OHDOT ORD Overview Training Guide

Department of Transportation
Office of CADD & Mapping Services

Ohio Dept. of Transportation

Office of CADD and Mapping Services

Updated: 01/25

Table of Contents

Section 100 OpenRoads Designer Introduction

OHDOT OpenRoads Training Configuration

This Ohio Department of Transportation training material is available for both internal and external users. This training guide is structured for ODOT's internal users with the training data located in the ProjectWise environment.

Training Guides

ODOT's training guides for ORD are available in the OHDOT CADD Standards. As these training guides are updated, the current version can be found in the following folder:

For an installation of the OHDOT CADD Standards within ProjectWise:

Ohio DOT Projects\Documents\03 Standards\CONNECT_Config\WorkSpaces\OHDOTCEv02\Standards\OHDOT Utilities\Training\OpenRoads\ORD Design\

For an installation of the OHDOT CADD Standards external to ProjectWise:

...\OHDOT\Standards\OHDOT Utilities\Training\OpenRoads\ORD Design\

The training guide is modular in approach, with each topic presented in individual documents. This allows ODOT to easily update individual sections without impacting other sections.

Training Datasets

For external users, the files used in this training are available in the OHDOT CADD Standards. See the following document for more information:

...\OHDOT\Standards\OHDOT Utilities\Training\Setting up Training.pdf

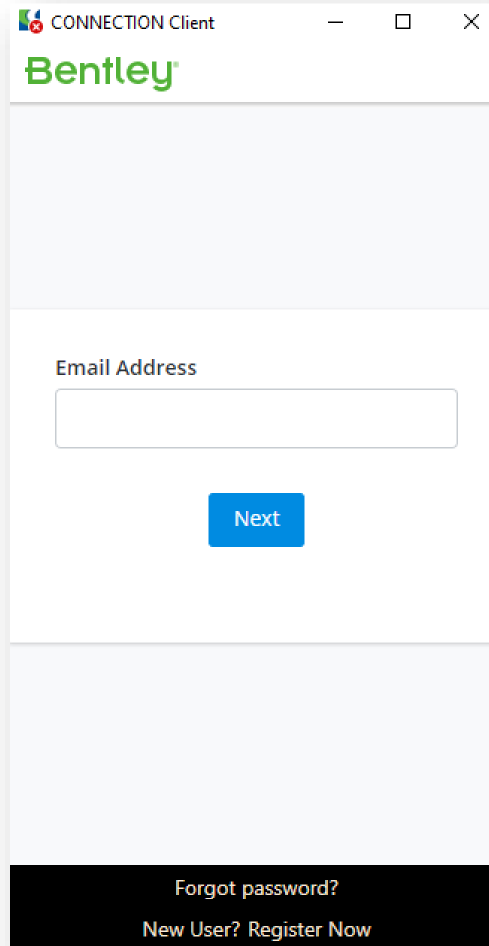
The training manual is presented in a modular approach; however, the training exercises are sequential with each training example building off the previous exercises. Additional datasets, with the sample project partially completed in various states appropriate for the exercises, are planned for a future update.

Bentley CONNECTION Client

Bentley Connect licensing requires users to log into their Bentley account to secure a software license.

ODOT users should log in using your ODOT email address. The password is automatically applied based on your ODOT credentials.

Contact your District CADD Administrator if you are unable to log into your Bentley account.



OpenRoads Designer Introduction

OpenRoads Designer (ORD) is Bentley's civil design software solution for surveying, drainage, subsurface utilities, roadway, and site design. ORD supersedes Bentley's previous design software packages InRoads, GEOPAK, MX, and Power Civil.

ORD contains all the functionality of MicroStation CONNECT Edition with the addition of Civil design capabilities.

ORD/SS10 Compatibility

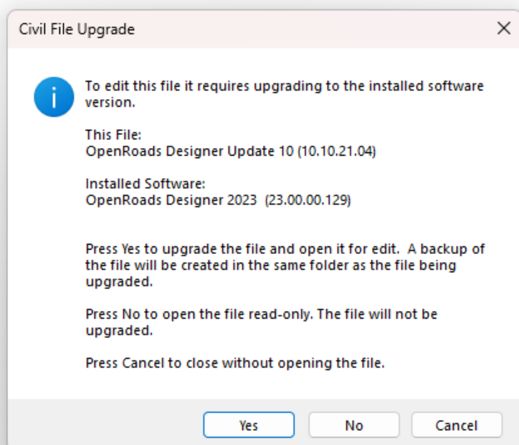
Many of the civil design and 3D modeling functions in ORD are identical in behavior and functionality to the OpenRoads tools that were available in previous Bentley civil design software offerings. However, the underlying technology has undergone significant enhancements and is not backwards compatible with the SELECTseries versions of InRoads, GEOPAK, MX, or Power Civil. Data can be exchanged between SS10 and ORD formats by use of XML files to export and import specific information. It is highly recommended that all users on the project, including any consultants or sub-consultants, use the same software versions.

ORD Software Updates

Bentley's software updates alternate between major and minor revisions.

- Major updates are indicated by the second number in the version number. These updates include new software functionality that changes the underlying software schema. Major updates are not backwards compatible with previous versions.

  In the example on the right, a file that was last edited with OpenRoads version 10.10 is opened with OpenRoads 23. Opening the file will upgrade the civil data to ORD 23. The file will no longer be editable in versions prior to 10.10 or 10.12.



- Minor updates do not change the underlying schema and do not require the file to be upgraded from a previous release with the same major version number.

As ODOT moves forward with software updates and the CADD Standards for ORD are updated, some of the updates to the CADD Standards will not be backwards compatible with prior versions of ORD due to changes to the schema. Therefore, it is important for both internal and external users of ORD and ODOT's CADD Standards for ORD to stay up to date with the current version of the software that is in use by ODOT.

The current ORD version that is in use by ODOT can be reviewed at the following web page:

> https://www.transportation.ohio.gov/working/engineering/cadd-mapping/cadd/current-cadd-software

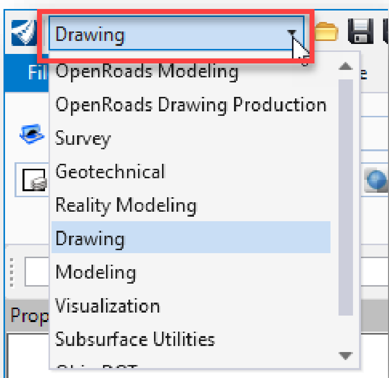
Ribbon Interface

ORD presents the user with a modernized ribbon interface, much like the various Microsoft Office products. Learning the ribbon is relatively easy but it will take a little time for users to get familiar with the new command layout.

The interface is organized by **WorkFlows**, **Tabs**, and the **Backstage**. Each is described below.

WorkFlows

The **WorkFlow** is selected by a drop-down menu in the upper left corner of the ORD interface.

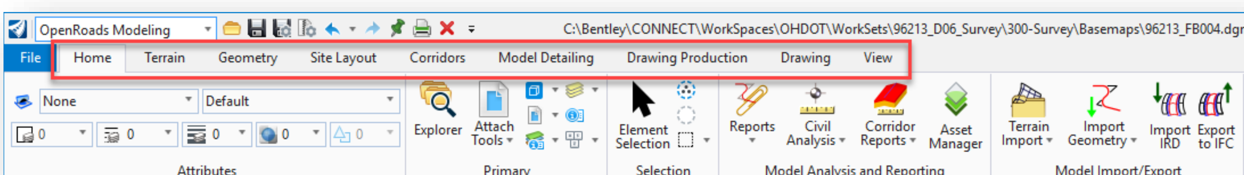


The **WorkFlows** present the various ORD commands logically organized according to a task. For example, the **Drawing** workflow contains commands that are used for typical MicroStation drawing tasks.

Some of the most used drawing functions are available in the other workflows.

Tabs

When a **Workflow** is selected, the various commands are organized into **Tabs** like the example below.

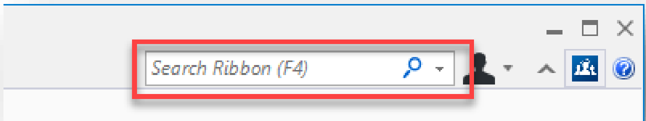


Each tab groups like commands into a logical organization of the various tools.

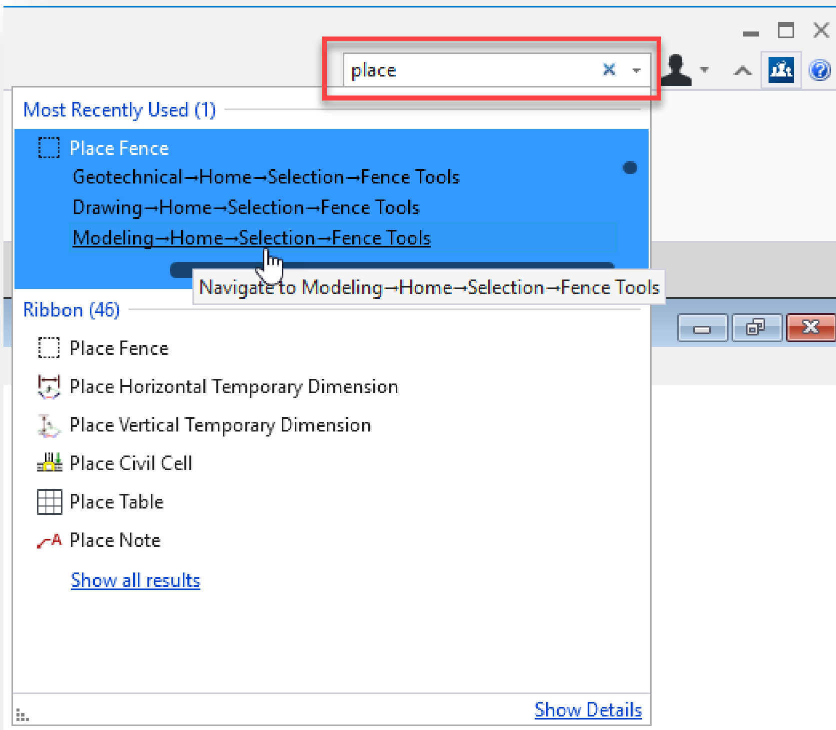
Note: Most **WorkFlows** contain a **Drawing** tab with commonly used MicroStation drawing tools.

Search Ribbon

The **Search Ribbon** key-in field, which is located at the top-right of the ORD user interface, is useful to quickly find a desired command as you learn the ribbon interface without manually searching the ribbon.



As you key-in the name of a command, the Search Ribbon function will show a list of commands matching the text string as it is entered. An example is shown below.



Most Recently Used commands matching the key-in entry are displayed at the top.

Floating the cursor over one of the results will expand the item to list the various places in the ORD interface where the command can be found.

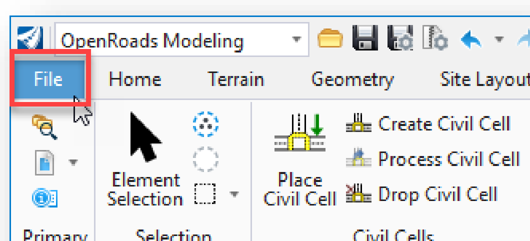
Click on the command name, without expanding the list, to run the command.

Click on the command location, with the list expanded, as shown at left, to navigate to the appropriate **Tab** in the ribbon.

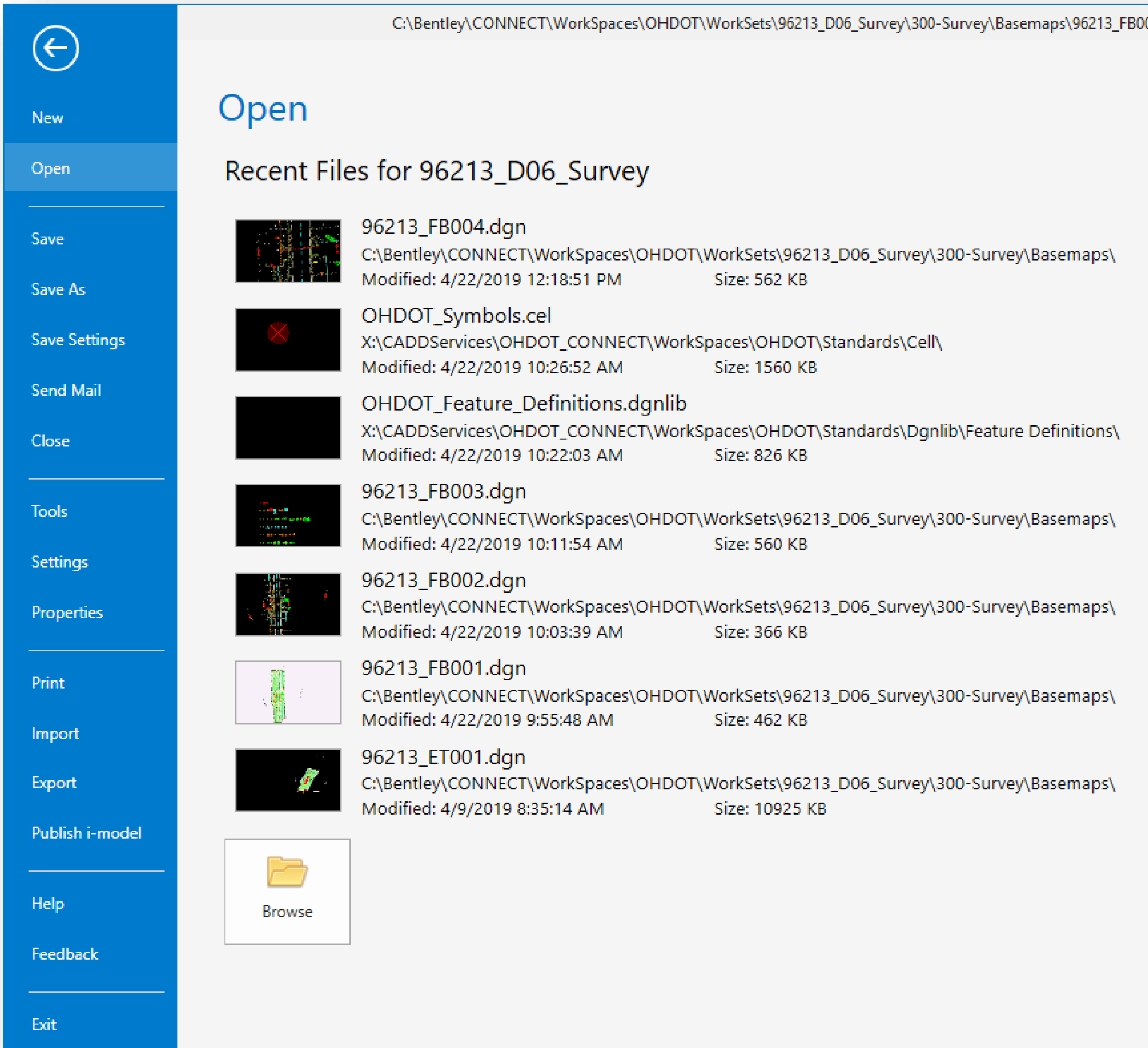
BackStage

The ribbon contains commands that are used for ORD and MicroStation related drawing and design tasks. Tools that are not command related, such as **Plotting**, **Design File Settings**, **User Preferences**, etc. are in the "**BackStage**" view.

To access the **BackStage**, select the **File** tab at the upper left of the ORD user interface.



The **BackStage** has a list of commands at left, along with a list of recently opened design file, as shown below:



We will not cover all the functions available from the **BackStage** in this introductory class. Two of the most accessed functions are listed below.

User Preferences

User Preferences are accessed from the **BackStage** by selecting **Settings > User > Preferences**.

Design File Settings

Design File Settings are accessed from the **BackStage** by selecting **Settings > File > Design File Settings**.

ProjectWise

ProjectWise, Bentley's engineering project collaboration software, is used to manage, share, and distribute engineering project content. ProjectWise integrates with Bentley applications and other products including Autodesk software and Microsoft Office.

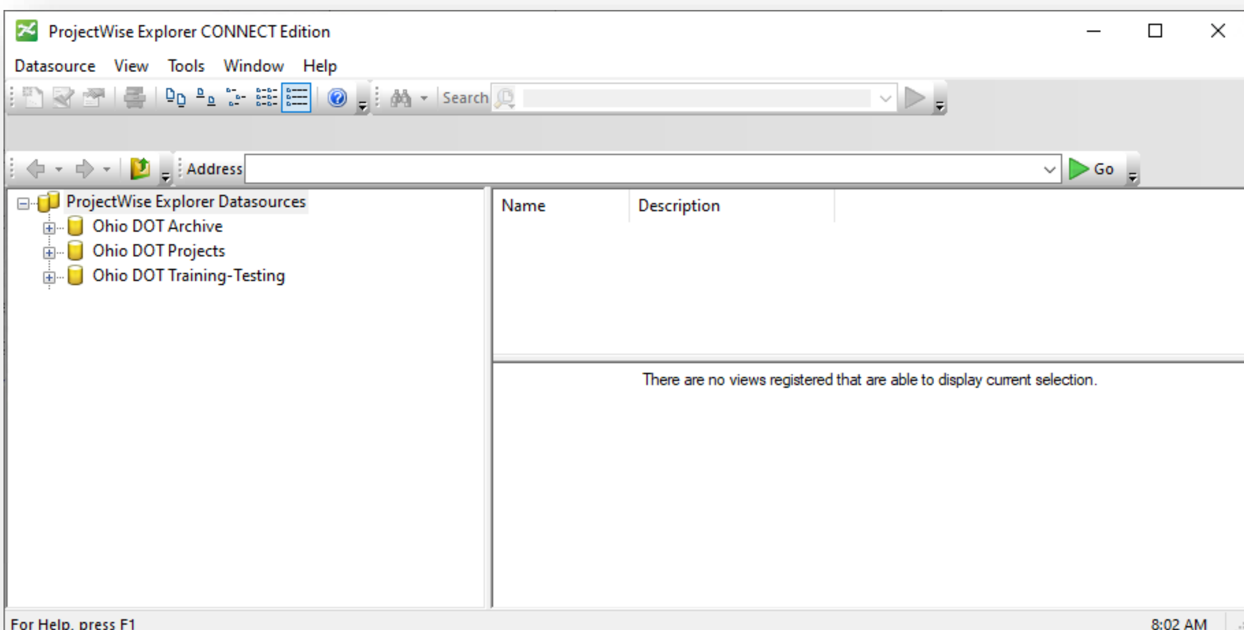
With the migration to OpenRoads Designer (ORD), all internal projects started using ORD will be housed in ProjectWise.

ProjectWise Explorer

ProjectWise Explorer is used to access project information stored in ProjectWise. The application is accessed using the desktop shortcut shown below.

Data Sources

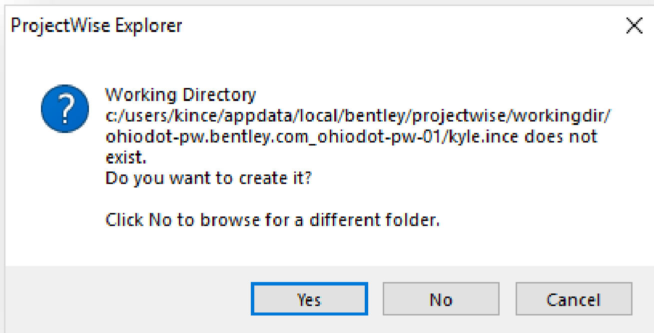
ODOT's ProjectWise environment is configured with three main datasources as described below:



- Ohio DOT Archive
    - Read Only repository for completed plan sets.
- Ohio DOT Projects
    - Active projects directory (directly replacing the "idrive").
    - All ODOT Connect projects will be housed here.
- Ohio DOT Training-Testing
    - Sandbox area for training and testing which mirrors the active projects directory.

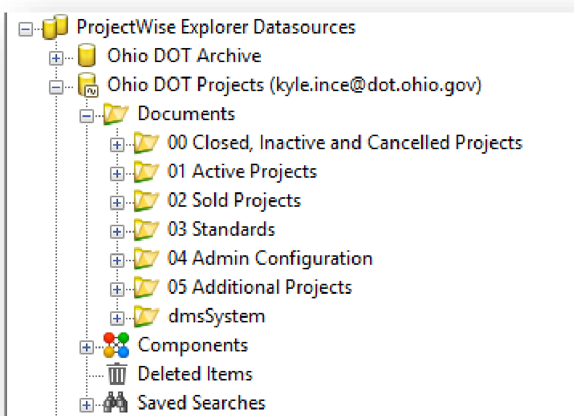
Select the datasource from the ProjectWise Explorer

- The first time you access ProjectWise on any PC, you will see the create working directory dialog.



- ProjectWise copies the working files to your local hard drive, called the Working Directory. It contains temporary copies of the document files that you checked or copied out from a ProjectWise server.

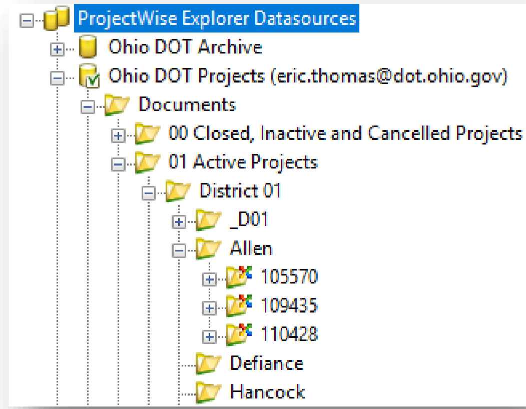
Ohio DOT Projects



- 00 Closed, Inactive and Cancelled Projects*
  - Archive of project data that is completed or cancelled.
- 01 Active Projects
  - Active PID projects (replacement of the project data folder).
- 02 Sold Projects
  - Project Data location when project is in construction.
- 03 Standards
  - CADD Standards, Help Documents, Templates, etc.
- 04 Admin Configuration
  - Access control models and other ProjectWise configuration documents.
- 05 Additional Projects
  - Area where projects without a PID may be worked on using OHDOT standards.

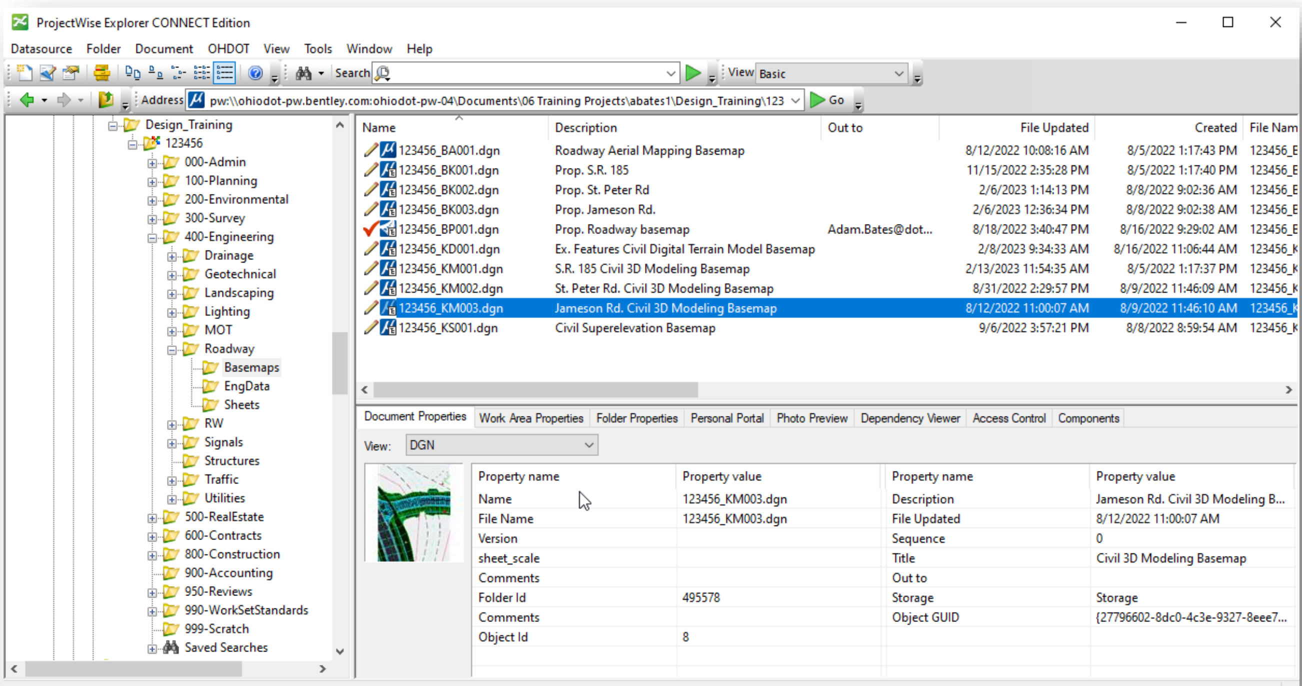
Active projects are organized by District, County, and PID as shown in the image to the right.

- The _D## folder is used for District-wide projects.



Opening Files with ProjectWise

Files are opened by double-clicking on the file name in the ***ProjectWise Explorer*** dialog. The file is opened with the associated program. Right-click on a file name and choose **Open With** to select the default program for a given file type.



The icons to the left of the file name are described below:

	File has Read/Write access • The user can check out the file for revisions. While the file is checked out locally, the file on the server is locked to other users.
	File has Read Only access • This symbol means the user can read the file but cannot edit the file. The user can view the file, but it remains available (and unlocked) to other users.
	File is Checked Out by you (others see the Locked symbol) • A copy of the file has been sent to your local working directory for editing. The file remains locked to other users until you check the file back in.
	File is Locked • The file is being accessed by another user (Checked Out or Exported) and is not available. Users can still open the file as Read Only.
	File has been Exported by you (others see the Locked symbol) • Files are typically exported to a local folder and then worked on outside of ProjectWise. When finished, the files are then Imported back into ProjectWise.
	File is in Final Status • No changes are allowed. This is an archived file status.

When a file is opened from *ProjectWise Explorer*, the file is copied to your local computer and is locked on the server. Other users can view the file but cannot edit the file while it is checked out by another user.

Changes to the file are not reflected on the server copy until one of the following actions is preformed:

1. The file is closed and checked back into the server using the **Check In** option.
2. The server copy is updated by choosing **File> Update Server Copy** in ORD.

Opening Multiple MicroStation Design Files

Double-clicking on a file name opens the selected file. By default, ProjectWise does not allow the user to open multiple design files simultaneously. If a MicroStation design file is currently opened, the software forces the user to close the current design file.

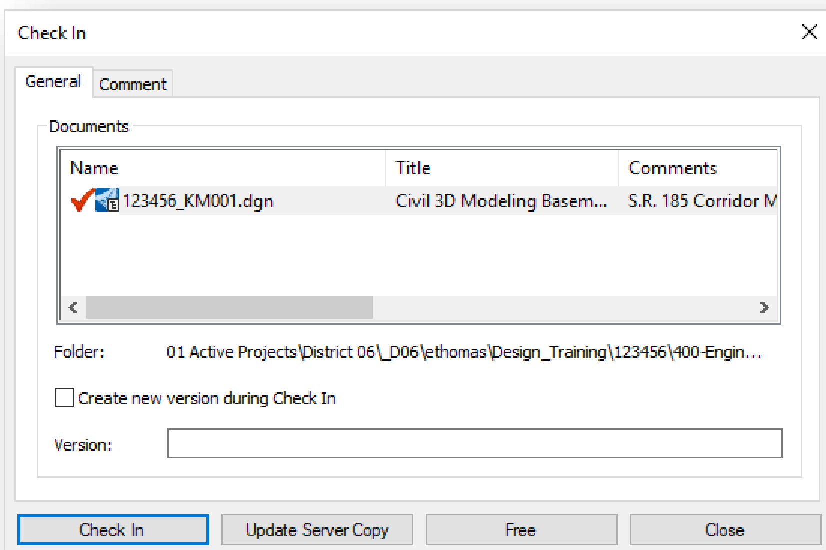
Multiple files can be opened with ProjectWise as follows:

- To open a second file for write access (assuming the file is not locked by another user), hold down the Shift key while double-clicking a file name.
- To open a second file for read access, right-click the file and choose **Open as Read-Only.**

Closing Files

After completing work on a document, in a supported application, ProjectWise will provide options for checking the document back into the server.

- Check In
  - Upload any document changes from your local copy to ProjectWise and mark the file available to other users.
- Update Server Copy
  - Updates any changes to ProjectWise but keeps the file checked out. (Also, under **File > Update Server Copy.**)
- Free
  - Checks document in but ignores any local document changes. This command also purges the document from your local working directory.
- Close
  - Dismisses the Check In window. Changes to the document are retained on your local copy but are not uploaded to ProjectWise and are not available for other users. The document remains checked out to you.



The OHDOT WorkSpace

This section presents a brief introduction to the OHDOTCEv02 WorkSpace. More information can be found in the **OHDOT CADD WorkSpace Basics** training guide, available in the following location within ProjectWise:

> **03 Standards\CADD Standards\CONNECT_Config\WorkSpaces\OHDOTCEv02\Standards\OHDOT Utilities\Training\OHDOTCADDWorkSpaceBasics**

OpenRoads uses a **WorkSpace** to define the CADD Standards for a WorkSet. A **Workspace** is a custom ORD environment or configuration. By selecting a workspace, you are customizing ORD for a specific discipline, project, or task. The workspace contains pointers to the CADD standards, DGN libraries, cell libraries, seed files, symbology resources, etc. necessary for the project.

ODOT maintains a custom WorkSpace for Open Roads Designer as described below. The CADD standards for ORD are maintained by the Office of CADD and Mapping Services.

When you access an ODOT project through ProjectWise Explorer, you enter a managed workspace environment. The CADD Standards are defined for ODOT projects and do not need to be specifically selected by the user.

ODOT's CADD Standards for Bentley CONNECT products are housed within ProjectWise in the following location:

Ohio DOT Projects > 03 Standards > CADD Standards > CONNECT_Config > WorkSpaces

Two WorkSpaces that have been configured for ODOT Standards are available, **OHDOTCEv01** and **OHDOTCEv02**, as described below.

The OHDOTCEv01 WorkSpace (Retired 3/6/23)

The OHDOTCEv01 WorkSpace has been superseded by the OHDOTCEv02 WorkSpace. This WorkSpace is provided to allow users to complete projects that were started with this WorkSpace. New projects are all configured to use the OHDOTCEv02 WorkSpace. OHDOTCEv01 will no longer be updated.

The OHDOTCEv02 WorkSpace

With the September 24, 2021, update to the ODOT CADD Standards, ODOT has released the OHDOTCEv02 WorkSpace. This workspace differs from the previous version primarily with the fonts that are used for plan production.

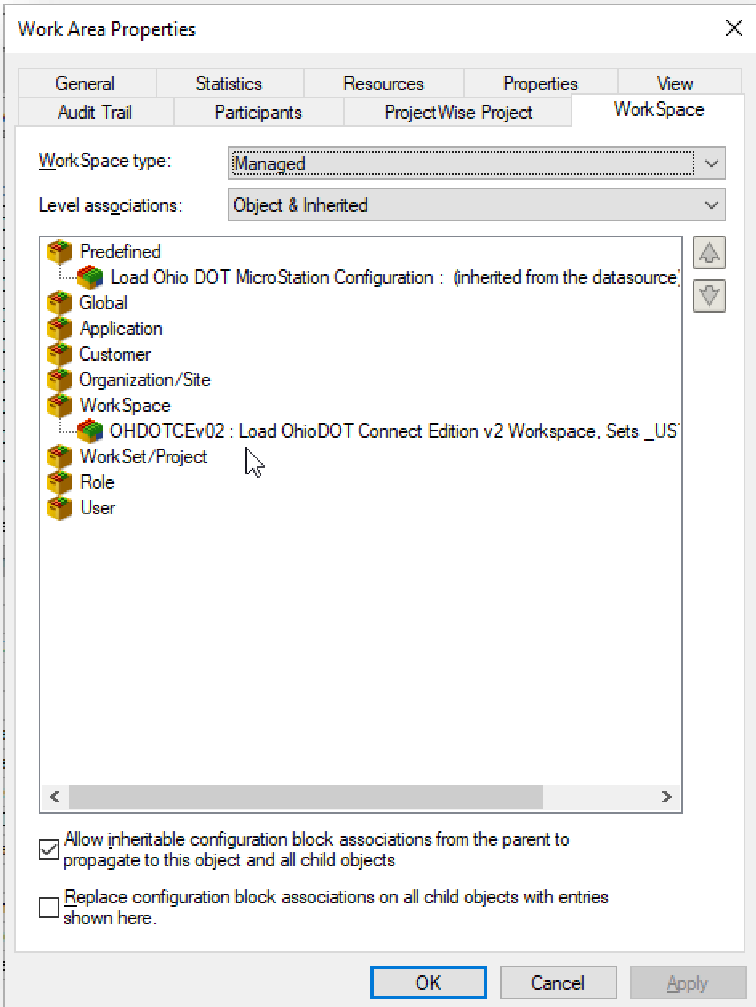
In the previous version, font Aerial was used for the annotation of both existing and proposed items. This made it difficult to differentiate between existing and proposed annotations. With the OHDOTCEv02 workspace, ODOT is moving to using font Calibri Italic for proposed items and font Calibri Light for existing items.

The OHDOTCEv02 WorkSpace also removes the old Drainage and Utilities dgn library (.dgnlib) from the standards.

Projects started with the OHDOTCEv01 WorkSpace should normally be completed using that WorkSpace. As new projects are started, the OHDOTCEv02 WorkSpace should be used.

When projects are created, the WorkSet is associated with the WorkSpace. ODOT's users do not need to do anything to select the appropriate WorkSpace for a project. The associated WorkSpace is automatically loaded when a design file is opened.

The **WorkSpace** that is assigned to a **WorkSet** can be reviewed in ProjectWise by right-clicking on the WorkSet folder name and choosing the **Properties** option.



From the ***WorkSet Area Properties*** dialog, select the **WorkSpace** tab to review the assigned WorkSpace as shown at left.

In the ODOT environment, users are not permitted to change the WorkSpace that is assigned to a WorkSet. Contact the ProjectWise team if the WorkSpace needs to be changed for a WorkSet.

The assigned WorkSpace can also be reviewed within a design file be selecting **File > Settings > Configuration > About Configuration**.

Exercises in this manual use the OHDOTCEv02 WorkSpace.

WorkSets

Projects in ORD are known as **WorkSets**. A WorkSet is used to house the files for each individual ODOT design project. WorkSets serve the following purpose:

- Used to store all related project information.
- Define the folder structure for the project.
- House project specific CADD Standards.

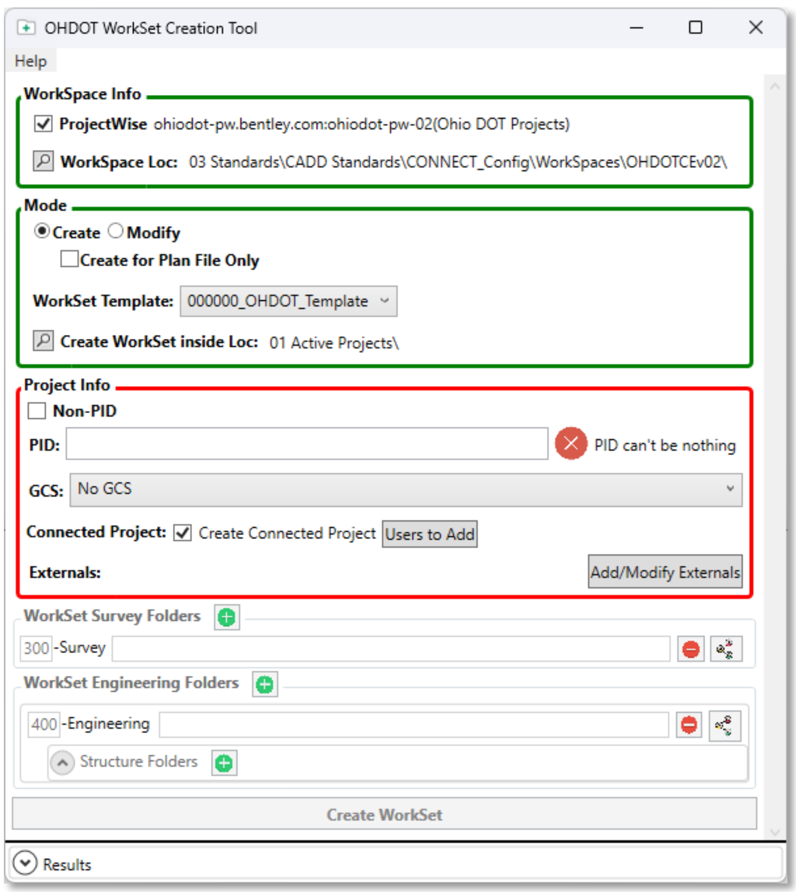
WorkSet Creation inside the ProjectWise Environment

Projects that have been assigned an ODOT Project Identification Number (PID) are created in the **Ohio DOT Projects > 01 Active Projects**.

Projects that do not yet have a PID number assigned but may be assigned a PID number at a future date are created in the **Ohio DOT Projects > 05 Additional Projects** data source.

In either case, the District ProjectWise Administrator is tasked with creating new projects using a custom application. The application is available in the OHDOT CADD Standards in the following location:

...\OHDOT\Standards\Applications\OHDOTCreateWorkSet.exe



See the application's support documentation for more information.

Geographic Coordinate Systems

OpenRoads Designer (ORD) contains Geo-Coordination features which allow users to specify the position of the design contents on the earth´s surface. Once that position is established, the design can be easily coordinated with other data for which the geographic location is known. A library of predefined Geographic Coordinate Systems (GCS) is available in ORD. Additionally, custom coordinate systems can be defined for projects that have been mapped to ground coordinates by defining a custom coordinate system.

When a GCS is initially selected, you are simply defining the coordinate system where the data resides. Choosing a GCS when one has not been previously defined does not re-project existing data in the design file to the selected GCS. The content of the design file will be re-projected when changing from one GCS to another.

Once a GCS is defined, ORD understands the geographic location of your design and provides additional capabilities such as:

- Referencing other geo-located designs and raster data.
- Displaying geographic latitude and longitude.
- Entering latitude and longitude data.
- Interfacing with a Global Positioning System device to correlate your physical position with the design on a mobile computer.

NOTE: The intent of the GCS is to define the location of the data for easy integration with other georeferenced data. It is not intended to be used “on the fly” to translate the data from grid to ground, or vice versa. Once defined, the survey data should be left in the defined coordinate system.

When a new WorkSet is created in ProjectWise, the ODOT application that is used to create projects provides the option to identify the GCS for the project. The project can be defined to use the appropriate State Plane coordinate zone, or the coordinate system can be left undefined in preparation for defining a custom coordinate system.

Before any work begins on the project, it is a best practice to discuss the coordinate system requirements with the District Survey Operations manager, and then to review the coordinate system assigned to the seed files for the project to ensure it has been correctly defined.

Grid Coordinates:

- If the project is mapped to grid coordinates, the coordinate system for the project is defined using one of the standard coordinate projection systems for Ohio.
- The coordinate system is assigned to the seed files for the project when the project is created.

Ground Coordinates:

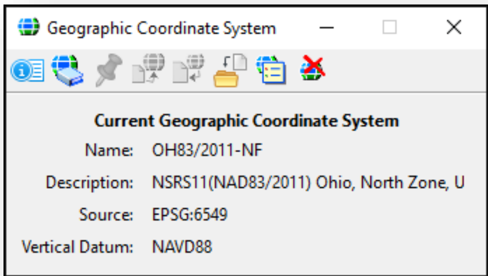
- If the project is mapped to ground coordinates, a custom Geographic Coordinate System (GCS) can be defined with the combined scale factor for the project. This step is not required, but it does provide advantages for users of the project data.
- If a custom coordinate system is used, the project is created with No GCS assigned. A DTY file is copied into the project that can later be customized by the Surveyor with the combined scale factor for the project.

Reviewing the current GCS

The seed files for the WorkSet are defined for the appropriate coordinate system. Open each seed file to ensure that the coordinate system is correctly defined. See files for the WorkSet are in the following folder:

990-WorkSet Standards\Seed

The GCS is defined by selecting **Drawing > Utilities > Geographic > Coordinate System** to access the Geographic Coordinate System dialog shown below.

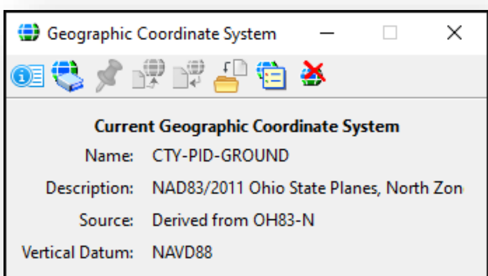


The current coordinate system is displayed on the dialog as shown at left. If no coordinate system has been defined, the **Name** will display "**None**".

If the correct system is assigned for the project, close the file, and continue reviewing the remaining seed files for the WorkSet.

If the coordinate system is undefined, or set incorrectly, it is best to coordinate with the District Survey Operations Manager before making any changes.

If a custom coordinate system has been defined by the Survey personnel, it will be named using the County and PID Number with the word "Ground" in the name like the example show below.



The process to define a custom coordinate system for projects mapped to ground coordinates is detailed in ODOT's training material for OpenRoads Survey.

WorkSet Folder Structure

All ODOT projects use a standard folder structure for the WorkSet. An Excel file detaining the intended usage for each folder can be found in the ODOT Standards in the following location:

- **Ohio DOT Projects > 03 Standards > CADD Standards > CONNECT_Config > WorkSpaces > OHDOTCEv02 > Standards\OHDOT Utilities\Documentation\CONNECT_Project_Folders.xlsx**

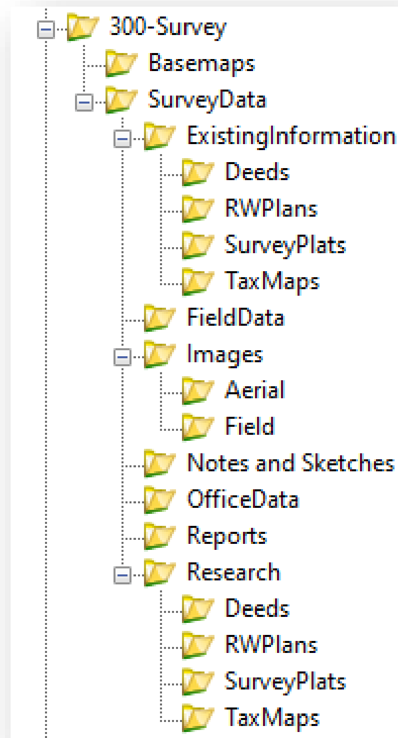
A summary of the folders used for ORD design and drafting is included below.

300-Survey

- 300-Survey\Basemaps\
  Used to store survey and existing basemap design files.
    DGN Files:
    - BA - Aerial Mapping
    - BC - Aerial and Ground Combined
    - FB - Ground Survey Basemap
        - No Alignments
        - No Right-of-Way
        - No Parcels
        - No Exceptions!
    - FD - New file for field terrain models
    - PC - New file for Point Cloud data

    Retired Files:

    - BE - No longer used for ORD Projects

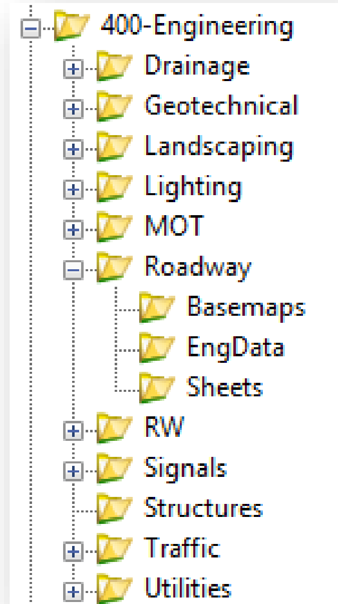


Notes:

- By default, ODOT's Roadway personnel have read-access to the 300-Survey folder and sub-folders.
- Designers will reference the Survey information. Any changes that are required by the design personnel must be coordinated with Survey.
- New folders can only be created at the bottom level folders.

400-Engineering

Each discipline specific folder in the 400-Engineering folder has three sub-folders as shown at right.



Alignments are stored in the following file:

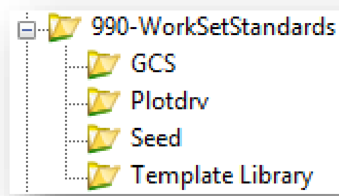
- BK – Geometry Basemap, located in the (400-Engineering\Roadway\Basemaps folder).
- One alignment per file. Use multiple files for multiple alignments. Example: 123456_BK001.dgn, 123456_BK002.dgn, etc.

Notes:

- ODOT's Surveyors have Write access to the 400-Engineering folder and sub-folders.
- ODOT doesn't recommend using container files for alignments. Example: 123456_BK000.dgn, 123456_BK100.dgn, etc.

990-WorkSetStandards

CADD Standards specific to a project are stored in the 990-WorkSet Standards folder. When a new project is created in ProjectWise, WorkSet specific CADD Standards are placed in these folders as detailed below:



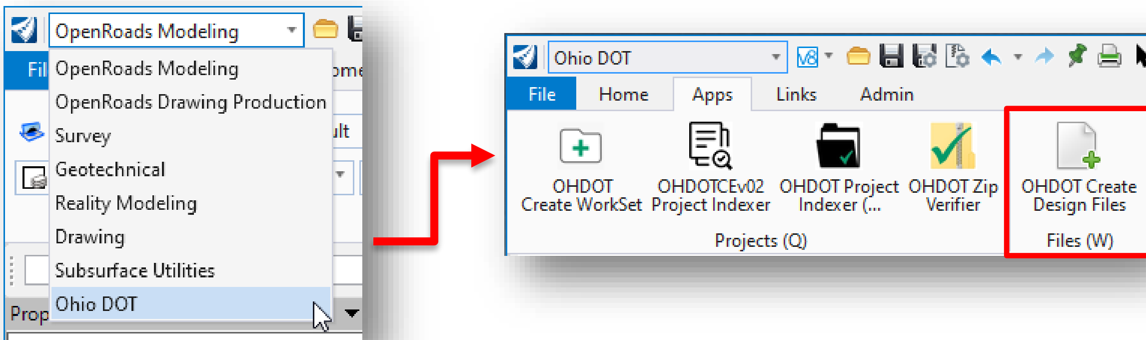
- GCS
  - This folder contains the DTY file defining the Geographic Coordinate System in the case where a custom scale factor is defined for the project. The DTY file is copied from the OHDOT CADD Standards and is named "######_Custom.dty" using the PID number for the project.
- Plotdrv
  - This folder contains the PSET file for the WorkSet. The file is copied from the OHDOT CADD Standards into this folder. The copied file is renamed "######.pset" using the PID number.
- Seed
  - Seed files are copied from the OHDOT CADD Standards and renamed using the PID number for the project.
  - These seed files are defined using the appropriate Geographic Coordinate System when the WorkSet is created.
  - The Design seed files are renamed with the ODOT PID Number.
  - The Drawing and Sheet seed files are not renamed with the PID number. It is necessary to maintain generic names for these two seed files for the ORD sheet clipping process.
- Template Library
  - The OHDOT_Templates.itl file is copied from the OHDOT CADD Standards into this folder. The copied file is renamed "######_templates.itl" using the PID number.

WorkSet Design File Names

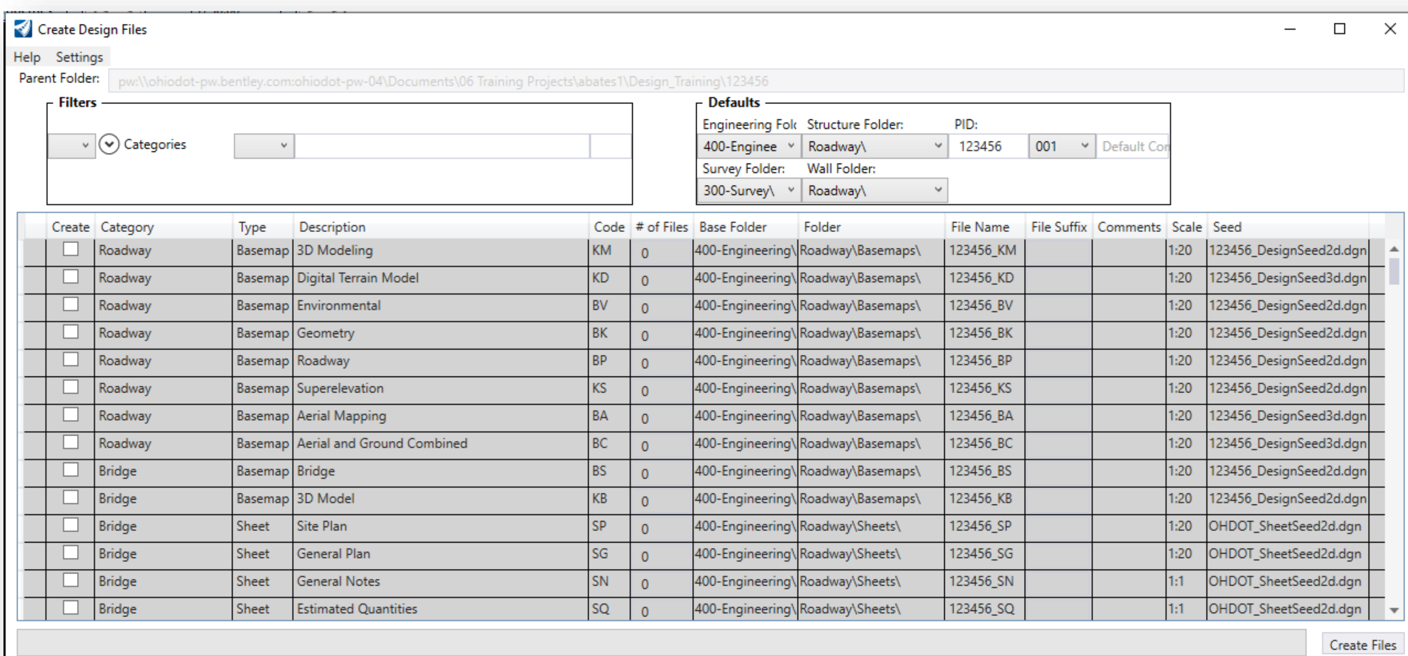
Design files are named using the format defined in the **Location & Design Manual, Volume 3 - Highway Plans**, **Section 1204.3.4.**

Creating MicroStation DGN Files

New Design Files are created within ORD by use of the OHDOT Create Design Files application. The application is accessed from the **Ohio DOT** Workflow by selecting the **OHDOT Create Design Files** icon as shown below.



The dialog is shown below. See the application help for more informati



Section 101 WorkSet Exercises

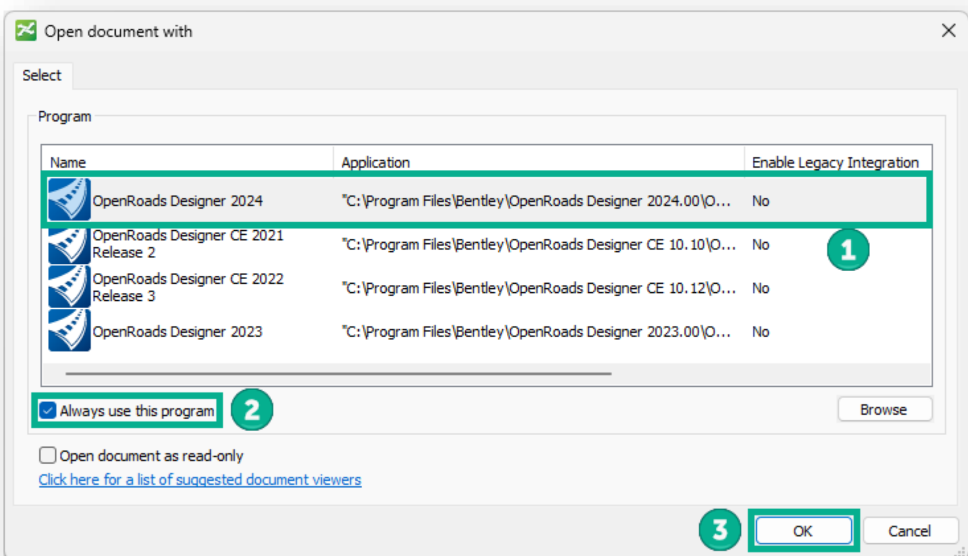
Exercise

In this exercise we will access ProjectWise Explorer to open a design file, explore the ORD ribbon interface, and create a new design file for the WorkSet.

Part 1: Accessing the Training Workset and Creating Design Files

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- From the training folder, browse to the **300-Survey\Basemaps** folder
- Right-Click on the file **123456_FB001.dgn** and choose the **Open With…** option to open the file with OpenRoads Designer XXXX. Toggle the **Always use the program** option **ON**



- Select **OK** to open the file.

Let's review the Geographic Coordinate System (GCS) assignment. The seed files for all ODOT projects should have a GCS defined.

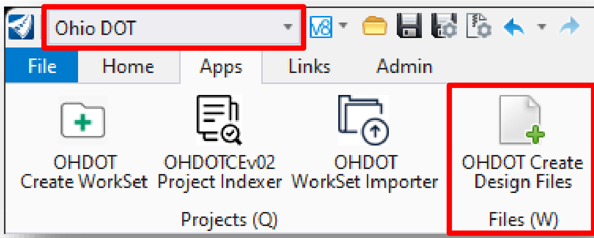
After opening the file, take the following steps:

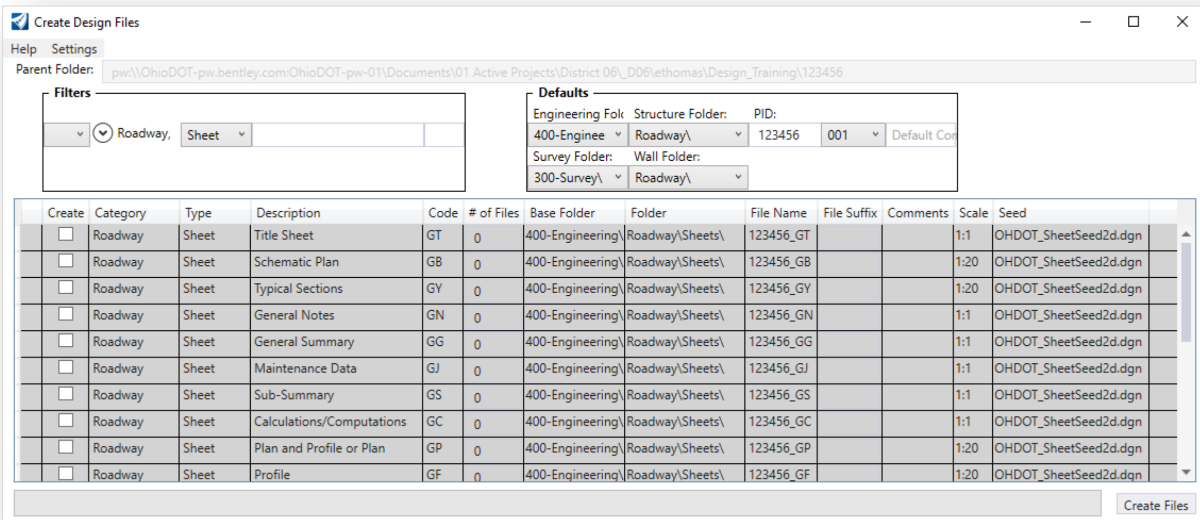
- Select the **Drawing** Workflow from the upper left of the ORD interface
- Select the **Coordinate System** icon from the **Utilities** tab to review the assigned coordinate system value.
    - What is the name of the currently assigned Geographic Coordinate System?

      ____
    - There is no need to change the coordinate system parameters. If the coordinate system has not been defined, contact the survey personnel.
    - It is good practice to make sure the GCS has also been defined for the Seed3d and the Drawing Seed files. Open those seed files to verify the GCS has been defined.
    - As each file is opened, be sure to check the files back into ProjectWise using the **Check In** option when prompted. Alternately, since no edits were made to the file, the **Free** option can be used to release the file without saving any changes.

Next, we will create some design files for the WorkSet.

- Select the **Ohio DOT** Workflow from the top left of the ORD interface.
- From the **Apps** tab, select the **OHDOT Create Design Files** icon





The application is opened as shown below:

- Create the following DGN files:
  - Set the Filters to **Roadway** and **Sheet** as shown above.
  - Create the following files:
    - Title Sheet
    - Schematic Plan
    - Typical Sections

  Which Seed File is used to create these design files? ______________________________

  Which folder are the files created in? ______________________________

  What should be entered for the **Comments** field for each file?

  Title Sheet: ______________________________

  Schematic Plan: ______________________________

  Typical Sections: ______________________________

  What should be entered for the **Scale** field for each file?

  Title Sheet: ______________________________

  Schematic Plan: ______________________________

  Typical Sections: ______________________________

- After creating the design files, close the **Create Design File** application

Part 2: Exploring the ORD Interface

- ***Without*** using the **Search Ribbon** key-in (unless you absolutely cannot find the command), locate the following commands and write down the location where you found the command:

  Place Smartline ______________________________________
  Place Text ______________________________________
  Edit Text ______________________________________
  Annotation Scale ______________________________________
  OpenRoads Line Between Points ______________________________________
  OpenRoads Spiral-Arc-Spiral ______________________________________
  OpenRoads Create Template ______________________________________
  OpenRoads Create Corridor ______________________________________

- What is the **Geographic Coordinate System** that is assigned to the file? ____________________
  How was the GCS assigned to the file? ______________________________________

- In which folder are the Seed files for the WorkSet located?

  ____

- In which folder is the OpenRoads Template Library for the WorkSet located?

  ____

- What is the name of the **User Preferences** file? ______________________________________

This completes this exercise.

- Exit the current file and check it back in to ProjectWise

Section 200 Working with ORD Survey Files

Survey Basemaps

Survey mapping processed with the OpenRoads Survey tools is written directly to a MicroStation design file without the use of any external databases. This "field book" (**FB**) design file is in the following folder within the ODOT WorkSet folder structure:

..\PID\300-Survey\Basemaps

The Survey information is processed in a 3D design file. Designers will work in 2D files and will attach the 3D survey information as a reference attachment.

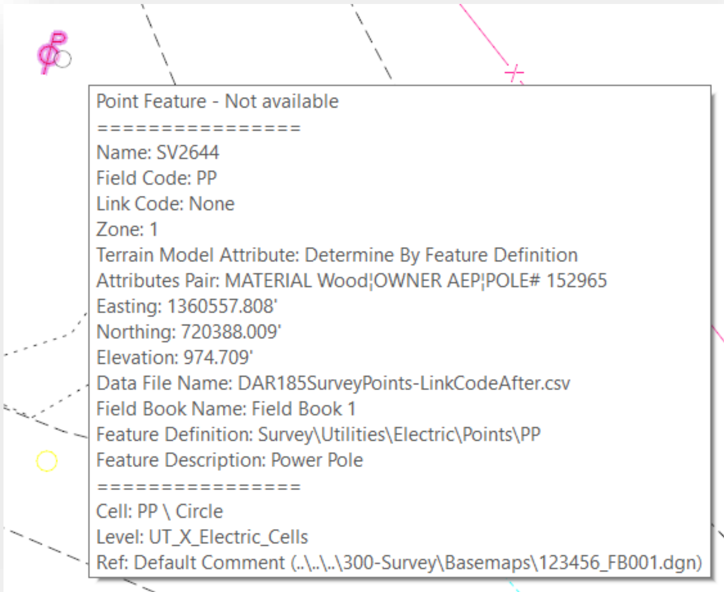
An existing ground terrain model is typically generated from the survey information. The terrain model is typically contained in the **FB** design file in cases where the terrain is comprised of only the ground survey information. In cases where the survey information is combined with other terrain data, such as LiDAR data, the combined terrain model is usually stored in a separate design file using the "**FD**" two-character code in the file name. Contact the survey personnel for clarification on which design file contains the terrain model that is to be used for the project, particularly in cases where a complex terrain model is generated from multiple sources in a separate **FD** design file.

Survey Data Integrity

Protecting the integrity of the survey information has always been a concern for ODOT's Survey personnel. The accuracy of the Survey information is the responsibility of the surveyor and therefore the original survey information should not be edited by design personnel. ***Under no circumstances should design personnel edit the survey data contained in the Survey field book FB design file***. If changes need to be made to the survey files, these changes should be coordinated with the originator of the survey data.

Survey Point Features

When survey data is processed, each field shot is stored as a **Point Feature** in a **Field Book** within the survey field book FB design file. For example, the field code "PP" is used to designate a power pole. When the survey data is processed, the power pole code is plotted in the design file using an ODOT standard cell element as shown below:

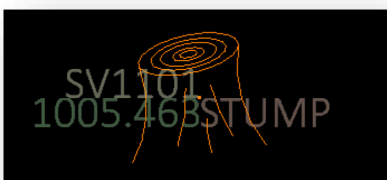


When the survey field book is attached as a reference, the data for an individual point can be reviewed as follows:

From the **Home** tab, choose the **Element Selection** command. Float the cursor over a referenced survey point and allow the cursor to rest over the element.

A pop-up dialog will appear showing the survey information for the point as shown at left.

With the December 2020 update to the OHDOT CADD Standards, macros have been added to annotate points like GEOPAK Survey. An example of the annotation placed by a macro for a point is shown below.



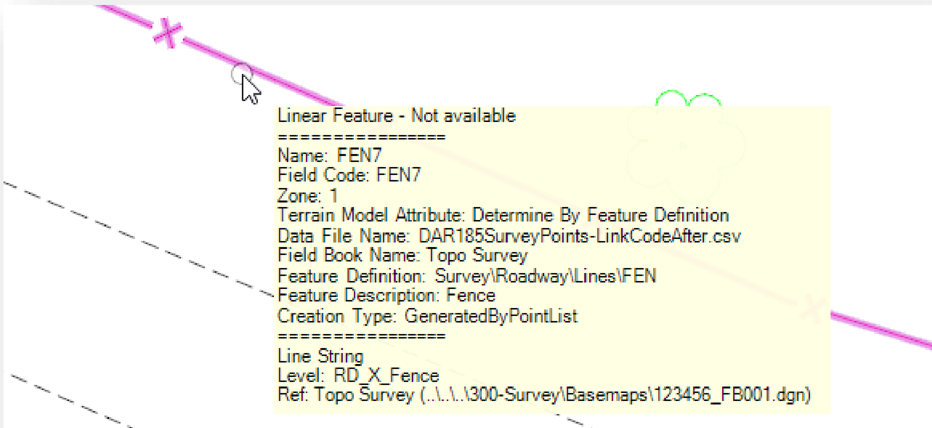
These annotations are placed as text elements in the file on the levels listed below.

Annotation	MicroStation Level
Point Name	SV_X_Text_Name
Point Elevation	SV_X_Text_Elevation
Field Code	SV_X_Text_Comment

The display size of the annotations is based on the **Annotation Scale** parameter.

Linear Features

A series of survey points representing curvilinear features can be connected as a **Linear Feature** as shown in the example below.



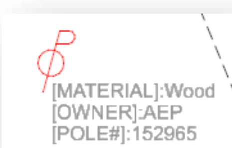
With the **Element Selection** tool active, floating your cursor over the referenced linear feature will open a pop-up menu with information about the feature line as shown in the example above.

Field Attributes

Many of the ODOT field codes allow for the use of attributes to store additional information with a specific survey shot. For example, the feature code "PP", used to specify a Power Pole, has the following attributes:

Material
Owner
Pole#

The attributes may or may not be plotted in the file depending on whether the survey personnel have elected to annotate the model. Annotations are not automatically placed in the design file as they were with previous software.



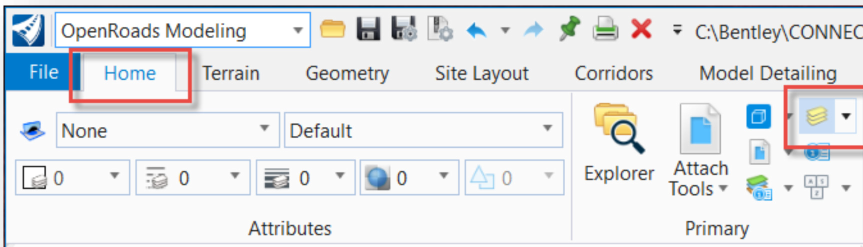
In the example at left, the annotation is placed in the file on the level SV_X_Text. These annotations are intended for information only. See the Annotating Survey Elements section for more information.

Attributes can also be reviewed in the pop-up dialog that is displayed when floating the cursor over a point as described on the previous page.

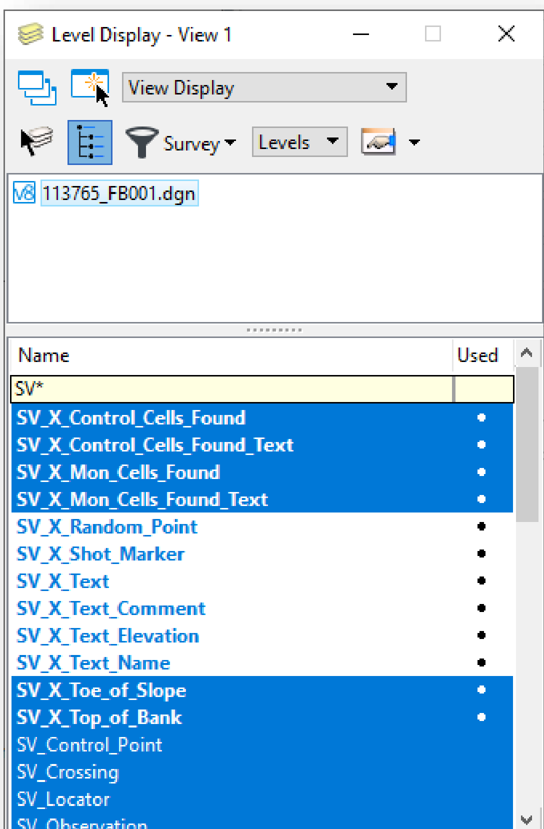
Design File Levels for Survey Data

The OHDOT CADD Standards include several MicroStation levels that are specific to survey information. These levels are grouped in the level library using the prefix "SV".

The levels can be turned on/off using the ***Level Display*** dialog. The ***Level Display*** dialog can be opened in all the various **Workflows** by selecting icon in the **Primary** group of the **Home** tab as shown below.



The ***Level Display*** dialog includes the ability to filter the dialog to only show specific levels. By selecting the Survey filter, the dialog will show only the survey levels.



The following levels are typically not shown on the plan and should normally be turned off after the survey basemap has been attached as a reference.

- PV_X_COP (center of pavement)
- SV_X_Break_Lines
- SV_X_Random_Point
- SV_X_Shot_Marker
- SV_X_Text_Comment
- SV_X_Text_Elevation
- SV_X_Text_Name
- SV_X_Text

Annotating Survey Elements

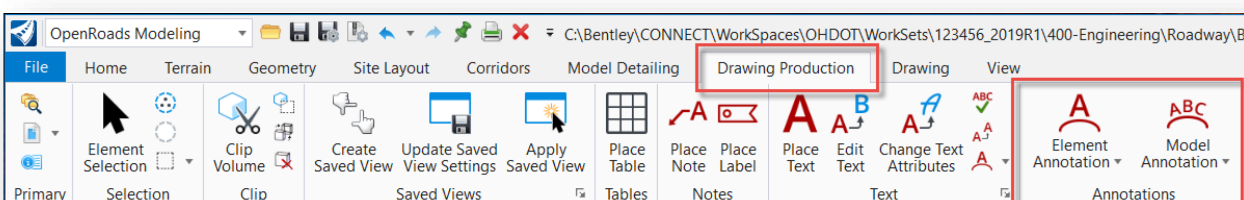
Annotation for Survey elements is not automatically placed in the FB design file when the survey data is processed as it was in PowerGEOPAK.

Survey elements can be annotated in the Field Book (FB) design file by the surveyor, or in any file that the FB design file is attached as a reference.

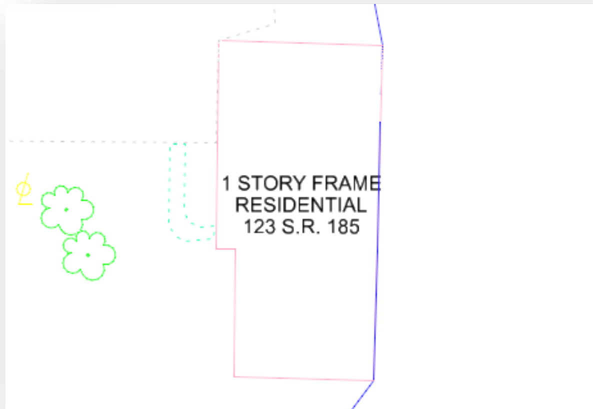
In the example below, the attributes for the building were entered by the surveyor as the data was collected in the field.

The **Feature Definition** assigned to the element controls how each Survey item is annotated. The annotation is also dependent on whether the surveyor entered any attribute data in the field as the survey data is collected.

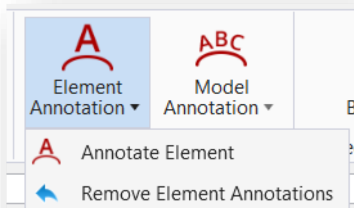
Annotation is placed in the active model by selecting an individual element, by a selection set, or for all the civil elements in the active mode by use of the annotation commands shown below:



An example of the annotation that is placed for a building using the **Element Annotation** command is shown below.



Annotations can be removed by manually deleting the text, or by use of the **Remove Element Annotation** or the **Remove Model Annotations** command.



Geographic Coordinate Systems

A Geographic Coordinate System (GCS) is used to define the position of the survey and design information on the Earth's surface. Once the position of the data has been established, the design can be easily coordinated with other data for which the geographic location is known such as Google Earth, Bing Maps, Geo-Referenced Imagery, or GIS systems.

Defining the Geographic Coordinate System is a requirement of the OHDOT CADD Standards. See the **Location & Design Manual, Volume 3 – Highway Plans**, **Section 1204.3.5** for more information.

A library of predefined GCS is available in ORD. Optionally, a custom GCS can be defined for projects that have been mapped to ground coordinates, if required.

The process to create a custom GCS for Grid-to-Ground conversions is the responsibility of ODOT's Survey personnel and is not covered in this manual. This manual will cover the basics of reviewing the GCS and create new design files using the same GCS that has been defied by the Survey personnel.

Note: The purpose of the GCS is to define the location of the data on the earth's surface. ODOT does not recommend changing the GCS to re-project the data.

WorkSet Seed Files and Geographic Coordinate Systems

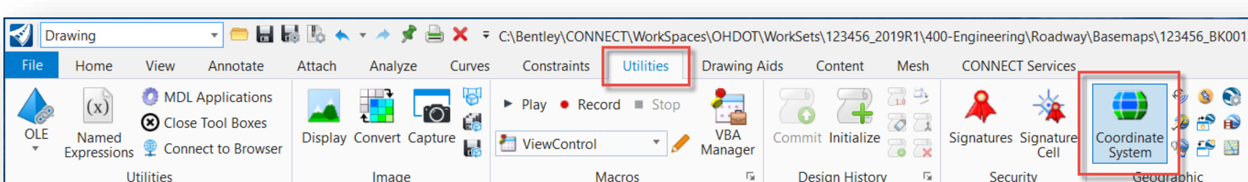
A seed file is an empty MicroStation design file that has been configured to meet ODOT requirements. The seed file is used as a template when new design files are created. The seed file selected by the user is copied and renamed to create the new file.

When a new project is created, 2D and 3D seed files are copied from the CADD Standards folder to the project's **990-WorkSetStandards\seed** folder. If necessary, the WorkSet seed files can then be modified by the survey personnel to define a custom GCS definition that is used to facilitate Grid-to-Ground projection.

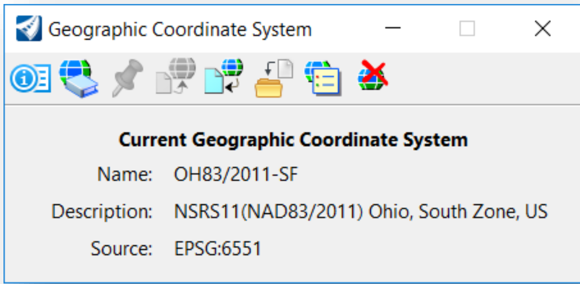
It is the responsibility of the Surveyor to define the appropriate GCS for the project data. As the designer for the project, it is your responsibility to ensure that all new design files created for the project use the same GCS as defined by the Surveyor.

Reviewing/Selecting the GCS Definition

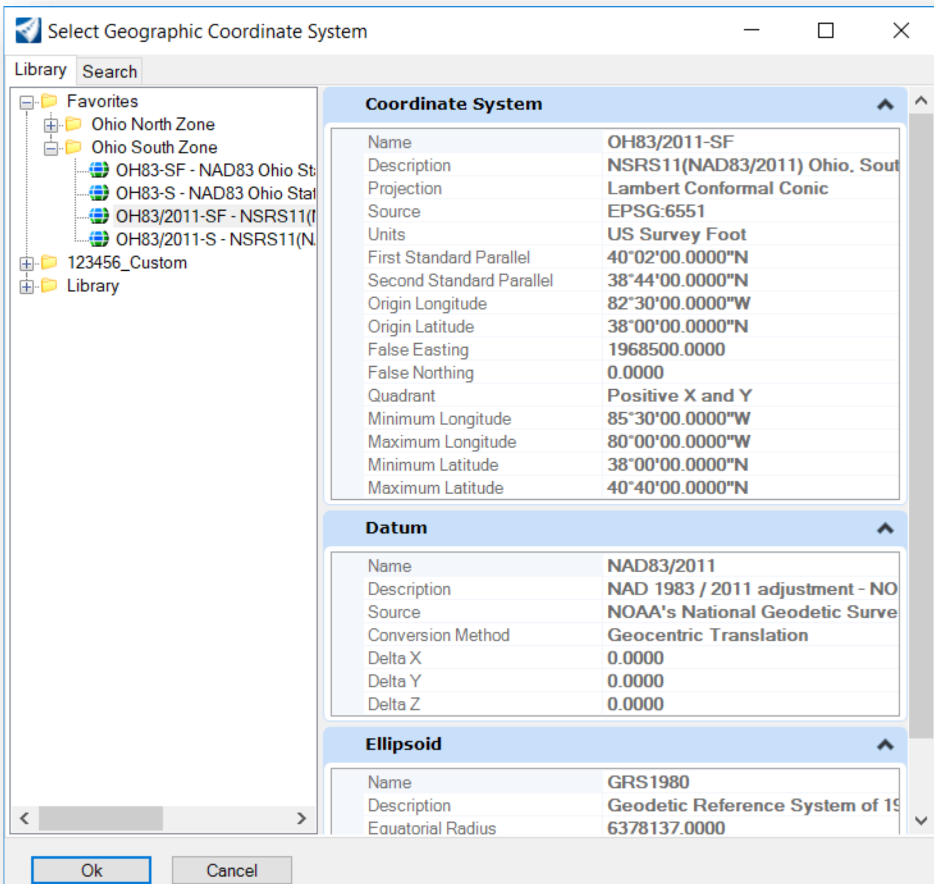
The GCS can be reviewed by selecting the **Drawing** WorkFlow, **Utilities > Coordinate System** as shown below.



The ***Geographic Coordinate System*** dialog shown below is opened.



To choose a GCS from the library, select the second icon, **From Library**. The ***Select Geographic Coordinate System*** dialog shown below is opened.

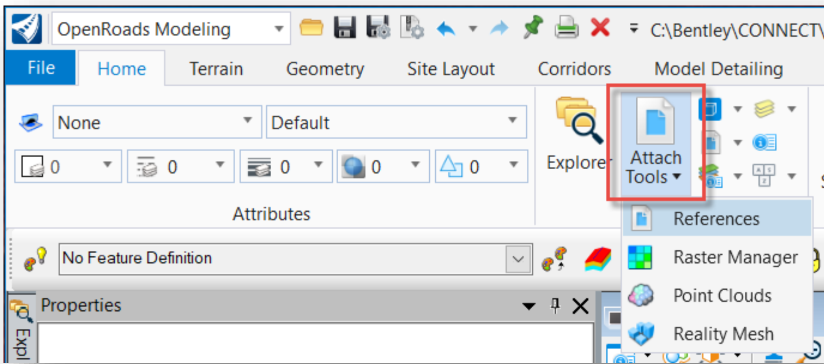


The OHDOT workspace defines the State Plane coordinate systems typically used on ODOT projects in the **Favorites** folder as shown above. Choose the appropriate GCS from the list. A different GCS can be selected by browsing the Library folder. The GCS definitions for Ohio shown below are available by selecting **Library > Projected (northing, easting) > North America > United States of America > Ohio**.

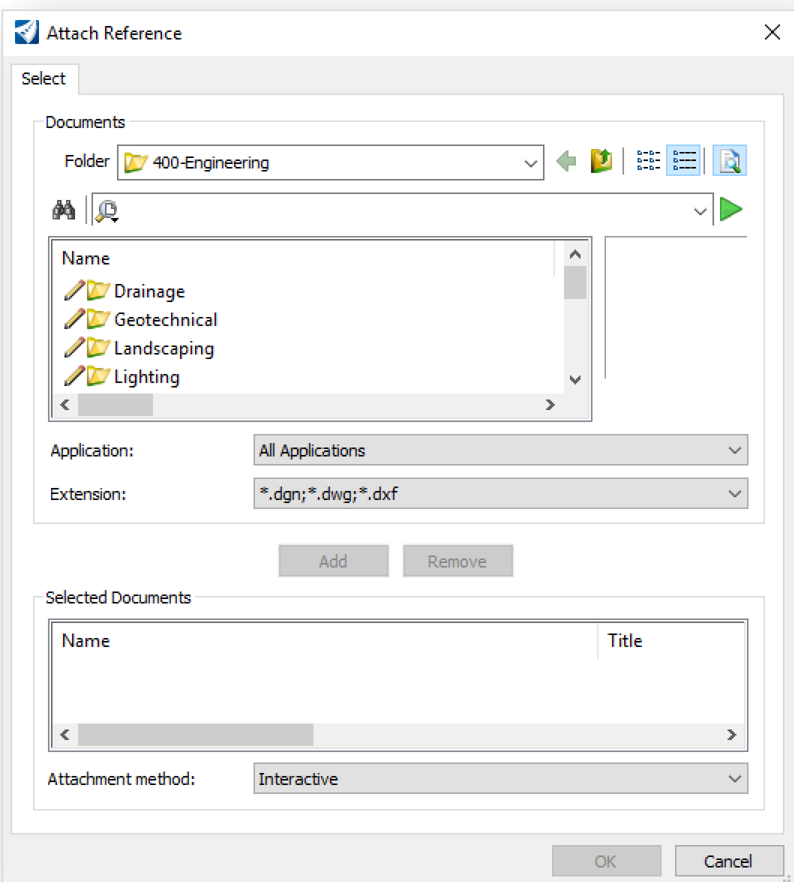
When a GCS is initially selected, you are simply telling MicroStation where the data resides. It is important to note that choosing a GCS when one has not been previously defined does not re-project existing data in the design file to the selected GCS. The data is only re-projected when changing from one GCS to another.

Reference Attachments

References are used to display the contents of another DGN file within the current ORD model. The **Home** tab in each WorkFlow contains the reference tools as shown below.



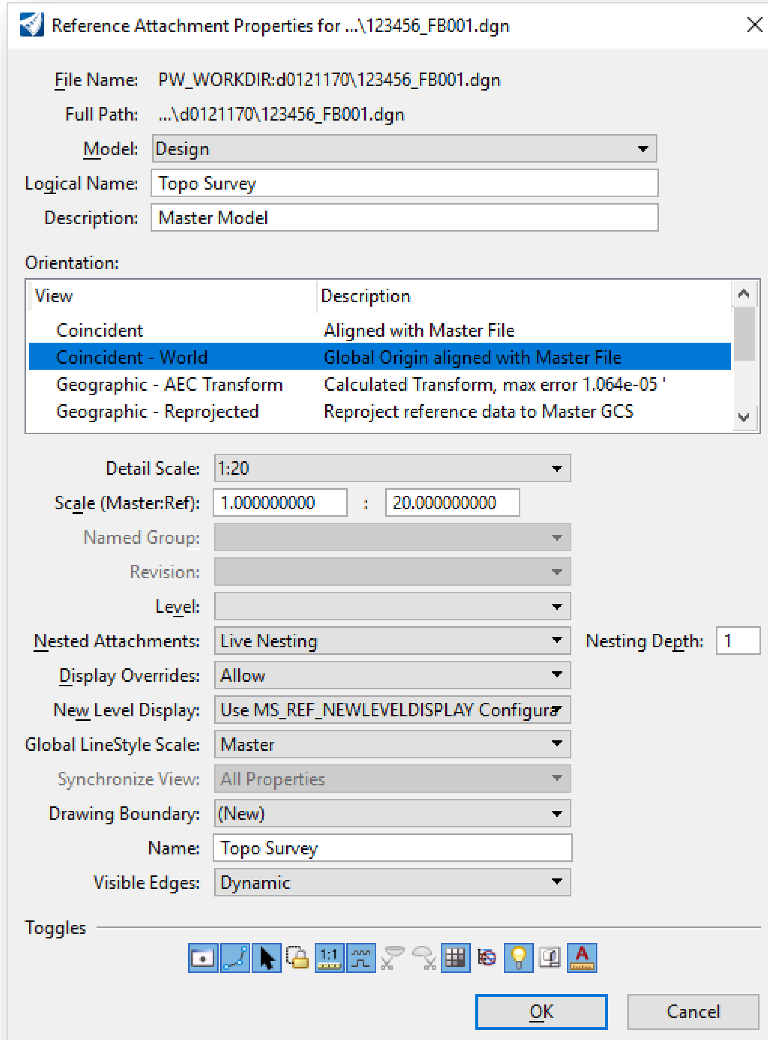
When attaching references in ProjectWise, the ***Attach Reference*** dialog shown below is opened.



The **Attachment Method** can be specified at the bottom of the dialog. If one of the methods is selected, the attachment is made without accessing the normal MicroStation dialog.

Browse to the file to be attached. Choose the Interactive method to access the MicroStation ***Reference Attachment Properties*** dialog, shown below, when the **OK** button is selected.

The various options available in the ***Reference Attachment Properties*** dialog are not covered in this manual. See the online help for more information.



When attaching the survey design file, the **Coincident -World** option is typically used since all the files in the project should be using the same GCS.

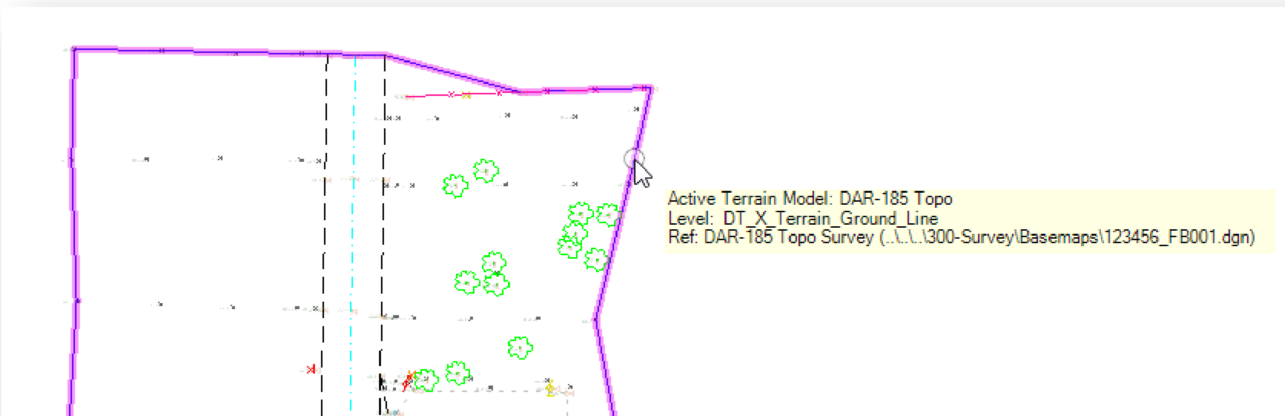
- Choose the **Geographic – Reprojected** option to re-project the referenced model to the coordinate system that is defined in the active model if the file being referenced uses a different GCS definition.
- If a GCS is not defined, choose the **Coincident – World** option.

Terrain Models

Terrain models generated by OpenRoads Survey are stored in a design file as a Terrain Model element type.

The terrain model can be accessed by design personnel by attaching the survey Field Book (FB) design file as a reference attachment. Depending on the project, the terrain model may be created in a separate Field Terrain design file (FD). Contact the survey personnel for clarification on which design file contains the terrain model that is to be used for the project, particularly in cases where a complex terrain model is generated from multiple sources and an FD file is used.

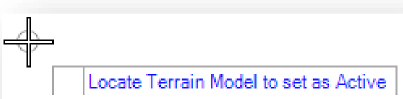
In the example below, the active file is **123456_BK001.dgn**. The terrain model is contained in the file **123456_FB001.dgn** which has been attached as a reference file.



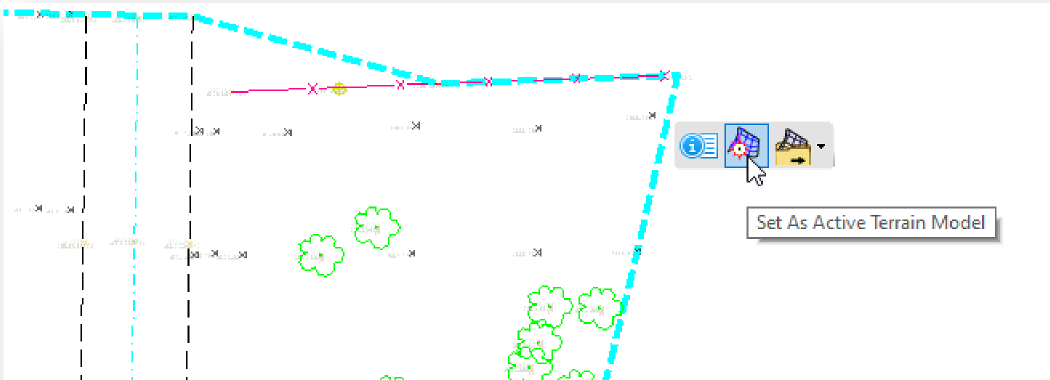
Defining the Active Terrain

After attaching the design file containing the terrain model element, it is a best practice to set the ground terrain model as the "Active Terrain". The active terrain model is used to define the existing ground for profiles and cross sections.

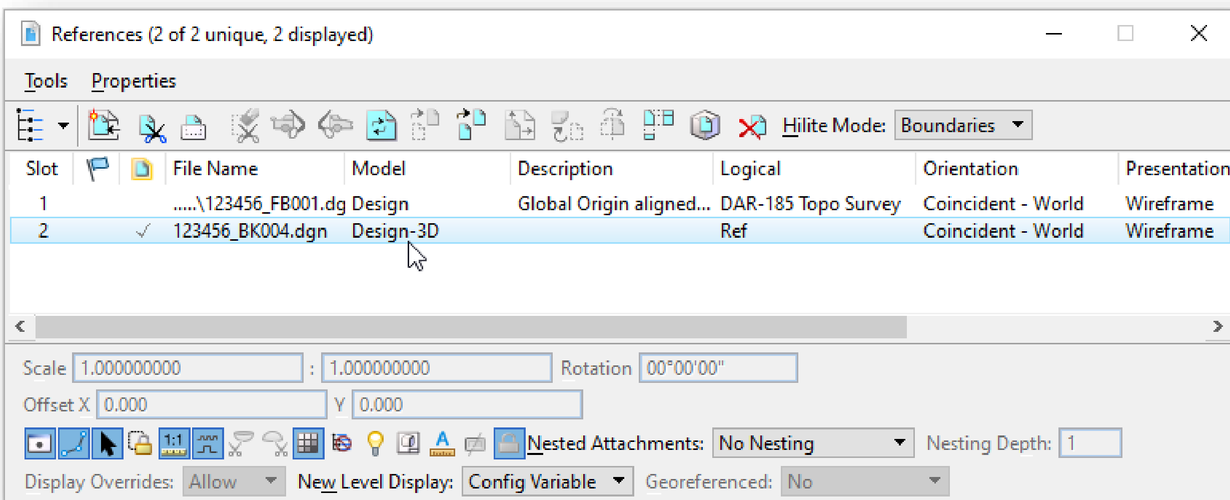
To define the active terrain, from the **Terrain** tab, choose **Active > Set Active**. Select the terrain model when prompted.



The active terrain can also be defined by selecting the **Set As Active Terrain Model** command from the pop-up menu when the terrain is selected using the **Element Selection** tool, as shown below.



When the terrain is activated, the software will automatically create a 3D model in the active design file. This 3D model is attached as a self-reference to the active file as shown below.

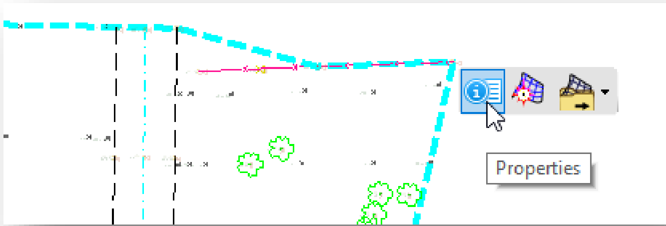


Terrain Model Display Settings

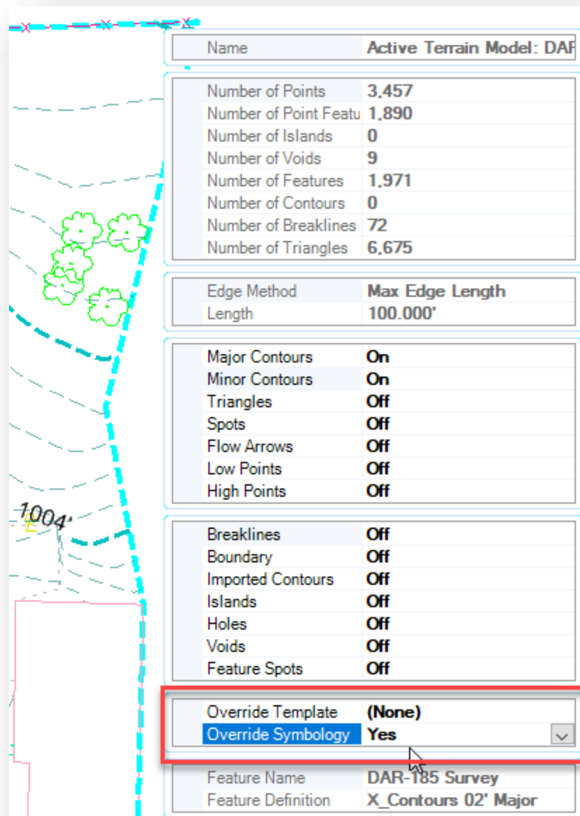
The display of the terrain model is controlled by the feature definition that is assigned in the design file that contains the terrain element. In the example above, the terrain is displayed using the **X_Boundary** feature definition.

The terrain display can be defined uniquely for the active file as follows:

- Use the **Element Selection** tool to select the terrain model element.
- Allow the cursor to rest over the terrain element. The menu shown below appears. Choose the **Properties** command.



The properties dialog appears as a pop-up dialog as shown below.



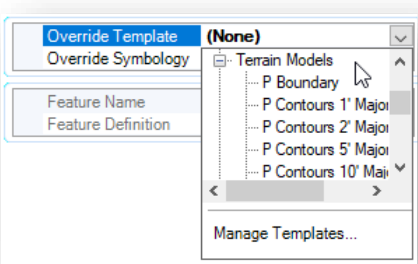
Since the terrain model is contained in a reference attachment, all the fields are ghosted and cannot be edited.

The user may wish to override the assigned feature definition to turn on/off contours and triangles or change the contour interval settings. This is accomplished as follows:

- Set the **Override Symbology** option to **Yes** to allow editing the terrain model properties.

  After setting the **Override Symbology** setting to **Yes**, the display of the terrain features can be toggled on/off using the options shown at right for turning on/off contours, triangles, etc. Note that the interval for the contours is controlled by the feature definition assigned to the terrain model, which cannot be edited across references.

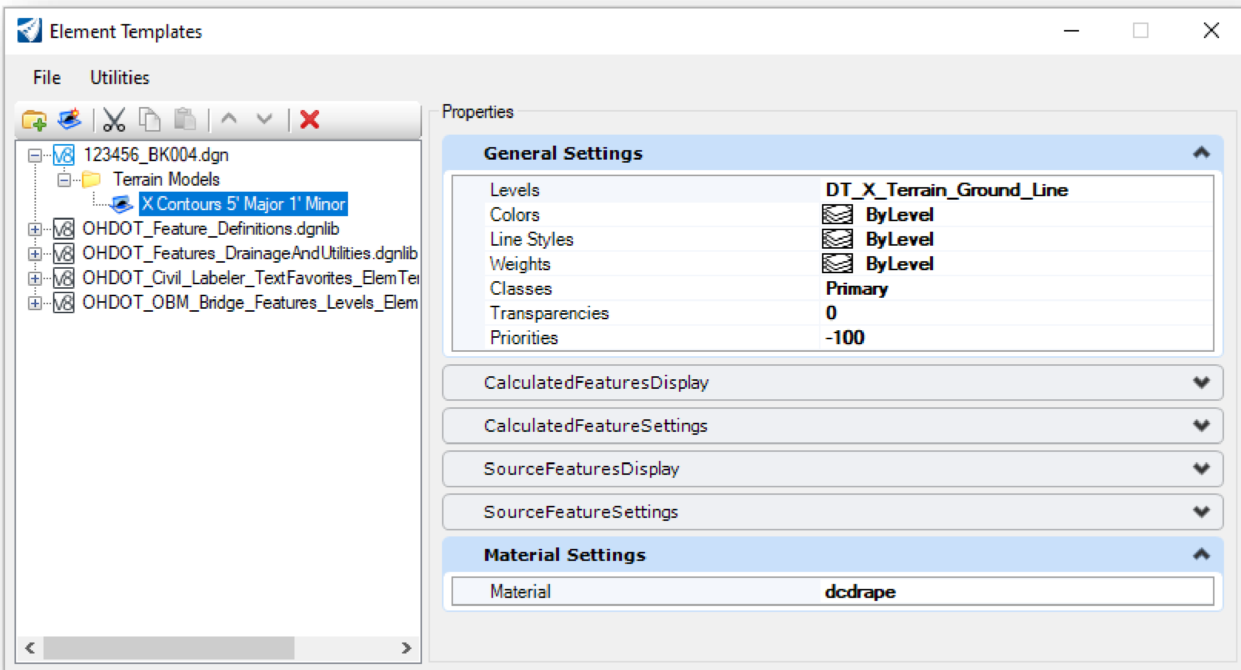
  The **Override Template** option is used to select a different element template for the terrain display. The OHDOT Standards contain several element templates for terrain display in the **Terrain Models** category, as shown below.



When one of the element templates is selected from the library, the element template definition is copied from the library to the active file. If necessary, the terrain display can be further customized by editing the local copy of the element template, as follows:

- From the **Home** tab, select the **Manage** option from the **Element Template** drop-down list (located in the **Attributes** group). This opens the ***Element Templates*** dialog.

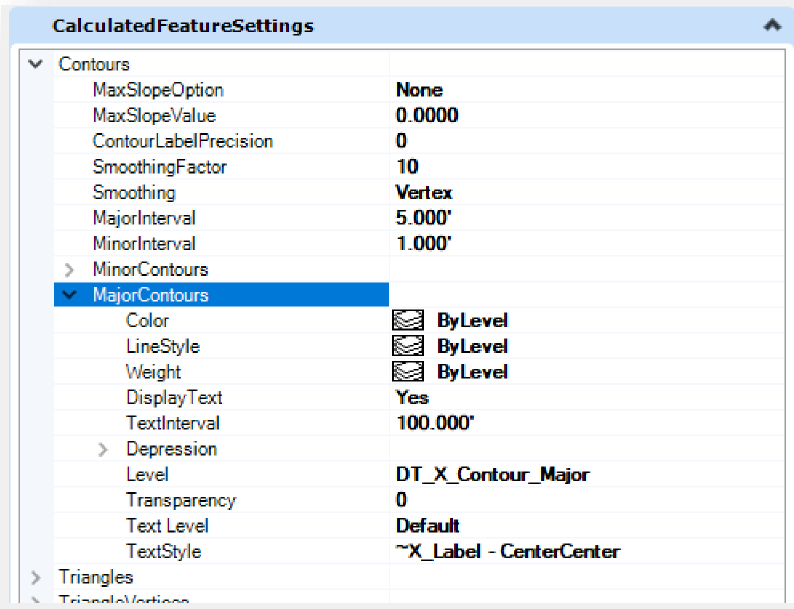
- The ***Element Templates*** dialog, shown below, lists any element templates that has been copied into the active file. Expand the active file to access the **Terrain Models** group. The settings for the element template can be edited in this dialog.



The **General Settings** group controls how the terrain is displayed in the cross sections. These values should not be changed.

The **Calculated Features Display** group allows turning on/off which features (contours, triangles, etc.) are displayed. Note that if the feature display is turned on, the display can also be turned on/off in the ***Level Display*** dialog.

The **Calculated Features Settings** group contains the settings for contour intervals and symbology settings.



Note: The major contour text labels are placed at an interval along the contour line, defined by the **DisplayText** and **TextInterval** parameters. This may result in contour labels falling into locations where they may overlap other elements or annotations. The **Label Terrain Contours** command (**OpenRoads Modeling > Terrain > Labeling**) can be used to annotate the major contour lines at user defined locations in the active file.

Section 300 Geometry Basemaps

Basemap Design Files

ODOT's CADD Standards detail the use of various basemap design files to store data such as plan, profile, corridor modeling, and cross section information. The basemaps store this information in real world dimensions, not scaled for a specific plan sheet presentation. These drawings are then referenced into sheet design files to produce the plan sheets for the project.

Several basemap files may be used depending on the project complexity. The ODOT CADD Standards allow for multiple basemap design files for each unique discipline (Drainage, Lighting, MOT, Roadway, Signals, etc.)

The following basemap design file types will be used in this training course:

DGN File Type	Two-character code	Location
Survey Field Book	FB	300-Survey\Basemaps \
Geometry	BK	400-Engineering\Roadway\Basemaps \
Right-of-Way	BR	400-Engineering\RW\Basemaps\
Superelevation	KS	400-Engineering\Roadway\Basemaps\
3D Modeling	KM	400-Engineering\Roadway\Basemaps\

The **OHDOT Create Design Files** application is used to create new design files using the seed files for the project.

See the **Location & Design Manual, Volume 3 - Highway Plans**, **Section 1204.3 File Management** for more details.

Geometry Basemaps

With OpenRoads Designer, all geometric information is stored in a MicroStation design file. A separate external database is not used as it was with Bentley's PowerGEOPAK software.

ODOT's CADD Standards designate a specific basemap design file for storing OpenRoads geometry. It is not recommended to store the OpenRoads geometry information in the same design file as the plan view graphics or the survey basemap. Geometry basemaps are stored in **400-Engineering\Basemaps** folder for the project.

Geometry basemap design files are named as follows:

nnnnnn_BK###.dgn

where:

nnnnnn = the six-digit PID number for the project
= a three-digit number for the file (001, 002, 003, etc.)

Example: 123456_BK001.dgn

Using Multiple "Federated" Basemaps

ODOT recommends a "federated" approach to store each geometric alignment in its own design file. This "federated" approach will allow multiple designers to work on different alignments simultaneously.

The **OHDOT Create Design Files** application is used to create geometry basemaps for the WorkSet.

Section 301 Geometry Basemaps Exercise

Overview

The sample project uses three alignments as listed below. Each alignment will be contained in its own design file.

Exercise A - Geometry Basemaps

For the sample project, the Survey personnel have created a design file containing the existing survey information:

File Name	Content
300-Survey\Basemaps\123456_FB001.dgn	Survey mapping and the existing ground terrain

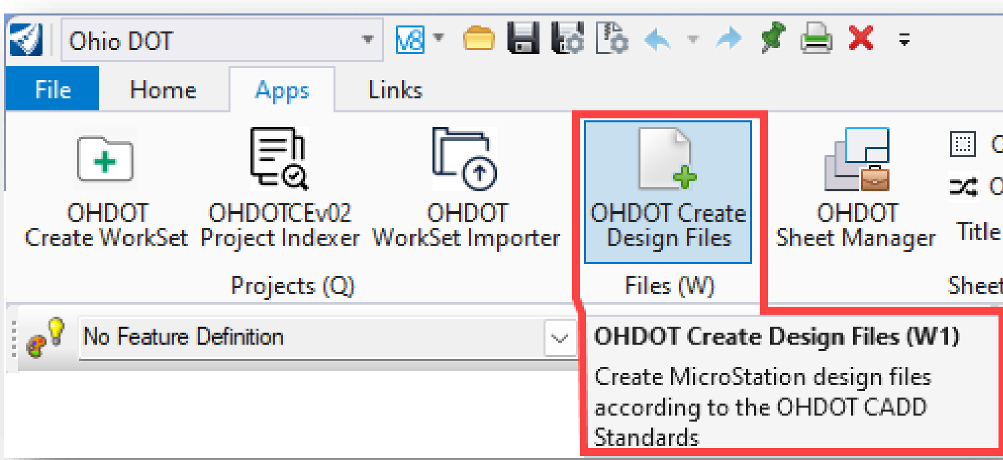
In this exercise we will create multiple design files to house the proposed geometry for the project.

- Open **ProjectWise Explorer**

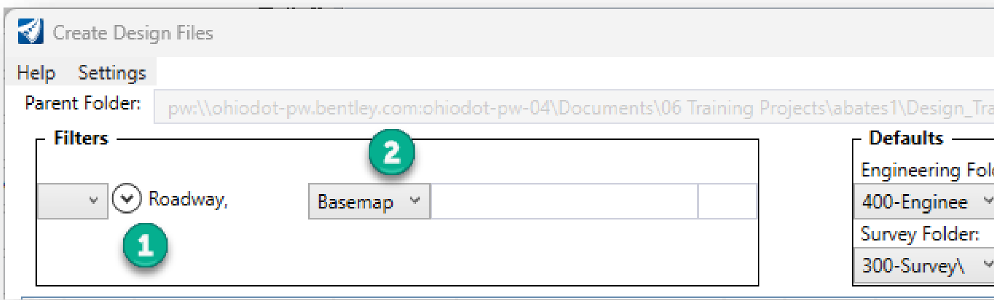
- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **400-Engineering\Roadway\Basemaps** folder.

Create new design files for the project using the **OHDOT Create Design Files** application. To create these files, the project FB file may be opened. If there aren't any basemap deign files defined for the project you may use one of the project seed files but not the recommended workflow.

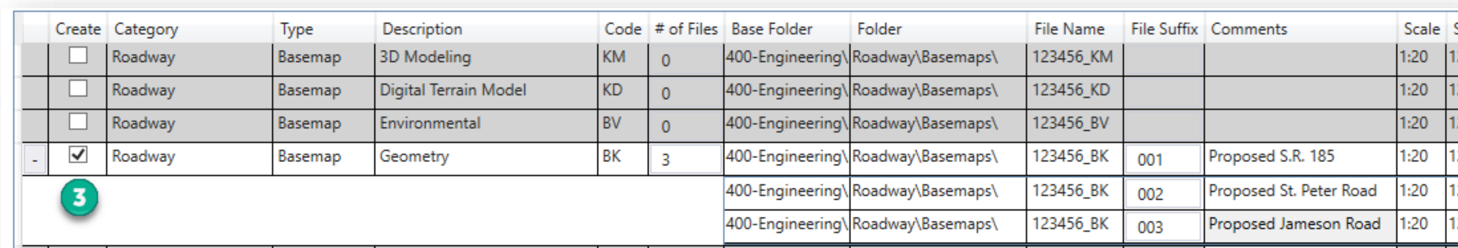
✓ Browse to the **300-Survey\Basemaps** folder.
- Open the **123456_FB001.dgn** file from the **300-Survey\Basemaps** folder.
- Select the **Ohio DOT** Workflow
- Select the **OHDOT Create Design Files** application



- Click on the **Category** drop down 1 and select the **Roadway Category**. Next click on the **Type** 2 and select **Basemap**.



- Within the **Create** column check the box 3 for Roadway Basemap Geometry BK file.



- Under column **# of Files** type in **3.** Using the information in the table below, fill out the columns **File Suffix** *(generally prepopulated)* and **Comments.**

File Name	Comments
123456_BK001.dgn	Proposed S.R. 185
123456_BK002.dgn	Proposed St. Peter Road
123456_BK003.dgn	Proposed Jameson Road

- To Create three new Geometry basemaps for the project click the **Create Files** button. This will open the last created file **123456_BK003.dgn**.

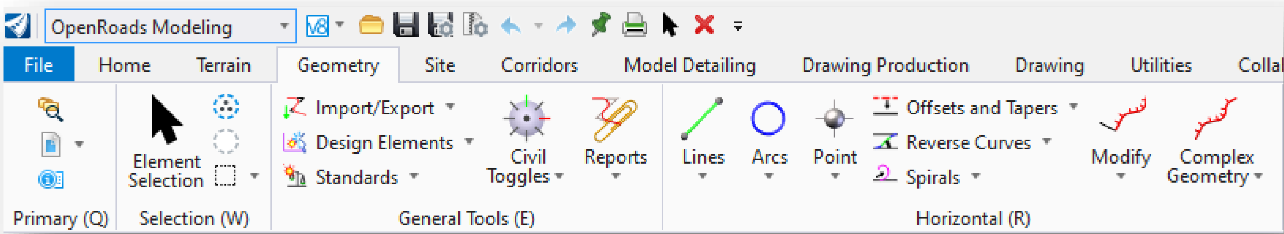
Note: If the user doesn't want to open the last created file. The user can turn this off by clicking the **Settings** to the upper left of the application and unchecking the **Open Last Created File**.

- Exit OpenRoads Designer (ORD), checking the file back in to ProjectWise. This completes this exercise.

Section 302 Horizontal Geometry

OpenRoads Geometry Tab Overview

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.

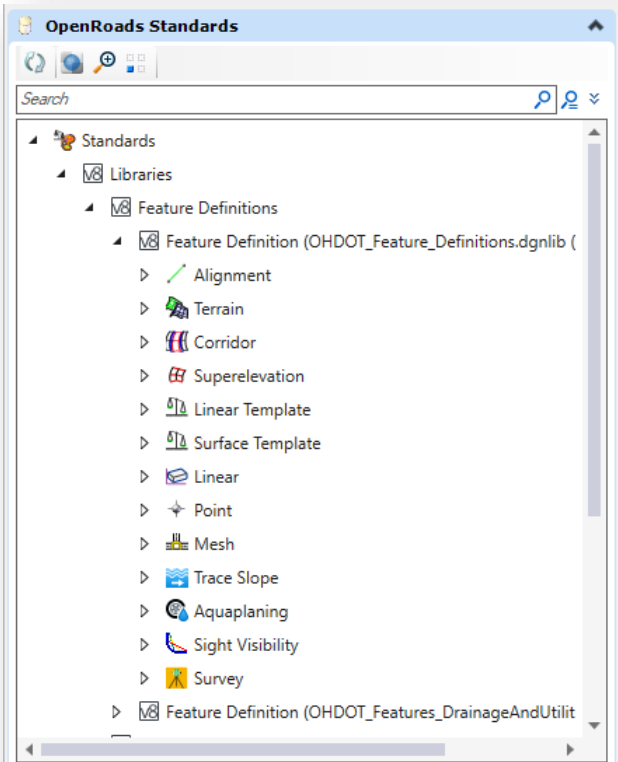
Feature Definitions

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.

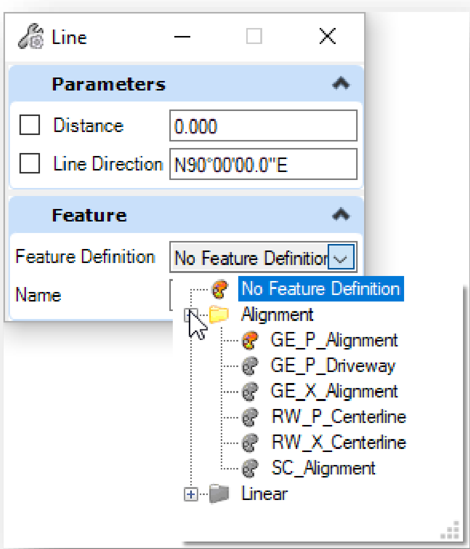


In addition to defining the symbology settings, feature definitions can also define annotation settings for each item.

For example, the **Alignment** feature definitions also define how an alignment will be annotated using the OpenRoads **Element Annotation** and **Model Annotation** 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.

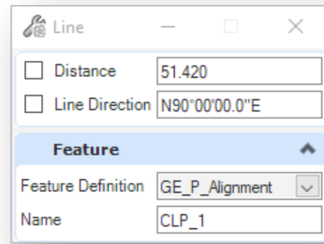
The feature definition is assigned when an element is placed. This can be accomplished by choosing the **Feature Definition** from the ***Tool Settings*** dialog as shown below.



Feature Name

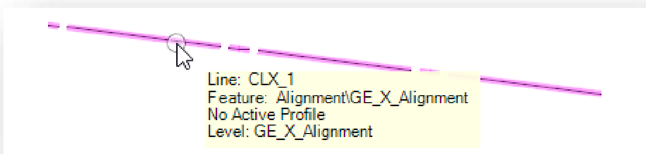
As civil geometry elements are placed, each element is given a feature name. The **Name** field displays the seed name that has been assigned to the selected feature definition.

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.

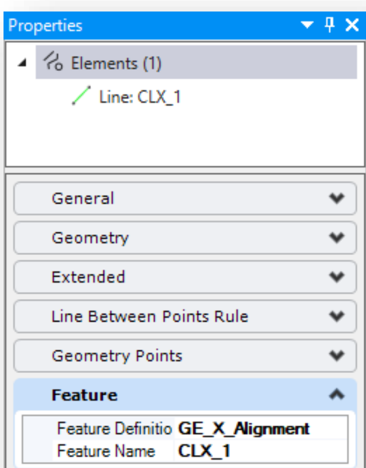


Note: By default, the software always displays the seed name, not the actual name of the element that is placed.

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).



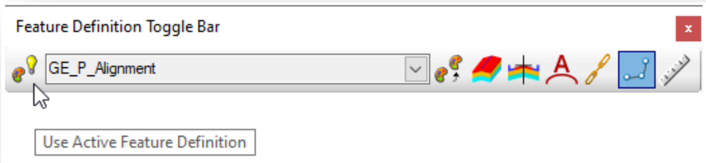
The **Feature Definition** and **Feature Name** for a selected element can be edited in the ***Properties*** dialog as shown below.



Standard element names for alignments are defined in the **<u>Location & Design Manual, Volume 3 – Highway Plans</u>**, **See section 1204.3.6.2**

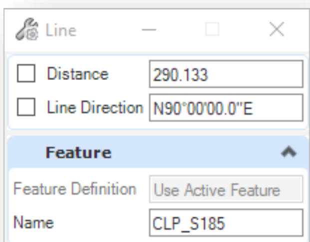
Feature Definition Toggle Bar

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**.

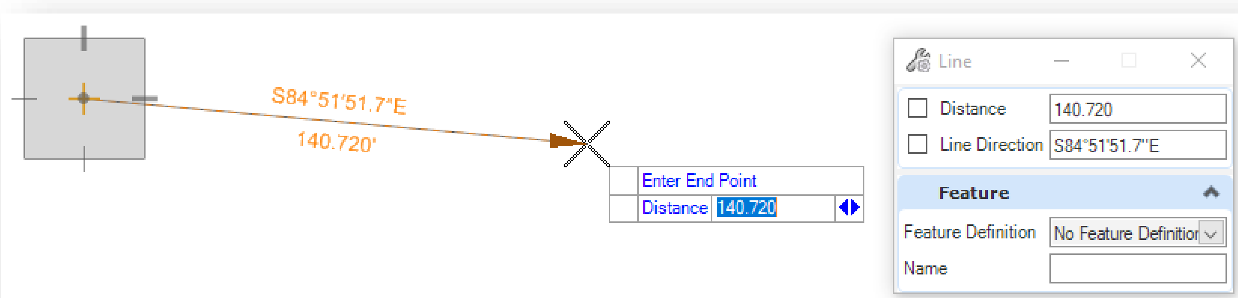


See the online help for a description of the remaining **Feature Definition Toggle Bar** icons.

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.

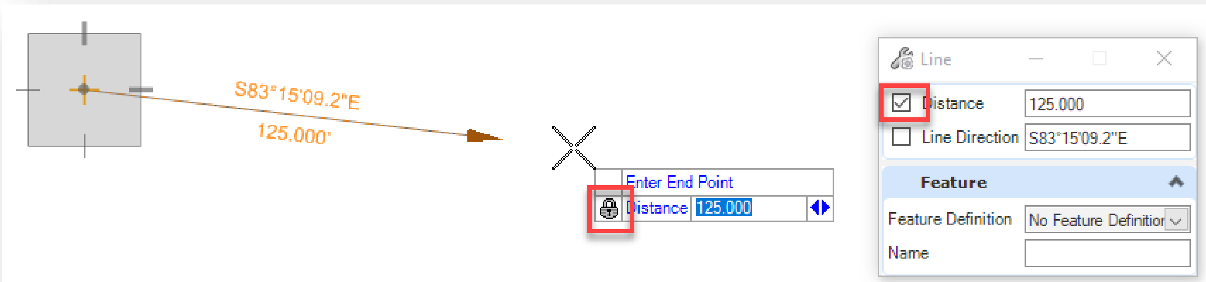
For example, a line drawn using the **Line Between Points** command has the following rules: **Distance** and **Line Direction** as shown below.



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.

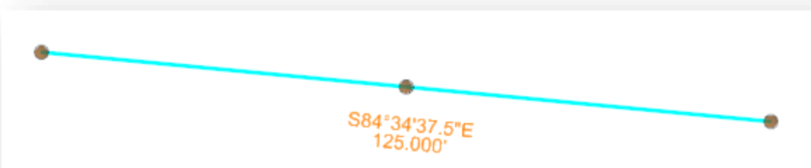
- When a parameter value is entered, it is locked as shown below. The parameter can be unlocked by unchecking the item in the ***Tool Settings*** dialog, or by pressing the **End** key on the keyboard while the parameter value on the cursor has the keyboard focus.



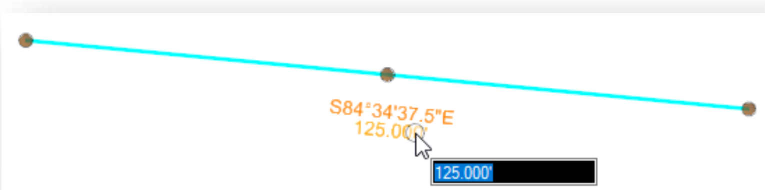
- After a parameter value has been keyed in, use the left and right arrow keys, or the **Tool Settings** dialog to enter values for the remaining parameters

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

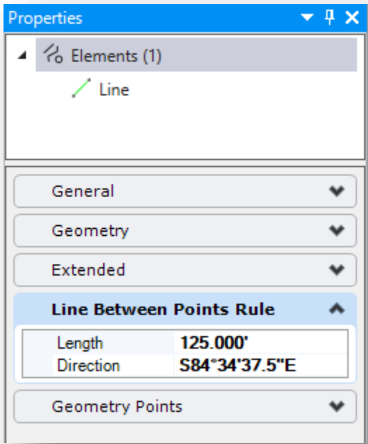
The rules can be reviewed and edited by using the MicroStation **Element Selection** tool to select the element. In the example below, the rules for the selected line are displayed.



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

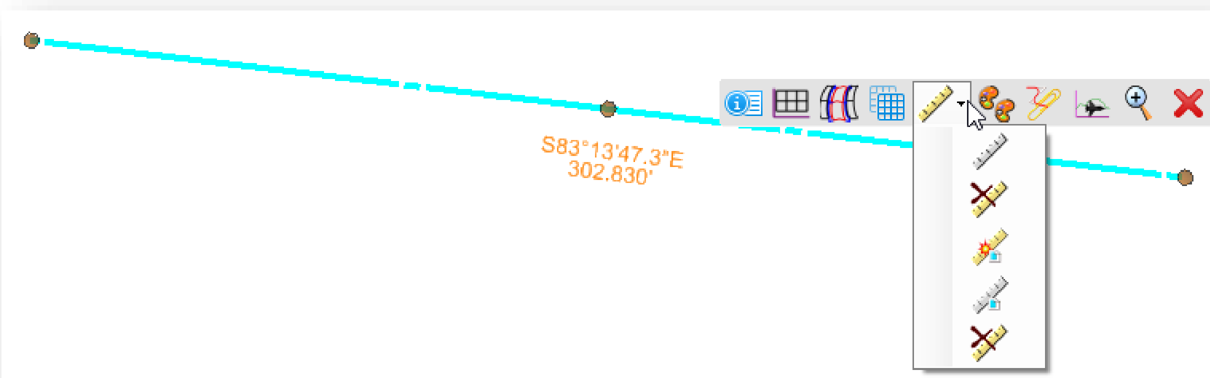


The rules for the selected element can also be reviewed and edited in the ***Properties*** dialog as shown below.



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)

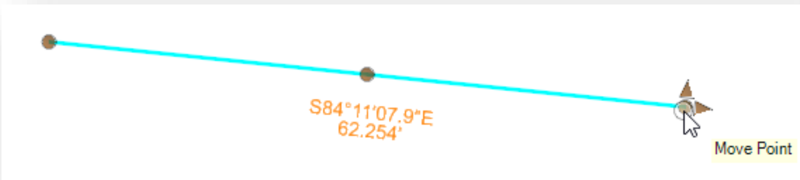
Selecting the Deactivate/Activate Rules and Remove Rule icons affect only the selected element.

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.



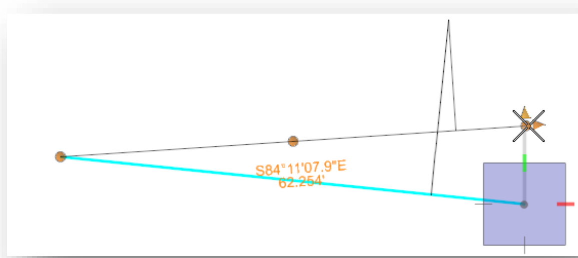
Tip: The size and color of the Manipulator Text can be adjusted in the **User Preferences**. From the MicroStation menu, select **File > Settings > User > Preferences** and then choose the **View Options – Civil** item on the left side of the dialog. The **Manipulator Size** or the **Manipulator Font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

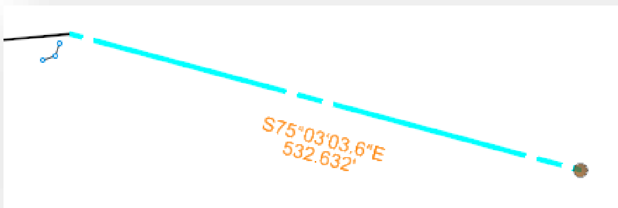


Snaps

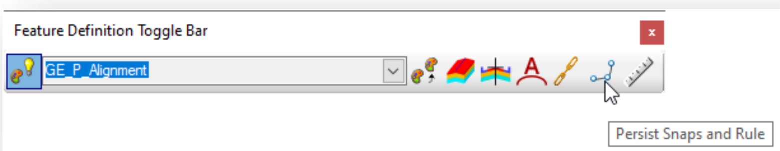
As geometry elements are placed in the design, the elements are "ruled" to one another using snaps.

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.

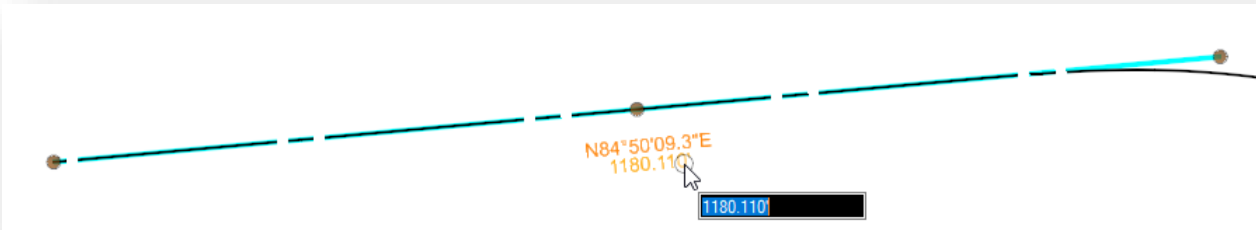


Editing Dynamic Text

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.

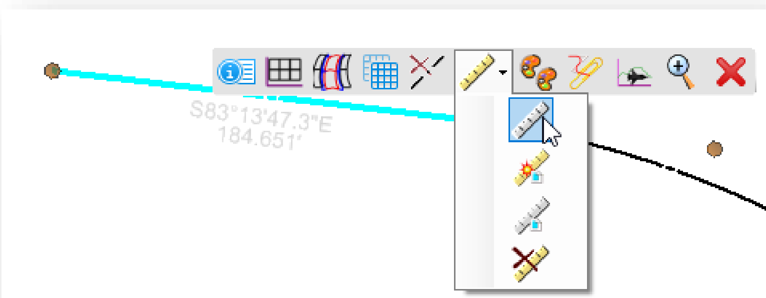
- Use the MicroStation Element Selection tool to select the geometry element
- Right-click on the element that you wish to edit
- The underlying element is selected, and the associated rules can be edited as shown below



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**.

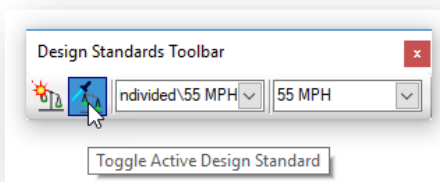
The design standards can be used for two purposes:

- 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

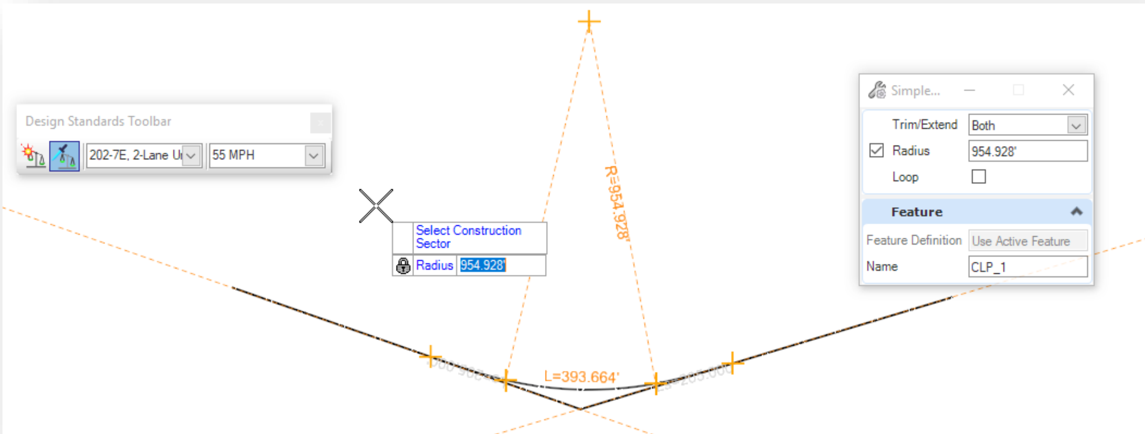
The example below details the process to apply a selected design standard to the **Simple Arc** command to place a curve between two tangent lines.

- 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**

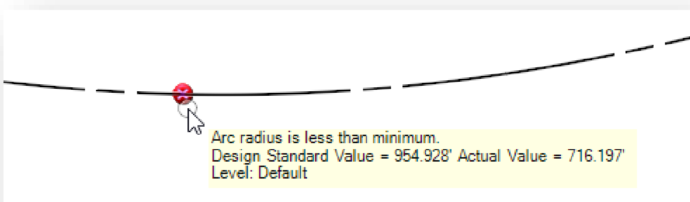
- The design standard defines a minimum transition length of 205 for the default radius. Spirals will be placed even though the **Simple Arc** command was initially selected.



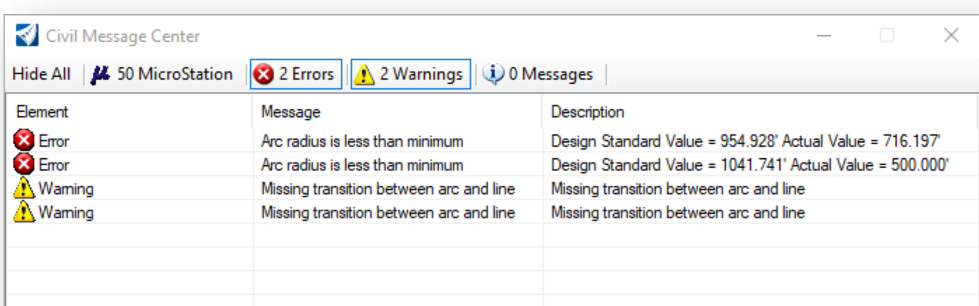
Design Standards can also be applied to an element after it has been created by using the **Set Design Standard** icon on the *Design Standards Toolbar*.

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 complete the operation.

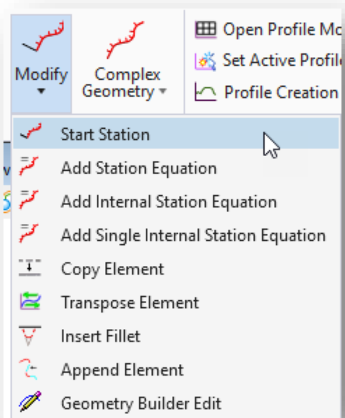
Be sure to enter a **Name** for the geometry. Standard geometry names are defined in the **Location & Design Manual, Volume 3 – Highway Plans, See section 1204.3.6.2**

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.

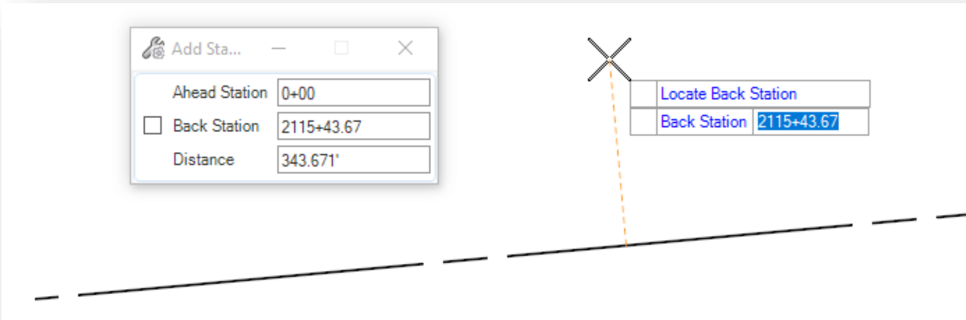
The station parameters can be edited by selecting the rule in the design file, or from the ***Properties*** dialog.

Add Station Equation

The **Add Station Equation** command, located below the **Start Station** command, adds parameters for the **Ahead Station** and **Back Station**.

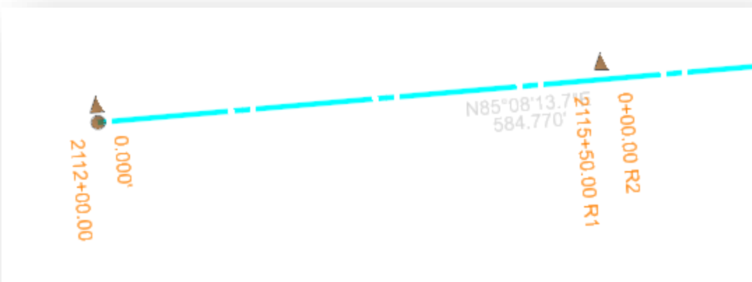
When the command is selected, the user is prompted to select the alignment to assign the station equation.

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.

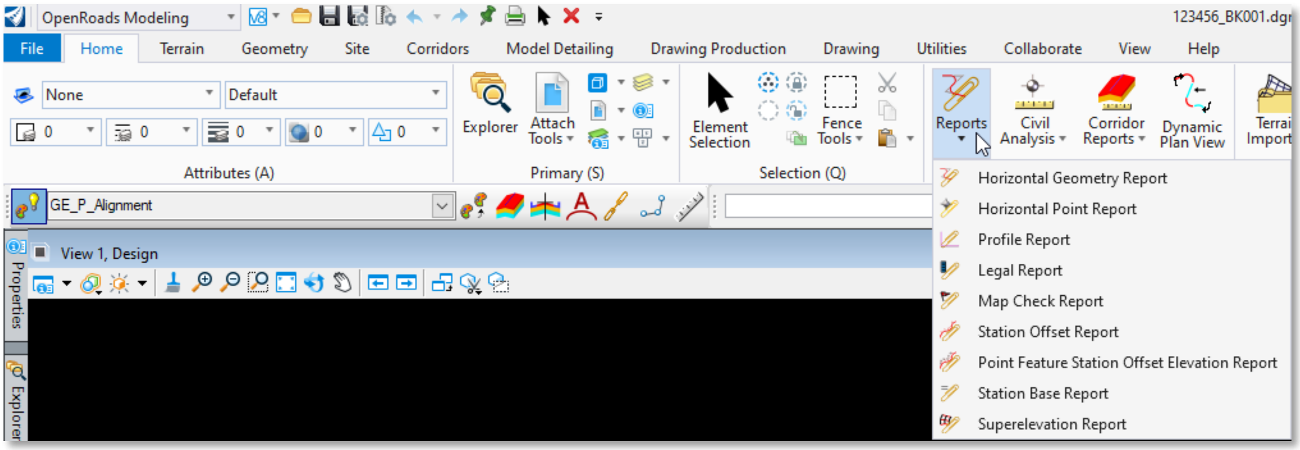
The station equation value can be reviewed and edited by selecting the element as shown below. The parameters can also be edited in the ***Properties*** dialog.



Reports

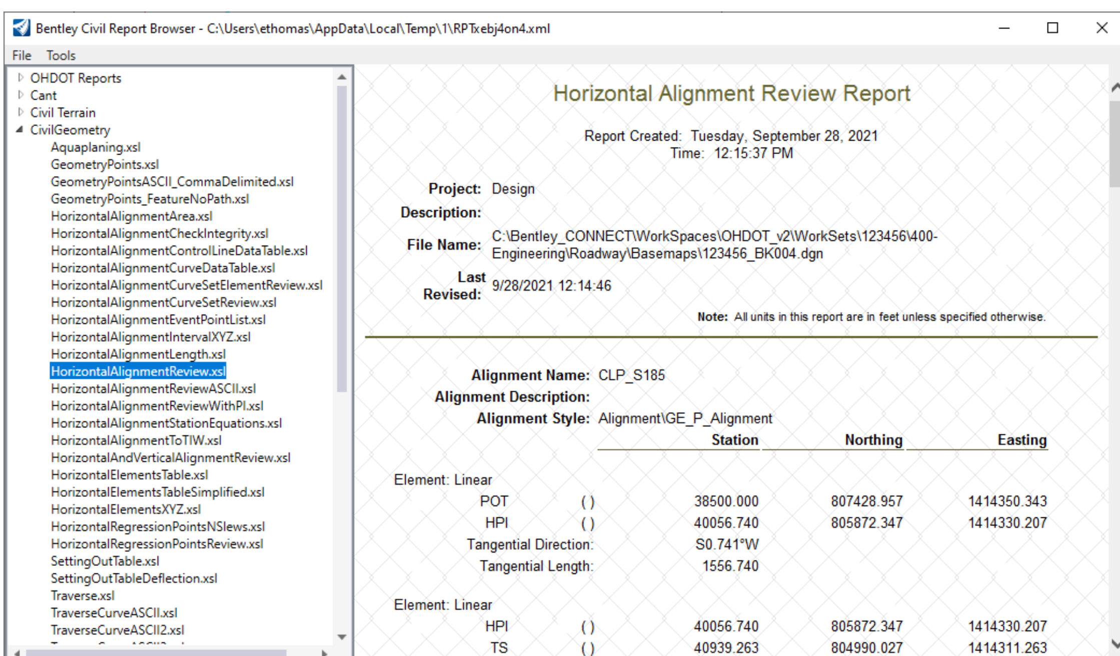
Several preconfigured reports are provided with the OpenRoads software that can be used to review alignment information.

Various reports can be accessed from the **Home** tab, as shown below.



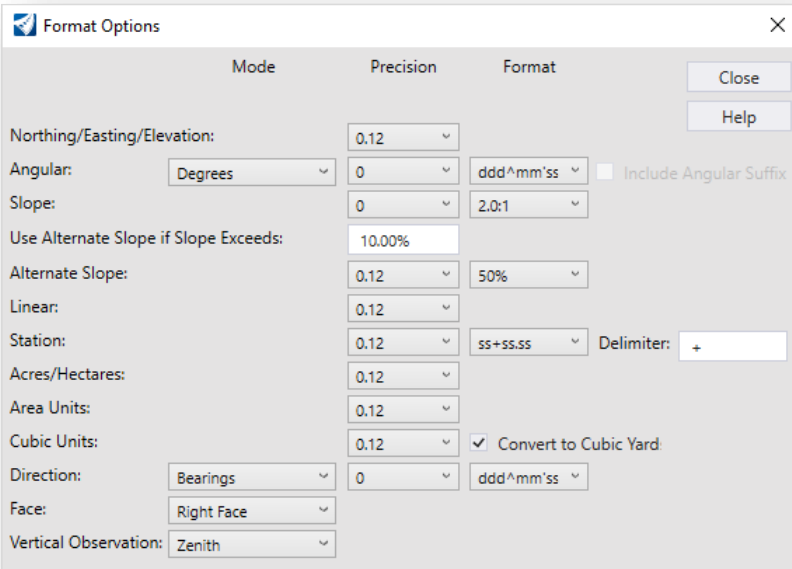
Additionally, reports can be generated by selecting a civil geometry element and choosing the Reports icon from the pop-up menu.

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.

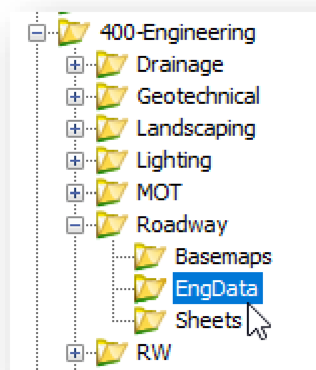


The format options are stored as a user setting in the Windows Registry and must be manually set by each user on their computer.

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. Reports should be saved to the **EngData** Folder of the project to the corresponding engineering discipline.



Section 303 Civil AccuDraw

Civil AccuDraw Overview

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

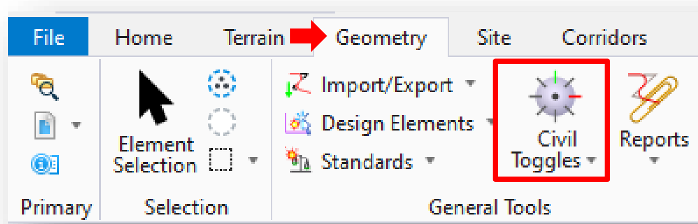
By default, data points in a 3D model are placed at the view's **Active Depth**. Where you snap a tentative point or place a data point in a blank part of a view, it will be located at the active depth. You can, however, snap a tentative point to an existing element at any depth in a view. When you accept such tentative points, the data point is placed at the level of the snap point.

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.

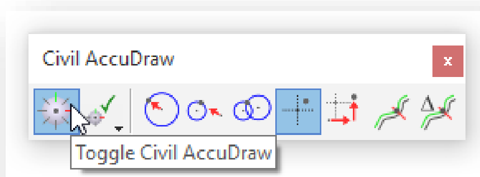
Civil Accudraw is activated by selecting the **Civil Accudraw** icon from the **Geometry** tab as shown below.

You can find Civil Accudraw tool set under these 2 workflows

- OpenRoads Modeling
- Survey



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

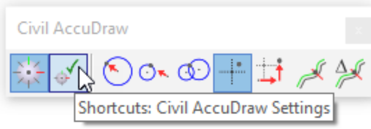


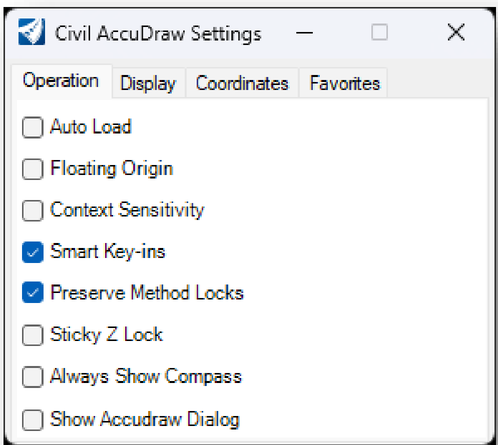
Civil AccuDraw performs many of the same functions as MicroStation AccuDraw but has greatly expanded capabilities for the civil designer. This guide will cover only a few of the functions available when using Civil AccuDraw. For more information, see the ORD online help.

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

Civil Accudraw Settings

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





Auto Load – (Default OFF)

Toggle this option ON to automatically load Civil Accudraw

Floating Origin - (Default OFF)

ON: the origin moves to the last point placed.

OFF: the origin remains fixed.

ODOT recommends turning this option off.

Context Sensitivity - (Default ON)

ON: the compass rotates in response to the context of various tools.

ODOT recommends turning this option off.

Smart Key-Ins - (Default ON)

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 - (Default ON)

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-station lock.

Sticky Z Lock - (Default OFF)

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.

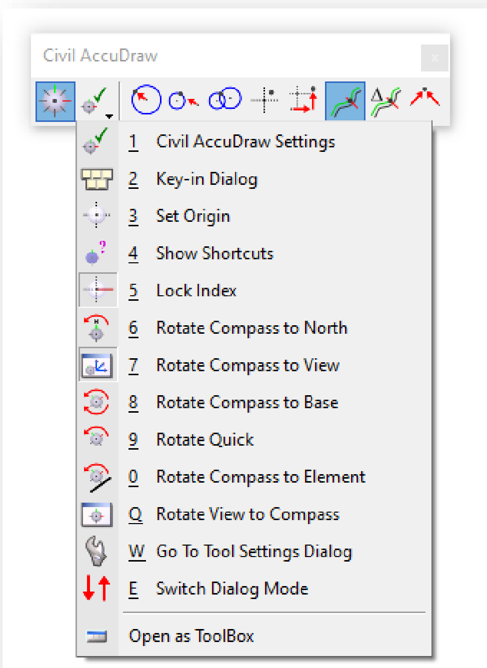
Always Show Compass - (Default OFF)

ON: the compass is always visible.

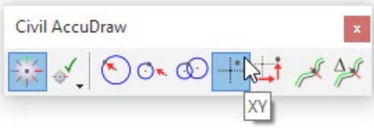
OFF: the compass is only visible when the appropriate input is required.

Civil Accudraw Shortcuts

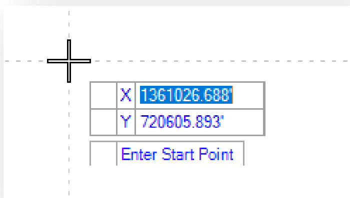
Hold down the second icon to reveal the shortcut menu shown below.



XY



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.



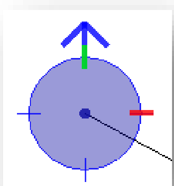
The **Tab** key is used to move through the key-in fields in the dialog.

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.

Civil AccuDraw Compass

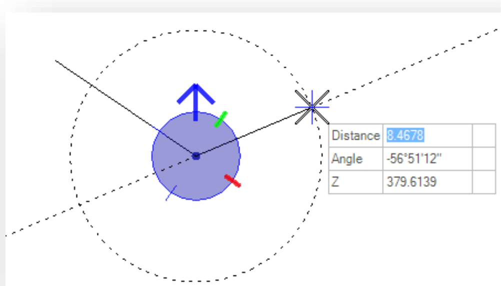
When the **Always Show Compass** setting is toggled on, a compass is displayed at AccuDraw's origin point when Civil AccuDraw is active.



The compass is always circular and is marked by a few 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:



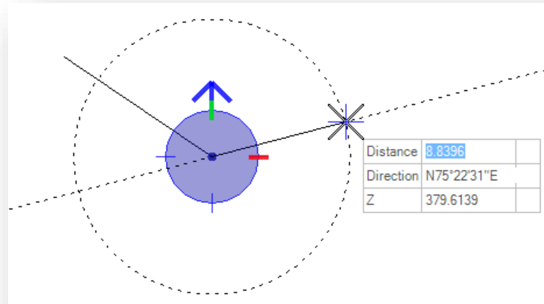
Note that rather than direction being shown, the label and value are now an **Angle**. This is because the context is based on the direction of the preceding line segment rather than an absolute direction.

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.

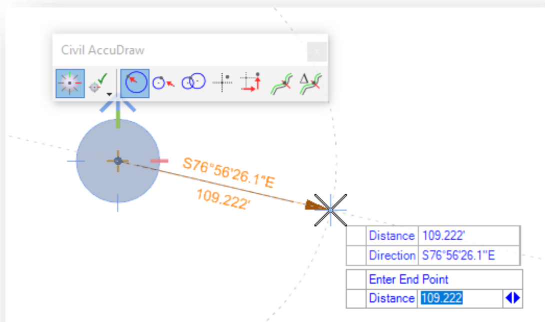
RQ is used to quickly and temporarily rotate the drawing plane.
RE rotates the drawing plane to match the orientation of a selected element.

Use the **T** (Top) shortcut or the corresponding drop-down to return to a pure direction.



Distance-Direction

The **Distance-Direction** ordinate is used to enter points in polar coordinates as shown below.



Angle Syntax

The input for angle and direction fields follows the **File > Settings File > Design File Settings > Angle Readout** settings.

Valid forms for the various settings are as follows:

DD.DDDD, Radians or Grads - Use the customary decimal input.

DD MM SS or DD MM - The following delimiters are supported between degrees, minutes, and seconds:

Colon - For example DD:MM:SS

^ (caret) to designate degrees, minute (’) and second (”) - For example DD^MM’SS”

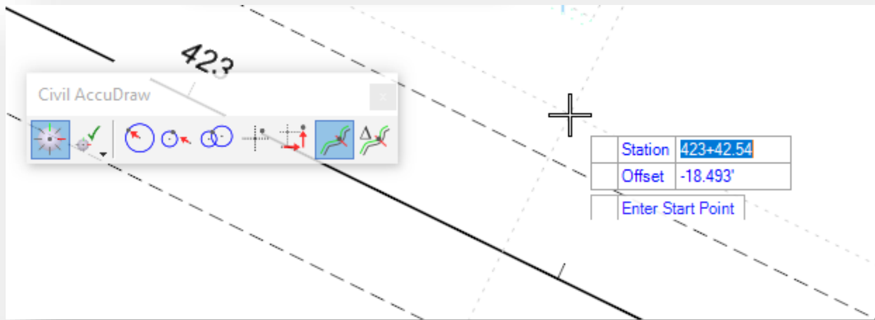
The following are not supported:

A space between degrees, minutes, and seconds because space launches the Civil AccuDraw popup menu.

Use d, m, and s to designate degrees, minutes, and seconds.

Station-Offset

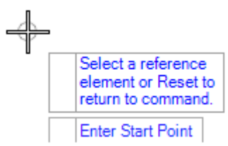
The **Station-Offset** ordinate is used with a reference element to enter information as a station and offset relative to a selected element.



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.

Defining a Direction-Direction Intersection Ordinate

Civil AccuDraw allows users to customize the dialog or add additional ordinates. This section documents how to add a direction-direction intersection ordinate. This is useful for computing a bearing-bearing intersection.

Take the following steps to add a direction-direction ordinate:

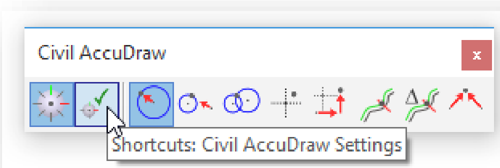
- Open **ProjectWise Explorer**

- Open the **Ohio DOT Projects** data source.

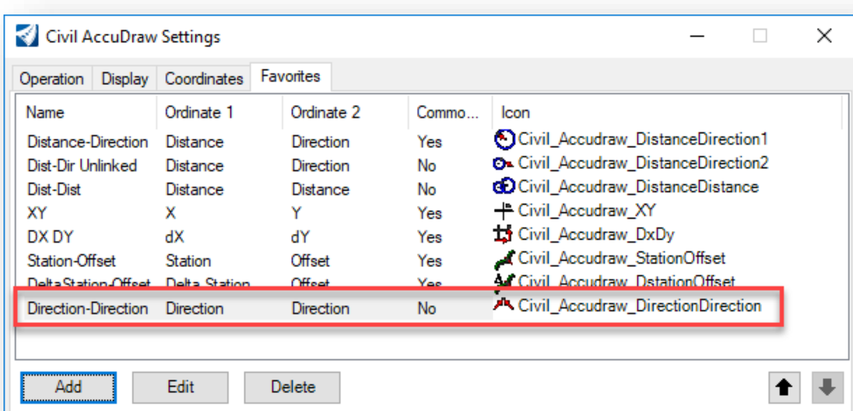
- Browse to the following folder to open the following design file:
  400-Engineering\Roadway\Basemaps\123456_BK004.dgn

✓ Open the Civil Accudraw dialog by choosing **Geometry > Civil Toggles > Civil AccuDraw**

✓ Click the second icon on the ***Civil AccuDraw*** dialog, **Shortcuts: Civil AccuDraw Settings**



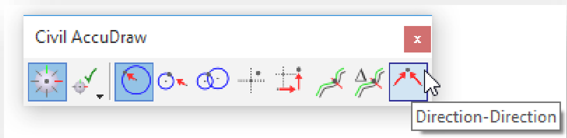
✓ Select the **Favorites** tab.



✓ Choose the **Add** button to add a new favorite.

- ✓ Configure the **Direction-Direction** ordinate as shown above.

- ✓ Close the Civil AccuDraw dialog and reopen it to see the added favorite, as shown below.



The added Civil AccuDraw favorite is saved in the following file:

C:\Users*username*\AppData\Local\Bentley\OpenRoadsDesigner\XX.0.0\prefs\accudrawFavorites.xml

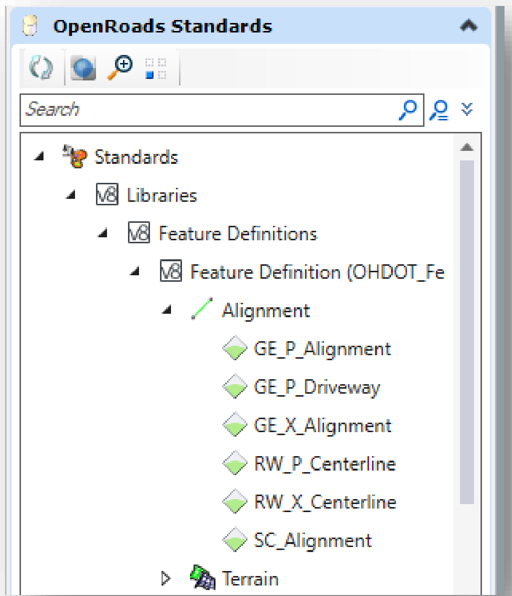
You will also find the Civil AccuDarw settings under the following Root directory.

C:\Users*username*\AppData\Local\Bentley\OpenRoadsDesigner\XX.0.0\prefs\accudrawSettings.xml

Note: *the XX indicates the ORD schema the user is using to open the file and modifying the accudraw settings.*

Section 304 Alignment Annotation

Alignment Feature Definitions



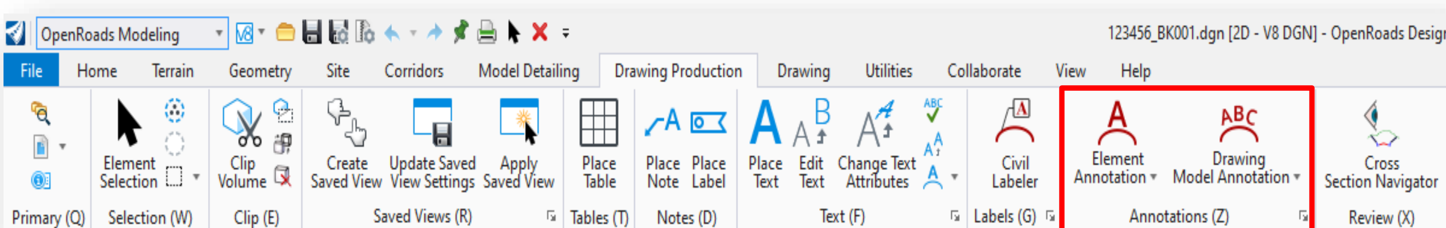
Alignment elements are assigned a **Feature Definition**, which controls the symbology of the element (level, color, line style, and line weight) as well as the annotation properties of the element. The OHDOT Standards include feature definitions for alignments, which can be reviewed from the *Project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

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

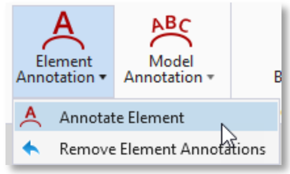
Alignment Annotation

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

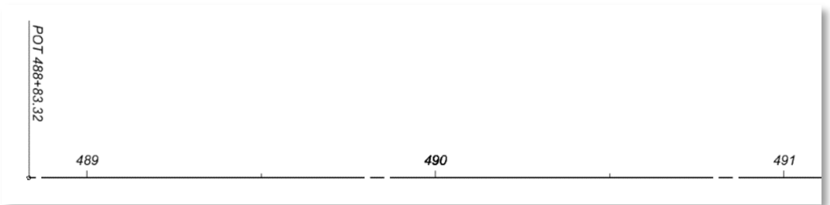


Individual elements are annotated by selecting the **Element Annotation** command from the **Drawing Production** tab. Two commands are available as shown below.

Annotate Element

The **Annotate Element** command is used to annotate one of the 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.

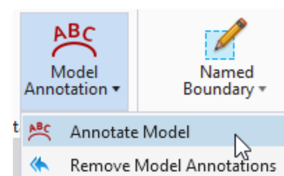


The annotation definition is part of the Alignment feature definition. The default annotation for alignment annotations using the OHDOT Standards places station labels every 100' like the example above.

Remove Element Annotations

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.

Model Annotation

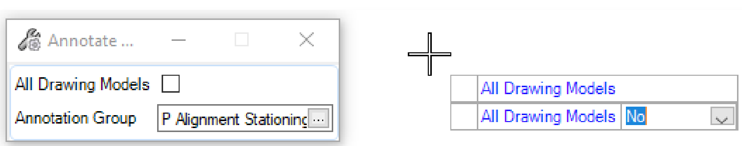


The **Model Annotation** command is used to annotate all of the elements in the active model, or all models, by selecting a specfic Annotation Group. Two options are available as shown below.

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.

Alignment Annotation Levels

For a typical ODOT plan set, the annotation for centerline stationing is displayed at different intervals along the alignment depending on the type of sheet showing the alignment information and the scale of the sheet (20-scale plan vs. a 500-scale schematic sheet for example). Depending on the sheet scale, the annotation may need to display stations at 100', 500', or 1000' intervals. These different station intervals are used to reduce clutter on the plan sheets.

The **Feature Definition** assigned to the alignment also includes an **Annotation Group** definition. The annotations are placed on separate levels as detailed below.

- GE_P_Alignment_XXX_Bearings
- GE_P_Alignment_XXX_Cardinal_Points
- GE_P_Alignment_XXX__Curve_Data
- GE_P_Alignment_XXX__Distances
- GE_P_Alignment_XXX__Stations

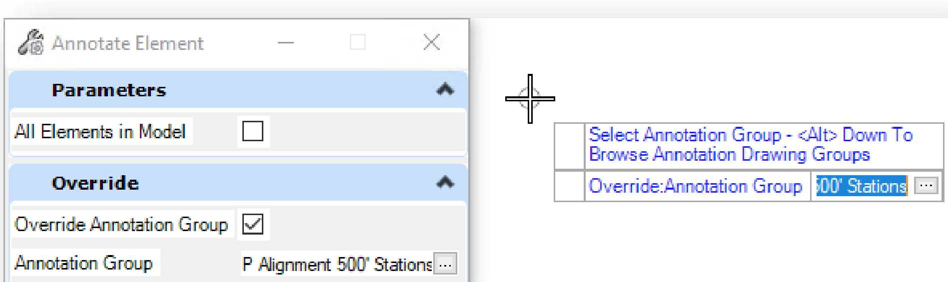
Where XXX = the station interval (100, 500, or 1000).

There are also three groups of levels for existing alignment annotation.

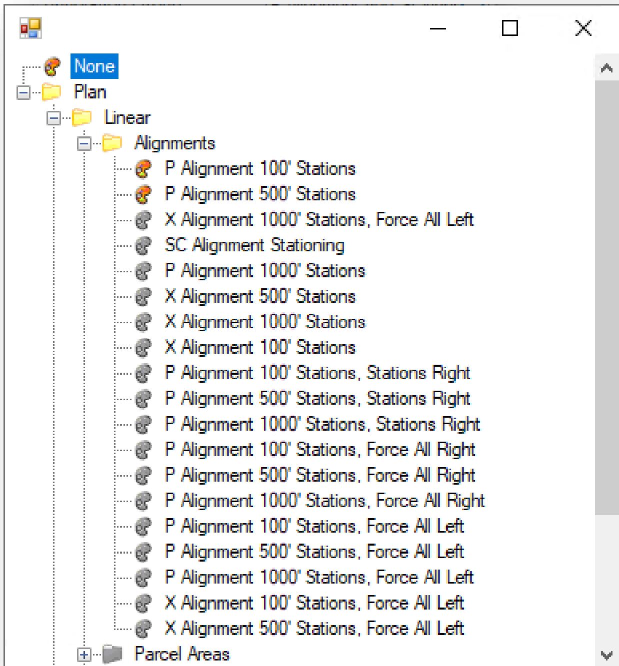
- GE_X_Alignment_XXX_Bearings
- GE_X_Alignment_XXX_Cardinal_Points
- GE_X_Alignment_XXX__Curve_Data
- GE_X_Alignment_XXX__Distances
- GE_X_Alignment_XXX__Stations

In older versions of the OHDOT CADD Standards that are compatible with OpenRoads version 10.09 and earlier, the default annotation definition was configured to place three sets of annotations on the 100, 500, and 1000 level groups at once. With this workflow, users were required to turn on/off the levels that they want to use for the plans depending on the desired station annotation interval.

This workflow has been changed with the OHDOT CADD Standards for OpenRoads version 10.10 and higher. By default, alignments are now annotated with only one set of annotations with stations shown at 100' intervals. If annotations are required using either the 500' or 1000' station interval, the default annotation definition can be overridden by toggling on the new **Override Annotation Group** setting as shown below.



ODOT provides several annotation definitions as shown below.



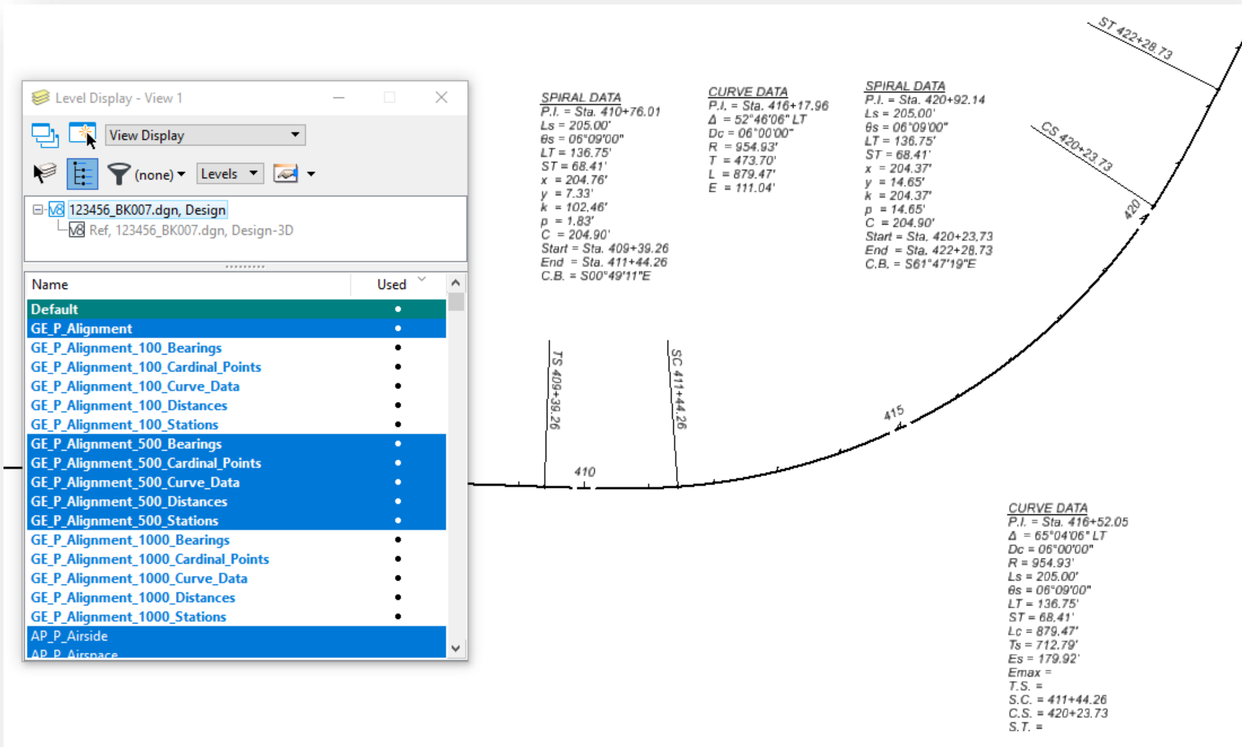
Note: Annotating an alignment with the **Override Annotation Group** option toggled on does not remove any previous annotations from the file. Use the **Remove Element Annotations** command to remove any previously placed annotations before re-annotating the element using an override.

The various annotation groups are summarized below. Annotation groups starting with the letter "P" are configured for proposed alignments. Annotation groups starting with the letter "X" are configured for existing alignments.

Annotation Group Name	Description
P Alignment – 100' Stations P Alignment – 500' Stations P Alignment – 1000' Stations	These are the normal annotation groups for most plan sheet applications. Station annotation is placed on the left side of the alignment. Spiral and curve key-point annotations are placed towards the curve center.
P Alignment – Force all Left 100' Stations P Alignment – Force all Left 500' Stations P Alignment – Force all Left 1000' Stations	These annotation groups force all station and key-point annotations to the left side of the alignment.
P Alignment – Force all Right 100' Stations P Alignment – Force all Right 100' Stations P Alignment – Force all Right 100' Stations	These annotation groups force all station and key-point annotations to the right side of the alignment.
P Alignment – Force Stations Right 100' Stations P Alignment – Force Stations Right 500' Stations P Alignment – Force Stations Right 1000' Stations	Station annotation is placed on the right side of the alignment. Spiral and curve key-point annotations are placed towards the curve center.

Annotation Groups for existing alignments are also available for each of the group definitions detailed in the table above starting with the letter "X" instead of "P".

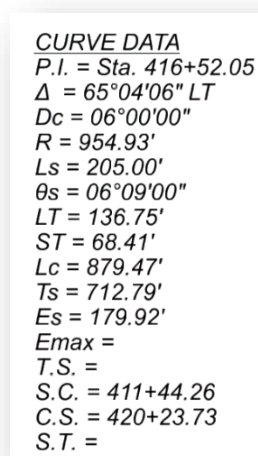
In the example below, the **Annotation Scale** value is set to 1:100 with the levels for 500' station intervals toggled on.



Additional Notes:

- The levels are named according to the station label intervals (100', 500', and 1000'), not the annotation scale value. For example, the **GE_P_100** group of levels are appropriate for multiple annotation scale values (1:10, 1:20, 1:50).

- In the event a Spiral-Curve-Spiral (SCS) is encountered, the SCS data is annotated at the location of the intersection of tangents. An example is shown at right. This data has been configured to include as much of the SCS annotation as the software currently allows. The text can be edited to include additional information for the Emax, T.S., and S.T. information. The individual spiral data is still placed in the file even when the combined SCS data is included.


CURVE DATA
P.I. = Sta. 416+52.05
Δ = 65°04'06" LT
Dc = 06°00'00"
R = 954.93'
Ls = 205.00'
θs = 06°09'00"
LT = 136.75'
ST = 68.41'
Lc = 879.47'
Ts = 712.79'
Es = 179.92'
Emax =
T.S. =
S.C. = 411+44.26
C.S. = 420+23.73
S.T. =

Section 305 Alignment Exercises

Alignment Exercises Overview

The following exercises step through the process to define proposed alignments for the sample project. Two proposed alignments will be developed for S.R. 185 and Jameson Road. The existing alignment for St. Peter Road will be used for the proposed improvements.

For these exercises, the following files have already been created in a previous exercise:

File Name	Content
300-Survey\Basemaps\123456_FB001.dgn	Survey mapping and the existing ground terrain
400-Engineering\Roadway\Basemaps\123456_BK001.dgn	Centerline of Proposed S.R. 185
400-Engineering\Roadway\Basemaps\123456_BK002.dgn	Centerline of Proposed St. Peter Road
400-Engineering\Roadway\Basemaps\123456_BK003.dgn	Centerline of Proposed Jameson Road

Exercise A - Proposed State Route 185

State Route 185 is the main alignment for the sample project. In this exercise we will define a new proposed alignment for State Route 185.

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.

✓ Browse to the **400-Engineering\Roadway\Basemaps** folder.

✓ Open the geometry basemap for State Route 185:

400-Engineering\Roadway\Basemaps\123456_BK001.dgn

✓ Attach the survey information as a reference file (**Home > Attach Tools > References**). The survey information is in the following design file:

300-Survey\Basemaps\123456_FB001.dgn

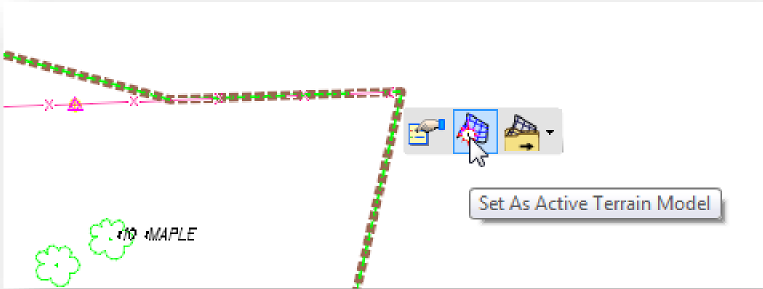
When attaching the reference file, which **Orientation** option should be used? **Coincident – World**, or **Geographic – Reprojected**?

Should **Live Nesting** be used?

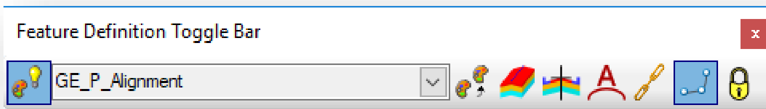
✓ Turn on/off levels for the survey information as necessary. Start by turning off the levels listed below, and any other levels that you may not want to see while working on the geometry for the project.

- PV_X_COP
- SV_X_Break_Lines
- SV_X_Random_Point
- SV_X_Shot_Marker
- SV_X_Text_Comment
- SV_X_Text_Elevation
- SV_X_Text_Name

✓ Select the terrain model graphics from the reference attachment and set it as the **Active Terrain** (select the terrain boundary and then hover the mouse over the boundary to get the pop-up toolbar seen below, a long right click on the boundary will bring up a context menu where the Set of Active Terrain Model button can be found as well).

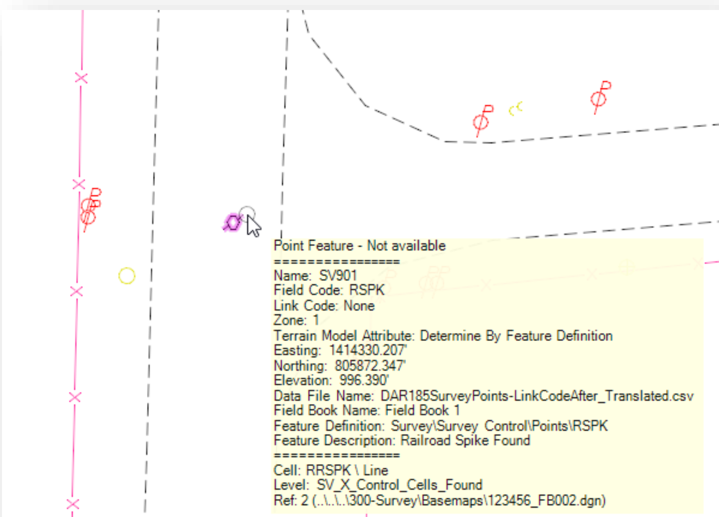


✓ From the **Geometry** tab, choose **Standards > Features Definition Toolbar**. Set the active feature to **Alignments > GE_P_Alignment**. Toggle on the **Use Active Feature Definition** icon.



✓ From the ***Geometry*** tab, choose the **Lines > Line Between Points** command.

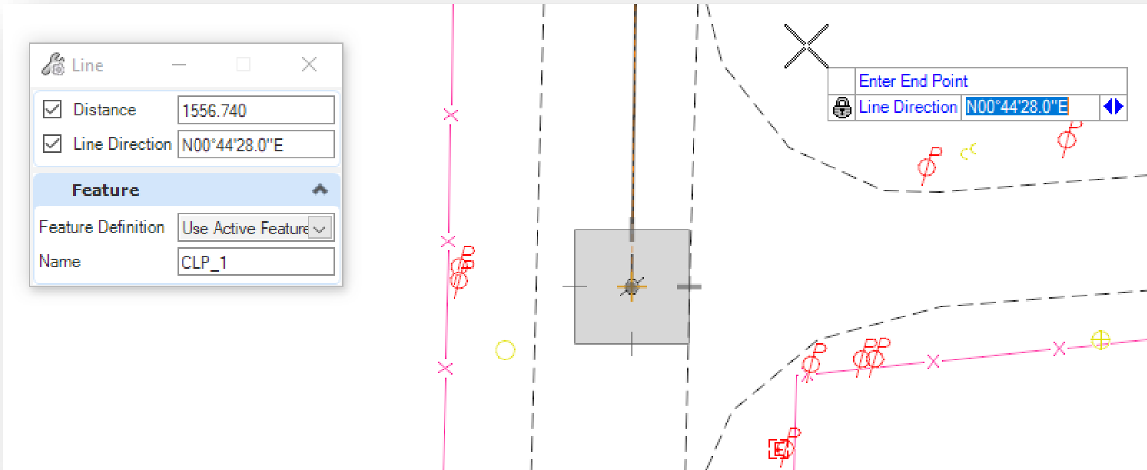
Start by placing the first line from the RSPK feature located at the intersection of St. Peter Road and S.R. 185. This is point SV901 in the Survey file. As shown below.



Use the MicroStation **Origin Snap** to ensure you are starting the first point of the line by snapping to the origin of the RRSKP cell.

✓ Define the second point of the line with a **Distance** value of **1556.74'** and a **Line Direction** value of **N00^44'28"E** as shown below.

Tip: Degrees, minutes, and seconds are entered using the colon character as a separator.
Example: n00:44:28e



✓ Use the **Line Between Points** command to store the following lines.

Define the second line starting from the RRSPK feature, point SV901, using the following settings:

Distance: 1595.31
Direction: s01:13:48w

Define the third line starting from the end of the second line.

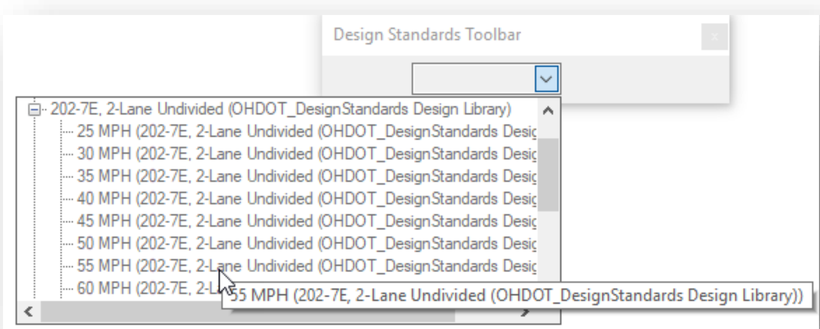
Distance: 1838.00
Direction: s63:50:18e

Define the fourth like starting from the end of the third line.

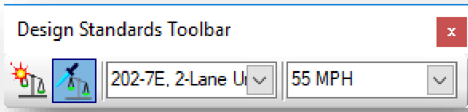
Distance: 650.00
Direction: s03:35:00w

✓ Next, use the ***Design Standards Toolbar*** to place a spiral-curve-spiral set. From the MicroStation ***Geometry*** tab, choose **Standards > Design Standards Toolbar** icon.

✓ Choose the **202-7E, 2-Lane Undivided 55 MPH** design standard as shown below.

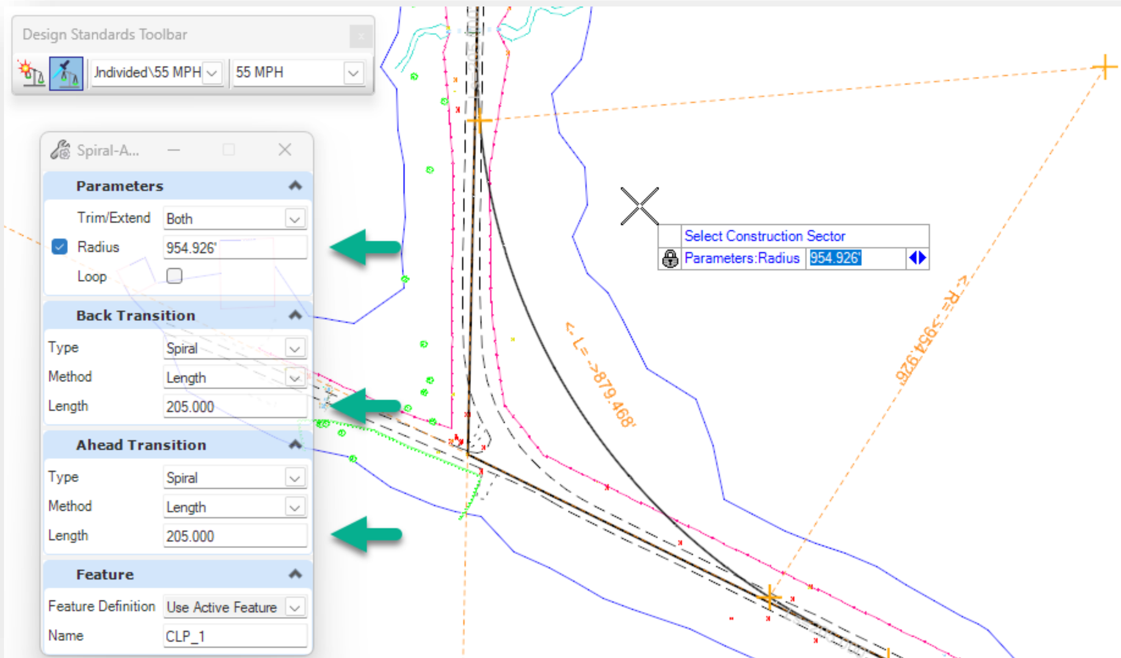


✓ Toggle on the second icon, **Toggle Active Design Standard**



✓ From the ***Geometry*** tab, choose the **Arcs > Arc Between Elements > Spiral Arc Spiral** command.

When using a **Design Standard**, the software will set the length of the spirals and the degree of curvature to the default values defined by the selected design standard.



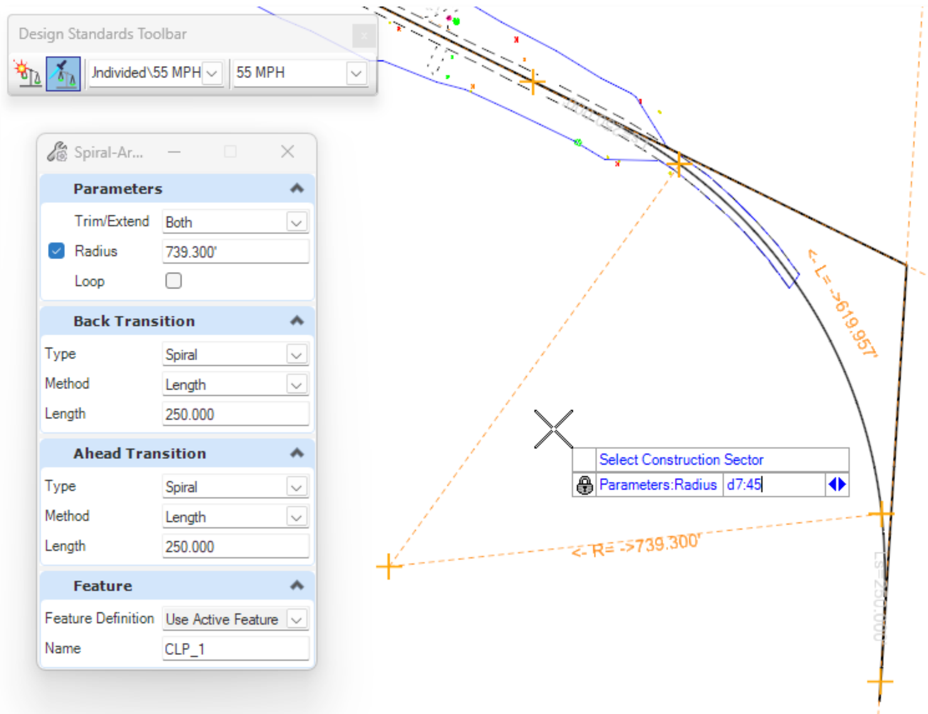
✓ Select **Trim/Extend Both** to complete the process.

Be sure to enter a left click (data point) to accept the **Trim/Extend** parameter. OpenRoads requires a left click to accept parameters as data is entered.

Use the **Spiral Arc Spiral** command to place the second curve for the alignment. For the second curve will match the existing alignment, the values defined by the active Design Standard will be overwritten.

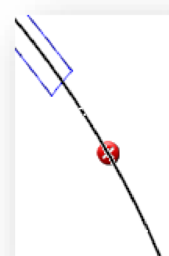
✓ Place the second spiral-curve-spiral using the following values:

Radius:	d7:45
Back Transition Length:	250
Ahead Transition Length:	250



Tip: The degree of curve can be set by keying in the degree value in the **Radius** field prefixed with a lower case “d”. Degrees, minutes, and seconds are separated by the colon character.

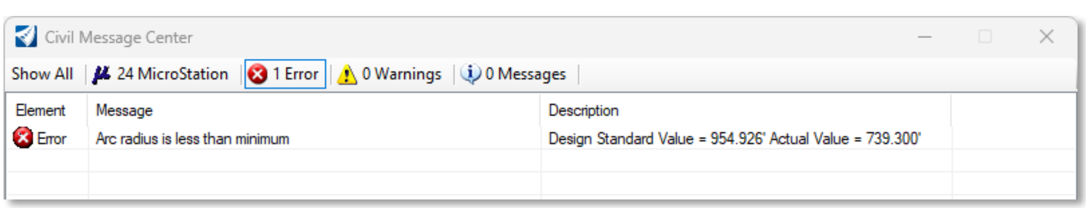
✓ Use the **Trim/Extend** and select the **Both** options to complete the process, issuing a left-click to accept the parameter.



The 7°45’ curve does not meet the currently selected 55 MPH Design Standard. A red X is displayed on the curve as shown on the right.

Design errors can be reviewed in the ***Civil Message Center*** dialog by selecting **Standards > Civil Message Center** from the **Geometry** tab.

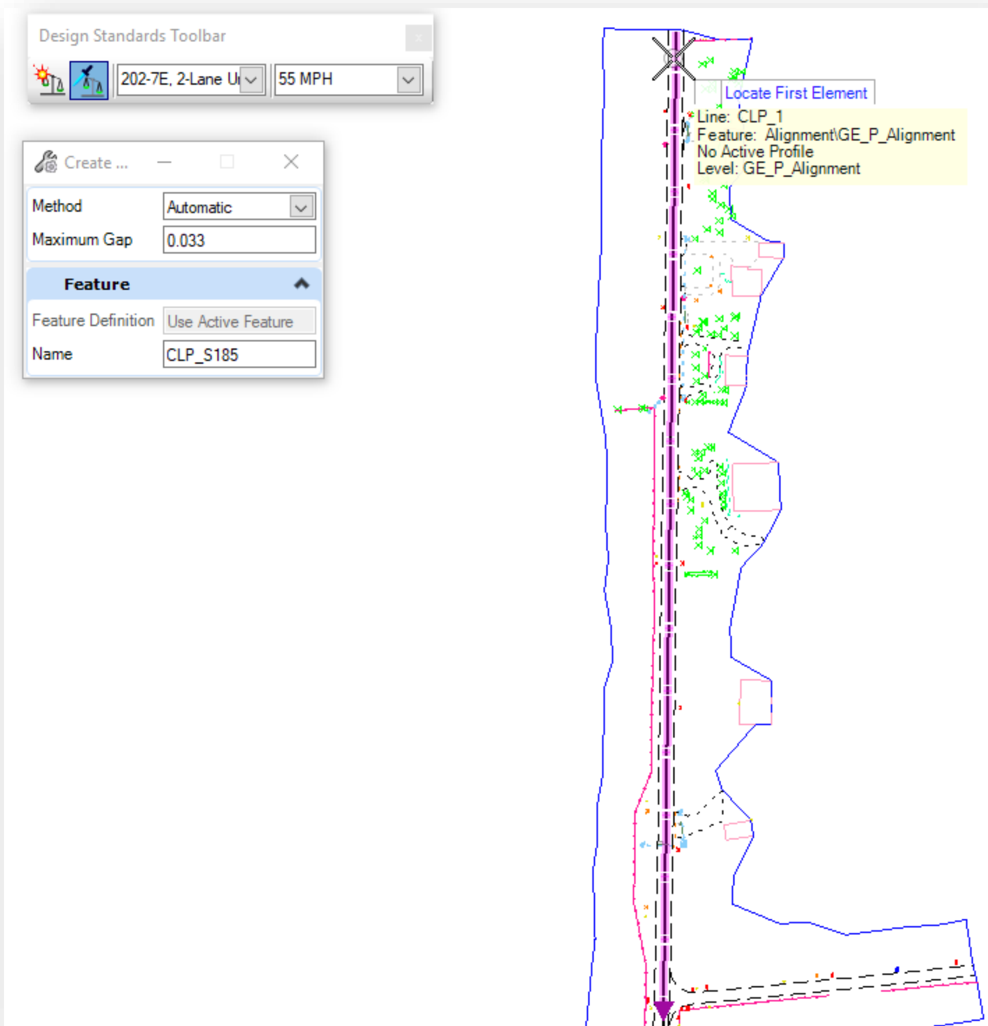
✓ Be sure to select the spiral-curve-spiral set to check the spiral lengths after the data is placed in the file. The Design Standard may have switched the length back to 205’. If the spirals were placed with a length of 205’, edit the rule for each spiral length to change the value to 250’



No action is required to continue designing the corridor model. This is strictly informational and can be ignored if the data is correct.

Next, create a complex chain from the individual elements that have been placed for the proposed centerline of S.R. 185.

- ✓ From the **Geometry** tab, choose the **Complex Geometry > Geometry > Complex By Elements** icon
  - Set the **Feature Definition** to the **Alignments > GE_P_Alignment**.
  - Key in the **Name** for the centerline element. See the **Location & Design Manual, Volume 3 – Highway Plans, Section 1204.3.6.2** for more information on naming geometry elements.
  - Using the **Automatic Method**, choose the first element to define the complex element. Note that the direction of the element is significant. When you move the cursor over the first element, an arrow is displayed indicating the direction the complex element will be defined.
  - When prompted to **Locate First Element**, select and accept the first element at the north end of the alignment as shown below.

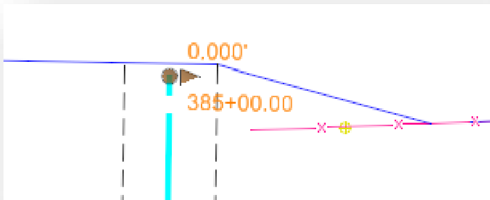


The chain is stored with a default beginning station value of 0+00. Use the **Start Station** tool to define the beginning station for the newly created alignment as detailed below.

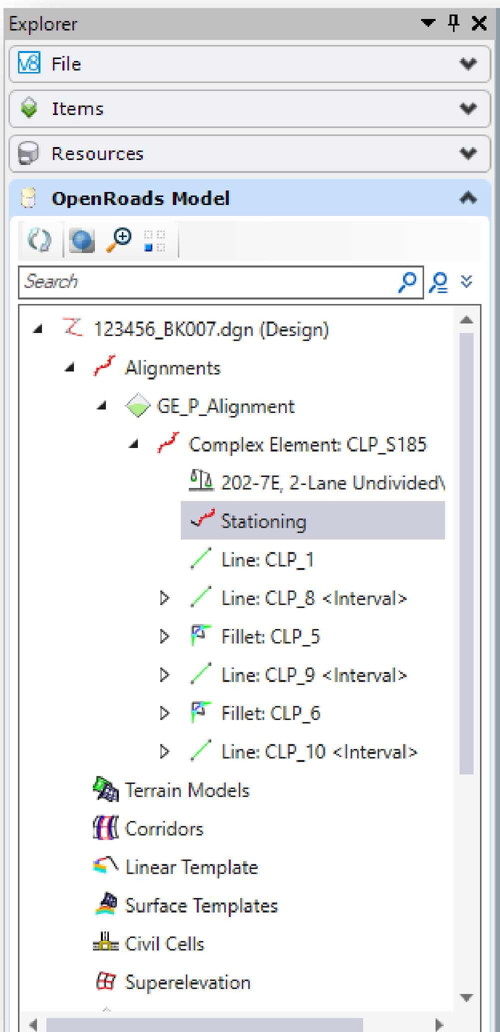
- ✓ From the **Geometry** tab, choose **Modify > Start Station**

The **Start Distance** parameter defines the location along the selected alignment where the **Start Station** value is applied. The **Start Distance** should be 0 and the **Start Station** is 385+00.00

The station value can be reviewed or edited by using the MicroStation Element Selection tool to select the complex element. That station rules are displayed as the example below.

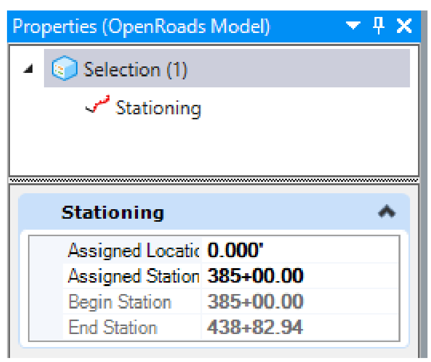


The parameters for the alignment can be reviewed using the *Project Explorer* dialog's **OpenRoads Model** group.

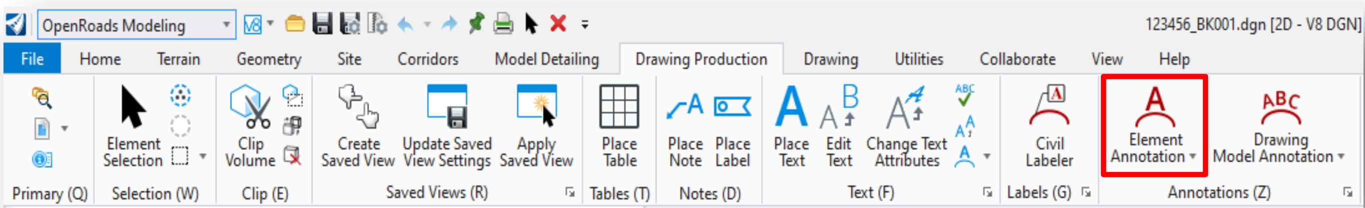


Various parameters can be reviewed and/or edited in *Project Explorer* by right-clicking on an item and selecting **Properties**.

For example, the **Assigned Station** value can be reviewed and edited in the ***Properties*** dialog as shown below.



✓ Annotate the alignment using the **Drawing Production** tab **Element Annotation** command.



✓ From the **Drawing Production** tab, change the **Drawing Scale** value to **1:50**. Note how the annotation is scaled, including station tics and leader lines.

This completes Exercise A

✓ Save settings and exit OpenRoads Designer. Be sure to check the file back into ProjectWise.

Exercise B - St. Peter Road

Take the following steps to define the centerline for St. Peter Road:

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.

✓ Browse to the **400-Engineering\Roadway\Basemaps** folder.

✓ Open the geometry basemap for St. Peter Road:

400-Engineering\Roadway\Basemaps\123456_BK002.dgn

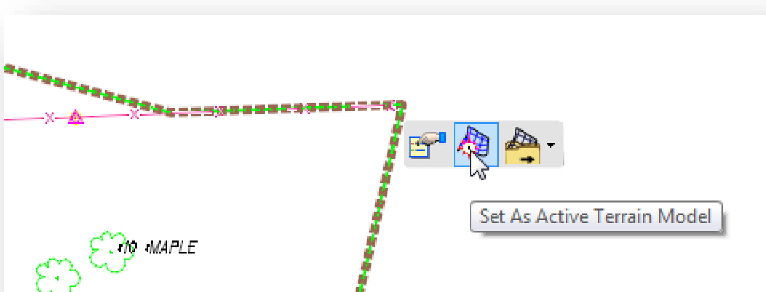
✓ Attach the survey information as a reference file using a **Nest Depth** value of **0**. The survey information is in the following design file:

300-Survey\Basemaps\123456_FB001.dgn

✓ Turn on/off levels for the survey information as necessary. Start by turning off the levels listed below, and any other levels that you may not want to see while working on the geometry for the project.

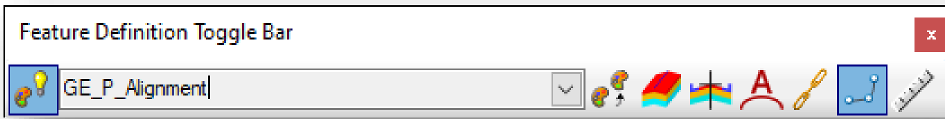
- PV_X_COP
- SV_X_Break_Lines
- SV_X_Random_Point
- SV_X_Shot_Marker
- SV_X_Text_Comment
- SV_X_Text_Elevation
- SV_X_Text_Name

✓ Select the terrain model graphics from the reference attachment and set it as the **Active Terrain** (select the terrain boundary and then hover the mouse over the boundary to get the pop-up toolbar seen below, a long right click on the boundary will bring up a context menu where the Set of Active Terrain Model button can be found as well).



Now we are ready to define the centerline alignment for St. Peter Road.

✓ From the **Geometry** tab, choose **Standards > Features Definition Toolbar**. Set the active feature to **Alignments > GE_P_Alignment**. Toggle on the **Use Active Feature Definition** icon.

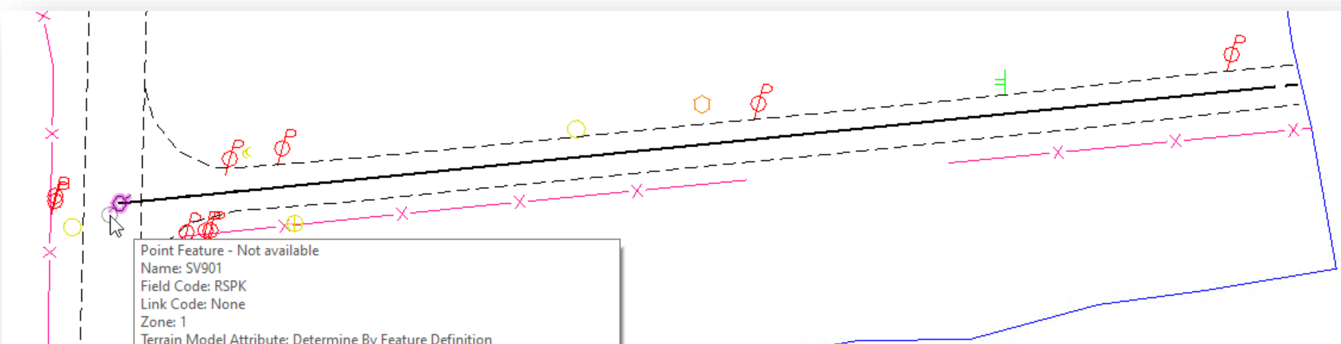


✓ From the **Geometry** tab, choose **Lines > Line Between Points**.

The survey crew has provided coordinates for the start of St. Peter Road. Use **CivilAccudraw** with the **XY ordinate** to enter the first point for the **Line Between Points** command with the following coordinates:

Easting = 1414827.690
Northing = 805922.456

The second point will end at survey point SV901. Use the MicroStation Origin Snap to snap to the survey point to end the line.

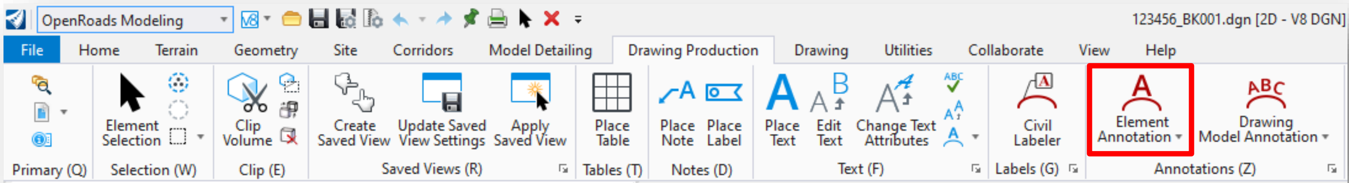


Use the **Start Station** tool to define the beginning station for the line as detailed below.

✓ From the **Geometry** tab, choose **Modify > Start Station**

The **Start Distance** parameter defines the location along the selected alignment where the **Start Station** value is applied. The **Start Distance** should be 0 and the **Start Station** is 15+00.00

- ✓ Annotate the alignment using the **Drawing Production** tab **Element Annotation** command.

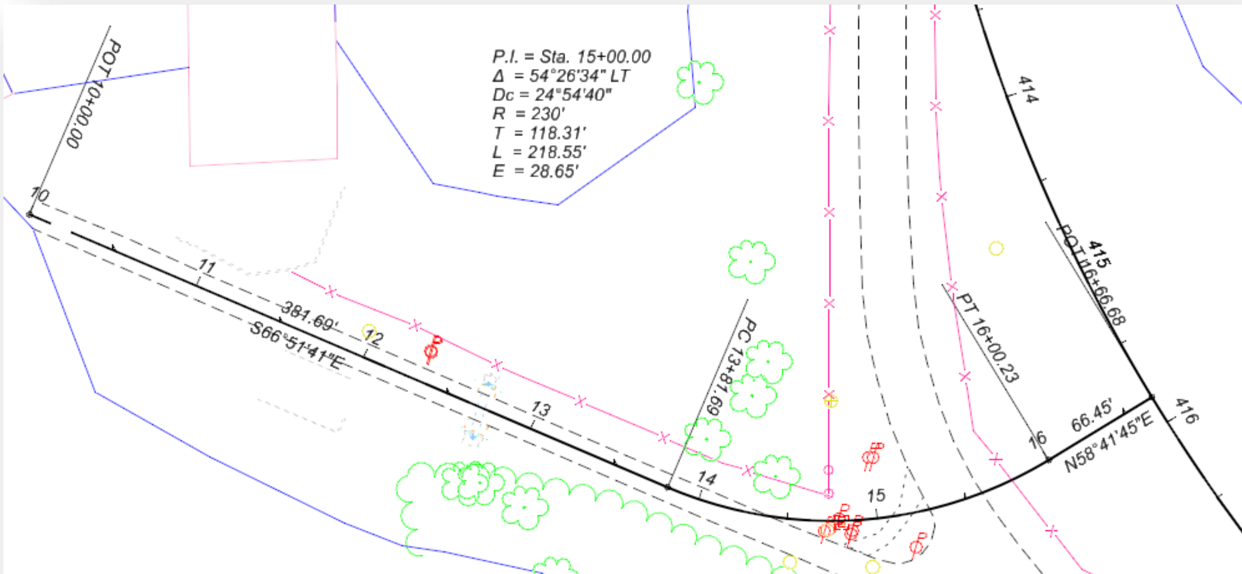


This completes Exercise B

- ✓ Save settings and exit OpenRoads Designer. Be sure to check the file back into ProjectWise.

Exercise C - Jameson Road Relocation

Jameson Road will be relocated by defining a new horizontal curve using the existing centerline as the back tangent and defining the ahead tangent that is perpendicular to the proposed S.R. 185 alignment.



Take the following steps to define the centerline for Jameson Road:

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.

✓ Browse to the **400-Engineering\Roadway\Basemaps** folder.

✓ Open the geometry basemap for Jameson Road:

400-Engineering\Roadway\Basemaps\123456_BK003.dgn

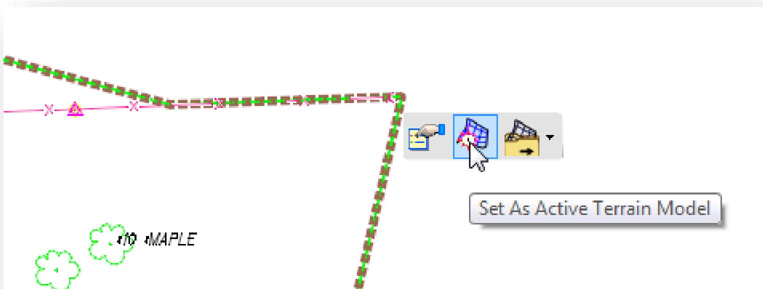
✓ Attach the survey information as a reference file using a **Nest Depth** value of **0**. The survey information is in the following design file:

300-Survey\Basemaps\123456_FB001.dgn

✓ Turn on/off levels for the survey information as necessary. Start by turning off the levels listed below, and any other levels that you may not want to see while working on the geometry for the project.

- PV_X_COP
- SV_X_Break_Lines
- SV_X_Random_Point
- SV_X_Shot_Marker
- SV_X_Text_Comment
- SV_X_Text_Elevation
- SV_X_Text_Name

✓ Select the terrain model graphics from the reference attachment and set it as the **Active Terrain** (select the terrain boundary and then hover the mouse over the boundary to get the pop-up toolbar seen below, a long right click on the boundary will bring up a context menu where the Set of Active Terrain Model button can be found as well).



✓ Attach the geometry basemap containing the centerline of proposed S.R. 185 as a reference using a **Nest Depth** value of **0**. The survey information is in the following design file:

400-Roadway\Basemaps\123456_BK001.dgn

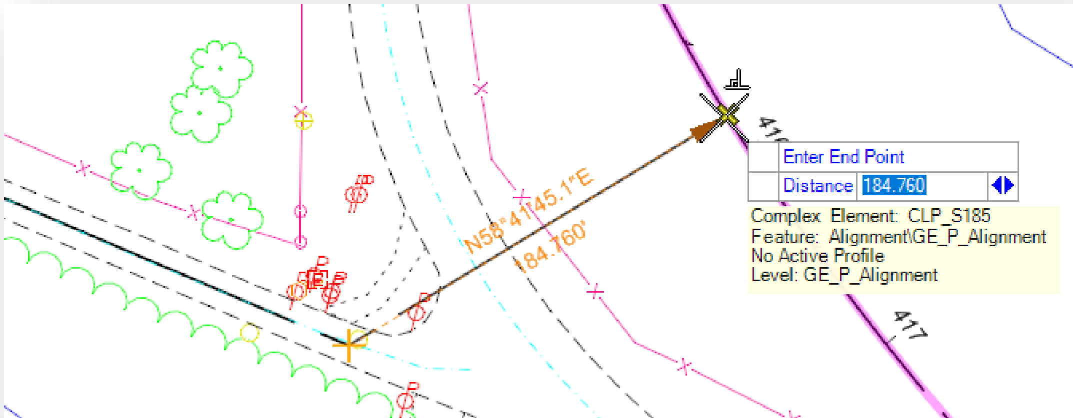
Now we are ready to define the centerline alignment for the Jameson Road relocation.

✓ From the **Geometry** tab, choose **Standards > Features Definition Toolbar**. Set the active feature to **Alignments > GE_P_Alignment**. Toggle on the **Use Active Feature Definition** icon.

✓ From the **Geometry** tab, choose **Lines > Line Between Points**. Draw the first tangent line for proposed Jameson Road using CivilAccudraw to enter the following coordinates.

Point 1	Easting = 1413832.052 Northing = 804471.371
Point 2	Easting = 1414291.830 Northing = 804274.892

✓ Place the second tangent line from the end of the first using the MicroStation Perpendicular Snap to place the line perpendicular to the proposed S.R. 185 alignment, as shown below.



✓ From the **Geometry** tab, select **Arcs > Arc Between Elements > Simple Arc** to place an arc with a 230' radius between the two tangent lines. Be sure to trim both elements.

✓ From the **Geometry** tab, select **Complex Geometry > Complex by Element** to create a complex element for proposed Jameson Road. Use a **Name** value of **CLP_JAMESON**

✓ From the **Geometry** tab, choose **Modify > Start Station**. Define the beginning station for the alignment with a **Start Distance** value of **0** and a **Start Station** value of **10+00**

✓ Annotate the alignment by selecting the **Drawing Production** tab **Element Annotation** command.

This completes Exercise C

✓ Save settings and exit OpenRoads Designer. Be sure to check the file back into ProjectWise.

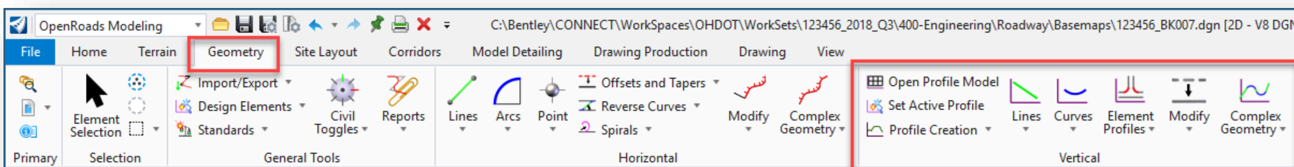
Section 400 Vertical Geometry

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.

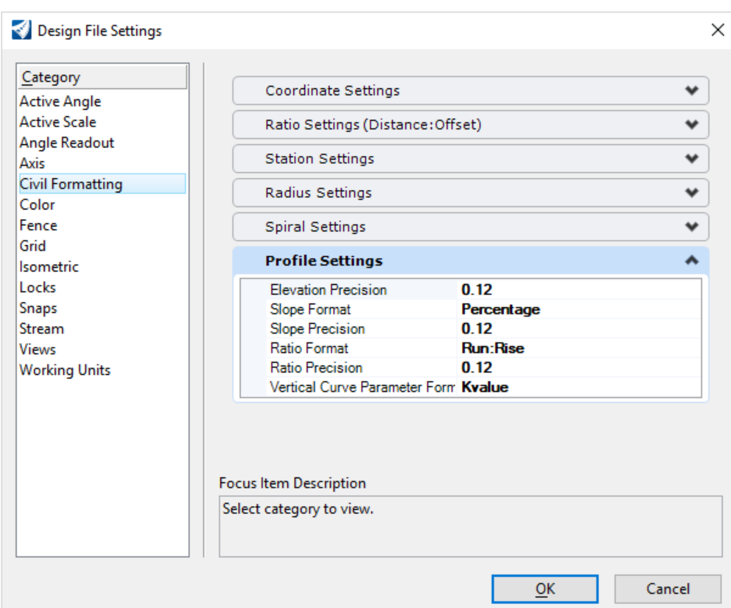
Vertical geometry tools are accessed from the **Geometry** tab of the **OpenRoads Modeling** workflow, as shown below.



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.

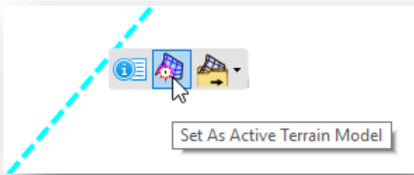
Profile Settings

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 ODOT seed files.

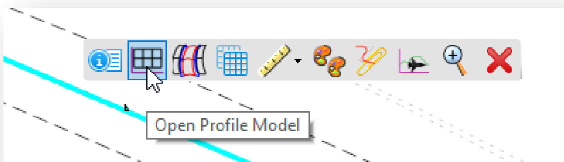


Existing Ground Profiles

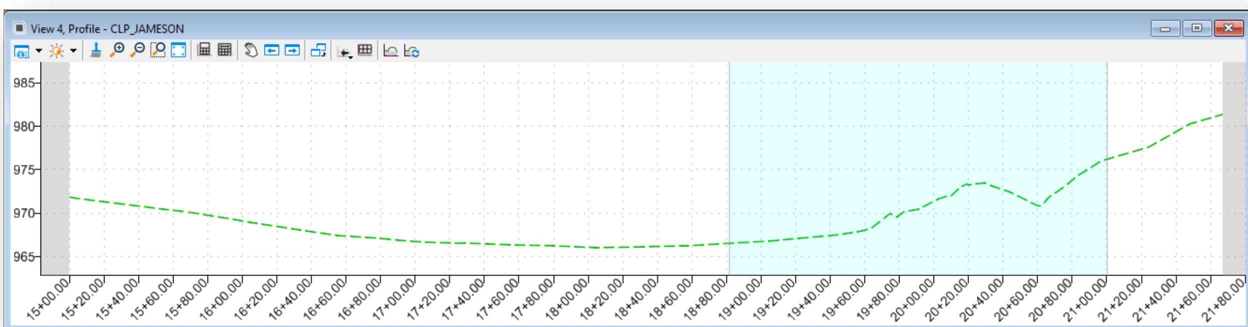
Existing ground profiles are extracted along the selected civil geometry element from the active terrain model. The active terrain model is set by selecting the terrain element and choosing the **Set As Active Terrain Model** icon as shown below.



To display 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.

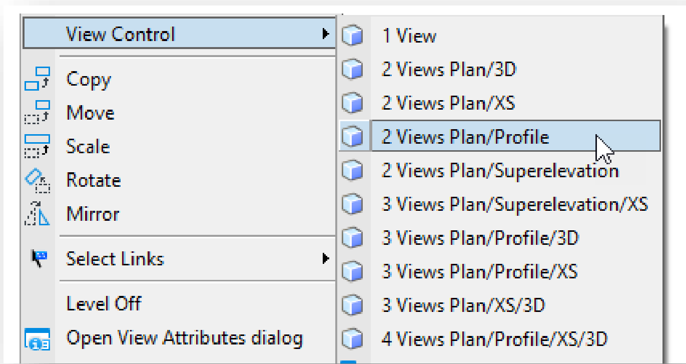


You are prompted to select or open a view window to use for the profile display. Open a new view window and then left click in the view window to display the profile. An example is shown below.

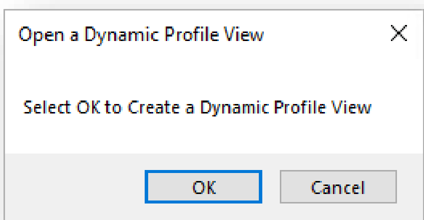


Note: The profile is displayed in the profile view for review and editing; however, the profile is not written to the design file. The selected view window is used as a temporary display of all the profiles associated with the selected alignment. The selected view window is not specifically associated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

Alternately, a profile can be displayed by holding down the left mouse button in the view window until the pop-up menu shown below is opened. Select the **2 Views Plan/Profile** option.



You are prompted to create a dynamic profile view as shown below. Select **OK** to continue.



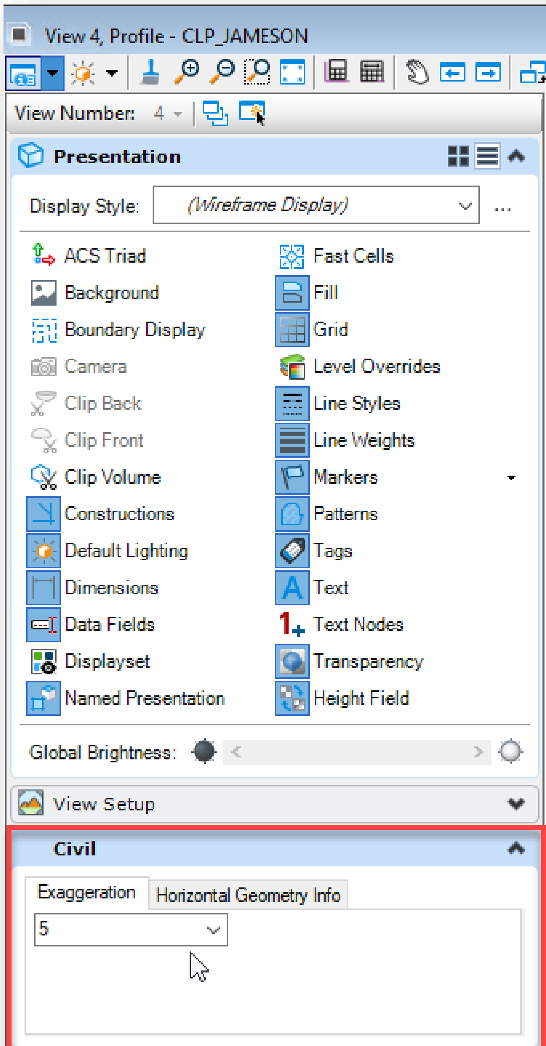
After selecting **OK**, the View 4 window is opened to display the profile graphics.

When prompted to select the plan element that will be used to display the profile. You are next prompted to **Select or Open View**. Left click in the newly opened View 4 window to select this view to display the profile.

The Profile Window

View Attributes

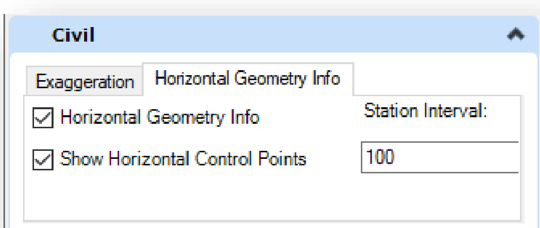
The View Attributes for a profile view contain an additional parameter to control the vertical exaggeration of the profile display, as shown below.



To change the profile display exaggeration factor, select the desired exaggeration factor from the drop-down list.

Tip: With the cursor in the profile view, holding down the shift key while scrolling the mouse wheel will dynamically change the profile exaggeration.

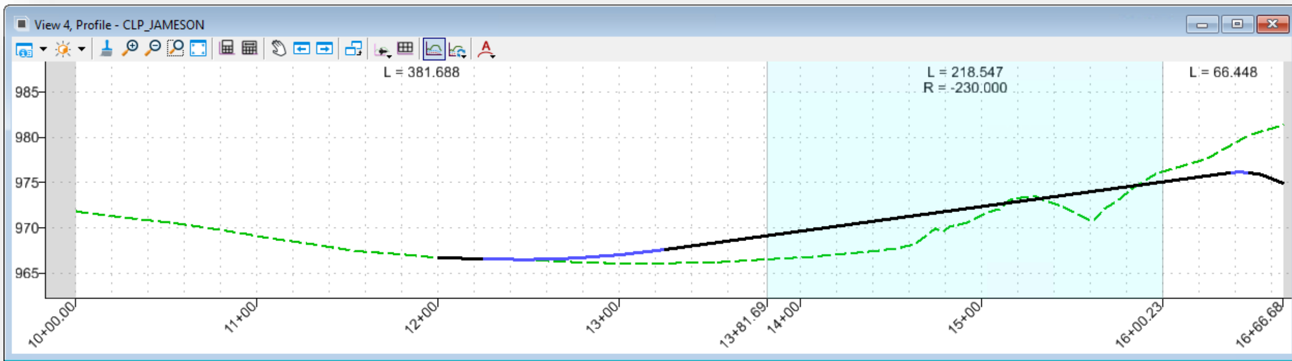
Version 10.10 adds additional controls for the profile view attributes under the **Horizontal Geometry Info** tab shown below.



The **Horizontal Geometry Info** option toggles on the display of the geometry tangent length, and length and radius for horizontal curves. This information is displayed across the top of the profile window as shown below.

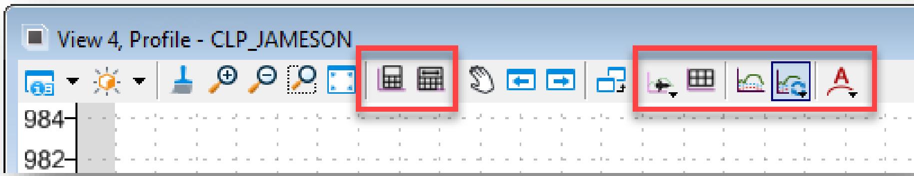
The **Show Horizontal Control Points** toggles on the display of the station of any key-points in the horizontal alignment across the bottom of the profile view.

The **Station interval** allows keying in an incremental value to show stations across the bottom of the profile window. A value of 100 is shown below.



Profile Commands

The **Profile** view window contains command icons specific to a profile view window as shown below.

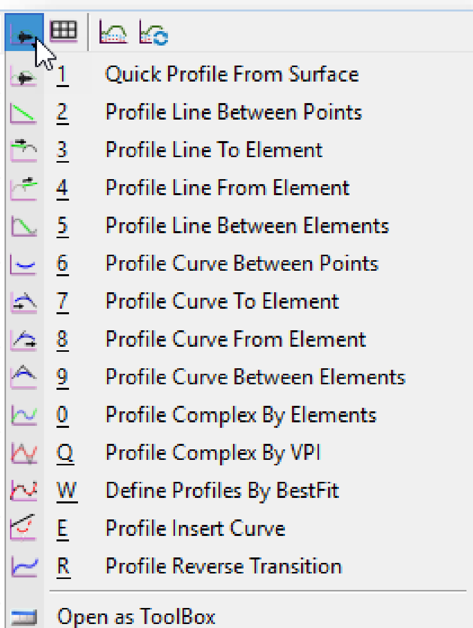


The following commands are available:

- Fit Plan Extents
- Fit Profile Elements
- Vertical Geometry Tools
- Attach All Profile Views
- Create 3d Cut
- Refresh 3d Cut
- Dynamic Annotation: Annotate Element

Vertical Geometry Tools

The vertical geometry tools can be accessed from the profile view's **Vertical Geometry Tools** icon as shown below.



See the online help for information on the individual **Vertical Geometry** commands.

These commands are also available in the OpenRoads ribbon menu by selecting **OpenRoads Modeling > Geometry**

Create 3d Cut/Refresh 3d Cut

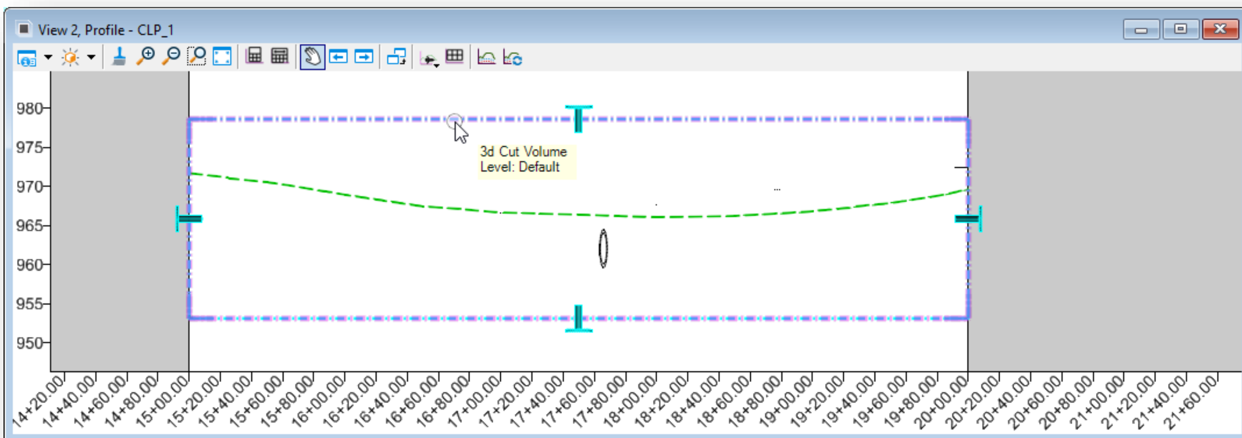
3D elements that cross the currently displayed profile, such as underground utilities, another corridor or bridge that passes over the active profile, can be displayed on the profile by choosing the **Create 3d Cut** or **Refresh 3d Cut** commands.

When the 3d profile is created, two Placement Methods are available:

- Corners - The **Corners** option allows the user to define a bounding box that defines both the horizontal and vertical extent of the 3D cut.

- Full Profile - The **Full Profile** options automatically sets the extents of the 3d cut.

In either case, a bounding box is drawn in the profile view line the example shown below.



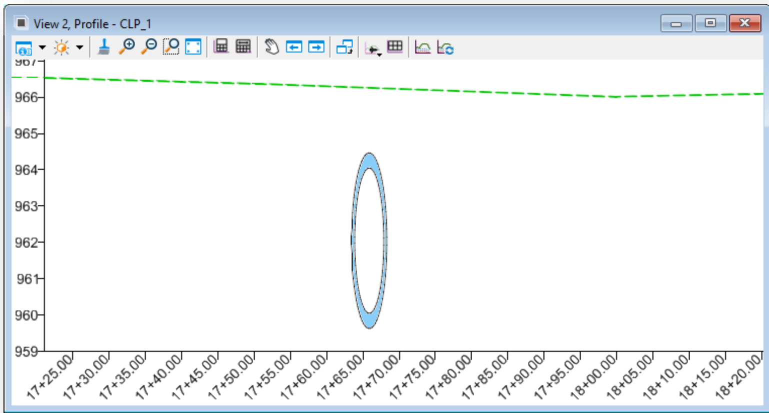
When the element is selected, four handles appear that allow adjusting the extents of the bounding area, as shown above.

Delete the bounding box to remove the 3d cut from the view.

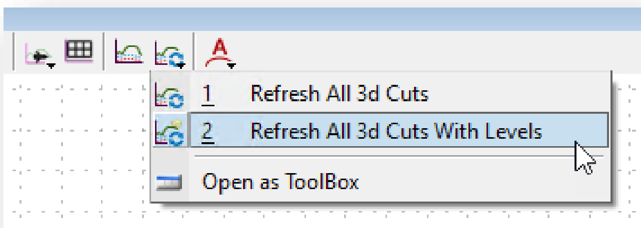
Use the MicroStation **Select Element** to select the bounding box. Allow the cursor to rest over the bounding element until the pop-up menu appears. Two options are available: **Properties** and **Delete**. Select Properties to access the parameters shown below.

Clip Start Station	True
Start Station	15+00.00
Clip Top Elevation	True
Top Elevation	978.580
Clip End Station	True
End Station	20+00.00
Clip Bottom Elevation	True
Bottom Elevation	953.112
Cut Offset	0.000'

The example below shows an existing 48” pipe that passes under the profile.

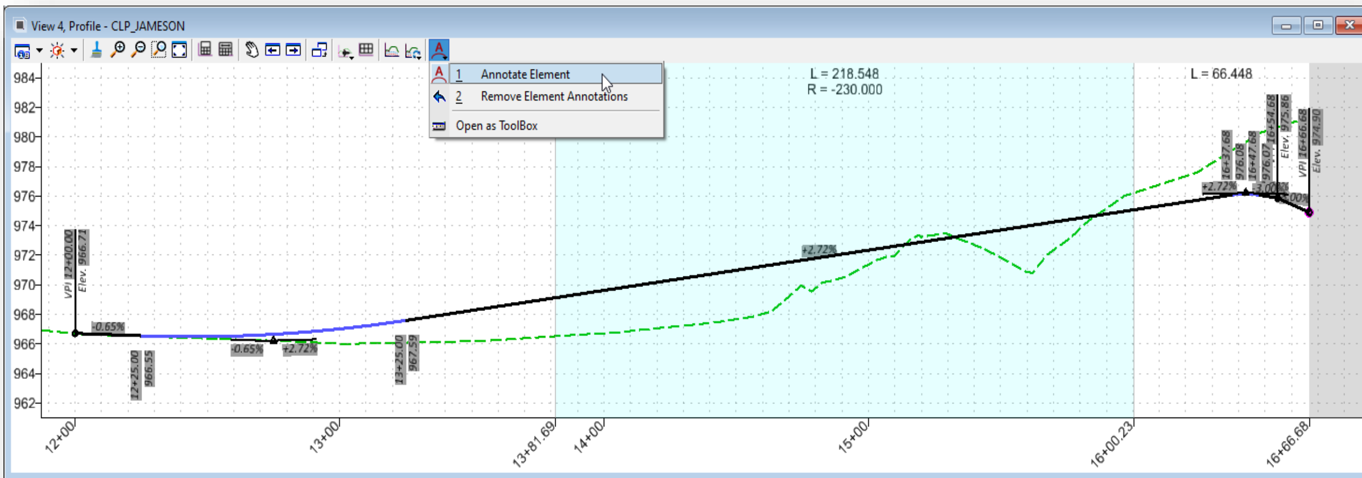


The 3d Cut graphics are a static representation of the elements in the 3d model at the time the 3d cut graphics were generated. To update the 3d cut graphics, the following commands are available in the Profile View window.



Annotate Profile Element

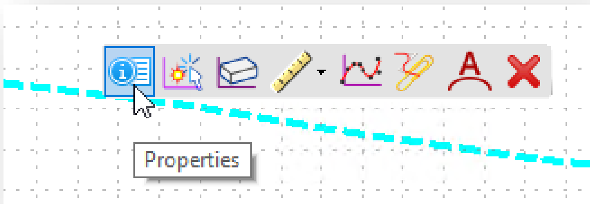
OpenRoads 10.10 adds the ability to annotate a profile directly in the profile window. Commands to **Annotate Element** and **Remove Element Annotations** are available as shown below.



Notes:
These annotations can be useful when working in the profile view, however, the same annotations will be placed in the drawing models when sheets are created.

Profile Pop-Up Menu

Selecting a profile graphic and allowing the cursor to rest on the element reveals a pop-up menu with commonly used commands as shown below.

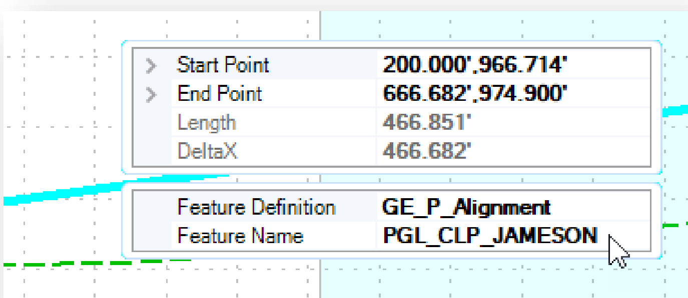


The following commands are available. See the online help for more information not covered in this document

- Properties
- Set as Active profile
- Rules
- Create Best Fit Profile
- Profile Report
- Annotate Element
- Delete

Properties

Select the **Properties** item to access the menu shown below. This is commonly used to edit the **Feature Definition** or the **Feature Name** for the selected profile.



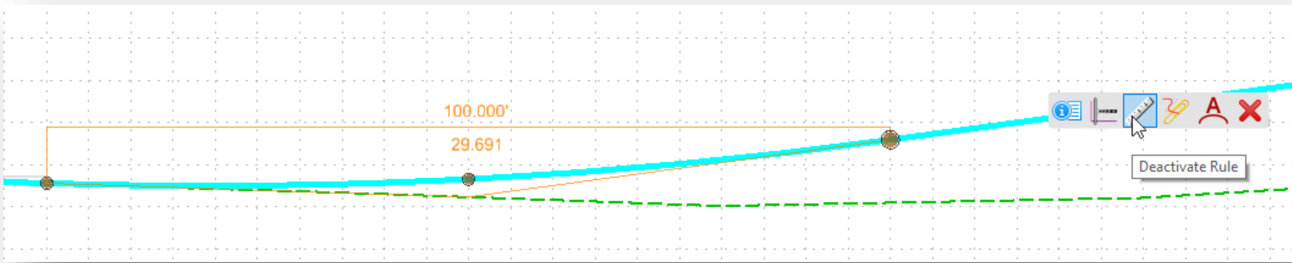
Set as Active Profile

This option is used to set the selected profile as the active profile. Once a profile is set active, it is used as the default profile for subsequent actions such as the corridor modeling process.

Rules

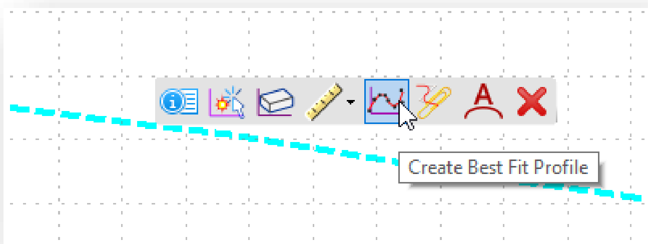
Like the Horizontal Geometry, profile elements have a set of rules that define the element parameters. In the example below, the rules for a vertical curve are shown.

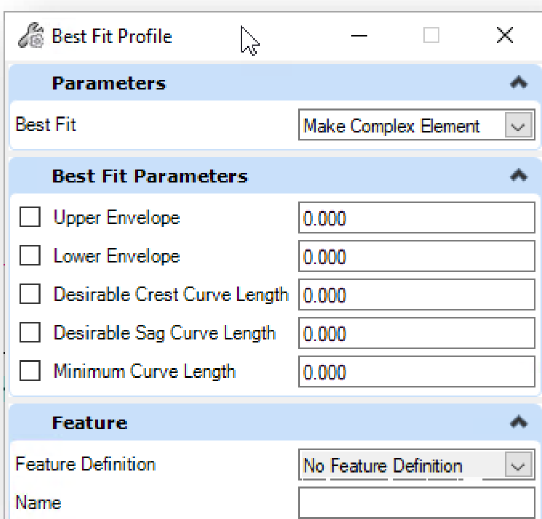
The rules may be activated (unlocked) or deactivated (locked) from the pop-up menu for the selected profile like the example below.



Create Best Fit Profile

For existing alignments, the pop-up menu includes the option to create a best fit profile as shown below.



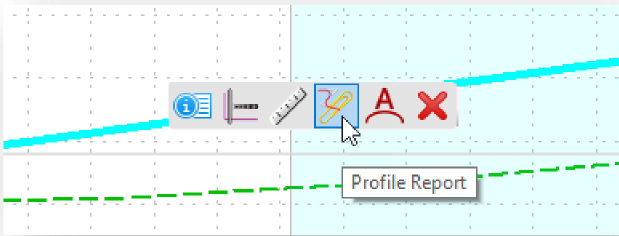


The dialog shown at left is opened.

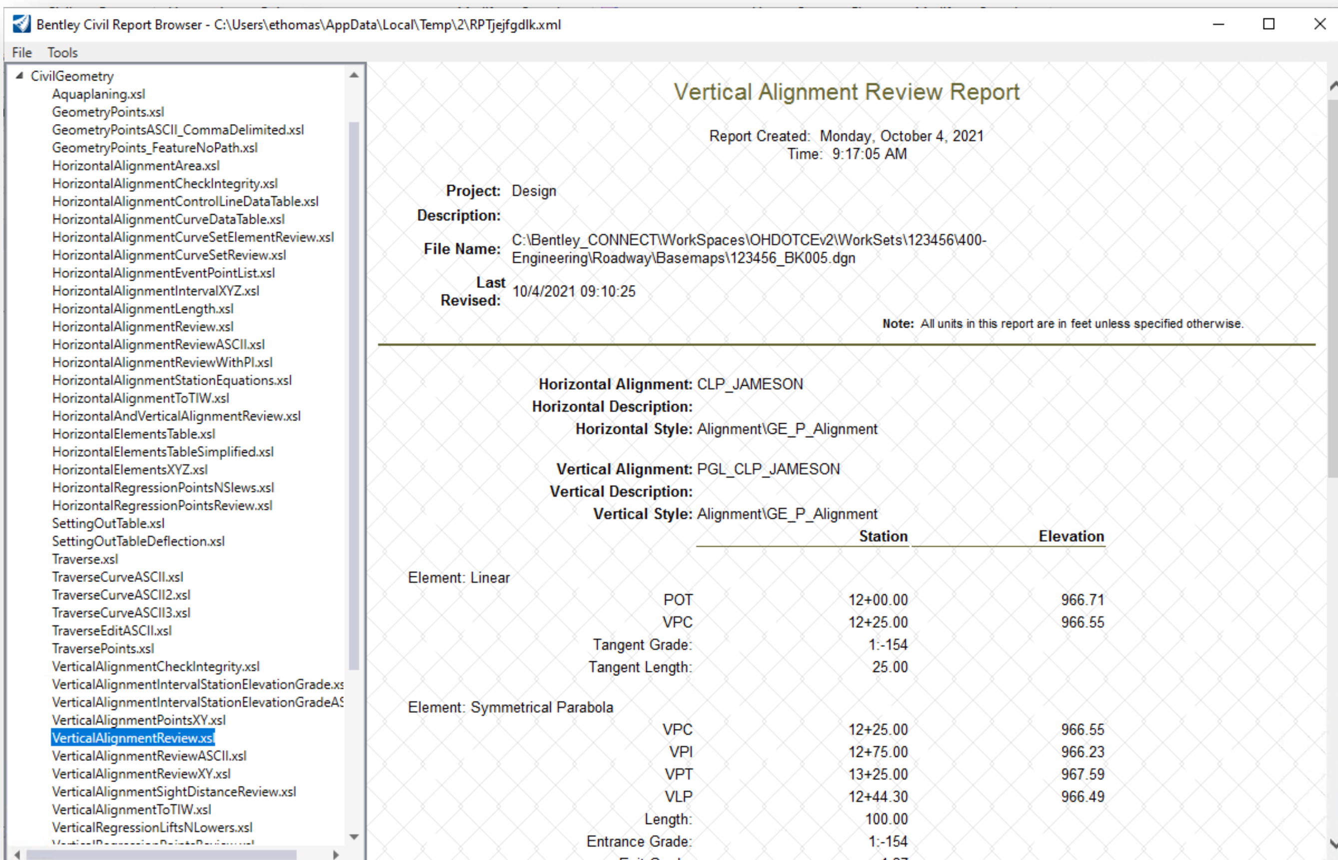
Various parameters are available to refine the best fit profile that is to be created. See the online help for more information.

Profile Report

Various vertical alignment reports can be generated by selecting the profile using the **Home > Element Selection** tool, and then choosing the **Profile Report** icon from the pop-up menu as shown below.



The default **Vertical Alignment Review Report** is shown below.

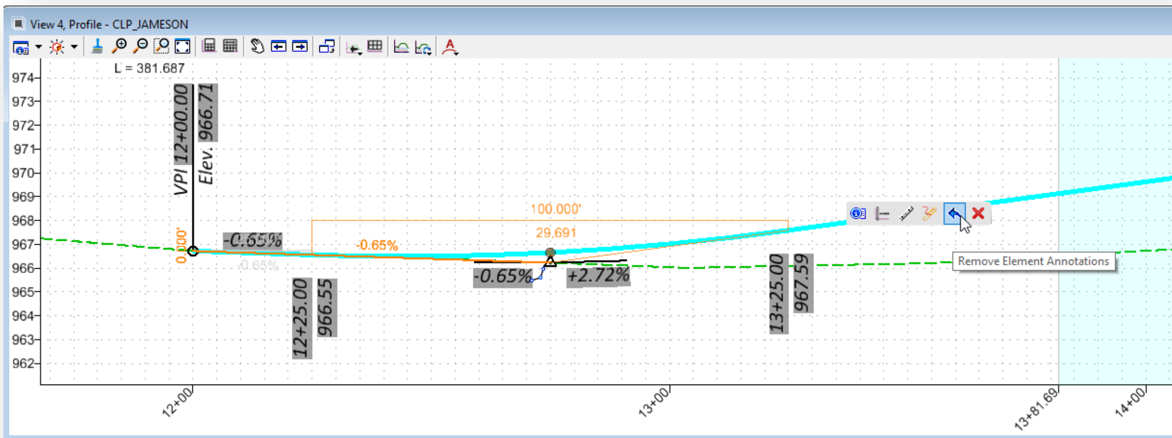


Various report formats can be selected from the list at the left of the dialog.

Annotate Element

The pop-up menu also provides the profile annotation commands, documented previously in this document, that are available from the profile window menu.

The annotations can be removed from a profile that has been previously annotated by choosing the **Remove Element Annotation** option, shown below.



Profile Feature Definitions

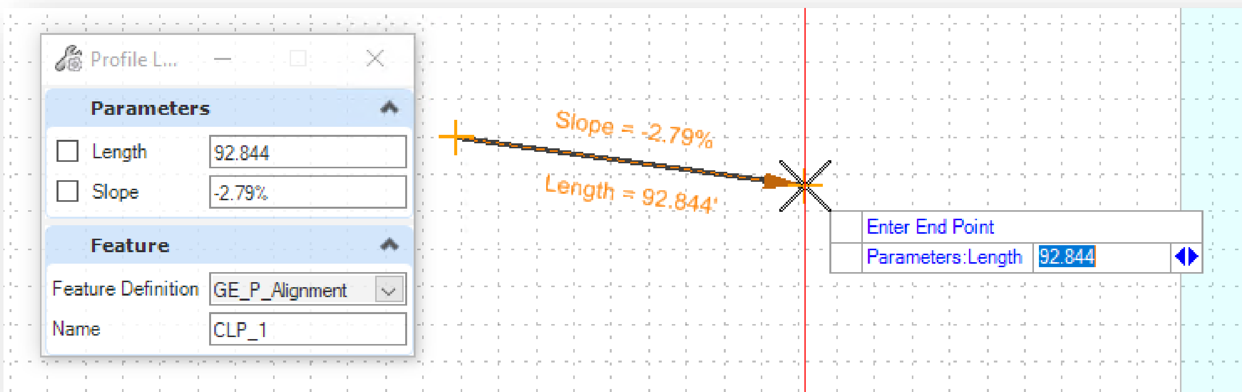
The symbology of vertical geometry is defined by Feature Definitions. The vertical geometry feature definitions define the symbology for the profile lines and vertical curves as well as the annotation parameters.

Feature definitions for vertical alignments are typically defined as follows:

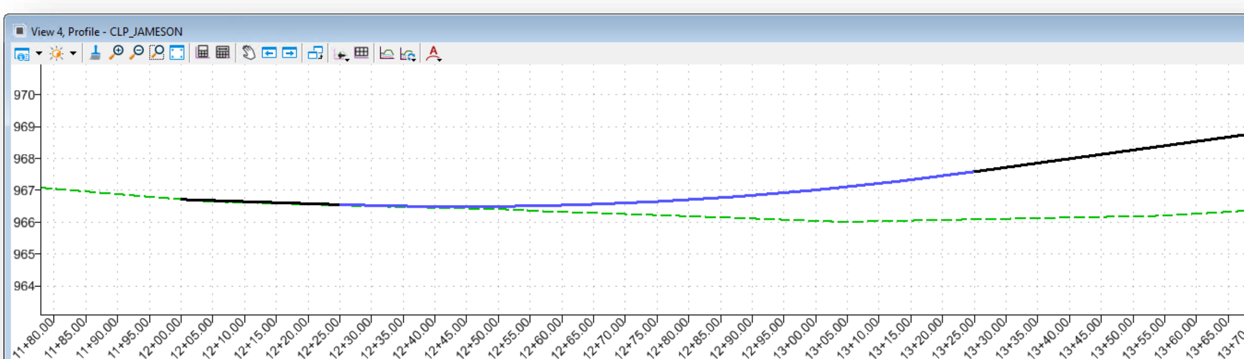
- Existing vertical alignments inherit their display parameters from the feature definition that has been assigned to the terrain model that was used to extract the profile.

- 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.

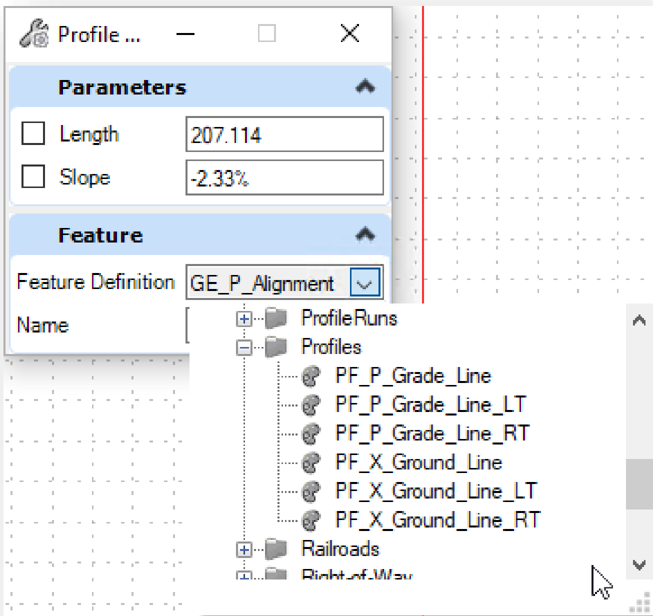
In the example below, a proposed profile line is drawn using the **Profile Line Between Points** command. The **Feature Definition** option is set to use the **Alignment > GE_P_Alignment** feature. This sets the symbology for the vertical geometry since the alignment feature definition also includes the profile feature parameters.



The feature definition is defined to differentiate between the tangent lines and vertical curves. Proposed profiles are defined to display the vertical curves with color 58, and tangent lines with color 0, as shown below.



There are cases where the user may need to display the profile information using different symbology other than the parameters defined with the horizontal geometry feature definition. The OHDOT Workspace includes additional features for profiles in the **Linear > Profiles** category as shown below.



Linear > Profiles > PF_P_Grade_Line
This option is typically used for utility profiles with no annotation definitions for vertical curve data.

Linear > Profiles > PF_P_Grade_Line_LT
Linear > Profiles > PF_P_Grade_Line_RT
Linear > Profiles > PF_X_Grade_Line_LT
Linear > Profiles > PF_X_Grade_Line_RT

These features are used for divided highways to draw the profile graphics on separate levels with different symbology's. This allows the user to turn on/off the levels for a specific left or right profile while designing the proposed vertical geometry with less clutter onscreen.

There are also profile definitions under the **Linear > Profile Runs** category. These features are used with Drainage and Utilities functions and are not used for vertical alignments.

Section 401 Vertical Geometry Exercises

S.R. 185 Proposed Profile

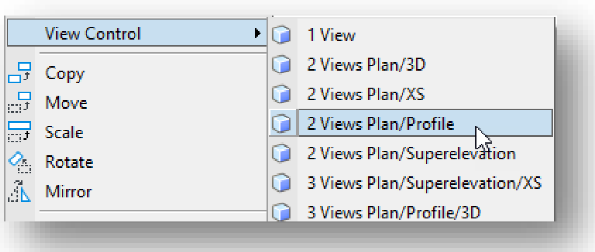
In this exercise, we'll define the proposed profile for State Route 185.

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Open the project with PID number **123456**.
- ✓ Browse to the **400-Engineering\Roadway\Basemaps** folder.
- ✓ Open the geometry basemap for State Route 185:

  400-Engineering\Roadway\Basemaps\123456_BK001.dgn

- ✓ Open a profile window for S.R. 185. This can be accomplished by selecting the alignment and then choosing the **Open Profile Model** icon from the pop-up menu, or by holding down the right-mouse button until the menu shown below appears and choosing the **2 Views Plan/Profile** option and then following the prompts.



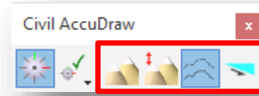
The existing ground is displayed in the profile view.

Use **Civil Accudraw** to store the tangent lines for the profile. Once these lines are stored, go back and store the vertical curves for the proposed alignment using the **Parabola Between Elements** command.

✓ Open the ***Civil AccuDraw*** dialog by choosing **Civil Toggles > Civil Accudraw** from the **Geometry** tab

- Toggle **Civil AccuDraw** on by choosing the first icon

With the profile window active, the **Civil AccuDraw** dialog will display icons relative to vertical geometry functions as shown



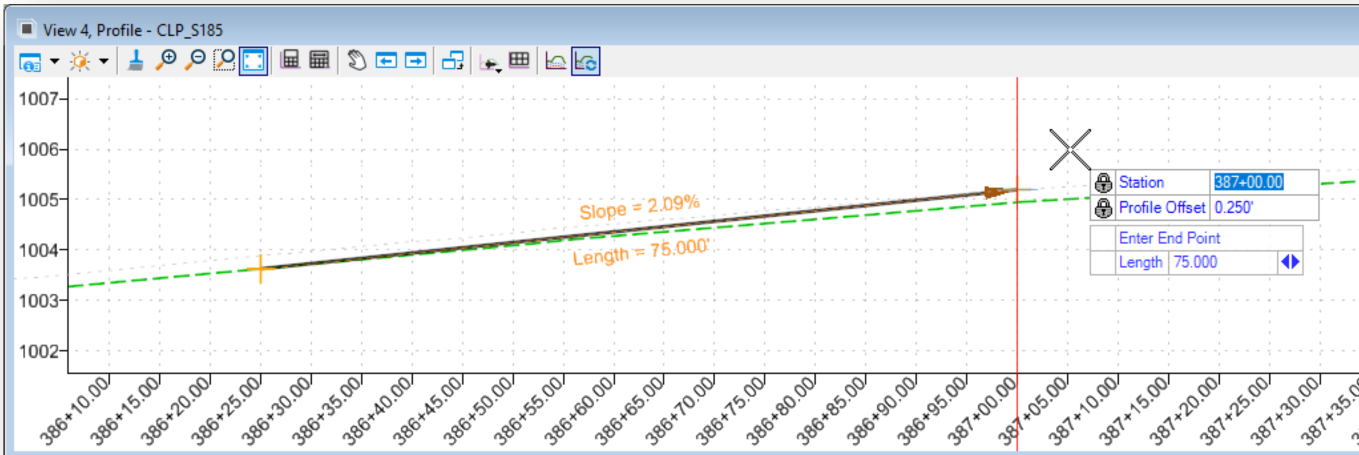
✓ Store the vertical tangent lined for the proposed profile for S.R. 185 using the values shown in the table below.

Use Civil Accudraw with the appropriate ordinate to facilitate entering the profile points.

VPI	Station	Civil Accudraw Ordinate	Offset/Elevation
1	386+25	ProfileOffset	Offset = 0
2	387+00	ProfileOffset	Offset = 0.25
3	392+00	Z	Elevation = 1014.07
4	398+00	Z	Elevation = 998.65
5	403+00	Z	Elevation = 993.65
6	410+00	Z	Elevation = 982.17
7	422+50	Z	Elevation = 966.61
8	426+00	Z	Elevation = 964.45
9	427+50	ProfileOffset	Offset = 0

✓ Start by choosing the **Lines > Profile Line Between Points** command to enter the first two VPI points listed above.

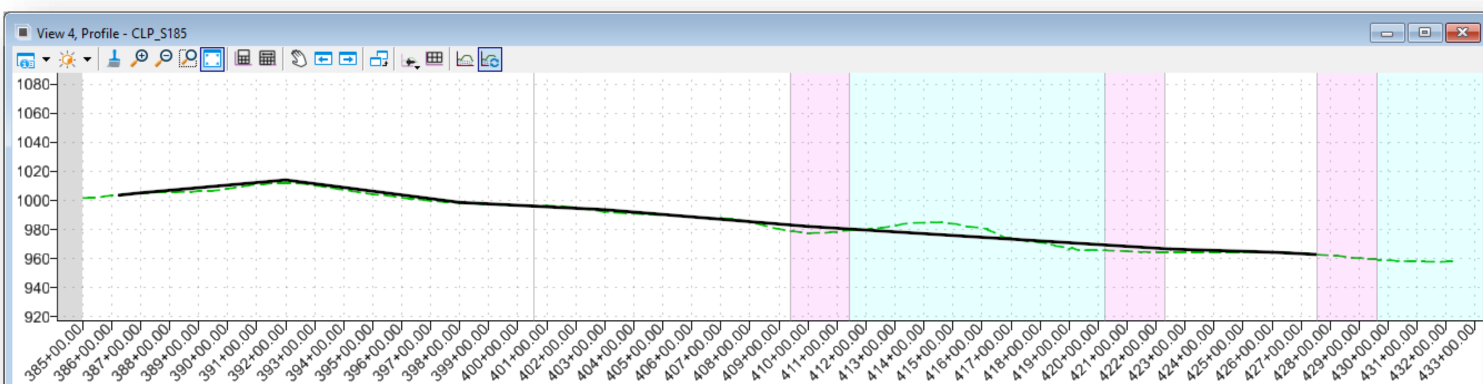
- Set **GE_P_Alignment** as the active **Feature Defintion**
- Toggle on the CivilAccudraw **ProfileOffset** ordinate
- With the **Profile Line Between Points** command active, tab so the **Profile Offset** field that is floating on the cursor has focus
- Type in the letter "**o**" to select the existing ground profile element as the origin for entering profile offset values.
- Select the existing ground profile. With the existing ground profile active, the cursor station and offset values will change relative to the cursor position in relation to the profile
- Key in the values for the first VPI point in the table above to enter the first point on the tangent line. Left-click to enter the first point
- Key in the values for the second VPI from the table above. Left-click to accept the second data point



- ✓ Take the following steps to place the second tangent line, defined by VPI 2 and VPI 3:
  - Start the line by snapping to the end of the first tangent line
  - Place the second point, defined by VPI 3 in the table above, by toggling the Civil AccuDraw ordinate to the Z mode
  - Enter the Station and Elevation for the point

- ✓ Continue placing the remaining tangent lines using the values in the table above and the approprite Civil AccuDraw ordinate.

  The tangent lines for the profile are defined like the example below

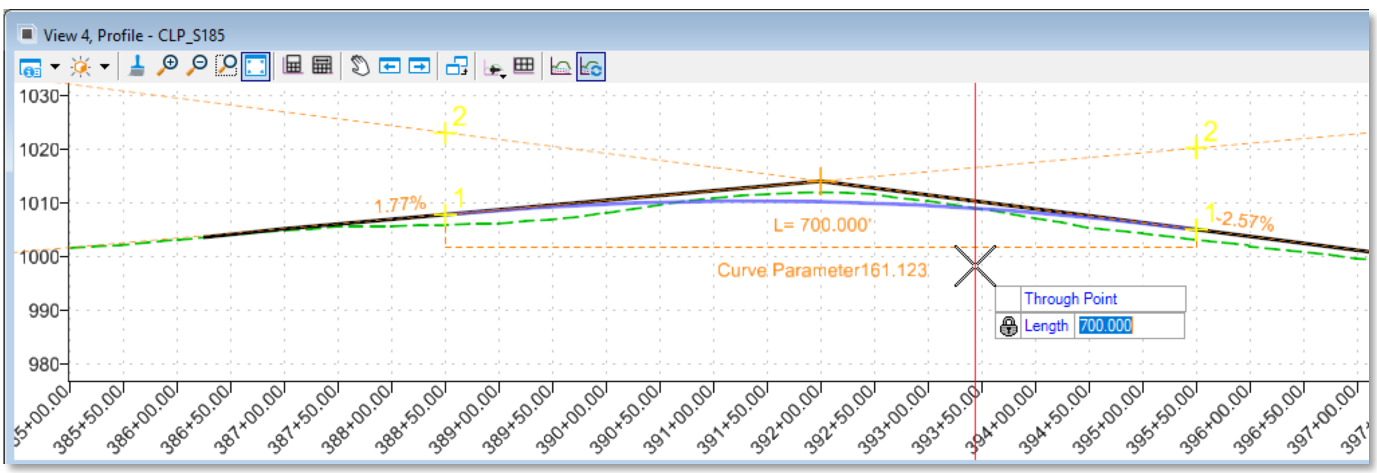


✓ After placing the tangent lines, from the **Geometry** tab, select **Curves > Profile Curve Between Elements > Parabola Between Elements** to place the vertical curves for the alignment

VPI	Station	Curve Length
1	386+25	
2	387+00	No Curve
3	392+00	700’
4	398+00	300’
5	403+00	300’
6	410+00	300’
7	422+50	450’
8	426+00	150’
9	427+50	

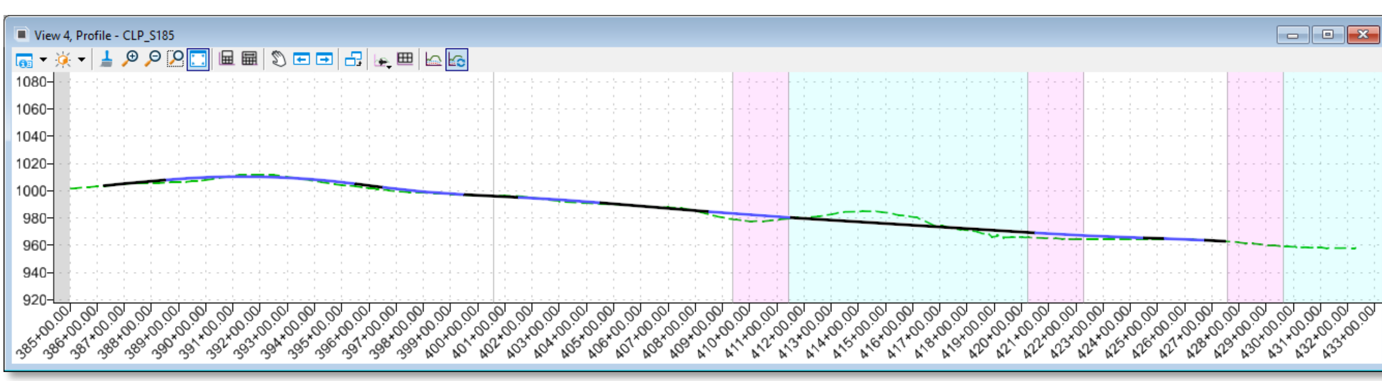
✓ Toggle off **Civil AccuDraw**. It is not needed to define the vertical curves

✓ Place the first curve at VPI 3 as shown below. Use the **Trim/Extend Both** option. Be sure to enter a left-click (data point) to accept the **Trim/Extend** parameter.



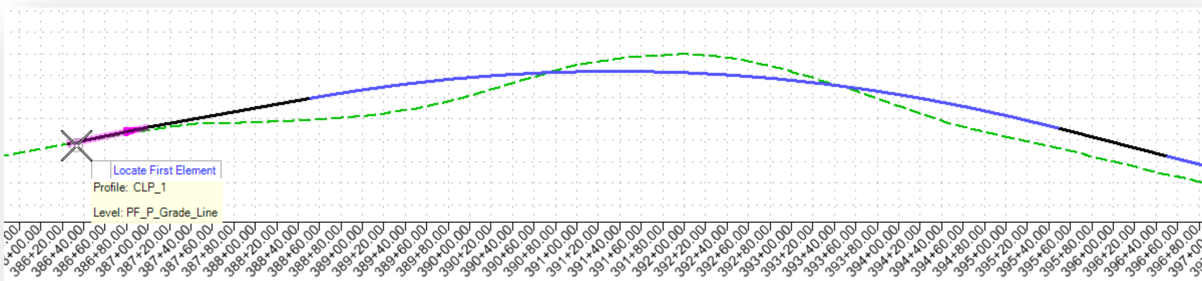
✓ Continue to place the remaing vertical curves

The vertical curves are defined like the example below



Next create a complex element for the vertical profile.

✓ From the **Geometry** tab, select **Complex Geometry > Profile Complex By Elements**
  - Set the **Feature Definition** to **GE_P_Alignment**
  - Define the **Name** as **PGL_CLP_S185** (See the **<u>Location & Design Manual, Volume 3 - Highway Plans</u>**, **Section 1204.3.6.2** for standards geometry element names.)
  - Using the **Automatic Method**, choose the first tangent element to define the complex chain



  - Left-click to accept the solution

Take the following steps to define the proposed profile as the active profile:

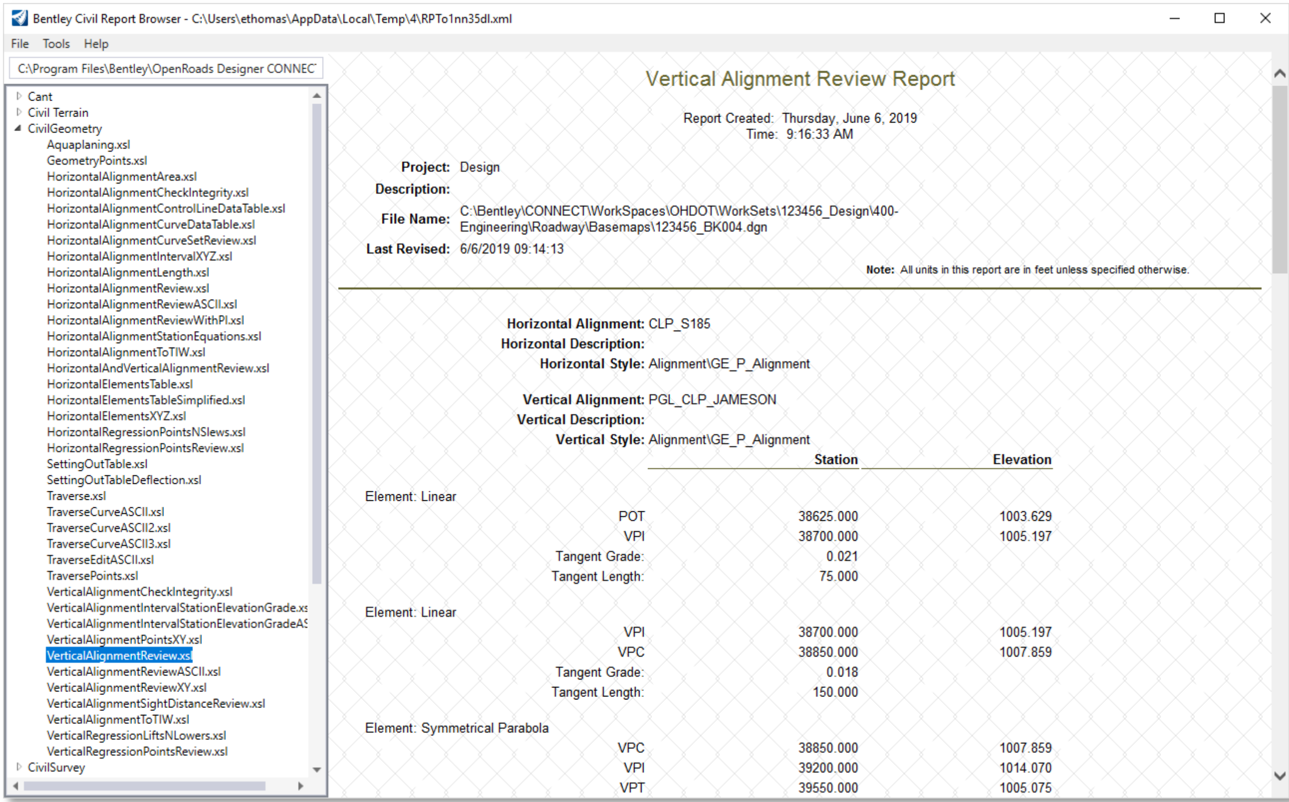
✓ Use the MicroStation **Select Element** tool to select the proposed profile line
  - Allow the cursor to rest on the selected element until the pop-up menu appears
  - Choose the **Set As Active Profile** icon

Review the profile by generating a report:

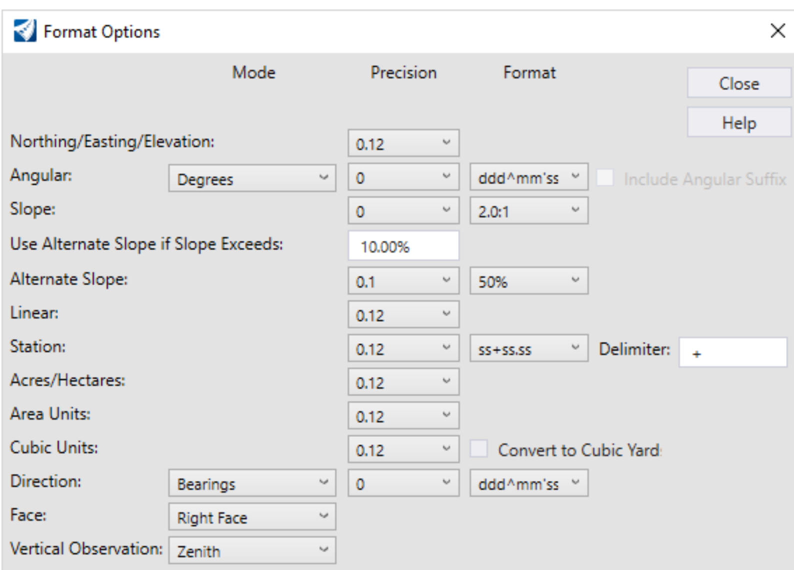
✓ Use the MicroStation **Select Element** tool to select the proposed profile line
✓ Allow the cursor to rest on the selected element until the pop-up menu appears
✓ Choose the **Profile Report** icon

A report is displayed in the ***Bentley Civil Report Browser*** dialog

The default **Vertical Alignment Review** report is displayed as shown below. Click on the other formats options to view the available options.



The format for the report, such as the number of decimal places, can be customized by choosing **Tools > Format Options**. Changes to these parameters are applied to the report when the ***Format Options*** dialog is closed. Suggested values for ODOT reports are shown below.



The report can be saved by choosing **File > Save As** from the **Bentley Civil Report Browser dialog.**

This completes this exercise.

- Exit OpenRoads Designer, checking the file back in to ProjectWise

St. Peter Road Proposed Profile

In this exercise, we'll define the proposed profile for St. Peter Road.

- Open **ProjectWise Explorer**

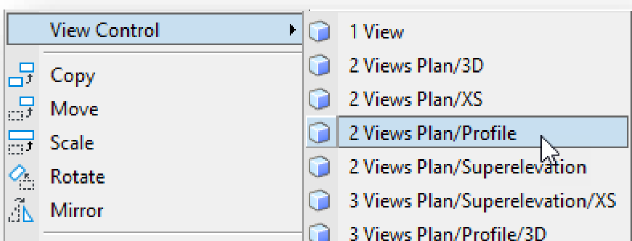
- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.

✓ Browse to the **400-Engineering\Roadway\Basemaps** folder.

✓ Open the geometry basemap for St. Peter Road:

400-Engineering\Roadway\Basemaps\123456_BK002.dgn

✓ Open a profile window for St. Peter Road. This can be accomplished by selecting the alignment and then choosing the **Open Profile Model** icon from the pop-up menu, or by holding down the right-mouse button until the menu shown below appears and choosing the **2 Views Plan/Profile** option and then following the prompts.



The existing ground is displayed in the profile view.

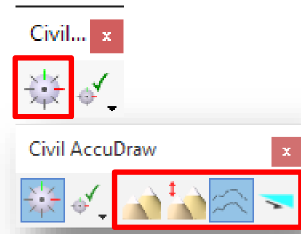
The profile for St. Peter Road will tie into S.R. 185. The profile grade point for the intersection of St. Peter Road with S.R. 185 can be drawn on the profile with the following steps:

✓ From the **Geometry** tab, choose the **Profile Creation > Profile Intersection Point** command and take the following steps:
   - When prompted to **Locate Element to Show Intersection**, choose the plan view graphic for St. Peter Road
   - When prompted to **Locate Element Which Intersects**, choose the plan view graphic for proposed S.R. 185

   A point representing the PGL location of S.R. 185 at the intersection with St. Peter Road is drawn on the profile

Next, use Civil Accudraw to store the tangent lines for the profile. Once these lines are stored, go back and store the vertical curves for the proposed alignment using the **Parabola Between Elements** command.

- ✓ Open the ***Civil AccuDraw*** dialog by choosing **Civil Toggles > Civil Accudraw** from the **Geometry** tab



  - Toggle Civil AccuDraw on by choosing the first icon

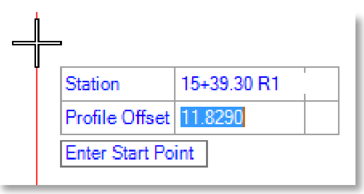
    With the profile window active, the Civil AccuDraw dialog will display icons relative to vertical geometry functions as shown

- ✓ Set **GE_P_Alignment** as the active **Feature Defintion**

- ✓ Store the proposed tangent lines for St. Peter Road using the values shown in the table below. Use Civil Accudraw with the appropriate ordinate to facilitate entering the profile points.

Station	Elevation	Civil Accudraw Ordinate
17+00	Meet Existing	ProfileOffset
18+25	997.8800	Z
19+70	995.6026	Z
20+00	Snap to S.R. 185 PGL Point	

Tips:
To use the **Civil Accudraw Profile Offset** ordinate, use the tab key to shift keyboard focus to the **Offset** field as shown below.



With focus in the **Offset** field, key in the “o” shortcut to select the profile element that will be used as the origin for the **Station** and **Profile Offset** values. You are prompted to **Select a reference element**. Select the exiting ground line for St. Peter Road.

- ✓ After placing the tangent lines, from the **Geometry** tab, select **Curves > Profile Curve Between Elements > Parabola Between Elements** to place the vertical curves for the alignment

  Toggle off Civil AccuDraw. It is not needed to define the vertical curves

Station	Curve Length
18+25	200’
19+70	10’

Use the **Trim/Extent Both** option when defining the curves.

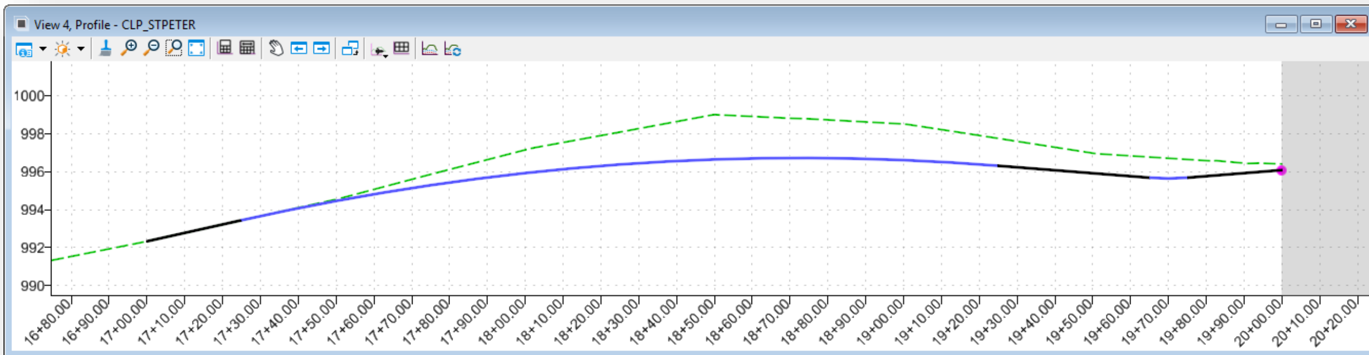
Create a complex element for the profile.

- ✓ From the **Geometry** tab, select **Complex Geometry > Profile Complex by Elements**
- ✓ Set the **Feature Definition** to **GE_P_Alignment**
- ✓ Set the **Name** to **PGL_CLP_STPETER** (See the **Location & Design Manual, Volume 3 - Highway Plans**, **Section 1204.3.6.2** for standards geometry element names.)
- ✓ Using the **Automatic Method**, choose the first tangent element to define the complex chain
- ✓ Left-click to accept the solution

Take the following steps to define the proposed profile as the active profile:

- ✓ Use the MicroStation **Select Element** tool to select the proposed profile line
- ✓ Allow the cursor to rest on the selected element until the pop-up menu appears
- ✓ Choose the **Set As Active Profile** icon

The completed profile should appear like the example shown below.



This completes this exercise.

- Exit OpenRoads Designer, checking the file back in to ProjectWise

Jameson Road Proposed Profile

In this exercise, we'll define the proposed profile for Jameson Road.

- Open **ProjectWise Explorer**

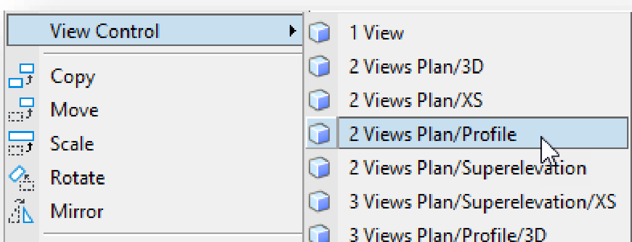
- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.

✓ Browse to the **400-Engineering\Roadway\Basemaps** folder.

✓ Open the geometry basemap for Jameson Road:

400-Engineering\Roadway\Basemaps\123456_BK003.dgn

✓ Open a profile window for Jameson Road. This can be accomplished by selecting the alignment and then choosing the **Open Profile Model** icon from the pop-up menu, or by holding down the right-mouse button until the menu shown below appears and choosing the **2 Views Plan/Profile** option and then following the prompts.



The existing ground is displayed in the profile view.

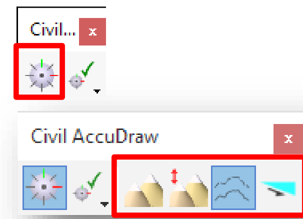
The profile for Jameson will tie into S.R. 185. The profile grade point for the intersection of Jameson with S.R. 185 can be drawn on the profile with the following steps:

✓ From the **Geometry** tab, choose the **Profile Creation > Profile Intersection Point** command and take the following steps:
  - When prompted to **Locate Element to Show Intersection**, choose the plan view graphic for Jameson Road
  - When prompted to Locate Element Which Intersects, choose the plan view graphic for proposed S.R. 185

  A point representing the PGL location of S.R. 185 at the intersection with Jameson Road is drawn on the profile

Next, use Civil Accudraw to store the tangent lines for the profile. Once these lines are stored, go back and store the vertical curves for the proposed alignment using the **Parabola Between Elements** command.

✓ Open the ***Civil AccuDraw*** dialog by choosing **Civil Toggles > Civil Accudraw** from the **Geometry** tab



- Toggle Civil AccuDraw on by choosing the first icon

  With the profile window active, the Civil AccuDraw dialog will display icons relative to vertical geometry functions as shown

✓ Set **GE_P_Alignment** as the active **Feature Defintion**

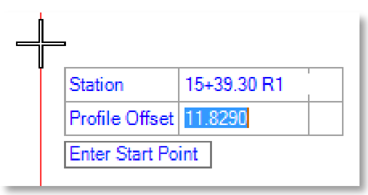
✓ Store the proposed tangent lines for Jameson Road using the values shown in the table below. Use Civil Accudraw with the appropriate ordinate to facilitate entering the profile points.

Note: Store the VPI points in this order:

- Store VPI 1 and VPI 2 first
- Then store the line from the point at the intersection with S.R. 185 moving backwards to define VPI 4 using a -8.0% slope for a lenth of -12'
- Next store VPI 3 moving backwards from VPI 4 using a -3.0% slope for a lenth of -12'
- Finally, place the tangent line between points VPI 2 and VPI 3

VPI	Station	Elevation	Civil Accudraw Ordinate
1	12+00	Meet Existing	ProfileOffset
2	12+75	-0.65% Slope from VPI 1	NA
3		-3.0% Slope, 12' left of VPI 4	NA
4		-8.0% Slope, 12' left of VPI 5	NA
5		Snap to S.R. 185 PGL Point	NA

Tips:
To use the **Civil Accudraw Profile Offset** ordinate, use the tab key to shift keyboard focus to the **Offset** field as shown below.



With focus in the **Offset** field, key in the "o" shortcut to select the profile element that will be used as the origin for the **Station** and **Profile Offset** values. You are prompted to **Select a reference element**. Select the exiting ground line for St. Peter Road.

✓ After placing the tangent lines, from the **Geometry** tab, select **Curves > Profile Curve Between Elements > Parabola Between Elements** to place the vertical curves for the alignment

Toggle off Civil AccuDraw. It is not needed to define the vertical curves

Station	Curve Length
VPI 2, Station 12+75	100'
VPI 3	10'
VPI 4	No Curve

Use the **Trim/Extent Both** option when defining the curves.

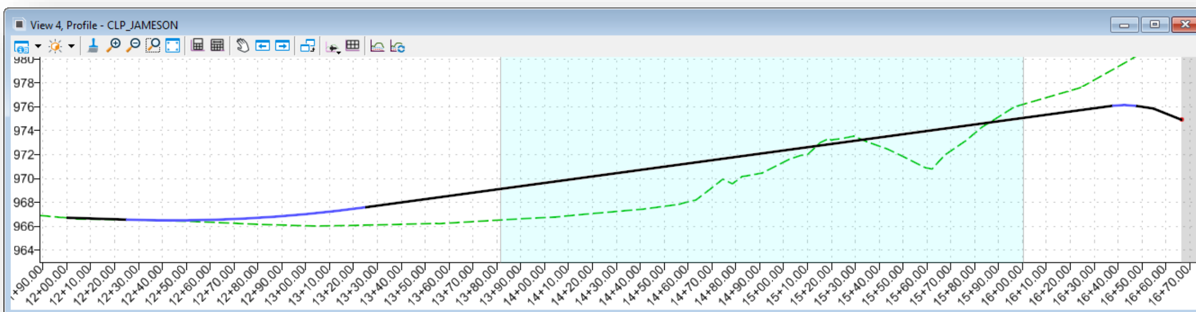
Create a complex element for the profile.

- ✓ From the **Geometry** tab, select **Complex Geometry > Profile Complex by Elements**
- ✓ Set the **Feature Definition** to **GE_P_Alignment**
- ✓ Set the **Name** to **PGL_CLP_JAMESON** (See the **Location & Design Manual, Volume 3 - Highway Plans**, **Section 1204.3.6.2** for standards geometry element names.)
- ✓ Using the **Automatic Method**, choose the first tangent element to define the complex chain
- ✓ Left-click to accept the solution

Take the following steps to define the proposed profile as the active profile:

- ✓ Use the MicroStation **Select Element** tool to select the proposed profile line
- ✓ Allow the cursor to rest on the selected element until the pop-up menu appears
- ✓ Choose the **Set As Active Profile** icon

The completed profile should appear like the example below.



This completes this exercise.

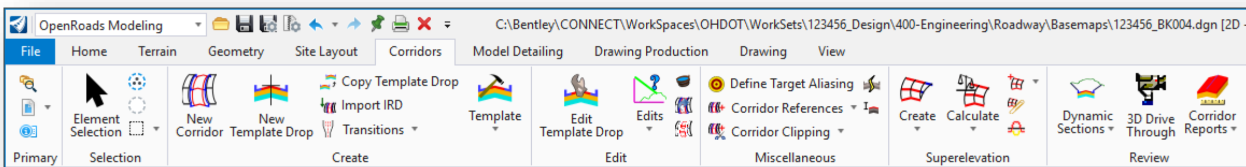
- Keep OpenRoads Designer open for exercise section 501 Corridor Modeling Basemap.

Section 500 Corridor Modeling

Corridor Modeling Overview

OpenRoads Designer provides several corridor modeling and site design tools to facilitate generating 3D representations of a new roadway or other type of surface. Designers work primarily in 2D files. The OpenRoads tools automatically generate the 3D model.

Tools for the design, creation, modification, management, and report functions are provided. The tools are accessed by choosing the Corridors tab as shown below.



Reference files are used to federate the data on a typical design project. Very small projects may have much of the data in a few files while larger projects will use multiple files for the geometry, terrain, superelevation, etc., and the 3D model. All these files can reference one another to present a complete model of the project.

Each road within the project is used to define a 3D corridor model representing the proposed design. A corridor consists of an alignment, profile, and a template defining the initial roadway typical section. Multiple templates may be applied within a corridor to better define the roadway. Additionally, transitions and other modifications to the template can be defined using various modification tools. As changes are made, the 3D corridor model is automatically updated.

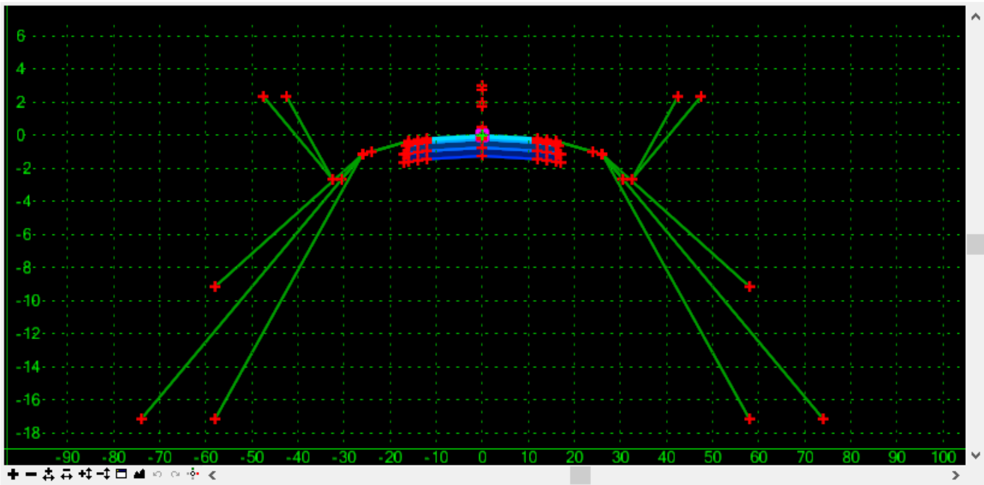
Corridor Modeling Workflow

The workflow for corridor modeling is summarized below.

1. Create Civil horizontal and vertical geometry element(s)
2. Create a DGN file for the corridor model
3. Attach references for the existing ground terrain and the alignment data
4. Define templates for the project
5. Set the active terrain (existing ground) if the template ties to ground
6. Create a corridor based on the horizontal and vertical elements
7. Add template drop(s)
8. Add horizontal and/or vertical controls for template points (optional)
9. Define any transitions and connections
10. Associate superelevation information
11. Review the results and adjust as necessary using additional controls, such as end condition exceptions, secondary alignments, parametric constraints, or target aliasing
12. Continue the process/review/modify

Templates

In simple terms, the template is a "smart" typical section that you "push" along an alignment to create the proposed 3D model. An example of a template for a 2-Lane undivided highway, with cut and fill slope options, is shown below.



Templates are stored in a template library (*.itl).

Templates are defined by inserting **Points**, **Components**, and **End Conditions**. A component is a set of points that define an open or closed shape. There is no limit to the number of points or components in a template.

ODOT Template Library

ODOT provides a sample template library in the following folder:

..\Standards\Template Library\OHDOT_Templates.itl

The templates contained in the ODOT library can be used as a starting point to define templates for specific projects. Many of the templates represent individual components that can be combined to generate the full template for a specific typical section design. For example, a template representing a lane of pavement can be combined with another template representing curb and gutter.

The templates for each project are stored in their own project specific template library. When a new project is created in ProjectWise using the ODOT Project Creation application, the ODOT template library is copied into the **990-WorkSet Standards\Template Library** folder for the project and renamed as follows:

######_Templates.itl
Where ###### = the 6-digit PID number for the project.

The template library is specified in the .cfg file for the WorkSet by the MicroStation configuration variable listed below:

CIVIL_ROADWAY_TEMPLATE_LIBRARY = $(_USTN_WORKSETSTANDARDS)Template Library/$(OHDOT_PID)_Templates.itl

Template creation is a key component to the Corridor Modeling process.

Corridors

Corridor Modeling Basemap Design Files

For ODOT projects, the 3D model is generated in a "KM" basemap design file. For smaller projects with only one alignment, it is acceptable to generate the entire model in one KM design file. For projects with more than one alignment, it is recommended that users take a federated approach to store each corridor model in its own KM design file.

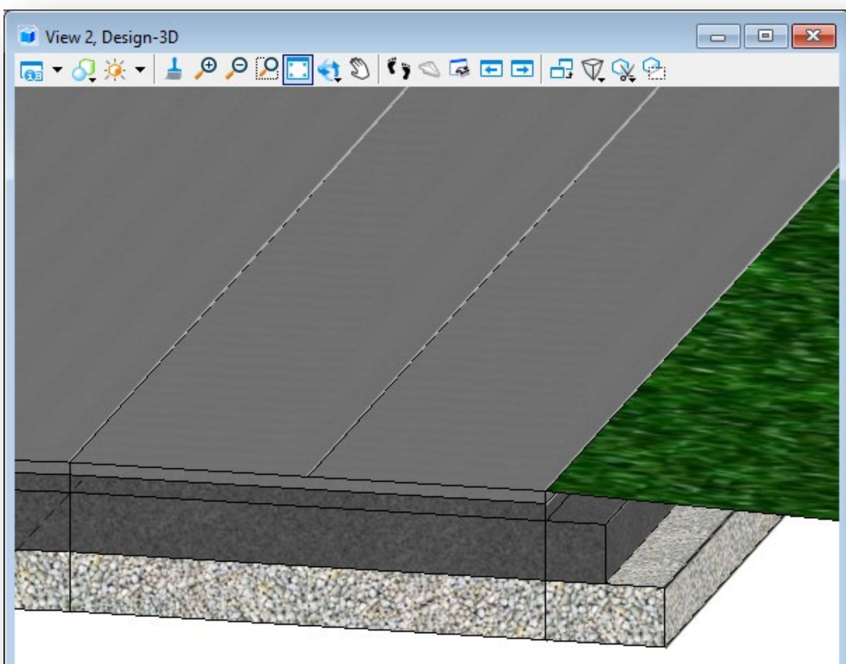
For projects with lengthy alignments, it may be desirable to create multiple KM files for a single alignment. When a corridor is defined, beginning, and ending stations are specified. Using this workflow, multiple corridor design files are referenced together.

The **OHDOT Create Design Files** application is used to create the KM design files for the project.

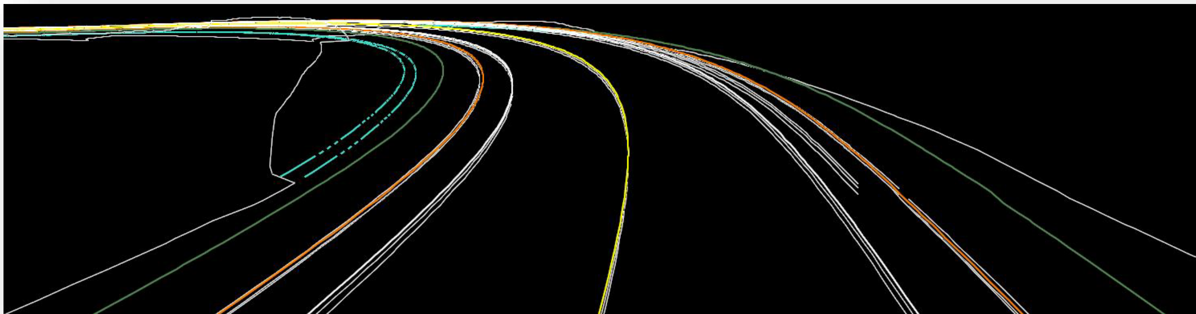
Corridor Model 3D Graphics

When a corridor is processed, the 3D model is generated in a separate MicroStation model named "Design-3D". The corridor graphics consist of 3D line strings and 3D surfaces for each design component.

The example below shows a portion of the pavement for a corridor with the individual pavement layers that were generated for the 3D model as well as a portion of the side slope.



When the template is processed, the template points are connected between template "drop" locations to form the longitudinal break lines that are used to create the proposed surface. An example of the longitudinal break lines plotted from a 3-dimensional design surface is shown below.

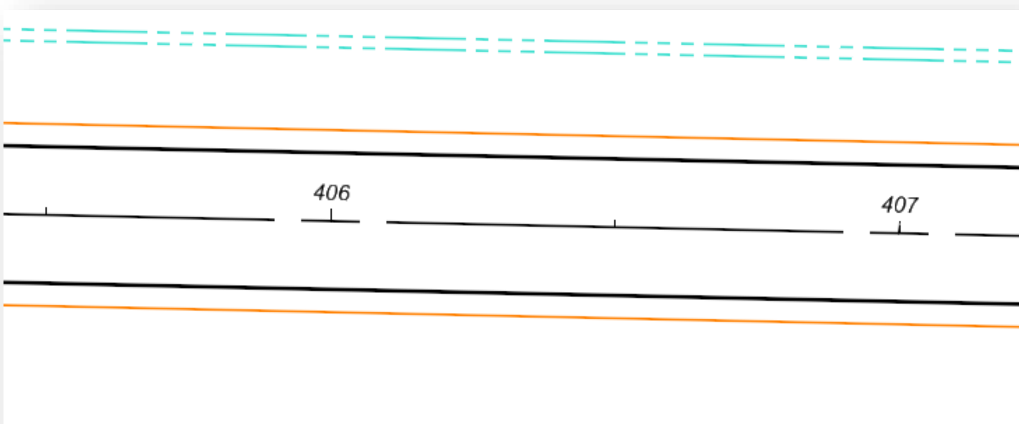


Only the top template points are used to create the proposed surface mesh. Points below the surface are excluded from the proposed design surface.

The sub-surfaces can be included in the proposed cross sections as "components" or alternate surfaces. These components can be used for volume calculations.

Corridor 2D Graphics

In addition to the 3D graphics, the corridor modeling process draws 2D plan view graphics. These graphics can be used to generate the plan sheets for the project. An example of the 2D plan-view graphics drawn as part of the corridor modeling process for the pavement, shoulders, and a ditch, is shown below.



Section 501 Corridor Modeling Basemap Exercise

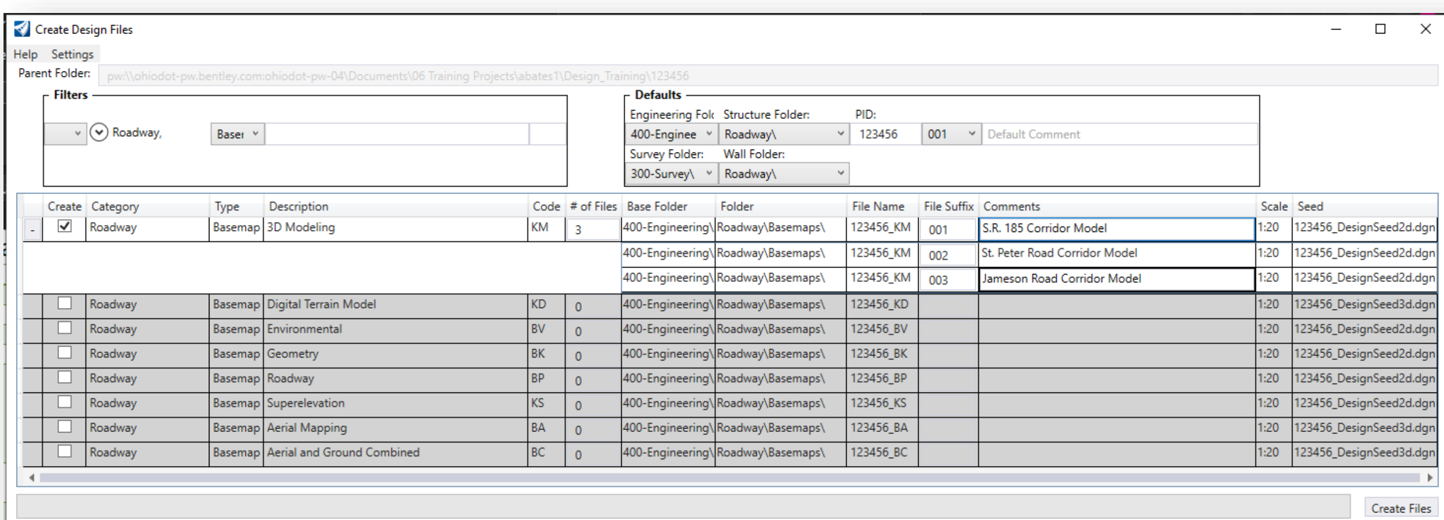
Exercise A: Corridor Modeling Design Files

In this exercise we will create separate design files for the 3D model of each alignment in the project.

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.
- Open the **123456_BK001.dgn** file or continue in **123456_BK003.dgn** if following exercise 401 completion.
- Select the **OHDOT Create Design Files** application (**OHDOT** WorkFlow) to create the corridor modeling basemaps for the project. Create the corridor basemap design files for the following alignments:

File Name	Comments
123456_KM001.dgn	S.R. 185 Corridor Model
123456_KM002.dgn	St. Peter Road Corridor Model
123456_KM003.dgn	Jameson Road Corridor Model



The settings in the **OHDOT Create Design Files** application are shown on the following page

✓ Once you've selected the corridor modeling basemap (KM) type fill out the **Comments** column
  - S.R. 185 Corridor Model
  - St. Peter Road Corridor Model
  - Jameson Road Corridor Model

✓ Choose the **Create Files** button to initiate the process. Three files are created.

Note: As you can see when you fill out the Comments column it fills in the file description in PW.

123456_KM001.dgn	Civil 3D Modeling Basemap S.R. 185 Corridor Model	123456_KM001.dgn
123456_KM002.dgn	Civil 3D Modeling Basemap St. Peter Road Corridor Model	123456_KM002.dgn
123456_KM003.dgn	Civil 3D Modeling Basemap Jameson Road Corridor Model	123456_KM003.dgn

This completes this exercise.

- Keep OpenRoads Designer open.

Section 502 Templates

ODOT Template Library

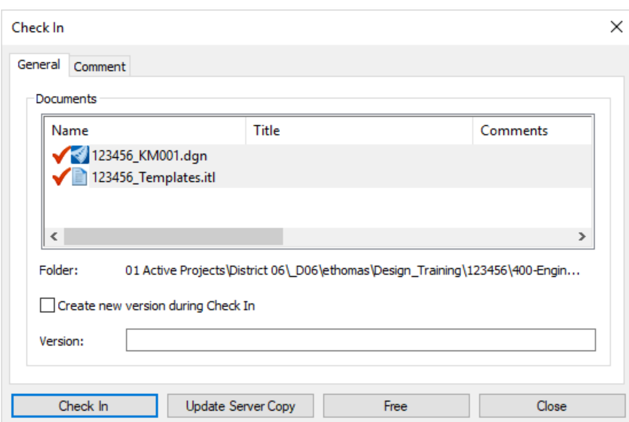
Templates are used to define the typical section for each alignment. Multiple templates can be used to define different typical section designs, or the template geometry can be modified using a variety of corridor editing tools.

When the WorkSet for a new project is created in ProjectWise, the **OHDOT_Templates.itl** library is copied from the OHDOT Standards to the **990-WorkSet Standards\Template Library** folder and renamed using the PID number for the project.

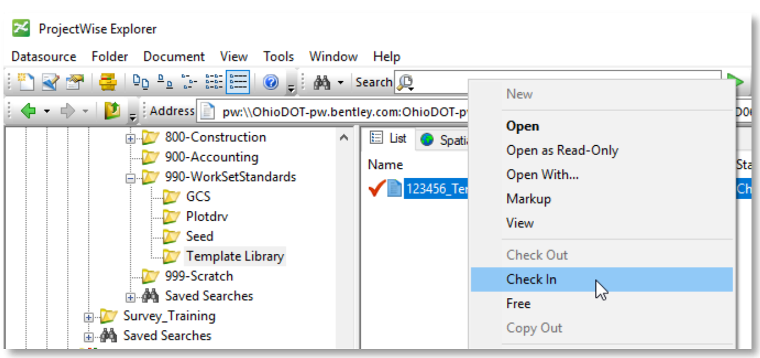
For users working outside of the ProjectWise environment, when the **Startup.mvba** application is processed on a new WorkSet, the template library is copied to the **990-WorkSet Standards\Template Library** folder as part of the program operation.

ProjectWise Template Library Check-in/Out

When a template library is opened in ProjectWise, the file is checked out to the current user and cannot be viewed or edited by other users. When the ***Create Templates,*** dialog is closed, the software does not ask the user to check the template library back into ProjectWise. The user is prompted to check the template library back into ProjectWise when the design file is closed, as shown below.

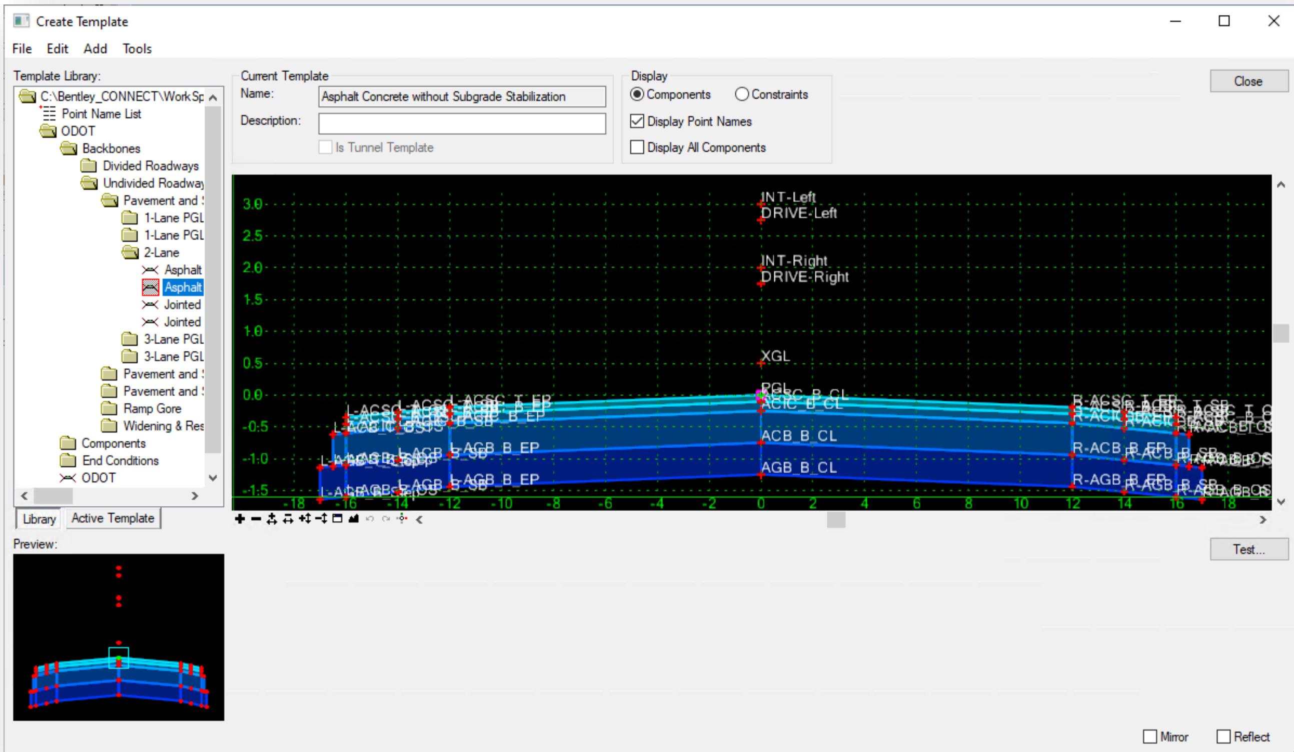


The template library can be checked back in after closing the template library from the ***ProjectWise Explorer*** dialog as shown below.



Create Template

Templates are created and edited in the template library by choosing **Template > Create Template** from the **Corridors** tab.



The dialog is split into several areas. Each area is briefly described on the following pages. See the online help for information not presented here.

Menu Commands

File

The **File** menu contains familiar commands for creating new Templates and new folders within the template library, as well as commands to save and open template libraries,

Edit

The **Edit** menu contains commands used when editing templates such as **Undo**, **Redo**, **Cut**, **Copy**, **Paste**, etc.…

Add

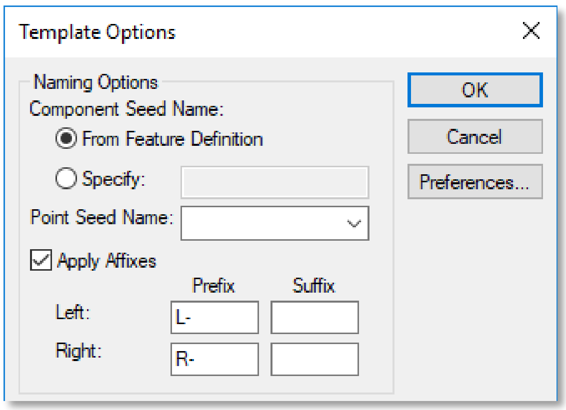
The **Add** menu contains commands to add points and components to the active template.

Tools

The **Tools** menu contains various tools for template creation. The Options and Dynamic Settings menu items are described on the following page.

Tools > Options

Selecting **Tools > Options...** displays the *Template Options* dialog which allows you to specify naming and step options for the current template. Use the **Dynamic Settings** dialog, detailed on the following page, to toggle these settings on/off as you design the template.



Naming Options>Component Seed Name > From Feature Definition indicates, when selected, the component name is derived from the style specified when the component is created.

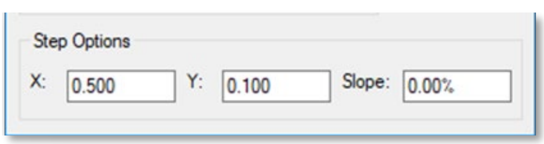
Specify indicates, when selected, the component name is specified in the field.

Point Seed Name allows you to specify the seed name for a point in the current template. Since point names must be unique, the seed name is appended with a different number each time a point is created.

Apply Affixes
Indicates when checked "on", that affixes are applied.

Left Prefix/Suffix specifies which prefix/suffix should be added to a point or component when the value is on the left of 0.0 (absolute).

Right Prefix/Suffix specifies which prefix/suffix should be added to a point or component when the value is on the right of 0.0 (absolute).



Step Options
These options are used to specify precision step input for components and points. These values can also be accessed from the *Dynamic Settings* dialog.

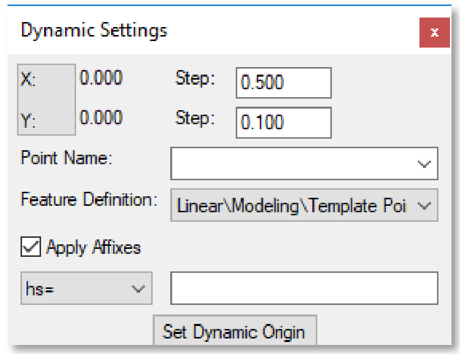
X Step specifies the step increment for the X distance from the dynamic origin. If this value is non-zero, then the cursor moves to the horizontal increments of the specified value as measured from the dynamic origin.

Y Step specifies the step increment for the Y distance from the dynamic origin. If this value is non-zero, then the cursor moves in vertical increments of the specified value as measured from the dynamic origin.

Slope specifies the step increment for slope from the dynamic origin. If this value is not 0, then the cursor moves in slope increments of the dynamic origin. Slope is applicable only when the dynamic settings are in horizontal/slope mode.

Tools > Dynamic Settings

Select **Tools > Dynamic Settings** to display the *Dynamic Settings* dialog, which is used for precision input of template components and to assign point names and styles when defining components.



X:Y/X:Slope toggles between X,Y step lock and X/Slope step lock.

X Value displays the current X distance from the dynamic origin.

X Step specifies the step increment. If this value is non-zero, then the cursor moves in horizontal increments of the specified value as measured from the dynamic origin.

Y value displays the current Y distance from the dynamic origin.

Y Step specifies the step increment. If this value is non-zero, then the cursor moves in vertical increments of the specified value as measured from the dynamic origin.

Point Name specifies the name of the point to be placed.

Point Style allows you to set the style of the point. The style is used to draw feature lines in the plan view. For ODOT projects, the style **CM_P_Feature_Line** should be used as the default style for most template points unless you want to assign a specific style that can be plotted in the display view 3D model.

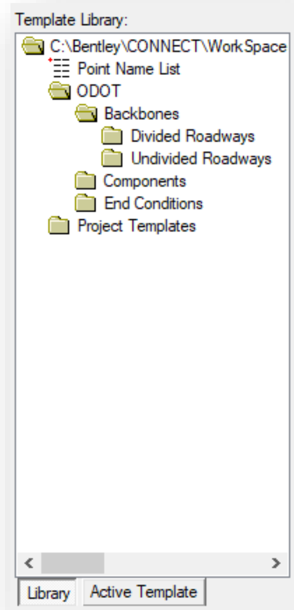
Apply Affixes specifies, when selected, that point name affixes (prefix or suffix) are to be applied to the point name when placing points. The **Affixes** are defined in the *Template Options* dialog.

The **Key-in** pull down is used to specify the type of key in to be performed.

- XY= key in absolute coordinates
- DL= key in delta coordinates from last point placed (defaults to the dynamic origin if it is the first point of a component.
- HS= key in horizontal delta distance and slope from last point placed.
- VS= key in vertical delta distance and slope from last point placed.
- OL= key in delta coordinates from dynamic origin.
- OS= key in horizontal delta distance and slope from dynamic origin.

Set Dynamic Origin sets the location of the dynamic origin.

Template Library



On the left side of the *Create Template* dialog is the **Template Library** area which contains a Windows-like folder structure. This area is used to manage the content of the current template library. Two tabs are supported for the **Template Library** area as described below.

Library

When the **Library** tab is selected, the **Template Library** displays the contents of the current template library in a familiar Windows folder structure as shown at right. The **Template Library** area also supports common Windows functions like drag, drop, cut, paste, etc.

If you double-click a template name, it becomes the active template.

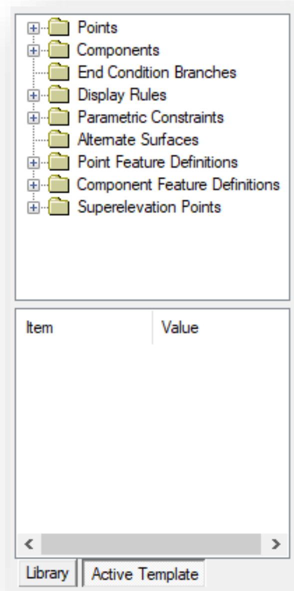
When a template is selected from the list, the **Preview** window is used to display a preview of the template.

Double-click the template name to set it as the active template.

The **ODOT** category contains various components and end conditions that can be used to build templates for your project.

The **Project Templates** category is used to create new templates for the active project.

Active Template



When this tab is selected, the **Template Library** area displays the contents of the active template as shown in the image to the right.

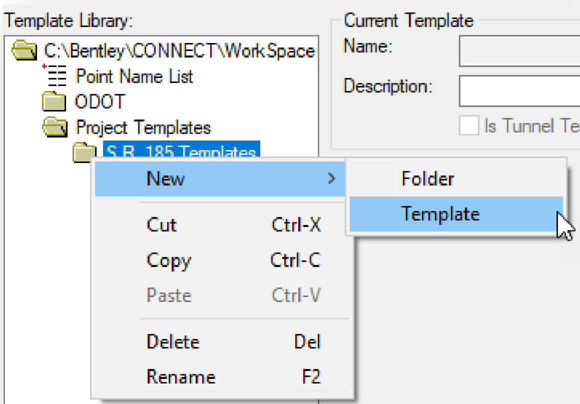
The **Active Template** option is used to review and edit template points, components, and parameters in a table format.

Creating New Folders and Templates

Templates can be organized into folders in the active template library. The Project Templates folder is intended to be used to create new folders and templates for a specific project.

To create a new folder, right click on the Project Templates folder and choose the **New > Folder** option.

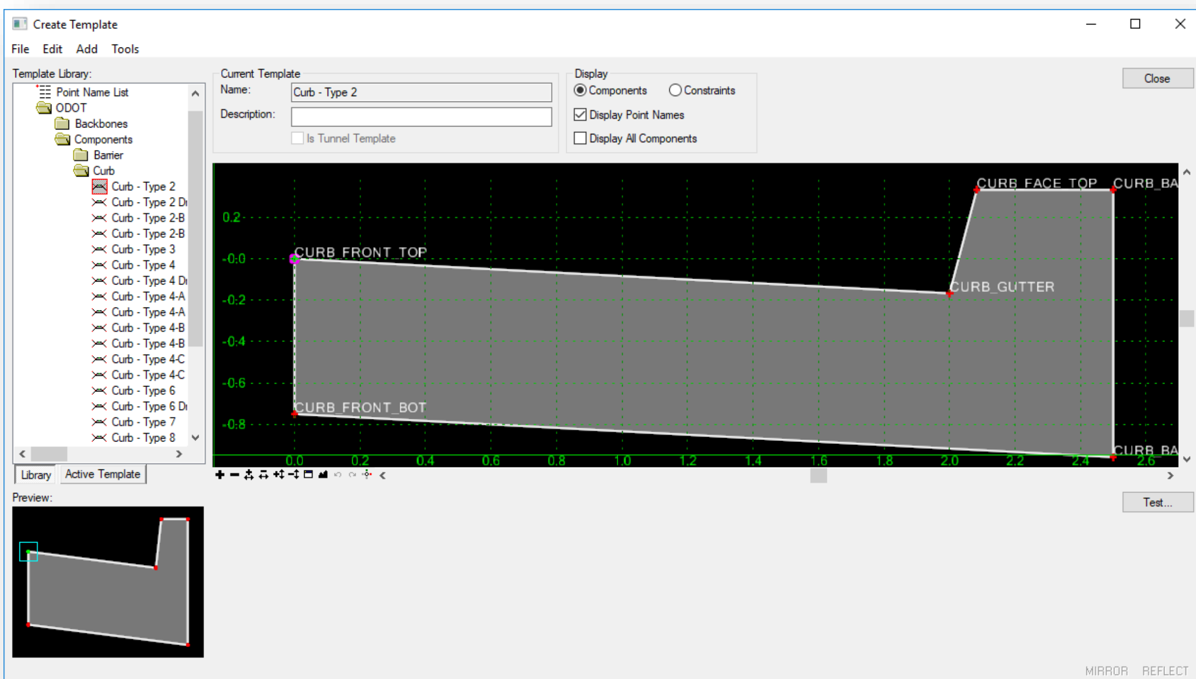
To create a new template, right click on the folder and choose the **New > Template** option.



You can also cut, copy, and paste templates from the ODOT folders to the desired Project Templates folder.

Current Template

The right side of the **Create Template** dialog is used to display and edit the current template.



Name
Shows the name of the current template.

Description
Allows you to review/edit the current template description.

Display
Components/Constraints
This option is used to set the display mode of the template. If **Components** are selected, then the template components are displayed. If **Constraints** are selected, then the point constraints for the template are displayed. Template points are always displayed.

Display Point Names
Toggle this option “on” to display the template point names.

Display All Components
When this option is checked on, any components that are “hidden” are displayed with dashed lines.

Current Template Window
The Current Template Window is used to display and edit the current template. The window uses its own graphics engine and is not a MicroStation view.

A magenta-colored box is used to designate the origin of the template.
At the bottom of the **Current Template** Window are view commands like the MicroStation view commands.

The two buttons that look like MicroStation View Previous and View Next commands are undo and redo commands used on template operations. The CTRL-Z keyboard shortcut can also be used to undo the last template operation.

Within the **Current Template Window,** you can right-click to access commonly used template functions. You can also double click template components for editing purposes.

Template Points

When the template is processed using the corridor modeling command, the points of a template are connected to form *breakline* features. Each point will be connected longitudinally to the next template drop location to form 3-D longitudinal breakline surface features. The breaklines are drawn using the symbology defined with the **Feature Definition** that is assigned to each point.

Since the template points are connected to generate the completed surface, establishing a standard name for template features that are used on each template is very important to the quality of the proposed design surface. For example, if a pavement point is named ACSC_T_CL on the first template, and it is named PAVT_SURFACE on the next, the software will not automatically connect these points to generate the breakline feature. ***Following a point naming convention is important to obtain the best results with Corridor Modeling.***

It is equally important to ensure that the feature definition assigned to points is consistent from template to template.

Each template point name must be unique to the template. The software automatically appends a number to the point name if the same name is used during template creation.

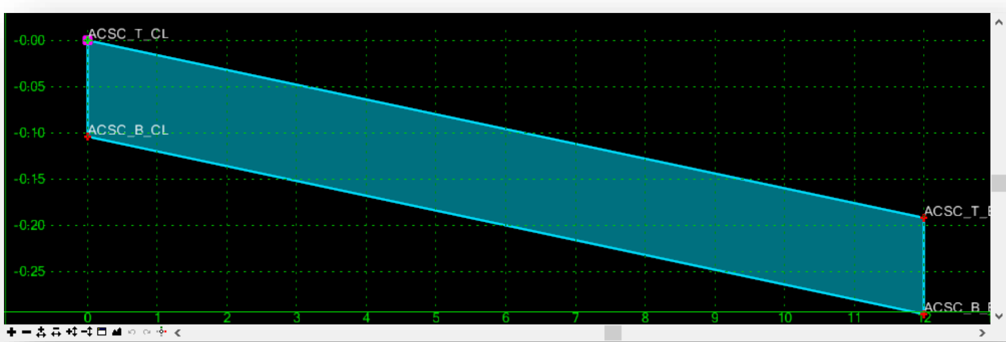
There is no limit to the number of points in a template.

Template Components

A component is a set of points that define an open or closed shape. Each component, whether open or closed, can represent a different material or area of interest. Components are named and have an assigned feature style. There are 6 types of components as described below:

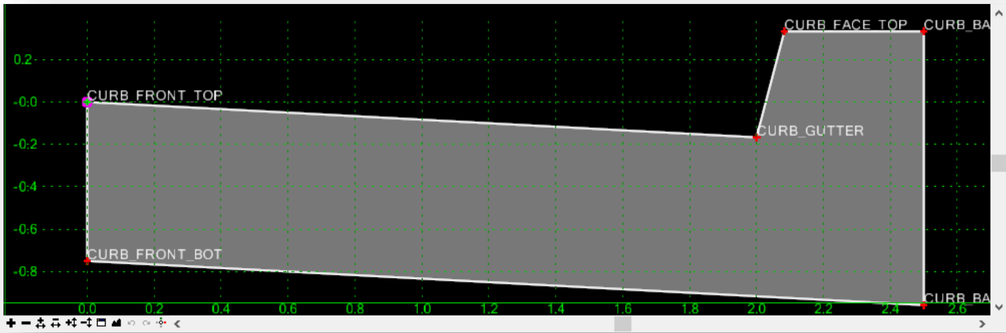
Simple Components

A simple component typically represents a section of pavement. It's a closed parallelogram (4 constrained points) that is defined by the slope and thickness.



Constrained Components

A constrained component consists of points that are all restricted to the movement of the first point. A constrained point is typically used to manage the behavior of other points in the template. When a point (parent) is moved, any constrained point (child) also moves. This restriction only affects the offset and elevation (x,y) of the restrained point and the relationship is unidirectional (movement of child point does not move the parent point). Fully constrained points are displayed with a red + symbol.



Unconstrained Components

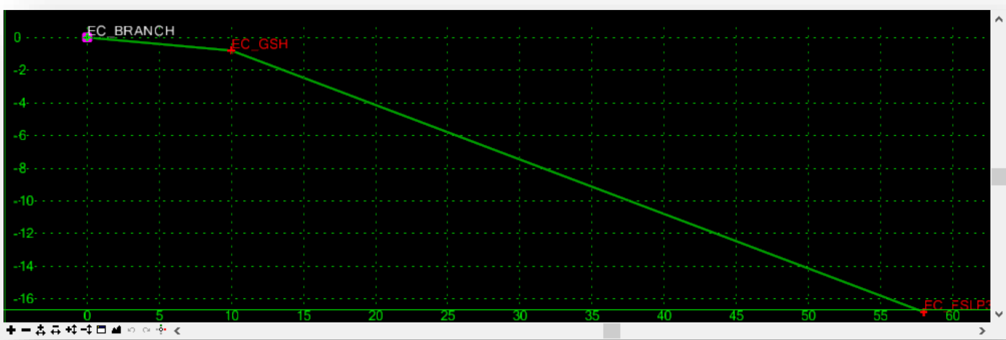
An unconstrained component is open or closed-shaped with no movement restrictions. Moving one point has no impact on the other points that make up the component.

Null Point

A null point is a template point that is purposely not related to a specific component. It's most often used as a reference for controlling other points.

End Condition

An end condition is a special open-shaped component that targets a surface, an elevation, or an alignment. End conditions are typically used to define cut and fill slope solutions.

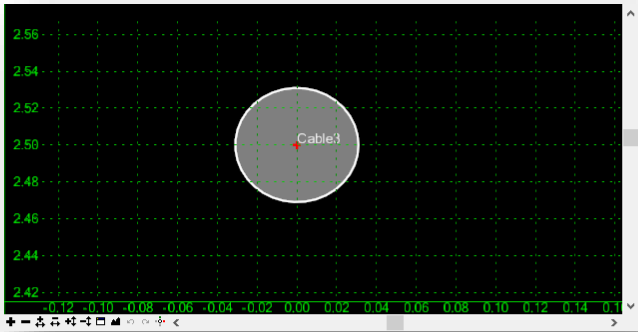


Overlay/Stripping Components

A specialized component type that is used for defining pavement planning and overlay materials.

Circle

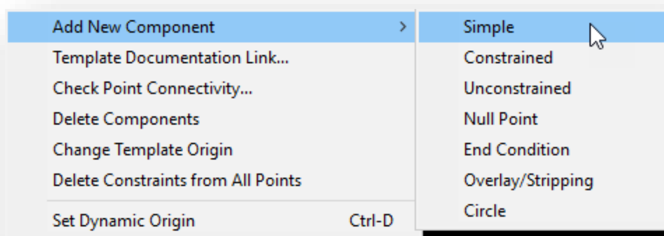
Circular components can be created as shown in the example below.



Creating Template Components

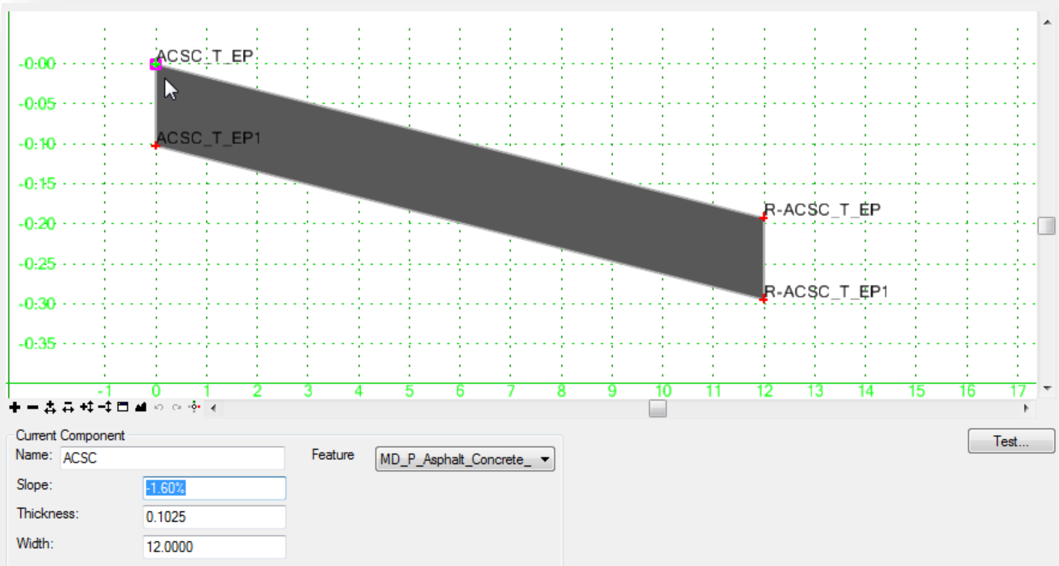
ODOT provides an example template library that contains several commonly used components. These components can be used to build templates. It may also be necessary to make specialized components to meet specific project requirements.

Components are added to the current template by selecting the component that you wish to add from the **Add** pull-down menu, or by right-clicking in the **Current Template** window to access the **Add New Component** menu shown at right.

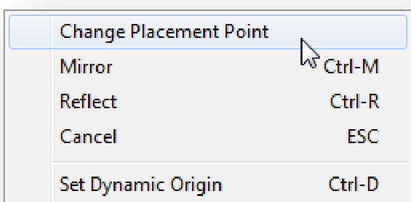


As new components are created, the **Current Component** portion of the *Create Templates* dialog is used to define the **Name**, **Style**, and any other relevant parameters for the component type being created.

In the example below, a simple component is created representing a single lane of pavement.



Before placing the component in the template, various options are available to control how the template is placed by right-clicking in the template window to reveal the menu shown below.



Change placement Point can be used to change which component point is used as the point placed at the point you specify. Each time this command is selected, the placement point will move in a clockwise direction to the next point in the component.

Mirror will create the original component as well as a mirrored component reflected horizontally about the template origin.

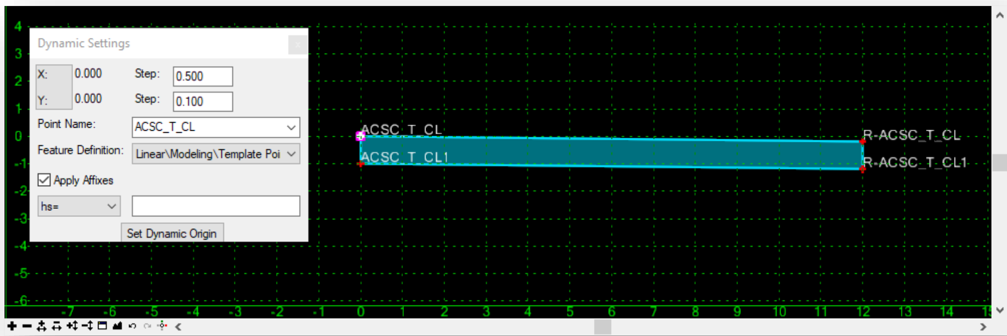
Reflect will create a single component reflected about the template origin.

The **Mirror** and **Reflect** options are toggles that are turned on or off and remain active. The on/off status of the **Mirror** and **Reflect** options can also be turned on/off by toggle switches at the bottom right corner of the *Create Templates* dialog.

Select **Set Dynamic Origin** to set the location of the dynamic origin. The dynamic origin is indicated by the purple box in the template display window.

Point Names and Affixes

When components are added to a template, the points that make up the component are named according to the **Point Name** and the **Pont Style** defined in the *Dynamic Settings* dialog (**Tools > Dynamic Settings**). The **Point Name** is automatically appended with the number 1, and the appended number is automatically incremented for each point in the component as shown in the example below.

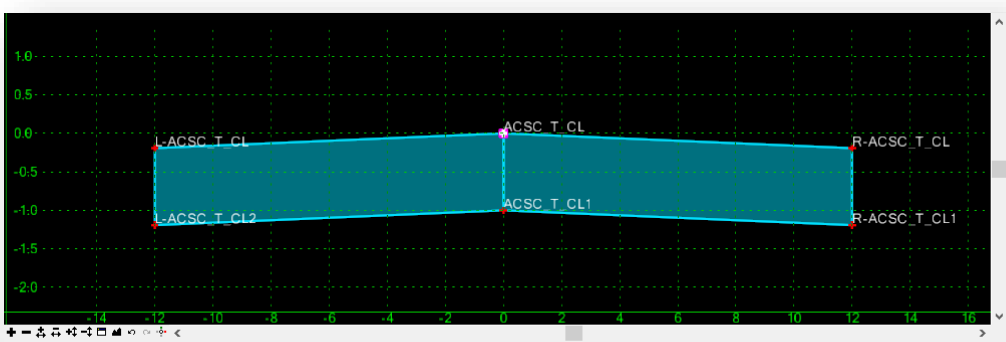


The **Point Name** can be keyed in or selected from the drop-down menu. When selected from the menu, the **Point Style** is automatically assigned for the point according to ODOT standards.

The point and component name designations are a bit cryptic. A listing of the point and component names can be reviewed in the following document in the OHDOT Standards:

..\OHDOT\Standards\Template Library\OHDOT_RoadwayTemplateDefinitions.xlxs

The **Apply Affixes** option is used to add a user defined prefix or suffix to the point name. In the example below a simple component was added with the **Mirror** option toggled on (as the component is being placed, right-click to reveal a menu to choose the **Mirror** option). The prefix of "L-" is applied to the points on the left side of the template. The prefix of "R-" is applied to the points on the right side of the template.

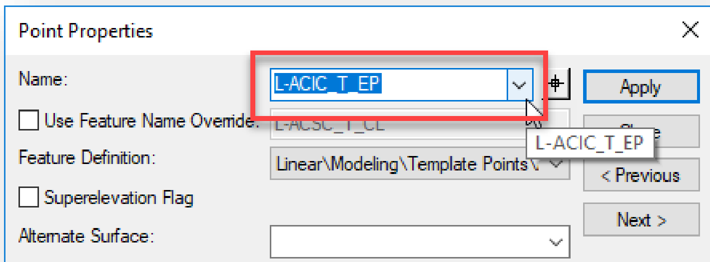


The affixes are defined in the *Template Options* dialog.

Tip: As a rule, affixes should not be applied to templates that contain individual components that will be used as "building blocks" for complete roadway templates. This is because the component may be used on either the left or right side of the template. Affixes should always be applied when creating completed templates for use with Roadway Designer.

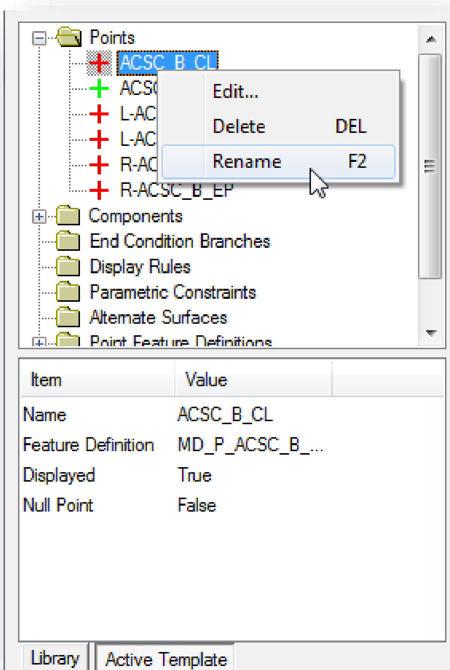
Note that all the points of the component are named using the currently defined **Point Name**. In the example on the previous page, the points were placed using the **ACSC_T_CL** (**A**sphalt **C**oncrete **S**urface **C**ourse **T**op **C**enter **L**ine) point name. After the component is placed, it is necessary to rename the points and assigned feature definitions to ensure that the points and 3D line strings are drawn using the appropriate standards. This can be done by double-clicking on each point to access the *Point Properties* dialog shown below.

The **Name** can be typed in or selected from the drop-down list. If the **Name** is selected from the list, the associated **Feature Definition** is automatically defined. The *Point Properties* dialog is covered later in this document.



Reviewing and Editing Point and Component Names

Point and component names can easily be reviewed and edited using the **Active Template** tab as shown below.



Right-click on a point or component name to reveal the menu shown at left. Choose **Rename** to rename the selected element.

The **Item** and **Value** list, shown below the elements, is used to review various parameters but cannot be used for editing.

Choose the **Edit** option for more editing operations.

Points and **components** can also be edited by double-clicking on a point or component in the template window. The *Point Properties* and *Component Properties* dialogs are covered later in this document.

Note: The **Feature Definition** cannot be edited using the **Active Template** editing options.

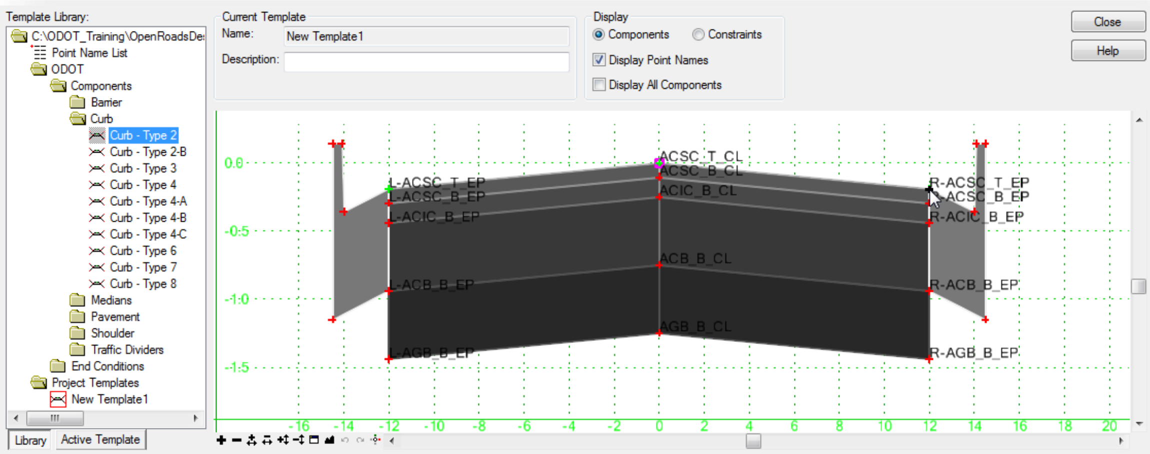
Drag and Drop Component Operations

A previously created template component can be included in the active template by use of drag and drop operations. Select a previously drawn component from the active template library and drag the component into the **Current Template** window. The component dynamically floats on the cursor. Release the mouse button to drop the component at the desired location.

While dragging the component in the **Current Template** window, right-click anywhere in the **Current Template** window to access the placement options discussed previously (**Mirror**, **Reflect**, etc…)

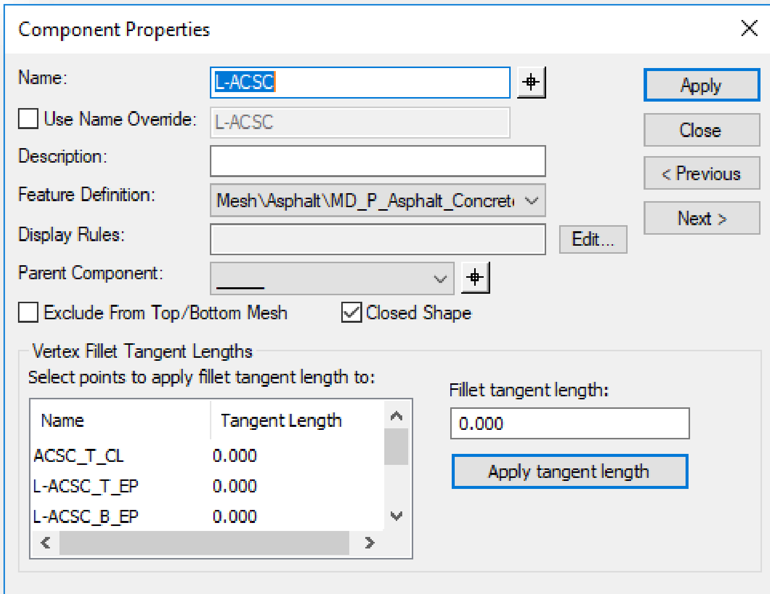
When connecting two components by dragging a component into the active template, the connecting point will change to a white colored plus sign prior to placing the component to indicate that the points are coincident. The point name from the previously placed component will be retained. Point names can be edited at any time during the template creation process.

In the example below, a previously defined curb and gutter component is placed at the edge of pavement in the active template (**New Template1**) by dragging the **Curb – Type 2** component and dropping it on the right edge of pavement point. The **Mirror** option is turned on, with **Apply Affixes** also toggled on, to place the curb and gutter on both sides of the pavement.



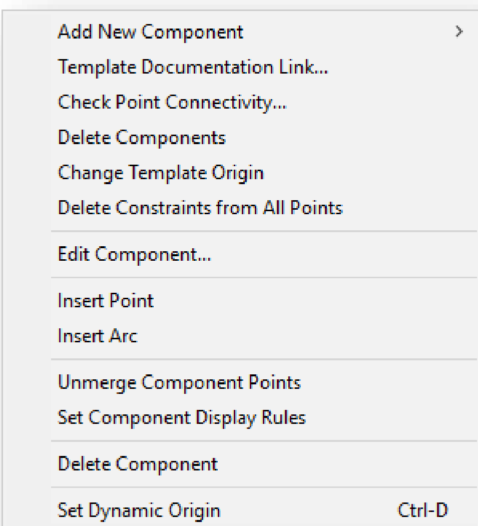
Component Editing

Components are edited by double-clicking on a component to open the ***Component Properties*** dialog shown below.



See the online help for details on each of these parameters.

Issue a right mouse click on a previously drawn component to access the menu shown below.



See the online help for details on each of these functions.

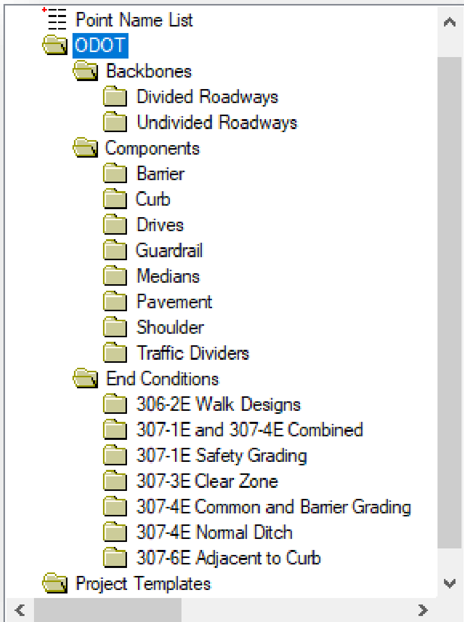
Mouse Wheel Functions While Creating Templates

The mouse wheel has the following functions when creating templates:

- Scroll the wheel to zoom in or out in the template window.
- Hold down the **Shift** key while scrolling the mouse wheel to zoom in or out in the vertical direction only.
- Hold down the **Ctrl** key while scrolling the mouse wheel to zoom in or out in the horizontal direction only.

The ODOT template Library

ODOT provides a template library with many template components defined that can be used as the starting point for creating and editing templates for your project. When a new WorkSet is created, the OHDOT template library is copied to the **990-WorkSet Standards\Template Library** folder.



The OHDOT template library has two main folders, **ODOT** and **Project Templates**.

The **ODOT** folder contains various **Components** and **End Conditions** that can be used to create completed templates.

The **Project Templates** folder is an empty folder to be used by the designer to create templates for the project.

The contents of the **ODOT** folder are summarized below:

Backbones
The “backbone” is defined as the pavement and shoulder portion of the template. The **Backbones** folder contains several common pavement and shoulder configurations for divided and undivided roadways. A variety of configurations are available for superelevated and non-superelevated roadways.

Components
The **Components** folder contains individual design components that can be used as the “building blocks” for a complete template. A Type 2 Curb and Gutter is an example of a component.

End Conditions
Several end conditions are defined for cut and fill conditions according to ODOT design standards.

Template Creation Workflow

While the process to create templates will have many variables, the general procedure to create templates is summarized below:

- Open the template library
- Create a new template or copy a template. Templates can be copied from one library to another by selecting **Tools > Template Library Organizer** from the ***Create Templates*** dialog.
- Drag and drop template backbones or individual components into the active template
- Create/edit the components as necessary
- Merge components as desired
- Review the point names and point name overrides
- Review the component names and component name overrides
- Test the template behavior
- Save the template library

Advanced Template Design

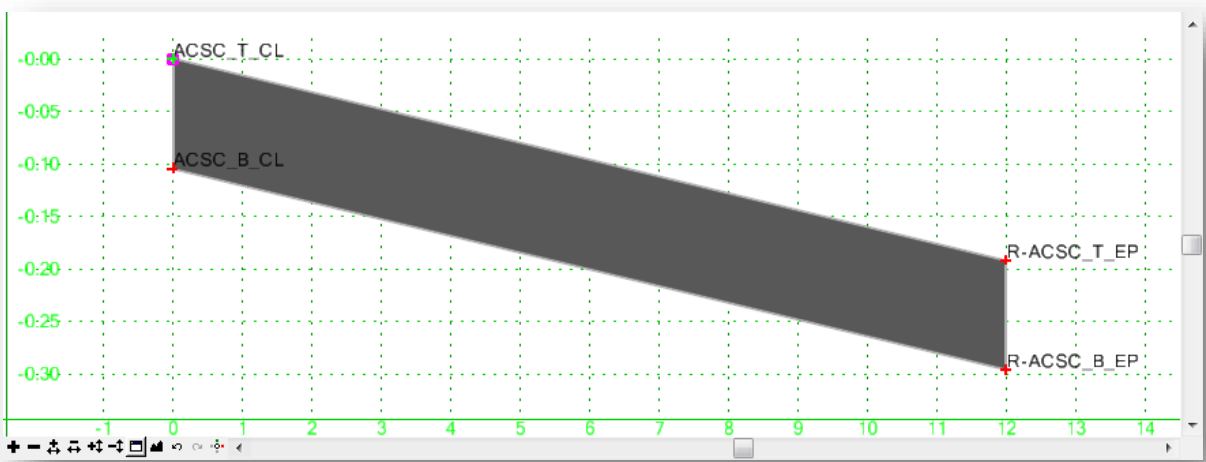
The corridor modeling tools provide a lot of options that can be used to create very complex templates that handle a variety of design scenarios. However, care should be taken not to create overly complex "Franken-Templates" to create a single template to account for all the design elements that occur over the length of the project. Overly complex templates can be hard to edit and even harder to diagnose when things don't work as expected. It's better to define a few simpler templates for specific station ranges than it is to define one overly complex template.

In this section, we will take a deeper look at the available template parameters.

Point Constraints

Point constraints are used to manage the behavior of points in a template. They are used so that if a point is moved in a template, either by the user editing the template or by the application of a horizontal or vertical control during design processing, all the points related to the point being moved behave in a rational and predictable manner.

For example, when a **Simple** component is inserted into the active template, it is comprised of four points which are connected to define the component as shown in the example below.



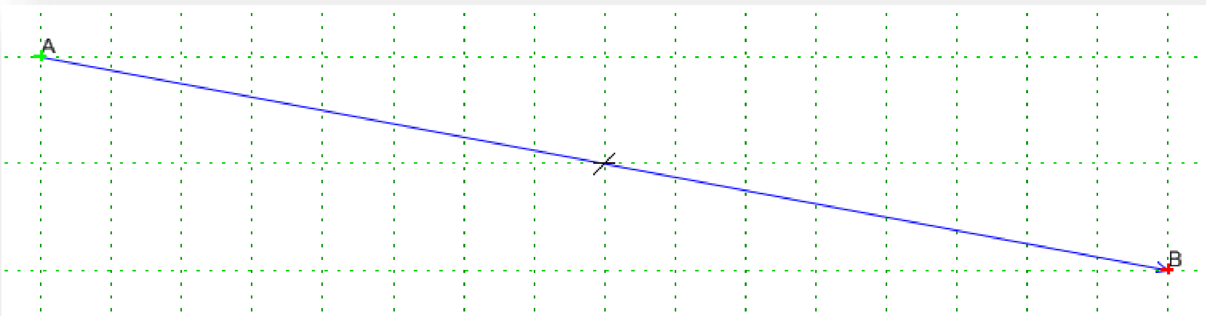
Three of the four points are constrained with relation to the insertion point. If the original insertion point is moved, the other three points move in relation to the first point.

A point can have at most two constraints. If two constraints are defined, the point is said to be "fully constrained". A point that is fully constrained is represented by a red plus sign. In the example above, points **R-ACSC_T_EP**, **R-ACSC_B_EP**, and **ACSC_B_CL** are fully constrained.

A point that is partially constrained, meaning that it has only one constraint on it, is shown as a yellow plus sign.

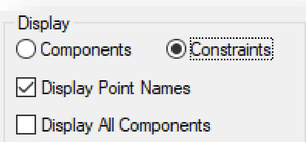
An unconstrained point is shown as a green plus sign. In the example above, point **ACSC_T_CL** is unconstrained.

Point constraints are two-dimensional and one-way. Two-dimensional means that the constraints can only affect the points offset and elevation (x and y coordinates in the cross-section view). One-way means there is a child-parent relationship between points. In other words, if point B is constrained by point A, point A is said to be the parent of point B and moving point A will affect point B, but you cannot move point B to affect point A. The example below shows a sample of this where the blue arrow indicates a parent/child relationship from point A to point B:



Constraints can also be labeled so that during processing, the value(s) of labeled constraint(s) may be changed. For example, the template may have a constraint label called **ACSC Thickness** that controls the thickness of the first pavement layer. This value can then be changed over one or more station ranges during the corridor modeling process.

The constraints for all the points in a template can be displayed by choosing the **Display Constraints** option on the *Create Template* dialog.



Constraints are displayed as blue lines between the parent and the child point. At the midpoint of the constraint line, there is a short white line designating the constraint type as follows:

- Horizontal line = Horizontal Constraint
- Vertical line = Vertical Constraint
- Sloped line = Slope Constraint

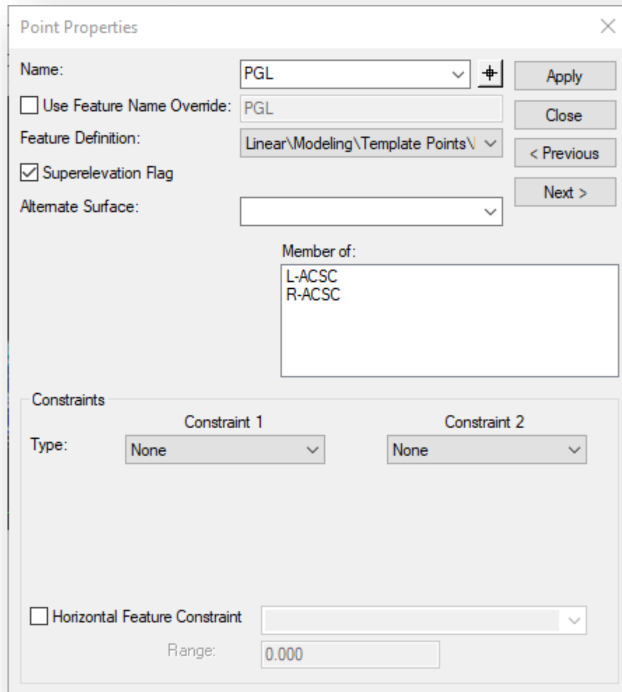
Multiple lines will be shown for points with two constraints.

Floating the cursor over a point will temporarily display an arrow designating the direction of the constraint (the arrow points to the child point) and a pop-up menu displaying information about the point.

The Point Properties Dialog

The ***Point Properties*** dialog is used to review and edit various properties associated with the template points, including the definition of the point constraints. To access the ***Point Properties*** dialog, in the ***Create Template*** dialog double-click on the point in the **Current Template** window. The ***Point Properties*** dialog, shown below, is opened.

The dialog contains the following options:



Name displays the point name. The point name can be keyed-in or selected from the drop-down list. If the point name is selected from the list, the corresponding **Feature Definition** is automatically assigned. Point names must be unique to the template.

Use Feature Name Override displays the name of the feature that will be created in the surface to correspond to the point. This field is optional. If it is blank, then the point name will be used as the feature name.

- The option is intended primarily for end condition components to create connectivity from one station to the next when the template end conditions change. For example, if you want all surface tie-in points to belong to one feature, then set the feature name of all the ending end condition points on each side of the template to the same name (i.e. all Cut and Fill end points on the right would be given the feature name R-Tie and all the ones on the left would be given the feature name L-Tie).
- If the point has a **Feature Name Override** defined, the point name is displayed in red in the Template window.

Feature Definition is used to define the feature definition of the point. If no style is specified, then it comes from the first component of which the point is a member.

The **Superelevation Flag** is used to identify the point as a candidate to be used for assigning superelevation control lines. This option should only be toggled on four pavement points on the surface of the pavement.

Alternate Surface allows you to specify the name of an alternate surface for a point. You may choose multiple alternate surface names. Select from the list of available surfaces in the active template.

Member Of indicates in which components the point is included.

Constraints

The **Constraints** portion of the ***Point Properties*** dialog allows you to review/edit the constraints on a point.

Type specifies the constraint type: **None**, **Horizontal**, **Vertical**, **Slope**, **Vector-Offset**, **Project to Surface**, **Project to Design**, **Horizontal Maximum**, **Horizontal Minimum**, **Vertical Maximum**, **Vertical Minimum**, **Angle Distance**. The constraints are described in more detail below.

Setting up the appropriate constraint types is critical to achieve the desired results when pavement layers, superelevation, and transitioning are introduced to the design.

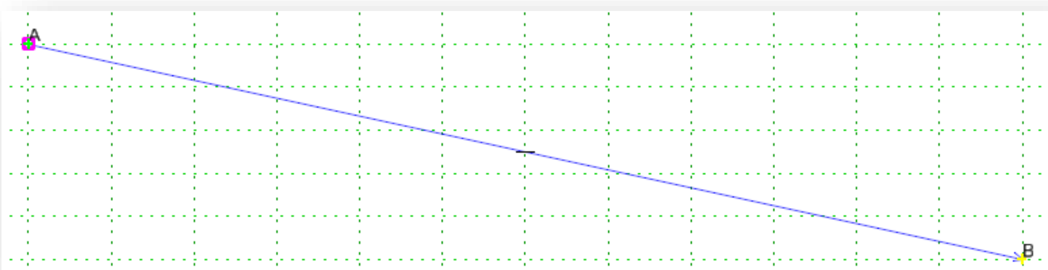
Label displays the optional label for the constraint. Constraints that are labeled can have their value changed during design processing. The same label name can be assigned to more than one constraint and more than one point. See Section 10.4 for details.

Horizontal Feature Constraint is used to allow a point to target elements in the design file that are defined with the specified **Feature Definition** within a specific **Range**. The use of this option is not recommended.

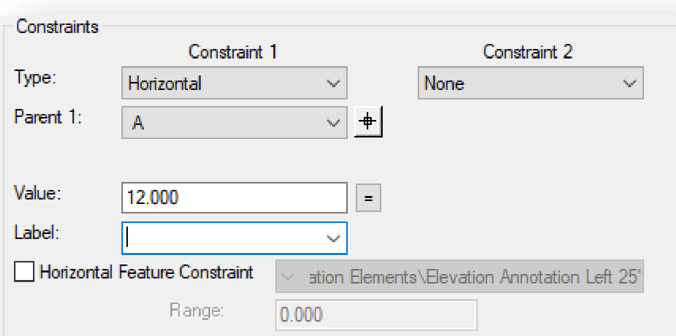
Point Constraint Types

Horizontal

The child point remains at the given horizontal distance from the parent point.



In the example above, point **B** has been constrained to point **A** with one constraint as shown below.

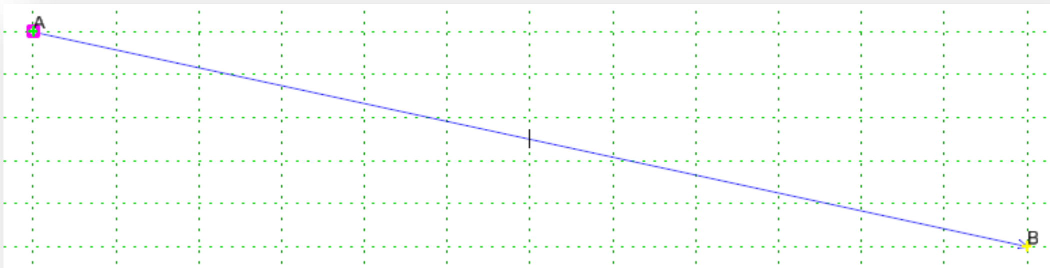


If point **A** is moved horizontally, point **B** will maintain its relative distance from point **A**.

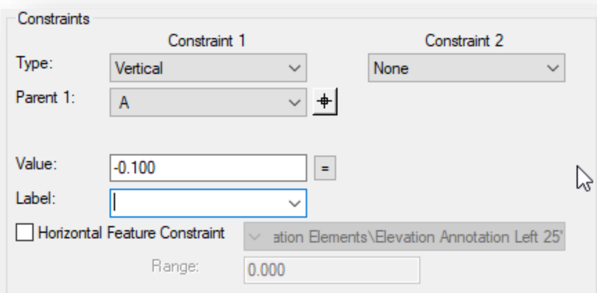
If point **A** is moved vertically, point **B** is unaffected.

Vertical

The child point remains at the given vertical distance from the parent point.



In the example above, point **B** has been constrained to point **A** with one constraint as shown below.

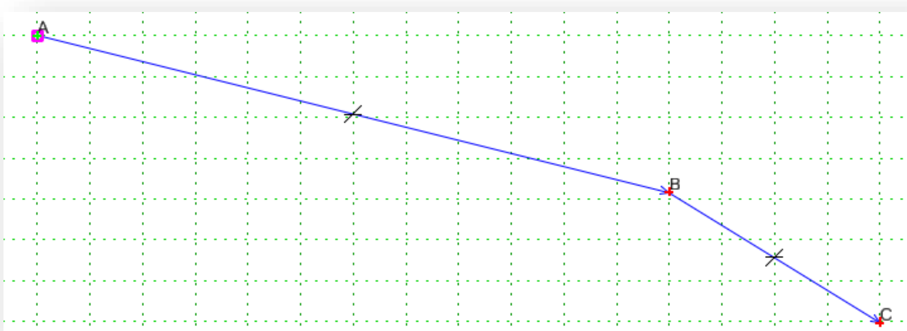


If point **A** is moved horizontally, point **B** is unaffected.

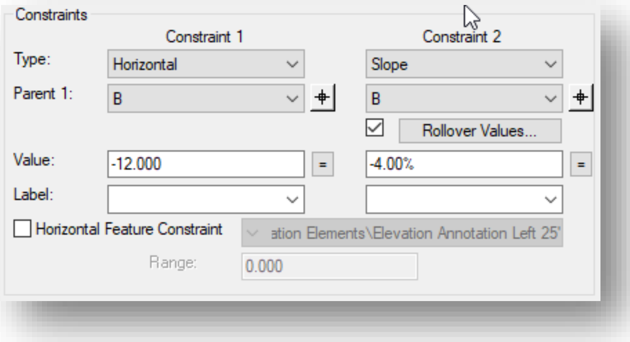
If point **A** is moved vertically, point **B** will maintain its vertical relationship to point **A**.

Slope

The child point will maintain the given slope from the parent point.

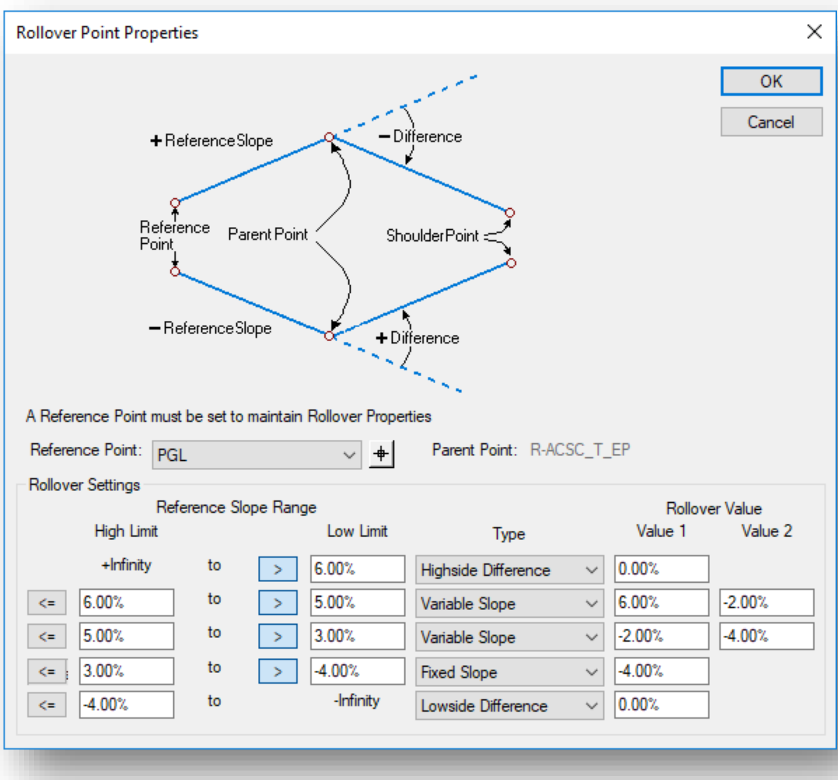


In the example above, point **C** has been defined with a **Horizontal** and a **Slope** constraint as shown below:



Slope constraints are absolute. Slopes going from lower-left to upper-right are positive regardless of whether the child point is to the left or right of the parent.

Slope constraints can also have **Rollover Values** assigned to them. Rollover values are used to set the slope constraint based on a variety of high side and low side slope parameters. A **Reference Point** is specified which defines the controlling slope to the parent point.



The example above documents the **Rollover Values** can as defined in ODOT's example templates for the shoulder break point behavior. See the online help for more information on the available parameters.

Horizontal Maximum

The child point has two parent points and remains at the given horizontal distance from the parent point that is farthest to the right (has the maximum horizontal or X value).

Horizontal Minimum

The child point has two parent points and remains at the given horizontal distance from the parent point that is farthest to the left (has the minimum horizontal or X value).

Vertical Maximum

The child point has two parent points and remains at the given vertical distance from the parent point that is highest (has the maximum vertical or Y value).

Vertical Minimum

The child point has two parent points and remains at the given vertical distance from the parent point that is lowest (has the minimum vertical or Y value).

Vector Offset

The child point has two parent points and will be projected onto the vector defined by the two parents. If the offset is not zero, then the child point will maintain a perpendicular offset from the parent vector at the specified offset value. Negative values indicate an offset to the left of the vector defined by the parent points. Positive values indicate an offset to the right.

Project to Surface

This constraint must be used in conjunction with one of the previously defined constraints. The other constraint will define the projection direction. The child point will then be projected to the surface with the name or parametric label given when the design is processed. If the surface does not exist, or no solution is found, then the point will remain where it is placed in the template.

Project to Design

This constraint is like the Project to Surface, except that the point is projected to the design surface of the template. A projection value is given to indicate whether the projection is to be to the left or to the right. The point must also be constrained by one of the previous constraints, excluding the Project to Surface, so that a direction for the projection may be determined. A negative value limits the projection to the left of 0; a positive value limits the projection to the right. A value of 0 will seek to the left and to the right of 0 to project the point. If no solution is found, then the point will remain where it is placed in the template.

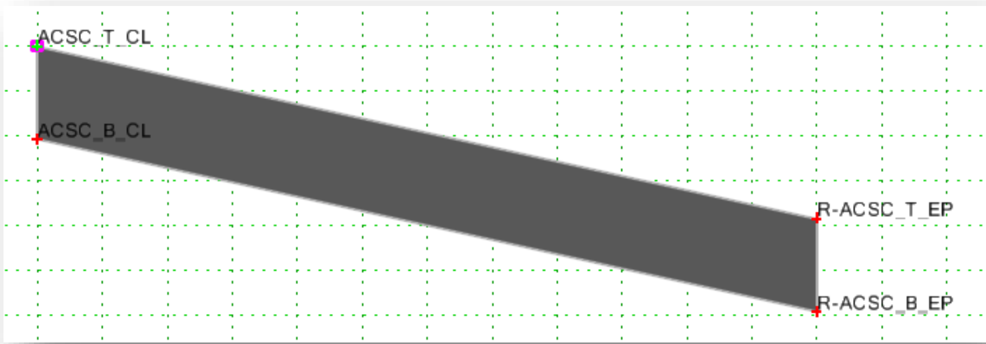
Angle Distance

This constraint takes two parent points, a distance, and an angle. The selected point is then fully constrained to the location defined by the first parent, and the angle from the first parent relative to the vector defined by the two parent points. This constraint creates a rigid-body rotation. When selected, no other constraint types are available.

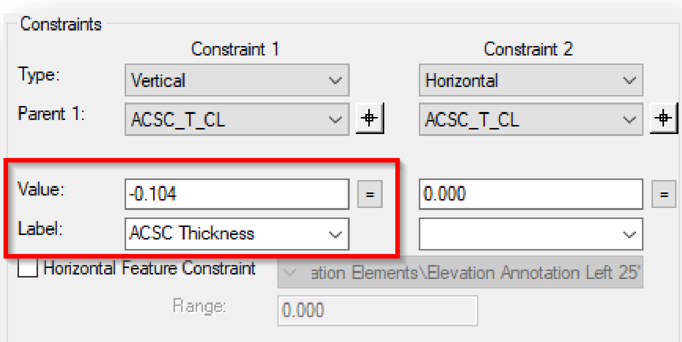
Constraint Labels

Constraint **Labels** can be assigned for points in your template that may have a variable value in your design, or a different value from one project to another. The value for the **Label** can be modified in the corridor modeling process. Assigning labels will allow your templates to be more flexible and therefore make them useful for more than one project.

In the example below, a simple component has been created representing the Asphalt Concrete Surface Course for one lane of pavement.

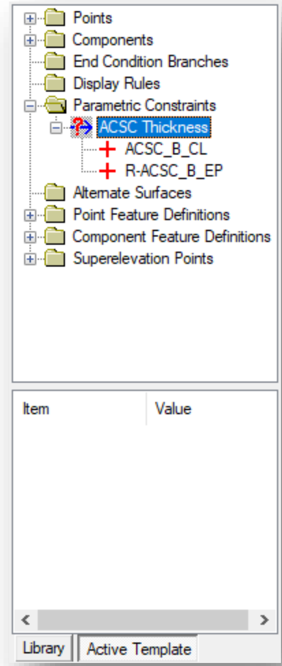


The depth of the pavement at points **ACSC_B_CL** and **R-ACSC_B_EP** has been defined as -0.1042' as shown below.



The thickness of the pavement can be changed by editing the template points, or by adding a **Label**. In the example above, the **Label** "ACSC Thickness" has been assigned for point **ACSC_B_CL** and for point **R-ACSC_B_EP**.

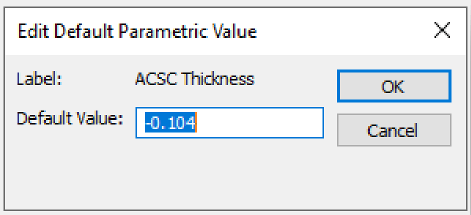
The **Label** name is user definable. The same label name is used for each point in the template that will use the value of the label. Once a label name has been assigned, it is available to be selected from the drop-down label list as other points are defined.



The value of the label can be changed in the **Create Template** dialog, or during the corridor modeling process.

To change the value in the **Create Template** dialog, select **Parametric Constraints** in the **Active Template** option. The points that have been assigned the label are listed as shown at right.

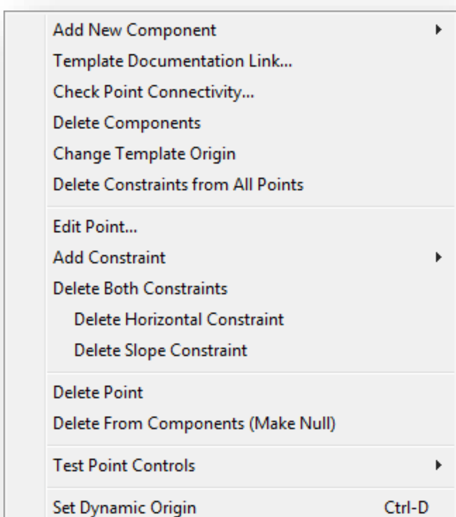
Double-click the name of the **Parametric Constraint** that you wish to edit. The *Edit Default Parametric Value* dialog show below is opened which allows you to edit the value.



The template graphics are automatically updated to reflect the new value.

Editing Points with the Right-Click Menu

In addition to the *Point Properties* dialog, points can be edited from a pop-up menu that is accessed by right-clicking on a point in the **Active Template** window. The menu is shown below.



Some of these commands are covered later in this training guide. See the online help for documentation of commands not covered in this document.

Adding Components to the Template

A component is a set of points that define an open or closed shape.

When defining new components using precision input, the sign of the distance is dependent on the parent-child relationship of the points. The parent is always the first point placed when creating template components. The following rules apply:

- A child placed to the right of the Parent point is at a positive distance
- A child placed to the left of the Parent point is at a negative distance
- A child placed above the Parent point is at a positive distance
- A child placed below the Parent point is at a negative distance
- The Slope is the algebraic slope

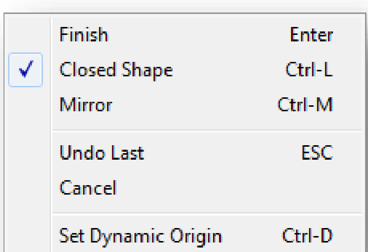
There are six types of components:

- Simple
- Constrained
- Unconstrained
- Null Point
- End Condition
- Overlay/Stripping

Each component type is described in further detail on the following pages.

As you add components to your template, the area under the active template window is used to display various parameters for the component. This is called the **Current Component** area.

Right-click in the current template window at any time during the component creation process to access the menu shown below:



Select **Finish** when you have completed creation of the component.

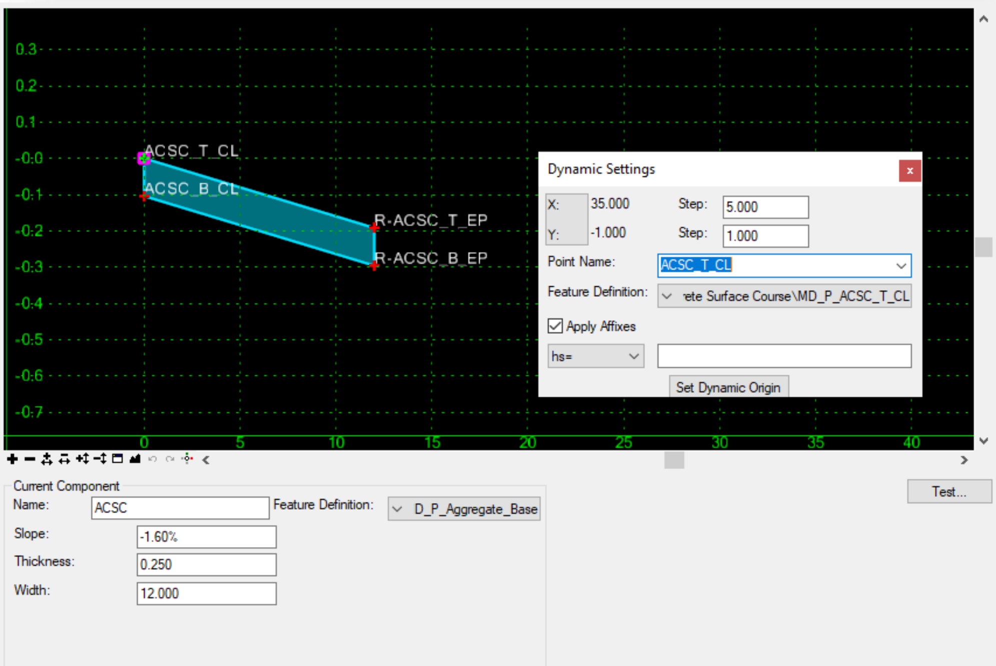
Creating Simple Components

A simple component consists of four points defined by **Slope**, **Thickness**, and **Width**. The top points are constrained by **Horizontal** and **Slope** constraints. The lower points are constrained to the points directly above them by a **Horizontal** and **Vertical** constraint.

To create a simple component, right-click in the active template window and select **Add New Component > Simple** or select **Add > Simple** from the drop-down menu of the *Create Template* dialog to initiate the command.

Define the **Name**, **Slope**, **Thickness**, **Width**, and **Feature** of the component before placing the component in the template.

The **Feature** is used to set the symbology for the graphics drawn to represent the component in the 3D model.



The *Dynamic Settings* dialog is used to facilitate component placement. The point names are derived from the **Point Name** selected in the *Dynamic Settings* dialog. Each point is appended with a number that is incremented. The points are assigned the **Point Style** feature that is active in the *Dynamic Settings* dialog.

Point names and the feature that is assigned to each point should be reviewed and edited after the component is placed.

Creating Constrained and Unconstrained Components

Constrained or **Unconstrained** components can be used to create any open or closed shapes in the template such as curb and gutter, median barriers, retaining walls, or complex pavement situations such as the shoulder break on the high side of superelevation.

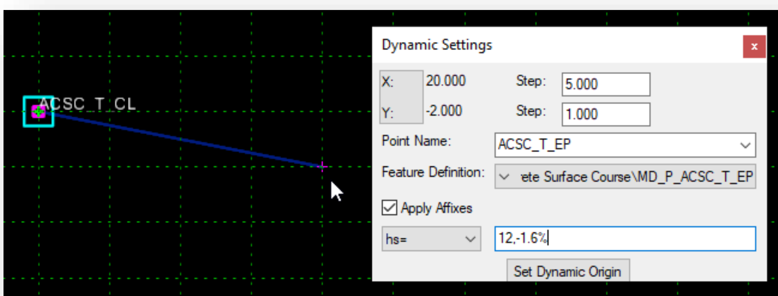
Constrained Components
To create a constrained component, right-click in the current template window and select **Add New Component > Constrained** or select **Add > Constrained** from the pull-down menu of the *Create Template* dialog to initiate the command.

When constrained components are created, both horizontal and vertical constraints are applied to each point.

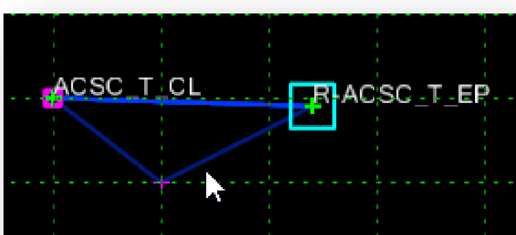
Unconstrained Components
To create an unconstrained component, right-click in the current template window and select **Add New Component > Unconstrained** or select **Add > Unconstrained** from the pull-down menu of the *Create Template* dialog to initiate the command.

When unconstrained components are created, no constraints are applied. This option is useful when you wish to define constraints other than Horizontal and Vertical constraints.

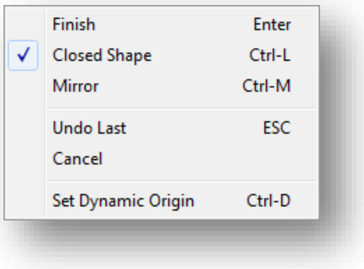
The *Dynamic Settings* dialog can be for precision input when creating the component. In the example below, point **ACSC_T_EP** will be created at a defined horizontal distance and slope from point **ACSC_T_CL** using a horizontal distance of 12 and a slope of -1.6% by using the **hs=12** option with a value of **12, -1.6%**.



Use the **Enter** key to execute the command.



Right-click in the current template window at any time during the component creation process to access the menu shown below:



- Select **Finish** when you have completed creation of the component.

- Components can be created as open or closed shapes. To toggle on/off the option to placed closed shapes, select **Closed Shape**.

- Select **Mirror** to create two components at the same time. The second component is a mirror image of the one you are defining. The component is mirrored about a vertical line passing through the **Dynamic Origin** location.

- Select **Undo Last** to undo the last point that was added to the component.

- Select **Cancel** to terminate the component creation process.

- Select **Set Dynamic Origin** to specify the location of the dynamic origin.

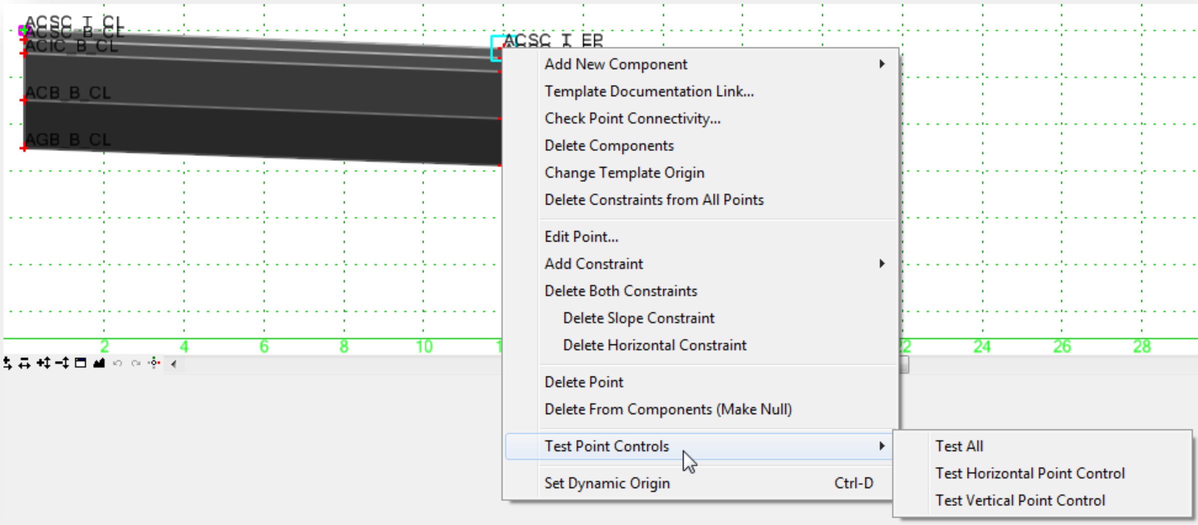
Null Points

A null point is a template point that is purposely not related to any component. It's most often used as a reference for controlling other points.

To create a null point, right-click in the current template window and select **Add New Component > Null Point** or select **Add > Null Point** from the pull-down menu of the *Create Template* dialog to initiate the command.

Testing Point Controls

The behavior of fully constrained points can be tested by right-clicking on the point and selecting **Test Point Controls** as shown below.



Select the desired option to test the behavior of a template as the point is moved either horizontally, vertically, or both at the same time.

ODOT Standard Point and Component Names and Features

As you create templates, each point and component in the template is assigned a **Name** and a **Feature Definition**. It is important to ensure that the points and components are named and assigned features consistently in each template.

Feature definitions for template points and components are defined in **OHDOT_Feature_Definitions.dgnlib**, which is attached by a configuration variable.

Point Names and Features

As the template is applied to the corridor, 3D line strings are drawn in the design file by connecting the points from template drop to template drop by their **Name**. The symbology of these line strings is controlled by the **Feature Definition** that is assigned to each point.

The ODOT workspace for OpenRoads includes several **Feature Definitions** that can be used for template points.

- The point feature definitions can be reviewed by browsing to the following folder in the Civil Standards tab:

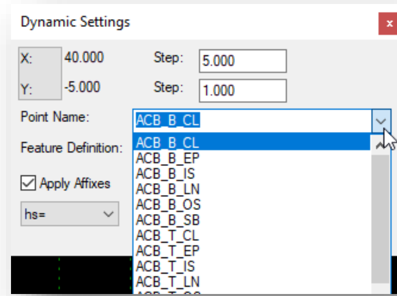
  Libraries > Feature Definitions > Linear > Modeling > Template Points

Features can be reviewed and assigned in a variety of ways.

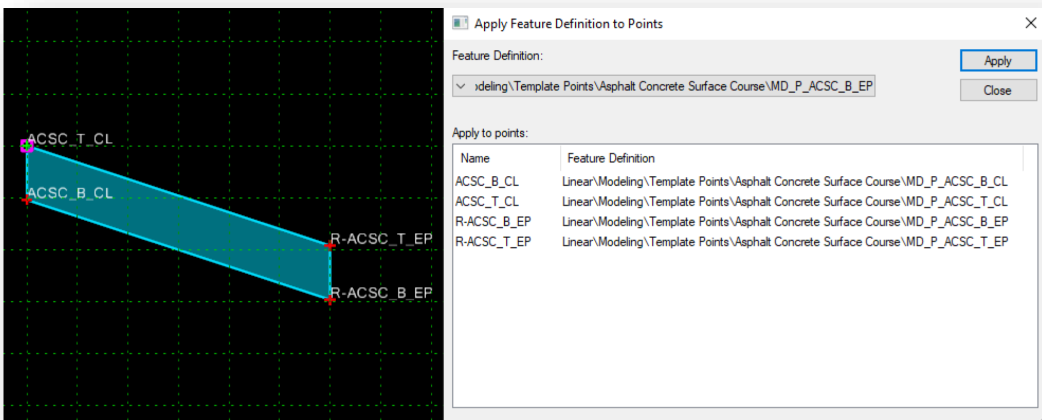
- We have already discussed the *Point Properties* dialog which is accessed by double-clicking the point in the **Current Template** window. The **Name** and **Feature Definition** can easily be edited using this dialog.

- The *Dynamic Settings* dialog can be used to define the **Point Name** that will be assigned to points as they are created. The **Point Name** field can be used to key-in a name for the point or to select a name from a predefined list as shown at right. It is recommended that you use the point names from this list when designing your templates.

  When a point name is selected from the list, the **Point Style**, which is the **Feature Definition**, is automatically set to the appropriate value. The **Point Style** list contains a list of all the **Feature Definitions** available in the OHDOTCEv02 workspace.



- Features can also be assigned to one or more points by selecting **Tools > Apply Feature Definition to Points** from the *Create Template* dialog as shown below.



Component Names and Features

Component Names and **Features** are defined by the user as each component is created.

- The component feature definitions can be reviewed by browsing to the following folder in the Civil Standards tab:

  Libraries > Feature Definitions > Mesh

Component Names and Features can be reviewed and edited in a variety of ways:

- Double-click on a component to open *the Component Properties* dialog.

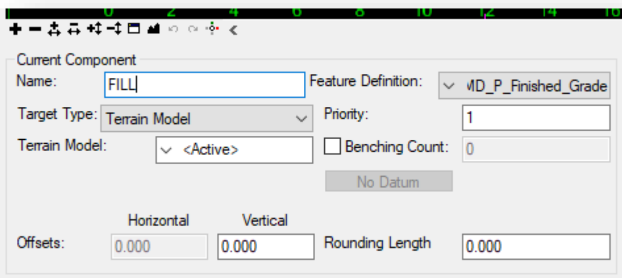
- Select **Tools > Apply Feature Definition to Components…** to apply a selected **Feature Definition** to one of more components in the template.

End Conditions

An end condition is a special open-shaped component that targets a surface, a feature of a surface, an elevation, or an alignment. End Conditions are commonly used for cut and fill definitions. End Conditions are added to the end of simple, constrained, or unconstrained components.

To create an end condition, right-click in the template window and select **Add New Component > End Condition** or select **Add > End Condition** from the *Create Template* dialog's pull-down menu.

When adding an end condition to the current template, the **Current Component** portion of the *Create Template* dialog displays the following options:



The **Name** field is used to specify the user defined name for the component.

The **Feature** is used to set the feature definition for the component.

Target Type specifies the type of target the End Condition is seeking. Targets include:

- **Terrain Model** – Seeks an OpenRoads Terrain Model
- **Elevation** – Seeks an elevation
- **Linear Horizontal/Vertical/Both** – Seeks the intersection with a linear element selected from a list.
- **Feature Definition Horizontal/Vertical/Both** – Seeks the intersection with a feature type.

Horizontal Offset specifies a horizontal offset from the target. This option only applies if the target is a horizontal target or has a horizontal aspect.

Vertical Offset specifies a vertical offset from the target. Applies only if the target is a vertical target or has a vertical aspect. For an elevation target, specify the elevation here.

Priority specifies the order in which end conditions are attempted in seeking a solution. Where more than one End Condition starts at the same point, this value determines the order in which End Conditions are attempted to find a solution. Lower numbers, which have a higher priority, are attempted first.

Benching Checkbox indicates, when on, the set of segments of the End Condition are repeated until a solution is found.

> The **Benching Count** indicates the maximum number of times that benching should occur. If this value is 0, then the repeat count is unlimited, and the benching End Condition must find its target for it to be valid. If any other number is entered, then if the benching is repeated for the specified

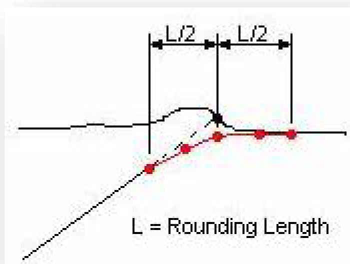
count and if the target is not reached, the benching End Condition will still be considered successful if an End Condition attached to the end of the benching End Condition is successful.

From Datum indicates, when on, the benching will use a datum elevation and step elevation to set the benching rather than using the End Condition segments as they are defined.

Datum Elevation specifies the elevation that represents the basis for the step elevation. Datum + Current bench elevation = n * Step Elevation.

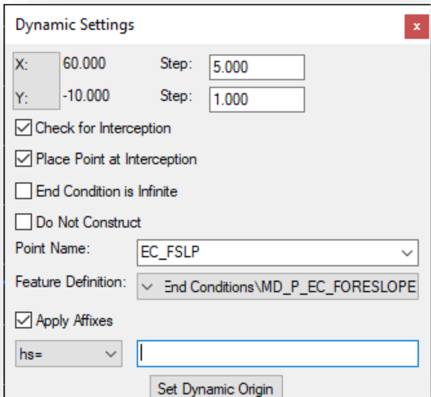
Step Elevation specifies the increment for benching when a datum is being used. Each bench elevation is always located at the end of the first segment (or the second point) of the benching End Condition.

Rounding Length specifies the length used to round the end condition. When this value is something other than 0.0, 4 additional points are added to the end of the end condition using a parabolic formula to smooth out the transition of the intersection of the end condition with the surface. This rounding effect is only applied when the target is a terrain model or an elevation. Any other end conditions attached to the end of this one is attached to the end of the rounded section.



End Condition Point Properties

As you are defining points for an end condition, the *Dynamic Settings* dialog includes additional options for the point definition as shown below.



Check for Interception specifies the preceding segment will be checked for interception of the target when the end-conditions are solved. If it is turned off, then the segment will be ignored when checking for interceptions.

Place Point at Interception specifies that when an interception is found along the preceding line segment, the line segment will be drawn to the interception. Otherwise, if the interception is found, the line segment will be drawn to its maximum width.

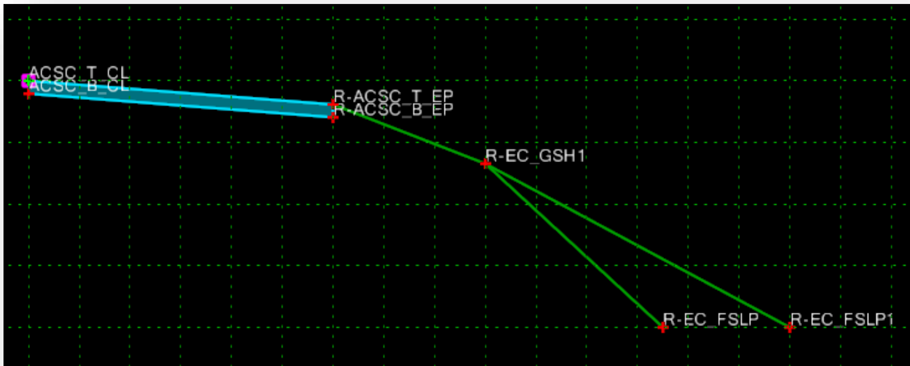
End Condition is Infinite specifies the last segment will be of infinite length when attempting to intercept the target. This option is applicable only to the last point on the end condition.

Do Not Construct specifies the point will be solved like all other points in the end-condition, but the point will be skipped when drawing the component segments.

These end condition point properties can be edited in the *Point Properties* dialog. Double click on an end condition point to review or edit the properties.

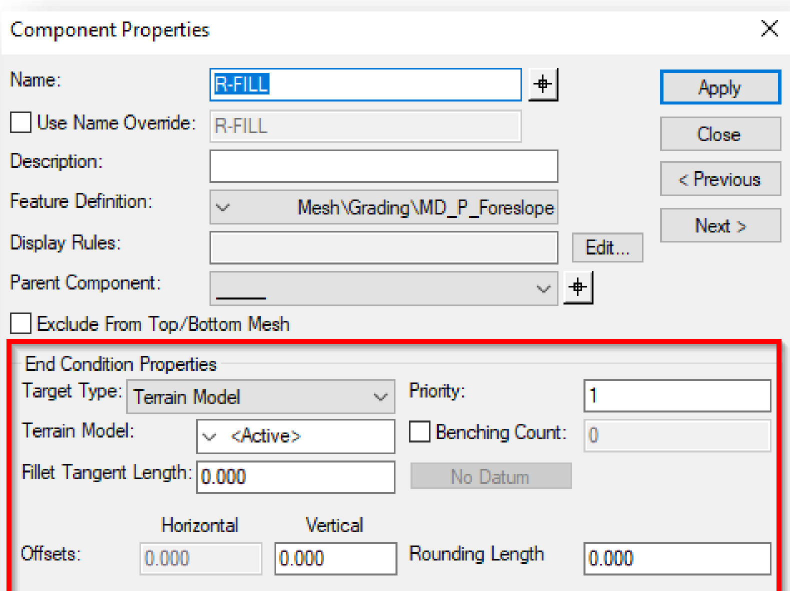
End Condition Priority

The end condition priority value is used when more than one end condition is defined starting at the same point. A simple end condition consisting of two fill lines is shown in the example below.

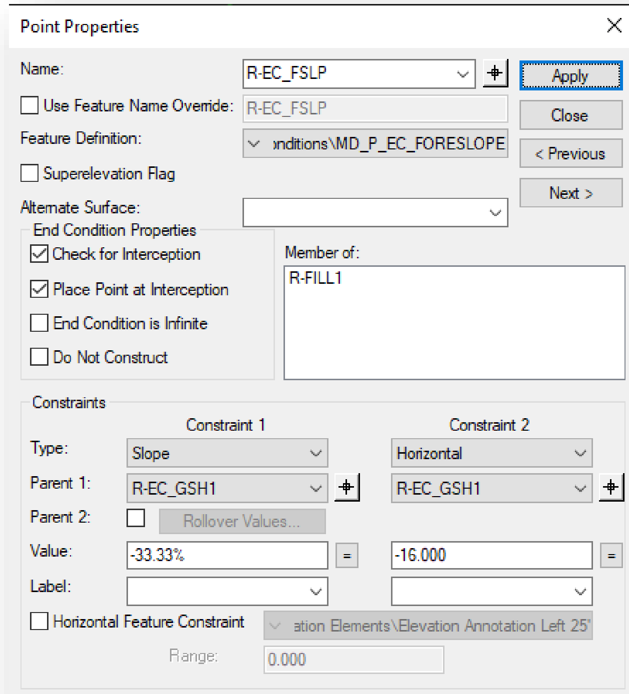


When the end condition is processed in corridor modeling, each end condition will be tested in the order specified by the user. The order is determined by the **Priority** value entered by the user during the end condition creation. The most desirable solution, and the first condition to be tested, is given a priority value of 1.

Double-clicking on a component will open the *Component Properties* dialog shown below. If the component is an end condition, the **End Condition Properties** are displayed. The **Priority** is set as shown below.



The properties for point R-EC_FSLP were defined as shown at right. **Check for Interception** is toggled “on”, and **End Condition is Infinite** is toggled “off”. The point is defined with a value of -33.33% Slope for a vertical distance of 16.0’ from the graded shoulder point R-EC_GSH1.



The properties for the point R-EC_FSLP1 were defined to specify a 2:1 slope with the **End Condition is Infinite** option toggled on. The priority for the end condition component R-FILL_2 is set to a value of 2.

With these settings, the end condition R-FILL_3 will be tested first (Priority=1) for an interception with the surface within the vertical distance of 16.0’. If an interception is found within the vertical distance, the end condition is used for the cross section and a point will be placed at the interception.

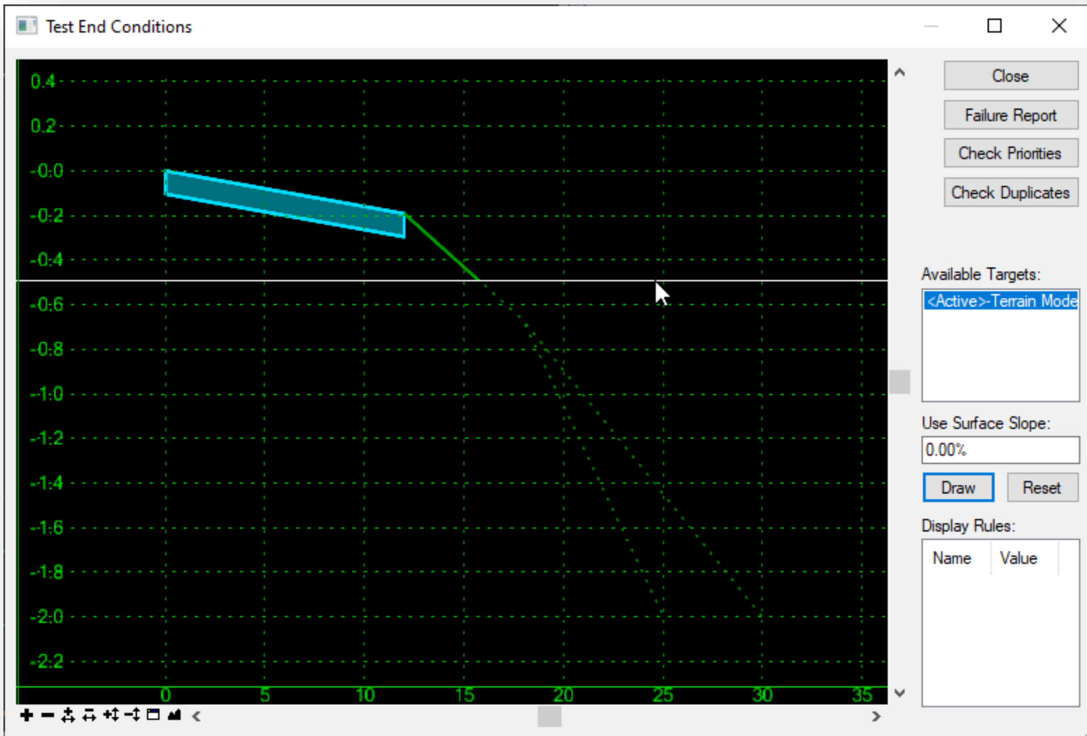
If no interception is found within 16.0’ vertically from point R-EC_GSH1, the condition fails and the software will test the next end condition, R-FILL_2, which has a Priority value of 2.

Since **End Condition is Infinite** is toggled on for point R-EC_FSLP1, the solution will solve for an interception with the surface regardless of the vertical constraint definition.

Testing End Conditions

End conditions can be tested in the *Create Template* dialog before processing the template with corridor modeling. Testing simulates how the end condition will behave when the corridor is processed.

This is extremely useful for complex end condition definitions with multiple solutions. To test the end condition, select the **Test** command, located below the **Current Template** window. The *Test End Conditions* dialog is opened.



The software will check for conflicts in the priorities defined in the template before opening the *Test End Condition* dialog. If conflicts exist, a warning is issued.

The dialog displays the current template. Non-solved end conditions, and their child components, are displayed in dashed lines. Normal components and solved end conditions are displayed as solid lines.

To test the end condition, select one of the available targets and then select the **Draw** button. Move your cursor over to the testing window. The end conditions will change from a dotted line to a solid line indicating the solution for the target interception.

Failure Report activates the *Results* dialog. When you fail to get a solution on one or both sides, the results show which components failed and which end condition start points were not solved.

Check Priorities tests the template for priority conflicts. When conflicts exist, a message is displayed indicating there are end conditions that start from the same point and have the same priorities. The *End Condition Priority Conflicts* dialog is displayed. If no conflicts exist, a message box indicating no conflicts is displayed.
Available Targets lists all the targets applicable to the current template. You can select the desired target and draw it in the display.

Use Surface Slope specifies the slope of the surface that is being placed. This is useful in certain instances where an end condition intersects a surface more than once, such as a ditch profile.

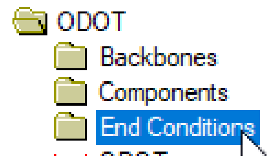
Draw draws the selected target. Surfaces and elevations are drawn as horizontal lines. Features and alignments are drawn as a "+" mark. As you draw a target, the display dynamically updates to show the solution(s).

Reset clears all the drawn targets.

Display Rules list the display rules for the current template. Left click the **Value** to toggle between **True** and **False**. Press the **Reset** button to restore the original **Display Rules** values.

OhioDOT Released End Conditions

The following section will help explain how to use and work with the Template end conditions that are released in the OHDOT standards.

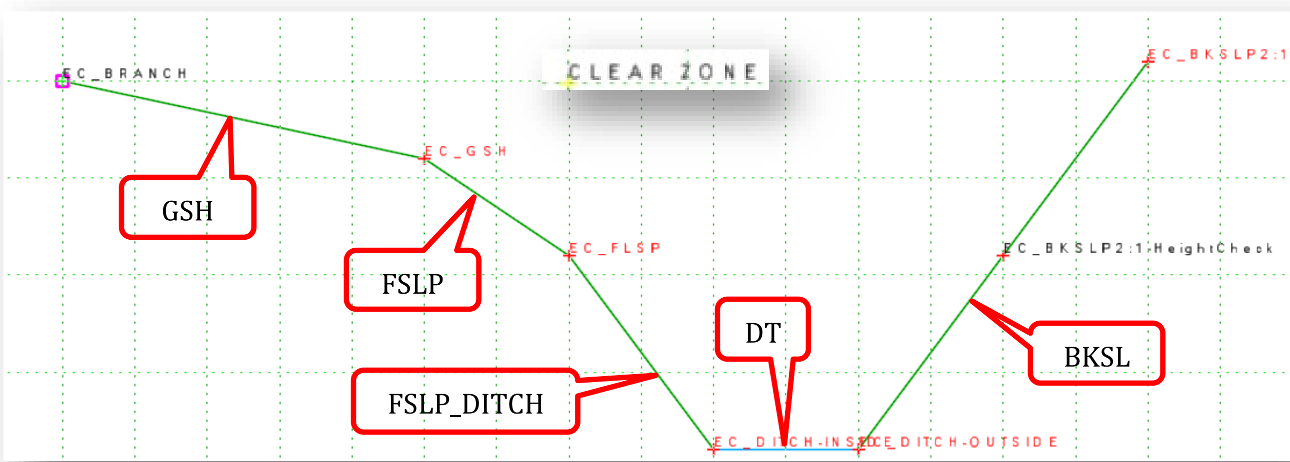


The end condition templates are based off the **Location & Design Manual, Volume 1 – Roadway Design, Section 300**. These templates can be found within the ODOT\End Conditions folder inside the worksets .itl file.

Standard Points and Components for End Conditions

While end conditions or grading can get very complicated in certain situations these templates were set up to cover most of the standard cases and can be used as a starting point for more complicated situations. To make sure all points and components will connect correctly from drop location to drop location the following points and components have been set up.

Grading with Ditches



See tables on the next page for more details on the points and components that make up this template.

- Note that the foreslopes consist of three components. A Graded Shoulder, Foreslope and Ditch Foreslope. If only two different foreslopes are needed, then instead of deleting the foreslope component, simply change the constraints on the EC_FLSP point so that the point moves to be at the same location as the EC_GSH point. This is done so that if another template is used in the corridor that uses three foreslopes the two template drops will be able to connect all the points and components together and not leave gaps in the model.

- Note that in some cases the EC_FLSP point will represent the clear zone location. Is this situation the CLEAR ZONE point should be used to set this distance and the EC_FLSP point should be horizontally constrained about the CLEAR ZONE POINT.

- Note the EC_BKSLP#:1-HeightCheck point is an optional point that can be added to the BKSLP component to represent the minimal ditch depth. Set the check for interception checkbox to be checked off to ensure the end condition fails if the minimum depth is not met, to ignore this remove the point or check on the check for intersection checkbox.

- Note when adding multiple end condition branches to a template, simply add a number to the end of point/component names and increment that for additional branches. Ensure override names remain the same.

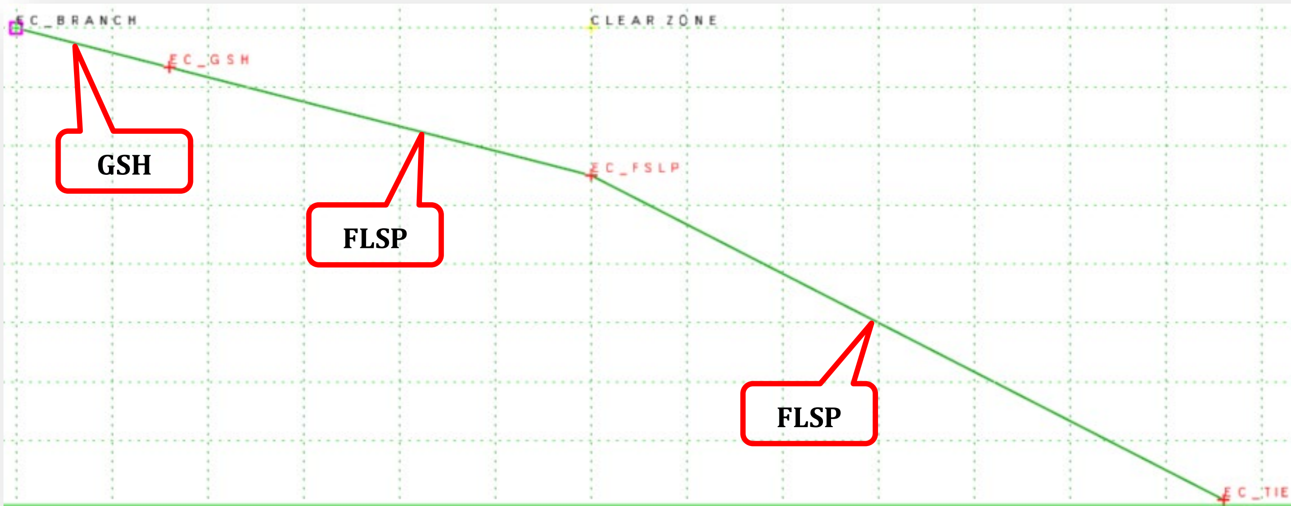
- Note the last point in an end condition branch should always have an override name of TIE. It can be used as a basis for construction limits.

- Note backslope points can optionally contain the slope in the name.

Point Name	Override Name	Desc	Constraint	Type	Parent	Label	NOTES
EC_BRANCH	--Merge with edge of shoulder point--						
EC_GSH	GSH	point for end of graded shoulder	Constraint 1	Slope	EC_BRANCH	GSH_SLOPE	Graded Shoulder slope
			Constraint 2	Horizontal	EC_BRANCH	GSH_WIDTH	width of graded shoulder
EC_FSLP	FSLP	point for end of foreslope, if not needed change constraint #2 value to 0, keep for connection to other templates	Constraint 1	Slope	EC_GSH	FSLP	Foreslope
			Constraint 2	Vertical	EC_GSH	FSLP_DEPTH	Depth of foreslope, if this point needs to match Clear Zone, then change to be horizontally constrained about CLEAR ZONE Point
EC_DITCH-INSIDE	DITCH-INSIDE	point for end of ditch foreslope/inside bottom of ditch	Constraint 1	Slope	EC_FLSP	DITCH_FSLP	Foreslope of ditch
			Constraint 2	Vertical	EC_FLSP	DITCH_DEPTH_NEGATIVE	depth of ditch
EC_DITCH-OUTSIDE	DITCH-OUTSIDE	point of outside bottom of ditch	Constraint 1	Horizontal	EC_DITCH-INSIDE	DITCH_WIDTH	width of ditch
			Constraint 2	Vertical	EC_DITCH-INSIDE		leave at 0
EC_BKSLP#:1	TIE	point for end of backslope	Constraint 1	Slope	EC_DITCH-OUTSIDE	DITCH_BKSLP	backslope
			Constraint 2	Vertical	EC_DITCH-OUTSIDE		height of backslope
EC_BKSLP#:1-HeightCheck		check point to check min depth of ditch-OPTIONAL	Constraint 1	Slope	EC_DITCH-OUTSIDE	DITCH_BKSLP	backslope
			Constraint 2	Vertical	EC_DITCH-OUTSIDE	DITCH_DEPTH	min. depth of ditch
EC_CZONE	CLEAR_ZONE	Point to represent clear zone - OPTIONAL	Contraint1	Horizontal	EC_BRANCH	CZ_WIDTH	Sets width of clear zone, can be used to move EC_FLSP point

Component Name	Override Name	Description	Parent
GSH	GSH	Graded Shoulder	
FSLP	FSLP	Foreslope-keep for connection to other templates	GSH
FSLP_DITCH	FSLP_DITCH	Ditch Foreslope	FSLP
DT	DITCH	Ditch	FSLP_DITCH
BKSLP	BKSLP	Backslope	DT

Grading without Ditches



See tables on next page for more details on the points and components that make up this template.

Unlike the ditch template which had three separate components for each part of the foreslope, there are only two components, the graded shoulder and the foreslope. Where the foreslope component consists of a middle point to allow an additional slope change. This was done to ensure components from drop location to drop location connect. If only two different foreslopes are needed, then instead of deleting the point, simply change the constraints on the EC_FSLP point so that the point moves to be at the same location of the EC_GSH point.

- Note when adding multiple end condition branches to a template, simply add a number to the end of point/component names and increment that for additional branches. Ensure override names remain the same.

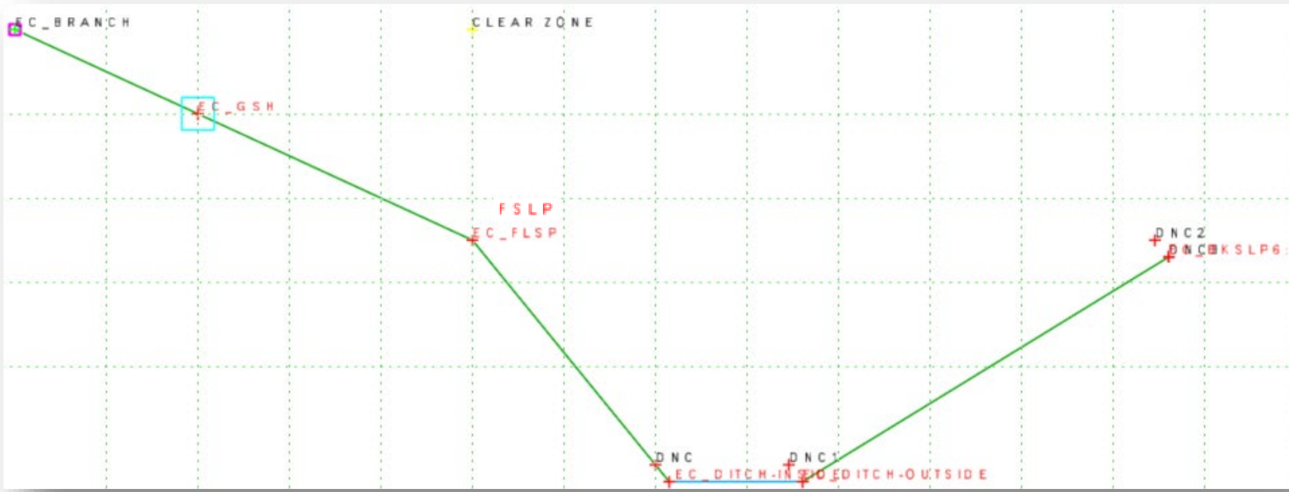
- Note the last point in an end condition branch should always have an override name of TIE. It can be used as a basis for construction limits.

Point Name	Override Name	Desc	Constraint	Type	Parent	Label	NOTES
EC_BRANCH	--Merge with edge of shoulder point--						
EC_GSH	GSH	point for end of graded shoulder	Constraint 1	Slope	EC_BRANCH	GSH_SLOPE	Graded Shoulder slope
			Constraint 2	Horizontal	EC_BRANCH	GSH_WIDTH	width of graded shoulder
EC_FSLP	FSLP	point for slope change of foreslope, if not needed change constraint #2 value to 0, keep for connection to other templates	Constraint 1	Slope	EC_GSH	FSLP	Foreslope
			Constraint 2	Vertical	EC_GSH	FSLP_DEPTH	Depth of foreslope
EC_TIE	TIE	point for end of foreslope	Constraint 1	Slope	EC_FSLP		Foreslope
			Constraint 2	Vertical	EC_BRANCH		Overall depth
EC_CZONE	CLEAR_ZONE	Point to represent clear zone - OPTIONAL	Constraint 1	Horizontal	EC_BRANCH	CZ_WIDTH	Sets width of clear zone, can be used to move EC_FLSP point

Component Name	Override Name	Description	Parent
GSH	GSH	Graded Shoulder	
FSLP	FSLP	Foreslope	GSH

Fixed Ditch Templates

These templates are set up to force a minimum ditch depth. Once the backslope becomes shorter than the minimum depth then the ditch will be pushed lower in order to maintain the required minimum depth of the ditch. These templates are useful when dealing with fill conditions.



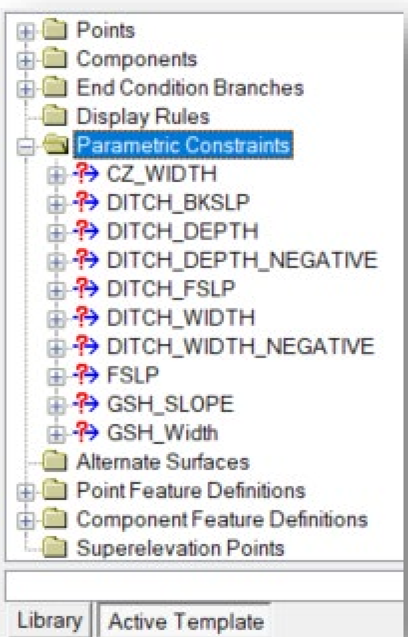
These templates have additional do not construct (DNC) points. These points are needed to set up the template to push the ditch to lower elevations in order to meet minimum ditch depths.

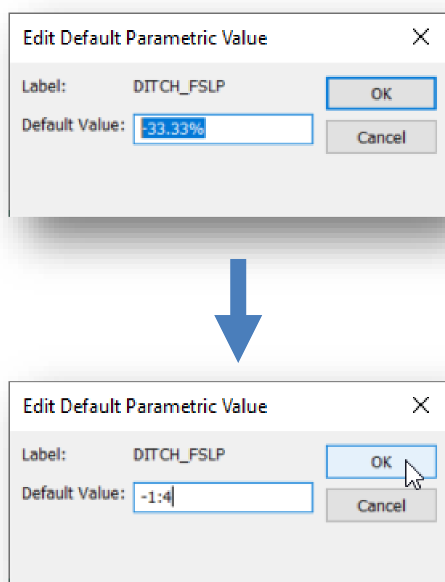
Point Name	Override Name	Desc	Constraint	Type	Parent	Label	NOTES
EC_BRANCH	--Merge with edge of shoulder point--						
EC_GSH	GSH	point for end of graded shoulder	Constraint 1	Slope	EC_BRANCH	GSH_SLOPE	Graded Shoulder slope
			Constraint 2	Horizontal	EC_BRANCH	GSH_WIDTH	width of graded shoulder
EC_FSLP	FSLP	point for end of foreslope, if not needed change constraint #2 value to 0, keep for connection to other templates	Constraint 1	Slope	EC_GSH	FSLP	Foreslope
			Constraint 2	Vertical	EC_GSH	FSLP_DEPTH	Depth of foreslope, if this point needs to match Clear zone, then change to be horizontally constrained about CLEAR ZONE Point
DNC		Do not construct point in line with Ditch foreslope/inside ditch	Constraint 1	Slope	EC_FSLP	DITCH_FSLP	Foreslope of ditch
			Constraint 2	Vertical	EC_FSLP	DITCH_DEPTH_NEGATIVE	Negative Depth of ditch
DNC1		Do not construct point parallel with outside ditch	Constraint 1	Vertical	DNC		
			Constraint 2	Horizontal	DNC	DITCH_WIDTH	Width of ditch
DNC2		Do not construct point parallel with backslope	Constraint 1	Slope	DNC1	DITCH_BKSLP	Backslope of ditch
			Constraint 2	Vertical	DNC1	DITCH_DEPTH	Depth of ditch
DNC3		Do not construct point lines up with end of backslope	Constraint 1	Slope	DNC2	DITCH_FSLP	Foreslope of ditch
			Constraint 2	Vertical	DNC2		
EC_BKSLP#:1	TIE	point for end of backslope	Constraint 1	Horizontal	DNC3		
			Constraint 2	Vertical	DNC3		
EC_DITCH-OUTSIDE	DITCH-OUTSIDE	point of outside bottom of ditch	Constraint 1	Slope	EC_BKSLP#:1	DITCH_BKSLP	Backslope of ditch
			Constraint 2	Vertical	EC_BKSLP#:1	DITCH_DEPTH_NEGATIVE	Negative Depth of ditch
EC_DITCH-INSIDE	DITCH-INSIDE	point for end of ditch foreslope/inside bottom of ditch	Constraint 1	Vertical	EC_DITCH-OUTSIDE		
			Constraint 2	Horizontal	EC_DITCH-OUTSIDE	DITCH_WIDTH_NEGATIVE	negative width of ditch
CLEAR ZONE		Point to represent clear zone - OPTIONAL	Contraint1	Horizontal	EC_BRANCH	CZ_WIDTH	Sets width of clear zone, can be used to move EC_FLSP

Component Name	Override Name	Description	Parent
GSH	GSH	Graded Shoulder	
FSLP	FSLP	Foreslope-keep for connection to other templates	GSH
DNC		Hidden component needed to push ditch down to maintain min. ditch depth	FSLP
BKSLP	BKSLP	Backslope	DNC
DT	DITCH	Ditch	BKSLP
FSLP_DITCH	FSLP_DITCH	Ditch Foreslope	DT

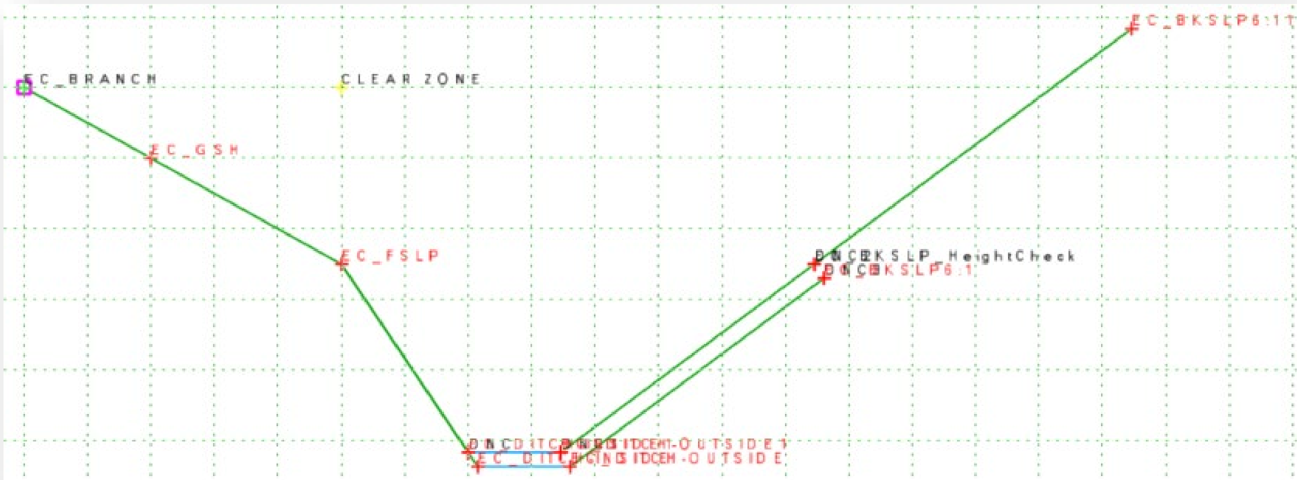
Modifying Fixed Ditch Templates

Since these templates are more complicated to edit, it is recommended you make edits via the Parametric Constraints (Labels). Simply select the Active Template Tab and then expand the Parametric Constraints Section. Double click on a given Constraint (Label) to change its value. For example, to change the Ditch foreslope, double click on the DITCH_FSLP constraint and a dialog will open displaying the current value. Change to the desired value and click okay.





The above screen shots of fixed ditch templates only show the fixed ditch part of the template. The actual templates delivered with the workspace take the [Grading with Ditches](#) template and combines it with the fixed ditch template. An example is shown below. Notice that some points are duplicated and end with a 1, that is okay. The Grading with Ditches part of the template has a higher priority than the fixed ditch part. This way, the backslope will search up to solve first and once the backslope reaches the minimum ditch depth the fixed ditch solves and pushes the ditch deeper to maintain the minimum ditch depth.



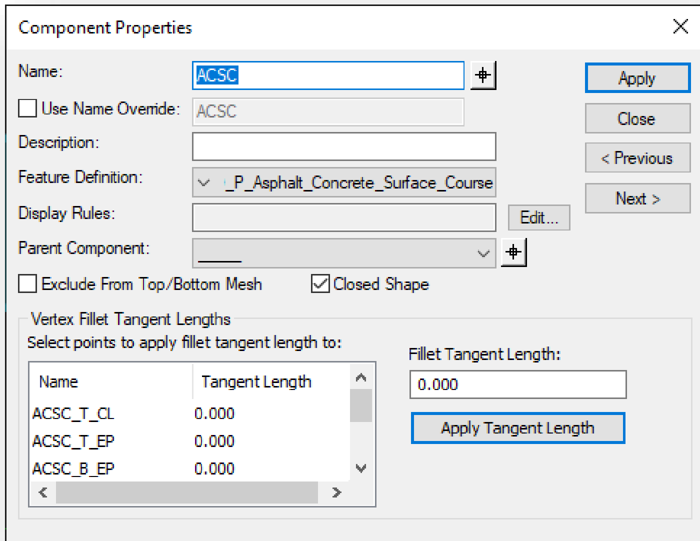
Using Fixed Ditch Templates

Since these ditches are more complicated it is recommended you use them as a starting point to get the ditch profile. Do not try to add point controls off these templates. The simple workflow is as follows.

1. Push fixed ditch template along alignment
2. Open Profile of inside ditch line. There might be bumps in profile and it might be desired to smooth out the profile. REMEMBER when drawing in a new profile to always stay at or below the current profile. The current profile represents the highest elevation that profile can be in order to solve. Going above that profile will cause your end condition to not solve in those areas.
   Also keep in mind the lower you draw in your profile, the further the horizontal distance of the ditch will be from the alignment.
   Make sure to complex and give a feature definition to your drawn in profile.
3. Project profile to centerline alignment.
4. Remove rules from projected profile!
5. Replace fixed ditch template with equivalent non-fixed ditch template
6. Add a vertical point control to the ditch point based off the projected ditch profile.

Component Properties

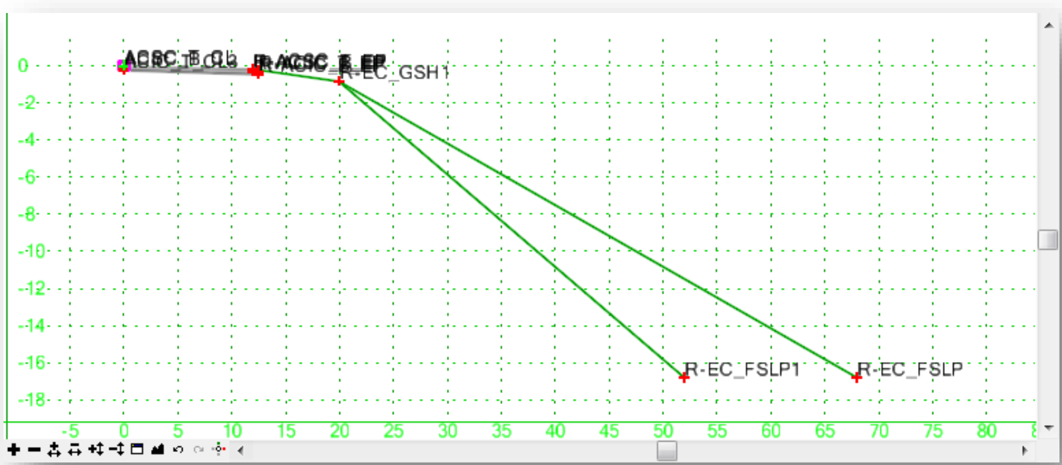
The properties of the component can be edited by double-clicking on the component. The *Component Properties* dialog is opened.



Name
Specifies the name of the current component. Select a different component using the locate button.

Use Name Override
The name override is used to specify the name used by the software to connect components from one template drop location to the next.

The option is intended to create connectivity from one station to the next when the template components change. For example, a template may contain more than one solution for a fill condition as shown below.



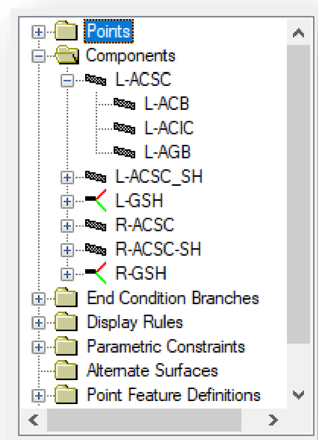
Two end conditions are defined for the fill slopes named R-FILL2 and R-FILL3. When the templates are processed, the software will evaluate each template drop location to determine whether the R-FILL2 or the

R-FILL3 component is used. 3D Mesh elements are created by connecting all the components by the name of the component. All the components named R-FILL2 will be connected to create a 3D Mesh element, and all the components named R-FILL3 will be connected to create another mesh element. This leaves a gap in components at the locations where the fill slope transitions from 3:1 to 2:1 since the components named R-FILL3 will not be connected to the components named R-FILL2. The solution is to assign the same **Override Name** for each component.

Description
Specifies an optional description of the component.

Feature Definition
Specifies the feature style of the component. This option is used for display and to define the component's material. ODOT has defined a specific group of styles to be used for components prefixed with the word "Component". When the components are plotted in MicroStation, either as a 3D model, or as cross sections, the symbology defined by the selected Style is used. Care should be taken to select the appropriate style for each template component.

Parent Component
Specifies the parent component. If a component has a parent component, then it is only displayed if the parent component is displayed. The parent component can be either a non-end condition component, or an end condition component. Specify the component or identify it using the locate button.
The **Active Template** listing can also be used to display and edit the parent child relationship between points and components. In the example at right, the **Components** category is open to show the parent/child relationship between the components in the tree. Drag and drop operations can be used to move one component under another.



Note: The component names are listed alphabetically, not from top to bottom as drawn in the template.

Exclude from Top/Bottom Mesh
Specifies that the component will be excluded from the top and bottom mesh elements that are created in the corridor modeling process.

Close Shape
This option is only available for non-End Condition components. Only closed shapes can be used to compute component quantities.

Section 503 Template Exercises

Exercise 1: Create Templates from Components

In this exercise we will assemble the template for St. Peter Road by dragging and dropping components from the ODOT template library.

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.
- Continue in the file **123456_KM003.dgn**. The design file that is used does not matter since we are editing the template library .itl file.

✓ From the Corridors tab, choose **Template > Create Template**. The ***Create Templates*** dialog is opened with the template library for the WorkSet loaded

✓ Create a folder named **St. Peter Road** to store all the templates for St. Peter Road. Right-click on the **Project Templates** folder and choose the **New > Folder** option.

Take the following steps to create the template for St. Peter Road:

✓ Right-click on the **Project Templates** folder and choose **New > Template** to create the first template for St. Peter Road. Name the template “St. Peter Road”.

✓ Open the ***Dynamic Settings*** dialog (**Tools > Dynamic Settings**) to define the **Step** increment values as 0.5 and 0.1. This is helpful to ensure that the initial pavement placement is constrained on the “grid”

✓ In the ***Dynamic Settings*** dialog, turn off the **Affixes** option. We will drag in a pavement backbone that already has the left and right sides defined.

✓ Drag the following pavement backbone into the template at the (0,0) origin:

> ODOT > Backbones > Undivided Roadways > Pavement and Shoulder 301-8E Uncurbed > 2-Lane > Asphalt Concrete without Stabilization

St. Peter Road will use a pavement width of 10’. Take the following steps to define the pavement width:

✓ Edit point **L-ACSC_T_EP** to define the **Horizontal** constraint with a value of **-10**

✓ Edit point **R-ACSC_T_EP** to define the **Horizontal** constraint with a value of **10**

Next add the end conditions to the template.

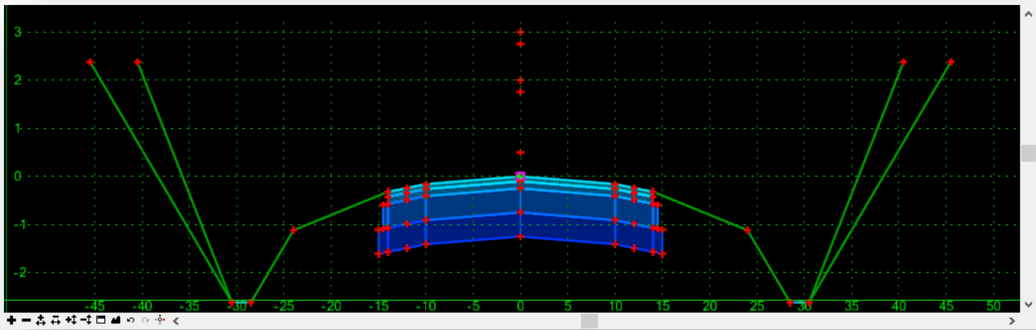
✓ In the *Dynamic Settings* dialog, turn on the **Affixes** option

✓ Drag the following component onto the edge of shoulder point.

ODOT > End Conditions > 307-4E Common and Barrier Grading > 307-4E Cut 3:1

Tip: Since this template is symmetrical, you can right-click while dragging the end condition into the template to activate the **Mirror** option (CTRL+M) to place it on both sides simultaneously.

The completed template should appear as shown below.



✓ When complete, save your template by choosing **File > Save** from the ***Create Template*** dialog pull-down menu.

Exercise 2: Creating the S.R. 185 and Jameson Road Templates

In this exercise we will create the templates for S.R. 185 for and Jameson Road

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.

✓ Open the file **123456_BK001.dgn**. The design file that is used does not matter since we are editing the template library .itl file.

S.R. 185 Template

✓ From the **Corridors** tab, choose **Template > Create Template**. The ***Create Templates*** dialog is opened with the template library for the WorkSet loaded

- Create a new template named **S.R. 185** in the **Project Templates** folder.
- Open the *Dynamic Settings* dialog to ensure the **Step** values are defined as desired.
- Create the S.R. 185 template by dragging in the components listed below.

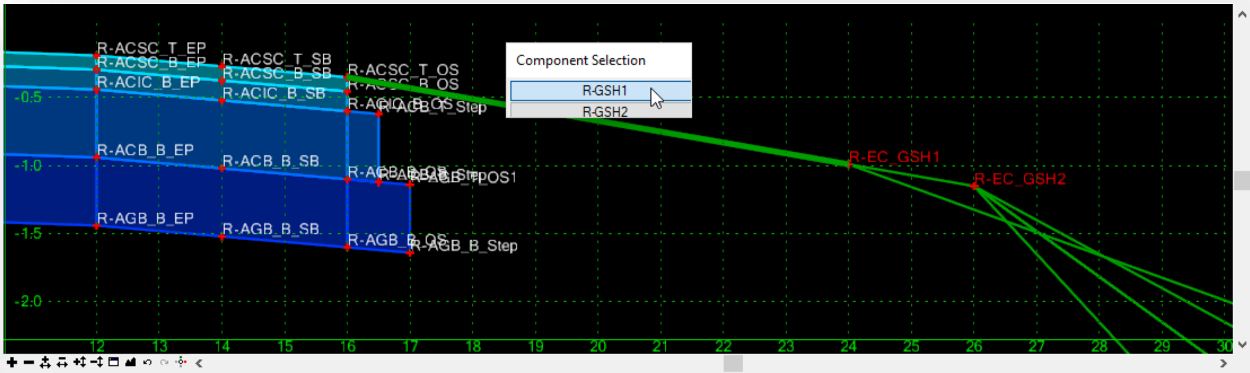
  Be sure to pay attention to the **Apply Affixes** setting (***Dynamic Settings*** dialog). Should this setting be toggled on or off as each of the components listed below is dragged into the template?

  ODOT > Backbones > Undivided Roadways > Pavement and Shoulder 301-8E Uncurbed > 2-Lane > Asphalt Concrete without Stabilization

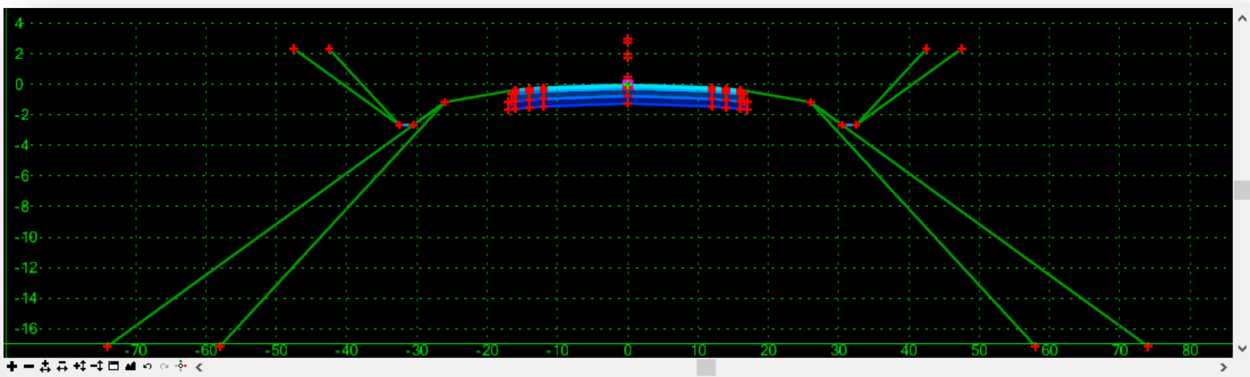
  ODOT > End Conditions > 307-4E Common and Barrier Grading > 307-4E 3:1 Cut, Fill Table

✓ After dragging in the end conditions, the next step is to remove some of the options. For this project, slopes of 3:1 or 2:1 will be used.

There are two different Graded Shoulder components in the end condition on each side of the template. The components R-GSH1 and L-GSH1 are used when the 6:1 slope is used. Remove these two components by right-clicking on the component and choosing the **Delete Component** option. Since these two graded shoulder solutions overlap, you are prompted to select the component you wish to delete as shown on the following page. Remove the R-GSH1 and L-GSH1 components. The 6:1 slope component is also removed since it has been defined as a "child" component of the graded shoulder 1 component.

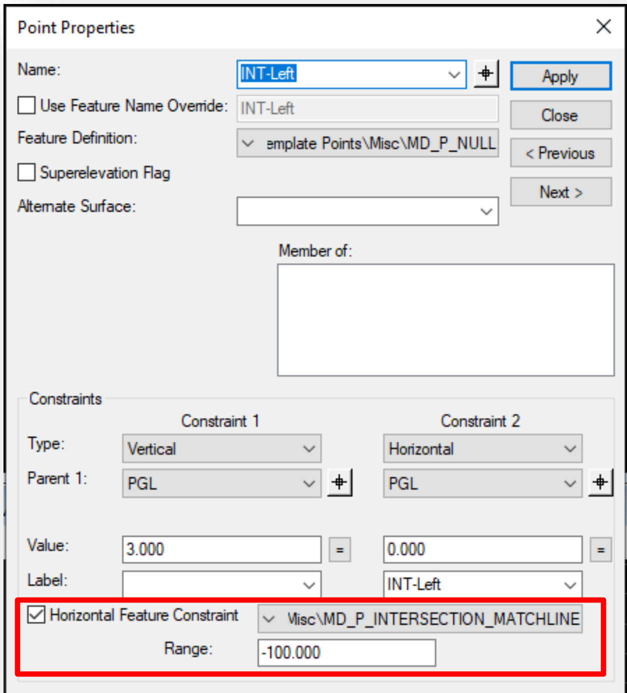


✓ Delete the 4:1 fill slope component on each side of the template. The completed template should appear as shown below.



Intersection and Driveway Display Rules

The backbone for the template contains four Null Points that are used to seek the intersection with another road (**INT-Left** and **INT-Right**) and points that are used to seek the intersection with a driveway (**DRIVE-Left** and **DRIVE-Right**). These points are defined to find elements of specific symbology defining the limits of an intersecting road or driveway. The properties for the point **INT-Left** are shown below.



The **Horizontal Feature Constraint** item defines the symbology for the element that the point will seek.

The **Range** defines the horizontal limit that the software will search for an element matching the defined symbology.

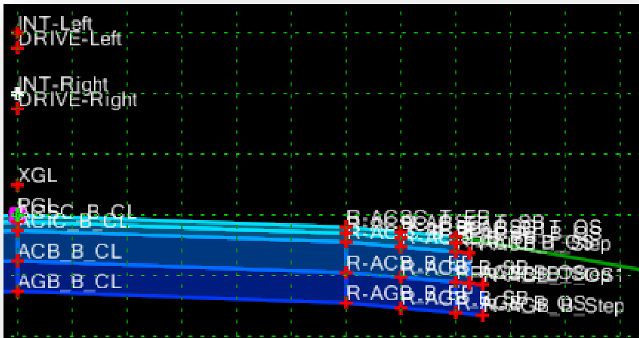
When the template is applied to a corridor, these points will seek for any elements matching the assigned symbology and will move to intersect the element. If the point moves, a **Display Rule** is applied to toggle off the display of selected components.

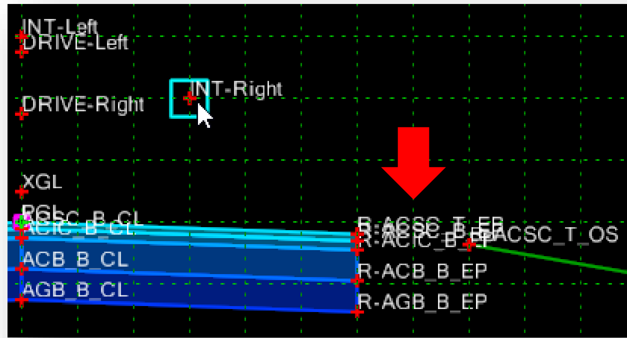
For example, if a driveway is found to the right, point **DRIVE-Right** moves to intersect the line that defines the limits of the drive. The display rule turns off the display of the shoulder components.

The behavior of the points can be tested in the template dialog. Take the following steps to test the behavior of the **DRIVE-Right** point:

✓ Right-click the point **DRIVE-Right**
  - Choose **Test Point Controls > Test Horizontal Point Control**
  - Move the cursor to the left and right of the template center point. Observe the behavior of the template components

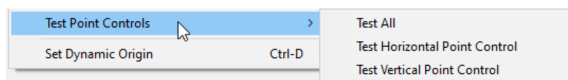
    As the point is moved to the right, the shoulder components on the right side of the template disappear. Notice that the end condition components are not defined to observe the display rules at this point, as shown below



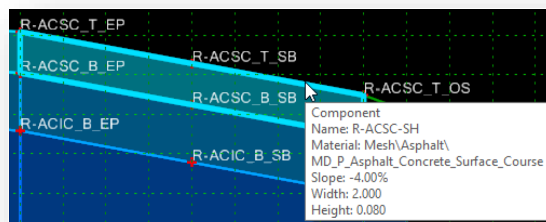


Take the following steps to assign the display rule to the end condition components:

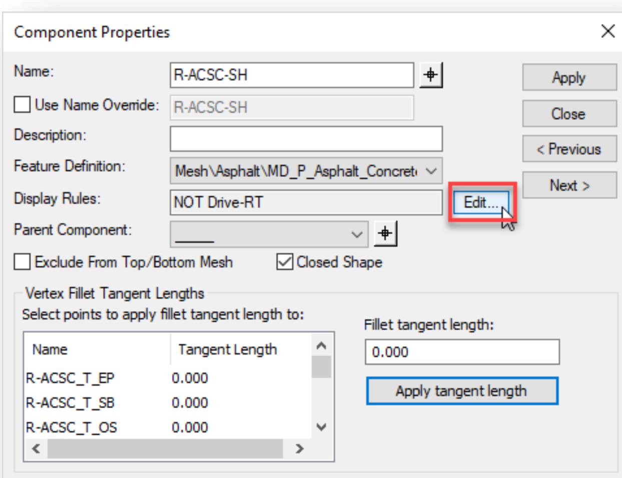
- ✓ Left click on the template window to terminate the testing process.

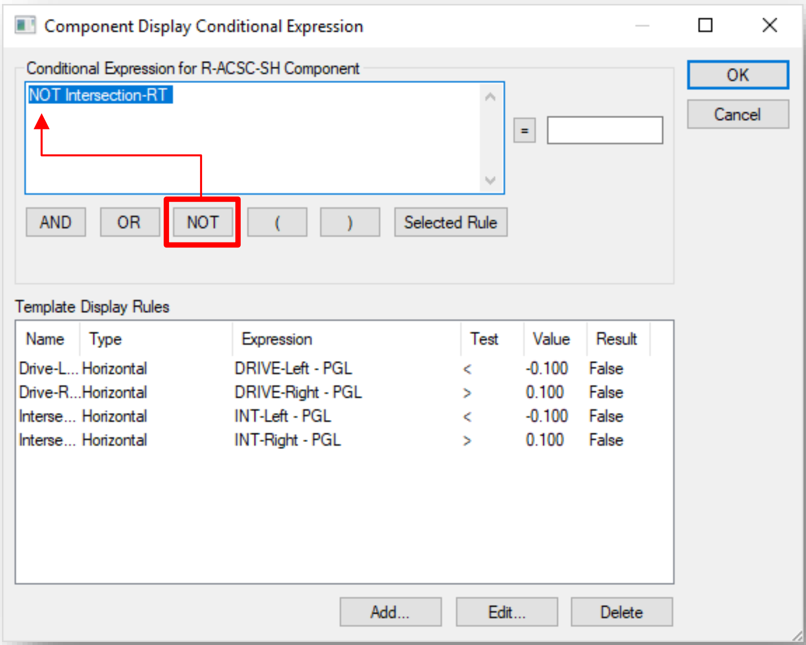


The **Display Rule** is applied to the top shoulder component. Take the following steps to review the display rule:



- ✓ Double-click the top component of the right shoulder, **R-ACSC-SH** (shown to the right)
- ✓ Select the **Edit** button to the right of the **Display Rule** parameter to open the ***Component Display Conditional Expression*** dialog. (shown below)





✓ Review the display rule settings. Select the **Cancel** button when complete.

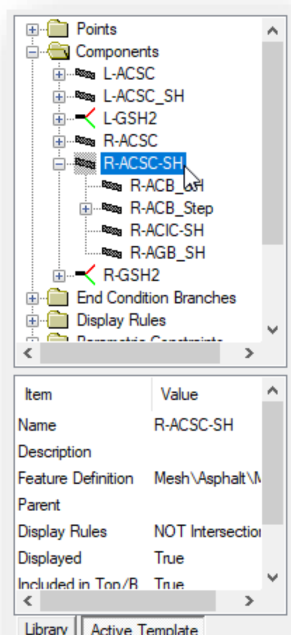
✓ From the ***Component Properties*** dialog, select **Close**.

Note that the display rule is only applied to the top component of the shoulder buildup in this template. Each of the sub-components of the shoulder are defined as “child” components with the top component as the parent component. Display rules assigned to the parent are inherited by the children.

This parent-child relationship can be reviewed or defined using the **Active Template** portion of the ***Create Template*** dialog as shown at right.

The parent component is at the top of the hierarchy for each branch. Components can be arranged in a hierarchal relationship by using drag-and-drop operations within the **Active Template** portion of the dialog.

If a display rule is set to turn off the display of the parent component, the display of each child component is also toggled off.



When an intersection or a driveway element is encountered, the display of the end conditions must also be turned off.

Take the following steps to apply displays rule to the parent component of the left and right end conditions:

- ✓ Double-click on the component **L-GSH2** to access the ***Component Properties*** dialog
  - Select the **Edit** button to the right of the **Display Rules**
  - Add the following display rule:
    NOT (Intersection-LT or Drive-LT)
  - Select **OK** to accept the display rule
  - Select **Apply** on the ***Component Properties*** dialog

- ✓ Test the point control by right-clicking on points **INT-Left** and **DRIVE-Left** to test the **Horizontal Point Control**.

  If the display rule is applied correctly, the shoulder and end condition will not display when point INT-Left is moved to the left, and the shoulder steps and the end condition will not display when point DRIVE-Left is moved to the left.

  Note: The **Library** option must be selected to test the display rule behavior. If the **Active Template** option is selected, the display rules cannot be tested. Be sure to select the **Library** option before testing the point and display rule behavior.

- ✓ Repeat the process to define the display rule for the right end condition by editing the **R-GSH2** component.

- ✓ When complete, choose **File > Save** from the ***Create Template*** dialog to save the changes to the template library.

Jameson Road Template

The Jameson Road template will use the same design as the S.R. 185 template.

- ✓ Right-click on the name of the S.R. 185 template
- ✓ Choose the **Copy** command
- ✓ Right-click on the **Project Templates** folder
- ✓ Choose the **Paste** command
- ✓ Rename the copied Folder and Template as "Jameson Road"

The pavement width for the proposed Jameson Road is 20'. Take the following steps to edit the template with the pavement width values:

- ✓ Double-click on the name of Jameson Road template to edit the template
- ✓ Edit points **R-ACSC_T_EP** and **L-ACSC_T_EP** to change the **Horizontal** constraint value to **10 & -10**

This completes this exercise.

- ✓ From the ***Create Template*** dialog, choose **File > Save**. Close the ***Create Template*** dialog.
- ✓ Exit OpenRoads designer, checking the design file back into ProjectWise

Additional Template Exercises

Simple Components

In this exercise we will create a simple template consisting of pavement and shoulders for a two-lane road.

- Open **ProjectWise Explorer**

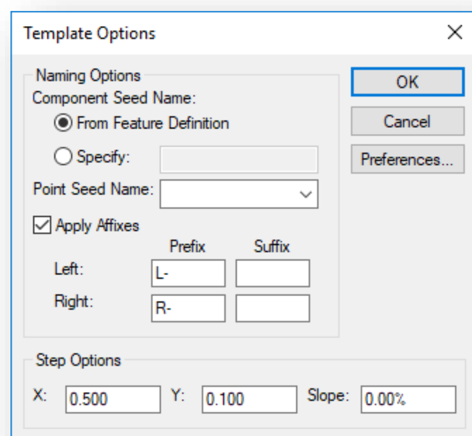
- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.
- Open the file **123456_BK001.dgn**. The design file that is used does not matter since we are editing the template library .itl file.

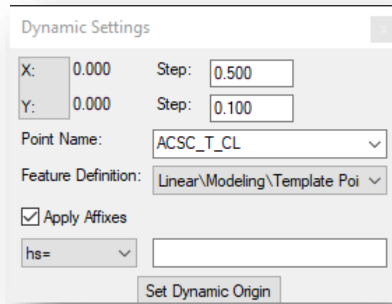
Take the following steps to create a new template for this exercise:

✓ From the **Corridors** tab, choose **Template > Create Template**. The ***Create Templates*** dialog is opened with the template library for the WorkSet loaded

✓ In the **Template Library** list, right-click on the **Project Templates** folder and choose the **New > Template** option. Name the new template **Exercise 1**.

- Select **Tools > Options** to open the *Template Options* dialog. Set the parameters for the dialog as shown on the right.





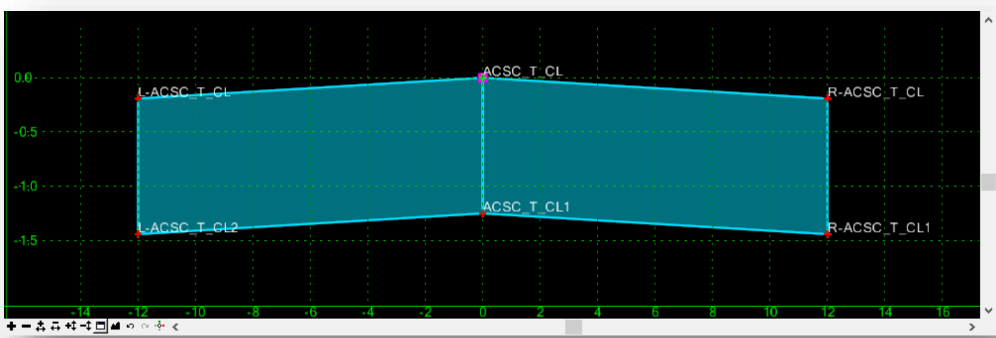
- Select **Tools > Dynamic Settings** to open the *Dynamic Settings* dialog. Set the parameters as shown at right.

- Right-Click in the template window and choose **Add New Component > Simple**. A simple component is floating on the cursor using the default settings for the **Slope**, **Thickness**, and **Width**.

- Set the parameters for the new component that will be created as follows:

  Name: ACSC
  Feature: MD_P_Asphalt_Concrete_Surface_Course
  Slope: -1.60%
  Thickness: 1.25"
  Width: 12

- Right-Click in the template window and choose the **Mirror** option (**Ctrl-M**). The simple component is now mirrored and appears on both the left and right sides. The component is mirrored about the **Dynamic Origin** location.
- Identify the template origin by issuing a data point at the (0,0) coordinate location in the template window. The component is constrained to the nearest point in the grid by the **Step** values defined in the *Dynamic Settings* dialog. The mirrored components are shown below.



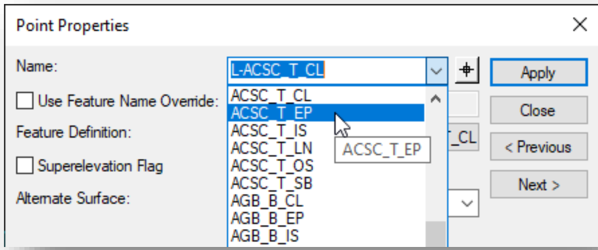
Tip: You can use the **XY=** option in the ***Dynamic Settings*** dialog to enter the (0,0) coordinate.

Take note of the point names for the components. Each point is named according to the **Point Name** parameter in the ***Dynamic Settings*** dialog. Points to the left and right of the origin are prefixed according to the affixes defined in the ***Template Options*** dialog shown on the previous page. The **Point Style** (feature definition) is the same for each point. It is necessary to review and edit the points to assign the appropriate point name and style.

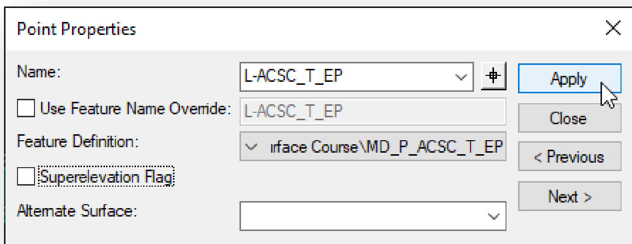
The ***Point Properties*** dialog is used to edit points.

✓ Double-click on point **L-ACSC_T_CL** to review and edit the properties of the point. Take the following steps in the **Point Properties** dialog:

- Change the point name to **ACSC_T_EP** by selecting the name from the list.



  - When the point name is selected from the list, the corresponding **Feature Definition** is automatically assigned for the point. Verify that the **Feature Definition** is now set to **MD_P_ACSC_T_EP**
  - The point name still needs to have a prefix assigned. Add the prefix **L-** to the point name. The point name and feature definition should be set as shown below.



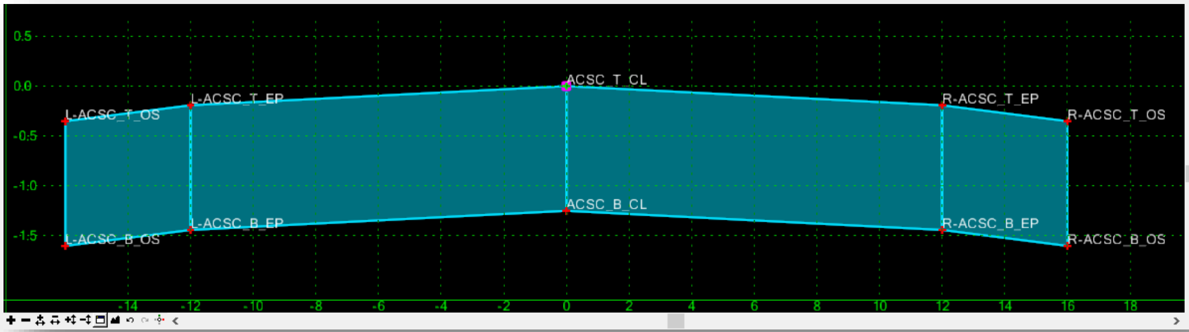
  - Select the **Apply** button to save the changes.

✓ Review the remaining points to set the appropriate point **Name** and **Feature Definition** for the top and bottom of the pavement layer. Remember, the feature definition that is assigned to each point will control the level symbology for the 3D line strings that are drawn in the corridor model.

✓ Add a simple component for the shoulder surface to the template using the **Mirror** option with the following settings:

Name:	ACSC-OS
Feature:	MD_P_Asphalt_Concrete_Surface_Course
Slope:	-4.00%
Thickness:	1.25”
Width:	4

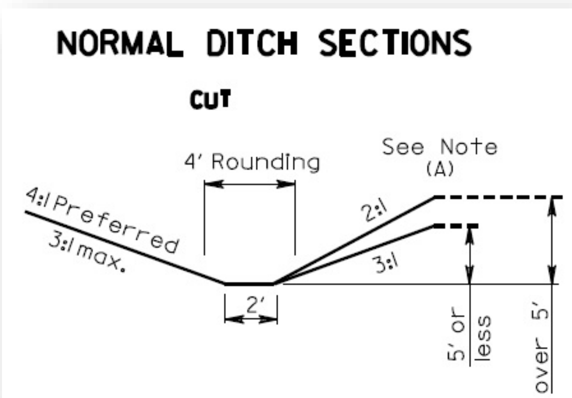
✓ Edit the point names for the shoulders by double-clicking on each point to open the *Point Properties* dialog. Be sure to pick the point names from the drop-down menu list so that the corresponding feature definitions are assigned. When completed, the points should be named as shown below.



- ✓ When complete, save your template by choosing **File > Save** from the ***Create Template*** dialog pull-down menu.

Creating and Testing End Conditions

In this exercise, you will create a simple cut end condition with a 3:1 foreslope and a 2' ditch. The backslope will contain two end conditions, a 3:1 slope and a 2:1 slope as shown below.



- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.
- Open the file **123456_BK001.dgn**. The design file that is used does not matter since we are editing the template library .itl file.

Take the following steps to create a new template for this exercise:

- ✓ From the **Corridors** tab, choose **Template > Create Template**. The ***Create Templates*** dialog is opened with the template library for the WorkSet loaded
- ✓ Right-click on the Project Templates folder and choose **New > Template**. Create a new template named **Normal Ditch**.
- ✓ Open the ***Dynamic Settings*** dialog and select the **Point Name** to "EC_HINGE" from the list. The **Point Style** is automatically set. Set the **Step** values as desired. Uncheck **Apply Affixes**.
- ✓ Add an end condition for the graded shoulder
    - Right-click in the template window and choose **Add New Component > End Condition**.
    - Start the End Condition at coordinate (XY=0, 0).
    - Name the component "GSH1"
    - Use the Feature "MD_P_Graded_Shoulder" for the component
    - Be sure to check off **Check for Interception** in the *Dynamic Settings* dialog after placing the first point.
    - Change the Point Name in the *Dynamic Settings* dialog to "EC_GSH1" After placing the first point but before placing the second point. By choosing the name from the list, the **Feature Definition** is automatically assigned.
    - Place the second point 6' horizontally from the first point at a slope of -8% (hs=6, -8%)
    - Right click and select Finish.
- ✓ Add an End Condition for the ditch foreslope.
    - In the *Dynamic Settings* dialog, select the **Point Name** EC_FSLP.
    - Right-click in the template window and choose **Add New Component > End Condition**.
    - Name the component "3:1 Foreslope"
    - Use the **Feature** "MD_P_Foreslope" for the component
    - Create the component with a vertical constraint of -1.5 at a 3:1 slope (vs=-1.5, -1:3) from the point EC_GSH1. Be sure to check off **Check for Interception** in the *Dynamic Settings* dialog after placing the first point.
    - Right click and select Finish.
    - Double click on the 3:1 Foreslope End Condition and set the Parent component to GSH1. Hit **Apply**.
- ✓ Add an End Condition for the 2' ditch bottom.
    - In the *Dynamic Settings* dialog, select the **Point Name** EC_DITCH.
    - Right-click in the template window and choose **Add New Component > End Condition**.
    - Name the component "DITCH"
    - Use the **Feature** "MD_P_Ditch" for the component
    - Create the component with a horizontal constraint of 2.0 and a vertical constraint of 0 (dl=2,0) from the point EC_FSLP. Be sure to check off **Check for Interception** in the *Dynamic Settings* dialog after placing the first point.
    - Right click and select Finish.
    - Double click on the DITCH End Condition and set the Parent component to 3:1 Foreslope. Hit **Apply**.

Next, define two end conditions for the backslope.

- ✓ In the *Dynamic Settings* dialog, select the **Point Name** EC_BKSLP
- ✓ Right-click in the template window and choose the **Add New Component > End Condition** option.
- ✓ Name the end condition "3:1 Backslope"
- ✓ Set the Feature for the end condition to "MD_P_Backslope"
- ✓ Create the component with a vertical constraint of 5' at a 3:1 slope (vs=5,1:3) from the point EC_DITCH. Be sure to check on **Check for Interception** in the *Dynamic Settings* dialog after placing the first point.
- ✓ Right click and select Finish.
- ✓ Double click on the 3:1 Backslope End Condition and set the Parent component to DITCH. Hit **Apply**.

Add a second End Condition for the 2:1 backslope using the settings detailed below.

- ✓ In the *Dynamic Settings* dialog, select the **Point Name** EC_BKSLP
- ✓ Right-click in the template window and choose **Add New Component > End Condition**.
- ✓ Name the end condition "2:1 Backslope"
- ✓ Set the Feature for the end condition to "MD_P_Backslope"
- ✓ Set the component **Priority** to **2**
- ✓ In the *Dynamic Settings* dialog, toggle on the **Check for Interception** and **End Condition is Infinite** options (options not available until the first point of the end condition is placed).
- ✓ Create the component with a vertical constraint of 5' at a 2:1 slope (vs=5,1:2) from the point EC_DITCH
- ✓ Right click and select Finish.
- ✓ Double click on the 2:1 Backslope End Condition and set the Parent component to DITCH. Hit **Apply**.

Next, review the point names and feature styles, as well as the Override names for the points and components.

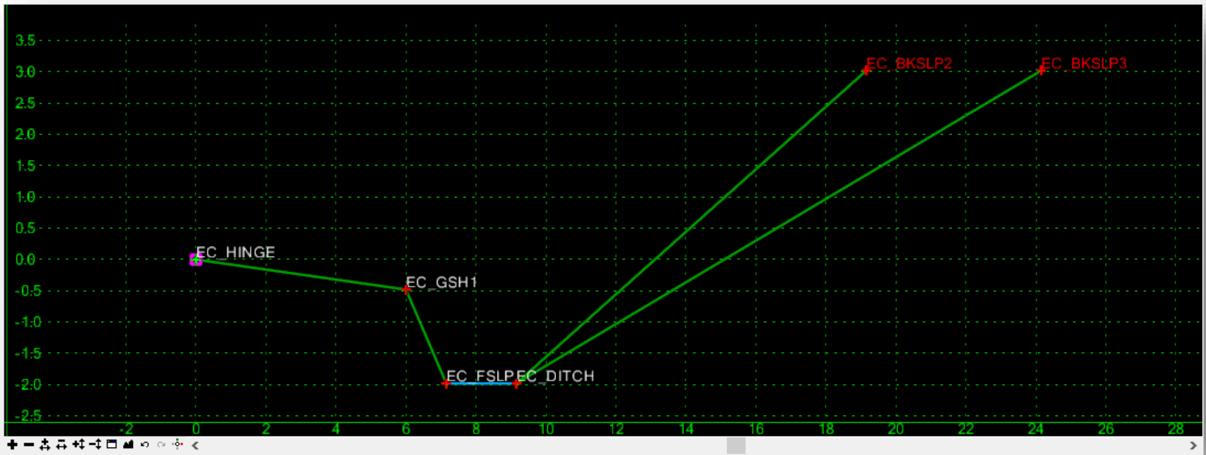
- ✓ Double-Click on the point **EC_BKSLP** to make the following change:
    - Toggle on the **Use Feature Name Override** option and enter an override name of "**BKSLP**" and change the name to "**EC_BKSLP3**" then hit **Apply**.
- ✓ Double-Click on the point **EC_BKSLP1** to make the following changes:
    - Toggle on the **Use Feature Name Override** option and enter an override name of "**BKSLP**" and change the name to "**EC_BKSLP2**" then hit **Apply**.

Review the component properties for the backslope components to set the override name.

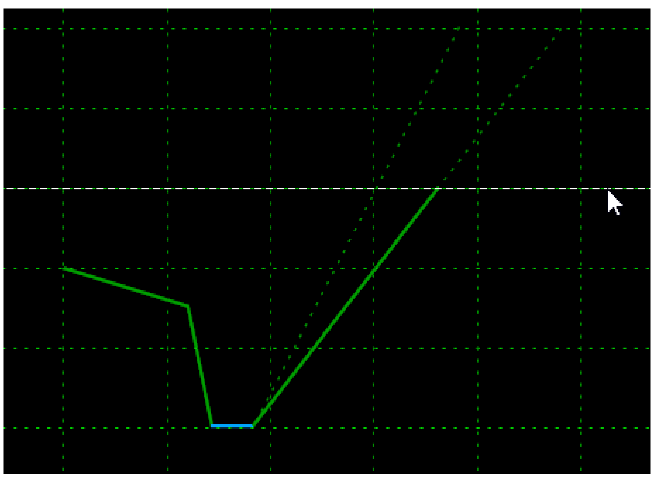
- ✓ Double-click the **2:1 Backslope** component. Toggle on the **Use Name Override** option and set the override name to "**Backslope**"

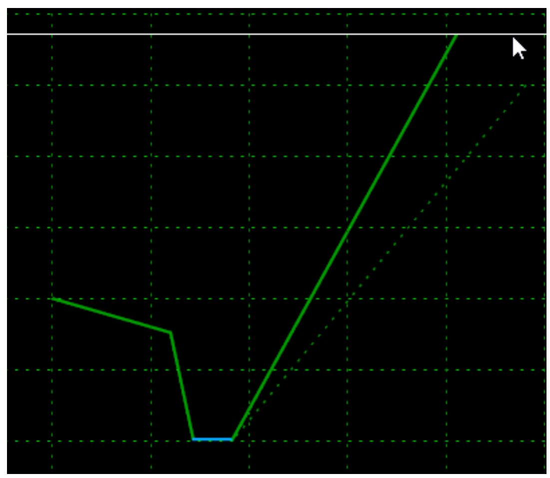
- ✓ Double-click the **3:1 Backslope** component. Toggle on the **Use Name Override** option and set the override name to "**Backslope**"

The completed template should appear as shown below.



- ✓ Use the **Test** option to ensure that the backslope components behave as intended. The 3:1 backslope should solve first for backfills up to 5' high. The 2:1 solution should solve for backslopes greater than 5'.





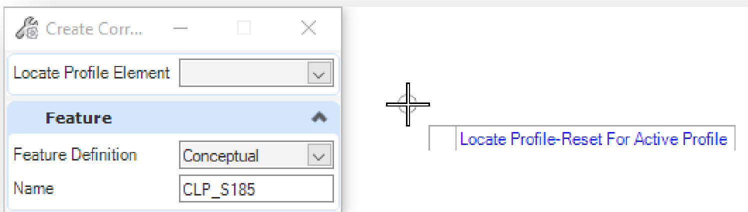
- ✓ When complete, choose **File > Save** from the ***Create Template*** dialog to save the changes to the template library.

Section 504 Corridor Modeling

Corridor Creation

A corridor model is created by applying a template to the selected horizontal and vertical geometry.

Corridor models are created by choosing the **New Corridor** command from the **Corridors** tab, or by selecting an alignment and choosing the Create Corridor command from the pop-up menu. When the command is selected, the software will step you through a series of prompts. These prompts appear on the cursor and can also be defined using the **Tool Settings** dialog.



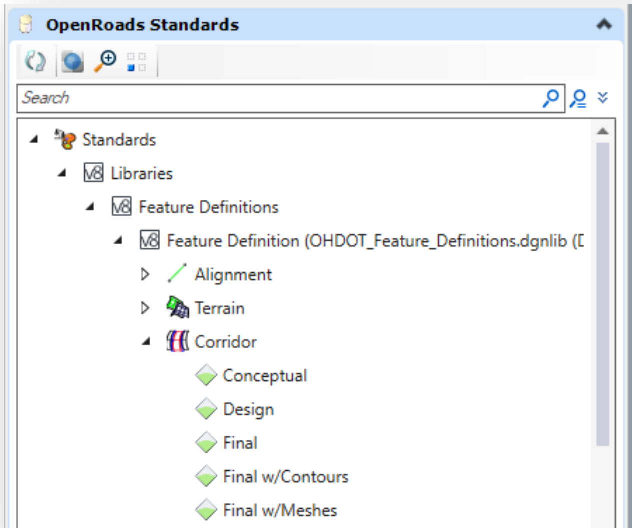
You are prompted to define the following parameters:

Locate Profile – Reset For Active Profile
Select the profile name from the drop-down list or issue a reset button to accept the **Active Profile** that is defined for the Horizontal Alignment.

Corridor Name
Define the name for the corridor. It is recommended that you use the same name as the horizontal alignment.

Feature Definition
The assigned Feature Definition defines a variety of parameters to control the template drop interval as well as which elements are drawn in the 3D model. (3D line strings, 3D components, surface meshes, etc.)

The feature definitions can be reviewed in the **Civil Standards** tab of the *Project Explorer* dialog as shown below.



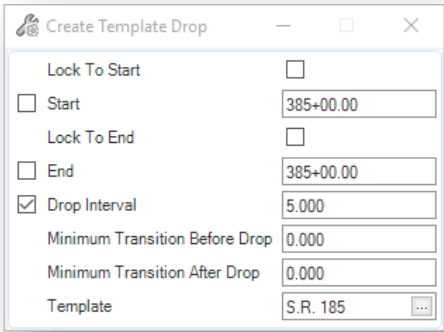
The feature definition assigned to a corridor can be changed at any time throughout the corridor modeling process.

Each Corridor feature definition is defined to provide more detailed modeling as you work from conceptual design to the final design. The settings for each Corridor feature definition are defined in the OHDOT standards and vary for each design stage.

Template Drop Interval Multiplier
When templates are applied to the horizontal and vertical geometry, the user is prompted to enter a template drop interval. ODOT recommends an interval of 5. The **Template Drop Interval Multiplier** parameter is used to specify a multiplier, which is applied to the initial template drop interval, to determine the actual interval of each template drop location according to the selected design stage. This is useful to speed up processing for early design stages where less detailed information is required. The ODOT design stages have been defined with the multiplier values shown in the table below.

Design Stage	Template Drop Interval Multiplier
Conceptual	5
Design	2
Final	1
Final w/Contours	1
Final w/Meshes	1

After defining the **Profile**, **Corridor Name**, and **Feature Definition**, you are prompted to define the **Template Drop** information as shown below.



The parameters can be defined in the ***Crete Template Drop*** window, as shown at left, or by dialogs floating on the cursor that will step you through the parameters.

Each parameter is defined below:

Lock to Start
Toggle this option on to lock the start of the corridor to the start of the horizontal alignment.

Start
Define the **Start** station for the corridor.

Lock to End
Toggle this option on to lock the end of the corridor to the end of the horizontal alignment.

End
Define the **end** station for the Corridor.

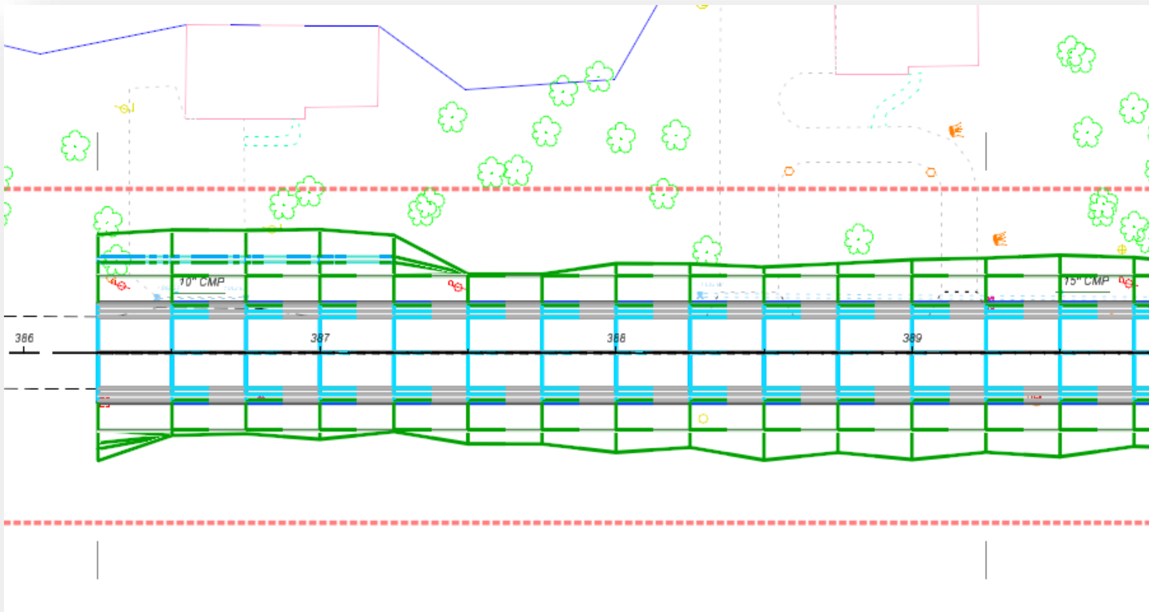
Drop Interval
This parameter is used to define the interval that the template will be applied to the corridor. We recommend a **Drop Interval** of **5** for ODOT projects.

Minimum Transition Before Drop / Minimum Transition After Drop
If they are non-zero, then a transition drop is created at the beginning/end of the template drop with a length greater than or equal to the value entered. The actual length is determined by how far it is between the new drop and the drop before/after the new drop. If there isn't enough space to meet the minimum, then the previous/next drop is shortened to accommodate the transition. If there is no previous/next drop, then no transition drop is created.

Template
Define the template from the library to be applied to the alignment and profile. The ... button to the right of the template name is used to browse the template library to choose the desired template.

After defining the ***Create Template Drop*** parameters, the template is applied to the selected horizontal and vertical alignments to create the corridor.

- A 3-Dimensional model named **Design-3D** is automatically created in the active design file and referenced to the active model.



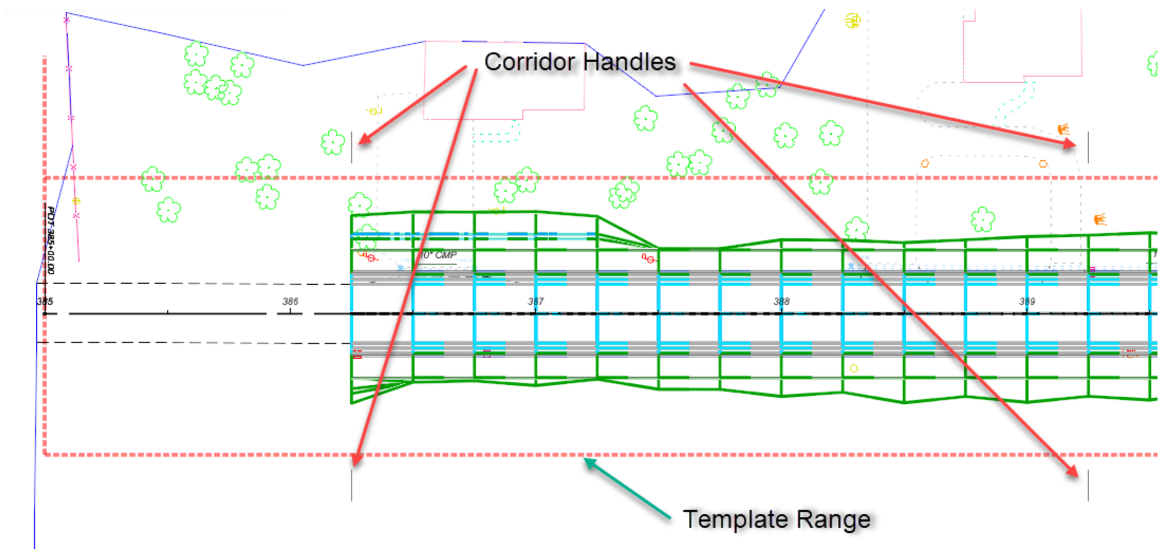
Note: The referenced graphics are not always desirable and can be turned off using the MicroStation Reference Attachment tools.

The 3d model of the corridor can be displayed by holding down the right-mouse button in the MicroStation view until the pop-up menu appears, and then choosing **View Control > 2 Views Plan/3D**.

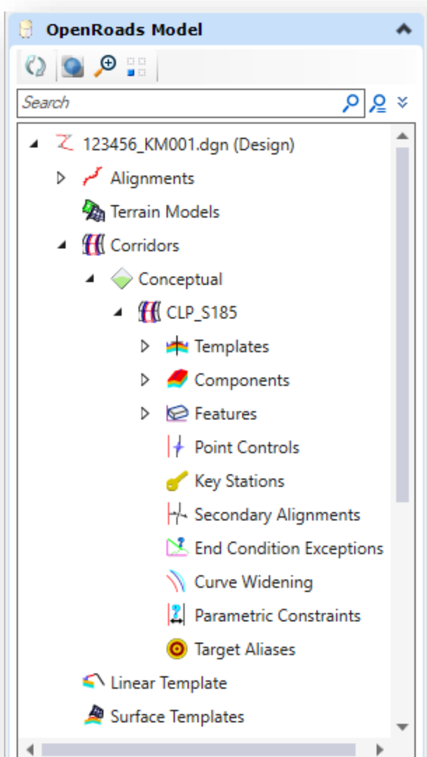
In addition to the 3d graphics, 2d graphics are drawn in the active model as described below:

- 2-Dimensional graphics are drawn in the active model as defined by the features assigned to the individual points in the template. The ODOT feature definitions that are assigned to points on the surface of the template, such as the edge of pavement, shoulder, or ditches, are the only template features that draw these 2D plan graphics.

- Graphics representing the length of the corridor model and the template drop range are created in the active model. This graphic includes several "handles" at intervals along the length of the corridor that can be easily selected to identify the corridor model as shown below.

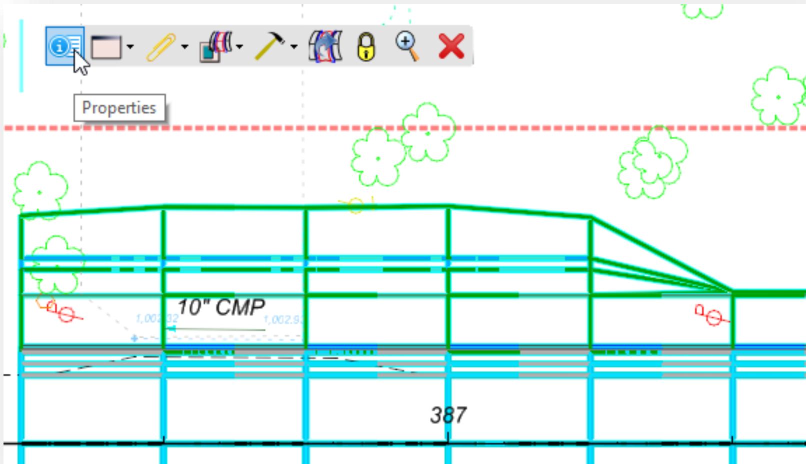


- The corridor is added to the **OpenRoads Model**, which can be reviewed in the ***Project Explorer*** dialog.



Corridor Editing Tools

Select one of the corridor handles, as shown below, to access a menu of common corridor commands.



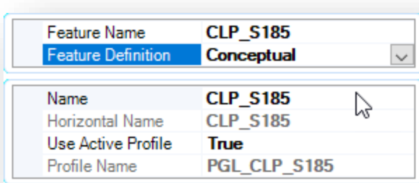
The commands in the menu are grouped as follows:

- Properties
- Corridor Views
- Reports
- Corridor References
- Corridor Creation Tools
- Lock – Deactivate Rule
- Zoom To
- Delete

Commonly used commands are detailed in this training guide. For additional information, see the online help.

Corridor Properties

Select **Properties** from the corridor pop-up menu to access the dialog shown below.

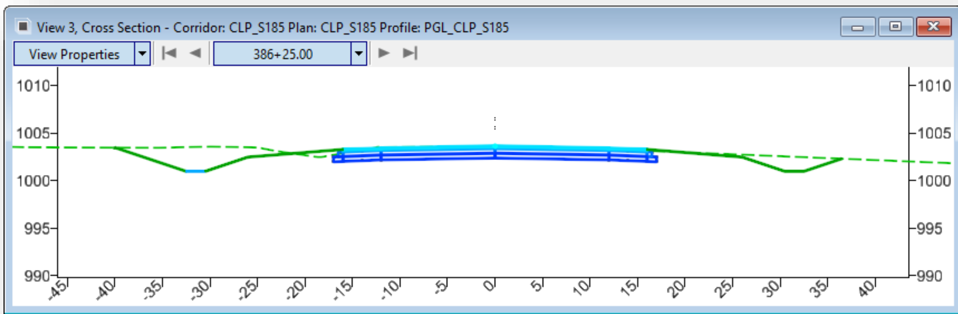


The corridor parameters shown above can also be edited in the ***Properties*** dialog when a corridor is selected.

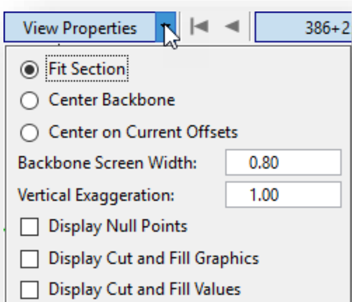
Corridor Views

The **Corridor Views** icon has one command, **Open Cross Section Model**. This command is used to create a dynamic cross section view to review the model by scrolling through cross sections.

When selected, you are prompted to **Open or Select View** for the cross-section display. In the example below, View 2 was selected to display the dynamic cross sections.

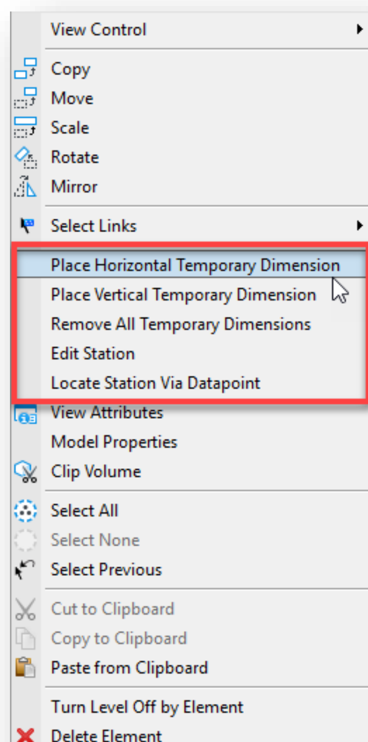


This dynamic view is a temporary display of the cross sections. The sections are not written to the design file using this command.



The **View Properties** contains parameters for adjusting the cross-section display, as shown at left.

Hold down the right mouse button in the cross-section view to access the pop-up menu shown at right. Five additional commands relevant to cross sections are available. See the online help for additional information.

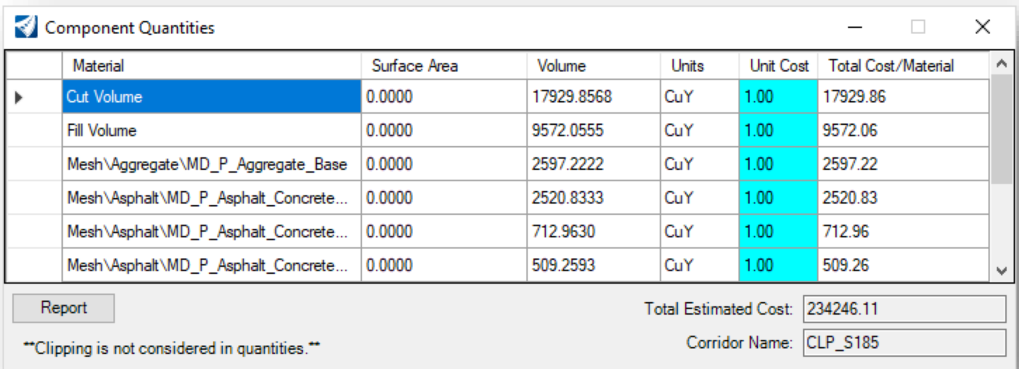


Reports

The **Reports** icon provides access to the following four reports:

- Corridor Component Quantities
- Design Input Report
- Results Report
- Milling Report

The **Corridor Component Quantities** report is used to generate a quick cost estimate for the selected corridor by assigning a unit cost for each component in the corridor templates.


Component Quantities

Material	Surface Area	Volume	Units	Unit Cost	Total Cost/Material
Cut Volume	0.0000	17929.8568	CuY	1.00	17929.86
Fill Volume	0.0000	9572.0555	CuY	1.00	9572.06
Mesh\Aggregate\MD_P_Aggregate_Base	0.0000	2597.2222	CuY	1.00	2597.22
Mesh\Asphalt\MD_P_Asphalt_Concrete...	0.0000	2520.8333	CuY	1.00	2520.83
Mesh\Asphalt\MD_P_Asphalt_Concrete...	0.0000	712.9630	CuY	1.00	712.96
Mesh\Asphalt\MD_P_Asphalt_Concrete...	0.0000	509.2593	CuY	1.00	509.26

Report

Total Estimated Cost: 234246.11

Corridor Name: CLP_S185

Clipping is not considered in quantities.

The remaining reports are offered within the software's online help documentation.

Corridor References

The following four **Corridor References** commands are available:

Add Corridor Reference
This tool is used to add graphical elements to the corridor processing. This must be done when a **Feature** is targeted in the template definition. This enables the software to process only the identified elements, which speeds up processing. For example, if the template targets a right-of-way line, the right-of-way lines must be included as a Corridor Reference for the template to find the lines.

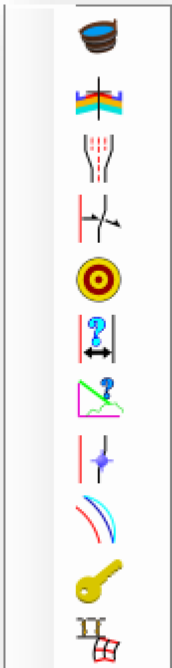
Remove Corridor Reference
This tool is used to remove graphical elements from the corridor processing.

Add Clipping Reference
This command is used to remove areas of overlap when working with multiple corridors. For example, in a corridor intersected by a crossing roadway, clipping is used to remove overlapping features within the intersection.

Remove Clipping Reference
This tool is used to remove any clipping references defied for a corridor.

Corridor Creation Tools

The **Corridor Creation Tools** menu contains commonly used corridor commands. The tools are listed below. These tools are detailed on the following pages.



Corridor Objects

Create Template Drop

Create Transition

Create Secondary Alignment

Define Target Aliasing

Create Parametric Constraint

Create End Condition Exception

Create Point Control

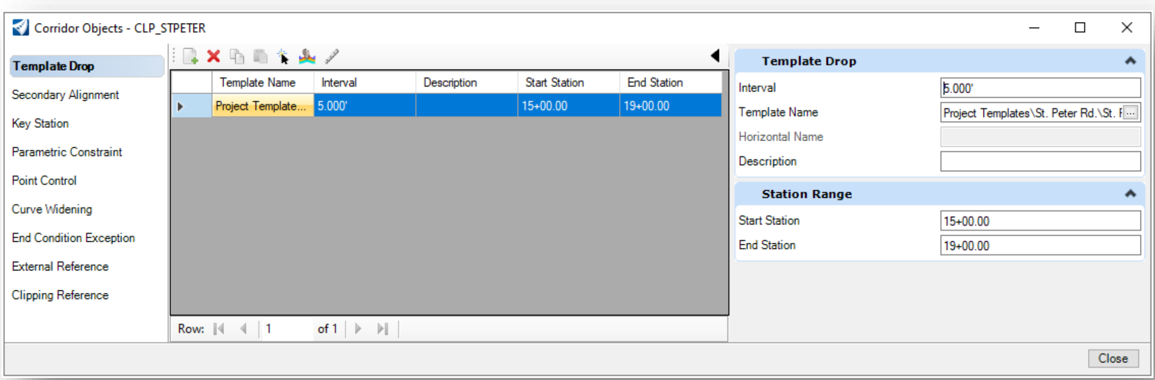
Create Curve Widening

Key Station

Assign Superelevation to Corridor

Corridor Objects

This option is a one-stop shop to view, create, and edit corridor objects. Most of the options contained in the **Corridor Creation Tools** can be accessed from this dialog as shown below.



Create Template Drop

This tool is used to create a new template drop location on the corridor.

Create Transition

A transition is created between templates of different names, as templates generally don't instantaneously change from one template to another. This tool creates the transition by selecting the two template drops which are adjacent to it. Once the transition location is created, it is up to the user to define how the transition is applied.

See the online help for more information.

Create Secondary Alignment

Secondary alignments are used to modify the direction of cross section processing. By default, at any given station, the cross section is created orthogonal to the main alignment. If a secondary alignment exists, then that portion of the cross section which lies outside the secondary alignment will be orthogonal to the secondary alignment instead of the main alignment.

See the online help for more information.

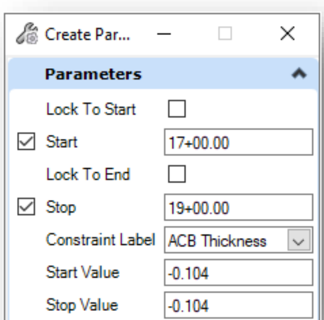
Define Target Aliasing

Target aliasing allows you to target other corridor surfaces or features or to set up a prioritized target list for end condition solutions on surfaces, features, and alignments.

See the online help for more information.

Create Parametric Constraint

Parametric constraints can be used to change one or more labeled constraint values of a template while the template is being processed in the corridor modeler.



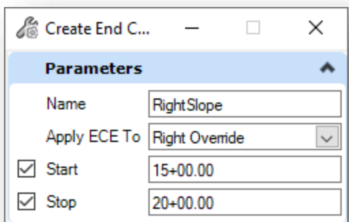
In the example at left, the **Constraint Label** “ACSC Thickness” is assigned in the template for the **Vertical** constraint of the Asphalt Concrete Surface Course point. The **Parametric Constraint** can be used to define a new value for a specified station range.

Create End Condition Exception

End Condition Exceptions are used to modify the behavior of an end condition solution without requiring the use of additional template drops. When an end condition exception is added, it must be edited to change its behavior. End condition exceptions come in two classes:

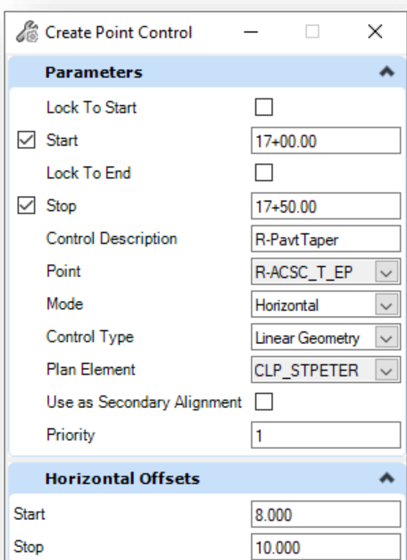
- **Overrides** allow you to replace or override the template drop end conditions on the left or right of the backbone. When you choose this option, you must edit the override to set up the new end condition. When the override exception is edited, the *Create Template* dialog is displayed allowing you to edit the end-condition.

- End condition **Transitions** are used where the end condition may change suddenly due to changes in the existing surface or other reasons, and you want the transition to be smooth over a specified station range rather than a sudden change over a short length.



Create Point Control

Point controls are used to override the normal locations of one or more points and or components in a cross section. Examples of this include lane widening, staying within the right-of-way, or maintaining a slope for a ditch.



Create Curve Widening

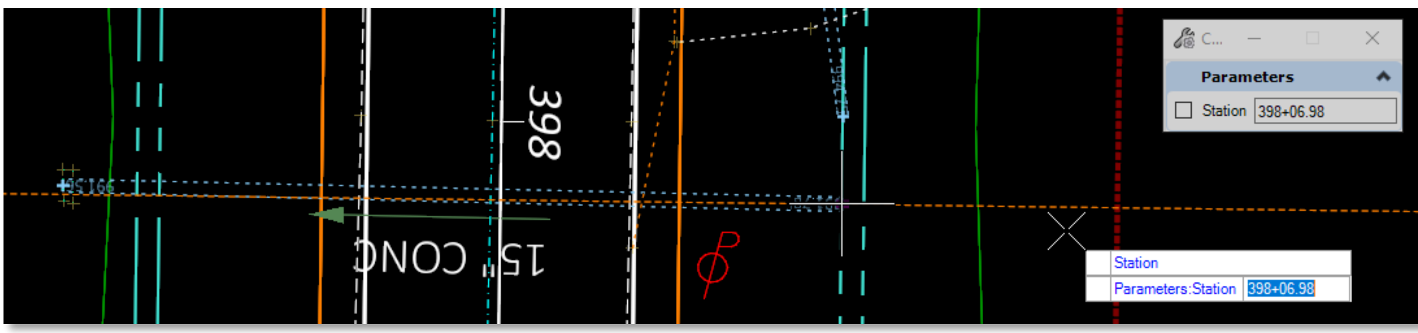
Curve widening is used to automatically create and apply horizontal controls to widen lanes and/or edge of pavement lines around curves, moving them further away from the centerline at each curve of the controlling alignment. The tool is used in conjunction with an ASCII file (*.wid) which contains parameters to define the widening.

Curve widening is defined in the **Location and Design Manual, Volume 1 Roadway Design**, **Section 301.1.3 Travelled Way Widening on Highway Curves**.

ODOT does not provide curve widening tables currently. Use point controls to manually assign the station ranges and offsets for curve widening.

Key Station

This command is used to add stations that are not coincident to the template interval to the corridor processing. For example, a key station can be added at a drive or culvert location to ensure the template is processed at that station.



Assign Superelevation to Corridor

Superelevation is covered in another section of this training.

Processing Order for Point Overrides

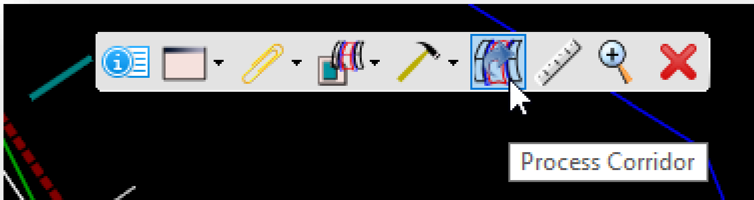
As you can see, there are several ways in the corridor processing to override the template definition for various constraints using Parametric Constraints, Point Controls, and Horizontal Feature Constraints.

This is generally the order in which OpenRoads solves the location of points and components at each template drop:

1. Template is dropped, and points are placed according to the point constraints stored in the template.
2. Parametric constraints are applied as defined in the template, and in the corridor.
3. Horizontal Feature constraints are applied to move points if the feature is found in the specified range.
4. Point controls are applied to the assigned points, overriding the corresponding constraint, and all points that are constrained back to the point-controlled point will be recalculated.
5. Component display rules are solved based on the current position of all points.
6. End conditions are solved by extending designated segments along the specified slope to seek their targets.

Process Corridor

Select the corridor line to highlight corridor then pop out menu with appear. Then click the icon to reprocess the corridor.



Lock - Deactivate Rule

This command can be used to temporarily deactivate processing rules on the corridor. This is useful when making edits so that the corridor does not automatically update as the edits are made. Once the edits have been completed, the rules can be turned back on, and the corridor reprocessed.

Zoom To

Select the command to zoom to the full extents of the corridor.

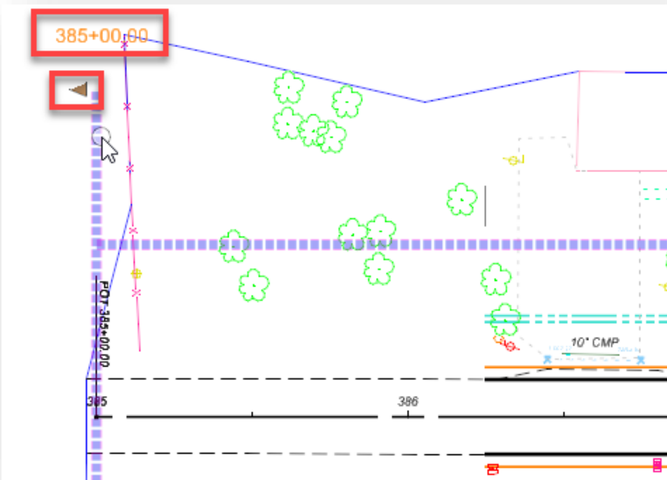
Delete

Select this command to delete the corridor.

Template Drop Tools

Edit the Template Range

When the template range graphic is selected, the range can be edited by accessing rule that defines the template drop location.



The template range can be edited by selecting the station and editing the value of by selecting the arrow icon to dynamically mode the start (or end) of the template range.

Template Pop-Up Menu

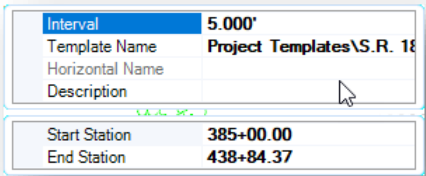
Select the template range graphic and let the cursor rest on the element to access the pop-up menu shown below.

The following commands are available:

- Properties
- Edit Template Drop
- Copy Template Drop
- Synchronize with Library
- Delete

Properties

Choosing **Properties** from the pop-up menu allows editing the parameters shown at left.



Edit Template Drop

When a template is applied to generate the corridor, the template definition is copied into the design file. The template definition can be edited in the design file, independent of the definition that is stored in the template library .itl file. Use this command to edit the template definition as applied to the corridor, not the template as defined in the library.

When the command is selected, the ***Editing Roadway Designer Template Drop*** dialog is opened. This tool is useful when you wish to make local changes to the corridor without editing the template library definition.

ODOT recommends making all changes to the template library and then using the **Synchronize with Library** command to apply the changes made in the library to the corridor model.

Copy Template Drop

This tool is used to copy a template drop to a new station range along the alignment by defining the beginning and ending station for the new template drop range.

Synchronize with Library

Select this tool to synchronize the local copy of the template that has been applied to the corridor with any changes that have been made to the template in the library .itl file.

Note: Synchronizing the template with the library will override and changes that may have been made to the corridor's template definition by using the **Edit Template Drop** command.

For consistency, ODOT recommends making changes to the template library and using the Synchronize with Library command to apply template changes to the corridor.

Section 505 Corridor Modeling Exercises

Exercise 1: S.R. 185 Corridor

This exercise illustrates the process of creating the initial corridor model. Additional exercises will be added in the future to illustrate different corridor editing tools.

The purpose of this exercise is to demonstrate various modeling tools, not to generate a model that accurately represents the best design of this type of project.

In this exercise we will create a corridor for State Route 185.

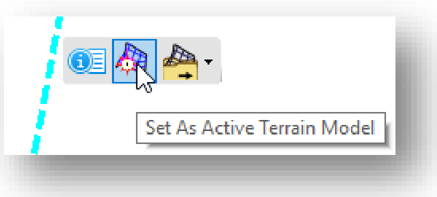
- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.
- ✓ Open the file **123456_KM001.dgn**.
- ✓ Attach the following design files as references (**Home > Attach Tools > References**):

  Geometry Basemap for S.R. 185
  401-Engineering\Roadway\Basemaps\123456_BK001.dgn
  - Set the **Nest Depth** parameter to a value of **0** when attaching the geometry basemap.

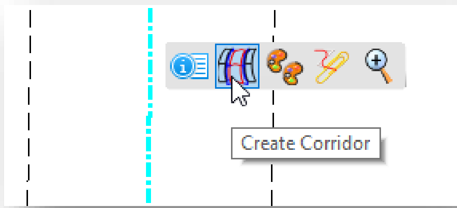
  Survey Basemap
  300-Survey\Basemaps\123456_FB001.dgn
  - Set the **Nest Depth** parameter to a value of **0** when attaching the survey basemap.

- ✓ Select the terrain model boundary and choose the **Set As Active Terrain Model** option

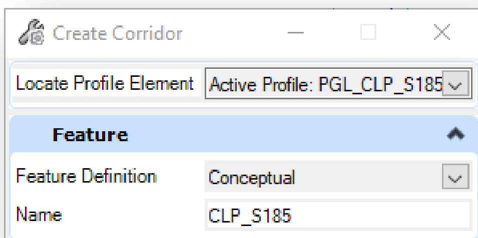


Take the following steps to create the corridor for S.R. 185:

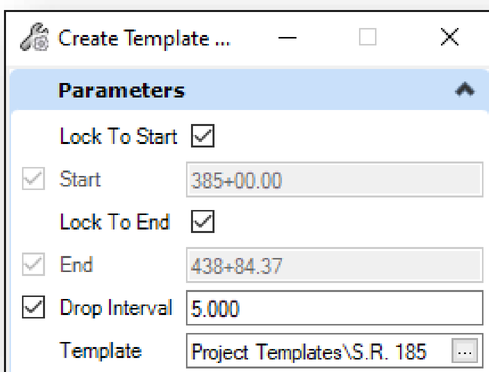
✓ Select the State Route 185 alignment and choose the **Create Corridor** command from the pop-up menu.



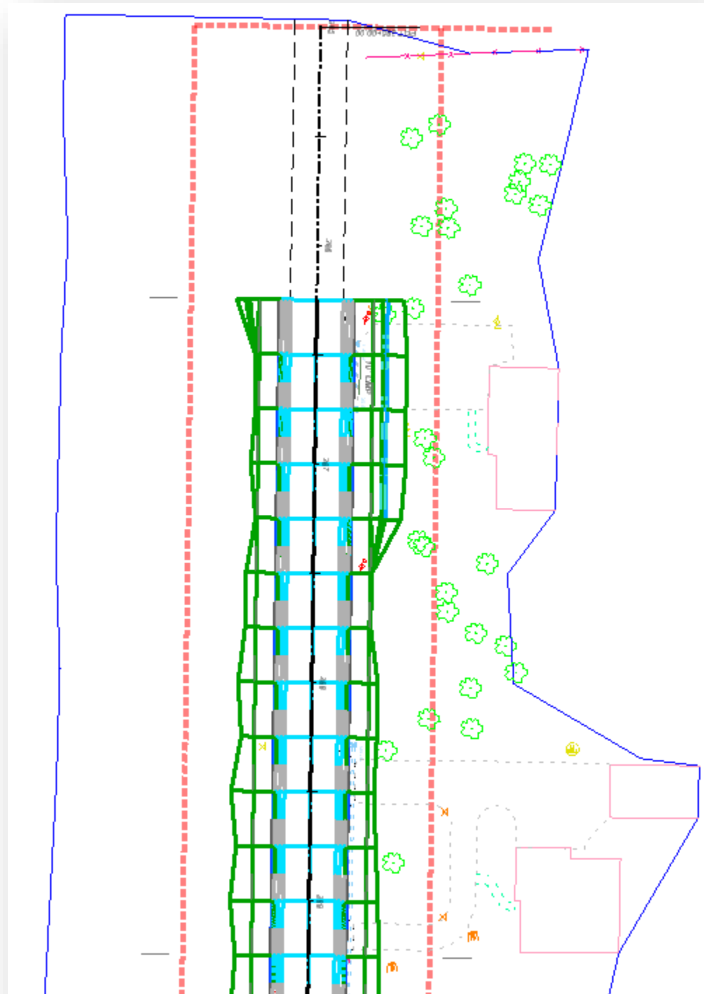
- Create a corridor named CLP_S185 using the **Active Profile**, and the **Conceptual Feature Definition**.



- Use the parameters below and the S.R. 185 template created in the previous exercise.



A portion of the corridor model is shown on the right.



The Design-3D model is attached as a reference. Most users elect to turn off the display of the referenced corridor graphics. Take the following steps to turn off the reference to display only the 2D information.

- ✓ From the **Home** tab, select **Attach Tools > References**
- ✓ Turn off the display of the Design-3D model.

The 3D model can be displayed in a separate view window. Take the following steps to open a new view to display the 3D model:

- ✓ Hold down the right mouse button in the empty space in the view window. Choose **View Control > 2 Views Plan/3D** from the pop-up menu.

This concludes this exercise.

- ✓ Close OpenRoads Designer and check the file back into ProjectWise

Exercise 2 (Optional): Parametric Constraints

This exercise demonstrates the use of a **Parametric Constraint** to override the pavement slope defined in the template to tie into the existing pavement at the beginning of the project

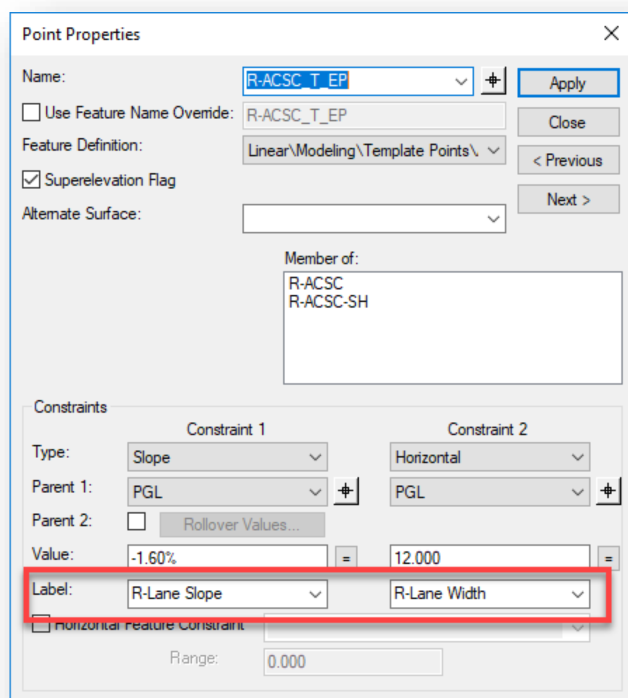
- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.
- Open the file **123456_KM001.dgn**.

To override the pavement slope using a **Parametric Constraint**, it is first necessary to define a **Label** for the point constraint that we wish to manipulate in the template definition. The label is defined in the point properties in the template.

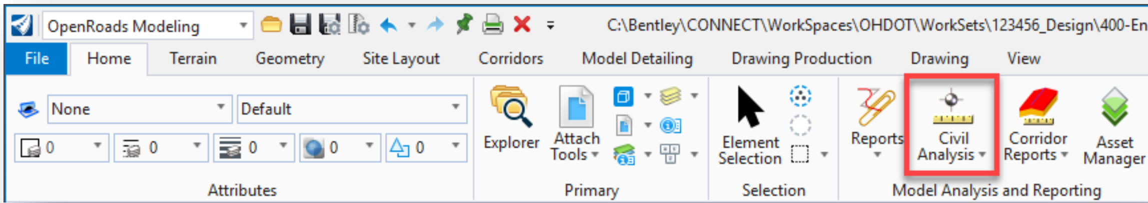
Many parametric labels are already defined in the ODOT template library. Take the following steps to review the labels assigned to the edge of pavement point in the S.R. 185 template:

- ✓ Open the ***Create Template*** dialog
- ✓ Double-click the **S.R. 185** template to make it active
- ✓ Double-click the point **R-ACSC_T_EP** in the template dialog
- ✓ Review the point properties. The edge of pavement point has two constraints assigned as shown below. Each constraint has a **Label** defined. These labels can be overridden in the corridor definition by assigning a parametric constraint.
- ✓ Close the template library after reviewing the point properties

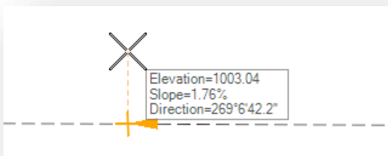


The next step is to analyze the existing pavement to get the existing elevation at the beginning of the project to compute the existing pavement cross slopes at station 386+25. To do this, we will use the **Analyze Point** command.

✓ From the **Home** tab, select **Civil Analysis > Analyze Point**.

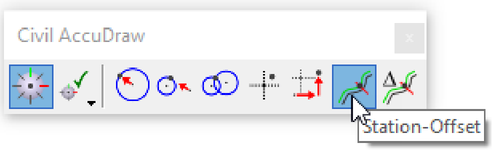


This command can be used to analyze the terrain model as well as 3D linear features. When prompted to **Select Element to Analyze Point**, choose the existing edge of pavement line on the west (right) side of the alignment. As you drag the cursor along the selected element, the **Elevation**, **Slope**, and **Direction** are reported as shown below.

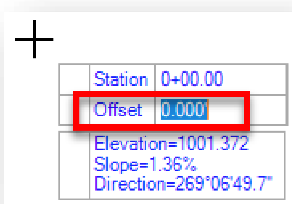


Civil Accudraw can be used to key in the station on the proposed centerline that you wish to analyze the selected feature.

- With Civil Accudraw toggled on select the **Station-Offset** ordinate.

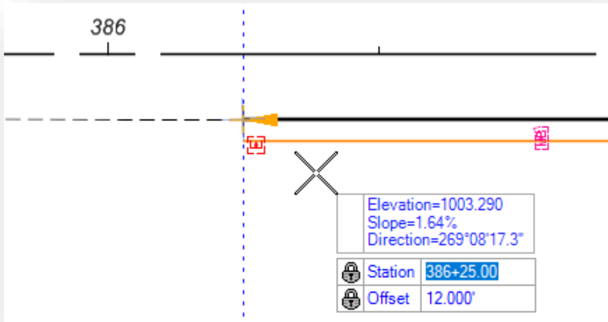


- In the Civil Accudraw settings floating on the cursor, select the Tab key to highlight the **Offset** field as shown below



With the keyboard focus set to the **Offset** field, key in the letter “o” to define the element that you wish to use for the station and offset ordinate. Select the centerline of proposed S.R. 185.

Once defined, you can now key in a **Station** value to get the elevation on the selected linear feature at the station location. The results are shown below.



Note: The **Offset** value is irrelevant as the command will project a perpendicular from the alignment to the selected linear feature to report the elevation on the linear feature.

Using the **Analyze Point** command, get the elevations for the left and right edge of pavement and the center of the existing pavement at station 386+25. Note, the surveyed line for the center of pavement is on the level **SV_X_COP**. Enter the elevation values in the table below.

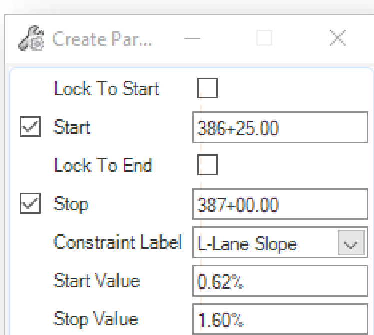
Selected Linear Feature	Elevation
Existing Left Edge of Pavement	
Proposed Center of Pavement	
Existing Right Edge of Pavement	

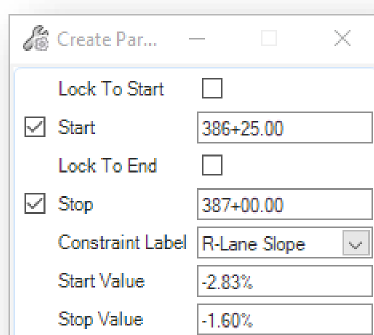
Compute the existing pavement cross slope and enter the value in the table below.

Side	Width	Cross Slope %
Left	12.0	
Right	12.0	

Take the following steps to define the **Parametric Constraints** to override the pavement slope at the beginning of the project.

- ✓ Select the **Create Parametric Constraint** command from the ribbon or from the pop-up menu that is accessed by selecting one of the corridor handles.
- ✓ Create parametric constraints for the **L-Lane Slope** and **R-Lane Slope** labels from station 386+25 using the slopes computed above, to end at 1.60% at station 387+00.

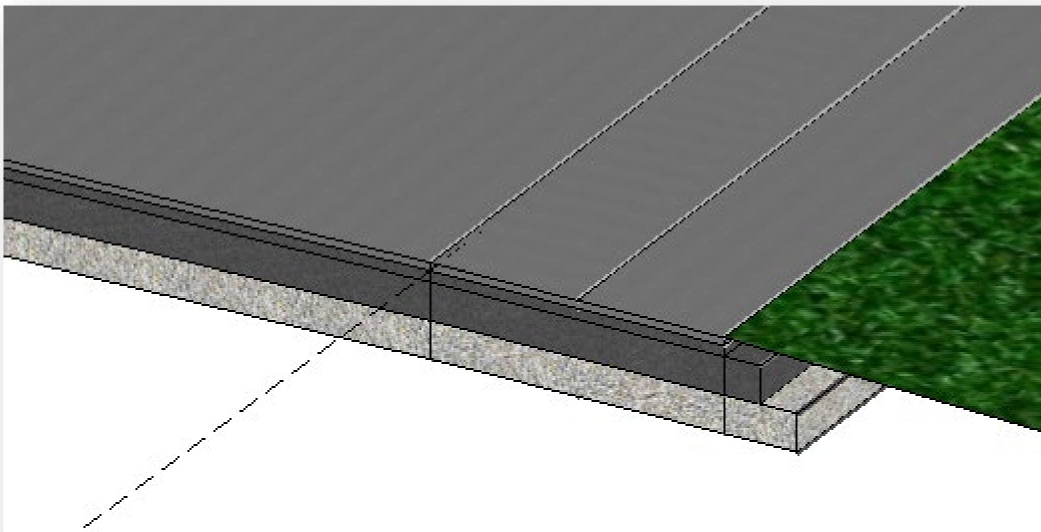




After the parametric constraints have been defined to set the value for the labels, the proposed pavement slope should meet the existing pavement in the 3D model, within an acceptable tolerance.

Whether it is necessary to adjust the model to this level of detail is debatable as the construction contractor will ensure that the proposed pavement ties into the existing pavement. However, this exercise illustrates the power of point controls and parametric constraints to override the initial template definition.

The 3D model for the right edge of pavement, after the parametric constraint for the proposed pavement slope was added, is shown below.



Exercise 3 (Optional): Ditch Profiles

The template definition for S.R. 185 contains end condition definitions for both cut and fill conditions. When the template is applied, the priority of each end condition is used to determine the order in which the end conditions are processed. As each option is evaluated, the first option that is successful will be used for the template drop location.

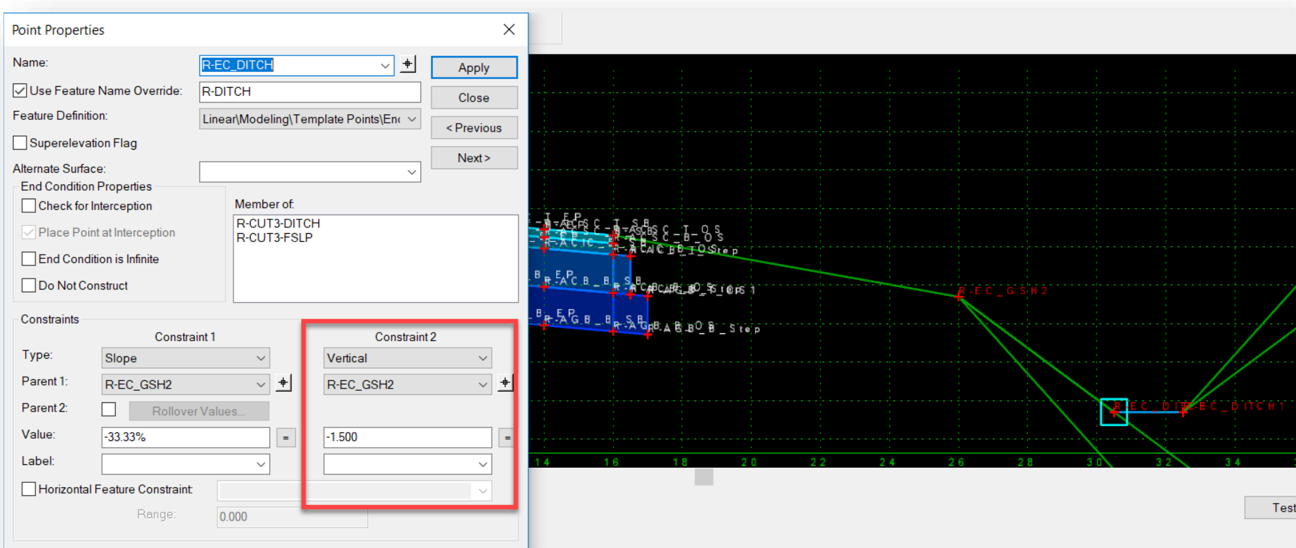
In many cases, it will be necessary to define a new profile for a ditch or even override the template solution to force a ditch in areas where the solution that was used for a range of template drop locations did not include a ditch. This exercise will look at two different ditch modification scenarios.

The length of the example project contains too many ditches for us to define all of them in this training course. We will look at two specific areas to introduce users to the available tools to define ditch profiles.

Part 1: Redefining a Ditch Profile with an Elevation and Grade Point Control

Point controls can be used to defile a ditch profile by a starting elevation and a grade for a range of stations.

The template used for S.R. 185 will draw a ditch in cut conditions. This ditch point **R-EC_DITCH** is set with a **Vertical** constraint value of -1.5' from point **R-EC_GSH2** as shown below.



Defining a point control for point **R-EC_Ditch** will also adjust the elevation for point **R-EC_Ditch1** since it is constrained vertically to point **R-EC_Ditch**.

In this example we will define a new profile for the ditch on the right side of the corridor from stations 390+75 to 394+00.

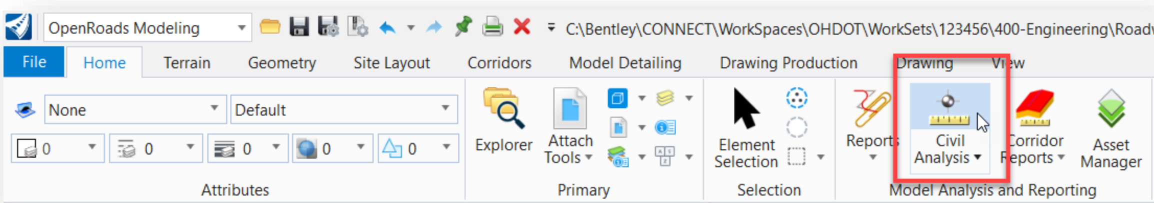
- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.

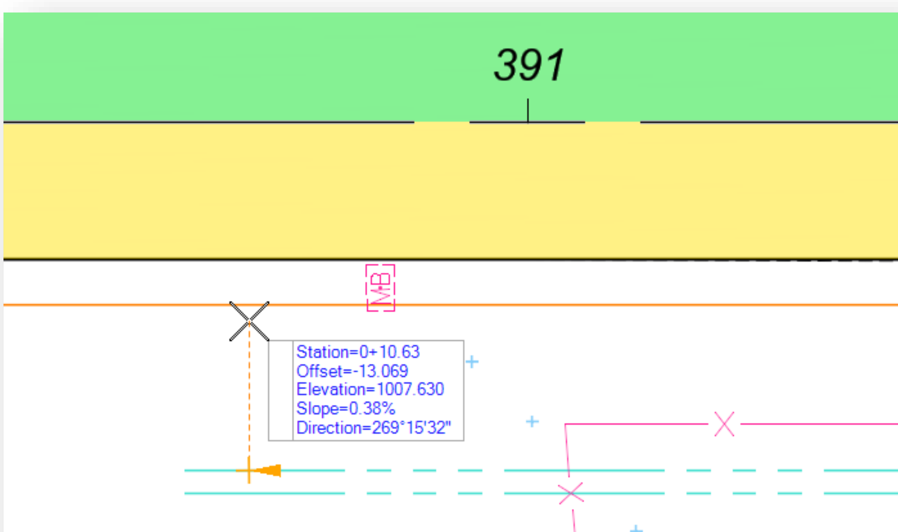
✓ Open the file containing the S.R. 185 corridor, **123456_KM001.dgn**.

Before defining the point control, it is necessary to evaluate the elevation of the current ditch that was drawn as part of the corridor modeling process. The Analyze Point command is used to analyze selected civil elements.

✓ From the **Home** tab, select the **Analyze Point** command.



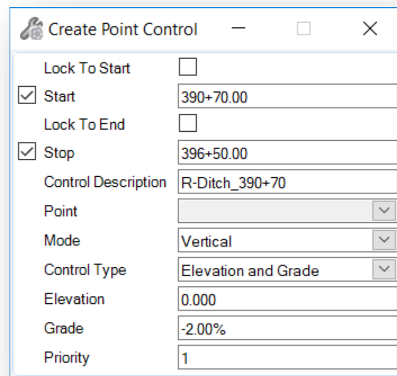
✓ Select the **R-Ditch** line when prompted to **Select Element to Analyze Point**. Values along the selected element is reported relative to the cursor location as shown below.



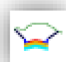
Note: The **Station** value reported in the dialog is computed relative to the selected element, not the centerline. Civil Accudraw can be used to specify a specific station location along the centerline element.

For this example, use the elevation at the beginning to the R-Ditch element by snapping the to ditch line graphic

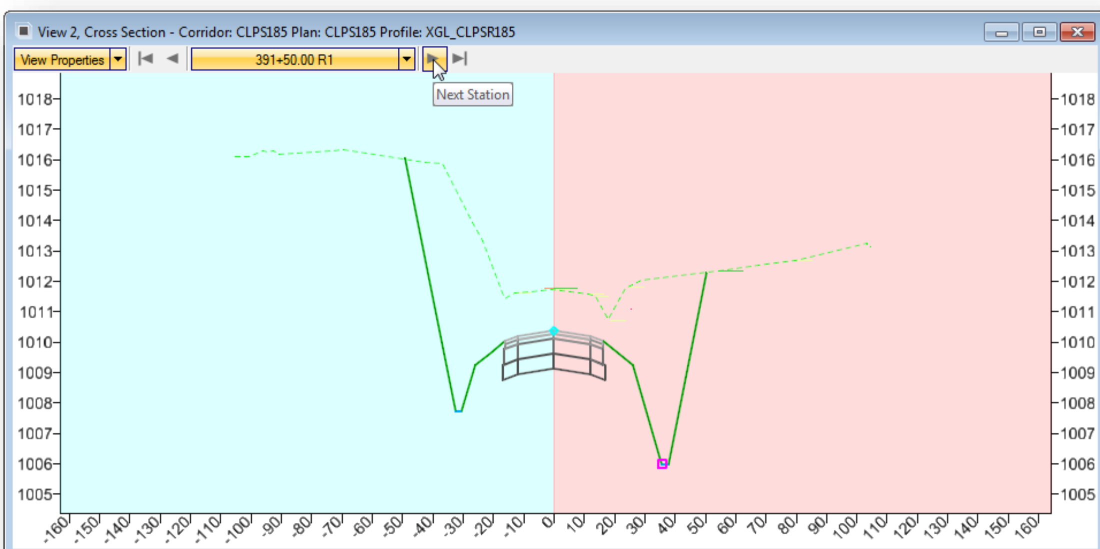
- ✓ From the **Corridors** tab, select **Edits > Create Point Control**. When prompted to **Locate Corridor**, select one of the handles for the S.R. 185 corridor model.
- ✓ Create the vertical point control using the settings shown at right below.



- ✓ When the point control is applied, the 3:1 slope from the graded shoulder to the ditch elevation is maintained. The ditch line moves out appropriately.



- ✓ The results can be reviewed by opening a cross-section view. From the *Tasks* menu, select **Tasks > Civil Tools > Corridor Modeling > Open Cross Section View**. When prompted, select the S.R. 185 corridor and then data point inside view 2. The cross section for Station 391+50 is shown below. The point control is indicated by the purple box drawn around point R-EC_Ditch.



- ✓ Review the 3D model to inspect the changes. This is not a legitimate design solution as the ditch does not have an outlet, but the example illustrates one way to manipulate a ditch profile.

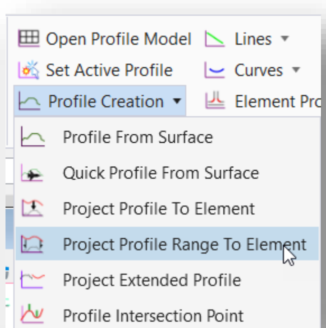
Part 2: Defining a Special Ditch Profile

When a corridor model is created, some of the Feature Definitions that have been assigned to the points in the template are configured to draw 2D plan view graphics into the corridor design file. These plan view graphics have a profile defined that can be reviewed in a profile window.

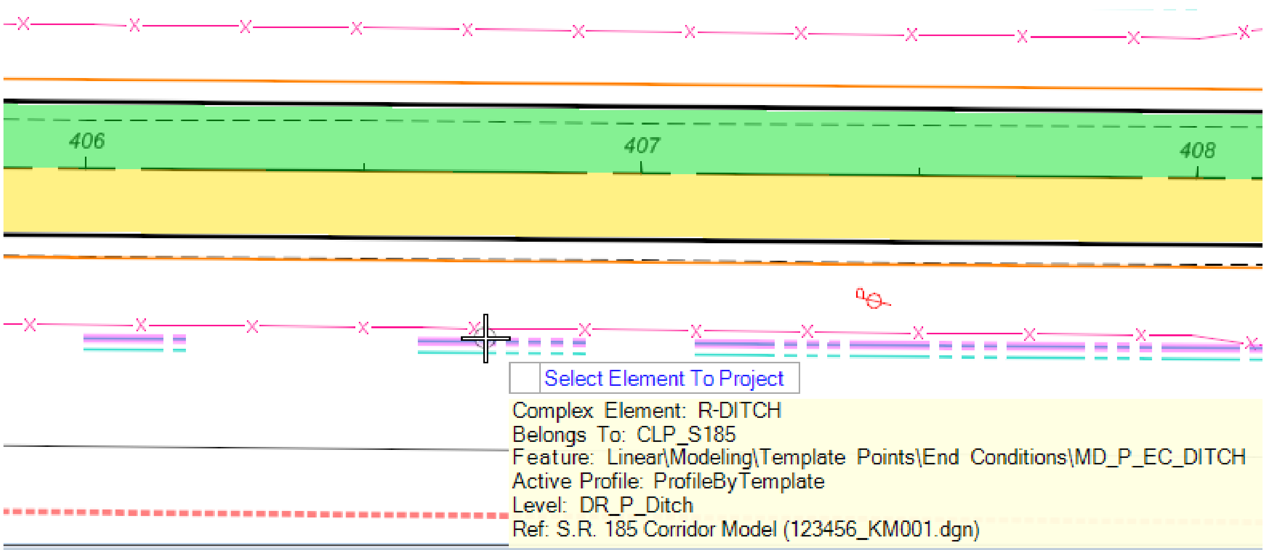
The **Feature Definition** MD_P_EC_DITCH is configured to draw 2D plan graphics in the corridor design file. The ditch profiles can be displayed relative to the centerline stationing in the profile window. All profiles must be defined in the same design file that contains the centerline geometry, the geometry basemap "BK" design file.

- Open **ProjectWise Explorer**

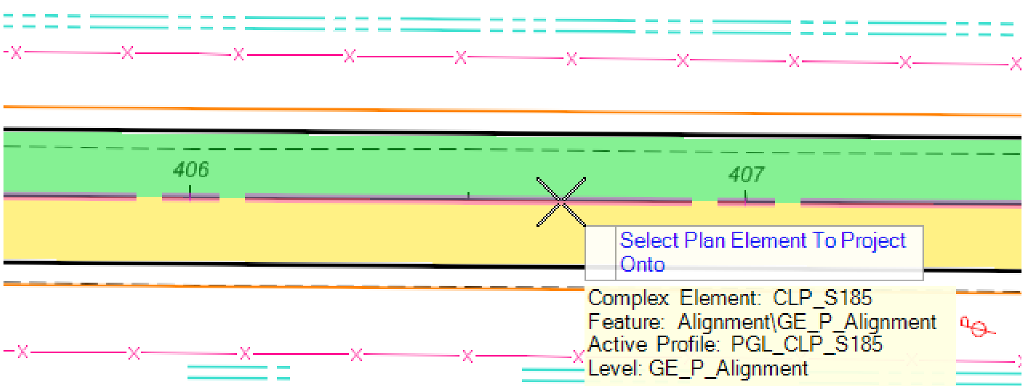
- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.
- ✓ To define a new profile for the ditch, open the file that contains the S.R. 185 alignment, **123456_BK001.dgn**.
- ✓ Attach a refence for the corridor model design file, **123456_KM001.dgn**
- ✓ In the plan view window, hold down the right-mouse button until the pop-up menu appears. Select the **View Control > 2 Views Plan/Profile** option. Follow the prompts to open a profile view for the S.R. 185 alignment.



Select Element to Project, select the inside ditch line (R-Ditch) on the right side of S.R. 185.



Select Plan Element to Project Onto, select the plan view S.R. 185 alignment element



Start Station:

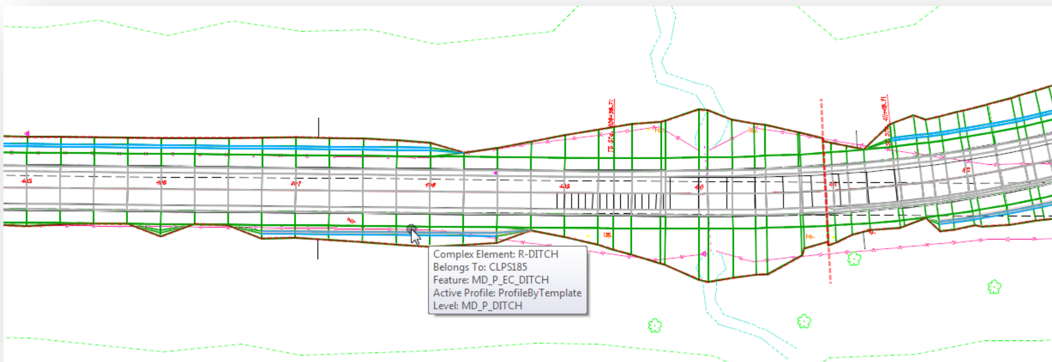
End Station:

✓ Open the design file for the S.R. 185 geometry:
..\Design\Geopak\Basemaps\12345_BK001.dgn

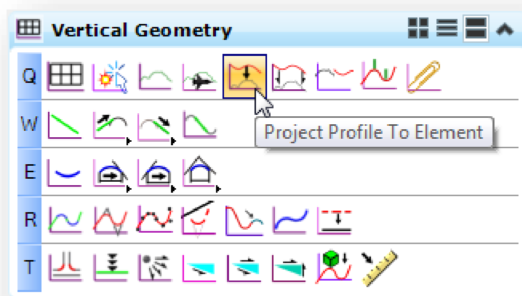
✓ Attach the Design model for the S.R. 185 corridor design file with a **Nest Depth** value of **0**.
..\Design\Geopak\Basemaps\12345_KM001.dgn

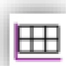
It is not necessary to attach the corridor model to design a new ditch profile. This is only necessary if you wish to display the ditch profile that was generated by the corridor model onto the S.R. 185 profile.

We will define a new ditch profile for the right side of the corridor from stations 405+00 to 412+00. Notice that the template did not place a ditch for the full station range.

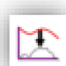


The profile of the ditch line as defined by the initial corridor model processing can be projected to the S.R. 185 profile by use of the **Project Profile to Element** command, or the **Project Profile Range to Element** command, contained in the **Tasks > Civil Tools > Vertical Geometry** group.



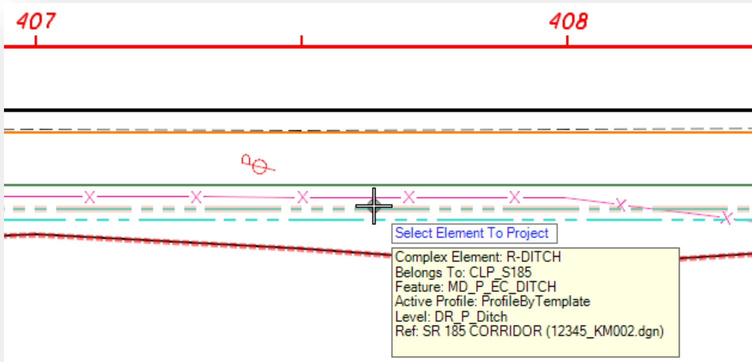


✓ To define the new ditch profile, select the centerline of S.R. 185 and choose the **Open Profile Model** command from the pop-up menu. Open view 2 to display the profile.

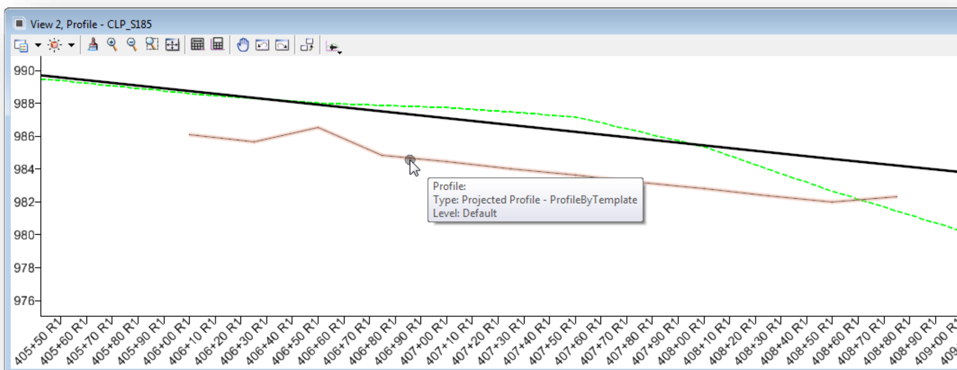


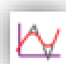
✓ To display the current ditch profile as defined in the corridor model, select the **Project Profile to Element** command.

- When prompted to **Select Element to Project**, choose the inside ditch line on the right side of the corridor



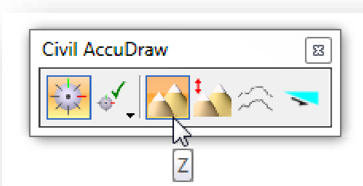
- When prompted to **Select Element to Project Onto**, choose the horizontal alignment for SR 185. The ditch profile is displayed in the profile view as shown below.





✓ In the *Profile* view window, choose the **Profile Complex by VPI** command.

✓ Toggle on *Civil Accudraw* and select the **Z** ordinate option.

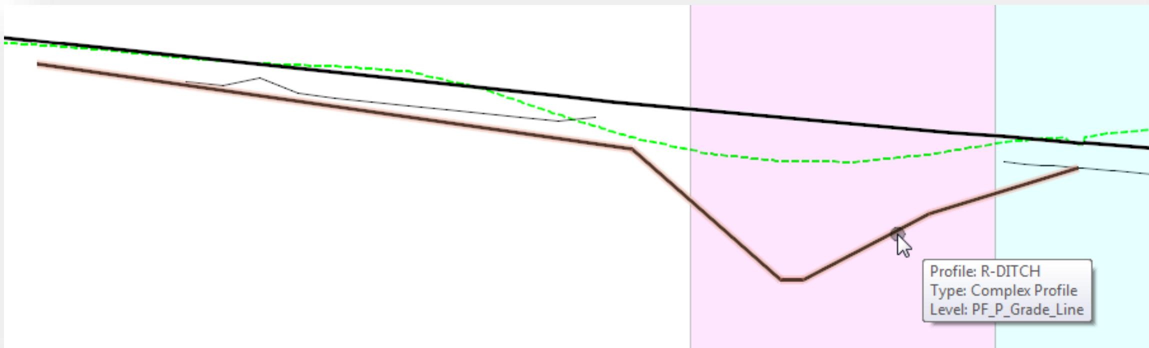


✓ Place a profile without any vertical curves (**Curve Length** = 0) using the following station and elevation values.

405+00	987.94
409+00	979.00
410+00	965.25
410+15	965.25
411+00	972.25

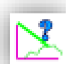
412+00 977.00

✓ After placing the profile, select the profile and choose the **Properties** icon from the pop-up menu. Define the **Feature Name** of the profile as **R-Ditch**. The ditch profile should appear like the example below.

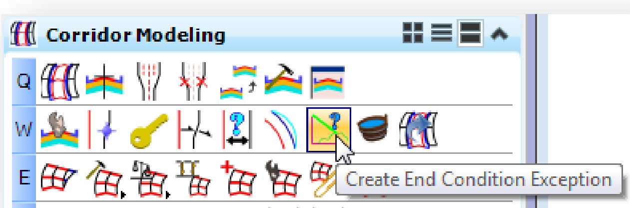


✓ Open the design file for the S.R. 185 corridor model:
..\Design\Geopak\Basemaps\12345_KM001.dgn

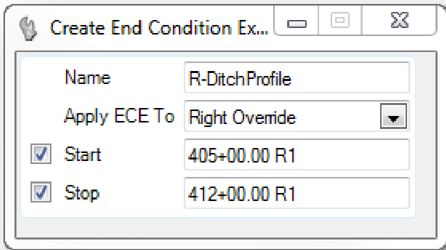
First we will define an **End Condition Override** to define a ditch for the full station range from 405+00 to 412+00.



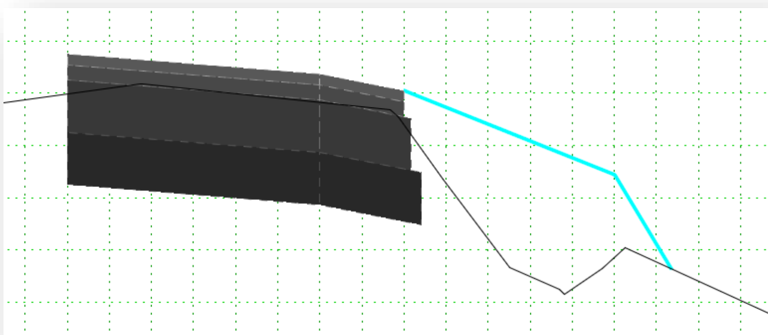
✓ From the *Tasks* menu, select **Tasks > Civil Tools > Corridor Modeling > Create End Condition Exception**. When promoted, select the S.R. 185 corridor.



✓ Define the end condition exception with the values shown below.



- ✓ The *Right Override* dialog is opened to display the right side template drop at station 405+00. Right-click in the template window and choose the **Delete Components** option to delete the graded shoulder and side slope lines. The deleted lines are still displayed as shown below.

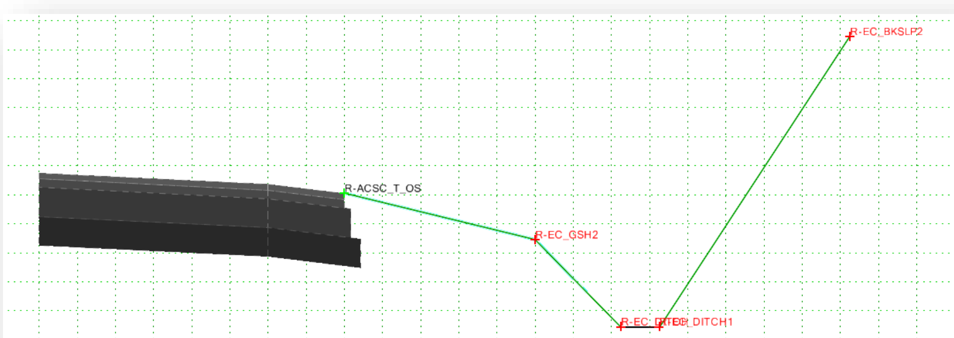


- ✓ Open the *Dynamic Settings* dialog (**Tools > Dynamic Settings**) and toggle on the **Apply Affixes** option

- ✓ Select the following end condition and drag it onto the edge of shoulder point:

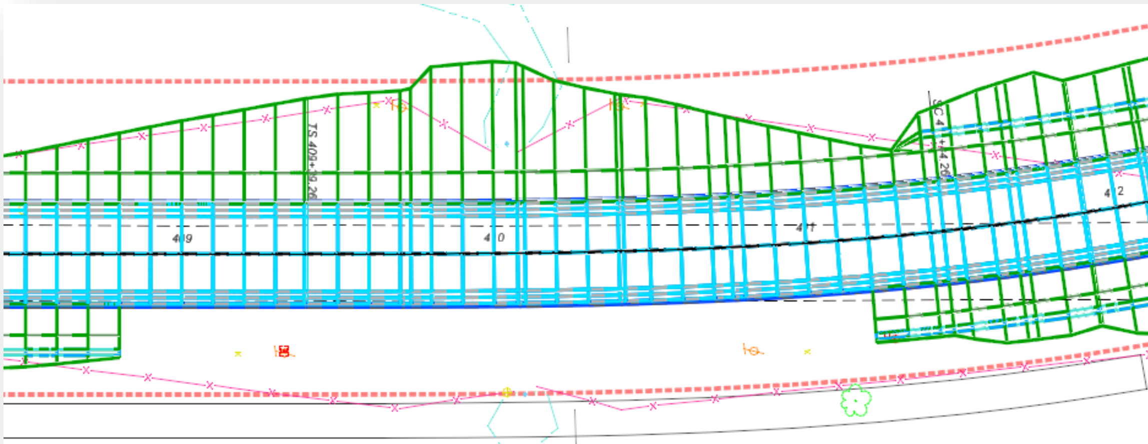
  ODOT > End Conditions > 307-4E Common and Barrier Grading > 307-4E Cut 3:1

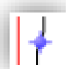
- ✓ Delete the 3:1 backslope component. The end condition override should appear as shown below.



When the dialog is closed, the corridor is reprocessed and the ditch is drawn for template drop locations where the template conditions are met.

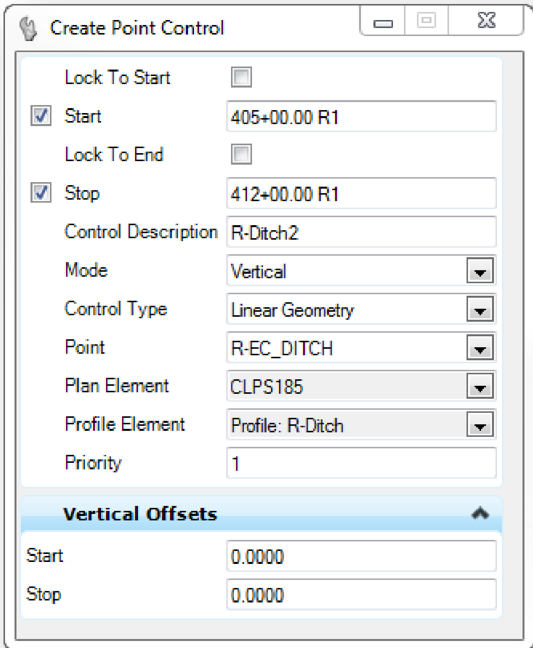
For this example, there will be stations where the end condition does not solve as shown below. We will remedy this by assigning the new ditch profile to the corridor.



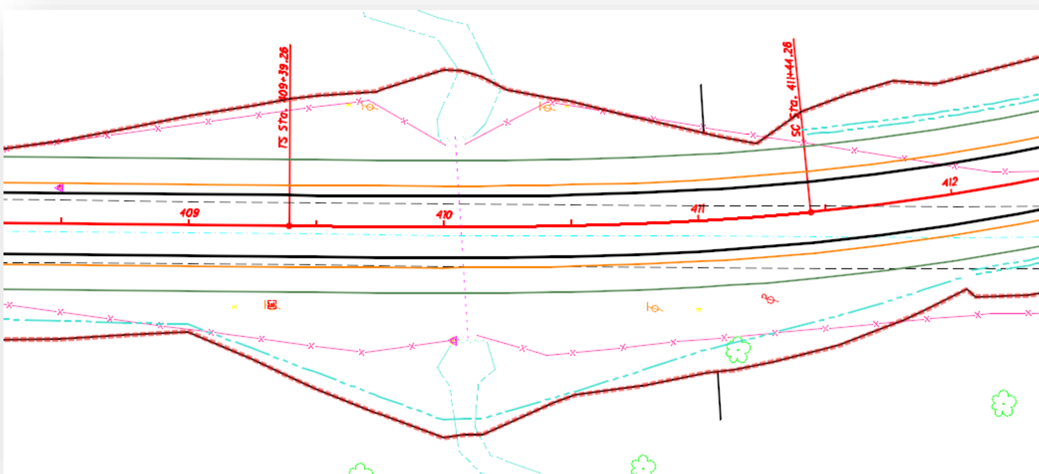


✓ From the *Tasks* menu, select **Tasks > Civil Tools > Corridor Modeling > Create Point Control**. Select the S.R. 185 corridor when prompted.

✓ Create the point control using the settings shown below.



When the ditch profile is applied, some of the template drops will display the new ditch line as shown in the example below



Normally a culvert would be designed for this location. Culvert design and grading around the culvert are beyond the scope of this training course. The purpose of this exercise is to illustrate the process to define and apply a special ditch profile in the corridor model.

Exercise 4 (Optional): End Condition Overrides

This exercise will demonstrate how to apply an end condition override to the side slopes for a specified station range.

- Open **ProjectWise Explorer**

- Open the **Ohio DOT Training/Testing** data source
- Under **Active Projects**, browse to your home District. Central Office employees use District 06
- Open the **_D##** folder. You should see your username in the folder. Open your username. Browse to the **Design_Training** folder.
- Open the project with PID number **123456**.
- Browse to the **401-Engineering\Roadway\Basemaps** folder.

✓ Open the file **123456_KM001.dgn**.

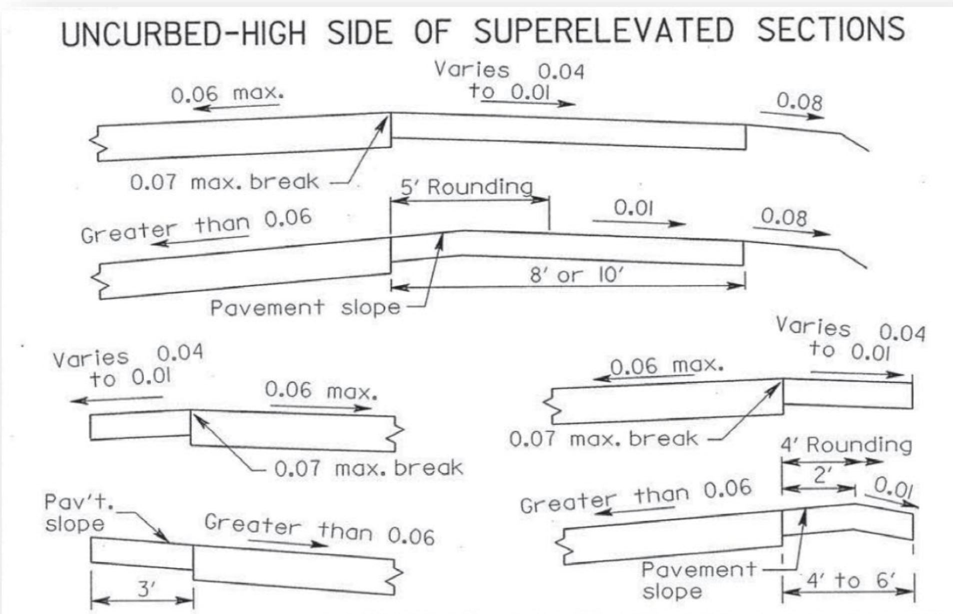
Section 506 Superelevation

Superelevation Overview

Superelevation commands calculate how much banking to apply to curves in the horizontal alignment to help offset centrifugal force. These commands also compute how the road will make the transition from normal crown to a fully banked curve and back again.

Pavement slopes are initially defined in the template. Superelevation is created and applied after the corridor is defined and the template drops have been assigned. This chapter will detail how superelevation is computed with OpenRoads as well as how to accurately model the shoulder break for the high side of Superelevation as defined in the **Location & Design Manual, Volume 1 - Roadway Design, Figures 301-8E, 301-9E, and 301-10E** using the Table Wizard with ODOT's ASCII Superelevation Tables.

For this example, we will reference the shoulder break for uncurbed roadways defined in **Figure 301-8E** shown below.

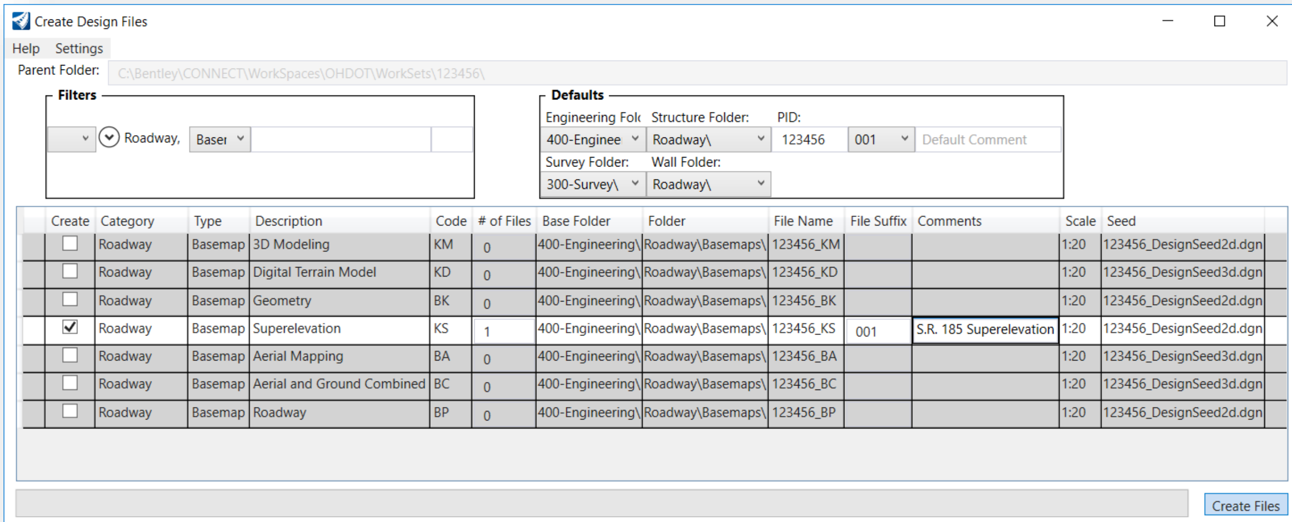


The behavior for the low side of superelevation is easily facilitated by applying a **Rollover Lock** to the shoulder point in the template definition.

The high side of superelevation presents some challenges. For pavement slopes greater than 3%, the shoulder slope maintains a 7% maximum break. This can also be facilitated with the **Rollover Lock** parameter in the template point properties. If the pavement slope exceeds 6%, a 2' break is introduced. Modeling this behavior cannot be modeled using the rollover lock. This chapter will document the procedure to model the superelevation using the ODOT template library.

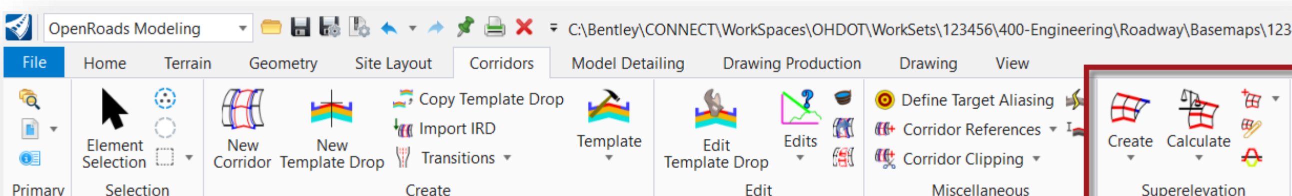
Superelevation Design Files

Superelevation graphics for each alignment are normally be placed in a separate "KS" design file. The **OHDOT Create Design files** application is used to create superelevation design files for the project as shown below.



Applying Superelevation to a Corridor

The Superelevation tools are located under the **Corridors** tab, as shown below.



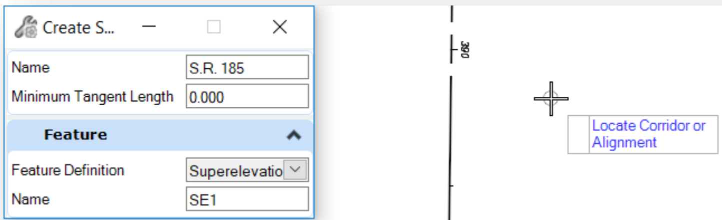
The following tools are available:

- Create Superelevation Sections
- Create Superelevation Lanes
- Create Superelevation Lanes by Road Template
- Calculate Superelevation
- Edit Superelevation rule File
- Import Superelevation
- Assign to Corridor
- Insert Station Cross-Slope
- Superelevation Editor
- Superelevation Report
- Open Superelevation View

Note: See the [Bentley online help documentation] for tools not covered in this training guide.

Create Superelevation Sections

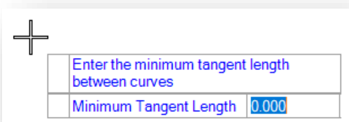
When the **Create Superelevation Section** tool is accessed, the user is prompted to enter a **Name** for the superelevation section. After accepting the Name, the user is next prompted to select the alignment as shown below.



Select the **Superelevation** feature definition from the ODOT library.

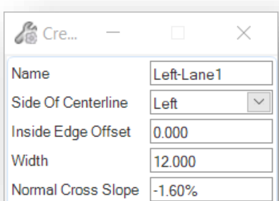
After the alignment is selected, enter the **Start** and **End Stations** for the superelevation definition. Use the <ALT> key to lock the values to the beginning and end of the selected alignment.

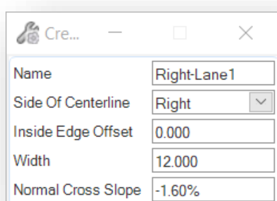
When prompted, enter a **Minimum Tangent Length** value of 0. If the tangent distance between two adjacent curves is less the specified value, the two curves and adjacent tangent are included in a single superelevation section. If the tangent distance between two adjacent curves is greater than the specified value, each curve and half of the adjacent tangent are included into each superelevation section. Enter a value larger than the total alignment length to create a single superelevation section.



ODOT recommends using a value of 0 to create a separate superelevation section for each curve set.

The user is next prompted to define the pavement lanes for the roadway. Each lane is entered like the example below. Reset (right-click) after defining all the pavement lanes.

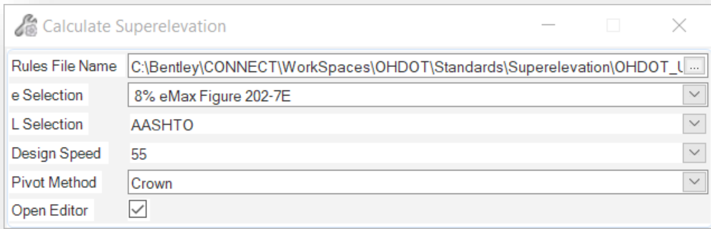




After the lanes have been defined, the user is prompted to select the superelevation rules file. Two rules files are provided as part of the OHDOT standards.

- OHDOT_Divided.xml
- OHDOT_Undivided.xml

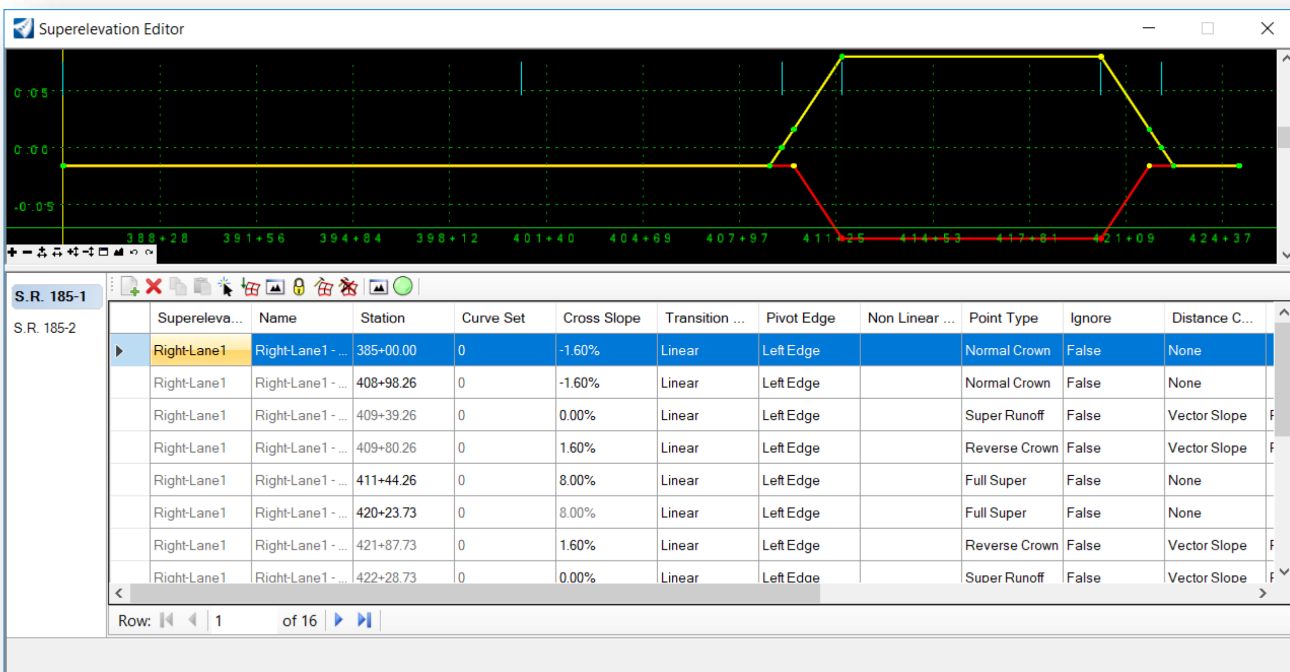
Select the appropriate file for the project.



Set the remaining parameters appropriately for the project.

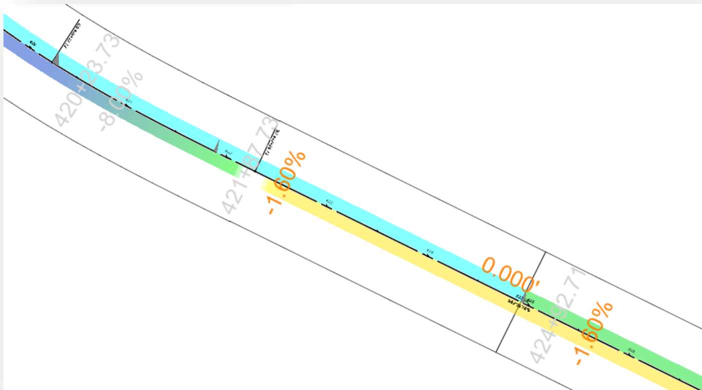
Note: The ODOT Superelevation Rules Files are provided as a starting point for the superelevation computation. It is the responsibility of the designer to review and adjust the values as necessary for the project.

If the **Open Editor** parameter is toggled on, the *Superelevation Editor* dialog is opened like the example below.



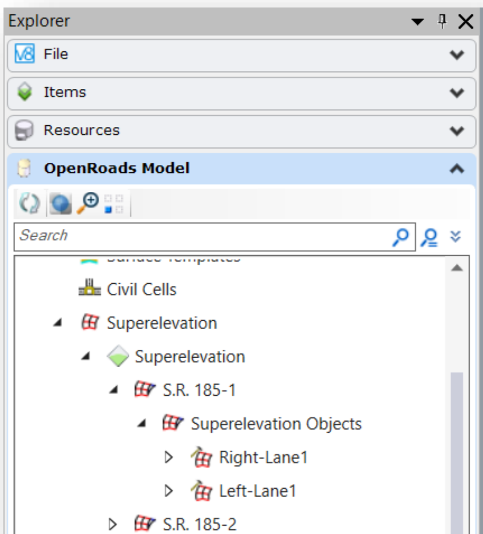
This dialog is used to review and edit the superelevation parameters. See the online help for information on the Superelevation Editor dialog.

The superelevation rules can also be reviewed and edited by selecting one of the pavement lanes as shown below. Each of the text items designating the station and cross slope can be edited by selecting the text.

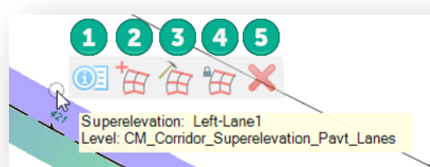


Superelevation values can also be reviewed and edited in the Properties dialog when the Superelevation Section graphic is selected.

Superelevation values can also be edited in the *Explorer* dialog's **OpenRoads Model** tab as shown below.



Selecting a superelevation shape, as shown at right, allows you to access a menu with the following commands:



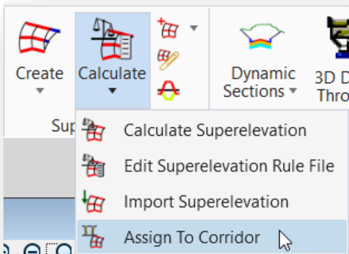
1. Properties
2. Insert Superelevation Station/Cross Slope
3. Create Single Control Line
4. Apply Shoulder Rollover Lock
5. Delete

Note: See the [Bentley online help documentation] for more information on these commands.

Assign a Superelevation to Corridor

After the superelevation values have been computed, the next step in the process is to assign the superelevation sections to a corridor. Before assigning the superelevation to the corridor, it is necessary to attach the reference file containing the superelevation definition (KS file) to the corridor model (KM file).

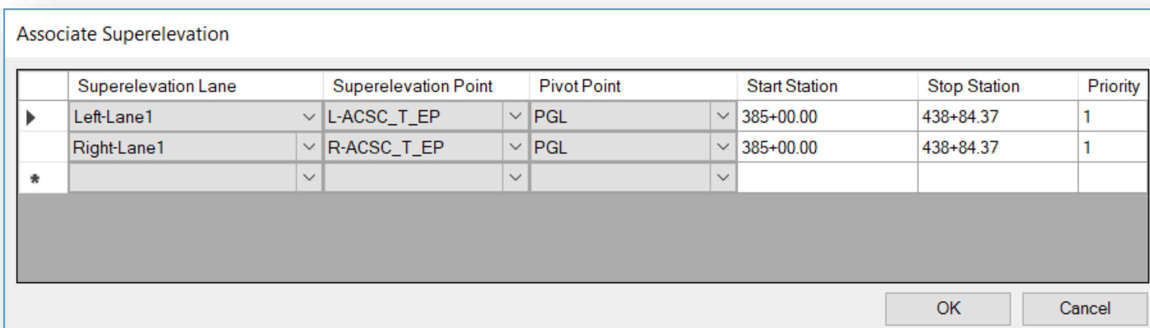
After the reference file containing the superelevation definition has been attached, it can now be assigned to the corridor. From the Corridor tab, choose the **Calculate > Assign to Corridor** command.



When prompted, select each superelevation section that you wish to apply to the corridor. Reset (right-click) after selecting each superelevation section.

When prompted, select the corridor.

After the sections and the corridor have been identified, the *Associate Superelevation* dialog is opened. This dialog is used to associate specific points in the template to the superelevation lanes.



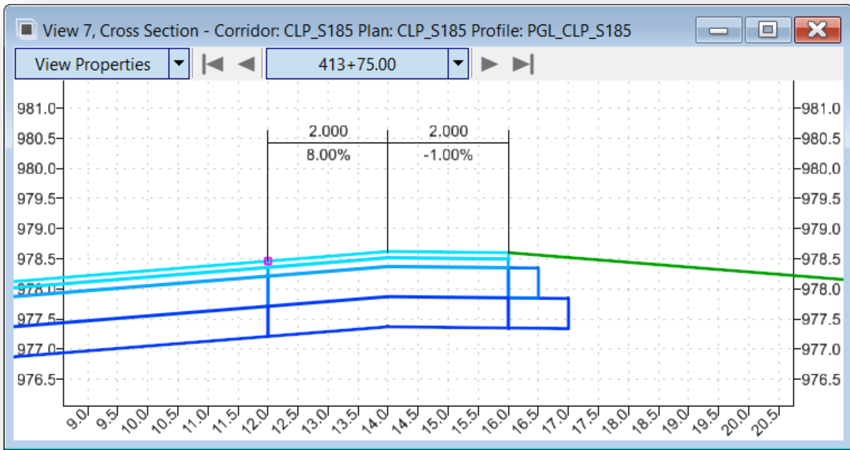
In the example above, the **Superelevation Lane Left-Lane1** and **Right-Lane1** are shown. The objects default to the points that have been identified with the **Superelevation Flag** toggled on in the template definition.

After reviewing and making any corrections to the assigned points for each lane, select **OK** to apply the superelevation to the corridor.

The model is automatically updated with superelevation values, which override the cross slopes defined in the templates.

Superelevation is applied to the pavement lanes. The shoulder slope is controlled by the rollover lock values defined in the template.

The example below shows a cross-section view displaying the slopes for the shoulder roll-over for the high side of superelevation using the OHDOT template.



Import Superelevation from CSV file

Many users may prefer to create a csv file with the superelevation transition information rather than use one of the xml files provided by ODOT to compute the superelevation values.

See the provided Bentley online help for a complete listing of all available parameters for CSV file import. The minimum required parameters will be discussed in this document.

The file format of the comma separated values file containing the required information is as follows:

Superelevation Lane, Station, Cross Slope, Pivot About

Data	**Description**
Superelevation Lane Name	Required – Links the data to an existing superelevation lane, therefore, the name in the CSV file must match the lane names created prior to importation.
Station	Required - Station where cross slope is defined.
Cross Slope	Required - Cross slope at the specified station formatted as a double value: (-2% = -0.02).
Pivot About	Required - Identifies which edge to pivot about. Generally, if you are rotating about the center line, the right lane would pivot about the left edge (LS) while the left lane would pivot about the right edge (RS). LS = left side (edge) RS = right side (edge)

Example input:

Right-Lane1, 386+25, -0.016, LS
Right-Lane1, 408+98.26, -0.016, LS
Right-Lane1, 409+39.26, 0.00, LS
Right-Lane1, 409+80.26, 0.016, LS
Right-Lane1, 411+44.26, 0.080, LS

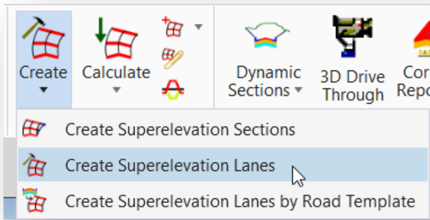
The workflow to import superelevation from a CSV file is as follows:

1. Create the CSV file, based on file format.
2. Create superelevation section and lanes, ensuring the lane names match those used in the CSV file.
3. Select the Import Superelevation tool.
4. Follow the heads-up prompts to choose the Superelevation Section and the CSV file to import.

Adding Addition Lanes

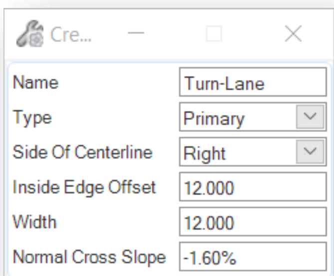
Additional lanes can be added to an existing superelevation section by use of the **Create Superelevation Lanes** command.

From the **Corridor** tab, select **Create > Create Superelevation Lanes**.



When prompted, select the superelevation section to create additional lanes.

The parameters for the superelevation lane are shown below.



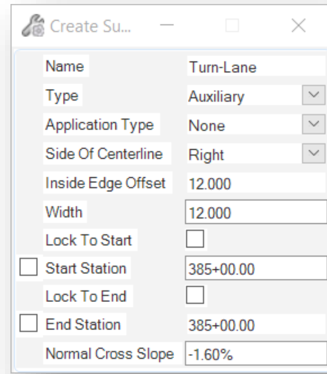
Define a **Name** for the lane definition.

Two values are supported for the **Type** parameter:

- **Primary** lanes - a normal cross slope is specified when the lane is defined and then the cross slope is calculated from the superelevation rate file or imported from a CSV file when the superelevation is calculated. Primary lanes extend the entire length of a section. The begin and end stations are defined by the section. Primary lanes are generally the through lanes that extend throughout the project.
- **Auxiliary** lanes - the cross slope is a fixed user-specified value or is automatically set to match an adjacent superelevation lane. Auxiliary lanes have a user-specified begin and end station but must be within a single section. Auxiliary lanes are lanes that are added and dropped along the alignment such as turn lanes.

When the **Type** is set to **Auxiliary**, the dialog is adjusted to reflect the parameters shown below:

The **Application Type** parameter is used to define how the cross slope will be defined. The parameter can be set as follows:

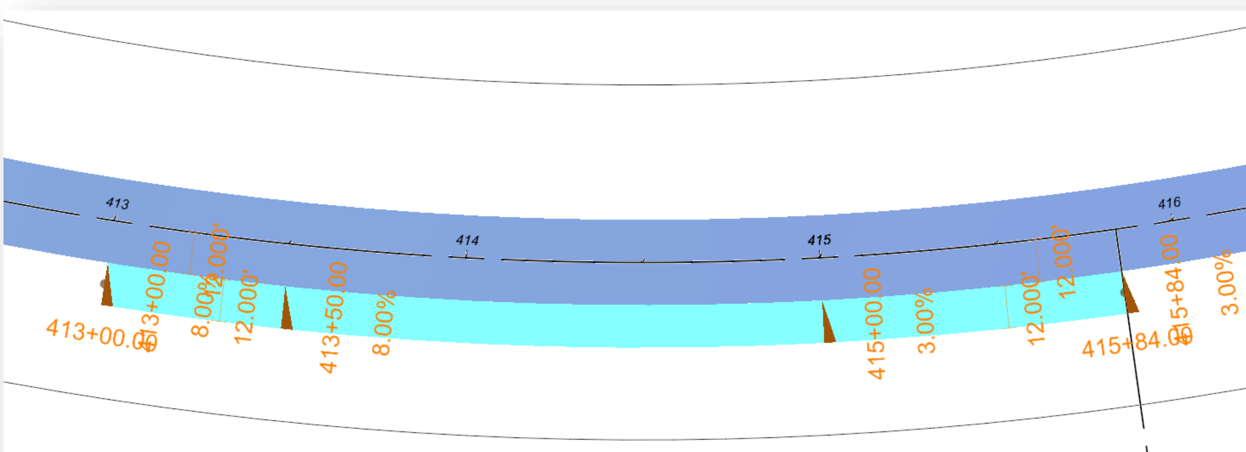


None - no cross slopes are assigned. Use this option if the cross slopes are defined by importing a CSV file with the superelevation values.

Constant - Fixed slope of specified value. If lane is transition, use this option, then modify one end in the editor or graphically after creation.

Follow Adjacent – This option projects cross slopes from the adjacent lane to this lane.

In the example below, an **Auxiliary** lane was defined for a turn lane. The cross slopes were defined by importing a CSV file with the values.



Section 507 Superelevation Exercises

Exercise 1: S.R. 185 Superelevation

In this exercise we will define the superelevation sections for the State Route 185 Corridor.

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.

- Browse to the **400-Engineering\Roadway\Basemaps** folder.

Create new design files for the project using the **OHDOT Create Design Files** application. To create these files, one of the existing BK files may be opened.

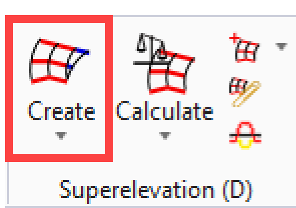
- Select the **OHDOT Create Design Files** application to create the superelevation basemap design files for the project.
    - Create the file listed below

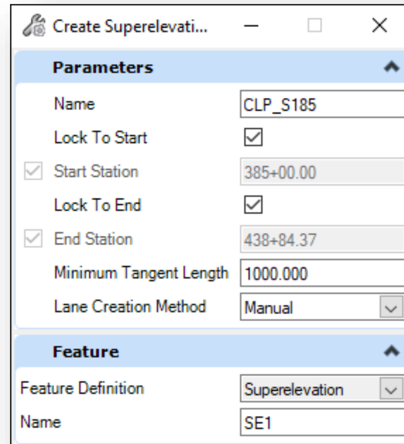
Design File name	Comments
123456_KS001.dgn	S.R. 185 Superelevation

- Open the file **123456_KS001.dgn**, checking the current file back in to ProjectWise

- Attach the following reference files:
  300-Survey\Basemaps\123456_FB001.dgn
  400-Engineering\Roadway\Basemaps\123456_BK001.dgn
  400-Engineering\Roadway\Basemaps\123456_KM001.dgn

✓ From the **Corridors** tab, select **Create Superelevation Sections**





- ✓ You are prompted to define the **Name** for the superelevation section. Use the same name as the horizontal alignment, CLP_S185.

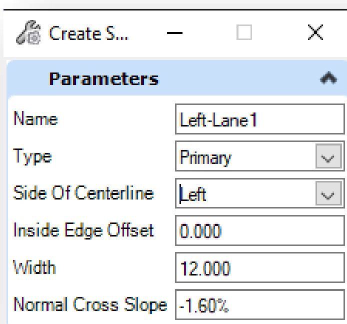
- ✓ Next you are prompted to select the alignment. Select the CLP_S185 alignment

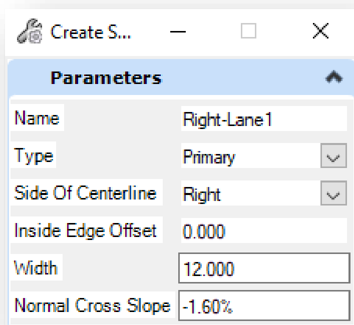
- ✓ Use the entire length of the chain for the start and ending stations by toggling on the **Lock to Start** and **Lock to End** options

- ✓ Before accepting the creation of the superelevation dialog select the feature definition and Name. (Shown to the right)

Next, the software will step you through the process to define the pavement lanes.

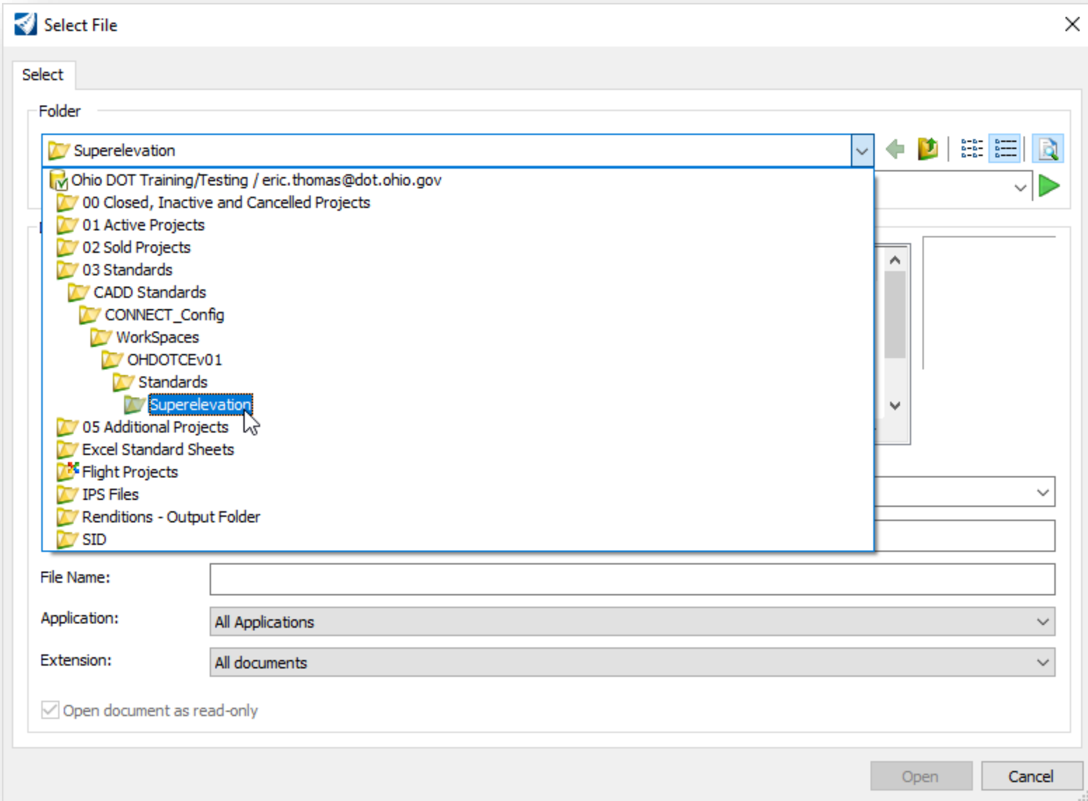
- ✓ Define two lanes for the superelevation as shown below



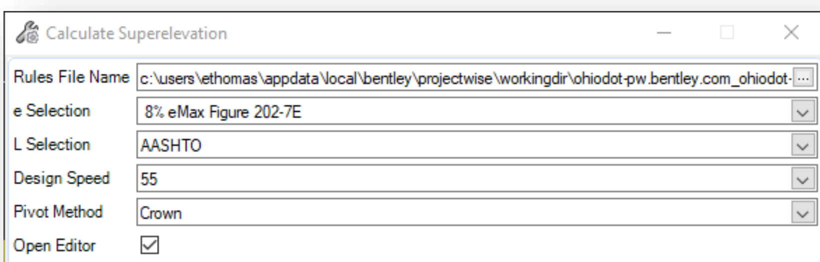


- ✓ After defining both lanes, right click to terminate the lane creation process. The *Calculate Superelevation* dialog is opened. Choose the **ODOT_Undivided.xml** file by navigating to the following location in the OHDOT standards

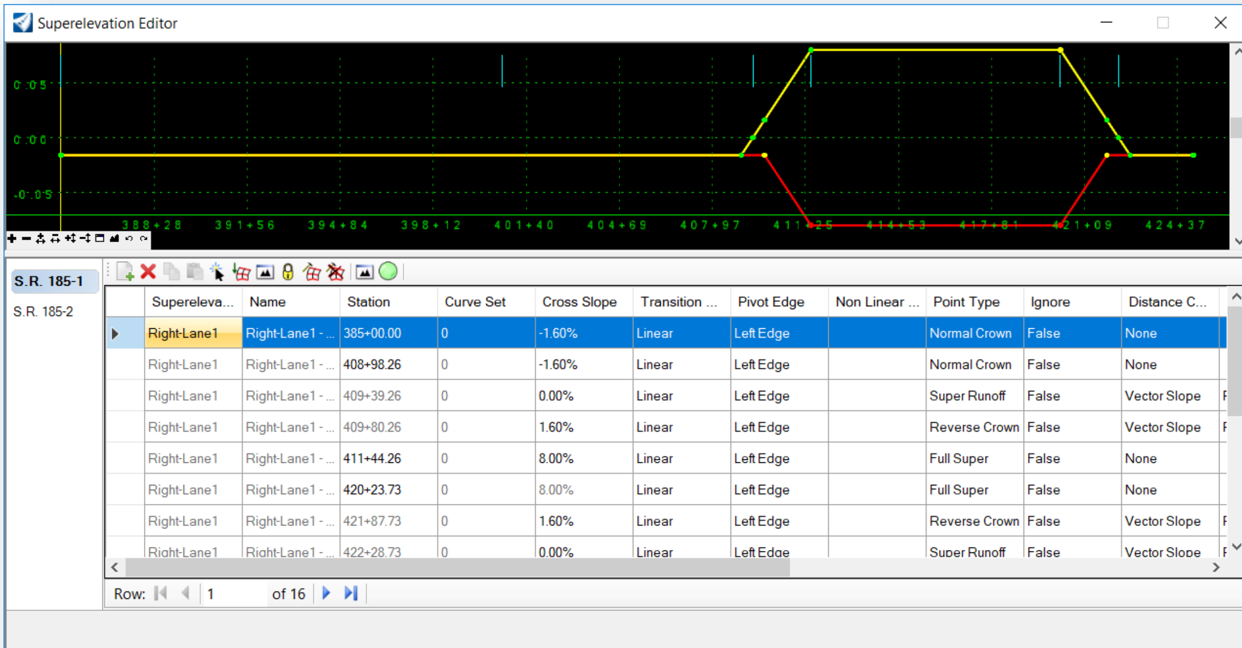
..\OHDOTCEv02\Standards\Superelevation



Use the settings shown below.



✓ After accepting the dialog settings, the *Superelevation Editor* dialog is opened.



Review the superelevation parameters and close the dialog when completed. The maximum superelevation rate should be 8.0% for this alignment.

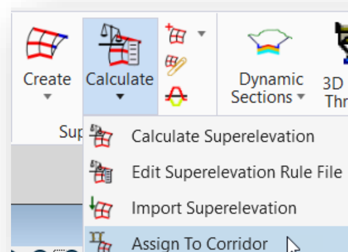
- ✓ Save settings in the superelevation design file

The **Assign Superelevation to Corridor** command is used to associate the superelevation settings with the corridor model and the template points. This assignment is made in the corridor design file.

- ✓ Browse to open the **12345_KM001.dgn** file containing the corridor for S.R. 185. Be sure to check the superelevation design file back into ProjectWise.

- ✓ Attach the superelevation definition design file, **12345_KS001.dgn**, to the corridor design file, **12345_KM001.dgn**.

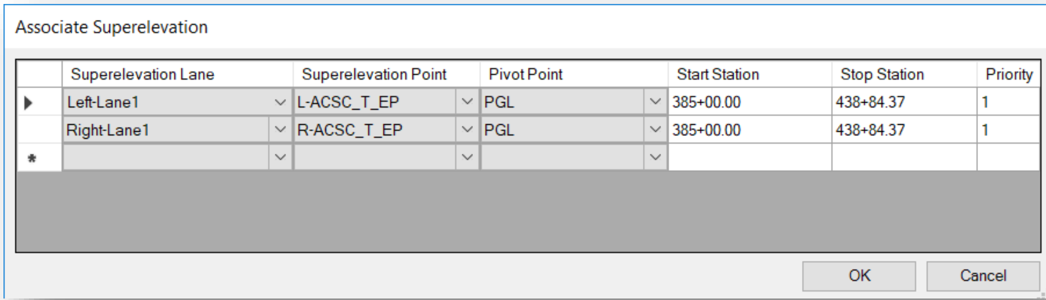
- ✓ From the Corridors tab, select the **Assign to Corridor** command to associate the superelevation settings with the corridor model. (Shown to the right)



- ✓ Choose the graphics that were drawn for the boundary of both superelevation sections, and then right-click (reset) to accept the sections

- ✓ When prompted to **Locate Corridor**, select one of the handles for the S.R. 185 corridor boundary.

- ✓ The *Associate Superelevation* dialog is opened. The dialog should default to the correct settings, but it is good practice to review the settings before accepting them.

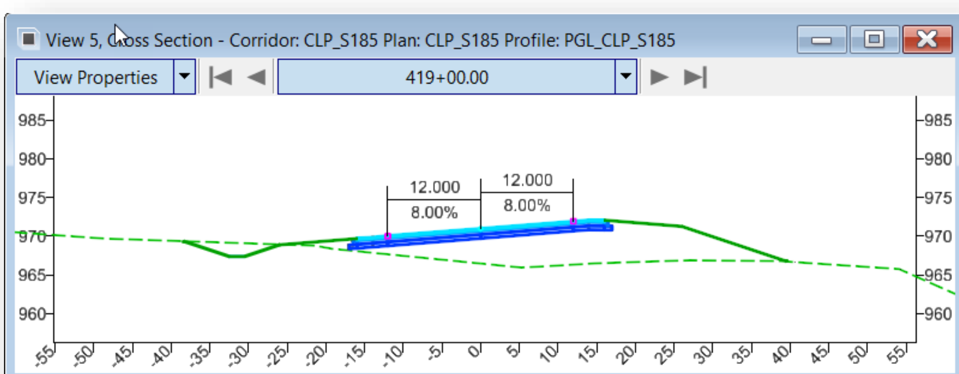


After associating the superelevation sections with the corridor, use the **Open Cross Section View** command to review the corridor.

- ✓ From the **Corridors** tab, select the **Dynamic Sections > Open Cross Section View** command.
  Advanced tip:
  *If a **multi-model view** isn't already open, you can use the **View Control** command by holding the right-mouse button within the design model view. Select **3 views Plan/XS/3D**. Then follow software prompts to view cross sections.*

- ✓ When prompted, select the corridor, and select a new view window to display the corridor.

- ✓ Add some temporary dimension lines to the cross-section view to facilitate reviewing the pavement width and cross slopes
  - Hold down the right-mouse button in the cross-section view.
  - Choose the **Place Horizontal Temporary Dimension** command
  - Identify the starting and ending points to add temporary dimension lines for each lane of pavement like the example below
  - Once you've placed Temporary dimensions you can also remove them by holding down the right-mouse button and select **Remove All Temporary Dimensions** command.



After reviewing the corridor cross sections, close the cross-section view window. Click Save settings to save view layout.

This completes the work on S.R. 185. Exit OpenRoads Designer and check the file back into ProjectWise.

Optional Superelevation Exercises

Optional Exercise 1: S.R. 185 Superelevation

In this exercise we will define the superelevation sections for the State Route 185 Corridor.

- Open **ProjectWise Explorer**

- Open the **Ohio DOT Training/Testing** data source
- Under **06 Training Projects**, browse to your provided folder. If you don't have a folder reach out to the CADD & Mapping office to have one created.
- You should see your username in the folder. Open your username. Browse to the **Design_Training** folder.
- Open the project with PID number **123456**.
- Browse to the **400-Engineering\Roadway\Basemaps** folder.

Create new design files for the project using the **OHDOT Create Design Files** application. To create these files, one of the existing BK files may be opened.

- Select the **OHDOT Create Design Files** application to create the superelevation basemap design files for the project.
  - Create two files, as listed below

Design File name	Comments
123456_KS001.dgn	S.R. 185 Superelevation
123456_KS002.dgn	Jameson Road Superelevation

- Open the file **123456_KS001.dgn**, checking the current file back in to ProjectWise
- Attach the following reference files:
  300-Survey\Basemaps\123456_FB001.dgn
  400-Engineering\Roadway\Basemaps\123456_BK000.dgn
  400-Engineering\Roadway\Basemaps\123456_KM001.dgn

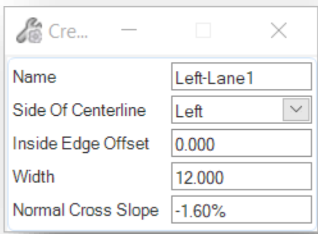
✓ From the **Corridors** tab, select **Create Superelevation Sections**

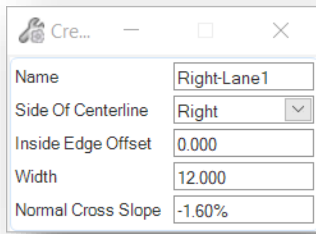
✓ You are prompted to define the **Name** for the superelevation section. Use the same name as the horizontal alignment, CLP_S185.

- ✓ Next you are prompted to select the alignment. Select the CLP_S185 alignment
- ✓ Use the entire length of the chain for the start and ending stations by toggling on the **Lock to Start** and **Lock to End** options

Next, the software will step you through the process to define the pavement lanes.

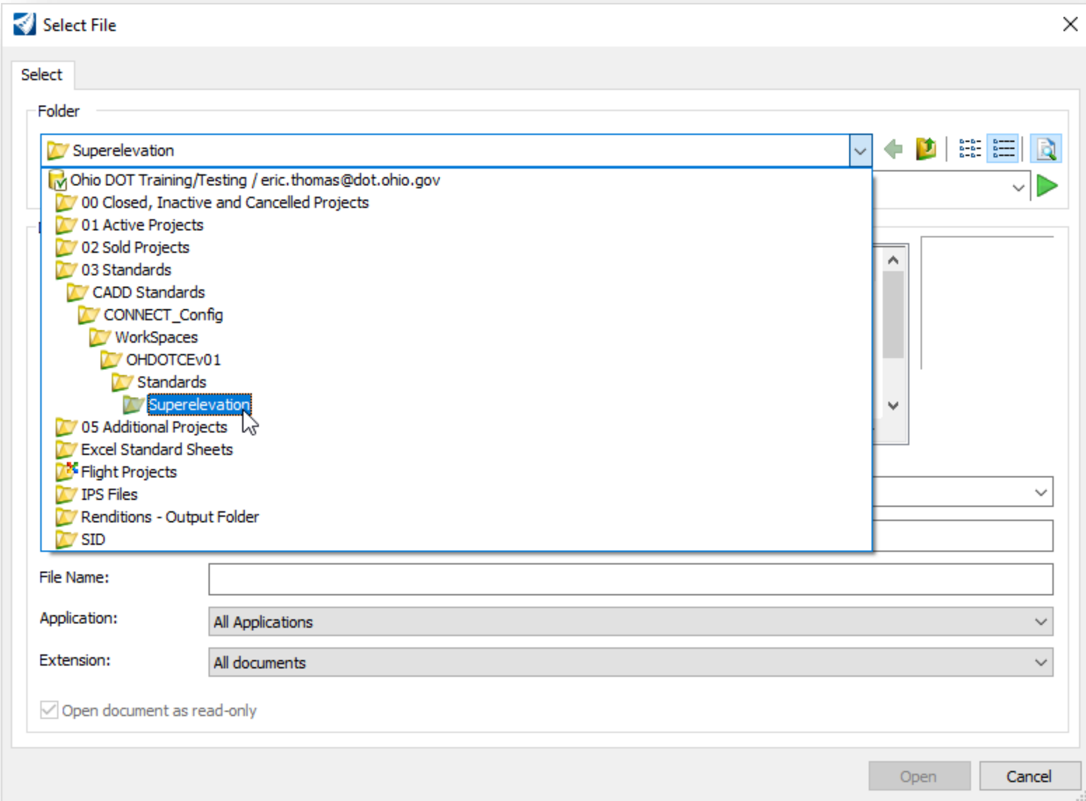
- ✓ Define two lanes for the superelevation as shown below



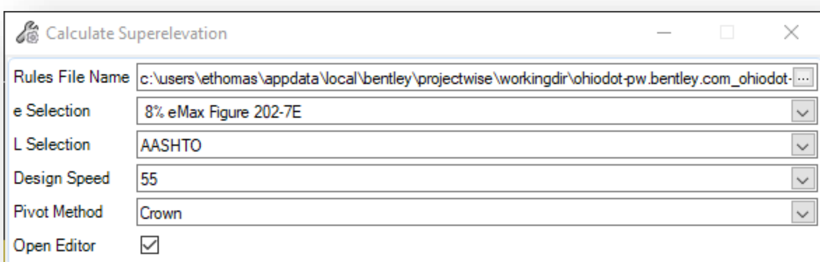


- ✓ After defining both lanes, right click to terminate the lane creation process. The *Calculate Superelevation* dialog is opened. Choose the **ODOT_Undivided.xml** file by navigating to the following location in the OHDOT standards

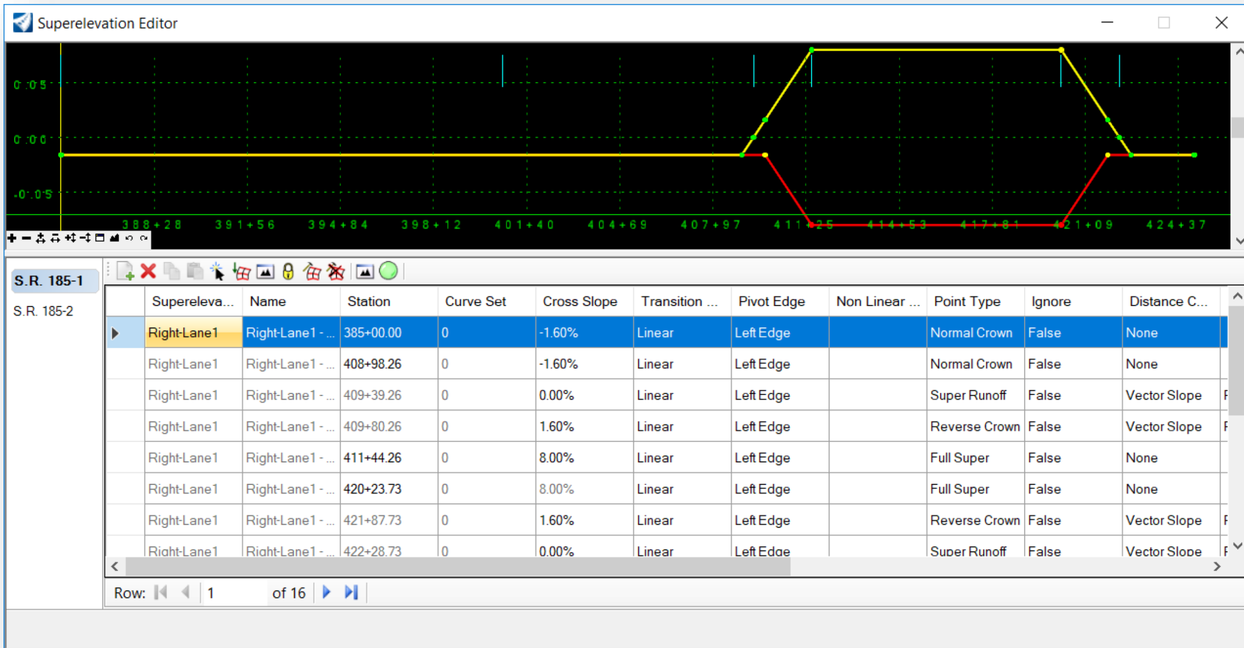
..\OHDOTCEv02\Standards\Superelevation



Use the settings shown below.



✓ After accepting the dialog settings, the *Superelevation Editor* dialog is opened.



Review the superelevation parameters and close the dialog when completed. The maximum superelevation rate should be 8.0% for this alignment.

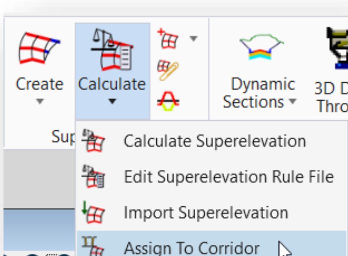
✓ Save settings in the superelevation design file

The **Assign Superelevation to Corridor** command is used to associate the superelevation settings with the corridor model and the template points. This assignment is made in the corridor design file.

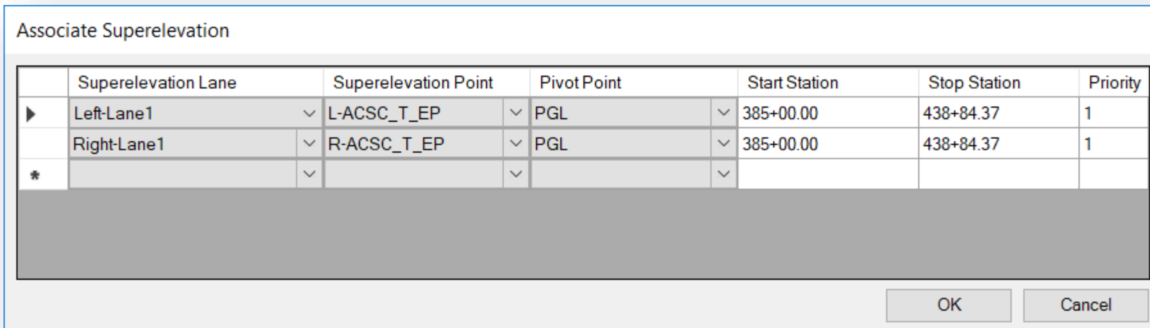
✓ Browse to open the **12345_KM001.dgn** file containing the corridor for S.R. 185. Be sure to check the superelevation design file back in to ProjectWise.

✓ Attach the superelevation definition design file, **12345_KS001.dgn**, to the corridor design file, **12345_KM001.dgn**.

✓ From the Corridors tab, select the **Assign Superelevation to Corridor** command to associate the superelevation settings with the corridor model.

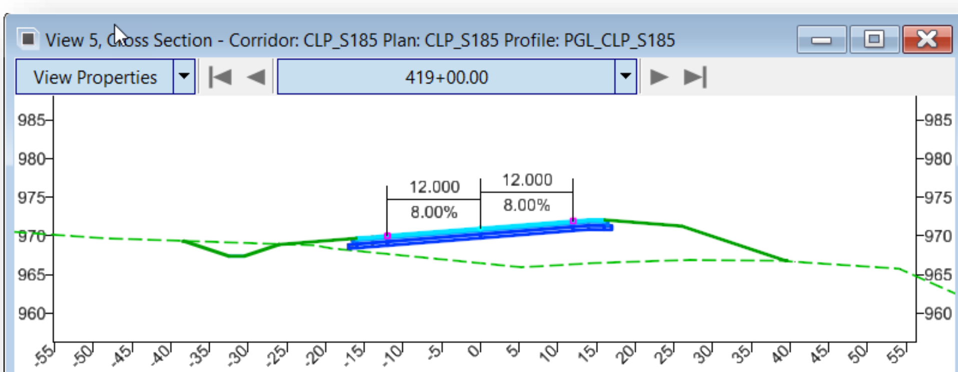


- ✓ Choose the graphics that were drawn for the boundary of both superelevation sections, and then right-click (reset) to accept the sections
- ✓ When prompted to **Locate Corridor**, select one of the handles for the S.R. 185 corridor boundary.
- ✓ The *Associate Superelevation* dialog is opened. The dialog should default to the correct settings but it is good practice to review the settings before accepting them.



After associating the superelevation sections with the corridor, use the **Open Cross Section View** command to review the corridor.

- ✓ From the **Corridors** tab, select the **Dynamic Sections > Open Cross Section View** command
- ✓ When prompted, select the corridor and select a new view window to display the corridor.
- ✓ Add some temporary dimension lines to the cross-section view to facilitate reviewing the pavement width and cross slopes
  - Hold down the right-mouse button in the cross-section view.
  - Choose the **Place Horizontal Temporary Dimension** command
  - Identify the starting and ending points to add temporary dimension lines for each lane of pavement like the example below



After reviewing the corridor cross sections, close the cross-section view window.

Optional Exercise 2: S.R. 185 and Jameson Road Turn Lane

This exercise will demonstrate the process of adding a right-turn lane to S.R. 185 at the Jameson Road intersection.

Part A - Adding the turn lane to S.R. 185

The first step is to add the turn lane for S.R. 185. Begin by defining the pavement slopes for the turn lane in a CSV file.

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.

- Browse to the **400-Engineering\Roadway\Basemaps\EngData** folder

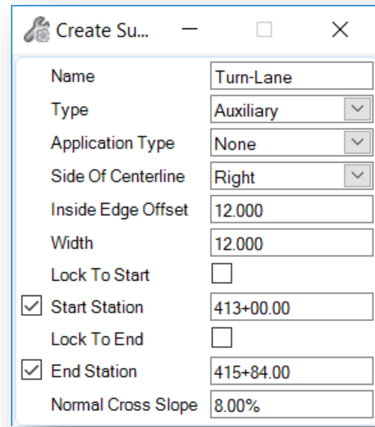
- Use Notepad to create an ASCII file in CSV format with the following values:

Lane Name	Station	Cross Slope	Pivot About
Turn-Lane	413+00	0.08	LS
Turn-Lane	413+50	0.08	LS
Turn-Lane	415+00	0.03	LS
Turn-Lane	415+84	0.03	LS

  - Name the file **SR185_Turn_Lane.csv**
  - Save the file in the following folder within ProjectWise:
    - \400-Engineering\Roadway\EngData\

- Open the file **123456_KS001.dgn**

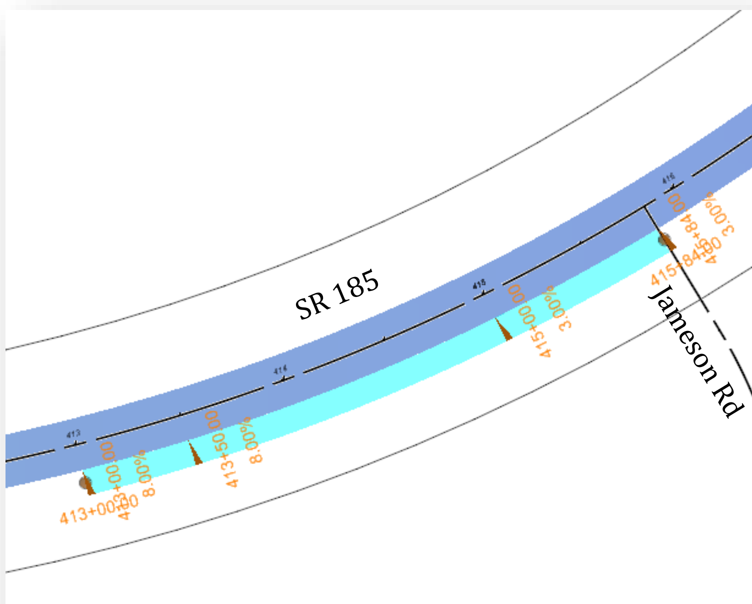
- From the **Corridors** tab **Superelevation** group, select **Create > Create Superelevation Lanes**



- When prompted to **Locate Superelevation Section**, select the first S.R. 185 Superelevation boundary element
- Use the settings shown on the right to create the turn lane

- From the **Corridors** tab **Superelevation** group, select **Calculate > Import Superelevation**
  - In the *Import Superelevation* dialog, browse to select the CSV file
  - When prompted to **Locate Superelevation Section**, select the first S.R. 185 Superelevation boundary element

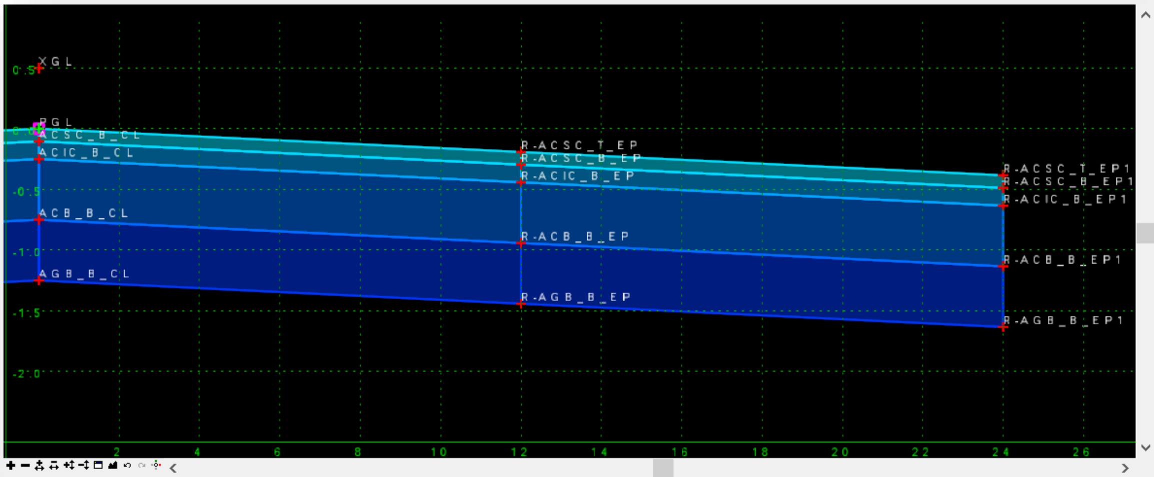
The slopes are assigned to the turn lane like the example below.



Next, create a new template with the turn lane to the S.R. 185 corridor. This is necessary since the pavement cross slope for the new turn lane does not match the mainline cross slope.

- ✓ Copy the S.R. 185 template and paste a copy in the Project Templates folder
- ✓ Rename the copied template "S.R. 185 Turn Lane"
- ✓ Double-click the template name to make it the active template
- ✓ Delete the shoulder and side slope components from the copied template
- ✓ Open the *Dynamic Settings* dialog and toggle on the **Apply Affixes** option

- ✓ Drag the pavement component **ODOT > Components > Pavement > Buildups > Asphalt Concrete without Stabilization** on to the right edge of pavement point. The new lane is added to the template like the example below.

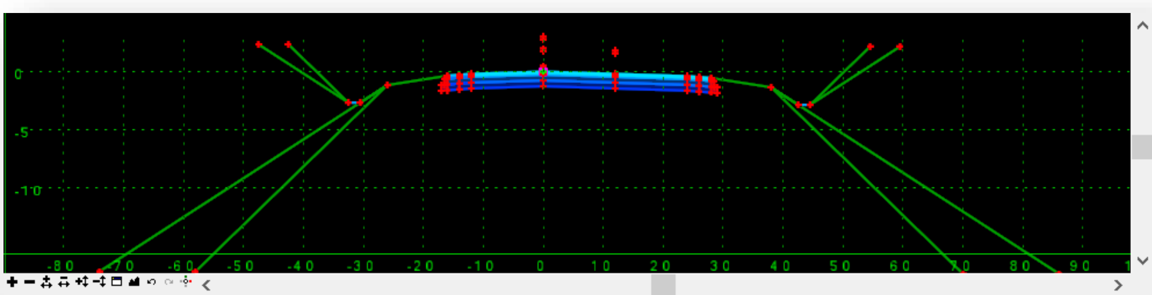


- ✓ Drag the shoulder component **ODOT > Components > Shoulder > 301-8E Asphalt Concrete Without Subgrade Stabilization** onto point **R-ASCS_T_EP**

- ✓ Drag the end condition **ODOT > End Conditions > 307-4E Common and Barrier Grading > 307-4E 3:1 Cut, Fill Table** on to the point **R-ACSC_T_EP1**

- ✓ Delete the following components from the end condition:
  - R-GSH1
  - R-FILL4-FSLP

  The template should appear like the example shown on the next page.



The shoulder component includes null points, **INT-Right** and **DRIVE-Right**, that seek out the intersection with a driveway or an intersecting road. When the shoulder component was added to the S.R. 185 Turn Lane template, these points are given the prefix **R-** and are named **R-INT-Right** and **R-DRIVE-Right**. These points will function as defined to remove the pavement steps in the event a driveway match line is encountered, and to remove the shoulder components in the event an intersection match line is encountered.

- ✓ Delete the points **INT-Right** and **DRIVE-Right**.

  Point **R-DRIVE-Right** is not needed for this template drop range as there are no driveways in this station range.

- ✓ Delete the point **R-DRIVE-Right**

- ✓ Test the behavior of points **R-INT-Right** by right-clicking on the point and choosing **Test Point Controls > Test Horizontal Point Control**.

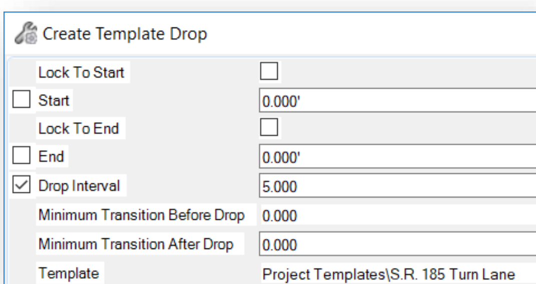
  Note that the shoulder components are removed appropriately, but the end condition components are not removed.

- ✓ Double-click on the component **R-GSH2** to access the *Component Properties* dialog.
  - o Assign component **R-ACSC-SH** as the **Parent Component**
  - o Select **Apply** and **Close** the *Component Properties* dialog

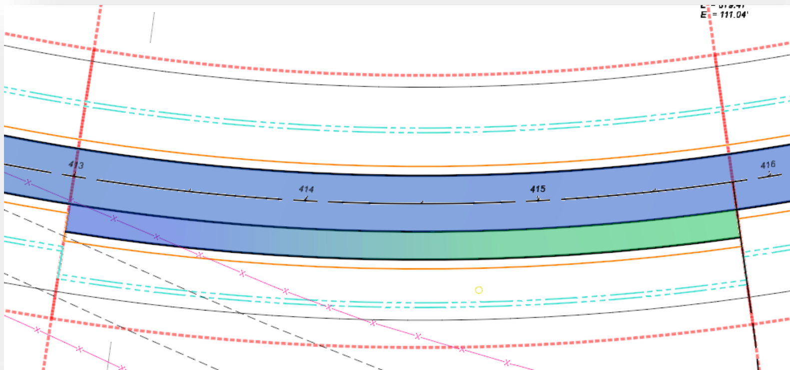
  Right-Click on point **R-INT-Right** and choose **Test Point Controls > Test Horizontal Point Control** to test the behavior of the point. The shoulder and the end condition should both be removed as the point is moved to the right.
- ✓ Save and close the template library

The next step is to create a new template drop for the turn lane template

- ✓ From the **Corridors** tab, select **New Template Drop**
  - o When prompted to **Locate Corridor**, choose one of the handles for the S.R. 185 corridor

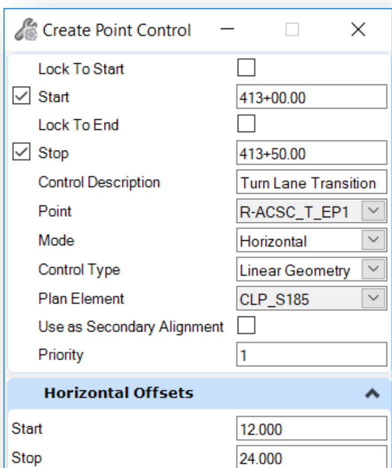


The turn lane is added to the corridor like the example below.

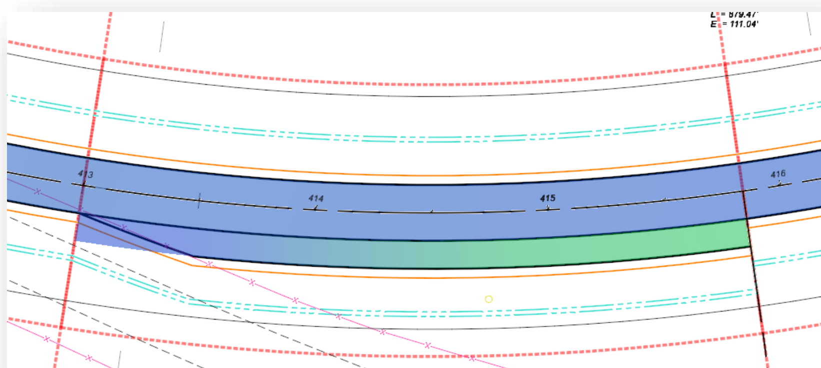


The turn lane will transition from 12' right of the centerline to 24' right of the centerline from station 413+00 to station 413+50. Add a point control to point R-ACSC_T_EP1 to define this transition.

- ✓ From the **Corridors** tab, select **Edits > Create Point Control**
  - ○ When prompted to **Locate Corridor**, choose one of the handles for the S.R. 185 corridor.
  - ○ Create the point control using the values shown below.



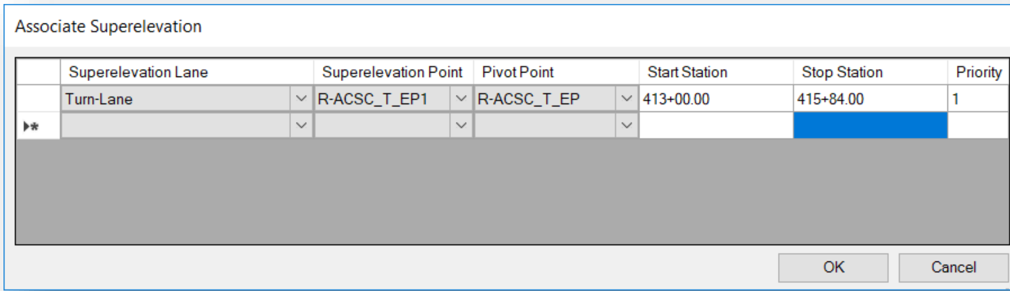
After the point control is defined, the turn lane should transition as shown in the example below.



The next step is to assign the superelevation transition to the new pavement lane.

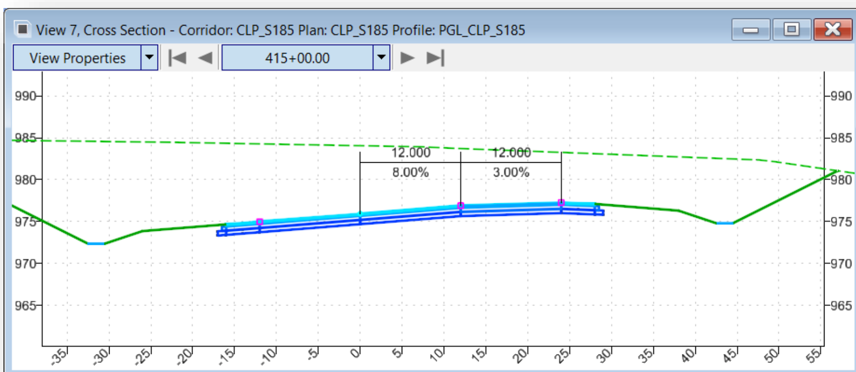
- ✓ From the **Corridors** tab **Superelevation** group, select **Calculate > Assign to Corridor**
  - ○ When prompted to **Locate First Superelevation Section**, choose the boundary element for the superelevation. Right-click to continue.
  - ○ When prompted to **Locate Corridor**, choose one of the handles for the S.R. 185 corridor.

The *Associate Superelevation* dialog is opened. This allows you to assign the superelevation to any of the lanes that have not already been defined. Enter the values shown below.



The superelevation transitions for the turn lane are assigned to the corridor.

The superelevation assignment can be reviewed in a Cross-Section view like the example below.



Take the following steps to open a cross-section view to review the superelevation values:

- ✓ Hold down the right mouse button in empty space in the view window until the pop-up menu appears.
  - Select **View Control > 2 Views Plan/XS**
  - When prompted, select **OK** to create the dynamic view
  - When prompted to **Locate Corridor or Alignment**, choose one of the handles for the S.R. 185 corridor.
  - When prompted to **Select or Open View**, issue a left-mouse click (data point) in the new view window that was opened. The first cross-section for the corridor is displayed in the view window
  - Scroll to station 415+00 (tip: use the Key-In option)
  - Hold down the right-mouse button in the cross-section view until the pop-up menu appears
  - Choose the **Place Horizontal Temporary Dimension** command
  - Follow the prompts to enter data points to define temporary dimension lines for the pavement lanes, like the example above.

This completes the work on S.R. 185.

Part B - Defining the Jameson Road Corridor

The next step is to create a corridor and define the superelevation for Jameson Road.

- ✓ Open the file **\400-Engineering\Roadway\Basemaps\123456_KS003.dgn**. Be sure to check the current file back into ProjectWise.

Section 508 Intersection Modeling

Exercise 1: St. Peter Road Corridor

The St. Peter Road corridor model will be created in its own design file. The design files for the corridor models were created in a previous exercise. Take the following steps to prepare the 12345_KM002.dgn file for the corridor modeling process.

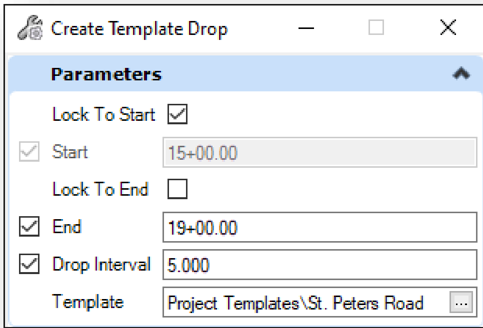
- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **400-Engineering\Roadway\Basemaps** folder.
- Open the file **123456_KM002.dgn**.
- From the **Home** tab, choose **Attach Tools > References** to attach the following references with a **Nest Depth** value of **0**:
    - Survey Field Book:
        - 300-Survey\Basemaps\123456_FB001.dgn
        - Set the survey terrain model as the active terrain model (Use the **Element Selection** tool to select the terrain boundary, allow the cursor to pause over the boundary element until the pop-up menu appears. Choose the **Set As Active Terrain** icon).
    - Geometry for proposed S.R. 185:
        - 400-Engineering\Roadway\Basemaps\123456_BK001.dgn
    - Geometry for Existing St. Peter Road:
        - 400-Engineering\Roadway\Basemaps\123456_BK002.dgn
    - S.R. 185 Corridor Model
        - 400-Engineering\Roadway\Basemaps\123456_KM001.dgn
    - Before closing the ***References*** dialog, turn off the display of the self-referenced 3D model, Design-3D

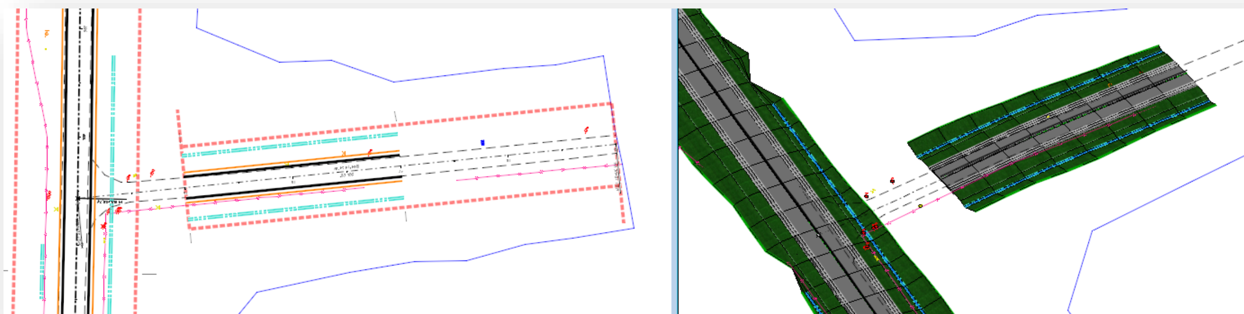
The template for St. Peter Road was created in a previous exercise. Take the following steps to create the corridor model for St. Peter Road

- Use the **Element Selection** tool select the alignment for St. Peter Road. Allow the cursor to rest on the alignment element until the pop-up menu appears. Select the **Create Corridor** icon.

- Create the corridor using the settings shown below.



The corridor is created like the example below.



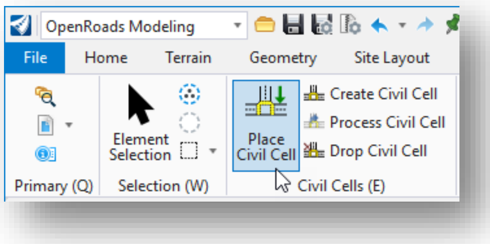
Exercise 2: St. Peter Road Intersection using a Civil Cell

This exercise will demonstrate the use of a civil cell for the intersection of St. Peter Road with S.R. 185.

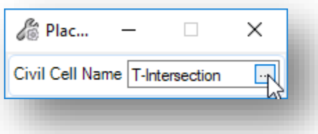
Part 1: Placing the Civil Cell

With the references attached and the existing terrain active, take the following steps to place a civil cell for the St. Peter Road intersection:

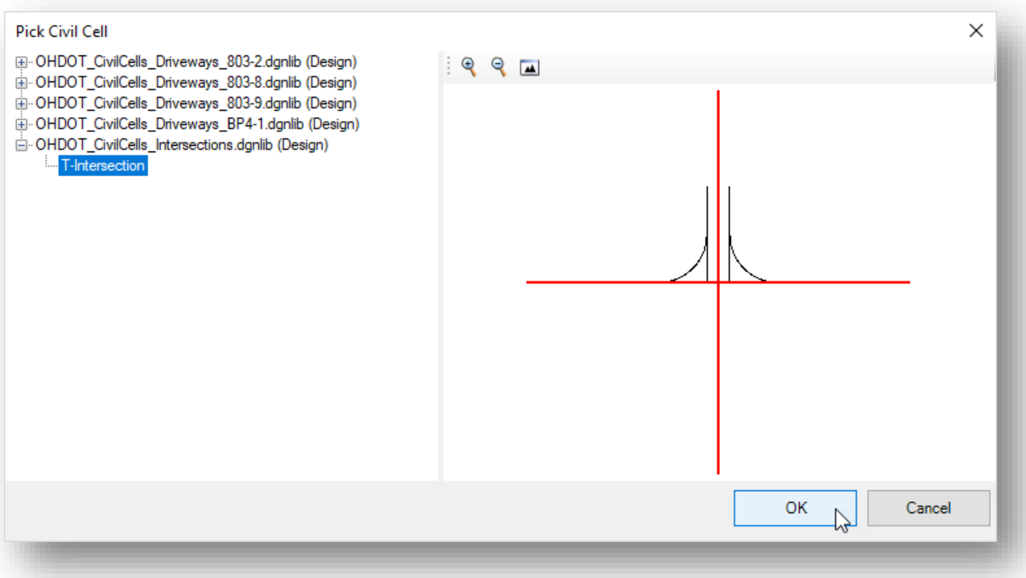
✓ From the **Model Detailing** tab, select the **Place Civil Cell** command



✓ Choose the **Browse** button to select the civil cell.



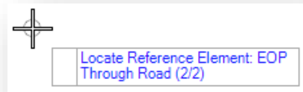
The ***Pick Civil Cell*** dialog is opened



- Choose the **T-Intersection** civil cell shown above and select **OK.**

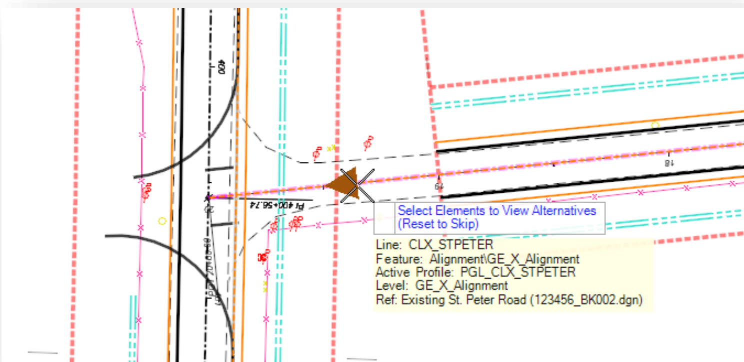
The civil cell requires two elements for placement: The centerline of the intersecting road and the edge of pavement of the through road.

- When prompted, select the element that represents the centerline of the intersecting road (St. Peter Road) and the through Edge of Pavement for S.R. 185.

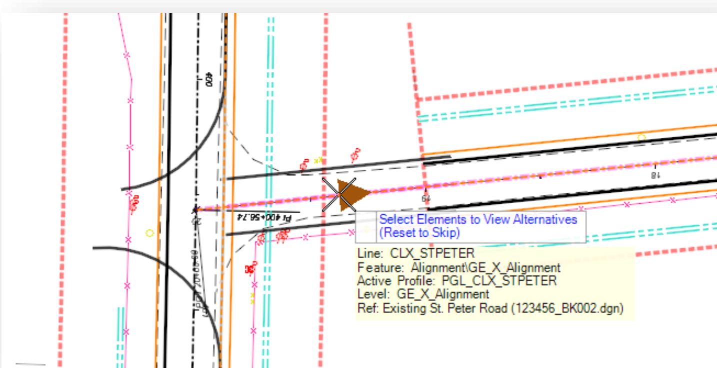


After selecting the two reference elements, the solution is displayed. Depending on the direction of the elements and how the civil cell was created, the solution may or may not appear correctly.

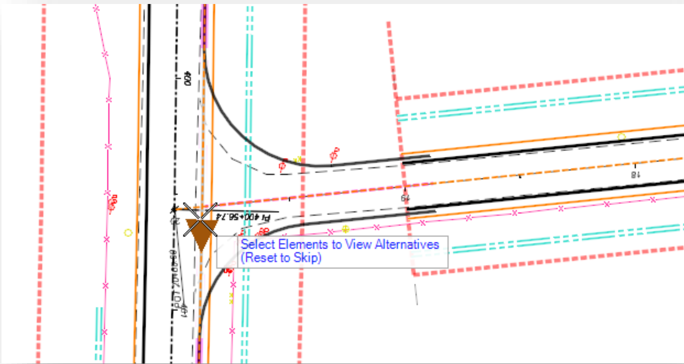
In the example below, the direction of the elements differs from the way the civil cell was created. This can be remedied by clicking on each of the reference elements to view different placement alternatives.



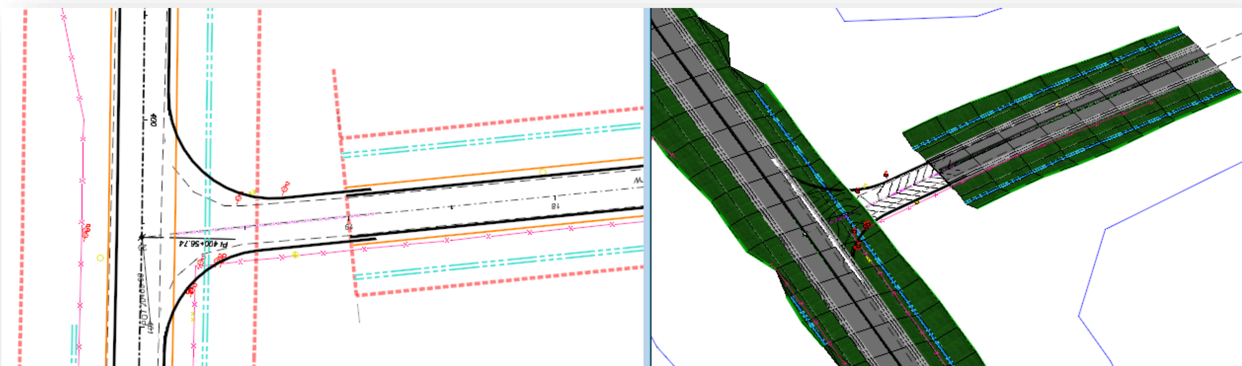
Left clicking on the center line of St. Peter Road will reverse the direction of the element for the civil cell placement as shown below.



The edge of pavement element can also be reversed by left-clicking the edge of pavement line.

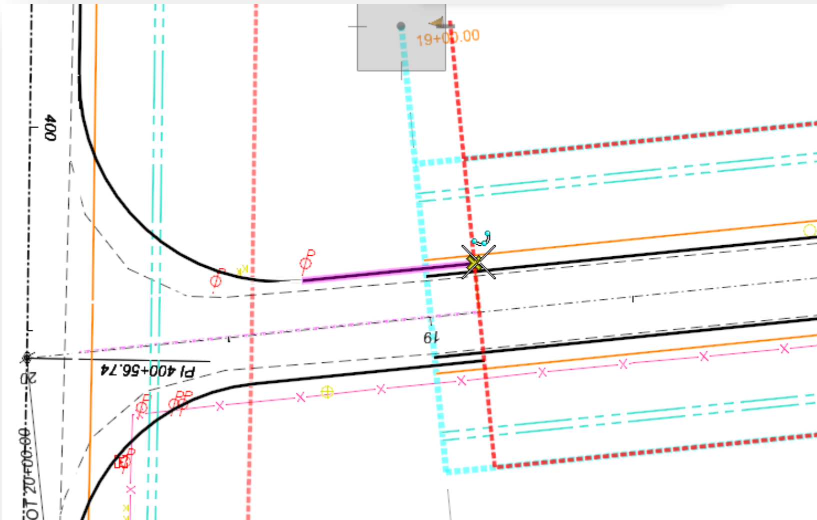


- ✓ When the solution appears correct, issue a reset (right-click) to skip the process of viewing alternatives.
- ✓ Issue a data-point (left click) to accept the civil cell placement. 2D and 3D graphics are placed to represent the surface of the intersection.

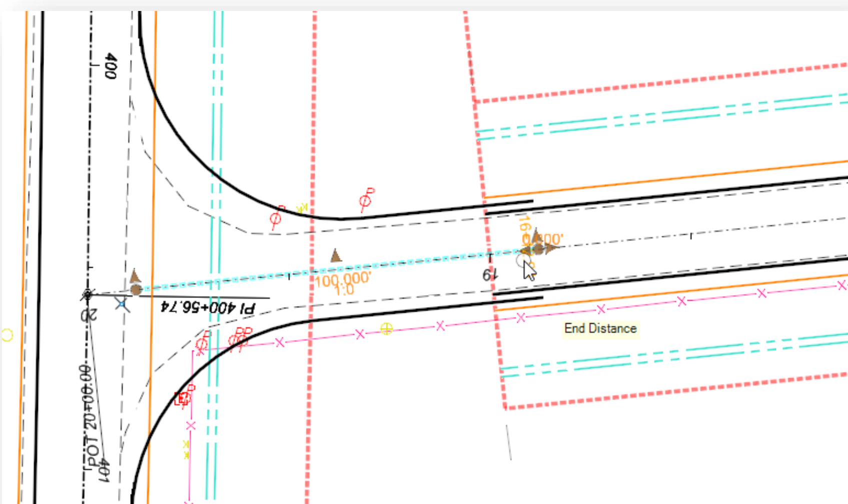


The length of the civil cell overlaps the St. Peter Road corridor. This can be corrected either by editing the civil cell length, or by adjusting the ending location of the corridor.

- ✓ To adjust the corridor, select the template range graphic, and then select the arrow handle to move the ending location of the corridor. The location can be specified by snapping to the end of the civil cell as shown below.

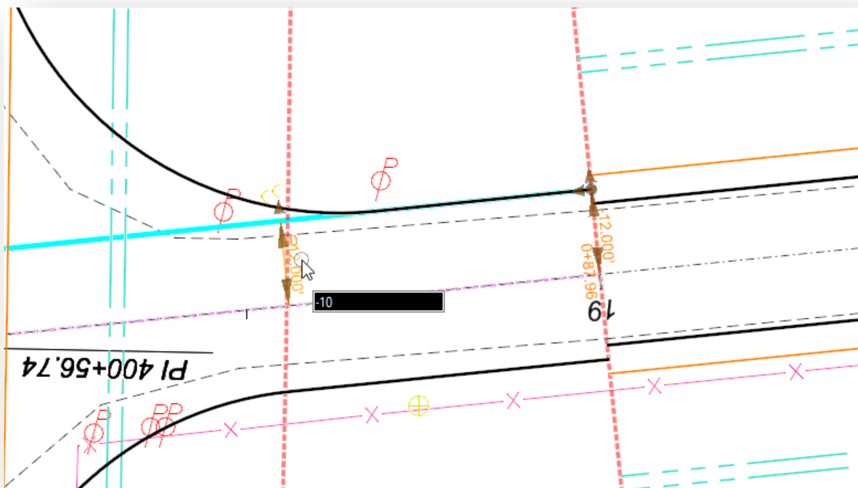


- ✓ To adjust the civil cell, select the civil cell element that overlays the St. Peter Road alignment. The length of the civil cell is a rule that can be edited (the value is 100.000’ in the example below), or the length can be dynamically modified by choosing the left arrow at the end of the civil cell and dragging the length to the desired location.



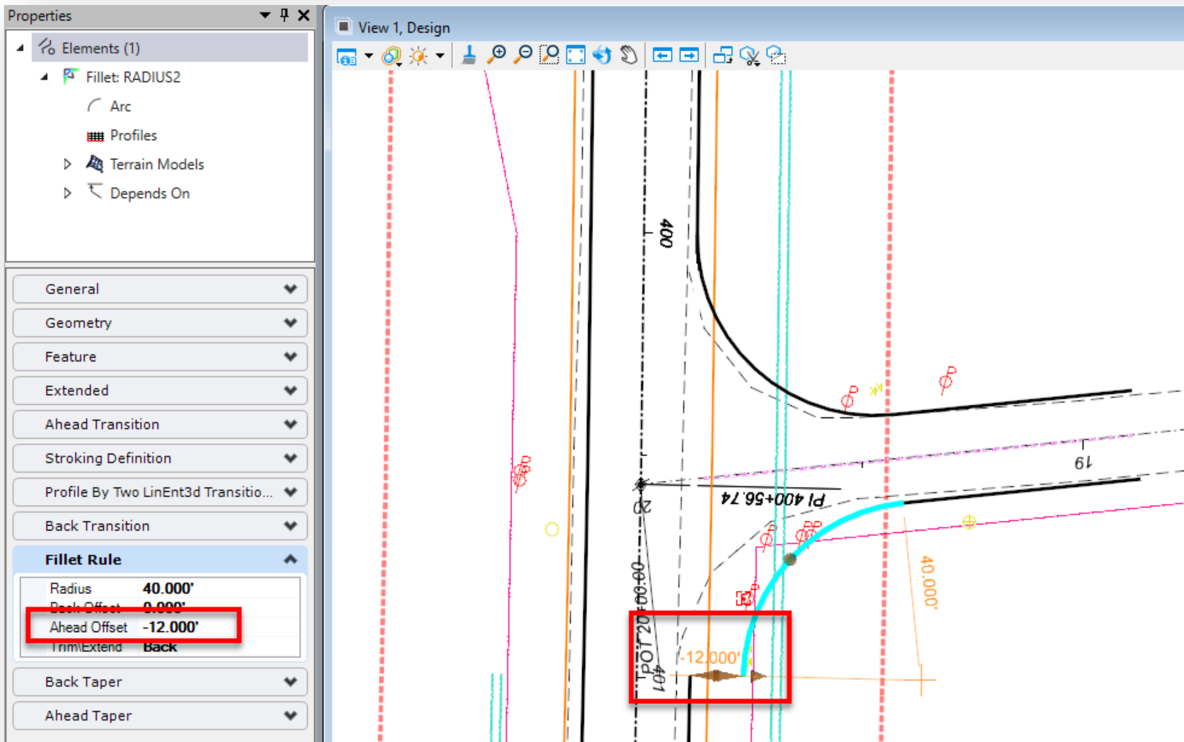
The civil cell can be edited to further meet specific project requirements. Take the following steps to edit the lane widths for the civil cell:

- ✓ Use the **Element Selection** command to select the edge of pavement on the north side of St. Peter Road
  - The underlying rules for the element are displayed but are grey cannot be edited until the underlying element is selected.
  - Right-click the element until the rules are editable
  - Select the center lane width ruled dimension. Change the value to -10
  - The edge of pavement is adjusted to the new value
  - Repeat this process to change the lane width for the opposite side to a value of 10



- ✓ The same type of editing can be applied to the radius returns.
  - Edit one of the radii returns to use a radius value of 40’
    - Observe the behavior of the civil cell elements. The interval displayed for the edge of pavement is automatically adjusted according to the new radius value and the surface terrain model for the civil cell is also adjusted in response to the radius change
  - Change the edited radius value back to 50’

- ✓ Additionally, the radius returns have start and ending offset values for the tie-in with the edge of pavement lines. This is useful for meeting differing pavement widths on each side of the intersection.
  - Select the radius return element on the south of St. Peter Road. The Properties dialog can be used to edit various parameters. From the **Fillet Rule** group, change the **Ahead Offset** value to **-12**. The fillet is adjusted as shown below.



✓ Change the **Ahead Offset** parameter back to **0**

Part 2: Defining the Pavement Buildup

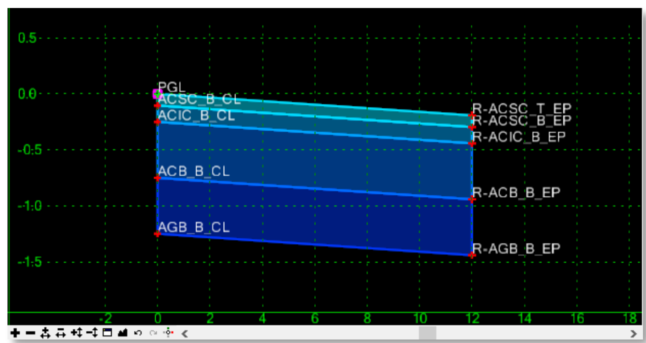
The pavement surface for the intersection is placed as a terrain model. The feature definition assigned to the terrain model is **P_Pavement_Surface** which is configured to display contour lines for the pavement surface at 0.1' intervals.

To complete the St. Peter Road pavement, a template will be applied to the surface terrain. Take the following steps to apply a template with the pavement buildup for St. Peter Road:

✓ From the **Corridors** tab, select **Template > Create Template**

- Copy the St. Peter Road Template.
- Paste the copy into the **Project Templates** folder.
- Rename the copied template to St. Peter Road Pavement.
- Double-click the St. Peter Road Pavement template to make it active.
- Right-click in the **Current Template** window and choose the **Delete Components** command. Delete the shoulders and the side slopes on both sides of the template.
- Delete the pavement layers for the left lane, leaving the pavement layers for the right lane in place.
- Delete the Null points above the pavement layers (right-click on each point and choose the Delete Point command).
- Close and save the template library.

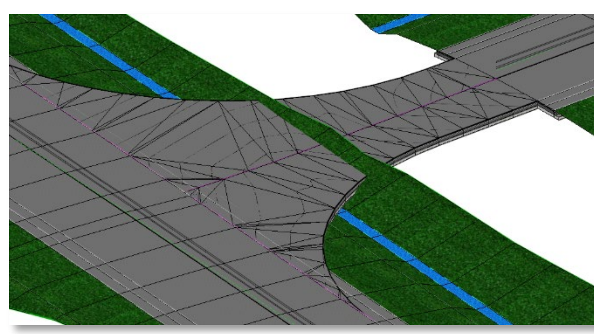
The completed St. Peter Road Pavement template contains only the pavement as shown below. The pavement slope and width of the components is irrelevant as the terrain model for the intersection will define the slope and width.



Take the following steps to assign the pavement template to the intersection terrain model.

✓ From the **Model Detailing** tab, select **Apply Surface Template**

- When prompted to **Locate a Terrain Model**, select the surface terrain model for the intersection.
- Select **No** for the **Apply Clipping Boundary** option.
- Choose the St. Peter Road Pavement template.
- Left click (data point) to accept the selection.
- Data point to accept the selection. The pavement template is applied to the terrain model as shown below.



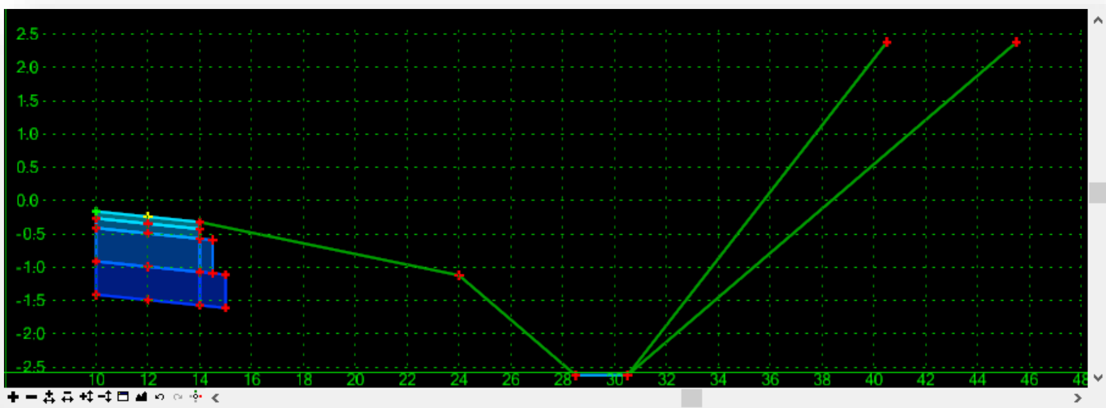
Part 3: St. Peter Road Shoulder Templates

A new template is necessary to define the pavement, shoulders, and side slopes for the St. Peter Road intersection area.

Take the following steps to create the radius return template:

✓ From the **Corridors** tab, select **Template > Create Template**
  - Copy the **St. Peter Road** template
  - Paste the copy into the **Project Templates** folder
  - Rename the copied template to **St. Peter Road Shoulder**
  - Double-click the **St. Peter Road Shoulder** template to make it active
  - Right-click in the **Current Template** window and choose the **Delete Components** command. Delete the pavement, the left shoulder, and end conditions on the left side. Leave the right shoulder and the right end conditions in place.
  - Delete the null points above the pavement layers (right-click on each point and choose the **Delete Point** command)

At this point, the template should appear like the example below.



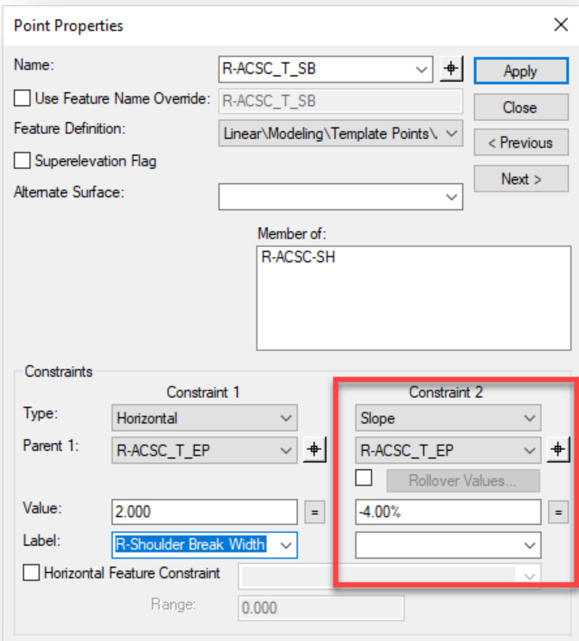
The shoulder components are currently located 10' right of the origin. The next step is to move the shoulder to the 0,0 origin in the template dialog.

✓ Open the ***Dynamic Settings*** dialog to ensure the **Step** values are defined

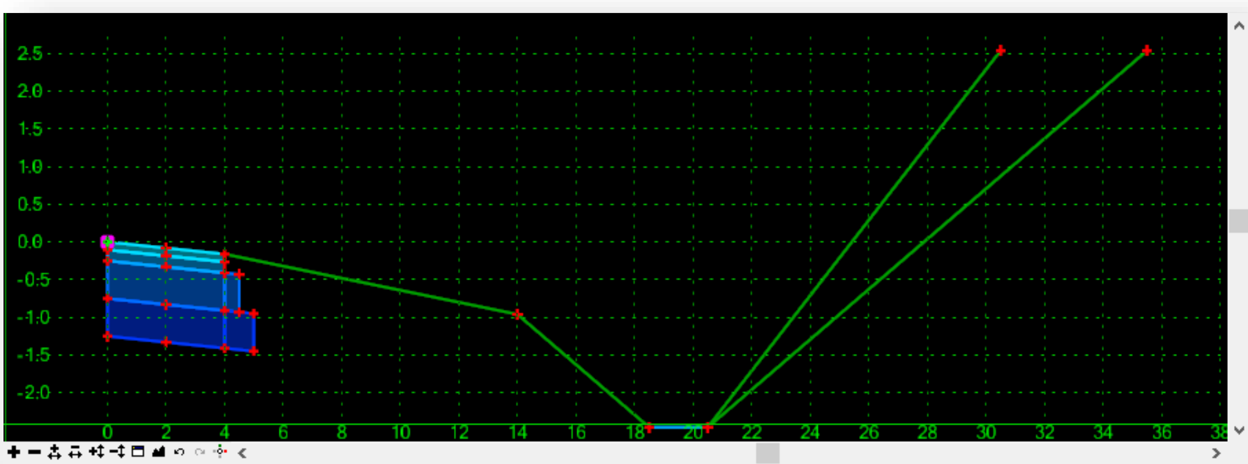
✓ Right-click on point **R-ACSC_T_EP** and choose the **Move Point** command. Move the point to the (0,0) position

Notice the point at the shoulder break is partially constrained (the point location is shown with a yellow plus sign indicating that only one constraint is defined). When the pavement lane was deleted, the rule for the slope was removed since it referred to point PGL as part of the rollover lock definition. Take the following steps to define a Slope constraint for point R-ACSC_T_SB:

✓ Double-click point **R-ACSC_T_SB** to access the *Point Properties* dialog. Add a **Slope** constraint to the point with a -4.00% slope from point **R-ACSC_T_EP** as shown below.



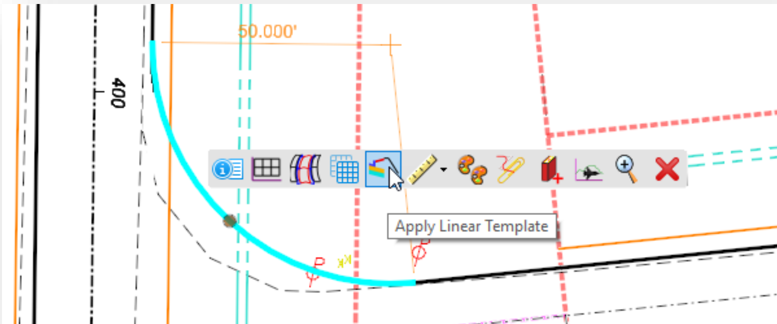
The completed template for the radius returns should appear as shown below



✓ Save and close the Template library.

Take the following steps to apply the St. Peter Road Shoulder template to the edge of pavement tangent lines and the radius return elements:

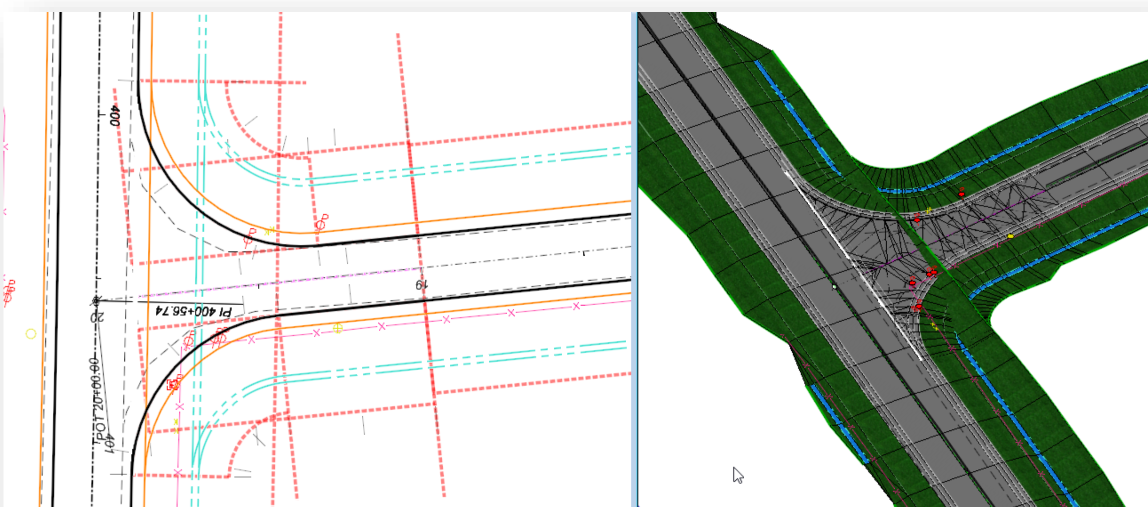
- ✓ Select the right radius return graphic and choose the **Apply Linear Template** command from the pop-up menu. Alternately, the **Apply Linear Template** command can be selected from the **Model Detailing** tab.



- ✓ Choose the **St. Peter Road Shoulder** template from the library
  - The template can be applied for a portion of the selected graphic, or the entire element. Define the **Start Station** and **End Station** to encompass the entire length of the element by using the <ALT> key
  - When applying a linear template, the template can be placed on either the left or right side of the selected element. When prompted to **Select Side – Reflect Option**, issue a data point on the side of the element that you wish to apply the template
  - Use the default **Exterior Corner Sweep Angle** value of 5 degrees

  The template is applied to the radius return. Repeat the process for the remaining edge of pavement lines and radius return.

When complete, the intersection should appear like the example below.



Part 4: Adjusting the S.R. 185 Corridor in the Intersection

The last step is to remove the left shoulder and end conditions on the S.R. 185 corridor through the intersection area.

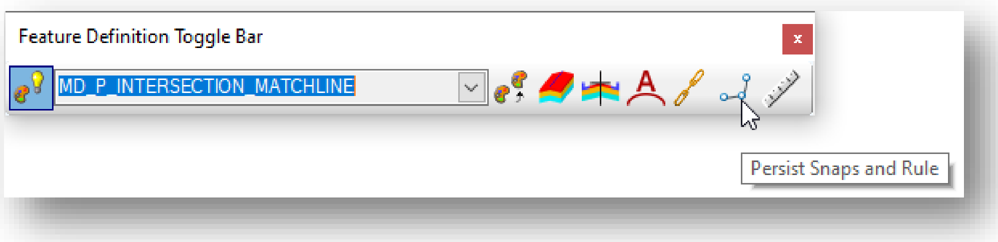
The template for S.R. 185 is defined with Null Points to seek an intersection with an element in the design that defines the limits of the intersection. If the intersection graphic element is found, a display rule is used to turn off the side slopes.

Take the following steps to draw a line in the design that will be used as the intersection element the null point will seek:

- ✓ Open the design file containing the corridor model for S.R. 185
  400-Engineering\Roadway\123456_KM001.dgn

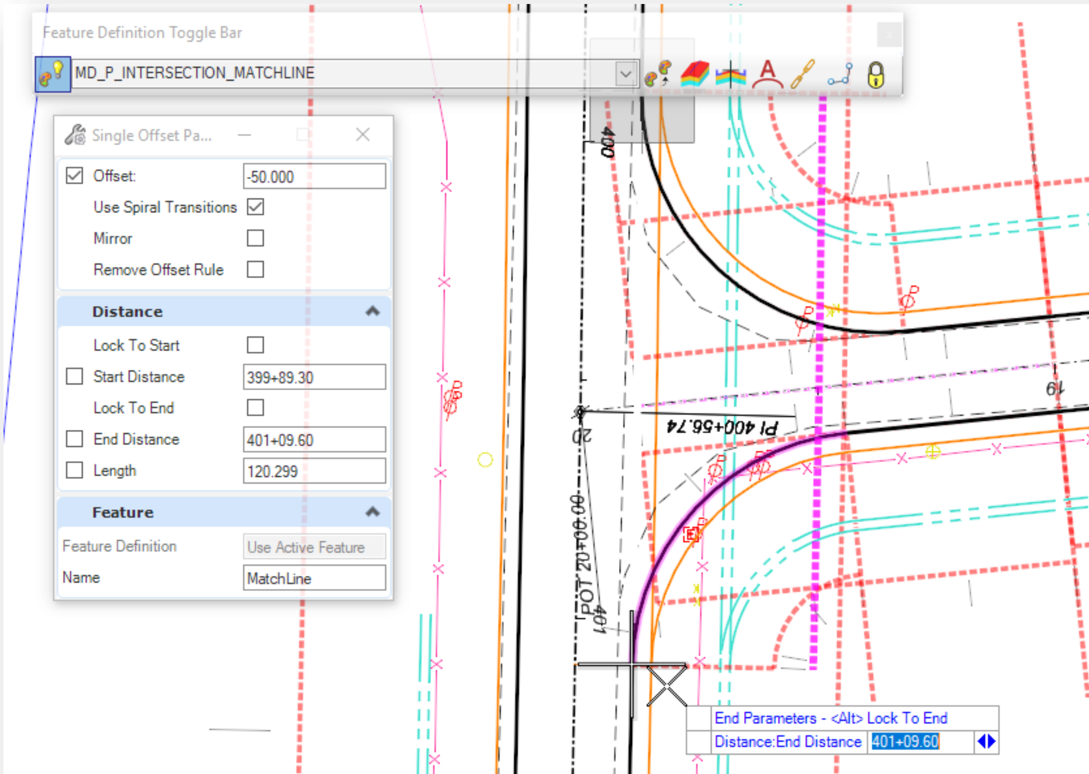
- ✓ Attach the St. Peter Road corridor as a reference
  - From the **Home** tab, select **Attach Tools > References** to attach the following file: 400-Engineering\Roadway\123456_KM002.dgn

- ✓ From the **Geometry** tab, choose **Standards > Feature Definition Toggle Bar**
  - Set the **Feature Definition** to **Linear\Modeling > Template Points > Misc > MD_P_INTERSECTION_MATCHLINE**
  - Turn on the Use **Active Feature Definition** option
  - Turn off the **Persist Snaps and Rule** option as shown below



- ✓ From the **Geometry** tab, **Horizontal** group, select **Offsets and Tapers > Single Offset Partial**
  - When prompted to **Locate Element**, choose the S.R. 185 alignment
  - Name the element **St. Peter Match Line**
  - Create an offset line 50' to the left of S.R. 185 by snapping to the radius returns of St. Peter Road

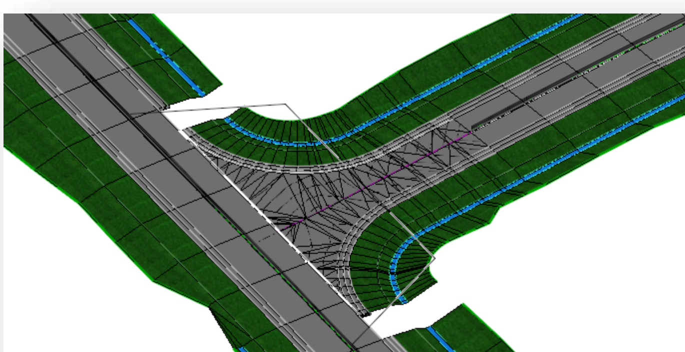
The completed line should be like the example shown below.



The match line element must be added as a corridor reference to the S.R. 185 corridor for the template to find the graphic. Take the following steps to add the match line element as a corridor reference:

✓ From the **Corridors** tab, select **Corridor References > Add Corridor Reference**
  - When prompted to **Locate Corridor**, choose the S.R. 185 corridor
  - When prompted to **Locate First Reference**, choose the match line element
  - Reset (right-click)

The element is added as a corridor reference and the corridor is reprocessed. The corridor model should appear like the example below.



Depending on the template drop interval and the location of the radius returns, there may be a gap in the model as shown above. This can be remedied by inserting a key station on the S.R. 185 corridor to ensure that the model is processed up to the station where the radius return starts.

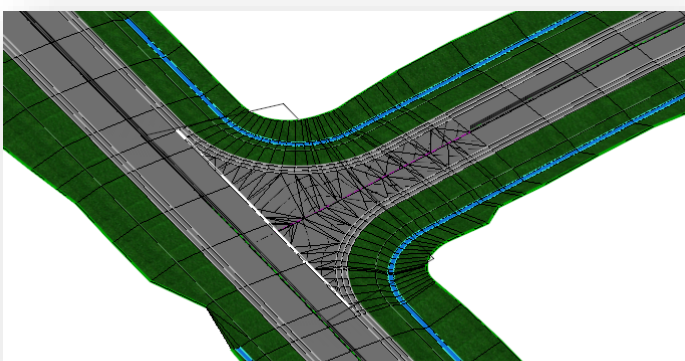
Take the following steps to insert a key station on the S.R. 185 corridor:

✓ From the **Corridors** tab, select **Edits > Create Key Station**
- Make sure the **Persist Snaps and Rule** option is toggled off in the Feature Definition toggle bar when adding key station locations to a corridor
- When prompted to **Locate Corridor**, choose the S.R. 185 corridor
- Identify the station by snapping to the radius return at the point where it meets the S.R. 185 edge of pavement but do not accept this location
- This function works best if you use a station value that is slightly outside of the range of the radius return. For example, if the station value is 399+89.30 at the snap point of the first radius return, key in a value of 399+89.29 that is just before the start of the radius return. For the second radius return, key in a station value that is just beyond the ending station value.

  Note: You can adjust the key-point location after it has been placed by selecting the line that is drawn at the key point location and then edit the station rule.

The key station graphic is drawn on the level **CM_Corridor_Key_Station**

There may be a small gap at the south end of the intersection as well. Insert another key station to close this gap.



This completes this exercise.

✓ Exit OpenRoads Designer and check the file back in to ProjectWise.

Optional Exercise 3: Intersection Modeling Step-by-Step

This section will step through the process to model the intersection of S.R. 185 and St. Peter Road using the OpenRoads tools without the use of a civil cell. This exercise is used in place of Exercise 2 Part 1 above.

Process Overview

The process to model the intersection will take several steps as summarized below:

- Define the Horizontal and Vertical geometry for each road
- Create Templates for each road
- Model the mainline corridor
- Model the side road corridor
- Add radius returns to the side road and define the vertical profile for each of the radius returns
- Modify the mainline corridor to remove any side slopes through the limits of the intersection
- Adjust the side road corridor to end at the radius return
- Create a surface terrain for the pavement surface of the side road in the intersection area. The surface terrain is defined by the radius returns, the edge of pavement for the mainline corridor, and the centerline and profile of the side road corridor.
- Apply a “surface template” to the surface terrain of the intersection. This will establish the pavement buildup components beneath the pavement surface terrain.

Apply linear templates to the radius returns to model the shoulders and side slopes for the intersecting side road.

Modeling the Intersection Pavement

The corridor for St. Peter Road was created up to station 19+25 in this example. Ending at this station will allow room for modeling the intersection with S.R. 185. The intersection will be modeled by first defining a terrain model representing the pavement surface for the intersection. Start by placing the edge of pavement lines and radius returns for the intersection.

The edge of pavement lines and the radius returns must be created using Civil Geometry tools. Take the following steps to create the edge of pavement lines:

✓ From the **Geometry** tab, open the *Features Definition Toggle Bar* by choosing **Standards > Feature Definition Toolbar**. Set the active feature definition to **Linear > Pavement > PV_P_Edge_of_Pavement**.

✓ In the *Features Definition Toggle Bar*, toggle on the **Use Active Feature Definition** option

✓ From the **Geometry** tab, choose **Offsets and Tapers > Single Offset Partial**. This command is used to place a civil geometry element at an offset from the selected alignment.

  - When prompted to **Locate Element**, choose the St. Peter Road alignment.
  - Enter a **Name** for the element a shown below, **R-EOP St Peter Road**.
  - Using an **Offset** of 10’, define the starting point of the offset element by snapping to the edge of pavement line at the end of the corridor as shown below.
  - Enter the **End Distance** by entering the <ALT> key to lock to the end of the alignment.
  - When prompted to **Mirror** the solution, choose **Yes**.
  - The mirrored line is named R-EOP St Peter Road1. Change the name to **L-EOP St Peter Road** by selecting the line element and choosing the **Properties** command from the pop-up menu.

Next, assign a profile to the newly created edge of pavement elements.

✓ From the **Geometry** tab, choose **Element Profiles > Profile By Slope From Element**

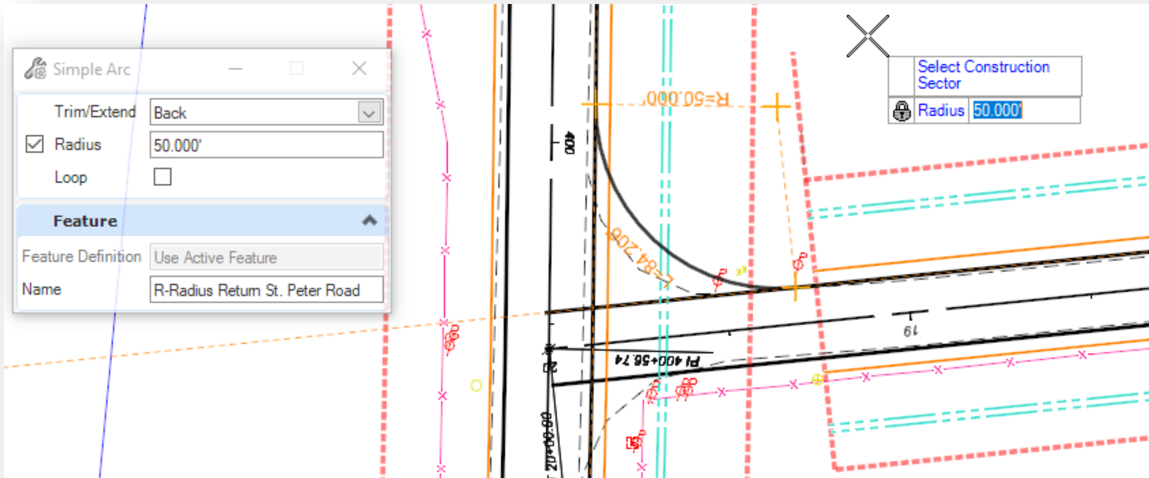
- When prompted to **Locate First Element to Profile**, choose one of the edges of pavement elements. Continue by choosing the other element when prompted to **Locate Next Element to Profile**. After choosing both elements, right click (reset) to end the selection process.
- Choose the centerline of St. Peter Road when prompted to **Locate Reference Element**.
- Enter a **Slope** value of -1.6%
- Choose the **Point Selection** mode **All**
- Use a **Profile Adjustment** of **None**
- Enter a **Vertical Offset** of **0**

Next, create the radius returns for the intersection.

✓ From the **Geometry** tab, select **Arcs > Arc Between Elements > Simple Arc**.

Create the first arc on the right side of St. Peter Road using a 50’ radius.

- Be sure to enter a **Name** of **R-Radius St Peter Road** for the radius return element.
- Choose the **R-EOP St Peter Road** line as the first element.
- When prompted to **Trim/Extend**, choose the **Back** option.

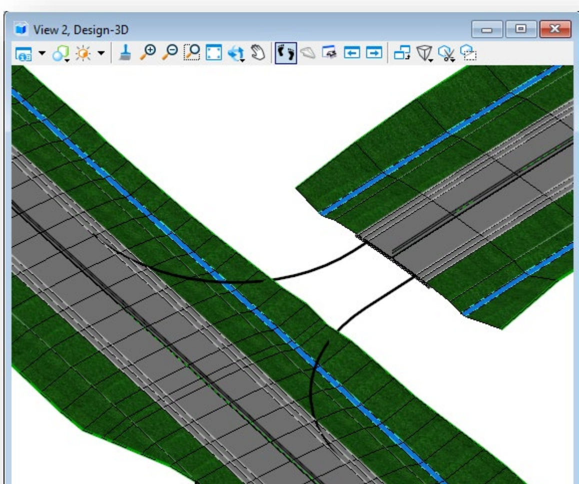


✓ Create the second arc on the left side of St. Peter Road with a **Radius** of 50', a **Name** of **L-Radius St Peter Road**, and use the **Trim/Extend Back** option.

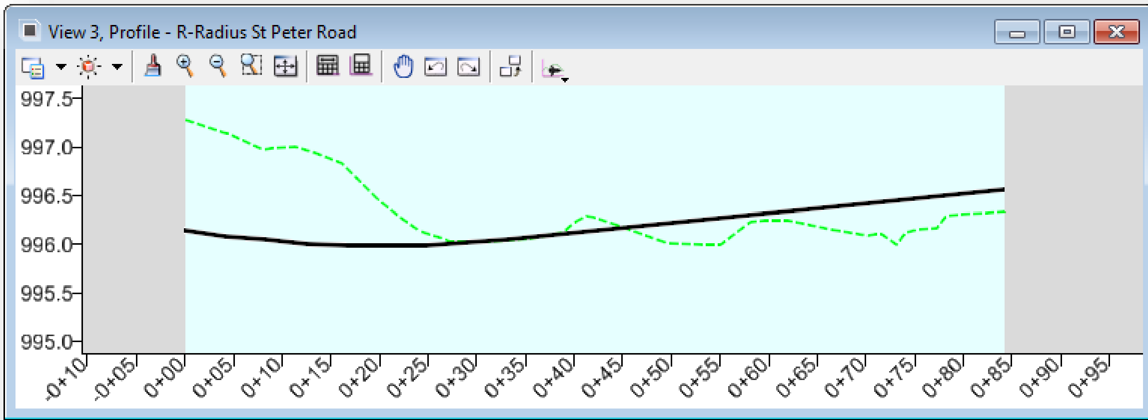
Next, assign a profile to the radius return elements using the **Quick Profile Transition** command.

✓ From the **Geometry** tab, select **Element Profiles > Quick Profile Transition**
  - After selecting the command, you are prompted to choose the **Quick Transition Method**. Select the **Parabolic** option.
  - When prompted to **Locate What to Define**, select one of the radii return elements. A quick profile is created for the element.
  - Repeat the procedure for the second radius return

After the profiles are assigned, the edge of pavement and radius return lines should appear in the 3D model like the example shown below.



The profile for each radius return can be reviewed and edited by selecting one of the radii returns and selecting the **Open Profile Model** command. The profile for **R-Radius St Peter Road** is shown below.



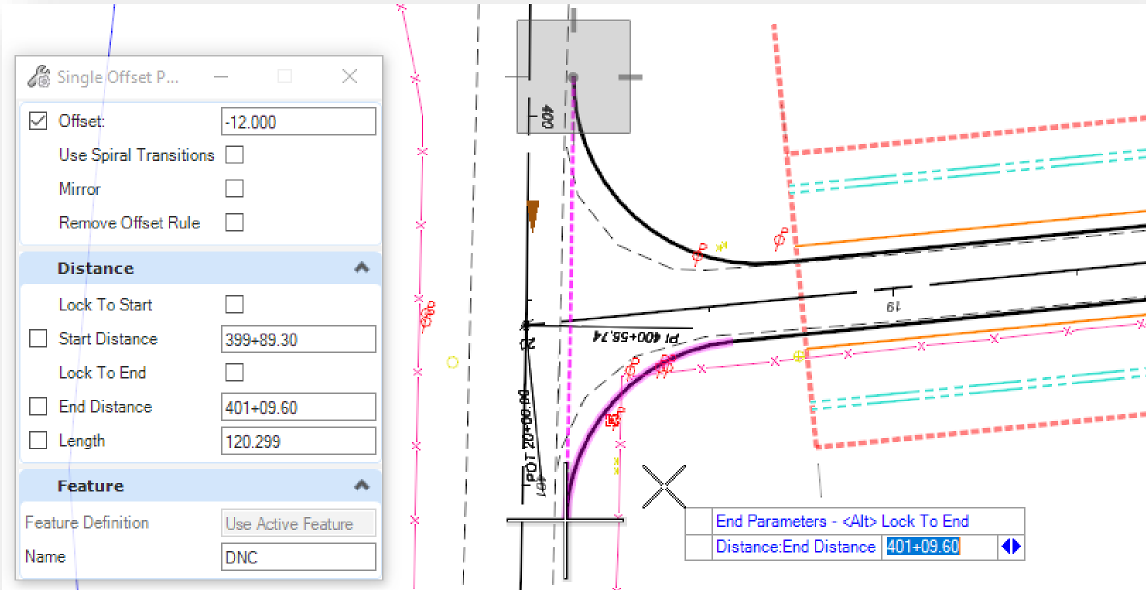
In addition to the two radius returns, it is necessary to define a line that follows the edge of pavement for S.R. 185 that extends from the ends of the radius returns. This is necessary to define the boundary of the intersection area that will be used to make the surface terrain model. We cannot use the edge of pavement line from the CLP_S185 corridor model as the line must be constrained to the limits of the radius returns.

Since this new edge of pavement line will overlap the edge of pavement lines already drawn for the CLP_S185 corridor we will use a different feature definition.

- ✓ In the *Features Definition Toggle Bar*, set the active feature definition to **Modeling > Template Points > Misc > MD_P_DNC**. The initials "DNC" stand for **Do Not Construct**. This feature definition can be used for linear elements used in the modeling process whose graphics will not be used for part of the plan production process. Graphics drawn with this feature are placed as construction elements on the level MD_P_DNC.

- ✓ From the **Geometry** tab, choose **Offsets and Tapers > Single Offset Partial**.
  - Create an offset line 12' to the left of the CLP_S185 alignment from the ends of the radius returns.
  - Because this new line is drawn on top of the edge of pavement line for S.R. 185, it is easier to temporarily turn off the reference file display of the S.R. 185 corridor model to ensure the line was drawn correctly. Open the *References* dialog and turn off the display of the 12345_KM001.dgn file.

The completed graphic should appear like the example below.

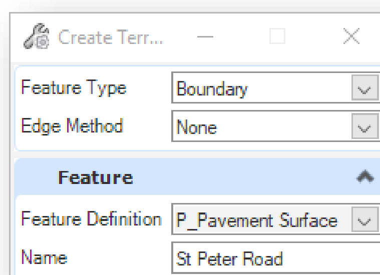


Next, assign a profile to this new DNC line.

✓ From **Geometry** tab, choose **Element Profiles > Profile By Slope From Element**.
- When prompted to **Locate First Element to Profile**, choose the DNC line
- Reset (right-click) to complete the location process
- When prompted to Locate Reference Element, choose the S.R. 185 alignment
- Define the profile with a -1.6% slope from the centerline of S.R. 185

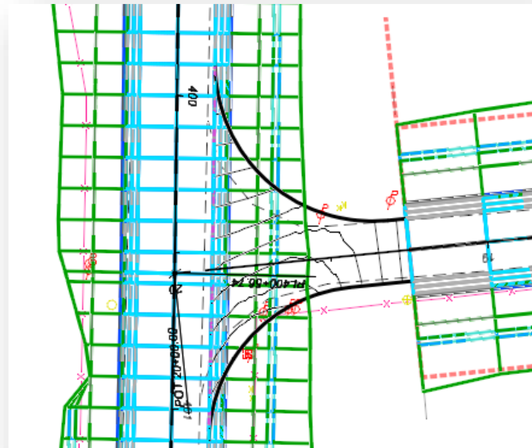
The edge of pavement and radius return lines will be used to define the boundary of a terrain model for the surface of the pavement in the intersection.

✓ From the **Terrain** tab, choose the **From Elements** command.



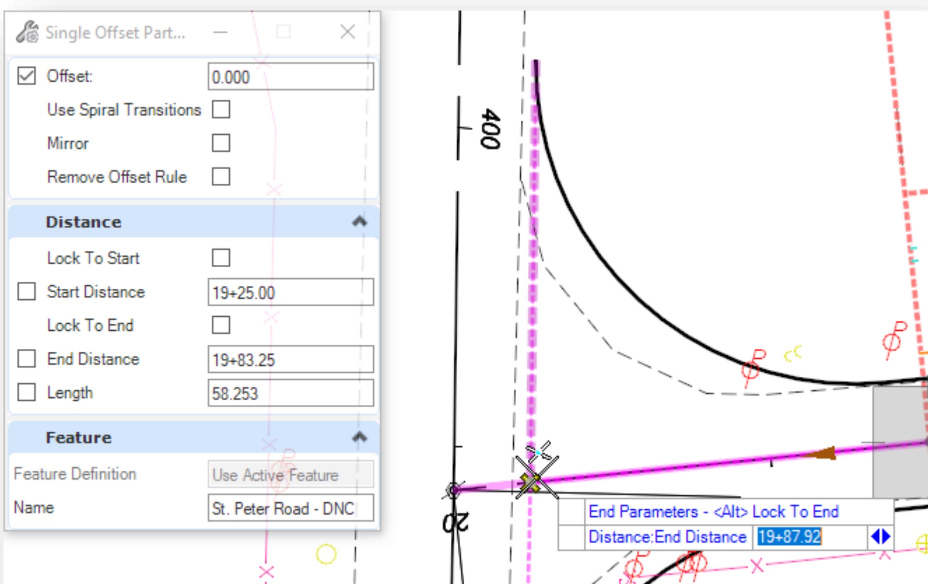
- Set the **Feature Type** to **Boundary**
- Set the **Edge Method** to **None**
- Set the **Feature Definition** to **Terrain > Proposed > P_Pavement Surface**. Since this is a terrain feature it cannot be selected from the *Feature Definition* tool bar.
- Set the feature **Name** to **St Peter Road**
- When prompted to **Locate Elements to Add**, choose the edge of pavement and radius returns for St. Peter Road as well as the DNC for S.R. 185 (five elements total – don't forget the edge of pavement lines for St. Peter Road between the radius returns and the corridor).
- Reset when the elements are selected.

The surface is created, and contour lines are drawn as shown on the right.



One more graphic is necessary to complete the pavement surface definition. The center line and profile of St. Peter Road will be added to define the crown of the pavement surface. This will be accomplished by creating one more partial offset element.

- ✓ In the *Features Definition Toggle Bar*, set the active feature definition to **Modeling > Template Points > Misc > MD_P_DNC**.
- ✓ From the **Geometry** tab, choose **Offsets and Tapers > Single Offset Partial** command.
- ✓ Choose the centerline of St. Peter Road when prompted to **Locate Element**.
    - Start the line at the end of the St. Peter Road corridor, **Station** 19+25 with an **Offset** value of 0.
    - Use the MicroStation **Intersection Snap** to end the line at the intersection of St. Peter Road with the S.R. 185 Edge of Pavement.



After placing the line, assign a profile to the line using the **Profile By Slope From Element** command.

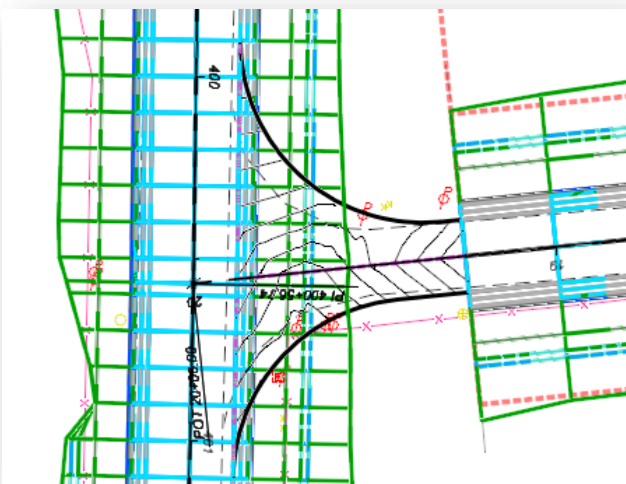
- ✓ From the Geometry tab, select Element Profile > Profile Slope from Element
- ✓ The **Reference Element** is the centerline of St. Peter Road.
- ✓ Since the new element overlaps the centerline of St. Peter Road, use a **Slope** value of **0.0%.**

The 3d line is created as shown below.

Add this new partial offset line to the surface terrain model as a break line.

✓ From the **Terrain** tab, select **Feature Management > Add Features**.
- When prompted to **Locate Terrain Model to Add Elements**, select the St. Peter Road pavement surface terrain model.
- When prompted to Locate Element to Add, select the DNC line at the center of St. Peter Road.
- Reset (right-click)
- Set the **Feature Type** to **Break Line**

After the break line element is added, the St Peter Road terrain model should appear like the example below.



The process to define the pavement buildup, add the shoulder and side slopes, and trim the S.R. 185 corridor model through the limits of the intersection are detailed in Parts 2, 3, & 4 of Exercise 2.

This completes this exercise.

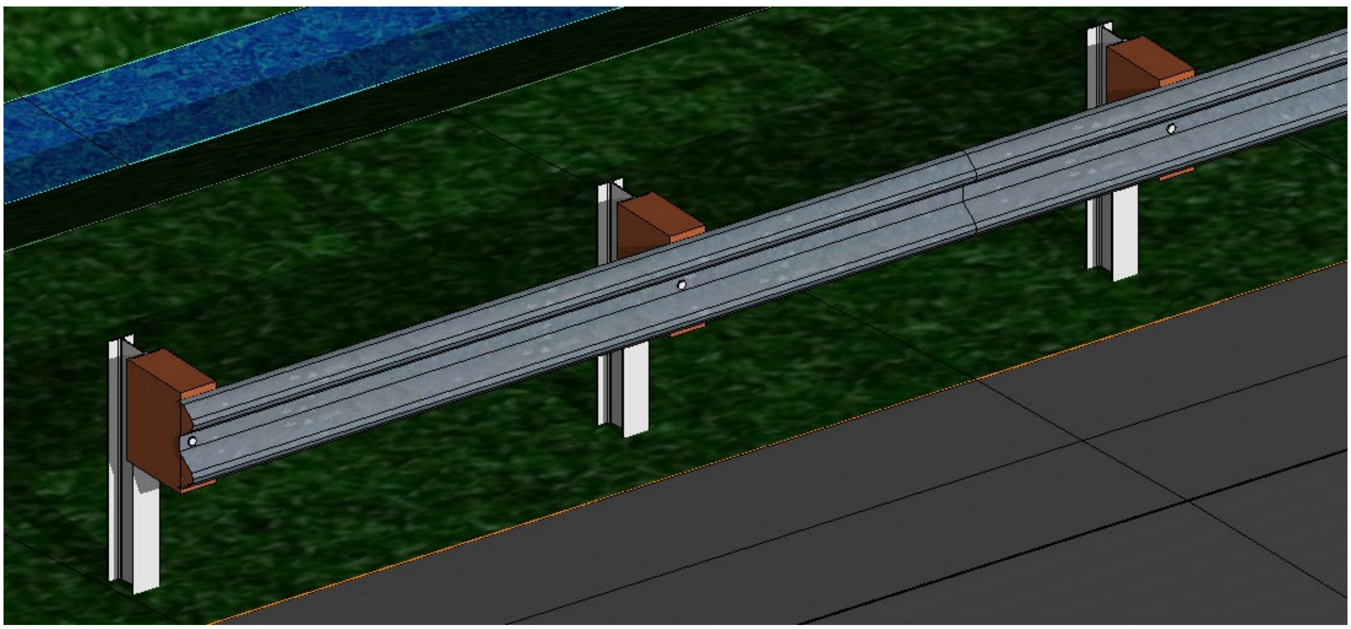
✓ Exit OpenRoads Designer and check the file back in to ProjectWise.

Section 509 3D Guardrail

Process Overview

ODOT's CADD Standards for OpenRoads Designer includes feature definitions, element templates, and cross section templates for the placement of 3D guardrail lines. The 3D guardrail graphics are intended for visualization purposes and should not be evaluated critically for strict compliance with design standards.

Note, the guardrail placement examples shown in this document are intended to demonstrate the process and do not necessarily represent proper guardrail design.



Placement of 3D guardrail runs requires the following software components:

- 3D linestrings are used to draw the posts.
  - The line strings for legacy guardrail types are in the file **OHDOT_3DGuardrail.rsc**.
  - New linestyles for the Midwest Guardrail Systems guardrail specifications are included in the file **OHDOT_MGS_Guardrail_3D.rsc**
- The rail, and the cables for a cable barrier, are drawn using OpenRoads templates.

The 3D guardrail is placed by first drawing the guardrail run in a 2D plan view using an OpenRoads feature. After a profile is defined for the 2D guardrail run, a 3D guardrail linestyle is added to the 3D model. The 3D linestring is made up of the guardrail posts and does not contain the rail faces.

The following feature definitions in **ODOT_Features_Design.dgnlib** are used to draw 3D MGS guardrail runs:

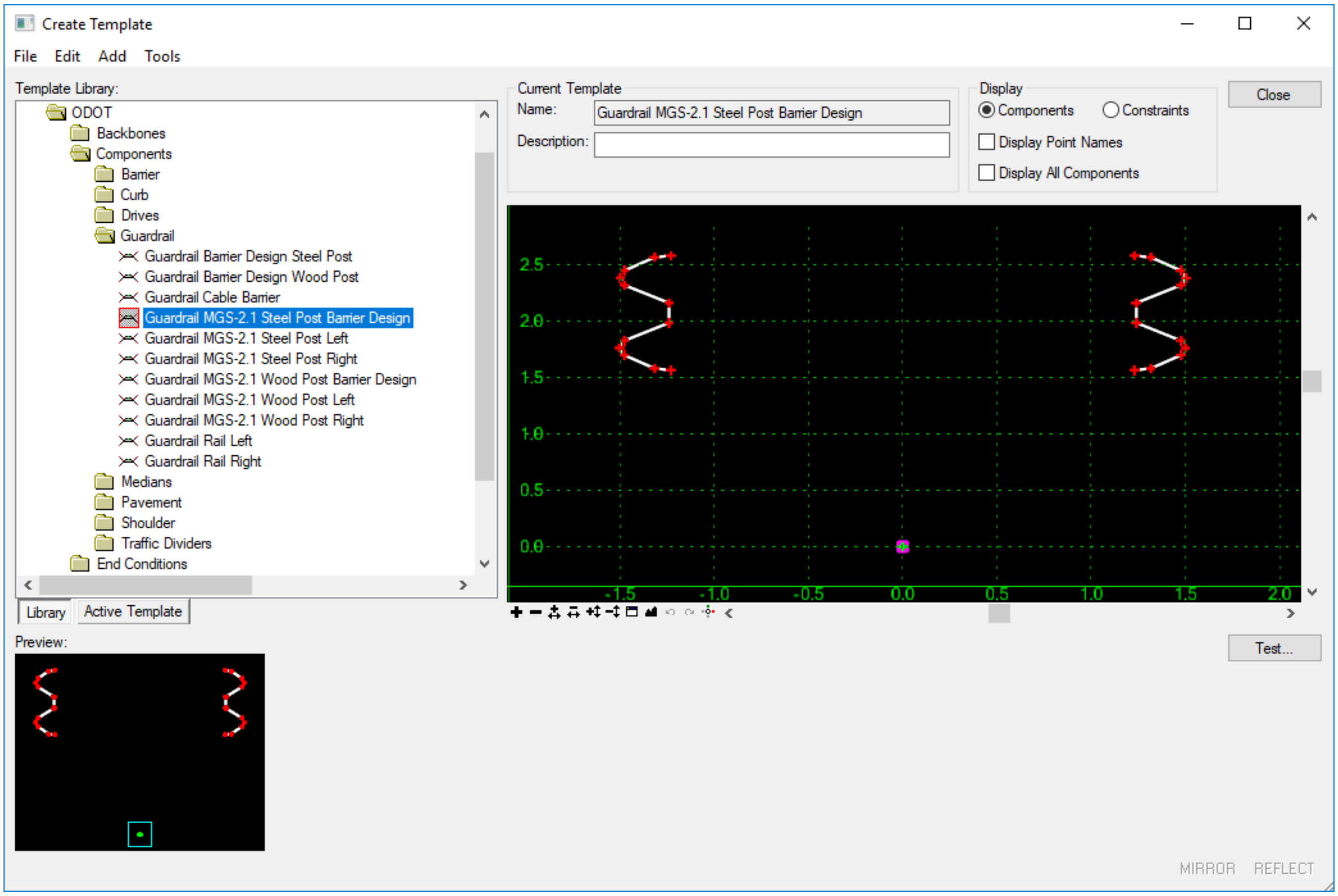
- Linear > Guardrail & Barriers > MGS-2.1 > RD_P_Guardrail MGS-2.1 Steel Post Barrier Design
- Linear > Guardrail & Barriers > MGS-2.1 > RD_P_Guardrail MGS-2.1 Steel Post Left
- Linear > Guardrail & Barriers > MGS-2.1 > RD_P_Guardrail MGS-2.1 Steel Post Right

- Linear > Guardrail & Barriers > MGS-2.1 > RD_P_Guardrail MGS-2.1 Wood Post Barrier Design
- Linear > Guardrail & Barriers > MGS-2.1 > RD_P_Guardrail MGS-2.1 Wood Post Left
- Linear > Guardrail & Barriers > MGS-2.1 > RD_P_Guardrail MGS-2.1 Wood Post Right

The rail itself, or the cables for cable barrier runs, are placed using a corridor template and the **Apply Linear Template** command.

Required Templates

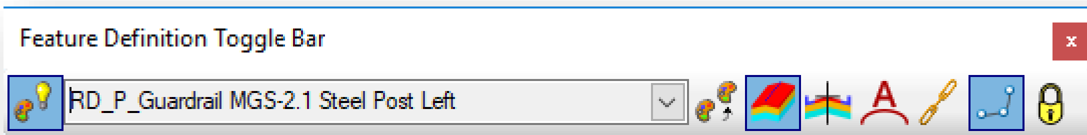
OHDOT_Templates.itl contains the templates shown below to draw the actual rail in the 3D model. The corridor templates must be present in the template library you are using for the project. These templates can be copied from **OHDOT_Templates.itl** by use of the **Template Library Organizer** function, if necessary.



Placing 3D Guardrail Elements

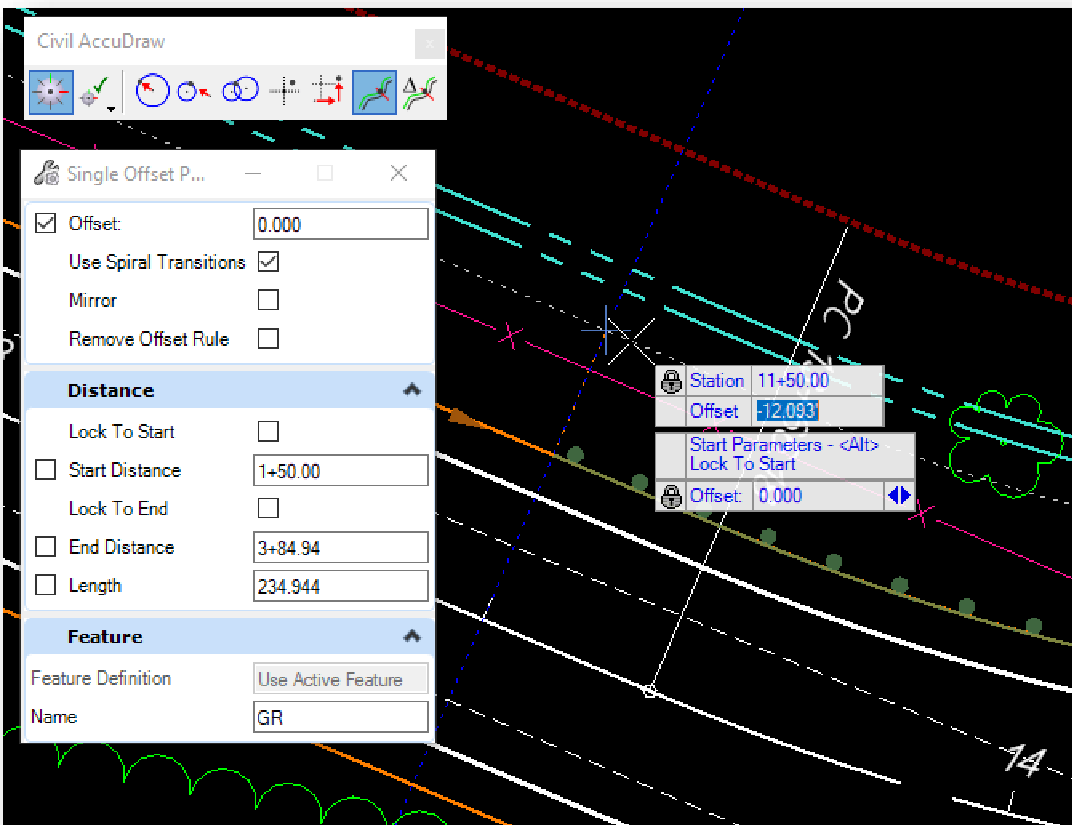
Take the following steps to place the 2D guardrail lines:

- Use the OpenRoads ***Feature Definition Toggle Bar*** to select the desired feature.
- Toggle on the **Use Active Feature Definition** and **Create 3D Automatically** icon as shown below.



- Use the OpenRoads **Geometry > Single Offset Partial** tool to place the 2D guardrail element.

  In the example below, the guardrail line is placed adjacent to the edge of shoulder. When prompted to select an element, the edge of shoulder line was selected. The **Offset** value is set to **0**.



Since the edge of shoulder element does not have stationing assigned, the centerline will be used as the reference element to define the station value. **Civil AccuDraw** is used to facilitate this task.

Take the following steps to use **Civil AccuDraw** to define the guardrail station limits:

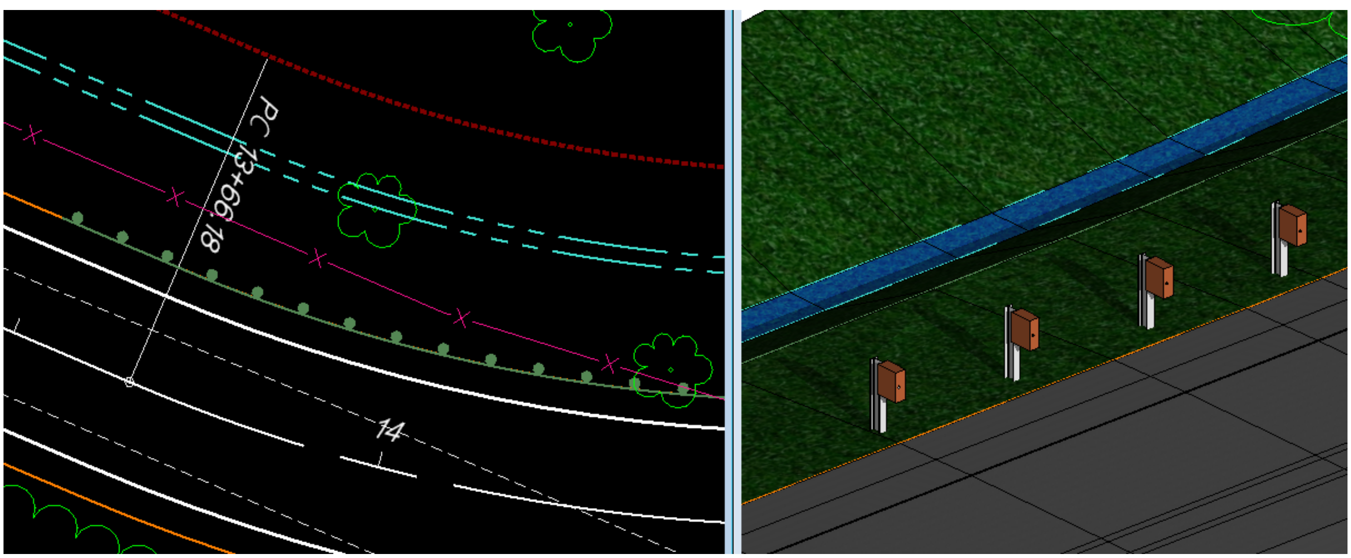
- Toggle on **Civil AccuDraw**
- Tab to the **Offset** field - there are two offset fields shown in the screen capture on the previous page: one for the Single Offset Partial command and one for Civil Accudraw. Be sure to toggle to the offset field for Civil Accudraw, which is the one below the Station value.
- With keyboard focus set to the **Offset** field, type in the letter "**o**"
- Civil AccuDraw prompts you to select the reference element or issue a reset to return to the active command. Select the centerline as the reference element.

After selecting the centerline element, the **Station** value should reflect the cursor location relative to the selected element. In the example on the preceding page, a **Station** value of **11+50** was keyed-in. The second **Offset** value, which is part of the Civil AccuDraw function, is irrelevant as the **Offset** value for the **Single Offset Partial** command is the value used to define the offset value, relative to the shoulder line, for the new element.

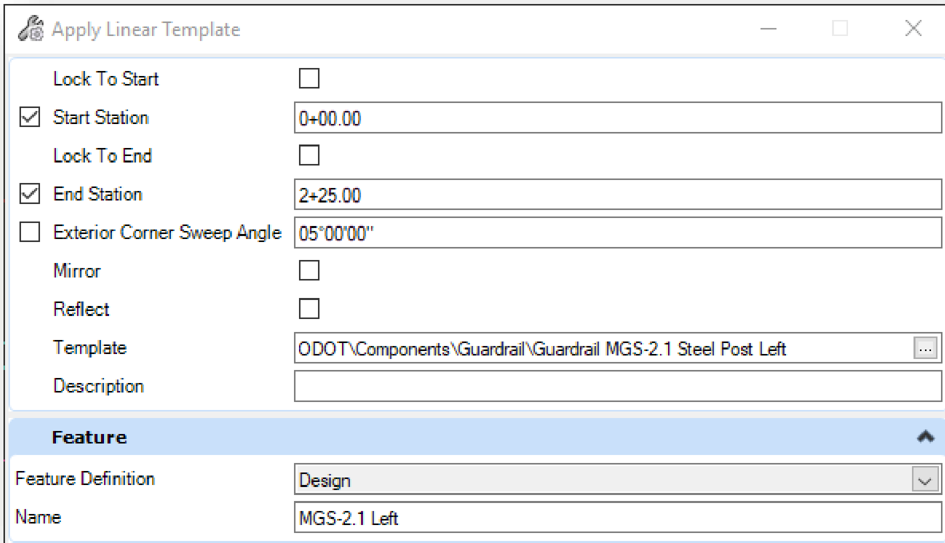
After accepting the starting location, the ending **Station** value or a **Length** can be keyed in. We recommend using the **Length** function rather than an ending **Station** value. This is due to the post spacing of the 3D guardrail line. Use the following guidelines:

- Guardrail runs should be drawn in 6.25' increments for the visualization of the posts to appear correct at the end of the run.
- Cable Barrier runs should be drawn in 10.5' increments for the visualization of the posts to appear correct at the end of the run.

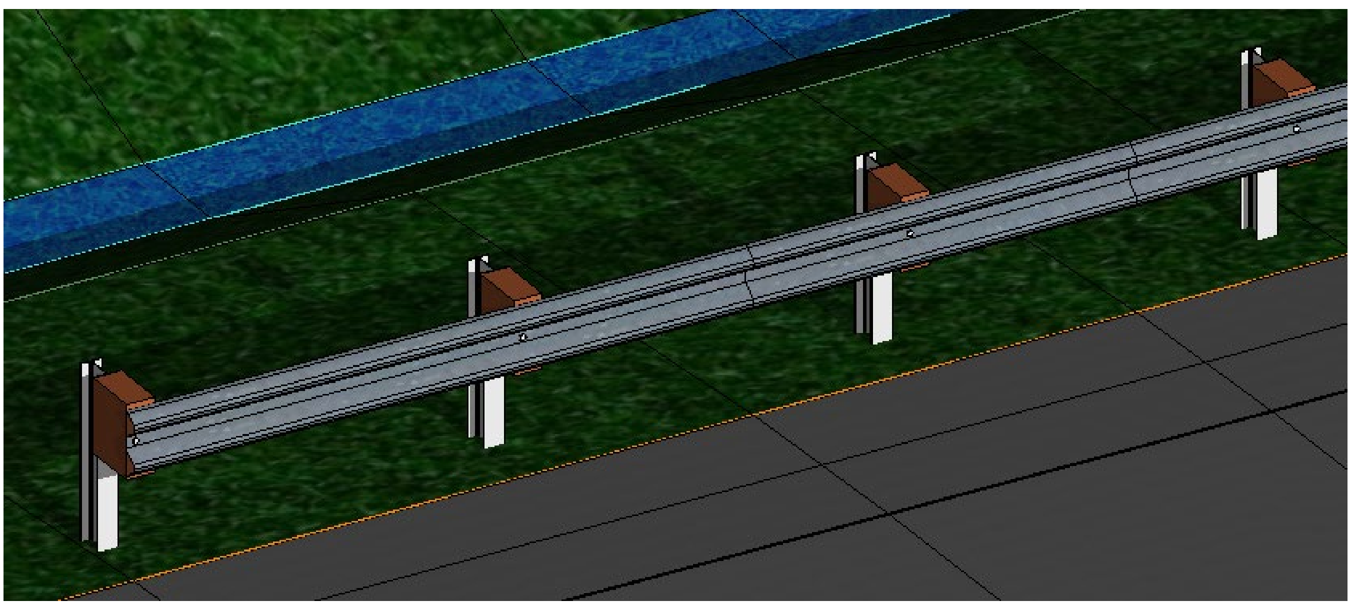
After the guardrail is placed, graphics for both the 2D and 3D view are drawn like the example below.



The rail is placed using the **Model Detailing > Apply Linear Template** command to assign the template to the guardrail run in the 2D view.



When complete, the guardrail run should appear like the example shown below.

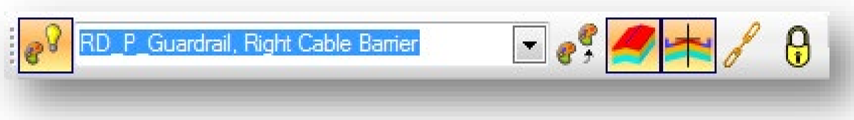


Placing a Cable Barrier Element

Placing a cable barrier run is like the process of placing a guardrail run. The cable barrier is typically placed offset from the edge of shoulder or from the ditch line. See the **Location and Design Manual** for cable barrier placement information. The **Single Offset Partial** command is used to place the cable barrier; however, the user will have to adjust the profile since the element is offset from the shoulder, or from a ditch line.

The following steps detail the process to place the cable barrier at an offset from the edge of shoulder.

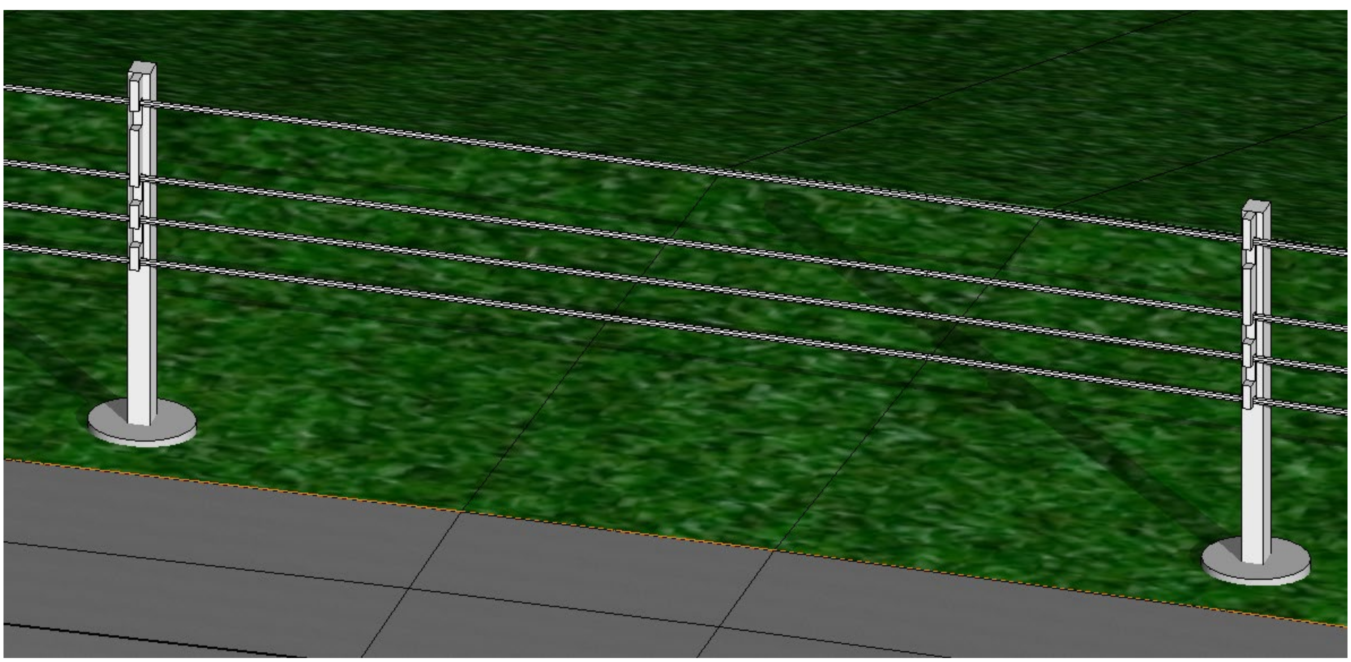
- From the *Feature Definition Toggle Bar*, select the desired feature definition and toggle on the **Use Active Feature Definition**, **Create 3D Automatically**, and **Use Feature Definition Template** icons as shown below



- Use the **Horizontal Geometry > Single Offset Partial** command to place the barrier, like the methodology described for placing guardrail on the preceding pages.

- After placing the cable barrier line, use the **Apply Linear Template** command to add the cables to the design file.

The cable barrier run is placed in the file like the example shown below.



In the example above, the cable barrier was placed at an offset of 1.0’ from the edge of shoulder.

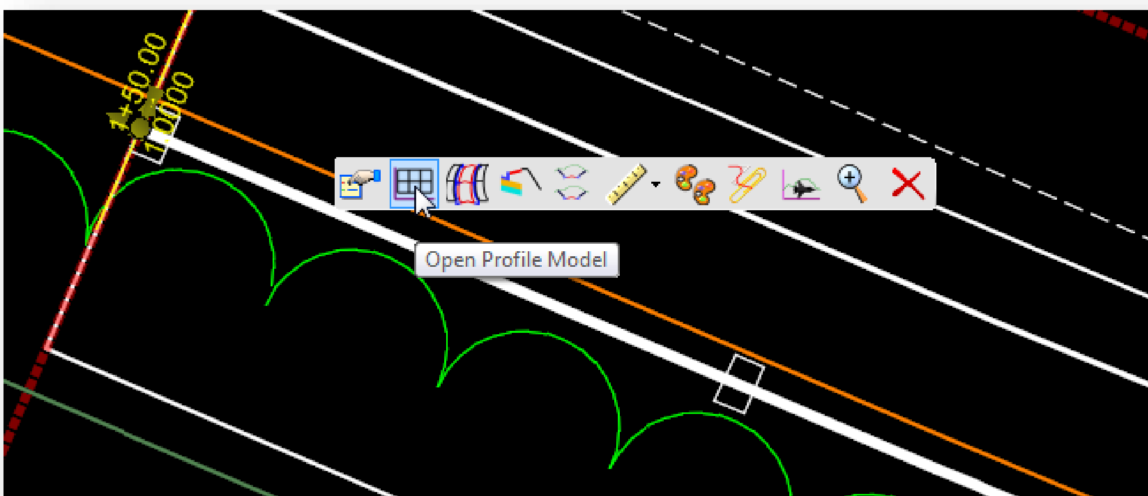
NOTE: There are different cable barrier types and attachment methods. The barrier type shown here is intended for visualization purposes and not intended to relay to the construction contractor that they must use a specific cable barrier type.

The **Single Offset Partial** command uses the same profile for the cable barrier as the selected element that it was offset from. The profile for the cable barrier can be adjusted using the **Profile By Slope From Element** command.

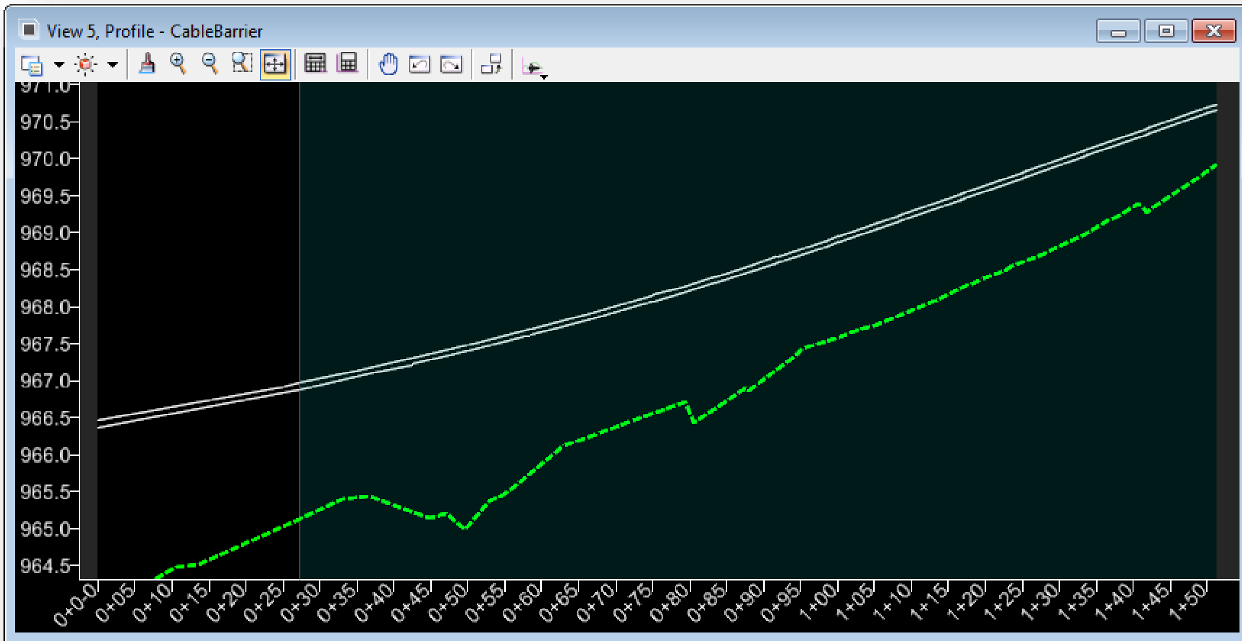
- Select the **Vertical Geometry > Profile By Slope From Element** command
- When prompted to **Locate First Element To Profile**, select the cable barrier line in the 2D view
- After selecting the cable barrier line, issue a reset button to move to the next step.
- When prompted to **Locate Reference Element**, select the element that was initially used to place the cable barrier (the edge of shoulder in this example)
- Enter the **Slope** for the graded shoulder, or the side slope, depending on location of the cable barrier line and the element that was used as the initial offset. In this example, a slope of **-8.0%** is used since the cable barrier line is on the graded shoulder which was defined as a -8.0% slope in the roadway template.

After defining the rest of the parameters, a new profile is created for the cable barrier line, but it is not set as the active profile. Take the following steps to set the new profile as the active profile:

- Select the 2D element representing the cable barrier line. From the pop-up menu, select the **Open Profile Model** command.

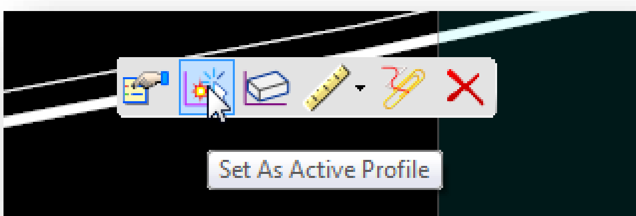


- When prompted, open a new view window, and issue a data point in the view to display the profiles for the cable barrier element. An example is shown below.



- Notice that there are two proposed profile lines. The lower line is the new profile, which was defined at a -8.0% slope from the shoulder.

- Select the lower profile line and choose the **Set As Active Profile** command.



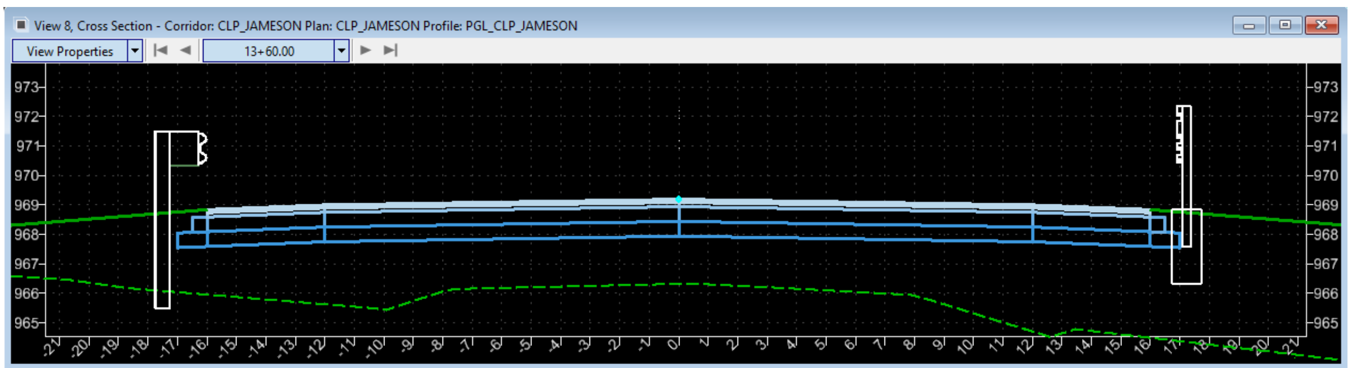
When the command is selected, the cable barrier is automatically adjusted to use the new profile.

Cross Sections

The display of the guardrail in the cross sections is automated by the feature definitions. A cell is placed in the cross section for each guardrail or cable barrier type. The following cells, some of which were added in the January 2017 CADD Standards release, are used for the display of the guardrail in the cross sections:

- GR_CABLE_LT_XS
- GR_CABLE_RT_XS
- GRBAR_MGS_STEEL_XS
- GRBAR_MGS_WOOD_XS
- GRBAR_STEEL_XS
- GRBAR_WOOD_XS
- GRLT_MGS_STEEL_XS
- GRLT_MGS_WOOD_XS
- GRLT_X_XS
- GRRT_MGS_STEEL_XS
- GRRT_MGS_WOOD_XS
- GRRT_X_XS

An example of the cells displayed in the cross section is shown below.



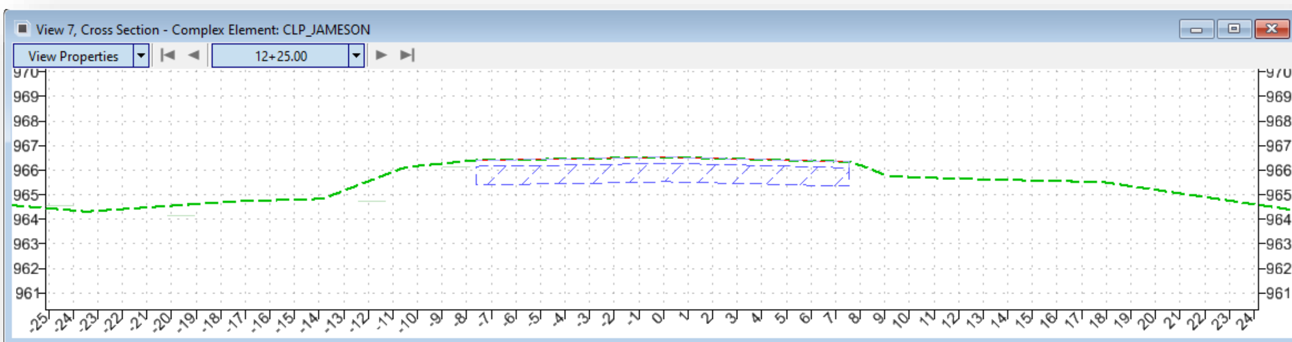
The cells should be automatically displayed. If the cells do not display, check the cell library **OHDOT_Symbols.cel** to ensure that you have the latest version of the cell library containing the cells listed above.

Section 510 Modeling Existing Features

Overview

Existing features such as the existing pavement, sidewalk, and driveways, can be modeled by creating a surface terrain model from the survey elements and then applying a surface template to the terrain.

This document will demonstrate the process to generate a terrain model for the surface of the existing pavement for Jameson Road. This surface terrain model will be created using the surveyed edge of pavement, center of pavement, and break line elements. After the surface terrain has been defined, a template with the existing pavement buildup will be applied to the surface terrain to create the 3D mesh elements representing the existing pavement buildup. Once defined as a 3D mesh element, the existing pavement can be displayed on the cross sections and used in quantity calculations. An example of the existing pavement as shown in a dynamic cross-section view is shown below.



Note: See Page 15 for alternative method using the Create Closed Mesh tool. This method doesn't support the Element Component Quantities.

Exercise 1: Modeling the Existing Pavement using a Surface Terrain Model

This section will demonstrate the process to create 3D mesh elements for the existing pavement. A portion of Jameson Road will be removed so we will create two separate surface terrains, one for the portion that will remain in place, and a second for the portion to be removed.

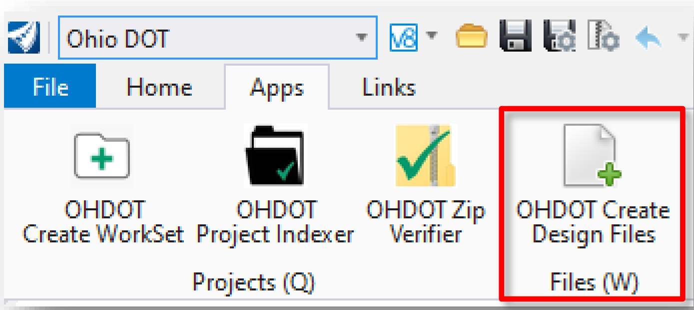
Part 1: Creating the Existing Features Design File

A new design file will be for the 3D mesh elements representing the existing features.

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **400-Engineering\Roadway\Basemaps** folder.
- ✓ Open one of the design files for the WorkSet. The file opened does not matter since it is being used to access the file creation application.

Create a new design file to use for modeling the existing features. Select the **OHDOT Create Design Files** application (**OHDOT** WorkFlow) to create the corridor modeling basemaps for the project.



- Use the Roadway Digital Terrain Model file type, “KD”, to model the existing features
- Choose the **Create Files** button to initiate the process.

Part 2: References

The surveyed edge of pavement and center of pavement elements will be used to generate the 3D mesh elements representing the existing pavement.

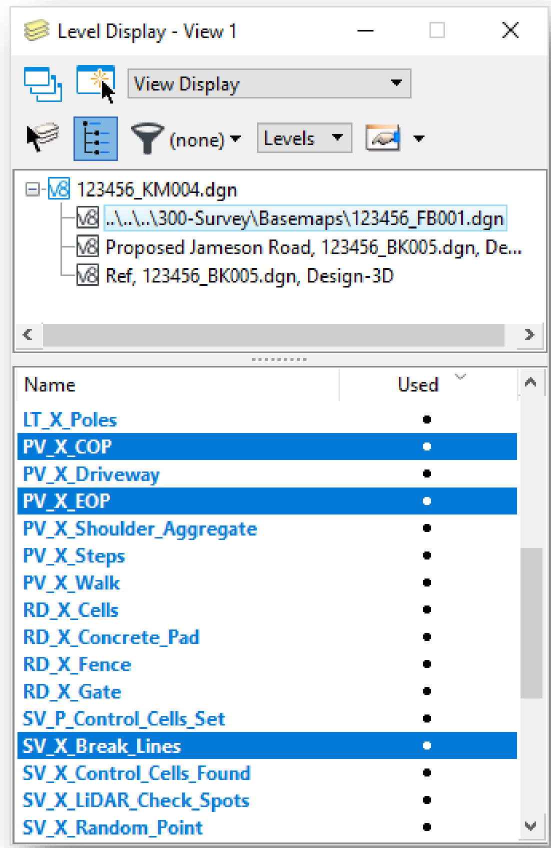
- ✓ Attach the following files as references (**Home > Attach Tools > References**):

  Geometry Basemap for Jameson Road:
  400-Engineering\Roadway\Basemaps\123456_BK003.dgn

  Survey Basemap:
  300-Survey\Basemaps\12345_FB001.dgn

Use the ***Level Display*** function (**OpenRoads Modeling > Home > Level Display**) to turn off all the levels in the attached survey design file (**123456_FB001.dgn**), except for the levels shown below

✓ From the **Home** tab, select the **Level Display** icon.
  - Select the file 123456_FB001.dgn in the top portion of the dialog.
  - Turn off all the levels by right-clicking in the lower portion of the dialog and choose the **All Off** from the pop-up menu.
  - After turning off all the levels, turn on the following levels by clicking on each level name as shown on the right.
    - PV_X_COP
    - PV_X_EOP
    - SV_X_Break_Lines



Part 3: Creating the Surface Terrain Model for the Existing Pavement

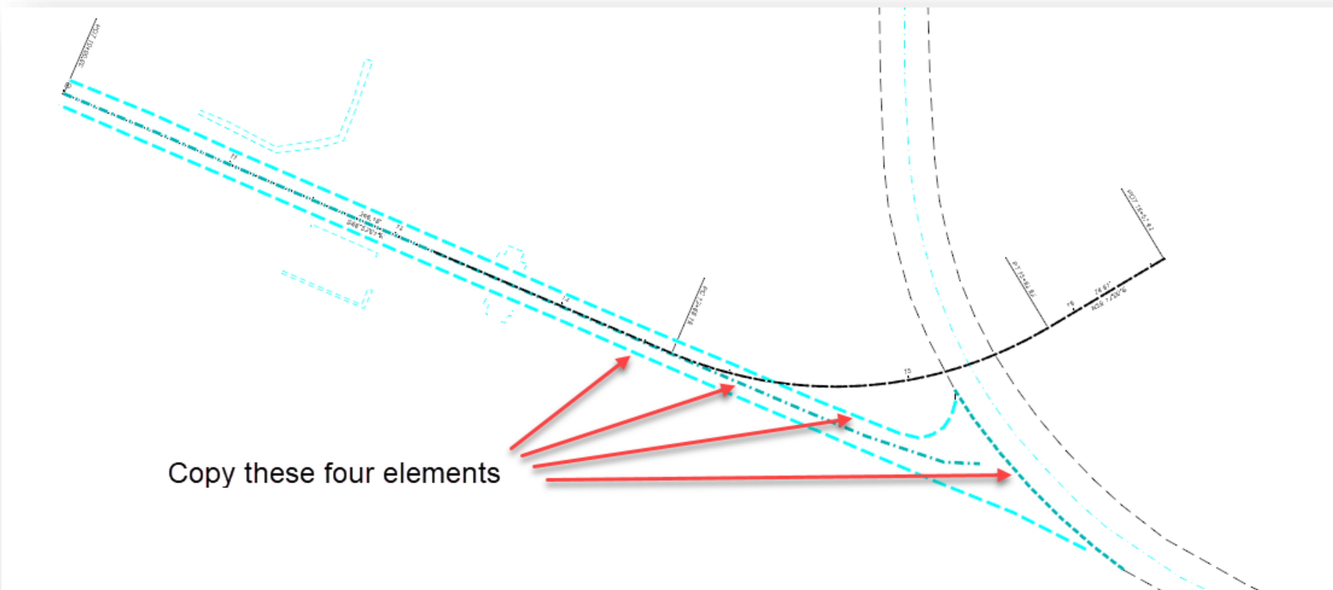
The surface terrain can be created for the existing pavement by directly using the referenced survey elements, or by copying the elements from the FB reference attachment into the active file.

In this example, we will create a surface terrain model limited to the station limits of the pavement removal areas, from station 12+00 to meet existing S.R. 185. A copy of the existing survey elements is required since the elements will be edited to specific station ranges.

Take the following steps to create the surface terrain for Jameson Road.

- ✓ Use the MicroStation **Copy Element** command (**Drawing > Copy**) to copy the existing edge of pavement, center of pavement, and break line elements from the Survey Field Book design file (FB) to the active design file. The elements to be copied are shown below.

  Be careful to copy the elements exactly on top of the original survey elements.

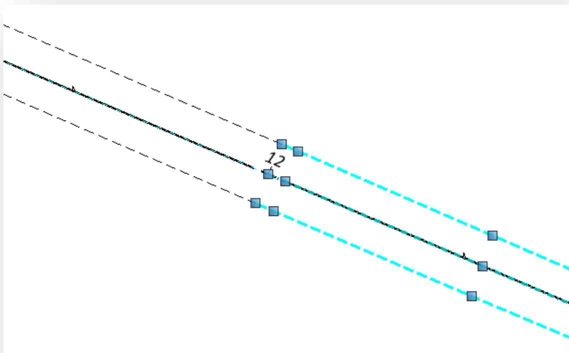


- ✓ After copying the elements, turn off the reference attachment display of the survey basemap design file, 123456_FB001.dgn (**Home > Attach Tools > References**).

Take the following steps to prepare the copied elements to generate the surface terrain model for the existing pavement:

✓ Use the **Break Element** command (**OpenRoads Modeling > Drawing > Break Element**) to break each edge of pavement line and the center of pavement line at station 12+00 of Jameson Road.

The result is shown below with the elements from station 12+00 forward selected to show that the lines are broken at station 12+00.

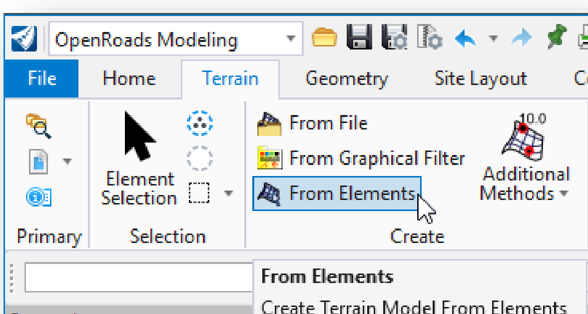


The edge of pavement elements will be used to define the boundary of the existing pavement surface terrain model. The surveyed center of pavement elements will be used as a break line to define the pavement crown. In this case, these elements are 3D graphics as generated from the Survey software and can be used to generate the surface terrain model directly. 2D elements can also be used if they have an OpenRoads profile defined.

Take the following steps to create the surface terrain for the pavement portion that will remain in place:

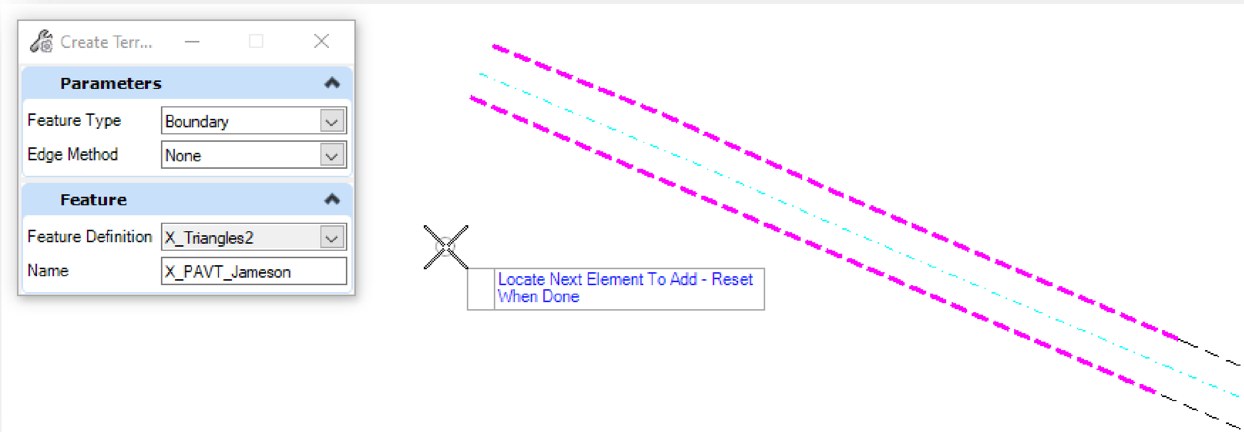
✓ From the ***References*** dialog (**Home > Attach Tools > References**), turn off the display of the file containing the proposed alignment for Jameson Road (123456_BK001.dgn)

✓ From the **Terrain** tab, select the **From Elements** command to create a terrain model from selected elements.

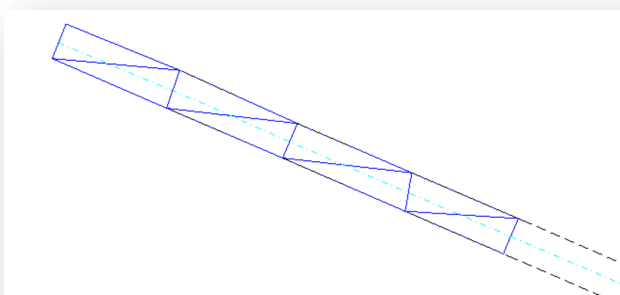


- Set the **Feature Type** to **Boundary**
- Set the **Edge Method** to **Remove Slivers**

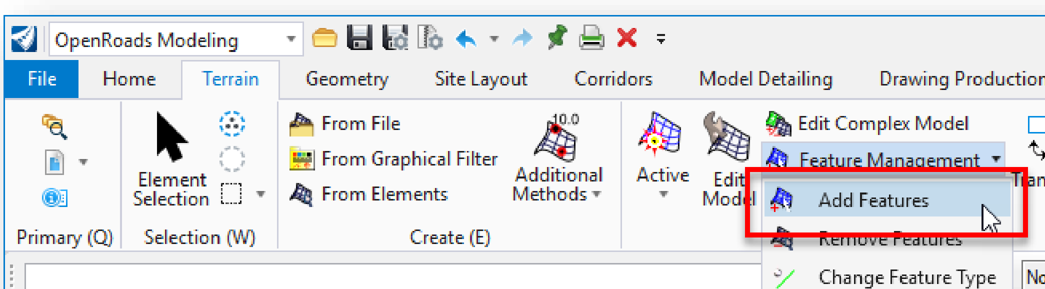
- Choose the **X_Triangles2** feature definition.
- Key-in the **Name** for the terrain model **X_PAVT_Jameson.**
- Select each edge of pavement element as shown below.



- Right-click (reset) when you are done selecting the elements.
- When prompted, left click to accept each parameter.
- After accepting each parameter, the terrain model is generated like the example below.

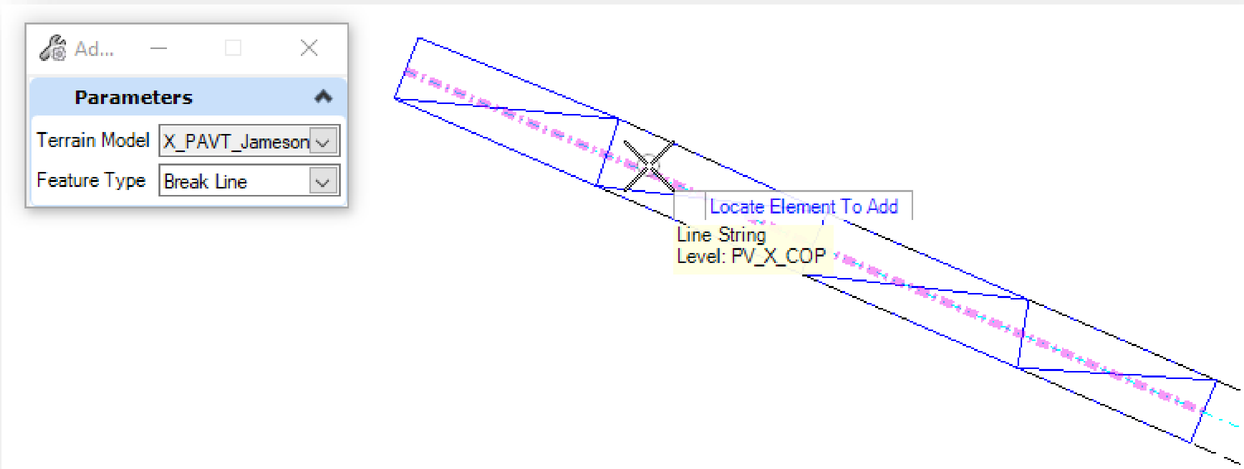


Take the following steps to add the center of pavement element to the terrain model as a break line.



From the **Terrain** tab, select **Feature Management > Add Features**

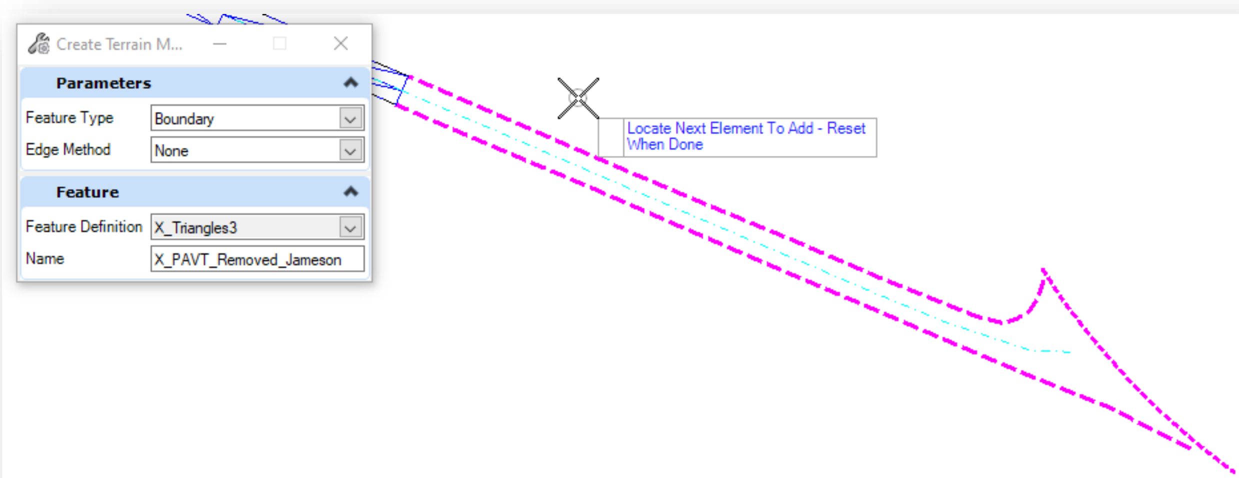
- Select the **Terrain Model** name from the drop-down list.
- Set the **Feature Type** parameter to **Break Line.**
- Select the center of pavement element as shown below.



Follow the prompts to accept the element. When complete, the center of pavement element is added to the terrain model as a break line.

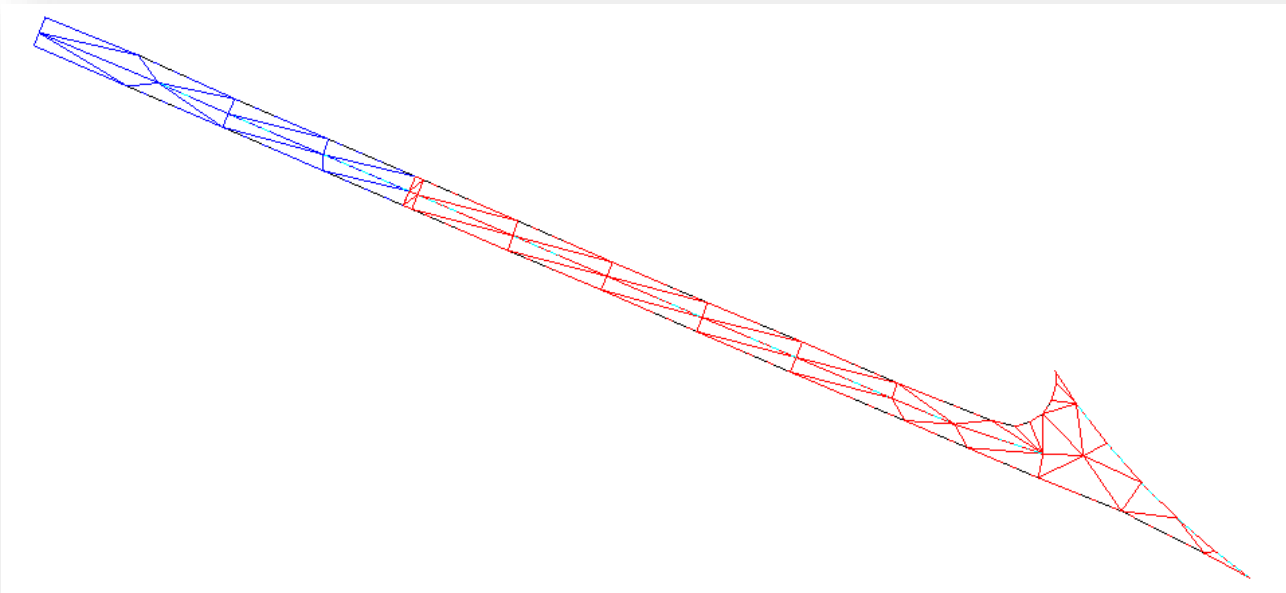
Following the steps above, create a second surface terrain for the portion of Jameson Road to be removed.

✓ Create a terrain named **X_PAVT_Removed_Jameson** from the edge of pavement and the break line at the intersection with S.R. 185.



✓ Add the crown of pavement element as a break line to the newly created terrain.

When complete, two separate terrains are created for the surface of existing Jameson Road like the example below.

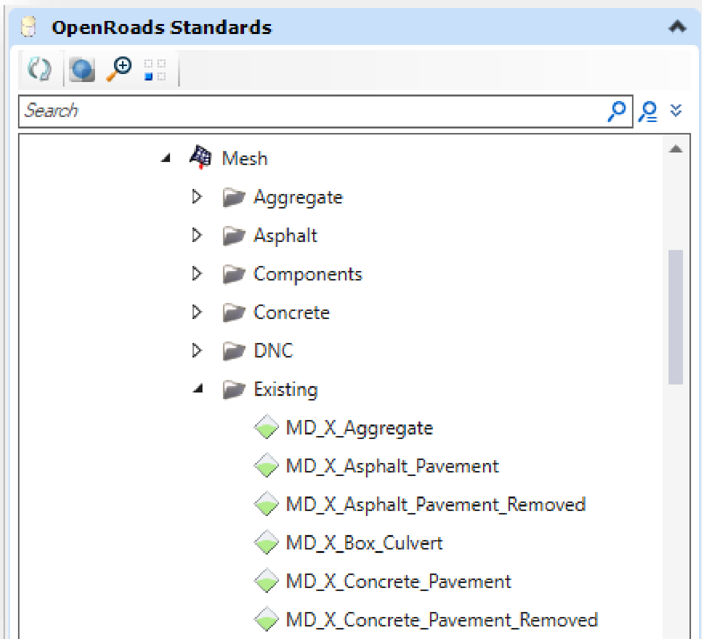


Part 4: Create the Surface Templates

OpenRoads templates can be applied to a terrain model to define the subsurface composition of the surface terrain. We will create two templates, one for the portion of Jameson Road that will remain in place, and a second for the portion to be removed.

Existing Pavement Feature Definitions

The OHDOT Standards contain several Mesh features for existing pavement as shown below.



Features for Asphalt and Concrete Removed features differ in the **Volume Option** parameter setting.

The features MD_X_Asphalt_Pavement and MD_X_Concrete Pavement have the **Volume Option** set to a value of **None**.

The features MD_X_Asphalt_Pavement_Removed and MD_X_Concrete_Pavement_Removed have the **Volume Option** set to a value of **Unsuitable**.

These settings are used to determine how the components are included in earthwork computations.

Creating an Existing Pavement Template

Take the following steps to create a template to the Jameson Road surface terrain:

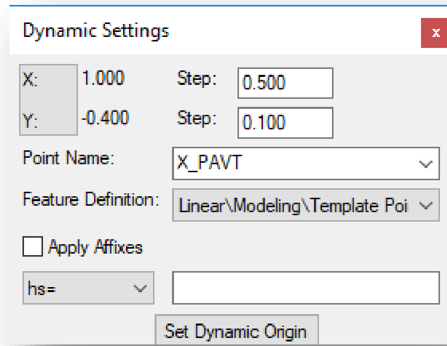
- ✓ From the **Corridors** tab, select **Template > Create Template**
- ✓ Right-click on the **Project Templates** folder and select **New > Template**
- ✓ Rename the new template "**Jameson_X_Pavt**"

Add components to define the pavement building of the exiting road.

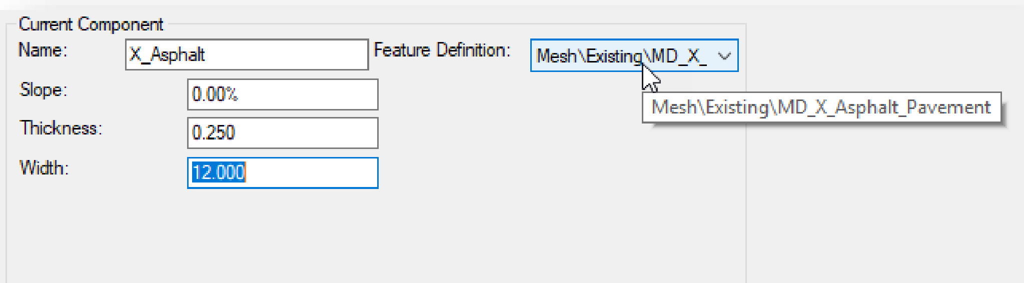
- ✓ Choose **Tools > Dynamic Settings** to open the dialog shown below.

Use the settings shown on the right. The **Feature Definition** is set as follows:

Linear\Modeling\Template Points\Scratch**Scratch1**

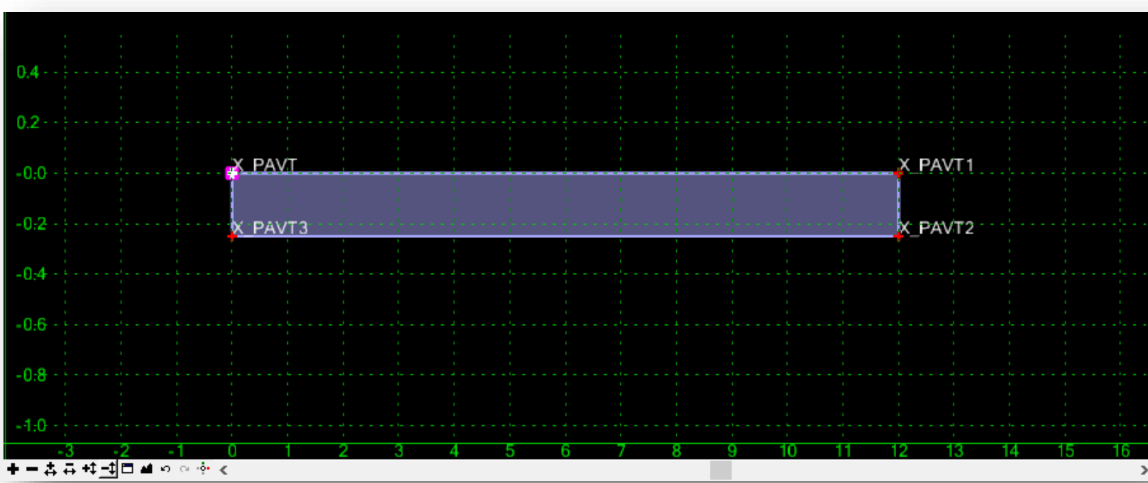


- ✓ In the **Current Template** window, right-click and choose **Add New Component > Simple.**
- ✓ A simple component consists of four points defining a rectangular shape. Set the parameters as shown below.

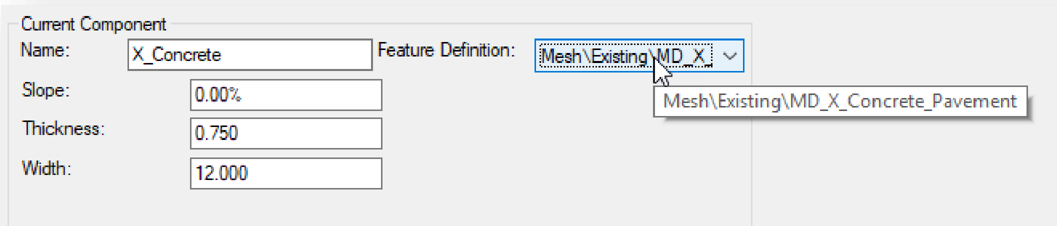


Note: The slope and width parameters are irrelevant as this is defined by the terrain surface.

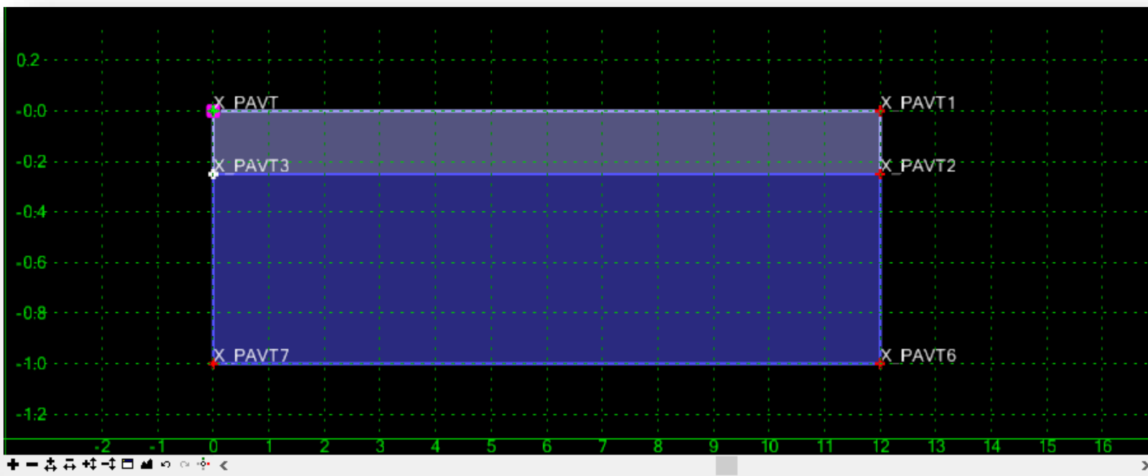
- ✓ Place the component at the 0,0 location as shown below.



- ✓ Right-Click in the **Current Template** window and choose **Add New Component > Simple** to add a component for a concrete pavement layer. Use the settings shown below.



- ✓ Place the component at the bottom of the previous component. The completed template should appear as show below



Creating an Existing Pavement Removed Template

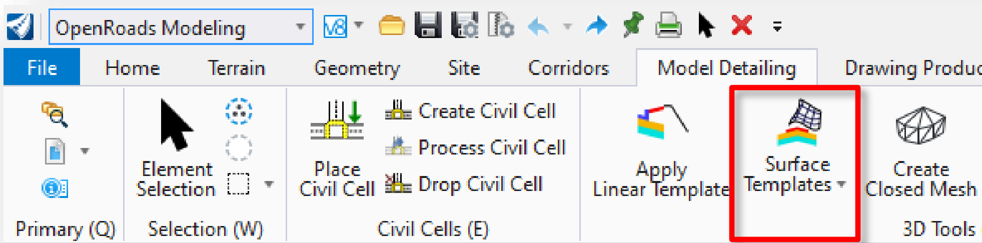
Take the following steps to create a template for the existing pavement removal areas.

- ✓ Right click the name of the Jameson_X_Pavt template and choose the **Copy** option.
- ✓ Right click on the Project Templates folder name and choose the **Paste** option.
- ✓ Rename the copied template as "Jameson_X_Pavt_Removed.
- ✓ Change the feature definition that is assigned to each of the two pavement layers as follows:
  - Right click the X_Asphalt component and choose the **Edit Component** command.
  - Change the **Feature Definition** to X_Asphalt_Pavement_Removed. Select the **Apply** button.
  - Edit the X_Concrete component to use the **Feature Definition.** .X_Concrete_Pavement_Removed. Select the **Apply** button and close the ***Component Properties*** dialog.
- ✓ In the ***Create Templates*** dialog, choose **File > Save** and then close the template library.

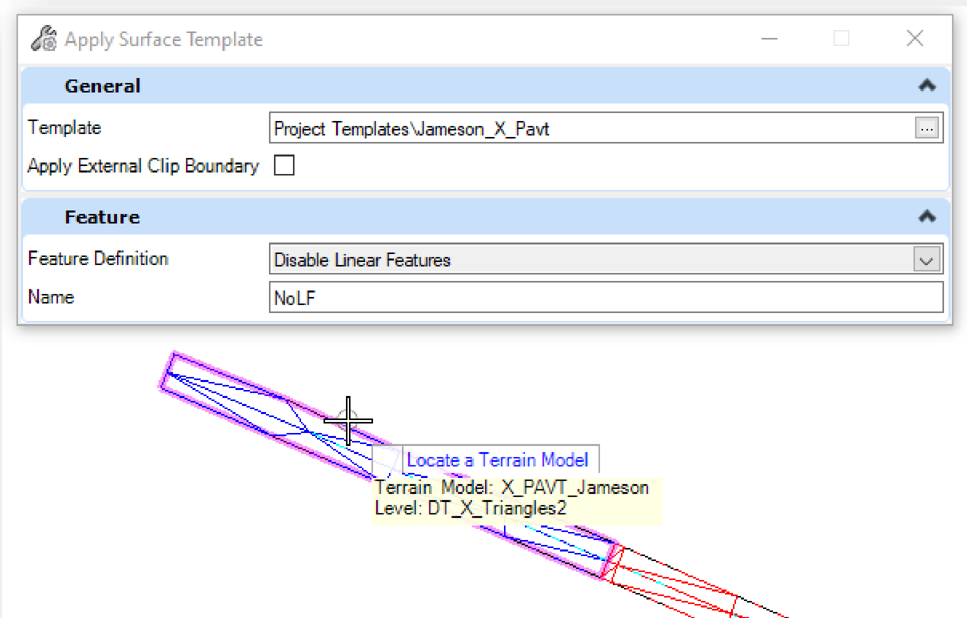
Part 5: Apply the Surface Template

The template is applied to terrain model using the **Apply Surface Template** command.

✓ From the **Model Detailing** tab, select **Apply Surface Template.**

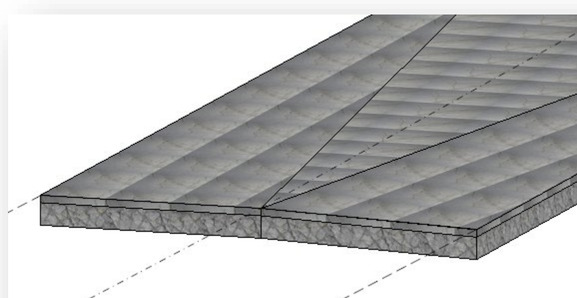


✓ Select the **Template** for the portion of Jameson Road that will remain in place to apply the template as shown below.



✓ When prompted, select the terrain model.

The template is applied to the 3D model, like the example at right.



Part 6: Dynamic Cross Section View

Take the following steps to display the elements in a dynamic cross-section view:

- ✓ Turn on the reference file display of the Jameson Road alignment (**123456_BK003.dgn**), which was turned off in a prior step in this training exercise (**OpenRoads Modeling > Home > Attach Tools**).

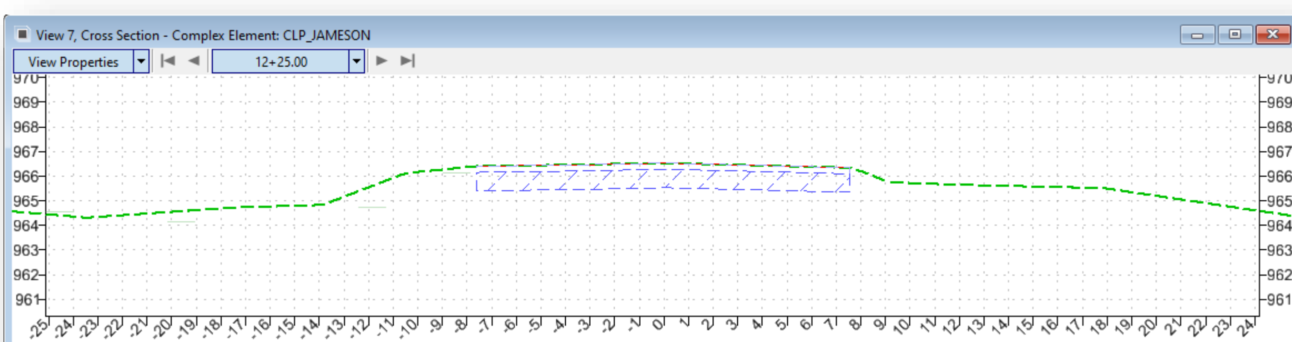
- ✓ Turn on the reference file display of the survey design file (**123456_FB001.dgn**), which was turned off in a prior step in this training exercise (**OpenRoads Modeling > Home > Attach Tools**).

- ✓ Use the Level Display dialog (**OpenRoads Modeling > Home > Level Display**) to turn on the following levels for the survey file (**123456_FB001.dgn**):
  - DT_X_Terrain_Boundary
  - DT_X_Terrain_Ground_Line

- ✓ Set the survey existing ground terrain model as the active terrain.
  - Choose the **Element Selection** tool (**OpenRoads Modeling > Home > Element Selection**).
  - Select the survey terrain model boundary. Allow the cursor to rest on the boundary element until the pop-up menu appears.
  - From the pop-up menu, choose the second icon to set the selected terrain model as the active terrain.

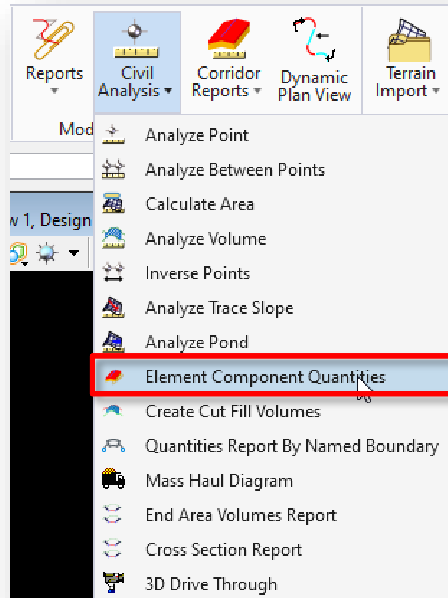
- ✓ Hold down the right-mouse button in the view window.
  Select **View Control > 2 Views Plan/XS** from the pop-up menu.

- ✓ When prompted, select **OK** to generate a Dynamic XS view.
  - When prompted to **Locate Corridor or Alignment**, choose the Jameson Road alignment.
  - Enter a **Left Offset** value of -100
  - Enter a **Right Offset** value of 100
  - Enter a **Station** value of 12+00
  - Enter an **Interval** of 50
  - When prompted to **Select or Open View**, enter a data point (left mouse click) in the cross-section view window. An example is shown below.



Part 7: Component Quantities

The OpenRoads component quantities report can be used with the surface terrain and mesh elements. Take the following steps to generate the component quantities for the Jameson Road pavement removal 3D mesh elements:

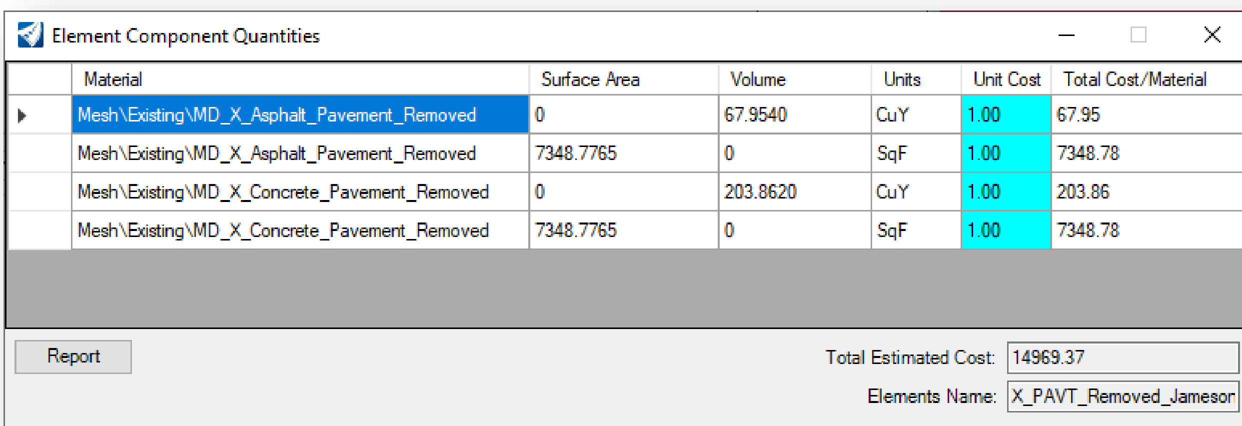


- ✓ From the **OpenRoads Modeling** workflow, **Home** tab, select **Civil Analysis > Element component Quantities** as shown at right.
- ✓ When prompted to **Locate First Element**, choose the surface terrain model for Jameson Road. Right-click (reset) when complete.
- ✓ The ***Element Component Quantities*** report is opened like the example below.

  Both the Volume and Surface Area are reported for each of the components as shown below.

  The **Unit Cost** can be assigned for each **Material** to generate a cost estimate.

  Select the **Report** button to send the quantities to the OpenRoads Report Browser. From there, the report can be saved in ASCII, Word, or Excel formats.


Element Component Quantities

Material	Surface Area	Volume	Units	Unit Cost	Total Cost/Material
Mesh\Existing\MD_X_Asphalt_Pavement_Removed	0	67.9540	CuY	1.00	67.95
Mesh\Existing\MD_X_Asphalt_Pavement_Removed	7348.7765	0	SqF	1.00	7348.78
Mesh\Existing\MD_X_Concrete_Pavement_Removed	0	203.8620	CuY	1.00	203.86
Mesh\Existing\MD_X_Concrete_Pavement_Removed	7348.7765	0	SqF	1.00	7348.78

Report

Total Estimated Cost: 14969.37

Elements Name: X_PAVT_Removed_Jameson

Alt. Exercise 2: Modeling the Existing Pavement using Create Closed Mesh

This section will demonstrate the process to create 3D mesh elements for the existing pavement using the Create Closed Mesh tool. A portion of St. Peters Rd. will be removed so we will create two separate surface terrains, one for the portion that will remain in place, and a second for the portion to be removed.

Part 1: References

The surveyed edge of pavement and center of pavement elements will be used to generate the 3D mesh elements representing the existing pavement.

✓ Attach the following files as references (**Home > Attach Tools > References**):

Geometry Basemap for St. Peters Rd:
400-Engineering\Roadway\Basemaps\123456_BK002.dgn

Part 2: Creating the Surface Terrain Model for the Existing Pavement

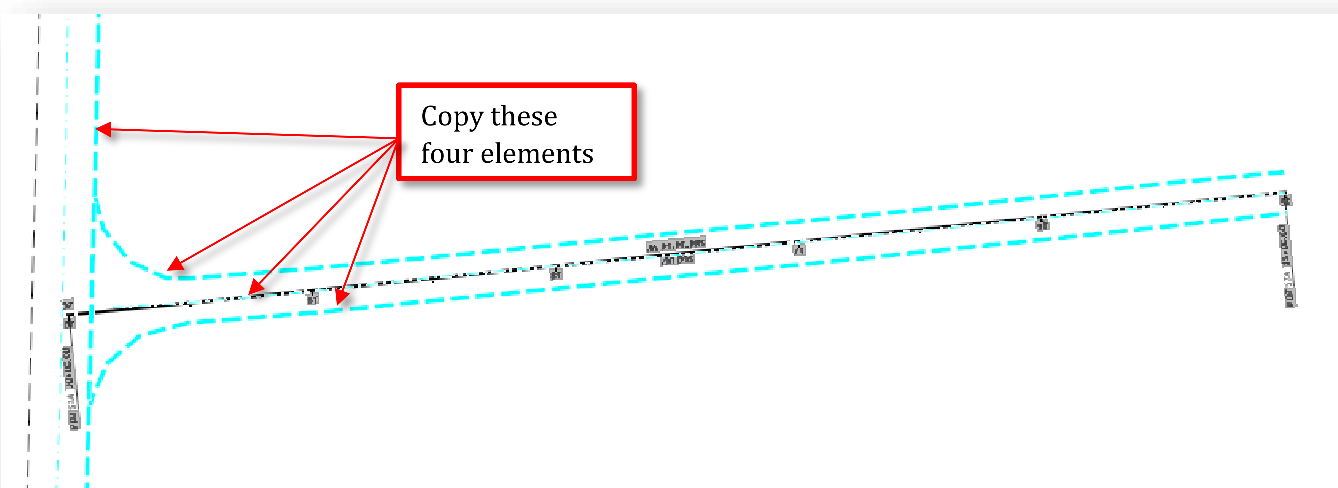
The surface terrain can be created for the existing pavement by directly using the referenced survey elements, or by copying the elements from the FB reference attachment into the active file.

In this example, we will create a surface terrain model limited to the station limits of the pavement removal areas, from station 17+00 to meet existing S.R. 185. A copy of the existing survey elements is required since the elements will be edited to specific station ranges.

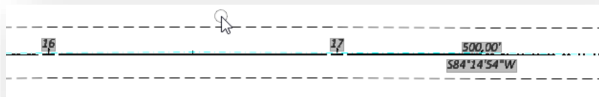
Take the following steps to create the surface terrain for St. Peters Rd.

✓ Use the MicroStation **Copy Element** command (**Drawing > Copy**) to copy the existing edge of pavement, center of pavement, and break line elements from the Survey Field Book design file (FB) to the active design file. The elements to be copied are shown below.

Note: Be careful to copy the elements exactly on top of the original survey elements.



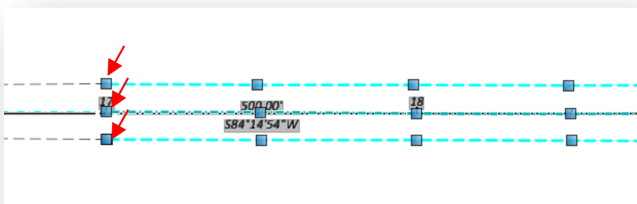
- ✓ After copying the elements, turn off the reference attachment display of the survey basemap design file, 123456_FB001.dgn (**Home > Attach Tools > References**).
- ✓ Then use the View Rotation: Rotate View by 3 points so that the St. Peters Rd. alignment annotations are in the correct orientation.



Take the following steps to prepare the copied elements to generate the surface terrain model for the existing pavement:

- ✓ Use the **Break Element** command (**OpenRoads Modeling > Drawing > Break Element**) to break each edge of pavement line and the center of pavement line at station 17+00 of St. Peters Rd.

  The result is shown below with the elements from station 17+00 forward selected to show that the lines are broken at station 17+00.



- ✓ Select the level **PV_P_Saw_Cut_Line**

- ✓ Use the **Place Line** command (**OpenRoads Modeling > Drawing > Place Line**) to draw a line to each point edge of pavement line at sta. 17+00 of St. Peters Rd.

The edge of pavement elements and sawcut element will be used to define the boundary of the existing pavement surface terrain model. The surveyed center of pavement elements will be used as a break line to define the pavement crown. In this case, these elements are 3D graphics as generated from the Survey software and can be used to generate the surface terrain model directly. 2D elements can also be used if they have an OpenRoads profile defined.

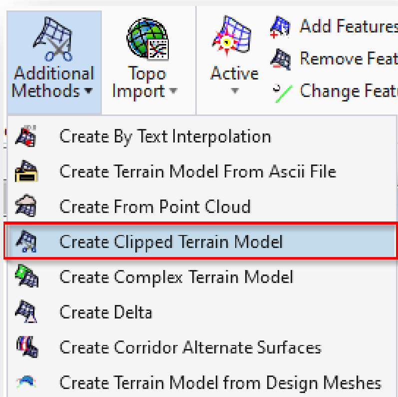
Take the following steps to create the surface terrain for the pavement portion that will remain in place:

- ✓ From the ***References*** dialog (**Home > Attach Tools > References**), turn off the display of the file containing the proposed alignment for St. Peters Rd. (123456_BK002.dgn). Then turn the survey file back on (123456_FB001.dgn).

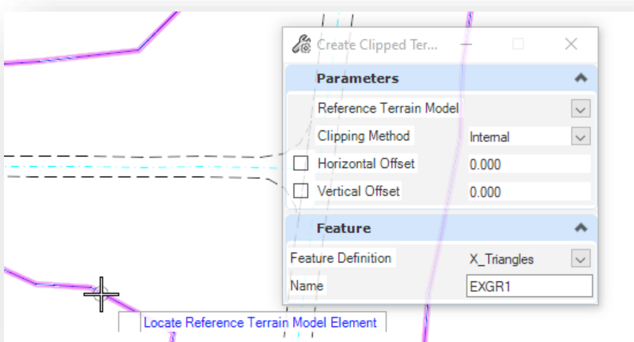
- ✓ From the ***Level Display*** dialog **(Home > Primary > Level Display)**, turn off all levels within the survey (123456_FB001.dgn) then turn on the **DT_X_Terrain_Ground_Line** & **DT_X_Terrain_Boundary** level.

✓ From the **Drawing** tab, select the **Create Complex Chain** command under the **Groups**. Then select all the EOP from Sta. 17+00 of St. Peters Rd. to S.R. 185.

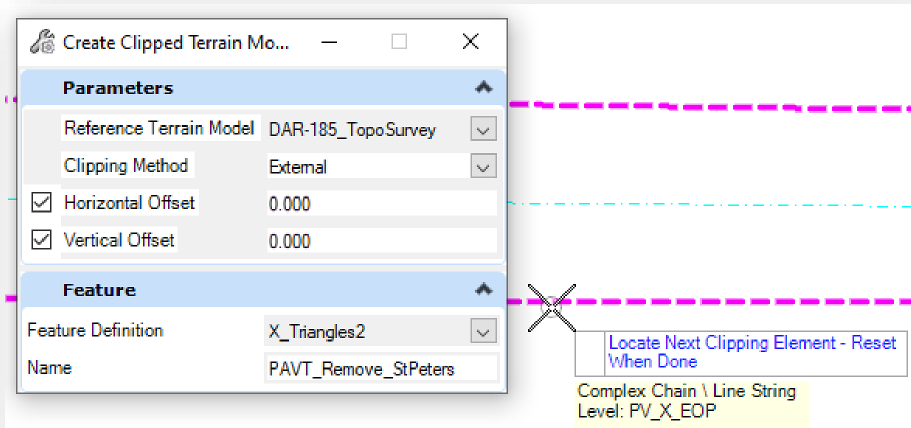
✓ From the **Terrain** tab, select the **Additional Methods** command to create a terrain model from selected elements.



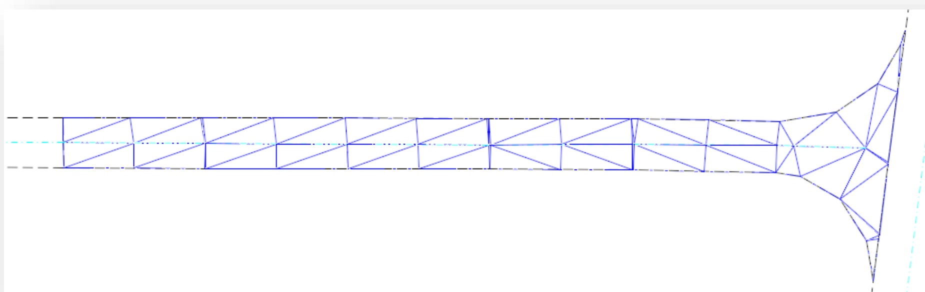
✓ Once the **Create Clipped Terrain Model** tool has been selected then survey terrain boundary.



- Set the **Clipping Method** to **External**
- **Horizontal Offset = 0.000**
- **Vertical Offset = 0.000**
- Choose the **X_Triangles2** feature definition.
- Key-in the **Name** for the terrain model **X_PAVT_Remove_StPeters**.
- Select each edge of pavement element as shown below.

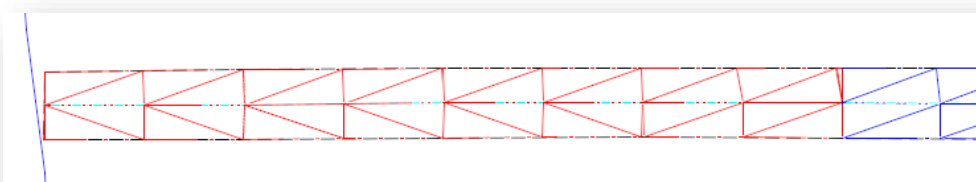


- Right-click (reset) when you are done selecting the elements.
- When prompted, left click to accept each parameter.
- After accepting each parameter, the terrain model is generated like the example below.



Note: this method automatically adds the center of pavement element to the terrain model as a break line.

- Repeat the process above for the Existing to Remain section of St. Peters Rd. using **X_Triangles3** feature definition and Key-in the **Name** for the terrain model **X_PAVT_Remain_StPeters**.

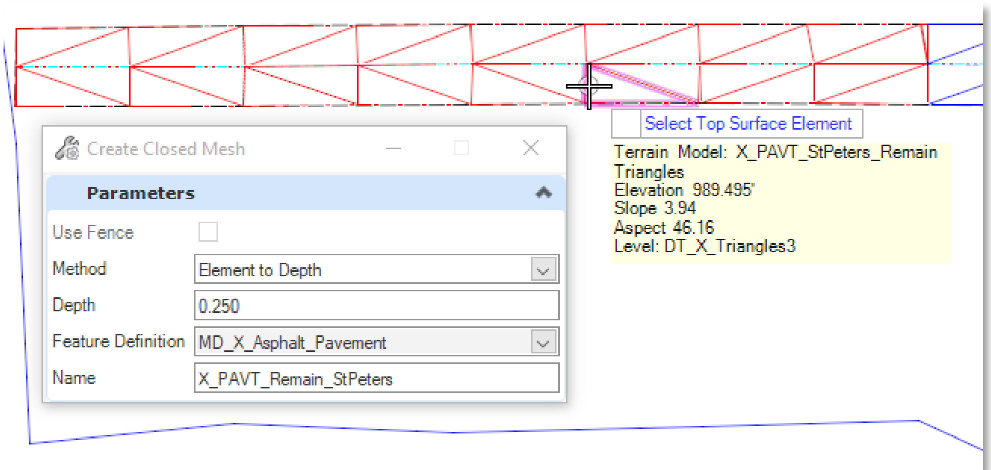


Part 3: Applying a Closed Mesh to a Surface

The Create Closed Mesh can be applied to a terrain model to define the subsurface composition of the surface terrain. We will create two templates, one for the portion of St. Peters Rd. that will remain in place, and a second for the portion to be removed.

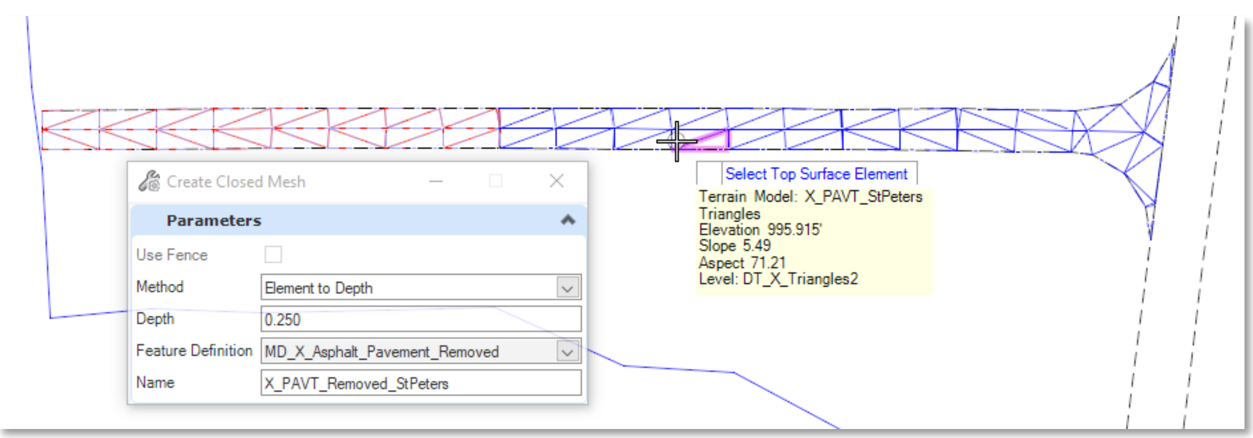
From the **Modeling Detailing** tab, select **Create Closed Mesh** tool under the **3D Tools** sections.

- Set the **Method** to **Element to Depth**
- **Depth = 0.25**
- Choose the **MD_X_Asphalt_Pavement** feature definition.
- Key-in the **Name** for the closed mesh **X_PAVT_Remain_StPeters**
- Data selected the first prompt.
- Select the surface of pavement element as shown below.

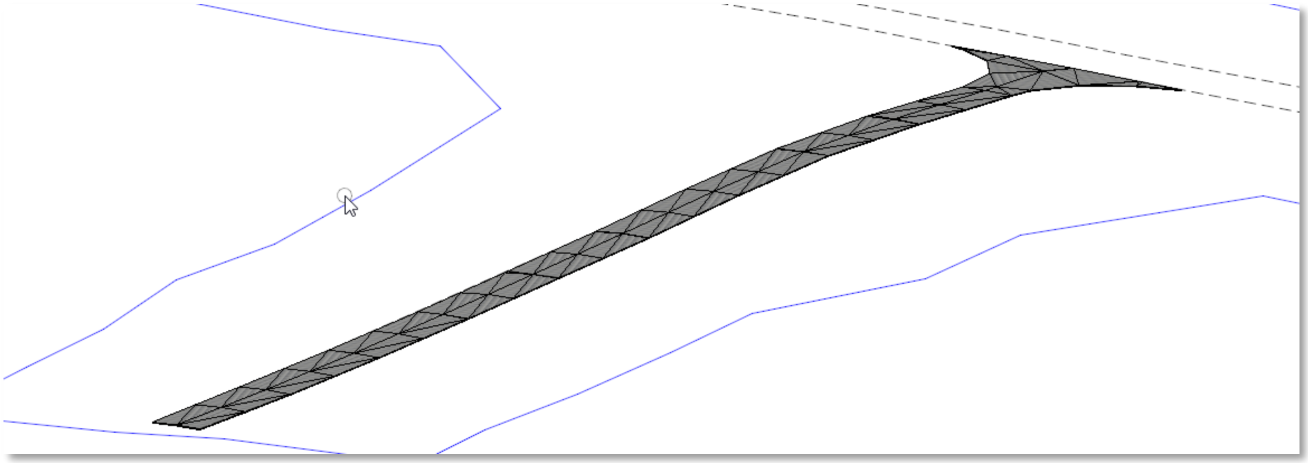


✓ Following the steps above, create a second closed mesh for the surface terrain for the portion of St. Peters Rd. to be removed.

- Choose the **MD_X_Asphalt_Pavement_Removed** feature definition.
- Key-in the **Name** for the closed mesh **X_PAVT_Removed_StPeters**.
- Select the surface terrain shown below.



✓ When complete, two separate closed mesh's are created form the surface of existing St. Peters Rd. like the example below.

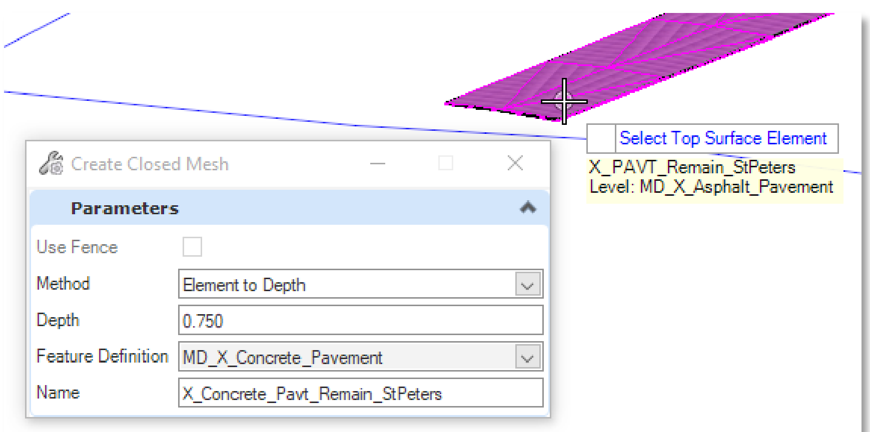


Tip: go to the view attributes  drop down to change the view display style to **Illustration:IgnoreLighting**.

Creating an Existing Asphalt Concrete Base

From the **Modeling Detailing** tab, select **Create Closed Mesh** tool.

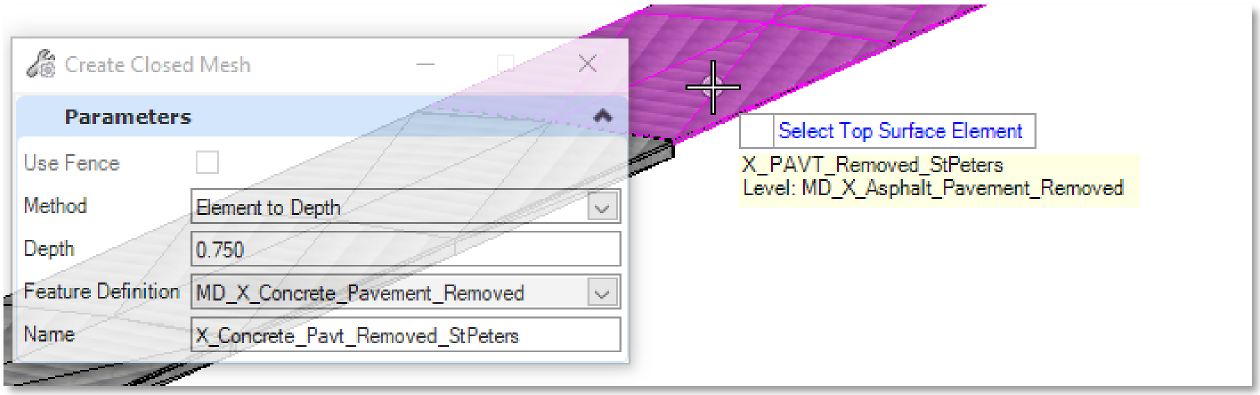
- Set the **Method** to **Element to Depth**
- **Top Mesh Side = Use Lowest**
- **Depth = 0.75**
- Choose the **MD_X_Concrete_Pavement** feature definition.
- Key-in the **Name** for the closed mesh **X_Concrete_Pavt_Remain_StPeters**
- Data selected the first prompt.
- Select the surface of pavement element as shown below.



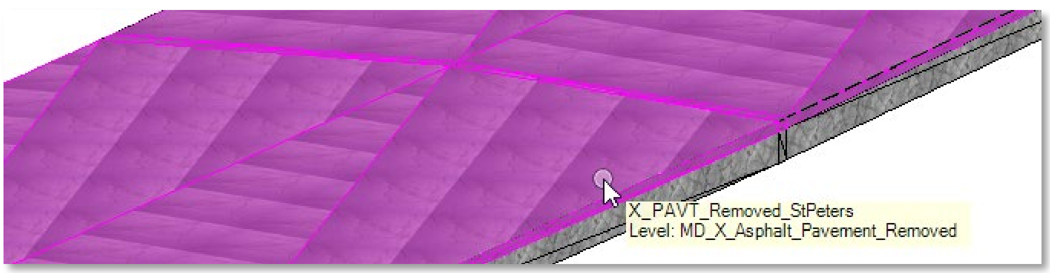
- A new Drop down with appear **Top Mesh Side** select **Use Lowest.**

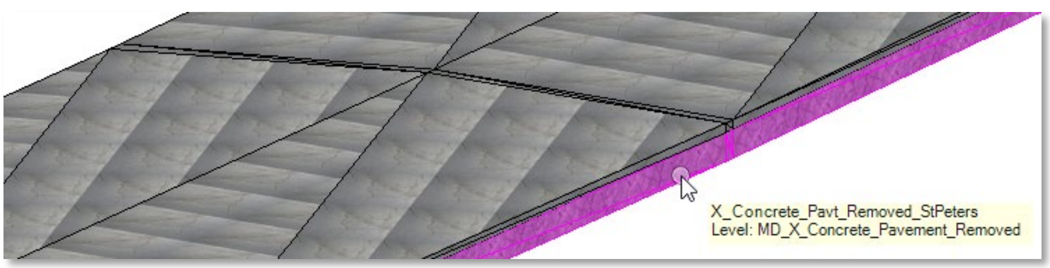
✓ Following the steps above, create a second closed mesh for the surface terrain for the portion of St. Peters Rd. to be removed.

- Choose the **MD_X_Concrete_Pavement_Removed** feature definition.
- Key-in the **Name** for the closed mesh **X_Concrete_Pavt_Removed_StPeters**.
- Select the surface terrain shown below.

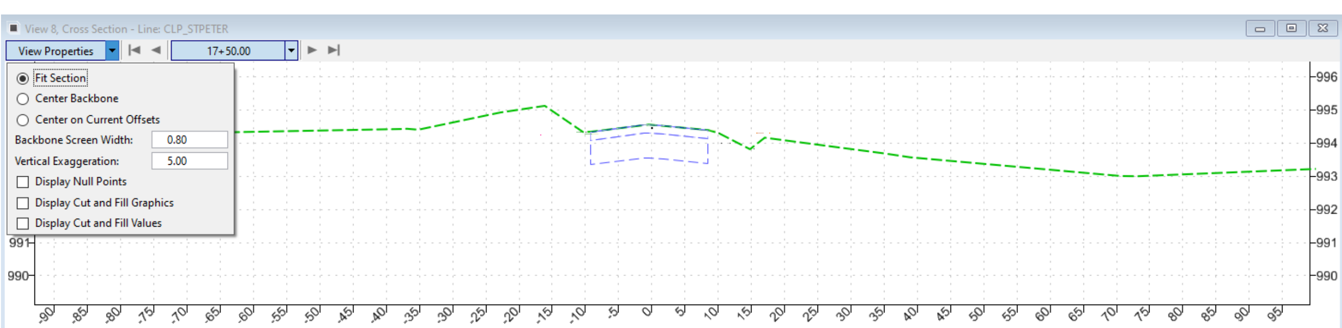


✓ When complete, you should have a Pavement layer and a concrete layer along St. Peters Rd. like the example below.





✓ Dynamic XS view of St. Peters Rd. use the **View Properties** shown below.



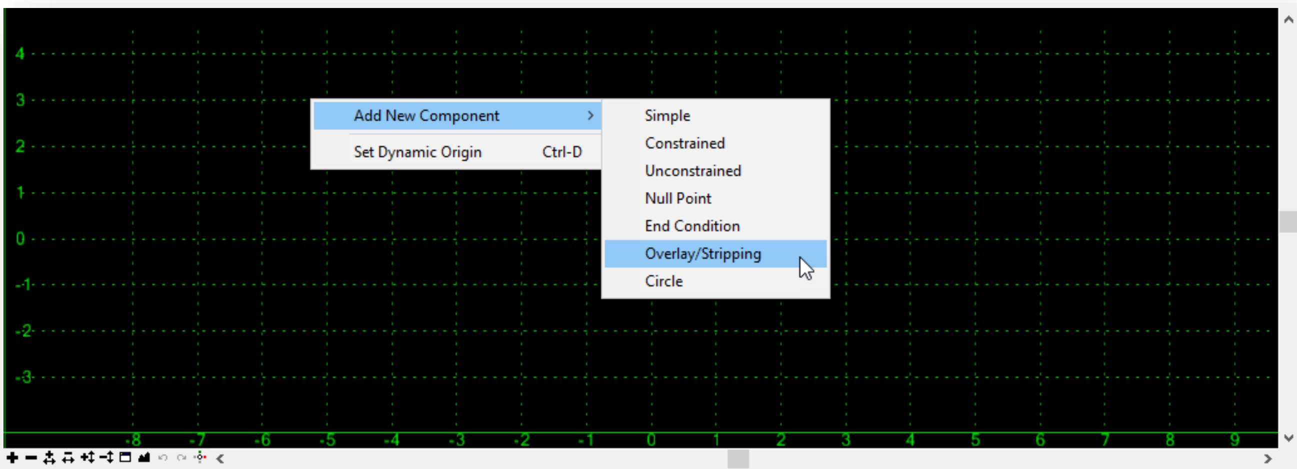
Section 514 Widening and Resurfacing

Overview

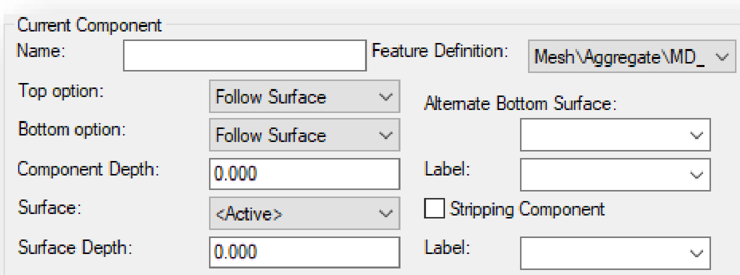
Widening and resurfacing projects can be accommodated using the OpenRoads Designer tools. Template components can be added to define pavement planning and wedge course components. This section will introduce users to the tools available for widening and resurfacing projects.

Template Components for Overlay and Stripping

From the **OpenRoads Modeling** WorkFlow, select the **Create Template** icon from the **Corridors** tab to open the ***Create Template*** dialog. To add an Overlay/Stripping component to the template, right-click in the current template window and choose **Add New Component > Overlay/Stripping** as shown below.



Several parameters are available to define the overlay/stripping component.



Name
Specifies the name of the new component.

Feature Definition
Specifies the feature style of the new component. This option is used for display and to define the component's material. The OHDOT standards contains two features for an overlay/stripping component:

Mesh\Resurfacing\MD_P_Pavement_Planing
Mesh\Resurfacing\MD_P_Wedge Course

Top Option
Specifies how the top of the component is defined. The following options are available:

Follow Surface
Specifies the top of the component will follow the surface.

Follow Component
Specifies the top of the component will follow the defined top of the component.

Follow Highest
Specifies the top of the component will follow the higher of the defined segments or surface.

Bottom Option
Specifies how the bottom of the component is defined. The behavior of these options can be different, depending on the selected Top Option.

Follow Surface
Specifies the bottom of the component will follow the surface at the defined depth but will not cross over the top of the component. The following options are available:

Follow Component
Specifies the bottom of the component will follow the defined top of the component at the defined depth but will not go above the top of the component.

Follow Lowest
Specifies the bottom of the component will follow the lower of the component/surface at the defined depths.

Follow Highest
Specifies the bottom of the component will follow the higher of the component/surface at the defined depths.

Component Depth
Sets the depth of the bottom below to the defined component line. This value cannot be negative.

Surface
Defines the surface to follow along. Does not support aliasing.

Surface Depth
Sets the depth of the bottom below the surface. This value cannot be negative.

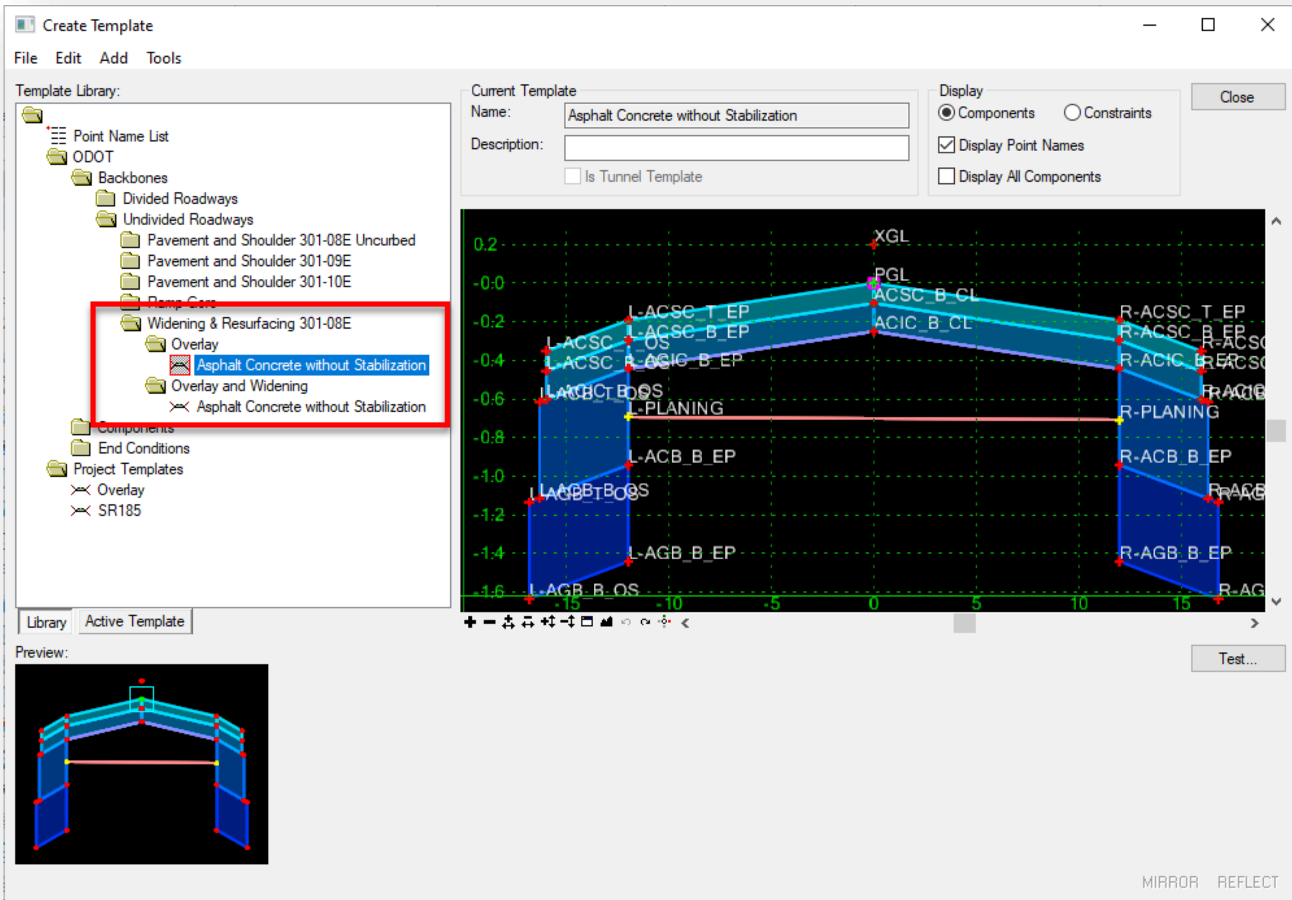
Alternate Bottom Surface
If this field is populated, when alternate surfaces are created from the Roadway Designer Create Surface command, the bottom of this component will generate an alternate surface of the specified name.

Label (Component Depth/Surface Depth)
Parametric label, used to change the default depth(s) from within Roadway Designer > Tools > Parametric Constraints.

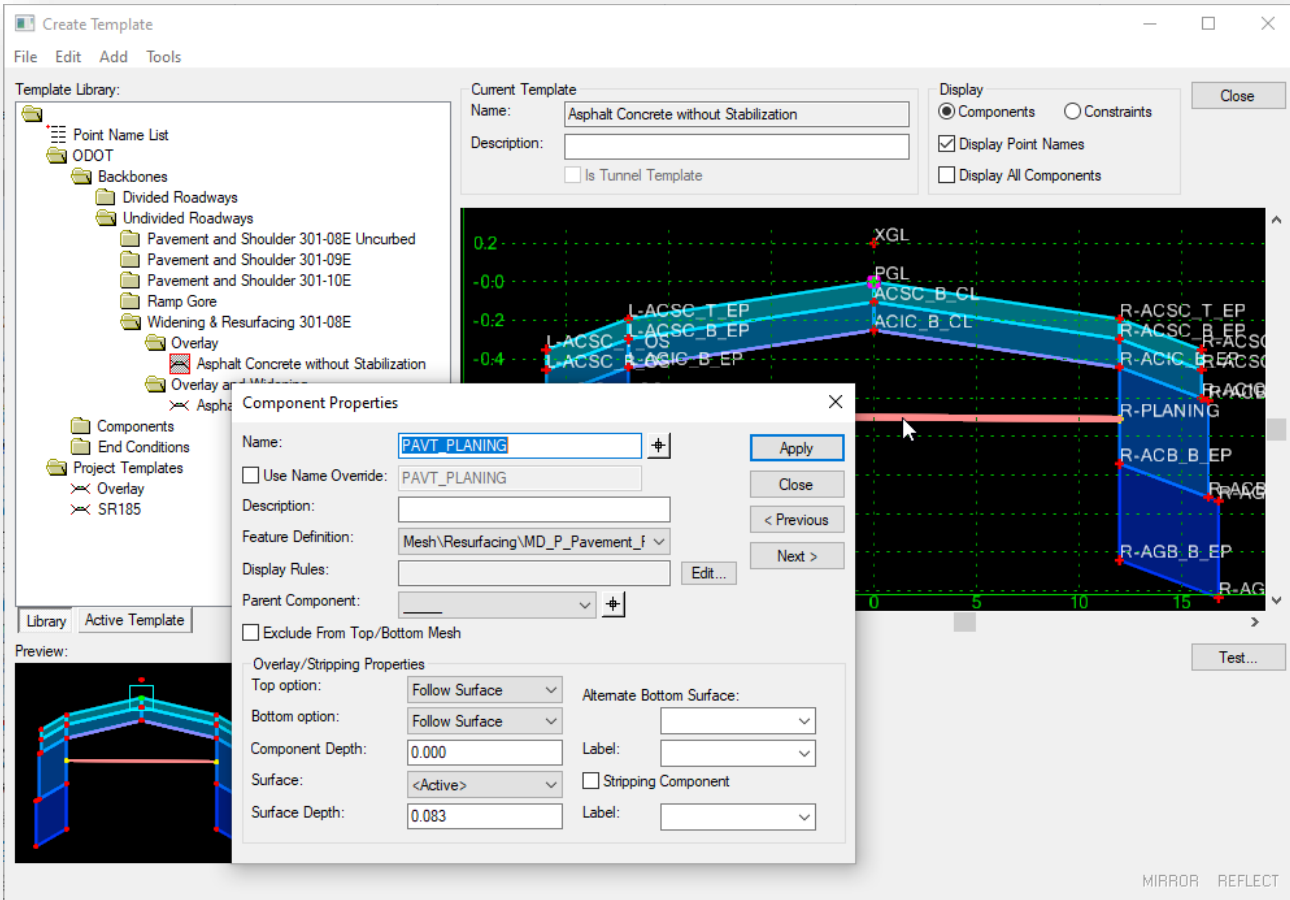
Stripping Component
Defines whether this component is designed to remove material or add material. Used in end-area volumes.

ODOT Example Templates for Overlay and Widening

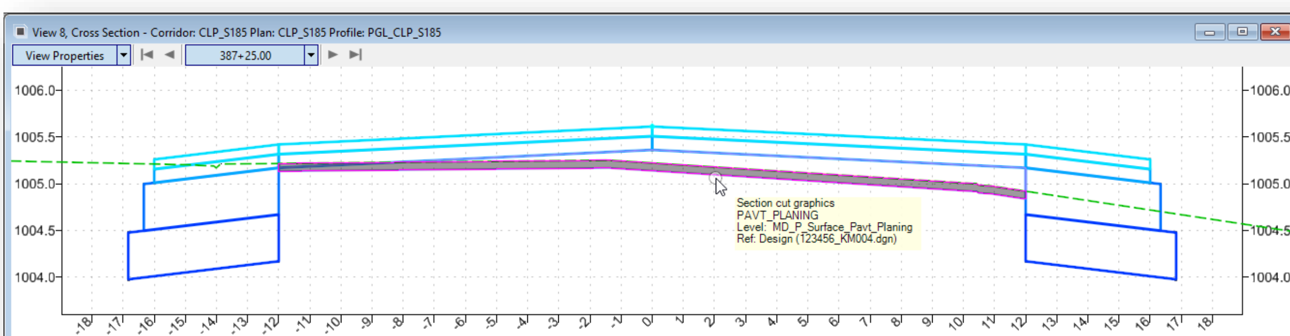
The **OHDOT_Templates.itl** library contains two sample templates for widening and resurfacing as shown below. These templates can be used as the starting point for customization to meet a variety widening and resurfacing project needs.



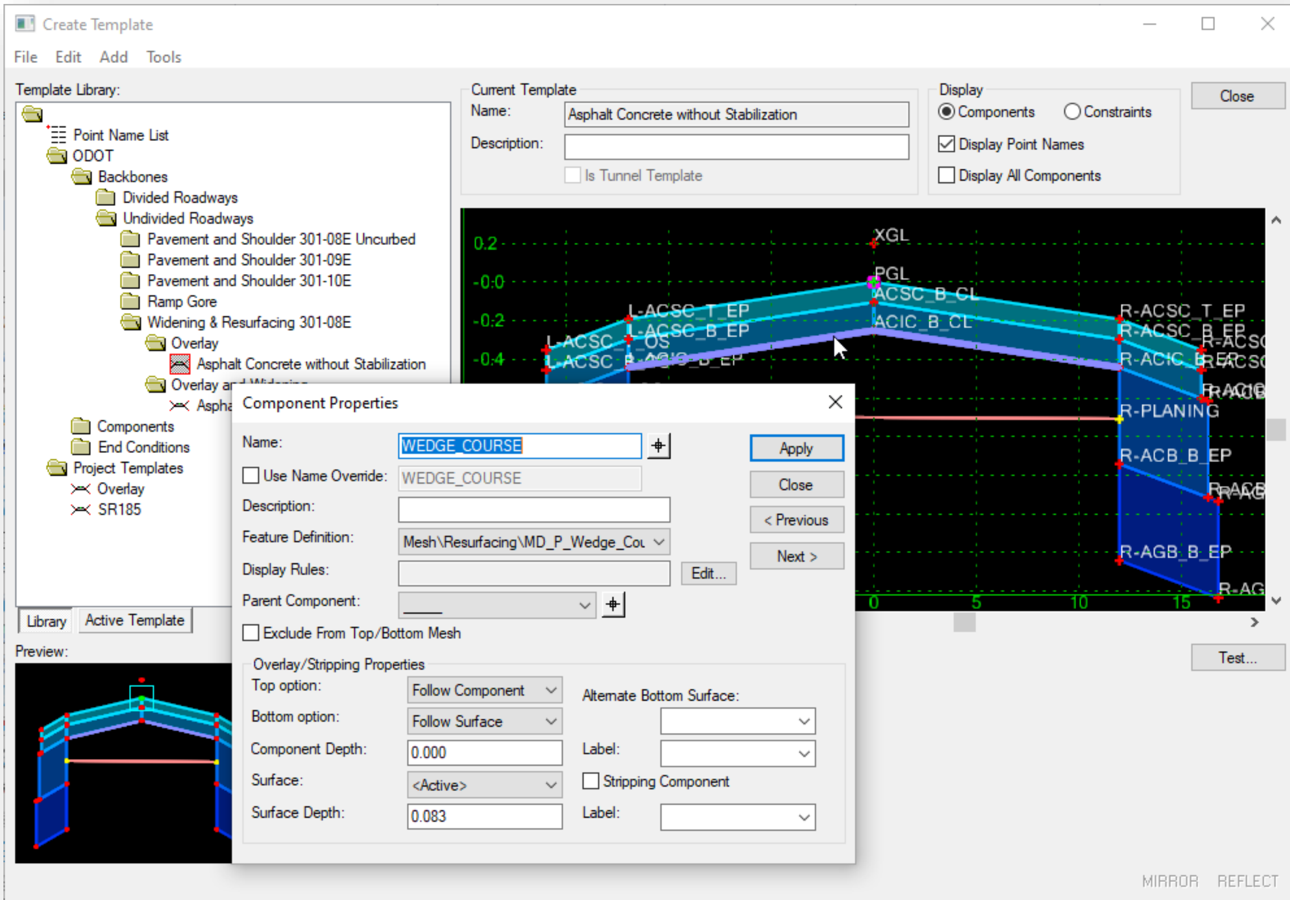
These templates have pavement planning and wedge course components defined. The pavement planning component is defined as shown below:



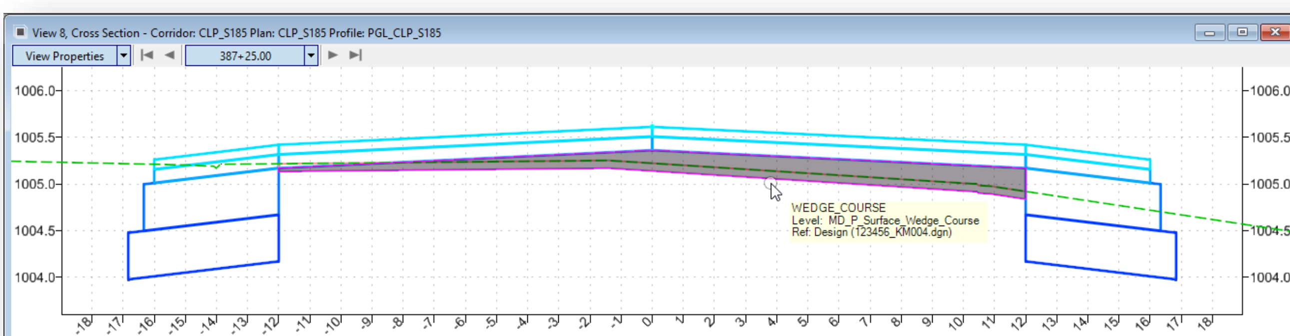
The pavement planing component is defined as a straight line across the template. The component is defined in the template below the pavement, but the actual location of the component within the template is not important. When the template is processed, the top of the pavement planing component will follow the existing terrain surface. The bottom is defined to follow the terrain surface with a depth of 1”. An example is shown below.



The wedge course component is defined to overlap the bottom of the Asphalt Concrete Intermediate Course layer as shown below.

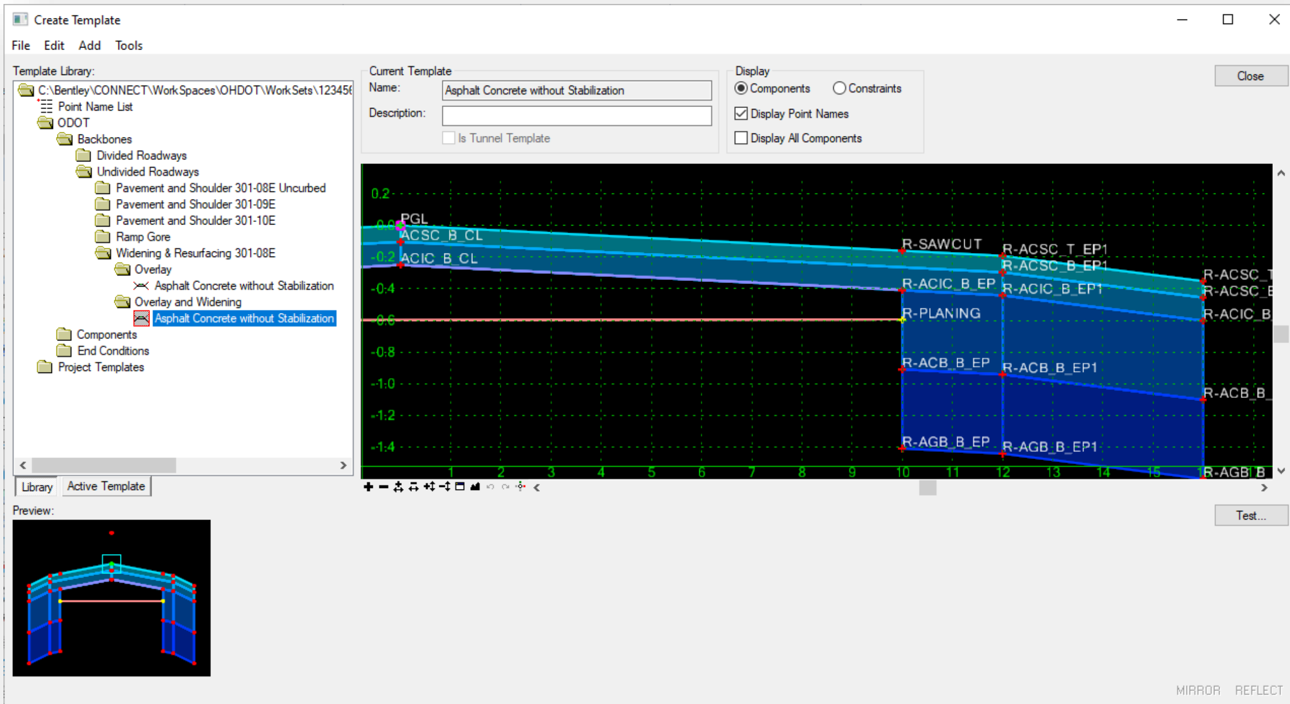


When the template is processed, the top of the wedge course component will follow the component definition. The bottom is defined to follow the terrain surface with a depth of 1" which is the depth of the pavement planing. An example is shown below.

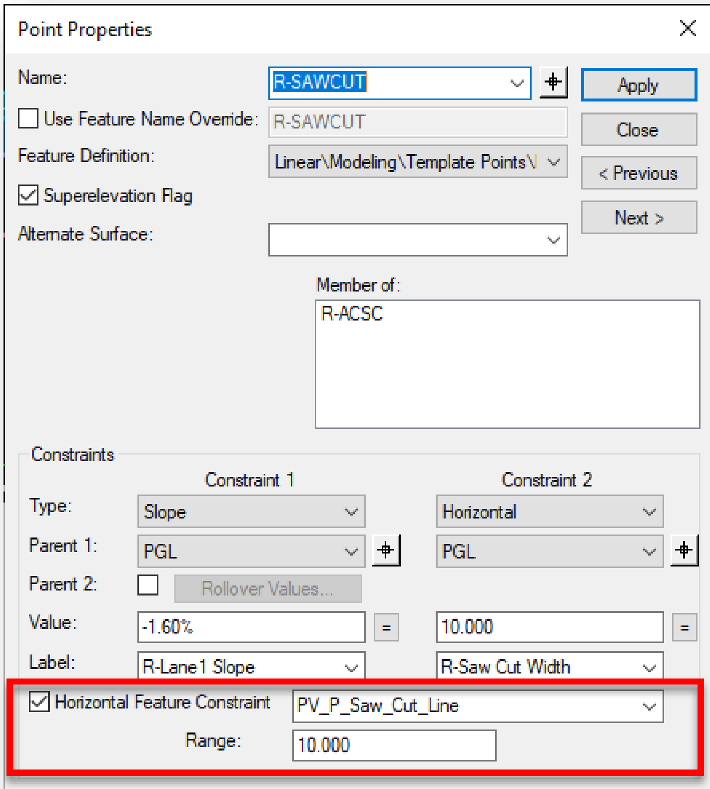


The Overlay and Widening template is configured with the same settings for the overlay/stripping components as the example shown above. The difference is that this template is configured to search for a saw cut line on the left and right sides of the centerline to define the limits of pavement widening.

The right side of the template, showing the R-SAWCUT point is shown below.

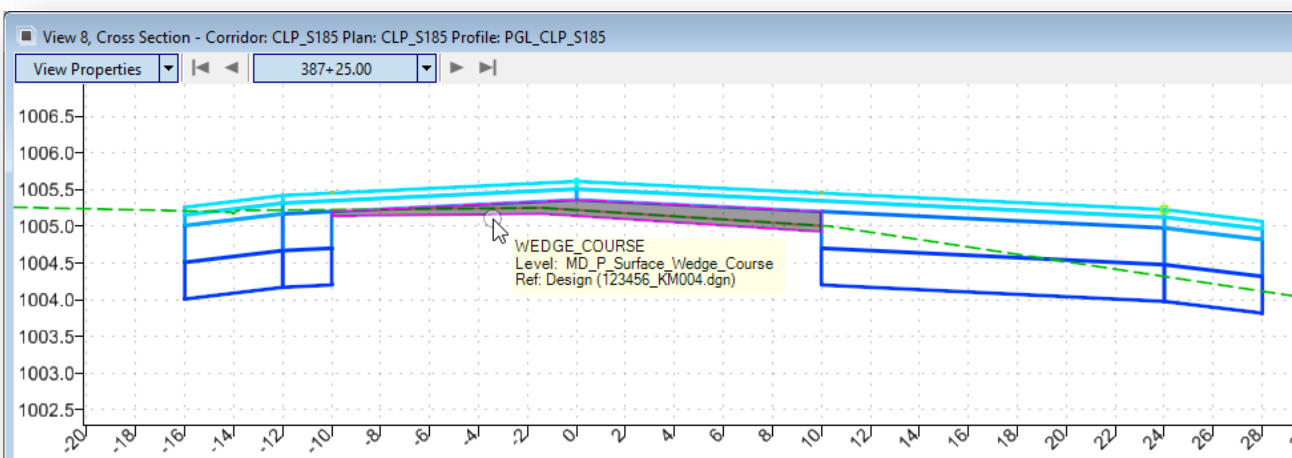


The L-SAWCUT and R-SAWCUT points are defined at a 10’ offset from the center of the template. These points are configured with a **Horizontal Feature Constraint** to search for a graphic defining the saw cut line location in the case that the sawcut is not at a constant offset from the centerline of the alignment. The point properties are shown below.



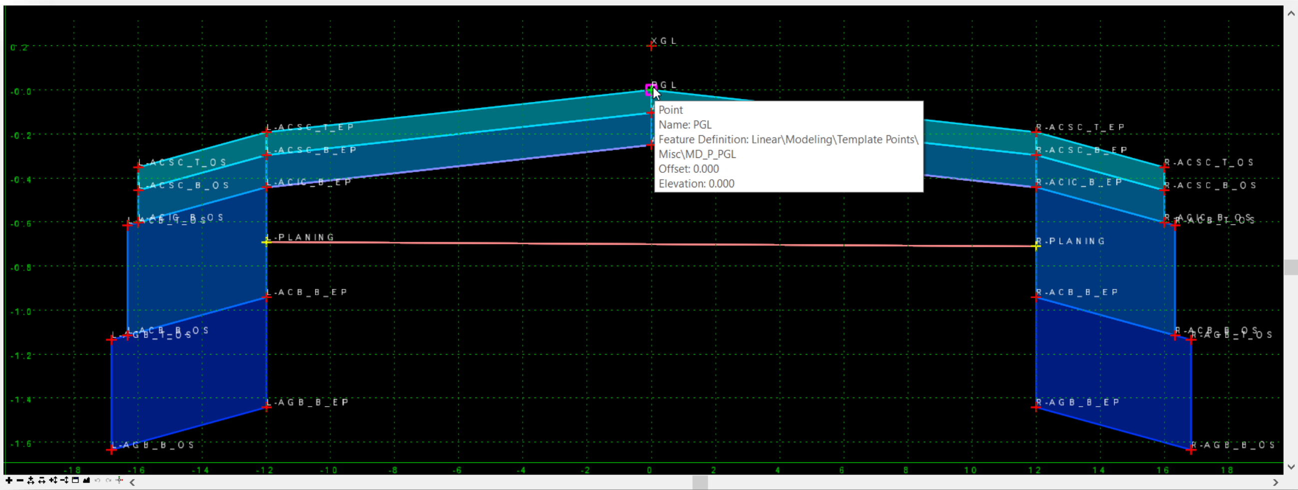
The elements representing the saw cut must be added as corridor references using the **OpenRoads Modeling > Corridors > Corridor References > Add Corridor References** command.

An example of this template, with an additional lane added to the right side, is shown below. The Sawcut and wedge course components are defined like the Overlay template example.



Processing Overlay Templates using the Existing Profile

The examples above were generated using a proposed profile to process the overlay template. The point named "PGL" is at the origin of the template as shown below and is applied to the alignment and profile.



For many projects, the user will want to process a template that overlays the existing pavement using the existing profile rather than a proposed profile. For example, the project may require planing off a portion of the pavement and replacing that with new asphalt pavement layers.

There are many variations for overlay project requirements. Two examples are illustrated below.

Example 1

For this scenario, we will overlay the existing pavement with 3" of new pavement using the 1.6% cross slope for the new pavement, as defined in the template. The following template is used for this example:

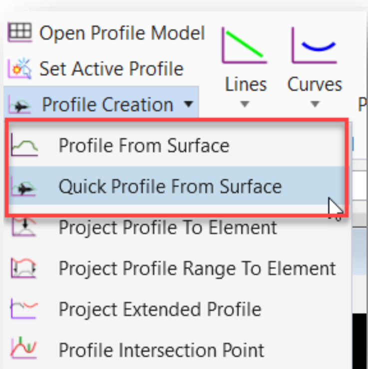
> **ODOT > Backbones > Undivided Roadways > Widening & Resurfacing 301-8E > Overlay > Asphalt Concrete without Stabilization**

The template can be processed as defined using the existing profile to place 3" of new pavement by planing the existing pavement as necessary.

Defining the Existing Ground Profile

The existing ground profile displayed in the profile view is read from the active terrain model. This profile is not a "named" profile that can be used for corridor modeling. It is necessary to generate a profile from the alignment and terrain model that can be used for corridor modeling.

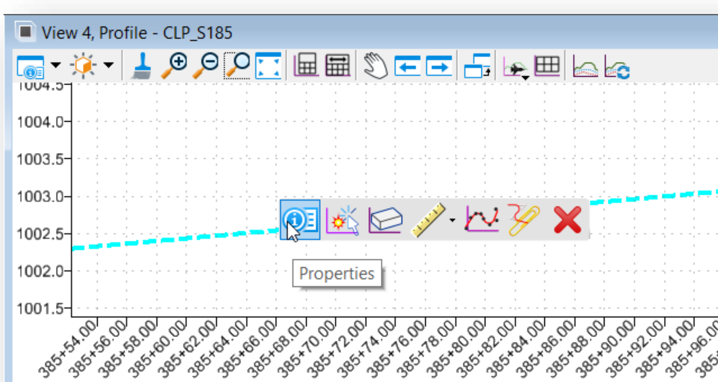
The **Profile From Surface**, or **Quick Profile From Surface** command, located in the **OpenRoads Modeling** workflow, **Geometry** tab, can be used to generate the existing profile.



The **Quick Profile From Surface** command extracts a profile from the selected alignment, using the selected terrain model, for the entire length of the alignment. No additional parameters are available.

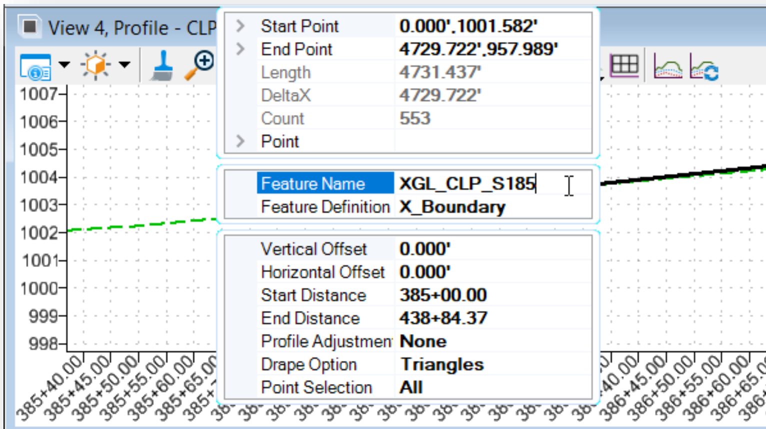
The **Profile From Surface** command allows additional parameters to define starting and ending stations, horizontal and vertical offsets, etc.

In the example below, the **Quick Profile From Surface** command was used to generate an existing profile. The existing profile lies on top of the profile that is displayed from the terrain model.



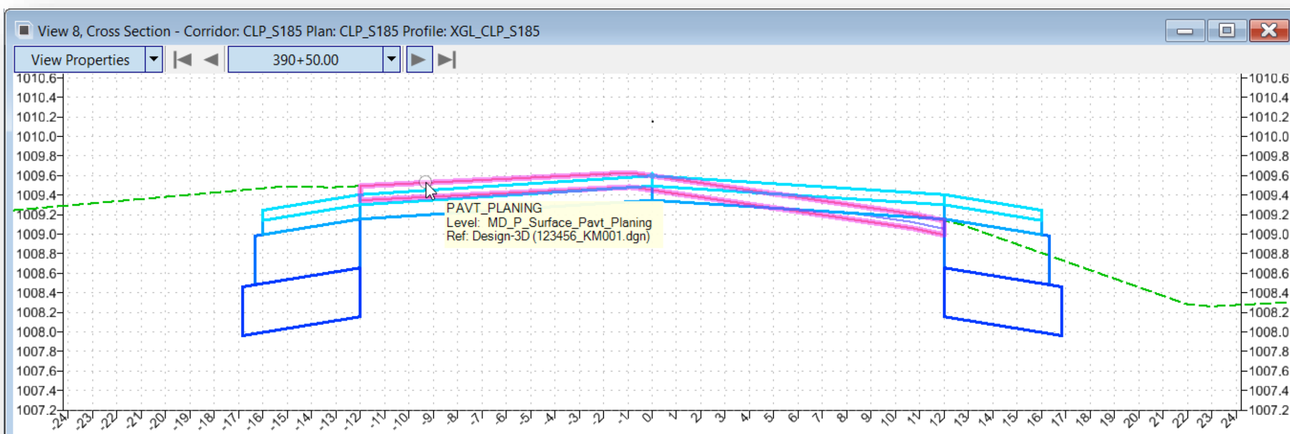
The newly created profile can be named using the **Properties** dialog from the pop-up menu when the profile is selected, as shown at left.

The profile is given a name according to the OHDOT CADD Standards as shown below.

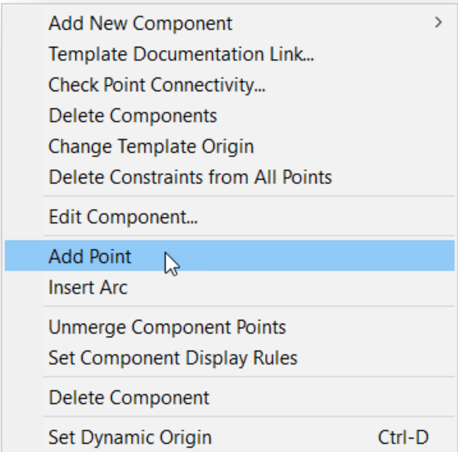


Defining the Pavement Planing Parameters

The template is applied like the example below. Note that the proposed 1.6% pavement slope does not match the existing pavement slope.



The pavement planing follows the existing pavement slope for both the top and bottom depths. This can be adjusted to follow the bottom the proposed pavement by editing the template as follows:



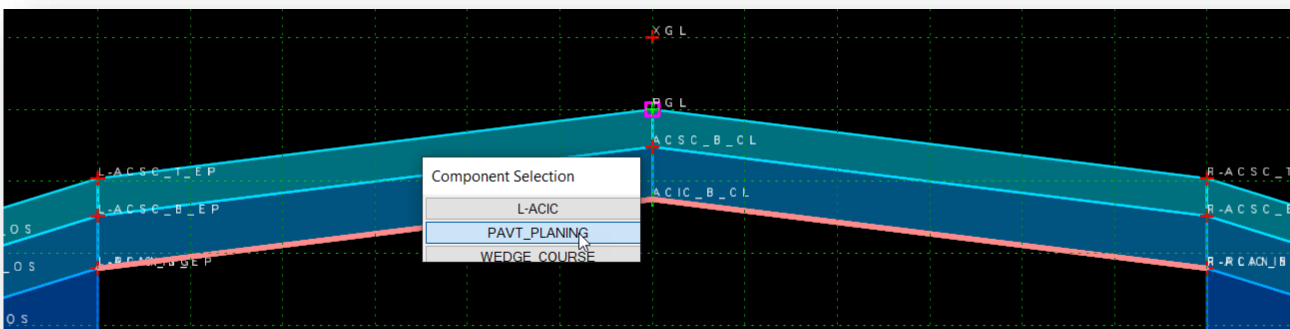
Add a point to the pavement planing component by right-clicking on the component and choosing the **Add Point** command.

Add a point to the component by clicking on top of the point **ACIC_B_CL** point.

Next, move point **L-Planing** on top of point **L-ACIC_B_EP** by right-clicking on point **L-Planing** and choosing the **Move Point** command.

Move point **R-Planing** on top of point **R-ACIC_B_EP** by right-clicking on point **R-Planing** and choosing the **Move Point** command.

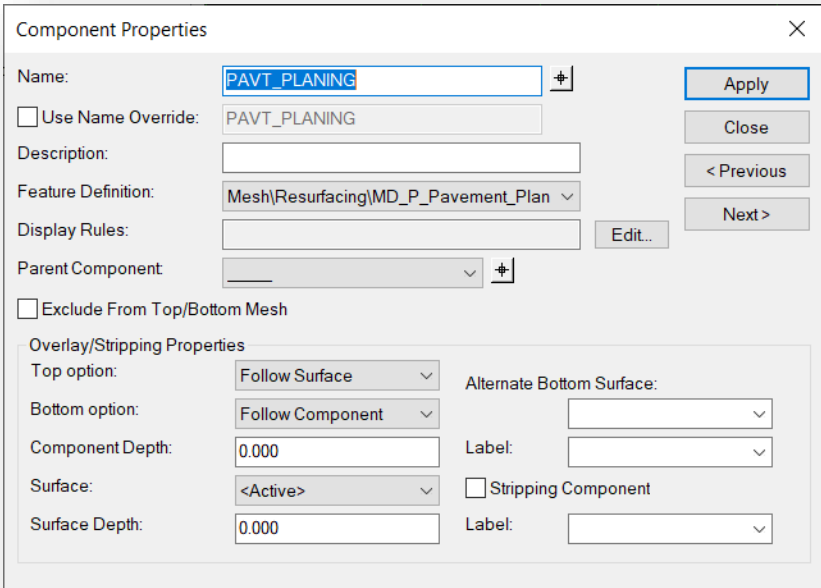
When complete, the **PAVT_PLANING** component now follows the bottom of the proposed pavement as shown below.



Note: There are now two points on top of each other at the left and right point locations. These points can be merged by right-clicking on the points and choosing the **Merge Point** command to remove the **L-PLANING** and **R-PLANING** points

Double-click the bottom of the pavement layers to edit the **PAVT_PLANING** component properties. Three components are defined at this location. Select the **PAVT_PLANING** component from the list to edit the properties.

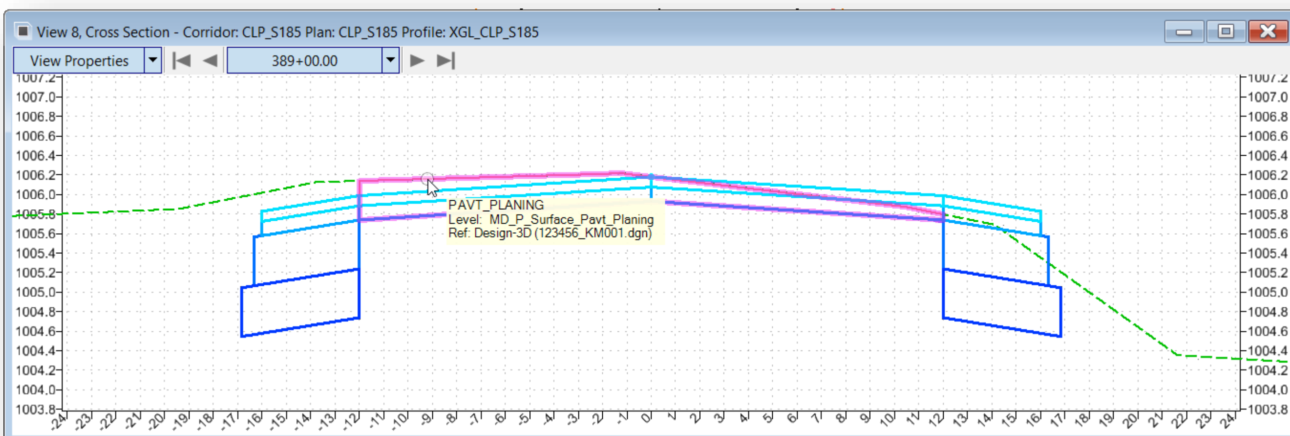
Note: The **WEDGE_COURSE** component is not needed for this example and can be deleted.



Change the **Bottom Option** to **Follow Component**.

Set the **Component Depth** to a value of **0**.

When the template is processed, the pavement planing now follows the top of the existing pavement and the bottom of the proposed pavement like the example below.



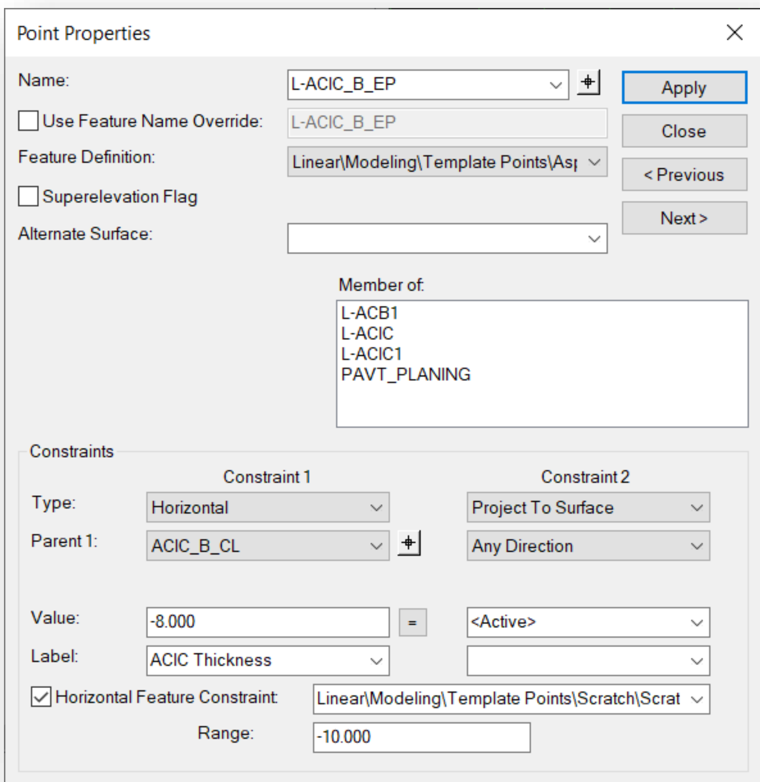
Example 2

For this scenario, we will overlay the existing pavement with 3" of new pavement by matching the existing pavement width and cross-slope. The following template is used as the basis for this example:

> **ODOT > Backbones > Undivided Roadways > Widening & Resurfacing 301-8E > Overlay > Asphalt Concrete without Stabilization**

A few changes need to be made to the template for this example:

1. Edit the constraints for points **L-ACIC_B_EP** and **R-ACIC_B_EP** to define these points so that they will find the existing edge of pavement lines and meet the pavement surface. The properties for the point **L-ACIC_B_EP** are shown below. Point **R-ACIC_B_EP** will be defined similarly, with at an offset of 8' to the right of point **ACIC_B_CL**.

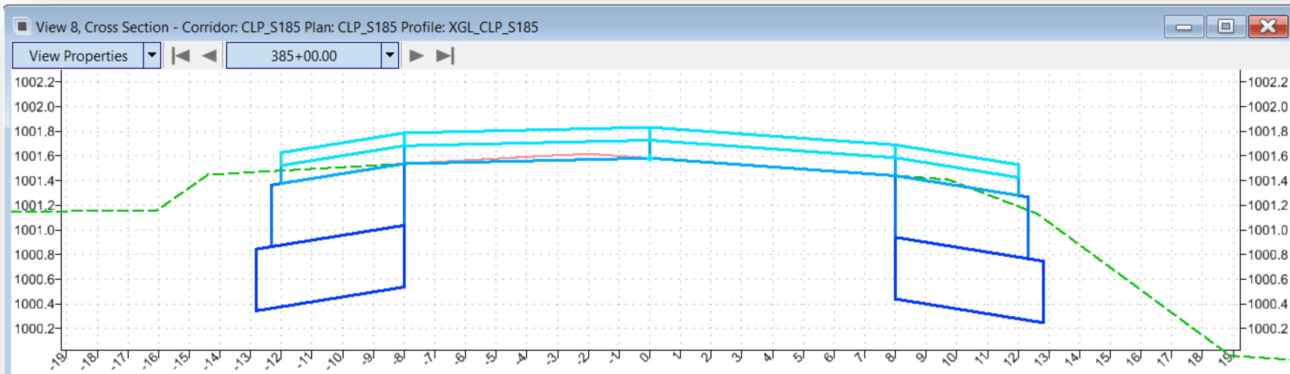


2. Edit the remaining EP points so that the pavement buildup is defined from the bottom-up instead of the top-down.

3. Delete the **WEDGE_COURSE** component. It is not needed for this example.

4. Modify the **PAVEMENT_PLANING** component to follow the bottom of the proposed pavement with a **Component Depth** value of **0**.

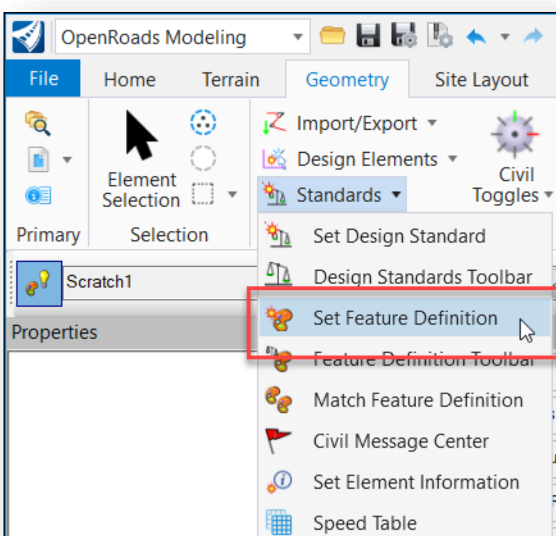
When the edited template is processed, the pavement slopes meet the existing terrain like the example below.



The left and right edge of pavement lines are defined with an offset of 12’ and do not meet the existing edge of pavement lines from the survey data. The **Horizontal Feature Constraint** parameter, which is defined in the template ***Point Properties*** dialog, can be used to set points **L-ACIC_B_EP** and **R-ACIC_B_EP** to meet the existing edge of pavement lines will be define the **Feature Definition** of the edge of pavement element.

With this release of the software, the **Horizontal Feature Constraint** parameter does not allow selecting Survey features. The surveyed edge of pavement lines must be copied from the survey design file to the modeling design file and then assigned a new feature definition that can be used for the modeling process.

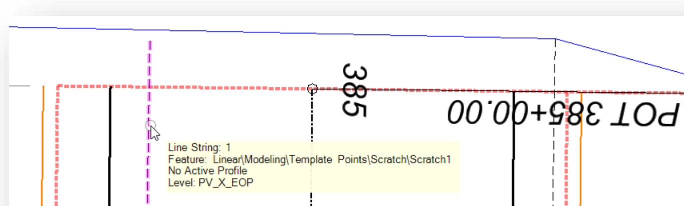
After copying the existing edge of pavement lines from the survey design file to the modeling design file, set the feature definition for the newly copied elements by using the **OpenRoads Modeling > Set Feature Definition** command as shown below.

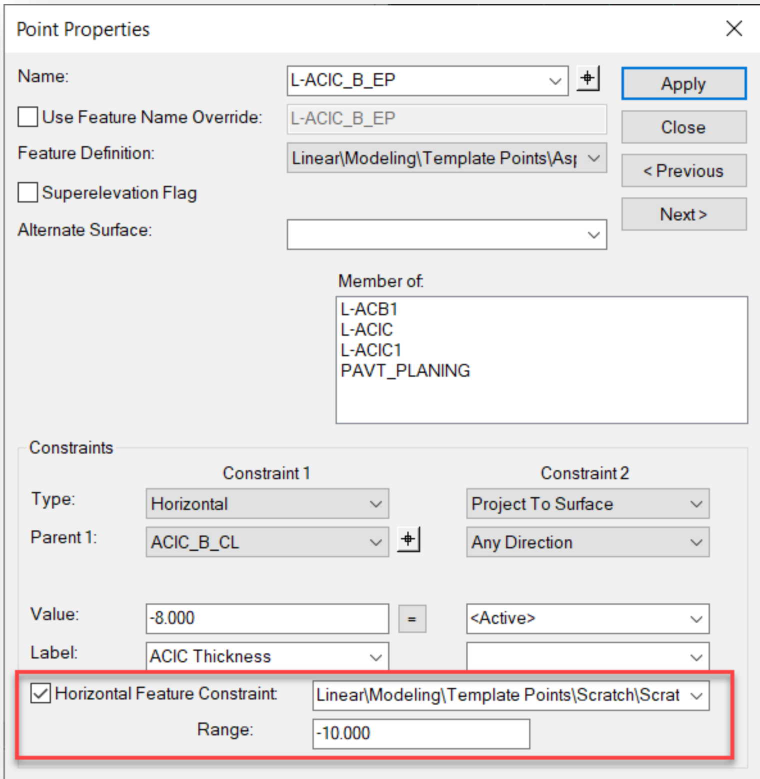


The feature **Linear > Modeling > Template Points > Scratch > Scratch1** is used in this example.

Select each copied edge of pavement element to assign the new feature definition.

The feature definition is assigned like the example below.

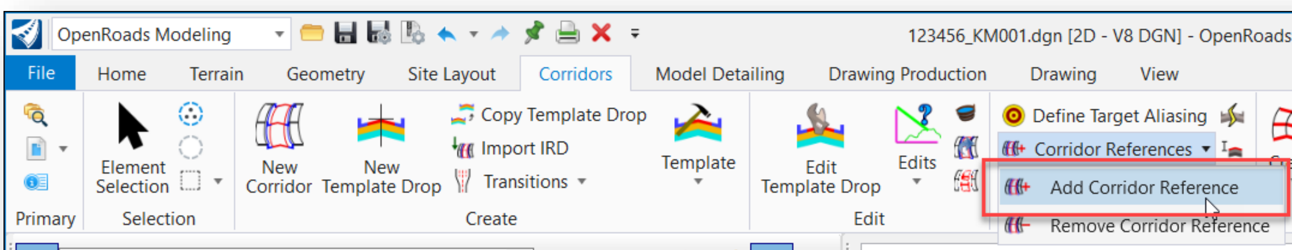




In the ***Create Template*** dialog, edit the point properties for points **L-ACIC_B_EP** and **R-ACIC_B_EP** to assign the **Horizontal Feature Constraint** parameter using the **Scratch1** feature like the example at left.

Note: The **Horizontal** offset value of **8'** is used to position the pavement points inside the limits of the existing pavement. The software will search from the location of the points, outward, for the intersecting elements. If the template points are already wider than the existing pavement the process will not find the existing pavement lines.

The edge of pavement elements must be added as a corridor reference for the modeling process to recognize the lines. Choose the **OpenRoads Modeling > Corridors > Add Corridor Reference** command.



- When prompted to locate the corridor, choose one of the corridor handle graphics to identify the corridor.
- Next choose the edge of pavement elements to add them as corridor references

Section 600 Right-of-Way

Right-of-Way Basemap Design Files

ODOT's CADD Standards detail the use of basemap design files to draw plan, profile, and cross section information for your project in real world dimensions. These drawings are then referenced into sheet design files to produce the plan sheets for the project.

Several basemap files may be used depending on the complexity of the project. The ODOT CADD Standards allow for multiple basemap design files for each unique discipline (Drainage, Lighting, MOT, Right-of-Way, Roadway, Signals, etc.)

The **OHDOT Create Design Files** application is used to create new design files using the seed files for the project.

Right-of-Way information can be placed in a single design file, or multiple depending on the size of the project and the number of people needing access to edit the information. For most projects, as a minimum, the use of two right-of-way basemap design files is recommended:

File Name	Purpose
400-Engineering\RW\Basemaps\######_BR001.dgn	Existing right-of-way information
400-Engineering\RW\Basemaps\######_BR002.dgn	Proposed right-of-way information

See the **ODOT CADD Engineering Standards Manual (OHDOT)**, **Section 300 File Management** for more details.

The right-of-way information is often generated by ODOT's survey personnel. all right-of-way information is stored in the 400-Engineering\RW\Basemaps\ folder regardless of who generates the information.

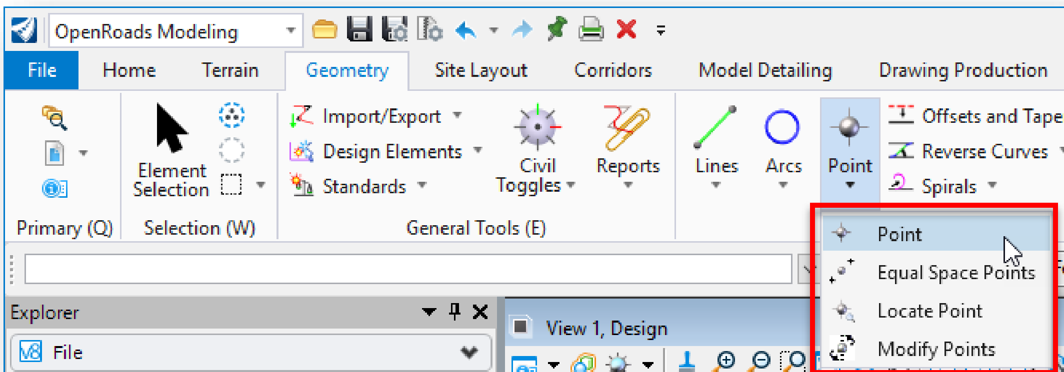
Right-of-Way Feature Definitions

The OHDOT WorkSpace contains several feature definitions intended for right-of-way areas, such as the parent tract area, or taking areas, as well as features for the right-of-way lines that are shown on the right-of-way plans.

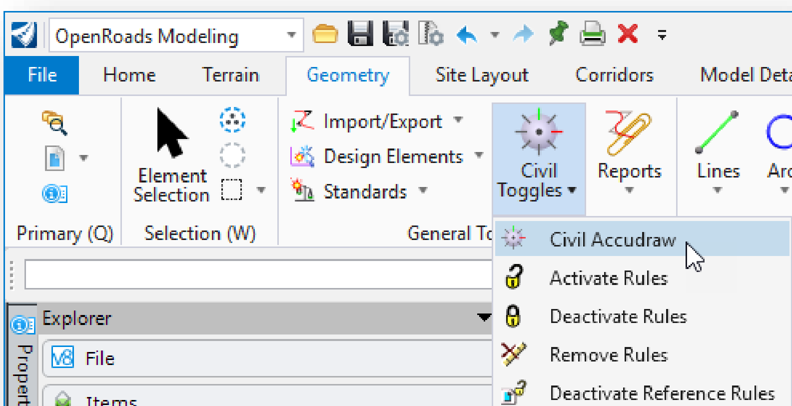
Point Features

Several features are available for storing right-of-way points in the **\Points\Control Points and Monuments** folder.

These features are used with the **Geometry > Point** commands, selected from the **OpenRoads Modeling** or the **Survey** Workflow.

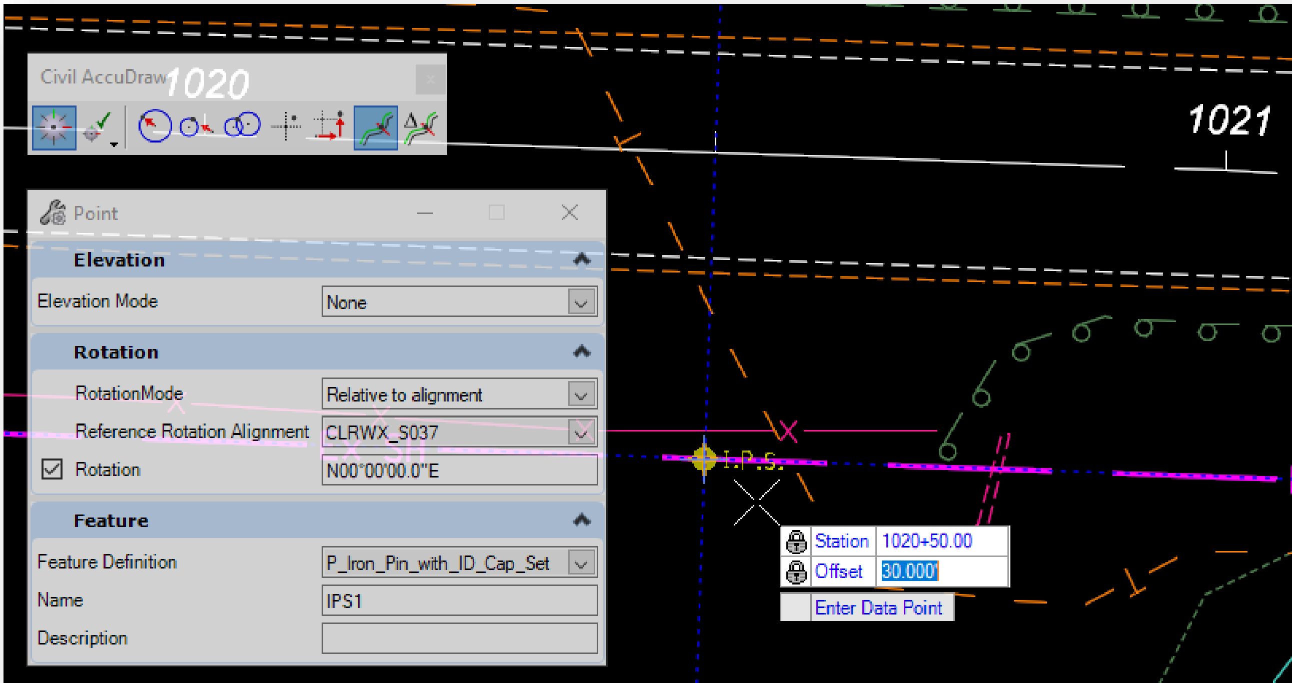


Civil Accudraw is used to place points at specific coordinates or at a station and offset location from a selected OpenRoads Alignment.

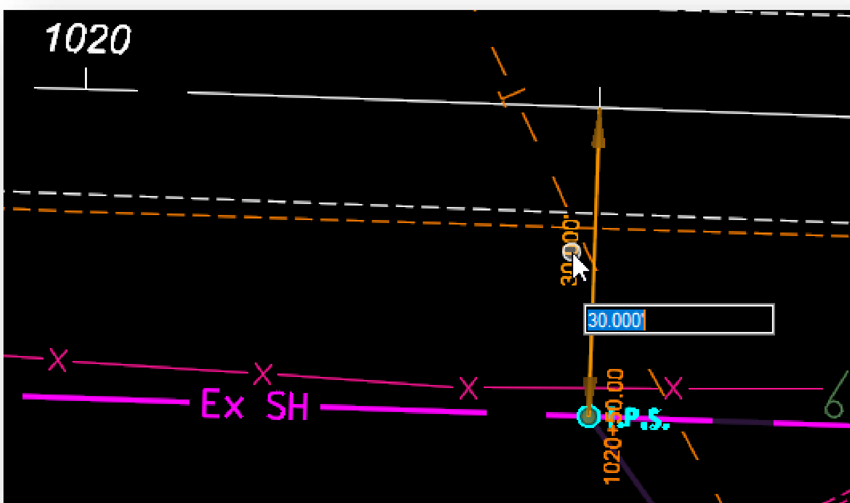


See the ***OHDOT_ORD_Design_303_CivilAccudraw*** guide for more information using Civil Accudraw.

An example placing an Iron Pin with Civil Accudraw is shown below.



The point element is placed with rules for the station and offset value. The rules can be edited by selecting the element and then clicking on one of the rules as shown below.



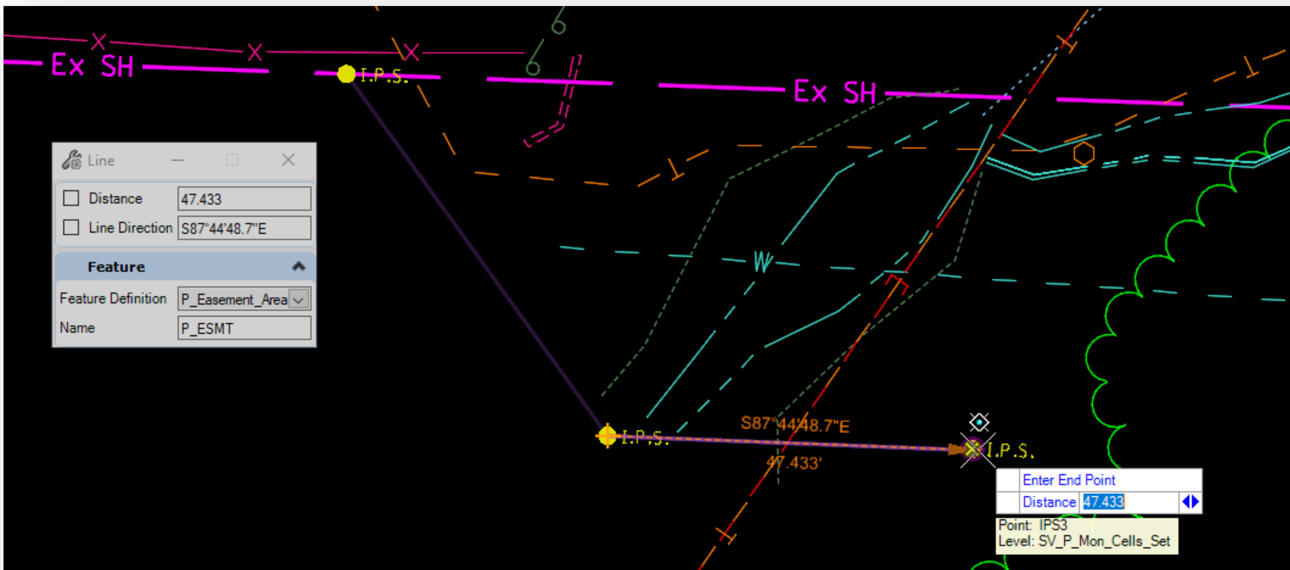
The rules for the selected point can also be edited in the **Point Constraints** section of the ***Properties*** dialog.

Parcel Area Features

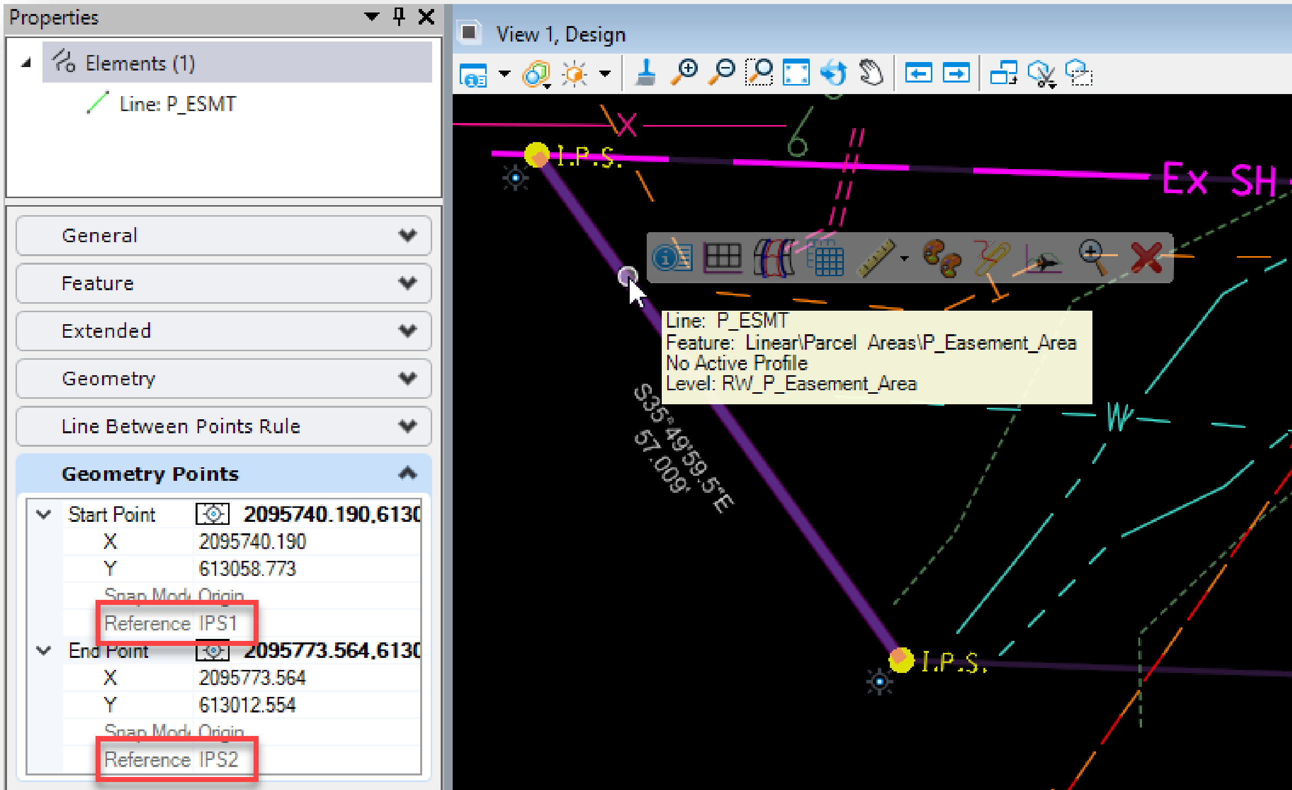
The parcel area features are used when defining the closed geometry that makes up a specific area, such as an easement or taking area. The graphics drawn for these areas are not intended to be shown on the final plans. Areas created using these features are intended to be used for computation purposes. These features are in the **\Linear\Parcel Areas** folder in the OHDOT Standards.

Areas can be placed using the OpenRoads Geometry tools to place lines and arcs. After placing the elements that define the closed area, the Complex Geometry command is used to create the closed shape.

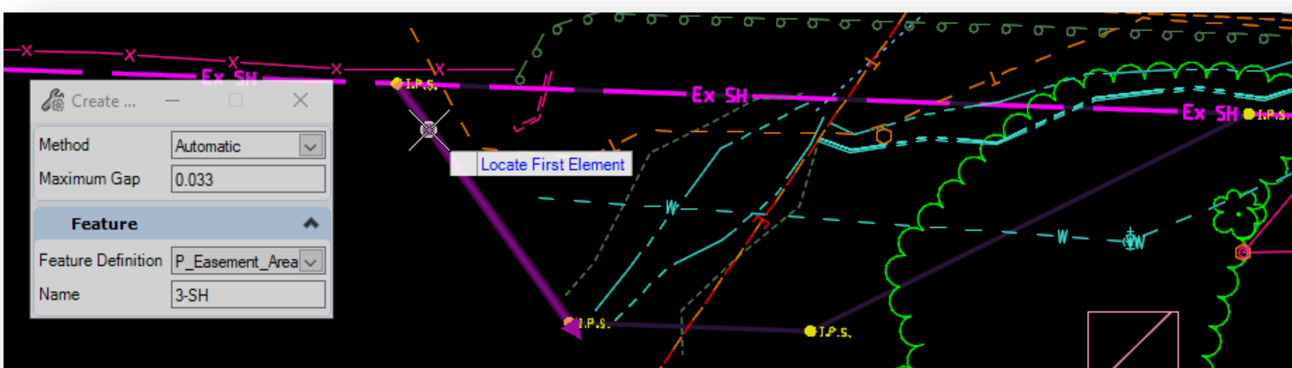
In the example below, the **Geometry > Lines > Line Between Points** command is used to place the lines defining the parcel area by snapping to previously placed geometry points. The MicroStation **Origin** snap is used to snap to the center of the geometry points.



Note: If the **Origin** snap is used to snap to the points when placing the curvilinear elements, the point name is associated with the endpoints of the curvilinear element. The ***Properties*** dialog shows the name of the points associated with the endpoints of the selected element as shown in the example below.



The **Geometry > Complex Geometry > Complex By Element** command is used to create the closed shape for the area. In the example below, an easement area is created. Note that the direction for the closed area, when using the **Automatic Method**, is indicated by the arrow drawn at the end of the first element selected, at the opposite end from the end closest to the location where the element is selected.



The closed geometry element is intended to be used to generate parcel areas and is not shown on the completed plan. The graphics that are placed when using these Parcel Area feature definitions are placed as construction class elements. These closed areas are useful to generate parcel reports.

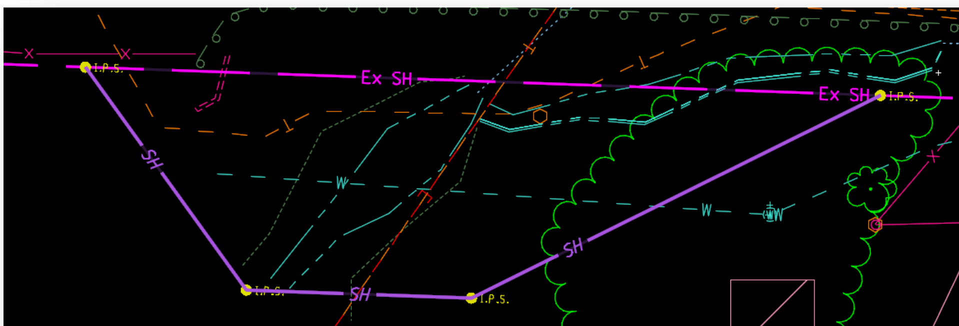
Right-of-Way Linear Features

Features for right-of-way and lines are in the **\Linear\Right-of-Way** folder in the ODOT Standards. These features are used to draw the right-of-way lines that will be shown in the right-of-way plans.

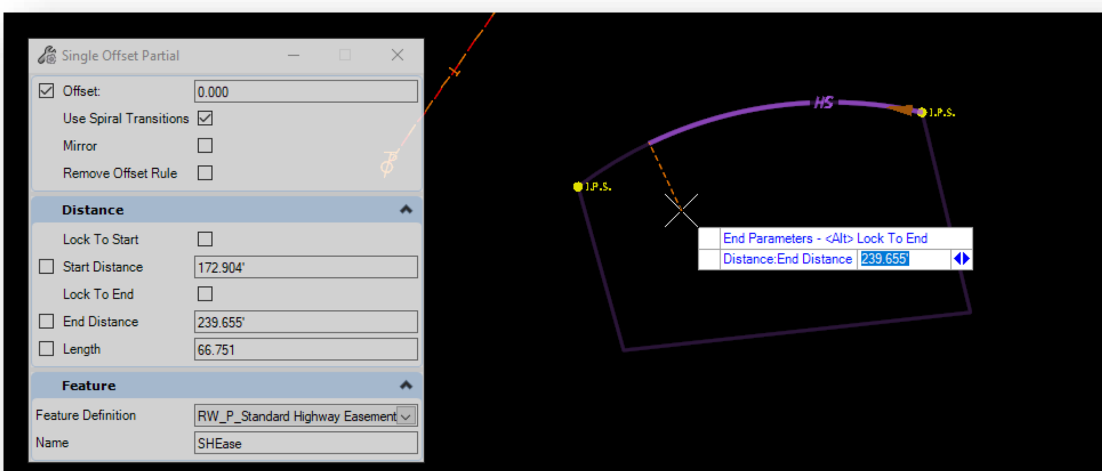
The right-of-way lines can be drawn by using the **Origin Snap** to snap to previously placed geometry points or by using the **Keypoint Snap** to snap to the corners of previously placed curvilinear elements used to define a closed parcel area.

If these lines are drawn by snapping to the underlying parcel element or the geometry points, a rule is created to link the lines to the underlying graphics. The right-of-way lines will respond to changes made to the underlying graphics.

In the example below, the feature definition **Linear\Right-of-Way\Easaements\RW_P_Standard Highway Easement** was used to draw the easement lines that will be shown on the right-of-way plans. The level for the underlying parcel area element can be turned off.



Tip: The **Geometry > Offsets and Tapers > Single Offset Partial** command can also be used to easily draw right-of-way lines along curves like the example below. **Note:** The element is drawn in the direction that the parcel area was stored. In the example below, the area was stored in the counterclockwise direction.

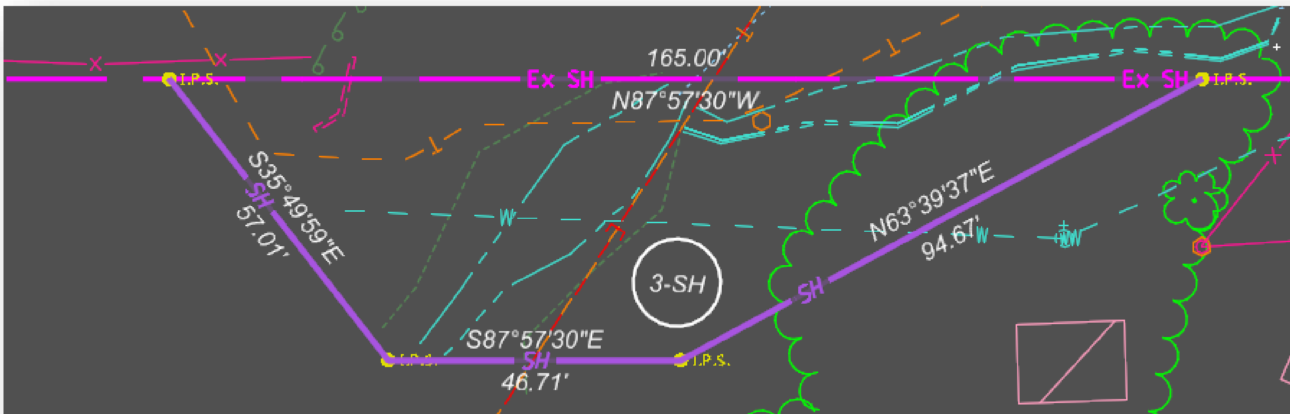


Parcel Annotation

The feature definitions for parcel areas have annotation definitions defined. The **Drawing Production > Element Annotation > Annotate Element** tool is used to annotate selected parcel area elements.

Parcel annotation is typically placed in the drawing model of the sheet design file. The process to clip sheets is detailed in the **OHDOT_Plan_and_Profile_Sheets** document.

In the example below, the closed element representing the easement area was annotated in the drawing model using the **Drawing Production > Element Annotation > Annotate Element** tool. The bearings, lengths, and the parcel name are annotated.



Annotation graphics can be moved or deleted as needed.

Note that if multiple areas adjacent to one another are annotated the bearing and length labels will be placed for each component of the complex element. This will result in overlapping labels that the user can manually delete.

If the parcel area includes an arc element, the radius, length, chord bearing, and chord length are annotated.

If the element is annotated again, previous annotations are left in place. The **Drawing Production > Element Annotation > Remove Element Annotations** command is used to remove previous annotations.

All annotations are drawn using the active Annotation Scale value.

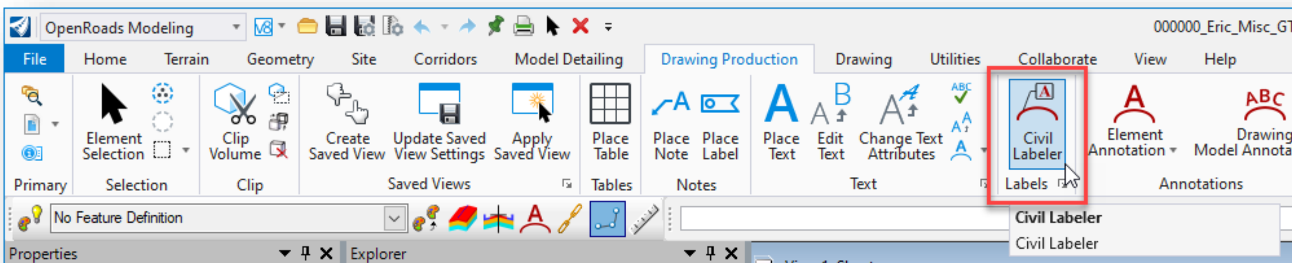
Parcel Labels

ORD supports placing labels that have fields computed from the selected geometry graphic or data point location. The label maintains an association to the element that is being labeled and is automatically updated if changes are made to the element.

Two separate tools that may be used for placing labels are documented below. The new **Civil Labeler** tool was added to the 2021 Release 1 version of ORD, Version 10.10, and is the preferred labeling tool moving forward. The older **Place Label** tool is still valid and may be used if desired.

Civil Labeler

A new annotation tool, **Civil Labeler**, has been added to the 2021 Release 1 version of ORD. As of this writing, the function is in a "tech preview" status as the tool is refined. The Civil Labeler can be accessed from the **OpenRoads Modeling** WorkFlow by selecting **Drawing Production > Labels > Civil Labeler** from the ORD Ribbon menu.



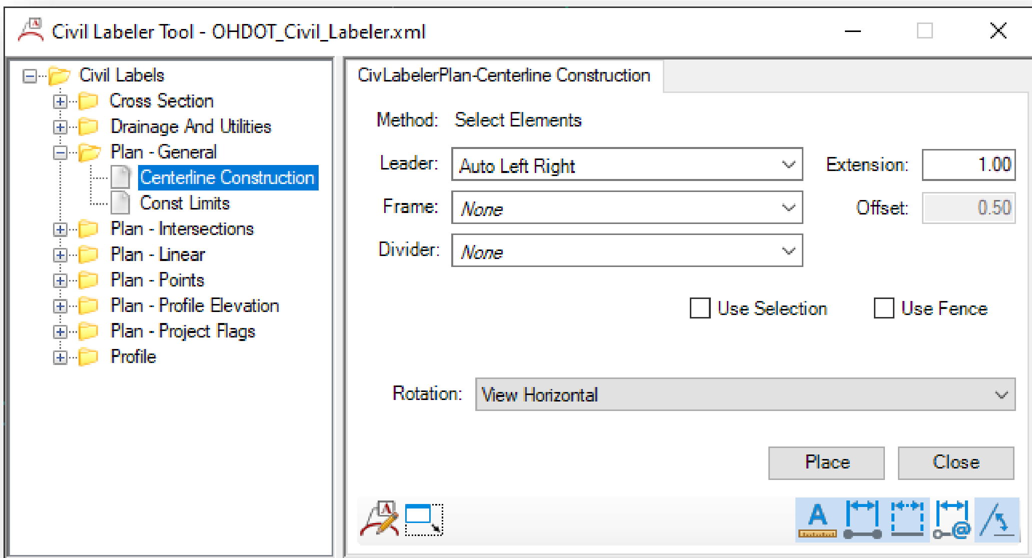
When the command is selected, the ***Civil Labeler Tool*** dialog is opened with the **OHDOT_Civil_Labeler.xml** file loaded as shown below.

The xml file is configured with multiple folders containing multiple label definitions as shown above. Each folder contains multiple labels. It is beyond the scope of this training guide to document each label option. A few of the common labels will be documented below. Contact the CADD and Mapping Support team with questions or requests to add additional labeling options.

Cross Section and Drainage and Utilities labels will not be covered in this document.

ODOT recommends placing most labels in the various Drawing models for the sheets. Labels should not be placed in the sheet model.

When a label is selected, the dialog presents several options to the user like the example below.



It is beyond the scope of this manual to document every option. The ODOT labels have been configured with the default options most used for each label type. Examples of commonly used labels are included below.

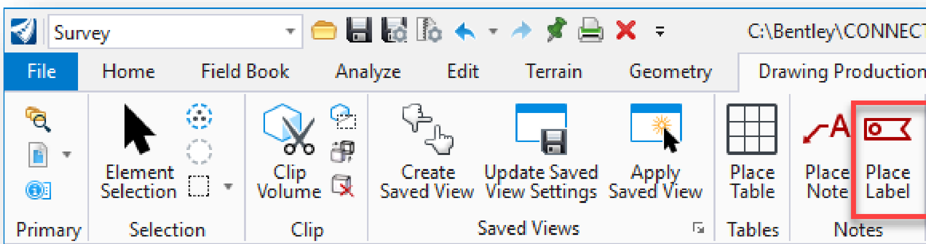
For more information, see the OHDOT_ORD_Design_800_Plan_and_Profile_Sheets training guide.

Place Label

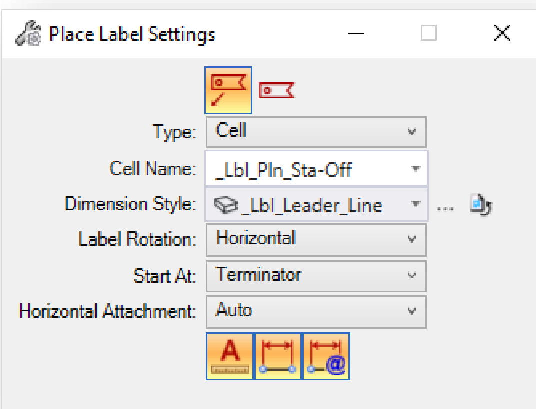
With a 2021 Release 1 version of OpenRoads Designer 10.10, ODOT recommends using the new **Civil Labeler** tools. This section is retained in the document for reference.

A label can maintain two types of associations. The first is regular association, in which if you move the element, the label remains at the same place, only the leader line moves along with the element. The second is relative association, in which the label will stay at the same location relative to the original snap point. That is, if you move the element the label will also move with it.

Several labels have been configured in the OHDOT standards for various types of labels. These labels are placed using the **Drawing Production > Notes > Place Label** command.



The **Place Label Settings** parameters are displayed in the **Tool Settings** dialog as shown below.



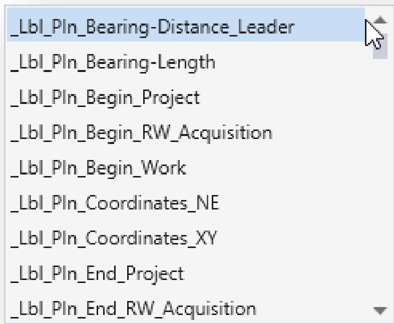
Labels can be placed with or without a leader line by selecting the corresponding icon at the top of the dialog.

The dialog has the following parameters:

Type

The **Type** option is used to place labels using a **Cell** or a **Text Favorite**. The cell option allows for labels that contain multiple computed annotations. When the cell option is selected, the dialog allows selection of the label from a cell library.

If **Cell** is selected, the **Cell Name** parameter lists all the cells found in the libraries defined by the configuration variable MS_CELLLIST. The labels that have been defined in the OHDOT standards are named with the prefix **_Lbl_** so that they will be listed first in the dialog. A partial list of the cells in the OHDOT WorkSpace that have been defined as labels is shown below.



If the **Type** parameter is set to **Text Favorite**, the **Favorite Name** item allows selection of a predefined text favorites.

In general, the **Cell** option is used for placing labels as they have been defined in the OHDOT WorkSpace. The text favorites are used as a component of the Cell labels, or other annotation functions within ORD such as the station annotation or the curve data. There may be occasions when using one of the text favorites directly is desired.

Dimension Style

The **Dimension Style** parameter us used to select a dimension style from the OHDOT standards that will be used to set the symbology settings for leader lines. Two options are provided in the OHDOT Standards that are used for the labeling process:

_Lbl_Leader_Line

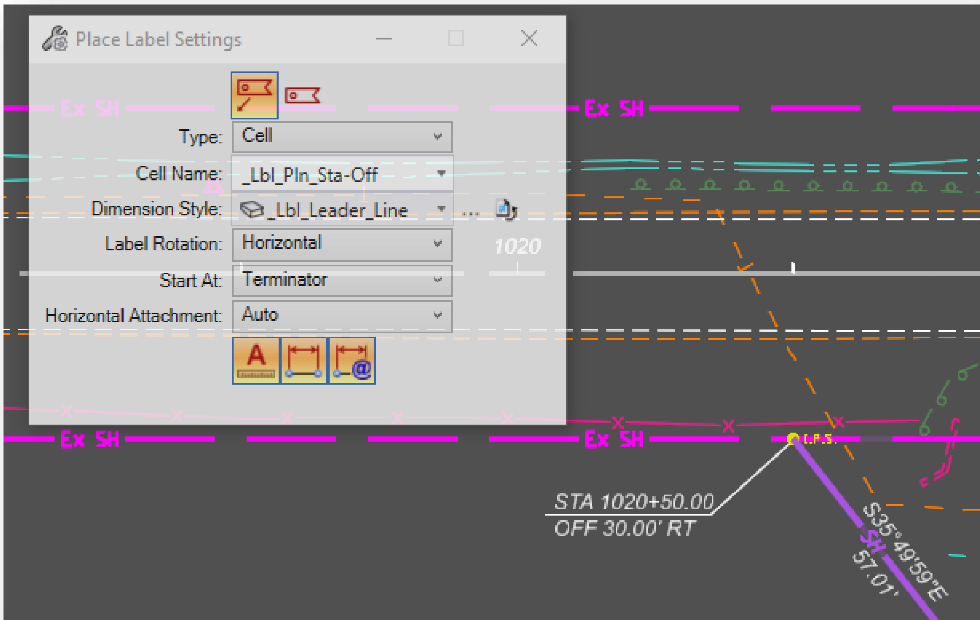
_Lbl_Leader_Line_With_Terminator

See the online documentation for a description of the remaining parameters.

Station and Offset Labels

Labels that have been defined using a station or offset text favorite require that the alignment is selected before the label is placed.

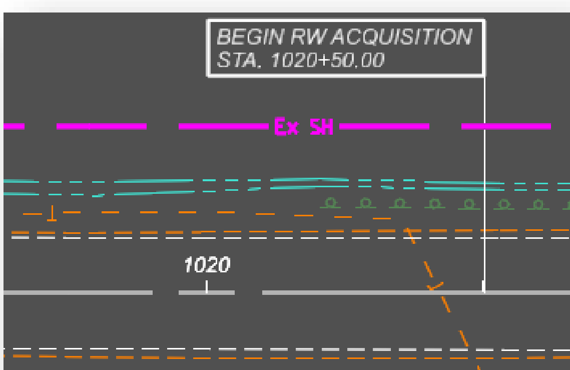
In the example below, the **_Lbl_Pln_Sta-Off** cell is selected.



The user is prompted to **Identify Element or Data Point** in the prompt field at the lower left corner of the ORD interface. Take the following steps:

- Select the alignment for the station and offset reference
- Select the location for the label. The **Start At** parameter is set to define whether the data point is at the **Terminator,** or the **Placement** point for the label.
- Additional labels can be placed without the need to re-identify the alignment

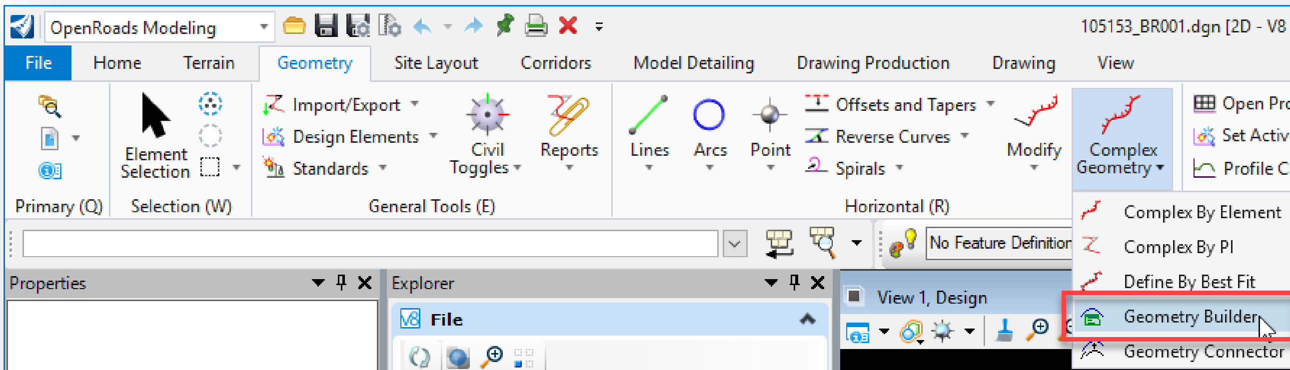
Additional label types have been defined for Right-of-Way plans. See the **Cell Name** drop-down list to review the available labels. The **_Lbl_Pln_Begin_RW_Acquisition** label is shown below.



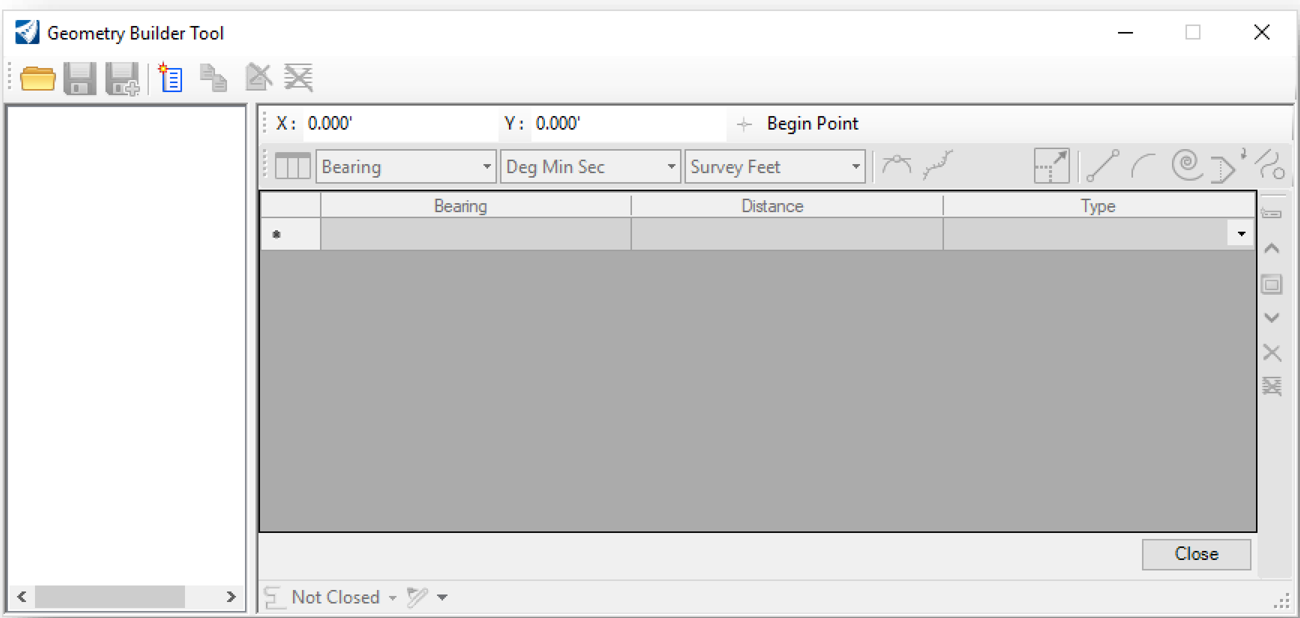
Geometry Builder

There are multiple ways to create a parcel in ORD. The **Geometry Builder** tool is specifically designed for creating and editing parcel elements.

Select **Geometry > Complex Geometry > Geometry Builder** from the **OpenRoads Modeling** workflow as shown below.



The ***Geometry Builder Tool*** dialog is opened as shown below.

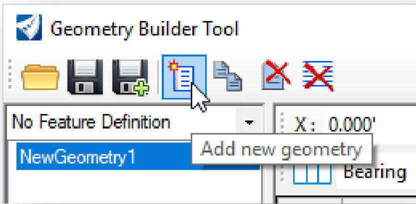


See the Bentley online help for options not covered in this document.

Creating Parcel Elements

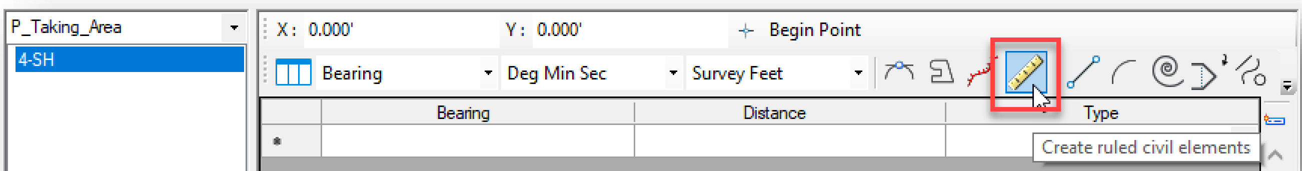
The steps to create a closed parcel area are summarized below.

- Select the **Add new geometry** button

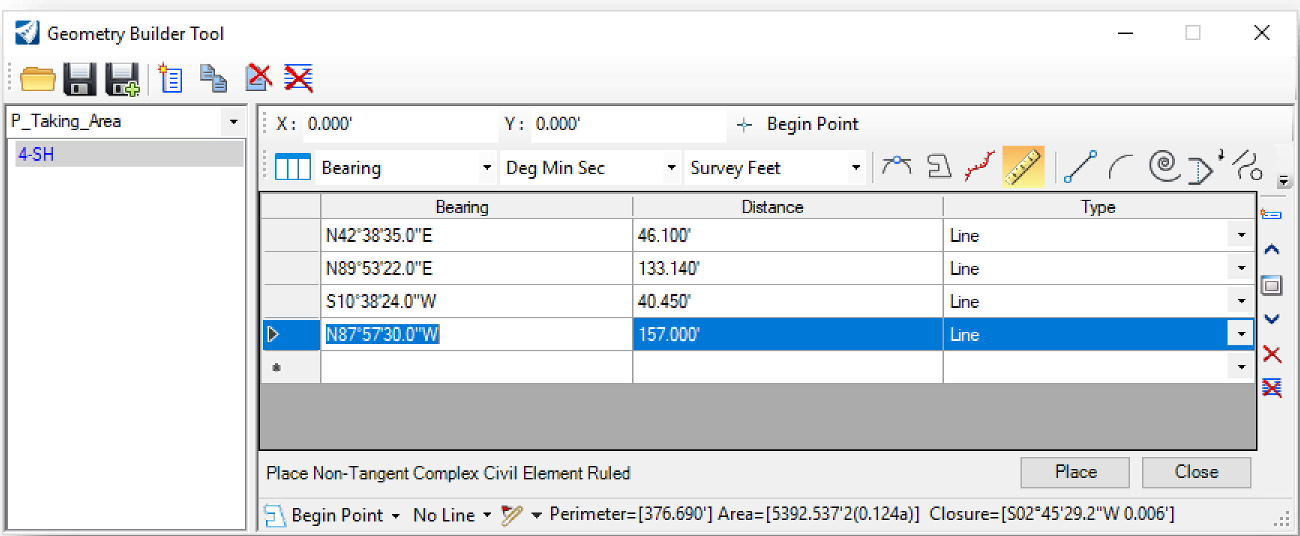


A geometry definition named **NewGeometry1** is created as shown at left.

- Choose the feature definition from the drop-down list. By default, No **Feature Definition** is defined as shown above.
- Edit the name of the geometry element by selecting the name, and then left-clicking on the name a second time once it has been selected.
- Toggle on the **Create ruled civil elements** option



- Enter the data to create the parcel area. An example is shown below.

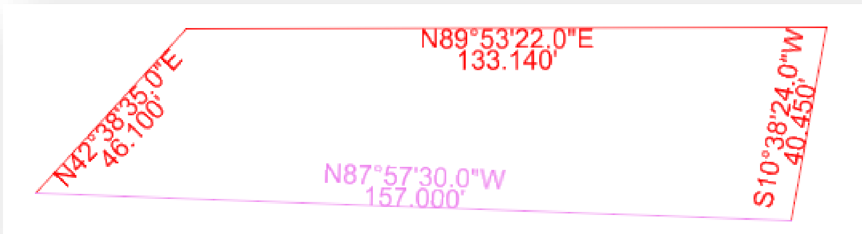


Information about the parcel, such as the perimeter, area, and closure of the shape, is displayed at the bottom of the geometry builder dialog.

- Select the **Begin Point** icon, or key-in the **(X, Y)** coordinates for the start of the parcel location.

  When the **Begin Point** icon is selected, the parcel graphics will drag on the cursor until a data point location is defined.

  The graphics are displayed in the file as temporary graphics like the example below. Changes to the parcel definition are dynamically reflected in the temporary graphics.



- Select the **Place** button to write the graphics to the design file. The parcel element is placed in the design file using the symbology defined by the selected feature definition.

Saving Parcel Definitions to an XML File

After the geometry has been defined, it is good practice to save an XML file with the geometry data.

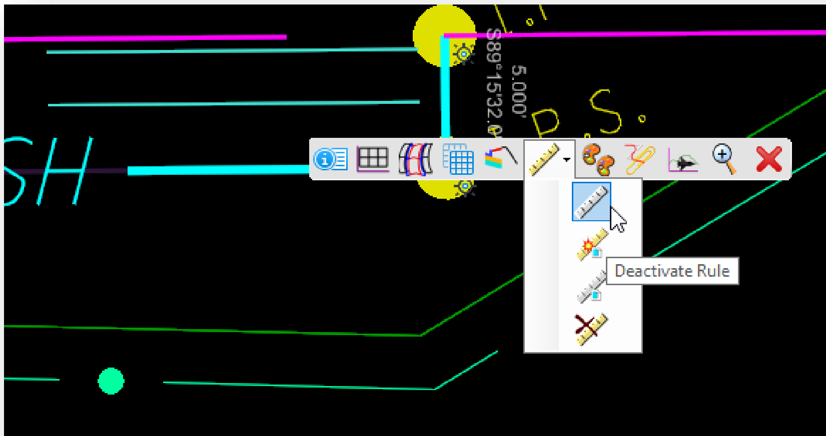
- Choose the **Save** icon at the top left of the ***Geometry Builder*** dialog.

  Note: In this release, the software does not allow saving the file directly in ProjectWise. Save the file to the desktop and then copy the file into ProjectWise in the **\400-Engineering\RW\EngData** folder for the WorkSet.

Previously saved XML files can be re-opened for editing using the **Open File** icon.

Locking Right-of-Way Elements

Parcel areas and right-of-way curvilinear elements can be locked by deactivating the rules. The rules for selected elements can be deactivated by selecting the element and choosing the **Deactivate Rules** command from the pop-up menu as shown below.

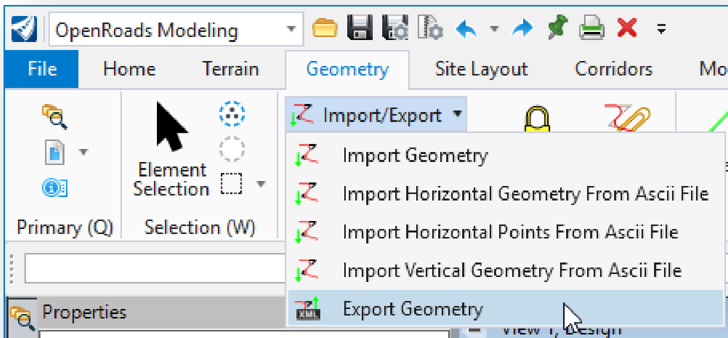


Alternately, the rules can be deactivated by choosing the **Geometry > Civil Toggles > Deactivate Rules** command and then selecting the element(s) to deactivate the rules.

Once deactivated, the element is locked and cannot be edited or deleted. Use the **Activate Rules** command, which can be selected from the pop-up menu or by choosing **Geometry > Civil Toggles > Activate Rules**, to reactivate the rules to allow editing or deleting the element.

Export Right-of-Way Geometry to XML

XML files can be saved with the parcel element definition as a back-up to the parcel data. This is useful to recreate the parcel should the data be inadvertently modified or deleted. To save an XML file for a parcel element, select the **Geometry > Import/Export > Export Geometry** command.



When prompted, choose the element that you wish to export to an XML file.

Save the XML file in the **\400-Engineering\RW\EngData** folder. Name the XML file using the name of the parcel element.

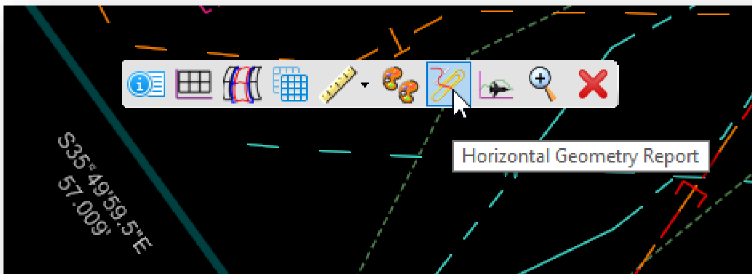
The **Geometry > Import/Export > Import Geometry** command is used to import the XML file if it is necessary to recreate the parcel element.

Reports

Various reports are provided for parcel areas and points.

Horizontal Geometry Report

Select a parcel area and let the cursor rest on the element until the pop-up menu appears. The **Horizontal Geometry Report** option can be used to generate a report for the area.

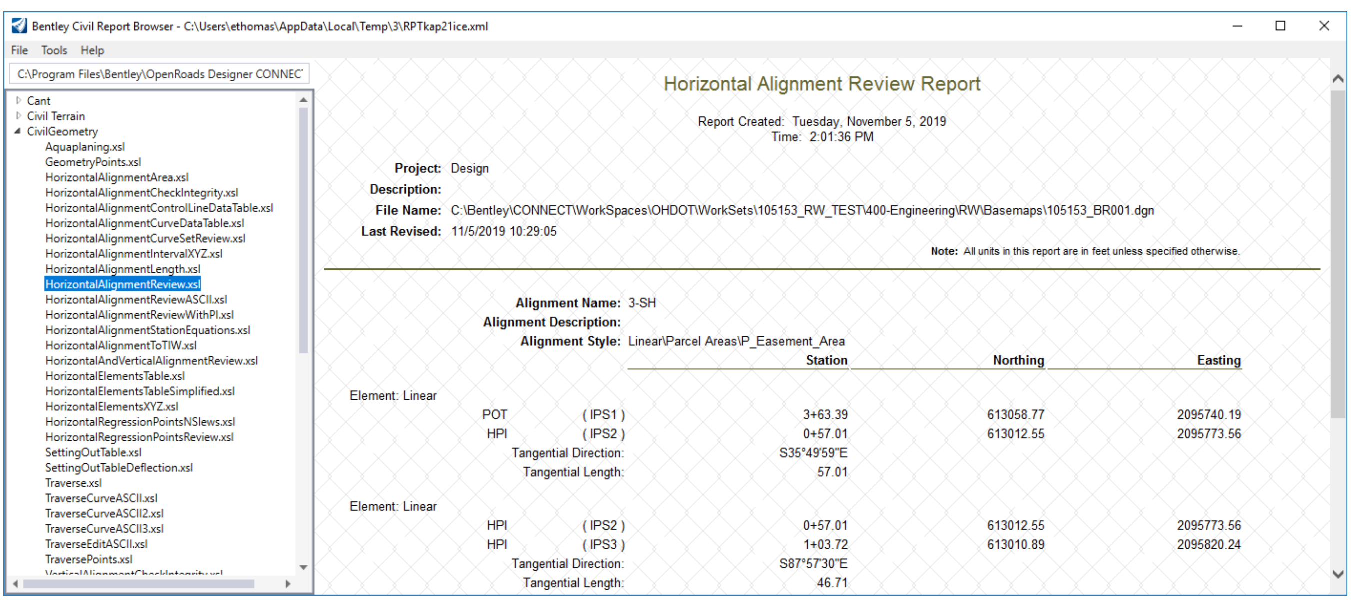


When the report icon is selected from the pop-up menu, the ***Report Browser*** dialog is opened with the default **Horizontal Alignment Review Report** selected.

In the ***Report Browser*** dialog, the **LegalDescription** and **Map Check** folders contain various reports for parcel areas.

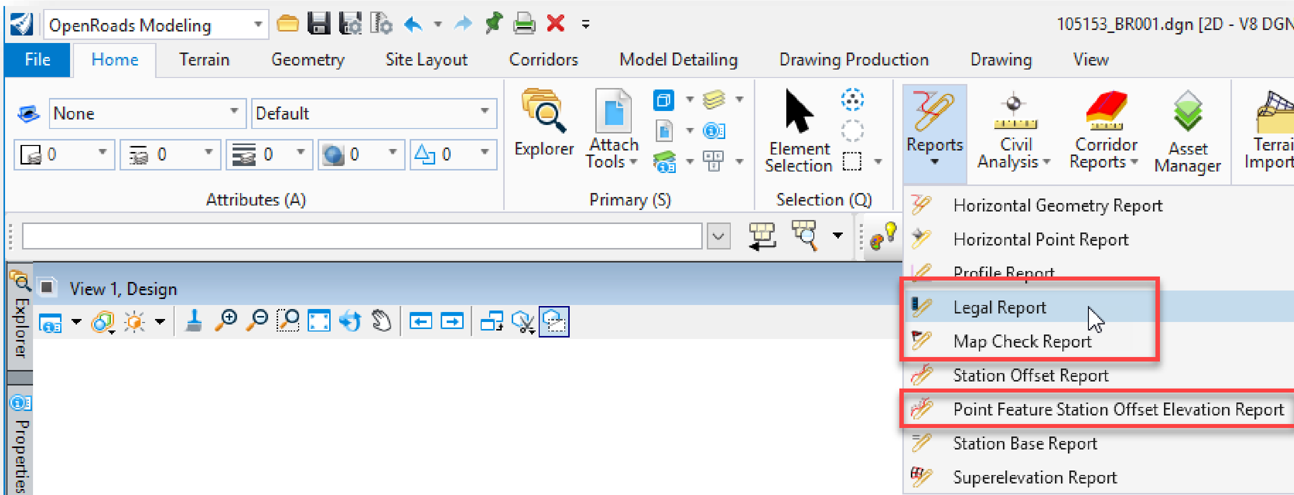
Depending on the report command that is used to open the ***Report Browser*** dialog, the selected report may not contain the desired information. For example, if the ***Report Browser*** dialog is accessed by selecting the **Horizontal Geometry Report** icon as shown above, selecting the **ParcelLayoutFromRefernce** report does not have the necessary information to generate the station and offset data for the report.

When using the **Horizontal Geometry Report**, the **Horizontal Alignment Review** report is the default report as shown below.



Parcel Reports

Various reports applicable to parcels are available on the **Home** tab in the **Reports** group as shown below.

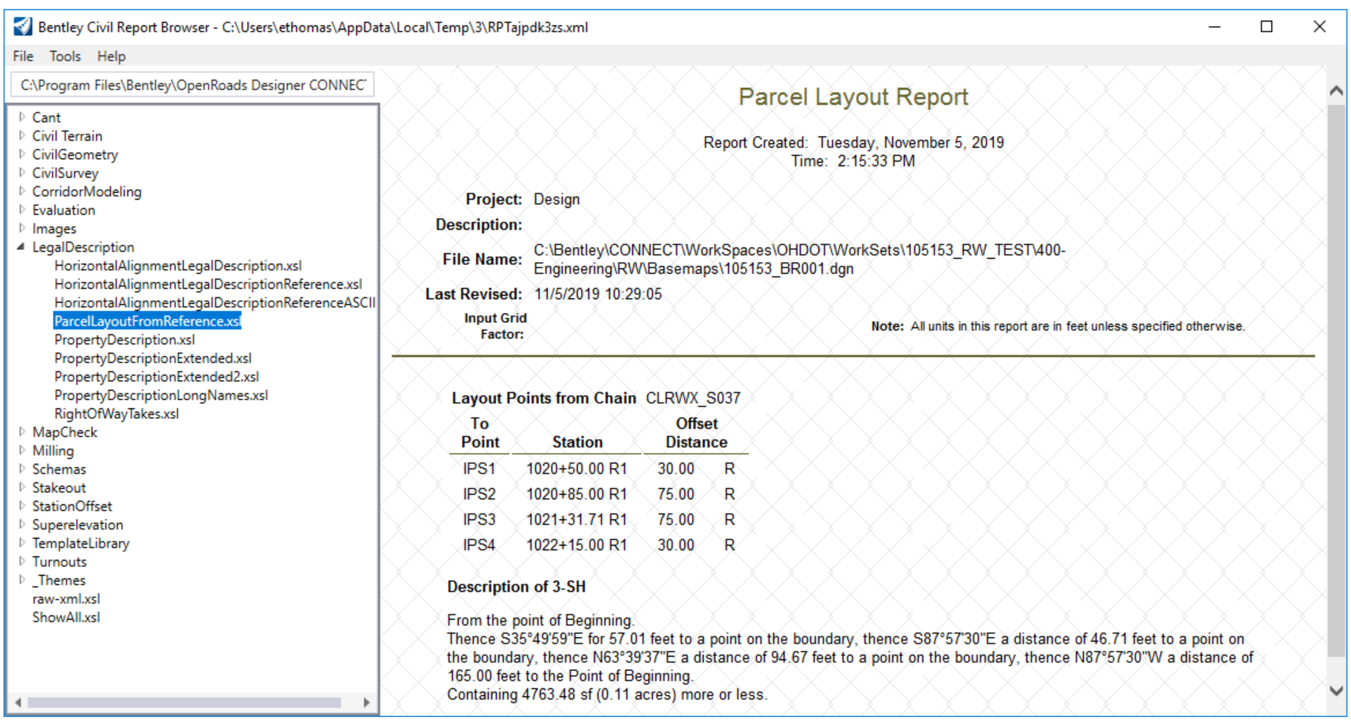


Legal Report

Legal descriptions can be generated by selecting **Reports > Legal Report** from the **Home** tab. When this command is selected, the user is prompted to enter the following information:

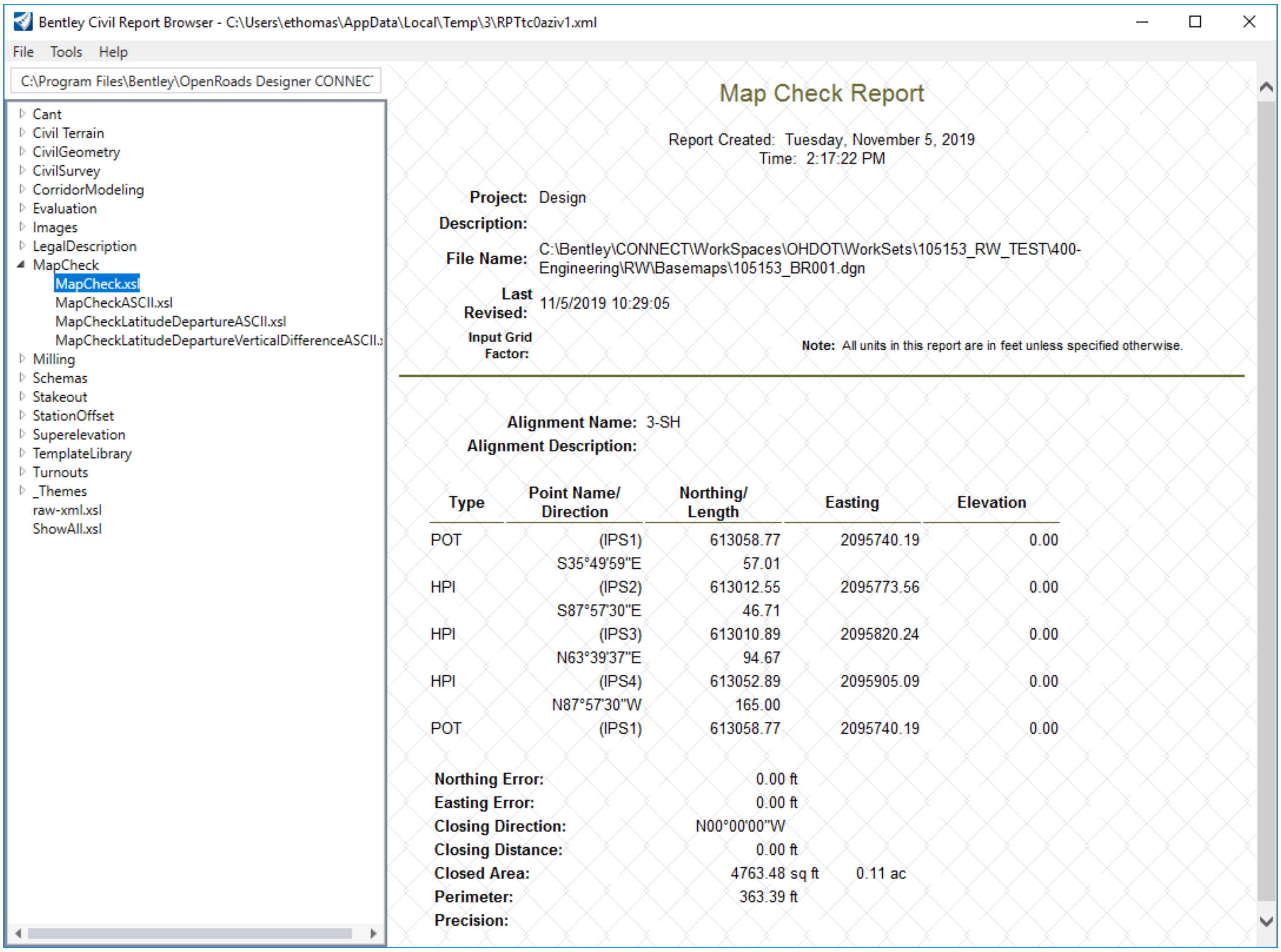
- Locate First Element
  - Choose the first parcel element to be included in the report
- Locate Next Element – Reset to Complete
  - Choose additional parcel elements to be included in the report or reset to complete the parcel selection process
- Locate Parent Element – Reset to Skip
  - Choose the parent tract element
- Locate Primary Reference – Reset to Skip
  - Choose the primary alignment for station and offset references
- Primary Reference Alignment: Bandwidth
  - Enter the bandwidth value
- Locate First Secondary Reference Element
  - Choose the secondary reference element for station offset references. Reset to skip this step

After entering the data as prompted, the report is generated. By default, the **LegalDescription HorizontalAlignmentLegalDescription** report is selected. Any of the reports in the **LegalDescription** category can be selected. If a reference element is included in the selection process, the station and offset information is included depending on the selected report.



Map Check Report

The map check report is generated by selecting **Reports > Map Check Report** from the **Home** tab. When this command is selected, the user is prompted to locate the parcel element for the report. An example of the report is shown below.


Bentley Civil Report Browser - C:\Users\ethomas\AppData\Local\Temp\3\RPTtc0aziv1.xml

File Tools Help

C:\Program Files\Bentley\OpenRoads Designer CONNEC

Cant
Civil Terrain
CivilGeometry
CivilSurvey
CorridorModeling
Evaluation
Images
LegalDescription
MapCheck
MapCheck.xsl
MapCheckASCII.xsl
MapCheckLatitudeDepartureASCII.xsl
MapCheckLatitudeDepartureVerticalDifferenceASCII.
Milling
Schemas
Stakeout
StationOffset
Superelevation
TemplateLibrary
Turnouts
_Themes
raw-xml.xsl
ShowAll.xsl

Map Check Report

Report Created: Tuesday, November 5, 2019
Time: 2:17:22 PM

Project: Design
Description:
File Name: C:\Bentley\CONNECT\WorkSpaces\OHDOT\WorkSets\105153_RW_TEST\400-Engineering\RW\Basemaps\105153_BR001.dgn
Last Revised: 11/5/2019 10:29:05
Input Grid Factor:

Note: All units in this report are in feet unless specified otherwise.

Alignment Name: 3-SH
Alignment Description:

Type	Point Name/ Direction	Northing/ Length	Easting	Elevation
POT	(IPS1)	613058.77	2095740.19	0.00
	S35°49'59"E	57.01		
HPI	(IPS2)	613012.55	2095773.56	0.00
	S87°57'30"E	46.71		
HPI	(IPS3)	613010.89	2095820.24	0.00
	N63°39'37"E	94.67		
HPI	(IPS4)	613052.89	2095905.09	0.00
	N87°57'30"W	165.00		
POT	(IPS1)	613058.77	2095740.19	0.00

Northing Error: 0.00 ft
Easting Error: 0.00 ft
Closing Direction: N00°00'00"W
Closing Distance: 0.00 ft
Closed Area: 4763.48 sq ft 0.11 ac
Perimeter: 363.39 ft
Precision:

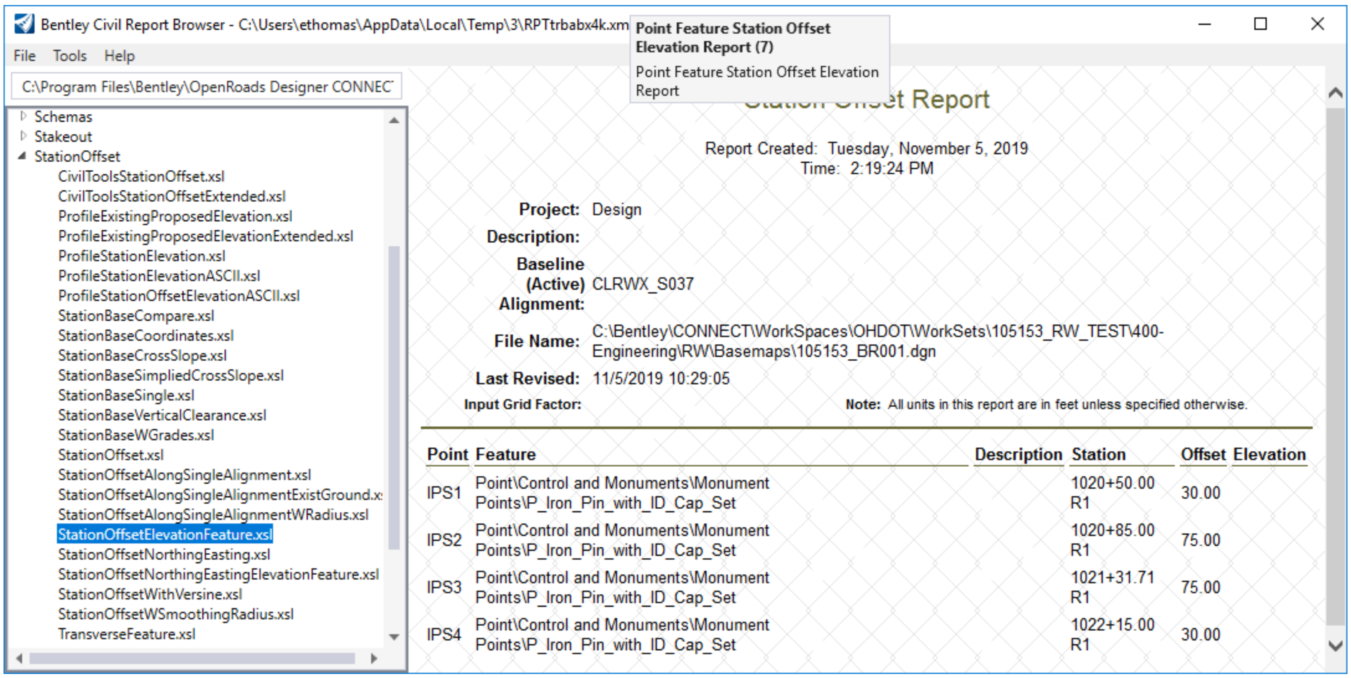
The **MapCheck.xsl** report is the default report for this command as shown above. Reports in other formats can be selected from the list at the left side of the dialog.

Point Feature Station Offset Elevation Report

This report is generated by selecting **Reports > Point Feature Station Offset Elevation Report** from the **Home** tab. When this command is selected, the user is prompted as follows:

- Locate Baseline Element
    - Select the alignment to be used for station and offset references
- Locate First Offset Element
    - Select the geometry point element to include in the report
- Locate Next Offset Element – Reset to Complete
    - Select additional geometry point elements or reset to complete the selection

An example of the report is shown below.

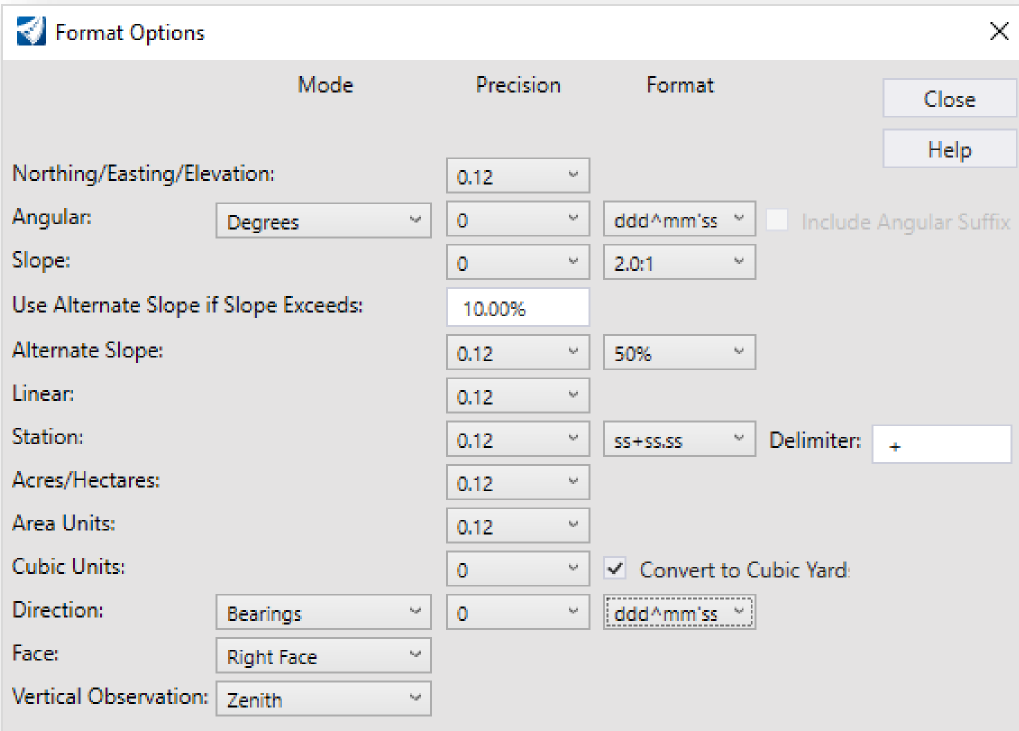


The **StationOffsetElevationFeature** report is the default report for this command. Note that a few other station and offset report formats are available for selection in the list on the left side of the dialog.

Report Format

The **Report Browser** allows the user to set the output format for the reports by choosing **Tools > Format options.**

The values shown below are recommended for most ODOT applications.



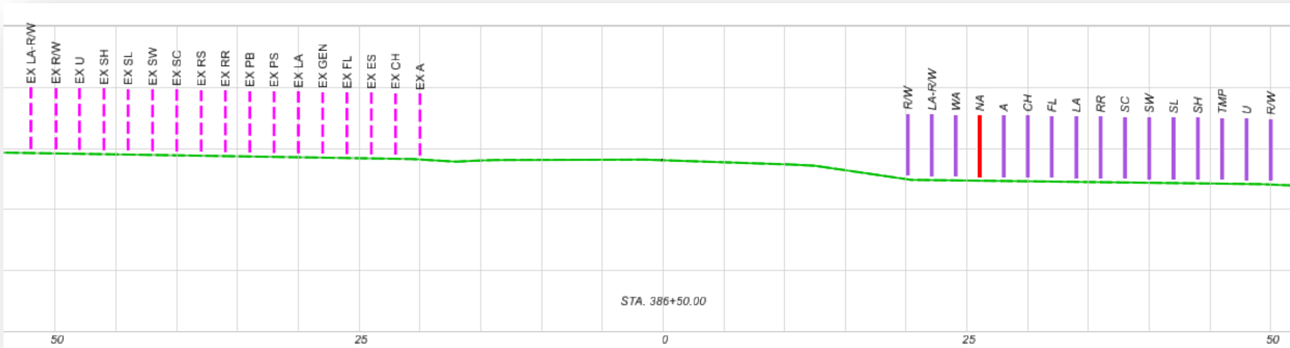
Saving Reports

Reports can be saved by choosing **File > Save As** from the ***Report Browser*** dialog.

Right-of-Way related reports are typically saved in the **\400-Engineering\RW\EngData** folder.

Displaying Right-of-Way on Cross Sections

The location of the Right-of-Way and Easement lines can be displayed in a dynamic cross-section view and on the cross-section sheets like the example shown below:

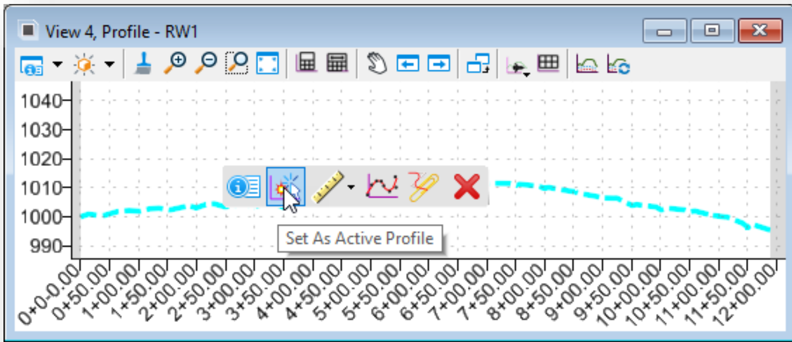


To display the Right-of-Way lines on the cross-sections, each Right-of-Way line must have a profile defined. This is accomplished by opening a profile view for each Right-of-Way line and defining the exiting ground profile as the active profile. When the profile is defined for right-of-way lines, a 3D line string is created in the 3D model for the line.

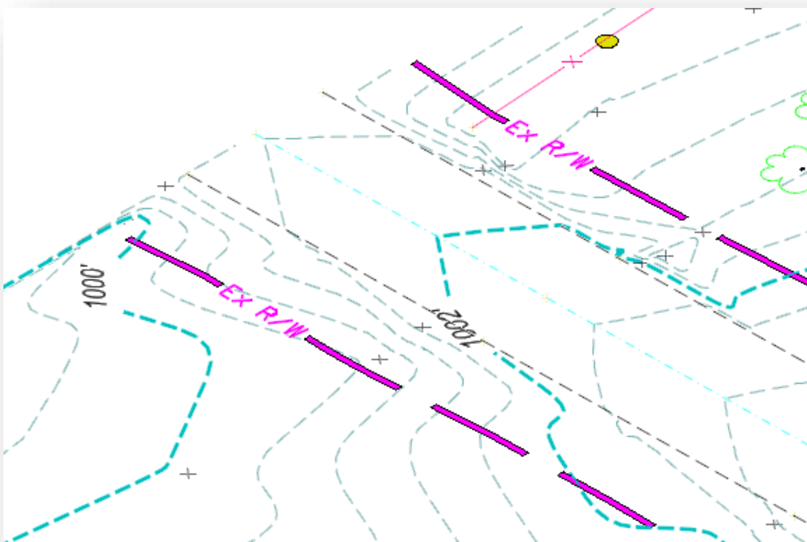
The workflow to assign a profile to the right-of-way lines is as follows:

- In the right-of-way basemap design file (BR), attach the design file containing the existing terrain
- Set the referenced terrain model as the active terrain by selecting the terrain and choosing the **Set As Active** Terrain option from the pop-up menu

- Open a profile model for each right-of-way line, either by selecting the element and choosing the Open **Profile Model** option, or by holding down the right-mouse button and choosing **View Control > 2 Views Plan/Profile** from the pop-up menu.
- Select the existing groundline in the profile window and choose the **Set As Active Profile** option from the pop-up menu



A 3D element is generated in the 3D model like the example below.



When cross sections are generated, if the cross section intersects a 3D Right-of-Way line in the 3D model, a cell is placed in the cross section at the right-of-way line offset and elevation.

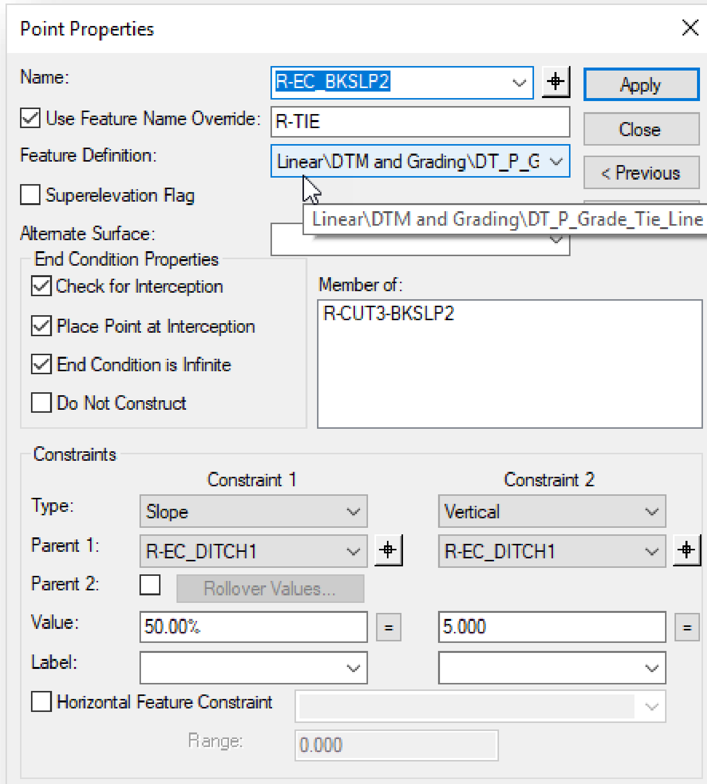
Construction Limits

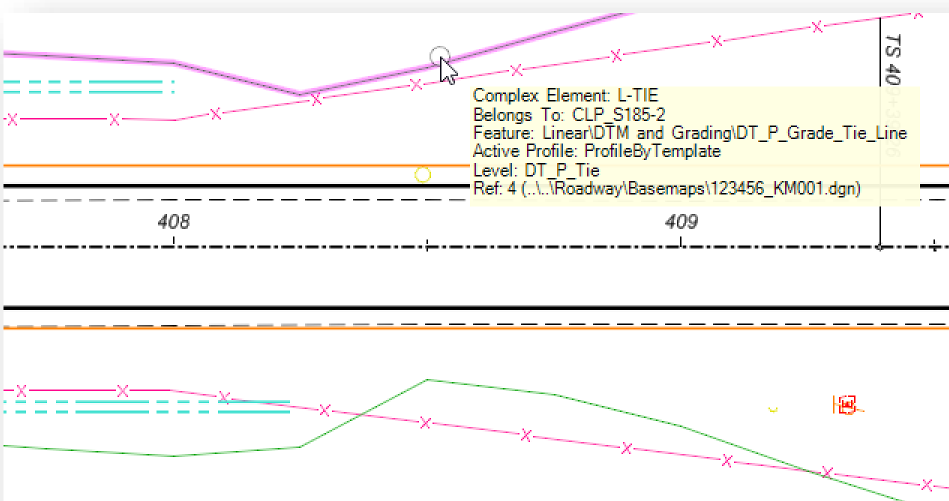
Construction limits can be generated from the graphics drawn during the 3D modeling process. There may be multiple ways to generate the construction limit lines using the template. This section documents that process that seem to work best for ODOT projects. The process is summarized below:

- Assign the **Feature Name Override** name consistently for the outer tie-down points in the template. This applies to all points in the template that tie to ground for both cut and fill conditions. In the example at right, the name R-TIE is assigned to all the outer tie-down points on the right side of the template.

- Assign the DT_P_Grade_Tie_Line **Feature Definition**, as shown at right, to all the outer tie down points in the template.

- When the template is processed, a continuous element is drawn in the 2D model at the outer tie-down location. This line will connect between different cut and fill transitions if the same **Feature Name Override** is used for all the outer tie-down points. An example is shown below.

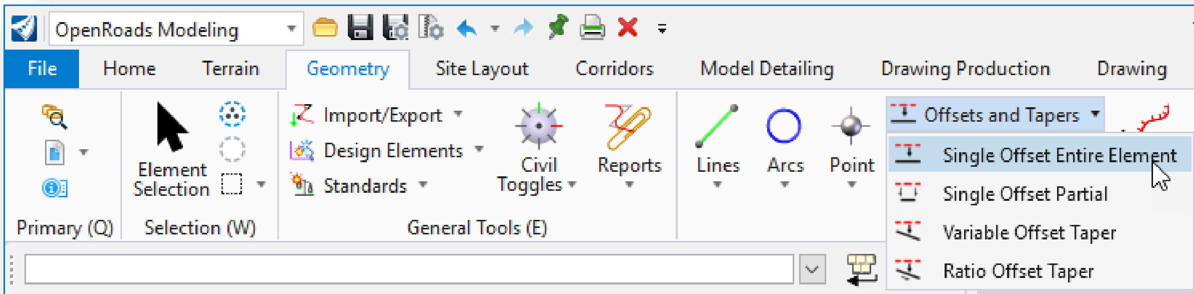




Construction limits can be generated from the outer tie-down lines as follows:

- The construction limits are normally drawn in the right-of-way design file (BR) containing the proposed right-of-way information. Attach the corridor design file (KM) containing the outer tie-down lines as a reference.

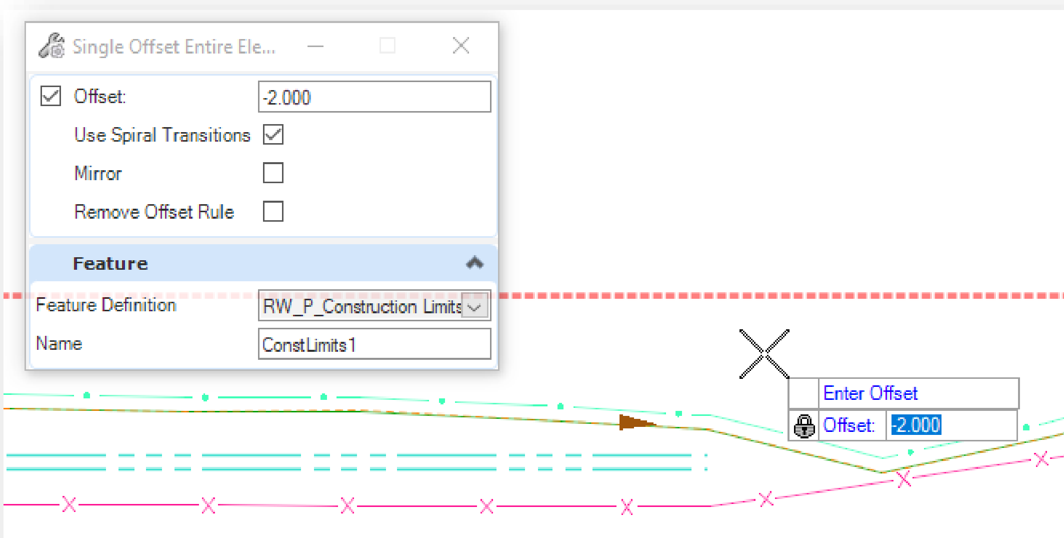
- From the **OpenRoads Modeling** Workflow, select **Geometry > Offsets and Tapers > Single Offset Entire Element**



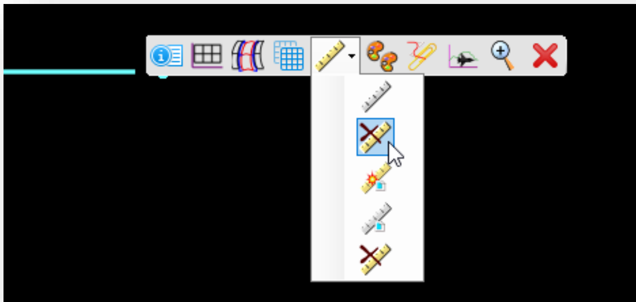
- Enter the desired **Offset** value

- Set the **Feature Definition** to **Linear\Right-of-Way\RW_P_Construction Limits**

- Select the DT_P_Tie element to create the new offset the element. An example is shown below.



- The newly created construction limit element is a ruled geometry element. To make edits to the element, it is necessary to remove the rules. Use the **Element Selection** tool to select the element and choose the **Remove Rule** option from the pop-up menu as shown below. After removing the rules, the element can be modified as desired using standard MicroStation drawing commands.



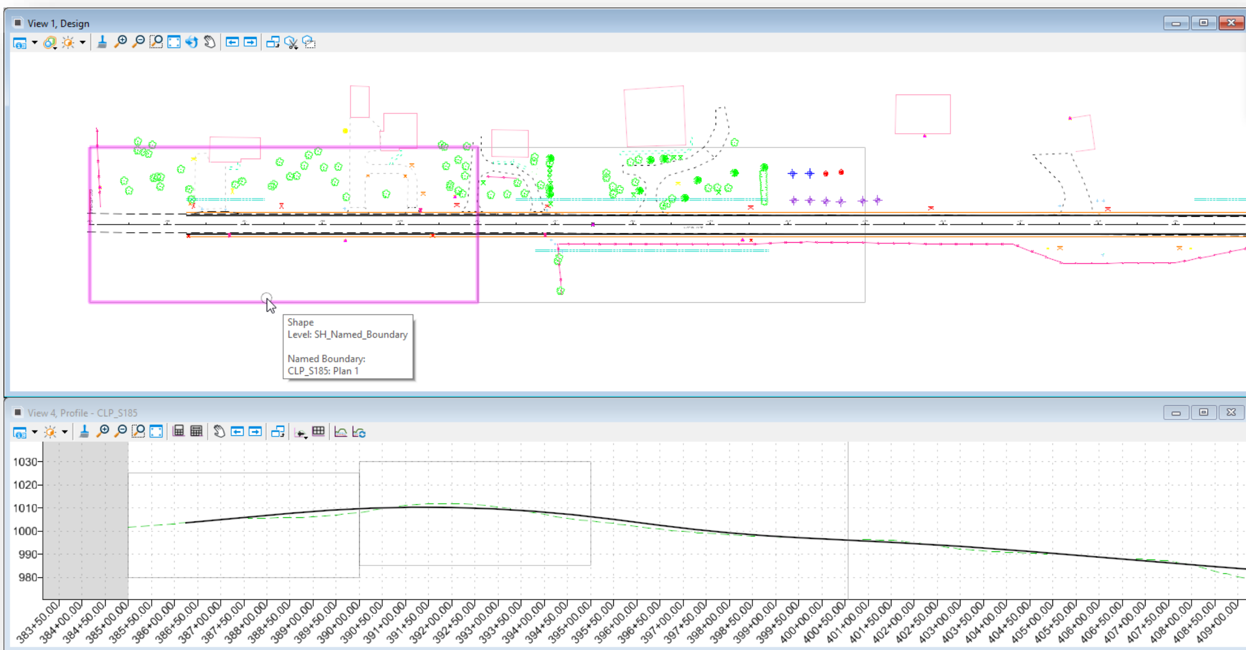
Rule Icons

- Deactivate Rule
- Remove Rule
- Activate Referencing Rules
- Deactivate Referencing Rules
- Remove Rule

Section 800 Plan and Profile Sheets

Process Overview

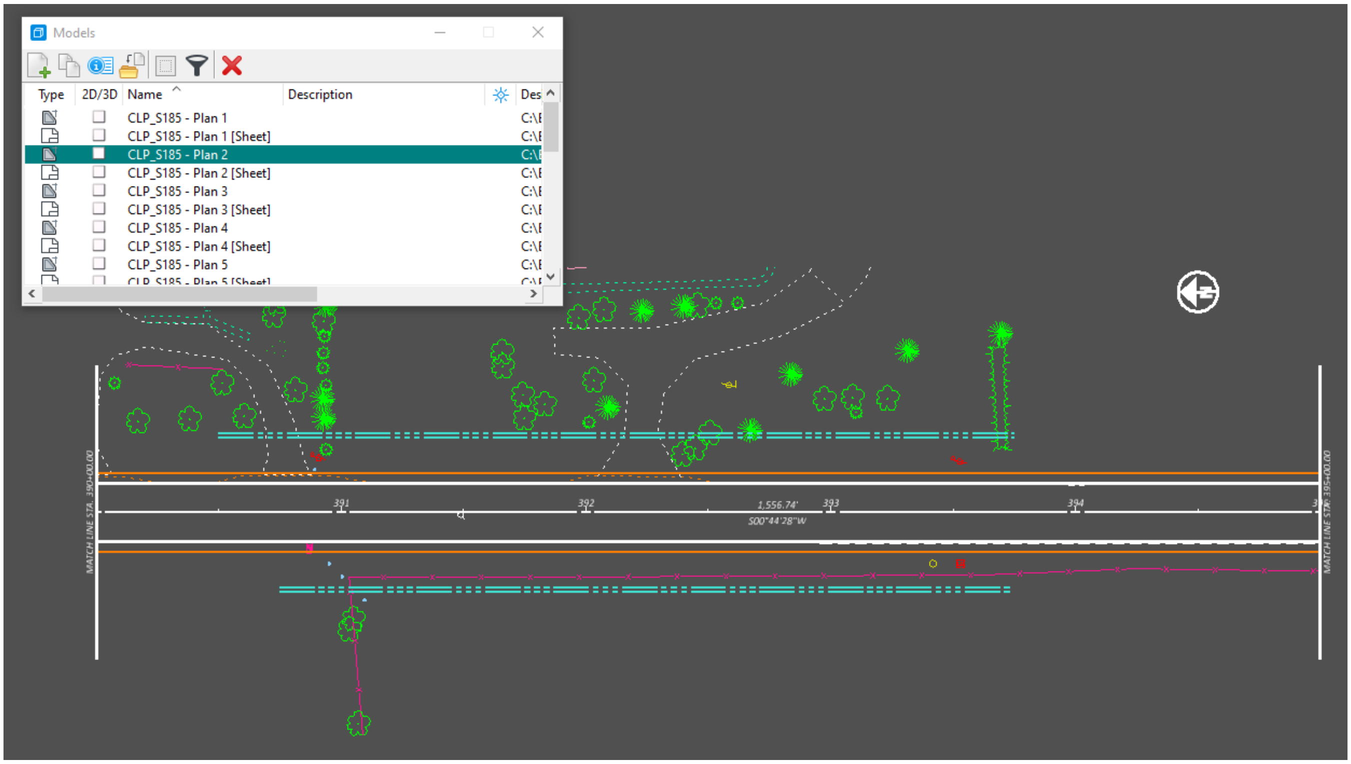
OpenRoads designer provides tools to easily create plan sheets in various formats such as plan only, plan and profile, or profile only. MicroStation **Named Boundaries** are used to define the clipping areas for the plan and profile portions of the sheets. In the example below, two named boundaries have been placed in the plan and profile views.



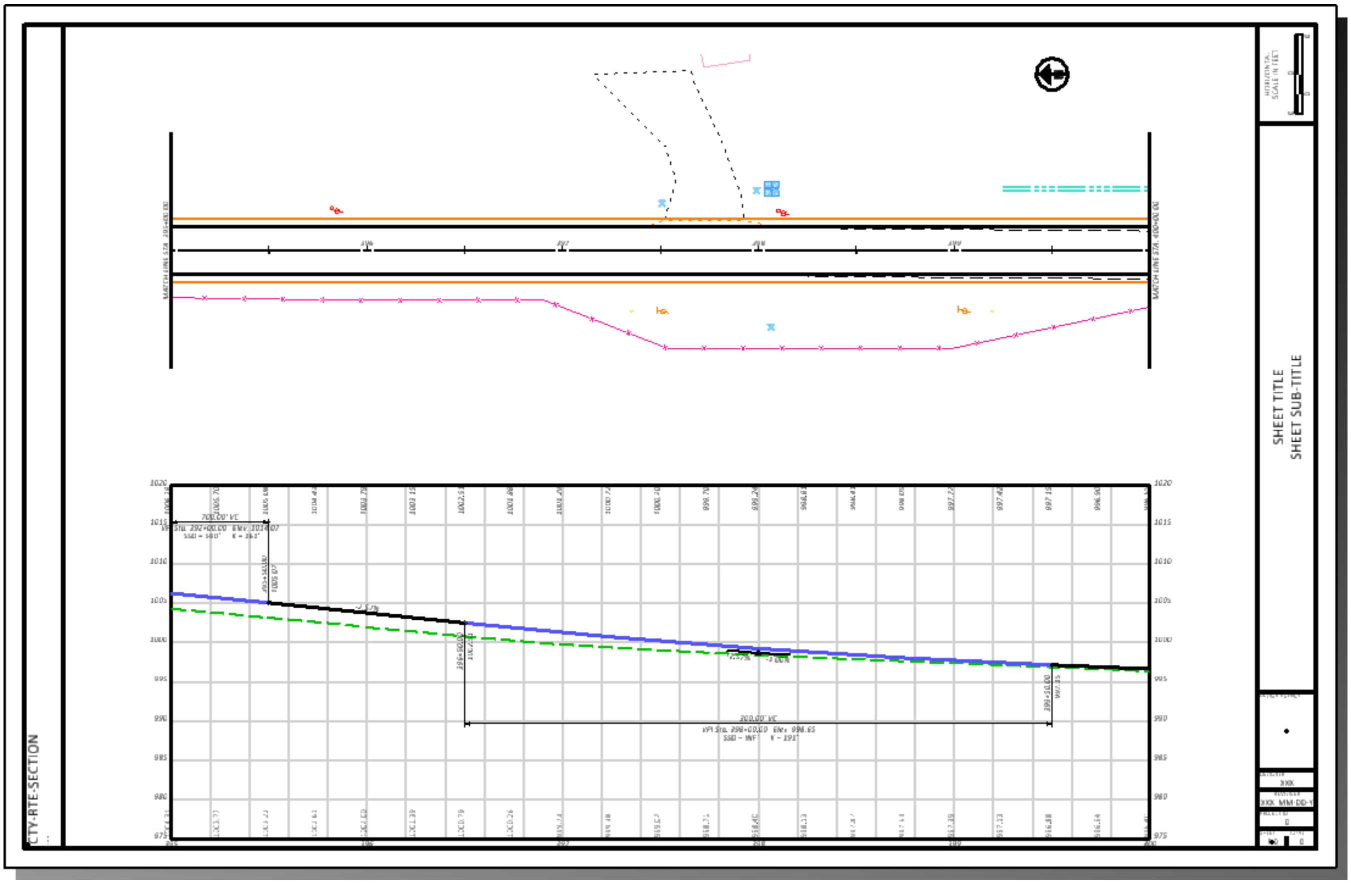
The various basemap design files required to assemble the sheets are attached as reference files to the **Design** Model in the sheet design file. In the example above, the geometry and survey basemaps are attached to the plan view with a profile window displayed.

Sheets can be clipped into the active design file, or separate design files.

When the sheets are generated, the software creates a MicroStation **Drawing Model** for each named boundary. An example of the drawing model for a plan view named boundary is shown below.



The drawing models are referenced to a sheet model with a sheet border to assemble the completed sheets for plotting and PDF generation as shown in the example below.



The sheet model is intended for plotting or PDF generation. Annotations should be placed in the drawing models, not in the sheet models.

File Types and Usage

Sheets can be clipped into a single file, or into multiple design files, one sheet per file. In either case, ODOT recommends using the fist file of a sheet series, the 000, 100, 200, etc. file, as the ***container*** file to attach references and place the named boundaries.

Each container file is used to generate the sheets for a single alignment. For example, a project with three alignments using the option to clip the sheets into separate files (one sheet per file) would typically use three container files as detailed below:

Design File	Usage	Models Used
GP000	Container file for S.R. 185	Design
GP001, GP002, GP003…	Plan and profile sheets for S.R. 185	Drawing and Sheet
GP100	Container file for St. Peter Road	Design
GP101, GP102, GP103…	Plan and profile sheets for St. Peter Road	Drawing and Sheet
GP200	Container file for Jameson Road	Design
GP201, GP202, GP203…	Plan and profile sheets for Jameson Road	Drawing and Sheet

The references are attached as illustrated below:

Basemap Files

The various basemap design files (survey, alignments, drainage, etc.) are used to draw the existing and proposed information for the project.

Container File

The container file's ***design*** model is used to attach the various basemap design files as references, and to place the named boundaries for the both plan and profile views.

Sheet File

The ***drawing*** models reference the container file's design model. The drawing models are also used for sheet annotation.

The ***sheet*** models references the drawing models. The sheet models are used for the sheet border and for plotting.

If all the sheets for a single alignment are to be clipped into one file, the container file is used for the sheets as follows:

Design File	Usage	Models Used
GP000	Container file and sheets for S.R. 185	Design, Drawing, and Sheet
GP100	Container file and sheets for St. Peter Road	Design, Drawing, and Sheet
GP200	Container file and sheets for Jameson Road	Design, Drawing, and Sheet

The references are attached as illustrated below:

Basemap Files

The various basemap design files (survey, alignments, drainage, etc.) are used to draw the existing and proposed information for the project.

Container File Design Model

The container file's ***design*** model is used to attach the various basemap design files as references, and to place the named boundaries for the both plan and profile views.

Container File Drawing Model

The ***drawing*** models reference the container file's design model. The drawing models are also used for sheet annotation.

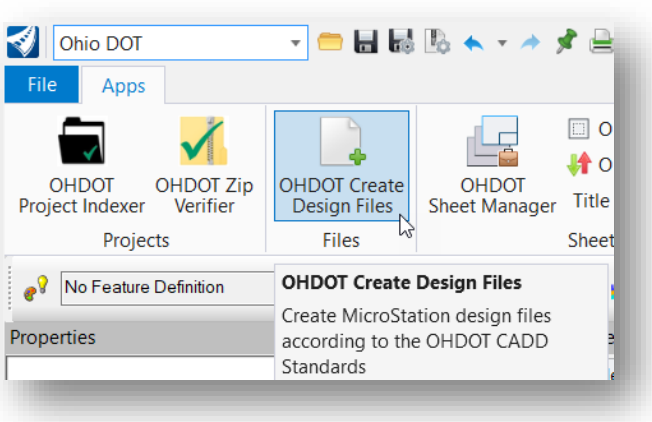
Container File Sheet Model

The ***sheet*** models reference the container file's drawing models. The sheet models are used for the sheet border and for plotting.

The container file is also used to control the reference attachment and level display settings for all the sheets from one location. More information on level display can be found in the Level Management section of this document.

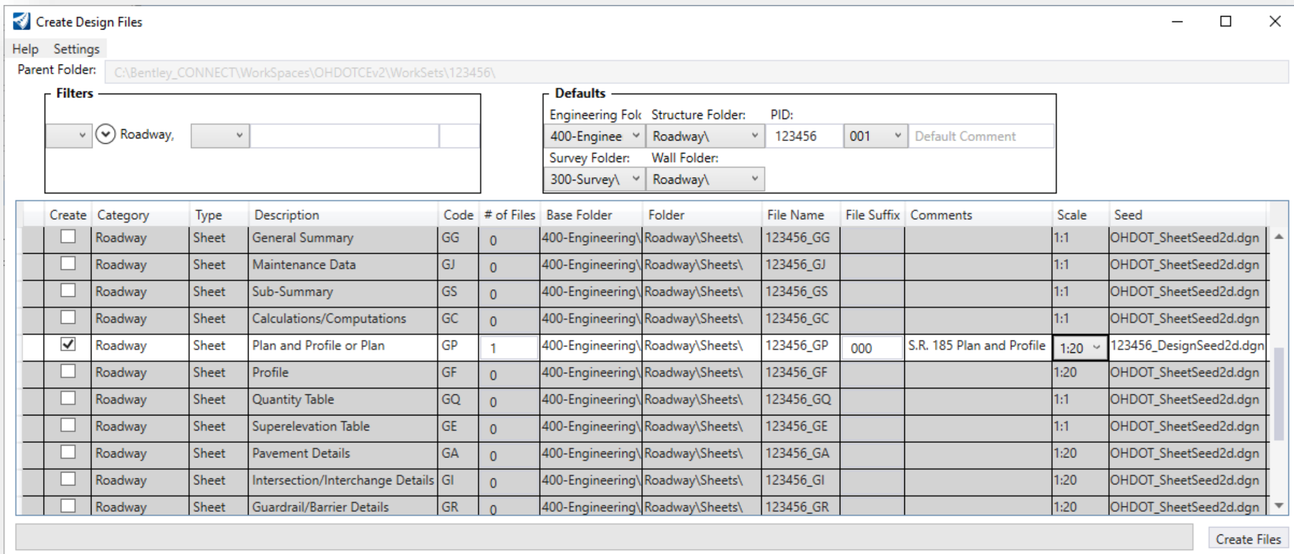
Creating the Sheet Container File

Sheet design files can be created using the **OHDOT Create Files** application. The application is accessed from the Ohio DOT WorkFlow by selecting the **New Design File** icon.



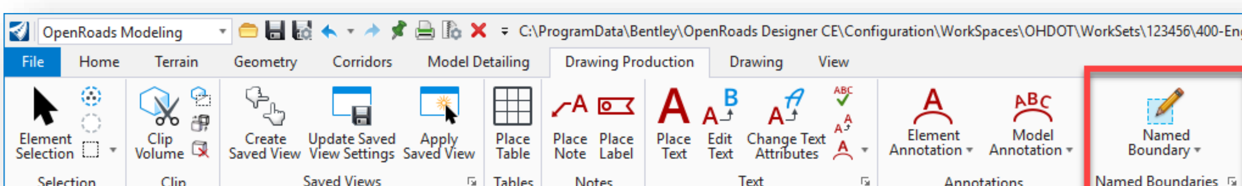
When creating sheet design files, it is important to consider the seed file that will be used to create the new file. Normally, a sheet design file is created using the **Sheet Seed** as defined by the OHDOT WorkSpace configuration. When creating design files for OpenRoads sheet clipping, it is important to use the **Design Seed**.

When the **File Suffix** value is entered, the Create Design Files application will automatically select the Design Seed if a suffix of 000, 100, 200, 300… is entered as shown below. This will ensure that the file is created with a design model instead of a sheet model. The design model is necessary for attaching the references that will be displayed in the sheets as well as laying out the named boundaries.

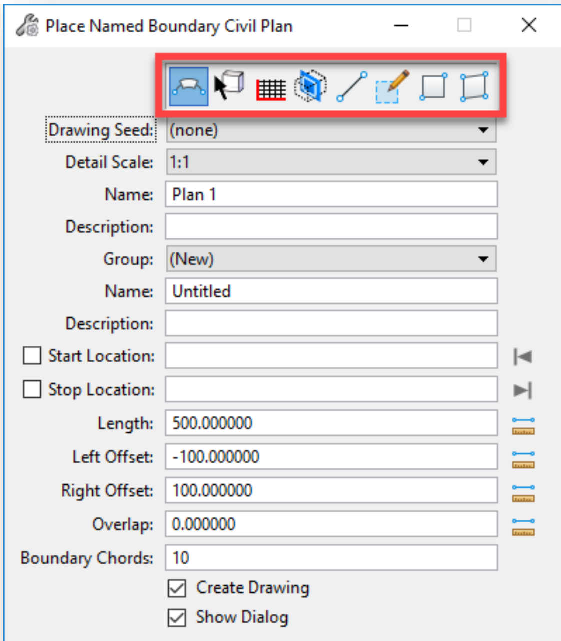


Named Boundaries

Sheets are clipped using the **Named Boundary** tool. From the **OpenRoads Modeling** workflow, select the **Drawing Production** tab to access the **Named Boundary** tool as shown below.



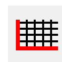
The *Place Named Boundary Civil Plan* dialog is opened as shown below.



The top of the dialog contains several icons to select the type of named boundary that you wish to place.

The following icons will be discussed in this document:

 Civil Plan

 Civil Profile

 Civil Cross Section

 Civil Cross Section 2 Points

ODOT has configured several sheet clipping options for plan, profile, and cross section sheets. These configurations are stored in a MicroStation DGN Library. The Configuration files are defined in the OHDOT_WorkSpace.cfg file as follows:

MS_DGNLIBLIST > $(_USTN_WORKSPACESTANDARDS)Dgnlib/Sheet Seeds/*.dgnlib

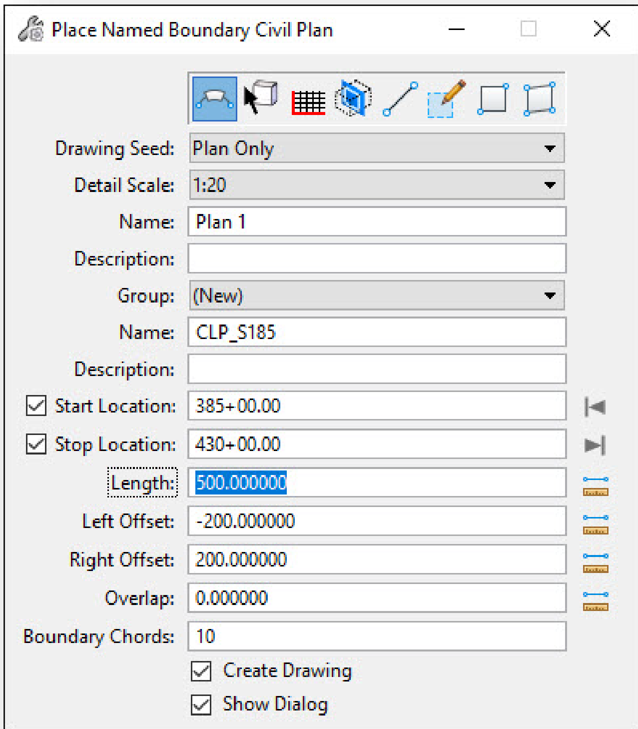
Named Boundary Level

The named boundaries are placed on the active level. Set the active level to **SH_Named_Boundaries** before placing the named boundaries in the design,

Sheet Clipping

Plan Only Sheets

From the OpenRoads Modeling WorkFlow, select **Drawing Production > Named Boundaries**. On the *Place Named Boundary Civil Plan* dialog, select the **Civil Plan** icon.



The **Drawing Seed** item is used to select the configuration for the named boundaries to be placed. Several options are available as defined by the OHDOT DGN Libraries.

For clipping Plan-Only sheets, select the **Plan Only** option for the **Drawing Seed** parameter as shown at left.

When one of the **Drawing Seed** options is selected, several parameters in the dialog are automatically populated with default values according to the configured settings.

The **Detail Scale** parameter is used to select the sheet scale. In the left example, 20-scale plan sheets will be generated.

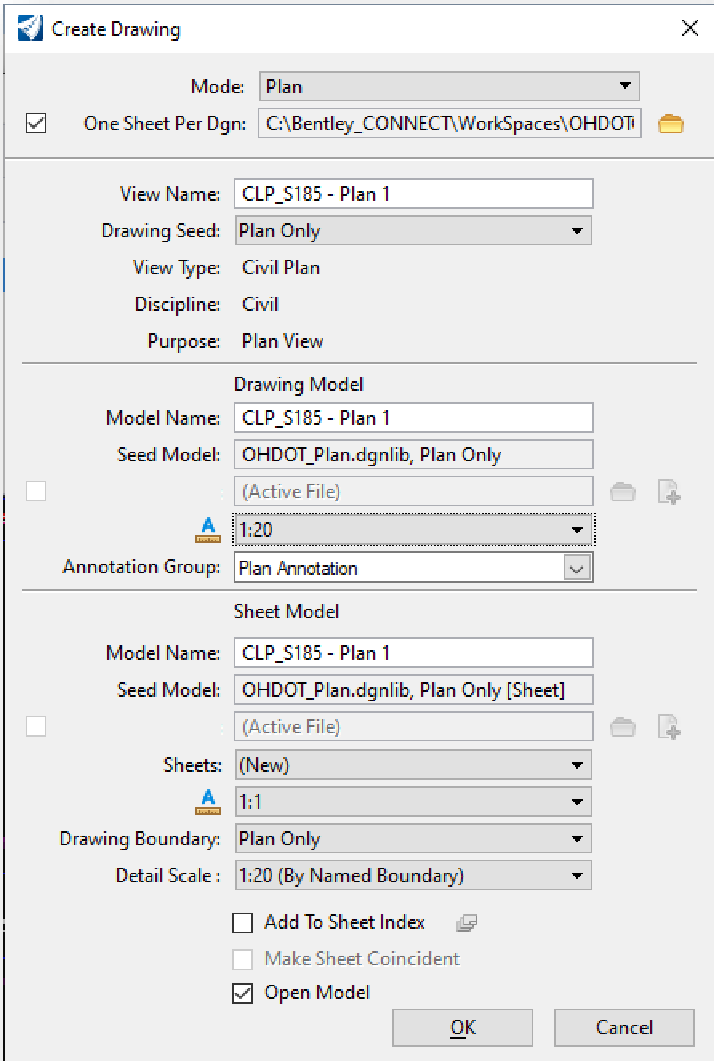
The **Length**, **Left Offset**, and **Right Offset** parameters are set according to the selected **Detail Scale** value.

Take the following steps to select the alignment and define the **Start Location** and **Stop Location** values:

- Issue a left mouse-click in the **Start Location** Field.
- Select the desired alignment that will be used as the centerline for the sheet clipping. The **Start Location** value will dynamically change as the cursor is moved within the view window.
- Identify the desired **Start Location** by issuing a data-point (left-click) within the view, by keying in a station value, or by selecting the icon to the right of the **Start Location** field to lock the starting location to the beginning of the selected alignment.
- After the **Start Location** has been established, identify the desired **Stop Location.** As the cursor is moved, the **Stop Location** will track the station value relative to the cursor position. Clipping borders are dynamically displayed as the cursor is moved. The stop location can be entered by issuing a right-mouse click in the view, keying in the desired station, or by selecting the icon to the right of the **Stop Location** field to lock the ending location to the end of the selected alignment. Note that the last sheet will always assume the value of the **Length** field even if the stop location is not equal to the **Length** value.

The **Overlap**, and **Boundary Chords** items are set according to the selected **Drawing Seed** item. These items should typically be left to their default values.

After defining the parameters, issue a data point (left click) in the plan view to accept the **Start Location** value. A second data point accepts the **Stop Location** value. A third data point accepts the placement of the named boundaries. If the **Create Drawing** and **Show Dialog** options are toggled on, as shown on the preceding page, the ***Create Drawing*** dialog is opened as shown below.



Some of the parameters for the *Create Drawing* dialog have been predefined in the DGN Library for the selected **Drawing Seed**.

If you wish to create a separate design file for each sheet, toggle the **One Sheet Per DGN** option on and set the path for the sheet file location.

The **View Name** parameter is used for the named views that will be created in the container file as part of the clipping process. This parameter is not normally changed by the user.

Drawing Model parameters

The **Model Name** is used as the seed name for each of the drawing models that will be created. This name is incremented by the software. In most cases, the default name is used.

Set the **Annotation Scale** value for the **Drawing Model** to the desired value. This parameter should default to the same value used for the **Detail Scale** in the ***Place Named Boundary*** dialog, but it is good practice to review the value to ensure it is set correctly.

Sheet Model parameters

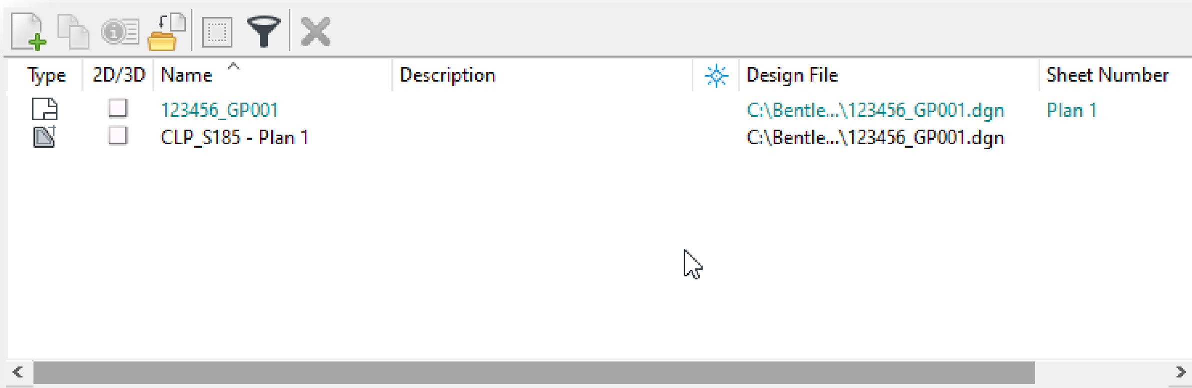
The **Model Name** entered for the sheet model is also the name that is used for the created design files when the **One Sheet Per Design** option is selected.

Do not change the **Annotation Scale** value for the **Sheet Model**. Leave this value at 1:1, as shown at above.

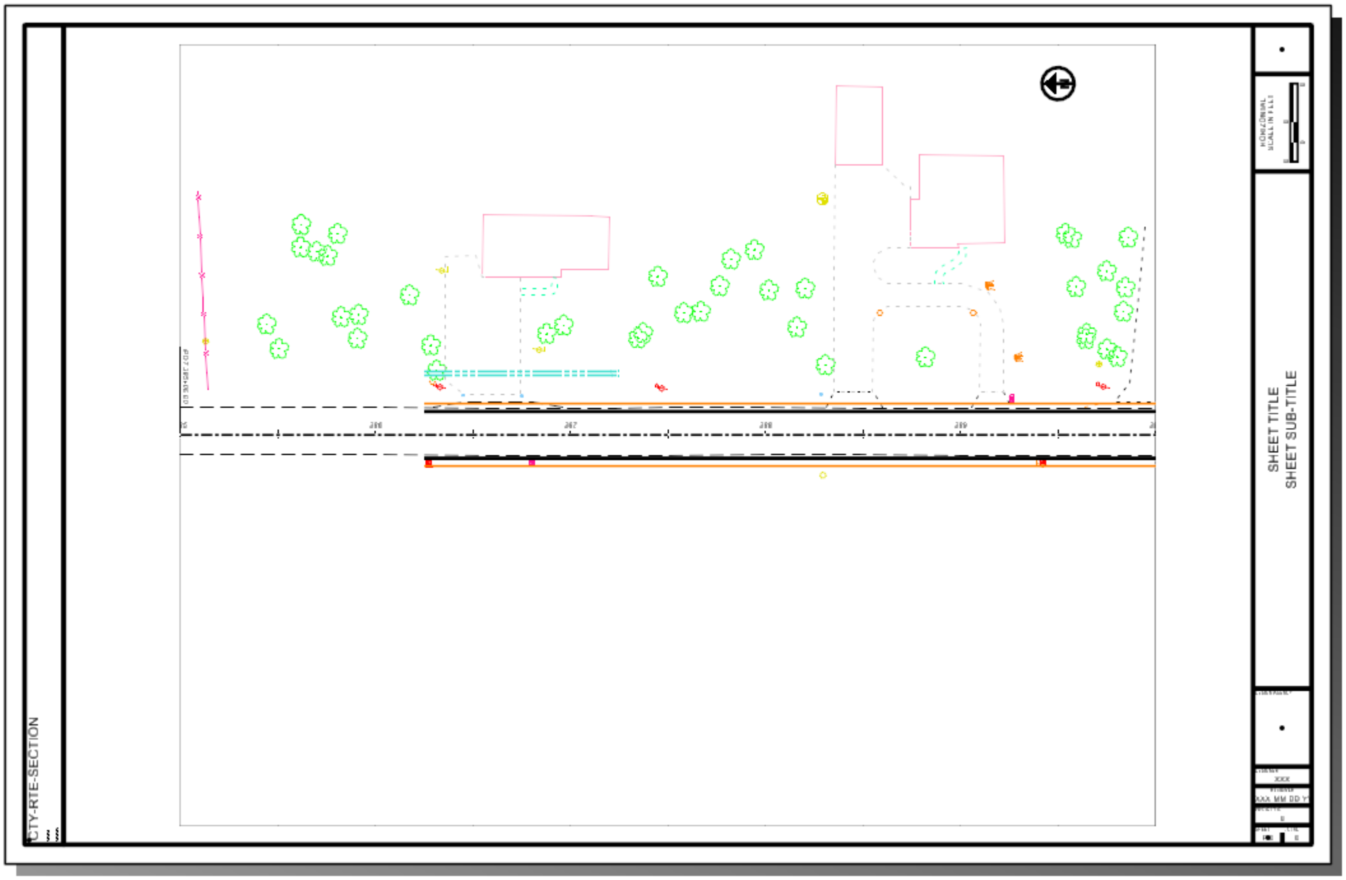
Set the **Detail Scale** value for the **Sheet Model** to the desired value. This parameter should default to the same value used for the **Detail Scale** in the ***Place Named Boundary*** dialog, but it is good practice to review the value to ensure it is set correctly.

After reviewing the dialog settings, select **OK** to continue the sheet clipping process. If **Open Model** is toggled on, the application will open the last created file upon completion.

An example of the models generated for a **Plan Only** sheet configuration when using the **One Sheet per DGN** option is shown below. Note the sheet model name is the same as the design file name.



An example of a completed sheet is shown on the next page.



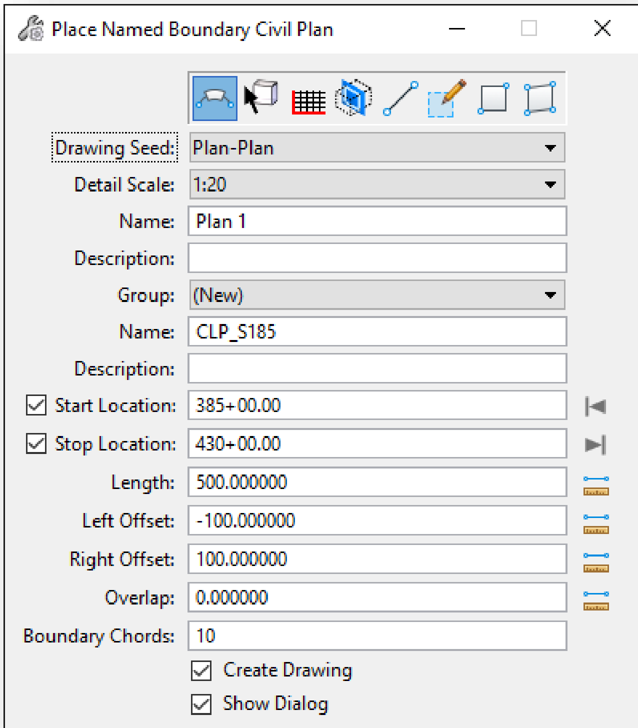
Notes:

- Each sheet design file contains a **Drawing** model corresponding to the plan view named boundary
- The **Drawing** models reference the original **Container** file
- The **Sheet** model references the **Drawing** model

Plan-Plan Sheets

The process to create dual-plan sheets is like the process to create plan-only sheets, as described above. The differences are detailed below.

From the OpenRoads Modeling WorkFlow, select **Drawing Production > Named Boundaries**. On the *Place Named Boundary Civil Plan* dialog, select the **Civil Plan** icon.

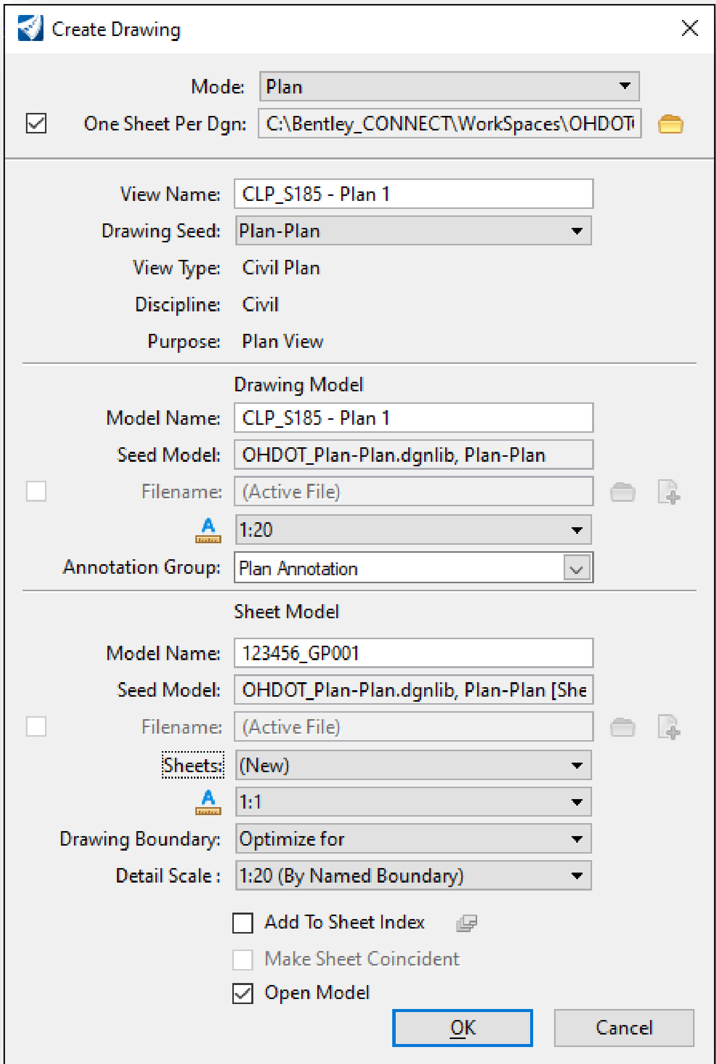


Select the **Drawing Seed** option **Plan-Plan** to generate sheets with two plan views.

The **Detail Scale** parameter defaults to 1:1. Set the parameter to the desired value. The **Length**, **Left Offset**, and **Right Offset** values are adjusted accordingly.

The process to define the alignment **Start Location** and **Stop Location** values is the same as described for the Plan-Only option above.

After defining the parameters, issue a data point (left click) in the plan view to accept the **Start Location** value. A second data point accepts the **Stop Location** value. A third data point accepts the placement of the named boundaries and opens the ***Create Drawing*** dialog, shown on the following page.



Some of the parameters for the *Create Drawing* dialog have been predefined in the DGN Library for the selected **Drawing Seed**.

If you wish to create a separate design file for each sheet, toggle the **One Sheet Per DGN** option on and set the path for the sheet file location.

The **View Name** parameter is used for the named views that will be created in the container file as part of the clipping process. This parameter is not normally changed by the user.

Drawing Model parameters

The Drawing **Model Name** is used as the seed name for each of the drawing models that will be created. This name is incremented by the software. In most cases, the default name is used.

Set the **Annotation Scale** value for the **Drawing Model** to the desired value. This parameter should default to the same value used for the **Detail Scale** in the ***Place Named Boundary*** dialog, but it is good practice to review the value to ensure it is set correctly.

Sheet Model parameters

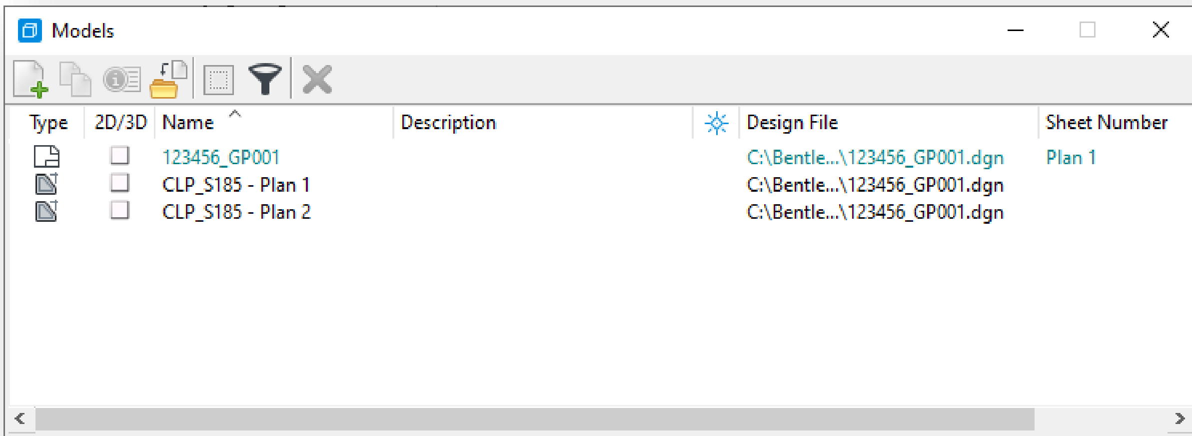
The Sheet **Model Name** entered for the sheet model is also the name that is used for the created design files when the **One Sheet Per Design** option is selected.

Do not change the **Annotation Scale** value for the **Sheet Model**. Leave this value at 1:1, as shown at above.

Set the **Detail Scale** value for the **Sheet Model** to the desired value. This parameter should default to the same value used for the **Detail Scale** in the ***Place Named Boundary*** dialog, but it is good practice to review the value to ensure it is set correctly.

After reviewing the dialog settings, select **OK** to continue the sheet clipping process. If **Open Model** is toggled on, the application will open the last created file upon completion.

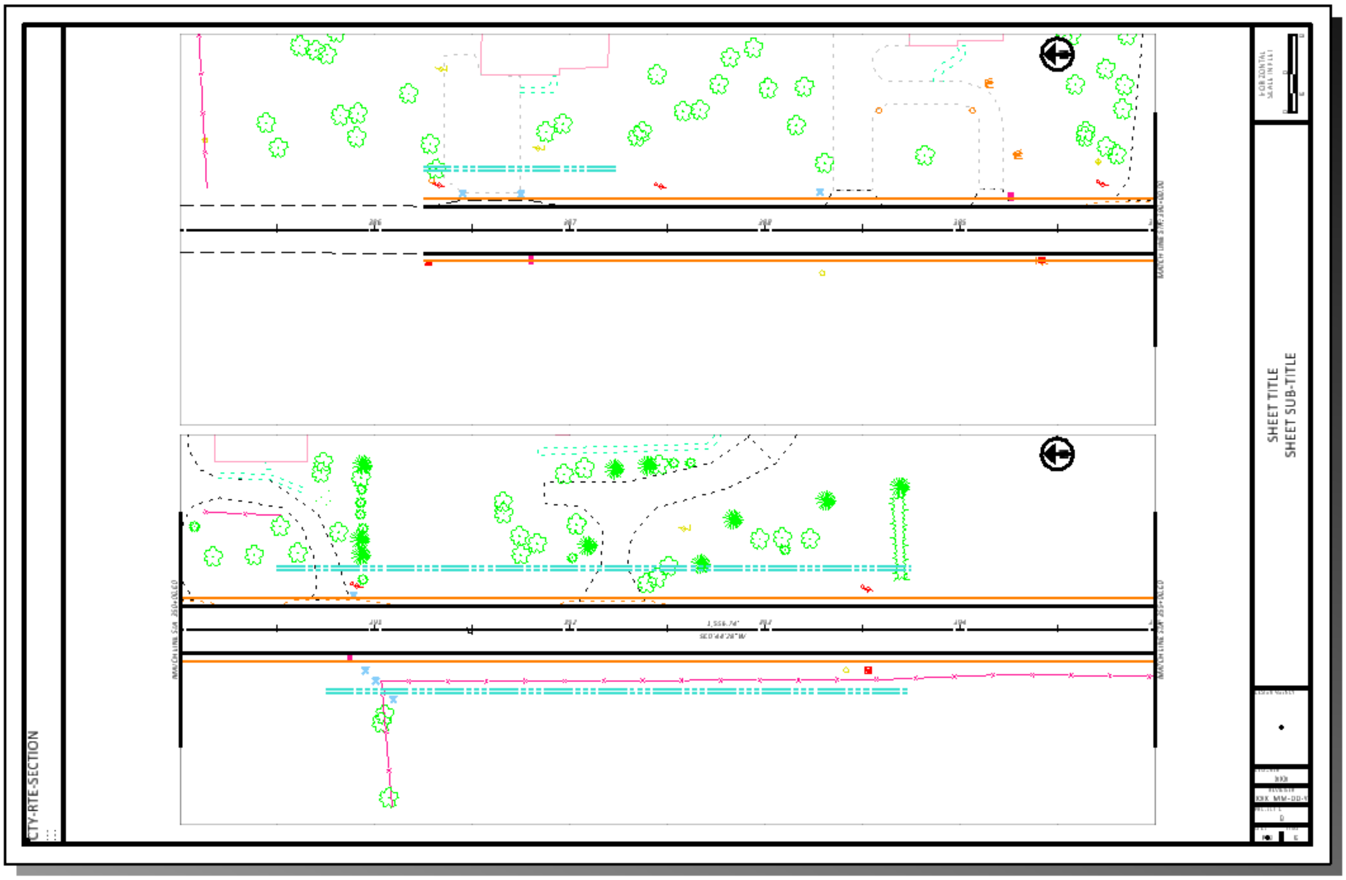
An example of the models generated for a **Plan Only** sheet configuration when using the **One Sheet per DGN** option is shown below. Note the sheet model name is the same as the design file name.



Select **OK** to continue the sheet clipping process.

After selecting **OK**, the software generates a drawing model for each named boundary. The drawing models are attached as references to a sheet model to generate the sheets.

An example of a Plan-Plan sheet is shown on the following page.

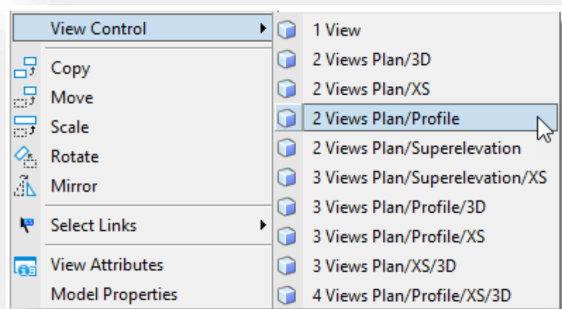

SHEET TITLE
SHEET SUB-TITLE
CTY-RTE-SECTION

Plan-Profile Sheets

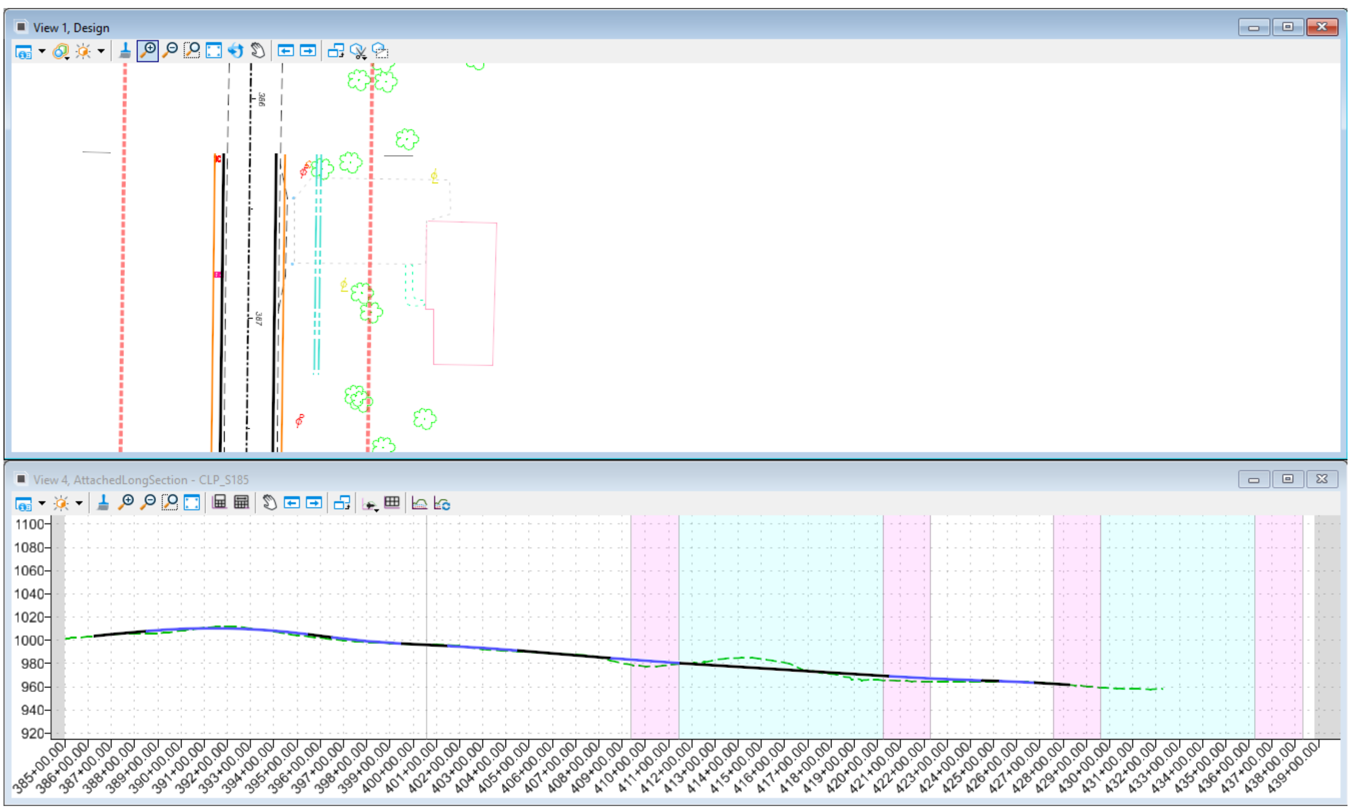
Plan and profile sheet clipping starts by configuring the MicroStation view windows to display both plan and profile information. This is facliltated using the right-mouse button menu. Hold down the right mouse button with the cursor positioned in a MicroStation view window until the pop-up menu appears. Select the **2 Views Plan/Profile** option as shown at right below.

Follow the prompts to open a profile window.

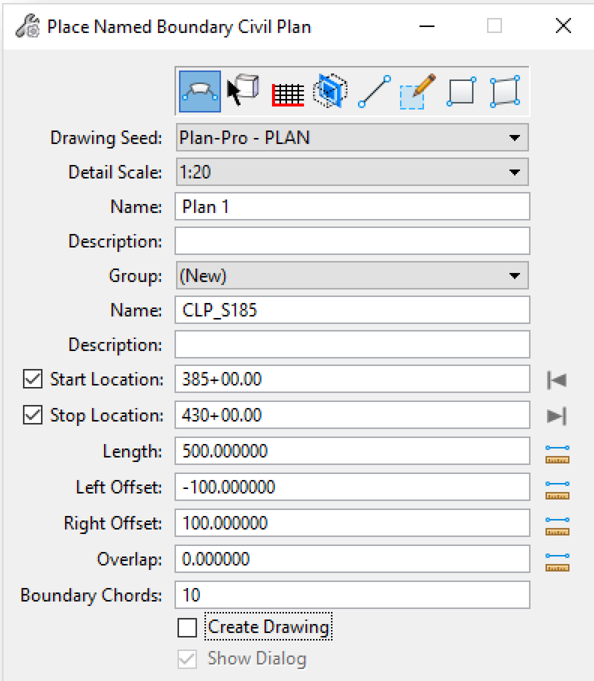
- Select **OK** when prompted to **Create a Dynamic Profile View**
- Locate the plan element
- Issue a left-click in the newly opened profile view to display the profile information for the selected alignment



When complete two view windows displaying the plan and profile are opened like the example below.



The named boundaries for plan and profile sheet clipping will be placed as two separate actions. From the **OpenRoads Modeling** WorkFlow, select **Drawing Production > Named Boundary** to initiate the process. The *Place Named Boundary Civil Plan* dialog is opened.



Select the first icon, **Civil Plan** to begin the process of placing the plan view named boundaries.

The **Drawing Seed** is used to select the configuration for various plan view named boundaries. Select the **Plan-Pro – PLAN** option.

The process to create the named boundaries for the plan view is the same as described previously in this document, with one significant difference:

Be sure to toggle ***off*** the **Create Drawing** option before placing the named boundaries as shown at left. It is necessary to generate the profile named boundaries before creating the drawings.

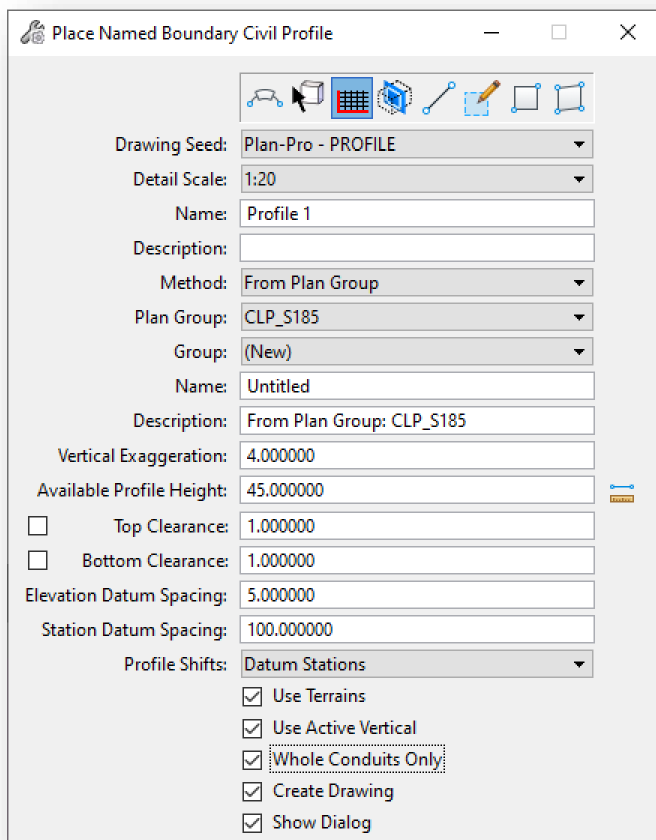
After the named boundaries for the plan view have been placed, take the following steps to create the named boundaries for the profile view:

- From the **OpenRoads Modeling** WorkFlow, select **Drawing Production > Named Boundary** to initiate the process. The *Place Named Boundary Civil Plan* dialog is opened.

- Select the third icon, **Civil Profile** to begin the process of placing the plan view named boundaries

- Select the **Drawing Seed** that corresponds to the previously selected plan view drawing seed. In this case, select **Plan-Pro – PROFILE** as shown at left.

- Set the **Detail Scale** to the desired horizontal plan scale, typically 1:20 or 1:50 for most ODOT plan and profile sheets



- Set the placement **Method** parameter to **From Plan Group**

- Key in the desired **Vertical Exaggeration** value

  The **Detail Scale** and **Vertical Exaggeration** parameters work together to determine the **Available Profile Height** parameter.

  The **Available Profile Height** value, as defined in the ODOT standards, is based on a profile grid that will measure 9" in height on the printed 22"x34" sheet. Typically, the **Available Profile Height** parameter is not changed by the user. It is best to allow the software to set this value as the **Detail Scale** and **Vertical Exaggeration** parameters are adjusted.

  For a typical ODOT project, 1:20 or 1:50 **Detail Scale** settings are the most often used values. The **Vertical Exaggeration** is then set to the desired value, typically a value of 4 for 1:20 scale plans, or a value of 5 for 1:50 scale plans.

  The **Available Profile Height** is computed as follows:

  Available Profile Height = Detail Scale * 9 / Vertical Exaggeration

  Example: A 20-scale plan with a 4:1 vertical exaggeration

  Available Profile Height = 20 * 9 / 4

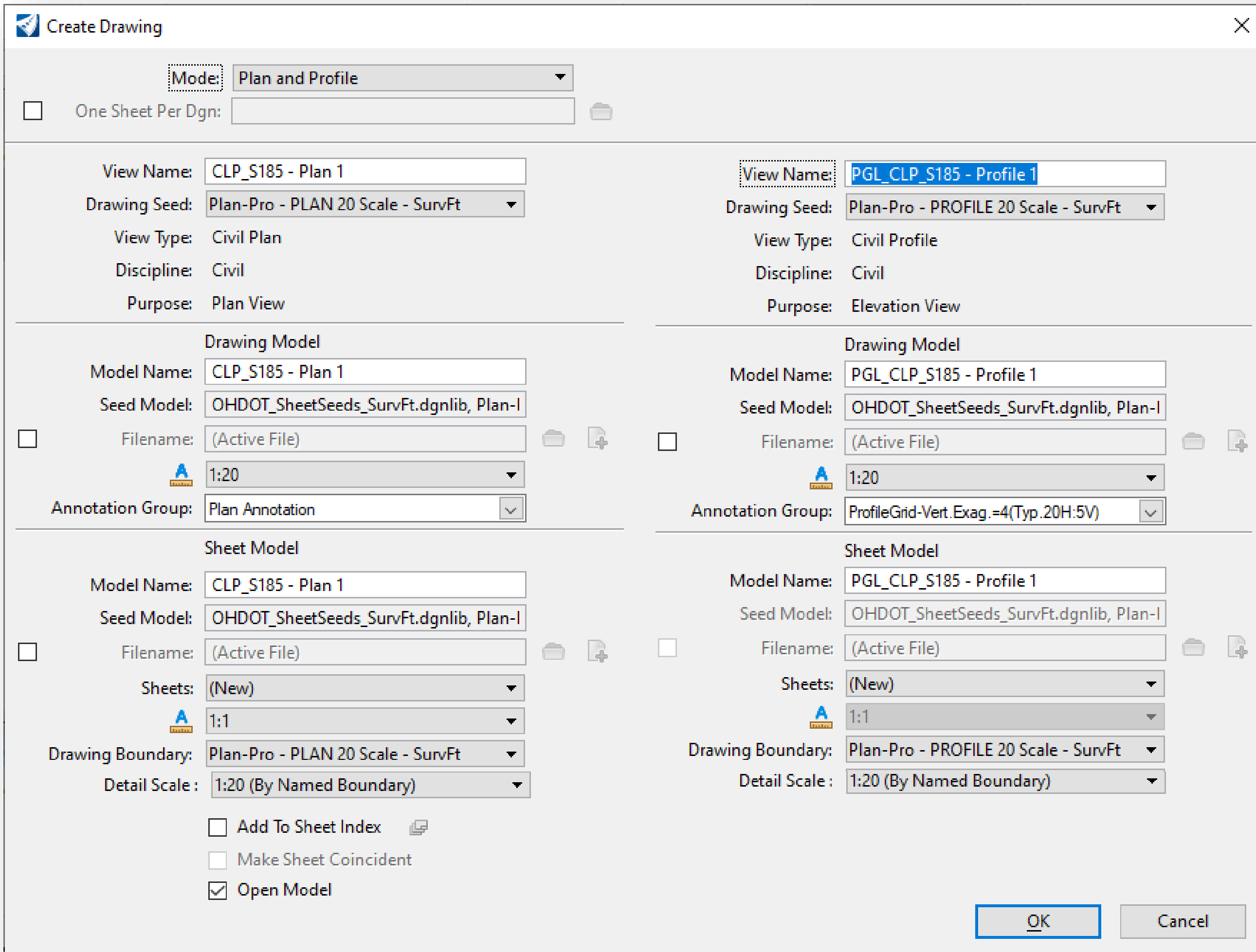
  Available Profile Height = 45

  Occasionally other horizontal scale values are used, such as 1:5, 1:10, 1:25, 1:40, etc... In each case, the user must set the **Vertical Exaggeration** to the desired value. The **Available Profile Height** is adjusted by the software corresponding to the **Detail Scale** and **Vertical Exaggeration** values entered.

For 5:1 Vertical Scale			For 10:1 Vertical Scale	
Horizontal Sheet Scale	**Vertical Exaggeration**		**Horizontal Sheet Scale**	**Vertical Exaggeration**
5	1		5	0.5
10	2		10	1
20	4		20	2
25	5		25	2.5
40	8		40	4
50	10		50	5

- Toggle on the **Create Drawing** option.

- Issue a data point (left click) in the profile view. The named boundaries are displayed in the view. Issue a second data point to accept the named boundary placement. The boundaries are drawn in the profile view and the **Create Drawing** dialog is opened.



The left side of the *Create Drawing* dialog lists parameters related to the plan view named boundaries. The right side corresponds to the profile named boundaries. Most of the parameters for the *Create Drawing* dialog have been predefined in the DGN Library for the selected **Drawing Seed**.

If you wish to create sheets in multiple files, toggle the **One Sheet Per DGN** option on and set the path for the sheet file location.

The **View Name** parameters are used for the named views that will be created in the container file as part of the clipping process. These parameters are not normally changed by the user.

Drawing Model parameters

The **Drawing Model Name** is used as the seed name for each of the drawing models that will be created, plan and profile. The name is incremented by the software. In most cases, the default name is used.

Set the **Annotation Scale** value for the **Drawing Model** to the desired value. This parameter should default to the same value used for the **Detail Scale** in the ***Place Named Boundary*** dialog, but it is good practice to review the value to ensure it is set correctly.

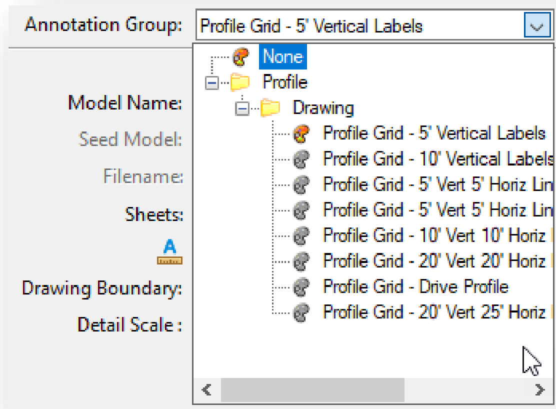
Sheet Model parameters

The **Sheet Model Name** entered for the Plan sheet model (left side of the dialog) is also the name that is used for the created design files when the **One Sheet Per Design** option is selected.

Do not change the **Annotation Scale** value for the **Sheet Model**. Leave this value at 1:1 for both plan and profile, as shown above.

The **Profile Annotation Group** setting (shown at right) will default to an appropriate annotation group for a 4:1 vertical exaggeration. If a different vertical exaggeration value was used when placing the profile named boundaries, different annotation options are available as shown at left.

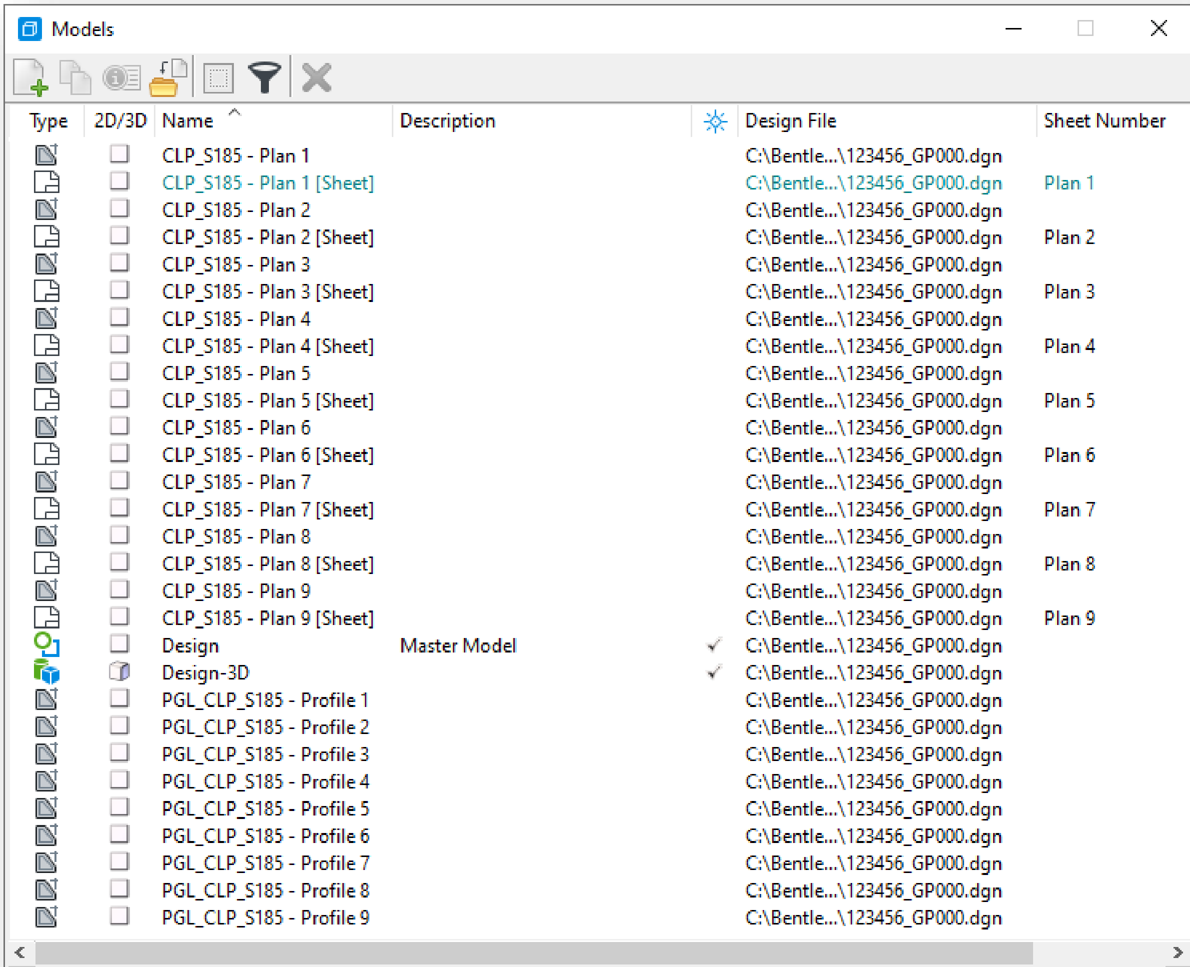
For sheets with a 5:1 Vertical Exaggeration, select the Profile Grid – 10' Vertical Labels option from the drop-down menu.



Set the **Detail Scale** value for the **Sheet Model** to the desired value for both the plan and profile. This parameter should default to the same value used for the **Detail Scale** in the ***Place Named Boundary*** dialog, but it is good practice to review the value to ensure it is set correctly.

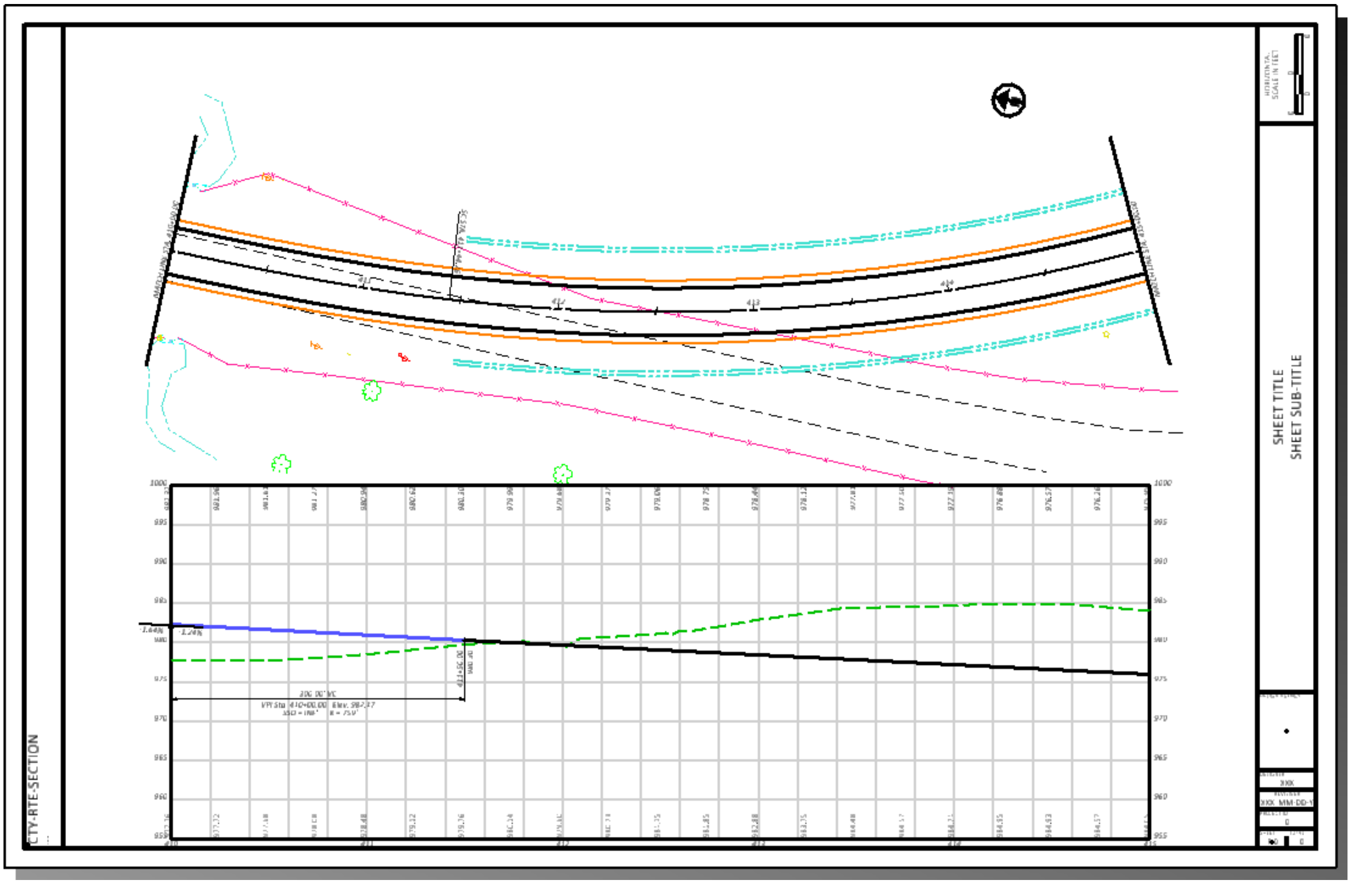
After reviewing the dialog settings, select **OK** to continue the sheet clipping process. If **Open Model** is toggled on, the application will open the last created file upon completion.

An example of the models generated for a **Plan Only** sheet configuration with all the sheets in one file is shown below.


Models

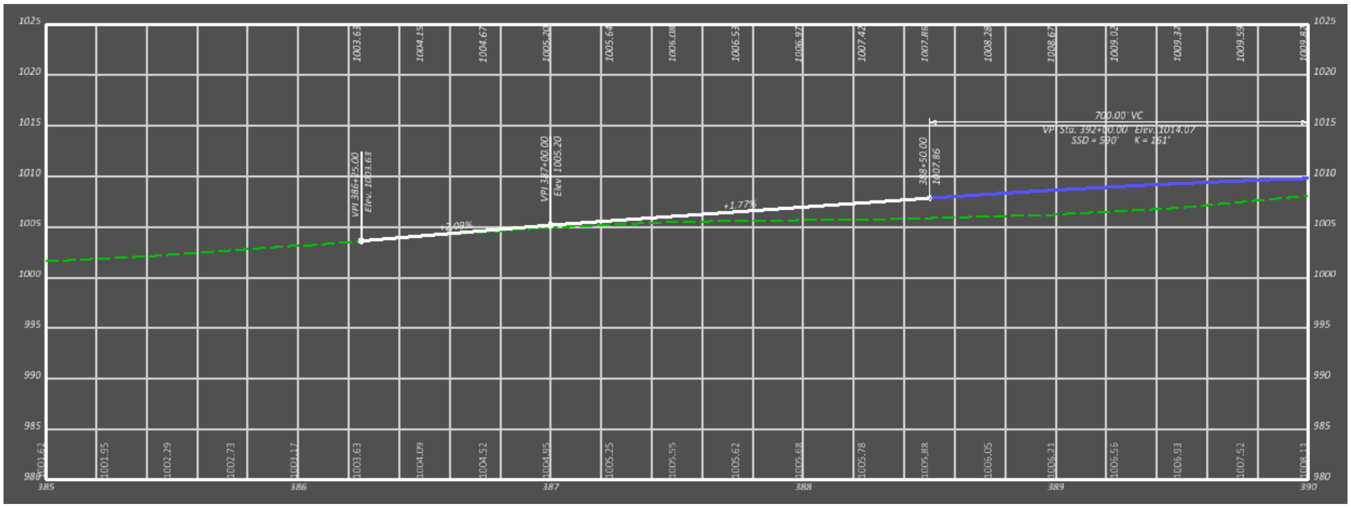
Type	2D/3D	Name	Description		Design File	Sheet Number
		CLP_S185 - Plan 1			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 1 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 1
		CLP_S185 - Plan 2			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 2 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 2
		CLP_S185 - Plan 3			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 3 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 3
		CLP_S185 - Plan 4			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 4 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 4
		CLP_S185 - Plan 5			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 5 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 5
		CLP_S185 - Plan 6			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 6 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 6
		CLP_S185 - Plan 7			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 7 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 7
		CLP_S185 - Plan 8			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 8 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 8
		CLP_S185 - Plan 9			C:\Bentle...\123456_GP000.dgn	
		CLP_S185 - Plan 9 [Sheet]			C:\Bentle...\123456_GP000.dgn	Plan 9
		Design	Master Model	✓	C:\Bentle...\123456_GP000.dgn	
		Design-3D		✓	C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 1			C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 2			C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 3			C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 4			C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 5			C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 6			C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 7			C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 8			C:\Bentle...\123456_GP000.dgn	
		PGL_CLP_S185 - Profile 9			C:\Bentle...\123456_GP000.dgn	

An example of a plan and profile sheet is shown below.


SHEET TITLE
SHEET SUB-TITLE
CTY-RTE-SECTION

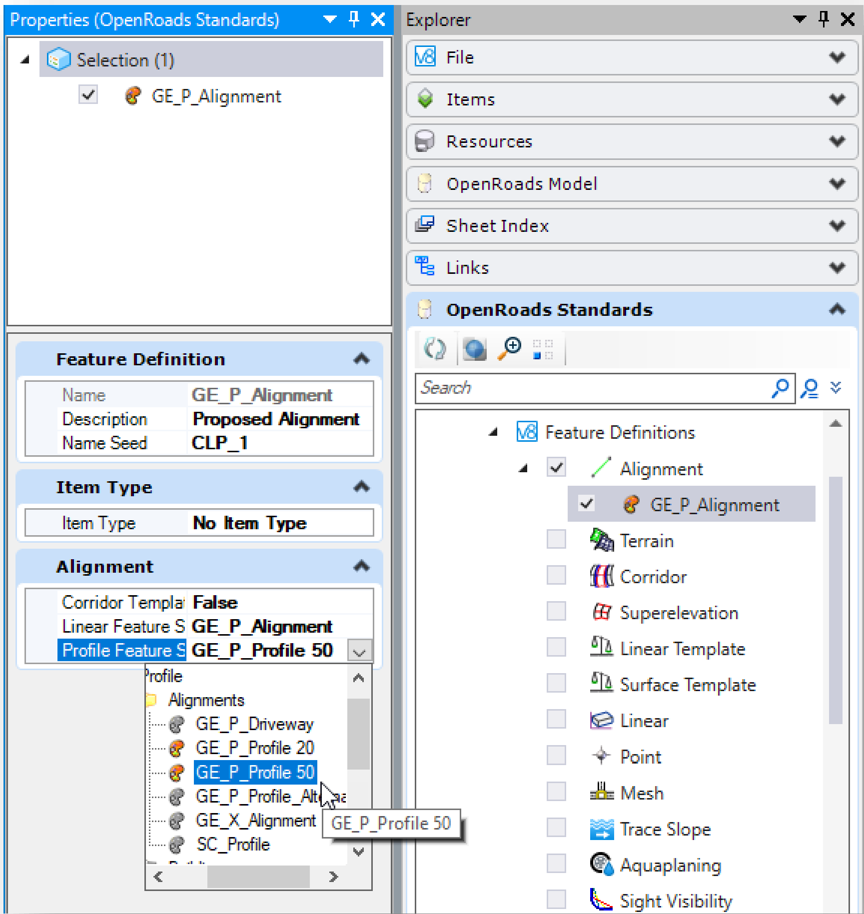
Profile Annotations

The drawing model for the profile contains the grid lines and annotations, like the example below.



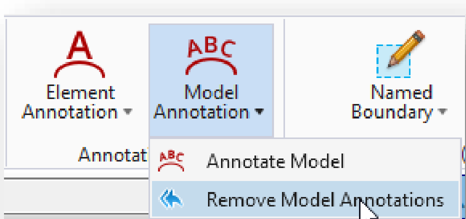
The annotation for the proposed profile elevations (shown across the top of the profile grid) is controlled by the Feature Definition assigned to profile itself in the BK design file. This feature definition defaults to 25' increments as shown above. This may not be desirable for 50-Scale plan and profile sheets and can be changed by the user. In the BK file, take the following steps to change the **Annotation Group** parameter:

- Open the BK file containing the profile
- Open the *Project Explorer* dialog
- From the *Project Explorer*, select the **OpenRoads Standards** tab.
- Expand the active design file to select the **Feature Definitions > Alignment > GE_P_Alignment** option as shown below
- In the *Properties* dialog, change the **Profile Feature** to the **GE_P_Profile 50** option as shown below

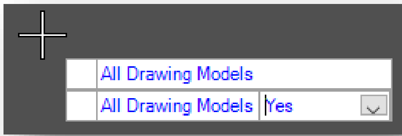


After making this change, it is necessary to redraw the annotation for the profile Drawing Models. Take the following steps in the sheet design file:

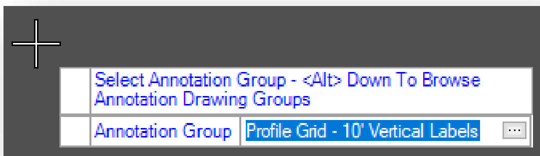
- Open the drawing model for the first profile
- From the **Drawing Production** tab, select **Model Annotation > Remove Model Annotations** as shown below



- When prompted, on the cursor menu, select **Yes** to remove the annotations from all the profile drawing models.



- After removing the annotations, redraw the annotation by selecting the **Model Annotation > Annotate Model** command from the **Drawing Production** tab
- When prompted to annotate all the drawing models, choose **Yes**
- Use the **Profile Grid – 10’ Vertical Labels** annotation group



The annotation is redrawn according to the select annotation group and the profile feature.

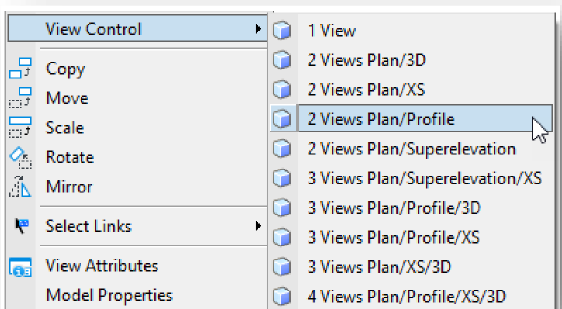
Single or Dual Profile Sheets

OpenRoads provides options to clip a sheet containing a single profile grid on the sheet, or two continuous profile grids across the top and bottom of a sheet. The following options are available:

- Pro-Pro
- Profile Only

The process is like creating named boundaries for the profile view detailed above. A profile view must be displayed to place the named boundaries. Take the following steps to open a profile window:

- Hold down the right mouse button with the cursor positioned in a MicroStation view window until the pop-up menu appears. Select the **2 Views Plan/Profile** option as shown below.

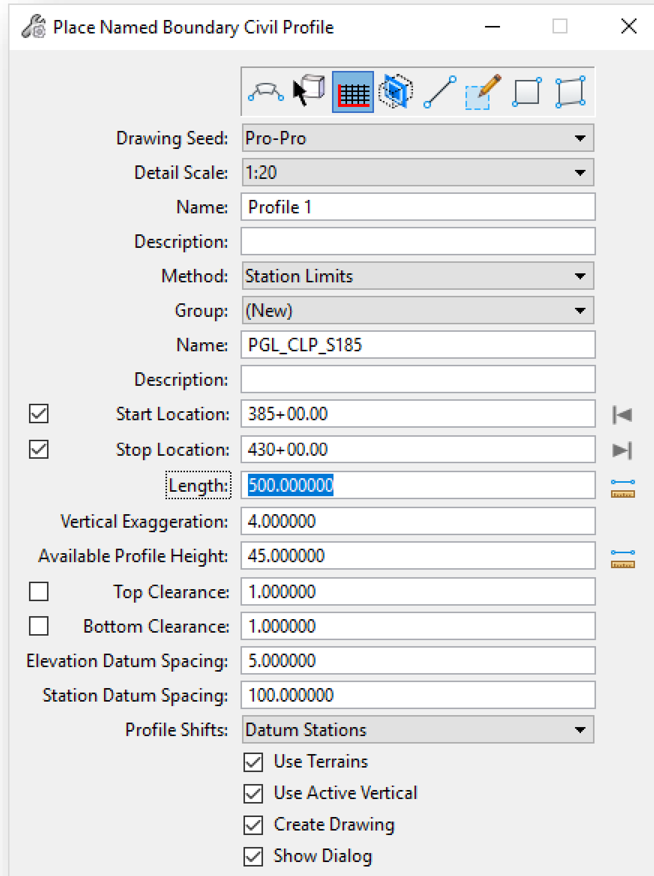


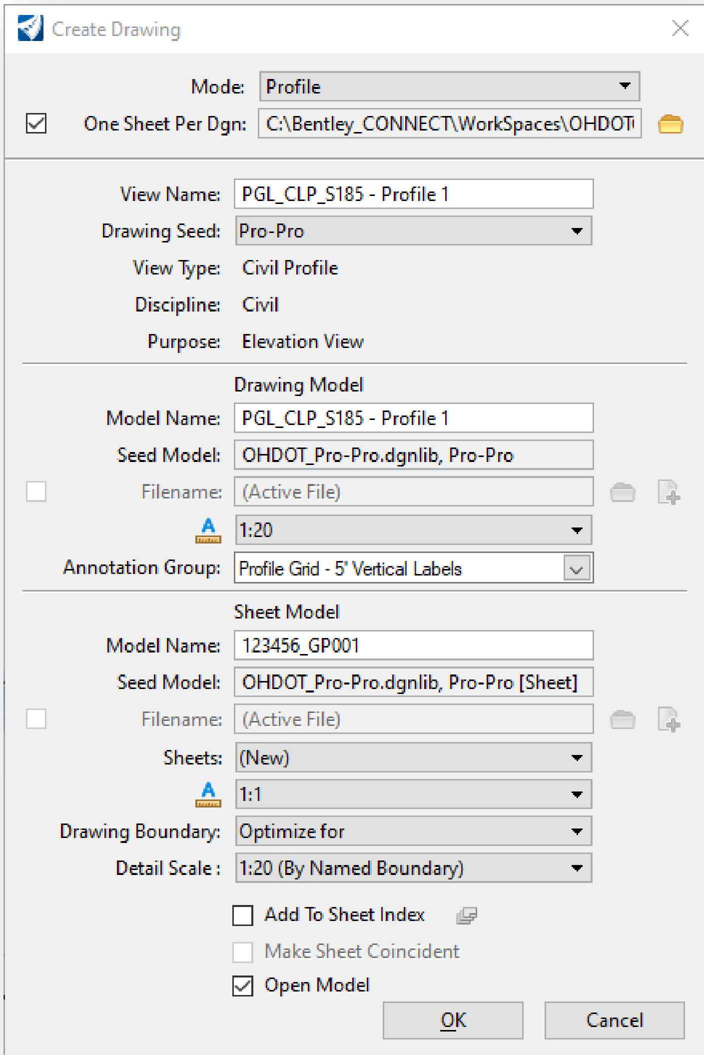
- Follow the prompts to open a profile window.
    - Select **OK** when prompted to **Create a Dynamic Profile View**
    - Locate the plan element
    - Issue a left-click in the newly opened profile view to display the profile information for the selected alignment

From the OpenRoads Modeling WorkFlow, select **Drawing Production > Named Boundaries**. On the *Place Named Boundary Civil Plan* dialog, select the **Civil Profile** icon.

From the **Place Named Boundary Civil Profile** dialog, take the following steps:

- Select the **Drawing Seed** (Pro-Pro or Profile Only)
- Select the **Detail Scale**
- Set the **Method** to **Station Limits**
- Left click in the **Start Location** dialog item
- Identify the profile view by issuing a left mouse-click in the profile view
- Define the **Start Location** and **Stop Location** parameters
- Set the **Vertical Exaggeration** value to **4** for 20-scale sheets, **5** for 50-scale sheets. The **Available Profile Height** parameter is adjusted automatically. See the Plan and Profile sheet section for more information.
- Issue another left mouse-click in the profile view to place the named boundaries and initiate the process. The *Create Drawing* dialog is opened as shown on the following page.





If you wish to create sheets in multiple files, toggle the **One Sheet Per DGN** option on and set the path for the sheet file location.

The **View Name** parameter is used for the named views that will be created in the container file as part of the clipping process. This parameter is not normally changed by the user.

Drawing Model parameters

The **Drawing Model Name** is used as the seed name for each of the drawing models that will be created. The name is incremented by the software. In most cases, the default name is used.

Set the **Annotation Scale** value for the **Drawing Model** to the desired value. This parameter should default to the same value used for the **Detail Scale** in the ***Place Named Boundary*** dialog, but it is good practice to review the value to ensure it is set correctly.

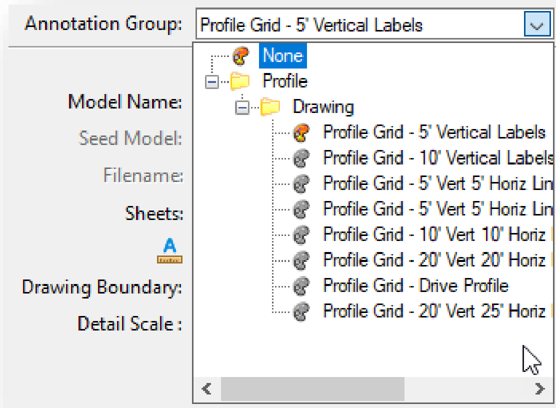
Sheet Model parameters

The **Sheet Model Name** is also the name that is used for the created design files when the **One Sheet Per Design** option is selected.

Do not change the **Annotation Scale** value for the **Sheet Model**. Leave this value at 1:1 as shown above.

The **Profile Annotation Group** setting (shown at right) will default to an appropriate annotation group for a 4:1 vertical exaggeration. If a different vertical exaggeration value was used when placing the profile named boundaries, different annotation options are available as shown at left.

For sheets with a 5:1 Vertical Exaggeration, select the Profile Grid – 10' Vertical Labels option from the drop-down menu.

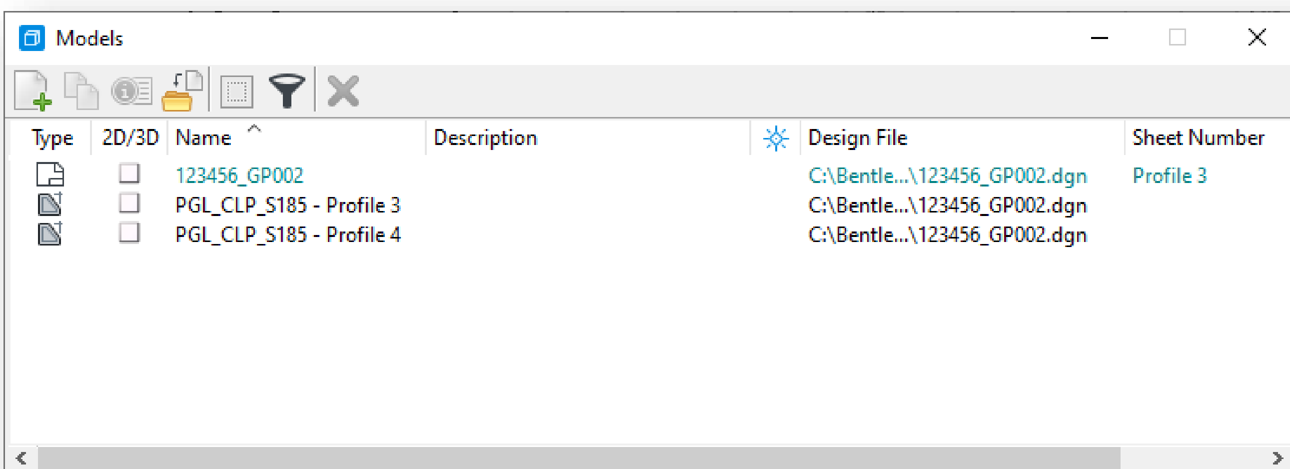


Set the **Detail Scale** value for the **Sheet Model** to the desired value. This parameter should default to the same value used for the **Detail Scale** in the ***Place Named Boundary*** dialog, but it is good practice to review the value to ensure it is set correctly.

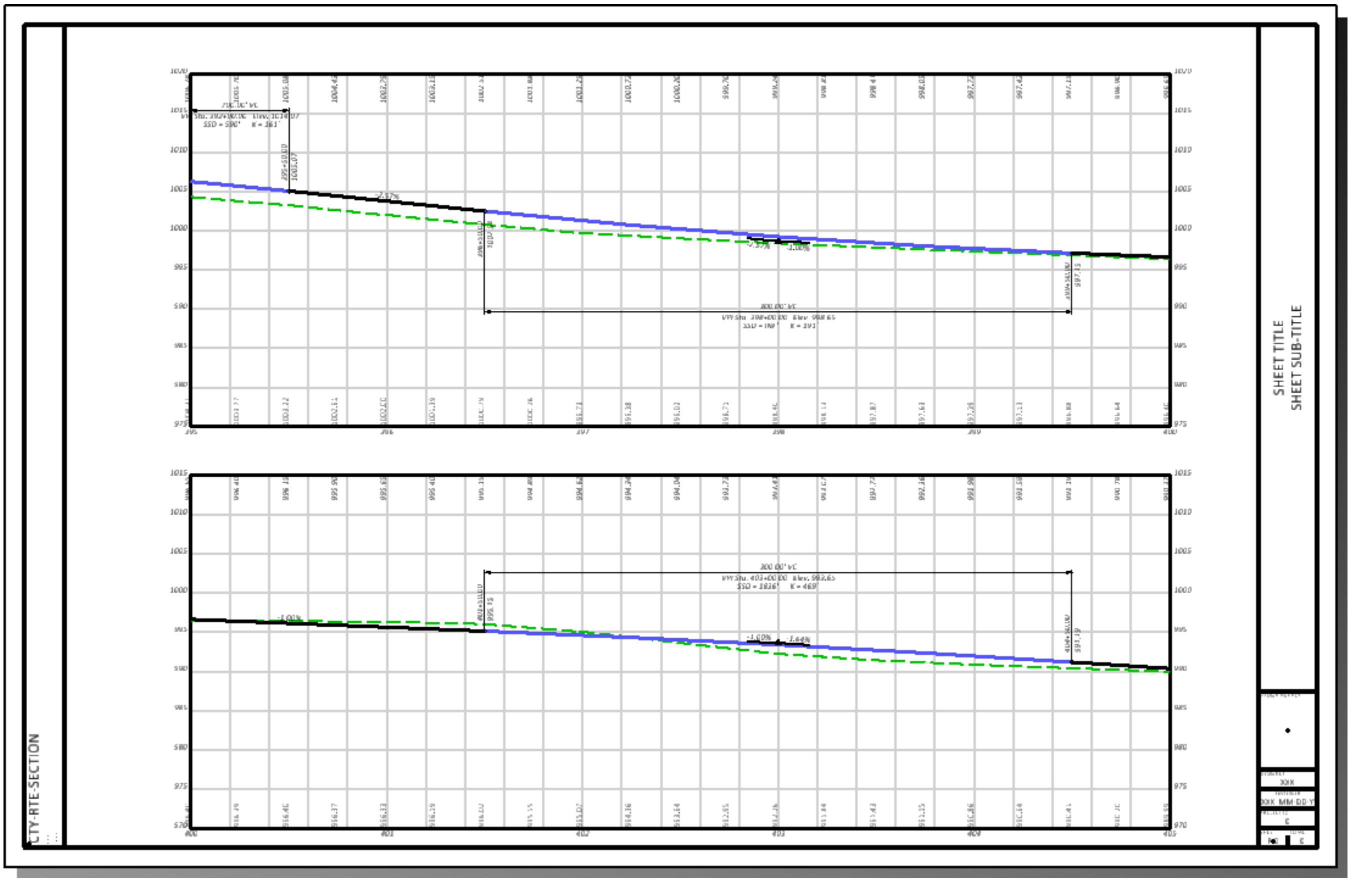
After reviewing the dialog settings, select **OK** to continue the sheet clipping process. If **Open Model** is toggled on, the application will open the last created file upon completion.

A drawing model is created for each of the named boundaries. The drawing models are referenced to the sheet model to generate the completed sheets. The drawing models contain the annotations for the profile. See the Profile Annotations section for details.

An example of the models generated for a Pro-Pro sheet configuration when using the **One Sheet per DGN** option is shown below. Note the sheet model name is the same as the design file name.



An example of a pro-pro sheet is shown on the following page.

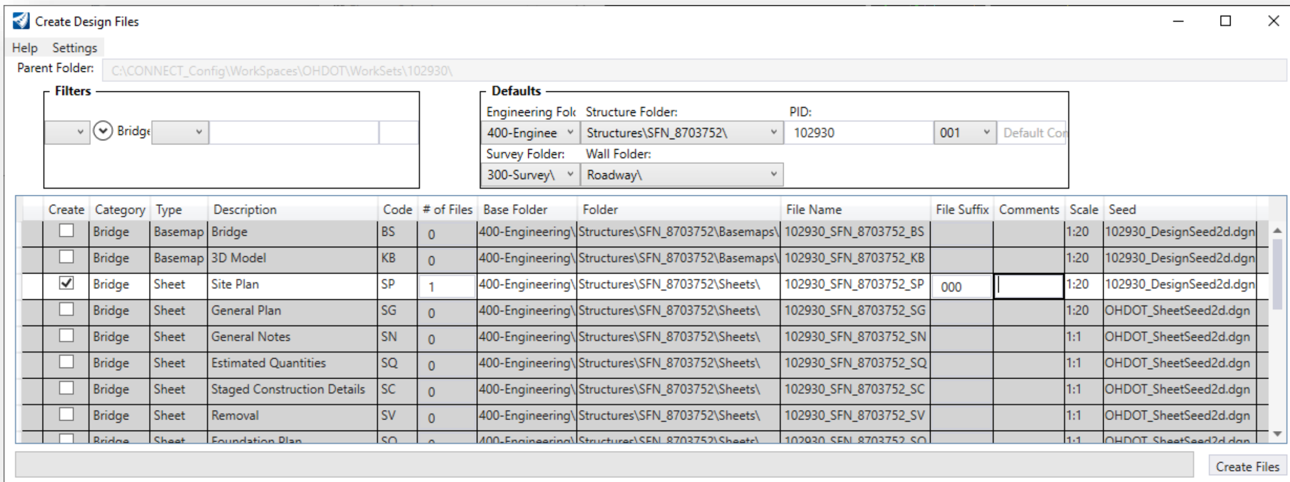

SHEET TITLE
SHEET SUB-TITLE
CTY-RTE-SECTION

Bridge Site Plan

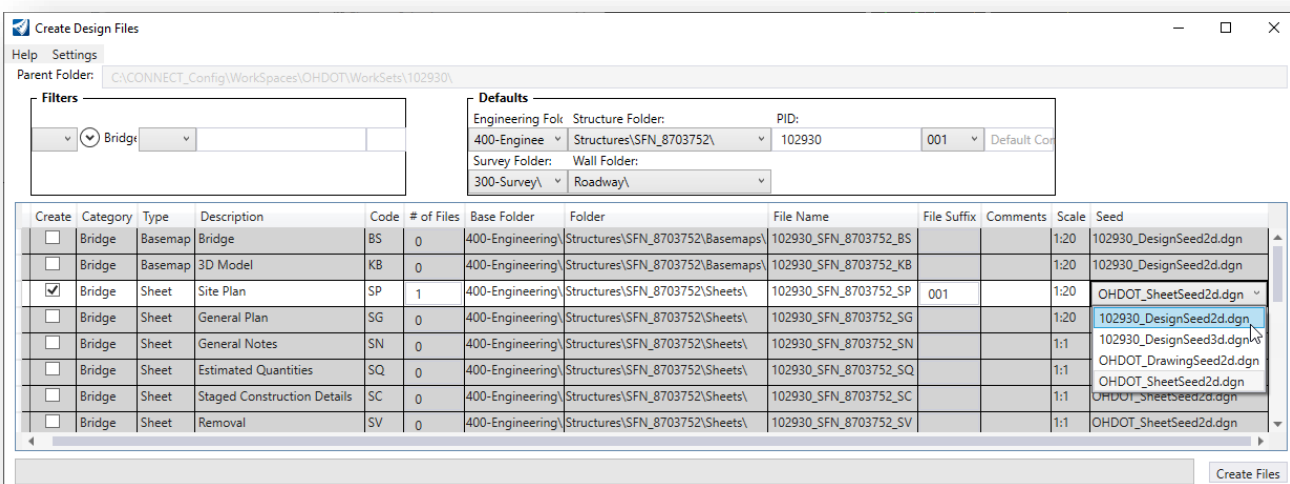
Bridge site plans can be generated using a container file or directly in the SP file. When the bridge site plan will consist of more than one sheet, the preferred process is to use a container file. See the *File Types and Usage* section of this document for more information on container files.

The **OHDOT Create Design Files** application is used to create the design file as shown below.

If a container file is used to generate multiple sheets, key in the appropriate 000, 100, 200, etc. **File Suffix**. The application will automatically set the **Seed** file to the Design Seed as shown below.



If a container file is not used, the **File Suffix** is set to the appropriate number for the site plan design file, **001** in the example below. The application will default to the Sheet Seed. Be sure to change the **Seed** file to the Design Seed as shown below.

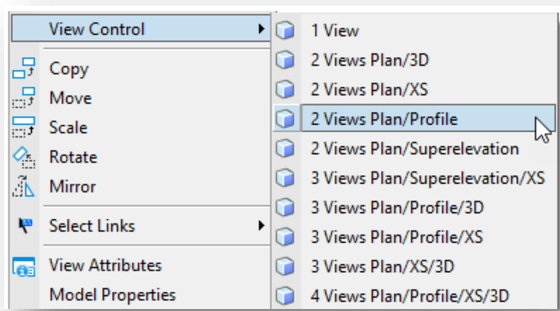


After creating the file, the next step is to attach the necessary design file references for the site plan.

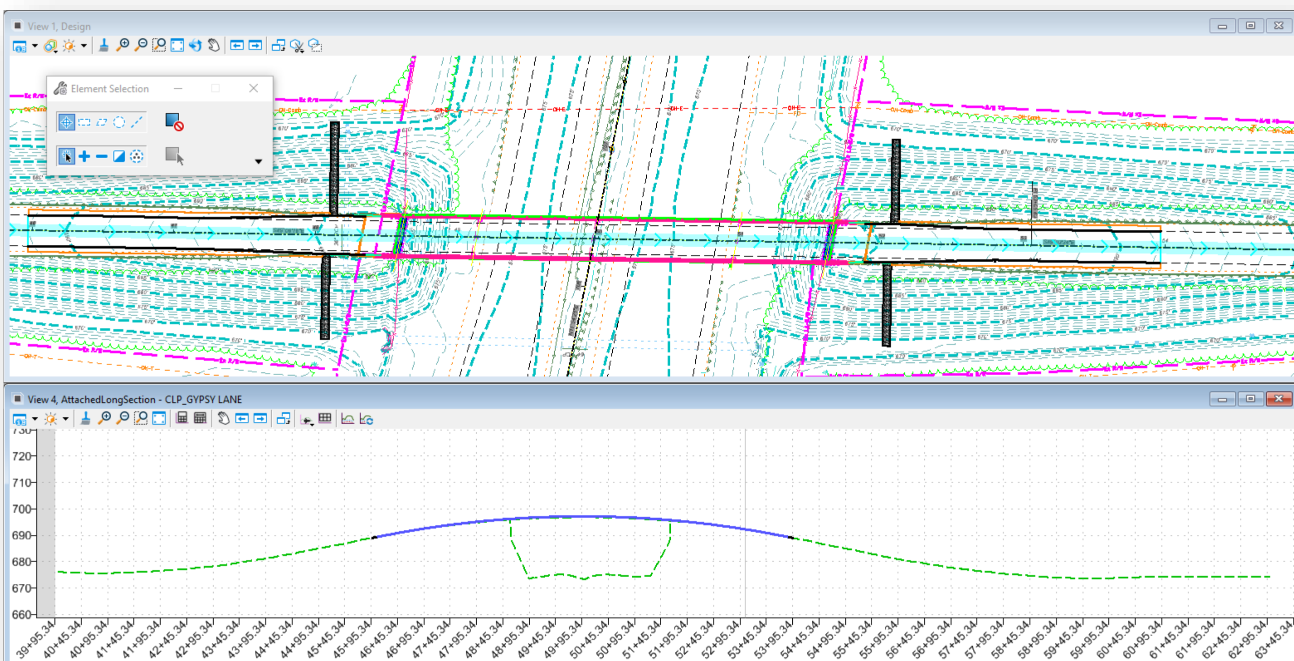
Like the plan and profile sheet clipping process, configure the MicroStation view windows to display both plan and profile information. This is facliltated using the right-mouse button menu. Hold down the right mouse button with the cursor positioned in a MicroStation view window until the pop-up menu appears. Select the **2 Views Plan/Profile** option as shown below.

Follow the prompts to open a profile window.

- Select **OK** when prompted to **Create a Dynamic Profile View**
- Locate the plan element
- Issue a left-click in the newly opened profile view to display the profile information for the selected alignment

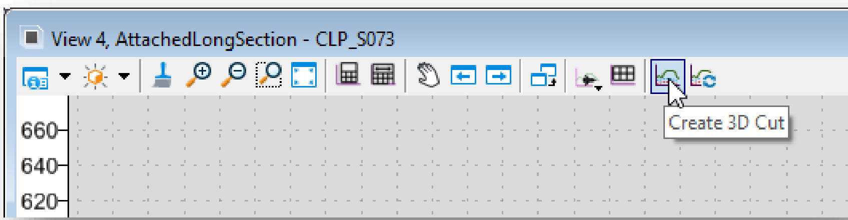


When complete two view windows displaying the plan and profile are opened like the example below.

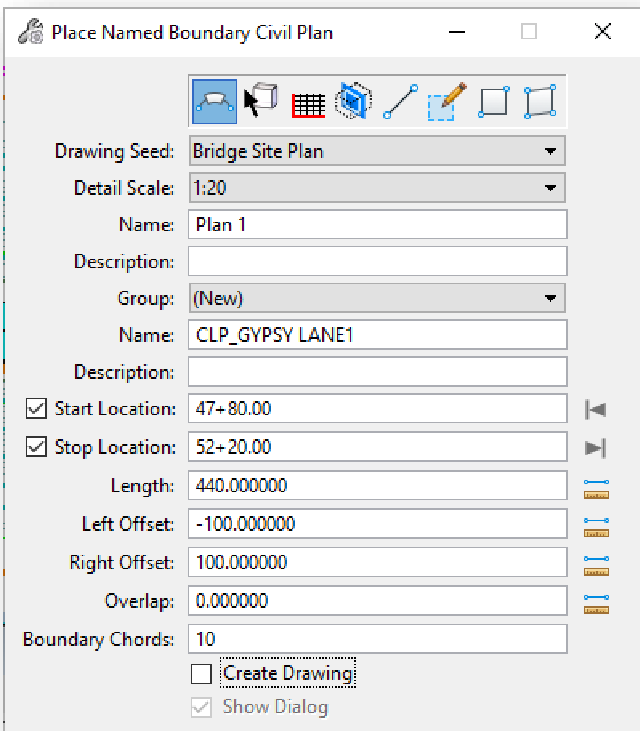


In this example, the horizontal geometry is defined in the \roadway\basemaps\102930_BK001.dgn file. Graphics that have been drawn in the profile window of the BK file will also be displayed here.

This example will make no attempt to define the bridge profile graphics or recommend a best practice for generating these graphics. The graphics can be drawn in the profile view in the SP design file. The **Create 3D Cut** command may also be used to read the Design-3D model and will display and 3d graphics encountered in the model.



The named boundaries for plan and profile sheet clipping will be placed as two separate actions. From the **OpenRoads Modeling** WorkFlow, select **Drawing Production > Named Boundary** to initiate the process. The *Place Named Boundary Civil Plan* dialog is opened.



Select the first icon, **Civil Plan** to begin the process of placing the plan view named boundaries.

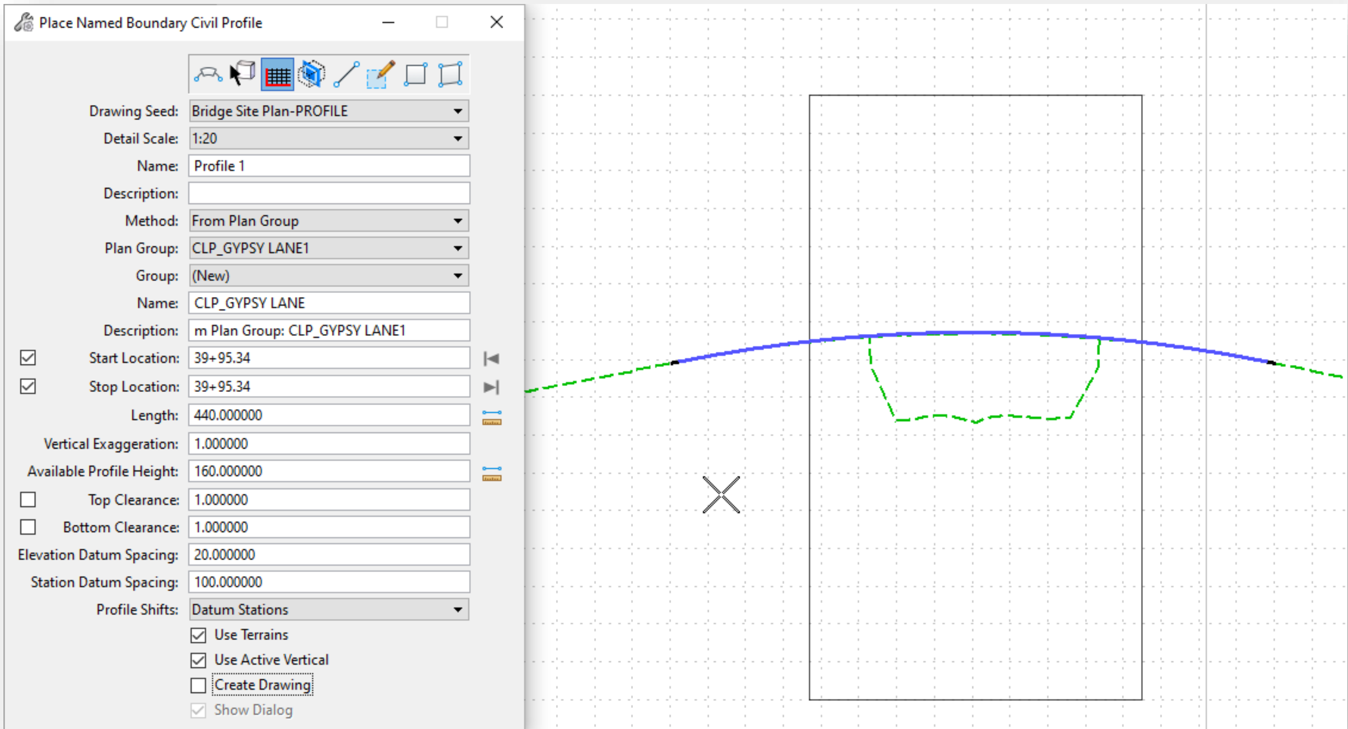
The **Drawing Seed** is used to select the configuration for various plan view named boundaries. Select the **Bridge Site Plan** option.

The process to create the named boundaries for the plan view is the same as described previously in this document.

Be sure to toggle ***off*** the **Create Drawing** option before placing the named boundaries as shown at left. It is necessary to generate the profile named boundaries before creating the drawings.

After the named boundaries for the plan view have been placed, take the following steps to create the named boundaries for the profile view:

- From the **OpenRoads Modeling** WorkFlow, select **Drawing Production > Named Boundary** to initiate the process. The *Place Named Boundary* dialog is opened.
- Select the third icon, **Civil Profile** to begin the process of placing the plan view named boundaries



- Select the **Drawing Seed** that corresponds to the previously selected plan view drawing seed. In this case, select **Bridge Site Plan – PROFILE** as shown at left.

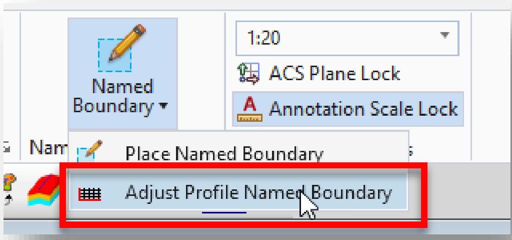
- Set the **Detail Scale** as desired. The **Vertical Exaggeration** and **Available Profile Height** parameters are automatically defined based on the selected **Detail Scale**. See the Plan and Profile Sheet section for more information.

- Set the placement **Method** parameter to **From Plan Group**.

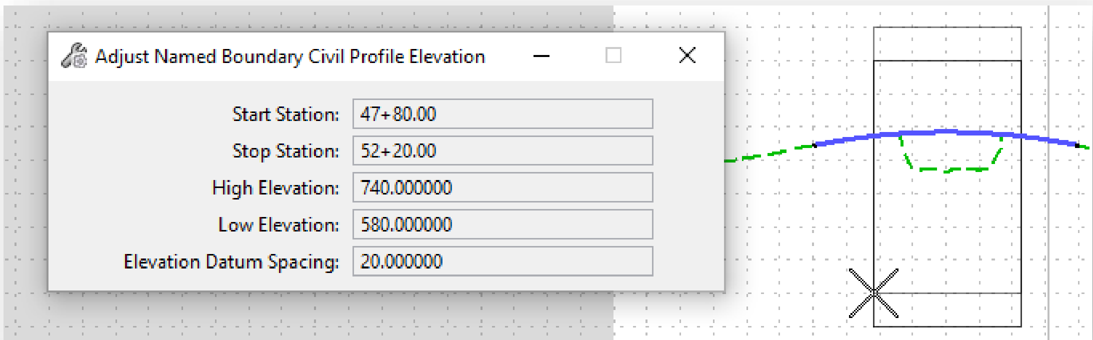
  The boundary is displayed as shown in the example above. The software determines the vertical location of the profile named boundary. Depending on the vertical extent of the profile graphics, the default placement location may need to be adjusted.

  - If the displayed boundary location is acceptable, toggle on the **Create Drawing** option before placing the boundary to proceed to the next step.
  - If the profile vertical location needs to be adjusted, toggle off the **Create Drawing** option before placing the named boundary. After placing the boundary, the **Adjust Profile Named Boundary** tool is used to adjust the vertical location of the boundary element as described below.

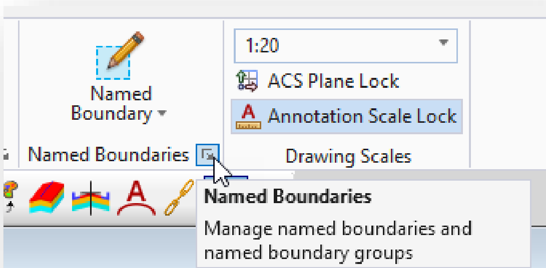
To adjust the named boundary vertically, select the **Adjust Profile Named Boundary** command from the **Drawing Production** tab as shown below.



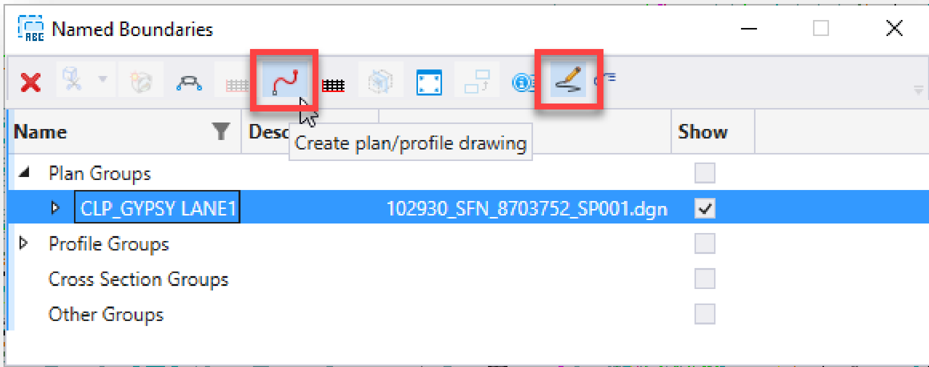
Select the named boundary element to adjust. The named boundary floats on the cursor and can be adjusted vertically based on the **Elevation Datum Spacing** parameter. Issue a left click (data point) to move the named boundary to the desired location.



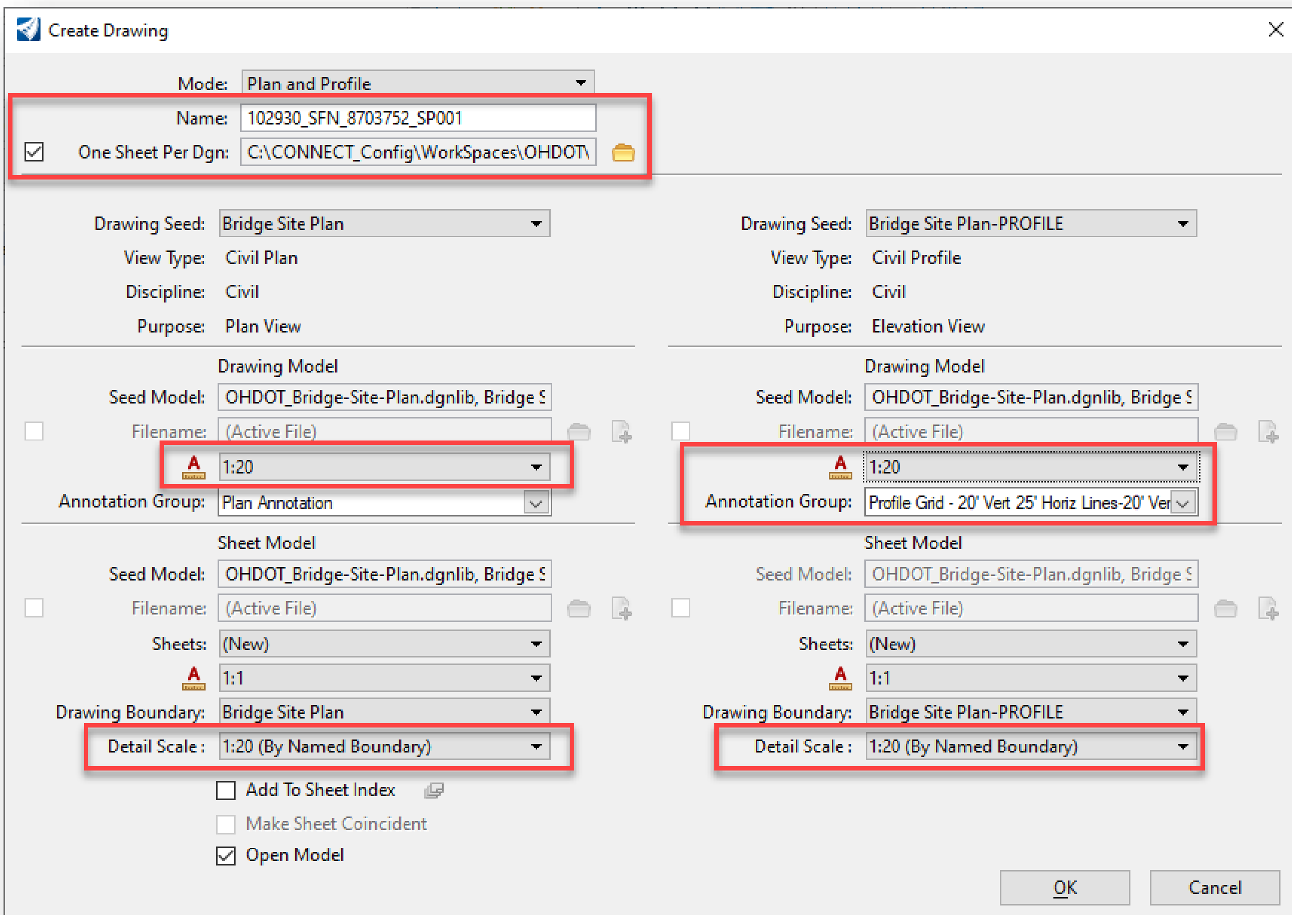
After adjusting the named boundary, the **Manage Named Boundaries** command is used to resume the sheet clipping process. Select the small icon in the lower right corner of the Named Boundaries group as shown below.



From the *Named Boundaries* dialog, select the **Plan Group** to process. Toggle on the **Show the Create Drawing** icon and then select the **Create plan/profile drawing** icon.



The *Create Drawing* dialog is opened as shown below.



The parameters, highlighted above, must be adjusted before selecting the **OK** button.

- If you are using a container file to place the named boundaries and will be generating separate SP files:

- Enter a **Name** according to the OHDOT Standard File names for the sheet type you wish to create. See the **Location & Design Manual, Volume 3 – Highway Plans, Section 1204.3.4** for more information.
- Toggle on the **One Sheet Per Design** option and select the discipline\sheets\ folder where the sheet design files are to be created.

- Set the **Drawing Model Annotation Scale** parameters, and the **Sheet Model Detail Scale** parameters to the intended scale for both the plan and profile settings as shown in the example above.

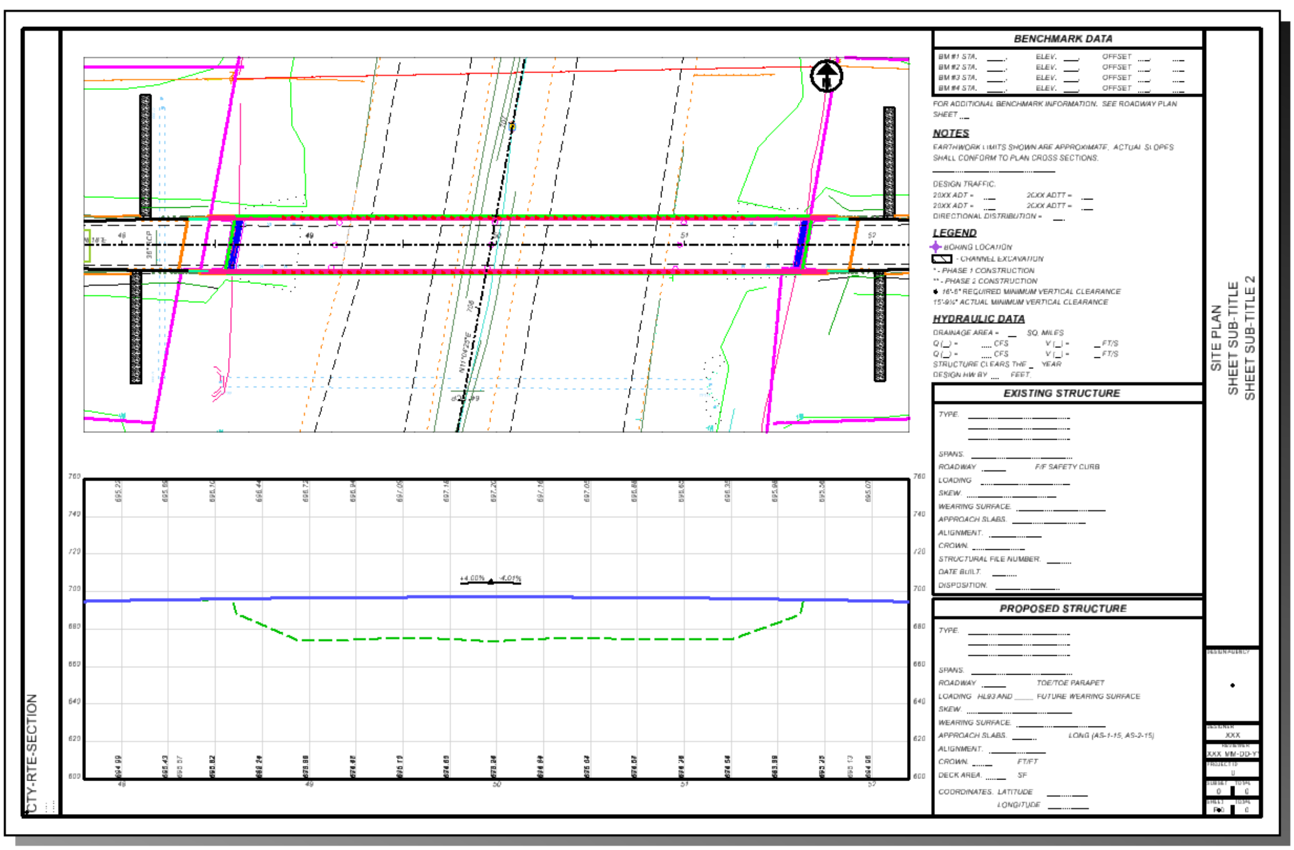
- The profile **Annotation Group** parameter is defaulted to the appropriate value for 20 Scale sheets with a 4:1 vertical exaggeration.

  For 20-Scale bridge site plans with a 1:1 vertical exaggeration, select the **Profile Grid – 20' Vert 25' Horiz Lines-20' Vert 100' Horiz Labels** option from the drop-down menu.

  For 10-Scale bridge site plans, select the **Profile Grid - 10' Vert 10' Horiz Lines-10'Vert 50'Horiz Label** option from the drop-down menu.

- Select **OK** to initiate the process.

The bridge site plan sheet is generated like the example below.



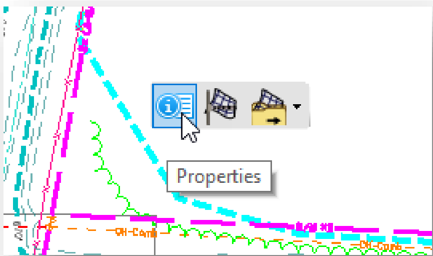
Notes:

- If using a **Container** file for the named boundaries and the references:
    - Each sheet design file contains a **Drawing** model for each of the named boundaries, one for the plan and one for the profile
    - The **Drawing** models reference the original **Container** file
    - The **Sheet** model references the **Drawing** models
- If the named boundaries and the references are defined in the **SP** file:
    - Each **SP** design file contains a **Drawing** model for each of the named boundaries, one for the plan and one for the profile
    - The **Drawing** models reference the design model in the **SP** file
    - The **Sheet** model references the **Drawing** models
- In either case, the profile drawing model includes both the grid lines and the annotation for the profile

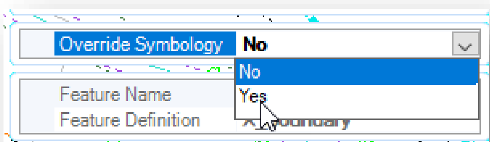
Terrain Model Contour Interval

If the existing ground terrain model contours will be show on the site plan, it is possible that the interval for the contour lines as saved in the survey "FB" design file may be set to display the contour interval that is required for the bridge site plan. Changing the contour interval display can be accomplished by using the terrain model **Override Symbology** setting.

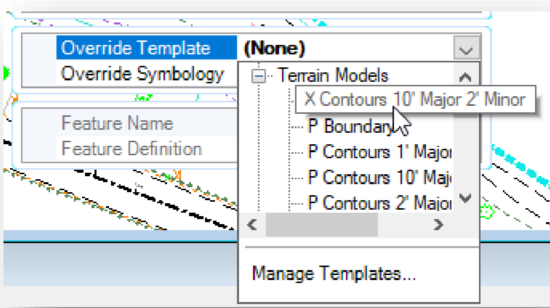
Select the terrain model boundary and choose the **Properties** command from the pop-up menu.



Change the **Override Symbology** setting to **Yes**.



The **Override Template** option is used to choose the Element Template with the contour interval that will be used to override the current settings as saved in the "FB" design file.

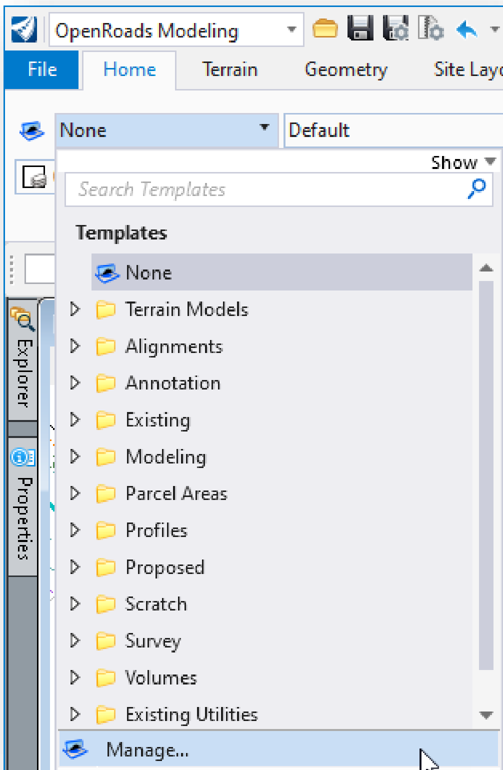


After selecting the element template, the changes are applied in the active design file. Note that the triangles may also be turned on when changing this setting and can be turned off using the MicroStation *Level Display* dialog.

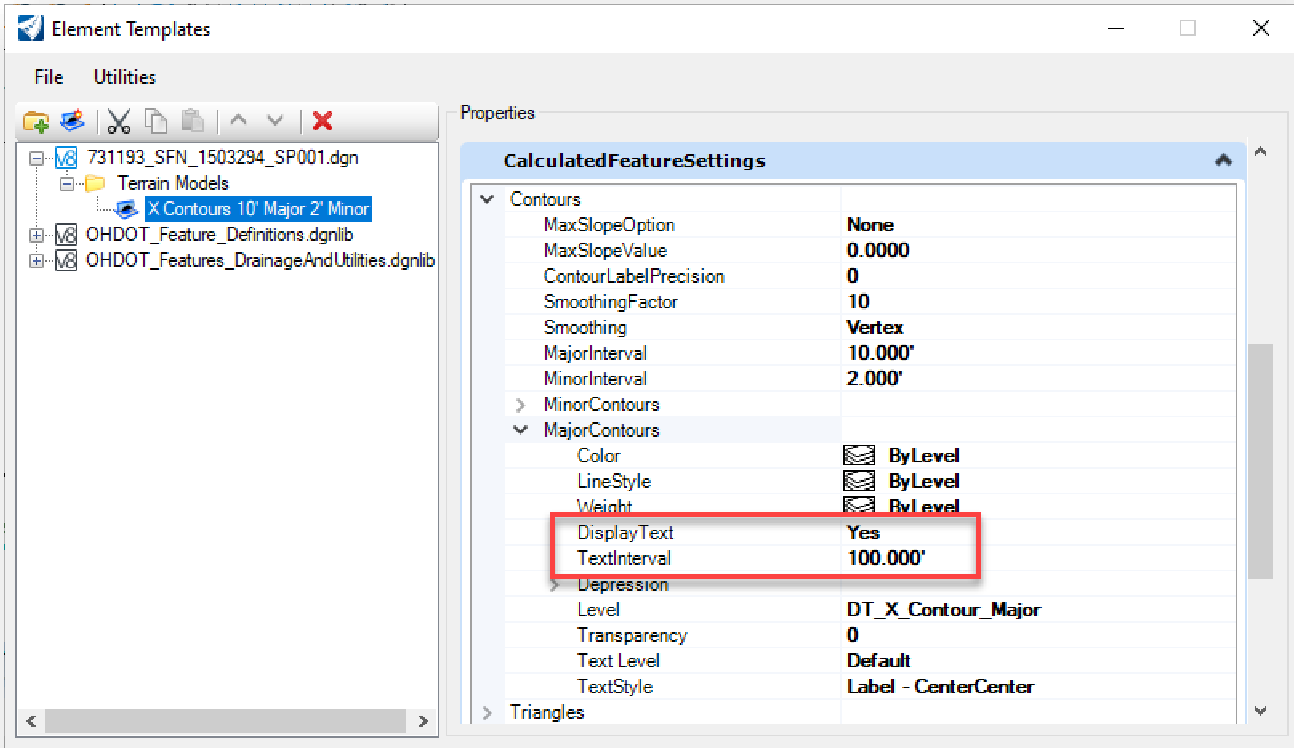
Note that OpenRoads adjusts the major contour text annotation location dynamically based on the view extents. This may result in text falling into locations that are undesirable. Unfortunately, there is no way to know where the text is going to fall on the printed sheet until the sheet is printed. There are two options to address this issue, both of which are accomplished by editing the element template that has ben copied into the active design file when the override template was selected.

1. Adjust the major contour text spacing. This option may not solve the problem as all it does is move the text to a different spacing interval.
2. Turn off the display of the major contour text and then manually annotate the major contour lines by placing the text at the desired location

In either case, the first step is to edit the element template that has been copied into the local file. From the **Home** tab, select the **Manage** command from the **Element Template** drop-down menu as shown below.

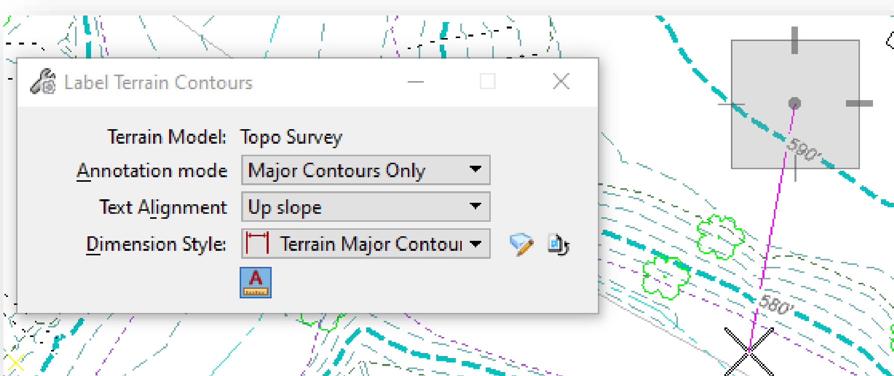


The *Element Templates* dialog is opened. Browse to the **Terrain Models** group to edit the template that had been copied into the design file as shown below.



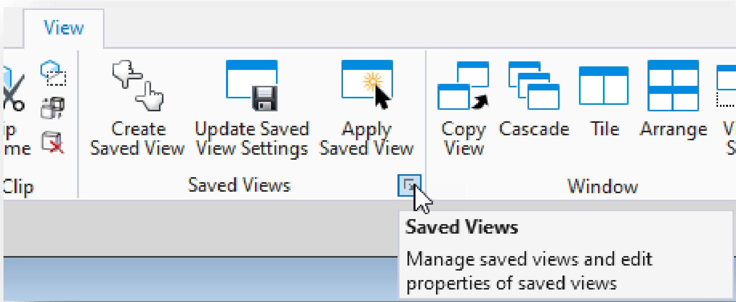
- Change the **TextInterval** option to a different value to adjust the text spacing.
- To turn off the text, toggle the **Display Text** option to **No**. Note: You may also have to change the **TextInterval** value for this change to take effect.

If you elect to turn off the text display, the **Terrain > Labeling > Label Terrain Contours** command is used to place contour labels at the desired locations by dragging a line through the contour(s) that are to be labeled. An example is shown below.

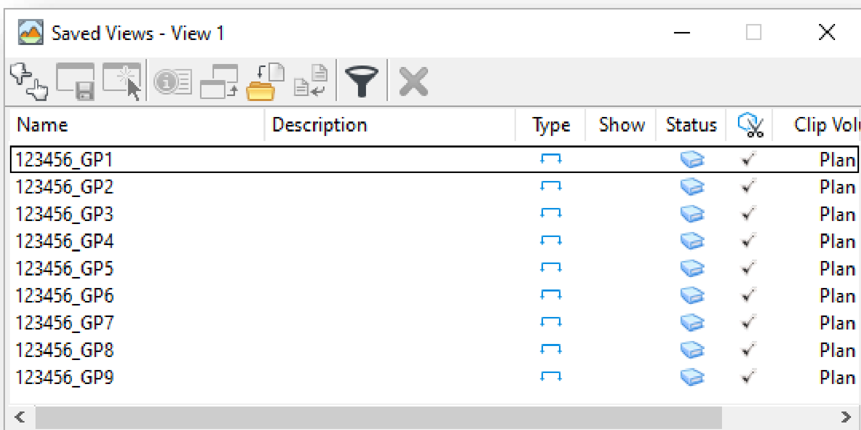


Named Boundaries and Saved Views

When the named boundaries are processed to generate the sheets, the software also creates a MicroStation saved view for each named boundary. The saved views can be reviewed by selecting **OpenRoads Modeling > View > Saved Views** as shown below.



The *Saved Views* dialog is opened, as shown below.



The saved views are named according to the name that is entered in the sheet generation process on the *Create Drawing* dialog.

Reprocessing Sheets

Occasionally it may be necessary to reprocess a set of sheets. The previously placed named boundaries can be reused, however, if the same name is used for the sheet files the software does not overwrite the previously created saved views, models, or design files. The user must delete the previously generated saved views, models, and design files before reprocessing the named boundaries.

If the sheets are all in one file, take the following steps before reprocessing the named boundaries:

- Open the *Saved Views* dialog. Delete the saved views corresponding to the sheets to be reprocessed
- Open the *Models* dialog. Delete all the drawing and sheet models corresponding to the sheets to be reprocessed

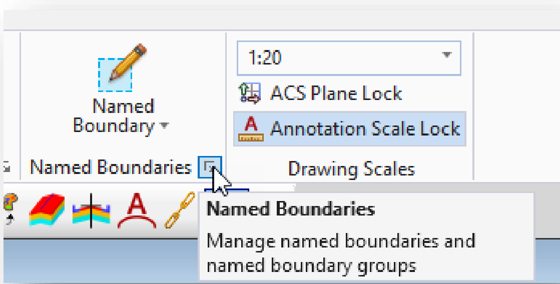
If the sheets are in separate design files, take the following steps before reprocessing the named boundaries:

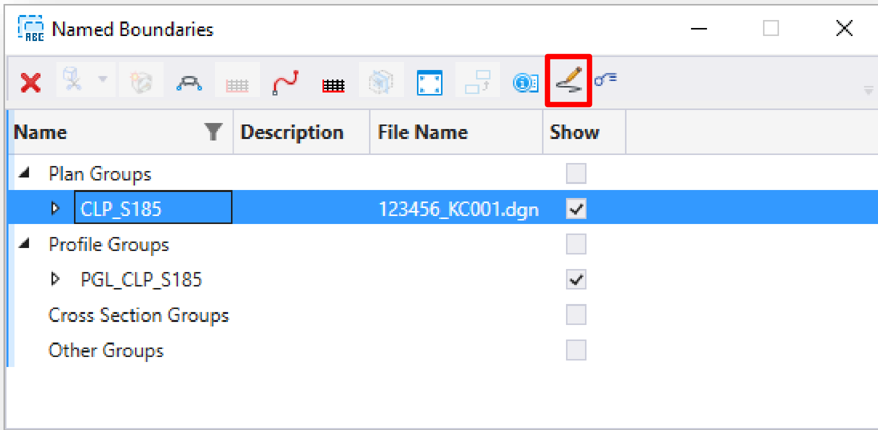
- In the KC design file, open the *Saved Views* dialog. Delete the saved views corresponding to the sheets to be reprocessed
- Delete all the sheet design files in the \discipline\sheets\ folder that are to be reprocessed

  Tip for ProjectWise users: Before deleting files, in ProjectWise Explorer right-click on the file name and choose the **Purge Local Copies** option.

The *Named Boundaries* dialog is used to reprocess the sheets from the previously placed named boundaries.

- From the **OpenRoads Modeling** WorkFlow, select **Drawing Production > Named Boundaries** as shown below. The *Named Boundaries* dialog is opened.





- Select the **Plan Group** to be processed.
- Toggle on the option to **Show the Create Drawing Dialog** 
- Select the appropriate icon to open the *Create Drawing* dialog (**Create plan drawing**, **Create Profile drawing**, **Create plan/profile drawing**). The *Create Drawing* dialog is opened and can be processed as previously detailed in this document.

Level Management

Managing Sheet Design File Levels from the Container File

Managing levels across multiple drawing and sheet models with multiple references (both direct and nested) can be confusing. When sheets are assembled, the reference files are attached as follows:

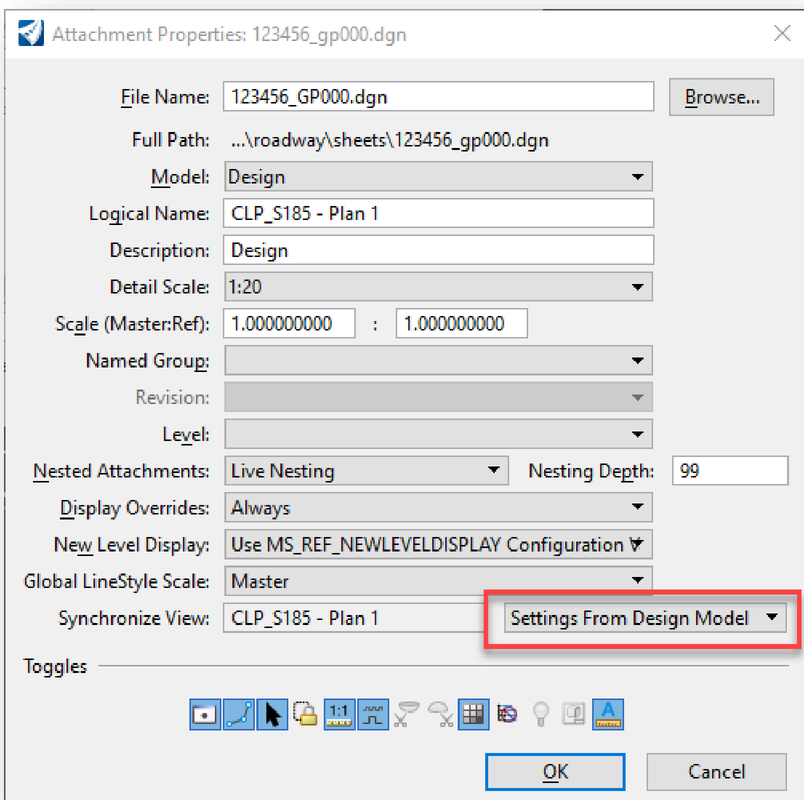
Basemaps > Container File (Design Model) > Sheet File (Drawing Model) > Sheet File (Sheet Model)

By default, the ODOT sheet clipping DGN Libraries are configured to allow users to manage the levels displayed on the completed sheets from the container file. This allows users to configure the levels displayed in the final sheets from a single location, the container file, with the changes reflected in every plan or profile sheet.

The 2021 Release 1 version of OpenRoads enhances this functionality with significant improvements:

- A new option has been added for reference attachments to synchronize the view settings from the attached file. The parameter is found in the ***Attachment Properties*** dialog for the selected reference file. Double-click the reference file name in the ***References*** dialog to access the ***Attachment Properties*** dialog shown below.

  The **Settings From Design Model** parameter has been configured as the default setting in the OHDOT Standards.



- The **Settings From Design Model** parameter also allows the view attributes settings in the container file to be reflected in the sheets. This allows turning off the display of construction class elements (Constructions) container file's View Attribute settings to carried through to all the sheets.
- Adding or removing reference attachments is reflected in the completed sheets

Tip: Use **View 1** when turning on or off plan view levels in the container file for the level changes to carry through to the drawing and sheet models.

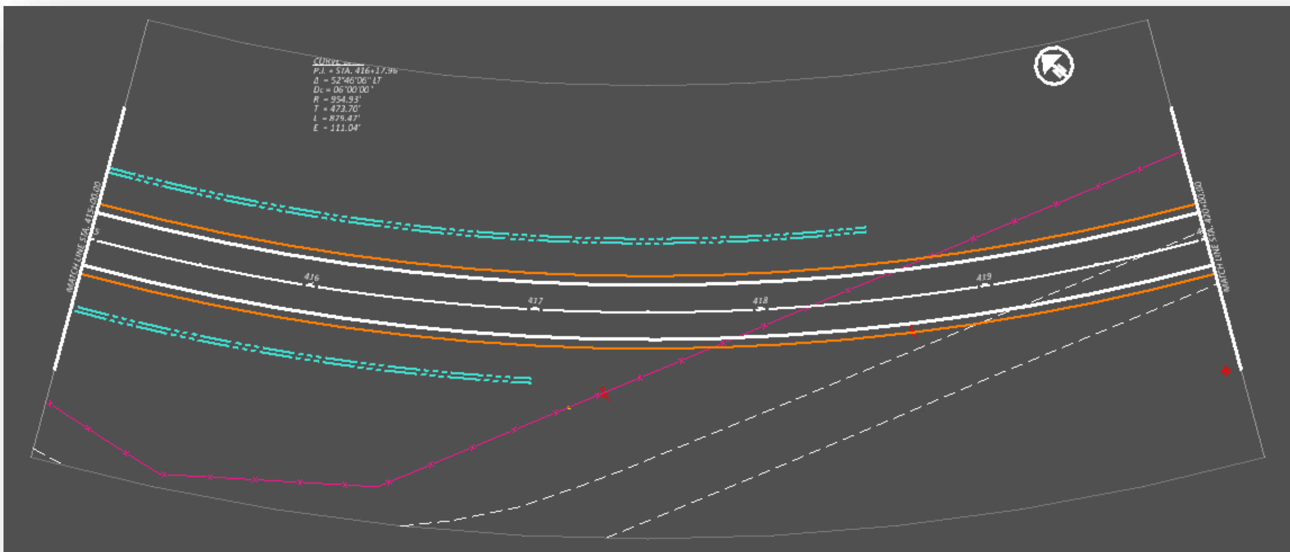
Sheet Annotation

Bentley's recommended workflow is for sheet annotations to be placed in the drawing models, not in the sheet models.

Plan View Alignment Annotation

Alignments are often annotated in the geometry basemap (BK) design file. These annotations can be shown on the completed sheets, however, editing the annotation in the BK file may impact other sheets in the WorkSet.

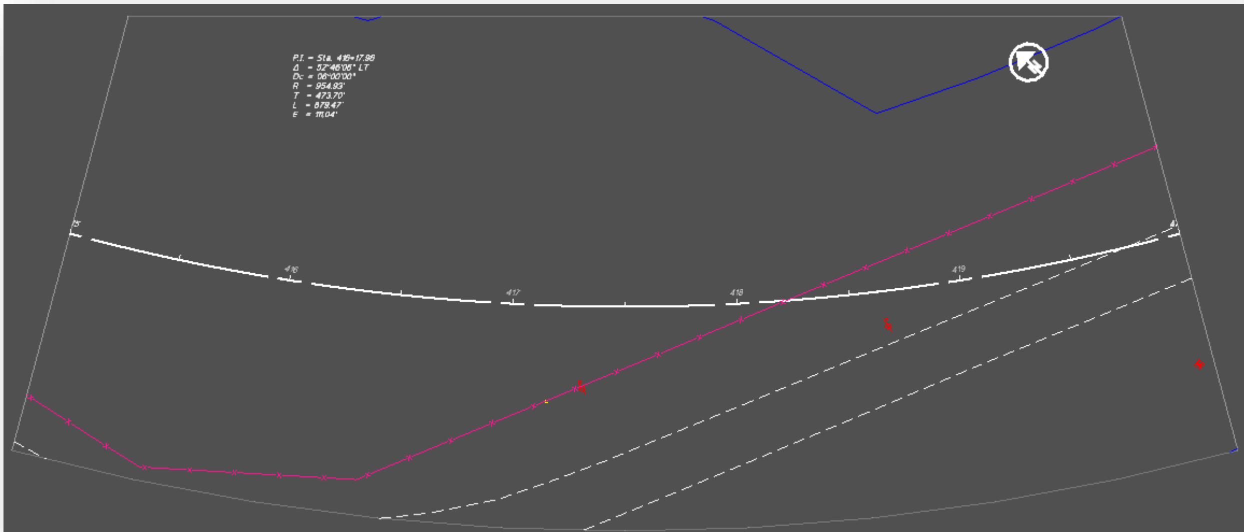
The example below shows the drawing model containing the plan view portion of a plan and profile sheet. The alignment annotation is referenced from the BK design file.



In this example, the curve data is partially clipped due to the named boundary limits. This can be resolved as follows:

Option 1 – Edit the Named Boundary.

Named boundaries can be edited, but there are limits to how much editing can be done. The boundary is chorded around a curve. In the example below, the vertices were deleted using the MicroStation Delete Vertex command. In this example, the named boundary was deleted so that the referenced curve data annotation is not clipped.



Option 2 – Place the Alignment annotation in the container file's Design Model

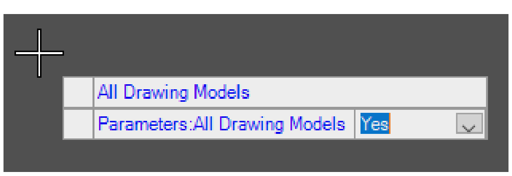
While this is not significantly different than alignment data placed in the geometry basemap, it does allow the user to move or delete unnecessary annotations without impacting other sheets.

Option 3 – Place the Alignment Annotation in the various Drawing Models

Using this option, the alignment is annotated in the drawing models for the limits of the named boundary. This method is advantageous because the annotation for each sheet is controlled by the drawing model for the sheet.

Take the following steps to use this option:

- From the container design file, turn off the alignment annotation levels from the attached BK file(s).
- In the drawing model for one of the sheets, from the *Drawing Production* tab, select the **Annotate Element** command. Select the alignment element to be annotated. OpenRoads version 10.10 adds the ability to annotate the element in multiple drawing models, as shown below.



Follow the prompts to annotate the selected alignment(s).

- Edit the annotation in the drawing models as necessary for each sheet

- Annotations added to the drawing models are not automatically visible in the sheet models. The attachment of the design model to the sheet is a "direct reference", not a "nested reference". As such, the level display is not automatically synchronized as nested references are. It is necessary to turn the annotation levels on for the drawing model attachment in each sheet model

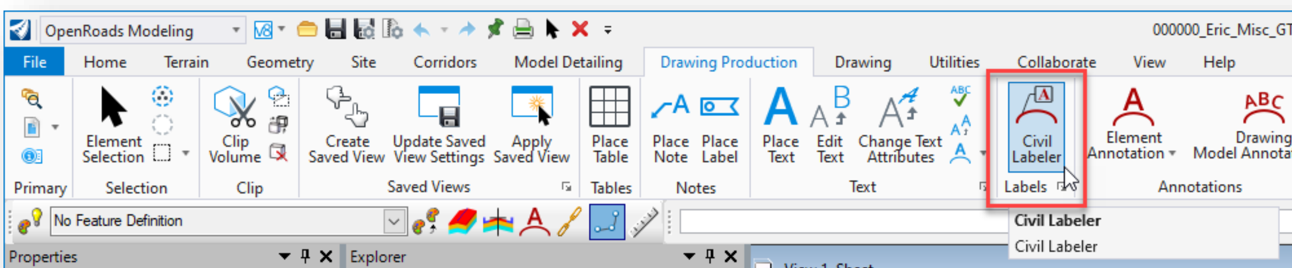
Plan View North Arrow Annotation

The north arrow symbol is placed in the plan view as part of the drawing model annotation. The location of the north arrow is defined in relation to the alignment.

The May 1, 2020 release of the OHDOT CADD Standards removed the north arrow placement block from the upper right corner of the sheet borders. It is no longer necessary to move the north arrow cell from the default location as defined in ODOT's CADD Standards to the upper right corner of the sheet border.

Civil Labeler

A new annotation tool, **Civil Labeler**, has been added to the 2021 Release 1 version of ORD. As of this writing, the function is in a "tech preview" status as the tool is refined. The Civil Labeler can be accessed from the **OpenRoads Modeling** WorkFlow by selecting **Drawing Production > Labels > Civil Labeler** from the ORD Ribbon menu.



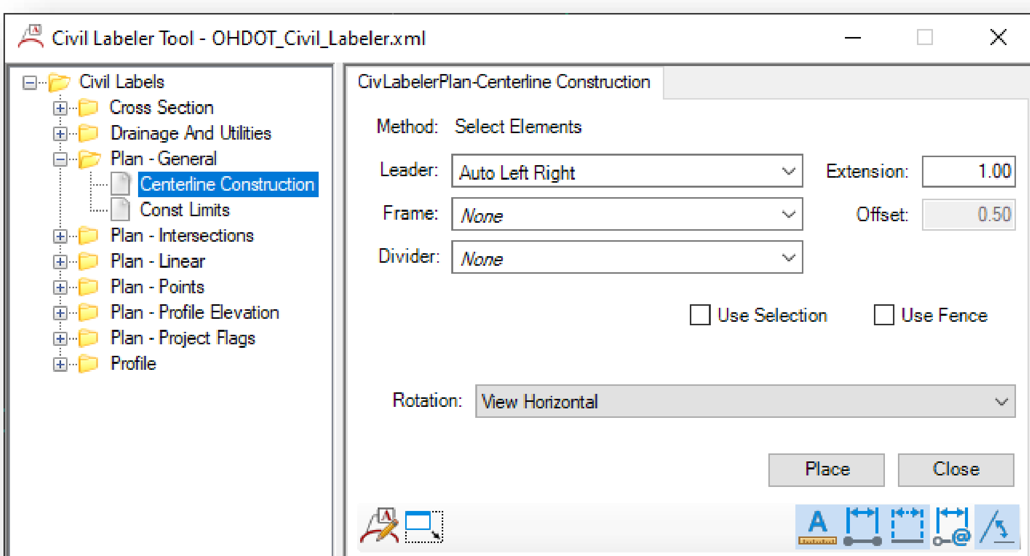
When the command is selected, the ***Civil Labeler Tool*** dialog is opened with the **OHDOT_Civil_Labeler.xml** file loaded as shown below.

The xml file is configured with multiple folders containing multiple label definitions as shown above. Each folder contains multiple labels. It is beyond the scope of this training guide to document each label option. A few of the common labels will be documented below. Contact the CADD and Mapping Support team with questions or requests to add additional labeling options.

Cross Section and Drainage and Utilities labels will not be covered in this document.

ODOT recommends placing most labels in the various Drawing models for the sheets. Labels should not be placed in the sheet model.

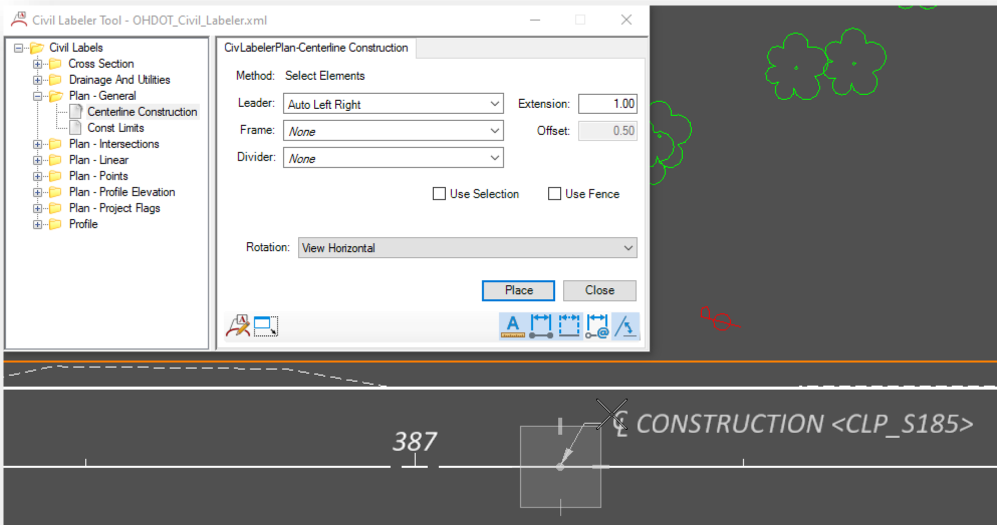
When a label is selected, the dialog presents several options to the user like the example below.



It is beyond the scope of this manual to document every option. The ODOT labels have been configured with the default options most used for each label type. Examples of commonly used labels are included below.

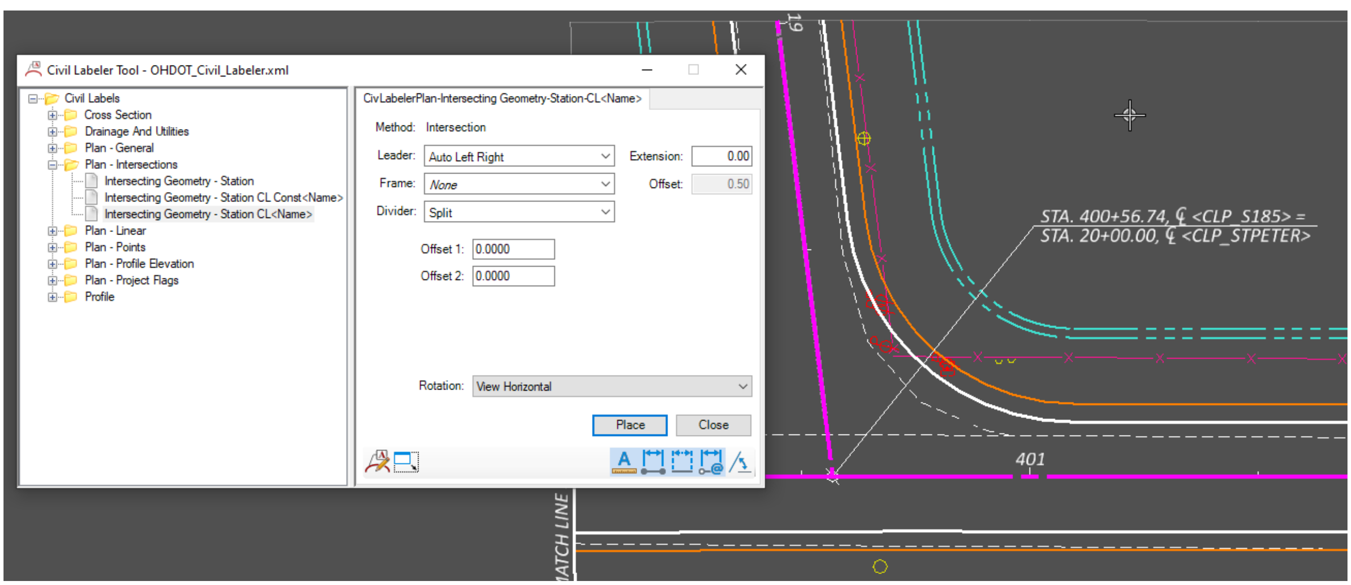
Plan - General Labels

This group of labels provides label options for annotating the centerline or construction limits. An example is shown below.



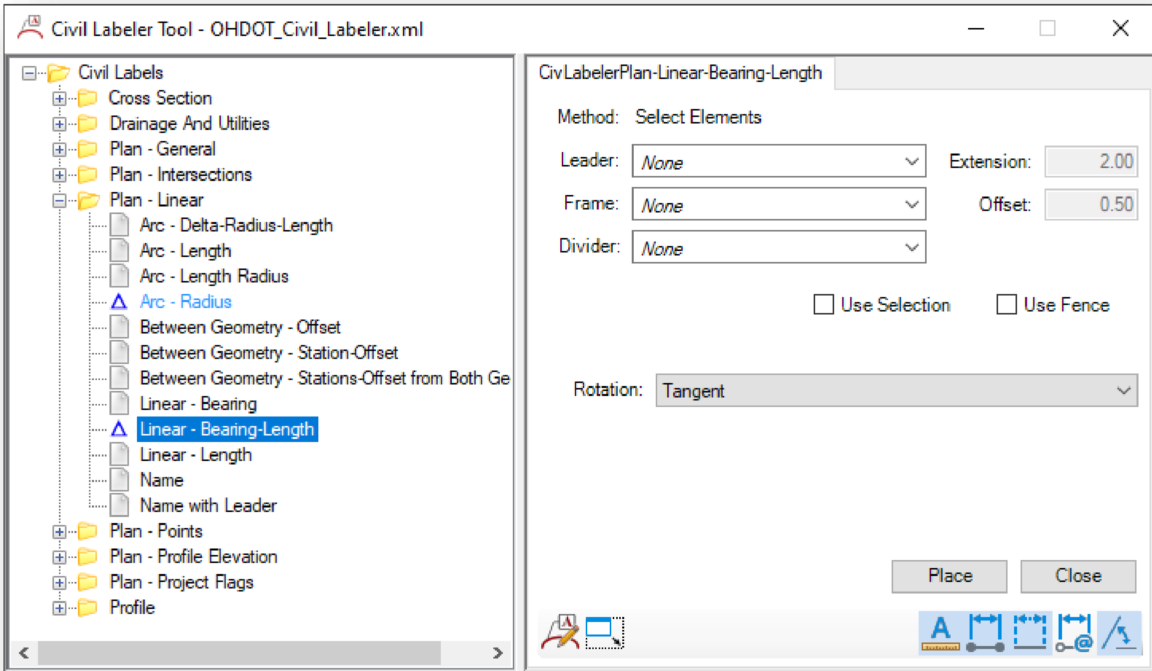
Plan - Intersection Labels

The Plan – Intersection group includes options to label the station of a point at the intersection of two alignments. In the example below, the user is prompted to select each alignment. The annotation is placed at the location defined by the user.



Plan - Linear Labels

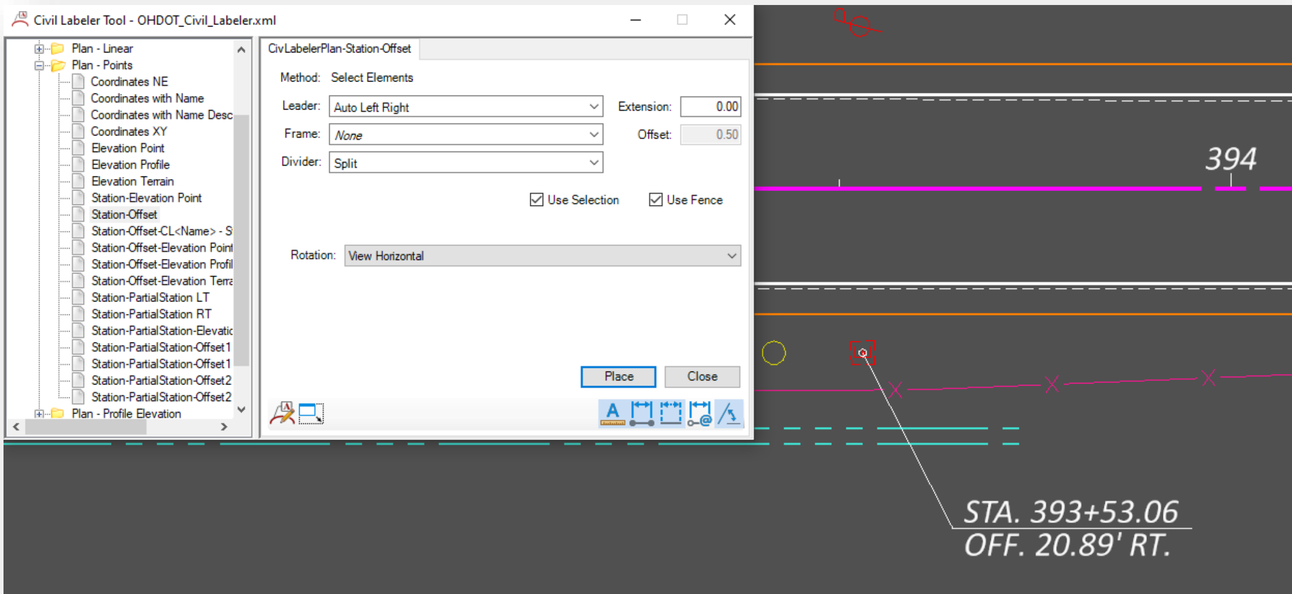
Various labels are available to annotate curvilinear elements as shown below



Plan - Points Labels

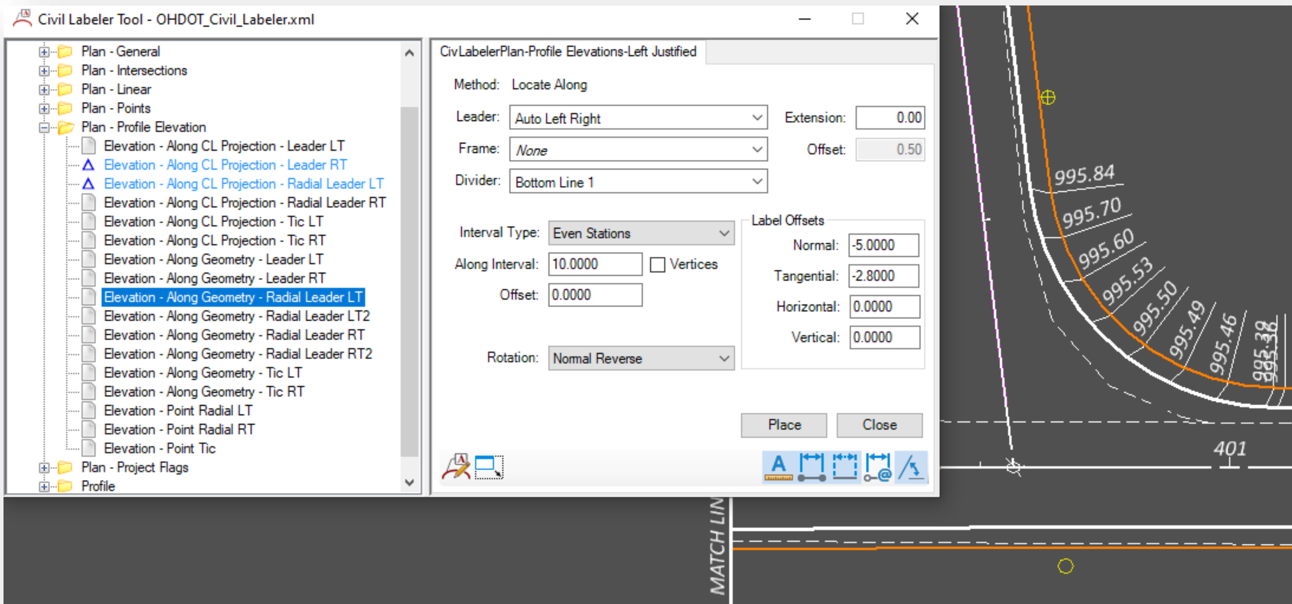
Various label options are provided for the annotation of points. In cases where a station and offset is to be annotated, after selecting the Place command it is required that you select the geometry element to be used as the station and offset reference.

An example using the **Station-Offset** label is shown below.



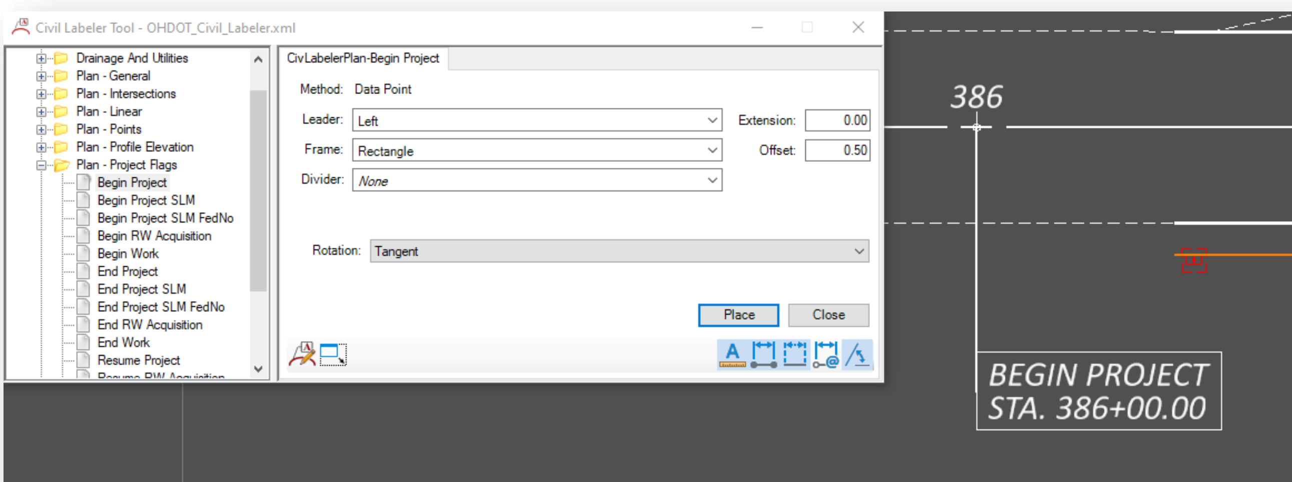
Plan - Profile Elevation Labels

Several labels have been constructed to annotate profile elevations in the plan view. The **Elevation – Along Geometry – Radial Leader LT** option is shown below.



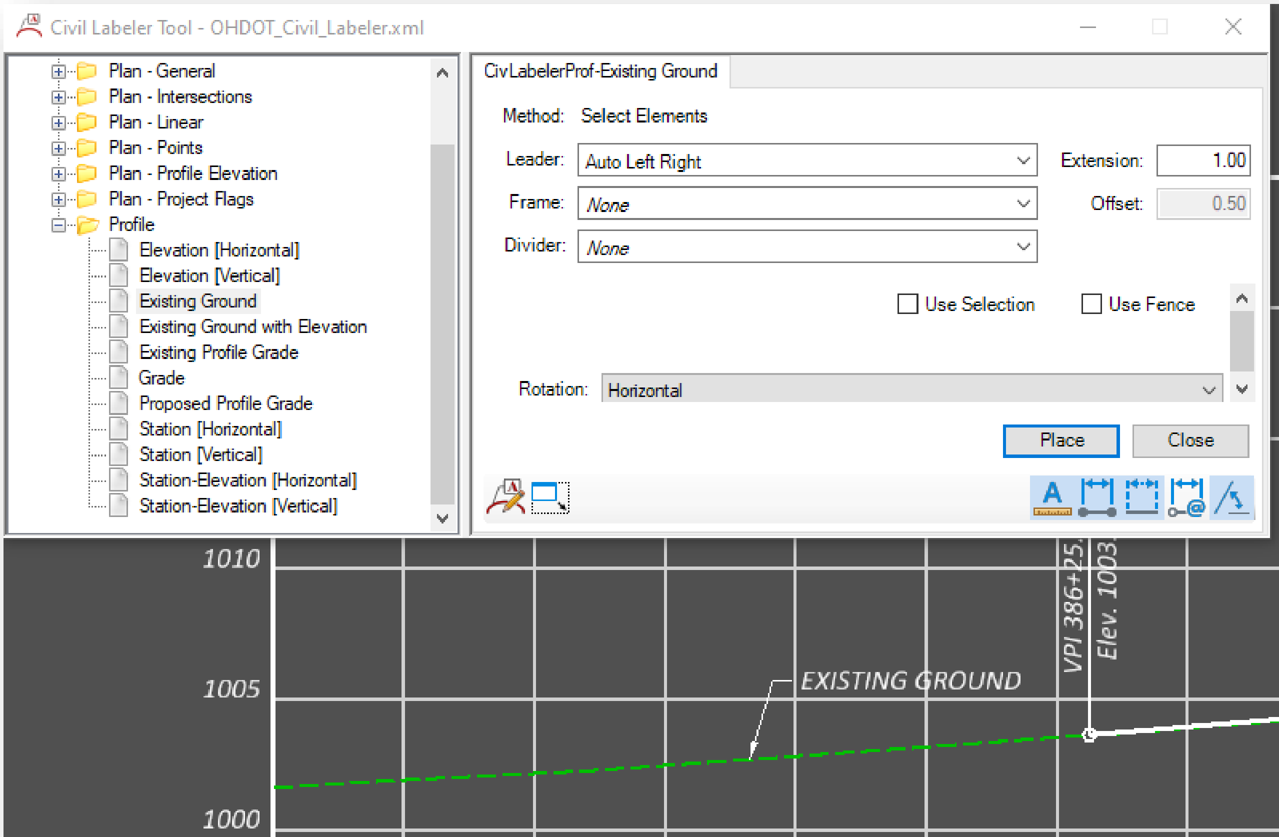
Plan - Project Flags Labels

The Plan – Project Flags group includes several labels as shown below.



Profile Labels

The Profile labels are configured for the profile drawing model. Various labels are provided as shown below.

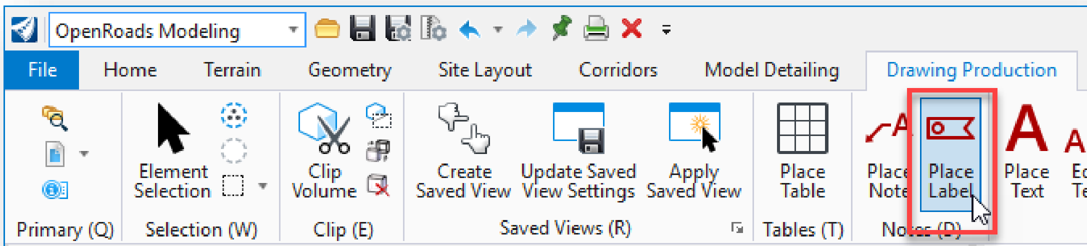


Computed Labels

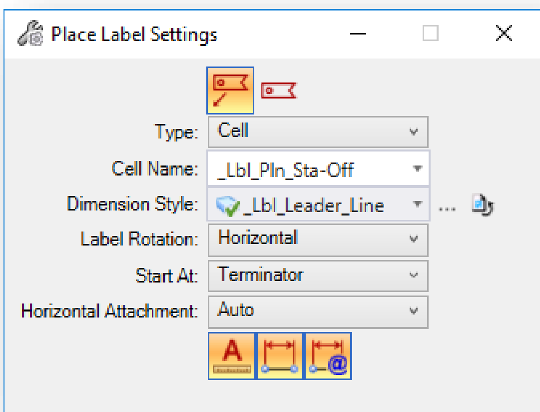
With the 2021 Release 1 version of OpenRoads Designer, ODOT recommends using the new **Civil Labeler** tools. This section is retained in the document for reference.

The **Place Label** command, located in the **Drawing Production** tab as shown below, is used to place various labels for plan production.

The labels contain Text Favorites which compute the value of the label based on selected geometry.



When the command is selected, the **Place Label Settings** dialog is opened as shown below.



Two icons are provided to place labels with or without a leader line.

The **Type** option is used to select placing either a label composed of a Text Favorite, or a label composed of a cell. An example of placing a cell is shown in this document.

The **Cell Name** parameter is used to select the label type. Several labels have been configured for ODOT users. The process to create labels is not documented in this manual.

Cells that have been configured as labels are named with an "**_Lbl**" prefix. Additional labels can be configured as requested.

The **Dimension Style** parameter is used to define the leader line for labels placed using the **Place Label with Leader** option. Two leader line dimension styles have been configured as shown below:

- _Lbl_Leader_Line
- _Lbl_Leader_Line_With_Terminator

The **Label Rotation** parameter is used to set the rotation of the label relative to the leader line: **Horizontal**, **Vertical**, or **Inline**.

Start At defines whether the label placement starts from the terminator of from the cell.

The **Horizontal Attachment** parameter sets the leader attachment side: **Auto**, **Left** or **Right**.

The three icons at the bottom of the dialog are as follows:

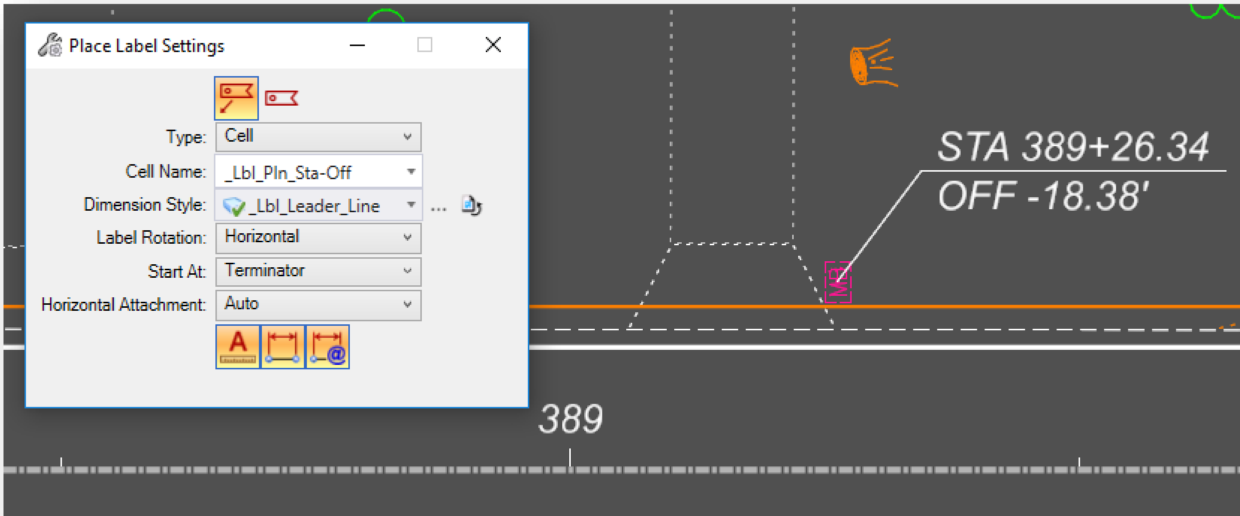
Annotation Scale
Turns on annotation scale. When this lock is on, the annotation scale is applied to the label.

Place Label With Regular Association
Sets regular association with the element. If you place a label with this setting turned on and the **Place Label With Relative Association** turned off, the label is placed with regular association. In this case, if you move the element, the label remains at the same place, only the leader line moves along with the element.

Place Label With Relative Association
(Enabled only when **Place Label With Association** is enabled) Sets relative association with the element. If a label is placed with relative association with an element, the label will stay at the same location relative to the original snap point. That is, if you move the element the label will also move with it.

To Place a Station and Offset Label with a Leader

- Select the **Place Label With Leader** icon.
- Select the **Type** value **Cell** with the **Cell Name** of **_Lbl_Pln_Sta-Off**
- Set the **Dimension Style**, **Label Rotation** method, and **Horizontal Attachment** method.
- (Optional) **Select Place Label With Regular Association** and then select **Place Label With Relative Association**.
- Enter a data point to select the alignment element that will be used to compute the station and offset values.
- Enter a data point to place the label with leader line.



Several label types have been configured in the OHDOT standards. Additional labels can be configured upon request.

Section 801 Plan and Profile Sheet Exercise

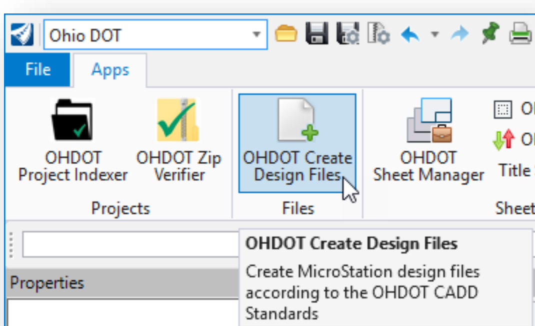
Exercise 1: S.R. 185 Plan and Profile Sheets

Part A: Sheet Design Files

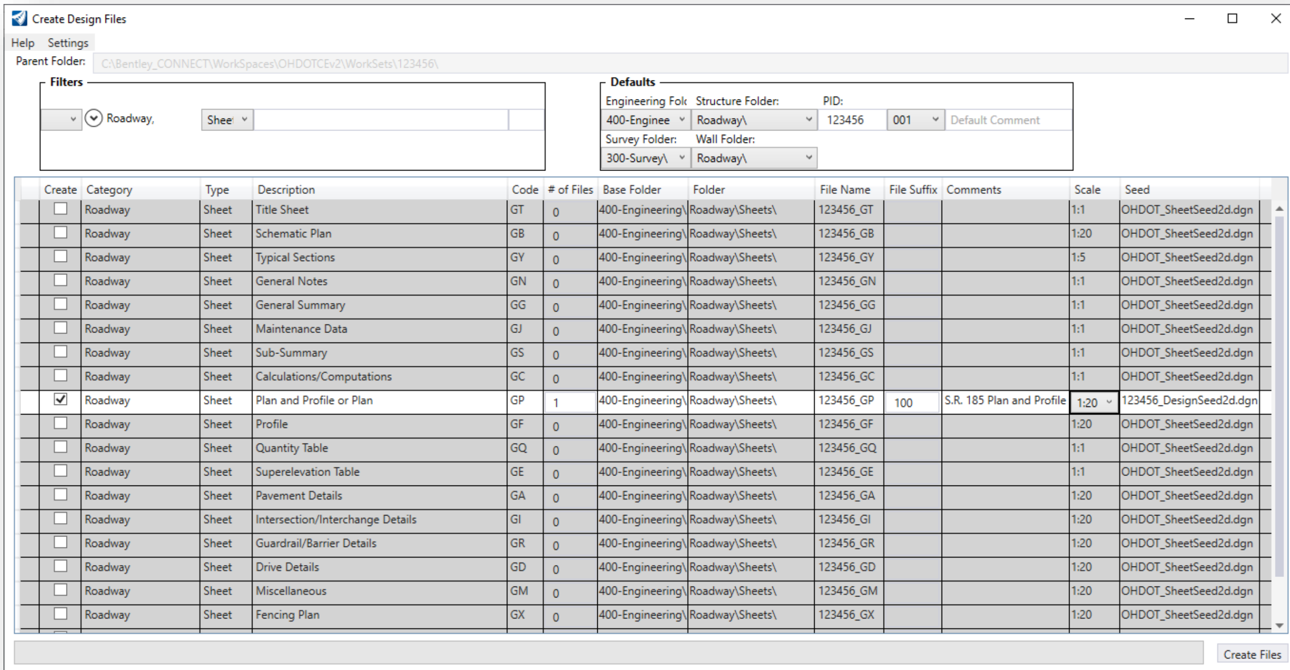
Take the following steps to create the design file for the plan and profile sheets:

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **402-Engineering\Roadway\Basemaps** folder.
- Open the file **123456_BK001.dgn**
- Select the **Ohio DOT** Workflow
- Select the **OHDOT Create Design Files** application



- Create the sheet design file for S.R. 185 using the settings below:



Create	Category	Type	Description	Code	# of Files	Base Folder	Folder	File Name	File Suffix	Comments	Scale	Seed
☐	Roadway	Sheet	Title Sheet	GT	0	400-Engineering\	Roadway\Sheets\	123456_GT			1:1	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Schematic Plan	GB	0	400-Engineering\	Roadway\Sheets\	123456_GB			1:20	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Typical Sections	GY	0	400-Engineering\	Roadway\Sheets\	123456_GY			1:5	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	General Notes	GN	0	400-Engineering\	Roadway\Sheets\	123456_GN			1:1	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	General Summary	GG	0	400-Engineering\	Roadway\Sheets\	123456_GG			1:1	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Maintenance Data	GJ	0	400-Engineering\	Roadway\Sheets\	123456_GJ			1:1	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Sub-Summary	GS	0	400-Engineering\	Roadway\Sheets\	123456_GS			1:1	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Calculations/Computations	GC	0	400-Engineering\	Roadway\Sheets\	123456_GC			1:1	OHDOT_SheetSeed2d.dgn
☑	Roadway	Sheet	Plan and Profile or Plan	GP	1	400-Engineering\	Roadway\Sheets\	123456_GP	100	S.R. 185 Plan and Profile	1:20	123456_DesignSeed2d.dgn
☐	Roadway	Sheet	Profile	GF	0	400-Engineering\	Roadway\Sheets\	123456_GF			1:20	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Quantity Table	GQ	0	400-Engineering\	Roadway\Sheets\	123456_GQ			1:1	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Superelevation Table	GE	0	400-Engineering\	Roadway\Sheets\	123456_GE			1:1	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Pavement Details	GA	0	400-Engineering\	Roadway\Sheets\	123456_GA			1:20	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Intersection/Interchange Details	GI	0	400-Engineering\	Roadway\Sheets\	123456_GI			1:20	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Guardrail/Barrier Details	GR	0	400-Engineering\	Roadway\Sheets\	123456_GR			1:20	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Drive Details	GD	0	400-Engineering\	Roadway\Sheets\	123456_GD			1:20	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Miscellaneous	GM	0	400-Engineering\	Roadway\Sheets\	123456_GM			1:20	OHDOT_SheetSeed2d.dgn
☐	Roadway	Sheet	Fencing Plan	GX	0	400-Engineering\	Roadway\Sheets\	123456_GX			1:20	OHDOT_SheetSeed2d.dgn

  - When the **File Suffix** value of 100 is entered, the application will change the selected seed file to use the **123456_DesignSeed2d.dgn** file.
  - Select **Settings > Open Last Created File**
  - Select the **Create Files** option
    The new file is created in the **402-Engineering\Roadway\Sheets** folder

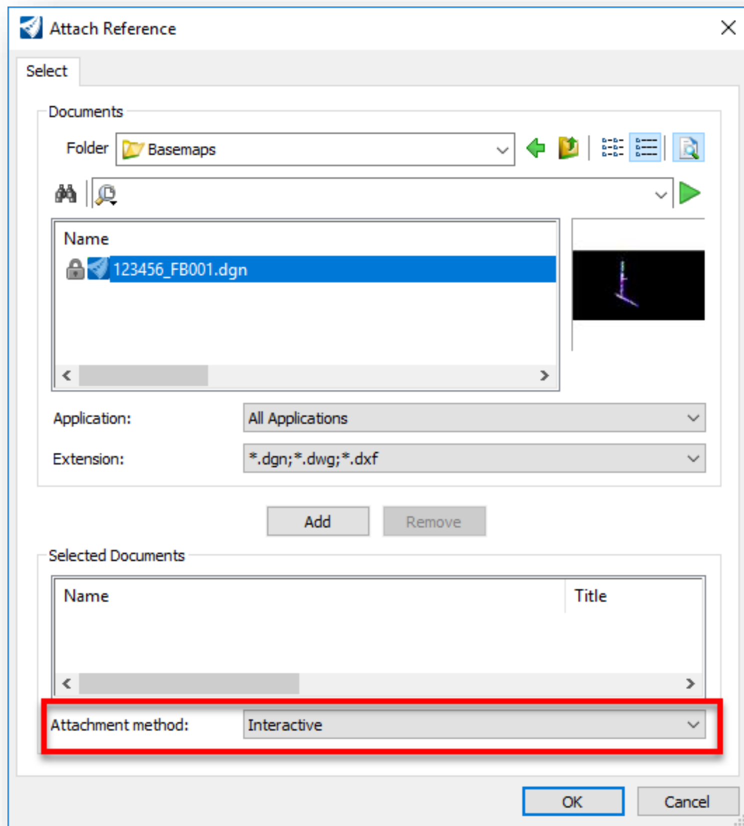
- When prompted, check the current file back into ProjectWise

Part B: Plan View References

- From the **OpenRoads Modeling** WorkFlow, select **Home > Attach Tools > References** to attach the files with the plan view information. Start by attaching the Survey Field Book (FB) design file from the following folder:

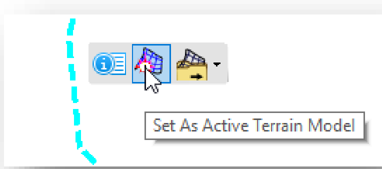
  300-Survey\Basemaps\123456_FB001.dgn

- When selecting the references, set the **Attachment Method** to **Interactive**. This will open the ***Reference Attachment*** dialog for each file as it is attached. This is necessary to review and set the **Nested Attachments** parameter for each file.

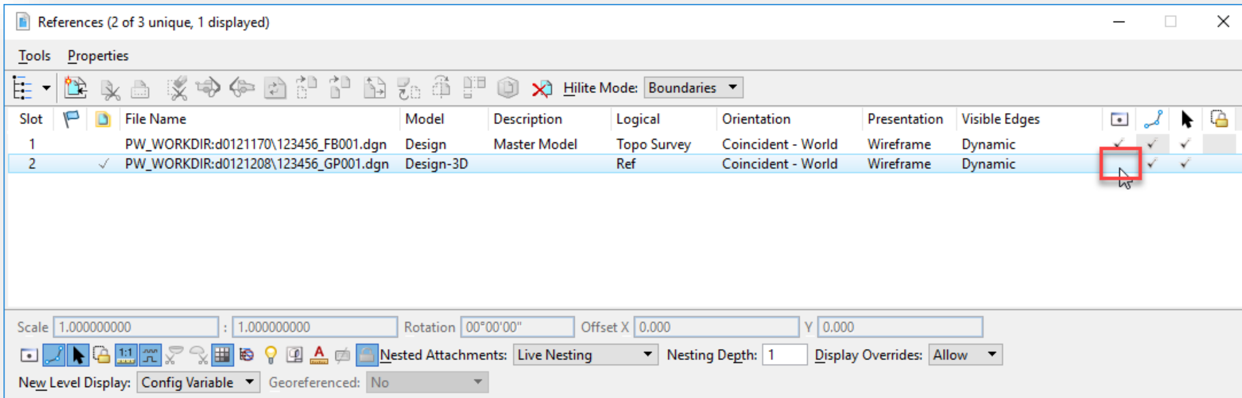


- From the ***Reference Attachment*** dialog, set the **Nested Attachments** parameter to **No Nesting**.

- After attaching the Survey Field Book design file (FB) set the terrain model as the active terrain by selecting the terrain model element and choosing the **Set As Active Terrain Model** icon from the pop-up menu.



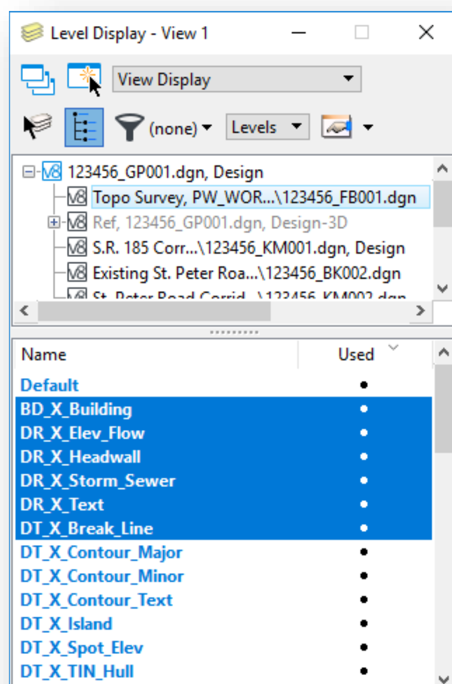
- The 3D model will not be displayed on the sheets. From the ***References*** dialog, turn off the display of the Design-3D model.



- Attach the following files using the **Nested Attachments** setting of **No Nesting**:
  - 401-Engineering\Roadway\Basemaps\123456_BK001.dgn (CL Proposed S.R. 185)
  - 401-Engineering\Roadway\Basemaps\123456_BK002.dgn (CL Existing St. Peter Road)
  - 401-Engineering\Roadway\Basemaps\123456_BK003.dgn (CL Proposed Jameson Road)
  - 401-Engineering\Roadway\Basemaps\123456_KM001.dgn (S.R. 185 Corridor Model)
  - 401-Engineering\Roadway\Basemaps\123456_KM002.dgn (St. Peter Road Corridor Model)
  - 401-Engineering\Roadway\Basemaps\123456_KM003.dgn (Jameson Road Corridor Model)
  - 401-Engineering\RW\Basemaps\123456_BR001.dgn (Existing Righ-of-Way)

- After the references are attached, from the **Home** tab, select the **Level Display** icon to turn on/off the levels that you wish to display in the Plan View of the sheets.
  - Select each reference file to turn on/off the desired levels

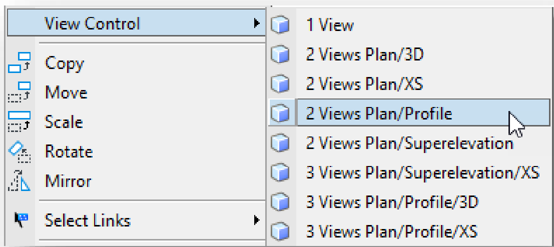
- Select **File > Save Settings** after you have completed the task of turning on/off the levels



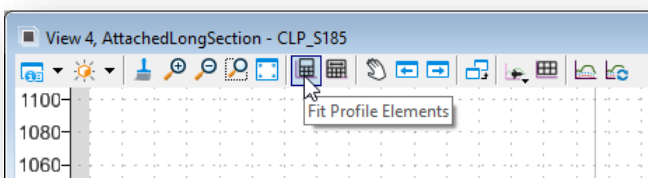
Part C: Profiles

A profile window must be displayed to lay out the named boundaries for the profile portion of the sheets. Take the following steps to open the profile window:

- ✓ Hold down the right-mouse button in the view window to access the menu shown below.
  - Select the **2 Views Plan/Profile** option and follow the prompts to open a profile view for the S.R. 185 Alignment



- ✓ If the profile is not displayed in the profile window, use the ***Level Display*** dialog (**Home > Level Display**) to turn on the levels for the existing and proposed alignments and then choose the **Fit Profile Elements** command from the profile window.



The profile is now ready to define the named boundaries.

Part D: Plan View Named Boundaries

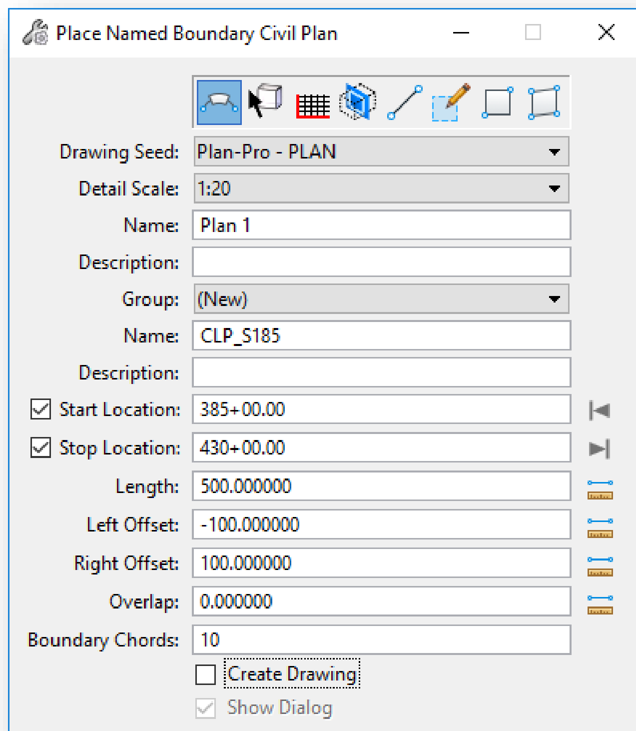
MicroStation Named Boundaries are placed to define the limits of each plan and profile sheet.

When the named boundaries are placed, they are spaced according to the **Starting Location** station value and the **Length** the selected sheet configuration file.

In some cases, placing the named boundaries from start to finish may result in boundaries that fall with the match line locations in the middle of a bridge or intersection.

Take the following steps to define the plan view named boundaries as a continuous run for S.R. 185:

- ✓ From the **Drawing Production** tab, select the **Place Named Boundaries** command
- ✓ From the ***Place Named Boundary*** command, take the following steps:

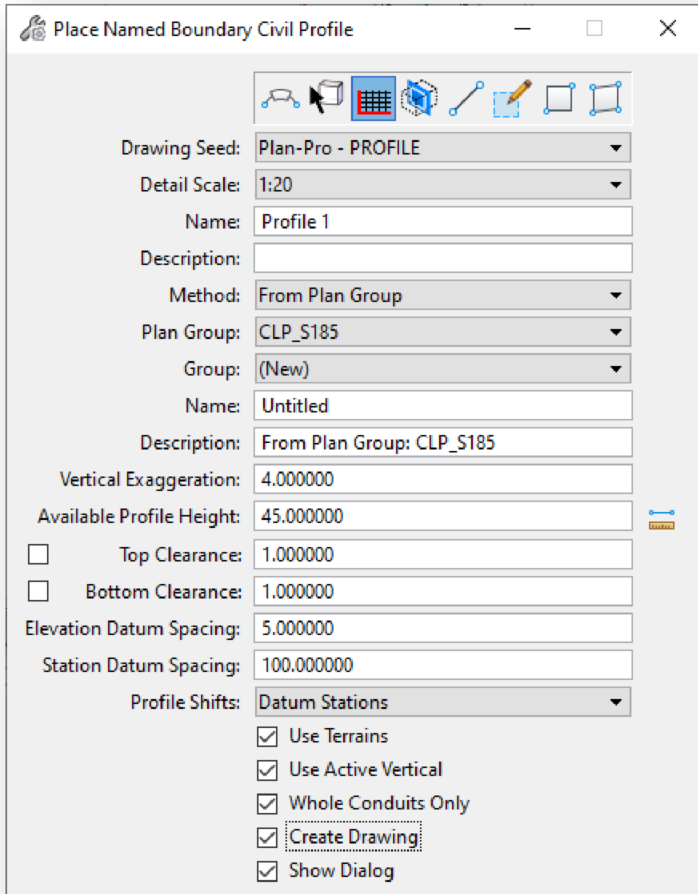


  - Select the **Civil Plan** icon (first icon at the top of the dialog)
  - Set the **Drawing Seed** to use the **Plan-Pro - PLAN** option
  - Toggle off the **Create Drawing** option at the bottom of the dialog.
  - Click inside the **Start Location** key-in field
  - Left-click on the plan view alignment graphic for S.R. 185. The dialog will track the start location station value relative to the cursor location
  - Key in a **Start Location** value of **385+00**
  - Key in a **Stop Location** value of **430+00**
    - Note that the default **Length** parameter for the **Plan-Pro - PLAN** option is **500'**. If the value does not default to an even 500.0', change the parameter setting in the dialog.
  - Issue a left-click in the plan view.
    - This will display the named boundaries before they are placed in the file.
    - Issue a left-click in the plan view (it may take two clicks) to accept the placement of the first two named boundaries.
    - The ***Place Named Boundaries*** dialog is automatically closed

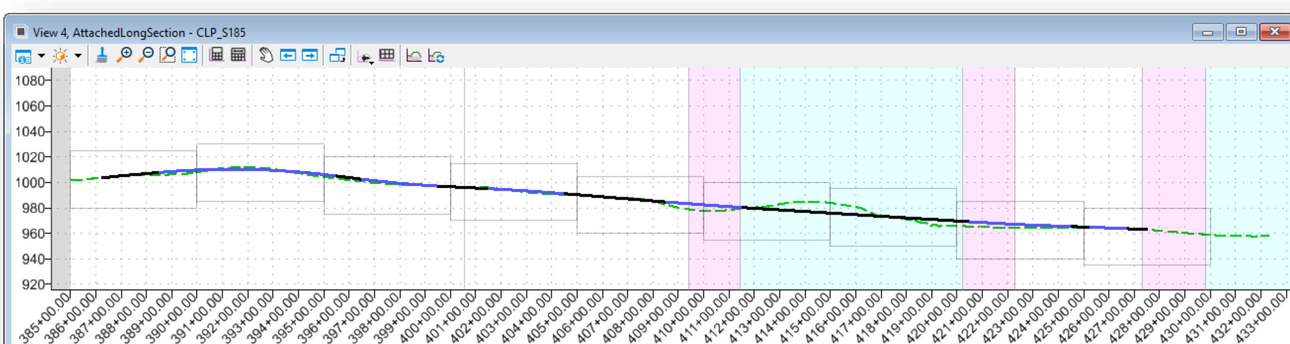
Part E: Profile Named Boundaries

Take the following steps to define the profile view named boundaries for S.R. 185:

- ✓ From the **Drawing Production** tab, select the **Place Named Boundaries** command
- ✓ From the ***Place Named Boundary*** command, take the following steps:
  - Select the **Civil Profile** icon (third icon at the top of the dialog)
  - Set the **Drawing Seed** to use the **Plan-Pro 20 - PROFILE** option
  - Set the **Method** to **From Plan Group**
  - Toggle **On** the following options:
    - **Use Terrains**
    - **Use Active Vertical**
    - **Whole Conduits Only**
    - **Create Drawing**
    - **Show Dialog**
  - Issue a left-click in the profile view (it may take three clicks) to accept the placement of the named boundaries for the profile.

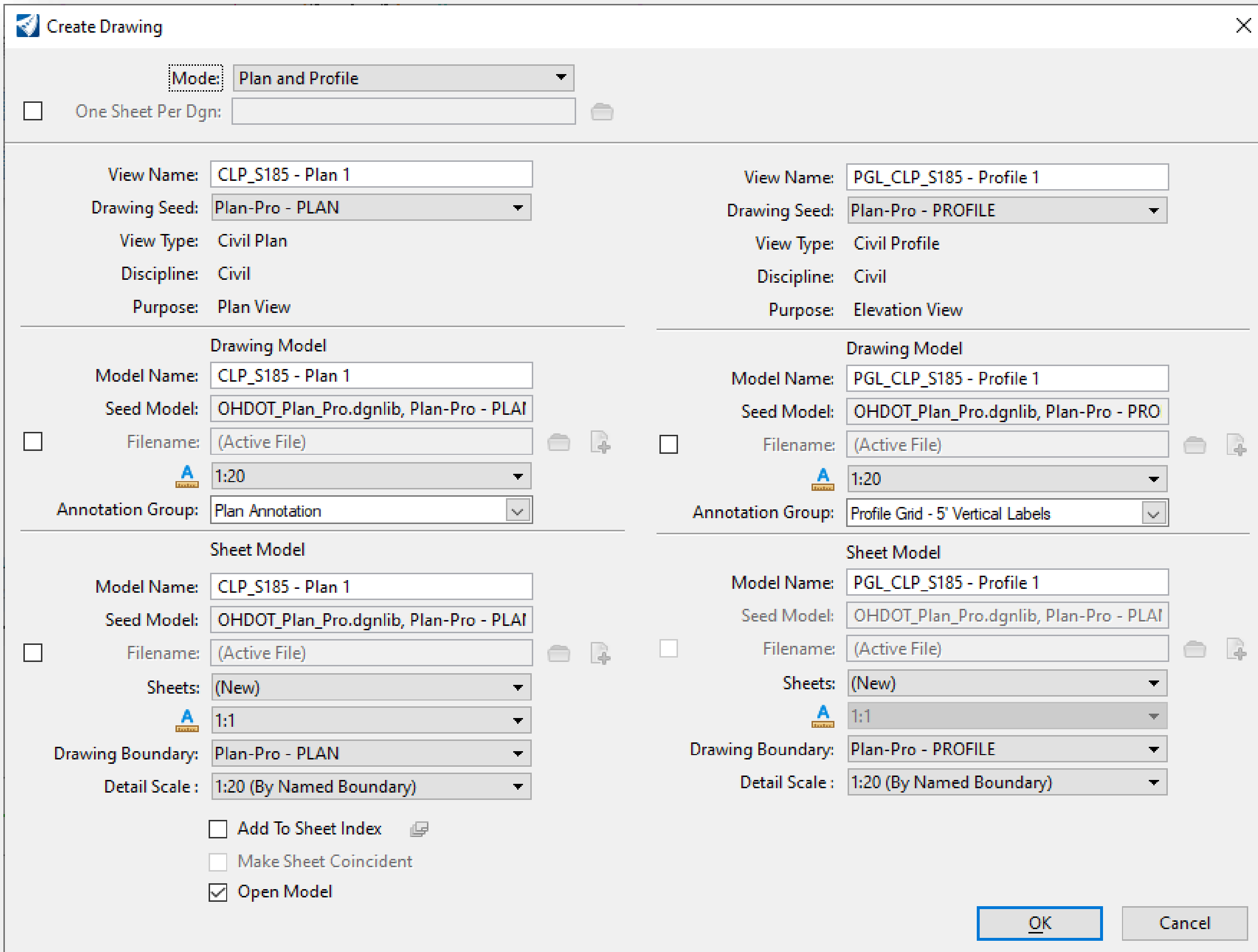


The boundaries are placed like the example below.



Part E: Create Drawing

If the **Show Dialog** option is toggled on as shown on the previous page, the ***Create Drawing*** dialog is opened.



- ✓ Toggle on the **One Sheet Per Dgn** option if you desire to create a new design file for each sheet. When this option is toggled on, the name entered for the plan view **Sheet Model** > **Model Name** parameter is used to create each sheet design file.

  ODOT recommendation: When using the **One Sheet Per Dgn** option, enter the **Sheet Model** > **Model Name** value using the PID number for the project with the two-character code for the sheet type and a three-digit number.

  Example: 123456_GP101

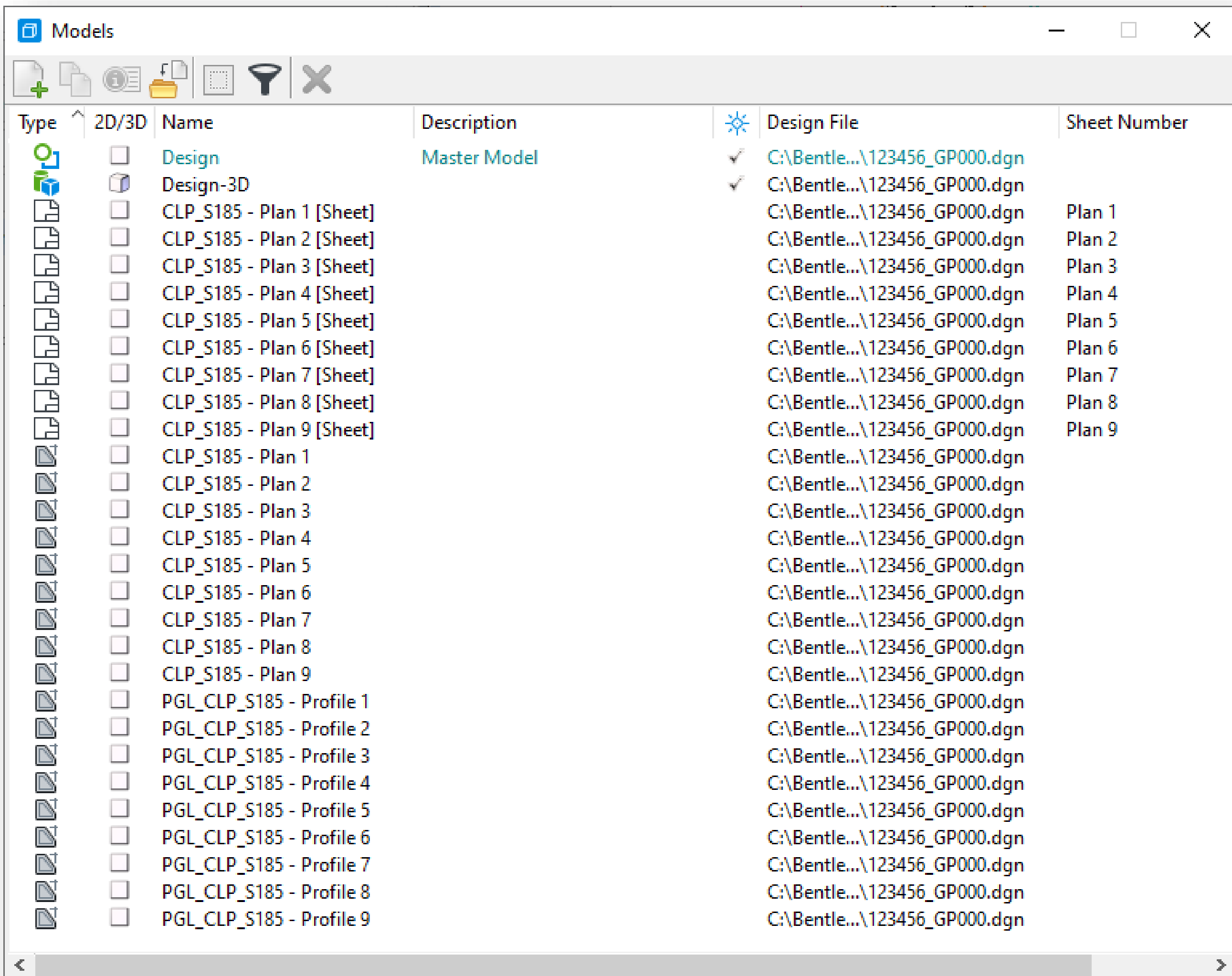
- ✓ Set the **Drawing Model Annotation Scale** and the **Sheet Model Detail Scale** parameters for both the plan and profile to the desired scale as shown above. Do ***NOT*** set the **Sheet Model Annotation**

Scale value. Leave this parameter at **1:1** as shown above.

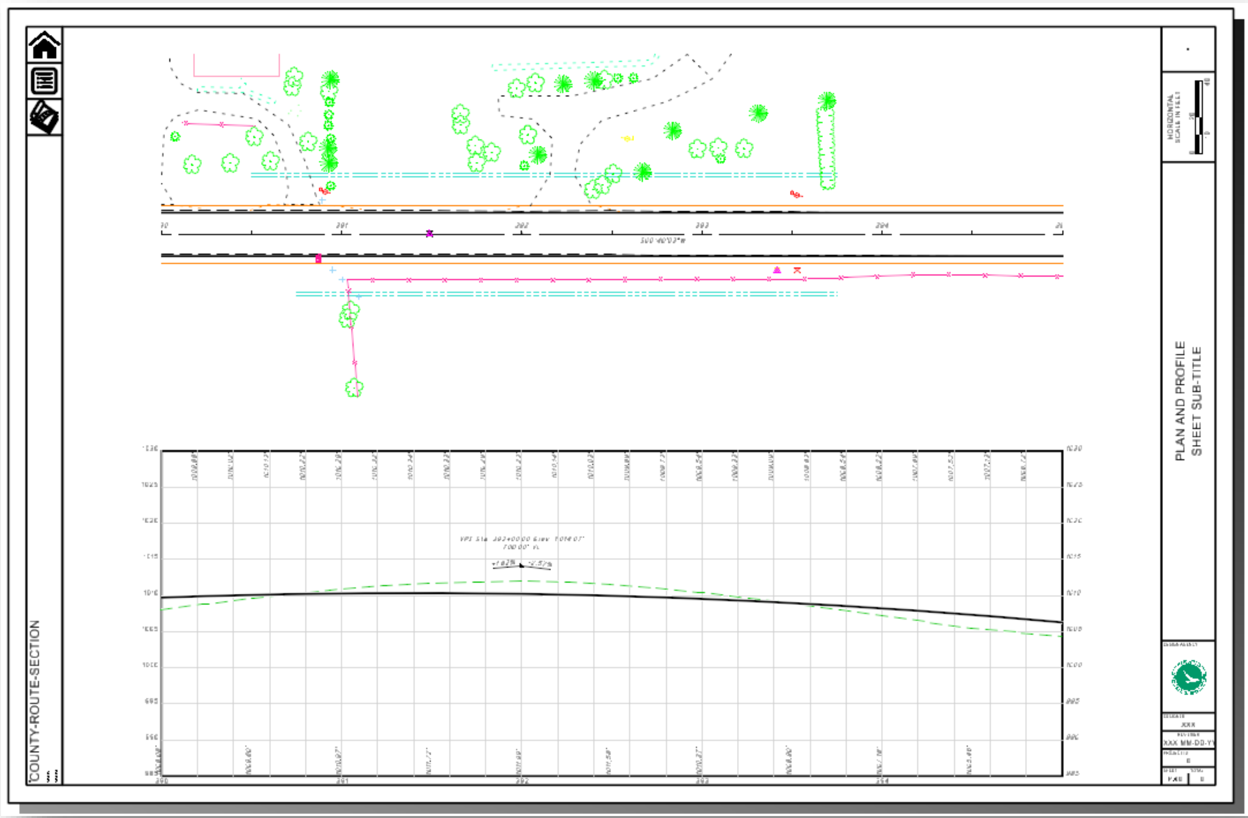
✓ Select the **OK** button to initiate the sheet creation process.

A drawing model is created for each named boundary. A sheet model is created for each sheet. The plan and profile drawing models are referenced to the sheet model

✓ Review the drawing and sheet models by choosing the model name from the **Models** dialog, as shown below.



An example of a completed sheet is shown below.



This completes this exercise. Exit OpenRoads Designer and check the file back into ProjectWise.

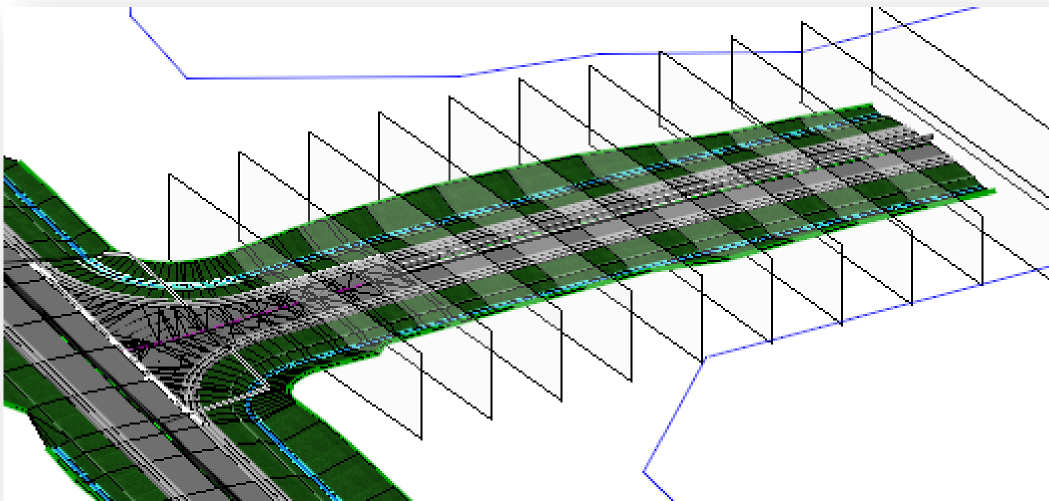
Section 802 Cross-Section Sheets

Cross-Section Sheet Process Overview

The process to generate cross-section sheets is very similar to the plan and profile sheet process.

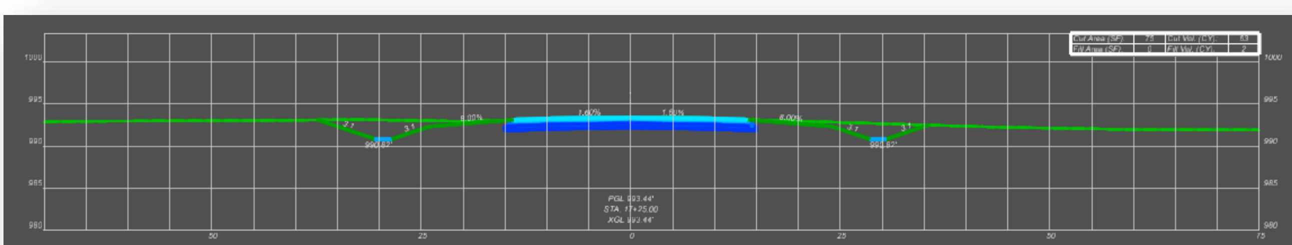
OpenRoads designer uses MicroStation **Named Boundaries** to define the cross section clipping areas. The named boundaries are placed in a **3D Design** model relative to a selected alignment.

The example below shows the 3D Named Boundaries placed relative to the St. Peter Road 3D model.

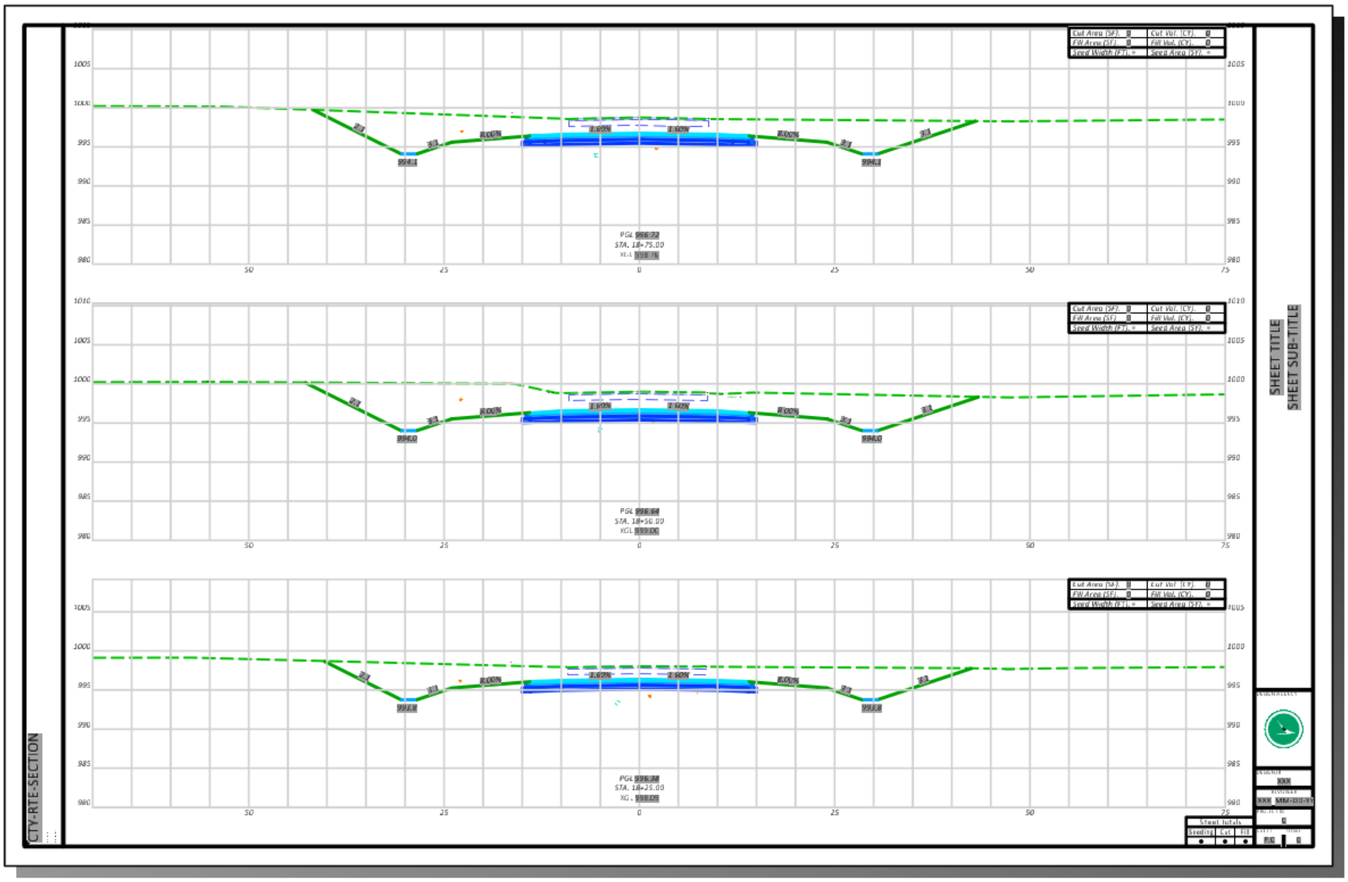


Before generating the named boundaries, a design file is created as a “container file” to attach references and place the named boundaries for the cross-section locations. The 3D model must be displayed in order to create the named boundaries. The cross sections are cut from the 3D model. In the example above, the geometry, survey, and corridor model basemaps are attached to the plan view with the 3D model displayed. The named boundaries are placed in the 3D model relative to a selected alignment.

When the sheets are generated, the software creates a MicroStation drawing model for each named boundary. An example of the drawing model for a cross-section is shown below.



The drawing models are referenced to a sheet model with a sheet border to assemble the completed cross-section as shown in the example below.



The sheet model is intended for plotting or PDF generation. Most annotations should be placed in the various drawing models, not in the sheet models.

File Types and Usage

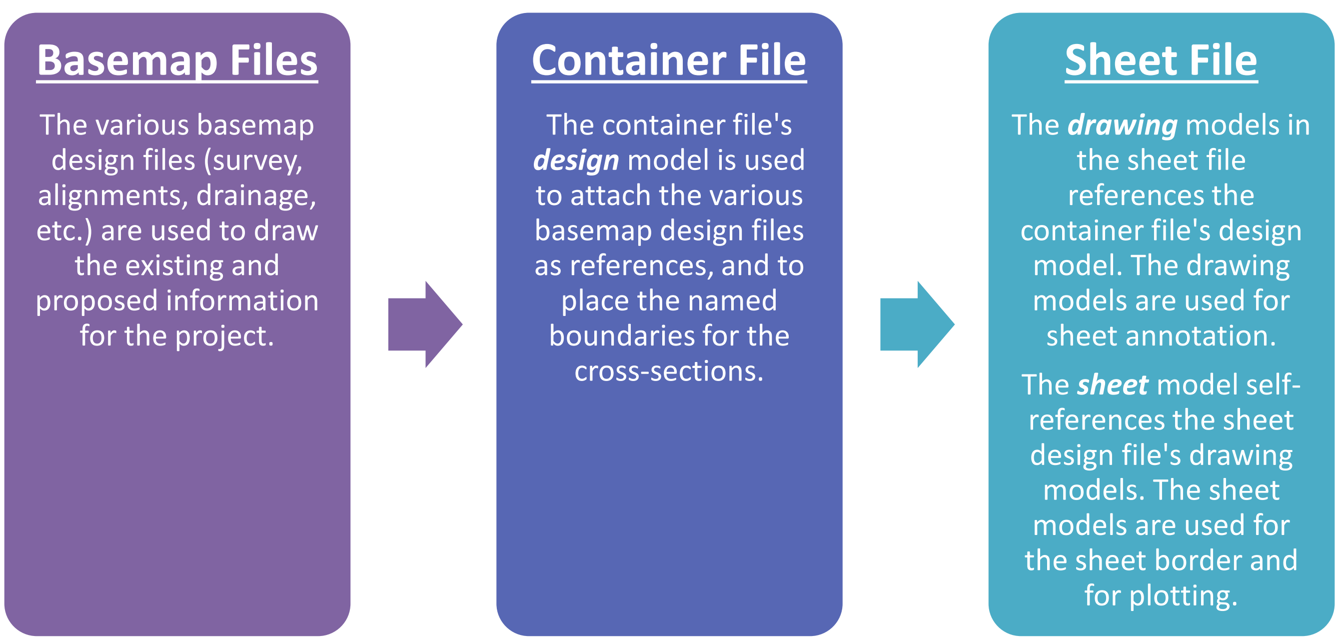
A "container file" is used to attach references for the information that will be shown in the cross sections as well as to place the named boundaries for the cross-section locations.

Sheets can be clipped into the container file itself, or clipped into multiple files, one sheet per design file, if desired.

Each container file is used to generate the sheets for a single alignment. For example, a project with three alignments would typically use three container files as detailed below:

Design File	Usage	Models Used
XS000	Container file for S.R. 185	Design
XS001, XS002, XS003…	Cross-section sheets for S.R. 185	Drawing and Sheet
XS100	Container file for St. Peter Road	Design
XS101, XS102, XS103…	Cross-section sheets for St. Peter Road	Drawing and Sheet
XS200	Container file for Jameson Road	Design
XS201, XS202, XS203…	Cross-section sheets for Jameson Road	Drawing and Sheet

The references are attached as illustrated below.



The container file is used to control the reference attachment and level display settings for all the sheets from one location. More information on level display can be found in the Level Management section of this document.

It is possible to put the named boundaries for multiple alignments in one container file if all the sheets will share the same reference attachments and level display settings.

End Area Volumes

End areas can be extracted from the 3D model for each named boundary and included on the completed sheets. This process requires 3D mesh elements for the cut and fill areas. The 3D mesh elements should be created in a separate design file. These 3D meshes are normally created before starting the sheet clipping process.

Cross-Section Sheet Summary

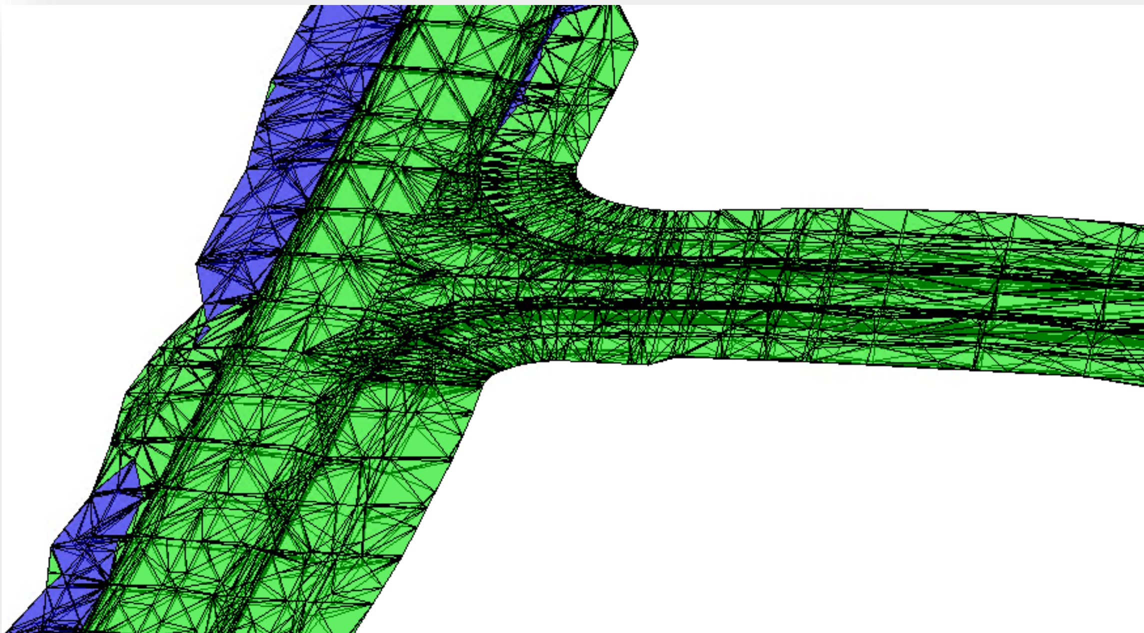
The steps to generate cross-section sheets are summarized below:

1. If the end area quantities are to be displayed, create a design file to contain the 3D mesh elements and run the **Create Cut Fill Volumes** command
2. Create a design file to act as the container file
3. In the container file, take the following steps to prepare for clipping the cross-section sheets:
    a. Attach the various design files containing the 3D models and survey information, underground utilities, 3D right-of-way lines, etc. that will be shown on the cross-sections. Also attach the design file containing the 3D mesh volume elements
    b. Assign the existing ground terrain model as the active terrain
    c. Open a 3D view of the model
    d. Toggle on/off the Global Display of the levels in the Design-3D model that you wish to be displayed in the cross-sections.
    e. Create **Named Boundaries** for the cross-section locations, but do not create the sheets yet.
    f. If end area quantities are to be displayed, run the **End Area Volume** report. Each named boundary is tagged with quantity values
4. Create the cross-section sheets. The sheets can be generated into the active design file (the container file) or into separate design files.

Create Cut and Fill 3D Meshes

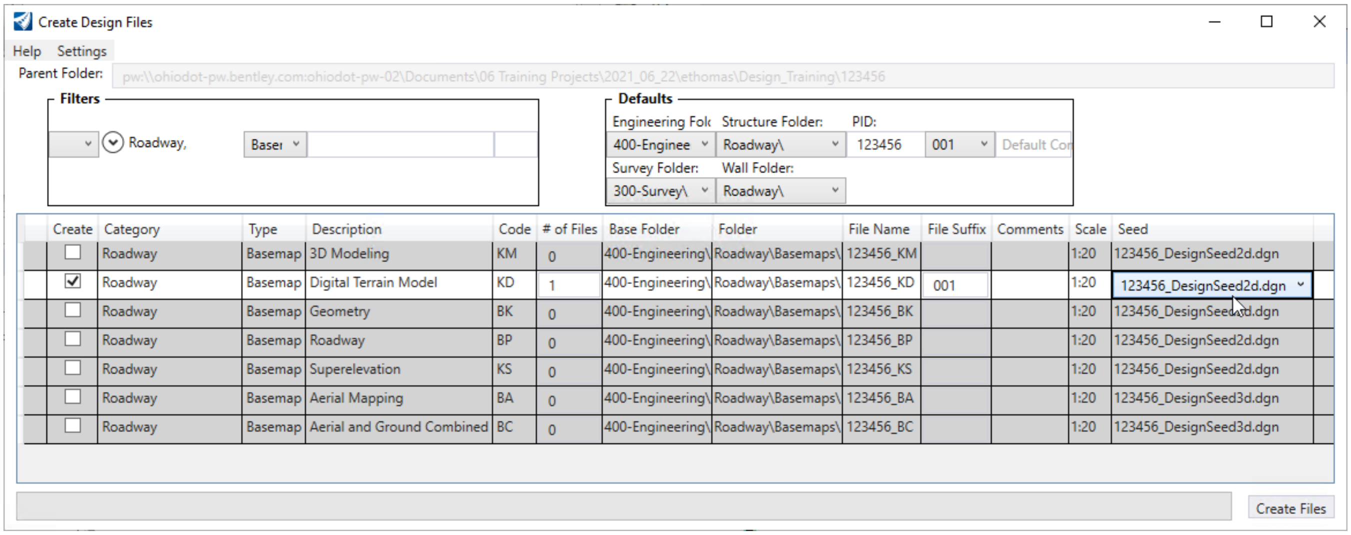
The **Create Cut Fill Volumes** command is used to generate 3D mesh elements for the cut and fill volumes. These elements are required to generate end area volumes for the cross-section sheets. This training guide will document the process to create simple cut/fill quantities.

An example of the 3D mesh elements is shown below.



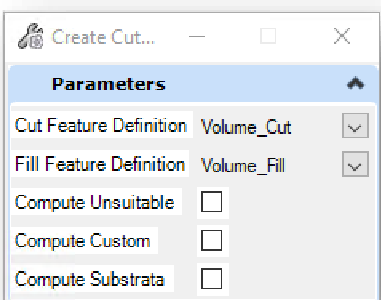
The 3D mesh elements are generated in the **Roadway Digital Terrain Model** file type.

When using the **OHDOT Create Design Files** application the **Seed** option will default to the **DesignSeed3d** file. Change this to the **DesignSeed2d** option as shown below.



After the file is created, the next step is to attach the design files as a reference that contains the existing ground terrain and the proposed corridor models.

The Create Cut Fill Volumes process will compare the lowest proposed component to the active terrain to generate 3D prismoid volumes. Be sure to set the existing ground terrain as the active terrain before running the command. The command is accessed by selecting **OpenRoads Modeling > Home > Civil Analysis > Create Cut Fill Volumes**.



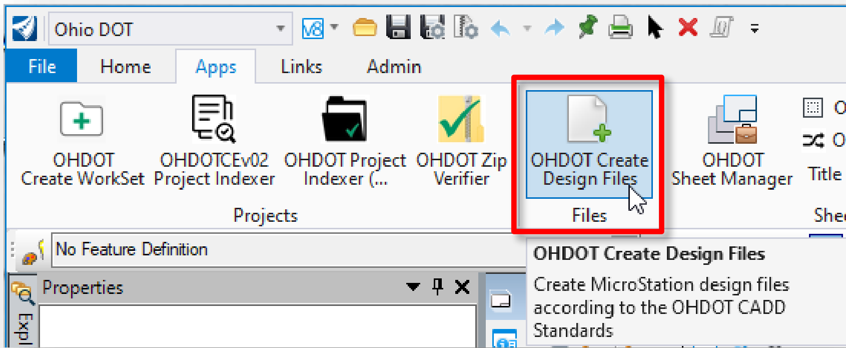
The tool settings for the command are shown at left.

Accept each option by issuing a left click as each option is presented on the cursor.

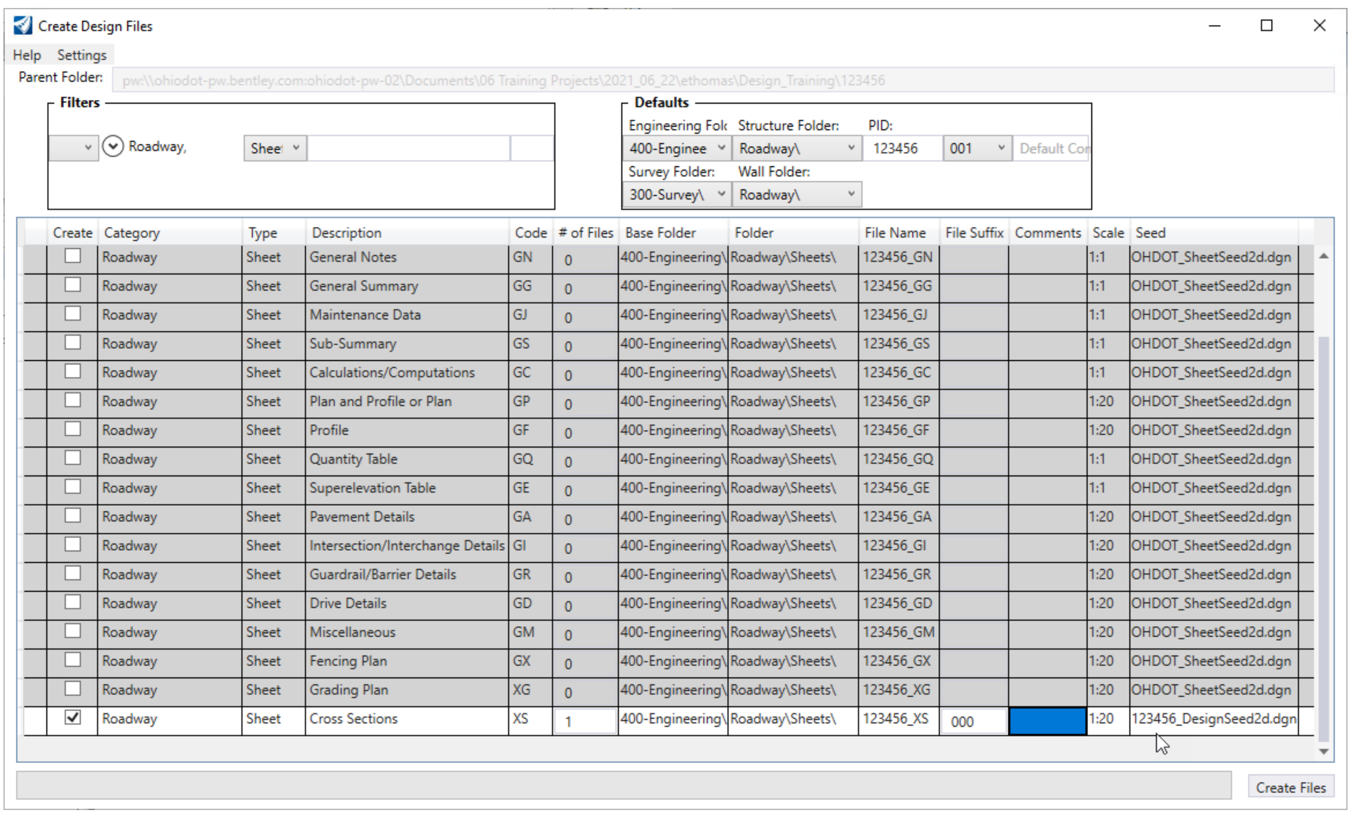
The 3D mesh elements for the cut and fill volumes are placed in the 3D model like the example shown on the previous page.

Creating the Container File

Design files are created using the **OHDOT Create Files** application. The application is accessed from the Ohio DOT WorkFlow by selecting the **New Design File** icon.



The application is opened as shown below.



In the example above, the file suffix of **000** has been entered for the **Cross Sections** sheet type. The application recognizes the 000, 100, 200, etc. series as container files and has automatically switched the **Seed** file to the appropriate design seed for the WorkSet.

Reference Attachments

After creating the new design file for the cross section named boundaries, attach the references necessary to generate the cross sections.

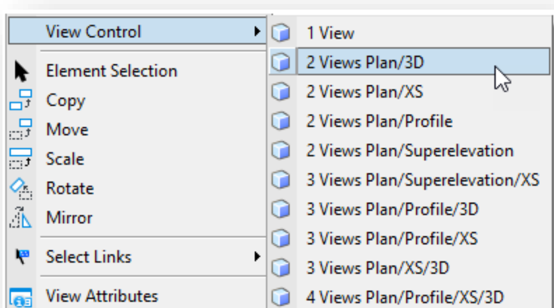
As a minimum, the following files will need to be attached:

- Survey basemap containing the existing ground terrain model.
    - **Note:** Be sure to set the existing ground as the active terrain after the file is attached. This will create a 3D model in the active cross section sheet design file
- Geometry basemap containing the alignments
- Corridor Modeling basemaps containg the applicable corridors

Several additional files may be included as attachments depending on the project:

- Any other files that contain 3D information (utilities, right-of-way, etc.) that is to be displayed on the cross sections
- The design file that contains the prismoidal volume areas, which is required to display end-area quantities on the sheets

After the references are attached, it is necessary to configure the view windows to display both plan and 3D model information. Hold down the right mouse-button with the cursor positioned in a MicroStation view window until the pop-up menu appears. Select the **2 Views Plan/3D** option as shown below.

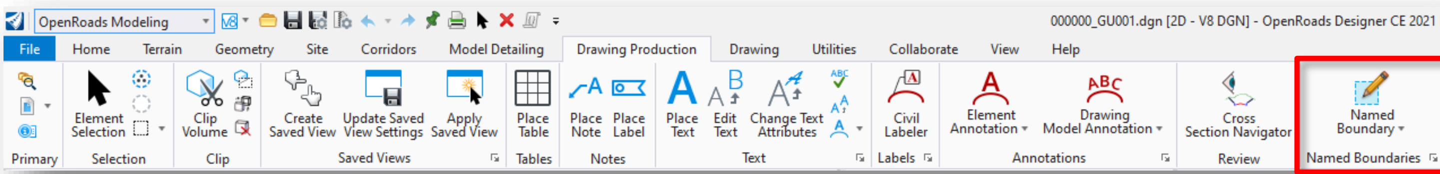


The graphics that will be displayed on the clipped cross sections are controlled by what is displayed in the 3D view. Toggle on/off the desired levels in the 3D view before proceeding.

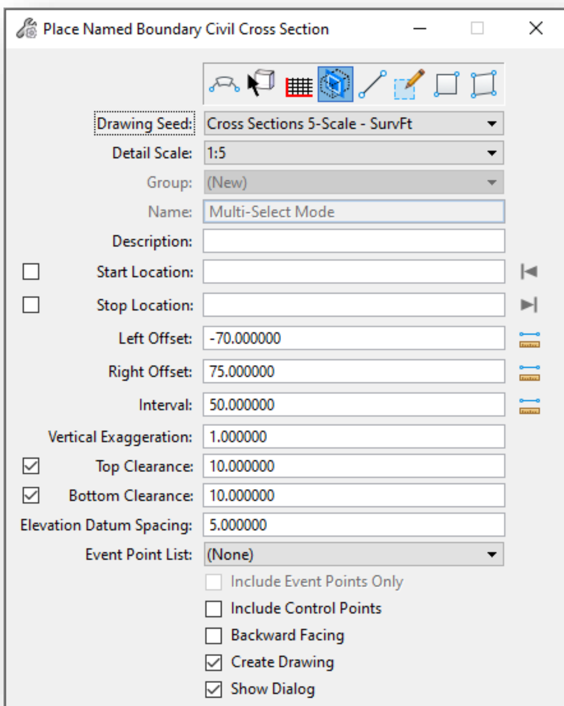
Named Boundaries

Sheets are clipped using **Named Boundary** tool. From the **OpenRoads Modeling** workflow, select the **Drawing Production** tab to access the **Named Boundary** tool as shown below.

Note: The Named Boundary tool can also be accessed from most of the default workflows provided within ORD.



The *Place Named Boundary Civil Plan* dialog is opened as shown below.



The top of the dialog contains several icons to select the type of named boundary that you wish to place.

The following icons will be discussed in this document:


Civil Cross Section


Civil Cross Section 2 Points

Note: Toggle the **Create Drawing** option off if generating end area volume quantities to be displayed on the cross-section sheets. The Named Boundaries must be placed before running the **End Area Volumes Report**. See End Area Volume Report for more information.

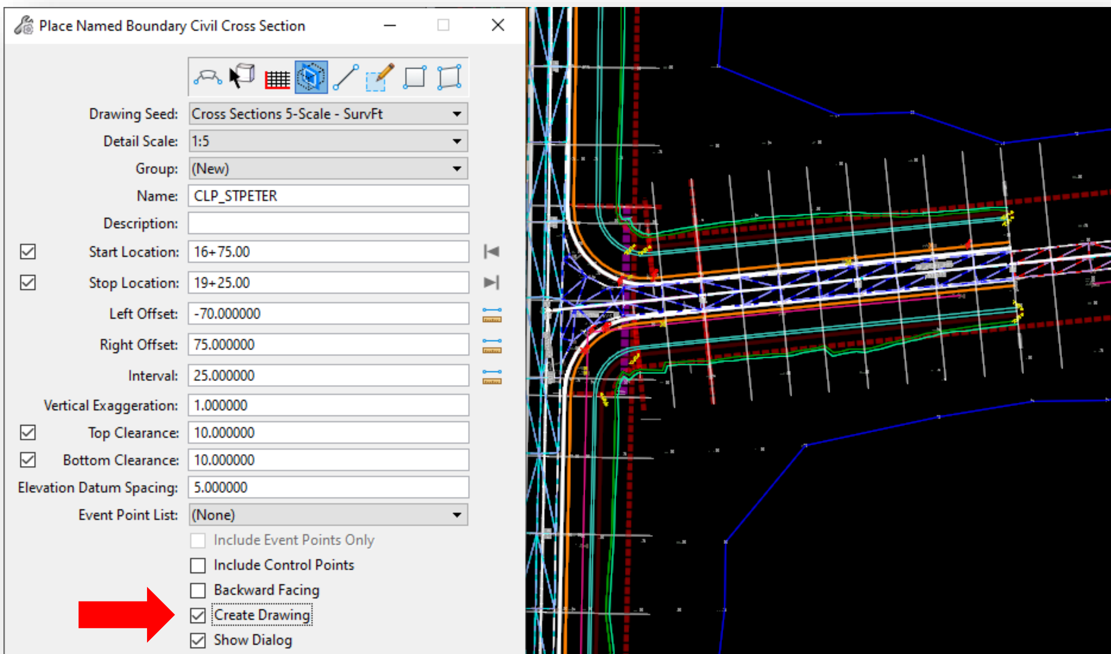
Select the fourth icon, **Civil Cross Section** to begin the process of placing the cross section named boundaries. The **Drawing Seed** item is used to select the configuration for placing the cross section named boundaries. Currently ODOT has two options that are available to define the named boundaries for cross section sheet clipping.

- Cross Sections 5-Scale
- Cross Sections 10-Scale

Issue a left mouse-click in the **Start Location** Field. And then select the desired alignment that will be used as the centerline for the cross-section clipping. The **Start Location** value will dynamically change as the cursor is moved within the view window. Note that the second **Name** field will change from “Untitled”, as shown above, to the name of the selected alignment.

- Identify the desired **Start Location** by issuing a data-point within the view, by keying in a station value, or by selecting the icon to the right of the **Start Location** field to lock the starting location to the beginning of the selected alignment.

After the **Start Location** has been established, identify the desired **Stop Location**. As the cursor is moved, the **Stop Location** will track the station value relative to the cursor position. Clipping borders are dynamically displayed as the cursor is moved. The stop location can be entered by issuing a right-mouse click in the view, keying in the desired station, or by selecting the icon to the right of the **Stop Location** field to lock the ending location to the end of the selected alignment. An example is shown below.



Create Drawing

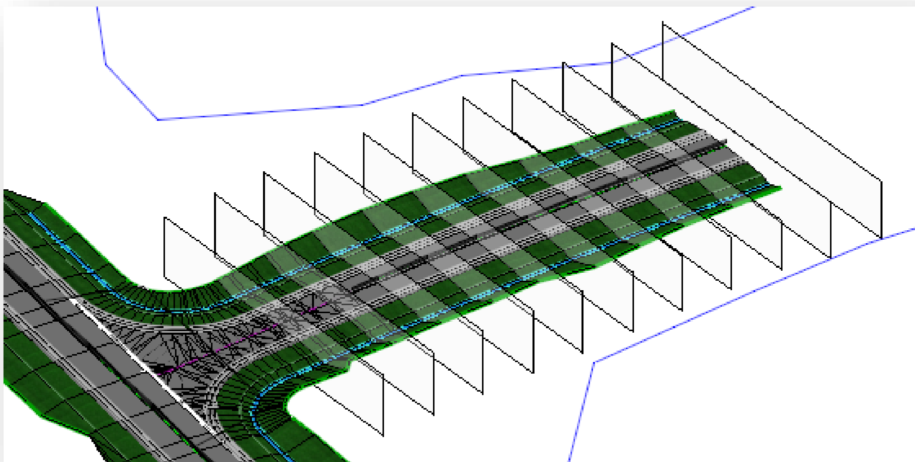
This option is used to initiate the process to create the sheets after the named boundaries are placed.

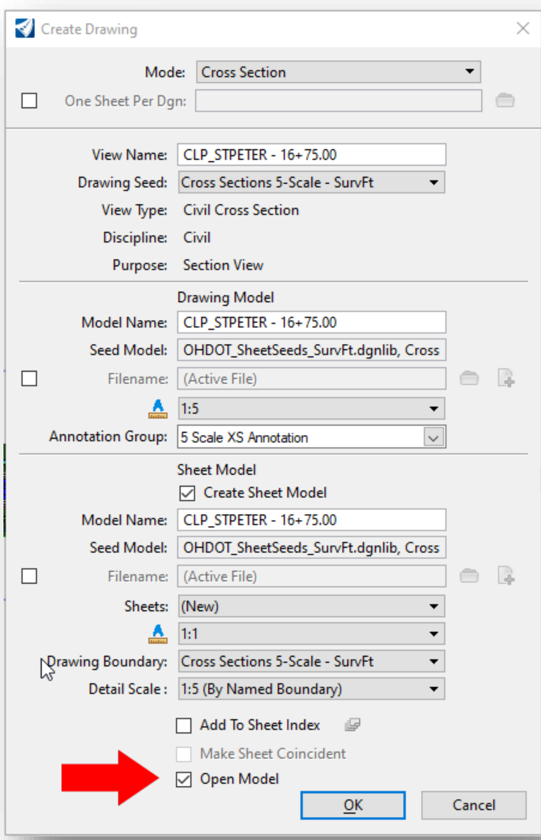
- Turn this option **OFF** if you want to update the named boundaries with the end area quantities before generating the sheets. See the End Area Volume Report section.
- Turn this option **ON** to generate the sheets without any end area values. The end areas can be updated later as detailed below.

Placing the Named Boundaries

After the parameters in the ***Place Named Boundaries*** dialog has been defined, it takes two left-mouse clicks to initiate the named boundary placement.

- Issue a left mouse-click to accept the **Start** and **Stop Location** values
- Issue another left mouse-click to accept the named boundary placement. Named boundaries are drawn in the 3D view representing the clipping limits for each cross section as shown below





If the **Create Drawing** and **Show Dialog** options are toggled on, the ***Create Drawing*** dialog, shown at left, is opened.

Toggle in the **One Sheet Per DGN** option if desired and set the path for the new cross-section sheets. When this option is selected, the **Sheet Model** > **Model Name** parameter is used to specify the name of the cross-section sheet design files.

Check to ensure that the **Drawing Model** annotation scale and **Sheet Model Detail Scale** parameters are set properly for the corresponding cross-sections.

Always set the **Sheet Model** annotation scale parameter to 1:1 as shown at left.

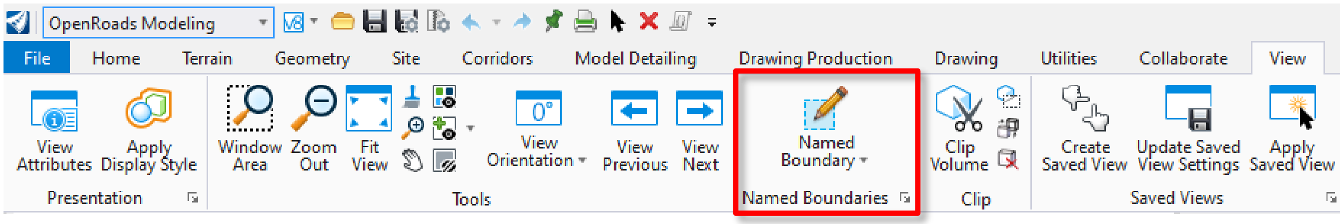
Select **OK** to initiate the sheet clipping process.

Open Model

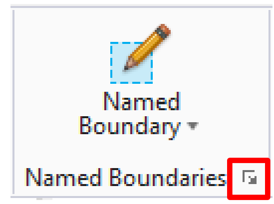
- Turn this option **OFF** if you don't want this process to open the last generated sheet.
- Turn this option **ON** if you want this process to open the last generate sheet.

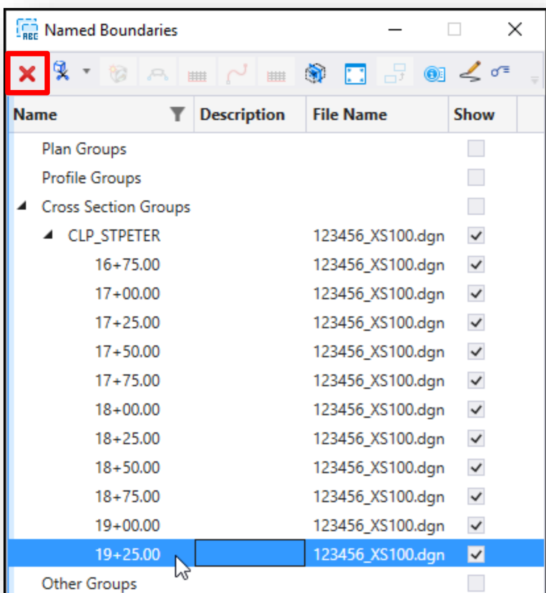
Deleting Named Boundaries

The **Named Boundaries** tool as you'll notice can be found under various workflows. But it will always be located under either the **View** tab or **Drawing Production** tab. For this example, we will be using the **OpenRoads Modeling** workflow.



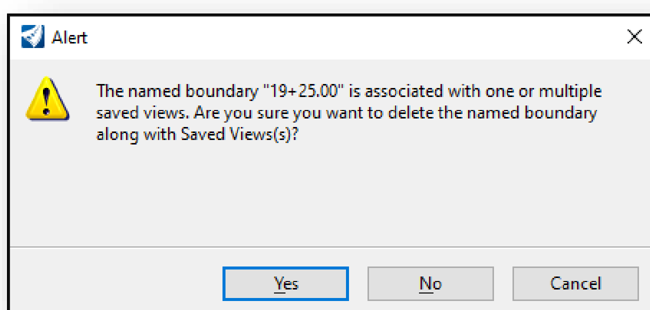
Open the Named boundaries Manager by clicking the Open Dialog arrow.





The Named Boundary Manager dialog shows all the named boundaries that have been placed within the dgn file that they were generated.

To delete these named boundaries right click and highlight the boundary then hit the red "X".



You will then be prompted with a warning dialog (shown below). Once you click "**yes**" the named boundary will be deleted from the Named Boundaries Manager and the Saved View will be deleted from the Saved View Manager the dgn.

Sheet Clipping Models and Saved Views

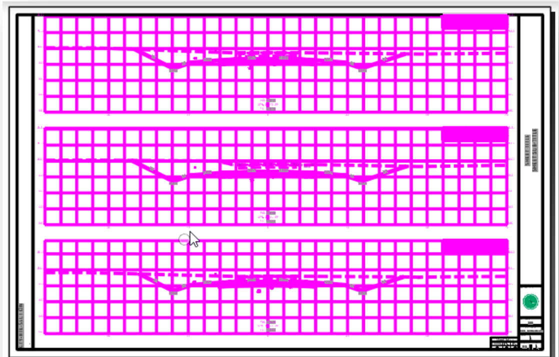
After clicking the OK button within the Create Drawing dialog software creates a drawing model that contains the cross-section annotations which is automatically generated. Also, within this process the software will create a “snapshot” of the cross sections design-3D model which is considered a Saved View. This will be done for each cross-section.

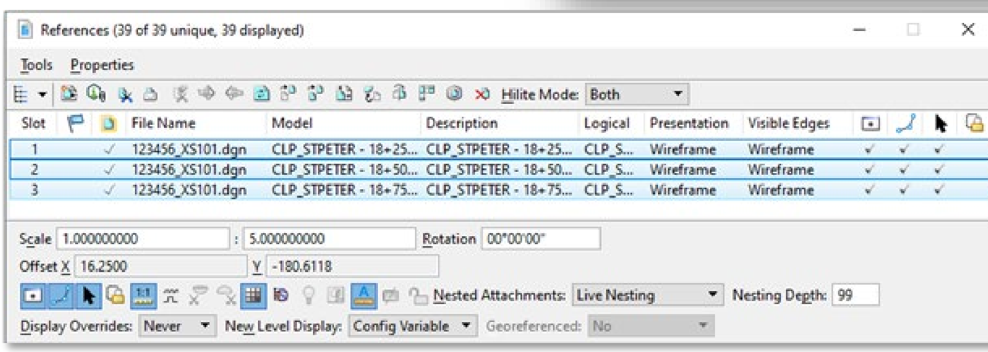
Cross Section Sheet models

These consist of

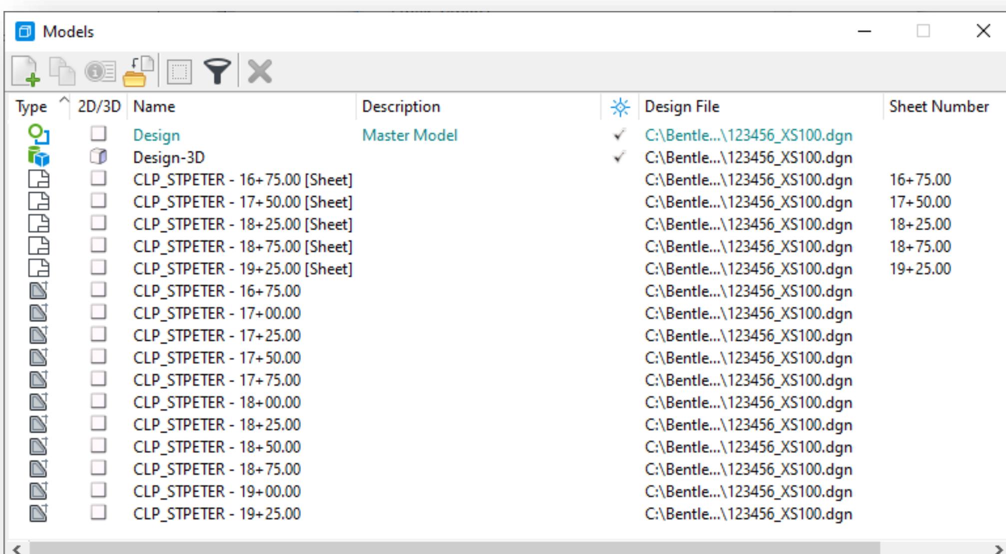
- Sheet borders.
- Referenced Cross section drawing models.
- Sheet totals for seeding, cut, and fill.

Note: No annotations should be placed within the sheet model apart from the listed above.

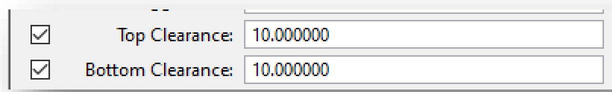


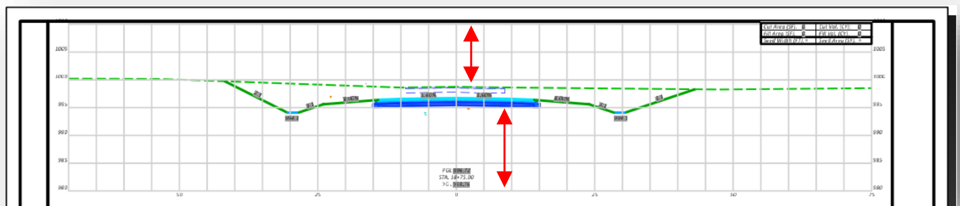


The **Models** dialog (**Home > Models**) can be used to review each of the newly created model, as shown below. The software will attempt to place as many cross-sections as possible on the sheets.

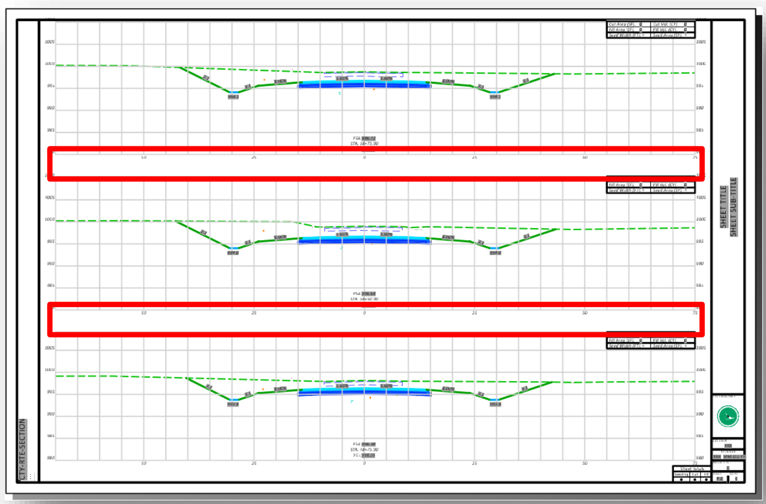


Note: The spacing above and below each cross-section, within the grid lines, is set in the *Place Named Boundary* dialog by setting the **Top Clearance** and **Bottom Clearance** values. The default values for the bottom clearance have been set to account for the presence of underground utilities, however, the value represents a default setting and may not be appropriate for all cases.



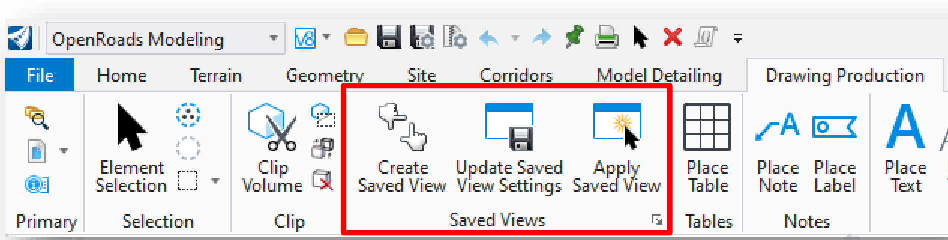


The spacing between the cross sections is set by a configuration variable.

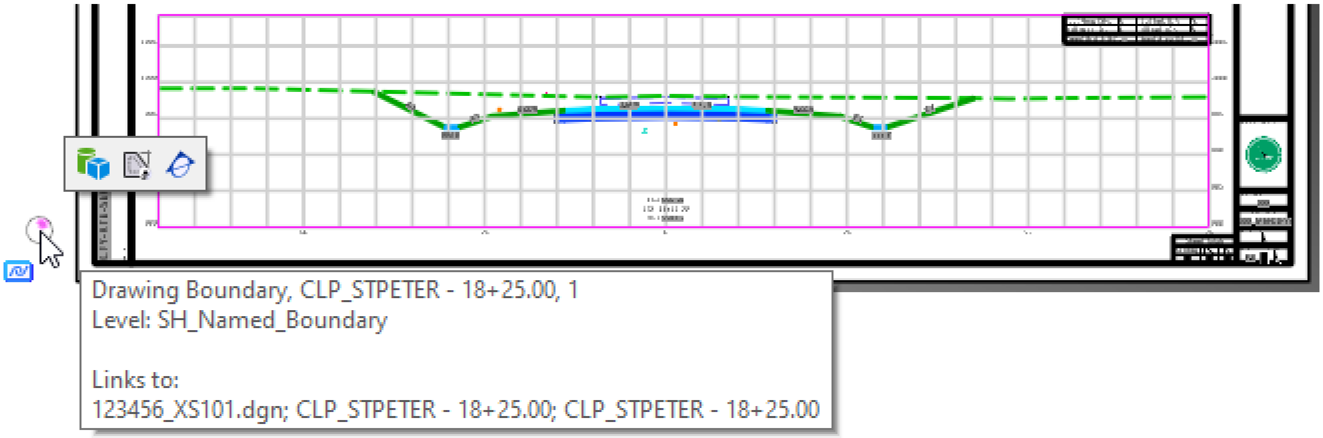


Saved Views

The **Saved Views** tools can be accessed under various workflows under the **Drawing Production** tab. The Saved View tab gives you the ability to apply views as well as to create new views that you can recall later. Views display as thumbnails sorted alphabetically. You can filter the list by using the search bar or change from thumbnails to a list by selecting the list display icon.



Saved views allow you to easily jump to points of interest in the model. A saved view remembers the camera location and orientation along with the levels and models that have been set to display.



Saved Views quick access dialog icons. This can be used to navigate between the cross sections from the sheet models.

 Open Design Model

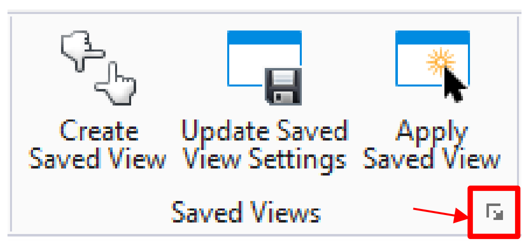
 Open Drawing Model

 Show Callouts *(Not used at this time*)

 Open Sheet Model (O*nly visible from the Drawing and Design models*)

Manage Saved Views

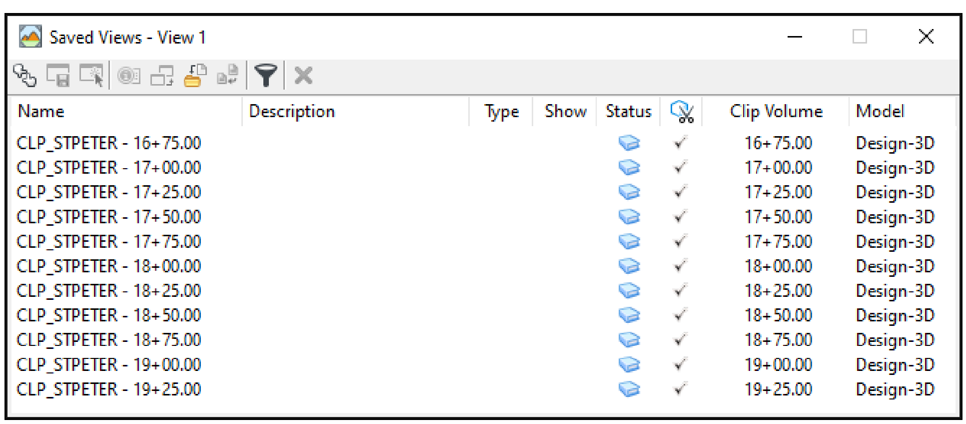
You can view all the **Save Views** of your cross sections within the design model where the cross sections were clipped from using the named boundaries tools. By clicking on the Manage Save Views (shown to the right).



Note: Anytime you use the named boundaries tools to create sheets it will always generate a saved view of that named boundary. *(This is a snapshot of how the view looks at the time of creating the named boundary).*

Within the Saved Views Manager dialog, you can select the saved view and update it from the Design-3D model view 1 if changes have been made after the cross sections named boundary has been created.

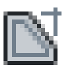
You can also delete the saved views within this dialog.



Note: In the case of the deletion of the Named Boundaries graphic element you will need to open the Saved View Manager and manually delete the saved view for that Named Boundary. If the saved view isn't deleted out of the Saved View Manager, you will run into issues when trying to recreate Named Boundaries.

Drawing Model vs. Sheet Model Usage

Each cross-section is displayed in a drawing model. The various drawing models are referenced to the sheets. Each sheet is contained in its own sheet model. The models are intended to be used as follows:



Drawing Models

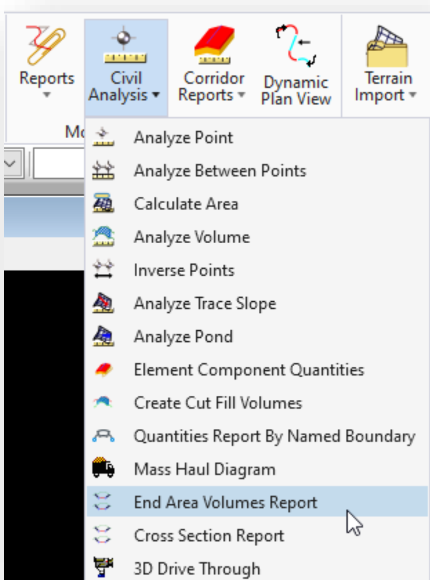
All annotations placed in the cross-sections should be placed in the drawing model. The cross-section grid area represents the clipping limits of the cross-section on the sheets; therefore, anything drawn outside of the grid will not be displayed on the cross-section sheets.

Sheet Models

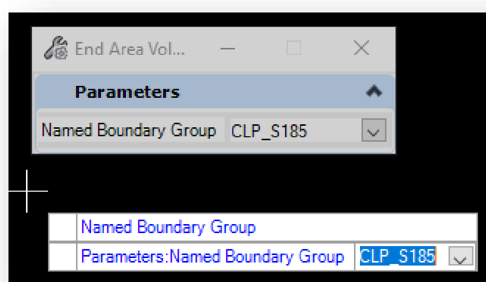
The sheet model is intended to be used for printing. All annotations, other than the sheet border information, should be placed in the various drawing models.

End Area Volume Report

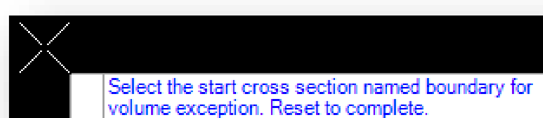
Quantities can be extracted from the 3D mesh elements (see Create Cut and Fill 3D Meshes) using a previously placed set of cross section named boundaries using the **End Area Volume Report**, shown below. When the command is processed, the named boundaries are tagged with the end area quantity values, which can then be used for cross-section annotation.



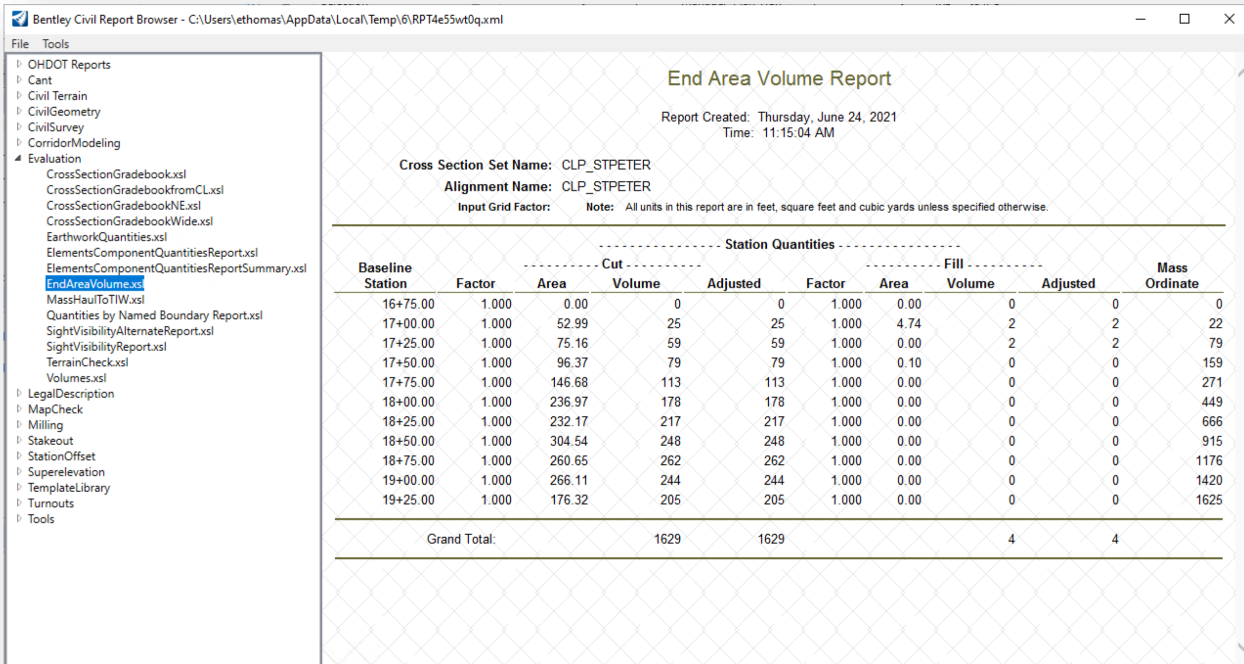
After selecting the command, choose the named boundary group that will be tagged with the quantity information.



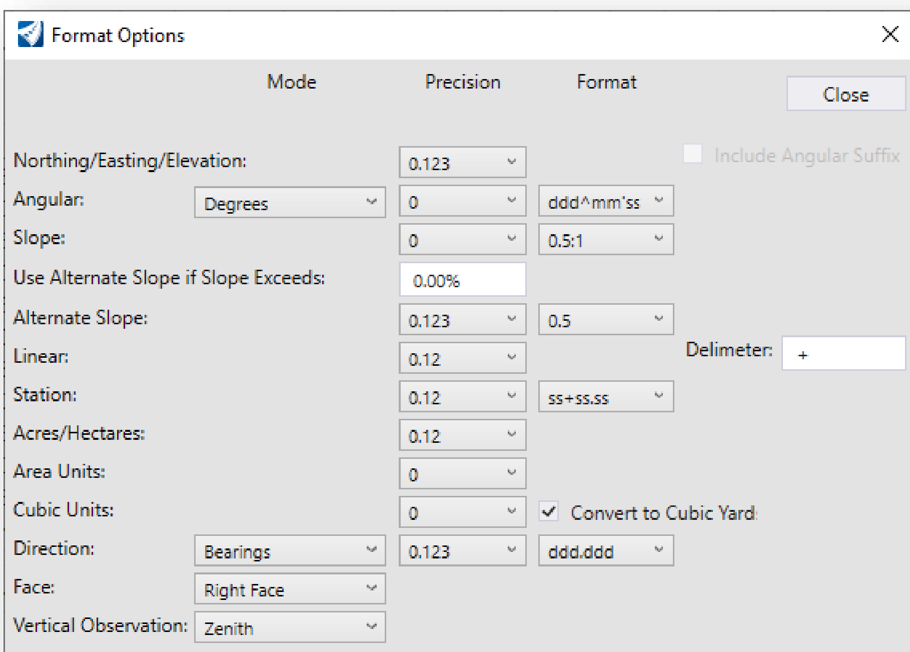
When prompted, select any named boundary group that you wish to exclude from the process. Right-click (reset) when complete.



After processing the **End Area Volume Report** is displayed as shown below.



Each named boundary is tagged with quantity information.
By default, the End Area Volume Report shows cut and fill volumes in Cubic Feet. From the ***Civil Report Browser*** dialog, select **Tools > Format Options** to access the dialog shown below.

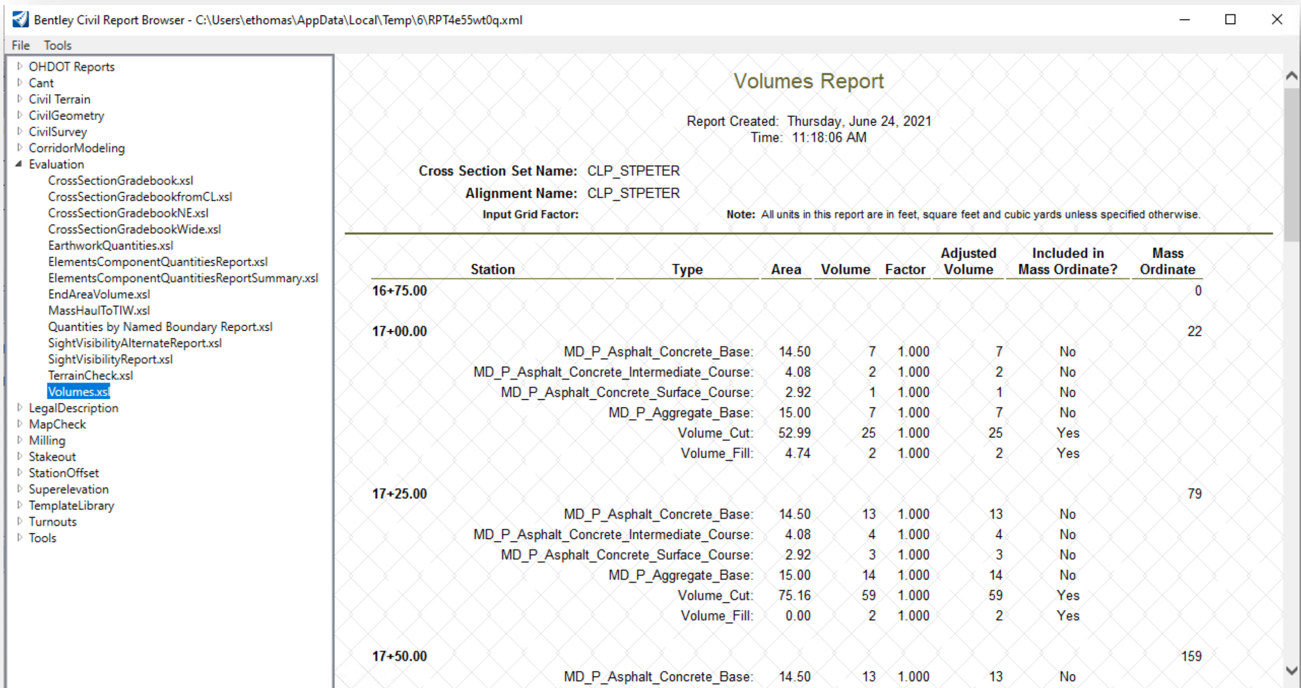


For ODOT projects, toggle on the **Convert to Cubic Yard** option as shown at left.

Select **Close** to apply the changes. Changes in format are applied to the current report values.

Various reports are available as shown on the previous page. It is good practice to save this report by choosing the **File > Save As** option.

Another report you may want to consider saving is the **Volumes** report, shown below. This report shows the quantities for each component.



Displaying End Area Quantities

End area volumes are annotated in a table in the drawing model for each cross section as shown below.

The table is placed as part of the cross-section annotation. The table uses ORD Civil Fields for the end area quantities. These fields are automatically updated using the current values assigned to the named boundary when the drawing model is opened.

Notes:

- The table is placed only if a 3D line string is encountered in the 3D model generated by a point named “PGL” or “EX_Table” in the template that was used to generate the corridor model.

- The Seeding quantities are not computed by ORD. These entries are included in the table for use to manually place seeding quantities in the table.

Updating the End Area Quantities

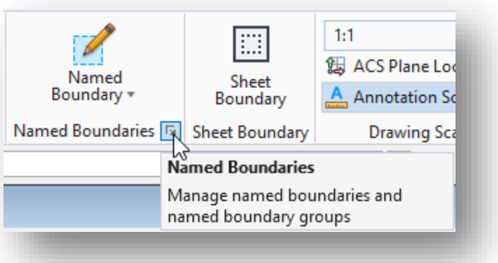
If changes are made to the model, the 3D mesh prismoidal cut and fill elements must be regenerated, and the end area volume report must be reprocessed. Take the following steps to update the quantities if changes are made to the model:

1. Open the DGN file containing the 3D mesh prismoidal volume elements
2. Delete the previously created prismoidal volume cut and fill elements
3. Run the **Home > Civil Analysis > Create Cut Fill Volumes** command
4. Open the cross-section container DGN file
5. Run the **Home > Civil Analysis > End Area Volume Report** command
6. Open each cross-section drawing model. The quantity fields are automatically synchronized with the earthwork quantity values that have been assigned to the named boundaries as each cross-section drawing model is opened

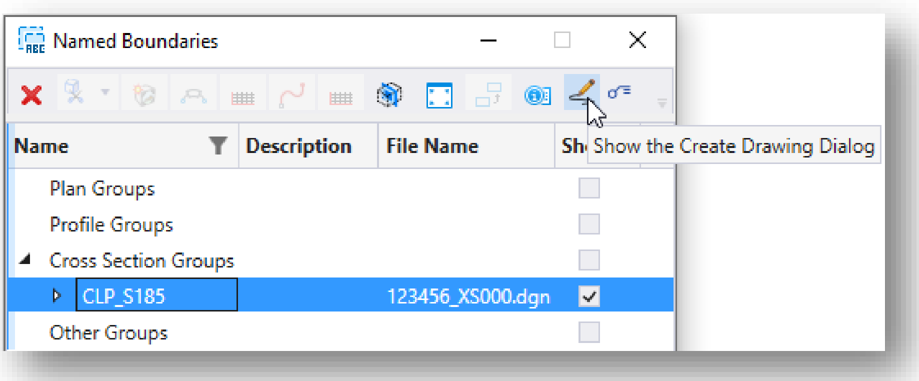
Processing Sheets from Previously Generated Named Boundaries

If the named boundaries were generated without creating the sheets, as is the case when running the **End Area Volume Report** after generating the named boundaries, the sheets can be generated from the named boundaries with the following steps:

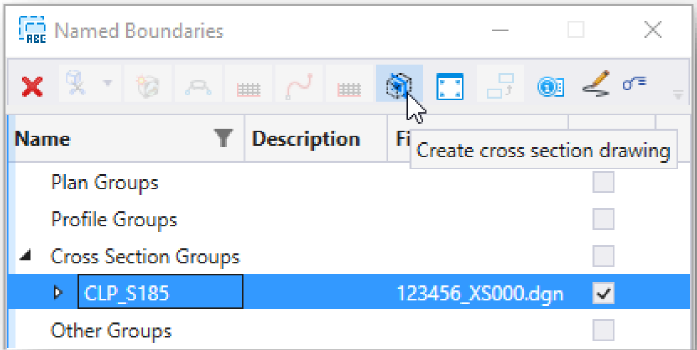
1. From the **Drawing Production** tab, select the icon at the lower right of the **Named Boundaries** group as shown below. The ***Named Boundaries*** dialog is opened.



2. In the ***Named Boundaries*** dialog, select the cross-section group from the list. Toggle the **Show the Create Drawing Dialog** option on as shown below.



3. Choose the **Create Cross Section Drawing** command. The ***Create Drawing*** dialog, previously described, is opened. Complete the ***Create Drawing*** dialog to generate the cross-section sheets.



Adding Cross Sections at Specific Stations

The initial cross section named boundary placement is defined by selecting an alignment and specifying an interval. Additional named boundaries for specific locations such as a driveway or a drainage structure can be inserted using the **Civil Cross Section 2 Points** option.

Additional cross section locations may be defined prior to, or after generating the initial sheets. Each workflow is summarized below.

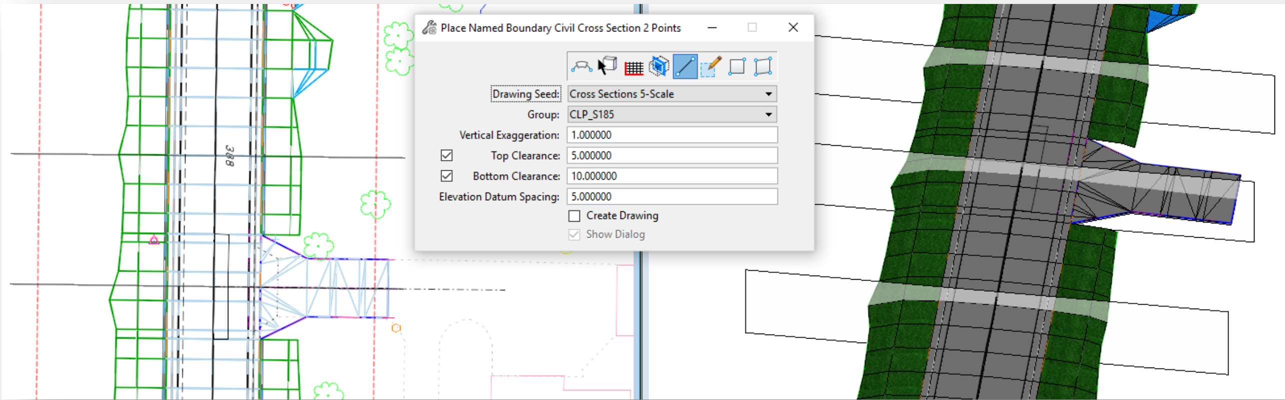
Adding Named Boundary Locations Before Generating the Initial Sheets

When the initial set of named boundaries is defined for the cross sections, be sure to toggle the **Create Drawing** option OFF. This allows the user the opportunity to define additional named boundaries at specific locations not covered by the initial named boundary interval placement.

Take the following steps to define additional cross section named boundary locations:

- From the **OpenRoads Modeling** WorkFlow, select **Drawing Production > Named Boundary** to initiate the process. The ***Place Named Boundary*** dialog is opened.
- If multiple locations are to be defined, toggle the **Create Drawing** option OFF
- Select the fifth icon, **Civil Cross Section 2 Points** to begin the process of placing the cross section named boundaries
- Select the desired **Drawing Seed** for the cross section named boundaries.
- Select the **Group**. The drop-down list will include any previously defined cross-section groups.
- Select the alignment for the named boundaries
- After the alignment has been identified, enter data points for the start and end of the named boundary location. A third data point is required to accept the named boundary placement
- To define additional named boundary locations, the path element (alignment) must be identified each time. Follow the prompts in the lower left corner of the MicroStation window.

In the example below, a named boundary has been added at a driveway location.



Tip: For best results, create any additional named boundaries using Civil AccuDraw to define the station and offset values for the boundary. Use the same offset values as the primary cross-sections.

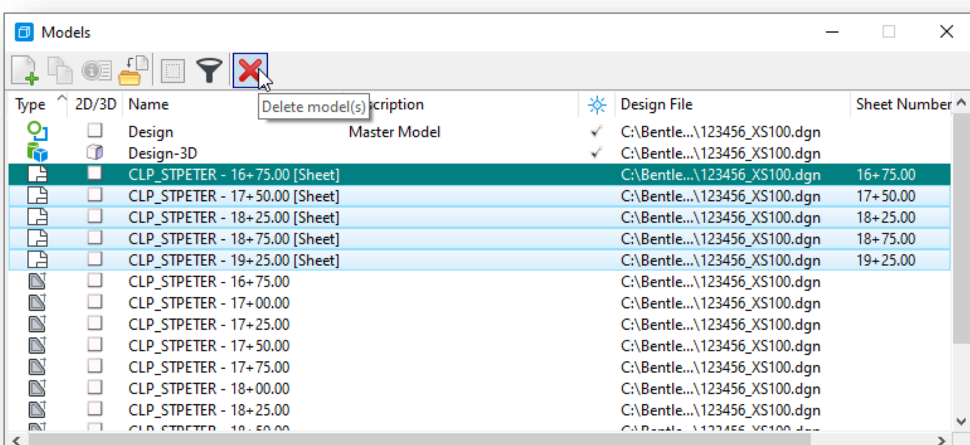
- Toggle on the **Create Drawing** option ON prior to defining the last boundary to launch the ***Create Drawing*** dialog.

  If you forget to toggle this option back on, the ***Create Drawing*** dialog can be accessed from the ***Named Boundaries*** dialog. See Processing Sheets from Previously Generated Named Boundaries, for more information.

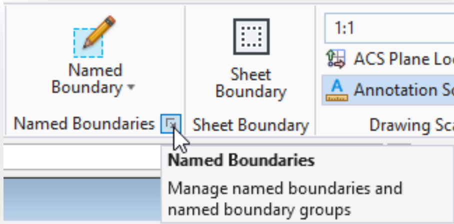
Adding Named Boundary Locations to Previously Generated Sheets

Occasionally it is necessary to add a cross-section location after the initial sheets have been generated. OpenRoads version 10.10 provides functionality to generate new sheet models while retaining previously generated cross-section drawing models.

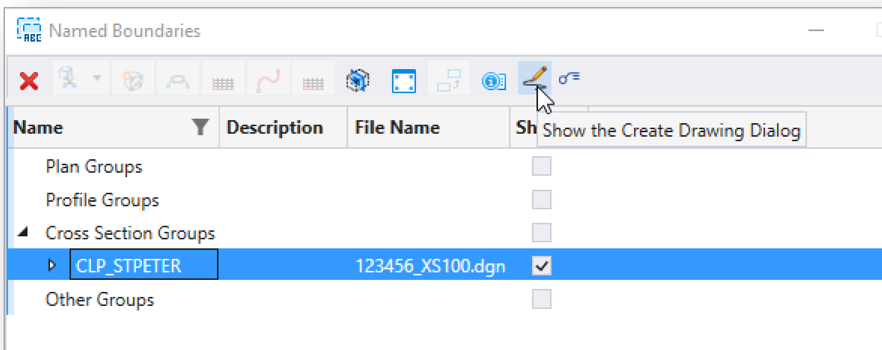
Before processing a new set of cross-section sheets, it is necessary to manually delete previously generated sheet models. Open the Models dialog from one of the OpenRoads WorkFlows by selecting the **Models** icon on the **Home** tab. Select the sheet models to be deleted, and then choose the **Delete Model(s)** icon like the example below.



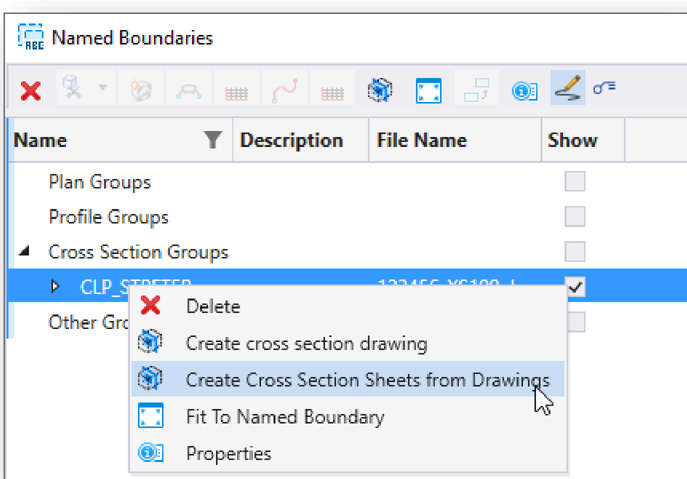
1. From the **Drawing Production** tab, select the icon at the lower right of the **Named Boundaries** group as shown below. The ***Named Boundaries*** dialog is opened.



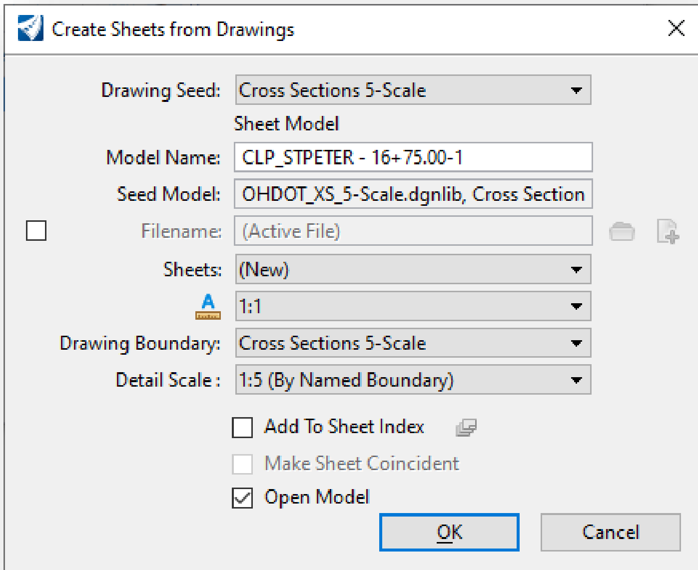
2. In the ***Named Boundaries*** dialog, select the cross-section group from the list. Toggle the **Show the Create Drawing Dialog** option on as shown below.



3. Right click on the named boundary group name and choose the **Create Cross Sections from Drawings** option.

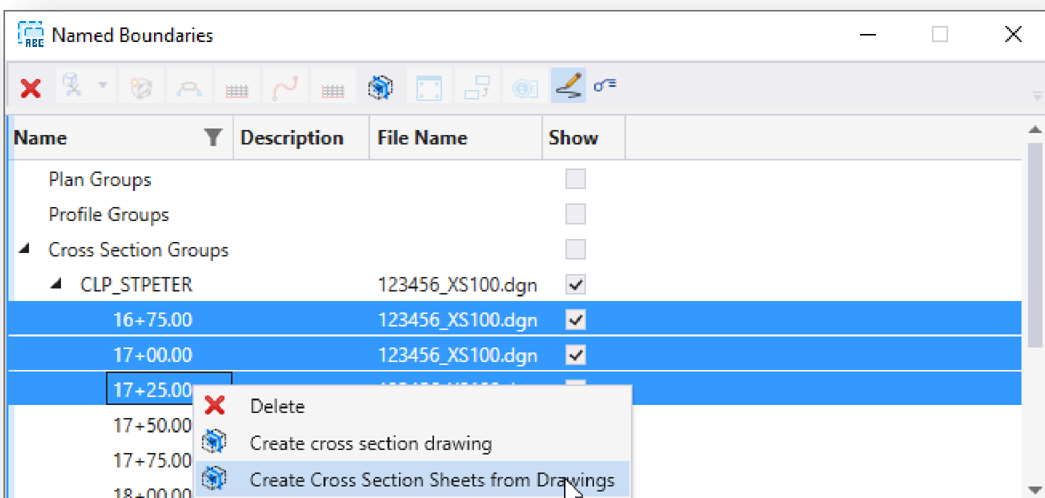


4. The ***Create Sheets from Drawings*** dialog is opened as shown below.



Select **OK** to initiate the process.

It is also possible to create new sheets from specifically selected named boundaries as shown below.



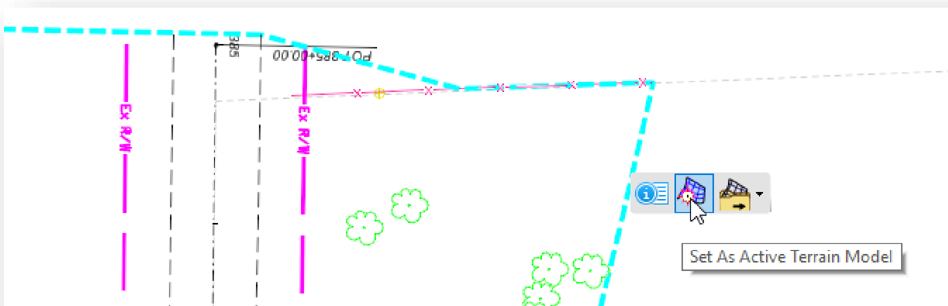
Displaying Right-of-Way on Cross-Sections

The location of both the existing and proposed Right-of-Way lines can be displayed in a dynamic cross-section view and on the cross-section sheets like the example shown below:

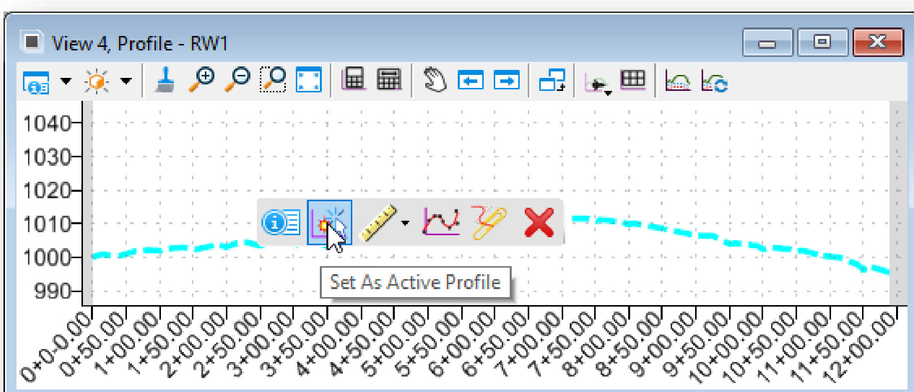
To display the Right-of-Way lines on the cross-sections, each Right-of-Way line must have an existing profile defined. This is accomplished by opening a profile view for each Right-of-Way line and defining the exiting ground profile as the active profile. When the profile is defined for right-of-way lines, a 3D line string is created in the 3D model for the line.

The workflow to assign a profile to the right-of-way lines is as follows:

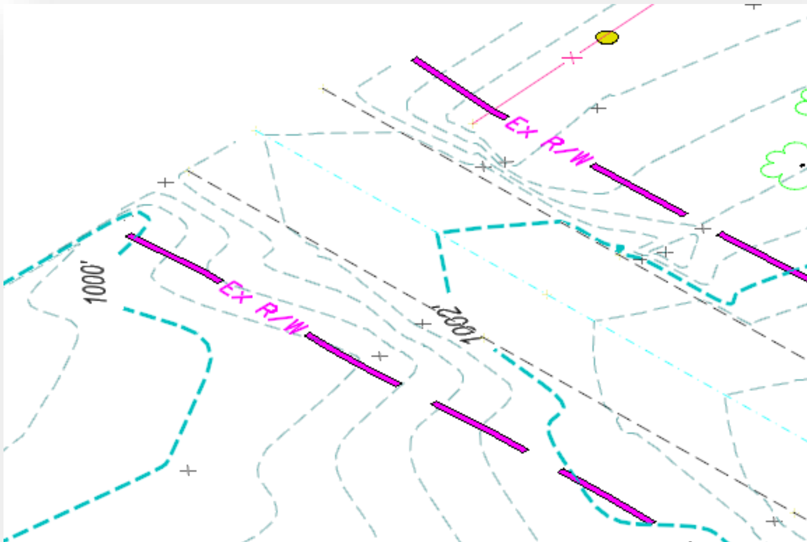
- In the right-of-way basemap design file (BR), attach the design file containing the existing terrain
- Set the referenced terrain model as the active terrain by selecting the terrain and choosing the **Set As Active** Terrain option from the pop-up menu



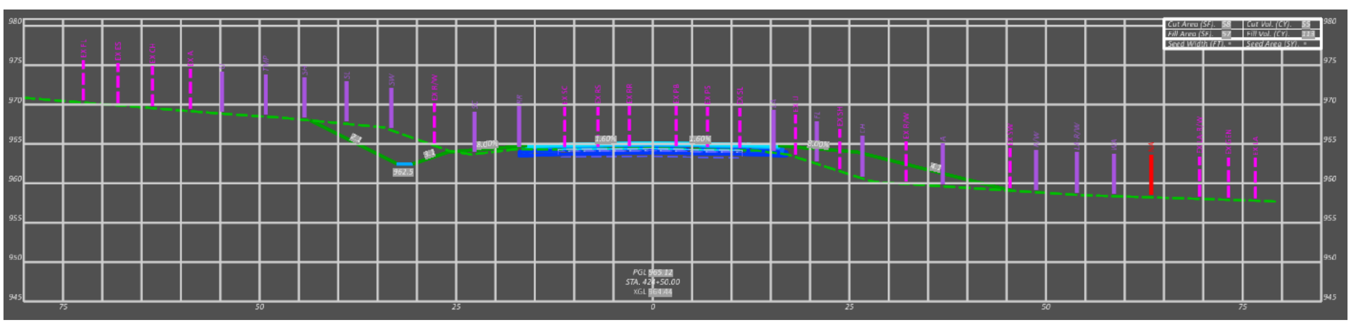
- Open a profile model for each right-of-way line, either by selecting the element and choosing the Open **Profile Model** option, or by holding down the right-mouse button and choosing **View Control > 2 Views Plan/Profile** from the pop-up menu.
- Select the groundline in the profile window and choose the **Set As Active Profile** option from the pop-up menu



A 3D element is generated in the 3D model like the example below.



When cross sections are generated, if the cross section intersects a 3D Right-of-Way line in the 3D model, a cell is placed in the cross section at the right-of-way line offset and elevation.



Section 803 Cross-Section Sheet Exercise

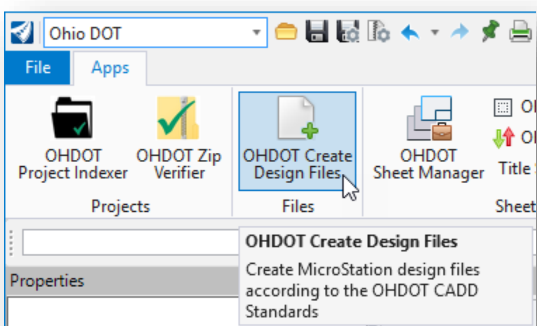
Exercise 1: S.R. 185 Cross Sections

Part A: Design Files

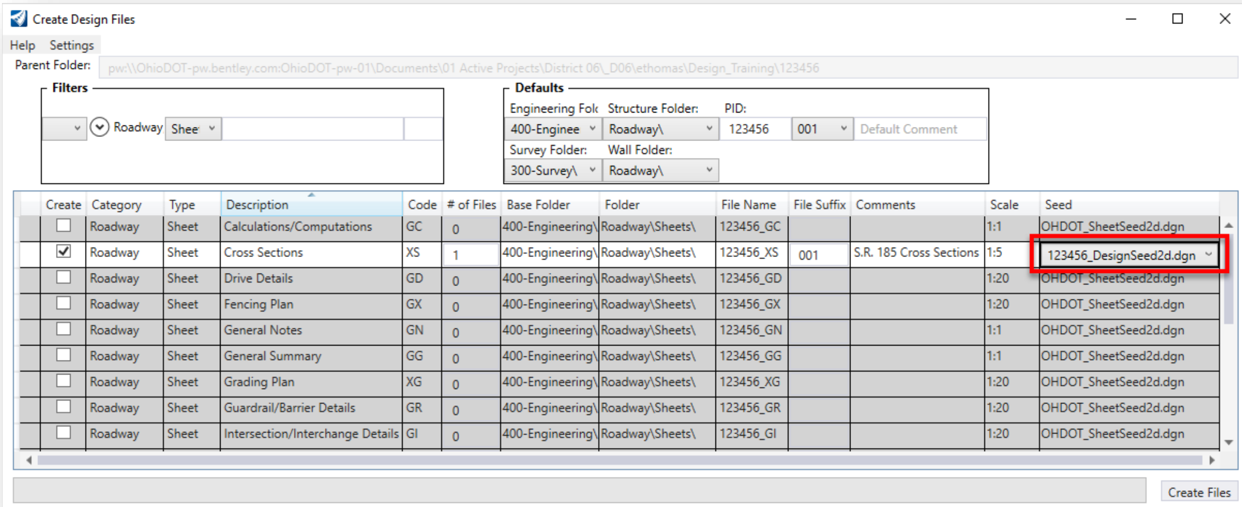
Take the following steps to create the design file for the cross-section sheets:

- Open **ProjectWise Explorer**

- Browse to the folder containing the training data. The folder location will be provided by the instructor for each class.
- Browse to the **402-Engineering\Roadway\Basemaps** folder.
- Open the file **123456_BK000.dgn**
- Select the **Ohio DOT** Workflow
- Select the **OHDOT Create Design Files** application



- Create the cross-section sheet design file for S.R. 185 using the settings below:



- Be sure to change the **Seed** file option to use the **123456_DesignSeed2d.dgn** file.
- Select **Settings > Open Last Created File**
- Select the **Create Files** option

  The new file is created in the **402-Engineering\Roadway\Sheets** folder

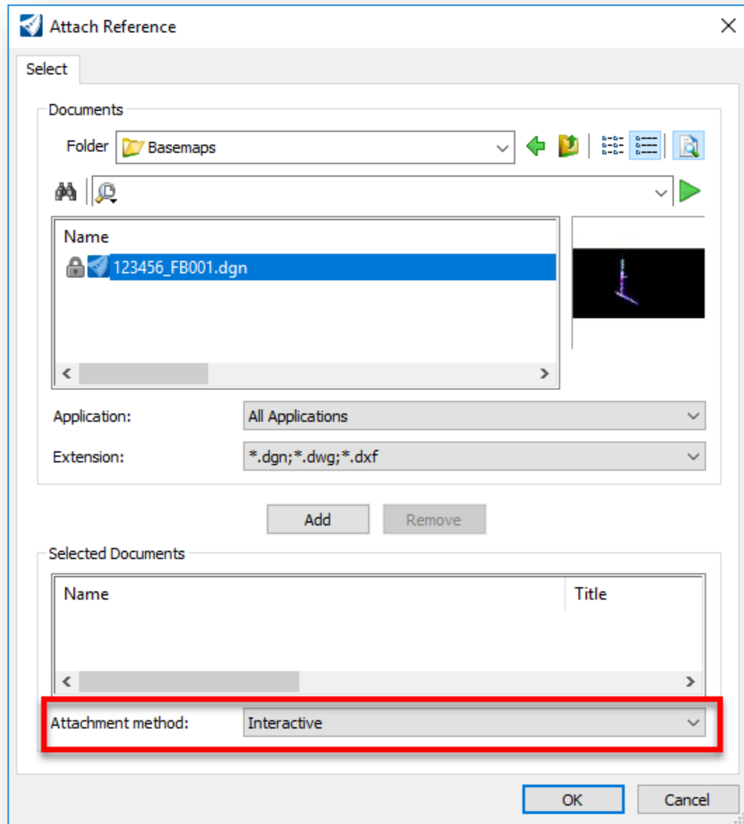
- When prompted, check the current file back into ProjectWise

Part B: Plan View References

- From the **OpenRoads Modeling** WorkFlow, select **Home > Attach Tools > References** to attach the files with the plan view information. Start by attaching the Survey Field Book (FB) design file from the following folder:

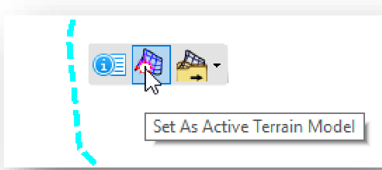
  300-Survey\Basemaps\123456_FB001.dgn

- When selecting the references, set the **Attachment Method** to **Interactive**. This will open the ***Reference Attachment*** dialog for each file as it is attached. This is necessary to review and set the **Nested Attachments** parameter for each file.



- From the ***Reference Attachment*** dialog, set the **Nested Attachments** parameter to **No Nesting**.

- After attaching the Survey Field Book design file (FB) set the terrain model as the active terrain by selecting the terrain model element and choosing the **Set As Active Terrain Model** icon from the pop-up menu.



The cross sections are extracted from the 3D model.

✓ Open a 3D view by holding down the right-mouse button in the view window and choosing **View Control > 2 Views Plan/3D**

Attach the files that contain the 3D information that will be displayed on the cross-sections. Set the **Nested Attachments** setting to **Live Nesting** with a depth value of **0** unless otherwise stated.

- 401-Engineering\Roadway\Basemaps\123456_BK001.dgn
- 401-Engineering\Roadway\Basemaps\123456_BK002.dgn
- 401-Engineering\Roadway\Basemaps\123456_BK003.dgn
- 401-Engineering\Roadway\Basemaps\123456_KM001.dgn
- 401-Engineering\Roadway\Basemaps\123456_KM002.dgn
- 401-Engineering\Roadway\Basemaps\123456_KM003.dgn
- 401-Engineering\RW\Basemaps\123456_BR001.dgn

- After the references are attached, select the 3D view so that it is the active view. From the **Home** tab, select the **Level Display** icon to turn on/off the levels that you wish to display in the cross-section sheets.
  - Select each reference file to turn on/off the desired levels
  - The only thing that is needed from the survey basemap (FB) design file is the terrain model. Turn off all the levels in the FB design file except the following
    - DT_X_TIN_Hull
    - XS_X_Ground_Line

- Select **File > Save Settings** after you have completed the task of turning on/off the levels

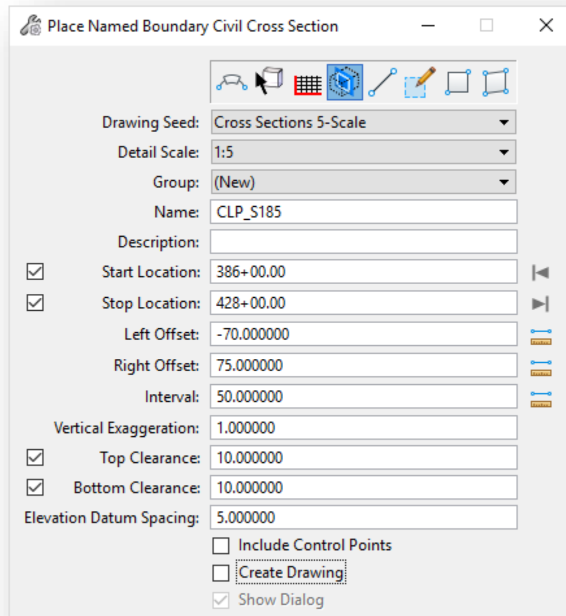
Part C: Placing Named Boundaries

MicroStation Named Boundaries are placed to define the limits of each cross-section.

Take the following steps to define the cross-section named boundaries for S.R. 185:

- ✓ From the **Drawing Production** tab, select the **Place Named Boundaries** command

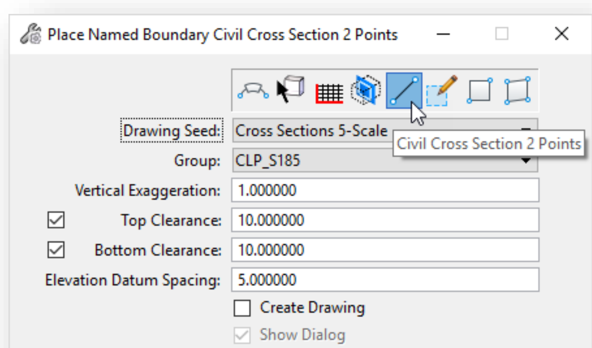
- ✓ From the ***Place Named Boundary*** command, take the following steps:
    - Select the **Civil Cross Section** command (4th icon)
    - Set the **Drawing Seed** parameter to **Cross Sections 5-Scale**
    - Toggle off the **Create Drawing** option at the bottom of the dialog.
    - Click inside the **Start Location** key-in field
    - Left-click on the plan view alignment graphic for S.R. 185. The dialog will track the start location station value relative to the cursor location
    - Key in a **Start Location** value of **386+00**
    - Key in a **Stop Location** value of **428+00**
    - Key in a value of **50** for the **Interval** parameter
    - Enter a data point (left-click) in the plan view to initiate the process to create the named boundaries for the cross-sections. It takes three clicks to place the boundary elements.



The boundaries are drawn for the cross sections and the ***Placed Named Boundary*** dialog is closed.

Additional cross sections at specific locations, such as at a driveway or culvert, can be specified by selecting the **Named Boundary** command and entering a single location by setting the **Start Location** and **End Location** to the same value. Set the **Group** parameter to use the same group as the previously placed named boundaries rather than **(New)** as shown above.

Cross section named boundaries can also be placed at a single location by using the **Civil Cross Section 2 Points** option as shown at right.

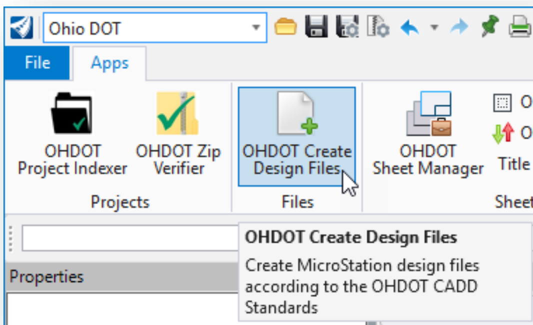


Part D: End Area Quantities

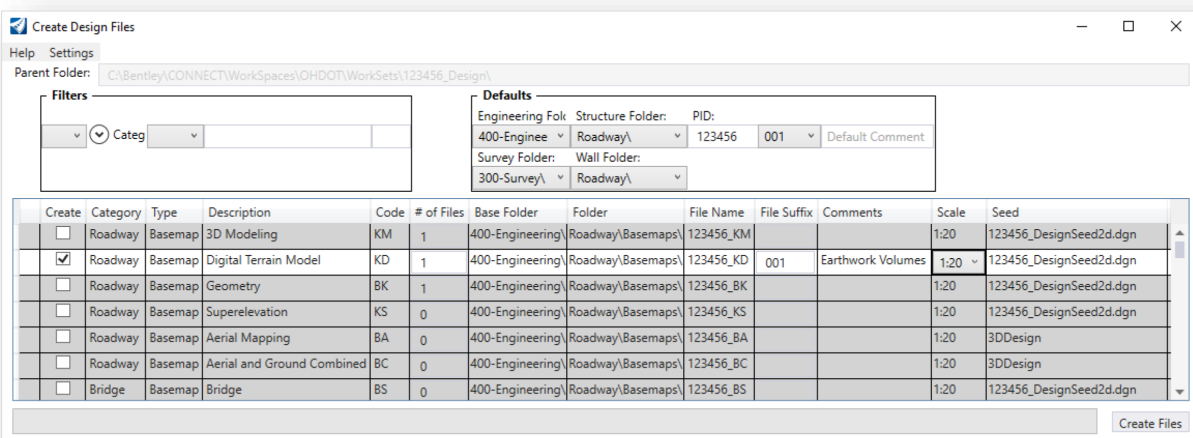
Quantities can be extracted from the 3D mesh elements using the location of a previously placed set of cross section named boundaries. The named boundaries are tagged with the quantity values, which can then be used for cross section sheet annotation.

Take the following steps to create a new design file to generate 3D mesh elements from the corridor model graphics.

- Select the **Ohio DOT** Workflow
- Select the **OHDOT Create Design Files** application



- Create the design file for 3D mesh elements using the settings below:



  - Be sure to change the **Seed** file option to use the **123456_DesignSeed2d.dgn** file.
  - Select **Settings > Open Last Created File**
  - Select the **Create Files** option

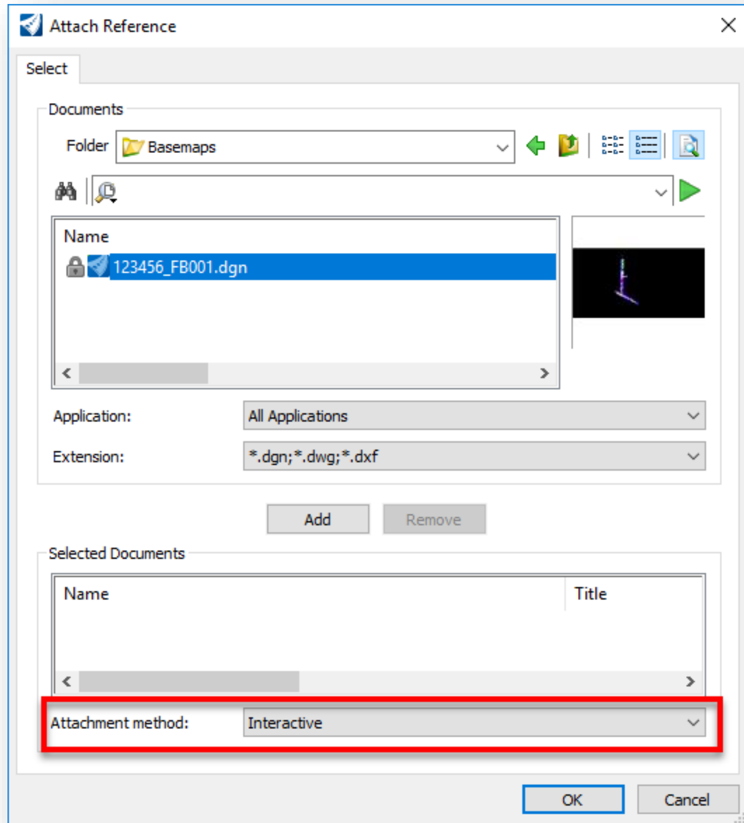
    The new file is created in the **402-Engineering\Roadway\Basemaps** folder

- When prompted, check the current file back into ProjectWise

- From the **OpenRoads Modeling** WorkFlow, select **Home > Attach Tools > References** to attach the files with the plan view information. Start by attaching the Survey Field Book (FB) design file from the following folder:

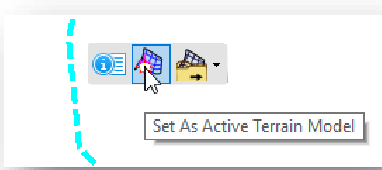
  300-Survey\Basemaps\123456_FB001.dgn

- When selecting the references, set the **Attachment Method** to **Interactive**. This will open the ***Reference Attachment*** dialog for each file as it is attached. This is necessary to review and set the **Nested Attachments** parameter for each file.



- From the ***Reference Attachment*** dialog, set the **Nested Attachments** parameter to **No Nesting**.

- After attaching the Survey Field Book design file (FB) set the terrain model as the active terrain by selecting the terrain model element and choosing the **Set As Active Terrain Model** icon from the pop-up menu.



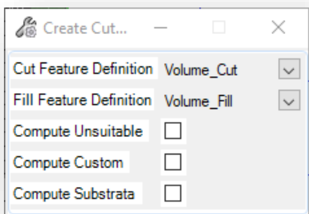
The 3D mesh elements are created from the 3D model.

✓ Open a 3D view by holding down the right-mouse button in the view window and choosing **View Control > 2 Views Plan/3D**

Attach the files that contain the 3D information that will be used to extract the volume elements. Set the **Nested Attachments** setting to **Live Nesting** with a depth value of **1**.

- 400-Engineering\Roadway\Basemaps\123456_BK000.dgn
- 400-Engineering\Roadway\Basemaps\123456_KM001.dgn
- 400-Engineering\Roadway\Basemaps\123456_KM002.dgn
- 400-Engineering\Roadway\Basemaps\123456_KM003.dgn

✓ From the **Home** tab, select **Civil Analysis > Create Cut Fill Volumes**
- Issue a data-point (left-click) to accept each of the parameters.



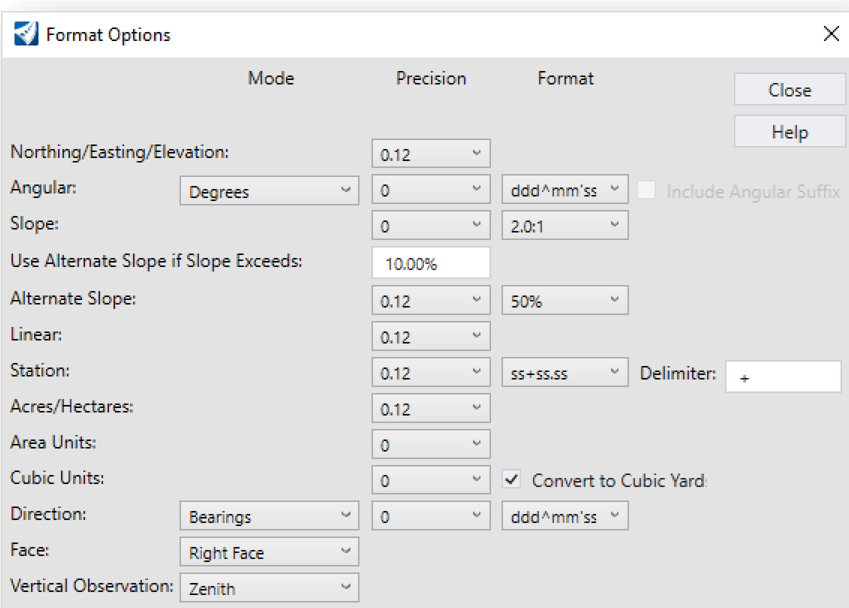
The 3D meshes are placed in the 3d Model on the following levels:

- VL_Cut_Volume
- VL_Fill_Volume

Part E: Cross Section Sheets

After placing the named boundaries and generating the 3D meshes for the cut and fill quantities, take the following steps to attach the design file containing the cut and fill 3D mesh elements to generate the end area quantities.

- ✓ Open the cross-section sheet design file. Be sure to check the current file back into ProjectWise.
  - **402-Engineering\Roadway\Sheets\123456_XS001.dgn**
- ✓ Attach the file containing the 3D mesh elements as a reference to the 2D plan view.
  - From the **Home** tab, select **Attach Tools > References**
  - Attach the following file with **No Nesting**: **402-Engineering\Roadway\Basemaps\123456_KD001.dgn**
- ✓ Turn off the display of the 3D mesh elements in the 3D view
  - Click the 3D view window to make it active
  - From the **Home** tab, select **Level Display**
  - Turn off the following levels:
    - VL_Cut_Volume
    - VL_Fill_Volume
- ✓ From the **Home** tab, select **Civil Analysis > End Area Volume Report**
  - When prompted, choose the **CLP_S185 Named Boundary Group**
  - Left-click to accept the selection. Follow the prompts to generate the end area volume report
- ✓ The report is displayed in the ***Civil Report Browser*** dialog. Select **Tools > Format Options** to adjust the output format of the report. The settings shown below are recommended for ODOT projects.

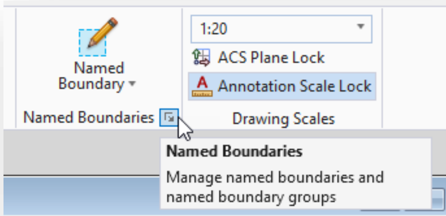


Changes to the **Format Options** are applied to the active report when the dialog is closed.

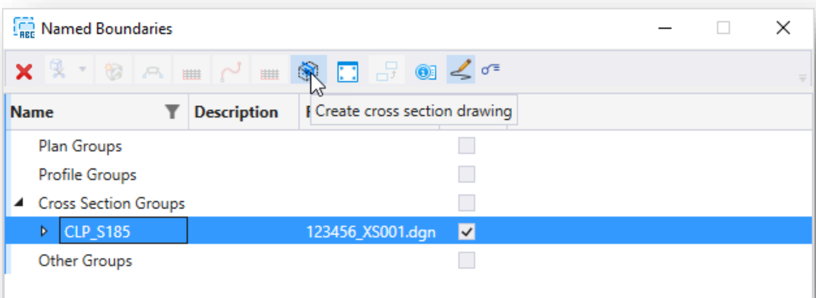
Close the report when complete.

Create the cross-section sheets.

- ✓ From the **Drawing Production** tab, select the **Manage Named Boundaries** icon in the lower right corner of the **Named Boundaries** group.



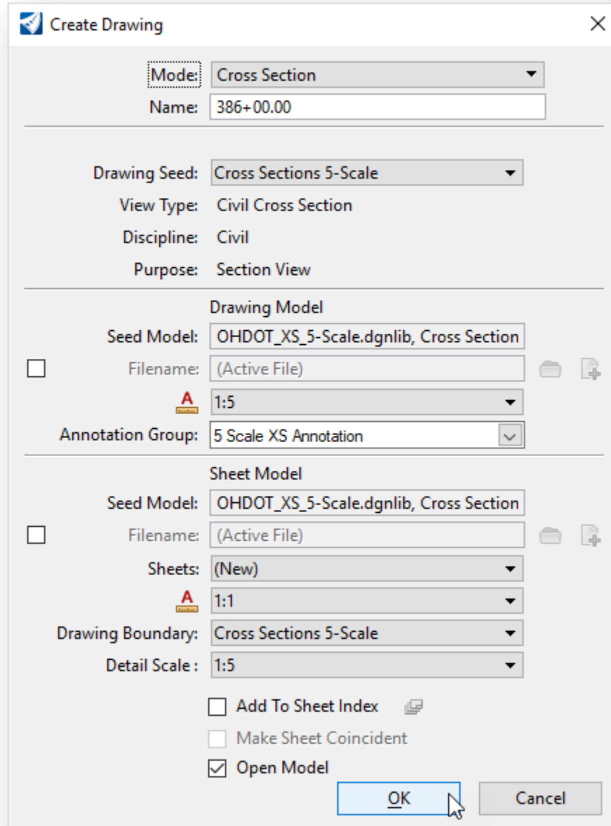
- ✓ From the ***Named Boundaries*** dialog, take the following steps:
  - ○ Toggle on the **Show the Create Drawing Dialog** option
  - ○ Select the **Cross Section Group** name from the list
  - ○ Select the **Create cross section drawing** icon.



The ***Create Cross Sections*** dialog is opened as shown below.

The settings on this dialog have been preconfigured for ODOT projects.

✓ Select **OK** to initiate the sheet creation process.



A drawing model is created for each named boundary.

The sheets are composed by referencing the drawing models.

Earthwork quantities are annotated on each cross section in the drawing model. If the **End Area Volume Report** is processed again, the quantity values are updated for each named boundary. The value shown in the drawing model is updated when each drawing model is opened. All the drawing models can be updated at once by removing the annotations from all drawing models and the re-annotating all the drawing models (**Drawing Production > Model Annotation > Remove Model Anntations/Annotate Model**)

Annotation is typically placed in the drawing models, not in the sheet models.

✓ Take some time to review the cross-section model and sheets.

This completes this exercise.

✓ Exit OpenRoads Designer, checking the file back in to ProjectWise

Section 900 Drainage and Utilities

Overview

Drainage and utilities, such as culverts, catch basins, manholes, telephone, water, gas lines, etc., must be defined in the 3D model to display this information on the roadway profile and cross sections. This guide will provide an overview of the process to model Drainage and Utility features in 3D, using the OpenRoads Drainage and Utility tools.

This guide will go over the modeling side of the Drainage and Utility tools. There is a separate guide for running analytics on drainage networks that is currently in development.

Configuration and setup

All the features set up for Drainage and Utility have been created within the OHDOT_Features_DrainageAndUtilities.dgnlib file. This file also contains all the settings for a Drainage & Utility Project, such as prototypes, catalogs, storm events, scenarios, flex tables, and default design constraints. The 2D and 3D cells that are used with the feature definitions are defined in the OHDOT_DraiangeAndUtilities.cel file. These cells should not be placed directly using the place cell tool. They are made specifically for use with the Drainage and utilities feature definitions.

Drainage & Utility Project

A Drainage & Utility Project contains all the settings for drainage and utilities. This includes prototypes, catalogs, storm events, scenarios, flex tables, and default design constraints.

Whenever a Drainage and Utilities tool is used a Drainage & Utility Project is created within that file if one does not already exist. This Drainage & Utility Project is a copy of all the Drainage and Utility settings that are set up in the OHDOT_Features_DrainageAndUtilities.dgnlib. Please note this means that any future updates to the OHDOT_Features_DrainageAndUtilities.dgnlib will not be used/applied to this file since the Drainage & Utility Project is already created. This does not copy all the feature definitions within the OHDOT_Features_DrainageAndUtilities.dgnlib file as those get copied into the file as they are used.

Drainage and Utility Features

Feature Definitions are currently defined as the following 3 different feature definition types

- Nodes – consists of point like features for example Catch Basins, Headwalls, Manholes…
- Conduit -consists of linear like features for example Gas line, Fiber Optic Line, and Culverts
- Drainage Area- consists of polygon like features for example catchment areas, ponds…

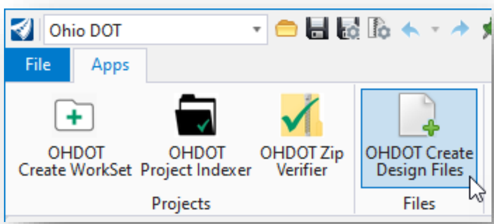
Use the OpenRoads Standards tab of Explorer within ORD to see the available feature definitions

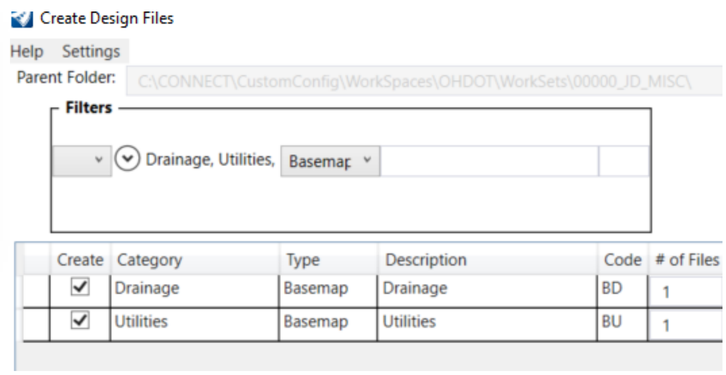
Modeling Existing Drainage and Utility Features

Majority of the time existing Drainage and Utility features will be displayed within the survey fieldbook (FB) file. It is important to understand these graphics from the survey fieldbook are not truly modeled and are not using the Drainage and Utilities Feature Definitions. But these graphics can be used to help create your Drainage and Utility model.

Creating Drainage and Utility models should be done within separate files. For Drainage a “BD” drainage basemap design file should be created and for Utility a “BU” Utility basemap design file should be used.

The **OHDOT Create Design Files** application can be used to create these files.



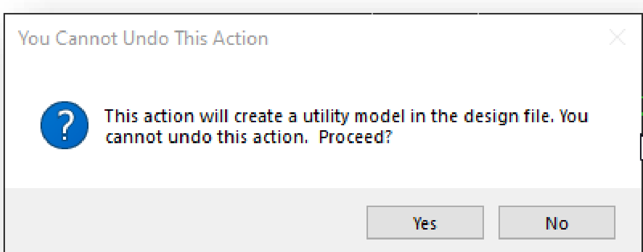


If existing drainage or utility graphics are contained with the projects Survey Fieldbook then the **Extract From Graphic** tool can be used to create existing drainage or utility models.

Extract From Graphic

The **Drainage and Utilities > Layout > Extract From Graphic** tool is used to extract 3D information from a 2D plan view graphic.

The first time a Subsurface Utility command is selected, the software prompts the user to create a utility model in the design file if one has not been previously created. Select **Yes** when prompted as shown below.

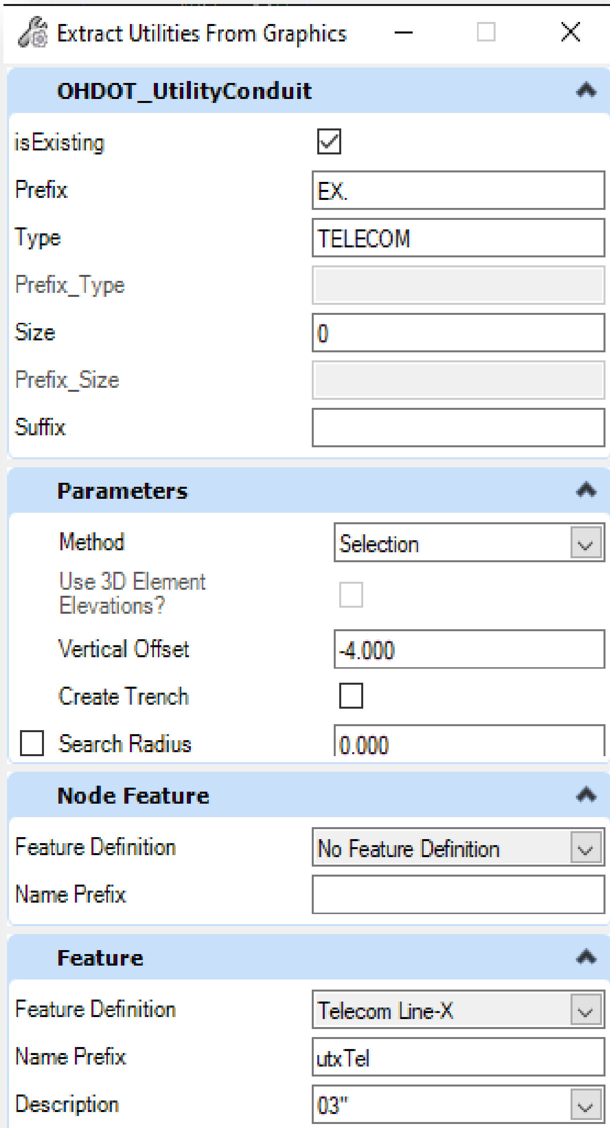


After the utility model has been created, the user must reselect the Drainage & Utility command to initiate the command.

The ***Extract Utilities from Graphic*** dialog is shown at left below.

Method

The elements that will be used to create the 3D model can be selected, using the **Selection** option, either by selecting a single element or by a selection set, or by use of a **Graphical Filter**, which creates 3D utility models for many different types of utility types as defined in the filter group.



When the **Method** parameter is set to **Selection**, the dialog inputs are as follows:

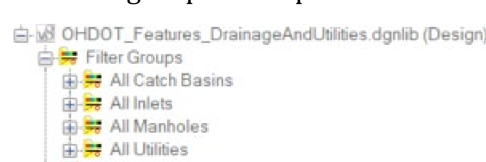
- **Vertical Offset** – Defines the depth of the utility like under the selected surface.
    - ***NOTE****: The depth value is assumed to be the depth to the top of the utility line.*
- **Create trench** – Defines whether to create the trench defined in feature definition. Currently, ODOT does not provide feature definitions for trenching.
- **Design Stage** – Selection of OpenRoads design stage which defines settings applied to the trench corridor, such as stroking tolerances
- **Feature Definition** – Assigns the selected feature definition to the resulting utility features
- **Name Prefix** – The prefix that is used to name the created 3D utility element
- **Description** – Pick list for the utility size

When the Method parameter is set to Graphical Filter, the dialog inputs are as follows:

- **Vertical Offset** – Defines the depth of the utility like under the selected surface.
    - ***NOTE****: The depth value is assumed to be the depth to the top of the utility line.*
- **Create trench** – Defines whether to create the trench defined in feature definition. Currently, ODOT does not provide feature definitions for trenching.
- **Select Filter**- defines which filter or filter group to use to extract utilities.
- **Design Stage** – Selection of OpenRoads design stage which defines settings applied to the trench corridor, such as stroking tolerances

Filter Manager

Filter Manager is set up with several filters and filter groups to make utility extraction easier. The OHDOT workspace has 4 main filter groups, imaged below. Each filter group is set up to extract existing features from graphics. For example, the All Catch Basins filter group will find all existing catch basin cells.



Filters are not set up for extracting drainage pipes as there is not currently a reliable way to extract the correct pipe size & material.

Utility line filters have default depths below the surface as defined by the table below.

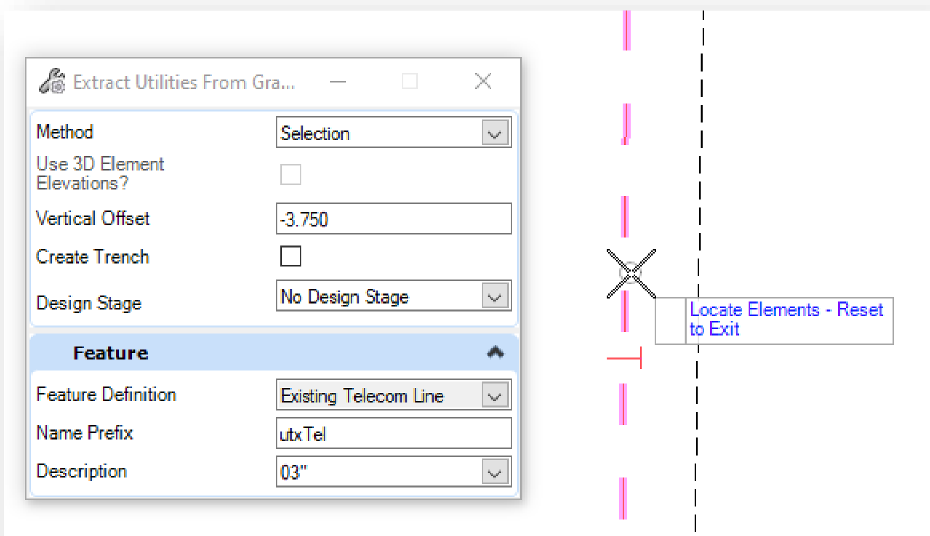
Utility Lines	Default Depth	Default Size
Cable, TV, Telecom, Fiber Optic Lines	-2	2”
Electrical Lines	-2.5	2”
Gas Lines	-3	6”
Water Lines	-4.5	6”
Oil Lines	5	6”

Extracting Utility Features

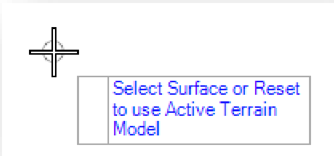
- To extract utility lines, you can use the **Extract From Graphic** tool in the select mode or the filter mode.

  In the **Selection Method**,

  - Set the Desired Feature Definition, Feature Name (Name Prefix), and size (Description).
  - Select the element(s) to be extracted to create the utility model
  - These selected utility elements may result from survey processes, GIS graphic data, OpenRoads Geometry, or other sources. But, in every case the elements are DGN graphic elements. In the example below, a MicroStation SmartLine was used to draw the telephone line graphics

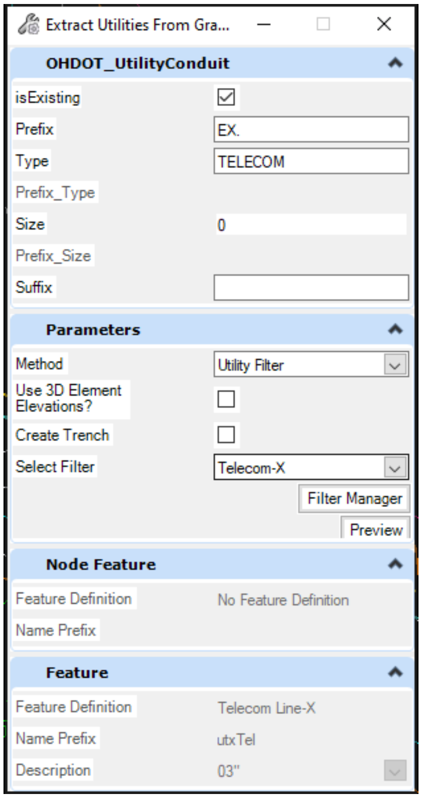


  - Issue a reset comment (right-click) to complete the selection process
  - When prompted, select the surface that will be used for the utility extraction. Reset (right-click) to use the active terrain model.



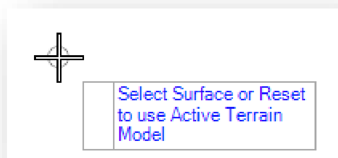
- Set the depth below the terrain to create the utility.
- Follow the prompts to accept the remaining options to complete the extraction process.

Tip 1: When extracting from a surface, keep the utility line graphics within the limits of the surface. If the plan view graphics for the utility line extend beyond the surface boundary the software will assume an elevation of zero for that portion of the utility line.



In the **Filter Method**,

- The Feature Definition, Feature Name (Name Prefix), and size (Description) will be grayed out since its being set within the filter.
- Select the desired filter (All Utilities), data point (left-click) to accept filter.
- When prompted, select the surface that will be used for the utility extraction. Reset (right-click) to use the active terrain model.



- Follow the prompts to accept the remaining options to complete the extraction process.

Tip 2: After extraction turn off display of elements used to extract utilities as the extracted utilities have a plan view display.

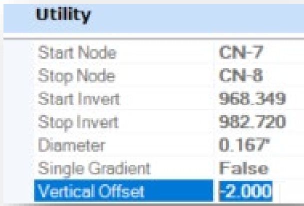
NOTE: You can always check over the extracted utilities and change pipe sizes and vertical offsets. Water lines and Gas lines will need to be checked to ensure proper pipe sizes and inverts. Sanitary lines are not included with the All-Utilities filter and should follow the same process as the following section, Extracting Drainage Features.

When should I use the Selection Mode versus the Filter method?

Both methods offer different benefits. In the selection mode you get to specify the desired depths for each extraction but for each feature a different extraction is required. Whereas the filter method can extract all utilities in one extraction, but the user does not get to specify depths but rather default depths are used.

Editing Utility Profiles

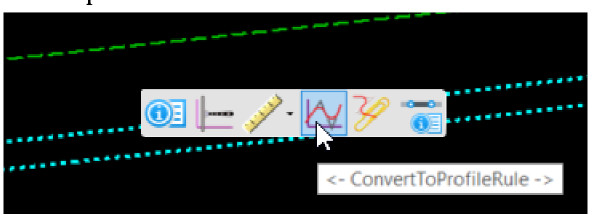
After the utility is extracted, it was a uniform depth under the surface. This depth can be changed in the utility properties.



But if more accurate data is known at certain locations of the utility the profile can be edited to account for the more accurate data by following the below steps.

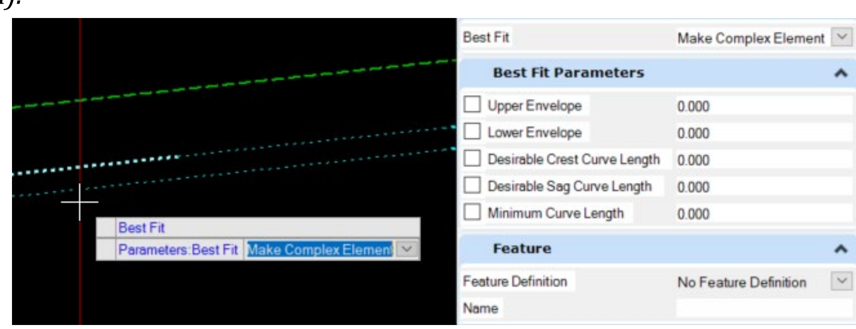
Caution should be taken with the direction the utility was extracted; it could be going the opposition direction of your centerline.

- Open the profile view of the utility.
- You could draw a new profile and make it active but if you want to utilize some of the current profile then follow the rest of the steps below
- Select the profile and convert to Profile Rule

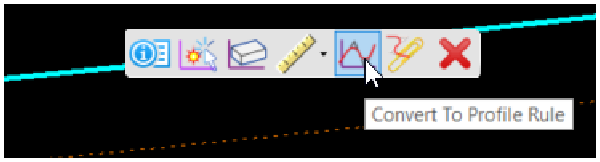


- Now profile the profile again and select create best fit profile

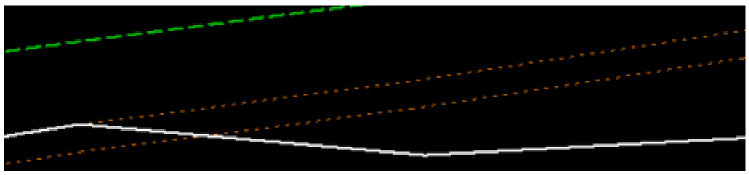
- Click through the prompts taking the defaults for the best fit profile *(**Note**: a feature definition does not matter as later when it becomes the active profile it will automatically get set to the utility feature definition).*



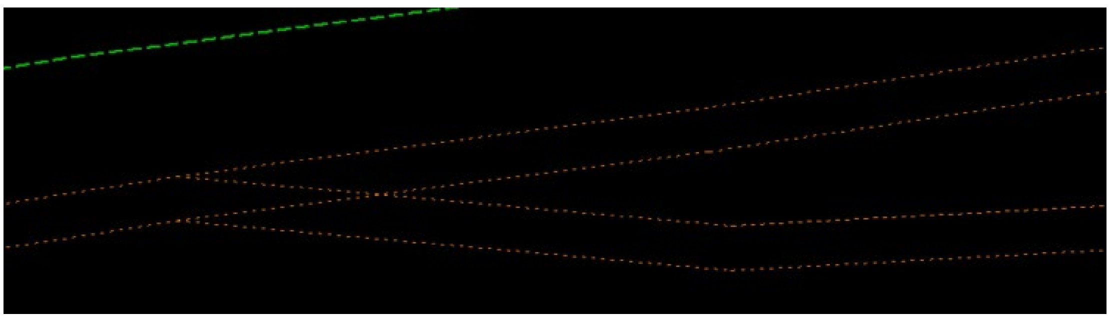
- Now select the newly created profile and convert to profile rule



- The newly created profile is now editable and can be modified as needed. While moving/inserting vertexes civil Accudraw can be used with profile offset ordinate to specify a depth below existing ground.



- Once modifications are complete make the new profile the active profile. You will notice the profile will automatically get assigned the utility feature definition.

Extracting Drainage Features

Extracting drainage features is a little more involved then extracting Utilities. There are so many variations of pipe sizes and materials that the Extract from graphics tool is not feasible to extract pipes. You can however use the Extract from graphics tool to extract the nodes. Nodes are required at each end of a conduit. Manholes, catch basins, Inlets are considered Nodes.

Typically, the existing drainage manholes, catch basins and inlets will be displayed in the survey fieldbook file as 2d cells. These cells can be used with the extract from graphics tools to create your drainage model.

During extraction the node (manhole, catch basin, inlet) invert elevation can be extracted at the same time. This is only possible is the survey fieldbook collected an additional point to represent the invert. This point will have a FLINE cell. If this point is not present, then a default invert for the node is used and will have to be set correctly after extraction.

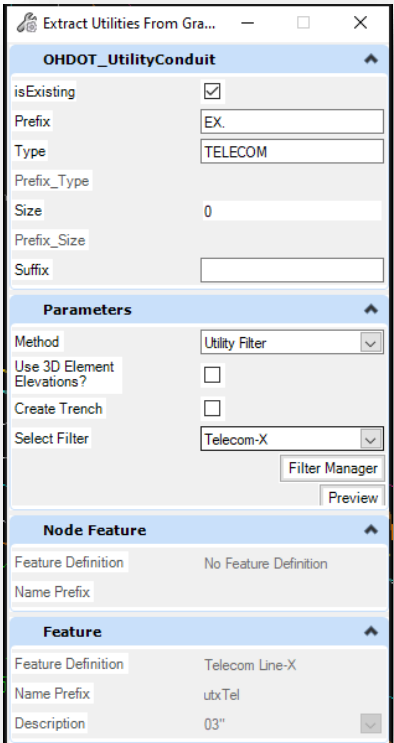
Notes:

- Drainage models should be extracted within a drainage basemap (BD).
- Use the 2D seed file to create the drainage basemap. The **Subsurface Utilities** software will generate a separate 3D model in the drainage basemap design file for the 3D drainage information
- The file is created in the **400-Engineering\Drainage\Basemaps** folder

Extracting 3D Drainage Nodes

The following actions can be used to create a 3D network of pipes and drainage structures.

Take the following steps to extract the Drainage Nodes.



1. Select the **Extract from Graphics** tool.
2. Set the **Method** to **Utility Filter**. Several filters are provided in the OHDOT CADD Standards for individual drainage items as well as groups of items.

3. Use the **Select Filter** item to choose **All Catch Basins** filter. This filter is configured to extract only the drainage cells for the catch basins and not the pipes. The pipes will be extracted individually as a separate process.

4. Toggle on the **Use 3D Element Elevations** option.

5. Accept the options as you are prompted on the cursor to initiate the extraction process.

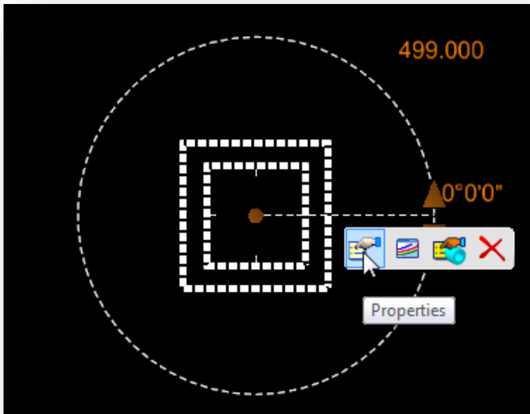
*Note: The **Design Stage** item is not relevant to this process.*

The 3D drainage structures are extracted using the elevation of the cells mapped by the Survey process. As each drainage structure is processed, the extraction process will search for a **FLINE** feature to establish the depth of the drainage structure. If a **FLINE** feature is not found, the depth is set using at a default value.

If the drainage basemap was created with a 2D seed file, which is the recommended best practice, **the Extract Utilities from Graphics** process will create a new model named **Design-3D** containing the 3D drainage structures. This new model is automatically attached as a reference to the current 2D model.

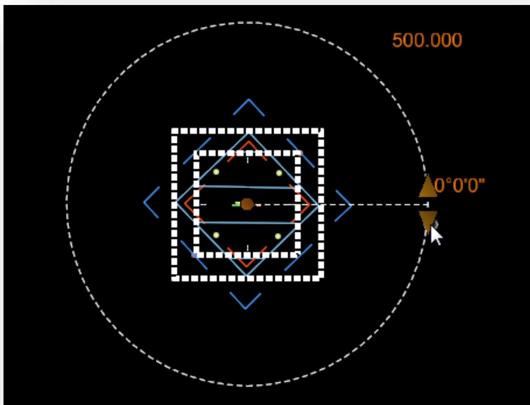
In addition to the 3D model, a 2D graphic is placed in the current 2D model for each extracted drainage structure nodes. This graphic is linked to the 3D model and can be used to make edits to the drainage structure.

In the example below, the reference file display of the survey basemap and the Design-3D model has been turned off to show the 2D graphic that is placed for one of the drainage structures.



Selecting the graphic reveals a pop-up menu that can be used to make edits to the drainage structure as well as two rules (**Node Top Elevation**, and **Node Rotation**) that can be edited by selecting the displayed text.

The extracted drainage structures do not consider the rotation angle of the original cells and are placed at a zero-degree rotation. The example at left below shows the original cell as referenced from the Survey basemap, and the 2D cell that was placed as part of the utility extraction process. It is necessary to rotate each structure manually.

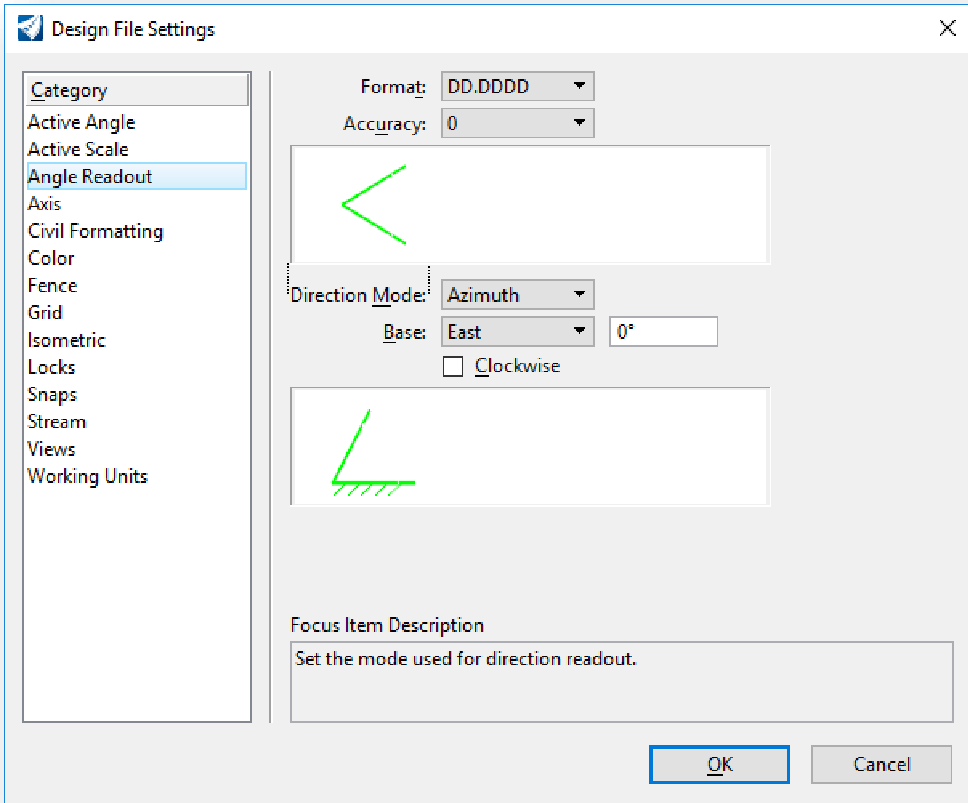


Once the 2D node is selected, parameters for the rotation of the 3D structure are displayed as shown at left. The angle can be keyed-in by selecting the angle text, or the 3D structure can be rotated by selecting one of the rotation arrows.

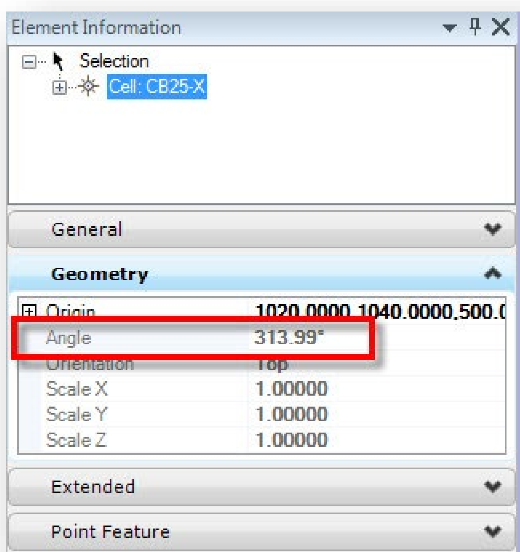
Typically, it is best to get the rotation angle of the original cell and use that value to key-in the rotation angle for the drainage node as described on the following page.

Take the following steps to key-in the rotation angle for the drainage structure:

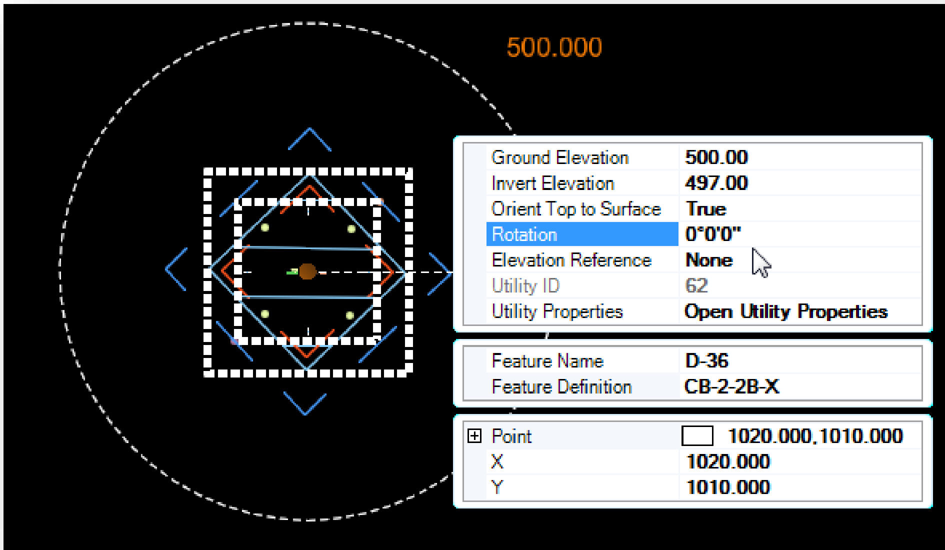
- ✓ Change the **Angle Readout** as shown below by selecting **File > Settings > File > Design File Settings**



✓ Use the MicroStation **Element Information** command to get the rotation angle of the original cell as shown below.

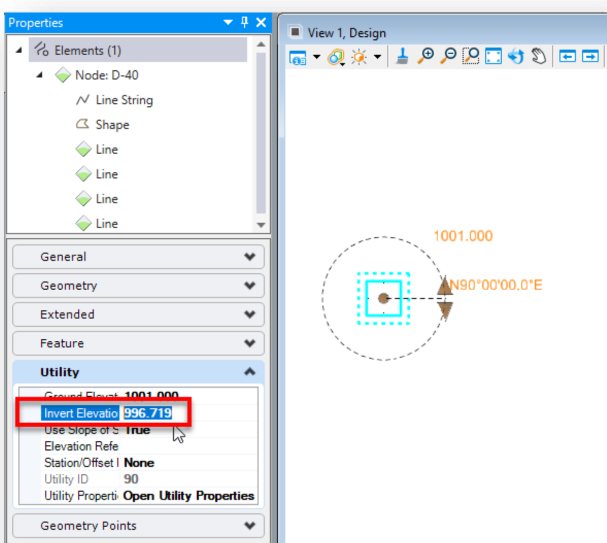


✓ Select the drainage node graphic and choose the **Properties** command from the pop-up menu. The **Rotation** can be modified as shown below. The graphics for the node in the 3D model are updated as changes are made to the properties.



After extracting the drainage structures and rotating them into place we are ready to extract the pipes.

After extracting the graphics, If the FLINE cell was not found or does not exist it is necessary to select each structure and manually key-in the **Invert Elevation** value. An example is shown below.

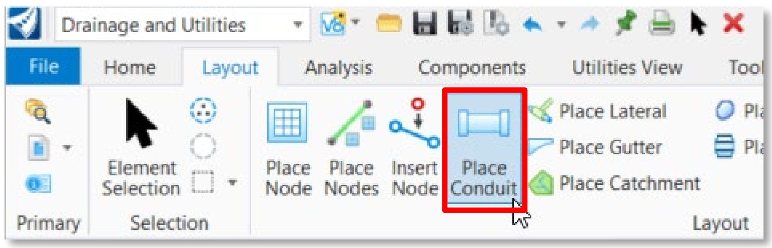


Note: The Filters are set up to search within a 12foot buffer of the cell. There is a chance that the process could not find the FLINE cell or find and use the wrong FLINE cell. Each structure should be verified the correct invert elevation is set.

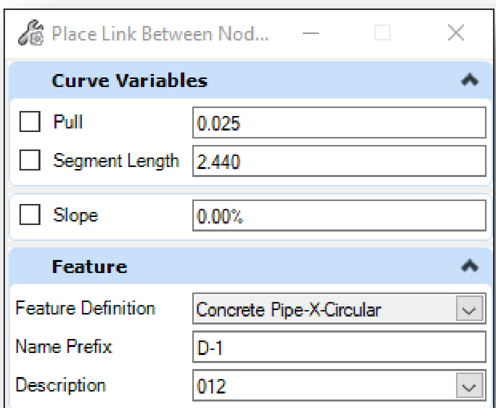
Repeat the process using the All Inlets and All Manholes filter groups.

Extracting Drainage Links

The **Extract Utilities from Graphics** command could be configured to extract pipe information; however, as of this writing the current release of the software will not connect the extracted pipes to the previously extracted nodes. Even though it is more cumbersome, it is better to place each pipe manually as a “link” using the **Place Conduit** command, shown below.



The dialog shown below is opened.



General information for the command can be found by selecting **Drainage and Utilities > Help > Help Contents** from the MicroStation ribbon menu.

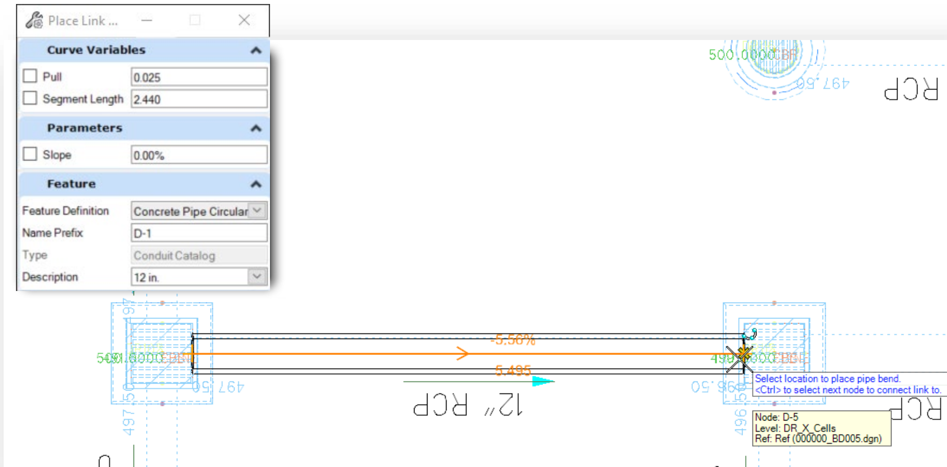
The **Feature Definition** is used to select the pipe type.

The **Description** item is used to select the pipe size.

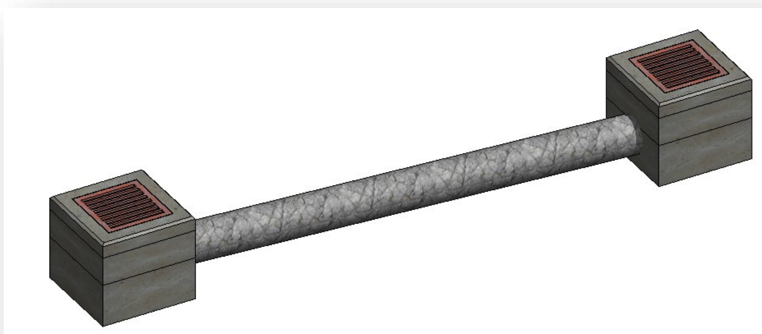
The **Name Prefix** is used to name each pipe and is incremented as you extract the pipes.

When the command is selected, you are prompted to **Select Start Node**.

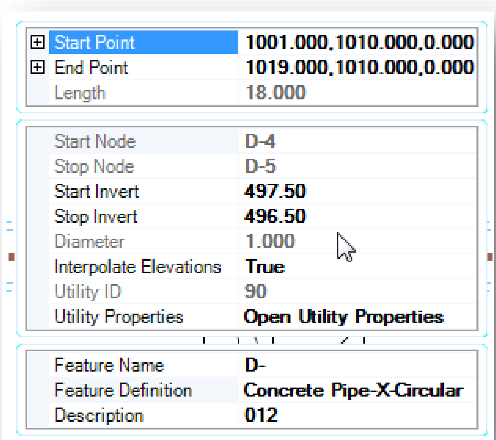
The ODOT 3D drainage items have specific connection regions on each side of rectangular structures. Circular structures have a circular connection region. When the first node is selected, the link starts from one of the connection regions. Follow the prompts to select the second node.



A line is placed in the current 3D model that can be used to modify the link. The extruded 3D link is placed in the 3D model as shown below.



The link is placed at the lowest elevation of the selected nodes. The elevation values can be edited by selecting the link in either the 2D or the 3D model, and then choosing the **Properties** icon from the pop-up menu.



From the *Properties* menu, shown at left, the **Start Invert** and **Stop Invert** values can be edited if necessary to set the correct elevations for the pipe.

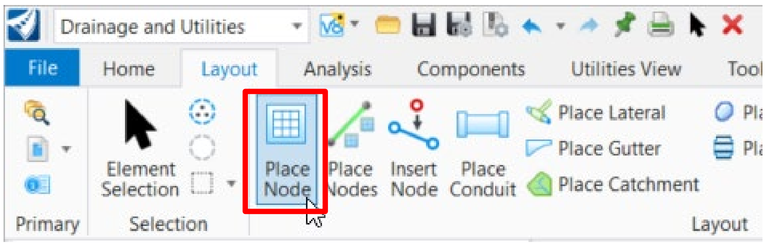
Note: *Sanitary manholes and conduits should be extracted following the same methods outlined in this section.*

Connecting Links to FDO or PIO Points

The field codes PIO (pipe inlet or outlet) and FDO (for direction only) can both be used to designate the end of a pipe. PIO is used to designate an inlet or outlet point for an open pipe of culvert. FDO (for direction only), is used to designate the direction of a pipe when the other end cannot be located.

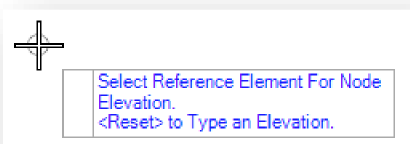
There must be a node defined at the PIO and FDO locations to place a link in the design file using the PIO and FDO points. The **Place Node** command is used to place nodes at the PIO and FDO locations.

The command is selected from the **Drainage and Utilities** WorkFlow as shown below.



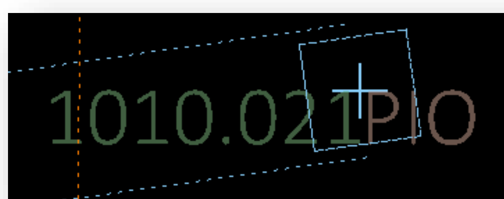
Take the following steps to create nodes for the PIO and FDO points:

- Select the **Place Node** command
- Set the **Feature Definition** to the **GenericNode-X** item
- When prompted as shown below, issue a reset (right-click) to type in the elevation for the node



- Type in the **Elevation** for the node
- Enter a data point for the node location by snapping to the PIO or FDO point
- A triangle is placed in the 2D and the 3D models at the location of the node. The direction of the triangle is irrelevant

The extracted node for the PIO and FDO points is placed as a triangle at each point on the level SC_Scratch1 as shown at right.



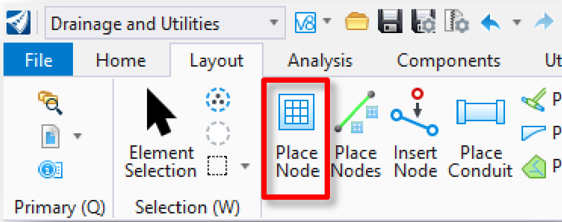
The **Place Conduit** command described in the previous section is used to place the 3D pipes between the nodes.

Tip: There is also a PIO filter that can be used with the **Extract from Graphics** tool instead.

Box Culverts

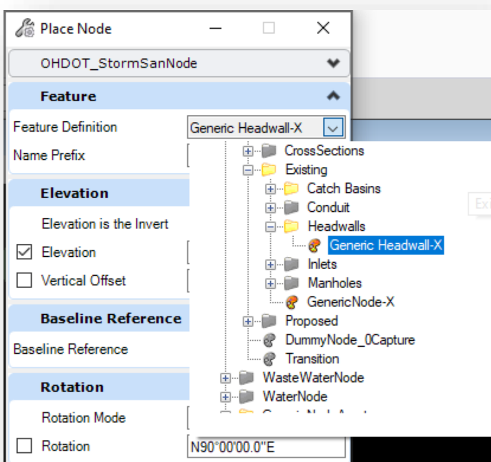
The graphics for existing storm water utilities are normally placed in a 2D model in the drainage (BD) basemap design file. The **Drainage and Utilities** software is used to place the box culvert in a 2D model. The software will automatically generate the 3D model for the box culvert.

After creating a drainage basemap design file, attach the survey basemap as a reference file to view the culvert shots. Box culverts are surveyed using the **CULV** field code to take a shot at the invert elevation at each end of the culvert.



Box culverts are placed as a link by the **Drainage and Utilities** software by use of the **Connect Conduit Between Nodes** command. A node must first be placed at each end of the proposed box culvert in the design file by use of the **Place Node** command. The generic headwall feature definition is used to define the location of each end of the box culvert.

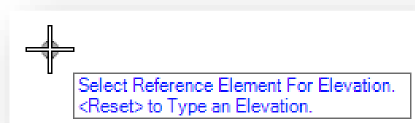
Select the **Place Node** command from the task list as shown below.



Place a node at each end of the box culvert using the **Generic Headwall-X** feature definition as shown above.

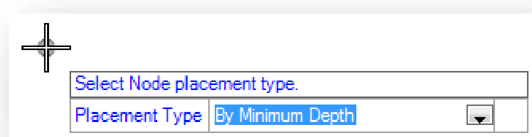
The user is prompted to select a reference element for the node elevation as shown below.

Nodes can be placed at a specific elevation, or relative to the selected surface or a corridor model. Issue a **Reset** to type an elevation.



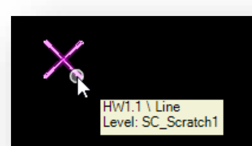
After the elevation reference is defined, the user is prompted to set the **Placement Type**.

The **By Minimum Depth** option is used to set the initial depth using a default value. The invert elevation can be edited after placement if necessary.

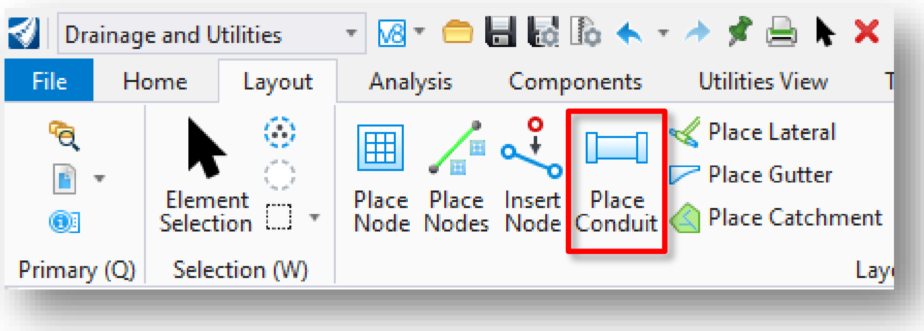


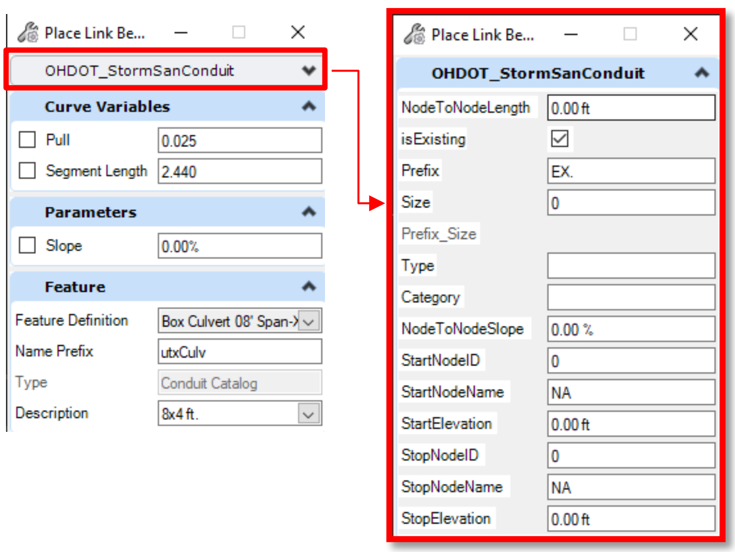
Next the user is prompted to set the location of the node. Place the node at the location of the CULV survey point.

The generic headwall node is placed using a X symbol on the level SC_Scratch1 as shown at right. The direction of the triangle, defined by the **Rotation** parameter, is irrelevant to the flow direction of any links connected to the node.



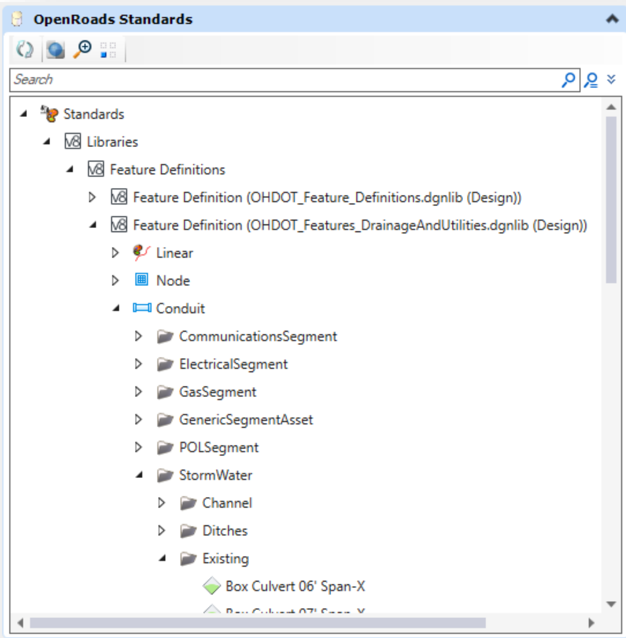
After the headwall points have been placed at each end of the culvert, the box culvert is placed using the **Place Conduit** command shown below.



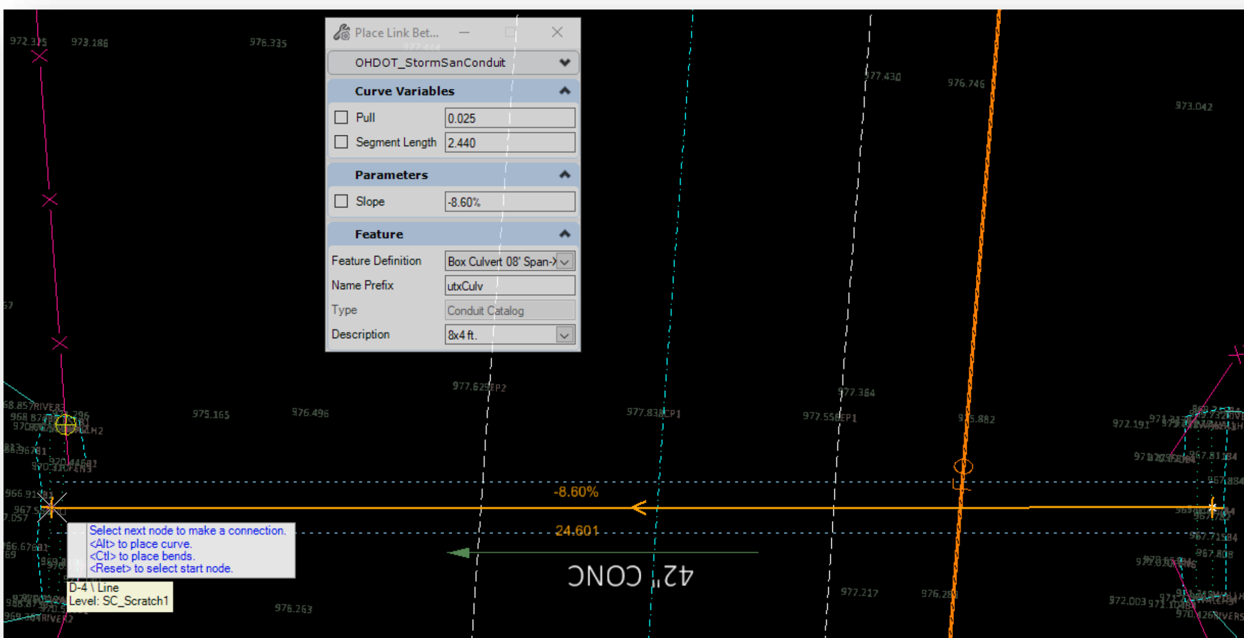


Feature definitions for existing and proposed box culverts are provided in the ODOT standards. The available features for existing box culverts are shown on the following page. The span and rise for each available culvert feature is defined by the **Description** field as shown at right above.

Available culvert sizes are based on the **Figure 1008-14** of the **ODOT Location and Design Manual – Volume 2, Drainage Design**.

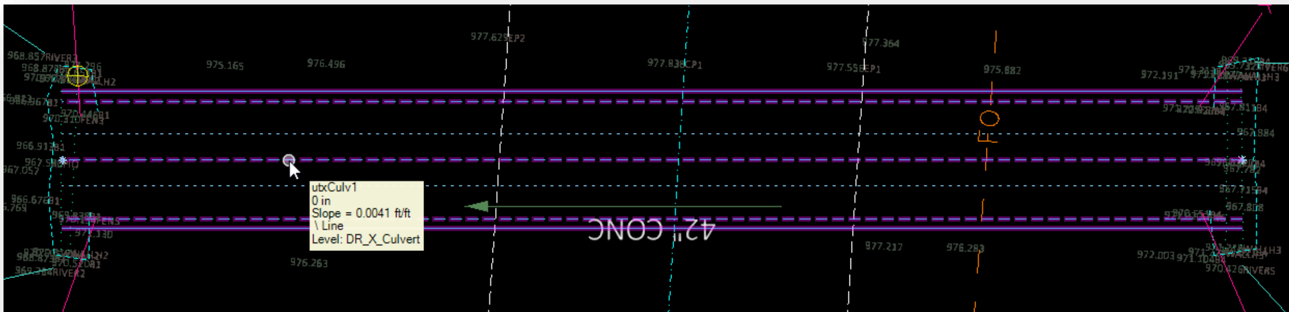


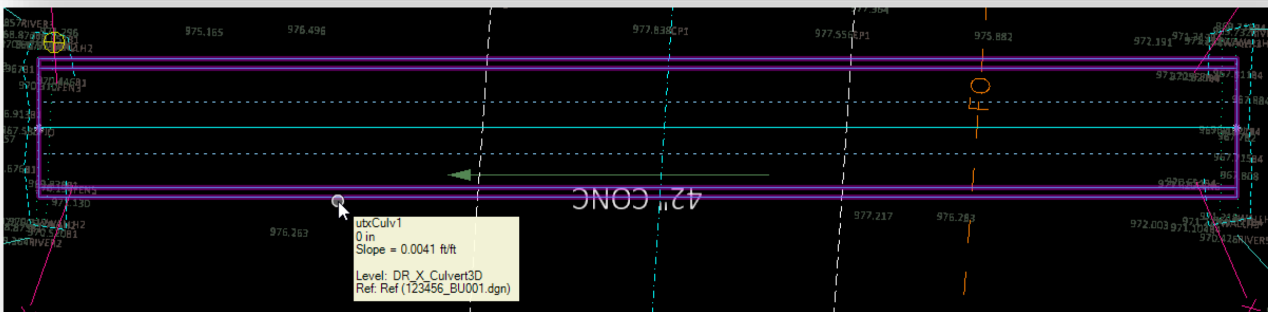
The culvert is placed by selecting each node as shown below.



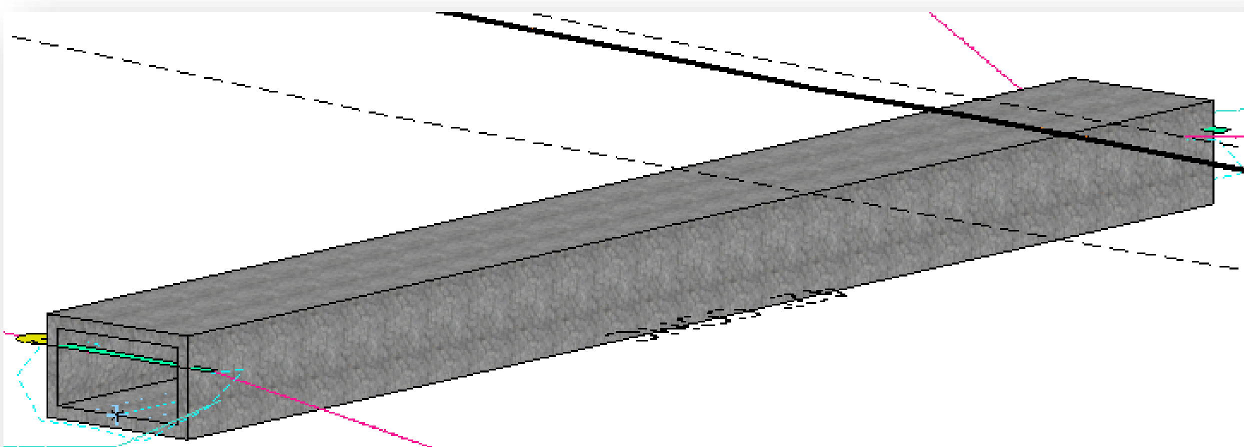
Two sets of graphics are drawn for the box culvert as shown below.

The culvert is drawn on 2 different levels DR_X_Culvert and DR_X_Culvert3D these will be shown on custom line styles in the 2D design model. The 2D design model can be referenced back to the survey basemap (BE) if desired or attached as a reference by the design personnel.





The Subsurface Utility software also creates a 3D model in the drainage basemap named "Design-3D" containing the 3D model of the box culvert as shown below.



Modeling Proposed Drainage Features

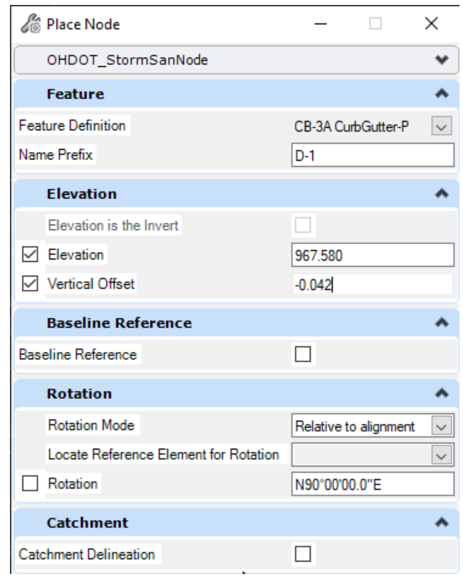
Creating a proposed drainage model is somewhat like modeling existing drainage features but the extract from graphics tool is not relevant.

Modeling the drainage features should be done within a drainage basemap (BD) file. If the proposed network will attach to the existing modeled network, then the proposed network MUST be created in the same file as the modeled existing network.

To place conduit, the ends of that conduit must have nodes. So, the first step is to place the nodes.

Placing Nodes

The **Place Node** and **Place Nodes** tools can be used to place Drainage nodes like Manholes, Catch Basins, Inlets, and headwalls.



The **Place Node** tool offers a couple different ways to place nodes. The workflow is listed below

- Set the desired feature definition and Name Prefix.
  Tip*: The feature name (name prefix) can be used to annotate call outs.*
- In the heads-up prompt you can select an element to be used for the grate/rim elevation or reset click (right click) to key in the elevation.
  Tip*: If placing curb/curb & gutter catch basins, use the CURB_GUTTER element as the elevation reference.*
- Next if a reference element as selected you can enter a vertical offset from that reference element. If a reference element was not selected, you can enter the desired elevation.

Tip*: When entering a vertical offset value don't forget the negative sign to offset below the selected element. Also, basic calculations are supported for example to vertically offset 0.5" below the reference element you can enter -0.5/12.*

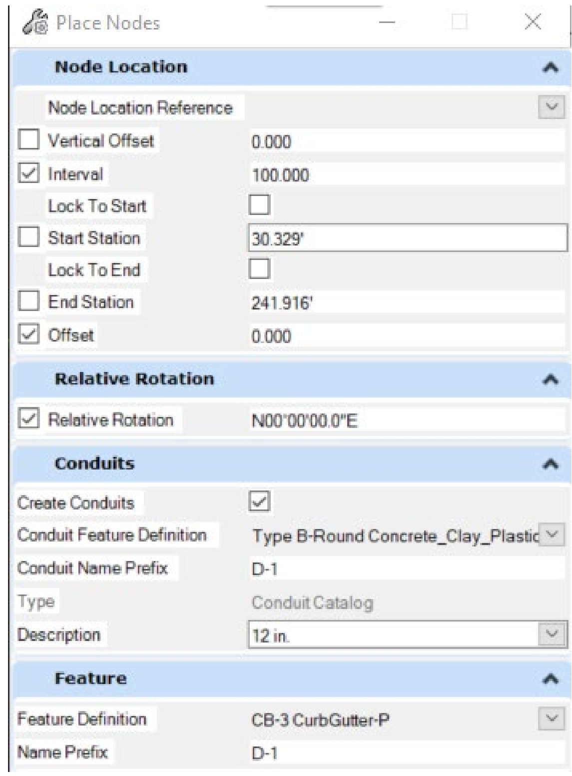
Note*: Don't forget about catch basin depressions when placing catch basins for curb/curb & gutter.*

- Next enter a data point (left click) at the location you want the node to be placed.

Tip*: Use C****ivil AccuDraw*** *station-offset ordinate to place the node at a desired station and offset.*

- Next you can specify the rotation of the node. There are two options, absolute or relative to alignment. Data point (left click) to accept the selected rotation mode.
- If relative to alignment is selected as the rotation mode the user is prompted to select the alignment to base the rotation off, then can enter the desired rotation and data point (left click) to complete the placement of the node.

The **Place Nodes** tool can be used to place a series of node at a given interval as well as place conduits between the nodes at the same time. The workflow is listed below.



- First fill out the tools dialog as desired
    - Check on Create conduits and set the feature definition, name prefix, and description (size) to desired values. Note only one feature and size can be set and will be used for all conduit placed. If conduits are not desired, then uncheck the create conduits checkbox. The name prefix will increment the number for each conduit placed.
    - Set the feature definition and name prefix to the desired feature for the node. The name prefix will increment the number for each node placed.
- The first prompt is to select the element to use as a reference for the locations of the nodes. This will typically be your centerline. Select the desired element.
- Next in the heads-up prompt you can select an element to be used for the grate/rim elevation or reset click (right click) to key in the elevation. If keying in an elevation, the same elevation will be used for all notes being placed whereas using an element for the reference elevation will use the elevation of that reference element at the given location of the nodes.

    ***Tip**: If placing curb/curb & gutter catch basins, use the CURB_GUTTER element as the elevation reference.*

- Next if a reference element was selected for node elevation you can enter a vertical offset from that reference element. This is not shown in the heads-up prompt and needs to be entered into the tools dialog. If a reference element was not selected, you can enter the desired elevation.

    ***Tip**: When entering a vertical offset value don't forget the negative sign to offset below the selected element. Also, basic calculations are supported for example to vertically offset 0.5" below the reference element you can enter -0.5/12.*

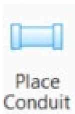
    ***Note**: Don't forget about catch basin depressions when placing catch basins for curb/curb & gutter.*

- Next specify the desired interval to place the nodes. And issue a Data Point (left click) to accept value
- Next enter in the start station, which is the station location of the first node, the station value is based off the first element reference you selected with this tool, which is the reference element to be used for node location. Issue a Data point to accept the value.
- Next enter the end station, which is the station location of the last node, the station value is based off the first element reference you selected with this tool, which is the reference element to be used for node location. Issue a Data point to accept the value.

    ***Note**: If the end location ends at the exact station of an interval the value might need to be bumped up slightly for the software to place a node at that location, the view window will show the nodes as a preview which can be used to see where the nodes will be placed.*

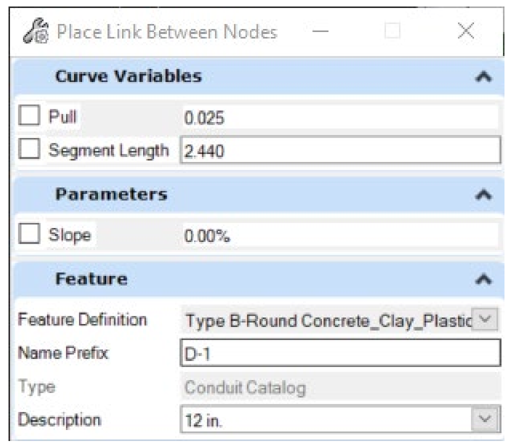
- Next enter in the desired offset. This is the offset from the selected element used for the node location. Issue a data point to accept the value.
- Next specify the desired relative rotation of the node. This is relative to the selected element used for the node location. Issue a data point to accept the value.
- Issue a final data point to finish placing the nodes and conduits.

Placing Conduit



To place conduit a node must exist at each and of the conduit. Think of a conduit as a link between two nodes. A link can't exist between two nodes if there are not two nodes.

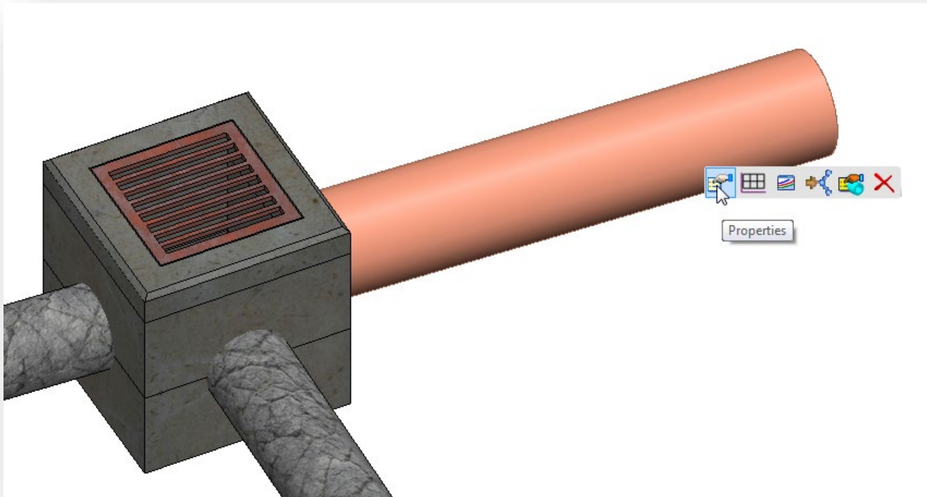
The workflow for placing conduit is listed below.



- Set the desired Feature Definition and Description (size)
- Select the start node where the start of the conduit will connect to the node.
    - There are connection regions defined on the nodes to specify the locations the conduit can connect to. In general, the connection regions are to the inside boundary of the node. Square nodes have connection regions at the center of each side while circular nodes have the connection region all around the inside edge.
- Select the end node where the end of the conduit will connect to the node
    - In rare situation a curve or bend can be introduced while placing the pipe by holding down the alt key or ctrl key while selecting the end node.

Reviewing and Editing Nodes and Links

After the nodes and links have been created, it is good practice to review the values for accuracy. The parameters of the 3D drainage information can be edited by using the MicroStation **Element Selection** tool to select a node or link, and then choosing the **Properties** icon from the pop-up menu as shown below.



The parameters are shown below. Note that the invert values for a link cannot be edited if the link was extracted from graphics.

Conduit Properties

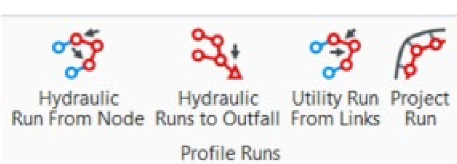
Start Point	1030.000,1040.000,0.000
End Point	1030.000,1030.000,0.000
Length	10.000
Start Node	D-21
Stop Node	D-20
Start Invert	499.75
Stop Invert	500.00
Diameter	1.000
Interpolate Elevations	True
Utility ID	145
Utility Properties	Open Utility Properties
Feature Name	D-1
Feature Definition	Concrete Pipe-X-Circular
Description	012

Node Properties

Origin	1020.0000,1010.0000,497
Angle	N90°0'0"E
Orientation	Top
Scale X	1.00000
Scale Y	1.00000
Scale Z	1.00000
Ground Elevation	499.00
Invert Elevation	496.00
Orient Top to Surface	True
Rotation	0°0'0"
Elevation Reference	None
Utility ID	30
Utility Properties	Open Utility Properties
Feature Name	D-4
Feature Definition	CB-2-2B-X
Point	1020.000,1010.000
X	1020.000
Y	1010.000

Currently the Node invert elevation will automatically change to match the lowest connected conduit invert elevation. Until this behavior is addressed by the software it is best practice to set all pipe begin and end invert elevations then set the node invert.

Creating Profile Runs



There are several tools available to help with creating profile runs. Profile runs can be created to view profiles of Drainage and Utility networks. Profile Runs can be as short as one pipe or "run" the whole drainage network.

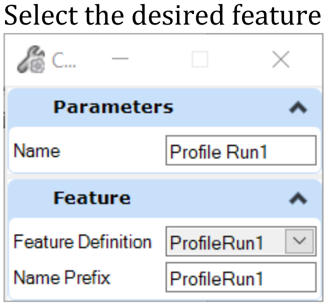
Setting	Description
Hydraulic Run From Node	Creates a profile run from a selected node to the outfall, or between two selected nodes. This tool only works on hydraulic (i.e. storm or sanitary) networks.
Hydraulic Runs to Outfall	Creates a profile run for every path, from the most upstream nodes to the outfall. These paths are sometimes known as trunks or branches. This tool only works on hydraulic (i.e. storm or sanitary) systems.
Utility Run From Links	Creates a profile run from selected links, for any type of utility (e.g. storm, communications, electric, etc.) The links must be consecutive, without gaps between them.
Project Run	Projects a profile run created using the tools above onto a linear element, which could be a road centerline, or another profile run.

The **Hydraulic Run From Node** and **Hydraulic Runs to Outfall** tools are great for a more automated profile run creation process and the **Utility Run From Links** tool is a more manual process. This Guide will cover the process for using **Utility Run From Links** tool.

Profile Runs must be created in the model containing the Drainage or Utility Network as well as the profile run can only include connected conduits (as in no branches or jumps in the network).

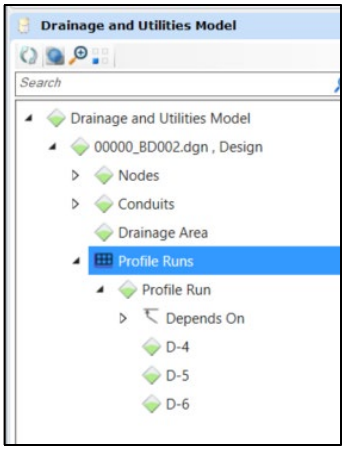
Follow the process below to create a Profile run using the **Utility Run From Links** Tool.

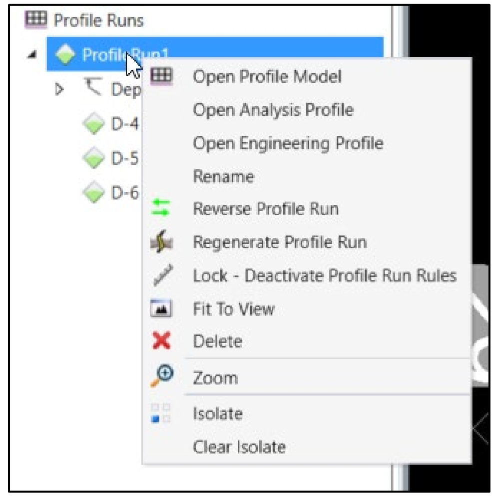
- Select the desired feature definition within the toolbox (there are ProfileRun1, ProfileRun2, ProfileRun3, and ProfileRun4 feature definitions created for this purpose.



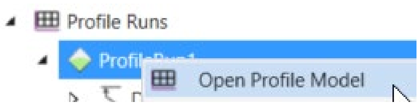
- Graphically select the first conduit desired to be included within the profile.
- Continue selecting conduits to be included within the profile.
- When done selecting conduit reset click (right click) to accept selection.

Profile runs can be found from within the Drainage and Utilities Model Tab of Explorer in the profile Runs section.

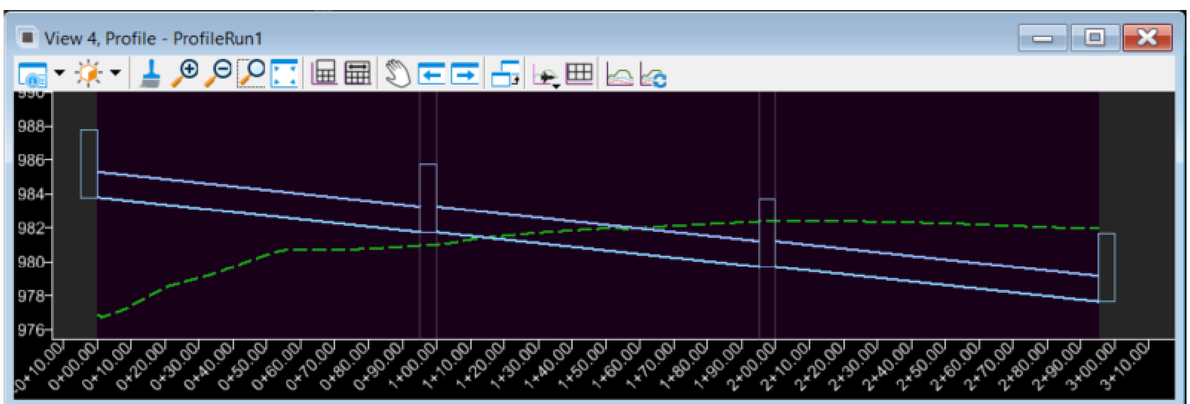




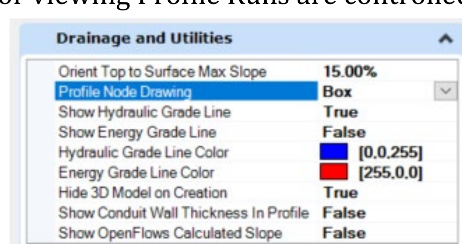
There are several options in the right click menu of a profile run, as shown above. The profile run will also have linear element in the 2d plan view to represent the profile run. If using the ProfileRun1-4 feature definitions, then the 3d plan view graphic will be on the SC_ProfileRun1-4 level(s). note that the priority on these levels is set to be under the DR_P_Storm_Sewer level so it might be covered up by that level.



Use the Open profile model tool to open the Profile Run's Profile. After selecting this tool simply click inside the view window that you want the profile view to be displayed in.



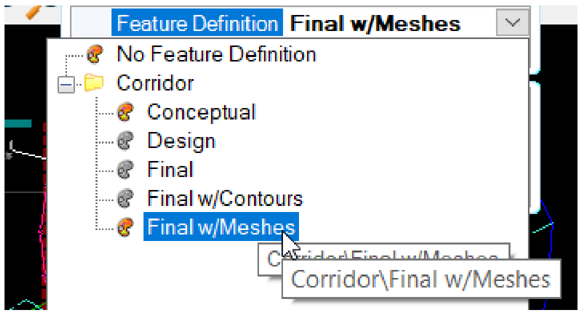
Some of the display settings for viewing Profile Runs are controlled via User Preferences > View Options – Civil.



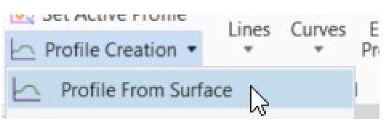
Showing Proposed Ground on Profile Runs

In the profile view of a profile run the active terrain will automatically be displayed along with the drainage network itself. But the proposed ground is not shown. Take the following steps to add proposed ground to the profile run.

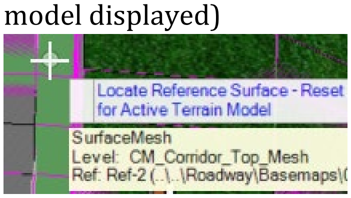
A Profile run typically will be within the extents of a corridor. The Corridor top mesh can be used to project its top elevation online the profile run. The corridor has design stages set up so that the top mesh is display when the design stage is set to Final w/Meshes. If you do not see the CM_Corridor_Top_Mesh level in use in the 3d Design model of your corridor file then you need to open that file, select the corridor graphic and change the corridor feature definition to Final w/Meshes as imaged below.



You can use the **Profile from Surface** tool to generate a dynamic profile from a surface.

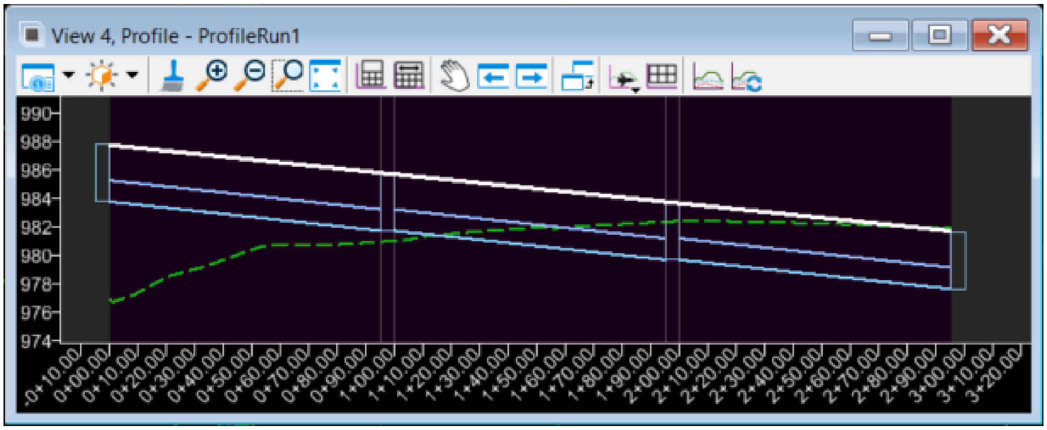


- When using this tool set the feature definition to Linear/Profiles/PF_P_Grade_Line.
- Select the profile run (not the conduit feature, but the actual profile run) that you want to create the proposed ground profile for.
- You can select multiple profile runs if desired.
- Reset click (right click) to accept selection and move on to the next step.
- Next select the surface to create the profile from. (You can select this from the view with the 3D model displayed)



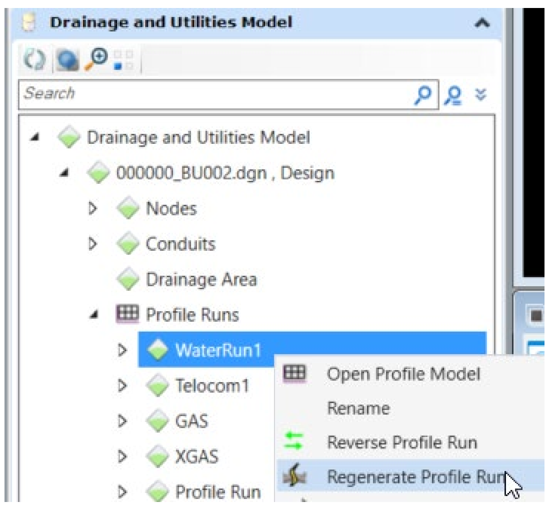
- Reset click to end surface selection after you have selected the desired surface
- Alt to lock to start and click to accept
- Alt to lock to end and click to accept
- Set point selection to all and click to accept
- Set profile adjustment to none and chick to accept
- Set Draping options to Triangles and click to accept
- Set Horizontal offsets to 0 and click to accept
- Set vertical offsets to 0 and click to accept

The profile is rules to the surface so if the surface changes this profile will automatically update with the changes.



Deleting projected profile runs

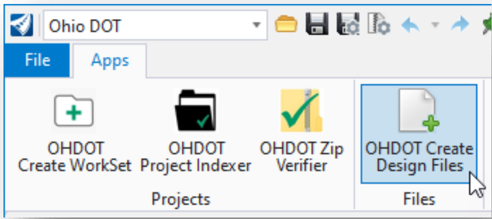
If a profile run was projected onto a centerline on accident or no longer is needed, you can open the project view and delete the conduit and profile run graphic, but you will notice that the nodes can not be deleted. This is a bug. The work around is to regenerate the profile run. This can be done from Explorer as shown in the image below.



Create Profile Run Sheets

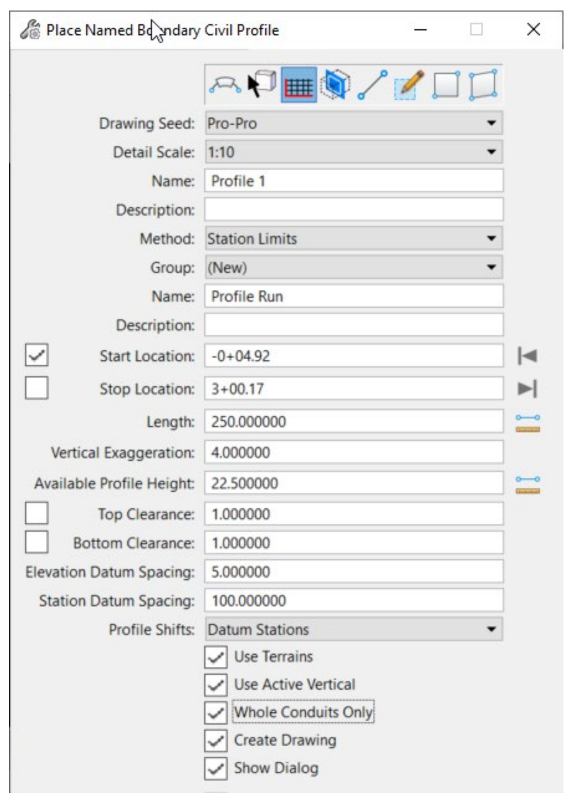
When putting together a plan set containing drainage there are a couple of options for how to display the drainage information. See L&D Vol. 3 section 1310.2 for details. One of the options is to show drainage information on a storm sewer profile sheet. This section will outline the steps to create these sheets.

Storm Sewer profile sheets should be created within the Drainage Profile (DF) file. Use the Create Design files application to create a Drainage Profile sheet file. Note that Named boundaries will be placed to create these sheets and as outlined in chapter 800, the named boundaries can be placed in the same file the sheets are generated in or a container file. For this example, we will put the named boundaries in the same file the sheets are generated in.

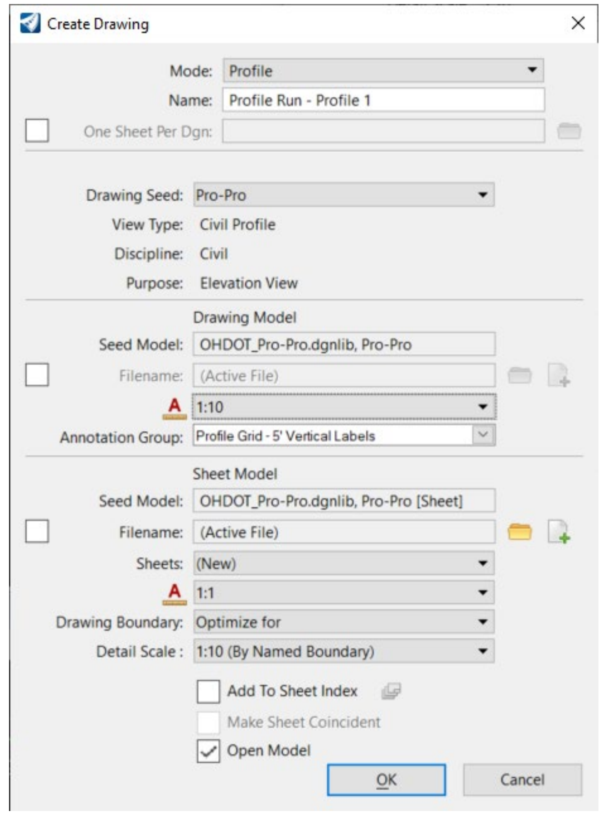


- Create a Drainage Profile sheet, don't forget to change the seed to 123456_DesignSeed2d.dgn

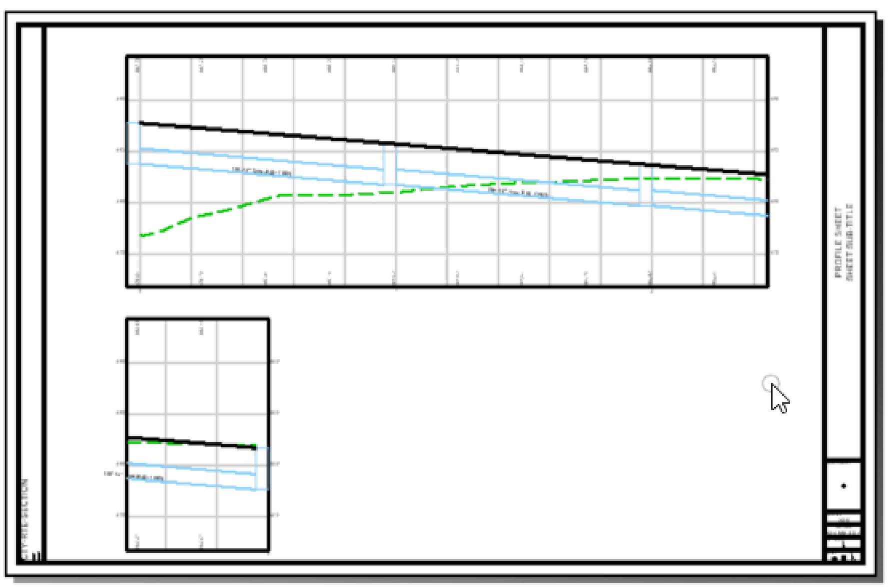
- After creating the file, add all desired references. This is typically the Fieldbook, alignment, corridors, and Drainage basemap files.
- Open the profile view of the Profile Run you would like to create a Storm Sewer Profile Sheet of.
- Select the **Place Named Boundary** tool and set the mode to profile, and the Drawing seed to either Pro-Pro or Profile Only. Pro-Pro seed will stack two named boundaries on top of each other in one sheet whereas Profile Only will take up the whole sheet for one named boundary.
- Set the scale to the desired value.
- Data point inside the Profile view to select the view to place the named boundaries.
- Data point to set the start location for the first named boundary.
- Data point to set the end location for the last-named boundary.
- Check on the Create drawing check box.
  Tip: *there is a check box for Whole Conduits Only*
- Data point to accept and place the named boundary



- In the create Drawing dialog that opens make sure to set the drawing model scale to match the scale set in the place named boundary dialog.



This will create a drawing model for each named boundary and sheet models. Annotation is placed in the drawing model.



Tip: You can bulk create profile sheets for all profile runs by adding them to a selection set before starting the Place Named Boundary tool. There is a Graphical filter named Profile Runs which can be used with the **Select by Graphical Filter** Tool to help create the selection set.

Creating Profile Run sheets for divided highways

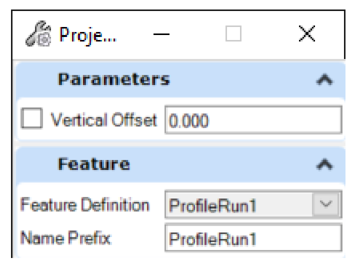
This Section is currently under development.

Setting up Drainage to show on Plan and Profile Sheets

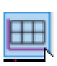
To have a drainage network to show up in the profile section of a centerline profile it first needs to be modeled using the Drainage and Utilities tools outlined in the Modeling Proposed Drainage Features section of this guide. After it is modeled a profile run can be created as outlined in the Creating Profile Runs section of this guide, and these profile runs can be projected onto the centerline profile. Crossing pipes can also be shown with the use of the 3d cut tool. This section will outline the steps to set this up.

Projecting profile runs

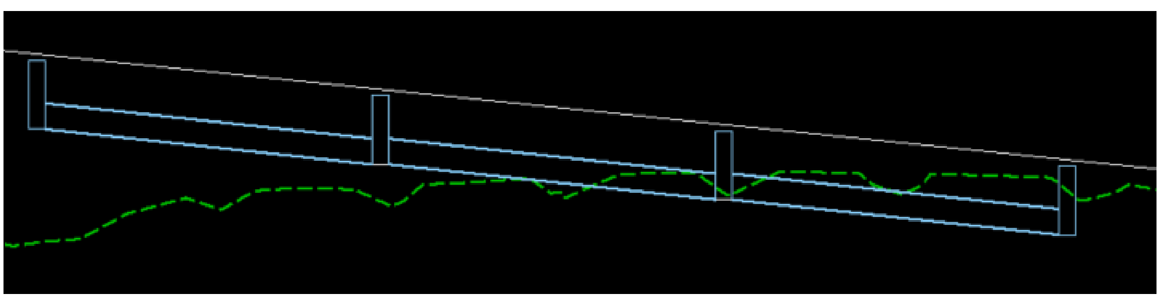
Profile runs can be projected onto other profiles with the **Project Run** tool. This projection can be done either from within the drainage basemap file or from a file that has referenced in the drainage basemap. Currently it is recommended to project profile runs onto alignments from within the drainage basemap. The following steps outline how to project profile runs.



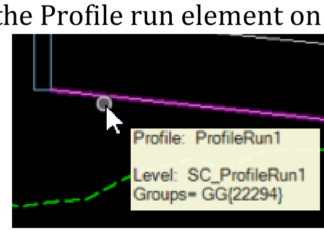
- Open the Drainage Basemap (BD) containing the drainage network you want to project to your center line alignment's profile
- Select the Project Run tool.
- Set the Feature Definition to one of the ProfileRun features (typically the same feature definition you created the profile runs with). Leaving the feature set to No Feature Definition will put a projection on the default level.
- Select the profile run you want to project
- Select the alignment you want to project the profile run onto.



- Open the profile view of the alignment to verify the profile run has been projected.



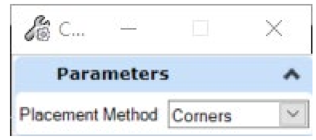
The profile run is projected perpendicular onto the alignment's profile. Notice if you turn off the DR_P_Storm_Sewer level you will see the Profile run element on the SC_ProfileRun1 level.



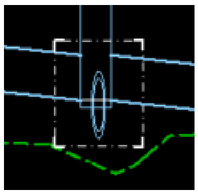
Showing Crossing Features

The previous section showed how to project a profile run onto an alignment but if there is a pipe crossing the road that is not shown. The create 3D cut tool can be used to show this crossing pipe.

- Select the create 3D cut button from within the Alignment' profile view.



- Data point to accept the placement method as corners
- Data point in the profile view at one corner to start the area of the 3d cut
- Next data point in the profile view at the opposite corner to create the 3d cut area.



This process can be repeated for any number of 3d cut areas needed. Note that these areas can be selected and changed in size. You must hit the refresh 3d Cut button to update the graphics within the 3d cut area anytime a change it made included level display. This is a static 3d cut so changes to the model would require a refresh to the 3d cut as well.

Setting up Utilities to show on Plan and Profile Sheets

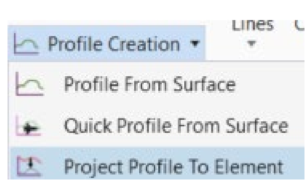
To have utilities show up in the profile section of a centerline profile it first needs to be modeled using the Drainage and Utilities tools outlined in the Extracting Utility Features section of this guide. After it is modeled its profile can be projected onto the centerline profile. Crossing utilities can also be shown with the use of the 3d cut tool. This section will outline the steps to set this up.

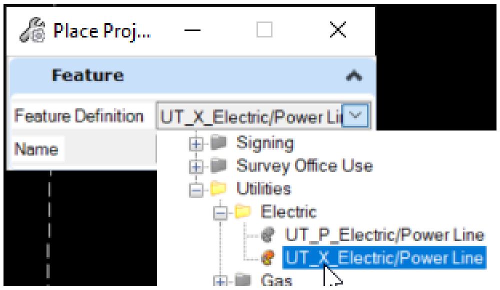
Projecting Profiles

The following steps show how to project utility profiles onto centerline profiles. ***This is a workaround workflow to using profile runs until some defects/enhancements are addressed with ORD. Utility text favorites for annotation will not be able to be used with this workflow! See alternative workflow for projecting profiles section for more info.***

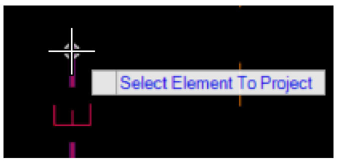
Currently to project a utility profile onto the centerline profile you must be within the file that contains the centerline element. This is typically a BK file.

1. Open the file containing the centerline alignment that the plan and profile sheet will be based upon.
2. Add a reference to the utility basemap.
3. Select the Project Profile to Element Tool
4. Select the appropriate feature definition



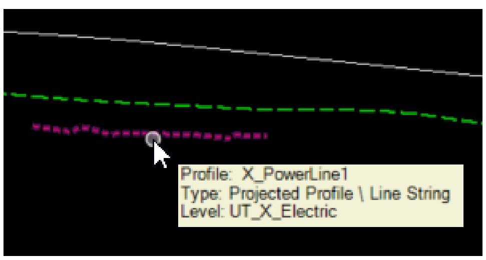


5. Select the Utility you want to project



6. Select the Centerline you want to project onto

7. Open Profile View of centerline to verify the utility profile was projected.



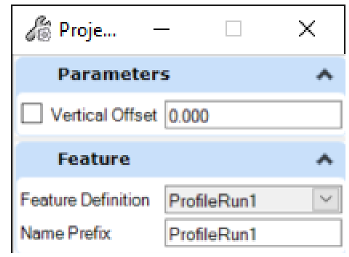
Note: *The Utility feature definitions are set up to project the top of the conduit (not the invert) whereas Sanitary and Drainage project the bottom (invert).*

Alternative workflow for projecting Utility Profiles

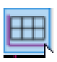
This workflow will project utilities onto centerline profile. This will allow for easier annotation of utilities with the use of text favorites, but the projection will show two lines (top and bottom of conduit). Typically for small diameter utilities we only show one line. There is currently a bug where the profile run graphic does not follow the profile of the pipe. When this is fixed this will be the preferred workflow.

To have a utility network show up in the profile section of a centerline profile it first needs to be modeled using the Drainage and Utilities tools outlined in the [Extracting Utility Features](#) section of this guide. After it is modeled a profile run can be created as outlined in the [Creating Profile Runs](#) section of this guide, and these profile runs can be projected onto the centerline profile. This section will outline the steps to set this up.

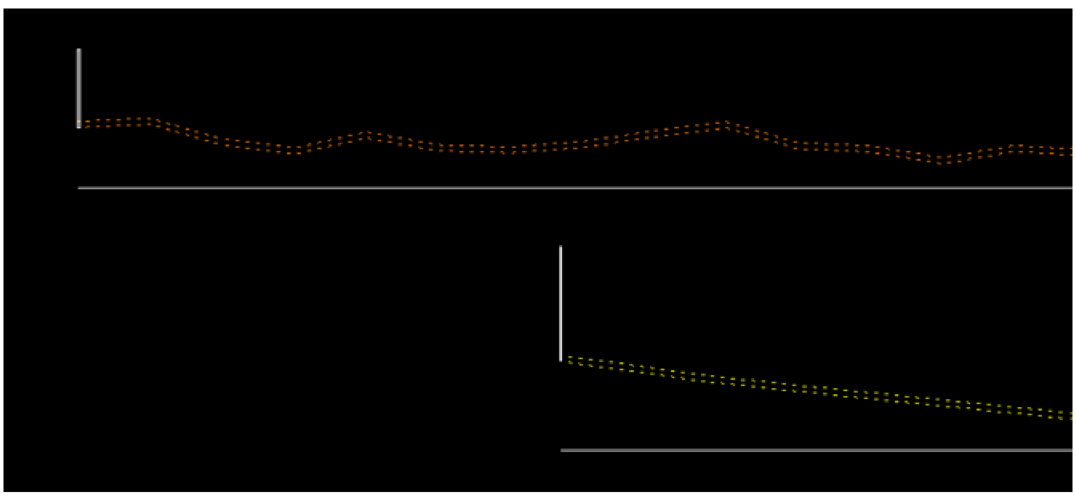
Profile runs can be projected onto other profiles with the **Project Run** tool. This projection can be done either from within the utility basemap (BU) file or from a file that has referenced in the utility basemap. Currently it is recommended to project profile runs onto alignments from within the utility basemap. The following steps outline how to project profile runs.



- Open the Utility Basemap (BU) containing the utility network you want to project to your center line alignment's profile
- Select the Project Run tool.
- Set the Feature Definition to one of the ProfileRun features (typically the same feature definition you created the profile runs with). Leaving the feature set to No Feature Definition will put a projection on the default level.
- Select the profile run you want to project
- Select the alignment you want to project the profile run onto.



- Open the profile view of the alignment to verify the profile run has been projected.



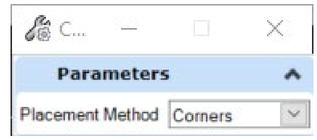
The profile run is projected perpendicular onto the alignment's profile. Since all conduits must connect to a node, you will notice nodes draw in, these are on the Default level and should be turned off. Also notice the profile run graphic and the bug that is does not follow the utility profile. This need turned off as well. This graphic will be on the SC_ProfileRun1 level is the ProfileRun1 feature definition was used. Also notice the two lines for the top and bottom of the conduit. In a future release of ORD when the profile run graphic is fixed to follow the conduit profile, the workflow will be to turn off the conduit and leave the profile run graphic on. The [Showing Drainage and Utilities on Plan and Profile](#) Section will show how to get these projected utilities to show up on profile sheets with the use of the attach all profile views button.

Showing Crossing Features

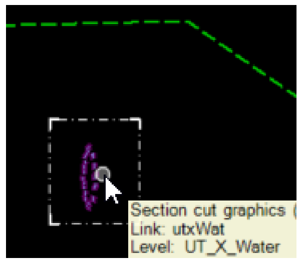
The previous section showed how to project a utility's profile onto an alignment but if there is a utility crossing the road that is not shown. The create 3D cut tool can be used to show this crossing utility.

- Open the Utilities Basemap.

- Select the create 3D cut button from within the Alignment' profile view.



- Data point to accept the placement method as corners
- Data point in the profile view at one corner to start the area of the 3d cut
- Next data point in the profile view at the opposite corner to create the 3d cut area.



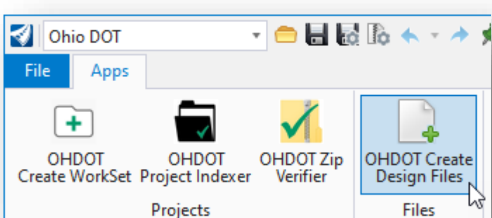
This process can be repeated for any number of 3d cut areas needed. Note that these areas can be selected and changed in size. You must hit the refresh 3d Cut button to update the graphics within the 3d cut area anytime a change it made included level display. This is a static 3d cut so changes to the model would require a refresh to the 3d cut as well.

Showing Drainage and Utilities on Plan and Profile sheets

The previous two sections showed how to set up drainage and utilities for plan and profile sheets. Now that the setup is done this section will show the steps for showing these features on a plan and profile sheet.

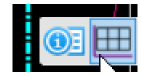
Note that Named boundaries will be placed to create these sheets and as outlined in chapter 800, the named boundaries can be placed in the same file the sheets are generated in or a container file. For this example, we will put the named boundaries in the same file the sheets are generated in. First, we will need to create a plan and profile sheet file to set up for our plan and profile sheet.

- Create a Plan and Profile sheet, don't forget to change the seed to 123456_DesignSeed2d.dgn

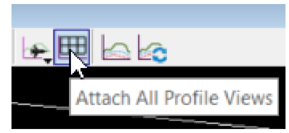


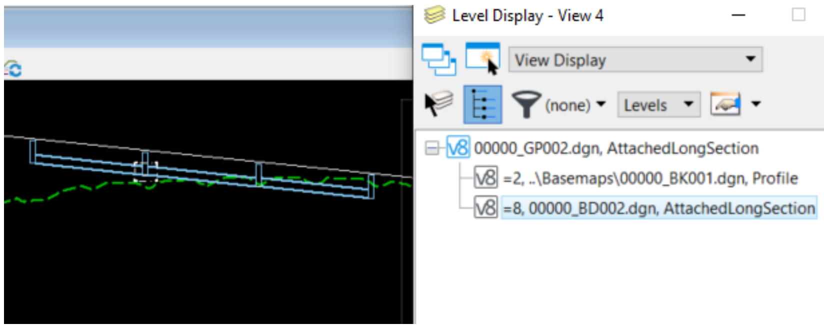
- After creating the file, add all desired references. This is typically the Fieldbook, alignment, corridors, Right of Way basemap, Drainage basemap, and Utility basemap files.

- Open the profile view of the alignment that the plan and profile sheet will be based on.

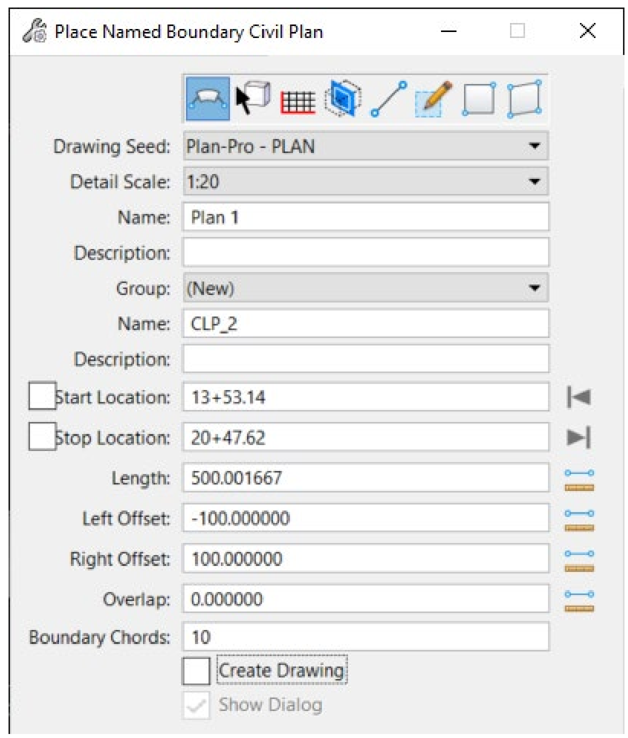


- Select the Attach All Profile Views button from within the profile view window. This will attach the profile view for any references that have drawn content for this profile view within that reference file.



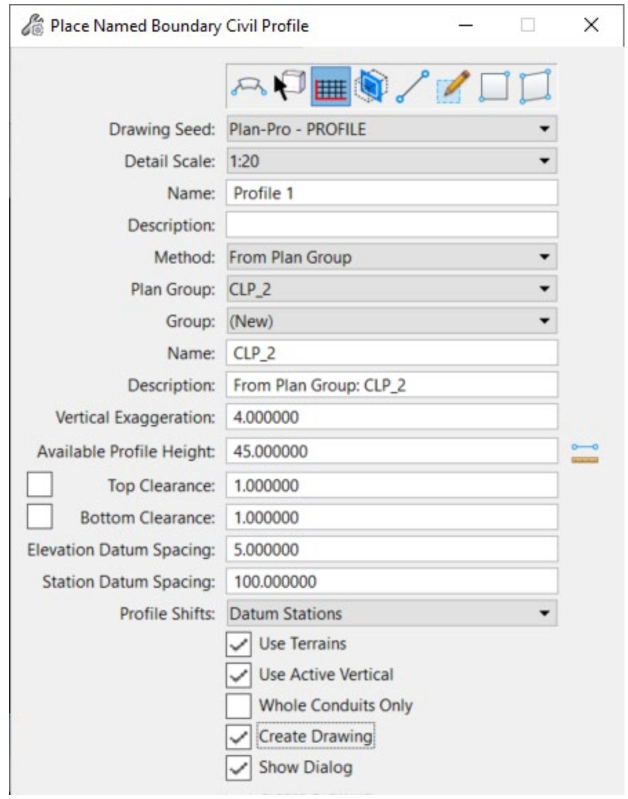


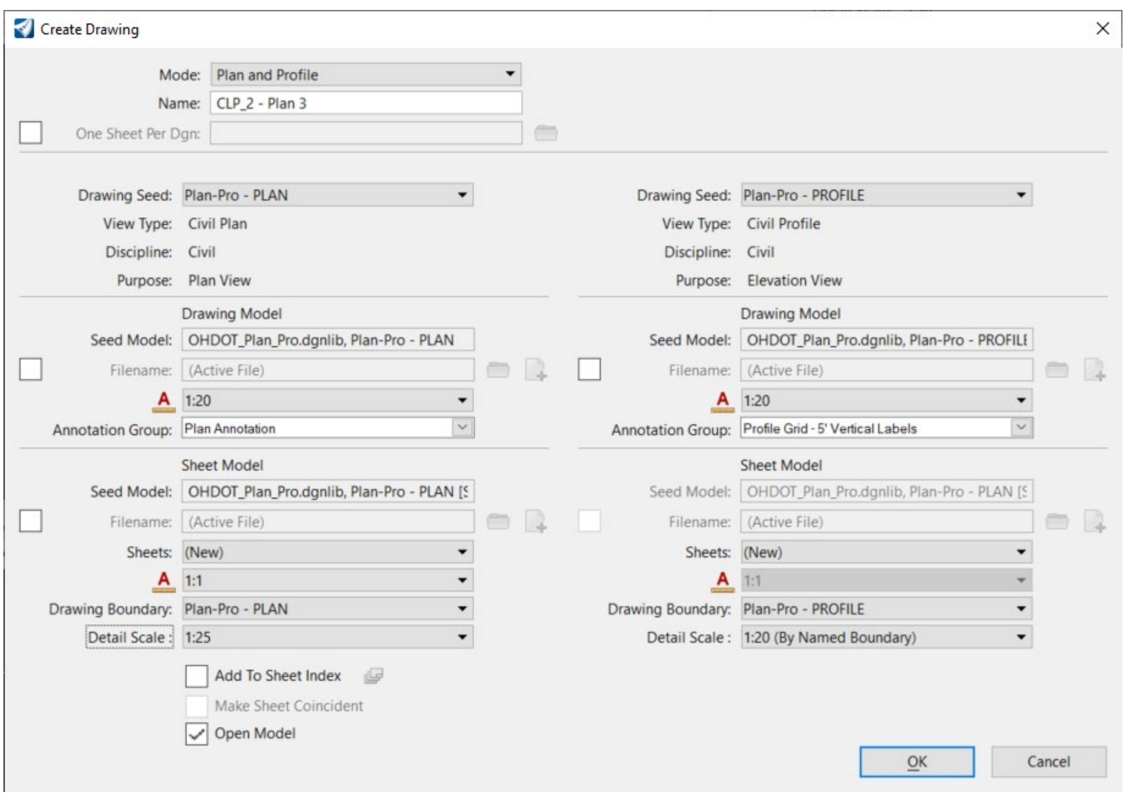
- Select the **Place Named Boundary** tool and set the mode to Civil Plan, and the Drawing seed to either Plan-Pro - PLAN.
- Set the scale to the desired value.
- Select the Center Line alignment which will be used to create named boundaries along.
- Key in or data point at the station you want the first named boundary to start at.
- Key in or data point at the station you want the last-named boundary to end at.
- Before issuing a final data point to place the plan view named boundaries make sure to check off create drawing, as we don't want to create the drawings until we set up our profile named boundaries.
- Now data point in the view window to accept and place the plan view named boundaries
- Select the **Place Named Boundary** tool again and this time set the mode to Civil Profile, and the Drawing seed to either Plan-Pro - Profile.
- Set the scale to the same set when placing the plan view named boundaries.
- Set the Method to From Plan Group and set the plan group to the group that was just created with the plan view named boundaries. this is typically the name of the selected alignment.

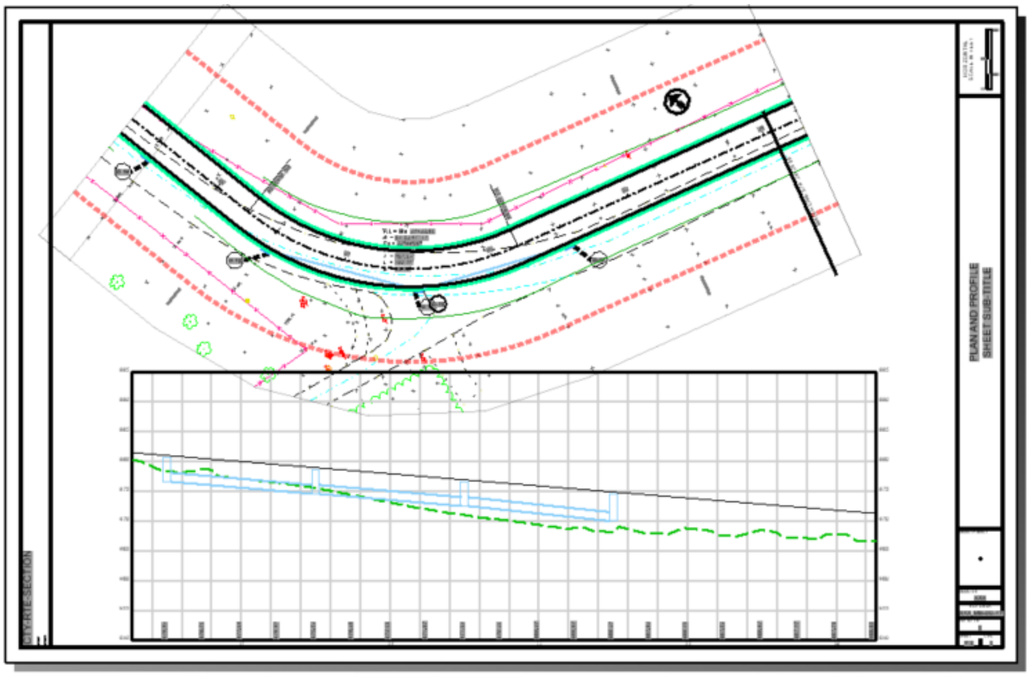


- Data point inside the view window containing your profile view.

- The profile named boundaries should now be shown in the profile view window. These are temporary graphics.
- Before issuing the final data point make sure to check on the Create Drawing and show dialog checkboxes since now, we are ready to generate the sheets.
- Now data point in the profile view to place the profile Named boundaries.
- Since Create Drawing was check on the software will not start generating the sheets. Since Show Dialog is check on a dialog will show up where you can review/edit settings for the sheet creation.
- Below shows the screen shot of the create drawing dialog. Make sure the Drawing model annotation scales match the scale you selected when placing the named boundaries, make sure the sheet model annotation scale is 1:1 and the sheet model detail scale matches the scale you selected when placing the named boundaries. Select OK and the software will generate the sheets.





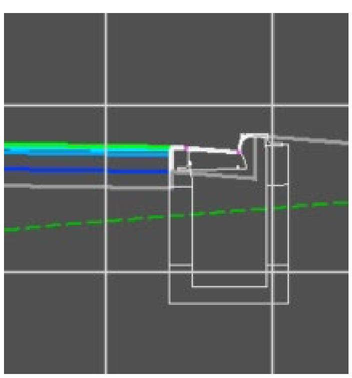


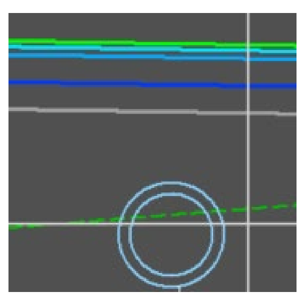
Showing Drainage and Utilities on Cross Sections

This section will explain how drainage and utility features are shown on cross section sheets. Refer to Chapter 800 of this guide for details on creating cross section sheets.

If you have modeled the drainage and utilities as outlined in this chapter with using the drainage and utilities tool, then your drainage and utilities should be modeled in 3D. When cross sections are generated, the graphics that are displayed on the cross sections are a slice of the 3d model. This means that when you cut cross sections if a drainage or utility feature is in 3d model at the location of the cross section then it will be displayed on the cross section.

If you want to show drainage nodes on cross sections, then a section cross must be cut at the location of the node.





Annotating Drainage and Utility Features

Several things have been set up to assist in the process of annotating drainage and utility features. Annotation of drainage and utility features starts with Text Favorites. Text Favorites are basically predefined text that contains text fields. These text fields are placeholders for property values. The properties can come from element properties, model properties, file properties, etc.

An excel file can be found within the OHDOT cadd standards that provides more detail on what text favorites are available for Drainage and Utilities and example of each one. The screen shot below shows some of the text favorites.

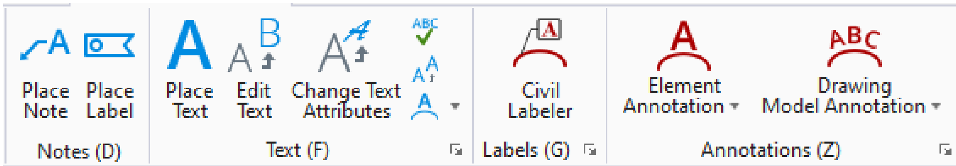
\..\WorkSpaces\OHDOT\Standards\OHDOT Utilities\Documentation\Drainage and Utilities Text Favorites.xlsx

**PLEASE NOTE CURRENTLY ANNOTATION FOR DRAINAGE AND UTILITY FEATURES DOES NOT WORK ACROSS A REFERENCE. WITH ORD 2020 R3. YOU CAN ANNOTATION ACROSS A REFERENCE WHILE YOU ARE IN THE FILE BUT WILL NEED TO COPY/PASTE THE TEXT FIELDS DOWN TO STATIC TEXT FOR PRINTING. THIS SHOULD BE FIXED IN AN UPCOMING RELEASE OF ORD. IF USING A VERSION PREVIOUS TO ORD 2020 R3 THEN ANNOTATION WILL NOT WORK ACROSS A REFERENCE. IF USING ORD 2020 R3 THEN SEE ANNOTATING IN ORD 2020 R3 SECTION FOR WORKFLOW. **

Text Favorite Name	Format	Example	Requirements
FeatureName	*Feature Name*	*D-2*	
Plan DrainagePipe-P	***Size Type***	***18" TYPE B***	Conduit
Profile DrainagePipe-P	***Length-Size Type @ Slope***	***100'-12" TYPE B @ -1.98%***	Conduit
Profile DrainagePipe-X	*EX. Size Type*	*EX. 12" RCP*	Conduit
Profile/XS DrainageNode-P	***STA XX+XX.XX, Offset Type, GRATE ELEV XXXX.XX***	***STA 386+50.00, 12.00' CB-3, GRATE ELEV 1003.65***	Catch Basin & CivilAccuDraw
Profile/XS DrainageNode-P_Unformatted	***STA XXXX.XX, Offset Type, GRATE ELEV XXXX.XX***	***STA 38650.00, 12.00' CB-3, GRATE ELEV 1003.65***	Catch Basin & Storm Feature
Profile/XS DrainageNode-P-MH	***STA XX+XX.XX, Offset Type, RIM ELEV XXXX.XX***	***STA 386+50.00, 12.00' MH-1, RIM ELEV 1003.65***	Manhole & CivilAccuDraw
Profile/XS DrainageNode-P-MH_Unformatted	***STA XXXX.XX, Offset Type, RIM ELEV XXXX.XX***	***STA 38650.00, 12.00' MH-1, RIM ELEV 1003.65***	Manhole & Storm Feature
Profile/XS DrainageNode-P-PipeEnd	***Size (direction) Elev at end of pipe***	***18" (N) 1002.13***	Conduit
Profile/XS DrainageNode-P-PipeStart	***Size (direction) Elev at start of pipe***	***18" (S) 1003.14***	Conduit
Profile/XS DrainageNode-X	*STA XX+XX.XX, Offset EX. Type, GRATE ELEV XXXX.XX*	*STA 386+50.00, 12'.00 EX. CB, GRATE ELEV 1003.65*	Catch Basin & CivilAccuDraw
Profile/XS DrainageNode-X_Unformatted	*STA XXXX.XX, Offset EX. Type, GRATE ELEV XXXX.XX*	*STA 38650.00, 12.00' EX. CB, GRATE ELEV 1003.65*	Catch Basin & Storm Feature
Profile/XS DrainageNode-X-MH	*STA XX+XX.XX, Offset EX. Type, RIM ELEV XXXX.XX*	*STA 386+50.00, 12.00' EX. MH, RIM ELEV 1003.65*	Manhole & CivilAccuDraw
Profile/XS DrainageNode-X-MH_Unformatted	*STA XXXX.XX, Offset EX. Type, RIM ELEV XXXX.XX*	*STA 38650.00, 12.00' EX. CB, RIM ELEV 1003.65*	Manhole & Storm Feature
Profile/XS DrainageNode-X-PipeEnd	*EX. Size direction Elev at end of pipe*	*EX. 12" PVC N 1002.13*	Conduit
Profile/XS DrainageNode-X-PipeStart	*EX. Size direction Elev at start of pipe*	*EX. 12" PVC S 1003.14*	Conduit
Profile/XS SanNode-P-MH_Unformatted	***STA XXXX.XX, Offset Type, RIM ELEV XXXX.XX***	***STA 38650.00, 12.00' MH-1, RIM ELEV 1003.65***	Manhole & Sanitary Feature
Profile/XS SanNode-X-MH_Unformatted	*STA XXXX.XX, Offset EX. Type, RIM ELEV XXXX.XX*	*STA 38650.00, 12.00' EX. CB, RIM ELEV 1003.65*	Manhole & Sanitary Feature
XS DrainagePipe-P	***Size Type***	***12" Type B***	Conduit
XS DrainagePipe-X	*EX. Size Type*	*EX. 12" STORM*	Conduit
Util-Comm-P-SizeType	***Size Type***	***6" FIBER OPTIC***	communication
Util-Comm-P-Type	***Type***	***FIBER OPTIC***	communication
Util-Comm-X-SizeType	*EX. Size Type*	*EX. 6" FIBER OPTIC*	communication
Util-Comm-X-Type	*EX. Type*	*EX. FIBER OPTIC*	communication
Util-Elec-P-SizeType	***Size Type***	***6" ELECTRIC***	Electrical
Util-Elec-P-Type	***Type***	***ELECTRIC***	Electrical
Util-Elec-X-SizeType	*EX. Size Type*	*EX. 6" ELECTRIC*	Electrical
Util-Elec-X-Type	*EX. Type*	*EX. ELECTRIC*	Electrical
Util-Gas-P-SizeType	***Size Type***	***6" GAS***	Gas
Util-Gas-P-Type	***Type***	***GAS***	Gas
Util-Gas-X-SizeType	*EX. Size Type*	*EX. 6" GAS*	Gas
Util-Gas-X-Type	*EX. Type*	*EX. GAS*	Gas
Util-Oil-X-SizeType	*EX. Size Type*	*EX. 6" OIL*	POL
Util-Oil-X-Type	*EX. Type*	*EX. OIL*	POL
Util-Water-P-SizeType	***Size Type***	***6" WATER***	Water
Util-Water-P-Type	***Type***	***WATER***	Water
Util-Water-X-SizeType	*EX. Size Type*	*EX. 6" WATER*	Water
Util-Water-X-Type	*EX. Type*	*EX. WATER*	Water

Typically, annotation should be done in the drawing models.

There are several tools available to assist in placing annotation.



Place Label: can place cells or text favorites with or without a leader line.

Element Annotation: if the element has an annotation group defined then element annotation will annotate the element.

Remove Element Annotation: removes the annotation that's associated with the element.

Model Annotation: annotates the entire drawing model.

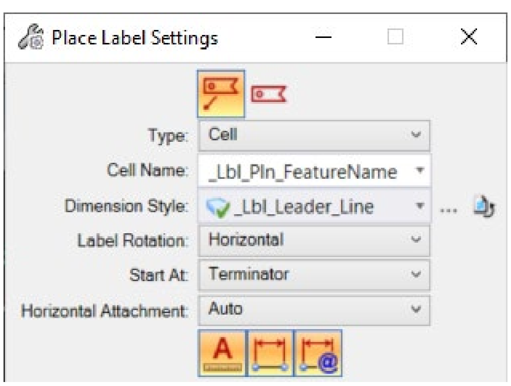
Remove Model Annotation: Removes all annotation from the model that was.

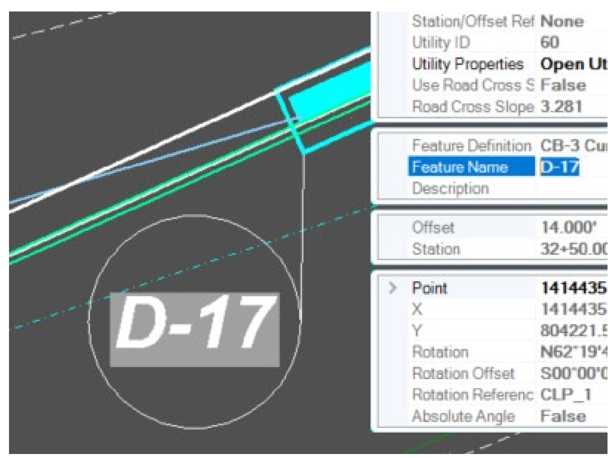
Plan View Drainage Annotation

Call outs

There are many ways to annotate a cell out. One way is to use _Lbl_Pln_FeatureName cell with the **Place Label** tool. This cell uses the Plan – Feature Name Text favorite and puts a circle around it. The Plan – Feature Name text favorite will display the feature name of the specified feature.

When using the place label tool make sure to select the element or the label will not get associated to that element and won't be able to pull properties from the element.



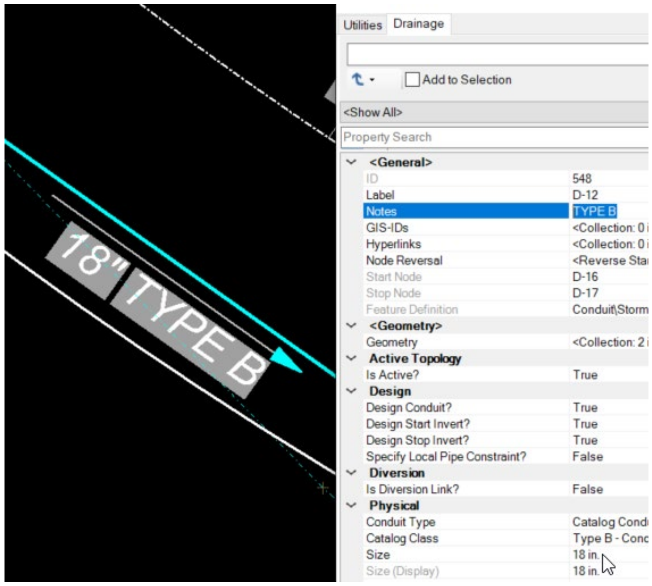


Note: Element annotation has been defined for Proposed Catch Basins, Manholes, and Inlets to place these call outs.

Plan View Drainage Pipe Annotation

Pipe Annotation has been defined on all proposed conduits. This means 

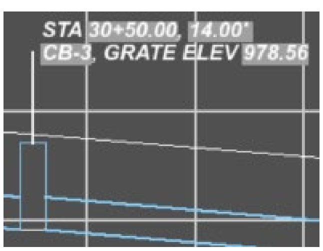
can be used. This will place a flow arrow in the direction of flow for the pipe as well as the size and type. Note that the type is coming from the Notes drainage property which has been preset on the feature definition.



** While annotation should typically be placed within drawing models. There are currently some bugs in the software, and it is best to place these pipe annotations in the Drainage Basemap file. **

Profile View Drainage Annotation

When profile sheets are generated, features that have annotation groups defined will automatically get annotated. Both node and pipe annotations for profile view have annotation groups defined so these elements should auto annotate. Note that more work needs to be done to the node annotation.



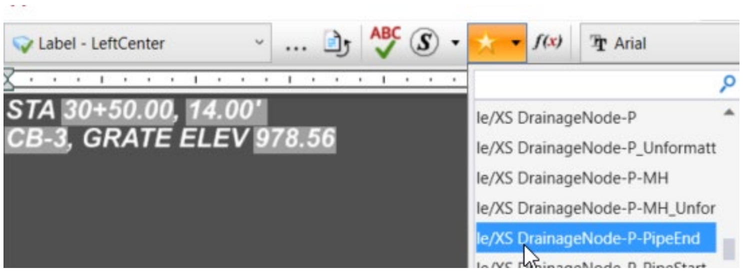
Node Annotation

As mentioned above node annotation should be auto annotated during sheet creation. The **Model annotation** and **Element annotation** tools could be used as well. But the annotation is missing the information about the connecting pipes. Currently this cannot be auto annotated. Follow the below steps to add in the annotation for each pipe that connects to the node.

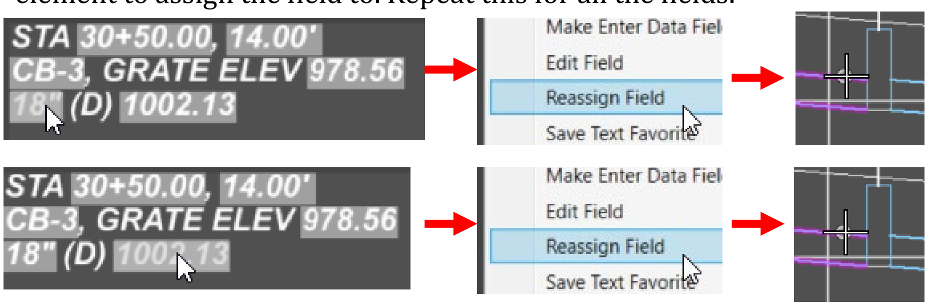
- Double click on text element or use **edit text** tool to open the text editor for the node annotation.
- Within the text editor insert a text favorite for the connected pipe. There are two text favorites to choose from for proposed nodes

The Profile/XS DrainageNode-P-PipeEnd will pull the end of the pipe invert elevation while the Profile/XS DraiangeNode-P-PipeStart pulls the start of the pipe invert elevation.

NOTE: *the pipe start/end locations represent the order/direction the pipe was drawn in not necessarily the direction of flow. Care must be taken to select the correct text favorite.*



- After selecting the text favorite, it will be inserted into the text editor dialog. But the fields within the text favorite still need to be assigned to an element to know the property values to display. To do this you simply right click on the field and select reassign then select the element to assign the field to. Repeat this for all the fields.



- The (D) standards for direction and must be manually typed in. This is the direction the pipe is going away from the node.

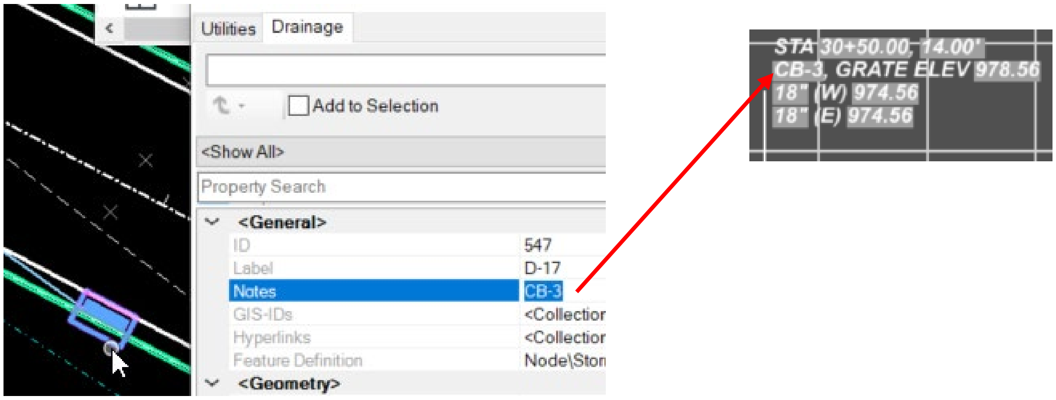
- Repeat the above two steps for any additional pipes connected to that node making sure to select the appropriate text favorite for each pipe.

This process is the same for existing drainage nodes with the exception that the Profile/XS DrainageNode-X-PipeEnd and Profile/XS DrainageNode-X-PipeStart should be used instead.

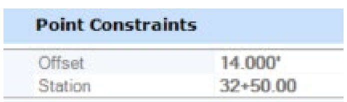
Additional Info

The auto annotation of nodes is using the Profile/XS DrainageNode-P or Profile/XS DrainageNode-P-MH text favorites (Profile/XS DrainageNode-X, and Profile/XS DraiangeNode-X-MH text favorites for existing). You could use the **Place Label** tool with these text favorites to achieve the same annotation.

The **structure type** field in the annotation is coming from the Notes drainage property as imaged below.

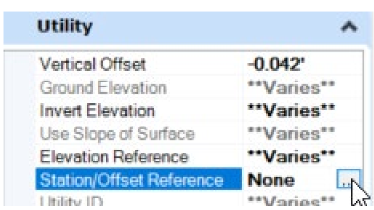


STATION AND OFFSET ANNOTATION



The Station and offset values are coming from a point constraint property that has been applied to the node. These point constraint properties are only added if the node was placed using civil AccuDraw station-offset ordinate or the **Place Nodes** tool. If the node was not placed using one of the above two mentioned methods, then the annotation will not have a station or offset property to get the value from and will be blank. If this is the case, you can either use civil AccuDraw station-offset ordinate while moving the node (to the same spot) to apply the needed point constraints or below lists an ***alternative workaround*** to annotating station and offset.

- In the Drainage basemap select all the drainage nodes and open the Properties dialog.



- In the properties dialog select the three horizontal dots in the Station/Offset Reference property.
- Now select the desired alignment to use that the station/offset reference.

- Now go back to the Profile Drawing model.

- Additional text favorites have been set up to use the Station/Offset Reference properties instead of the point constraint properties, but there are limitations to the formatting of these text favorites. The text favorites available are Profile/XS DrainageNode-P_Unformatted, Profile/XS DrainageNode-P-MH_Unformatted, and Profile/XS SanNode-P-MH_Unformatted (Profile/XS DrainageNode-X_Unformatted, Profile/XS DrainageNode-X-MH_Unformatted, Profile/XS SanNode-X-MH_Unformatted text favorites for existing).

- Use the Place label tool and select the Profile/XS DrainageNode-P_Unformatted text favorite for inlet nodes, Profile/XS DrainageNode-P-MH_Unformatted text favorite for storm manholes, or Profile/XS SanNode-P-MH_Unformatted text favorite for sanitary manholes.

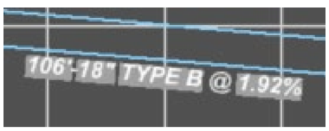
- After placing the label, you will notice that the station value is not formatted correctly, this is a current limitation of the software that will eventually get fixed.

- Edit the text element and type over the station text field to format it as desired. Note that this will make the station value static and if the node is moved the value will **not** dynamically update.

- Follow the Node Annotation section for the steps on annotation the connecting pipes.

Pipe Annotation

As mentioned above pipe annotation should be auto annotated when profile sheets are generated.



Model Annotation and **Element Annotation** tools could be used as well. This auto-annotation is using the Profile DrainagePipe-P text favorite (Profile DraingePipe-X text favorite for existing). You could use the **Place Label** tool to achieve the same annotation.

The Length value in this text field is the center of node to center of node length. The slope value is also based off the center of node to center of node length. Just as in the plan view annotation the Type is coming from Notes Drainage Property.

Cross Section View Drainage Annotation

Cross section drainage annotation is very similar to Profile drainage annotation except that element annotation tool will not work for cross sections and currently nothing is auto annotated.

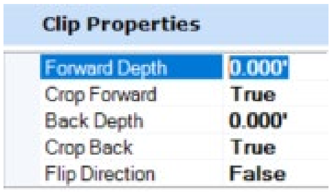
Cross section annotation should be done in the drawing models that are created from named boundaries.

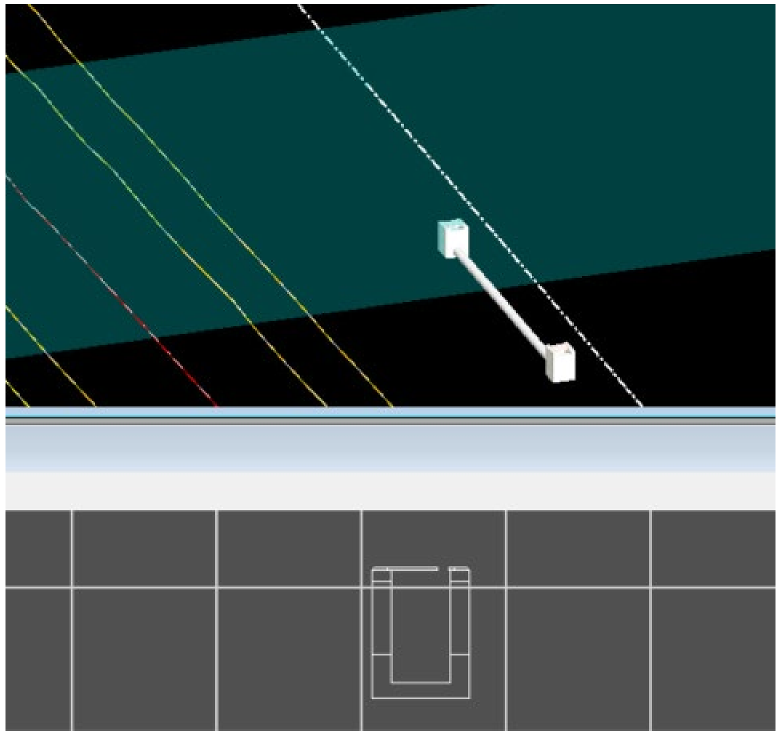
Node Annotation

Unlike Profiles, drainage networks are not projected to the nearest cross section. If you want to show a drainage node on a cross-section, then a cross section must be cut at that location. The same text favorites that are used to annotate nodes in a profile view are used to annotate nodes in a cross-section view. Please follow the Profile View Drainage Annotation – Node Annotation Section for the workflow.

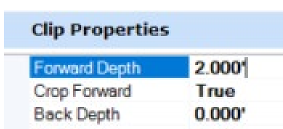
When annotating connecting pipes (as outlined in the Profile View Drainage Annotation – Node Annotation Section) there is a good chance that the connecting pipe will not be visible on the cross section making it hard to pull annotation values for that pipe. Follow the steps below to temporary display the connecting pipes to set up the connecting pipe annotation.

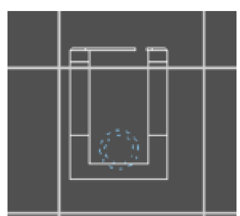
- Select the 3d named boundary from the 3D model for the cross section containing the drainage node that has connecting pipes that are not visible and change the Forward or Back Depth properties. This will widen the cross-section slice to show a thicker section of the 3D. the Forward depth will widen the named boundary in the up-station direction. The value needs to be great enough to get the named boundary to cut where the pipe is.





- After the value is updated notice how the connecting pipe shows up in the cross-section view. Now the connection pipe annotation can be added to the node. (Follow the steps outlined in the [Profile View Drainage Annotation – Node Annotation Section](#)).



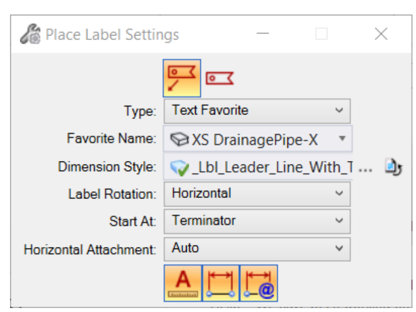


- Change the Forward/Back depth back to zero. And repeat of the other side if needed.

Pipe Annotation

Pipes will only be shown in the cross section if that pipe is crossing the Cross section named boundary.

The following steps outline how to annotate these pipes.



- Select the **Place Label** Tool. Select **type** then set as **Text Favorite** and use the XS DrainagePipe-X for existing pipes and XS DrainagePipe-P for proposed pipes.

- Select the pipe and place the annotation

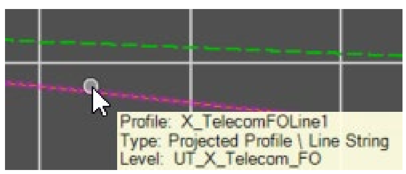
Just as in the plan view and profile view annotation the Type is coming from Notes Drainage Property.

Profile View Utility Annotation

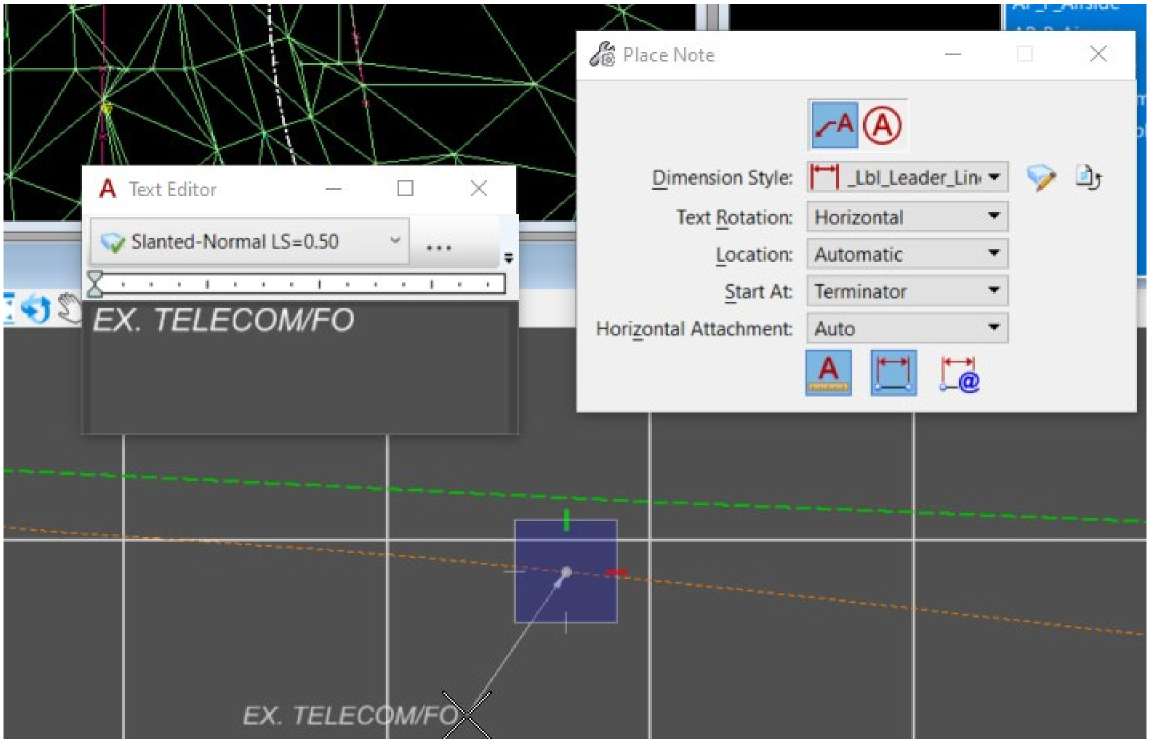
While there are several text favorites set up for utility annotation, these will not work if the workflow outlined in the Setting up Utilities to show on Plan and Profile Sheets - Projecting Profiles section. This is because that workflow is just creating profiles using regular linear feature definitions and NOT an actual conduit feature definition, meaning that none of the utility properties exist on those elements. This workflow is used because currently projecting the actual utility shows two lines (the top and bottom of the conduit). Typically, we only show one line for these utilities as the diameter is small and two lines adds a lot of unneeded clutter. If you would like to use text favorites to annotate utilities, then follow the alternative workflow for projecting utilities section.

- If the Setting up Utilities to show on Plan and Profile Sheets - Projecting Profiles section was used to show utility profiles on centerline profile, then annotation will be manual using the **Place Note** tool.

By hovering over the element, you can see what feature/level it is on to know what utility it is

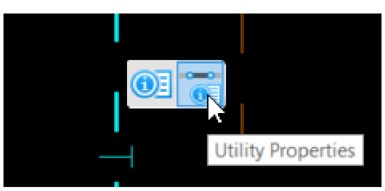


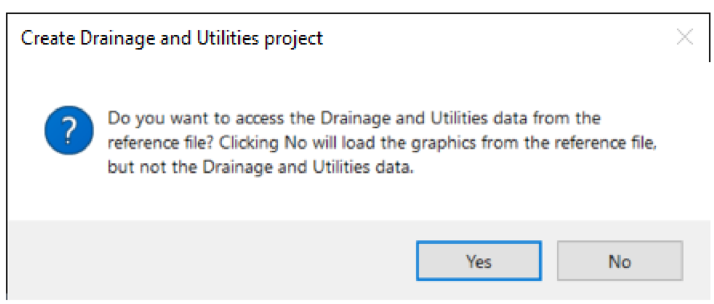
Then use the place note tool to place the annotation as shown in the screen shot below.



- If the [Alternative workflow for projecting utilities]() section was used to show utility profiles on centerline profile, then annotation can be dynamically pulled from the utility properties using text favorites with the **Place Label** Tool. (Remember if you're missing the projected utility runs to use the Attach all profile views button).

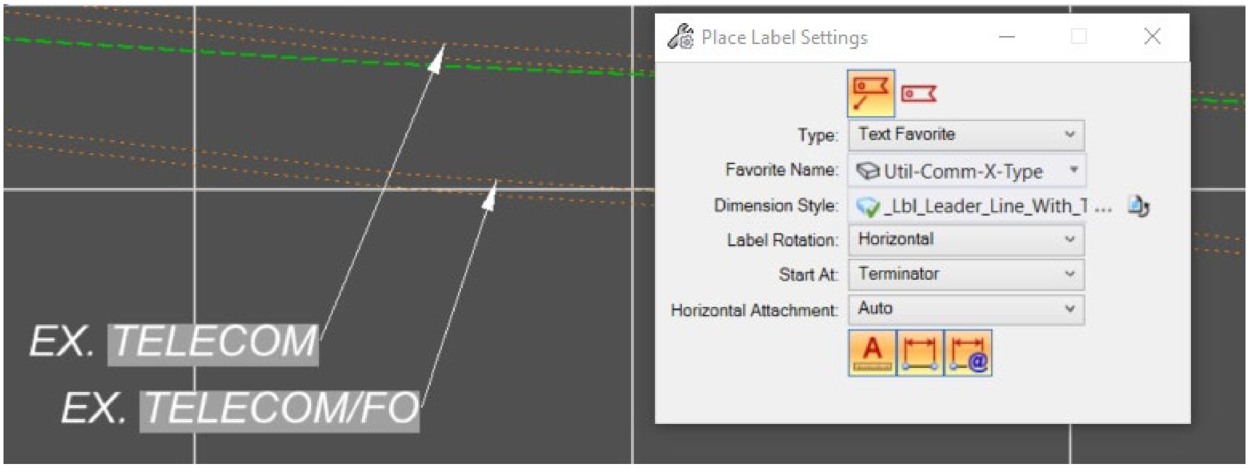
Remember before annotating Drainage & Utility features that the Drainage & Utility Project needs to be created in this project so that the drainage & Utility properties can be accessed. This is done easily by simply opening the Utility properties of any of the Drainage and Utility elements that are referenced into the file.



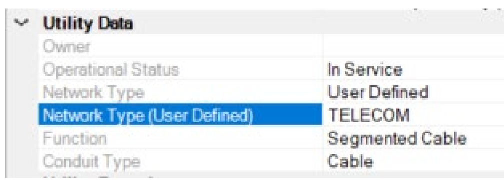


Now simply use **Place Label** tool. Note the different text favorites available for utilities.

Text favorites are specific to the utility type. For each type there is a text favorite for annotating just the utility type and another for the utility size and type.



The Utility text favorites are pulling the Network Type (User Defined) property. This has been predefined on the utility feature definitions with appropriate values.



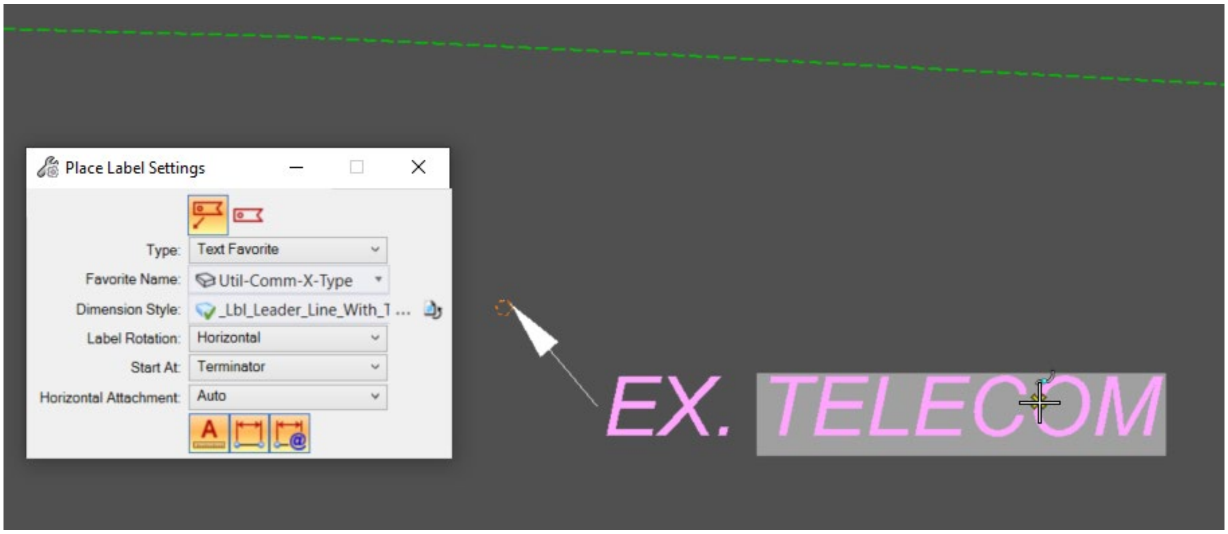
Currently this method does show a line for the top and bottom of the utility. You could also follow this workflow [Setting up Utilities to show on Plan and Profile Sheets - Projecting Profiles] to show utility as one line in addition to this workflow, [Alternative workflow for projecting utilities] to use with the text favorites and turn off levels from the utility run projections.

Cross Section View Utility Annotation

Cross section utility annotation is very similar to Profile Utility annotation.

Cross section annotation should be done in the drawing models that are created from named boundaries.

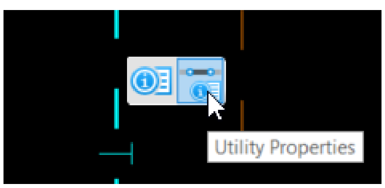
Use The **Place Label** Tool with the Utility text favorites to annotate Utilities in cross section drawing models.

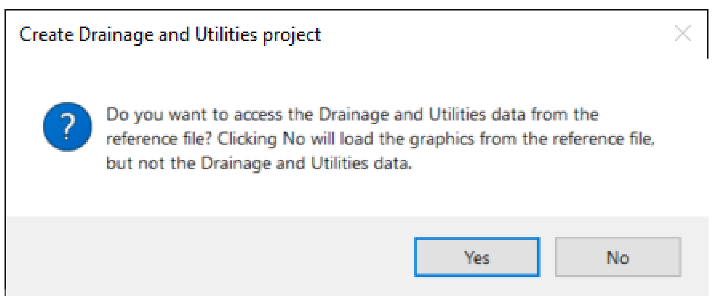


See the Profile View Utility Annotation Section for more details in the Utility Text Favorites.

Annotation in ORD 2020 R3

With ORD 2020 R3 it is possible to annotate drainage and utility features across a reference. There are still some bugs that will be fixed in a future release of ORD. For now, to annotate Drainage & Utility features across a reference you first open the utility properties of any referenced in node/conduit. You will be prompted with the below dialog. Select yes and it will then expose the Drainage & Utility properties from the reference, which can then be used within text favorites.





There are two major bugs when annotates Drainage and Utility features across a reference in ORD 2020 R3.

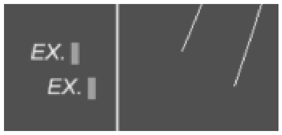
1. Auto-Annotation (element/model annotation) does not associate fields in text favorites to the actual element. (It does not work).

   To fix this, every field within the text favorite needs to be re-assigned to the element.

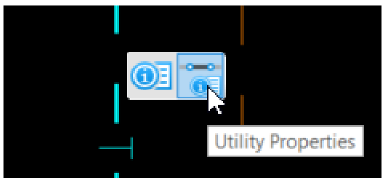
2. Annotation values do not hold after you close and reopen the file.

   The workaround for this is to change the fields to static text. This will make it so the values **WILL NOT** update if the Drainage & Utility network changes.

After you close and reopen the file, you will notice that the Drainage & Utility annotation are blank.



To fix this you must again open the Utility properties.



Now the text field should refresh, and the annotation will have the correct values.

When printing sheets to PDF this does have the possibility to print to PDF without the values. To alleviate this possibility, you can copy and paste the text fields down to static text.

OpenRoads Software Version

This document was prepared using the following software version:

OpenRoads Designer CONNECT Edition – 2020 Release 3 Update 9 – Version 10.09.00.91

This document was updated using the following software version:

OpenRoads Designer CONNECT Edition – 2021 Release 2 Update 10 – Version 10.10.21.04

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