RoadEng®
Civil Training

Version 5.1

Softree Technical Systems Inc.

Document Version - September 1, 2010

Trade Marks

AutoCAD is a registered trademark of Autodesk.
Criterion and LaserSoft are registered trademarks of Laser Technology, Inc.
Hewlett-Packard, HP and LaserJet are trademarks of Hewlett-Packard Company
Microsoft Windows 2000, NT, XP, Vista, Microsoft Word, and Microsoft Excel are trademarks of Microsoft Corporation
Terrain Tools® and RoadEng® are registered trademarks of Softree Technical Systems Inc.

1191 West 3rd Street
North Vancouver BC
Canada V7P 0A1

Table of Contents

1. Getting Started

This manual is formatted as a hands-on tutorial, which can be used by novice or experienced users. Step by step examples use prepared documents and data files to illustrate tools needed for common RoadEng® tasks. The document is laid out as if you were doing a road design project from original ground survey to completed construction documents.

Files

The tutorial files referred to in the examples are installed to your desktop and the *Layouts and Settings* folder by the special installation provided with this manual. You can install this from the training CD provided.

If you wish to use this manual with another version of RoadEng, example files can be found on the training CD (*Examples* folder) and should be copied to your *desktop*. In addition, there are files that need to be copied to your *Layouts and Settings* folder. You can find the RoadEng® *Layouts and Settings* folder on your hard drive by using the *Module* | *Setup* menu, *Install* tab. All these files have the word *training* in the file name to distinguish them from any production counterparts.

Note: If you do *not* run the special training installation, you need to copy the training CD contents to your hard drive.

[CD drive]\Examples → <hard drive Desktop>\Examples
[CD drive]\Examples\RoadEng → <hard drive Layouts and Settings>

Most of the following examples end with the phrase: "… do not save changes". If you modify the tutorial files, they will no longer work with the steps in the exercise; this will prevent you, or someone else, coming back and doing the exercise again. In the event that the files do get modified, you can always delete and re-copy from the CD.

Demonstration Mode

RoadEng® requires a password or hardware key (dongle) for each licensed copy. *Demonstration Mode* allows previewing of functions before purchasing. In *Demonstration Mode* printing and file saving are disabled. RoadEng® reverts to *Demonstration Mode* whenever non-permitted function groups are enabled; in this event, a warning will appear on your screen.

Note: Many of the examples in this manual can be successfully completed in *Demonstration Mode*.

Function Groups

Some RoadEng® (and Terrain Tools®) products have certain features disabled (*RoadEng Civil Lite* for example); we classify these optional features by *function group*.

If you don't have a password or key, then only a few Terrain Module *function groups* are permitted without entering *Demonstration Mode*. To view or change the enabled function groups:

1. Select *Module* | *Setup* from the menu bar; an Options Dialog box appears. Click on the *General* tab.

2. Click on the *Menus* button to open the Menu Customization Dialog box.

Function group items with a red circle before them are permitted in *Demonstration Mode* only. A checkmark beside an item indicates it has been enabled.

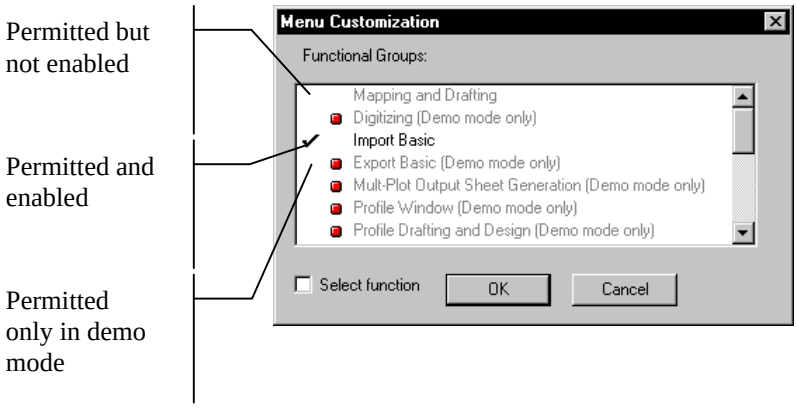


Figure 1.1: Function Groups Displayed in the Menu Customization Dialog

Function Groups Required for Examples
All required function groups are listed prior to each example in this manual. If you do not have permission to use all the required function groups, you may wish to skip the example. Also note that some function groups may be disabled even if *you* have permission to use them – this is so users with a lesser license can still do the example.

If you attempt to open a tutorial file containing function groups that are not permitted in your licensed software you will be prompted with the message box below:

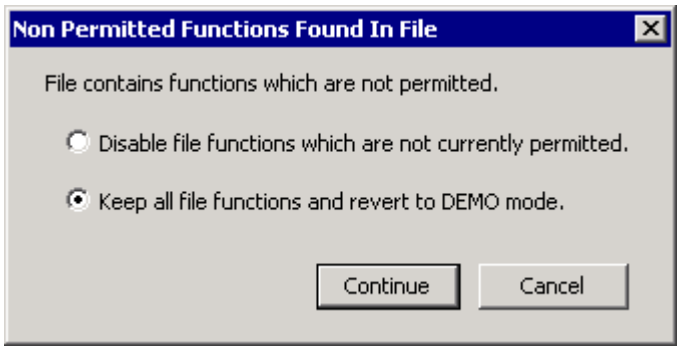


Figure 1.2: Function Groups Not Permitted Prompt

To continue the example you must respond *"Keep all file functions and revert to demo mode"*. In demo mode printing and saving are disabled.

On-line Help

Help information is available by choosing the *Help* menu or pressing <F1> on your keyboard. The On-line Help includes detailed technical information about menus, dialog boxes, and operation of the program.

It may be useful to refer to the On-line Help while working through the examples in this manual.

Tutorial Units

Most examples in this tutorial are in Imperial units (feet). To correctly follow the examples ensure Imperial (ft) units are enabled in the *Module* | *Setup* dialog box. If other units are used they will be specified at the start of the example. The procedures and concepts described apply to all unit systems.

Checkpoints 

Checkpoints (identified by the checkmark above) indicate the beginning of an example or a place where you can jump in without completing any earlier steps. All files required to start from a checkpoint are included on the CD.

Conventions

- The following conventions are used throughout the manual:
- Menu functions are delimited by a line "|". *File*|*Open* means to click on *File* in the menu bar and then select *Open* from the drop down menu.
- Dialog box control (like buttons) and heading names are *italicized*.
- The symbols "< >" contain keyboard functions. For example < shift-enter> means: hold down the *Shift* key and press the *Enter* key.
- File names and path names are **bold.**

2. Functional Overview

The RoadEng product comes with three *modules*:

Survey/Map

Terrain

Location

Each of the modules can be started from the Windows Start menu, a desktop shortcut or from the *Module* menu in either of the other modules.

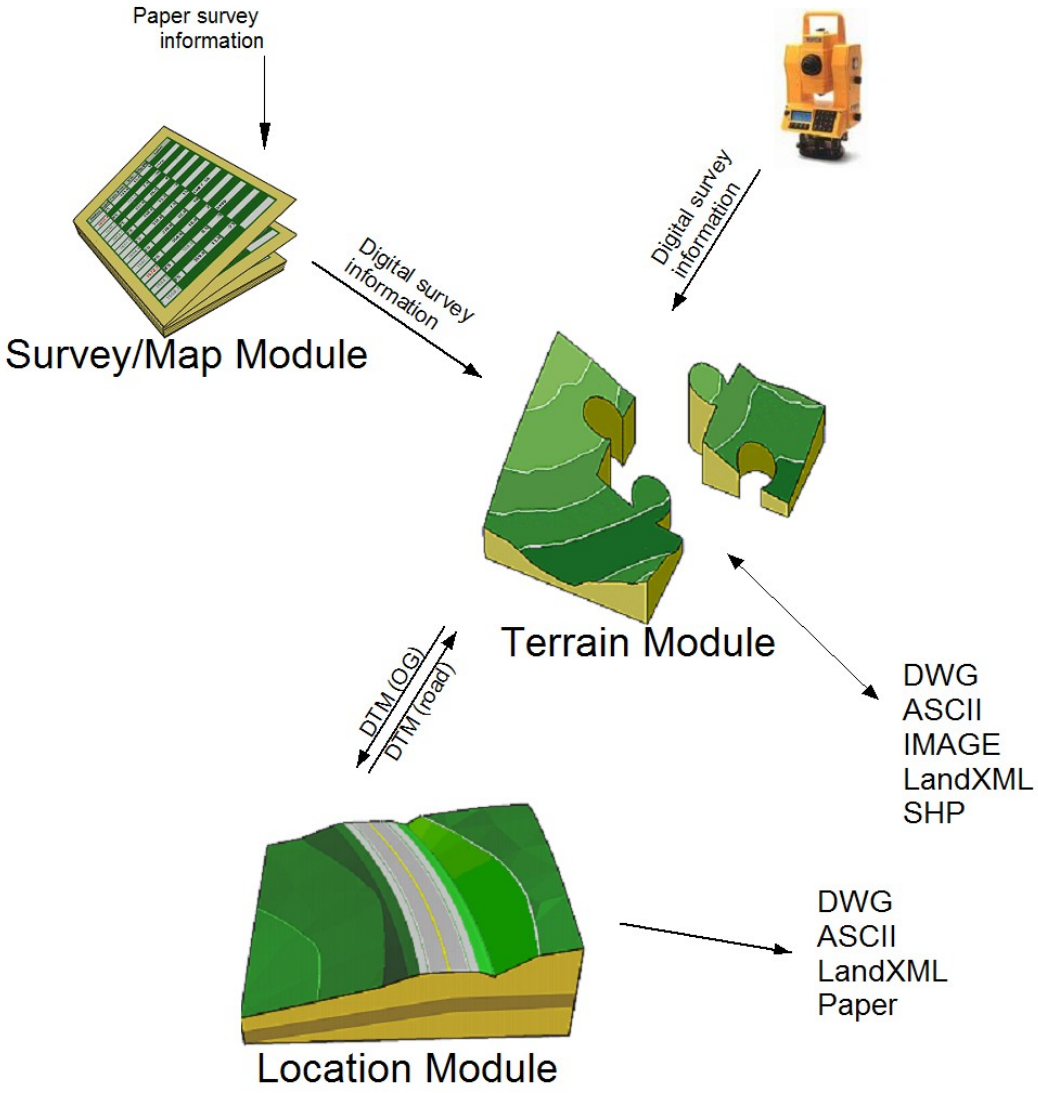


Figure 2.1: Relationship between modules

Survey/Map Module

This module is used primarily to type paper survey notes into the computer. Azimuths, distances and slopes are entered and reduced to coordinates. Facilities exist to add perpendicular side shots to a traverse so that a ribbon of terrain, suitable for a road design, can be easily captured with basic survey instruments.

Survey/Map also contains tools for adjusting traverses with respect to each other or to known coordinates.

Terrain Module

The Terrain Module provides basic CAD facilities for assembling and manipulating 2D and 3D points and features. Information can be imported from external sources like survey files, CAD files and image files. Three dimensional coordinates can be incorporated into a digital terrain model (DTM):

DTMs can be used for:

- Contour generation
- Section and profile display
- Volume calculations
- Pad, pit and site design (grading)
- 3D viewing
- Original ground for road design (Location module)

The Terrain module is also a capable mapping tool with control of line types, colors, symbols, hatching and labelling styles.

Location Module

This is the module used to design road alignments. Location requires an original ground terrain (provided by the Survey/Map and/or Terrain modules). The designer controls cross section templates, alignment location and curves. Location provides real time feedback of volumes, mass haul, road footprint, cross sections, grades, etc.

Location can also export designed surfaces back to the terrain module where they can be merged into a composite surface. This is the most common way to prepare the original ground for an intersection design.

3. Importing ASCII Survey Files

The Terrain Module will accept a variety of different ASCII files by allowing the user to configure the import format. This example illustrates the use of the import functions to read a topographic survey file created by a total station data collector.

A Typical Data File

The file (excerpt below) consists of a *sequence number*, *X*, *Y*, *Z* and *code* separated by tabs.

501	100005.519	669380.4079	374.3334144	SIGN
502	100005.4794	669377.6708	378.4704648	TOB
503	100005.455	669381.2522	373.6119528	DITCH
504	100005.5069	669382.2581	373.6689504	SHOULDER!
505	100005.5678	669383.4834	373.6997352	EP!
506	100004.9978	669360.2576	381.6608064	SPOT
507	100006.4914	669386.827	373.7369208	CLP!
508	100004.7662	669349.7755	383.6218896	SPOT
509	100024.0052	669385.6383	373.5726336	CLP
510	100021.4448	669349.0349	382.406652	SPOT
511	100023.7247	669382.4074	373.5458112	EP
512	100023.9594	669381.0297	373.482108	SHOULDER
513	100023.9625	669380.2037	373.415052	DITCH
514	100022.8043	669363.6653	379.8536472	SPOT
515	100041.7689	669378.9815	373.1014128	DITCH
516	100023.7491	669376.4882	377.8693992	TOB
517	100041.6561	669379.9873	373.180356	SHOULDER
518	100041.2203	669375.9365	376.1890368	TOB
519	100041.6409	669381.3284	373.2388776	EP
520	100042.0036	669384.5227	373.2394872	CLP

*Figure 3.1: Excerpt from **Survey1.txt***

Setting up an Import Format

1. Open the Terrain Module; select menu *Module*/*Setup*, and click on the *Units* tab. Select *Imperial (ft)* units if necessary. The import software cannot detect units from the information in an ASCII file.

2. Click on the *Import* tab and press the *Open* button. Browse to find the import options file **training Normal.iop** (if this file is not in your Layouts and Settings folder, see *Getting Started, Files* section). See figure below. Press *Open* to read the file.

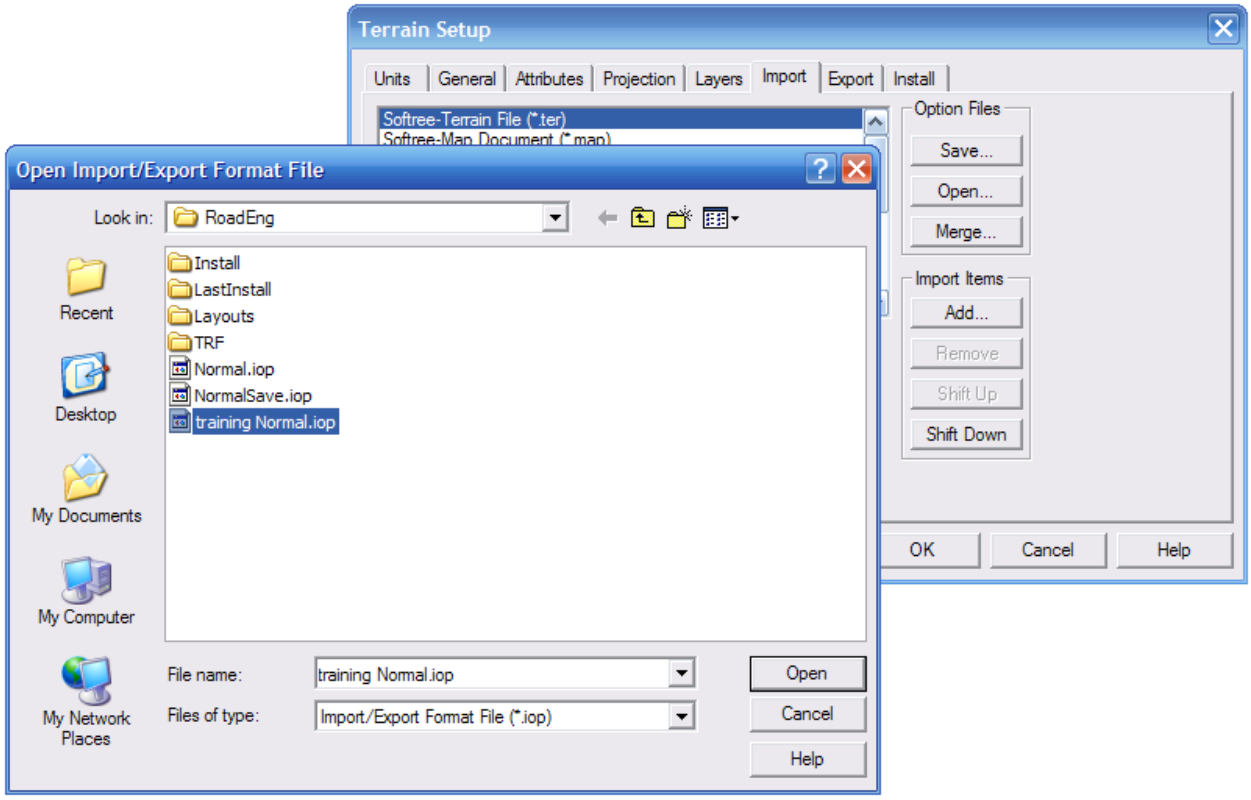


Figure 3.2: Opening an import/export format file from Terrain Setup.

Note: the default import/export formats are read from Normal.iop (found in your Layouts and Settings folder). Modify Normal.iop to change your defaults.

3. Select the format called "ASCII (x,y,z,code)", then press the *Add* button to open the *Define New File Format Options* dialog as shown in the figure below.

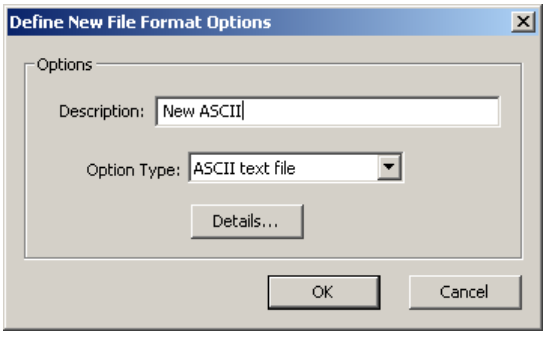


Figure 3.3: Define New File Format Options Dialog

Note: when you create a new import format, it will initially be a copy of the one selected when you press the *Add* button ("ASCII (x,y,z,code)" in this case).

4. Type "New ASCII" in the *Description* field and then click on the *Details* button to open the *Import ASCII Options* Dialog box shown below.

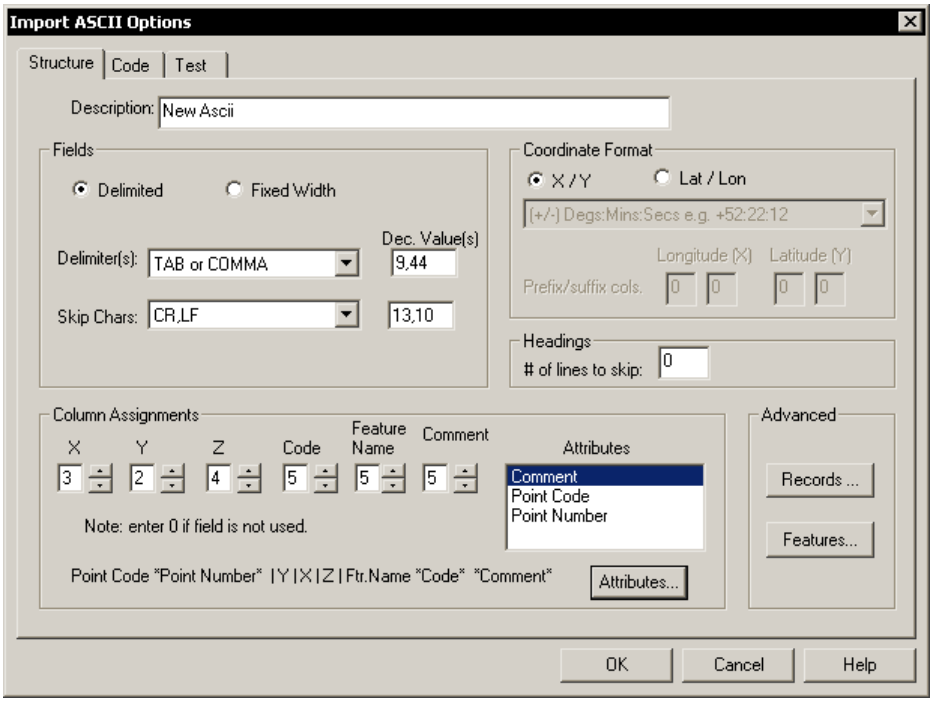


Figure 3.4: Import ASCII Options Dialog – Structure Tab

The *Import Ascii Options* dialog box allows you to describe the format of external files. Several options are available to identify, select and format incoming coordinate data. Detailed descriptions of the options in this dialog box are available by pressing <F1>.

5. Change the *Column Assignments* in the dialog box to match the figure above. Our file contains [point #, Y, X, Z, code] in each line.

You have now set up the import format to read data from the correct columns in the file.

6. Go to the top of the dialog box and select the *Code* tab (figure below). Here you can assign properties, symbols and line-types to the incoming points.

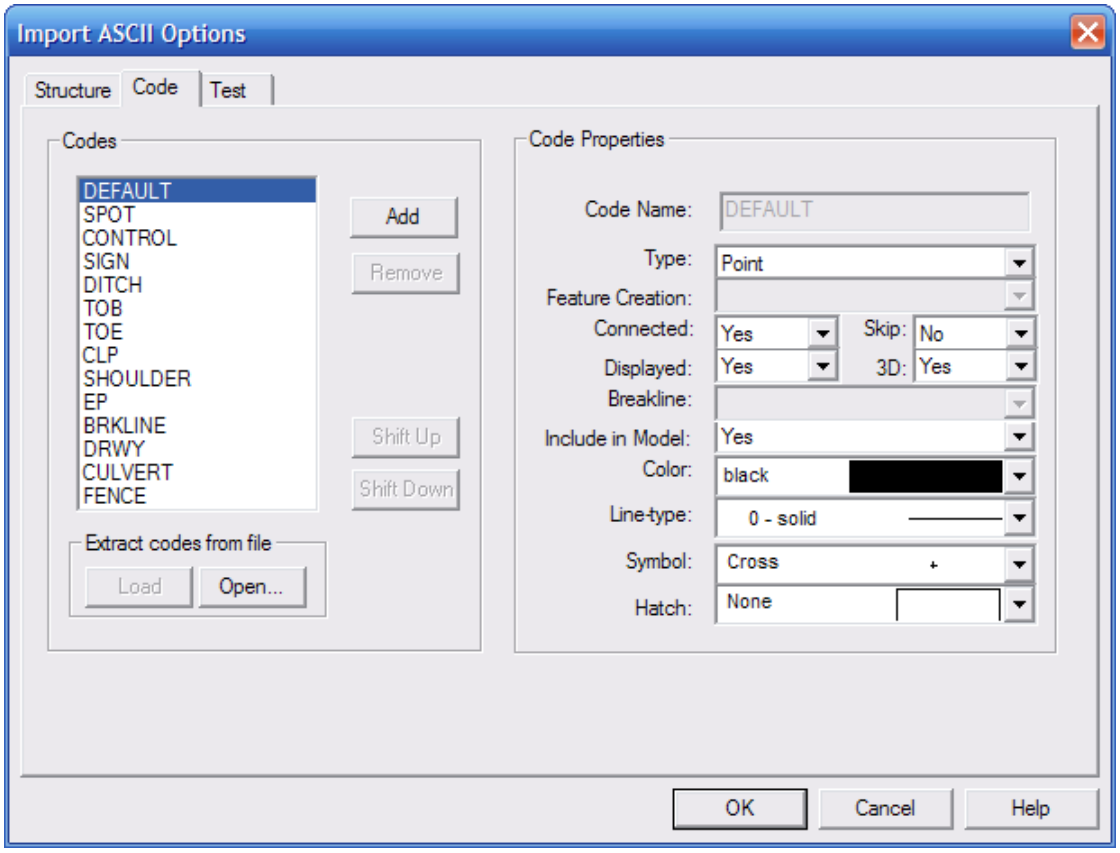


Figure 3.5: Import ASCII Options Dialog – Code Tab

7. Change your default code properties to match those shown in the figure above; 3D points with a black cross symbol.

8. Press the *Open* button and select the file called **Examples\ASCII Import\survey1.txt**. Press *Open*. This will extract all of the codes found in the file.

9. Select the **CONTROL** code found in the codes list. Note that the options initially are the same as DEFAULT. Change the *Color* to **navy** and *Symbol* to **Circle with cross**.

10. Select **EP** (Edge Pavement) in the code list and type in * beside EP in the Code Name. The "*" is a wild card – any code starting with "EP" will fall into this category. Change *Type* to **Polyline**. Change *Connected* to **Connect All by Code**. Change *Breakline* to **Yes**. Change the *Color* to **Blue**. Change *Symbol* to **None**. Points with the EP code will be connected together (in the order found in the file) and made into a blue breakline. The **Connect All by Code** property ensures that codes like EPL and EPR form separate features even though they both fit the EP* specification.

11. Select code name **CLP** (Center Line Pavement) in the code list. Change *Type* to **Polyline**. Change *Connected* to **Connect All**. Change *Breakline* to **Yes**. Change the *Color* to **Red**, *Line-type* to **dash-dot**.

12. Go to the top of the dialog box and select the *Structure* tab again. Press the *Features* button in the *Advanced* section on the lower right. The dialog shown in the figure below will appear.

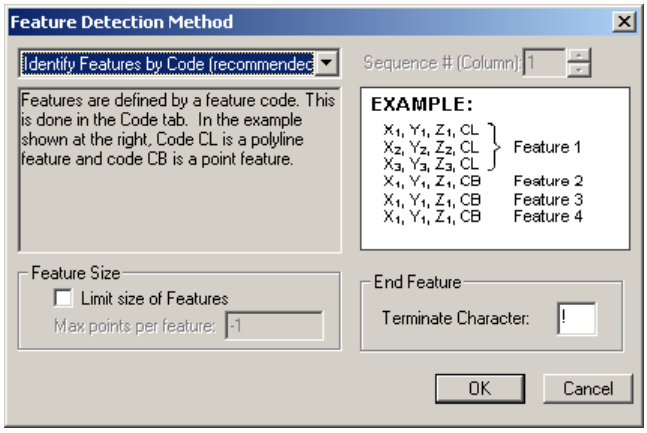


*Figure 3.6: This dialog box allows you to limit the length of **polyline** features by defining a termination character to be found in the point code.*

An exclamation point, "!", is defined as the termination character in the *Feature Detection Method* dialog box. If you refer to the **Survey1.txt** (see figure at start of this exercise), you will see many of the point codes end with "!"; this means that a connected feature breaks after this point and a new feature will be created when the next point of this type is encountered. The *EP* polyline code (defined above) will import as two breaklines (left and right) because of a strategically placed "!" in the survey point codes.

13. Press Cancel to exit the *Feature Detection Method* dialog box.

14. To test the specification, go to the *Test* tab (see figure below).

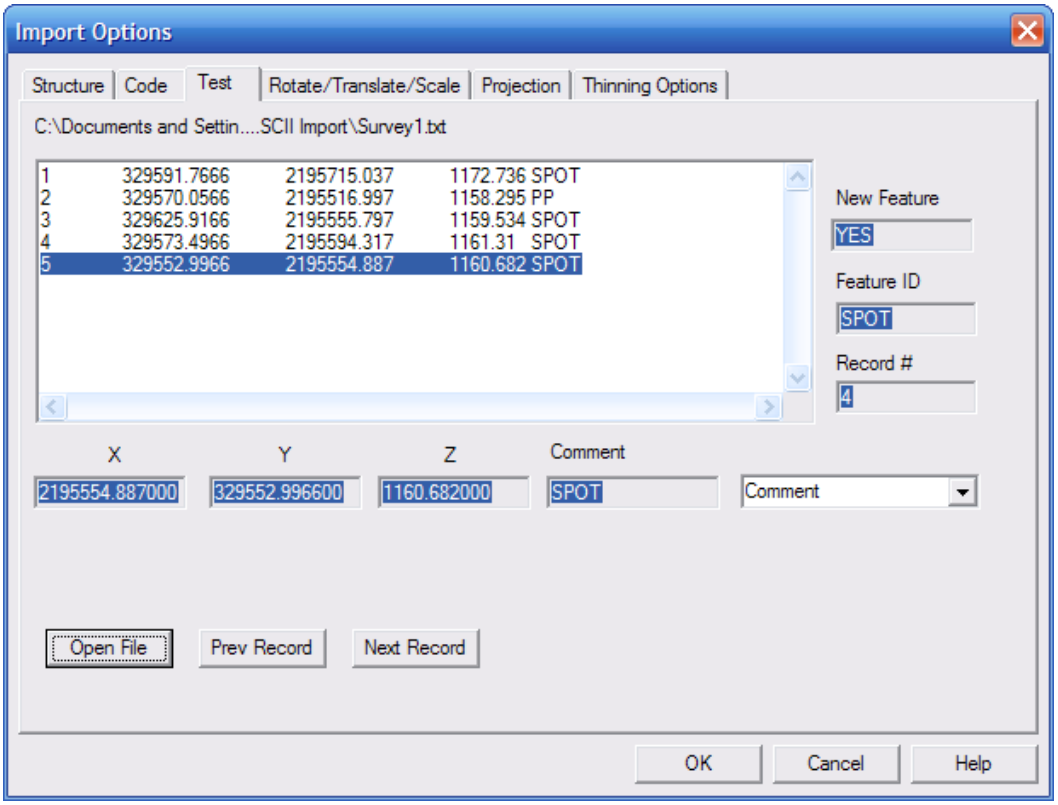


Figure 3.7: Import ASCII Options Dialog – Test Tab

15. Press the *Open File* button and choose **Survey1.txt.**

16. Press the *Next Record* button several times. At the bottom of the dialog box the values of X,Y,Z and comment are displayed. Confirm that the incoming fields are being correctly interpreted; if not return to the other tabs to modify the format.

17. When satisfied, press *OK* to return to the *Define New File Format* dialog and *OK* again to add the new format to the list.

18. To save the new import specifications for future use, press the *Save* button. Normally, you would choose **Normal.IOP** and write over it (to update your default settings) – do this only if you are working on a computer used for training only, otherwise save as **training.IOP** or ***Cancel*** to avoid changing your defaults.

19. Press OK to close the *Terrain Setup* dialog box.

Now we'll use the import format we've created to open the survey data file.

20. Select menu *File* | *Open*. Change *Files of type* to *New ASCI I* (at the bottom of the list). Open **Survey1.txt**. You will be presented with the *Import Options* dialog box to allow last minute changes. Press *OK* to import the file.

21. Use the layouts drop down in the standard toolbar (or menu *File* | *Retrieve Screen Layout*) to open **training ASCII import.ilt**. This will set up your options and windows to look like the figure below.

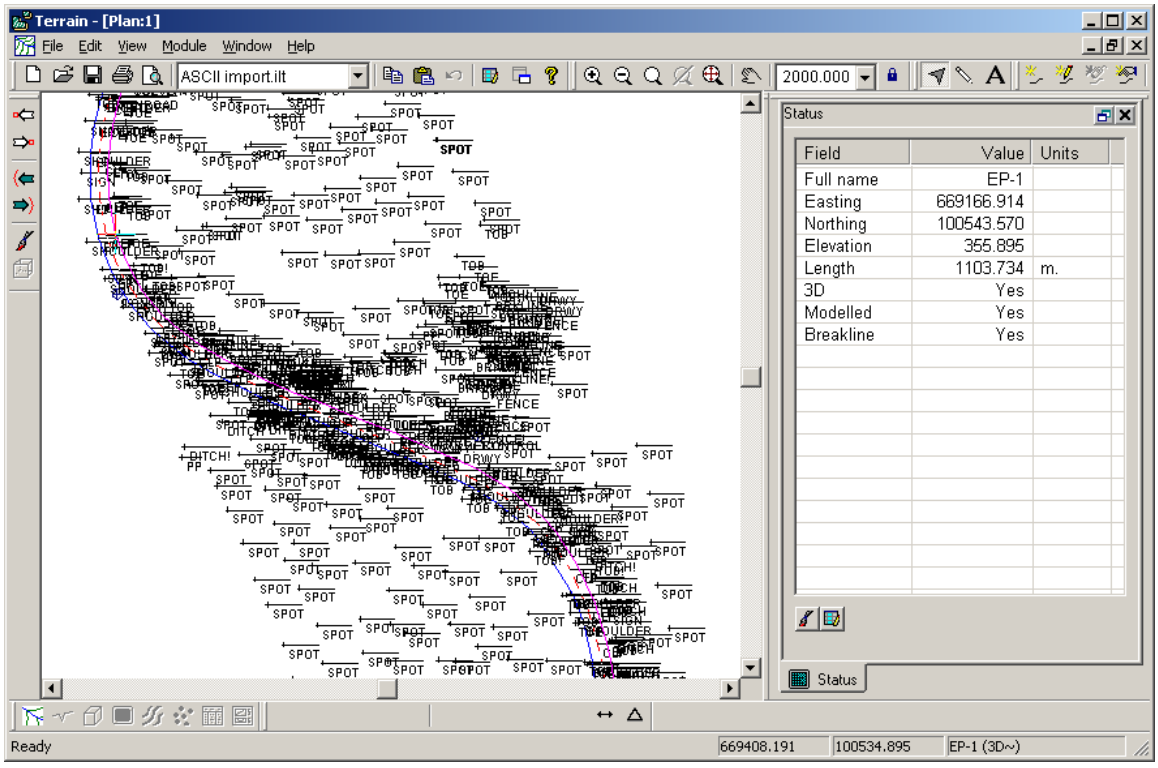


*Figure 3.8: Plan window after importing **Survey1.txt**. The right **EP** feature is selected (note the properties displayed in the status window).*

Note that there are many point codes that have not been formatted or connected to form breaklines. In the next steps, we will re-read the same data with a prepared import format.

22. Select menu *File* | *Open*. Change *Files of type* to *ASCII 2 (#,y,x,z,code)*. Open **Survey1.txt**. When prompted to save changes, choose *No*.

23. You will then be presented with the *Import Options* dialog; click on the *Code* tab to see the extra codes defined – no changes are required. Press *OK* to import the file.

24. Use the layouts drop down in the standard toolbar (or menu *File* | *Retrieve Screen Layout*) to open **training Normal.ilt**. This will set up your options and windows to look like the figure below.

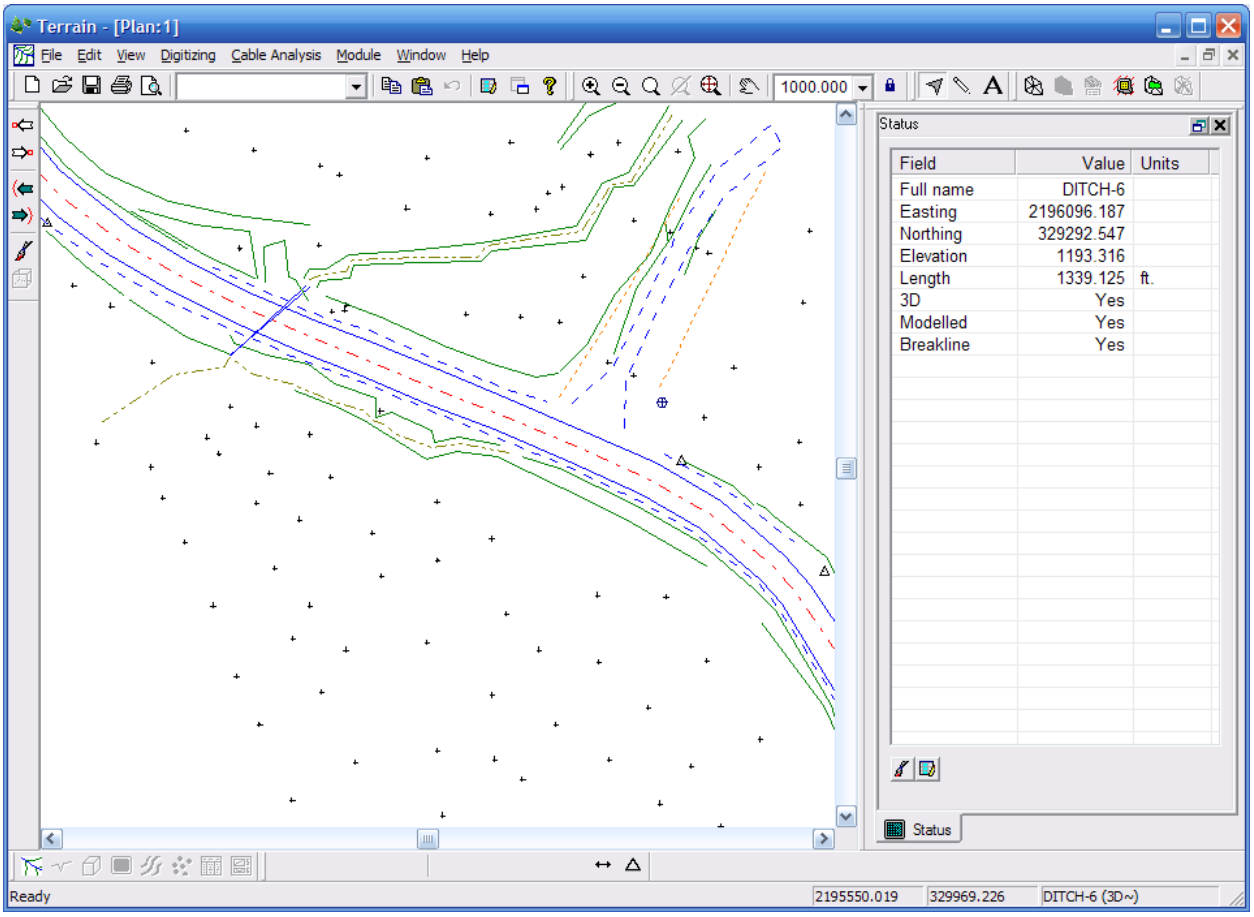


Figure 3.9: ***Survey1.txt*** *imported with more point codes defined.*

You may wish to select features with the mouse to see what properties are displayed in the *Status* area.

25. To conclude this exercise, choose menu *File* | *New*; do not save changes.

4. Creating a DTM with Contours

In this exercise, you will open a file containing 3D data (imported in the *Importing ASCII Survey Files* exercise above) and create a *Digital Terrain Model* (DTM).

You will also generate major and minor contour lines.

Note: The digital model is represented by a *Triangular Irregular Network* (TIN); for this reason, menus, documentation and help files often refer to a Digital Terrain Model as *TIN* model.

1. Open the Terrain Module; select menu *File* | *Open*. Browse for file **Examples \DTM\Topo 1.ter**. Press *Open* to load the file (figure below).

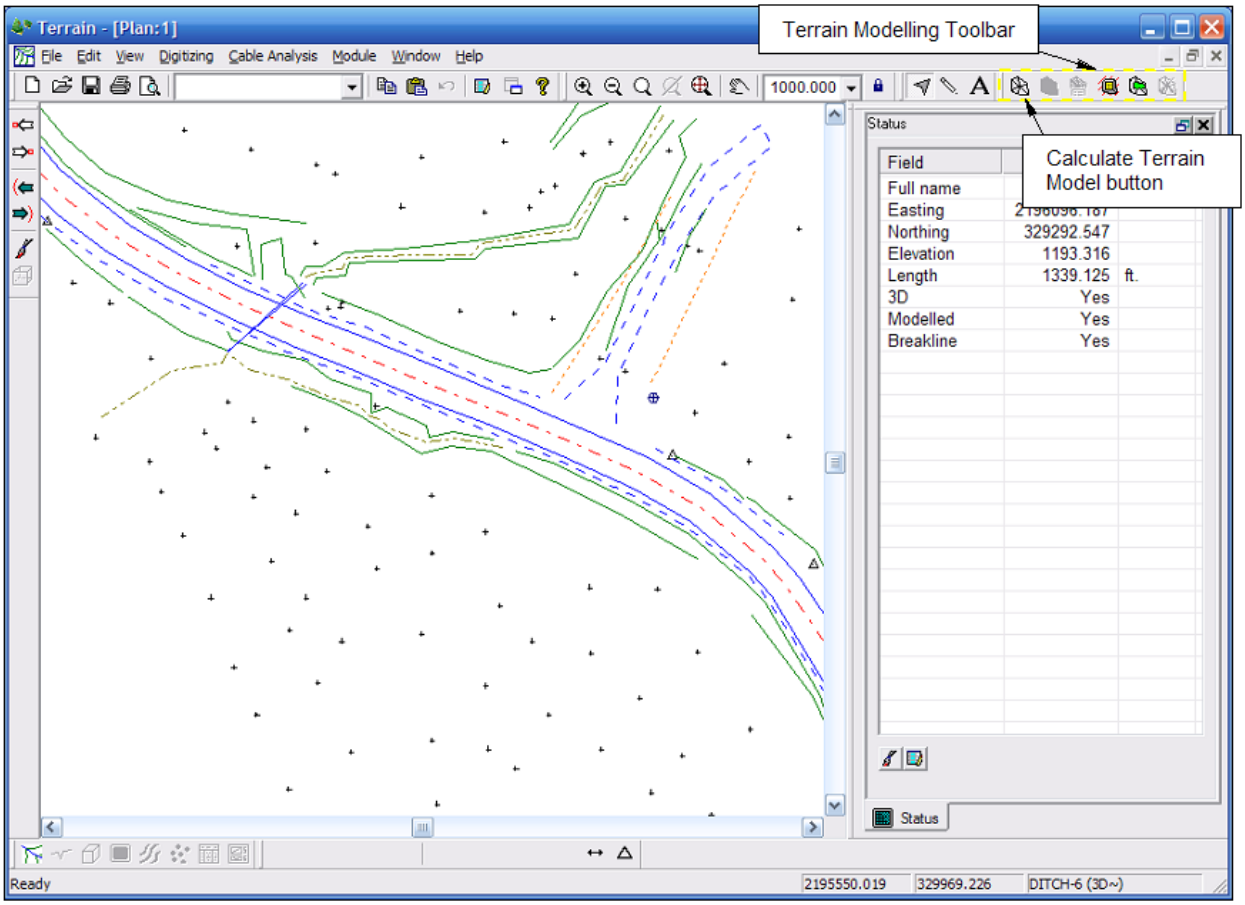


*Figure 4.1: Terrain file **Topo 1.ter**.*

2. Click the *Generate TIN* button  in the toolbar (see figure above). You can also use the menu *Edit* | *Terrain modeling* | *Calculate Terrain Model*. This will open the dialog box shown below.

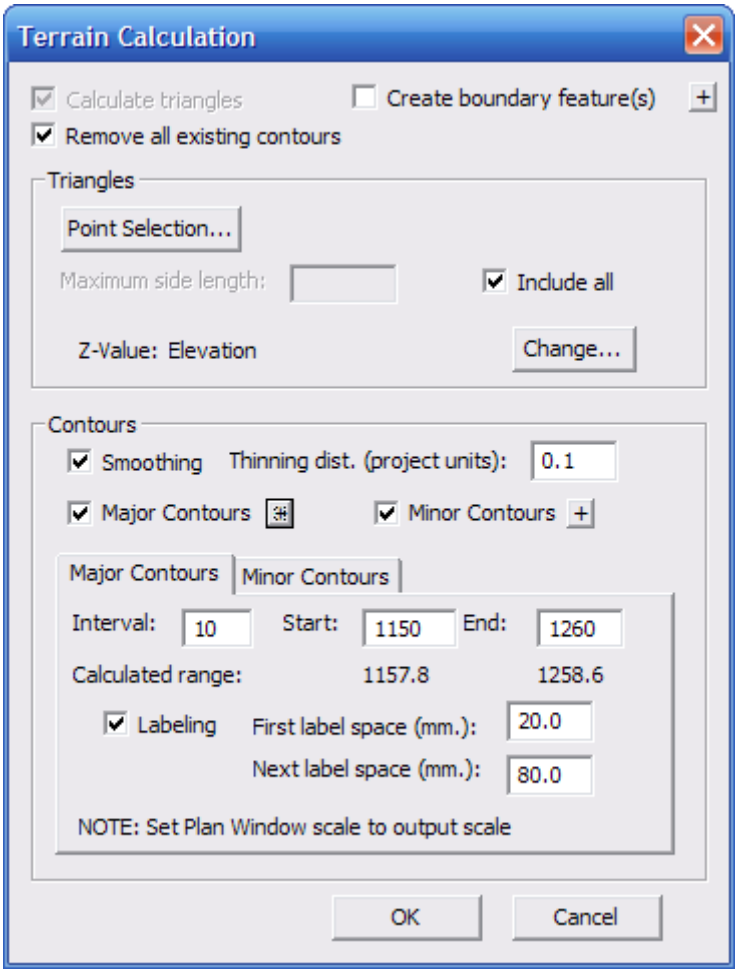


Figure 4.2: Digital Terrain Model calculation dialog box. Optional contours enabled.

Contour specification

3. In the contours area, make sure that *Major* and *Minor Contours* are turned ON. If you click on the button the Major or Minor Contours check boxes, you can change the color and line type used for the contour lines. Optional contour *Smoothing* (controlled by Thinning distance) rounds the corners where contours cross triangle sides – smoothed contours do not match the model elevation exactly.

Note: Default contour line types and colors are stored in the **Normal.ilt** screen layout. Any changes made after a new document is created are saved with the document.

4. Click on the *Major Contours* tab and set the interval to 10 and make sure *Labeling* is turned ON as shown above. You also need to set the *Start* elevation to be a multiple of 10 (1150 in this case).

5. Click on the *Minor Contours* tab and set the interval to 2 and make sure *Labeling* is turned OFF. You also need to set the *Start* elevation to be a multiple of 2.

6. Press the *OK* button to generate both TIN and contours. The figure below shows the result.

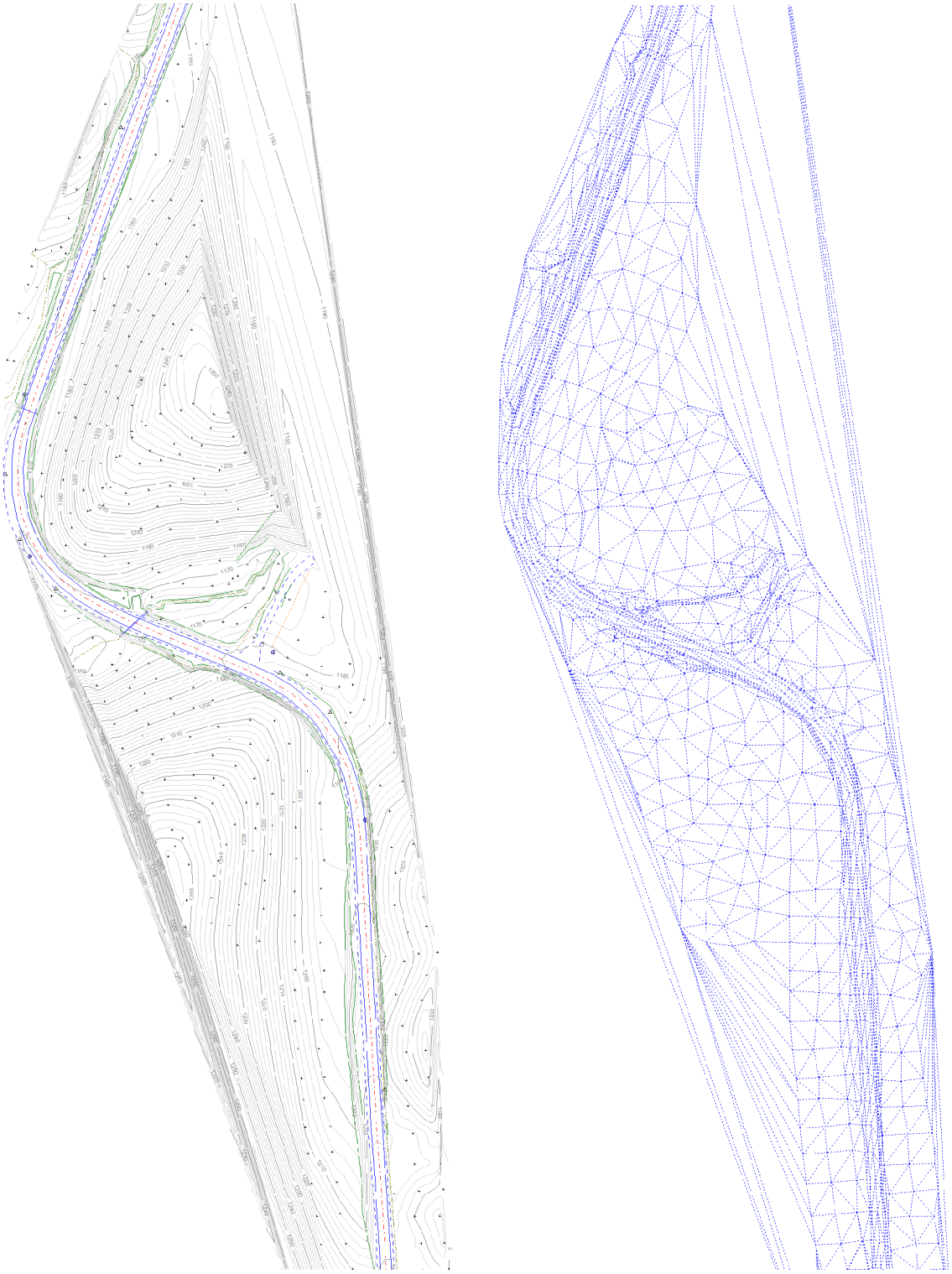


Figure 4.3: Contours generated without boundary or length limitation. Underlying triangles shown at right.

The figure above shows how a *Triangular Irregular Network* (TIN model) is created from 3D data points. Once the TIN model has been generated, contours are formed by creating a straight-line segment across each triangle (see figure below).

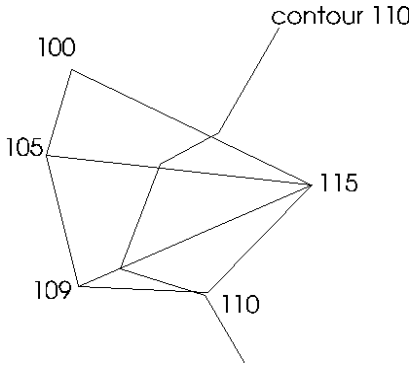


Figure 4.4: Contour formation from TIN model. Elevations between known elevation points are interpolated. If contour smoothing has been enabled, the contours will be less angular.

Limiting Triangles

In this example, the triangles (and resulting contours) on the upper right and lower left of the model are unrealistic – elevations are being interpolated between points very far apart. There are two ways to prevent these unrealistic triangles:

- Create a boundary polygon (with property *TIN boundary*).
- Limit triangle length.

A boundary polygon will limit triangle formation to an area of interest – this can also be useful when your data set is very large or when you wish to merge a small DTM into a larger one. TIN boundaries will be covered in other exercises.

In this example we will limit the triangle length.

7. Click the *Generate TIN* button  in the tool bar to re-open the *Terrain Calculation* dialog box.

8. Turn ON the *Calculate triangles* check box, turn OFF *Include all* and set the *Maximum side length* to 150 (see figure below). If you set this value too small, there will be holes in your model.

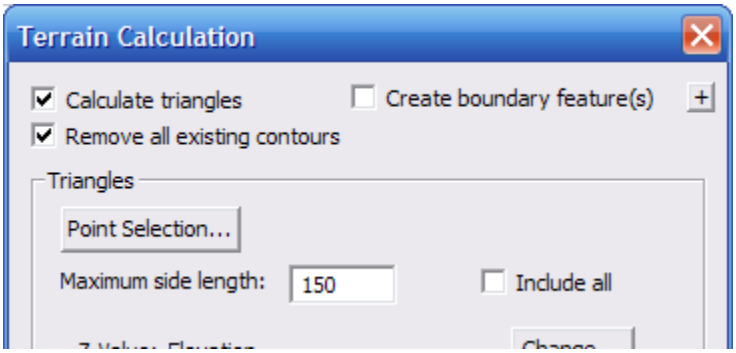


Figure 4.5: Terrain calculation with triangle Maximum side length limited.

9. Press *OK* to recalculate triangles and contours. Your Plan window should look similar to the figure below.

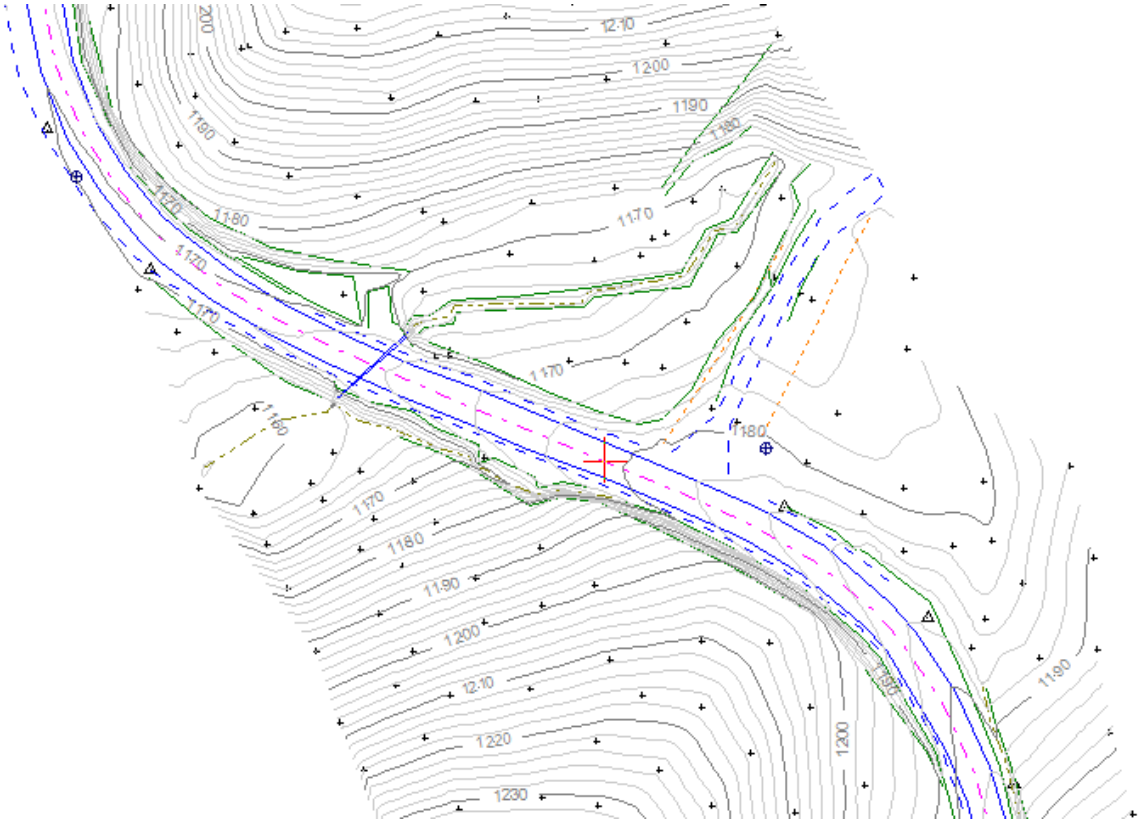


Figure 4.6: Terrain model with triangles limited to 150 feet.

At this point you may wish to experiment with some of the other options in the Terrain Calculation dialog box. Once the dialog box is open type <F1> to see detailed help information.

10. Choose menu *File* | *New*; do not save changes.

5. Moving Around in the Plan Window

In this exercise you will use the *Zooming* and *Panning* functions to change the Plan view. You will also select features with the mouse to examine their properties in the Status window. Many of these functions work in other graphics window types.

1. Open the Terrain Module; select menu *File* | *Open*. Browse for file **Examples \DTM\Topo with issues.ter**. Press *Open* to load the file (figure below).

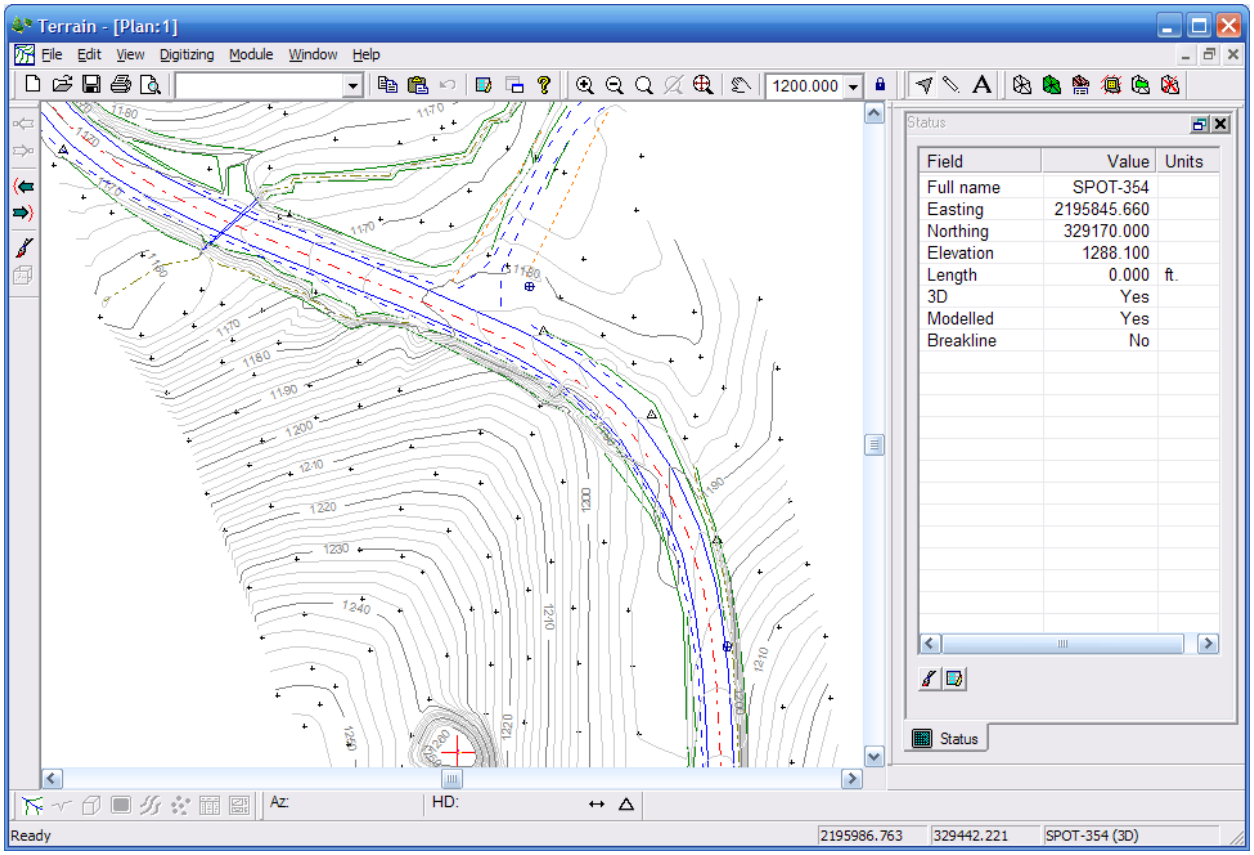


*Figure 5.1: Terrain file **Topo with issues.ter**.*

Selecting features with the mouse

2. Right click in the Plan window and make sure that *Select with mouse* is checked in the context menu. Your cursor will look like an arrow .

3. Click on the red center line feature.

When you click on a feature with the selection cursor , several things happen:

- It becomes the *current feature* and is highlighted by changing color to magenta.
- The point nearest where you clicked becomes the current point and is indicated with a red cross.
- The status window shows information about the new current point and feature if applicable.
- The status bar shows the current feature name (lower right corner of application window).

4. Click on the Next and Previous point buttons and note how the current point moves along the selected feature. <Ctrl-N> and <Ctrl-B> have the same effect.

Note: If you move the current point (<Ctrl-N> and <Ctrl-B>), all windows will automatically scroll to make the new current point visible.

5. Try clicking and dragging with the mouse to see how window selection works.

6. Hold the <shift> key and click on a feature. This allows you to add and remove features from a selection set. It works with click and drag also.

Zooming and Panning

The *Zoom Tools* toolbar allows you to *zoom in, zoom out, zoom to window, end zoom, zoom extents* and *pan* respectively. The function of these tools is mostly self evident with a little experimentation.

The middle roller mouse button is dedicated to zoom and pan functions. If these functions do not work as described below, it is likely because of mouse software that has been configured to override the default behaviour – check your control panel.

7. Move your mouse cursor over the Plan window and click and drag with the *middle* mouse button; even a roller button can be "clicked". Note that the mouse cursor changes into the *Pan* hand , and the plan image moves with your mouse.

Note: The dedicated middle mouse *Pan* function can be much more convenient than scroll bars. You can turn scroll bars off to save space (right click, *Active Window (Plan) Options, General* tab, clear *Scroll bars*).

8. Move your mouse cursor to a point of interest then roll the *middle* mouse button *away* from you. Note how the image zooms in and how the point of interest stays under the mouse. This is quite different from using the *Zoom 200%* button which always keeps the center of the screen in the same place.

9. Similarily, use the middle roller mouse to zoom out by rolling towards you.

10. Practice zooming and panning while you look for interesting features of the model. Note that the scale changes (tool bar) every time you zoom in or out. Also note that the text remains the same size (although this is an option) and that the symbol sizes and line thicknesses remain unchanged (figure below.

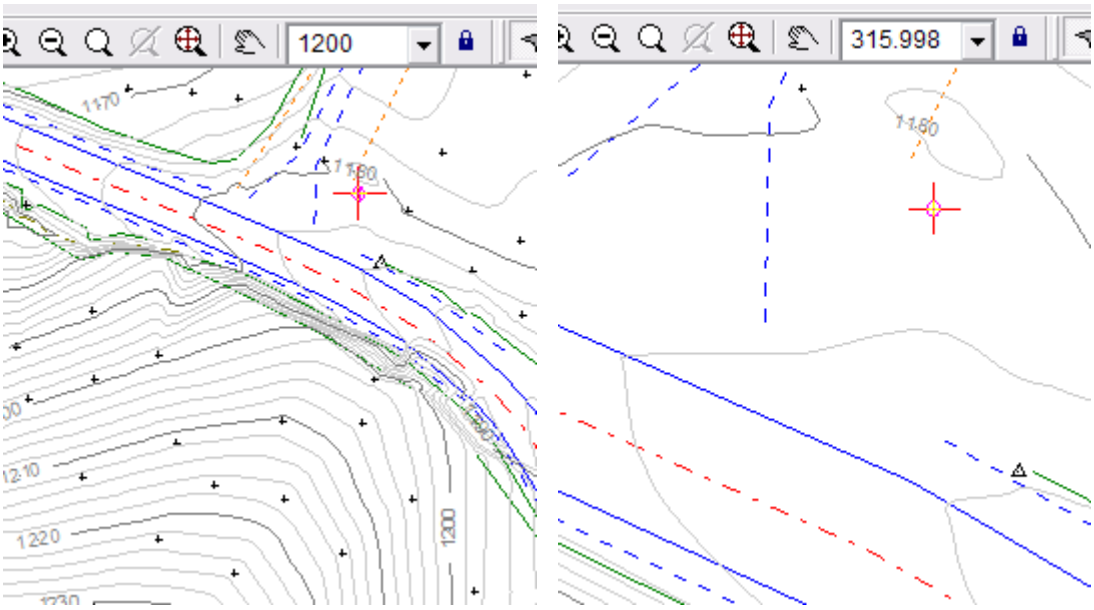


Figure 5.2: Before and after a zoom operation with scale un-locked.

11. Set the scale to 1200 in the toolbar (note this is a natural scale, the same as 1” = 100 ’).

12. Click the *lock scale* button , so it appears depressed, and try a few zoom operations. Note that this time, the scale does not change but the text, symbols and lines appear magnified or shrunk (figure below).

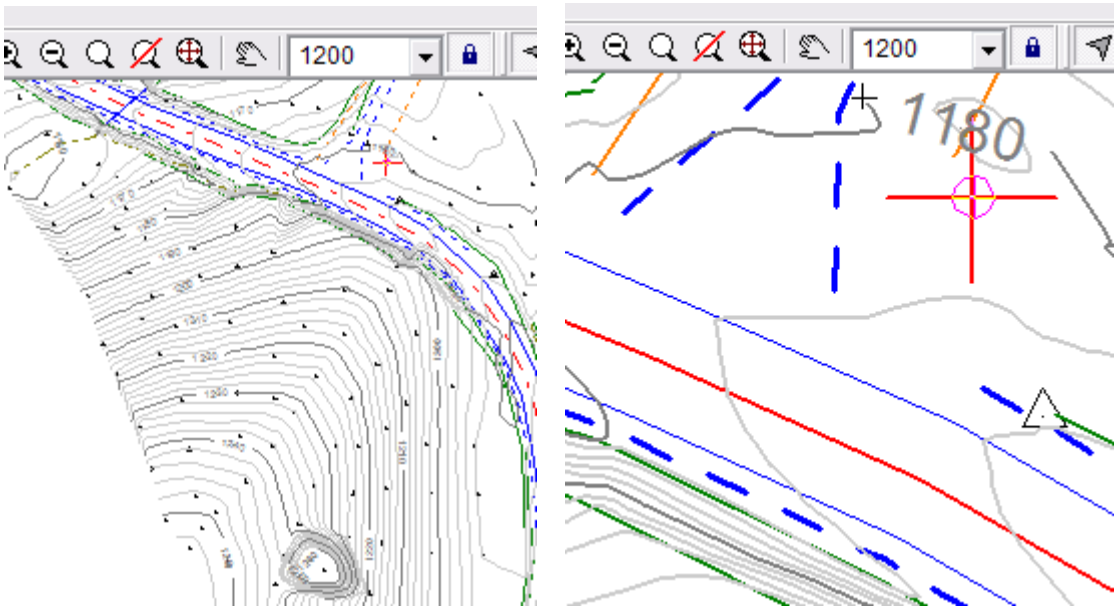


Figure 5.3: Zoom out and zoom in, respectively, with scale locked.

13. When you have finished experimenting with mouse feature selection and moving around, *File | New,* do not save changes.

6. 3D Moving Around in the 3D Window

In this exercise you will use the *Zoom*, *Pan* and *Rotation* functions to change the 3D view. You will also use the current point to help navigate in the 3D window and to help to find corresponding points in Plan and 3D views.

1. Open the Terrain Module; select menu *File* | *Open*. Browse for file **Examples \DTM\Topo with issues.ter**. Press *Open* to load the file.

2. Select menu *Window* /*New Window* /*Graphics* /*3D* and a 3D window will appear on your screen.

   The rendered surface should be visible; if it is not, press the *Zoom extents* button in the tool bar (this does not always work if your model contains stray points).

3. Use menu *Window* | *Tile Vertically* to show Plan and 3D windows side by side (see figure below).

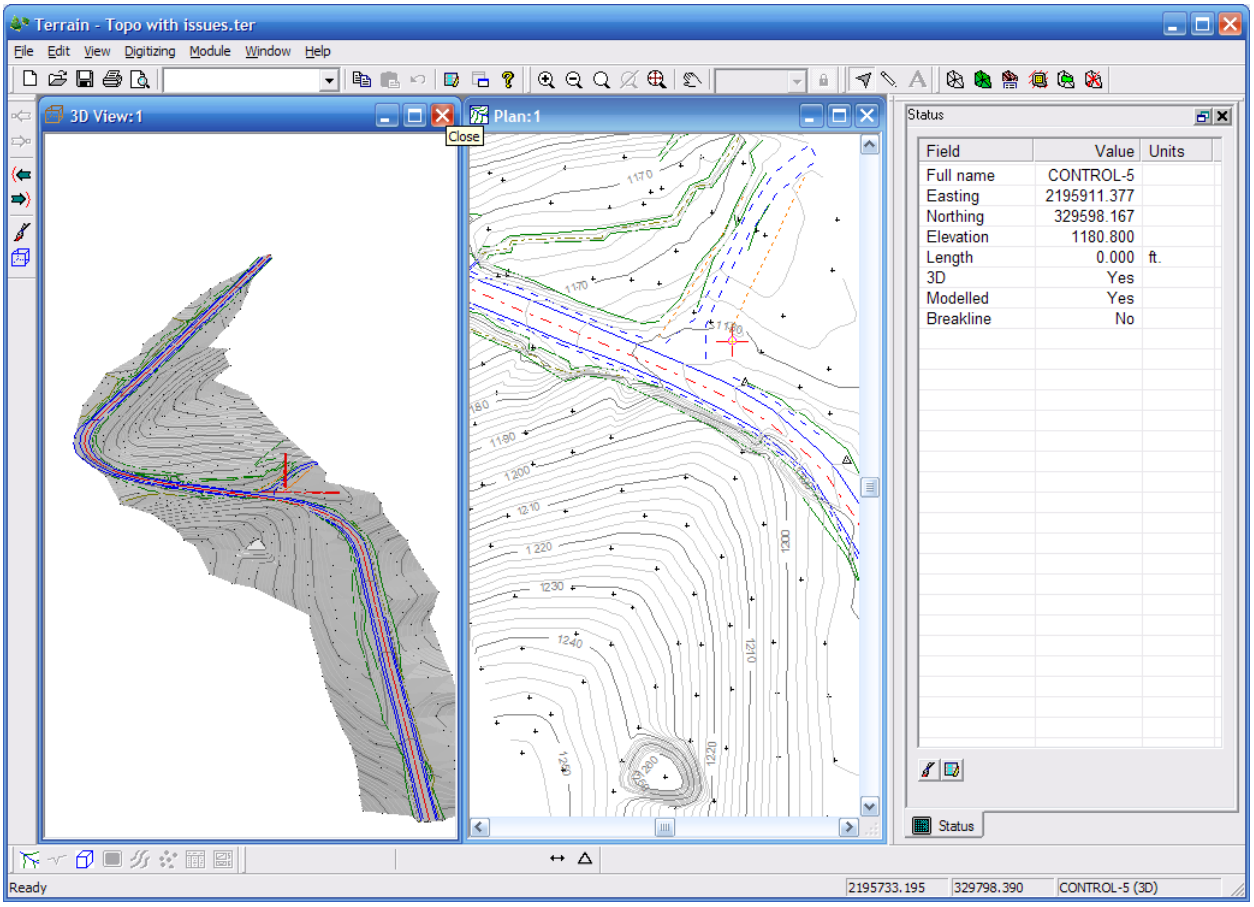


*Figure 6.1: 3D and Plan windows showing file **Topo with issues.ter**.*

Now we need to move around in the two windows to find problems with the model. In the 3D window, *Zooming* and *Panning* behave in a similar way to the Plan window (see previous exercise *Moving Around in the Plan Window*).

4. Use the zoom tools in the tool bar or the *middle* mouse to move around in the 3D window.

Rotating the 3D image

The 3D window, however, also allows you to rotate the image.

5. Click and drag with the *left* mouse and notice how the 3D view changes. It may take a little practice to get the hang of it.

6. Make sure you have a current point defined by clicking with the selection cursor on a feature in the **Plan** Window. Note that the current point is represented by a three dimensional red cross in the 3D window.

3D window options

7. Right click in the 3D window and select the menu *Active window (3D) Options*. The dialog box shown below will pop up.

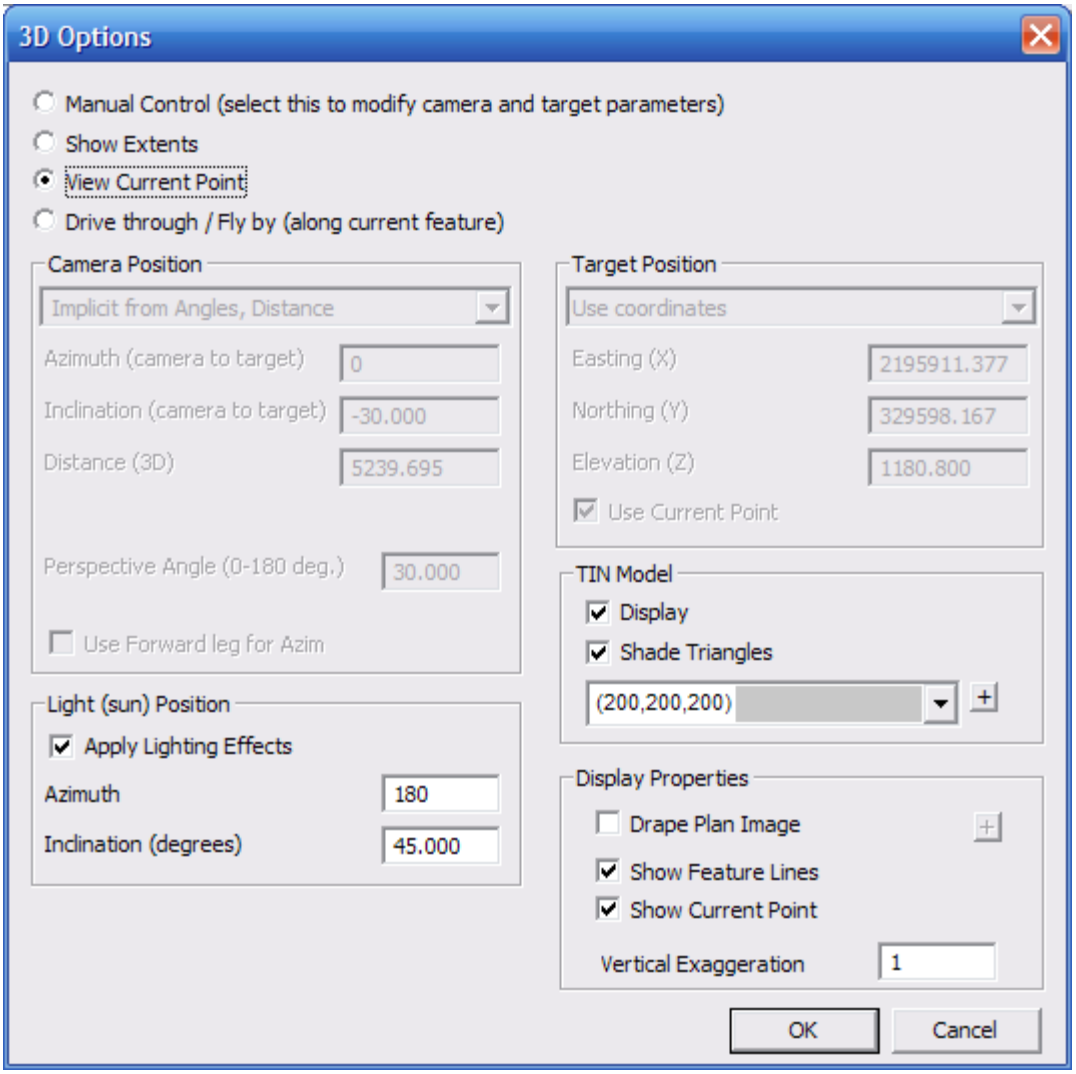


Figure 6.2: 3D window options dialog box.

The *3D Options* dialog box allows you to change many of the rendering options including camera and target positions. You may wish to experiment with some of these options if time permits. Don't forget about the <F1> help key.

8. Click on the *View Current Point* radio button near the top left of the *3D Options* dialog box. This sets the target position to the current point.

9. Press *OK* to accept the change and close the dialog box.

10. Select different current points in the Plan window with the mouse or by typing the <Ctrl-N> or <Ctrl-B>. Notice how the 3D view tracks the current point.

Note: If your 3D window is empty or if it doesn't rotate in a predictable way, use the 3D window options to change the mode to *View Current Point* (Right click and select the menu *Active window (3D) Options*). You must have current point selected. This will scroll the image into view and change the rotation point to the current point.

11. When you have finished experimenting with the 3D window, select menu *File | New*; do not save changes.

7. Finding and Repairing DTM problems

In this exercise you will use the 3D window to help find problems with a DTM. You will also remove bad data points from the model and tag critical features as breaklines. It is possible to find all the problems with this model by looking carefully at the contours (especially as they are closely spaced). However the 3D window often makes this task quicker and easier.

You should already be familiar with moving around in the Plan and 3D windows (previous two exercises)

1. Open the Terrain Module; select menu *File* | *Open*. Browse for file **Examples \DTM\Topo with issues.ter**. Press *Open* to load the file.

2. Select menu *Window* /*New Window* /*Graphics* /*3D* and a 3D window will appear on your screen.

3. Use menu *Window* | *Tile Vertically* to show Plan and 3D windows side by side.

Removing a bad point from the model

4. Adjust the Plan and 3D views until you can see the bad elevation point shown below.

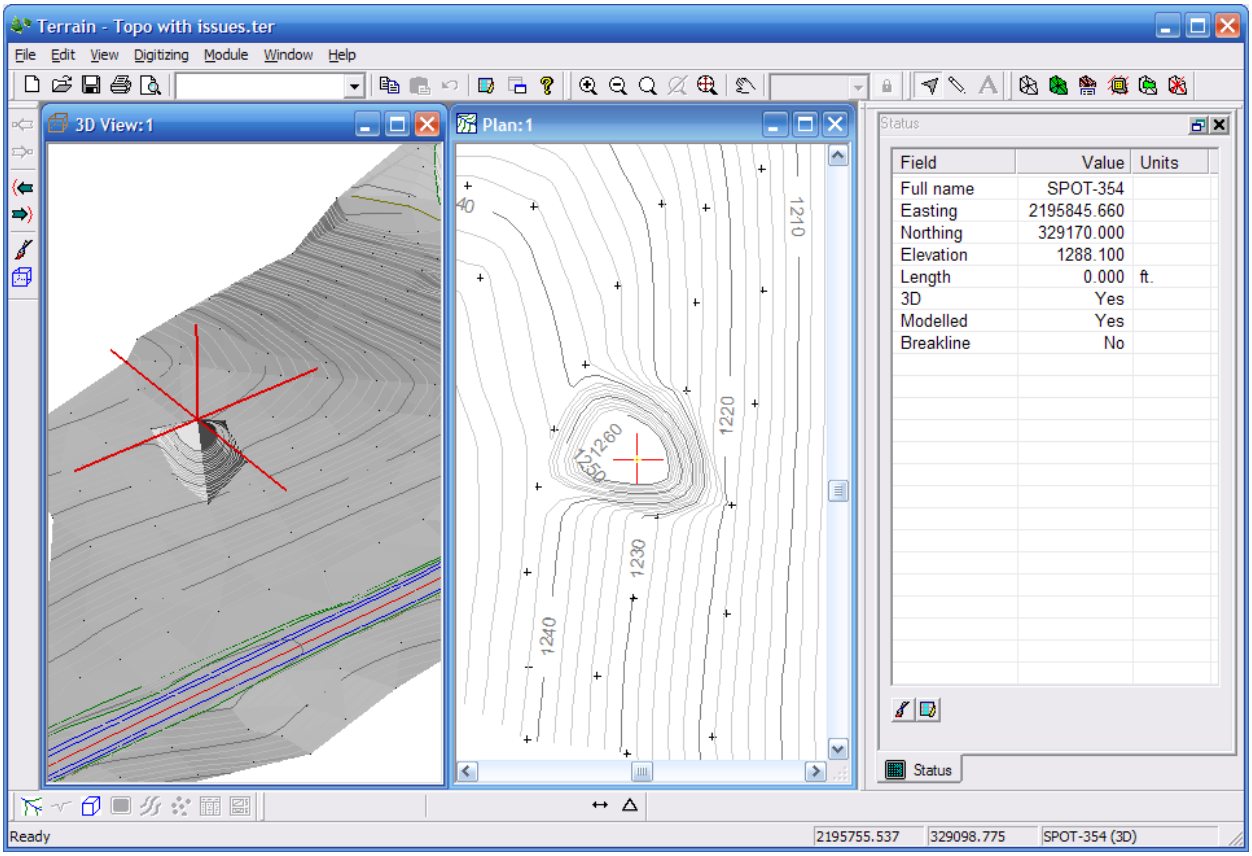


Figure 7.1: Bad elevation point shown in 3D, Plan and Status windows.

5. Select the bad point in the Plan window with the mouse . You know you've selected the correct point when the 3D window shows the current point on top of the anomalous spike (figure above). Note that the Status window shows that this point is a *3D modelled* point – it is part of the TIN surface.

At this point you could delete the feature but then there will be no record of this point. Instead we will remove it from the TIN model.

6. Use the menu *Edit* | *Modify Selected Feature(s)* | *Properties* to open the Feature properties dialog box shown below. Note that the *Modify Selected Feature(s)* menu is also available when you right click in any of the windows shown – you can also use the speed key <Ctrl-E>, as shown in the menu text.

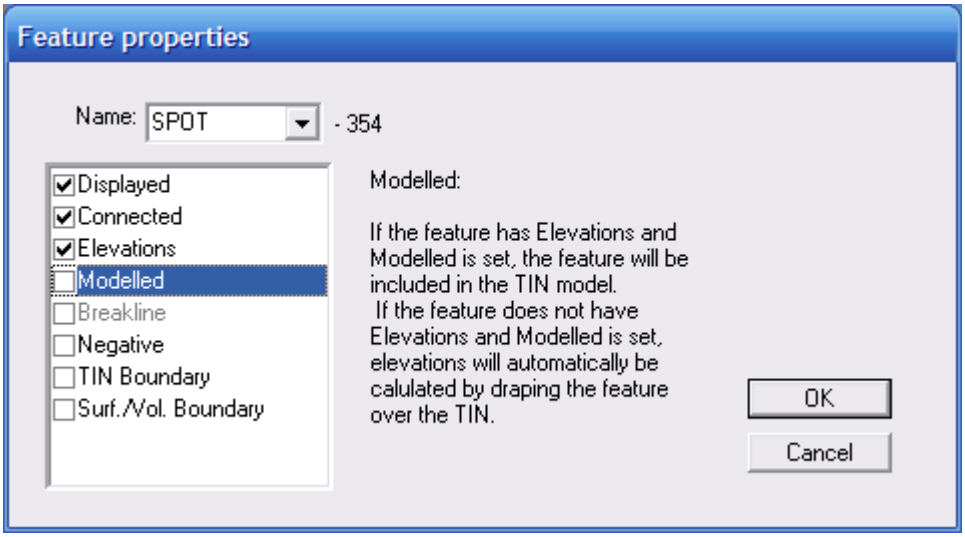


Figure 7.2: Feature properties dialog box.

7. Clear the *Modelled* property so the point feature will no longer be part of the model.

8. Press *OK* to accept the change and close the dialog box.

9. When warned that "existing triangles will be cleared" respond *OK*.

Note: The above procedure is typical of most Terrain Module operations:

First, select features of interest (sometimes the *current feature* and *current point* are important).
Second, use the *Modify Selected Feature(s)* menu to do something to the selection set.

10. Use the *Generate TIN* button in the tool bar to open the *Terrain Calculation* dialog box (see *Creating a DTM with Contours* exercise above). The settings for this dialog box were configured when this file was created; you don't need to adjust anything.

11. Press *OK* to recalculate the DTM and the contours. Note that the anomalous spike in the model has disappeared.

Defining breaklines

12. Adjust the Plan and 3D views until you can see along the curve in the road shown below.

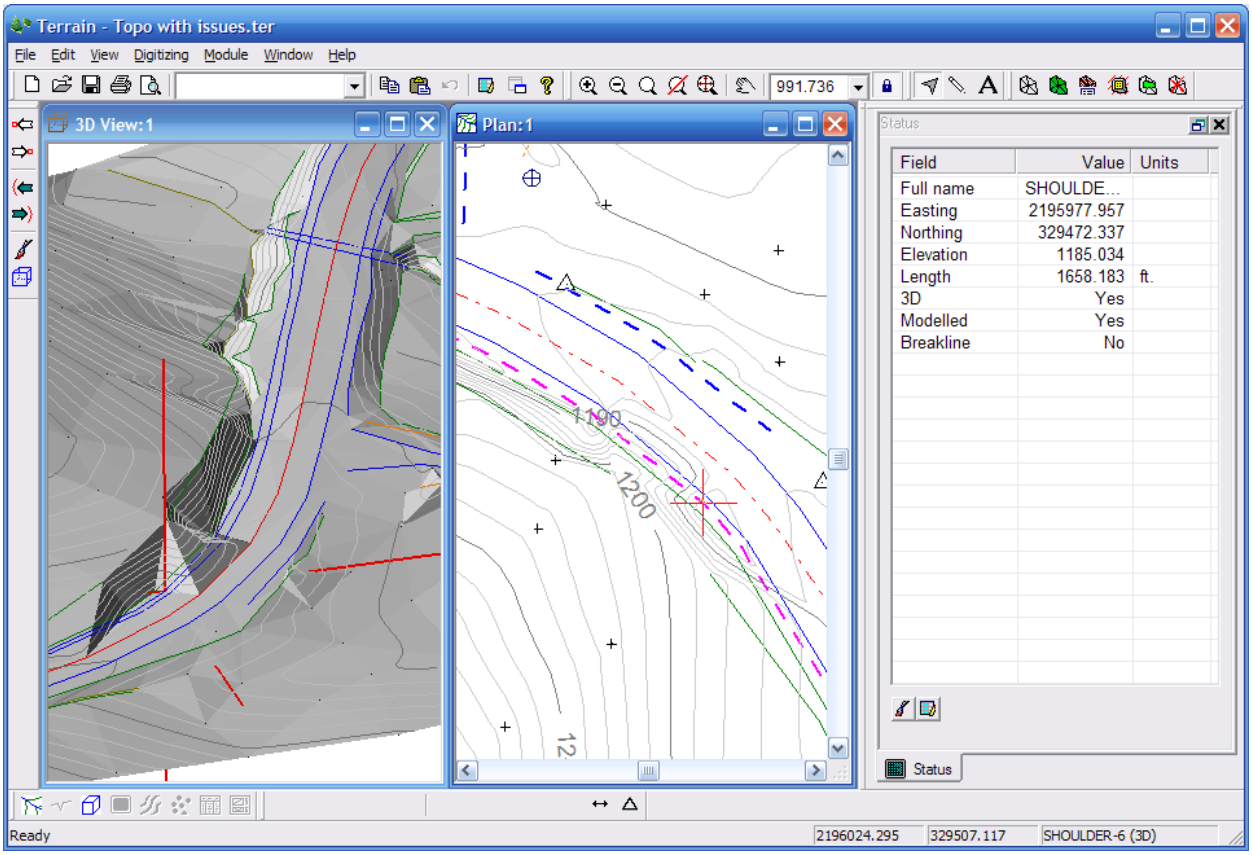


Figure 7.3:3D and Plan contours showing bad triangles caused by missing breaklines.

What looks like a land slide in the figure above is actually a triangle formed by connecting centerline survey points with their nearest neighbour, a top of bank point. We know that the shoulder of the road should be a smooth and continuous line; in terrain modelling terms this is a *breakline*. Some typical breaklines are listed below:

- Road shoulder
- Ditch bottom
- Top of cut
- Toe of fill
- River bank

13. Select the SHOULDER feature as shown in the figure above. Note that the properties shown in the Status window indicate that this is NOT a *breakline*.

14. Open the *Feature Properties* dialog box (as in step 6 above) and Turn ON the *Breakline* property.

15. Press *OK* to accept the change and close the dialog box.

16. Recalculate the Terrain Model  (as in steps 10 and 11). Note that the Model looks a little better.

17. Use the *Delete TIN/Contours* button (or menu *Edit* | *Terrain Modelling* | *Delete Terrain Model / Contours*) to remove contours. This will make the following step easier.

18. Find other features that should be tagged as breakline (), and repeat the steps above. Note that you can use the <shift> click (or click and drag) technique to select more than one feature at a time and then change their properties all at once.

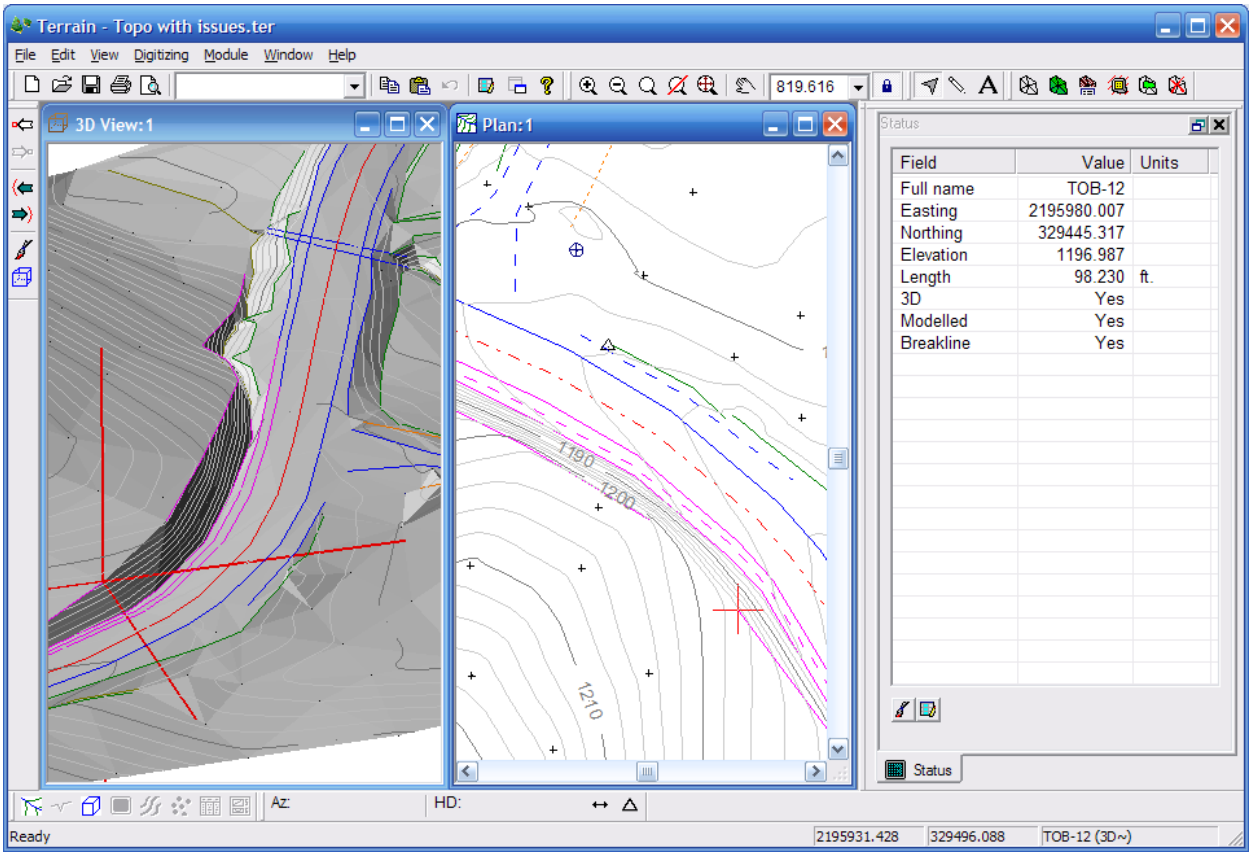


Figure 7.4: Model after key features tagged as breaklines and the model has been recalculated

19. Select menu *File* | *New*; do not save changes.

8. Creating breaklines

We have seen in exercise 3, *Importing ASCII Survey Data*, that breaklines can be created automatically. Sometimes, however, it is easier to simply connect the dots. In this exercise you will add some breaklines to a data set that consists of nothing but points.

To perform this task you will learn about the following Terrain functions:

- Select features by name.
- Join points to create a polyline feature.
- Create a new feature.
- Draw and edit features with the mouse.
- Format feature colors, symbols and line styles.

1. Open the Terrain Module; select menu *File* | *Open*. Browse for file **Examples \DTM\Topo no breaklines.ter**. Press *Open* to load the file (see figure below).

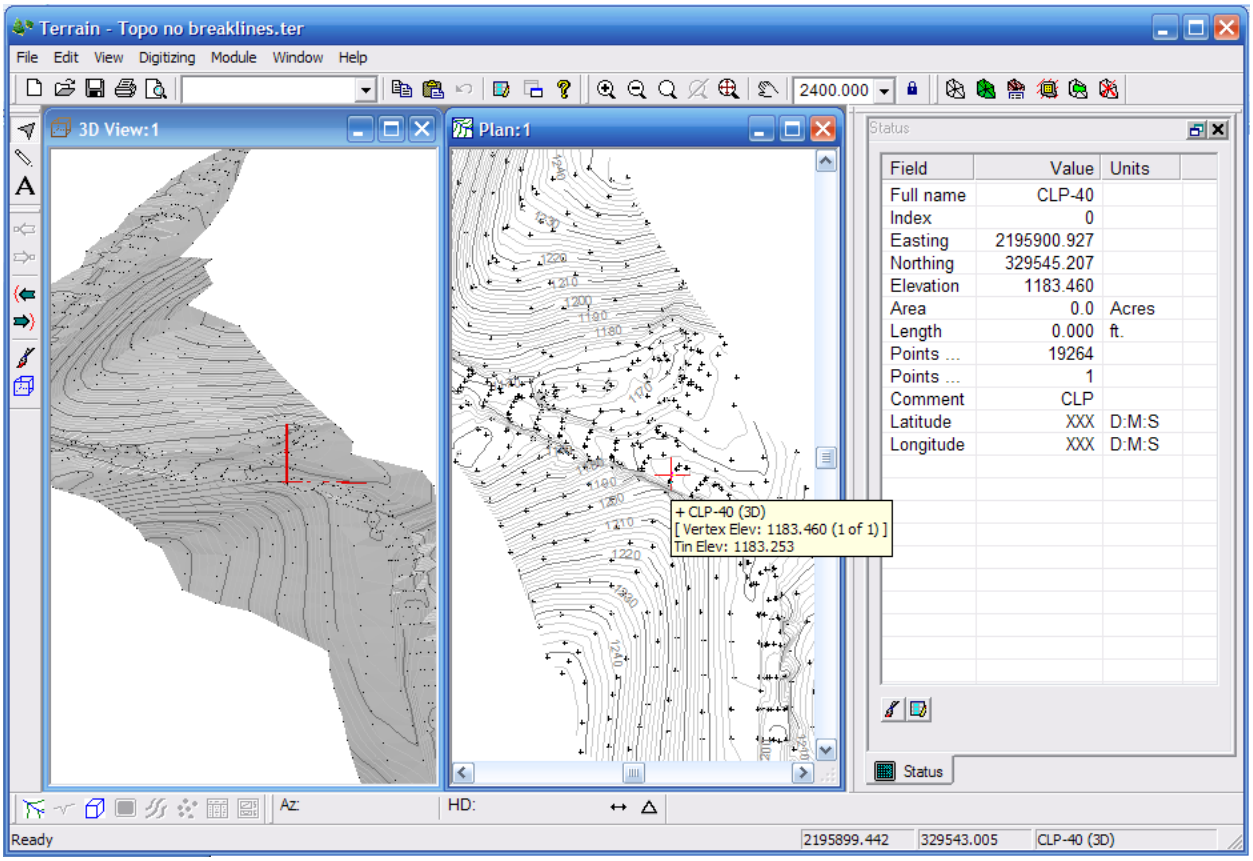


*Figure 8.1: 3D and Plan windows showing file **Topo no breaklines.ter**.*

Notice that the road is not well defined. As shown in the previous exercise, breaklines are required to define the surface realistically. It would also be nice to see other surveyed features like pavement edges and the road center line. Fortunately, the survey data for this file was imported so that features are named by the survey point code (see exercise *Importing ASCII Survey Data*).

Selecting Features by Name

2. Hover your mouse cursor over a point in the Plan window and note the information tooltip window that appears after a moment (see figure above). This is a subset of the Status window information available after you select a point.

3. Zoom in and select or hover over points to find out their names. You will notice that the road center line points are named “CLP”.

4. Select menu *Edit* | *Select Feature(s)* | *By Name* to open the dialog box; press the *Advanced* button to expand the dialog box as shown below.

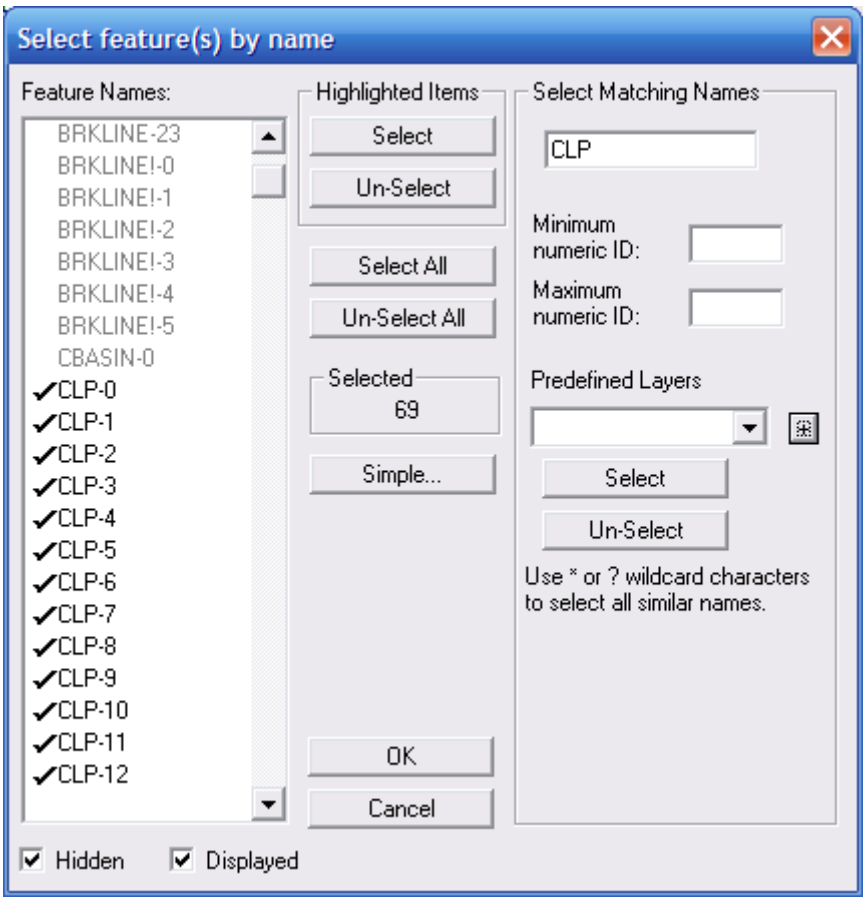


Figure 8.2: Select features by name dialog box with the Select Matching Names area exposed.

5. Press the *Un-Select All* button in the center of the dialog box.

Note: The *Select features by name* dialog box allows you to add/remove feature(s) to/from the existing selection set. This can be very powerful if you want to select a group of features that don’t share the same name. However, most selection operations will start with *Un-Select All* (if the initial number selected is not zero).

6. Type “CLP” at the top of the *Select Matching Names* area and press the *Select* button underneath on the right hand side. Note that the number *Selected* is now 69 and that the CLP items are checked in the list (you may have to scroll down).

7. Press *OK* to accept the change and close the dialog box.

Joining points to create a polyline feature

Now that the CLP points are selected (highlighted magenta) we can connect them together and format the resulting polyline.

8. Use menu *Edit* | *Modify Selected Feature(s)* | *Join*, or <Ctrl-J>, to connect all the CLP points into one polyline feature.

9. When warned that “existing triangles will be cleared” respond *OK*.

Modifying feature formatting

10. Use menu *Edit* | *Modify Selected Feature(s)* | *Linetypes, Symbols*, or <Ctrl-L>, to display the dialog box below.

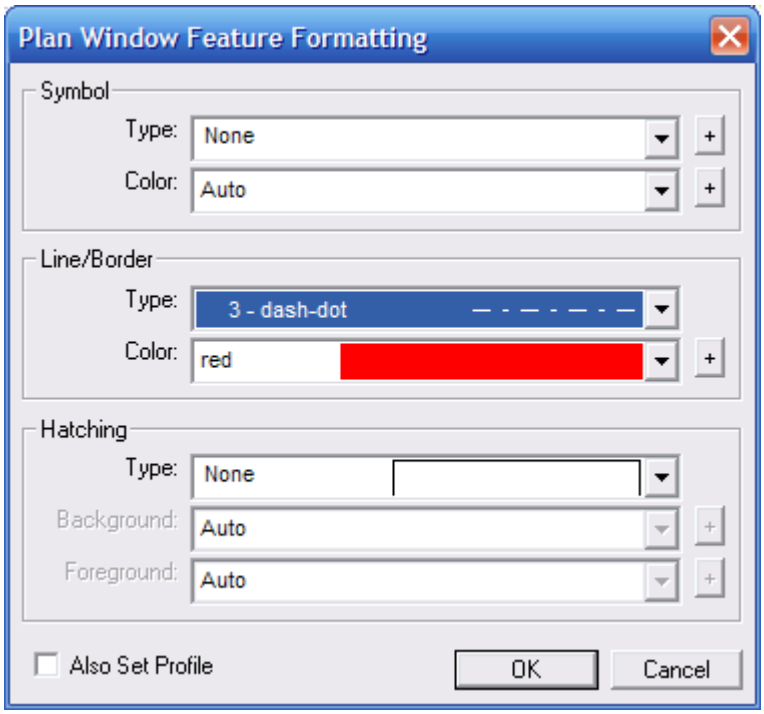


Figure 8.3: Feature formatting dialog box.

11. Change the *Symbol Type* to None, the *Line/Border Type* to dash-dot and the *Color* to red as shown in the figure above.

12. Press *OK* to accept the change and close the dialog box.

The center line is now visible and represented by a polyline as desired. It should also be a breakline as it represents the crown of the pavement.

13. Use the Properties dialog box <Ctrl-E> to tag the CLP feature as a breakline (as in the *Finding and Repairing DTM problems* exercise above).

Now let's try the same process with the pavement edge (EP) points.

14. As in step 4 above, use the *Select features by name* dialog box to select all EP points.

15. Again use <Ctrl-J> to join them. The results are pictured below.

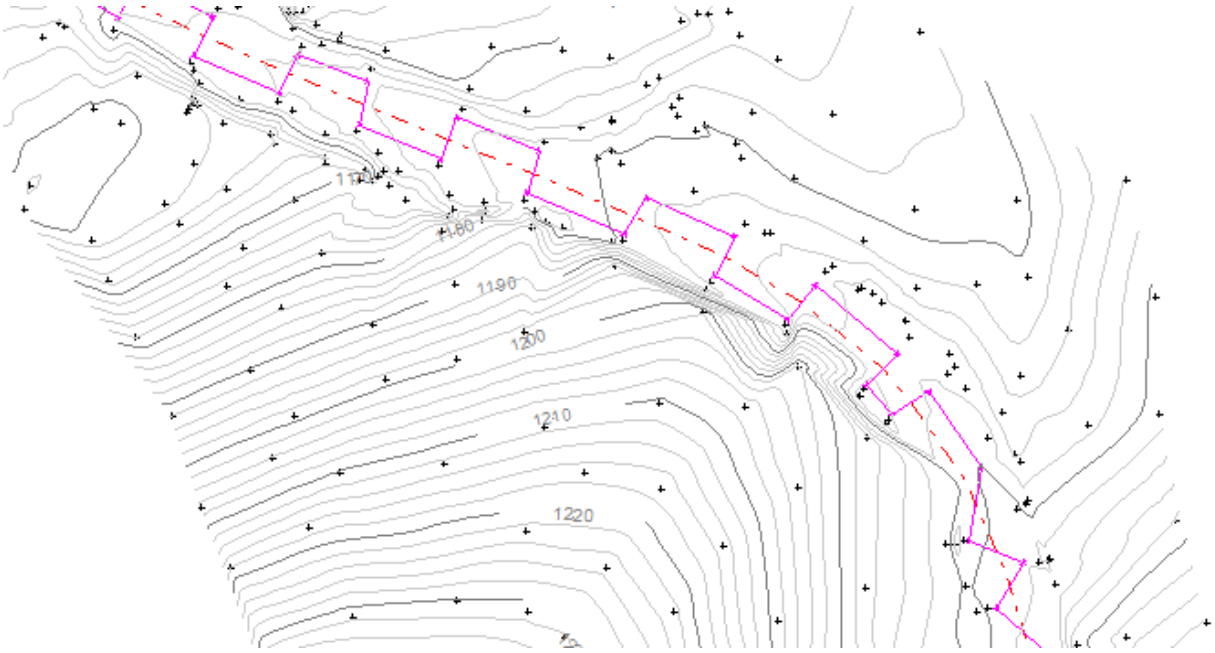


Figure 8.4: Pavement edges connected using the Join function.

The polyline created above connects one side of the road to the other; the join function connects each point to its nearest neighbour. If the points had been coded EPL (left) and EPR (right) then this procedure would have produced satisfactory results (in two operations).

In this case, it is easier to connect the dots. We will make the EP points easy to find and then create a new breakline feature to connect them manually.

16. Use the *Edit* | *Undo Join* menu or <Ctrl-Z> to restore the loose points.

17. As in step 10 above, use the formatting dialog box <Ctrl-L> to change the EP points to have a distinctive color and symbol (as below).

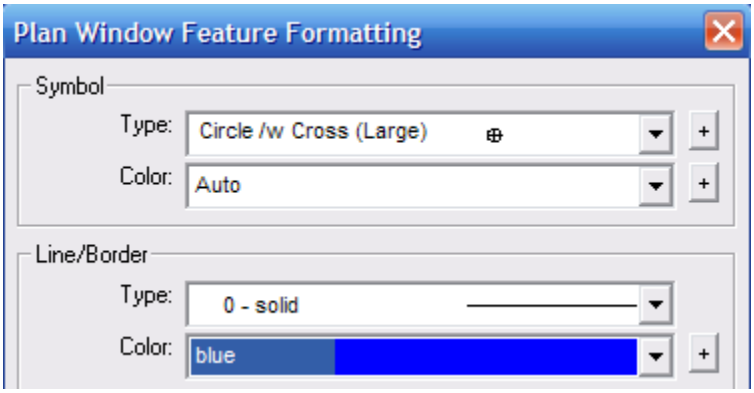


Figure 8.5: Use formatting to make it easy to find points of a given type.

18. Use the *Delete TIN/Contours* button (or menu *Edit* | *Terrain Modelling* | *Delete Terrain Model / Contours*) to remove contours. This will make the following steps easier.

Creating a new feature

19. Use the menu *Edit* | *New Feature* menu to open the now familiar Feature Properties dialog.

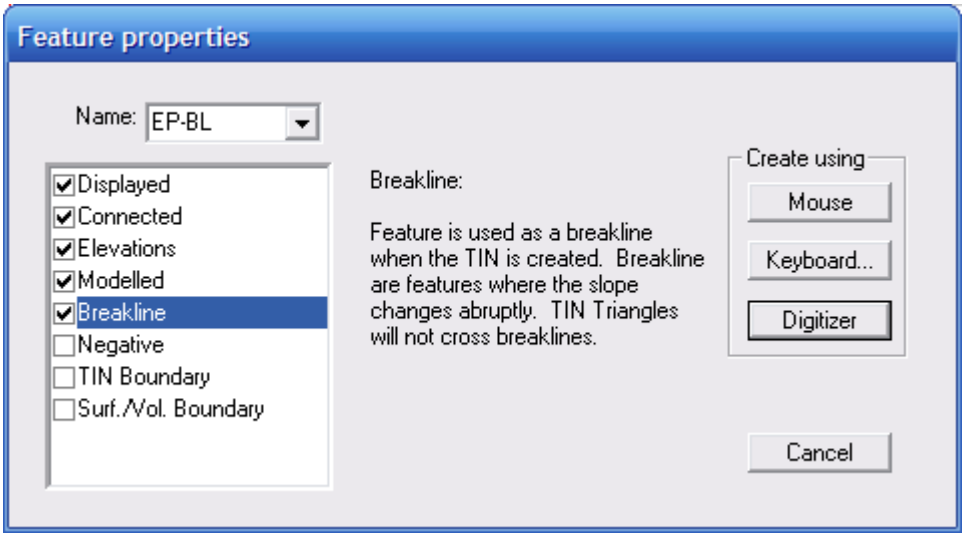


Figure 8.6: The Feature Properties dialog used to prepare a new feature.

20. Change the *Name* to EP-BL and check the *Breakline* property as well as the others shown in the figure above.

21. Press the *Mouse* button to close the dialog box and create the new feature.

22. When you are prompted to define the *Elevation* value, just press *OK* to take the 100ft default. We will be snapping to existing points and picking up their elevations.

Drawing with the mouse

When you are in *Edit/Insert points* mode, the mouse cursor will change to indicate what will happen when you click the mouse.

New point is added at either end of the current feature.

New point is inserted in between existing points of the current feature.

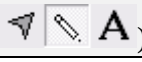
Existing point is captured for editing.

23. Your mouse cursor has changed to a pencil  indicating that you are in *Edit/Insert points* mode. Left click anywhere in the Plan window (mouse down and up again) to create a new point.
24. Move the red cross over an EP point and click a second time to anchor the new point. Note that the *Elevation* shown in the Status window is the elevation of the EP survey point (if it is 100, then the snap failed – you may have been too far from the EP point).

Note: *Snap to Point* is an option set in the Plan window options (right click, *Active Window (Plan) Options, General* tab). Settings like this are saved in the document and in screen layouts.

25. Continue adding points to your new break line:
    a. Click with the pencil  cursor to create a new point.
    b. Move the red cross over an EP point and click a second time to anchor the new point.
26. Try editing a point:
    a. Move your mouse over an existing point in the new feature; note that the cursor changes to a box  .
    b. Click the mouse the capture the point.
    c. Move the red cross to a new position and click a second time to re-anchor the point.
27. Delete a point:
    a. Move your mouse over an existing point in the new feature; note that the cursor changes to a box  .
    b. Click the mouse the capture the point.
    c. Type the <delete> key.
28. Insert a point:
    a. Move your mouse over an existing segment in the new feature; note that the cursor changes to a pencil with a cross  .
    b. Click the mouse to create a new point.
    c. Move the red cross to a desired position and click a second time to anchor the point.

29. Stop when you have done enough points to get the hang of editing with the mouse. Make sure you have tried deleting a inserting points as well as adding new ones at the end of the feature.

Note: you can edit the points of any feature. First select the feature, then right click and select menu *Edit/Insert points with mouse* (you can also choose the pencil button in the Mode tools toolbar ).

Your new feature should look similar to the figure below. Note that the new breakline (EP-BL) is separate from the original survey points (EP) although its vertices share the same coordinates.

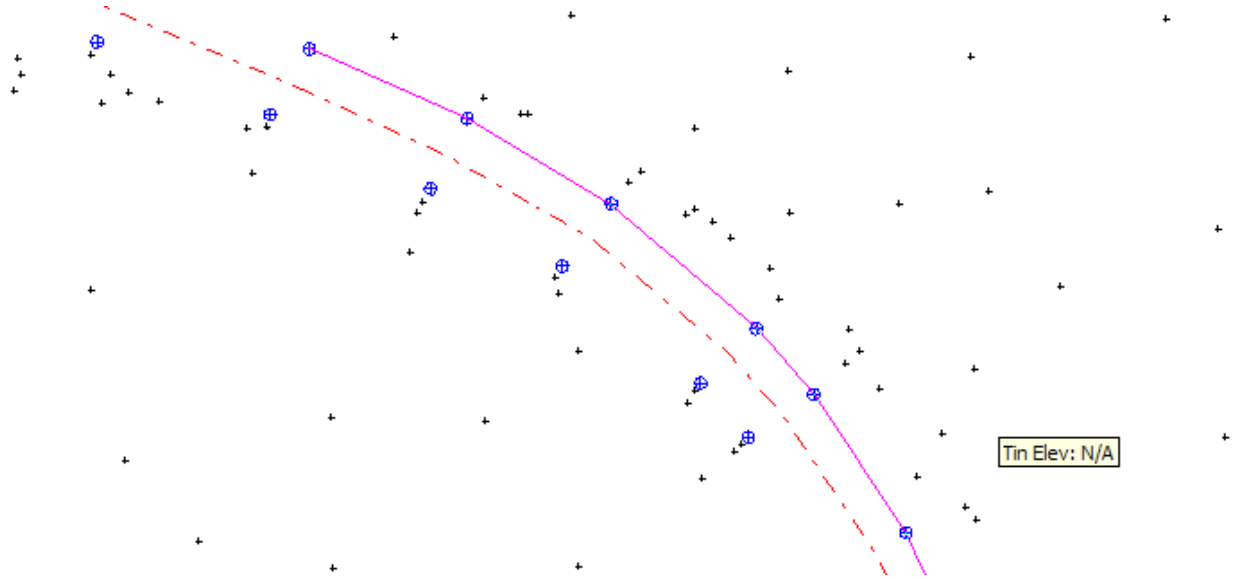


Figure 8.7: New EP breakline feature.

There is another way to connect the dots that is similar to the first method we used to connect the CLP points. You will now create a breakline for the other side of the pavement.

30. Change back to selection mode : right click and select menu *Select with mouse*.

31. Click on one of the EP points to select it (it will turn pink).

32. <Shift> click on the next EP point: hold the <shift> key, left click on the EP point, release the <shift> key. Now two points should be pink.

33. Type <Ctrl-J> to join the two points. Now you have a two point polyline.

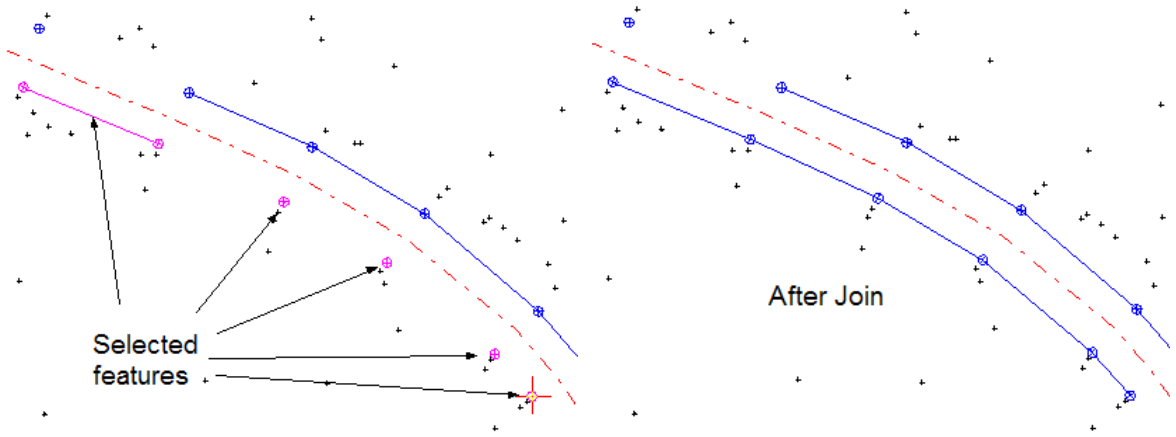


Figure 8.8: Joining points a few at a time. Select a few points (<shift> click), then join them <Ctrl-J>.

34. Make sure the new two-point polyline remains selected and <shift> click to select a few more EP points (left side of figure above). It doesn't matter which order you select the points.

35. Type <Ctrl-J> to join them all into a bigger polyline.

36. Continue this process until you are comfortable with the process. Then set the properties of the new polyline to *Breakline* <Ctrl-E>.

If time permits you may wish to create breaklines for other point types using any of the methods above.

37. Select menu *File* | *New*; do not save changes.

9. New Location Design

In this exercise you will create a short road alignment.

1. Open the Location Module (if you are running the Terrain module, you can use the *Module* | *To Location Design* menu).

2. Select menu *File* | *New* to open the file open dialog box below.

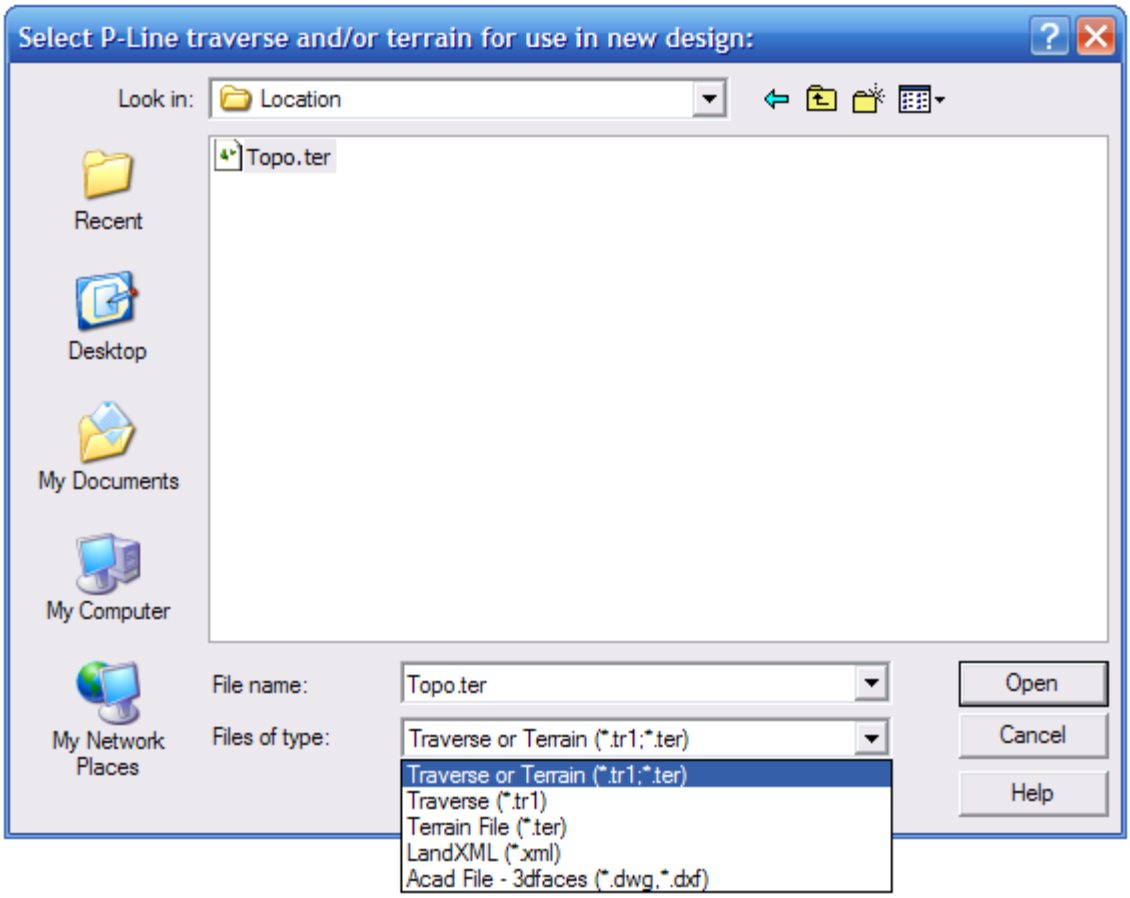


Figure 9.1: File | New first prompts you to define the original ground.

To create a new road alignment in the Location module, you first need to create an original ground DTM. This is usually done by reading survey data into the Terrain module, and then creating a surface with contours (see previous exercises). However, it is possible to import DTM surfaces from other applications by using LandXML or DWG (3d faces) file formats.

3. Select file **Examples\Location\Topo.ter** and press Open. After a moment you will be prompted with the dialog box below.

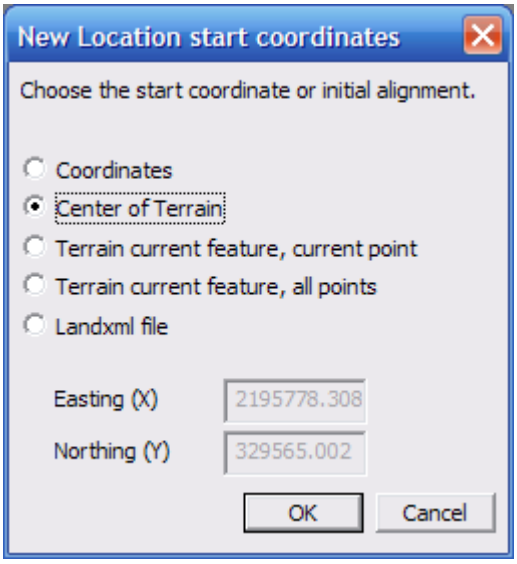


Figure 9.2: The New Location start coordinates dialog box allows you to select a start coordinate or to define an existing alignment.

4. Choose *Center of Terrain* (we will define our start coordinate later) and press *OK* to continue.

The look of your screen depends on the contents of the default *Screen Layout* (**Normal.dlt**) in your RoadEng Settings and Layouts folder (you can find this folder in the *Module* | *Setup* dialog box, *Install* tab).

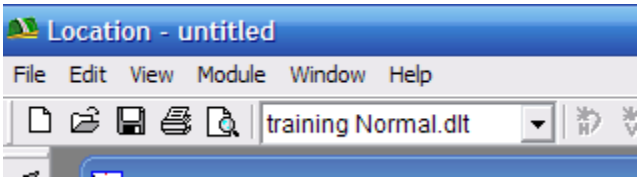


Figure 9.3: The Screen Layout *drop-down in the standard tool bar.*

5. Use the *Screen Layout* drop-down in the standard tool bar (figure above) to open **training Normal.dlt**. This will set up your screen to look like the figure below.

6. If you are working on a training computer or if you have not yet configured your Normal screen layout, you may wish to use the menu *File* | *Save Screen Layout* to over-write **Normal.dlt** with the current settings.

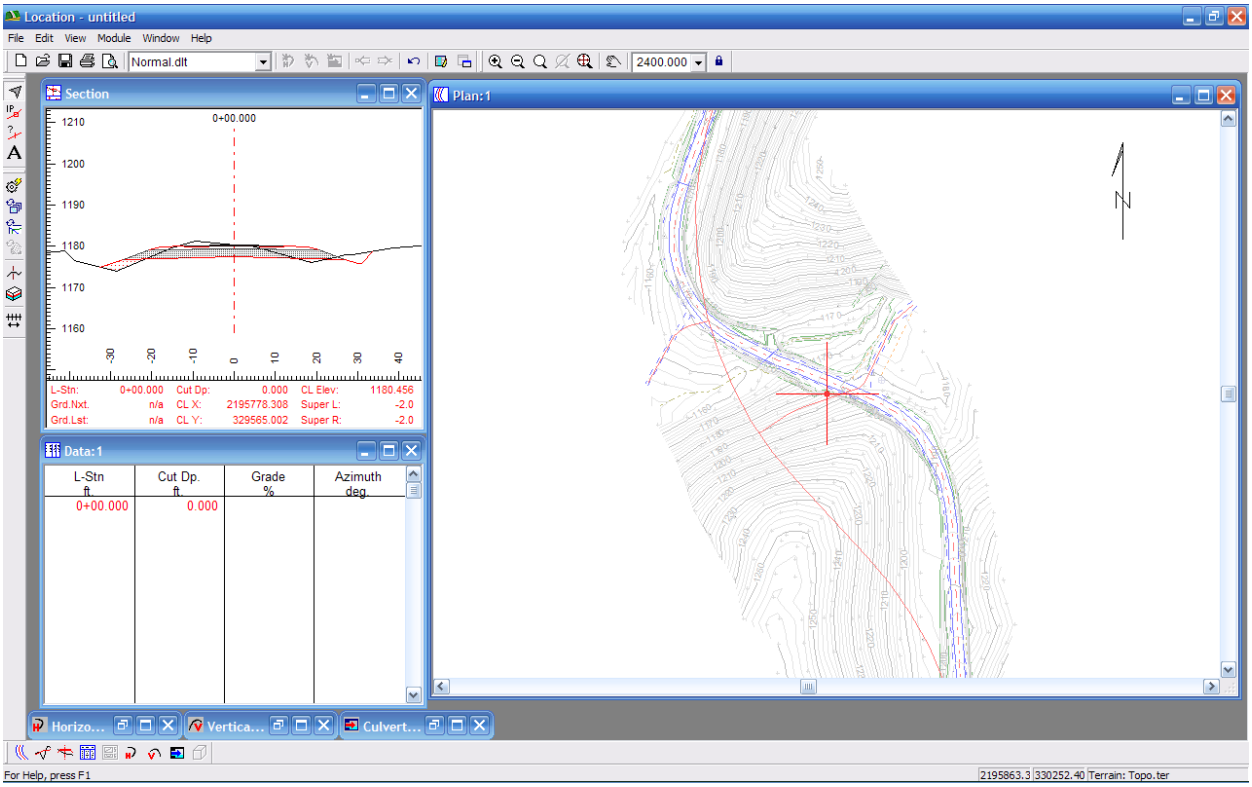


Figure 9.4: The Location module after starting a new alignment.

In the figure above, you can see the original ground DTM in the Plan background; the line work is faded so it doesn't overwhelm the new alignment features. A red line (also in the background) shows the proposed new alignment, mostly to the left of the old road.

The shape of your cross section depends on the contents of the default *Template table* (**Normal.tpl**) in your RoadEng Settings and Layouts folder The next few steps will load templates for this exercise.

7. Choose menu item *Edit /Edit Templates*, or press tool bar button , to open the *Template Table Editor* shown below.

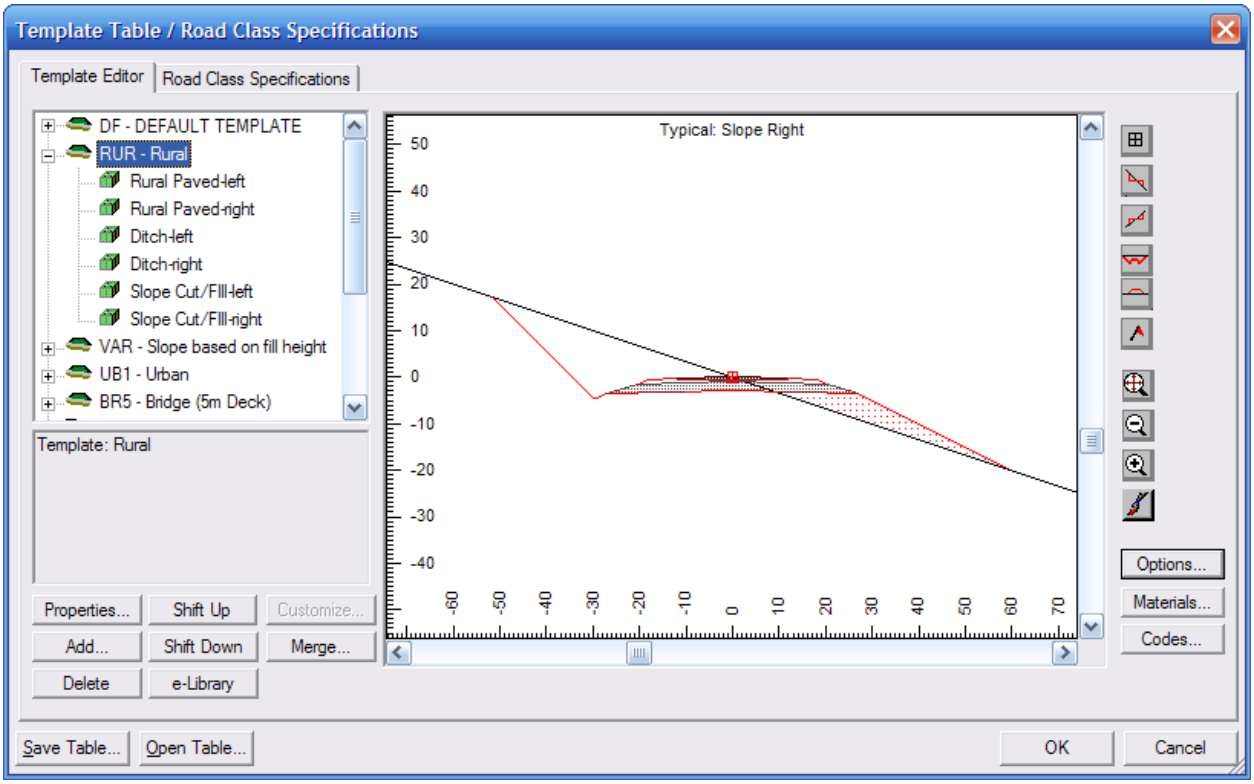


Figure 9.5: Template Table Editor *dialog Box.*

8. Press the *Open Table* button and open **training Normal feet.tpl**.

9. If you are working on a training computer or if you have not yet configured your default templates, you may wish to use the *Save Table* button over-write **Normal.tpl**. Note that templates depend on length units (feet or meters).

10. Press *OK* to close the *Template Table Editor* and *Yes* to recalculate the cross sections.

Templates will be discussed in more detail in future exercises.

Horizontal alignment

In the following steps you will create a horizontal alignment by creating intersection points (IPs) with the mouse. IP editing in the Location module is very similar to point editing in the Terrain module (exercise 8 *Creating breaklines* above).

11. Right click in the Plan window and select the *Add/Edit IP tool* .

12. Edit the start point of the design:
    a. Move your mouse over the existing point (red cross); note that the cursor changes to a box ⌜ ⌝ ⌞ ⌟.
    b. Click the mouse to capture the point.
    c. Move the red cross down to where the proposed alignment leaves the existing road (the Plan window will scroll automatically).
    d. Click a second time to re-anchor the point.

13. Zoom in and fine tune the start point (same process) so that it lies on the old centerline, just south of where the proposed road leaves the old one (figure below left).

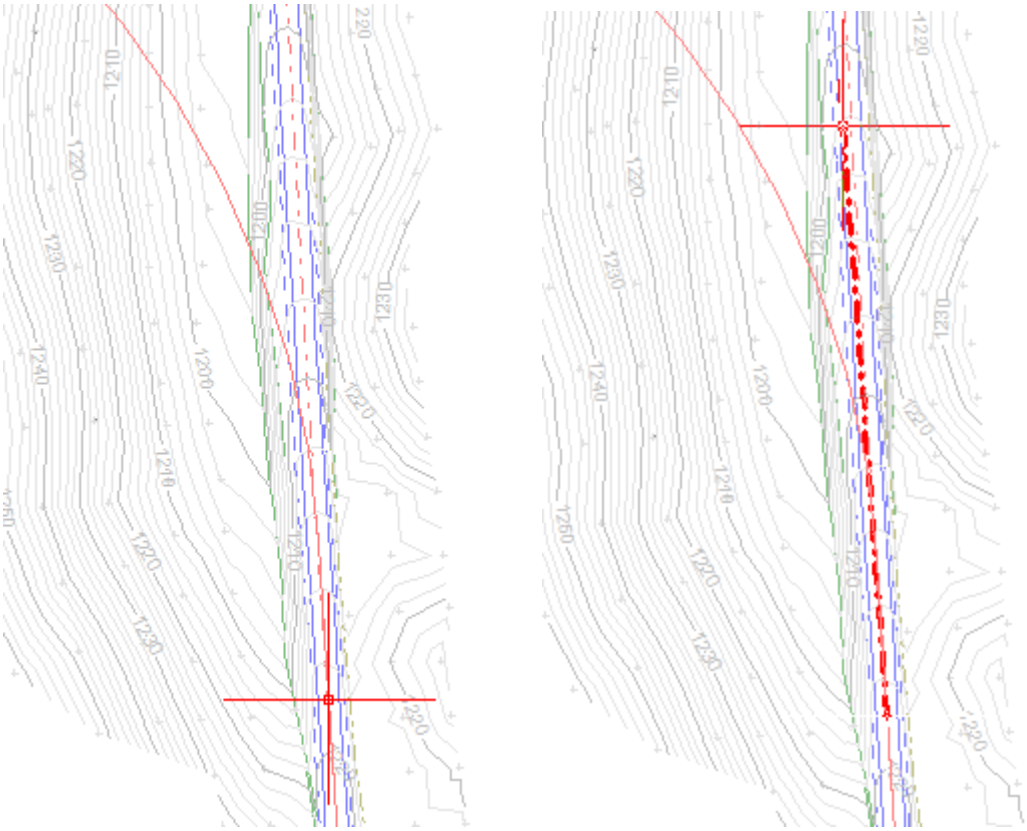


Figure 9.6: Drawing the horizontal alignment.

14. Create a new IP:
    a. Click with the pencil cursor (away from the existing point) to create a new point.
    b. Move the red cross to the position shown in the figure above right (approximately).
    c. Click a second time to anchor the new point.

15. Edit your IP:
    a. Move your mouse over an IP; note that the cursor changes to a box.
    b. Click the mouse to capture the IP.
    c. Move the red cross to a new position and click a second time to re-anchor the IP.

16. Insert an IP:
    a. Move your mouse over a segment between IPs; note that the cursor changes to a pencil with cross.
    b. Click the mouse to create a new IP.
    c. Move the red cross to the desired position (not important) and click a second time to anchor the IP.

17. Delete an IP:

a. Move your mouse over the IP created above; note that the cursor changes to a box ⌜ ⌝ ⌞ ⌟.
b. Click the mouse to capture the IP.
c. Type the <delete> key to remove the IP.

18. Continue this process until you have an alignment similar to the one in the figure below on the left.

Note: Don't forget about the *Edit* | *Undo* command, <Ctrl-Z>.

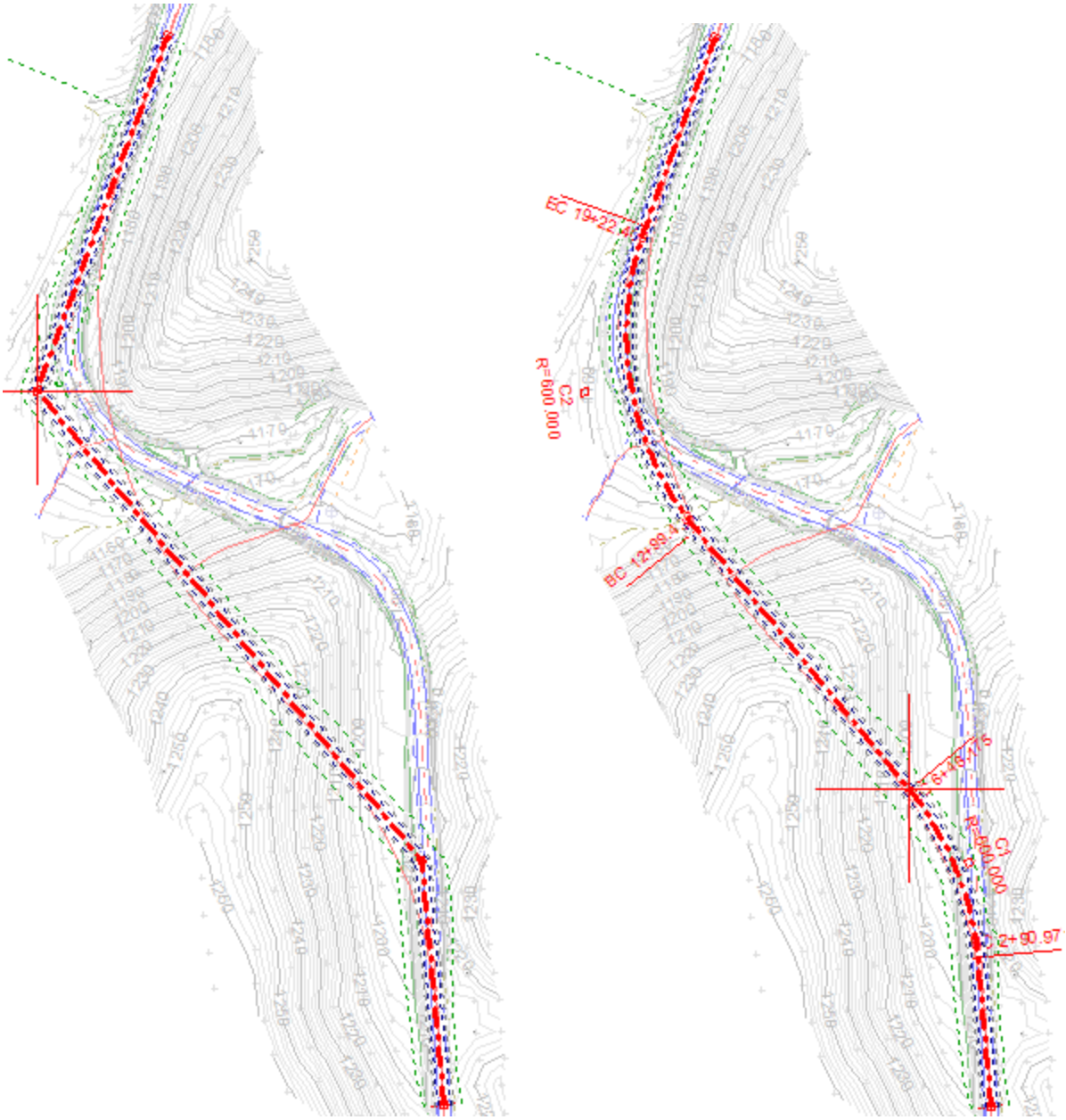


Figure 9.7: Horizontal alignment before and after curves applied.

Adding horizontal curves

To create a horizontal curve, you identify an IP then you define a curve between the tangents it defines. Horizontal curves are created and edited using the *horizontal curve panel*.

19. Restore the Horizontal curve panel using the *Window Tools* toolbar in the lower left part of the screen (figure below)

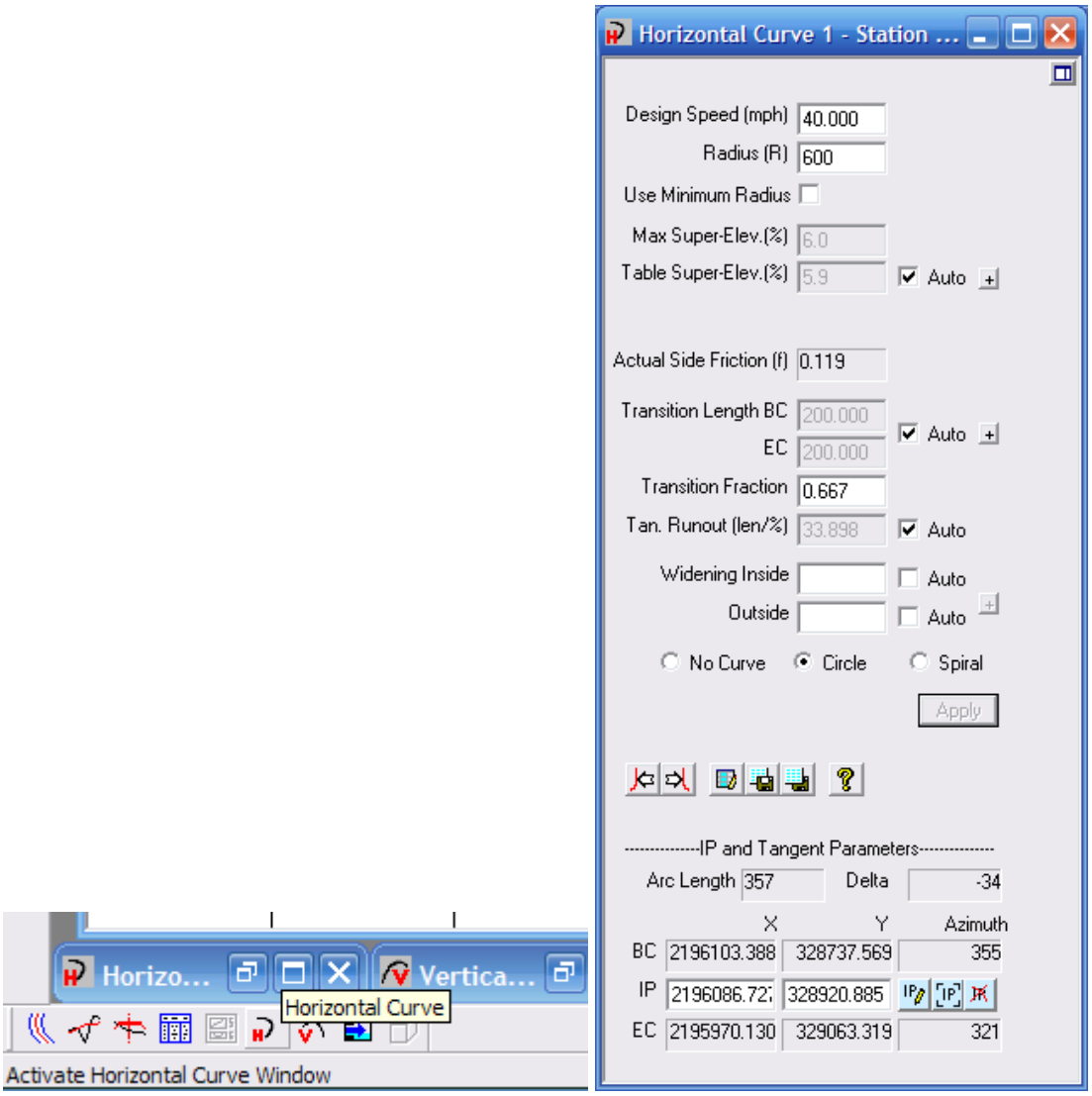


Figure 9.8: The Window Tools toolbar (left) allows you to restore the Horizontal Curve panel (right).

20. Use the *Previous IP* or *Next IP* button to move to the second IP in the alignment (watch the Plan window).

21. Press the *Get Default Curve* button to set up the parameters as shown in the figure above.

Note: The curve panel controls are disabled until the current is point is an IP between two tangents. Most of the controls are still disabled until you either select *Circle* or *Spiral* or press the *Get Default Curve* button .

Note: Default curves and associated tables are stored with your template table. The default template table is Normal.TPL.

22. Press the *Apply* button to create the first curve.

23. Use the *Next IP*  button to move to the third IP in the alignment and repeat to the steps above to create the second curve.

24. Return to the Plan window and edit one of the curve IPs. Notice that when the IP is captured, the curves update dynamically as you move the mouse. Also notice that you aren't allowed to overlap the curves or to push a curve off the end of the road.

Note: The current cross section is shown in the Plan window as a red cross. When you have finished editing a curve, the current cross section is the End Curve (EC) point.

25. Go back to the curve panel and increase the radius of the two curves:
    a. Use the *Previous IP*  or *Next IP*  button to select a curve.
    b. Type a new radius (it must be small enough to fit or you will see an error message).
    c. Press the *Apply* button.

26. Modify the IP locations and continue adjusting the radius and see if you can get the new alignment to line up with the proposed alignment in the background.

27. *File* | *Close*, do not save changes.

10. Vertical Alignment

This exercise follows on from the previous. You must create a horizontal alignment before you can create vertical alignment.

In the following steps you will create a vertical alignment by creating vertical intersection points (VIPs) with the mouse. VIP editing in the Profile window is very similar to IP editing in the Plan window.

1. Use menu *File* | *Open* to open **Examples\Location\ Align stage 1.dsn.** This should look just like your design at the end of the previous exercise.

2. Use the *Screen Layout* drop-down in the standard tool bar (Figure 9.3 above) to open **training Profile.dlt**. This will set up your screen to look like the figure below.

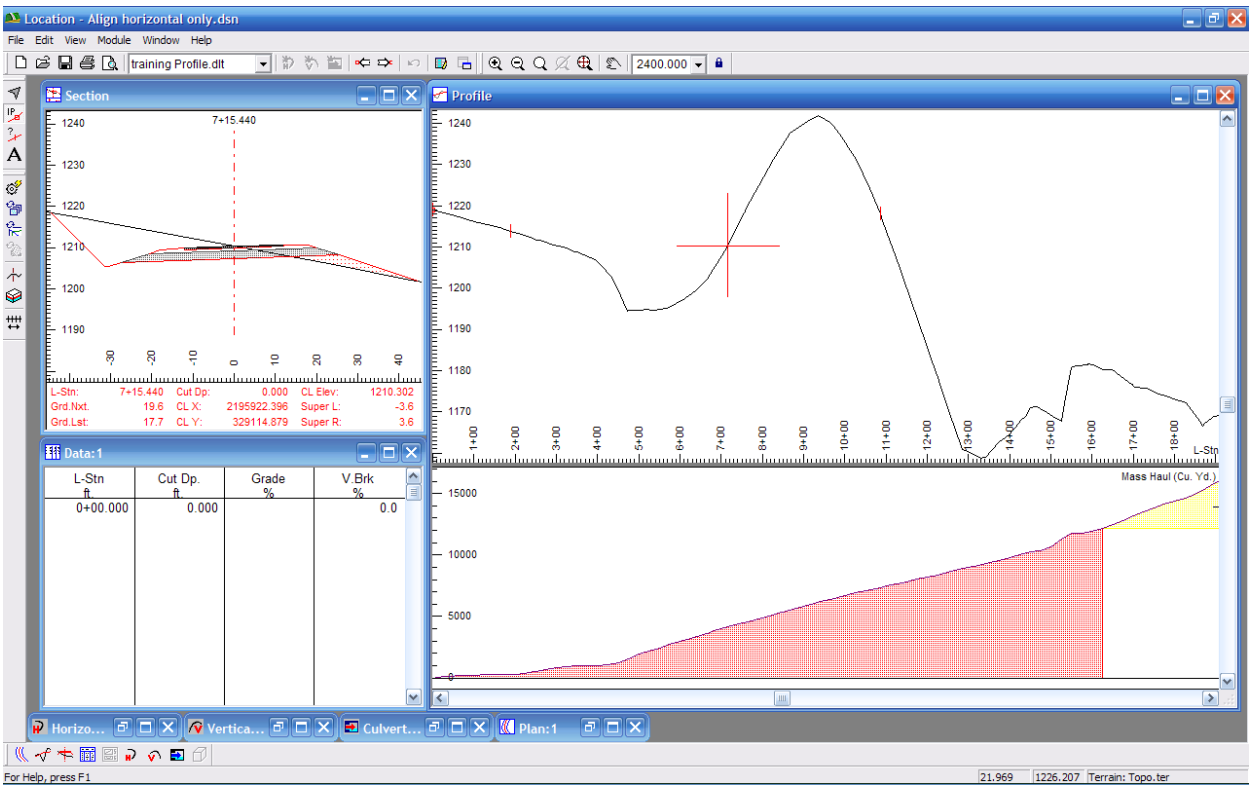


Figure 10.1: Location design with no vertical alignment. The Profile window shows the ground line only.

3. Right click in the Profile window and select the *Add/Edit IP tool* .

4. Click anywhere to the right of station 0+00 to create a new VIP.

5. Move the captured point to a desired position and click again to anchor the point

6. As in the previous horizontal alignment exercise, use the mouse to
    a. Create a new VIP at the end of the existing alignment.
    b. Edit an existing VIP.
    c. Insert a VIP between existing VIPs.
    d. Delete a VIP

There are a few subtle differences between editing in the Plan and Profile windows:

- In the profile, you can't have a backwards segment (if you insert a point between two existing VIPs, you are restricted to that station range).
- In the profile, you can insert a point no matter where your mouse is (in the Plan you must mouse over a segment).
- Profile editing is constrained by the length of the horizontal alignment (if you remove one end of your horizontal alignment, you will generally remove some vertical alignment).

7. Continue editing the Vertical Intersection Points until you have a design similar to the one in the figure below.

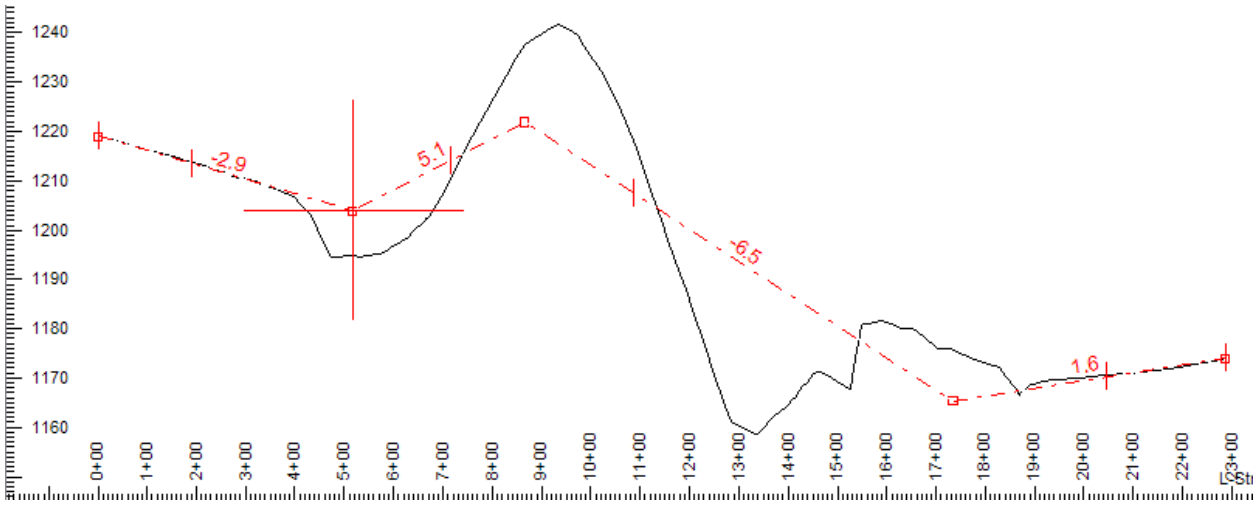


Figure 10.2: Vertical alignment before curves applied.

Adding vertical curves

Vertical curves are very similar to horizontal curves: you first identify a VIP, and then you define a curve between the tangents it defines. Vertical curves are created and edited using the *vertical curve panel*.

8. Open **Align stage 2.dsn** (do not save changes), so the following steps behave correctly.

9. Restore the Vertical curve panel (figure below)

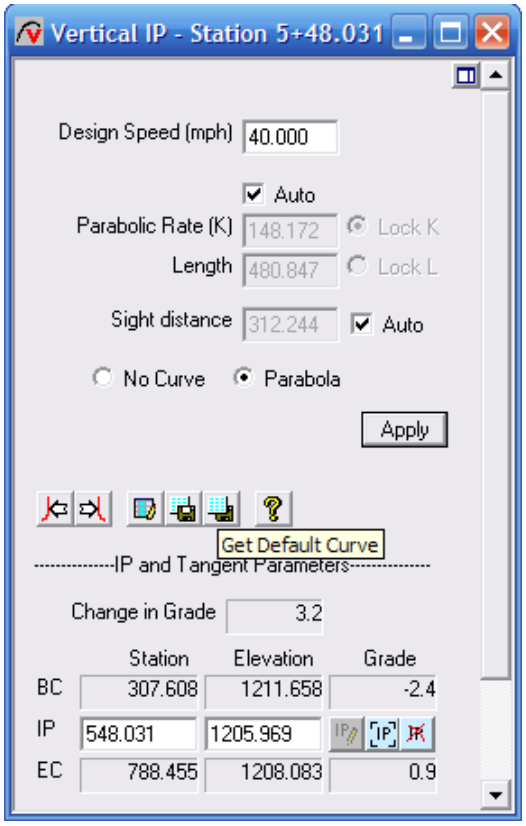


Figure 10.3: Vertical curve panel with automatic curvature selected.

10. Use the *Previous IP* or *Next IP* button to move to the second VIP in the alignment (watch the Profile window).

11. Press the *Get Default Curve* button. Then Press the *Apply* button to create the first curve.

12. Use the *Next IP* button to move to the third VIP in the alignment.

13. Again press the *Get Default Curve* button. Notice that you are warned that the curve does not fit the tangents and the *Apply* button is disabled (figure below).

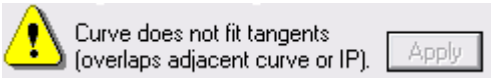


Figure 10.4: Curve is too long for tangents.

Note: When a curve does not fit tangents (vertical or horizontal), you can either:
Shorten the curve.
Shorten the previous and/or next curve.
Move intersection points to reduce the angle between tangents or to lengthen the tangents.

In this case, as the curve is already defined as the shortest curve allowed for 40mph (*auto* is checked), the only way to shorten the curve (without changing VIP locations) is to reduce the speed.

14. Change the Speed value to 20; notice that the error goes away.

This speed is too slow (the whole point here is to increase the design speed of this section of road). Instead we will lower the third VIP elevation to reduce the grade change.

15. Go to the Profile window and move the third VIP down a bit.

16. In the Vertical curve panel, again press the *Get Default Curve* button  . Did the error go away? If not, move the IP down a little further and try again.

17. When the 40 mph curve fits, *Apply* it.

18. Move ahead and apply the same curve to the fourth VIP.

19. Return to the Profile window and edit the third curve VIP. Notice that when the IP is captured, the curves update dynamically as you move the mouse. Also notice that you aren't allowed to overlap the curves or push a curve off the end of the road.

Note: The current cross section is shown in the Profile window as a red cross. When you have finished editing a curve, the current cross section is the End Vertical Curve (EVC) point.

20. Continue editing the all the VIPs until you get this middle point as high as possible. (figure below). Keep the first and last segments tangent to the original road.

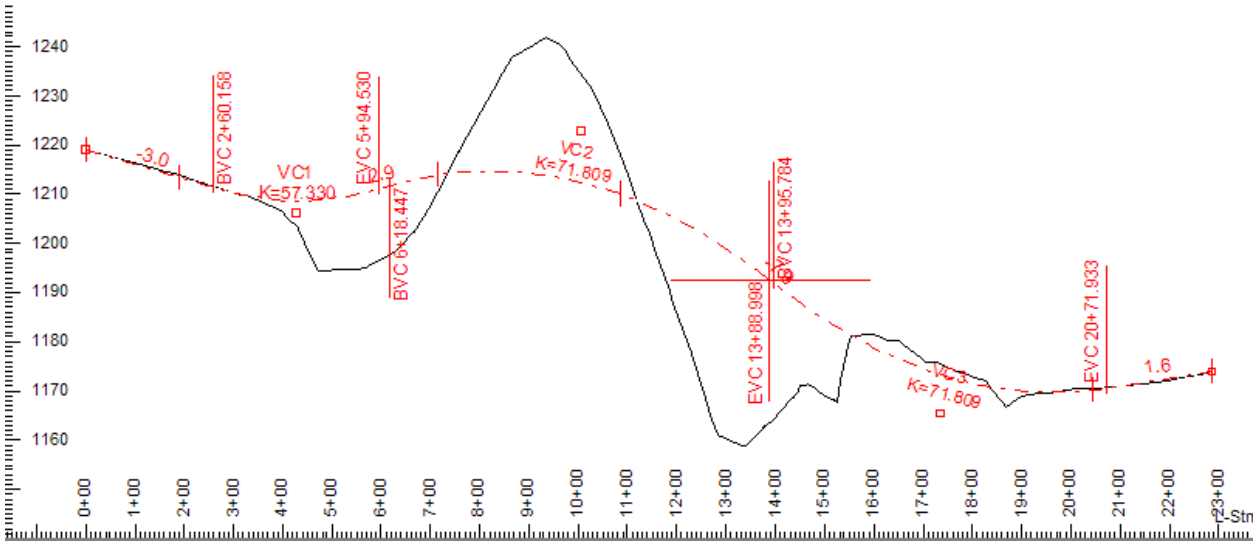


Figure 10.5: Vertical alignment after curves applied.

You should also try to make the mass haul balanced. We will cover mass haul in more detail in following exercises.

21. *File* | *Close*, do not save changes.

11. The Mass Haul diagram

The *Mass Haul* diagram gives you quick, qualitative information about cut and fill volumes and movements. In this exercise, you will explore the options available for configuring this graphic.

Mass Haul is a graphic representation of accumulated volume; at any station, the value is the accumulated *cut volume* minus the accumulated *fill volume* up to that point. The difference in Mass Haul between two points indicates the volume of surplus (positive difference) or deficit (negative difference).

As of this printing, all sub-grade material is included in the mass haul calculation regardless of material type. For example, if you are cutting only silt and filling only shot rock, your mass haul may still show a balance.

1. Use menu *File* | *Open* to open **Examples\Location\ Align stage 3.dsn.**

2. Use the *Screen Layout* drop-down in the standard tool bar (Figure 9.3 above) to open **training Profile only.dlt**. This will set up your screen to look like the figure below.

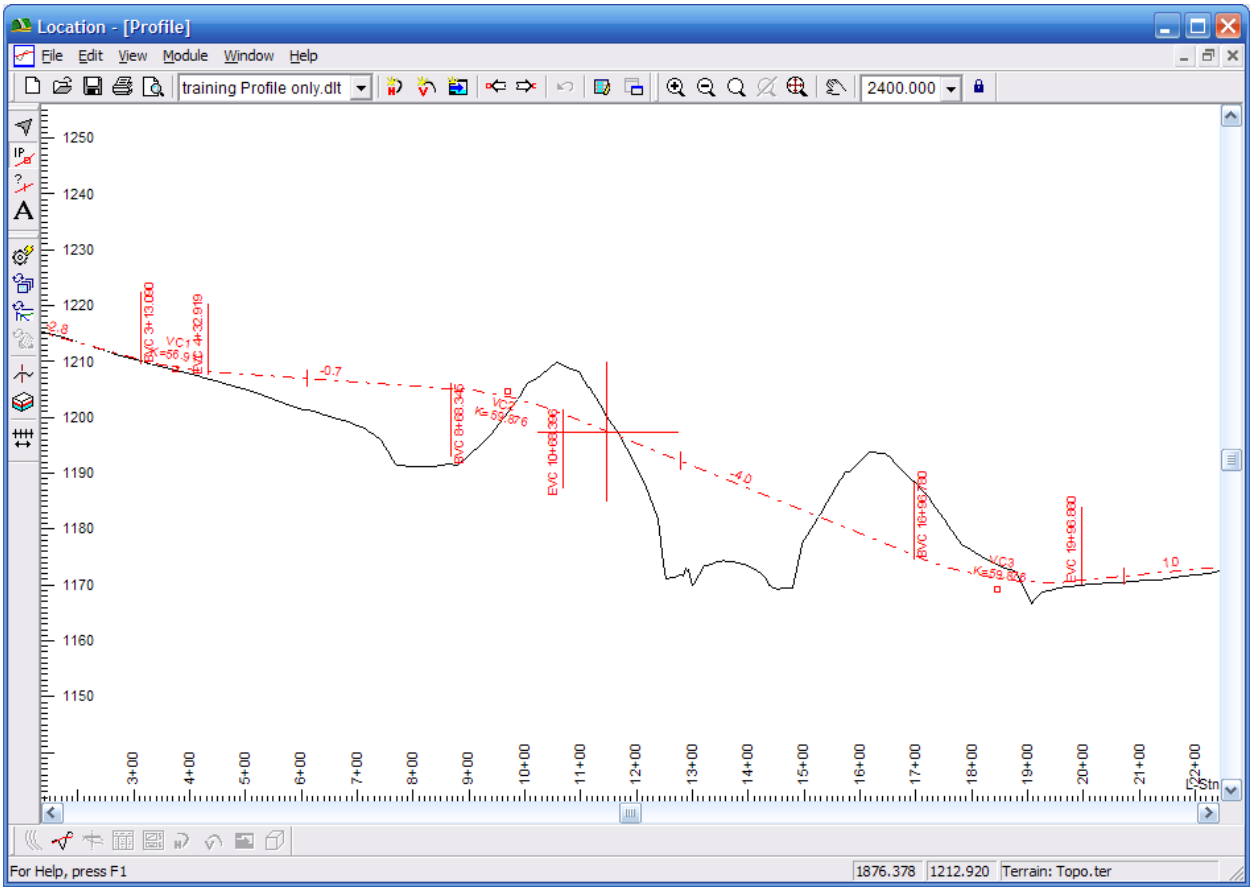


Figure 11.1: Location design after opening ***training Profile only.dlt*** *screen layout* .

3. Right click in the Profile window and select *Profile Options* to open the dialog box shown below left.

4. Press the *Select* button at the bottom of the *Sub-Windows* area. This will display the two list dialog box shown below right.

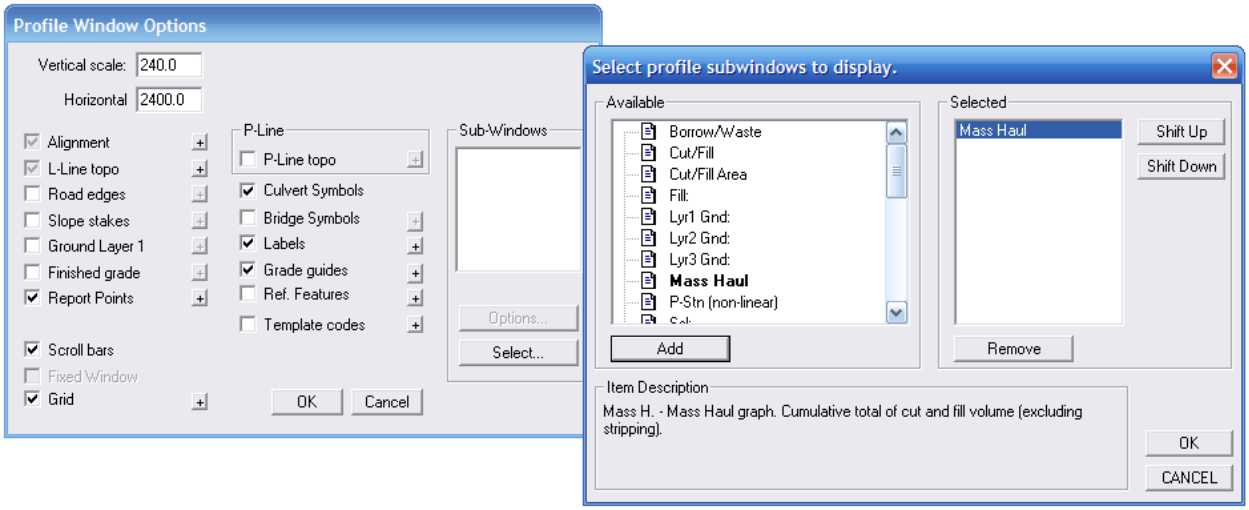


Figure 11.2: Select sub-windows *for the profile window.*

5. Select *Mass Haul* on the left and click *Add* (or double click) to add it to the *Selected* list (figure above).

Note: The profile sub-windows area can display multiple items. All sub-windows will share the same horizontal axis (station) with the profile window.

6. Press *OK* and *OK* again to accept changes and close the dialog boxes.

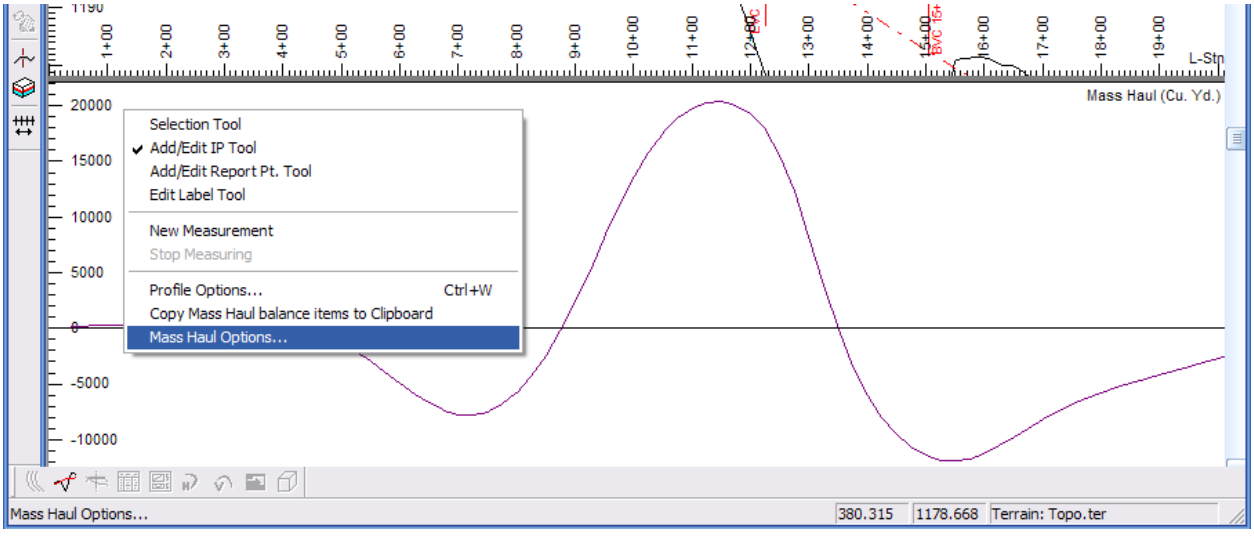


Figure 11.3: Mass haul displayed in a profile sub-window.

7. Move your mouse over the divider between the main profile window and the mass haul; when it changes to the sizing cursor ↕, click and drag up to make more room for the mass haul.

8. Right click in the mass haul window and select menu *Mass Haul Options* to open the dialog box shown in the figure below.

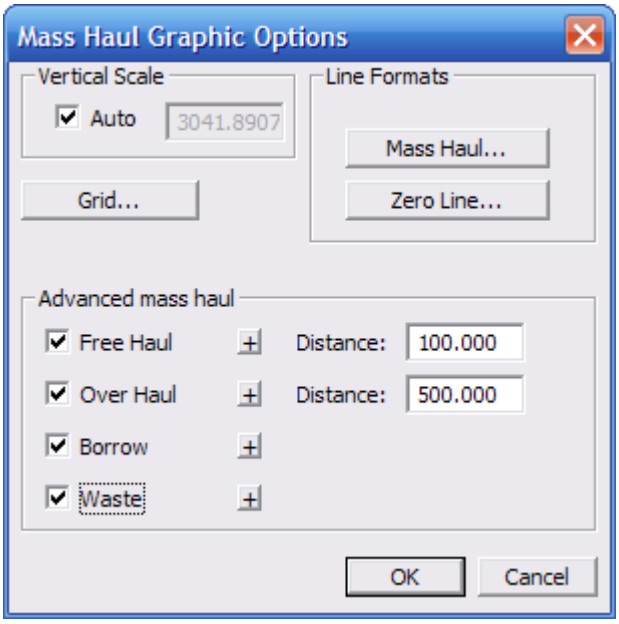


Figure 11.4: Mass Haul *options with all features turned on.*

The concepts behind the mass haul diagram are discussed in detail in the help document.

9. Type <F1> and read the help text if you are unfamiliar with terms such as *Free Haul*, *Over Haul*, *Borrow* and *Waste*. Close the help window when you are done.

10. Press the *Grid* button to display the common grid and axis label control. Notice that the horizontal axis is disabled – it would be identical to the Profile axis and therefore redundant.

11. *Cancel* to close the grid options.

The *Mass Haul* and *Zero Line* buttons allow you to control the line style and color of the basic graphic items (shown in Figure 11.3)

12. Turn on all four *Advanced mass haul* items (figure above).

The *Free Haul Distance* and *Over Haul Distance* are controlled by the fields on the right.

The  button beside each item allows you to control the hatching style and color. The hatching in the figures below was chosen for black and white printing; you don't need to change the current values.

13. Press *OK* to accept changes and close the options dialog box.

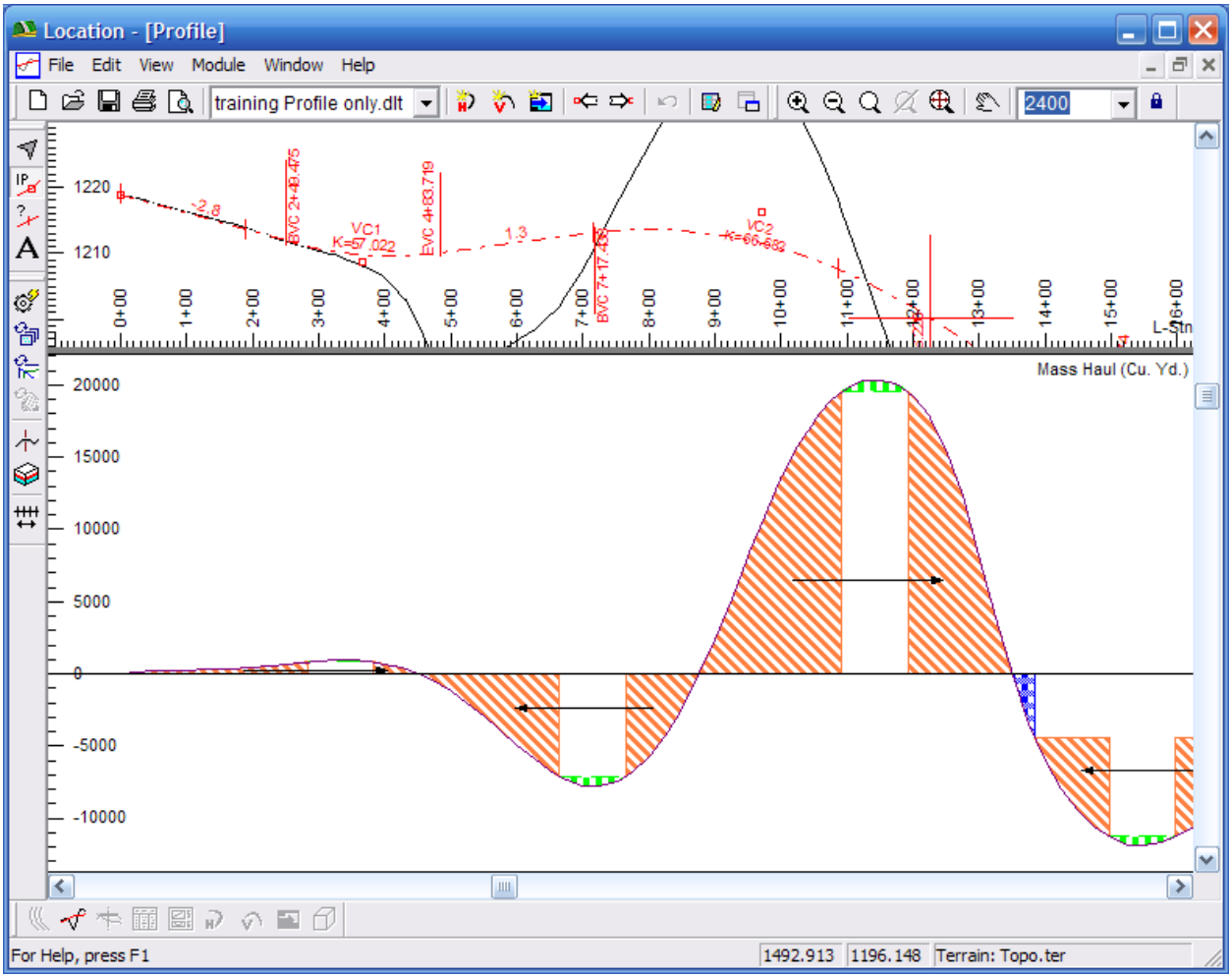


Figure 11.5: Mass haul with advance features turned on. (Note that the hatch styles are different from the ones on your screen so that we can print this document in black and white)

	Free Haul	Material which is pushed or pulled a distance less than the *Free Haul distance* (100 ft.).
	Over Haul	Material moved beyond *Free Haul Distance* (100 ft.) and less than the *Overhaul Distance* (500 ft.).
	Borrow	Material which must be trucked in from outside the road project.
	Waste	Material which must be trucked outside the road project (End haul).

It is possible to modify the mass haul to account for borrow and waste.

14. Choose menu *Edit* | *Assign Parameters by Range* (tool bar button). Select the *Borrow/Waste* tab (figure below).

15. Enter 450 for the *Station* and 8000 for the *Volume*. Press the *Add/Update* button.

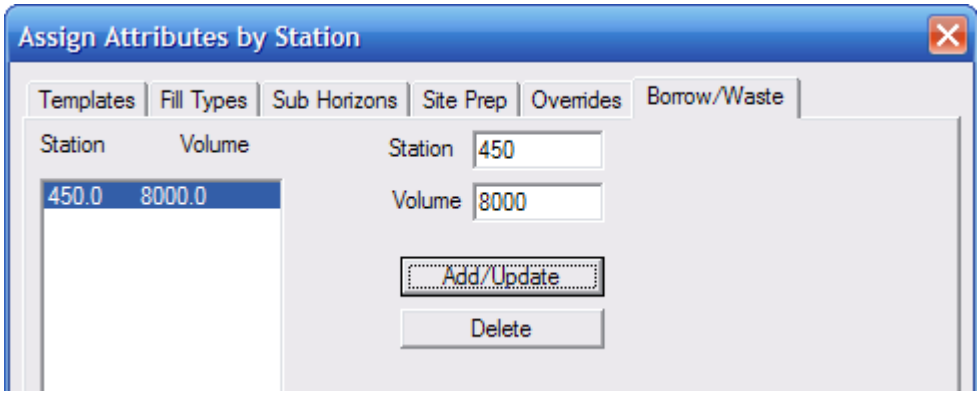


Figure 11.6: Earthwork Borrow/Waste dialog box.

16. Press *OK* and respond *Yes* to the "Recalculate road alignment?" prompt.

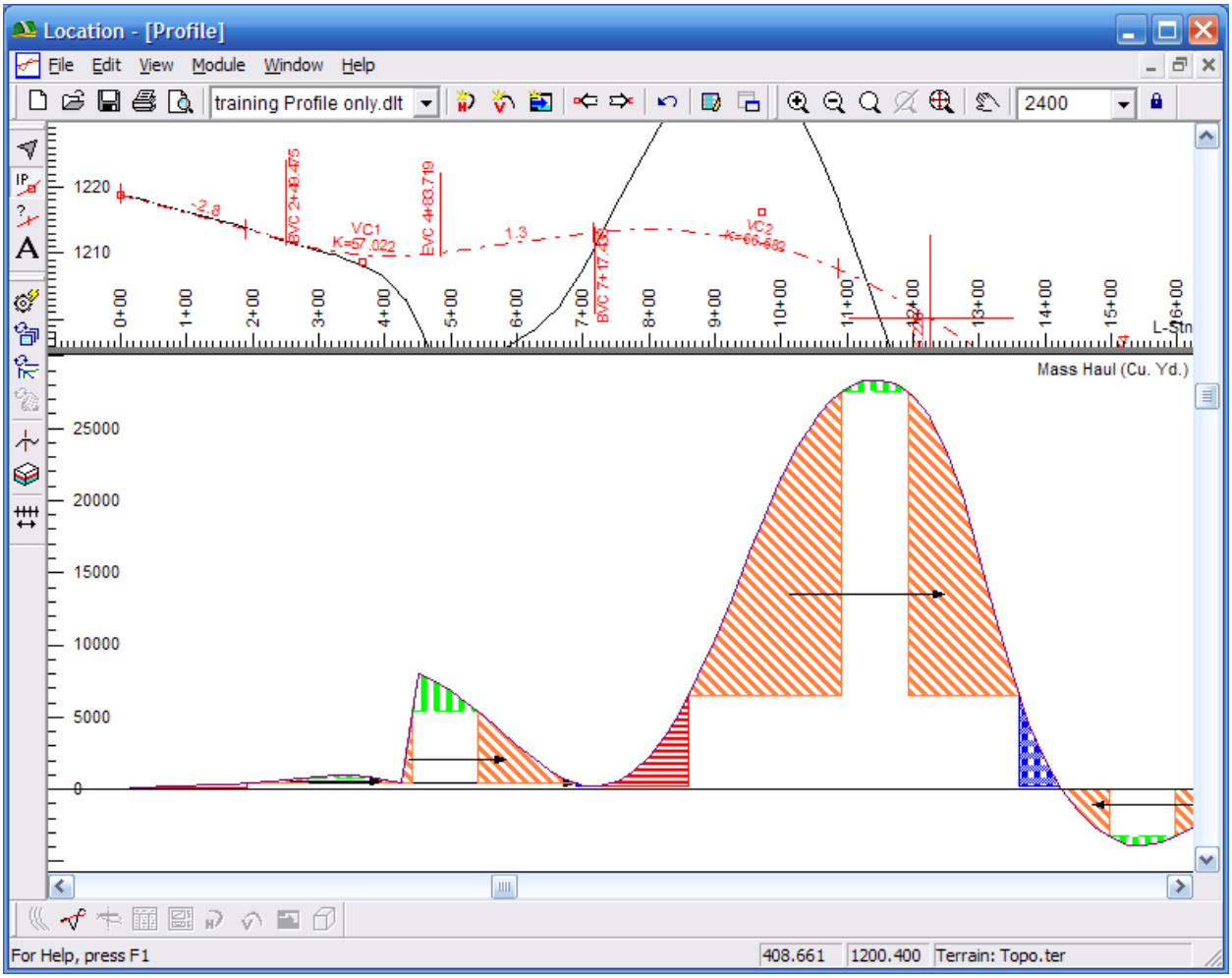


Figure 11.7: Mass Haul after 8000 cubic yards borrowed at station 4+50. Now there is no need to pull *material back across the gully.*

17. File | Close. Do not save changes.

12. Volume Optimization

In this exercise we will continue working on the realignment design from the previous exercises. This project has the following goals and constraints:

- Realign less than 2500 feet of road.
- Increase design speed to 40 mph.
- Grades less than 8%.
- Tangent to existing road (vertical and horizontal).
- Balance sub-grade volumes.
- Minimize cut and fill volumes.
- Minimize right of way (road foot print)
- Template constraints:
    - Lane width.
    - Shoulder width.
    - Material types.
    - Material thicknesses.
    - Cut and fill slopes
- Vertical Curve constraints:
    - Design speed.
    - Sight Stopping distance.
- Horizontal Curve constraints:
    - Design speed.
    - Super-elevation.

Our goal here is to adjust both vertical and horizontal alignment within the constraints given so that we minimize and balance volumes. We start by assuming that the template given is correct.

1. Use menu *File* | *Open* to open **Examples\Location\ Align stage 3.dsn**.

The outer most lines on either side of the alignment are the *slope stakes* or catch points (configurable in the *Plan Options* dialog box). The area inside the slope stakes is the footprint of the road that must lie within the right of way.

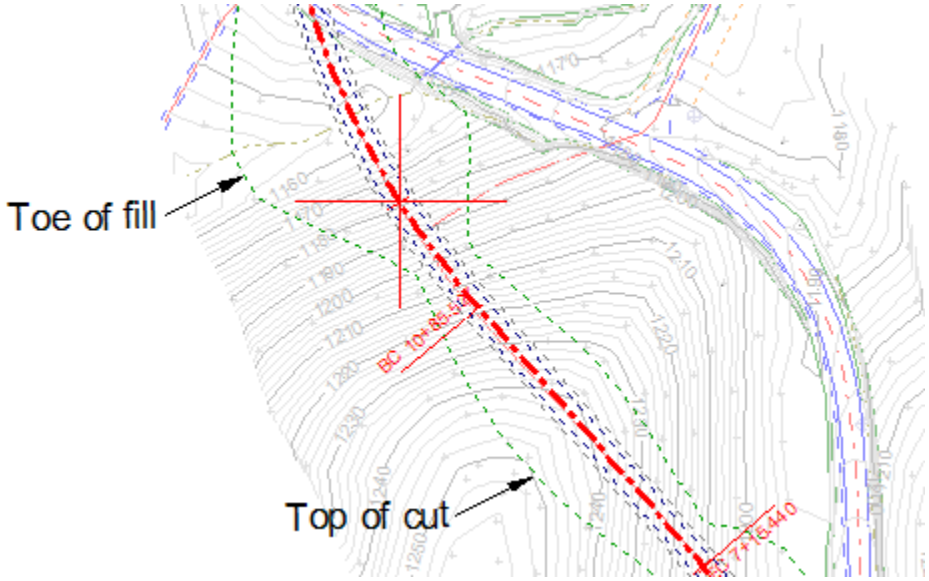


Figure 12.1: Green dot slope stake (catch point) lines in the plan indicate the footprint of the design.

2. Bring the Profile widow to the front and notice shape of the ground and the quantities indicated by the mass haul diagram (figure below).

3. Enter selection mode (right click choose *Selection Tool*) then click on the alignment to view a cross section in the middle of the through cut (red cross in the profile shown below).

Note: You can change the cursor to selection mode temporarily by holding down the <Ctrl> key.
Note: You have found a cross section to select when the cursor has a plus symbol attached .

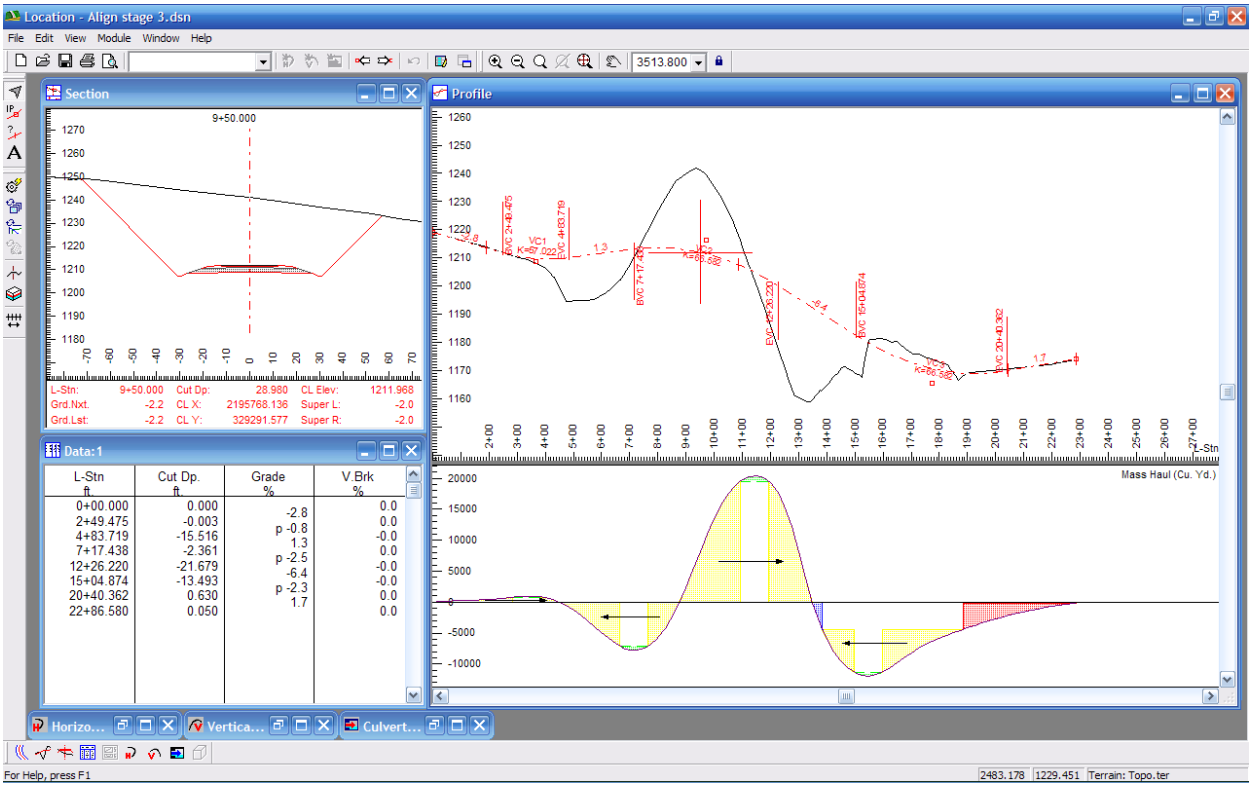


Figure 12.2: The road passes through a hill and therefore has large cut volumes.

4. Bring the Plan window to the front; you can see the contours representing the hill under the red cross.

5. Restore the horizontal curve panel and navigate to the first curve.

6. Check the *Use Minimum Radius* check box (figure below). Notice that the radius can be reduced to 600ft without reducing the design speed.

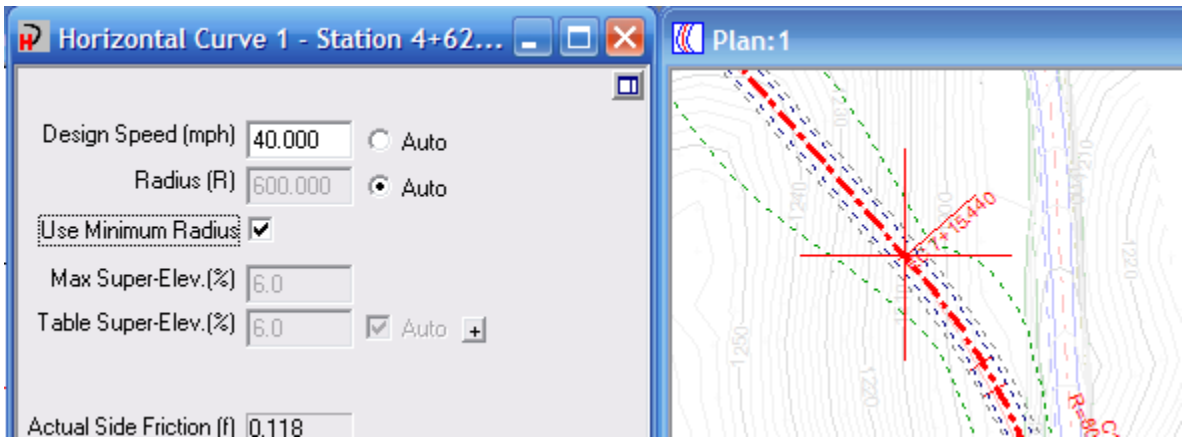


Figure 12.3: Use Minimum Radius *defines the smallest curve possible given design speed, and the super-elevation table.*

7. Apply the smaller 600ft radius to both curves. This will give you some room to move the horizontal alignment so it is not so far up the hill.

8. Return to *Add/Enter* mode .

9. In the Plan window, move the IP for the South curve North and move the IP for the North curve South (both while maintaining tangency with the old road). You will have to stop when the curves bump into each other and become an S curve (figure below)

10. Bring the Profile widow to the front and notice the new shape of the ground (figure below).

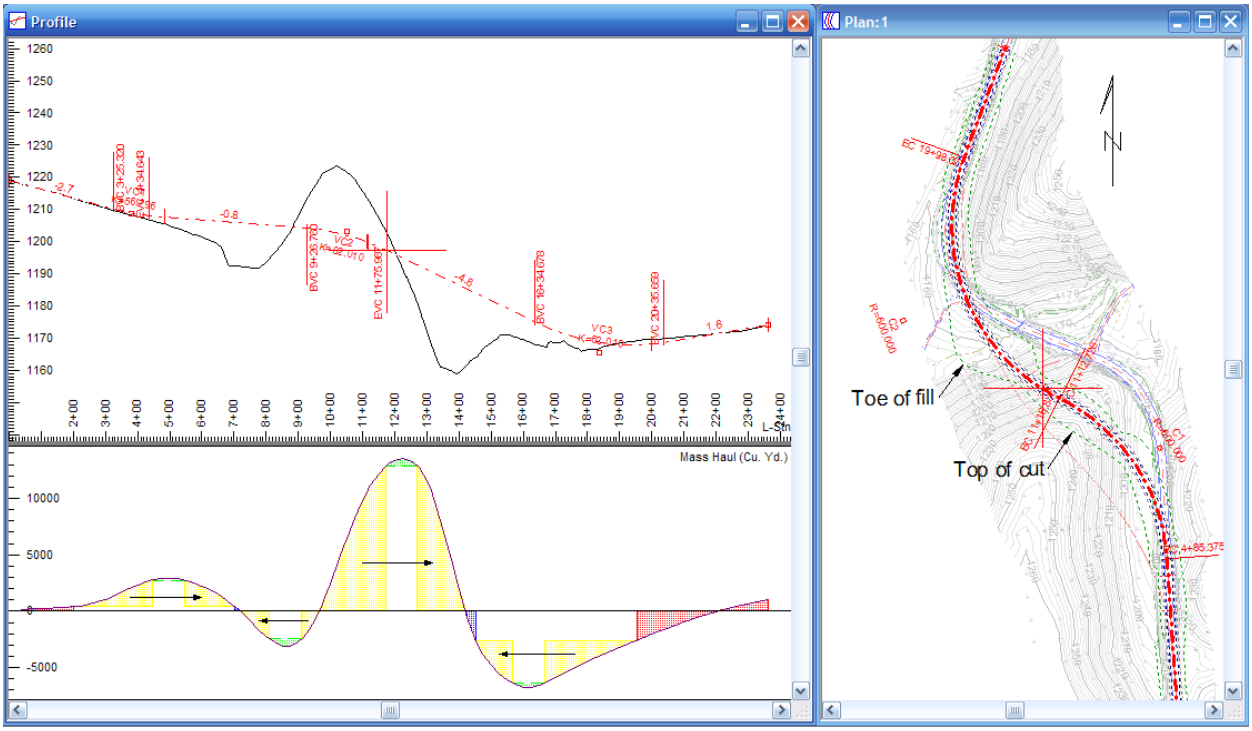


Figure 12.4:When the alignment is much closer to the old road (but still 40mph) the volumes are significantly reduced.

11. Now adjust the vertical alignment to further optimize the volumes indicated by the mass haul diagram (figure above).

In Figure 12.4 above, the material cut to pass through the hill is about 16000 cu. yds. (3000 + 13000). In Figure 12.2 , before horizontal alignment, the comparable quantity is 27000 cu. yds. The footprint is also smaller and the grades have been reduced.

In reality, it may be considered unsafe to have an S curve. Continue modifying the design until you are comfortable with:

- Vertical and horizontal IP editing (including adding and removing IPs).
- Vertical and horizontal curve editing.
- Moving around the various Location windows.

12. *File* | *Close*, do not save changes.

13. Setting up a screen layout

In this exercise you will configure the window locations and some window options to emphasize horizontal curves (for the next exercise).

1. Use menu *File* | *Open* to open **Examples\Location\ Align stage 4.dsn.** This should look similar to your design at the end of the previous exercise.

2. Restore the horizontal curve panel and click on the *Dock Panel Window* button at the top right of the panel.

3. Double click in the Plan window *title bar* to maximize. Your screen should look similar to the figure below.

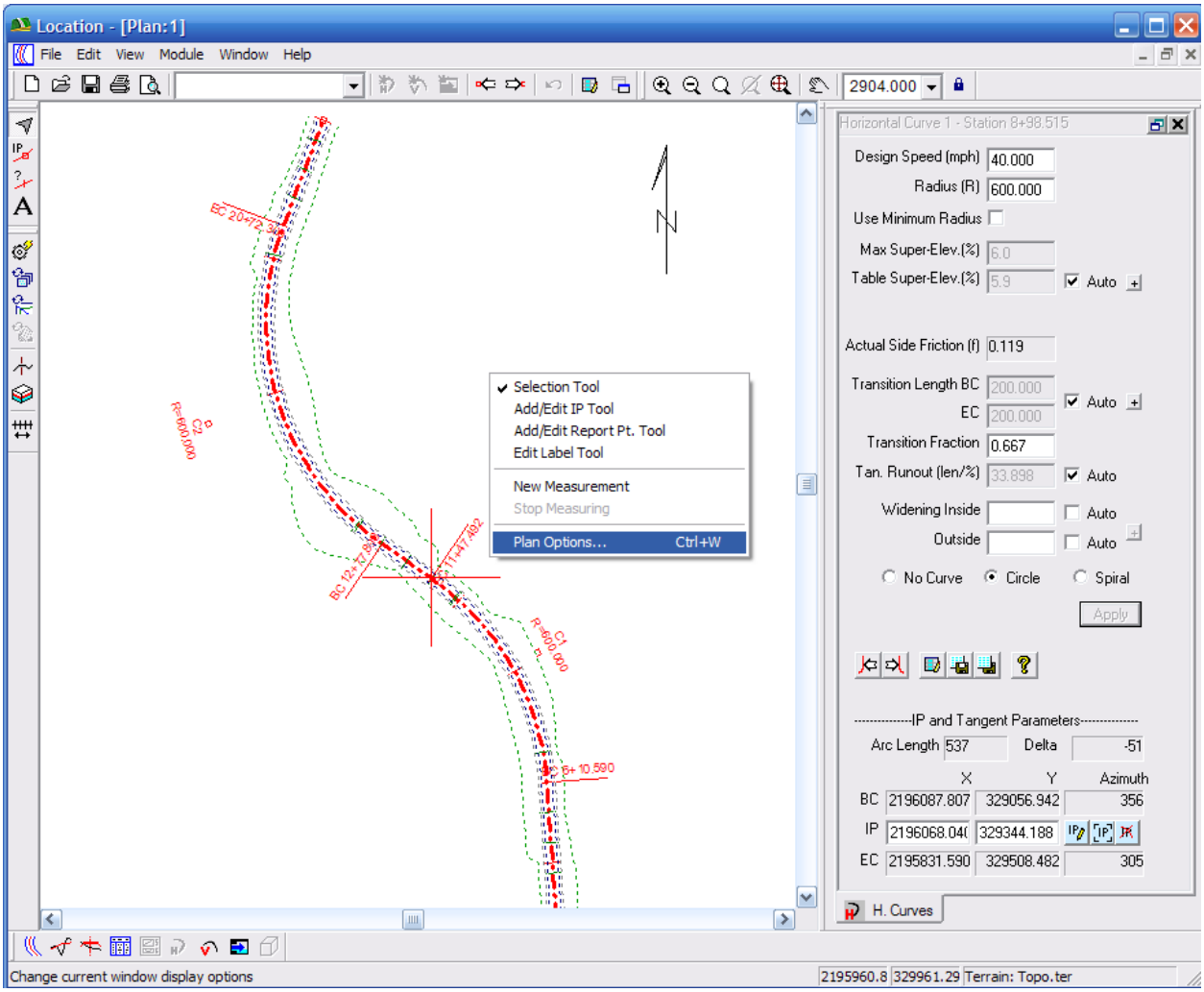


Figure 13.1: A screen layout with docked curve panel and maximized plan window.

4. Open the Plan window *Options* dialog box (right click, *Plan Options*) and turn off *Background* display (figure below, left).

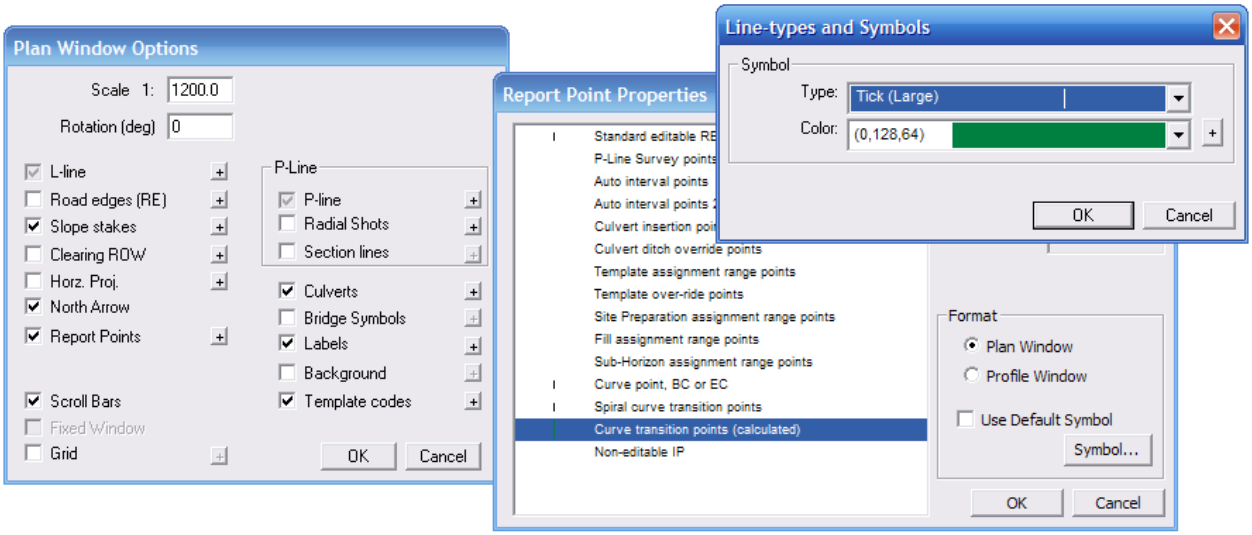


Figure 13.2: Plan Window Options, Report point Properties *and* Symbol Formatting *dialog boxes.*

5. Click the [+] button beside *Report Points* to pop up the *Report Point Properties* dialog box (figure above, center)

6. Select *Curve Transition Points (calculated)* in the list and then press the *Symbol* button.

7. Change the symbol to *Tick (Large)* and dark green (as in figure above, right).

8. Press *OK*, *OK*, and *OK* again to accept changes and close all dialog boxes.

The changes you've made in the last few steps have changed the *look* of the screen but they have made no changes to the actual design – no alignment or cross section changes. These changes and the rest of the *Screen Layout* can be saved to your hard drive for later use.

9. Drop down the *Screen Layouts* tool bar item and right click on the *Softree* folder at the top (figure below left). Choose the *Save New Layout* menu to display the dialog box show below to the right.

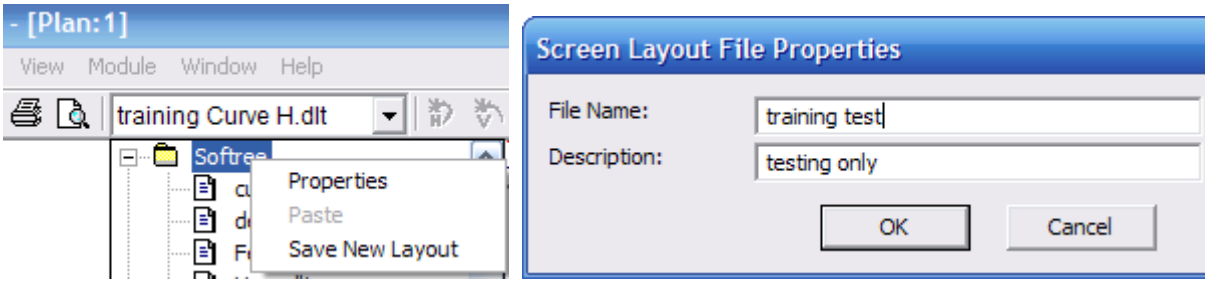


Figure 13.3: Saving a screen layout with the Screen Layouts tool bar.

10. Name the screen layout (as in the figure above) and press the *OK* button.

You have just created a screen layout that will appear in your *Screen Layouts* tool bar item for easy retrieval. Alternately, you could have used the menu *File* | *Save screen layout*.

11. Select menu *File* | *Save screen layout*. The folder displayed in the file-save dialog box is the RoadEng *Settings and Layouts* folder. You should be able to see your new file (**training test.DLT**) created above (figure below).

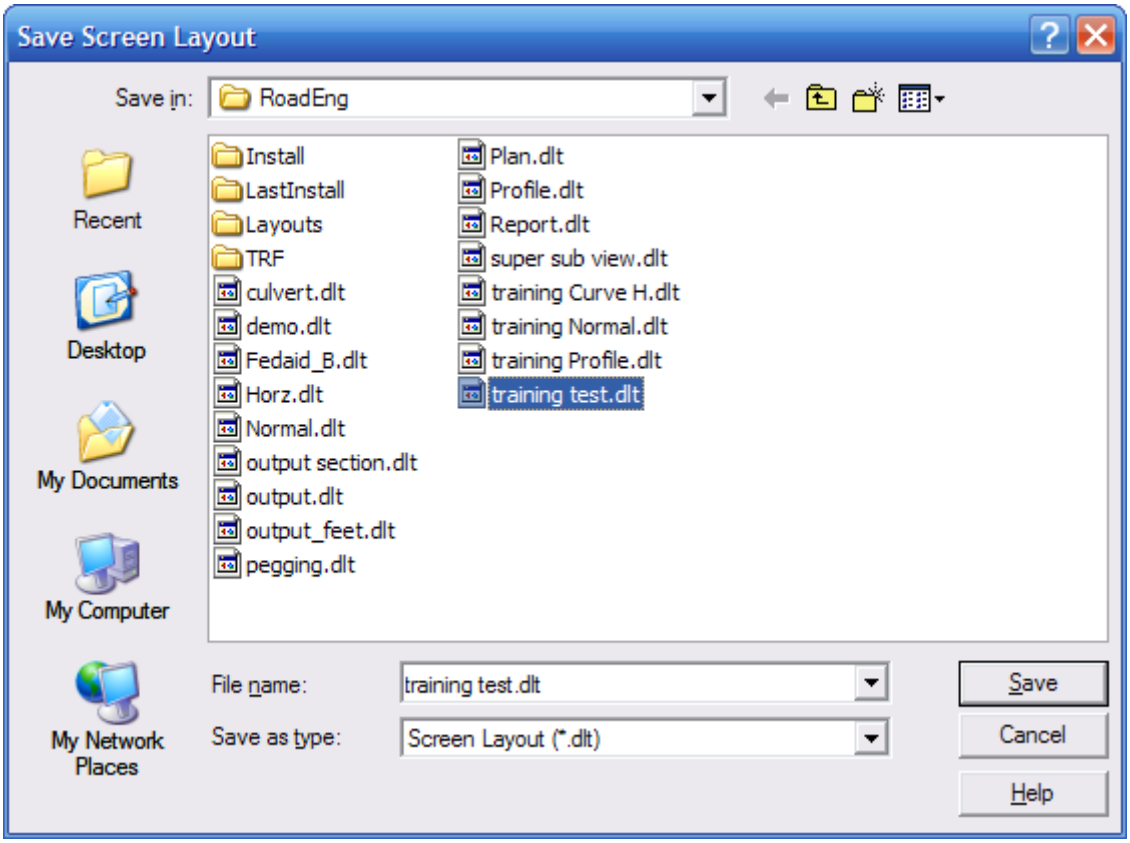


Figure 13.4: Screen layout files are saved to the RoadEng Settings and Layouts *folder.*

12. Press Cancel to close the file-save dialog box.

13. A screen layout called **training Curve H** has been installed; select it using the *Screen Layouts* tool bar item and note how it differs from your screen layout.

14. Try opening some of the other screen layouts available.

Screen layout facts

- Right click on a screen layout in the *Screen Layouts* tool bar item to
  - Change properties
  - Delete
  - Copy
  - Save
- Right click on a folder (*Softree* or *Custom*) in the *Screen Layouts* tool bar item to
  - Change properties (only the *Custom* folder can be changed here)
  - Paste a screen layout that was recently copied
  - Save new layout (define name and description)

- The *Custom* folder is often defined on a network drive so that the layouts are accessible to all users.
- The menu *File* | *Save screen layout* allows you to save a screen layout anywhere but only those in the *Custom* or *Softree* folders will appear in the *Screen Layouts* tool bar.
- The menu *File* | *Retrieve screen layout* allows you to open a screen layout file anywhere including those in the *Custom* or *Softree* folders.
- You can change the *Softree* folder (RoadEng Settings and Layouts folder) from the menu *Module* | *Setup*, *Install* tab. Do not do this unless you understand the consequences; more than just screen layouts are stored in this folder. The most common change is to put *Settings and Layouts* into your *Documents* folder (private to one user only).

15. When you have finished experimenting with screen layouts, *File* | *Close*, do not save changes.

14. Horizontal curve details

In this exercise, we will examine the *Horizontal Curve Panel* in detail.

1. Use menu *File* | *Open* to open **Examples\Location\ Align stage 4.dsn.**

2. Using the *Screen Layouts* tool bar item, select **training Curve H** screen layout. Your screen should look like the figure below.

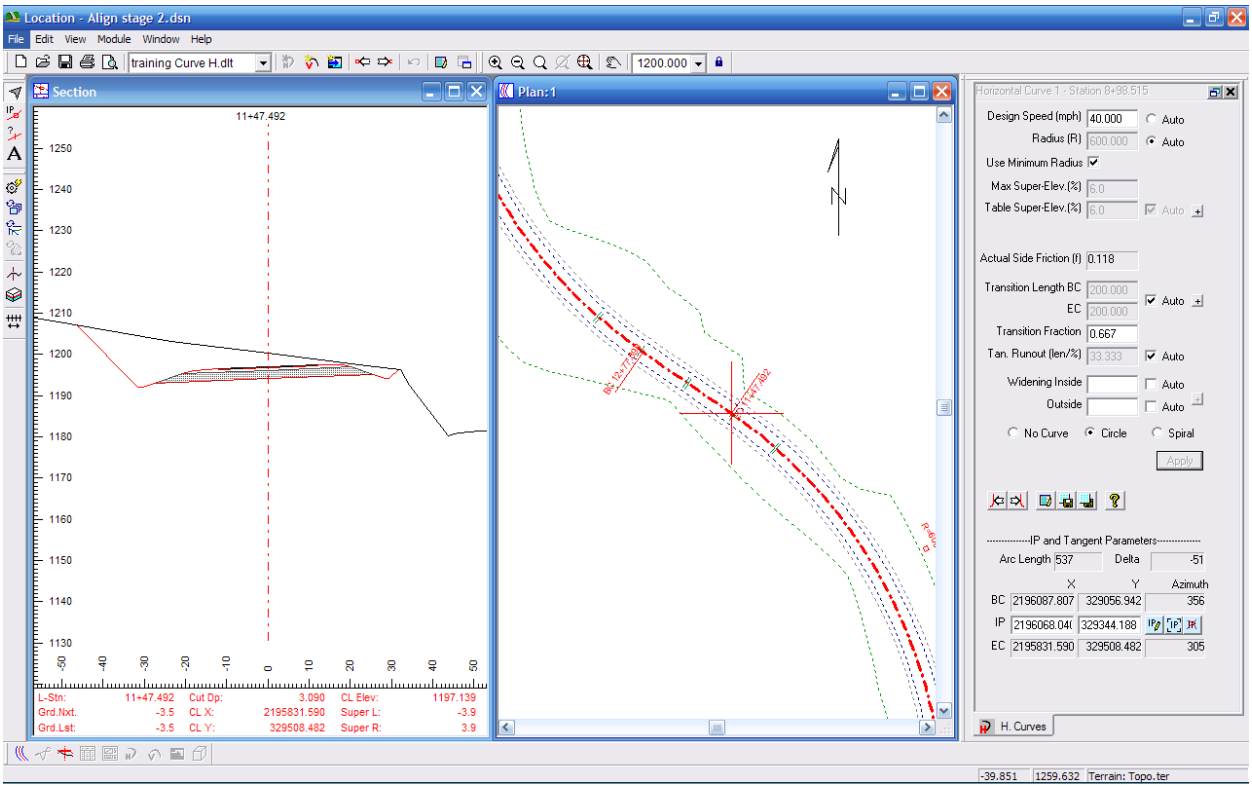


Figure 14.1: A screen layout with docked curve panel, section window and plan window.

Using help

There are too many possible curve configurations to cover them exhaustively in this exercise. So the first thing you need to know is how to view the help files for the curve panels.

Skip ahead to step 8 if you are already familiar with the RoadEng help documents.

3. Type the <F1> key for context sensitive help. You will be presented with the window shown below.

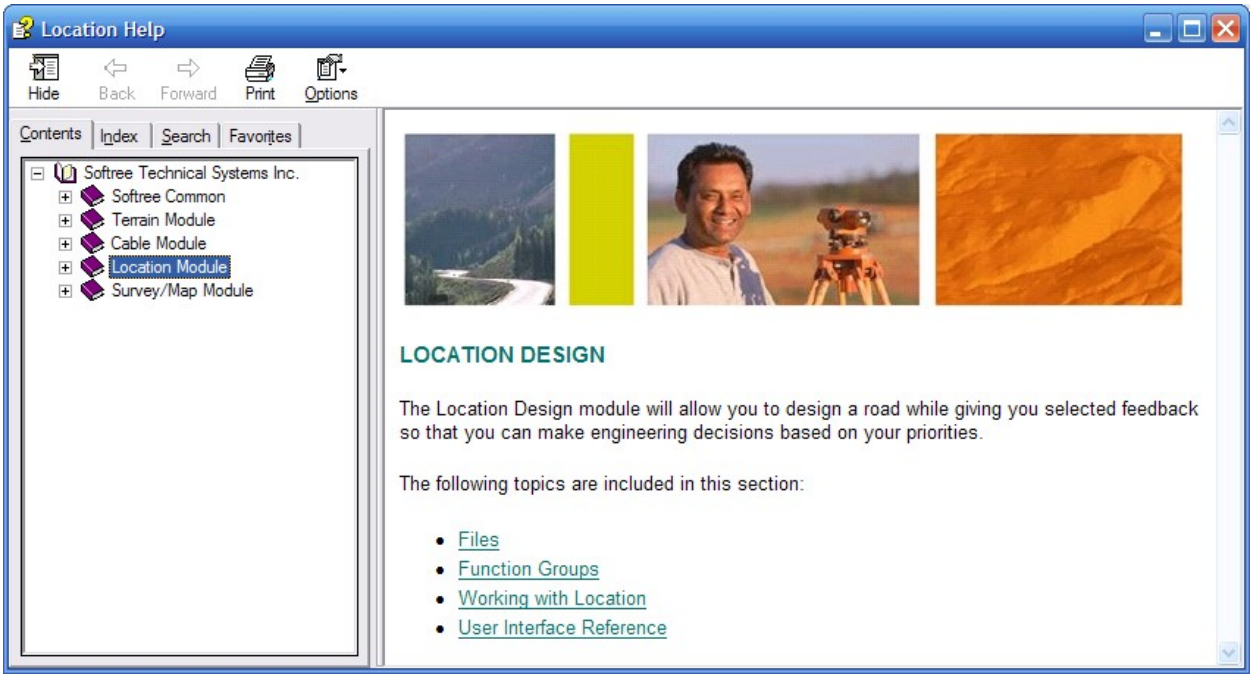


Figure 14.2: The front page for Location help.

The curve panels are not dialog boxes (although they look like they are) so the <F1> key opens a general page. You could also have used menu *Help | Location.*

The quickest way to find the curve panels in the help document is to use the *Search* tab.

4. Select the Search tab and type “curves” into the text box and press *List Topics*. The first topic in the list is *Horizontal Curves Panel - Advanced Mode*; double click on this item or press *Display*. The help window should now look like the figure below.

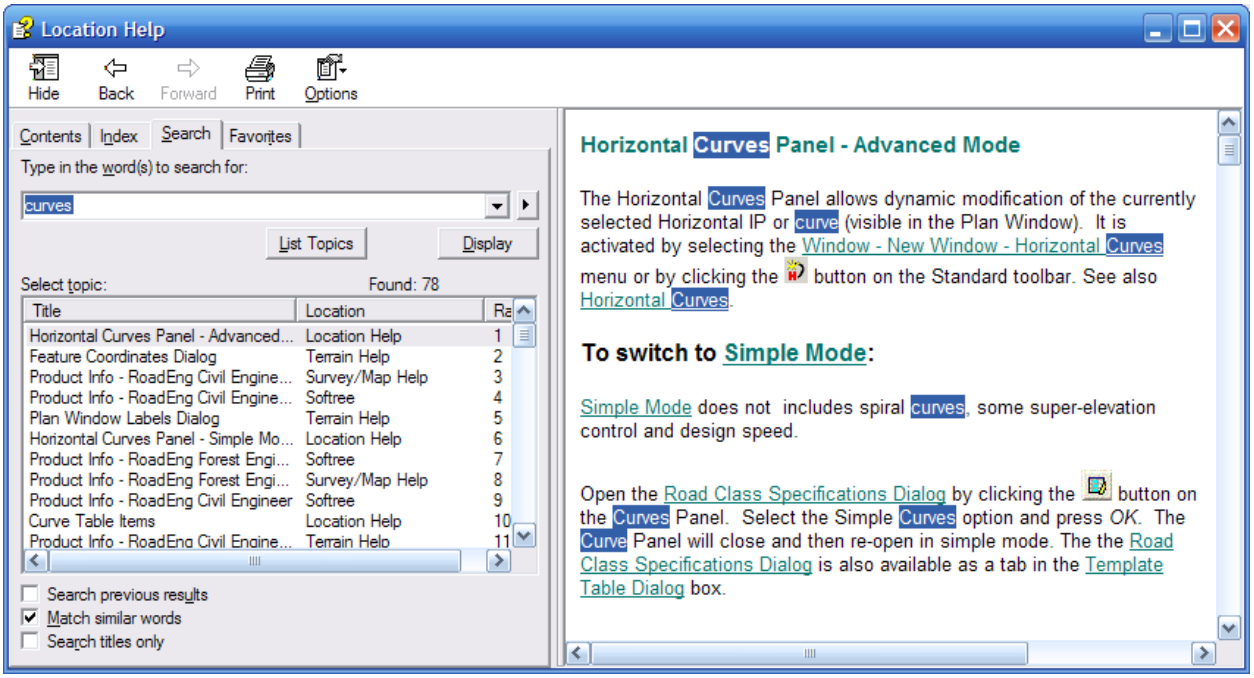


Figure 14.3: Horizontal Curves Panel - Advanced Mode *help page.*

This page has information about all the controls in the horizontal curve panel. It also has many links to related topics.

5. Select the *Contents* tab (on the left side of the help window). Note that the contents show the location of the current page in a tree list with related topics nearby (figure below).

6. Click on *Vertical Curves Panel - Advanced Mode* in the list to display its help page.

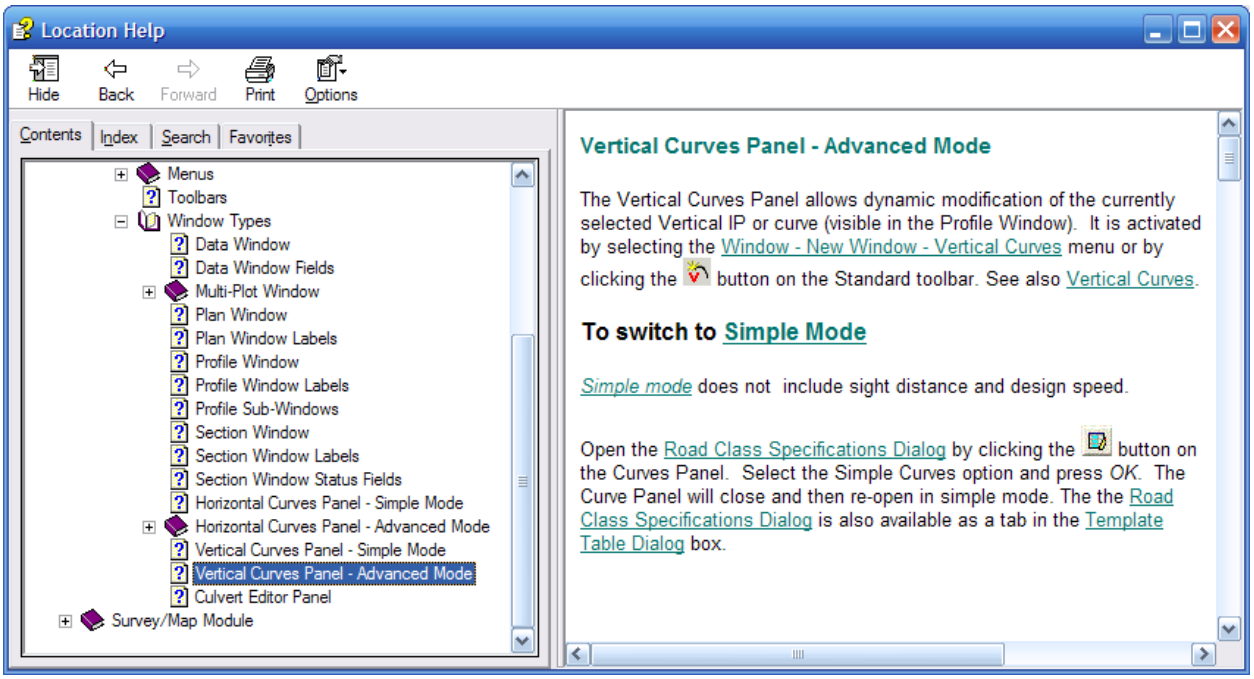
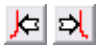


Figure 14.4: Help window showing the Contents *tree and* Vertical Curves Panel - Advanced Mode *help page.*

7. Close the Help window.

Radius, Design Speed and Super-elevation

The most common way to define a safe horizontal curve is by using a super-elevation table. However you may define curves manually if you wish.

8. The first curve should already be current, if it is not navigate  to the first curve.

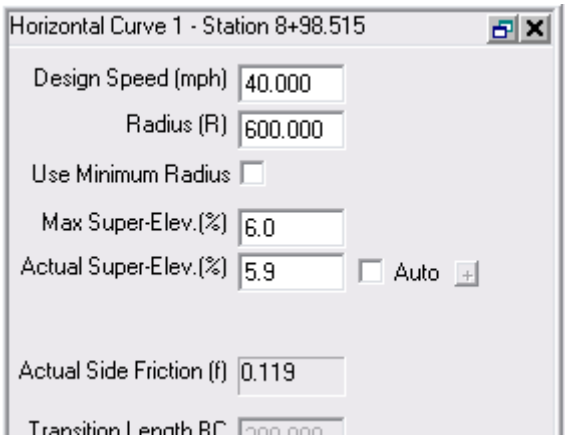


Figure 14.5: The top part of the horizontal curve panel

9. Turn off the *Use Minimum Radius* check box and then turn off the *Auto* check box (figure above) and note that you can then define the super-elevation for a given curve manually.

The *Side Friction* factor is the coefficient of friction required to keep a vehicle on the road; alternately, it is the sideways acceleration felt by the driver (as a fraction of the acceleration of gravity, the "g-force"). The smaller the better.

10. Try a few values for *Design Speed, Radius* and the *Actual Super-elevation* and notice what happens to the *Side Friction* factor.

Design Speed (mph)	Radius (feet)	Super-Elevation (%)	Side Friction
40	600	6	0.118
40	600	0	0.178
40	1200	3	0.059
30	600	3	0.070

11. Turn the *Auto* check box back ON and press the [+] button beside it to open the *Auto Super Elevation Options* dialog box (figure below).

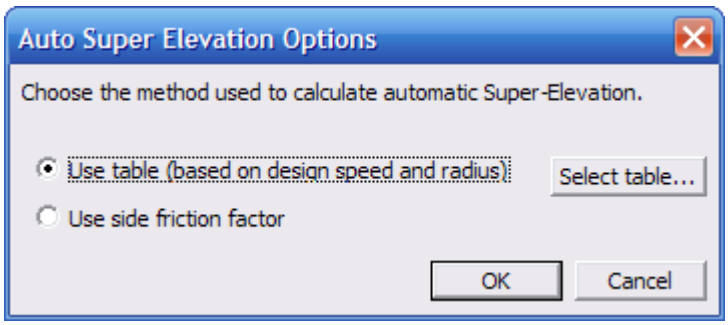


Figure 14.6: Auto Super Elevation Options *dialog box.*

The *Use side friction factor* method is defined in the AASHTO 2001 handbook. It relies on a table of "safe" side friction factors to calculate maximum speed given radius (or minimum radius given speed) using physical principals. It is possible to use this method to calculate the values for a super-elevation table; the two methods need not give different results. The *side friction factor* method is discussed in the help text and will not be discussed further here.

12. Press the *Select Table* button to open the *Lookup Table* dialog box (figure below).

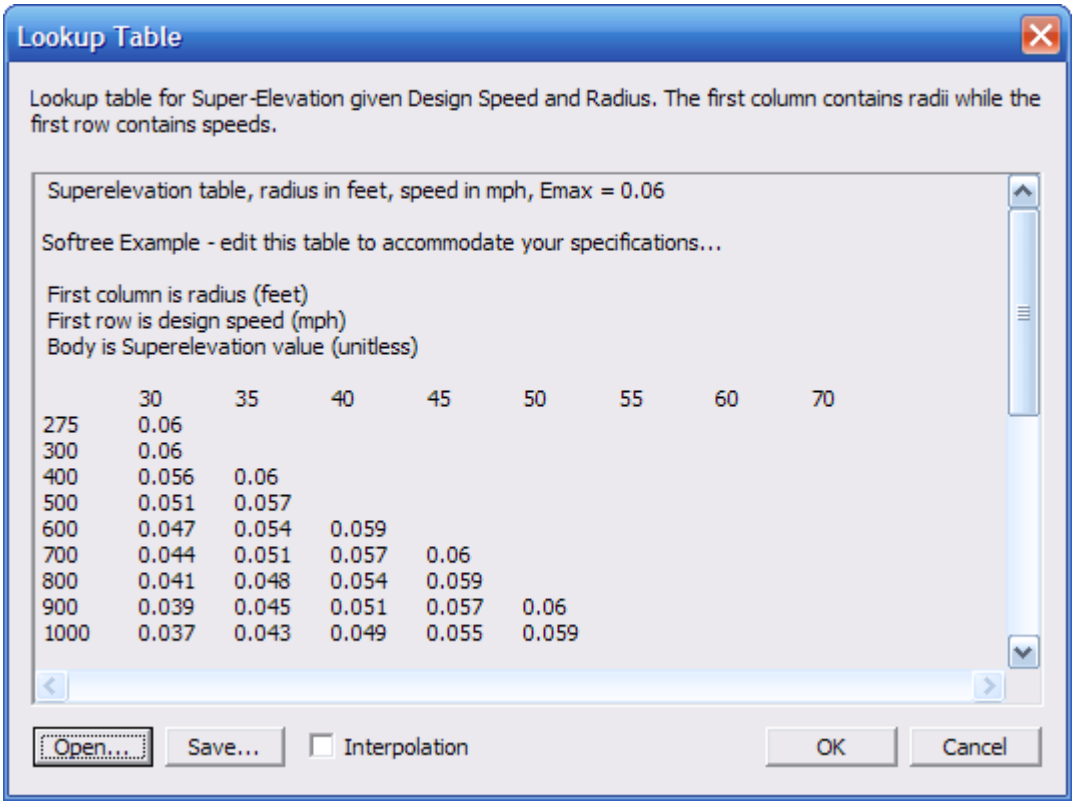


Figure 14.7: Super elevation Lookup Table *dialog box.*

Super-elevation table facts

- The blank area of the table represents unsafe combinations of radius and speed.
- The top of each column in the body of the table is the maximum super-elevation. Given a speed, the corresponding radius is the minimum allowed; given a radius, the corresponding speed is the maximum allowed.
- Super-elevation is defined by rise over run – the tangent of the angle (not %)
- *Interpolation* means that if the speed or radius is in between table entries, the super-elevation value will be linearly interpolated between the values in the body of the table.
- This table is stored in the template table (inside a design document or in an external file).
- Super-elevation tables can be imported/exported from/to simple text files (comma or tab delimited). Spreadsheet programs can also import and export these files.
- Lines starting with "#" characters are excluded from the table when importing text files – the comments at the top of the table in the figure above were marked in this way.
- Softree does not (as of this printing) provide officially approved tables.

13. Press *Cancel* to close the table and *Cancel* again to close the *Auto Super Elevation Options* dialog box.

Sometimes when your options are limited, it is best to design alignment with the smallest safe radius – you can always increase the radius later if you have room.

14. Check the *Use Minimum Radius* check box and then check the *Auto* radio button beside *Radius*. In this mode, you can type a desired *Design Speed* and the software will use the super-elevation table to find the minimum safe radius. Try a few values.

If you type a speed outside the table (20 mph for example) you will see the error message below left. If you type a speed greater than 40 mph (45 mph for example) you will see the warning below right. You can ignore the warning and apply the curve anyway if you wish.

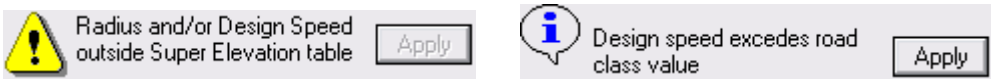


*Figure 14.8: Curve error (*Apply *button disabled) and warning (*Apply *button still available).*

15. Check the *Auto* radio button beside *Design Speed*. In this mode, you can type a desired *Radius* and the software will use the super-elevation table to find the maximum safe speed. Again, try a few values.

Road class specifications

Each curve has its own *Design Speed*; the maximum design speed for the entire road is specified in the *Road Class Specifications*.

16. Press the *Road Class Specifications* button to open the dialog box shown below. This dialog box is also available from the Template Table editor (Edit | Edit Templates, *Road Class Specifications* tab).

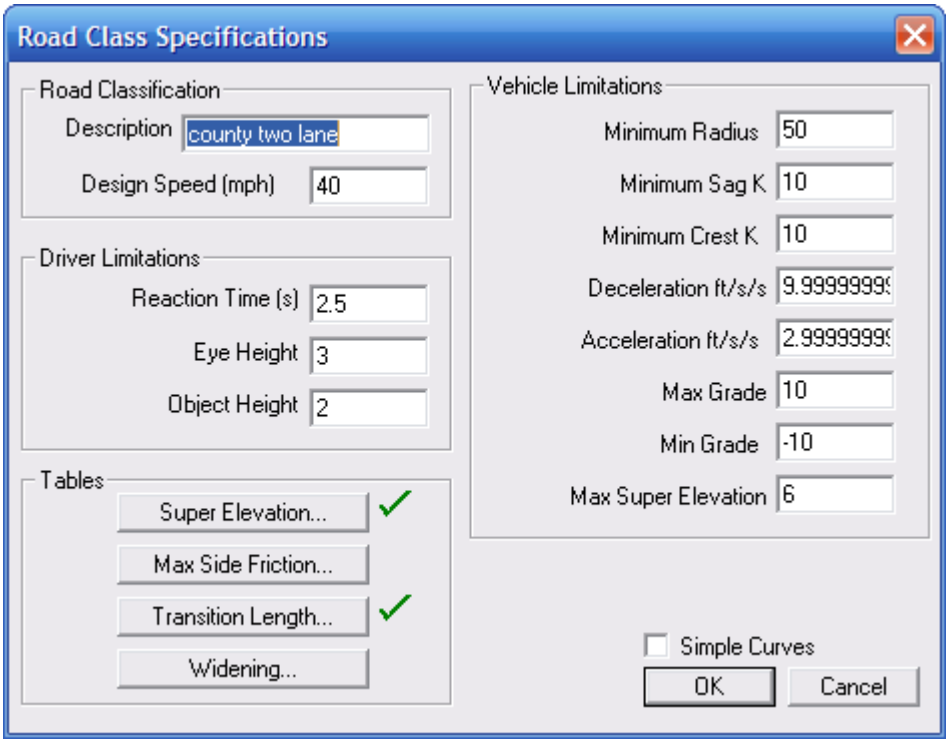


Figure 14.9: Road Class Specifications *dialog box.*

Road Class Specifications are stored with your *Template Table*. Each class of road will have a few typical cross section templates, a design speed and other parameters that are common from road to road. You should have a template table (TPL file) for each class of road you may design.

If you are using a super-elevation table, the parameter in the *Road Class Specifications* most important for horizontal curves is the *Design Speed*. The other parameters are mostly concerned with vertical curves, are redundant or can be accessed directly from the horizontal curve panel. *Transition Length* and *Widening* tables are accessible from the curve panel and are discussed below.

If you turn on the Simple Curves check box you disable many features to reduce complexity – this is often used for private access roads.

Type <F1> for more information.

17. Press *Cancel* to close the *Road Class Specifications* dialog box.

Curve transitions

Transition Length

The *Transition Length* is the distance from half crown to full super, also known as the *Super Elevation Runoff*; it is labeled *Length of Runoff* in the figure below. In a spiral curve the *Transition Length* is also the spiral distance (from *tangent to spiral* (TS) to *spiral to circle* (SC) and from *circle to spiral* (CS) to *spiral to tangent* (ST).

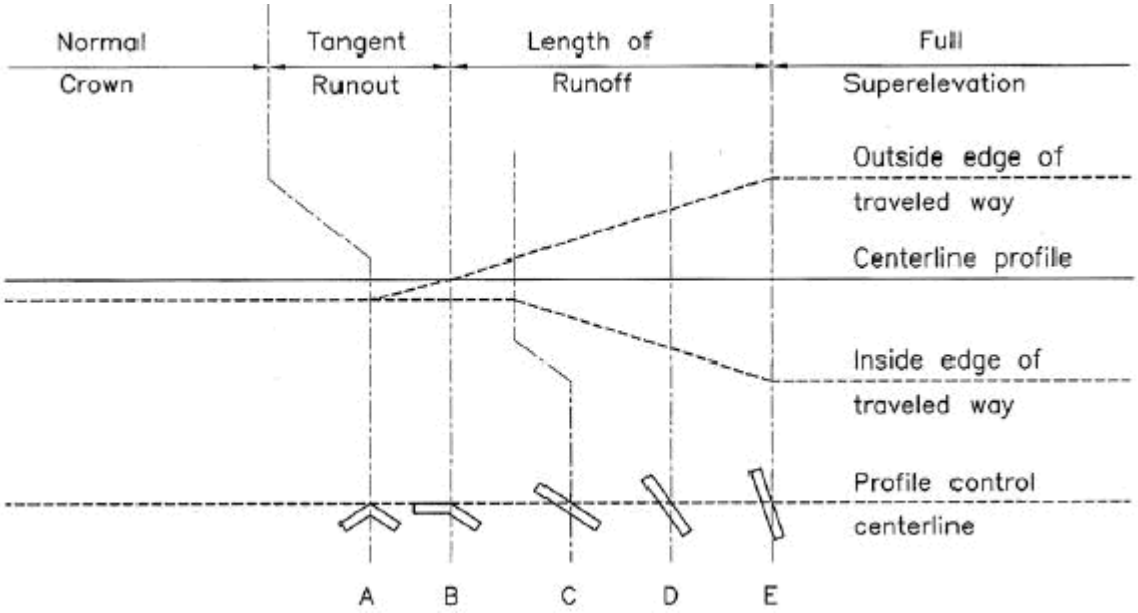


Figure 14.10: Cross fall behavior upon entering a curve.

18. Clear the *Auto* box beside the two *Transition Length* values and note that you can now manually enter a different value for BC (begin curve) and EC (end curve).

19. Check the *Auto* box beside the two *Transition Length* values then press the associated + button. The table shown below will be displayed.

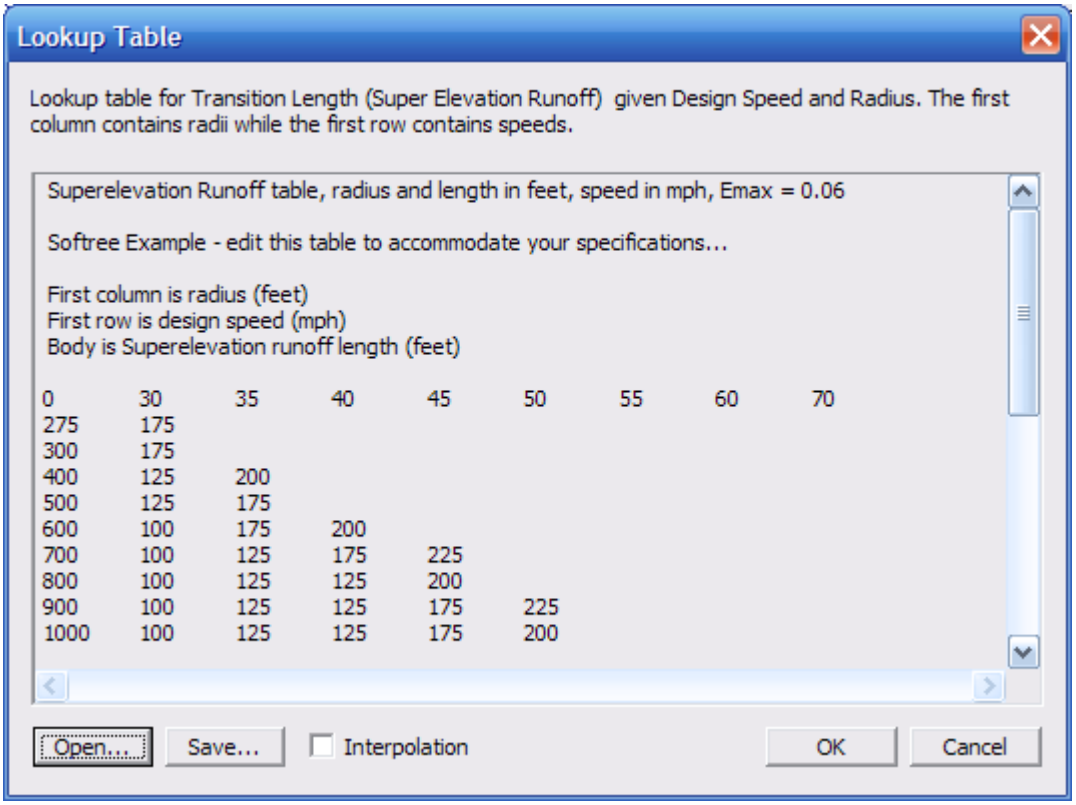


Figure 14.11: Transition Length *table.*

When *Transition Length* is automatic, you are forced to use the same length for begin and end of curve. Like the super-elevation table, this table can be imported/exported from/to an external text file. To make changes, export, edit and re-import.

20. Press *Cancel* to close the *Transition Length* table.

Transition Fraction

For a circular curve, the *Super Elevation Runoff* may start before the curve BC and end after; similarly, at the end of the curve, the transition starts before the EC and ends after. The *Transition Fraction* is the amount of transition that happens outside the curve (before BC and after EC). For example, if *Transition Length* is 90 feet:

- If *Transition Fraction* is 1.0, the *Super Elevation Runoff* starts 90 feet before the BC point and full super-elevation is reached at BC
- If *Transition Fraction* is 0.667, a common standard, the *Super Elevation Runoff* starts 60 feet before BC and full super-elevation is reached 30 feet beyond BC.

In a spiral curve the *Transition Fraction* is not used (the *Super Elevation Runoff* always happens in the spiral section).

Tangent Runout length

Tangent Runout length is the distance from full crown to half crown (see cross fall behaviour figure above).

21. Clear the *Auto* check box associated with *Tan. Runout*. Note that you can now enter a *length* manually.

22. Set the *Auto* check box associated with *Tan. Runout*. Now the caption reads *len./%*; the automatic value displayed is the tangent runout length for each % of crown (if your crown is 2%, you multiply this number by two to get the total length). When set to automatic, the *Tangent Runout* happens at the same **rate** as the *Super-Elevation Runoff*.

Curve Widening

Small radius curves require lane widening to accommodate large vehicle off tracking. The *Widening* fields allow you to define a different widening distance for inside and outside lanes. Note that your cross section template must have curve widening built in for these values to have any effect.

As with other curve parameters, you can extract widening values from a table by setting the *Auto* check box. If time permits, you may wish to experiment with this feature. There is a widening table called **Examples\Location\WideningFeet.tbl**.

23. *File* | *Close*, do not save changes.

Viewing curve transitions

Now we will look more closely at the curves we have created to see how the super elevation is applied along the alignment.

1. Use menu *File* | *Open* to re-open Examples\Location\ Align stage 4.dsn.
2. Using the *Screen Layouts* tool bar item, select **training Curve H** screen layout.
3. Use the next or back buttons in the tool bar (NOT in the curve panel) to move the current point to the tangent before the first curve (figure below).

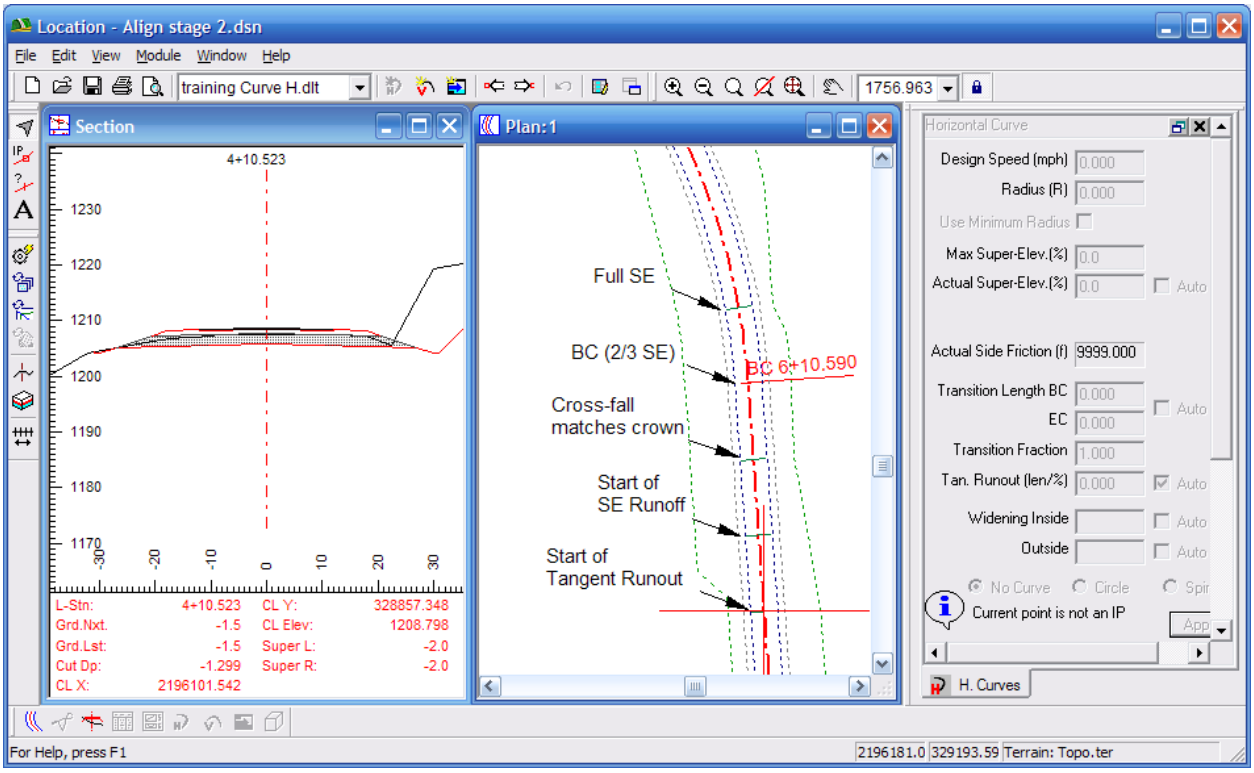


Figure 14.12: Section window text area shows super-elevation while the plan window has curve transition report points displayed (green tick).

Cross sections are calculated at all the horizontal curve transition points (unless you explicitly turn off this feature). In this screen layout, these points have been given a green tick symbol so you can see them in the plan window.

4. Use the next or back buttons in to move the current point to the first (South most) tick. Note that the cross section shows full crown (Super L = -2.0%, Super R = -2.0%) (figure above).

5. Click the next button and watch the section window; continue clicking the next button, the outside (right) cross fall is increasing. Stop when you get to the next tick mark.

The second tick mark is the end of *tangent runout* and the beginning of *super-elevation runoff*. The right hand cross-fall is 0.0%. Note that the station is 133.3 ft. less than the BC station – 2/3 of our transition length.

6. Continue clicking the next button until you get to the next tick mark. The outside lane has rotated until it is tangent to the crown slope on the other side (Super L = -2.0%, Super R = +2.0%). From here on both sides will rotate.

7. Continue clicking the next button until you get to the BC point. Here the super-elevation is about 4% (2/3 of the final value).

8. Continue clicking the next button until you get to next tick mark. This is full super-elevation.

9. Use the horizontal curve panel to change the transition fraction to .5 and re-apply the curve. Note how transition points move.

Curve transition overlap

What happens between the two curves?

10. Use the same technique to step through the cross sections near the tangent between the two curves.

Note: When there is not enough distance between curves for the full transition, the software will skip the crown cross sections and go directly from one super-elevation to the other (with one flat cross section in between in the case of an S curve).

11. Bring the Profile window to the front, the sub-window has been configured to show the super elevation (figure below).

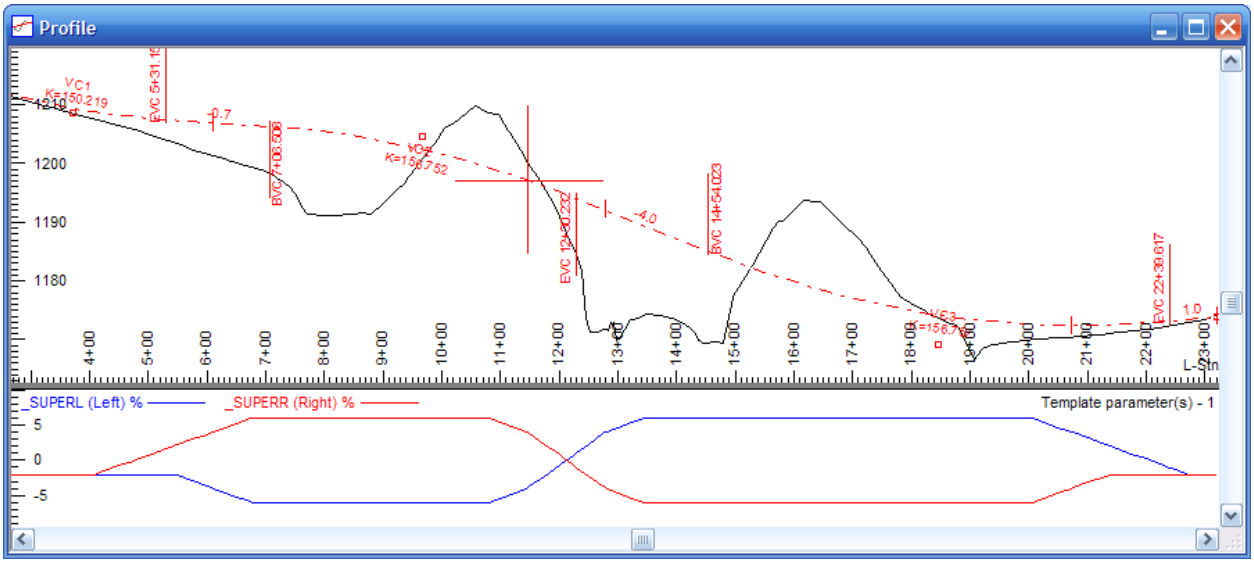


Figure 14.13: Profile Template Parameters *sub-window with super elevation displayed.*

12. If time permits, move the curves closer or farther apart and repeat your cross section audit.

13. *File* | *Close*, do not save changes.

`

15. Vertical Curve Details

In this exercise, we will examine the *Vertical Curve Panel* in detail.

1. Use menu *File* | *Open* to open **Examples\Location\ Align stage 4.dsn.**

2. Using the *Screen Layouts* tool bar item, select **training Curve V** screen layout. Your screen should look like the figure below.

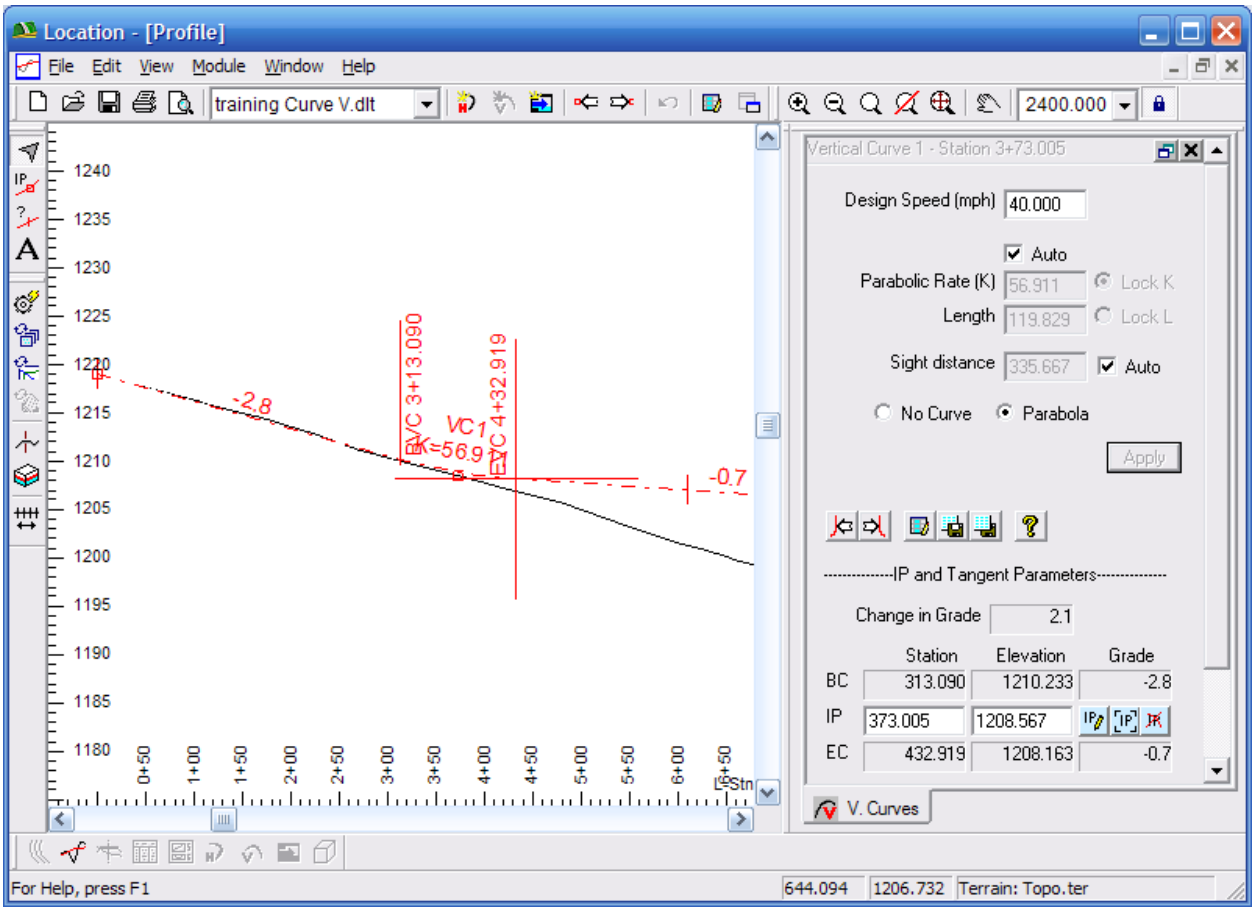


Figure 15.1: A screen layout with docked curve panel and profile window.

3. Use the Previous IP and Next IP buttons to move to the first vertical curve.

Automatic curvature

This curve has been configured to find the smallest possible curve that has a safe sight stopping distance. The *Auto* check box is set for both *Parabolic Rate (K)* (curvature) and *Sight distance*.

4. Change the *Design Speed* to 30. Notice how the sight stopping distance drops, as does K and the length of curve.

5. Set the *Design Speed* back to 40.

6. Press the *Road Class Specifications* button to open the dialog box shown below.

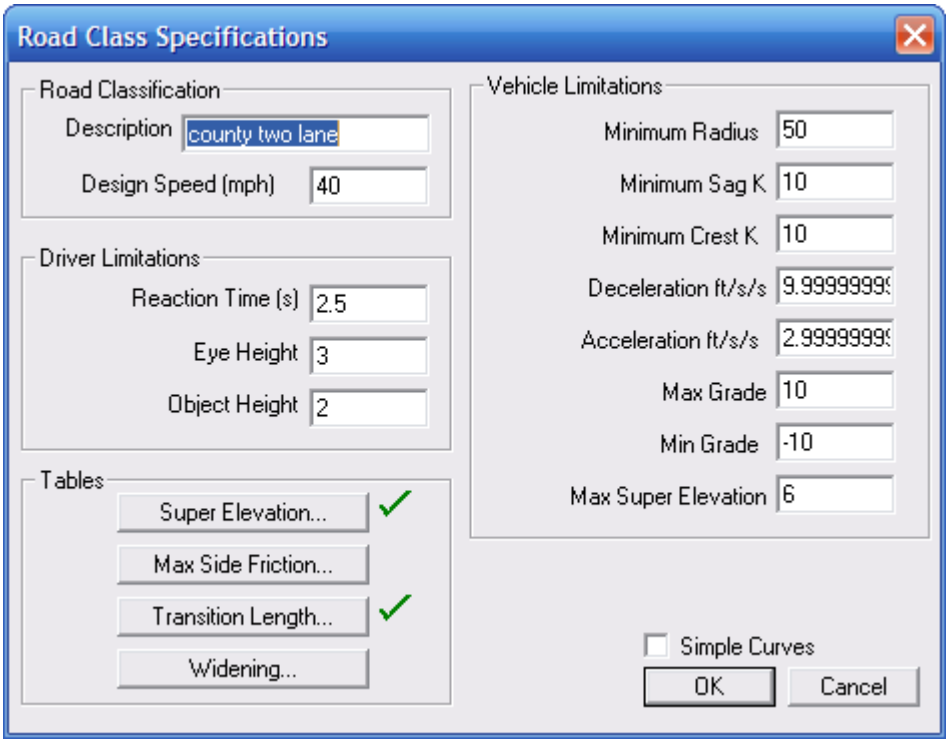


Figure 15.2: Road Class Specifications *dialog box.*

When *Sight distance* (stopping) is automatic in the vertical curve panel, it is calculated from the steepest grade in the curve and the following values from the *Road Class Specifications*:

- *Design Speed*
- *Reaction time*
- *Deceleration*

When curvature, *K*, is automatic in the vertical curve panel, it is calculated from the required *Sight distance* and the following values from the *Road Class Specifications*:

- *Eye Height*
- *Object Height*

Type <F1> for more information.

7. Press *Cancel* to close the *Road Class Specifications* dialog box.

8. Capture the VIP point for this curve in the profile window and move it up and down. Notice how the values in the curve panel are kept up to date. Also notice that with the curvature set to automatic, both the length and the curvature (K) change as the VIP is moved.

Locked K

9. Clear the *Auto* check box below the design speed. Make sure the *Lock K* radio button is selected and *Apply* your change.

10. Repeat the experiment from step 8. Now the K value is constant and the length changes as you move the VIP up and down. Notice that the "K is too small for desired sight distance" warning appears and disappears (larger grades require longer stopping distances and therefore larger K values).

Locked length

11. Select the *Lock L* radio button and *Apply* your change. Again, capture the VIP and move it with the mouse. Curves with constant length will never bump into one another when you raise or lower the VIP; however the curvature changes dramatically.

Editing VIPs with the curve panel

So far we have created and edited intersection points only with the mouse (both vertical and horizontal). The curve panels also allow you to create and edit IPs.

12. Re-open **Examples\Location\ Align stage 4.dsn** (do not save changes), and select screen layout **training Curve V**.

13. Use the Previous IP and Next IP buttons to move to the second vertical curve.

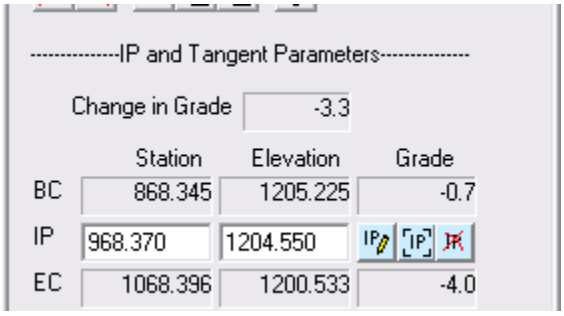


Figure 15.3: The bottom part of the vertical curve panel.

14. Change the elevation of the IP to 1210 (shown as 1204.550 in the figure above) and *Apply* your change. Note how the curve moves vertically.

Note: You can fine tune your alignment by making small changes to the VIP *Station, Elevation* values.

15. Press the *Modify IP* button to open the *Modify Vertical IP* dialog box (figure below).

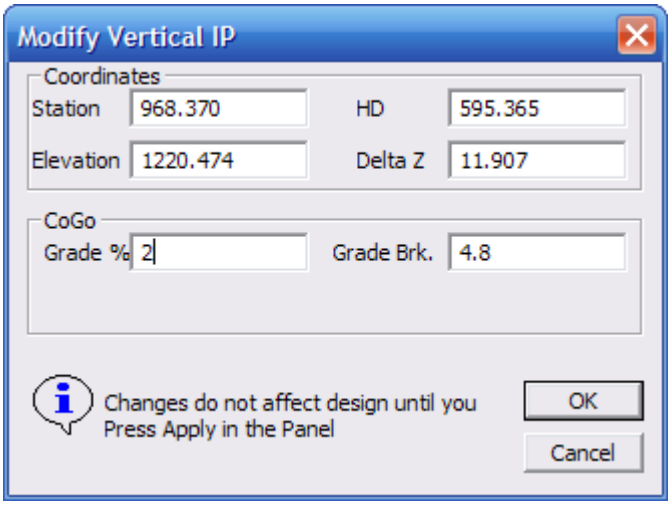


Figure 15.4: The Modify Vertical IP *dialog box allows you to set the grade of the previous tangent (among other things).*

16. Type "2" into the *Grade %* field (figure above) and press *OK* to close the dialog box. Your *Elevation* value in the curve panel has been updated.

17. Press *Apply* to change the curve.

Note: You can also edit *horizontal* alignment in the *Horizontal* curve panel in an analogous way.

18. *File* | *Close*, do not save changes.

16. Materials and Stripping

So far our design has ignored the quality of the material in the original ground. If you looked a little closer you would find that all subgrade cut and fill material is classified as *OB* (overburden). This is a common practice and produces acceptable results (provided that you assign a reasonable expansion factor to *OB* so that the Mass Haul is approximately correct).

In this exercise we will add some more realism to our design by defining materials in the original ground and in the subgrade fill. We will also strip topsoil from the original ground before applying templates.

Defining sub-surface layers

1. Open **Examples\Location\Align stage 4.dsn**.
2. Choose menu *Edit* | *Edit Ground Types* to open the *Ground Types Editor* (figure below).

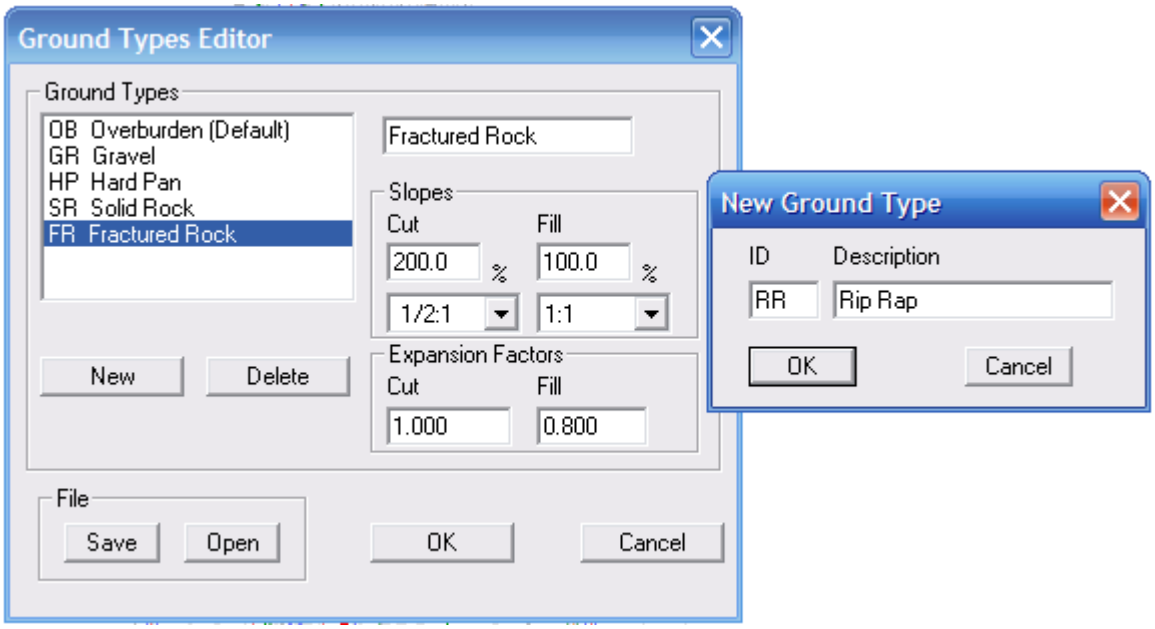


Figure 16.1: The available ground and subgrade fill materials in the Ground Types Editor.

3. Press the *New* button and create a new material called *RR – Rip Rap* (as in the figure above right). You will use this as subgrade fill in following steps. Press *OK* to close the *New Ground Type* dialog box and to add the new material to the list.
4. Set the *RR* fill angle to 100% (1:1).
5. Also create a material called *CS-Clay Silt* with cut angle 1:1 and fill angle 3:1.
6. Press the *Save* button to open the file *Save-As* dialog box. Notice that the default folder is the RoadEng *Settings and Layouts* folder. Press *Cancel* to close the dialog box.

Note: The default ground table is called **RoadEng.gnd** (the only default that is not called “normal”).

7. Press OK to accept changes and close the *Ground Types Editor*. Respond *No* to the recalculate prompt (we didn't change anything that is in use).

8. Choose menu *Edit | Assign Parameters by range* or press the tool bar button to open the *Assign Attributes by Station* dialog box. Select the *Sub Horizons* tab (figure below).

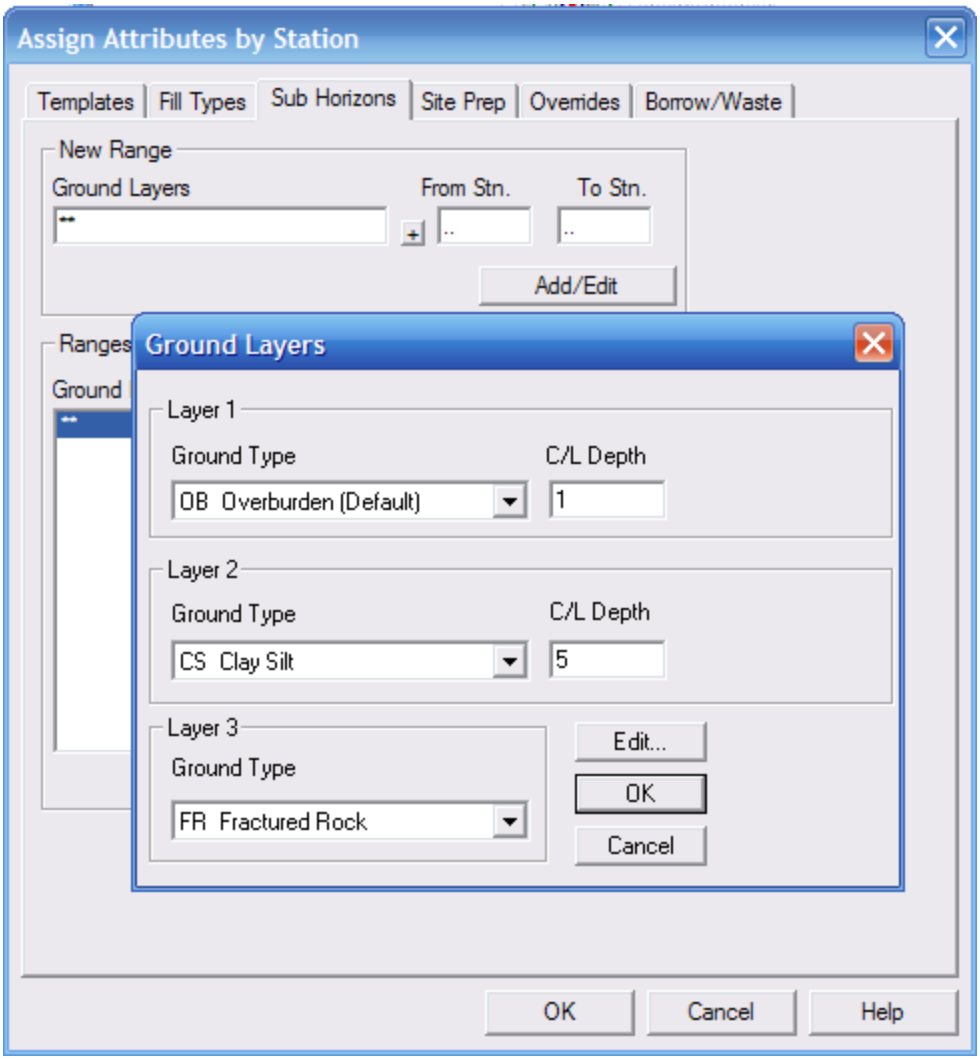


Figure 16.2: Defining materials in the original ground.

9. Press the button beside the *Ground Layers* field to open the *Ground Layers* dialog box (figure above).

Note: If you define *reference surfaces* (Terrain files with a DTM) you can use them to define the *C/L depth* values. This would require bore hole data or similar to create enough subsurface points to make surface models.

10. Select the three subsurface layers shown above and then set the layer depths. Press OK to close the dialog box and fill the Ground Layers field in the Sub-Horizons dialog box.

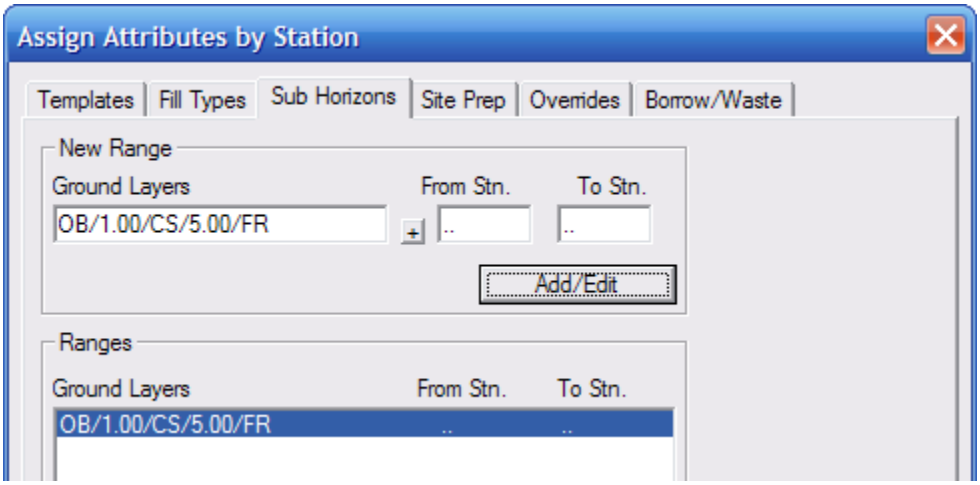


Figure 16.3: Sub-horizons have been applied to the entire alignment

11. Leave the *From/To Stn.* values as ".." and press the *Add/Edit* button. This will apply the new layer arrangement to the entire alignment.

Note: The most common mistake made in the assignments dialog box is to skip the *Add/Edit* step. If you Press *OK* before the ranges are updated, nothing happens.

12. Press OK to accept changes and close the dialog box. Respond *Yes* to the re-calculate prompt.

13. Select a cross section that has some cut in it. Notice that the new ground layers are displayed.

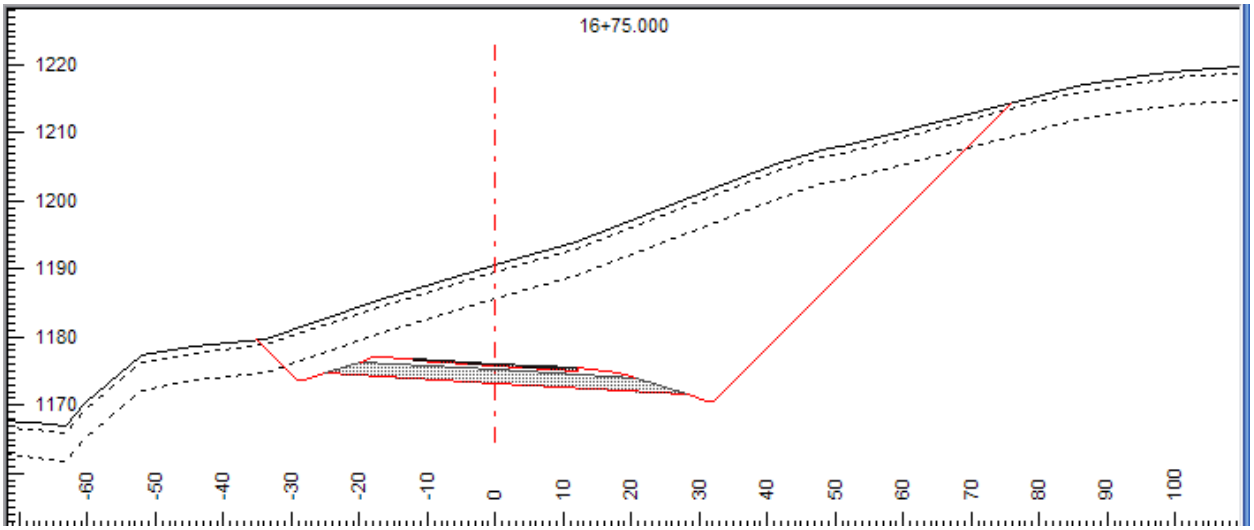


Figure 16.4: ground layers shown in the section window.

The design total volumes haven't changed because of the new ground layers. However the program is now keeping track of three categories of cut volume which can be reported separately.

Stripping

Now we will remove the top layer before building each cross section.

14. If you had trouble with the previous steps or if you wish to start here, open **Examples\Location\Align stage 5.dsn**.

15. Choose menu *Edit* | *Assign Parameters by range* or press the tool bar button to open the *Assign Attributes by Station* dialog box. Select the *Site Prep* tab (figure below).

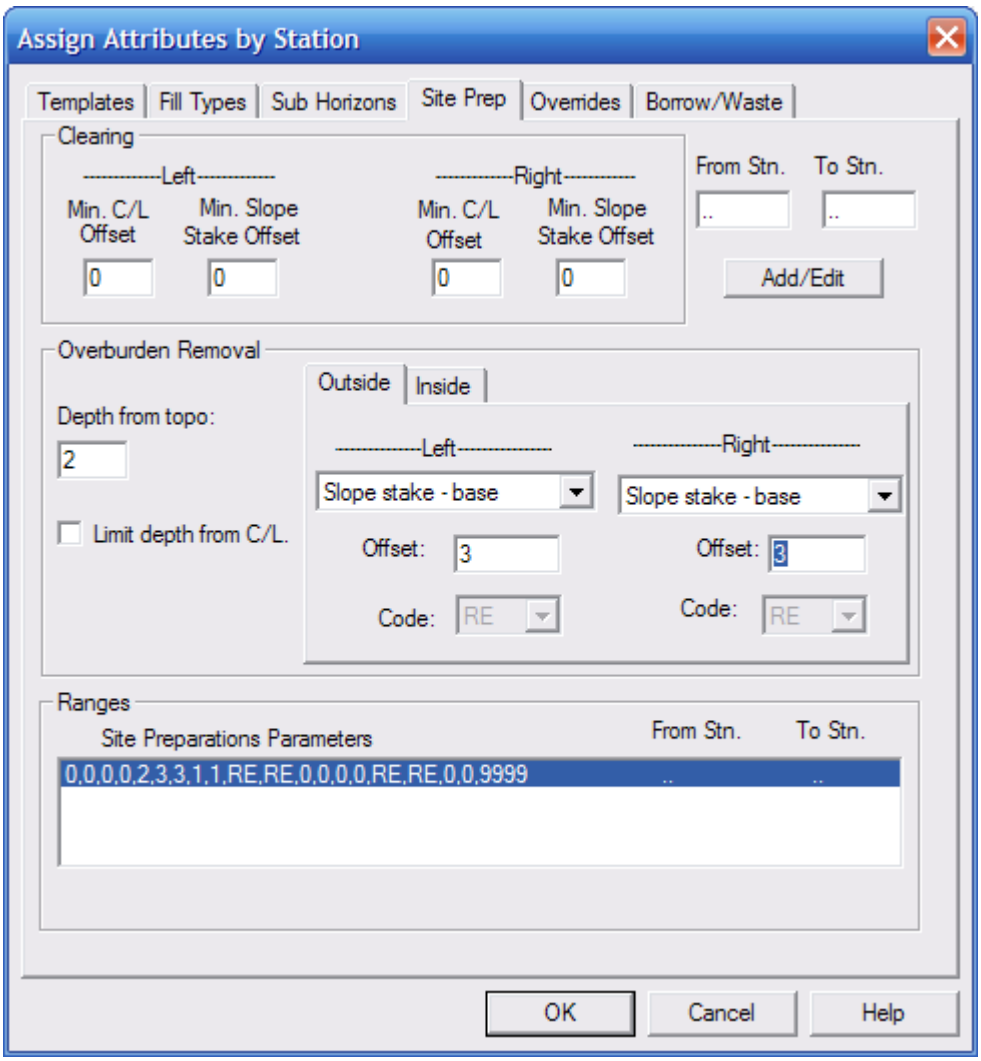


Figure 16.5: Site Preparation dialog set up for stripping (it also can control clearing offsets).

16. In the *Overburden Removal* area, Set the *Depth from topo* to 2 feet.

17. Leave the default zero offsets in the *Inside* tab.

18. Set the *Outside* controls to *Slope stake – base*, 3 feet *Offset* (both sides as in the figure above). This will strip 3 feet *outside* the template footprint.

19. Press the *Add/Edit* button.

20. Press *OK* to accept changes and close the dialog box. Respond *Yes* to the re-calculate prompt.

21. Zoom in on the cross section left or right hand side. Notice that the stripping line is displayed (figure below).

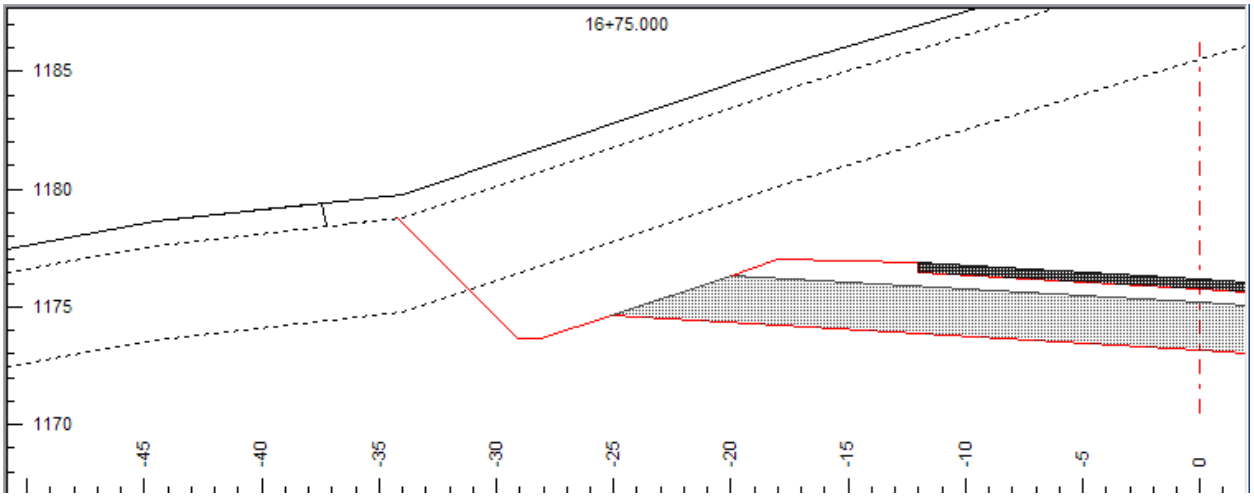


Figure 16.6: The top ground layer has been stripped off.

The total volume of cut will have been reduced and fill increased. The OB cut volume will now be zero and there is now a non-zero *Stripping* volume available for reporting.

Some important facts about stripping:

- Stripped material is excluded from the mass haul – it is assumed to be unsuitable for fill.
- The depth stripped will be the value assigned in the Site Prep dialog or the top surface layer thickness, whichever is **least**. In the above example the top layer (OB) is only 1 foot thick so that is the stripped depth.
- Stripping happens before the template is applied to a cross section.

Using materials to control templates

The templates we have used so far have fixed cut and fill slopes. In the following steps we will change the final slopes component so that it extracts slopes from the materials in the ground or in the fill.

22. If you had trouble with the previous steps or if you wish to start here, open **Examples\Location\Align stage 6.dsn**.

23. Choose menu *Edit | Assign Parameters by range* or press the tool bar button to open the *Assign Attributes by Station* dialog box. Select the *Templates* tab (figure below).

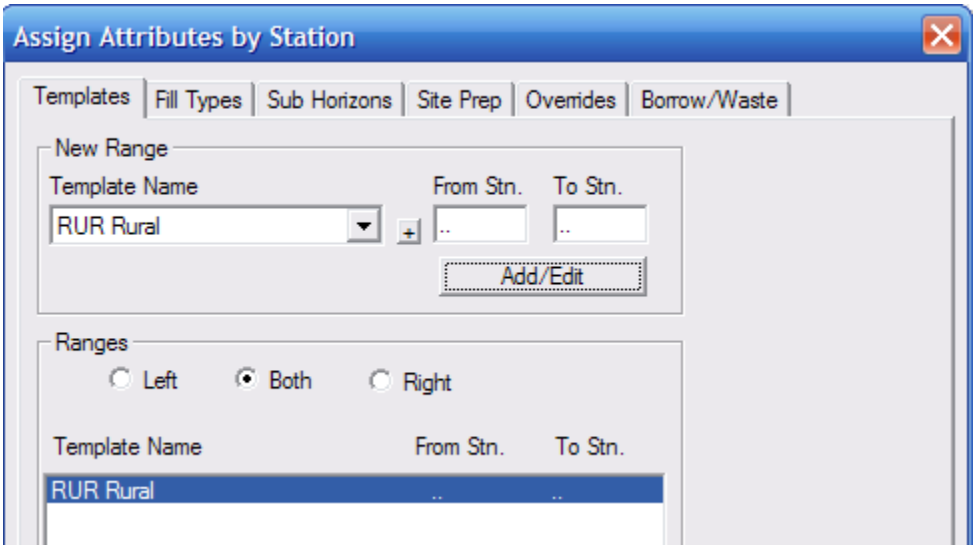


Figure 16.7: Assigning the RUR rural template to the entire alignment

24. In the *Template Name* control, choose *RUR Rural*. Leave the default ".." in the *From/To* fields.

25. Press the *Add/Edit* button, this will apply the new template to the entire alignment**.** The dialog box should appear as in the figure above.

Note: The most common mistake made in the assignments dialog box is to skip the *Add/Edit* step. If you Press *OK* before the ranges are updated, nothing happens.

26. Press *OK* to return to the main screen. Respond *Yes* to "Recalculate road alignment".

27. Select menu *View* | *Jump to Station* (or <Ctrl-J>) and type station 1675. Press *OK* to update the current section.

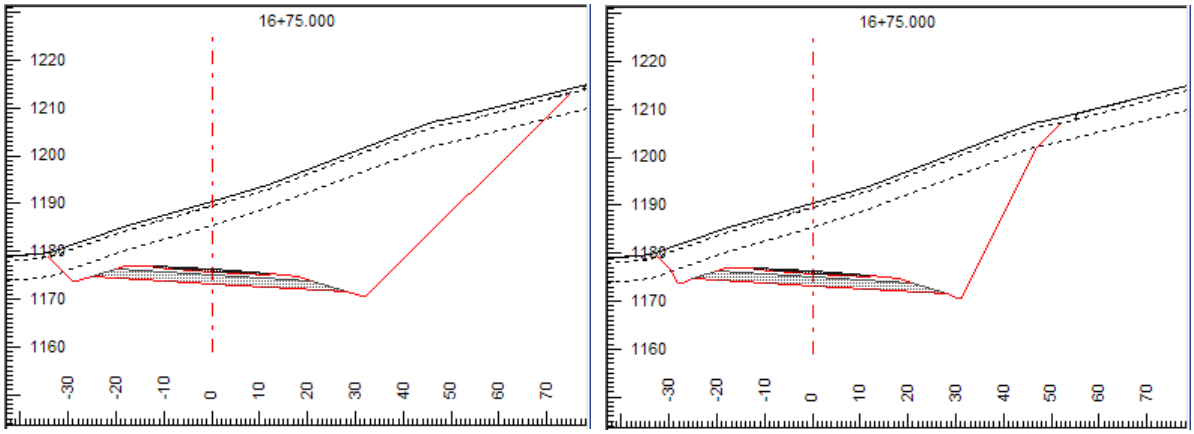


Figure 16.8: Template on the left has fixed slope values, while the one on the right has slopes set to Auto.

The *RUR Rural* template has the cut and fill slope parameters set to automatic. The result of this change is to significantly reduce the cut material due to the steeper cut angle in the deeper FR layer.

This technique is used in the Culverts exercise later.

28. Choose menu *File* | *Close*. Do not save changes.

17. Templates - Introduction

Cross section templates allow you to set parameters such as road width, surfacing depths, shoulders, ditches and cut/fill slopes.

Templates interact with topography, super-elevation, sub-surfaces, and alignment(s) to produce final design cross sections. It is important to understand that templates are not static; they adapt to each cross section.

Users will typically create a set of standard templates for use in common design situations. *Templates*, *template components* and road *class specifications* are stored in a template table.

This example will introduce you to templates and the *Template Table Editor*.

1. In the Location module, open **Examples\Location2\bluff_road.dsn**.

Note: If you are using RoadEng® Lite or RoadEng® Civil Assistant and the "Non Permitted Functions Found in File" dialog appears, choose "Keep all functions and revert to DEMO Mode".

2. Choose menu item *Edit /Edit Templates*, or press tool bar button , to open the *Template Table Editor* shown below.

3. Click on the slope right button (to the right of the graphic) to prepare for the following section.

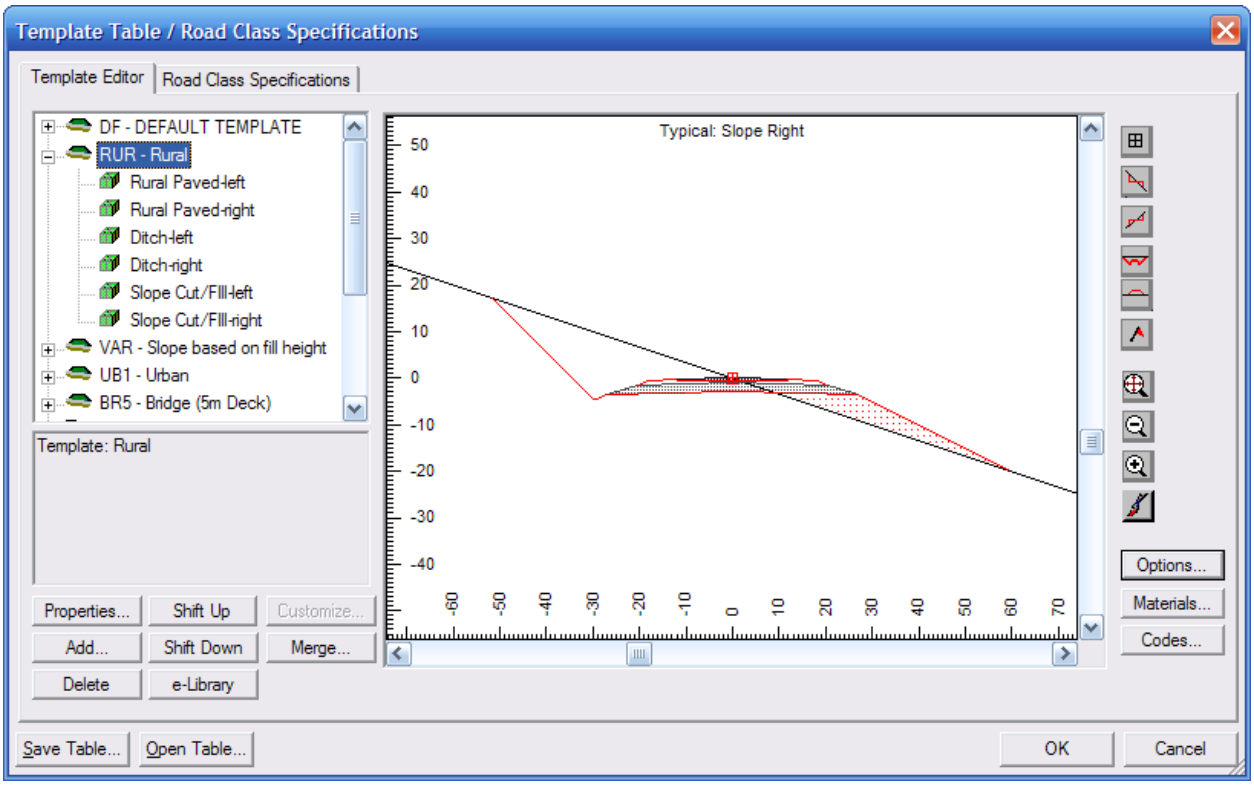


Figure 17.1: Template Table Editor *dialog Box.*

4. Click on the + sign beside the template *RUR-Rural* to view its components as shown in the figure above.

Template properties

The template editor shows you the templates contained in a table and the components contained in each template.

5. Right click on the *RUR-Rural* template and choose *Properties*.

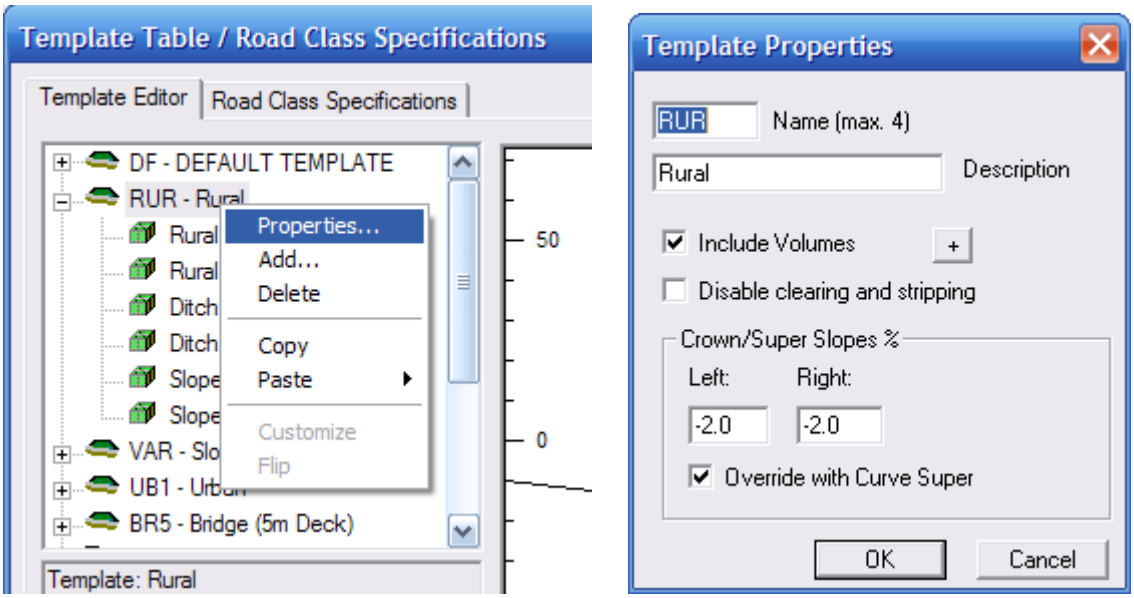


Figure 17.2: Template Properties Dialog.

There are relatively few controls in the *Template Properties*, most template flexibility is at the *Component* level. Aside from the template *Name* and *Description*, the most commonly used properties are the cross fall slopes.

6. Change the *Crown/Super Slope* to -5% on the left and +5% on the right. Press *OK*.

Notice how the Roadway component is altered by the new crown slopes – this is what happen when this template is applied inside a curve with 5% super-elevation (the template property *Override with Curve Super* must be enabled). Some components are designed to adjust themselves to the prevailing crown or super-elevation slope.

7. Re-open the *RUR-Rural* template *Properties* and reset the crown to -2 both sides, Press *OK*.

Creating and Deleting templates

Although there is an *Add* button (and context menu), you will find the most intuitive way to create a new template is to copy an existing template, paste it back into the table and then re-name and modify it.

8. Right click on the *RUR-Rural* template and select *Copy*.

9. Right click again and select *Paste | As New*. The new template will appear at the bottom of the list.

10. Select the new *xxx-Rural* template and use the *Shift Up* button to move it up the list.

You could also open the properties and rename the template to RUR2 or similar. You would also want to change at least one property or component to make the template different in a useful way.

Note: The fewer templates you have the easier it is to maintain them.

11. Right click on the new *xxx-Rural* template and select *Delete* to remove it.

Template Components

There are four types of template components:

- Custom
- Roadway
- Ditches
- Slopes

Roadway, *Ditches*, and *Slopes* components are included for backward compatibility and their behavior is mostly self explanatory (and there is always <F1>). Custom components have replaced and improved upon their features. You can tell when you are looking at an old style component – the properties dialog box is quite different from the *Custom* components properties (figure below).

In this document, we will work exclusively with *Custom* components.

Template Component properties

12. Right click on the *Rural Paved-left* component of the *RUR-Rural* template and select *Properties* (or double click) to open the *Custom Component Properties* dialog box (figure below).

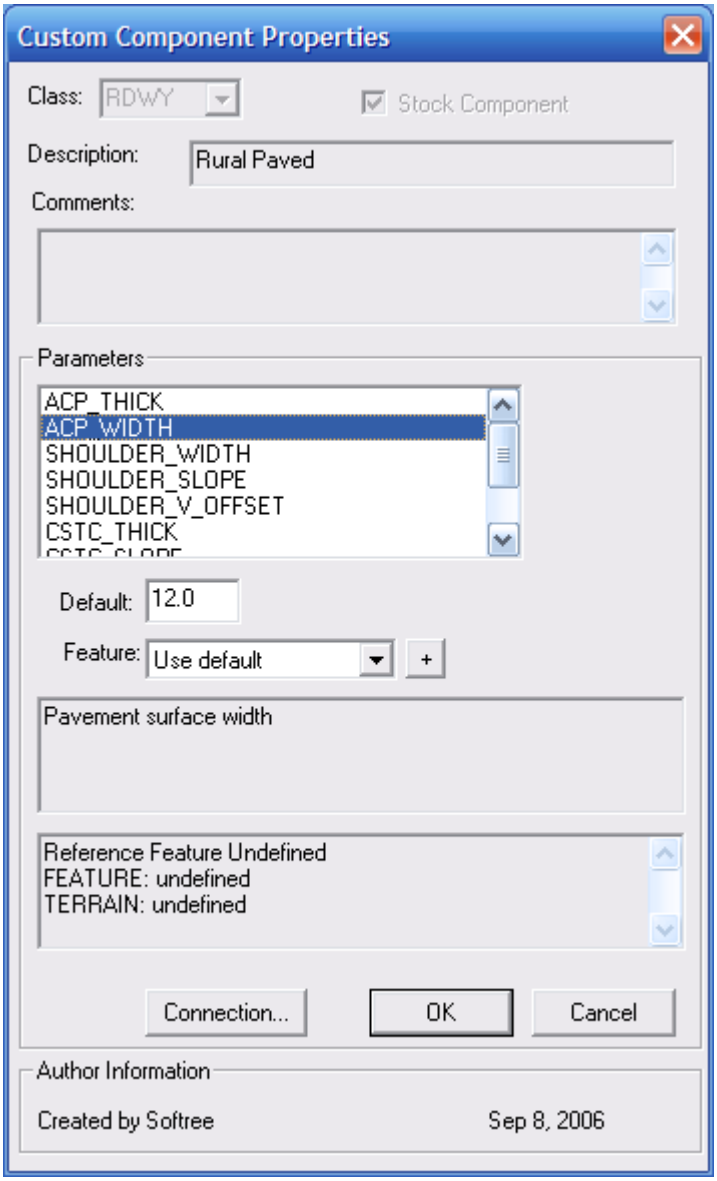


Figure 17.3: Rural Paved *roadway component properties*

This component allows you to change various parameters such as pavement thickness, lane width shoulders, sub-base and base thicknesses and slopes.

13. Change the lane width by selecting *ACP_WIDTH* in the *Parameters* list and setting the *Default* value to 16.

14. Press *OK*. to accept changes and close the properties dialog box. Notice the road has widened on the left.

15. Similarly, open the *Ditch-left* properties dialog box (figure below left).

16. Note the available parameters and then press *Cancel* to close the properties dialog box.

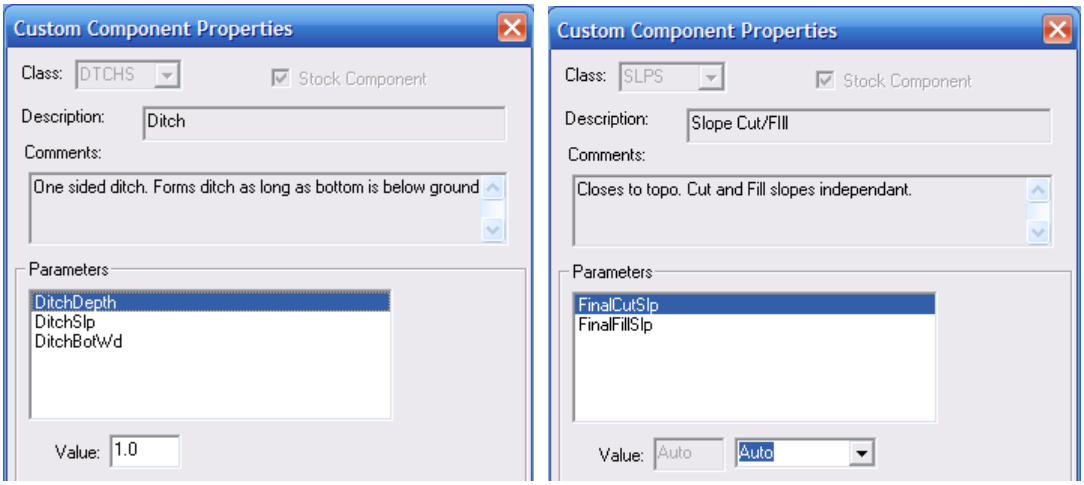


Figure 17.4: Ditch and slopes component properties.

17. Open the *Slope Cut/Fill-left* properties dialog box (figure above right).

In this component you can change the final closing slope. If *Auto* is chosen, as is shown in the figure above, the slope will be taken from the *material types* found in the ground (cut slope) or fill (fill slope). Choosing anything other than *Auto* will fix the slope and ignore the material types.

18. Press *Cancel* to close the properties dialog box.

The ground slope buttons (shown at the right) allow you to view typical cross sections.

Slope Right

Slope Left

Cut

Fill

19. Select the slope left button  . Notice how the black ground line changes and how the template accommodates.

20. Click and drag in the template graphic area. Note that you can change the template position and see how it will behave in different situations.

Since Softree added this ability to change the template position (up and down, for example) the difference between *Cut*  and *Fill*  cross sections has become irrelevant.

Note: The middle mouse pan and zoom functions work in the template graphic area.

21. Click on the Split screen view button in the upper right corner  . The screen shown below will appear showing four ground situations at the same time. Each template position can be adjusted with a mouse click and drag.

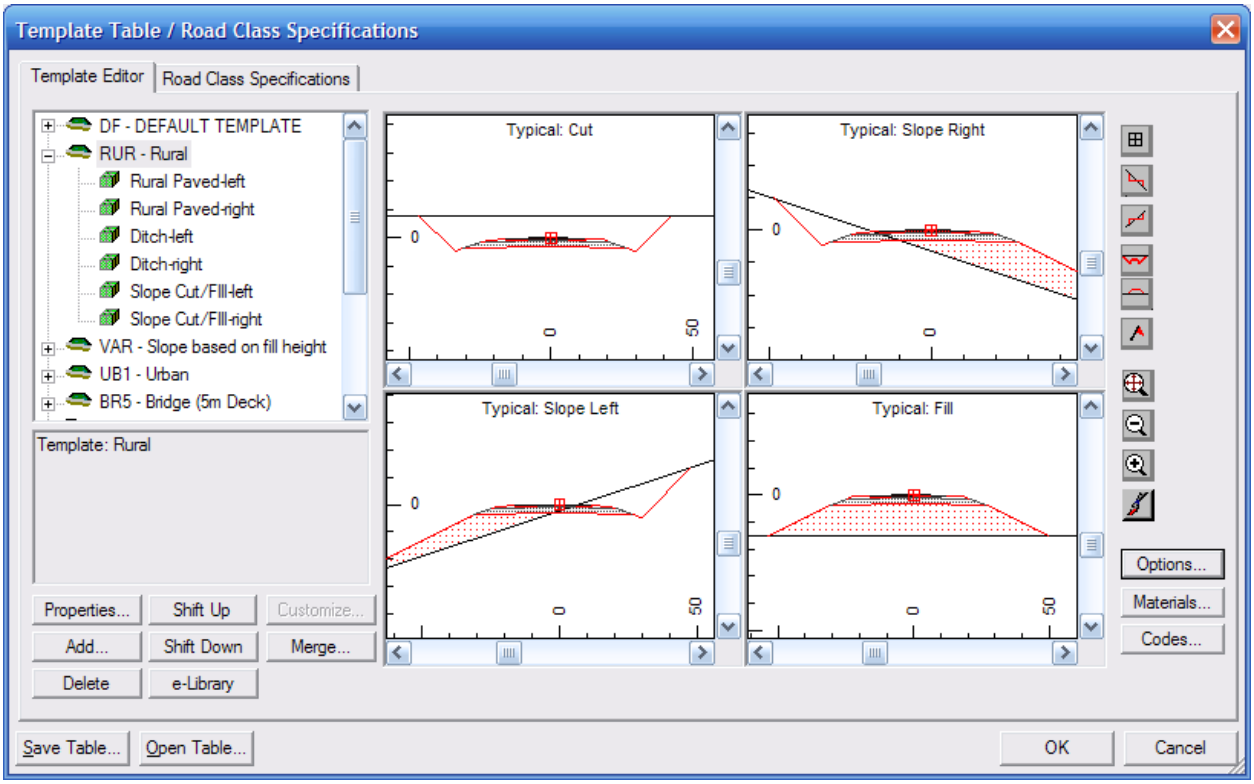


Figure 17.5: Split screen view showing the 4 typical sections.

22. Click on the station button  on the right hand side of the screen, and enter the value 300. Press *OK*.

On the screen you will see the template applied to station 300 of the design. This allows you to quickly see how the template will appear before it is assigned.

23. Click on the slope right button  to prepare for the following section.

Working with Components

Template *components* are interchangeable building blocks. A template table can also contain optional folders containing re-usable components.

24. Scroll down the template tree list and open the *Slopes Components* folder (figure below).

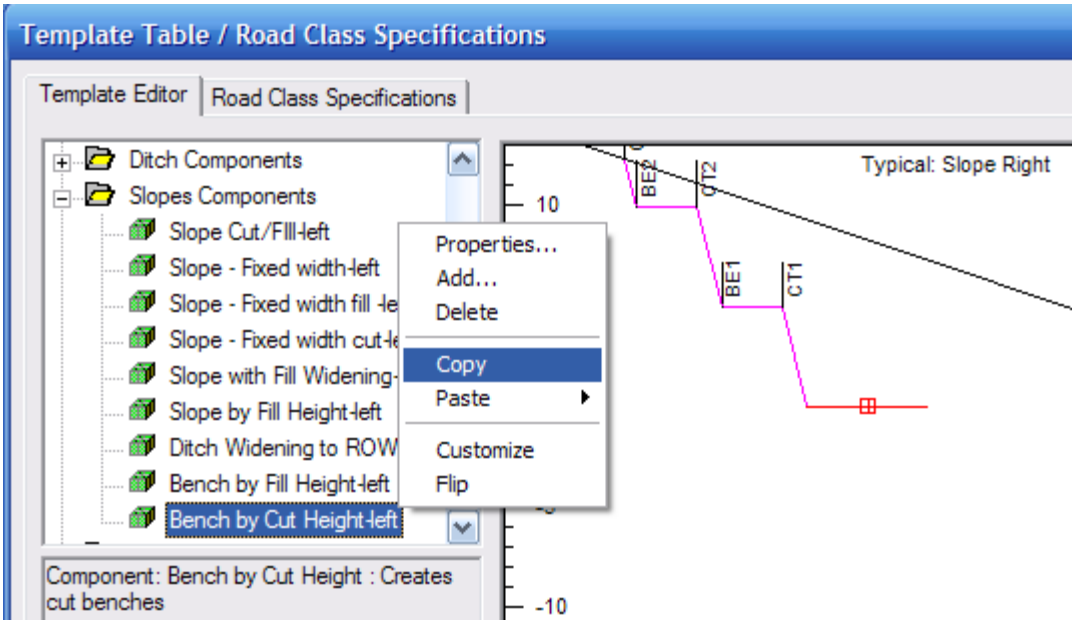


Figure 17.6: Copying a component from a Folder.

25. Right click on the *Bench by Cut Height-left* component and select *Copy*.

26. Scroll up until you can see the *RUR-Rural* template again.

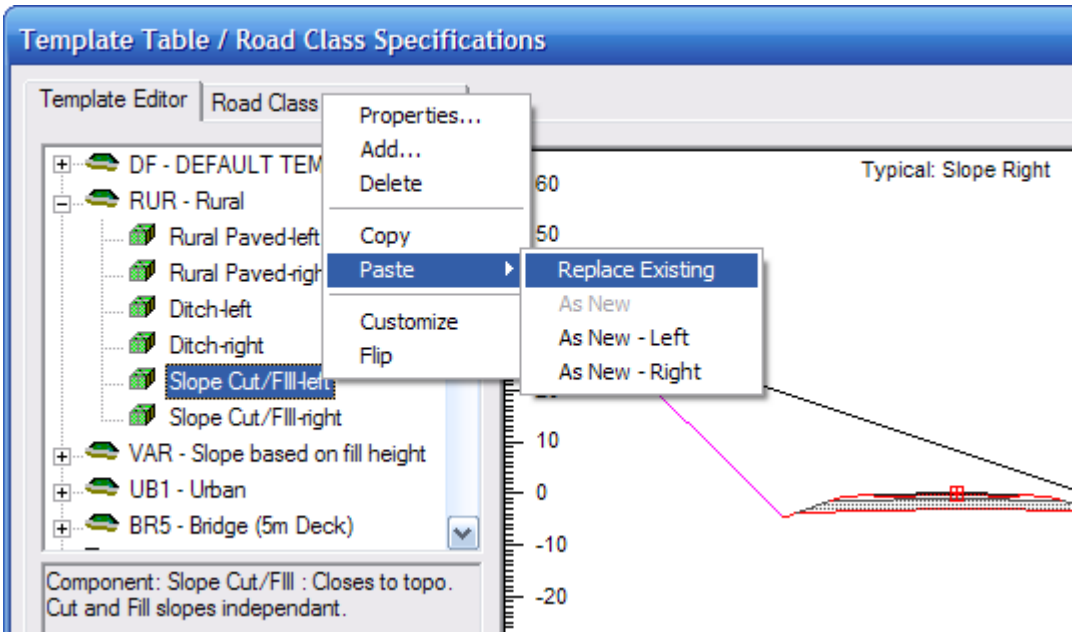


Figure 17.7: Pasting a component into a Template.

27. Right click on the *Slope Cut/Fill-left* component in the *RUR-Rural* template and choose menu *Paste* | *Replace Existing* (figure above).

28. Click and drag the template down until you can see the cut benches – figure below (you may also need to zoom out).

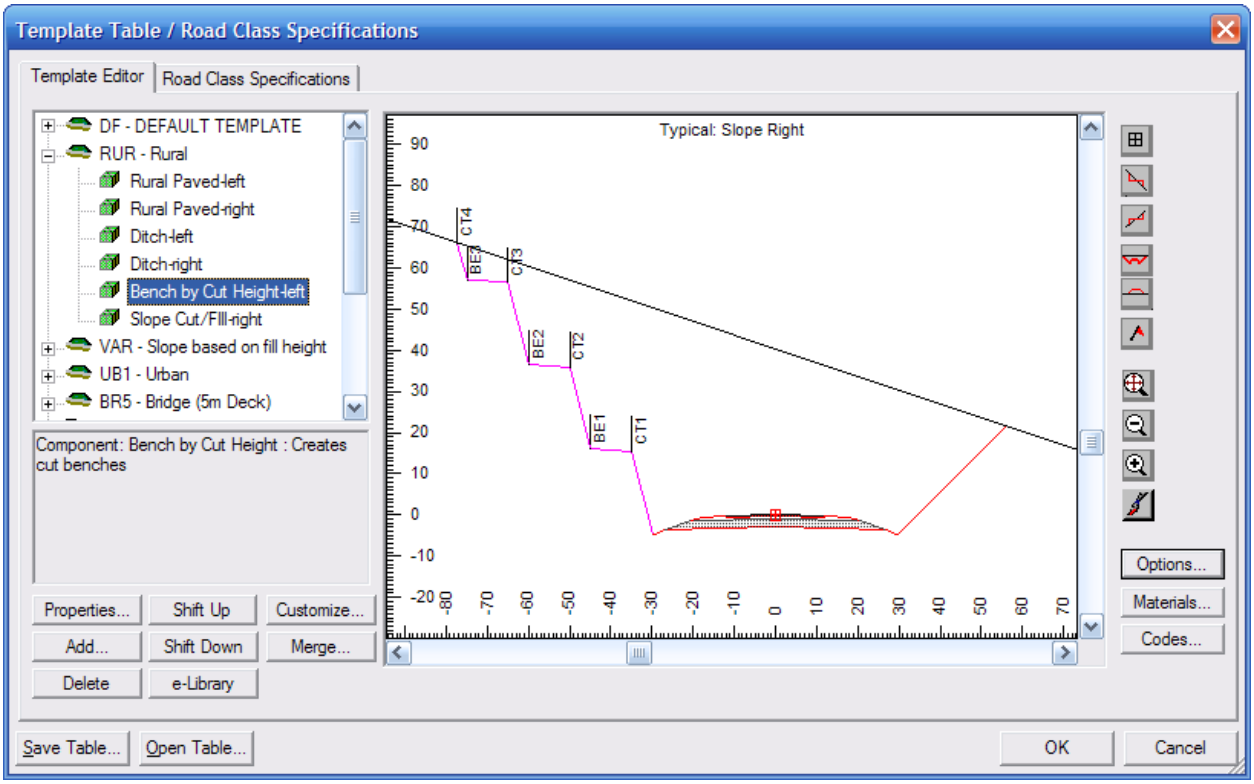


Figure 17.8: The new slopes component is only on the left side.

At this point you might want to change the new component's properties. Is the bench wide enough? Are the steps high enough?

29. Open the properties dialog box for the new component and change *BenchWidth* to 15. Press *OK* to see the change.

Notice that the cut bench component is only on the left side. We will copy it (with its new bench width) to the right.

30. Right click on *Slope Cut/Fill-right* and choose menu *Delete.*

31. Right click on *Bench by Cut Height-left* and select *Copy*.

32. Right click again and select *Paste | As New - Right*. Notice the template now has cut benches on both sides.

Note: The order of components is important; components should be arranged from the center line out. The left/right order is unimportant.

33. Use the Shift Up button to move your cut bench component to the top of the list. Notice what happens to the drawing.

34. Restore the order.

The e-Library

Softree maintains a library of template components on the web.

35. If you are connected to the web, press the e-Library button to open the dialog box in the figure below

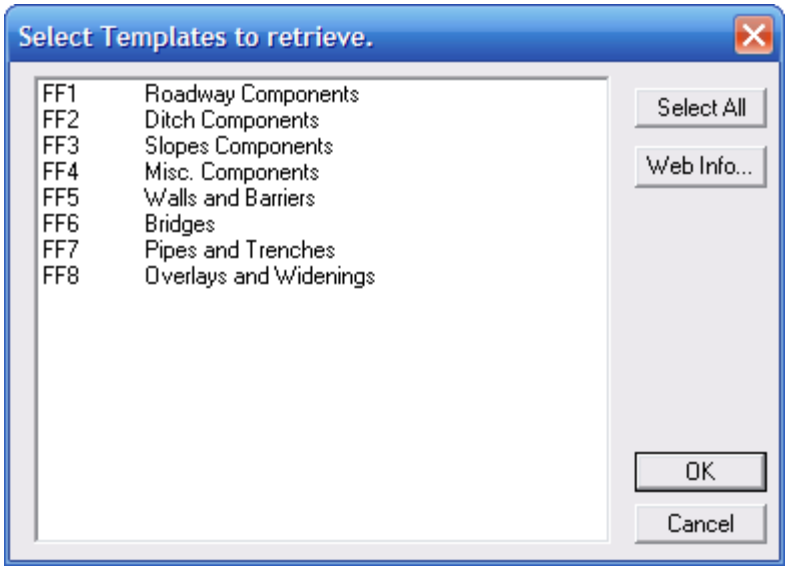


Figure 17.9: Component folders available from Softree's web site.

36. Press *Cancel* to close the dialog box.

Template symmetry

Although it is possible to make two sided template components, it is strongly recommended that you make and use one sided components. Two sided components have twice as many parameters; with a one-sided component you can set up the parameters on the left and then duplicate the component for the right.

There are situations where you need asymmetric templates (a guard rail on one side for example); however, you can accomplish this by assigning one template (guard rail) to one side of the road and another template (no rail) on the other side. See *Template Assignments* exercise following.

It is also possible to over ride the values of template parameters along the alignment. This reduces the number of templates you need in your table. See *Template Assignments* exercise following.

37. Press *Cancel* to exit the template editor and then *File* | *Close*, do not save changes.

18. Template Assignments

Assigning a Roadside Barrier to a Range of Stations

Templates can be assigned to a range of stations. The following example will demonstrate how this is done by adding a road side barrier to one side of a road.

1. In the Location module, open **Examples \Location2\bluff_road.dsn**.

Note: If you are using RoadEng® Lite or RoadEng® Civil Assistant and the "Non Permitted Functions Found in File" dialog appears, choose "Keep all functions and revert to DEMO Mode".

Creating a new template

2. Select menu *Edit* | *Edit Templates* or press tool bar button , to open the template table editor.
3. Right click template *RUR-Rural* and select menu *Copy*.
4. Right click again and select menu *Paste* | *As New* to create a new template. The new template (*xx0-Rural*) is highlighted and appears at the bottom of the list.
5. Use the *Shift Up* button to move the new template to just under *RUR-rural*.
6. Click on the Properties button and change the *Name* of the new template to BAR and the *Description* to "Rural Paved with Barrier". Press *OK*.

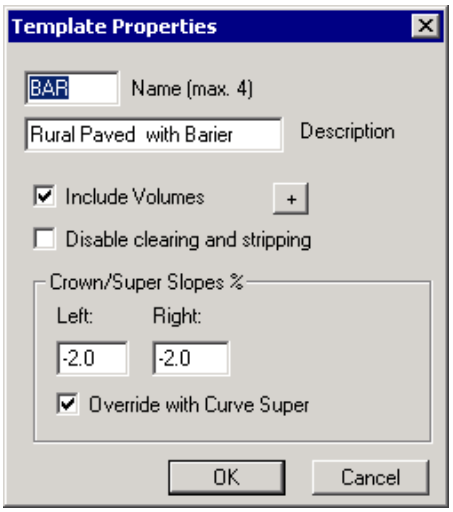


Figure 18.1: Template Properties

Now that you have a new template, you need to add the barrier component to it.

7. Open the *Walls and Barriers* folder and choose *Barrier-left*. Right click and *Copy* it to the clipboard.

8. Locate new template *BAR- Rural Paved with Barrier* template created above. Right click and choose menu *Paste* | *As New* to add the new barrier component. It will appear at the bottom of the components list.

9. Open the *Properties* of the *Barrier-left* component and change the *CL_OFFSET* parameter to **15**. Press *OK* to exit the properties dialog box. Your template should now appear as in the figure below

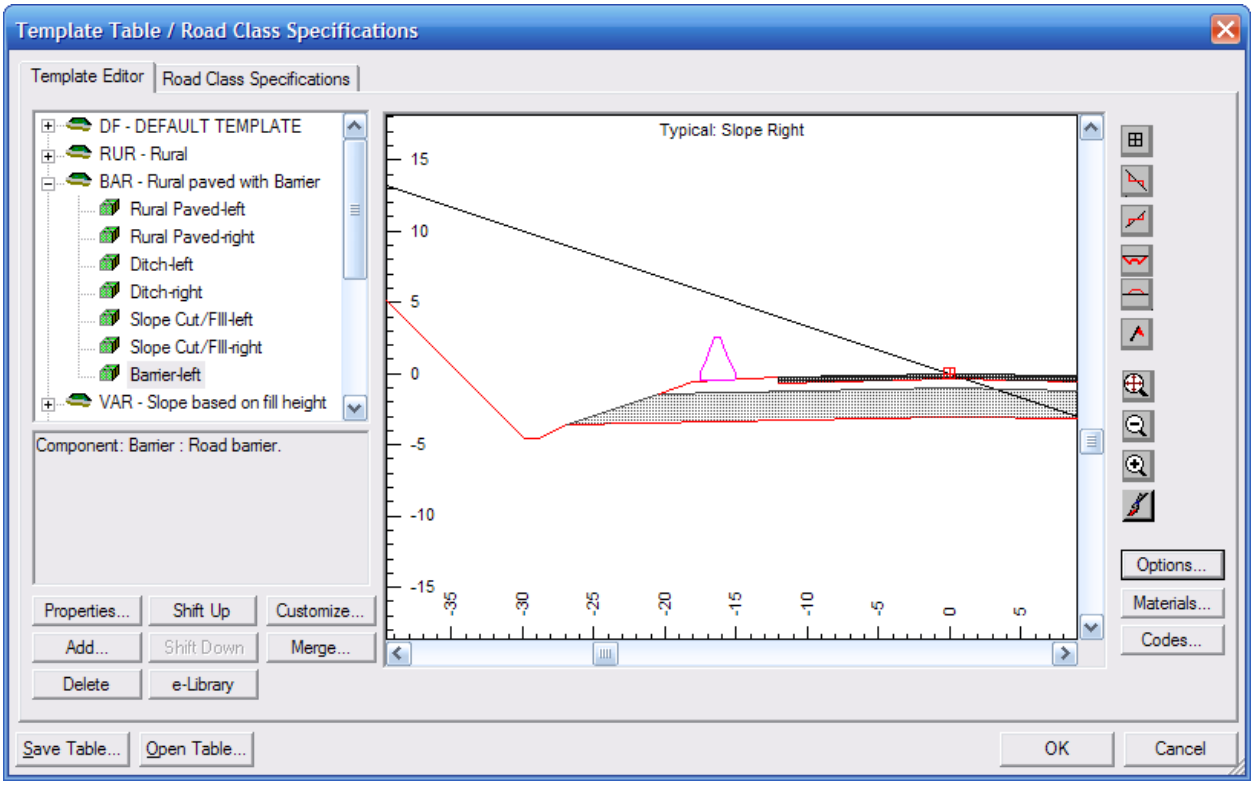


Figure 18.2: Template with Barrier

To make this template more useful, we will put the barrier on both sides

10. Right click on *Barrier-left* and *Copy* it to the clipboard.

11. Right click again and select menu *Paste* | *As New- Right.*

12. Press *OK* to accept the changes and close the template editor. It is all right to respond *No* to the recalculate prompt because the new template has not been assigned yet.

Assigning the template

13. If you had problems with the previous steps or if you wish to start here, open **Examples \Location2\bluff_road-2.dsn**

14. Choose menu *Edit* | *Assign Parameters by range* or press the tool bar button to open the *Assign Attributes by Station* dialog box (figure below). Select the *Templates* tab.

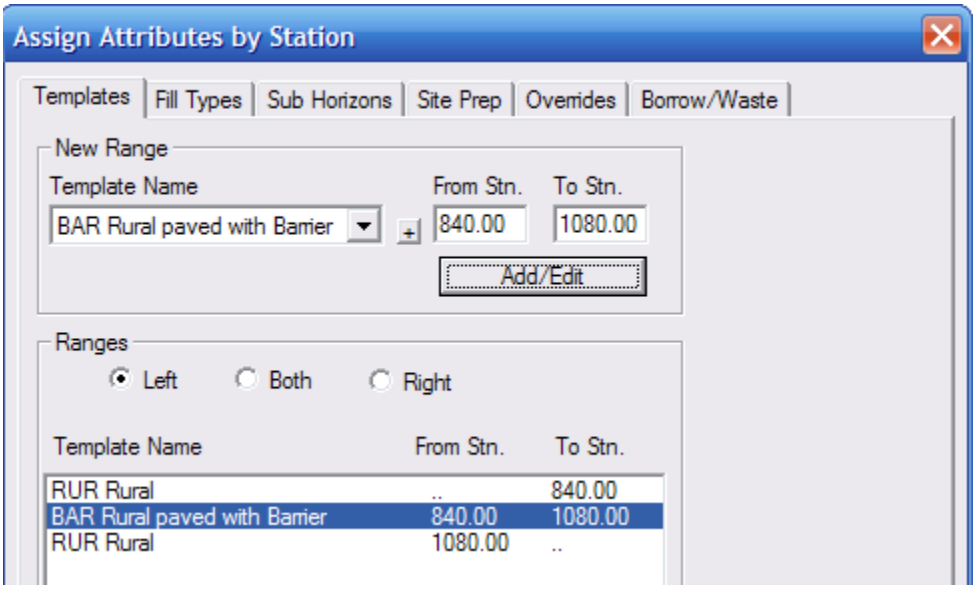


Figure 18.3: Assigning a template to a station range.

The barrier will be placed between stations 840 and 1080 but only on the left hand side.

15. In the *Ranges* area, select *Left*. Do this first because it resets the template name and range fields.

16. In the *Template Name* control, choose *BAR Rural Paved with Barrier*. In the *From Stn*. edit box enter **840** and in *To Stn* enter **1080.**

17. Press the *Add/Edit* button**.** The dialog box should appear as in the figure above.

Note: The most common mistake made in the assignments dialog box is to skip the *Add/Edit* step. If you Press *OK* before the ranges are updated, nothing happens.

18. Press *OK* to return to the main screen. Respond *Yes* to “Recalculate road alignment”.

19. Select menu *View* | *Jump to Station* (or <Ctrl-J>) and select station 1000. Press OK to update the current section.

Adjust the view in the Section window so you can see the road side barrier.

20. Choose menu *File* | *Close*. Do not save changes.

19. Template Parameter Overrides

The previous section demonstrated how an entire template can be assigned to a range of stations. To do this a new template was created and assigned to a station range.

It is often desirable to change a single template parameter such as road or shoulder width, ditch depth, etc. for a range of stations. *Template Parameter Overrides* provides an easy way to do this.

Creating a Turning Lane

This example will demonstrate parameter overrides by creating a turning lane at an approach to an intersection.

1. Open **Examples\Location2\bluff_road.DSN**.

Note: If you are using RoadEng® Lite or RoadEng® Civil Assistant and the "Non Permitted Functions Found in File" dialog appears, choose "Keep all functions and revert to DEMO Mode".

2. Choose menu *Edit* | *Assign Parameters by range* or press the tool bar button to open the *Assign Attributes by Station* dialog box (figure below). Select the *Overrides* tab.

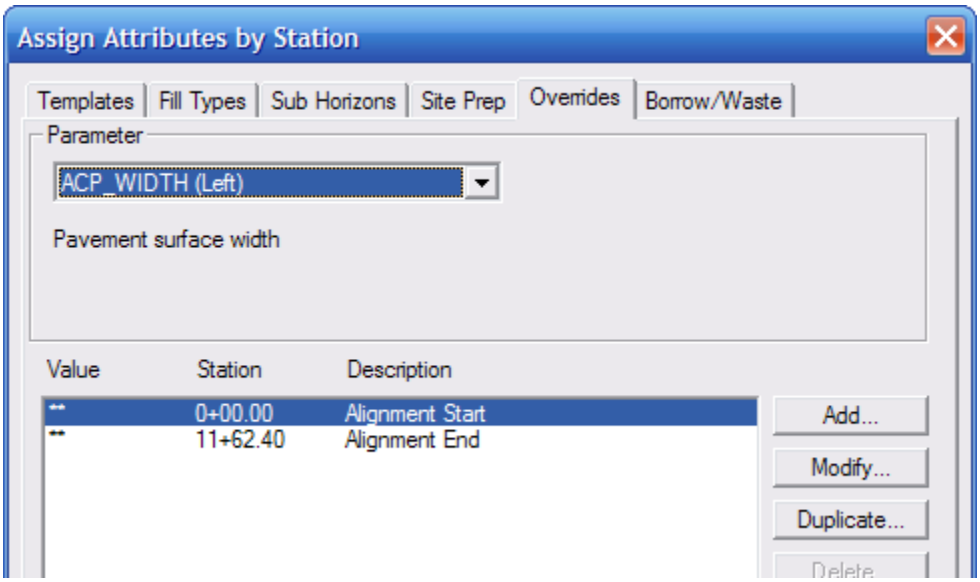


Figure 19.1: The assign template overrides dialog box.

3. In the *Parameter* control, choose *ACP_WIDTH (left)*.

4. Click on the first entry in the override list (*Station* 0+00) and press the *Modify* button. Turn off *Use Default* and set the *Value* to **30** (figure below left). Press *OK*.

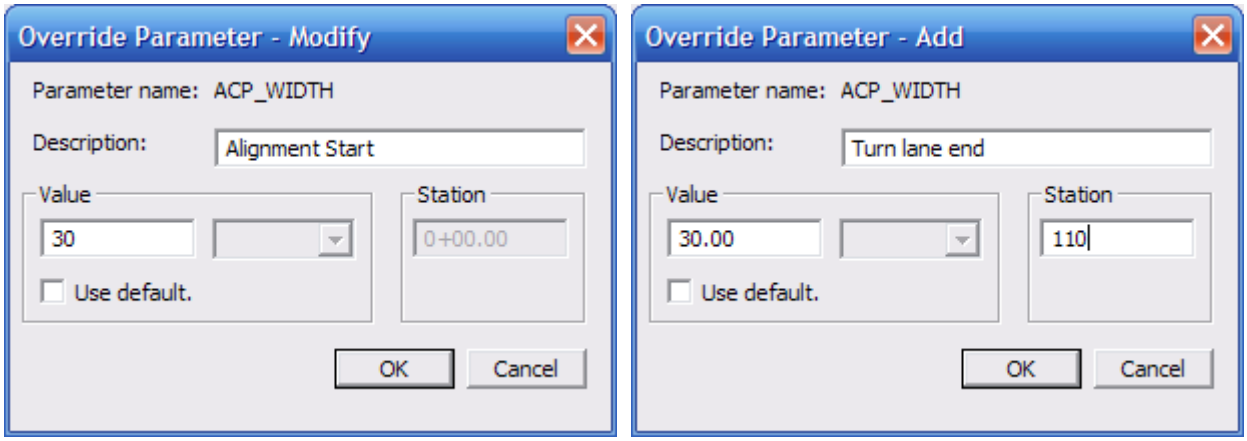


Figure 19.2: Override Parameter Dialog.

5. Press the *Add* button and add another *Value* of **30** at *Station* **110**. Also change the *Description* to read "Turn lane end" (figure above right). Press OK.

6. Press the *Duplicate* button and set the *Use default* check box. Change the *Station* to **200**. Change the *Description* to "Turn lane taper end". Press *OK*. Your override list should be the same as the figure below.

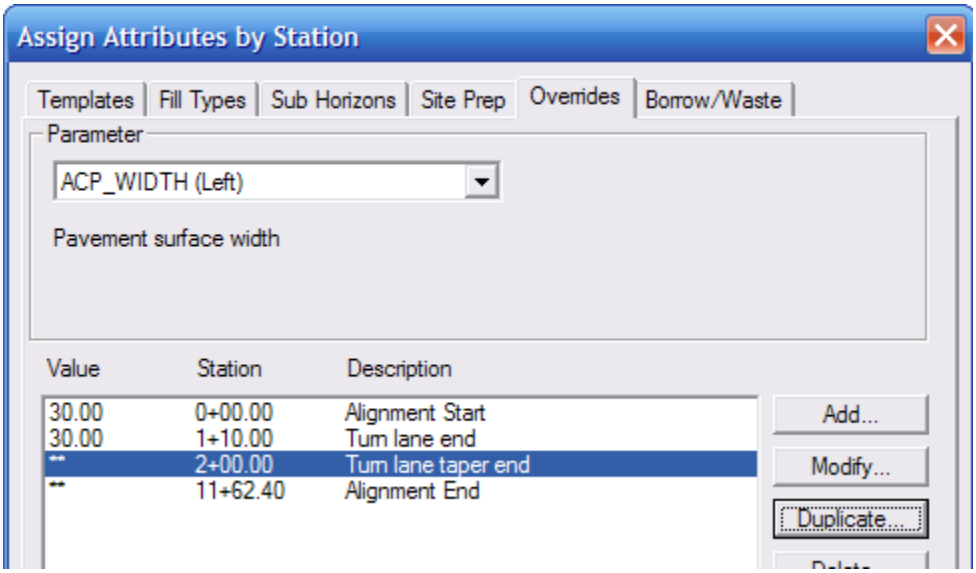


Figure 19.3: Override list for turning lane at start of alignment.

7. Press *OK* to return to the main screen. Respond *Yes* to "Recalculate road alignment".

8. Scroll and zoom the plan window to the beginning of the design. Notice the road edges in blue now display the additional lane width.

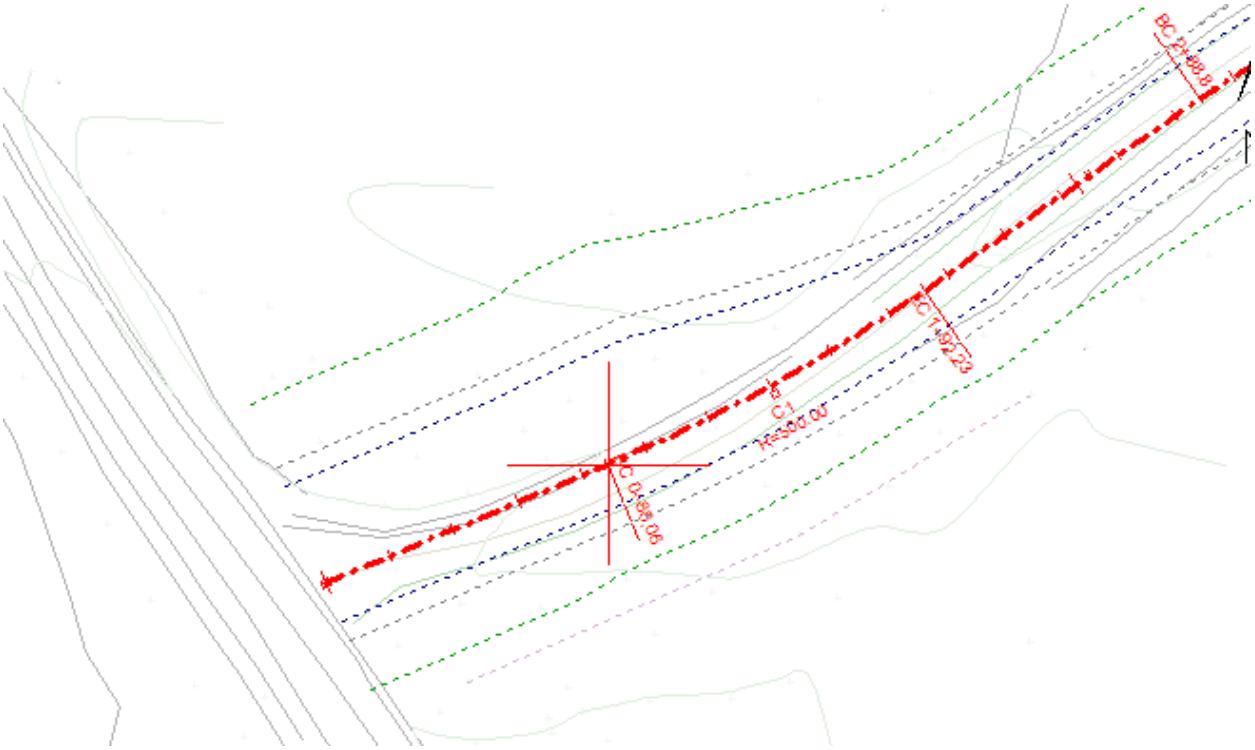


Figure 19.4: Turning lane defined by template parameter overrides.

9. Choose menu *File* | *Close*. Do not save changes.

20. Templates - Advanced

RoadEng templates are very flexible. Templates can be built to accommodate a wide range of design situations. Building custom templates can be quite complicated. It is best done by individuals with a computer programming aptitude. Careful design and testing is required to ensure a custom template functions correctly. Fortunately, most users do not have to get down to this level of detail. Template components allow template programmers to package their work in an easy to use format for use by others.

This section will describes some of the basic concepts required to apply template components. The details of creating template components are not covered. Before we begin some basic concepts and definitions are required.

Template Components Concepts

Template Parameters

Template components have parameters allowing you to configure the object for your specific design. Template parameters can be any one of the following:

User	This is the most common type of parameter. It can be a numerical value or a slope percent.
Reference Feature X Offset	Allows you to specify an optional horizontal alignment instead of a numerical offset from center-line. See Reference Features for more information.
Reference Feature Y Offset	Allows you to specify an optional vertical alignment instead of a numerical offset from center-line. See Reference Features for more information.
Reference Surface	Allows you to specify a surface.
System	This variable is a predefined value such as super elevation or curve widening. It is only of interest to a template programmer and will not be discussed here.
Expression	This variable is used to enter a mathematical expression. It is only of interest to a template programmer and will not be discussed here.

Codes

Each template component has a set of pre-defined template codes. These point codes can be displayed in the Plan, Profile, Section or Data windows. In Profile and Plan the codes are connected to form linear features such as a ditch-line or sidewalk offset.

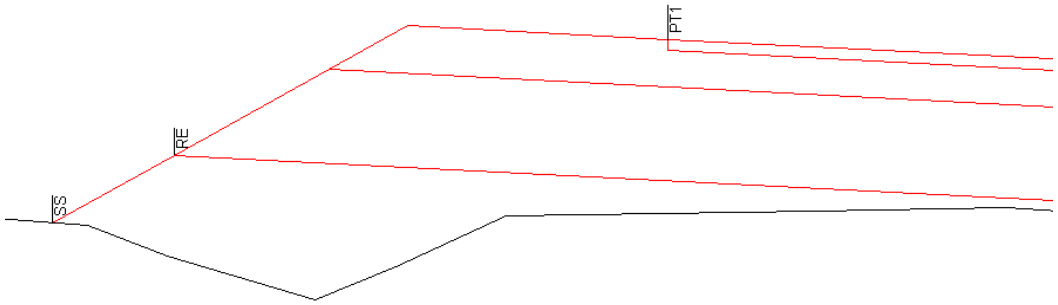


Figure 20.1: Template Codes

Surfaces

Template surfaces are used to track and report material volumes. Each template can define up to 16 surfaces plus sub-grade. Material volumes are calculated between surfaces. Thus we can calculate and report cut and fill volumes below the sub-grade surface and up to 16 material fill volumes.

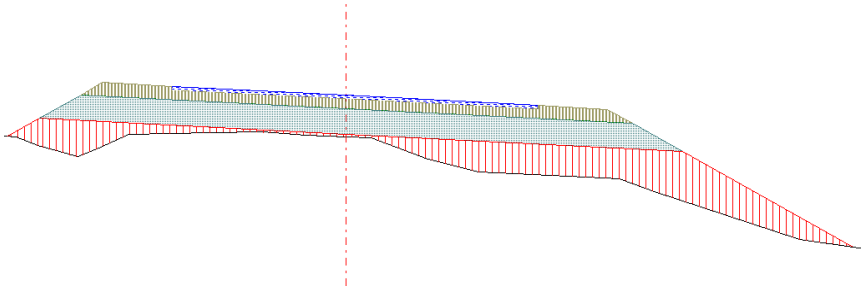


Figure 20.2: Template Surfaces and Enclosed Materials

Display and Reporting of Ditch Lines

1. Open **Examples\Location\bluff_road.dsn**.

Note: If you are using RoadEng® Lite or RoadEng® Civil Assistant and the "Non Permitted Functions Found in File" dialog appears, choose "Keep all functions and revert to DEMO Mode".

2. Choose menu *Edit* | *Edit Templates* or press tool bar button , to open the template table editor.

Formatting template layers

3. Press the Options button (lower right) to open the *Template Display Format* dialog box (figure below).

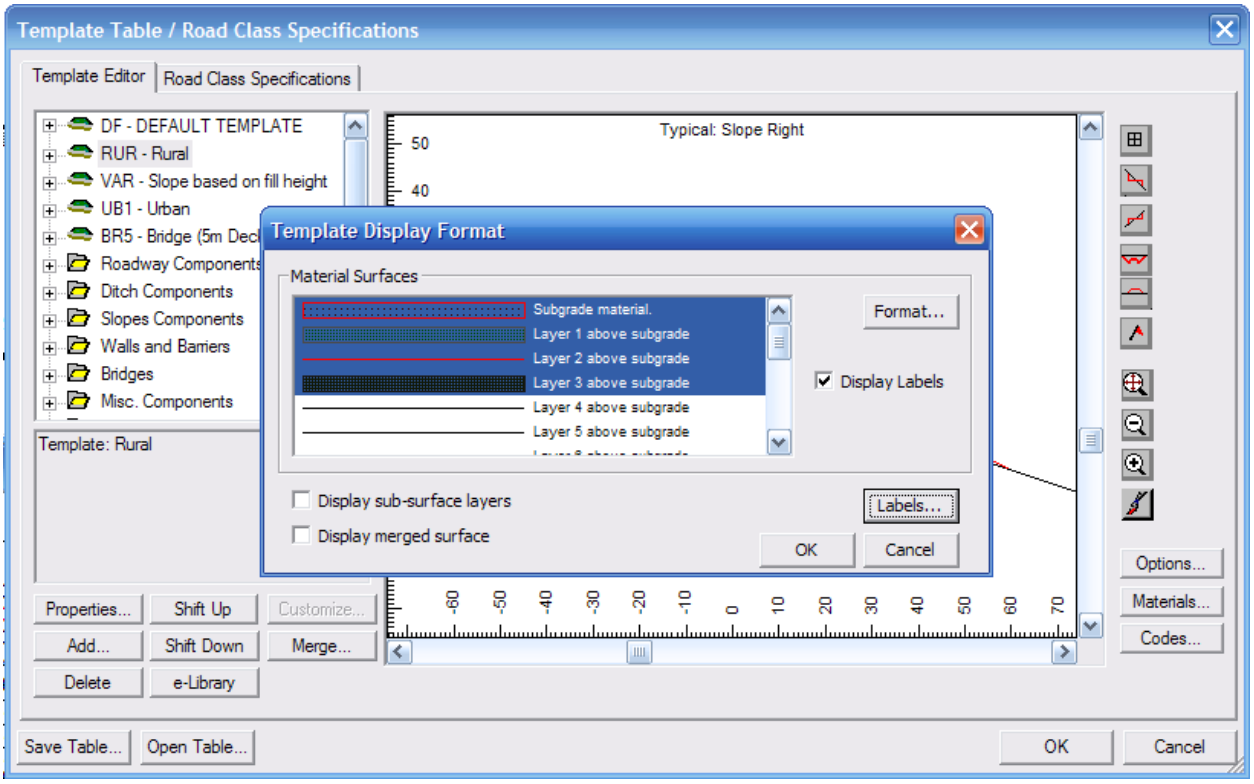


Figure 20.3: Template table editor and the Template Display Format *dialog.*

4. Select the first four items (<Ctrl> click each item) and turn on *Display Label*.

In version 5.1, a combo box below the list has been added to allow you to *Show* layers for either the *Current Template*, *Assigned Templates* or *All surfaces*.

5. Open the Label Selection and Formatting Dialog box (figure below).
    a. If you are using version 5.0, press the *Labels* button (lower right figure above).
    b. In version 5.1, go to the *Labels* section on the **left** side of the dialog box and press the *Format* button.

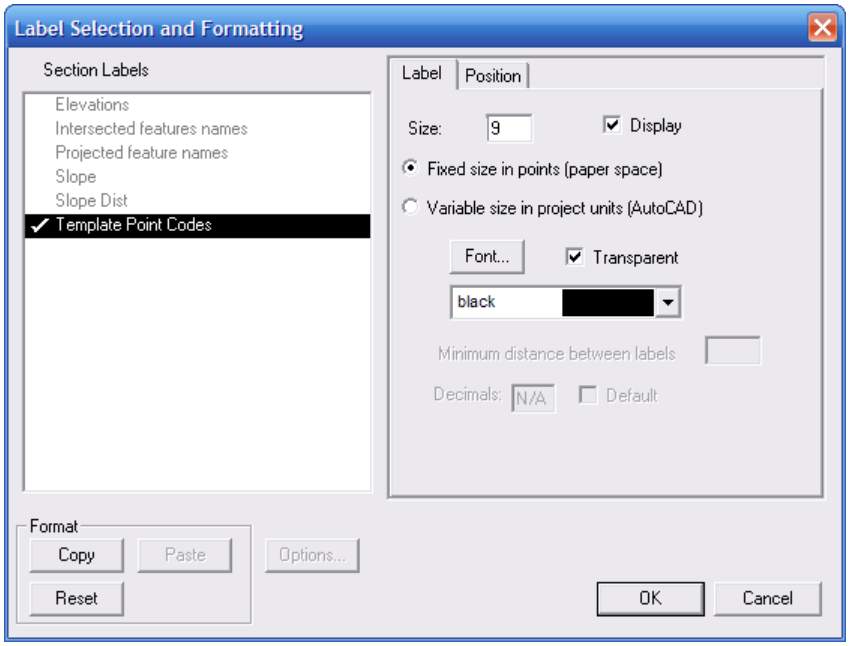


Figure 20.4: Label Selection and Formatting Dialog

6. Ensure that only *Template Point Codes* labels are displayed. You can also change label font, color and position in this dialog box (we will keep the current values). Press *OK* to close the dialog box.

7. Select only the *Subgrade material* item and then press the *Format* button to open the *Line-types and Symbols* dialog box (figure below).

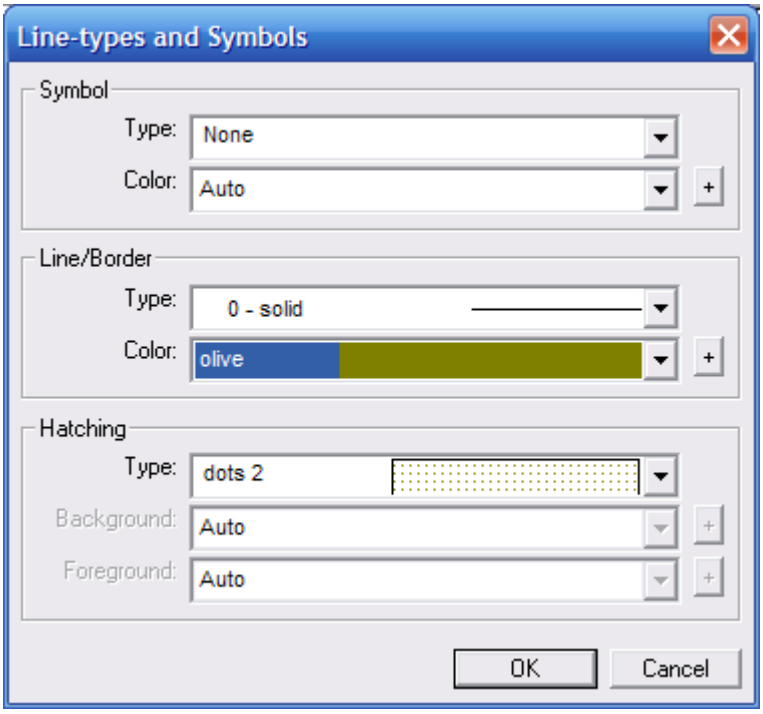


Figure 20.5: The Line-types and Symbols *dialog box allows you to change line style (including symbols), hatch style and color*

8. Change the color of the subgrade to olive (figure above) and press OK to accept changes and close the dialog box.

9. In version 5.1, set the *Hatch Fill areas* check box.

10. Press *OK* again to return to the template editor. If you click and drag your template up, you will see the olive colored fill.

Note: The above formatting is displayed in the section window and can also be accessed from the *Section Window Options* dialog box.

11. Click on the + sign beside the template *RUR-Rural* (or double click) to view its components as shown in the figure below.

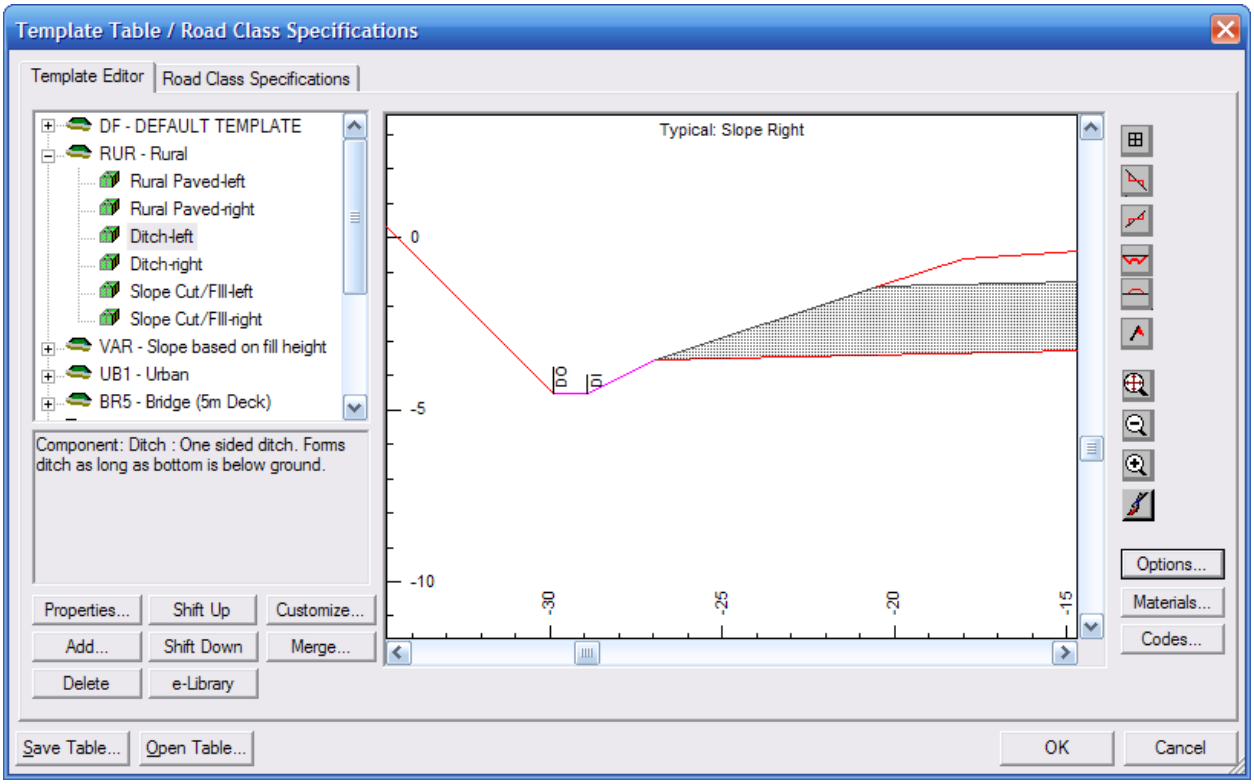


Figure 20.6: Ditch template codes displayed in the template editor.

12. Select each of the components and notice how the line work associated with the selected component is highlighted in magenta.
    a. In version 5.0, codes are only displayed for the selected component; the Section window, however, will display all codes (**if** labels are enabled).
    b. In version 5.1, tooltips will display point codes (and other information) when you hover the mouse over a point in the template, even if labels are not displayed.

Note: The template layer formatting that we have modified in the exercise is stored in **Screen Layouts**. This includes line style, hatch style, color and label formatting for subgrade and each layer above subgrade.

Reporting template point codes

Point codes can be displayed graphically in the *Plan*, *Profile* and *Section* windows. The *Data* and *Status* windows can display numeric information such as point code coordinates or centerline offsets.

The following steps will display the ditch lines in the Plan Window.

13. While still in the template editor, press the *Codes* button to open the dialog box shown below.

14. Select *Plan* in the *Window* list and notice that several point codes are already shown in the *Code* list. PT1 is the pavement edge, SB3 is the shoulder edge.

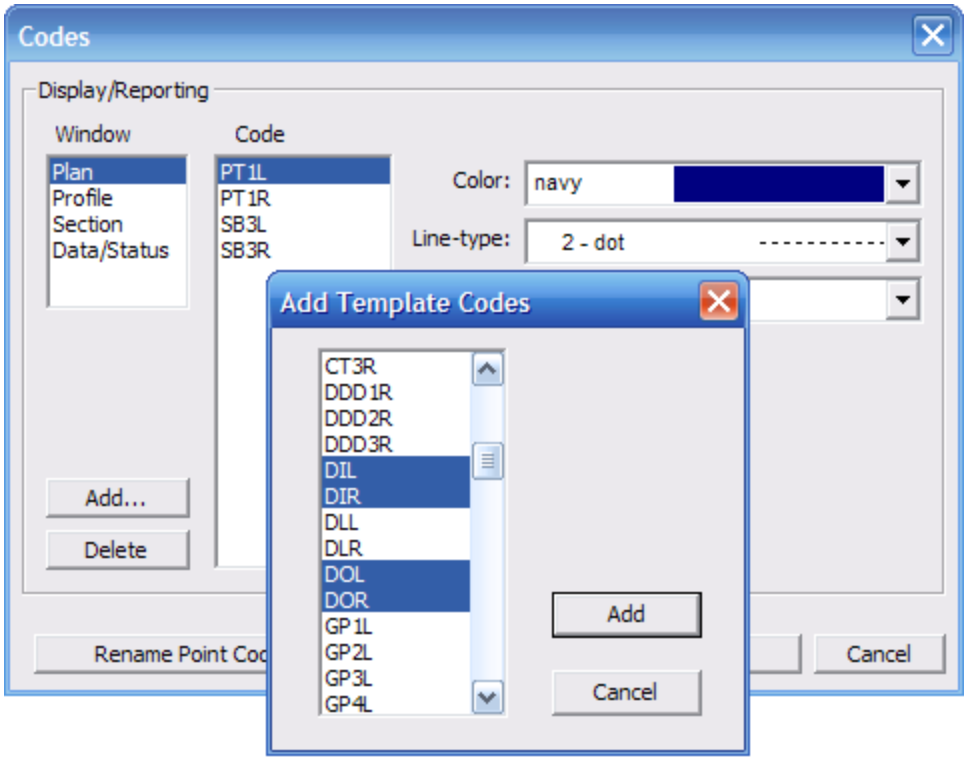


Figure 20.7: Adding template codes for display in the Plan window.

15. Click the *Add* button and select all the ditch point codes (figure above). Note that *L* and *R* have been added to the point codes to distinguish left from right.

16. Click *Add* to close the selection dialog box.

17. With the new codes still selected, choose a *blue dash* line as shown below.

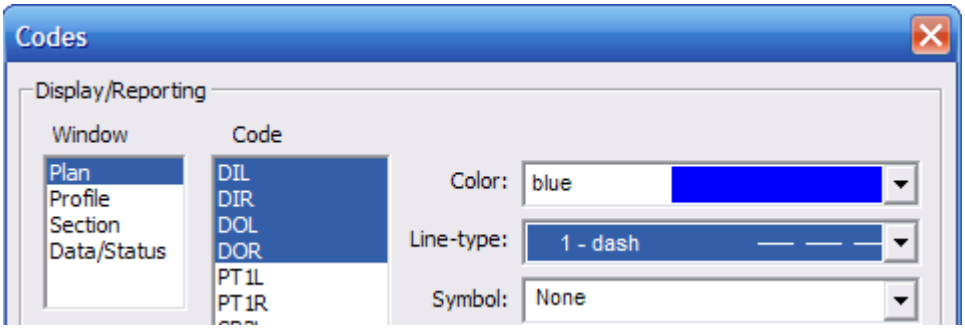


Figure 20.8: Plan Template Codes format control.

18. Press *OK* to close the *Codes* dialog box and *OK* again to close the template table editor. Respond *Yes* to “Recalculate Road Alignment”.

19. Adjust the Plan Window view so you can see the new ditch lines.

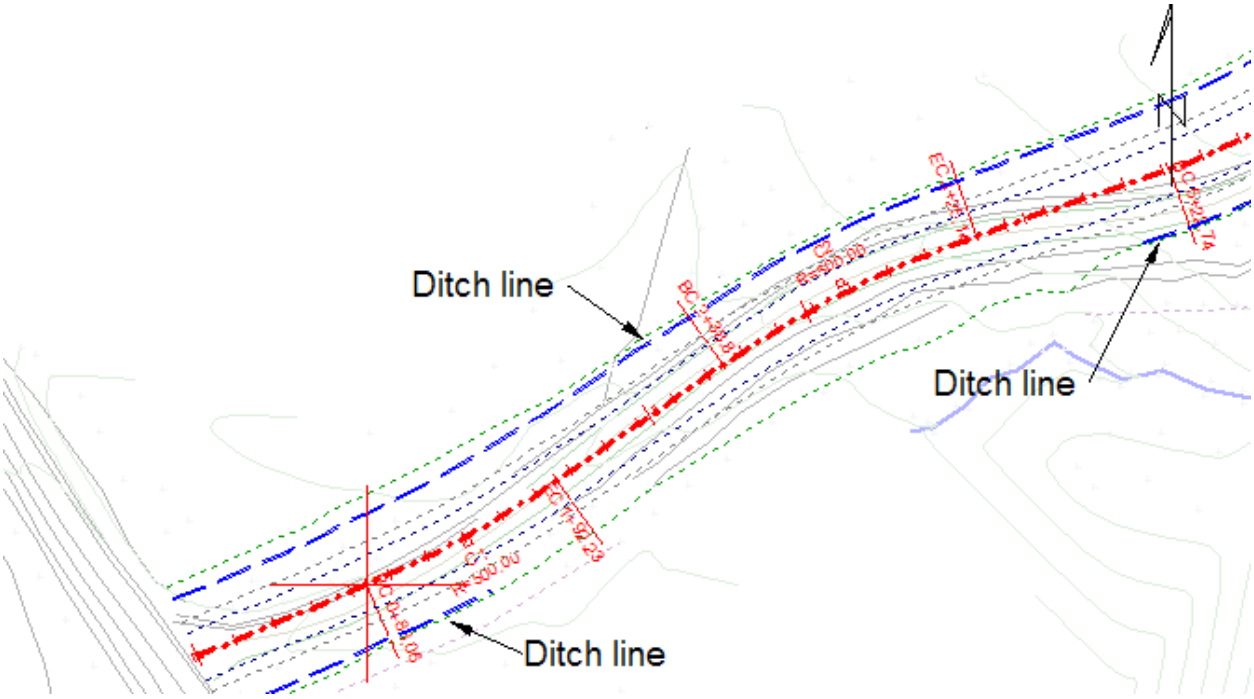


Figure 20.9: Ditch lines in Plan View

Note: Template point code display options we have changes in this exercise are saved in the **Template Table**.

The following steps will display the ditch line offsets in the Data Window.

20. Choose menu *Edit* | *Edit Templates* or press tool bar button , to re-open the template table editor.

21. Press the *Codes* button to open the dialog box (as above) and select *Data/Status* in the *Window* list.

22. Click the *Add* button and select all the ditch point codes (as in the previous exercise) and click *Add* to close the selection dialog box.

23. Press *OK* to close the *Codes* dialog box and *OK* again to close the template table editor. Respond *Yes* to “Recalculate Road Alignment”.

24. Right click in the Data window and choose *Data Options*. The dialog box below left will appear.

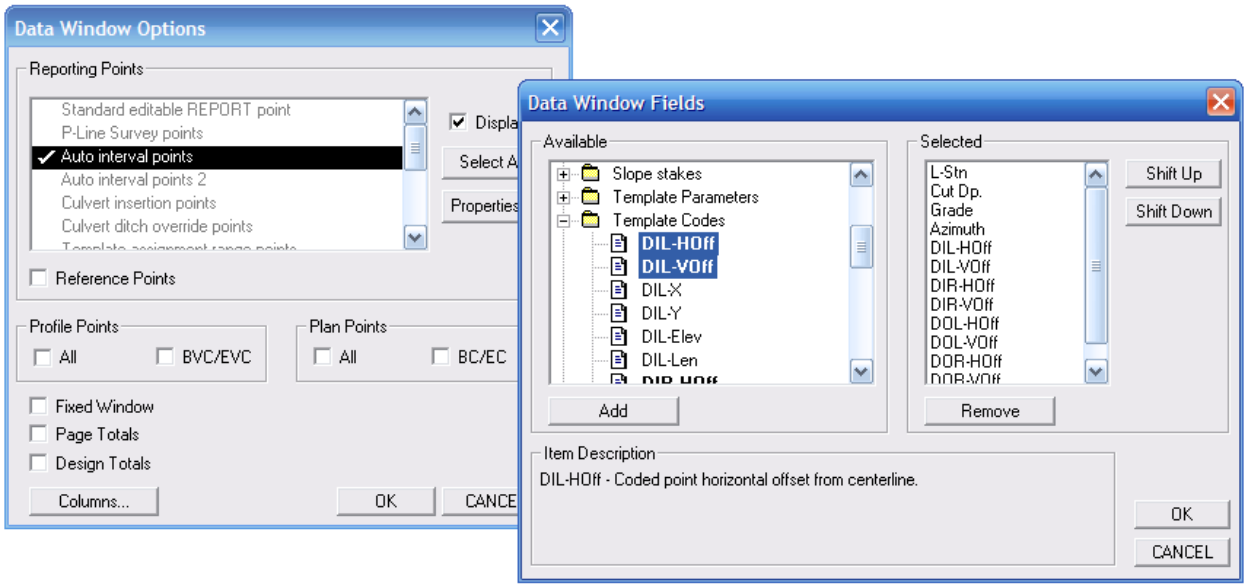


Figure 20.10: Selecting point code offsets for display in the Data window.

25. Turn off the *Profile Points* and *Plan Points* check boxes. Select *Auto interval points* in the list and turn on the *Display* check box (this should be the only item displayed).

26. Press the *Properties* button and select *Auto interval points*, note that the interval has been set to 100ft. *Cancel* to close the *Report Point Properties* dialog.

27. Press the *Columns* button and add the *Add* HOff and VOff for each of the available point codes (figure above right).

28. Press *OK* and *OK* again to update the Data display (figure below).

Data:1

L-Stn ft.	Cut Dp. ft.	Grade %	Azimuth D:M:S	DIL-HOff ft.	DIL-VOff ft.	DIR-HOff ft.	DIR-VOff ft.	DOL-HOff ft.	DOL-VOff ft.	DOR-HOff ft.	DOR-VOff ft.
0+00.00	-0.31			-28.90	-4.54	28.90	-4.54	-29.90	-4.54	29.90	-4.54
1+00.00	-0.43	3.8		-30.73	-5.33	27.34	-2.83	-31.73	-5.33	28.34	-2.83
2+00.00	-1.06	3.8		-29.76	-4.79			-30.76	-4.79		
3+00.00	-1.33			-27.34	-2.83			-28.34	-2.83		
4+00.00	-2.02		℄ 62:15:27	-27.34	-2.83			-28.34	-2.83		
5+00.00	-3.84	7.1		-28.90	-4.54	28.90	-4.54	-29.90	-4.54	29.90	-4.54
6+00.00	-2.99			-29.90	-4.56	28.90	-4.54	-30.90	-4.56	29.90	-4.54
7+00.00	-2.51		℄ 46:33:44	-29.90	-4.56	28.90	-4.54	-30.90	-4.56	29.90	-4.54
8+00.00	-2.01	4.9		-27.34	-2.83	30.73	-5.33	-28.34	-2.83	31.73	-5.33
9+00.00	-1.08		℄ 41:41:35	-27.34	-2.83			-28.34	-2.83		
10+00.00	-1.55		℄ 53:09:08	-27.34	-2.83	30.73	-5.33	-28.34	-2.83	31.73	-5.33
11+00.00	-1.33	8.0		-31.91	-6.31			-32.91	-6.31		
11+62.40	-3.17	8.0		-28.90	-4.54			-29.90	-4.54		

Figure 20.11: Data window showing point code offsets.

Note: The data window can be exported to a file (menu *File* | *Export Data to ASCII*) or the clipboard (*Edit* | *Copy to Clipboard* | *Data Window Ctrl+C*). This tabular data can be read by a spread sheet application.

You can also add the point code offsets to your Section window *Status* area (below the graphic). Right click in the Section window and choose *Section Options*; press the *Fields* button.

29. Choose menu *File* | *Close*. Do not save changes.

21. Culverts

In this exercise you will assign a culvert to the road realignment design.

1. Open **Examples\Location\ Align stage 7.dsn**.
2. In the plan window, zoom in to the creek crossing (figure below)

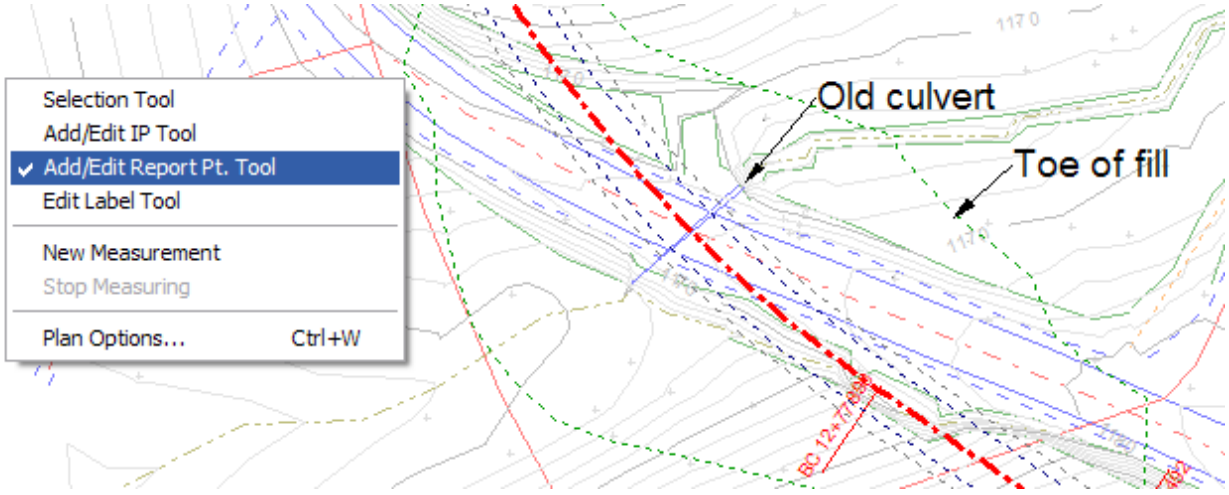


Figure 21.1: Creek crossing location; you can see the old culvert in the background

3. Right click in the plan window and choose *Add/Edit Report Pt. Tool* (figure above). Your mouse cursor will change to the pencil with question mark .
4. Click near the culvert to create a new *report point*. Move the red cross until it is over the culvert and click a second time to anchor the new point. You cross section window will update to show the new location (figure below).

Note: Report points can only follow the existing alignment.

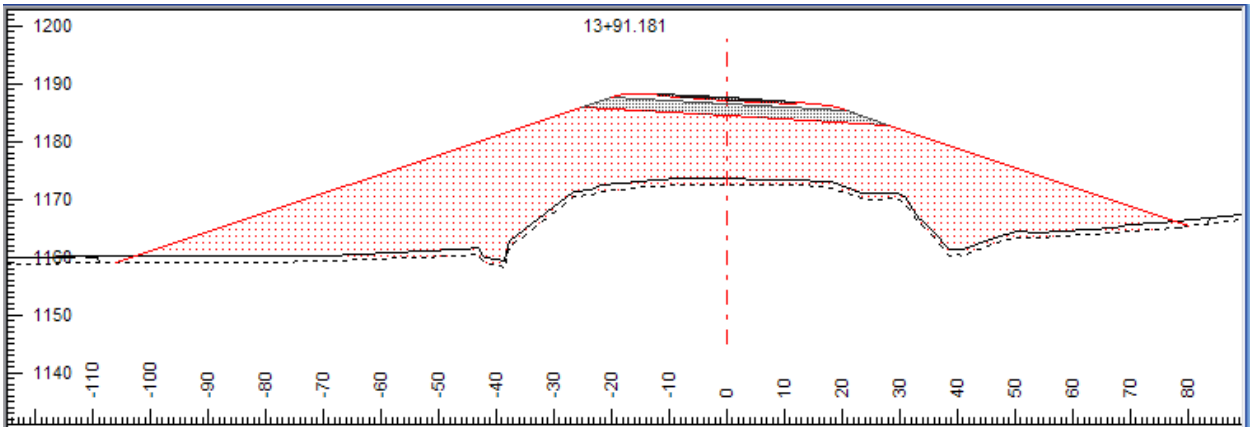


Figure 21.2: Desired culvert location.

5. Select screen layout **training Culvert.dlt**. Your screen should look similar to the figure below.

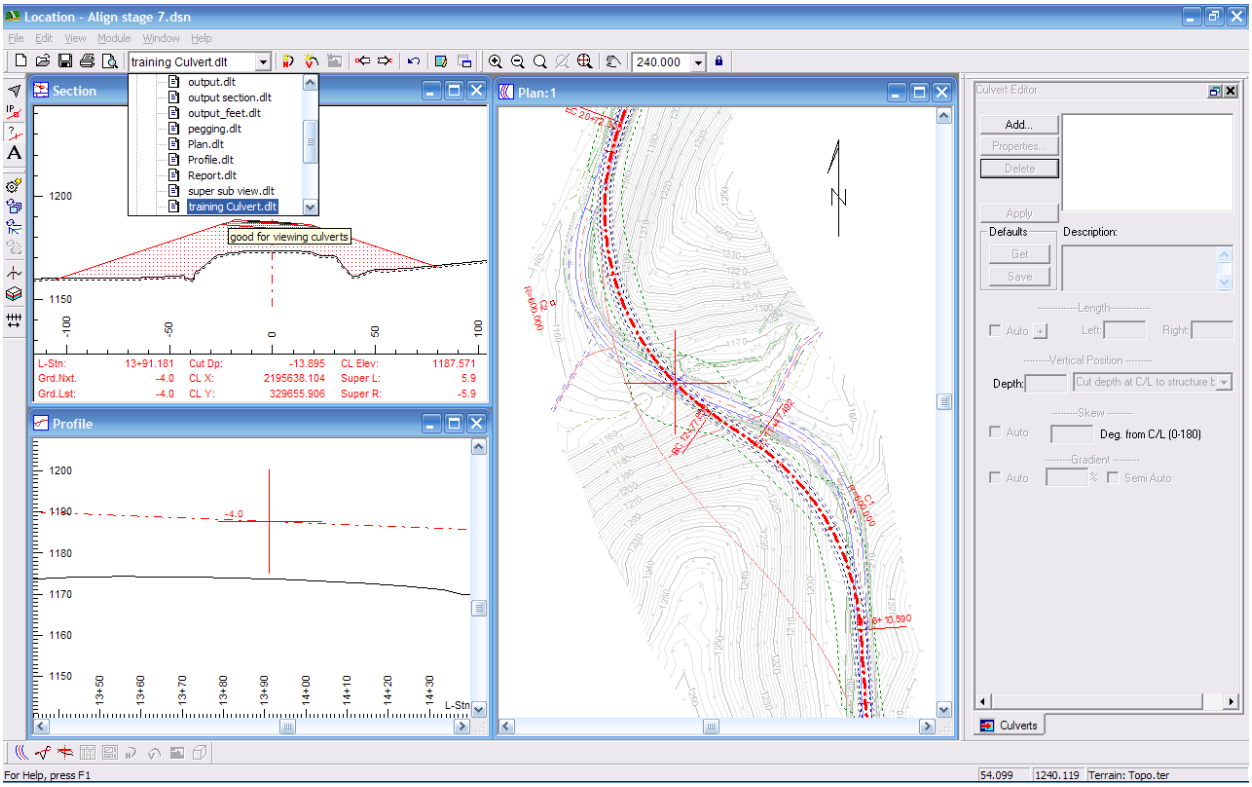


*Figure 21.3: Screen layout **training Culvert.dlt***

6. Press the *Add* button in the Culverts Panel (right side of screen) to open the dialog box shown below. Note that the *Station* defaults to the current cross section.

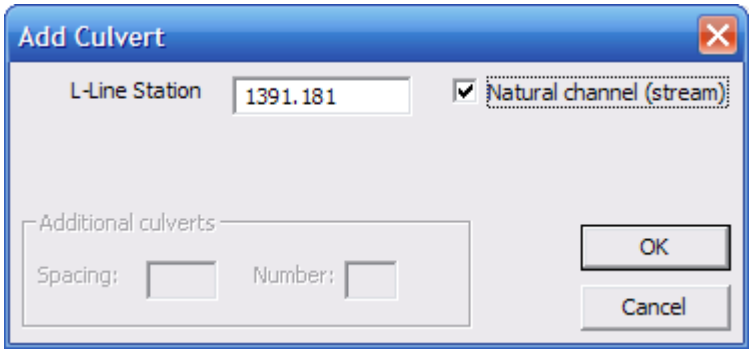


Figure 21.4: Add Culvert dialog box.

7. Set the *Natural channel (stream)* check box (the alternative is a cross drain). Press *OK* to create the culvert.

8. If the profile window is not showing the correct station, press *next* and then *back* buttons in the tool bar. Whenever you change the current section this way all window scroll to show the new current point.

Your culvert should be visible in all three windows.

9. Press the *Properties* button (top left of Culverts panel) to open the dialog box shown in the figure below.

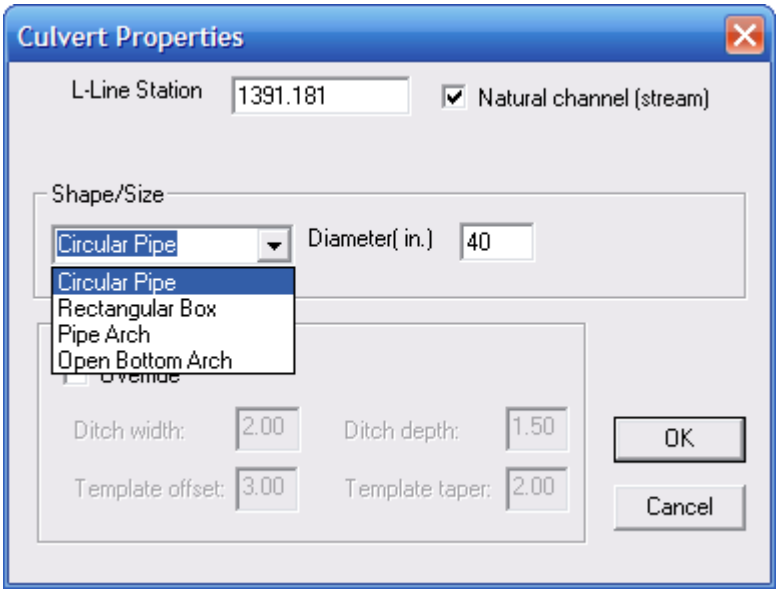


Figure 21.5: The culvert properties dialog box allows you allows you to change the location, size and shape of a culvert.

10. Change the *Diameter* to 60 inches and press *OK* to close the *Culvert Properties* dialog box.

11. Change the *Depth* to 8 feet, figure below left. And Press *Apply* to see the changes.

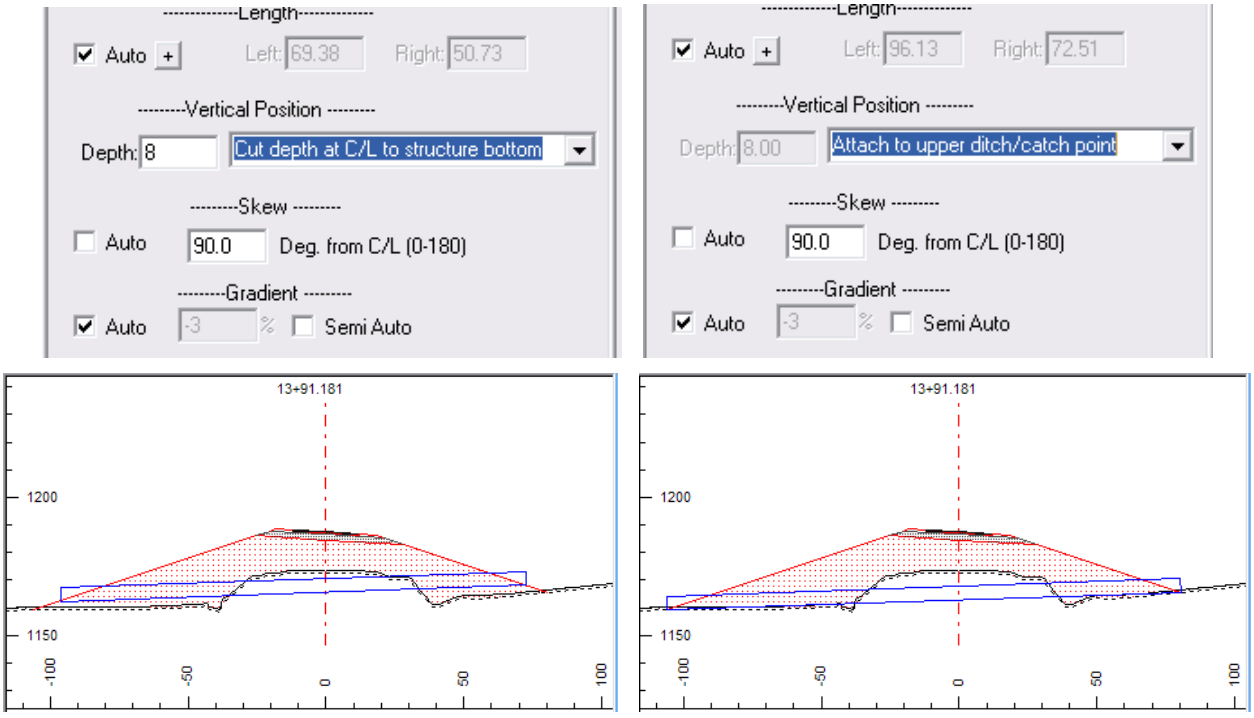


Figure 21.6: culvert elevation controlled by cut depth at center line (left) and by catch points (right).

12. Change the *Vertical Position* type to *Attach to upper ditch/catch point*, figure above right. Press *Apply* to see the changes.

The latter method (*Attach to upper ditch/catch point*) is more reliable for creating a stream culvert that sits on the bottom of a stream bed.

13. Press the *Save* button and respond *OK* to the to save stream default prompt. The next time you create a *Natural Channel* culvert, this will be the initial configuration.

Changing fill material

Now we will change the road fill material to rip rap so that we can

- Steepen the fill and shorten the culvert
- Reduce the amount of fill required
- Reduce the footprint in the wetland (and the right of way)
- Prevent scouring.

14. Choose menu *Edit* | *Assign Parameters by range* or press the tool bar button to open the *Assign Attributes by Station* dialog box (figure below). Select the *Fill Types* tab.

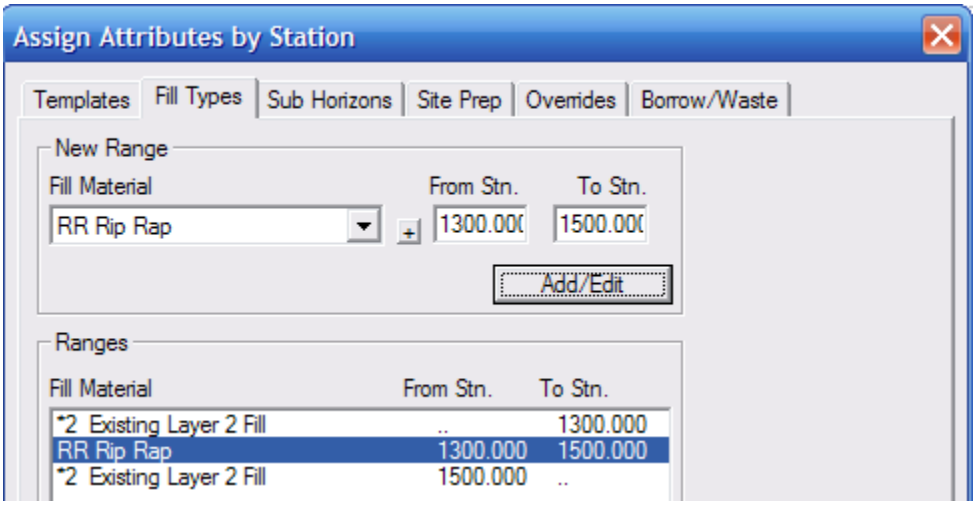


Figure 21.7: Applying special fill (rip rap) near the culvert.

15. Choose fill material *RR Rip Rap* (this comes from you *ground types* table) and set the station range from 1300-1500. Press the *Add/Edit* button.

16. Press the button to open the *Ground Types* editor. You can add materials here if you wish. Press *Cancel* to close the *Ground Types* editor again.

17. Press *OK* to close the dialog box. Respond *Yes* to the recalculate prompt.

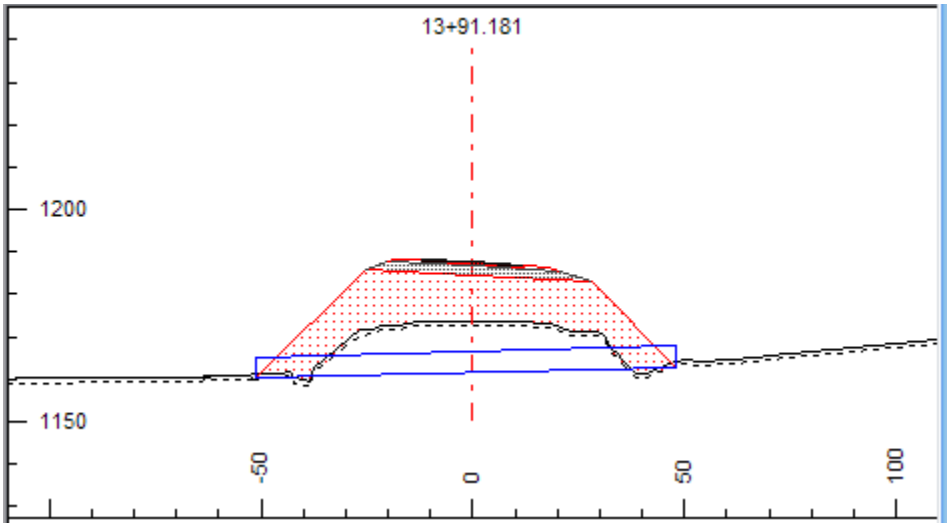


Figure 21.8: The creek crossing with rip rap fill and a much shorter culvert.

You can also see the change in the road footprint in the plan window.

Note that the template applied here has a fill slope defined as *Automatic*; if the template was set up with a fixed slope, you would have to change the fill slope for the desired station range using the Overrides tab in the *Edit | Assign Parameters by range* dialog box (see Template Parameter Overrides).

18. Choose menu *File | Close*. Do not save changes.

22. Multi-plot Output

Multi-plot is a page layout tool for creating output. Any of the main windows (Plan, Profile, Data, and Section) can be placed on a Multi-Plot sheet with other items such as a legend, a scale bar, a bitmap graphic, a Terrain file, Curve Tables, Template assignments, or a title block. This feature can be incorporated with screen layouts to retrieve most commonly used options and formats. In this section, you will learn how to create a Multi-Plot of a standard Plan over Profile Window.

To do the examples in this section the *P-Line Design, Multi-Plot Output Sheet Generation* and *Sub-surfaces* function groups should be enabled. See *Function Groups* in the introduction above and in the On-line help for more information.

Multi-Plot Introduction

In this example you will create a Multi-Plot output sheet containing Profile and Plan sub-views. We will discuss automatic pagination.

Creating and Positioning Sub-Views

1. Choose menu File | Open, Select **Examples\Location\Align stage 8.DSN.** Press *Open*.

Note**:** If you are using RoadEng® Lite or RoadEng® Civil Assistant and the “Non Permitted Functions Found in File” dialog appears, choose “Keep all functions and revert to DEMO Mode”.

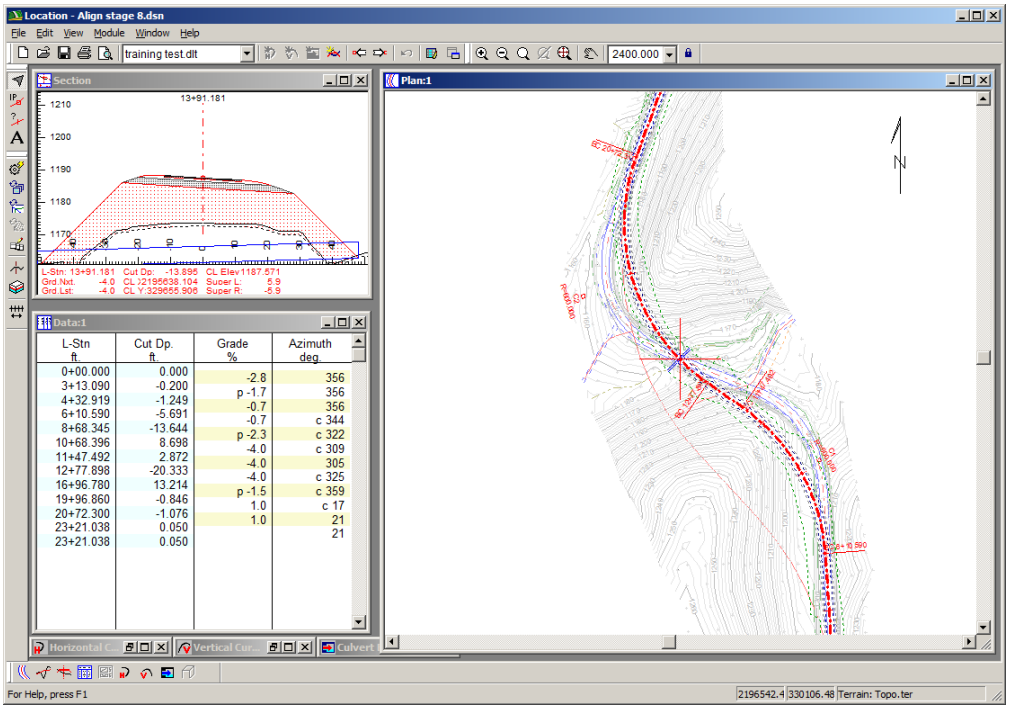


Figure 22.1: ***Align stage 8.DSN***

For our purposes, this design is considered complete from an engineering point of view. Now we want to produce output that a contractor can use to bid on and/or build the road.

2. From menu Window|New Window, select Multi-Plot. This will display the Multi-Plot Window.

3. Maximize the Multi-Plot window.

Printer setup

The orientation and size of the blank sheet, within the Multi-Plot Window, is determined by the selected printer set-up.

NOTE: Multi-Plot setups apply to a specific paper size and orientation.

4. Use menu File|Print Setup to open the setup dialog box, change the orientation to Landscape, and the Paper size to 8.5” by 14” (legal). Press *OK*.

Adding graphic sub-views

Now let’s add some content to our page.

5. From the menu Edit|New Sub-view, select *Plan:1*. A Plan *Sub-view* should appear in the center of your Multi-Plot Window.

Note: The Plan sub-view is an image of the main Plan window. If you don’t have a Plan *window* displayed (see the Window menu) then you can’t create a Plan *Sub-view*.

Notice that there are 8 handles that you can click and drag to change the size of the sub-view. Click and drag anywhere else on the sub-view ✣ to move it. The <delete> key will remove the selected sub-view(s).

Notice, also, that the Plan is rotated automatically to best fit the rectangle with increasing stations running from left to right. In this case the Plan has been rotated automatically by approximately 90 degrees. See section: *Plan Rotation.*

6. Resize and reposition the Plan sub-view as shown below.

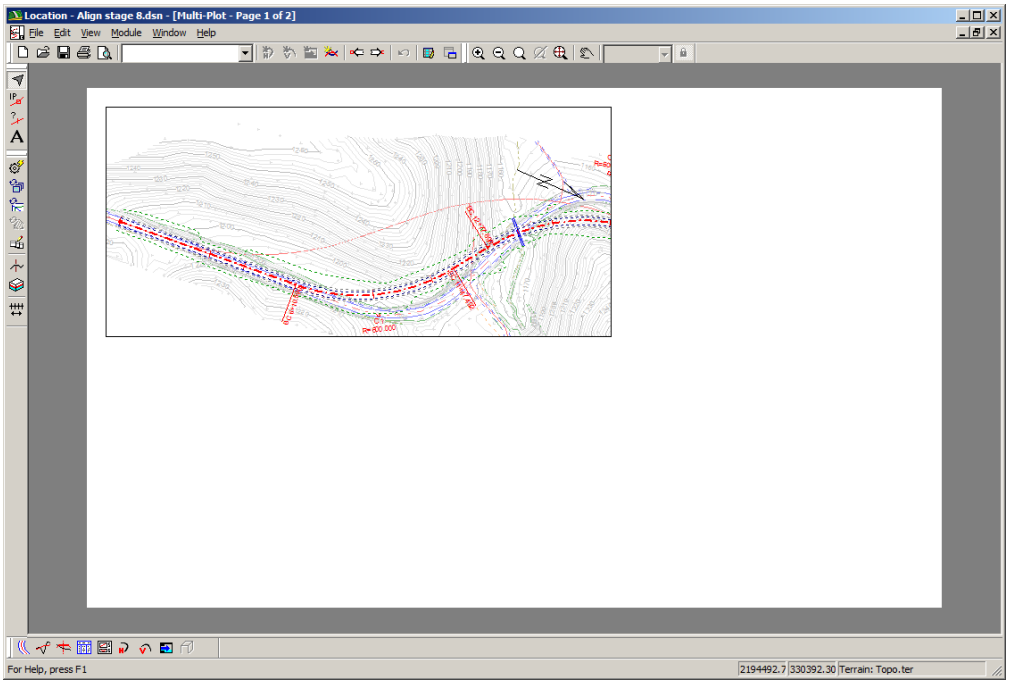


Figure 22.2: Plan Sub-view After Sizing and Positioning

7. From menu Edit | New Sub-view, select *Profile:1*. A Profile sub-view should appear in the center of the Multi-Plot. Adjust it to fit under the Plan sub-view (don't worry about misalignment at this point).

A click on a sub-view will select it and deselect the previous sub-view. See also the note below.

NOTE: When you click outside all sub-views and drag the mouse you will create a selection rectangle. All sub-views inside or crossing the rectangle will be selected when you release the mouse. Also, <Ctrl> click allows you to select/deselect sub-views without affecting the selection state of other sub-views. Group selected sub-views can be deleted or moved together.

Grid options

Here we turn on a grid to make it easier to align the Plan and Profile sub-views.

8. Select View | Multi-Plot Options to open the dialog box below. You can also access this dialog box by right clicking in the Multi-Plot Window.

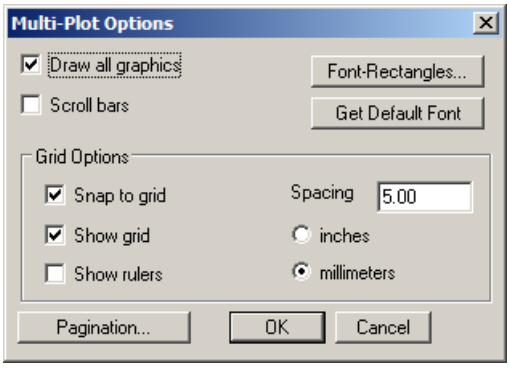


Figure 22.3: Multi-Plot Options Dialog Box.

9. Select *Show grid* and *Snap to Grid* and set the *Spacing* to **5**mm as shown above. Press *OK*. A dot grid will cover the entire Multi-Plot sheet

10. Now adjust the size and position of both the Plan and Profile sub-views so they are aligned as in the figure below

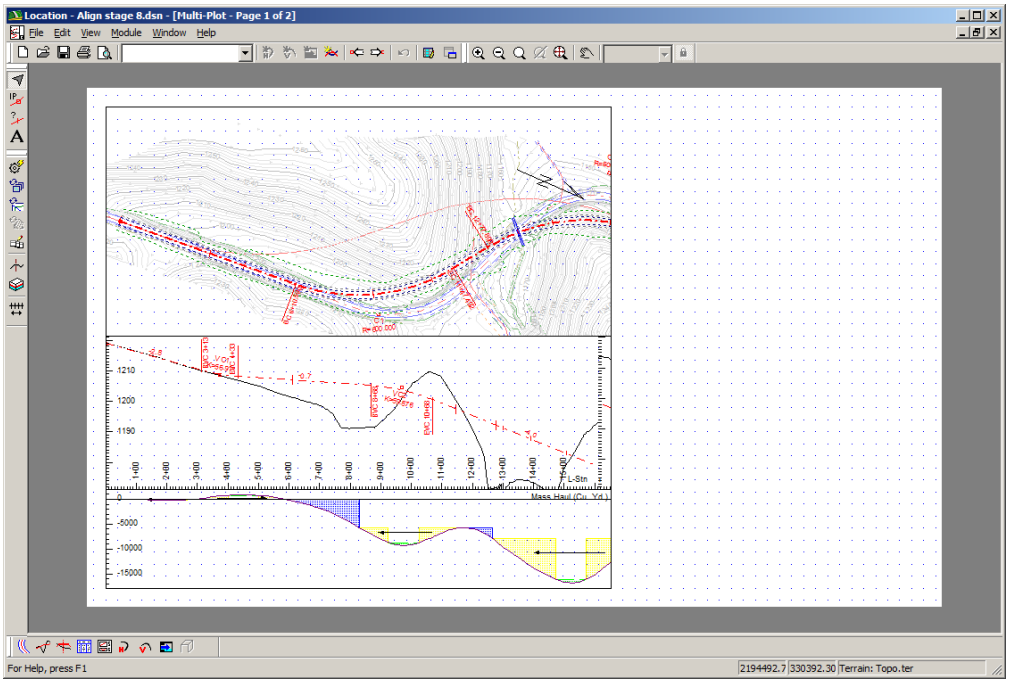


Figure 22.4: Multi-Plot after grid enabled.

Sub-View options

Every sub-view has options accessible from the right click menu.

11. Right Click on the Plan sub-view and select menu Plan:1 Sub-view Options to open the dialog box below left.

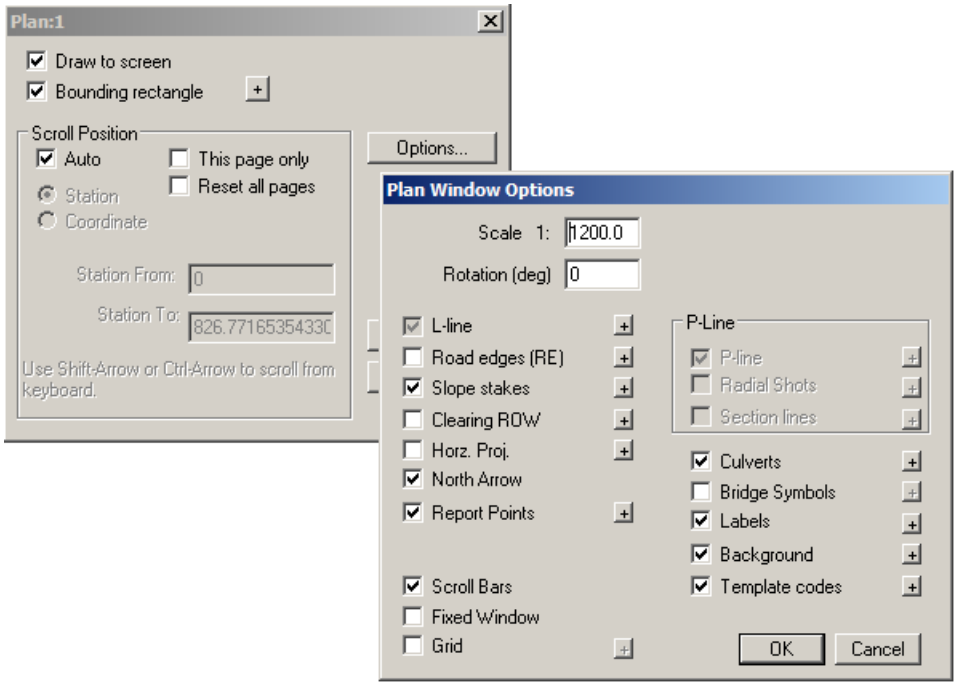


Figure 22.5: Plan sub-view options and Plan Window Options – shared by the main Plan window and the Multi-Plot Plan sub-view.

12. Click on the *Options* button to open the *Plan Window Options* dialog box (above right).

NOTE: The same dialog box you see now is available when the *main* Plan Window is active. All the viewing options defined for the main Plan Window are displayed in the Multi-Plot Plan sub-view (with the exception of the rotation angle).

13. Change the *Scale* to *1200* and then Press *OK* and *OK* again to close the options dialog boxes.

14. Similarly, access the Profile options and set the scales to 120 and 1200. Remove the Mass haul display (optional) using the *Select* button in the *Sub-Windows* area.

15. Press *OK* to close all *Options* dialog boxes.

Adding a Scale Bar

16. Select menu Edit | New sub-view; choose *Scale Bar*. A Scale bar will appear in the middle of your sheet.

17. Right click on the new scale bar and select *Scale Bar Sub-view Options* menu.

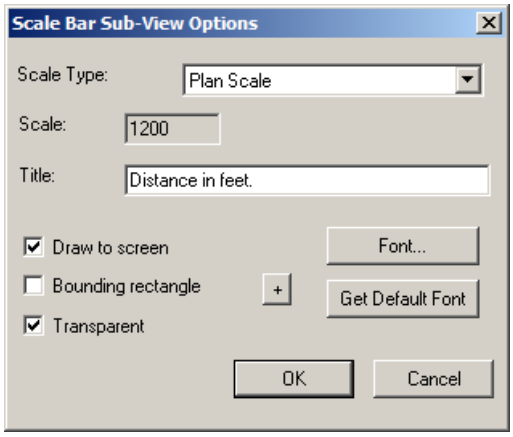


Figure 22.6: Scale Bar Sub-view Options Dialog Box

18. Keep the default *Plan Scale* from the *Scale Type:* pull-down box. Add in the optional *Title* "Distance in feet". Press *OK*.

19. Resize and reposition the Scale Bar sub-view, until it appears as in the figure below (also see notes below).

NOTE: If you accidentally click on the Plan sub-view the scale bar will be inaccessible underneath. Use the menu Edit | Shuffle Front to Back or <Control + K> to allow access to the scale bar.

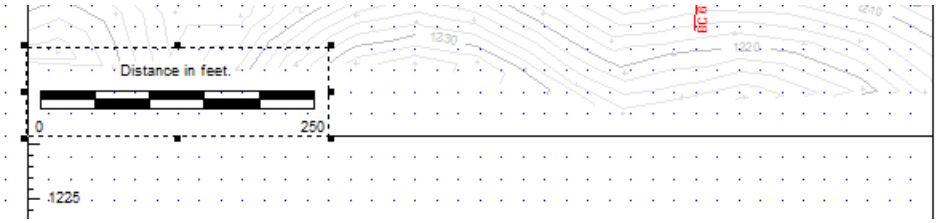


Figure 22.7: Scale bar in lower left corner of Plan sub-view.

Adding Rectangle items

Rectangles can hold typed text or of many pre-defined text items.

20. Select menu Edit | New sub-view; choose *Rectangle*. A Rectangle will appear in the middle of your sheet and the options dialog box shown below will appear.

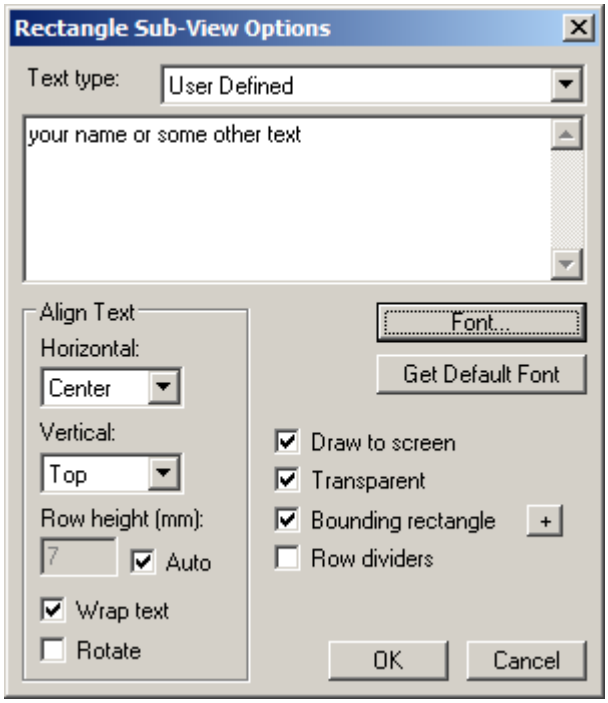


Figure 22.8: The Rectangle Sub-view Options Dialog *opens automatically when you create a new rectangle but you can also access it from a right click on any rectangle sub-view.*

Note: Sometimes it is useful to use an *empty* rectangle just for its border graphic (*User Defined*, no text).

21. Type your name or some other text in the text box. Multiple lines are allowed. Change horizontal alignment to *Center*. Check the *Wrap text* option. Click on the *Font* button and change the size to 20. Press *OK* and *OK* again.

22. Arrange the new rectangle to the lower right as in the figure below.

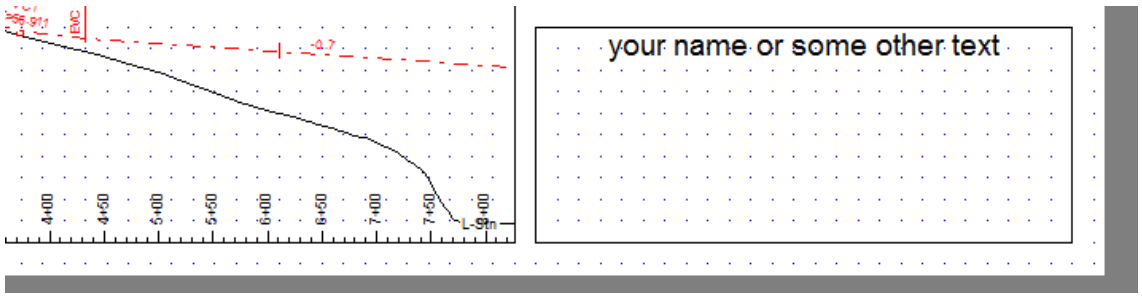


*Figure 22.9: A New Rectangle Sub-view with C*entered, Wrapped, User Defined *text in a Large* Font.

23. Create two more rectangles. In the first select *Print Date* from the *Text type* combo box. In the second select *Page X of N*.

24. Arrange the two new rectangles to fit in the first rectangle. Notice how the snap to grid feature helps line up edges. Don't forget about Edit | Shuffle Front to Back, <Control + K>, if you get one rectangle stuck behind the other.

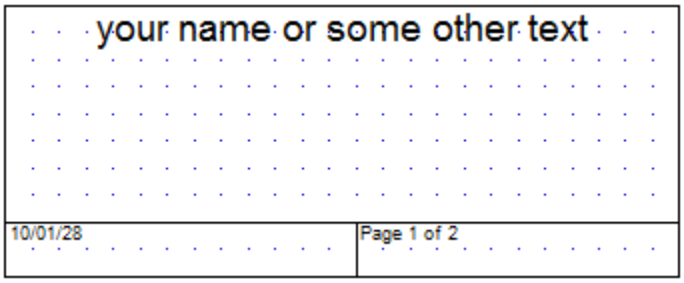


Figure 22.10: The Beginning of a Title Block

25. Continue to the next section or File | Close. Do not save changes.

Pagination

The Location Multi-Plot Window can automatically produce as many pages as required to show the entire design. In this section we will explore some of the pagination options.

If you are continuing from the exercise above you may omit steps 1 and 2.

1. Choose menu File | Open, select **Examples\Location\Align stage 8.DSN**. Press *Open*.
2. Retrieve screen layout **training multi 2.DLT**.

Notice that the Multi-Plot Window title bar says that this is Page 1 of 3.

3. Use <Ctrl + N> to see what all the pages look like (menu View | Jump to Page also works).
4. Choose menu View | Multi-Plot Options to open the dialog box below. You can also access this menu by right clicking in the Multi-Plot Window.
5. Click on the *Pagination* button.

Notice that the current page size is set to *Use Plan or Profile width* – reducing the width of the Profile sub-view, for example, would reduce the page size from the current value of 905.5 feet (in the disabled edit box above). You will now change the setting to a *Fixed* width (figure below) - this is the recommended paging method when page station ranges are constant.

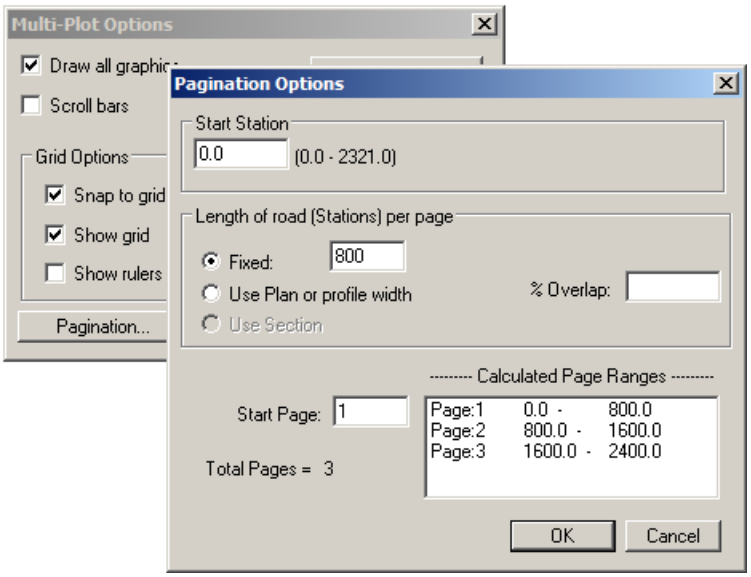


Figure 22.11: Multi-Plot Paging Options

6. Change the Length to *Fixed*, **800** and the overlap to **0** as shown above. Press *OK* and *OK*, again.

Again page through the design, <Ctrl + N>; notice the overlap between pages and the gap at the start of page 1.

The Multi-Plot window can also display section graphics; in this case the station range per page is not always a constant. The optional steps below give you a chance to look at a typical section multi-plot showing curve transition points.

7. Retrieve screen layout **training multi section.DLT**.

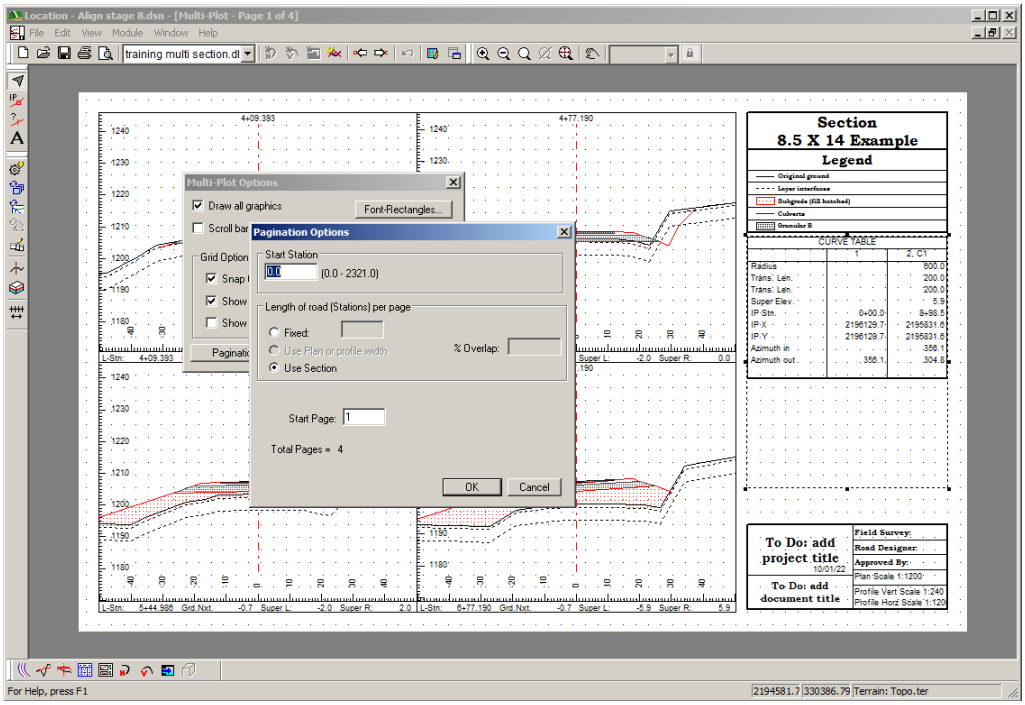


Figure 22.12: Section multi-plot example.

8. Page through the document <Ctrl + N> then check out the pagination options (as in the steps above).

9. Retrieve screen layout **training multi 3.DLT** to restore the Plan over Profile layout.

Multi-Plot Plan Rotation

In this example the Plan sub-view is acceptable on most pages. The automatic pagination puts the page start station on the left side of the Plan sub-view and the end station on the right. This approach does not always work so it is possible to set the Plan sub-view scroll position and rotation angle manually.

If you are continuing from the exercise above you may omit steps 1 and 2.

1. Choose menu File | Open, select **Examples\Location\Align stage 8.DSN**. Press *Open*.

2. Retrieve screen layout **training multi 3.DLT**.

3. Use <Ctrl + N> or <Ctrl + B> to get to page **3**.

4. Right Click on the Plan sub-view and select menu Plan:1 Sub-view Options to open the dialog box below left.

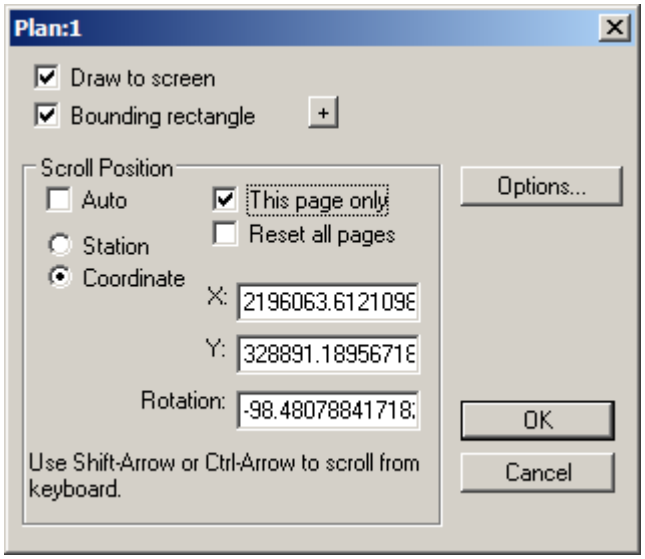


Figure 22.13: Plan sub-view options set to scroll manually.

5. Clear the *Auto* check box and click on the *Coordinate* button and set *This page only* (as above). Press *OK*.

Note the Plan position has not yet changed; we didn't change coordinates or rotation angle yet.

6. Type <Shift + arrow> to scroll or <Ctrl + arrow> to rotate (See note below). Try to get the Plan sub-view to look like the one in the figure below.

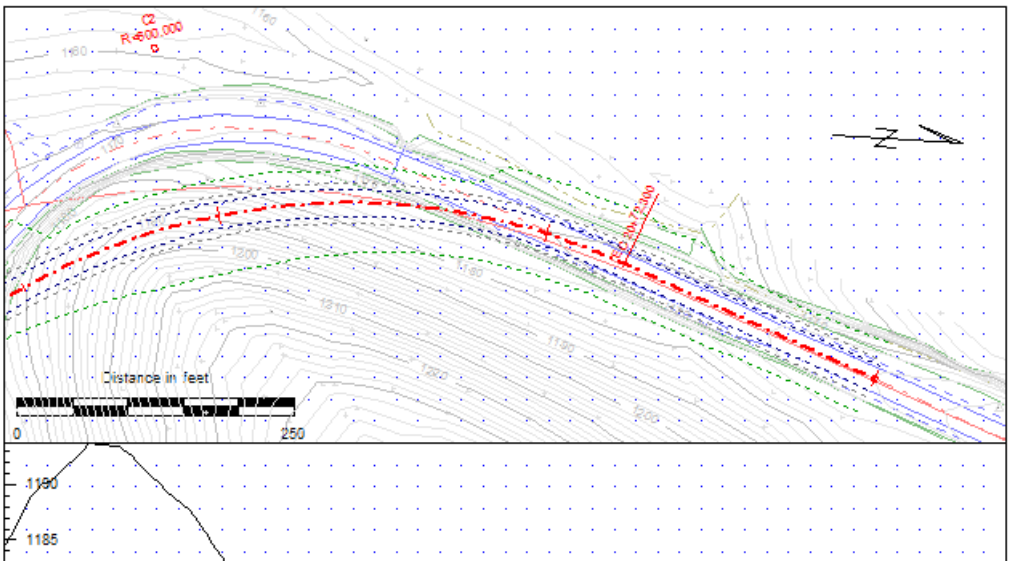


Figure 22.14: Plan Sub-view after manually scrolling and rotating

7. Use <Ctrl + N> or <Ctrl + B> to view all the pages.

NOTE: Manual alterations to the Plan or Profile position and orientation can also be done by selecting the window and then pressing <shift + arrows>. A prompt may remind you that *Your Plan/Profile sub-view is set to scroll with the current page station range. Do you wish to scroll manually instead?* This operation will disable the *Auto* check box as in step 5 above. <Shift + arrows > will scroll the plan or profile in the direction of the arrow. <Ctrl + arrows> will rotate the Plan sub-view around its center.

8. Continue to the next section or File | Close. Do not save changes.

Multi-Plot Layouts

In the previous Multi-Plot exercises you have used *screen layouts* (DLT files) several times. Screen layouts allow you to make your output standardized and save you the considerable trouble of setting up Multi-Plot page layouts for every design.

In these exercises we will copy and paste multi-plot items, explore a couple of new sub-views and save the result for future use.

To do the examples in this section the *P-Line Design, Multi-Plot Output Sheet Generation* and *Sub-surfaces* function groups should be enabled (see *Function Groups* in the introduction above and in the On-line help for more information)

Copy and Paste of Multi-Plot items

This exercise will add a title block to a Multi-Plot sheet. We will do this by merging the current Multi-Plot with a commonly used title block screen layout.

If you are continuing from the exercise above you may omit steps 1 and 2.

1. Choose menu File | Open, select **Examples\Location\Align stage 8.DSN**. Press *Open*.

2. Retrieve screen layout **training multi 3.DLT**.

3. Select and <delete> the existing title block items so that only the Pan and Profile remain (figure below).

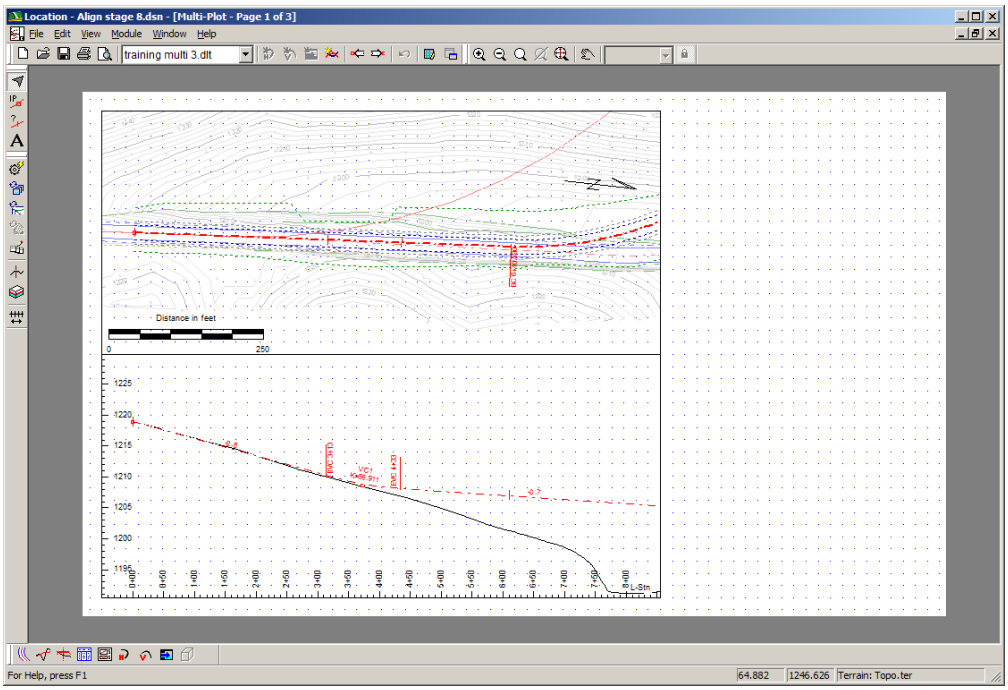


Figure 22.15: Multi-Plot after loading Screen Layout and removing title block rectangles.

When we are finished, we want to have a screen layout we can use for output from any road design. Start by creating a new screen layout to hold the current setup:

4. Create a new screen layout:

    a. Drop down the screen layouts from the main toolbar (figure below left)

    b. Scroll to the top and right click on the *Softree* folder.

    c. Choose menu *Save New Layout* to open the dialog box below right.

    d. Fill out the *File Name* (**training Output**) and the optional *Description*.

    e. Press *OK* to save the file.

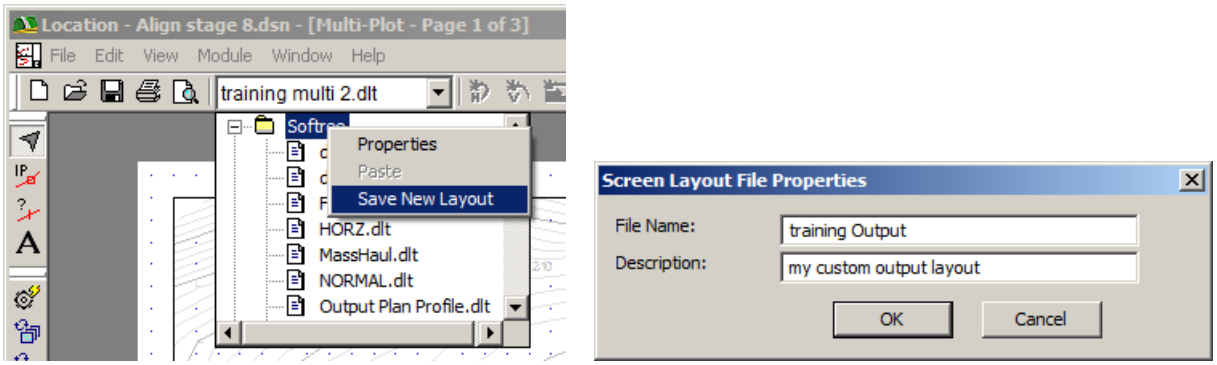


Figure 22.16: Creating a new layout file. This file will represent the screen and options as you currently see them.

Note that you could also have used the menu File | Save Screen Layout.

5. Retrieve screen layout **training multi section.DLT**.
6. Click and drag from the lower right corner to select only the title block as shown below.
7. Type <Ctrl + C> to copy the selection to the clipboard (or use menu Edit | Copy).

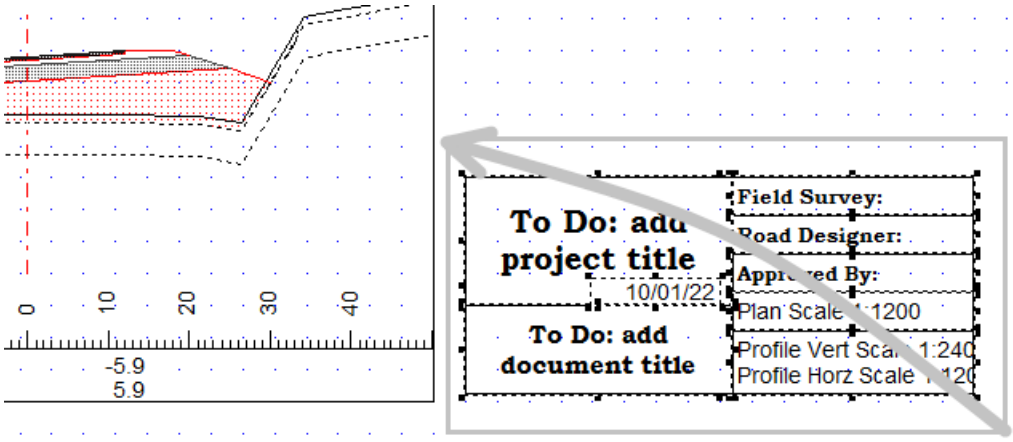


*Figure 22.17: Selecting multiple sub-views (*rectangles *in this case) with a mouse click and drag.*

8. Retrieve the custom screen layout created earlier: **training Output.DLT**.
9. Type <Ctrl + V> to Paste the title block on your page (or use menu Edit | Paste).
10. While the multiple items are still selected, click and drag ✣ to move them to the top right corner.

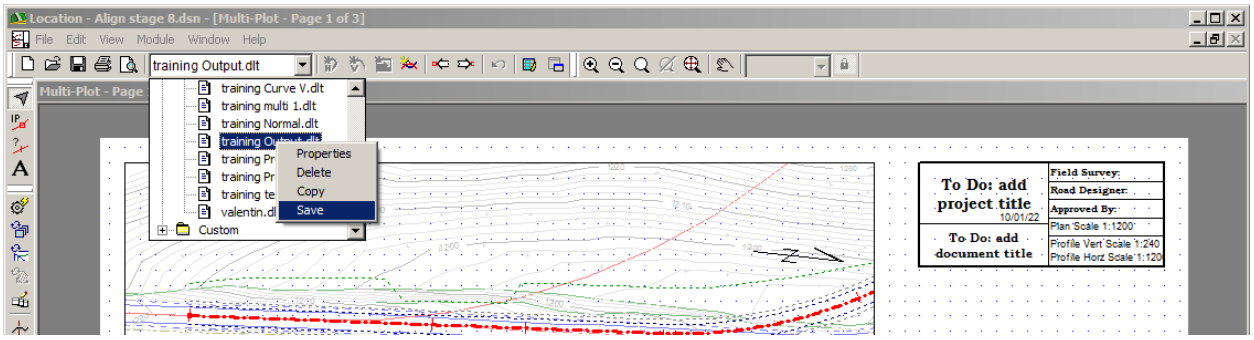


Figure 22.18: Saving over an existing Screen Layout.

11. Save your modified screen layout:

    a. Drop down the screen layouts from the main toolbar (figure above)

    b. Right click on **training Output.DLT**.

    c. Choose menu *Save* to save the file.

Note that you could also have used the menu File | Save Screen Layout.

Add a Legend

In this section we will create legend sub-view item and examine some of its options.

If you are continuing from the exercise above you should omit steps 1 and 2.

1. Choose menu File | Open, select **Examples\Location\Align stage 8.DSN**. Press *Open*.

2. Retrieve screen layout **training multi 4.DLT**.

3. Choose menu Edit | New Sub-View | Legend to create a legend item.

Most of the legend items created automatically need to be removed; some of those remaining will need to be renamed.

4. Right click on the legend and choose menu Legend Sub-View Options.

5. Click on the Items button to display the dialog box below.

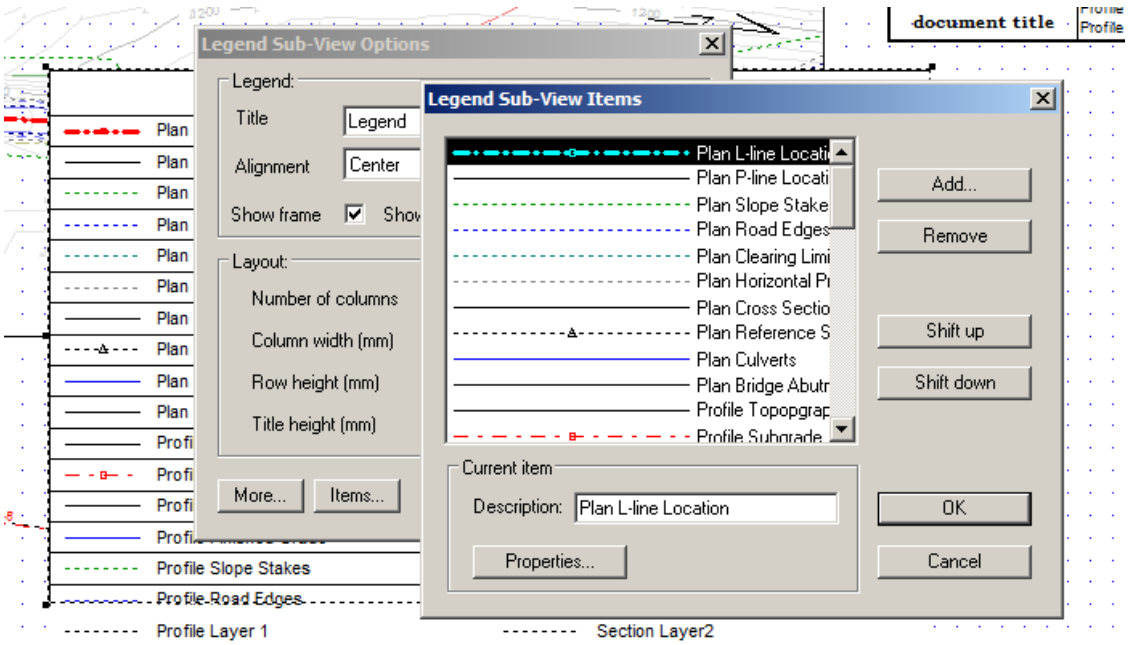


Figure 22.19: Dialog box for modifying legend items

6. Select and *Remove* all but the items shown below left (multiple select is allowed: <Ctrl + Click> or <Shift + Click>).

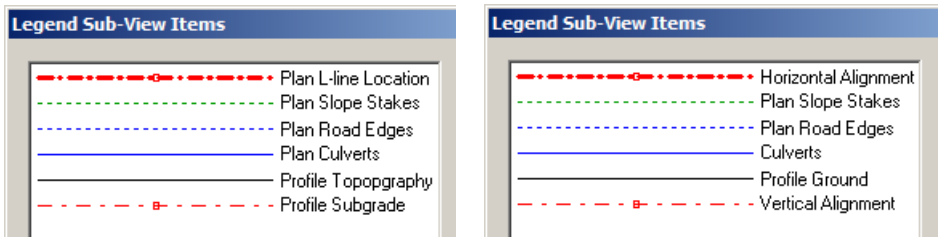


Figure 22.20: Legend with fewer items (left) and new Descriptions *(right).*

7. Select items one at a time and change the *Description* as above right.

At this point you may wish to experiment with the other buttons. The *Properties* button allows you to change the line, symbol and hatching for any item.

8. Press *OK* once to return to the main legend options dialog.

9. Change the *Number of columns* to 1 and press *OK*.

10. Finally move and size your legend so it fits nicely on the right of the Plan and Profile graphics; see figure below.

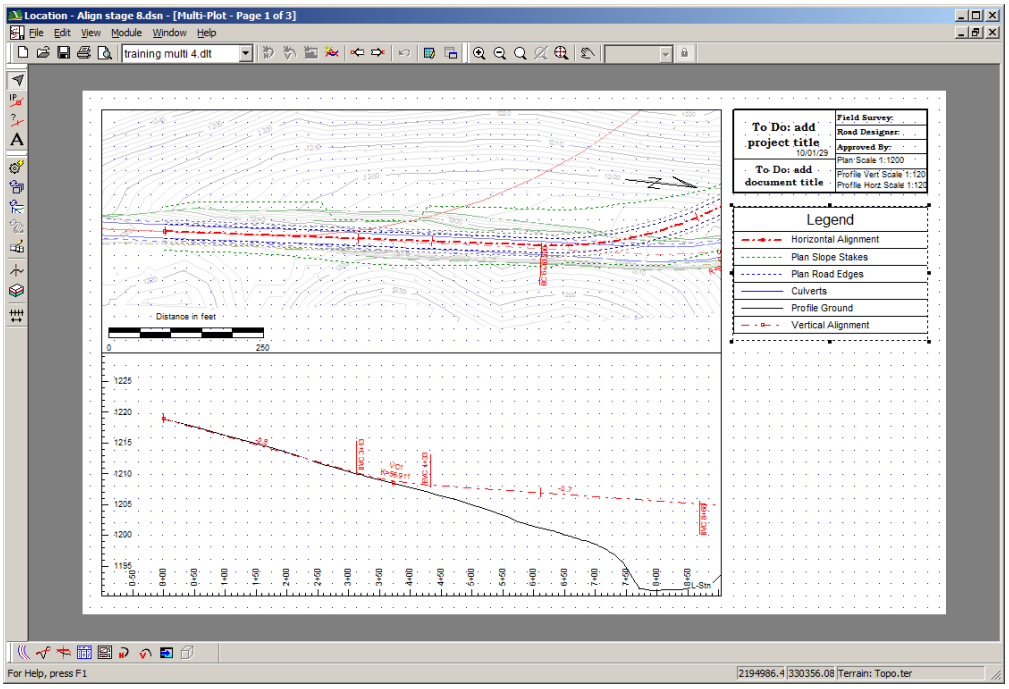


Figure 22.21: Legend added to layout.

11. Save your modified screen layout: **training Output.DLT.** See instructions in previous exercises.

Add a Curve Table

In this section we will create a horizontal Curve Table sub-view and examine some of its options.

If you are continuing from the exercise above you should omit steps 1 and 2.

1. Choose menu File | Open, select **Examples\Location\Align stage 8.DSN**. Press *Open*.
2. Retrieve screen layout **training multi 5.DLT**.
3. Choose menu Edit | New Sub-View | Tables | Horizontal Curves to create the table.
4. Move and size the table until it fits on the right of the Plan and Profile graphics.
5. Right click on the table and select menu Horizontal Curve Table 1: Sub-View Options to open the *Horizontal Curve Table* dialog box (figure below, top).

Scroll Position Auto here means that the table items shown occur in the station range for the current page (see pagination exercise on page 128).

6. Press the *Options* button to open the *Curve Table Options* dialog box (figure below, right).
7. Change the Width to 20mm.
8. Select *Design Points All* to include points of intersection (IPs) with no curve attached (new in version 5.1).
9. Press the *Add/Remove* button to open the *Curve Table Fields* dialog box (figure below, left).

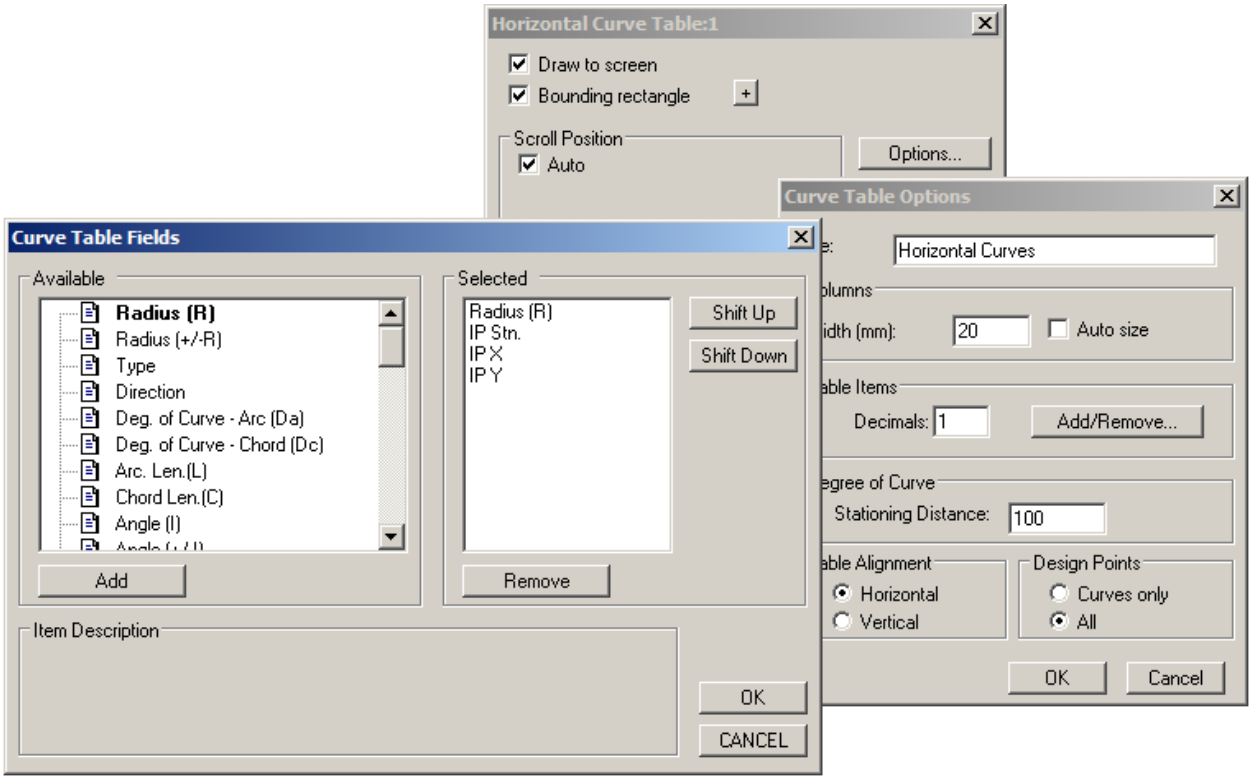


Figure 22.22: Horizontal Curves table options dialog boxes.

10. *Add* and *Remove* items (double click works) until you have only *Radius (R)*, *IP Stn*, *IP X* and *IP Y* in the selected column.

11. Press OK in all three dialog boxes to see the results (figure below)

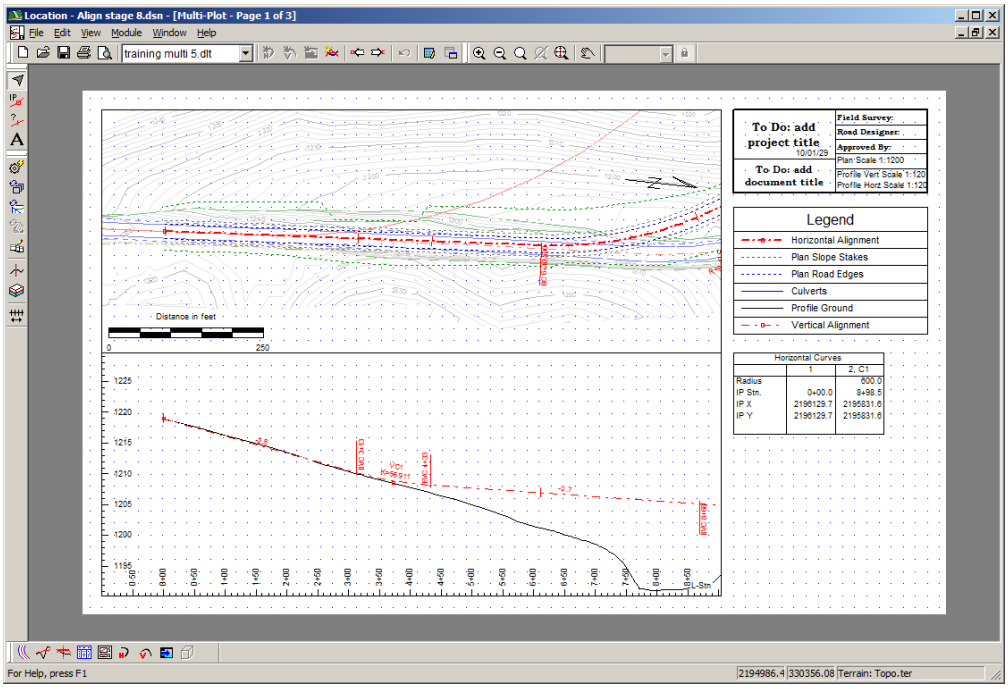


Figure 22.23: Horizontal curve table after configuration.

12. Save your modified screen layout: **training Output.DLT.** See instructions in previous exercises.

If you had trouble with the preceding exercises, screen layout **training multi 6.DLT** contains the final results.

At this point you may want to experiment with some of the other curve table options or maybe create a vertical curve table. You can also use <Ctrl + N> to check all the pages.

13. File | Close; do not save changes.

23. Creating a composite surface

In this section we will export the designed surface from the Location module and merge it into the original ground surface in the Terrain module. The resulting composite surface is ideal for presentation; it is also a starting point for designing an intersecting road.

Exporting Designed surfaces

1. Open the Location module.
2. Choose menu File | Open, select **Examples\Location\Align stage 8.DSN**. Press *Open*.

Assuming that this design is finished, we want to export the designed surface.

3. Choose menu File | Save As to open the file save dialog box.
    a. Set the *type* to *Terrain File (*.ter).*
    b. Change folders to **Examples\Composite**.
    c. *Name* the output file **Road Surface xx,** where *xx* are your initials (we don't want to write over the example file).

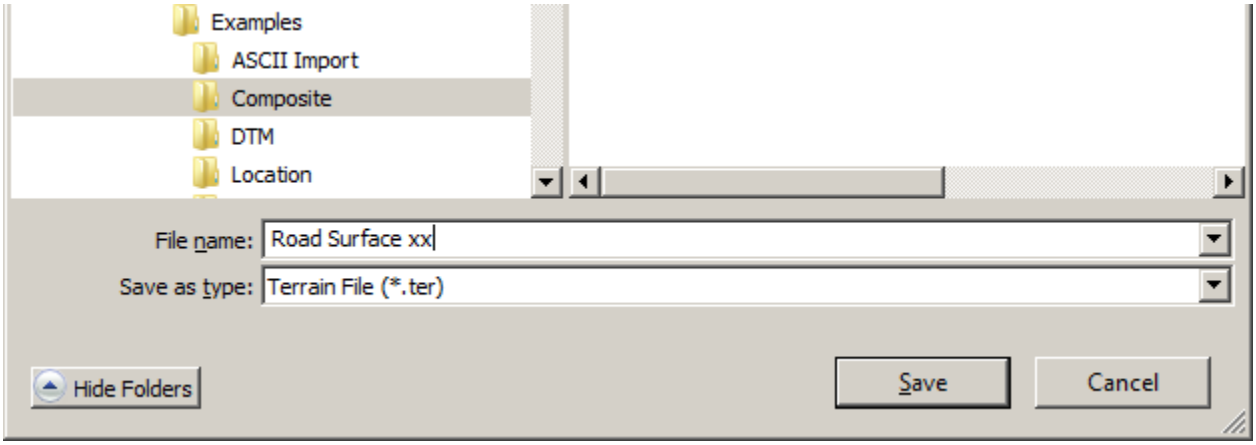


Figure 23.1: File Save As dialog box ready to export a Terrain file from the Location module.

4. Press *Save*; the *Export to Terrain* options dialog box will open (figure below).

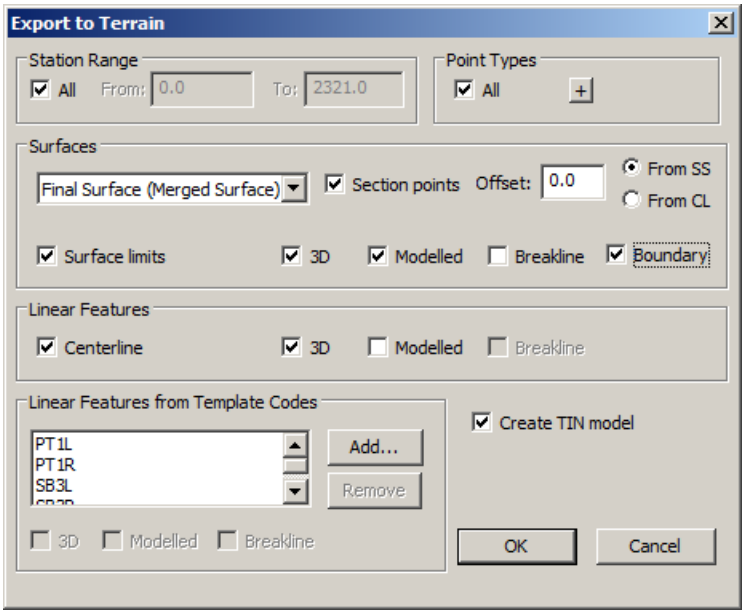


Figure 23.2: Export to Terrain *options dialog box.*

The *Export to Terrain* function can be used for quite different purposes; you may wish to:

- generate a construction surface for staking or digitally controlled grading
- export alignments for use as reference features in another design
- export alignments for use as displayed features in a map or other plan drawing
- export the designed sub-grade or finished grade to create a composite designed surface.

Most of the items in the dialog box are set correctly by default; we will only explicitly deal with some of the features below. Type <F1> to see a description of every control in the *Export to Terrain* dialog box.

5. Make sure the *Surface* selected is *Final Surface (Merged Surface)*. We want to export the surface of the road as if it were complete.

6. Make sure that *Section points* is checked and that *Offset* is 0.0 From *SS*. We will export data up to the slope stake lines (SS) but no further; in other words, we will export the disturbed area.

7. Make sure that the *Surface Limits* item is set and also set the *Boundary* check box to the right of it. This will limit our surface to the stay within the stake lines (SS).

8. Ensure that the *Create TIN model* check box is set.

Although data points will be exported for all template points that define the surface, it is often desirable to make point codes into linear features; connect the dots. There should already be four items in the *Linear Features from Template Codes* list:

- PT1L – pavement edge (left)
- PT1L – pavement edge (right)
- SB3L – shoulder edge (left)
- SB3R – shoulder edge (right)

All of these point codes will be connected together to create break-lines (select an item to see its properties below).

We would like to add ditch bottom features to the list.

9. Press the Add button to open the dialog box shown below.

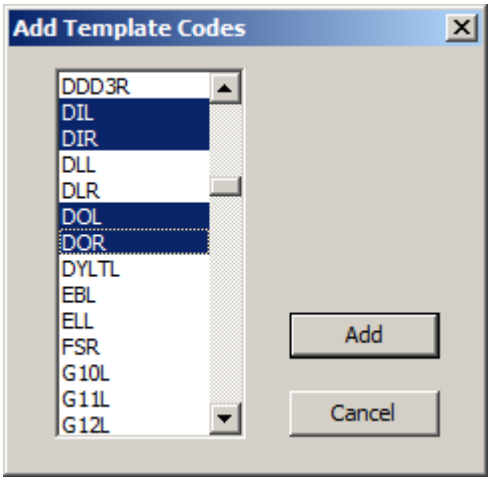


Figure 23.3: Add Template Codes *dialog box after selecting ditch bottom point codes.*

10. Scroll down and select the DIL, DIR, DOL and DOR point codes (ditch inside, outside left and right) as shown in the figure above (use <Ctrl + click> to select/deselect multiple items).

Note: Template codes are defined in the template editor.

11. Press the *Add* button to close.

12. Select all the new items and set the *Breakline* check box.

We have finished setting the options for export. It is useful to note that these options are saved with the Location design when you save it.

13. Press the *OK* button to export the Terrain file.

Merging Terrains

Now we will merge the designed surface created above with the original ground terrain to make a composite.

1. Open the **Terrain** module (the Location module menu Module | To Terrain is handy).

2. Choose menu File | Open, select **Examples\Location\topo.ter**. Press *Open*.

First, we need to save this file to a new location so we don't ruin our location design by modifying the original ground terrain.

1. Choose menu File | Save As to open the file save dialog box.

    a. Leave the *type* as *Terrain File (*.ter).*

    b. Change folders to **Examples\Composite**.

c. *Name* the output file **Composite Surface xx,** where *xx* are your initials (we don't want to write over the example file).

d. Press *Save* to copy the file.

2. Choose menu Edit | Terrain Modeling | Merge or press the *Merge terrain* toolbar button . This will open the *Merge Surface Dialog* box (below).

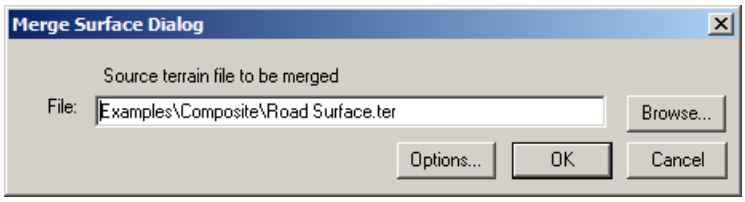


Figure 23.4: Merge Surface Dialog *box.*

3. *Browse* for **Examples\Composite\Road Surface.ter** or the file you created in the first part of the exercise.

4. Press *OK* to merge the Terrains.

Now we need to re-calculate the surface.

5. Choose menu Edit | Terrain Modeling | Calculate Terrain Model or press the *Generate TIN* toolbar button . This will open the *Terrain Calculation* Dialog box.

6. Keep the existing settings and press *OK* to recalculate the triangles and contours.

7. Use menu Window | New Window | Graphics | 3D to create a 3D view if desired.

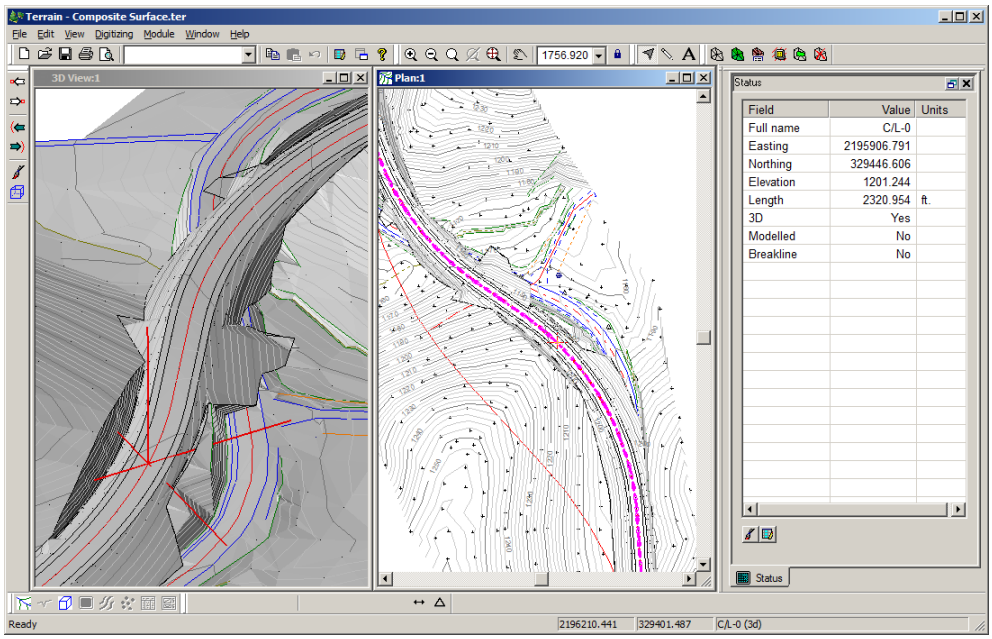


Figure 23.5: Composite surface showing designed road merged with original ground.

If you had trouble with the previous exercise, open **Examples\Composite\Composite Surface.ter**.

Iterative alignment design

This composite surface model could now be used as the *original ground surface* for a new Location design. We might do this to design an intersecting road, driveway or overpass. This would ensure grade elevations are coincident (or grade separation in the case of an overpass) and would avoid any double counting of volumes. We might also wish to design the other direction for a divided highway.

24. Calculating As-built Volumes

This example will demonstrate how to calculate the volume between an original ground surface and several as-built surfaces (stripping, undercut and final grade).

RoadEng® Civil version 5.1123 or later is required for this example.

Prior to starting this example the following TIN surfaces were created from re-measure survey data:

OG.ter	original ground
WG.ter	waste surface after the stripping of topsoil
UC.ter	undercut (or sub-cut) surface.
FG.ter	final grade.

We have also placed the centerline alignment in **OG.ter** as a 'draped feature' (*Properties* set to *Modeled* but no *Elevations*).

Note: Files for this example are located in the **Examples\As-Built** folder (usually on your desktop).

Setup of Alignments and Surfaces

We'll start by creating a new Location design.

1. Start the Location module and choose menu File | New.

2. When prompted for the original ground terrain, select file **OG.ter**; press *Open*.

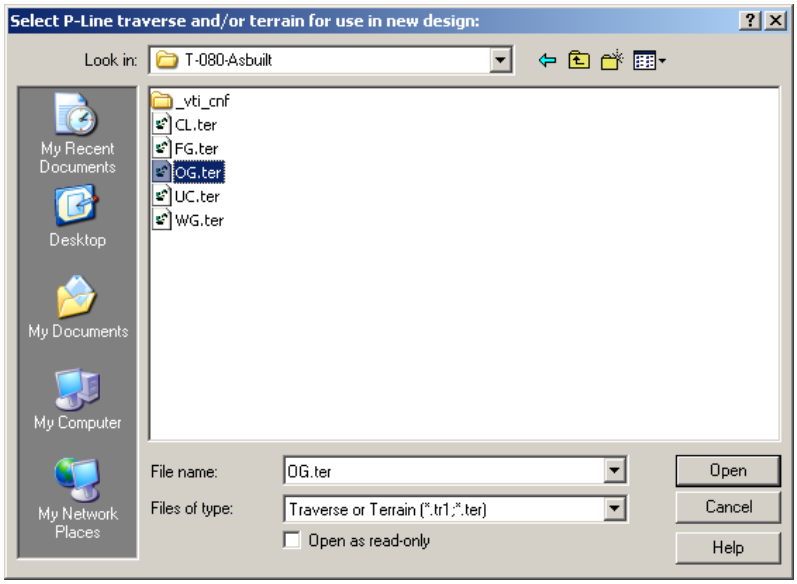


Figure 24-1: New Location Design Dialog

3. When the *New Location start coordinates* dialog appears, choose *Terrain current feature, all points*.

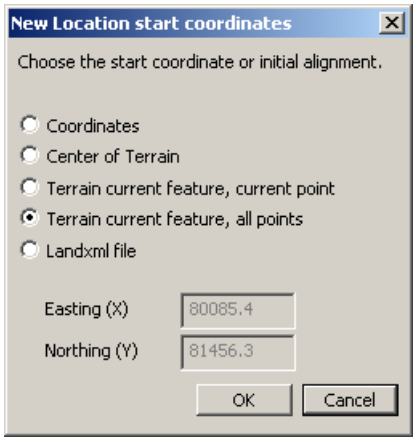


Figure 24-2: New Location start coordinates dialog. The selected option will read the initial alignment from the original ground terrain.

We will now retrieve a screen layout (**training As-Built.dlt**) this will ensure your screen displays match those in this example.

4. Choose menu File | Retrieve Screen Layout and *Open* **training As-Built.dlt**.

5. Choose menu Module | Setup to open the dialog box below. Select the *Alignment* tab.

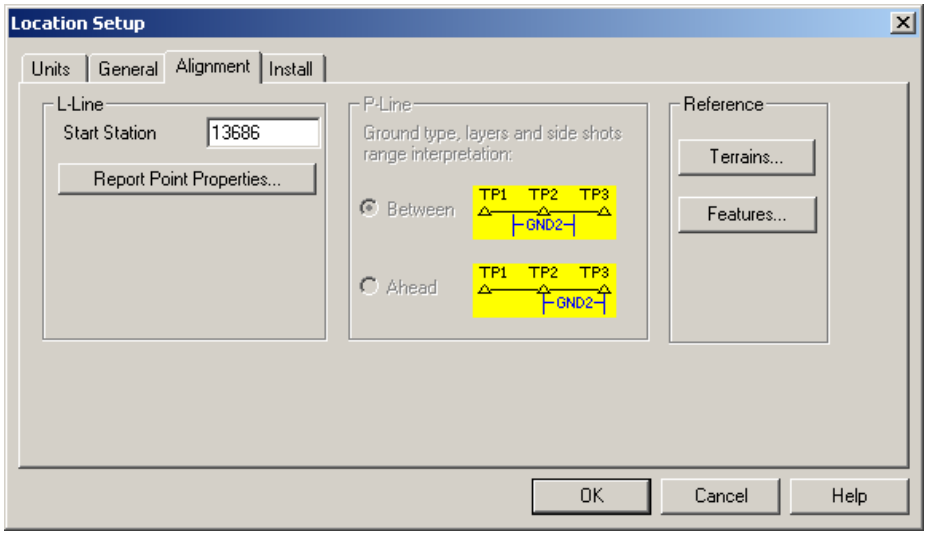


Figure 24-3: Setup *dialog,* Alignment *tab.*

6. Set the *Start Station* to **13686** (don't close the dialog box yet!).

We will now define the as built surfaces as *reference surfaces* in our design file.

Reference surfaces are used for display and control of templates and volumes. For this example we will set them to be the *stripped*, *undercut* and *final* surfaces respectively.

7. In the current dialog box (figure above), click on the *Terrains* button to open the dialog box below.

8. Scroll down in the surface list and select *Reference terrain 1*.
    a. Click the *Browse* button and *Open* file **WG.ter**.
    b. Type in a *Description* (“stripped surface”)
    c. Turn on *Include in cross section*
    d. Set the *Layer* to *Srf 10* (we’ll change the layer description later in this example).
9. Similarly, set *Reference terrain 2* to **UC.ter** (“under-cut”) and *Layer Srf11*.
10. Similarly, set *Reference terrain 3* to **FG.ter** (“final grade”) and *Layer Srf12*.

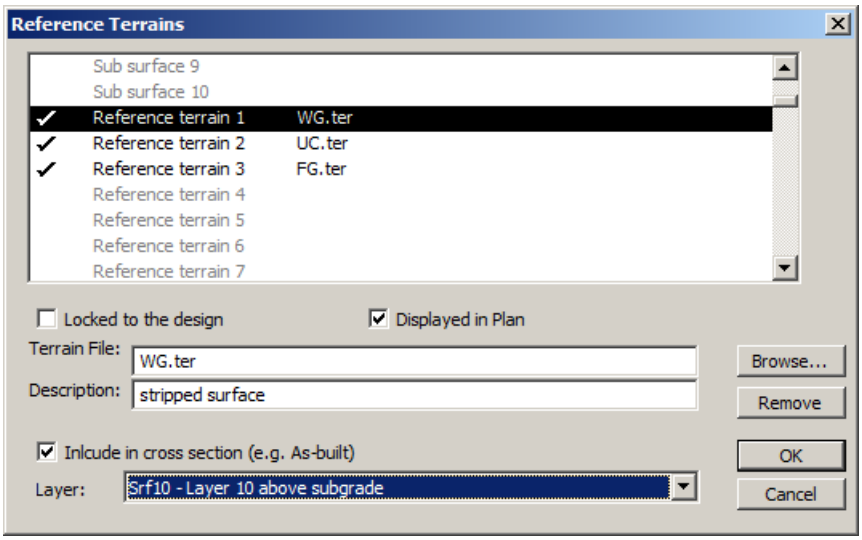


Figure 24-4: New Location start coordinates dialog

Note: The order of reference terrain layers is important if we are to get realistic volume calculations later. They should be in chronological order: *stripping* before *under-cut* before *final grade*.

11. Press *OK* twice to return to the main screen.

We will now setup the *As-built surfaces* descriptions.

12. Choose menu Edit | Edit Templates (or use *Edit Templates* toolbar button) to open the template editor.
13. Click on the *Open Table* button (lower left), browse to the folder containing this example and *Open* **training AsbuiltSurfaces.tpl**.

14. Click on the *Materials* button (lower right) to open the dialog box shown below.

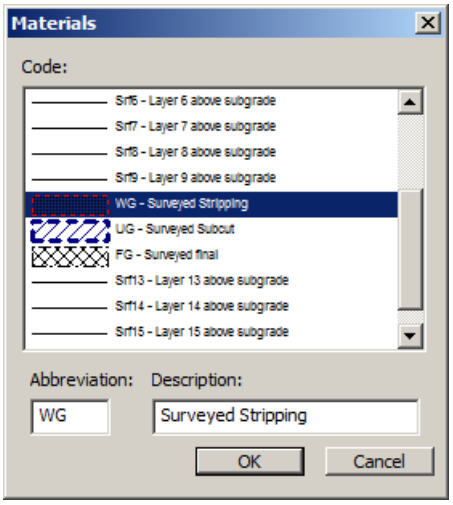


Figure 24-5: As-built surface descriptions

15. Scroll down and you will notice that material descriptions have been set for Srf10, Srf11 and Srf12.

Note: Material names are stored in *template tables* (and the template table is stored with the design document).

On the other hand, Material *line type*, *color* and *hatch* information is stored with screen layout files. The ones you see now came from settings stored in **training As-Built.dlt**.

16. Press *Cancel* to return to the template editor and *OK* to return main screen.

17. Respond *No* to the *Recalculate road alignment* prompt.

We will now disable the design template so that the only the As-built surfaces will be included in our calculations.

18. Choose menu Edit | Assign Parameters by Range (or press the *Assign by Range* toolbar button) to open the dialog box shown below.

19. Select the *Templates* tab (if it is not already selected).

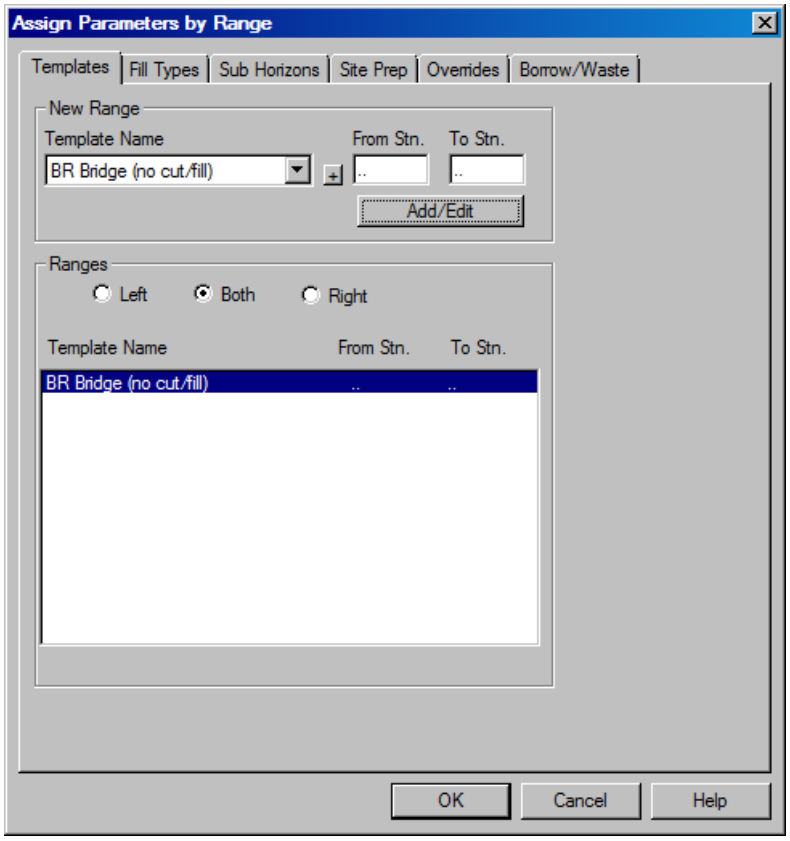


Figure 24-6: Template Assignments

20. In the *Template Name* area, choose *BR- Bridge (no cut/fill)*. Press the *Add/Edit* button to assign this NULL template to the entire length of the road (*From* and *To* are set to "**..**"). Press *OK* to return to the main screen and choose *Yes* to *Recalculate road alignment*.

21. Select menu View | Jump to Station; type **13740** in the station field and Press *OK* to set the current station and update the cross section.

Your screen should now look like the figure below.

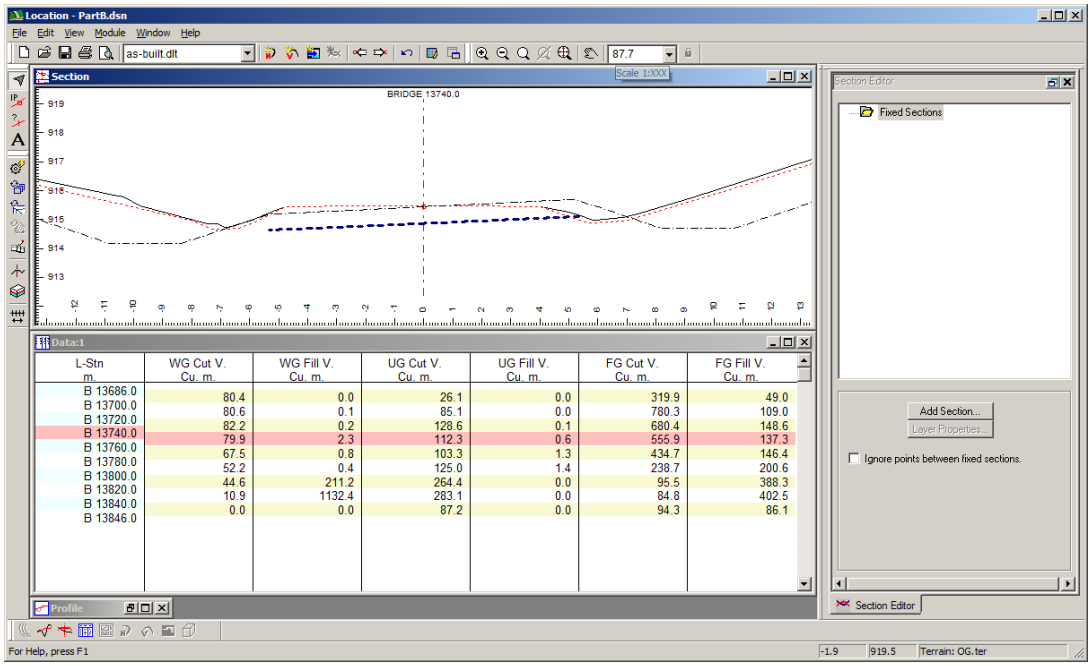


Figure 24-7: As Built Surfaces

We now have a volume report and cross section graphics based on our re-measure surveys. In many cases we could stop here, however, in this case there is one obvious problem with the volume calculations. *WG* (stripping) should never fill.

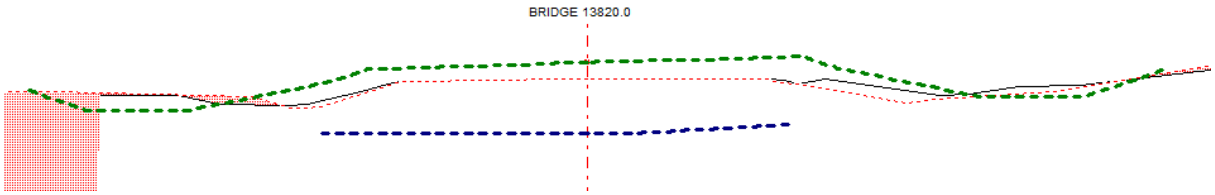


Figure 24-8: Section with stripping fill area shaded.

The figure above shows how the stripped surface extends outside the original ground surface on the left (station 13820); this generates the large fill shown in the volumes display. The WG surface needs to be *tied off* so it does not extend outside OG.

Adjusting As-built Surfaces

As we found at the end of the previous section, survey data (and the resulting surfaces) are not always perfect. It is not the fault of the surveyor if the stripped surface survey extends a few meters beyond the original ground survey; and yet this will produce large fictitious fill volumes. In some cases it is desirable to create a surface that was not surveyed or to fix obvious glitches in the survey.

Rather than try and modify the *surfaces* we will first create *fixed cross sections* from the existing surfaces and then modify the cross sections layers.

Creating fixed cross sections

If you are continuing from the previous section you can skip the next step.

1. Start the Location module and choose menu File | Open. *Open* file **PartB.dsn** from the example files included with this exercise.

This location design is currently displaying four surfaces (extracted from surveyed Terrain files):

OG.ter	original ground
WG.ter	waste surface after the stripping of topsoil
UC.ter	under-cut (or sub-cut) surface.
FG.ter	final grade.

2. In the *Fixed Section* Panel on the right side of the screen click on the *Add Section* button to open the dialog box below.

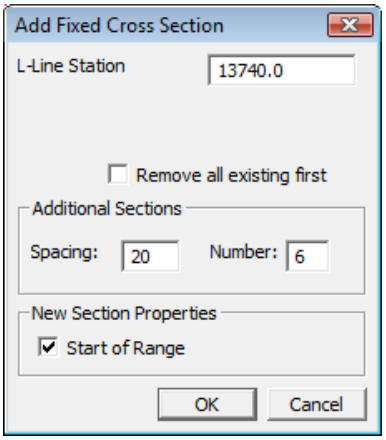


Figure 24-9: Add Fixed Cross Section *Dialog*

3. Enter *L-line Station* **13740**, *Spacing*: **20** and *Number* **6**. Also set *Start of Range* (this will be explained later in the Volumes section) Press *OK*.

Notice that the new items appear in the *Section Editor* in a tree format.

4. Click on the ⊞ controls in the tree to expand a section (figure below)

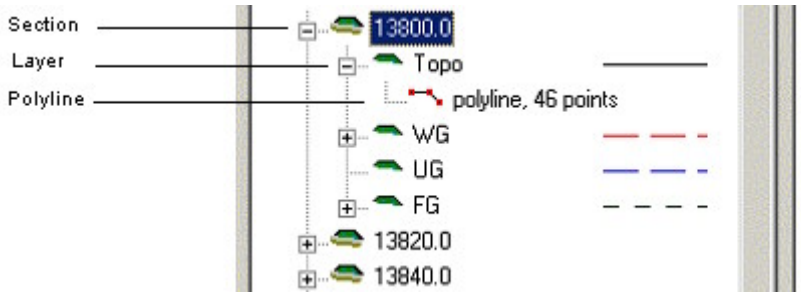


Figure 8: Section Editor *Tree*

We have created fixed sections from station 13740 to 13840. Because we set the *Start of Range* property for all sections, no other cross sections are calculated in this 100m range. Although the layer *polylines* were created from the surfaces (draped) the surfaces are now ignored inside this 100m range. See *Fixed Ranges* in the *Volume Calculations* section below.

Editing layer polylines

We will now edit the as-built surfaces on a cross section to "tie them off" correctly.

5. In the Section Editor tree-control click on section **13800** (you can also use *Next Point* button  to navigate to the correct section).

6. Expand the tree by clicking on the  button beside **13800** and further by clicking on the  button beside **WG**.

7. Click on the *polyline* item immediately under WG.

Your screen should appear as shown in the figure below.

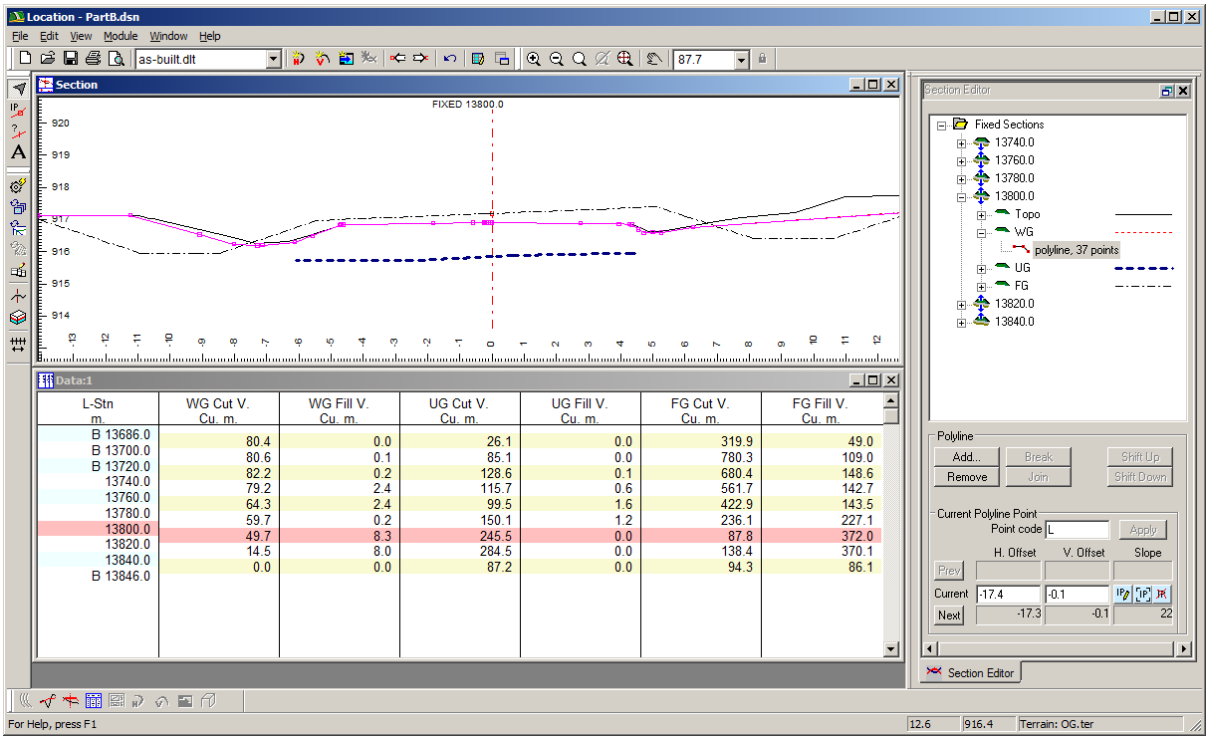


Figure 24-10: WG polyline selected for editing

Note that the **WG** layer is highlighted magenta and that the points have a small box symbol; this should remind you of a selected feature in the Terrain module.

8. Right click in the section window and notice that there are four edit modes available (figure below left). You can also select edit modes in the toolbar (below right)

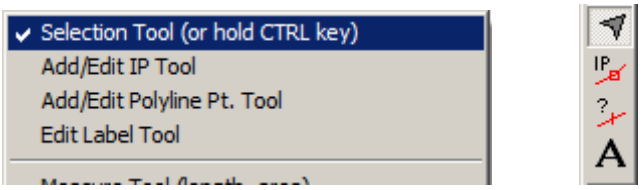


Figure 24-11: Section edit modes. The first and third apply to editing fixed section layer polylines.

The *Add/Edit Polyline Pt. Tool* allows you to edit polyline nodes the same way you edit vertical IPs in the Profile window or feature points in the Terrain module. We will use the *Selection Tool* for the following steps.

9. In the Section window scroll and zoom to the upper left corner (figure below).

First, we will delete the left-most portion of the polyline (the piece that falls off downwards). We do this by first breaking the polyline and then deleting the left most section (the portion which falls off).

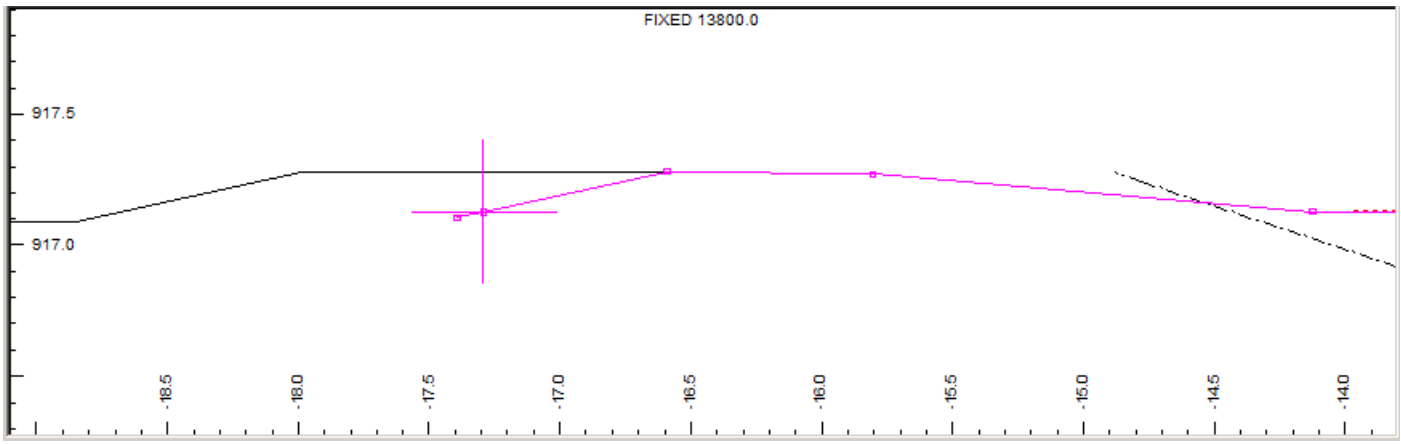


Figure 24-12: WG polyline

10. Move the selection cursor over the point at -17.3 offset from centerline (2nd to last point on the left - see cross position in figure above) and left click to select it.

This sets the current point and updates the data at the bottom of the Section Editor Panel.

11. In the Section Editor Panel press the *Break* button.

We have broken the surface into 2 polylines. Notice the box symbols only appear on the points to the left of the current point, indicating that the left polyline is selected. Also, in the Section Panel tree, the new two-point polyline is highlighted.

12. Click on the *Remove* button in the Section Editor panel. The leftmost polyline is removed.

Next we'll edit the leftmost point to tie the polyline into topo.

13. Right click and choose menu *Add / Edit Polyline Pt. Tool*.
    a. Move the cursor over the left most point in the remaining WG polyline; when it changes to a box, left click to capture the point.
    b. Move the point vertically up to topo (note the cursor change as you reach snap range) and left click again to anchor the point.

The figure below shows the cross section after the end point has been tied to topo.

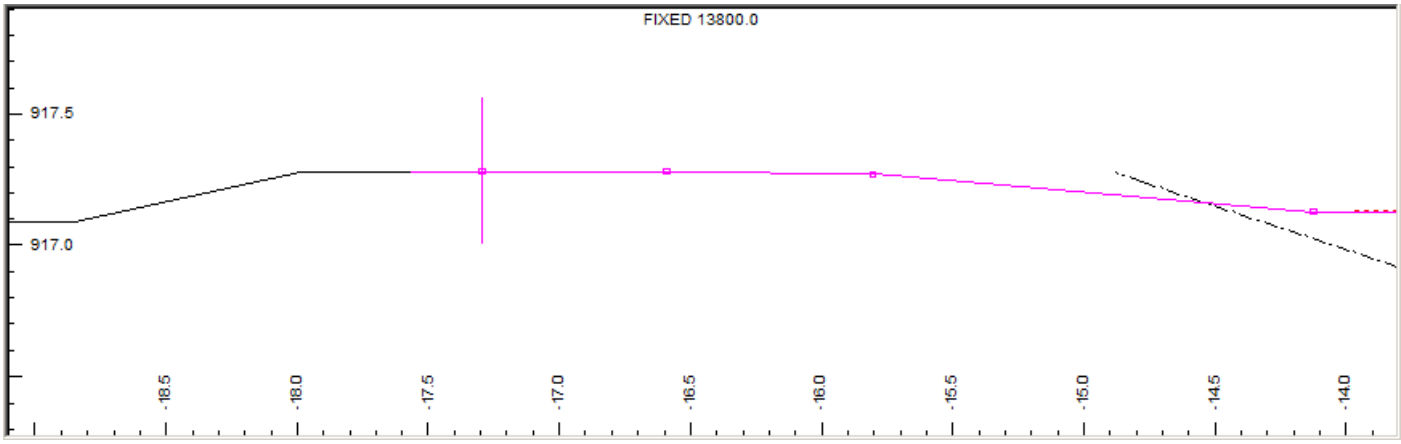


Figure 24-13: Cross Section 13820 After editing

You have probably noticed the editing of fixed section is very similar to editing in other parts of RoadEng. The cross section editor allows you to do the following:

- Add, remove or change a point on any layer or polyline.
- Add or remove layers or polylines.
- Change the order of the layers (more on this in the Volumes section).
- *Break* or *Join* polylines.
- Add or change point codes.
- Enable or disable display and volume calculations for a layer.

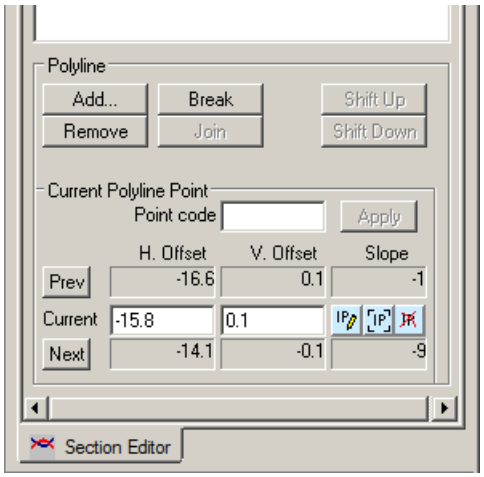


Figure 24-14: Cross Section Editor controls for polyline *edit.*

Volume Calculations

In the Location module, cross section end areas (and therefore volumes) are calculated using a layer merging process which simulates the construction sequence. With *fixed* cross sections you control the shape and order of the layers so it is important to understand this process.

Step 1. *Layer 2* is compared with *layer 1* (usually topo) and the difference is calculated; this defines the cut and fill areas for *layer 2* (in this case stripping).

Step 2. Layers 1 and 2 are *merged* to create a new *merged surface* (in this case the stripped surface)

Step 3. The *merged surface* replaces *layer 1*, *layer 3* replaces *layer 2* and the two steps above are repeated.

Step 4. Repeat until all layers processed.

The figure below shows the progression of the merged surface.

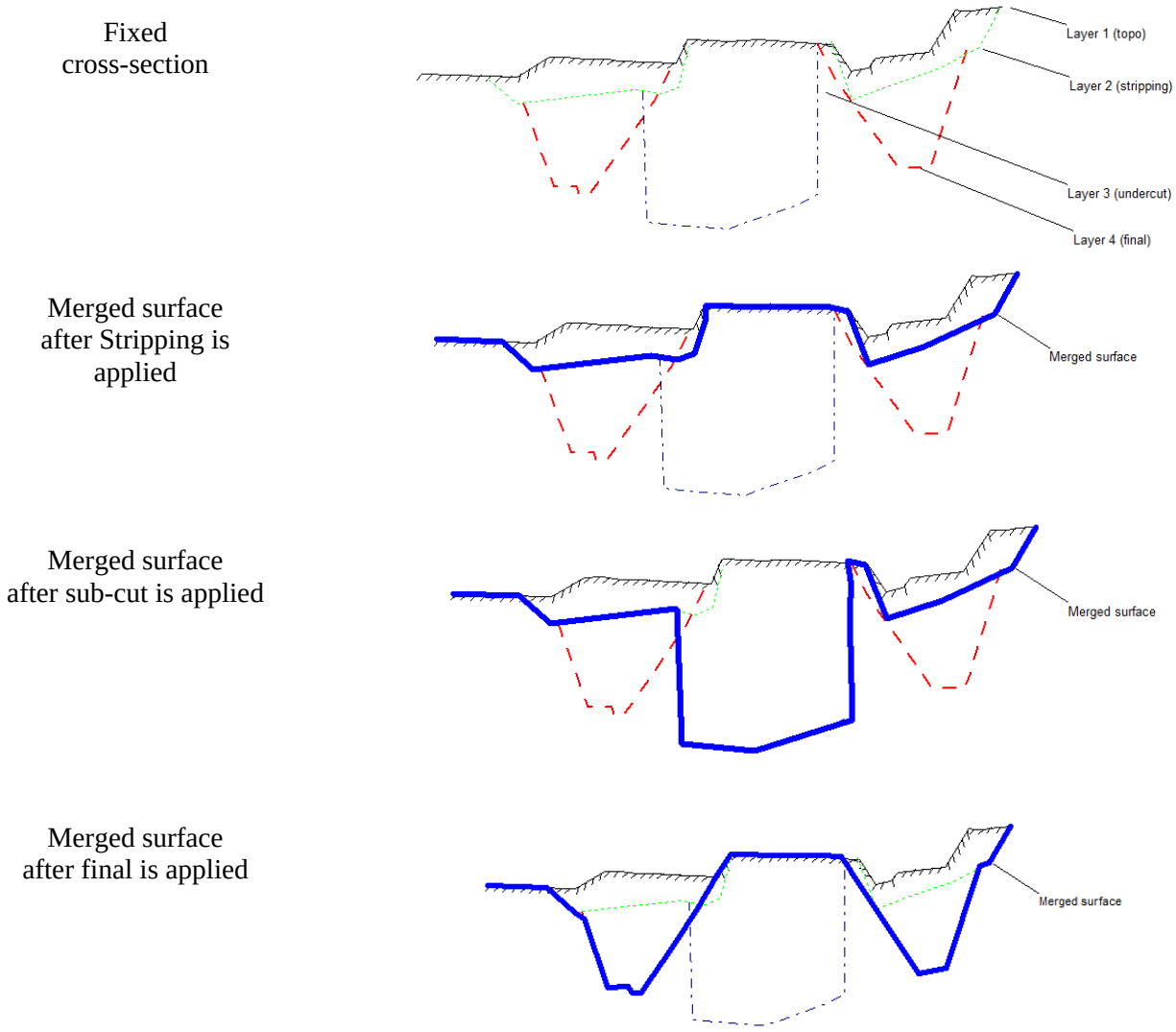


Figure 24-15: Volume Calculations

Note: the order of the layers in the Section Editor Tree-control is very important. Cut fill volumes are calculated between the current layer and the merged surface.

Checking Areas and Volumes

There are several visual aids available to verify cross sectional areas and volumes. This section will explore hatching areas and the information available in hover tips.

If you are continuing from the previous section you can skip the next step.

1. Start the Location module and choose menu File | Open. *Open* file **PartC.dsn**.

2. Right click on the Section window and choose *Section Options*.

3. In the *Section Options* dialog box, click on the + button beside *Template* to activate the *Template Display Format* dialog (figure below).

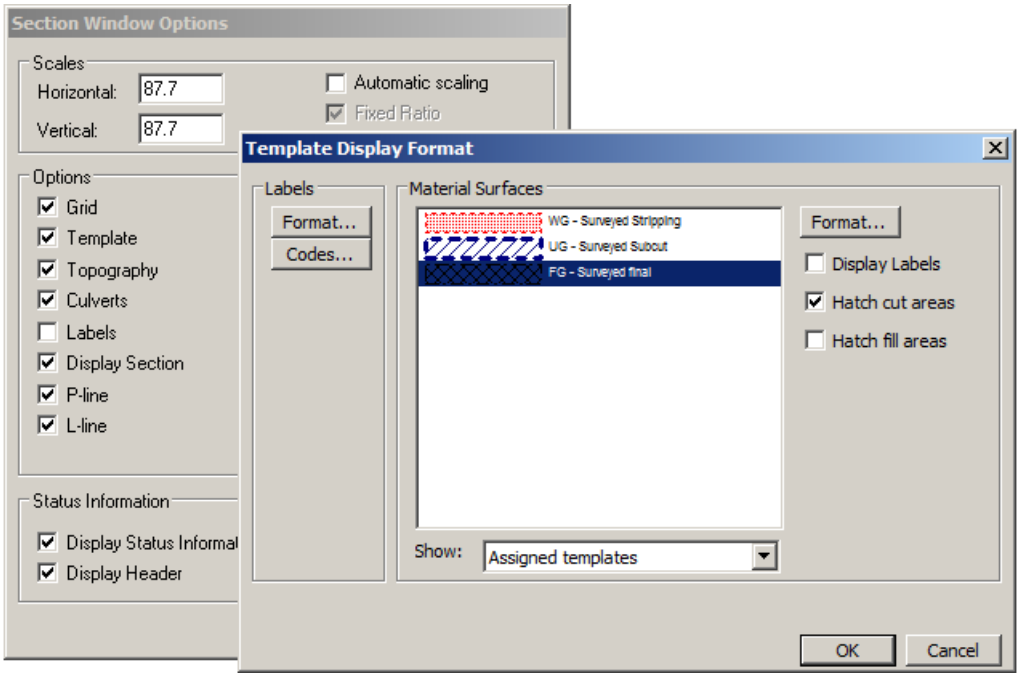


Figure 24-16: Template Display Format *dialog*

4. Set the *Hatch Cut Areas* check box for each of the three material surfaces.

Note that you can control the color, line type and hatch of the selected material by clicking on the *Format* button.

5. Press *OK* twice to return to the main screen.

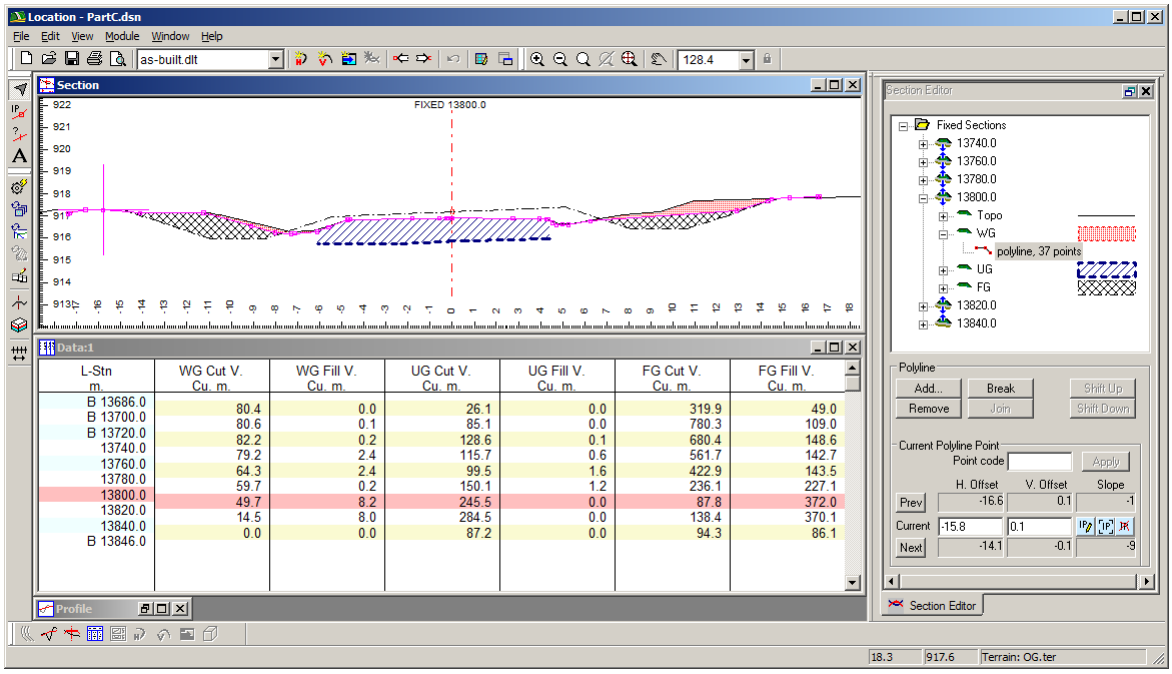


Figure 24-17: Section Window with Cut Areas Hatched

Note the hatched areas correspond to the areas used for volume calculations.

6. In the Section window, hover your cursor over one of the cut areas (figure below).

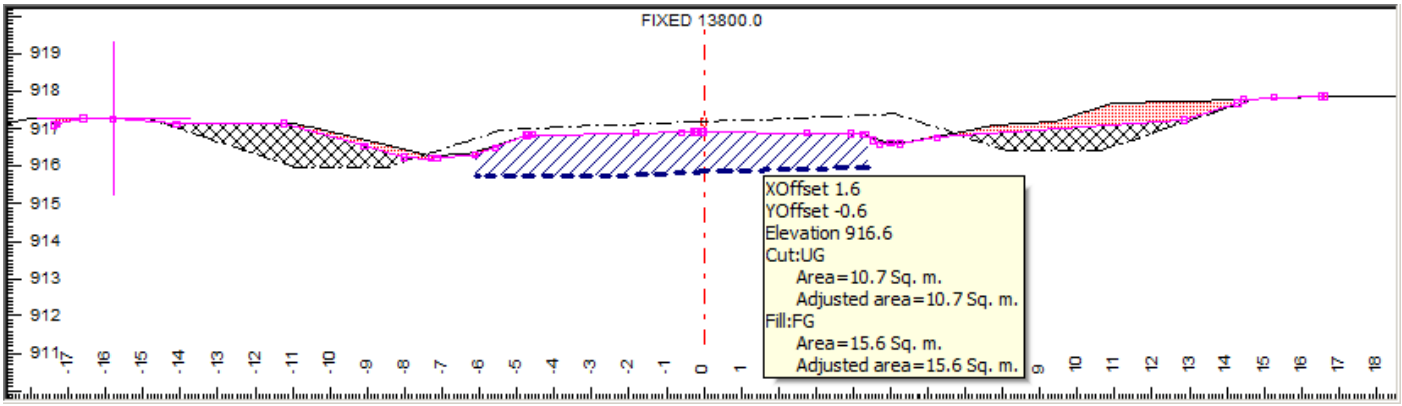


Figure 24-18: Section Window hover tip showing cut and fill areas.

The hover tip shown in the figure above indicates the position of the cursor with respect to the centerline as well as cut and fill information. You may want to try turning on *fill* hatching to see how the final grade is filled after the sub cut is extracted.

Note: Right click in the Section window and notice the hatching options at the bottom of the context menu. The layers available for hatching cut or fill depend on where you click.

Fixed Ranges

Fixed Sections divide the alignment into *fixed ranges* for calculation and reporting. A *fixed range* starts at a *fixed* cross section with the *Start of Fixed Range* property set (figure below right) and extends to the next *fixed* cross section. In the range between these two fixed sections no other cross sections are calculated – the volumes are calculated from the fixed section end areas only.

If a fixed section does not have the *Start of Fixed Range* property set, then cross sections at reporting points following are calculated using the template (or in this example the Terrain surfaces).

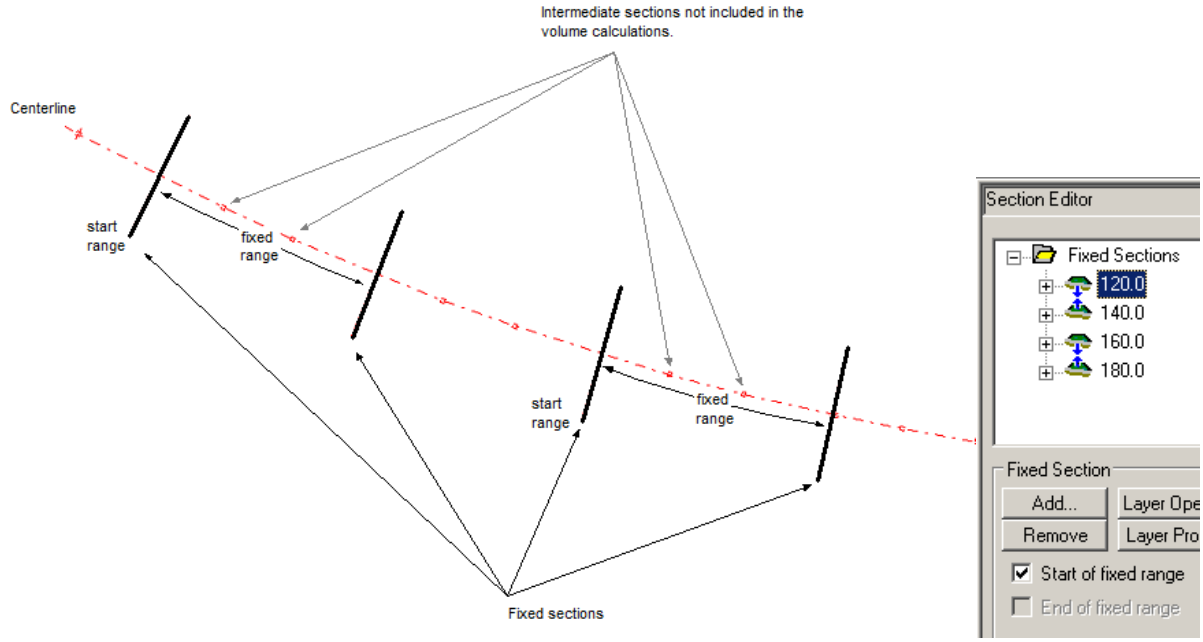


Figure 24-19: Fixed sections and Intermediate Reporting points.

Note: When the *Start of Range* property is set on a *fixed section*, volumes are calculated from the average end areas of this *fixed* cross sections and the one following. Design points in between are ignored.

Note: The Section Editor tree-control indicates the ranges with arrows in the layer icons and by the state of the *Start of fixed range* and *End of fixed range* check boxes.

In this example the 100m range from station 13740 to 13840 is *fixed*.

7. File | Close; do not save changes.

Working with Design Files

In this section, we will calculate quantities between the designed and constructed sub-grade surfaces.

We start by opening an existing road design file.

1. Start the Location module and choose menu File | Open. *Open* file **PartD.dsn** included with this example.

This design contains the desired shape of the sub-grade as well as the layers above sub-grade.

Adding a surveyed layer

The actual, as-built, sub-grade was surveyed, imported into the Terrain module and made into a TIN surface; the result is **FG.ter**. We will now define the as-built surface as *reference surface*.

2. Choose menu Module | Setup to open the *Location Setup* dialog box.

3. Select the *Alignment* tab and press the *Terrains* button to open the *Reference Terrains* dialog box (figure below).

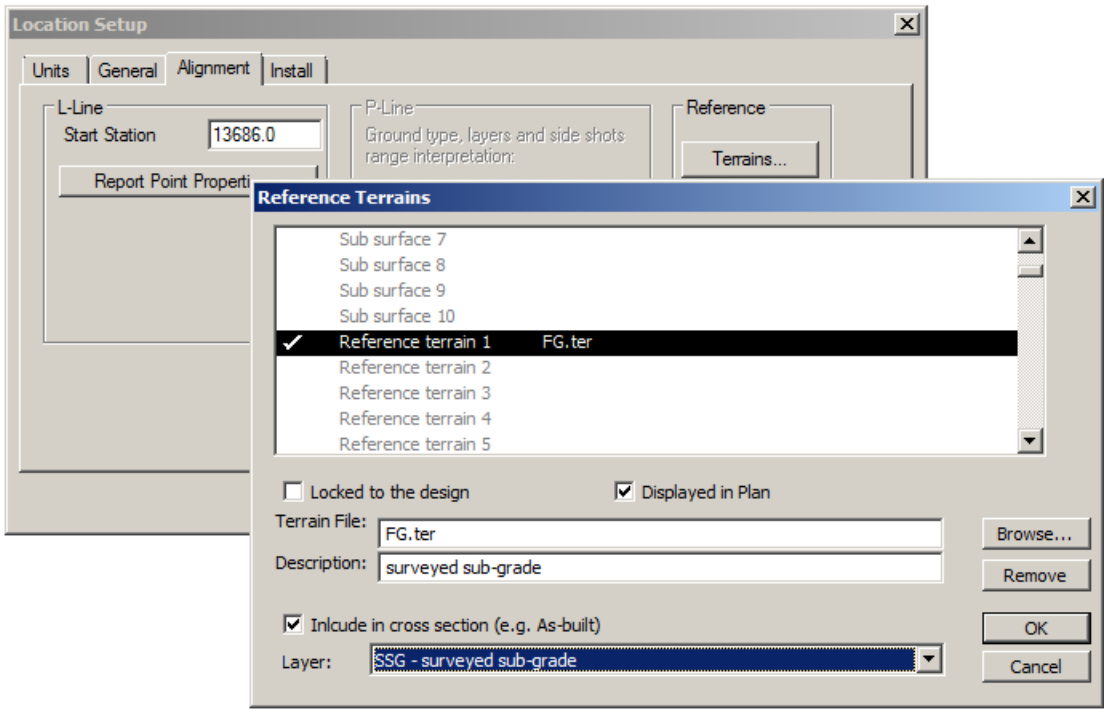


Figure 24-20: Adding the surveyed final grade surface as a reference terrain.

Now we will create a reference terrain that contains a surface to be displayed in the cross section (and used to calculate volumes!).

4. Scroll down in the surface list and select *Reference terrain 1*.
    a. Click the *Browse* button and Open terrain file FG.ter.
    b. Type a *Description*, if desired.
    c. Check *Include in cross section*.
    d. Choose the *SSG Layer*.

5. Press OK twice to return to the main screen.

Your screen should look similar to the figure below.

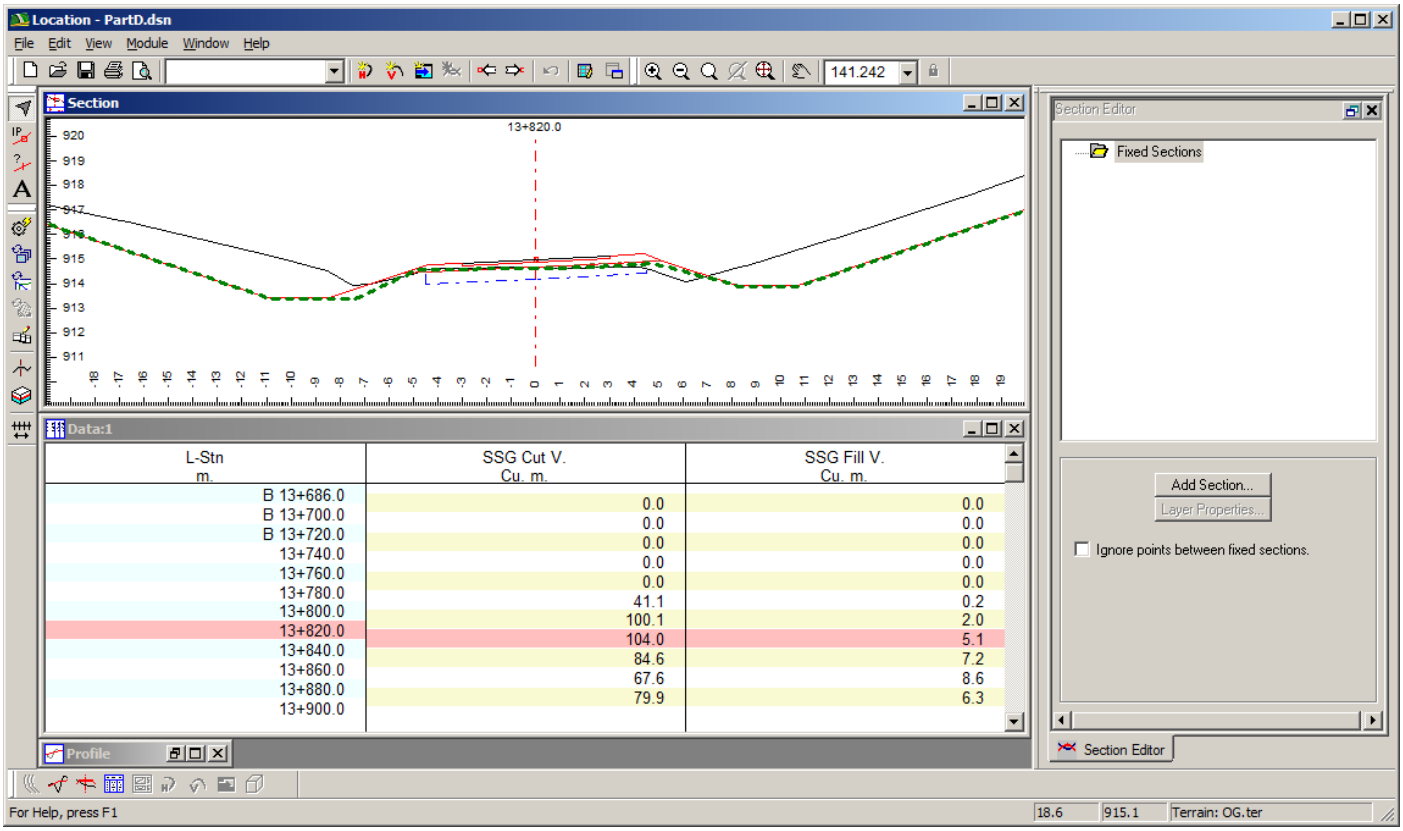


Figure 24-21: Design after reference terrain surface (heavy dashed line) added to cross section.

The heavy dashed line is the surveyed sub-grade (green on your monitor); it is behaving like another template component added after the designed template has been calculated. The cut volumes for layer SSG, shown in the Data window (figure above), are mostly due to the surveyed layer cutting off the designed layers above sub-grade.

6. Right click in the Section window and turn on *Hatch all cut areas*. Zoom in to see the surveyed sub-grade cut (cross hatch – figure below).

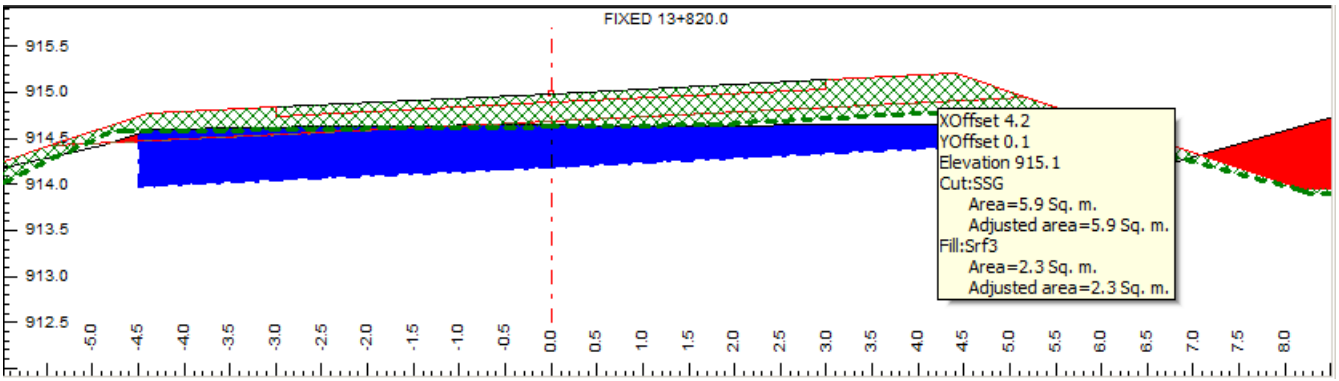


Figure 24-22: Cross section with all cut areas hatched.

Note: Adding a Terrain surface to your sections (using a *reference terrain*) changes the volume calculations and the final merged surface.

7. Right click in the Section window and turn *Hatch all cut areas* off again.

Removing layers using Fixed Sections

To see the difference between the surveyed sub-grade and the designed sub-grade we need to remove un-needed layers. The easiest way to do this is to create *fixed* cross sections. This will also allow us fix any bad cross sections by *tying off* layers (see exercises above).

8. In the Fixed Section Panel on the right side of the screen:
    a. Click on the *Add Section* button.
    b. Enter *L-line Station* **13820**, *Spacing* **20** and *Number* **4**.
    c. Also set *Start of Range* (this is explained in the Volumes section above)
    d. Press *OK*.

Your screen should see your new fixed sections in the Section Editor as shown in the figure below.

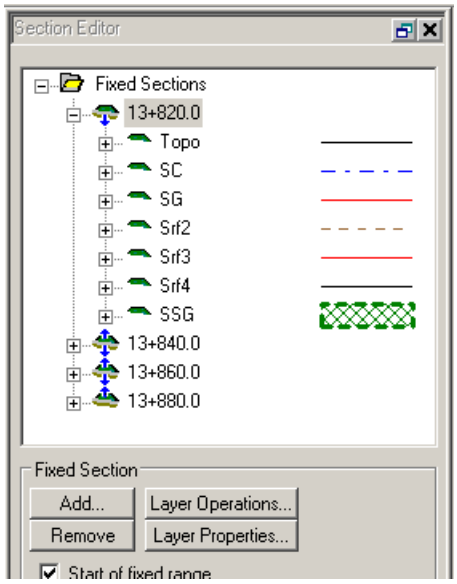


Figure 24-23: Design file with Final As-built surface

We will now turn off the display and volume calculation for some of the unused surfaces.

9. In the Fixed Section Panel on the right side of the screen, click on the *Layer Properties* button (figure above) to open the Fixed Section Layer Properties dialog box (figure below).

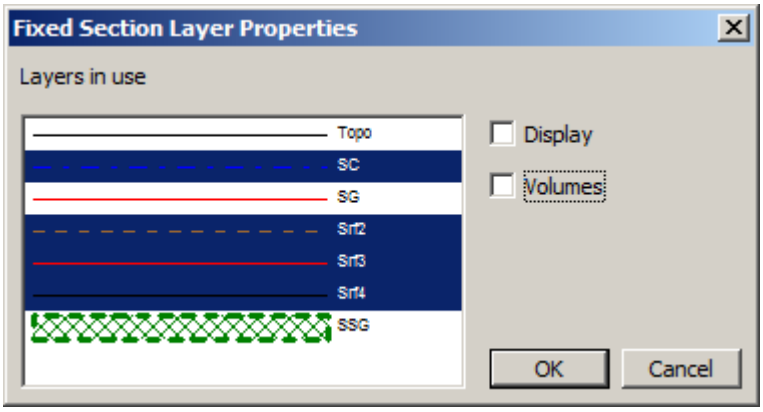


Figure 24-24: Fixed Section Layer Properties Dialog

10. Turn off the *Display* and *Volumes* properties for all layers except Topo, SG and SSG. Press *OK* to accept and close.

The three layers between SG and SSG are the ones we really needed to disable. We could also have removed these layers to achieve the same effect (*Remove* button).

Viewing differences

Now we are ready to examine the differences between designed and as-built sub-grade surfaces. The figure below shows Location set up to display a few of the key points of interest. For brevity, we will not go through explicit step by step examples for this section; these features are covered in other exercises.

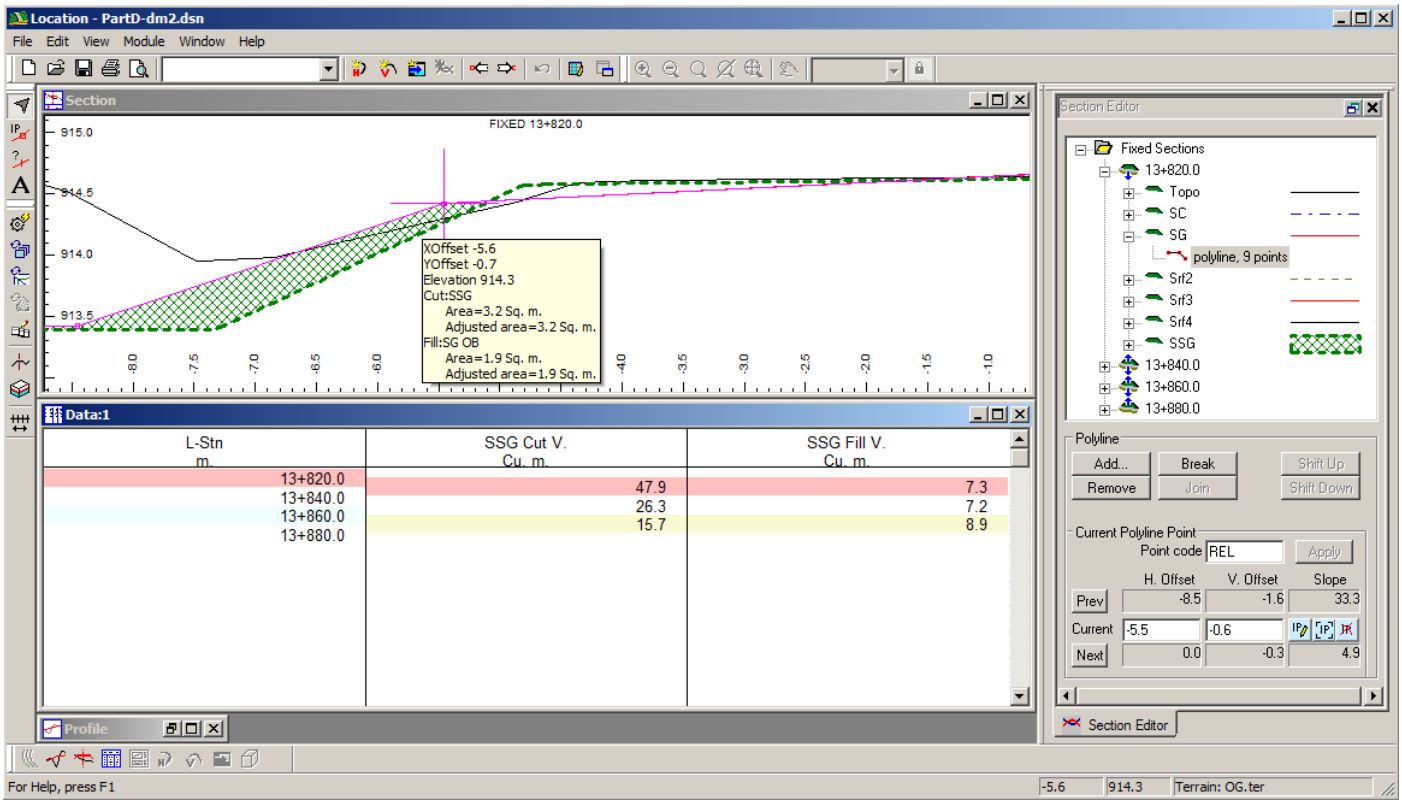


Figure 24-25: Design vs. As-built Quantities

By looking at the Section display, you can easily see where the surveyed sub-grade (SSG) is below the designed sub-grade (SG). The hover tips display cross sectional areas.

The Data window has been set up to show the volumes between the designed and surveyed sub-grade; this is a quantitative measure of over-build and under-build.

The bottom part of the Section Editor panel shows coordinates and slopes for points and segments in the layer *polylines*. If you click in the Section window to select the outer edge of the designed roadway (figure above) you can see that the designed slopes are **33.3%** and **4.9%** respectively. There are too many small segments to get a reasonable measure of the surveyed slopes using this method. However the *Measure tool* shows actual grades of about **47%** and **3.6%** respectively.

What other discrepancies can you find?

This completes our example.

11. Choose menu File | Close; don't save changes.

Creating Surfaces for Pay Quantities

This example we will calculate sub-cut pay quantities where the sub-cut material must be below both stripping (WG) and sub-grade (SG). To do this calculation we will automatically create a new surface (SCTop: sub-cut top) which runs along the top of the sub-cut. The SCTop surface is the minimum of WG and SG.

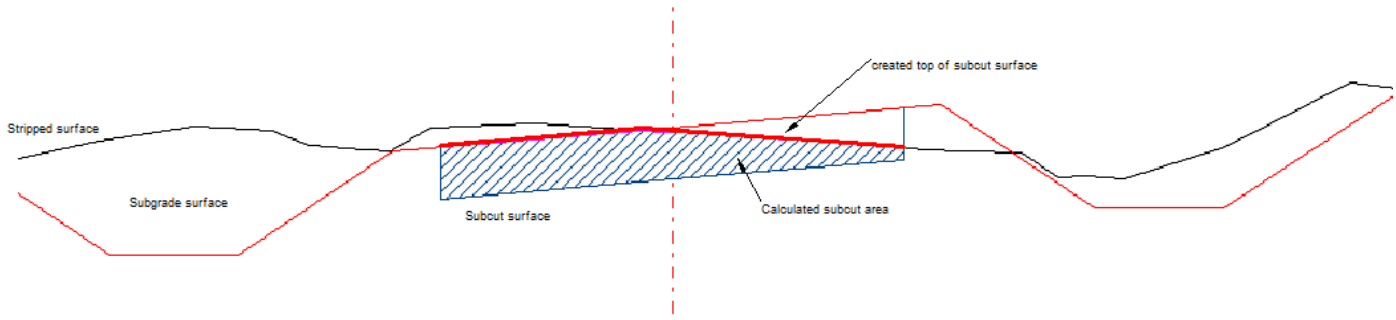


Figure 24-26: Top of Sub-cut Surface

We start by opening an existing design file created from an as-built survey.

1. Start the Location module and choose menu File | Open. *Open* file **PartE.dsn**.

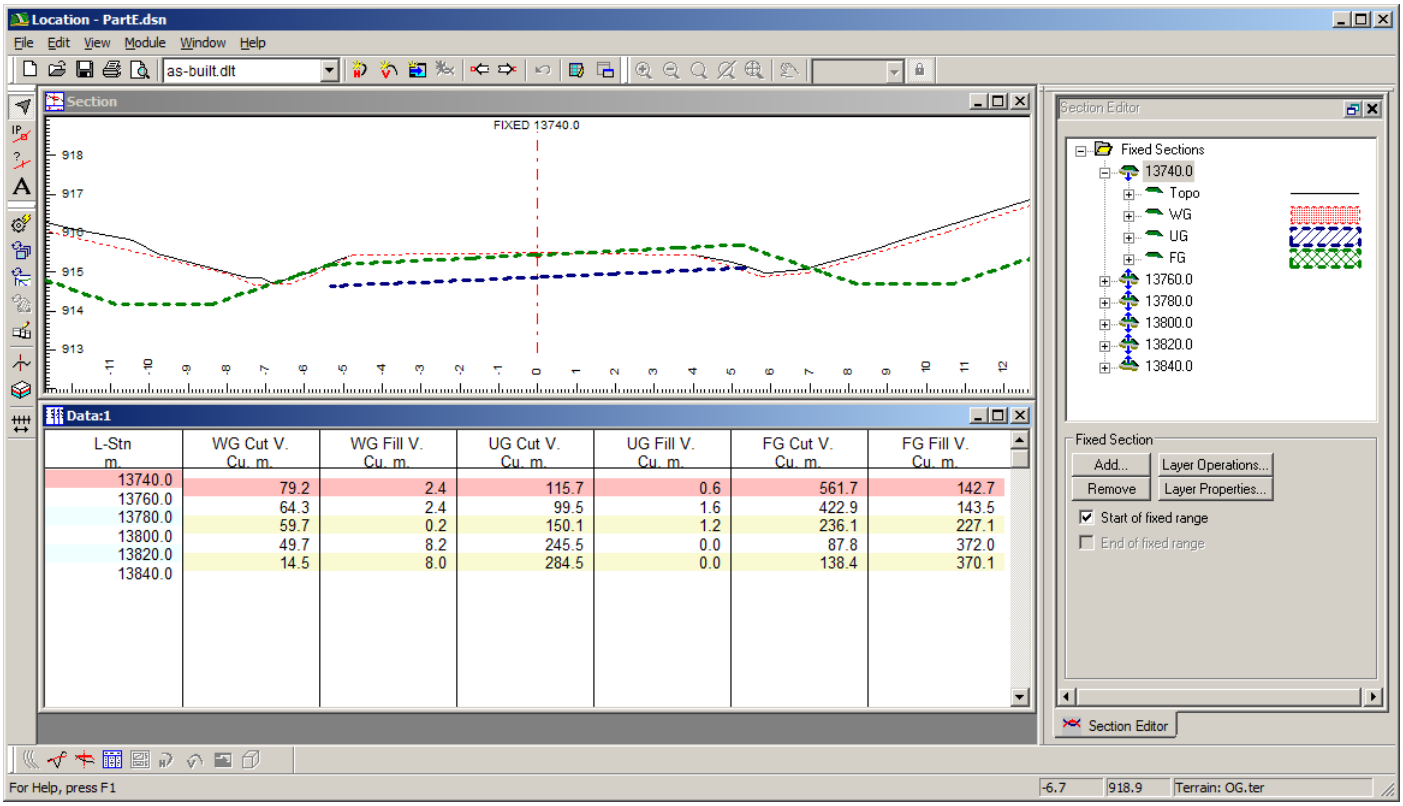


Figure 24-27: Design created from an as-built survey, with Fixed Sections

2. In the Section Editor tree-control, select station 13740 and press the *Layer Operations* button to open the *Layer Operations* dialog box shown below.

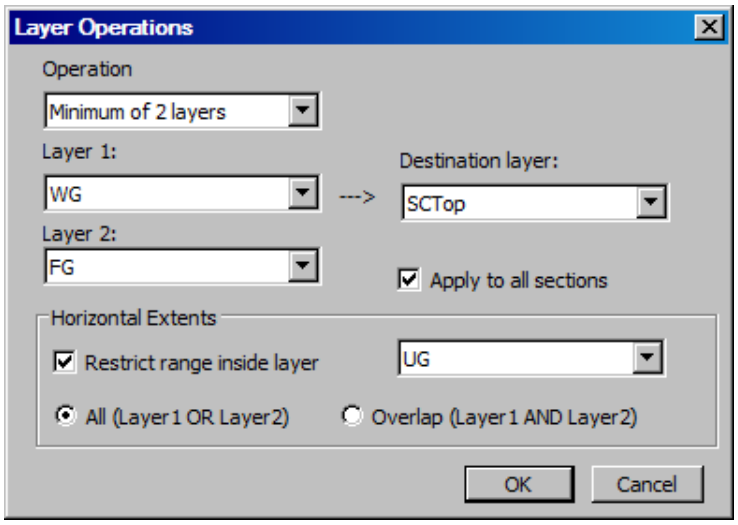


Figure 24-28: Layer Operations dialog

3. Set the parameters as shown in the figure above.

When you press OK, These parameters will create a new surface (SCTop) which is the minimum (in elevation) of WG and FG. This process is applied to all cross sections. The *Horizontal Extents* of the new surface will not extend outside layer UG. SCtop will exist anywhere either WG or FG exist (union).

4. Press *OK* to create the new Layer.

The next step will shift the new SCTop surface so that it is processed after WG and before SC. The new processing order will be:

Topo	original ground
WG	stripping
SCTop	sub grade (above sub-cut)
UG	Sub-cut
FG	Sub-grade (outside sub-cut)

Cut and fill volumes are calculated after each step and a new merged surface is created (see *Part C: Volume Calculations*).

5. In the Section Editor tree-control, select station **13740** and press the button to expand the tree (if necessary).
    a. Select the new SCTop layer (it should be at the bottom).
    b. Press the Shift Up button until SCTop is between WG and UG.
    c. Press the *Apply to All Sections* button to update the order on the other sections.

The next step will remove any of the UG layer which extends above WG or FG (figure below). Sub-cut (UG) should never fill.

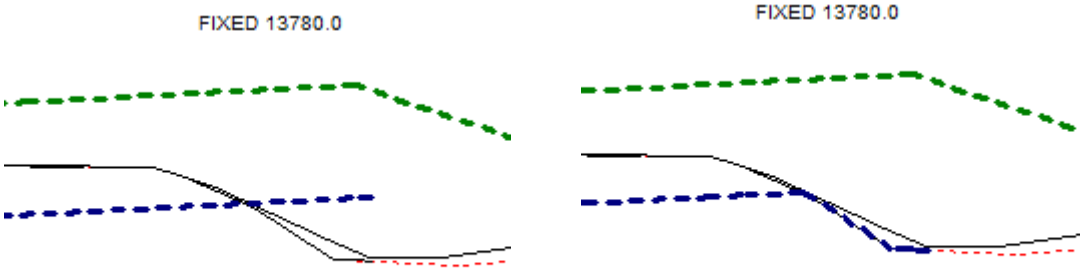


Figure 24-29: Before (left) and after (right) trimming (step below).

6. In the Section Editor tree-control, select station 13740 and press the *Layer Operations* button to activate the Layer Operations dialog. Set the parameters as shown in the figure below.

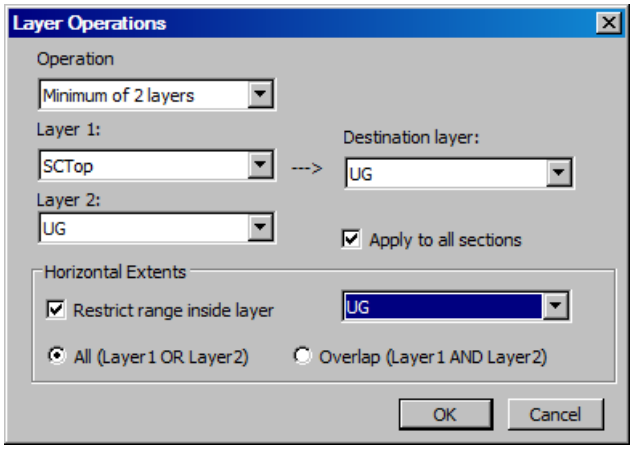


Figure 24-30: Trimming UG above WG or FG

7. Press *OK* and answer yes when prompted to overwrite layer UG.
8. With the mouse in the sub cut area in the cross section, right click and select *Hatch Area for UG*.

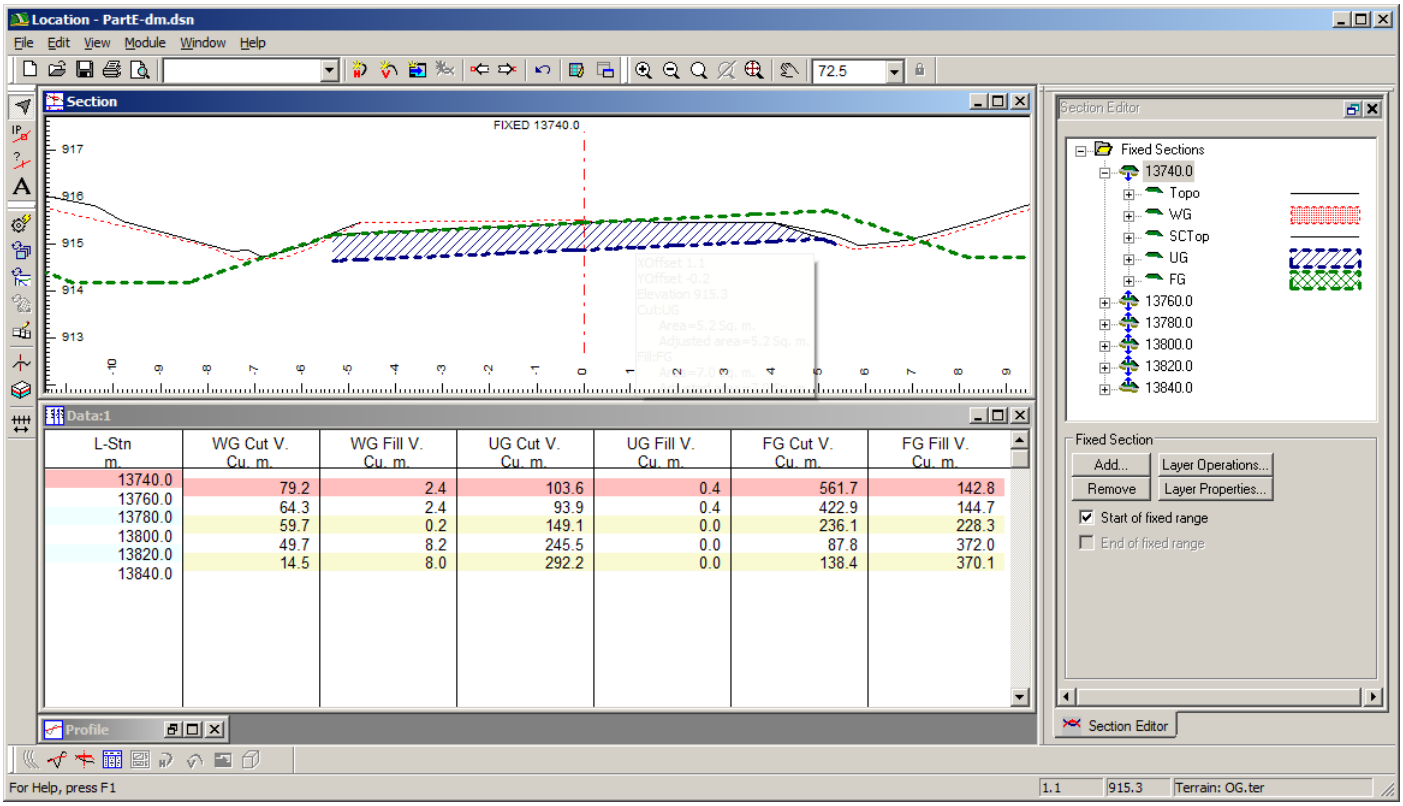


Figure 24-31: Sub cut area

The Data report now accurately shows the pay volume for under-cut under sub-grade.

This completes our example.

9. Choose menu File | Close don't save changes.

25. Index

G

H

I

J

L

M

N

O

P

R

S

T

U

V

W

Z

Training Course Evaluation Form

Date: _____

Location: _____

Course: _____

Instructor: _____

Instructor:	Excellent	Very Good	Good	Fair	Poor
1. Knowledge of subject matter					
2. Listening Skills					
3. presentation skills/delivery					
4. Overall instructor rating					
5. Topic covered in too much detail					
Course Content:	Excellent	Very Good	Good	Fair	Poor
1. Did course achieve its objectives					
2. Exercises					
3. Use of class time					
4. Overall course rating					
Materials:	Excellent	Very Good	Good	Fair	Poor
1. Overall quality of course material					
2. Potential value as future reference material					
3.Value of presentation material					
4. Flow /structure of information					

Please elaborate on any of the above. Note highlights and suggest possible ways to increase the quality, relevance and/or value of the session.