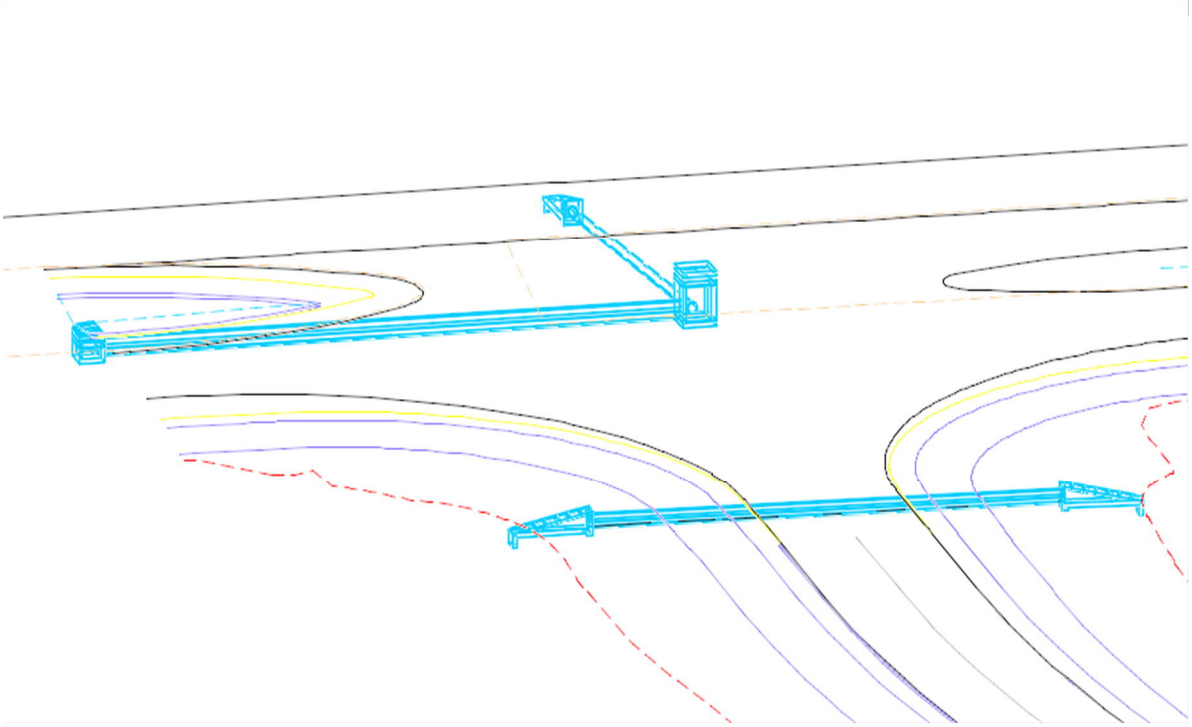
SUDA

OpenRoads Designer

(Formerly GEOPAK Drainage)

CONNECT Edition



TDOT

Roadway Design Division

October 2022

TN TDOT Department of Transportation

Table of Contents

Preface

Purpose & Need

The Subsurface Utility Design and Analysis (***SUDA***) OpenRoads Designer (ORD) Manual is the fourth document in a series of **six** training manuals released by the Tennessee Department of Transportation (TDOT) that is available through a digital, interactive **flipbook** format. Bentley's ORD software is being adopted and implemented statewide by TDOT as the new **3D modeling** design software, which will ultimately replace both MicroStation V8i and Geopak (SELECT Series 2). This manual provides an introduction to the **Drainage and Utility modeling** workflow in ORD and focuses on hydraulic analysis and drainage calculations for storm drain networks. Terrain is discussed briefly in the Appendix but refer to the Roadway Design II (ORD) Manual for a more in depth review of the terrain tools. The completed drainage network created in the manual can be referenced in **Appendix C**.

Disclaimer

The ***SUDA*** Manual is developed based on OpenRoads Designer CONNECT Edition 2021 Release 2, Version **10.10.21.04**. The TDOT ORD workspace (**10.10.21.04_10.14.2022**) complies with the latest CADD standards and should be used in conjunction with this manual. It can be downloaded on the TDOT CADD Support website under **OpenRoads Designer Information >> ORD Resources >> Support Files**. If you have any technical issues or recommendations for this manual, please contact TDOT CADD Support at TDOT.ORD@tn.gov.

Training Videos

The ***SUDA*** Manual has accompanying training videos which are intended to be utilized as you go through the exercises. Video icons have been inserted throughout the flipbook, providing direct access to the applicable video. In general, the videos contain instructional guidance and additional tips and tricks, as well as an informational bar at the bottom of the screen to help stay on track.

Revisions

The ***SUDA*** Manual will be revised over time as a result of future ORD software releases and procedural & workspace updates. All revisions will be documented by WSP/TDOT and included on the **Revision History** page at the end of the manual. TDOT CADD support will announce updated manual versions when they become available via emailed Instructional Bulletins. The updates will also be posted on the ORD TDOT webpage.

ORD Training Manuals

The ***SUDA*** Manual is one of **six** ORD training manuals available. Each manual has its own icon and color associated with it, which are maintained throughout the applicable manual and videos to help the user with wayfinding.

Chapter 1. Course Overview

Course Description and Objectives:

This course introduces users to the **Drainage and Utilities** [also known as **Subsurface Utility Design and Analysis** (SUDA)] tools available in the OpenRoads Designer (ORD) CONNECT software. The **SUDA** acronym will be referenced throughout this manual.

At the conclusion of this course, participants will be able to:

1. Understand the workflow for Drainage and Utilities through the creation of a 3D model (nodes, conduits/links and catchment areas).
2. Understand the drainage system layout tools and hydraulic analysis tools.
3. Understand the hydraulic model components, design constraints and Prototypes.
4. Learn how to run Scenarios and Alternatives, and how to manage and create Flex Tables, Queries, and calculation reports.
5. Explore other tools such as StormCAD and SewerCAD to create hydraulic analysis for closed and open channel drainage network systems.

The topics covered in this class are:

1. Introduction to SUDA
2. Developing a Drainage and Utilities File
3. Terrain Setup and Management
4. Layout a Closed Drainage Network
5. Additional Software
6. Alternate Runoff Methods
7. Common Errors During Computations
8. Creating Separate Topologies for each Scenario
9. Snap Mode Tool Box
10. LiDAR Data Access
11. Workflow for Copying Files with Drainage and Utilities Data
12. Utilities Conflicts Tool

Target Audience:

This course is designed for anyone using the modeling tools to design storm drains, culverts, and sanitary sewers.

- Project Development

Pre-Requisites:

- Familiarity with TDOT's Drainage Manual, Standard Drawings, Specifications and Roadway Design Guidelines.
- A working knowledge of Windows 10
- Fundamental understanding of drainage calculations and Rational method
- Fundamentals (ORD) Manual and Roadway Design I (ORD) Manual

Chapter 2. Introduction to SUDA

SUDA is the utility and hydraulic design component of ORD. It provides comprehensive feature-based 2D and 3D modeling combined with dynamic model-based design computation and analyzation.

2.1 Objectives

At the conclusion of this chapter, participants will be able to:

1. Understand the drainage feature definitions and symbologies.
2. Understand the Drainage and Utilities workflow.
3. Navigate the ribbon interface.

2.2 Lecture: Feature Definitions

Feature Definitions essentially tell the software what each model object represents (e.g., differentiating between Type 12 and Type 14 catch basins). Each feature definition has an associated **Feature Symbology** used to create the civil model both in 2D and 3D. Drainage Feature Definitions have been set up per the **TDOT CADD Standards** and hydraulic design criteria.

TDOT has multiple Feature Definition libraries to manage survey, roadway, drainage, and utility features. This manual will only cover those applicable to **Drainage**. Feature Definitions, in general, are discussed in further detail in the Roadway Design I (ORD) Manual.

Table 1 below provides the summary of the three types of feature definitions in SUDA, along with their descriptions.

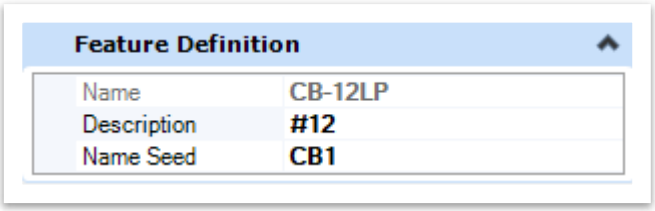
TABLE 1. DRAINAGE FEATURE DEFINITION TYPES

Feature Definition Type	Description
Nodes	A node represents an inlet or manhole and is a point defined in the drawing by the user.
Conduits	Known as links, conduits are linear features that connect two nodes.
Drainage Areas	Known as catchment areas, a drainage area is a closed boundary. Single drainage area flows are all tied to a single node in a one-to-one relationship.

2.2.1 Feature Definition Properties

All feature definitions have the following three properties: **Name**, **Description**, and **Name Seed** (Figure 1). The name seed is the default name for that piece of geometry. If there are multiple elements with the same feature definition, the name seed will increment by **1** with every new element placed.

FIGURE 1. DRAINAGE FEATURE DEFINITION PROPERTIES



2.2.2 Feature Symbologies

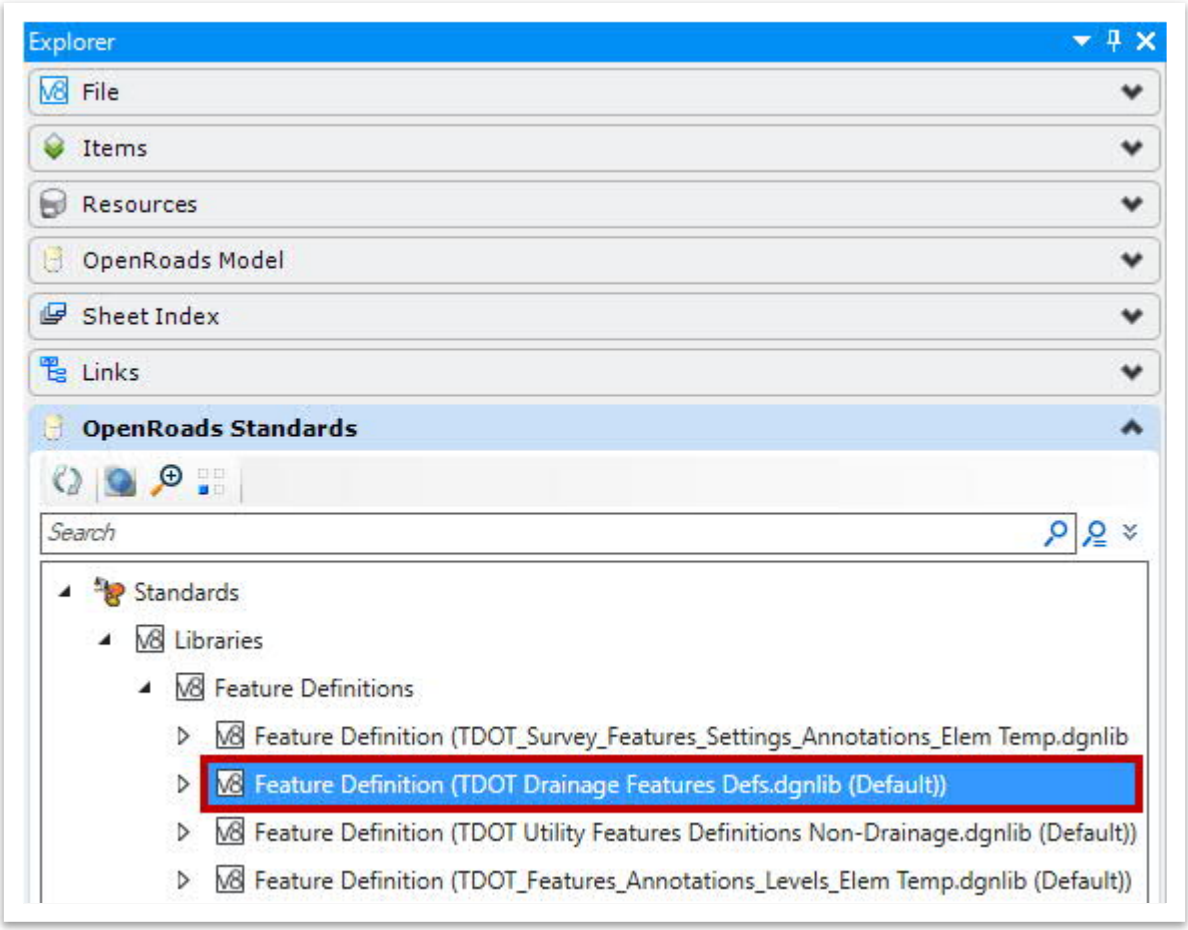
Feature Symbology settings are defined for each drainage feature and let the software know what element templates to assign each feature definition. These settings determine **the appearance of each element and the level associated with the geometry**. Depending on the element, there may be symbology for 2D and/or 3D objects. The feature symbologies are located within the **TDOT Drainage Feature Defs.dgnlib**.

2.2.3 Drainage Feature Definitions

Drainage Feature Definitions are located within the **TDOT Drainage Features Defs .dgnlib**. While these feature definitions may not be edited, they can be reviewed.

Within the **Explorer**, you can expand the **OpenRoads Standards** tab and browse to the following location to review the drainage feature definitions: **Standards >> Libraries >> Feature Definitions >> TDOT Drainage Features Defs.dgnlib** (Figure 2).

FIGURE 2. TDOT DRAINAGE FEATURES DEFS DGNLIB LOCATION



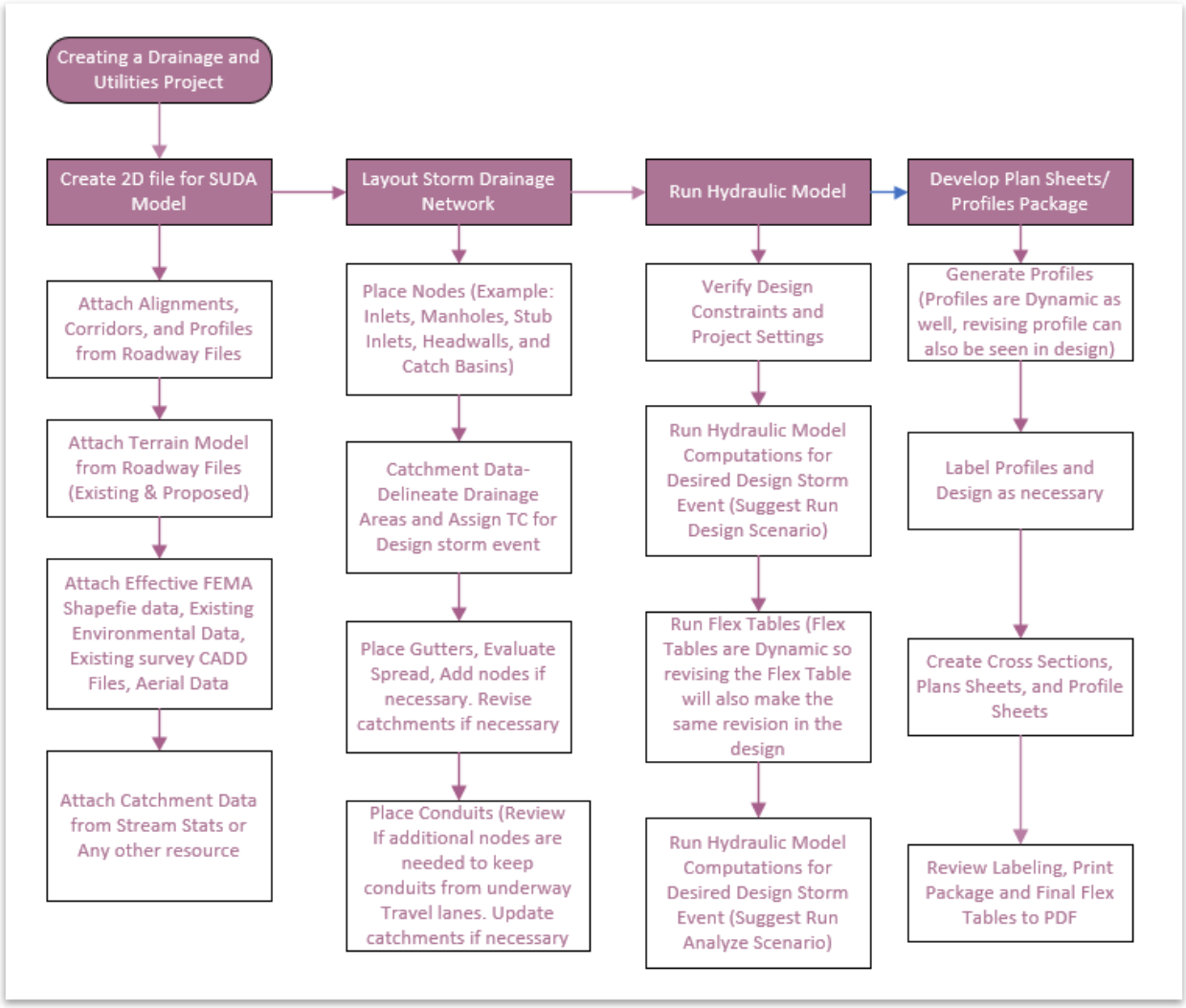
2.3 Lecture: Drainage and Utilities Workflow

The **Drainage and Utilities Workflow** houses all the tools needed for designing drainage networks using a model-centric approach. The general process to setup an ORD file to design and analyze a drainage network and to print redline drainage sheets is as follows (refer to Figure 3 on Page 7):

1. Create a blank **2D** ORD file using the **TDOTSeed2D** with the standard TDOT naming convention for the SUDA model.
    - Attach the terrain model from the roadway files (or create one using the **TDOTSeed3D** with the standard TDOT naming convention). Use the correct terrain depending on the existing/proposed conditions. At times, the project will need to use a merged terrain (portions of existing and proposed combined). Refer to the Roadway Design I (ORD) Manual for the process to create a merged terrain.
    - Attach USGS quad maps and imagery.
    - Export the kml file.
    - Attach existing dgn files (e.g., survey, wetland, etc.).
    - Attach proposed dgn files (e.g., alignments, corridors, profiles, and other roadway files).
2. Place standard redline drainage items.
    - Flow arrows (e.g., density).
    - Existing ditches (e.g., ditch detail locations).
    - Callouts for existing drainage conditions.
    - Existing drainage areas.

3. Layout the storm drainage network.
    - Place Nodes to layout the inlets, catch basins, manholes and endwalls.
    - Review and assign **Tc** values to all catchment areas. The catchment areas are either automatically generated by the software or will need to be manually created and then assigned to the applicable inlet. In addition, the user can use a combination of the two to get a **weighted C** for the rational method.
    - Place gutters to evaluate the spread criteria and finalize the catch basin and inlet locations/spacing.
    - Place conduits to connect the nodes.
4. Run the hydraulic model.
    - Verify that the project settings and Design Constraints meet the project's requirements.
    - Run the hydraulic model computations (Design or Analyze) for the desired storm event. This encompasses either designing a proposed system or analyzing the pipe sizes used to build the network.
    - Review the results and revise if necessary. If the **Design** option is used, software designs the sizes per criteria and the initial sizes will be overwritten with proposed sizes. If the **Analyze** option is used, the initial sizes are used in the analysis to verify if the criteria is met.
    - Use reporting and Flex Table tools (see Section 5.10.4) to generate reports, if necessary.
5. Develop Plan Sheets.
    - Create plan, profile, and cross-section sheet models, which will then be referenced into the roadway sheets. **Note:** When designing the storm drainage system, coordinate (if necessary) with the roadway engineer when creating these sheet models.
    - Add annotation and labels as necessary.

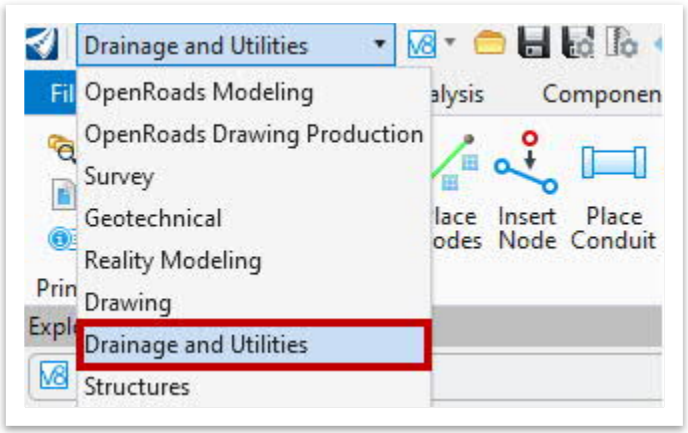
FIGURE 3. DRAINAGE AND UTILITIES WORKFLOW GRAPHIC



2.3.1 Navigating the Ribbons

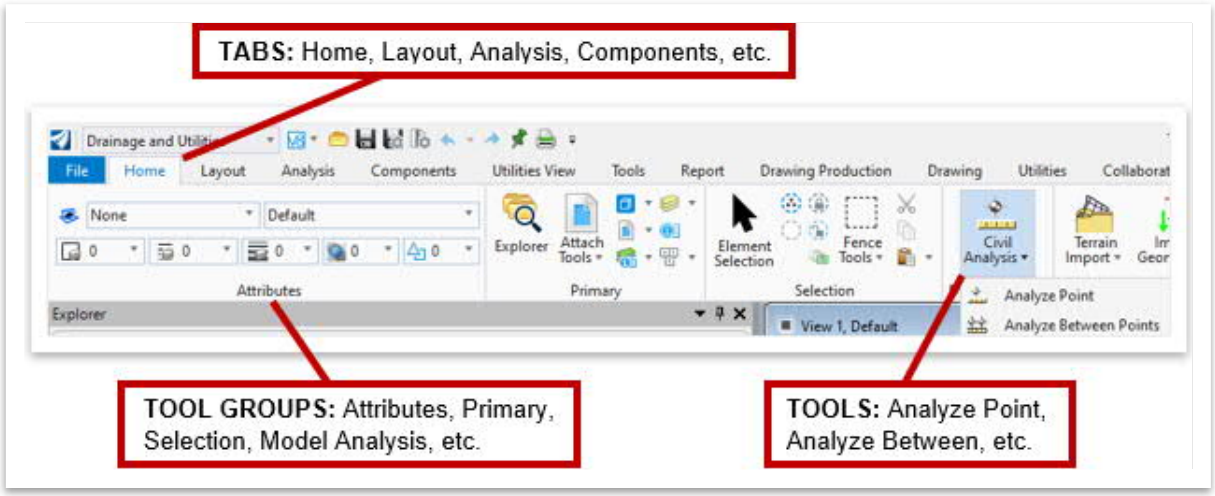
This workflow can be accessed in any ORD file within the **Quick Access Tool bar** located at the top left corner (Figure 4).

FIGURE 4. DRAINAGE AND UTILITIES WORKFLOW



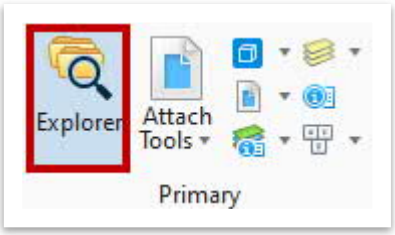
Within the Drainage and Utilities workflow, there are several tabs, tool groups and tools that are arranged in ribbons from left to right (Figure 5).

FIGURE 5. TABS, TOOL GROUPS, TOOLS



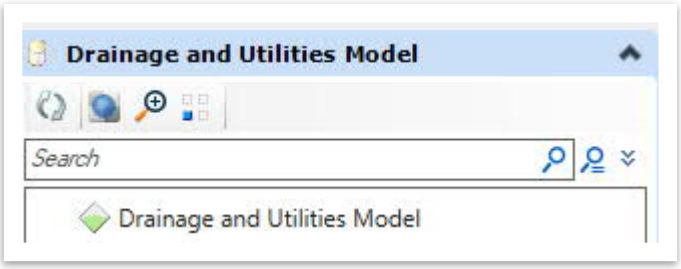
It is recommended to open the **Explorer** (**Drainage and Utilities >> Home >> Primary**) (Figure 6) and have it docked on the left side of the screen.

FIGURE 6. EXPLORER



Within the **Explorer**, the **Drainage and Utilities Model** tab should be used for managing and reviewing all drainage and utility data (Figure 7). **Note:** In ORD, all drainage files are embedded in the **dgn** file.

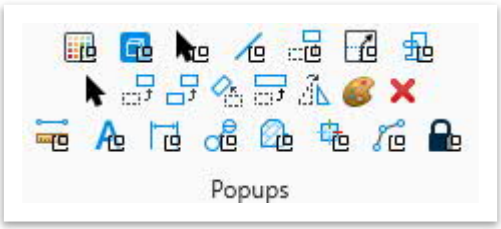
FIGURE 7. EXPLORER – DRAINAGE AND UTILITIES MODEL



2.3.2 Shortcut Keys

In V8i, users may recall hotkey commands such as typing **3** and **1** to copy an element or typing **3** and **2** to move an element. In ORD, hotkeys are setup differently. To access the **hotkey popup menu** (Figure 8), the user will need to press spacebar and then move the cursor over the applicable icon (move, copy, rotate, etc).

FIGURE 8. SHORTCUT KEYS POPUP MENU



Chapter 3. Developing a Drainage and Utilities File

The proposed drainage and utilities models need to be created in unique files, as a given file can only have one active drainage and utilities model at a time. It is recommended to break the project out into sections to help with production and to enhance the performance of the software.

3.1 Objectives

At the conclusion of this chapter, participants will be able to:

1. Create a file in ORD and navigate to the Drainage and Utilities workflow.
2. Access and reference existing imagery and data maps via Bing, USGS Quad Maps and USGS StreamStats.

3.2 Exercise: File Creation

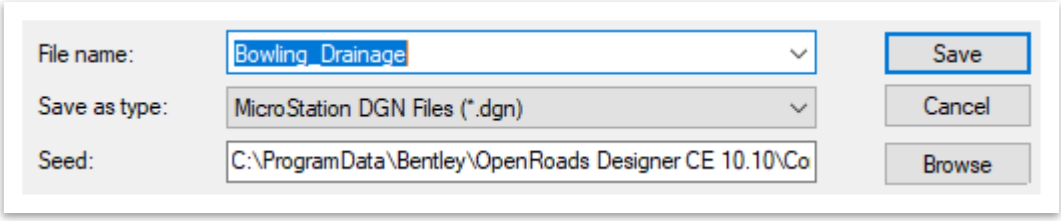
In this exercise, we will create a file utilizing the **SUDA_Training** workset, which will contain the drainage utility model and allow the user to review the SUDA modeling workflow. This file will be used throughout the remaining chapters of this manual.

Take Note!

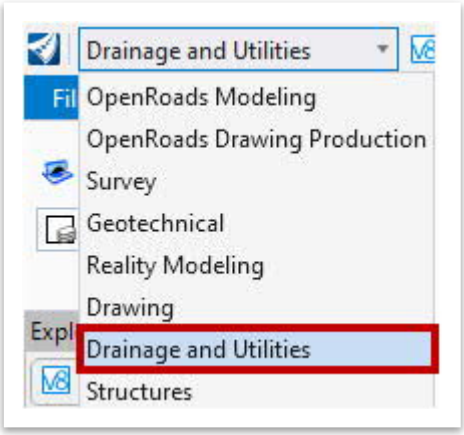
*On an actual project, remember to use the TDOT ORD naming convention when creating **all** design files. The file names used in this manual serve as general guidance for training purposes only. The procedures are intended to teach the necessary tools and TDOT's ORD workspace. For more information, refer to the **TDOT ORD File Naming Convention Standards** document.*

1. First, move the provided **class files** to the following location within File Explorer: **C:\ProgramData\Bentley\OpenRoads Designer CE 10.10\Configuration\WorkSpaces\TDOT_Standards\WorkSets\SUDA_Training\dgn**. These files will be utilized later in the manual.
2. Open ORD from your desktop. The **TDOT_Standards** workspace should still be active after taking the Roadway Design I (ORD) training. Go ahead and select the **SUDA_Training** workset, which will be used for the duration of this manual.

3. Create a new file and name it **Bowling_Drainage**. Select the **TDOTSeed2D.dgn** and click **Save**. **Note:** By default, the software should save the file in the workset dgn subfolder: **C:\ProgramData\Bentley\OpenRoads Designer CE 10.10\Configuration\WorkSpaces\TDOT_Standards\WorkSets\SUDA_Training\dgn**.



4. The **Bowling_Drainage.dgn** is now created and will be referred to as the **design file** or **drainage utility file** throughout the manual.
5. Make sure that the **Drainage and Utilities** workflow is selected in the upper left corner of the ORD interface, which will be used throughout this manual.



Take Note!

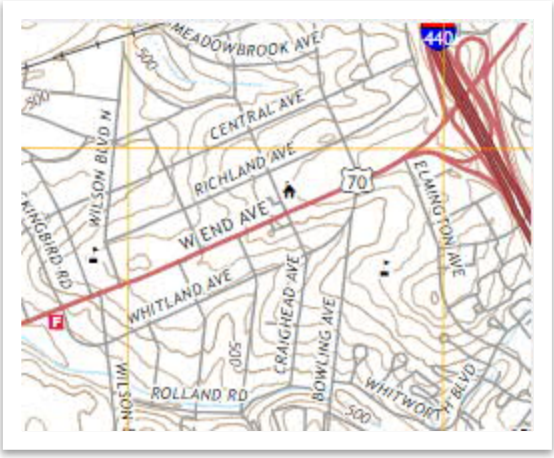
*The **Drainage and Utilities** workflow is selected as the active workflow only when working in the drainage design file or in a non-drainage utility file (e.g., Sanitary Sewer, Water, Gas, etc.). There are other workflows that should be used when manipulating roadway, terrain, or survey files.*

6. Go ahead and attach the **Bowling_Geometry.dgn** file as a reference using the **Coincident World** attachment method. By default, ORD should open to the **SUDA_Training\dgn** folder. Click **Fit View** and notice the geometry.

3.4 Lecture: USGS Quad Maps

The **USGS Quad Maps** for a given project location can be accessed here: https://store.usgs .gov/map-locator. This tool allows the user to select from current or historical USGS quad maps as a PDF. Figure 9 shows an example of a USGS topo map imported into ORD.

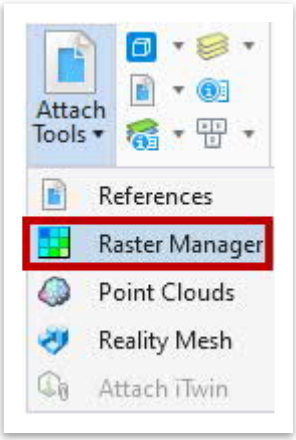
FIGURE 9. SAMPLE QUAD MAP



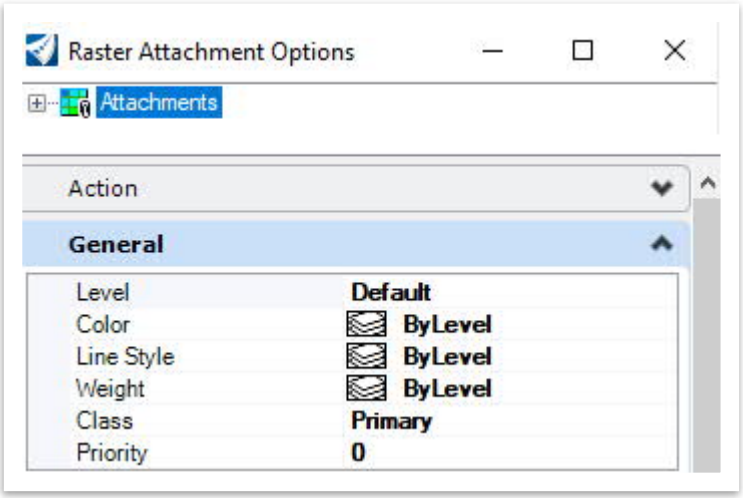
3.4.1 Exercise: Georeferencing a USGS Quad Map PDF

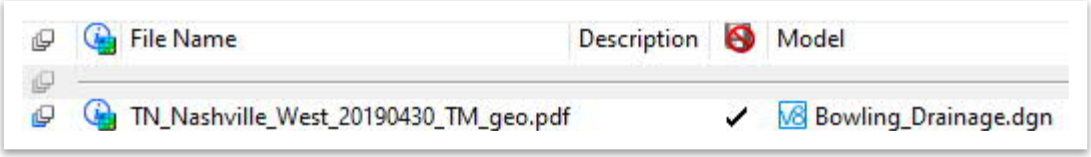
In this exercise, we will open the provided Quad Raster format (Bowling Avenue; Nashville, TN; latitude: 36.120803, longitude: -86.82618) and rectify the projection using known locations around the project location. We will continue to utilize the same **Bowling _Drainage.dgn** file.

1. First, open the **Raster Manager** window (**Drainage and Utilities >> Home >> Primary >> Attach Tools**).

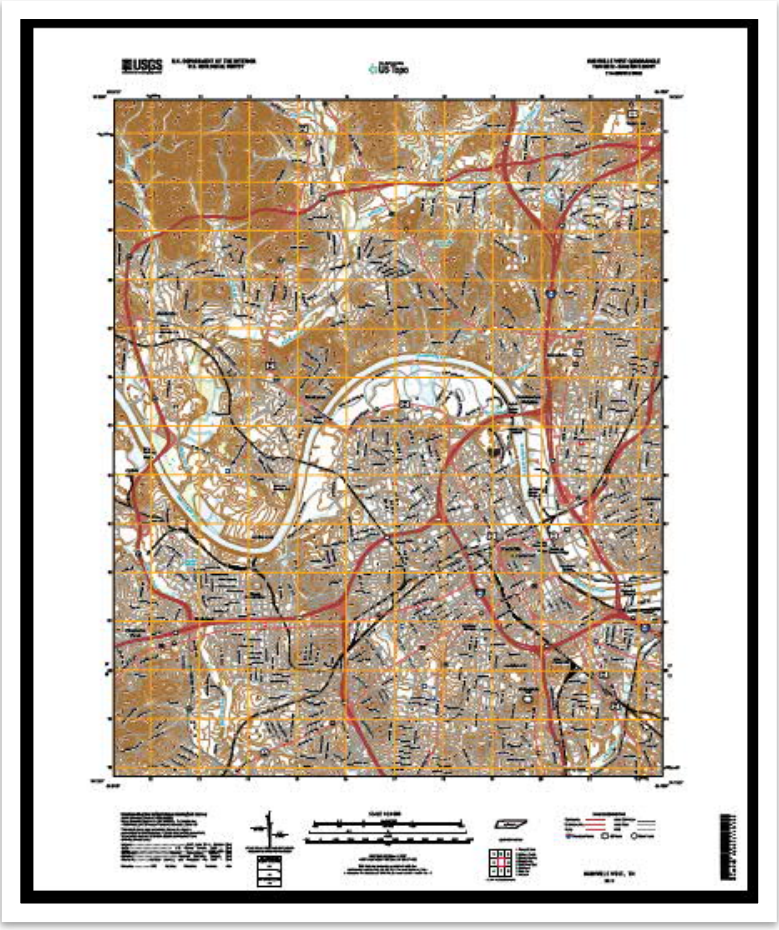


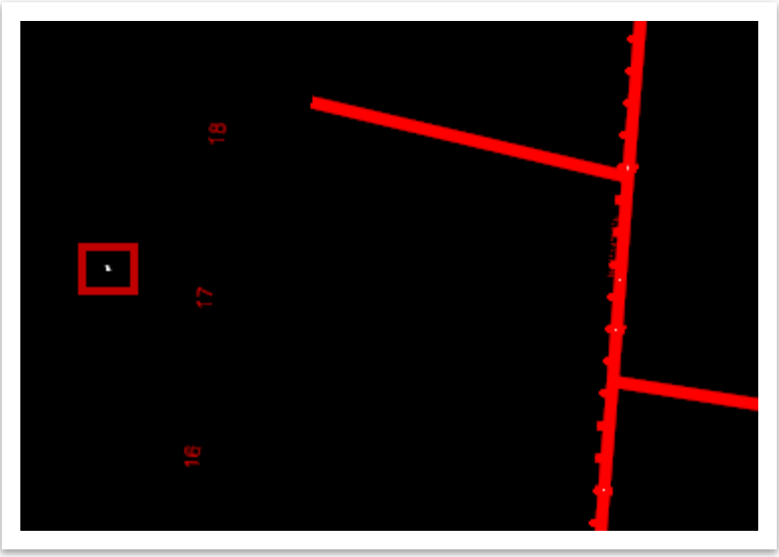
2. Within the **Raster Manager** window, go to **File >> Attach >> Raster** and select the **TN_Nashville_West_20190430_TM_geo.pdf** file located within the **SUDA_ Training\dgn** folder. Leave the default settings as-is and then click **Attach**.





3. Zoom out and locate the raster. Within the **Raster Manager** window, right click on the PDF and select **Move**. Go ahead and move the PDF closer to where the geometry file is located, as highlighted below.

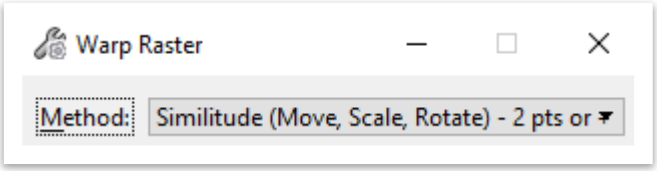




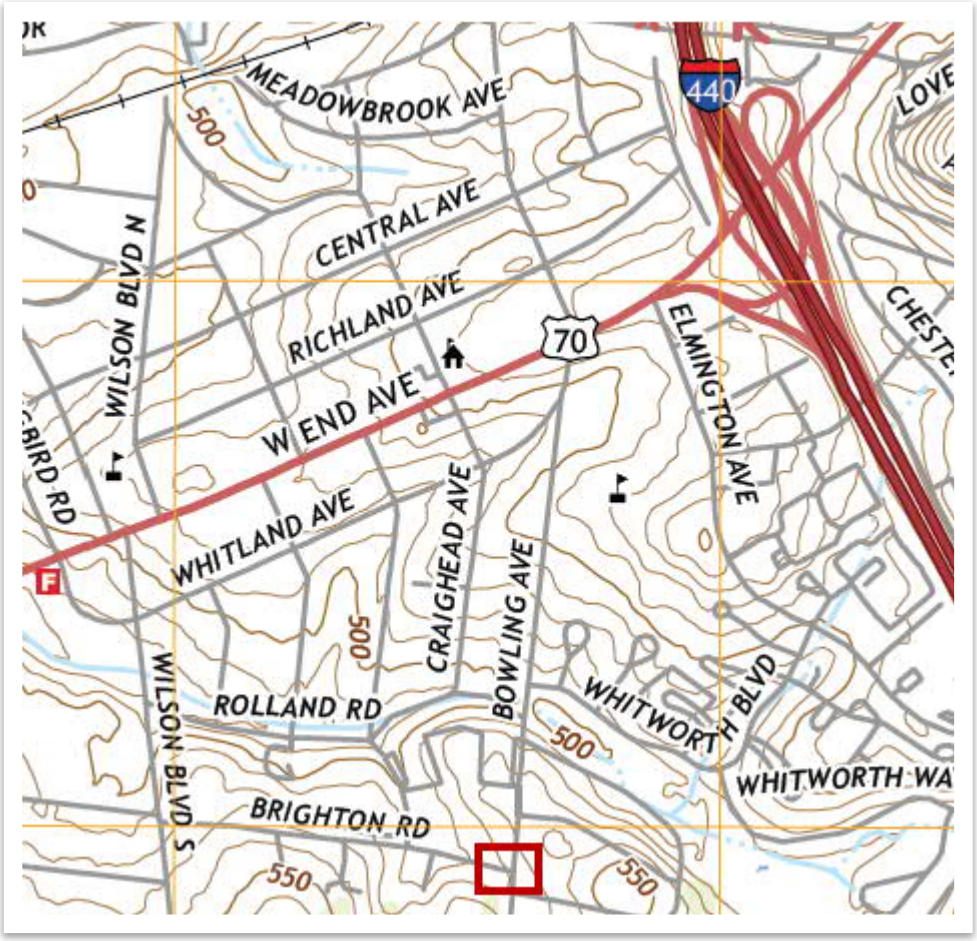
4. Now, set the **Background Map Type** back to **Hybrid**. We are now going to fix the projection of the PDF but we need two reference points. For this project we will use the **Brighton Rd** and **Rolland Rd** intersections, as highlighted below.

5. Since there is not any geometry linework at the Rolland Rd intersection, we need to draw a line so we can snap to the location easier. Open the **Place Smart Line** tool (**Drainage and Utilities >> Drawing >> Placement**) and draw a line like what is shown below.

6. Next, within the **Raster Manager** window right click the PDF again and select **Warp**. We will use the **Similitude (Move, Scale, Rotate) – 2 pts or +** method.



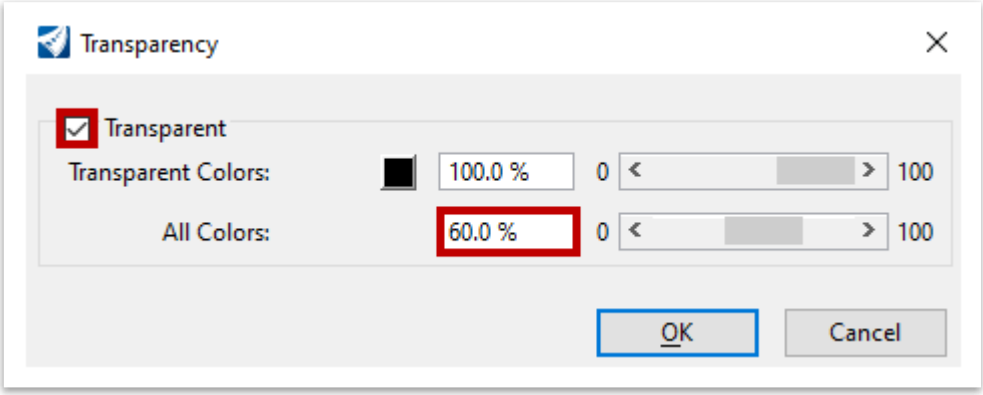
7. Notice the prompt in the lower left corner: **Enter image point**. Zoom into the PDF and left click in the general area of the **Brighton Rd** and **Bowling Ave** intersection.



8. Notice the next prompt in the lower left corner: **Enter monument point**. Left click the equivalent point within the referenced geometry. **Note:** Give the software a minute to process. The quad map is quite large and computer processors vary.

9. Repeat the previous two steps for the **Rolland Rd** intersection. Right click to accept and then give the software a second to process. You should see the PDF align with the geometry.

10. To visualize the PDF relative to the imagery, we need to adjust the transparency. Within the **Raster Manager** window, right click on the PDF and select **Transparency**. Toggle on **Transparent** and then change the **All Colors** percentage to **60%** and click **OK**.



11. Now, the projection of the raster file is fixed against the background imagery.

12. Once you have reviewed, go ahead and turn off the raster and close the **Raster Manager** window. Set the **Background Map Type** back to **None** within **View Attributes**. You can also delete the smartline at the **Rolland Rd** intersection. **Note:** To turn off the raster, open the **Raster Manager**, select the raster (TN_Nashville_West_20190430_TM_geo.pdf), and then turn off **View 1**.

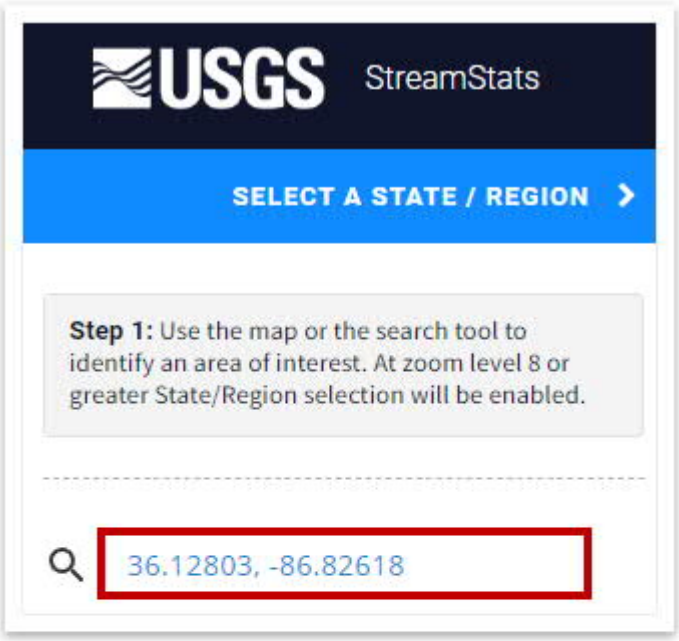
3.5 Lecture: Using the USGS StreamStats GIS Program

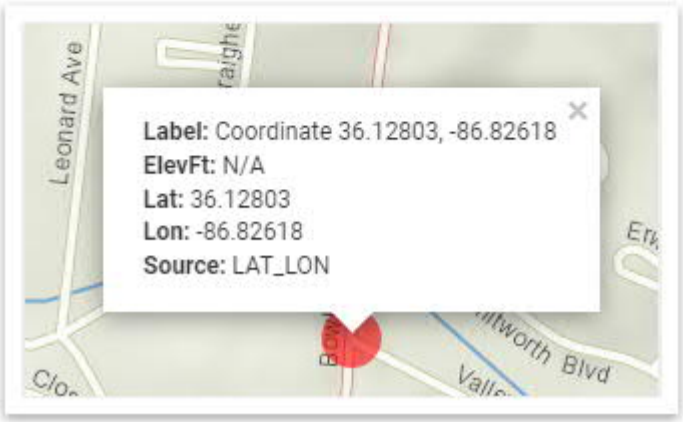
The **USGS StreamStats** can be accessed here: https://streamstats.usgs.gov/ss/. StreamStats utilizes USGS gage stations and regression equations to develop various flow statistics for a selected point discharge within a watershed or drainage basin. Refer to Chapter 6 in the Survey (ORD) Manual for a further discussion on StreamStats.

3.5.1 Exercise: Obtaining USGS StreamStats Flow Data Maps

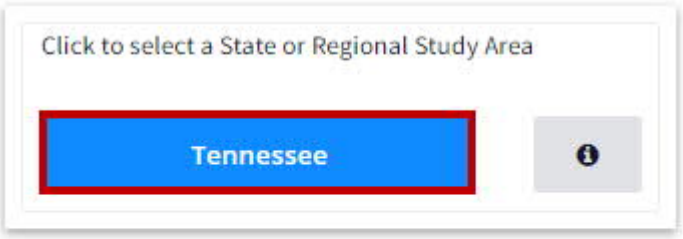
In this exercise, we will learn how to navigate the U.S. Geological Survey's StreamStats GIS program to obtain flow data maps, which will be used as a background. We will continue to utilize the same **Bowling_Drainage.dgn** file.

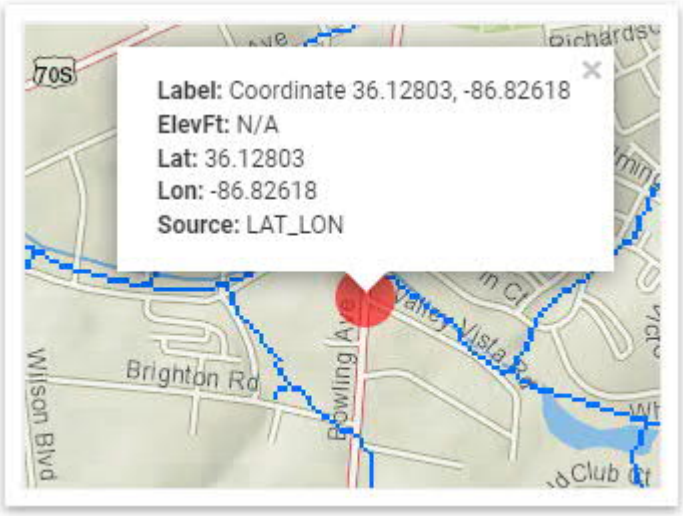
1. First, open the following link: https://streamstats.usgs.gov/ss/. In the upper left corner, key-in the coordinates for Bowling Avenue (**36.12803, -86.82618**) and then click **Enter**.



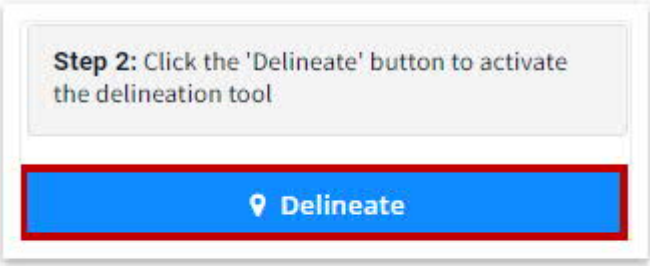


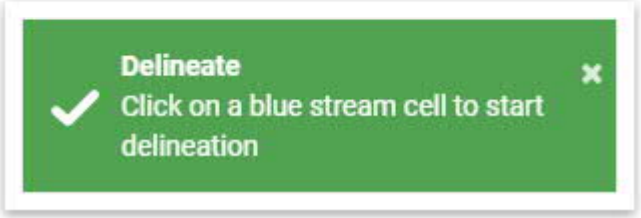
2. Next, click **Tennessee** to select the state and notice the blue National Hydro lines appear.



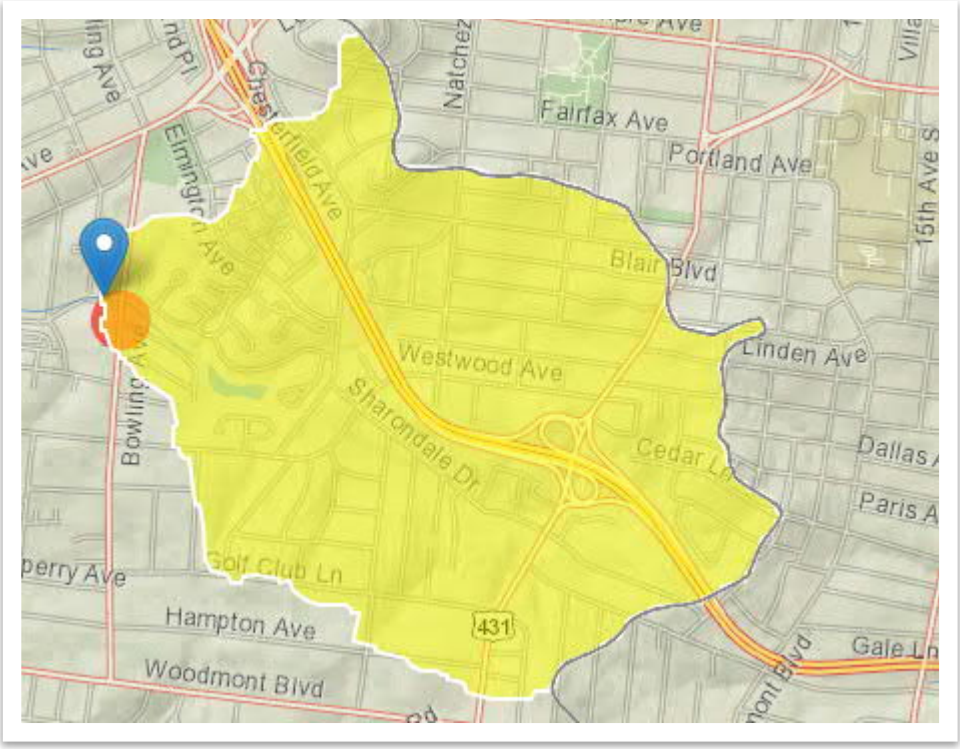


3. Now, click **Delineate** to activate the delineation tool. Notice the green prompt in the lower right corner: **Click on a blue stream cell to start delineation**.

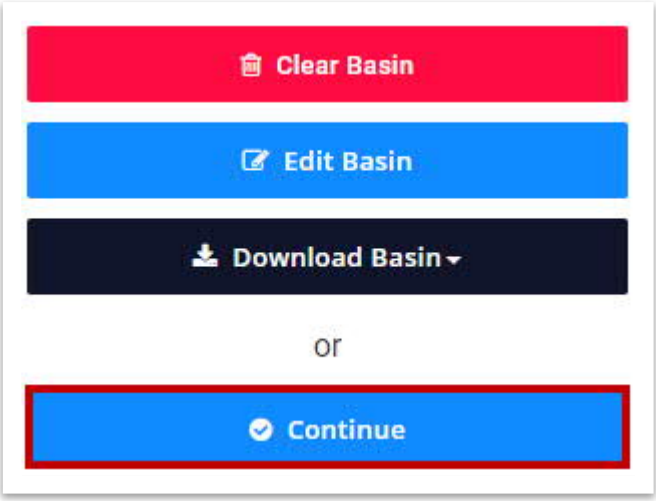




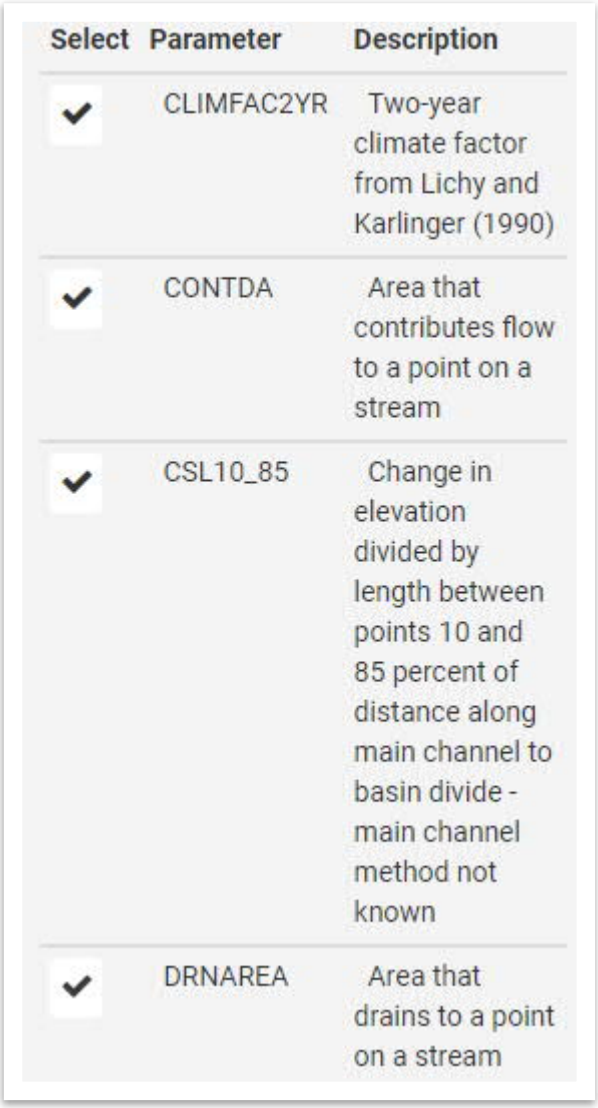
4. Pick any point along the stream crossing **Bowling Avenue** to delineate the watershed. Give the software a minute to validate the point and complete the delineation.



5. Click **Continue** and let it query the regression regions with your basin as it builds the report. **Note:** You can also download the basin as a **ShapeFile** and import into ORD.



6. We can now select a scenario or specific basin characteristics. For this exercise, expand **Basin Characteristics** and select the following **Parameters** and then click **Continue**.
    a. CLIMFAC2YR
    b. CONTDA
    c. CSL10_85
    d. DRNAREA



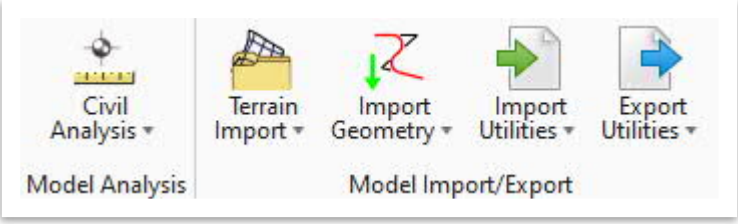
Select	Parameter	Description
✔	CLIMFAC2YR	Two-year climate factor from Lichy and Karlinger (1990)
✔	CONTDA	Area that contributes flow to a point on a stream
✔	CSL10_85	Change in elevation divided by length between points 10 and 85 percent of distance along main channel to basin divide - main channel method not known
✔	DRNAREA	Area that drains to a point on a stream

7. Lastly, click **Open Report** to view your report on-screen. You could then print or download the report and save it in the application location. For this exercise, go ahead and close the report and then close the USGS StreamStats website. **Note:** The report could be used as a background file in ORD.

Chapter 4. Terrain Setup and Management

The **Model Analysis** and **Model Import/Export** tools assist with terrain creation and analyzation (Figure 10). Terrain can be managed in various ways. The tools below will allow you to import terrain from various types such as las, xyz, and tin generated by V8i. These tools can also be used to import drainage and utilities models. The appropriate tool needs to be selected since the options do distinguish between drainage and utilities.

FIGURE 10. TERRAIN MODEL TOOLS



4.1 Objectives

At the conclusion of this chapter, participants will be able to:

1. Create a terrain from LAS files and clip.
2. Create a complex terrain.
3. Review the calculated features of a terrain.
4. Analyze different elements of a terrain (point, between points, trace slope, pond).
5. Import and export drainage utility files.

4.2 Lecture: Terrain Models

Because drainage areas oftentimes extend outside the surveyed terrain, an **extended TIN surface** is needed to provide the additional terrain data within the drainage areas.

Within ORD, **contour lines can be displayed from a surface but not drawn into a file**. This means that whenever a user changes the view, the contours will re-process and then re-display. While this is not a concern when working with smaller terrains, it does present an issue when working with larger extended terrains. At a minimum, the extended terrain will cause the software to lag during re-processing and likely result in the terrain file crashing. Therefore, it is recommended to **turn off the contour display within a terrain until the terrain has been trimmed to its final size**. In addition, never select a feature definition that will automatically display contours when loading or complexing terrains.

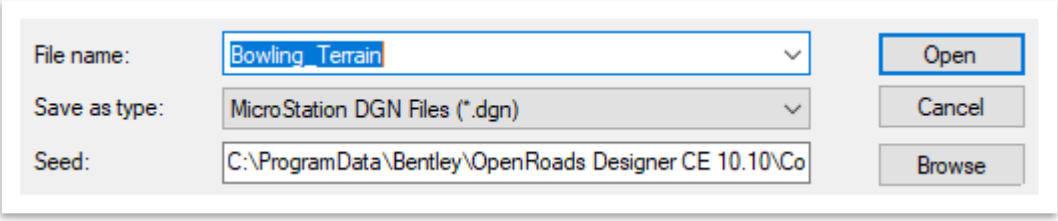
Take Note!

*If you still experience software crashes while loading contour features, it is recommended to export the terrain as a **LandXML** file. Then, use **ESRI ArcGIS** to process the data to a contour set.*

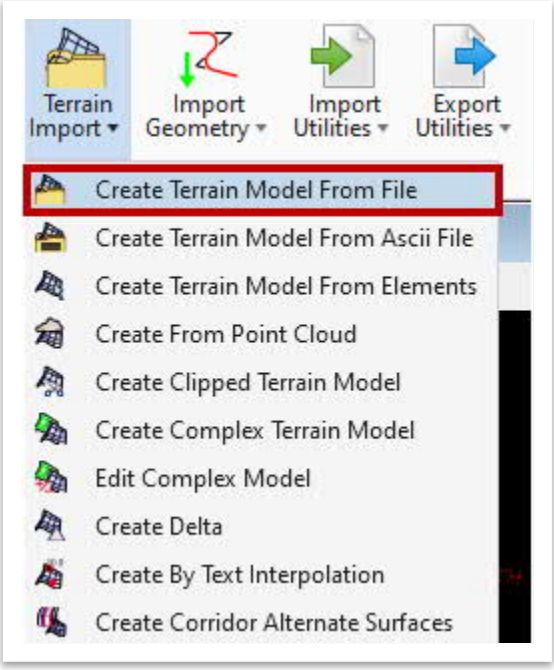
4.2.1 Exercise: Terrain Model Creation – LAS Files

In this exercise, we will create a terrain model from LiDAR tiles (LAS files) that will be built upon and used in this manual.

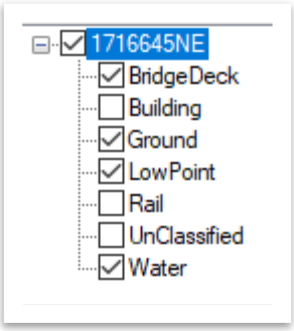
1. Create a new file and name it **Bowling_Terrain**. Select the **TDOTSeed3D.dgn** and click **Save**.



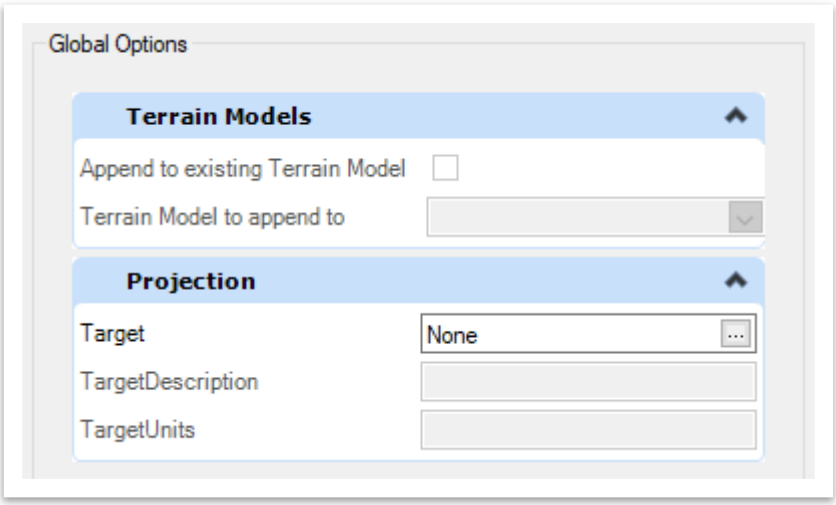
2. Click **Fit View** and notice the embedded survey data text. This is seen because the **TDOTSeed3D** file was utilized. Go ahead and delete all survey data, which is on the three **SUR** levels below, since it is not applicable for this file.
    - SUR - CTRL - Text
    - SUR - Project Information and Notes
    - SUR - UTL - Owners
3. Attach the **Bowling_Geometry.dgn** file as a reference using the **Coincident World** attachment method and then click **Fit View** once again.
4. Next, we will import the **.las** files to create the terrain. Open the **Create Terrain Model From File** tool (**Drainage and Utilities >> Home >> Model Import/ Export >> Terrain Import**).



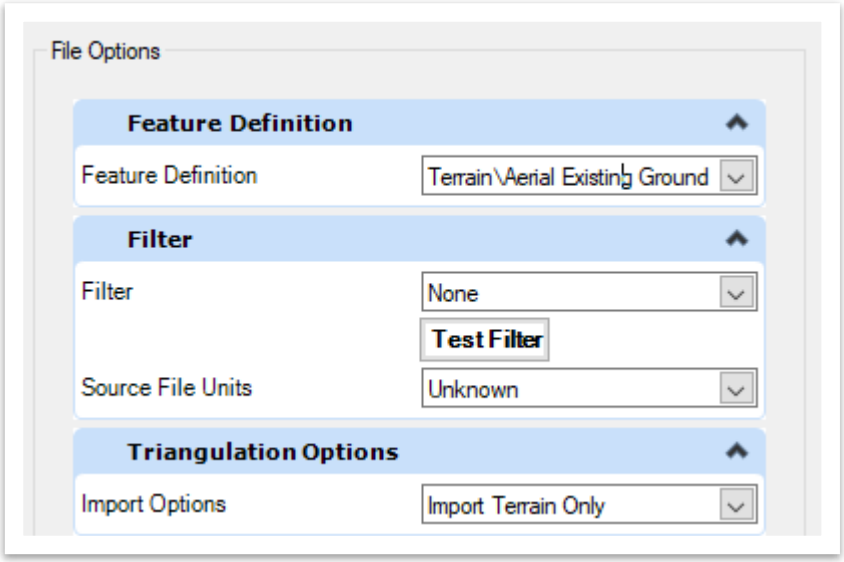
5. Select the **1716645NE.las** file and then click **Done** and give the software a minute to load. **Note:** To prevent the software from crashing, import one tile at a time. The process on how to generate the .las files is covered in **Appendix F**.
6. Within the **Import Terrain Model(s)** dialog box, uncheck the **Building**, **Rail** and **UnClassified** filters.



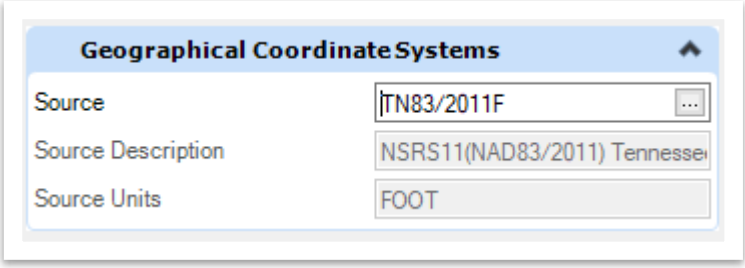
7. Under **Global Options >> Projection**, make sure the **Target** is set to **None**.



8. Under **File Options**, select the following settings.
    a. **Feature Definition:** Terrain\Aerial Existing Ground
    b. **Filter:** None
    c. **Source File Units:** Leave as-is. This field will take care of itself and disappear once the geographic coordinate system is selected in the next step.
    d. **Triangulation Options:** Import Terrain Only

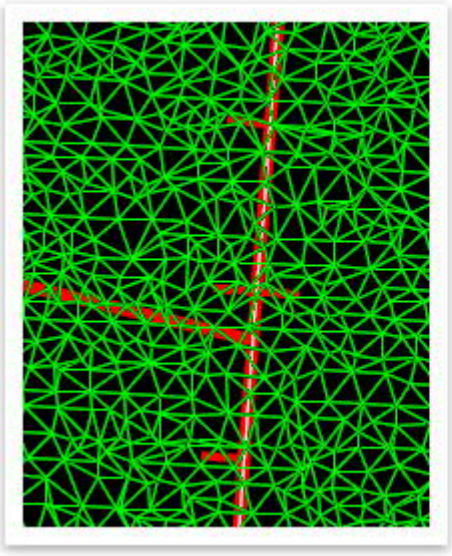


9. Under **Geographical Coordinate Systems**, click the ellipses next to the **Source** field. You should already have the correct coordinate system saved as a Favorite (**TN83/2011F – NSRS11 (NAD83/ 2011) Tennessee State Plane Zone, US Foot**). If not, you can browse to it here: **Library >> Projected (northing, easting, …) >> North America >> United States of America >> Tennessee**.



10. Click **OK** and then click **Import** and give the software a minute to process. Close the **Import Terrain Model(s)** dialog box.

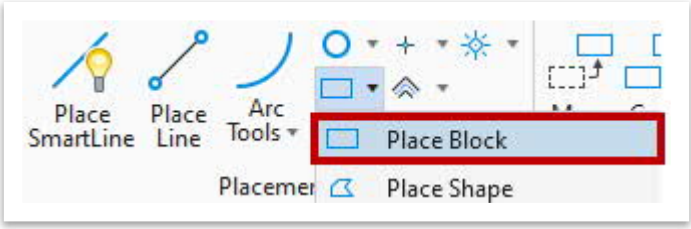
11. Repeat Steps 4-10 to import the **1716653SE.las** file. Close the **Import Terrain Model(s)** dialog box. By default, the triangles will be turned on. Select the boundary of each tile and open the **Properties** within the heads-up display. Go ahead and turn the **Triangles** off. **Note:** If you wanted to turn on the contours, it would take some time due to the size of the tiles and would slow the software down considerably.



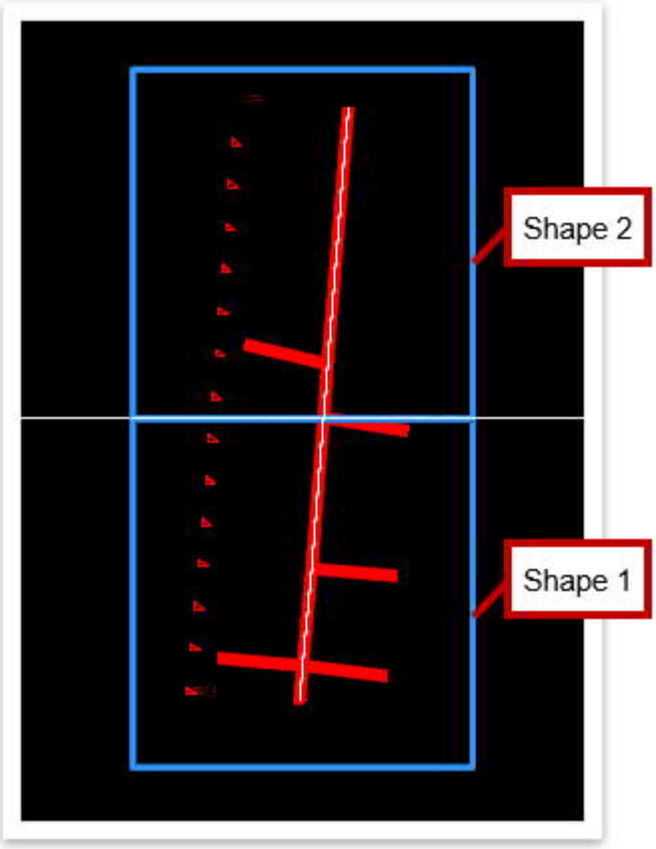
4.2.2 Exercise: Terrain Model Creation – Clipped Terrain Model

In this exercise, we will create clipped terrain models from the imported terrain data in the previous exercise. We will continue to utilize the same **Bowling_Terrain.dgn** file.

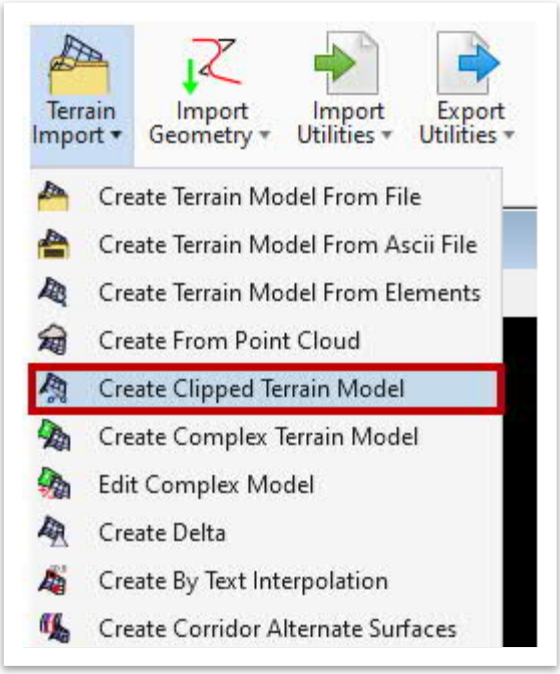
1. To decrease file size and improve design efficiency, we will now **clip** the LiDAR tiles to just beyond the project area extents. Open the **Place Block** tool (**Drainage and Utilities >> Drawing >> Placement >> Polygon Tools**).



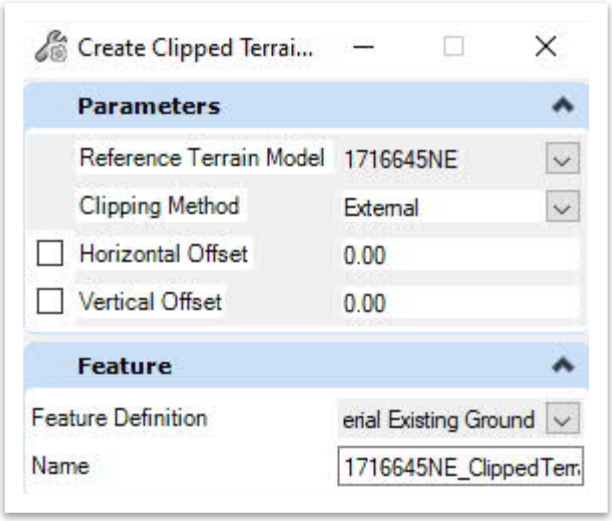
2. Draw two shapes around the **outer extent** of the project's area: **one for <u>each</u> tile** so that it is completely enclosed, as shown below. Within the **Properties**, go ahead and update both shapes <u>color</u> and <u>line weight</u> to **4** for better visibility. **Note:** Your shapes do not need to match exactly to what is shown. Also, drawing a shape for each tile rather than one overall shape that spans both tiles will allow the file to process more quickly.



3. Next, open the **Create Clipped Terrain Model** tool (**Drainage and Utilities >> Home >> Model Import/Export >> Terrain Import**).

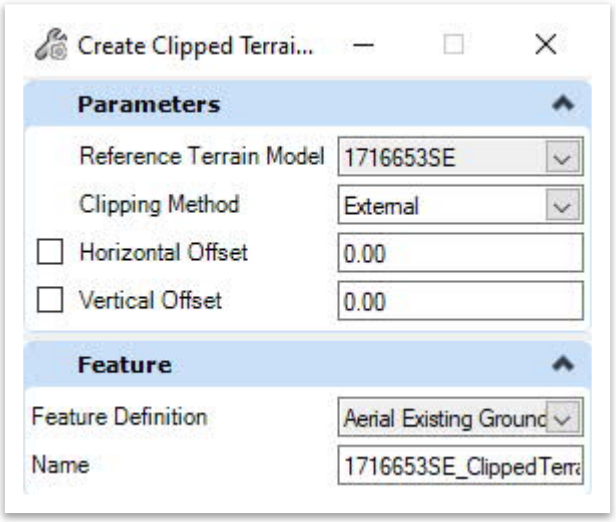


4. We will first clip the **1716645NE** tile. Within the **Create Clipped Terrain Model** dialog box, select the following settings.
    a. **Reference Terrain Model:** 1716645NE
    b. **Clipping Method:** External
    c. **Horizontal/Vertical Offsets:** 0.00
    d. **Feature Definition:** Terrain\Aerial Existing Ground
    e. **Name:** 1716645NE_ClippedTerrain



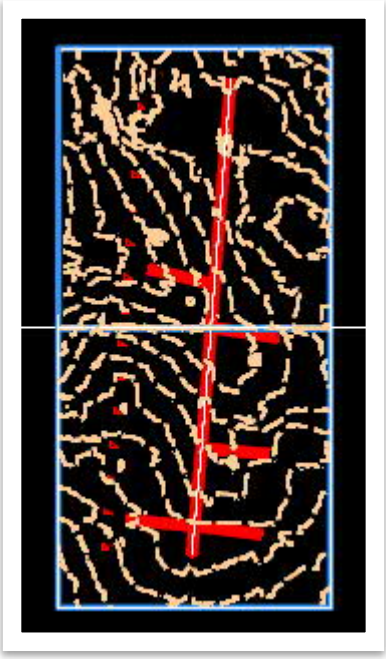
5. Notice the cursor prompt: **Locate Clipping Element**. Left click to select the bottom clipping shape you created in Step 2 and then **right** click to reset.

6. Left click to accept the remaining prompts until you get to **Locate Reference Terrain Model Element**. Within the **Create Clipped Terrain Model** dialog box, select the following settings.
    a. **Reference Terrain Model:** 1716645SE
    b. **Clipping Method:** External
    c. **Horizontal/Vertical Offsets:** 0.00
    d. **Feature Definition:** Terrain\Aerial Existing Ground
    e. **Name:** 1716645SE_ClippedTerrain

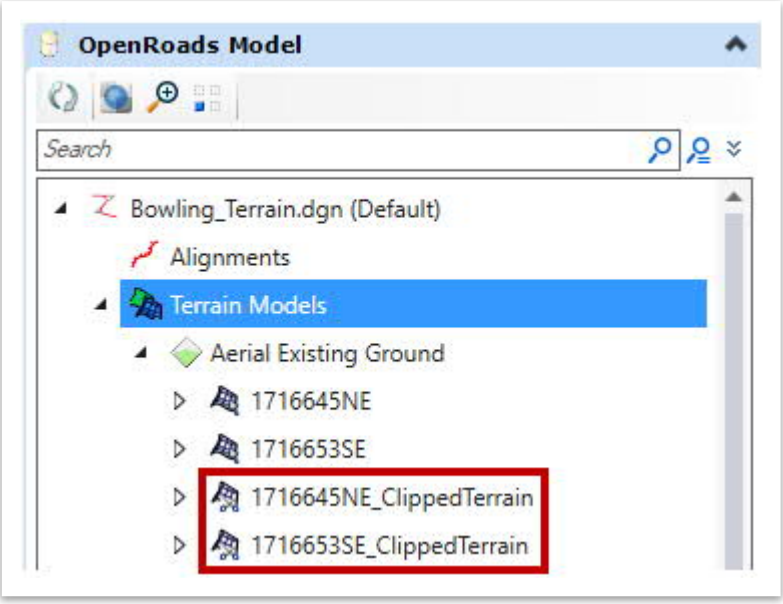


7. Notice the cursor prompt: **Locate Clipping Element**. Left click to select the top clipping shape and then **right** click to reset. Left click to accept the remaining prompts. Right click to close the tool.

8. By default, the triangles will be turned on within the clipped terrain. Let's now view the major contours. Select each clipped terrain boundary and open the **Properties** within the heads-up display. Turn off the **Triangles** and turn on the **Major Contours**. Once reviewed, go ahead and turn the contours back off.



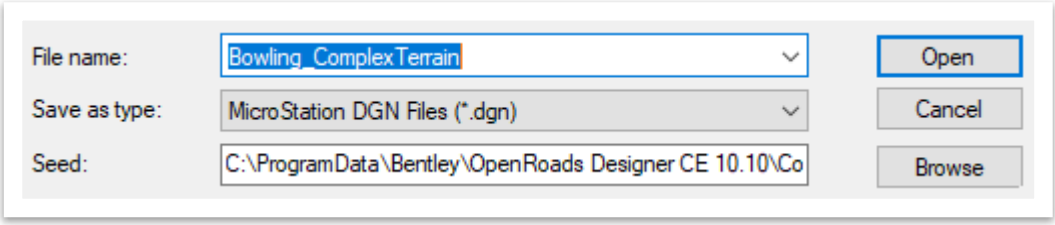
9. Within the **Explorer**, notice that the two clipped terrains are now listed under **Open Roads Model >> Bowling_Terrain.dgn (Default) >> Terrain Models >> Aerial Existing Ground**.



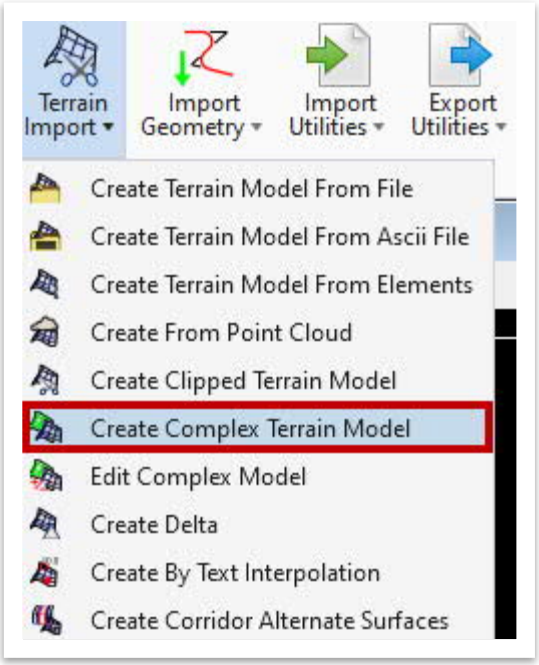
4.2.3 Exercise: Terrain Model Creation – Complex

In this exercise, we will create a complex terrain model by appending the clipped terrains and then create a final extended terrain model.

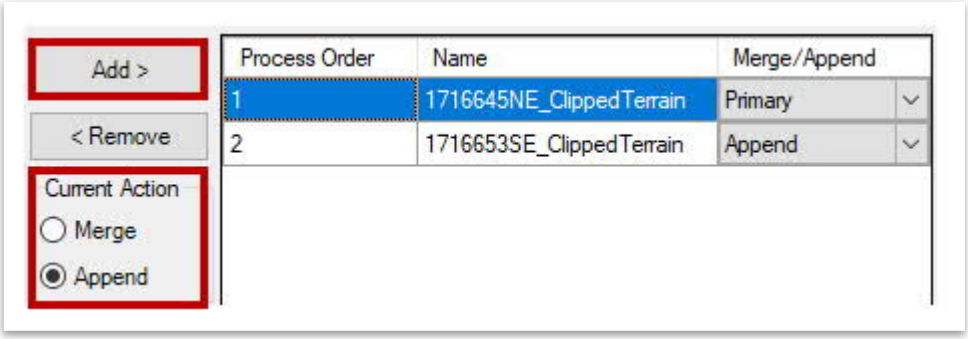
1. Create a new file and name it **Bowling_ComplexTerrain**. Select the **TDOTSeed 3D.dgn** and click **Save**.



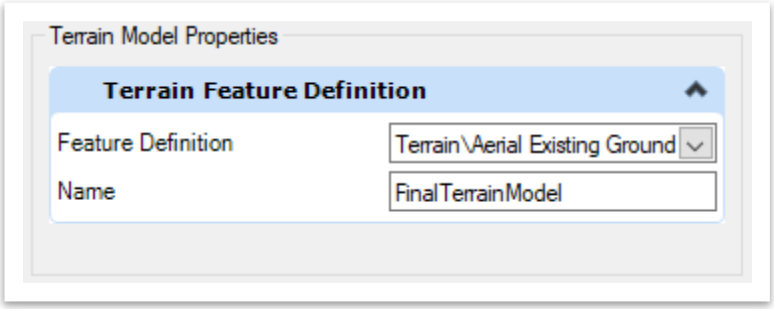
2. Click **Fit View** and notice the embedded survey data text. Once again, this is seen because the **TDOTSeed3D** file was utilized. Go ahead and delete all survey data, which is on the three **SUR** levels below, since it is not applicable for this file.
    - SUR - CTRL - Text
    - SUR - Project Information and Notes
    - SUR - UTL – Owners
3. Attach the **Bowling_Geometry.dgn** file and the **Bowling_Terrain.dgn** file as references using the **Coincident World** attachment method. Click **Fit View**.
4. Next, we will **append** the two clipped terrain models from the previous exercise to create one **complex** model. Open the **Create Complex Terrain Model** tool (**Drainage and Utilities >> Home >> Model Import/Export >> Terrain Import**).



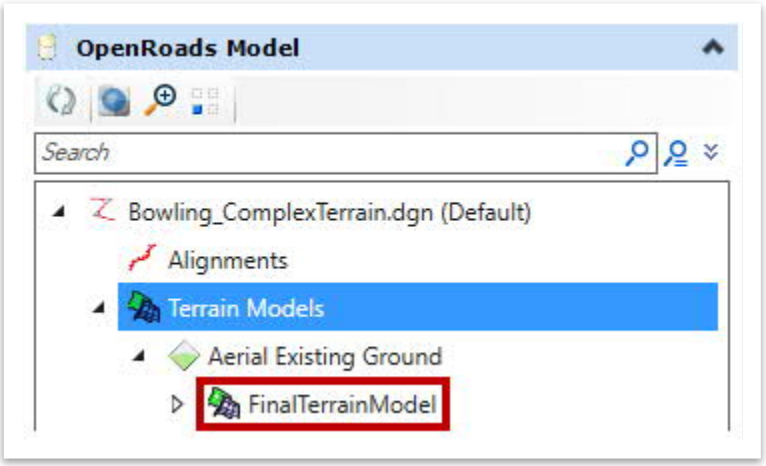
5. Notice that the **Create Complex Terrain Model** dialog box appears showing the original terrains plus the clipped terrains on the left side. Select the two clipped terrains (**1716645NE_ClippedTerrain** and **1716653SE_ClippedTerrain**) and **Add** them to the right side. Set the **Current Action** to **Append**. Leave the **Merge/Append** options as shown below.



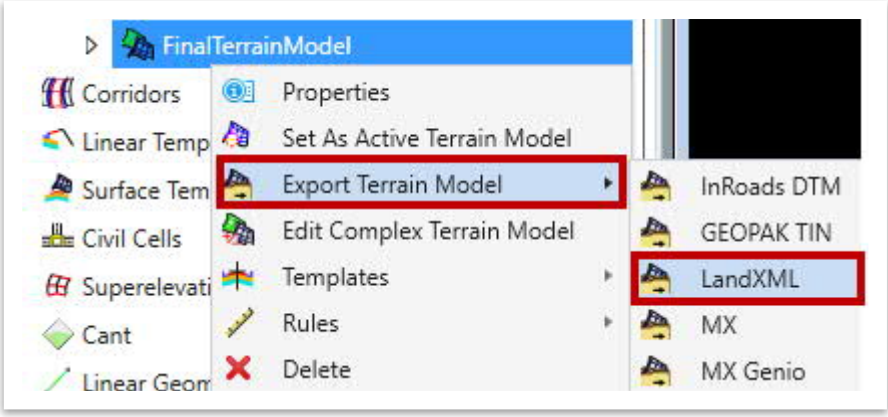
6. Under **Terrain Model Properties**, select the following settings and then click **Finish**. By default, the triangles will turn on by default, so turn them **off**.
   a. **Feature Definition:** Terrain >> Aerial Existing Ground
   b. **Name:** FinalTerrainModel



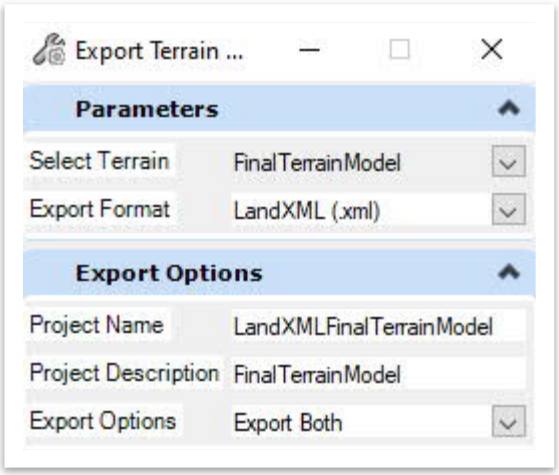
7. Within the **Explorer**, notice that the **FinalTerrainModel** is now listed under **Open Roads Model >> Bowling_ComplexTerrain.dgn (Default) >> Terrain Models >> Aerial Existing Ground**.



8. We will now export the final combined terrain model to a **LandXML**. Right click **FinalTerrainModel** within the **Explorer** and select **Export Terrain Model >> LandXML**.



9. Within the **Export Terrain Model** dialog box, select the following settings.
    a. **Select Terrain:** FinalTerrainModel
    b. **Export Format:** LandXML (.xml)
    c. **Project Name:** LandXMLFinalTerrainModel
    d. **Project Description:** FinalTerrainModel
    e. **ExportOptions:** Export Both

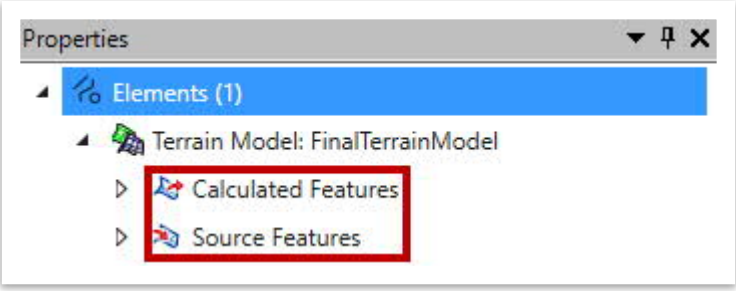


10. Left click through the prompts to accept. Within the **Export Terrain** window, browse to the **dgn** subfolder within your **SUDA_Training** workset and then click **Save**. Go ahead and close the **Export Terrain Model** dialog box.

4.3 Lecture: Calculated Features

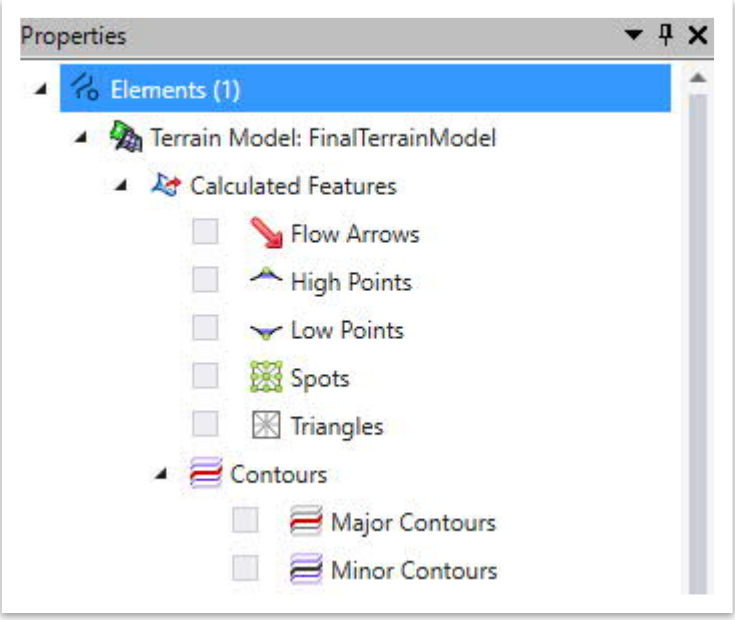
Once a terrain model has been created, you can click on the terrain border to open its properties and then display various features. Under the active terrain model there are two features: **Calculated Features** and **Source Features** (Figure 11).

FIGURE 11. CALCULATED AND SOURCE FEATURES



Under **Calculated Features** you can control the display of the flow arrows, high and low points, spots, and triangles, plus the contours (Figure 12). Each feature has a designated level so that the user can turn each feature on and off, as necessary.

FIGURE 12. CALCULATED FEATURES - CATEGORIES



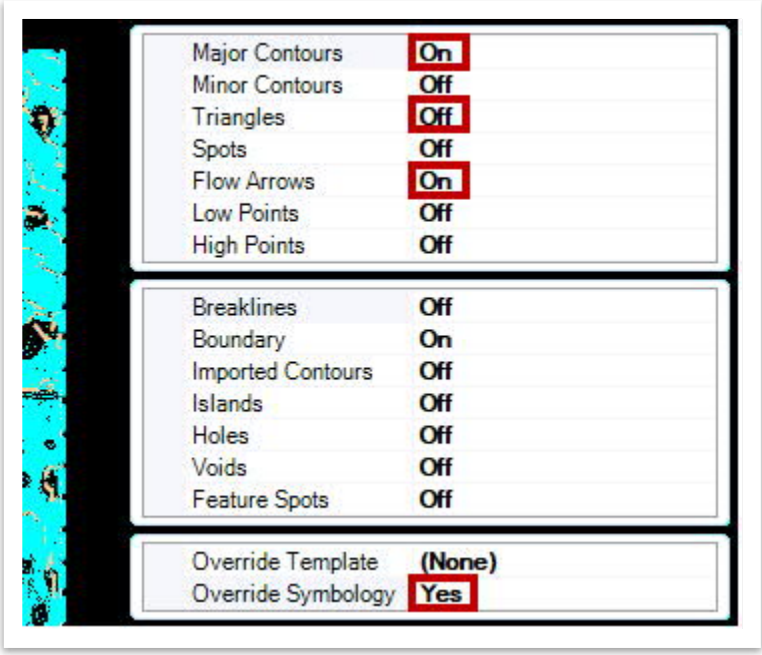
Take Note!

Flow arrows and contours will be the mostly used features. If there is an existing or proposed roadway that is part of the terrain, the high and low points are helpful.

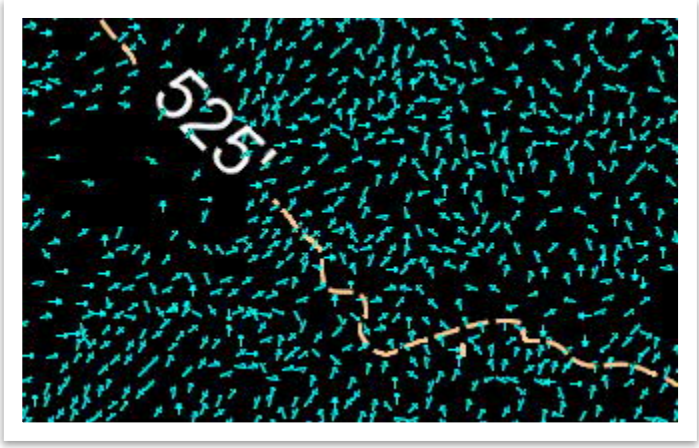
4.3.1 Exercise: Reviewing the Calculated Features

In this exercise, we will review some of the calculated features of the terrain that was previously created. We will open back up the **Bowling_Drainage.dgn** file.

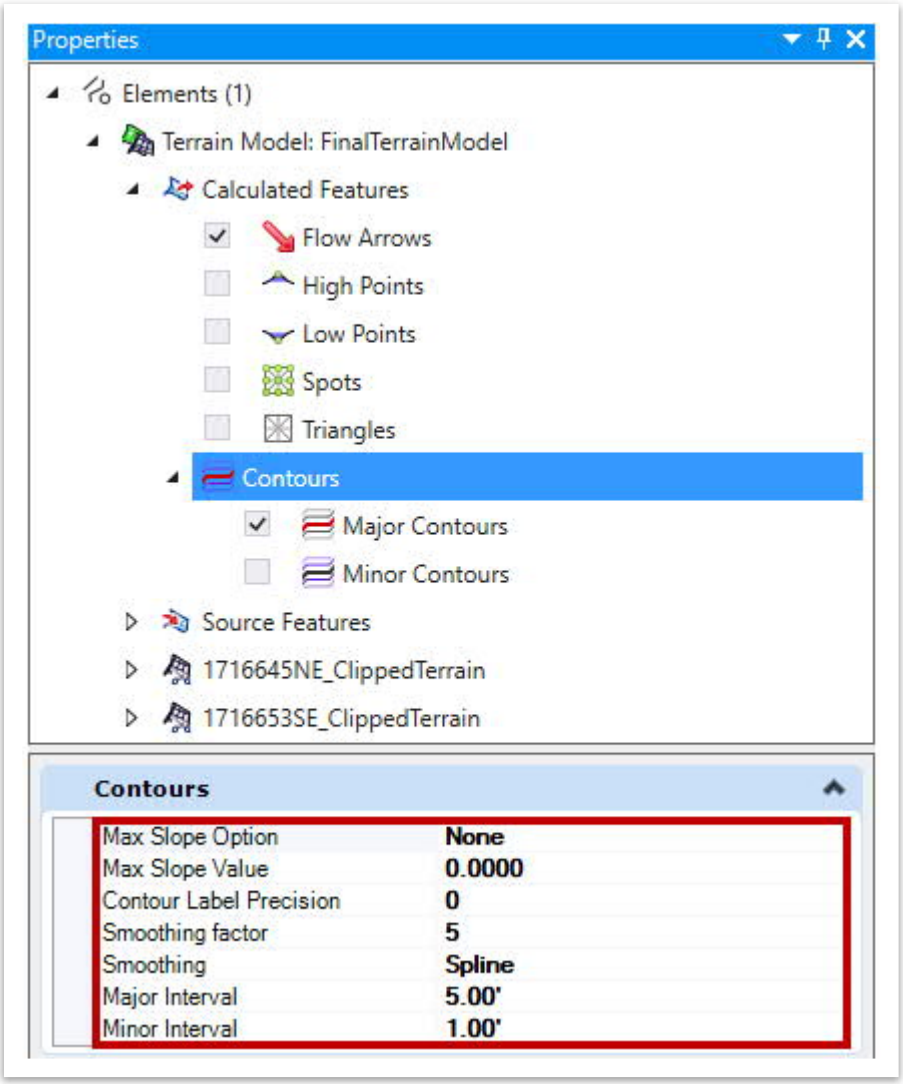
1. Attach the **Bowling_ComplexTerrain.dgn** file as a reference using the **Coincident World** attachment method. Click **Fit View** and then zoom in to the project area. Go ahead and turn on the **major contours** and **flow arrows** for the complex terrain. **Note:** You will need to set the **Override Symbology** to **Yes** before you can manipulate the terrain options within the heads-up display.



2. The density of the flow arrows is based on triangulated density of the terrain data. **Note:** If you want to change the density, you can create different terrain dgn files with various densities.



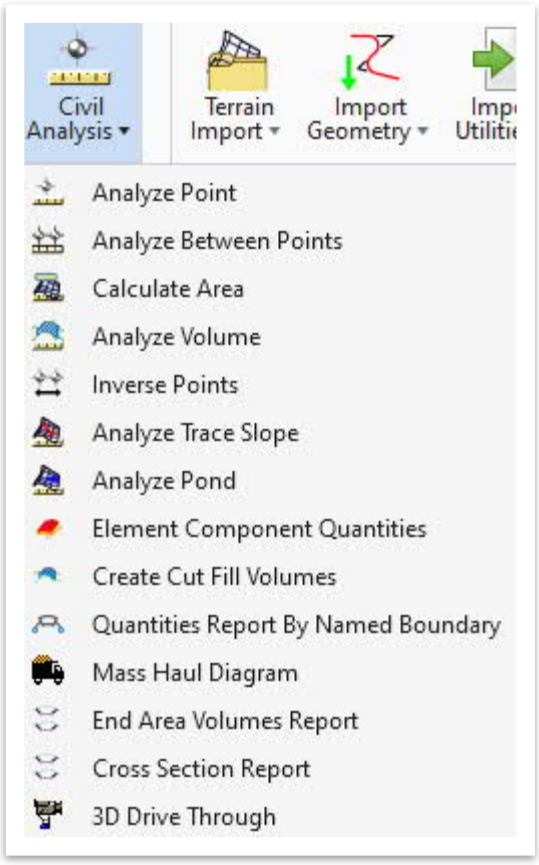
3. In addition to flow arrows, another one of the calculated features is **contours**, which are broken into two categories: **Major** and **Minor**. Unlike the flow arrows, the contour intervals can be changed in the file without generating new contours from scratch. Go ahead and select the complex terrain and then review the **Contours** properties.



4.4 Lecture: Terrain Analysis

The **Civil Analysis** tools (Figure 13) help analyze various terrain features to build a drainage model and are located here: **Drainage and Utilities >> Home >> Model Analysis**. The terrain must be referenced before these tools can be utilized when building the drainage model.

FIGURE 13. CIVIL ANALYSIS TOOLS



Analyze Point: Dynamically review information at any given point on a selected 2D or 3D design element, terrain model or mesh.

Analyze Between Points: Review elevation, slope, distance and delta information between two selected points of a terrain model or mesh.

Calculate Area: Measures the area defined by fenced limits located within the active terrain limits.

Analyze Volume: Compute volumes between two models or a model and plane, and optionally place the results in the file at a user-defined location.

Inverse Points: Computes and displays the distance and direction between each pair of specified points. Several types are supported: linear, arc, radial, perpendicular and by element.

Analyze Trace Slope: This tool dynamically traces a path on a terrain model or mesh surface and **works best in the 3D model**. Two Trace Methods are supported: Maximum Slope Trace and Constant Slope Trace.

- **Maximum Slope Trace:** This method traces slope in most direct (steepest) direction. **This method is most suitable for drainage analysis**.
- **Constant Slope Trace:** Traces a user-defined slope value. This is useful, for instance, where the user needs to determine a path for a channel or ditch (or another design element) if that feature has a maximum slope Design Constraint.

Analyze Pond: Traces a point downstream to a low point and fills it giving the volume, maximum depth, and maximum elevation. In addition, the pond delineation is graphically displayed.

Element Component Quantities: Report includes component (concrete, topsoil, shoulders, cut and fill volumes) surface area, volumes, units of measure, unit cost, and component and total cost. **This is not applicable to SUDA modeling**.

Create Cut Fill Volumes: Used to calculate cut and fill volumes between two surfaces, usually an existing terrain and a surface (terrain or mesh) and creates a 3D mesh solid with volume attributes. **This is not applicable to SUDA modeling**.

Quantities Report by Named Boundary: Used to calculate volume quantities for specific station ranges and/or sheet clipping boundaries. **This is not applicable to SUDA modeling**.

Mass Haul Diagram: Used to display earthwork volumes along an alignment, based on sectional volume computation. **This is not applicable to SUDA modeling**.

End Area Volumes Report: Used to calculate end Area Volume reports based off the 3D model and cross-section named boundary group. **This is not applicable to SUDA modeling**.

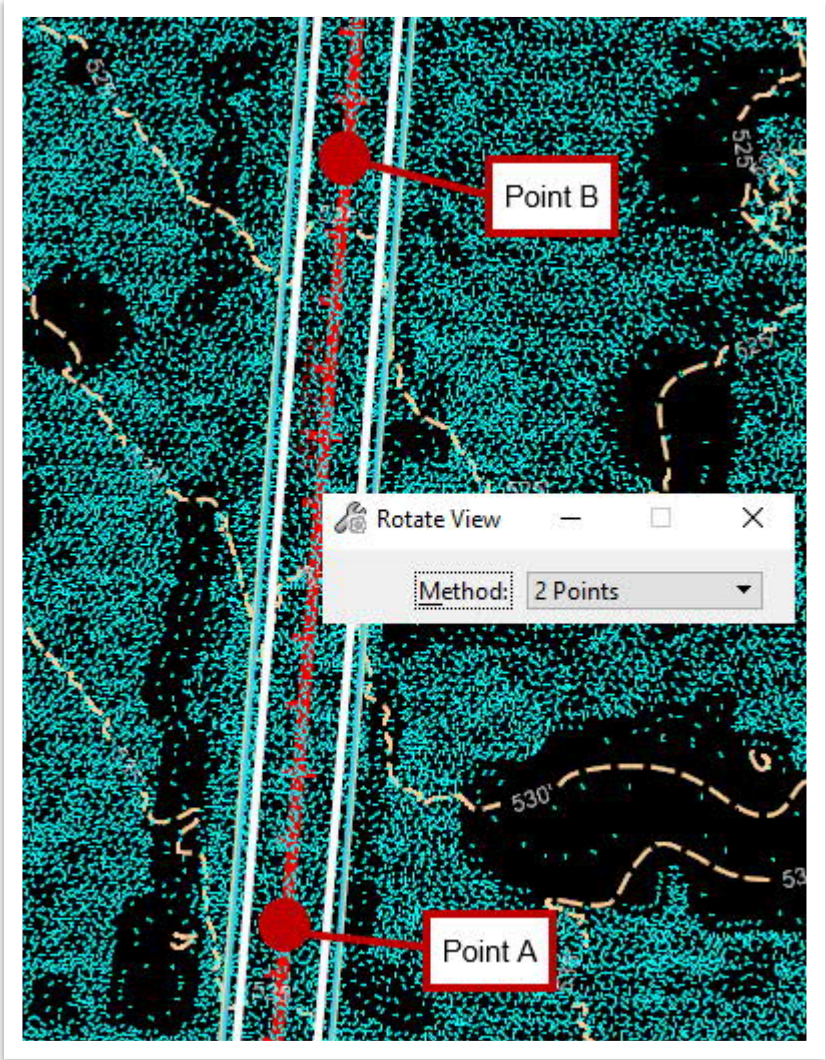
Cross-Section Report: Used to calculate cross-section area reports based off the 3D model and the cross-section named boundary group.

3D Drive Through: Gives the user the capability of viewing 3D models in a more realistic visualization.

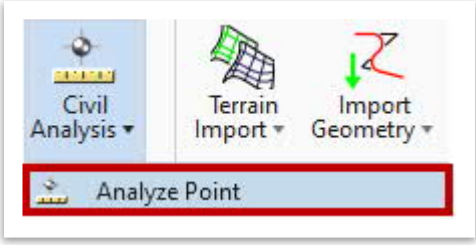
4.4.1 Exercise: Civil Tools – Analyze Point

In this exercise, we will analyze a single point on the referenced complex terrain and view the associated information. We will continue to utilize the **Bowling_Drainage.dgn** file.

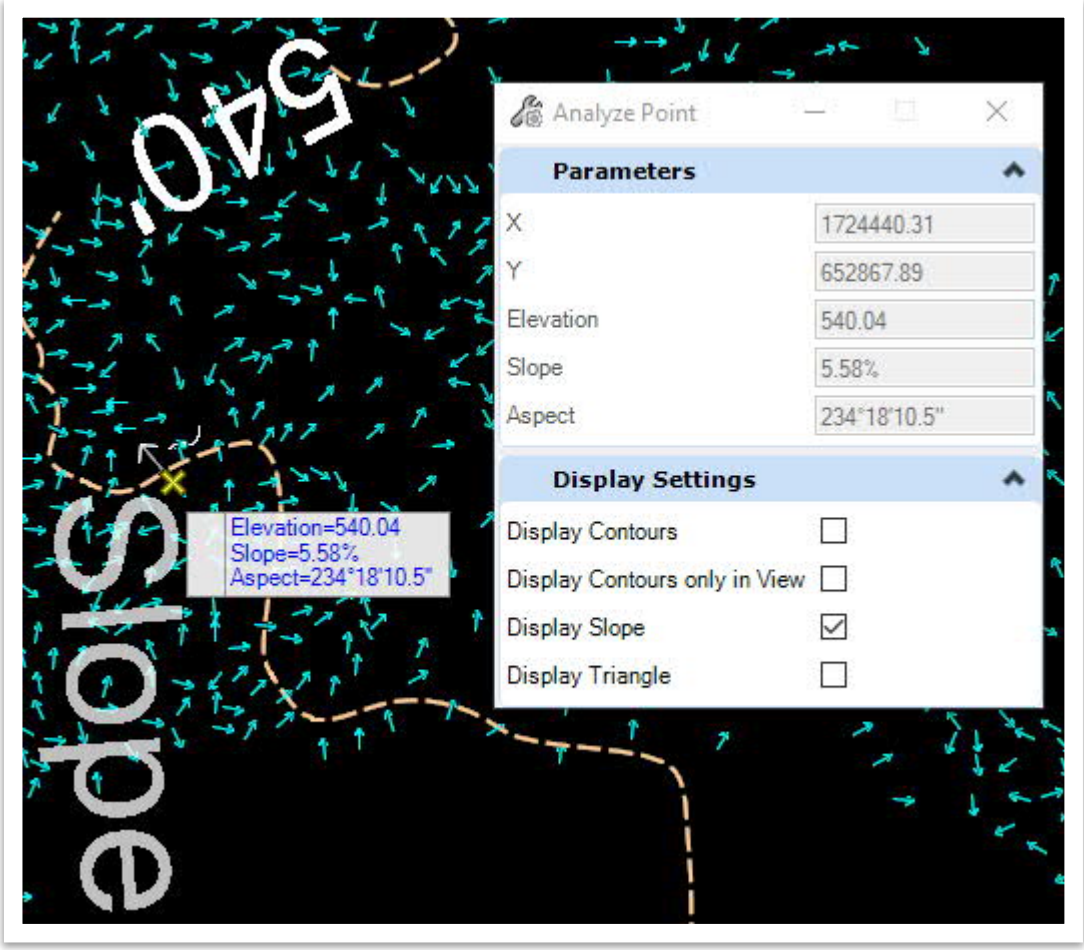
1. Attach the **Bowling_Corridor.dgn** file as a reference using the **Coincident World** attachment method. Click **Rotate View** and select the **2 Points** method. Rotate the view approximately **90** degrees in the **clockwise** direction by selecting 2 points along the imaginary **Y-axis**. Click anywhere near Point **A** and then click a second time near Point **B**.



2. Next, open the **Analyze Point** tool (**Drainage and Utilities >> Home >> Model Analysis >> Civil Analysis**).



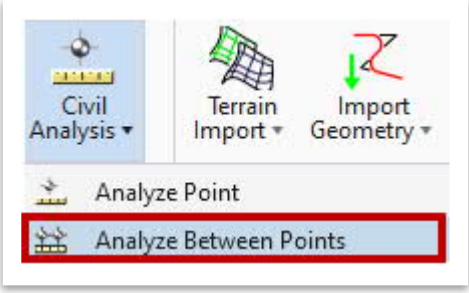
3. Notice the cursor prompt: **Select Element To Analyze Point**. Select any contour of the referenced complex terrain file. Notice the **Analyze Point** dialog box appears with an arrow pointing from the selected point. The **arrow** represents the **flow direction** from that point. The cursor also shows **Elevation**, **Slope** and **Aspect** values dynamically as the cursor is moved around. In the **Analyze Point** window, toggle on the **Display Slope** box (if necessary) to see the dynamic slope from any selected point. **Note:** The data shown in the screenshot will vary depending on where your cursor is.



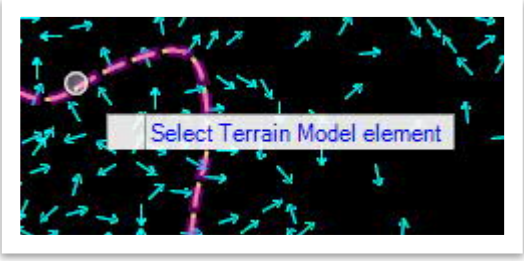
4.4.2 Exercise: Civil Tools – Analyze Between Points

In this exercise, we will select two points on the referenced complex terrain and view associated information between the two points. We will continue to utilize the same **Bowling_ Drainage.dgn** file.

1. Open the **Analyze Between Points** tool (**Drainage and Utilities >> Home >> Model Analysis >> Civil Analysis**).

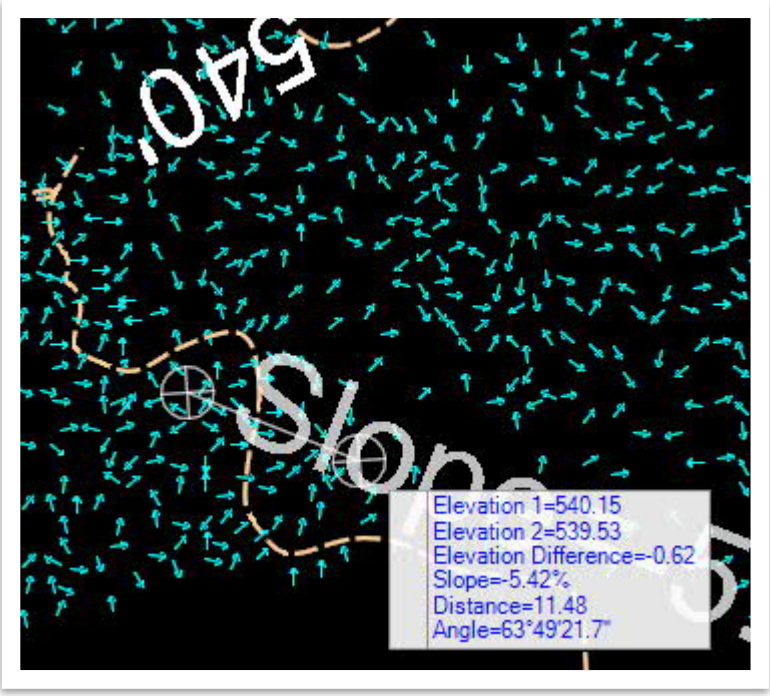


2. Notice the cursor prompt: **Select Terrain Model element**. Select any contour of the referenced complex terrain.

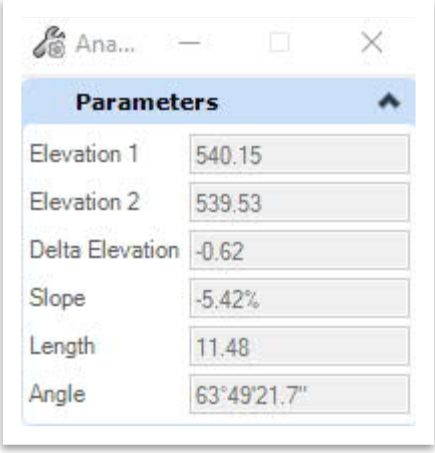


3. Notice the next cursor prompt: **Select Start Point**. Pick any point as a start point.

4. Now, move the cursor in any direction. Notice the **slope** between the two points is displayed along the cursor direction. In addition, the elevations at both points plus the corresponding **Elevation Difference**, **Slope**, **Distance** and **Angle** are shown dynamically as the cursor is moved around. Select a **second point** and notice the slope is physically placed in the file. **Note:** The data shown in the screenshot will vary depending on what 2 points you selected.



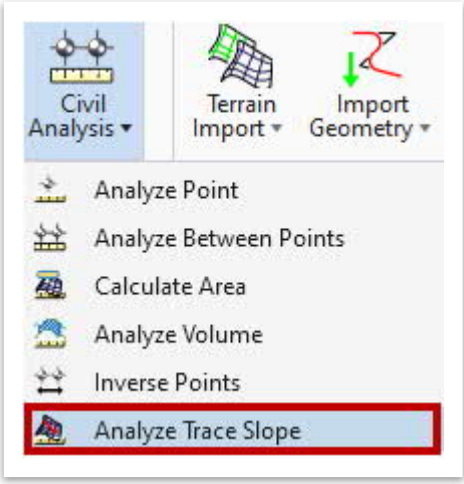
5. You will also notice that the same data auto-populates within the **Analyze Between Points** dialog box. **Note:** The data shown in the screenshot will vary depending on what 2 points you selected.



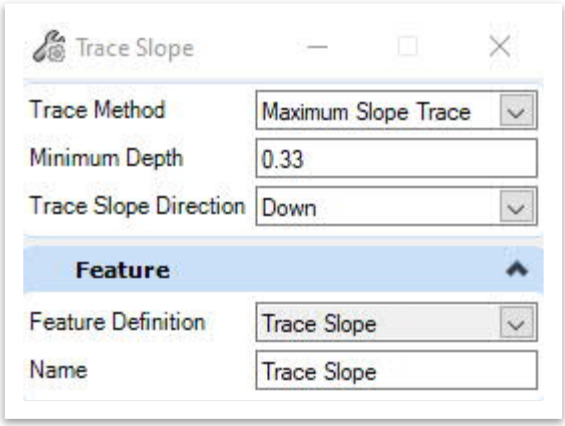
4.4.3 Exercise: Civil Tools – Analyze Trace Slope

In this exercise, we will utilize the **Analyze Trace Slope** tool, which corresponds to the path of a drop of water down a selected terrain model. This tool has two different options: **Maximum Slope** and **Constant Slope**. The former is the appropriate tool for analyzing terrain. We will continue to utilize the same **Bowling_Drainage.dgn** file.

1. Open the **Analyze Trace Slope** tool (**Drainage and Utilities >> Home >> Model Analysis >> Civil Analysis**).



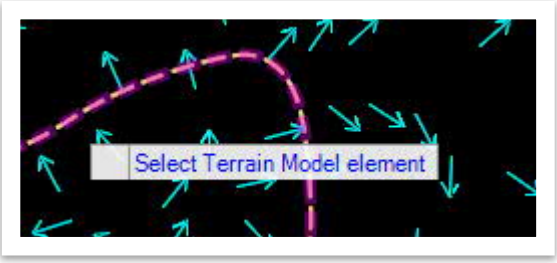
2. Select the following settings in the **Analyze Trace Slope** dialog box.
   a. **Trace Method:** Maximum Slope Trace
   b. **Trace Slope Direction:** Down
   c. **Feature Definition:** Trace Slope



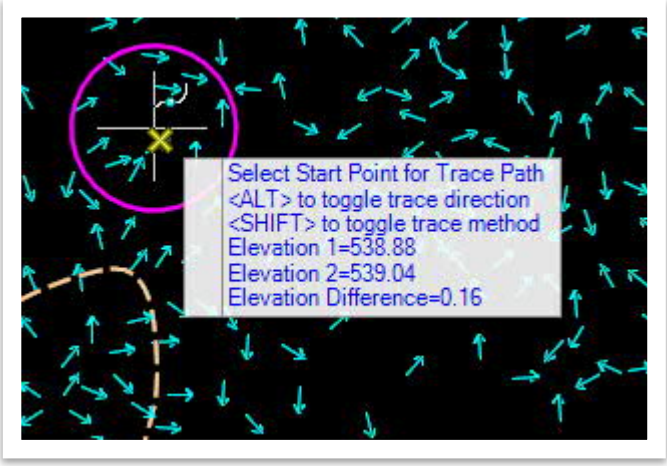
Take Note!

*The **Trace Slope Direction** does not refer to the direction of flow but rather references the beginning point. For example, if the starting point is downhill, you would select **Down** and then vice versa for uphill.*

3. Notice the cursor prompt: **Select Terrain Model element.** Select any contour of the referenced complex terrain.



4. Notice the cursor prompt: **Select Start Point for Trace Path**. Select any point as the Start point. This results in a **flow path** shown from the cursor to the furthest point that the water drop reaches. This tool helps in analyzing flow patterns for a project. **Note:** The data shown in the screenshot will vary depending on what start point you selected.

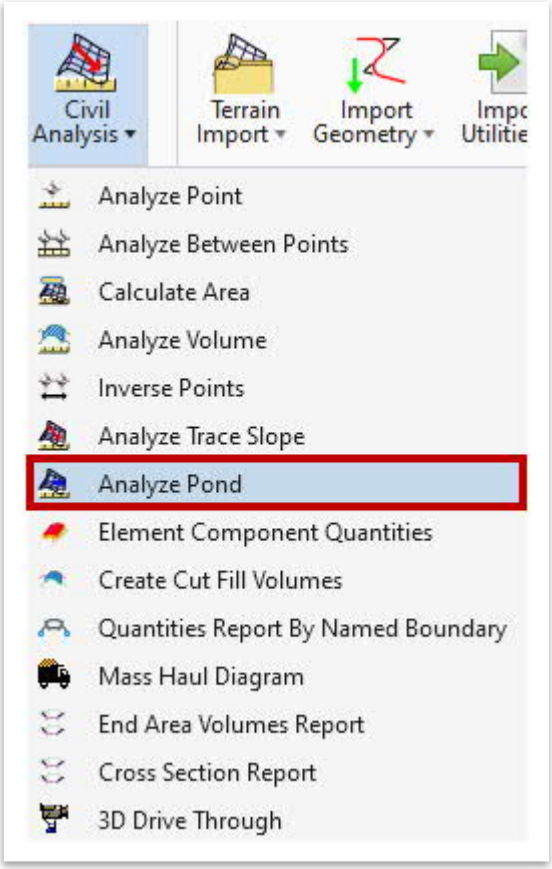


5. For this exercise, go ahead and hit **UNDO** to remove the flow path from the file.

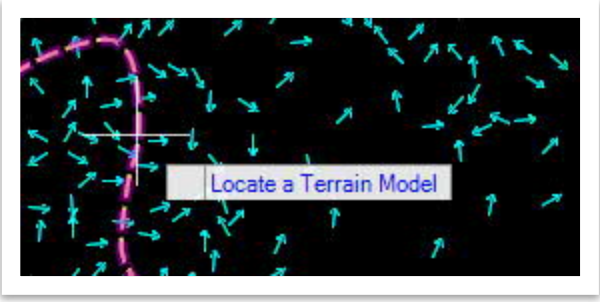
4.4.4 Exercise: Civil Tools – Analyze Pond

In this exercise, we will identify the ponds on the referenced complex terrain and look at the associated information. We will continue to utilize the same **Bowling_Drainage.dgn** file.

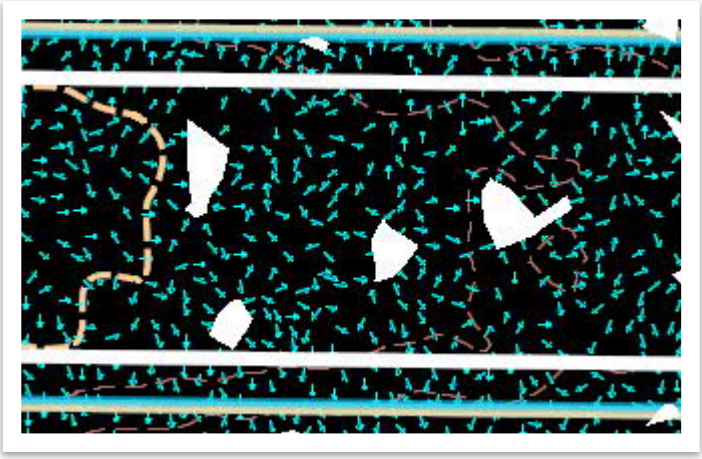
1. Open the **Analyze Pond** tool (**Drainage and Utilities >> Home >> Model Analysis >> Civil Analysis**).



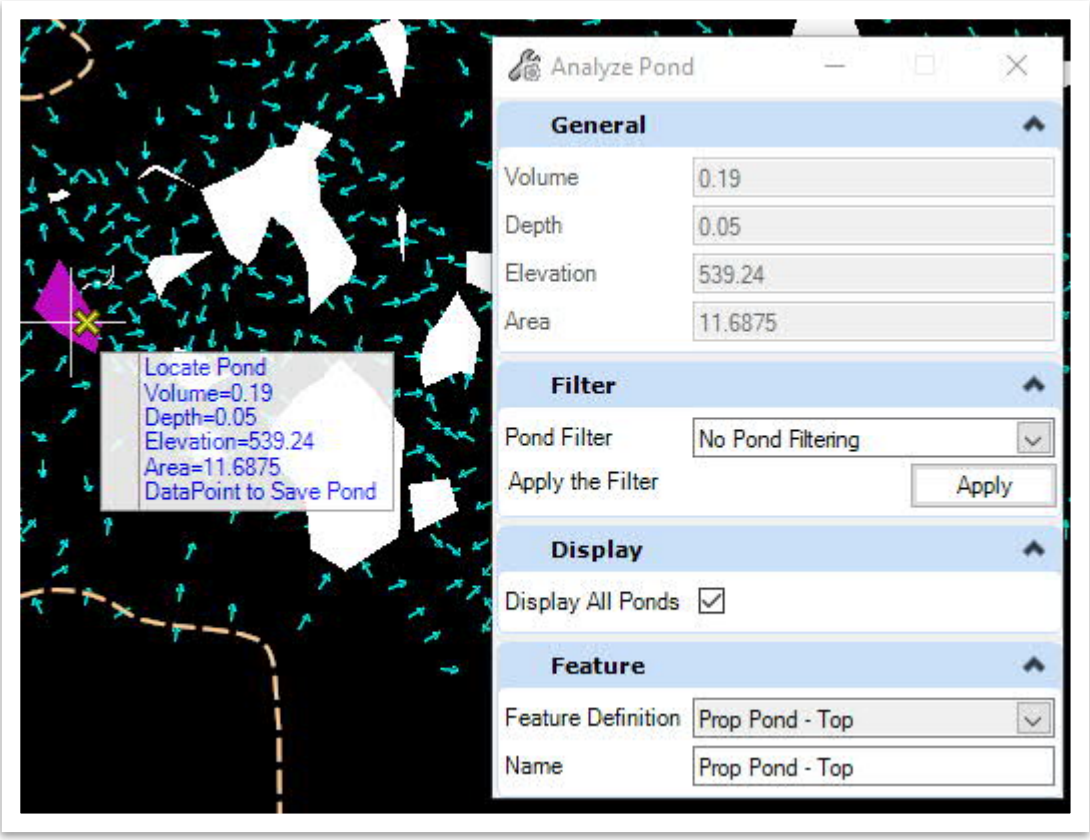
2. Notice the cursor prompt: **Locate a Terrain Model**. Select any contour of the referenced complex terrain file.



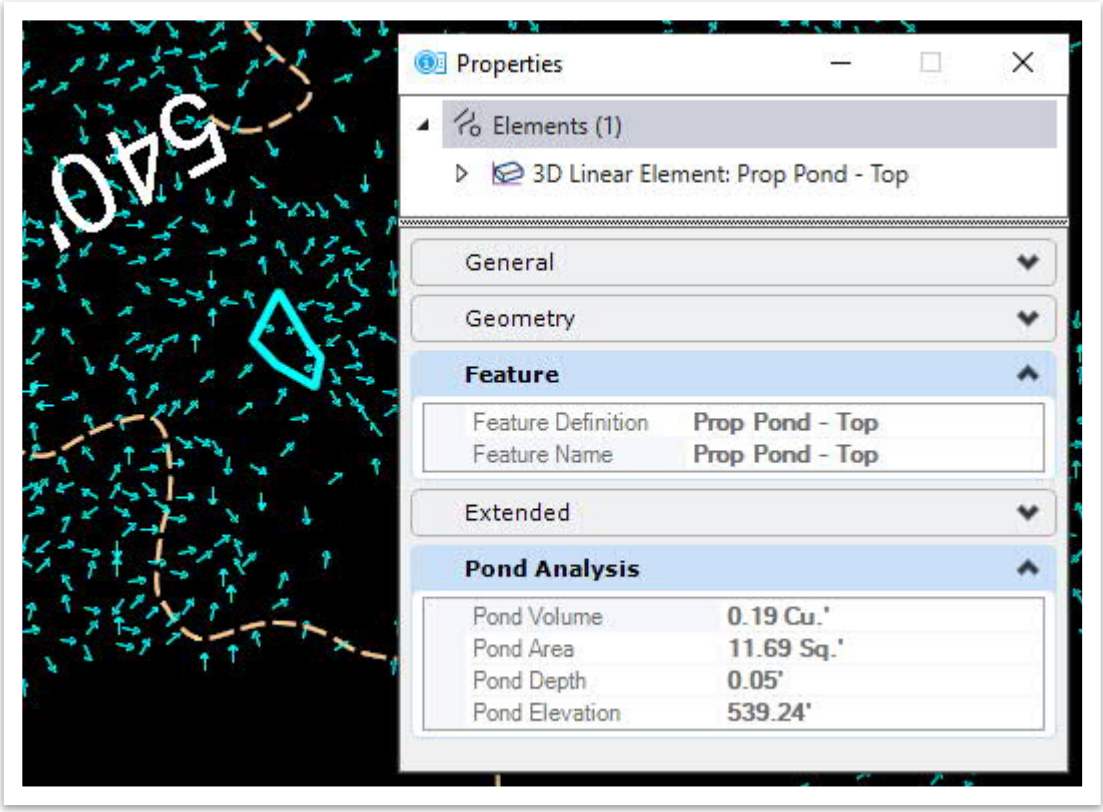
3. Zoom out and notice that several areas filled in with a solid white color. These are points where **ponding** areas have been identified.



4. The **Analyze Pond** dialog box should open. Hover over the different pond areas and you will notice that the pond data will populate. Within the dialog box, select the **Prop Pond - Top** feature definition (**Linear >> Roadway >> Pond**) and then left click on the shape as shown below. This will add the pond area to the model. **Note:** Units for **Volume** and **Area** are **cubic feet** and **square feet** respectively.



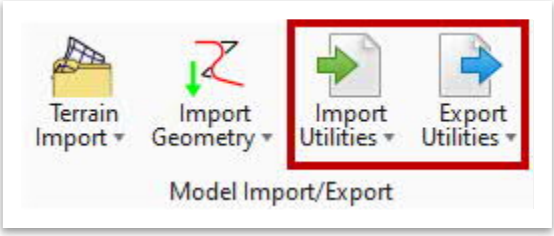
5. Now hit **ESC** to clear the tool. This will clear all ponds initially identified in the **Default** view other than the one that was added in the previous step. Go ahead and select the pond boundary and open its **Properties**. Notice that the **Pond Volume** and **Pond Area** are shown under **Pond Analysis**. Once reviewed, deselect the pond and turn off the flow arrows.



4.5 Lecture: Import/Export Model

Refer to the Roadway Design I (ORD) Manual for the **Terrain Import** and **Export Geometry** tools. This section will focus on the Importing and Exporting of Drainage Utility files (Figure 14).

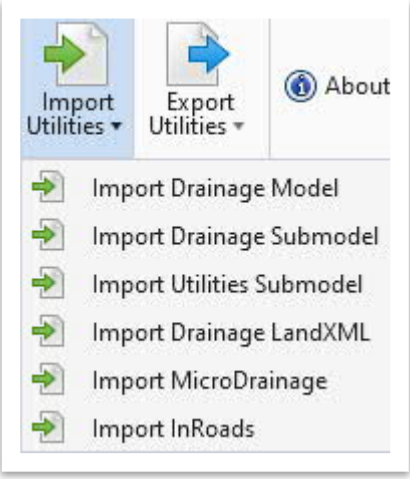
FIGURE 14. IMPORT/EXPORT UTILITIES



4.5.1 Import Utilities

The **Import Utilities** tools provide a set of sub-tools to import a utility model into the active dgn file. Applications that are supported include OpenFlows Model, OpenFlows Submodel, LandXML, MicroDrainage and InRoads files (Figure 15). **TDOT's current drainage models are contained within. GDF (geopak) files and cannot be imported into ORD using these tools**.

FIGURE 15. IMPORT UTILITIES



However, once a <u>new</u> project is started and the drainage model has been created solely in ORD, the **two** applications **listed below** will become applicable.

- **<u>Import OpenFlows Model</u>:** This tool imports a utility model to a Microstation file (DGN) if there is no existing utility model already linked to the target file. This tool imports a file with a ***.stsw** format. **Once the OpenFlows Model is imported, the user can continue to build a proposed utility model**.

- **Import OpenFlows Submodel:** This tool imports a utility model to a Microstation file (DGN), even if the target file has an existing utility model. The file imported will have a ***.sqlite** format. It is important to note the following when importing an OpenFlows Submodel:

Take Note!

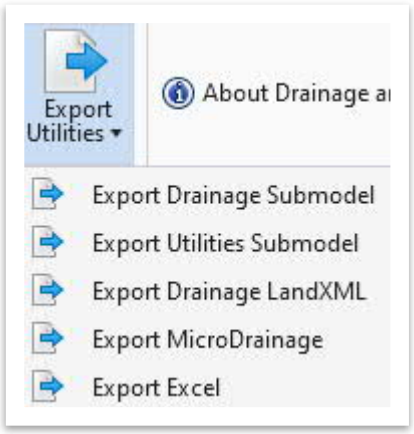
The OpenFlows Submodel (.sqlite file) overwrites any scenarios and elements with the same label in the target file.*

  - If the user imports a ***.sqlite** file into an existing utility model, and both files have a node with label CB1, the node in the imported file overwrites (replaces) the node in the target file.
  - Any portion of the working model can be exported by selecting specific elements such as pipes, catchment types/runoff and default constraints.

4.5.2 Export Utilities

The **Export Utilities** tools provide a set of tools to be able to export SUDA hydraulic utility model to various applications (Figure 16). Applications supported include: OpenFlows, MicroDrainage, LandXML file and Excel file.

FIGURE 16. EXPORT UTILITIES



Take Note!

*User preference and project needs determine which export tool to use. The software does distinguish between drainage and utilities models. Although they appear to be treated the same, they are managed differently in the software. Future ORD releases will show them separately in the **Explorer** under **Drainage and Utilities Model**.*

Chapter 5. Layout a Closed Drainage Network

This chapter focuses on laying out a closed drainage network system, which is comprised of nodes (inlets, junctions, and outfalls), conduits, catchments, and gutters. The information shows how to use the drainage and utilities tools to input the data. The user should review the most current TDOT Drainage Manual and make sure that conformance is met for the appropriate hydrologic and hydraulic items needed to perform the computations. For the flow of getting spread computations as the user lays out the inlets, a scenario analysis must be run. Section 5.11 contains a detailed description of scenarios and flex tables for the completion of a drainage network.

5.1 Objectives

At the conclusion of this chapter, participants will be able to:

1. Utilize the Layout tools to create a drainage model which includes:
    a. Nodes (inlets)
    b. Endwalls
    c. Catchments
    d. Gutters
    e. Conduit (pipes)
2. Add riprap at culvert outlets.
3. Create, review, and edit profile runs.
4. Verify project design constraints.
5. Run inlet spread scenarios.
6. Run and manipulate Flex Tables.
7. Understand how to use the Civil Labeler tool.

5.2 Lecture: Layout Tools

The **Layout** tools are used to build a utility model. In this section of the manual, the user will learn how to place drainage nodes/conduits and how to build a storm drain network.

Refer to the example drainage network shown in **Appendix C** for the overall drainage network layout.

Place Node: Places utility point features such as catch basins, manholes, valves, bends and endwalls.

Place Nodes: Used to create nodes that are equally spaced along a linear element.

Insert Node: Inserts a node within a conduit by splitting the conduit into two conduits.

Place Conduit: Creates conduit links by connecting two independent nodes.

Place Lateral: Places utility point features such as catch basins, manholes and valves while simultaneously creating a connecting conduit to a trunk line.

Place Gutter: Defines the path of bypass flow and the shape of gutter between inlets of a drainage network.

Place Catchment: Defines an area which could either be for a catchment or a pond.

Place Pond: This tool creates a Pond.

Place Low Impact Development: This tool creates a Low Impact Development, which protects the water quality using natural processes.

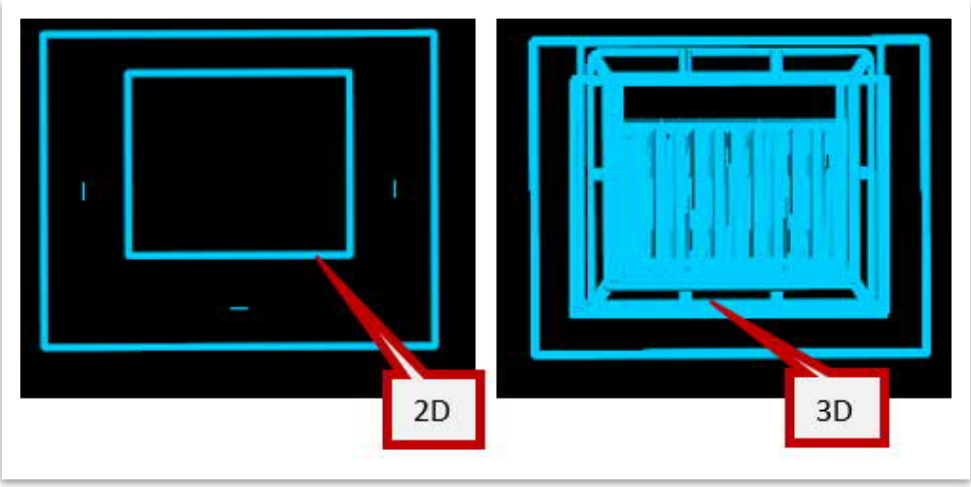
Extract from Graphic: Creates utility models from graphic elements. **This tool is not in the scope of this manual**.

Filter Manager: Allows the creation and management of filters and filter groups. **This tool is not in the scope of this manual**.

5.3 Lecture: Nodes

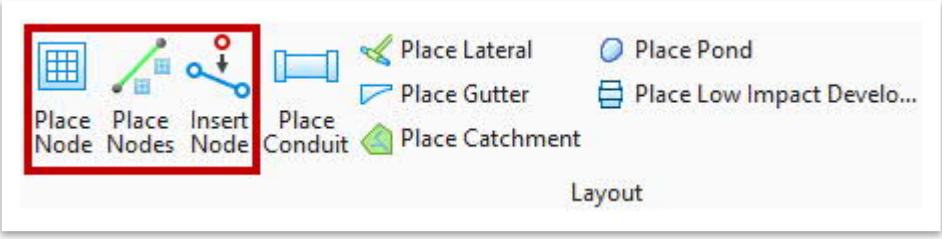
Nodes are interactive solid features containing information and drainage properties that allow them to be utilized in a drainage network design. Nodes act as points in a network that the storm drainage may flow in and out of as well as being a junction between two or more conduits. Common nodes include catch basins, manholes, junction boxes and endwalls. Figure 17 shows sample nodes in both 2D and 3D views when laid out in a drainage utility model.

FIGURE 17. 2D AND 3D NODES



There are **three** tool options to place nodes in ORD: **Place Node**, **Place Nodes**, **Insert Nodes** (Figure 18). Each option is discussed in more detail below and can be accessed here: **Drainage and Utilities >> Layout >> Layout**. When initiating these tools for the first time in a design file, the software will need to embed the necessary workspace files for Drainage and Utilities projects into the file and will result in the user needing to open the applicable tool a 2nd time to use it.

FIGURE 18. NODE TOOL OPTIONS

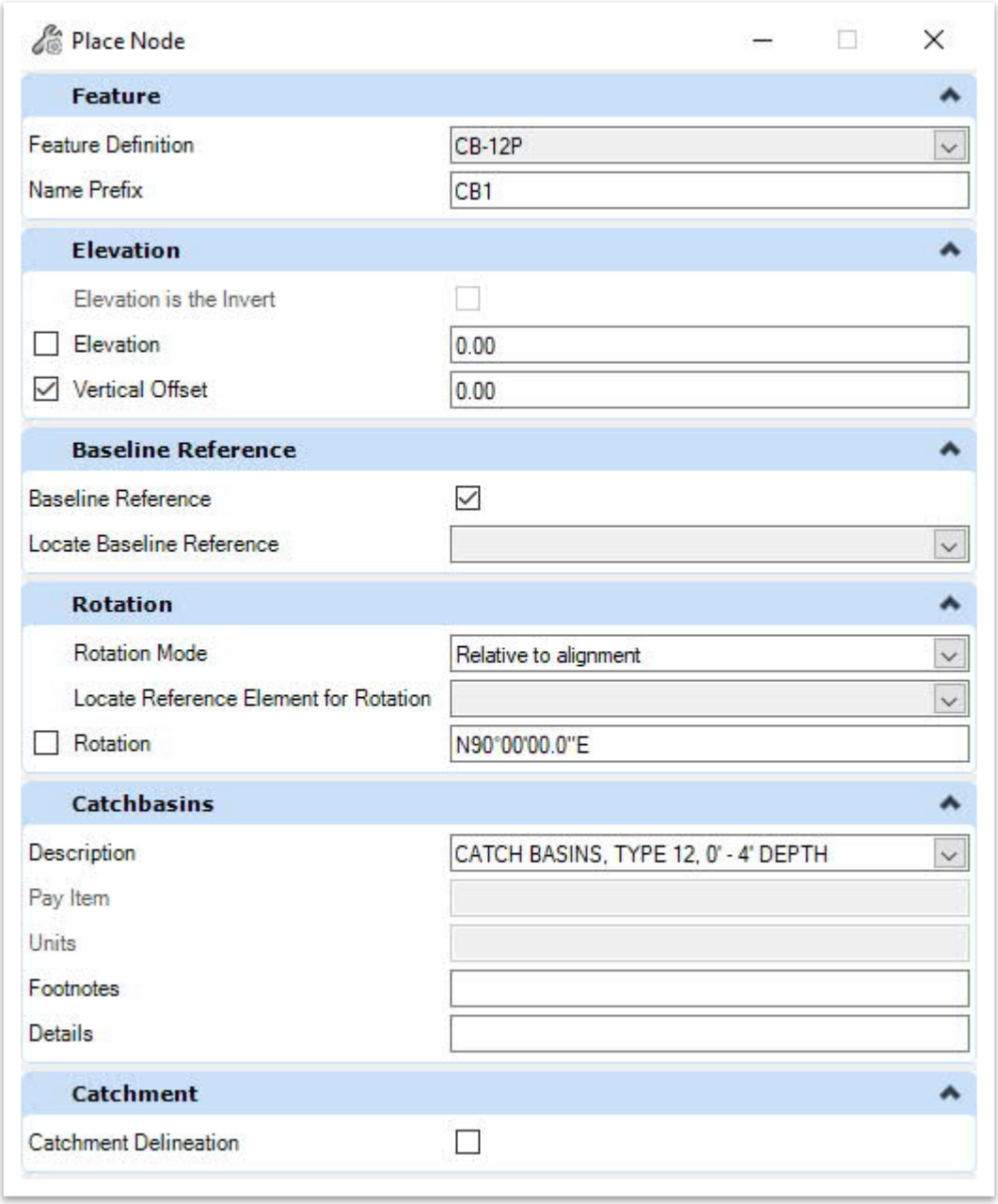


Take Note!

*When placing a node that will intake flow, use the **gutter flow line** as the reference instead of a roadway reference. This will place the grate at the lowest part in the gutter. For non-inlet nodes, use the proposed tin. If the corridor is used, sometimes the structure top is placed higher than the pavement.*

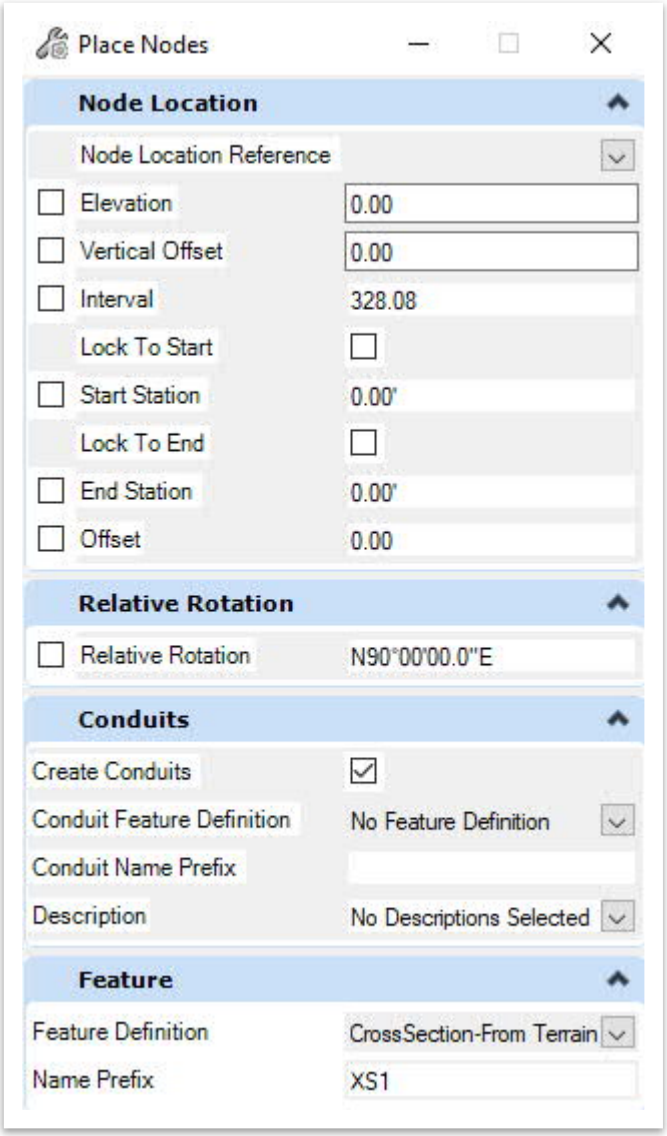
- **Option 1 (Place Node):** Allows the user to place **one** node at time (Figure 19). Some of the key options are described below.
  - Feature Definition: Use the drop-down to find the correct node (structure).
  - Baseline Reference: This is referred to as the alignment. If used, toggle this option on and then select the correct baseline.
  - Catchment Delineation: Toggle this option on if you have pre-drawn catchments. Click through the prompts to get the catchment associated with the node.

FIGURE 19. PLACE NODE DIALOG BOX



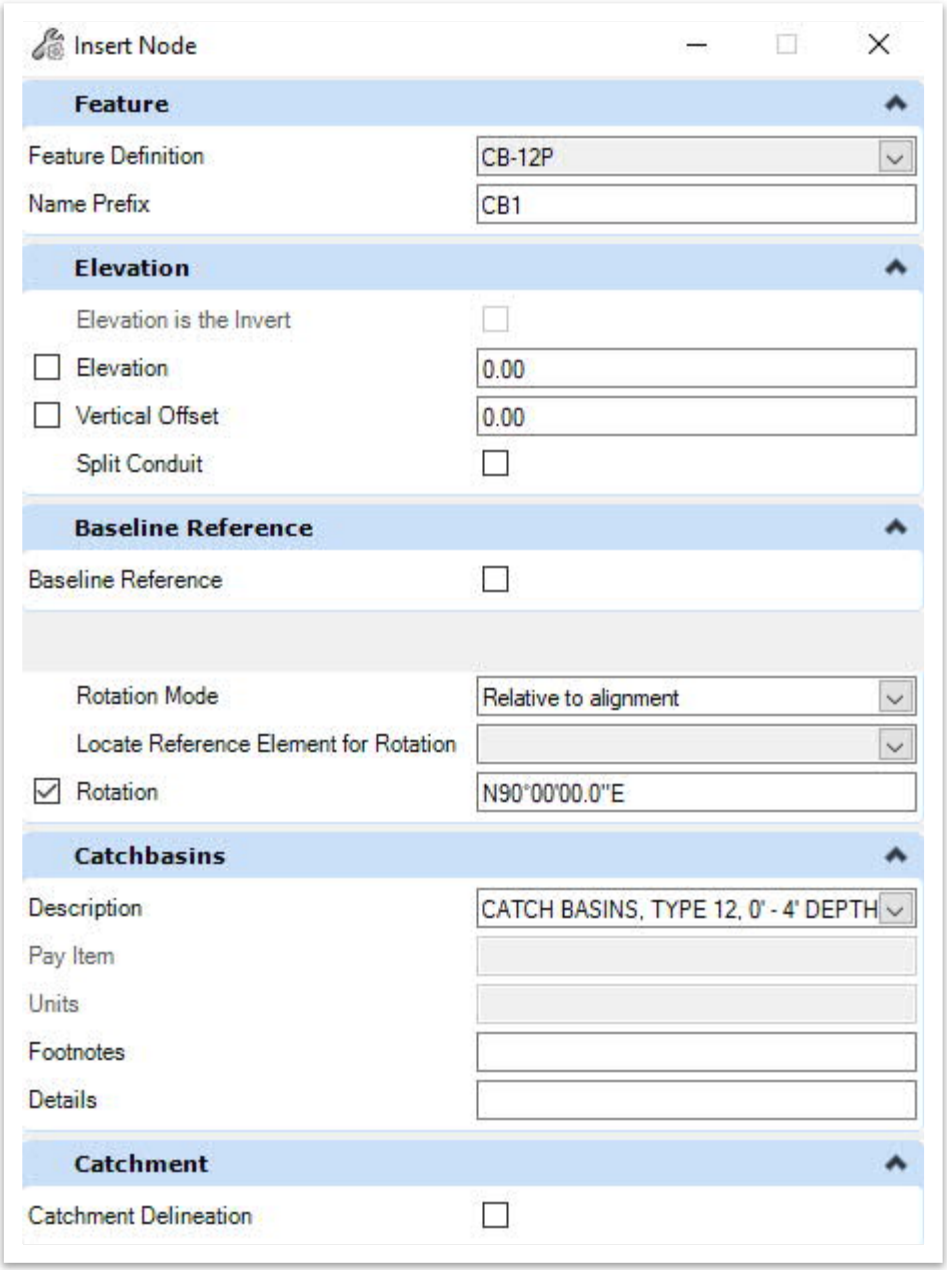
- **Option 2 (Place Nodes):** Allows the user to place **multiple** nodes along an element (e.g., along a curb and gutter section or along a barrier wall) (Figure 20). Some of the key options are described below.
  - Node Location: This would be the alignment. The user can place the same type of node along a determined length.
  - Create Conduits: This option allows the user to add a conduit at multiple locations, thus avoiding the need to place individual conduits in a file.

FIGURE 20. PLACE NODES DIALOG BOX



- **Option 3 (Insert Node):** Allows the user to insert a node along a conduit that has already been placed (Figure 21). This option would only be necessary if making a revision to a hydraulic model where the network has been layed out. This allows a previously placed conduit to be split, thus removing the need to manually add new conduit.

FIGURE 21. INSERT NODE DIALOG BOX



All of the nodes (cells) in the TDOT ORD workspace are based off of TDOT standard drawings. A user could open the **Inlet Catalog** (**Drainage and Utilities >> Components >> Catalog >> Catalog**) if the cell dimensions needed to be checked (Figure 22 and Figure 23).

FIGURE 22. INLET CATALOG

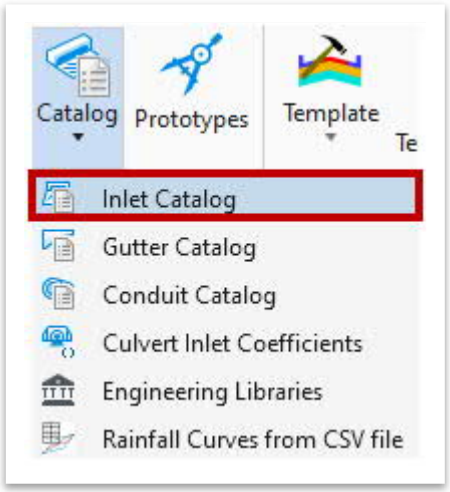
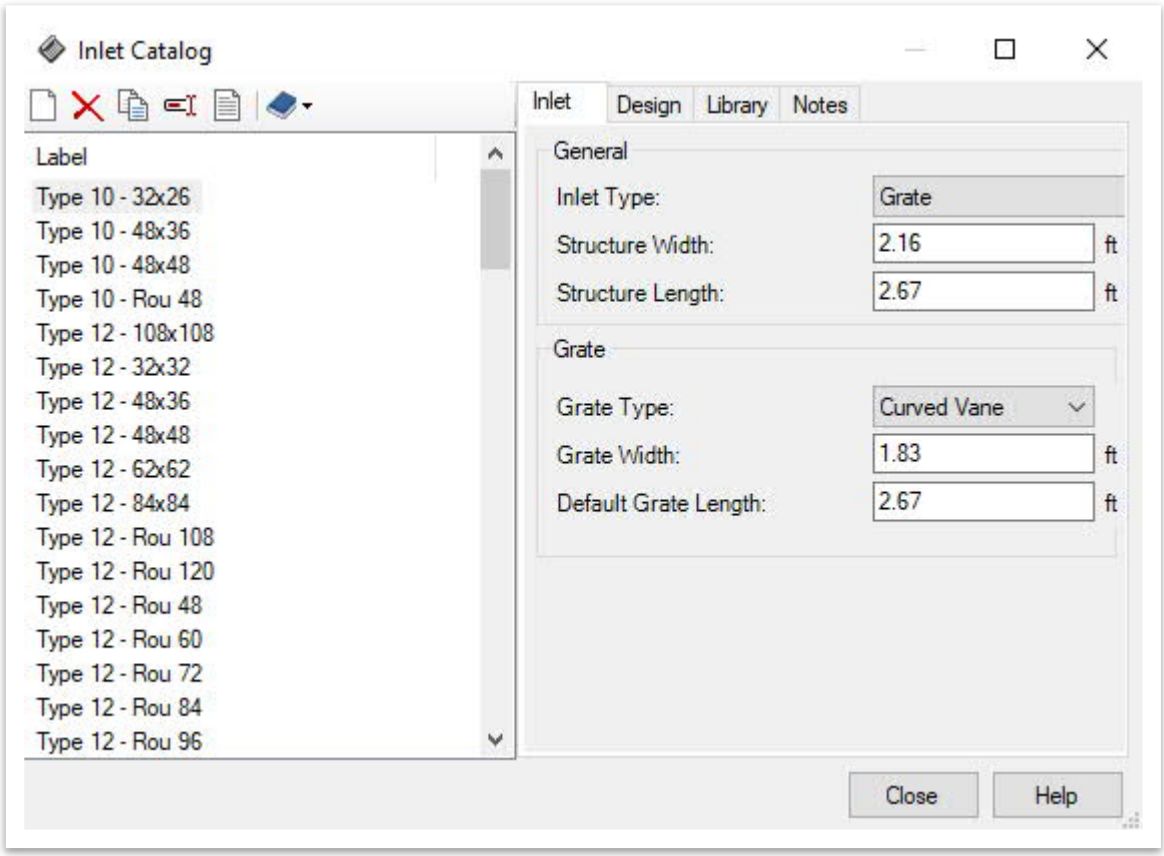


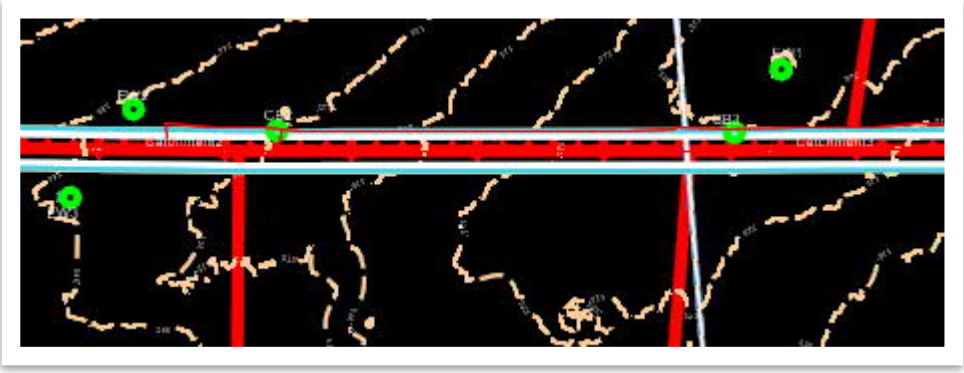
FIGURE 23. INLET CATALOG DATABASE



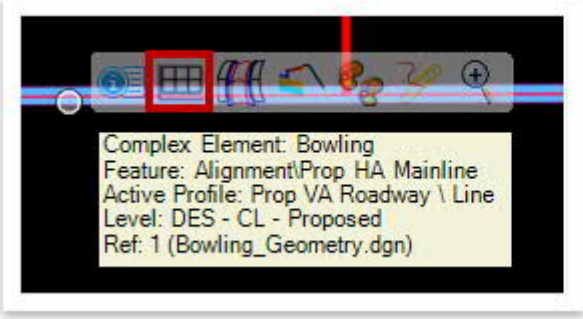
5.3.1 Exercise: Placing Nodes

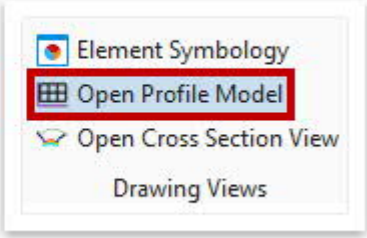
In this exercise, we will begin to build our drainage model by placing four nodes (three catch basins and one endwall). We will continue to utilize the same **Bowling_Drainage.dgn** file.

1. Attach the **Bowling_Catchment samples.dgn** file as a reference using the **Coincident World** attachment method.
2. Zoom in and notice the referenced labels and green circles indicating approximate catch basin locations: **CB1** (far right), **CB2** (far left) and **CB3** (between CB1 and CB2). In addition, approximate locations for endwalls (**EW1**, **EW2** and **EW3**) are also identified.

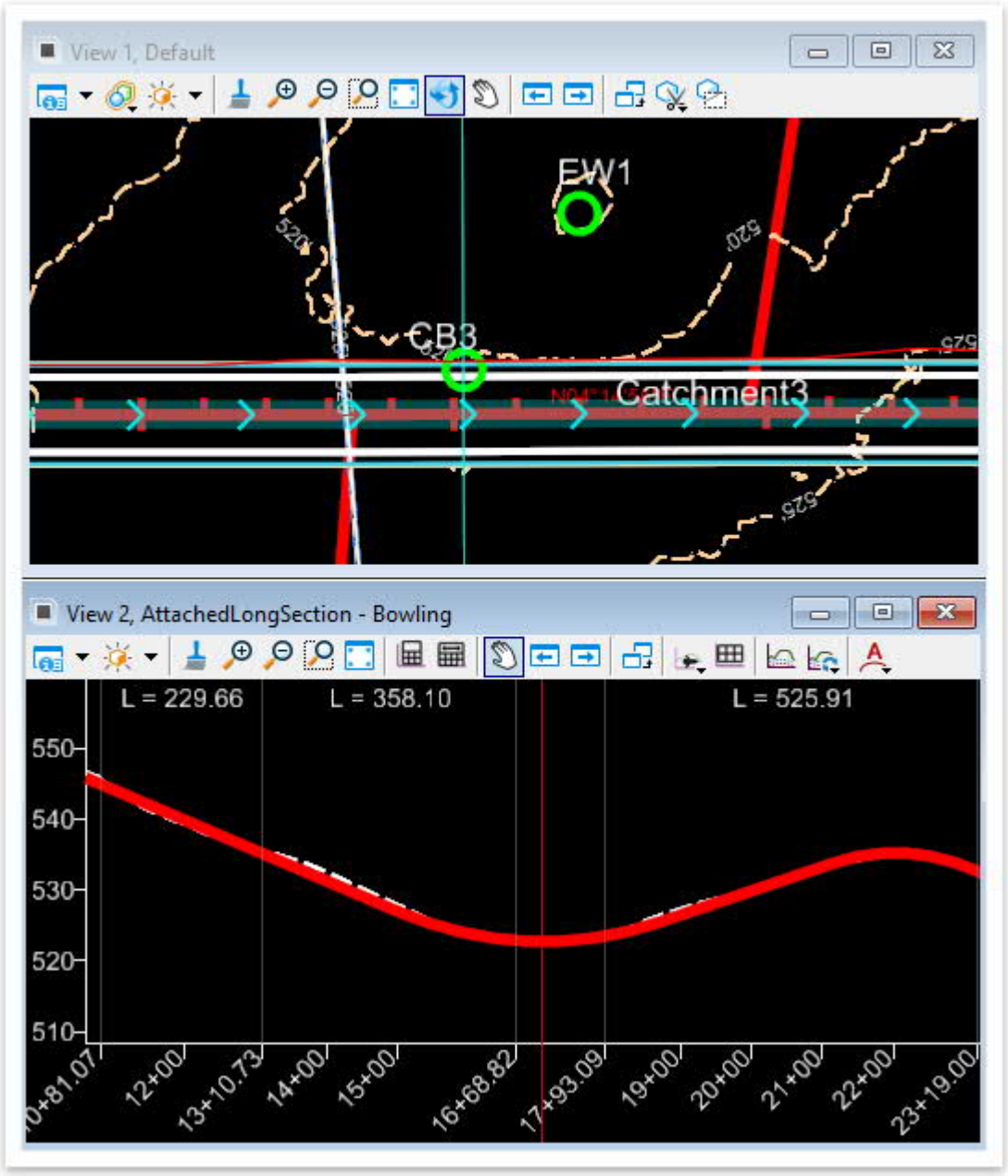


3. We will now place catch basins at the approximate locations shown. **The location of CB3 is a sag point on the roadway**. Before we place any catch basins, it is recommended to review the **centerline profile**. Select the **Bowling** centerline and open the **Open Profile Model** tool. It can either be accessed within the heads-up display (2nd icon) or within the ribbon under **Drainage and Utilities >> Utilities View >> Drawing Views**.

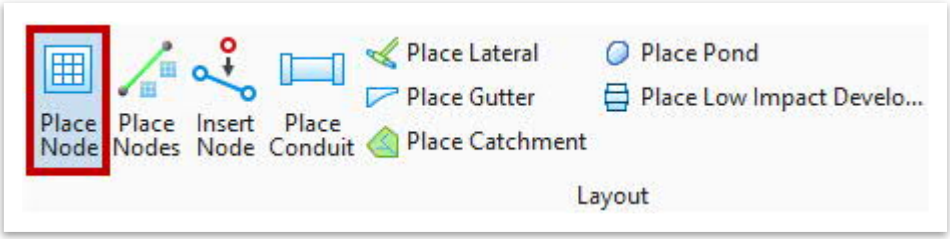




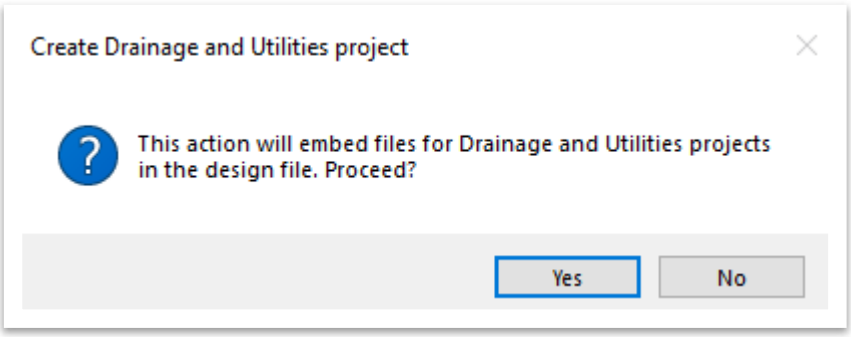
4. The cursor prompts to **Select or Open View**. Open **View 2** and left click anywhere within the window. Notice that the roadway profile has a **sag** at the midway point. Once reviewed, go ahead and **close the profile window**. **Note:** If the profile view is active, you will see light blue directional arrows along the alignment in plan view.



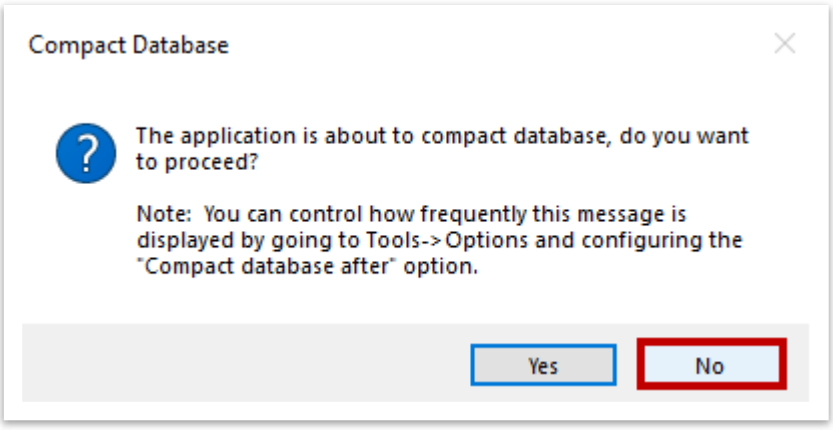
6. Go ahead and open the **Place Node** tool (**Drainage and Utilities >> Layout >> Layout**).



7. A warning will display asking if you want to proceed with embedding files for Drainage and Utilities projects in the design file. Click **Yes**.

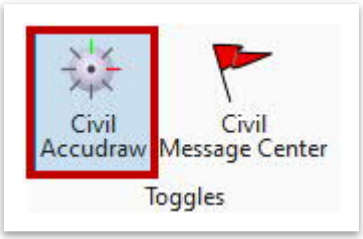


8. You should get a **Compact Database** alert asking if you want to proceed. Click **No**. **Note:** It is recommended to always click **No** unless the database (i.e., the drainage dgnlib) has been updated and needs to be compacted.

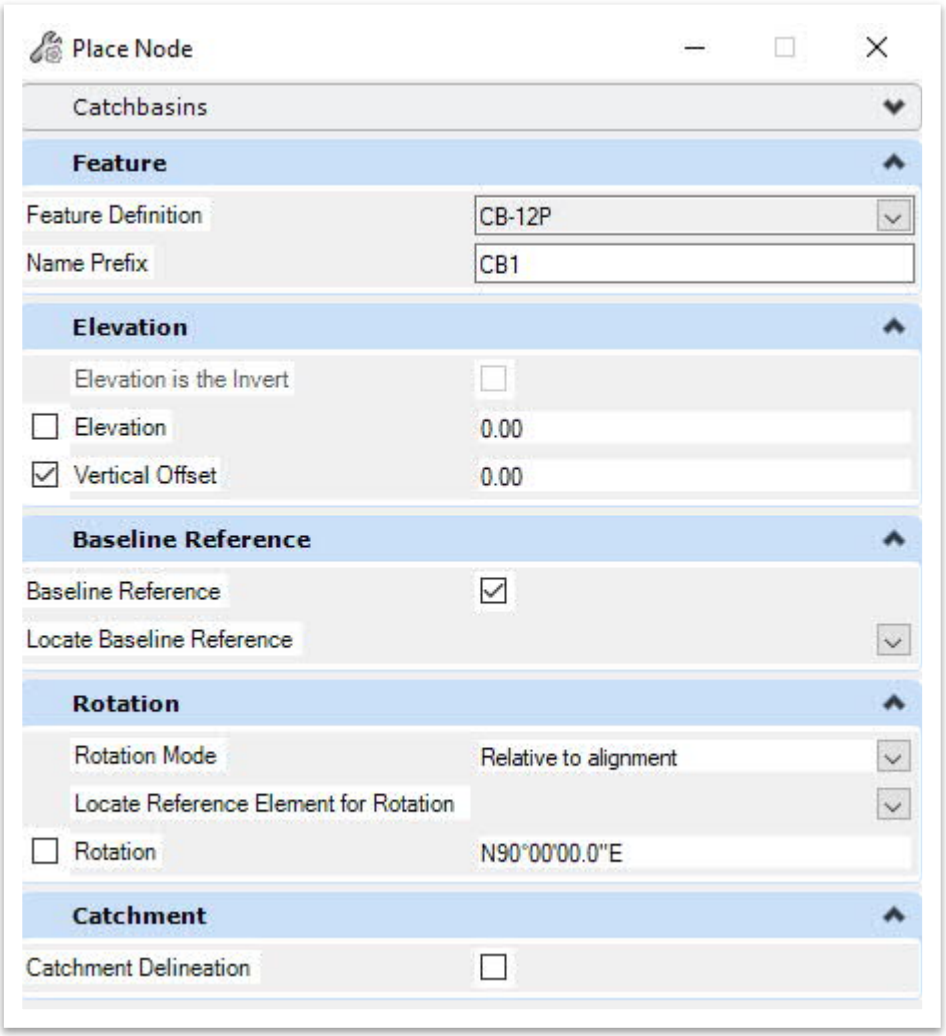


9. It will take a minute for the software to create the drainage model. The previous two steps are only seen when creating a drainage model in a **new** file. SUDA's hydraulic seed file that runs the background calculations is loaded from the **TDOT Drainage Feature Defs.dgnlib**. Now that the utility model has been created in the design file, all drainage modeling tools will open the first time from now on in this file.

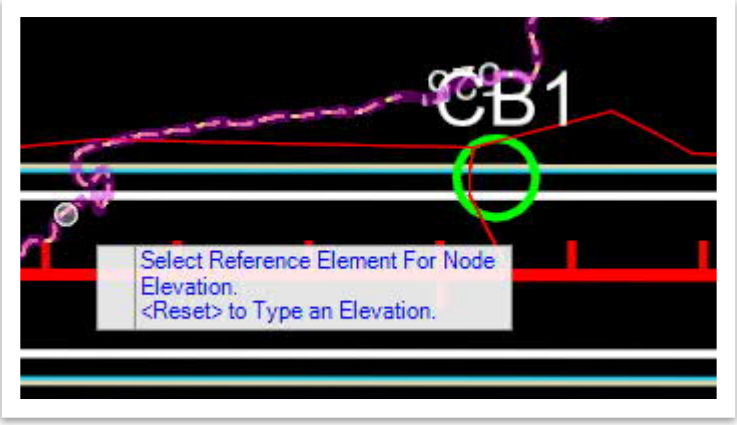
10. Before placing any nodes, open the **Civil Accudraw** tool (**Drainage and Utilities >> Layout >> Toggles**). Toggle **Civil Accudraw** ON and then select the **Station-Offset** icon (2nd from right). This will assist in more accurate node placement. **Note:** As a reminder, you cannot have regular Accudraw on at the same time.



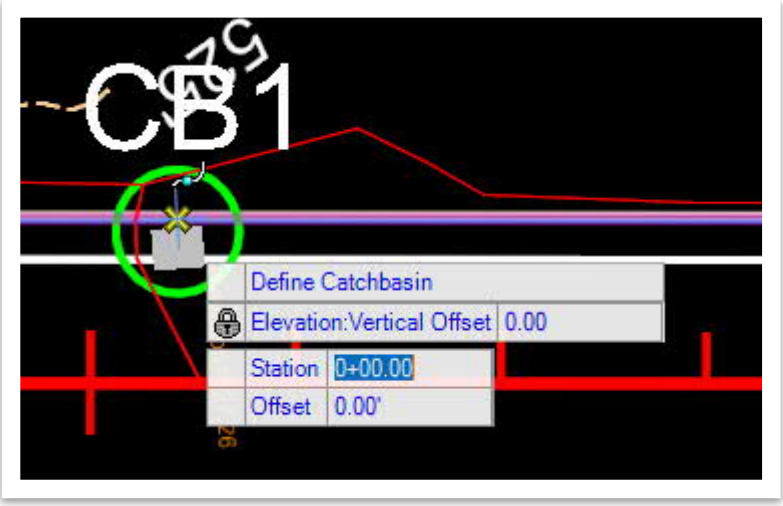
11. Open the **Place Node** tool again. Within the **Place Node** dialog box, select the **CB-12P** feature definition (**Node >> StormWaterNode >> Storm Drains >> Proposed >> Type 12-13**). By default, the **Name Prefix** should be **CB1**. Match the remaining settings as shown below. **Note:** You can ignore the elevation value.



12. Notice the cursor prompt: **Select Reference Element for Node Elevation. <Reset> to Type an Elevation**. Select any contour of the referenced complex terrain file and left click to accept.

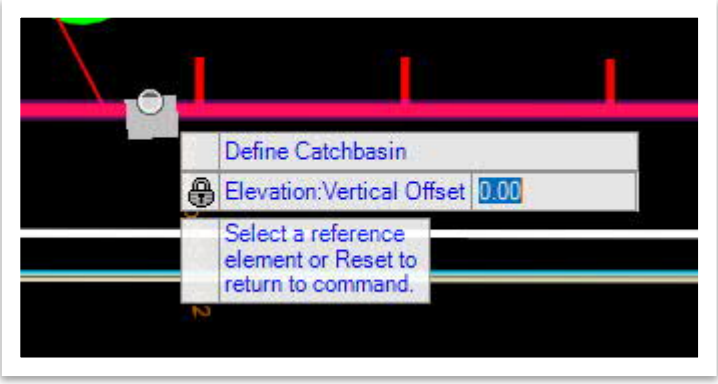


13. Hit **TAB** to move the selection from **Vertical Offset** to **Station**. **Note:** You may need to refresh the **Civil AccuDraw** tool if you do not see the Station and Offset options on your cursor.

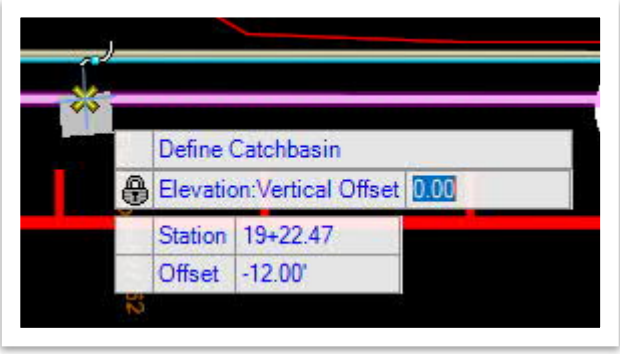


14. Move the cursor towards the location of **CB1**. Press the **letter O** on the keyboard. Notice the cursor prompt: **Select a reference element or Reset to return to command**. **Note:** If ever the Sta/Off functionality does not seem to work, you will likely need to re-select the reference element in which to base the Sta/Off by doing this step again.

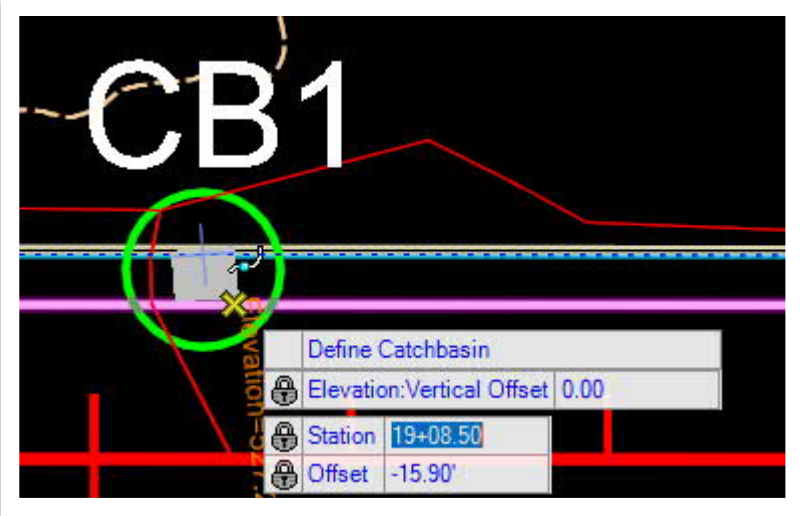
15. Select the **Bowling** centerline.



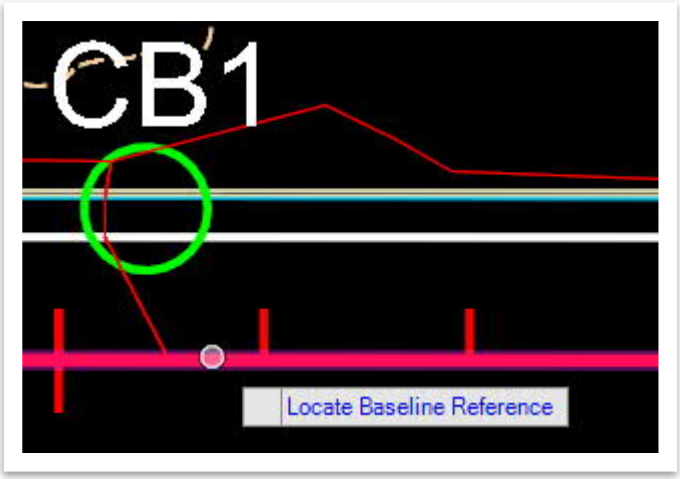
16. Move your cursor around and notice that **Civil AccuDraw** is now linked to the alignment.

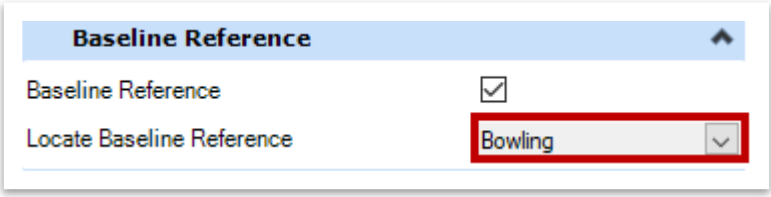


17. Move the cursor towards the location of **CB1**. Hit **TAB** once again to edit the station and offset. Key-in station **19+08.50** and offset **-15.90'** for this exercise. Hit the **TAB** or **Enter** key after each entry so that it locks in place. Left click to accept.

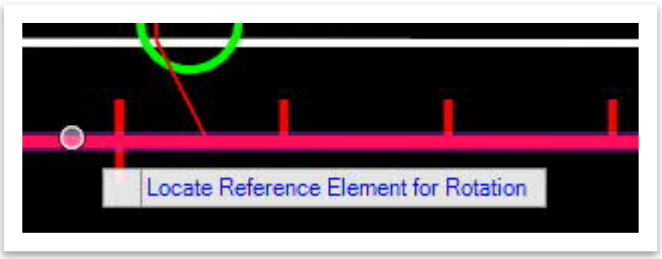


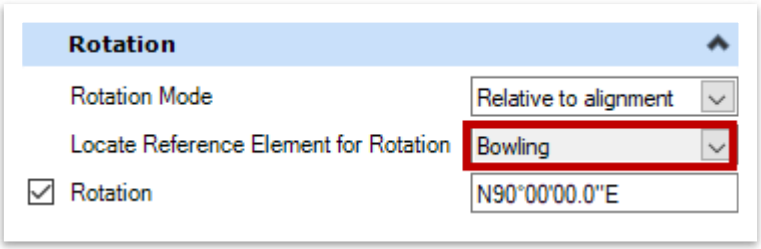
18. Notice the cursor prompt: **Locate Baseline Reference**. Select the **Bowling** centerline. Alternatively, you could select the Baseline Reference in the **Place Node** dialog box. Left click to accept.



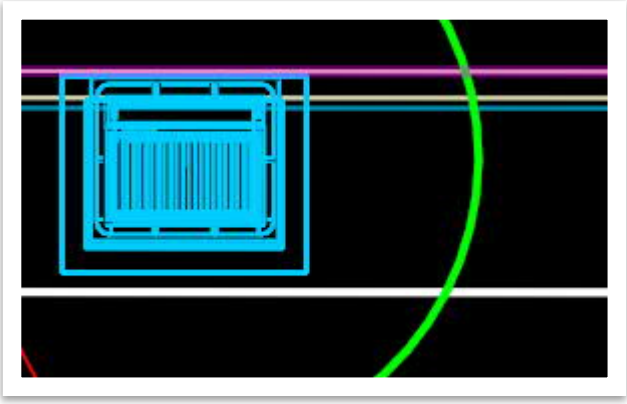


19. Next, you will be prompt to select the **Rotation Mode**. Left click to accept and a new prompt will appear asking you to **Locate Reference Element for Rotation**. Select the **Bowling** centerline and hit **Enter**. Left click to accept the rotation. Alternatively, you could select the Rotation Mode in the **Place Node** dialog box. In this exercise, we are checking the Rotation box and leaving it at 90 degrees. Left click to accept and then hit **ESC** to clear the tool.

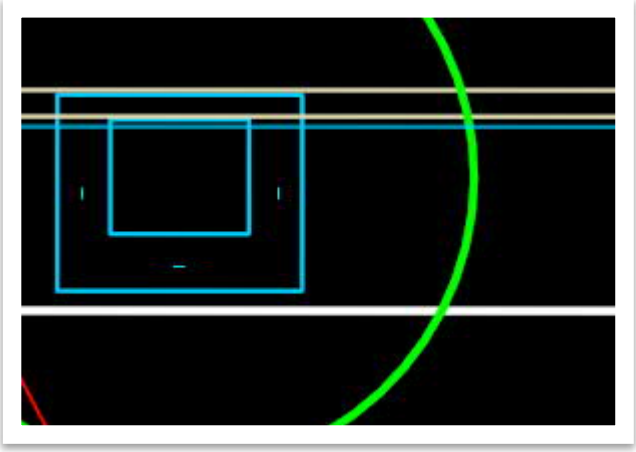




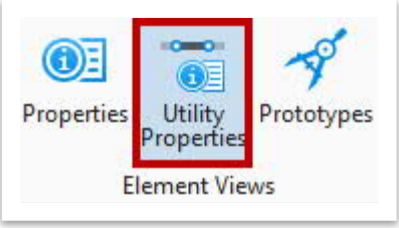
20. The light blue catch basin is now placed in both **2D** and **3D** views.



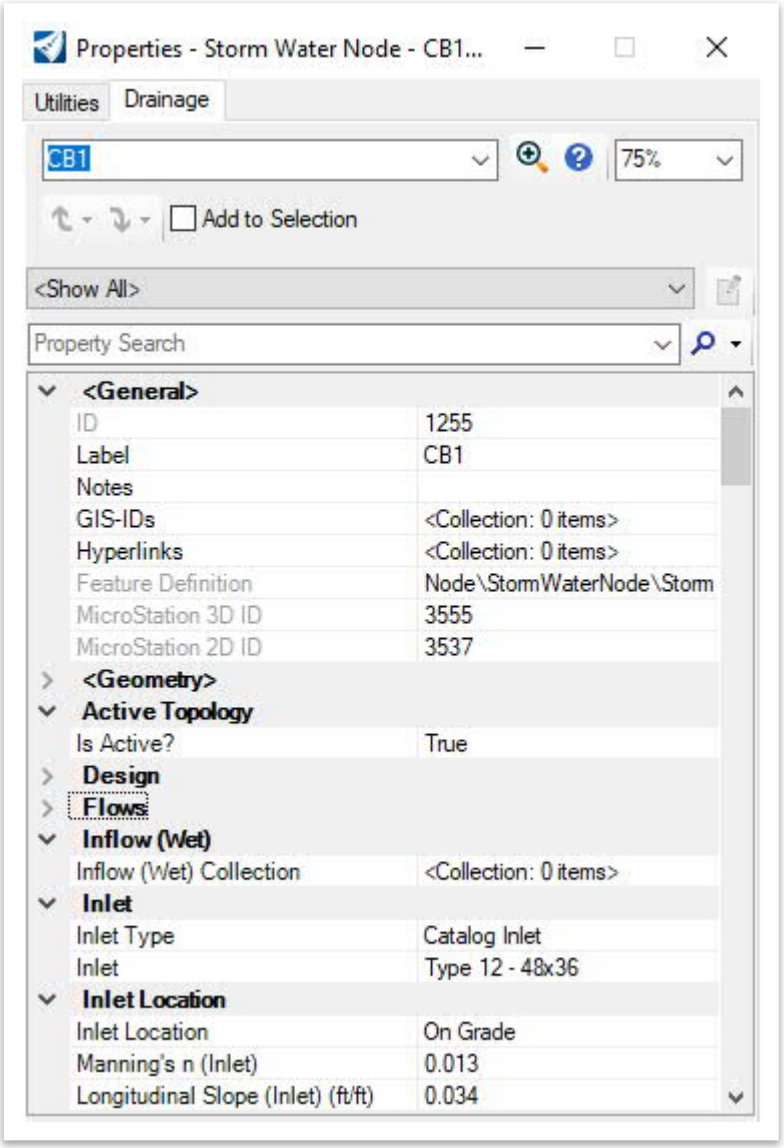
21. For clarity, let's view the model in **2D**. Within the **References** window, turn off the **Bowling_Drainage.dgn** file (**Model: Default-3D**). The catch basin should now only have the 2D linework displayed.



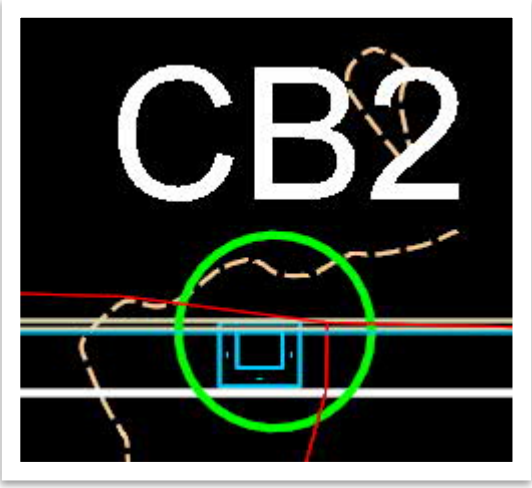
22. Before we place the other nodes, open the **Utility Properties** (**Drainage and Utilities >> Utilities View >> Element Views**).



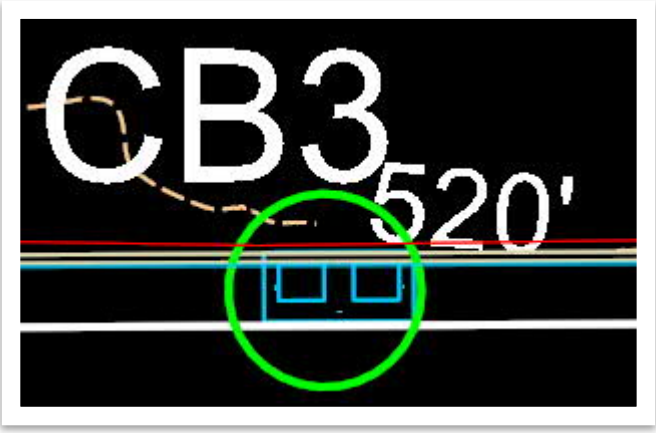
23. Select **CB1** and review the properties. Close the window once finished.



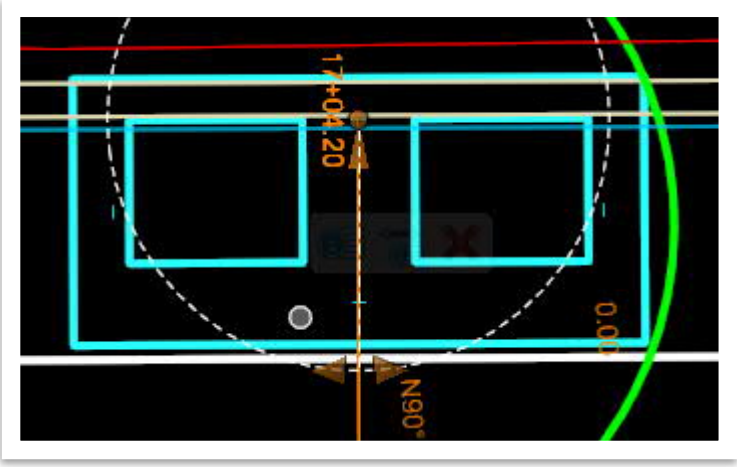
24. Open the **Place Node** tool once again. Let's place the second catch basin, **CB2**. Start by selecting any contour of the referenced complex terrain and then repeat Steps 17-19 to place another **CB-12P**. There is no need to change the **Name Prefix** within the **Place Node** tool, even if it reverts to **CB1**. This catch basin will automatically be numbered **CB2**. The catch basins will be assigned consecutive numbers as they are placed. Key-in station **13+40.00** and offset **-15.90'**. Keep the same **Rotation Mode** and **Rotation** angle.



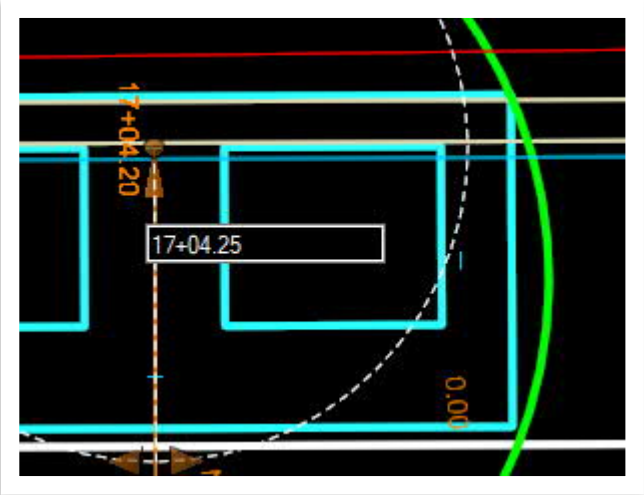
25. Now, let's place the third and final catch basin (**CB3**) near the sag point at the intersection. This catch basin will be a **Type CB-14S**. Within the **Place Node** dialog box, select the applicable feature definition (**Node >> StormWaterNode >> Storm Drains >> Proposed >> Type 14**). Once again, there is no need to change the **Name Prefix** within the **Place Node** tool, even if it reverts to **CB1**. This catch basin will be numbered **CB3** automatically. Repeat Steps 17-19 to place the structure. Key-in station **17+04.20** and offset **-15.90'**. Keep the same **Rotation Mode** and **Rotation** angle. Once placed, hit **ESC** to clear the tool. **Note:** If all the inlets will be placed along the curb line, relative to the centerline, the rotation angle should be **90°**.



26. Lastly, let's update the **CB3** station. Select the catch basin and notice that the manipulators (orange text) will highlight showing the station, rotation, and vertical offset.



27. Click the orange station text and key-in **17+04.25** and then hit **Enter**. Notice that the structure location automatically updates to the precise sag location along the roadway.



5.4 Lecture: Endwalls

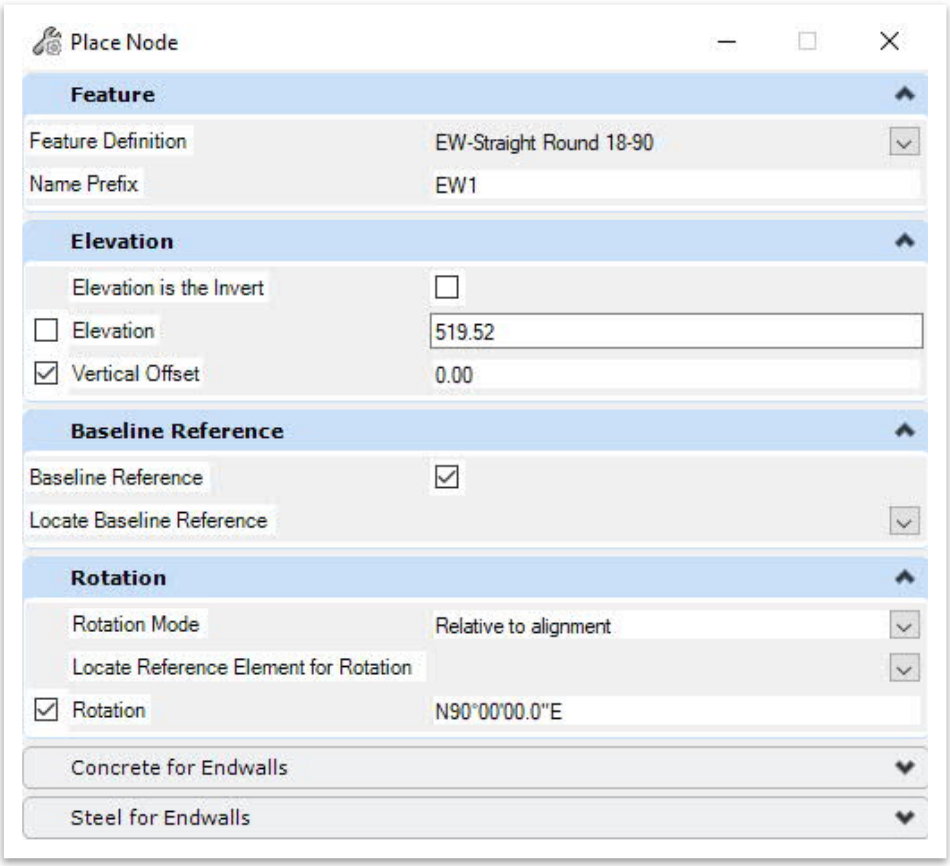
An endwall is a structure designed to protect the ends of a pipe (beginning with 18” RCP) and increase efficiency by directing the flow and reducing erosion issues. Keep in mind that endwalls cannot be placed within the clear zone. The latest TDOT endwall standard drawings can be accessed here:

https://www.tn.gov/content/tn/tdot/roadway-design/standard-drawings-library/standard-roadway-drawings/pipe-culverts-and-endwalls.html

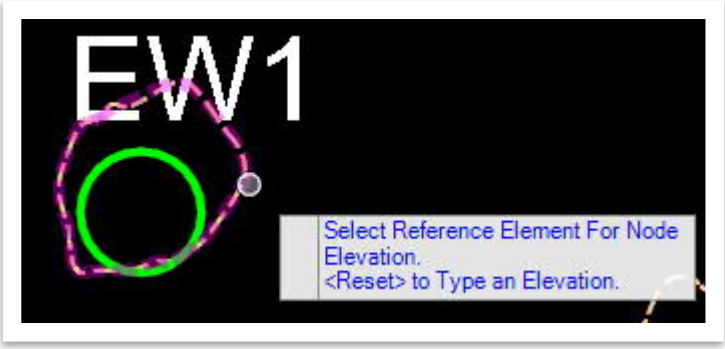
5.4.1 Exercise: Place Endwall

In this exercise, we will place an endwall north of the road to outlet the flow from a catch basin. We will continue to utilize the same **Bowling_ Drainage.dgn** file.

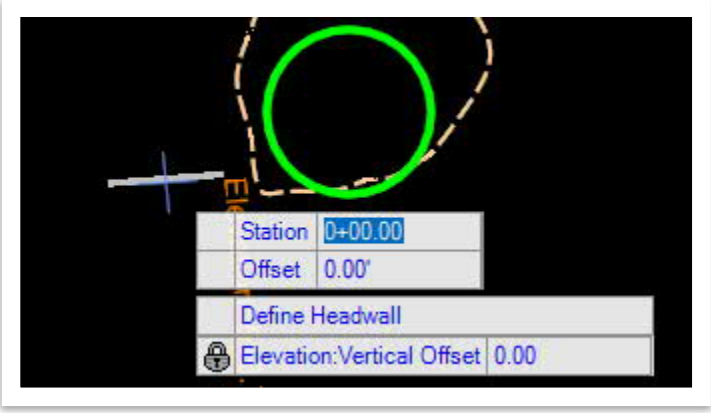
1. Zoom in to the **EW1** green circle, which is where we will place an endwall (outlet). Make sure Civil AccuDraw is still turned on. Open the **Place Node** tool (**Drainage and Utilities >> Layout >> Layout**). Within the **Place Node** dialog box, select the **EW-Straight Round 18-90** feature definition (**Node >> StormWaterNode >> Endwalls >> Cross Drains >> Straight >> Round >>18”-30”; Skew 90**). Match the remaining settings as shown below. **Note:** Toggling on the **Elevation is the Invert** option enables the endwall to daylight at the ground elevation for positive downstream flow. You can ignore the elevation value.



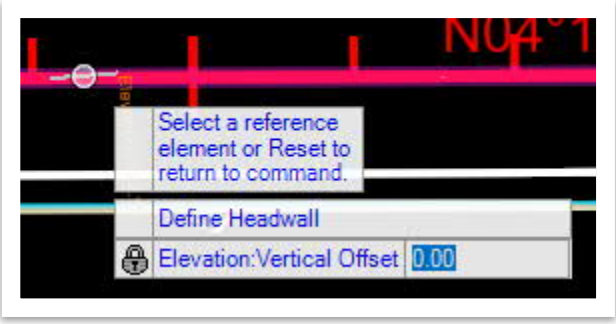
2. Notice the cursor prompt: **Select Reference Element For Node Elevation. <Reset> to Type an elevation**. Select any contour of the referenced complex terrain file and left click to accept.



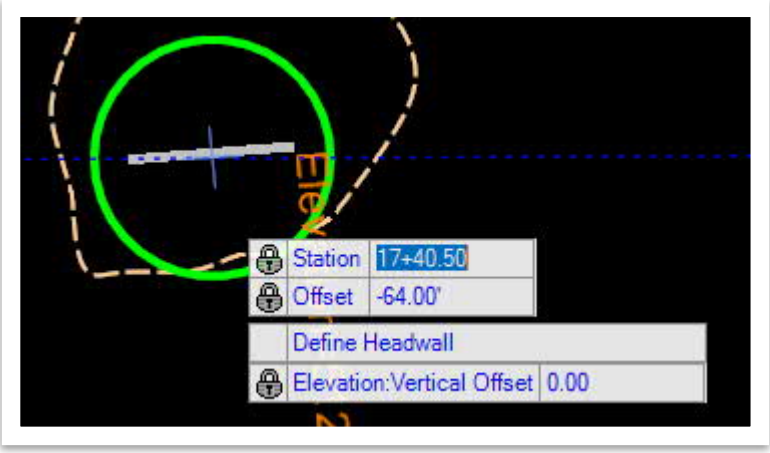
3. The user will be prompted to **Define Headwall**. The **Vertical Offset** should already be set to **0.00**. Hit **TAB** to move the selection from **Vertical Offset** to **Station**. **Note:** You may need to refresh the **Civil AccuDraw** tool if you do not see the Station and Offset options on your cursor. Also, the software prompt still says **Headwall** even though TDOT uses the term Endwall



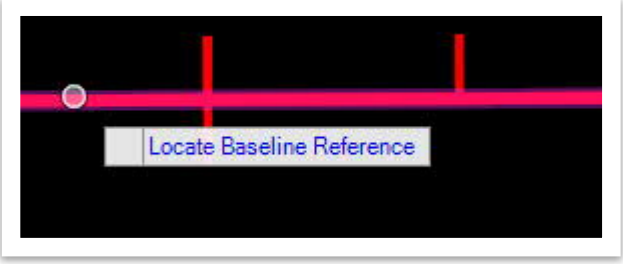
4. Move the cursor towards the location of **EW1**. Press the **letter O** on the keyboard. Notice the cursor prompt: **Select a reference element or Reset to return to command**.
5. Select the **Bowling** centerline.



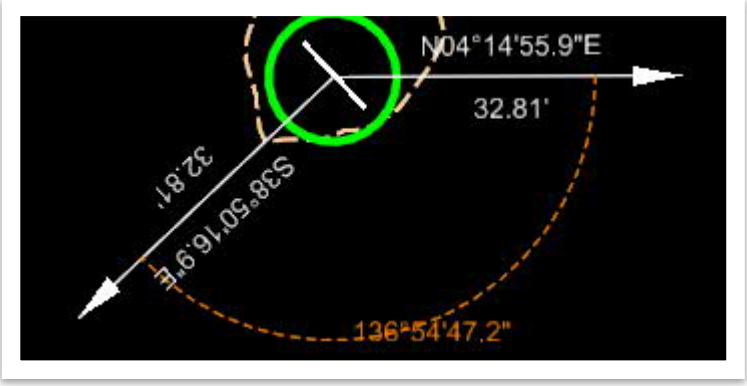
6. Move the cursor towards the location of **EW1**. Hit **TAB** once again to edit the station and offset. Key-in station **17+40.50** and offset **-64.00'** for this exercise. Hit the **TAB** or **Enter** key after each entry so that it locks in place. Left click to accept.



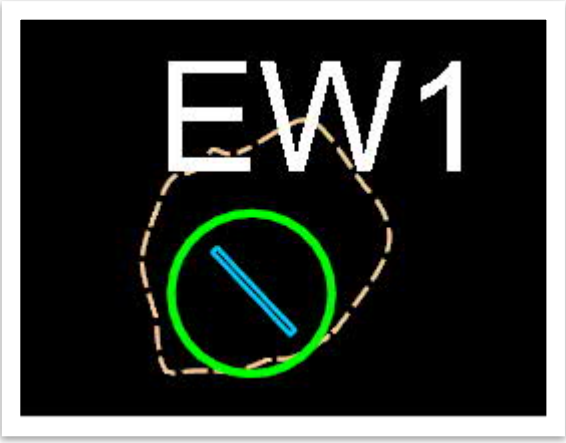
7. Notice the cursor prompt: **Locate Baseline Reference**. Select the **Bowling** centerline. Alternatively, you could select the Baseline Reference in the **Place Node** dialog box. Left click to accept.



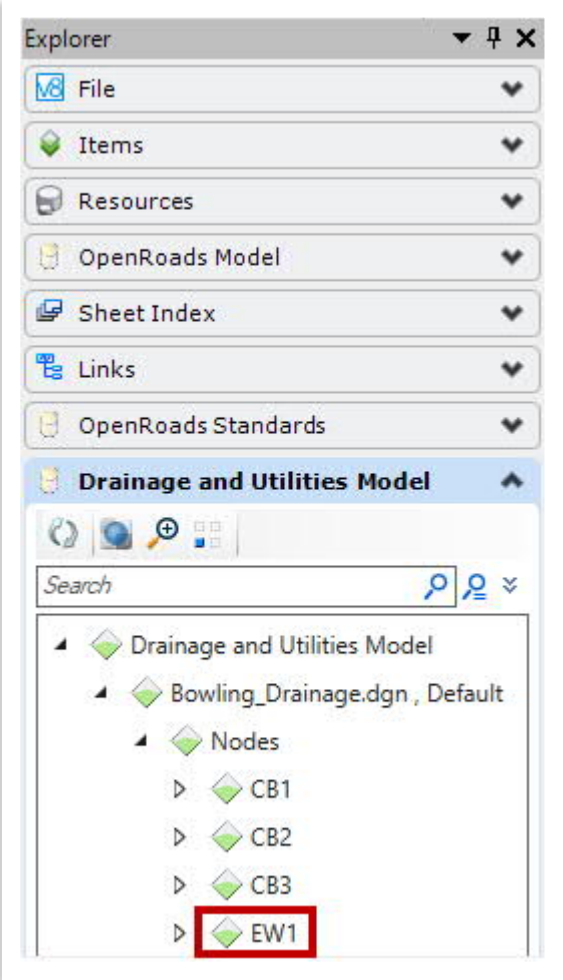
8. Next, you will be prompt to select the **Rotation Mode**. Left click to accept and a new prompt will appear asking you to **Locate Reference Element for Rotation**. Select the **Bowling** centerline and hit **Enter**. Left click to accept the rotation. Alternatively, you could select the Rotation Mode in the **Place Node** dialog box. In this exercise, key-in a rotation angle of **S43^05'12.8"E**. Left click to accept and then hit **ESC** to clear the tool.



9. Notice that the endwall is now placed.



10. Within the **Explorer**, notice that **EW1** is now shown under **Drainage and Utilities Model >> Bowling_Drainage.dgn >> Nodes**.

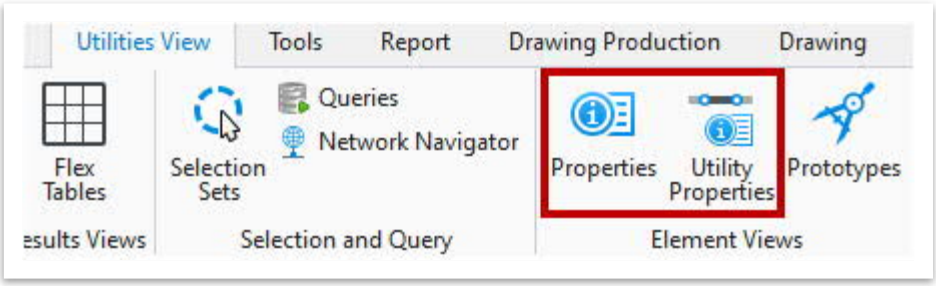


5.5 Lecture: Element Properties vs. Utility Properties

The **Utilities View** tab contains two sets that help mitigate the design file. **It is highly recommended that the user accesses these properties as the model is being built**. These properties can be accessed from any of the following three locations:

- **Utilities View Tab: Properties** and **Utility Properties** for any drainage element (Figure 24).

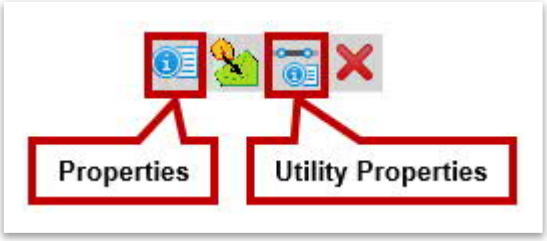
FIGURE 24. UTILITIES TAB – PROPERTIES



  - **Properties:** Provides the general element information (MicroStation/CADD) and attributes for a selected element in the design.
  - **Utility Properties:** Provides the utility-specific properties (inputs and outputs) for any feature in a drainage or non-drainage utility file. Utility Properties display the properties for Subsurface Utilities Engineering and Hydraulic Analysis for the selected element.

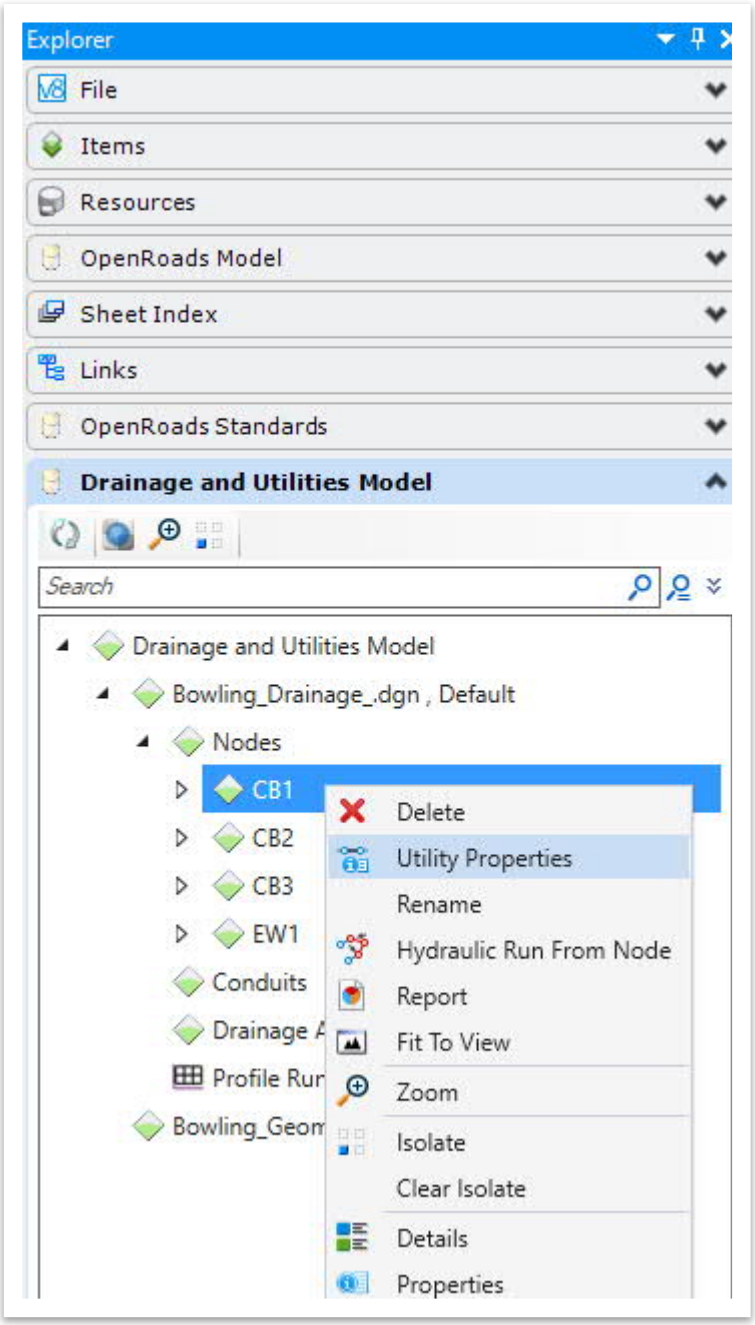
- **Heads-up Display:** Element Properties and Utility Properties may also be accessed in the heads-up display when an element is selected (Figure 25).

FIGURE 25. UTILITY PROPERTIES FROM HEADS-UP DISPLAY



- **Explorer: Utility Properties** can also be viewed from the **Explorer >> Drainage and Utilities Model >> Bowling_Drainage.dgn >> Nodes >> CB1 >> Right Click >> Utility Properties** (Figure 26).

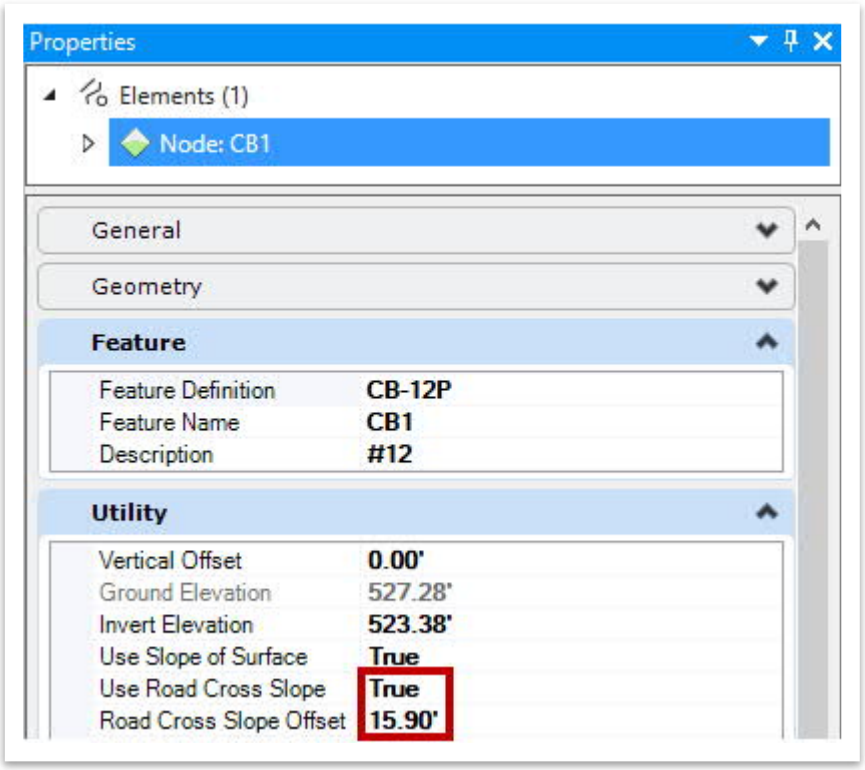
FIGURE 26. UTILITY PROPERTIES FROM EXPLORER



5.5.1 Exercise: Review Node Properties

In this exercise, we will link the nodes to the geometry and view the element **properties** and **utility properties**. We will continue to utilize the same **Bowling_Drainage.dgn** file.

1. Open the **Element Selection** tool and select **CB1** and open the element **Properties** dialog box (**Drainage and Utilities >> Utilities View >> Element Views**). The properties show the type of cell, feature defnition, terrain data and other information related to the structure. Update the following two fields below:
    a. **Use Road Cross Slope:** True (should be set by default)
    b. **Road Cross Slope Offset:** 15.90’ (**Note:** All lanes slope towards the gutter on both sides of the crown of the roadway.)

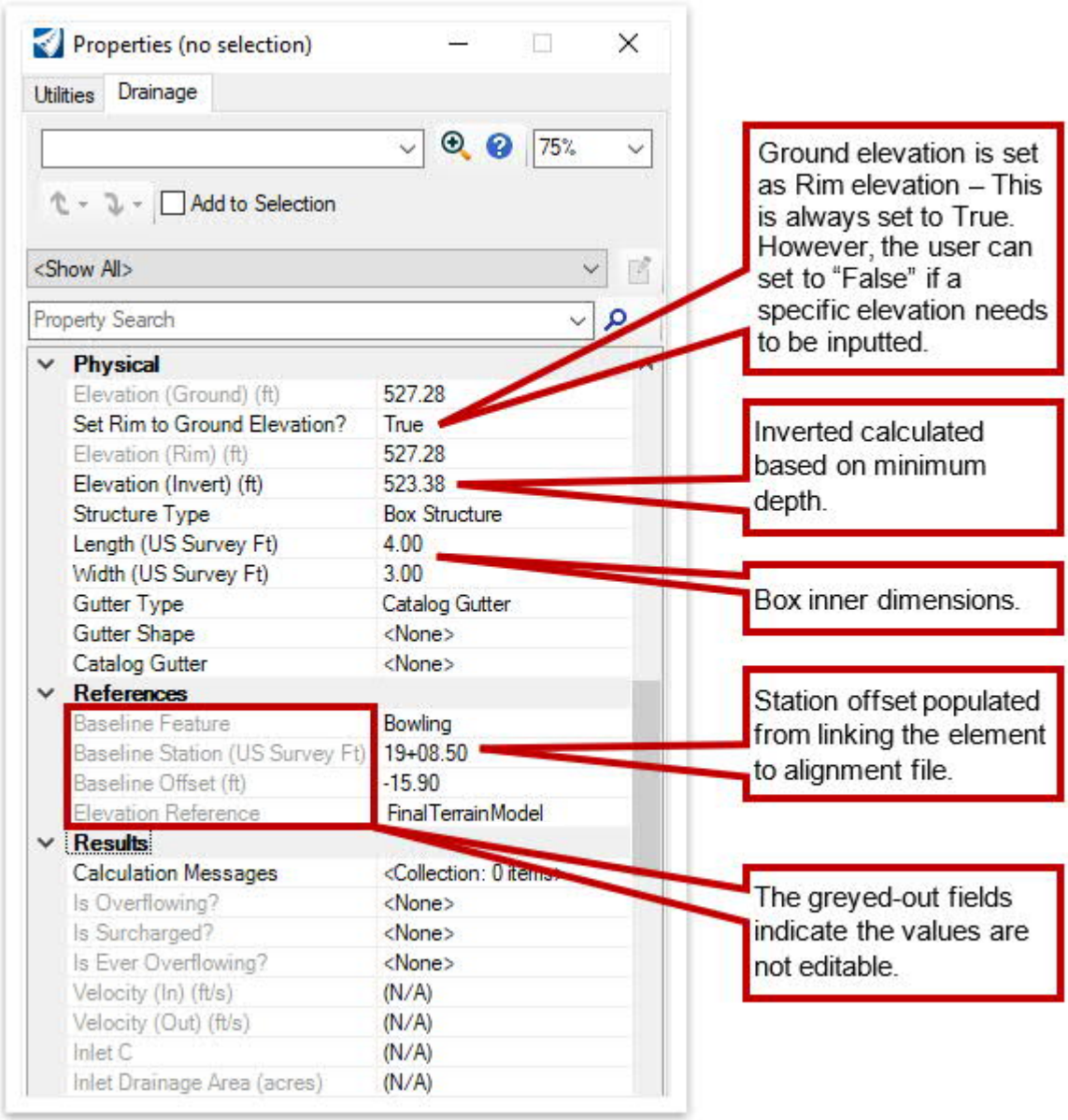


Take Note!

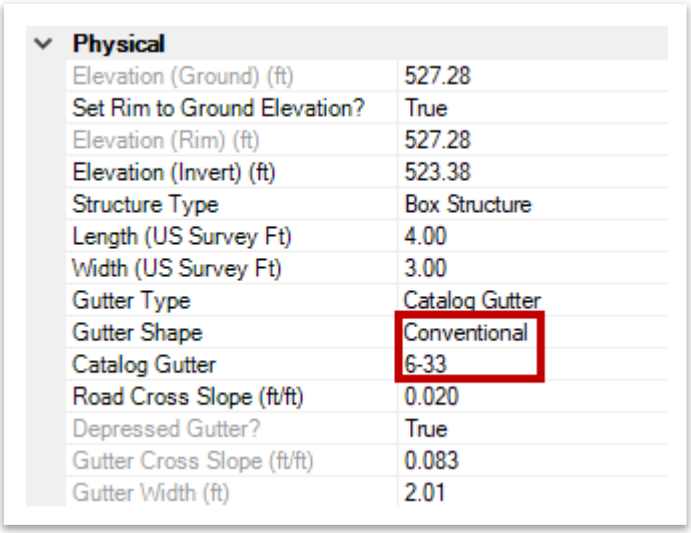
*The **Use Road Cross Slope** field should be set to **True** for all catch basins used within Curb & Gutter sections. This can be read directly from the active terrain.*

*The **Road Cross Slope Offset** field should be the width (ft) of the roadway that flows towards the gutter. The user should review and edit this field for all catch basins in each project.*

2. Next, select catch basin **CB1** (if not still selected) and open the **Utility Properties** dialog box (**Drainage and Utilities >> Utilities View >> Element Views**). Make sure you have the **Drainage** tab activated. The Utility Properties contain the design parameters that run the hydraulics for the model. It also contains the **physical** properties of the nodes, and provides reference to the terrain and geometry files. Review the property descriptions as noted below. **Note:** The blank drop-down menu under the **Drainage** tab works in conjuction with the arrows below. It will show the nodes and conduits that have been selected. Also, your Drai nage tabs (e.g., Physical, References, etc) may be in a different order than what's shown below.



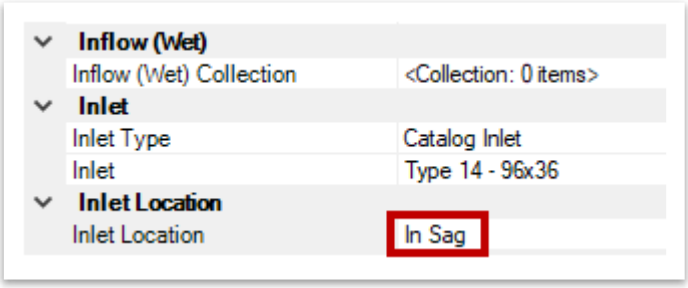
3. Now, we need to update the **Gutter Shape** and **Catalog Gutter** for each of the three catch basins within the Utility Properties under **Physical**. Select **CB1** and then update the following two settings. Repeat and apply the same settings for **CB2** and **CB3**.
    a. **Gutter Shape:** Conventional
    b. **Catalog Gutter:** 6-33 (**Note:** This will vary from project to project so it is important to verify the project's typical section before selecting the appropriate gutter.)



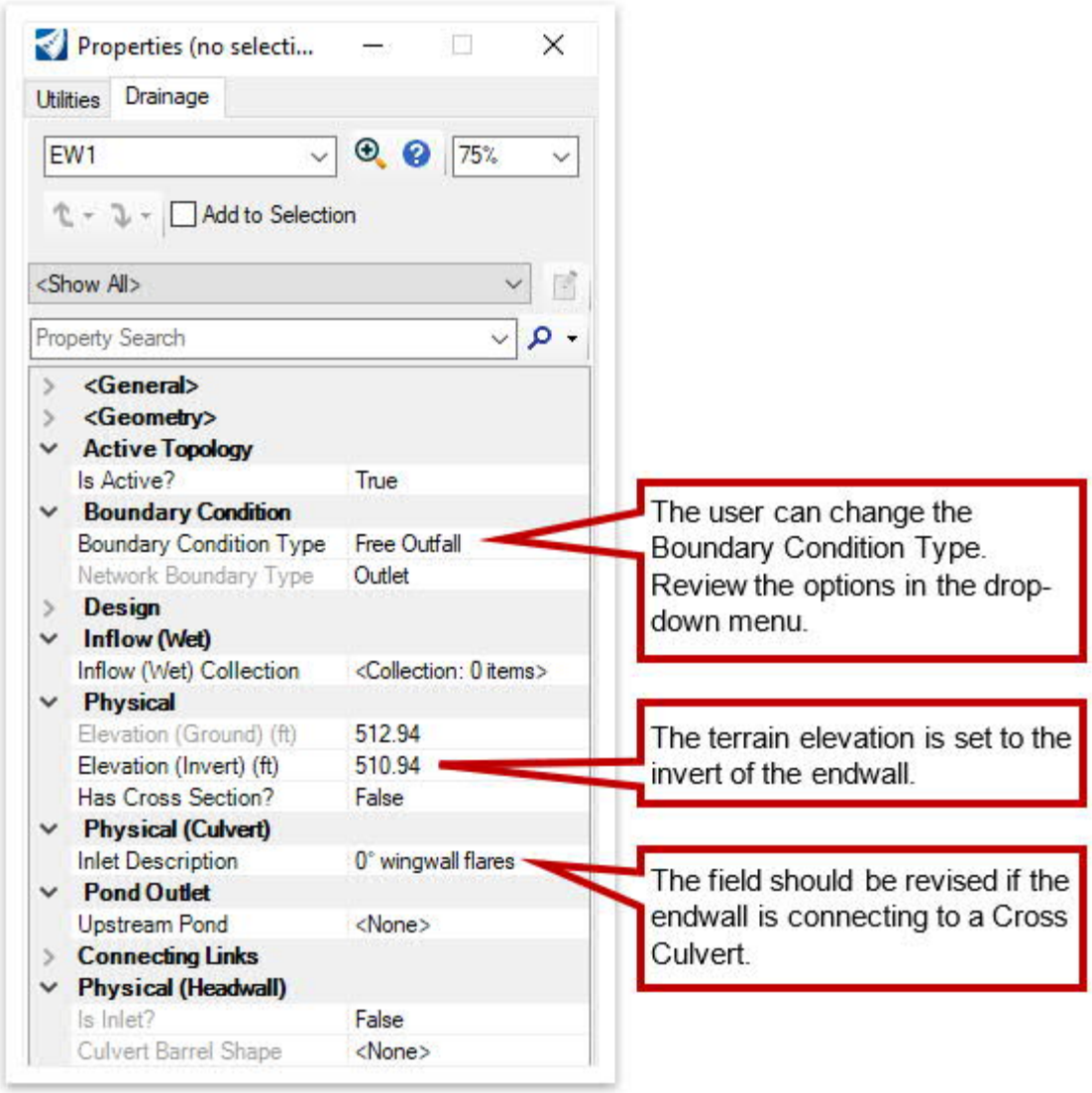
Take Note!

*The user can apply the **Gutter Type** individually to each inlet or by selecting the field within the Flex Tables (discussed in Section 5.10.4).*

4. Next, select catch basin **CB3** (if not still selected) and look at the **Inlet Location** within the Utility Properties. Select **In Sag** from the drop-down menu, which will ensure that **CB3** will not have any by-pass flow and that all flow is captured at this location. **Note:** Once you select In Sag, several fields under Inlet Location will disappear.



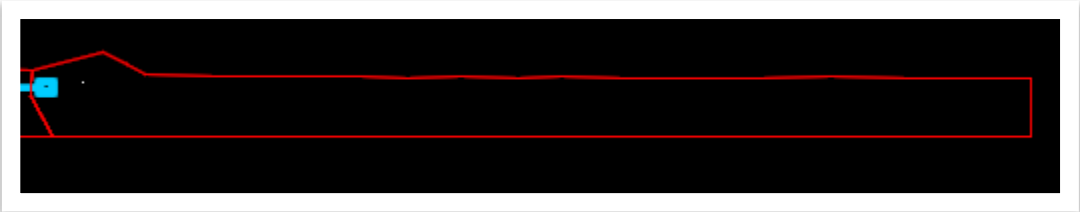
5. Lastly, review the element **Properties** and **Utility Properties** for endwall **EW1** and then close the dialog box once finished. **Note:** The **Utility Properties** are shown below.



5.6 Lecture: Catchments

Catchments are interactive polygon type features containing information and drainage properties that allow them to be utilized in a drainage network design. They act as drainage areas in which all drainage attenuates to a common point. Figure 27 shows an example of a catchment area in **2D** view.

FIGURE 27. CATCHMENT AREA



Drainage catchments get the rainfall into the collection and conveyance systems. Within ORD, there are various options for delineating a catchment area:

- Delineation along with the **Place Node** tool (most common method) – Discussed in Exercise 5.3.1.
- Picking points (Pick Points method).
- Choose a shape that acts as the boundary (Pick Shape method).
- Flood the area between graphics (Flood Fill method) – Not commonly used.

Typically, catchment areas are automatically delineated when catch basins or inlets are placed in the model. The user will need to manually delineate catchment areas (using the **Pick Shape** or **Pick Points** method within the **Place Catchment** tool) when the model involves other nodes such as endwalls or ditches that take flow.

5.6.1 Exercise: Placing Catchments

In this exercise, we will place three catchments manually (#1 – #3) by selecting pre-drawn areas using the **Pick Shape** method within the **Place Catchment** tool. We will continue to utilize the same **Bowling_Drainage.dgn** file.

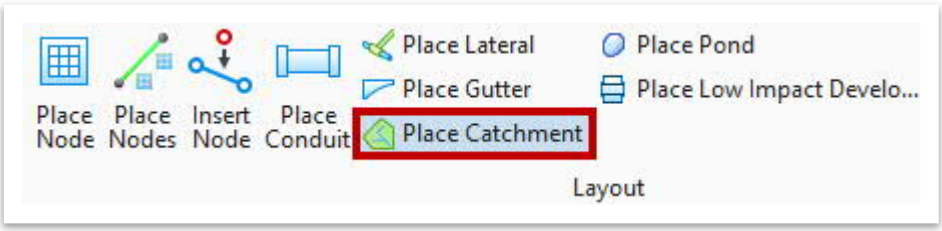
1. First, turn off the following reference files: **Bowling_Geometry.dgn** and **Bowling_Corridor.dgn**. This will allow us to see the catchment shapes and IDs more clearly, which are within the **Bowling_Catchment samples.dgn** reference file on the **SUR - DRG - Area Shapes** level. For the **Place Catchment** tool to work, the shapes need to be in the active file so go ahead and copy Catchments **1-3** and paste them.

Take Note!

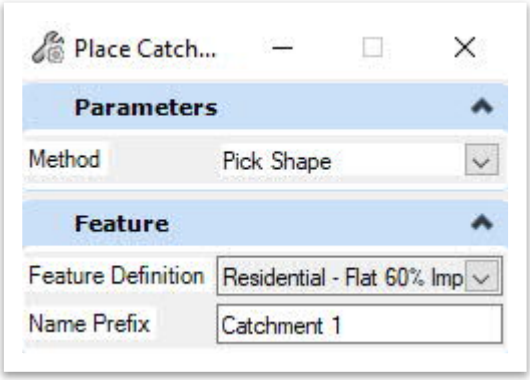
*Each land use category is assigned a **land use coefficient** that represents the runoff characters of that category. These runoff coefficients are assigned to Catchments. When defining catchment areas based on their land use and type, the selected Feature Definition will apply the associated **Land Use** / **Runoff Coefficient**.*

These values are incorporated within predefined prototypes in the TDOT Drainage Features Defs.dgnlib. These prototypes are built based on Table 4-2 of the TDOT Drainage Manual.

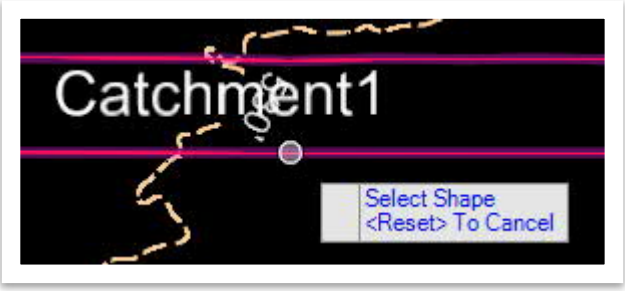
2. Now, open the **Place Catchment** tool (**Drainage and Utilities >> Layout >> Layout**).



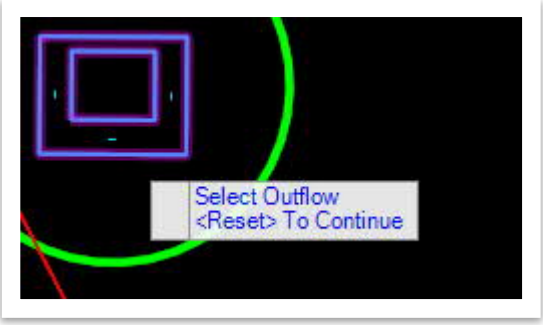
3. Within the **Place Catchment** dialog box, select the following settings.
    a. **Method:** Pick Shape
    b. **Feature Definition:** Residential - Flat 60% Imp (**Drainage Area >> Catchment >> Urban**)
    c. **Name Prefix:** Catchment 1



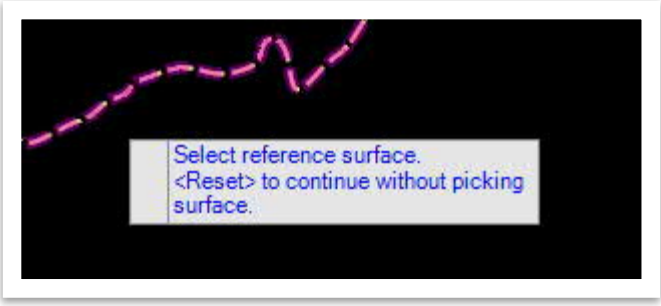
4. Notice the cursor prompt: **Select Layout Method**. Left click to accept **Pick Shape**. Notice the next cursor prompt: **Select Shape**. Select **Catchment 1**.



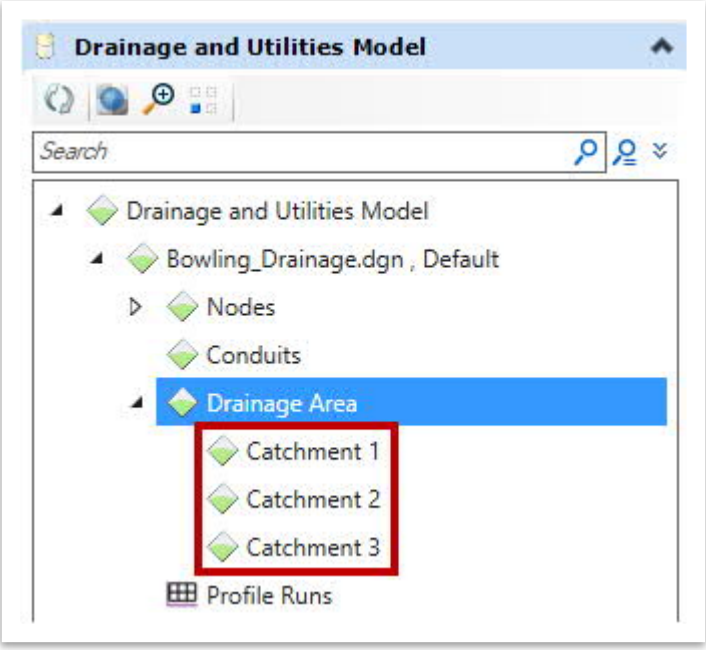
5. Notice the next cursor prompt: **Select Outflow**. Select the appropriate catch basin draining the area (**CB1** in this case).



6. Notice the next cursor prompt: **Select reference surface**. Select any contour within the complex terrain file and left click to accept.



7. Repeat Steps 3-6 to place Catchment #2 and #3. Make sure to update the **Name Prefix** to **Catchment 2** and **Catchment 3** respectively before placing. Then, hit **ESC** to clear the tool.
8. Confirm that the **three** new drainage areas (**Catchments 1-3**) have been added to the **Drainage and Utilities** model within the **Explorer**.



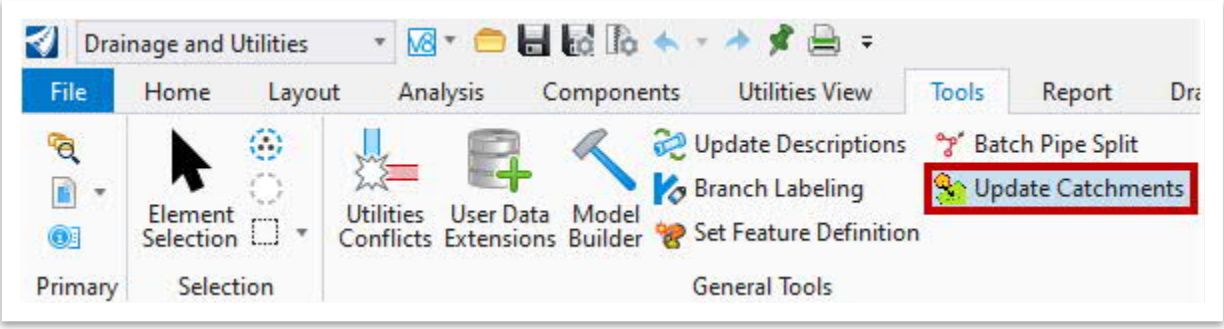
Take Note!

Catchment delineation can be done in combination (manual and automatic). This is useful for when there is a mixture of flow type (e.g., roadway and offsite drainage).

5.6.2 Update Catchments

New functionality has been added in the latest software version to update multiple delineated catchments. The **Update Catchments** tool (Figure 28) uses a selection set of inlets and will update any catchments that have been automatically delineated at one time, thus resulting in a more efficient workflow.

FIGURE 28. UPDATE CATCHMENTS TOOL



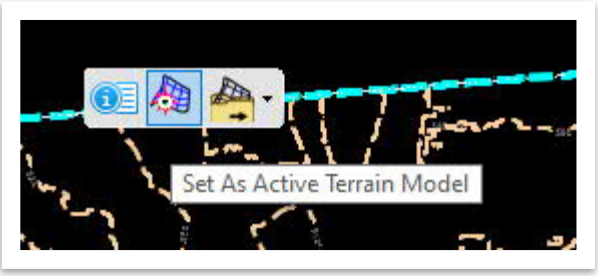
In addition, the user can now apportion catchment runoff to gutters that flow to a sump inlet. If a catchment is connected to a sump inlet, which has two gutters connected to it, then the flow from the catchment should be apportioned between the gutters, as opposed to the same flow being assigned to both gutters.

For manually delineated catchments, the user would select the catchment and then update the necessary vertex/vertices, thus resulting in an updated area (acres) in the Utility Properties under **Results**.

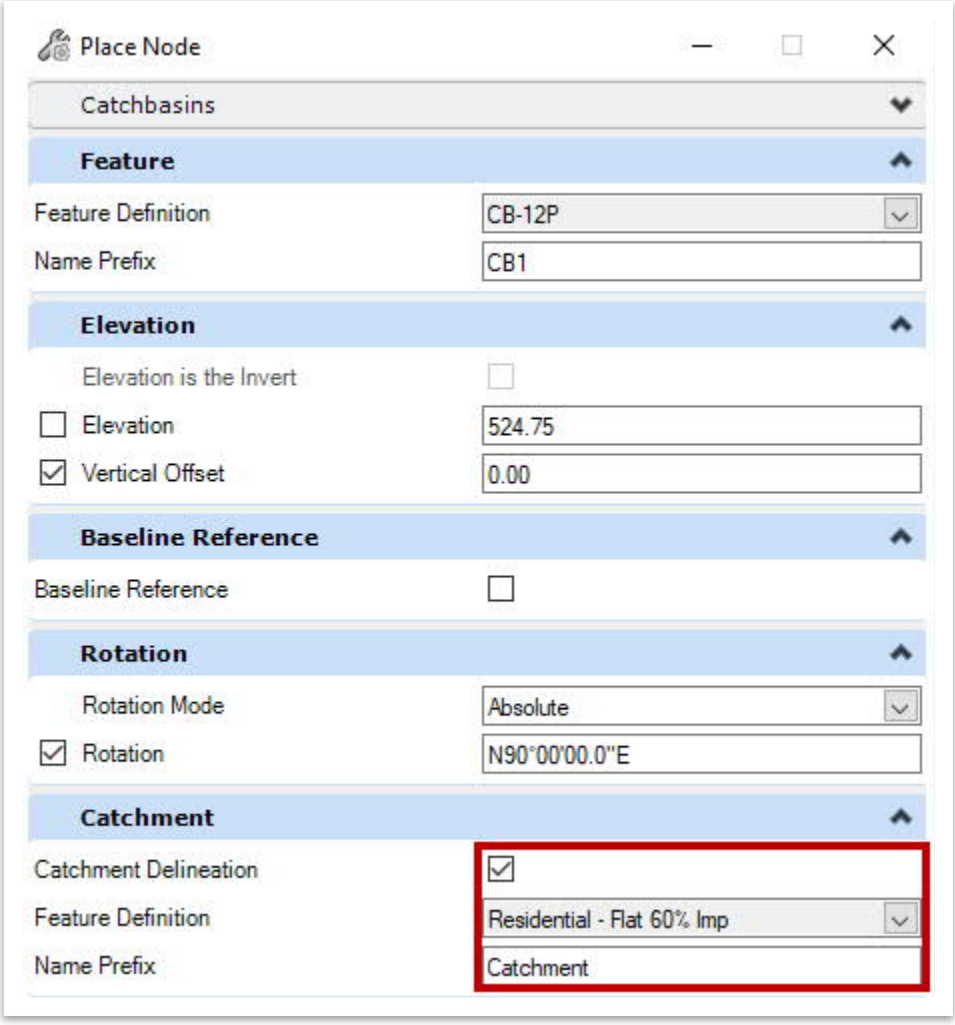
5.6.3 Exercise: Update Catchments

In this exercise, we will add two additional catch basins with automatic catchment delineation toggled on, and then use the Update Catchments tool. As a reminder, the catch basins that were placed earlier in the chapter had this toggle turned off.

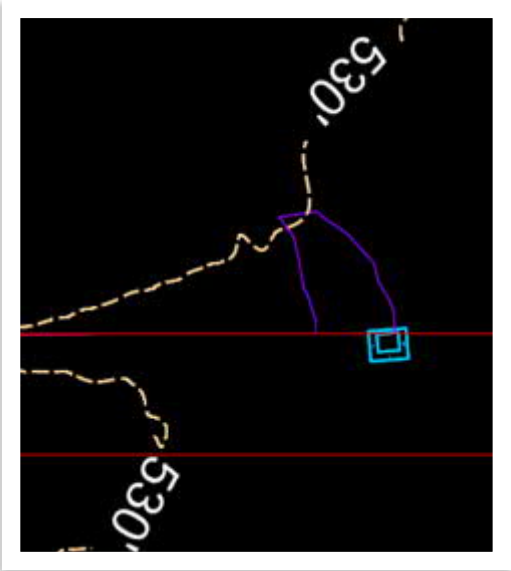
1. Within the **Bowling_Drainage** file, go to **File >> Save As** and name the file **Bowling _Drainage_Update Catchment**.
2. Select the complex terrain boundary and then **Set As Active Terrain Model**.



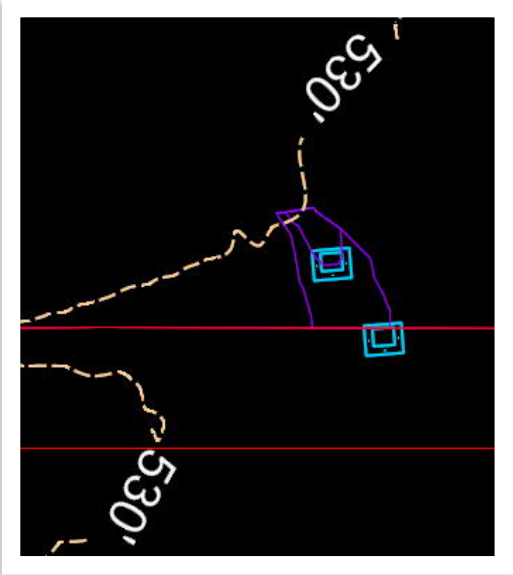
3. We will now add two additional catch basins so that we can apply the Update Catchment tool properly. Open the **Place Node** tool (**Drainage and Utilities >> Layout >> Layout**). Within the **Place Node** dialog box, select the following settings and then match the remaining as shown below. **Note:** You can ignore the elevation value.
    a. **Feature Definition:** CB-12P (**Node >> StormWaterNode >> Storm Drains >> Proposed >> Type 12-13**).
    b. **Name Prefix:** CB1
    c. **Catchment Delineation:** Toggle on.
    d. **Catchment Feature Definition:** Residential - Flat 60% Imp (**Drainage Area >> Catchment >> Urban**).
    e. **Catchment Name Prefix:** Catchment



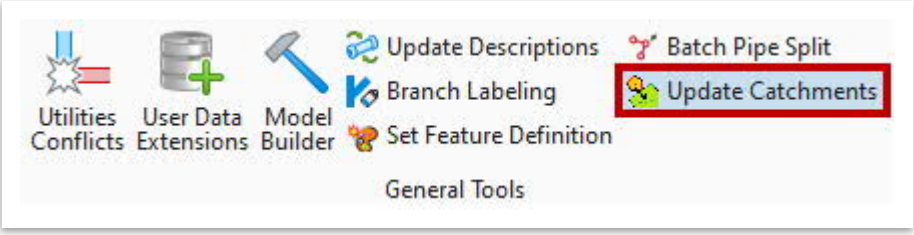
4. Notice the cursor prompt: **Select Reference Element for Node Elevation. <Reset> to Type an Elevation**. Select any contour of the referenced complex terrain file and left click to accept. Move the cursor anywhere within **Catchment3** and the left click through the prompts to accept placement and notice the automatic delineation. **Note:** Your delineation area will look different depending on where you placed the catch basin.



5. Place a second catch basin within the delineated area of the first catch basin using the same structure type and catchment.



6. Next, select both catch basins and the open the **Update Catchments** tool (**Drainage and Utilities >> Tools >> General Tools**).



7. Notice the cursor prompt: **Data Point To Accept Selection Set Escape To Reject Selection Set**. Left click to accept and then notice the catchments automatically update.

Take Note!

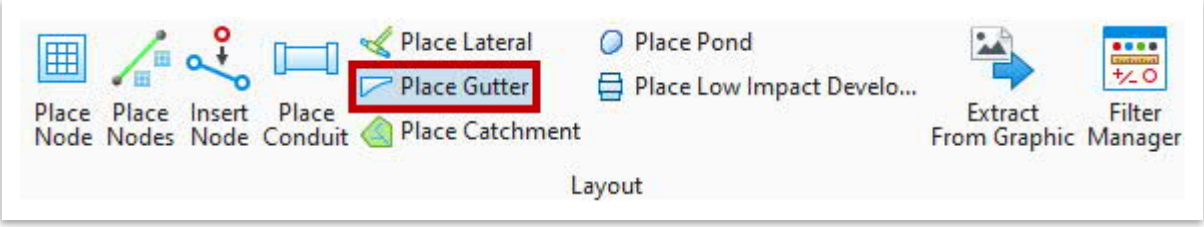
*If you hover over a structure that has automation delineation, the **Update Catchment For Inlets** tool should show in the ribbon.*

5.7 Lecture: Gutters

Gutters are defined between on-grade catch basins located in the gutter pan to assign the bypass flow to the downstream catch basin. The **Place Gutter** tool (Figure 29) can be accessed here: **Drainage and Utilities >> Layout >> Layout**. It is used to define:

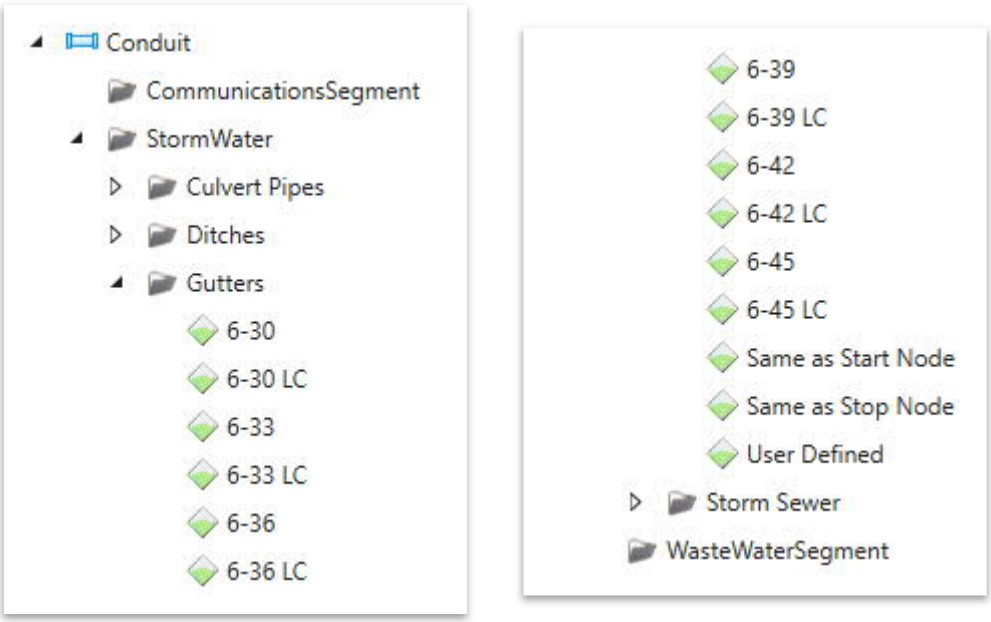
- The path of bypass flow between inlets/catch basins.
- The shape of gutter between inlets.

FIGURE 29. PLACE GUTTER TOOL



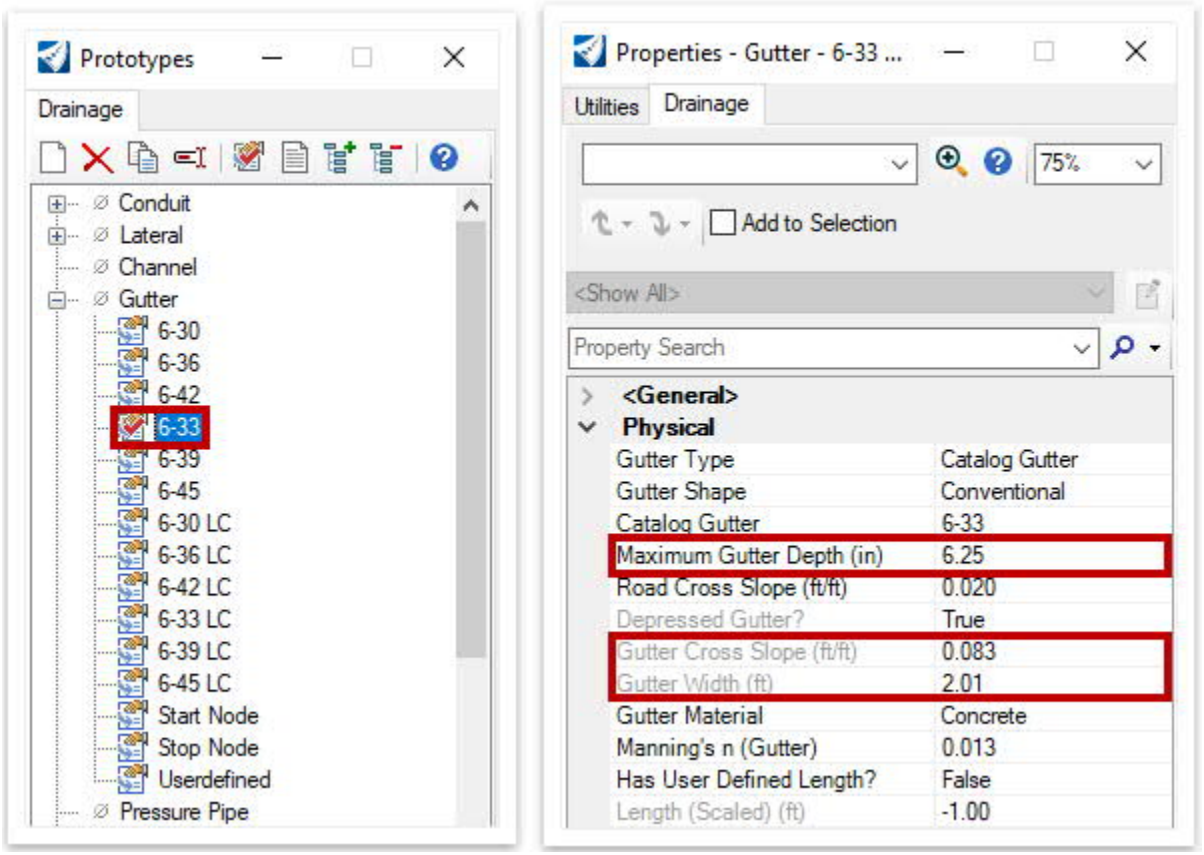
The TDOT Drainage Feature Defs.dgnlib provides **multiple** feature definitions for gutters (Figure 30). The gutter types and nomenclature correspond to TDOT's gutter types defined within the TDOT standard drawings. The feature definitions represent the dimensions for the gutters per the TDOT standard drawings.

FIGURE 30. GUTTER FEATURE DEFINITIONS



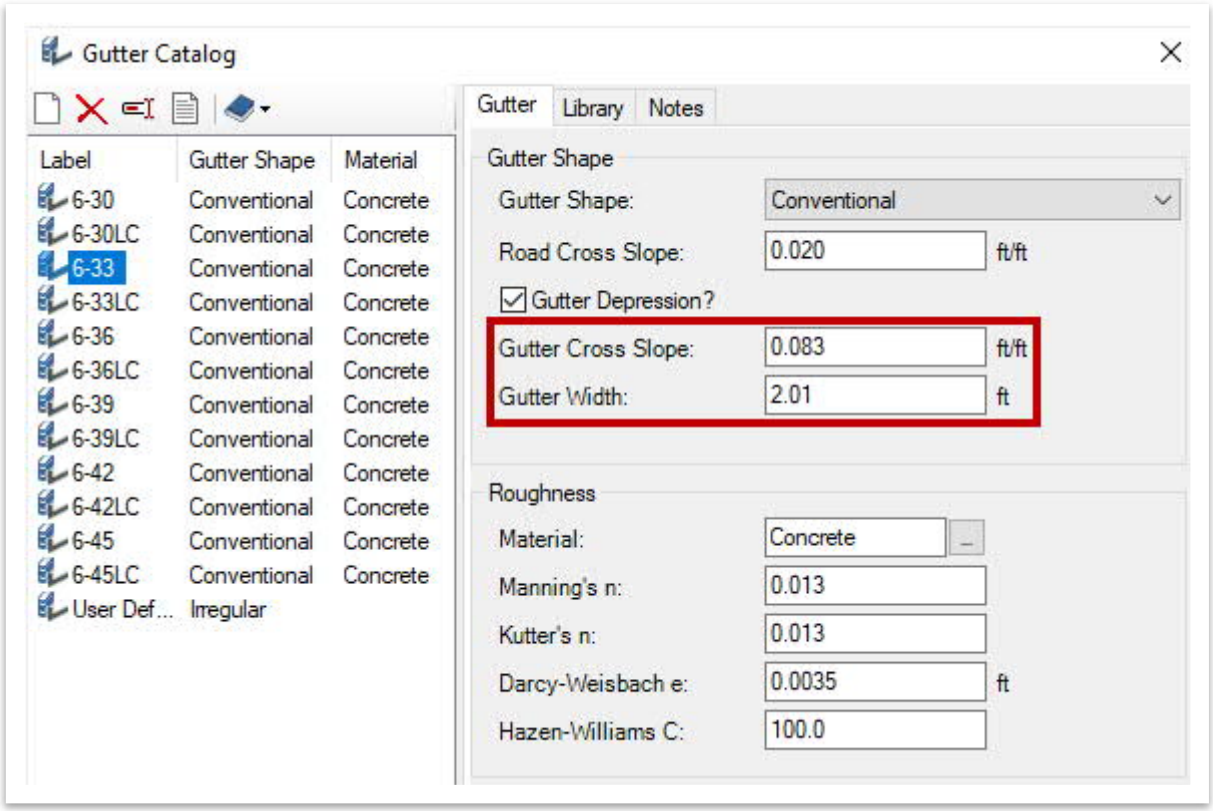
Each gutter feature definition is linked to a **Gutter** prototype (**Drainage and Utilities >> Components >> Catalog >> Prototypes**) which contain the corresponding dimensions and slopes per TDOT standard drawings. The **6-33** gutter prototype and its properties are shown in Figure 31. The user will not need to edit any prototype.

FIGURE 31. GUTTER PROTOTYPE EXAMPLE



Each prototype is linked to the corresponding gutter within the **Gutter Catalog** (**Drainage and Utilities >> Components >> Catalog >> Catalog**). The **6-33** gutter and its properties are shown within the gutter catalog in Figure 32. The gutters listed match the properties of those in the prototypes and appear within the Utility Properties of a catch basin or inlet. The user can then assign the corresponding gutter to the catch basin.

FIGURE 32. GUTTER CATALOG



Take Note!

*The **Road Cross Slope** value will be overwritten by the dynamic road cross slope obtained by the inlet.*

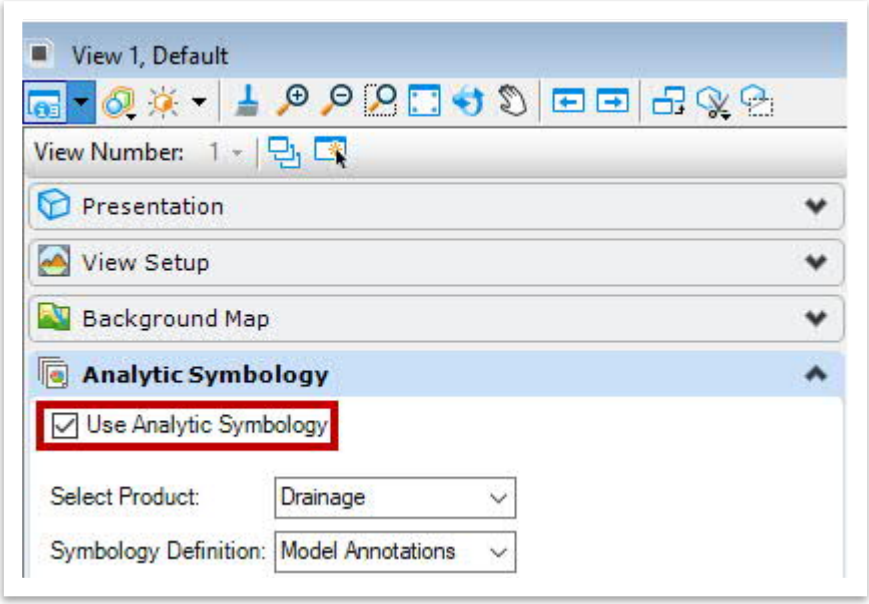
This tool also helps the user to **calculate the gutter spread calculations** and **design inlet spacing** prior to building the complete network. The next exercise illustrates the spread and bypass flow analysis using the place gutter tool. The spreads at all catch basins, including the first one, are summarized in the results and the spread width of the runoff is visualized on the dgn.

5.7.1 Exercise: Placing Gutter Flow

In this exercise, we will assign gutters to the catch basins within the drainage model, view the analytic symbology, and then run the gutter spread analysis. We will open back up the **Bowling_Drainage.dgn** file.

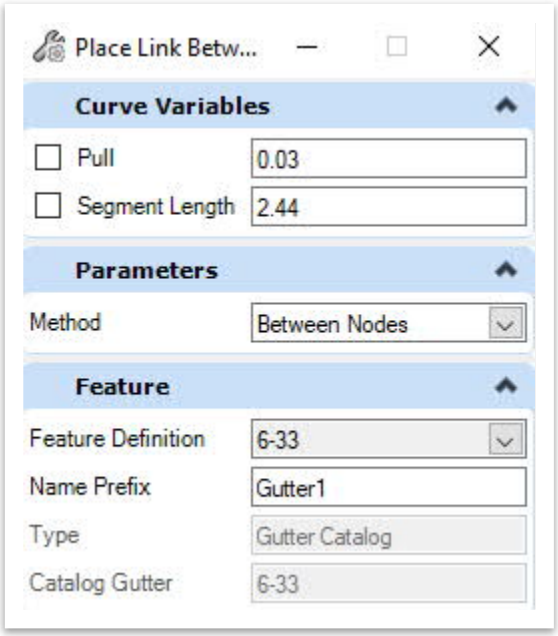
1. Open the **Place Gutter** tool (**Drainage and Utilities >> Layout >> Layout**).

2. Now, open **View Attributes** and toggle on the **Use Analytic Symbology** box. **Note:** The **Analytic Symbology** only needs to be turned on when viewing gutters. No other storm drain feature needs this view.



3. Zoom in and notice the **model annotations** that were automatically created at each structure. They are scaled to whatever the active drawing scale is (1”=50’ in this exercise) but could be changed if necessary. The **Rim** and **Inv** elevations are shown below for **CB1**. **Note:** These annotations are only used during modeling and will not be shown on the final plans.

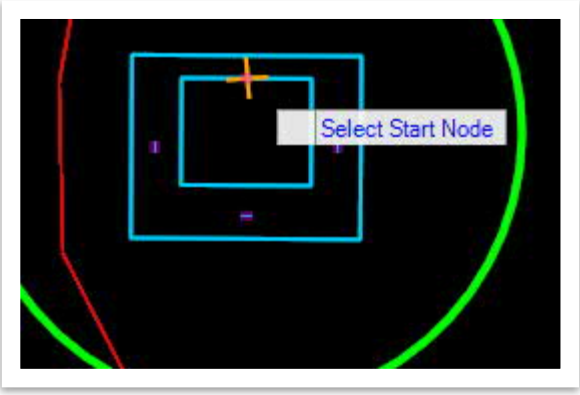
4. Within the **Place Link Between Nodes** dialog box, select the following settings.
   a. **Method:** Between Nodes
   b. **Feature Definition:** 6-33 (**Conduit >> Stormwater >> Gutters**)
   c. **Name Prefix:** Gutter1



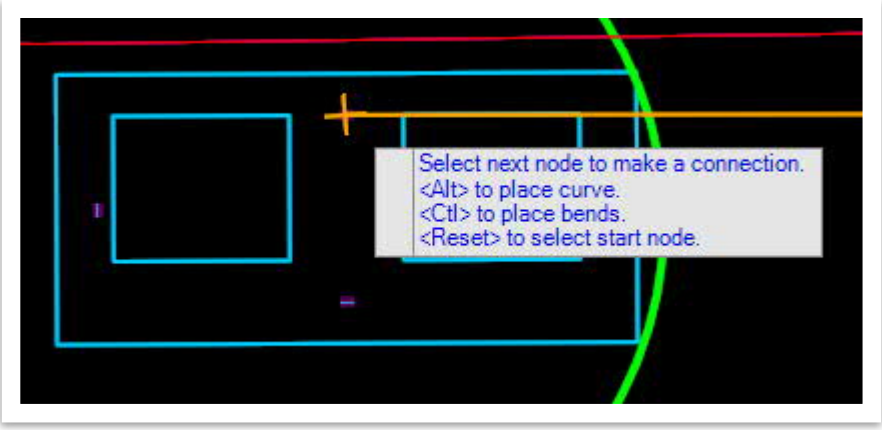
Take Note!

*The **Between Nodes** method carries the bypass flow to the downstream node. This is the preferred method to select since the proposed design of the gutter connects the upstream inlet to the downstream inlet. The **Trace Slope** method carries the bypass flow along the downstream flow direction based on the terrain and may be used on small projects such as bridge replacement jobs or short length widening projects, where the terrain can be more precise. If the **Trace Slope** method is used, it creates intermediate sections along a gutter element by sectioning a terrain model and in some locations, will not be representative of the actual conditions on the ground.*

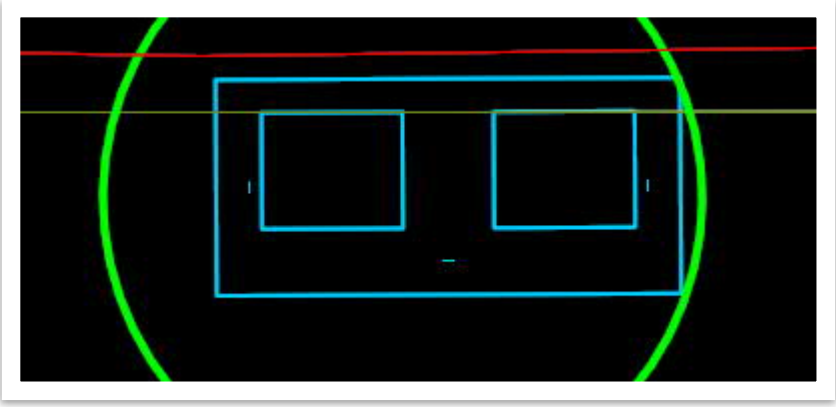
5. Notice the cursor prompt: **Select Start Node**. An **orange crosshair** should appear on the cursor once you hover over a **connection point**. For **CB1**, snap to the top connection point as shown below, which is where the link will start.



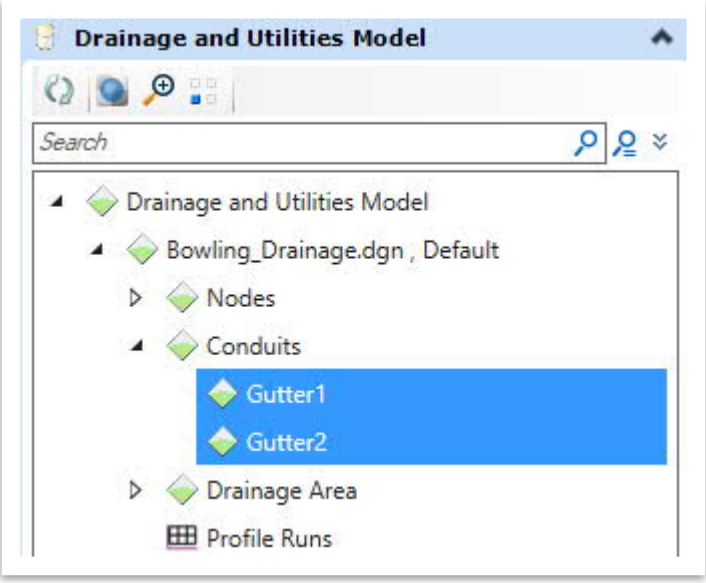
6. Notice the next cursor prompt: **Select next node to make a connection**. For **CB3**, snap to the top connection point as shown below. The **green** gutter should now be placed between **CB1** and **CB3**. Bypass flow will go to CB3.



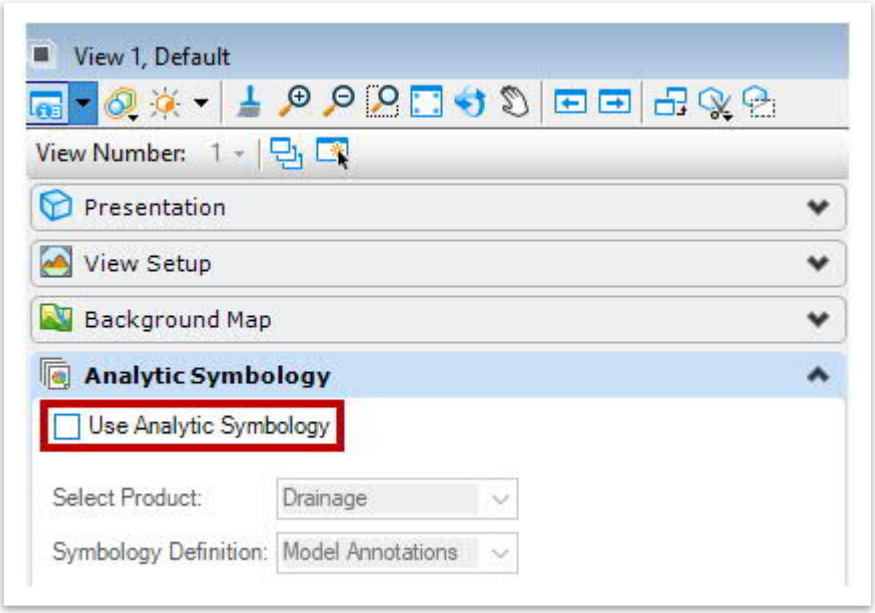
7. Next, place another gutter in the same manner starting at **CB2** and connecting to **CB3**. Bypass flow will again go to CB3. Once completed, hit **ESC** to clear the tool.



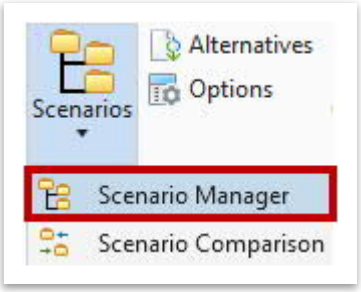
8. Confirm that the two new gutter conduits (**Gutter1** and **Gutter2**) have been added to the **Drainage and Utilities** model within the **Explorer**. In general, all **on-grade inlets** will need a gutter to be assigned on a given project. **Note:** If the conduits are not showing, click the **Refresh** icon under the **Drainage and Utilities Model** header. If the conduits still do not show, close and re-open the **Explorer**.



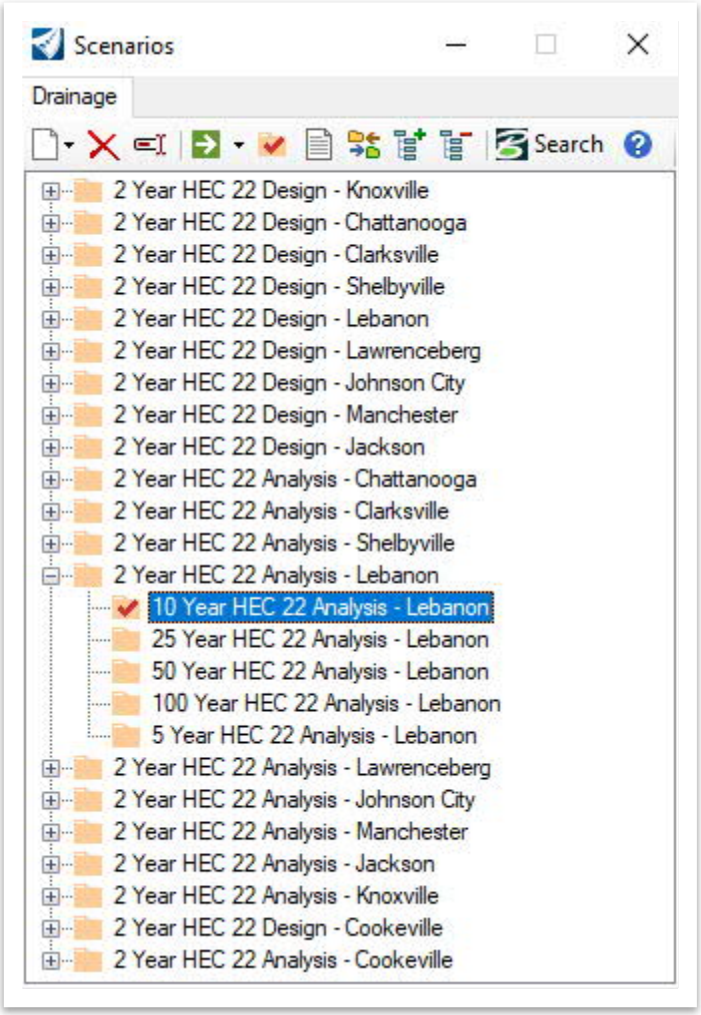
9. Go ahead and open **View Attributes** once again and toggle off the **Use Analytic Symbology** box.



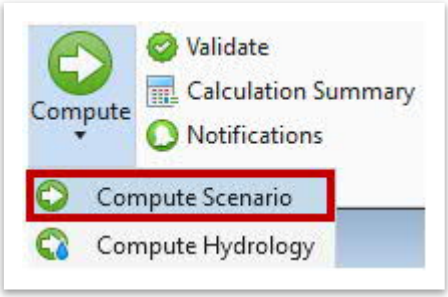
10. Now, we will run the **gutter spread** analysis. Open the **Scenario Manager** tool (**Drainage and Utilities >> Analysis >> Calculation >> Scenarios**).



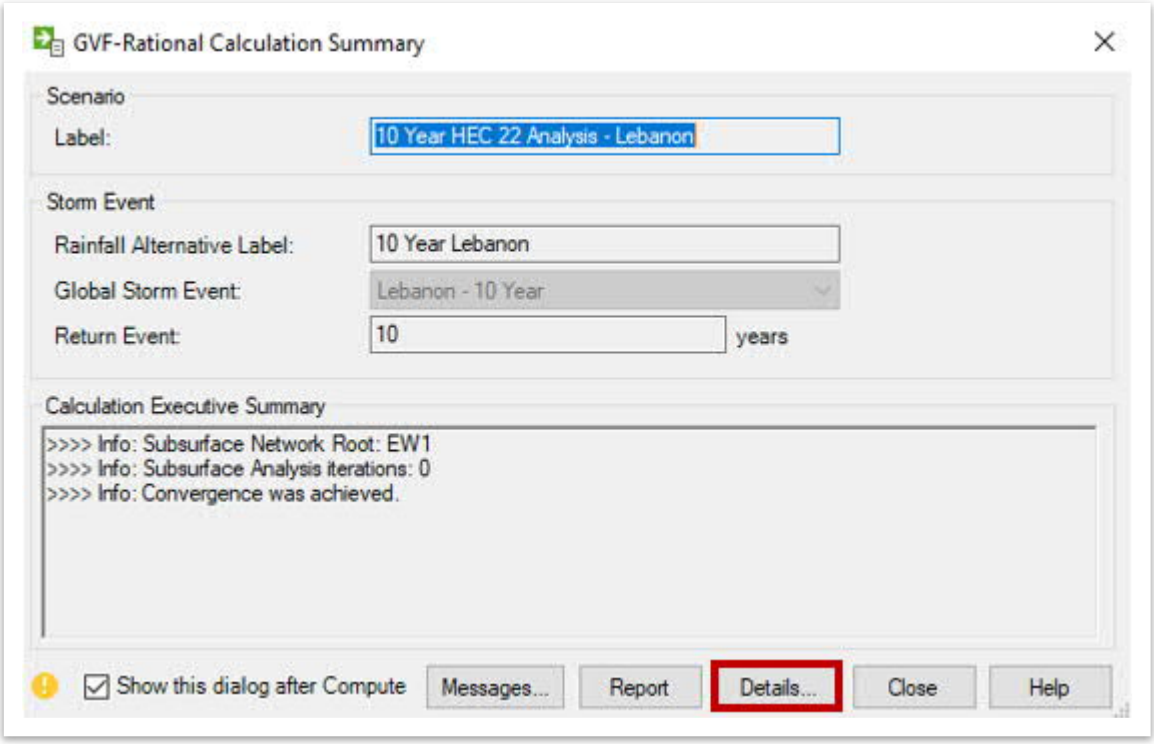
11. Expand the **2 Year HEC 22 Analysis - Lebanon** folder and right-click on **10 Year HEC 22 Design - Lebanon** and select **Make Current**, which will add a red check mark next to the scenario. Close the Scenario Manager once you are done.



12. Open the **Compute Scenario** tool (**Drainage and Utilities >> Analysis >> Calculation >> Compute**).



13. The **GVF-Rational Calculation Summary** has now been generated. Go ahead and click on the **Details** button.



14. Click on the **Inlet Summary** tab at the top and notice the summarized data, including the spread calculations.

Label	Inlet Type	Catalog Inlet Type	Catalog Inlet	Flow (Captured) (cfs)
CB1	Catalog Inlet	Combination	Type 12 - 48x36	0.47
CB2	Catalog Inlet	Combination	Type 12 - 48x36	0.15
CB3	Catalog Inlet	Combination	Type 14 - 96x36	0.85

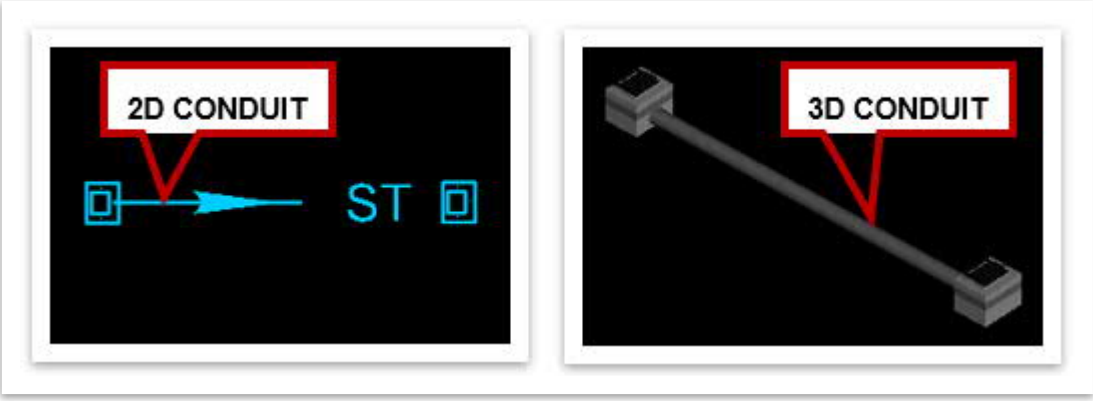
Flow (Total Bypassed) (cfs)	Bypass Target	Capture Efficiency (Calculated) (%)	Depth (Gutter) (in)	Spread / Top Width (ft)
0.00	CB3	100.0	1.6	1.64
0.00	CB3	100.0	1.0	1.03
0.00	(N/A)	100.0	2.6	4.66

15. Notice that the spread at **CB3** meets the criteria since it is lower than **8.0'**, which is set in the standards that load with ORD per Section 7.03.3.7 of the TDOT Drainage Manual. If the spread was higher, you would need to add more inlets downstream of CB1 or CB2 to capture additional runoff or physically move CB1 or CB2 further downstream, so that the spread at CB3 would meet the criteria. **Note:** This process will be a trial and error method until an acceptable spread is obtained.
16. Close the **Calculation Detailed Summary** window and then the **GVF-Rational Calculation Summary** window. You likely will see a **Civil Message Center** open on either the side or bottom of the screen. Go ahead and close that as well for now. We will learn more about **Scenarios** in Section 5.11.1.2.

5.8 Lecture: Conduits

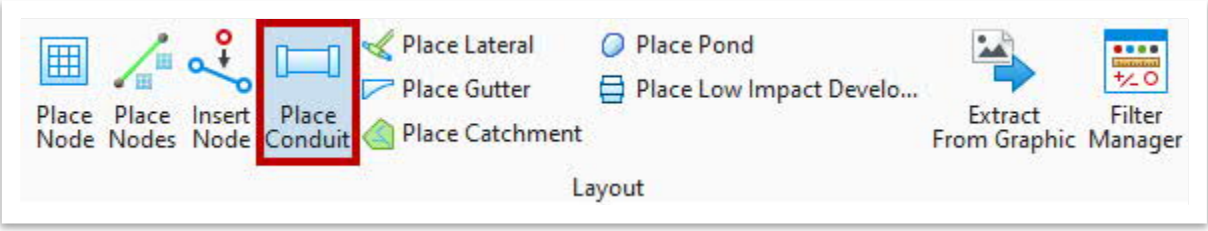
Conduits are interactive line type features containing information and drainage properties that allow them to be used in a drainage network design for runoff conveyance. Conduits act as the medium by which storm drainage travels between nodes. The most common conduits are **storm drains**, which may be **circular**, **rectangular**, **elliptical**, **arch pipes**, etc. Storm drain pipes also come in various materials such as **concrete**, **corrugated metal pipe**, **HDPE**, etc. Figure 33 shows a sample conduit in both **2D** and **3D** views when laid out in a drainage utility model.

FIGURE 33. SAMPLE CONDUIT



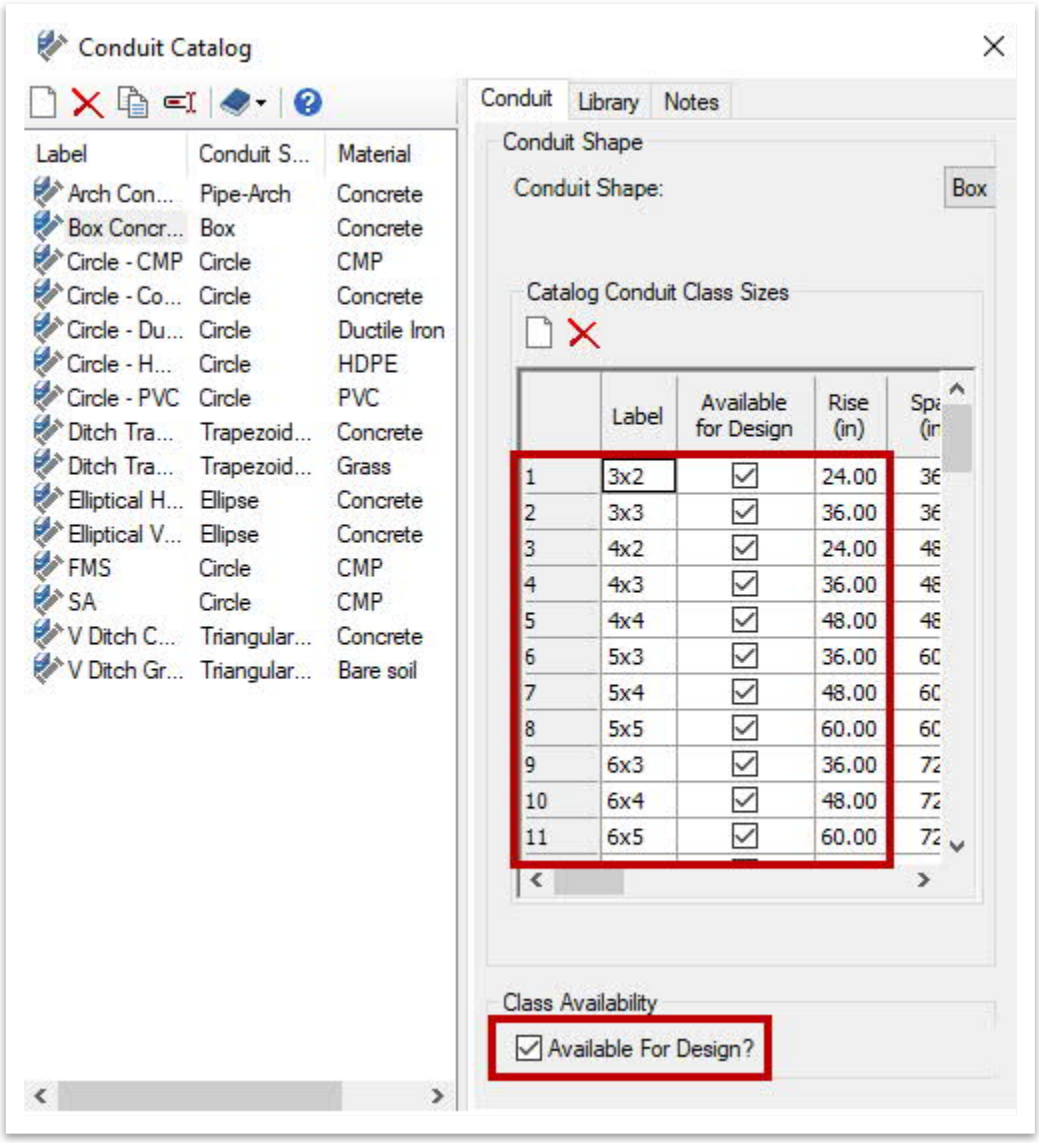
The **Place Conduit** tool (Figure 34) allows the user to place a conduit such as a pipe or ditch between nodes and can be accessed here: **Drainage and Utilities >> Layout >> Layout**. The user selects the type of conduit by selecting the appropriate Feature Definition. The **size of the conduit is determined using the description field**. The user selects the start node and the end node to layout a conduit. The **slope is automatically determined** by the software unless the user has defined a slope in the tool dialog box.

FIGURE 34. PLACE CONDUIT TOOL



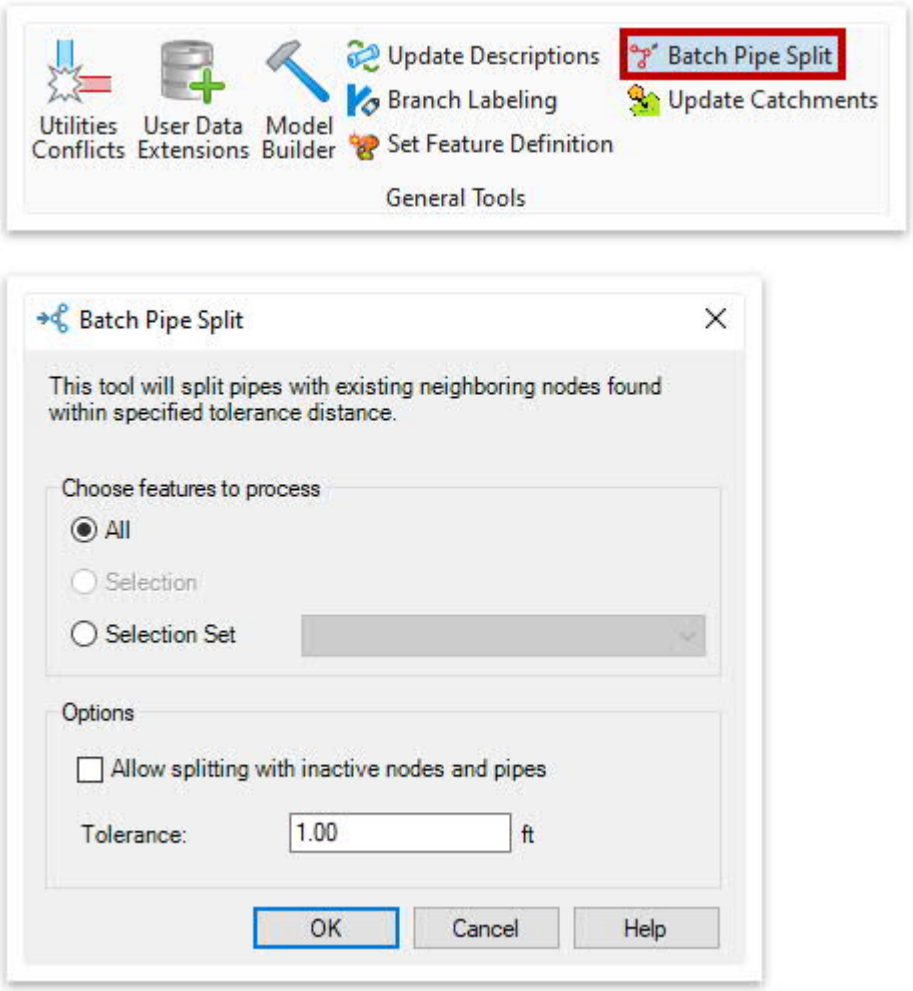
The **Conduit Catalog** lists all common conduit sizes and materials available for design. The user should review this catalog and specifically the **Class Sizes** (Figure 35), and only checkmark the sizes under **Available for Design** that are required for the project. The software will only use the selected pipes to compute the model under the **Design** calculation option.

FIGURE 35. CONDUIT CATALOG - CLASS SIZES



The **Batch Pipe Split** tool (Figure 36) allows the user to split multiple pipes in a series. For example, if a user is working in an area with catch basins and needs to add additional catch basins to reduce spread, this tool can be used to add the appropriate number of structures along the gutter at one time. This tool can be accessed here: **Drainage and Utilities >> Tools >> General Tools**. The user can then analyze various terrain features to build a drainage model.

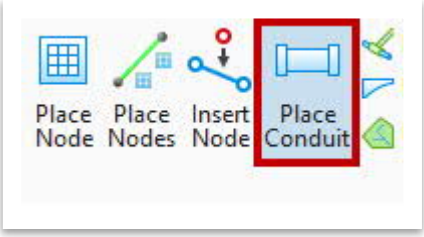
FIGURE 36. BATCH PIPE SPLIT TOOL



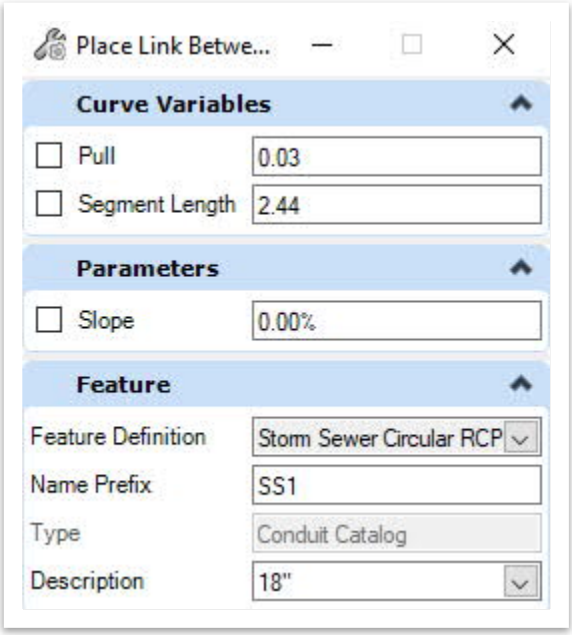
5.8.1 Exercise: Placing Conduits

In this exercise, we will connect the catch basins and an endwall with **conduits**. We will continue to utilize the same **Bowling_Drainage.dgn** file.

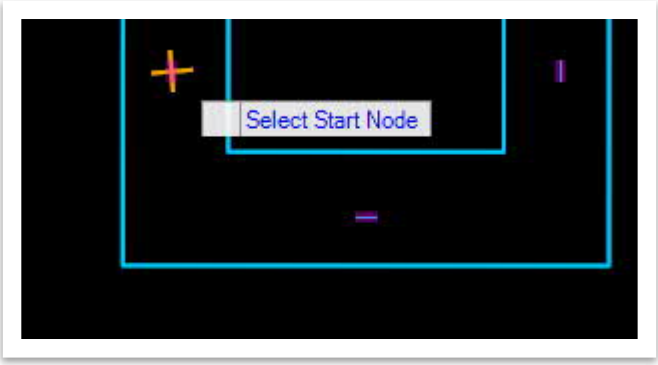
1. Turn off the **Bowling_ComplexTerrain.dgn** reference file, so the catch basin and endwall IDs can be seen more clearly. Open the **Place Conduit** tool (**Drainage and Utilities >> Layout >> Layout**).



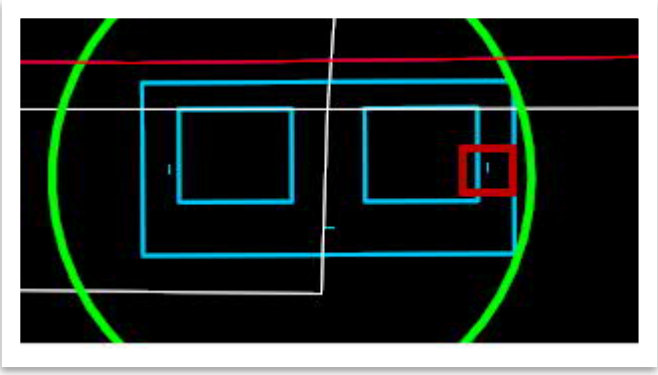
2. Within the **Place Link Between Nodes** dialog box, select the following settings.
    a. **Pull / Segment Length / Slope:** Unchecked (**Note:** These are only used for placing Curved Conduits. Ignore any values shown.
    b. **Feature Definition:** Storm Sewer Circular RCP (**Conduit >> StormWater >> Storm Sewer >> Proposed**)
    c. **Name Prefix:** SS1
    d. **Type:** Conduit Catalog (**Note:** This is greyed out because the conduit is selected from a pre-defined set of conduit catalogs.)
    e. **Description:** 18”



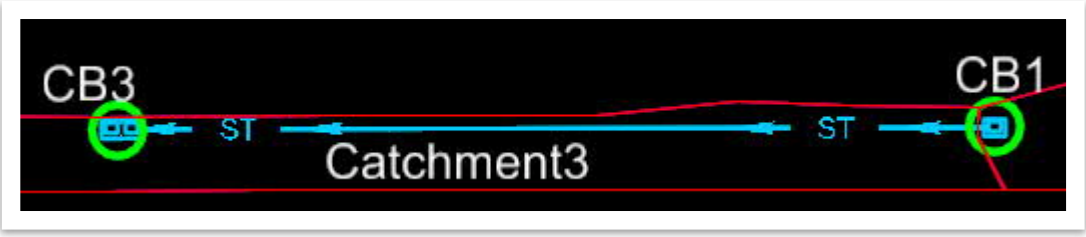
3. Zoom into **CB1** and snap to the left connection point. **This is the upstream node for the conduit**. **Conduits are laid upstream to downstream in the direction of flow**. **Note:** For this exercise, you can ignore any alert icons at the catch basins.



4. A white conduit line should now appear with the end attached to the cursor. Notice the cursor prompt: **Select next node to make a connection**. Move down to **CB3** and select the far right connection point, highlighted in red. Left click to accept placement of the conduit and then right click to clear the tool.

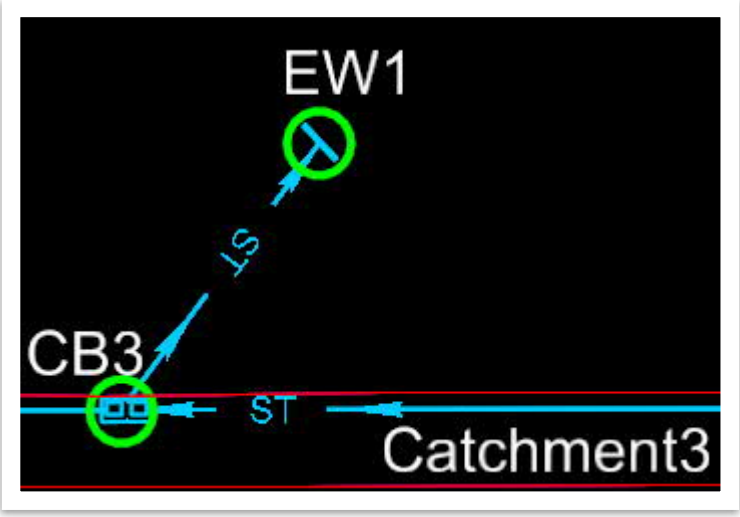


5. You should now see a proposed blue **ST line** with flow arrows. **Note:** The **Default** level has been turned off for screenshot clarity.



6. Next, do the same process to place conduit from **CB2** to **CB3**. This conduit will have the same features as **SS1**. Once again, there is no need to change the **Name Prefix** within the **Place Link Between Nodes** tool, even if it reverts to **SS1**. This conduit will be numbered **SS2** automatically. **Note:** The **Default** level has been turned off for screenshot clarity.

7. Lastly, do the same process to place conduit from **CB3** to **EW1**. This conduit will have the same features as **SS1** and **SS2**, and the naming should auto-increment to **SS3**. **Note:** You will not see separate connection points on the endwall. Also, the **Default** level has been turned off for screenshot clarity.

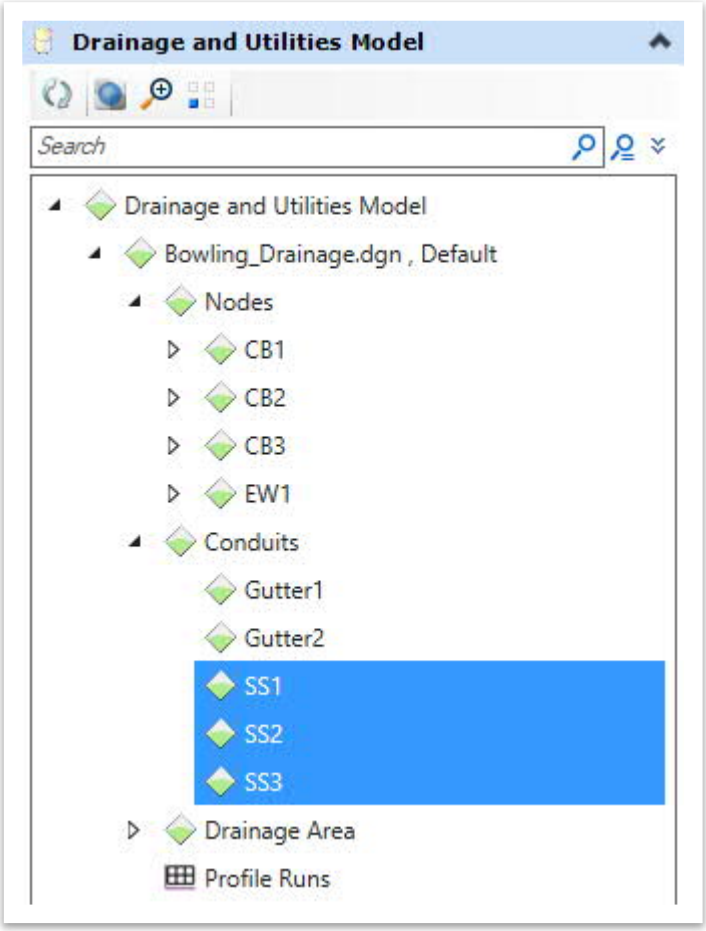


Take Note!

For skewed pipe connections, it is recommended to use a structure with a round bottom with the appropriate grate.

8. Catch basin **CB3** should now have **three** pipes connected to it. Turn off <u>all</u> reference files as well as the **Default** and **SUR - DRG - Area Shapes** levels in the active file to view the layout shown below.

9. Lastly, confirm that the **three** new conduits (**SS1 – SS3**) have been added to the **Drainage and Utilities** model within the **Explorer**. **Note:** As a reminder, if the conduits are not showing, click the **Refresh** icon under the **Drainage and Utilities Model** header. If the conduits still do not show, close and re-open the **Explorer**.

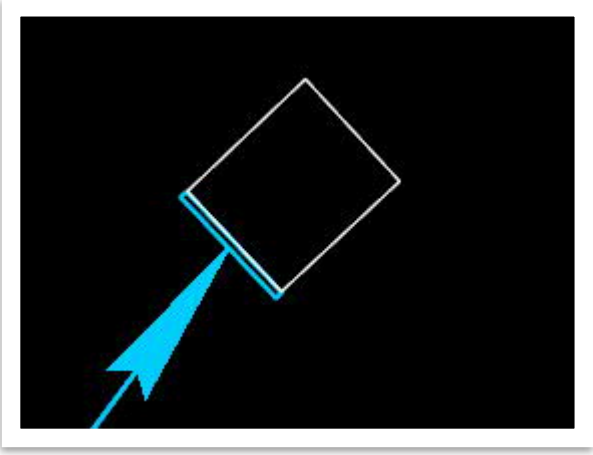


10. The storm network along **Bowling Avenue** has now been built.

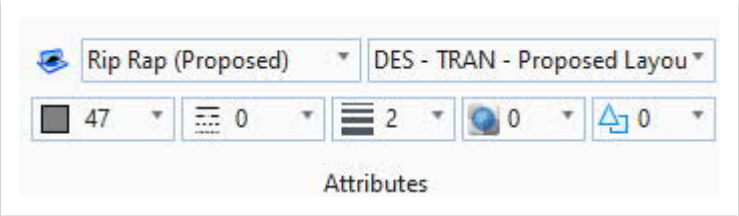
5.8.2 Exercise: Adding Riprap at Culvert Outlets

In this exercise, we will learn how to place riprap patterning at the outfall of a culvert. Riprap dimensions are calculated per Section 6.05.5 of the TDOT Drainage Manual. The dimensions used in this exercise are arbitrary. We will continue to utilize the same **Bowling_Drainage.dgn** file.

1. Attach the **Bowling_RipRap.dgn** file as a reference using the **Coincident World** attachment method. The other reference files should still be turned off. The proposed **apron** at outfall **EW1** has been pre-defined and measures **10 ft x 8.5 ft** (Length x Width).

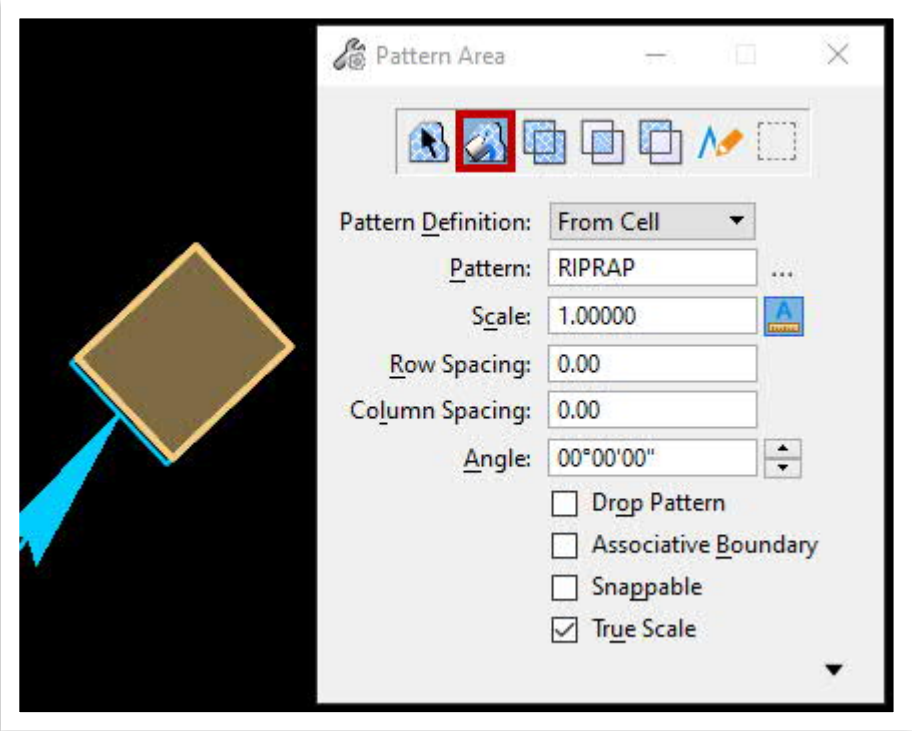


2. Now, select the **Rip Rap (Proposed)** element template (**Design >> Area Patterns >> Roadway**) so that the riprap will be placed with the correct symbology.

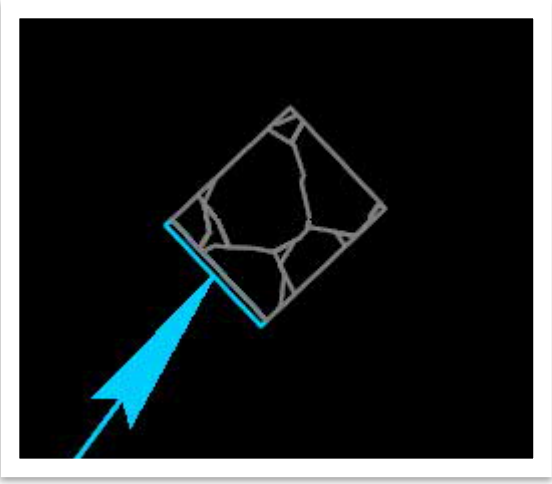


3. Next, open the **Pattern Area** tool. The quickest method is to use the search ribbon in the upper right corner of the ORD interface. **Note:** You could also find the tool in the **Drawing** workflow under **Annotate >> Patterns**.

4. Within the **Pattern Area** dialog box, notice that the **RIPRAP** pattern is automatically linked. Select the **Flood** option (highlighted in red) and then left click once anywhere within the white apron square. The flood area should highlight, as shown below. **Note:** All patterns should be placed at a scale of **1** and will automatically size to the active drawing scale. (i.e., 1"=50').



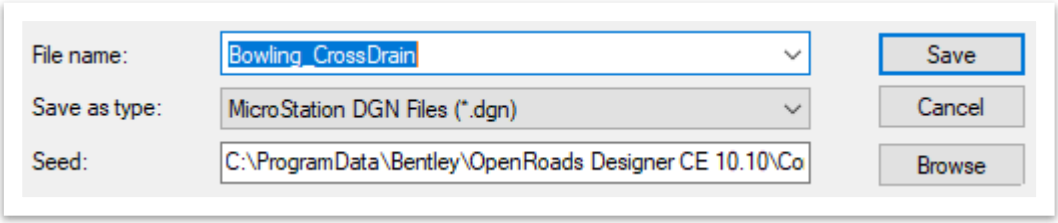
5. Left click again and notice the placement alert. Click **OK**. The riprap is now placed at the correct scale with the applicable symbology. Go ahead and turn off the **Bowling_RipRap.dgn** reference file..



5.8.3 Exercise: Cross Drain Conduit

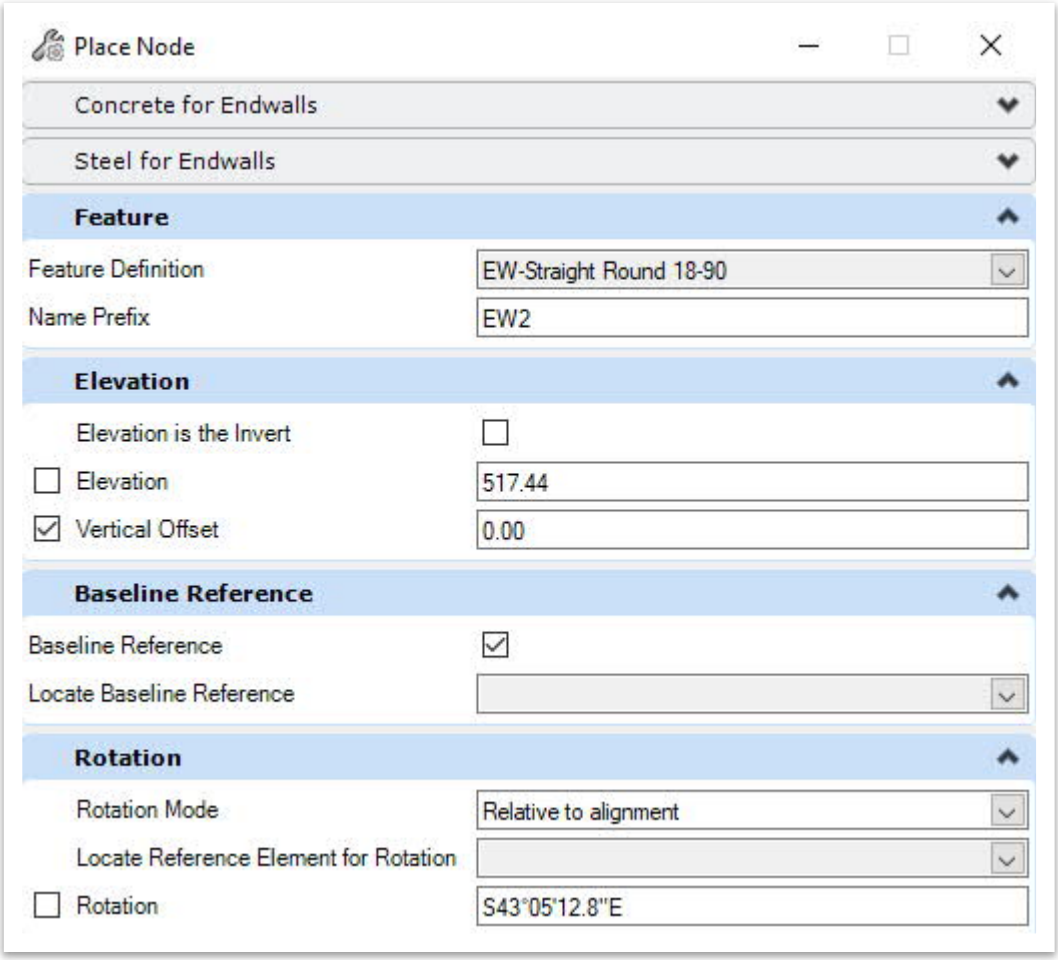
In this exercise, we will place a cross drain conduit between **EW2** and **EW3**.

1. Create a new file and name it **Bowling_CrossDrain**. Select the **TDOTSeed2D .dgn** and click **Save**.

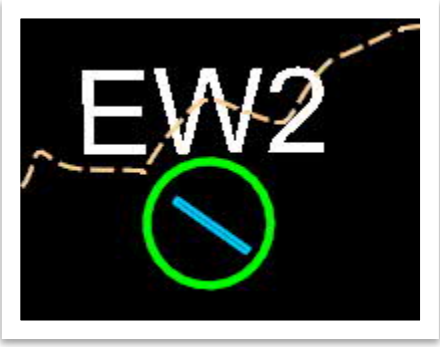


2. Attach the following reference files using the **Coincident World** attachment method and then click **Fit View**. Rotate the view similar to what was done in Exercise 4.4.1 Step 1.
    - Bowling_Catchment samples.dgn
    - Bowling_Geometry.dgn
    - Bowling_ComplexTerrain.dgn (turn on major contours)

3. Open the **Place Node** tool. Remember, since this is new file, you will need to click **Yes** to embed files for a Drainage and Utilities project in the design file and click **No** for the Compact Database alert. Open the **Place Node** tool again. Within the **Place Node** dialog box, select the **EW-Straight Round 18-90** feature definition (**Node >> StormWater Node >> Endwalls >> Cross Drains >> Straight >> Round >> 18”-30”; Skew 90**). Go ahead and update the **Name Prefix** to **EW2** since we already have an EW1 in the project. Match the remaining settings as shown below. **Note:** You can ignore the elevation and rotation values.

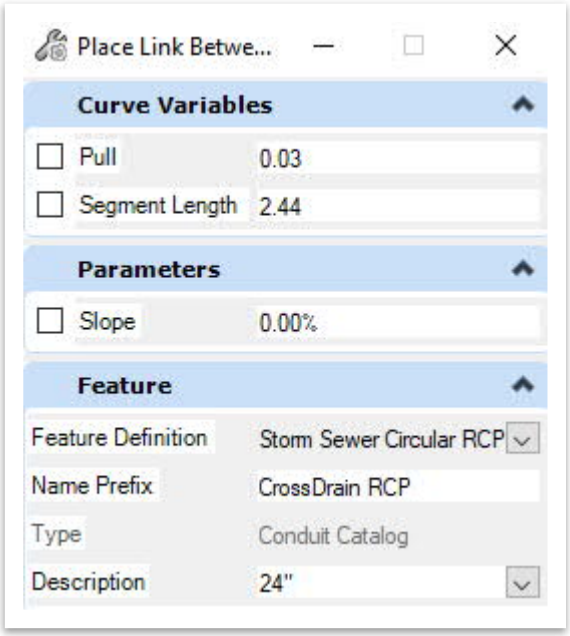


4. Notice the cursor prompt: **Select Reference Element For Node Elevation**. Select any contour of the referenced complex terrain file and left click to accept. Notice the next cursor prompt: **Define Headwall**. The **Vertical Offset** should already be set to **0.00** so left click within the center of the green circle to accept (don't snap to the center). Notice the next cursor prompt: **Locate Baseline Reference**. Select the **Bowling** centerline. Left click to accept the **Rotation Mode** and then select the **Bowling** centerline again and place the endwall with a similar angle as shown below. Turn off the **Bowling_ CrossDrain.dgn Default-3D** reference file.

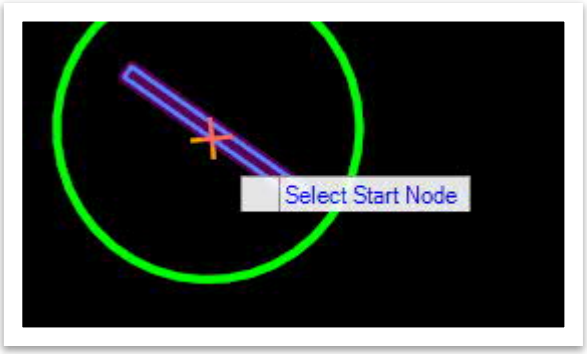


5. With the **Place Node** tool still activated, repeat the previous step to place a second endwall at **EW3** using the same settings. Remember to update the **Name Prefix**.

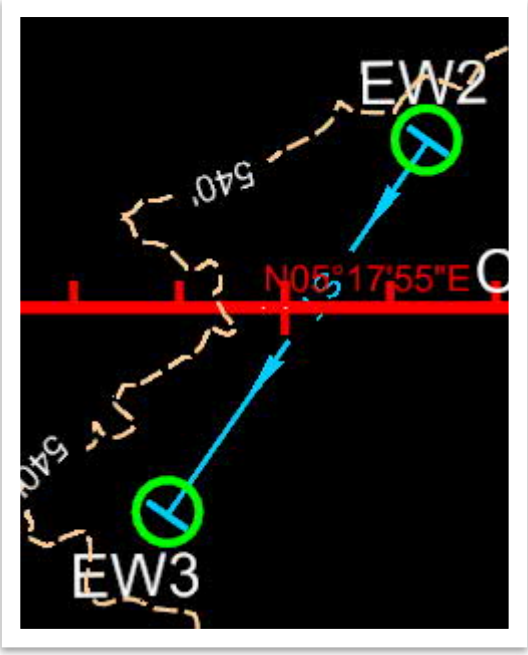
6. Next, open the **Place Conduit** tool. Within the **Place Link Between Nodes** dialog box, select the following settings.
    a. **Pull / Segment Length / Slope:** Unchecked (**Note:** These are only used for placing Curved Conduits. Ignore any values shown.
    b. **Feature Definition:** Storm Sewer Circular RCP (**Conduit >> StormWater >> Storm Sewer >> Proposed**)
    c. **Name Prefix:** CrossDrain RCP
    d. **Type:** Conduit Catalog (**Note:** This is greyed out because the conduit is selected from a pre-defined set of conduit catalogs.)
    e. **Description:** 24”



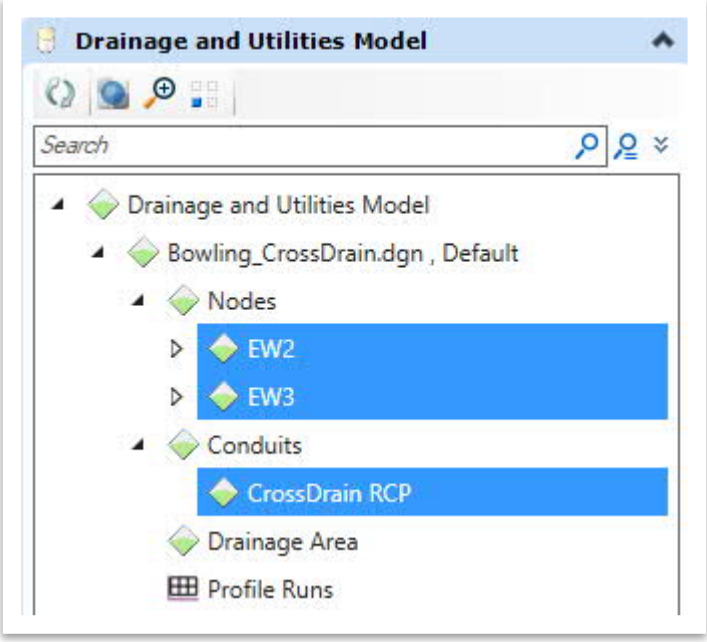
7. Zoom into **EW2** and snap to the connection point.



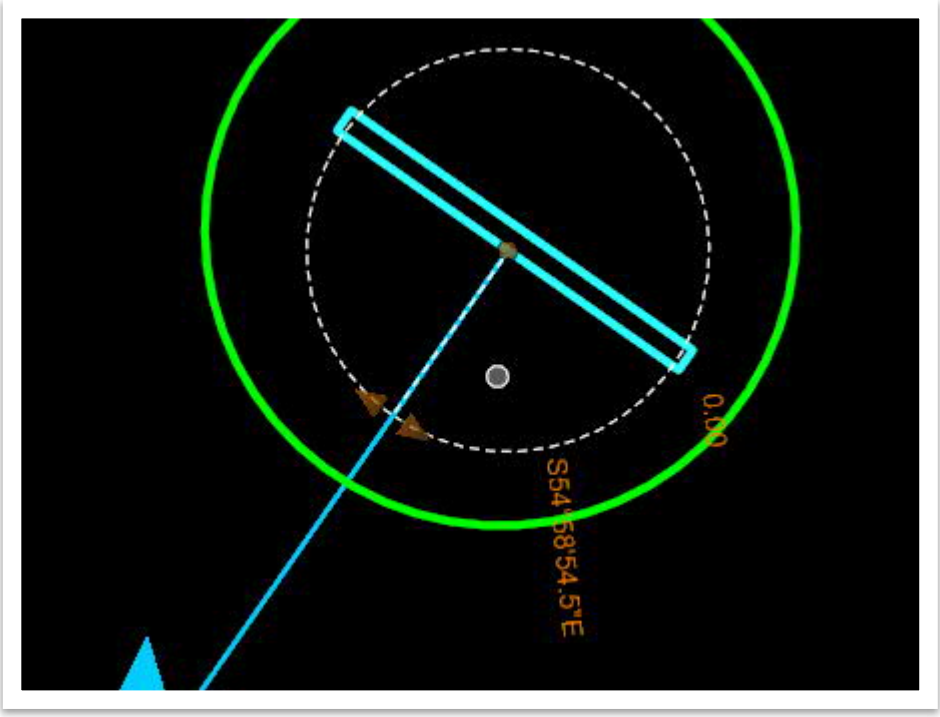
8. Notice the cursor prompt: **Select next node to make a connection**. Move downstream to **EW3** and snap to the connection point and then hit **ESC** to clear the tool. You should now see a proposed blue **ST line** with flow arrows.



9. Confirm that the **two** new endwalls (**EW2** and **EW3**) and the new conduit (**CrossDrain RCP**) have been added to the **Drainage and Utilities** model within the **Explorer**. **Note:** As a reminder, if the conduit does not show, click the **Refresh** icon under the **Drainage and Utilities Model** header. If the conduit still does not show, close and re-open the **Explorer**.



10. To verify if the endwalls are aligned with the cross drain, select **EW2** so that the orientation circle and origin point appear. If the orientation radii on the circle does not align with the **direction of the outgoing cross drain**, you can hold the **two manipulators** and rotate the structure to align with the cross drain. You would repeat this step for EW3 if any adjustments were necessary.



5.9 Lecture: Generating Drainage Profiles

A **profile** is a graph that plots an attribute across a distance along a section of piping, such as ground elevation. Profiles also show other characteristics, such as **Hydraulic Grade**, **Velocity**, and **Flow Depth**.

Profiles can be defined by selecting a series of adjacent elements. Only **conduits**, **channels**, **ditches**, and **gutters** can be part of a profile. The profile created will display the structures selected, as well as relative ground elevation and water elevations within the pipe or ditch. The **Profile Runs** tools allows the user to create, view, and edit profile views of elements in the network.

There are **two** locations where the user can select the **profile building tools**:

- **Within the Ribbon (Drainage and Utilities >> Layout >> Profile Runs)** (Figure 37). There are four different methods to create a profile:

FIGURE 37. PROFILE RUN TOOLS

Hydraulic Run from Node: This tool builds a profile by connecting nodes.

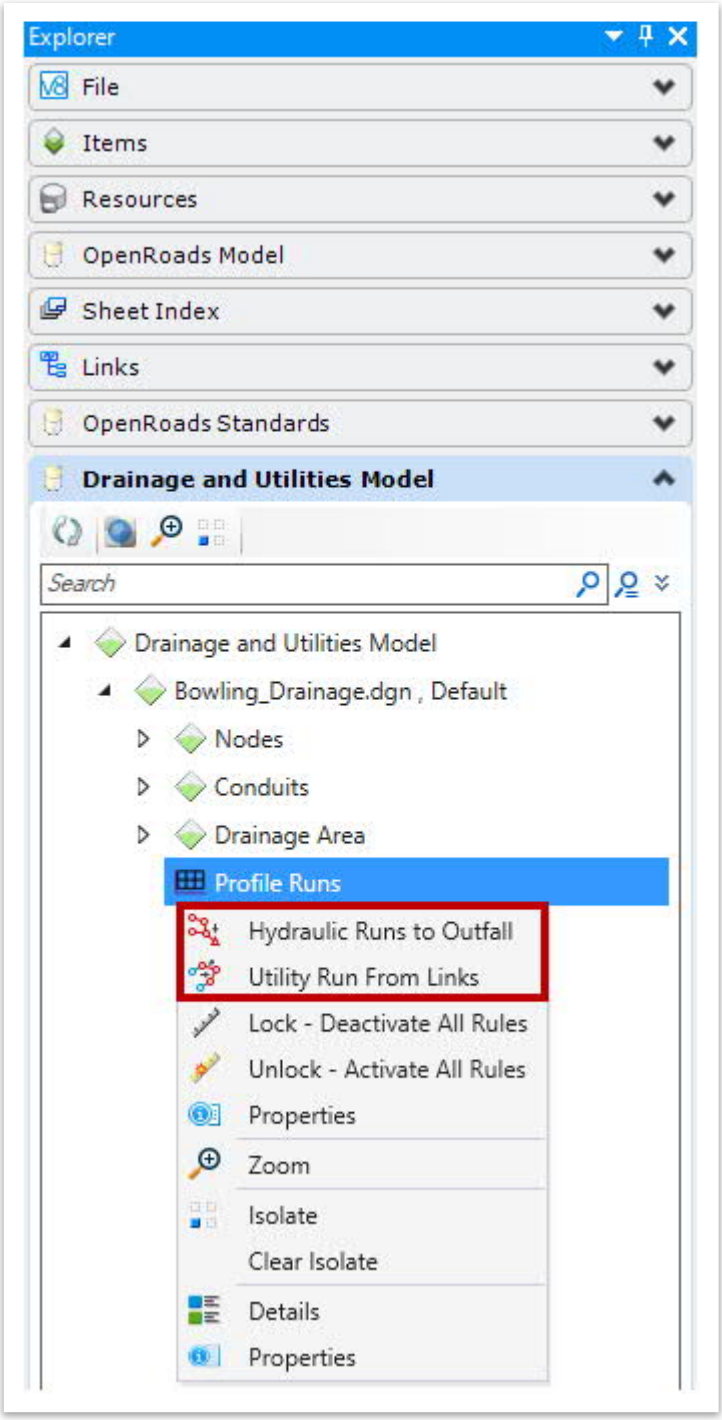
Hydraulic Runs to Outfall: This tool builds a profile by selecting the outfall node. Profiles for lateral connections to the trunk line are also created.

Utility Run From Links: This tool creates a profile by selecting applicable conduits.

Project Run: This tool helps project a storm drain network with respect to the roadway geometry/alignment.

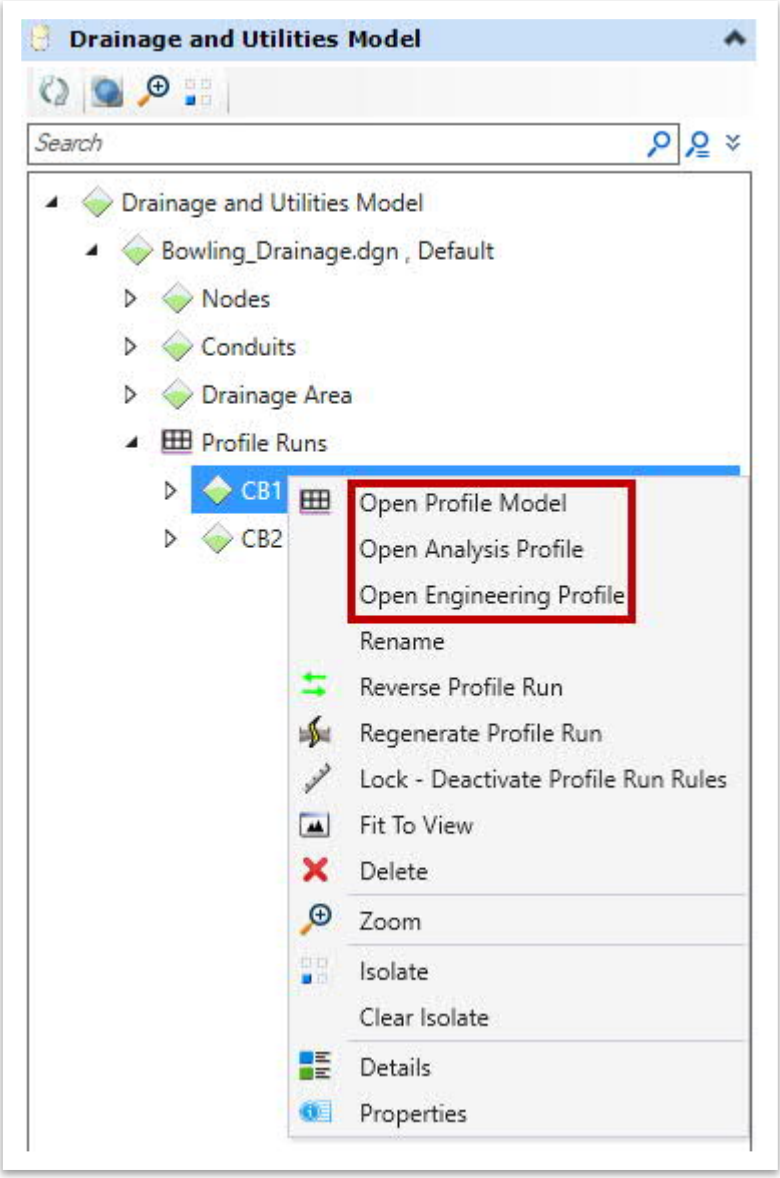
- **Within the Explorer (Drainage and Utilities Model >> Profile Runs)** (Figure 38). Right click on Profile Runs.

FIGURE 38. EXPLORER - PROFILE RUNS TOOLS



The **Utility** model will populate with profiles once they are created. The profile can be viewed in **three** different ways, as highlighted below (Figure 39). To access the different profile options, right click on the applicable run under **Profile Runs**.

FIGURE 39. EXPLORER - VIEW PROFILE RUNS



Open Profile Model:

Generates a View that presents the nodes and conduits of a utility model in the profile view. The profile for a utility conduit can be viewed with respect to the project's vertical geometry. This view helps to review ground cover issues, judge if pipe slopes are in-line with roadway slopes and identify any utility conflict issues. This view also represents the profile model that is referenced on to profile sheets.

Open Analysis Profile:

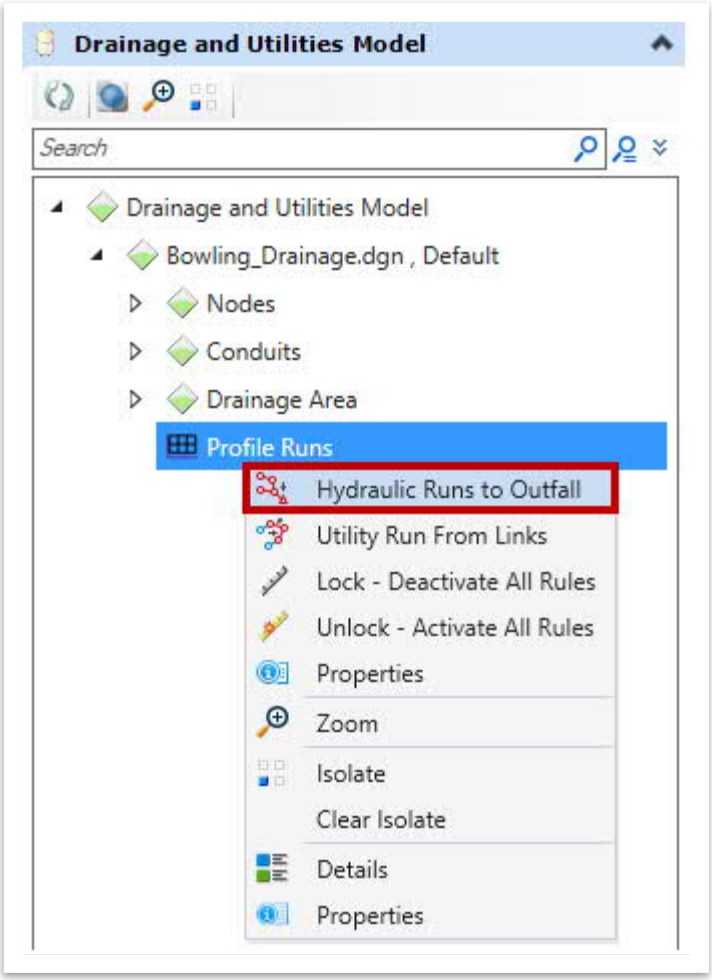
Allows for review of the profile showing the EGL (Energy Grade Line) and HGL (Hydraulic Grade Line). Several annotation edits can be made, and the profile can be exported to a dgn fill.

Open Engineering Profile: Allows profile generation at a desired engineering scale that is different from the profile sheets. Various display options can be changed such as editing both the structure annotations and the axis labels. A profile report can also be generated.

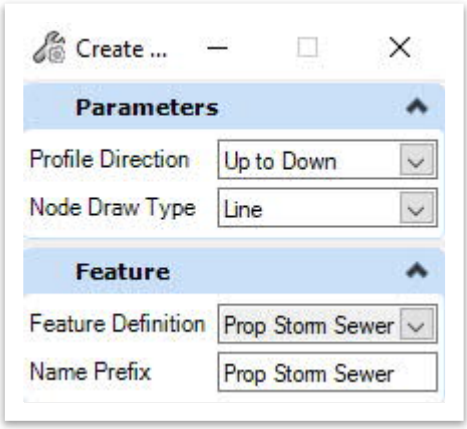
5.9.1 Exercise: Create, Review, and Edit Profile Runs

In this exercise, we will create profile runs between the catch basins and the downstream endwall, and then examine the **Profile Model** and **Engineering Profile** of a selected run. In addition, custom annotation will be added to the Engineering Profile. We will open back up the **Bowling_Drainage.dgn** file.

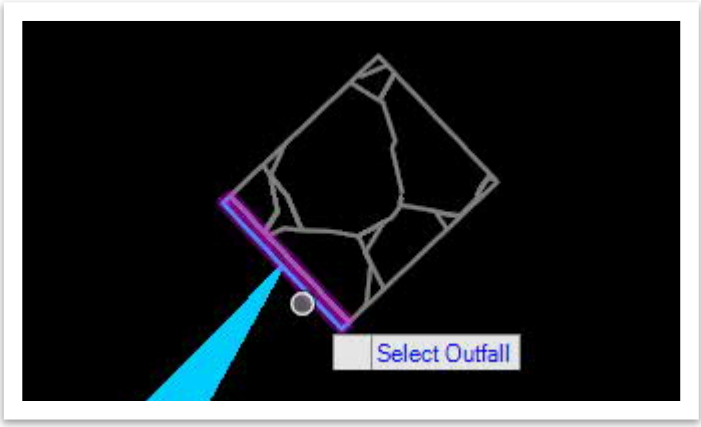
1. Make sure the **Default** view is active and the **Default** and **SUR - DRG - Area Shapes** levels are turned off. Right click on **Profile Runs** within the **Explorer** and select **Hydraulic Runs to Outfall**.



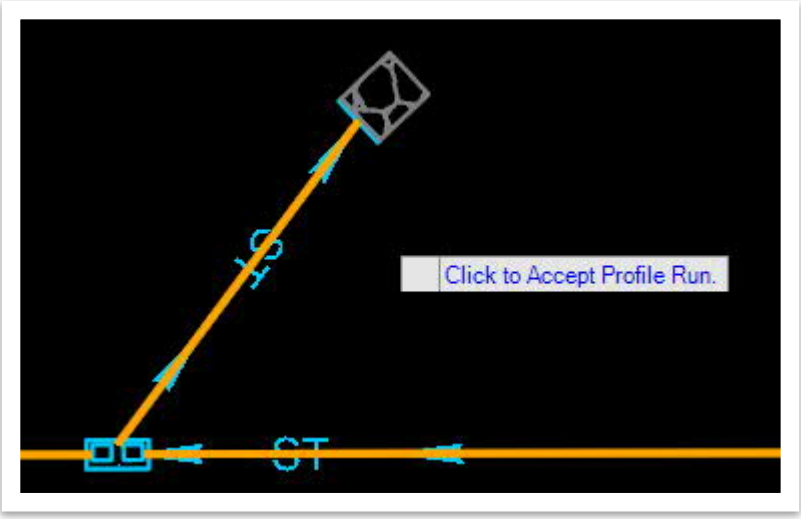
2. Within the **Create Reaches To Outfall** dialog box, select the following settings.
    a. **Profile Direction:** Up to Down
    b. **Node Draw Type:** Line
    c. **Feature Definition:** Prop Storm Sewer (**Linear >> Profiles >> Drainage >> Proposed**)
    d. **Name Prefix:** Prop Storm Sewer



3. Notice the cursor prompt: **Select Outfall**. Select **EW1**. **Note:** Turn on the **Bowling_Catchment samples.dgn** reference file if you forget the structure names.



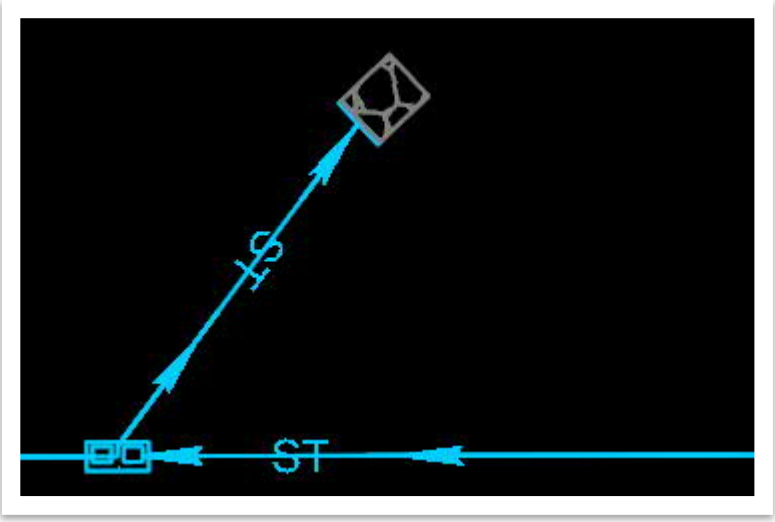
4. The tool traces profiles up through the network. The traced profiles are highlighted in orange. Left click to **Accept Profile Run**.



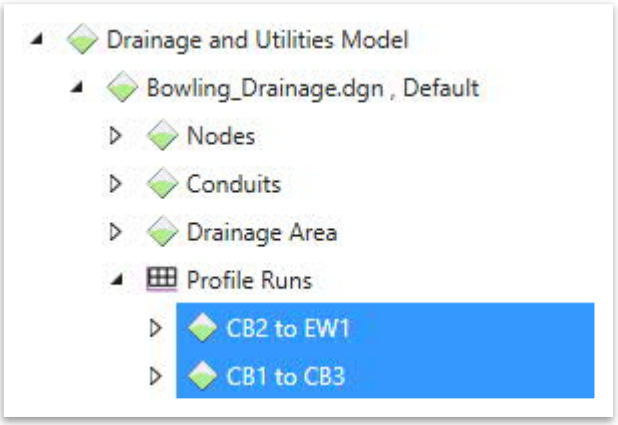
Take Note!

*The software takes a minute to automatically create **Profile Runs**. **Do not click anything immediately after selecting an outfall**. Wait until the orange lines are replaced with the blue profile lines.*

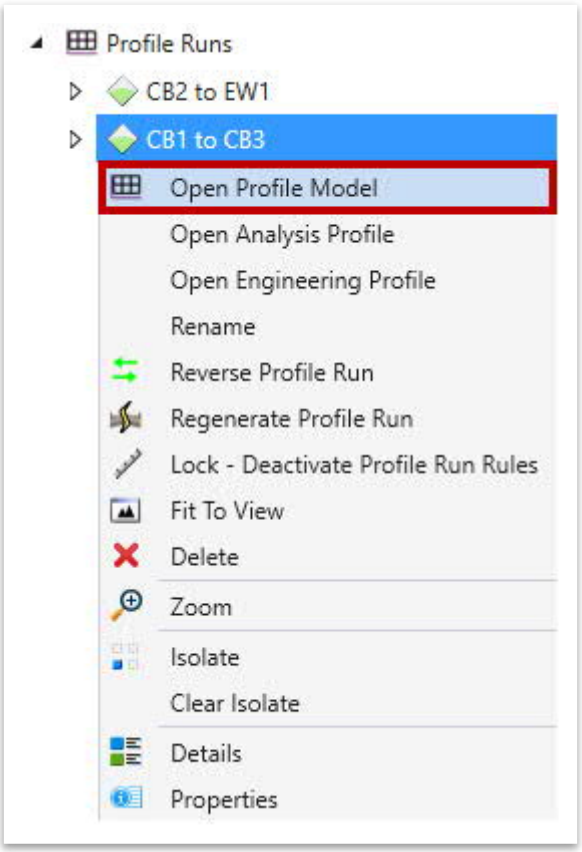
5. The profile runs are now added on the **DES - PRO - Drainage - Storm Sewer** level. Go ahead and toggle it off and on to see the added profile runs. Since **CB3** is acting as a **junction** between conduit pipes **SS1** and **SS2**, one of those pipes will be treated as a **lateral**. **Note:** When a profile is created using **Hydraulic Runs to Outfall**, the software defines the mainline and the laterals. However, if the **Utility Run from Links** option was selected, it would allow the user to select the mainline and lateral lines, based on the downstream conduits selected.



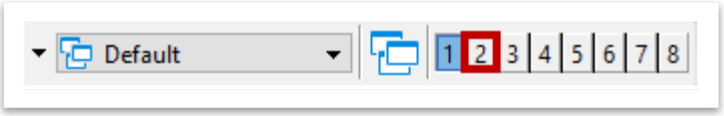
6. Now that the profile runs have been created, they will appear in the drainage model (**Explorer >> Drainage and Utilities Model >> Bowling_Drainage.dgn >> Profile Runs**). It appears that **SS2** and **SS3** have been chose as the main branch / reach whereas **SS1** has been designated as the lateral for the Profile Run.



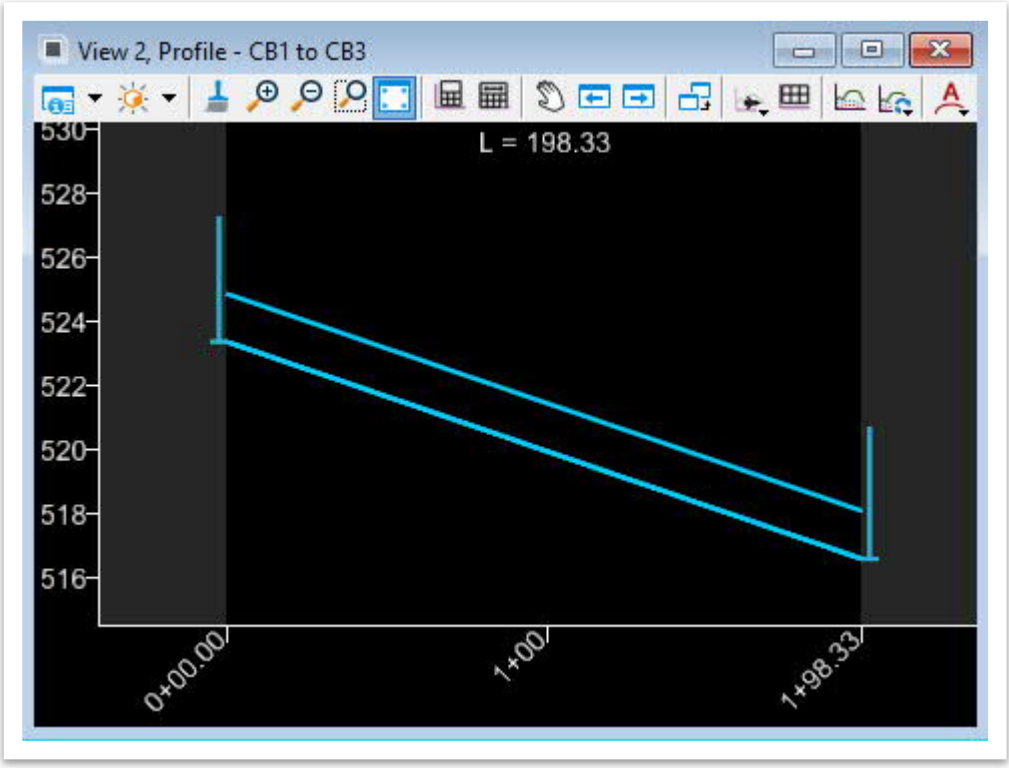
7. Next, right click on Profile Run **CB1 to CB3** and select **Open Profile Model**.



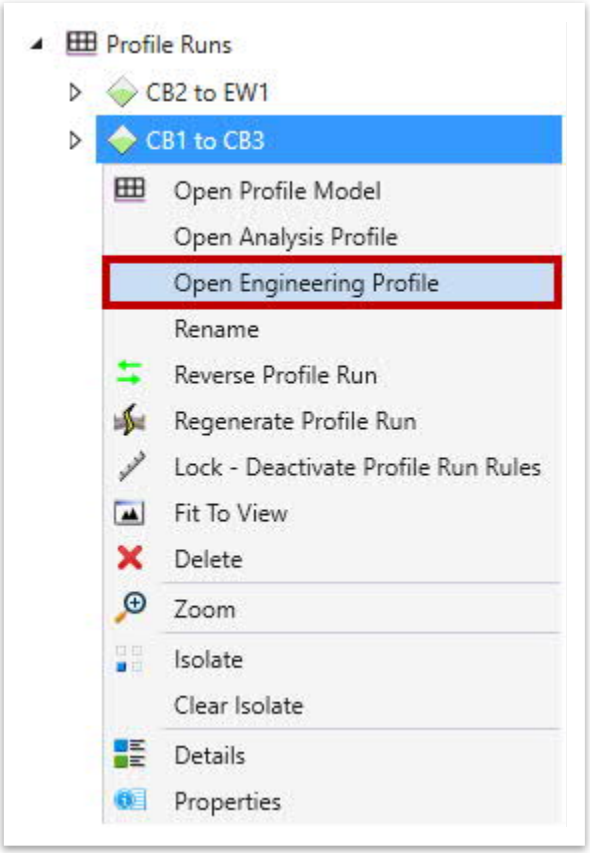
8. Notice the cursor prompt: **Select or Open View**. Select **View 2** at the bottom of the drawing window. **Note:** Ultimately, you could select any view.



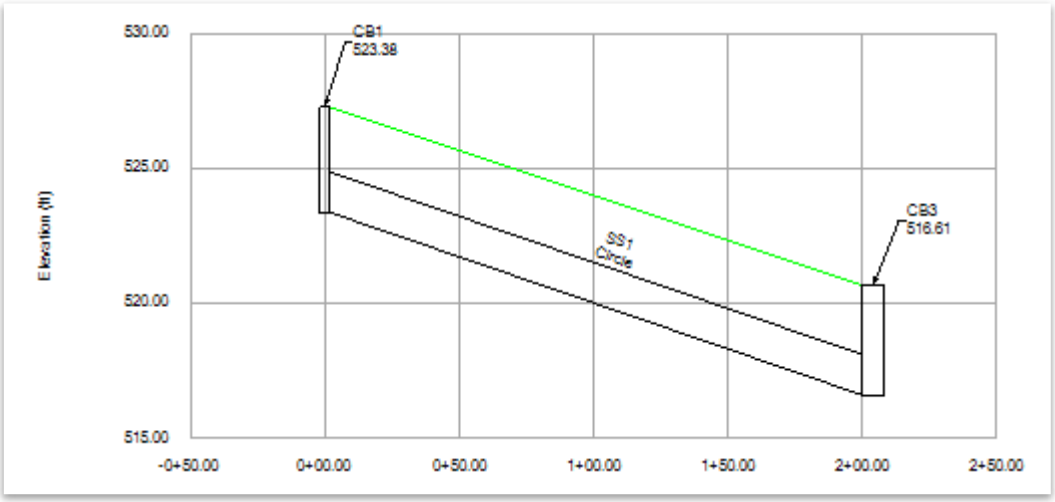
9. Once **View 2** opens, left click anywhere within that view and notice that a **Profile** of the selected **Profile Run** is displayed. This profile is shown along the **pipe length** and not along the project's alignment. Once reviewed, close **View 2**. **Note:** You may review the profile model of the other profile run (**CB2 to EW1**) in the same manner.



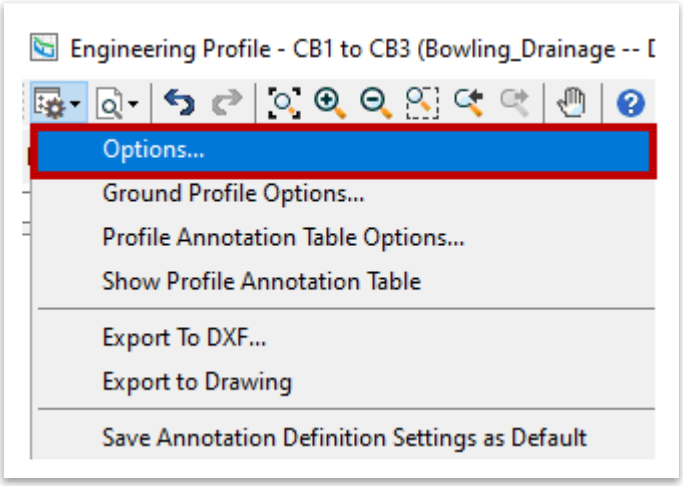
10. Now, right click on profile run **CB1 to CB3** and select **Open Engineering Profile**. This profile is shown along the **pipe length** and not along the project's alignment.

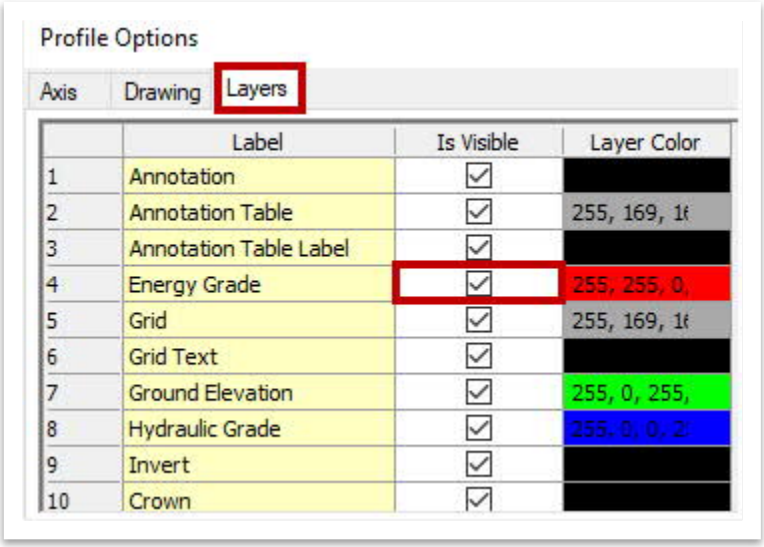


11. In the **Engineering Profile** window, review the **Nodes**, **Conduit**, **Ground Elevations**, **Inverts**, **etc**. to verify if the inlets and pipe are connected correctly in the direction of flow, along the storm drain. **Note:** You may review the engineering profile of the other profile run (**CB2 to EW1**) in the same manner.



12. Lastly, within the **Engineering Profile** window, go to **Options >> Layers** tab. Toggle on the **Energy Grade** option so that it is visible in the profile with the selected **Layer Color**. Once reviewed, close the **Engineering Profile** window. **Note:** Change these options based on preference and refer to the TDOT standards to detail the profile as necessary. Also, if the **EGL** does not appear, make sure the **Show Energy Grade Line** setting is set to **True** under **File >> Settings >> User >> Preferences >> View Options – Civil**.





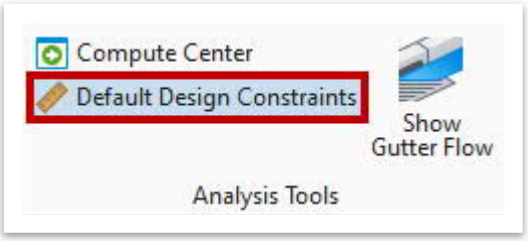
Take Note!

*Showing the Energy Grade Line (**EGL**) will allow you to understand the changes in head of the water flowing through a pipe.*

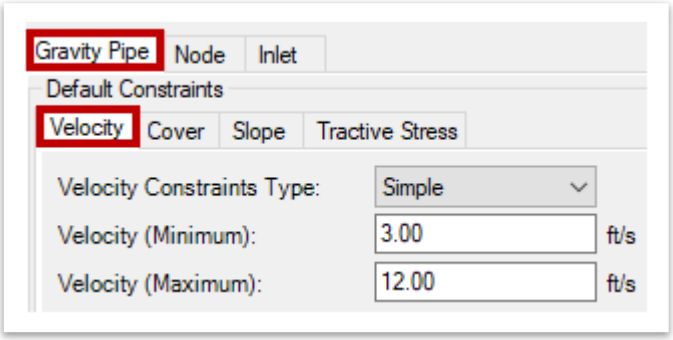
5.10 Exercise: Verify Project's Design Constraints

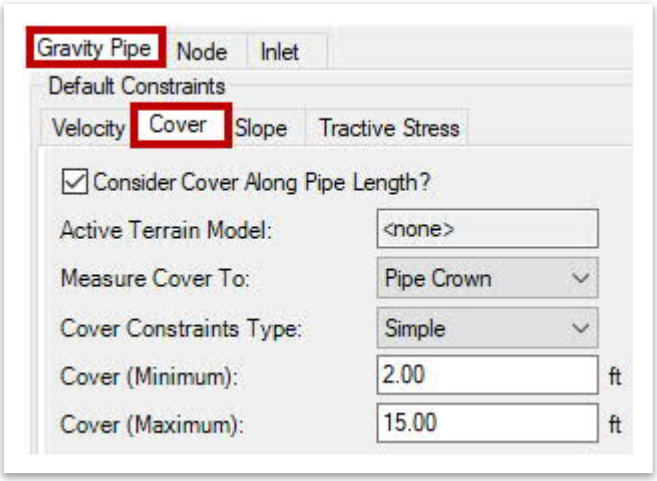
In this exercise, we will optimize the drainage design analysis by revising the default design constraints. All networks will be run at one time, unlike in GEOPAK where individual networks were run. We will continue to utilize the same **Bowling_Drainage .dgn** file.

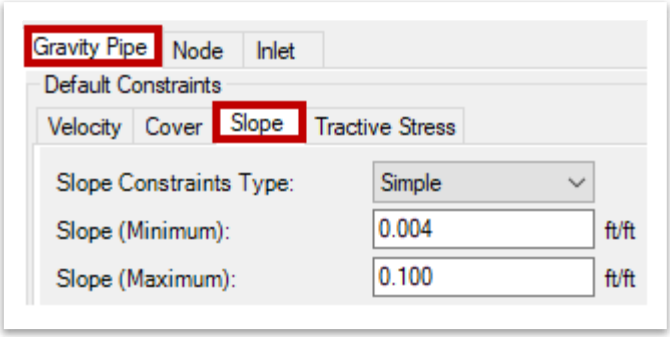
1. Open the **Default Design Constraints** tool (**Drainage and Utilities >> Analysis >> Analysis Tools**).

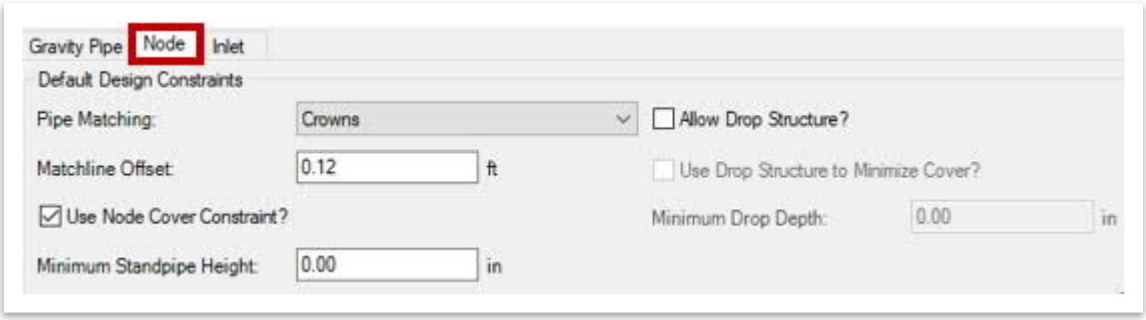


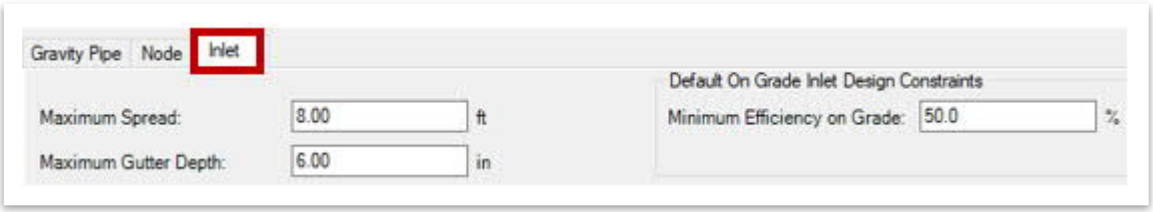
2. It is important to review the following tabs prior to computing the model and ensure that the correct values are applied. Go ahead and confirm the following values in the **Default Design Constraints** window. Update any fields, as necessary, so they match what is shown below. Then close the window.









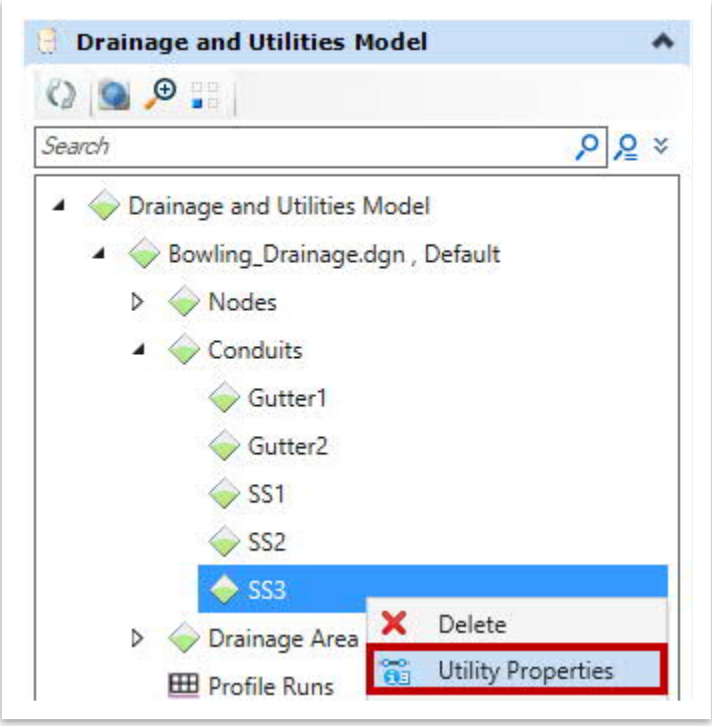


Take Note!

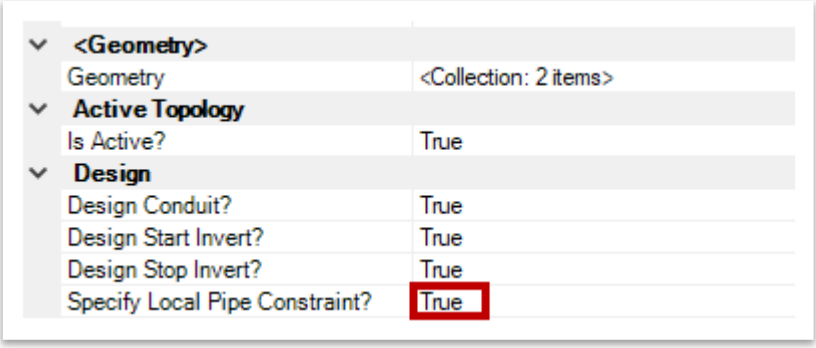
*The **Cover (Minimum)** value must be revised according to project specific requirements. Per Chapter 7 in the TDOT Drainage Manual, this value for RCP's is 12.00 inches measured from the bottom of the subgrade to the outside surface of the pipe. Other material types are also covered in the same chapter. For example, if the pavement depth accounts for a depth of 15.00 inches, then the pipe's minimum cover would be 12.00 + 15.00 = 27.00 inches = **2.25 ft**.*

3. For this project, the pavement depth shown in the typical section is 16.20 inches, so the **Cover (Minimum)** would be **2.35** ft. These minimum/maximum cover values would be applied to design or analyze all conduits across the entire project. **Note:** The value that first loads with the creation of the dgn file could be different than what is shown above, since that value is loaded from the pre-defined standards in the **TDOT Drainage Features Defs.dgnlib**. Hence, this value must be revised in the project's design file. If there are any pipes that need more cover, they can be edited individually, which is shown in the remaining steps.

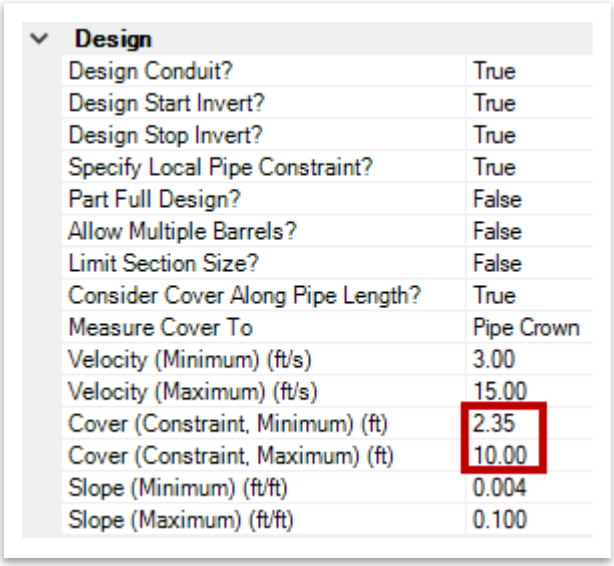
4. Next, we will examine conduit **SS3** to review the process of how to apply a **pipe specific cover constraint**. Open the **Utility Properties**, as shown below. ORD has the capability to input a minimum/maximum **cover** range for a given conduit. In this project, **SS3** is connected to a **Type 14** inlet. Per TDOT's Drainage Manual requirements, an **18"** pipe connected to a **Type 14** inlet requires a cover of **2.35'**.



5. Within the **Drainage** tab, we first need to change the **Specify Local Pipe Constraint?** field to **True** under **Design**. You will notice additional fields appear once this field is changed.



6. The minimum and maximum cover constraints are specific to **SS3**. The remaining conduits will be designed based on the cover constraints seen in Step 2. Go ahead and set the **Cover (Constraint, Minimum) (ft)** to **2.35** and the **Cover (Constraint, Maximum) (ft)** to **10.00**, as required by TDOT. Then, close the **Utility Properties**. **Note:** This process would only be applied to conduits that are an exception where the default constraints are not practical and should be documented in the project calculations.



5.11 Lecture: Drainage Output

The final steps in building a drainage model involve producing a drainage output in ORD. In Geopak, the user would need to run the network and build a report to see the output of the drainage network. When naming the output files, they should be named based on the outfall of the run (network) with a unique suffix identifier (e.g., Outfall_**1**). It is recommended to keep all runs in case the user needs to go backwards when optimizing the drainage networks.

5.11.1 Alternatives, Scenarios and Flex Tables

Scenarios and Alternatives let the user create, analyze, and recall an unlimited number of variations of a drainage and utility model. Alternatives are part of Scenarios and when applied together, they let the user review, document, and compare results from several variations possible to the model. Flex Tables provide all inputs and outputs for drainage elements.

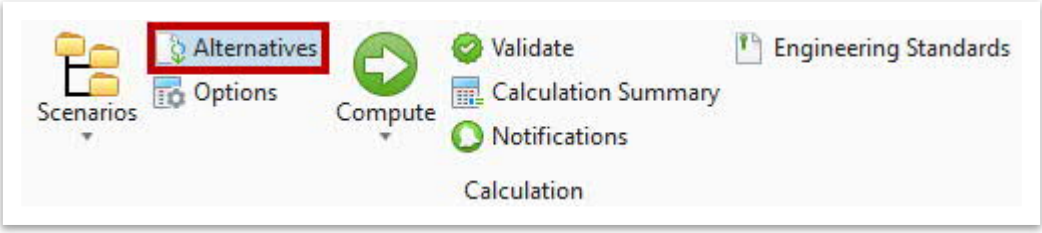
5.11.1.1 Alternatives

An Alternative is a logically organized set of properties, which include Physical Alternatives, Design **Alternatives** and **Rainfall Runoff Alternatives**. All these pre-defined properties make up a Scenario, which is applied to design or to analyze a storm drain system.

- **Physical Alternatives:** Criteria that will apply to the physical data of the network's elements, such as elevations, sizes, and roughness coefficients.
- **Design Alternatives:** Groups engineering criteria that will be applied during calculations such as velocity limits, spread criteria and pipe slope etc.
- **Rainfall Runoff Alternatives:** Allows different storm events (2, 10, 25, 50 and 100 year) to be used in calculations. Each storm event is programmed for all **ten** (10) cities within the TDOT Drainage Feature Defs.dgnlib.

The **Alternatives Manager** lets the user create, view, and edit the alternatives that make up the project scenarios (Figure 40). The tool can be located here: **Drainage and Utilities >> Analysis >> Calculation**. We will talk more about Scenarios later in this section.

FIGURE 40. ALTERNATIVES TOOL

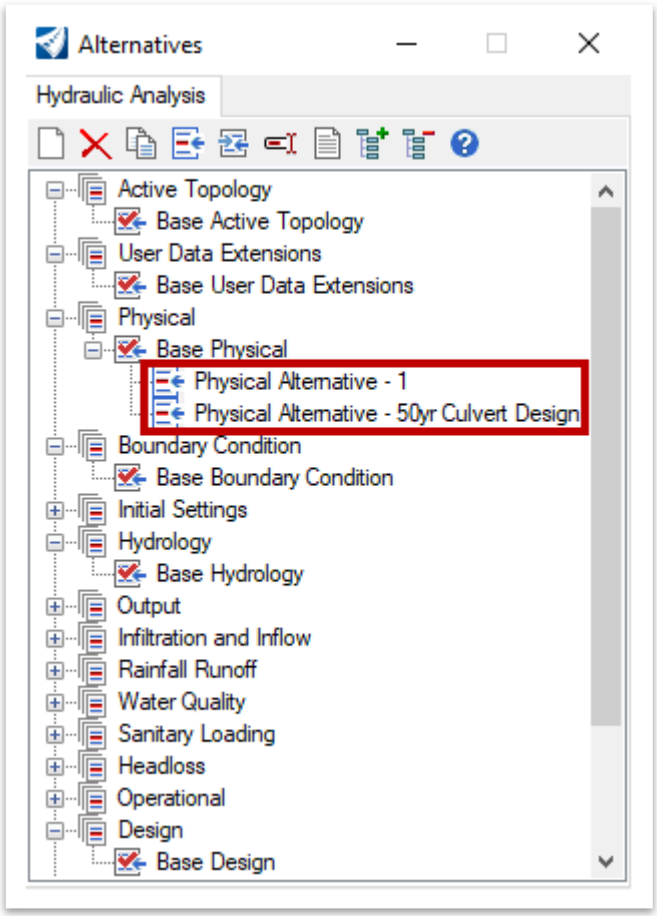


Within the **Alternatives Manager**, there are expandable folders for each alternative. There are two kinds of Alternatives: **Base Alternatives** and **Child Alternatives**.

- **Base Alternatives:** Contains data pertaining to the project for **all** elements in the model. Within the properties, there are **check marks** shown for **every element** in the first column.
- **Child Alternatives:** Inherits its data from Base Alternatives. This data can be altered by the user and is applicable to only **some of the elements** of the model. The check marks can be **customized per the elements** needed in this alternative. These alternatives can be created by right clicking on any Base alternative and selecting **New**.

Figure 41 shows an example of two **Child Alternatives** (highlighted in red) that have previously been created under the **Base Alternative** called **Base Physical**.

FIGURE 41. CHILD ALTERNATIVES

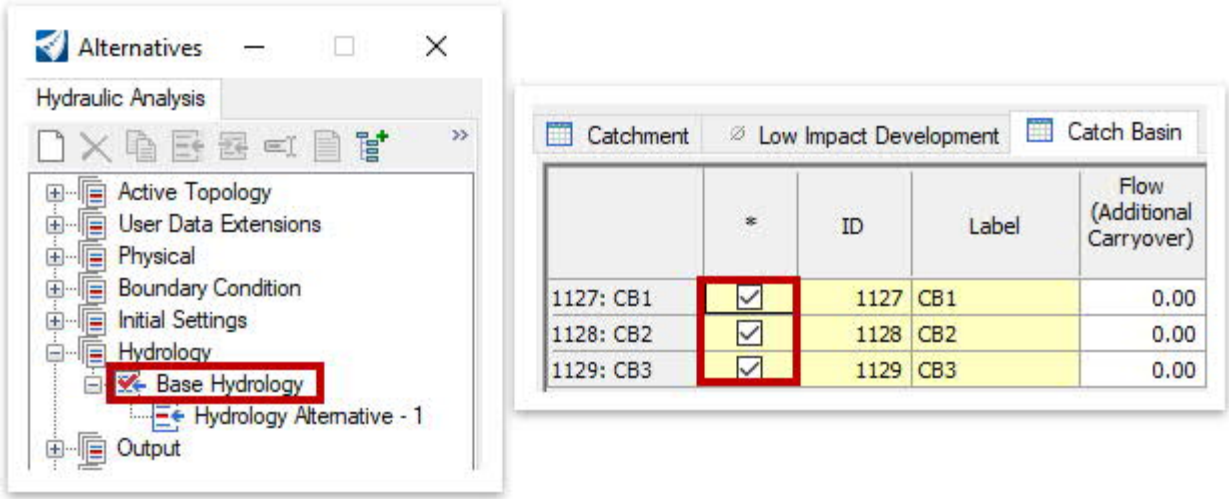


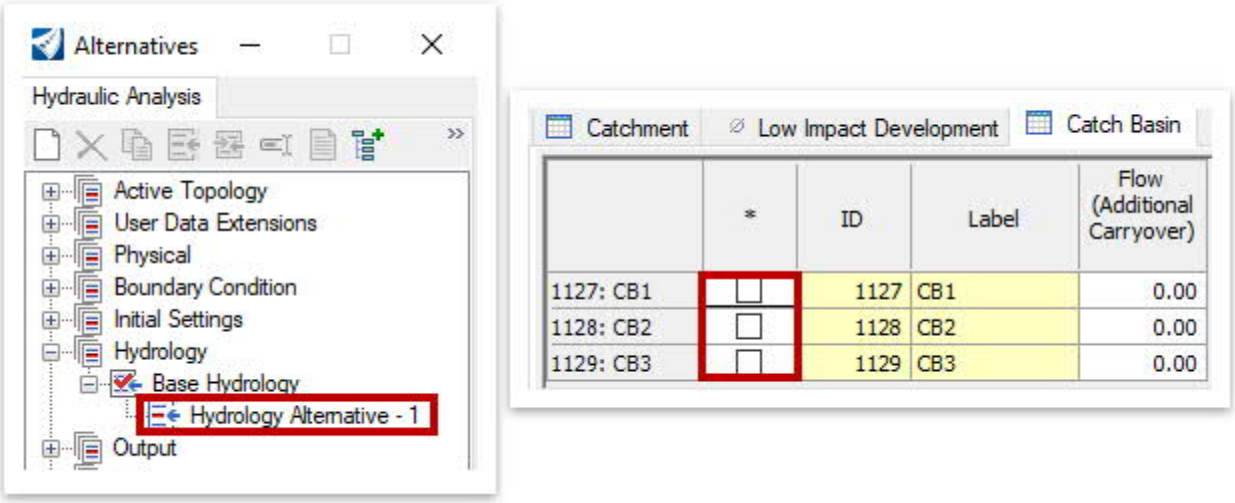
Editing Alternatives:

- Base and Child Alternatives can be edited.
- Expand the **+** sign so that all alternatives are shown.
- **Right click on the alternative** and select **Open** to view the properties.
- Within the Base and Child alternatives, regardless of the element, the **yellow** fields are **non-editable** whereas the **white** fields are **editable**, irrespective of the check boxes in the first column.
- In the Base alternative, the first column contains a series of check boxes, which means that the record on that line has been locked and the data is specific to that alternative.
- In the Child alternative, the first column does not contain check boxes, which means that the record on that line is inherited from its higher-level parent alternative. The records can be edited (e.g., add/remove elements as part of the child alternative).

For example, the **Base Hydrology** alternative (Figure 42) has check boxes for CB1, CB2 and CB3, which means these three elements are part of this alternative and cannot be removed. The child alternative (**Hydrology Alternative - 1**) shown on the next page does not have check boxes for CB1, CB2 and CB3, which means any of the elements can be omitted from applying the child scenario. In both alternatives, **Flow (Additional Carryover)** data can be inputted since the column is **white**, if a project has known flows that need to be considered.

FIGURE 42. EDIT ALTERNATIVES



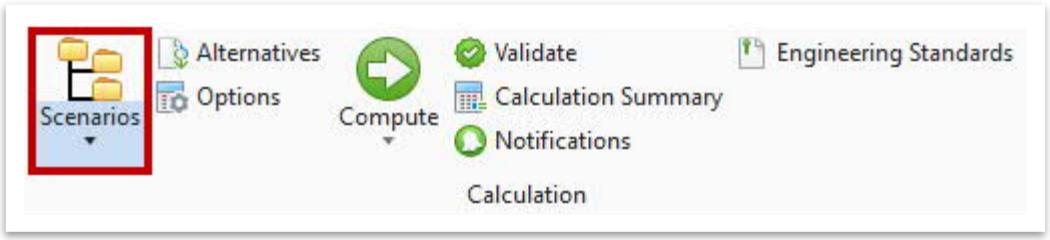


5.11.1.2 Scenarios

Scenarios are a list of **Alternatives** and **Calculation Options**, which can be viewed in the **Properties** window by right clicking on any scenario. **Scenarios** let the user set up an **unlimited number of What If?** situations for the model.

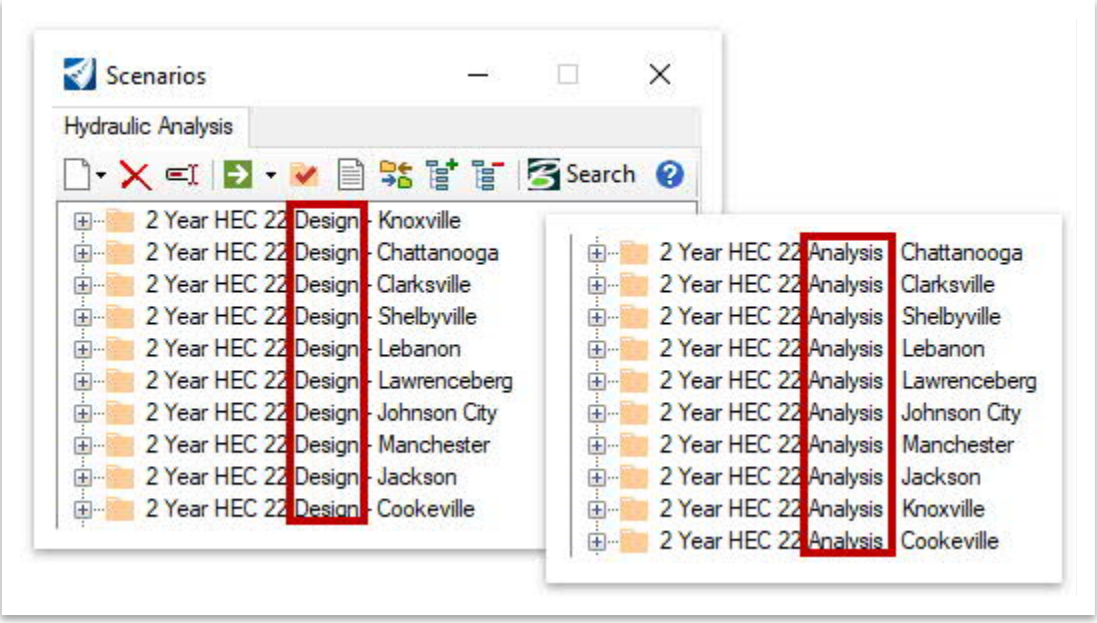
A **Scenario** contains all the input data (in the form of Alternatives), calculation options, templates to write results and notes associated with a set of calculations. The user can **create**, **edit,** and **manage** any scenario, using the **Scenario Manager** tool located here: **Drainage and Utilities >> Analysis >> Calculation >> Scenarios** (Figure 43).

FIGURE 43. SCENARIOS TOOLS



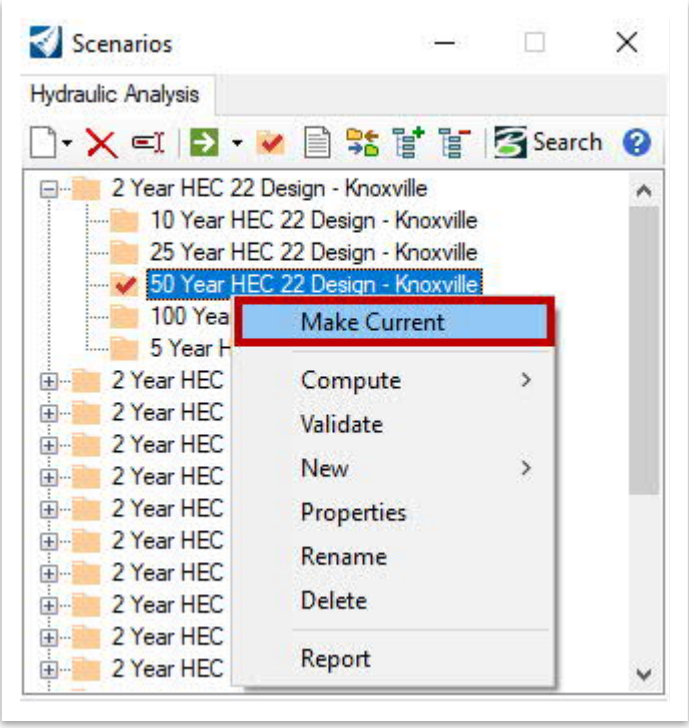
Every project is **designed** or **analyzed** using a base scenario. The **TDOT Drainage Feature Defs.dgnlib** lists all Scenarios that are needed to run any storm event. Within the Scenario Manager, **Design** and **Analysis Options** are available for all storm rainfall events (2, 5, 10, 25, 50 and 100 year) for the **10** cities (Figure 44). This means, the user can choose to **design** or **analyze** the system within any of the ten cities and for any storm event.

FIGURE 44. DESIGN AND ANALYSIS OPTIONS



These Scenarios are automatically loaded into any project file when a Drainage and Utility model is created. The user will set any of the Scenarios as the **Base Scenario**. For the example below, the **50 Year HEC 22 Design - Knoxville** is the base scenario since the project is designed for a **50-yr. storm** and is in **Knoxville**. In order to set this Scenario as the **Base Scenario**, the user would right click and select **Make Current** (Figure 45). You will then notice a red check mark to indicate that the scenario is active.

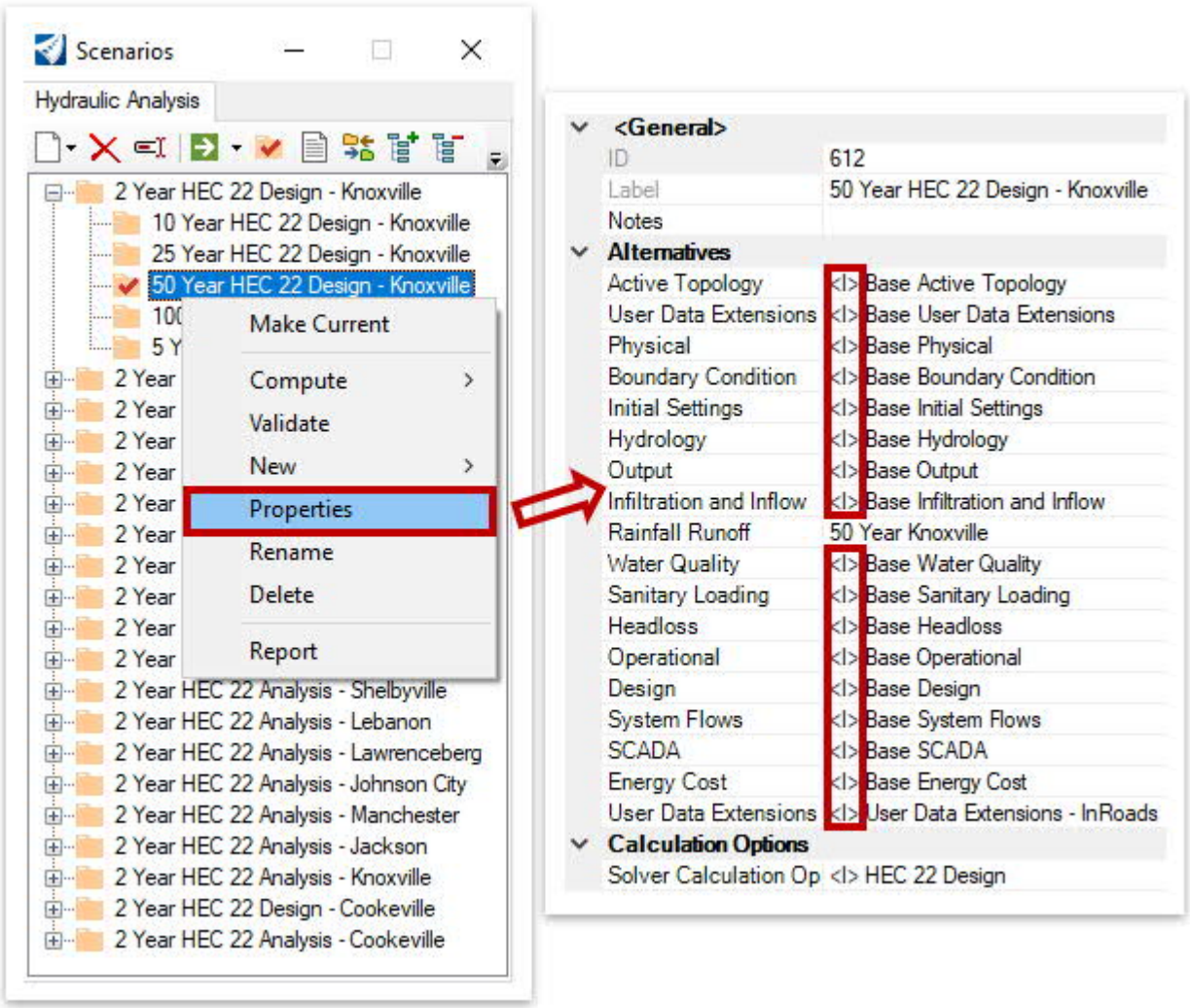
FIGURE 45. SCENARIOS - MAKE CURRENT OPTION



The user will not see a folder named **Base Scenario**, since the **TDOT Drainage Feature Defs.dgnlib** is customized for TDOT's standards.

Once the **Base** Scenario is set, the user can review the properties (Figure 46). Notice that the **inherited** properties highlighted below are indicated with an **<I>**. This means these alternatives were inherited from an initial scenario created for the project and are typically the same between all scenarios. Notice that the **Rainfall Runoff** Alternative is the only alternative that differs between the scenarios. The model can then be computed and will provide results under those conditions. In addition, if an alternative has an **<I>** in front, it can be inherited to a new Scenario. **Note:** The user doesn't need to create any new scenarios, unless approved by TDOT.

FIGURE 46. SCENARIO PROPERTY MANAGER



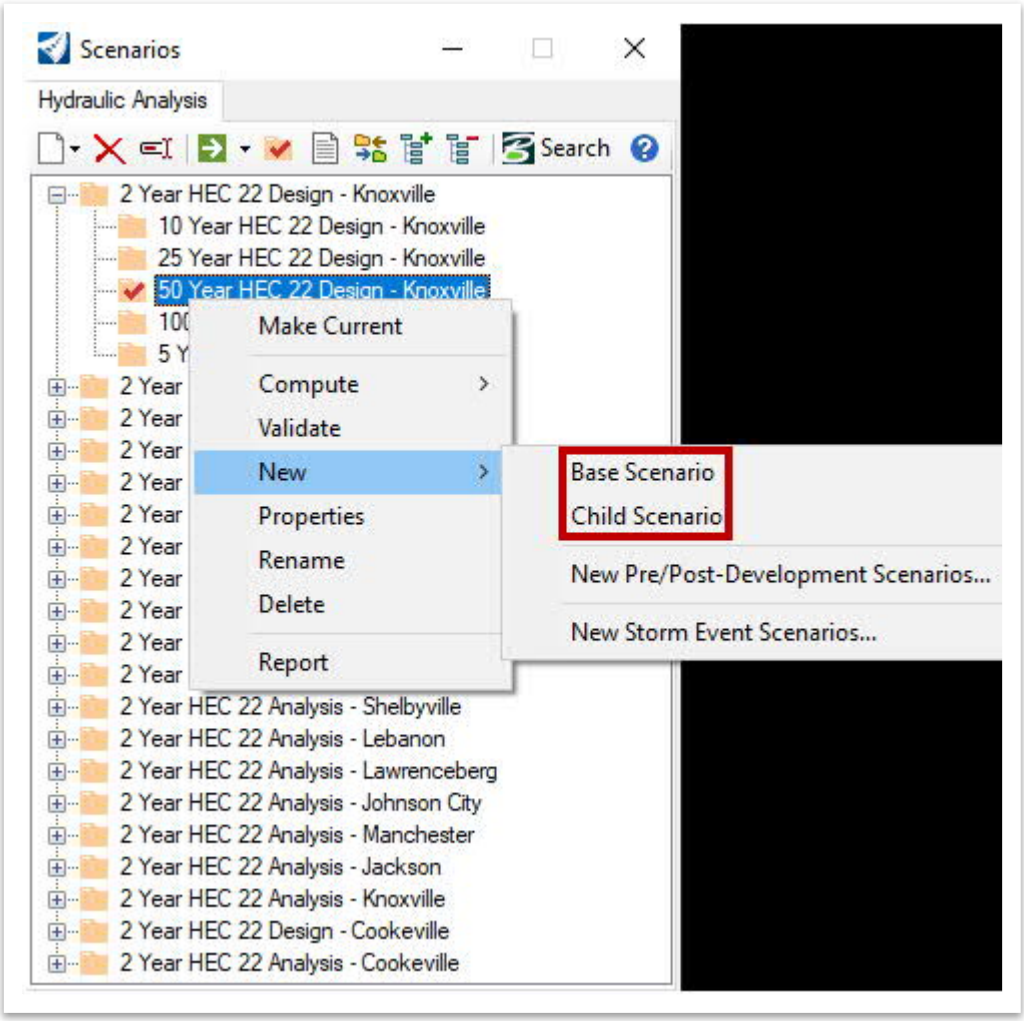
Take Note!

*Changes made in the **drawing** model, **Properties** and **Flex Tables** will automatically make changes to the values in the active alternative.*

Two categories of Scenarios can be created: **Base Scenarios** and **Child Scenarios** (Figure 47). They can be created by right clicking on any scenario and selecting **New**.

- **Base Scenarios:** Contain all the user's working data. It is the **current scenario** on which the model computes results. The user will need to set one of the scenarios as the **Current** scenario. **Note: The user will not typically need to create a Base Scenario within TDOT's workspace**.
- **Child Scenarios:** Inherits its data from its **Parent Scenario**. More precisely, its **Alternatives** and **Calculation** options are links to the same options as its **Parent**. This can be useful in situations where a new scenario will use most of the model data from an existing scenario, with only minimal changes to one of the Alternatives.

FIGURE 47. BASE - CHILD SCENARIOS



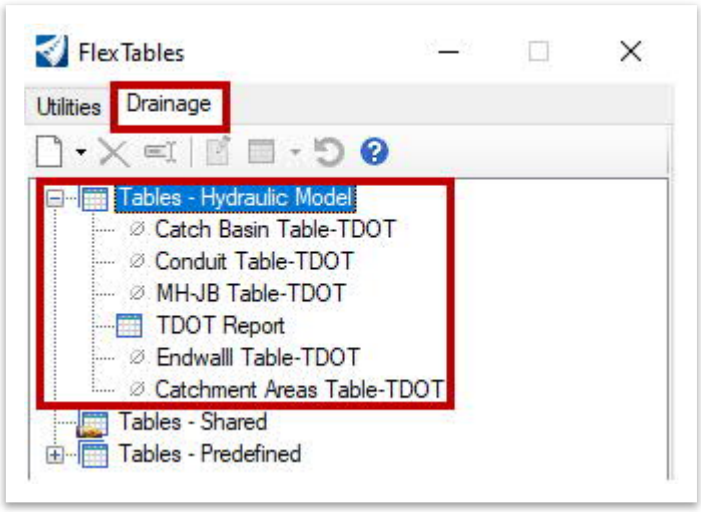
5.11.1.3 Flex Tables

The software has two sets of Flex Tables: **Utilities** (e.g., electric, gas, water, etc.) and **Drainage** (e.g., storm and sanitary). These are shown on two different tabs within the Flex Tables window. Closed network systems will utilize the Drainage tab.

The **Flex Tables** window shows a list of all the custom Flex Tables currently available for the user (Figure 48). Within the **Drainage** tab, Flex Tables are pre-defined per TDOT's preferences under **Tables - Hydraulic Model** and are loaded from the **TDOT Drainage Features Defs.dgnlib**. Flex Tables for Catch Basins, Conduits, MH-JB, TDOT Report, Outfalls and Catchment Areas provide all inputs and outputs for each element. Flex tables under **Tables - Predefined** represent Bentley formatted tables.

The user can click on the **blue question mark** icon to access the **Drainage and Utilities CONNECT Edition Help** browser. Here, the user can learn the theory behind the software and obtain troubleshooting support.

FIGURE 48. FLEX TABLES

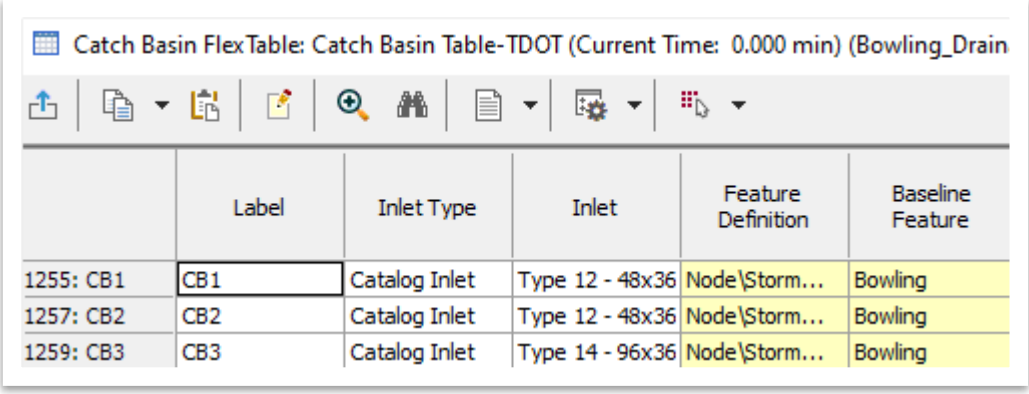


Take Note!

*Flex Tables can be accessed from both the **Analysis** tab and the **Utilities View** tab. The user should always work in the **Drainage** tab within the **Flex Tables** window.*

To open a TDOT flex table, simply double click on any table ending with **TDOT** in the name under **Tables - Hydraulic Model**. The **Catch Basin Table-TDOT** table is shown below (Figure 49).

FIGURE 49. FLEX TABLE - CATCH BASIN TABLE-TDOT



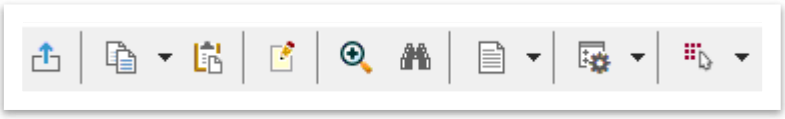
	Label	Inlet Type	Inlet	Feature Definition	Baseline Feature
1255: CB1	CB1	Catalog Inlet	Type 12 - 48x36	Node\Storm...	Bowling
1257: CB2	CB2	Catalog Inlet	Type 12 - 48x36	Node\Storm...	Bowling
1259: CB3	CB3	Catalog Inlet	Type 14 - 96x36	Node\Storm...	Bowling

Columns containing **editable data** are displayed with a **white background**. These columns can be edited directly in the flex table and the changes are applied to the model. Although the flex tables are dynamically tied to the design, it is not recommended to make the changes in the flex table. Using the flex tables only for review purposes helps on the data management side. While the design model will update dynamically, the calculations will still need to be re-run.

Columns containing **non-editable data** are displayed with a **yellow background** and correspond to model results calculated by the program and composite values. The content in these columns will change if any information is changed in either the design file or **Utility Properties**, or if a computation is run.

The tools to edit the Flex Tables are located at the top of the window and are described below (Figure 50).

FIGURE 50. EDIT FLEX TABLES



Export to File: Allows the user to export tabular data to a separate file in .shp, .txt or .csv formats.

Edit: Allows the user to add or remove available columns of various data fields from the current Flex Table as well as rearrange the column order.

Zoom To: Zooms to the highlighted element.

Find: Creates a text search within the table like that of the **Find** function in Microsoft Word.

Report: Generates a printable report for the given Flex Table and its respective fields. The report can also be generated in XML format.

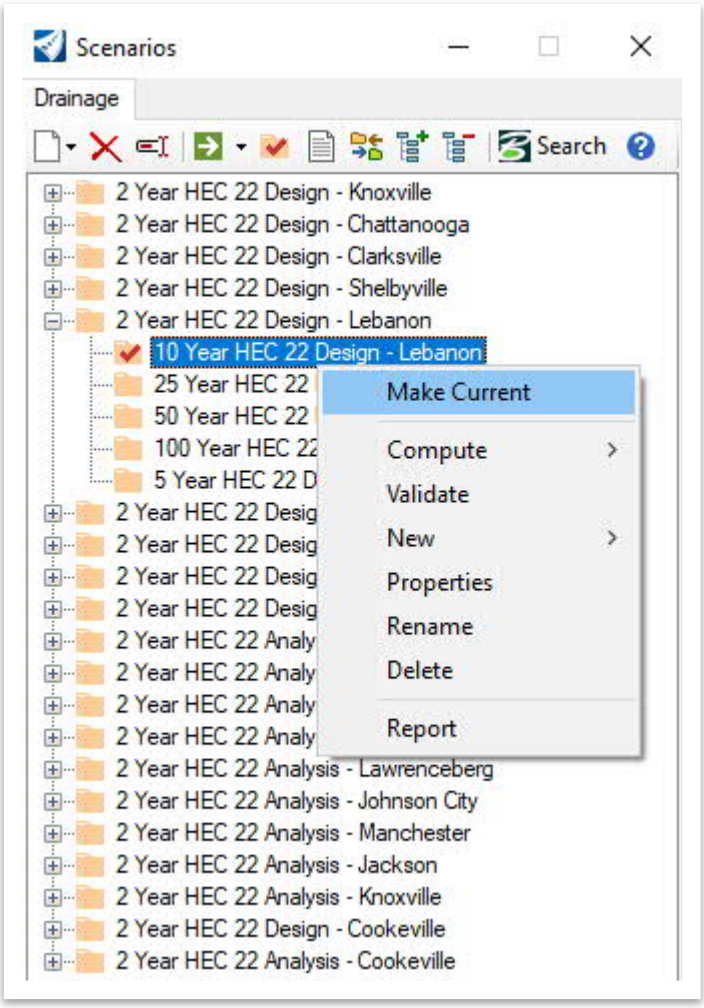
Options: Allows the user to create and edit selection sets (user-defined groups of network elements) so that the group of elements can be manipulated together. Elements can also be re-labeled in the Flex Table.

Select in Drawing: Allows the user to highlight any single row or set of rows and select elements to be highlighted in the dgn for edits or adjustments.

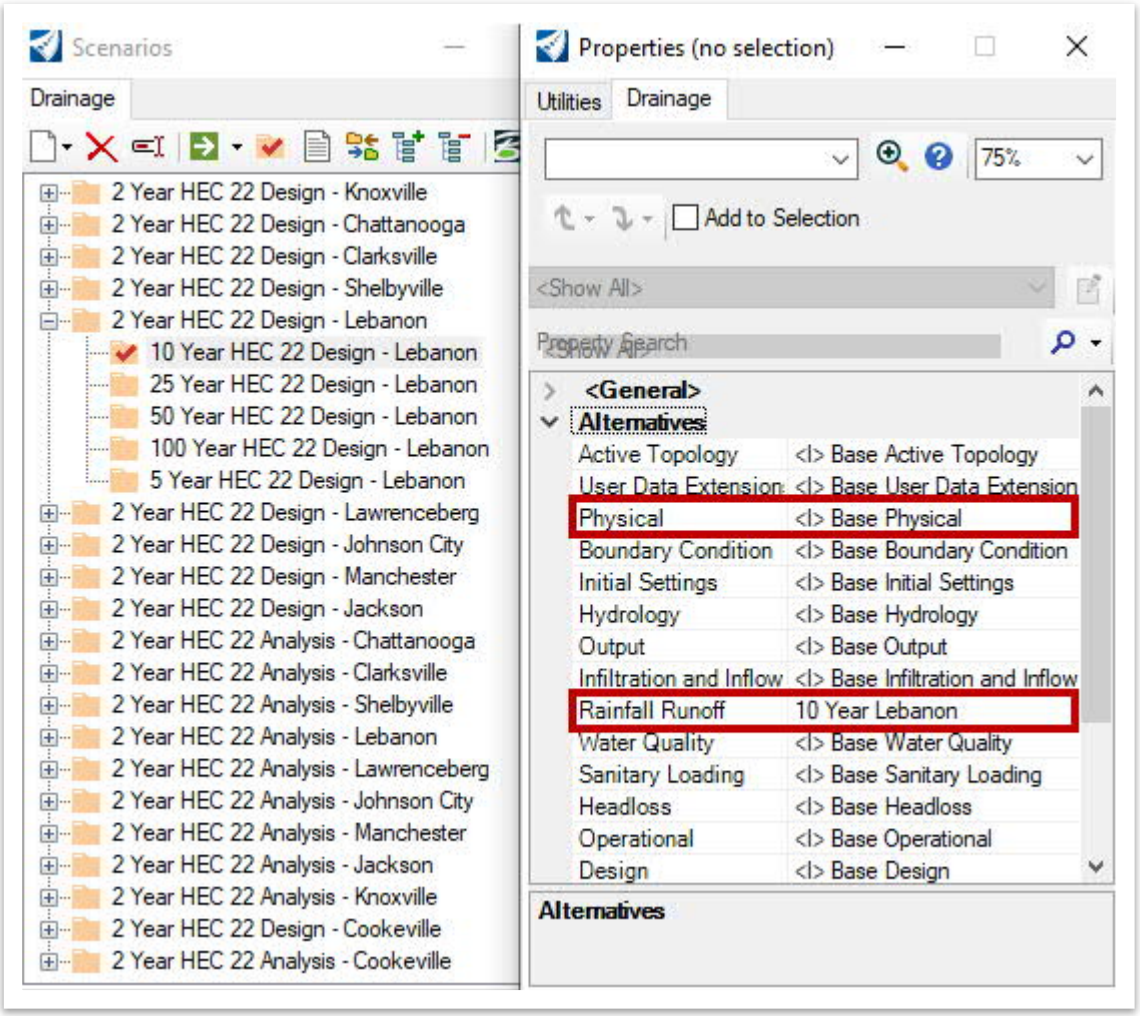
5.11.1.4 Exercise: Running Scenarios and Intro to Flex Tables

In this exercise, we will learn how to apply scenarios, create new alternatives, and link the alternatives to the scenario. We will apply the **10 Year Lebanon Design** scenario which will determine if the current proposed design in the model meets the criteria for the 10 Year design storm intensities within the Lebanon area based on the TDOT Drainage design guidelines. The design function will optimize the conduit (pipe) size and box depths to ensure minimum / maximum cover, hgl elevations, and pipe capacity. Spread calculation checks for the inlets should be done prior to this step (and prior to placing conduit). We will continue to utilize the same **Bowling_Drainage.dgn** file.

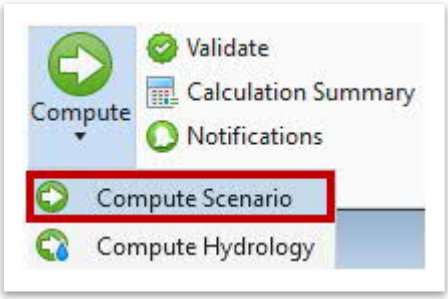
1. First, go to **File >> Save As** and name the file **Bowling_Drainage_ALT**. Then, open the **Scenario Manager** tool (**Drainage and Utilities >> Analysis >> Calculation >> Scenarios**). The Base scenario we will use for the current project is **10 Year HEC 22 Design - Lebanon**. Right click and **Make Current**. You should see a red checkmark next to the scenario once current.



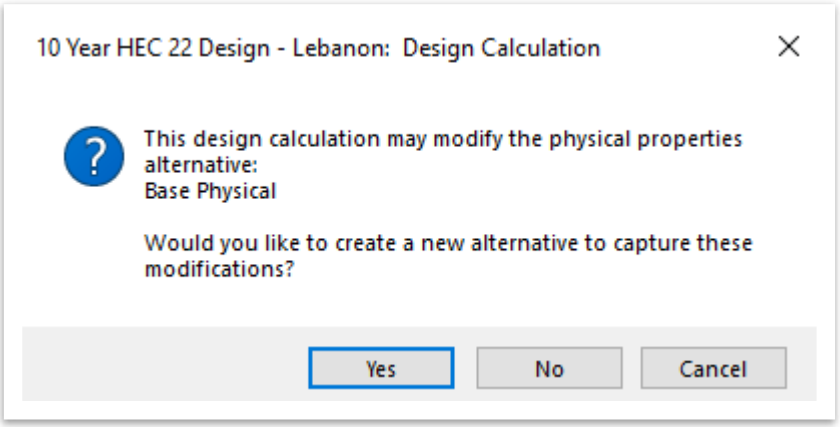
2. If no new alternative is created, the results from this scenario will be written to the **Base Physical** alternative. Double-click on the **10 Year HEC 22 Design – Lebanon** scenario to open the **Utility Properties**. Notice under **Alternatives** that **Physical** is set to **<I> Base Physical** and **Rainfall Runoff** is set to **10 Year Lebanon**. Close both windows once you are finished reviewing.

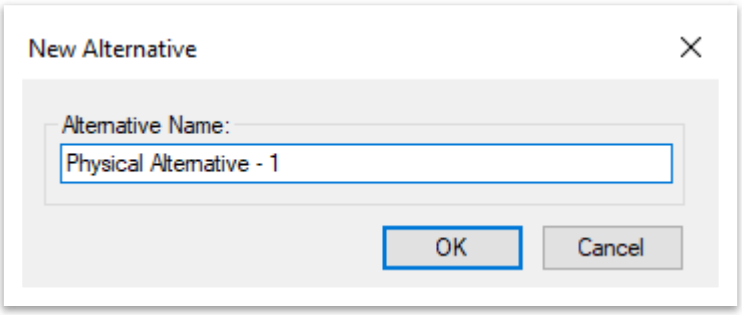


3. Now, open the **Compute Scenario** tool (**Drainage and Utilities >> Analysis >> Calculation >> Compute**).

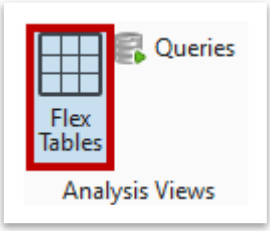


4. You should get a **Design Calculation** alert. Click **Yes** to create a new alternative. Leave the default name as-is (**Physical Alternative - 1**) and then click **OK** and give the software a minute to process. Go ahead and close the **GVF-Rational Calculation Summary** and **Civil Message Center** windows.

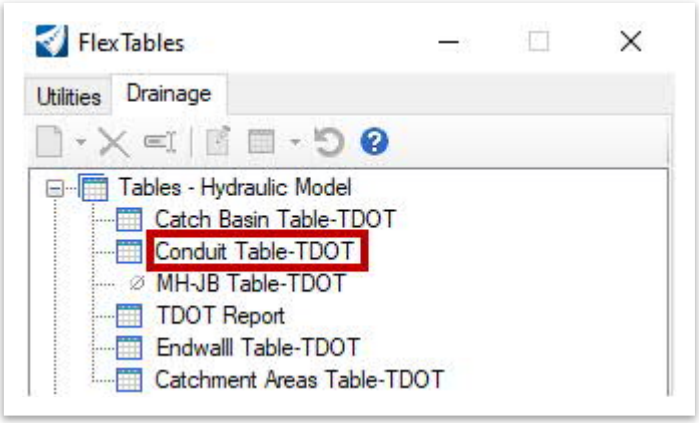




5. Next, open the **Flex Tables** tool (**Drainage and Utilities >> Analysis >> Analysis Views**).



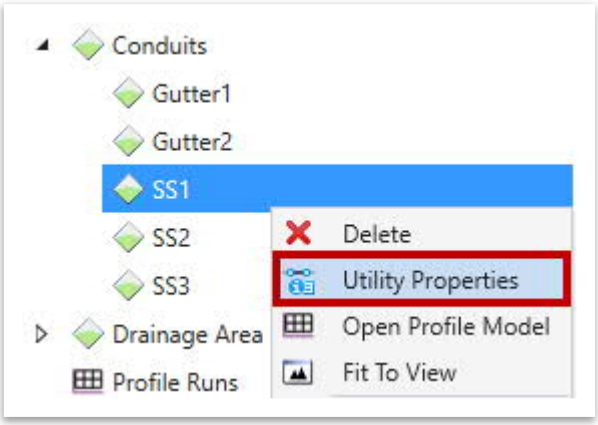
6. Within the **Flex Tables** window, double click on the **Conduit Table-TDOT** flex table under the **Drainage** tab.



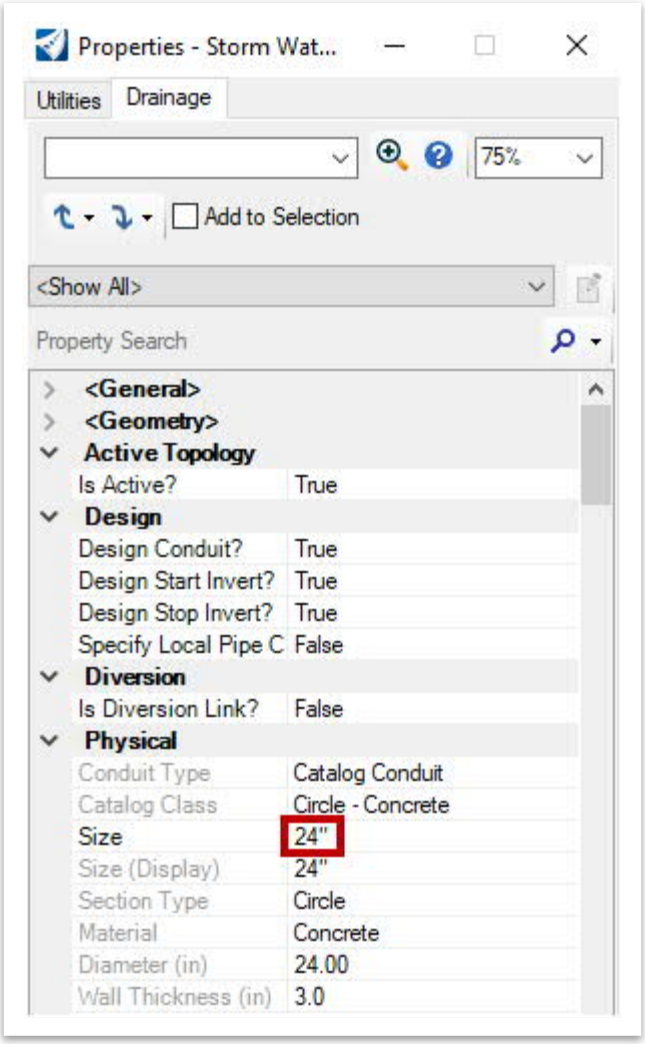
7. We are going to update the size of **SS1**. Under the **Size** column, change the size from 12" to **24"** using the drop-down. Notice the **Diameter (in)** column automatically updates. Close the flex table when you are finished.

Label	Feature Definition	Catalog Class	Start Node	Stop Node	Conduit Type	Conduit Description	Material	Section Type	Diameter (in)	Size
SS1	Conduit\...	Circle - C	CB1	CB3	Catalog Conduit	Circle24.00in	Concrete	Circle	24.00	24"
SS2	Conduit\...	Circle - C	CB2	CB3	Catalog Conduit	Circle12.00in	Concrete	Circle	18.00	18"
SS3	Conduit\...	Circle - C	CB3	EW1	Catalog Conduit	Circle12.00in	Concrete	Circle	18.00	18"

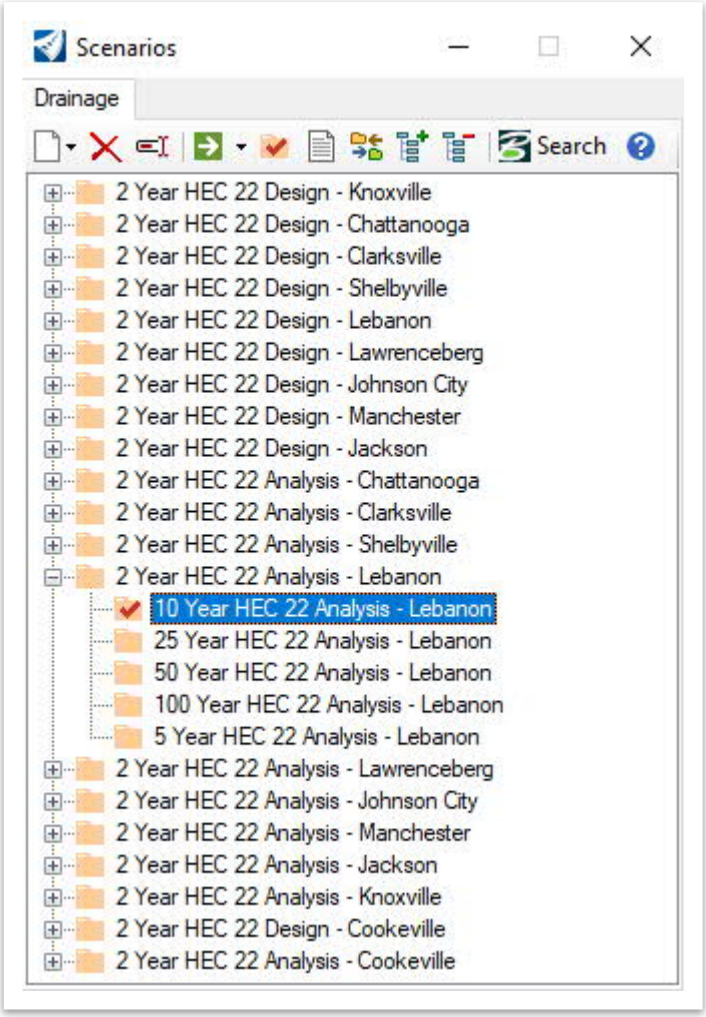
8. Next, go to the **Drainage and Utilities** model within the **Explorer** and expand **Conduits**. Right click on **SS1** and select **Utility Properties**.



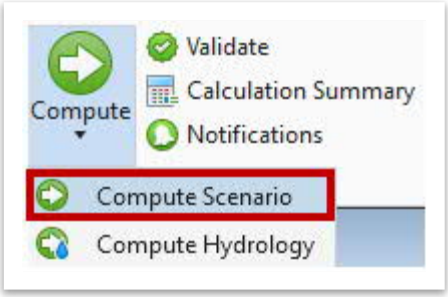
9. Under the **Drainage** tab, notice that the SS1 **Size** has been updated to **24”**. Close the utility properties once you are finished reviewing.



10. Now, let's run the **Analysis** scenario. Open the **Scenario Manager** tool once again (**Drainage and Utilities >> Analysis >> Calculation >> Scenarios**). Select the **10 Year HEC 22 Analysis - Lebanon**. Right click and **Make Current**. You should see a red checkmark next to the scenario once current.



11. Now, open the **Compute Scenario** tool once again (**Drainage and Utilities >> Analysis >> Calculation >> Compute**).



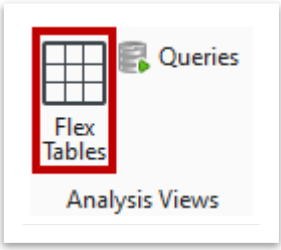
12. Lastly, open the **Flex Tables** tool once again (**Drainage and Utilities >> Analysis >> Analysis Views**). Within the **Flex Tables** window, double click on the **Conduit Table-TDOT** flex table under the **Drainage** tab. Confirm that the SS1 **Size** is **18”**. Once reviewed, close the **Conduit Table-TDOT** flex table. **Note:** For comparison purposes, the analysis run will utilize the original value to run the model. Also, the size shown is based on the analyzed size, which is what the software determined was sufficient for design.

Label	Feature Definition	Catalog Class	Start Node	Stop Node	Conduit Type	Conduit Description	Material	Section Type	Diameter (in)	Size
SS1	Conduit\...	Circle - C	CB1	CB3	Catalog Conduit	Circle 18.00in	Concrete	Circle	18.00	18"

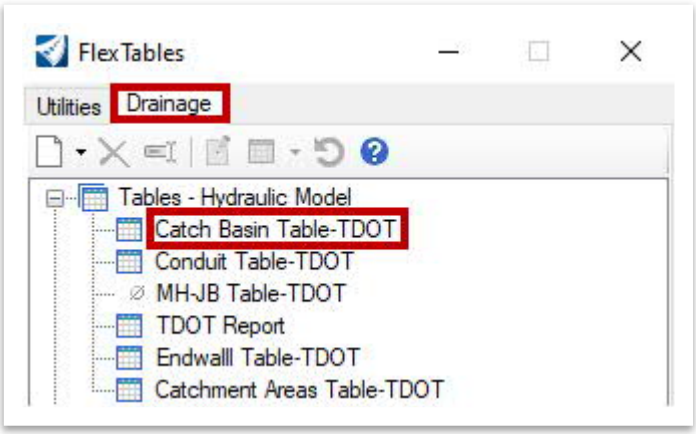
5.11.1.5 Exercise: Editing Flex Tables

In this exercise, we will show how to edit the Flex Tables in case a project specific scenario arises requiring a special table. We will open back up the **Bowling_Drainage .dgn** file.

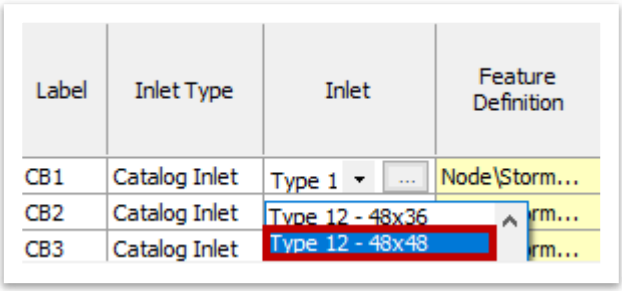
1. Open the **Flex Tables** tool (**Drainage and Utilities >> Analysis >> Analysis Views**).



2. Within the **Flex Tables** window, select the **Drainage** tab and then double click on **Catch Basin Table-TDOT** flex table.



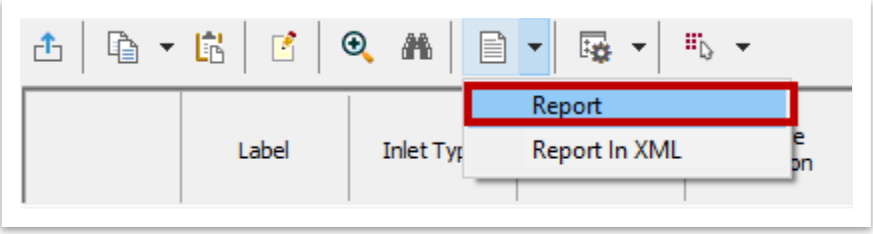
3. Within the **Catch Basin Table-TDOT** flex table, update the CB1 **Inlet** type to **Type 12 - 48x48** and then close the window.



4. Next, open the **Catchment Areas Table-TDOT** flex table. The software should populate the **Time of Concentration (min)** to be **5.000** which meets the minimum since no value was keyed in to override it. However, there is an issue with the current software version so go ahead and manually key in **5.000** for all three catchments. You can highlight all three fields and then right click on **Global Edit**.

Label	Outflow Element	Area (User Defined) (acres)	Runoff Coefficient (Rational)	Time of Concentration (min)	Flow (Total Out) (cfs)
Catchment 1	CB1		0.550	5.000	0.47
Catchment 2	CB2		0.550	5.000	0.15
Catchment 3	CB3		0.550	5.000	0.85

5. Now that we have manipulated the flex tables, we are going to apply a unique name to the Catch Basin flex table. Once again, open the **Catch Basin Table-TDOT** flex table (if not already open) and then open the **Report** tool and let the software generate the results.

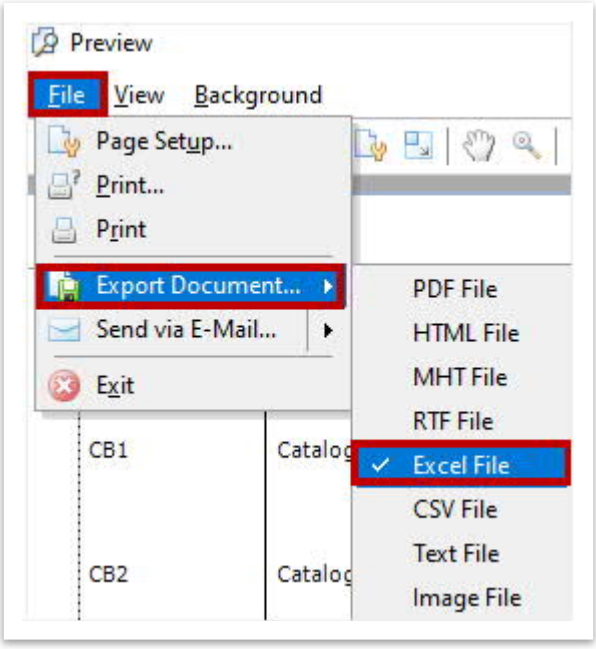


6. The report **Preview** window should appear and look like what is shown below. **Note:** Only a portion of the report is shown in the image below.

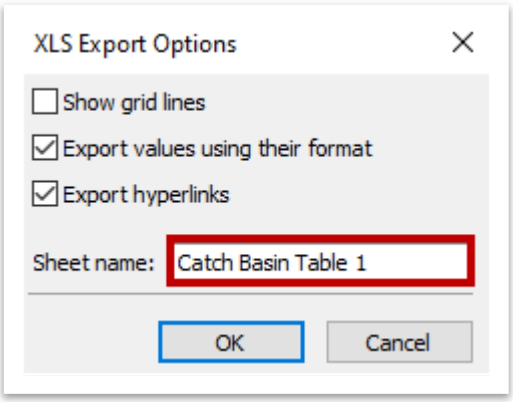
FlexTable: Catch Basin Table-TDOT

Label	Inlet Type	Inlet	Feature Definition	Baseline Feature	Baseline Station (US Survey Ft)
CB1	Catalog Inlet	Type 12 - 48x36	Node\StormWaterNode\Storm Drains\Proposed \Type 12-13\CB-12P	Bowling	19+08.50
CB2	Catalog Inlet	Type 12 - 48x36	Node\StormWaterNode\Storm Drains\Proposed \Type 12-13\CB-12P	Bowling	13+40.00
CB3	Catalog Inlet	Type 14 - 96x36	Node\StormWaterNode\Storm Drains\Proposed \Type 14\CB-14S	Bowling	17+04.25

7. Next, we need to export the table as an excel file. Within the report **Preview** window, go to **File >> Export Document >> Excel File**.



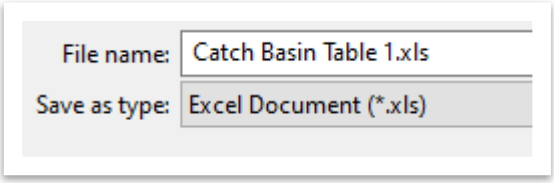
8. Within the **XLS Export Options** window, rename the **Sheet name** to **Catch Basin Table 1**. Then, highlight the name and copy it (**CTRL+C**) and click **OK**. **Note:** By default, the sheet name is **Sheet1**.



Take Note!

*After each scenario run, the necessary flex tables needed to complete a review of the entire system will be created for the specific inlets and conduits. As the system is optimized, there will be multiple flex tables that represent each run, thus each flex table will require a unique name such as **Catch Basin Table 1**. It is recommended to rename in Windows Explorer.*

9. Lastly, use **CTRL+V** and update the name of the excel file in the **Save As** window to match the sheet name (**Catch Basin Table 1**) and then click **Save**. **Note:** Make sure and save the file in the **SUDA_Training** dgn subfolder.

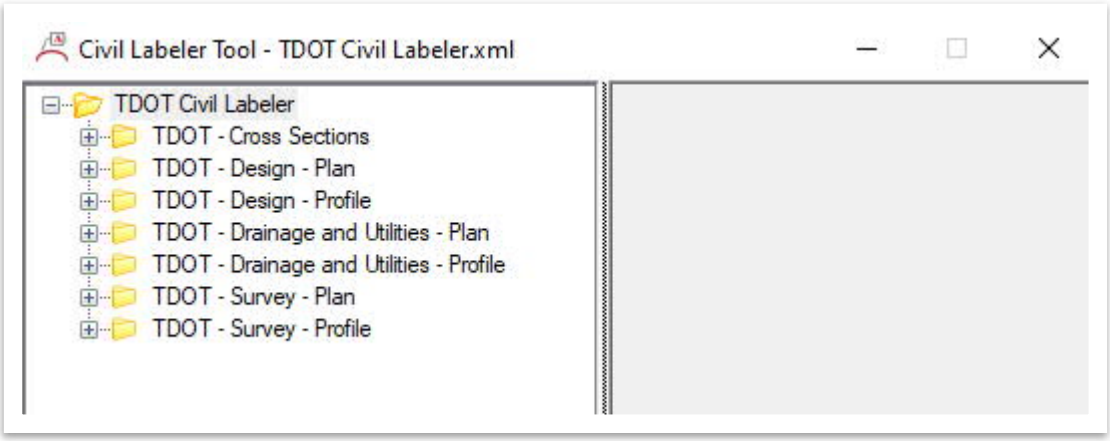


10. You can open the excel file to review if you'd like. Once you are done, go ahead and close the report preview window and close all flex tables.

5.12 Lecture: Civil Labeler Tool

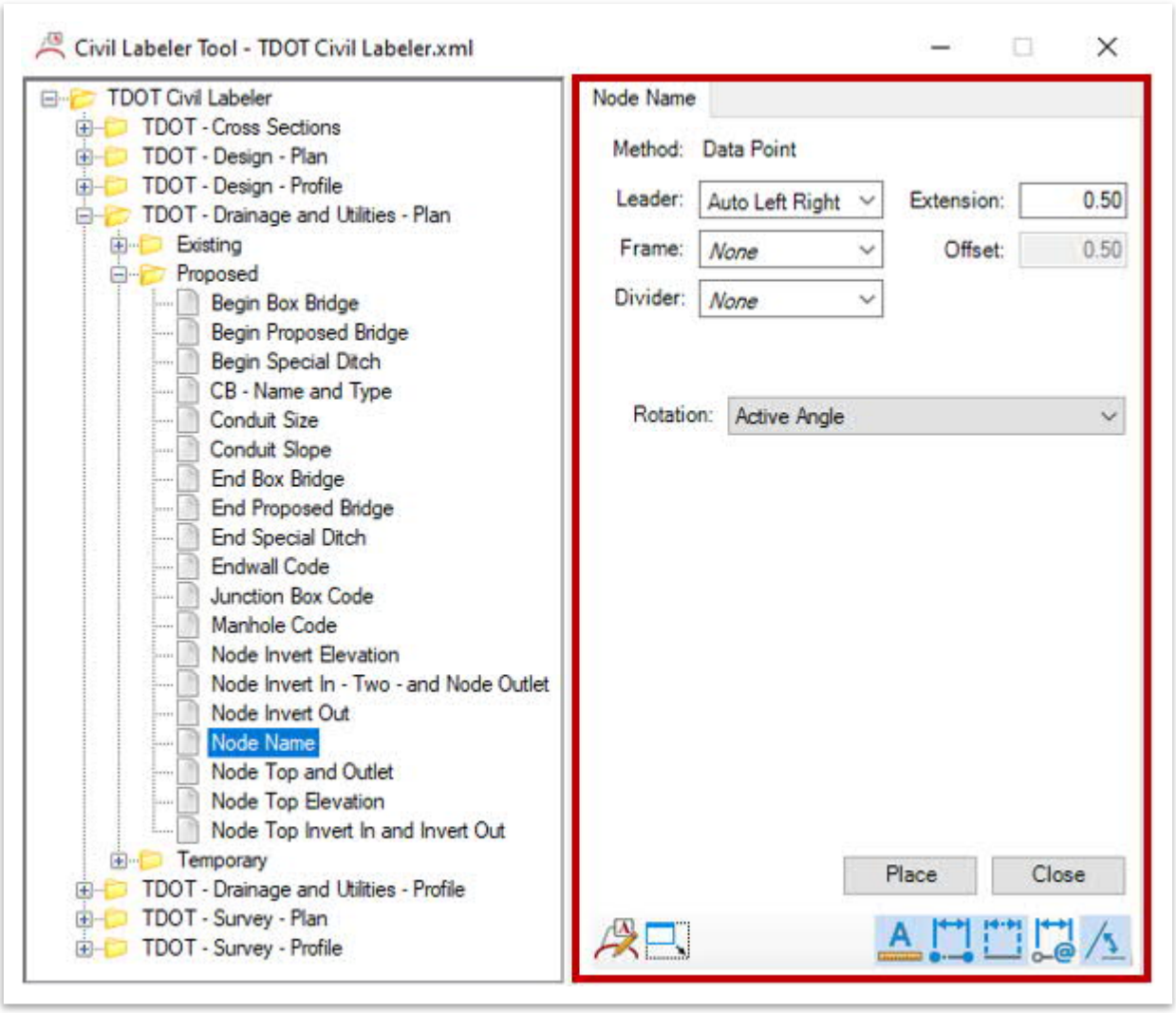
As a reminder, the Civil Labeler is a new tool in ORD 10.10 located here: **Drainage and Utilities >> Drawing Production >> Labels**. The Civil Labeler tool will eventually replace the place label tool. The TDOT Civil Labeler tool is built to replace the labels created for the Place Label tool on a 1 to 1 basis with additional labels created for the enhanced capabilities of the Civil Labeler tool. When opening the tool, give the software a minute to open. The TDOT Civil Labeler tool is broken down into the following seven sub folders (Figure 51). The **TDOT - Drainage and Utilities - Plan** and **TDOT - Drainage and Utilities - Profile** will be the primary folders utilized when dealing with SUDA annotation, which are further broken down into **Existing**, **Proposed** and **Temporary**.

FIGURE 51. CIVIL LABELER TOOL



To place a label, open the applicable folder structure and select the desired label. A **placement options** dialog will open on the right side of the window. There are several drop downs and key ins within the dialog to modify how the label is placed. The TDOT Civil Labeler has default values, but the user may want to modify them in some cases. The **Node Name** label is shown below with the default settings (Figure 52).

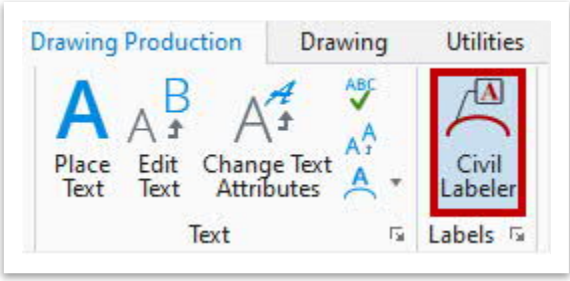
FIGURE 52. CIVIL LABELER TOOL – LABEL SETTINGS + EXAMPLE



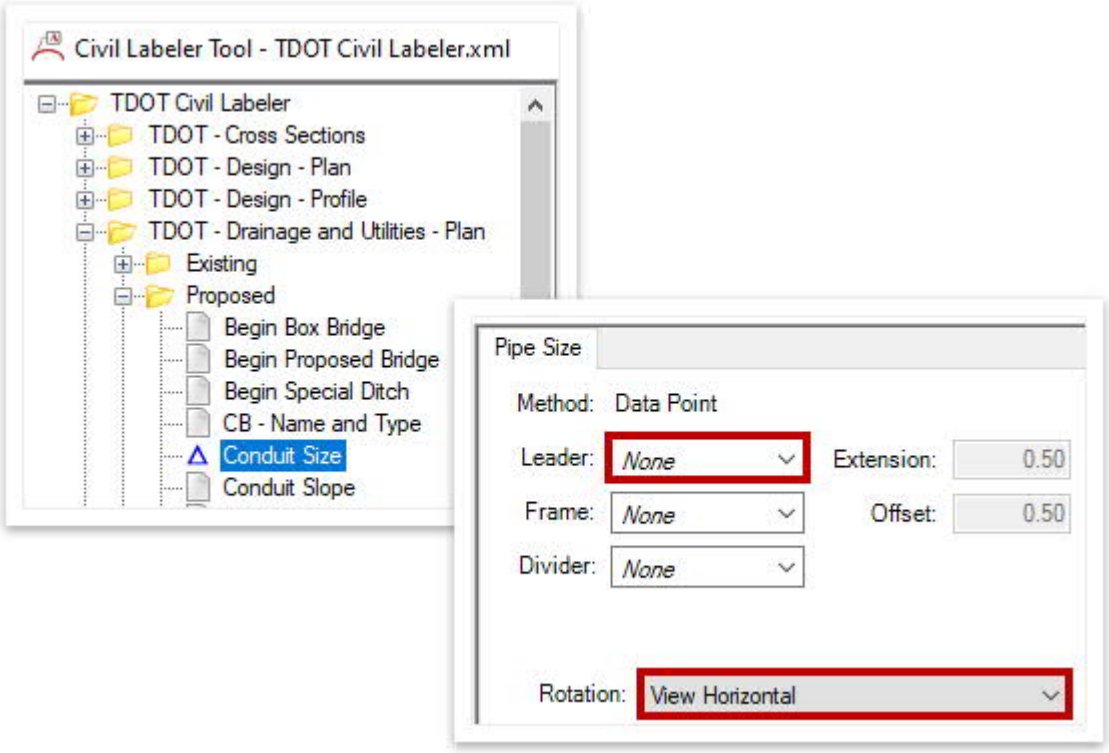
5.12.1 Exercise: Using the Civil Labeler Tool

In this exercise, we will learn how to use the **Civil Labeler** tool that has been setup within the TDOT ORD workspace. This will allow the model to be labeled in preparation for plan sheet development. We will continue to utilize the same **Bowling_Drainage.dgn** file.

1. First, open the **Civil Labeler** tool (**Drainage and Utilities >> Drawing Production >> Labeler**). **Note:** Turn off the **DES - PRO - Drainage - Storm Sewer** level.

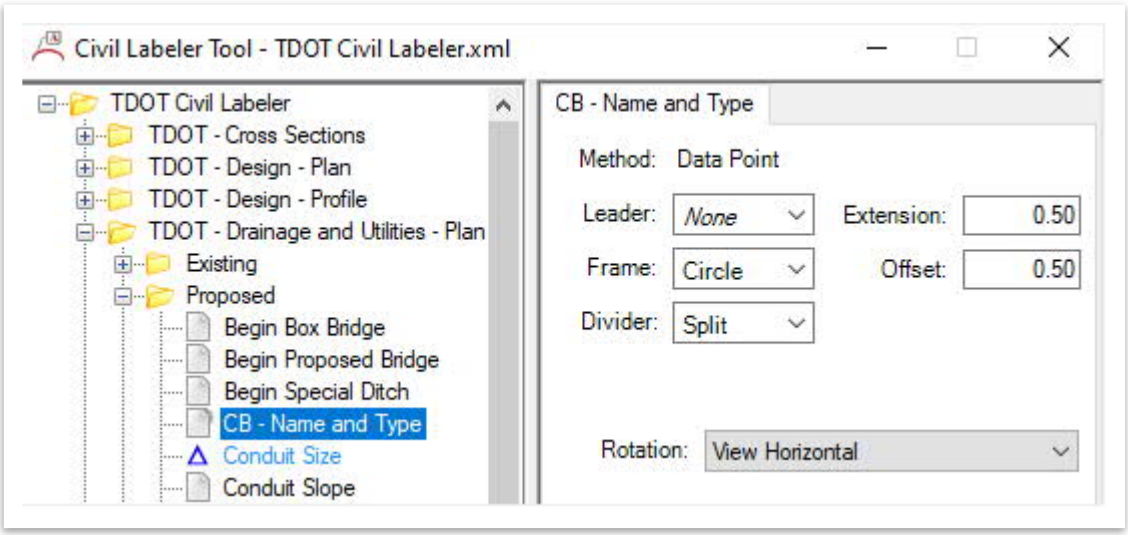


2. Within the civil labeler window, select the **Conduit Size** label (**TDOT Civil Labeler >> TDOT - Drainage and Utilities - Plan >> Proposed**). Change the **Leader** to **None** and the **Rotation** to **View Horizontal**. Leave all other predefined TDOT settings as-is. Click **Place**.



3. Notice the prompt in the lower left corner: **Data Point > Identify Conduit**. Left click to select the **SS1** conduit and then left click along the conduit. The label should now appear on your cursor. Then, left click again anywhere above the conduit to accept placement. **Note:** You may want to turn off the **DES - PRO - Drainage - Storm Sewer** level (profile run) to make it easier to select the conduit.

4. Next, select the **CB - Name and Type** label (**TDOT Civil Labeler >> TDOT - Drainage and Utilities - Plan >> Proposed**). Leave all predefined TDOT settings as-is. Click **Place**.



5. Notice the prompt in the lower left corner: **Data Point > Identify Catchbasin**. Select **CB1** and left click again. The label should now appear on your cursor. Left click again anywhere near the catch basin to accept placement. There are other labels within the Civil Labeler that will be utilized when labeling drainage items. The placement would follow a similar process. Go ahead and close the Civil Labeler once completed.

Chapter 6. Additional Software

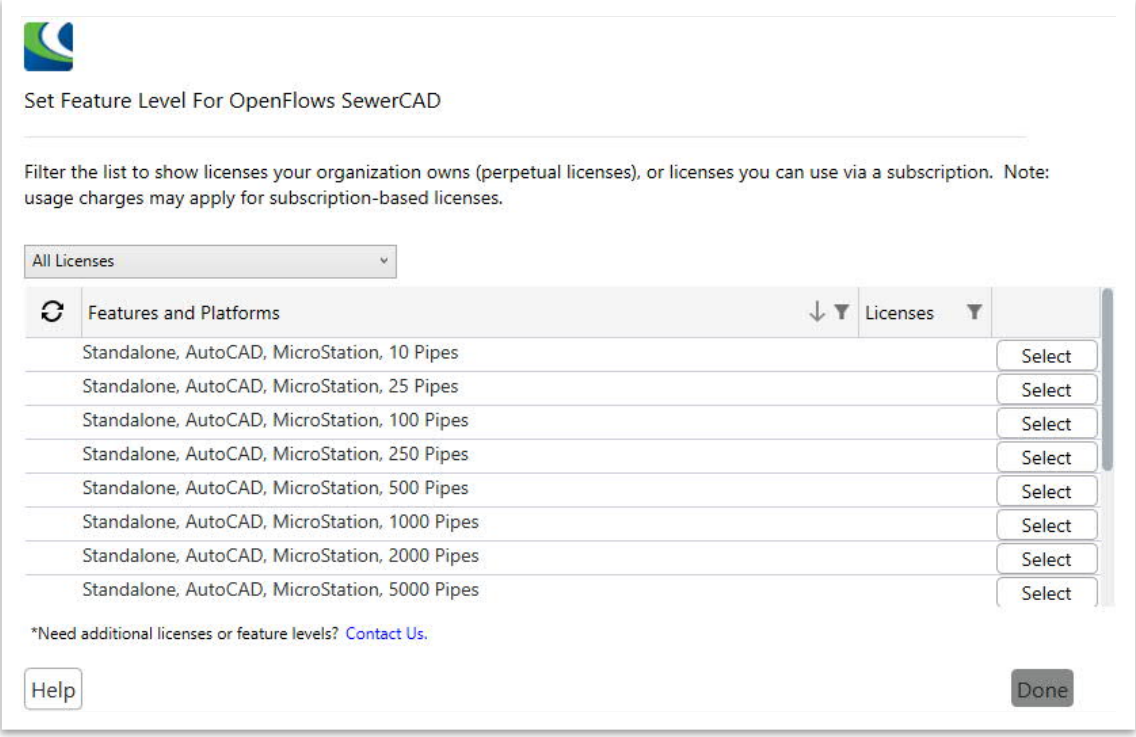
The **Analysis Tools** within the **Drainage and Utilities** workflow are predominately utilized to create a hydraulic analysis for closed and open channel drainage network systems. Additionally, there are tools such as StormCAD, SewerCAD, etc. that can also be accessed within the same workflow after license activation.

6.1 Lecture: Additional Software Activation

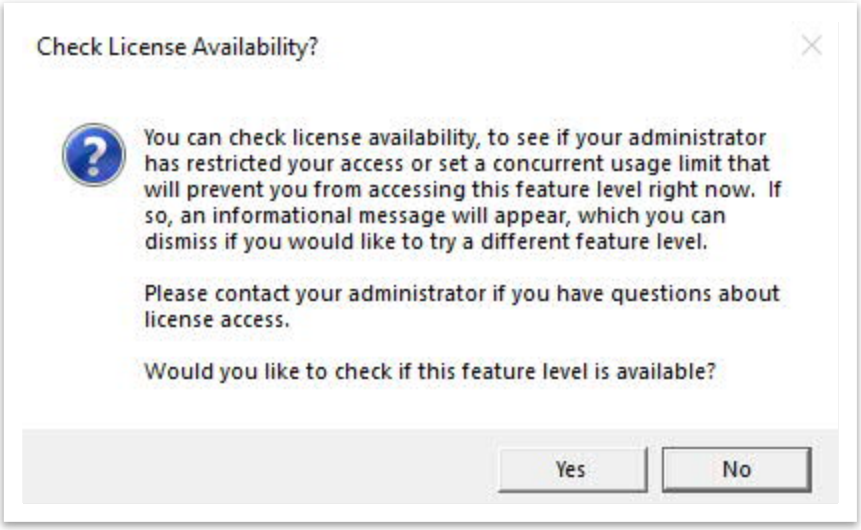
The following process can be utilized to manipulate license activation for the additional software within the Drainage and Utilities workflow. The process for **SewerCAD** activation is shown below.

1. To activate **SewerCAD**, open the license under **Drainage and Utilities >> Tools >> License Activation**.

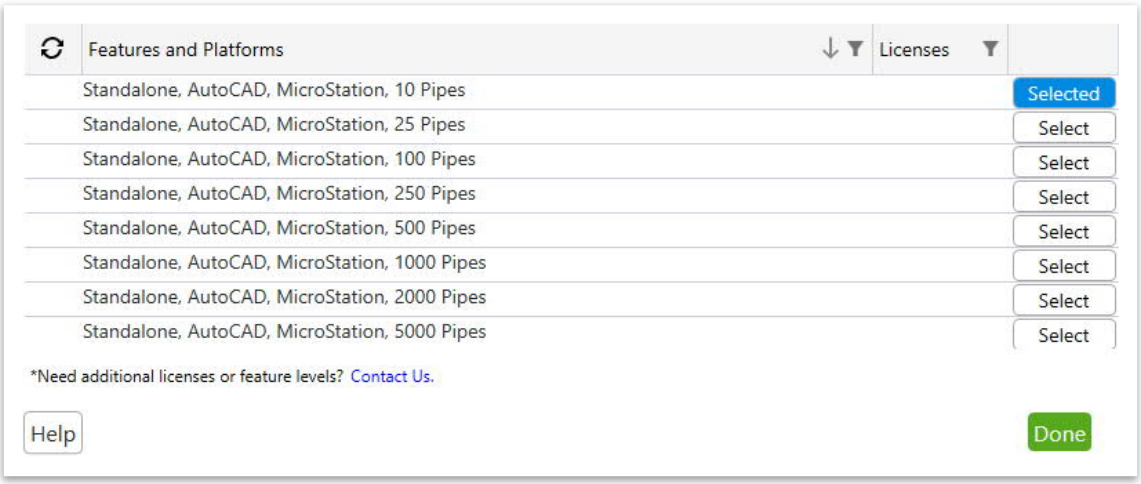
2. A Bentley **Feature Level Selection** window will open, showing all licenses your organization owns. Select the appropriate license for your specific project.



3. A warning message will appear prompting you to check license availability. Click **Yes**.



4. Once the license is confirmed, the option will say **Selected** and the **Done** button will turn green. Click **Done**.



5. To access the licensing tool and confirm the SewerCAD license on your machine, navigate to the Bentley **License Management Tool** by keying it into your Windows search bar.

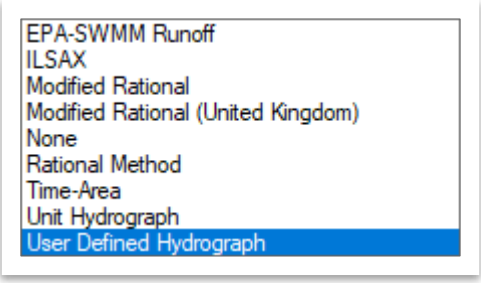
Revision History

DATE (MONTH/YEAR)	AUTHOR/EDITOR	IB #	SECTIONS MODIFIED

Appendix A. Alternate Runoff Methods

The Drainage and Utilities tool allows for other types of hydrologic calculations outside of the Rational Method (Figure 53). The TDOT Drainage Standards manual should be reviewed to determine which method is the most appropriate. You can access the Bentley links below for more insight on each method.

FIGURE 53. HYDROLOGIC CALCULATION OPTIONS

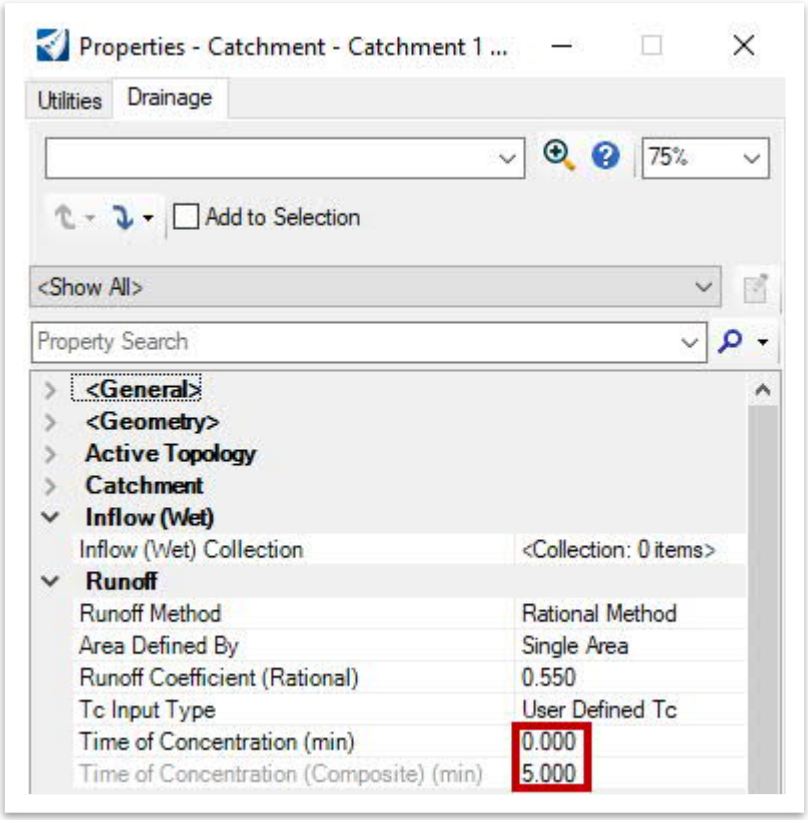


- **EPA-SWMM Runoff**:
  https://docs.bentley.com/LiveContent/web/Subsurface%20Utilities%20CONNECT%20Edition%20Help-v2/en/GUID-4EBC55D276B04A279825FE75ACD34347.html
- **ILSAX**:
  https://docs.bentley.com/LiveContent/web/Subsurface%20Utilities%20CONNECT%20Edition%20Help-v2/en/GUID-CECDC254-311C-4BE4-B6FE-4C246494DB9F.html
- **Modified Rational**:
  https://docs.bentley.com/LiveContent/web/Subsurface%20Utilities%20CONNECT%20Edition%20Help-v2/en/GUID-85A442CDB33D4B1684EE9E795BA6BABE.html
- **Rational Method**:
  https://docs.bentley.com/LiveContent/web/Subsurface%20Utilities%20CONNECT%20Edition%20Help-v2/en/GUID-8CEDC8A483BD4187BE37079B7E081F36.html
- **Time-Area**:
  https://docs.bentley.com/LiveContent/web/Subsurface%20Utilities%20CONNECT%20Edition%20Help-v2/en/GUID-E17D7A28-B33B-40B4-9473-C4EFD803A05F.html
- **Unit Hydrograph**:
  https://docs.bentley.com/LiveContent/web/Subsurface%20Utilities%20CONNECT%20Edition%20Help-v2/en/GUID-54B1B72A474E4C85826B54926F4416D8.html

Appendix B. Common Errors During Computations

Here are some common examples of **User Notifications / Warnings / Errors** and an explanation on how to fix them:

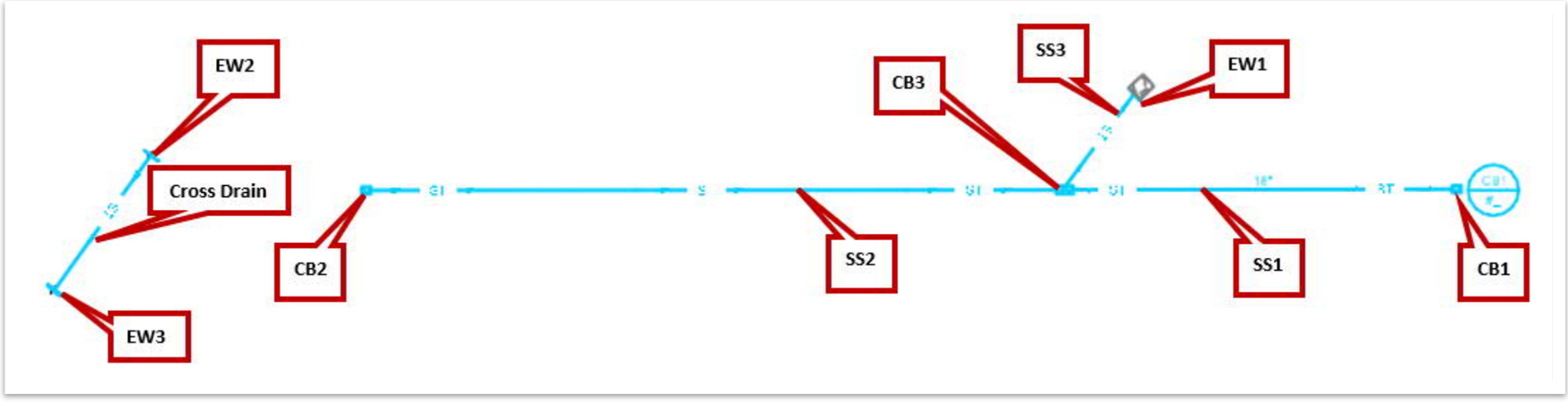
1. *Time of concentration for catchment is less than the minimum Tc value defined in the calculation options. The minimum Tc value was used.*
    - Meaning: No **Tc** value was defined for the **Catchment** in the **Utility Properties** by the user. Since the **Tc Input Type** field is seeking a **User Defined Tc**, the software is first looking for a value entered by the user. Since this value is shown as 0.00, the software is instead using the predefined minimum **Tc** value (**5 min**) as shown in the greyed-out **Time of Concentration (Composite)** field. If any **Tc** value higher than 5 min needs to be used, select the applicable catchment, and open the **Utility Properties**, and enter the value manually in the highlighted field below. You may also enter 5 (if you agree that the catchment has a minimum Tc of 5 min) in the highlighted field to see the warning disappear. **Note:** You can also select the catchment here: **Explorer >> Drainage and Utilities Model >> Drainage Area**.



2. *There is no gutter leaving this 'On Grade' catch basin. Bypassed flow is directed to the subnetwork outfall.*
    - Meaning: There is no gutter defined in the utility model to convey the bypass flow from an **on-grade** inlet to the next inlet. Use the **Place Gutter** tool (**Drainage and Utilities >> Layout >> Layout**) to place a gutter from one node to the other.
3. *Road Cross Slope should be greater than zero.*
    - Meaning: Verify the **road cross slope** that is defined within the **Gutter Catalog** located here: **Drainage and Utilities >> Components >> Catalog >> Catalog**. If any "**-**"**signs** are seen, change them to positive.
4. *Conduit Section Size is not selected, or the selected Section Size doesn't exist in the Conduit Catalog.*
    - Meaning: Within the conduit **Utility Properties**, select the **conduit size**.
5. *There are no design lengths available in the Inlet Catalog. Set the design inlet opening property to False or add design length.*
    - Meaning: Under **Utility Properties** for Inlets, **Set the Inlet Design Opening** to **False**. This means that the software will not design the inlets and will use the default dimensions for the inlets.
6. *The width of ponding exceeds the maximum spread constraint for this inlet.*
    - Meaning: The inlet cannot handle the incoming flow of the utility model and the spread calculated exceeds the allowable spread. **Increase the capacity** of the inlet (e.g., triple grate inlet) or add an **intermediate** inlet between the upstream inlet and the inlet that is not meeting spread. At times, adjusting the upstream inlet's location also helps with alleviating the downstream spread widths.
7. *The upstream connected endwall's culvert coefficients data is ignored because the conduit link is not set as a culvert.*
    - Meaning: The conduit is not being modeled as a culvert. Within the conduit **Utility Properties**, change the **Is Culvert** field from **False** to **True**.

Appendix C. Example Drainage Network

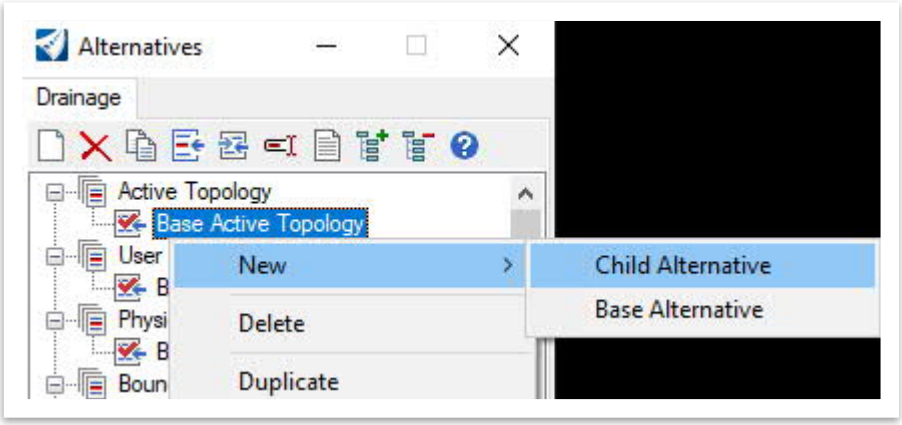
An overview of the completed project is shown below for reference throughout the class. This includes the plan view location and annotation of the catch basins, endwalls and conduits (including the cross culverts, SA line and driveway culvert).



Appendix D. Creating Separate Topology for each Scenario

Before running a scenario (either on-site or cross drains), the active topology to the respective elements needs to be checked. The following example shows how to create a topology scenario. The user can make variations using the child alternative.

1. Name the Active topology (e.g., On-site Systems) and then check all On-site Systems that need to run for a 10-yr scenario.
2. Then, create a **Child Alternative** folder under Topology and give it a name (e.g., Cross Drain Structures). Check all cross drains that need to be run for a 50-yr scenario.

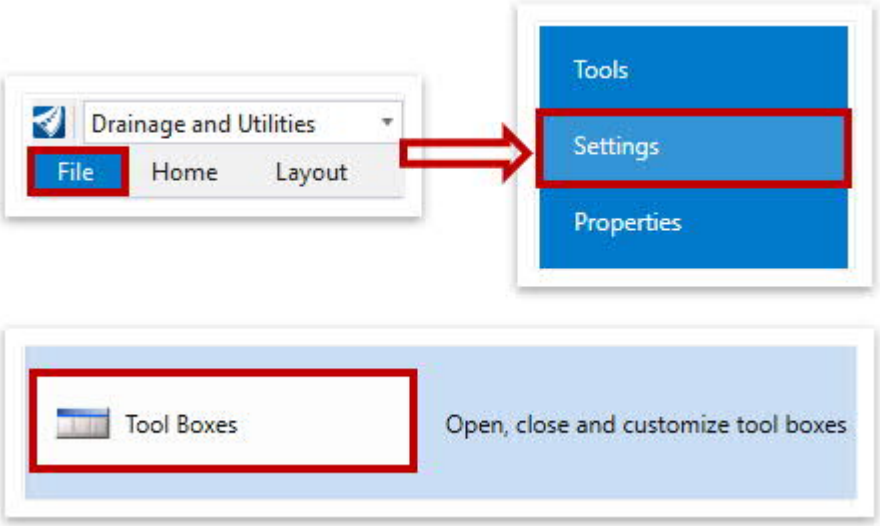


3. To keep all alternatives, the user should create a child alternative with a unique name and then link the respective topologies to the scenarios. This will run the scenarios only on those elements, which will minimize the need to check / uncheck the topology each time. If running an analyze scenario, select the analyze scenario and link the desired topology that you want to run on it.

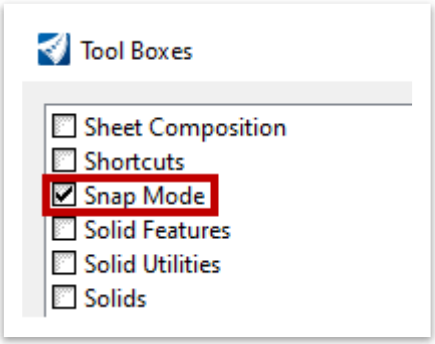
Appendix E. Snap Mode Tool Box

It is highly recommended to keep the **snap mode** tool box docked within ORD, as the snaps are used quite frequently. The following steps are one method to achieve this, which can be done in any design file. The tool box will remained docked each time you open the software thereafter.

1. Go to **File >> Settings >> Tool Boxes**.



2. Scroll down and toggle on the **Snap Mode** option.



3. Dock the tool box anywhere within your screen. The screenshot below shows it docked at the bottom.

Appendix F. LiDAR Data Access

There are multiple ways to access LiDAR data, which can then be used in ORD to create extended surfaces. For consultants, it will also require a data conversion process on the backend. The overall process for each consultant option is further detailed in the next two exercises.

- **TDOT:** Browse to the following location on the TDOT network and select the applicable tile(s): **\\tdot05nas 001\GIS\LiDAR**.
- **Consultants (if using Google credentials):** Exercise F.1.
- **Consultants (if not using Google credentials):** Exercise F.2.

F.1 Exercise: State of Tennessee LiDAR Coverage

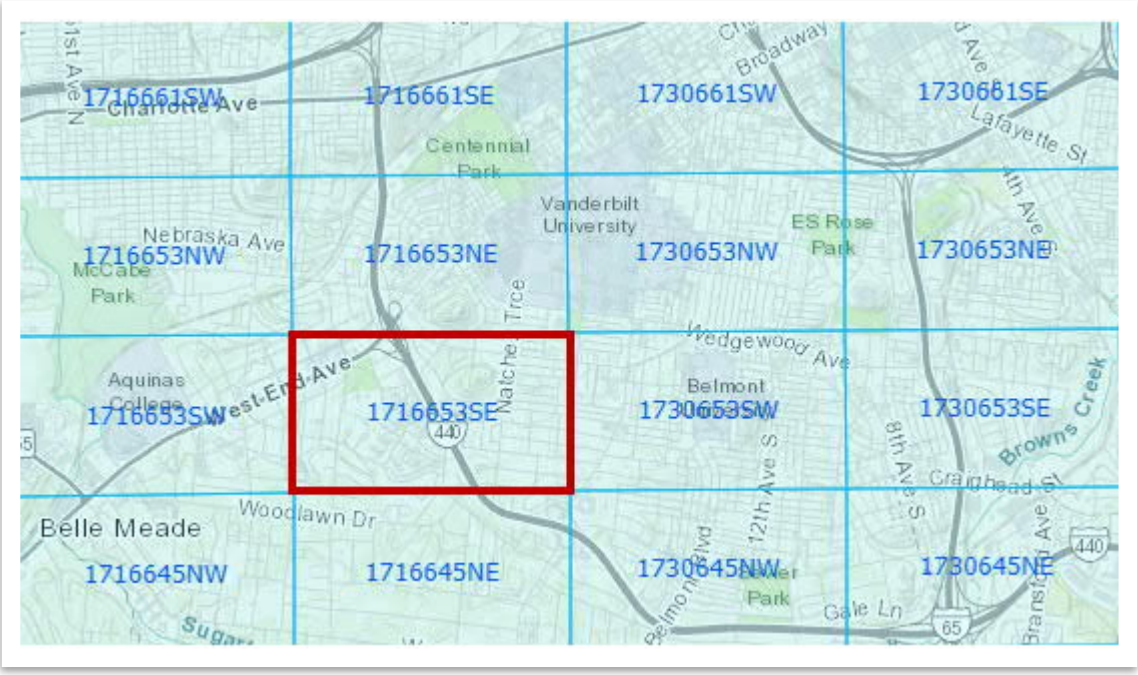
In this exercise, we will show how to download LiDAR data from the **State of Tennessee LiDAR Coverage** website. We will use the Bowling Avenue project location in Davidson County, TN.

1. Sign in to your google account and then click the following link to open the **State of Tennessee LiDAR Coverage** website. **Note:** Do not use Internet Explorer. **Google Chrome** was used for the exercise images, but other current browsers may be used. **Note:** In the future, this download option may not require Google credentials.

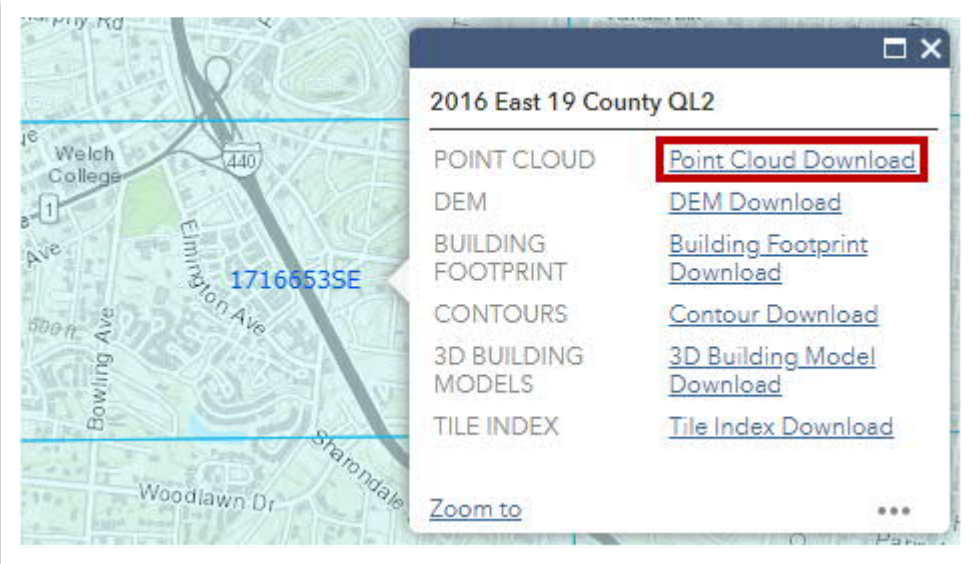
   https://tnmap.maps.arcgis.com/apps/webappviewer/index.html?id=1bf015da6e10402487a6bae1c0f71385

2. In the upper left corner, key-in **Davidson County, TN** in the search bar and then hit **Enter**. This will narrow the search location to the applicable area.

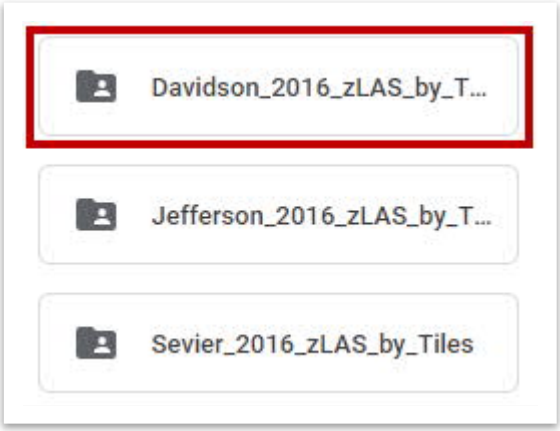
3. Zoom in to the project area (Bowling Avenue) by clicking the **+** sign in the upper left corner or by using your mouse wheel. We will download the highlighted tile shown below: **1716653SE**.



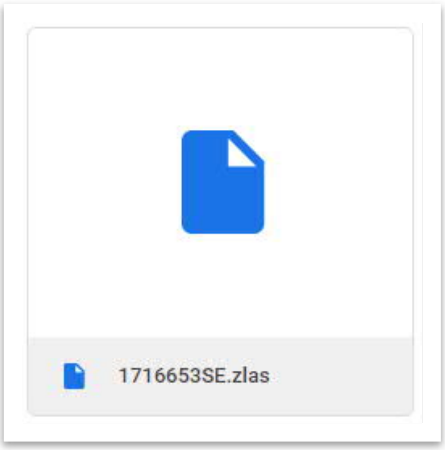
4. Left click anywhere within the tile and select **Point Cloud Download**.



5. You'll notice that various county folders appear within **Google Drive**. Double click on **Davidson County** for this exercise.



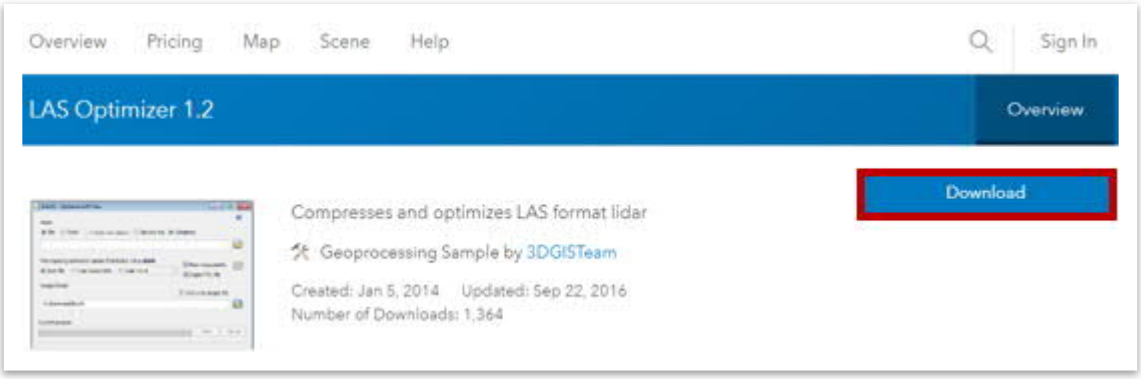
6. Within the **Davidson County** folder, notice all of the tiles are listed in **.zlas** format. Scroll down and right click on **1716653SE.zlas** and select **Download**. Accept any prompts that might appear. Save the file in the following location: **C:\Program Data\Bentley\OpenRoads Designer CE 10.10\Configuration\WorkSpaces\TDOT_Standards\WorkSets\SUDA_Training\dgn**. **Note:** To download the data, you will need to be signed into Google Drive with your Google credentials. The Google credentials may not be requried in the future.



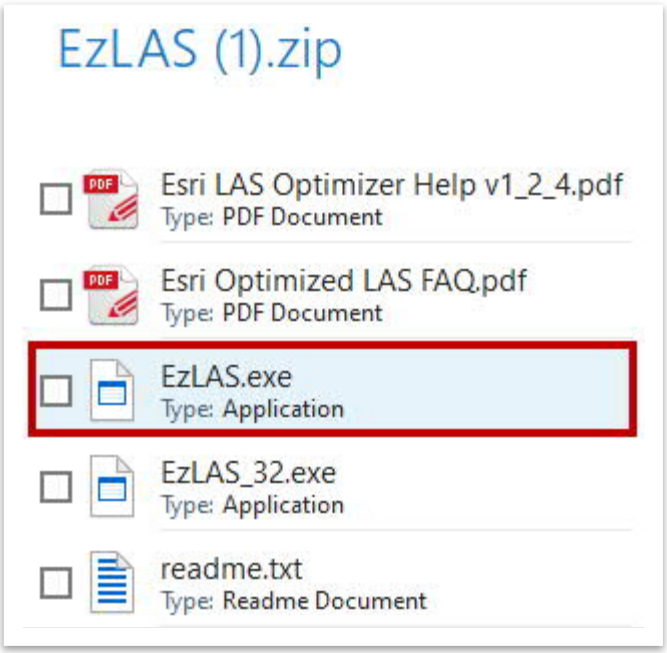
7. Since the file is in a **.zlas** format, it needs to be converted to a **.las** format to import into ORD using an LAS converter. Click on the following link to access the **LAS Optimizer 1.2** converter website. **Note:** If you already have the converter installed, open it and skip to Step 10.

   https://www.arcgis.com/home/item.html?id=787794cdbd384261bc9bf99a860a374f

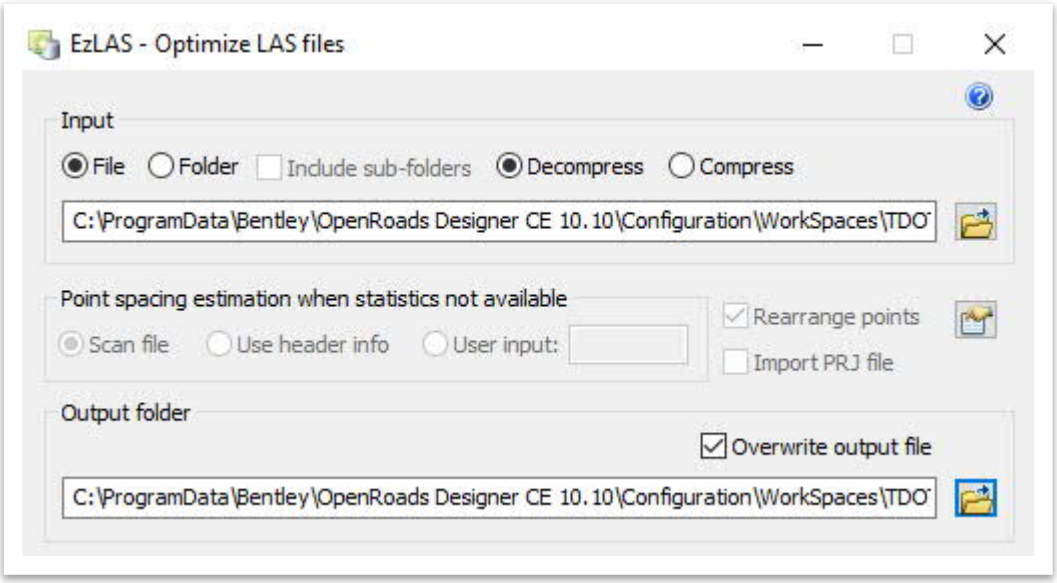
8. Select **Download** in the upper right corner to download the **EzLAS.zip** file.



9. Open the zip file and then double click on the **ExLAS.exe** file. If prompted, click on **Yes, open this file**. **Note: WinZip Pro** was used in the image, but you may have a different program.



10. An **EzLAS - Optimize LAS files** window should open. Under **Input**, select **File** and **Decompress** and then browse to the **.zlas** file. Under **Output folder**, select the folder where the .las file will be saved, which should be the same dgn subfolder within the **SUDA_Training** workset folder.

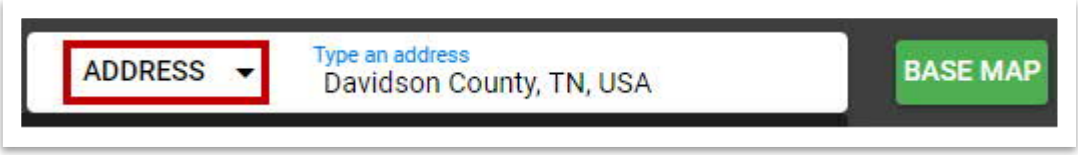


11. Click **Start** to begin the conversion process from **.zlas** to **.las** format. If you had multiple tiles, you will need to do this process for each tile separately. Once completed, you should see the .las file in your dgn subfolder and can now use it in ORD. **Note:** Since this .las file was part of the class files, it will simply save over the previous version.

F.2 Exercise: USGS 3DEP LiDAR Explorer

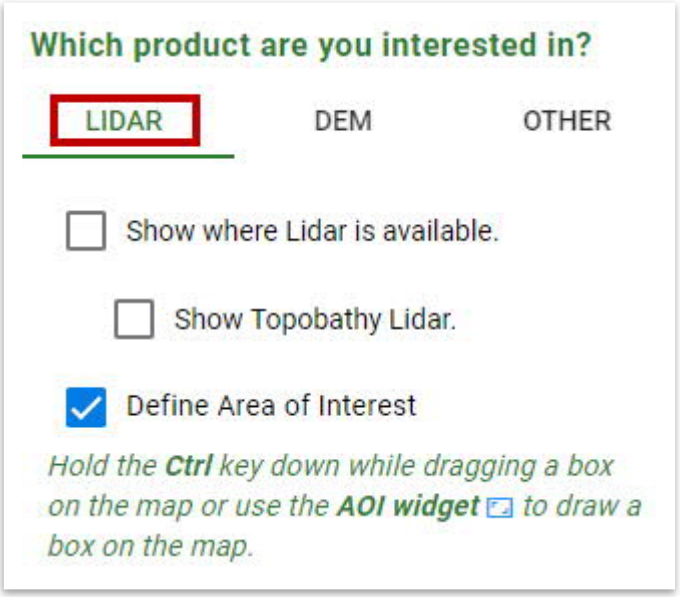
In this exercise, we will show how to download LiDAR data from the **USGS 3DEP LiDAR Explorer** website. We will use the Bowling Avenue project location in Davidson County, TN.

1. Click the following link to open the **USGS 3DEP LiDAR Explorer** website to retrieve the map tile data: https://apps.nationalmap.gov/lidar-explorer/#/. **Note:** Do not use Internet Explorer. **Google Chrome** was used for this exercise, but other browsers may be used.
2. In the upper right corner, set the search category to **Address** and then key-in **Davidson County, TN, USA** and select it from the drop-down menu. This will narrow the search location to the applicable area.

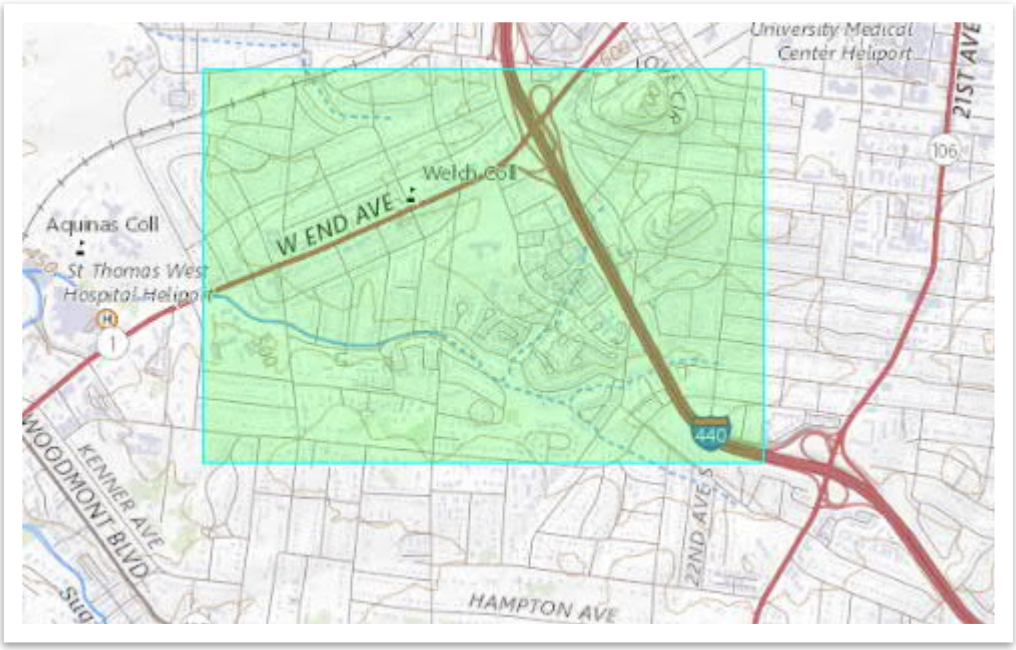


3. Zoom in to the project area (Bowling Avenue) by using your mouse wheel.

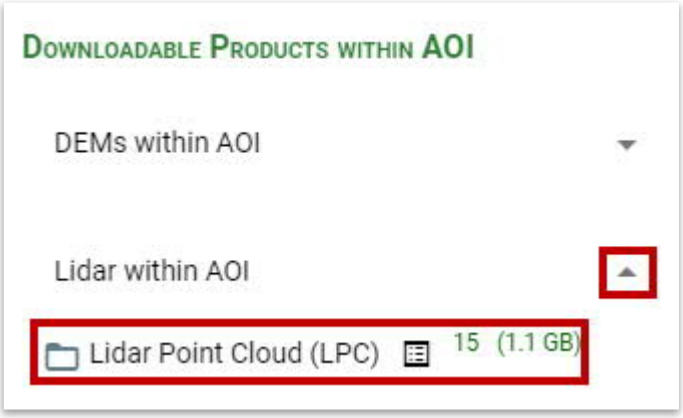
4. On the left side of the screen, choose the **LIDAR** product and select **Define Area of Interest**.



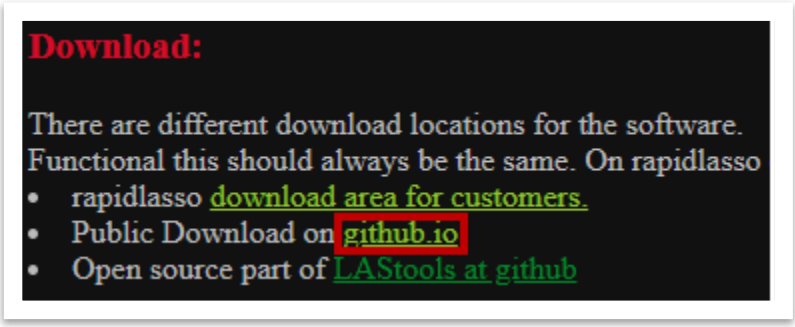
5. Follow the USGS instruction to define the area of interest. **Note:** Your area does not need to match the image exactly.



6. For the defined area of interest, expand the **Lidar within AOI** folder on the right side of the screen and click on **Lidar Point Cloud (LPC)**.

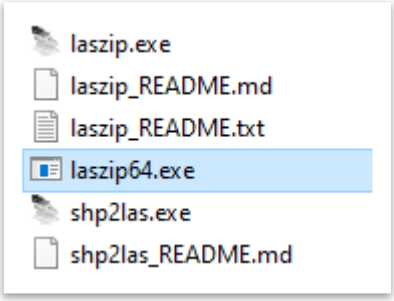


7. Within the **Filter results**, scroll down and click on the download icon next to **TN Eastern-2-16-B16-Del1_2016_1716653SE_LAS_2017**. Accept any prompts that might appear. Save the file in the following location: **C:\Program Data\Bentley\OpenRoads Designer CE 10.10\Configuration\WorkSpaces\TDOT_Standards\WorkSets\SUDA_Training\dgn**.
8. Notice the tile is saved in a **.laz** format. It needs to be converted to a **.las** format to import into ORD. Click on the following link to access the **LAStools** website. https://lastools.github.io/
9. Scroll down to the bottom under **Download** and click the **github.io** link next to **Public Download**.

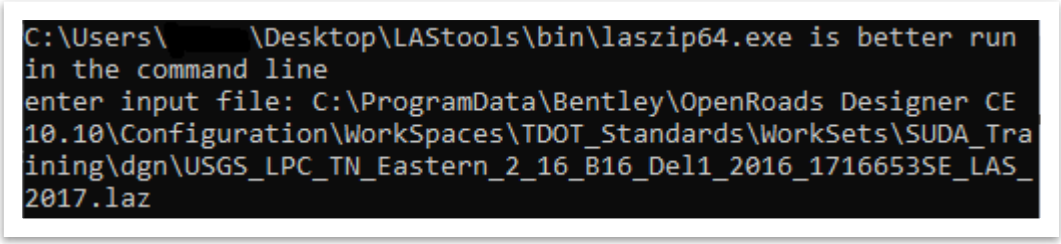


10. Next, click on **LAStools.zip** at the top of the screen.

12. Unzip the **LAStools** folder to the dgn subfolder within the **SUDA_Training** workset folder. Within the folder, browse to the **bin** subfolder and click on the **laszip64.exe** file, which will open a command prompt window.



11. Next to **enter input file**, key-in the file path of the **.laz** file and then hit **Enter**: **C:\ProgramData\Bentley\OpenRoads Designer CE 10.10\Configuration\WorkSpaces\TDOT_Standards\WorkSets\SUDA_Training\dgn\USGS_LPC_TN_Eastern_2_16_B16_Del1_2016_1716653SE_LAS_2017.laz**.



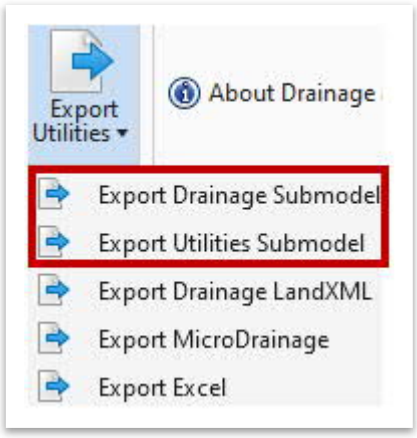
12. Lastly, next to **enter output file**, key-in the same file path but change the extension to **.las** and then hit **Enter**. Give the software a minute to process and then close the command prompt window. You should then notice that the .las file has been created in your dgn subfolder and can now use it in ORD.

Appendix G. Workflow for Copying Files with Drainage and Utilities Data

At times, the user may need to create a new drainage and/or utilities file. This could be necessary due to a corrupt file or perhaps the reference alignment breaks within the model. The best workflow for copying a file so that the model would not need to be redone would be to create a new dgn file and then import the drainage and utilities. In the current ORD version (10.10.21.04), a change was implemented in the software that keeps it from loading referenced databases by default. While this helps, manually copying a design file that contains Drainage and Utility data must still be avoided. This includes copying/pasting in Windows Explorer and similar processes in ProjectWise.

If you have any Design files that have been created in this manner, you could open them in the Civil product and do a **File >> Save As** to create a new Design file. The databases in this file will have unique **GUIDs**. You can also export data using the **Export Submodel** options (**Drainage and Utilities >> Home >> Model Import/Export >> Export Utilities**) (Figure 54), and then import into a new design file.

FIGURE 54. EXPORT DRAINAGE / UTILITIES SUBMODEL TOOLS



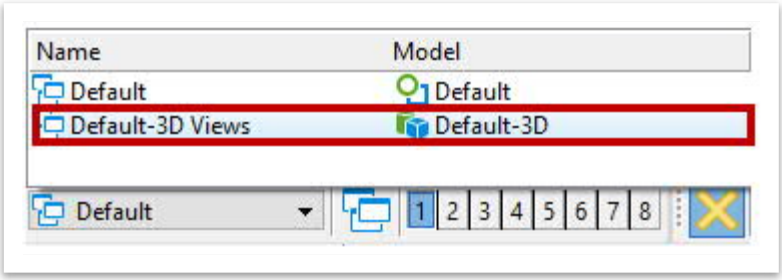
Appendix H. Utilities Conflicts Tool

The purpose of the **Utilities Conflicts** tool is to identify conflicts between the existing and proposed drainage and utility models so that design adjustments can be made, if necessary. Bentley has indicated that this tool should be enhanced in the next software release.

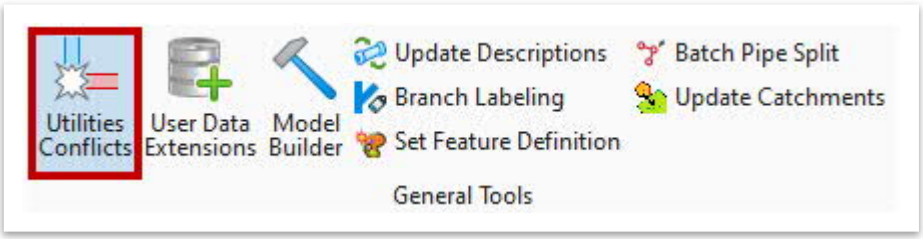
Within the tool, the user can detect physical **clashes** and **clearances** between elements. Conflict detection must be done in the **3D model** because if not, the options within the **Subsurface Utility Conflict Detection** window are disabled. This tool works with the active dgn file. Reference files cannot be used by this tool at this time, so the user will need to export their drainage and utility models (separate dgn files per the ORD Naming Convention Standards) and then import them into a single dummy file. The dummy file will not be part of the drainage submittal requirement.

The following example shows how you would utilize the tool within the dummy file on a project.

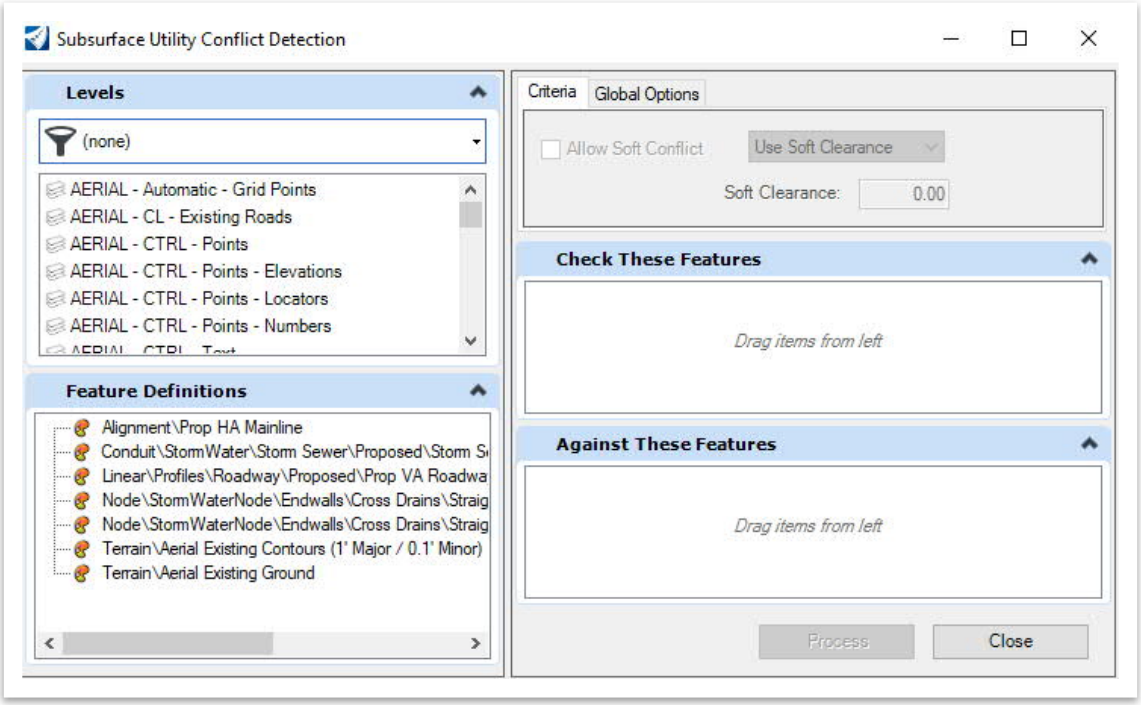
1. First, confirm that the **3D** view is active.



2. Then, open the **Utilities Conflicts** tool (**Drainage and Utilities >> Tools >> General Tools**).



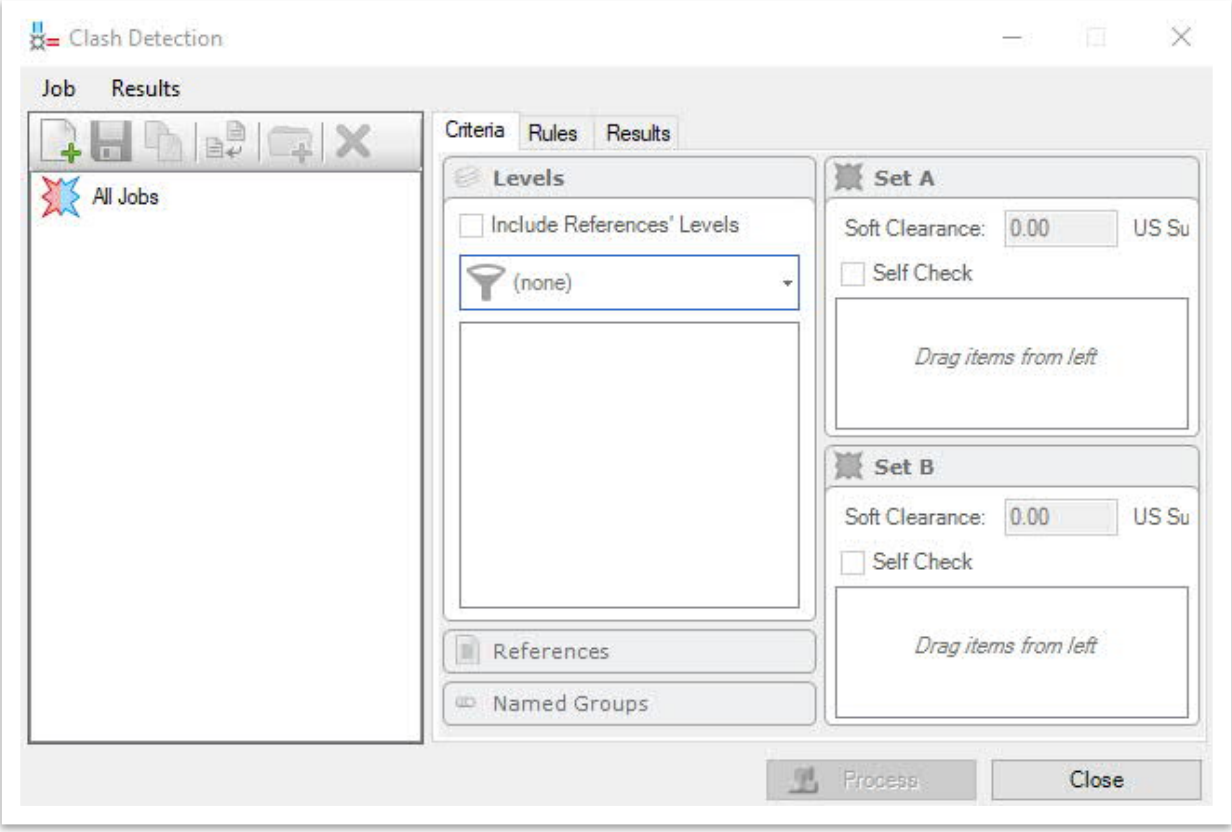
3. The **Subsurface Utility Conflict Detection** window should now display. **Note:** You may have different **Feature Definitions** shown.



4. You have the option to either search for conflicts between <u>different</u> **Feature Definitions** or within the <u>same</u> **Feature Definition** that are listed on the left side. For example, if you wanted to see conflicts between different sizes of the same pipe, you would search for conflicts within the same feature definition. Conflicts may also be analyzed based on **Levels**. <u>All detected clashes are stored as **conflict nodes** in the dgn model</u>.

Clash Detection Tool (V8i)

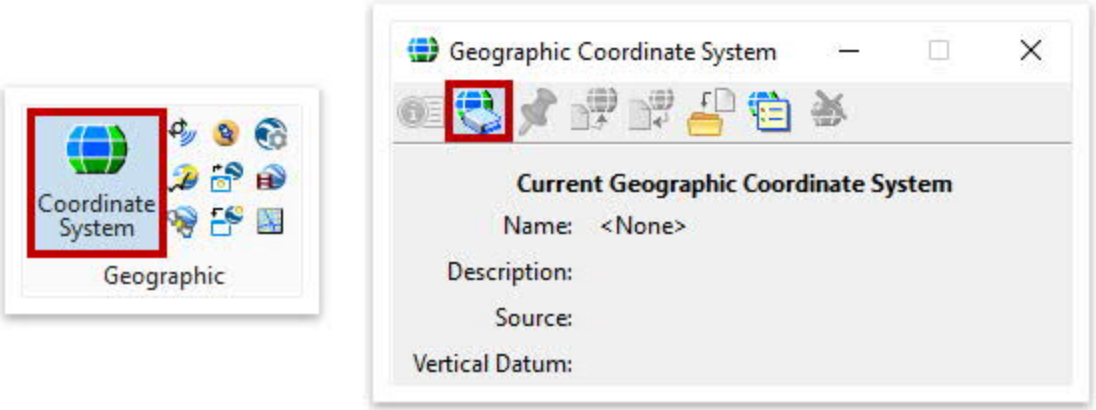
Until the enhancement for the Utilities Conflicts tool is available, the legacy V8i **Clash Detection** tool can be used to determine if there are any conflict issues by adding references under the **Criteria** tab (**Drainage and Utilities >> Collaborate >> Clash Detection**). **Note:** The 3D view must be active to see the Clash Detection tools in the ribbon.



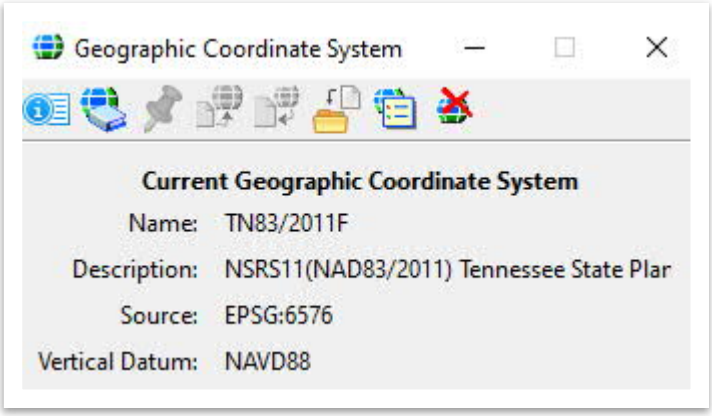
Appendix I. Bing Map Imagery

At times, a geographical background (Bing Map Imagery) might be needed to further enhance or visualize a project's location, which requires the applicable coordinate system to be set. The TDOT seed files have the coordinate system **set by default**, but if using an older file, follow the first two steps to set it.

1. Open the **Coordinate System** tool (**Drainage and Utilities >> Utilities >> Geographic**). Click the **From Library** option (second icon from the left). Notice that by default there is not a coordinate system associated with the file.



2. The correct coordinate system should be saved as a Favorite (**TN83/2011F – NSRS11 (NAD83/ 2011) Tennessee State Plane Zone, US Foot**). If not, you can browse to it here: **Library >> Projected (northing, easting, …) >> North America >> United States of America >> Tennessee**. Once selected, click **OK** and then close the **Geographic Coordinate System** dialog box.



3. Open **View Attributes**. Under **Background Map**, select **Hybrid** under **Background Map Type**. Give the software a minute to load and then notice that the imagery appears.