a PI. The titles of the columns are shown in gray on the top. The columns are: *A/A, Ch, H, Dx, Slope%* and *Radius.* The program fills in the first column automatically with the serial number of the cross section and the user cannot intervene. In the other columns you can enter the PI data from left to right. It is not mandatory to fill all columns. Only the first column of the table appears filled.

To select a field, left-click in it and then start typing. If you wish to restore the data of the cell, press the *Esc* key. Press the *Enter* key to confirm the data you have entered and to move to the next field on the right. You can also move between the fields by using the up, down, right and left arrow keys. If you are at the end of a row, you can press the *Enter* key to go to the first field of the next row.

The horizontal position of a PI can be defined in the following ways:

- In the Ch. column you can specify the PI chainage.
- If you know the distance of a point from its previous point, but you don't know its chainage, type the distance in the *Dx* column.

The position of a point with respect to the elevation can be defined in the following ways:

- In the *H* column you can specify the PI elevation.
- In the *Slope%* column you can type the slope of the point with respect to its previous point.

In the *Radius* field you can type the value of the PI rounding radius. When you enter data in one of the table's cells, the other cells of the same row are updated automatically.

Regarding the tangent curve PIs, if you wish to insert a new PI at the end of the table, select the last field of the last PI and press *Enter*. If you wish to insert a new PI between two existing ones, select the immediate next PI and press the *Ctrl+I* keys. The new PI will be inserted right above the selected one and at the half distance. If you wish to delete a PI, simply select the desired PI and press the *Ctrl+D* keys.

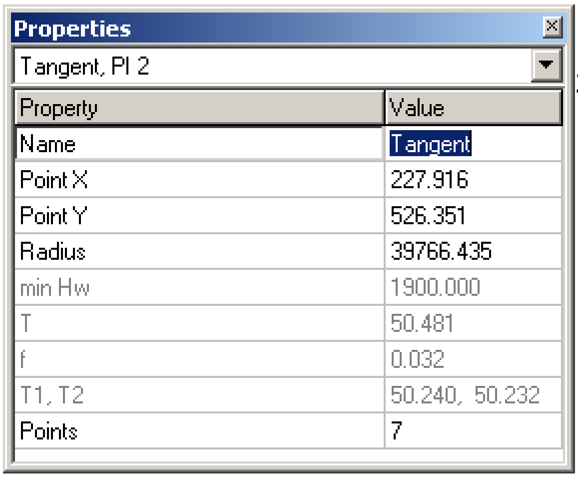
Conversion of a Terrain Line to Tangent Curve

To convert a terrain line to a tangent curve, select the desired terrain line and from the menu that is displayed if you right-click in the drawing area run the *Copy terrain to new tangent curve* command.

Inserting and Editing Tangent Curve Radii

To complete the calculation of the road profile, you must insert the rounding radius for every PI. This is done from the *Properties* command window.

How to insert the value of a PI rounding radius



1. Select (highlight) the desired PI and run the *Properties* command.
2. From the window of the *Properties* command enter the value of the rounding radius in the*Radius* field. In this window you can also view the minimum curvature radius of the curve, which ensures the required sight distance. The minimum radius is calculated based on the value you have entered in the *Eye elevation*, in the *Road preferences* of the *Profile* menu.

When you close the screen window, the road profile appears rounded according to the value of the curvature radius, but only if this value is compatible with the other data (namely, only if the curvature radius does not overlap the curvature radii of the adjacent PIs).

You can also edit the curvature radii from the dialogue box of the *Line edit* command. Select the desired tangent curve and in the last column, type the new radii values.

Finally, when a tangent line has been calculated, you can change the curvature radius of a PI and view the result in real time. If you select with the left mouse button the flags of the desired PI, you will notice that their color changes. You can increase or decrease the PI radius by left-clicking the (left or right) flag. Move the mouse to the left or to the right to increase or decrease the flag, respectively. Finally, click again the left mouse button to finish the procedure, or click the right mouse button to restore the flags to their original position. The yellow flags indicate overlapping of successive circular arcs which are not displayed.

8.11 Editing a Tangent Curve

You can edit the tangent curves that you have inserted, so as to give them the desired properties.

Changing the Name of a Tangent Curve

To change the name of a tangent curve, click the  button from the window of the *Line edit* command. A dialogue box appears on the screen, where you must type a new name. Next, click the *OK* button.

You can also edit the name of a tangent curve from the window of the *Properties* command. Simply run the command and in the *Name* field, type the desired name.

Moving a Tangent Curve

You can move a tangent curve both horizontally (Chainage) and vertically (elevations). The steps to follow are those described above for moving the terrain lines. The only difference refers to the relative position of the two points that define the movement, while their difference in the horizontal direction is also taken into account (in addition to the vertical direction).

You can also move the tangent curve from the window of the *Line edit* command. The procedure is the same as the one described above for the vertical movement of terrain lines.

Adding a PI to a Tangent Curve

You can add one or more points to a terrain line. The insertion of a PI to a tangent curve will not affect the other tangent curves.

You can do the same from the dialogue box of the *Line edit* command.

Moving a Tangent Curve PI

You can move the PI of a tangent curve by applying the same procedure that you would use for the terrain lines. A PI can only be moved within the boundaries of its two adjacent PIs. The same action is possible from the dialogue box of the *Line edit* command. You can also move points from the window of the *Points* command, as well as from the window of the *Properties* command.

Deleting a Tangent Curve and Tangent Curve PIs

Delete a tangent curve or a PI in the same way that you would use to delete a terrain line or a terrain line point.

8.12 Road Profile Calculation Data

The road profile that corresponds to the tangent curve of a road is calculated automatically upon the insertion of the road's tangent curve and based on the initial rounding radii of the tangent curve PIs. The road profile is re-calculated automatically every time you edit the data of any of its PIs. To view the data of a PI, select the PI and run the *Properties* command.

In the resulting window, you can see the tangent length and the curve arrow in the *T* and *d* fields respectively.

Moreover, on the axes at the bottom of the drawing area, you can see the road profile data for every cross section.

Copying a Road Profile to a Terrain Line

You can copy the road profile of a tangent curve to a terrain line. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Copy road profile to new terrain* command. A new terrain line will be created, whose points will coincide with the road profile of the tangent curve.

8.13 Mandatory Points

The mandatory points enable the setting of the coordinates from which the road profile will pass.

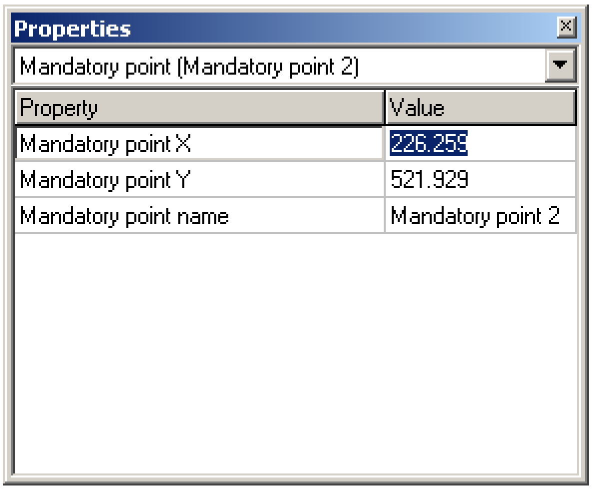
How to create mandatory points

1. From the *Profile* menu, run the *New mandatory point* command, or click the  button in the *Edit* section of the *Elements Drawing* toolbar.
2. Use the mouse to define the position of the mandatory point.

The names of the inserted points are determined automatically by the program upon insertion of the points. Select a point and right-click in the drawing area to open the *Properties* shortcut menu. Enter the new name in the *Mandatory point name* field.

Moving and Renaming Mandatory Points

How to move or rename the selected Mandatory Points

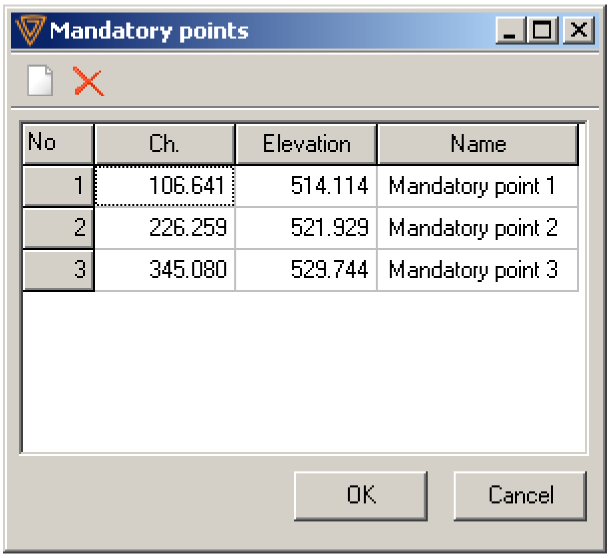


1. Select the *Mandatory Point* you wish to move, by clicking the left mouse button when the pointer is on the point.
2. Open the *Properties* command window by right-clicking in the drawing area. In the *Mandatory point name* field type the new name, if you wish to change it, and in the *Mandatory point X* and *Mandatory point Y* fields type the coordinates.

To move a point faster, select it, activate it and move it to its new position by using the mouse.

Editing Mandatory Points

From the *Profile* menu, run the *Mandatory Points Edit* command. The following window will appear on the screen:



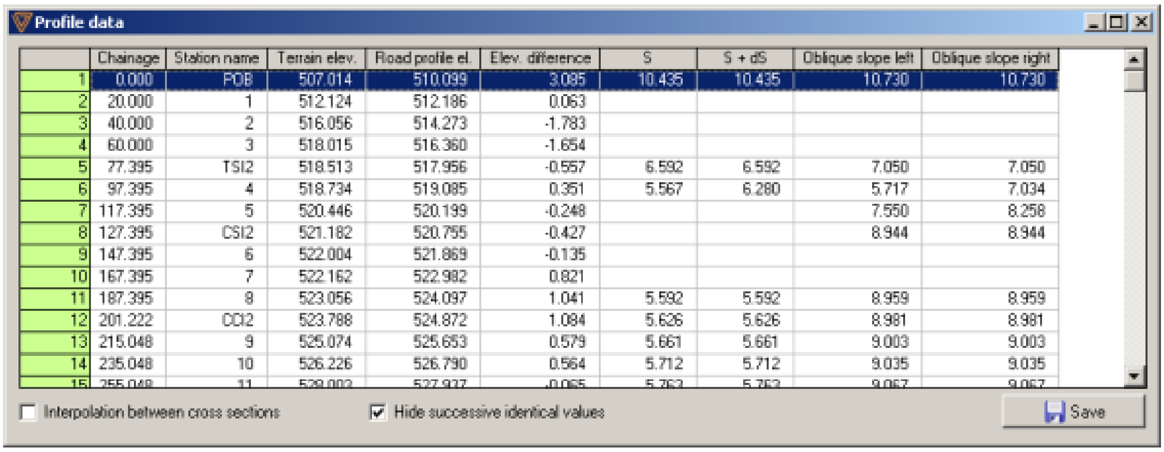
This window is comprised of the following columns:

- *No:* The serial number of every point.
- *Ch.:* The chainage of every point.
- *Elevation:* The elevation where a specific mandatory point is located.
- *Name:* The name of every point.

In this window you can edit all data of a point. If you wish to add a point and you know its *Chainage* and *Elevation*, click the respective button; an empty line will appear on the screen where you can enter the characteristics of the point. If you wish to delete a mandatory point, click the respective button.

8.14 Profile Data

The data displayed on the profile axes can be viewed in the form of a table, from the window of the *Display Data* command of the *Profile* menu.



	Chainage	Station name	Terrain elev.	Road profile el.	Elev. difference	S	S + dS	Oblique slope left	Oblique slope right
1	0.000	POB	507.014	510.099	3.085	10.435	10.435	10.730	10.730
2	20.000	1	512.124	512.186	0.063				
3	40.000	2	516.056	514.273	-1.783				
4	60.000	3	518.015	516.360	-1.654				
5	77.395	TSI2	518.513	517.956	-0.557	6.592	6.592	7.050	7.050
6	97.395	4	518.734	519.085	0.351	5.567	6.280	5.717	7.034
7	117.395	5	520.446	520.199	-0.248			7.550	8.258
8	127.395	CSI2	521.182	520.755	-0.427			8.944	8.944
9	147.395	6	522.004	521.869	-0.135				
10	167.395	7	522.162	522.982	0.821				
11	187.395	8	523.056	524.097	1.041	5.592	5.592	8.959	8.959
12	201.222	CCI2	523.788	524.872	1.084	5.626	5.626	8.981	8.981
13	215.048	9	525.074	525.653	0.579	5.661	5.661	9.003	9.003
14	235.048	10	526.226	526.790	0.564	5.712	5.712	9.035	9.035
15	255.048	11	528.003	527.937	-0.065	5.763	5.763	9.067	9.067

To save the data of the window in a text file, click the 

button. The *Save* dialogue box appears on the screen. Go to the subfolder where you wish to save your file and type the file name. Finally, click the *Save* button.

If you tick the Interpolation between cross sections check box, the program checks the profile data in the *S,S+DS, Oblique slope left* and *Oblique slope right* columns with respect to a fixed distance and shows the results by adding new rows to the table. The distance between the chainages where the check takes places is defined from the *View* menu → *Display Options* command → *Table data* → *Display data* group.

If you tick the *Hide successive identical values* check box, the program only shows the first of a column's values that are identical and in adjacent cells. Obviously, if all cells of a row are empty, the program will not show the row at all.

The aforementioned two check boxes do not affect only the data displayed in the dialogue box, but also the data of the axes displayed in the program window.

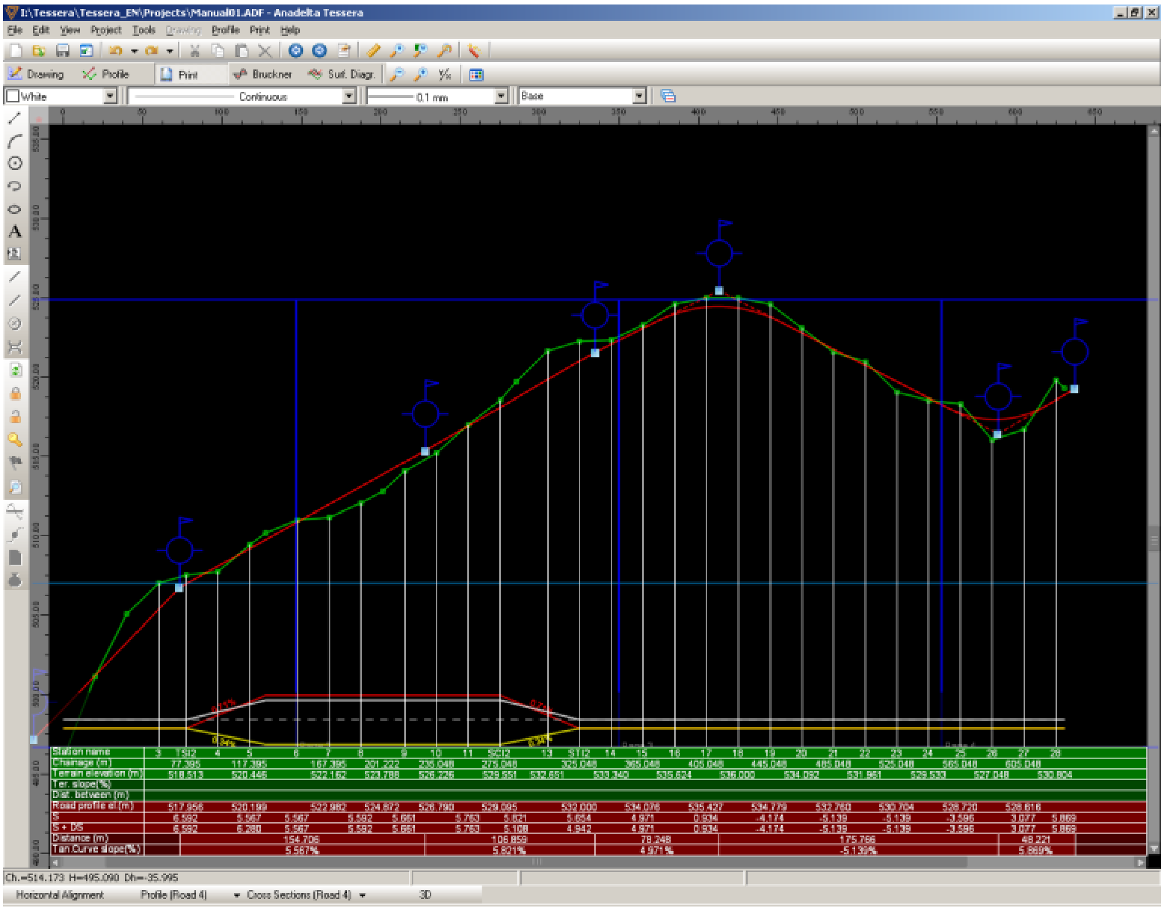
8.15 Printing

The profile printing and the creation of a freezed segment take place in a different mode than the profile editing, as we have already mentioned. To go to this mode, click the 

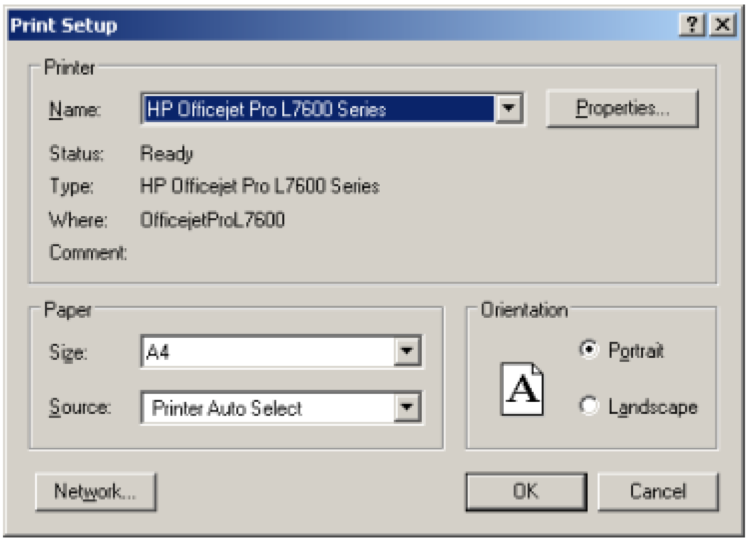
button on the *Status* bar.

In this mode you can create a freezed segment and print the profile. You can also define the positions where the flags will be displayed during printing. If you wish to return to the profile editing mode, click again the relevant *Profile* button.

This mode has many differences compared to the profile-editing mode. First of all, you cannot edit any lines. When you click the button, the *Print* group of the *Elements drawing* toolbar is activated; the print pages are displayed in the drawing area in blue, one next to the other, from left to right. The drawing must be positioned at the top of the page, between the first and the second horizontal lines.

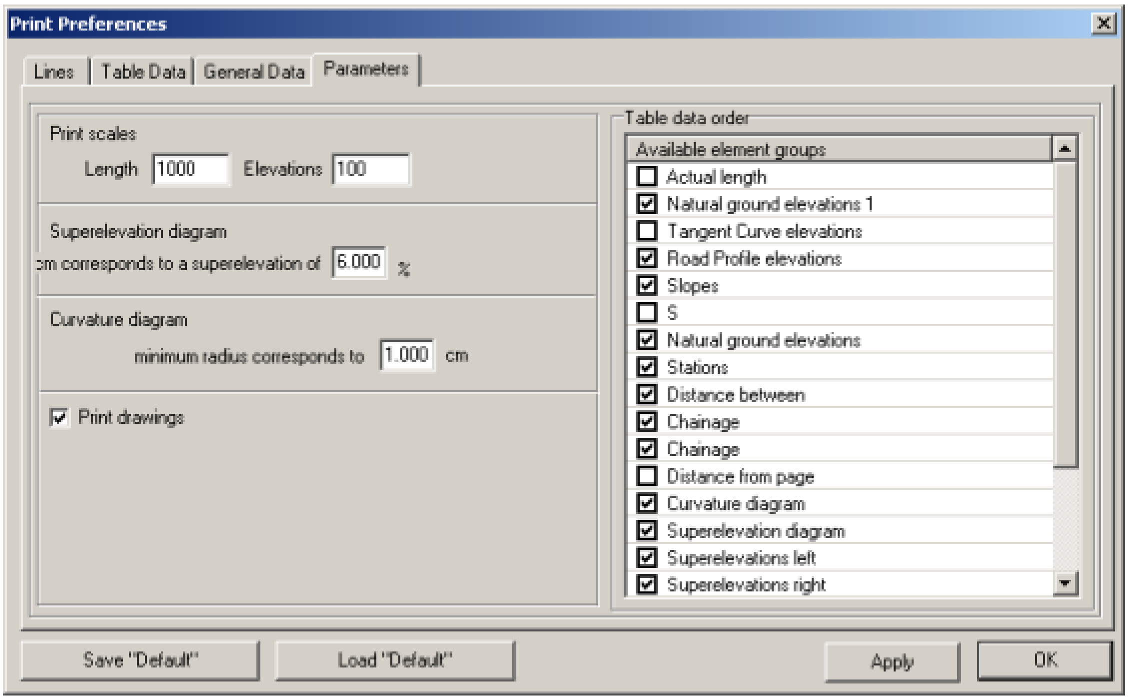


The space between the second and the third line is occupied by the data axes or tables of the profile, and some space on the left of the first page is reserved for the scales and the titles of the axes. Very often, in order for the elevation of the *Profile* to fit in the page margins, you must create a freezed segment or a break. You can have as many breaks as necessary. If there are more than one pages in the printing area, then the pages are separated by blue vertical lines. The dimensions of the print pages are proportional to the true dimensions of the paper and to the scale that has been set. You can set the true dimensions of the paper from the *Print* menu → *Page Setup* command → *Paper* group → *Size* field.



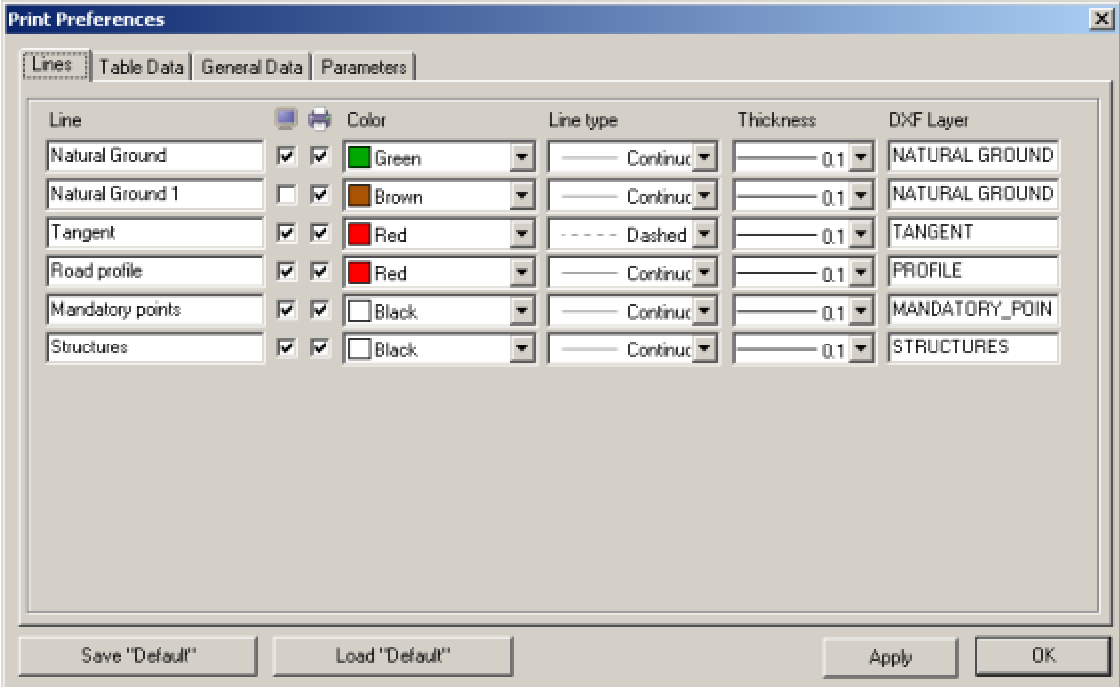
Based on these dimensions, and more specifically on the page height, the program calculates how many centimeters will be occupied by the profile drawing and how many by the data axes, always with reference to the print scale that you have set.

Use the *Profile print* preferences command of the *Print* menu to select the data that will be printed in the profile. Set the elevation and length scales from the *Parameters* tab.



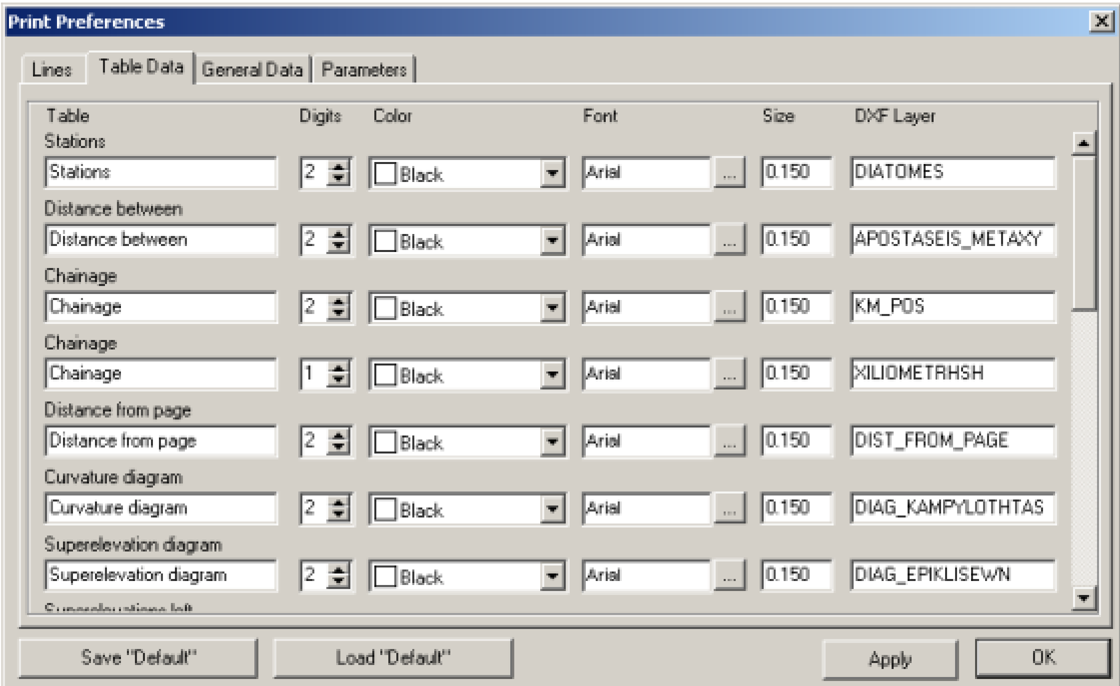
In the same tab you can set which data will be printed as axes or tables at the bottom of the page, and in which order. The available data axes or tables are shown on the right of the frame in the order they will appear on the printed profile. To select whether a table will appear or not, tick or untick respectively the check box beside the table's name. To change the order of appearance, select the table (left-click on it) to highlight it (the field fills with color) and then drag and drop the table's name up or down.

In the *Lines* tab



you can specify which lines will be printed and displayed, as well as the properties (color, line type, thickness) that the lines will have both on screen and on the printout.

In the *Table Data* tab,



you can specify the form of the data that will be printed on the axes (axis title, decimals, font color, font, size, etc.). To apply the changes you have made, click the *Apply* button. To save your settings in order to apply them to every project, click the *Save Default* button. To restore the default settings, click the *Load Default* button. To close the dialogue box and return to the *Profile*, click the *OK* button.

Creating a Freezed Segment

To be able to move within the screen of the print mode, proceed as described above. Namely, you can use the up, down, left and right arrow keys to move to the respective directions, or use the mouse wheel; to move by double-step, hold the *Shift* key down and press the respective arrow key. In this screen you can also move the whole profile or only a part of the profile in order to create a freezed segment. To do this, use the *PageUp* and *PageDown* keys.

To create a freezed segment, run the *Freeze segment* command of the *Print* menu. The whole procedure is as follows: Highlight the last point of the area to which you wish to assign a certain reference level and run the *Freeze segment* command from the *Print* menu, or click the  button, or press the *Ctrl+L* keys.

Note that a new horizon elevation appears on the selected station. Now the area before the current point is “frozen”. That means that its position on the screen will not change. Namely, if you move the drawing within the screen (*PageUp* and *PageDown* keys), the whole drawing will move up and down except for the frozen area. Repeat the freezing procedure for the other areas of the profile to create the freezed segment. The vertical step according to which the profile moves when you press the *PageUp* and *PageDown* keys also depends on the current magnification, but you can set it from the *Tools* menu → *General Options* command → *Profile* tab → *Minimum moving step for unfreezed segments* field.

The *Unfreeze segment* command of the *Print* menu “unfreezes” the (frozen) area where the current profile belongs. You can run this command by clicking the  button or by pressing the *Ctrl+U* keys; you can also run the *Unfreeze All* command to unfreeze the whole profile.

Editing Flags

In the print mode, a flag with the PI data is displayed on every PI. To move a flag you must first select it. Next, activate it (by left-clicking inside the circle of the selected flag), move the crosshair to the desired flag position and click again the left mouse button. If you don’t want a particular flag to be displayed, select it and from the menu that is displayed if you right-click in the drawing area, run the *Hide flag* command. If you want to view again the particular flag, select the respective tangent curve PI and from the menu that is displayed if you right-click on the screen, run the *Show flag* command. To delete all flags, go to the *Print* menu, open the *Flags* secondary menu and run the *Delete All* command. To restore the deleted flags, run the *Reset all* command from the same menu.

If you have moved a flag, you can align it, in order for the flag to get the same chainage as the tangent curve PI. To do this, select the flag and run the *Align by Ch.* command from the menu that is displayed if you click the right mouse button. You can do the same for all stations by running the *Align all by Ch.* command. If you want, apart from the horizontal alignment, to bring all flags to the elevation of the selected one, from the same menu run the *Align all by Ch., H* command; if you wish to align all flags so that their distance from the PI is the same as the one of the selected flag, from the same menu run the *Align all by Ch., Dx* command.

Preview and Print

By running the *Print profile* command of the *Print* menu, you can preview and print the profile.

To print from the *Print* profile window, where you can preview a page to be printed, click the last button with the printer icon. In the *Print* dialogue box, make the desired settings. Click the *OK* button to start printing.

8.16 Export to DXF File

From the *Profile* workspace you can export four types of DXF files. These files contain the Profile (with or without the Superelevation Diagram) or the separate Superelevation Diagram or one of the Surface Diagrams or the Bruckner Diagram.

How to export data to a ASCII file

1. From the *File* menu and the *Export to DXF* submenu, select *Profile* or *Superelevation diagram* or *Bruckner diagram* or *Surface diagram*.
2. Enter the name of the file in the dialogue box of the *File* Name field.
3. In the *File Type* field, select the type of the file according to the desired version, *14* or *2000*.
4. In all cases, click *Save* to finish the creation of the file.

To set the form of the elements and how they will be displayed, as well as which elements will be exported to your file and to which layer, go to the *Print* menu and open the dialogue box of the *Profile Print Preferences* command, in the case of the *Profile*, or the *Superelevation Diagram Print Preferences* command in the case of the Superelevation Diagram.
For this action, it is best to preview first the whole drawing or part of it. Before exporting the file, you must have also set the proper drawing scales by which the data font size will be determined. Throughout the program, the sizes of the data (names, terrain point elevations, PI names, station names and any other text) are given in cm. In a DXF file this size is translated into "universal" m. The conversion takes place by using the drawing scale.

8.17 Export to ASCII File

To export a curve or a tangent curve or any other element to an ASCII file, go to the *File* menu and from the *Export to ASCII* submenu run the *Terrain profile* or *Tangent curve* or *Bruckner diagram* command. The *Save as* dialogue box will appear on the screen. In this box you must specify the path and the name of the file that will be created. When you have finished, click *OK* to create the file. To export *Drawing elements, Line types* and *Hatches,* follow the instructions for the respective commands of the *Horizontal Alignment*.

8.18 Printing the Superelevation Diagram

You have the option to print the superelevation diagram separately. Its form is determined according to the selections you have made in the dialogue box of the *Superelevation diagram print preferences.* To print the diagram, run the *Print superelevation diagram* command of the *Print* menu.

8.19 Bruckner Diagram

When you go to the *Bruckner* mode, the Bruckner diagram is displayed in the drawing area. The Bruckner toolbar is also activated at the left of the screen. The elements of the diagram derive from the quantities that were calculated during the cross section calculation in the *Profile* workspace. If the cross sections change, when you enter the *Bruckner* mode, the program will show you the relevant confirmation message for updating the diagram. If you wish to update the *Bruckner* diagram at any other time, click the button on the Bruckner toolbar.

At crossings with other roads, it is possible to have local increase of cuttings or fillings. In this case, the line of the cubes is vertical. To insert a crossing you must be in the *Bruckner* mode.

How to insert a crossing in the Bruckner diagram

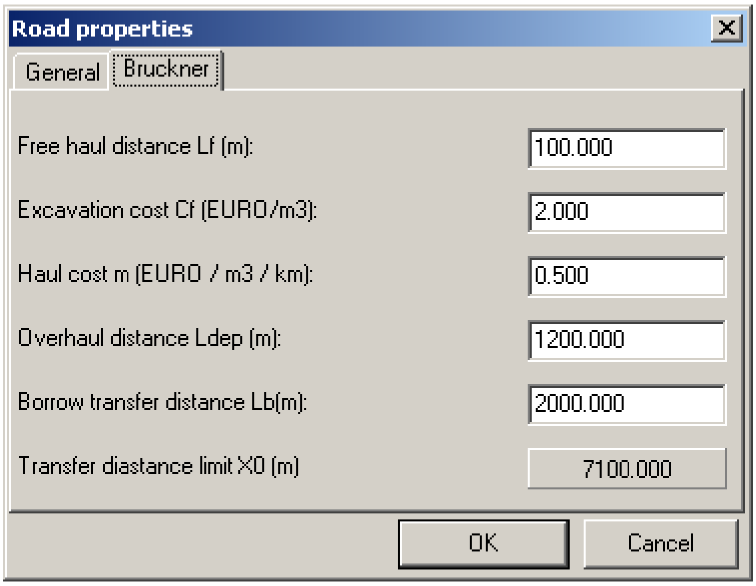
1. Click the button on the *Bruckner* toolbar. A white vertical line appears in the drawing area and moves according to the mouse movement, snapping the chainages of the diagram cross sections.
2. Left-click to select the chainage where you will insert the crossing.
3. Select the side to which the crossing will be moved, by left-clicking a point on the left or on the right of the line. Depending on where the pointer is located, a white arrow appears on top of the line, showing you which part will be moved.
4. When you move the mouse pointer, the selected part of the diagram also moves. Move the diagram to the desired position and left-click to finish the procedure.

You can cancel the insertion procedure at any time by pressing the *Esc* button.

How to insert a haul line into the Bruckner diagram

1. Click the button on the *Bruckner* toolbar. A blue horizontal line appears in the drawing area and moves vertically, following the mouse movement. The length of the line changes according to the vertical position of the line, so that the line always starts and ends on the line of the cubes.
2. Left-click to select the vertical position where you will insert the line.
3. Select the first point of the line and then the second point by clicking the left mouse button. The points where the line can stop, are points where the line intersects the diagram. When you approach a point, its color changes from white to blue. The haul line will be drawn between the two selected points.

The insertion of the haul line into the drawing area indicates the way in which the cuttings will be hauled and moved towards the fillings. Note that you can specify the critical distances of the diagram in the *Bruckner* tab of the dialogue box of the *Road properties* command.



You can also specify what the overhauls will be and where they will be hauled, or what the borrows will be and where they will be transferred from. Also, the transfer line is drawn on every concavity, so as to trim the edges of the convexes or concaves. The relevant areas in m3 and the start and end chainages of every section are also shown. You can insert more than one haul line, as long as they do not overlap horizontally.

If desired, you can change the number of concavities that a haul line crosses. Select the haul line and from the menu that is displayed when you right-click in the drawing area, run the *Line section points* command. The line will be displayed again with the maximum length it can get, and you must specify the new section points as previously.

If you are not satisfied with the position of a haul line, you can shift it vertically. Left-click on the line to select it, so that its color changes from yellow to blue, and then activate it by left-clicking it again. Next, transfer the line by moving the mouse pointer to the desired position and left-click to finalize the line position. Next, the program will prompt you to specify the start and end points of the haul line, as you did for the insertion of the line.

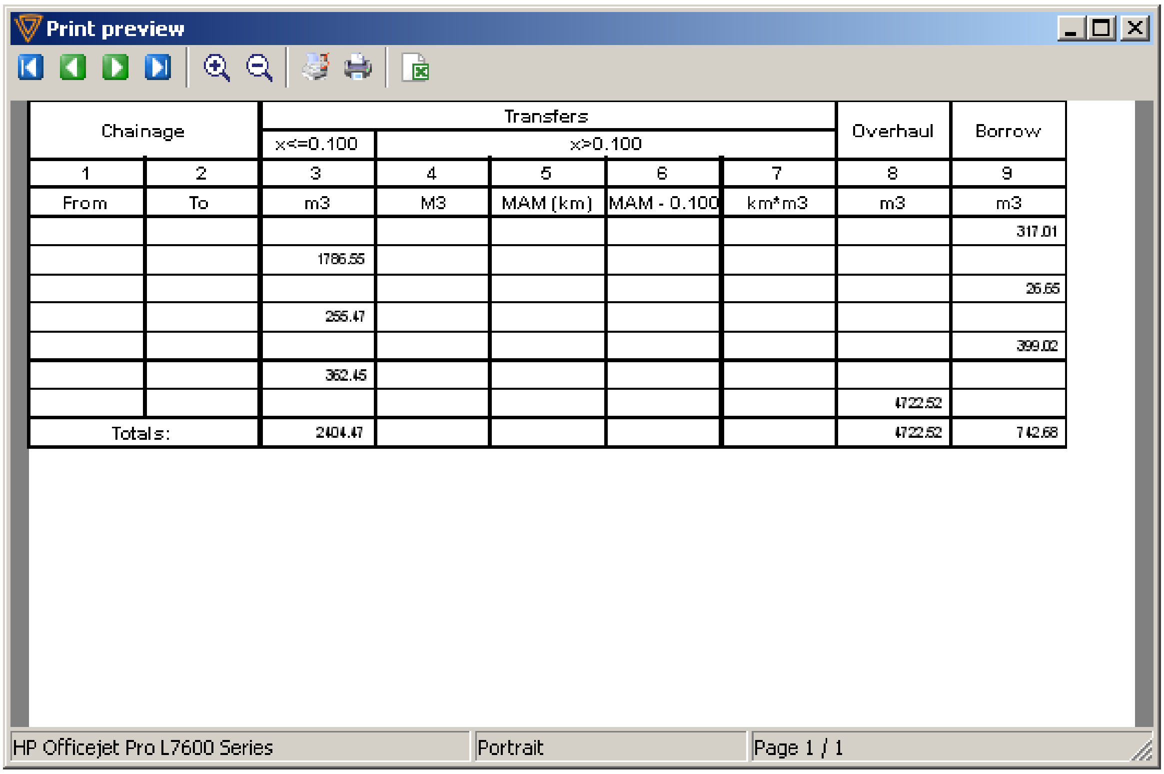
If you wish the program to move the selected haul line to the appropriate position in order to minimize the filling transfers, run the *Optimize haul line* command. To do this, click the  button on the *Bruckner* toolbar.

You can delete a haul line very easily.

How to delete a haul line

1. Select the haul line you wish to delete.
2. From the *Edit* menu, run the *Delete* command, or, from the menu that is displayed if you right-click on the screen, run the *Delete Objects* command, or press the *Del* key.

After the insertion and optimization of the haul lines, the program calculates automatically all mass hauls that are required for the construction of a road. These actions are reflected on the *Mass haul diagram* (Bruckner) and the calculations are summarized in the *Mass haul table*. To print this Table from the *Bruckner* mode, run the *Print mass haul diagram* command from the *Print* menu. The *Print preview* window will open as shown in the figure.



Chainage		Transfers					Overhaul	Borrow
		x<=0.100	x>0.100					
1	2	3	4	5	6	7	8	9
From	To	m3	M3	MAM (km)	MAM - 0.100	km*m3	m3	m3
								317.01
		1786.55						
								26.65
		255.47						
								399.02
		362.45						
							4722.52	
Totals:		2404.47					4722.52	742.68

To view the first page, click the button. To view the previous, the next or the last page, click the buttons respectively. Click the button to zoom in on the displayed page or click the button to zoom out. This feature does not affect the print size. Click the button to set the print options. Click the button to print the table. Click the button to open Microsoft Excel and to transfer the table's data into this program.

Chapter 9

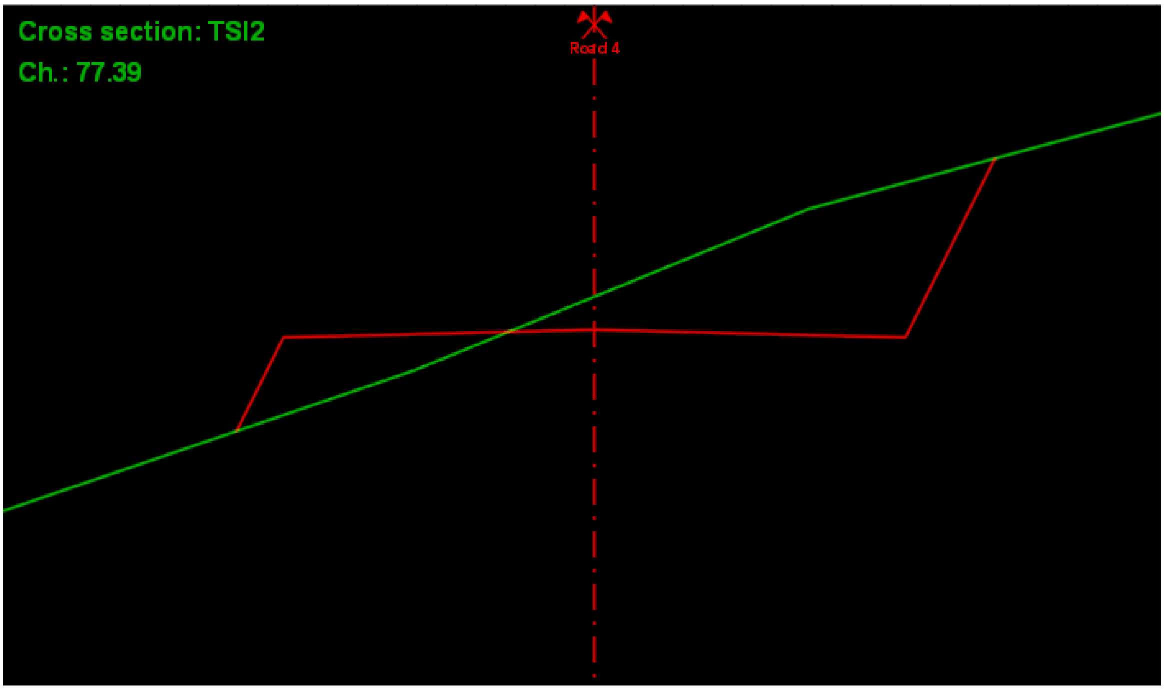
9 Cross Sections

9.1 Introduction

The cross sections of a road are created and edited in a workspace other than the *Horizontal Alignment* and *Profile* that we have described in previous chapters. The *Cross Sections* workspace offers special functions that can cover the needs of the most demanding users. Before starting the detailed presentation of every function, and for the better understanding of how this workspace works, we shall describe briefly the structure and philosophy of the *Cross Sections* workspace.

The tab of the *Cross Sections* workspace contains the cross sections of a road. Every cross section of the road is identified by a chainage and a name. The chainages and names of the cross sections can derive either automatically, if this information already exists either in the *Horizontal Alignment* or in the *Profile* workspaces of the specific road, or they can be imported from a serial text file, or, the user can enter them one by one.

A *Cross Sections* project is comprised of one or more cross sections.

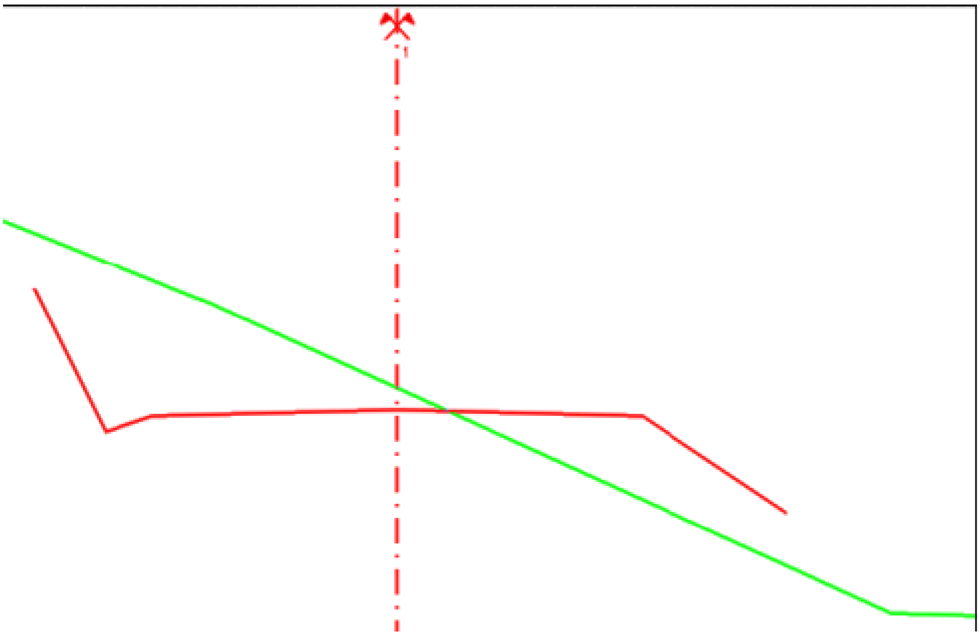


Every cross section is identified by its chainage, which is the main value used for arranging the cross sections in their workspace. You can view and edit one by one the cross sections of a project on the screen. The program also offers special operations for batch editing.

Any intervention on the natural relief of the ground until the road acquires its final form is represented in a cross section by using the appropriate lines. When a road's cross section is in the initial stage, namely before you start configuring it, it usually contains only one line, the *"Natural ground"* line. The Natural ground line depicts the ground surface before any intervention. In most cases, at least one more line is added, the *"Artificial ground"* line, during the final stage of the cross section processing. The Artificial ground line depicts the final surface of the ground, as this will derive after the execution of tasks, either in the case of a cutting or a filling. Regarding the cross sections, you must use further lines to fully describe the geometry of the street. For this reason, the program offers a variety of lines that the user can select and use according to the project's needs and characteristics (e.g. benches, unsuitables, paving layers, etc.).

The simplest form for processing a cross section is to design every line by defining each point individually, as you would do if you were to design the cross sections manually. In this case, the program must calculate the areas that are formed between the lines, in order to draw up the mass table. For this reason, the lines are grouped in main categories - families, depending on their functional role. For example, in the aforementioned cross section, the program will automatically calculate separately the two areas that are formed between the natural ground and the artificial ground, in order for you to know the surface of the cutting and filling.

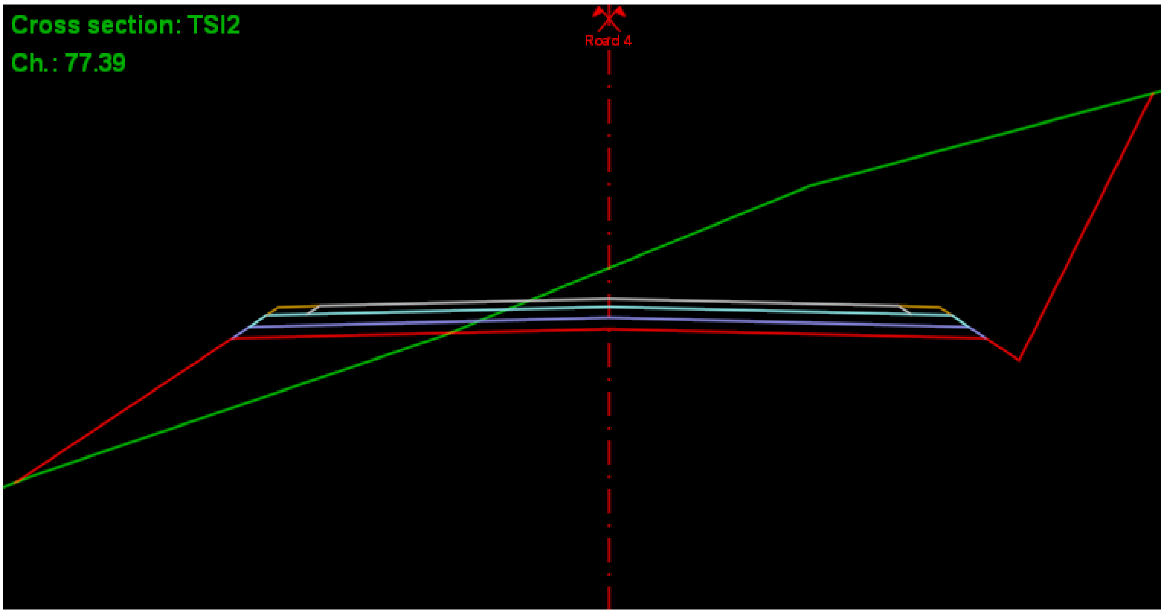
In a more advanced processing stage, the user may insert, e.g. the Artificial ground, without having defined accurately the starting and end points. The program will define these points during the calculation. The program will automatically extend or cut-off the edges of the Artificial ground, so that the latter "wraps" upon the Natural Ground.



The use of special lines for every task enables the program to automatically calculate the sections of the lines, as well as the produced (lengths-areas), in order to draw up quickly and easily the tables of quantities. The arrangement and standardisation of the lines, as well as the cross reference of the relevant calculated quantities to the mass table columns are referred to shortly in the program as "*Typical Tasks*". Of course, the user is free to parameterize the lines – members of the aforementioned families and to create new ones, according to the specific requirements of every project.

The typical cross sections are a parametric way to describe the lines that comprise a cross section. Just like cross sections, they are also comprised of four main parts, the branches, the topsoil, the sideslopes and the pavings.

The typical cross sections, which constitute the most automated stage in the cross section processing, are also based on the typical tasks. By using a typical cross section, the user does not create one by one the lines of every cross section, but he defines the form of the cross section qualitatively and geometrically, both in the cutting and filling. In practice, this means the creation of a cross section like the one shown in the figure:



The user must have defined, always in the typical cross section, the characteristics of the cross section, such as the number of branches, the way that the road profile elevation will be applied, the number of paving layers, the thickness and the geometry (widths, superelevations, sideslope layers' slopes) of the layers, the sequence of the sideslopes' slopes, as well as the extent of the topsoil. A typical cross section can be defined in an absolute or in a dynamic way. When using the absolute definition, the main sizes that characterize a typical cross section, e.g. road profile elevation, pavement width, roadway width, superelevations, have fixed values and they do not depend on the chainage where the typical cross section will be applied. When using the dynamic definition, the values of the main sizes of the typical cross section are not fixed, but they change based either on the chainage or on the relevant diagrams (profile, superelevation, width, etc.), or by special functions (as in the case, e.g. of the shoulder superelevations calculation).

Having all the aforementioned data when calculating automatically the cross section, the program will determine the form of the cross section (cutting, filling or composite cross section), create automatically the required lines, place them by elevation and by horizontal alignment, set their points based on the geometrical characteristics that you have defined, and finally it will calculate the areas or the lengths of the lines.

The typical cross sections, as well as the typical tasks, can also be found in separate files of an external library (directory ...\Anadelta\Tessera\Csec) and you can import them at will, into every cross section project – file, thus creating an internal library of Typ. Cross Sections and Tasks, especially set up for the particular project. The user is also enabled to export to the external library both the set of the typical tasks that he has created for a project as well as the typical cross sections that have been created by the user or have derived from modifications in existing cross sections.

You can also create, edit, import and export various diagrams to/from serial text files (ASCII). The diagrams in the cross sections workspace are used for stating a change in a value, e.g. superelevation, width, elevation, etc. with respect to the chainage. The user is also enabled to import and export several other data of his choice. For example, you can easily import or export the points of a line, e.g. the Artificial ground line or the Natural ground line from/to a serial text file (ASCII), or to import – export certain drawings to/from external library files, or to proceed to the batch assignment of the excavation percentages of every cross section (soil, rock, etc.) based on an external file.

Finally, some of the program's special functions enable you to import, copy, cut, paste, delete, and edit multiple lines for a group of selected cross sections.

This chapter describes the operations that you can carry out in the *Cross Sections* workspace. These operations are classified into the following categories:

- *File operations:* Select files for editing, import data from files, export data to files, view ASCII files for reading, delete files.
- *Cross section operations:* Add and delete a cross section, change the chainage.
- *Calculation operations:* Calculation of cross sections, assignment of drawings, creation of lines, copy, deletion and move of lines. The operations of this category can be carried out in two easy ways, either for multiple cross sections at the same time or separately for every cross section.
- *Printing operations:* Setting of printing and printer parameters, cross sections printing, mass tables printing.
- *Operations for files with multiple roads (Professional version):* Join cross sections of various roads into a composite cross section, change the current road, add a new road, delete an existing road.

The stages that are usually followed during the design of a road’s cross sections are:

1. Creation of workspace and cross sections.
2. Cross section management.
3. Set up of typical tasks.
4. Creation and editing of lines.
5. Update of cross sections with the profile, width and superelevation diagrams.
6. Cross section calculation – application of typical cross section.
7. Addition of special lines to the cross sections.
8. Operations of batch cross section management.
9. Operations of batch line management.
10. Calculation of a road’s areas.
11. Mass table drawing-up – calculation of volumes.
12. Printing of cross sections, export to files.

9.2 Creating a Project

With *Anadelta Tessera* you can start creating a new *Cross Section* project in two ways:

1. You can request the automatic creation of the cross sections based on the elements that you have inserted in the *Horizontal Alignment* or in the *Profile* workspaces.
2. You can create a new file that will contain only the *Cross Sections* workspace. The file that you are creating is empty, namely it doesn't contain any cross sections, and you must create them manually. This procedure is useful in case you wish to use the program only for carrying out measurements on a project.

Creating a New Cross Section Project

This procedure is useful in case that the road project has been designed and you wish to use the program only for measuring the project's cross sections.

How to create a new Cross Sections file

From the *File* menu, select the *New Project…* submenu and run the *Cross Sections* command.

The file that you are creating is empty, namely it doesn't contain any cross sections or lines. All elements, such as cross sections, lines, etc., must be either created by the user or imported from a text file (ASCII).

A *Cross Sections* project is comprised of one or more cross sections. In order for the program to calculate an area, every cross section must have at least two lines the one of which being the Natural Ground (e.g. Natural Ground and Artificial ground). All points of a line comprised in a cross section, have a set of coordinates (x, h), that corresponds to the distance of the point from the road axis (distance from axis) and to the absolute elevation of the terrain at the point.

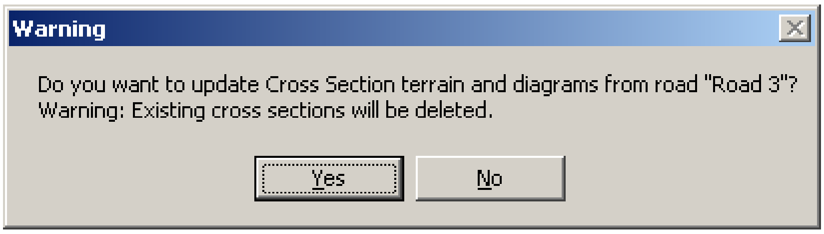
Creating or Editing Cross Sections from the Horizontal Alignment

This procedure is performed after you have completed the horizontal design of the road in the *Horizontal Alignment* workspace and you wish to create and edit the road's cross sections.

This action enables the program to create the cross sections and the Natural Ground line for every cross section in the *Cross Sections* workspace. If your project does not have a terrain model or points, the cross sections will be created anyway, but they will not have the Natural Ground line.

How to create the Cross Sections from the Horizontal Alignment

1. From the *Horizontal Alignment* workspace, select the road whose *Cross Sections* you wish to create or update.
2. From the *File* menu, run the *Update cross sections* command. The following confirmation message will appear on the screen:



Click *Yes* to continue. The program will create the cross sections.

To run the command, you must have performed stationing on the selected road.

You can also create the *Cross Sections* from the *Horizontal Alignment* in the dialogue box of the *Manage* command of the *Project* menu. Select the road from the relevant list at the top of the dialogue box and then click the *From Horizontal Alignment to Cross Sections* button in the *Quick Update* group. The procedure is the same as the one described above. A message appears at the end of the procedure, informing you that the procedure has been completed.

When this type of update takes place, the program creates the road's cross sections, together with the main terrain line and its points for every cross section. Also, the pavement width diagram, the roadway width diagram, the left and right superelevation diagrams, the left and right semiaxis diagrams, as well as the diagrams that the user has created from a drawing line (diagrams of the existing road and other diagrams) are also transferred. Note that through this type of update the cross sections are created again from the beginning and any data that existed in the *Cross Sections* workspace before the update will be deleted.

Alternatively, you can fully define which elements of the *Cross Sections* will be updated. To do this, use again the *Manage* command. From the *Update* group of the *Updates* tab in the command's dialogue box, select *From Horizontal Alignment* in the first drop-down list and *To Cross Sections* in the second drop-down list. Next, select which updates will take place by ticking or unticking the relevant check boxes and then click the *Update* button.

The program will create the *Cross Sections* workspace, if there is no such workspace for the selected road; next, in the *Cross Sections* workspace, the program will insert the cross sections with the Natural Ground line if you have requested so, or by default if this is the first time this workplace is being created.
You can use the aforementioned commands to update the *Cross Sections*, in case you have modified the road design in the *Horizontal Alignment* workspace.

Creating a Cross Section Project from a Profile Project

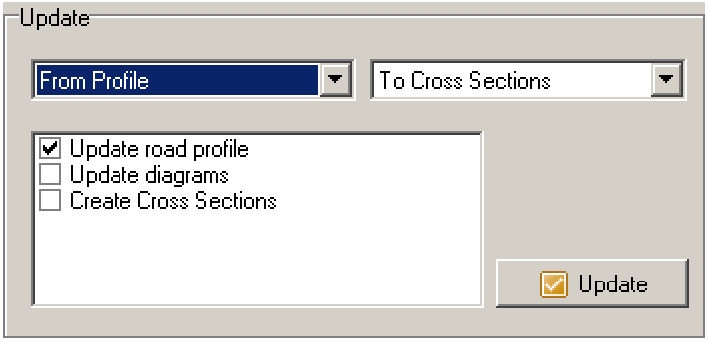
The elements that the *Profile* supplies to the *Cross Sections* workspace are the station names with the respective chainages, an indicative Natural Ground with two points, and the road profile diagram.

The first point of the natural ground is on the cross section axis and at the elevation defined by the Profile, while the second point is a little further to the right, at the same elevation.

How to create the Cross Sections from the Profile

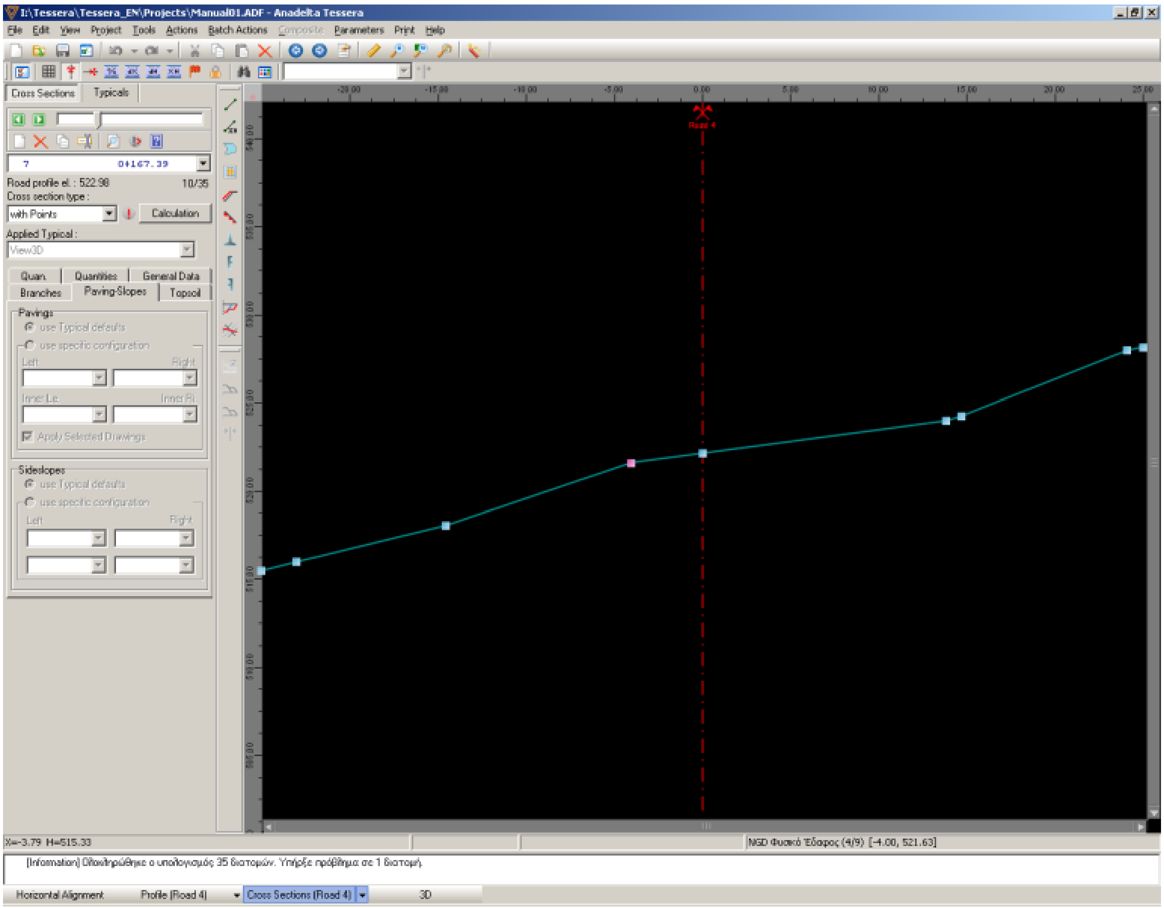
1. Go to the *Profile* workspace that corresponds to the road whose *Cross Sections* you wish to create or update.
2. From the *Project* menu, run the *Manage* command.
3. In the *Update* group of the *Updates* tab, select *From Profile* in the first drop-down

list and *To Cross Sections* in the second drop-down list and then tick the *Create Cross Sections* check box. Click the *Update* button to create the cross sections. If you wish to also update the *Cross Sections* with the road profile diagram, tick the *Update* road profile check box. To update the other diagrams, tick the relevant check box.

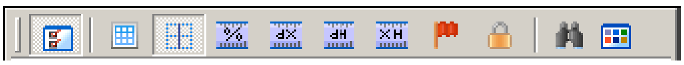


9.3 The Workspace

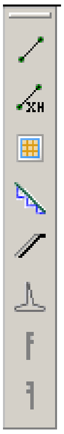
This paragraph describes the *Cross Sections* workspace. The particular workspace has many differences compared to the other two workspaces that have been described so far. To go to the *Cross Sections* workspace, click the *Cross Sections* tab at the bottom of the screen, over the status bar, or run the *Cross Section* command from the *View* menu. The tab is displayed only if you have carried out the cross section creation procedure.

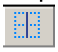


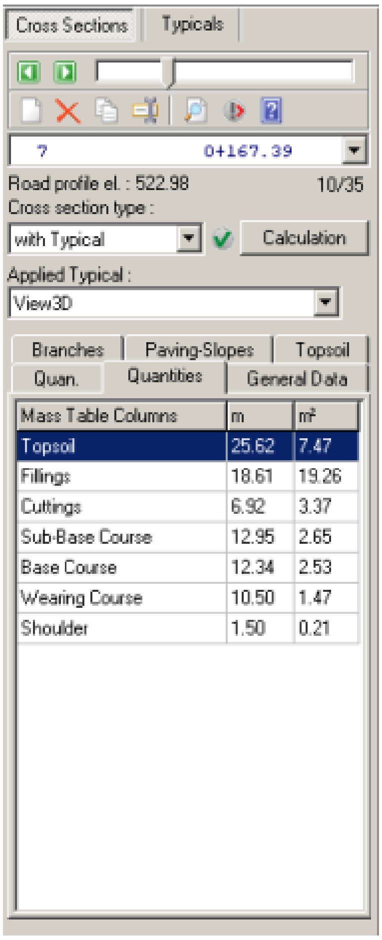
Similarly to the *Horizontal Alignment* and *Profile* workspaces, at the top of the screen, you will find the familiar menu bar, with the *File, Edit, View, Project, Tools* and *Help* menus. There are also four new menus: *Actions, Batch Actions, Parameters* and *Print.* Below the menu bar, you will find the main toolbar and below the latter, you will find the View toolbar, which contains buttons that carry out operations relevant to the elements that are displayed on the screen, especially in the drawing area.



At the left of the program's window you will find the *Lines* toolbar, whose functions refer to the creation and editing of the cross sections lines.



There are also the scroll bars, while in the middle of the cross section (at the point where the x abscissa equals to zero), you can see the road axis, which is displayed as a red axial line. If you do not wish to view the axis line, click the  button on the View toolbar. If you wish to view the axis line, click again the same button. If you wish to view a grid in the drawing area, click the second button on the View toolbar.



To select a line on the screen, place the crosshair over the line and click the left mouse button. A selected line is highlighted (it changes color) and a small blue box appears on every point of the selected line. The active point of the line is displayed with a pink box. If you have selected the fifth, sixth, seventh or eighth button of the *View* toolbar or any combination of them, then the slopes, the horizontal distances, the vertical distances and the coordinates of the points are displayed over the straight parts or the points of the selected line.

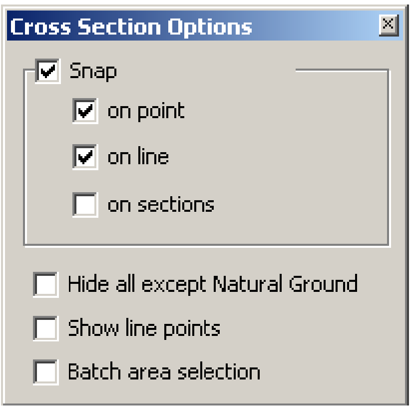
After the calculation of the cross section, you can choose to also view the calculated areas on the screen. Simply left-click on the region whose area you wish to view. The area is activated immediately and the region is filled with color, so that its boundaries can be viewed clearly.

In the drawing area you can see all lines that comprise a cross section, and you can select and edit those lines. In the *Cross Sections* workspace, you can view and edit only the selected cross section.

The *Cross Section* data management window is displayed at the left of the main screen window. The management window is primary, since here you can execute most of the operations referring to a cross section, while you can also view all parameters of the cross section calculation, as well as the calculated quantities/areas. The name and the chainage of the current cross section (the one you see on your screen) in the file, are displayed in the *Cross Section selection* drop-down list of the window. You may choose not to display the management window. Simply click the first button from the left of the View toolbar . If you wish to display again the window, click again the same button. If the management window is closed, the name and the chainage of the cross section appear in green letters at the top left of the drawing area.

The status bar is displayed at the bottom of the screen and exactly below you can see the message window.
The status bar is comprised of four sections. The first section shows the coordinates of the current position of the crosshair (distance from axis and absolute elevation). The second section enables you to display the *Input Form,* if available. Next to the input form, in the third section, you can see the type of the line (e.g. Natural Ground) where the crosshair is located, or read various messages – instructions that the program shows to guide you during the execution of the various commands. The fourth and last sections display information on the selected line (total line points, serial number of the selected point and coordinates of the selected point).

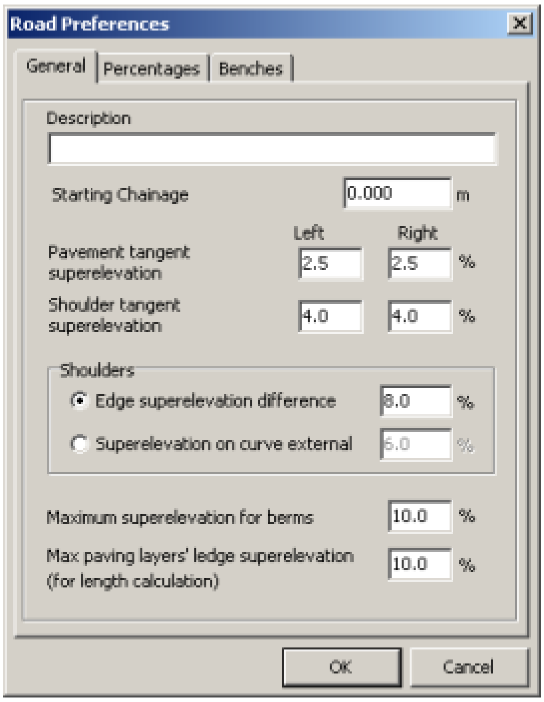
The message window is the space where the program displays various messages about the problems that are detected during the calculation of a cross section. The messages are divided in three categories: information, warning and error.



The snap can be set from the dialogue box shown in the figure. This box appears by running the *Options* command of the View menu or by clicking the button on the *View* toolbar or by pressing the *F2* key. If you select the *Show line points* check box, the points of all lines that are included in the cross section are displayed. If you select the *Batch area selection* check box, you can select all areas that are created after the cross section calculation by pressing and holding the *Ctrl* key.

9.4 Road Properties

If you run the *Road preferences* command from the *Parameters* menu, the following dialogue box will appear on the screen:



The box has three tabs, where you can make various settings for the road, the benches and the excavation percentages of the cross sections.

In the *General* tab, you can enter the description of the road in the *Description* text field and, if you wish to change the starting chainage of the road, enter the value in the *Starting Chainage* text field. In the shoulders group you can specify the shoulder calculation way and the values of the left and right elevation of the shoulders, as described in the next chapter. The *Benches* tab contains all the necessary settings for the automatic creation of benches, while in the *Percentages* tab you can specify the percentages of every soil category and the bulking factors for every category. When you have finished, click the *OK* button to save your changes and return to the *Cross Sections* workspace.

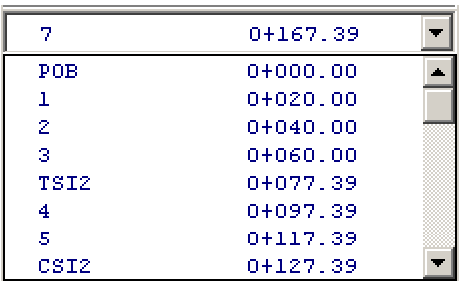
9.5 Cross Sections Management

Moving Between Cross Sections

For moving between cross sections, the *Cross Section* data window features two buttons, one drop-down list and a scroll bar, but you can also use the dialogue box of the View toolbar.

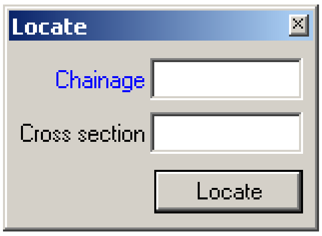
If you click the  button of the *Cross Section* data window, the program transfers you to the previous cross section (based on the chainage) while the  button transfers you to the next cross section. You can do the same by pressing the *Page Up* or *Page Down* keys respectively. The cross sections are sorted out in ascending order according to their chainage.

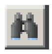
There are also two extra functions, enabling you to go to any of the project's cross sections. The first function is executed from the *Cross Section selection* drop-down list in the *Cross Section* data window. In this list, you can see the name and the chainage of the current cross section. If you left-click on the list, a menu appears where all the cross sections of the current project are sorted out by chainage, and you can choose the cross section where you wish to go.



The second function that you may use to find a cross section, is executed with the assistance of the scroll bar, which is located at the top of the *Cross Section data* window.

Here, you can select the cross section that will appear on the screen as follows: Left-click on the rectangular box on the scroll bar. The name and the chainage of the current cross section in the file are displayed in the *Cross Section selection* drop-down list. Hold the left mouse button down and scroll until you see the name of the desired cross section; then release the mouse button.

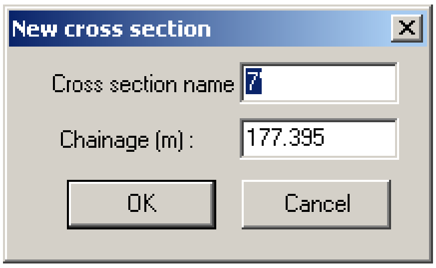


The last search function can be executed with the *Cross Section* data window open or closed. Click the  button on the *View* toolbar to open the dialogue box. The box has two text fields and a button. If you wish to go to a cross section whose chainage is known, type the chainage in the *Chainage* field and then click the *Locate* button. If there isn't any cross section at the specified chainage, the program will transfer you to the immediate next cross section. If you wish to go to a cross section whose name is known, type the name in the *Cross section* field and then click the *Locate* button.

Cross Section Insertion

In the *Cross Sections* workspace, the user can add the desired cross sections to a road.

How to insert a new cross section



1. From the *Cross Section* data window, click the button.
2. The dialogue box appears on the screen, where you must enter the data of the new cross section (name and chainage). The program fills automatically these fields (namely, it gives the name of the current cross section with an accent and selects a chainage with respect to the next cross section), but you can still enter the values you desire.
3. When you have finished, click the *OK* button to create the new cross section, which will be displayed in the drawing area. This cross section will be empty, namely it will not have any lines.

You can also add a cross section when importing lines from ASCII text files. Another feature is the batch import of cross sections from another project.

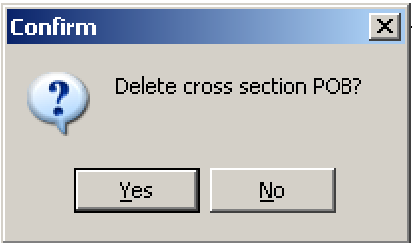
Changing the Cross Section Chainage and Name

This paragraph describes the way in which the name and the chainage of a cross section are specified. This can be done by clicking the button. A dialogue box appears on the screen. In the *Name* field, type the new name (code) of the cross section by using any combination of capital letters and numbers.

In the *Chainage* field, type the chainage (in meters) of the cross section in the project. This data will appear in the section that shows information on the current cross section. Although the same file may contain two cross sections with the same name, it is not possible to have two cross sections with the same chainage. Therefore, if you accidentally type a chainage that has already been assigned to another cross section, the program will show you the following error message,

informing you that the cross section cannot be transferred.

Deleting Cross Sections



To delete a cross section, click the  button. In the resulting window, click *Yes* to confirm that you wish to delete the cross section, otherwise click *No*. You can also delete multiple cross sections at once.

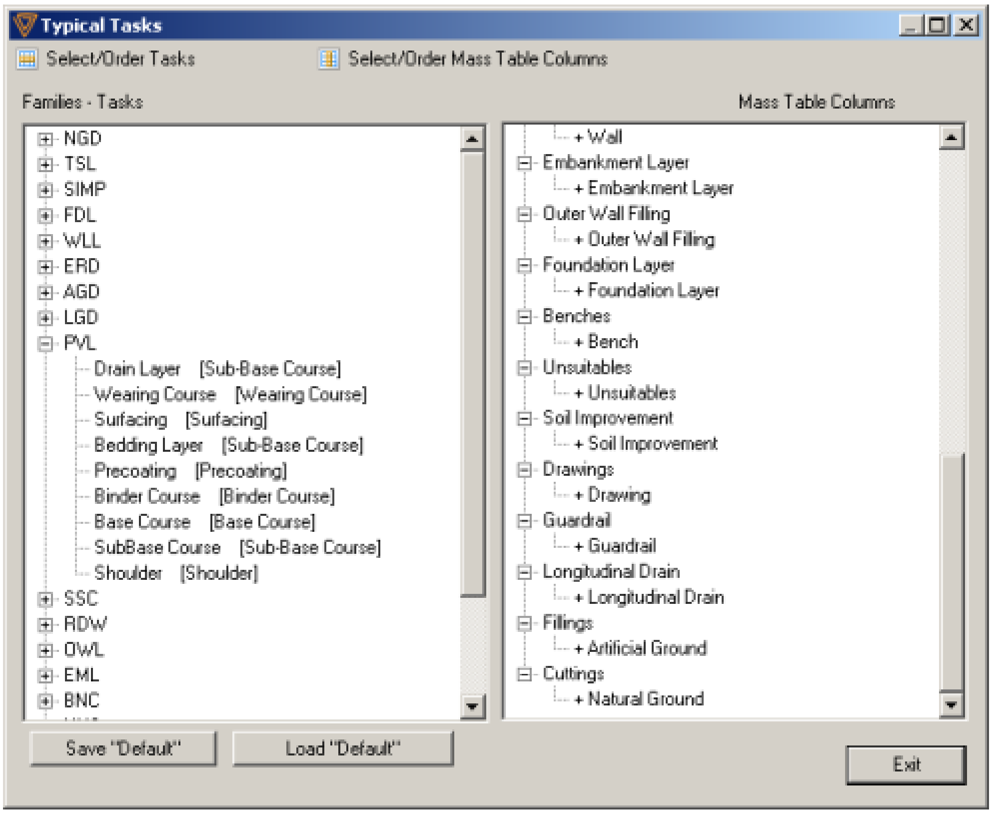
9.6 Typical Tasks

9.6.1 Creation - Management of Typical Tasks

The lines comprising each one of the program's cross sections are grouped in families. This sorting out of the lines has a qualitative meaning. It allows every line, depending on its family, to locate cross-sections and to calculate areas, by using the appropriate lines. It also serves to determine how the lines will appear both on the screen and on the printouts, as well as to the absolute parametric drawing-up of the mass and paving tables.

This standardisation of the lines (tasks) and the linkage of the calculated quantities to the mass table columns are referred to shortly as *Typical Tasks.* The determination of the typical tasks is one of the main operations of the program and must be done before starting to draw the cross-station.

From the *Parameters* menu, run the *Typical Tasks* command to open the window in which you will specify the typical tasks.



Import of Typical Tasks from a File

To import typical tasks from an external file into the current project, go to the *File* menu, open the *Insert* submenu and run the *Typ. Tasks* command. Before running this command, however, you must be certain that you do not need the current typical tasks that you are using for your project. It is advisable to export them first into a file and then to replace them, because the import of a typical tasks file deletes the current typical tasks and replaces them with the ones contained in the file.

You can import the typical tasks either from a typical tasks file (*.ATE) or you can use the typical tasks of another *Tessera* file (*.ADF).

How to import typical tasks from another file

1. From the *File* menu, select the *Import* submenu and run the *Typ. Tasks* command.
2. The *Open* dialogue box appears on the screen. From the *File Type* field, select the type of file (*.ATE or *.ADF), then select the desired file and click *Open*.

Creation of Typical Tasks Files (*.ATE)

The typical tasks can be saved in the external library (directory ...\ANADELTA\TESSERA\CSEC). The default extension of these files is ATE.

How to export typical tasks to a text file

1. From the *File* menu, select the *Export* submenu and run the *Typ. Tasks* command.
2. The *Export Typ. Tasks* dialogue box appears on the screen. Enter the file name and click the *Save* button.

Typical Tasks of a New File

The typical tasks of a new project are also imported automatically from a typical tasks file (*.ATE). This file is selected from the *General Options* dialogue box of the *Tools* menu. More specifically, the full name and the path of the file are displayed in the *New project's typical cross sections* field of the *Cross Sections* tab. If you wish to select a new file, click the  button. Select the desired typical tasks file from the dialogue box that will appear on the screen. Once you've made your selection, click the *Open* button to return to the *General Options* dialogue box, and then click the *OK* button to close the dialogue box and return to the main screen.

9.6.2 Families

The typical tasks are comprised of families. The main available task families are those listed below, together with a description of the behavior of every family's lines during the calculation and the area measurement of the cross sections.

NGD (NATURAL GROUND): This family – task represents the natural ground level before any intervention into the cross section. The *Natural Ground* line of every cross section does not extend nor is it cut-off automatically. It always remains unchanged during the calculation of cross sections and contributes to the formation of the *Natural ground* envelope. It cannot be used in the typical cross section.
TSL (TOPSOIL): This family – task represents the excavation line for the removal of the *Topsoil.* It is always below the natural ground, usually parallel to it, at a depth of 0.2 - 0.3 m, without this being mandatory. After the calculation of a cross section, this task contributes to the formation of the *Natural ground* envelope. The tasks belonging to this family cannot be used in the typical cross section.
SIMP (IMPROVEMENT, FOUNDATION): The tasks of this family usually represent improvements that are being made always under the natural ground and sometimes under the road (in the case of a cutting) for the improvement of the ground, or excavations for the foundation of walls. These tasks require a section with the current natural ground envelope. After being calculated, they contribute to the formation of the Natural ground envelope. The tasks belonging to this family cannot be used in the typical cross section.
FDL (FOUNDATION LAYER): The tasks of this family usually represent lines over the natural ground envelope, after the natural ground has been formed by the *Topsoil* and the *Improvements - Foundations.* These are layers of materials for the foundation of walls, or layers of materials before the filling. After being calculated, they contribute to the formation of the Natural ground envelope. They cannot be used in the typical cross section.

NGD.ENV. (NATURAL GROUND ENVELOPE): This is the bottom envelope that derives from the natural ground and the topsoil. If there are also tasks of the SIMP and FDL types, these also form the natural ground envelope according to the order they are inserted. The natural ground envelope is a fictitious line and does not correspond to a task, but it is used as a boundary for the creation of the *Artificial ground* line. It is comprised of the lower elevation points of the lines belonging to the NGD, TSL, SIMP families and of the upper elevation points of the lines belonging to the FDL family.

AGD (ARTIFICIAL GROUND): This task represents the final level of the artificial ground works in a cross section. In order to be calculated, it requires a section with the final natural ground envelope. Its top part where the paving and the encasing solids will be laid is the *Ledge.* The lateral oblique parts are the sideslopes. After being calculated, it shapes the artificial ground envelope. The tasks belonging to this family can be used in the typical cross section.

PVL (PAVING LAYERS): The tasks of this family represent the paving layers, e.g. Courses (Surfacing, Wearing Course, Binder Course, SubBase Course), mining materials (Base, SubBase, Drain Layer, Grading Layer), roadway shoulders, special constructions that are being used mainly for the detachment – endistancement of the encasing solid from the edge of the course, and generally lines over the artificial ground that comprise the paving. They require a section with the current artificial ground envelope, which they also shape after they have been calculated. The tasks belonging to this family can be used in the typical cross section.

GTT (GUTTER): The tasks of this family represent the encasing solids of the roadway's paving materials (kerb channels, gutters, N.Jersey). They are sorted out in the cross section by branch (left, right) and by side (left, side, centre). They are either supported on or shape the underlying PVL. They are calculated together with the corresponding PVL and they require a section with the artificial ground envelope, which they also shape after they have been calculated. The tasks belonging to this family can be used in the typical cross section.

SSC (COATING): The tasks of this family usually represent the sideslope coatings with topsoil in the fillings (parallel to the sideslopes or not) or the construction of retention zones or rock traps in the cuttings, on the external side of the gutters. They require a section both with the natural ground envelope and with the current artificial ground envelope, which they also shape after they have been calculated. The tasks belonging to this family can be used in the typical cross section.

AGD.ENV. (ARTIFICIAL GROUND ENVELOPE): This is the top envelope that derives from the tasks of the AGD, PVL, SSC and GTT families. It is a fictitious line and does not correspond to any task. It assists during the calculation of all tasks above the Artificial Ground.

EML (EMBANKMENT LAYER): The tasks of this family represent layers that are laid over the natural ground envelope (e.g. the separation of the filling in various filling materials). They must be laid by elevation, from the lower to the upper one. They are laid under the *Artificial Ground* and they are calculated starting from the lower one. They require a section both with the *Artificial ground* and with the current envelope of the artificial ground's bottom layer, which they also shape after they have been calculated. They cannot be used in the typical cross section.

EML.ENV. (EMBANKMENT LAYER ENVELOPE): It is the envelope that derives from the natural ground envelope and the tasks of the EML family, which shape it according to the order they are applied. The envelope of the artificial ground's bottom layer is a fictitious line and does not correspond to any task, but it assists during the calculation of all tasks under and inside the Artificial ground and over the natural ground envelope.

BNC (BENCH): The tasks of this family usually represent the anchoring benches (tasks for the stabilization of the fillings) or improvements that do not shape the natural ground envelope (namely, they do not contribute to the shaping of the Artificial ground). They require a section with the bench envelope or with the Artificial ground, depending on which one is the first to meet. After having being calculated, they shape the bench envelope. They cannot be used in the typical cross section.

BNC.ENV. (BENCH ENVELOPE): It is the bottom envelope that derives from the natural

ground envelope, the artificial ground and the tasks of the BNC family, which shape it according to the order they are applied. The bench envelope is a fictitious line and does not correspond to any task, but it assists during the calculation of all works under the Artificial ground and the natural ground envelope.
UNS (UNSUITABLES): The tasks of this family represent works referring to unsuitables (e.g. excavation works, whose products cannot be reused for filling). They require a section with the unsuitables envelope, which they shape after they have been calculated. They cannot be used in the typical cross section.
UNS.ENV. (UNSUITABLES ENVELOPE): It is the bottom envelope that derives from the natural ground envelope and the tasks of the UNS family, which shape it according to the order they are applied. The unsuitables envelope is a fictitious line and does not correspond to any task, but it assists during the calculation of all tasks under the natural ground envelope.
DRW (DRAWING): The tasks – lines of this family do not affect nor do they contribute to the calculations of sections, they do not extend and they can be of any shape. The user can either draw them freely or import them from a drawing library. These tasks can be used in the typical cross section.
WLL (WALL): The tasks of this family represent the supporting – backing walls that stabilize the slideslopes of cutting and filling, respectively. These are the first to shape the artificial ground envelope. If they are active**, then they shape the Artificial Ground task during the calculation of sections. These tasks cannot be used in the typical cross section.
ERD (EXISTING ROAD): The tasks of this family usually represent the artificial ground envelope of an existing road. They are used when making improvements on an existing road. During the cross section calculation, the lines of the AGD, PVL and GTT tasks are formed in such a way, so as to coincide with the particular task, at the part where the ERD line is higher than them, in order to outline the roadway of the existing road. The tasks if this family cannot be used in the typical cross section.
GRL (GUARDRAIL): This family represents the guardrails with barriers that are inserted automatically by the program, when applying the typical cross section. They do not take part in calculations. Their position depends on the elevation of the filling and they can be used in the typical cross section.
LGD (LONGITUDINAL DRAIN): The tasks of this family usually represent the configuration that takes place for the placement of longitudinal drains under the artificial ground envelope. They require a section with the artificial ground envelope and they can be used in the typical cross section.

* Additional parameters (branch, road side, bearing line etc.) for tasks of the PVL, GTT and WLL types that are inserted outside the typical cross section can be specified from the *Properties* dialogue box.
**Active are the walls whose value in the *Shapes Artificial Ground* field is *Yes*.

9.6.3 Families - Tasks

In the left part of the *Typical Tasks* window there is the *Families – Tasks* group. The code of every family is shown in capital letters in the table of the *Families – Tasks* group. The family codes have been strictly defined by the program and it is not possible for the user to change them. When you select a family of lines, you can view the members of every family that correspond to the various types of a family's tasks - members. Left-click on the + symbol next to each task family or double-click on the family name, to view the tasks of this family. You can select a task by left-clicking on it. The tasks are displayed in a tree structure below the code of the family to which they belong, and they are connected to their family with a line.

Regarding the calculation of the areas, the quantities that result automatically during the calculation of a cross section refer to the whole task and not to an individual line that belongs to this type of task. Therefore, you must create as many types of tasks as the quantities you wish to measure separately. If, for example, you intend to use two base layers in a cross section, and you wish the program to calculate the lengths or the areas separately for each layer, you must define two different types of tasks, e.g. "Base 1" and "Base 2". With these two tasks and through the proper linkage of the quantities, the quantities can be calculated either as two separate final quantities-columns of the mass table, or both quantities can be seen as a single final quantity in the same column of the "Base" mass table. It is also possible to combine the above two possibilities.

In order to create a new task you can:

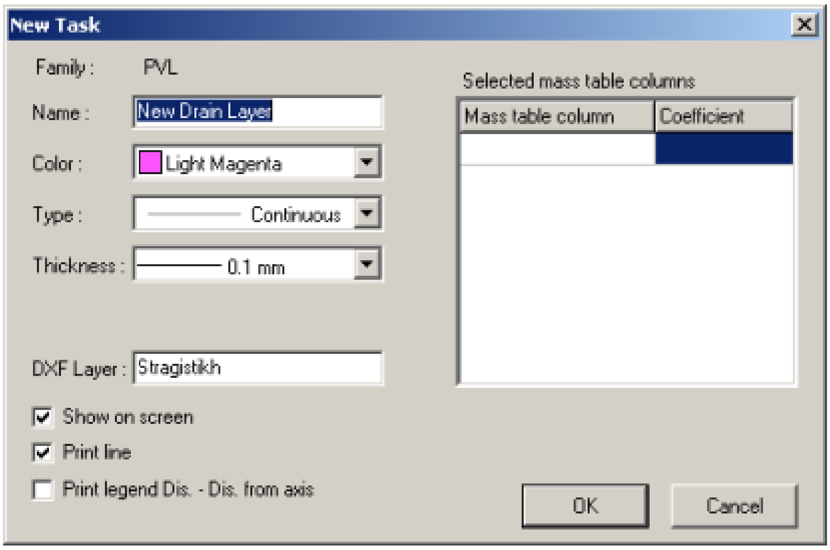
1. *Add - define* the new task in the task group.
2. *Create* the mass table column where the task will send its quantities, if you wish the task to be calculated separately.
3. *Link* the particular task with the desired column of the mass table.

<u>Adding a New Task</u>

You can add to every family as many members – task types as you wish, but you can delete only the members that you have added. The only exception constitute the tasks of the Natural ground, Topsoil and Artificial ground, which are unique in their family.

How to create a new task

1. Run the *Typical Tasks* command of the *Parameters* menu and then right-click on the name of an existing task that belongs to the same family to which the new family will belong. From the displayed menu, select *New Task.* Alternatively, right-click on the family name and select *New Task for family...*.
2. This option opens the dialogue box where you can specify the parameters for the new task.



3. In the *Name* field, type the name of the new task. The color, the line type (full, axial, dashed, dotted, etc.) and the thickness of the line on the printout are defined from the relevant drop-down lists. In the *Dxf Layer* field, type the name of the Layer to which the line will belong when the drawing is exported to a DXF file. If you do not wish the line to be displayed on the screen or on the printout, untick the Show on screen and Print line check boxes respectively. If you wish to print the line coordinates as a legend below the drawing of every cross section, tick the *Print legend Dis. - Dis from axis* check box.
4. To exit the dialogue box, click *OK* to confirm the changes you have made or click

Cancel to return to the main screen.

You can link the quantities from the right part of the box in the *Selected mass table columns* group, or in the *Mass Table Columns* group.

The default parameter values are the corresponding values of the existing task that you clicked before, in case you have selected a task.

<u>Setting the Task Properties</u>

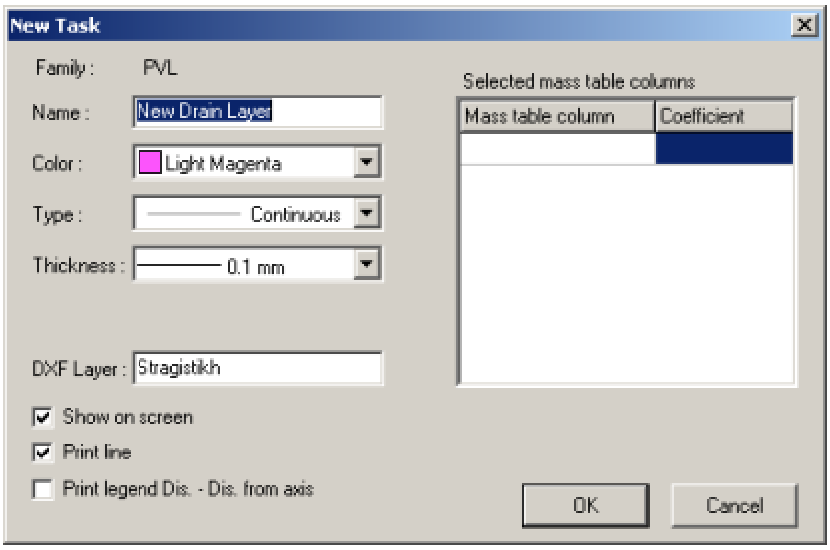
To set the properties of a task, select the desired task, click the right mouse button, and from the displayed menu run the *Task properties* command. Then proceed as described above for the new tasks.

<u>Deleting Tasks</u>

To delete a task, select the task, click the right mouse button, and from the displayed menu run the *Delete task* command.

<u>Linking a Task with Columns</u>

The calculated quantities of a task are linked with the relevant columns of the Mass Table by assigning a task quantity to a column, from the following dialogue box:



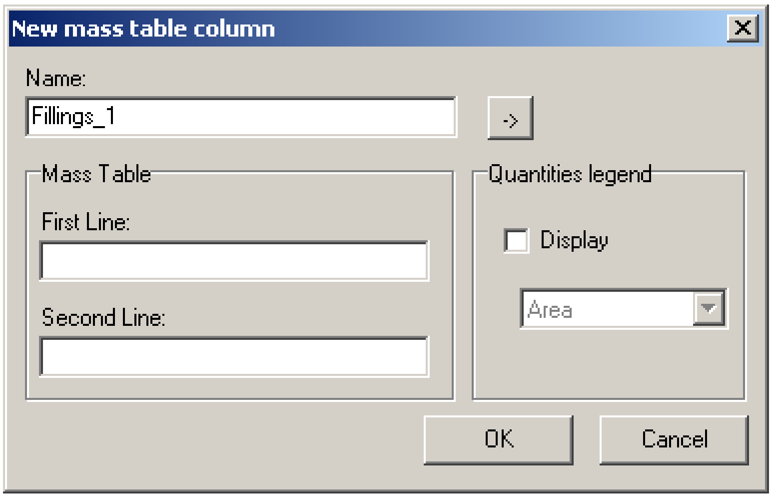
Here you can define to which columns of the mass table the selected task will contribute; these columns will be displayed in the table of the *Selected mass table columns* group. To add a new column, right-click on the table and from the displayed menu, open the *Add column* submenu and then select a column. This menu contains all columns of the *Mass Table Columns* group. In the *Coefficient* column of the table, type the coefficient in the row where the column name appears. To remove a column, select the desired column, click the right mouse button and from the displayed menu run the *Remove mass table column* command.

9.6.4 Mass Table Columns

At the right part of the *Typical Tasks* window there is the *Mass Table Columns* group. This group corresponds to the columns of the mass – paving table. The user is free to create and delete final quantities at will. Every task of the *Families – Tasks* group can be linked to as many *Mass Table Columns* as you wish. This enables you to further group certain quantities and to view them all in one column of the mass table. This linkage is specified by the user, who can also add/remove a whole quantity or a percentage of a quantity to/from a mass table column. Below each column you can view which tasks have been assigned to that particular column.

Adding a New Mass Table Column

To create a new mass table column, run the *Typical Tasks* command of the *Parameters* menu, right-click on an existing column, and from the displayed menu, run the *New mass table column* command. This option opens the dialogue box where you can specify the parameters for the new column. The values in the fields of this dialogue box are the same as the ones in the existing column that you had selected before running this second command.



In the *Name* field, type the column – quantity name, which will appear both in the dialogue box of the typical tasks and in the cross section legend on the printout. Apart from the name, there are also two groups, one for the way in which the quantity will be displayed in the mass table and one for the way in which the quantity will be displayed in the cross section drawing legend (quantities legend).

In the *Quantities Legend* group, tick or untick the *Display* check box, depending on whether you wish or not to view the particular quantity in the cross section drawing legend (quantities legend). From the *Quantity type* drop-down list, select the type of the quantity that the program will calculate. Namely, select the area or the length.

Regarding how the quantity will be displayed in the mass table, there are two lines in which you can describe the particular quantity. If you wish, you can copy in the *First Line* field the name that you specified in the *Name* field, by clicking the  button.

Setting the Parameters for a Mass Table Column

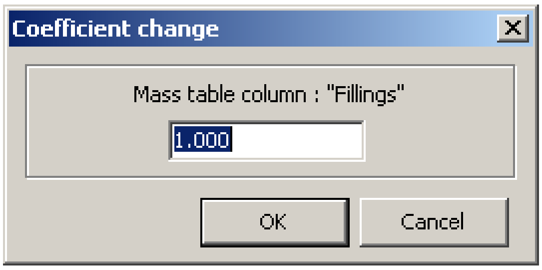
To set the parameters for a mass table column, select the desired column and from the displayed menu, run the *Mass table column properties* command. Then proceed as described above for the new columns.

Deleting a Column

To delete a Mass Table column, select the column, click the right mouse button, and from the displayed menu, run the *Delete mass table column* command.

Linking a Column with Tasks

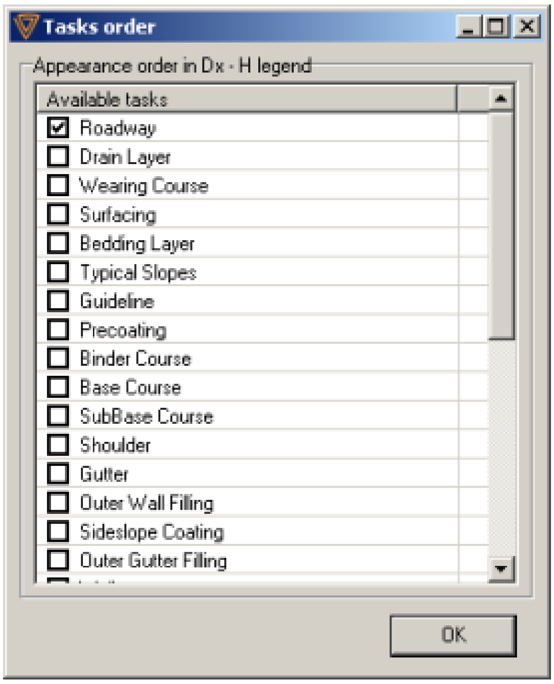
A Mass Table column can be linked with the appropriate quantities of calculated tasks. To do this, right-click on a column and from the displayed menu open the *Add new task* submenu and select the task you wish to assign to the selected column. This submenu contains all tasks of the *Family – Tasks* group. A quantity is transferred to a column by means of a coefficient, which, when multiplied by the quantity, gives the value that will be added numerically to the selected final quantity. Initially, this coefficient equals to one. To change the coefficient of a task, select the task that has been added below the column, click the right mouse button and from the displayed menu run the *Coefficient Change...*command. The following dialogue box appears on the screen,



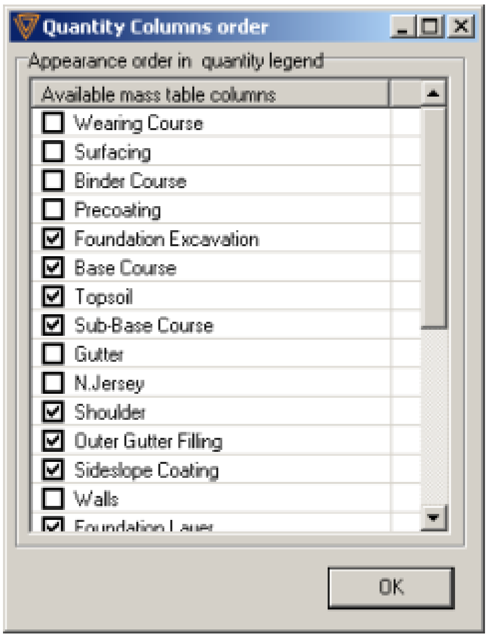
and you can type the new coefficient. If you wish to delete a task from a column, select the desired task, click the right mouse button and from the displayed menu run the *Remove task* command. The program will prompt you to confirm the deletion by showing you the relevant confirmation message. Click *Yes* to delete the task and complete the procedure.

9.6.5 Order of Columns and Tasks

At the top of the initial dialogue box of the Typical Tasks, there are two buttons. By clicking the *Select/Order Tasks* button, the following dialogue box appears:



while by clicking the *Select/Order Mass Table Columns* button, the following dialogue box appears:



In the first field of the *Appearance order in Dx-H legend* group, you can select the lines whose coordinates will be printed on a legend at the bottom of the cross section. In the second field of the *Appearance order in quantity* group, you can select the areas or lengths that will be printed below the name and chainage of the cross section. If you want to view a quantity or a task, tick the check box next to its name. If you wish to change the order of appearance, then use the mouse to select the desired quantity or task and then drag-and-drop it to the desired position.

9.7 Creation - Automatic Insertion of Lines

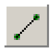
The existence of natural ground is a necessary prerequisite for the calculation of a cross section. As a general rule for the lines that comprise the cross sections, if you haven't requested their automatic creation by the program, you must insert them yourself. The user can insert lines in two ways:

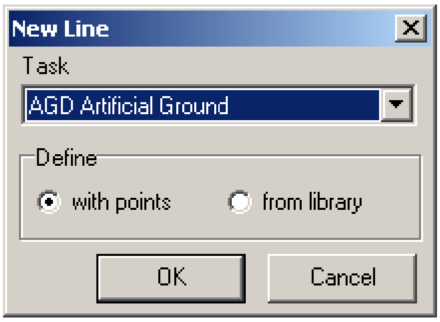
1. By creating a line within the *Anadelta Tessera* environment.
2. By proceeding to the batch import of lines from *ACS* text files.

Creating Lines

You can insert individual lines to a cross section by specifying each line's points. To insert a line, you must first go to the cross section to which you wish to add the line.

How to create a line with the mouse

1. From the *Actions* menu, run the *Insert Line* command, or click the  button on the *Lines* toolbar.
2. The following dialogue box appears on the screen:



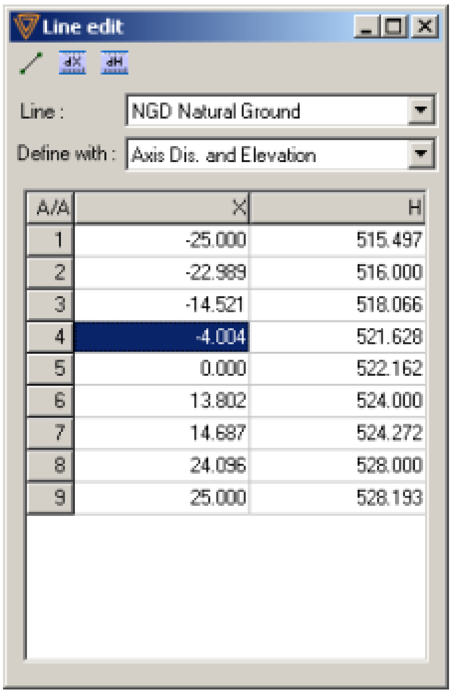
3. From the *Task* drop-down list, select the typical task to which the line will belong (e.g. [NGD] Natural Ground for the natural ground), and in the *Define* group tick the with points check box and then click *OK*. The *with points* option means that the line will be defined by the manual insertion of points, while the *from library* option means that a user-selected new drawing will be imported from the drawings library. This option is very useful for line categories such as [DRW] Drawing, [WLL] Walls, etc., which have a default form.
4. Define the line points with the mouse within the drawing area, using also the snap feature or the *Input Form* command.
5. To finish the point insertion procedure, click the right mouse button anywhere on the screen.

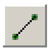
Creating a Line by Typing Absolute or Relative Coordinates

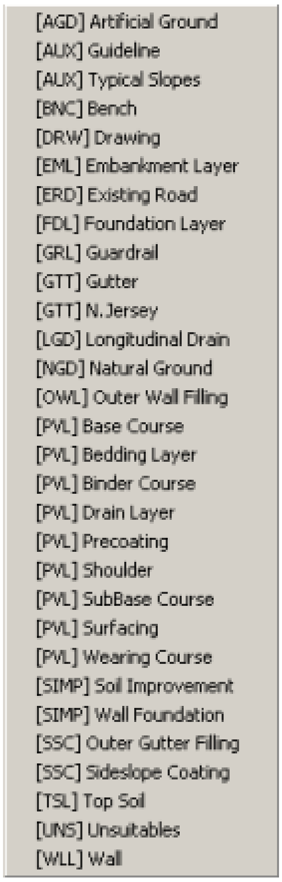
You can insert a line by typing its data on a list. This data can be either the absolute coordinates, namely the distance from axis *(X)* and the absolute elevation *(H)*, or data relevant to the previous point of the line, such as the distance from the previous point and the elevation discrepancy. The usual direction in which the points of a line are inserted is from left to right. For closed shapes, use the clockwise direction.

How to create a line from the window of the Edit line command

1. From the *Actions* menu, run the *Edit line* command, or click the  button on the *Lines* toolbar. The following window appears on the screen:



2. Click the  button. A menu with all the available typical tasks will be displayed, and you must select the task to which the line will belong.



3. When you select the typical task, you will return to the first window. In the *Line* drop-down list at the top of the box, you should see the name of the line that will

be created (same as the selected typical task). From the *Define with* drop-down list, select the type of data that you will input.

A table occupies the main part of the dialogue box. Each line of the table corresponds to a point of the selected line. The titles of the columns are shown in gray at the top. The number of columns varies and depends on how you have selected to define the points. The available columns are: *A/A, Ch, H, Dx or Length* and *Dh or Slope*. The A/A column is always displayed and shows automatically the serial number of the cross section. The user cannot intervene into this column. In the other columns you must enter the data of the points.

To select a field, left-click in it and then start typing. Press the *Enter* key to confirm the data you have entered and to move to the next field on the right. You can also move between the fields by using the up, down, right and left arrow keys. If you are at the end of a row, you can press the *Enter* key to go to the first field of the next row.

You can insert points in five ways:

1. If you know the X, H coordinates (distance from axis and elevation in meters) of the points that will comprise the line, select from the *Define with:* drop-down list the the *Axis Distance* and *Elevation* option and type the relevant data in the *X* and *H* columns respectively.
2. If you know the horizontal and vertical distances of a point from its previous one, select from the *Define with:* drop-down list the *Dx* and *Dh* from previous option, and then enter the horizontal distance in the *Dx* column and the vertical distance in the *Dh* column.
3. If you know the slope of a point with respect to its previous point and the horizontal distance between these two points, select from the *Define with:* drop-down list the *Dx* and *Slope from previous* option, and then enter the horizontal distance in the *Dx* column and the slope in the *Slope* column.
4. If you know the slope of a point with respect to its previous point and the vertical distance between these two points, select from the *Define with:* drop-down list the *Dh* and *Slope from previous* option, and then enter the vertical distance in the *Dh* column and the slope in the *Slope* column.
5. If you know the slope of a point with respect to its previous point and the slope distance between these two points, select from the *Define with:* drop-down list the *Length* and *Slope from previous* option, and then enter the distance in the *Length* column and the slope in the *Slope* column.

Every time you enter data into one column, the other columns are updated automatically.

To insert a new point at the end of the table, simply select the last column of the last row and press *Enter*. If you want to insert a new point between two existing points, select the immediate next point and press the *Ctrl+I* keys. The new point will be inserted right above the selected one and at the half of the distance from the previous point. This is the initial value, but you can change it. If you wish to delete a point, simply select the row that contains the data of that point and press the *Ctrl+D* keys.

From the same dialogue box you can carry out further operations to edit a line.

9.8 Importing a Line from a Text File

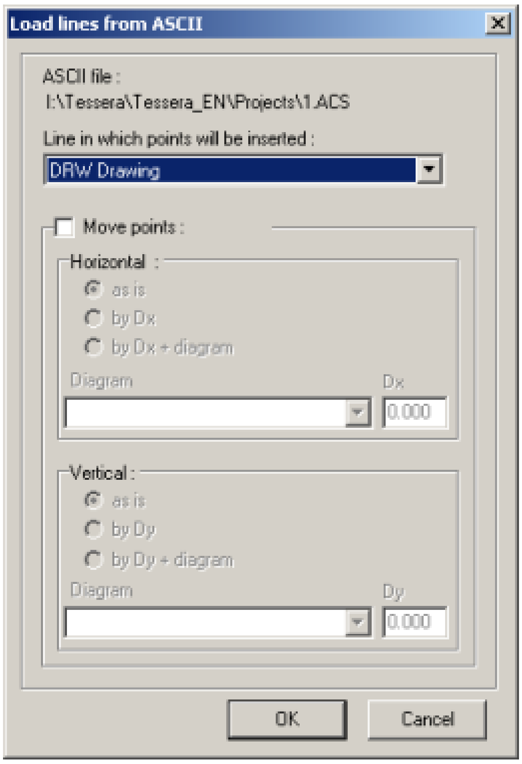
Importing a Line from a Text File (ACS)

This procedure is useful in case you want to import into your file the terrain survey for every cross section (mostly for purposes of measurement) which is available in digital form (ASCII text files). You can apply this procedure at any given time, either during the initial stage of the design or at a later stage, when processing your design.

Especially for the Natural Ground import, the program will also create cross sections, if necessary. Namely, the program reads the text file and updates the Natural Ground line of all cross sections in your project. If the text file contains lines referring to cross sections that are not included in the project, then the program will create these cross sections and will load the Natural Ground line to them.

How to import a line from an ASCII text file

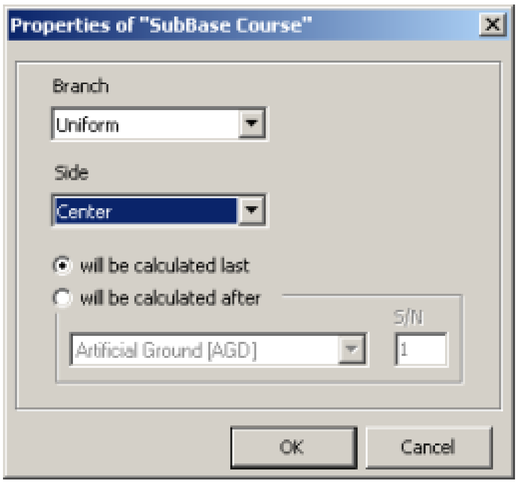
1. From the *File* menu, select the *Import from ASCII* submenu and then run the Lines command. The *Import Lines from ASCII* dialogue box appears on the screen.
2. Select the desired file and click *Open*. If there aren't any cross sections in your project (e.g. the file has derived from the creation of a new cross section project), then the program will assume that the line refers to the Natural Ground line and will create the cross sections and the Natural Ground that corresponds to every cross section.
3. If there is even one cross section in your project, the program will display the following dialogue box,

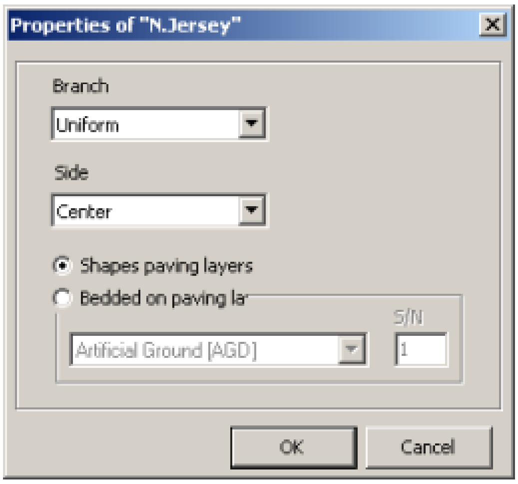


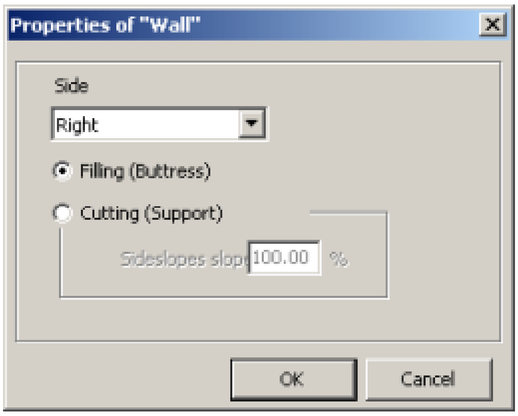
where you must define, from the *Line in which points will be inserted* drop-down list, the line in which the points will be inserted. You can also select, if desired, whether the lines will be moved horizontally or vertically when inserted.
To do this, tick the *Move points:* check box and next define the offset value. In the *Horizontal* and *Vertical* groups, you can define the horizontal and vertical offsets, respectively. If you wish to move the line by a fixed value, tick the by *Dx* or by *Dy*

check box and type the distance in the *Dx* and *Dy* text fields respectively. The as is option means that the line will not be moved towards the selected direction. If you want to move the line towards a direction by a variable distance, which will be different from cross section to cross section and will be determined by the program from a diagram, select the by *Dx + diagram* or by *Dy + diagram* option respectively. In this case, you must select from the Diagram drop-down list the diagram that will be used. You can also add a fixed length to the values of the diagram; simply enter the desired length in the *Dx* and *Dy* text field. If required, you can also choose to invert the diagram signs.

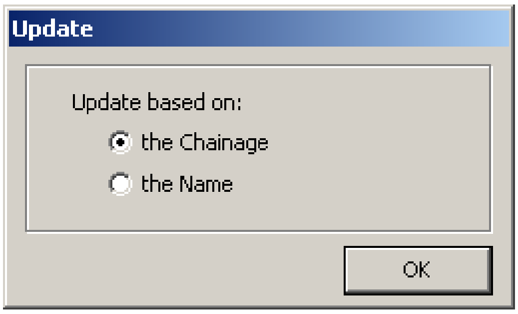
4. After you have made the desired settings, click the *OK* button to proceed to the next stage.
5. If you have selected a typical task of the *WLL, GTT* or *PVL* families, the program will display the following dialogue boxes:







6. Specify the properties of the inserted line and click the *OK* button to proceed.
7. Finally, the program will ask you to specify the way in which it will check whether a cross section in the ACS file exists in your file. Namely, you must select whether the check will be carried out based on the cross section's name or chainage, as shown in the figure below:



If you want the cross sections to be updated based on their names, click the Name button, or, if you prefer the update to take place based on the chainage, click the *Chainage* button.

8. Click the button for the update to take place; the line is displayed in the drawing area.

Specifications of Text Files (ACS).

The cross section data in a file of this type is arranged by groups. Every group-cross section starts with a row with an * and is followed by a row that includes the name and chainage of the cross section (or the distance from the previous cross section) separated by at least one space (e.g. AA 0.00). The cross section points are shown in the next rows (one per row). Every row of a point has two numbers, the first being the distance of the point from the axis and the second its elevation; the two numbers are separated by at least one space.

If the first row of the file has a number, then it is a file with relative chainages, otherwise the file contains absolute chainages. In the case of absolute chainages, the number written next to the name of every cross section represents the absolute chainage of the cross section and the order in which the cross sections appear in the file is insignificant. In the case of relative chainages, the number at the beginning of the file represents the starting chainage and the number written next to the name of every cross section represents its distance from the previous cross section; in this case, the order in which the cross sections appear in the file is critical.

9.9 Editing Lines

The program offers various secondary functions, enabling you to edit the lines you have inserted, so as to give them the desired form and properties.

Main Line Properties

The appearance of a line, both on screen and on the printout, is defined from the *Typical Tasks* dialogue box. From there you can select whether a line will appear on the screen and whether it will be printed. You can also select the color, the type and the width of the line.

Moving a Line

When you move a line, the whole line is transferred, without any changes regarding the orientation or the relative positions of the points with respect to each other. To ensure the necessary accuracy, use the *Input Form* command or the snap feature.

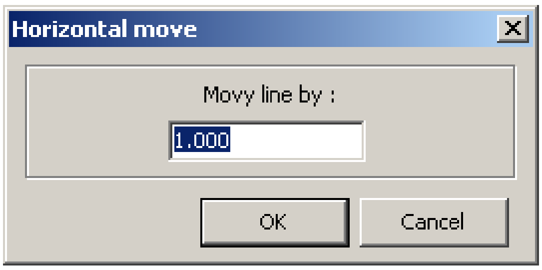
How to move one or more lines at the same time with the mouse

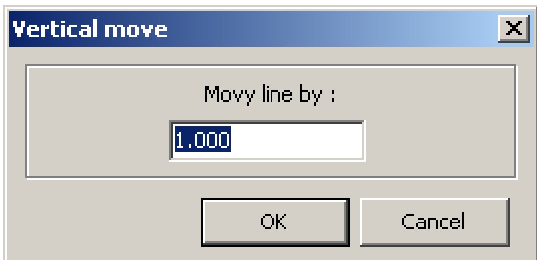
1. Select the lines you wish to move.
2. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Transformations* command and then run the *Move* command.
3. Specify the reference point, based on which the line will be moved.
4. Specify the end point, namely the point where the reference point will be placed.

Once you have specified the second point, the line is moved to its new position.

How to move a line horizontally or vertically

1. Select the line you wish to move.
2. From the shortcut menu that is displayed when you right-click in the drawing area, select the *Edit line* command, or click the  button on the *Lines* toolbar. The particular line must appear selected (highlighted) in the *Line* drop-down list of the resulting dialogue box. Otherwise, select it.
3. To move the line horizontally or vertically, click the  button or the  button, respectively. One of the following dialogue boxes appears on the screen, depending on the button that you clicked:





   In all cases, type the value of the offset in meters in the *Move line by:* field.
4. When you have finished, click the *OK* button to move the line and to return to the *Edit Line* dialogue box. Now you can view the new coordinates of the line that you moved. To return to the main screen, click the exit button on the right of the title bar.

Note that it is also possible to move multiple lines at the same time.

LineRotation

You can rotate a line by selecting a center and a rotation angle towards the horizontal direction.

How to rotate lines

1. Select the lines you wish to rotate.
2. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Transformations* command and then run the *Rotation* command.
3. Use the mouse or the *Input Form* command to specify a point that will become the rotation center.
4. Set the rotation angle either by specifying again a point on the screen, or by entering the angle with the *Input Form* command. If the angle value is positive, the line will be rotated in a counterclockwise direction.

Once you have set the rotation angle, the rotation result appears on the screen.

Changing the Line Size

You can scale up or down a set of selected lines, while maintaining their proportions with respect to each other. To scale the lines up or down, specify a reference point and a length, which will be used as the scale coefficient, or enter the value of the coefficient.

How to change the size of lines

1. Select the lines you wish to scale up or down.
2. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Transformations* command and then run the *Scale* command.
3. Use the mouse or the *Input Form* command to specify the reference point.
4. Set the scaling percentage by specifying a point on the screen with the mouse or by typing the percentage value (e.g. “200” for doubling the size) with the use of the *Input Form*.

Obviously, to scale up an object, the percentage that you will enter must be higher than 100, while to scale down you must enter a percentage lower than 100.

Mirroring Lines

You can mirror certain lines, either by the horizontal or by the vertical axis, by specifying an axis point. The original lines will be deleted.

How to mirror lines

1. Select the lines you wish to mirror.
2. From the shortcut menu, which is displayed if you right-click in the drawing area, select the *Transformations* command and then select *Mirror by X* or *Mirror by Y*, depending on the desired mirroring direction.
3. Use the mouse or the *Input Form* command to specify a point from which the horizontal or the vertical mirroring axis will pass.

Once you have set the point, the mirroring result is displayed on the screen.

Inverting the Direction of Lines

Lines are usually inserted in a clockwise direction. If you wish to change the direction of a line, namely if you want the first point to become last, the second point to become last but one, etc., run the *Invert direction* command.

How to invert the direction of a line

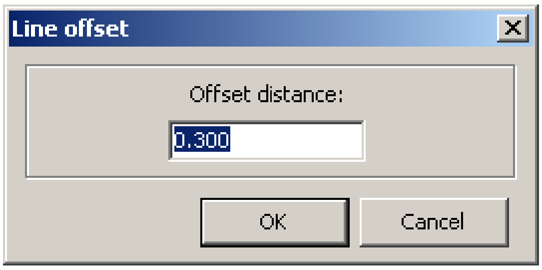
1. Select the desired line.
2. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Transformations* command and then run the *Invert direction* command.

Parallel Offset of Lines

You can offset a line in parallel.

How to offset a line in parallel.

1. Select the line you wish to offset.
2. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Transformations* command and then run the *Offset* command. The following dialogue box appears on the screen:



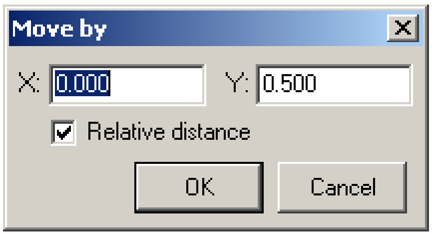
3. In the *Offset distance* text field, type the value of the offset in meters.
4. This offset is first performed separately for every part of the line, offsetting it in parallel to the initial point. In this way, a new line is created and every part of this new line has a fixed distance from the original line (offset function). Therefore, if the value you have entered in the *Offset distance* field is positive, the offset will be upwards (assuming the direction is clockwise), while if the value is negative, the offset will be downwards.
5. When you have finished, click the *OK* button. This closes the dialogue box, returns you to the main screen and applies the offset.

Line Duplication

You can create a duplicate of a line, offset by a certain distance.

How to duplicate a line

1. Select the line you wish to duplicate.
2. From the menu that is displayed if you right-click anywhere in the drawing area, run the *Duplicate* command.
3. The following dialogue box appears on the screen,



where you must enter the offset data of the new line with respect to the old one. If the *Relative distance* check box is ticked, in the X and Y text fields enter the horizontal (Dx) and the vertical (Dy) distance between the new and the old line. If the *Relative distance* check box is not ticked, then in the *X* and *Y* text fields enter the coordinates that the new point will have, namely the point that you had selected before running this command.

4. Click *OK.* This closes the dialogue box and applies the duplication.

The new line that has been created will appear on the screen. This line belongs to the *Drawing* typical task. It is also possible to duplicate multiple lines at the same time.

Deleting a Line

You can delete one or more lines, if desired.

How to delete lines

1. Select the lines you wish to delete.
2. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Delete objects* command, or click the  button on the main toolbar, or press the *Del* key.

If you accidentally delete a line, you can restore it by using the Undo command. It is also possible to delete multiple lines at the same time.

9.10 Editing Line Points

Inserting a Line Point

You can insert one or more points to a line.

How to insert points to a line from the main screen

1. Select the line point next to which you will insert another point.
2. From the shortcut menu that is displayed if you right-click on the screen, run the *Expand to the beginning* or *Expand to the end* command, depending on which side you wish to insert the point (the points of the boundaries have a counterclockwise direction). This command adds a new point to the line on screen, before or after the point you originally selected. The original point is highlighted in red, and you can move the new point by using the mouse pointer.
3. Set the position of the new point either by clicking the left mouse button or by using the *Input Form* command.
4. To finish the point insertion procedure, click the right mouse button. The insertion procedure for the last point that would be inserted to the line is cancelled and the procedure ends.

You can also insert a point from the window of the *Edit line* command.

Moving a Line Point

You can move the points of a line, so as to give the line the desired shape.

How to move a line point

1. Select the line you wish to edit and the point you wish to move.
2. Activate the selected point. The point will start moving, following the movement of the mouse pointer.
3. Specify the new position of the point.

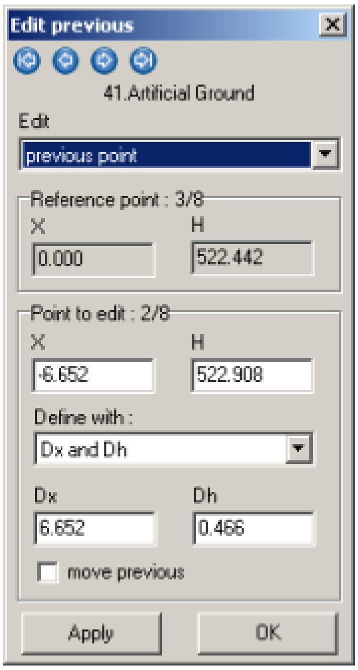
You can also move a point from the window of the *Edit line* command.

Editing a Line Point

From the dialogue box of the *Edit point* command you can edit the position of a line point with respect to its previous or next point.

How to edit a line point

1. Select the line you wish to edit.
2. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Edit point* command. The following dialogue box appears on the screen:



Select a line point and then edit the selected point or its next point or its previous point. The name of the selected line is shown at the top of the box. To change line, activate the box and select the new line from the drawing area. The selected point is the reference point and its data are shown in the *Reference point:* group. You can change the reference point from the toolbar at the top of the box. If you click the first button, the first point of the line will become the reference point. If you click the second button, the previous (from the selected) point will become the reference point. If you click the third button, the next point will become the reference point, while if you click the fourth button, the last point of the line will become the reference point.

3. After you have set the reference point, specify the point to be edited by choosing *previous point* or *current point* or *next point* respectively from the *Edit* drop-down list.
4. Edit the point by entering the appropriate data in the Point to edit group. Simply type the new coordinates of the point in the *x* and *y* text fields. You can also specify the position of the point with respect to the reference point. By selecting from the *Define with:* drop-down list the *Dx* and *Dh* option, you can edit the relative distances from the reference point, by typing the relevant values in the *Dx* and *Dh* fields. The *Dx and slope* option allows you to edit the values of the point's horizontal distance and slope with respect to the reference point. With the *Dh and slope* option, edit the value of the point's vertical distance and slope with respect to the reference point, while with the *Length and slope* option edit the value of the point's distance and slope with respect to the reference point. When the *move previous* check box is ticked, the points on the left of the point that you're moving are also moved by the same distance. When the *Move next* check box is ticked, the points on the right of the point that you're moving are also moved by the same distance.
5. For every change you are making, click the *Apply* button to move the point and update all fields with the new values.
6. To close the dialogue box and return to the main screen applying the changes, click the *OK* button.

<u>Deleting a Line Point</u>

You can delete one or more points of a line.

How to delete a line point

1. Select the line you wish to edit.
2. Select the point you wish to delete.
3. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Delete Point* command or press the *Ctrl+Del* keys.

You can do the same from the dialogue box of the *Edit line* command.

Copying Line Points

You can copy the points of a line to another line.

How to copy the points of a line

1. Select the line whose points you wish to copy. From the menu that is displayed if you right-click in the drawing area, run the *Copy points* command.
2. Select the line to which the points will be copied. From the menu that is displayed if you right-click in the drawing area, run the *Paste points* command. The points of the line will be erased and the new points will be copied.

If you run the *Paste points* command without having selected a line, the program will create a new line and will open a dialogue box, where you must select the project to which the new line will belong. By clicking the *OK* button, the program will create the new line to which the points of the initial line will be copied.

9.11 Creation - Management of Diagrams

When the cross sections are updated from the profile, this means that the profile diagram is inserted. This diagram contains all of the profile elements (PIs, curvature radii) and serves for the determination of the roadway elevation on every cross section.

How to update the Cross Sections from the Profile

1. Go to the *Cross Sections* workspace of the road you wish to update.
2. From the *File* menu, run the *Update from Profile* command. A message will appear on the screen, informing you that the update has been completed successfully. To proceed and return to the cross sections workspace, click *OK*.

You can also update the *Cross Sections* from the *Profile* by using the *Manage* command of the *Project* menu. This command opens the following dialogue box: To proceed to the update, either click the *From Profile to Cross Sections* button in the *Quick Update* group of the *Updates* tab, or click the *Update* button of the *Update* group, after you have selected *From Profile to Cross Sections* in the two drop-down lists of the group. If you use the second way, you can select whether the diagrams and the chainages of the road's cross sections will be updated too, by ticking or unticking the relevant check boxes. You can use the aforementioned commands to update again the *Cross Sections*, in case you make any changes in the *Profile* workspace.

Note that you can also create and edit the profile diagram from the cross sections workspace.

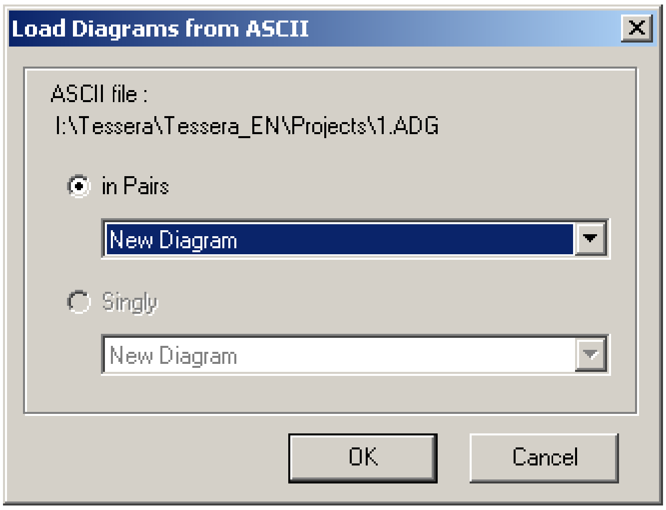
Management of Cross Section Diagrams

From the diagram management window, you can view the profile diagram, as well as all diagrams that are transferred to the *Cross Sections* when updating from the other two workspaces. From the same window you can create your own diagrams, in order to use them either during the cross section calculation that is based on the typical cross section, or for operations that are based on diagrams.

To open the management window, run the *Diagrams* command of the *Parameters* menu. The lines that have been created or edited in this way, can be copied to the *Profile* and to the *Horizontal Alignment,* by using the *Manage* command.

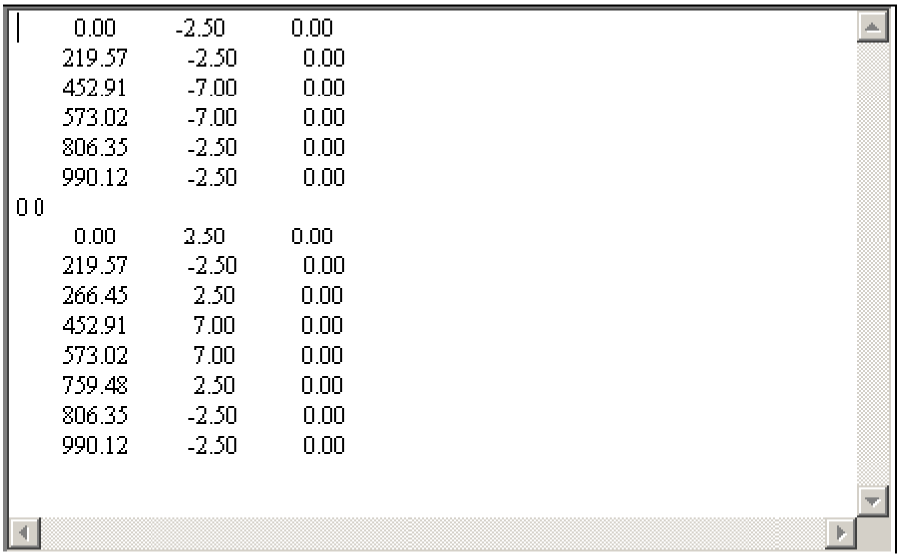
Importing Cross Section Diagrams from Text Files

To import diagrams from an ASCII text file, go to the *File* menu, select the *Import from ASCII* submenu and run the *Diagrams* command. A dialogue box appears on the screen. Select the desired diagram file and click the *Open* button.
The following dialogue box appears on the screen,



where you must select the file format.

The file must have the format shown below,



```
    0.00    -2.50    0.00
  219.57    -2.50    0.00
  452.91    -7.00    0.00
  573.02    -7.00    0.00
  806.35    -2.50    0.00
  990.12    -2.50    0.00
0 0
    0.00     2.50    0.00
  219.57    -2.50    0.00
  266.45     2.50    0.00
  452.91     7.00    0.00
  573.02     7.00    0.00
  759.48     2.50    0.00
  806.35    -2.50    0.00
  990.12    -2.50    0.00
```

and the ADG extension. The data of the diagram in a file of this type is arranged in three columns. Every row has three numbers, the first being the chainage of a point in the diagram, the second being the value of the diagram size and the third being the rounding radius on the point; these three numbers are separated by at least one space. The same file may contain diagrams that are separated in left and right diagram. The end of every group – diagram is marked by a row with two zeros. If the text file contains a diagram, tick the *Singly* check box and from the drop-down list, select the diagram to which you wish to import the text file data. If both diagrams (left and right) are included in the text file, tick the in *Pairs* check box and from the relevant drop-down list select the diagram in which the data will be saved.

Click the *OK* button to import the diagram to the file. If you select *New Diagram* from the drop-down list, the screen displays the relevant dialogue boxes, in which you must type the names of the diagrams to be imported. Type the name of every diagram and then click the *OK* button.

Exporting Cross Section Diagrams to Text Files

The drawings can be saved in an external text file.

How to export the drawings to an ASCII file

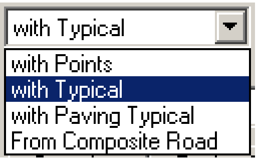
1. From the *File* menu, select the *Export to ASCII* submenu and run the *Diagrams* command.
2. The following dialogue box appears on the screen: If you wish to include only one diagram in the text file, tick the *Singly* check box and from the drop-down list select the diagram you wish to export. If you want to export a pair of diagrams to the text file, tick the in *Pairs* check box and from the relevant drop-down list select the desired diagram.
3. The relevant dialogue box appears on the screen.
4. Specify the path and the name for the file you wish to save. The selected diagram will be saved in a file that will have the name that you specified and the ADG extension.
5. Click *OK* to create the file.

9.12 Cross Sections Calculation

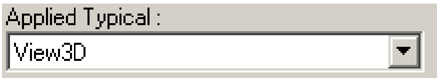
For the cross sections to be calculated, the user or the program must create the main components of the lines, then extend or trim parts of these lines, so as to create between them closed surfaces, and finally measure the area of these surfaces. The calculation of a cross section and the setting of the calculation parameters are carried out through the *Cross Sections Data* management window. This operation can be carried out collectively for all of the project's cross sections.

Before you press the 

button for the program to "close" the cross section, you must first select the type of the cross section from the *Cross Section Type* drop-down list.



Select one of the following types: *with Points, with Typical, with Paving Typical, from Composite Road*. If you prefer to apply a typical cross section, use the drop-down list to select the typical to be applied.

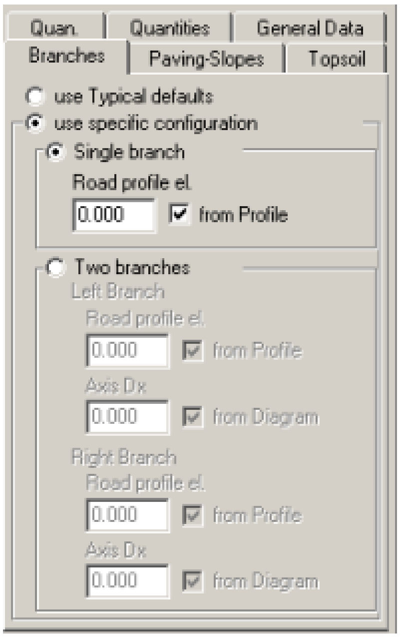


The cross section type determines the way in which the program will proceed during the calculation, namely:

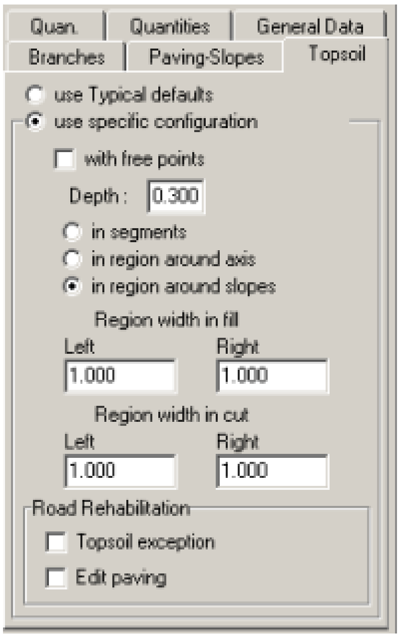
- When the cross section type is *with Points,* the program does not create any new lines, but extends or trim the existing lines.
- If you have selected the with *Typical* type, the program deletes the existing lines of the PVL, GTT, AGD, LGD and SCC families and creates new lines of the PVL, GTT, AGD, LGD, DRW, GRL and SCC families. If the cross section includes lines of other families, these lines will not be deleted and they will be used for the shaping of the cross section's typical. The lines that are inserted to the cross section and the way in which these lines will be calculated, are directly related to the selected typical cross section that will be applied.
- When calculating *with Paving Typical*, the procedure is the same as the one of the calculation *with Typical*; the only difference here is that the typical's paving section (that corresponds to a cutting or a filling) is applied to every cross section, leaving the *Artificial Ground* line unchanged, in the exact shape that it had before the paving typical is applied.

Cross Section Calculation Parameters

The cross sections, as well as the typical cross sections, are comprised of four layers: the branches, the topsoil, the sideslopes and the pavings. You can define the form of these elements from the following tabs of the *Cross Sections Data* window:



Branches: The cross sections can have one or two branches. A single- branch cross section is the one whose paveway is continuous. An axis appears in the middle of the single-branch cross sections; the road profile elevation is applied to this axis. The single-branch cross sections are split in two sections, the left and the right section. If the paveway is split, then the cross section is constituted of two branches. The two branches can have a fixed distance between them or the distance can vary from one cross section to another. To set the parameters for a particular cross section, go to the *Branches* tab of the *Cross Sections Data* window.
The central axis also appears in the two-branch cross sections,
but there are also two secondary semiaxes, one on the left of the central axis and the other on its right. Each of the semiaxes corresponds to a branch. The road profile elevation is applied to both semiaxes. Every branch of the cross sections is split in two parts, left and right.



Topsoil: You can set the topsoil type in four different ways. If you use any of the first three ways, the topsoil is created automatically by the program and it is parallel to the Natural Ground line. By using the fourth way, you can create an independent line. To select one of the four ways, go to the *Topsoil* tab of the *Cross Sections Data* window.
You can also select whether the topsoil will extend parallel to the Natural Ground line and in segments (which are user-defined) or around the road axis or around the points where the Artificial Ground Line intersects with the natural terrain or, finally, whether under the Natural Ground there will be a line having its own geometry. In the first three cases you must enter the topsoil depth, while in the cases where there are slopes in a region around the axis or the slopes, you must enter the right and left distances separately for the cuttings and the fillings. If you do not wish to have any topsoil, the value in the *Depth* text field must be zero. All values are in meters (m). If you have selected the topsoil to be in segments, then use the mouse to specify these segments in the drawing area. Select the topsoil line and more specifically the right part of the segment you wish to modify, and from the menu that is displayed if you right-click in the drawing area, run the *Change Topsoil* command. In this way you can specify whether there will be topsoil in the particular segment.

Cross Section with Points

If the cross section is *with Points*, then, when the program searches for intersections, it will not delete any lines, but it will attempt to close the cross section by extending or trimming the appropriate lines. For calculations of this type, you must set the parameters referring to the branches and the topsoil.

The roadway elevation cannot be defined parametrically if the cross section is *with Points* (you cannot use the typical cross section). The road profile elevation is the elevation of the intersection point between the axis (or the semiaxes in the case of two-branch cross sections) and the Artificial Ground line.

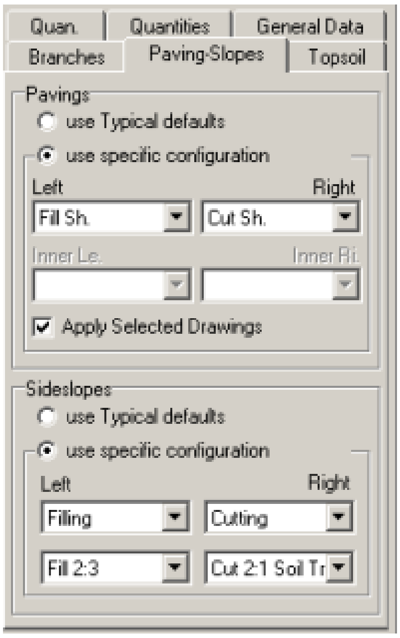
If your cross section has two branches, in order to view both semiaxes you must set their distances from the main axis of the cross section.

Cross Section with Typical

If you select the *with Typical* calculation way, the existing paving lines that have been inserted automatically by the program, as well as the Artificial Ground line, regardless of how it was inserted, are both deleted and re-created based on the user-selected typical cross section. The user is not limited when using this method for calculating a cross section. If you wish, some of the parametrical values of a cross section to which a typical cross section is applied, can be different than the values that you have set for the typical cross section, without having to change the typical cross section itself.

To set the topsoil, from the *Topsoil* tab you can select the same settings that you have selected for the typical cross section by ticking the use *Typical defaults* check box. If you wish to configure the topsoil of the particular cross section in another way, tick the use *specific configuration* check box and define the way in which the topsoil will be calculated.

To define the branches and the road profile elevation on the axis (or the semiaxes), from the *Branches* tab, you can also select between the defaults and the specific configuration, by ticking the respective check box. In the second case, you can choose whether a cross section will have one or two branches by ticking the relevant check box. To set the road profile elevation, type the desired value in the *Road profile el* (absolute way). If you wish the roadway elevation value to be relative to the road profile diagram, tick the *From Profile* check box (relative way). For two-branch cross sections, if you wish both semiaxes to have a fixed distance from the main axis of the cross section, type the distance in the Axis Dx text field both for the left and for the right semiaxis. If this distance is not fixed, then the program can calculate the distance based on the cross section chainage and a specific diagram. All you have to do is tick the *From Diagram* check box and the program will calculate the distance that derives from the Dx diagram of the left or the right semiaxis.



To create the paving and slope lines, you must set their parameters. To do this, go to the *Paving - Slopes* tab, which contains two groups, one for the *Pavings* and one for the *Slopes*. Here too, you can tick in every group the use *Typical defaults* check box to apply the typical to the cross section or you can use a *specific configuration* by ticking the relevant check box.

In addition, by unticking the *Apply Selected Drawings* check box, you can deselect, at the particular cross section, the drawings that have been applied based on the typical. To display again these drawings, tick the same box.

Pavings: This is the set of lines and structures over the Artificial Ground and the Artificial Ground Ledge. These lines comprise the cross section paveway and they can be created either directly by the user or from the typical and then be applied to the cross section.
Sideslopes: This group refers to the sloped segments of the Artificial Ground. They can be created either by defining a sequence of slopes from the typical or the user can create them by designing them directly into a specific cross section. The sideslopes are applied until an intersection with the final natural ground envelope is found. Their configurations for the cutting and for the filling are different.

As you will see analytically in the next chapter, a typical cross section in *Anadelta Tessera* may contain many sets of pavings and many sets of sideslopes. However, only one set of pavings or sideslopes is selected and applied when the specific typical cross section is first applied to a cross section. Next, for a particular cross section, you can select any of the predefined configurations that you have defined in the typical and apply this configuration both to the left and to the right part of every branch. To do this, tick the *with specific configuration* check box and select the desired configurations from the relevant drop-down lists. The same applies to the sideslopes. Namely, here too you must tick the *with specific configuration* check box in order to select from the drop-down lists the type of the configuration (cutting or filling) and then the slopes sequence for the specific configuration.

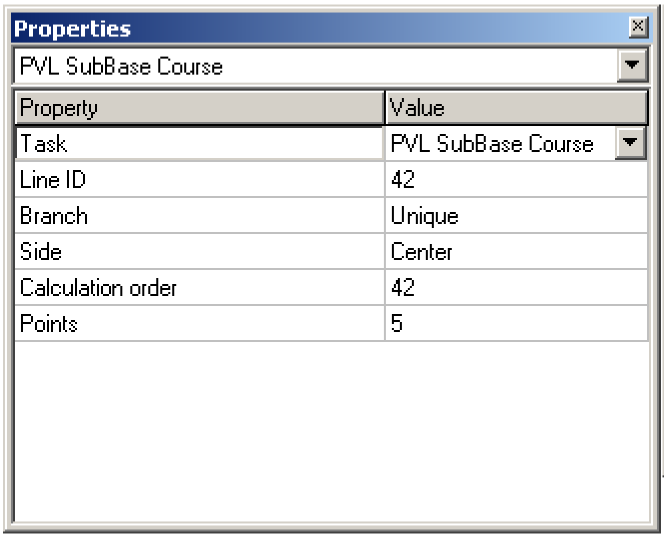
Cross Section with Paving Typical

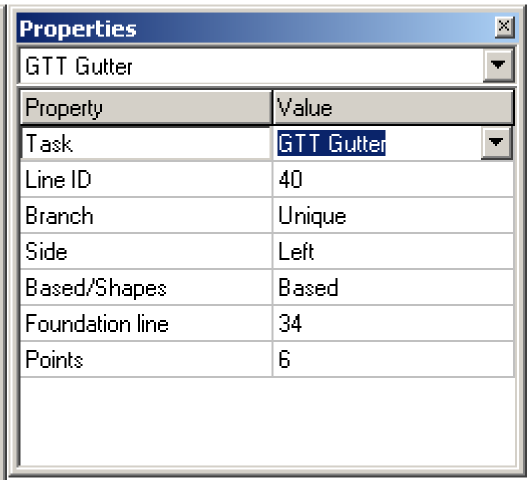
The calculation method with paving typical is the same as the one with typical, the only difference is that the Artificial Ground task will not be deleted. Namely, during the calculations, the program uses all elements of the typical cross section that refer to the paving, but it does not use the elements that refer to the Artificial Ground. The Artificial Ground is calculated in the same way as in the case of the calculation *with Points*.

The settings that you can make are the same as the settings of the cross section with typical; the only difference here is that the *Sideslopes* group is inactive.
The lines referring to the paving are the lines of the PVL (Paving Layer), SSC (Coating) and GTT (Gutter) types. The cross section type does not play any role as regards the "closing" of the other lines (e.g. benches, improvements, unsuitables, etc.), since these lines have nothing to do with the use of the typical cross section.

LineCalculation Properties

Every line that belongs to the PVL (paving layer) or to the GTT (gutter) family has certain additional properties compared to the lines that are defined from the typical tasks and that play a role in the calculation – closing of the particular lines and therefore of the cross sections containing these lines. You can set these properties from the following dialogue boxes.





To open either of these boxes, select a line of the PVL or GTT family, click the right mouse button and from the displayed menu select *Line properties*.

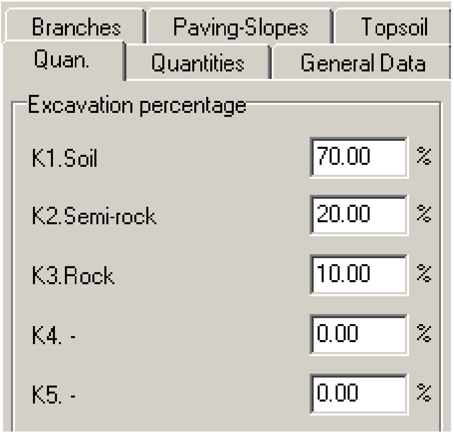
Since a cross section may have two branches, a basic information for a line belonging to either of the above families is the branch to which this line belongs. In the case of a single-branch cross section, your only option is *Unique,* while in the case of a two-branch cross section, you can select *Left, Right* or *Undefined*, depending on the branch to which the line belongs. You can also select the side to which the line belongs by entering the appropriate side, namely *Left, Right, Centre* or *Undefined*, in the relevant field. Finally, if a line belongs to the encasement solids (GTT), you must define whether the solid will be based on a paving layer or whether it will shape the paving layer. If the solid is based on a line, in order to define this line you must select the Foundation line field, click the three-dot button on the left of the field, and use the mouse pointer to select the foundation line of the solid within the drawing area. The field is updated with a number, which corresponds to the particular line, and once you've run the calculation of the typical, the solid will be based on this line.

Especially for the encasement solids, their branch side also defines the behavior of the layers when they meet such solids. Therefore, if they belong to the *Left* or *Right* side, the layers meeting this solid end there and they do not continue on the other side of the solid, namely outside the road. On the contrary, when the solid is in the *Centre*, then the layers it meets up to the foundation layer are automatically split in two identical layers, one on the left and one on the right of the solid.

As you will see in details in the next chapter, which refers to the typical cross sections, all of the aforementioned lines are sorted out in order of calculation. If the cross section derives by applying a typical *(with Typical or with Paving Typical)*, the order of calculation for every line is predefined. If the cross section is calculated *with Points*, the calculation order of the lines belonging to the aforementioned families, corresponds to the order these lines were created or inserted. In order for you to get the desired result, you must set the calculation order manually. To do this, run the Line properties command as described above and then select the *Calculation Order* field. Click the three-dot button on the left of the field, and use the mouse pointer to select the line after which the selected line will be calculated.

Excavation Percentage

To set the excavation percentage for the current cross section, open the *Quan.* tab of the *Cross Sections Data* window.

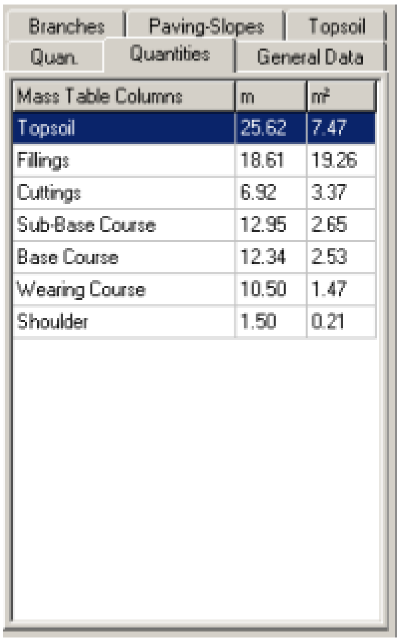


In the tab fields, type the excavation percentages, which can be grouped in up to five categories. If you change the percentages, click again the *Calculation* button to calculate the new quantities. You can also proceed to the batch calculation of the excavation percentages for all cross sections.

9.13 Quantities - Areas

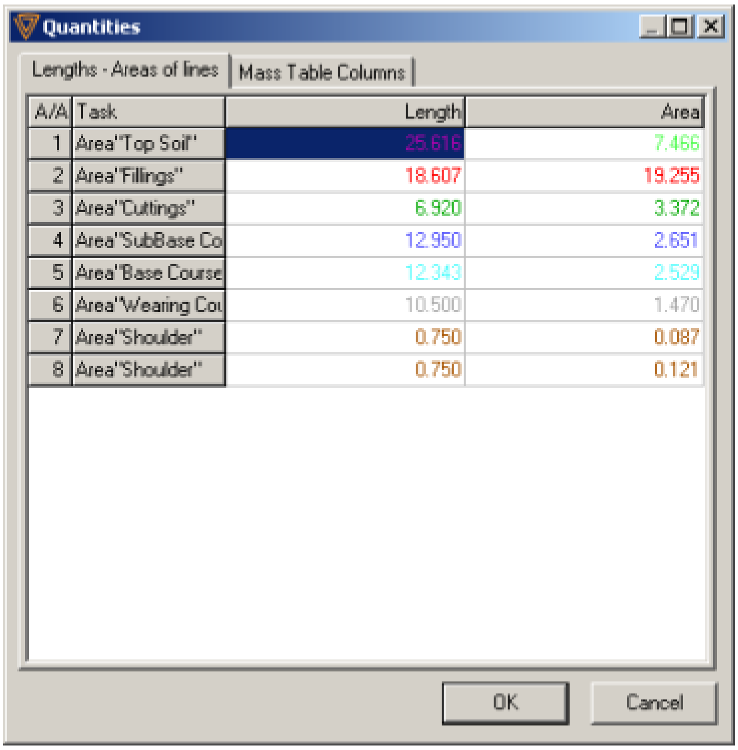
During the calculation of a cross section, the program calculates the length and the area of every cross section line. You can select the area by clicking the left mouse button on the relevant region within the drawing area. By selecting the area, the relevant region is colored. The data of the selected area appears at the bottom of the *Status bar*.

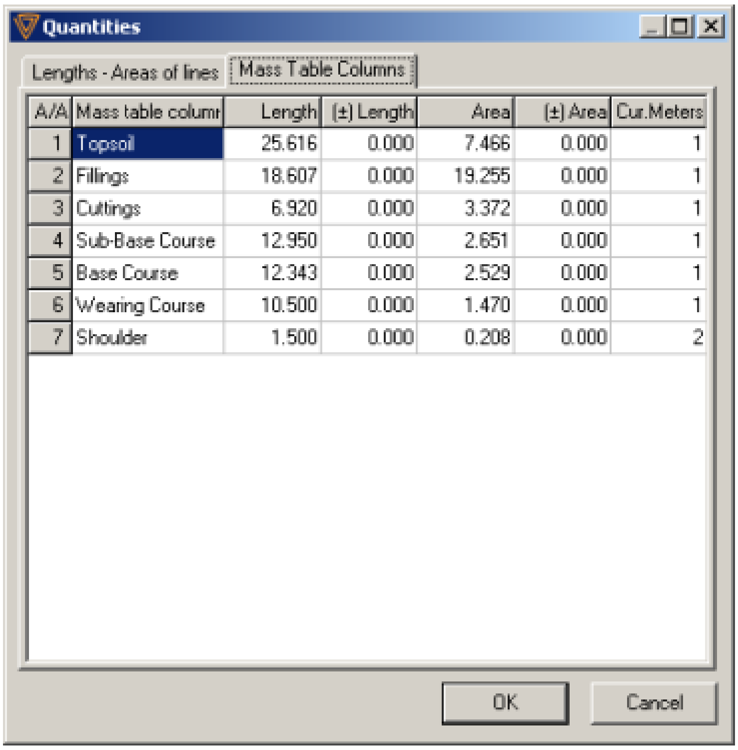
From the *Quantities* tab of the *Cross Section Data* window, you can quickly check all calculated quantities (length - area) of the current cross section, as these quantities are linked together in the columns of the mass table.



In the window of the *Quantities* command of the *Actions* menu, you can view all calculated quantities of the current cross section. You can also run this command by clicking the  button on the *Lines* toolbar.

This command opens the following dialogue box, which contains two tabs:





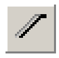
In the *Lengths – Areas of lines* tab, you can see in a table all the lines of the current cross section, as well as the lengths and areas that have been calculated for these lines.

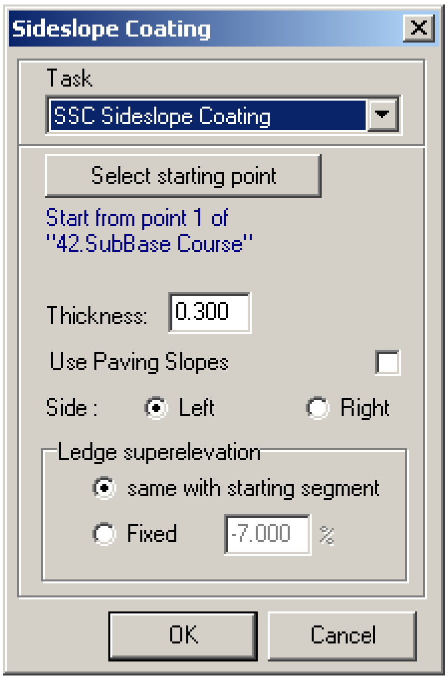
In the *Mass Table Columns* tab of the same window, you can see all columns of the mass table for the particular cross section, together with the *Length, Area* and *Current Meters* quantities for each column. If desired, you can change the quantities contained in these columns. If the cross section is recalculated, the quantities are restored automatically in their original values. If you want to add or subtract a fixed value to/from the quantities, then enter the desired value in one of the *(±)Length* and *(±)Area* columns, respectively.

9.14 Sideslopes Coating

The program can create automatically a sideslope coating, which is parallel to the sideslopes of the current cross section, even if such coating does not exist in the typical cross section. The sideslopes coating is inserted after the calculation of the cross section.

How to create the sideslopes coating for a cross section

1. From the *Actions* menu, run the Insert *Sideslopes Coating* command, or click the  button on the *Lines* toolbar.
2. The following dialogue box appears on the screen:



3. In this box, select from the *Task* drop-down list the typical task to which the new line will belong. The list shows only the typical tasks that belong to the SSC family. The task to select is usually the *Sideslopes Coating*.
4. The next step is the determination of the line's starting point. Select the line that contains the point from which the coating will begin and make this point current. Then click the *Select starting point* button. You will notice that the message exactly below the field changes from *Starting point not defined* to a message that describes the selected point, e.g. Start from point 2 of Shoulder. The Side field is updated automatically with the side of the selected line that contains the starting point of the sideslopes coating. Next, in the *Thickness* field, type in meters the distance of the coating from the sideslope. If you wish to also use the paving sideslopes for creating the line, tick the *Use Paving Slopes* check box.
5. Click the *OK* button to close the dialogue box; the line will be created.

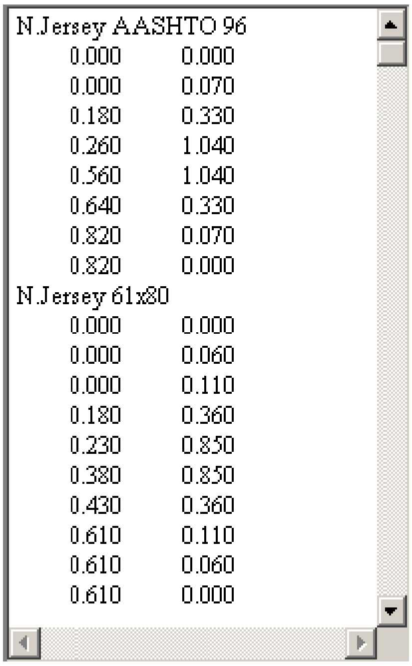
9.15 Drawings

The term drawings means predefined line types, which are stored in an internal library and can be imported as a new line to a cross section. The drawings can be imported to the internal library either from a text file, or they can be created by the user.

Creating an Internal Drawing Library

To save a line of a cross section in the internal drawing library of a project, select the line and from the menu that is displayed if you right-click in the drawing area, run the *Add to Drawings* command of the *Tools* submenu.

To import drawings from a text file, go to the *File* menu, select the *Import from ASCII* submenu and run the *Drawing Library* command. The *Drawing Library from ASCII* dialogue box appears on the screen. Select the desired file and click *Open*.



```
N.Jersey AASHTO 96
      0.000    0.000
      0.000    0.070
      0.180    0.330
      0.260    1.040
      0.560    1.040
      0.640    0.330
      0.820    0.070
      0.820    0.000
N.Jersey 61x80
      0.000    0.000
      0.000    0.060
      0.000    0.110
      0.180    0.360
      0.230    0.850
      0.380    0.850
      0.430    0.360
      0.610    0.110
      0.610    0.060
      0.610    0.000
```

The text file with the drawings must have the format shown below, and the .ADR extension. The drawing data in a file of this type is arranged by groups. The first row of every group-drawing contains the name of the drawing. The next rows contain the points of the drawing. Every row has two numbers, the first being the X coordinate of the point and the second the Y coordinate; these two numbers are separated by at least one space. The end of every group–drawing is marked by an empty row.

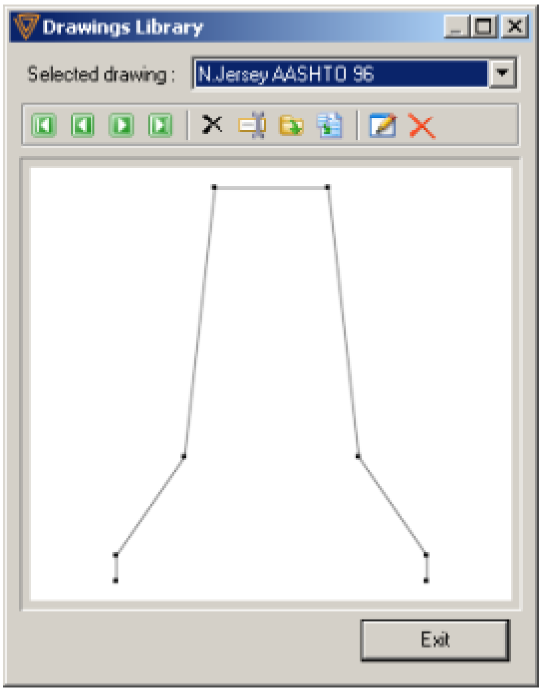
You can also import drawings from the drawing management window.

<u>New Project's Internal Drawing Library</u>

The drawings of a new project can also be imported from an external file of drawings (*.ADR). You can select this file from the *General Options* dialogue box of the *Tools* menu. More specifically, the full name and the path of the file are displayed in the *New project drawing library* field of the i tab. If you wish to select a new file, click the button. Select the desired drawings file from the dialogue box that will appear on the screen. Once you've made your selection, click the *Open* button to return to the *General Options* dialogue box, and then click the *OK* button to close the dialogue box and return to the main screen.

Drawings Management

You can manage the drawings from the window of the *Drawing Library* command of the *Parameters* menu.



This command opens the management window that contains all drawings of a project, enabling you to view and edit the drawings of a project's internal library. The drawing that has been selected from the *Selected drawing* drop-down list appears in the centre of the window. To view the drawings, use the toolbar at the top left of the box . The first button shows the first drawing. The second button shows the previous (from the selected) drawing. The third button shows the next drawing and the fourth button shows the last drawing.

If you wish to give a new name to the selected drawing, click the button; this opens a dialogue box, where you can enter the name by which the file will appear in the library. You can even delete the selected drawing from the project's library by clicking the button. To delete all drawings, click the button.

To import drawings from ASCII text files, click the button. To import the drawings contained in the library of another *Tessera* project, click the button. In both cases, the familiar dialogue box will open, and you can select the desired file and then click the *Open* button.

Export of Drawings to an ASCII Text File

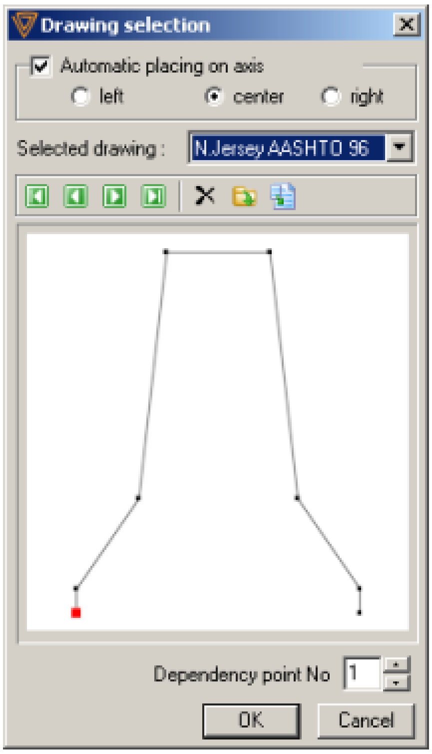
The drawings of a project's internal drawing library can be saved in an external text file.

How to export the drawings to an ASCII file

1. From the *File* menu, select the *Export to ASCII* submenu and run the *Drawing Library* command.
2. Specify the path and the name for the file in which you will save your drawings. The file will have the .ADR extension.
3. Click *OK* to create the file.

Importing a Drawing to a Cross Section

To import a drawing to a cross section, proceed in the same way as you would do for creating a line, and in the dialogue box, tick the *from library* check box. Click *OK* to open the following dialogue:



From the *Selected Drawing* drop-down list, select the drawing to be imported. The buttons on the toolbar have the same functions as the buttons in the management window.

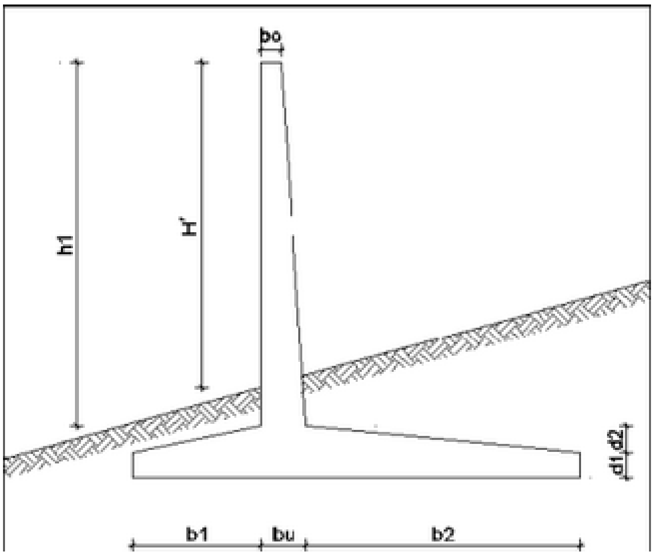
In the *Dependence point* field, specify the point whose position you must define in order to insert the drawing to the cross section.

Click the *OK* button to close the dialogue box. The screen displays the drawing, which you must position either by using the mouse to select a point within the drawing area (in this case it is recommended to activate the snap on point feature), or by using the *Input Form.* This operation can be carried out collectively for all of the project's cross sections.

9.16 Walls

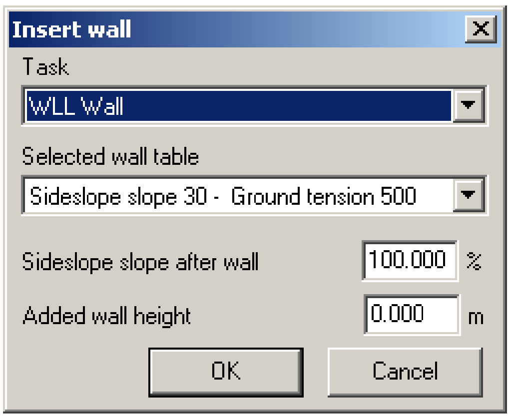
Generally, the walls in *Anadelta Tessera* are divided in two categories, depending on the position they will get. They can be either *Filling* walls, commonly used for supporting the soil at the foot of the filling slopes, or *Cutting* walls, which are commonly used to retain the soil of the cutting slope. You can insert walls collectively for all of the project's cross sections.

Generally the walls can be inserted like any other line. You can also insert a wall by selecting from a table containing the dimensions of the wall, based on the wall's position in a cross section. With this method, you can insert a wall to the current cross section and specify only the distance of the wall from the cross section axis. The height, the type and the individual dimensions of the wall are calculated automatically by the program. The dimensions are selected according to the net height of the wall, from wall tables, which contain geometrical characteristics based on the allowable tension of the ground and the slope of the filling sideslope behind the wall. The dimensions specified in these tables are as follows:



To insert a wall with this method, run the *Insert Wall* command from the *Actions* menu, or click the  button on the Lines toolbar.

This command opens the following dialogue box,



where you can set the wall insertion parameters. Note that, before inserting a wall in this way, you must have calculated the cross section.

From the *Task* drop-down list, select the typical task to which the inserted line will belong. By left-clicking on the list, you can view all typical tasks of the family (WLL); select one of the displayed tasks. In the *Selected wall table* drop-down list, choose the table that will be used for the selection of the appropriate wall. You can select among ten tables, which contain all of the wall's geometrical data.

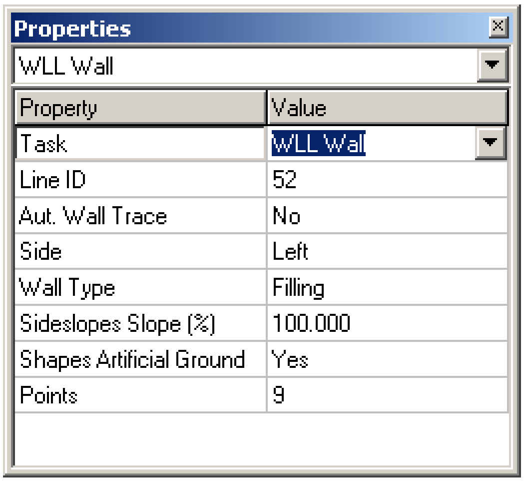
In the *Sideslope slope after wall* field, type the value of the sideslope slope outside the wall. A sideslope after the wall will only exist in the case of a support wall. In the *Added wall height* field, type the height that the wall will have above the natural ground or the top envelope, depending on whether the wall is positioned in a cutting or filling, respectively.

In all cases, click *OK* to insert the wall to the cross section, or click *Cancel* to abort the procedure. The *OK* button transfers you to the screen, where you must specify the position of the wall. The shape and the dimensions of the wall change each time, in order for the wall to be adapted to the existing geometry of the lines. Click the left mouse button to finalize the position of the wall, or click the right mouse button to stop the procedure without inserting the wall.

Properties of the WLL Family Lines

You can select whether the presence of the walls in the cross section will modify or not the Artificial Ground. If you select the Artificial Ground not to be affected, then its configuration will not depend on whether it meets or not the wall line, while in the opposite case, the Artificial Ground will change according to the type and shape of the wall. On the contrary, the Paving Layers, the Encasement Solids and the Sideslopes Coating will always stop on the wall line.

To edit the properties of a wall, select the wall and run the *Properties* command.



The above figure shows the dialogue box with the wall properties that you can edit.

If you want the wall line to take part in the shaping of the Artificial Ground, select Yes in the *Shapes Artificial Ground* drop-down list. Otherwise select No (inactive wall); the wall line will exist in the cross section (mainly for the measurement of the cross section area) but it will not modify the *Artificial Ground*. If you select Yes (active wall), then the Artificial Ground will change from the first point of its intersection with the Wall line, according to your selections as regards the other properties of the wall.

For the Artificial Ground to be shaped again, after you have activated or deactivated the wall, the cross section must be recalculated.

Importing Wall Tables

To import tables from an ASCII text file, go to the *File* menu, select the *Import from ASCII* submenu and run the *Walls Library* command. The *Import Walls Library from ASCII* dialogue box appears on the screen. Select the desired file and click *Open*.

The file must have the format shown below,

0.00	100.00						
0.40	0.70	0.30	0.30	0.30	0.50	0.30	0.00
0.90	1.20	0.30	0.30	0.30	0.80	0.30	0.00
1.20	1.50	0.30	0.40	0.60	0.80	0.30	0.20
1.70	2.00	0.30	0.40	0.60	1.00	0.30	0.20
2.20	2.50	0.30	0.40	0.60	1.30	0.30	0.20
2.70	3.00	0.30	0.50	0.75	1.45	0.30	0.20
3.20	3.40	0.30	0.50	0.90	1.90	0.30	0.30
3.70	3.90	0.30	0.60	0.95	2.45	0.30	0.30
4.20	4.30	0.30	0.60	1.00	3.00	0.40	0.30
4.70	4.80	0.30	0.60	1.40	3.10	0.40	0.30
5.20	5.20	0.30	0.70	2.00	3.10	0.40	0.40
5.70	5.70	0.30	0.70	2.20	3.20	0.40	0.40
6.20	6.20	0.30	0.80	2.20	3.30	0.40	0.40
6.60	6.60	0.30	0.90	2.50	3.10	0.50	0.40
7.00	7.00	0.30	1.00	2.80	3.20	0.50	0.50

and the .AST extension. The wall data in a file of this type is sorted out in groups, depending on the allowable ground tension and the ground's sideslope slope behind the wall. The first row of every group contains two values, the sideslope angle in degrees and the allowable ground tension. These two values are separated by at least one space (e.g. 0 300). The next rows show the dimensions of the walls with respect to the useful wall height H?. By order of appearance, the data shown in the table are: H?, h1, bo, bu, b1, b2, d1 and d2. When importing a wall, the program, based on the calculated H?, provides the dimensions of the wall by using the other values of the row corresponding to the H?. The end of every group is marked by an empty row.

Wall Tables of a New File

The wall tables of a new project are also imported from a wall table file (*.AST). This file is selected from the *General Options* dialogue box of the *Tools* menu. More specifically, the full name and the path of the file are displayed in the *New project wall library* field of the *Cross Sections* tab. If you wish to select a new file, click the ... button. Select the desired typical tasks file from the dialogue box that will appear on the screen. Once you've made your selection, click the *Open* button to return to the *General Options* dialogue box, and then click the *OK* button to close the dialogue box and return to the main screen.

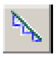
Exporting Wall Tables to an ASCII Text File

The wall tables can be saved in an external text file.

How to export wall tables to an ASCII file

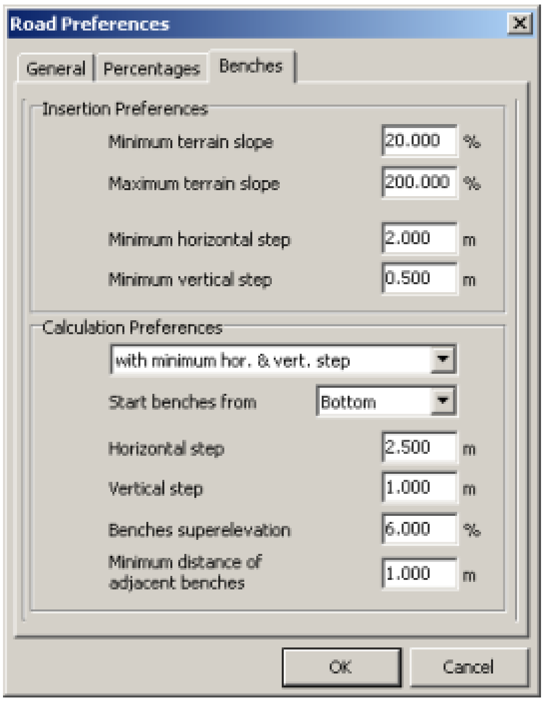
1. From the *File* menu, select the *Export to ASCII* submenu and run the *Walls Library* command.
2. Specify the path and the name for the file you wish to save. The wall tables will be exported to a file with the (*.AST) extension.
3. Click *OK* to create the file.

9.17 Benches

The *Insert Bench* command of the *Actions* menu is used for inserting automatically a bench into the current cross section. You can also run this command by clicking the  button on the *Lines* toolbar. This command results in the creation (if required for the cross section) of a bench line, which always belongs to the *Bench Typical Task* (BNC). The shape and the insertion way of the benches depend on the parameters that you have set in the *Benches* tab of the *Road Preferences*. You can also insert benches collectively for all of the project's cross sections.

Bench Preferences

To set the bench insertion preferences, run the *Road Preferences* command of the *Parameters* menu and then open the *Benches* tab. The resulting dialogue box has two groups.



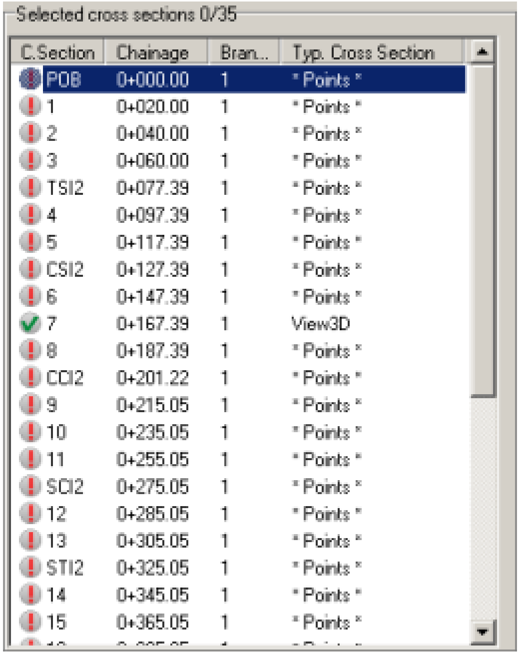
The *Insertion Preferences* group contains settings referring to the criteria according to which the program will determine whether a bench will be inserted or not to a cross section. In the *Minimum terrain slope* (%) and *Maximum terrain slope* fields, enter the minimum and maximum slope that two successive points of the natural sideslope must have, in order for a bench to be inserted between them. In the *Minimum horizontal step* and *Minimum vertical step* fields, enter the minimum horizontal and vertical distance that two successive points of the natural terrain must have, in order for a bench to be inserted between them. A bench cannot be inserted between points whose slope is between the limits that you have set, but their in-between vertical distance does not exceed the minimum value.

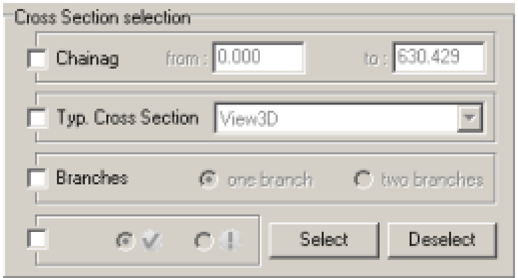
The *Calculation Preferences* group contains settings referring to the shape of the inserted bench. In the *Start benches from* field, specify the way in which the benches will be inserted. Namely, if you wish to insert the first complete step to the highest or to the lowest point of the section to which the bench will be inserted, from the relevant drop-down list select *Top* or *Bottom,* respectively. In the *Horizontal step* field, enter the horizontal distance – width of the benches, while in the Vertical step field enter the bench height. In the *Benches superelevation* field, enter the superelevation of each bench. When inserting benches automatically, the program uses the insertion preferences to determine the segments to which the bench lines will be inserted and then shapes the lines according to the bench preferences. If there are two adjacent bench segments and you wish to join them into a single part, enter the desired value in the *Minimum distance of adjacent benches* field. The program joints automatically the benches whose in-between distance is smaller than the value which has been entered in this field and leaves the benches whose distance is longer than this value unaffected.

The shape of the benches is determined according to the method that you have selected from the drop-down list at the top of the *Bench Preferences* group. With the option with minimum horizontal step, the width of the benches equals to the value that you have entered in the *Horizontal Step* field. If this cannot be applied (usually at the end of the line), the resulting bench is wider but never narrower. The option with minimum vertical step is similar, only here the bench height remains fixed at the value that you have entered in the *Vertical Step* field. With the option *with minimum horizontal and vertical step*, the width of the benches is the value in the *Horizontal Step* field and their minimum height is the value that you have entered in the *Vertical Step* field. If this cannot be applied, the program will make the benches wider.

9.18 Batch actions

Many of the operations of the *Cross Sections* workspace can be carried out collectively for a group of cross sections. Most of these operations can be found in the *Batch Actions* menu. Any command contained in this menu opens a relevant dialogue box. These dialogue boxes contain, among others, two common groups: *Selected cross sections* and *Cross section selection*.





From these groups you can select the cross sections you wish to edit.

The *Selected cross sections* group is comprised of a window with four columns, which contain the names of all cross sections of the current road, their chainages, the number of the cross section's branches, and the calculation way of the cross section. To select a cross section from the table, left-click on the cross section and its line will appear colored. An icon appears on the left of every cross section's name. The ✔ icon appears next to the calculated cross sections, while the ! icon appears next to the uncalculated cross sections. The number of the cross sections that have been selected for editing and the total number of cross sections appear next to the name of the group (e.g. 10/26).

To select only one cross section, select the cross section from the window and press the *Space* bar on the keyboard. If you wish to deselect the cross section, click again the same button. You can also carry out these operations by using the *Insert* and *Delete* keys respectively, or by double-clicking with the mouse. The selected cross sections appear in blue fonts.

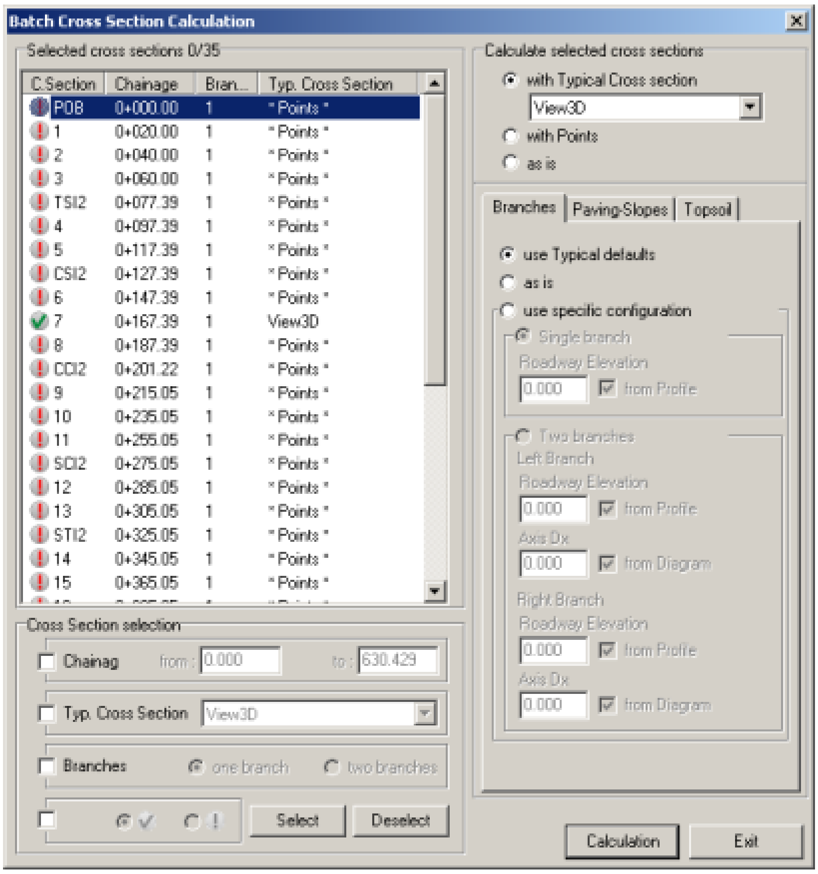
To select all cross sections, click the *Select* button at the bottom of the *Cross Section* selection group. To deselect all cross sections, click the *Deselect* button.

The program gives you four more options. You can select or deselect certain cross sections between two chainages. To do this, tick the *Chainage* check box and then enter the respective chainages in the *From* and *To* text fields. Click the *Select* button to add the cross sections to the existing set of the selected cross sections or click the *Deselect* button to remove them.

If you wish to select the cross sections that have been calculated based on a typical cross section, tick the *Typ. Cross Section* check box and then select the typical cross section from the drop-down list. You can also select only single-branch cross sections or only two-branch cross sections; to do this, tick the *Branches* check box and then select *one branch* or *two branches*, respectively. Finally, you can select only the cross sections that have been calculated or only those that have not been calculated, by ticking the last check box and making the appropriate selection. In all cases, click the *Select* button to add the cross sections to the existing set of selected cross sections or click the *Deselect* button to remove them. You can also combine these four options. Namely, you can select the uncalculated cross sections between two chainages.

9.18.1 Batch Cross Section Calculation

The actions referring to the calculation of a cross section can be carried out collectively for all of the project's cross sections by running the *Calculation* command from the *Batch Actions* window. The purpose of this command is to carry out batch calculations in the cross sections of the current road of a project. The calculations may also include settings for the parameters of the branches, the topsoil, the pavings, the sideslopes or/and all of the above. The window where you can make the settings and the batch calculation is as follows,



and appears as soon as you run this command.

The dialogue box is comprised of three groups: In the *Selected cross sections* and *Cross Section selection* groups you can choose the cross sections to which the calculations will be applied. In the right part of the window you will find the *Calculate selected cross sections* group, where you can specify the type of the calculations that will be applied, as well as the corresponding parameters. In the same group, you can specify the type of the batch calculation tasks for the selected cross sections. To define the type of the calculations, proceed in the same way as you would for a single cross section. The calculations can be performed with *Typical Cross section* or with *Points*. The as is option means that the calculation will be repeated with the same *Typical,* which had already been used in another calculation and is different for every cross section, or without *Typical*, for any cross section that had been calculated *with Points*. This is very helpful when recalculation is required after editing the terrain model, something that could affect the natural ground of many cross sections, or after changing the diagrams (profile, width, superelevation), something that affects the cross sections' calculation. It is also required for the batch recalculation of cross sections after editing one or more typical cross sections that had been used in a previous calculation. The as is option in the *Topsoil* and *Branches* tabs has the same meaning, namely the way in which the topsoil or the branches are calculated remains unchanged.

In all cases, click the *Calculation* button for the program to start the batch calculation. At the end of the calculation, the program informs you on the results by displaying the relevant message in the message window at the bottom of the program screen. Any problems that may have occurred will appear in detail in the message window. Click the *Exit* button or press the *Esc* key to close the dialogue box and return to the program's main screen.

Correction of Batch Calculation Errors

After the batch calculation, the message window shows three message categories:

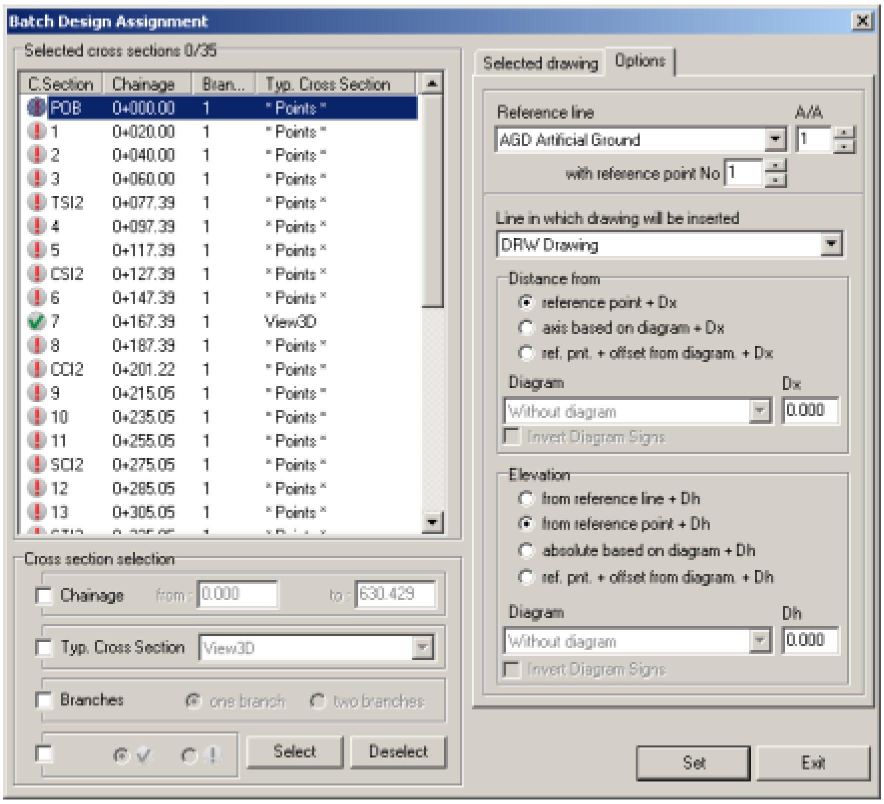
- Information, referring to the start and the end of the calculation process.
- Warnings, referring to problems that occurred during the calculation of specific cross sections and that the program solved by itself. In this case, you must check whether the program has applied the correct solution to the particular cross sections.
- Errors, referring to problems that occurred during the calculation of specific cross sections and couldn't be solved. In this case, you must go to the cross sections involved, correct the problems listed in the message and request recalculation of the cross section.

Every warning or error shows the name of the cross section in which the problem occurred. To go to the cross section, select the corresponding message from the message window in order to highlight it and double click on it. Or, from the menu that is displayed if you right click in the drawing area when the message is selected, run the *Cross section selection* command.

To delete a message, select it, click the right mouse button, and from the displayed menu run the *Delete message* command. To delete all messages of a category, select the *Delete* submenu of the aforementioned menu and then select the category of messages that you wish to delete. To delete all messages from the aforementioned submenu, run the *All* command. To save the messages in a text file, from the *Save* submenu select the category of messages you wish to save. The *Save* dialogue box opens and you must select the folder where the file will be saved, as well as the file name.

9.18.2 Batch Drawing Assignment

The actions referring to drawings can be carried out collectively for all of a project's cross sections by running the *Drawings Assignment* command of the *Batch Actions* menu. This command opens the following dialog box:



Here you can insert – create new lines into certain cross sections of a project, by using a template line, which must exist as a drawing in the project's internal drawing library.

In the *Selected cross sections* and *Cross Section selection* groups, which can be found in the left part of the window, you can choose the cross sections to which the calculations will be applied. It the right part of the window there are two tabs that enable you to assign drawings and insert them into the selected cross sections. From the *Selected drawing* dropdown list, select the library drawing to be inserted.

From the *Options* tab, select the line to be created and the way in which the drawing will be placed both horizontally and with respect to the elevation. From the *Line in which drawing will be inserted* dropdown list, select the typical task in which the selected drawing – template will be inserted in the cross sections, e.g. drawing, N. Jersey, etc. From the *Reference line* dropdown list, select the line based on which the new line will be inserted. From the *Reference line S/N* field, specify the serial number of the line that will become the reference line. The S/N matters only if the selected reference line is a line that can be found in a cross section more than once. Next, you must select a reference line point, based on which the drawings will be applied to every cross section. Select this point from the *with reference point No* field. The S/N of a point is shown in the status bar when the point has been selected within the drawing area. If you wish to insert the drawing to the last point of the line, you can enter a value higher than the available points, e.g. 2000, and the drawing will be inserted to the last point of the reference line. You can also enter negative values in this field. If you wish to insert the drawing to the last but one point of the line, enter the -1 value, and so on.

In the *Distance from* group, specify the horizontal position of the drawing in the cross section. You can insert a drawing in three ways:

- The first way is the *reference point* + *Dx*. If you select this way, the drawing will be inserted in such a way, so that its dependence point has the same X coordinate with the reference line point that the user has entered in the *with reference point No*. The horizontal distance that you enter in the *Dx* text field is added to this coordinate.
- The second way is *axis based on diagram + Dx,* namely the drawing dependence point will be inserted in such a way, so that it has a fixed distance from the cross section axis, depending on the cross section's chainage. The distance is calculated for every cross section from the diagram that the user selects from the *Diagram* dropdown list (e.g. roadway width diagram). When diagrams referring to the left part of the cross section are to be used, you must tick the *Invert Diagram Signs* check box, so that their values correspond to the left part of the cross section. The horizontal distance that you enter in the *Dx* text field is added to this diagram. If you want to use only the fixed distance of the *Dx* field with respect to every cross section's axis, select the Without diagram option.
- The third and last way to insert a drawing is to combine the above two ways. With this method, the X coordinate of the drawing's dependence point is the sum of the distance entered in the *Dx* field, the X coordinate of the reference point and the value of the diagram at every cross section's chainage.

In the *Elevation* group, enter the position that the drawing will get with respect to the elevation. There are four ways to insert a drawing with respect to the elevation:

- The first way is the *from reference line + Dh*. If you select this way, the dependence point of the inserted drawing will have the elevation of the reference line at the distance from axis which was specified in the previous group. The vertical distance that you enter in the *Dh* text field is added to this coordinate.
- The second way is the *from reference point + Dh*. If you select this way, the dependence point of the inserted drawing will have the elevation of the point which was specified in the *with reference point No* field. The vertical distance that you enter in the *Dh* text field is added to this coordinate.
- The third way is the *absolute based on diagram + Dh.* In this case, the drawing's dependence point will have a variable elevation, depending on the cross section's chainage. This elevation is calculated for every cross section from the diagram that the user selects from the *Diagram* dropdown list (e.g. road profile diagram). If you want to invert the signs of the diagram's values, you must tick the *Invert Diagram Signs* check box. The vertical distance that you entered in the *Dh* text field is added to the value that is calculated based on the diagram. If you want to use only the fixed distance that you have entered in the *Dh* field, select the Without diagram option.
- The fourth and last way to insert a drawing is a combination of the second and third ways. With this method, the elevation of the drawing's dependence point is the sum of the distance entered in the *Dh* field, the elevation of the reference point and the value of the diagram at the cross section's chainage.

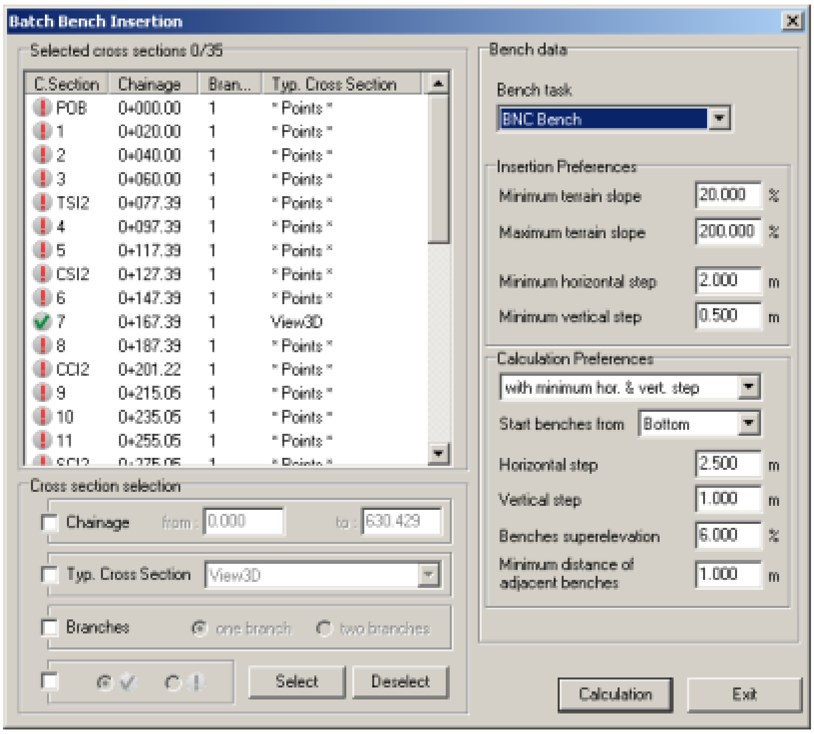
After you have set all parameters, click the *Set* button. The program will inform you that the procedure has been completed. If an error occurs during the process, the relevant message will be displayed in the message bar at the bottom of the screen.

If you have selected a typical task of the WLL, GTT or PVL families, the program will display the relevant dialogue boxes shown in the figures. In these boxes you must set the properties of the lines that will be inserted into the cross sections, e.g. Branches, Side, etc., and then click the *OK* button to continue and finish the procedure.

When you have finished assigning drawings, click the *Exit* button to close the dialogue box and return to the main screen.

9.18.3 Batch Bench Assignment

The actions referring to benches can be carried out collectively for all of a project's cross sections by running the *Bench Insertion* command of the *Batch Actions* menu. The purpose of this command is to insert collectively bench lines of a predefined form into the cross sections of the current road of a project. This command results in the following dialogue box,



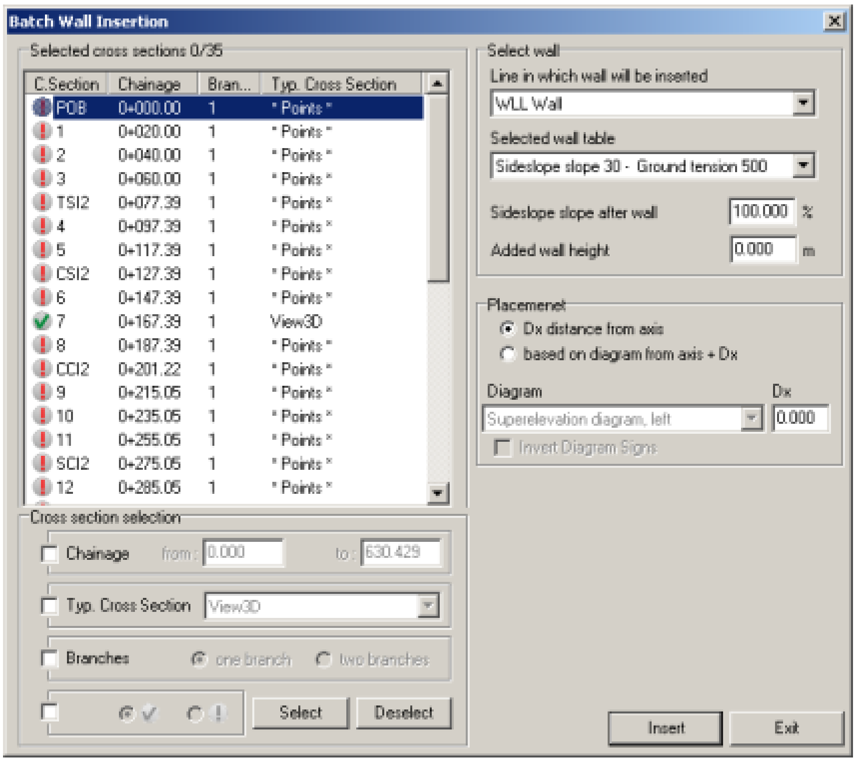
which is comprised of three groups. The *Bench data* group contains the bench insertion and calculation parameters, which are similar to those that you can set by running the *Road preferences* command. From the dropdown list at the top of the group, select the BNC typical task, to which the lines that will be created will belong.

After you have set all parameters, click the *Calculation* button. The program will inform you that the procedure has been completed. If an error occurs during the process, the relevant message will be displayed in the message bar at the bottom of the screen.

When you have finished inserting benches, click the *Exit* button to close the dialogue box and return to the main screen.

9.18.4 Batch Wall Assignment

The actions referring to walls can be carried out collectively for all of a project's cross sections by running the *Wall Insertion* command of the *Batch Actions* menu. The purpose of this command is to insert collectively wall lines of a predefined form into the cross sections of the current road of a project. This command results in the following dialogue box,



which is comprised of four groups. The *Select wall* group contains the wall insertion parameters, which are similar to those that you can set when inserting the wall into a cross section.

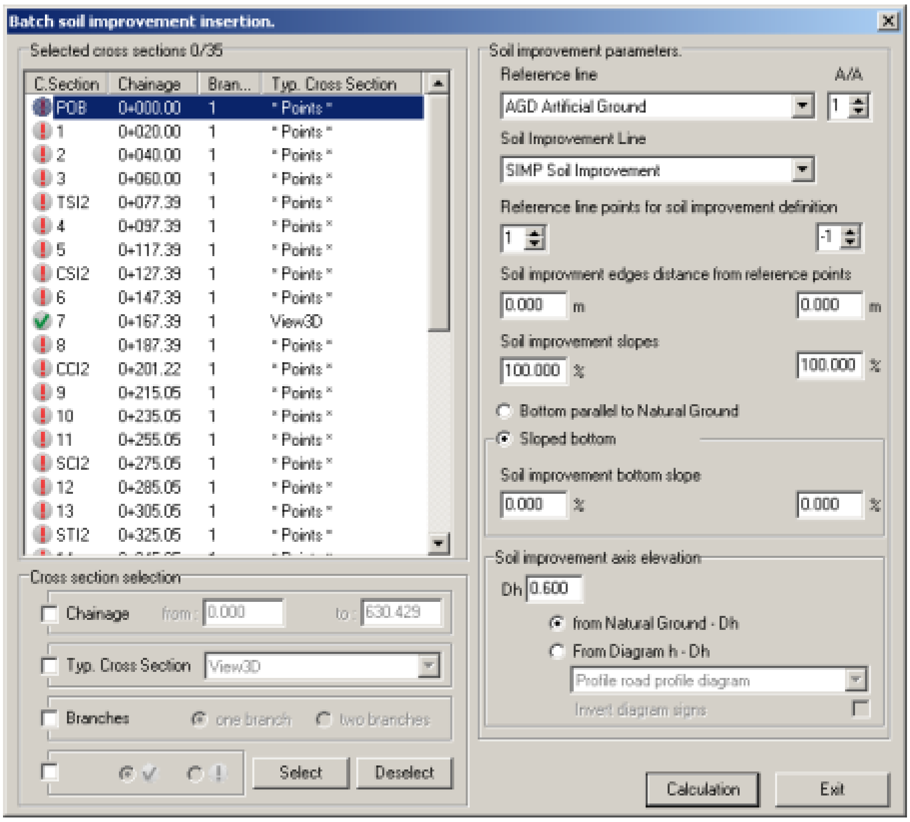
The *Placement* group refers to the selection of the wall's place, since it cannot be selected with the mouse during the batch insertion. If you want the wall to be placed at a fixed distance from the axis of the cross section, tick the *Dx distance from axis* check box and type the distance in the *Dx* field. It is also possible to add a second distance to the fixed one; this second distance will derive from a diagram, depending on the cross section's chainage. To do this, tick the *based on diagram from axis + Dx* check box and from the *Diagram* dropdown list select one of the diagrams included in the file. When diagrams referring to the left part of the cross section are to be used, you must tick the *Invert Diagram Signs* check box, so that their values can correspond to the left part of the cross section. For example, if you wish to insert a wall into the left side of the pavement and at a distance of two meters from the end of the pavement, tick the *based on diagram from axis + Dx* check box, select the left pavement width diagram from the *Diagram* dropdown list, type 2 in the *Dx* field and tick the *Invert Diagram Signs* check box.

After you have set all parameters, click the *Insert* button. The program will inform you that the procedure has been completed. If an error occurs during the process, the relevant message will be displayed in the message bar at the bottom of the screen.

When you have finished inserting walls, click the *Exit* button to close the dialogue box and return to the main screen.

9.18.5 Soil Improvement Insertion

The actions referring to soil improvement can be carried out collectively for all of a project's cross sections by running the *Soil Improvement Insertion* command of the *Batch Actions* menu. This command opens the following dialog box:



This dialogue box is comprised of four groups. In the *Selected cross sections* and *Cross Section* selection groups, which can be found in the left part of the window, you can choose the cross sections to which the calculations will be applied. In the right part of the window there are two more groups, enabling you to set the *Soil improvement parameters* and the *Soil improvement axis elevation*.

More specifically, in the Soil improvement parameters group you can set the following:

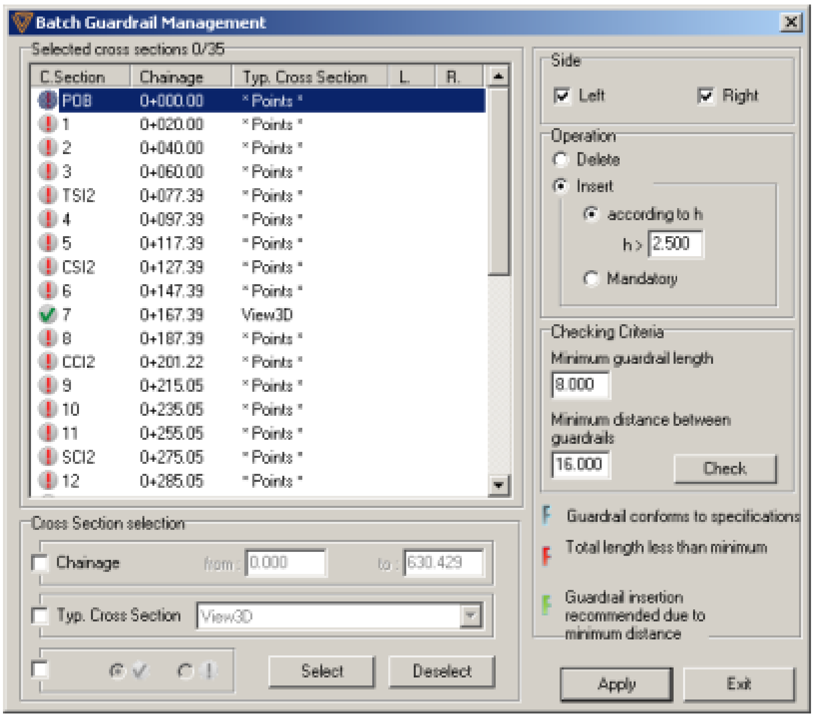
- *Reference line:* Select the line that will determine the X coordinates of the terrain points from which the sideslopes of the new *Soil Improvement* will start. This line is usually the *Artificial Ground*.
- *Reference line S/N:* Enter the serial number of the line that will become the reference line. The S/N matters only if the selected reference line is a line which can be found in a cross section more than once.
- *Soil Improvement Line:* Select the typical task in which the new line will be inserted, e.g. drawing, soil improvement, etc.
- *Reference line points for soil improvement definition:* In these fields you can set the reference line points that will determine the edges of the *Soil Improvement sideslopes* (usually 1 for left and -1 for right).
- *Soil improvement edges distance from reference points:* In these fields you can set an additional distance from the selected reference points.
- *Soil improvement slopes:* Set the slope that the sideslopes will have on the left and right part of the soil improvement.
- *Soil improvement bottom superelevations:* Define any desired superelevations that the bottom will have on the left and right part of the soil improvement.

In the *Soil improvement axis elevation* you can set the following:

- *Dh:* In this field, enter the elevation of the Soil Improvement with respect to the *Natural Ground* or with respect to a *Diagram*.
- *Invert diagram signs:* Tick this option when the *Dh* is calculated based on a *Diagram* and the signs of the diagram values must be inverted.

9.18.6 Guardrail management

The actions referring to guardrails can be carried out collectively for all of a project's cross sections by running the *Guardrails Management* command of the *Batch Actions* menu. The purpose of this command is to insert guardrails collectively into the cross sections of the current road of a project, according to the predefined form that you have chosen. This command results in the following dialogue box,



which is comprised of three groups. In the Side group, select the cross section side to which the guardrail will be inserted. To place guardrails left and right, tick both boxes. In the *Operation* group, select the task you wish to perform.

- *Delete:* If you select this option and click the *Apply* button, all guardrails are deleted from the selected cross sections.
- *Insert:* Select this field to activate the following options:
  - *according to h:* In the *h>* field, enter the minimum *Dh* according to which the guardrails will be inserted (The Dh is calculated between the elevation of the guardrail reference point and the sideslope foot.
  - *Mandatory:* The program inserts guardrails in all of the selected cross sections.

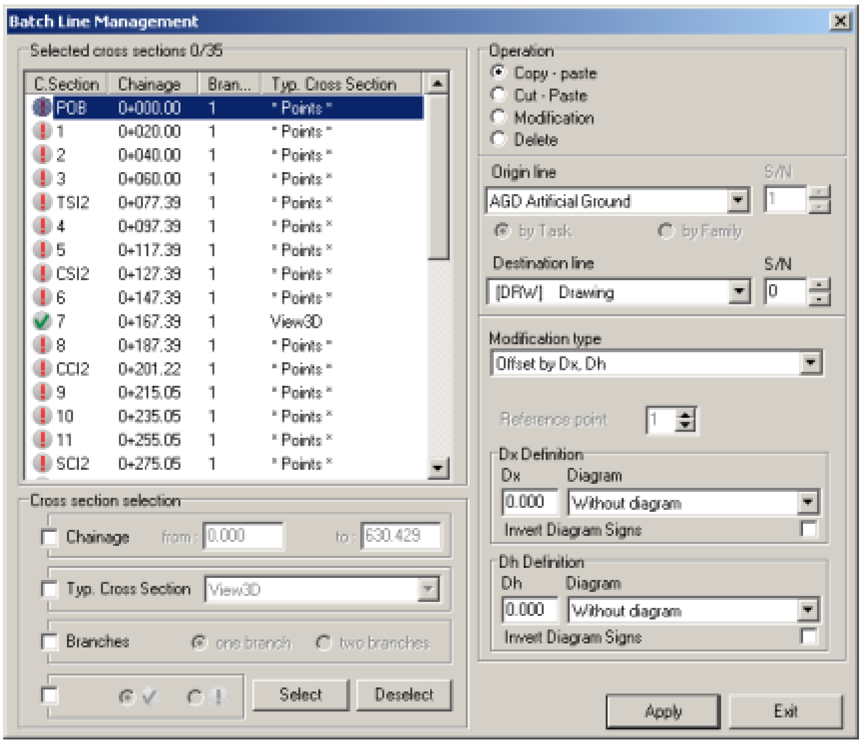
The F symbol appears for all cross sections in which guardrails have been placed. After the placement of guardrails, the program performs an automatic check based on the *Checking Criteria* that you have specified. If you change these criteria, click again the *Check* button to apply the new criteria.

The *Checking Criteria* group has the following fields:

- *Minimum guardrail length:* In this field, set the minimum acceptable/ applicable length of a cross section or a set of cross sections, in which a guardrail can be placed. If the applicable length of a set of cross sections with guardrail is shorter than this value, the F symbol appears for these cross sections.
- *Minimum distance between guardrails:* In this field, set the minimum acceptable distance between two successive sets of cross sections with guardrail. If the distance between two successive sets of cross sections with guardrail is shorter than this value, the F symbol appears for the intermediary cross sections.

9.18.7 Batch Line Management

Anadelta Tessera offers operations for batch line management. These operations can be carried out from the dialogue box of the *Line management* command of the *Batch Actions* menu.



From this box, you can perform four operations (copy, cut, modification, delete), as described below.

The dialogue box is comprised of three groups. The *Operation* group is comprised of three sections. From the top section you can select the type of the operation to be performed, from the middle section you can select the lines to which the selected operation shall refer to, and from the bottom section you can set the line modification type and its parameters.

Batch Line Copy

You can copy the lines of all cross sections belonging to a typical task.

How to copy simultaneously all lines of a typical task

1. Open the dialogue box of the *Line management* command and select the cross sections you wish to edit.
2. Tick the *Copy – Paste* check box in the middle section of the *Operation* group.
3. From the *Origin line* dropdown list of the middle section, select the typical task to which the copied lines belong. If certain cross sections contain more than one lines of the same typical task (e.g. two base courses), then in the S/N field enter the serial number of the line you wish to copy. If this serial number is zero, then the *by Task* and *by Family* check boxes are enabled. This means that the program will copy all lines of the selected cross sections belonging to the typical task or family of the selected typical task, respectively, depending on the ticked check box.
4. From the *Destination line* dropdown list of the middle section, select the typical task of the lines to which the data of the lines that you selected in the *Origin line* will be copied. If certain cross sections already contain lines of the particular typical task,

then in the *S/N* field enter the zero value. With this option, the program creates new lines without affecting the existing lines of the selected typical task. If you enter a value higher than zero, then the new lines that have this number will be inserted and placed in the cross sections and any existing lines with the same number will be deleted.

5. If you wish to modify the origin line, select the type of the modification from the *Modification type* dropdown list of the bottom section. In the remaining fields, make the necessary settings according to the selected modification type.
6. Click the *Apply* button to carry out the operation for the selected cross sections. The program will prompt you to confirm that you wish to copy the lines, by showing you the relevant confirmation message. To complete the procedure, click *Yes*. If the serial number of the typical task that you selected from the *Destination line* dropdown list is zero and the typical task belongs to the WLL, GTT or PVL family, the program will display the relevant dialogue boxes, enabling you to set the properties of the lines that will be created, such as Branches, Side, etc. After you have set the properties for the particular lines, click the *OK* button to proceed. To close the dialogue box, click the *Exit* button.

Batch Line Cut

You can transfer the lines of a typical task to another typical task, which already exists or will be created in any cross section, as necessary. The procedure is similar to the batch line copy, only in this case the origin lines are deleted permanently.

Line Modification

You can modify the lines of all cross sections belonging to a typical task.

How to modify simultaneously all lines of a typical task

1. Open the dialogue box of the *Line management* command and select the cross sections you wish to modify.
2. Tick the *Modification* check box in the top section of the *Operation* group.
3. From the *Origin line* dropdown list of the middle section, select the typical task of the lines to be modified. If certain cross sections contain more than one lines of the same typical task (e.g. two base courses), then in the *S/N* field enter the serial number of the line you wish to modify. If the selected serial number is zero, then the *by Task* and *by Family* check boxes are enabled. This means that the program will modify all lines of the selected cross sections belonging to the typical task or family of the selected typical task, respectively, depending on the ticked check box.
4. If you wish to modify the origin line, select the type of the modification from the *Modification type* dropdown list of the bottom section. In the remaining fields, make the necessary settings according to the selected modification type.
5. Click the *Apply* button to carry out the operation for the selected cross sections. To close the dialogue box, click the *Exit* button.

Deleting Lines

You can delete the lines of all cross sections belonging to a typical task.

How to delete simultaneously all lines of a typical task

1. Open the dialogue box of the *Line management* command and select the cross sections you wish to edit.
2. Tick the *Delete* check box in the top section of the *Operation* group.
3. From the *Origin line* dropdown list of the middle section, select the typical task of the lines to be deleted. If certain cross sections contain more than one lines of the same typical task (e.g. two base courses), then in the *S/N* field enter the serial

number of the line you wish to delete. If the selected serial number is zero, then the *by Task* and *by Family* check boxes are enabled. This means that the program will delete all lines of the selected cross sections belonging to the typical task or family of the selected typical task, respectively, depending on the ticked check box.
4. Click the *Apply* button to carry out the operation. The program will prompt you to confirm that you wish to delete the lines, by showing you the relevant confirmation message. To complete the procedure, click *Yes*. To close the dialogue box, click the *Exit* button.

Modification Types

From the *Modification type* dropdown list in the bottom section of the *Operation* group of the *Batch Line Management* dialogue box, you can select the type of the modification that will be applied to the selected lines of the selected cross sections. The selectable modification types, as well as their relevant settings, are explained in detail below.

Line offset

The line offset can be performed in four ways:

1. To offset lines both horizontally and vertically, from the *Modification type* dropdown list select *Offset by Dx, Dh*. In the *Define Dx* group, set the horizontal offset. The distance is given according to the cross section's chainage from the diagram that you select from the *Diagram* dropdown list. If you want to invert the signs of the selected diagram's values, you must tick the *Invert Diagram Signs* check box, so that their values correspond to the left part of every cross section. The distance that you enter in the *Dx* text field is added to this diagram. If you want to use only the fixed distance that you have entered in the *Dx* field, select the *Without diagram* option. In the *Define Dh* group, set the vertical offset. The procedure that you must follow to set the vertical offset is the same as the one for setting the horizontal offset.
2. To offset the lines horizontally by defining the new coordinate of a line point, from the *Modification type* dropdown list select *Move reference point to X*. In the reference point *No field* select the reference point of the line. In the *Define X* field, enter the new X coordinate. The coordinate is given according to the cross section's chainage, from the diagram that the user selects from the *Diagram* dropdown list. If you want to invert the signs of the selected diagram's values, you must tick the *Invert Diagram Signs* check box, so that their values correspond to the left part of every cross section. The value entered in the *X* text field is added to this diagram. If you want to use only the fixed value of the *Dx* field, select the *Without diagram* option.
3. To offset the lines vertically by defining the new coordinate of a line point, from the *Modification type* dropdown list select *Move reference point to H.* In the *reference point No* field, select the reference point of the line. In the *Define H* group, enter the new elevation of the reference point, as described above for the horizontal offset.
4. To offset lines in parallel by setting the distance of the parallel offset, from the *Modification type* dropdown list select *Offset*. In the *Define D* group, set the offset distance.

Line extension

To extend lines, you must add a new point to the selected lines, either at the start or at the end of the lines:

1. To add a new point at the start of the lines, by setting the horizontal and vertical distances from the first point of the line, from the *Modification type* dropdown list

select *Extend to start with Dx, Dh*. In the *Define Dx* group, set the horizontal distance of the new point from the first point of the line. In the *Define Dh* group, set the vertical distance. If you type the –1000 value in the *Dh* field, then the elevation of the new point is determined by the program in such a way, so that the new straight part has the same slope with the first part of the line.
2. To add a new point at the end of the lines, by setting the horizontal and vertical distances from the last point of the line, from the *Modification type* dropdown list select *Extend to end with Dx, Dh*. Set the distances as described above for the extension to start.

Line interpolation

You can add a point to a line in two ways:

1. To add a point at a fixed distance from the start of the axes, from the *Modification type* dropdown list select *Interpolation at X*. In the *Define X* group, define the position of the new point by entering the distance from the start of the axes. The elevation of the new point is determined by the program in such a way, so that the slopes of the straight parts of the line remain unchanged.
2. To add a point at a fixed distance from a certain point of the line, from the *Modification type* dropdown list select *Interpolation at Dx from reference point*. In *the Reference point No* field, select the reference point of the line. In the *Define Dx* group, set the distance of the new point from the reference point. The elevation of the new point is determined by the program in such a way, so that the slopes of the straight parts of the line remain unchanged.

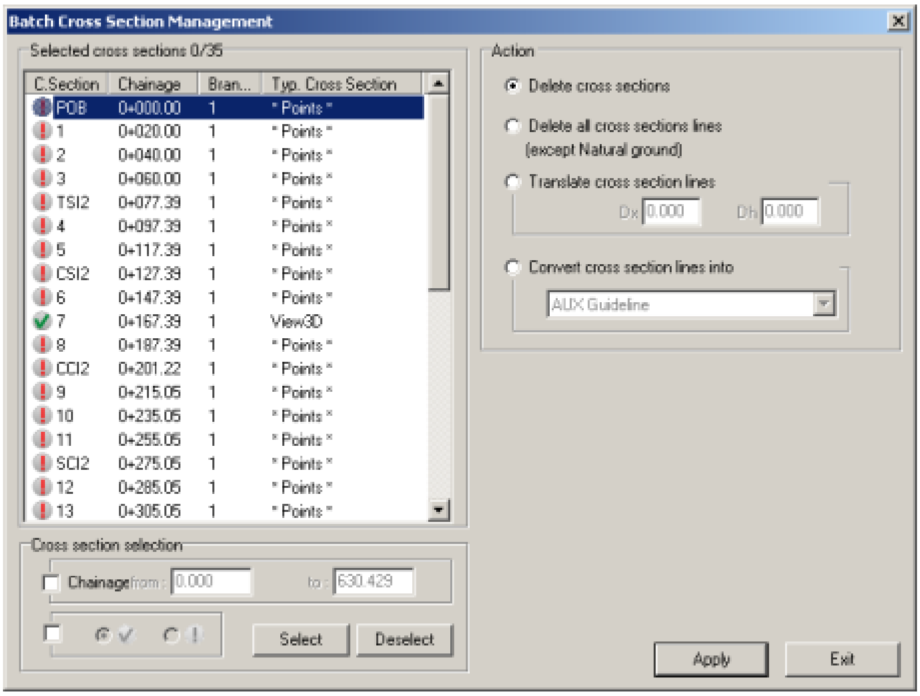
Cropping of lines

There are four ways to crop a line:

1. To remove a line segment before a certain distance from the start of the axes, from the *Modification type* dropdown list select *Crop before X*. In the *Define X* group, enter the distance from the start of the axes. The line segment that will be removed is located on the left of the user-defined distance.
2. To remove a line after a certain distance from the start of the axes, from the *Modification type* dropdown list select *Crop after X*. In the *Define X* group, enter the distance from the start of the axes. The line segment that will be removed is located on the right of the user-defined distance.
3. To remove the line segment before a certain distance from a reference point, from the *Modification type* dropdown list select *Crop at Dx before reference point*. In the *Reference point No* field, select the reference point of the line. In the *Define Dx* group, set the distance from the reference point.
4. To remove the line segment after a certain distance from a reference point, from the *Modification type* dropdown list select *Crop at Dx after reference point*. In the *Reference point No* field, select the reference point of the line. In the *Define Dx* group, set the distance from the reference point.

9.18.8 Batch Cross Sections Management

Anadelta Tessera offers features for the batch management of cross sections. These operations can be carried out from the dialogue box of the *Cross sections management* command of the *Batch Actions* menu.



From this box, you can perform four operations (cross section deletion, line deletion, line offset, modification of typical task), as described below.
The dialogue box is comprised of three groups. The desired operation is selected from the *Action* group.

Batch Cross Section Deletion

You can delete simultaneously as many cross sections as you wish from your project.

How to delete simultaneously multiple cross sections

1. Open the dialogue box of the *Cross sections management* command and select the cross sections you wish to delete.
2. From the *Action* group, tick the *Delete cross sections* check box. To delete the selected cross sections, click the *Apply* button. The program will prompt you to confirm that you wish to delete the cross sections, by showing you the relevant confirmation message. Click *Yes* to proceed or click *No* to cancel the procedure.
3. When you have finished, click the *Exit* button to close the dialogue box and return to the Cross Sections workspace.

Deleting all Lines, apart from the Natural Ground

You can delete all lines of the selected cross sections, apart from the line of the Natural Ground task.

How to delete all lines of the selected cross sections

1. Open the dialogue box of the *Cross section management* command and select the cross sections you wish to edit.

2. In the *Action* group, tick the *Delete all cross sections lines (except Natural ground)* and click the *Apply* button to proceed. The program will prompt you to confirm that you wish to delete the cross sections' lines, by showing you the relevant confirmation message. Click *Yes* to proceed or click *No* to cancel the procedure.
3. When the procedure has been completed, click the *Exit* button to close the dialogue box and return to the Cross Sections workspace.

<u>Moving cross section's lines</u>

You can move all lines of the selected cross sections. The lines are moved at fixed, user-defined distances (horizontally and vertically).

How to move all lines of the selected cross sections

1. Open the dialogue box of the *Cross section management* command and select the cross sections you wish to edit.
2. In the *Action* group, tick the *Translate cross section lines* check box. In the *Dx* and *Dh* fields, type the horizontal and vertical distance, respectively, and then click the *Apply* button to proceed. The program will prompt you to confirm that you wish to move the cross sections' lines, by showing you the relevant confirmation message. Click *Yes* to proceed or click *No* to cancel the procedure.
3. When the procedure has been completed, click the *Exit* button to close the dialogue box and return to the Cross Sections workspace.

<u>Changing the Typical Task of all Lines</u>

You can convert all lines of the selected cross sections, apart from the line of the Natural Ground, into lines of the DRW or AUX typical tasks. This feature can be very helpful in case you wish to view two different configurations in the same cross section. Namely, you can first apply a typical cross section, then convert simultaneously all lines into a *Drawing* and finally apply a second typical cross section to the project's cross sections. In this way, you will be able to view both of the applied solutions to every cross section.

How to change the typical task of all lines of the selected cross sections

1. Open the dialogue box of the *Cross sections management* command and select the cross sections you wish to edit.
2. In the *Action* group, tick the *Convert cross section lines into* check box.
3. From the dropdown list, select one of the typical tasks [AUX] *Guideline* or [AUX] *Typical Slopes* or [DRW] *Drawing* and click the *Apply* button to carry out the procedure. The program will prompt you to confirm that you wish to convert the cross sections' lines, by showing you the relevant confirmation message. Click *Yes* to proceed or click No to cancel the procedure.
4. When the procedure has been completed, click the *Exit* button to close the dialogue box and return to the Cross Sections workspace.

9.18.9 Batch Cross Section Import from another Project

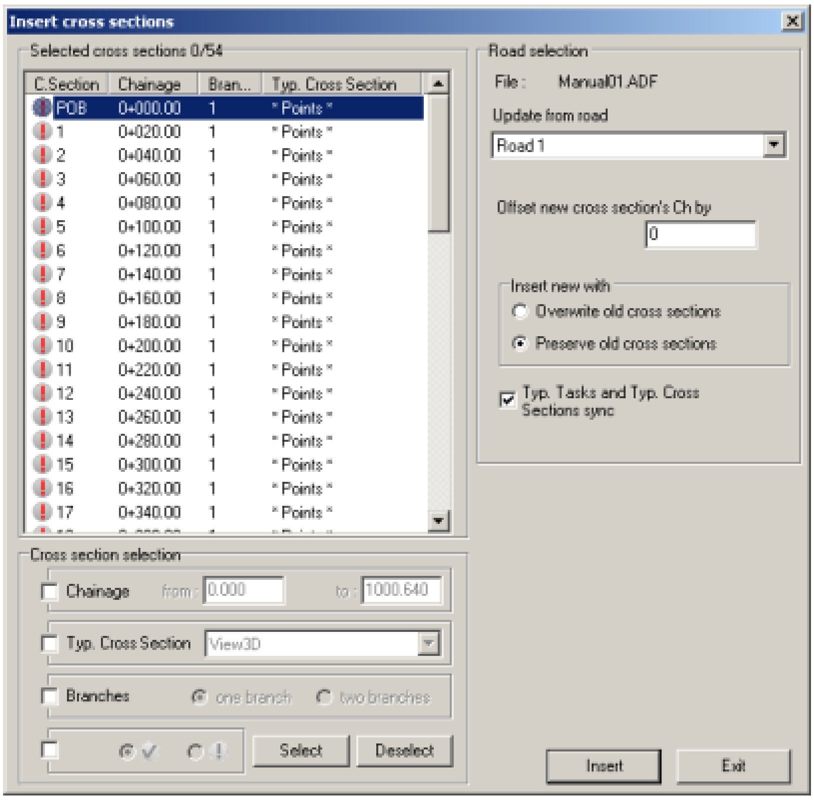
Anadelta Tessera enables you to import into your current project one or more cross sections from another project. This action imports all of the desired cross sections simultaneously.

How to import cross sections from another project

1. From the *File* menu, select the *Import* submenu and run the *Cross Sections* command.
2. From the resulting dialogue box, select the file from which the cross sections will be

copied. Next, click *Open*.

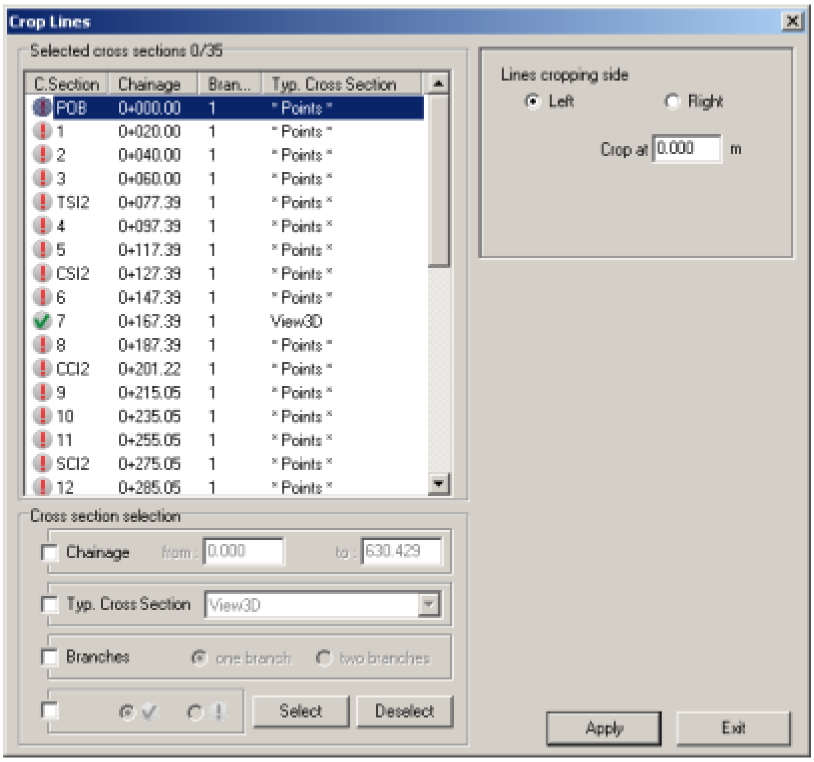
3. The following dialogue box appears on the screen:



4. The displayed cross sections correspond to the file you selected previously; select which of these cross sections will be imported into your project. Make your selections in the *Selected cross sections* and *Cross Section* Selection groups.
5. From the dropdown list of the *Road selection* group, select the road whose cross sections you wish to add into your project.
6. In the *Insert new with* group, define how the cross sections will be added. If you tick the *Overwrite old cross sections* check box, the program will insert all of the selected cross sections and if a cross section's chainage coincides with the chainage of an existing cross section, the latter will be deleted. If you tick the Preserve old cross sections check box, the program will only insert the selected cross sections if their chainages are different from the chainages of the cross sections that already exist the file.
7. In the *Offset new cross section's Ch by* text field, set the distance by which the new cross sections' chainages will be offset. Please note that the actions of step 6 above will be based on the new chainages.
8. If you tick the *Typ. Tasks* and *Typ. Cross Sections sync* box, the typical cross sections and the typical tasks of the source file will also be copied to your current project.
9. When you have finished, click the *Exit* button to close the dialogue box and return to the Cross Sections workspace.

9.18.10 Batch Cropping of Lines

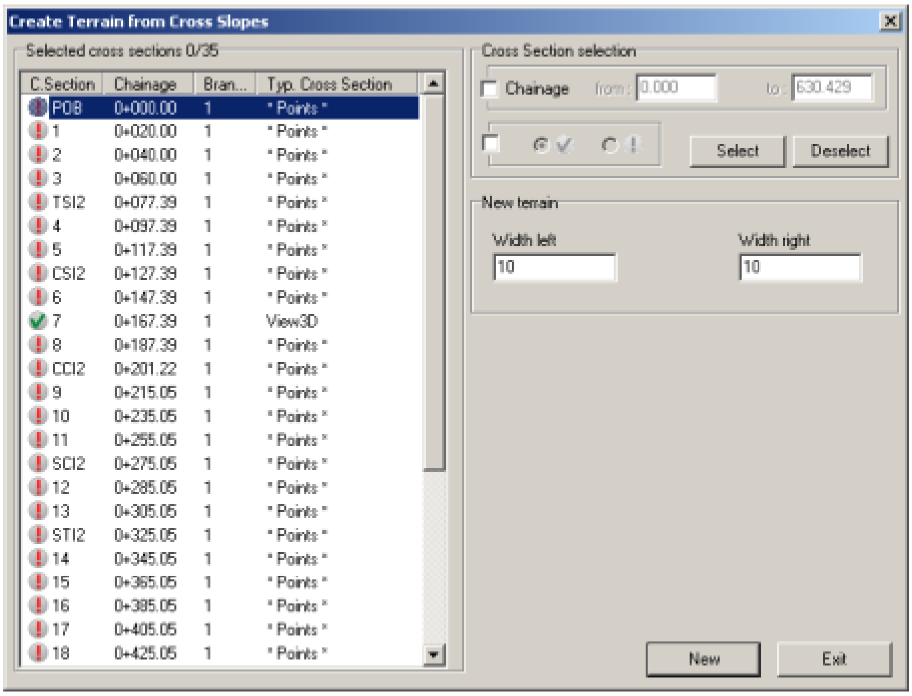
After the cross section calculation, you can batch crop the right or the left part of the cross section at a user-defined distance from the axis. To do this, select the *Crop Lines* commands from the *Batch Actions* menu. The following window will appear on the screen:



Select the cross sections you wish to crop. You can also select the *Lines cropping side*, as well as the cropping distance from the axis by entering the desired value in the *Crop at* field. If the segment to be removed is at the left of the axis, then you must first choose the left side and then enter a negative value in the *Crop at* field. Namely, in this case it is not enough to just select the left side.
The *Crop Lines* command removes the selected segment from the cross sections, closes the cross sections and calculates them *with Points*, leaving the Natural Ground line unaffected.

9.18.11 Natural Ground from Cross Slopes

Anadelta Tessera enables you to create terrain from cross slopes. This feature is required only in cases where an approximate presentation of the natural ground is needed, when the only known data are the profile and the cross sections. To create terrain from cross slopes, run the *Natural Ground from Cross Slopes* command from the *Batch Actions* menu. The following dialogue box will appear on the screen:



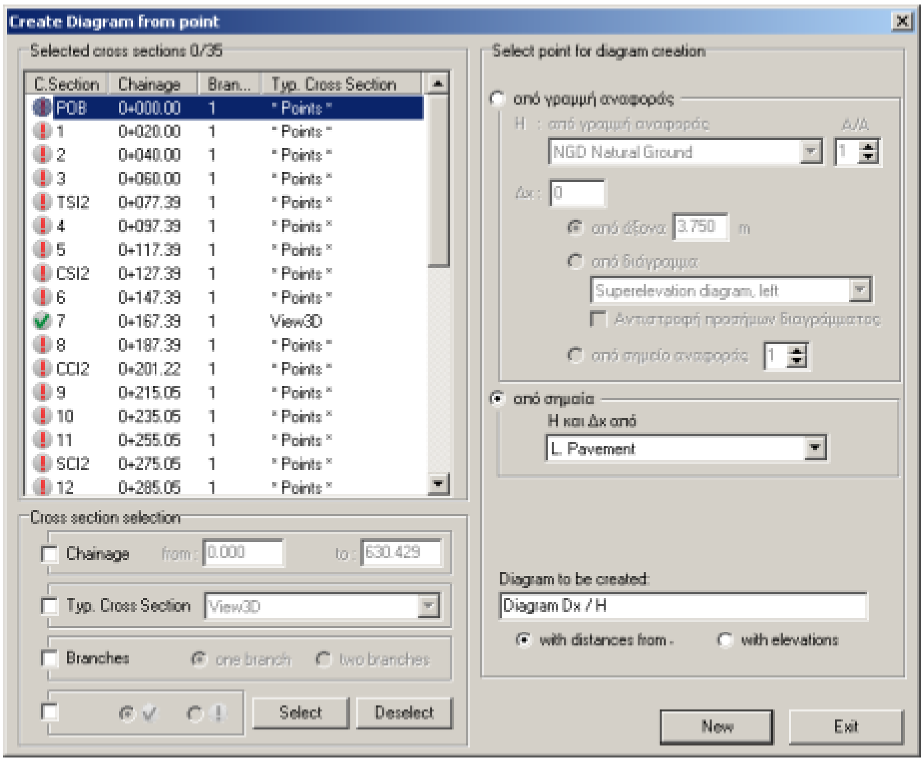
To create a new terrain from cross slopes, you must first make the *Cross Slopes Diagram* (if it does not exist already). To create a diagram, proceed as follows:

- From the *Parameters* menu, select the *Diagrams* command.
- In the resulting dialogue box, click the  button. From the resulting dropdown list, select the diagram you wish to make (in this case, the *Natural ground cross slope diagram*). The values that you will enter for the slopes will be in (%), namely for a 10% slope, simply type 10. Negative slope values mean that the ground descends from left to right.
- Enter the desired values and click the *OK* button to close the window.

In this window, it is also possible to select multiple cross sections by clicking the *Select* button, or you can select cross sections based on their *Chainage*.
In the *Width left* and *Width right* fields of the *New ground* section, type the boundaries of the ground that will be created.
Click the *New* button to erase the old ground and create a new one based on your settings.

9.19 Creating Diagram from Point

The cross sections workspace enables you to create a Dx or H diagram, regardless of whether the cross sections are calculated or there is only the natural ground line. This Dx or H diagram will be saved in the cross sections diagrams (Parameters - Diagrams). To do this, select the *Diagram from Point* command of the *Tools* menu. The following window will appear on the screen,



and you must select the cross sections whose points will be included in the diagram to be created. Select the desired cross sections from the *Selected cross sections* and *Cross section selection* groups.
You can select the desired point in two ways: either by specifying directly the point or by settings the point's distance from the axis. Both ways are explained in detail below:
Directly:

- By selecting the reference line through the *from reference line* field, you can enter directly the number of the reference line's point, based on which the diagram will be created (-1 for the last point, -2 for the last but one point, etc.).
- By selecting the *from flag* option, the point to be exported is the one you have already marked with a flag in the cross section (e.g. Pavement, Roadway, etc.).

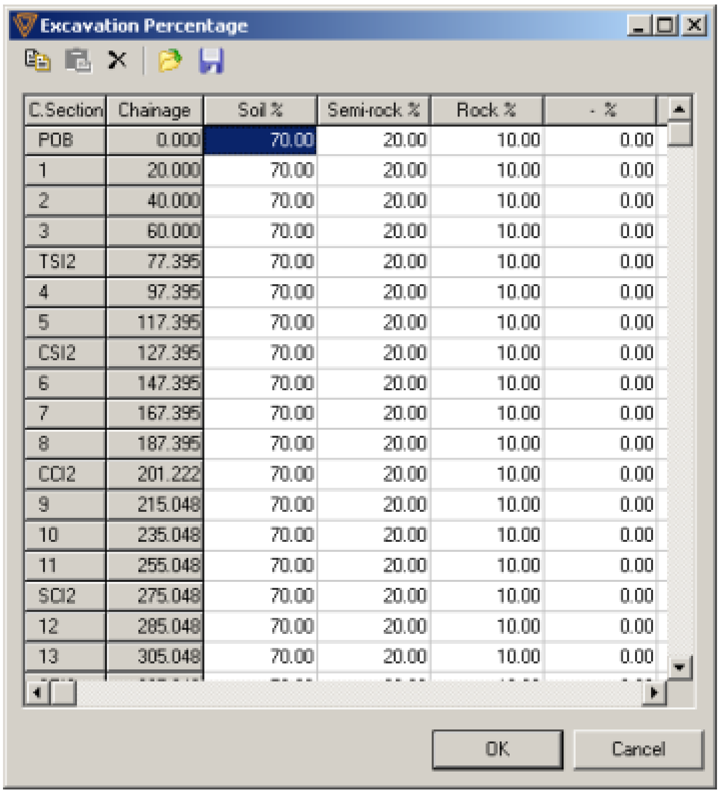
Indirectly:

- By selecting the reference line and then the *from axis* option, you can set directly the point's distance from the axis. When the point is located to the right of the axis, the distance has positive values, while when the point is located to the left of the axis, the distance has negative values.
- By selecting the reference line and then the *from diagram* option, the point's distance from the axis will be determined according to the diagram's values. If you wish to increase the distance determined from the diagram, enter the value of the additional distance into the *Dx* field. If you are referring to the left side and the diagram's values are positive (e.g. pavement width, roadway width), tick the *Invert diagram signs* check box.

In the *Diagram to be created* field, you can enter the desired name of the diagram. The *with distances from* and *with elevations* options refer to the type of the new diagram, namely the diagram will be created based either on the point's relative distance from the axis or on the absolute point elevation (point profile). To save the new diagram, click the *New* button.

9.20 Excavations

The *Excavation Percentage* command of the *Tools* menu determines the excavation percentage for all of the project's cross sections, so that you don't have to define them separately for every cross section. You must, of course, use this option to determine the excavation percentage before calculating and printing the Mass Table or the cross sections drawings. The use of this command results in the following window:



C.Section	Chainage	Soil %	Semi-rock %	Rock %	- %
POB	0.000	70.00	20.00	10.00	0.00
1	20.000	70.00	20.00	10.00	0.00
2	40.000	70.00	20.00	10.00	0.00
3	60.000	70.00	20.00	10.00	0.00
TSI2	77.395	70.00	20.00	10.00	0.00
4	97.395	70.00	20.00	10.00	0.00
5	117.395	70.00	20.00	10.00	0.00
CSI2	127.395	70.00	20.00	10.00	0.00
6	147.395	70.00	20.00	10.00	0.00
7	167.395	70.00	20.00	10.00	0.00
8	187.395	70.00	20.00	10.00	0.00
CCI2	201.222	70.00	20.00	10.00	0.00
9	215.048	70.00	20.00	10.00	0.00
10	235.048	70.00	20.00	10.00	0.00
11	255.048	70.00	20.00	10.00	0.00
SCI2	275.048	70.00	20.00	10.00	0.00
12	285.048	70.00	20.00	10.00	0.00
13	305.048	70.00	20.00	10.00	0.00

The first and second columns show the names of the cross sections of the current road (Cross Section) and their chainages. It is not possible to edit this information here, you can only change it from the window of the *Cross Section Data* command, separately for every cross section.

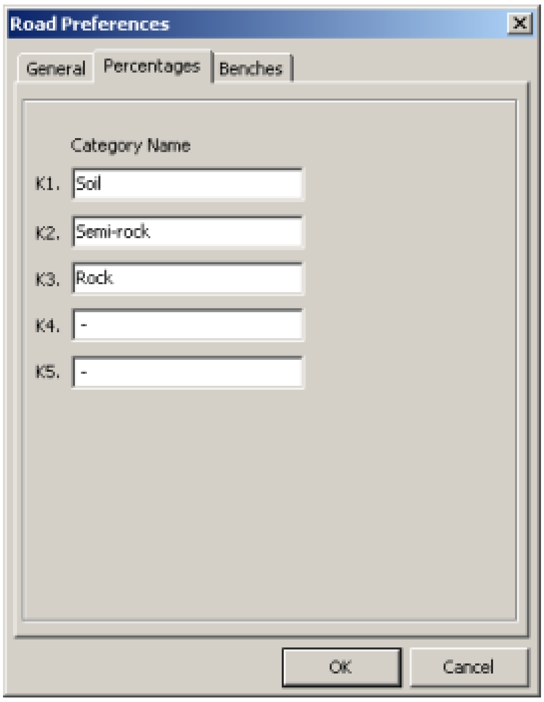
The next columns show the excavation percentages, which can be separated in up to five categories. In these columns you can enter the percentages for every cross section, by typing their values in the fields of the values table. The values are entered in percentages, from 0% to 100%. The sum of a cross section's excavation percentages must be equal to one hundred.

To move between the columns, use the *Tab* key for moving from left to right, or the *Shif+Tab* keys for moving from right to left. You can also use the arrow keys to move up, down, right and left, while if you use the mouse you can go directly to the desired field by clicking the right or left mouse button.

To copy a whole set of consecutive cells, select the desired cells and click the  button at the top right of the window. To select multiple fields, hold the *Shift* down and use the arrow keys. Next, select the area to which the cells will be copied and click the  button. If you wish to zero the values of certain cells, select the cells by following the aforementioned copy procedure and then click the  button.

In all cases, when you have finished, click the *OK* button to apply the new percentages to all cross sections. Finally, you can import and export the excavation percentages to/from a text file.

Names of Categories



To change the names of the excavation percentages, open the *Parameters* menu and run the *Road Preferences* command. In the *Percentages* tab you will find the categories' names (K1, K2, K3, K4, K5), which are originally assigned by default by the program (e.g. Rock, Semi-rock, Soil). More specifically, for every category there is the *Category Name* text field. Type the desired name in this field and then click *OK* to save your settings. The titles of the columns in the window, as well as the titles of the mass table subcolumns are updated automatically.

Importing Percentages from a Text File

You can import into your project all excavation percentages for every cross section, if they are available in electronic form (ASCII text files). You can do this at any given moment, either during the initial stage of the design or at a later stage, when processing your design, and as many times as you wish.

How to import excavation percentages from a text file

1. From the dialogue box of the *Excavation percentage* command, click the  button.
2. A dialogue box appears on the screen. Select the desired file and click the *Open* button. The file to select must have the form shown in the next section.
3. Finally, the program will ask you to specify the way in which it will check whether a cross section in the APS file exists in your project. Namely, the check will be based on the cross section's name or chainage.

Click the *Name* button for the check to be carried out based on the cross section name, or click the *Chainage* button if you prefer the check to be carried out based on the chainage.

Click either button for the update to take place; the new percentages are displayed on the screen.

Specifications of Percentages Text Files (APS)

The cross section data in a file of this type is arranged by columns. Each row of the file corresponds to a cross section. The form of these files is as follows:

POB	0.000	70.00	20.00	10.00	0.00	0.00
1	20.000	70.00	20.00	10.00	0.00	0.00
2	40.000	70.00	20.00	10.00	0.00	0.00
3	60.000	70.00	20.00	10.00	0.00	0.00
TSI2	77.395	70.00	20.00	10.00	0.00	0.00
4	97.395	70.00	20.00	10.00	0.00	0.00
5	117.395	70.00	20.00	10.00	0.00	0.00
CSI2	127.395	70.00	20.00	10.00	0.00	0.00
6	147.395	70.00	20.00	10.00	0.00	0.00
7	167.395	70.00	20.00	10.00	0.00	0.00
8	187.395	70.00	20.00	10.00	0.00	0.00
CCI2	201.222	70.00	20.00	10.00	0.00	0.00
9	215.048	70.00	20.00	10.00	0.00	0.00
10	235.048	70.00	20.00	10.00	0.00	0.00
11	255.048	70.00	20.00	10.00	0.00	0.00
SCI2	275.048	70.00	20.00	10.00	0.00	0.00
12	285.048	70.00	20.00	10.00	0.00	0.00
13	305.048	70.00	20.00	10.00	0.00	0.00
STI2	325.048	70.00	20.00	10.00	0.00	0.00
14	345.048	70.00	20.00	10.00	0.00	0.00
15	365.048	70.00	20.00	10.00	0.00	0.00
16	385.048	70.00	20.00	10.00	0.00	0.00
17	405.048	70.00	20.00	10.00	0.00	0.00
18	425.048	70.00	20.00	10.00	0.00	0.00
19	445.048	70.00	20.00	10.00	0.00	0.00
20	465.048	70.00	20.00	10.00	0.00	0.00
21	485.048	70.00	20.00	10.00	0.00	0.00
22	505.048	70.00	20.00	10.00	0.00	0.00
23	525.048	70.00	20.00	10.00	0.00	0.00
24	545.048	70.00	20.00	10.00	0.00	0.00
25	565.048	70.00	20.00	10.00	0.00	0.00
26	585.048	70.00	20.00	10.00	0.00	0.00

The file is comprised of the following columns: *Cross section*, showing the names of the current road's cross sections, *Chainage*, showing the chainage of every cross section, and five additional columns, *K1, K2, K3, K4* and *K5*, showing the excavation allocation percentages for every cross section.

Exporting Percentages to a Text File

You can export from your project to a text file all excavation percentages for every cross section.

How to export the excavation percentages to a text file

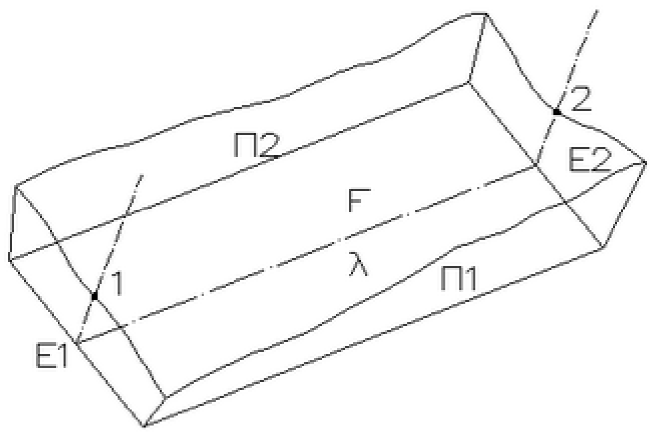
1. From the dialogue box of the *Excavation percentage* command, click the *Save* button.
2. In the resulting dialogue box, specify the path and the name of the file and click the *Save* button.

9.21 Mass Table

Before starting describing the creation of a mass table, it is worth mentioning in brief certain general information that will help you better understand this procedure.

The total mass volume is the sum of the individual volumes between two successive cross sections of the road.

These individual volumes are limited, as shown in the figure below:



1. By the E1 and E2 vertical surfaces of two successive cross sections.
2. By the Π1 and Π2 surfaces of the sideslopes on both sides.
3. By the surface of the roadway.
4. By the F surface of the natural ground.

The necessary data for the calculation of the individual volumes are the areas of the E1 and E2 surfaces and the λ distance between them.

Therefore, by applying the approximate formula:

$$\boxed{V = \frac{E_1 + E_2}{2} \times \lambda}$$

we can calculate the volume contained between the 1 and 2 cross sections for a given λ distance between them.

Mass Volume Calculation Methods.

Two methods are commonly used for the approximate calculation of the mass volume: the method of average surfaces and the method of applicable lengths.

Let’s assume that in a complete cutting (or filling) you have the successive cross sections 1,2,3,..., whose respective areas are E1, E2, E3,... The distances between the cross sections are λ1, λ2, λ3, ...

Method of Average Surfaces.

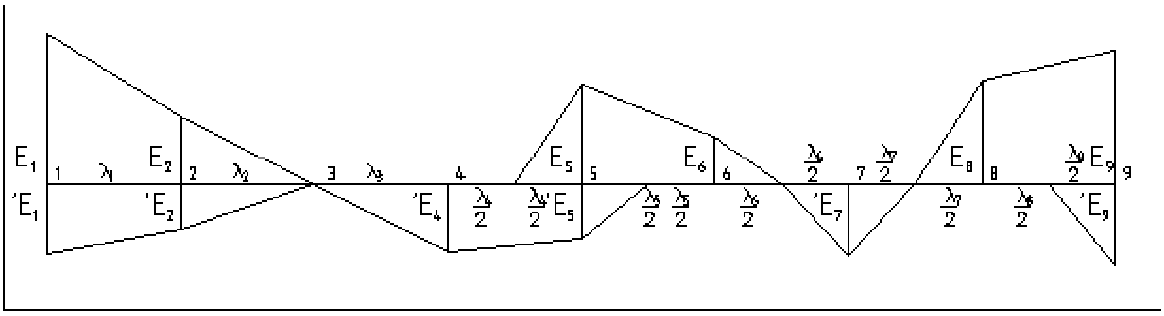
By applying the familiar formula,

$$\boxed{V = \frac{E_1 + E_2}{2} \times \lambda}$$

the result will be:

$$\boxed{V = \frac{E_1 + E_2}{2} \times \lambda_1 + \frac{E_2 + E_3}{2} \times \lambda_2 + \ldots}$$

The $\boxed{\frac{E_1 + E_2}{2}, \frac{E_2 + E_3}{2}, \frac{E_3 + E_4}{2}, \ldots}$ are referred to as average surfaces and therefore the calculation of the V volume with the use of the above formula is referred to as method of average surfaces.



The formula commonly used is:

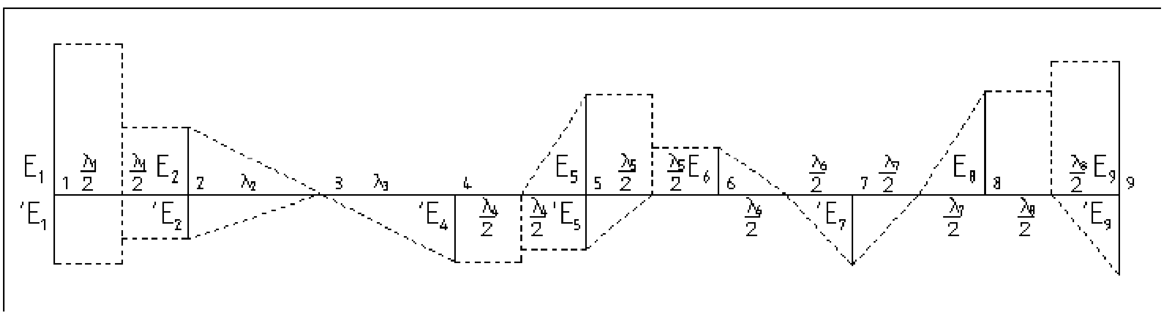
$$\boxed{V = \frac{E_1 + E_2}{2} \times \lambda_1 + \frac{E_2 + E_3}{2} \times \lambda_2 + \ldots + \frac{E_{v-1} + E_v}{2} \times \lambda_{v-1}}$$

Method of Applicable Lengths.

The above formula can also be written as follows:

$$\boxed{V = \frac{\lambda_1}{2} \times E_1 + \frac{\lambda_1 + \lambda_2}{2} \times E_2 + \frac{\lambda_2 + \lambda_3}{2} \times E_3 + \ldots}$$

The $\boxed{\frac{\lambda_1}{2}, \frac{\lambda_1 + \lambda_2}{2}, \frac{\lambda_2 + \lambda_3}{2}, \ldots}$ are referred to as applicable lengths and therefore the calculation of the V volume with the use of the above formula is referred to as method of applicable lengths.



The formula commonly used is:

$$V = \frac{\lambda_1}{2} \times E_1 + \frac{\lambda_1 + \lambda_2}{2} \times E_2 + \ldots + \frac{\lambda_{v-2} + \lambda_{v-1}}{2} \times E_{v-1} + \frac{\lambda_{v-1}}{2} \times E_v$$

Through the typical tasks, the user can define the number of the available columns that may be displayed in the mass table, as well as their titles (the title must be short and it cannot occupy more than two lines per column). The program calculates, if applicable, the length and area of every typical task – line of a cross section (for the method of average areas, the program calculates the average length and area between two successive cross sections). During the mass table calculation, the length or the area are multiplied by the applicable length or by the distance between two successive cross sections (depending on the method that is applied) in order to give the respective surface and volume quantities. These quantities are summed up cumulatively. Each column of the mass table can be divided in a maximum of twenty (20) subcolumns and when the mass table is being created, you can select the subcolumns you wish to be displayed. These subcolumns are the Area with the respectively produced Volume, the Length with the respectively produced Area, the Current Meters, and, for every category of allocation percentages (K1, K2, K3, K4 and K5), the percentage, the allocated area and the allocated volume.

As shown in the figure below (mass table):

1st Line Left
2nd Line Left
3rd Line Left
4th Line Left
5th Line Left

1st Line Right
2nd Line Right
3rd Line Right
4th Line Right
5th Line Right

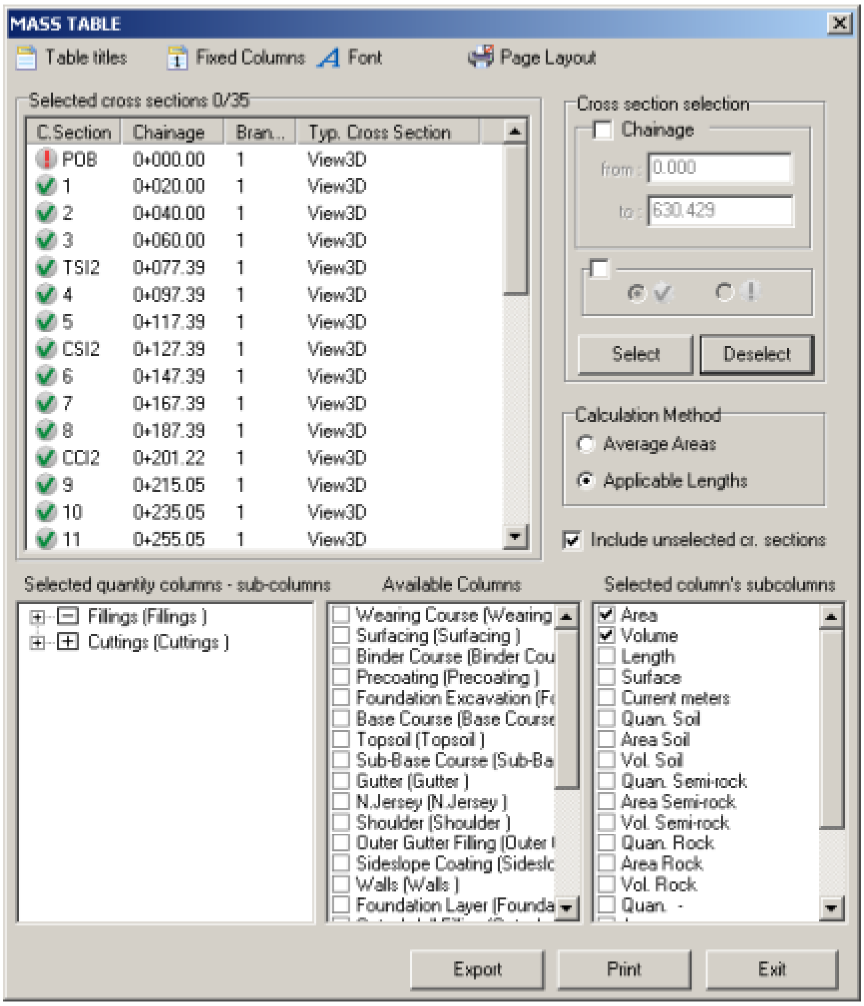
MASS TABLE

General Data				Fillings		Cuttings	
Cross section Name	Chainage Position	Distance Between	Applicable Length	Area	Volume	Area	Volume
POB	0.000	0.00	10.00	0.00	0.00	0.00	0.00
1	20.000	20.00	20.00	40.00	800.00	22.86	457.20
2	40.000	20.00	20.00	2.25	45.00	47.63	952.60
3	60.000	20.00	18.70	1.90	35.52	40.88	764.25
TSI2	77.395	17.39	18.70	7.68	143.58	24.21	452.61
4	97.395	20.00	20.00	18.03	360.60	13.68	273.60
5	117.395	20.00	15.00	9.73	145.95	16.15	242.25
CSI2	127.395	10.00	15.00	8.29	124.35	17.84	267.60
6	147.395	20.00	20.00	9.37	187.40	12.91	258.20
7	167.395	20.00	20.00	7.14	142.80	3.37	67.40
8	187.395	20.00	16.92	18.97	320.88	3.88	65.63
CCI2	201.222	13.83	13.83	22.07	305.23	6.11	84.50
9	215.048	13.83	16.92	16.44	278.08	9.93	167.97
10	235.048	20.00	20.00	19.26	385.20	11.39	227.80
11	255.048	20.00	20.00	14.71	294.20	16.63	332.60
SCI2	275.048	20.00	15.00	9.96	149.40	23.92	358.80
12	285.048	10.00	15.00	6.25	93.75	30.27	454.05
13	305.048	20.00	20.00	0.00	0.00	37.49	749.80
STI2	325.048	20.00	20.00	0.00	0.00	28.74	574.80
14	345.048	20.00	20.00	0.04	0.80	11.01	220.20

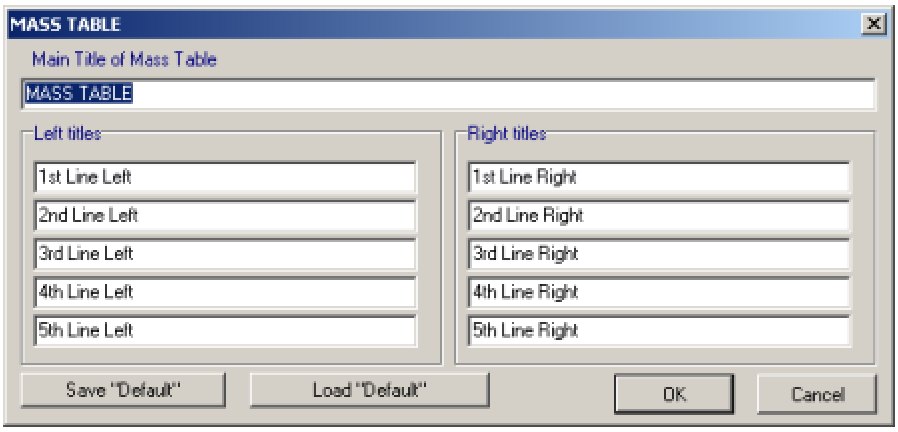
In addition to the user-selected columns, the mass table always contains the columns that show the cross section's name, the cross section's chainage, the distance between and the applicable length (if this is the selected method), based on which the volumes and surfaces will be calculated.

9.22 Mass Table Creation

To open the mass table creation window, go to the *Print* menu and run the *Print mass table* command.

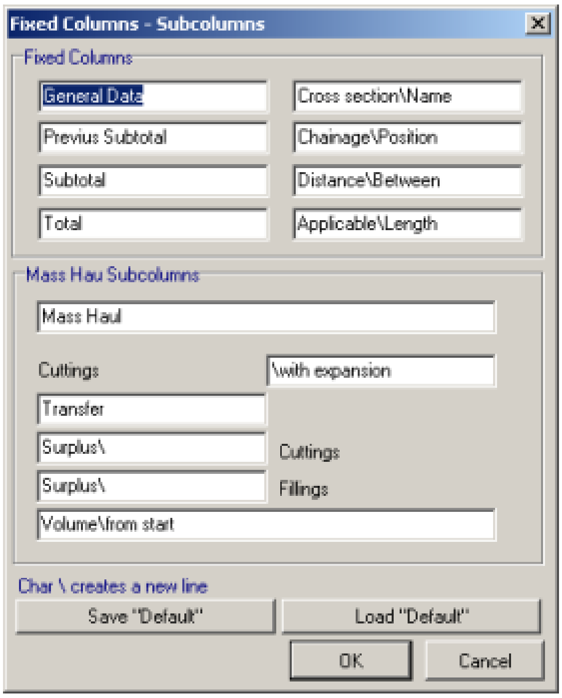


When creating the mass table, you can first select some general information that you wish to view at the top of the mass table's first page. To do this, use the fields of the window shown below:



To open this window, click the *Table Titles* button. In the *Main title of mass table* field, enter the main title of the table. In the next two groups, enter the left and right titles respectively.

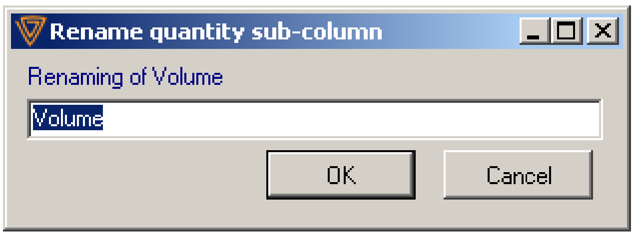
To change the names of the titles for the *Fixed Columns* and the *Mass Haul Subcolumns*, click the *Fixed columns* button to open the window shown below:



In the two groups of the window, enter the titles for the axis data and for the mass haul, respectively.

Next, select the columns, namely the quantities that will be printed in the mass table, as well as the quantity type (area, volume, length, etc.). From the *Available Columns* check list, which contains all columns of the *Mass Table* that were defined in the typical tasks, select one or more columns by clicking in the box next to the column's name. The column's name will be immediately displayed on the *Selected columns - subcolumns* List. From the *Subcolumns* check list, select the desired subcolumns you wish to include in the current column. The columns and subcolumns of the *Mass Table* are displayed in the order they were entered. If you wish to change their order of appearance, use the mouse to select the column or the subcolumn you want to move, then drag-and-drop it to the desired position.

If you want to change a subcolumn's name, select the subcolumn, click the right mouse button, and from the displayed menu run the *Rename* command. In the resulting dialogue box,



type the new name of the subcolumn and click the *OK* button.

If you have selected to calculate the mass table for a set of cross sections by applying the method of applicable lengths, then you can choose whether the applicable length of the first and last cross section of the set will be calculated with the contribution of the previous and next cross section, respectively, by enabling the *Include unselected cr. sections* option.

Next, you can proceed to the calculation of the mass table by clicking the *Print* button. When the calculation is completed, the *Mass Table Preview* window opens and you can print the mass table.

Mass Haul

Especially for the *Mass Haul* column, please note that:

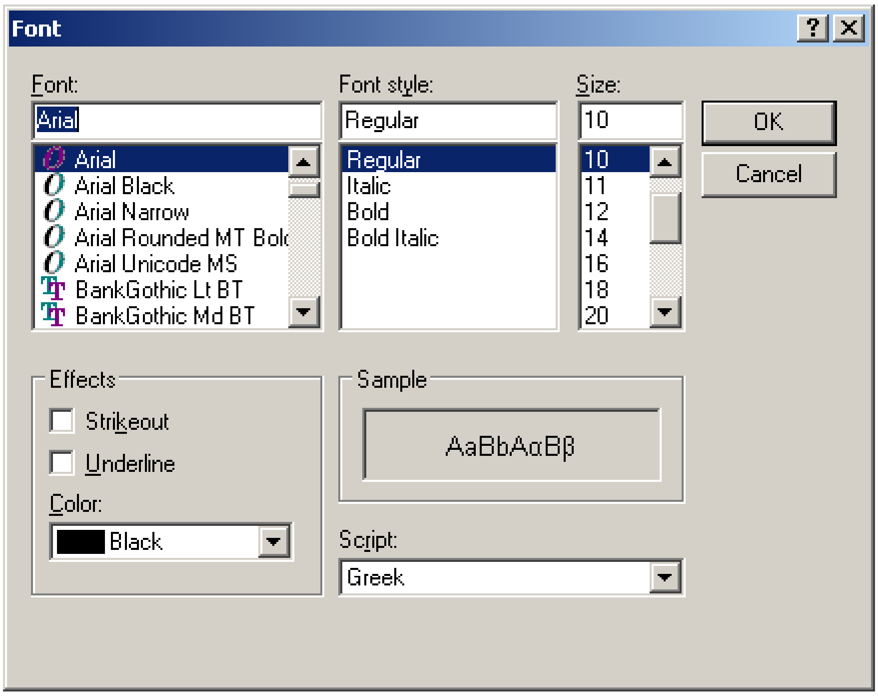
- The program will display the mass haul for two quantities. The first quantity is the one that has been marked with an + in the *Selected columns-subcolumns* list and is deemed positive (usually the cuttings) and the other quantity is the one that has been marked with an – and is deemed negative (usually the fillings). To mark a quantity as positive or negative, select the quantity from the *Selected columns-subcolumns* list, click the right mouse button, and from the displayed menu, select *Positive for mass haul* or *Negative for mass haul*, respectively.
- When selecting columns, the table must also show the columns that contribute to the mass haul (e.g. cuttings and fillings) and then the mass haul column.
- Before the calculation of the table, you must also define the relevant expansion coefficients, in order that they are properly included in the volume calculation.

Text Font

You can select the font that will be used for printing the mass table, as well as the font size and style.

To change the font or the font size – To use bold or italics in selected text or numbers

1. In the *Mass Haul* dialogue box, click the *Font* icon.
2. In the dialogue box of the *Font* command,



select the desired font from the *Font* list, and then select the desired script from the *Script* list.

3. From the *Size* list, select the desired font size.
4. From the *Style* list, select the desired font style. The *Sample* preview field shows how the text will be displayed according to your settings.
5. In all cases, click the *OK* button to close the dialogue box, save your settings and return to the previous dialogue box.

9.23 Mass Table Printing - Export

You can print the *Mass Table* through the environment of *Tessera*. You can also transfer the *Mass Table* data to Microsoft Excel, for further processing.

Mass Table Printing- Export

To print the Mass Table, follow the two simple steps below.

1. Select a printer and set its properties (paper size, orientation, etc.).
2. Request the calculation of the mass table quantities and the layout of the mass table according to the print preferences.

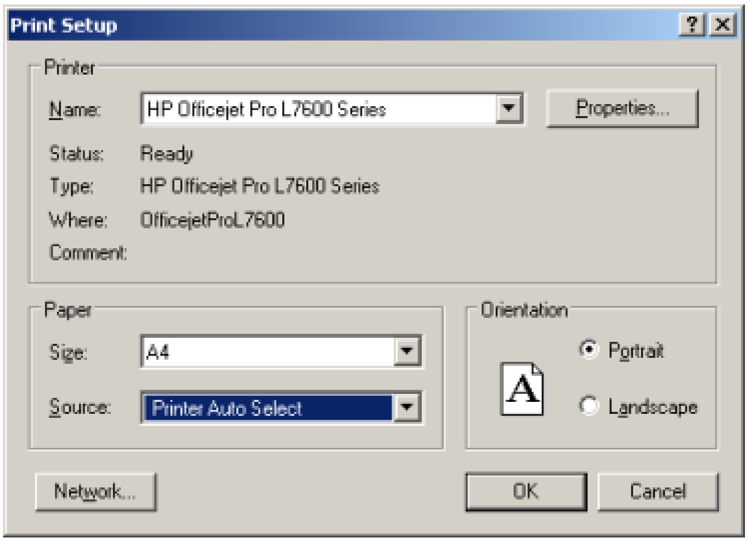
The steps for printing the *Mass Table* are described in detail below.

Printer Selection – Print Properties Setting

First you must set the print preferences, such as the orientation and the paper size.

How to set the print preferences

1. From the *Print* menu, run the *Mass Table Print* command and then click the  icon.
2. In the *Print parameters* dialogue box,



   select one of the available printers from the drop-down list of the *Printer* group. The other fields of the group show information about the selected printer, such as the status, the model, the port or the location of the printer, as well as any additional user-defined information – comment.
3. To change the print options, click the *Properties* button.
4. In the *Size* field (paper size) of the *Paper* group, select from the drop-down list one of the available paper sizes. In the *Source* field, select one of the available paper feed trays.
5. In the *Orientation* group, select the orientation of the text on the paper for printers that support landscape orientation (Horizontal) or portrait orientation (Vertical).

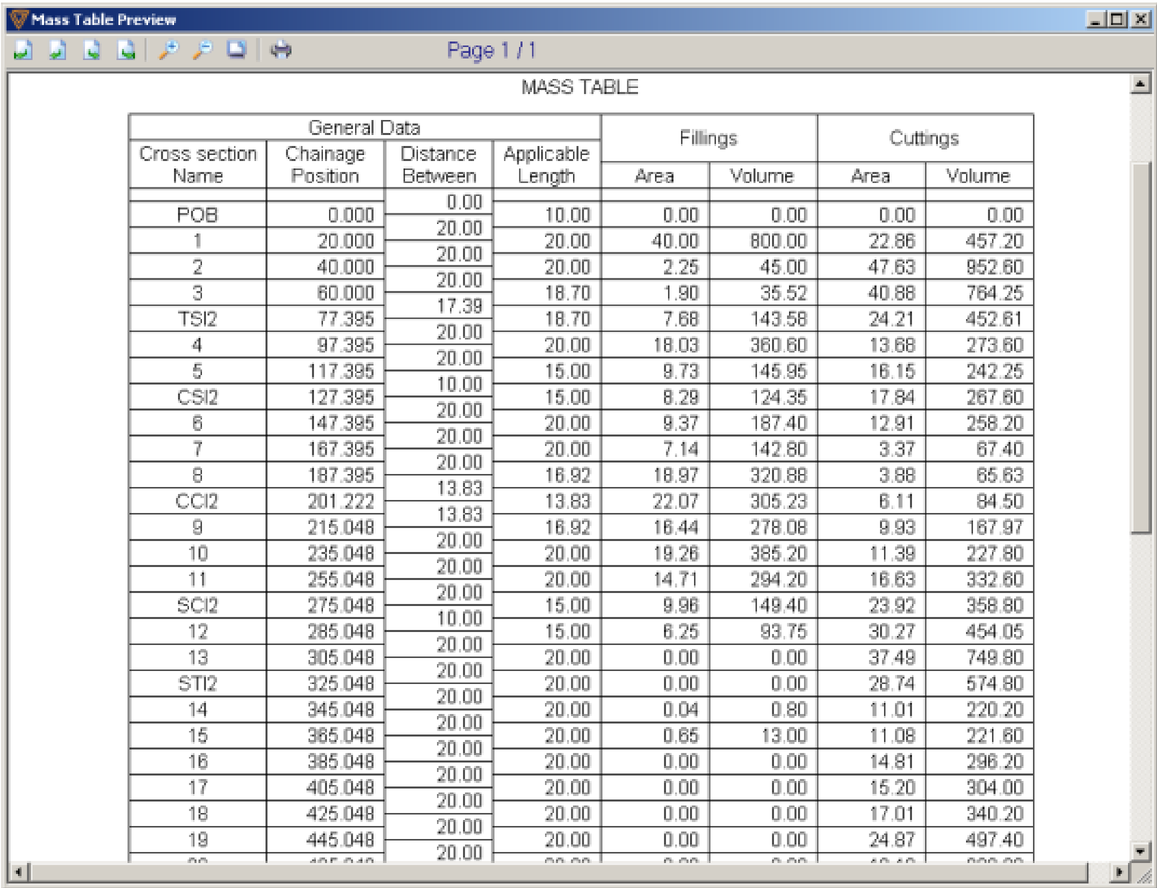
If you change the current printer, the settings that are supported by the new printer will be kept, while all other settings will be replaced by the default settings of the new printer.

<u>Mass Table Preview</u>

When you have finished setting the print preferences according to the aforementioned procedure, you can first display the print preview of the *Mass Table* and then print the table.

To preview a Mass Table

1. Click the *Print* button. The *Mass Table Preview* window appears on the screen.



MASS TABLE

General Data				Fillings		Cuttings	
Cross section Name	Chainage Position	Distance Between	Applicable Length	Area	Volume	Area	Volume
POB	0.000	0.00	10.00	0.00	0.00	0.00	0.00
1	20.000	20.00	20.00	40.00	800.00	22.86	457.20
2	40.000	20.00	20.00	2.25	45.00	47.63	952.60
3	60.000	20.00	18.70	1.90	35.52	40.88	764.25
TSI2	77.395	17.39	18.70	7.68	143.58	24.21	452.61
4	97.395	20.00	20.00	18.03	360.60	13.68	273.60
5	117.395	20.00	15.00	9.73	145.95	16.15	242.25
CSI2	127.395	10.00	15.00	8.29	124.35	17.84	267.60
6	147.395	20.00	20.00	9.37	187.40	12.91	258.20
7	167.395	20.00	20.00	7.14	142.80	3.37	67.40
8	187.395	20.00	16.92	18.97	320.88	3.88	65.63
CCI2	201.222	13.83	13.83	22.07	305.23	6.11	84.50
9	215.048	13.83	16.92	16.44	278.08	9.93	167.97
10	235.048	20.00	20.00	19.26	385.20	11.39	227.80
11	255.048	20.00	20.00	14.71	294.20	16.63	332.60
SCI2	275.048	20.00	15.00	9.96	149.40	23.92	358.80
12	285.048	10.00	15.00	6.25	93.75	30.27	454.05
13	305.048	20.00	20.00	0.00	0.00	37.49	749.80
STI2	325.048	20.00	20.00	0.00	0.00	28.74	574.80
14	345.048	20.00	20.00	0.04	0.80	11.01	220.20
15	365.048	20.00	20.00	0.65	13.00	11.08	221.60
16	385.048	20.00	20.00	0.00	0.00	14.81	296.20
17	405.048	20.00	20.00	0.00	0.00	15.20	304.00
18	425.048	20.00	20.00	0.00	0.00	17.01	340.20
19	445.048	20.00	20.00	0.00	0.00	24.87	497.40

2. Click the maximize button in the title bar to view the window on full screen.
3. This window is comprised of the following items:

- *Preview:* the bottom part of the window where the image of the whole or part of the table's page is displayed.
- *Toolbar:* this is located at the top of the window. It carries the buttons that control the transition from page to page and the modification of the size of the page's viewable area .

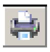
  More specifically, the buttons of the toolbar are the following in order of appearance:
- *First page:* Left-click on the first button to view the first page.
- *Next page:* Left-click on the second button to view the next page.
- *Previous page:* Left-click on the third button to view the previous page.
- *Last Page:* Left-click on the fourth button to view the last page.
- *Zoom In:* Left-click on the fifth button to zoom in on the page. The Zoom feature does not affect the print size.
- *Zoom Out:* Left-click on the sixth button to zoom out of the page. The Zoom feature does not affect the print size.

- *Full Frame:* Left-click on the seventh button to return to the original view of the mass table.
- *Print:* Left-click on the eighth button to set the options before printing and to print the Mass Table.
- *Page Information:* This section displays the number of the current page and the number of the total pages of the mass table.

Mass Table Print

You can print the whole *Mass Table* or part of it in one or more copies.

How to print the Mass Table

1. In the preview window, click the  button.
2. In the dialogue box of the *Print* command, make the desired settings.
3. Click the *OK* button to start printing the Table.

If desired, you can print only part of the *Mass Table* in more than one copy.

Exporting the Mass Table to Microsoft Excel

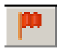
You can transfer the *Mass Table* data to Microsoft Excel for further processing.

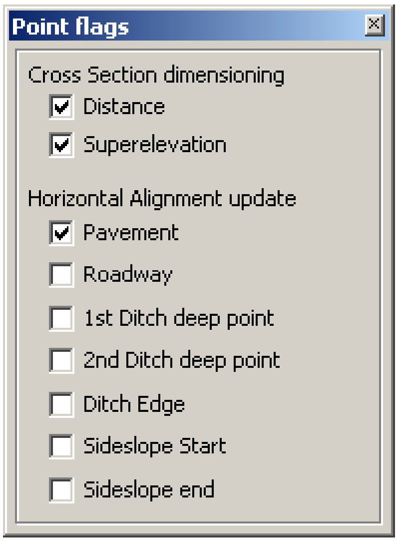
How to transfer the Mass Table data to Microsoft Excel

1. In the *Mass Haul* dialogue box, click the 

button.
2. In the resulting dialogue box, specify the path and the name of the file and click the *Save* button.

9.24 Dimensioning Points - Roadlines

To go to the dimensioning mode, lick the  button on the View toolbar. In the cross section you can see the dimensioning and roadlines points, which determine the dimensions on the printout and create the roadlines in the Horizontal Alighment.



For a point to be taken into account in the printout dimensioning, select the point and from the *Flags definition* form, select *Distance* if you wish to view the distance between two successive points, or *Superelevation* if you want to view the superelevation of the segment to which the point belongs. If you want to view both the *Distance* and the *Superelevation*, select both options.
During cross section calculation *with Typical*, dimensioning is performed automatically if these points have already been defined in the typical cross section.

From the second section of this form you can define how the *Horizontal Alignment* roadlines will be created. More specifically, you can define the cross section points, whose distances from the axis will be transferred to the *Horizontal Alignment* when you run the Update roadlines from *Cross Sections to Horizontal Alignment* command. This kind of definition allows for a fully parametric configuration of the roadlines in the *Horizontal Alignment,* so that the roadlines reflect the calculated structures in the *Cross Sections* in the best possible way.

9.25 Print Setup

In order to print-draw the cross sections, use the commands of the *Print* menu. The commands of this menu are:

1. *Page Setup*.
2. *Print Preferences*.
3. *Cross Section Printing*.

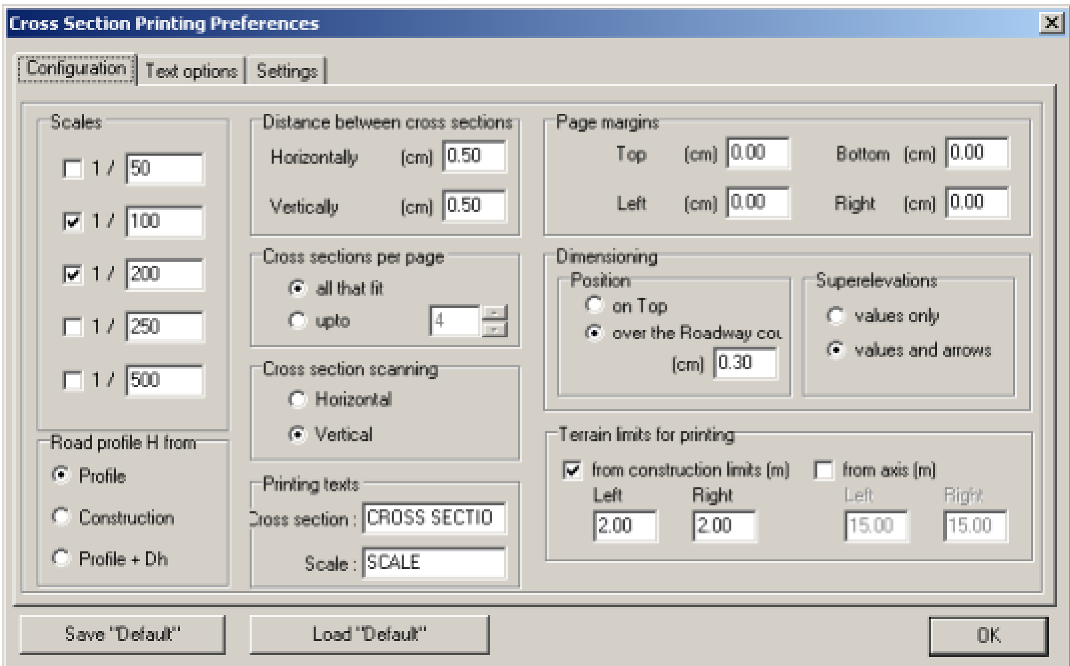
These commands must be used in the order they appear in the print menu.

Print Preferences

The cross sections printing settings include the scale, the page margins, the font size on the printout, as well as some more detailed settings. To make the aforementioned settings, run the *Print preferences* command of the *Print* menu. The resulting window has three tabs:

1. The *Configuration* tab.
2. The *Text options* tab.
3. The *Settings* tab.

Configuration



The *Configuration* tab enables you to set multiple scales, so that the program may select the most appropriate one. In the *Scale* group, you can set up to five scales. To enable or disable a scale, tick or untick respectively the check box to the left of the scale. You can also set directly the value of every scale, by typing it in the right check box.

In order to print the cross sections, the program places the cross sections starting from the top left part of the page and continuing down or to the left, depending on your selection in the *Cross section scanning* group. For every cross section, the program checks whether it fits into the available height and width of the page, and if not, it places the cross section by using a lower scale. If the cross section cannot be printed in any scale, then the program adds this cross section to those that do not fit in the page and proceeds to the next cross section. When printing is completed, the program informs you on the cross sections that were not printed; these cross sections remain selected.

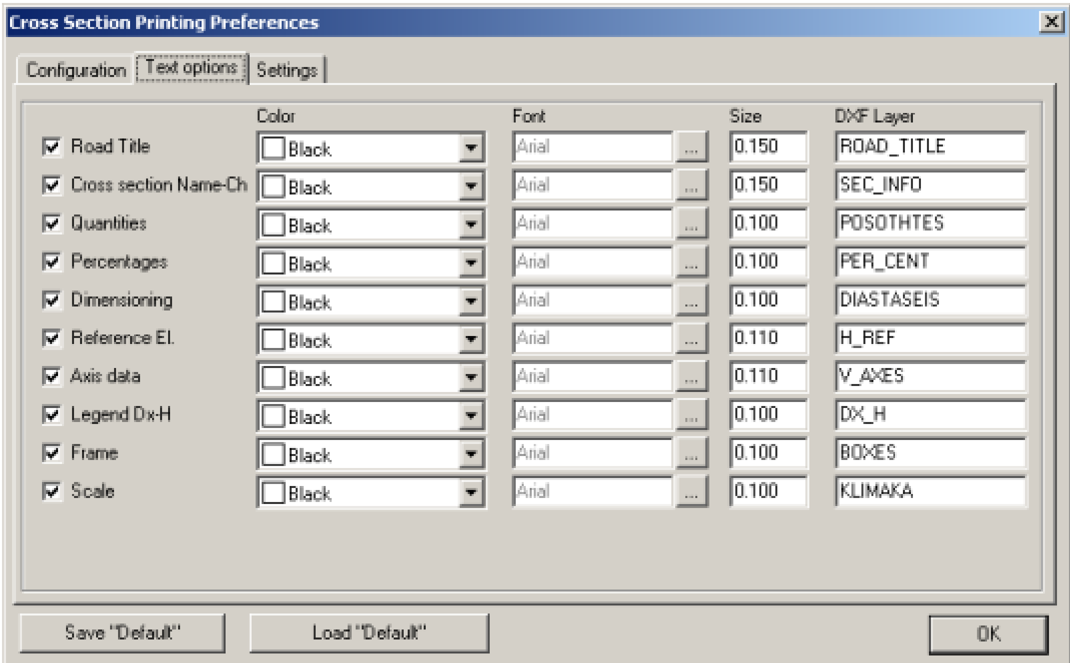
In the *Cross sections per page* group, tick the *Up to* check box and enter the desired value to set the maximum number of cross sections that will be printed on one page. If you wish to print on one page all of the cross sections that fit, tick the *all that fit* check box.

In the *Distance between cross sections* group, set the distances between two cross sections that will be printed on the same page. In the *Horizontally* (cm) and *Vertically* (cm) fields, enter the horizontal and vertical distance, respectively. In the same way (in cm) you can set, if desired, the margins that will be left around the page, by entering the relevant values in the *Top* (cm), *Bottom* (cm), *Left* (cm) and *Right* (cm) fields of the *Page margins* group.

In the *Natural ground printing limits* group, set in meters the natural ground and the topsoil that will be printed left and right to the extremes of the cross section works and from the axis. If you wish to print the whole natural ground, simply enter a high value, e.g. 200. If you tick both check boxes, namely *from construction* (m) or/and *from axis* (m), the program calculates both distances and draws the natural ground up to the longest resulting distance.

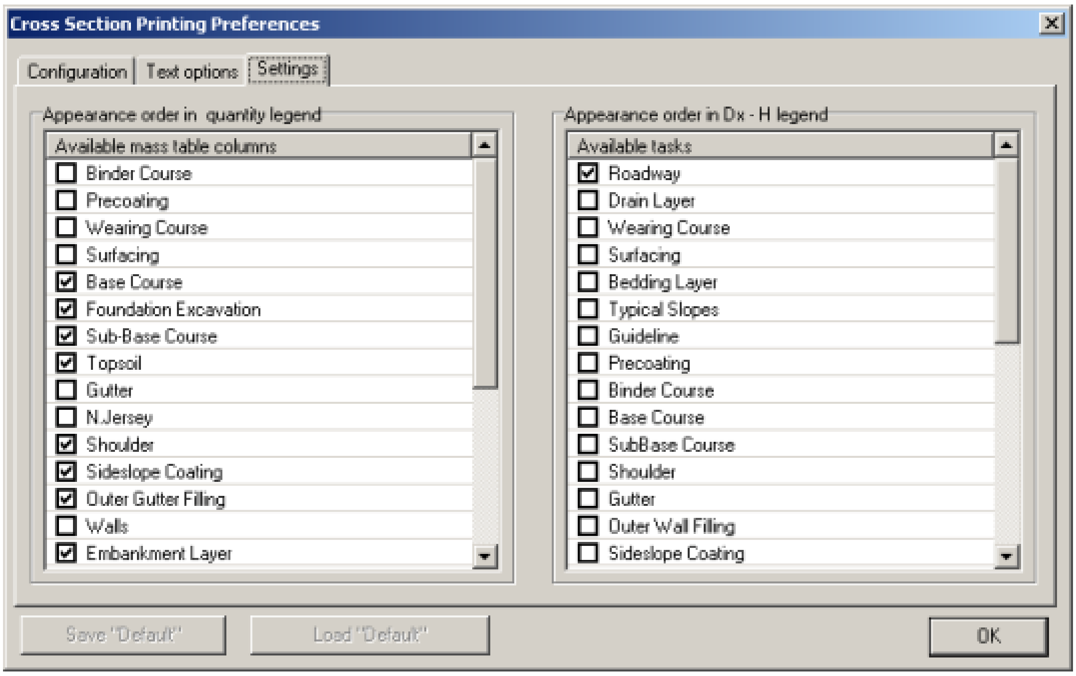
The *Dimensioning* group refers to the dimensioning of the cross sections' roadway and to the printing of the superelevations. If you wish to place the dimensions on top of the cross section, tick the *Position* check box. If you wish to place the dimensions at a fixed distance, over the roadway, tick the *Over the roadway* check box and in the field below, enter the distance in centimeters. In the *Superelevations* group, select whether the printout will include only the values of the superelevations or both their values and their arrows.

Text options



In the *Text options* tab, you can choose which of the available texts will be displayed on the cross section printout, by ticking the respective check boxes. You can also select the font, the font color and the font size. If the cross sections are to be exported in a DXF file, in the same tab you can select the layer to which a text will be exported.

Settings

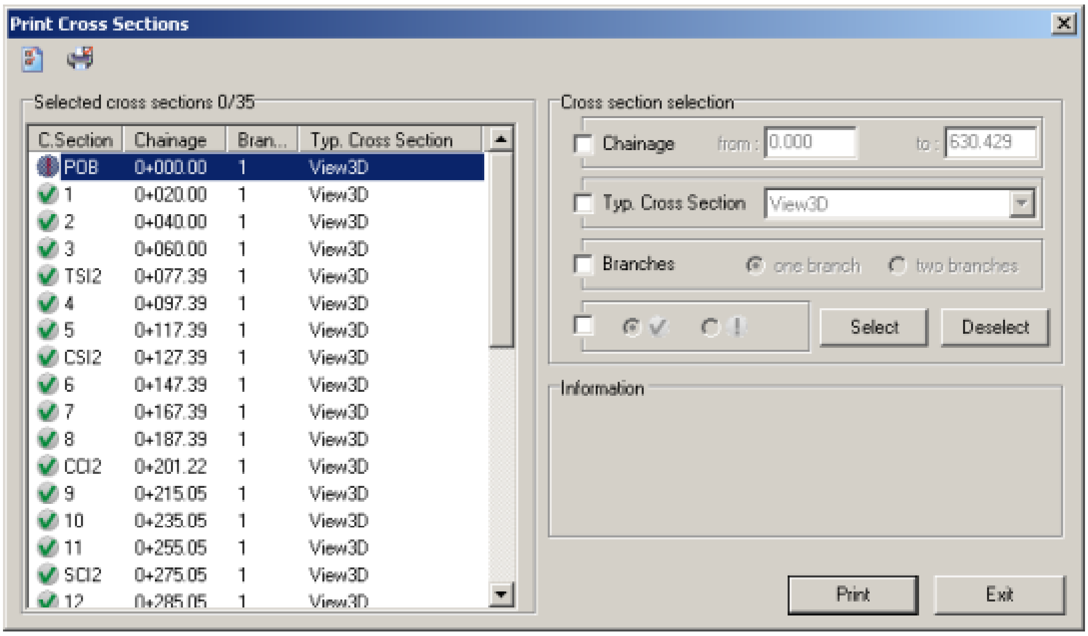


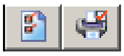
This *Settings* tab is comprised of two groups.

In the *Appearance order in quantity legend* group, select the areas or the lengths that will be printed below the name and chainage of the cross section. In the *Appearance order in Dx-H legend* group, select the lines whose coordinates will be printed in a legend below the cross section drawing. Note that these settings are also available in the dialogue box of the *Typical Tasks* command.
The procedure is the same for both groups: If you want to print a quantity or a task, tick the check box next to its name. If you wish to change the order of appearance on the printout, use the mouse to select the desired quantity or task, and then drag-and-drop it to the desired position.

9.26 Cross Sections Printing

To print cross sections, run the *Print cross sections* command of the *Print* menu.



The  buttons at the top of the resulting window open the dialogue boxes for setting the print preferences and the page setup preferences, respectively. From the *Selected cross sections* and *Cross section selection* groups, select the cross sections that will be printed.

To preview and then print the selected cross sections, click the *Print* button. A window will appear on the screen, showing the print preview for the selected cross sections.

To print the cross sections drawing, click the  button in the preview window. The dialogue box of the *Print* command opens and you can make your last settings before printing the cross sections. Next, click the *OK* button to start printing.

To close the preview window, click the *Exit* button. The *Information* group shows you how many cross sections have been selected, how many pages are required for printing the selected cross sections, and how many cross sections fit in the page margins, according to the page setup. If you printed the cross sections (not just previewed them), then only the cross sections that didn't fit in the page margins will remain selected, otherwise all cross sections that you selected before the preview will remain selected until you print them.

9.27 Data Export

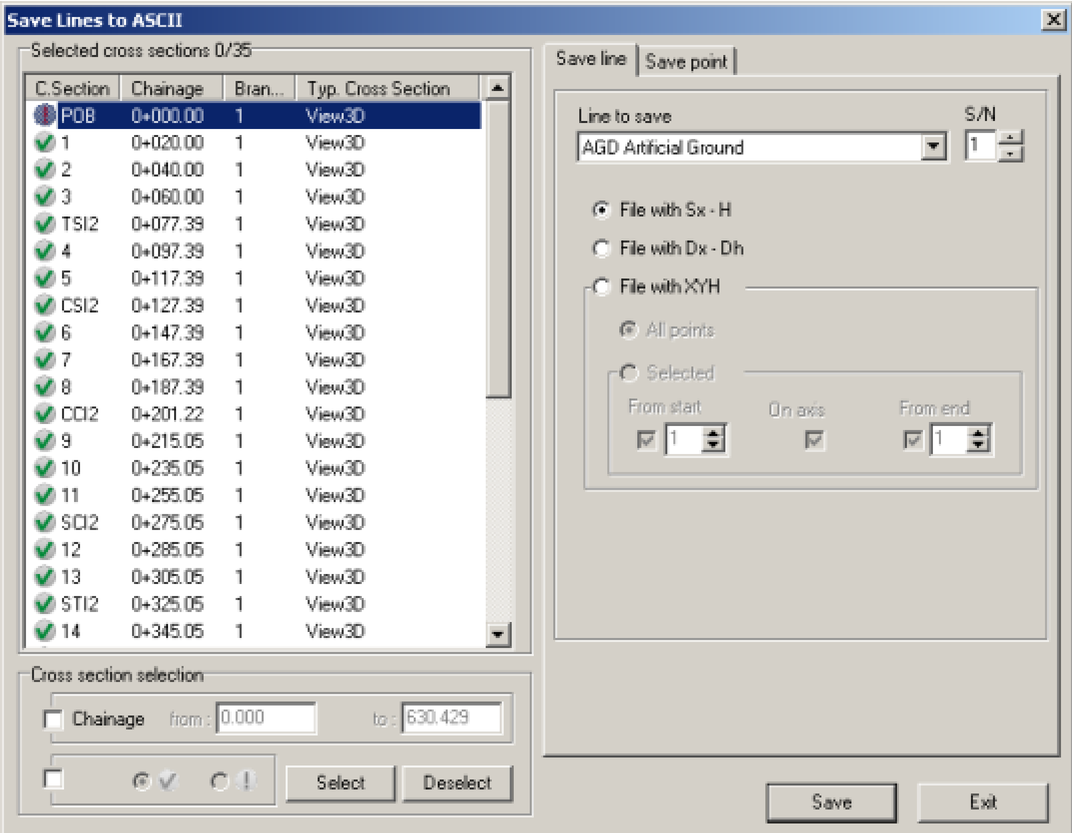
Exporting Lines to an ASCII File

The cross sections' lines can be saved in an external text file.

How to export the cross sections' lines to an ASCII file

1. From the *File* menu, select the *Export to ASCII* submenu and run the *Lines* command.

2. The following dialogue box appears on the screen:



From the *Selected cross sections* and *Cross section selection* groups, select the cross sections whose lines will be included in the new file.

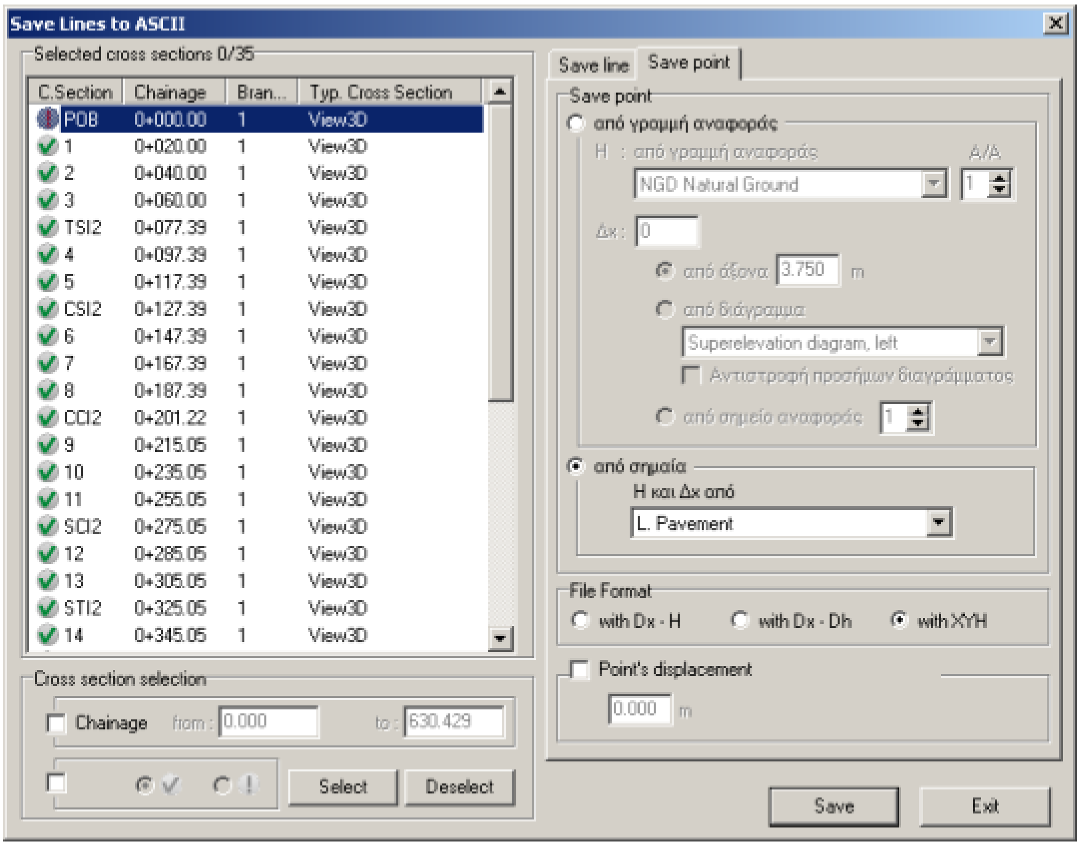
3. Next, open the *Export line* tab and select the line you wish to export as well as the file type. Select the line from the *Line to save* drop-down list and in the *S/N* field enter the line's serial number, in case that a cross section has more than one lines of the same typical task. If the value you enter is 0, then all cross section's lines of the selected typical task will be exported. In this case, the points between the lines are separated in the text file by a set of two numbers, -98765 -98765. This separation is also taken into account when importing multiple lines from a text file. You can create three types of files. The *File with Dx – H* option results in a file, in which the line coordinates are determined according to the distance from the axis and the absolute elevation. With the *File with Dx – Dh* option, the line coordinates are determined according to the distance from the axis and the elevation discrepancy from the profile elevation in the cross section. Finally, when the *File with X,Y,H* option is selected, the coordinates of the lines' points are determined according to the *Horizontal Alighment* of the road. If you select the *All points* option, the resulting file contains all points of the selected line. If you select the *Selected* option, the resulting file contains up to three points, depending on the

ticked check boxes. One of these points is on the axis and the other two are defined from the start or/and the end of the line. The points in the file appear in ascending order according to the chainage, and the points of the same cross section are shown from left to right.

4. After you have set all parameters, click the *Export* button. Specify the path and the name of the file you wish to create.
5. Click on *OK* to create the file. The lines' coordinates will be exported to the specified file that will have the ACS extension.

How to export one line point per cross section to an ASCII file

From the *File* menu, select the *Export to ASCII* submenu and run the *Lines* command. The following dialogue box appears on the screen:



From the *Selected cross sections* and *Cross section selection* groups select the cross sections whose points will be exported to the new file.

Next, open the *Export point* tab. You can export a point in two ways: either by specifying directly the point or by setting the point's distance from the axis. Both ways are explained in detail below:

<u>Directly:</u>

- Select the reference line and from the *from reference point* field, enter directly the number of the reference line's point that you wish to export (-1 for the last point, -2 for the last but one point, etc.).
- With the *from flag* option, the point to be exported is the one you have already marked with a flag in the cross section (e.g. Pavement, Roadway, etc.).

<u>Indirectly:</u>

- By selecting the reference line and then the *from axis* option, you can set directly the point's distance from the axis. When the point is located to the right of the axis, the distance has positive values. When the point is located to the left of the axis, the distance has negative values.

- By selecting the reference line and then the *from diagram* option, the point's distance from the axis will be determined according to the diagram's values. If you wish to increase the distance determined from the diagram, enter the value of the additional distance into the *Dx* field. If you are referring to the left side and the diagram's values are positive (e.g. pavement width, roadway width), tick the *Invert diagram signs* check box.

The *Point displacement* option enables you to move the selected point on the cross section according to the distance that you enter in the relevant field; the displacement does not affect the point's elevation.
After you have set all parameters, click the *Export* button. Specify the path and the name of the new file and click *OK* to create the file.

<u>You can also export the following items to an ASCII file:</u>

- *Percentage Export:* This command exports the percentages of every cross section. The resulting file will have the .APS extension and shall include the Cross section name, the Chainage, and the percentages of the Soil, the Semi-Rock and the Rock.
- *Cross Sections Data:* With this command you can view in a file all of a cross section's data. The file will have the .TXT extension and shall include the Cross section name, the Chainage, the X and H of the ground (point on axis), the Azimuth, the cross section Angle, the cross section Distance from the main axis, the Oblique width of the left and right pavement, and the Oblique width of the left and right roadway.
- *Calculation Data:* This type of export allows you to view in a table the way in which the cross sections have been calculated. The resulting file has the .TXT extension and includes the following information: the Cross section name, the Chainage, whether or not the chainage has been calculated, the Calculation way (with Typical or with Points), the Typical cross section that has been used, the Paving left and right to the axis, the left and right Sideslopes, as well as whether there is a left and right Guardrail.
- *Applied Walls:* This command exports all data of the applied walls. The resulting file will have the .TXT extension and shall include the Cross section name, the Chainage, the Side to which the wall has been applied, the wall height, and the wall type (Filling/ Cutting).

Exporting Files to DXF

From the dialogue box of the *Typical Tasks* command of the *Parameters* menu, you can select the form and the appearance of the lines, as well as the lines that will be exported to the DXF file and the layer to which they will be exported.

The form of all other elements that will be exported, is defined from the dialogue box of the *Print Preferences*command of the *Print menu*. Type the names of the layers for every category of elements in the respective *Dxf layer* text box.

How to export data to a DXF file

1. From the *File* menu, select *Export to DXF*.
2. Select the desired cross sections from the dialogue box that appears on the screen. From the *Selected cross sections* and *Cross Section Selection* groups, select the cross sections that will be printed.
3. Click the *Export to Dxf* button. The *Export Cross Sections to Dxf* window will open, showing you a preview of the cross sections. The window is similar to the *Print Cross Sections* window and most icons are the same in both windows. The only difference here is the *Dxf Export* icon instead of the printer icon.
4. Click this icon and in the resulting dialogue box, enter the name of the DXF file in the *File Name* field.
5. In the *File Type* field, select the type of the file according to the desired version, *14*

or 2000.

6. Finally, click *Save* to complete the creation of the file.

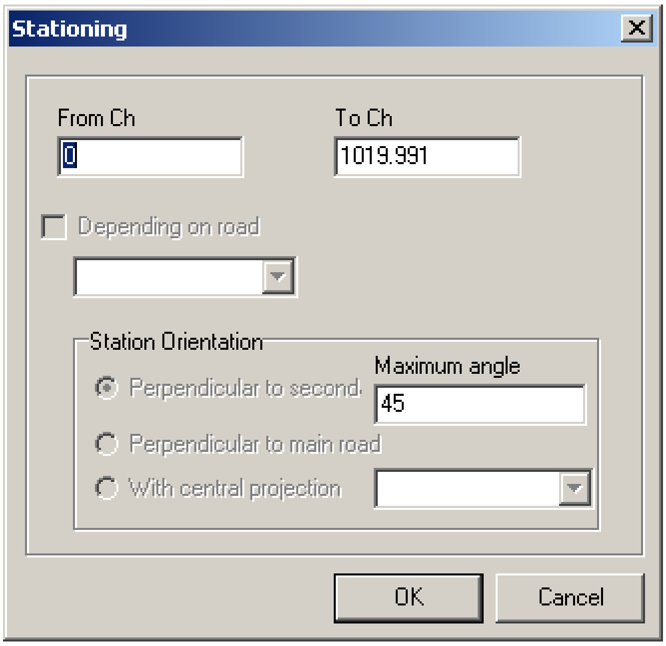
All of the selected cross section will be included in the same DXF file. The layout will be the one that you have selected in the *Print Preferences*. Before exporting the file, you must have also set the correct drawing scale, which also determines the font size. Throughout the program, the data size (names, terrain point elevations, PI names, station names and any other text) is given in cm. In a DXF file, this size is translated into "universal" m. The conversion takes place by using the drawing scale.

Chapter 10

10 Composite Cross Sections

10.1 Horizontal Alignment

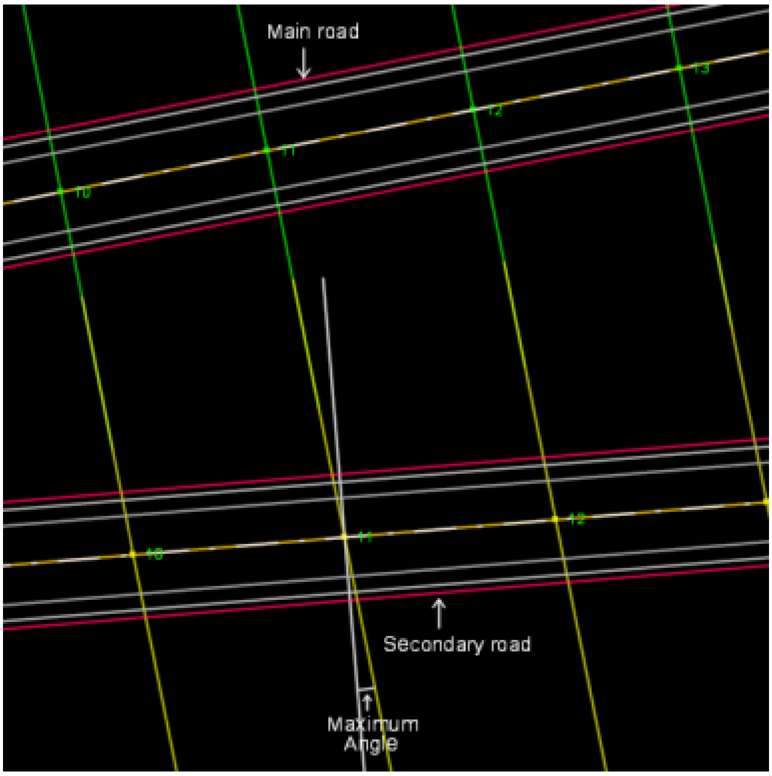
The composite cross sections presuppose that at least one or more roads (dependent – secondary) in the Horizontal Alignment workspace must have cross sections on the extension of another road's (main road) cross sections. To create composite cross sections, proceed as follows:
After the creation of the roads (main and secondary or only secondary) and the stationing of the main road, you must proceed to the stationing of the secondary road based on the main one. To do this, from the *Road Design* menu, select the *Stationing* submenu and then run the *Stationing (Partial/Dependent)* command. In the resulting window,



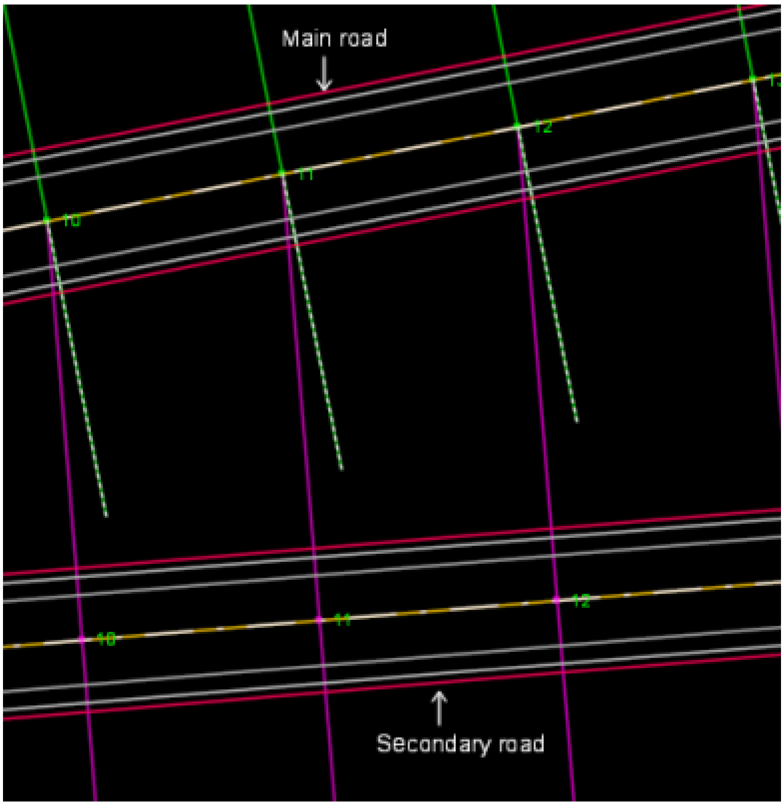
select whether the dependent stationing will be applied to the whole secondary road or to a specific part of it. This part is defined in the *From Ch.* and *To Ch.* fields. Tick the *Depending on road* field to choose the road based on which the stationing of the secondary road will take place.

In the *Station Orientation* group, select one of the available options:

- *Perpendicular to main road:* With this option, the cross sections of the secondary road become extensions of the main road's cross sections. In the *Maximum angle* field, enter the maximum allowable angle from the perpendicular to the secondary road's axis; the cross sections that have an angle higher than this value will not be inserted in the secondary road.

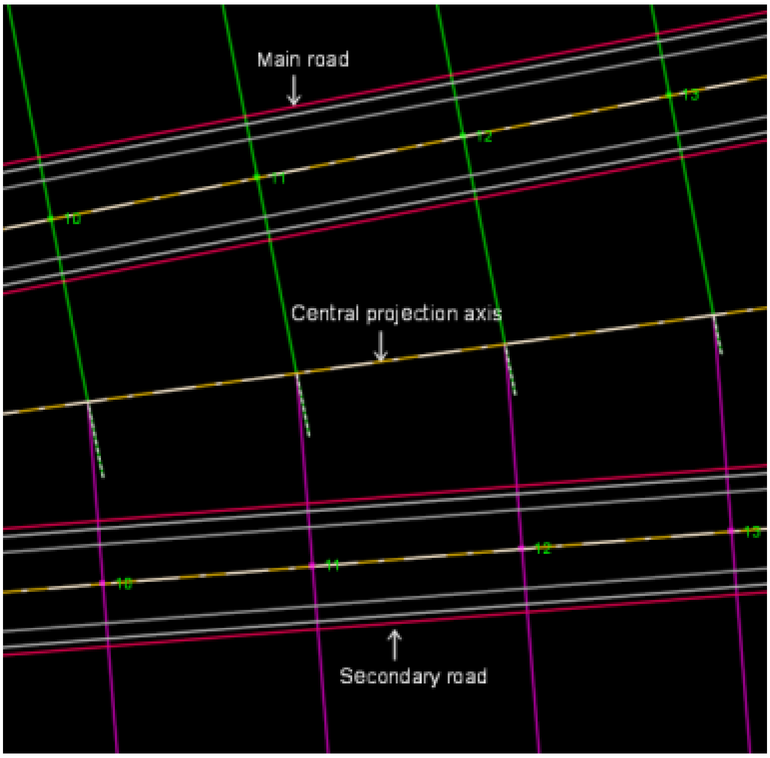


- *Perpendicular to secondary road:* With this option, the positions of the main road's cross sections are projected to the secondary road. The resulting cross sections are perpendicular to the secondary road. This type of dependent stationing does not generally result in composite cross sections.



- *With central projection:* If you select this option, then, apart from selecting the main road on which stationing will be based, you must also select another road –

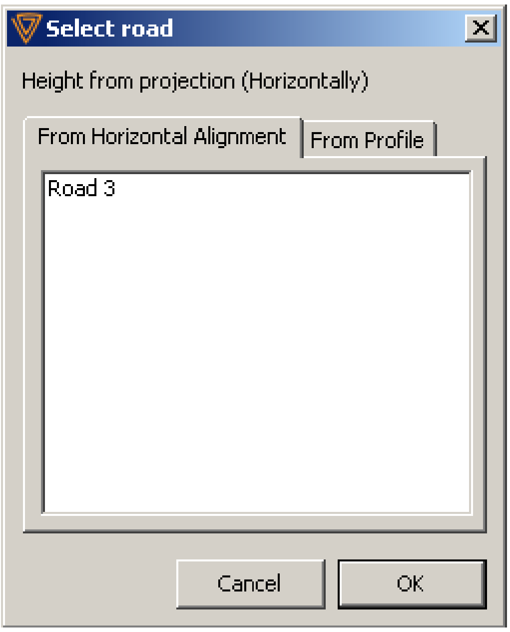
axis, on which central projection will be based. With this option, the cross sections that will be created in the secondary road can be used for the creation of composite cross sections. The resulting composite cross sections will be placed at two intersecting levels, whose sections will be vertical lines, passing from the points of intersection with the central projection axis.



Next, the appropriate updates must be carried out separately for every road, from the Horizontal Alignment to Profile and from the Horizontal Alignment to Cross Sections. Then you must create the composite cross sections from the main road. To do this, select the main road, open the *Project* menu, run the *Manage* command and in the Update group, select from *Horizontal Alignment to Composite Cross Sections*.

10.2 Profile

An important tool for studying the profiles of roads that are involved in the creation of composite cross sections is the *New Tangent Curve from projection* command of the *Profile* menu. This command enables you to view the road profile of a secondary road in the current profile.
The elevation can be calculated either directly from the road profile of the secondary road or from the road profile diagram that exists in the horizontal alignment of the secondary road.
Note that a road's cross sections do not always depend on the other road.
For the aforementioned operation, open the *Profile* menu and run the *New Tangent Curve from projection* command. The resulting window that appears on the screen,



has two tabs, which include all of the project's available roads. To update using the data of the road profile diagram of the Horizontal Alignment, select the road that is displayed in the *From Horizontal Alignment* tab. To update using the data of the road profile curve of the profile workspace, select the corresponding road in the *From Profile* tab. Click the *OK* button to confirm your selection and return to the *Profile* workspace, or click *Cancel* to undo your selections.

10.3 Cross Sections

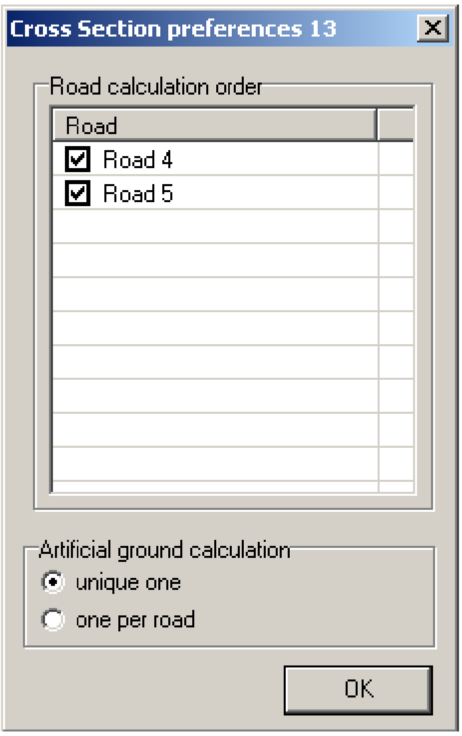
In the cross section's workspace of every road, you will notice that the axis of the other road appears at the positions of the composite cross sections. The axis of the current road appears in red, while the axes of all other roads appear in blue.
After the calculation of cross sections (separately for every road), you will notice that the outline of the every separately calculated cross sections appears in the cross sections space. To view the composite cross sections, you must go to the respective workspace either by using the drop-down list in the *View* toolbar, or by clicking the respective button at the bottom of the screen (in the same way that you are transferred to the cross sections of a road).
The *Oblique Widths* must be used for the dependent cross sections (of the secondary road), so that the widths, which are increased due to the obliquity of the cross section, as well as the superelevations and slopes are defined more accurately. For the batch calculation of the cross sections with the use of the oblique values, select *Calculation* from the *Batch Actions* menu and then open the *Paving-Slopes* tab to enable the *Use oblique values* option. If you want to apply the oblique values to a particular cross section, click the  button.
After you have selected the cross sections of the secondary road by using the oblique values, keep in mind that the values of the typical cross section will be distorted (e.g. encasing solids, paveway – shoulder superelevation, sideslopes slopes).

<u>Cross Sections Composition</u>

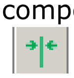
In the *Composite Cross Sections* workspace, you will notice that a new menu *(Composite)* appears next to the *Batch Actions*; this *Composite* menu is enabled only if you are currently on the composite road. Therefore, to display all calculated cross sections in the composite cross sections space, from the *Composite* menu select the *Update Composite Cross Sections* command. To update only the current cross section, from the same menu select *Update Composite* or click the  button. All lines of the calculated cross sections of both roads will appear on the screen.

<u>Composite Cross Section Preferences</u>

To define the preferences for the calculation of composite cross sections, from the *Composite* menu select the *Composites' Preferences* command. In the resulting window that appears on the screen,



you can select the *Road calculation order* and the method that will be used for the composition of the cross sections.
To change the road calculation order, select the road (e.g. Road 1) and drag it below Road 2, if, for example, you want Road 1 to be the second road that will be calculated. If you want to disable a road so that it does not appear at all in the *Composites* workspace, left-click in the relevant check box to untick it. In the *Composite cross section with* group, select a unique artificial ground if you want the cross sections to have a single artificial ground line, or select many different artificial grounds, if you want the composite cross sections to have different artificial grounds.

Click the *OK* button to confirm the changes and return to the *Composite Cross Sections* workspace.
To set the preferences for a particular cross section, from the *Composite* menu select *Cross section preferences* or click the  button. The same window appears, only this time the changes you are making are applied only to the current cross section.
After the update of the composite cross section and the setting of the preferences, you can proceed to cross section calculation by running the *Calculation* command of the *Batch Actions* menu. Now all composite cross sections will have been calculated *with Points*. You will notice that some green dashed lines have appeared on the screen. These lines demarcate the boundaries of every cross section's ground, so that splitting may take place on the proper axis when you update the individual spaces from the composite cross sections. You can also enter a splitting axis manually, by clicking the  button.

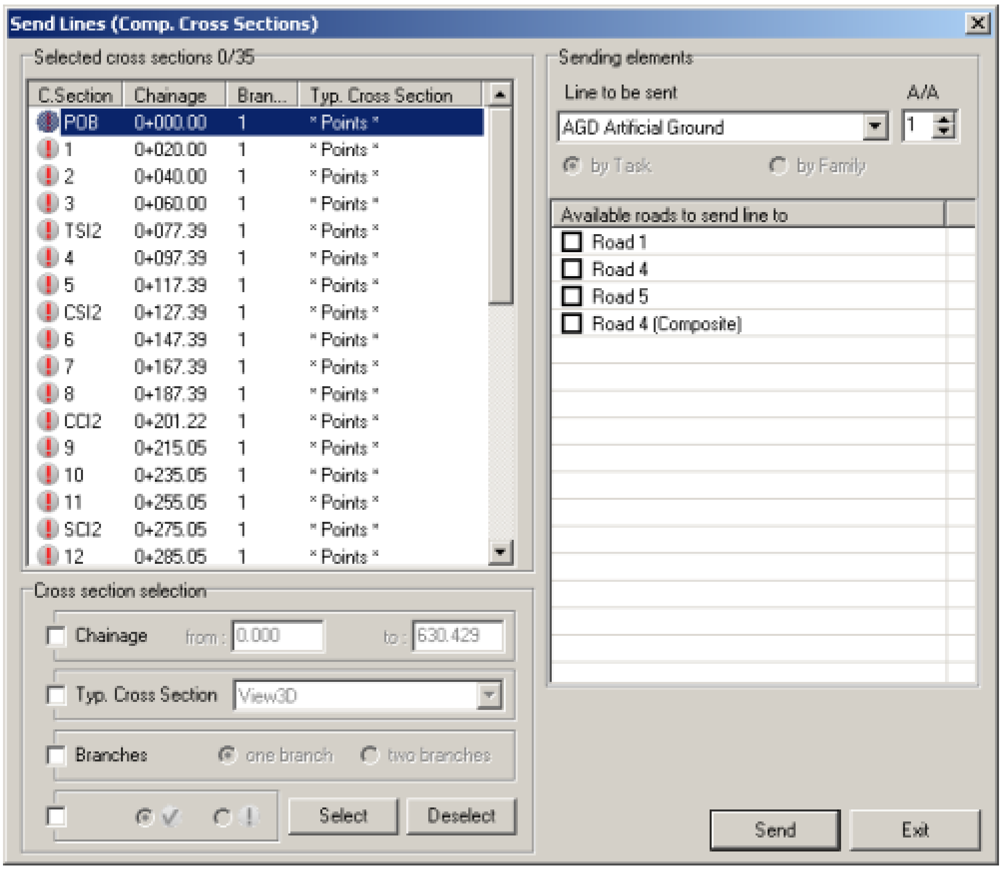
Additional Features

The *Composite* menu offers the following additional options:

- *Update Individuals from Composites:* This command updates the individual cross sections from the composite ones.
- *Synchronize Typical Tasks:* With this option, you can synchronize the typical tasks of the composite road with the ones of the individual roads.
- *Recreate Links:* Use the link recreation option when you make certain modifications (such as deletion or update from the horizontal alignment) in the cross sections of the secondary road and you want to update the composites.

Sending Lines

Another feature is the exchange of lines between the roads that comprise the composite road. To do this, run the *Send Lines (Comp. Cross Sections)* commands of the *Batch Actions* menu. The following window will appear on the screen:



In the *Line to be sent* field of the *Sending elements* group, select from the available lines of the drop-down list the line you wish to send from the composite cross sections to another road (or roads). In the *S/N* field, enter the serial number of the line that will be sent to another road. The S/N matters only if the selected line is a line that can be found in a cross section more than once (-1 to select the last line, -2 to select the last but one line etc.). To send lines of the same typical task or family, enter 0 in the *S/N* field and then select one of the two options (*by Task* or *by Family*). In the *Available roads to send line* to group, select the road to which the selected line will be sent. Click the *Send* button to send the line. Click the *Exit* button to close the dialogue box and return to the *Cross Sections* workspace.

Chapter 11

11 Typical Cross Sections

11.1 Overview

In general, there are three ways to create a project's typical cross sections in *Anadelta Tessera*. In the first way, cross sections are created automatically from the *Horizontal Alignment* workspace and, apart from the natural ground line in every cross section, they also include the pavement width diagram, the superelevation diagram and the shoulder width diagram. In the second way, cross sections result from the data of the *Profile* workspace. In the third way, the user inserts the cross sections in the *Cross Sections* workspace and the natural ground of every cross section is inserted either collectively from ASCII files (e.g. ACS files), or by the user, separately for every cross section. In all of the aforementioned cases, the user can either select to calculate the cross sections based on a typical cross section, or to insert the data of the other cross sections' lines (artificial ground, paving layers, etc.) manually.

Every cross section project has a library of typical cross sections, which are displayed in the *Typical selection* drop-down list, in the left part of the *Typical Cross Section Data* window. To open the *Typical Cross Section Data window*, go to the typical cross sections workspace by clicking the *Typicals* button at the top of the *Cross Section Data* window. To return to the cross sections workspace, click the *Cross Sections* button.

A typical cross section can be either created in the typical cross sections workspace or imported from a file (*.ATD or *.ADF). To import a typical cross section from a file, go to the *Cross Sections* workspace, open the *File* menu, select the *Import* submenu and run the *Typ. Cross Sections* command.

You can also save one or more typical cross sections in an external file, in order to use them in another project or keep them as a backup copy. To do this, go to the *File* menu, open the *Export* submenu and run the *Typ. Cross Sections* command.

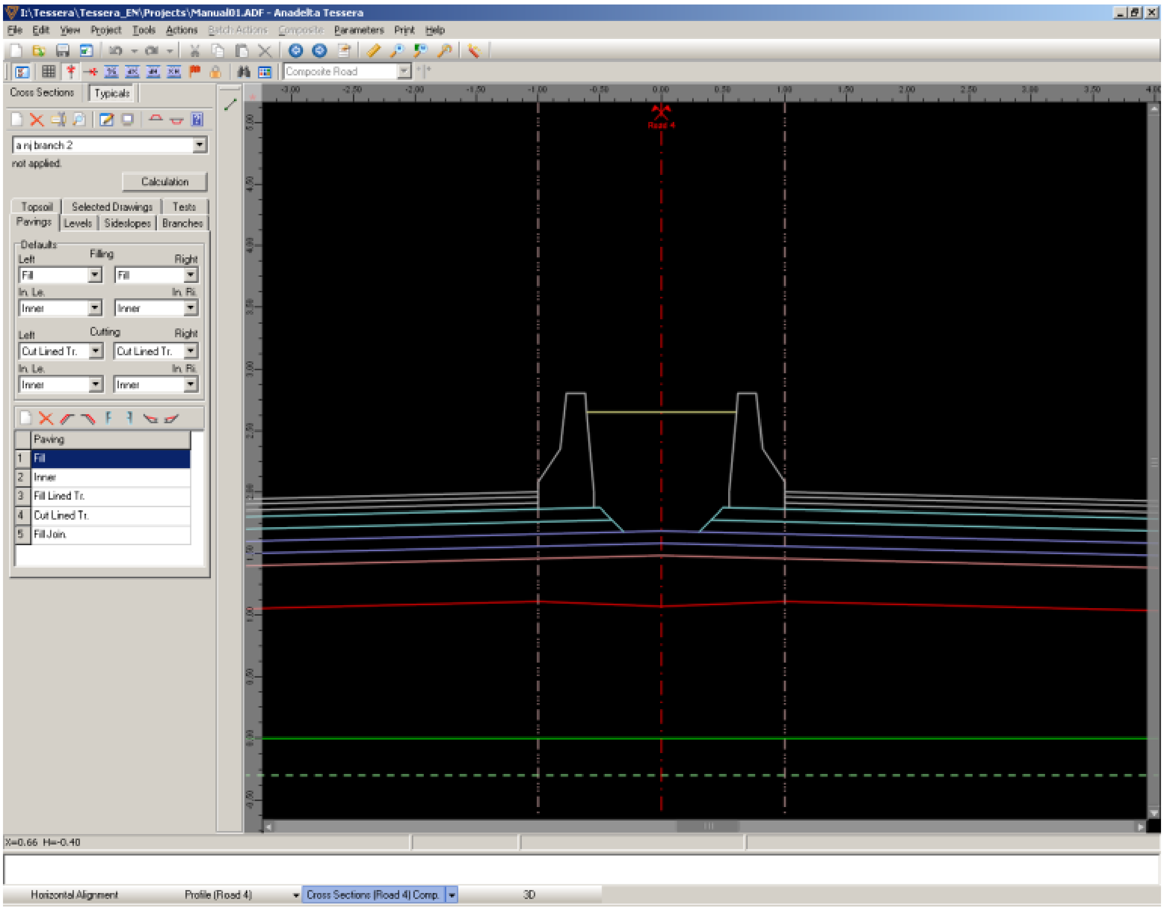
A typical cross section has two formations, one for the filling and one for the cutting. The program enables you to define separately the left and the right part of the cross section for each one of these two formations. Besides, a typical cross section can be either defined in an absolute way, namely with specific widths and superelevations, or in a dynamic way, according to the respective diagrams.

When a typical cross section is applied to – calculated in a cross section of the project, it is actually adjusted to the data of the current cross section. This applies especially to the dynamic typical cross sections, whose values (widths, superelevations, etc.) depend on diagrams. These typicals are adjusted to every cross section according to their chainage and to the diagram values for the particular chainage.

11.2 Operations

To describe the typical cross section mode, we must create a new file.

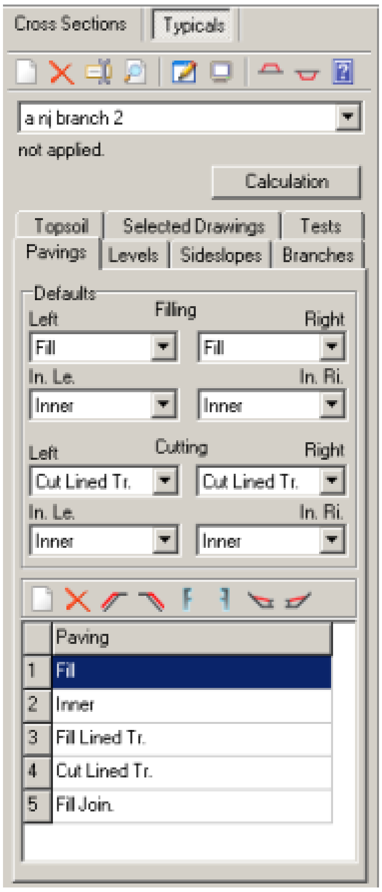
Left-click on the *Typicals* button to enable the typical cross sections' creation and editing mode and to display one of the typical cross sections that are included in our project:



This mode enables you to create and test a typical cross section, by making separately the settings for all main parts of the typical.

The program screen in the *Typicals* workspace is very similar to the window of the *Cross Sections* workspace and has the same menus. The difference here is that certain commands of the *Actions, Parameters* and *Print* menus are disabled. This window also features the main toolbar and the View toolbar. The *Cross Sections* toolbar has only one button, the one that creates the line.

A new window appears now in the left part of the program screen (in the same place where the Cross Section Data window appeared in the *Cross Sections* workspace). This is the *Typical Cross Section Data* management window, which is very similar to the previous one.



The management window is primary, since here you can make most of the settings referring to a typical cross section, while you can also view all values of the calculation parameters for the current typical cross section. The name of the current typical cross section (the one you see on your screen) is displayed in the *Typical Cross Section Selection* drop-down list of the window. Exactly below this list, there is a number, showing you to how many cross sections the current typical cross section has already been applied in your current project.
If you wish to enlarge the drawing area, you may choose not to display the management window. To do this, click the  button in the View toolbar. If you wish to display again the *Typical Cross Section Data* window, click again the same button. When the management window is closed, the name of the typical cross section appears in blue letters at the top left of the drawing area.

The message window is displayed at the bottom of the screen and exactly below you can see the status bar, just like in the *Cross Sections* workspace.

When defining the typical cross section, and generally every time you define slopes or superelevations, the values must be entered in percentages, while every time you define distances, elevations, Dx, Dy values, they must be entered in meters. E.g., for a 2/3 or 1/1.5 slope (where 2/3 is the ratio of y/x) enter 66,6666 or for a 1:1 slope enter 100, while for a 3:1 slope enter 300).

11.3 Management

Importing Typical Cross Sections from a File

To import typical cross sections from an external file into the current project, use the *Typ. Tasks* command of the *Import* submenu of the *File* menu. You can import typical cross sections either from another project of *Anadelta Tessera* (*.ADF), or from a typical cross sections text file. Note that the typical cross sections text files that are exported from *Anadelta Tessera* have the *.ATD extension.

How to import typical cross sections from another file

1. From the *File* menu, select the *Import* submenu and run the *Typ. Cross Sections* command.
2. The *Open* dialogue box appears on the screen. From the *File Type* field, select the type of file (*.ATD or* .ADF), then select the desired file and click *Open*.

Creation of Typical Cross Sections Files (*.ATD)

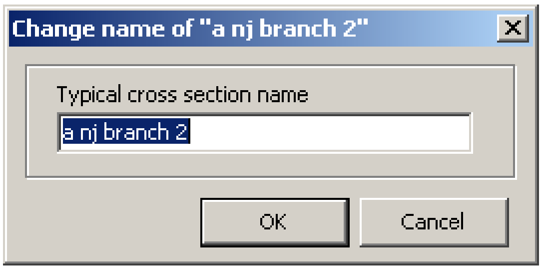
You can save one or more cross sections of a project in the program's external library *(C:\Program Files\Anadelta\Tessera\Csec)* as a typical cross sections file. The default extension of these files is .ATD.

How to export typical cross sections to a text file

1. From the *File* menu, select the *Export* submenu and run the *Typ. Cross Sections* command.
2. The *Save* dialogue box appears on the screen. Enter the name of the file and then click the *Save* button.

Changing the Name of a Typical Cross Section

To change the name of the current typical cross section, click the *Name Change*  button in the toolbar at the top of the management window. In the resulting dialogue box,



type the new name in the relevant field. Now the new name will be displayed in the typical cross section selection list.

Typical Cross Section Selection

To select a typical cross section of the project, use the drop-down list in the *Typical Cross Section Data* window.
This is the *Typical Selection* list, where you can see the name of the current typical cross section. If you left-click on the list, a menu appears with all typical cross sections of the current project, and you can choose the typical you wish to edit. When you select a typical cross section, below the list you can see whether this typical has been applied or not to any of the current project's cross sections.

TypicalCross Section Creation

By default, every new file contains some typical cross sections. You can use these typicals as they are or you can edit them in order to meet the requirements of your projects. In addition, you can create as many new typical cross sections as you want.

How to create a new typical cross section

1. From the *Typical Cross Section Data* window, click the *New Typical Cross Section* button.
2. In the resulting dialogue box that appears on the screen, enter the data of the new typical cross section (name).
3. When you have finished, click the *OK* button to create the new typical cross section.

The clicking of this button creates a new typical cross section, which is an identical copy of the one that was the current typical cross section before you started the creation process. The new typical cross section will be displayed in the drawing area and in the *Typical Selection* drop-down list.

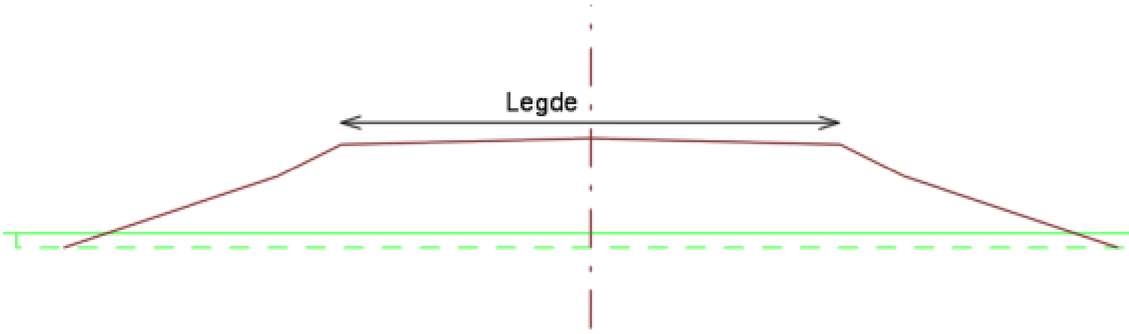
Deleting a Typical Cross Section

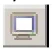
To delete a typical cross section, click the *Delete Typical* command in the typical cross sections management window. In the confirmation message that appears, click *Yes* to confirm that you wish to delete the typical cross section, otherwise click *No*.

11.4 Main Parts

A typical cross section is comprised of certain main parts, which can be defined in a number of ways. These main parts are described below.

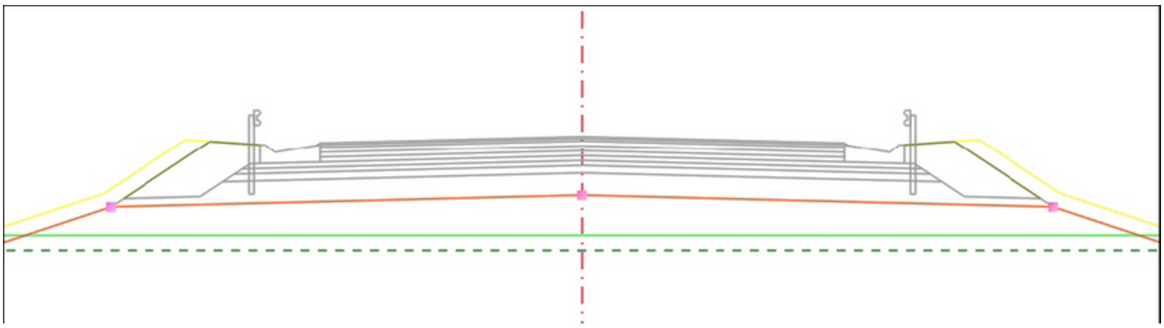
The simplest form of a typical cross section has only the Artificial Ground line. An example of such a typical cross section is shown below:



Select it to display the cross section. To view the cutting and filling formations, click the  or the  button, respectively. The Artificial Ground is comprised of a single line. The top part of the line (the one where the paving layers are seated) is the ledge, while the side sloped parts are the sideslopes. To view the ledge parts, click the  button. The reason for this distinction is that the points of the Artificial Ground's ledge line are defined in a different way than the sideslopes points.

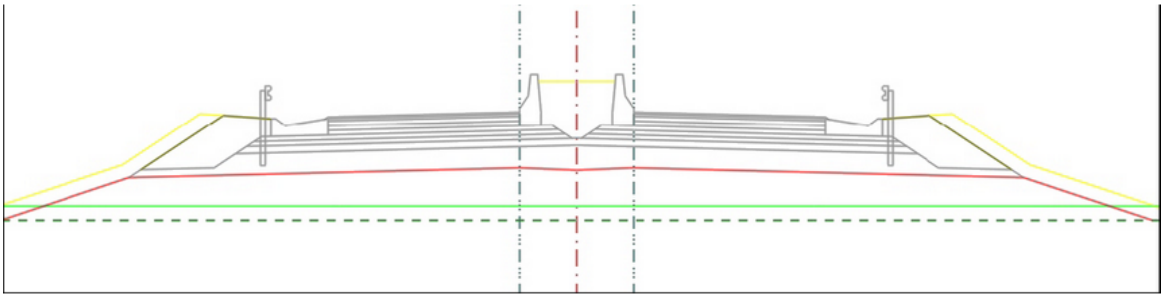
The number of a typical cross section's branches is defined in the *Branches* tab. The type of the branches is not binding, since the same typical may be used for cross sections with one or two branches.

When the two branches option has been selected, you can view on the screen the left and right semiaxes, as well as the formations of the typical inside the branches. To better understand the aforementioned information, from the drop-down list select the typical cross section *A nj b2*:



The cross section that is displayed, corresponds to the single-branch formation. Next, from the *Typical cross section points* window, open the *Branches* tab. In this tab, choose the *two branches option*.

Now you can see on your screen what the typical's formation will appear when applied as a two- branch typical:



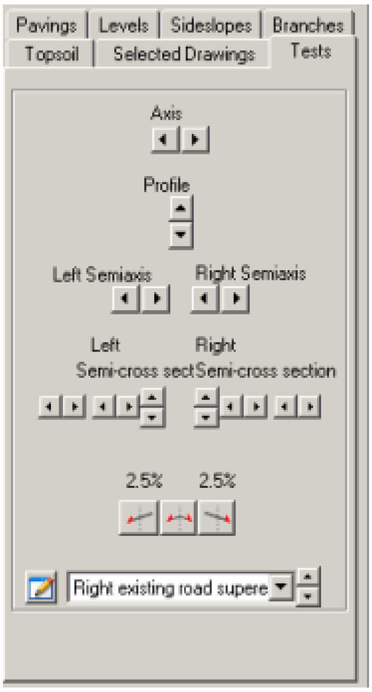
Therefore, you can use this typical in your project both as a single-branch typical and as a two-branch typical.

In the *Sideslopes* tab, enter the sideslope slopes for the current typical cross section. In a typical cross section, you must define the minimum sideslope slopes both for the cutting and for the filling. The sideslope slopes are not unique, meaning that in the same typical, you can enter different sets of sideslope slopes and use them accordingly when the typical is applied to your road's cross sections. For example, in the A nj b2 typical cross section, two different sideslope formations have been defined for the cutting. In the *Sideslopes* tab at the bottom of the window, you can see the available sideslope formations for the cuttings and for the fillings. Display the formation of the cross section in a cutting by clicking the  button and from the *Left Cutting* drop-down list select a different sideslope formation. The selected formation appears to the left of the cross section. To restore the cross section in its original state, select from the drop-down list its previous formation.

The meaning of the term paving in this manual and in the program includes all of a cross section's elements on the Artificial Ground (paving layers, shoulders, encasing solids, etc.), as well as the Artificial ground ledge. The left and right paving formation is not unique. Namely, in the same typical you can define various paving formations, which can be applied during the calculation of your project's cross sections. Select, for example, the typical cross section A nj b2. Next, from the *Typical Cross section Data* window, select the *Pavings* tab. At the bottom of the window, you can see the available pavings, which can be applied whether the cross section is in a cutting or in a filling. In the *Left Filling* drop-down list of the *Defaults* group, select *Fill Lined*. instead of *Fill Lined Tr.*. A new formation will be displayed in the drawing area, with a triangular ditch and a different shoulder formation. Note that although a typical cross section supports many different paving formations, all pavings must always have the same number of layers and their layers must have the same thickness.

11.5 Tests

Through the tests we can see how the typical cross section behaves in various superelevations and widths values; these values are taken from the virtual diagrams that simulate here the true superelevation and width values to which the current typical cross will be applied. Please note that the typical cross section is not modified during the tests, it is only adapted to-tested in various diagram values. When the *Tests* tab is selected, the layout of the *Typical Cross Section Data* window is displayed as shown below:



More specifically, you can change (to increase or to decrease the value) the axis position, the distance of the left and right semiaxis (two-branch cross sections), the roadway width and the pavement width. In a similar way, you can also change (to decrease or to increase the value) the profile elevation and the half-cross section elevation. The values of these sizes can vary for the two sides of the road, this is why there are two sets of buttons, one for each side. Every time you change a value, the typical cross section drawing changes accordingly. To view the current value of a particular size, place and leave the mouse pointer over one of the buttons that control this size.

At the bottom of the tab there are three buttons . Use the first button to change the superelevations of the road, like on a left turn. By clicking the second button, the superelevations get their values on the tangent, while the third button changes the superelevations like on a right turn.

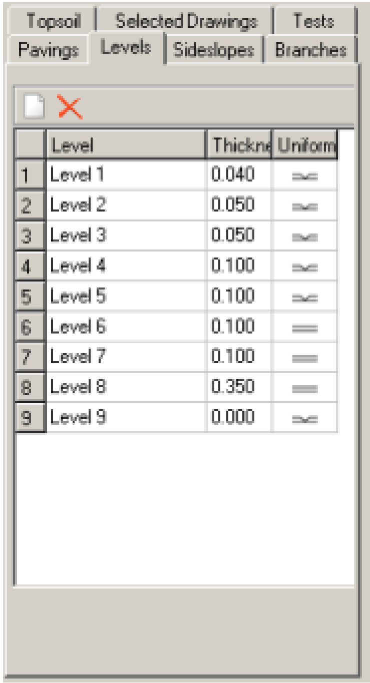
At the bottom of the tab you will find the button, which opens a dialogue box; in this box you can choose the desired diagram and enter directly the desired values. Finally, from the drop-down list at the bottom of the tab, you can choose any of the user-created diagrams and with the assistance of the buttons on the left of the list you can decrease or increase their values.

11.6 Creation and Insertion of Levels

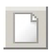
Every typical cross section is divided horizontally into levels (layers). The levels of the typical cross section represent the true paving layers and any change in the levels is the only reason for the creation of a new typical cross section.

Namely, you will have to create a new typical cross section or to modify the existing one only if you wish to change the number or the thickness of the typical cross section's layers in a part of your project. If you wish to change the branches, the sideslope slopes, the formation of the paving layers or the formation of the shoulder, to change the encasing solids or to insert ditches, etc., you don't have to create a new typical cross section.

Every typical cross section has at least one level that includes by default the Artificial Ground ledge; this is the bottom level. You can define as many additional levels as you wish, namely as many as the desired paving layers of your typical cross section.



In the *Levels* tab, you can define the number of the paving layers and the thickness of each paving layer, but you cannot define the number and the positions of the paving layer's points. You can do this in the drawing area, by opening the *Pavings* tab and running the *Edit point* command.

To insert the layers, click the  button. Click the button as many times as the desired number of layers. The order in which the layers appear in the *Levels* tab, from top to bottom, signifies their order by elevation. The elevation of the axis (profile elevation) is applied to the axis of the highest layer.

If there are already some layers and you create a new one, the new layer will have the same characteristics as the layer that you had selected before creating the new one and it will be added exactly below the selected layer.

For every layer you must define the following:

- *Thickness:* Set the thickness of each layer at its axis. To set the thickness, in the *Thickness* column and in the field that is on the same row with the layer's name, type the value in meters.

- *Uniform:* If your typical has two branches, you must specify whether a layer is uniform, namely whether a layer is common to both branches. In case that the current typical cross section has two branches, a third column, the *Uniform* column is displayed in the *Levels* tab. Double-click in the field of this column to make the respective layer uniform or not. This action does not affect the use of the layer in the single-branch cross section.

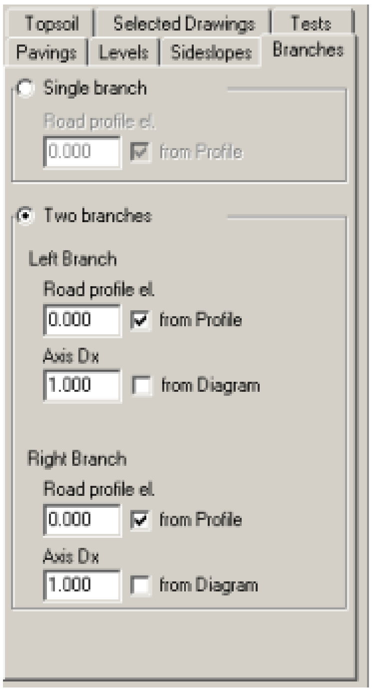
To delete a layer, select the layer's field and click the  button. Keep in mind that you cannot delete the last/bottom layer, which corresponds to the Artificial Ground line.

To specify the *Layer Material*, namely the material that the layer will be made of, you must change the typical task to which the layer belongs. You can do this in the drawing area, by opening the *Properties* command window. After you have selected the line, open the command's window by pressing the *F11* key or by running the command from the menu that is displayed if you click the right mouse button. Change the material from the drop-down list of the *Task* property, which contains all typical tasks that have been created in the relevant window.

11.7 Setting the Branches

The typical cross section can be defined as single (one branch) or with two branches. In *Anadelta Tessera*, you can use the same typical both for the parts where the road has a single branch and for the parts where the road has two branches.

Here we must underline that the definition of a typical cross section as a single-branch cross section or a two-branch cross section is not binding. It only means that the semiaxes of the branches are displayed in the drawing area and that you can insert lines into the inner pavings. During cross section calculation, your choice is applied as default, but you can change it for certain cross sections.



To set the number of branches, open the Branches tab and tick the *Single branch* check box for a typical with one branch or the *Two branches* check box for a typical with two branches. Any change in the number of branches will be reflected immediately in the typical's drawing area. In the case of a two-branch typical, both semiaxes are displayed on the screen.

If you select the two branches option, the *Left Inner Pav.* and *Right Inner Pav.* drop-down lists are enabled in the *Pavings* tab, both for the cutting and for the filling. From these drop-down lists, select the formations that the layers will have on the inner sides of the two branches; this formation refers to the pavings.

Setting the Road Profile Elevation for the Branches

You can set the road profile elevation either in an absolute way or according to the profile diagram. In the first case, type the road profile elevation in the *Road Profile el.* text field; in the second case, tick the respective *From Profile* check box. Do the same separately for every branch if the typical has two branches.

The road profile elevation is always applied to the cross section axis and at the highest level. If the cross section has two branches, then the road profile elevation is defined and applied separately to every branch. If you wish the profile elevation to be applied lower or higher than the value of the road profile diagram, type the relevant distance (in meters) in the *Road Profile el.* text field. In this case, the roadway elevation of the cross section to which the typical will be applied, will be the algebraic sum of the user-defined value and the value that the road profile diagram has at the chainage of the particular cross section. Note that a positive distance value shifts the cross section upwards, while a negative value shifts the cross section downwards.

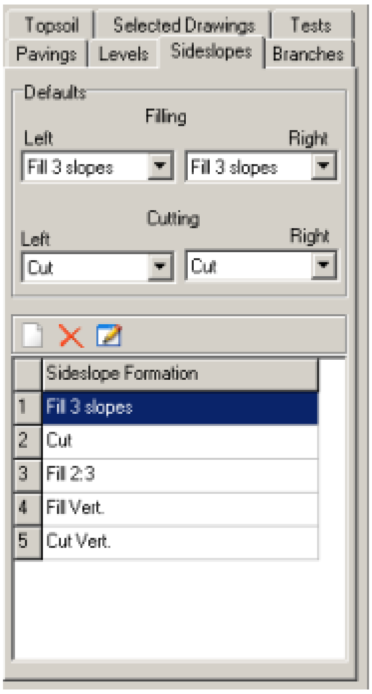
Distance of the Semiaxes of a Two-Branch Typical

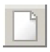
The distance of the left or right semiaxis of a two-branch typical cross section can be either fixed or it may depend on a diagram (Dx of left or right semiaxis). In the first case, type in the *Axis Dx* text field the desired distance (in meters), while in the second case tick the *from Diagram* check box. To combine the above two cases, enter a zero value in the text field and at the same time tick the check box. In this way, the distance of the semiaxes in every cross section will be the algebraic sum of the value in the text field and the value of the Dx of left or right semiaxis diagram at the respective chainage.

11.8 Sideslopes

When defining the typical cross section, you must set the sideslopes, both for the fillings and for the cuttings. To define the formation of the sideslopes, open the *Sideslopes* tab.

As mentioned earlier, a typical cross section may contain multiple sideslope formations. To define a sideslope formation, set the sequence of the sideslope's slopes.



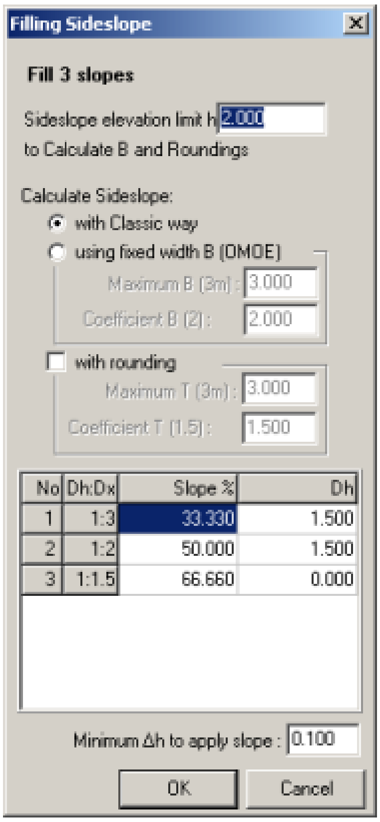
At the bottom of this tab, you will see all the existing sideslope formations for the current typical cross section. Every typical cross section must have at least two sideslope formations, one for the filling and one for the cutting. To create a new formation, select one of the formations that are listed in the table (e.g. Filling 1:3, 1:2, 2:3) and click the  button. A new formation, which is a copy of the selected one, will be displayed on the screen. The initial name of this new formation is comprised of the word New and the name of the sideslope formation that you had selected before clicking the button (e.g. New Filling 1:3, 1:2, 2:3). To change the name of the formation, simply type the new name in the formation's field. When you have finished, press *Enter* to confirm the name. To delete a formation from the table, select the desired formation and click the  button.

In the *Defaults* group, define the sideslope formations that you wish they are applied by default to the current typical cross section, both for the left and for the right part, as well as for the cutting and for the filling. The formations are selected from the respective drop-down lists of this group. The selected sideslope formation is applied instantly and appears in the workspace of the typical cross section. The default sideslope formations are also applied during the cross sections calculation, unless otherwise selected in the *Cross Sections Data* window during the calculation of a particular cross section or in the dialogue box of the *Calculation* command in the *Batch Actions* menu during batch calculation.

11.9 Setting the Sideslopes

To edit a formation's slopes, select the desired formation and click the  button. In the resulting dialogue box that appears on the screen, you can edit the sideslopes.

Setting the Sideslope Slopes



In the table at the bottom of the dialogue box, enter the slopes and the corresponding elevations. The slopes towards the natural ground are always positive and their values must be positive, regardless of the side, and regardless of whether they are referring to a cutting or a filling; all elevations are also deemed positive.
To define the formation of a cutting's slopes, in the table at the bottom of the dialogue box enter the sideslope slopes and the respective elevation for every slope. The order in which you must enter the slopes' values, is from the roadway to the ground. For the last slope, the program ignores the elevation value that you have entered and applies the slope up to the required elevation. For the cuttings' slopes, you may start from the first slope and continue setting some slopes, which will be applied by default, regardless of whether there is an intermediate section with the natural terrain. The program applies these mandatory slopes for the user-defined elevation, and then tests the first non-mandatory slope up to the particular elevation, and if no section with the natural ground is found, continues with the next slope, and so on.
To define a set of slopes as mandatory, select the last slope of the set and click the *Ctrl+Y* keys; use the same combination of keys to restore a slope to its non-mandatory state. The mandatory slopes are marked in blue color. The slope or the slopes that correspond to the ditches can be set as mandatory. In the case of a triangular ditch having a slope of 2/3, namely 66.66% downwards and a depth of 40cm, enter the mandatory slope -66.66 for an elevation of 0.4. The negative sign in the slope value is mandatory, because instead of approaching the ground, we are moving away from it, while the elevation, as mentioned above, does not require a sign.

You can form the filling slopes in two ways. In the first way, the calculation of the slope is based on the values entered in the relevant table, as in the case of the cutting. To use this method, tick the *Classic way* check box.

Contrary to the cuttings, the order in which you must enter the sideslope slopes and the respective elevations is from the ground to the roadway. For the last slope of the filling (the slope immediately after the roadway), the program ignores the elevation value that you have entered and applies the slope up to the required elevation. The program first tests the first user-defined slope up to the respective elevation, and if no section with the natural ground is found, continues with the next slope and so on.

The second way that you may use for the calculation of the filling slopes is based on the Road Works Design Directives. To apply this method, tick the *using fixed width B* check box and in the two text fields below enter the respective values for the calculation. In the *Sideslope elevation limit h* text field, enter the maximum sideslope elevation that will be calculated with this method. For elevations higher than this value, the sideslope is calculated with the Classic way.

In the *Minimum Dy to apply slope:* field, enter the minimum elevation that the last slope of a sideslope formation (cutting or filing) must have, in order to be inserted into a cross section. If the slope elevation is lower than the value that you have entered in the field, then the slope is rejected and the last slope to be applied is the previous slope in the list of the particular sideslope formation.
If you wish to have round sideslope edges (in the cutting or/and in the filling), tick the *with rounding* check box and enter the geometrical data of the rounding in the relevant text fields.

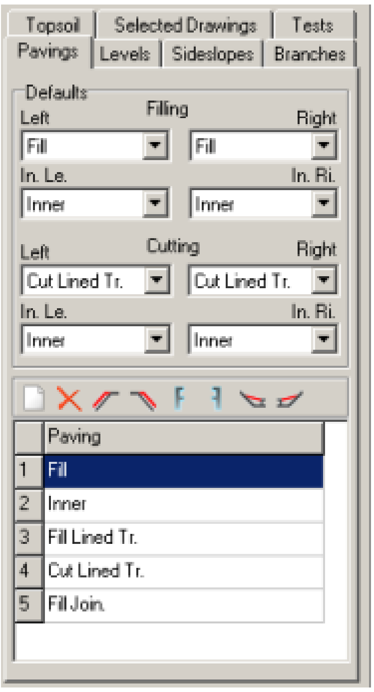
To confirm your changes for a sideslope formation, click the *OK* button. This saves the changes, closes the dialogue box and returns you to the *Sideslopes* tab. To view these changes in the typical cross section's workspace, you must have selected the edited sideslope formation in one of the four drop-down lists of the defaults group of the tab.

We remind you the three important points that you must always take into account when setting the sideslope slopes:

- The slopes towards the natural ground are deemed positive and entered as positive values, regardless of the side, and regardless of whether they are referring to a cutting or a filling.
- All sideslope elevations must be positive. If you give a negative sign to an elevation, the program will ignore it. The direction of the sideslopes is determined solely by the sign of their slope.
- The slopes referring to ditches (negative or zero values for trapezoid ditches) must be characterized as mandatory, in case you wish the whole ditch to be applied, even if there isn't any intermediate section with the natural ground.

11.10 Pavings

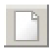
This paragraph describes how to set the parameters of the pavings, namely of the part over the Artificial Ground line.



As we have already mentioned, *Anadelta Tessera* supports the existence of multiple paving formations in the same typical cross section. By default, there are minimum two paving formations for the single-branch cross sections (cutting - filling) and three paving formations for the two-branch cross sections (cutting - filling - inner). Of course, you can insert more than one paving formations for the cutting or the filling, as you can see in the example of the figure: there are two formations for the filling, one for the cutting and one inner paving in case the typical cross section is defined with two branches.

In the *Defaults* group, set the paving formations that you wish to be applied by default to the current typical cross section, both for the cutting and for the filling, as well as for the right, left and inner part of the typical two-branch cross sections. The selected formations are displayed in the typical cross section's workspace, in the part where the formations will be applied according to your selection. Note that any changes in the formation of the pavings can only be made from the typical cross section's workspace, therefore you can change only a paving that has been selected in the *Defaults* group. The pavings are selected from the respective drop-down lists of this group. The selected paving is displayed instantly in the drawing area. To see an example, in the *Typical* mode select the A nj b2 typical. This typical has five pavings. While in the drawing area you have selected to view the cross section in a filling, in the *Defaults* group you will notice that the Fill. paving appears selected in the filling both for the left and for the right part of the cross section. From the respective drop-down list, select the *Fill Lined Tr.* paving for the right part of the filling. You will notice that, when the new paving is applied, the cross section changes in the drawing area.

The paving formations that you select from the *Defaults* group are also applied during the cross sections calculation, unless otherwise selected in the *Cross Sections Data* window during the calculation of a particular cross section or in the dialogue box of the *Calculation* command in the *Batch Actions* menu during the batch calculation.

To create a new paving, simply click the  button in the *Pavings* tab. A new line appears in the table below, where you can type the name of the new paving formation. The initial name of this new formation is comprised of the word New and the name of the paving formation that you had selected before clicking the button. Thus, the creation of a new paving results in a copy of the paving that you had selected before clicking the button.

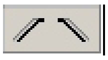
To delete a paving, select it by left-clicking in the respective field of the table and then click the  button. To change a paving's name, simply type the new name in the paving's field. When you have finished, press *Enter* to confirm the new name.

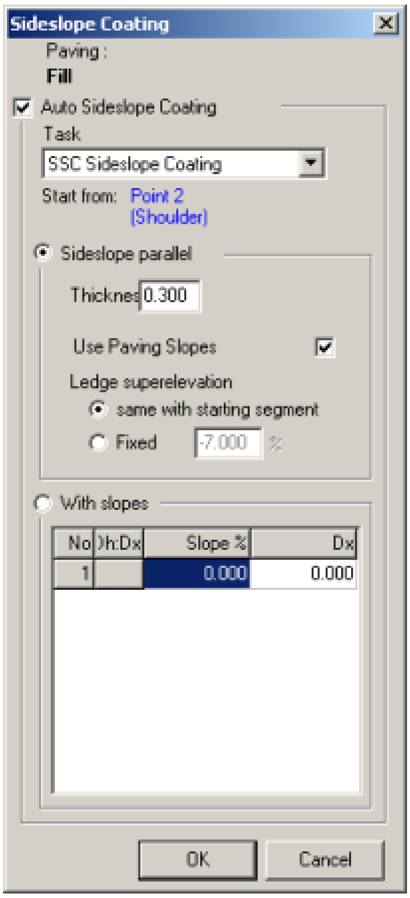
In the *Pavings* tab you define the number of pavings, as well as which of these pavings will be applied by default to the various parts of the typical cross section. The form of the paving (widths, layer superelevations, existence and types of shoulders, encasing solids, etc.) is defined from the drawing area. For this purpose, commands are used for inserting and editing lines in a typical cross section, as well as the *Edit point* command for the accurate calculation of a point's position in the typical.

11.11 Paving Sideslope Coating

The sideslope coating is defined from the *Pavings* tab and can be created automatically in a typical cross section. The coating is either parallel to the sideslopes (usually for the fillings) or to the slopes (usually for the cuttings).

How to set the automatic sideslope coating

1. From the *Pavings* tab of the *Typical Cross Section Data* window, click one of the  buttons, depending on whether the coating will be applied to the left or to the right of the typical cross section. The coating will be applied to the paving that has been selected in the *Defaults* group and according to whether you have selected the cutting or the filling.
2. The following dialogue box appears on the screen:



   For the sideslope coating to be applied automatically, tick the *Auto Sideslope Coating* check box. From the *Task* drop-down list, select the typical task to which the new line will belong. The list shows only the typical tasks of the SSC family. The typical task that you must select is usually *Sideslope Coating* for the filling or *Outer Gutter Filling* for the cutting.
3. Next, you must define the sideslope coating starting point. To define the point, left-click on the *Not defined* phrase below the drop-down list to display the *Select reference point phrase* at the top left corner of the drawing area. Select the desired point and then click the *Accept point phrase*. Now the field is updated and displays the selected point, e.g. *Starting from point 2 of the Shoulder*.
4. If the sideslope coating refers to a filling, tick the *Sideslope parallel* check box. Next, in the *Thickness* field, type in meters the distance of the coating from the sideslope. If you wish to also use the paving sideslopes (in addition to the artificial ground

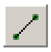
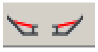
sideslopes) for the creation of the line, tick the *Use Paving Slopes* check box. If the filling refers to the cutting and to the creation of a debris removal zone, tick the *With slopes* check box. In the list below, enter the slopes and the horizontal step (distance) according to which every slope will be applied. The step of the last slope must always be zero. The program extends the last slope until it finds a section with the top or the bottom envelope.

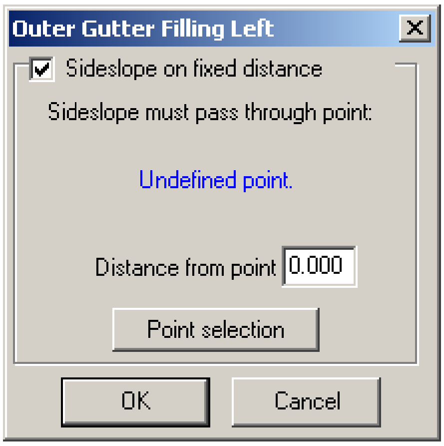
5. Click the *OK* button to close the dialogue box and create the line.

11.12 Outer Gutter Filling of Fixed Width

The *Outer gutter filling of fixed width* line is a special case of the Debris Removal Zone. When this line is defined to have a fixed width, then its end is a mandatory point through which the cross section's sideslope must pass.

How to define the Outer Gutter Filling of fixed width

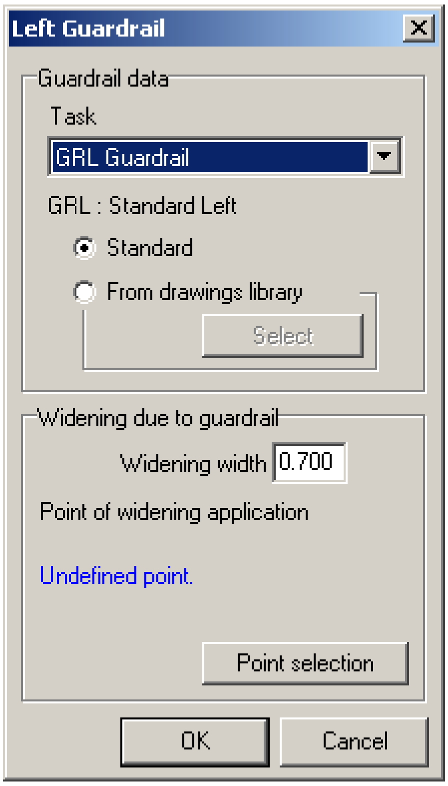
1. If there isn't any Outer Gutter Filling line, click the  button to create a new one. In the resulting window, select the *Paving* side, the Level of the new line and the type of the line you wish to create. Here, select the *Outer Gutter Filling* line. Use the *Edit point* command to select the calculation way (e.g. Dx and slope from previous) of the end point of the line.
2. Next, in the *Pavings* tab of the *Typical Cross Section Data* window, click one of the  buttons, depending on whether the outer gutter filling line of fixed width will be applied to the left or to the right of the typical cross section.
3. The following dialogue box appears on the screen:



4. Select the mandatory point through which the sideslope will pass and tick the *Sideslope on fixed distance* check box.
5. Click the *Point selection* button to confirm the point that you selected in the previous step.
6. In the *Distance from point* field, enter the additional distance of the Outer Gutter Filling.
7. Click the *OK* button to close the dialogue box.
8. Click the cross section *Calculation* to view the new form of the cross section.

11.13 Guardrails Support

To define how and where the *Guardrails* will appear in the cross section, open the *Pavings* tab of the *Typical Cross Section Data* window and click one of the  buttons, depending on whether the guardrail refers to the left or to the right part of the typical cross section. The clicking of either button results in the dialogue box shown below:



From the *Task* drop-down list of the *Guardrail* data group, select the task by which the guardrail will be inserted into the cross section. Next, select the design of the guardrail. This can be either *Standard,* or you can select it from the drawings library, by choosing the drawing that represents the particular guardrail. To do this, tick the *From drawing library* check box. To select the drawing that will be used for the guardrail, click the Select button to open the drawings library and then select both the drawing that will be used and the drawing's dependence point, which will determine the exact position of the guardrail in the cross section. With the use of the *Edit point* command, you can depend this point on another point of the typical cross section in order to define the exact position of the guardrail.

In the *Widening due to guardrail* group, you can select whether the cross section will be widened due to the guardrail. In the *Widening width* field, enter the desired widening of the cross section. The point to which the widening will be applied, is displayed below the relevant field and appears selected on the cross section. To change this point, select another appropriate point and click the *Point selection* button.

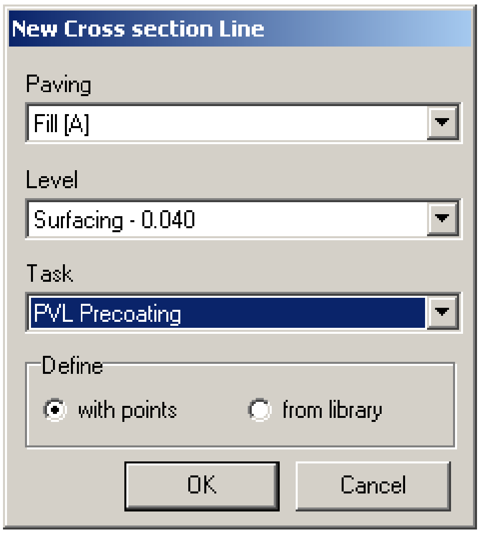
To confirm your settings and close window, click the *OK* button.

Note that in the *Typicals* mode the guardrails are always displayed, regardless of any elevation discrepancies, in order to remind you to define them correctly, so that they can be inserted into certain cross sections that use the particular typical.

To insert the guardrails into a given cross section, go to the *Cross Sections* workspace and click one of the  buttons, depending on whether the guardrail refers to the left or to the right part of the cross section. If you wish to calculate the guardrails in order to use them in all cross sections on user-defined conditions, run the *Guardrails Management* command of the *Batch Actions* menu.

11.14 Insertion - Editing of Lines

To insert a new line into a typical cross section, run the *Insert line* command of the *Actions* menu or click the  button. The following dialogue box appears on the screen:



This command is mostly used for inserting left and right pavement gutters, shoulders, and generally lines that correspond to a paving, apart from the lines that are inserted originally when creating a level. From the *Paving* drop-down list of the box, select the paving to which the new line will belong. Only the pavings that you have selected in the *Defaults* group of the *Pavings* tab in the *Typical Cross Section Data* window are available on the list. Next, select the level and the typical task to which the new line will belong from the *Level* and *Task* drop-down lists, respectively.

Finally, from the *Define* group select how the line will be inserted and then click the *OK* button to insert the line. The *with points* option means that the line will be defined by user-inserted points, while the *from library* option means that the line will have the form of a predefined drawing, which the user will select from the drawings library.

In the first case, define the line points either with the mouse, within the drawing area (you can also use the snap feature), or with the use of the *Input Form* command.

Editing Lines

The program offers various secondary functions, enabling you to edit the lines that you have inserted into a typical cross section, so as to give them the desired shape and properties.

<u>Moving a Line</u>

When you move a line, the whole line is transferred, without any changes regarding the orientation or the relative positions of the points with respect to each other. To ensure the necessary accuracy, use the *Input Form* command or the snap feature.

How to move one or more lines simultaneously by using the mouse

1. Select the lines you wish to move.
2. From the shortcut menu that is displayed when you right-click in the drawing area, select the *Move* command.

3. Specify the reference point based on which the line will be moved.
4. Specify the end point, where the reference point will be placed.

Once you have specified the second point, the line is moved to its new position.

Deleting Lines

You can delete one or more lines, if desired.

How to delete lines

1. Select the lines you wish to delete.
2. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Delete objects* command, or click the  button in the main toolbar, or press the *Del* key.

If you accidentally delete a line, you can restore the line by using the *Undo* command. Please keep in mind that the program will not allow you to delete a line on which other lines depend.

Editing line points

Inserting a Line Point

You can insert one or more points to a line.

How to insert points to a line

1. Select the line point next to which you wish to insert the new point. Note that the program allows you to insert the new point further than the cross section's axis with respect to the current point.
2. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Extend to start* or *Extend to end* command. The first command appears when the current point is on the left side of the road, while the second command appears when the current point is on the right side of the road. When the current point is at the cross section's axis, then both commands appear simultaneously. By running either command, a new point appears, which is highlighted in red. Move the mouse pointed to select the new point's position.
3. Set the position of the new point either by clicking the left mouse button or by using the *Input Form* command.
4. To finish the point insertion procedure, click the right mouse button. The insertion of the last points is cancelled and the procedure stops.

Offsetting a Line Point

You can offset a line point, so as to give it the desired shape.

How to offset a line point

1. Select the line you wish to edit and the point you wish to offset.
2. Activate the selected point. The point will start moving, following the movement of the mouse pointer.
3. Specify the new position of the point.

Deleting Line Points

You can delete one or more points of a line.

How to delete a line point

1. Select the line you wish to edit.
2. Select the point you wish to delete.
3. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Delete Point* command or press the *Ctrl+Del* keys.

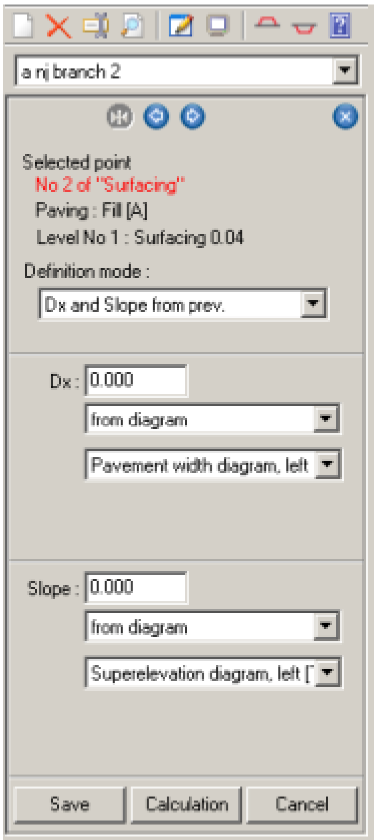
You cannot delete points on which other points depend. To do that, you must first use the *Edit point* command to edit all the dependent points and then you can delete the point.

11.15 Calculating the Position of a Line Point

To edit the points of a line in a typical cross section, use the *Edit point* command. With this command you can select the way in which a line point is calculated in the typical cross section, by correlating the position with a reference position in the same typical cross section (e.g. axis, a different point of the same or another line, a diagram, etc.).

How to define the position of a line point

1. Select the line you wish to edit.
2. Select the point you wish to edit.



3. From the shortcut menu that is displayed if you right-click in the drawing area, run the *Edit point* command or click the button in the *Typical Cross Section Data* window. The layout of the window changes, allowing you to define the position of a point. The name and the data of the selected line are shown at the top of the window. The selected point that you are editing is written in red letters. You can change the selected point either by using the buttons at the top of the window or by selecting the point from the drawing area, after having activated the box.
4. *From the Definition mode* drop-down list, select the desired way to define a point. There are five ways to define a point's position:
5. *Dx and Dy:* With this option you can define a point's position by entering the horizontal and vertical distance of the point from a reference position.
6. *Dx and Slope from prev.:* With this option you can define a point's position by entering the point's horizontal distance from a reference position and the slope of the straight segment between the current point and the previous point of the line (the point closest to the cross section's axis). This way is mainly used for defining

points of the top paving layer (e.g. traffic layer).

7. *Dy and Slope from prev.:* With this option you can define a point's position in a similar way as previously, by entering the point's vertical distance from a reference position and the slope of the straight segment between the current point and the previous point of the line (the point closest to the cross section's axis). This way is mainly used for defining sideslope points.
8. *Length and Slope from prev.:* With this option you can define a point's position by entering the length and the slope of the straight segment between the current point and the previous point of the line (the point closest to the cross section's axis).
9. *Section and Slope from prev.:* With this option you can define a point's position by entering the slope of the straight segment between the current point and the previous point of the line (the point closest to the cross section's axis) and the distance of this segment from the intersection with another segment of the typical cross section. This way is mainly used for defining points of a paving layer with respect to the sideslopes of the previous layer (e.g. Base, SubBase).
10. *Slope and Slope from prev.:* With the option you can define a point's position as a section between two half-lines; the first half-line is defined from the slope from another defined point and the other from the slope of the straight segment between the current point and the previous point of the line (the point closest to the cross section's axis).
11. Detailed explanations on how to fill in the relevant fields for each case are given hereunder.
12. To calculate the typical cross section according to the new data, click the *Calculation* button. To save the changes that you have made, click the *Save* button. To cancel all changes and restore the typical into its original state, click the *Cancel* button.

Setting the Dx or the Dy:

As we have previously seen, you can define a point by entering its horizontal or vertical distance from a reference position.

The Dx or the Dy distance can be set as a fixed distance from the axis (or the semiaxes if the cross section has two branches) for the horizontal distance (e.g. 3 m from the axis) and from the road profile elevation that you have set for the vertical distance. To do this, type the distance in the *Dx* or in the *Dy* field, and from the drop-down list below the field select *from axis*.

If you wish the distance to be measured from another previous point of the line, select *from point*. In this case, the phrase *Not defined* will appear below the drop-down list. This means that the point from which the distance will be measured has not been defined yet. To define the point, left-click on the *Not defined* phrase to display the *Select reference point phrase* at the top left corner of the drawing area. Select the desired point and then click the *Accept point* phrase. The field will be updated and the selected point will be displayed. To select a different point, repeat the procedure.

If you wish the horizontal or the vertical distance to be calculated from a diagram, then from the first drop-down list select *from diagram* and next, from the drop-down list that is displayed below, select the diagram based on which the point will be calculated. The *T* that appears next to every diagram's name indicates a typical cross section diagram, which can be edited from the *Tests* tab. In the typical's workspace these diagrams are virtual, but when the typical is applied to certain cross sections of the project, the program will search and apply the width of the particular (real) diagram at the cross section's chainage. The *from rev. diagram* option is the same, only here the values of the diagram are applied with the reverse sign. In the *Dx* or *Dy* text field, you may enter a fixed distance, which will be added to the current value of the diagram.

Settingthe Slope

To set the slope of the straight segment between the current and the previous point, choose one of the following options. If you choose the *absolute* option, type the point's slope in the text field. A positive slope means that the point will be placed at a higher elevation with respect to the previous point of the line, while a negative slope means the opposite.

If you wish the slope to be calculated from a diagram, then from the drop-down list select *from diagram* and next, from the drop-down list that is displayed below, select the diagram based on which the slope will be calculated. The *T* that appears next to every diagram's name indicates a typical cross section diagram, which can be edited from the *Tests* tab. In the typical's workspace these diagrams are virtual, but when the typical is applied to certain cross sections of the project, the program will search and apply the slope of the particular (real) diagram at the cross section's chainage.

The *from rev. diagram* option is the same, only here the values of the diagram are applied with the reverse sign. In the *Slope* text field, you may enter a fixed slope, which will be added to the current value of the diagram.

If you wish to apply the slope of another straight segment, then in the drop-down list select *from segment*. Select the segment in the same way that you select the reference point for the horizontal and vertical distances. Note that a segment's slope is defined from the second point that has been placed on the segment, which is usually located away from the axis, regardless of the side. Use this option if you wish the paving layers to have the same superelevation with the top layer, or if you wish the layer sideslopes to have the same slope with the sideslopes of the top layer. In this way, any changes in the slopes of the top layer are applied instantly to the other layers.

To set the slope of the road's shoulder, in the drop-down list select *from function for.* A shoulder's slope can be calculated in two ways: with *Variable superelevation* and with *Pavement superelevation*. The calculation way is selected from the drop-down list. More specifically:

<u>Calculation with Variable Superelevation</u>

To define how the shoulder' slope will be calculated, go to the *Parameters* menu, run the *Road preferences* command, and in the *Shoulders* group of the *General tab* select the desired calculation way by ticking the respective check box.

If you tick the *Edge superelevation difference* check box, enter the maximum value in the text field next to the box; the superelevation will be formed by segment, based also on the values in the *Shoulder tangent superelevation* fields. The superelevation of the shoulders on the tangent is the value that the user enters in the relevant field. For the inner side (downslope – lower side) of a turn, when the pavement superelevation exceeds the shoulder superelevation on tangent, then the shoulder superelevation follows the pavement superelevation. For the outer side (upslope – higher side), where an edge is formed, the shoulder superelevation maintains the tangent value, as long as the edge superelevation difference is lower than the value that you have entered in the Preferences. When the superelevation difference between the shoulder and the pavement becomes equal to the *Edge superelevation difference*, then every time the pavement superelevation is increased, the shoulder superelevation is decreased (the shoulder is "raised").

Namely, in the last case, if:

q	is the current pavement superelevation
dq	is the *Edge superelevation difference*

eu is the shoulder superelevation at the high side
then the following formula shall apply:
|q| +| eu| = |dq|

If you tick the *Superelevation on curve external* check box, enter the absolute shoulder superelevation value in the text field next to the box; the superelevation will be formed by segment, based also on the values in the *Shoulder tangent superelevation* fields. The shoulder superelevation on the tangent and on the inner side of the turns is the value entered in the *Shoulder tangent superelevation* field, while the shoulder superelevation on the outer side of the turn is the value entered in the *Superelevation on curve external* field.

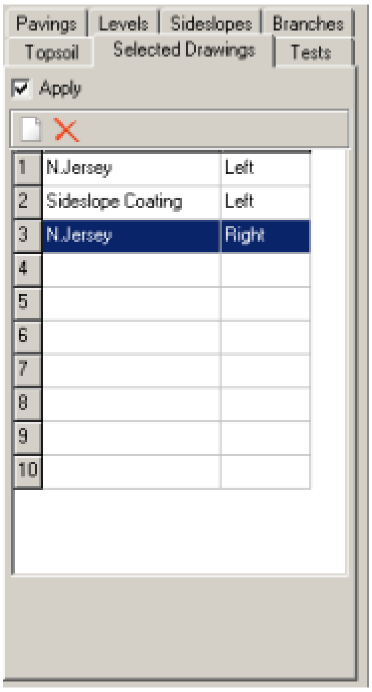
Calculation with Pavement Superelevation

This calculation is based on the value that you have entered in the *Shoulder tangent superelevation* field. The superelevation of the shoulders on the tangent is the value that the user enters in the relevant field. For the inner side of a turn, all information mentioned as regards the previous calculation applies here too. Namely, when the pavement superelevation exceeds the shoulder superelevation on tangent, then the shoulder superelevation becomes equal to the pavement superelevation. Regarding the outer side, however, when the pavement superelevation becomes different than the value that you have entered for the tangent, then the shoulder superelevation becomes equal to the pavement superelevation.

Setting the Section

To define the position by using the *Section and slope* method, you must select the segment according to which the section will be determined. To select the segment, proceed as you would do for selecting the segment for setting a slope. Use this method when a line must extend up to a user-defined segment. This method usually serves for the calculation of the paving width based on the sideslope of a previous layer (at a higher level) of the typical cross section.

11.16 Selected Drawings



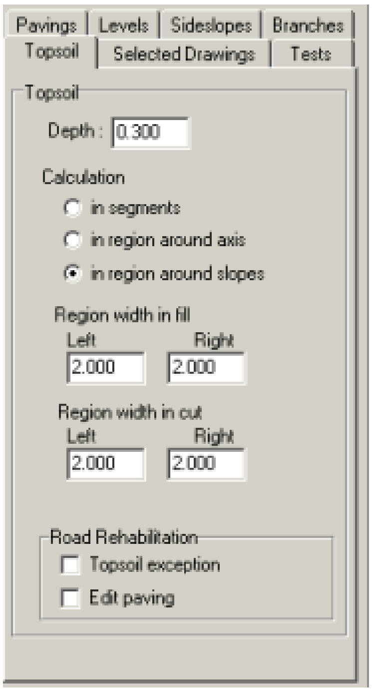
From the typical cross section workspace, you can insert certain additional standard drawings to specific positions, based on the main axis of the cross sections or based on two semiaxes for two-branch cross sections. To insert drawings to the axis, open the *Selected Drawings* tab of the *Typical Cross Section Data* window. This tab includes a list of all of the selected drawings that are applied to the current typical cross section, as well as two buttons for inserting and deleting drawings to/from the typical. To insert a drawing, click the  button, which opens a box, enabling you to select a drawing from the current drawings library. In addition, from the drop-down list at the top of the box, select the task to which the line will belong and exactly below select the position of the line with respect to the cross section's axis (center, right or left). Usually the drawing is inserted in the middle of the axis for the single-branch typical cross sections (N. Jersey), while in the case of two-branch typical cross sections, the drawing is inserted to the left or to the right of the semiaxes of the right and left branch, respectively.

To delete a selected drawing while working in the typical cross section workspace, select the drawing from the list of the *Selected Drawings* tap by clicking the left mouse button and then click the  button.

If you don't want to apply the selected drawings, but you wish to keep them, untick the *Apply* field.

11.17 Topsoil

From the *Topsoil* tab of the *Typical Cross Section Data* window, you can define the formation of the topsoil, the clearance of the top soil, and the way that the topsoil is calculated in a typical cross section.



First you must define the topsoil clearance depth, namely the distance of the topsoil line from the natural ground line. In the *Depth* field, enter the desired distance in meters. Next, you must define the natural ground area in which the topsoil will be calculated. You can leave this area free in the typical, in order to define it separately for every cross section of the project, or you can select its automatic calculation either for a fixed distance from the axis of every cross section or for the whole construction zone plus a distance from the sideslope edges.

- With the in *segments* option, the user defines and forms the topsoil separately for every cross section.
- With the in *region around axis* option, the topsoil clearance is determined automatically in a region above and below the axis, as this region is determined for every side from the *Region width in fill* or *Region width in cut* field, and according to the cross section type. This region is separate from the construction zone that will derive during the cross section calculation.
- With the in *region around slopes* option, the topsoil clearance is determined automatically in a region above and below the axis that extends beyond the construction zone. This region is determined for every side from the *Region width in fill* or *Region width in cut* field, and according to the cross section type. In this case, the values in the fields indicate the distances from the sideslope edges that demarcate the construction zone.

Finally, if the particular typical cross section is to be used for the rehabilitation of an existing road, select the *Topsoil exception* option from the *Road rehabilitation* group at the bottom of the tab. With this option, the topsoil will not be calculated for the part of the cross section that corresponds to an existing paving; it will be calculated only for the parts left and right to the paving. The existing road is indicated according to every cross section’s chainage by two diagrams *(Exist. Road diagram right and Exist. Road diagram left)*, which can be updated automatically from the horizontal alignment, provided that they are depicted there in the form of lines. Apart from the topsoil line in the region of the existing road, the program also enables you to exclude the paving part that coincides with the older paving of every cross section in the project, by simply selecting the *Edit paving* option from the *Road Rehabilitation* group.

Note that you can view immediately every change that you are making in the topsoil tab, since the topsoil is immediately updated in the drawing area, while you can also view the new settings in the typical cross section's workspace.

Chapter 12

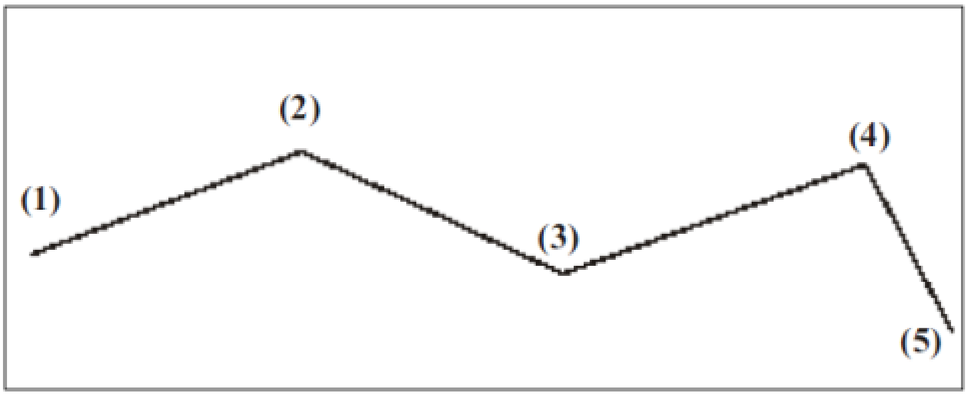
12 Drawing

12.1 Lines

The simplest object in *Anadelta Tessera* is the zigzag line, which is referred to as drawing line in the program. The drawing lines can be comprised of one or more straight segments and they can be closed or open. To draw these lines, you must define the points of their peaks.

Drawing a Line

A line is comprised of a series of connected straight segments, which can be managed as a single object.



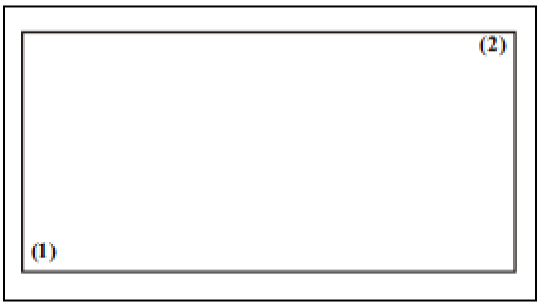
How to draw a drawing line

1. From the *Drawing* menu, run the *Line* command, or from the *Drawing* group of the *Drawing Elements* toolbar click the button. You will see the highlighted box in the middle of the mouse cross, showing you where the line point will be inserted.
2. Click the left mouse button to define the start point (1) of the line.
3. In the same way define the end point of every straight segment of the line (e.g. (2), (3), (4) and (5)).
4. When you have finished setting the peaks of the line, click the right mouse button to end the procedure.

After you have inserted a line, you can convert it into a breakline, a road tangent or even a boundary of an existing road.

Drawing a Quadrilateral

You can easily and quickly draw lines in the form of a rectangle.



How to draw a quadrilateral

1. From the *Drawing* menu, select the *New Quadrilateral* command.
2. Click the left mouse button to define the first angle (1) of the quadrilateral.
3. In the same way, define the opposite angle (2) of the quadrilateral. The insertion is completed after the second click and the quadrilateral remains selected.

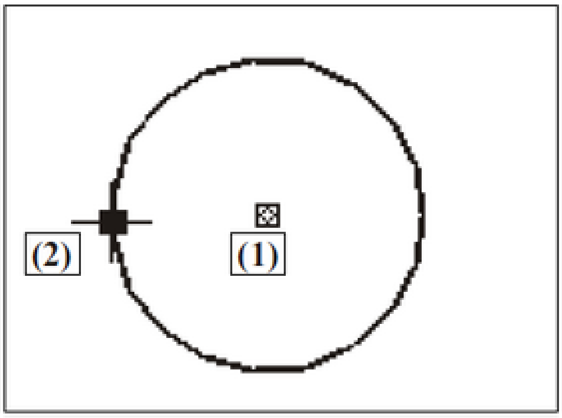
The resulting quadrilateral is a rectangle, whose sides are parallel to the X and Y axes.

12.2 Curves

The program enables you to draw curves, such as circles, ellipses, circular arcs or elliptical arcs.

Drawing a Circle

To draw a circle, you must define its radius and center.

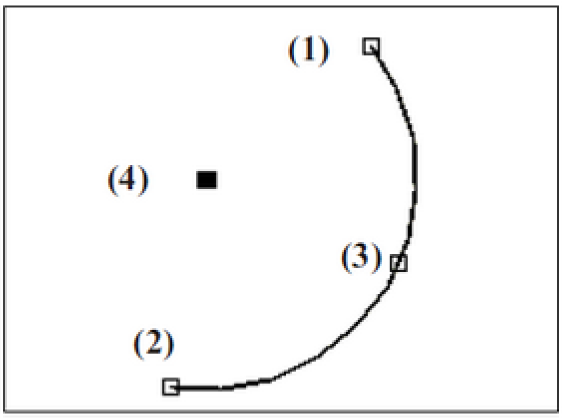


How to draw a circle

1. From the *Drawing* menu, select the *New Circle* submenu and then run the *Center, Radius* command; or, from the *Elements Drawing* toolbar, click the button.
2. Click the left mouse button to define the center point (1).
3. Click again the left mouse button to define a point for the circumference of the circle (2). The insertion is completed and the circle remains selected.

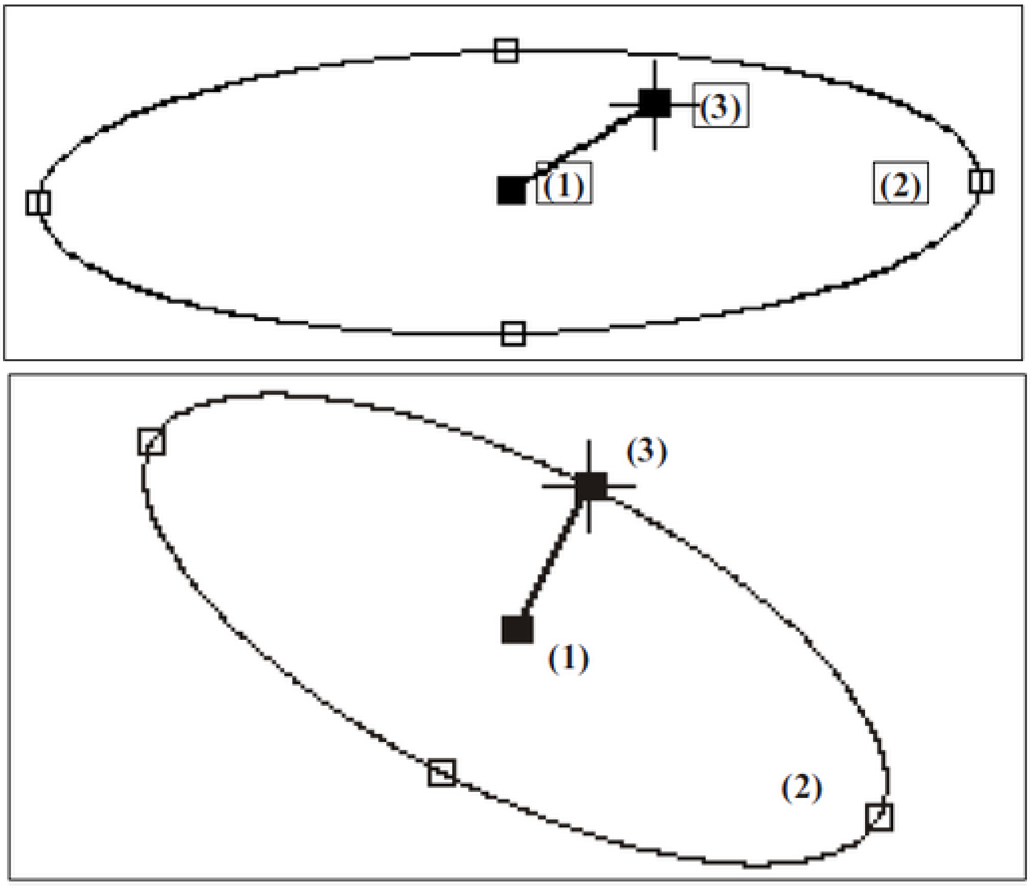
Drawing a Circular Arc

There are two ways to draw a circular arc. The main way is to define three points — the first point, the final point and the intermediate point of the circular arc. You can also define the center, the beginning and the end. By default, *Anadelta Tessera* draws the circulars arcs in a counterclockwise direction.



Drawing an Ellipse

You can create complete ellipses and elliptical arcs. The main method for drawing an ellipse is to define its center, the length of the primary and secondary axis, and finally the orientation of the axes. Both axes are vertical to each other. The order in which you define the axes is totally insignificant.



How to draw an ellipse (first way)

1. From the *Drawing* menu, select the *New Ellipse* submenu and run the *Center, Radii* command.
2. Click the mouse button to define the center (1) of the ellipse.
3. Set the length and the orientation of the first axis by defining a point (2) in the drawing area. The axis radius is equal to the distance of the point from the center of the ellipse.
4. Set the length of the second axis by defining a third point (3) in the drawing area. The axis radius is equal to the distance between the center and the point of the second axis.

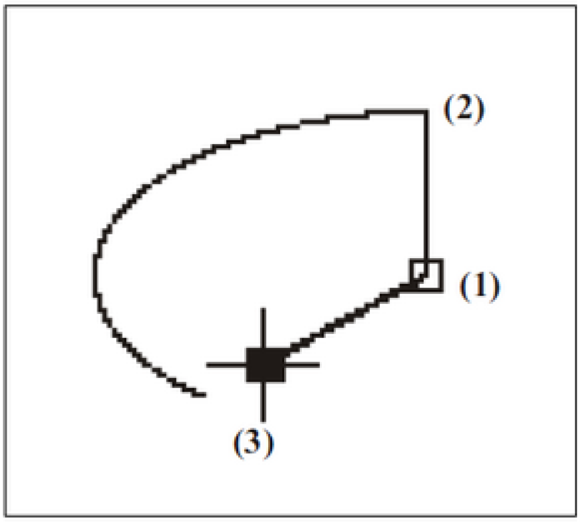
The elements of the ellipse can be defined in a different order.

How to draw an ellipse (second way)

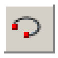
1. From the *Drawing* menu, select the *New Ellipse* submenu and then run the *Center, Radius, Angle* command; or, from the *Elements Drawing* toolbar, click the  button.
2. Click the mouse button to define the center (1) of the ellipse.
3. Set the length of the first axis by defining a point (2) in the drawing area. The axis radius is equal to the distance between the two points.
4. Set the length and the orientation of the second axis by defining a third point (3) in the drawing area. The radius and the orientation of the axis are equal to the distance and the orientation between the center and the point of the second axis.

Drawing an Elliptical Arc

To draw elliptical arcs, you must define all elements of the ellipse, by using the second way that is described in the previous paragraph. The arc starts at the point by which you defined the length and the orientation of the second axis. Next, you must define the end of the elliptical arc. The arcs are always drawn in a counterclockwise direction.



How to draw an elliptical arc

1. From the *Drawing* menu, select the *New Ellipse* submenu and then run the *Elliptical Arc,* command; or, from the Elements Drawing toolbar, click the  button.
2. Define the center (1) of the ellipse.
3. Set the length of the first axis by defining a point (2) in the drawing area. The axis radius is equal to the distance between the two points.
4. Set the length and the orientation of the second axis by defining a third point (2) in the drawing area.
5. The radius and the orientation of the axis are equal to the distance and the orientation between the center and the point of the second axis, similarly to point (3) of the second way that was described in the previous paragraph.
6. Set the end of the arc by defining a point (3).

12.3 Clothoids

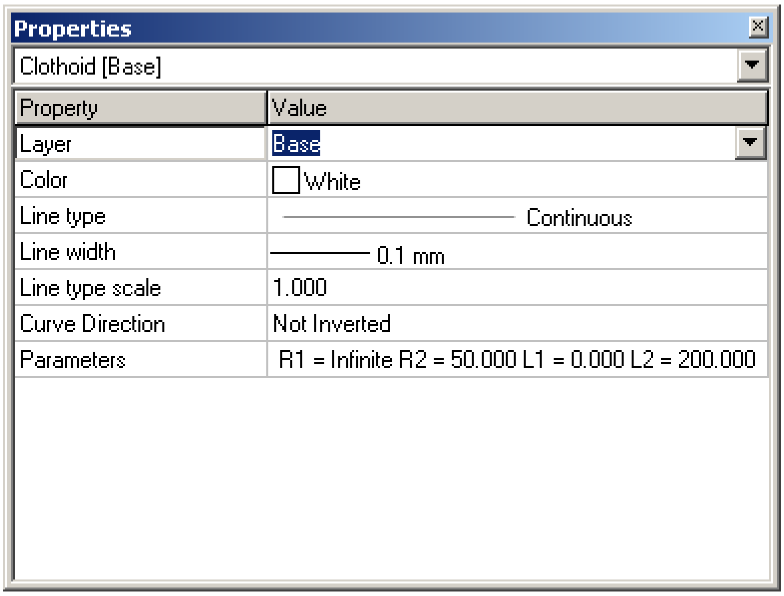
Drawing a Clothoid Curve

Anadelta Tessera offers the unique feature of drawing clothoid curves.

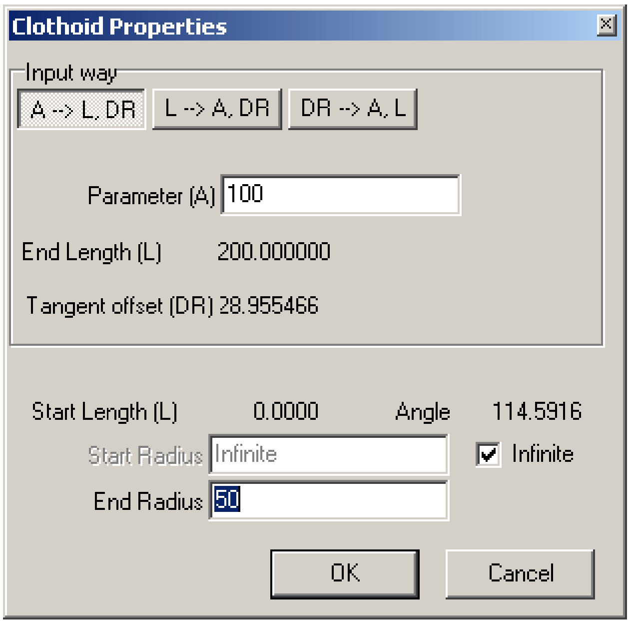
To insert a clothoid, run the *New Clothoid* command of the *Drawing* menu and define the main tangent of the clothoid in the same way that you define a straight segment. The clothoid is inserted with the second mouse click and remains selected. When the length of the clothoid is zero, its tangent appears on the screen; this tangent also serves as a selection line (handle). Next, define the data – parameters of the clothoid.

How to input the clothoid's data

1. Open the *Properties* command window.
2. Select the clothoid you wish to edit.
3. In the *Properties* command window, select the value of the *Parameters* property. Click the three-dot button that is displayed when this field is selected.

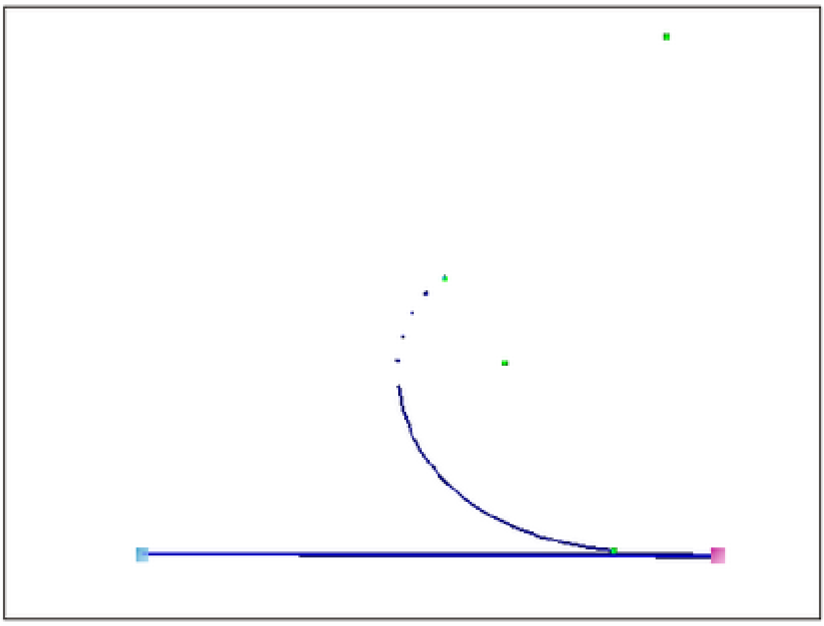


The following dialogue box appears on screen:



1. In the three fields of the *Input way* group, you can set the parameter A of the curve, the length L of the clothoid, as well as the offset DR. According to literature, these three values are interrelated with the $\varepsilon = \frac{L^2}{24R}$ and $L = \frac{A^2}{R}$ formulas. Depending on the selected calculation way for these values, when you click the respective button at the top of the group, only one of the three text fields is active, while the other two fields show the data values that the program calculates automatically.
2. In the *Start Radius* and *End Radius* fields, enter the radii of the circular arcs at the start and the end of the clothoid. If you wish the clothoid to start from the tangent, tick the *Infinite* check box. The *Start Length (L)* field shows the clothoid length that the program has calculated from the tangent up to the start radius (if the latter is not infinite). Namely, this value corresponds to the length of the initial segment that is not drawn on the screen. Finally, the *Angle* field shows the calculated value of the DR offset angle.
3. Click the *OK* button to close the dialogue box.

The clothoid curves can be edited too, like any other drawing object. Also, every time you insert a new drawing element, the characteristic parts of the clothoid appear in green (start and end curvature centers, start and end point of the clothoid) so that you can apply the snap feature to them.



Drawinga Parabola

Anadelta Tessera enables you to draw various types of parabolas.

To insert a parabola, run the *New Parabola* command of the *Drawing* menu and then draw the main tangent of the parabola in the same way that you define a straight segment. The parabola, whose length is zero, is inserted with the second mouse click and remains selected. When the length of the parabola is zero, its tangent is drawn on the screen; this tangent also serves as the parabola’s selection line (handle). Next, define the data – parameters of the parabola in the same way that you define the clothoid data. The additional feature that characterizes the parabola is selected from the *Curve Type* field.

12.4 Texts

The text enriches a drawing by providing important information. You can use texts to give information about the nature of the design, e.g. title, surveyor, project owner and supervising service, as well as explanations in the study.

Creating a Text

To create one or more text lines, use the *New Text* command. Every text line is a separate object that can be moved, centered and edited as desired.

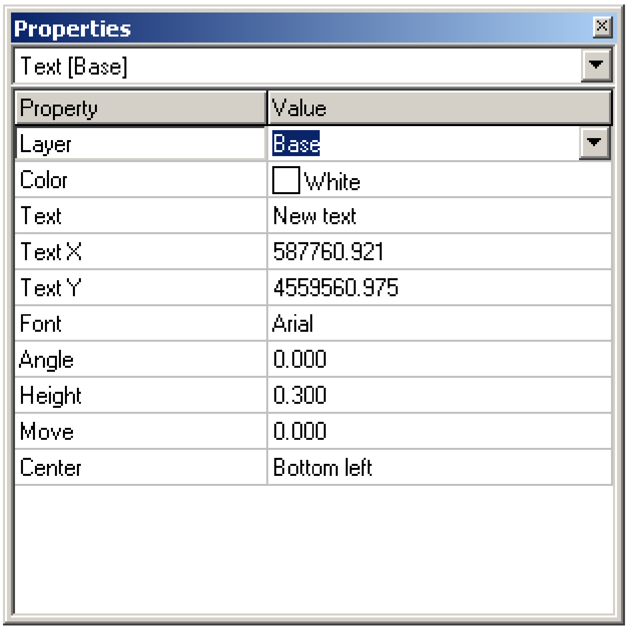
How to create a text

1. From the *Drawing* menu, run the *New Text* command or, in the *Elements Drawing* toolbar, click the button.
2. Use the mouse to select where the first character (1) will be inserted.

A new text object, entitled *New Text*, is inserted in the project.

Text Editing

Texts are inserted with default properties. To give them the form and the content of your choice, edit them in the window of the *Properties* command.



Properties

Text [Base]

Property	Value
Layer	Base
Color	White
Text	New text
Text X	587760.921
Text Y	4559560.975
Font	Arial
Angle	0.000
Height	0.300
Move	0.000
Center	Bottom left

Text Typing

In the field of the *Text* property in the *Properties* window, type the desired content of the text and press *Enter*.

Text Font and Font Height

The font height of the new texts is 0.3 cm on paper and the font type is *Arial*. The font size on screen is similar to the font size on the printout, according to the print scale.

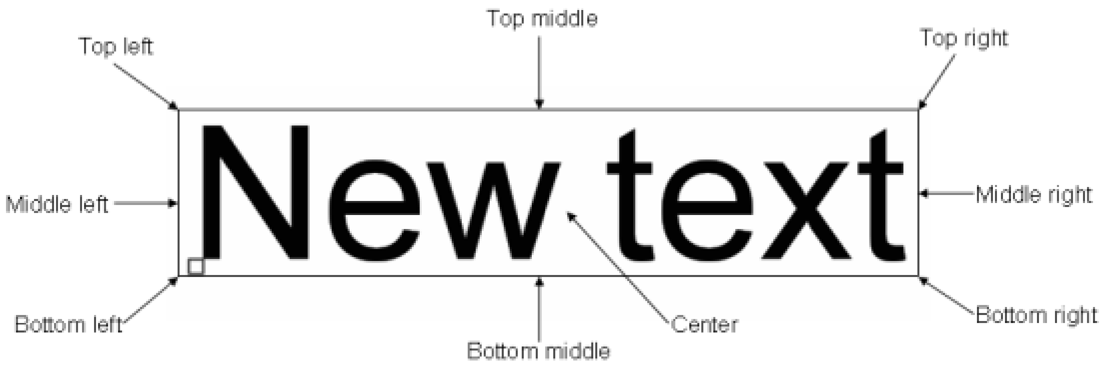
To change the text font, select the text you wish to edit and open the window of the *Properties* command. In the field of the *Font* property, click the three-dot button . In this box, you can choose a new font or whether the characters will appear bold or italic.

To change the font height, open the window of the *Properties* command and in the *Height* field type in centimeters the new height.

Text Alignment

For every text, the initial point of alignment is the bottom left, the distance from this point is equal to zero and the rotation angle is also zero.

Every text has a center point. A center point is the point of the box that determines the place of the text on the screen. There are nine available places in the box that can be defined as center points.



To change the center point, open the window of the *Properties* command and select a value from the drop-down list of the *Center* property.

12.5 Images

To insert one or more images, use the *Image* command.

How to insert an image

1. From the *Drawing* menu, run the *New Image* command or, from the *Elements Drawing* toolbar, click the button.
2. Define the point at which the bottom left corner of the image will be inserted. An image with the *Anadelta Software* logo is inserted into the project.
3. Next, open the *Properties* command window.
4. In the field of the Image property, click the three-dot button which is located to the right of the field. A dialogue box appears on the screen. Select the image file that will be inserted into the drawing.

12.6 Object Properties

You can specify the main properties of the new drawing objects before creating the objects. The values of the properties that will be applied to the new objects are referred to as current values.

Defining the Color for the New Objects

You can set the value of the *Color* property for the new drawing objects.

How to set the current Color

1. Deselect all objects.
2. From the *Object Properties* toolbar or from the window of the *Properties* command, select one of the colors available in the *Color* drop-down list or select *Other…* to define a new color.

Defining the Line Type for the New Objects

You can set the value of the *Line type* property for the new drawing objects.

How to define the current type of line

1. Deselect all objects.
2. From the *Object Properties* toolbar or from the window of the *Properties* command, select one of the line types available in the *Line type* drop-down list.

Defining the Line Type Scale for the New Objects

You can set the value of the *Line type scale* property for the new drawing objects.

How to define the current Line type Scale

1. Deselect all objects.
2. Open the window of the *Properties* command and type the desired value in the field of the *Line Type Scale* property.

Defining the Line Width for the New Objects

You can set the value of the *Line width* property for the objects that will be created.

How to define the current Line width

1. Deselect all objects.
2. From the *Object Properties* toolbar or from the window of the *Properties* command, select one of the line widths available in the *Line width* drop-down list.

12.7 Layers

With the help of the layers, you can further categorize the drawing objects. The layers are groups in which you can organize and group various drawing objects. This grouping facilitates the management of the *Horizontal Alignment* data. For every group, you must set certain values for the main properties. The main properties of a drawing object can be set either based on the object's layer, or they can be different from the values of the layer's properties.

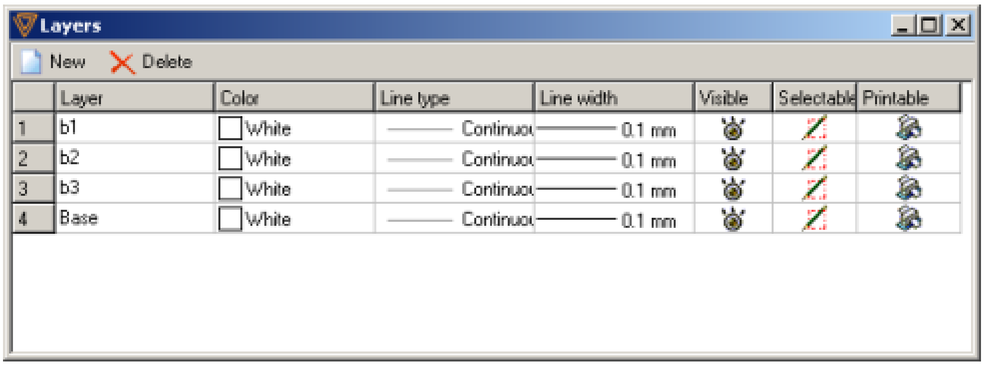
Working with Layers

The drawing objects always belong to a layer. This can be the base layer or a user-created layer. In every layer there are specific values for the *Color,* the *Line type* and the *Line width* properties. You can use the layers to organize your drawing in groups of objects, in order to manage and edit the drawing objects easily and quickly.

For example, you can create a layer that shall include the outlines of the buildings. The desired properties for the drawing objects (color, line type and line thickness) can be assigned to the layer instead of being assigned separately to every object. To draw a new outline, you must make the layer of these lines current and then start drawing. You don't have to define again the line type, the line color and the line width every time you draw an outline.

Every time a new design begins, *Anadelta Tessera* creates automatically a layer, which is entitled Base. By default, the property values of the *Base* layer are *Black* for the *Color, Continuous* for the *Line type* and 0.1 mm for the *Line width*. You cannot delete or rename the base layer.

To manage and edit the layers, open the *Layers* window.



How to open and close the Layers management window

1. From the *Drawing* menu, run the *Layers* command, or press the *F9* key, or click the button in the toolbar. The management window appears on the screen.
2. Proceed to the desired changes.
3. When you have finished, click the exit button to close the management window and return to the main screen.

Creating and Renaming Layers

You can create and rename a layer for every set of lines, and also assign the common properties of the lines to this layer. By grouping the objects in layers, you can easily control their appearance and make changes quickly and efficiently.

How to create a new layer

1. Open the Layers management window.
2. Click the *New Layer* New button at the top of the window.

This creates a new layer with a unique name, e.g. *New Layer, New Layer 1, New Layer 2,* etc. After you have created a layer, you can rename it. The property values of the new layers are *Black* for the *Color*, *Continuous* for the *Line type* and *0.1 mm* for the *Line width*, but you can change them, if desired.

Changing the Name of a Layer

You can change the name of a layer at any given time, in order to show clearly what the layer represents in the drawing. You cannot, however, change the name of the *Base* layer.

How to rename a layer

1. Open the *Layers* management window.
2. Left click on the layer's name field to activate it.
3. Type the new name and press *Enter*.
4. Click the exit button to close the management window and return to the main screen.

How to make a layer current

You always draw in the current layer. When a layer is current, you can create new objects in this layer and these new objects may have the same properties as the layer (provided that the object's properties have the *Layer* option enabled). When the current layer is non visible or non selectable, you cannot create new drawing objects.

How to make a layer current

1. Deselect all objects.
2. From the *Object Properties* toolbar or from the window of the *Properties* command, select one of the layers available in the *Layer* drop-down list.

Checking the Layers

Anadelta Tessera will not display the objects that are drawn in non visible layers. Also, the program will not print the objects that are drawn in non printable layers. If you want to hide temporarily some of the drawing objects while working on details of your drawing, you can make a layer or a set of layers non visible. The layers containing elements that don't have to be printed, such as auxiliary lines, can also be made non visible.

Checking the Layers' Visibility

The non-visible layers will not be displayed on the screen, but you can still print them.

How to make a layer visible or non-visible

1. Open the *Layers* management window.
2. Select the fields of the layers that you want to make visible or non visible.
3. Double-click on the *Visible* field.
4. When you have finished, click the exit button to close the management window

and return to the main screen.

Checking the Layers' Printability

You can select which layers will be printed. If the layer contains information that doesn't have to appear on the final drawing (for example, auxiliary lines), you can select not to print this information. In this case, the layer's objects are shown on the screen, but they will not be printed. Therefore, you don't have to erase the data before printing.

How to make a layer Printable/Non printable

1. Open the *Layers* management window.
2. Select the fields of the layers that you want to make printable or non printable.
3. Double-click on the *Printable* field.
4. When you have finished, click the exit button to close the management window and return to the main screen.

Checking the Layers' Selectability

You may often want to edit objects in specific layers, while viewing the objects of other layers that cannot be edited. You cannot edit or select the objects of a non-selectable layer, but the objects are visible when the layer is visible.

It is possible to use the snap feature on objects of a non-selectable layer. You can make a layer non selectable, non visible and non printable. You can also change the color, the line width and the other main properties.

How to make a layer Selectable/Non selectable

1. Open the *Layers* management window.
2. Select the fields of the layers that you want to make selectable or non selectable.
3. Double-click on the *Selectable* field.
4. When you have finished, click the exit button to close the management window and return to the main screen.

Assigning Color to a Layer

You can assign a color to a layer from the management window of the Layers command.

How to assign a color to a layer

1. Open the *Layers* management window.
2. Select a layer and click on the respective color field.
3. Select one of the colors available in the drop-down list of the *Color* property, or select *Other …* to define a new color.
4. When you have finished, click the exit button to close the management window and return to the main screen.

Assigning the Line Type to a Layer

Every line type presents the visual information in a different way. A line type can be a continuous line or a repeated pattern of dashes, dots and spaces, enabling you to distinguish the role of a line. The name and the definition of the line type describe the specific sequence of characters, the relevant sizes and their characteristics.

How to assign a line type to a layer

1. Open the *Layers* management window.
2. Select a layer and click on the respective field of the Line type property.
3. From the drop-down list of the *Line type* property, select one of the available line types.
4. When you have finished, click the exit button to close the management window and return to the main screen.

Assigning the Line Width to a Layer

The line width adds thickness to the objects that are drawn by using a specific line. All drawing objects can be viewed thickened and with texts both on the screen and on the printout. By defining the line width for a layer or an object, you can simulate the object's width on the printout. By changing the width of objects and layers, you can enhance the presentation quality of your drawings.

How to assign a line width to a layer

1. Open the *Layers* management window.
2. Select a layer and click on the respective field of the *Line width* property.
3. From the drop-down list of the *Line width* property, select one of the available widths.
4. When you have finished, click the exit button to close the management window and return to the main screen.

Deleting a Layer

You can delete any layer, at any given time when processing a drawing. You cannot, however, delete the *Base* layer. When a layer is deleted, its objects are transferred by default to the base layer.

How to delete a layer

1. Open the *Layers* management window.
2. Select the desired layer and click the Delete button.
3. *Anadelta Tessera* will prompt you to confirm that you wish to delete the layer, by showing you the relevant dialogue box.
4. When you have finished, click the exit button to close the management window and return to the main screen.

Using the Layers' Properties

When you draw by using a layer, it is recommended that the default properties are those of the layer. The objects whose main properties have the *Layer* value acquire the properties of their layer. The same can apply for every new object that will be created. When designing in this way, you are able to maintain a well-organized drawing.

If you want the color, the line type or the line width of a particular object to be different than those of the layer, you can change the properties of this object. The properties of an object can have either a fixed value (e.g. red for color) or the respective value of the layer. In the first case, the object's value prevails over the layer's value.

12.8 Object Import

Importing Drawing Objects from DXF files

If your drawing objects are available in digital form (DXF) and you wish to import them into your design, you can do this anytime, either during the first stage of your design or while editing your design, and for as many times as you wish.

You can import any drawing object (lines, circular arc, texts, etc.) from a DXF file. You can also do this with the simultaneous import of terrain model elements. From a DXF file you can import objects such as line, polyline, circle, arc, ellipse, elliptical arc. Regarding their main properties, the imported objects maintain their original layer, which will be transferred to another layer.

How to import drawing objects from DXF files

1. From the *File* menu, select *Import from Dxf* and then run the *Import Drawing from DXF* command.
2. From the dialogue box of the *Open* window, select the DXF file and click *Open*. After the program has read the file, select, if necessary, *zoom extents*.

Importing Drawing Objects from ASCII files

By using the Import *Drawing* command, which is the first command of the *Import from ASCII* submenu, you can import drawing lines from text files.

How to import terrain points from a text file

1. From the *File* menu, select the *Import from ASCII* submenu and then run the *Drawing Data* command. This command will be displayed in the menu only if you first select the creation of a new file or open an existing one. From the dialogue box that appears on the screen, select a terrain point file (such files have usually the .RLN extension).
2. From the *Browse* field, find the subfolder where your file is located. Select the desired file and click *Open*.
3. Confirm by clicking OK. The program reads the lines and imports them to the file. If a problem occurs during the file transfer, the program will display the relevant error message on the screen.

12.9 Object Editing

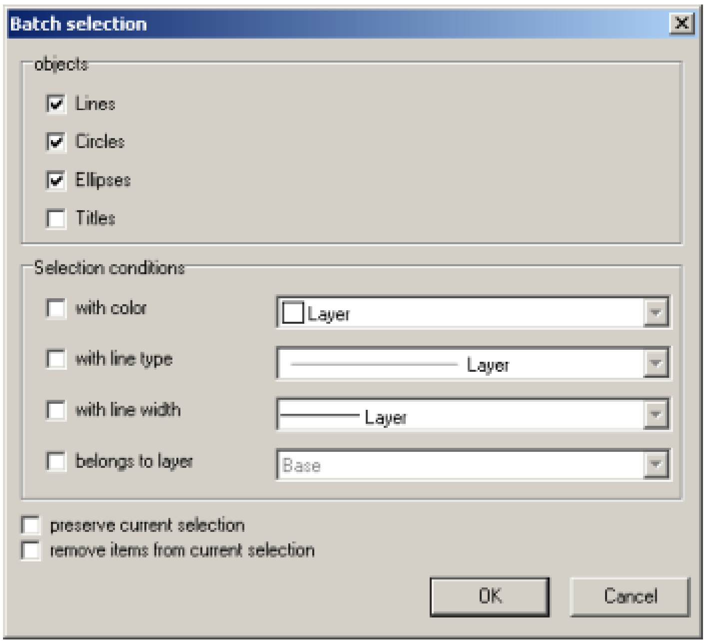
There is only one way to edit objects in the *Anadelta Tessera* environment. First select the objects and then edit them. In this section, you will learn how to select objects, how to view the selected objects on the screen, how to change their main properties, and how to carry out general and special editing operations.

Selection of Objects

For the lines and curves, the aforementioned data shall apply. For the drawing objects, there are two additional methods of selection.

<u>Filtering All Selected Objects</u>

With the *Batch Selection* command,



you can select a set based on the common properties and the category of the drawing objects. For example, you can select simultaneously all the red drawing lines of the *Horizontal Alignment*, without selecting any other object, or, you can select all objects except for these lines.

To create a set of selected objects based on the value of the *Color, Line type* or/and *Line width* properties, you should first see whether the property has the Layer value. For example, an object can be black, either because its value in the *Color* property is *Black* or because it has the *Layer* value and the value in the layer's *Color* property has been set to *Black*.

How to create a set by using the Batch Selection command

1. From the *Edit* menu, select the *Batch Selection* command or press the *Ctrl+A* keys.
2. In the *Objects* group of the *Batch Selection* window, tick the respective check box (es) to the left of every category of drawing objects that you wish to select. Note that there are four categories of drawing objects: the lines (*Lines* check box), the circles that also include the circular arcs (*Circles* check box), the ellipses that also include the elliptical arcs (*Ellipses* check box) and the texts (*Titles* check box).
3. In the *Selection conditions* group, define the selection criteria for sorting out the objects. This group includes all of the identifiable properties of the drawing objects.

To enable a selection condition, tick the relevant check box on its left.
4. Enter the property value by selecting it from the drop-down list to the right of the ticked box.
5. Click *OK*.

Anadelta Tessera selects all drawing objects that match the user-selected criteria and closes the dialogue box of the Batch Selection command.

By default, *Anadelta Tessera* creates a new set of selection. To add this set to the already selected objects, tick the preserve current selection check box. To remove this set from the already selected objects, tick the *remove items from current selection* check box.

Select All

With the *Select all* command of the *Edit* menu, you can easily select all visible objects of the enabled modes.

To select all objects of an enabled mode

From the *Edit* menu, run the *Select all* command or press the *Shift+Ctrl+A* keys.

Editing the Main Properties

In *Anadelta Tessera* you can edit the main properties of the drawing objects in two ways: either from the *Object Properties* toolbar or from the window of the *Properties* command.

Copy Objects

You can copy one or more drawing objects in your current file or from one file to another.

Copy Objects

With the *Copy* command, the objects are copied to *Clipboard*.

How to copy objects to Clipboard

1. Select the objects you wish to copy.
2. From the *Edit* menu, select the *Copy* command or press the *Ctrl+C* keys, or click the copy icon.

The objects are copied to Clipboard. The reference point of the objects is the intersection point of the diagonals of the fictitious box that contains the selected object. If you want to define manually the reference point for the set of selected objects, run the *Copy with Basepoint* command.

How to copy objects to Clipboard, by selecting the reference point

1. Select the objects you wish to copy.
2. From the *Edit* menu, select the *Copy with Basepoint* command.
3. Set the reference point.

Cut Objects

With the *Cut* command, the drawing objects are deleted from the current file and transferred to *Clipboard*.

How to transfer objects to Clipboard

1. Select the objects you wish to cut.
2. From the *Edit* menu, select the *Cut* command or press the *Ctrl+X* keys, or click the cut icon.

Paste Objects

The objects that you copied or transferred to Clipboard can be imported again to the same or to another file.

How to import objects from the Clipboard

From the *Edit* menu, select the *Paste* command or press the *Ctrl+V* keys, or click the paste icon.

The objects in the Clipboard will be imported to the drawing.

Moving Objects

When you move objects, the objects are moved as they are, without any changes regarding the orientation or their relative positions with respect to each other. To ensure the necessary accuracy, use the *Input Form* command or the snap feature.

How to move one or more objects simultaneously

1. Select the drawing objects you wish to move.
2. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Transformations* submenu and then run the *Move* command.
3. Set the reference point.
4. Set the end point.

Moving Drawing Objects with the Mouse

You can move certain categories of drawing objects by using the points. To do this, activate the center of a circle or a circular arc, of an ellipse or an elliptical arc, or the insertion point of a text character.

How to move an object with the mouse

1. Select the drawing object you wish to move.
2. Activate the appropriate point. For texts and images, simply left click on them again.
3. Specify the new position of the point.

Moving Drawing Objects by Using the Coordinates

From the window of the *Properties* command, you can move categories of certain drawing objects, such as circles or circular arcs, ellipses or elliptical arcs, images and texts. Simply select the desired object and open the window of the *Properties* command. In the fields of the *Center X* and *Center Y* properties for the curved objects, and *Text X* and *Text Y* for texts and images, type the new coordinates of the center point. The drawing objects will be offset, so that their centers are transferred to the position indicated by the new coordinates.

Especially for texts and images, you can change the value of the *Move* property, in order to move the letters or the image without offsetting the center point. The offset is equal to the value that the user entered in the field, both for the X and for the Y axis.

Object Rotation

To rotate the drawing objects, select a reference point and a rotation angle towards the horizontal direction. The objects always rotate in a counterclockwise direction.

How to rotate an object

1. Select the drawing objects you wish to rotate.
2. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Transformations* submenu and then run the *Rotate* command.
3. Set the reference point.
4. Set the rotation angle either by specifying a point on the screen, or by entering the angle with the use of the *Input Form* command.

Rotation of Texts, Ellipses or Images

From the window of the *Properties* command, you can rotate texts, ellipses or images. Simply select the text or the ellipse that you wish to rotate and open the window of the *Properties* command. In the *Text Angle* or *Angle* field, type the horizontal rotation angle (the rotation direction is counterclockwise and the angle is entered in degrees). The text or the ellipse or the image will rotate around its center.

Changing the Size and Shape of Objects

You can scale up or down a set of selected objects, while preserving their proportions with respect to each other. To scale the objects up or down, specify a reference point and a length, which will be used as the scale coefficient, or enter the value of the coefficient.

How to change the size of the selected objects

1. Select the drawing objects you wish to scale up or down.
2. From the shortcut menu that is displayed if you right-click in the drawing area, open the *Transformations* submenu and run the *Scale* command.
3. Set the reference point.
4. Set the scaling percentage either by specifying a point on the screen or by typing the percentage value (e.g. “200” for double size) with the use of the *Input Form*. When you define a point on the screen, Anadelta Tessera calculates the distance between the reference point and the user-defined point; the scaling percentage is the one tenth (1/10) of this distance.

Obviously, to scale up an object, you must enter a percentage higher than 100, while to scale down an object you must enter a percentage lower than 100.

Changing the Size of Circles

To make a circle bigger or smaller, activate a point on its circumference and specify a new position for this point. When you move this point, the circle radius is increased or decreased accordingly, but the center of the circle remains fixed. You can do the same from the dialogue box of the *Properties* command. Simply select the circle and open the window. Type the new value in the field of the *Radius* command.

Changing the Shape of an Ellipse

To change the lengths of an ellipse’s axis, activate a point on the circumference of the ellipse and specify a new position for this point. When you move this point, the ellipse radii are increased or decreased accordingly, but the center of the ellipse remains fixed. You can do the same from the dialogue box of the *Properties* command. Simply select the ellipse and open the window. Type the new values in the fields of the *X Radius* and *Y Radius* properties.

<u>Changing the Shape of Circular and Elliptical Arcs</u>

To change the shape of an arc, activate a point on the circumference of the arc and specify a new position for this point. When you move this point, the arc radius is increased or decreased, and the arc center also changes accordingly. You can do the same for the circular arcs from the dialogue box of the *Properties* command. Simply select the circular arc and open the *Properties* window. Type the new value in the field of the *Radius* command. The arc will be drawn again, so that its center point remains fixed and the positions of the three points of the arc's circumference match the new radius value.

Deleting Objects

You can delete all of the selected objects.

To delete all the selected objects

1. Select the drawing objects you wish to delete.
2. From the *Edit* menu, select the *Delete* command, or press the *Del* key, or click the  icon.

You can do the same by selecting the *Delete objects* command from the shortcut menu that appears when you right-click in the drawing area.

How to delete all drawing points

From the *File* menu, select the *Delete* submenu and then run the *Drawing Data* command.

12.10 Editing Lines

The program offers some additional editing features regarding the position and shape of the drawing lines.

To edit a drawing line

1. Select the drawing line you wish to edit.
2. From the shortcut menu that is displayed when you right-click in the drawing area, select the desired editing command.

Mirroring Drawing Lines

You can mirror the drawing lines, either by the horizontal or by the vertical axis, by specifying an axis point. The original object will be deleted.

How to mirror drawing lines

1. Select the drawing lines you wish to mirror.
2. From the shortcut menu that displayed if you right-click in the drawing area, select the *Transformations* submenu and run the *Mirror (X axis)* or *Mirror (Y axis)* command, depending on the desired direction.
3. Specify the point through which the horizontal or the vertical mirroring axis will pass.

Breaking Drawing Lines

You can break a drawing line into one or more objects. The line can be broken only at certain of its peaks – points.

<u>Breaking Drawing Lines in Straight Segments</u>

You can break a drawing line in smaller lines; each of these lines will have only two points.

How to break drawing lines into straight segments

1. Select the drawing lines you wish to break.
2. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Tools* submenu and then run the *Break in segments* command.

<u>Breaking a Drawing Line in Two Lines</u>

You can break a drawing line in two individual lines based on a reference point.

How to break a drawing line in two

1. Select the drawing line you wish to break.
2. Select the appropriate point.
3. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Tools* submenu and then run the *Break at point* command.

The line will be broken in two segments. The selected point will become the start of one line and the end of the other line.

Editing Drawing Line Points

You can delete or move the points of a line. You can also add points to a drawing line.

Deleting a Drawing Line Point

You can delete one or more points of a drawing line.

How to delete a drawing line point

1. Select the drawing line you wish to edit.
2. Select the point you wish to delete.
3. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Delete Point* command.

Offsetting a Drawing Line Point

You can offset the points of a drawing line, in order to give the line the desired shape.

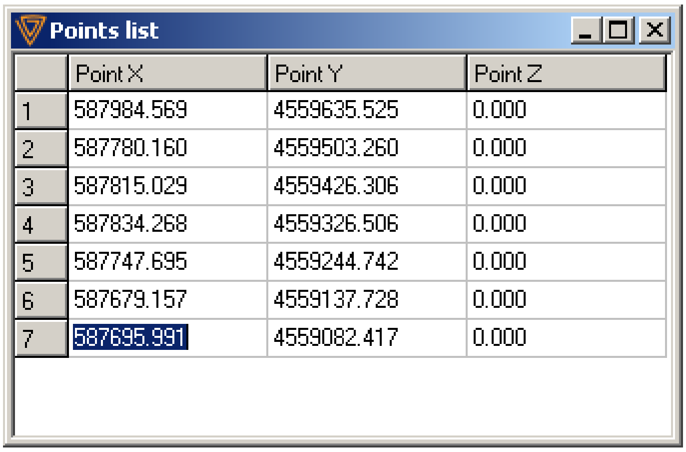
How to offset a drawing line point

1. Select the drawing line you wish to edit.
2. Activate the point you wish to move.
3. Specify the new position of the point.

Batch Move of Drawing Line Points

From the *Points* property of the *Properties* command, you can change the points' positions simultaneously.

How to move simultaneously the points of a drawing line


Points list

	Point X	Point Y	Point Z
1	587984.569	4559635.525	0.000
2	587780.160	4559503.260	0.000
3	587815.029	4559426.306	0.000
4	587834.268	4559326.506	0.000
5	587747.695	4559244.742	0.000
6	587679.157	4559137.728	0.000
7	587695.991	4559082.417	0.000

1. Select the drawing line you wish to edit.
2. From the shortcut menu that appears if you right-click in the drawing area, open the window of the *Properties* command. In the field of the *Points* property, click the three-dot button, which is located to the right of the field. The window of the *Points List* command appears on the screen. This window has two columns, which correspond to the X and Y coordinates. Every row corresponds to a line point.
3. Change the values in the desired fields by typing the new coordinates.
4. Click *OK*.

Inserting a Drawing Line Point

You can insert one or more points to a drawing line.

How to insert points to a drawing line

1. Select the drawing line you wish to edit.
2. Select the point before or after the point you wish to insert.
3. From the shortcut menu that is displayed if you right-click in the drawing area, select *Extend towards start* or *Extend towards end*, depending on which direction you wish to insert the new point. The order of the points is the same as the order in which they were inserted.
4. Specify the position of the new point. Continue with the next point. To finish the point insertion procedure, click the right mouse button or press the *Esc* key.

Converting Drawing Lines

You can convert a drawing line into a breakline, a road tangent or a diagram.

How to convert a drawing line

1. Select the drawing line you wish to convert.
2. From the shortcut menu that is displayed if you right-click in the drawing area, select the *Convert* submenu and then select the desired conversion, i.e. breakline, road, existing road diagram or generic use diagram.

Please keep in mind that in order for a line to be converted to a breakline, all points of the line must coincide with terrain points, while in order for a line to be converted to a diagram, there should be at least one road in your project. In the case of more than one road, the program will display a dialogue box and you must select the road for which the diagram will be created.

12.11 Data Export

Exporting Drawing Data to a Text File

You can create a file that will contain the drawing lines of your project.

How to export the drawing objects to an ASCII file

1. From the *File* menu, select the *Export to ASCII* submenu and run the *Drawing Data* command.
2. Specify the path and the name for the file you wish to save. The terrain points will be exported to a file with the .RLN extension.
3. Click *OK* to create the file.