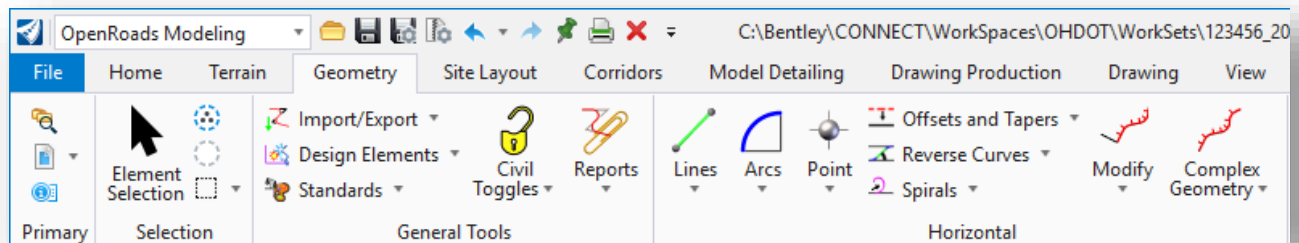# OpenRoads Designer: Horizontal Geometry

OpenRoads Geometry Tab

The OpenRoads Geometry commands are selected from the **Geometry** tab of the **OpenRoads Modeling** workflow, as shown below.

The commands are separated into two main categories, **General Tools** and **Horizontal** as shown above.

This training guide will not document each geometry tool that is available in the OpenRoads interface. See the online help for commands not detailed in this document.
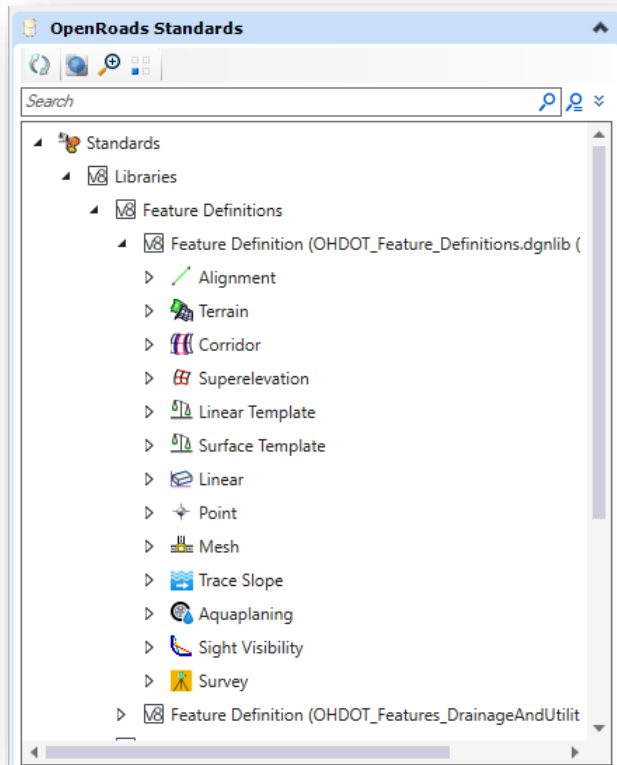
Feature Definitions

As you draw information using the OpenRoads horizontal geometry tools, OpenRoads **Feature Definitions** are used to define the plan view representation of an object as well as how the object will appear in other contexts such as the profile, cross sections, and 3D.

What is a Feature?
Simply stated, a feature is anything in your design that represents a real-world object such as:

- Curb and Gutter
- Pavement
- Catch basins, Manholes, Inlets
- Walls
- Ditches
- Cut and Fill Slopes
- Etc.

Features are defined in MicroStation DGN Libraries. The OHDOT configuration defines which feature libraries are available for ODOT use. The available feature definitions can be reviewed in the *Explorer* dialog as shown below.


In addition to defining the symbology settings, feature definitions can also define annotation settings for each item.

For example, the **Alignment** feature definitions also define how an alignment will be annotated using the OpenRoads **Element Annotation** and **Model A**nnotation commands. These tools are described later in this document.

The features are organized by categories. Expanding a category will reveal the features available in the category.

The feature definition is assigned when an element is placed. This can be accomplished by choosing the **Feature Definition** from the ***Tool Settings*** dialog as shown below.
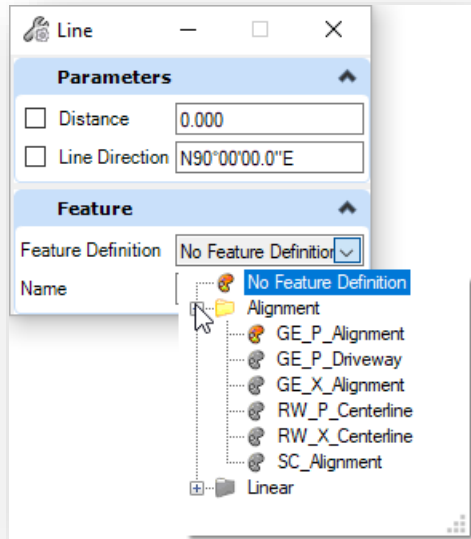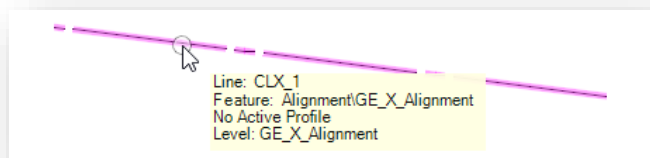

Feature Name

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 right, the **Line Between Points** command has been selected. The **GE_P_Alignment** feature definition has a seed **Name** prefix of **CLP_1** 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 "CLP_1", the second is named "CLP_2", etc.
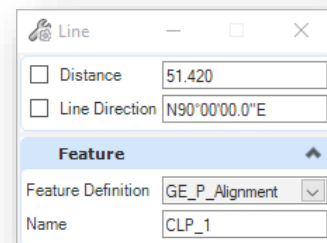
Note: By default, the software always displays the seed name, not the actual name of the element that is placed.

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.
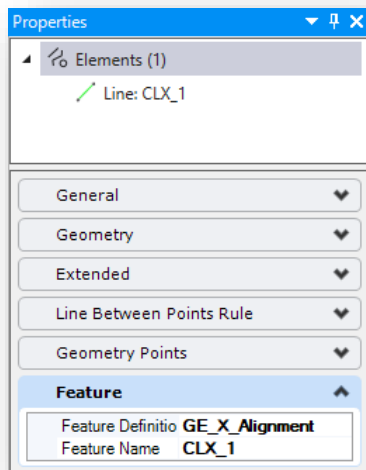
Standard element names for alignments are defined in the **ODOT CADD Engineering Standards Manual**.
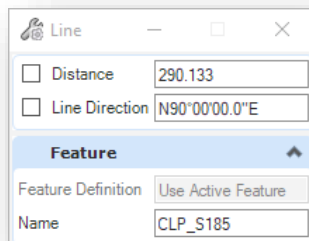
Feature Definition Toggle Bar

The **Feature Definition Toggle Bar** is accessed from the ribbon by selecting **OpenRoads Modeling > Geometry > Standards > Feature Definition Toolbar.**
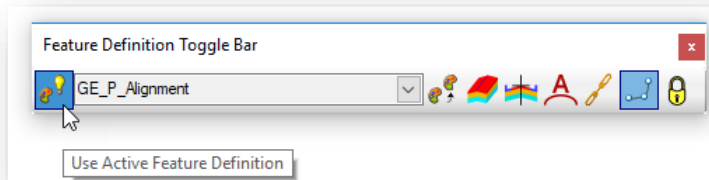
Toggle on the first icon to automatically assign the selected feature as the active feature as you use various OpenRoads Geometry commands.

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**.

See the online help for a description of the remaining **Feature Definition Toggle Bar** icons.

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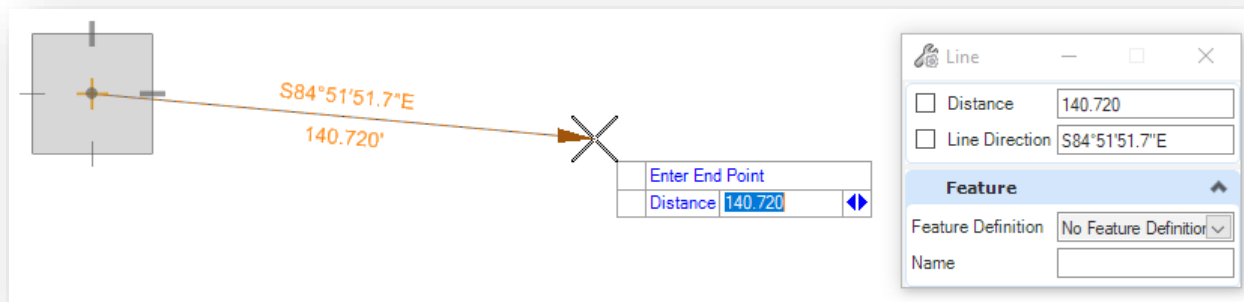
These parameters can be entered in the *Tool Settings* dialog, shown at right above, or by using the parameters floating on the cursor.

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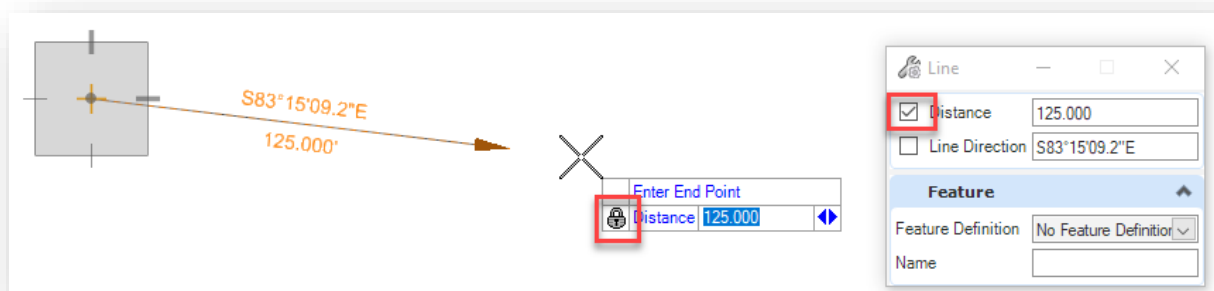
- After a parameter value has been keyed in, use the left and right arrow keys, or the **Tool Settings** dialog to enter values for the remaining parameters

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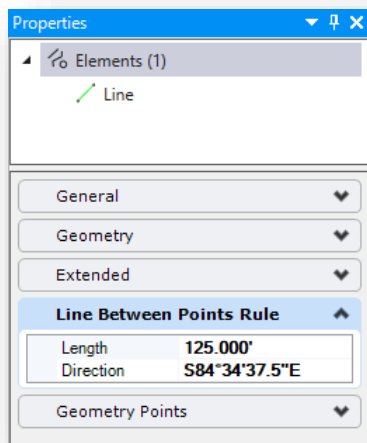
MicroStation's **Manipulate** commands (Copy, Scale, Rotate, etc.) cannot be used on ruled geometry. The rule must be removed to use these commands.

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.
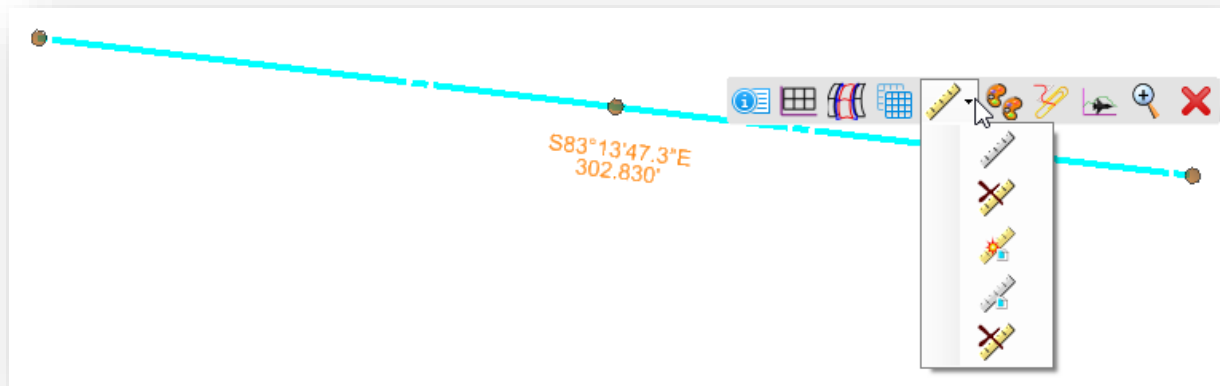
The following options are provided:
- Deactivate Rules (Activate Rules is the selected elements rules are currently deactivated)
- Remove Rule
- Activate Referencing Rules
- Deactivate Reference Rules
- Remove Rule (Remove Referencing Rules)

Selecting the Deactivate/Activate Rules and Remove Rule icons affect only the selected element.

Reference rules are elements upon which the chosen element is dependent, thus a curve between elements depends on the two other elements that define it. This command allows deactivating the rules for those other elements.

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.
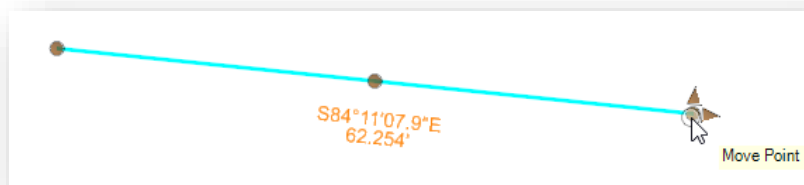

Tip: The size and color of the Manipulator Text can be adjusted in the **User Preferences**. From the MicroStation menu, select **File > Settings > User > Preferences** and then choose the **View Options – Civil** item on the left side of the dialog. The **Manipulator Size** or the **Manipulator Font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

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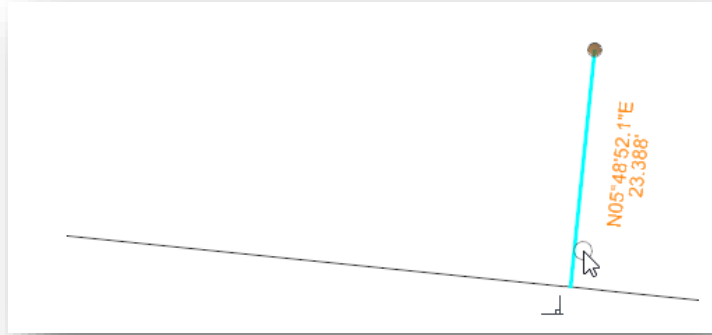
If the horizontal line is modified by changing the direction of the line, the design intent for the perpendicular line is preserved. The **Perpendicular rule**, and the ending location are preserved as shown below.
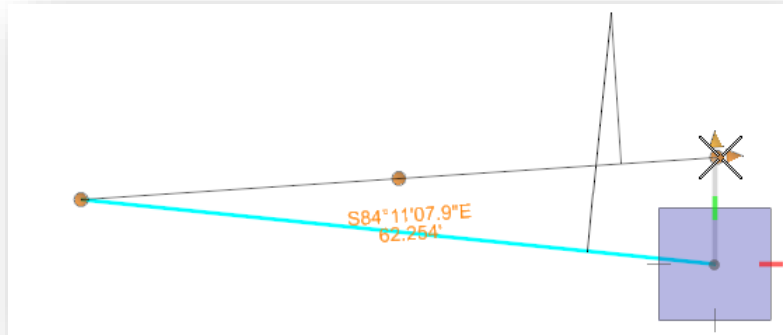
Snaps

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.
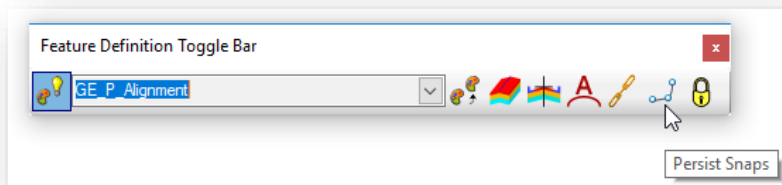
Editing Dynamic Text

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.
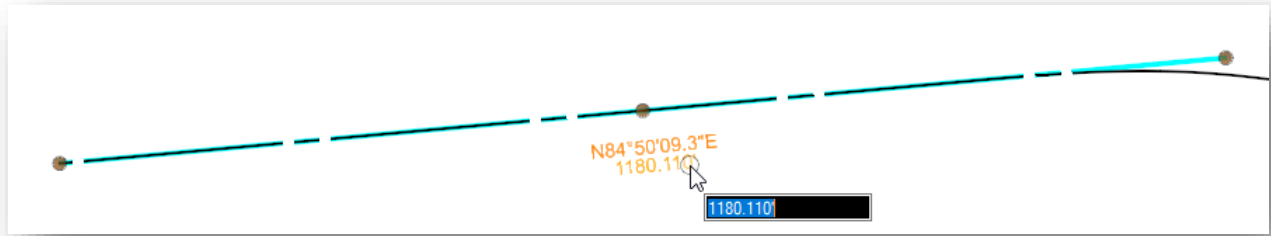

Take the following steps to select the underlying element.

- 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

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 **Deactivate Rule** option as shown below.
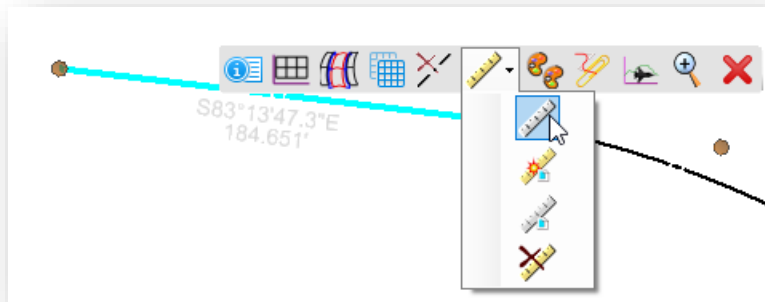
Multiple elements can be locked at once by choosing **Geometry > Civil Toggles > Deactivate Rules**.

Design Standards

Design Standards are used to monitor various alignment parameters on OpenRoads geometry elements. The standards are stored in the DGN Library **OHDOT_DesignStandards.dgnlib**.

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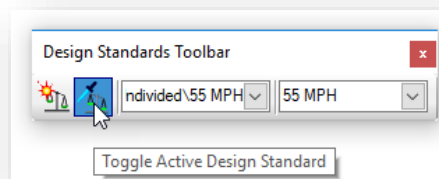
The toolbar contains a drop-down list to select the design standards and the design speed. The vertical design standard, displayed in the drop-down menu to the right, is automatically set when the horizontal design standards is selected.

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.
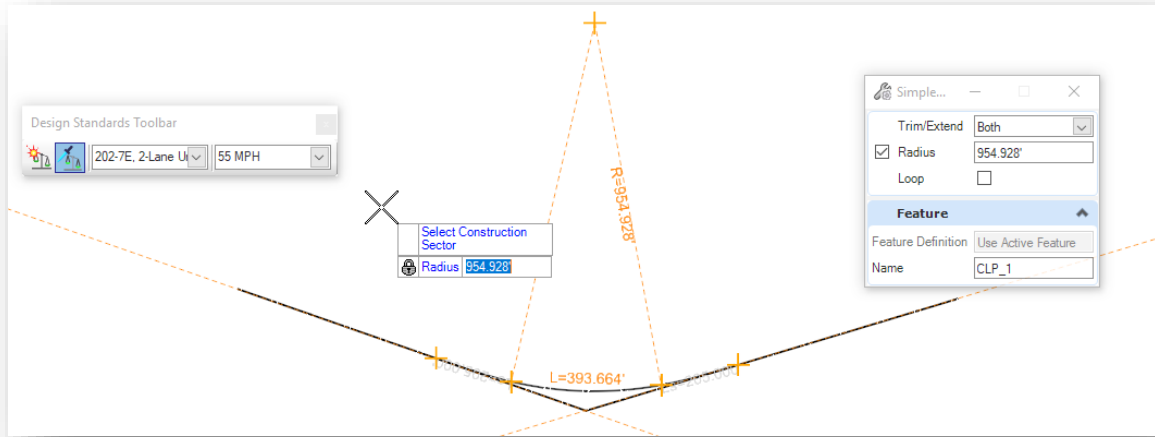
To Apply a Design Standard

The example below details the process to apply a selected design standard to the **Simple Arc** command to place a curve between two tangent lines.

- The *Design Standards Toolbar* is set to the ODOT standard **202-7E, 2-Lane Undivided** with a design speed of **55 MPH**

- The **Toggle Active Design Standard** icon is checked on

- Using the **Simple Arc** command, the **Radius** value defaults to the value defined in the selected design standard, **954.9279**

- The design standard defines a minimum transition length of 205 for the default radius. Spirals will be placed even though the **Simple Arc** command was initially selected.

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*.

To Review a Design Standard Violation

If a design standard is violated, the software will display an icon on the element with the violation like the example below. Floating the cursor over the icon will display the error message.
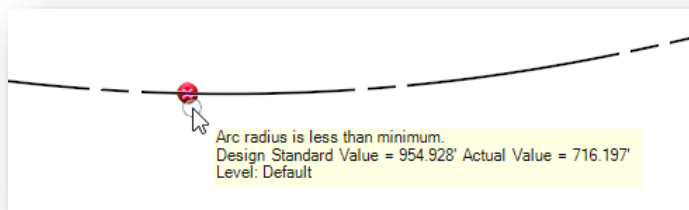
Errors can also be reviewed in the **Civil Message Center** by selecting Geometry > Standards > Civil Message Center. Toggle on the **Error** option to view any errors.
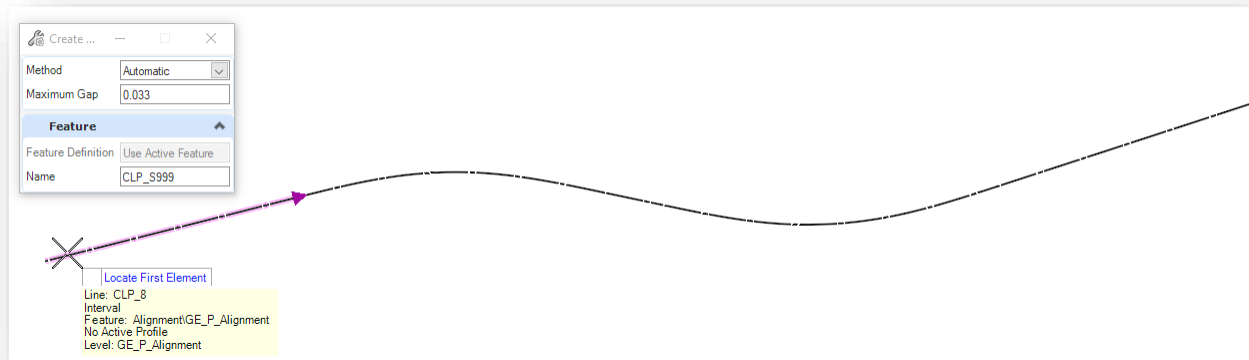

You can zoom to the error by right-clicking on the error and choosing the **Zoom To** option.

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. Standard geometry names are defined in the **CADD Engineering Standards Manual**, **Section 306.2**.

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.

Stations and Station Equations

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.
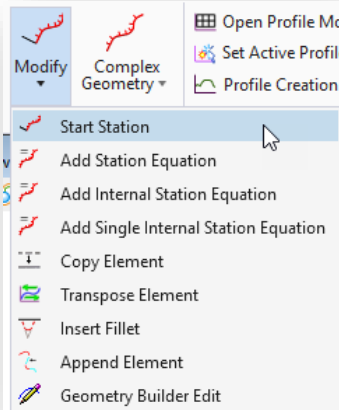
The command has two parameters:

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.


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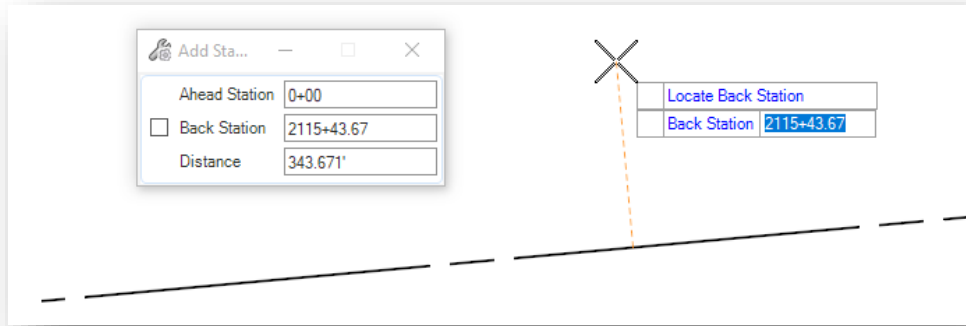
The ahead station is stored as a "region" much like the legacy GEOPAK format. When the **Ahead Station** value is entered, it is not necessary to define the region number.

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.
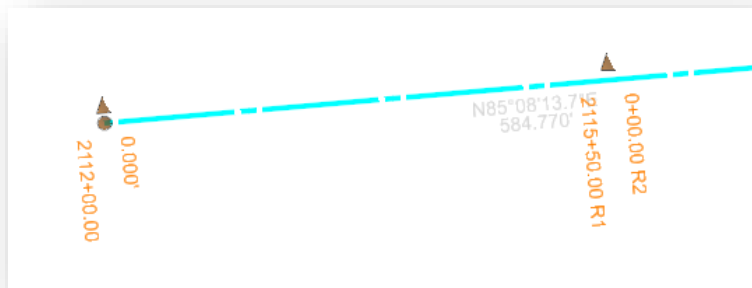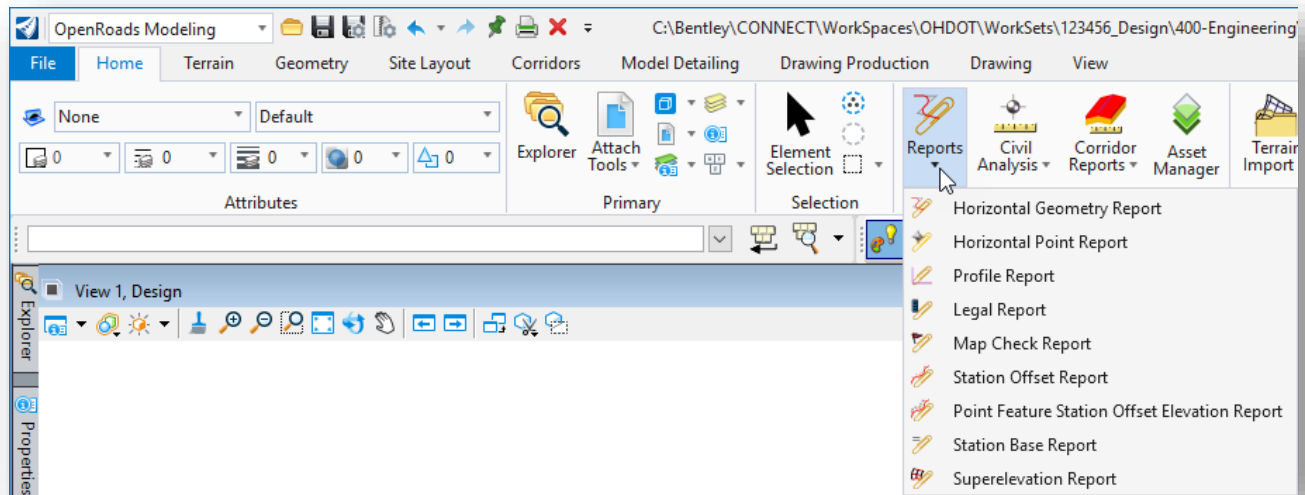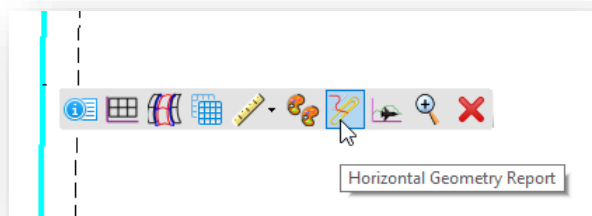
Reports

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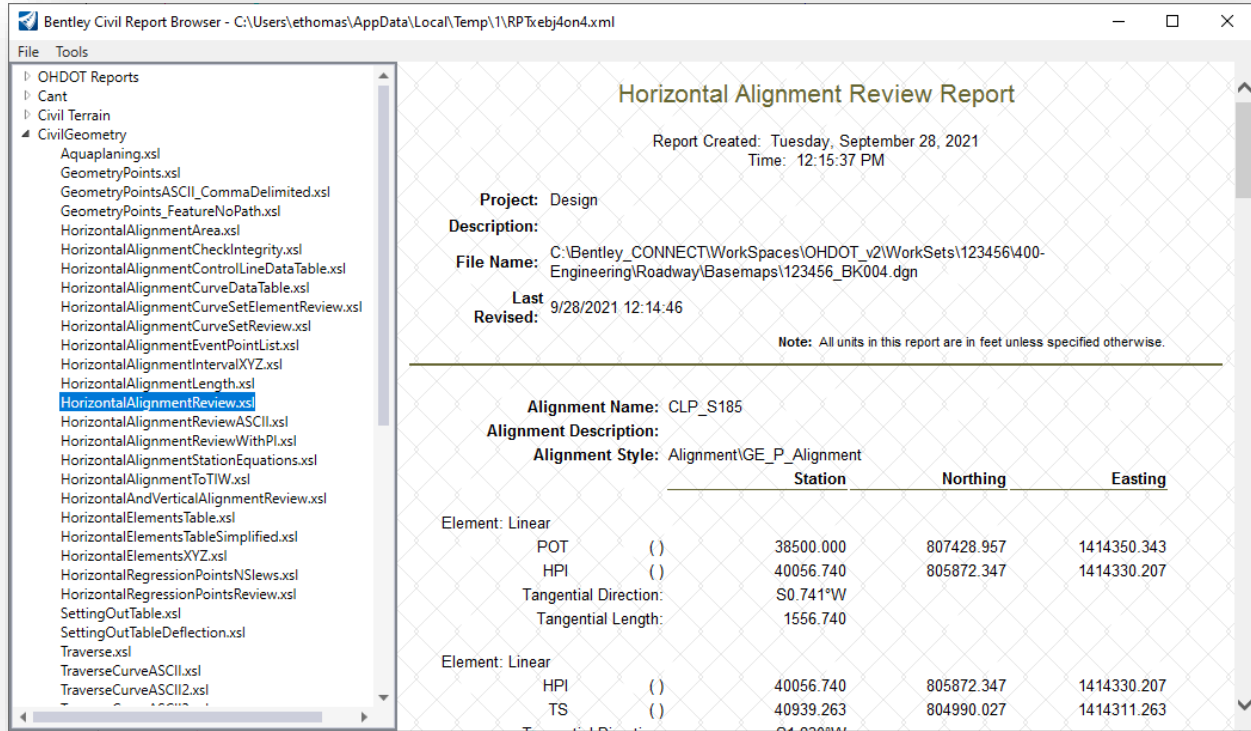
Various reports can be accessed from the **Home** tab, as shown below.

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.
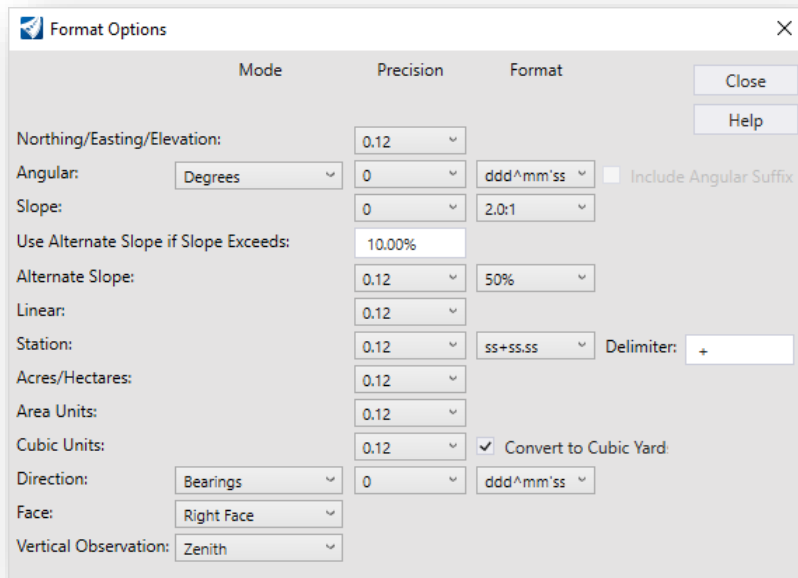
The format options are stored as a user setting in the Windows Registry and must be manually set by each user on their computer.

The settings shown at left are recommended for ODOT 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.

OpenRoads Software Version

This document was prepared using the following software version:

OpenRoads Designer CONNECT Edition – 2021 Release 1 Update 10 – Version 10.10.01.03

Contacts

For any questions, suggestions, or problems with this document please contact the ODOT Office of CADD and Mapping Services by use of the following form on the ODOT website:

https://odot.formstack.com/forms/cadd_servicerequest