RoadEng®

Tutorial for Civil Applications

Version 9

Softree Technical Systems Inc.

Document Version - December 17, 2019

Trade Marks

AutoCAD and Civil 3D are registered trademarks of Autodesk.
Criterion and LaserSoft are registered trademarks of Laser Technology, Inc.
Microsoft Windows 7, 8, 10, Microsoft Word, and Microsoft Excel are trademarks of Microsoft Corporation
Terrain Tools® and RoadEng® are registered trademarks of Softree Technical Systems Inc.

Suite 215 - 1000 Roosevelt Crescent
North Vancouver, BC
Canada V7P 344

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 set out as if you were doing a road design project from original ground survey to completed construction documents.

Installation

The tutorial files referred to in the following examples can be installed from Softree's Support web site:

- Go to the *Support-Documentation Updates* page on Softree's web site: https://support.softree.com/product-updates/Documentation-Tutorials.
- Once *SoftreeTutorials.exe* has been successfully downloaded.
- *Double-click* on the file to begin installation.

During the installation you will be prompted to select which content to install, we recommend installing all the available tutorial options.

Documents

The tutorial files (data sets) will be installed in the folder below by default:

C:\Users\Public\Documents\softree\training90\RoadEngCivil

We will refer to this folder as **<RoadEngCivil>** in the examples below. It is possible to change this folder at install time; you can also copy it to a new location afterwards if you wish.

Recommendation: To make accessing files easier as you work through the tutorial, we suggest pinning the <RoadEngCivil> folder to your Quick Access menu. To do so, open Windows Explorer, navigate to the folder RoadEngCivil. Right-click on the folder, select "Pin to Quick Access". This will now make the folder available on the left-hand side of Windows Explorer (see figure below).

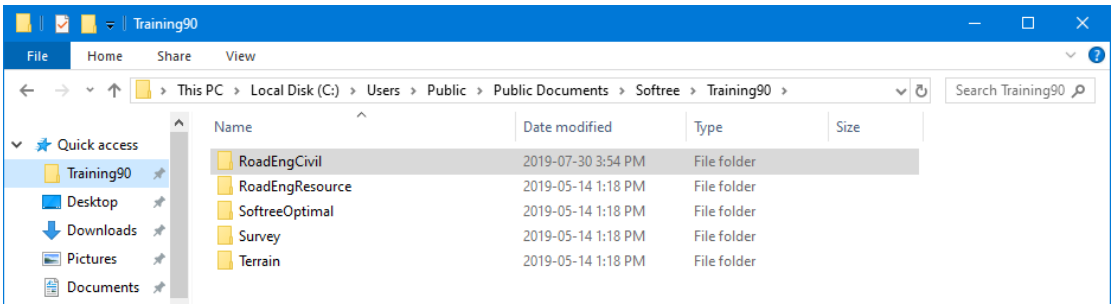

Figure 1-1: Quick Access in Windows Explorer

Don't Save Files (in most cases)

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.

If a file gets modified, delete the files in the training folder. Then re-install the tutorial files (per the original steps).

C:\Users\Public\Documents\softree\training90\RoadEngCivil

Defaults and Layouts

The setup and layout files are stored the folder below by default:

C:\ProgramData\Softree\RoadEng

It is possible to change this folder, so we will refer to it as **<Defaults and Layouts>** in the examples below. A folder containing training specific files has also been added to this location:

<Defaults and Layouts>\

Note: You can always determine the actual **<Defaults and Layouts>** folder by running the Terrain Module, selecting menu *Setup* | *Location Setup* | *Install* tab.

If Softree Optimal is installed after RoadEng, the default folder will be:

C:\ProgramData\Softree\SoftreeOptimal\

Function Groups

Some RoadEng® and Terrain Tools® products have certain features; we classify these optional features by *function group*.

To view the features enabled with your license:

1. Select *Setup* | *Module Setup* and click on the *General* tab.
2. Click on the *Menus…* to open the Menu Customization Dialogue box.

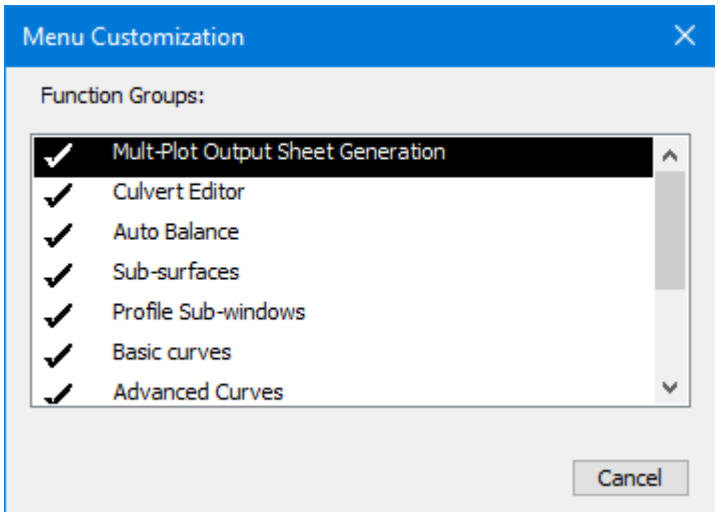

Figure 1-2: Function Groups Displayed in the *Menu Customisation* Dialogue

Note: Specific function groups are required to do certain 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.

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, dialogue 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.

Additional help is available through the Softree Knowledge Base:

https://www.support.softree.com/knowledge-base

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 *Setup |Setup Module Setup | Units* tab | Units: *Imperial (ft)*. 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.

Screen Layouts

Screen layouts are small files that save display options (window positions, labels, scales etc). Many of the examples in this training manual include a step to retrieve a screen layout; this change provides multiple view options in one quick step.

The *screen layout* drop-down control can be found in the Standard toolbar in all modules (figure below), *View | Screen Layout*:

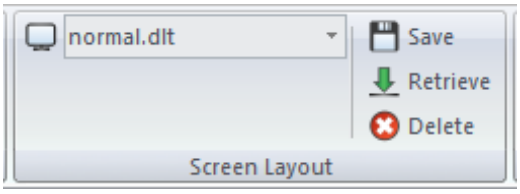

Figure 1-3: Accessing *Screen Layouts* Group

With the drop-down expanded, you can:

- <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 *Save screen layout* button 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 *Retrieve screen layout* button allows you to open a screen layout file anywhere including those in the *Custom*, *Training* or *Softree* folders.

- The *Delete screen layout* button opens up the screen layout folder where you can multiple layouts to delete.

- You can change the *Softree* 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).

Note: Screen layouts were updated in Version 9. Softree recommends 'updating' any legacy user screen layouts to update their behavior. Version 9 layouts work better when moved between monitors of differing screen resolutions.

To 'update' your screen layouts:

If your legacy screen layout contains multi-plot information, please open your legacy screen layout in the multi-plot window first:

Select *Multi-Plot* tab | *Add New* ▼ | *Retrieve Other Layout.* Select ***Multi-Plot Old Screen Layout (.dlt)*** from the file type drop-down in the *Retrieve Screen Layout Dialog*. Select your legacy layout. Once open, press *Save Chapter* in the Multi-Plot ribbon.

Conventions

The following conventions are used throughout the manual:

- Menu functions are delimited by a line "|". File | Open means to click on Location File button in the corner of the menu bar and then select Open from the drop-down menu. Dialogue 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

Softree software solutions are sold as modular products. Depending on the product you have purchased, it could include up to three *modules*:

1. Survey/Map
2. Terrain
3. Location

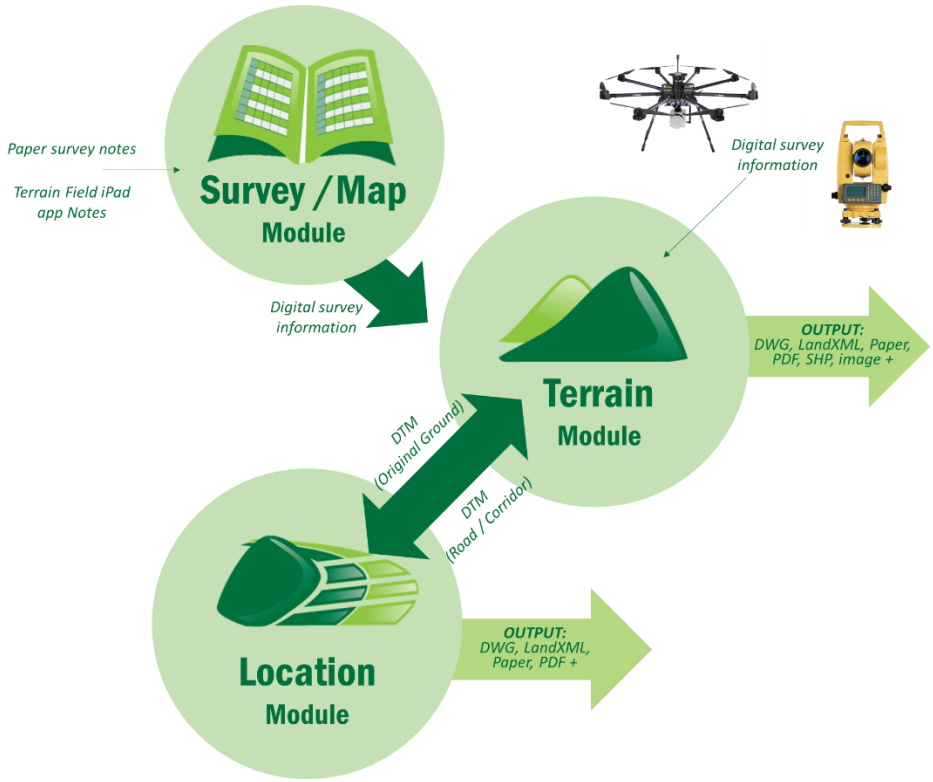

Figure 2-1: Relationship Between the *Modules*

Each of the modules can be started from the Windows Start menu, a desktop shortcut or from the *Setup* tab within either of the other 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 ground 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.

Typical RoadENG Work Flow for Designing a Road

1. The Terrain module is used to import and verify survey data of existing conditions. Possible data sources include total station (XYZ files), LiDAR, or GIS maps (shape, dwg, dgn etc.).
2. Using the Terrain module, break-lines and other linear features are identified and connected. A TIN (Triangular Irregular Network) surface representing original ground (OG) is created. The resulting linework and TIN surface is saved in a *.TERX file. NOTE: it may be useful to create several terrain files (e.g. one with the TIN model and one with planimetric linework).
3. A new design is created in the Location module, based on an OG surface (.TERX file from step 2).
4. The road cross section is created or adjusted using the *Template Editor*.
5. A horizontal alignment is created or adjusted using the mouse or explicitly in the *Horizontal Alignment Panel*.
6. A vertical alignment is created or adjusted using the mouse or explicitly in the *Vertical Alignment Panel*. Vertical optimization (Softree Optimal) can also be used in this step.
7. Steps 4-6 are repeated until the designer is satisfied with the result. In addition to *Plan*, *Profile* and *Cross Section* views, various reporting tools provide the designer with feedback. This includes volumes, mass haul diagram, and cost reporting.
8. Construction documentation is prepared using the Multi-Plot window. This documentation is printed directly or exported to CAD (*.dwg).
9. LandXML or ASCII files can be saved for construction staking.

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.

Note: section for file install folders (<RoadEngCivil> and <Defaults and Layouts>).

A Typical Data File

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

1	329591.7666	2195715.037	1172.736	SPOT
2	329570.0566	2195516.997	1158.295	PP
3	329625.9166	2195555.797	1159.534	SPOT
4	329573.4966	2195594.317	1161.31	SPOT
5	329552.9966	2195554.887	1160.682	SPOT
6	329561.9466	2195602.537	1164.661	SPOT
7	329531.5866	2195563.567	1166.9	SPOT
8	329527.9066	2195628.777	1177.279	SPOT
9	329500.6266	2195578.507	1177.822	SPOT
10	329482.4666	2195641.327	1190.244	SPOT
11	329456.7666	2195598.247	1192.141	SPOT
12	329433.7266	2195654.027	1204.384	SPOT
13	329407.6066	2195614.587	1206.786	SPOT
14	329396.5866	2195673.697	1216.893	SPOT
15	329374.2266	2195630.877	1218.22	SPOT
16	329347.4766	2195697.547	1231.632	SPOT
17	329321.3566	2195653.237	1235.406	SPOT
18	329296.9066	2195704.397	1242.378	SPOT
19	329276.1266	2195665.097	1244.316	SPOT
20	329247.7166	2195711.457	1248.812	SPOT

Figure 3-1: Excerpt from *Survey1.txt*

Setting Up an Import Format

1. Open the Terrain Module.
2. *Setup* | *Module Setup* | *Units* tab, Units: *Imperial (ft)*. The import software cannot detect units from the information in an ASCII file.
3. Click on the *Import* tab | *Open…* button. Browse to find the import options file <SettingsLayouts>\Training**training Normal.iop**. Press *Open* to read the file.

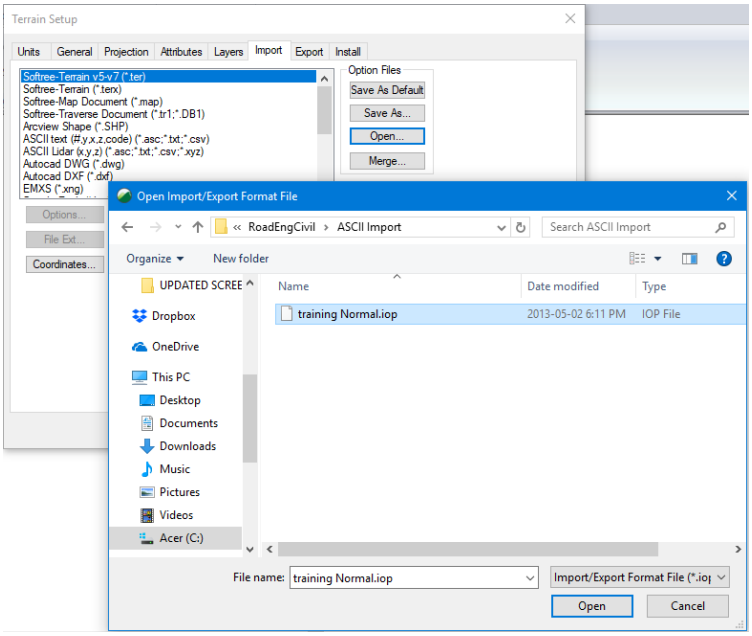

Figure 3-2: Opening an Import/Export Format File from Terrain Setup

Note: .iop files contains the format for types of files that can be imported into Terrain. This format is necessary for importing files such as data from total stations, LIDAR, and so on. The *Open…* button **replaces** all the.iop formats in the list of the Terrain Setup Import tab and the *Merge…* button **adds to** the list of all the importable file formats.

4. Now will create a new ASCII import format. Select the format called "*ASCII (x,y,z,code)*", then press *Add…* button to open the Define New File Format Options dialogue as shown in the figure below.

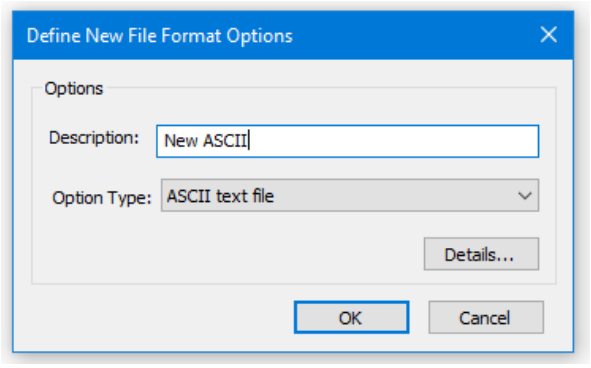

Figure 3-3: Define New File Format Options Dialogue Box

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

5. Within the open dialogue box type ***"New ASCII"*** in the Description field and then click on *Details…* button to open the next 'Import ASCII Options Dialogue' box shown below.

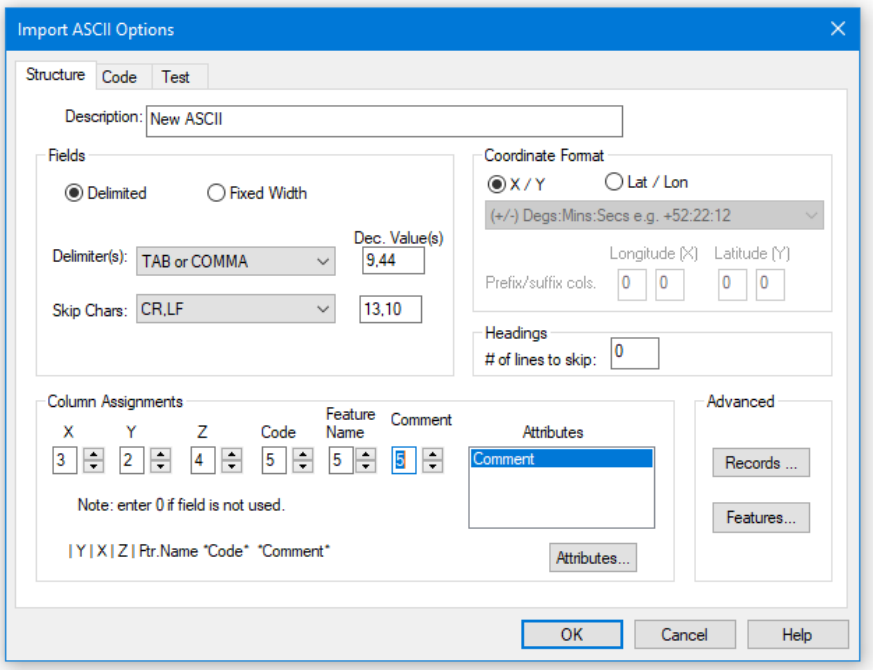

Figure 3-4: *Import ASCII Options* Dialogue Box – Structure Tab

The *Import ASCII Options* dialogue 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 dialogue box are available by pressing <***F1***>.

6. Change the Column Assignments in the dialogue box to match the figure above (X=**3**, Y=**2**, Z=**4**, Code=**5**, Feature Name=**5** and Comment=**5**). 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.

7. On the top of the *Import ASCII Options* dialogue box with the *Structure* tab selected. Press the *Features…* button in the Advanced section on the lower right. Ensure *Identify Features by Code (recommended)* is selected. The dialogue shown in the figure next page.

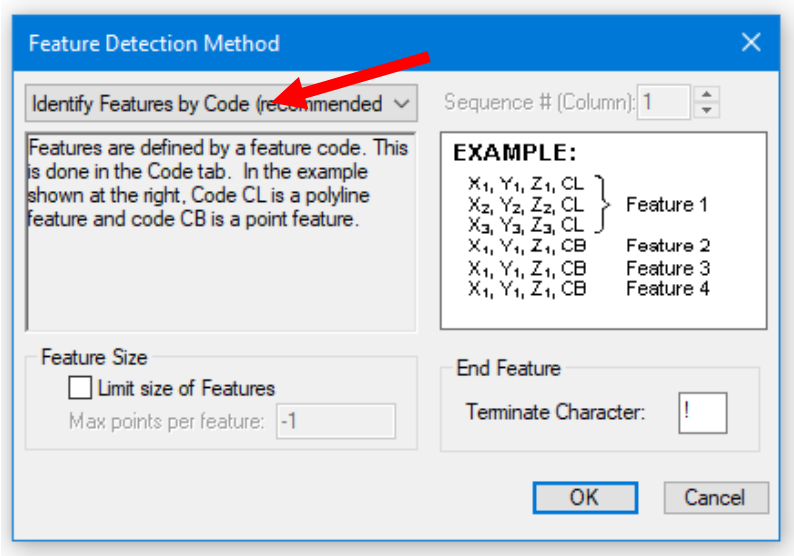

Figure 3-5: *Feature Detection Method* Dialogue Box

8. The dialogue box above 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 dialogue 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.

9. Press *OK* to exit the *Feature Detection Method* dialogue box.

10. Within the existing dialogue box and select the *Code* tab (figure next page). Here you can assign properties, symbols and line-types to the incoming points. For example, when importing survey data you may can to connect center line or edge of road points.

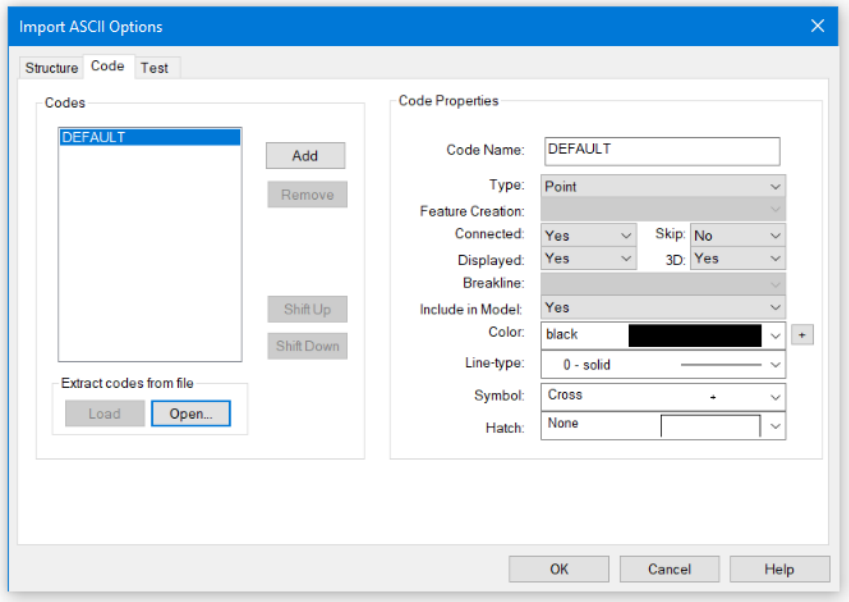

Figure 3-6: Import ASCII Options Dialogue – Code tab

11. Within the *Import ASCII Options* dialogue box select the *Code* tab. Change your default code properties to match those shown in the above figure; 3D points with a black cross symbol.

12. Press *Open…*, within the Import ASCII Options dialogue and select <RoadEngCivil>\ASCII Import**survey1.txt**. This will extract all the codes found in the file.

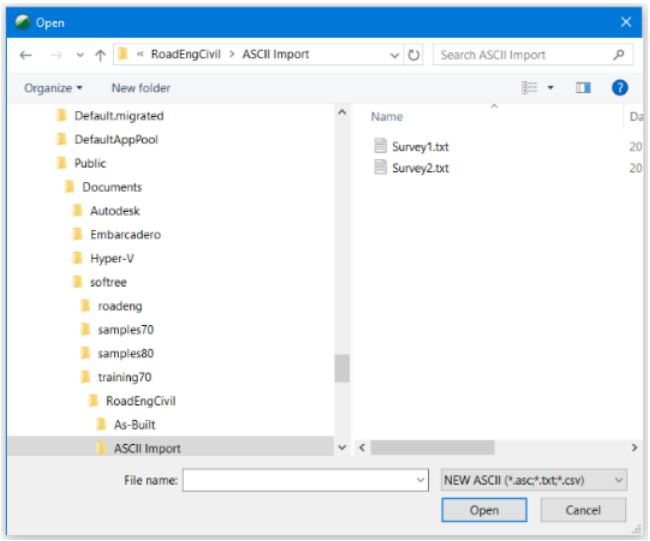

Figure 3-7: Opening file Survey1.txt

13. Select *CONTROL* code found in the codes list. Note that the options initially are the same as *DEFAULT*. Make the following changes:

    - *Color*: **navy**
    - *Symbol*: **Circle /w Cross**

14. Select *EP* (Edge Pavement) in the code list and type in ***** beside *EP*, eg. '*EP**', in the Code Name. The "*" is a wild card – any code starting with "EP" will fall into this category. Make the following changes:
    - *Type*: **Polyline**
    - *Feature Creation*: **Connect All by Code**
    - *Breakline*: **Yes**
    - *Color*: **blue**
    - *Symbol*: **None**

Points with the EP code will be connected (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.

15. Select code name *CLP* (Center Line Pavement) in the code list. Make the following changes:
    - *Type:* **Polyline**
    - *Feature Creation:* **Connect All**
    - *Breakline:* **Yes**
    - *Color:* **red**
    - *Line-type:* **3-dash-dot**

16. To test the specification, go to the *Test* tab (see Figure 3-8):

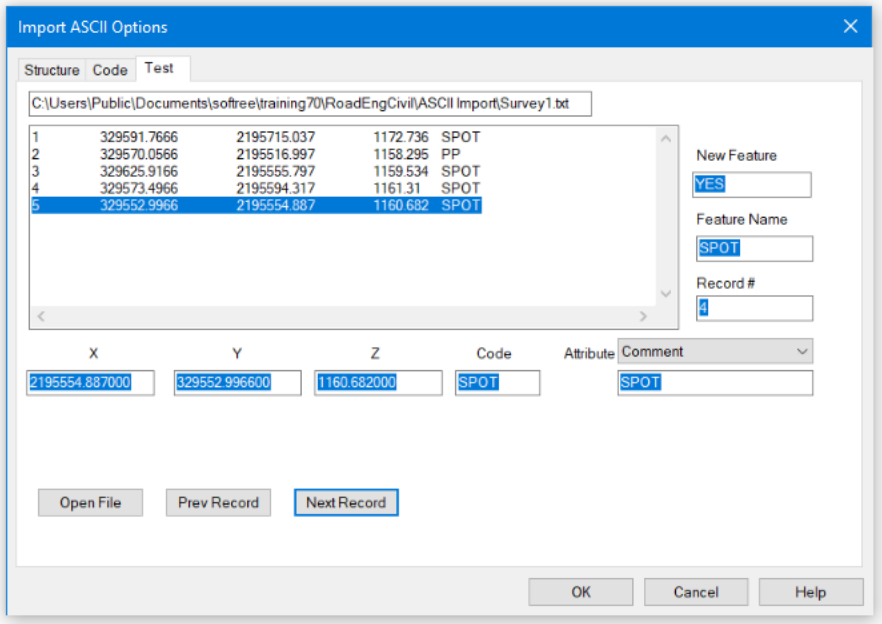

Figure 3-8: Import ASCII Options Dialogue – *Test* tab

17. Click Open File and open <RoadEngCivil>\ASCII Import**Survey1.txt**.
18. Press *Next Record* several times. At the bottom of the dialogue 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.
19. When satisfied, press *OK* to return to the Terrain Setup dialogue box.

20. To save the new import specifications for future use *Setup* | *Module Setup* | *Import* tab | *Save As…* 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 tutorial or training, otherwise save as **new training.iop** or press *Cancel* to avoid changing your defaults.

21. Press *OK* to close the Terrain Setup dialogue box. Now we'll use the import format we've created to open the survey data file.

22. *File* | *Open*. Change dropdown menu in lower right-hand corner to *New ASCII (*.asc, *.txt, *.csv)* (at the bottom of the list). Open <RoadEngCivil>\ASCII Import**Survey1.txt**. You will be presented with the Import Options dialogue box to allow last minute changes. Press *OK* to import the file.

23. Softree Warning appears: "Incoming coordinate system and units are undefined. OK to continue without conversion?" Click *Continue*.

24. Select the *View*| *Screen Layouts* | select **training Normal.ilt** from the dropdown menu. This will set up your options and windows to look like the Figure 3-9.

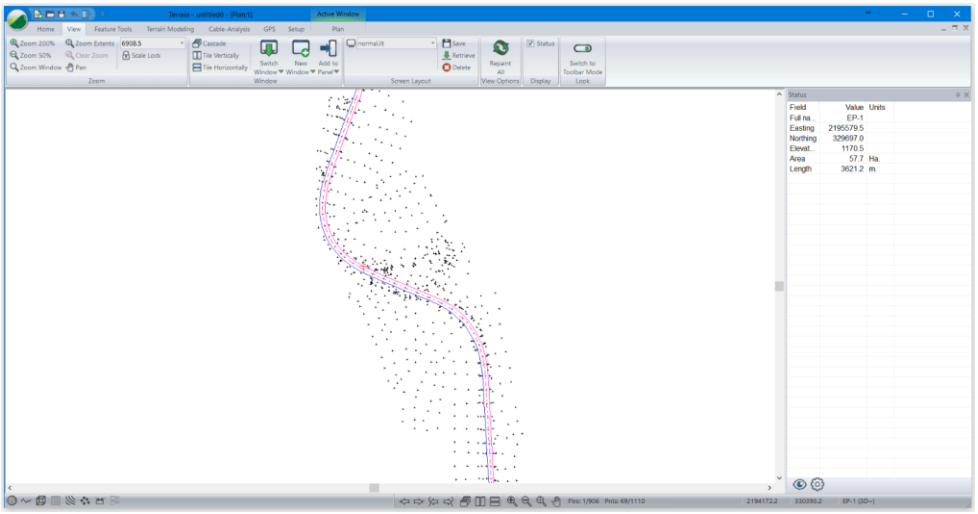

Figure 3-9: Plan Window after Importing *Survey1.txt*.

Note: The right **EP** feature is selected (note the properties displayed in the status window). Also 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.

Turn on the feature labels:

25. <Right-click> in the Plan window | Select Feature(s)>All, <right-click> | select *Modify Selected Feature(s)* | *Labels…*

26. <Double-click> on '*Comments (at feature points)*' and '*Feature Name*'. Press *OK*.

To reduce the size of the labels:

27. Zoom in by scrolling with the mouse wheel until the label font size is smaller and readable.

28. Press the *Scale Lock* . Now *zoom out* by rolling the wheel on the mouse. The labels will remain the size of what they were when they were locked.

We will now open the same file, with more point codes defined:

29. *File* | *Open*. Change Files of type to *ASCII 2 (#,y,x,z,code)*.
30. Open <RoadEngCivil>\ASCII Import**Survey1.txt**. When prompted to save changes, choose *No*.
31. This will open the *Import Options* dialogue; click on the *Code* tab to see the extra codes defined – no changes are required. Press *OK* to import the file. Softree Warning message hit *Continue*. Your options and windows to look like Figure 3-10.

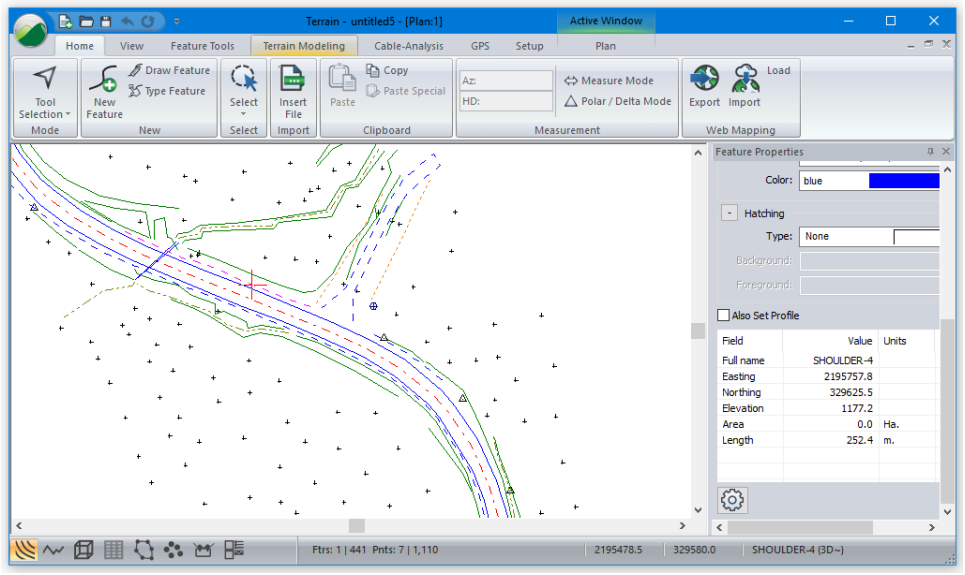

Figure 3-10: *Survey1.txt* Imported with More Point Codes Defined

32. You may wish to select features with the mouse to see what properties are displayed in the Status area. The status panel on the right hand of the display. To add more attributes, press the button at the bottom of the status window then press *Add/Remove…* button. Select the features you would like to add or remove.
33. *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) 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.
2. *File* | *Open* <RoadEngCivil>\DTM**Topo 1.terx**.

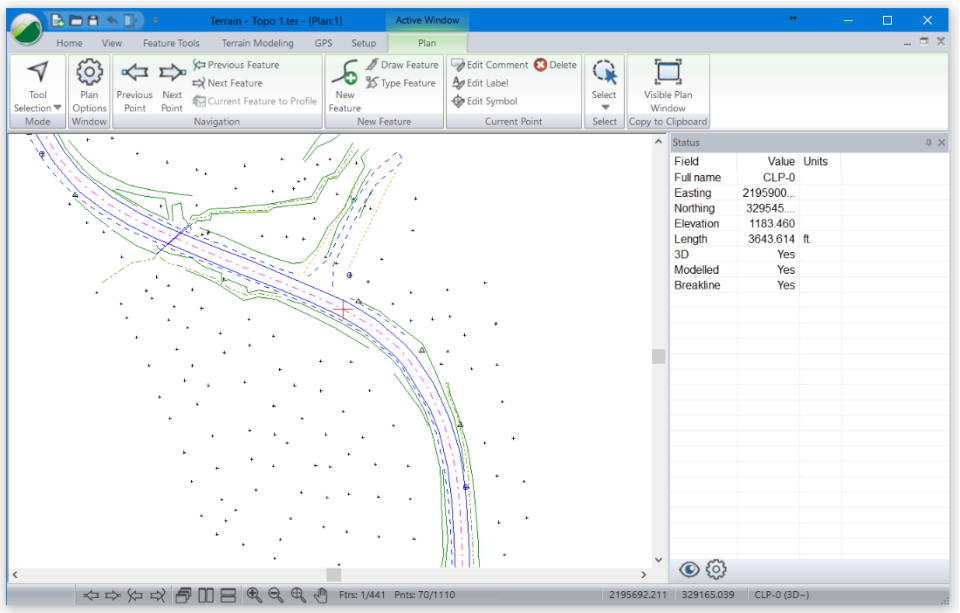

Figure 4-1: Terrain file *Topo 1.terx*

3. *Terrain Modeling* | *Generate TIN*. This opens the *Terrain Calculation* dialogue box (figure below).

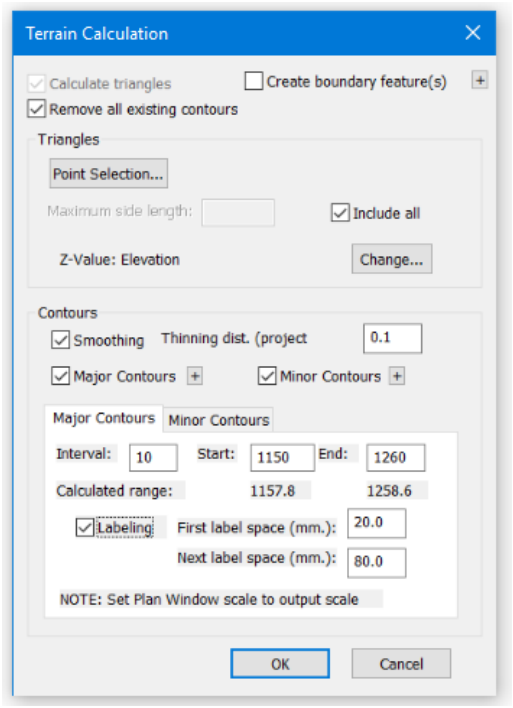

Figure 4-2: Digital *Terrain Calculation* Dialogue Box with Both Major and Minor Contours Enabled.

Contour Specification

4. Ensure the *Labeling* box is checked in the *Major Contours* tabs. Clicking on the ⊞ button adjacent to the Major or Minor Contours check boxes, allows you to 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.

5. Click on the *Major Contours* tab and set the Interval: **10** and check the *Labeling* box as shown above. You also need to an even number before the start of calculated range. Set the elevation Start: **1150**.

Click on the *Minor Contours* tab and set the Interval: **2** and make sure *Labeling* box is unchecked. You also need to set the Start elevation to be a multiple of 2, Start: **1150**. Press *OK* to generate both TIN and contours.

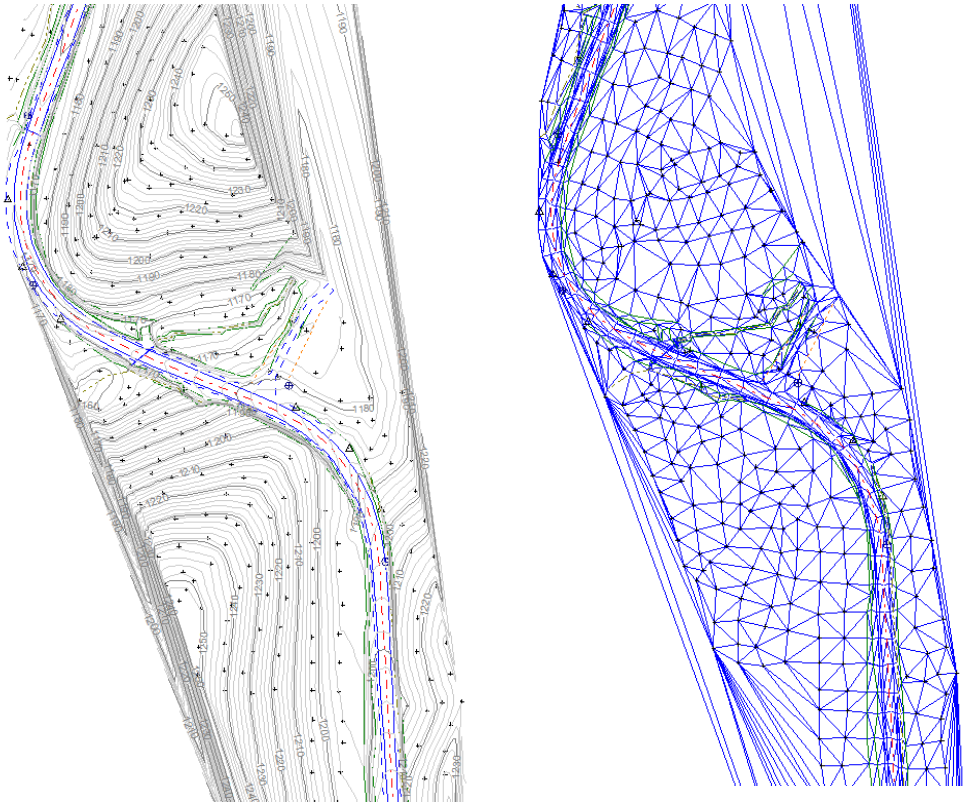

Figure 4-3: Contours Generated without Boundary or Length Limitation. *Underlying Triangles Shown* on Right

6. To display triangles in the model first delete the contours: *Terrain Modelling* | *Delete TIN* | select *Delete Contours* box | *OK*. Then, *<right-click> in* plan view select *Active Window (Plan) Option…*| *Surface* tab | check *Triangle outlines* box.

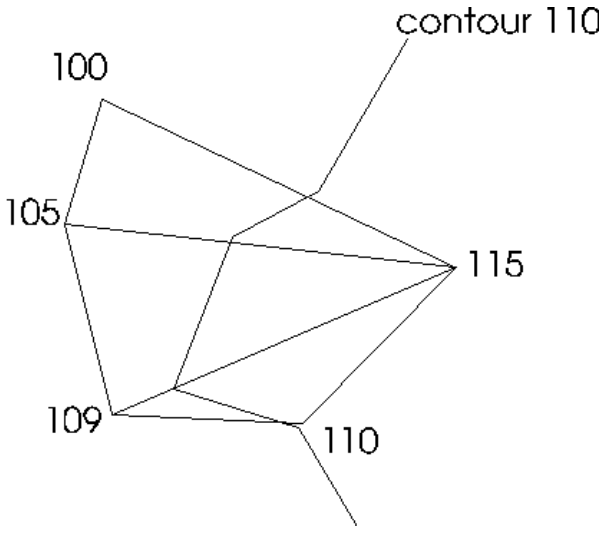

Figure 4-4: Contour Formation from *TIN model*. Elevations between known Elevation Points are Interpolated. Enabling Contour Smoothing causes Contours to be Less Angular

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. If triangles are still displayed, turn them off: <*Right-click*> in plan view | *Active Window (Plan) Options…* | *Surface* tab | uncheck *Triangle outlines* box | *OK*.
8. *Terrain Modeling* | *Generate TIN*. This re-opens the Terrain Calculation dialogue box.
9. Check *Calculate triangles* box, un-check *Include all* and set the Maximum side length: **150** (see Figure 4-5).

Note: Maximum side length of your triangles should be set as small as possible to create an accurate TIN. However, if you set this value too small, there will be holes in your model. As a rough guide LiDAR data could use a triangle length of 50 or smaller. A trial and error approach can work for this. Put in a number, generate the 3D model and if the model has "holes" in it, increase the side length.

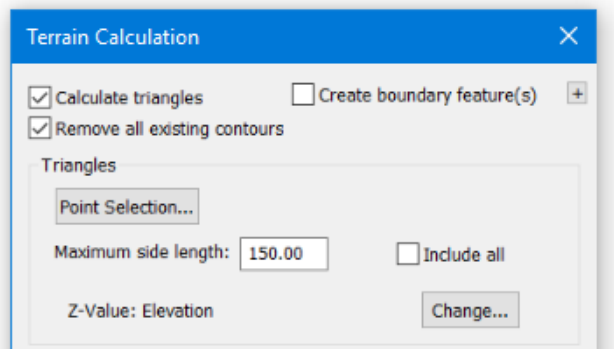

Figure 4-5: *Terrain Calculation* with Triangle Maximum Side Length Limited

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

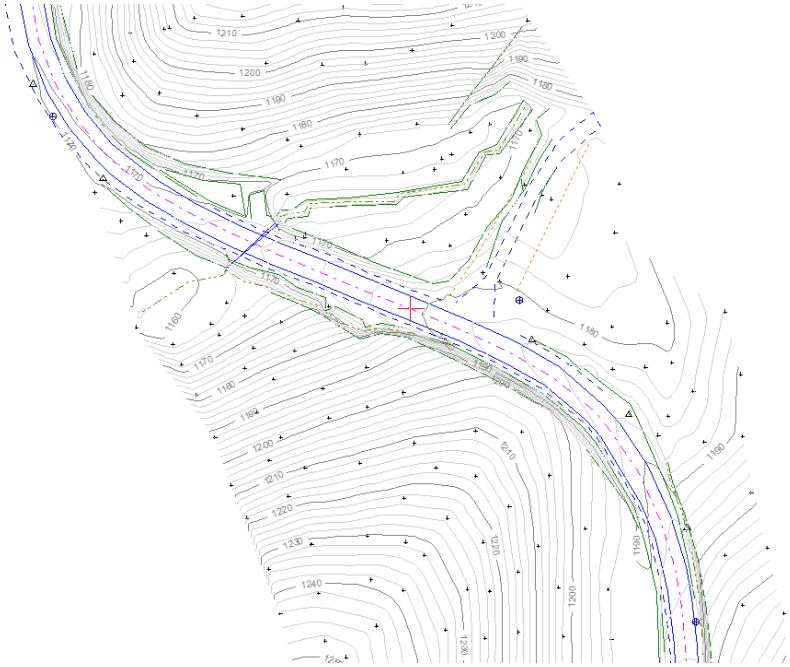

Figure 4-6: *Terrain Model* with Triangles Limited to 150 feet

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

11. *File* | *Save As*, this opens the file *Save-As* dialogue box. Notice that the default folder is the RoadEng Settings and Layouts folder. *Cancel* to close the dialogue box; we will not save this table.
12. *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.

Note: section for file install folders (<RoadEngCivil> and <Defaults and Layouts>).

1. Open the Terrain Module.
2. *File* | *Open* <RoadEngCivil>\DTM**Topo with issues.terx**

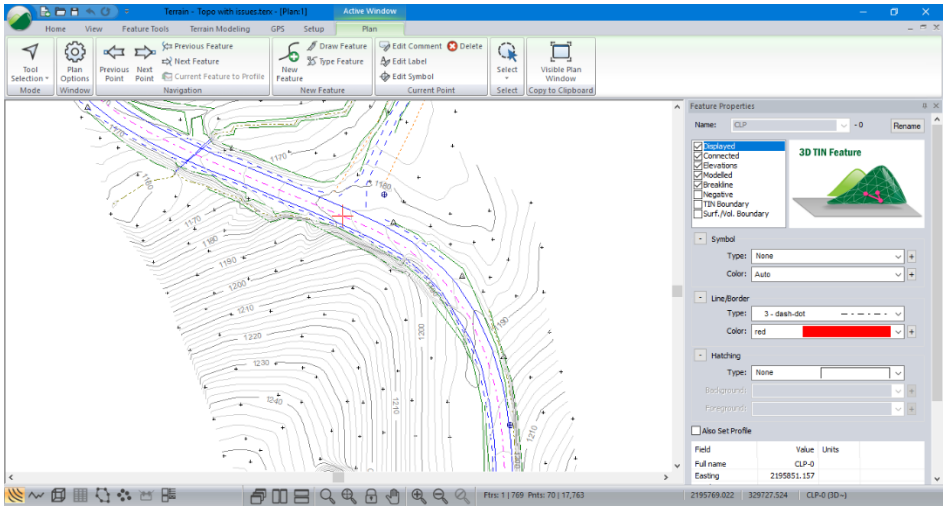

Figure 5-1: Terrain file *Topo with issues.terx*

Selecting Features with the Mouse

3. <*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 .
4. *Click* on the red center line *(CLP)* 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).

5. *Plan* | *Next Point* and *Previous Point* 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.

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

7. Hold the <*shift*> key and *click* on a feature. This allows you to add and remove features from a selection set.

Zooming and Panning

View | *Zoom* 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 behavior – check your control panel.

8. 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) Window Options…* | *General* tab | check *Scroll Bars*).

9. 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. If you use the *Zoom 200%* the center of the screen is always in the same location.
10. Similarly, use the middle roller mouse to zoom out by rolling towards you.
11. 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 thickness remain unchanged (Figure 5-2).

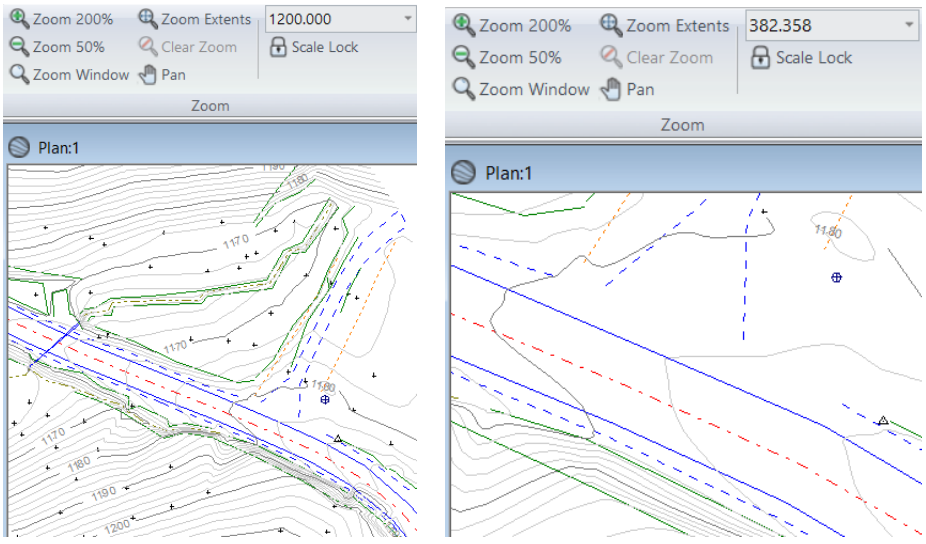

Figure 5-2: Before and After *Zoom Operation* with *Scale Un-locked*

12. Set the scale to **1200** in the toolbar (note this is a natural scale, the same as 1" = 100").

Note: The mouse roller will change the scale box in the tool bar once you have selected it. This can be confusing. See step 14 below.

13. Press the *Scale Lock*.
14. *Click* in the Plan window to move the mouse focus away from the scale bar, 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 5-3).

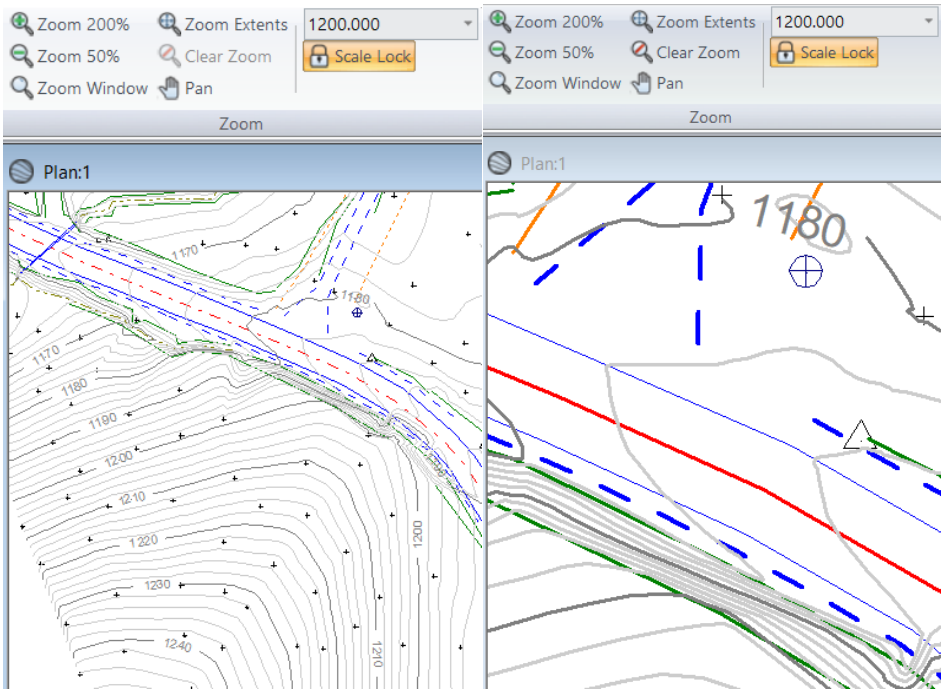

Figure 5-3: *Zoom Out* and *Zoom In*, Respectively, with Scale Locked

15. When you have finished experimenting with mouse feature selection and moving around, select menu *File* | *New*. Do not save changes.

6. 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.

Note: Refer to *Getting Started* section for file install folders (***<RoadEngCivil>*** and ***<Defaults and Layouts>***).

1. Open the Terrain Module.
2. *File* | *Open* <RoadEngCivil>\DTM**Topo with issues.terx**
3. Select *View* | *New Window* | *3D.* A 3D window will appear on your screen. The rendered surface should be visible; if it is not, press *Zoom Extents* in the *View* tab of the tool bar (this does not always work if your model contains stray points).
4. *View* | *Tile Vertically* to show Plan and 3D windows side by side (see figure below).

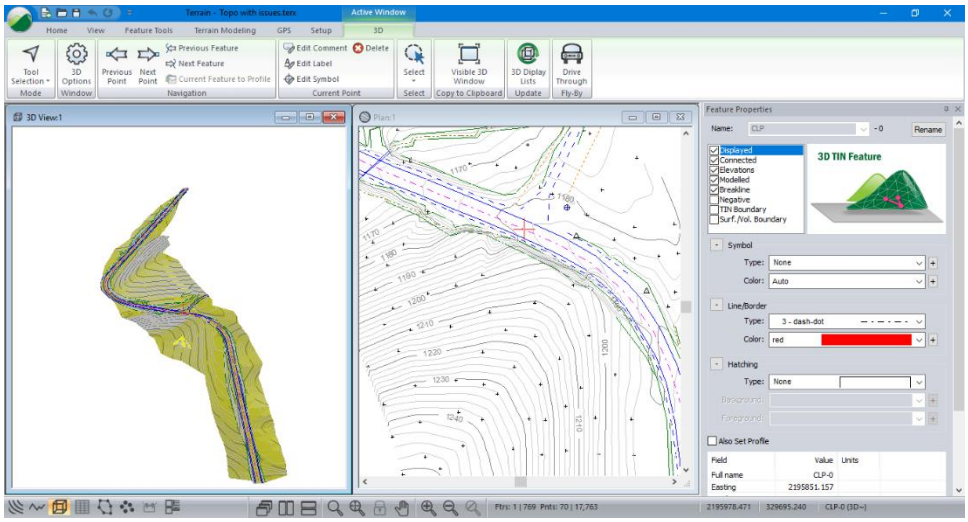

Figure 6-1: 3D and Plan Windows depicting file *Topo with issues.terx*

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*).

5. Use the zoom tools in the tool bar or the middle mouse wheel button to move around in the 3D window.

Rotating the 3D image

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

6. In the 3D window, *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.

3D Window Options

7. 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.
8. <*Right-click*> in the 3D window | *Active Window (3D) Options…* The dialogue box shown below will pop up.

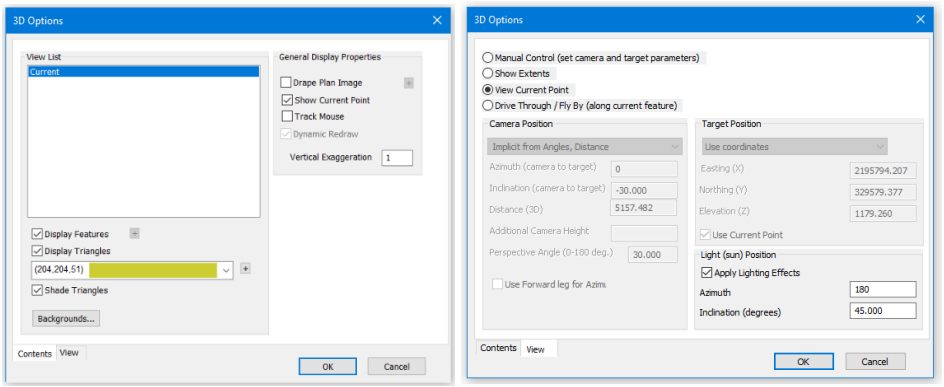

Figure 6-2: 3D Options Dialogue Box

The *3D Options* dialogue 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. Press <*F1*> or the help key for more information.

9. Select the *Contents* tab. Turn on the *Track Mouse* check box.
10. Select *View* tab; select *View Current Point*. This sets the target position to the current point.
11. Press *OK* to accept the change and close the dialogue box.
12. 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.

Notice that when you move the mouse over the 3D surface, there is a cursor tracking your position in the Plan window. Similarly, if you move the mouse over the Plan window, a line will appear in the 3D window indicating your position.

If you click on the surface (left mouse), the view will change to center that point, and if you click and drag (left mouse) the image will rotate about the point you first click on.

Note: If the 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*> |*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.

13. When you have finished experimenting with the 3D window, *File* | *New*. Do not save changes.

7. Finding / 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).

Note: Refer to *Getting Started* section for file install folders (***<RoadEngCivil>*** and ***<Defaults and Layouts>***).

1. Open the Terrain Module.
2. *File* | *Open* <RoadEngCivil>\DTM**Topo with issues.terx**.
3. *View* | *New Window* | *3D* from dropdown. A 3D window will appear on your screen.
4. Use menu *View* | *Tile Vertically* to show 3D and Plan windows side by side.

Removing a Bad Point From the Model

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

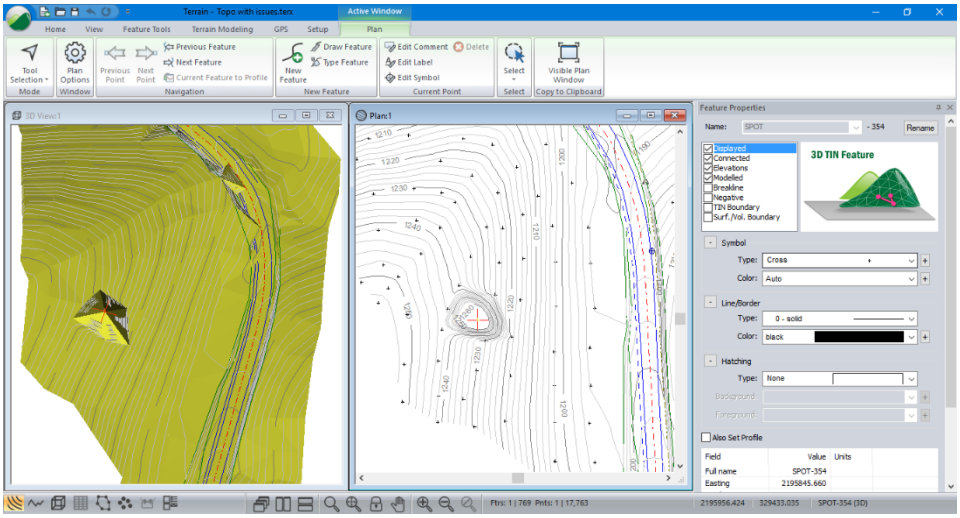

Figure 7-1: *Bad Elevation Point* Displayed in 3D, Plan and Feature Properties Windows

6. Select the bad point in the Plan window or the 3D 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 modeled 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.

7. In the *Feature Properties panel*, clear the *Modelled* property so the point feature will no longer be part of the model. Press *Apply*.
8. 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.

9. Select the *Terrain Modeling | Generate TIN* in the tool bar to open the Terrain Calculation dialogue box (see Creating a DTM with Contours exercise above). The settings for this dialogue box were configured when this file was created; you don't need to adjust anything.
10. Press *OK* to recalculate the DTM and the contours. Note that the anomalous spike in the model has disappeared.

Defining Breaklines

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

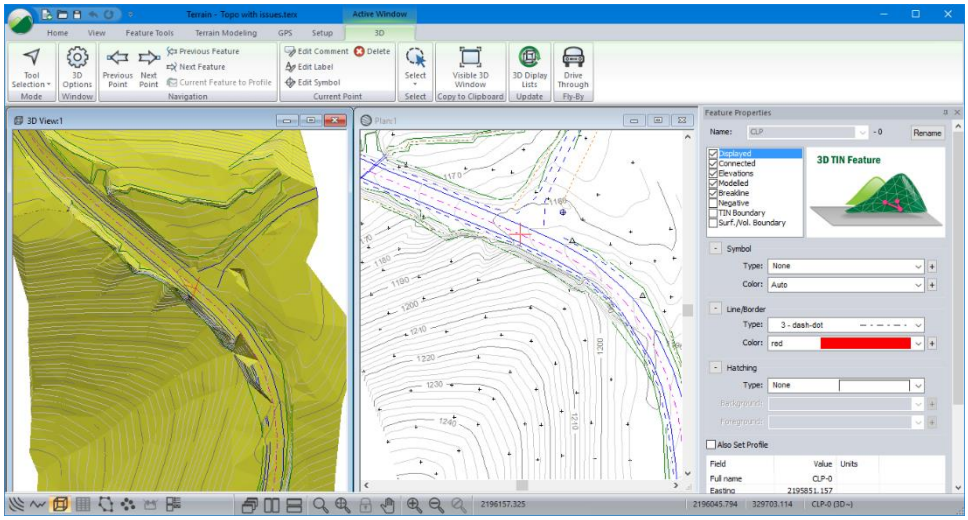

Figure 7-2: 3D and Plan Contours Depicting *Bad Triangles* Caused by Missing Breaklines.

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

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

12. Select the *EP feature*. Note that the properties the Status window indicate that this is NOT a breakline.
13. In the *Feature Properties* panel, turn *ON* the *Breakline property*. Press *Apply*.
14. Recalculate the Terrain Model: *Terrain Modeling* | *Generate TIN* (as in steps 9 and 10). Note that the Model looks a little better.
15. *Terrain Modeling* | *Delete TIN* | check *Delete Contours* box. This will make the following step easier.
16. Find other features that should be tagged as breakline *(EP, TOE, TOB)*, 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.

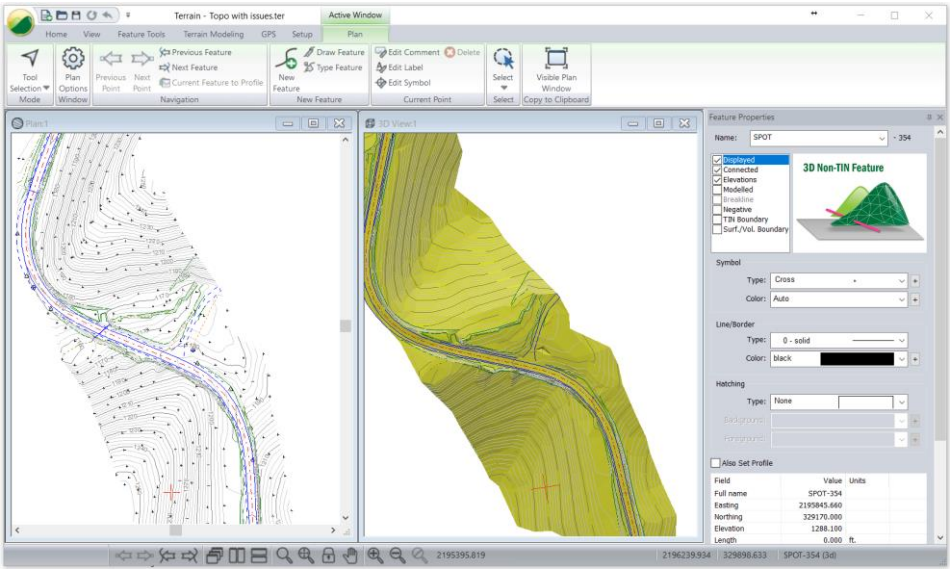

Figure 7-3: Model after Features made *Breaklines* and Model Re-Calculated

17. *File* | *New*. Do not save changes.

8. Creating Breaklines

We have seen in the Chapter, *Importing ASCII Survey Files*, 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.

Note: Refer to *Getting Started* section for file install folders (***<RoadEngCivil>*** and ***<Defaults and Layouts>***).

1. Open the Terrain Module.
2. *File* | *Open* <RoadEngCivil>\DTM**Topo no breaklines.terx**.

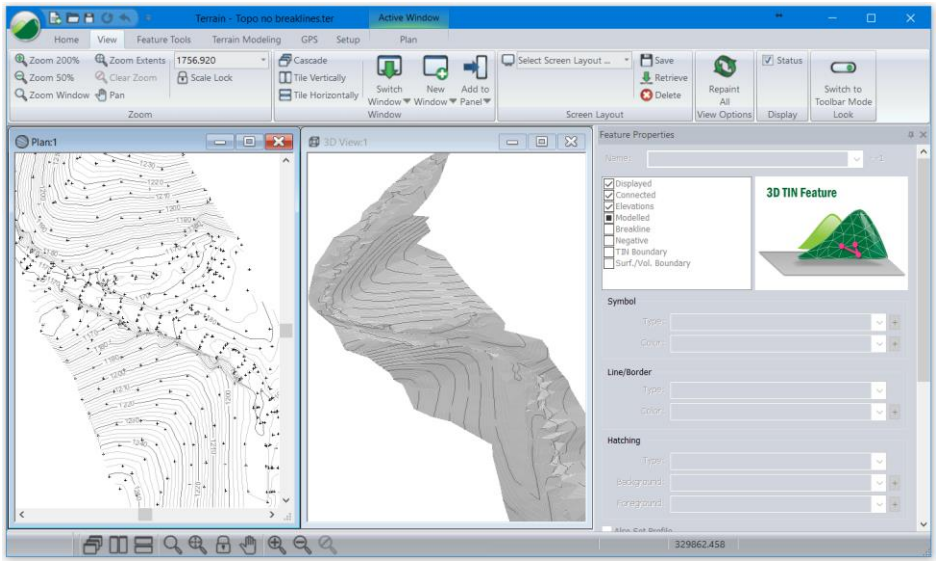

Figure 8-1: 3D and Plan Windows, File: *Topo no breaklines.terx*.

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.

Selecting Features by Name

3. 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* information available after you select a point (lower portion of *Feature Properties* panel).
4. *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”.

5. *<Right-click>…* | *Select Feature(s)* | *By Name…* Press the *Advanced…* button to open the dialogue box.

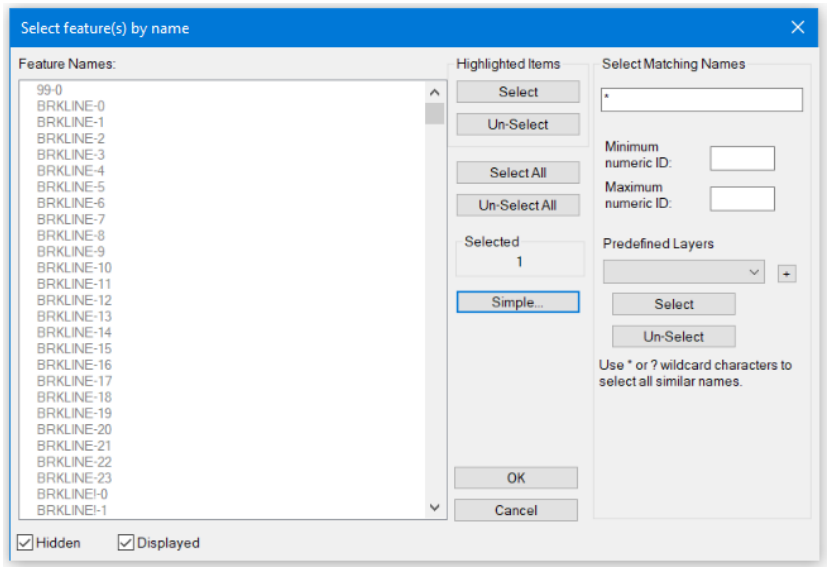

Figure 8-2: *Select Features by Name* Dialogue Box with the *Select Matching Names* area Visible

6. Press *Un-Select All* button in the center of the dialogue box.

Note: The *Select features by name* dialogue 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).

7. Press *Advanced…* button | type "**CLP**" at the top of the Select Matching Names area | press *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).

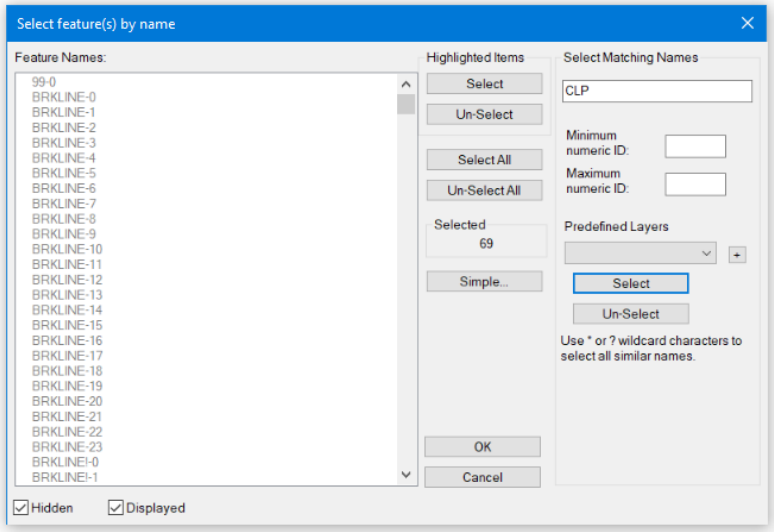

Figure 8-3: Select Features by Name Dialogue Box with CPL Selected

8. Press *OK* to accept the change and close the dialogue 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.

9. *Feature Tools* | *Join* or <*Ctrl-J*>, to connect all the CLP points into one polyline feature. When warned, "existing triangles will be cleared", respond *OK* button.

Modifying Feature Formatting

10. <Right-click> in plan view| *Modify Selected Feature(s)* | *Linetypes, Symbols* or <Ctrl-L>, to display the dialogue box below. Alternatively, you could use the *Feature Properties Panel*.

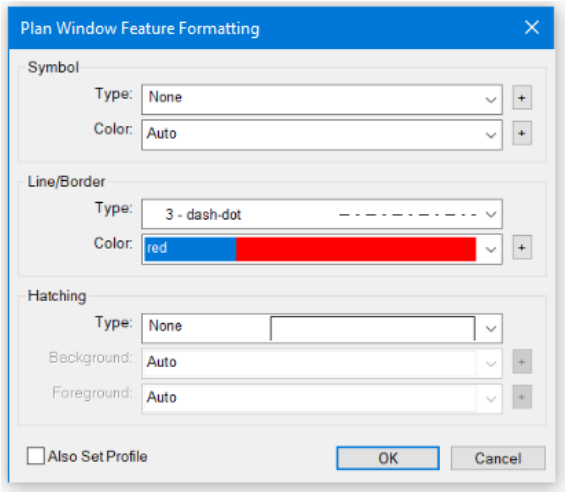

Figure 8-4: Feature Formatting Dialogue Box

11. Within the Plan Window Feature Formatting set Symbol Type: *None*, the Line/Border Type: *3-dash-dot* and the Color: *red* as shown in the figure above.
12. Press *OK* to accept the change and close the dialogue 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 *Feature Properties* panel to set the CLP feature as a *Breakline* (as in the Finding and Repairing DTM problems exercise above). Press *Apply*.
14. Now let's try the same process with the edge of pavement (*EP*) points.
15. As in step 5 above, use the Select features by name dialogue box to *select* all *EP* points.
16. Again use <*Ctrl-J*> to join them. The results are pictured below.

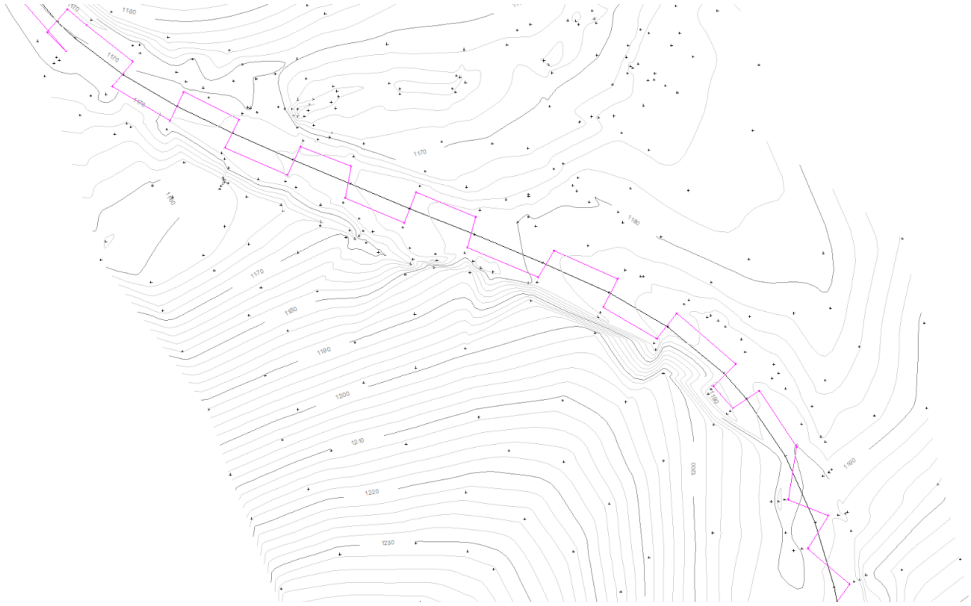

Figure 8-5: Pavement Edges Connected using the Join Function

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.

17. Use the undo button or <*Ctrl-Z*> to restore the loose points.
18. As in step 10 above, use the formatting dialogue box <*Ctrl-L*> to change the *EP* points to a distinctive *color* and *symbol* (as below).

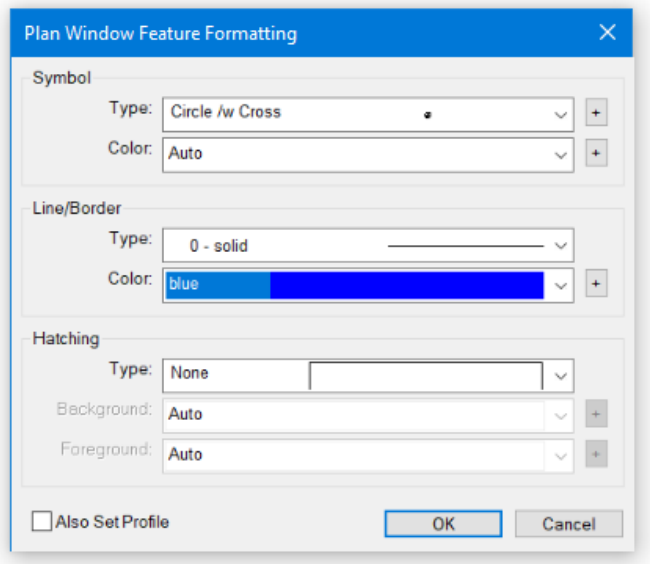

Figure 8-6: *Formatting* can Make it Easier to Locate Points of a Given Type

19. Select the *Terrain Modeling* | *Delete TIN*. Check the *Delete Contours box* and press *OK* button. This will make the following steps easier.

Creating a New Feature

20. *Home* | *New Feature*.

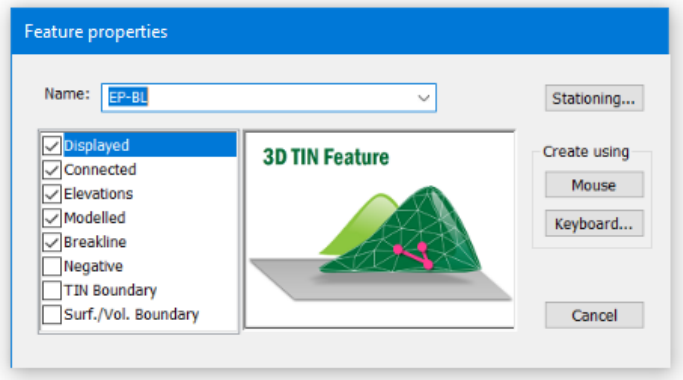

Figure 8-7: The *Feature Properties* Dialogue used to Prepare a *New Feature*

21. Change the *Name: EP-BL* and *check* the *Breakline* property box as well as the others shown in the figure above.
22. Within the Feature Properties dialogue box press *Mouse* button to close the dialogue box and create the new feature.
23. When you are prompted to define the *Elevation* value, just press *OK*; keep *Default elevation*: 100.0. 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.

24. 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. Your cursor changes to a cross.
25. *Move the cross* over an EP point the cursor changes  to indicate you are ready to snap. *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.

26. 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.
27. 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.
28. 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.
29. 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.
30. Stop when you have done enough points to get the hang of editing with the mouse. Make sure you have tried deleting and 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 *Home* tab| *Mode* group | *Tool Selection* button | *Edit/Insert Points with Mouse* from dropdown menu.

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.

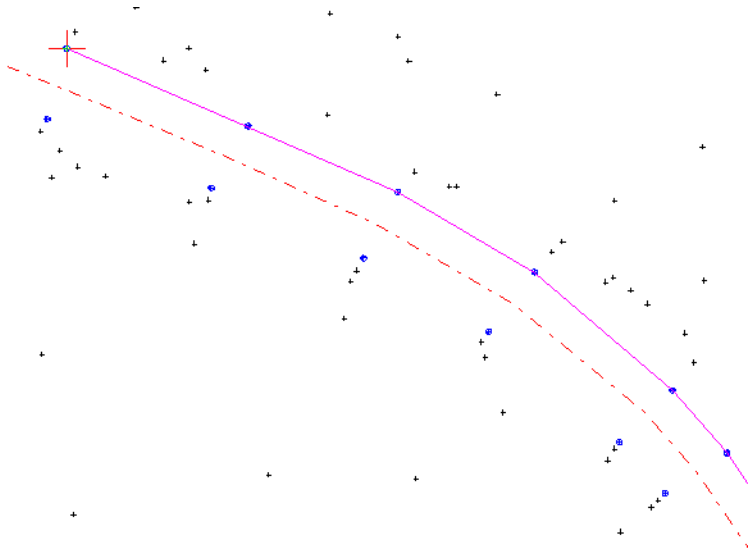

Figure 8-8: New EP *Breakline Feature*

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

31. Change back to selection mode : <*Right-click*> |select menu *Select with mouse*.
32. *Click* on one of the EP points to select it (the point will turn pink).
33. <*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.
34. Type <*Ctrl-J*> to join the two points. Now you have a two-point polyline.

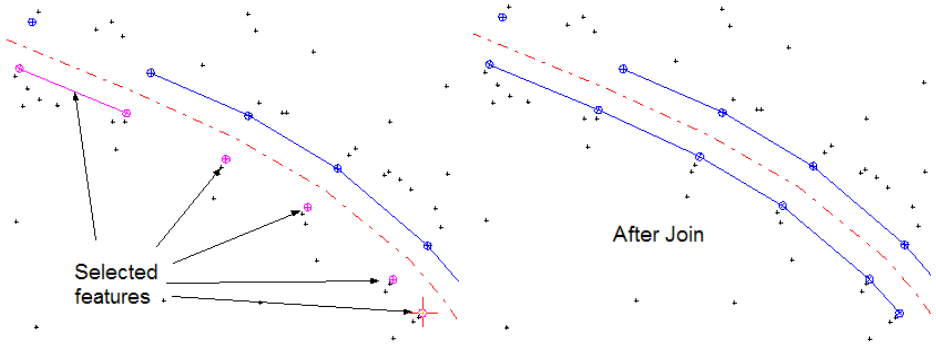

Figure 8-9: *Joining Points* a Few at a Time,
Select a Few Points (<*shift*>+ <*Right-click*>), then Join Them <*Ctrl-J*>

35. 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.
36. Type <*Ctrl-J*> to join them all into a bigger polyline.
37. 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.

38. *File* | *New*. Do not save changes.

9. Working with LiDAR

LiDAR (**Li**ght **D**etection **A**nd **R**anging) surveys produce very large amounts of relatively accurate three dimensional point data. The data includes points representing laser light scattered from the ground (bare earth), foliage, buildings, transmission lines and other objects. This data is usually broken into tiles, each containing a few million points.

Size and Accuracy Considerations

- The 32-bit version of RoadEng® is limited to approximately 5 million points. The 64-bit version of RoadEng® can handle more points depending on the speed of the user's CPU processor and amount RAM, 10 million points is reasonable.

- Interpolating the LiDAR into regular grid format is not recommended, because this creates points by interpolation (lost accuracy). For accuracy purposes, it is better to work with the raw data points.

- When importing LiDAR data, it is very important to group points together instead of making feature for every point. Features require a significant amount of memory (much more that a point) so it is best to store thousands of points per feature.

It is not uncommon to have data sets with hundreds of millions of points (well exceeding the recommended maximum of 10 million points). This limitation is generally not a problem for most corridor projects, if points outside the area of interest are thinned. Consider a relatively large road project say 20 kilometers (~ 12 miles). Assume that your LiDAR horizontal resolution is 1 meter (3 feet) and that you have identified a corridor that is 200 meters (~656 ft.) wide along a preliminary alignment. This yields about 4 million data points.

Importing LiDAR in ASCII format

Large data sets need to be loaded in such a way that they use the least amount of memory possible. In the next section, you will load a prepared LiDAR import format from a *.**IOP** (Input/Output Parameters) file.

Note: If your data is in LAS format, many of the steps in the next section are not required. However, the corridor thinning technique is required for both formats. LAS format is the preferred format for LiDAR, as it is compact and loads fast.

1. *Open* the *Terrain* module.
2. *File* | *Open* <RoadEngCivil>\LiDAR**Empty.terx**
3. *Setup* | *Module Setup* button. This opens a *Terrain Setup* dialogue box.
4. Select the *Import* tab.
5. Check if *LiDAR (x,y,z,code)* already existing in the dropdown menu. If that format in not present press *Merge…* button and browse to find the import options file. <RoadEngCivil>\LIDAR **\Lidar2.iop**. See figure below. IOP files are Import/Export File Formats were previously created.

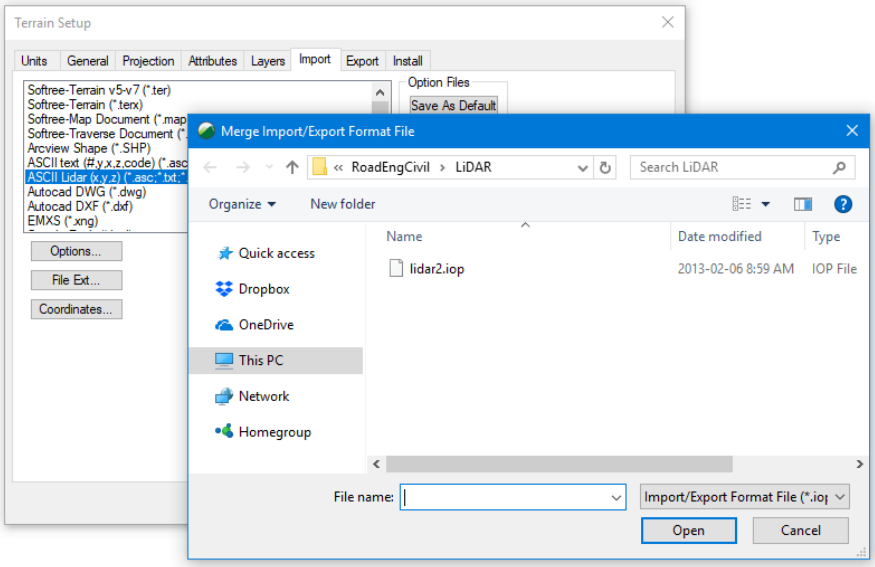

Figure 9-1: Changing *Import Options* by Opening an IOP File

6. Press *Open* (if it was not present in your list) and press *OK* to close the *Module Setup* dialogue box.

Setting up a Linear Corridor Feature

Now you will read in a proposed center line and later use it to create an area of interest.

7. *Home | Insert File*. Ensure your file type drop-down is set to *Shape (Arc) (*.shp)* – should be at the bottom of the list. Browse for file <RoadEngCivil>\LiDAR\ **ProposedAlignment.SHP**. Press *Ok.*

8. The *Import options* dialogue box below appears. Press *OK*.

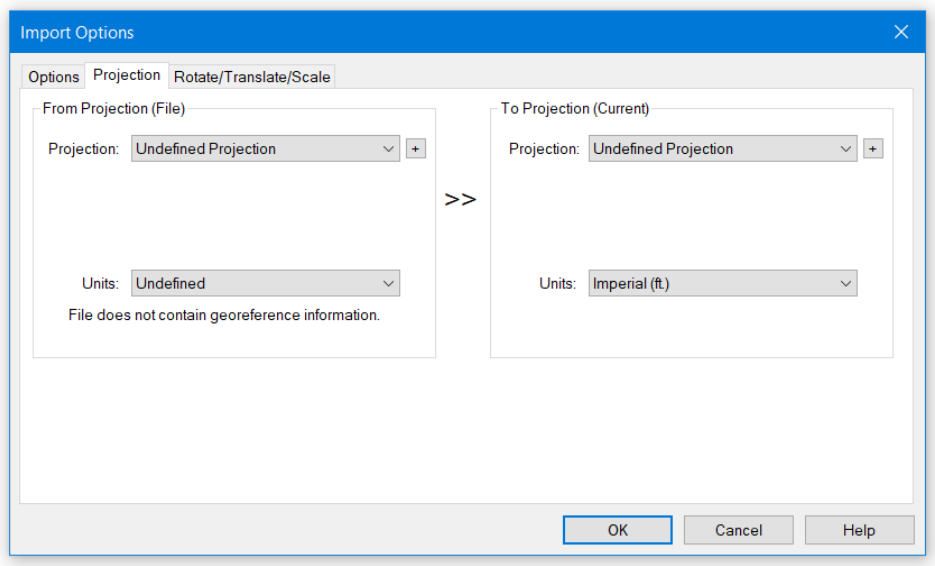

Figure 9-2: *Import Options* Dialogue Box

9. Softree Warning stating "Incoming coordinate system and unit are undefined. OK to continue without conversion?". *Enable* "Do not show this message again". Press *Continue* to load the proposed center line shown below. The reason for this is the coordinate system and units in **empty.terx** are correct.

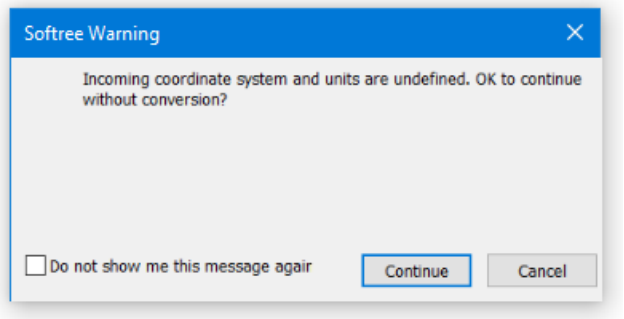

Figure 9-3: Softree Warning Message

Note: The Coordinate system undefined errors shown in the above steps will not happen is the Shape PRJ file is available.

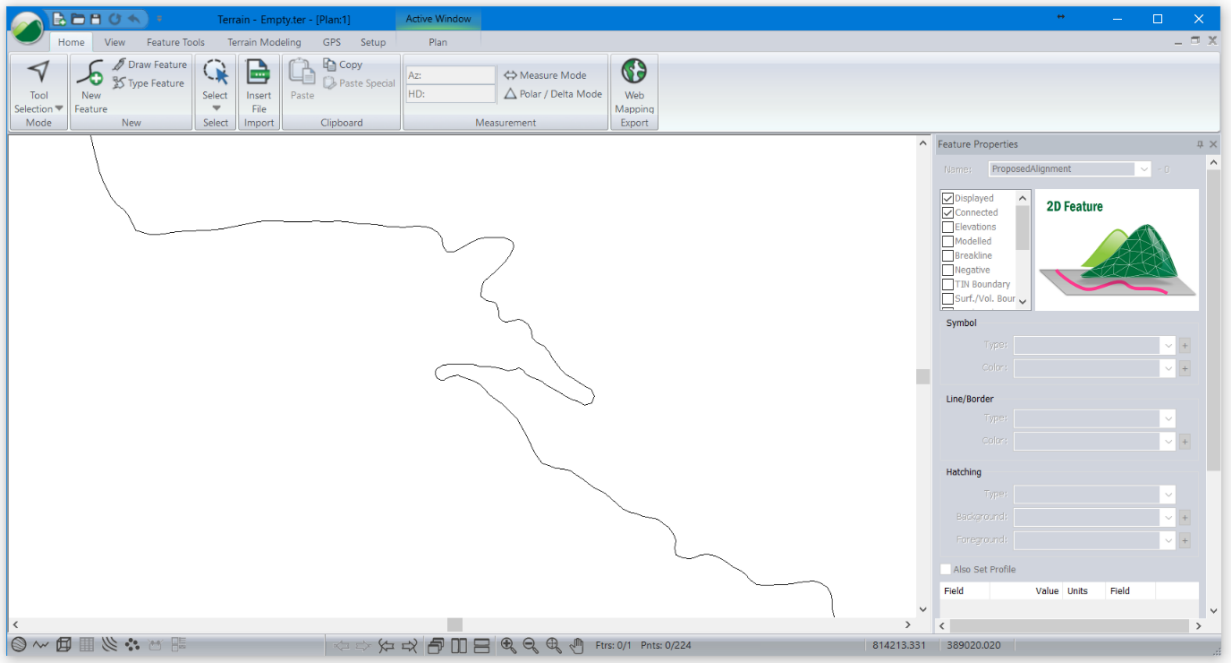

Figure 9-4: *Proposed Road* Center Line

Now that the road center line has been brought in, we can bring in the LiDAR data. This example only contains approximately 700,000 points to save download and file read time. This example use *.txt files but could be other file types. A common LiDAR file format it *.LAS.

In the following steps, we will read in the data at full resolution in the area of interest (AOI) and skip some points outside this area. In addition, we will follow some important guidelines to prevent slow draw times and memory overload.

Bring in the LiDAR data: *Home* | *Insert File* button. Set the *file type drop-down* to *ASCII Lidar (x,y,z)*. (This is the one at the bottom of the list.)

10. Select both *TXT* files included with this example. Press *Open*. **Tile_1E_BaldEarth_SP.txt**; **Tile_2E_BaldEarth_SP.txt**

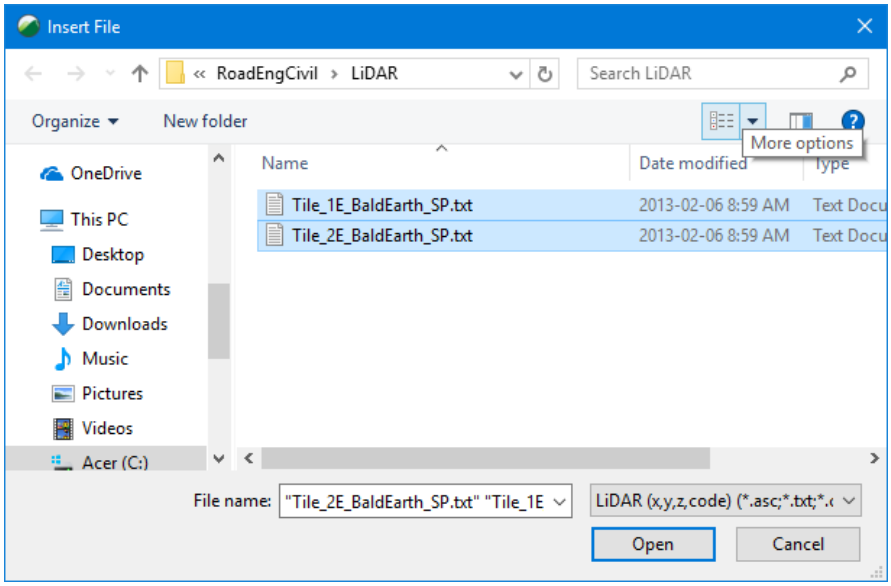

Figure 9-5: Inserting Multiple *LiDAR Files*

Note: Using *Home* | *Insert File* unlike *File* | *Open* that allows multiple files at once and does not clear existing features from your Terrain.

You will now be presented with the import options (in case you want to make last minute changes).

11. Click the *Test* tab then *Next Record* button a few times to see what the file looks like. Note that the *X, Y, Z* fields are showing the correct values (figure below). This indicates that the options set in the *Structure* tab are working correctly.

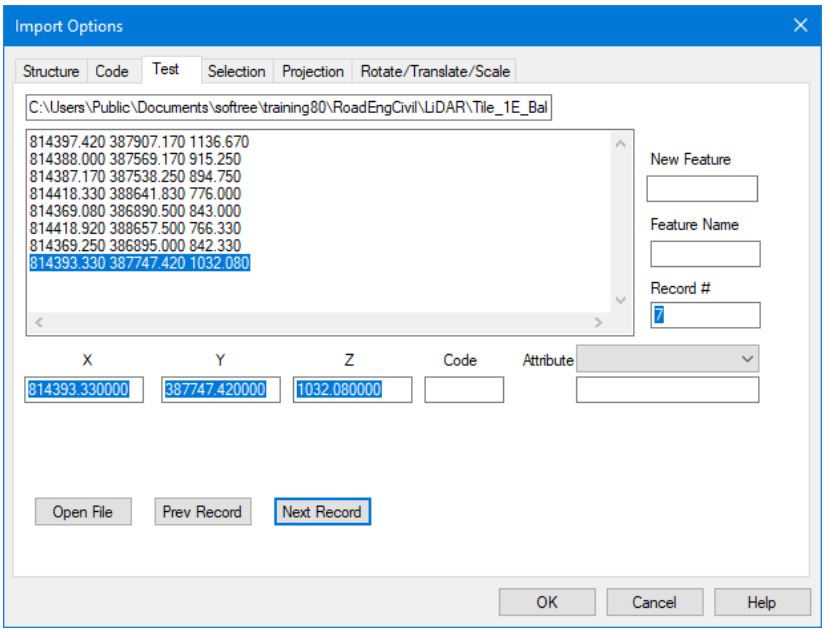

Figure 9-6: The *Test Tab* after Pressing the *Next Record* Button a Few Times.

Note: Memory Usage / Display Speed Rules & Guidelines

Other *Import Options* have been setup to avoid using more memory than necessary and to make the resulting Terrain display manageable. The following rules are necessary when importing large data sets:

A. Do not attach comments or other attributes to every point.

B. Do not allow very large numbers of points in features.

C. Do not make every point into a separate feature.

D. Do not attach symbols to every point.

E. Do not turn on labels (such as Elevation) that will display at every point.

If you use the standard LiDAR import options these guidelines will be taken care of for you.

12. Click on the *Structure* tab. Notice that there are no *Attributes* defined in the *Column Assignments* section (Rule A).

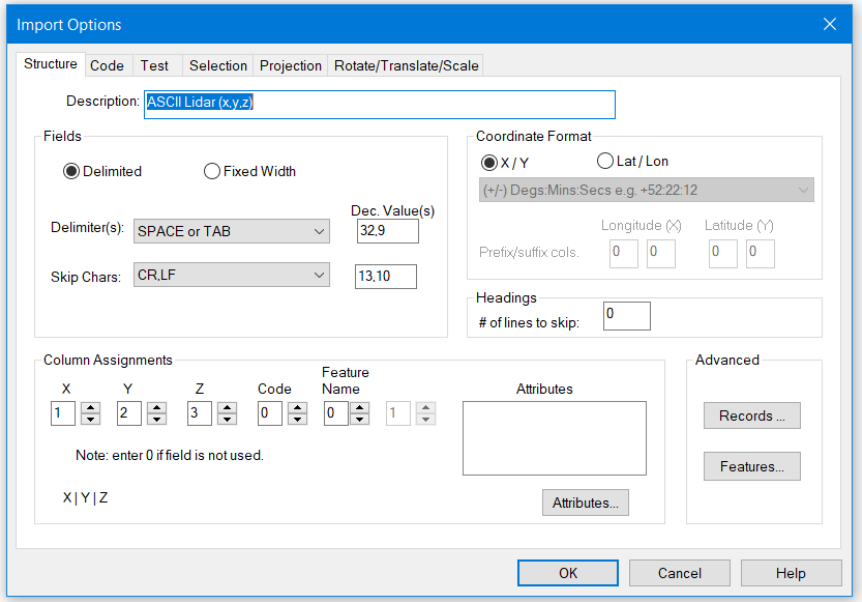

Figure 9-7: The *Structure Tab* defines the Location of the *X,Y,Z Coordinates*

13. Press the *Features…* button in the *Advanced* section (lower right).
    - Enable *Limit size of Features.*
    - Set Max points per feature: **1000**.
    - *OK*.

The reason for this is when LiDAR points are grouped in features the program works better.

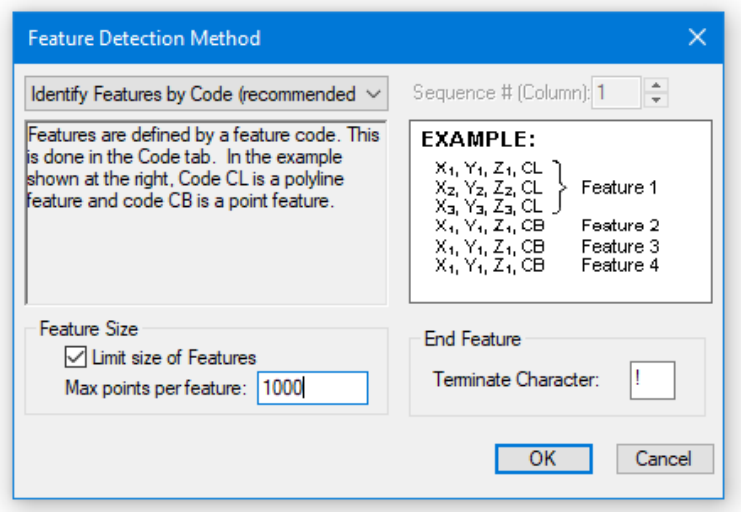

Figure 9-8: Feature Size Limited to *1000* Points

14. Select the *Code* tab. For the DEFAULT Code type ensure the following fields are set:
    - *Type*: **Polyline**
    - *Feature Creation*: **Connect All**
    - *Connected*: **No**

Also note that no symbol is defined (rule *D* above). In some cases, it makes sense to turn off the *Displayed* property, to speed draw time later.

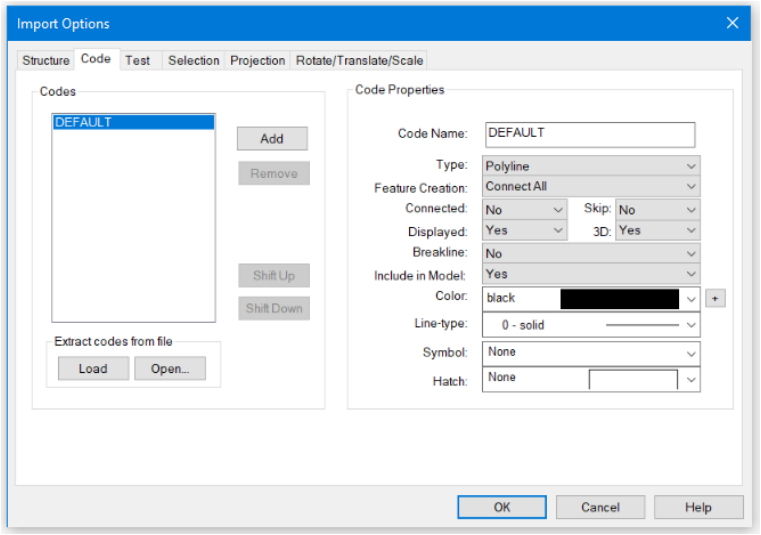

Figure 9-9: *Point Code Properties* suitable for Large Data Set Import

15. Click on the *Selection* tab to show the options below:

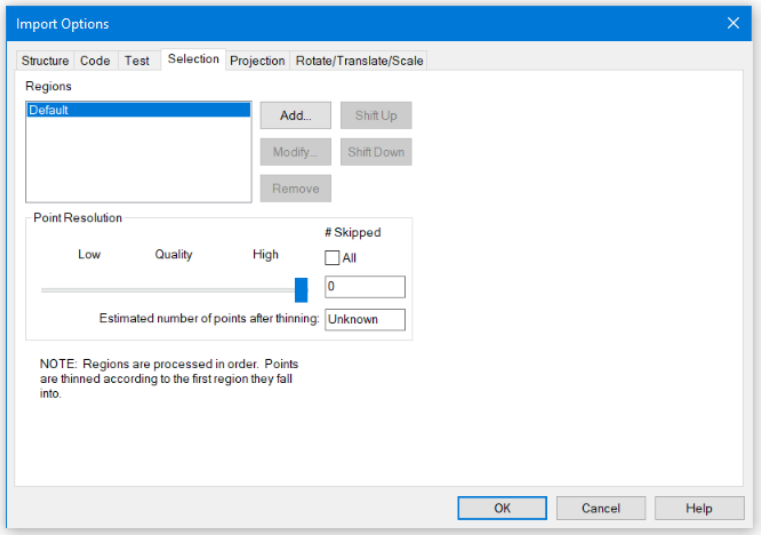

Figure 9-10: Import Options Dialogue Box with *Selection* Tab Selected

16. Press *Add…* button to open the Filtering Region options as shown below.

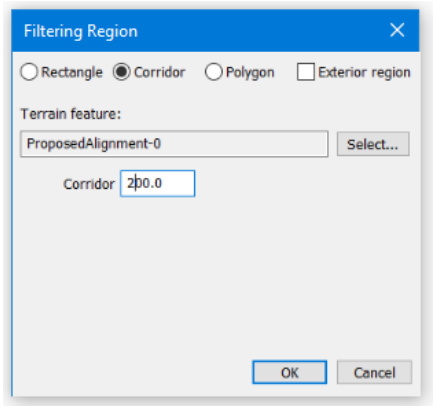

Figure 9-11: *Filtering Region* Dialogue Box Options

17. Set the *Corridor* button as shown above.
18. *Select…* button, <*double-click*> on the alignment feature: "*ProposedAlignment-0*" to select it. Press OK to return to the *Filtering Region* dialog. Set the *Corridor width* **200.0**. Press *OK*.

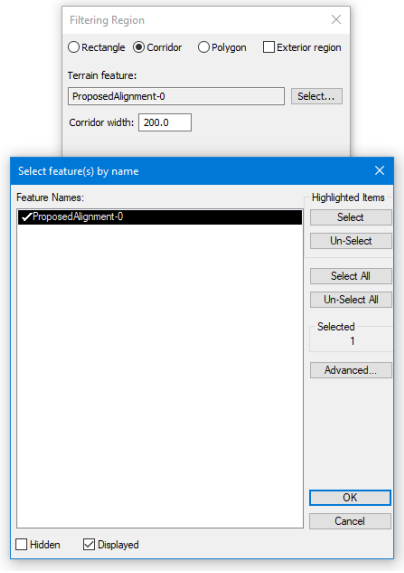

Figure 9-12: *Defining a Corridor* with a Linear Feature

19. *Click* on **Default** in the list and set the Point Resolution to **9** (refer to Figure 9-13).

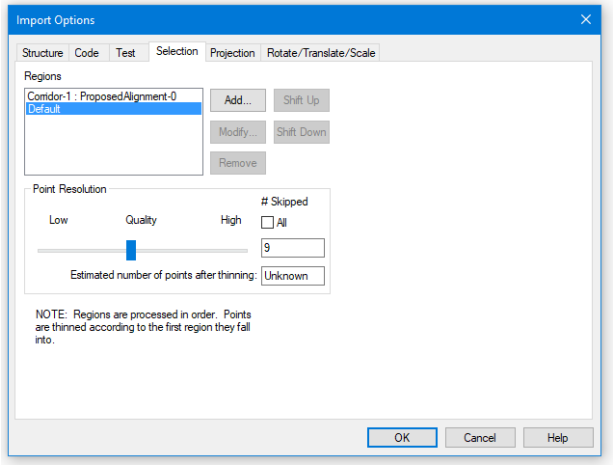

Figure 9-13: These *Setup Options* will skip most points outside of Corridor-1.

20. Press the *OK* button to read the data. It will take a couple of minutes to import about 69,000 points out of the 680,000 available.

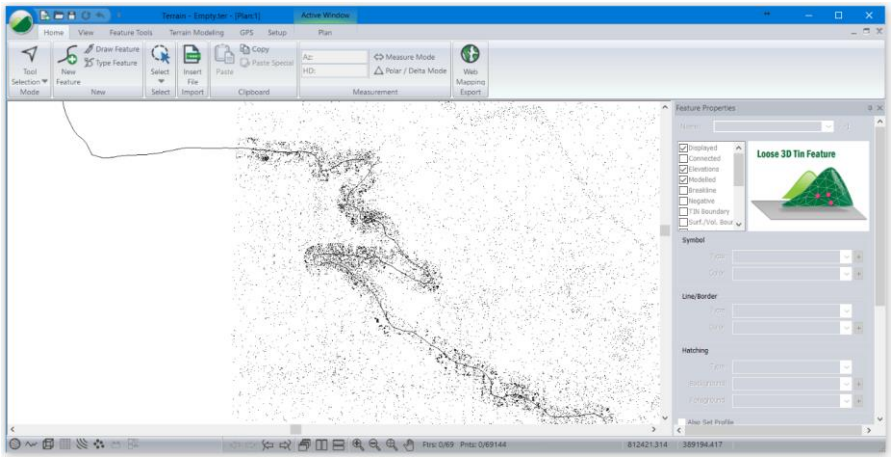

Figure 9-14: *Full Point Density along a Corridor*, Reduced Density Outside.

Note : There are additional, more accurate data filtering options available post-import. For more information on these please consult the *Softree Knowledge Base.*

21. *File* | *New*. Do not save changes.

10. New Location Design

In this exercise, a short road alignment will be created.

Note: See Getting Started section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

Open the Location Module or if you are running the Terrain Module, use the *Setup | Open Location* Module Button.

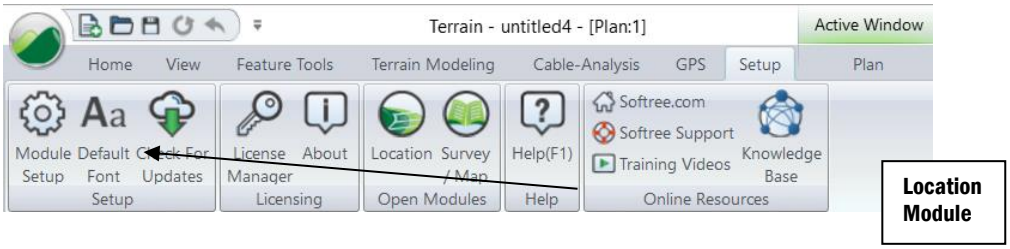

Figure 10-1: Setup Tab | Open Modules Group | Location Button

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.

1. Open the Location Module. *File | New File.*
2. Select *Terrain Surface*, and press the *Browse…* button and open <RoadEngCivil>\Location**Topo.terx**. Press *OK.*

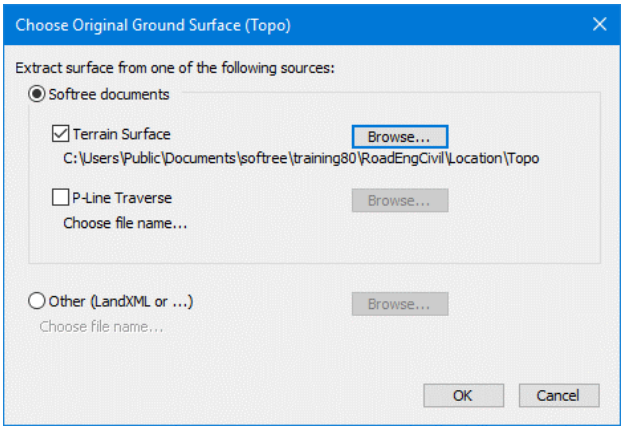

Figure 10-2: *File | New File – Opening File* to Define the Original Ground

3. The *Initial Alignment* dialogue box will appear. This allows you to select a start coordinate or to import an existing alignment.

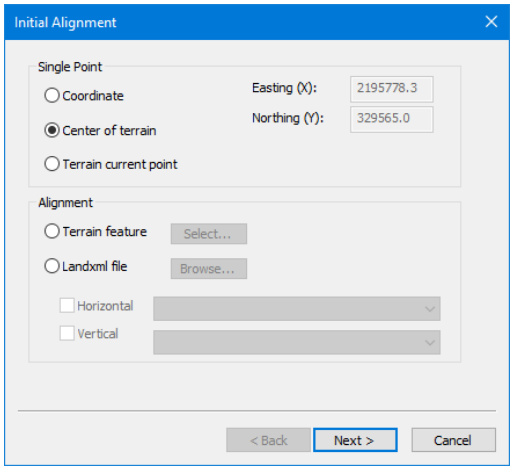

Figure 10-3: The *Initial Alignment* Dialogue Box

4. Choose *Center of Terrain* (we will define our start coordinate later).
5. Press *Next >*, keep *Standard Template* checked. Press *Finish*.

The look of your initial screen depends on the contents of the default Screen Layout (normal.dlt).

6. To change screen layout, select *View* | *Screen Layout* group, select layout **training Normal.DLT** from the training folder in the dropdown list.

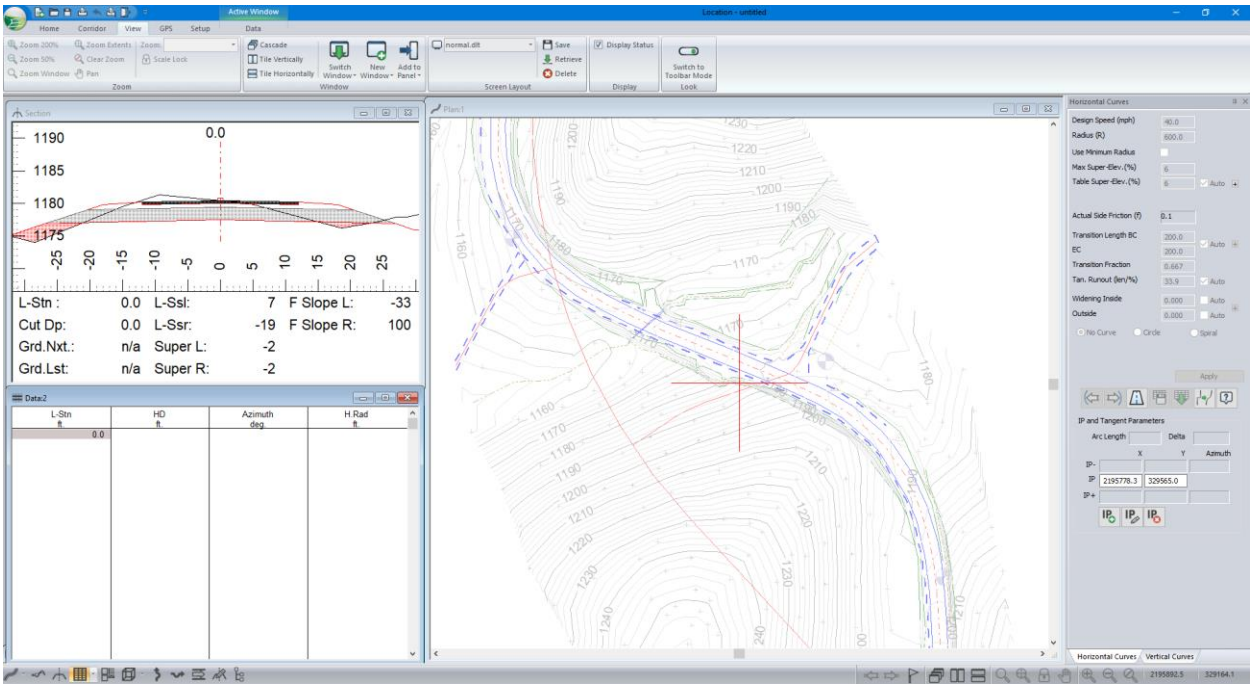

Figure 10-4: The *Location Module* after Begining a New Alignment

The red line in the figure above depicts the proposed new alignment, mostly to the left of the old road. 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.

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

7. *Home* | *Templates*. This will open the *Template Editor*.

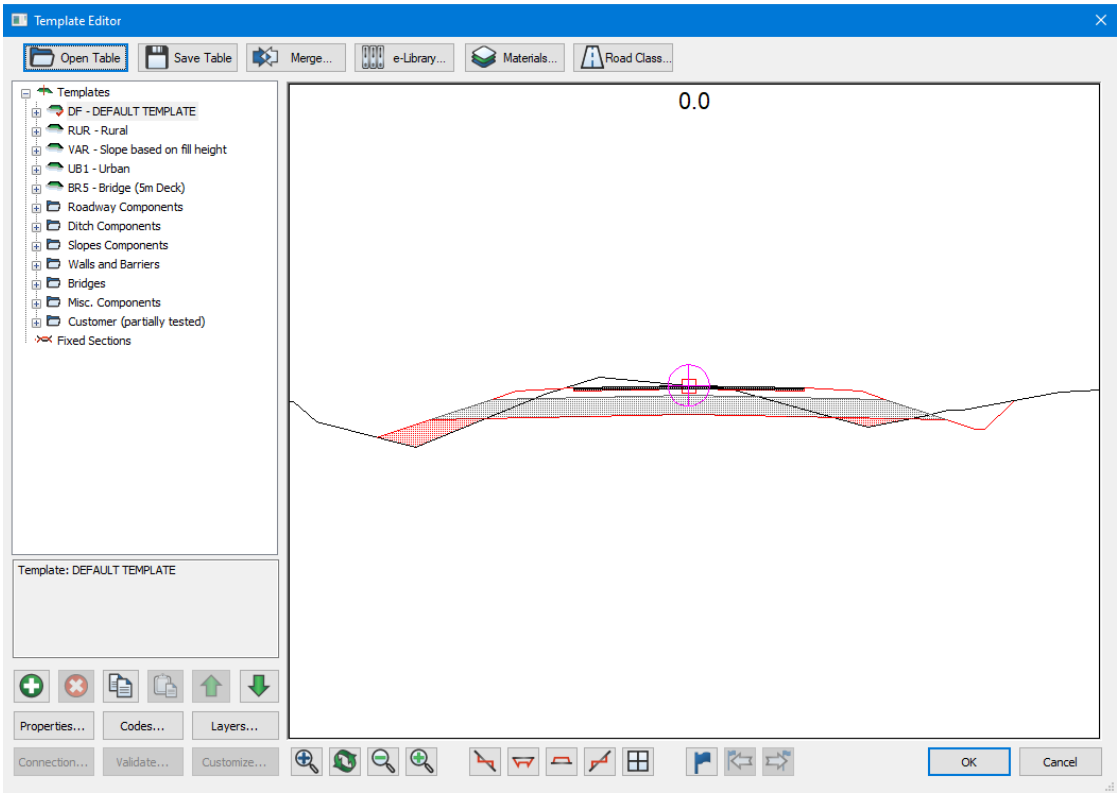

Figure 10-5: *Template Editor* Dialogue Box

8. Within *Templates Editor,* press *Open Table***,** select <Defaults and Layouts>\training**training Normal feet.tpl.** *Open.*
9. If you are working on a training computer or if you have not yet configured your default templates, you may wish to save template: *Save Table* | all selected, press *OK* | over-write **Normal.TPL** | say *Yes* to replace existing. Note that templates depend on length units (feet or meters).
10. *OK* to close the *Template Table Editor*. *OK* 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 similar to point editing in the Terrain module (exercise 8. Creating breaklines above).

11. <*Right-click*> (in the Plan window) | *Add/Edit IP tool* .

Note: Although already displayed in this file, to get your contours to display in plan-view <*Right-click*> | *Plan Options…* |check *Background* box. The existing road alignment is in the background as a guide for drawing the new road. The red line is the proposed new alignment.

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

13. *Zoom in* and fine tune the start point of the new alignment, so that it lies on the old centerline, just south of where the proposed road leaves the old one (Figure 10-6 below left).

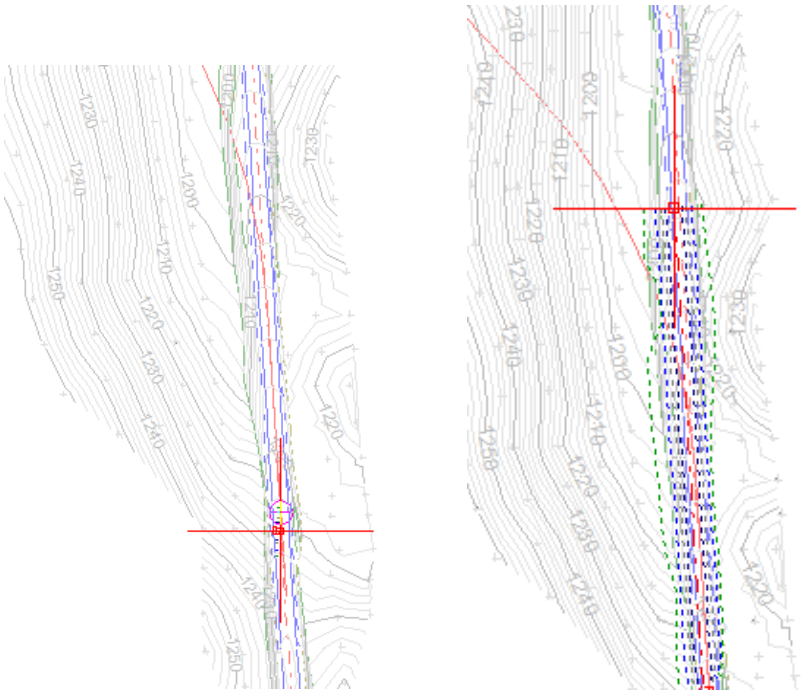

Figure 10-6: Drawing the *Horizontal Alignment*. Image on Left depicts the Starting Point of the Alignment. Image on Right Shows next IP

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

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

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

17. Delete an IP:
    - *Move* your mouse over the IP created above; note that the cursor changes to a box .
    - *Click* the mouse to capture the IP.
    - Type the <*delete*> key to remove the IP.

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

Note: Don't forget *Undo* command, <*Ctrl-Z*> or undo button.

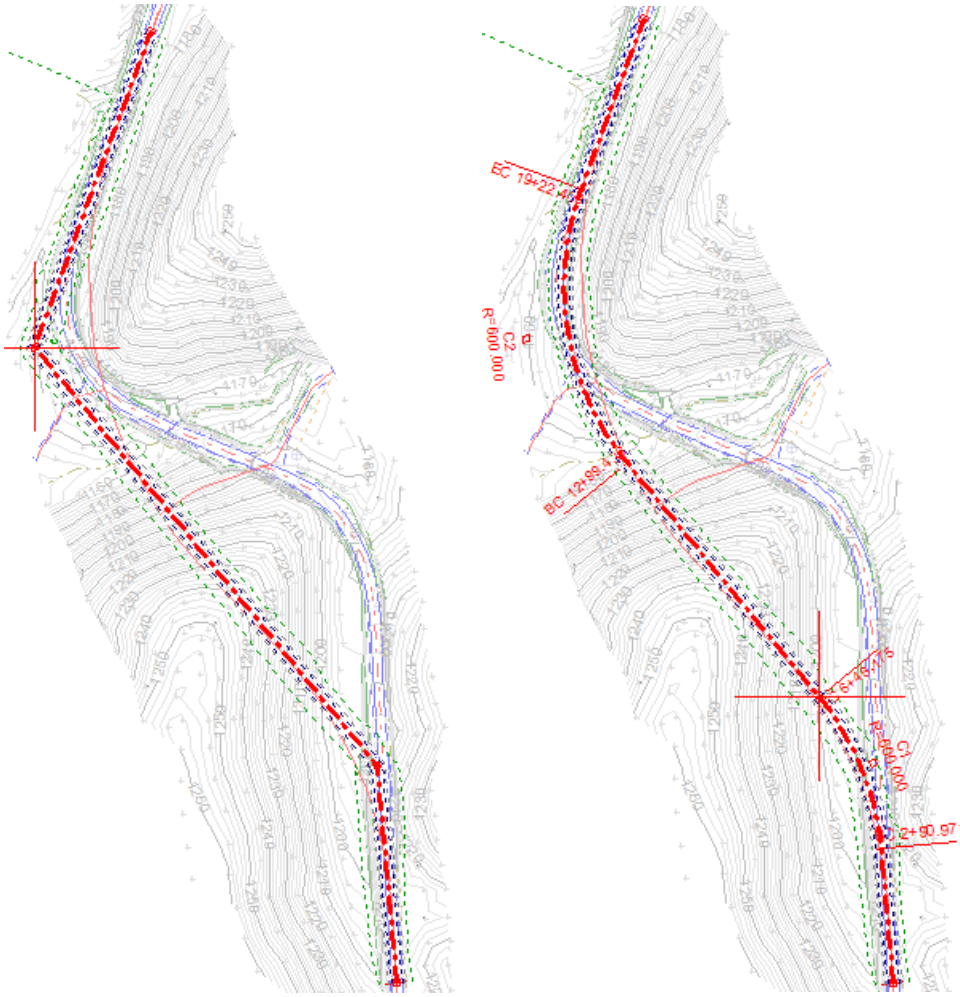

Figure 10-7: *Horizontal Alignment* Before and After Curves Applied

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*.

If it is not open, you can open a new curve panel using the Horizontal Curve Panel button in the window navigation toolbar.

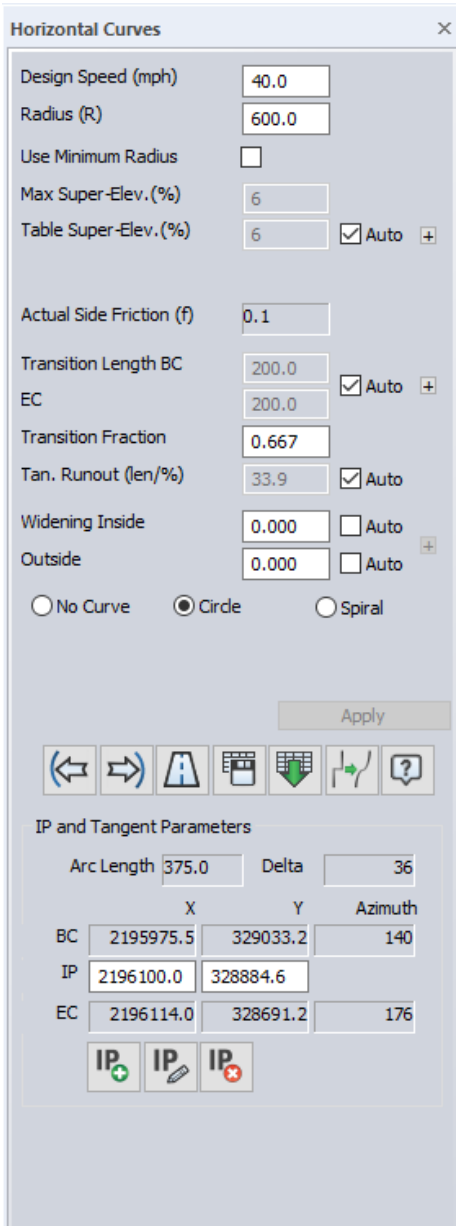

Figure 10-8: *Horizontal Curve Panel*

18. If not already shown, press the *Activate Horizontal Curve Panel* button using the Window Tools toolbar in the lower left part of the screen (figure above).
19. Use the *Previous IP* or *Next IP* buttons to move to the second IP in the alignment (watch the Plan window).

20. Press *Get Default Curve* button to set up the parameters as shown in the Figure 10-8: *Horizontal Curve Panel*, 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 .

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

21. Press the *Apply* button to create the first curve.
22. 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.
23. 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.

24. Go back to the curve panel and increase the radius of the two curves:
    - Use the *Previous IP* or *Next IP* button to select a curve.
    - Type a new radius (it must be small enough to fit or you will see an error message).
    - Press the *Apply* button.
25. 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.
26. *File* | *Close*. Do not save changes.

11. 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 similar to IP editing in the Plan window.

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 1.dsnx** in the Location Module**.** This should look just like your design at the end of the previous exercise.
2. *View* | *Retrieve* <Defaults and Layouts>\Training**training Profile.dlt**.

This will set up your screen to look like the figure below.

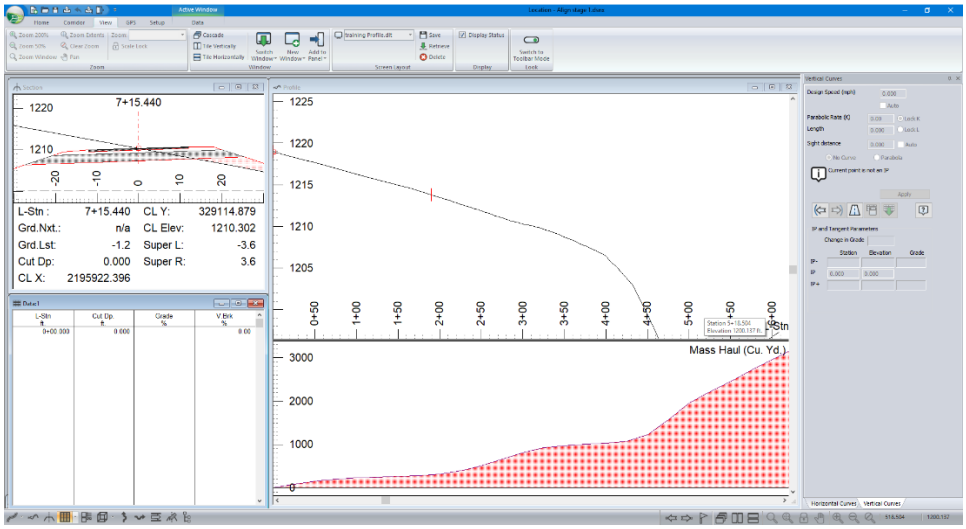

Figure 11-1: *No Vertical Alignment*, Profile Window Shows the Ground Line Only

3. <*Right-click*> in the Profile window and select the *Add/Edit IP tool* .Click anywhere to the right of station 0+00 to create a new Vertical intersection point (VIP). Move the captured point to a desired position and click again to anchor the point.

As in the previous horizontal alignment exercise, we can use the *mouse* to:

- *Create* a new VIP at the end of the existing alignment.
- *Edit* an existing VIP.
- *Insert* a VIP between existing VIPs.
- *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).

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

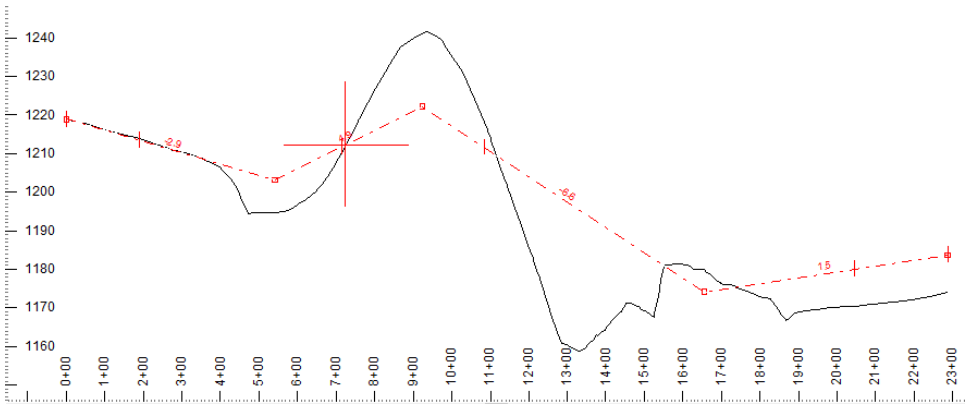

Figure 11-2: *Vertical Alignment* before Curves Applied

5. *File* | Close. Do not save changes.

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*.

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 2.dsnx**.
2. *View* | *Retrieve* <Defaults and Layouts>\Training**training Profile.dlt**.
3. If necessary, activate the *Vertical Curves* panel, *View* | *Add to Panel* button | *Vertical Curves*.

Note: If vertical curves is grayed out in the *Add to Panel* dropdown this means that it is already open. Look at the base of Horizontal Curve panel, one can switch between Horizontal Curves and Vertical Curves.

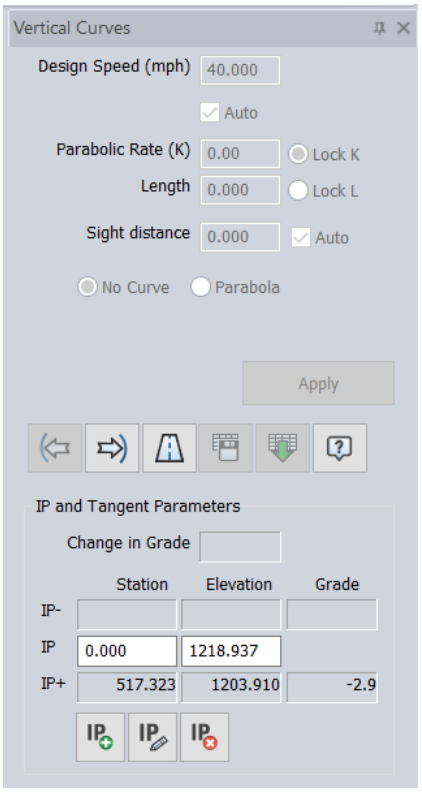

Figure 11-3: *Vertical Curves* Panel with Automatic Curvature Selected

4. Use the *Previous IP* or *Next IP* buttons to move to the second VIP in the alignment (watch the Profile window).
5. Press the Get *Default Curve* button. Then press the *Apply* button to create the first curve.
6. Use the *Next IP* button to move to the third VIP in the alignment.
7. 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).

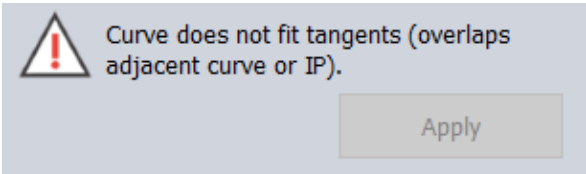

Figure 11-4: *Curve Warning*

Note: When a curve does not fit tangents (vertical or horizontal), you can either:
a. Shorten the curve.
b. Shorten the previous and/or next curve.
c. 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 40 mph (*auto* is checked), the only way to shorten the curve (without changing VIP locations) is to reduce the speed.

8. Change the Design Speed **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.

9. Go to the Profile window and move the third VIP down a bit. In the Vertical curve panel, again press the *Get Default Curve* button  . The error should have gone away. Press *Apply*.

10. Return to the Profile window and edit the third curve VIP. <*Right-click*> in profile window, *Add/Edit IP Tool*. Then c*lick* on intersection point at vertical curve 3 to capture it. Move the VIP up and down the curves will update dynamically as you move the mouse.

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.

11. Continue editing 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.

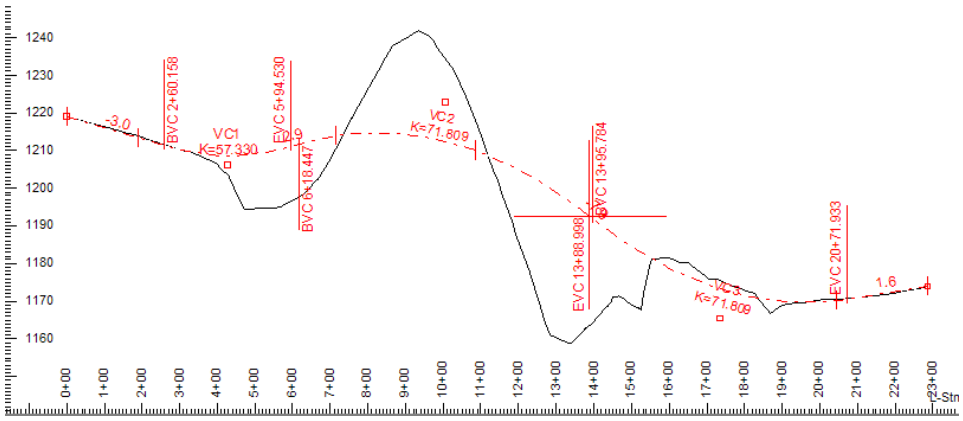

Figure 11-5: *Vertical Alignment* after Curves Applied

You should also try to make the mass haul balanced. Mass haul is covered in more detail in following exercises.

Note: To delete multiple points on the alignment or entire alignment click on *Home* |*Delete Range* button. In you uncheck the *All Points* box and check *Profile (Vertical IPs)* to delete the entire profile.

12. *File* button | *Close*. Do not save changes.

12. Cross Section 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*.

Template Editor

Note: Refer to *Getting Started* section for file install folders (<RoadEngCivil> and <Defaults and Layouts>).

1. In the Location module, open <RoadEngCivil> \Location2**bluff_road.dsnx**.
2. *Home* | *Templates*, to open the *Template Table Editor* shown below.
3. Click on *slope right*  (to the right of the graphic) to prepare for the following section.

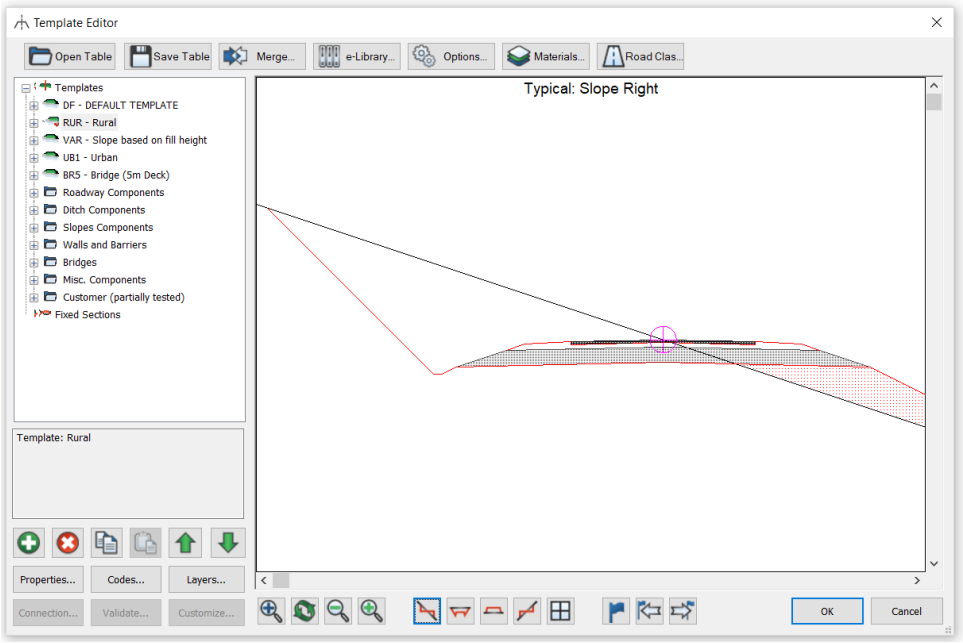

Figure 12-1: *Template Table Editor* Dialogue Box

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

The template editor shows you the templates  contained in a table and the components  contained in each template. These are the items you commonly work with.

Template Properties

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

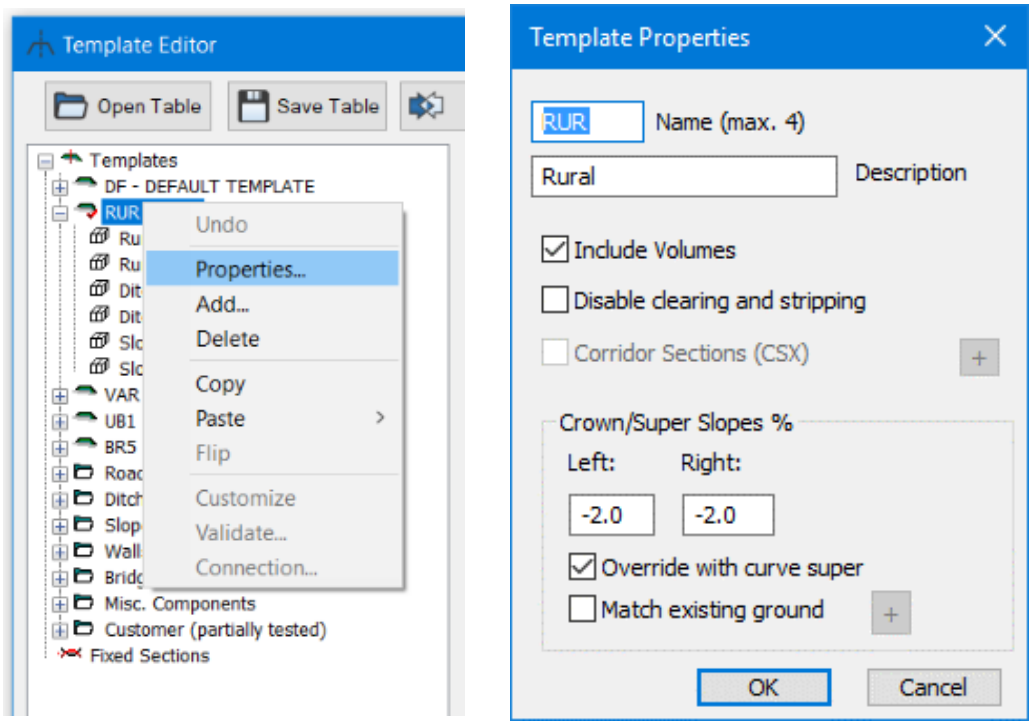

Figure 12-2: *Template Properties* Dialogue Box

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 happens 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. <*Right-click*> on the RUR-Rural template and choose *Undo Modify* to restore the initial settings.

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*> | *Paste* > | *As New*. The new template will appear at the bottom of the list.

10. Select the new xx0-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 xx0-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 dialogue boxes are quite different from the *Custom* components properties (figures below).

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

Template Component Properties

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 (usually distance) 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 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 (rarely used).

12. <*Right-click*> on the *Rural Paved-left* component of the *RUR-Rural* template | select *Properties…* to open the *Template Component Properties* dialogue box (Figure 12-3).

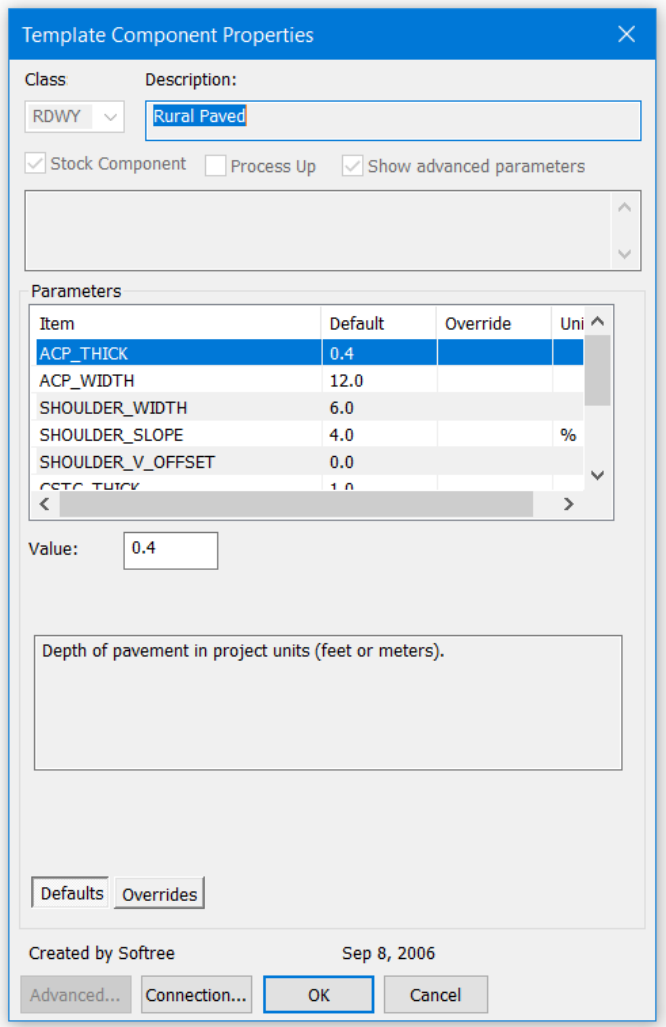

Figure 12-3: Rural Paved *Roadway Component* Properties

This component allows you to change various parameters such as pavement thickness, lane width shoulder width, and various thicknesses and slopes.

13. Change the lane width:
    - Select *ACP_WIDTH* in the *Parameters* list; note the pavement highlight in the graphic.
    - Change the *Default* value to **16.0**; note that the graphic updates immediately.

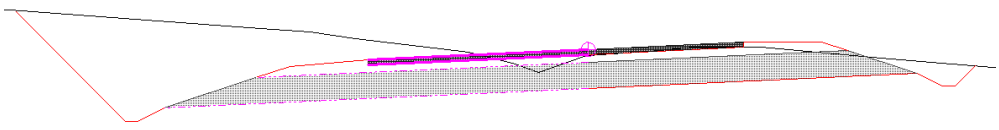

Figure 12-4: Template Graphic Depicting the Effect of Changing the *ACP_WIDTH* Variable While the *Component Properties* Dialogue is Open

Note: When there is a *Feature* option for a given parameter, it means that you can use a reference feature centerline offset to define the parameter instead of the default value. *Reference Features* can be defined by pressing the button.

14. Press *OK* to accept changes and close the properties dialogue box. Notice the road has widened on the left.
15. Similarly, open the *Ditch-left* properties dialogue box (Figure 12-5 below left).
    - Select each available parameter and note the highlighted dependent parts in the graphic. Also notice that the options and description change with each selection.
    - Press *Cancel* to close the properties dialogue box.

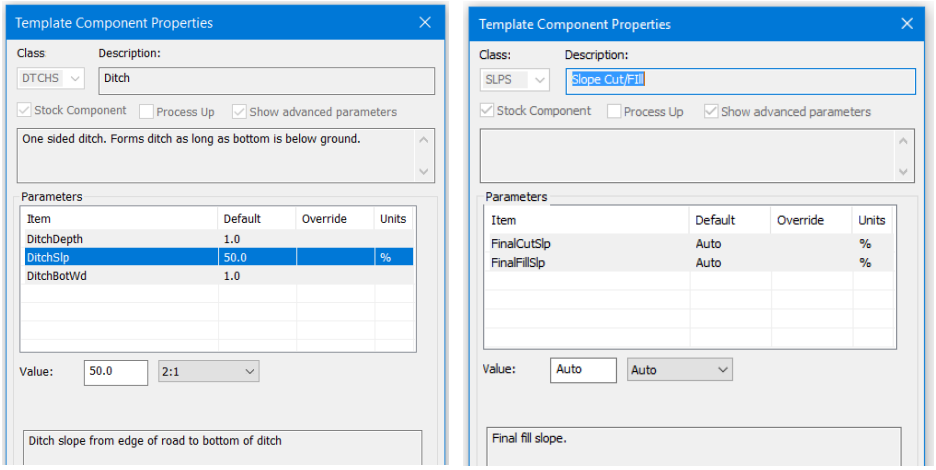

Figure 12-5: *Ditch and Slopes* Component Properties

16. Open the *Slope Cut/Fill-left* properties dialogue 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.
17. Press *Cancel* to close the properties dialogue box.

The ground slope buttons (shown below) allow you to view typical cross sections.

- Slope Right
- Slope Left
- Cut
- Fill

18. Select the *slope left* button Notice how the black ground line changes and how the template accommodates.
19. *Click* and *drag* the cross in the template graphic area. Note that you can change the template position and see how it will behave in different situations.
20. Softree added this ability to change the template position (up and down, for example), the difference between Cut and Fill cross sections have become irrelevant.

Note: The middle mouse pan and zoom work in the template graphic area. To turn off/on labels in the Template Editor: *Options* | *Select all the layers* | *Select/de-select Display Labels*

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

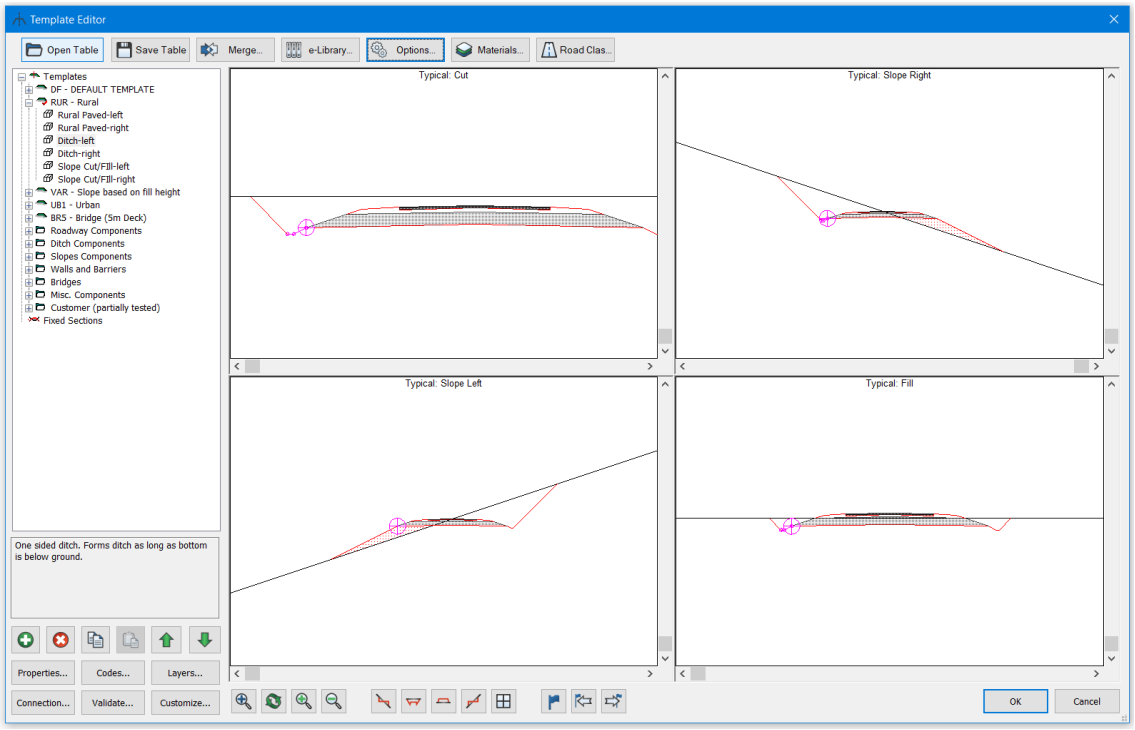

Figure 12-6: Split Screen View Showing the *4 Typical Sections*

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.

Working with Components

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

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

24. Scroll down the template tree list and *open* the *Slopes Components* folder (Figure 12-7).

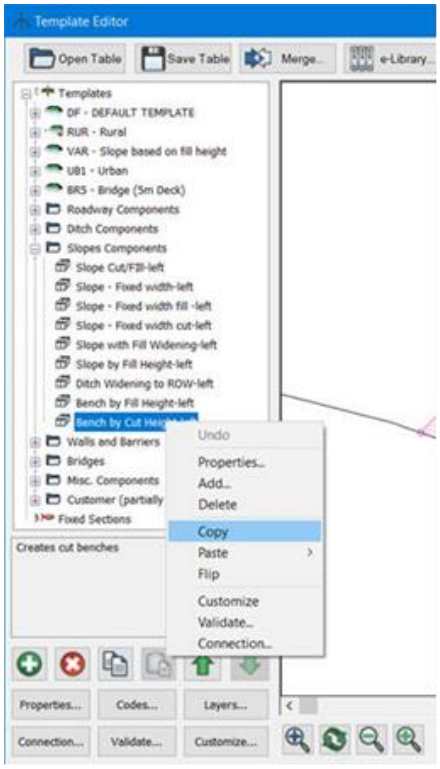

Figure 12-7: *Copying a Component* from a Folder

25. <*Right-click*> on the *Bench by Cut Height-left* (note to select by cut and not by fill) component and select *Copy*.
26. *Scroll up* until you can see the *RUR-Rural template* again.

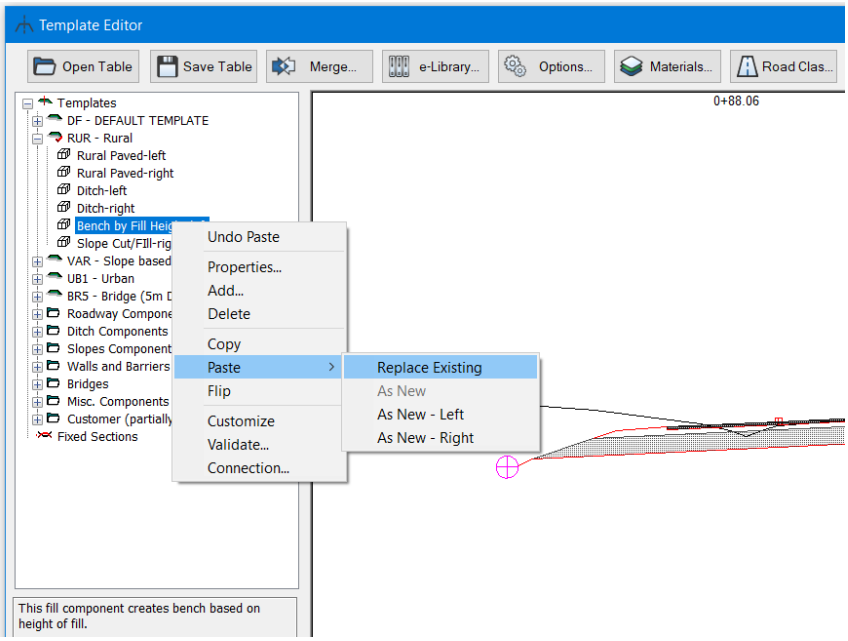

Figure 12-8: Pasting a Component Into a *Template*

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

You will be prompted with the dialogue box shown below. This allows you to copy parameters between similar components. In this case we just want to overwrite the component **not** the parameters.

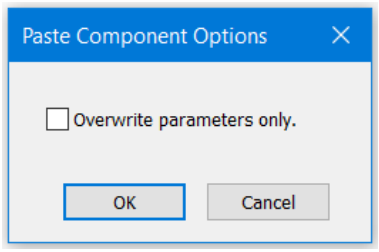

Figure 12-9: *Paste Options* when Replacing an Existing Template Component

28. Leave the *Overwrite parameters only* option *unchecked* (as shown in the figure above) and press *OK*.
29. *Click* and drag the template down until you can see the *cut benches – figure below* (you may also need to zoom out).

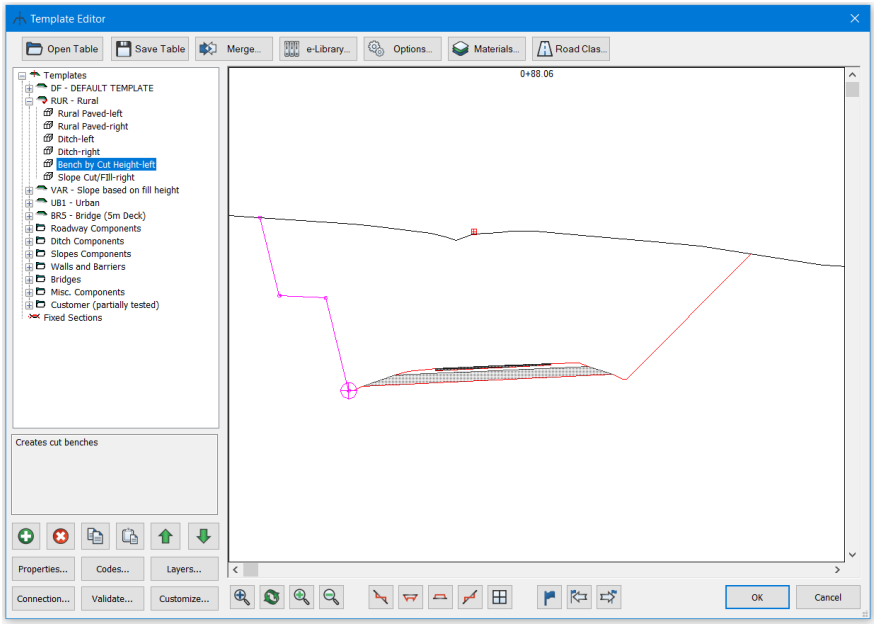

Figure 12-10: The *New Slopes Component* is only on the Leftside

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

30. Open the properties dialogue box for the new component and change *BenchWidth* to **15**. Press *OK* to commit the change.

31. Notice that the cut bench component is only on the left side. We will copy it (with its new bench width) to the right:
    - <*Right-click*> on Slope Cut/Fill-right and choose menu Delete.
    - <*Right-click*> on Bench by Cut Height-left and select Copy.
    - <*Right-click*> again *Bench by Cut Height-left* 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.

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

The e-Library

Softree maintains a library of template components on the web.

34. If you are connected to the web, press the *e-Library* button, within the template editor, to open the dialogue box in the figure below:

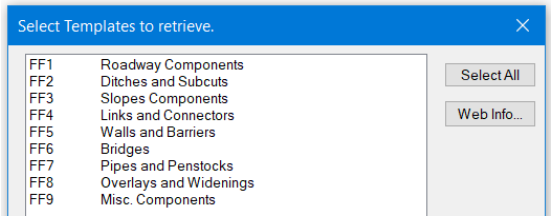

Figure 12-11: *Component Folders* available from Softree's Website

35. Press the *Web Info…* button. The Template Resource page lists the available components and their parameters.

Note: If you are using components it is always best to load or reload them directly from the e-Library.

36. Press *Cancel* to close the dialogue box.
37. *File* | *Close*. Do not save changes.

13. Volumes & Mass Haul

Note: Getting started section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 3.dsnx**.
2. *View* | *Retrieve Screen Layout* <Defaults and Layouts>\training**training Profile Only.dlt**. This will set up your screen to look like the figure below.

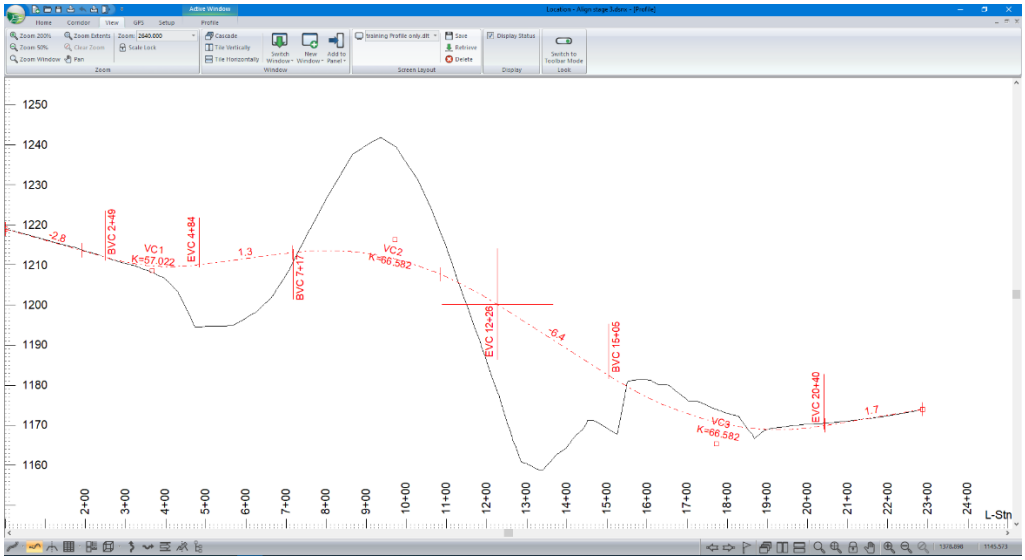

Figure 13-1: Location Design - *training Profile Only.dlt* Screen Layout

Displaying Cut and Fill Volumes in Data Table

3. Activate a new Data Window from the bottom navigation toolbar.

Change the columns displayed in the data window:

4. <Right-click> | *Data Options*...
5. Click the plus next to *Point Types* (Rows), this will open the *Point Type Selection* dialogue.
6. De-select any current selected point types (<*double-click*>).
7. <*Double-click*> on *Auto interval points* (xx.000 ft). If the interval is not currently set to 20.0 ft, press *Properties*... to configure. Adjust the *Interval* to **20.0 ft**. Press *OK* to return to the previous dialogue.
8. Press *OK* to return to the *Data Window Options* main dialogue.
9. Press *Columns*.... Add the following fields as shown in the figure below:

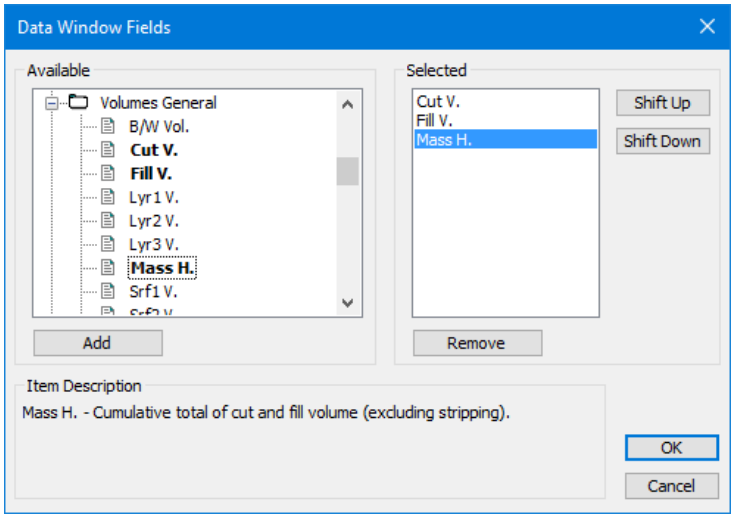

Figure 13-2: Configuring Data Window Fields

10. These are the subgrade volumes. *Cut V.* is the subgrade cut volume, *Fill V.* is the subgrade fill volume.
11. Press *OK* twice to return to the Data Window as shown below:

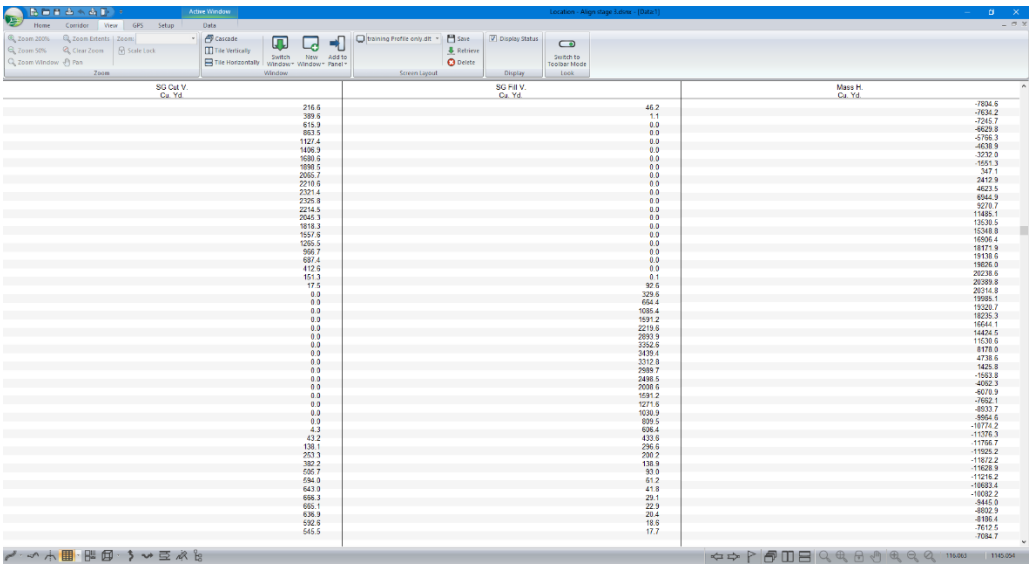

Figure 13-3: Data Window with Sub-grade Volumes

Note: Data Window details can be copied and pasted directly into an Excel spreadsheet. To do so, *<right-click> | Copy Data to Clipboard*, then open Excel and paste.

Displaying a Mass Haul Graph

The *Mass Haul* graph gives you quick, qualitative information about cut / 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).
The default setting for mass haul includes only sub-grade material, however it is possible to select specific materials to be included.

12. Activate the Profile window  from the bottom navigation toolbar.
13. <<*Right-click*>> in the Profile window | select *Profile Options…* to open the dialogue box shown below left.
14. Press *Select…* at the bottom of the *Sub-Windows* area. This will display the two-list dialogue box shown below on the right.

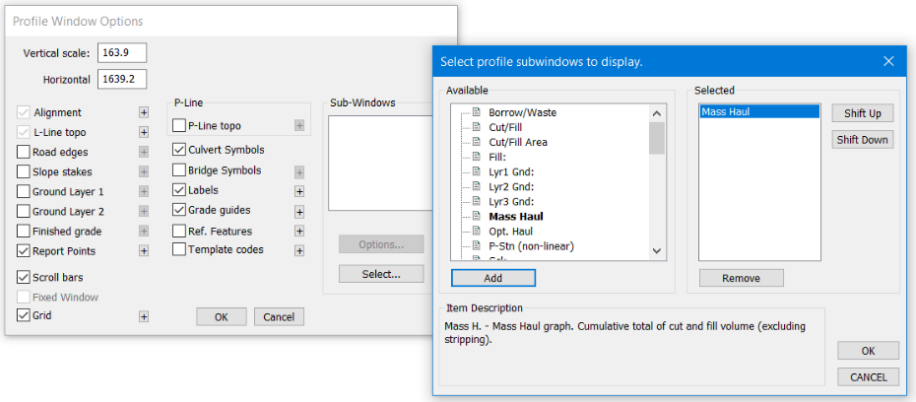

Figure 13-4: *Profile Sub-Windows* to Display

15. Select *Mass Haul*, press *Add* (or <Double-click>) to add it to the *Selected* list as shown 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.

16. Press *OK* button twice to accept changes and close the dialogue boxes.

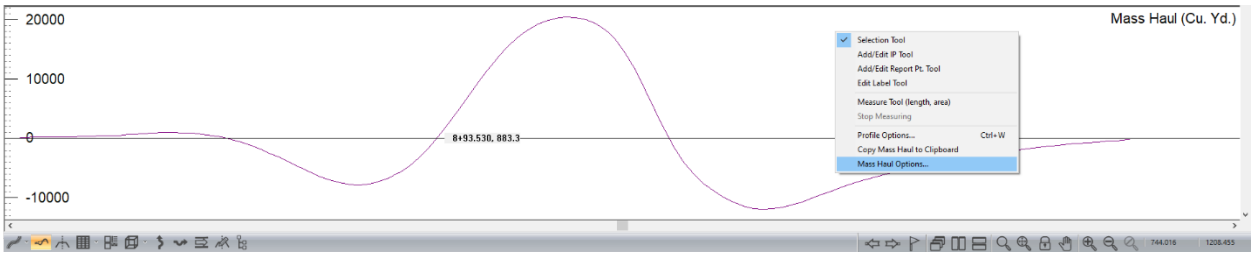

Figure 13-5: *Mass-Haul* Displayed in a *Profile Sub-Window*.

17. 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.
18. <*Right-click*> in the mass haul window | select *Mass Haul Options…* to open the dialogue box shown in the figure below.

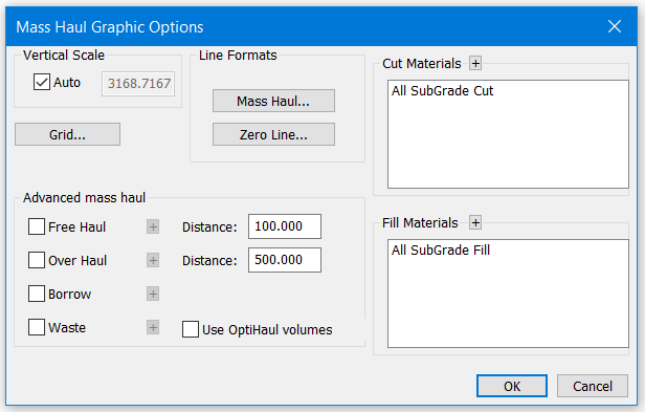

Figure 13-6: *Mass Haul Graphic Options* Dialogue Box

19. The concepts behind the mass haul diagram are discussed in detail in the help document. 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.
20. Within *Mass Haul Graph Options* dialogue box, 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. *Cancel* button to close the grid options.
21. The *Mass Haul…* and *Zero Line…* buttons allow you to control the line style and color of the basic graphic items as shown in the Figure below.
22. Turn on all four *Advanced mass haul* items. This enables hatching.

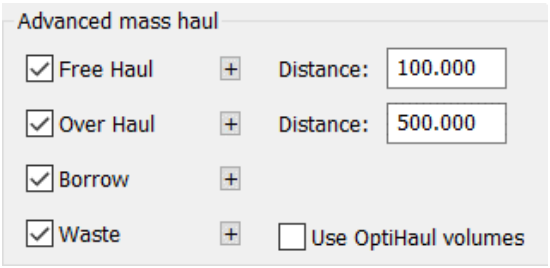

Figure 13-7: *Advanced Mass Haul* Items Enabled

The *Free Haul Distance* and *Over Haul Distance* are controlled by the fields on the right (see Figure above).

The *plus* 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.

23. Press *OK* to accept changes and close the options dialogue box.

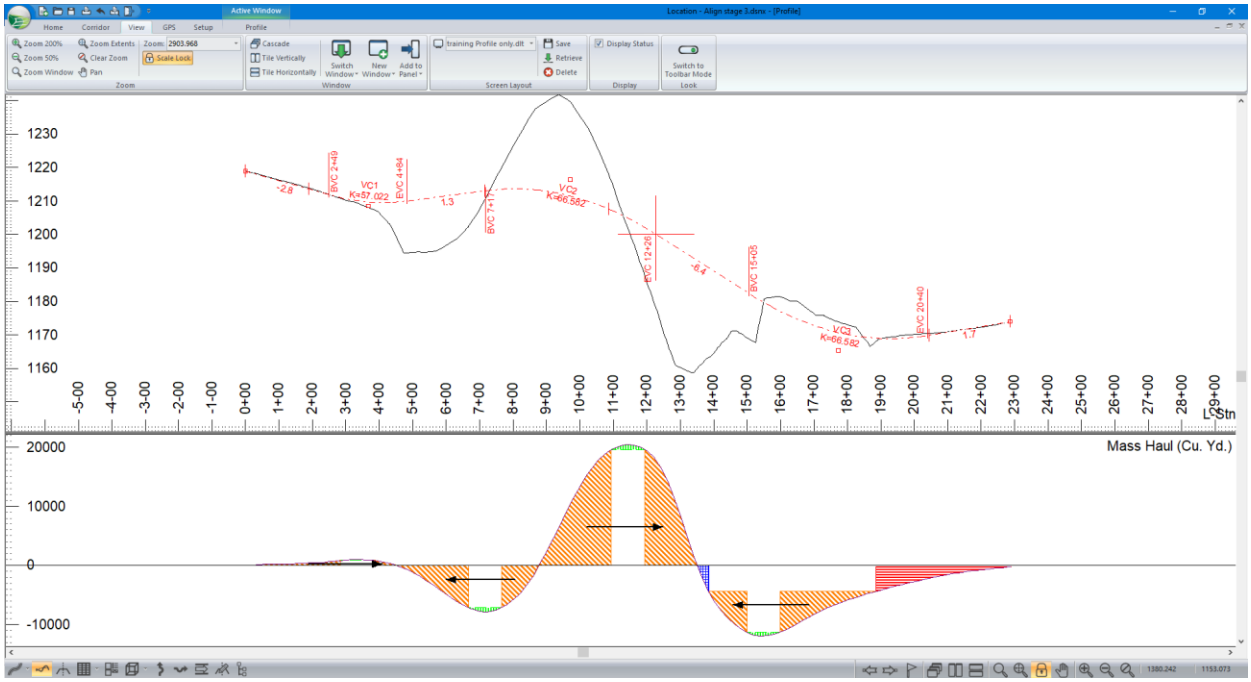

Figure 13-8: *Mass Haul* with *Advanced Features* Turned On

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).

24. It is possible to modify the mass haul to account for borrow and waste. We will add a new *Borrow* pit:
    - *Home | Assign by Range*.
    - Select the *Pits* tab.
    - *Add…* Set the *access station* to **450**. Press *Ok.*
    - Select *Borrow* and uncheck the *Variable volume (smart pit)*.
    - In the *Volume (Cu. Yd.)* capacity box enter **8000**.

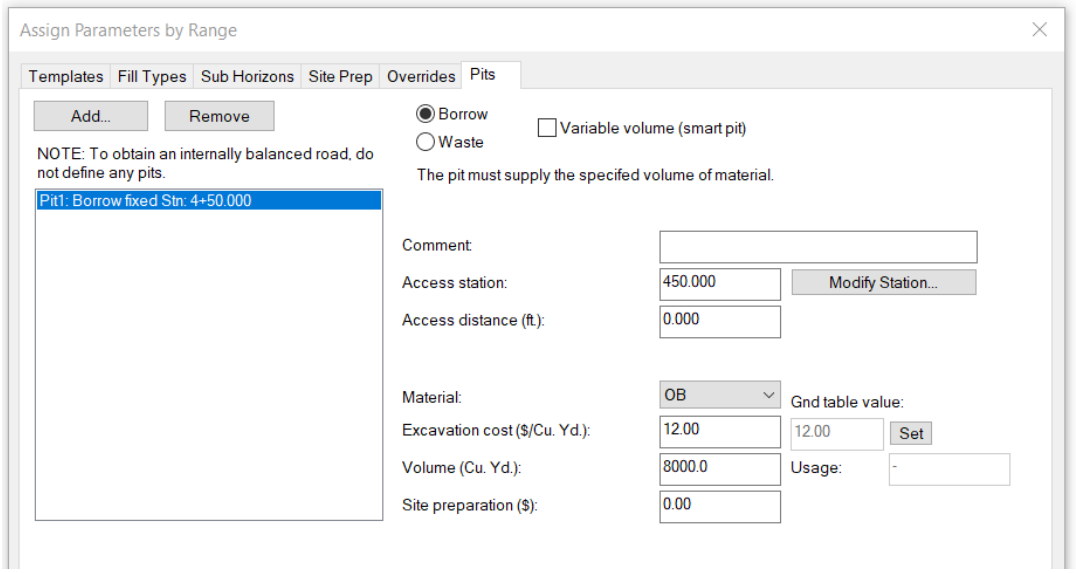

Figure 13-9: *Assign Parameters by Range, Pits* Tab

- Press *OK*.

25. Check both Update entire design and Re-Cost.
26. Respond *OK* to the "Recalculate road alignment?" prompt.

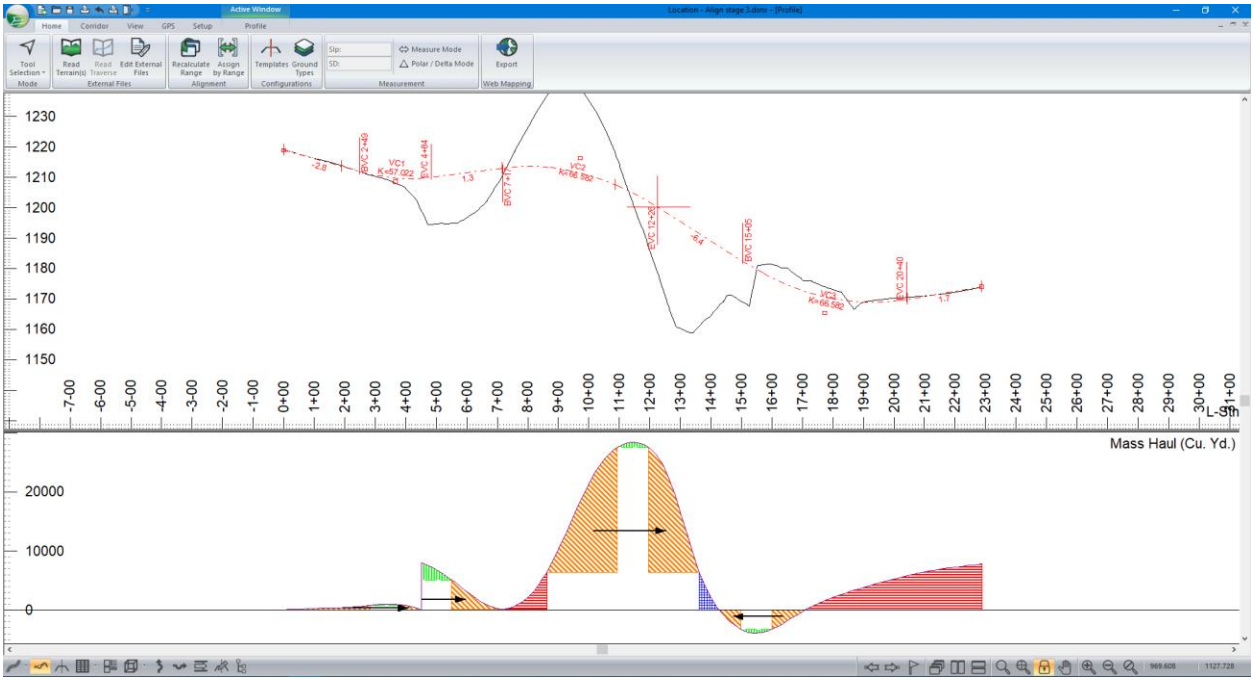

Figure 13-10: *Mass Haul* after 8000 Cubic Yards Borrowed at Station 4+50

Now there is no need to *pull* material back across the gully.

27. *File* | *Close*. Do not save changes.

14. Alignment Design and Volume Balancing

In this section, we will explore some of the RoadEng® features for alignment design and earthwork balancing. We will also introduce several *Softree Optimal* features such as *Design Time Costing*, *Optimal Haul Calculation*, *Smart Pits* and *Quick Fit Profiles*. These features are included in the basic RoadEng® software and are very useful for speeding up workflow, comparing alignments and reducing construction costs.

These functions are covered more completely in the *Softree Optimal* documentation (downloadable from the documentation section of the Softree Support website).

NOTE: This exercise assumes that you are familiar with mouse editing of vertical and horizontal alignments.

Alignment Design Objectives

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.

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 3.dsnx** in Location Module.

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

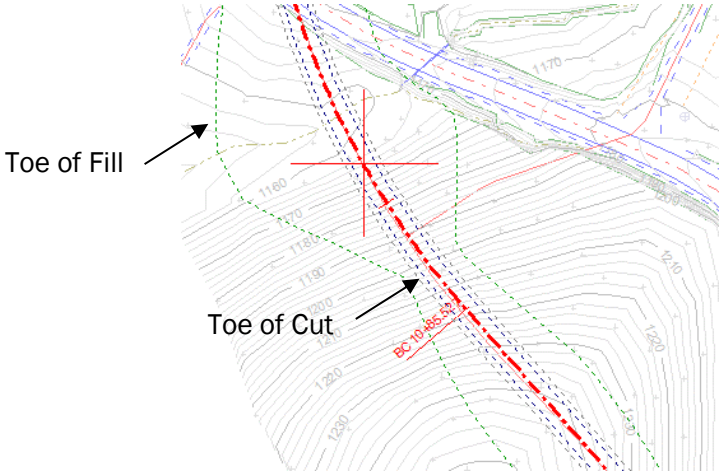

Figure 14-1: Green Dot Slope Stakes (catch point) Lines in the Plan Indicates the Footprint of the Design

2. Press the *Activate Profile Window* button , in windows toolbar, to open the Profile window using in the windows toolbar 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).

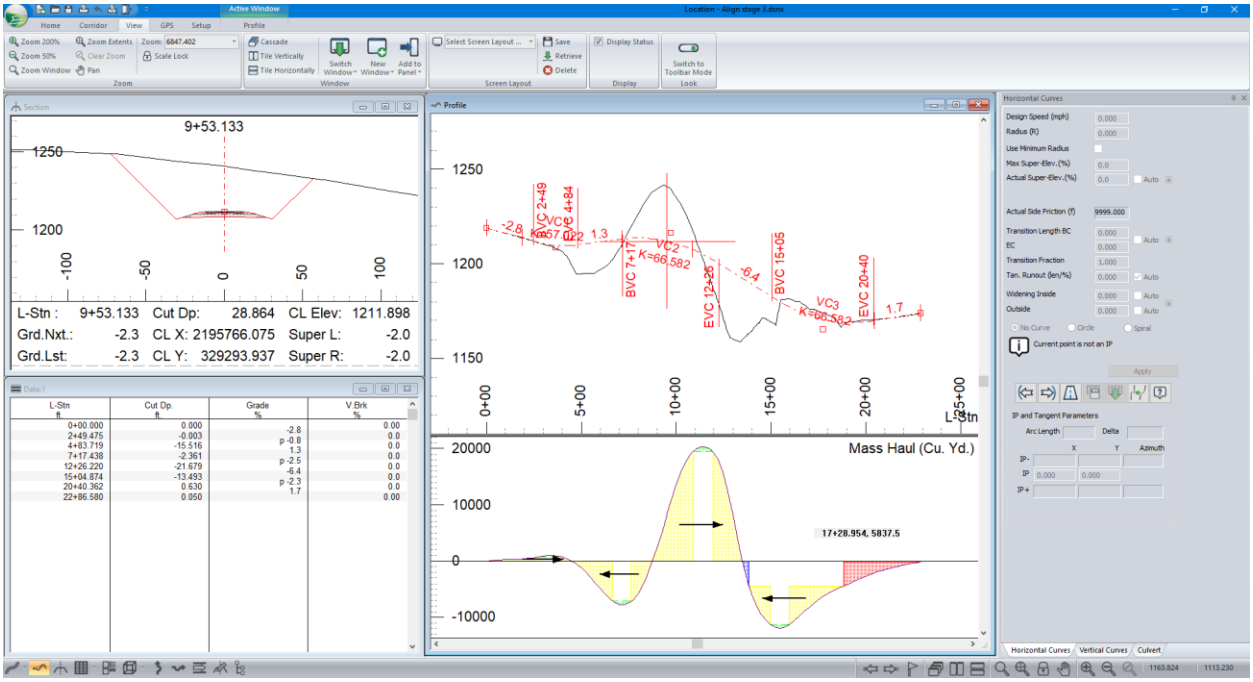

Figure 14-2: The Road Passes Through a Hill and Therefore has Large Cut Volume.

4. Press *Activate Plan Window* button  , in windows toolbar. You can see the contours representing the hill under the current point.

5. Press *Activate Horizontal Curve Panel* button  , in windows toolbar. To navigate *Previous IP*  or *Next IP*  buttons to the first curve.

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

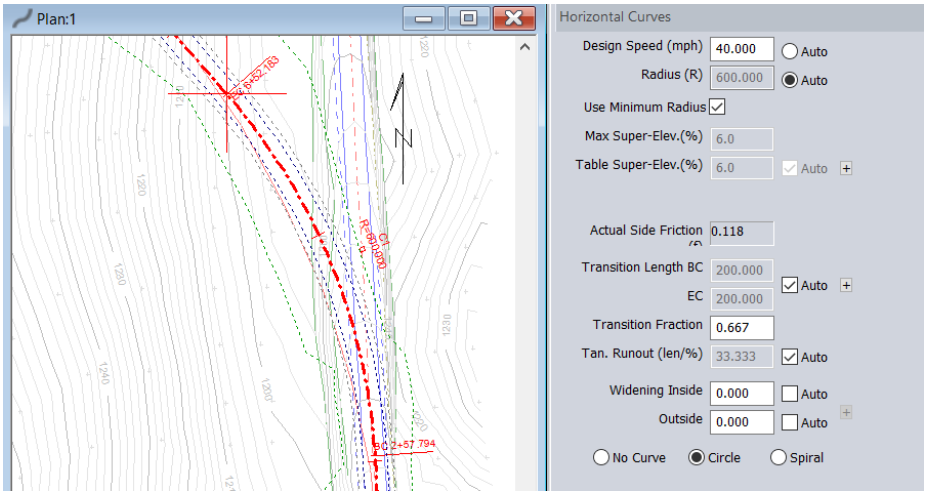

Figure 14-3: *Use Minimum Radius* Checked Defines the Smallest Curve Possible Given Design Speed, and The Super-Elevation Table

7. *Apply* the smaller 600 ft 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. Change to edit mode ; <*Right-click*> choose *Add/Edit IP Tool*.
9. In the Plan window, move *C1* (the IP for the South curve) North and move *C2* (the IP for the North curve) South (both while maintaining tangency with the old road).

If you move the *C2* too far south, the cross section will extend beyond the edge of the surface model (figure below). Be sure that the fill areas of the road do not fall off the topographic surface. You will never be able to balance the mass haul diagram this way.

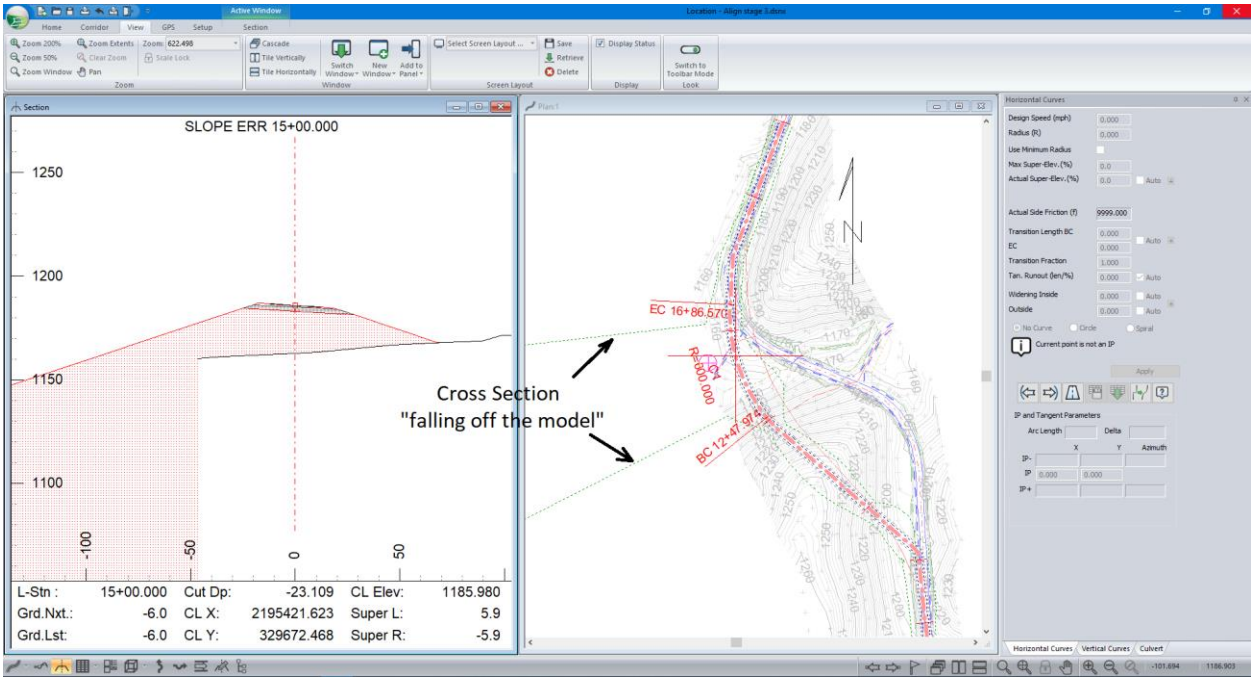

Figure 14-4: *Plan* and *Section* Windows Depicting What Occurs When The Road Section Extends Beyond the Original Ground Surface Model

10. Note that a captured curve IP will stop moving (and there will be an audible beep if you have sounds on) when the curves bump into each other and become an S curve.
11. Open *Activate Profile Window* button and notice the new shape of the ground (figure below).

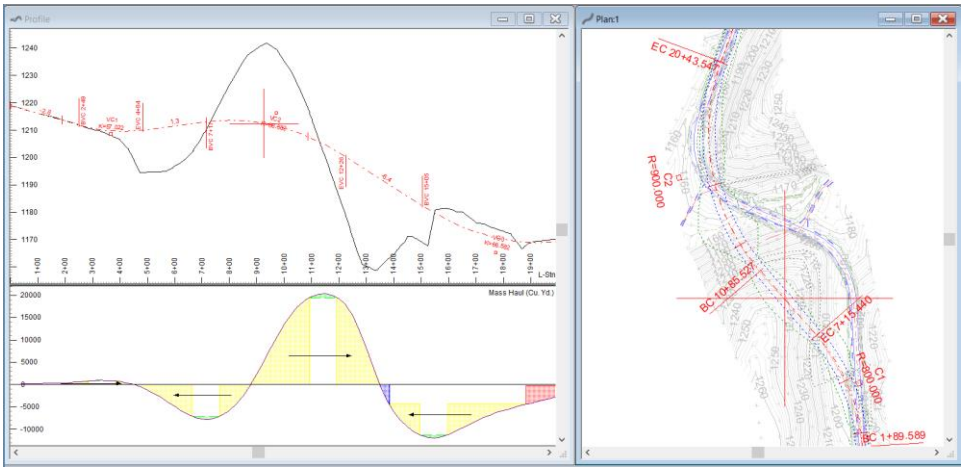

Figure 14-5: Alignment Closer to the Old Road (but still 40mph) Significantly Reduces the Volumes

12. Adjust the vertical alignment to balance the volumes; use the mass haul diagram (figure above shows an almost balanced Mass Haul).

In Figure 14-5 above, the material cut to pass through the hill is about *28,000* cu. yds. (8000 + 20000). After the grades have been reduced alignment the footprint becomes smaller.

Your design will be different from the example in the figure, but you can use the same techniques to quickly evaluate its quality.

13. Note that 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.

14. *File* | *Close*. Do not save changes.

Softree Optimal Design Tools

RoadEng® contains several features from the Softree Optimal technology. This section will briefly introduce these functions.

Note: These functions are covered more completely in the *Softree Optimal* documentation (downloadable from the documentation section of the Softree Support website).

The following features are available:

- ***Design Time Costing*** - dynamically calculates the cost of a design based on cut, fill and material movement.
- ***Optimal Haul Calculation***- determines the best (lowest cost) way to move material.
- ***Smart Pits*** - Automatically determines the pits to borrow and waste material.
- ***Quick Fit Profile*** - Quickly calculates a starting vertical alignment which matches your curvature and grade constraints.

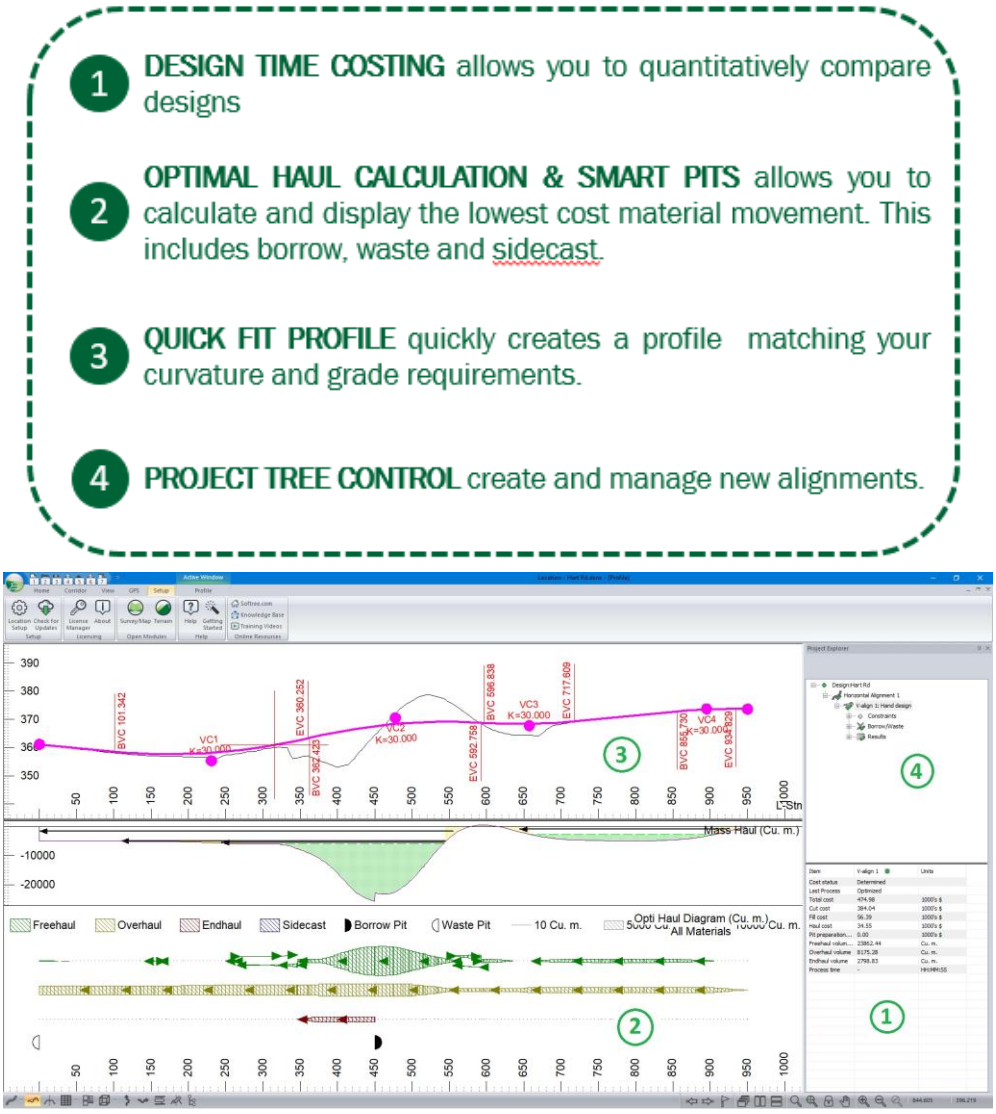

Figure 14-6: *Softree Optimal Features*.

Design Time Costing

Cost reporting and feedback is useful at all stages of design (preliminary, detailed and construction estimation).

Design Time Cost Reporting is the ability to accurately evaluate the cost of a particular design interactively before it is complete. *Softree Optimal* provides interactive and automated feedback to report earthwork costs. This functionality is extremely useful for manual design and is a prerequisite for optimization.

Earthwork cost calculations are based on material excavation, embankment, movement and borrow/waste locations.

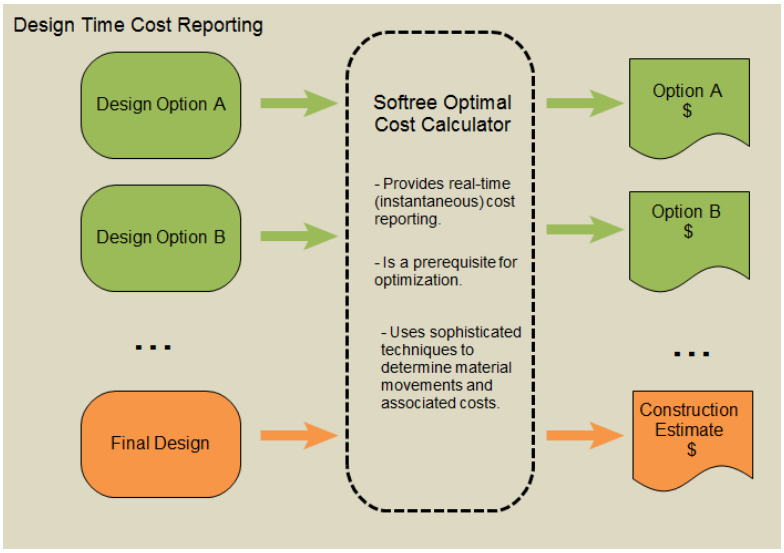

Figure 14-7: Design Time *Cost Reporting*

Optimal Haul Calculation

When *Softree Optimal* calculates the cost of an alignment, it determines the lowest cost prescription (or recipe) for moving material. We call this the *Optimal Haul*. The *Optimal Haul* is a detailed description of how material is moved along the alignment, and from/to borrow/waste pits.

Traditionally the mass haul diagram has been used to represent material movements, however it has some drawbacks. The *mass haul diagram* does not fully expose the *Optimal Movement Prescription.* It does not provide a detailed schedule of earth movement between stations and it does not handle the concept of material quality introduced in the case of multiple materials.

The *Optimal Haul Diagram* addresses these two deficiencies.
The Optimal Haul *diagram* illustrates the *Optimal Movement Prescription* (as determined by Softree Optimal).

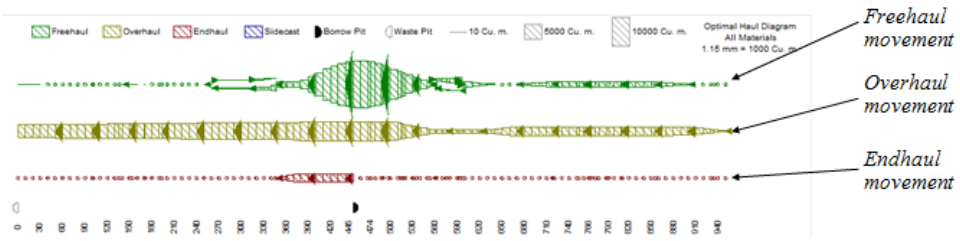

Figure 14-8: *Optimal Haul Diagram*

Smart Pits

The *smart pit* feature allows the user to determine the best location to borrow or waste material from a set of pits. Each Pit has the following information:

Access station - location on the alignment from which the pit is accessed.

Distance - from access station to the borrow/waste site (sometimes called *dead-haul* distance).

Elevation - at the pit. Press the *Get from Alignment* button to assign the same elevation as at the *Access Station*.

Material - available (borrow pit only).

Excavation $ - Cost to excavate (borrow only).

Waste quality - The minimum material quality required (non-variable only).

Capacity limit - Maximum volume of borrow or waste (*variable* only).

Volume - Exact amount of borrow or waste (non-*variable* only).

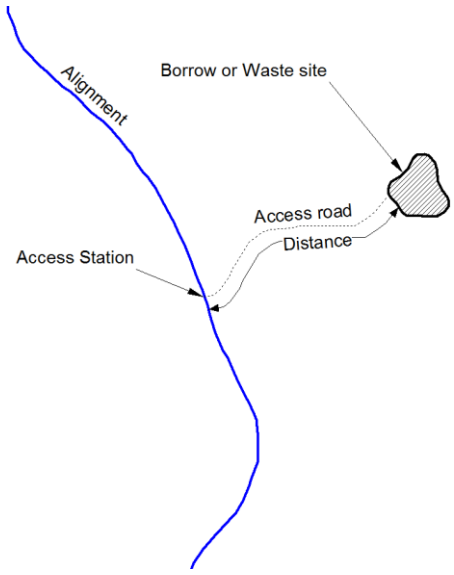

Figure 14-9: *Smart Pits* account for access road distance

Quick Profile

Quick Profile generates, if possible, the closest profile to the ground considering all the geometric constraints defined by the user. The cost of this alignment will also be calculated.

The Quick Profile feature is very useful for determining if an alignment is feasible based on K values, min/max grades and predetermined control points.

15. Setting Up a Screen Layout

In this exercise, you will configure the window locations and some window options to emphasize horizontal curves (for use in *Horizontal Curve Details*).

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**)

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 4.dsnx** in Location Module**.**
2. Maximize the plan window.

Note: <Double-clicking> in any window title bar will maximize it. Arrange your screen to resemble Figure 15-1.

3. Ensure the *Horizontal Curve Panel* is visible.

Note: Once the panel is open the user can click on the *Auto Hide* to maximize the screen viewing space.

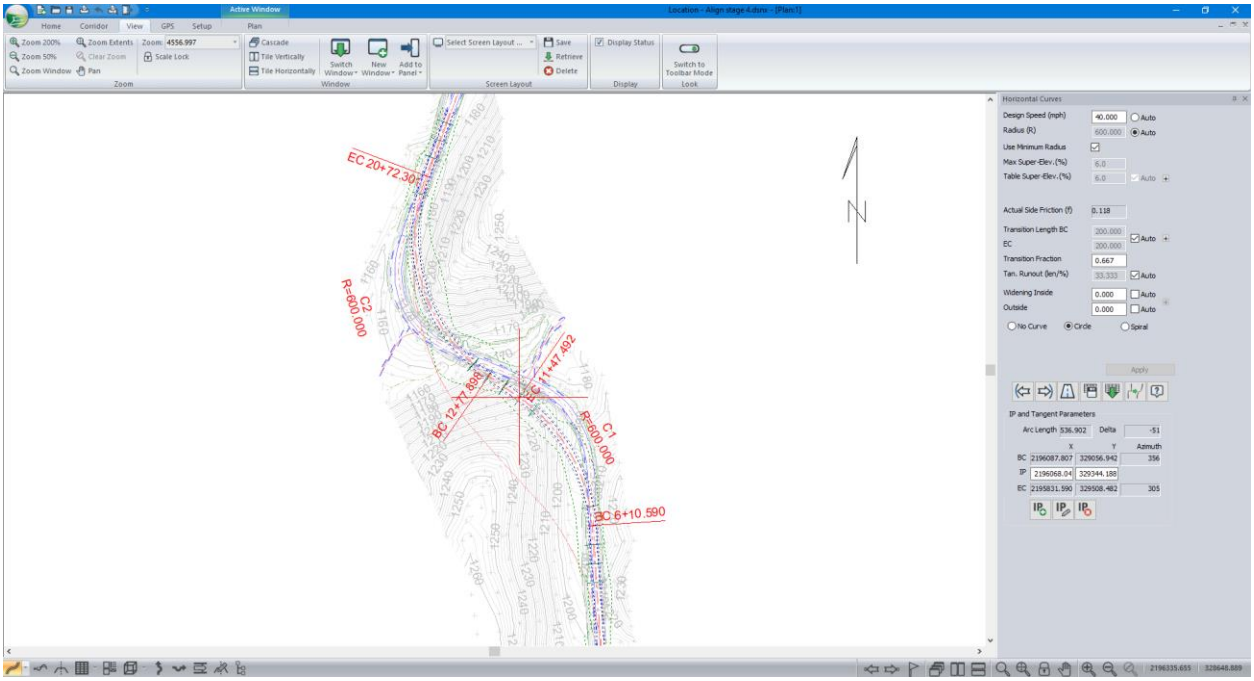

Figure 15-1: A Screen Layout with Docked Curve Panel and Maximized Plan Window

4. Open the *Plan Window Options* dialogue box. <*Right-click*> | *Plan Options…* and check the box next to *Background* display.
5. Press *Ok*. The Plan Window should be updated to look similar to the image above.
6. *Plan* | *Plan Options, click* the button beside *Report Points* to pop up the Report Point Properties dialogue box (figure below, center).

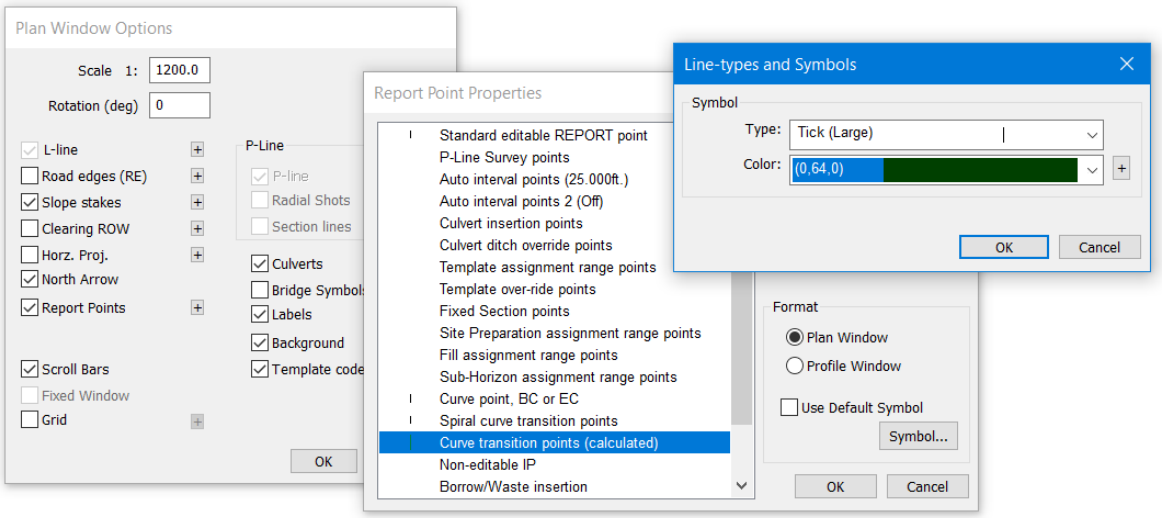

Figure 15-2: Plan Window Options, Report Point Properties and Symbol Formatting Dialogue Boxes

7. Select *Curve Transition Points (calculated)* in the list and then press the *Symbol…* button. If not already set, change the symbol to *Tick (Large)* and dark green (0,64,0) (as in figure above, right). Press *OK*, *OK*, and *OK* again to accept changes and close all dialogue boxes.

The changes you've made in the last few steps have changed the *look* of the screen but they have not modified 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.

8. Save the current configuration in a new screen layout:
   - *View* | *Save Screen Layout* button
   - This will open the default Settings and Layouts location. <*Double-click*> to open the Training folder.
   - Type **Training Test** in the *File name* field. Refer to figure below.

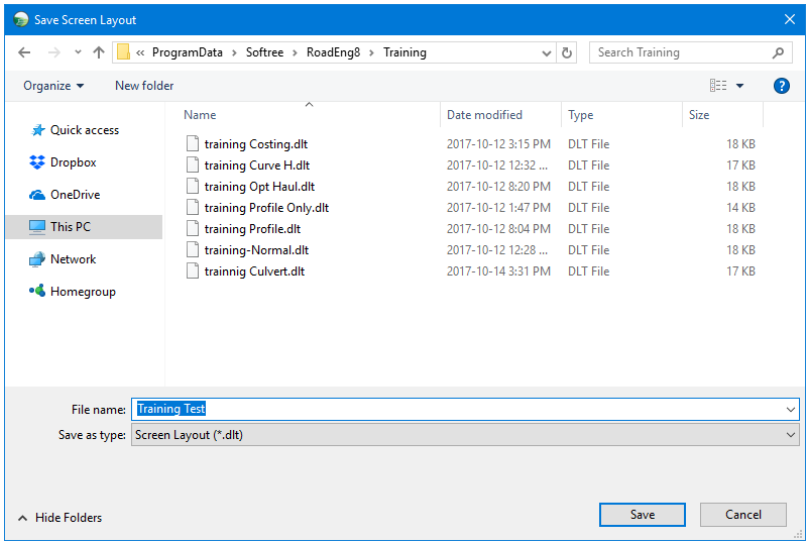

Figure 15-3: Saving a *Screen Layout*: View | Save Button

- Press *Save* button

9. You will be prompted with a reminder about save locations for screen layouts. Press *OK* to continue.
10. You have just created a screen layout that will appear in your Screen Layouts tool bar item for easy retrieval. **Training Test.dlt** is now in the dropdown menu within the training folder.
11. *View* | *Select Screen Layout* (dropdown) to retrive another screen layout: **training Curve H.dlt** (located in the *Training* folder).
12. Try opening some of the other screen layouts available.

Note: When accessing screen layouts or opening new windows (Profile, Plan, Section, Data, 3D, Multi-Plot) window can get lost behind one another or minimized. Using *View* | *Cascade* or *Tile Vertically* can be useful find open windows.

Screen Layout Facts

- The *Custom* folder is often defined on a network drive so that the layouts are accessible to all users.
- You can change the *Softree* folder (RoadEng Settings and Layouts folder) from the menu *Setup* | *Location 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 *Defaults and Layouts Folder* into your *Documents* folder (private to one user only).

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

16. Horizontal Curve Details

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.

If you are already familiar with the RoadEng help documents, you may wish to skip this exercise.

1. *Setup* | *Help* button or <*F1*> key for context sensitive help. You will be presented with the window shown below.

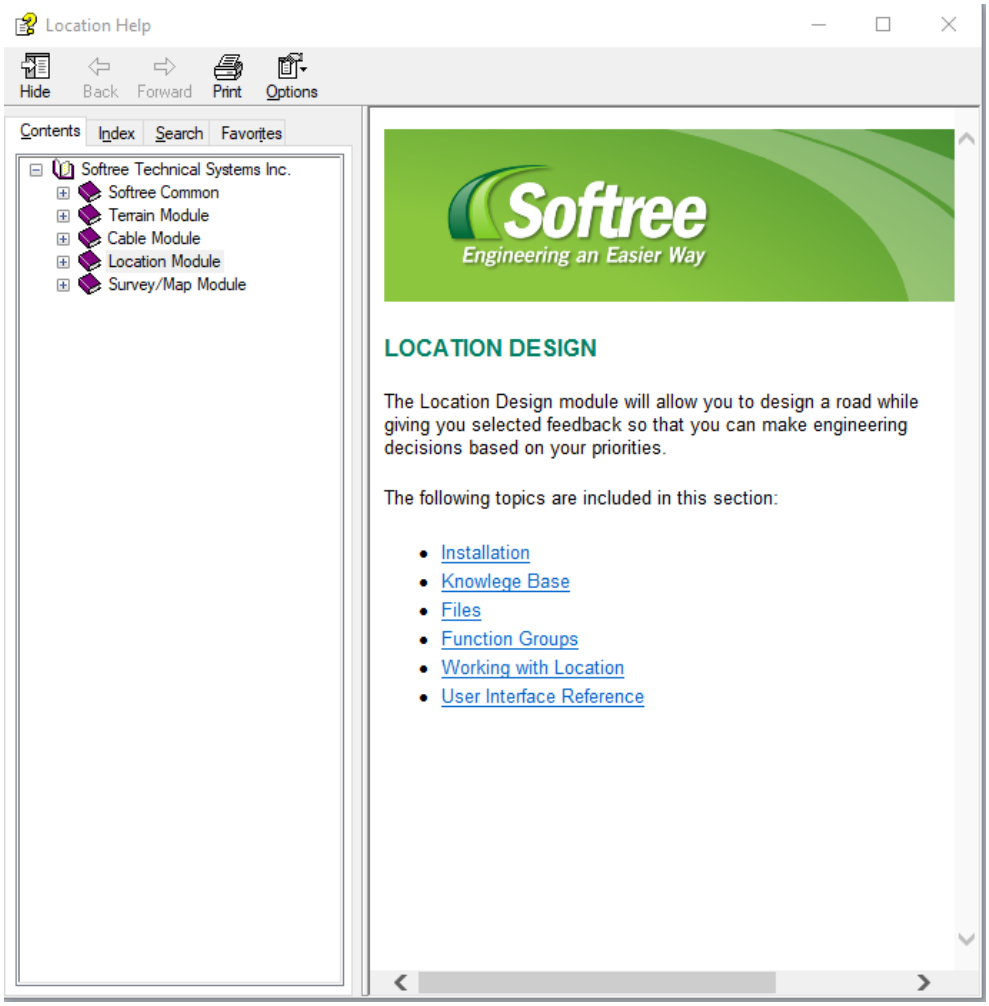

Figure 16-1: The Front Page for Location Help

The curve panels are not dialogue boxes (although they look like they are).

2. Select the *Search* and type "*curves*" into the text box and press *List Topics*. Select *Horizontal Curves Panel - Advanced Mode* | <*Double-click*> on this item or press *Display*. The help window should now look like the figure below.

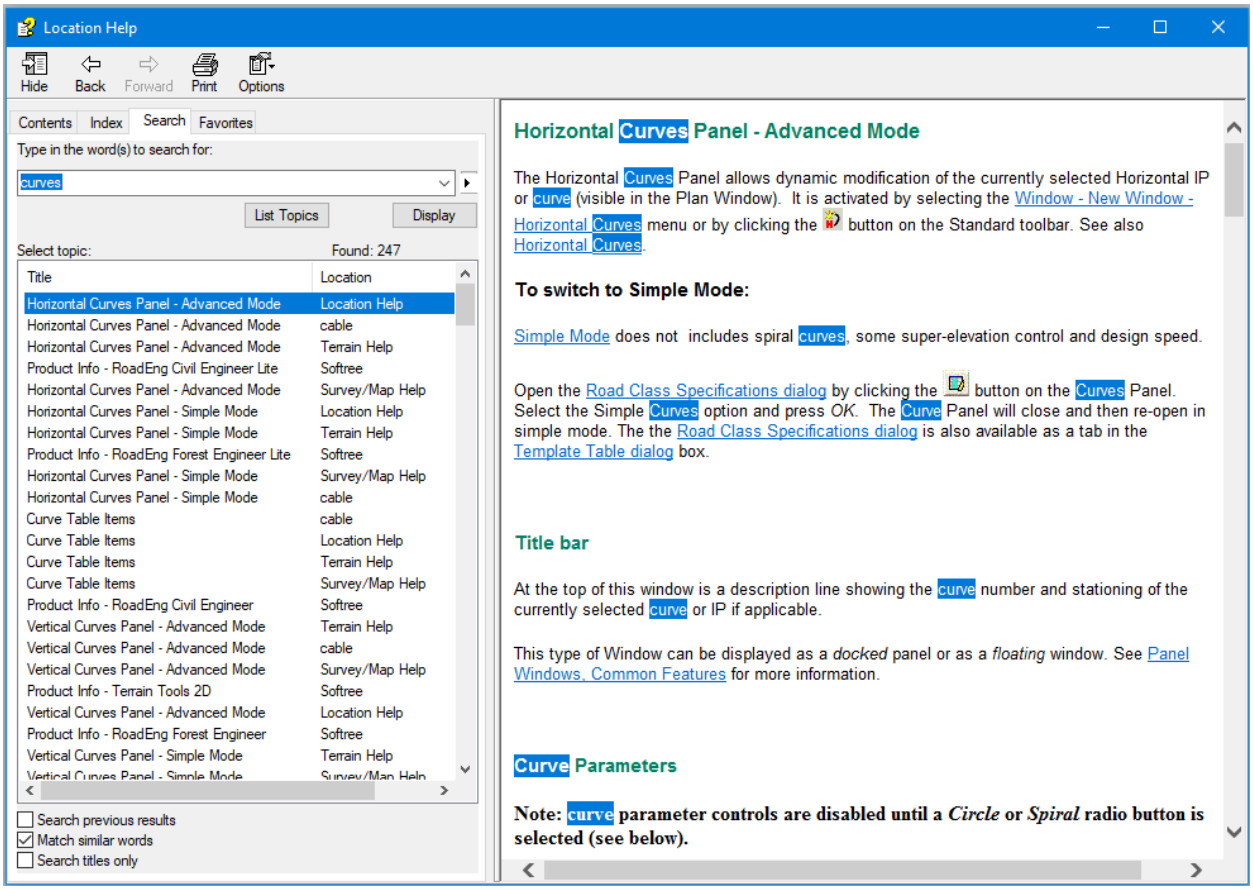

Figure 16-2: Horizontal Curves Panel - Advanced Mode Help Page

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

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

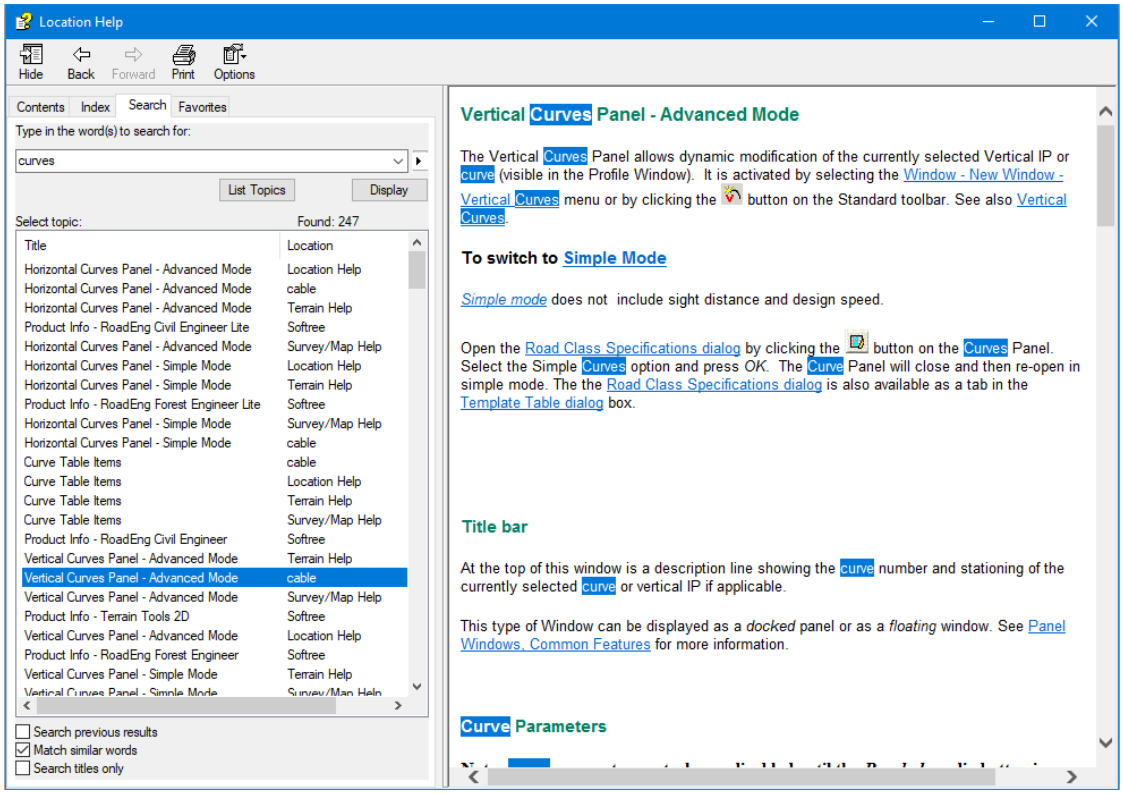

Figure 16-3: Help Window depicting the *Contents* Tree and *Vertical Curves Panel - Advanced Mode* Help Page

4. *Close* the Help window.

Horizontal Curve Panel

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

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 4.dsnx** in Location Module.
2. *View* | *Retrieve Screen Layout* <Defaults and Layouts>\training**training Curve H.dlt**.
3. If prompted by a *Recalculate Range* dialogue, press *OK* to proceed.

Your screen should look like the figure:

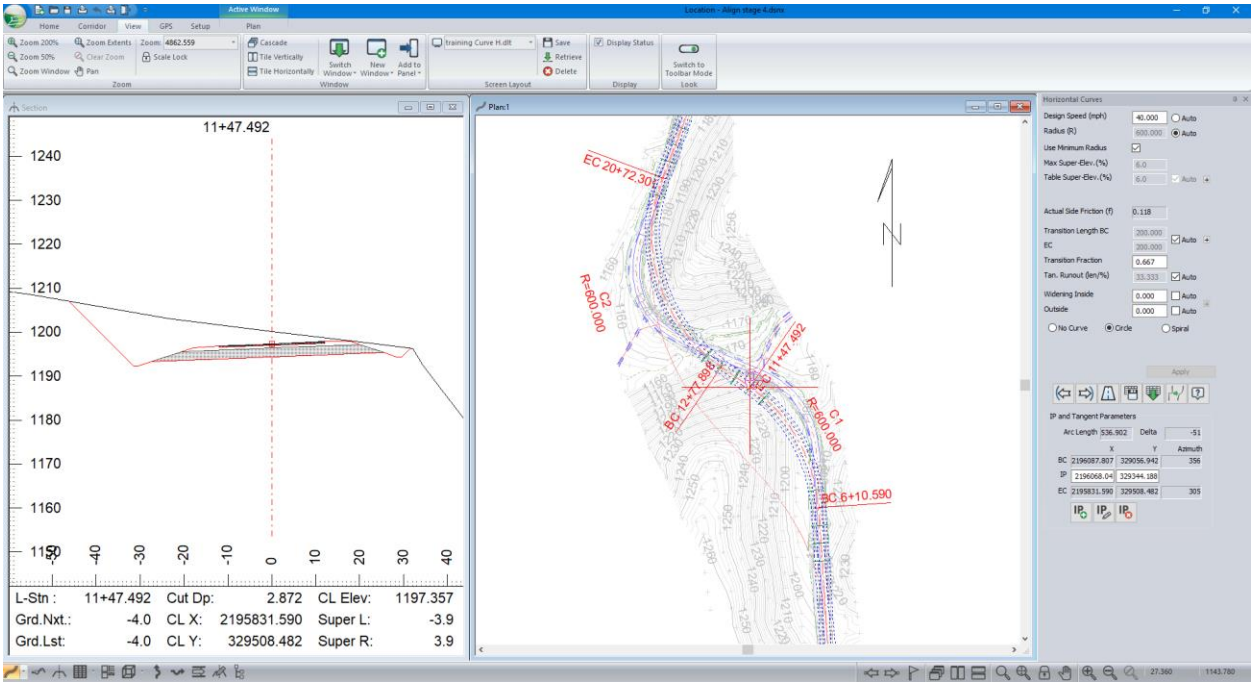

Figure 16-4: A *Screen Layout* with Docked Curve Panel, Plan Window and Section Window

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.

4. The first curve should already be current. If it is not, navigate  to the first curve.

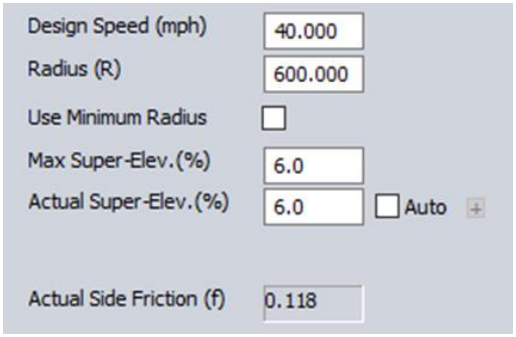

Figure 16-5: The Top Part of the Horizontal Curve Panel

5. Uncheck *Use Minimum Radius* box and then turn off the *Auto* check box (Figure 16-5). Note that you can then define the super-elevation for a given curve manually.

The *Actual Side Friction* 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.

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	0.07

Figure 16-6: Side friction factor for various speeds, curve radii and super-elevation values.

6. Turn the *Auto* check box back *ON* and press the *plus* button beside it to open the *Auto Super Elevation Options* dialogue box (Figure 16-7).

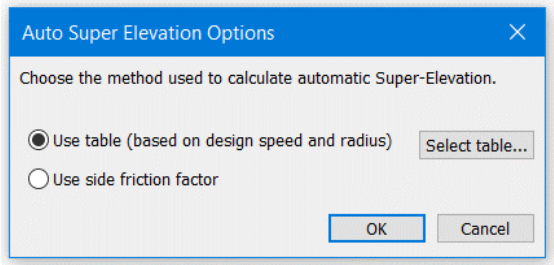

Figure 16-7: Auto Super Elevation Options Dialogue Box

The use of the *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.

7. Press the *Select Table* button to open the *Lookup Table* dialogue box (Figure 16-8).

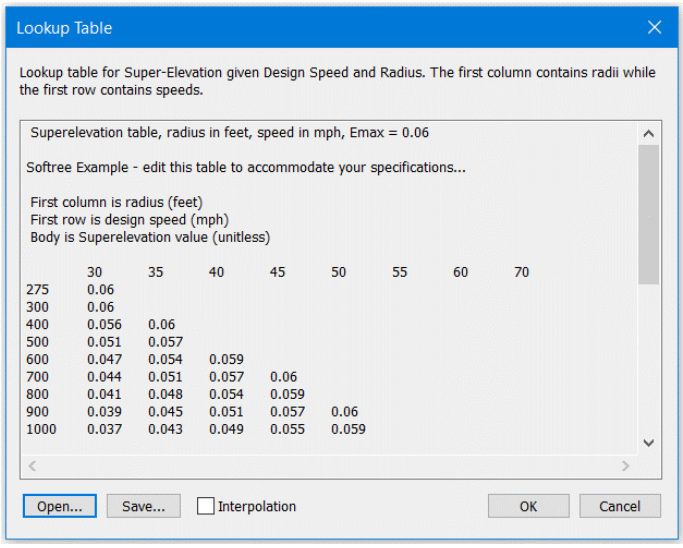

Figure 16-8: Super Elevation *Lookup Table* Dialogue Box

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. However, there are metric and English versions included in the RoadEng install (<Defaults and Layouts>***Superelevation EMax 06.tbl*** and ***Superelevation EMax 06 - Feet.tbl***, respectively).

Note: *.tbl files can be viewed and edited using notepad.

8. Press *Cancel* to close the table and *Cancel* again to close the *Auto Super Elevation Options* dialogue 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.

9. *Check* the Use Minimum Radius box and then check the *Auto* 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.

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

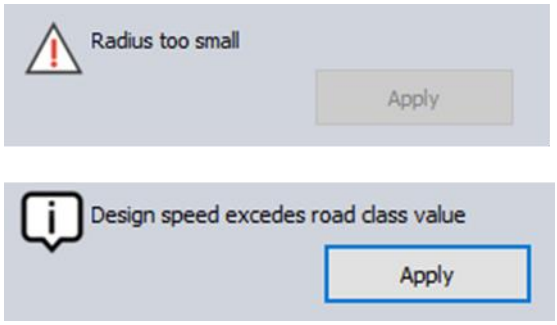

Figure 16-9: Curve Error Messages

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*.

11. Press the *Road Class Specifications* button  to open the dialogue, as shown in Figure 16-10 (below).

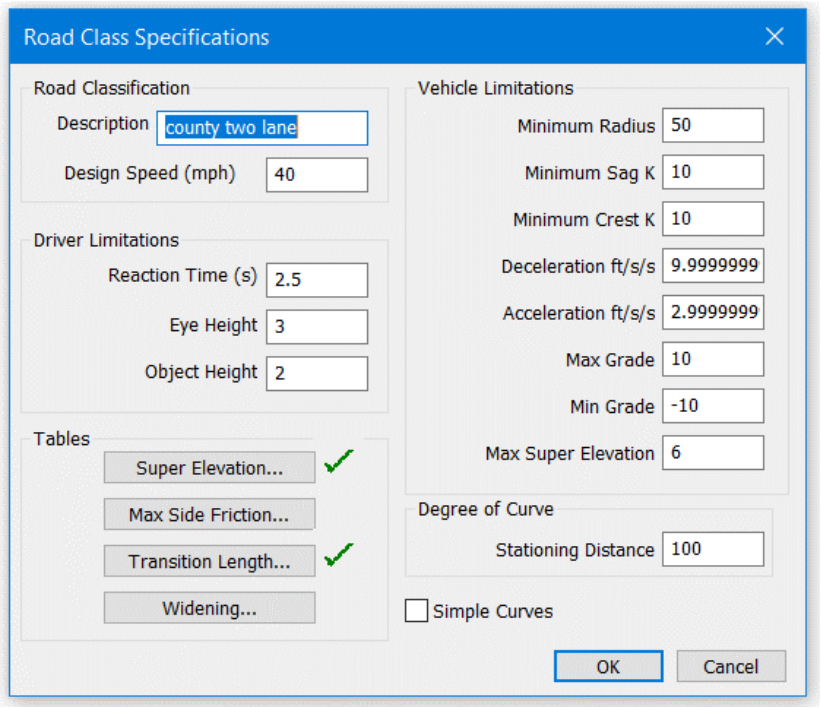

Figure 16-10: Road Class Specifications Dialogue Box

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. You can type <*F1*> for more information.

12. Press *Cancel* button to close the *Road Class Specifications* dialogue 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).

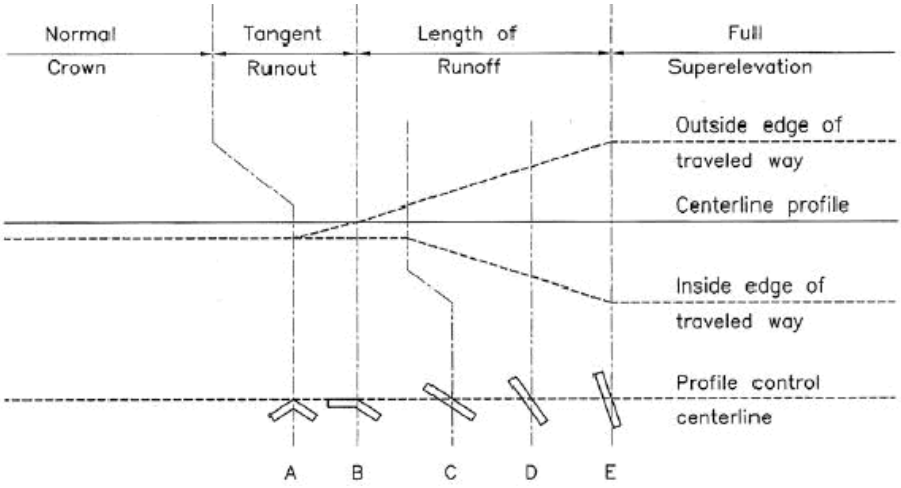

Figure 16-11: *Cross Fall Behavior* when Entering a Curve

13. 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), shown in the figure below:

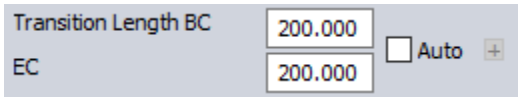

Figure 16-12: Manually Configuring Transition Length

14. Re-check the *Auto* box beside the two *Transition Length* values then press the associated *plus* button. The table shown below will be displayed.

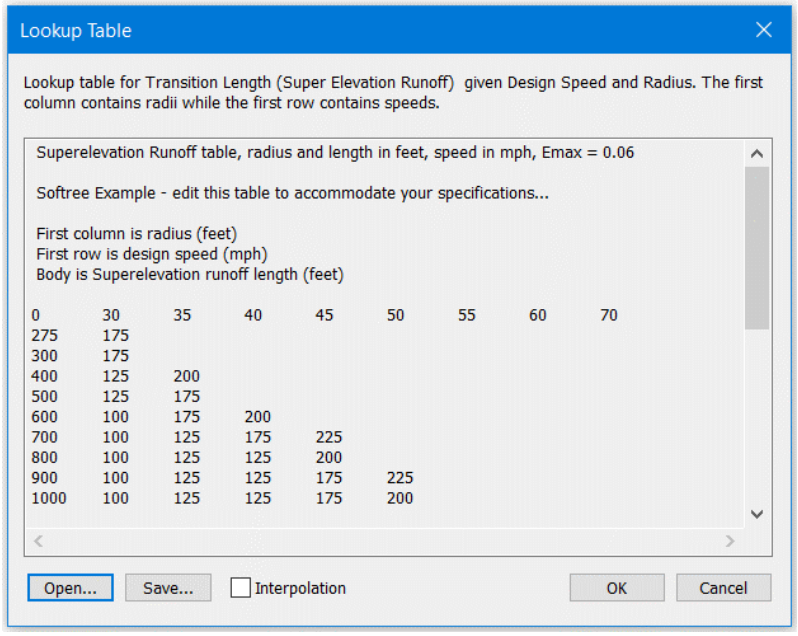

Figure 16-13: *Transition Length* Table.

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.

15. Press *Cancel* to close the Transition Length *Lookup 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 = 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 behavior figure above).

16. Clear the *Auto* check box associated with *Tan. Runout (len/%)*. Note that you can now enter a *length* manually.
17. Re-check the *Auto* check box associated with *Tan. Runout (length)*. 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: Your cross-section template must have curve widening built-in for these values to have any effect.

18. 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: <RoadEngCivil>\Location**WideningFeet.tbl**.
19. *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. *File* | *Open* to open <RoadEngCivil>\Location**Align stage 4.dsnx** in Location Module.
2. Use the Screen Layouts tool bar control to retrieve <Defaults and Layouts>\training**training Curve H.dlt**.

3. If prompted by a *Recalculate Range* dialogue, press *OK* to proceed.

4. Use the *next*  or *back*  buttons in the bottom navigation tool bar (NOT in the curve panel) to move the current point to the tangent before the first curve (figure below). This is before station 4+10.

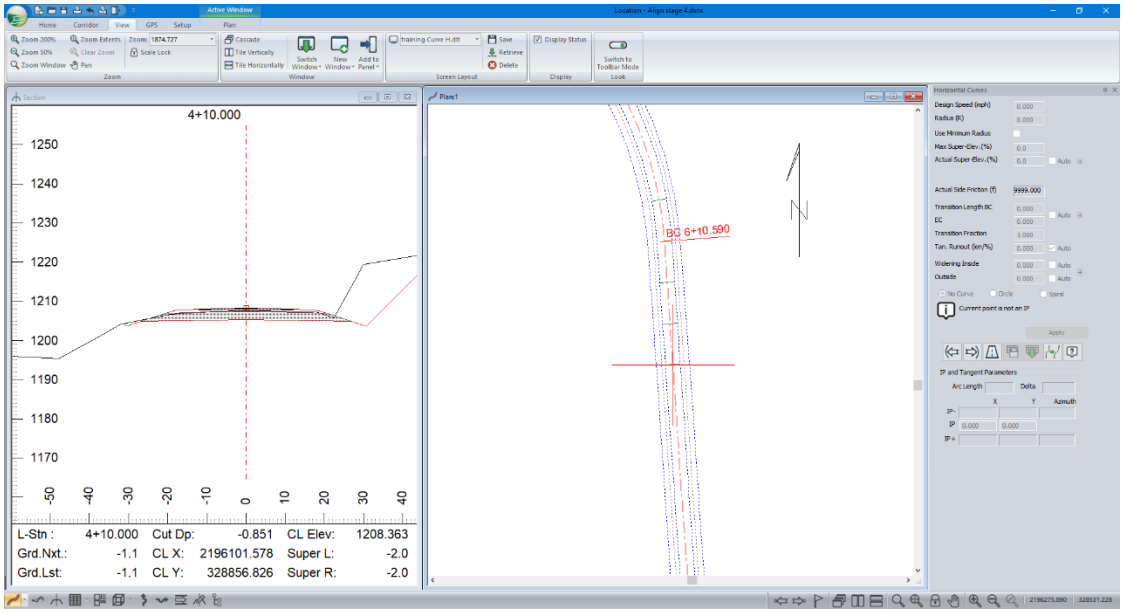

Figure 16-14: Section Window Text Area Depicts Super-Elevation While the Plan Window has a Curve Transition Report Points Displaying (Green Tick)

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.

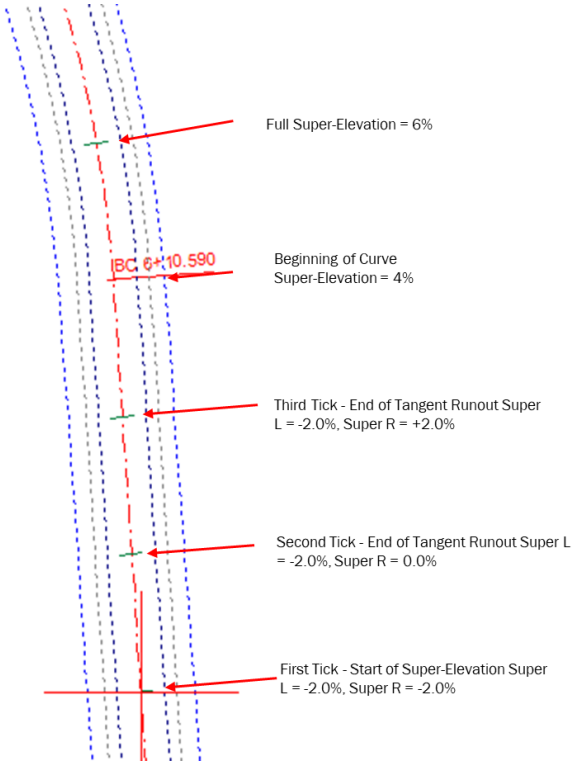

Figure 16-15: Plan Window Curve Transition Report Points Displayed (Green Tick)

5. Use the *next* button to move the current point to the first (southmost) tick.

The cross section shows full crown with super slope of 2% (Super L = -2.0%, Super R = -2.0%).

6. 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.

7. 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.

8. 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).
9. Continue clicking the *next* button until you get to next tick mark. This is full super-elevation (6%).
10. Use the *Horizontal Curve Panel* to navigate to the end of curve 1. Change the *Transition Fraction* to **0.5** and re-apply the curve. Note how transition points move.

Curve Transition Overlap

What happens between the two curves?

11. Use the *next* button 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).

12. Bring the Profile window to the front and maximize.

The sub-window has been configured to show the super elevation (refer to figure next page).

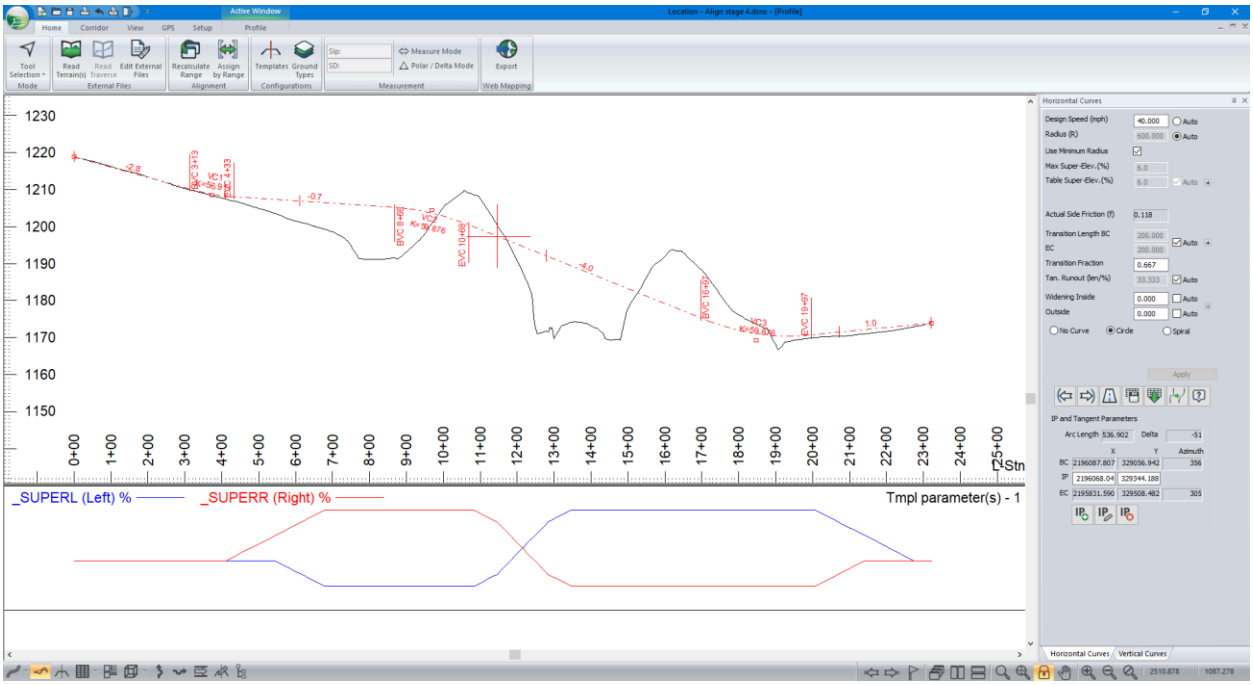

Figure 16-16: Profile *Template Parameters* Sub-Window with Super Elevation Displayed.

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

The super elevation graph is useful to view transition of super elevation. To display graph: *<Right-click>* in profile window, select *Profile Options.* Press the *Sub Windows Select…* button, select *Tmpl parameter(s) – 1.*

14. *File | Close*. Do not save changes.

17. Vertical Curve Details

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

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

1. In Location, *File*| *Open* <RoadEngCivil>\Location**Align stage 4.dsnx**.
2. Use the Screen Layouts tool bar control to retrieve <Defaults and Layouts>\training**training Curve V.dlt**.

Your screen should look like Figure 17-1.

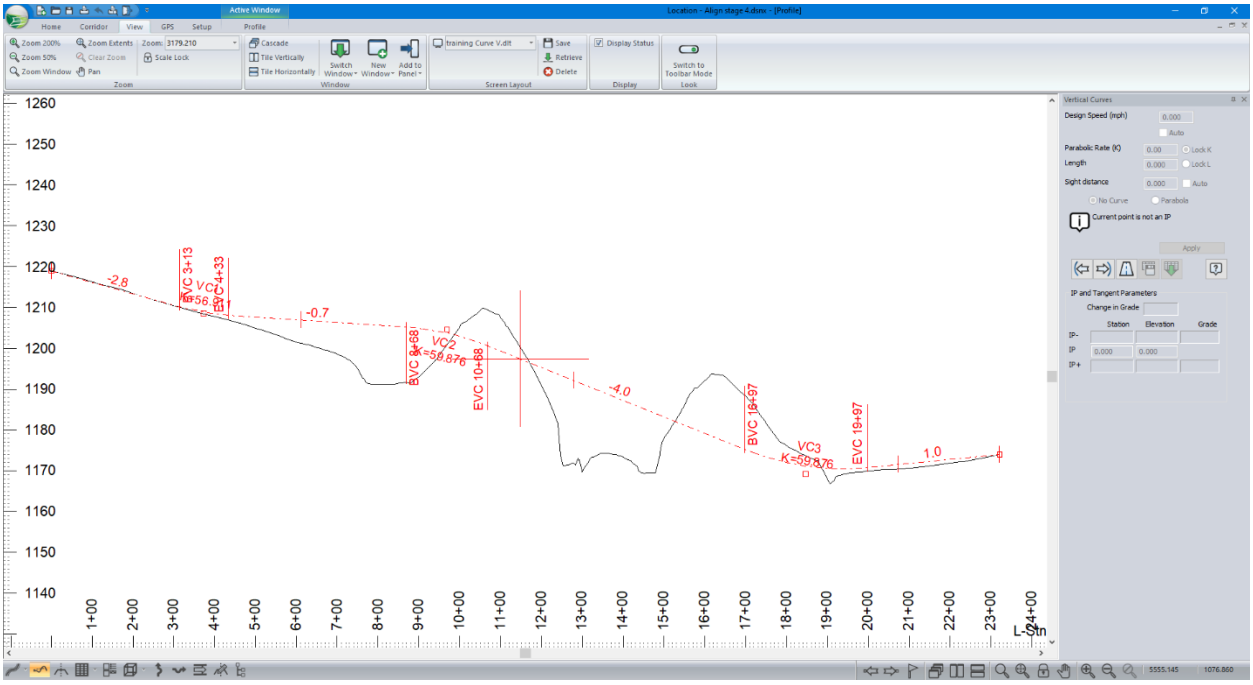

Figure 17-1: *Vertical Alignment* with Docked Vertical Curve Panel

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

Automatic Curvature

4. This curve has been configured to find the smallest possible curve that has a safe sight stopping distance. Ensure the *Auto* check box for *Sight distance* is enabled and *check* Parabola button.
5. Change the Design Speed to **30**. Notice how the sight stopping distance drops, as does K and the length of curve.
6. Set the Design Speed back to **40**.
7. Press the *Road Class Specifications* button to open the dialogue box shown below.

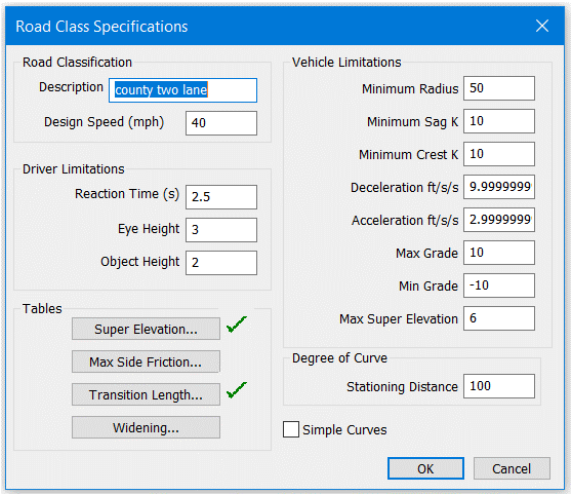

Figure 17-2: *Road Class Specifications* Dialogue Box

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

Note: You can type <*F1*> for more information.

8. Press *Cancel* to close the *Road Class Specifications* dialogue box.
9. <*Right-click*> in profile window, select *Add/Edit IP Tool*>move cursor to *capture* the *vertical intersection point* for this curve. Move it up and down.

Notice how the values in the curve panel are kept up to date. With the curvature set to automatic, both the length and the curvature (K) change as the VIP is moved.

Locked K

10. Clear the *Auto* check box below the *design speed*. Make sure the *Lock K* button is selected and press *Apply*.
11. Repeat the experiment from step 9. 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

12. Select the *Lock L* 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.
13. *File* | *close*. Do not save changes.

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.

1. *File* button | *Open* <RoadEngCivil>\Location**Align stage 4.dsnx**.
2. Retrieve screen layout <Defaults and Layouts>\training**training Curve V.dlt**.
3. Use the *Previous* IP and *Next* IP buttons to move to the second vertical curve.

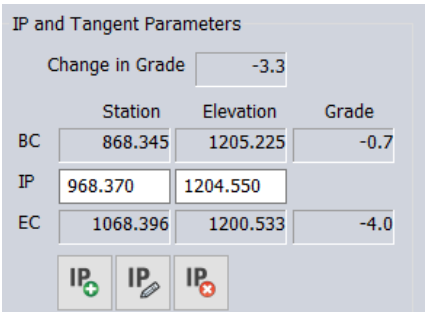

Figure 17-3 : The Bottom Part of The *Vertical Curve Panel*

4. 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.

5. Press the *Modify IP* button to open the *Modify Vertical IP* dialogue box (Figure 17-4).

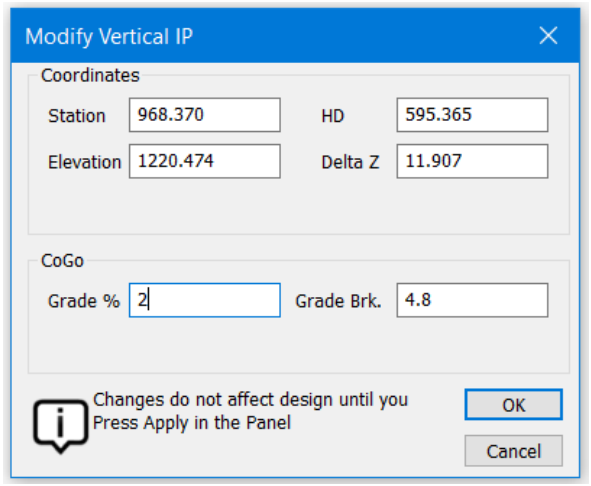

Figure 17-4: The *Modified Vertical IP* Dialogue Box Allows the User to Set the Grade of the Previous Tangent (Among Other Things)

6. Type "**2**" into the *Grade %* field (Figure 17-4) and press *OK* to close the dialogue box.

Your Elevation value in the curve panel has been updated.

7. Press *Apply* in vertical curve panel to change the curve.

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

8. *File* | *Close*. Do not save changes.

18. Materials and Stripping

So far, our "Align Stage" 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. *File* | *Open* <RoadEngCivil>\Location**Align stage 4.dsnx**.
2. Select *Home* | *Ground Types*.
3. Create a new ground type for use in subgrade fill:
    - Select the **FR Fractured Rock** item.
    - Press the *New...* button to open the *New Ground Type* dialogue box (figure below right). Note that we start with a copy of the selected item.
    - Fill in the ID *RR* and Description *Rip Rap*.
    - Press *OK*.

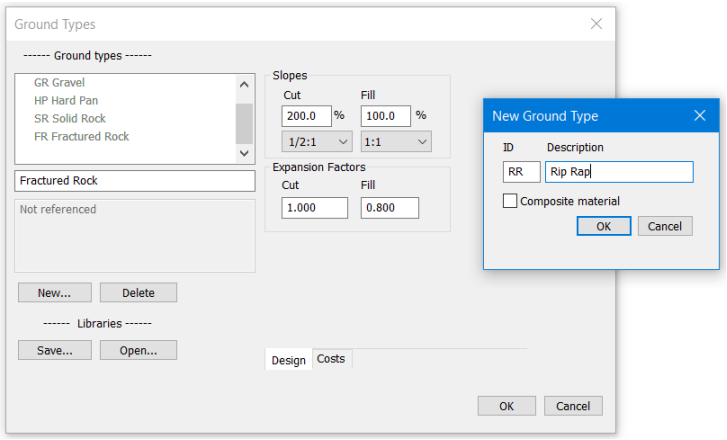

Figure 18-1: The Available Ground and Subgrade Fill Materials in the *Ground Types Editor*.

Note: The *Composite Material* option can be used for uncertain or mixed materials found in the original ground – F1 for help.

Note that the RR fill angle is set to *100% (1:1);* this is suitable for our purposes, so there is no need to change it. The cut angle is not important as we will not assign this material as native ground.

4. Create another material called *CS-Clay Silt* with *cut slope* of *1:1* and *fill slope 3:1*.

Note: The default ground table is called **NORMAL.GDX**.

5. Press *OK* to accept changes and close the *Ground Types Editor*. Respond *Cancel* to the recalculate prompt (we didn't change anything that is in use).

Assign original ground materials to a station range:

6. *Home | Assign by Range*
7. Select the *Sub Horizons* tab (Figure 18-2 below left).

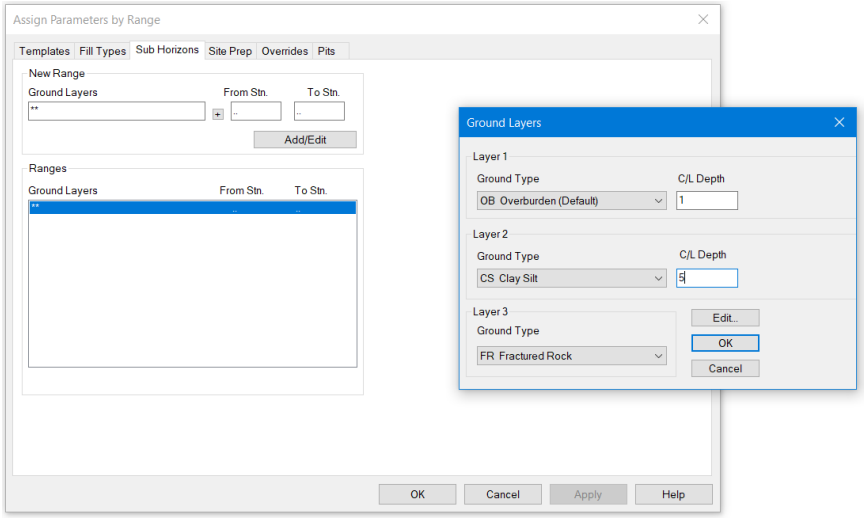

Figure 18-2: *Defining Materials* in The Original Ground.

8. Press the ⊞ button beside the *Ground Layers* field to open the *Ground Layers* dialogue box (Figure 18-2 above right).

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 like create enough subsurface points to make surface models.

9. Select the three subsurface layers shown above (*OB*, *CS*, *FR*) and then set the layer depths (**1**, **5**).
10. Press *OK* to close the dialogue box and fill the *Ground Layers* field in the *Sub-Horizons* dialogue box.

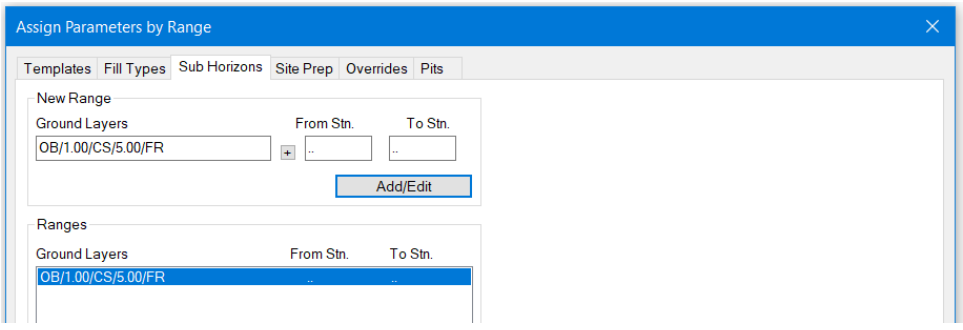

Figure 18-3: *Sub-Horizons* Have Been Applied to The Entire Alignment

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 dialogue 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 dialogue box.
13. Respond *OK* to the *Re-calculate Range* prompt.
14. Select a cross section that has some cut in it (ie stn 17+25). Notice that the new ground layers are displayed.

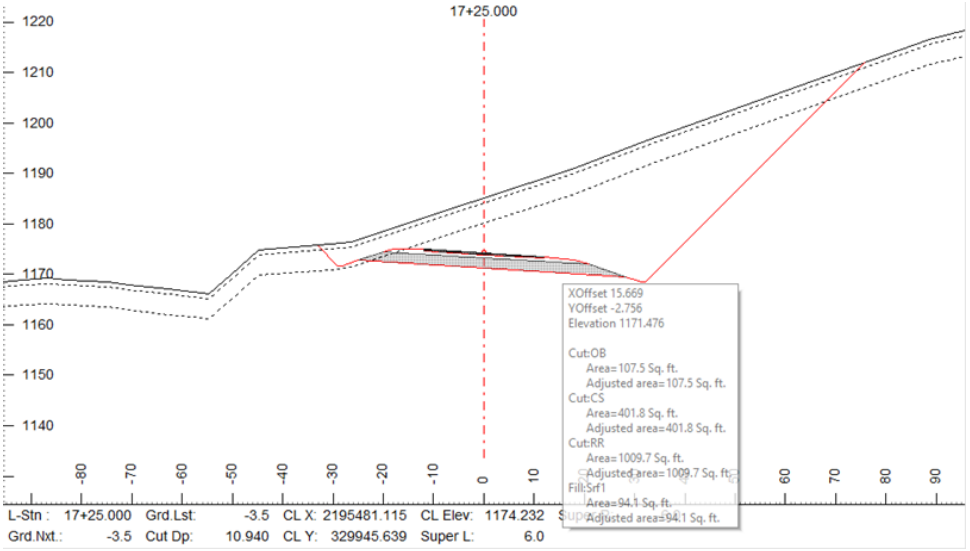

Figure 18-4: *Ground Layers* Depicted in The Section Window.

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.

15. *File* | *Close*. Do not save changes.

Stripping

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

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 5.dsnx**.
2. *Home* | *Assign by Range*. Select the *Site Prep* tab (figure below).

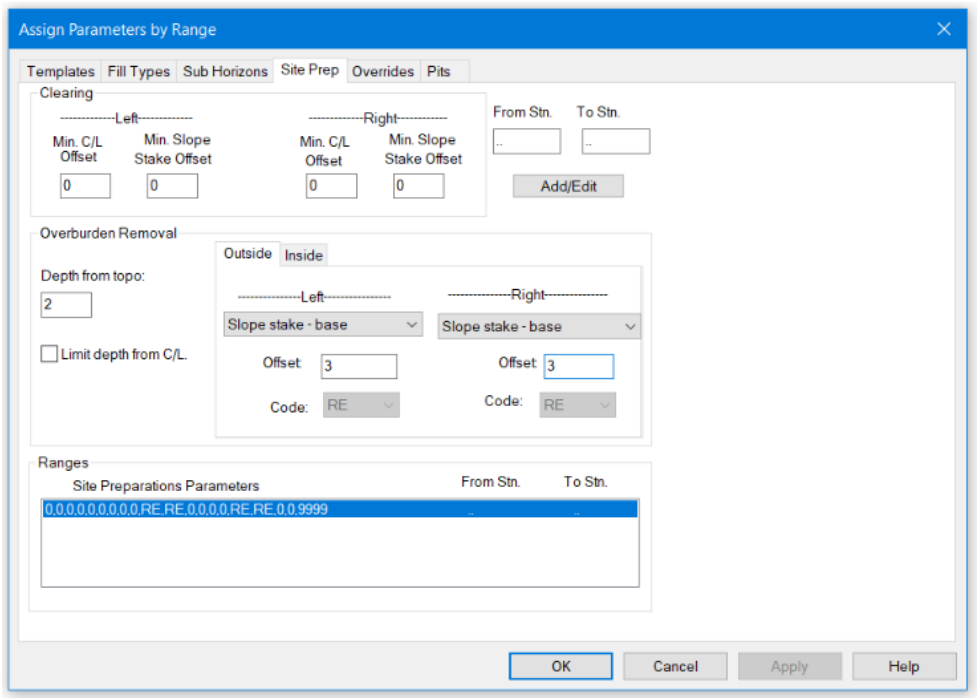

Figure 18-5: *Site Preparation* Dialogue Set Up for Stripping
(It also can control clearing offsets)

3. In the *Overburden Removal* area, set the Depth from topo to **2**. Leave the default zero offsets in the *Inside* tab.
4. Set the *Outside* controls to *Slope stake – base*, Offset: **3** feet (both sides as in Figure 18-6). This will strip 3 feet outside the template footprint.
5. Press the *Add/Edit* button under the From Stn. To Stn. boxes.
6. Press *OK* to accept changes and close the dialogue box. Respond *OK* to the re-calculate prompt.
7. Zoom in on the cross section left or right-hand side. Notice that the stripping line is displayed (Figure 18-6).

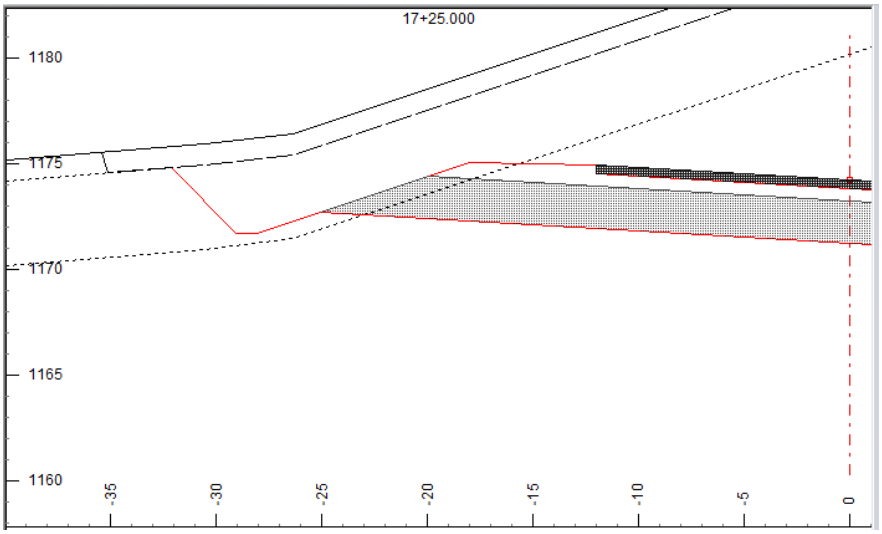

Figure 18-6: The Top Ground Layer has to been *Stripped* Off

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 dialogue 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.

8. *File* | *Close*. Do not save changes.

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.

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 6.dsnx**.
2. *Home* | *Assign by Range.* Select the *Templates tab* (Figure 18-7).

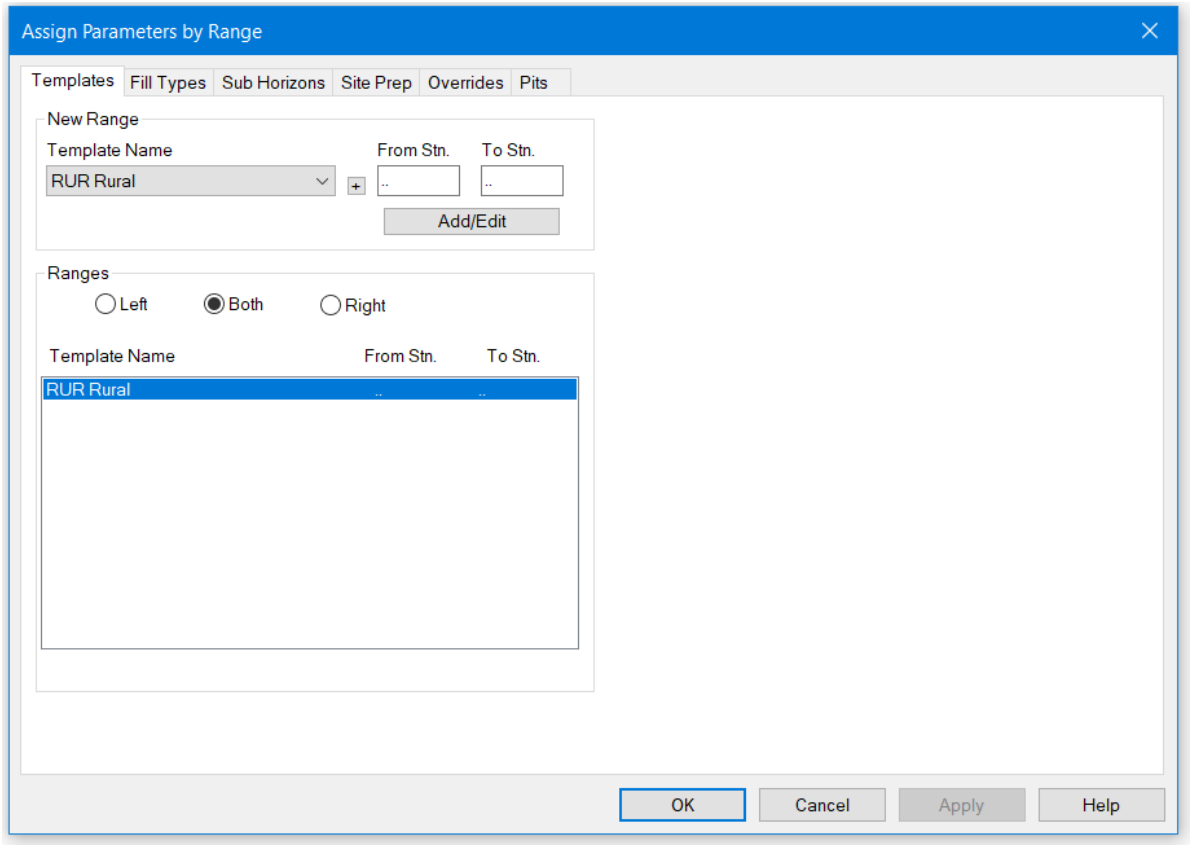

Figure 18-7: Assigning the RUR Rural Template to The Entire Alignment.

3. In *Template Name*, choose *RUR Rural*. Leave the default ".." in the From/To fields.
4. Press the *Add/Edit* button, this will apply the new template to the entire alignment. The dialogue box should appear as in the figure above.

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

5. Press *OK* to return to the main screen. Respond *OK* to "Recalculate road alignment".
6. Click on the section window window to select it. *Section | Jump to Station* type in <*Ctrl-J*> and type station **1675**. Press *OK* to update the current section.

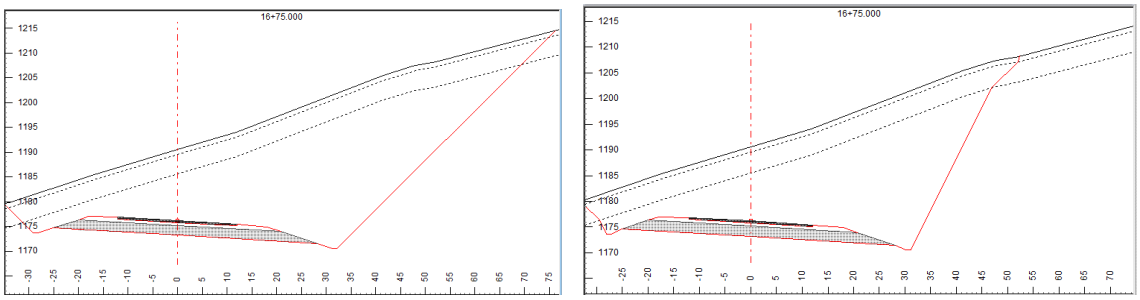

Figure 18.8: Template on The Left has Fixed Slope Values, while The One on The Right Has Slopes Set to *Auto*

7. 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.

8. *File* | *Close*. Do not save changes.

19. 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.

Note: Refer to *Getting Started* section for file install folders (<RoadEngCivil> and <Defaults and Layouts>).

1. *File* | *Open* <RoadEngCivil>\Location2**bluff_road.dsnx**.

Creating a New Template

2. Select *Home* |*Templates*, to open the template table editor.
3. *<Right-click>* RUR-Rural and select menu *Copy*.
4. *<Right-click>* Templates 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 type in **BAR** in *Name* and **Rural Paved with Barrier** in *Description*. Press *OK*.

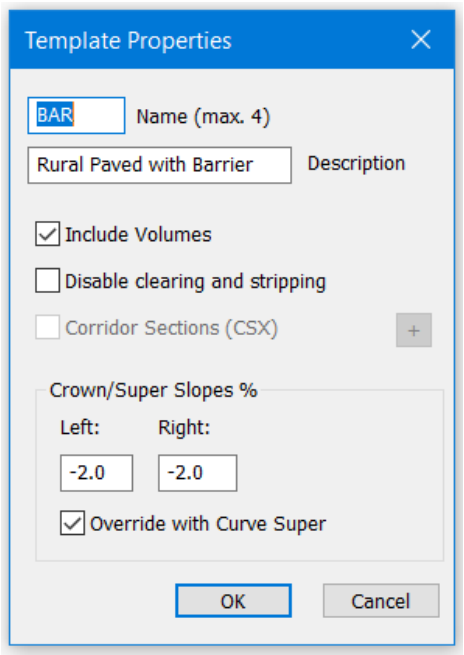

Figure 19-1: *Template Properties*

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

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 Barrier-left component *Properties…* | change *CL_OFFSET* parameter to **15**. Press *OK* to exit the properties dialogue box. Your template should now appear as in the figure below:

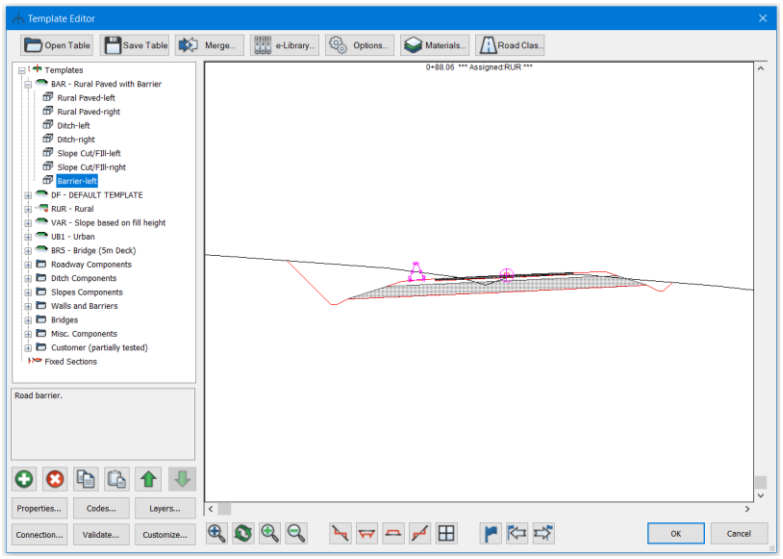

Figure 19-2: Template with *Barrier*

10. To make this template more useful, we will put the barrier on both sides. <*Right-click*> on Barrier-left and *Copy it* to the clipboard. <*Right-click*> again on Barrier-left and select menu *Paste | As New- Right*.
11. Press *OK* to accept the changes and close the template editor. It is all right to respond *Cancel* to the recalculate prompt because the new template has not been assigned yet.

Assigning the Template

12. If you had problems with the previous steps or if you wish to start here, open <RoadEngCivil>\Location2**bluff_road-2.dsnx**.
13. *Home | Assign by Range* to open the *Assign Parameters by Range* dialogue box (Figure 19-3). Select the *Templates* tab.

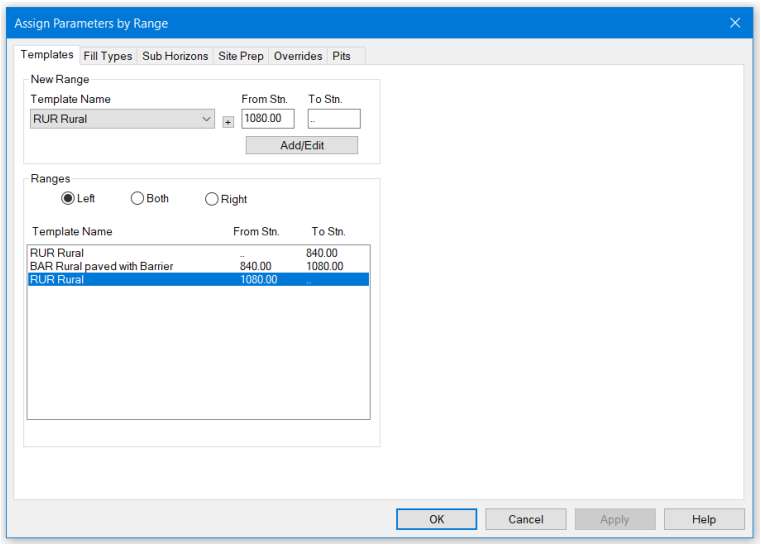

Figure 19-3: *Assigning a Template* to a Station Range

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

14. In the *Ranges* area, select *Left*. Do this first because it resets the template name and range fields:
    - 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**.
    - Press the *Add/Edit* button. The dialogue box should appear as in the figure above.

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

15. Press *OK* to return to the main screen. Respond *OK* to "*Recalculate road alignment*".
16. Select the section window *Section | Jump to Station* or *<Ctrl-J>Jump to Station* 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.

17. *File | Close*. Do not save changes.

20. 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.

Note: Refer to *Getting Started* section for file install folders (<RoadEngCivil> and <Defaults and Layouts>).

1. *File* | *Open* <RoadEngCivil>\Location2**bluff_road.dsnx**.
2. *Home* | *Assign by Range* | Select the *Overrides* tab.

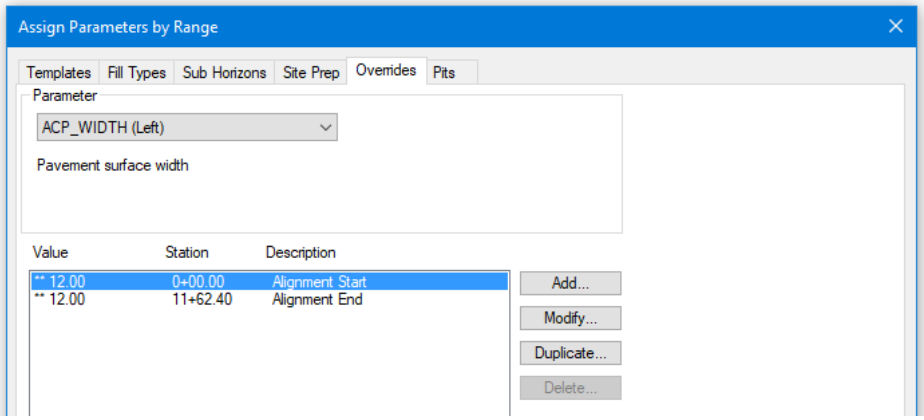

Figure 20-1: The *Assign Parameters By Range* with Overrides Tab Selected

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 ***30.0*** (figure below left). Press *OK*.
5. Press *Add…* another Value of *30* at Station *110*. Also change the Description to read "*Turn lane end*" (figure below right). Press *OK*.

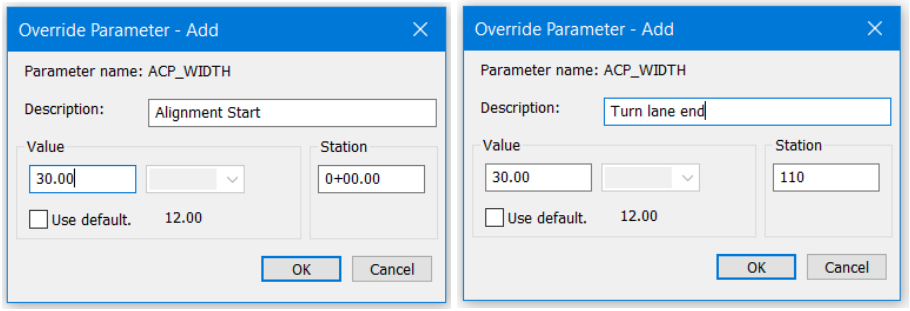

Figure 20-2: *Override Parameter* Dialogue

6. Press *Duplicate…* and *check* 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.

Note: Without adding the taper at a specific station it will be difficult to see the transition in width of the road.

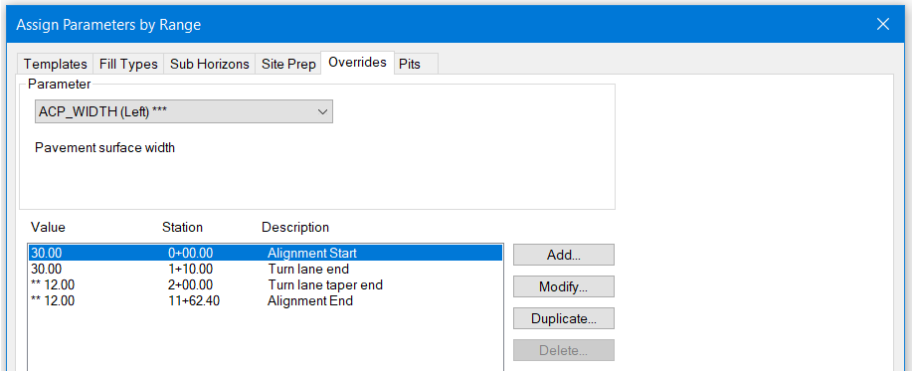

Figure 20-3: *Override List* for Turning Lane at Start of Alignment

7. Press *OK* to return to the main screen. Respond *OK* 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.

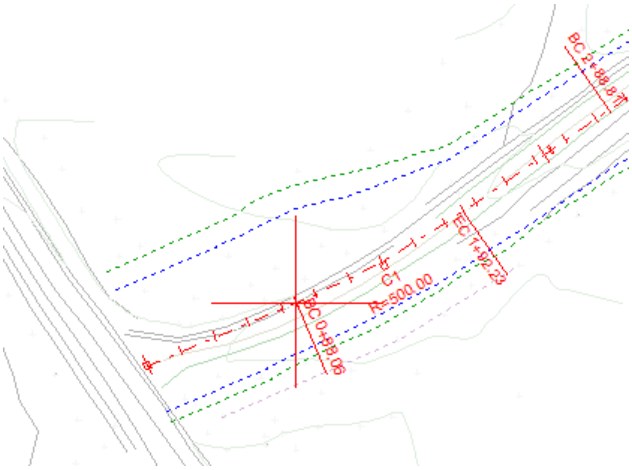

Figure 20-4: *Turning Lane* Defined by Template Parameter Overrides.

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

21. Templates – Display and Reporting

Before we begin some basic concepts and definitions are required.

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.

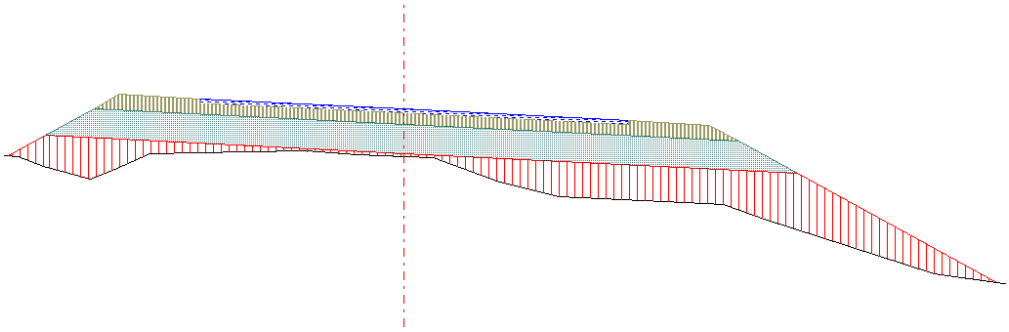

Figure 21-1: *Template Surfaces* and Enclosed Materials

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.

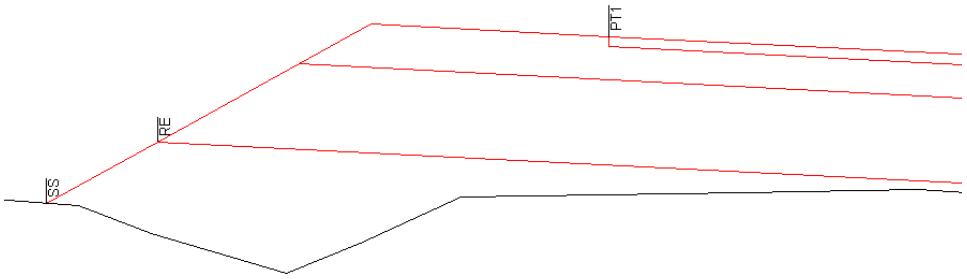

Figure 21-2: *Template Codes*

Display and Reporting of Template Layers

Formatting Template Layers

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

1. *File* | *Open* <RoadEngCivil>\Location2**bluff_road.dsnx**.
2. Maximize the Section window and zoom in.
3. <*Right-click*> in the Section window and choose *Section Options…*
4. Click on the ⊞ button beside the *Template* check box to open the Template Display Format dialogue box (Figure 21-3).

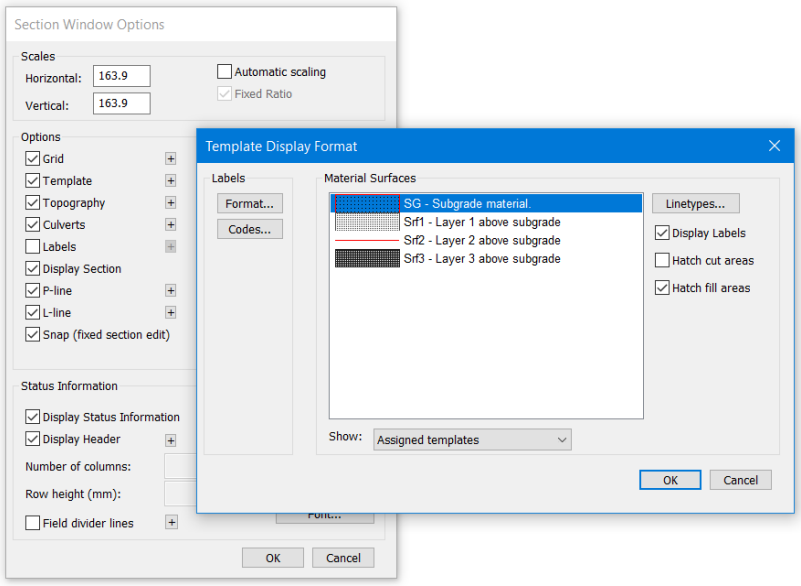

Figure 21-3: Section Window Options and The *Template Display Format* Dialogue

5. Select the first item (*SG – Subgrade material*) and set the *Display Labels* check box (as shown above).

6. Select each of the other layers and note that *Display Labels* is not checked. Only template items from the Subgrade layer will display labels.
7. Select SG, in the *Labels* section on the left-side of the dialogue box, press the *Format…* button.

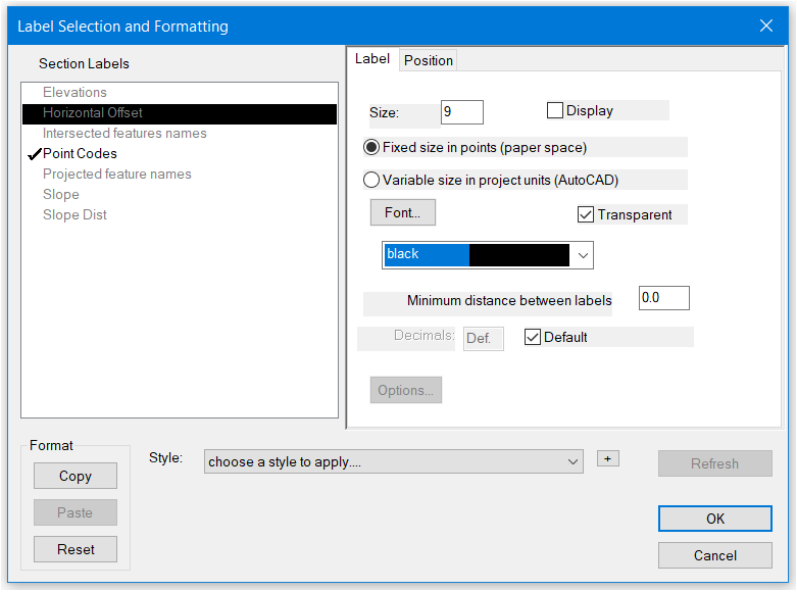

Figure 21-4: *Label Selection* and *Formatting* Dialogue Box

8. Ensure that only *Point Codes* labels are displayed as shown in the figure above. You can also change label font, color and position in this dialogue box. Press *OK* to close the dialogue box.
9. Back in the *Template Display Format* dialogue, press the *Linetypes…* button on the right-hand side to open the *Line-types and Symbols* dialogue box (Figure 21-5).

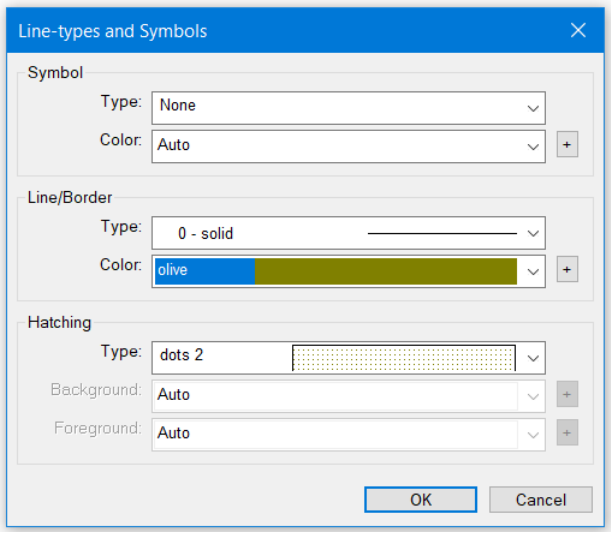

Figure 21-5: The *Line-types and Symbols* Dialogue Box Allows the user to Change Line Style (including symbols), Hatch Style and Color

10. Change the color of the subgrade to *olive* (figure above) and press *OK* to accept changes and close the dialogue box.
11. Press *OK* to return to Section Window Options.
12. Ensure the *Labels* check box is checked; our code labels will not display unless we turn on this master switch.
13. Press *OK* to return to the main screen.

You should see the template point code labels for the subgrade surface (Figure 21-6).

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.

14. Move your mouse over a template point that is not displayed (for example the shoulder edge) and hold your mouse there (hover). You will see a tool tip displaying the point code along with some other information (Figure 21-6).

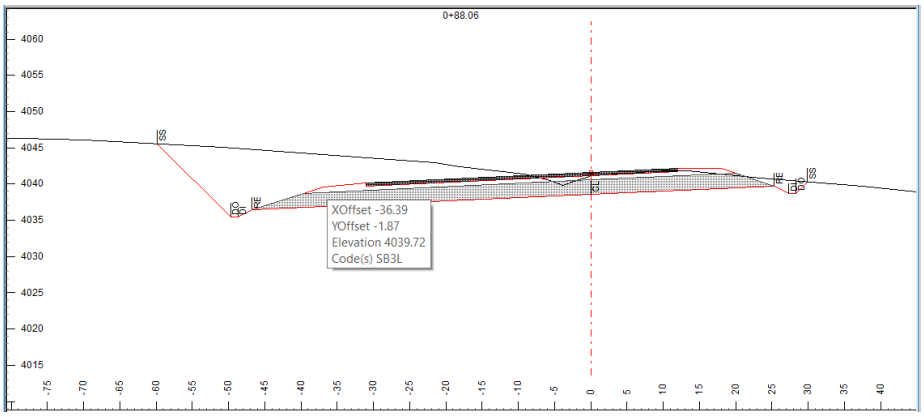

Figure 21-6: *Hover Tips* in The Section Window

15. Try hovering in other places and see what information appears in the tip. You can display:
    - Mouse elevation, vertical and horizontal offsets
    - Cut and fill material
    - Cross sectional areas
    - Template leg length and slope
    - Point codes
16. <*Right-click*> in the ditch cut area (above the ditch bottom and below the original ground) and select *Hatch Cut Area for SG* from the menu. This shows the olive hatching you defined above.
17. <*Right-click*> in the same place to turn off the subgrade hatching.

Display and Reporting of Ditch Lines

Reporting template point codes

In the exercise above we displayed the same point code labels in the section window. Point codes can also 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.

18. If you are not continuing from above, open <RoadEngCivil>\Location2**bluff_road.dsnx.**
19. If you are continuing from above, restore the Section window so the Plan window is visible again.
20. <*Right-click*> in the Plan window and select *Plan Options…*
21. Ensure that the *Template Codes* option is selected; press the button beside it to open the *Codes* dialogue box.

Notice that several point codes are already shown in the *Code* list. PT1 is the pavement edge, SB3 is the shoulder edge; L and R designate Left and Right.

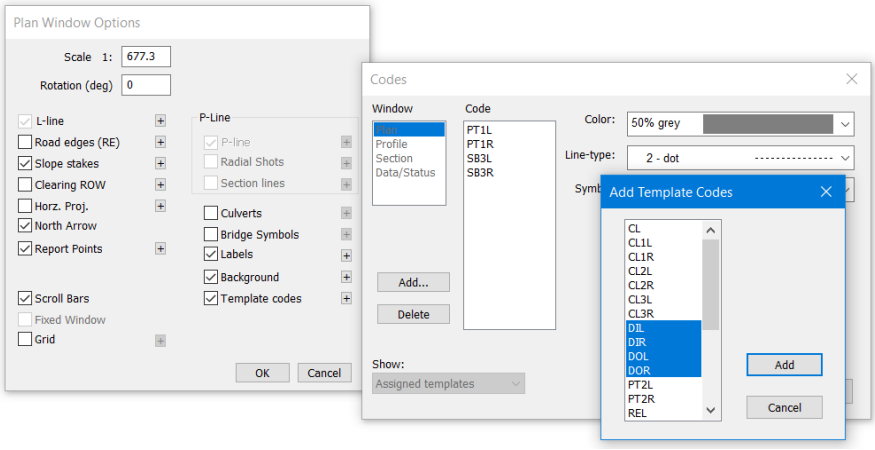

Figure 21-7: *Adding Template Codes* for Display in the Plan Window

22. Click the *Add…* button and select all the ditch point codes: DIL, DIR, DOL, DOR (see figure above). Click *Add* to close the selection dialogue box.
23. With the new codes still selected, choose a blue dash line as shown below.

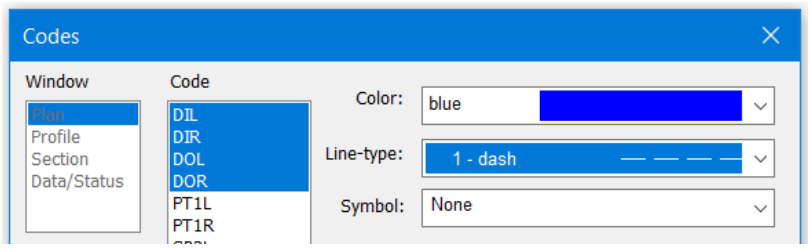

Figure 21-8: *Plan Template Codes* Format Control

24. Press *OK* to close the *Codes* dialogue box and respond *OK* to *Recalculate Range*.

25. Press *OK* again to close the *Plan Window Options* dialogue.
26. Adjust the Plan Window view so you can see the new ditch lines.

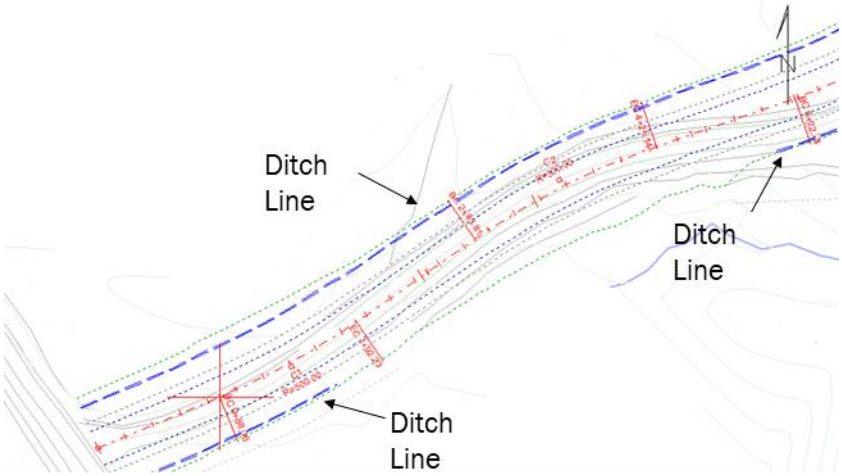

Figure 21-9: *Ditch lines* in Plan View

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

27. Next, we will set up the data window. With the Data Window active, *Data | Data Options*.
28. First set up the desired spacing of data rows:
    - *Click* the [+] button next to *Point Types* to open the *Point Type Selection* dialogue box (Figure 21-10 below left).

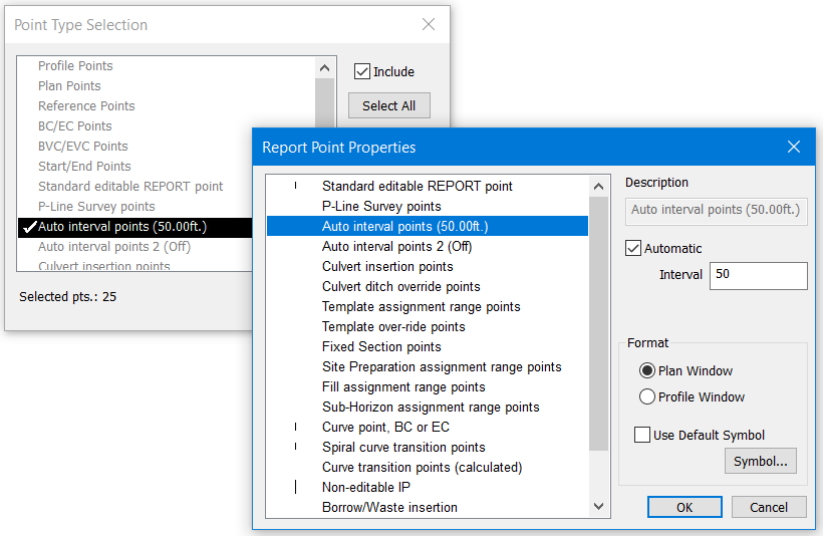

Figure 21-10: Setting the *Data Window* to *Display Information every 50 ft*

    - Ensure that only the Auto Interval points are *checked* for display (as in figure above left).

- Press the *Properties*… button to show the *Report Point Properties* dialogue box (Figure 21-10).
- Select Auto interval points and *check* the *Automatic* box with an Interval of **50**.
- Press *OK* (there will be a re-calculation).
- Press *OK* again to return to the *Data Window Options* dialogue box.

29. To set up desired columns to display: press *Columns*… to open the *Data Window Fields* dialogue (Figure 21-11)

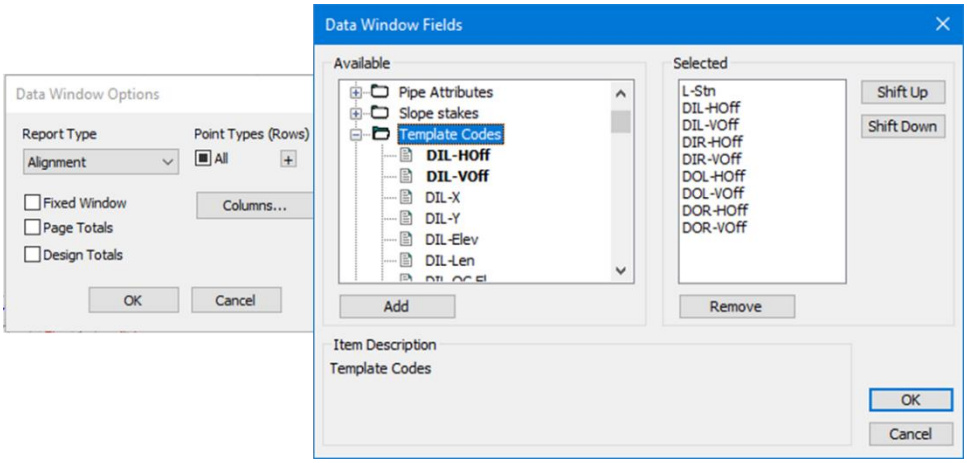

Figure 21-11: Selecting *Point Code Offsets* for Display in the Data Window.

30. Remove all but *L-Stn,* expand the *Template Codes* folder and *Add* **DIL, DIR, DOL, DOR Hoff** and **VOff** for each of the available Template Codes (Figure 21-11).
31. Press OK to close the *Columns* dialogue. Press OK again to close *Data Window Options*.

Your data window should now look like Figure 21-12 below.

Data:2

L-Stn ft.	DIL-HOff (N/A) ft.	DIL-VOff (N/A) ft.	DIR-HOff (N/A) ft.	DIR-VOff (N/A) ft.	DOR-HOff (N/A) ft.	DOR-VOff (N/A) ft.
3+00.00	-27.34	-2.83				
3+50.00	-27.34	-2.83				
4+00.00	-27.34	-2.83				
4+50.00	-28.66	-4.30				
5+00.00	-28.90	-4.54	28.90	-4.54	29.90	-4.54
5+50.00	-29.90	-4.56	28.90	-4.54	29.90	-4.54
6+00.00	-29.90	-4.56	28.90	-4.54	29.90	-4.54
6+50.00	-29.90	-4.56	28.90	-4.54	28.94	-4.54
7+00.00	-29.90	-4.56	28.90	-4.54	29.90	-4.54
7+50.00	-27.93	-3.53	29.23	-4.49	30.23	-4.49
8+00.00	-27.34	-2.83	30.73	-5.33	31.73	-5.33
8+50.00	-27.34	-2.83	30.73	-5.33	31.73	-5.33
9+00.00	-27.34	-2.83				
9+50.00	-27.34	-2.83	30.73	-5.33	31.30	-5.33
10+00.00	-27.34	-2.83	30.73	-5.33	31.73	-5.33
10+50.00	-31.25	-5.78	27.29	-2.46	28.29	-2.46
11+00.00	-31.91	-6.31				

Figure 21-12: Data Window Showing *Point Code Offsets*

There are two export options for data from the Data Window:

- *Data | Export Data to ASCII.*

- Data can be copied to clipboard: (*Data* | *Visible Window* or <*Right-click*> | *Copy Data to Clipboard* or use the hotkey <*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:

32. In the Section Window *Status* area (below the graphic). <*Right-click*> in the Section window and choose *Section Options*; press the *Fields…* button. Here you can add and remove fields for reporting.
33. Press *OK* to close the *Section Window Status Fields* dialogue. And *OK* again to close *Section Window Options*.
34. *File* | *Close*. Do not save changes.

22. Culverts

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

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 7.dsnx**.
2. In the plan window, zoom in to the creek crossing near stn 14+00 (figure below).

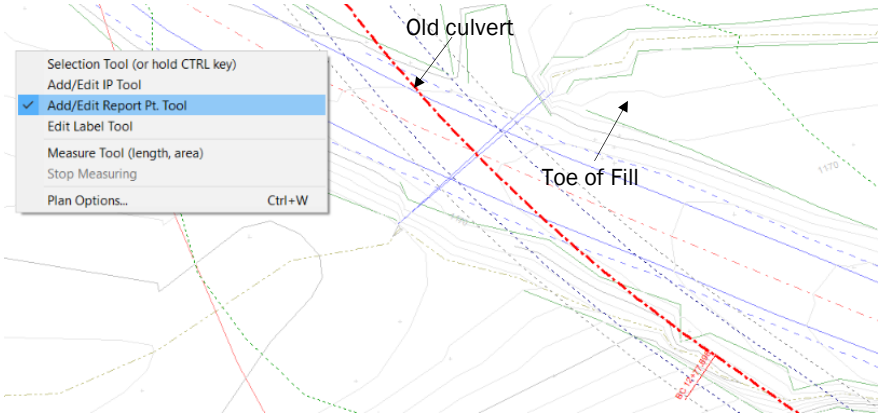

Figure 22-1: Creek Crossing Location; with the Old Culvert in the Background

3. <*Right-click*> in the plan window and choose *Add/Edit Report Pt. Tool*
4. Your mouse cursor will change to the pencil with question mark . 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. Your cross section window will update to show the new location (Figure 22-2).

Note: Report points can only follow the existing alignment.

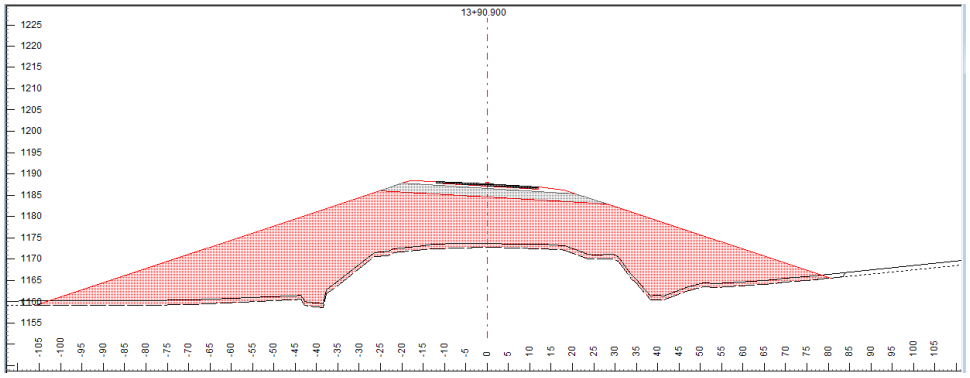

Figure 22-2: Desired Culvert Location

5. Select screen layout <Defaults and Layouts>\training**training Culvert.dlt**. Your screen should look similar to the figure below.

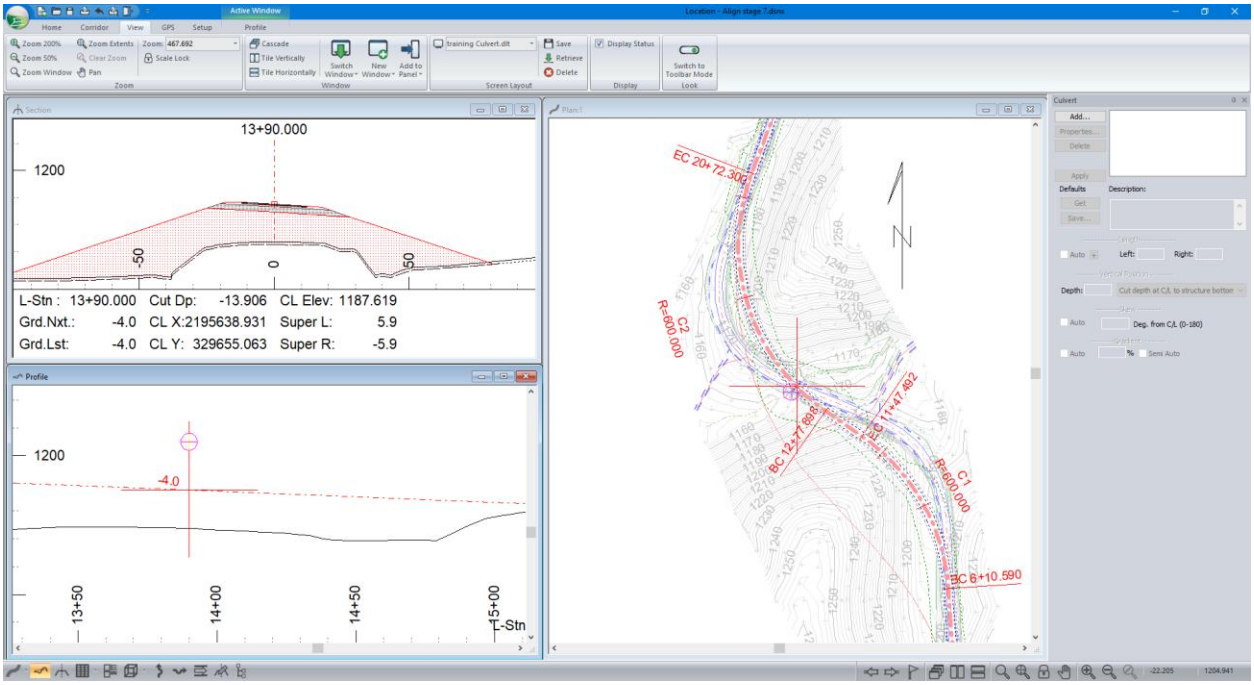

Figure 22-3: Screen Layout Training *Culvert.dlt*

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

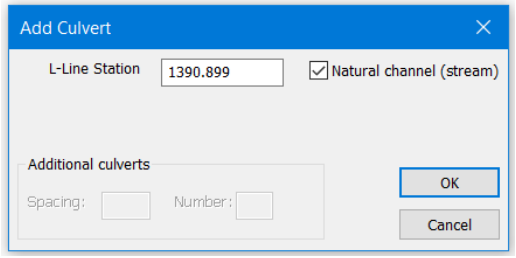

Figure 22-4: *Add Culvert* Dialogue Box

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 the *next*  and then *back*  buttons in the navigation tool bar. Whenever you change the current section this way all windows scroll depicting the new current points. Your culvert should be visible in all three windows.

9. Press *Properties…* (top left of *Culverts panel*) to open the *Culvert Properties* dialogue box shown in Figure 22-5.

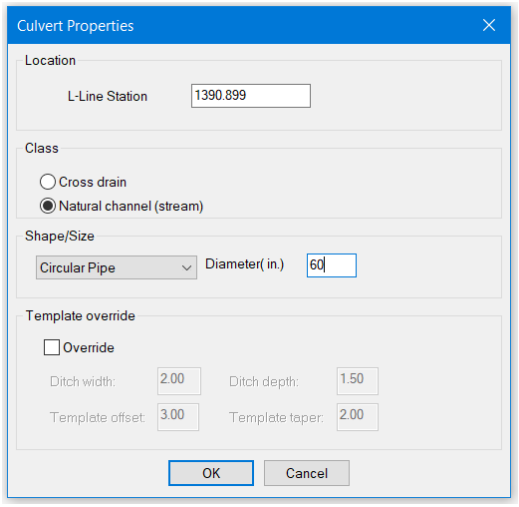

Figure 22-5: The *Culvert Properties* Dialogue Box allows the user to change the Location, Size and Shape of a Culvert.

10. Change the Diameter to **60** inches and press *OK* to close the *Culvert Properties* dialogue box.
11. In the *Culverts Panel*, change the *Vertical Position* dropdown menu to *"Cut depth to C/L to structure bottom"* the Depth to **8** feet, figure below left and press *Apply* button to see the changes.

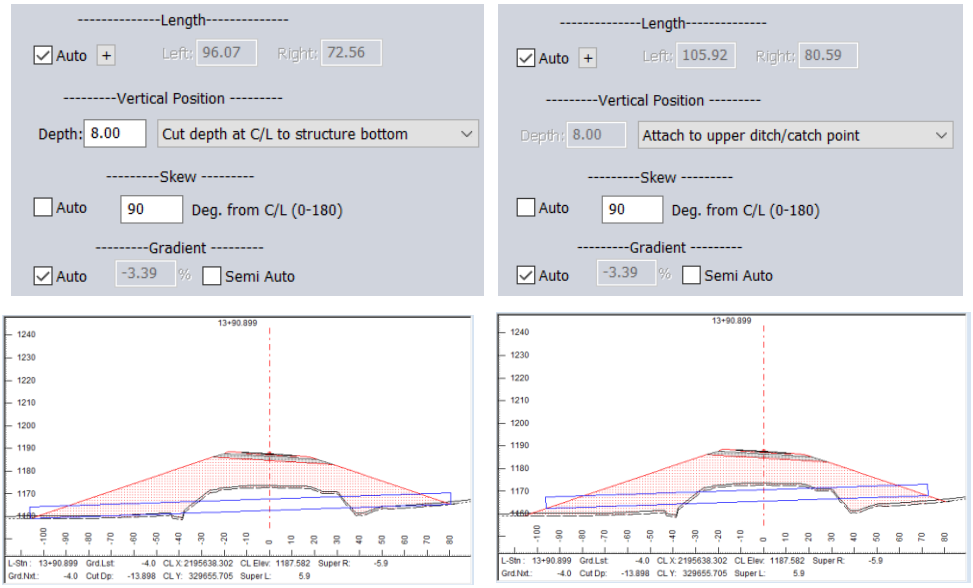

Figure 22-6: Culvert Elevation Controlled by Cut Depth at Center Line (Left) and by Catch Points (right)

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…* in the culverts panel. Uncheck the *Save to disk for future designs box* and respond *OK* to the 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. *Home* | *Assign by Range*, select the *Fill Types* tab.

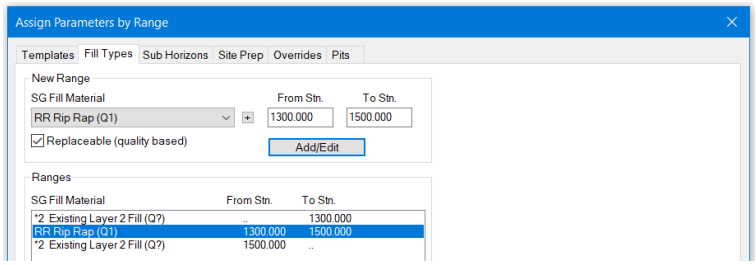

Figure 22-7: Applying Special *Fill (Rip Rap)* Near the Culvert

15. Choose fill material *RR Rip Rap* (this comes from your ground types table) and set the station range from **1300** -**1500**. Press the *Add/Edit* button.
16. Press the ⊞ button next to SG Fill Material type 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 dialogue box. Respond *OK* to the recalculate prompt.

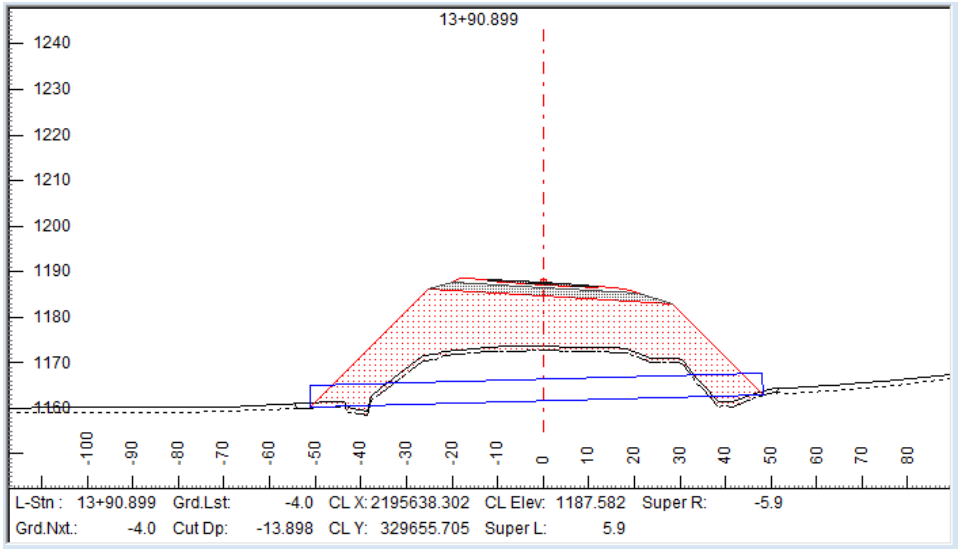

Figure 22-8: The *Creek Crossing* with Rip Rap Fill, Steeper Slopes Resulting in a Shorter Culvert

As shown in Figure 22-8, the length of the culvert has been reduced. 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: *Home | Assign by Range| Overrides* tab. Refer to section *Template Parameter Overrides*.

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

23. Labels

Annotation and labelling is available in the Plan, Profile and Section Windows. This section describes methods and procedures to control label formatting and positioning.

Label Classes

Labels are displayed according to their *Class Format* and *Point Format* (optional). The menu *View (Plan, Profile or Section) Options* provides access to class label formatting. The *Edit label tool* button in the toolbar allows you to modify individual labels (*point formatting*) with the mouse. The Section window does not allow *point formatting*.

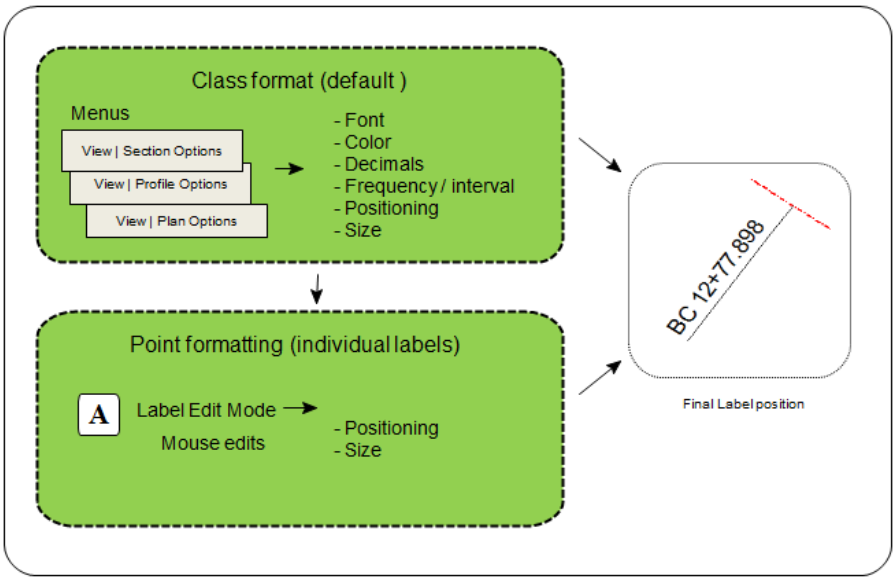

Figure 23-1: *Label Rendering*

All Label formatting is saved with your document of course, but only *class* formatting can be saved to a *Screen Layout*.

Note: When you retrieve a *Screen Layout*, you are setting all class label formatting (as well as the other saved layout options).

Class Label Formatting

The Plan Window is used in the following example; however, the same principles apply to the Profile and Section windows.

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 8.dsnx**.

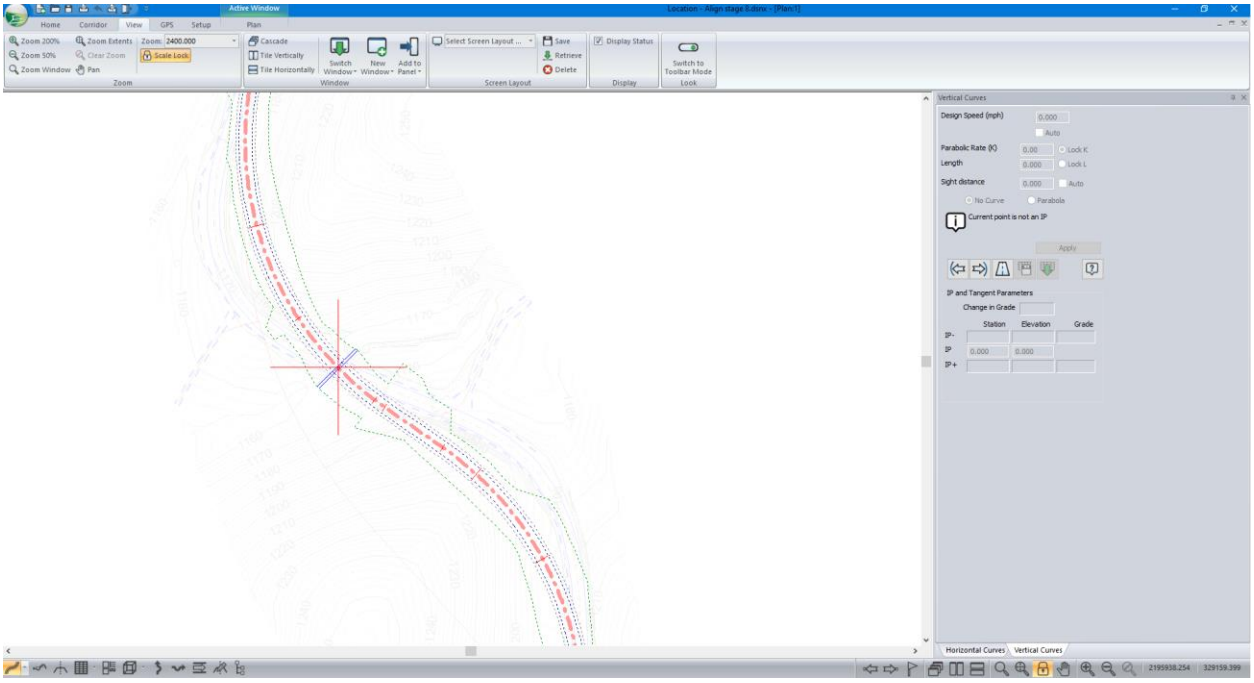

Figure 23-2: *Align Stage 8.dsnx*

This example demonstrates how to display:

- Station labels at reporting points (set at 200ft intervals)
- Horizontal curve labels at the IPs
- Culvert description labels.

These label types are representative; other labels behave in a similar way.
To create labels at equal intervals, you need to set up *Report Points* to generate cross sections where you want your labels. Most labels can only be displayed at existing cross sections.

2. In the plan window, <*Right-click*> | *Plan Options…* click on the ⊞ button next to *Report Points*. This will open the *Report Point Properties* dialogue box.
3. Select *Auto Interval Points 2* to create points on **200** ft intervals (figure below). Press *OK* twice to return to the main screen.

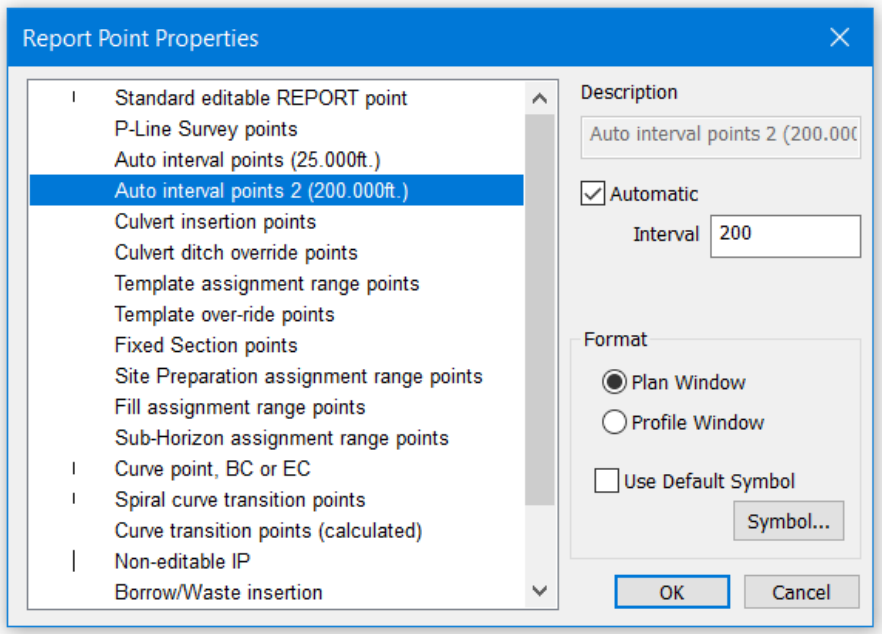

Figure 23-3: *Report Point Properties* Dialogue

4. Set up station label options:
    - <Right-click>, select Plan Options…
    - Click on the ⊞ button beside *Labels* to open the *Label Selection and Formatting* dialogue box.
    - Turn on *LStn Report Points (Intervals)* by <Double-clicking> (or select, then set the *Display* check box).

Note: LStn stands for Location Station

    - Set the *Interval spacing* to **200** (Figure 23-4).
    - Press *OK* twice to return to the main screen.

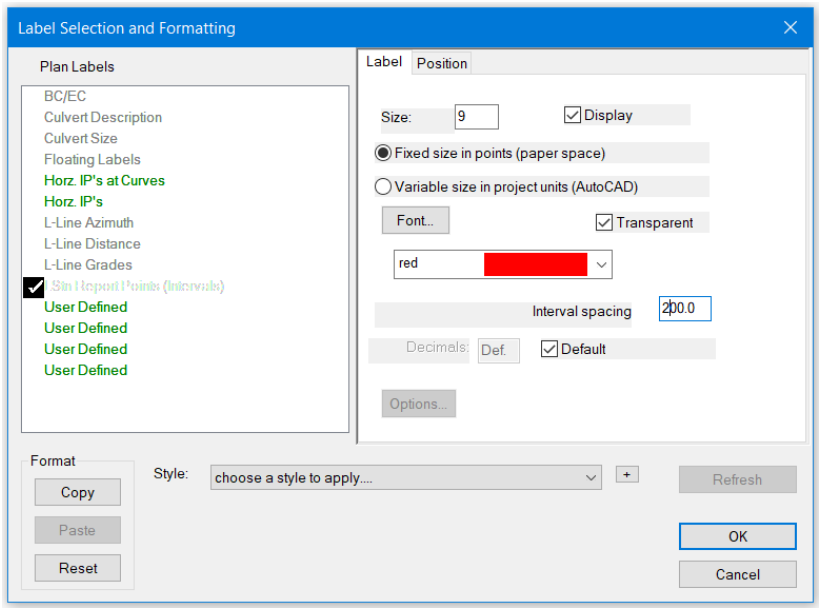

Figure 23-4: Plan Window *Label Selection and Formatting*

After the Plan window refreshes the screen should appear with stationing every 200 feet.

5. The format of the stationing (xx+yy in this case) is controlled in *Setup* | *Location Setup* | *Units* tab | *Stationing*: *Traditional S+dd e.g. 12+01* from dropdown.

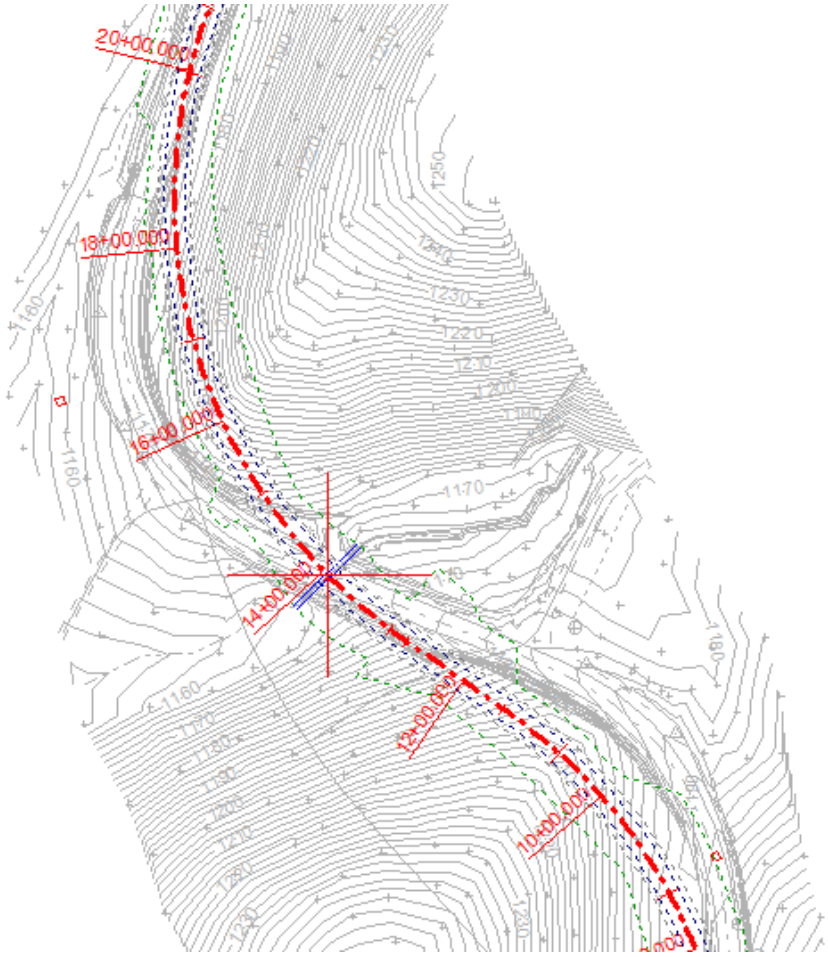

Figure 23-5: Stationing at Fixed Intervals

6. Next, horizontal curve information is added. Another option is to use the *Curve Tables* in the Multi-Plot window (see *Multi-Plot Report Builder*).
7. Set up curve labels for display:
    - Again, open the *Plan Label Selection and Formatting* dialogue box. <*Right-click*> | *Plan Options*... click the ⊞ next to *Labels*.
    - Turn on display of *Horizontal IP's at Curves*.
    - Press *OK*, *OK* to return to the main screen. Notice the curve radius labels.

Some labels have configurable content. In the next section, we will add to the contents of this label.

8. Add Begin Curve (BC) and End Curve (EC) stations to the Horizontal IP's at Curves label:
    - Open the *Plan Label Selection and Formatting* dialogue box.
    - Select *Horizontal IP's at Curves* and click on *Options*... button.

Note: Label classes shaded green will have *Options* available. Refer to the

section below.

- Click on the *Add Attribute…* button and then add *BC Station* and *EC Station* (Figure 23-6).

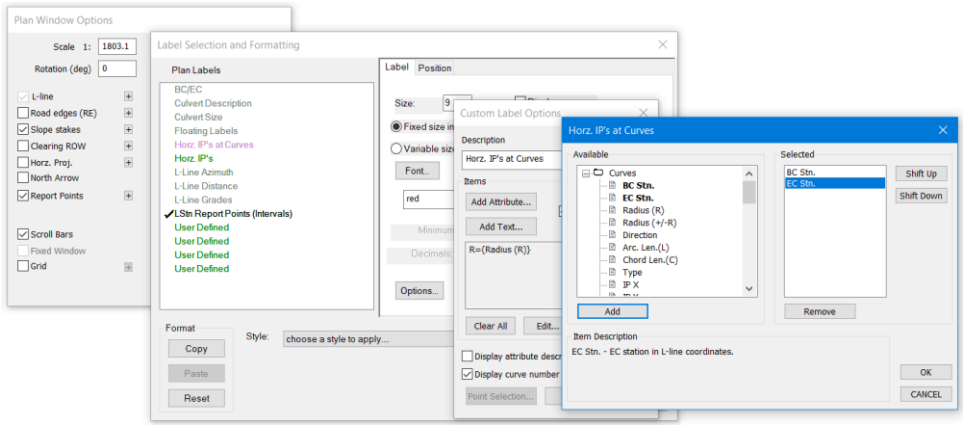

Figure 23-6: Displaying Horizontal IP's Labels at Curves

- Press *OK* to update the *Items* list.
- Press *OK* three more times to return to the main screen.

Notice that the additional BC/EC information is now displayed (Figure 23-7).

Figure 23-7: *Curve Information Labels*

There is a pre-configured label called *BC/EC*; you might want to use this as an alternative to displaying station values at the curve IP.

9. To turn on labels for the attributes that were just added: *Plan Options…* click the ⊞ sign next to *Labels*, select *Horz. IP's at Curves,* select *Options…* check *Display attribute descriptions*. Press *OK*, *OK*.

Now we will display a label at the culvert just before station 14+00. We will also look briefly at the *Position* formatting.

10. Add a culvert information label:
    - Open the *Plan Label Selection and Formatting* dialogue box.
    - Select (<*double-click*>) *Culvert Description* class.
    - Select the *Position* tab (Figure 23-8).
    - Change the *Connector* combo box to **Circle**.
    - Press *OK* twice to return to the main screen.

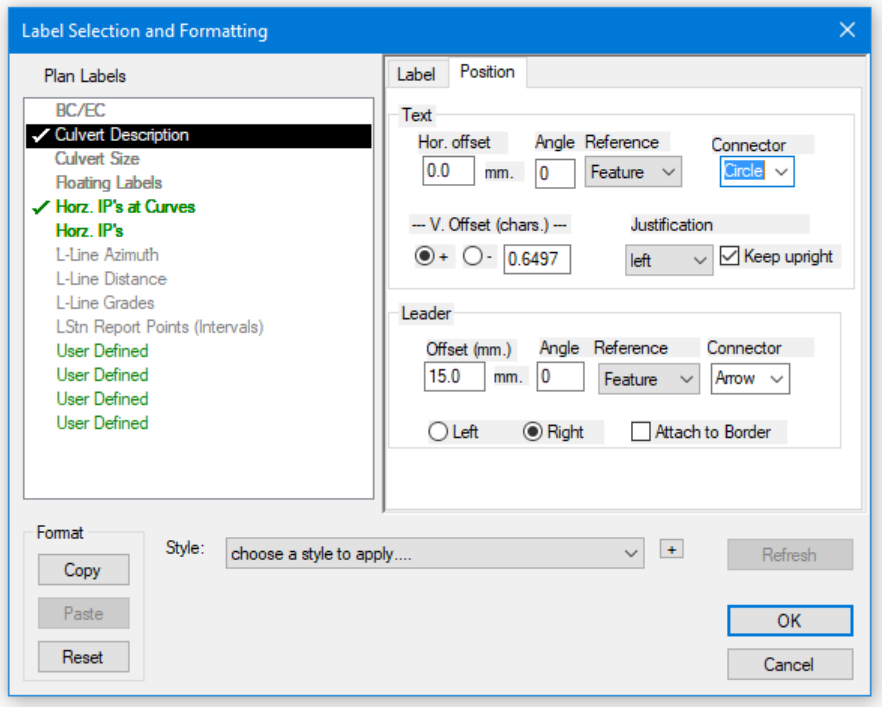

Figure 23-8: *Culvert Information Labels*

Figure 23-9: *Culvert Pipe Display*

11. At this point you may want to experiment with some of the *Position* options to see how they modify the label formatting; see also <*F1*> help.

Note: It is possible to use Point Label Formatting (see section below) to modify the *Position* settings for the class.

12. *File* | *Close*. Do not save changes.

User Definable Labels

It is possible to create user definable labels and display these labels at reporting points along the alignment. User definable labels can consist of *attributes* and *static text*. The following is a partial list of attributes:

- L-Stn l-line stationing
- V.Brk vertical grade break
- CL X centerline X
- CL Y centerline Y
- CL Elev centerline Z

The next example will demonstrate how to create a user defined label in the Plan window.

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 8.dsnx**
2. *Plan* | *Plan Options* or <*Right-click*> *Plan Options*...
3. *Click* on the ⊞ button adjacent to *Labels* to open the *Label Selection and Formatting* dialogue box.
4. In the *Label Selection and Formatting* dialogue, <*Double-click*> on the first *User Defined* label class (or click to select, then check *Display*).
5. Click on the *Options*... button to open the *Custom Label Options* dialogue box (figure Figure 23-10).

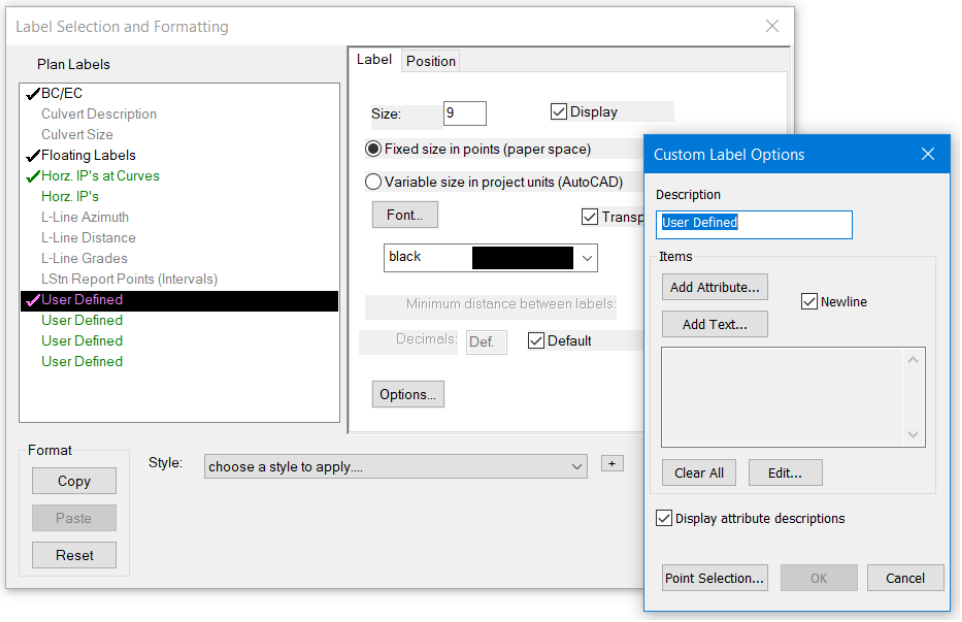

Figure 23-10: *Custom Label Options* Dialogue Box

6. Set up the new custom label:
   - Change *Description* to **Interval**.
   - Press the *Add Attribute*... button to open dialogue box in the figure below.
   - Add *L-Stn, CL X* and *CL Y* as shown below from the *L-Line* folder.

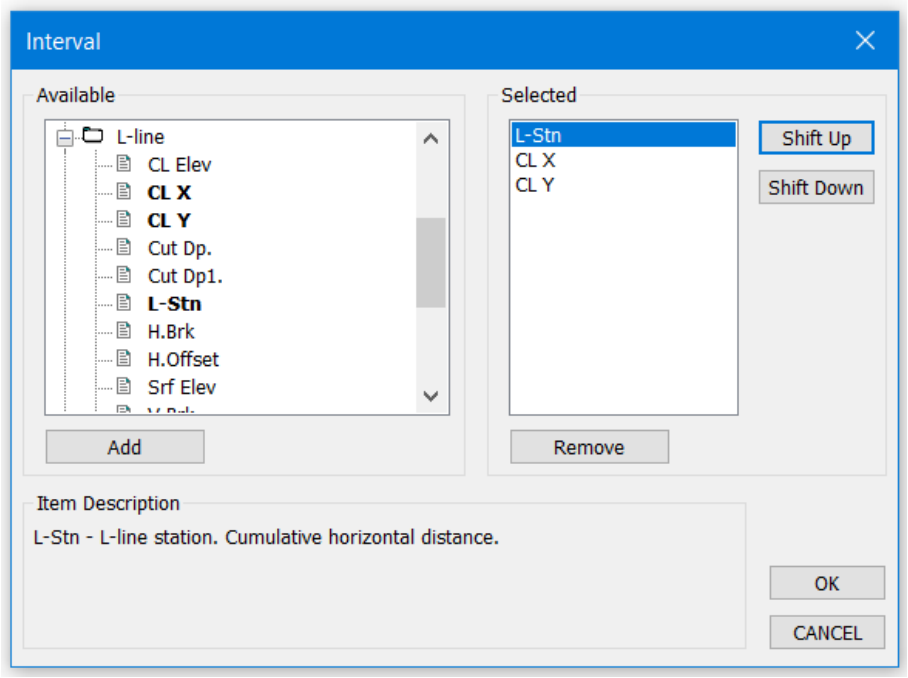

Figure 23-11: Add Attributes Dialogue

7. Press *OK* to return to the *Custom Label Options* dialogue and the updated *Items* list (Figure 23-12).

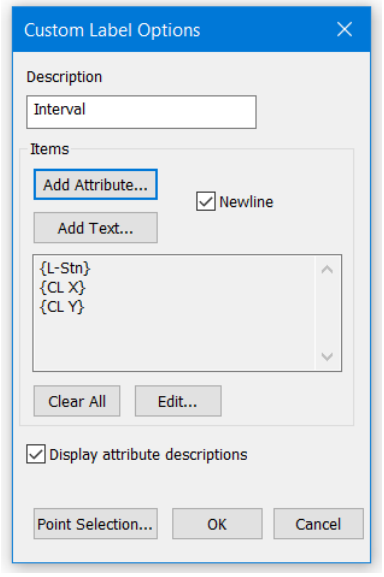

Figure 23-12: *Custom Label Options* Dialogue Box after Modifications

8. Click *Point Selection...* button to open the *Point Type Selection* dialogue box. Include only *Auto interval points 2* (Figure 23-13). <*Double-click*> or select and set *Include*.

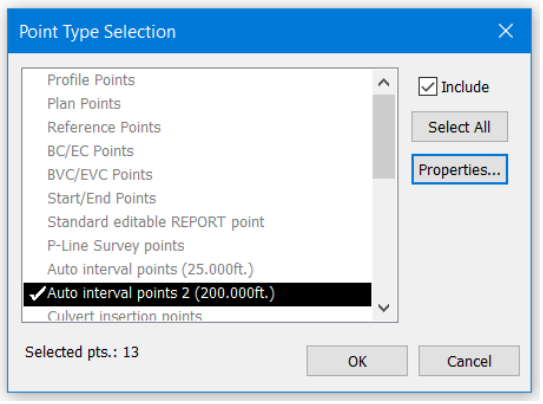

Figure 23-13: *Point Type Selection* Dialogue Box

Auto interval points 2 is set to **200** ft; you can change this value by clicking on the *Properties* button.

9. Press *OK* three time to return to the main screen.

Your Plan window should appear as shown below.

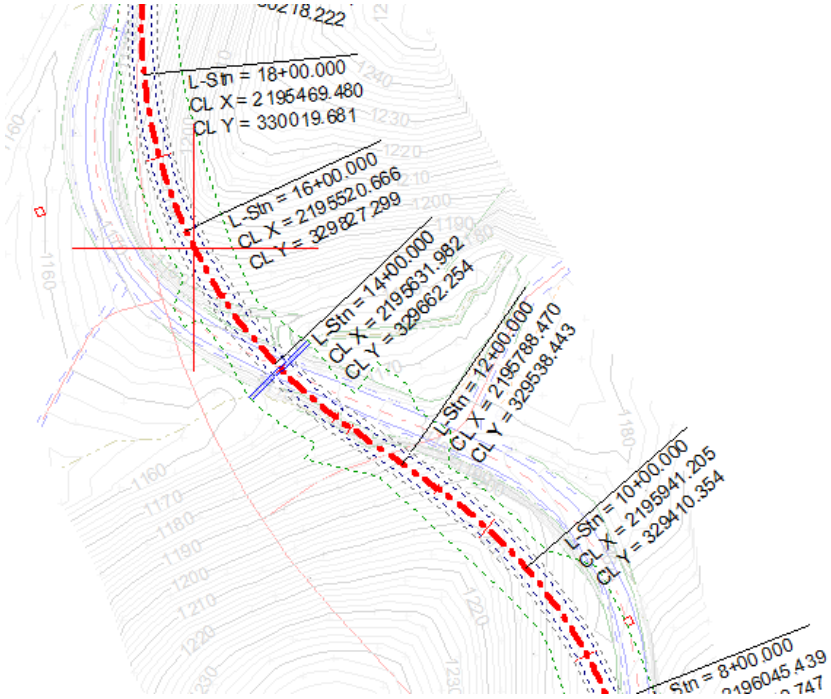

Figure 23-14: *User Defined Labels* (Stn, X, Y at 200' Spacing)

The text before the numeric value (e.g. “L-Stn =”) is included because Display attribute descriptions was set in the options (refer to figure above).

10. If you are not satisfied with these descriptions, clear this check box and enter your own:
    - *<Right-click> Plan Options…*
    - ⊞ next to *Labels*
    - Select *Interval*

- Press the *Options…* button
- *Edit…* opens the *User Label Edit* dialogue box
- Type: "Station = {L-Stn}"
- Press *OK* four times to exit all the dialogue boxes.

11. *File* | *Close*. Do not save changes.

In the exercises above, we changed class label formatting; all labels with the same class had the same formatting. Class label formatting is saved with your document, but it can also be saved in *Screen Layouts* for use in other documents.

Point Label Formatting

Editing Labels with the Mouse

It is often necessary to control the position and format of individual labels. In this section Label Edit mode will be used to adjust individual labels.

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 8a.dsnx**.
2. <*Right-click*> in the Plan window and select *Edit Label Tool* from the context menu. This will change the cursor to the *Edit Label tool* cursor.
3. Move your mouse cursor over the red 12+00 label, when the cursor changes to a simple cross, *left click* once.
4. The label is now selected and should look like the one in the figure below.

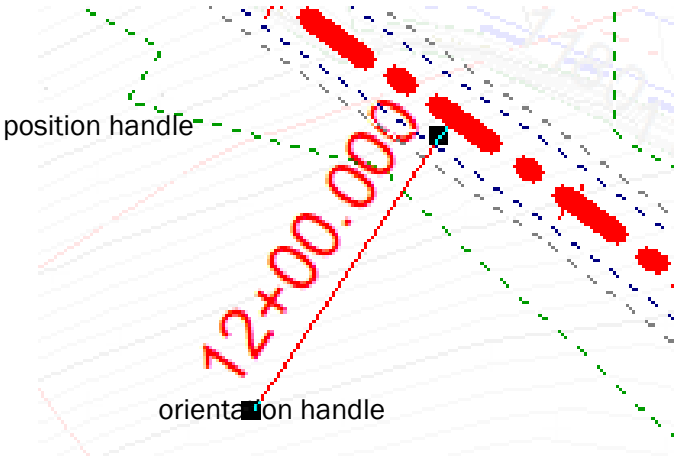

Figure 23-15: Selected Label with *Handles Visible*

5. Move your *mouse pointer* over each of the handles (black squares); the one farthest from the road centerline is a rotation handle .
6. When in label edit mode, *click* and *drag* on a selected label handle to move or rotate the label.
7. Re-orient and re-position the label until it appears as shown below.

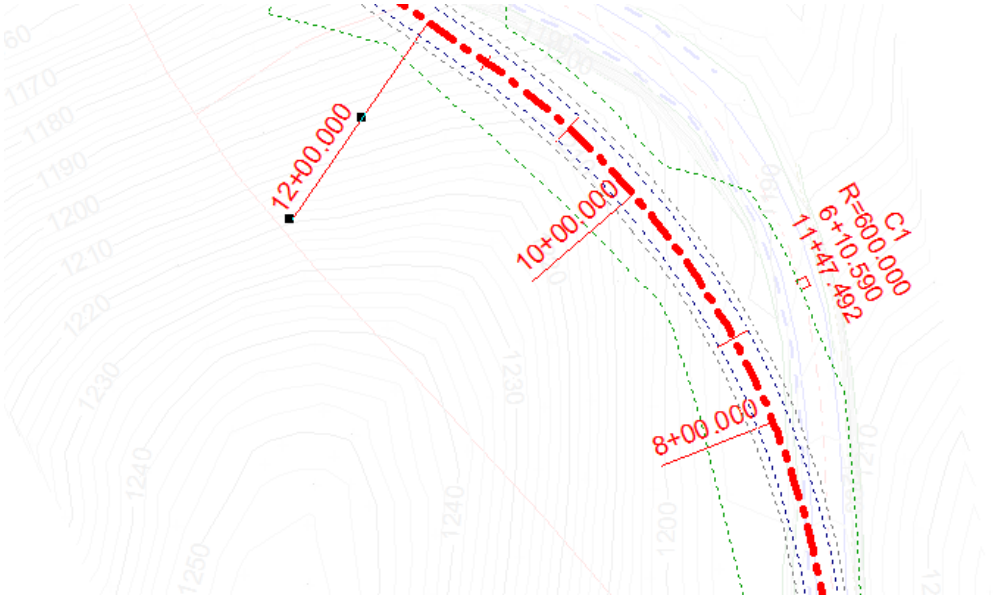

Figure 23-16: Label after Re-Positioning

8. <*Double-click*> on the text of the label you just edited. This opens the *Label Selection and Formatting* dialogue box.

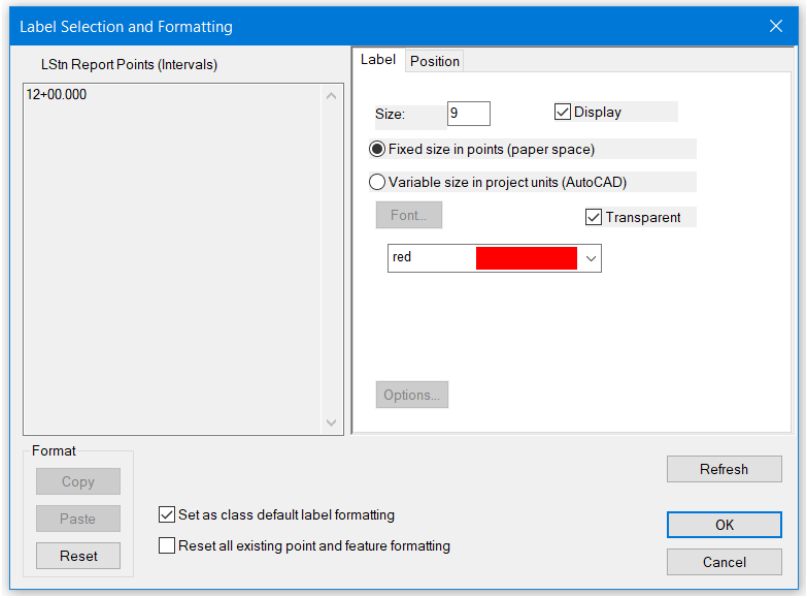

Figure 23-17: Label Selection and Formatting Dialogue Box

9. Select *Set as class default label formatting*. Press *OK*.

10. Unlock the scale by clicking on the *lock scale* button  and *zoom extents*  in the windows toolbar.

Your Plan window should now appear as shown below.

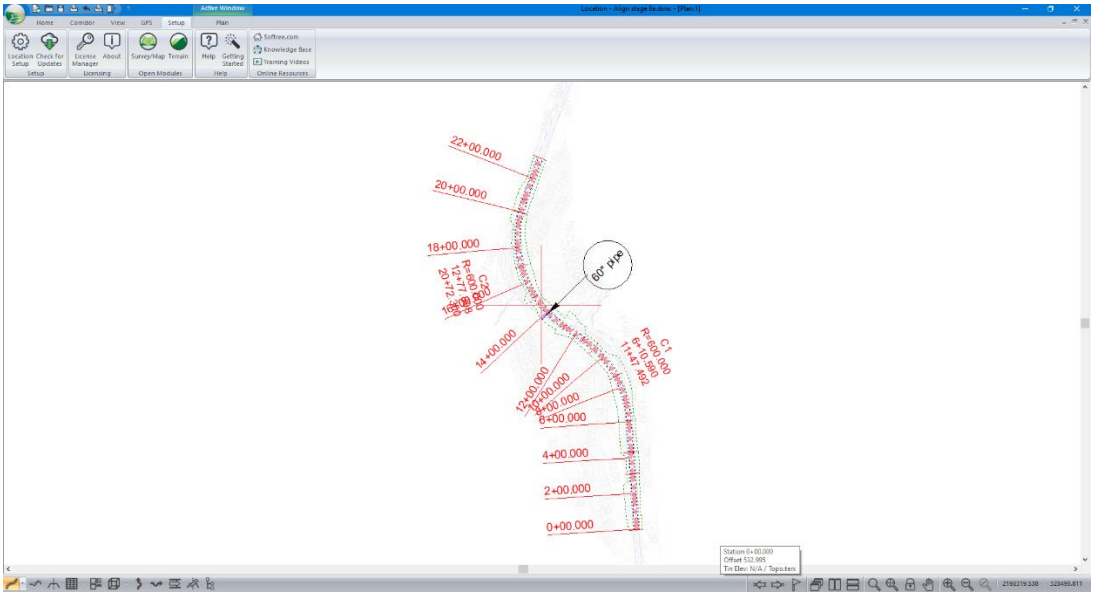

Figure 23-18: Plan View after Change to Class Format

Floating Labels

Floating Labels can be added anywhere in the Plan or Profile Windows. Floating Labels can have their *anchor point* moved.

11. The *Edit Label tool* cursor should still be enabled. If not, <*right-click*> in the Plan window and select *Edit Label Tool*.
12. <*Right-click*> anywhere in the Plan Window. Select *Plan Options*. Press the *plus* button beside Labels. <*Double-click*> on *Floating Labels* to turn then on. Press *OK* twice to return to the main screen.
13. With the label edit tool , *click* the *left mouse button* anywhere in the Plan Window to open the Label Selection and Formatting dialogue box.
14. Change "xxxxxxxx" to "***Rock cut section***". *Click* on the *Position* tab and change the *Leader* to have a ***28***mm Offset and an **Arrow** Connector.
15. Press *OK*; the floating label will appear where you first clicked the mouse.

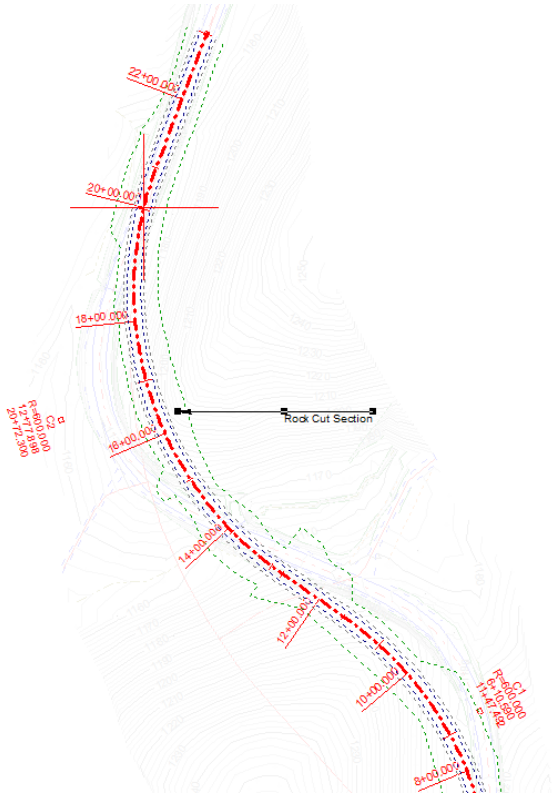

Figure 23-19: Plan with *Floating Label*

16. *File* | *Close*. Do not save changes.

Profile Sub View Labels

For presentation purposes, it is often useful to display information below the Profile. The following example will create sub-view labels for station, FG (final ground) and OG (original ground).

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 8a.dsnx.**
2. Activate the Profile Window .
3. <*Right-click*> | Profile Options…
4. In the Sub-Windows area click on the *Select…* button:
   - Scroll down the *Available* list and select *Custom-Label 1*, press *Add*. Custom Label -1 now appears in the *Selected* list.
   - Similarly add *Custom Label -2* and *Custom Label - 3*.
   - Press *OK* to return to the *Profile Window Options* dialogue box.

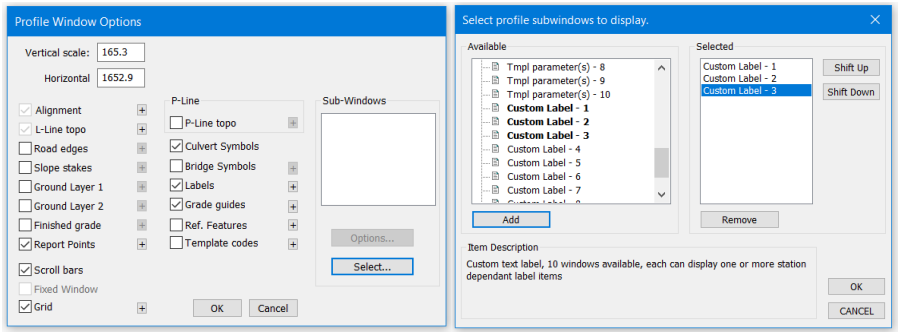

Figure 23 20: *Profile Window Options* Dialogue & *Sub-Windows* Selection

5. Within the *Sub-Windows* area, select *Custom Label-1*.
6. Press *Options…*. When the *Profile Custom Label Sub-Window Options* dialogue box appears:
   - Change the *Description* to **Stn**.
   - Click on *Add Attribute…*
   - Add *L-Stn* (L-Line folder) to the *Selected list* and press *OK*.
   - Uncheck the *Display attribute descriptions* box.
   - Click *Point Selection…* and set the point type selection to *Auto Interval 2 (200' intervals)*.
   - Press *OK*.

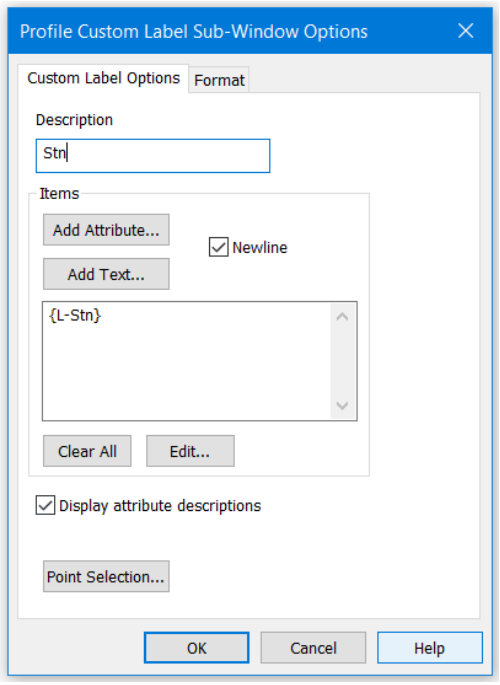

Figure 23-20: Custom Label Profile Sub-Window Options Dialogue

We will now do similar steps to setup labels (in *Custom Label-2* and *Custom Label-3*) for FG (final ground) and OG (original ground).

7. With *Custom Label-2* selected:

- Click on *Options…*.
- Change the *Description* to **FG**
- Click on the *Add Attribute…* button.
- Add *CL-Elev* (L-Line folder) to the *Selected* list and press *OK*.
- Uncheck *Display attribute descriptions*.
- Click on *Point Selection…* and make sure the point type selection is set to be *Auto Interval 2 (200' intervals)*.
- Press *OK* twice.

8. With *Custom Label-3* selected:
   - Click on *Options…*.
   - Change the *Description* to **OG**
   - Click on the *Add Attribute…* button.
   - Add *GND-Elev* (*Ground Layers* folder) to the *Selected* list and press *OK*.
   - Uncheck Display attribute descriptions.
   - Click on Point Selection… and make sure the point type selection is set to be Auto Interval 2 (200' intervals).
   - Press OK three times to return to the main screen.

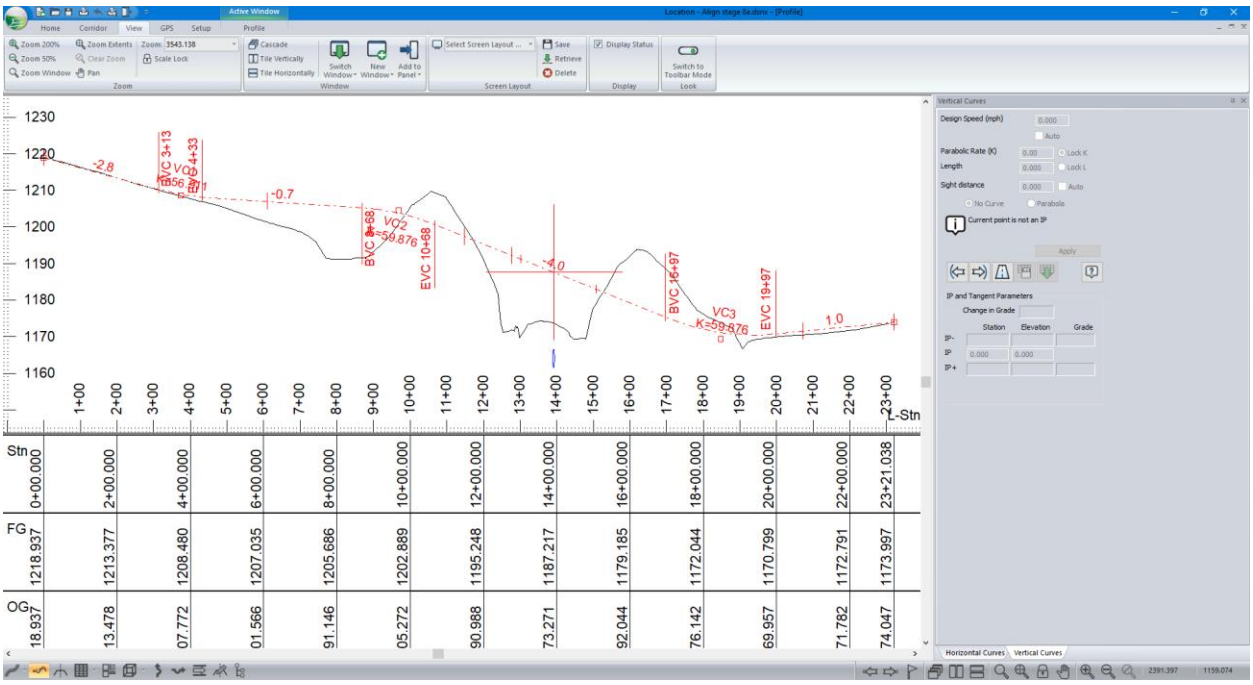

Figure 23-21: Profile Sub-View Labels

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

24. Multi-Plot Report Builder

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.

As of Verstion 8, Multi-Plot layouts are no longer included in standard screen layouts. There are two unique layout file types available to Multi-Plot in the Location module:

- Book Layout file (**.blt**) – a book layout file is a collection of chapter layouts.
- Chapter Layout file (**.clt**) – a chapter layout file contains the information for a single layout type. The number pages within each chapter are defined by that Chapter's pagination settings.

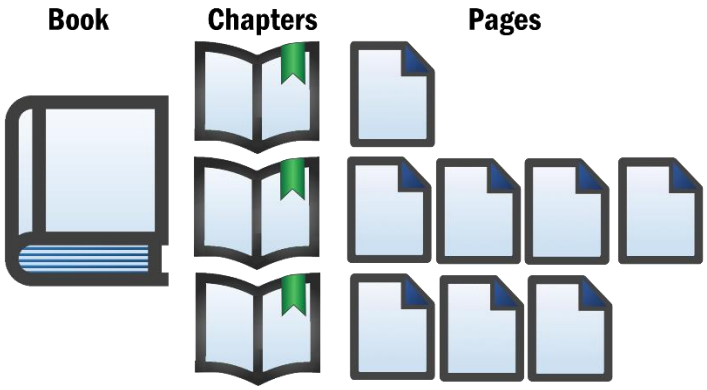

Figure 24-1: Multi-Plot Structure

In this section, you will learn how to create a Multi-Plot book with several chapters, including a title page and a standard Plan over Profile.

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.

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**)

Creating and Positioning Sub-Views

1. *File* | *Open* in Location module. <RoadEngCivil>\Location**Align stage 9.dsnx**.
2. Change screen layout: *View* | *Screen Layout* dropdown, select ***training Normal.dlt*** from dropdown list.

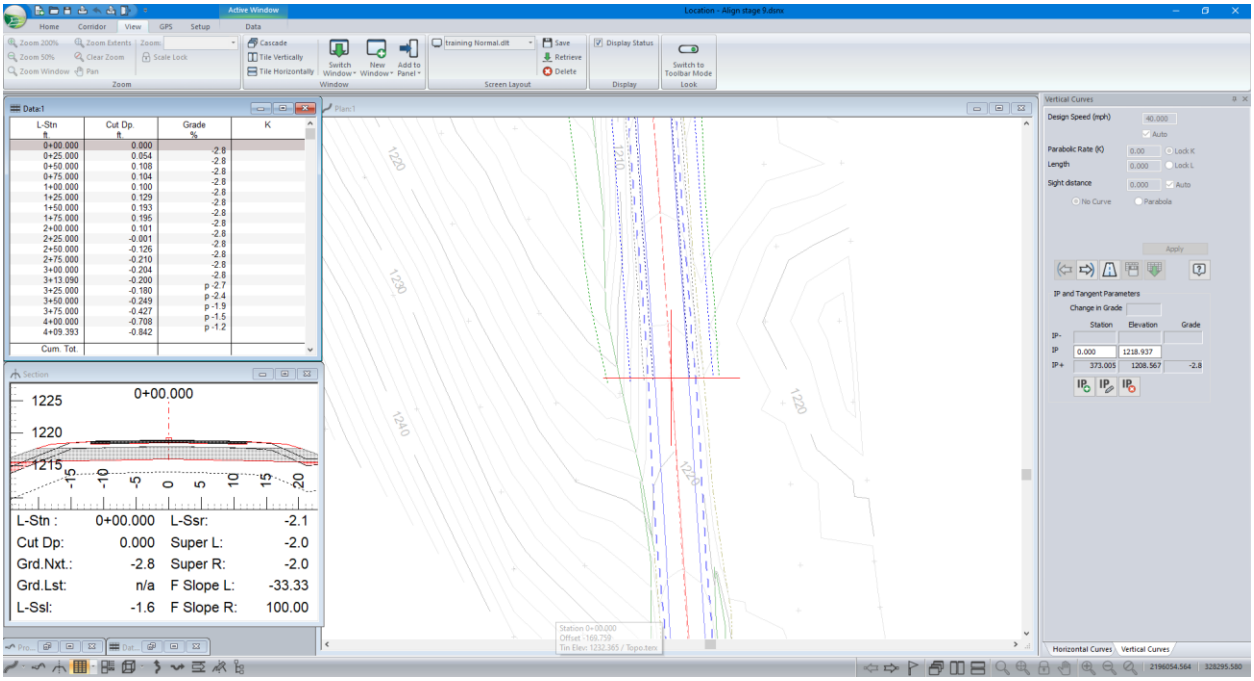

Figure 24-2: *Align stage 9.dsnx*

For our example 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.

3. Press the *Multi-Plot*  in the windows toolbar. This opens a multi-plot window and display bar on the left-hand side. Note that multi-plot is organized in the Book, Default and Page levels. The window opens on the page level.

Configuring Your Page Size

The orientation and size of the blank sheet, within the Multi-Plot Window, defaults to 11" x 17" horizontal. Five standard engineering page sizes are available as pre-set options in the *Page Size* dialogue:

ANSI	Size (mm)	Size (inches)
ANSI A	215.9mm x 279.4 mm	8.5" x 11"
ANSI B	279.4mm x 431.8 mm	11" x 17"
ANSI C	431.8mm x 558.8 mm	17" x 22"
ANSI D	558.8mm x 863.6 mm	22" x 34"
ANSI E	863.6mm x 1117.6 mm	34" x 44"

Figure 24-3: *Multi-Plot Page Size* Defaults

4. *Multi-Plot* | *Units*. Change to *Imperial*.
5. *Multi-Plot* | *Page Size* | *Custom Size* to open the *Page Size* dialogue box.
    - Ensure the *orientation* to **Landscape**.
    - Ensure the *Paper size* to **11" x 17"** (Size B)

- Press *OK*.

Note: the screen view is determined by the Page Size and Page Orientation controls in the ribbon toolbar. When printing, the printer setup must be confirmed to match the screen setup.

6. *File*| *Printer Setup* to open the *Print Setup* dialogue. Set the paper size and orientation to match our screen size and layout (11x17" and landscape). This is the paper size that governs in the print preview. This depends on the type of printer and paper size the printer can handle.

Configuring our Chapter

The first chapter we will create is a Plan over Profile layout.

7. <Right-click> on the *Default* chapter, this should allow you to edit its name. Select *Rename Chapter*, change the name to **PlanProfile.**

The Location Multi-Plot Window can automatically produce as many pages as are required to show the entire design. Before we insert a Sub-View, it would be more appropriate to set the number of pages to avoid any potential rework. In this section, we will explore some of the pagination options.

2. *Multi-plot | Pagination:*
   - Make sure **Fixed** is selected as *Length of road (Stations) per page*.
   - Enter a value of **800** Feet.
   - Enter an *overlap* value of **0%** as seen below in Figure 24-4.
   - Press *OK* to accept these *Pagination Options*.

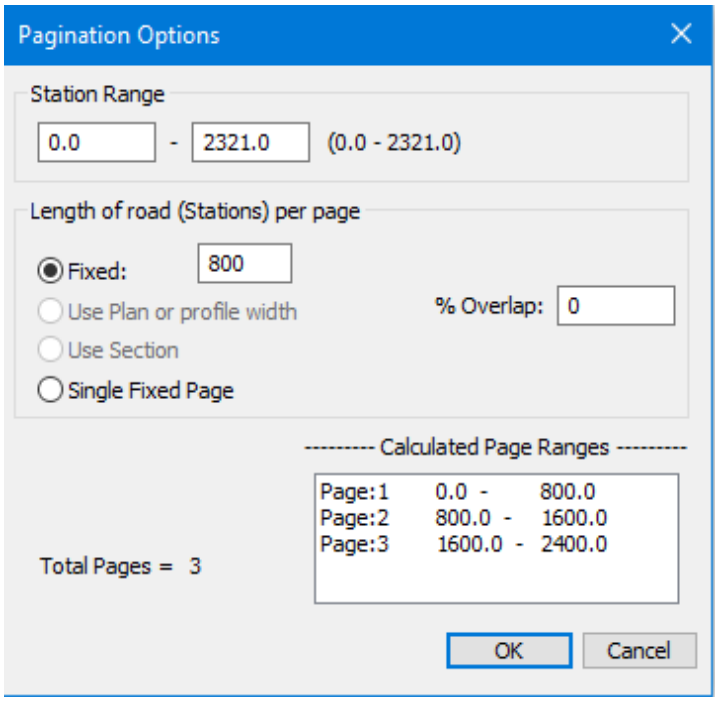

Figure 24-4: *Pagination Options*

Adding Graphic Sub-views

Now let's add some content to our page. It is recommended to always add items in Chapter-mode. This means the sub-view will be shown on all pages within that chapter.

8. With the *PlanProfile* chapter selected, *Multi-Plot* | *New Sub-view* | *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*. The scale and positioning of the Plan window is controlled within Multi-Plot, and can differ from your main window.

9. <Double-click> on the Plan sub-view. Change the *Scale* to 1: **800**. Press *OK*.
10. 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 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: *Multi-Plot Plan Rotation.*

11. *Resize* and *reposition* ✥ the Plan sub-view to look similar to the following figure.

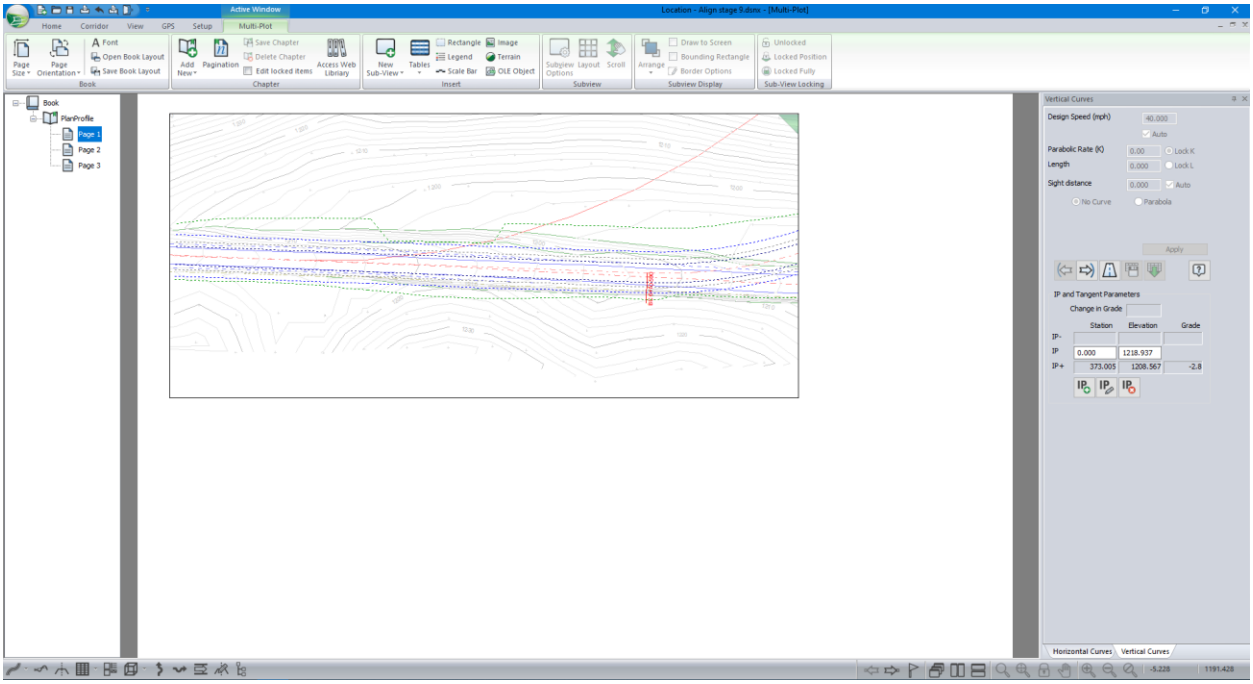

Figure 24-5: *Plan Sub-View* after Sizing and Positioning

12. *Multi-Plot* | *New Sub-view* | *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).

13. <Double-click> on the Profile Sub-view to open its options. In the *Sub-Windows* area, press *Select*. *Remove* Mass Haul from the *Selected* area. Press *OK* twice.

Note: Changing the profile sub-view options in Multi-Plot does not impact the main Profile window.

Note: 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.

14. Right-click on the screen, select *show Grid and Snap to Grid*, and set the *Spacing* at **0.25"**, as shown in the figure below:

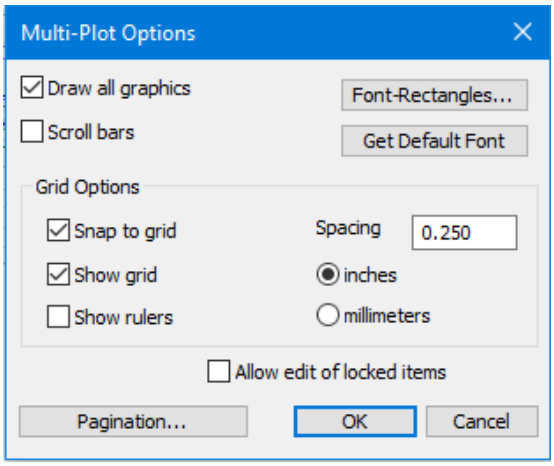

Figure 24-6: *Multi-Plot Options* box

A dot grid will cover the entire Multi-Plot sheet.

15. Now adjust the size and position of both the Plan and Profile sub-views so they are aligned as in the figure below. Alternatively, you can also use the align tools. With both sub-views selected, *Sub-View* | *Align* | *Align Left*.

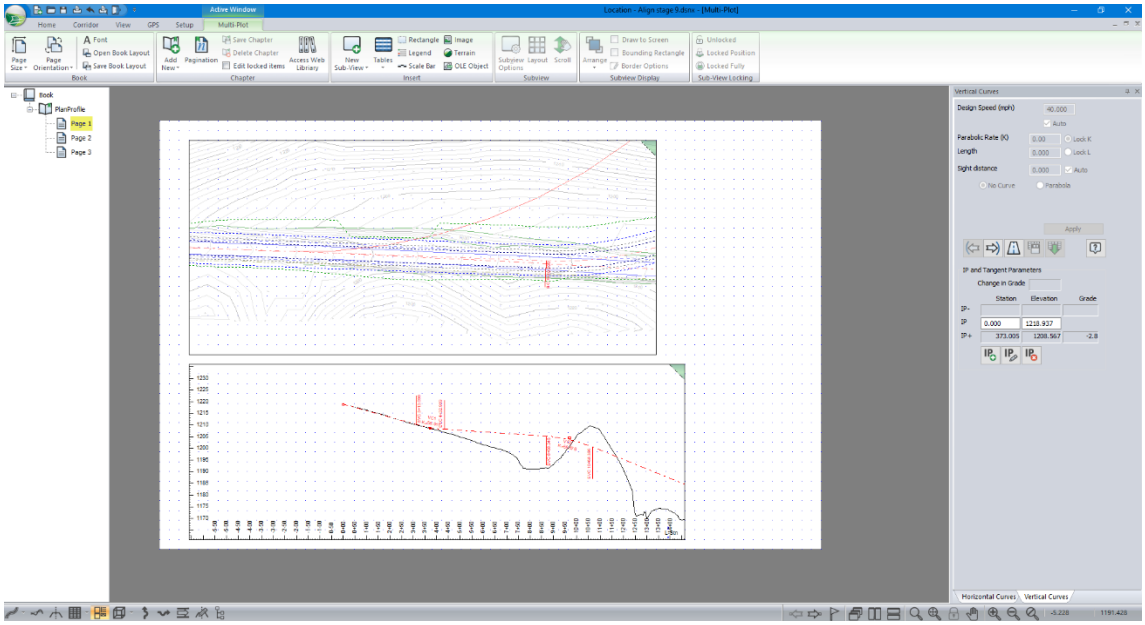

Figure 24-7: Multi-Plot After *Grid Enabled*

These two sub-views are now set-up on all 3 pages of our Plan Profile Chapter. But, the scale for our new profile sub-views is not ideal. Let's adjust so the horizontal scale of the Plan and Profile is the same.

16. <*Double-click*> on the Profile sub-view. Change the *Horizontal Scale* to 1: **800**. Press *OK*.

Note: If you have two sub-view windows vertically on top of one another, such as a plan view and profile view, to ensure they are in line (starting at the same station) be sure the scale is the same.

Note: Sub-view windows can be selected/deselected and deleted. When you click outside all sub-views and drag the mouse you will create a selection rectangle. All sub-views inside 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.

Note: The same dialogue box can be opened by left <Double-click>ing on the Sub window sub-view.

Adding a Scale Bar

17. *Multi-Plot* | *Insert Scale Bar*. A Scale bar will appear in the middle of your sheet.
18. <*Double-click*> on the new scale bar to open the *Scale Bar Sub-view Options* menu.

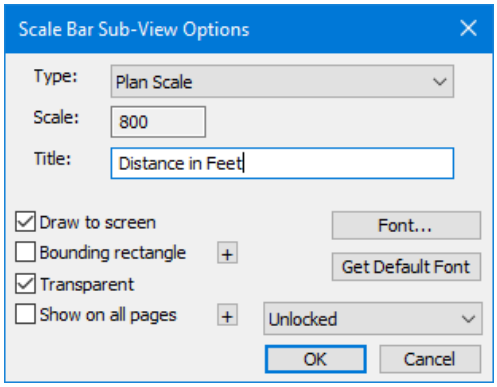

Figure 24-8: *Scale Bar Sub-View Options* Dialogue Box

19. Keep the Type: *Plan Scale,* ensure the *Scale* is set to **800**. Add in the optional Title: **Distance in Feet**. Press *OK*.
20. Resize and reposition the Scale Bar sub-view, until it appears as in the figure below (also see notes below).

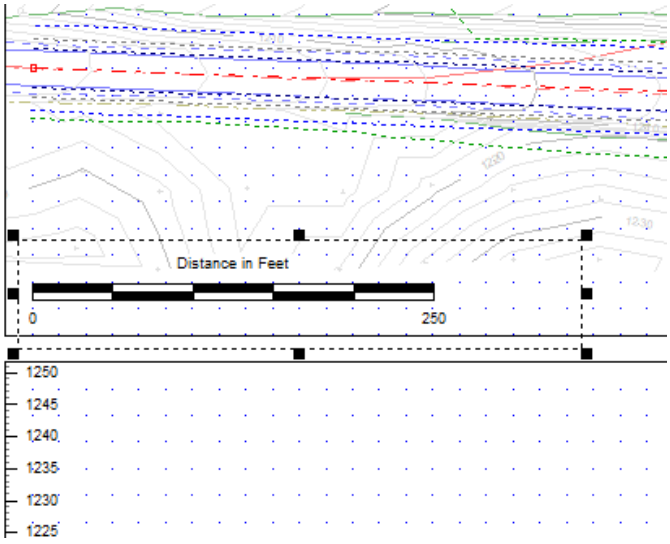

Figure 24-9 : *Scale Bar* in Lower Left Corner of Plan Sub-View

Adding Rectangle Sub-View Items

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

21. *Multi-Plot* | *Insert Rectangle*. A Rectangle will appear in the middle of your sheet with the options dialogue box as shown below.

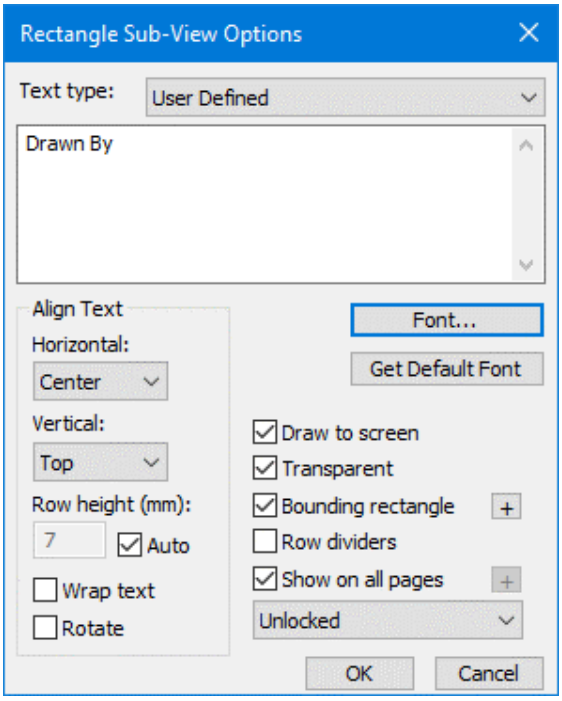

Figure 24-10: The *Rectangle Sub-View Options Dialogue*

The *Rectangle Sub-View Options* Dialogue opens automatically when you create a new rectangle, but you can also access it from a <right-click> on any rectangle sub-view and selecting *Rectangle Sub View Options* or by <double left-clicking> on the *Rectangle Sub-view.*

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

22. Type **"Drawn By"** 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 **12**. Press *OK* twice.

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

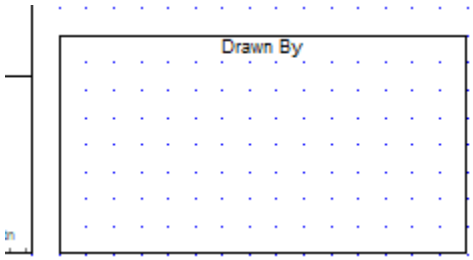

Figure 24-11: A *New Rectangle Sub-View* with Centered, Wrapped, User Defined Text in a Large Font

23. Create two more rectangles:
    - In the first, select **Print Date** from the *Text type* drop down menu.
    - In the second, select **Page X of N** from the *Text type* drop down menu.

24. Arrange the two new rectangles to fit in the first rectangle as in the figure below.

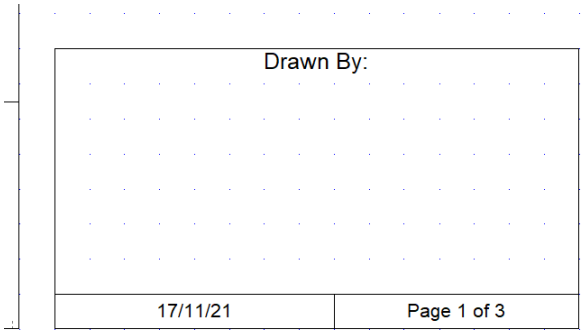

Figure 24-12: The Start of a *Title Block*

Notice how the snap to grid feature helps line up edges.

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.

25. Use the *Previous Page* and *Next Page* buttons in the *Multi-Plot* ribbon to scroll through the 3 pages. Alternatively, you could also click on the pages in the navigation panel, or use <Ctrl+b> and <Ctrl +n>.

You will notice that the Plan sub-view layout on page 5 doesn't fit. We will manually scroll the position of this page.

26. Click on Page 2 in the Multi-Plot navigation pane.
27. Select the Plan sub-view. *SubView* | *Scroll* to open the *Sub-view Options* dialogue below.

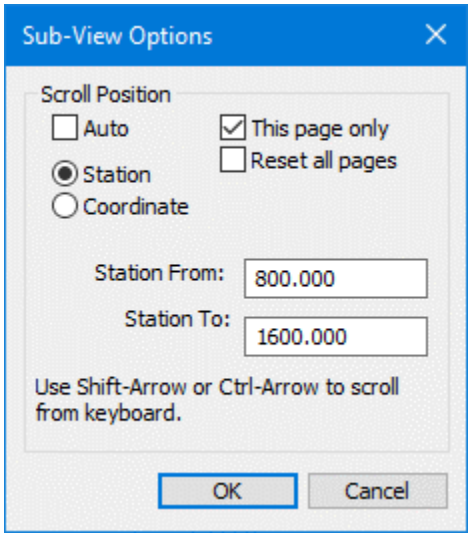

Figure 24-13: *Plan Sub-View* after Manually Scrolling and Rotating

28. Clear the *Auto* check box and check *This Page Only* (as above). Press *OK*.

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

29. Type <Shift + arrow> to scroll. Respond OK to the manual scroll prompt.
30. Use <Shift + arrow> to adjust the Plan sub-view so that the curve is fully visible and no longer located below the scale bar. Try to get the Plan sub-view to look like the one in the figure below.

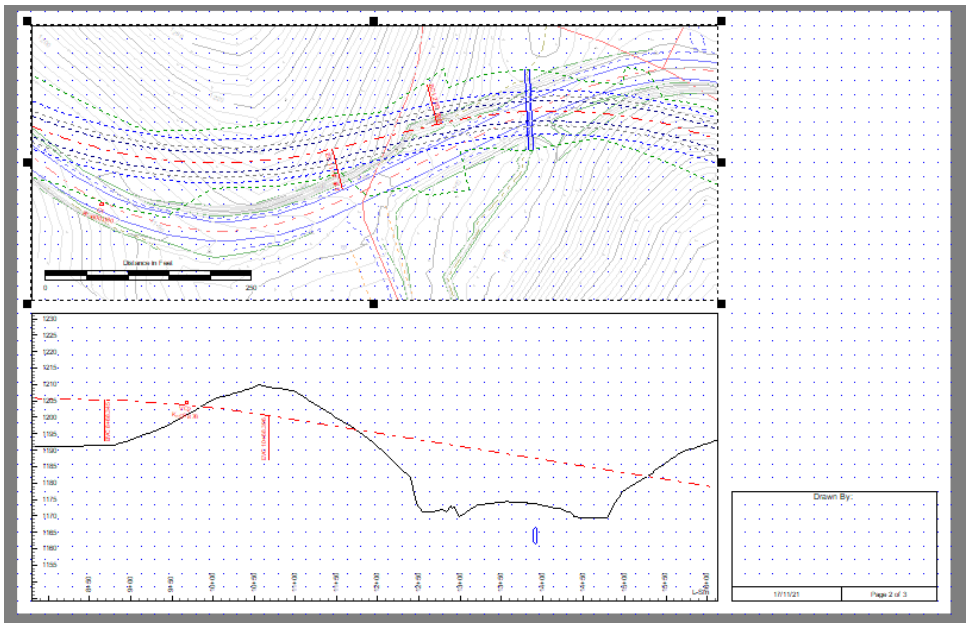

Figure 24-14: *Plan Sub-View* after Manually Scrolling and Rotating

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 28 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.

31. *File* | *Close*. Do not save changes.

Multi-Plot Chapters

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

Copy and Paste of Multi-Plot Items

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

1. *File* | *Open* <RoadEngCivil>\Location**Align Stage 10.dsnx.**
2. Select and <delete> the existing title block items so that only the Plan, Profile, scale bar remain.

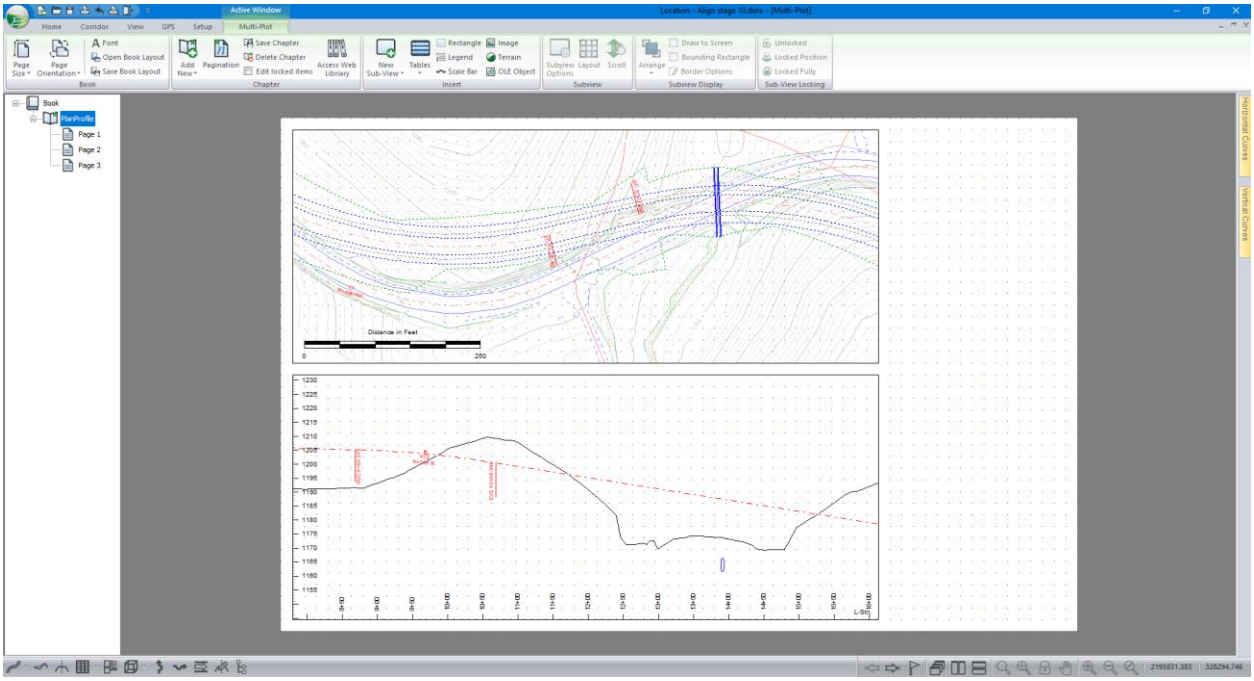

Figure 24-15: *Multi-Plot* After Removing Title Block Rectangles

3. *Multi-plot* | *Add New* Chapter | *Retrieve Other Layout*. Select *screen layout* <Defaults and Layouts>**\Training\Title Block.clt.** Press *Open*.

You will now have a second chapter with the title block we would like to copy.

4. Click and drag from the top right corner to select all the sub-views (rectangles in this case) of the title block as shown in Figure 24-16.

5. Type <Ctrl + C> to copy the selection to the clipboard (or use menu *Edit* | *Copy*).

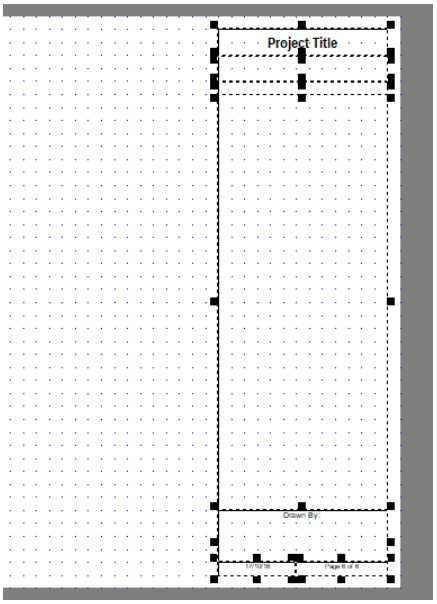

Figure 24-16: Selecting Multiple Sub-Views
(*Rectangles in this Case*) With a Mouse Click and Drag

6. Click on the PlanProfile chapter. Type <Ctrl + V> to Paste the title block on your page (or use menu *Edit* | *Paste*).

Your screen should appear as shown below:

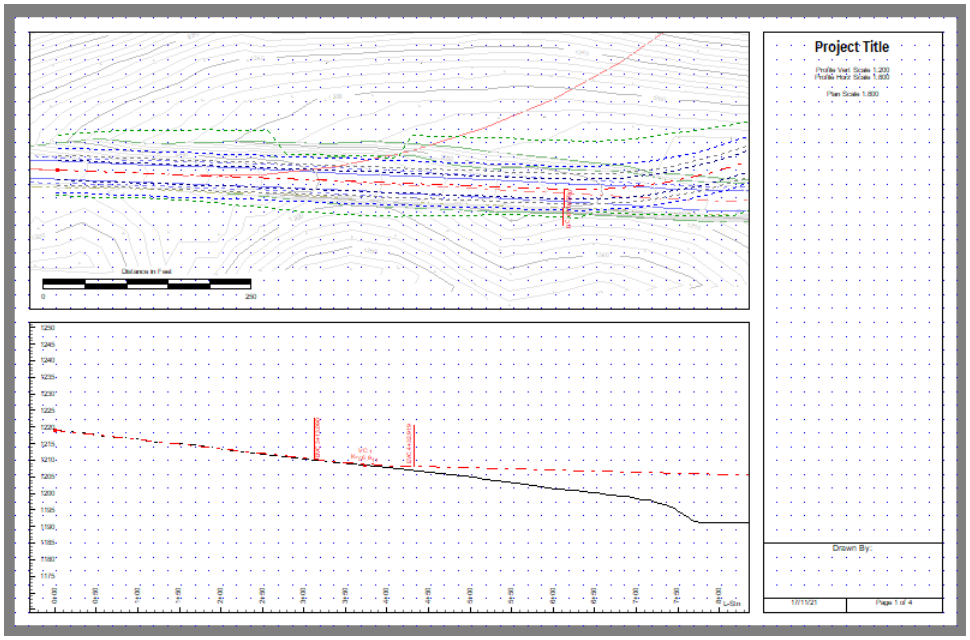

Figure 24-17: Updated Title Block

7. We can save our new Plan profile chapter layout for future use: *Multi Plot* | *Save Chapter* to open the *Save Chapter* dialogue. (optional)
8. We no longer need the Default chapter. Click on *Default in the* navigation panel, <right-click> select *Delete Chapter*.

Add a Legend

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

9. With the *PlanProfile* chapter selected, select menu *Multi-Plot* | *Insert 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.

10. <Double-click> on the legend to open *Legend Sub-View Options*.
11. Click on the *Items* tab of the *Legend Sub-View* dialogue box.
12. Select and *Remove* all but the items shown below on the top. Multiple select is allowed use <Ctrl + Click> or <Shift + Click>.

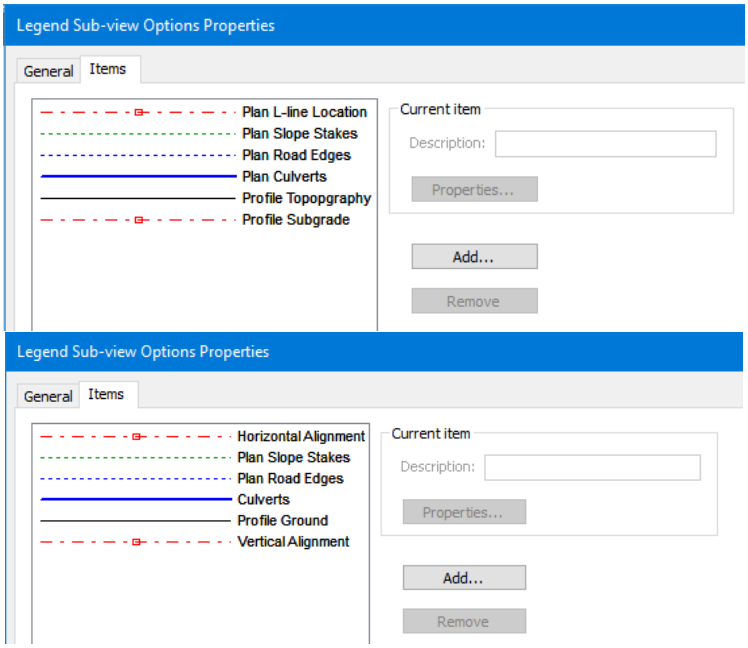

Figure 24-18: Legend with *Fewer Items* (TOP) and *New Descriptions* (BOTTOM).

13. Select items on the left one at a time and change the *Description* as in figure above on the bottom.

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.

14. Click on the *General* tab, change the *number of columns* to **1** and press *OK*.
15. Finally move and size your legend so it fits nicely on the right side of the Plan and Profile graphics. See Figure 24-19.

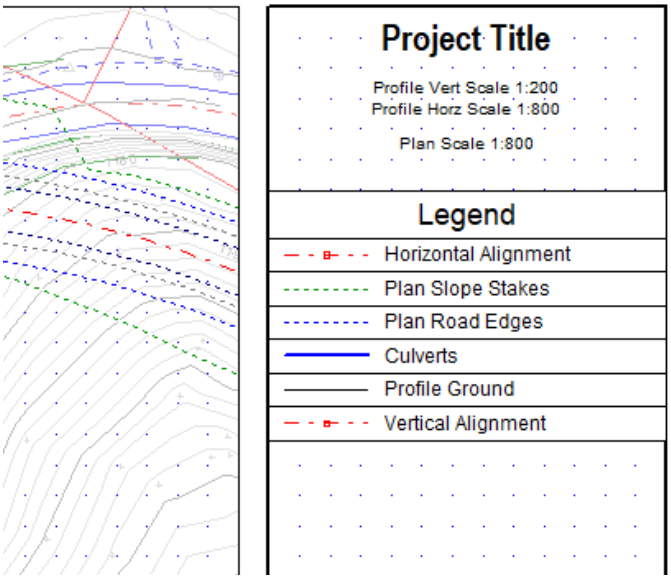

Figure 24-19: *Legend Added* to Layout

Add a Curve Table

In this section, we will create a horizontal Curve Table sub-view and examine some of its options.

16. With the *PlanProfile* chapter selected, *Multi-Plot* | *Tables* | *Horizontal Curves* to create the table.
17. Move and size the table until it fits on the right of the Plan and Profile graphics. We can make further adjustments to its layout and content:
    - <Double-click> on the Horizontal Curve Sub-View table to open Curve Table Options.
    - Change the *Column Width* to **25mm**.
    - Select *Design Points All* to include points of intersection (IPs) with no curve attached.
    - Press the *Add/Remove* button to open the *Curve Table Fields* dialogue box shown in the Figure below on the right.

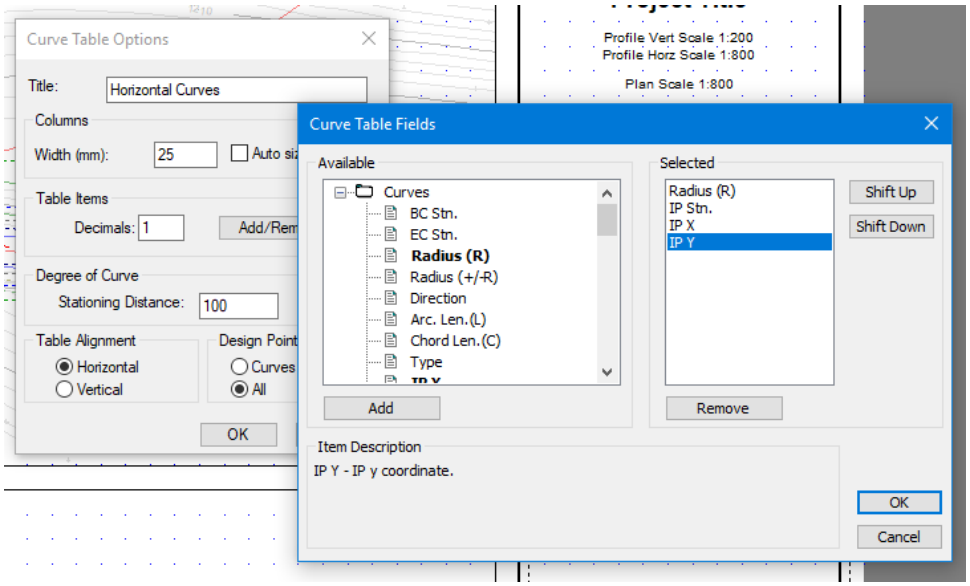

Figure 24-20: *Horizontal Curves Table Options* Dialogue Boxes

18. *Add* and *Remove* items (<double-click> works) until you have only **Radius (R)**, **IP Stn**, **IP X** and **IP Y** in the *Selected* column as in Figure 24-20.
19. Press *OK* in both dialogue boxes to see the results as shown below.

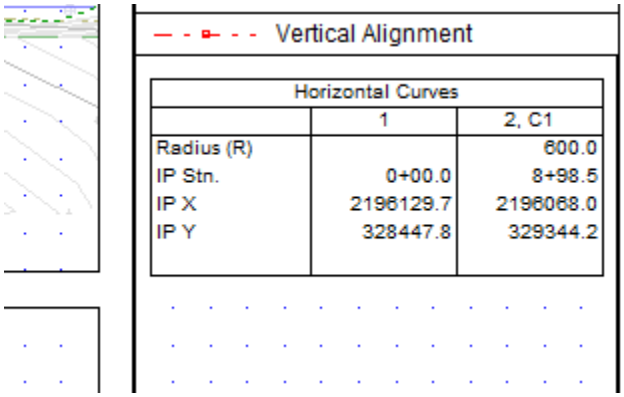

Figure 24-21: *Horizontal Curve Table* after Configuration

Saving Layouts

If you only have one chapter, you can save the layout as an individual chapter layout file (.clt): *Multi-Plot* | *Save Chapter* button.

If you have multiple chapters and want to use their layouts together in a future file, you can save the layout as an book layout file (.blt): *Multi-Plot* | *Save Book* button.

32. *File* | *Close*. Do not save changes.

25. Fixed Section Editor

Fixed sections are cross section at set stationing along the alignment of the road. These can be useful for determining cut/fill volumes and slope.

Fixed sections allow you to explicitly change the geometry of the cross-section. They can be used to quickly create specific design elements. However, fixed sections will not respond to alignment changes; therefore, they should only be used after the alignment has been finalized or for calculating existing (as-built) quantities.

1. *File* | *Open* <RoadEngCivil>\Location**Align stage 2.dsnx**
2. Retrieve screen layout: *View* | *Screen Layout*, select **training Normal.dlt** from dropdown menu.
3. Open section editor: press the *section editor* button in the *window navigation toolbar*.

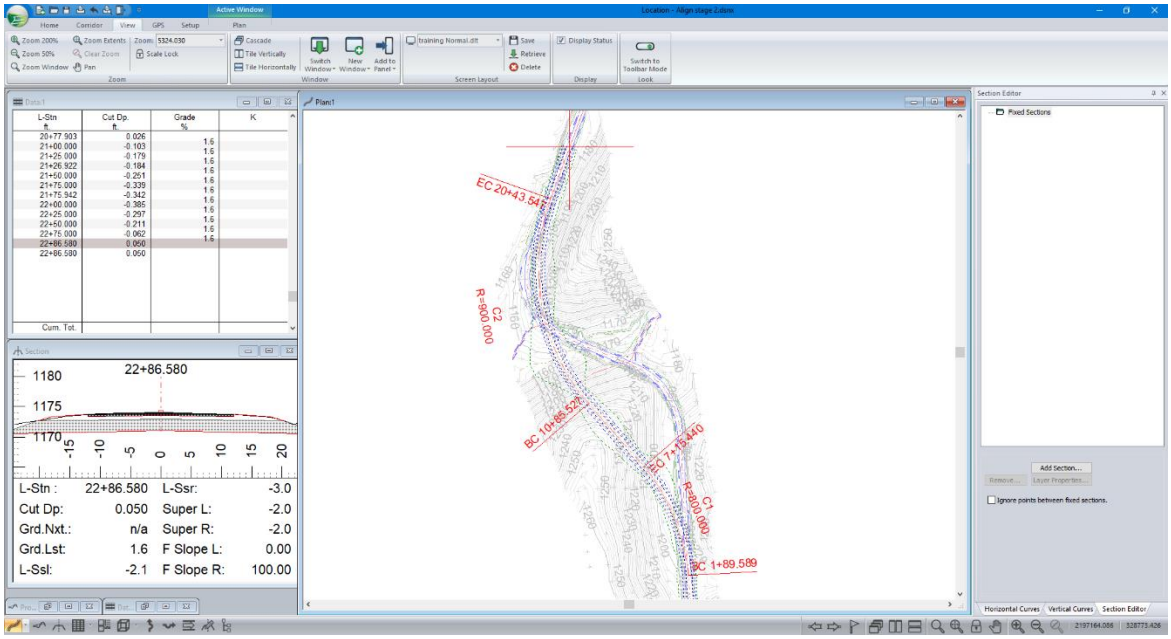

Figure 25-1 – *Screen Layout* for Cross Sections

4. To allow cross sections volume calculations to be based on equal end area calculation, *check* the box *Ignore Points Between Fixed Sections* in the lower part of *Cross Section Editor* panel.
5. Press the *Add Section…* in the *Cross Section* panel.

   Within the *Add Fixed Cross Section* dialogue, enable *Station Range, check Whole Road* and check *Start of Range*.

The next step is to assign the station interval where you would like the cross sections:

6. Press the + next to *Point Types*. Press *Select All* button. *Uncheck* Include box.
7. *<Double-click>* on *Auto interval points (25.000 Ft).*

8. Press *Properties…* button. Ensure *Automatic* is checked, set the *Interval* to **20.0**. Press *OK* three times to exit all the dialogues.
9. Each section will be displayed in Cross Section panel. Refer to Figure 25-2.

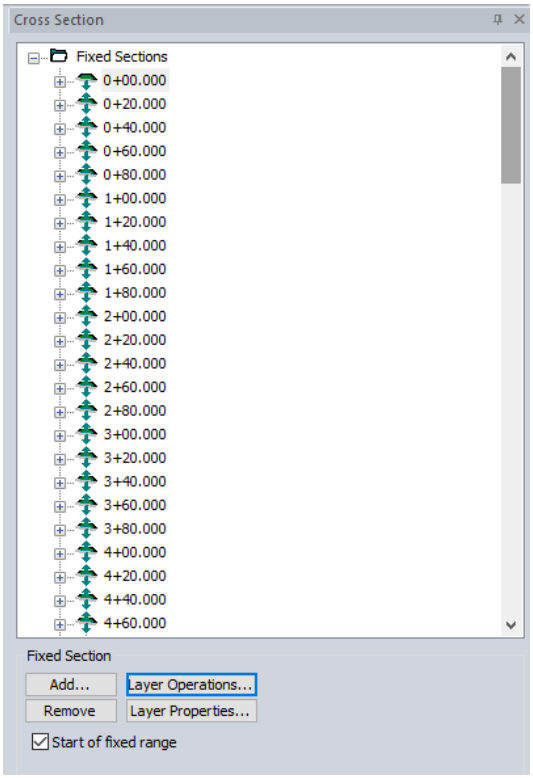

Figure 25-2 *Cross Section Panel*

One can now select each cross section and view it in the section panel.

Editing Layer of Individual Cross Sections:

10. Maximize the Section window.
11. In the *Cross Section Editor* panel, click on the station you would like to edit, in this example we will edit *0+40.000*.
12. Press the + next to the section. Click on the *plus* next to SG. Select polyline, 7-point refer to Figure 25-3.

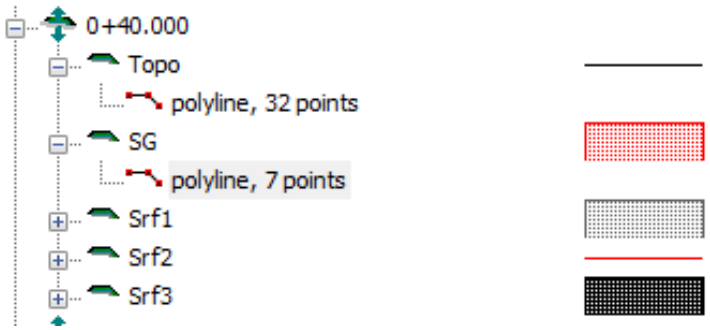

Figure 25-3 *Cross Section Editor* with *SG Polyline* Selected

13. The subgrade (SG) layer should now be highlighted in magenta in the section window.
14. <*Right-click*> in the section window | select the *Add/Edit Polyline Pt. Tool*. Click and drag the center point IP point. Refer to Figure 25-4.

One can notice the *SG Cut Volume* at Station 0+40.00 will increase.

Individual layers can be modified following this procedure.

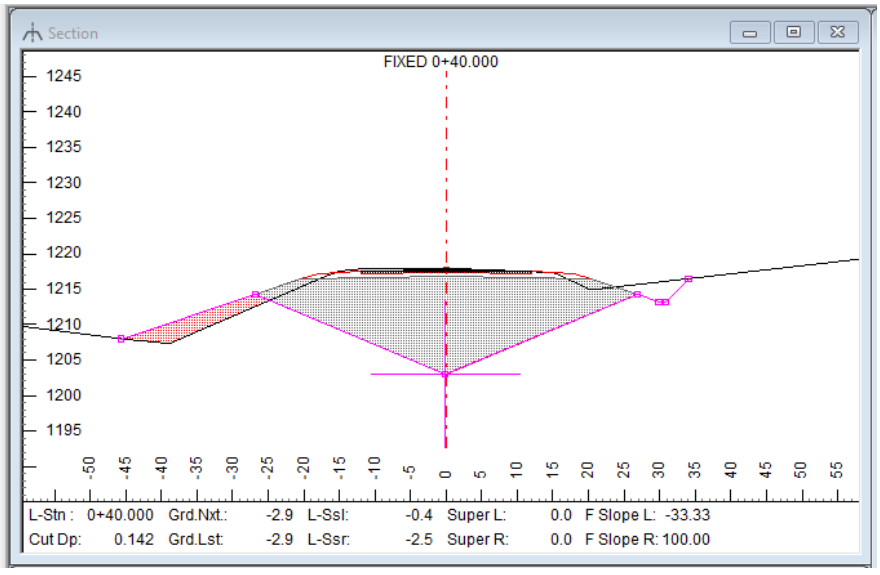

Figure 25-4 Cross Section Panel with *SG Polyline Selected*

Fixed Section 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 are processed.

The figure below shows the progression of the merged surface:

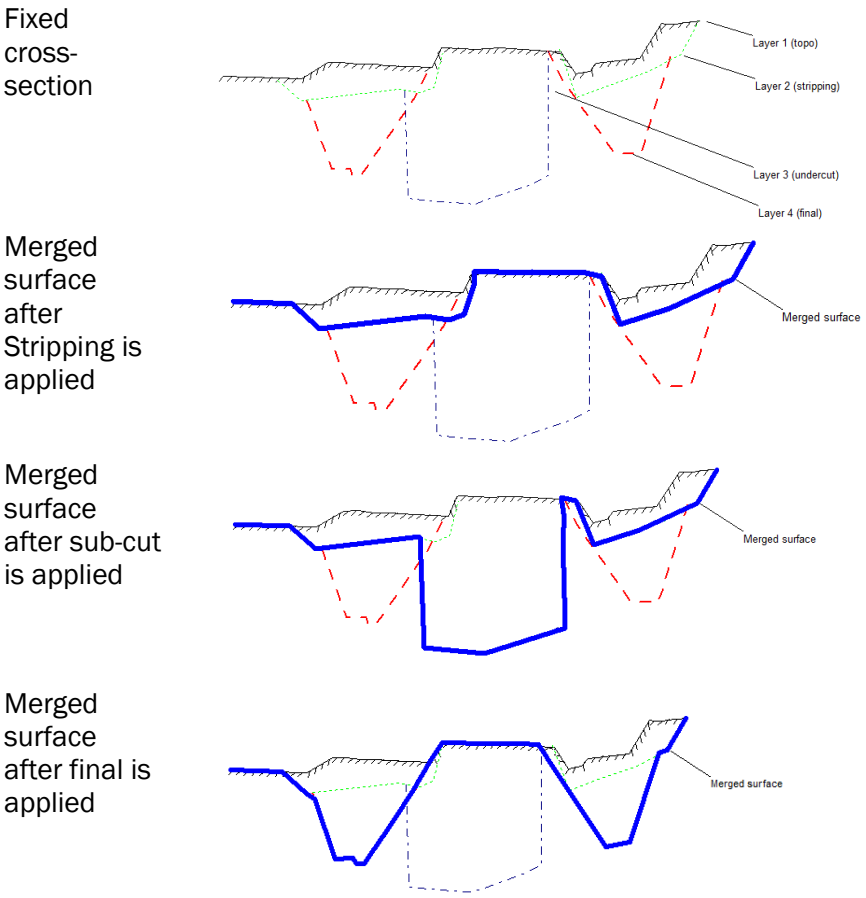

Figure 25-5: Depiction of *Merged Surface Progression*

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.

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 box checked off (see below), then all the cross sections between the two fixed sections will follow the original template—the average volume between them will not be calculated.

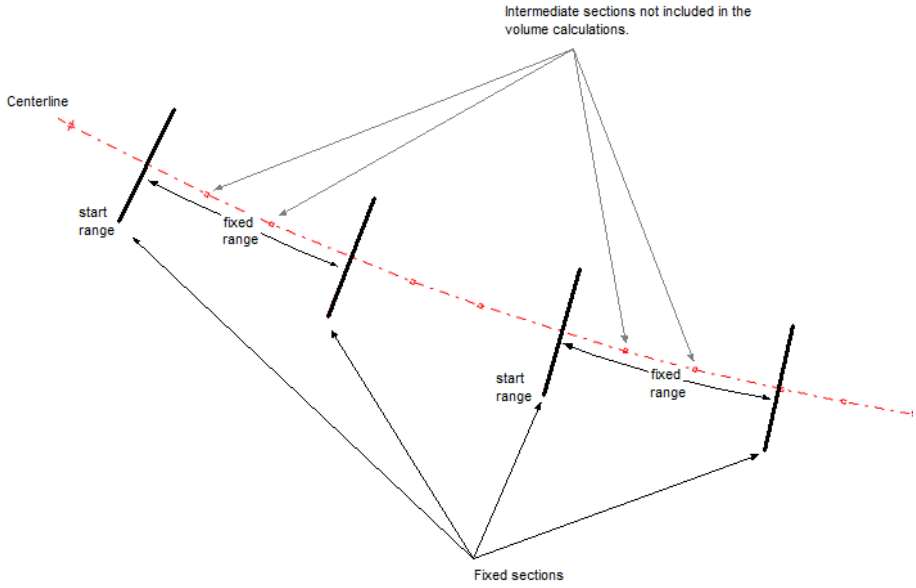

Figure 25-6: Fixed Sections and Intermediate Reporting Points

26. Creating a Composite Surface

In this section, the designed surface from the Location module will be exported and merged 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.

Note: Refer to *Getting Started* section for file install folders (**<RoadEngCivil>** and **<Defaults and Layouts>**).

Exporting Designed Surfaces

For this example, we will assume that this design is finished, and we want to export the designed surface.

1. *File* | *Open* in Location. <RoadEngCivil>\Location**Align stage 9.dsnx**.
2. Choose menu *File* | *Save As* to open the file save dialogue box.
    - Set the type to Terrain File (*.terx).
    - Change folders to <RoadEngCivil>\Composite
    - Name the output file **Road Surface XXX.terx,** where XXX is your initials (we don't want to write over the tutorial file).

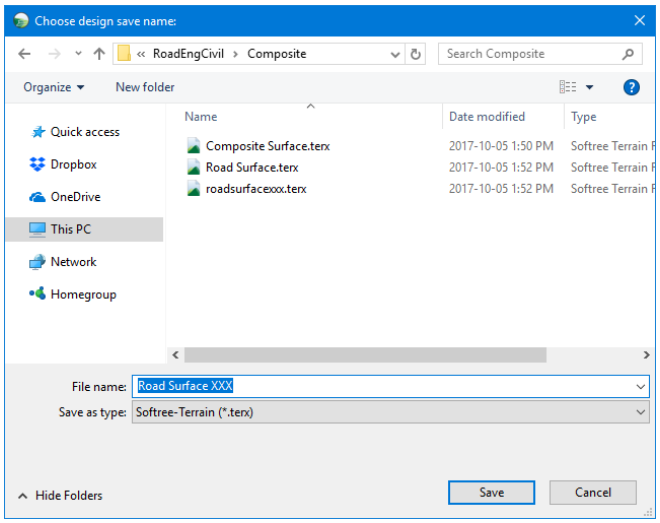

Figure 26-1: *File Save As* Dialogue Box Ready to Export a Terrain File From The Location Module

3. Press *Save*; the *Export to Terrain* options dialogue box will open (Figure 26-2).

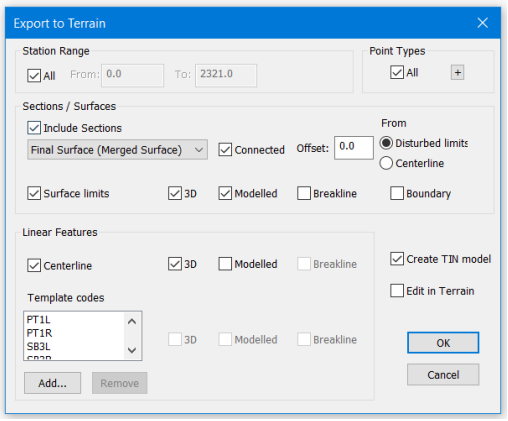

Figure 26-2: *Export To Terrain* Options Dialogue Box

Note: The Export to Terrain function can be used 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 dialogue 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 dialogue box.

4. Make sure *Final Surface (Merged Surface)* in Sections / Surfaces. We want to export the surface of the road as if it were complete.
5. Make sure that *Include Sections* is checked and that Offset: **0.0**, From *Disturbed limits*. We will export data up to the slope stake lines but no further; in other words, we will export the disturbed area.
6. Make sure that the *Surface Limits* is checked and also check the *Boundary* to the right of it. This will limit our surface to the stay within the stake lines (SS).
7. Ensure that the *Create TIN model* check box is set.
8. 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. This “connects the dots”. There are currently 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)
9. All of these, point codes will be connected together. Ensure *Breakline* box is checked (select an item to see its properties below).

10. Next add the ditch bottom features: *DIL*; *DIR*; *DOL*; *DOR* to the list of *Template codes*. Press the *Add*... button to open the dialogue box shown below. These codes represent ditch inside, outside left and right as shown in the figure above (use <*Ctrl* + *click*> to select/deselect multiple items).

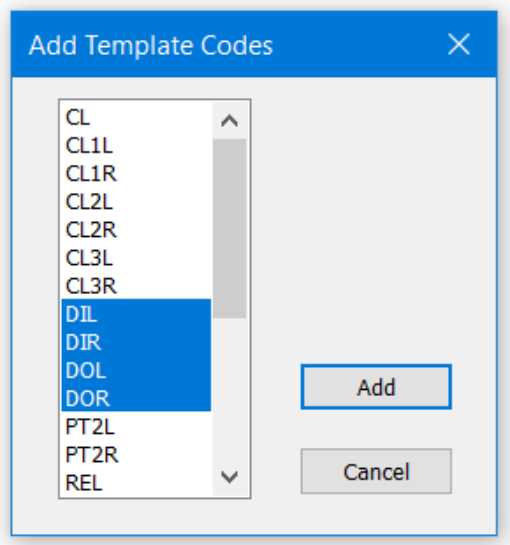

Figure 26-3: *Add Template Codes* Dialogue Box after Selecting Ditch Bottom Point Codes.

11. 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.

12. Press the *OK* button to export the Terrain file.

Merging Terrains

Next the designed surface created above will be merged with the original ground terrain to make a composite.

13. Open the Terrain module (the Location module menu *Setup* | *Terrain* button is handy).
14. *File* | *Open* <RoadEngCivil>\Location**Topo.terx**.
15. The first step is to save this file to a new location so the location design is not ruined by modifying the original ground terrain.
16. Use *File* | *Save As* to open the file save dialogue box.
    - Ensure file type is set to Softree-Terrain File (*.terx).
    - Change folders to <RoadEngCivil>**Composite**.
    - *Name* the output file **Composite Surface xx,** where *xx* is your initials (this will prevent writing over the example file).
    - Press *Save* button to copy the file.
17. Bring in file from previous exercise: *Terrain Modeling* | *Merge Terrain* button. This will open the *Merge Surface Options* Dialogue box (below).
18. *Browse…* for <RoadEngCivil>\Composite**Road Surface XXX.terx** (the file you created in the previous exercise).

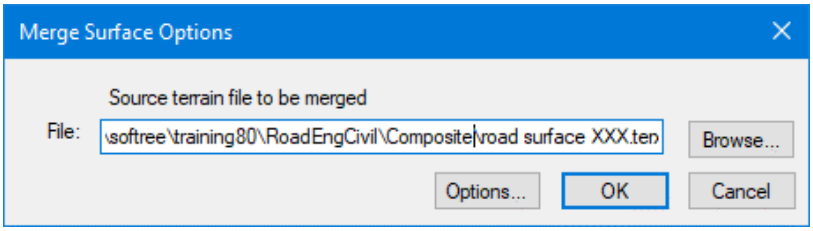

Figure 26-4: Merge Surface Dialogue Box

19. Press *OK* to merge the Terrains. A "Warning No Space for Undo" appears. Say *OK* to continue.

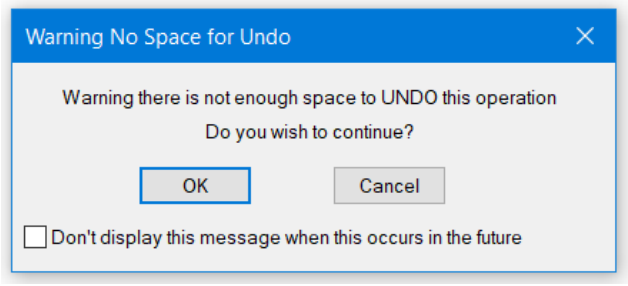

Figure 26-5: Warning Dialogue Box

20. Now we need to re-calculate the surface. *Terrain Modeling* | *Generate TIN* button. This will open the *Terrain Calculation* Dialogue box.
21. Keep the existing settings and press *OK* to recalculate the triangles and contours.
22. *View* |*New Window* | *3D* from drop down menu. A 3D window will appear on your screen.

23. *View* | *Tile Vertically* Button to see the 3D and Plan windows side by side.
24. *File* | *New* do not save changes.

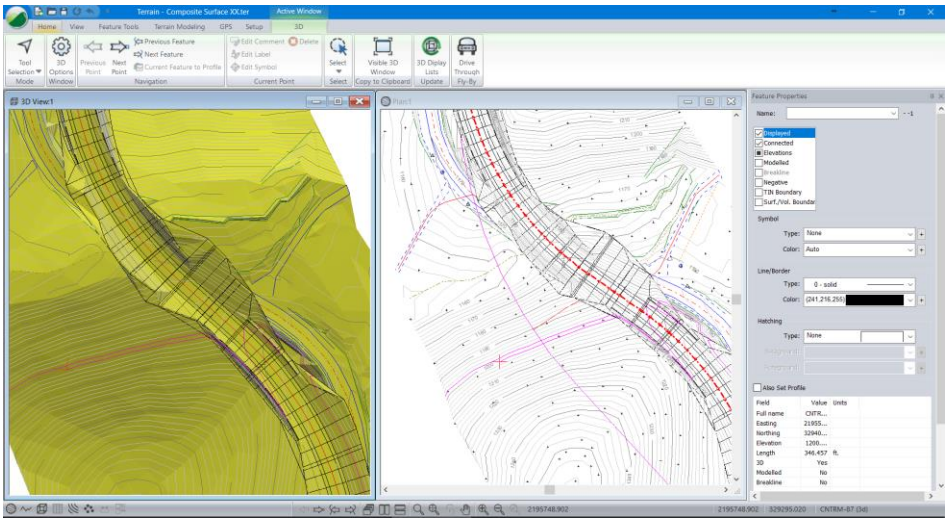

Figure 26-6: Composite Surface Showing Designed Road Merged with Original Ground.

Iterative Alignment Design

This composite surface model could now be used as the *original ground surface* for a new Location design. This could be used 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.

Index

E

F

G

H

I

J

L

M

N

O

P

Q

R

S

T

U

V

W

Z