CONNECTICUT DEPARTMENT OF TRANSPORTATION

DIGITAL DESIGN ENVIRONMENT GUIDE

CONNECT EDITION

Volume 3.1 – OpenRoads Designer Alignments

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Course Overview

In these exercises you will learn how to place Horizontal and Vertical Alignments. This guide will not document each tool that is available on the OpenRoads interface. See the online help for commands not detailed in this document.

OpenRoads Designer CONNECT Edition Help

Skills Taught

- Select the proper seed file to create a Base Model
- Align a file so it's in the proper geospatial location
- Horizontal Centerline Layout
- Horizontal Alignment Annotation
- Vertical Profile Creation

Exercise 1 – Creating a Base Model Geometry File

This section instructs users how to create an alignment base model design file for a project. The alignment base model file will reside within the project folder structure the ***…|Highways|Base_Models*** folder. Depending on the complexity of the project one or more alignment files can be created. Horizontal alignments and vertical alignments (profiles) are stored in the alignment DGN file(s). References files such as survey and terrain files are attached without nesting.

1.1 Startup

Before attempting to open or create DGN files users should make sure the following is in place:

1. CTDOT users should have the CTDOT CONNECT DDE synced through SharePoint with the COMPASS Project Synced along with the CAD Configuration.
2. Consultants should have CTDOT DDE properly installed or be syncing to the CTDOT DDE SharePoint/COMPASS system.
3. Make note of the **Coordinate System** you will be working in. If you have existing survey data, you will need to find out what system is being used **(NAD 83/NAVD 88 or NAD 27/NAVD 29).**
4. Log on to the CONNECTION Client. Bentley Connect licensing requires users to log into their Bentley account to secure a software license. CTDOT users should log in using your CTDOT email address and Bentley password.If you do not see the dialog box, select the ^ icon on the bottom Windows Screen. Click on the Connection Client Icon and select **Open**.

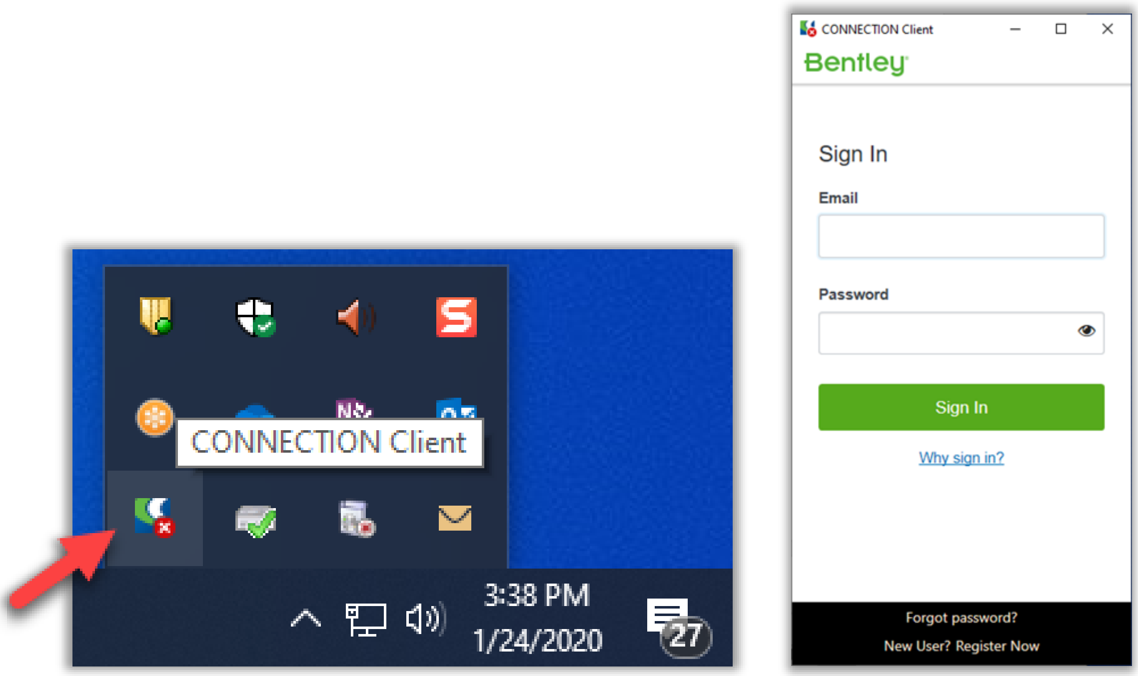


Figure 1 CONNECTION Client System tray

5. Launch the Application.
   - **Consultants**
     Start the software via an appropriate **CTDOT DDE** icon
   - **CTDOT employees**
     On your desktop double click on the **CAD Accounting** icon.

6. On the CT DOT Accounting Menu there will be select ***Compass OpenRoads CE***
   In the ***Run Program*** field select the needed program, the ***Available Account*** (funding source) and ***Resource Type***. Click on the **Start** button to load the program.

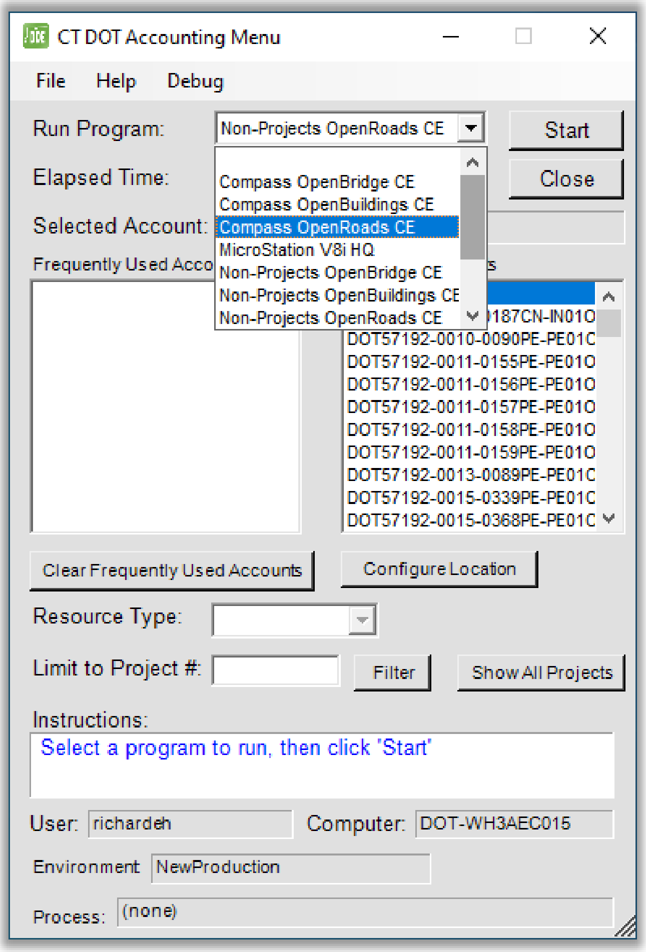


Figure 2 CAD Accounting dialog box

7. After launching the program, a Welcome Screen for ***OpenRoads Designer*** will appear.

8. Select **Custom Configuration**, using the small drop-down arrows select the Workspace **CT_Workspace**, the needed **WorkSet** and **Role**.

Note: *If you do not see the Project Number listed under Workset, please request a Compass/CAD Setup using this link [New CAD Project Request]*

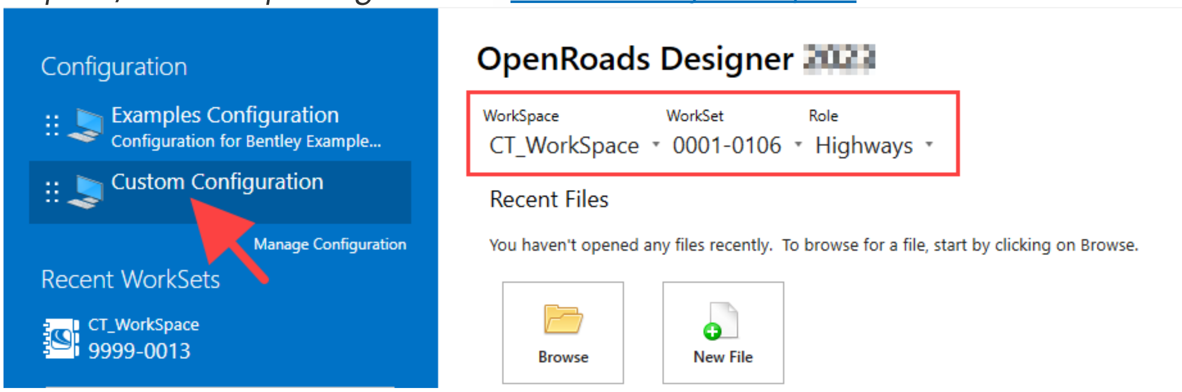


Figure 3

1.2 Creating a New File

1. Select the **New File icon**. Create a DGN file from the civil 2D seed and save it to the ***Base_Models*** folder. Use the file-naming conventions as described in Volume 16. Example: ***HW_CB_1234_1234_Alignments01.dgn***

> **Warning:** Do not copy DGN files created with V8i SELECTseries or InRoads SS2, SS3, SS4, or SS10 to the new CTDOT CONNECT Project/WorkSet folders.

2. On the New dialog box click the **Browse** button to select the proper seed file. ***…CT_Configuration|Organization|Seed|Road|Seed2D - CT RoadDesign.dgn***

   If the survey was done in an old Datum, use the 2D Seed Files in this folder …***CT_Configuration|Organization|Seed|GCS|***

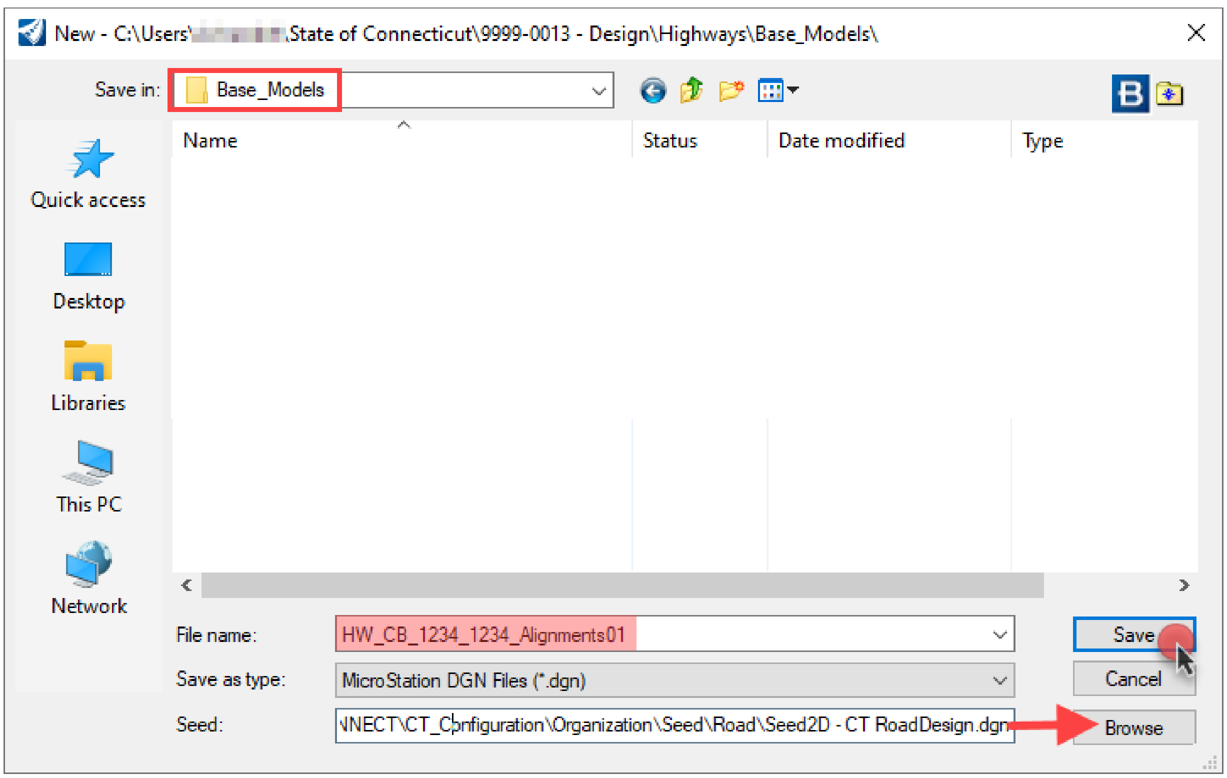


Figure 4 New File

3. After the DGN file is created open File Explorer and browse to the file, **right click hold** and select **View online**.

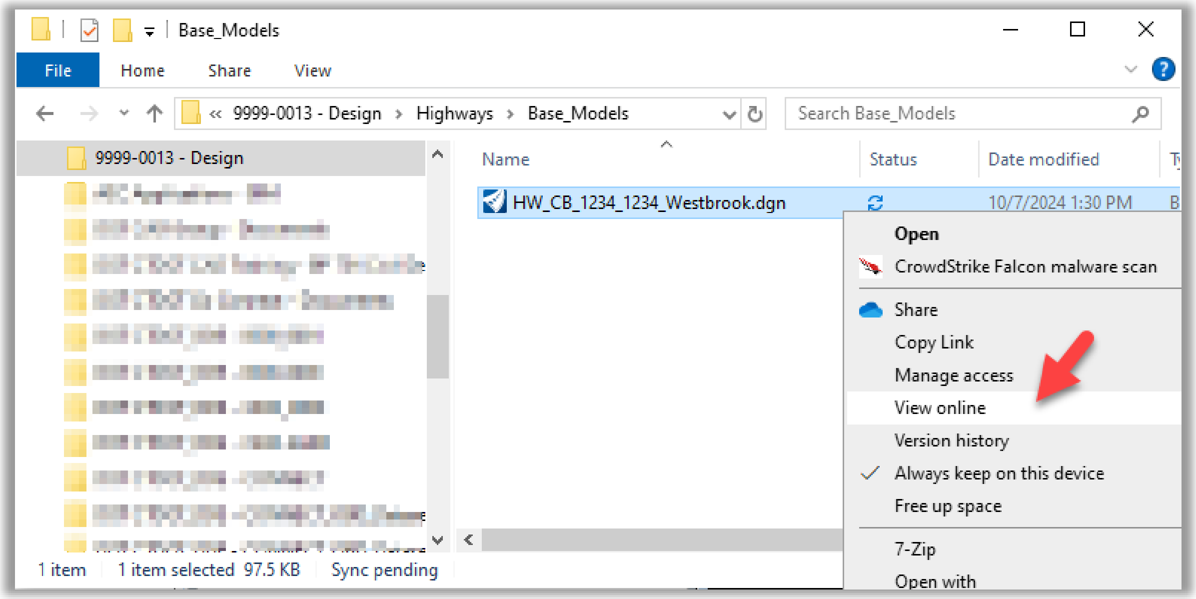


Figure 5 File Explorer View online tool

4. The Projects SharePoint site will open, sort by **Date**, click on the **three dots**, select ***More > Check Out.***

   Note: *When you are done working on the DGN file, exit the program and go back to the SharePoint Site and* ***Check In*** *the file.*

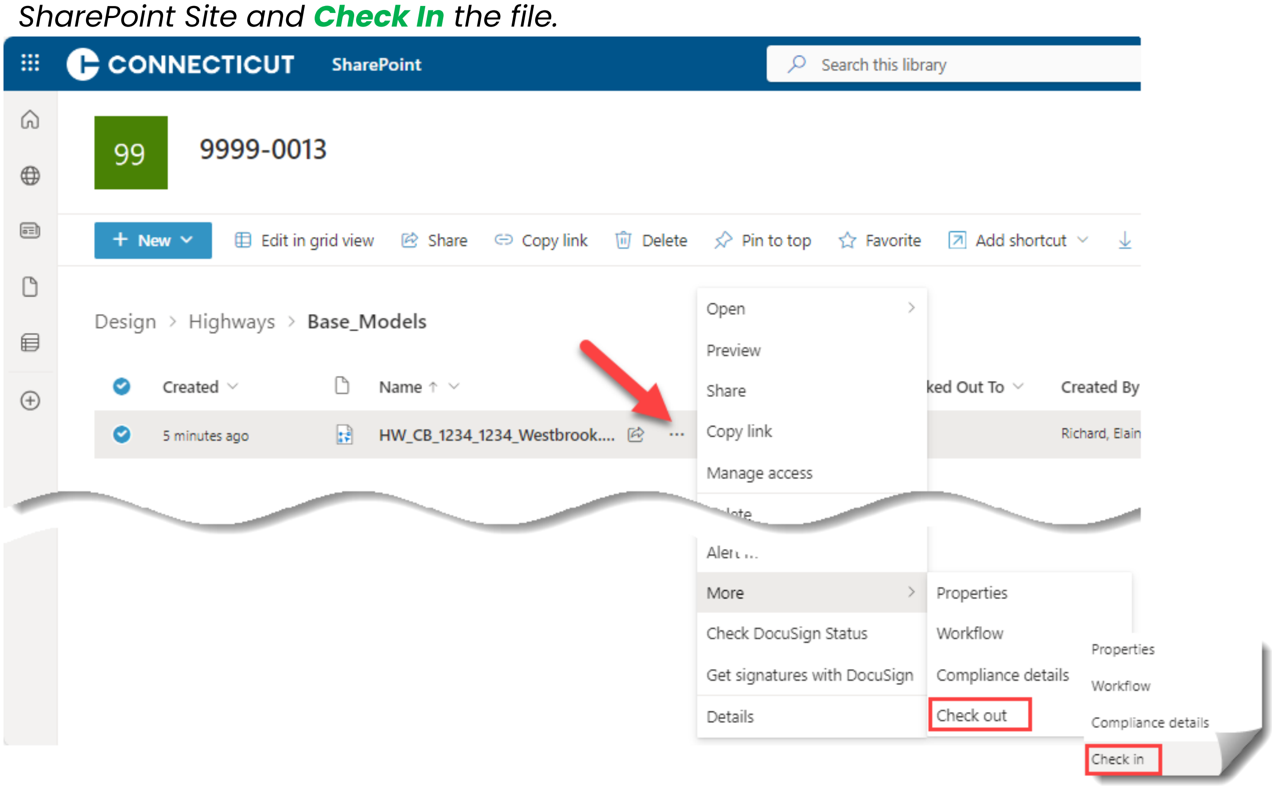


Figure 6 SharePoint Check out

1.3 Referencing

1. Reference in the survey and terrain file(s) using **no nesting**. Turn off levels as needed.
2. ***TIP*:** 'Save Settings' when all levels are turned off as needed.
3. Activate the terrain by clicking on the terrain boundary, hover over the boundary and from the pop-up menu select/click the **Set Terrain Active** tool.
4. Click again on the terrain boundary and set the override symbology to **Yes**. Then to help with the horizontal alignment creation, you can turn on the contours. Save Settings.

Figure 7

5. Review the file and **Save Settings**.

Exercise 2 - OpenRoads Review

2.1 General Tools and Interface

2.1.1 Tools

Tools can be accessed a variety of ways, including the Ribbon, Search and Context Sensitive Menus. Ribbons are arranged by Workflows, which are selected from a pull-down in the upper left corner of the software.

To activate the OpenRoads Modeling workflow use the pick list in the upper left corner if it is not already active.

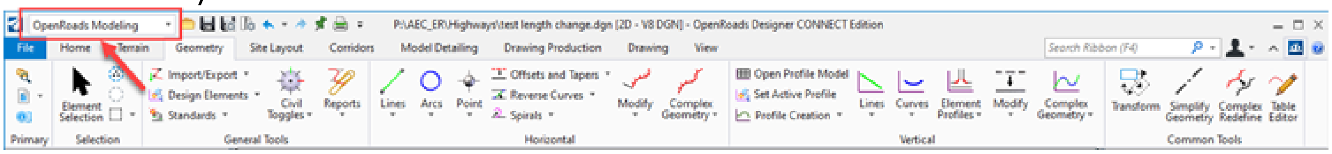


Figure 8

The ribbon menu will reflect the OpenRoads Modeling tools. The tools are organized into categories on the ribbon tabs.

- **Home** - Common tools such as Attributes, Explorer, Attach Reference Tools, Models and Level Display.
- **Terrain** - Terrain modeling tools.
- **Geometry** - Civil AccuDraw and geometry tools.
- **Site Layout** - Parking, pad, pathway, vertical geometry and grading site design tools
- **Corridors** - Superelevation and corridor modeling tools.
- **Model Detailing** - Civil Cells and 3D tools (Linear Templates, Surface Templates, etc.).
- **Drawing Production** - Saved views, notes, text, annotations, and plans production (cross section, plan, and profile) tools.
- **Drawing** - Commonly used MicroStation drawing tools.
- **View** - Commonly used view control tools.

A Search Ribbon tool is also available at the right side of the title bar. Use it to find tools across the multiple ribbons. When you are not sure where to find a tool, simply type the command in the Search Ribbon field and a search will be performed across all ribbon menus.

2.1.2 Models and Views

"Real World" Civil Engineering starts with a horizontal plane and a set of engineering rules. Vertical design follows the horizontal paths and follows a different path when it come to rules. For example, horizontal geometry typically uses circular arcs, vertical geometry commonly uses parabolas. Combine the two geometries to get 3D results.

Some OpenRoads dgn's have a single 2D or 3D models, but many design files have Multiple Models.

The old InRoads software used an "external" proprietary file (.alg) to store geometry data. OpenRoads Designer exclusively uses the .dgn format (no external files) to store geometry data, the software stores geometry types in different models within a .dgn file.

- Horizontal Geometry is stored in the default 2D Model.
- Vertical Geometry is stored in a Profile Model view and associated to the horizontal geometry.

Note: *If a horizontal element has vertical information - one or more Profiles - then it "owns" a single Profile Model containing all its profile information. Profile Models are visible only by using the View Profile Model tool).*

Many design files contain a 2D and 3D Model, typically named Default and Default-3D, respectively. 3D Elements are created, managed and stored in the default 3D Model.

2.1.3 View Control

In OpenRoads Designer CONNECT Edition, an enhanced right-click view control option has been added.

- Enables quick display of multiple views in one model including plan, 3D, dynamic profiles, dynamic cross sections and superelevation.
- The "1 View" option will reset a view back to default if accidently set to a dynamic cross section or profile view.
- When selecting a dynamic profile or cross section view, the grid will be turned on for the view.
- Right Click in the View 1 Window and select the desired preset View Control window set-up.

After Selecting the desired tool select Save Settings.

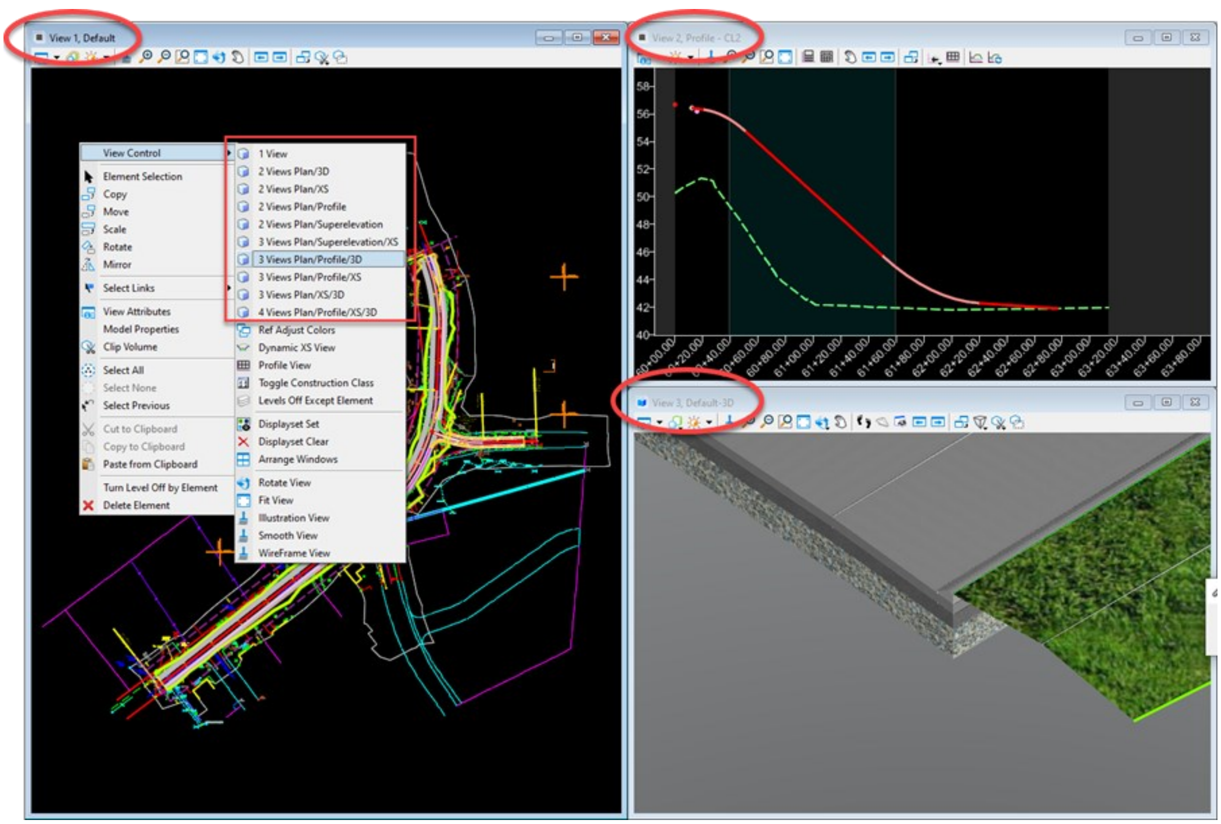


Figure 9

2.1.4 Multi-Model View

Using the View Control tools creates a Multi-Model View. This can also be done manually. Open two views, on the View Attributes dialog box set View 1 to Default and View 2 to Default-3D. The active model will display both models in two separate views. Issue a Save Settings before moving on.

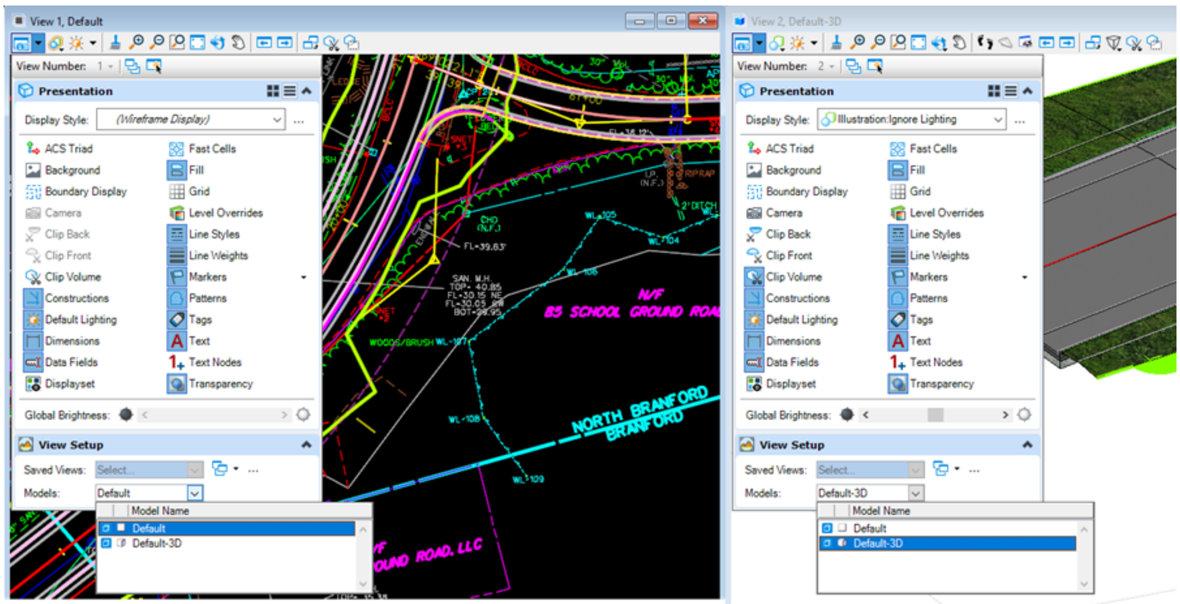


Figure 10

2.1.5 Switching Between Models

Use the Set Active View Group tool to move between Models in your DGN file.

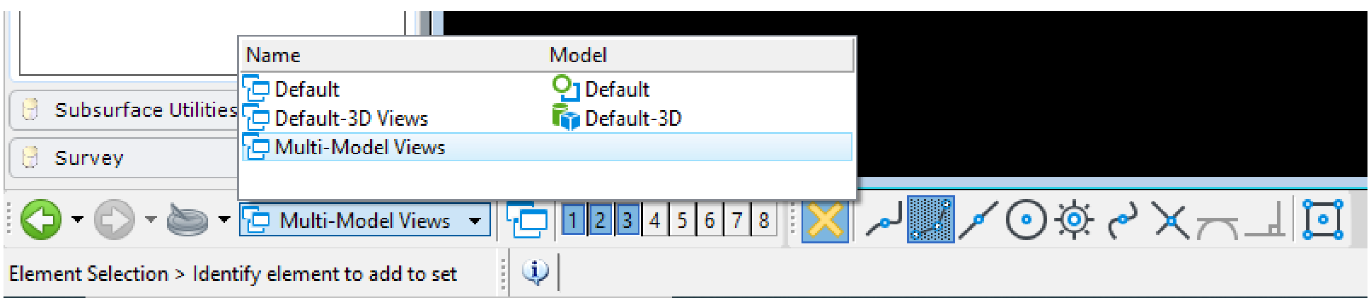
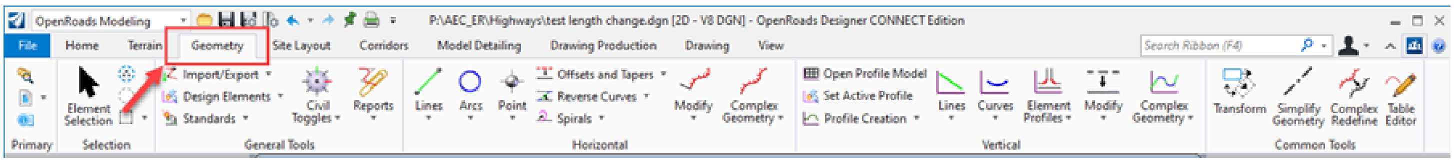


Figure 11

2.2 Geometry Tools

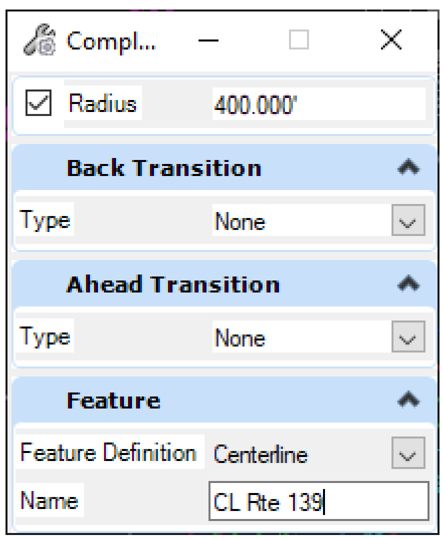
2.2.1 Feature Naming

The OpenRoads Geometry commands are selected from the **Geometry** tab of the **OpenRoads Modeling** workflow. The commands are separated into four main categories, **General Tools**, **Horizontal**, **Vertical** and **Common Tools** as shown below.

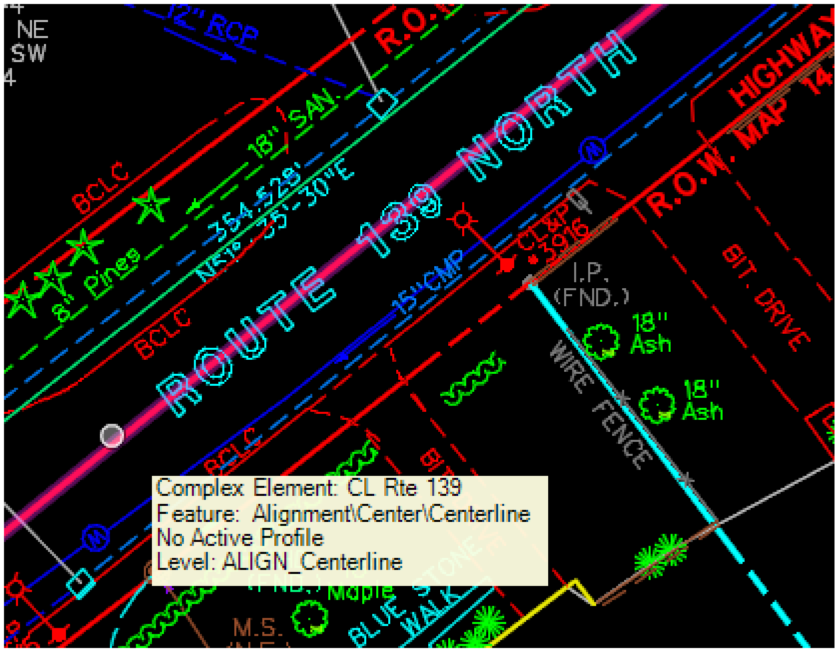


As civil geometry elements are placed, each element is given a feature name. The Name field displays the seed name that has been assigned to the selected feature definition.

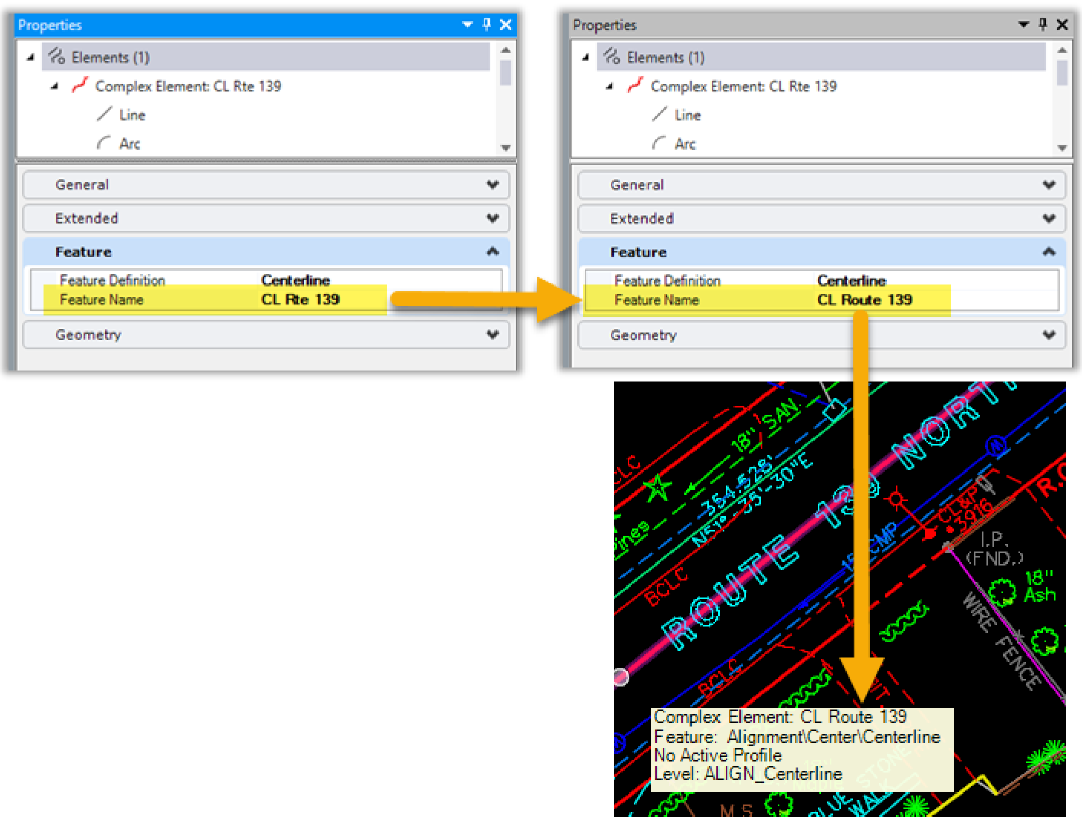
In the example at below, the **Complex By PI** command has been selected. The Centerline feature definition has a seed Name prefix of ***CL*** defined. If the name value is not changed, the software will automatically increment the name of each line placed. For example, the first line placed is named “CL”, the second is named “CL1”, etc., is it recommended to give a short descripted name as shown below.



The element name can be reviewed by using the **Select Element** tool and floating the cursor over the element as shown below (you do not need to select the element).



The **Feature Definition** and **Feature Name** for a selected element can be edited in the Properties dialog as shown below.

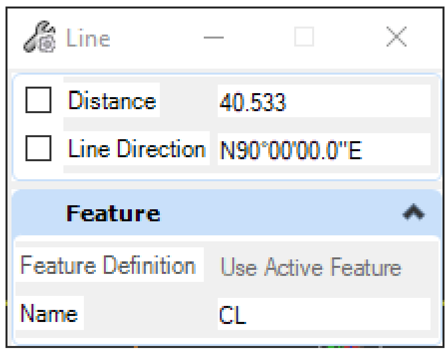


2.2.2 Feature Definition Toggle Bar

The **Feature Definition Toggle Bar** is accessed from the ribbon by selecting **OpenRoads Modeling > Geometry > Standards > Feature Definition Toolbar**.

Toggle on the first icon to automatically assign the selected feature as the active feature as you use various OpenRoads Geometry commands.

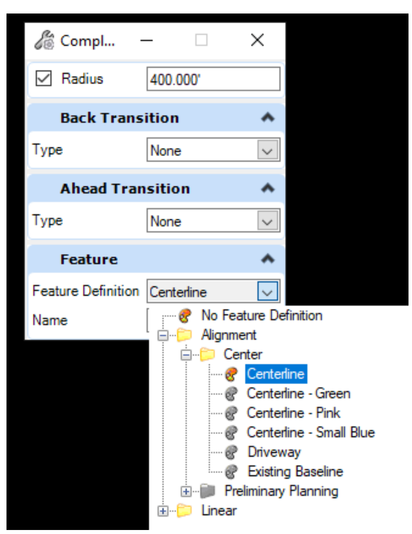
In the example below, the **Line Between Points** command is selected. The **Feature Definition** is defined by the active feature selected in the **Feature Definition Toggle Bar**.



2.2.3 Centerlines

Standard centerlines will use the following Feature Definition:

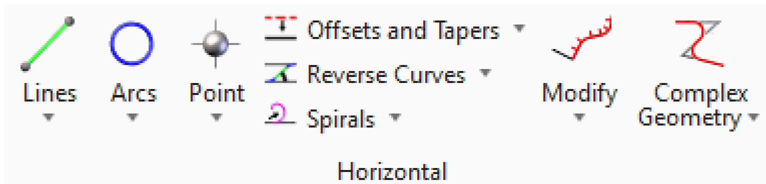
- Centerlines: **Alignment/Center/Centerline**
- Driveway Centerlines: **Alignment/Center/Driveway**



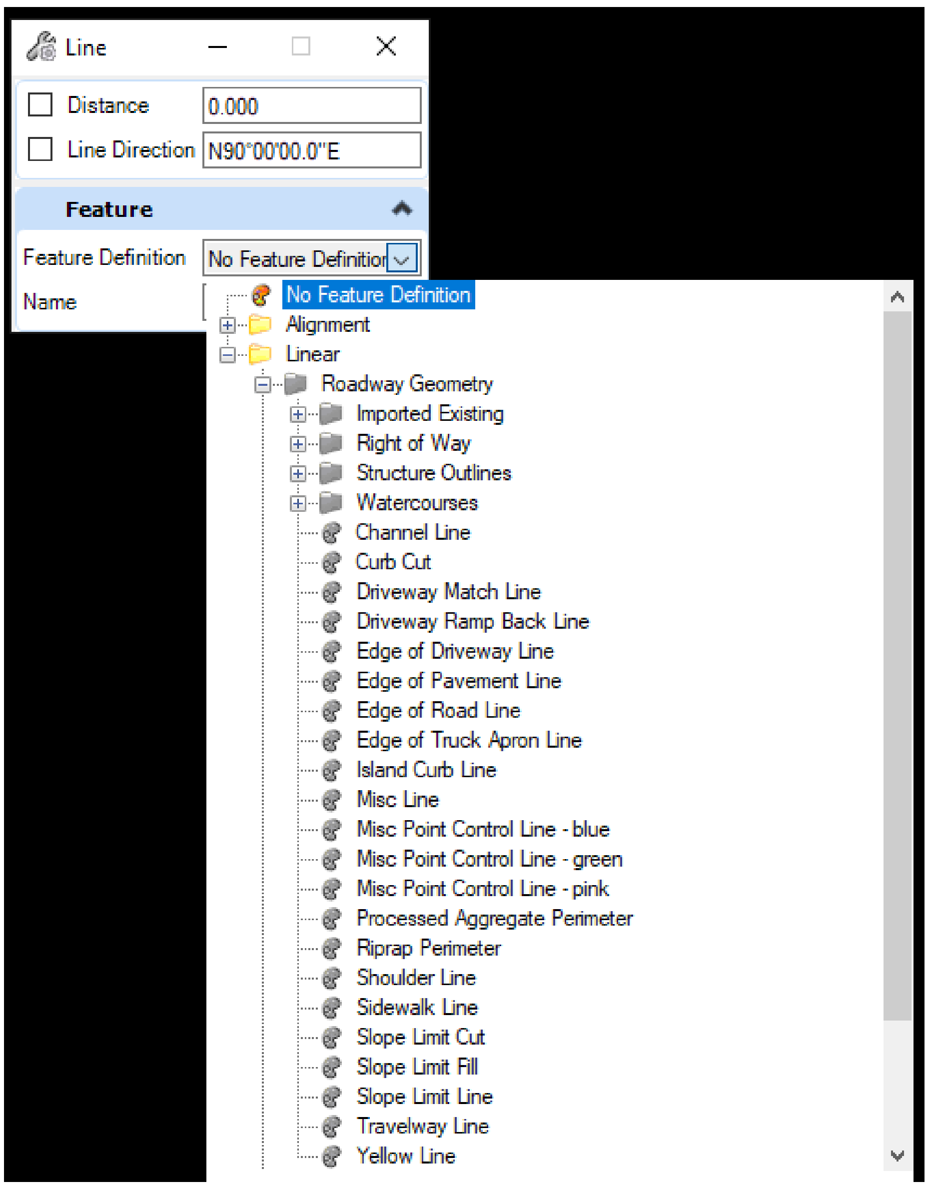
Miscellaneous alignment feature definitions are available if needed for out of the ordinary circumstances.

2.2.4 Non-Centerline Roadway Geometry

Non Centerlines Geometry can be placed with any of the Horizontal tools.



These feature definitions can be found in the **liner/Roadway Geometry** folder.

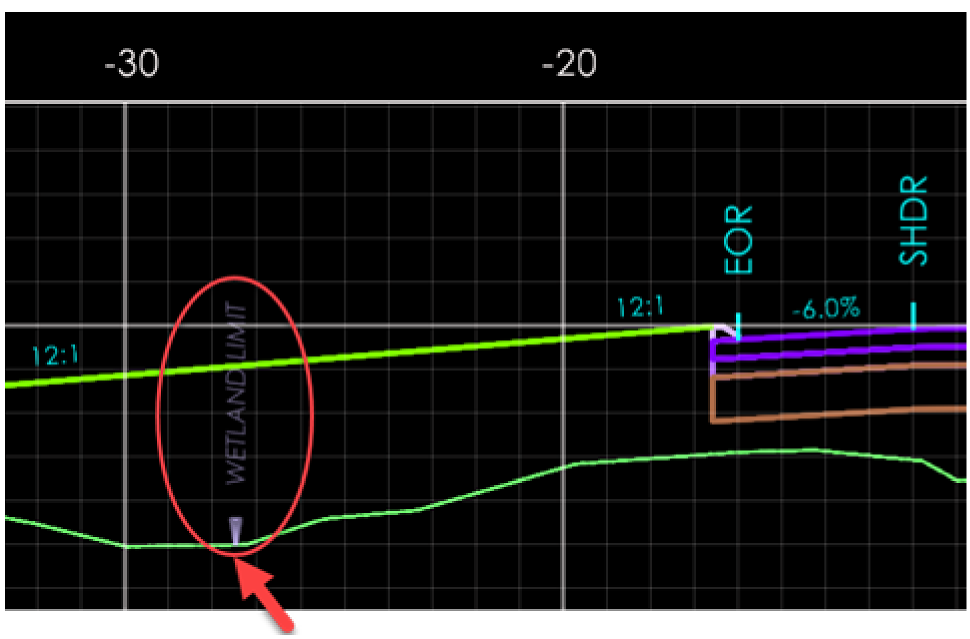


Feature Definitions are further broken down into folders for:

- Placing Right of Way lines
- Placing Watercourses
- Placing Structure Outlines
- Importing Existing **Warning:** Currently OpenRoads Terrains and Survey Features do not have the same functionality as InRoads. With InRoads existing linear features in a DTM could be displayed as a crossing feature on cross section sheets. This is not available with OpenRoads, crossing features will need to be brought in as a Geometry Feature in order to be displayed on Cross Sections. Bentley Recommends using the **3D by Plan Profile** tool to recreate the needed crossing features.

2.2.5 Existing Features

For a quick overview on Importing Existing Features watch this video of a Wetland Line being placed over the Existing Survey Feature.



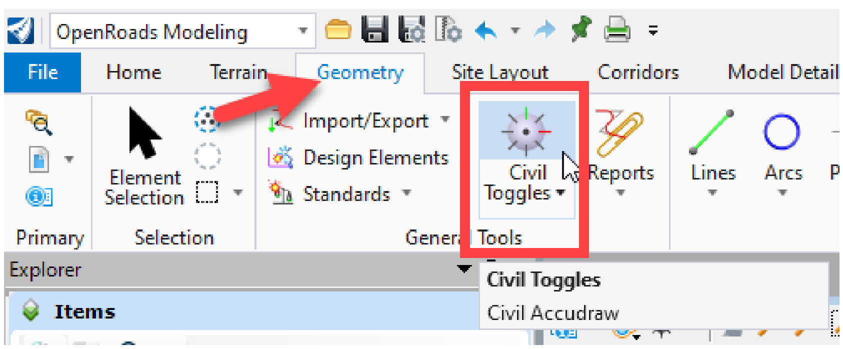
2.2.6 Civil Accudraw

Placing elements in a 2D model is like manual drafting — all elements appear on the same plane, the sheet of paper.

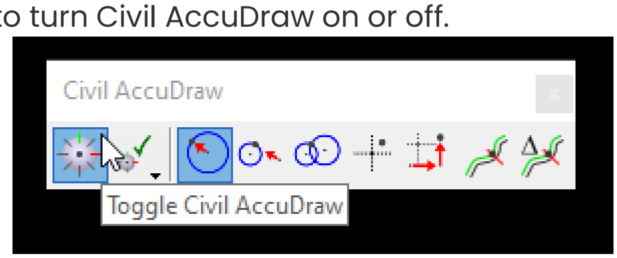
- In 3D, you place elements in space — horizontally (for example, a floor), vertically (for example, a wall), or at any other angle or direction (for example, a sloping roof).
- By default, data points in a 3D model are placed at the view's Active Depth. Where you snap a tentative point or place a data point in a blank part of a view, it will be located at the active depth. You can, however, snap a tentative point to an existing element at any depth in a view. When you accept such tentative points, the data point is placed at the level of the snap point.

Civil AccuDraw and its drawing plane let you place elements away from the active depth. Often this improves productivity, since you need not constantly change the active depth.

Civil Accudraw is activated by selecting the Civil Accudraw icon from the Geometry tab as shown below.



The first icon is used to turn Civil AccuDraw on or off.

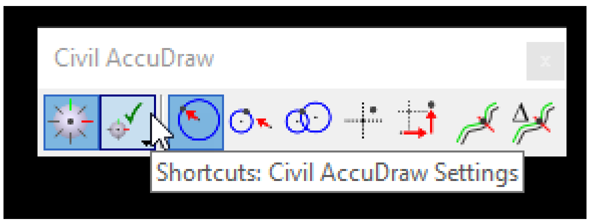


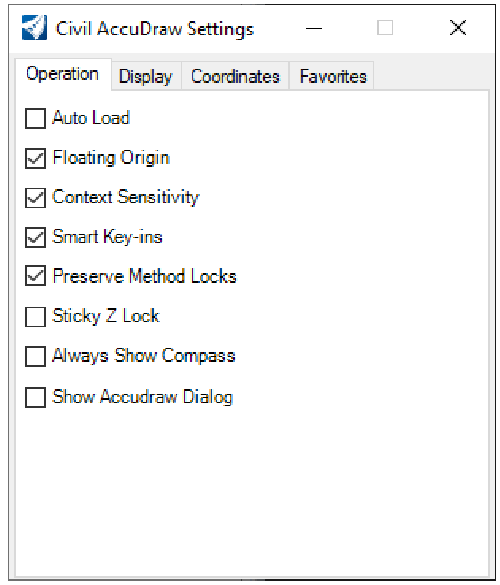
Civil AccuDraw performs many of the same functions as MicroStation AccuDraw but has greatly expanded capabilities for the civil designer.

Warning: *Simultaneous use of Civil AccuDraw and MicroStation AccuDraw will cause errors. Close MicroStation AccuDraw when using Civil AccuDraw.*

2.2.6.1 Civil AccuDraw Settings

Civil AccuDraw Settings are accessed by selecting second icon as shown below.





Auto Load – Toggle this option ON to automatically load Civil AccuDraw.

Floating Origin - *CTDOT recommends turning this option off.*
ON: the origin moves to the last point placed. OFF: the origin remains fixed.

Context Sensitivity - *CTDOT recommends turning this option off.*
ON: the compass rotates in response to the context of various tools.

Smart Key-Ins -
ON: Civil AccuDraw interprets a number as positive or negative, depending on the direction of the pointer from the compass.
XY ordinates only: Smart Key-ins cause Civil AccuDraw to move the focus to either the x or the y field depending on the pointer position.

Preserve Method Locks -
ON: the locked values remain locked after switching the ordinate methods. For example, using station-offset ordinates and a lock station of 1+00, switch to distance-direction ordinates. Station 1+00 will remain locked while inputting distance-direction. The resulting effect will be a distance-direction-stationlock.

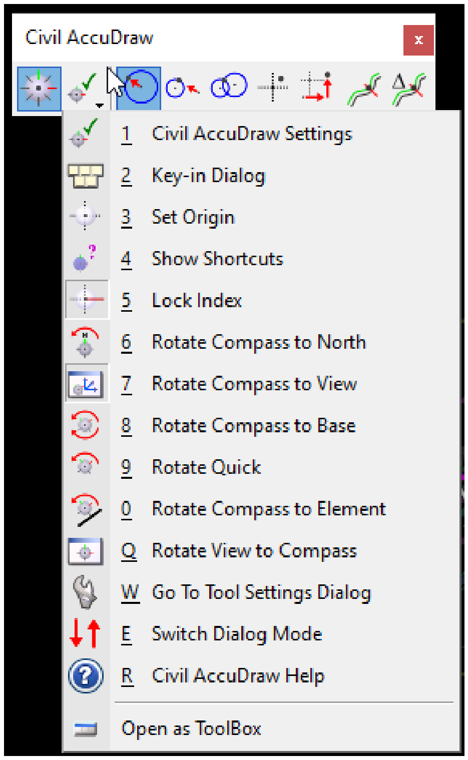
Sticky Z Lock -
ON: inputs to Z ordinates remain locked (sticky) until they are changed. This is useful in a 3D DGN to control the Z ordinate. For example, input 100.00 for the Z ordinate and the value remains constant until changed.
OFF: The Z ordinate follows the cursor dynamics.

This also controls whether the origin for delta Z measurements follows the XY origin or is independent of XY.
ON: the origin for delta Z is completely independent of the XY origin. For example, the user can set the origin for dXdY measurements from point A and set dZ to be measured from a second point B. The Z origin does not change until reset by the user. The effect will be apparent when drawing a linestring. When drawing a linestring, the dynamic orientation of the compass will adjust to align with the most recent line segment. The Z origin will remain fixed wherever the user placed it at in the beginning.
Note: When Accudraw ordinates are set to X,Y the sticky Z toggle does not apply. This is because, by definition, XY mode is always assumed to be absolute.

Always Show Compass -
ON: the compass is always visible.
OFF: the compass is only visible when the appropriate input is required.

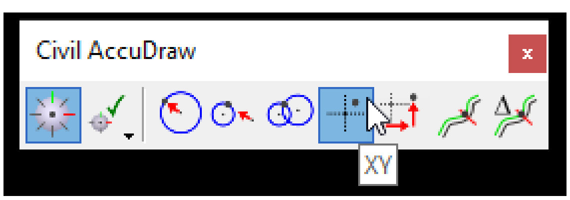
2.2.6.2 Civil AccuDraw Tools

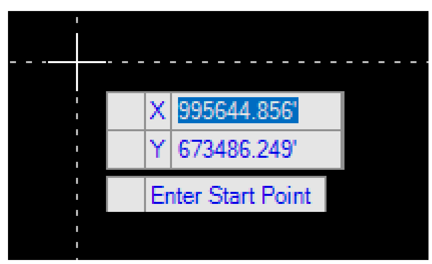
Hold down the second icon to reveal the shortcut menu shown below.



2.2.6.3 XY Tool

When the **XY** mode is selected and a civil command is active, the coordinates of the current cursor position are displayed in a floating dialog as shown below.



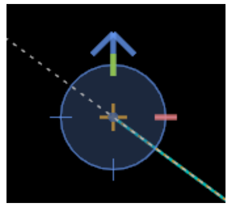


The **Tab** key is used to move through the key-in fields in the dialog.
Values entered in the floating dialog are locked when the Enter key is pushed.
To unlock a value, tab to the field and push the End key.

2.2.6.4 Civil AccuDraw Compass and Curser Input

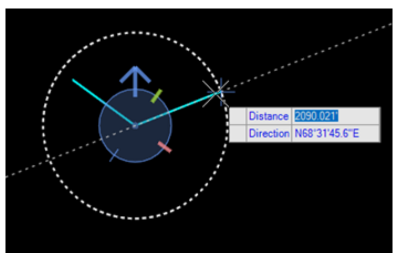
When the **Always Show Compass** setting is toggled on, a compass is displayed at AccuDraw's origin point when Civil AccuDraw is active.

The compass is always circular and is marked by a number of tics. The default is 4 compass points which can be changed in the settings.



The north arrow on the compass will always point to north as defined by selecting **File > Settings > File > Design File Settings > Angle Readout**.

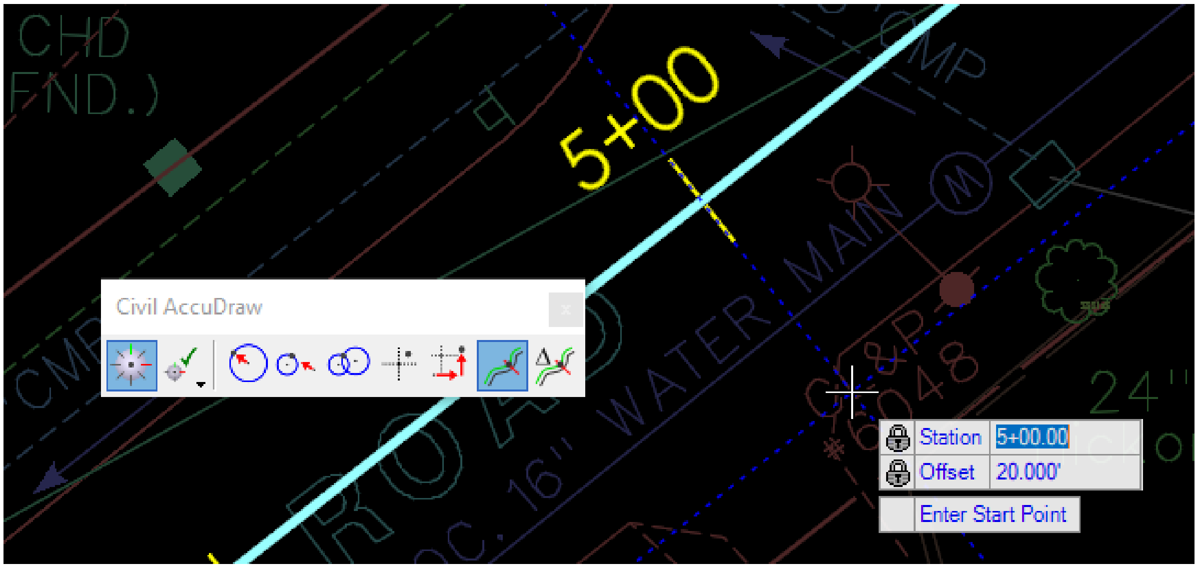
If the *Civil AccuDraw Settings* dialog (shown above) has the **Context Sensitive** option turned on (default), then the compass auto-rotates in a similar fashion to MicroStation AccuDraw:



- The **Distance-Direction** ordinate is used to enter points in polar coordinates as shown above.
- The input for angle and direction fields follows the **File > Settings File > Design File Settings > Angle Readout** settings.
- Note that rather than direction being shown, the label and value are now an Angle. This is because the context is based on the direction of the preceding line segment rather than an absolute direction.
- The compass can be rotated with the **V, B, T, RQ, RE** shortcuts.
- **V** rotates the drawing plane to align with the view axes.
- **T** rotates the drawing plane to align with the axes in a standard Top view.
- **RQ** is used to quickly and temporarily rotate the drawing plane.
- **RE** rotates the drawing plane to match the orientation of a selected element.
- Use the **T** (Top) shortcut or the corresponding drop-down to return to a pure direction.

2.2.6.5 Station Off-set

The **Station-Offset** ordinate is used with a reference element to enter information as a station and offset relative to a selected element.



The reference element can be a MicroStation element or an ORD Geometry element. If a MicroStation element is selected, the beginning of the element is assumed to be station 0+00. If a civil element is selected with stationing defined, the station value of the selected element is used.

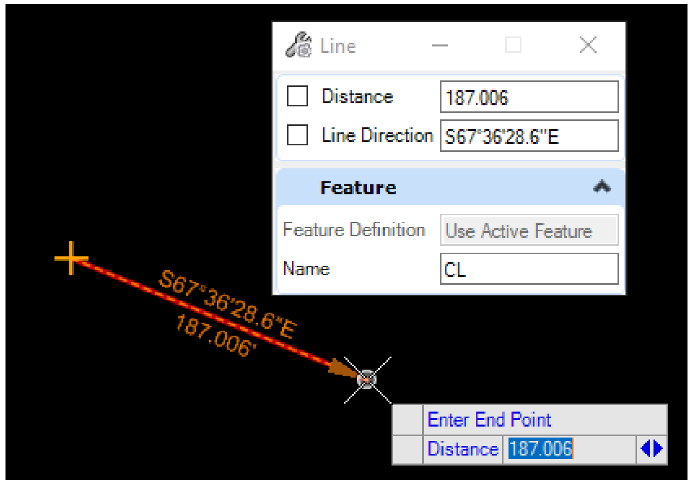
Take the following steps to select the reference element with the **Station-Offset** ordinate active:

- Tab to activate the **Offset** field that is floating on the cursor
- Enter the letter "**O**" shortcut to set the origin. You are prompted to select the reference element.

Once selected, data can be entered using station and offset values relative to the reference element as shown above.

2.2.7 Geometry Rules

Civil Geometry elements drawn using the OpenRoads tools are "ruled" graphics that behave according to the geometric rules that define how the element was placed.

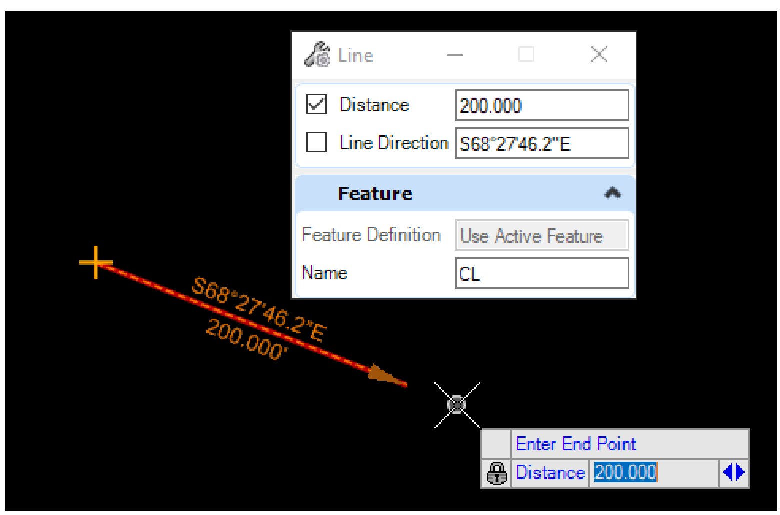


For example, a line drawn using the **Line Between Points** command has the following rules: **Distance** and **Line Direction** as shown below.

These parameters can be entered in the **Tool Settings** dialog, shown at right above, or by using the parameters floating on the cursor.

If you use the input field that floats on the cursor, the left and right arrow keys are used to switch between multiple parameters.

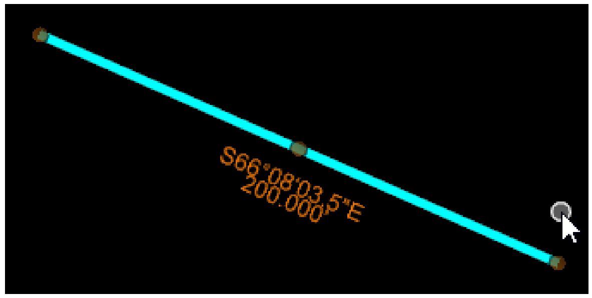
When a parameter value is entered, it is locked as shown below. The parameter can be unlocked by unchecking the item in the **Tool Settings** dialog, or by pressing the End key on the keyboard while the parameter value on the cursor has the keyboard focus.



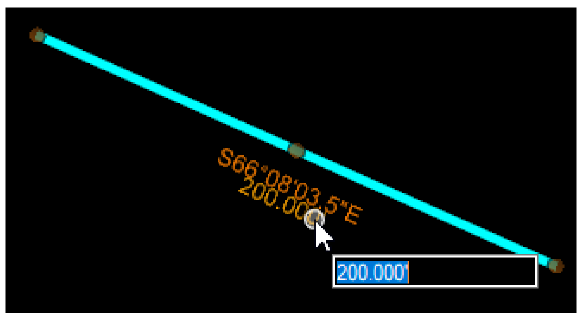
After a parameter value has been keyed in, use the left and right arrow keys, or the **Tool Settings** dialog to enter values for the remaining parameters.

Issue a data point (left-click) to place the element.

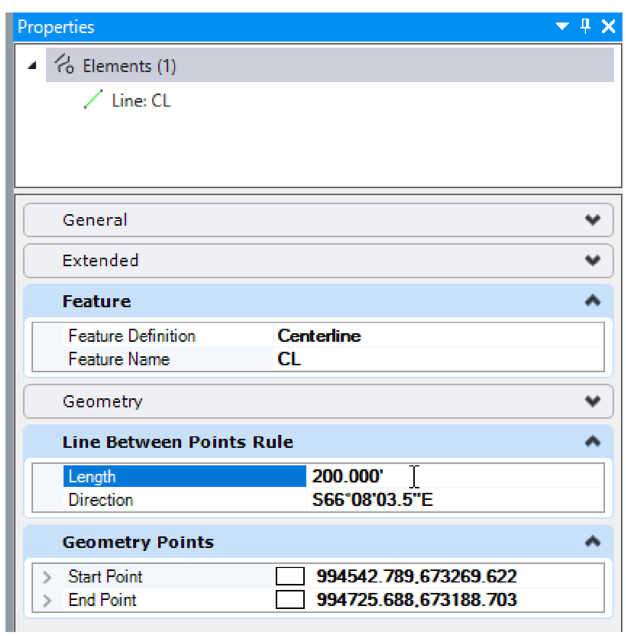
The rules can be reviewed and edited by using the MicroStation **Element Selection** tool to select the element. In the example below, the rules for the selected line are displayed.



The rules can be edited by left-clicking on one of the parameters as shown below.



The rules for the selected element can also be reviewed and edited in the **Properties** dialog as shown below.

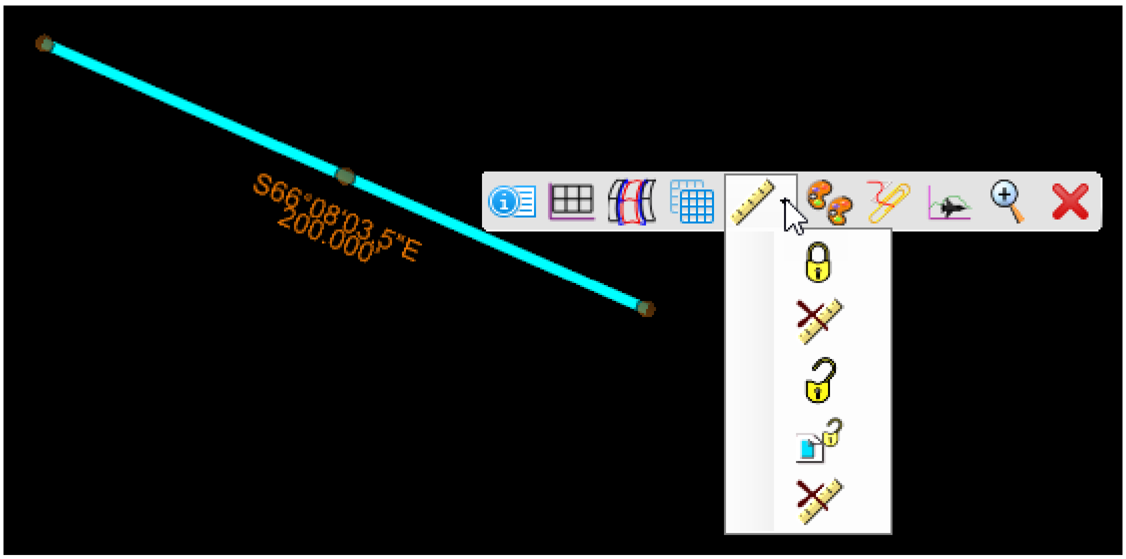


MicroStation's **Manipulate** commands (Copy, Scale, Rotate, etc.) cannot be used on ruled geometry. The rule must be removed to use these commands.

To remove a rule, select the element and let the cursor rest on the selected element until the pop-up menu appears. Choose the **Rules** menu as shown below.

The following options are provided:

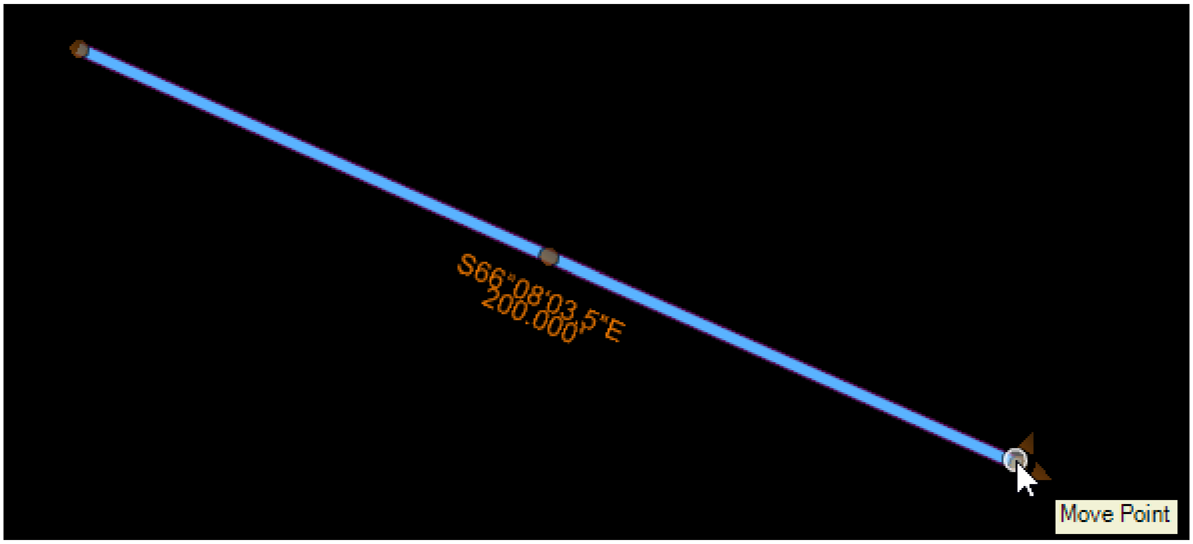
- Lock - Deactivate Rules
- Remove Rule
- Unlock - Activate Referencing Rules
- Lock - Deactivate Reference Rules
- Remove Rule



2.2.8 Element Manipulators

OpenRoads Geometry elements have **Manipulators** that are displayed in MicroStation when an element is selected. These manipulators can be used to edit the selected element graphically or by entering a value.

Clicking on one of the key-point manipulators will display direction arrows to edit the line, as shown on the example below.

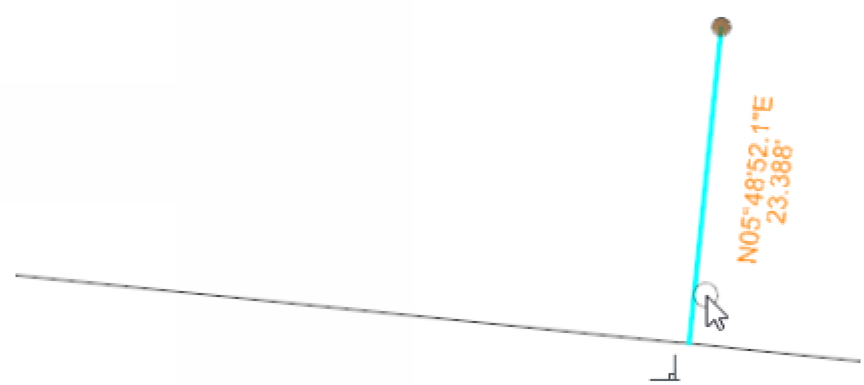


Tip: The size and color of the Manipulator Text can be adjusted in the **User Preferences**. From the MicroStation menu, select ***File > Settings > User > Preferences*** and then choose the **View Options – Civil** item on the left side of the dialog. The **Manipulator Size** or the **Manipulator Font Scale** parameters can be used to adjust the text size.

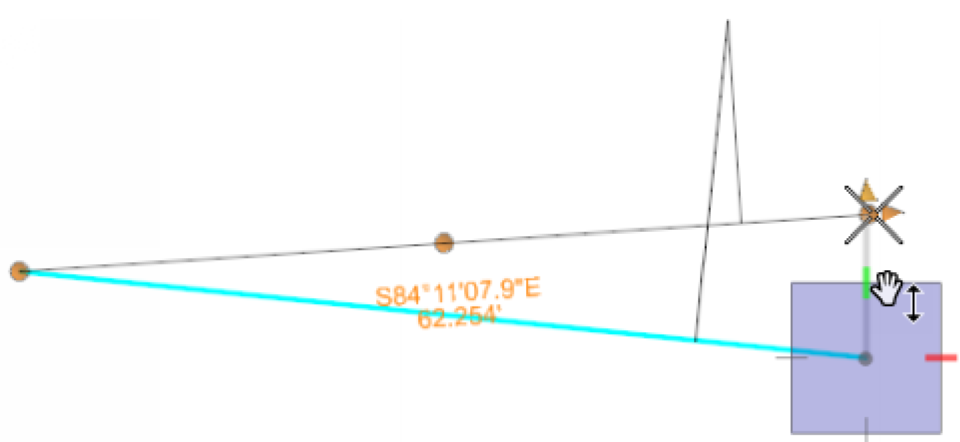
2.2.9 Design Intent

As geometric elements are drawn in MicroStation using the OpenRoads Horizontal Geometry tools, the software retains information about not only **what** element was drawn, but **how** it was drawn. This is referred to as the **Design Intent**.

In the example below, the horizontal line was drawn first. The second line was drawn using the **Perpendicular** snap to draw the line at a perpendicular form the horizontal line to a specific ending point.



If the horizontal line is modified by changing the direction of the line, the design intent for the perpendicular line is preserved. The **Perpendicular rule**, and the ending location are preserved as shown below.



2.2.10 Snaps

As geometry elements are placed in the design, the elements are "ruled" to one another using snaps.

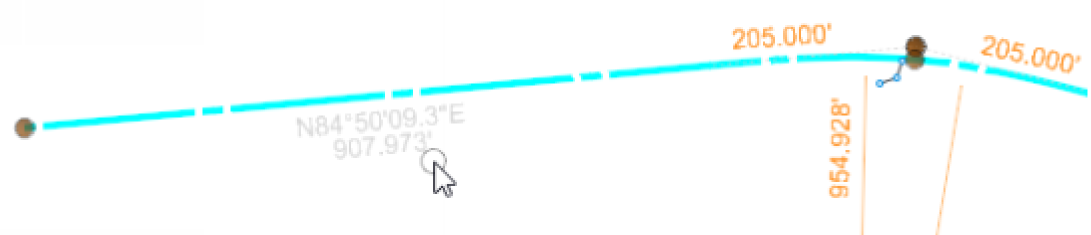
The most common snap is the Key Point snap. In the example below, the second line was placed using a Key Point snap. When the line is selected, the icon for the snap mode that was used is displayed.

If the first line is modified, the second line will maintain its relationship to the first line according to the Key Point snap rule.

Occasionally you may wish to create civil geometry elements that are not "ruled" to other elements. Turn off the **Persist Snaps icon** on the **Feature Definition Toggle Bar** to place civil geometry elements by snapping to other geometry without creating a rule.

2.2.11 Editing Dynamic Text

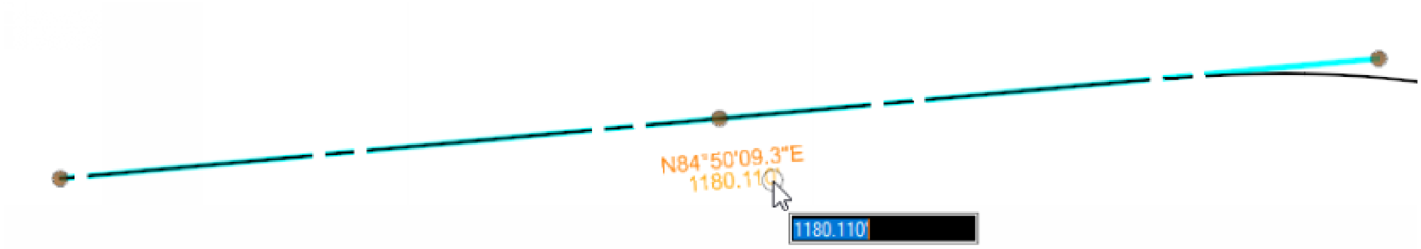
When a civil geometry element is selected, and the various rules are displayed, some rules are shown in grey and cannot be edited by left-clicking on the text. To edit these rules, you must select the underlying geometry directly.



Take the following steps to select the underlying element.

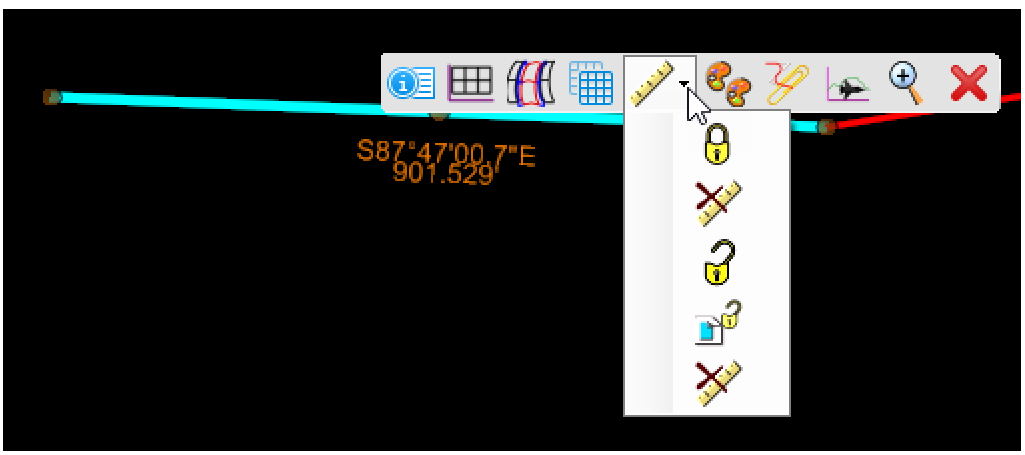
Use the MicroStation Element Selection tool to select the geometry element • Right-click on the element that you wish to edit.

The underlying element is selected, and the associated rules can be edited as shown below.



2.2.12 Deactivating Rules

Geometry elements that have active rules can be selected, edited, and deleted at any time. Deactivating the rules will lock the element in place so that it cannot be edited or deleted unless the rules are reactivated. This is useful to lock an alignment in place once the design has been completed.



To deactivate the rules on a given element, select the element and let the cursor rest on the element until the pop-up menu appears. Choose the **Lock – Deactivate Rule** option as shown below.

2.3 Design Standards

Design Standards are used to monitor various alignment parameters on OpenRoads geometry elements. The standards are stored in DGN Libraries in the following location:

...\CT_Configuration\Organization-Civil_CT_Civil Standards - Imperial\Dgnlib\Design Standards

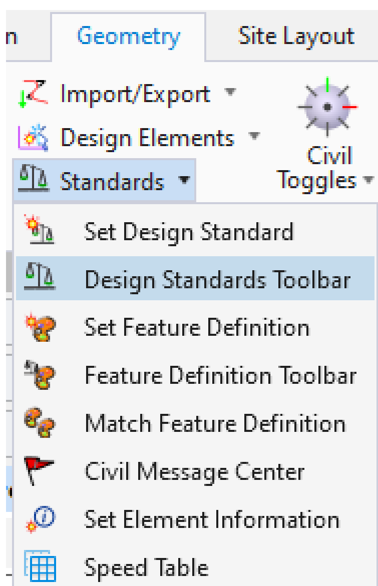
The design standards can be used for two purposes:

- Provide default values for element creation tools such as radius of a curve for a selected CTDOT standard.
- Check and report on the compliance of a geometric element with the design standard that has been assigned to the element.

When a design standard is violated, the user is notified in two ways:

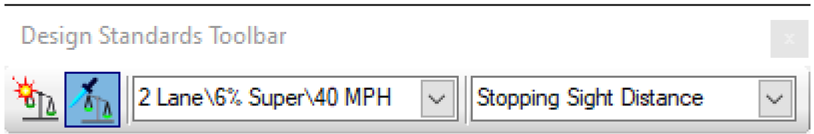
- An icon is displayed on the graphics to indicate that there is a problem.
- The Civil Message Center reports any errors.

The **Design Standards Toolbar** allows you to select and apply a design standard. From the OpenRoads Modeling **Geometry** tab, select ***Standards > Design Standards Toolbar*** to access the dialog shown below. Design Standards can also be applied to an element after it has been created by using the Set Design Standard icon on the Design Standards Toolbar.



The toolbar contains a drop-down list to select the design standards and the design speed. The vertical design standard, displayed in the drop-down menu to the right, is automatically set when the horizontal design standards is selected.

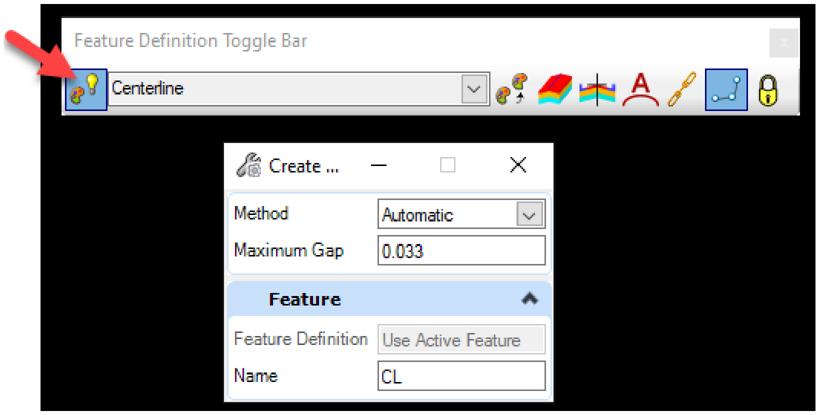
There are two icons located to the left of the drop-down menus. The first, **Set Design Standard**, is used to apply the current design standard to selected geometric elements. The second, **Toggle Active Design Standard**, is used to set the selected design standards active for relevant commands.



2.4 Complex Geometry

Individual civil geometry elements are connected together as a single entity using the **Complex By Element** command.

Select ***Geometry > Complex Geometry > Complex By Element*** to access the command.

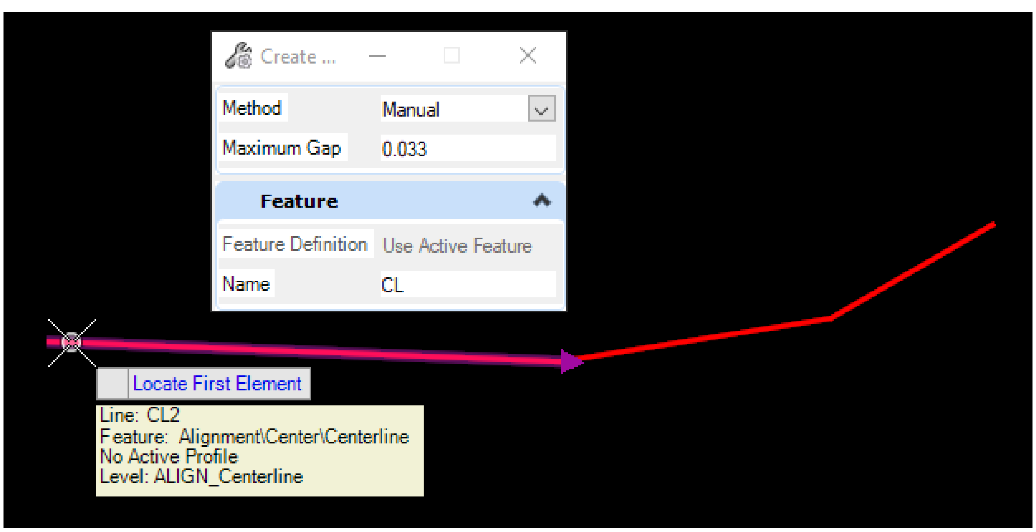


In the example above, the **Feature Definition** is set to **Use Active Feature** as defined in the ***Feature Definition Toggle Bar***.

When the **Method** is set to **Automatic**, the software will automatically select elements to make up the complex geometry by searching for elements with endpoints that connect within the **Maximum Gap** value. If a fork is found, where two elements are found within the **Maximum Gap** tolerance, the user is prompted to identify which path to accept to compete the operation.

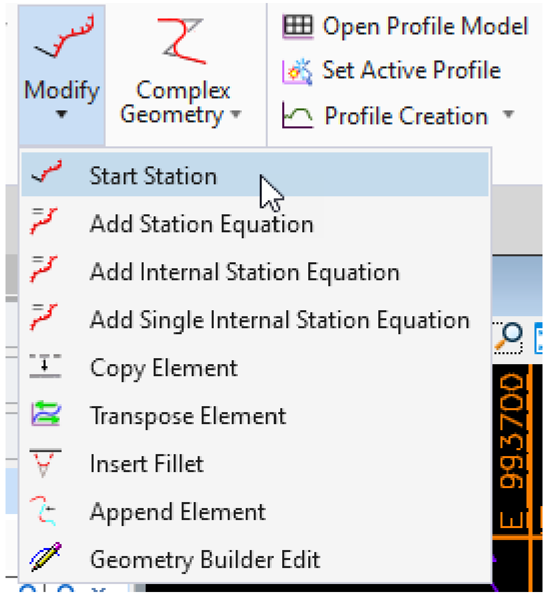
Be sure to enter a **Name** for the geometry.

When locating the starting element an arrow is displayed defining the direction to store the element. Complex elements can be stored in the opposite direction than they were originally defined by selecting the other end of the graphics.



2.5 Start Station

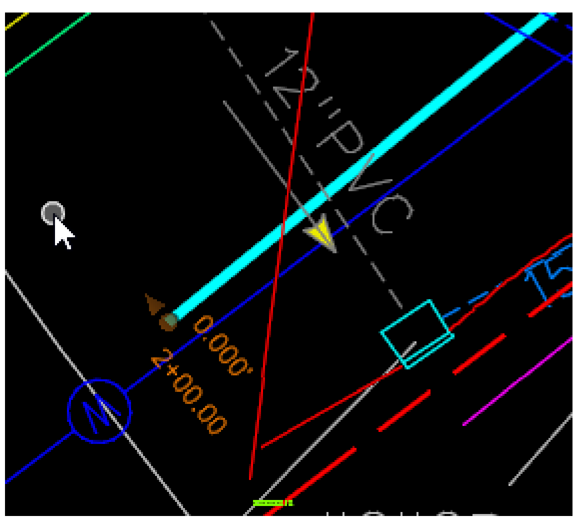
When an OpenRoads alignment is stored, the software assumes a beginning station value of 0+00. The station can be redefined by use of the **Start Station** command.



The command has two parameters:

- **Start Distance** The distance along the alignment to assign the Start Station. The beginning station is computed based on the Start Distance and the Start Station value.
- **Start Station** The station value to be assigned at the Start Distance location.

After the beginning station has been defined, selecting the alignment will display the station rules as shown below.

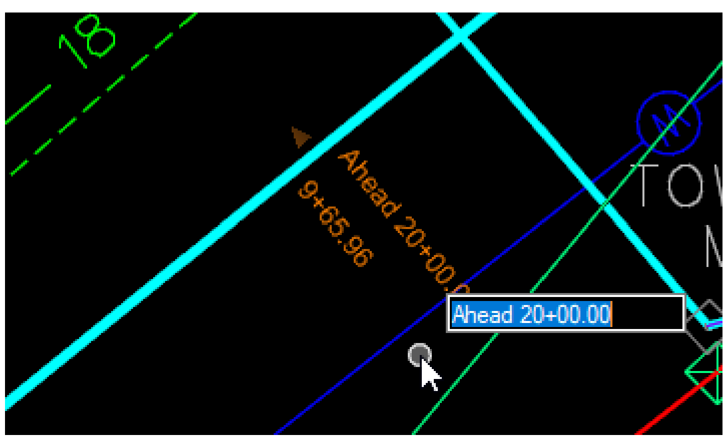


2.6 Station Equations

The station parameters can be edited by selecting the rule in the design file, or from the Properties dialog. **Add Station Equation -** The Add Station Equation command, located below the Start Station command, adds parameters for the Ahead Station and Back Station. When the command is selected, the user is prompted to select the alignment to assign the station equation.

After the alignment has been selected, the location of the equation can be identified graphically, or by keying in the **Back Station** value.

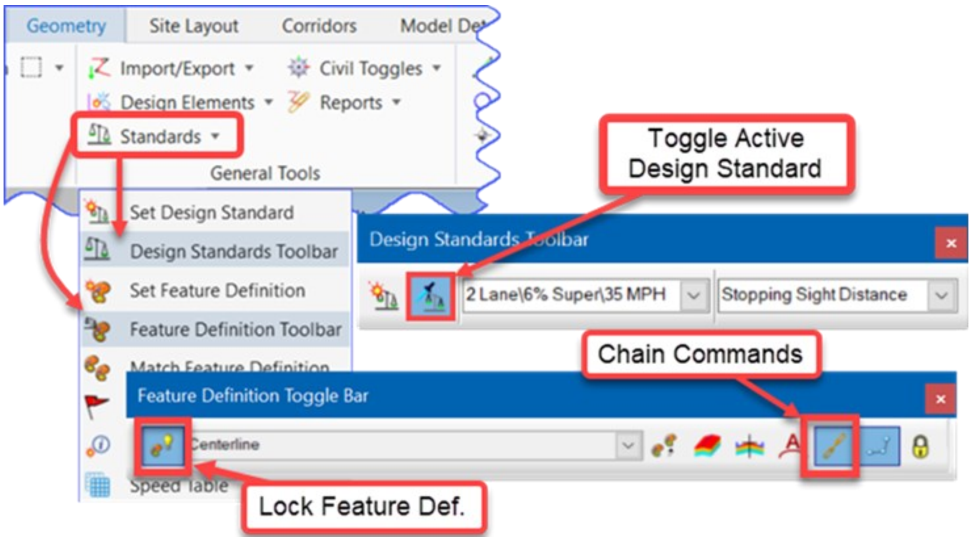
The ahead station is stored as a "region" much like the legacy GEOPAK format. When the **Ahead Station** value is entered, it is not necessary to define the region number.



The station equation value can be reviewed and edited by selecting the element as shown below. The parameters can also be edited in the **Properties** dialog.

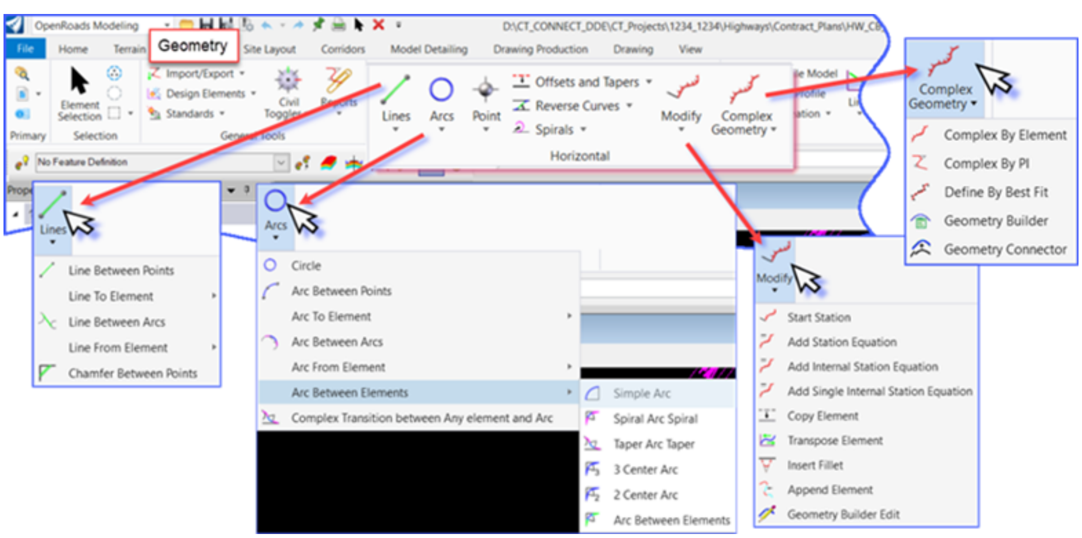
2.7 Interface Setup

1. If not already docked, from the **Geometry** ribbon select the **General Tools** tool, select and dock the **Design Standards Toolbar** and the **Feature Definition Toolbar**. These tools can be used to better adhere to design standards.



2. From the **Design Standards Toolbar**: select the AASHTO Design Standards for the project design as needed, example: ***6% Super*** (2 Lane AASHTO Design Standards 2018 – Imperial Design Library) – 35 MPH (2 Lane AASHTO Design Standards 2018 – Imperial Design Library) and ***Stopping Sight Distance*** (AASHTO Design Standards 2018 – Imperial Design Library). Turn on the ***Toggle Active Design Standard***.

3. From the **Feature Definition Toolbar**: select the active feature definition for ***Alignment - Center – Centerline*** and **Lock** the feature definition. Activate the **Chain Command**, this will chain the tangents together, but still remain individual elements.

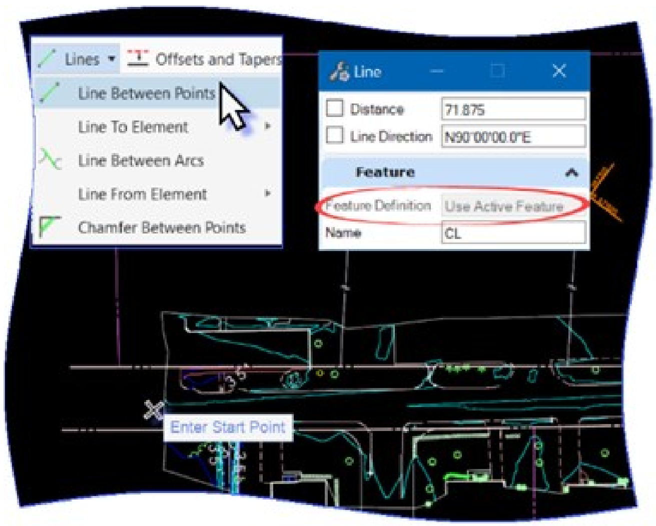


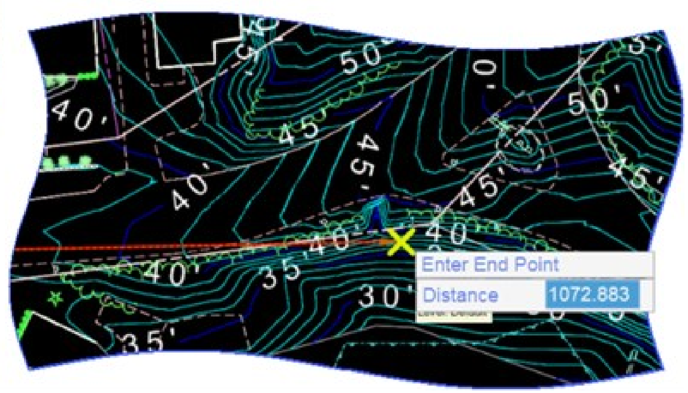
Exercise 3 - Horizontal Alignements

3.1 Creating Alignments

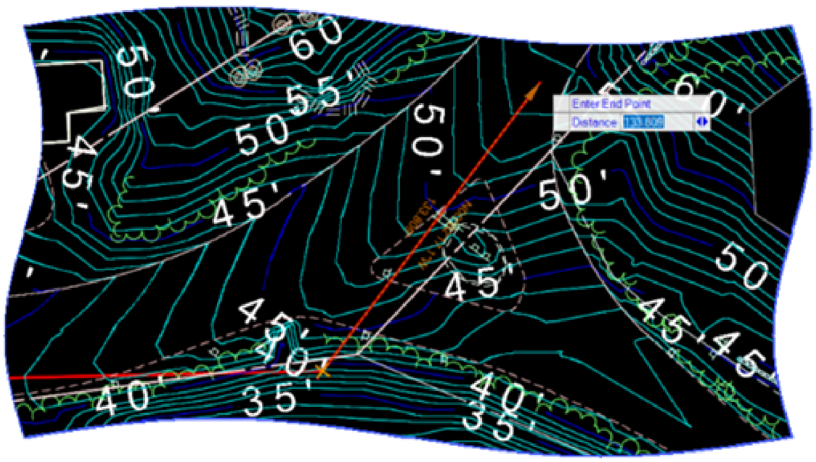
In this module the creation of a horizontal alignment with the PI method will be described. There are several methods to create horizontal alignments for more on other methods or other alignment tools please see Bentley's OpenRoads Help – Horizontal Geometry.

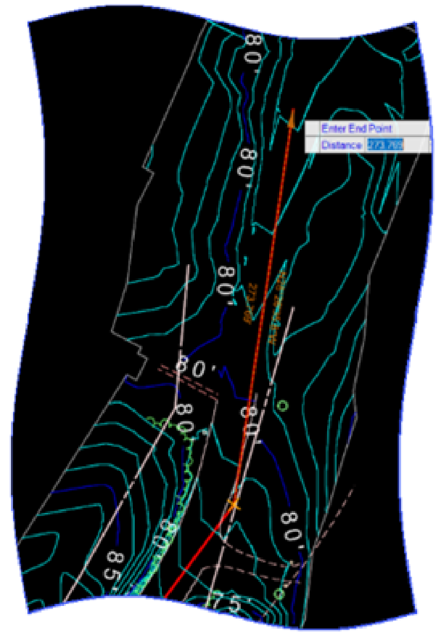
1. From the **Horizontal tool group** select the **Lines – Line Between Points** tool. In the **Line** toolbox, **Feature Definition** should be set to: ***Use Active Feature***, this should be greyed out, and it will set the feature tangent line to the active feature definition picked in the previous step. In 'View 1, Default' select the first PI location. Follow the command prompts attached at the cursor or at the bottom left – ***Enter Start Point***. Example: At the left beginning, at about the middle of the existing roadway – left click. ***Enter End Point***. Example: somewhere close to the intersection – left click.





2. Because the Chain Command was activated, it prompts ***Enter End Point***. Select a point for the next PI, again it prompts ***Enter End Point***. Select the end of the horizontal alignment. ***Right-click*** to end selecting a new end point, select the **Element Section** tool to end and get out of the place line command.

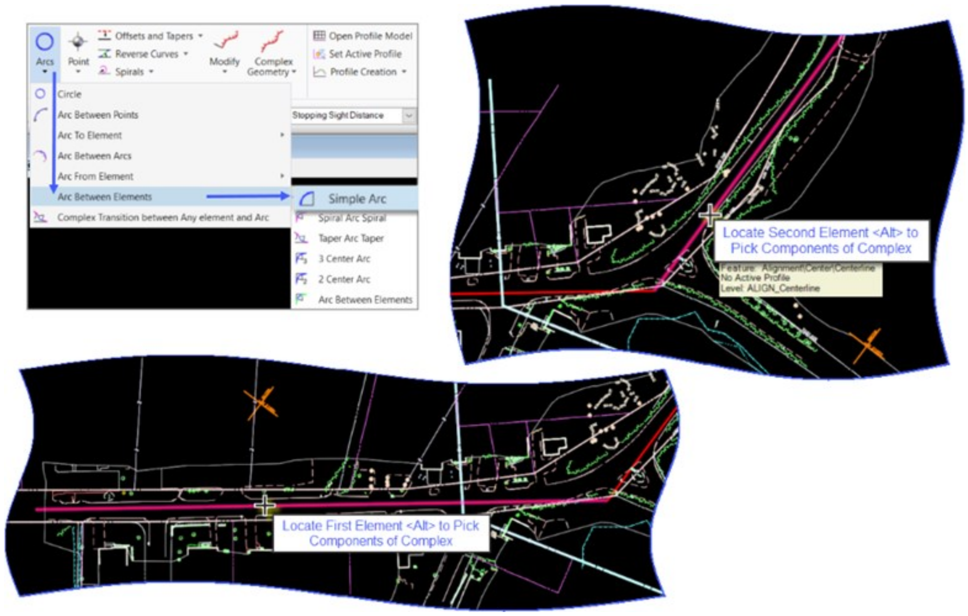




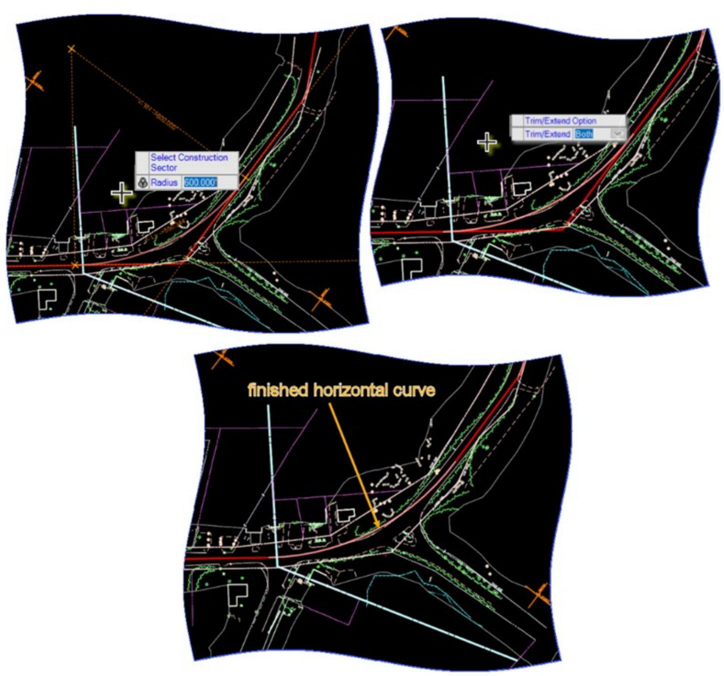
3. Four PI locations have been established with tangents shown. ***Turn Off*** the contours for the terrain.

4. To place a curve (arc) between two tangents select the **Arcs – Arc Between Elements – Simple Arc** tool. The **Simple Arc** toolbox opens. The Feature Definition should be set to ***Use***

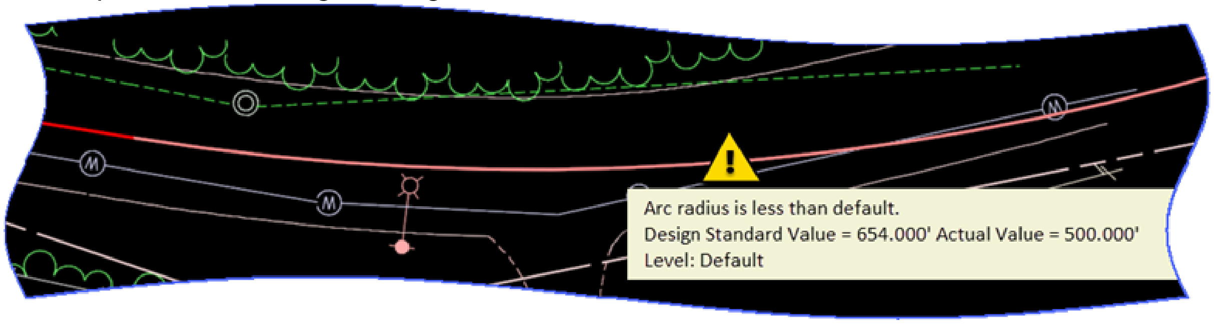
Active Feature. With the Design Standards active the radius is set to the value for the design speed, example: Radius – 654.000'. Now follow the command prompts. ***Locate First Element*** – select the first tangent; ***Locate Second Element*** – select second tangent; ***Select Construction Sector*** with ***locked Radius 654.00'*** – datapoint to select the construction sector; ***Trim/Extend Option*** – choose ***Both*** and datapoint (left-click) to accept.



5. **Repeat** this for the next curve (arc).



6. **Review** the Horizontal Curves. The radii can be adjusted, if the radius doesn't comply with the design standard an advisory marker will be placed on the radius.

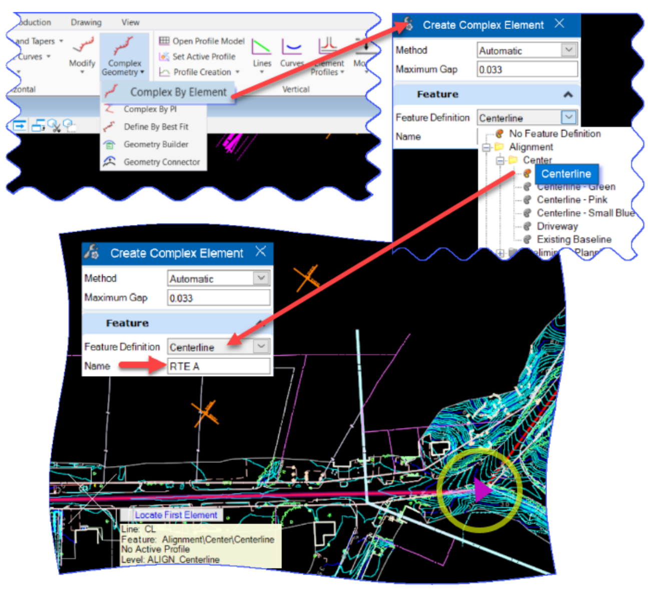


NOTE: *You still can keep this radius value, just be aware this is not to design standards picked for the alignment. The design standards tools do not have to be used when designing the alignments, the tools are design aides.*

7. **Complex** the horizontal alignment, turn the individual elements into one complex line-string. Select the ***Complex Geometry – Complex By Element*** tool.

   The **Create Complex Element** toolbox opens, set the following:

   - **Method** select either: ***Automatic*** or ***Manual***, example: Automatic.
   - **Maximum Gap** set to ***0.033*** (max. distance between elements that will be complexed).
   - **Feature Definition** is set to Use Active features if feature definition is locked or set to as example: ***Centerline***.
   - **Name** type a name for the horizontal alignment, example: ***RTE A***



   Follow the prompts: ***Select First Element*** – select the beginning of the first tangent, an arrow should appear showing the direction of the alignment. ***Left Click*** to accept direction.

All of the horizontal alignment should be highlighted/selected. ***Left Click*** again to ***Accept Complex***.

3.2 Adjustments

3.2.1 Adjustments to the Horizontal Alignment

Depending on the **Design Intent**, the placement and relationship between the elements will control how individual elements of the horizontal alignment can be modified.

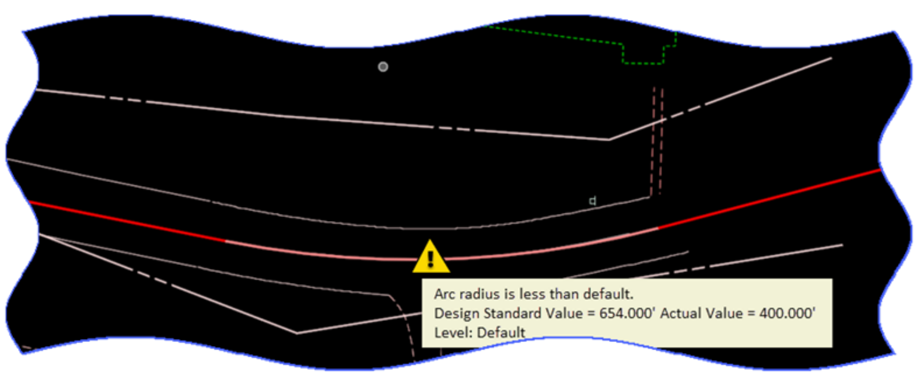
For more information on placements and relationship of elements see Bentley's OpenRoads Help.

In the example the PI method was used, this will allow for modifications to the PI locations and the radii of curves.

3.2.2 Adjustment to PI Location and Radii of Curves

1. Using the **Element Selection tool** click on the horizontal alignment; the horizontal alignment is highlighted and all features of it will be visible.
2. ***Select the Second PI location*** and move it to a new location; see the horizontal alignment move. Move the PI to the new location.
3. Move the third PI location. ***Select the third PI*** and move the PI to the new location, move the alignment to better fit with the existing roadway.
4. Now adjust the radius for a better fit. Increase the first radius to 800 feet, see what happens. ***Click on the radius***, this will activate the radius components that can be adjusted. Click on the **radius** (orange text) a box with the radius will appear, here change the radius to 800.00'. Press ***Enter***. Review the arc, click again on the radius and type in ***700.00 feet*** as radius. ***Click on the second radius***, enter a new radius of 400 feet. Press ***Enter***. We will leave this radius to better match in with the existing curve. You can see there is now an **exclamation icon**, meaning the design standards are <u>not</u> met.

   These are just a few of the options to control and edit horizontal alignments, see Bentley's OpenRoads Help for more detailed edit options.

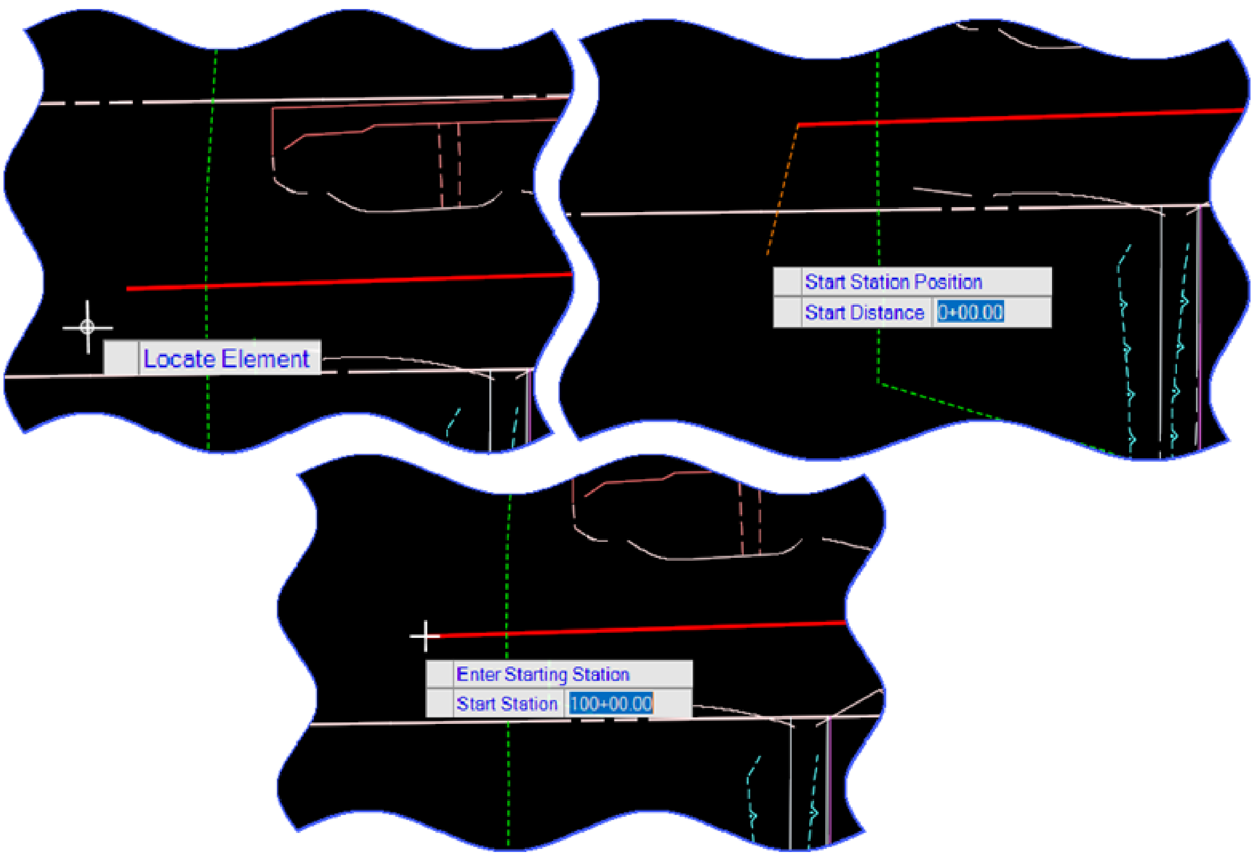


3.3 Define Start Station for Horizontal Alignment

The **Start Station** assigns stationing to an element. A station value and a position along the element for that station value are assigned. If no station value is assigned, then the beginning station is assumed to be zero (0+00.00).

1. In the **Horizontal toolbar** click on the **Modify** tools – select the **Start Station** tool.
2. Follow the prompts: ***Locate Element,*** select the beginning of the alignment. ***Start Station Position***, enter 0.00 as Start Distance. ***Enter Starting Station***, enter Start Station 100+00.00 and click within the view.
3. Click on the alignment, it will now display the start station and the offset.

NOTE: *Any changes to the alignment will also update the Stationing.*

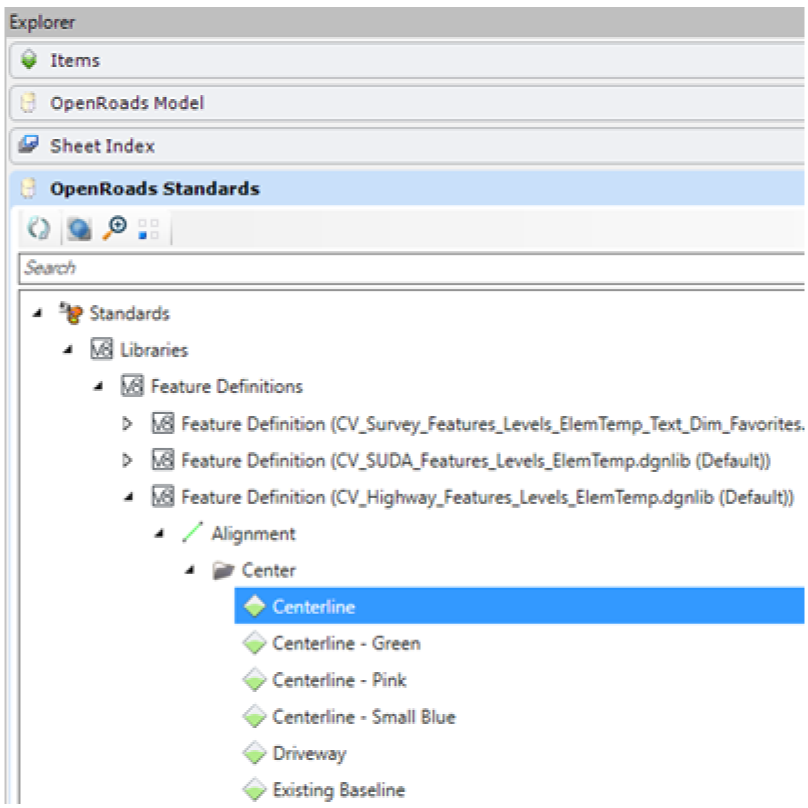


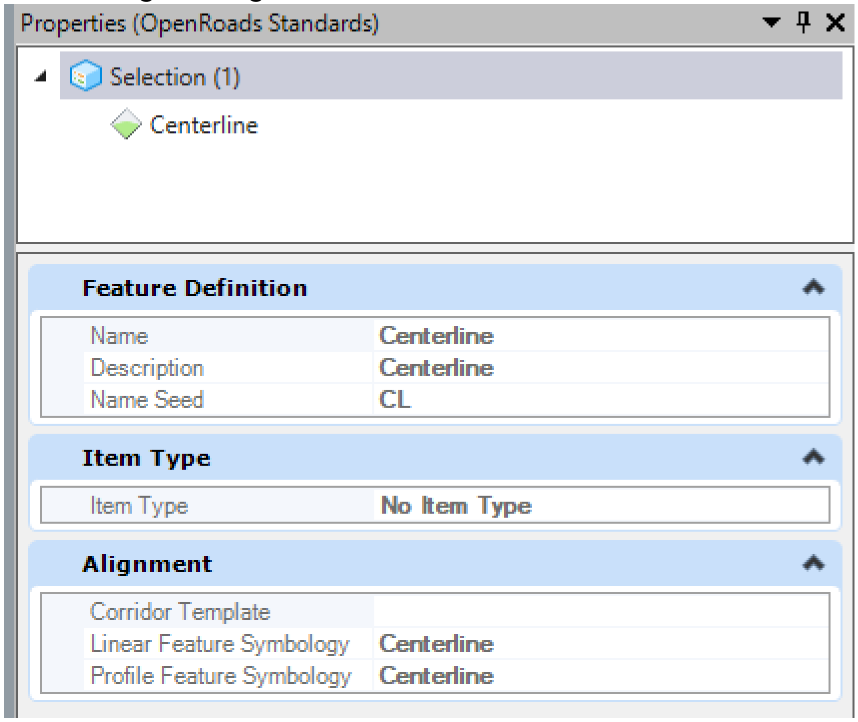
3.4 Horizontal Alignment Annotation

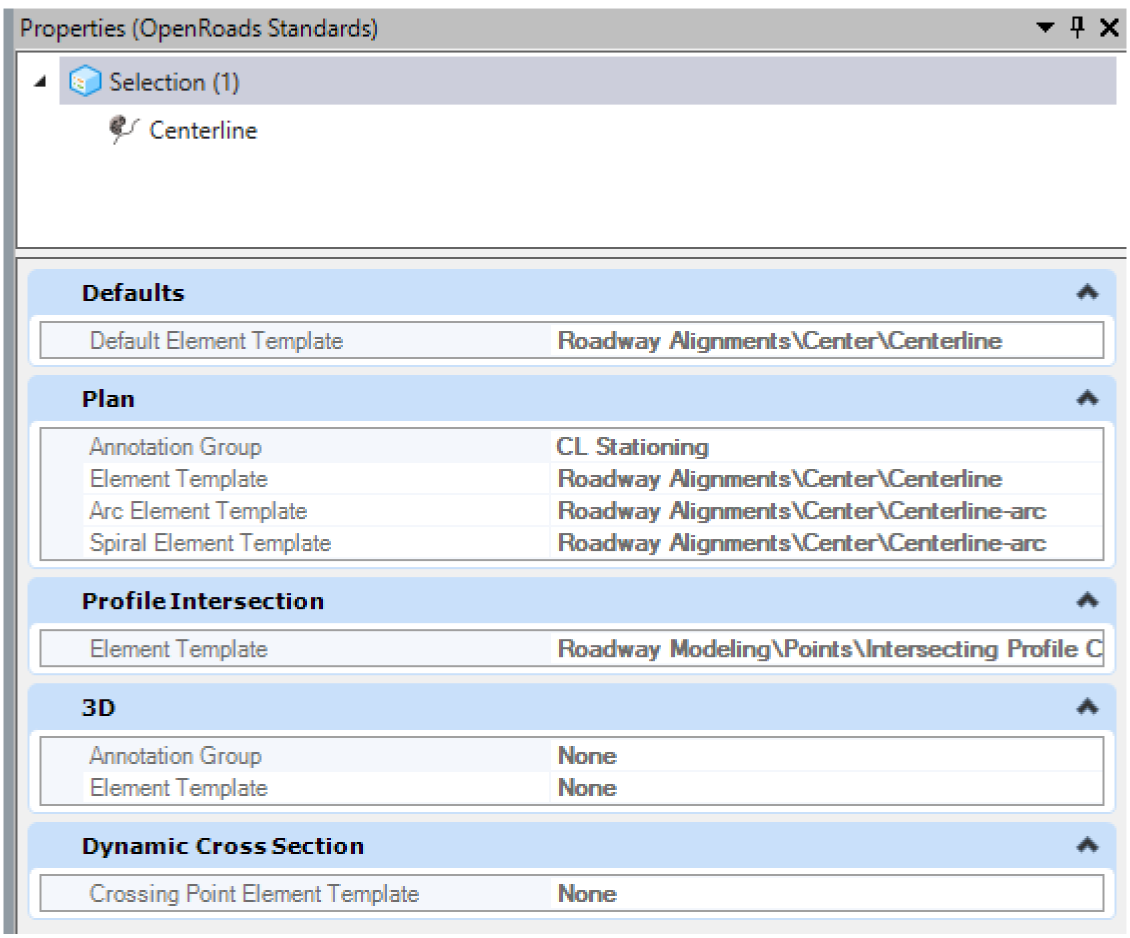
3.4.1 Annotation Groups

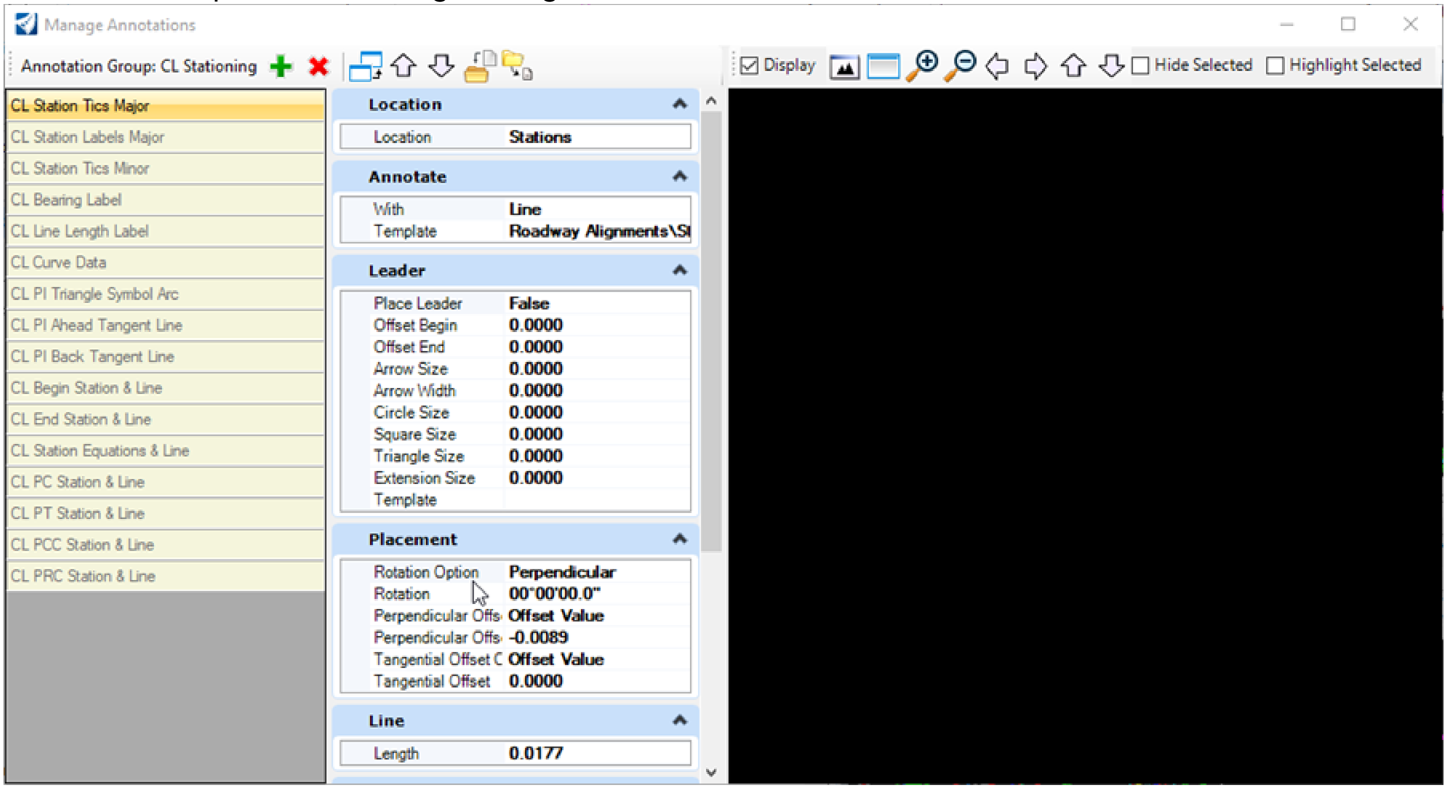
Alignment elements are assigned a **Feature Definition**, which controls the symbology of the element (level, color, line style, and line weight) as well as the annotation properties of the element. The CTDOT Standards include feature definitions for alignments, which can be reviewed from the Project Explorer dialog, as shown below.

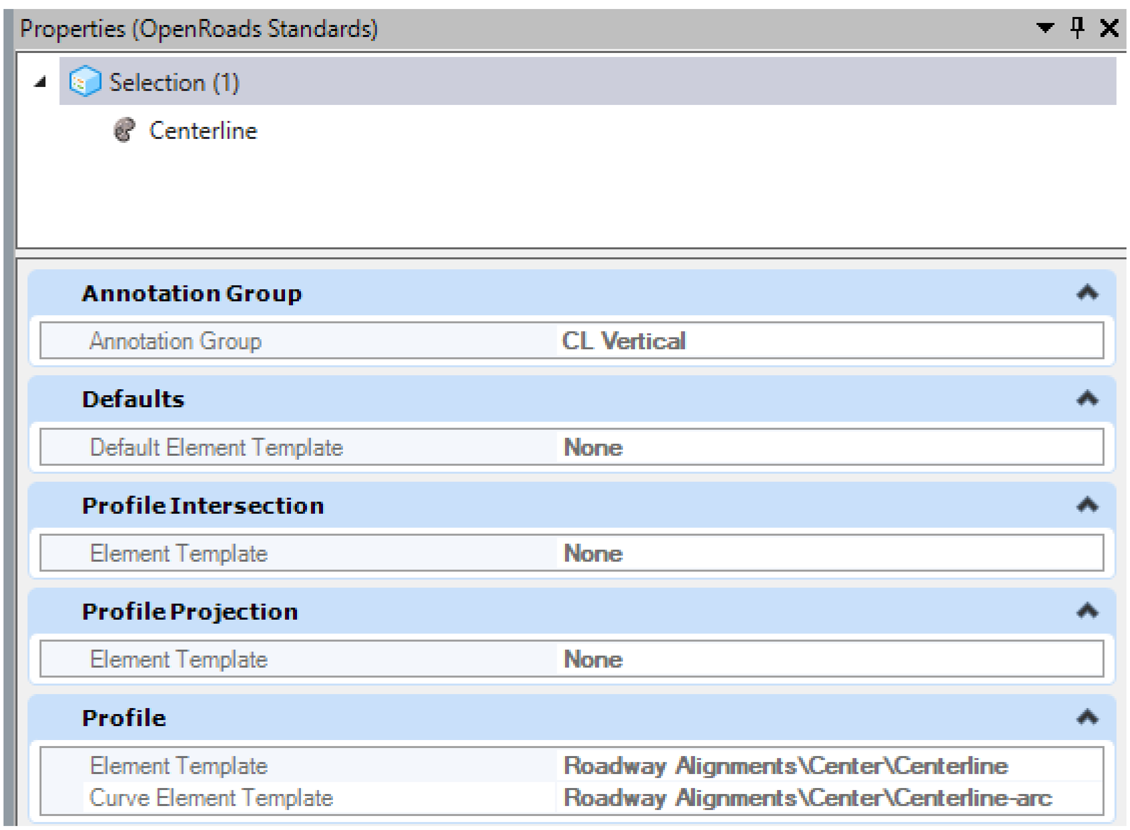
There are several levels of configuration that has been set to get the annotation to display to CTDOT standards.

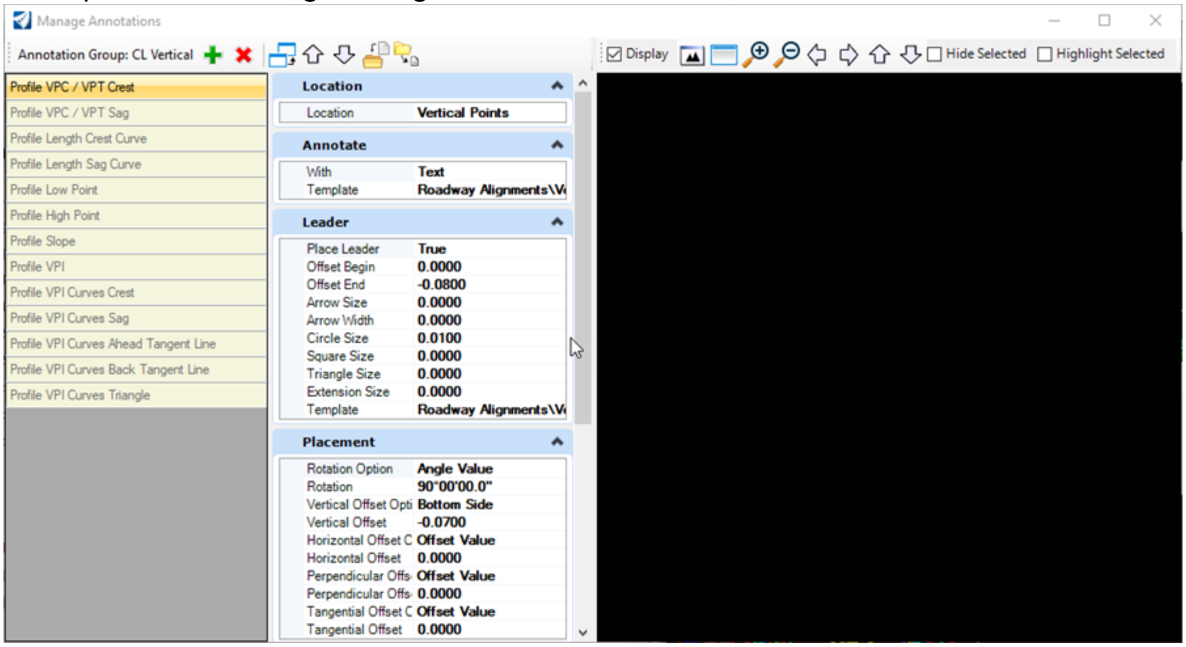



Properties (OpenRoads Standards)
Selection (1)
Centerline
Feature Definition
Name
Centerline
Description
Centerline
Name Seed
CL
Item Type
Item Type
No Item Type
Alignment
Corridor Template
Linear Feature Symbology
Centerline
Profile Feature Symbology
Centerline


Properties (OpenRoads Standards)
Selection (1)
Centerline
Defaults
Default Element Template
Roadway Alignments\Center\Centerline
Plan
Annotation Group
CL Stationing
Element Template
Roadway Alignments\Center\Centerline
Arc Element Template
Roadway Alignments\Center\Centerline-arc
Spiral Element Template
Roadway Alignments\Center\Centerline-arc
Profile Intersection
Element Template
Roadway Modeling\Points\Intersecting Profile C
3D
Annotation Group
None
Element Template
None
Dynamic Cross Section
Crossing Point Element Template
None


Manage Annotations
Annotation Group: CL Stationing
Display
Hide Selected
Highlight Selected
CL Station Tics Major
CL Station Labels Major
CL Station Tics Minor
CL Bearing Label
CL Line Length Label
CL Curve Data
CL PI Triangle Symbol Arc
CL PI Ahead Tangent Line
CL PI Back Tangent Line
CL Begin Station & Line
CL End Station & Line
CL Station Equations & Line
CL PC Station & Line
CL PT Station & Line
CL PCC Station & Line
CL PRC Station & Line
Location
Location Stations
Annotate
With Line
Template Roadway Alignments\St
Leader
Place Leader False
Offset Begin 0.0000
Offset End 0.0000
Arrow Size 0.0000
Arrow Width 0.0000
Circle Size 0.0000
Square Size 0.0000
Triangle Size 0.0000
Extension Size 0.0000
Template
Placement
Rotation Option Perpendicular
Rotation 00°00'00.0"
Perpendicular Offs Offset Value
Perpendicular Offs -0.0089
Tangential Offset C Offset Value
Tangential Offset 0.0000
Line
Length 0.0177


Properties (OpenRoads Standards)
Selection (1)
Centerline
Annotation Group
Annotation Group CL Vertical
Defaults
Default Element Template None
Profile Intersection
Element Template None
Profile Projection
Element Template None
Profile
Element Template Roadway Alignments\Center\Centerline
Curve Element Template Roadway Alignments\Center\Centerline-arc

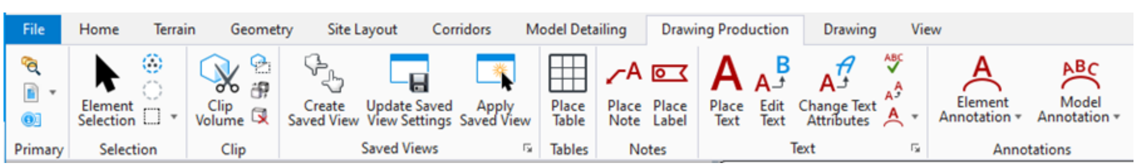


Note: *the alignment feature definition controls the symbology and annotation settings for both the horizontal geometry and the proposed vertical alignment.*

3.4.2 Annotation Tools

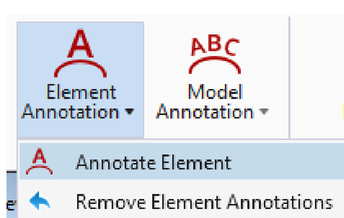
When geometric alignments are placed in the file, the annotation is not automatically generated by the software. The user must identify the alignments that are to be annotated either individually, by selecting the specific alignments, or by annotating all of the alignments contained in the active model, or all models in the active file.

Annotation commands are selected from the **OpenRoads Modeling** WorkFlow in the **Drawing Production** tab, as shown below.



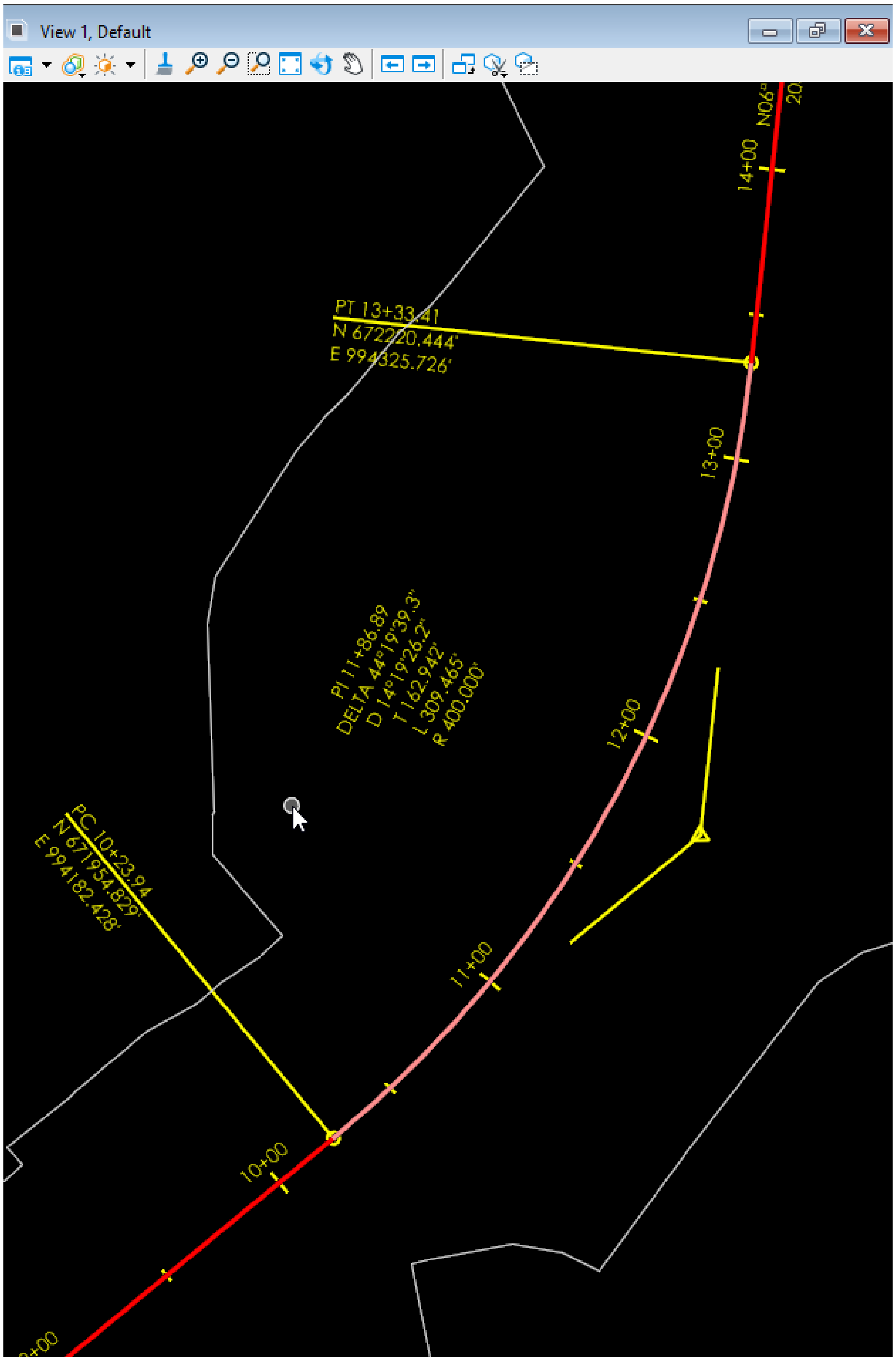
Element Annotation

Individual elements are annotated by selecting the Element Annotation command from the Drawing Production tab. Two commands are available as shown below.



3.4.3 Annotation Element

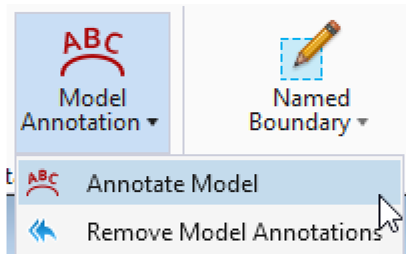
The **Annotate Element** command is used to annotate one of more selected elements. The command prompts to select the OpenRoads element to be annotated. Multiple elements can be selected. Reset (right-click) to complete the selection process and initiate the annotation placement. An example of annotation at the beginning of an alignment is shown below.



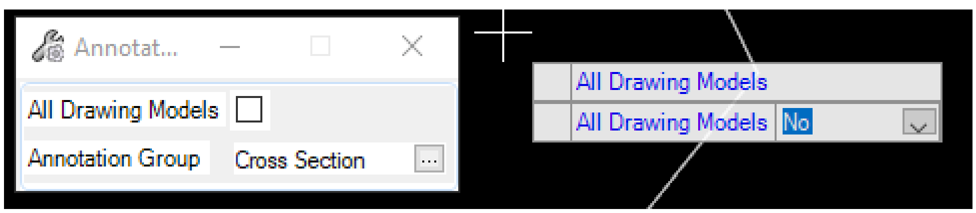
The **Remove Element Annotations** command is used to remove the annotation from selected elements. When prompted, select the elements to remove the annotation. Reset (right-click) to end the selection process and initiate the removal process.

3.4.4 Model Annotation

The Model Annotation command us used to annotate all of the elements in the active model, or all models, by selecting a specific Annotation Group. Two options are available as shown below.



ANNOTATE MODEL – Select this command to annotate all the elements in a model, or in all models, by selecting a specific Annotation Group. When the command is selected, the user is prompted as shown below. Issue a data point (left mouse-click) to initiate the process.



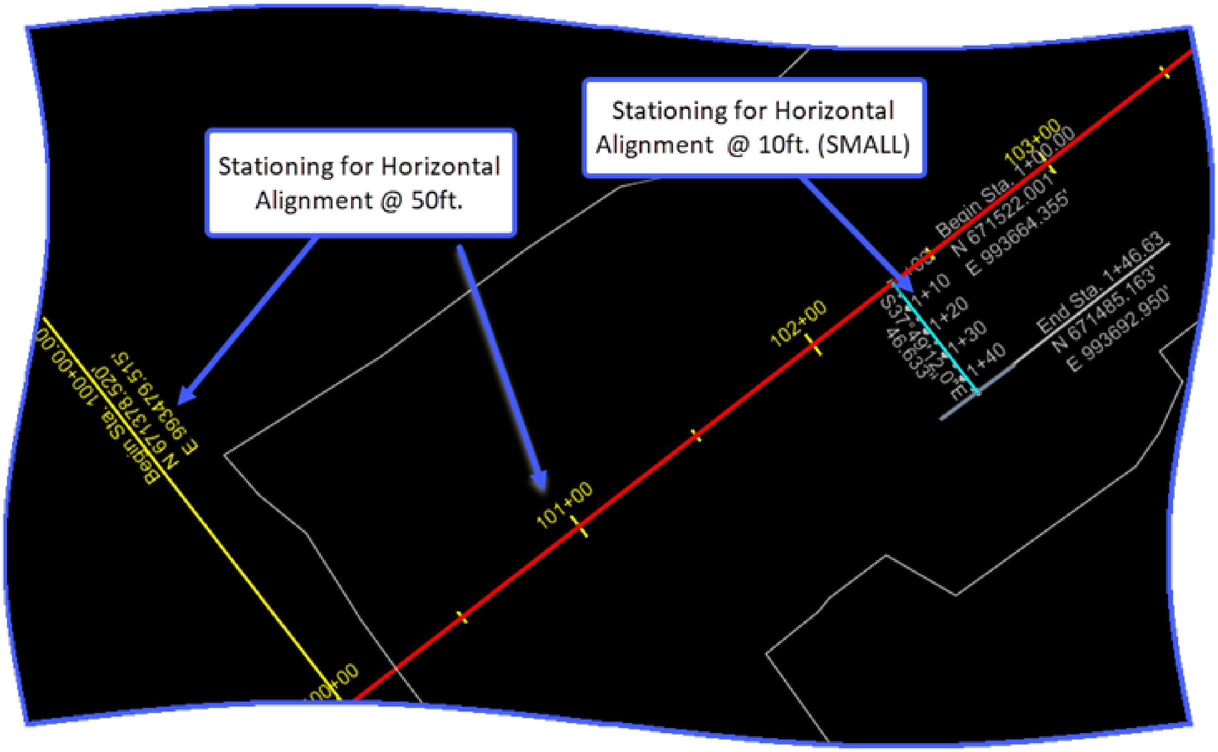
REMOVE MODEL ANNOTATIONS – Select this command to remove all annotations from the active model or all models in the active file.

A Horizontal Alignment needs the following annotation:

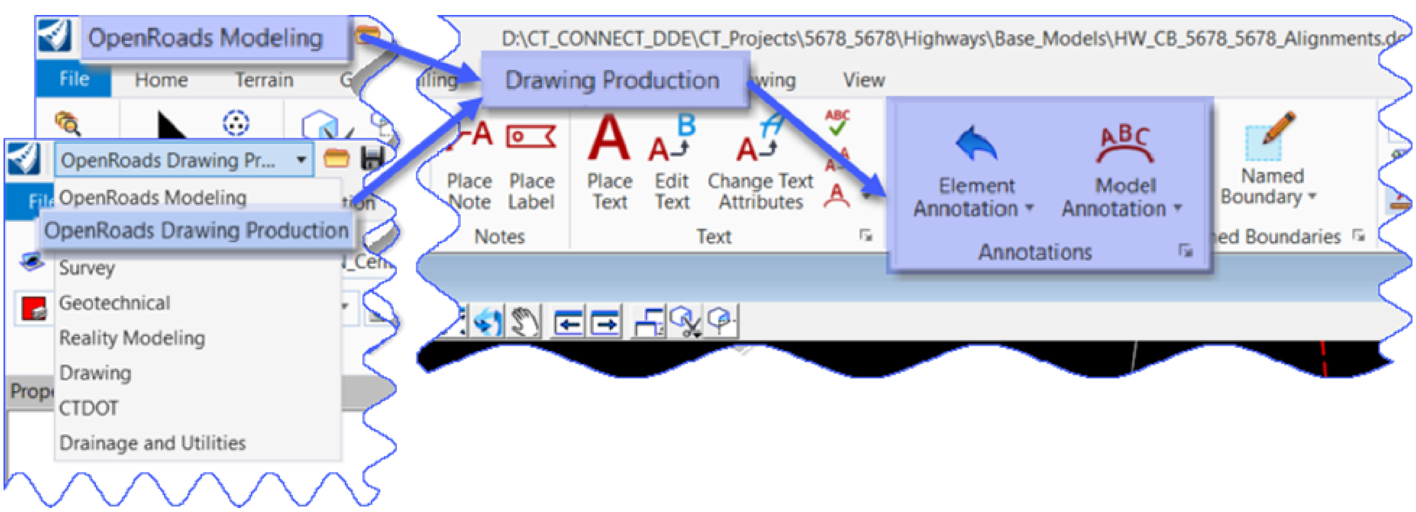
- Stationing,
- PC's, PI's, PT's,
- Curve Data and
- Bearings

This has been automated; the annotation will be in the design file (alignment dgn-file) and can be done after the alignment is created. Some text adjustments to location of PC, PI, PT, curve data and bearings can be made, but this should be done sparingly if at all. If adjustments to the alignment are made, the text will also adjust to original text location/placement.

Example: Horizontal Alignment for RTE A (normal annotation) and Private Road (small annotation), see below.

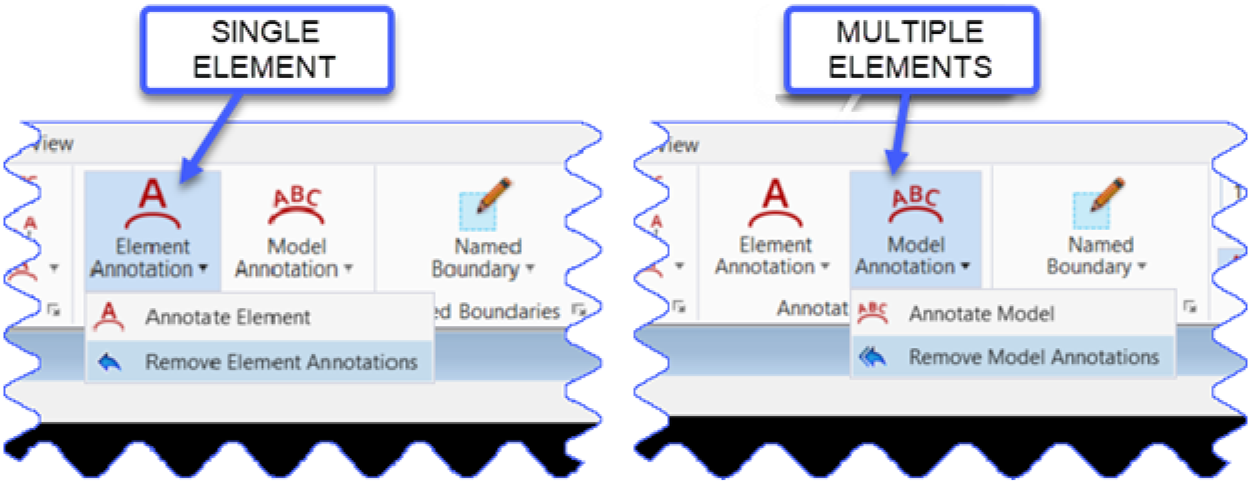


1. ***Open*** the alignment file (this file should reside in the Base_Model folder), if not already open. Select the **OpenRoads Modeling** workflow or the **OpenRoads Drawing Production** workflow.

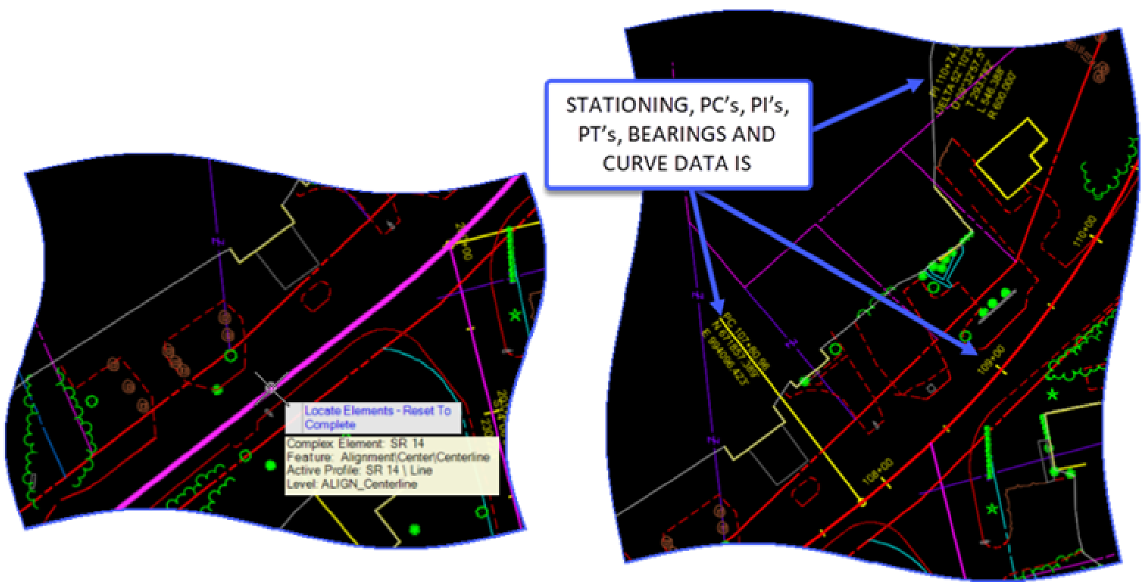


2. Click on **Drawing Production** tab. These annotations tools are available: ***Annotate Element*** (for single element), ***Remove Element Annotation***, ***Model Annotation*** (for multiple

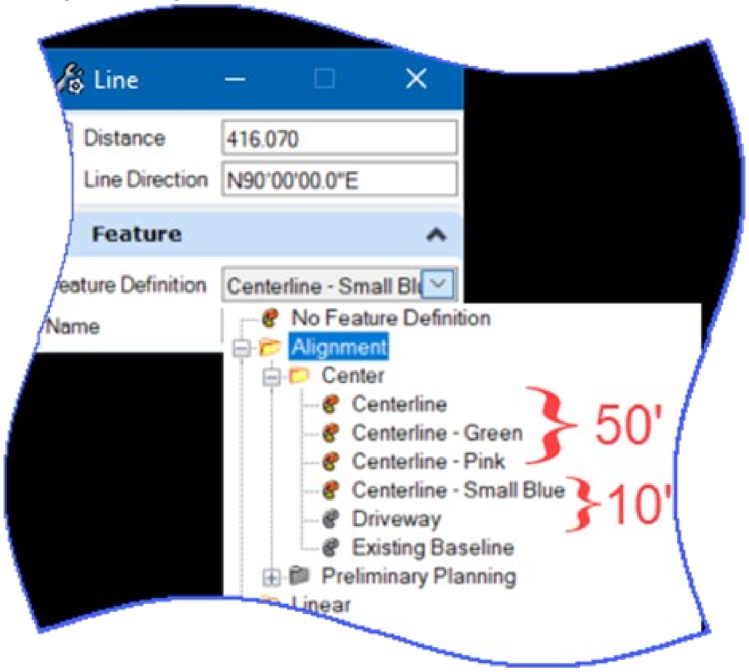
elements) and ***Remove Model Annotation***. As an example, we will use ***Annotate Element*** tool.



3. Click the **Annotate Element** tool, follow the prompts. ***Locate Elements – Reset To Complete***. Click on the alignment (example: SR 14), after it is highlighted, ***right-click to Reset***.



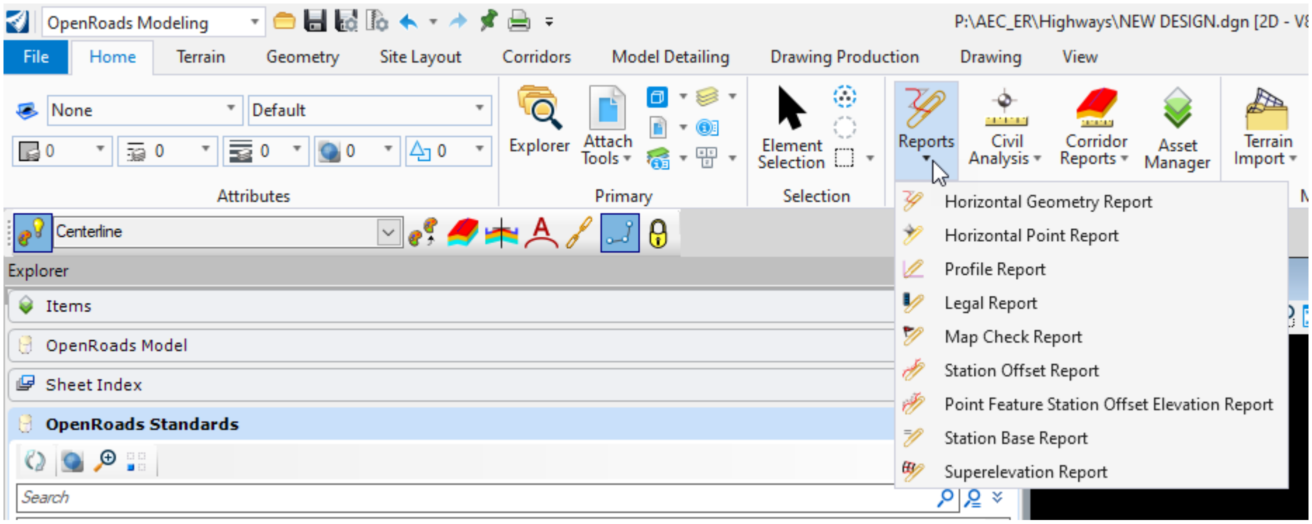
4. The alignment is now annotated/labeled.
5. For alignments using **Centerline**, **Centerline – Green** and **Centerline – Pink** the stationing will be every **50ft**.
6. For alignments using **Centerline – Small Blue** and **Driveway** the stationing will be every **10 ft**.



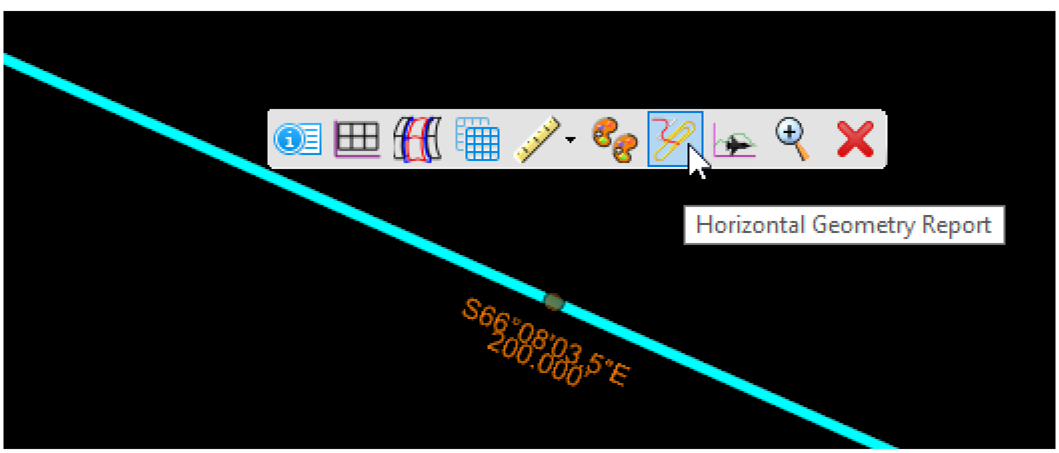
7. To remove annotation, use the **Remove Element Annotation** tool, follow the prompts. ***Locate Elements – Reset To Complete***. Click on the alignment, after it is highlighted, ***right-click to Reset***. The annotation is removed.

3.5 Horizontal Reports

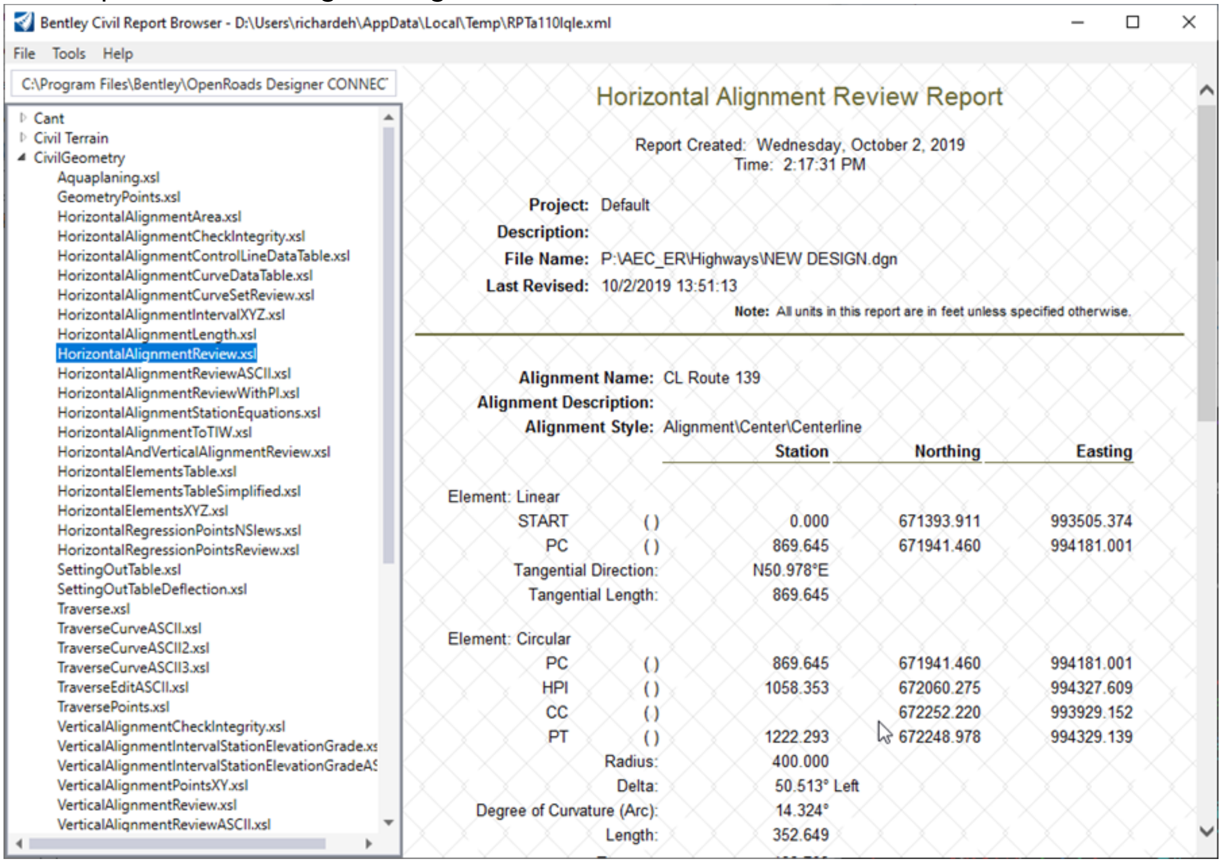
Several preconfigured reports are provided with the OpenRoads software that can be used to review alignment information.
Various reports can be accessed from the **Home** tab, as shown below.



Additionally, reports can be generated by selecting a civil geometry element and choosing the Reports icon from the pop-up menu.

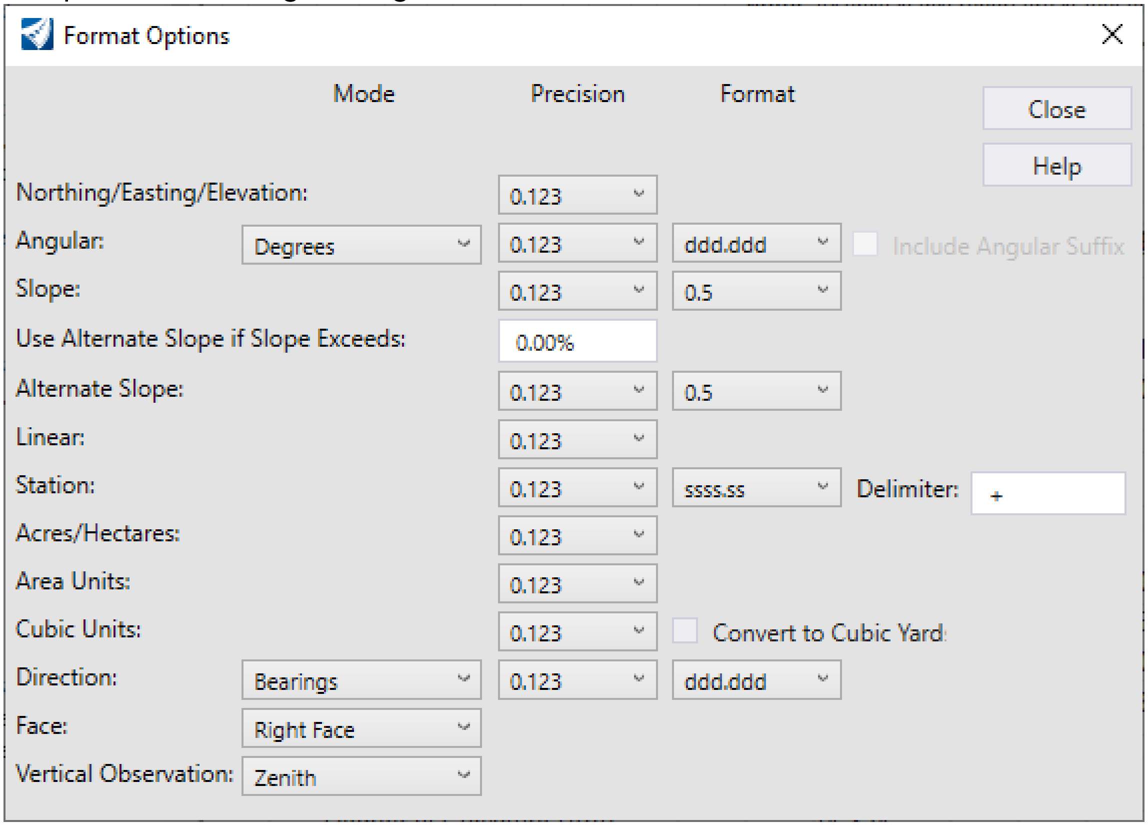


When a report is generated, the default report for the element type is displayed. The example below shows the default report for an alignment.



Several reports are available and can be selected from the list at left. Depending on the element type that the selected report has been configured to accept, it may not show any information.

The format of the report can be modified by selecting ***Tools > Format Options***. The dialog below is opened.



The format options are stored as a user setting in the Windows Registry and must be manually set by each user on their computer. The settings shown above are recommended for CTDOT projects. Changes made to the Format Options dialog are dynamically applied to the current report when the dialog is closed.

Reports can be saved in various formats using the **File › Save As...** command from the main Report Browser dialog.

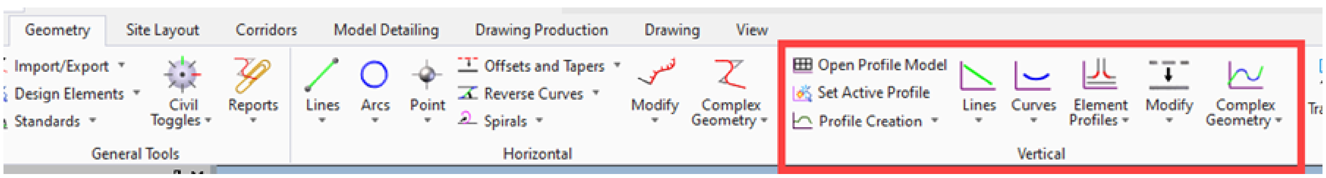
Exercise 4 – Profiles

In this module the creation of a vertical alignment (proposed profile) with the PI method will be described. There are several methods to create vertical alignments for more on other methods or other alignment tools please see the Bentley's OpenRoads Help – Vertical Geometry.

OpenRoads Designer stores vertical alignments as a component of the horizontal alignment in the same design file as the horizontal geometry. This is true for both existing vertical alignments, which are extracted from an OpenRoads Terrain Model, and proposed vertical alignments.

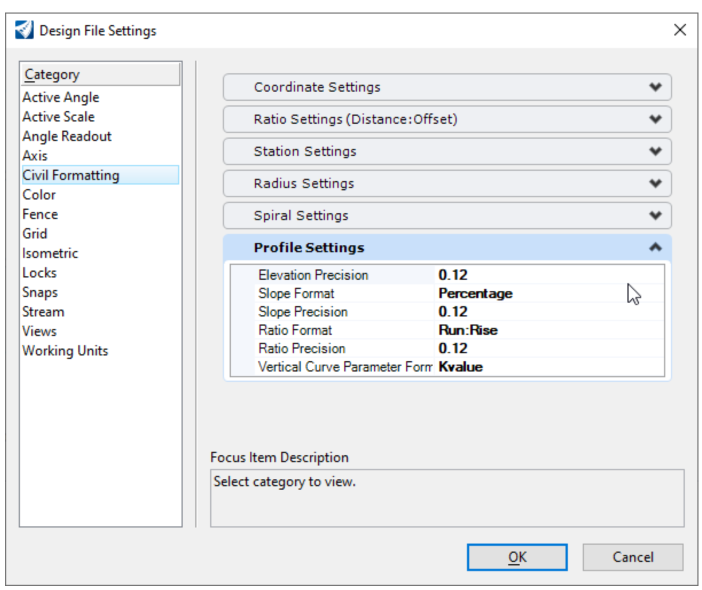
Although the vertical geometry is stored in the design file, the graphical display of the vertical alignment information is not plotted in the design file. Vertical alignments are displayed in a profile window for review and editing.

Vertical geometry tools are accessed from the **Geometry** tab of the **OpenRoads Modeling** workflow, as shown below.



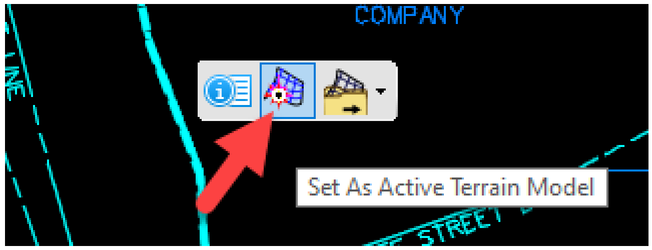
4.1 Profile Settings

The Design File settings contain parameters for formats as you work with vertical alignments. ***Select File > Settings > File > Design File Settings*** and then click the **Civil Formatting** item from the list on the left of the dialog. The **Profile Settings** are set to the values shown below in the CTDOT seed files.

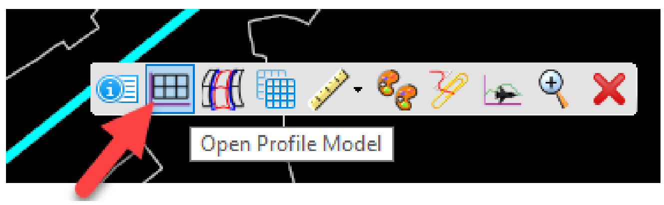


4.2 Existing Ground Profiles

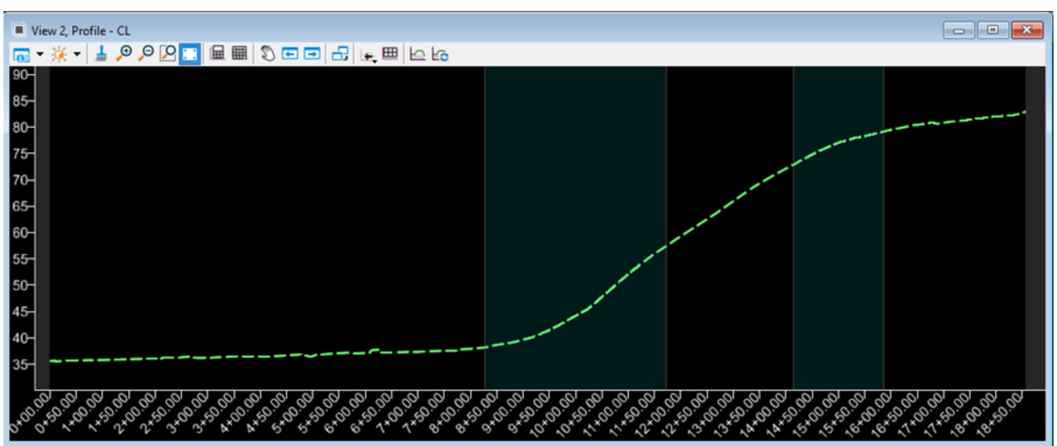
Existing ground profiles are extracted along the selected civil geometry element from the active terrain model. The active terrain model is set by selecting the terrain element and choosing the **Set As Active Terrain Model** icon as shown below.



To extract the existing profile, select the alignment graphics and let the cursor rest on the graphic element until the pop-up menu appears. Choose the **Open Profile Model** icon as shown below.



You are prompted to select or open a view window to use for the profile display. Open a new view window and then left click in the view window to display the profile. An example is shown below.



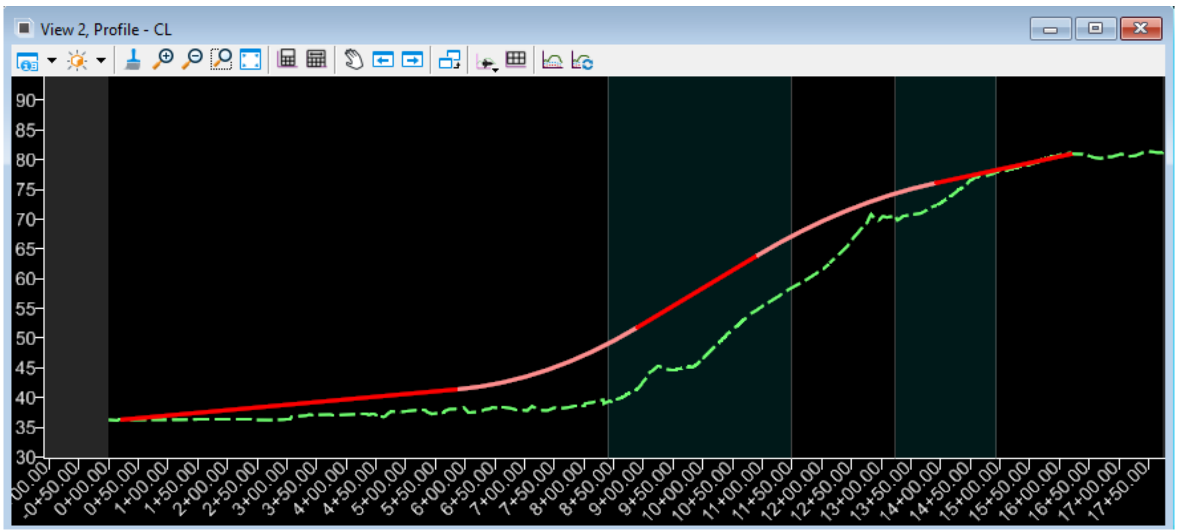
Note: *The profile is displayed in the profile view for review and editing; however, the profile is not written to the design file. The selected view window is used as a temporary display of all the profiles associated with the selected alignment. The selected view window is not specifically associated with an alignment/profile and can be used to display the profiles from different geometric alignments as needed.*

4.3 Profile Feature Definitions

The symbology of vertical geometry is defined by Feature Definitions; however, the feature definition for profiles is not assigned by the user. Feature definitions for vertical alignments are defined as follows:

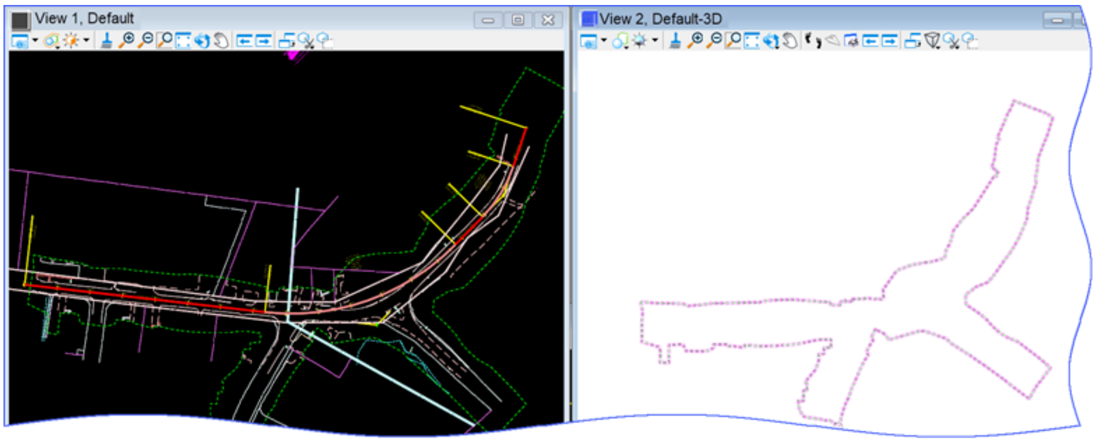
- Existing vertical alignments inherit their display parameters from the feature definition that has been assigned to the terrain model that was used to extract the profile.
- Proposed vertical alignments inherit their display parameters from the feature definition that has been assigned to the horizontal geometry that the vertical geometry is associated with.

To help differentiate between the tangent lines and vertical curves, proposed profiles are defined to display the vertical curves with color 20, and tangent lines with color 1.

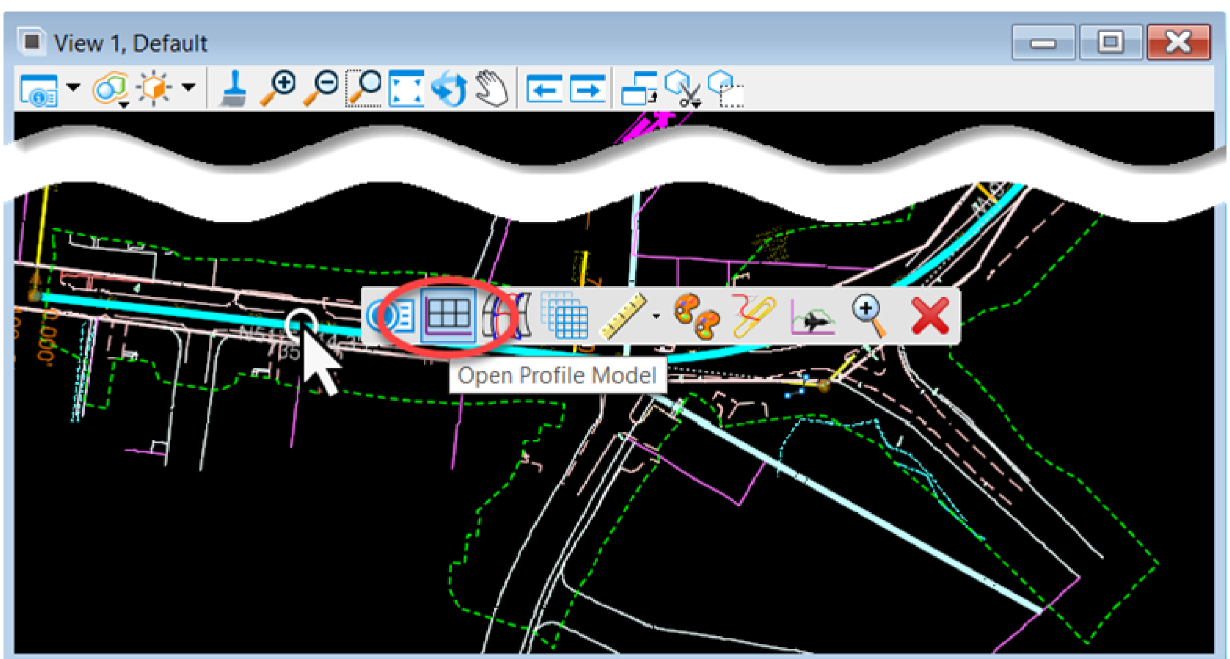


4.4 Creating Vertical Alignments

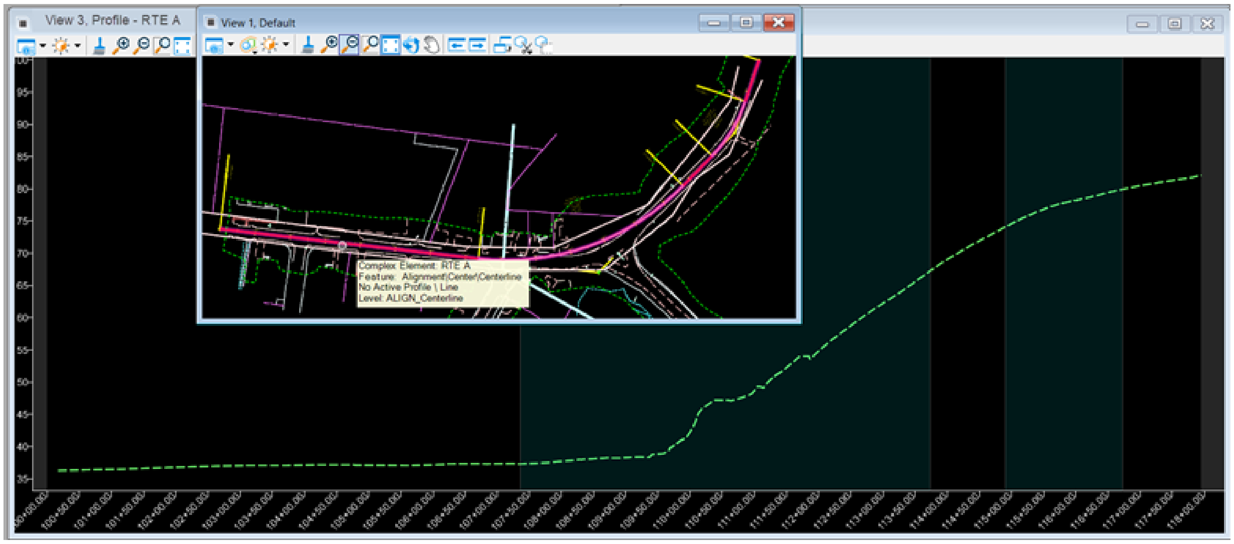
1. There should be 2 view windows open: **View 1, Default** showing the survey, terrain and the horizontal alignment; **View 2, Default-3D** showing the terrain, there are no other 3D items. The 3D terrain should only have the Boundary turned on.



2. If not already docked, from the **Geometry** ribbon select the **General Tools** tool and select and dock the ***Design Standards Toolbar*** and the ***Feature Definition Toolbar*.** These tools can be used to better adhere to design standards.
3. Next open another view window (View 3 Default) for the profile view of the existing terrain at the horizontal alignment, a longitudinal cross section of the existing terrain at the centerline.
4. In the horizontal view (View 1, Default) with the **Element Selection** tool select the Centerline (the horizontal alignment previously created). Example: RTE A.



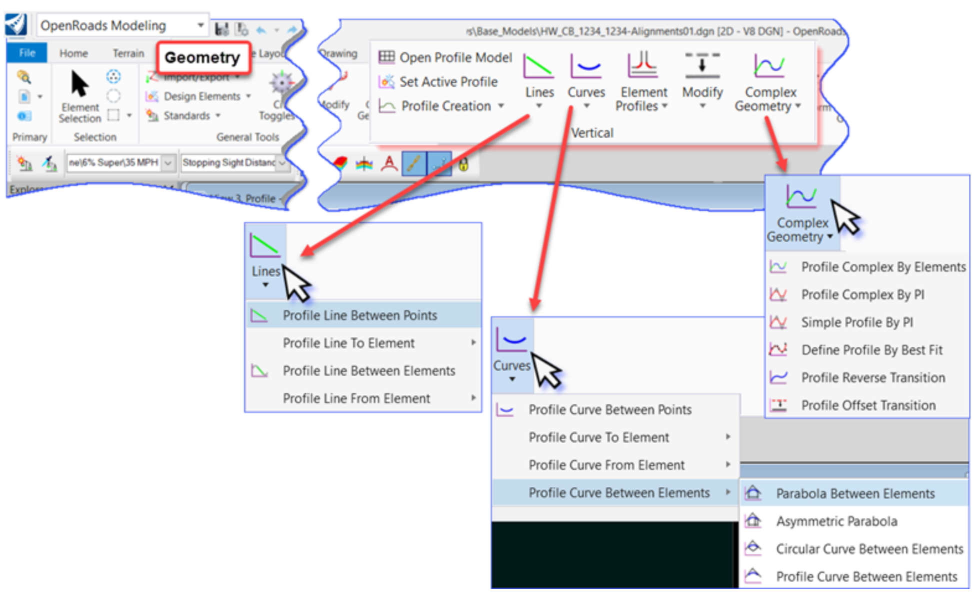
5. Hover over the horizontal alignment for the context menu (pop-up menu) to come up and select the tool. Follow the prompts. Select the Open View (View 3, Default) window. Click within the view and the existing terrain profile will be shown (Terrain must be active).



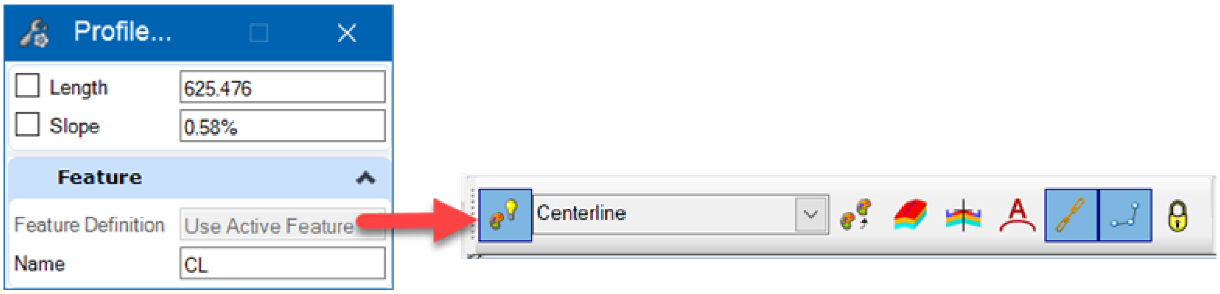
6. The view window's name will also change, example: from View 3, **View 3, Profile – RTE A**.

7. The **Design Standards previously** selected should still be active if previously activated. Example: ***6% Super*** (2 Lane AASHTO Design Standards 2018 – Imperial Design Library) – 35 MPH (2 Lane AASHTO Design Standards 2018 – Imperial Design Library) and ***Stopping Sight Distance*** (AASHTO Design Standards 2018 – Imperial Design Library).

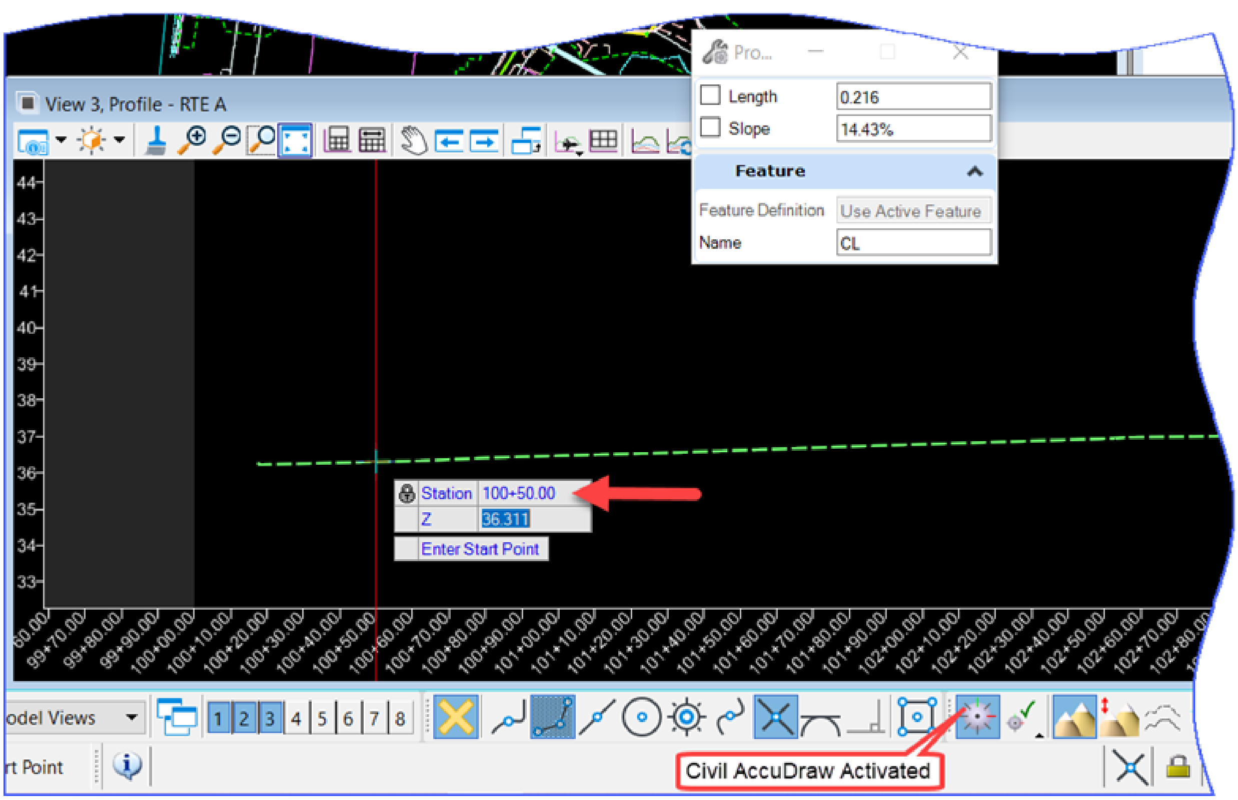
8. From the **Feature Definition Toolbar**: select the active feature definition for ***Alignment - Center – Centerline*** and ***Lock*** the feature definition. Activate the ***Chain Command***, this will chain the tangents together, but still remain individual elements.



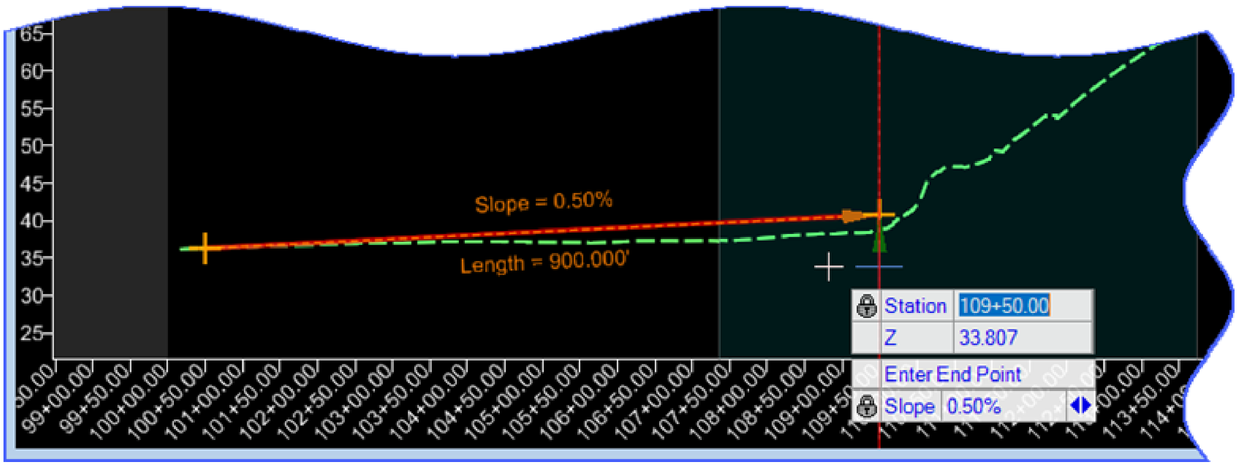
9. Next from the **Vertical tool group** select the ***Lines – Profile Line Between Points*** tool. In the **Line** toolbox, the **Feature Definition** should be set to: ***Use Active Feature***, this should be greyed out, and it will set the feature tangent line to the active feature definition picked in the previous step.



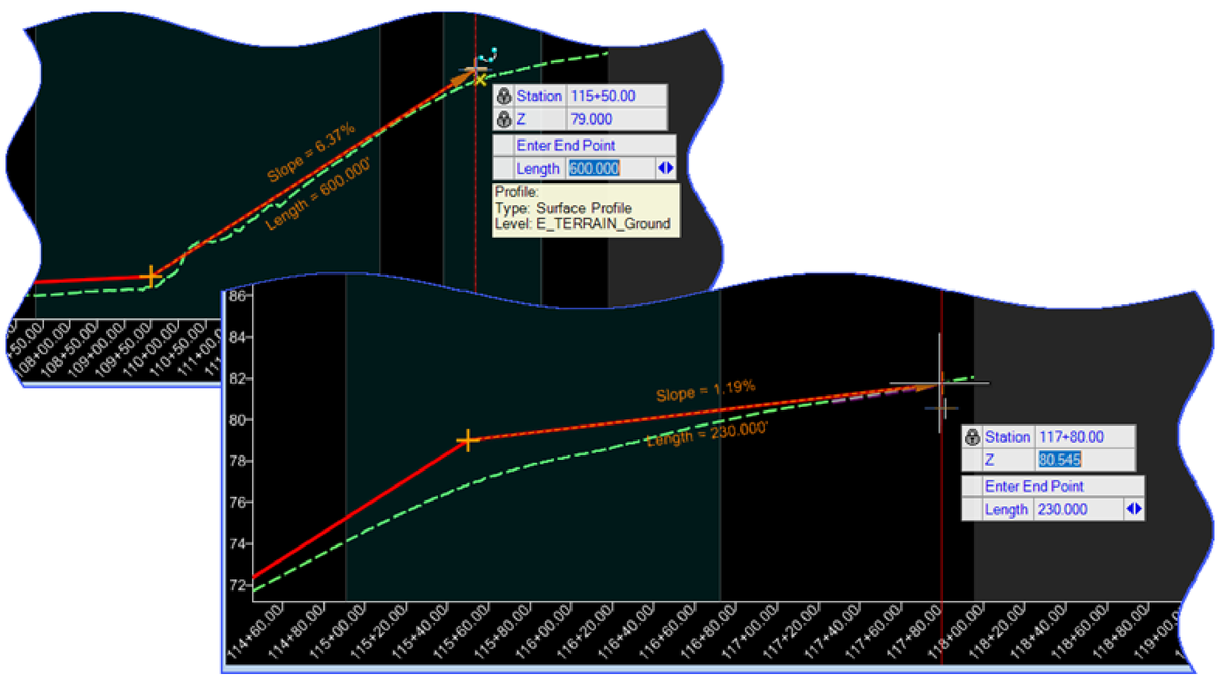
10. In "**View 3, Profile – RTE A**" (example) ***select the first PI location***. For VPI 1 (Vertical Point of Intersection) you can just click to the beginning of the existing profile, or you can use **Civil Accudraw** and to use ***Station and Elevation*** for the VPI method. ***Click*** on the **Civil Accudraw** icon bring the courser back into the view, now there is ***Station and 'Z'*** (for elevation) attached to the courser. ***Type in the Stationing*** (100+50.00) and hit the ***Tab key***, the 'Z' value for elevation can be added or left blank. The view will now show a vertical red line at Station 100+50, ***snap*** to the beginning of the terrain and ***Accept (left click)***.



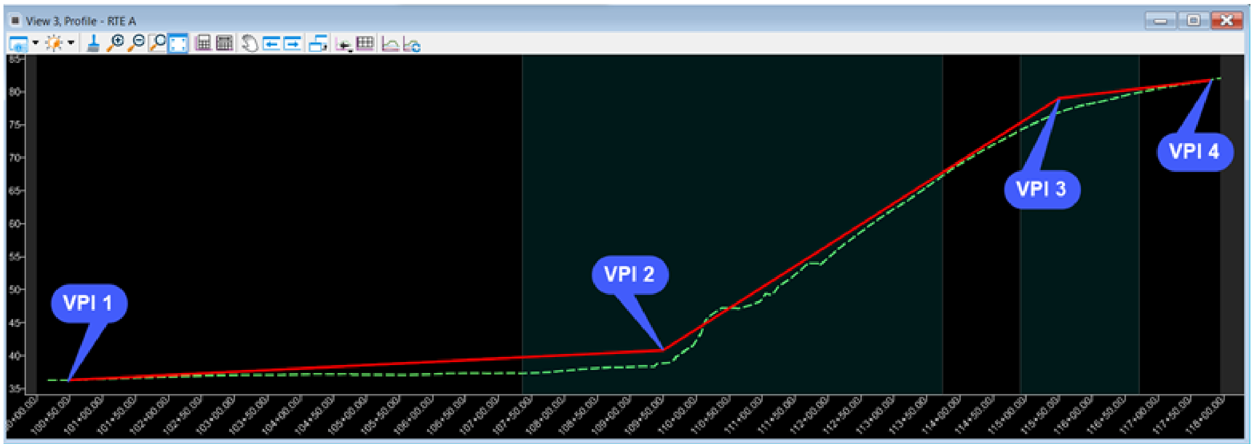
11. Move the courser and pick the next VPI (VPI 2) by ***Station, Elevation or Distance***, example: Station: 109+50, hit the ***Tab*** key; you can enter the Elevation, or Tab again, enter a length, or use arrow key to enter a desired slope.



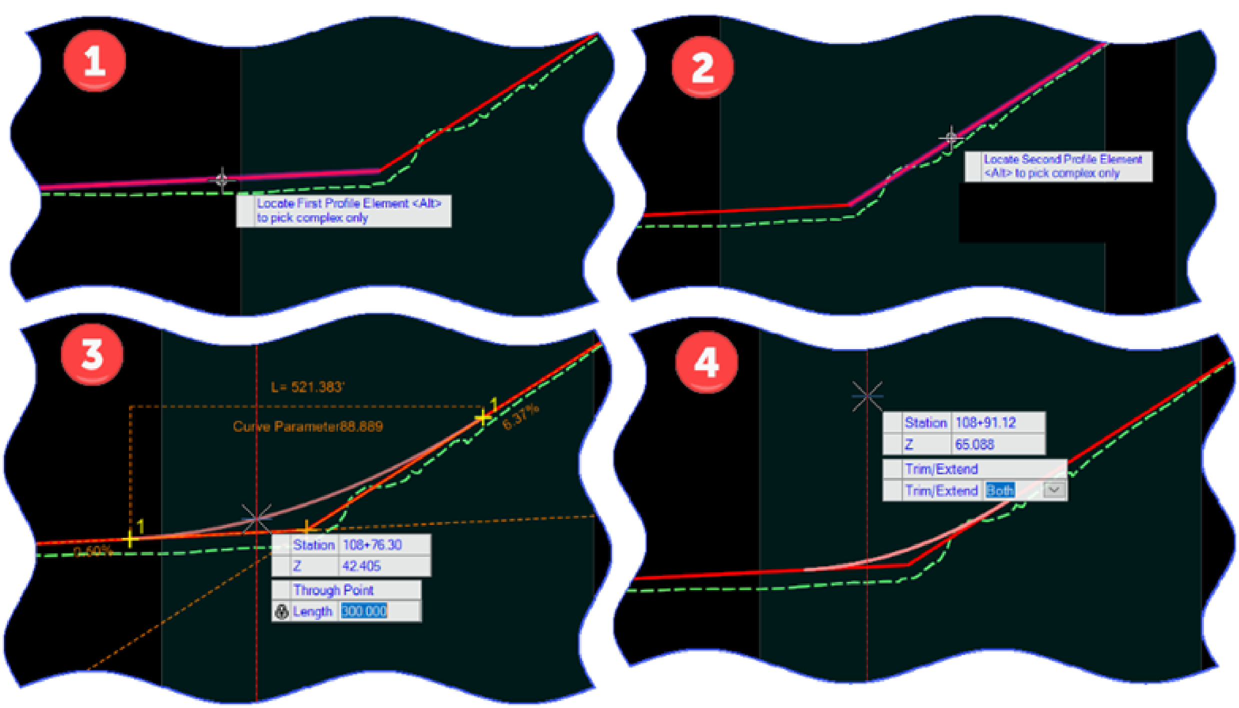
12. ***Repeat*** this for VPI 3, example: Station: 115+50, Tab key, enter Elevation 79.00, Tab key. Click the left mouse button to accept and place the VPI.
13. For the last VPI (VPI 4) click to the end of the existing profile or enter a Station, example: Station 117+80, tab (now station is locked) click on the existing profile to match in to. Click the Right mouse button to stop the command and select the Element Selection tool to fully exit the command.



14. ***Review*** your vertical alignment (proposed profile). The Feature Definition Lock and the Design Standard Lock are still on (set). Making it unnecessary to enter a Feature Definition.

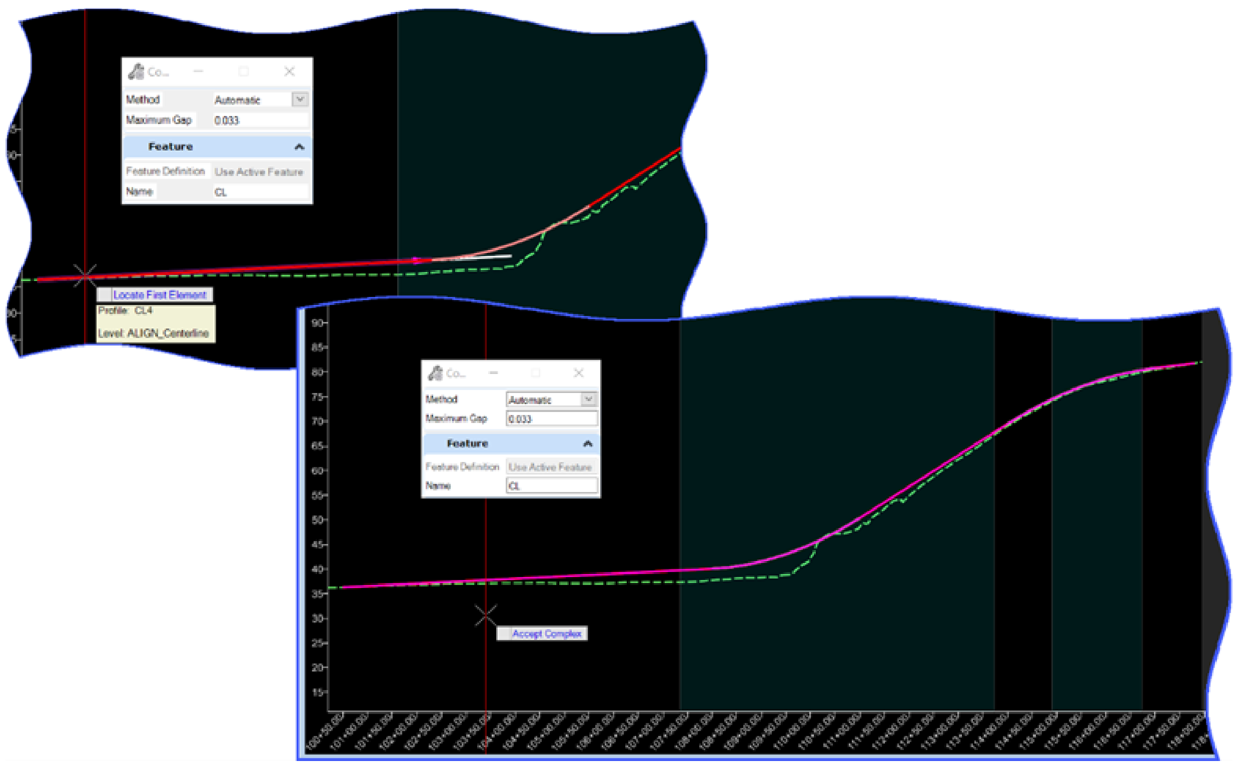


15. Next the vertical curves will be placed. Select the **Curves – Profile Curve Between Elements – Parabola Between Elements** tool. Follow the prompts. ***Locate First Profile Element*** – click on the first tangent. ***Locate Second Profile Element*** – click on the second tangent. Next ***define the curve by entering the length***, example: 300.00 (feet). ***Accept*** (left-click), ***Trim/Extend*** use the arrow keys to go to Both, ***Tab*** key. ***Accept*** (left-click). Repeat these steps for the next curve.

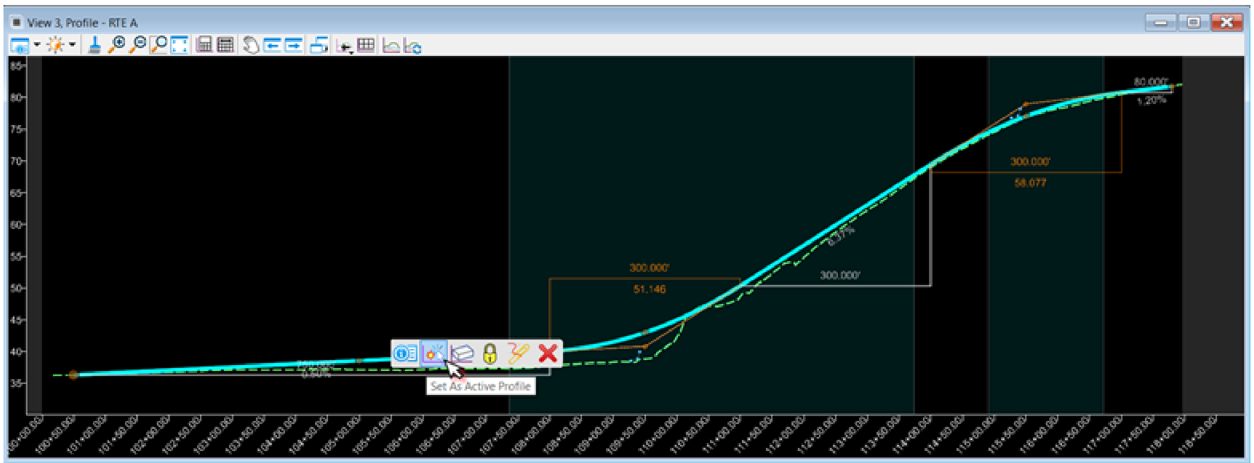


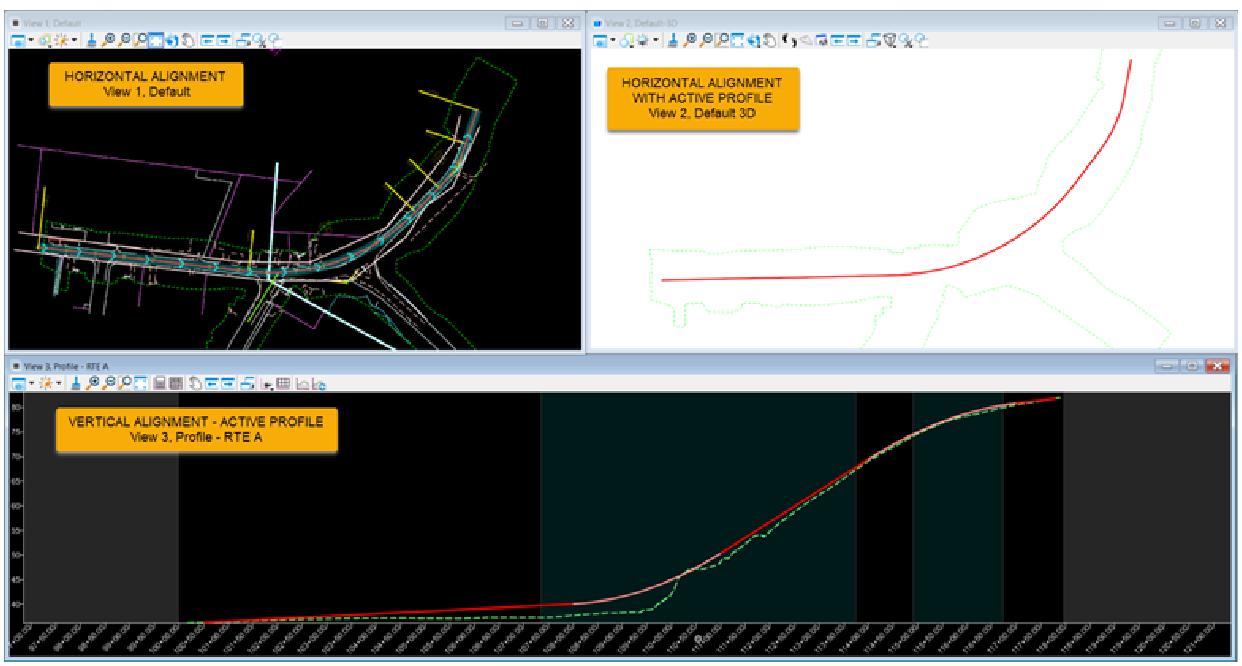
16. ***Review*** the vertical alignment (proposed profile). Make adjustments if needed. Using the ***Element Selection*** tool click on the first tangent, depending on the rules (relationships) with which the tangent was placed ***Length, Slope, Station and Elevation*** can be changed. The ***Length of Curve*** can be adjusted.

17. Complex the vertical alignment, select **Complex Geometry – Profile Complex By Elements**. Set the following: ***Method to Automatic***, ***Maximum Gap to 0.033***. Feature Definition: Use Active Feature (greyed out – Feature Definition locked previously), ***Name: enter the alignment name*** – example: RTE A. Follow the prompts. ***Locate First Element***. Select the beginning of the vertical alignment, the entire alignment will highlight. ***Accept*** (left-click).



18. Set the profile active. Using the ***Element Selection*** tool, select the profile, from the pop-up menu select: ***Set As Active Profile***. This will associate this profile with the horizontal alignment, and you can now see the alignment also in the 3D view (View 2, Default-3D). ***Save Settings***.





The profile in the alignment dgn file is not annotated, for annotation of profiles see Volume 13

Revisions