ORD – DU
Introduction and Workflows

April 2023

Table Of Contents

Introduction

Welcome to Drainage and Utilities (DU), the Texas Department of Transportation (TxDOT) training module for the storm sewer system design and analysis software capabilities within Bentley OpenRoads Designer (ORD). The purpose of this manual is to provide training with exercises and to serve as a technical reference for engineers and designers.

DU is a comprehensive design and analysis application that builds sewer networks that are integrated with and into 3-dimensional roadway design corridors. This course does not include instruction on performing unsteady analysis using Bentley's CivilStorm products. A tutorial for CivilStorm will be developed later.

TxDOT recently switched software from GEOPAK Drainage to DU for the design of storm sewer networks. There was a brief transition period where designers were instructed to use Power GEOPAK SS4; those who are familiar with the SS4 software should find the transition easy as the hydraulic concepts and criteria are the same. Users not familiar with SS4, should be able to design and analyze a storm sewer network using the ORD DU software following this Workflow Manual.

Objective: Introduce and serve as an instructional guide on how to design and analyze a storm sewer network using Bentley ORD DU software. The engineer/designer will need a fundamental understanding of drainage design and the Rational Method that is used throughout ORD DU.

This platform includes storm drainage design and analysis as well as utilities engineering with capabilities to include conflict detection and non-drainage attribution. ORD DU has two drainage calculation options: **Analysis** *will not change* the physical characteristics of the storm drainage; **Design** *may change* the physical characteristics.

It is the user's responsibility to adhere to the TxDOT Drainage Manual, TxDOT Hydraulics & Drainage Directives and Standards.

Drainage components have been set up to follow TxDOT standards and methods. The engineer is responsible for verifying drainage items properties.

DU uses the terrain model referenced into a DGN file to display the storm drainage and other utilities in 2D and 3D. Drainage structures and conduits are fully integrated in graphic and database functions.

Use DU to:

- Analyze the existing storm drainage for a project site, a single system or multiple.
- Analyze the existing storm drainage and upgrade for a project site as needed/needed.
- Analyze and/or design proposed storm drainage system(s) for a project site
- Analyze internal drainage structures – driveway culverts and ditches.
- Analyze conflicts between existing utilities and proposed drainage.

The designer will be working in the **2D** model and DU will create the 3D model the same way as in ORD.

Content Outline:

This Manual is a step-by-step tutorial that will guide the participant through the steps necessary to design a simple storm sewer network.

The topics in this manual include the following:

- Project Components Establishing Required Design Files
- Project Description and Design Criteria
- Drainage Area and Nodes
- Laying Out a Network
- Design and Analysis using Scenarios and Alternatives
- Computation, Review, and Profiles

- FlexTables – Creating Custom Reports

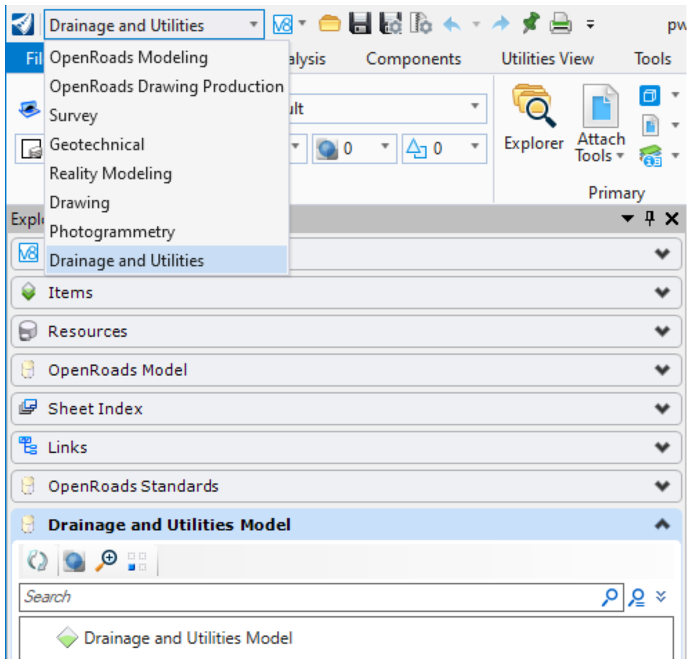
- Plan Preparation (Hydraulic Data Sheets, Drainage Area Sheets, Plan and Profile Sheets, Culvert layout sheets, lateral sheets), and annotation

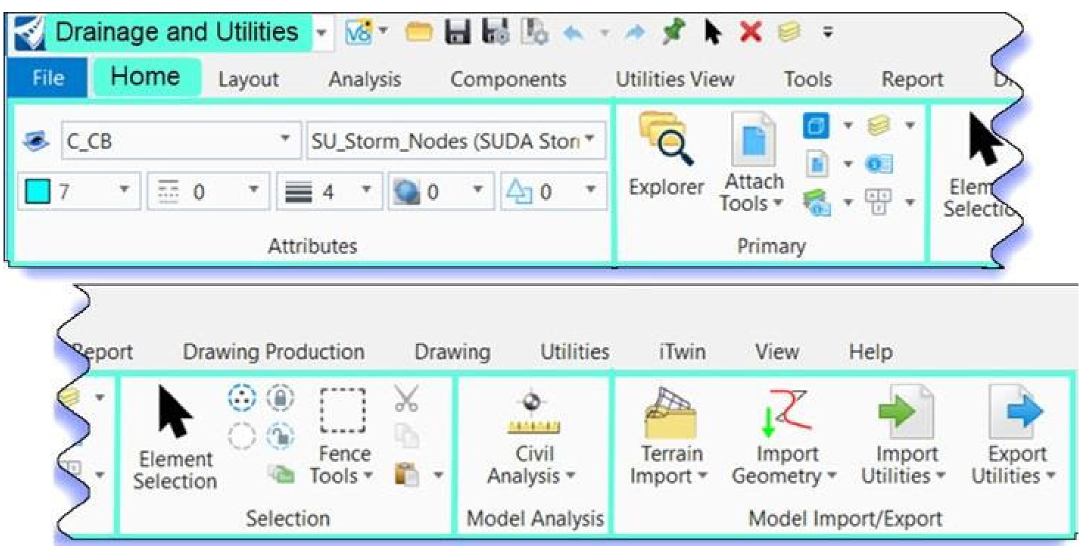
The next section gives a brief explanation of the tools and menu bars available for the user to become familiar with the workspace.

Drainage and Utilities tools

This guide will not document each tool that is available in the ORD DU interface. See the Bentley Online Help for commands not detailed in this document. The Tools below are commonly used:

Navigate to the Drainage and Utilities Ribbon:

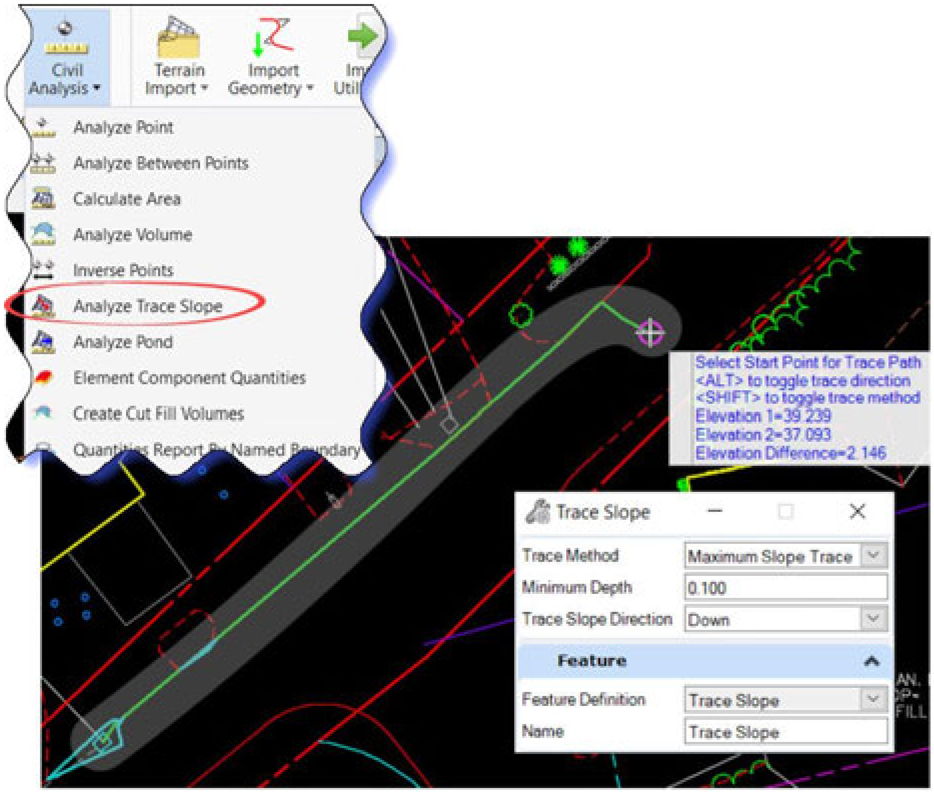




1. Home tab

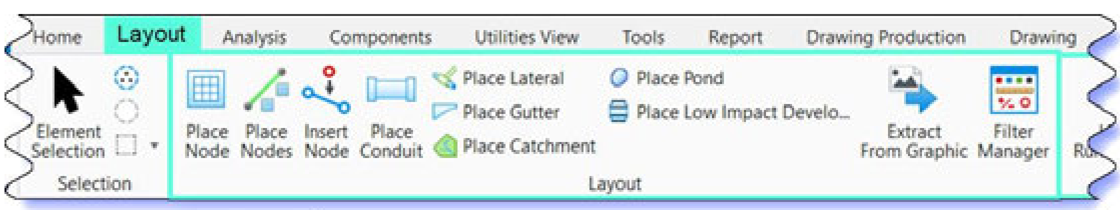
- **Attributes** – with most workflows, it shows all attributes that are active
- **Primary** – opens the explorer, properties, attach tools for references and raster files, level manager, level displays (common MicroStation tools)
- **Selection** – houses the Element Selection tool, Select All, Select None, and Fence tools
- **Model Analysis** – Civil Analysis gives the user the option to analyze myriad items.

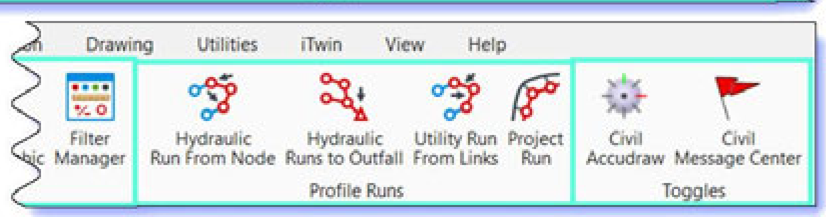
A useful tool for the drainage design is the **Analyze Trace Slope** - The Trace Slope tool shows the down/up slope direction of the slope at any one point within the active terrain. The Trace Slope tool should be used in its separate file and referenced into the drainage file as needed.



- **Model Import/Export** - Various import and export options.
  - **Terrain Import** – Creates a terrain model from File, ASCII File, Elements, and Point Cloud etc., see Bentley Online help for more Terrain options.
  - **Import Geometry** from various file types.
  - **Import Utilities** – Import drainage data from various other software, e.g., InRoads and Export Utilities to various other software, see Bentley Online help.
    Terrain import, geometry import or import utilities should be done in a separate file and referenced into the drainage file as needed.
  - **Export Utilities** – Export drainage data to various other software, see Bentley Online help.

2. Layout tab

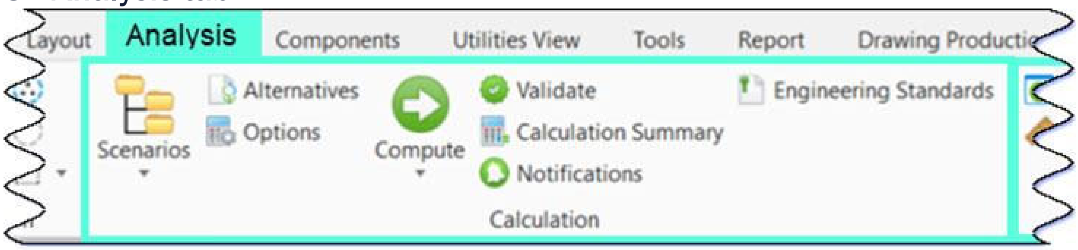


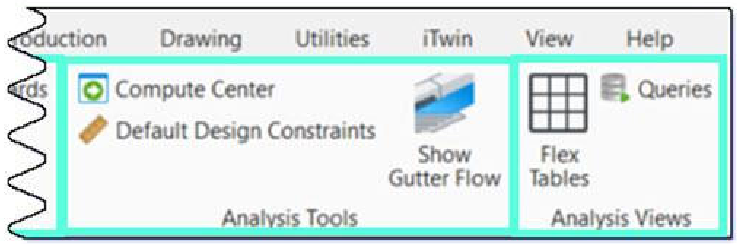


The **Layout** tab allow the user to build hydraulic and utility networks by placing utility structures, conduits, culverts, ditches, drainage areas, gutters, etc. The utility structures are used with clash detection tool. The utilities can be below the surface, on the surface or above the surface.

- **Layout** - This is the main tool to place utilities, such as drainage structures and pipes. This will be explained in more detail under "Drainage System Layout."
- **Profile Runs** – These are profiles along a series of structures. Different profiles can be created from the Start Node or the Outfall, and Links can be added to a profile run.
- **Toggles** – Will open/activate Civil Accudraw and the Civil Message Center.

3. Analysis tab

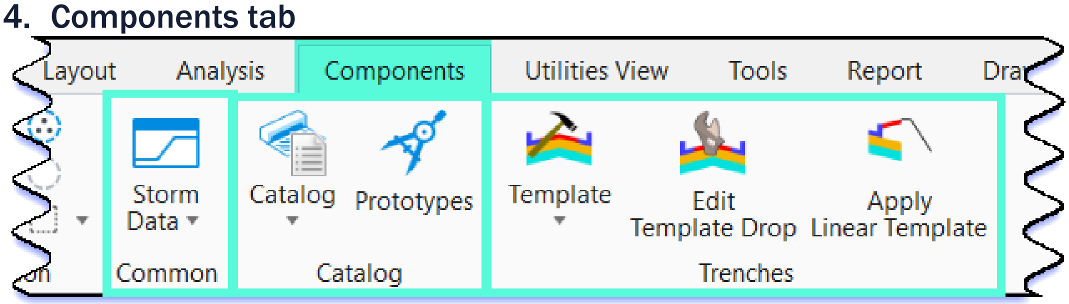




The **Analysis** tab sets calculation parameters, standards, constraints, and scenarios for drainage items and will be explained in more detail under Drainage Computation. The Analysis tab holds the tools to analyze and design hydraulic systems (storm sewer, ditches, culverts)

- **Calculation** - This is the center for drainage calculations.
- **Analysis Tools** - Holds the Compute Center (one toolbox for access to calculations). Default Design Constraints are set to TxDOT standards and should be reviewed for project specific constraints (e.g., max. spread, conduit slope). Show Gutter Flow will show the path of any overflows from catch basins.
- **Analysis Views - Houses the flex tables** – TxDOT-specific flex tables for hydraulic analysis have been set up. Queries can be used to report on a specific item.

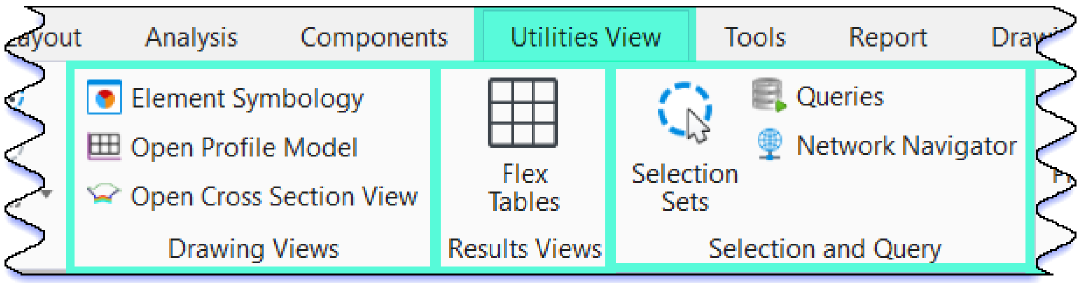
4. Components tab

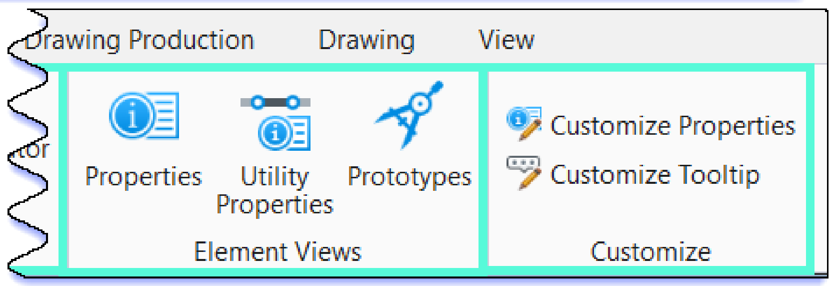


The **Components** tab is used to show project-specific storm data, drainage properties, and settings. It will be explained in more detail under Section 4 - Drainage and Computation.

- **Common** - Create – import Storm Data, select Global Storm Events.
- **Catalog** - Access to engineering libraries, inlet – gutter – conduit catalogs, culvert inlet coefficients, and rainfall curves from csv files. Prototypes for drainage items are set to TxDOT Standards and can be updated to project-specific data if needed.
- **Trenches** – Access to template tools, edit template drop, and apply linear template for trenches used in the design.

5. Utilities View tab

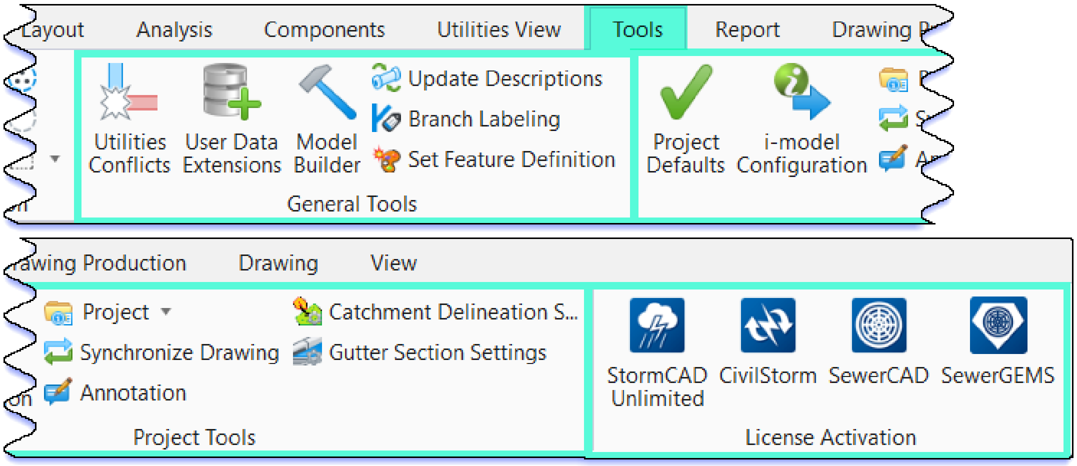




The **Utilities View** tools open views of various drainage properties.

- **Drawing Views** - Element symbology allows control of how elements and associated label are displayed; open profile model generates a view presenting a desired feature in profile; open cross-section view creates a dynamic cross-section of a selection item. These are the ORD Modeling tools.
- **Results Views - Houses the Flex Tables** – TxDOT-specific Flex Tables for hydraulic analysis have been set up, and the user can view input data and results for all elements of a specific type in a tabular format.
- **Selection and Query** - Create, edit, and navigate selection sets; display all queries in the current hydraulic model; open the network navigator.
- **Element Views** - Properties displays a dialog box containing selected elements' associated properties. Utility Properties displays the subsurface utilities and hydraulic analysis of a selected element. Prototypes displays and allows edit to default values for elements in a network. Prototypes have been set up to TxDOT Standards. The engineer needs to verify the items in the project.
- **Customize** - Customize Properties allows customization and changes to default user interface. Customize Tooltip allows customization of tooltips that appear when hovering over an element.

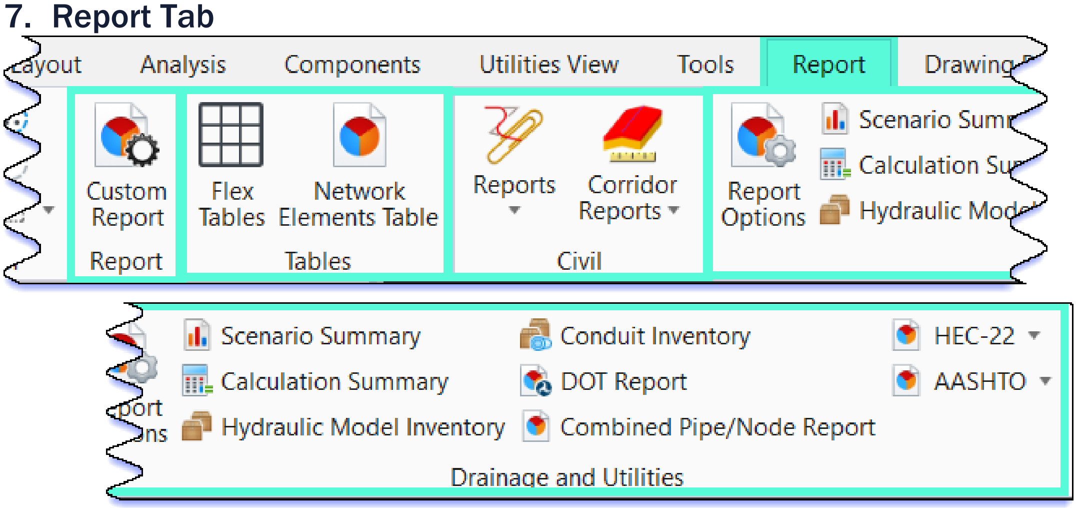
6. Tools Tab



The **Tools tab** is divided into general tools and project tools. including “Utilities Conflict” used for conflict detection.

- **Tools** - Utilities conflicts allow detection of physical clashes and clearances between elements. User data extensions allow adding data fields to the hydraulic model toolbox. Model builder allows existing GIS assets to construct a new model or update an existing model. Update descriptions of conduits in the model. Branch labeling sets the labels of conduits. Set feature definition allows setting/locking a feature definition.
- **Project Tools** - Houses project defaults, i-model configuration to export drainage calculations, project properties, synchronizes drawing, and annotation tool to label the drainage items.
- License Activation - **StormCAD Unlimited**. for systems that have greater than 100 nodes. **CivilStorm** is used for unsteady flow analysis.

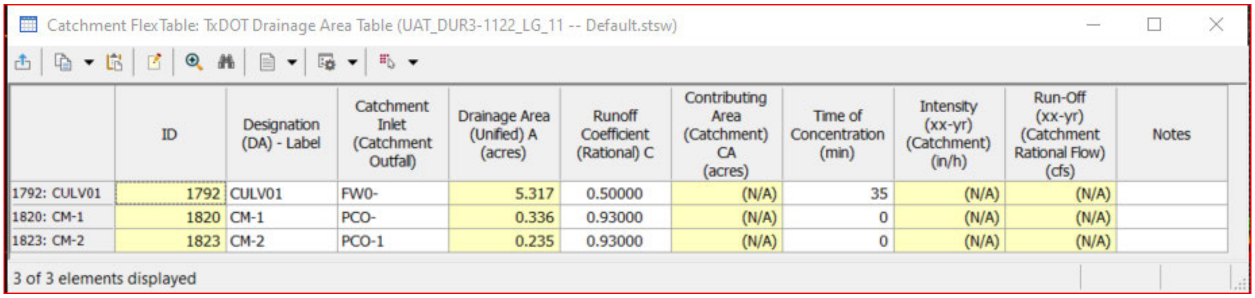
7. Report Tab



The Report tab gives the user fast access to various reports for Tables, Civil and Drainage and Utilities.

- **Report** - Custom Report allows the user to assemble a wide variety of model input, results, graphs, etc., in a customized report.

- **Tables** - Access to all Flex Tables and Network Elements Table for the utility project.



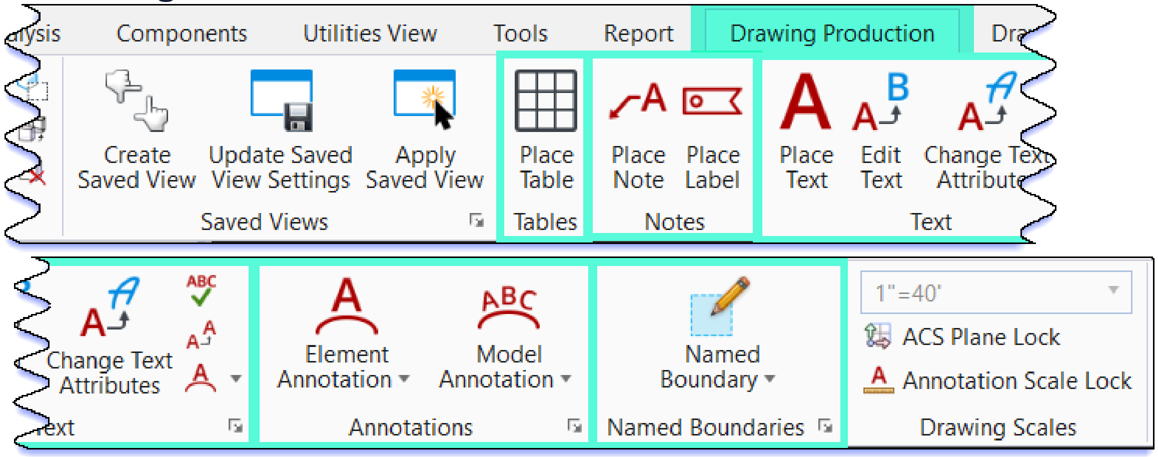
Catchment FlexTable: TxDOT Drainage Area Table (UAT_DUR3-1122_LG_11 -- Default.stsw)

	ID	Designation (DA) - Label	Catchment Inlet (Catchment Outfall)	Drainage Area (Unified) A (acres)	Runoff Coefficient (Rational) C	Contributing Area (Catchment) CA (acres)	Time of Concentration (min)	Intensity (xx-yr) (Catchment) (in/h)	Run-Off (xx-yr) (Catchment Rational Flow) (cfs)	Notes
1792: CULV01	1792	CULV01	FW0-	5.317	0.50000	(N/A)	35	(N/A)	(N/A)	
1820: CM-1	1820	CM-1	PCO-	0.336	0.93000	(N/A)	0	(N/A)	(N/A)	
1823: CM-2	1823	CM-2	PCO-1	0.235	0.93000	(N/A)	0	(N/A)	(N/A)	

3 of 3 elements displayed

- **Civil** - Access to civil reports such as horizontal geometry report and corridor reports.
- **Drainage and Utilities** - Access to various reports and summaries for the drainage design.

8. Drawing Production Tab



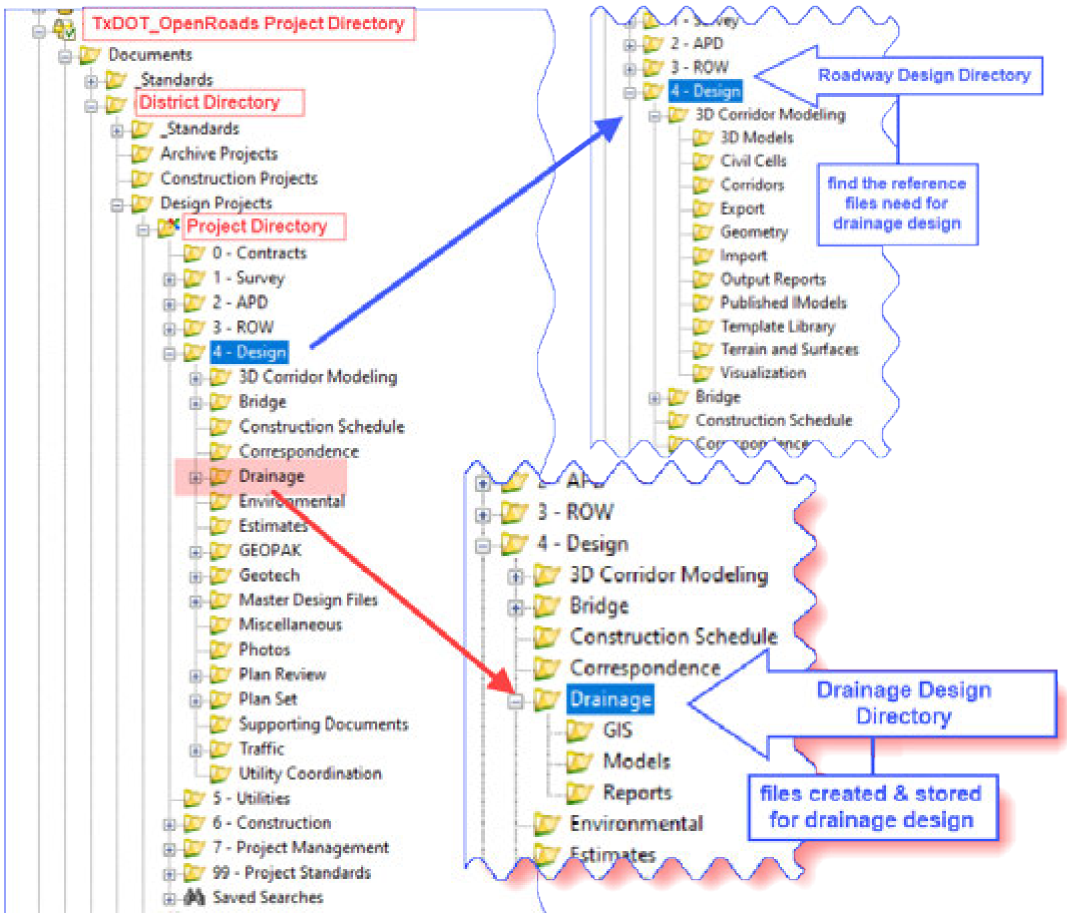
The **Drawing Production** tab is divided into tables, notes, text, annotation, and named boundaries. Explanations are found in detail under the **Plans production sections** of this workflow manual.

- Drawing Production tools enable the user to place notes and labels; place and edit text; annotate an individual element or an entire model; and enable sheet production through the Named Boundary tool.

Project Setup

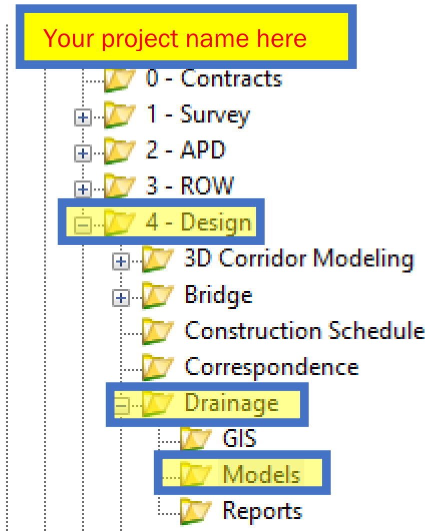
TxDOT recommends that all design projects are completed and saved in Projectwise. Let's get started with a project. First step is to set up the drainage designs files in the right project folders. This guide assumes the TxDOT Design PM has set up a project folder for the project. All TxDOT projects will have a similar folder structure as the one shown below.

1. Folder structure and reference files location.



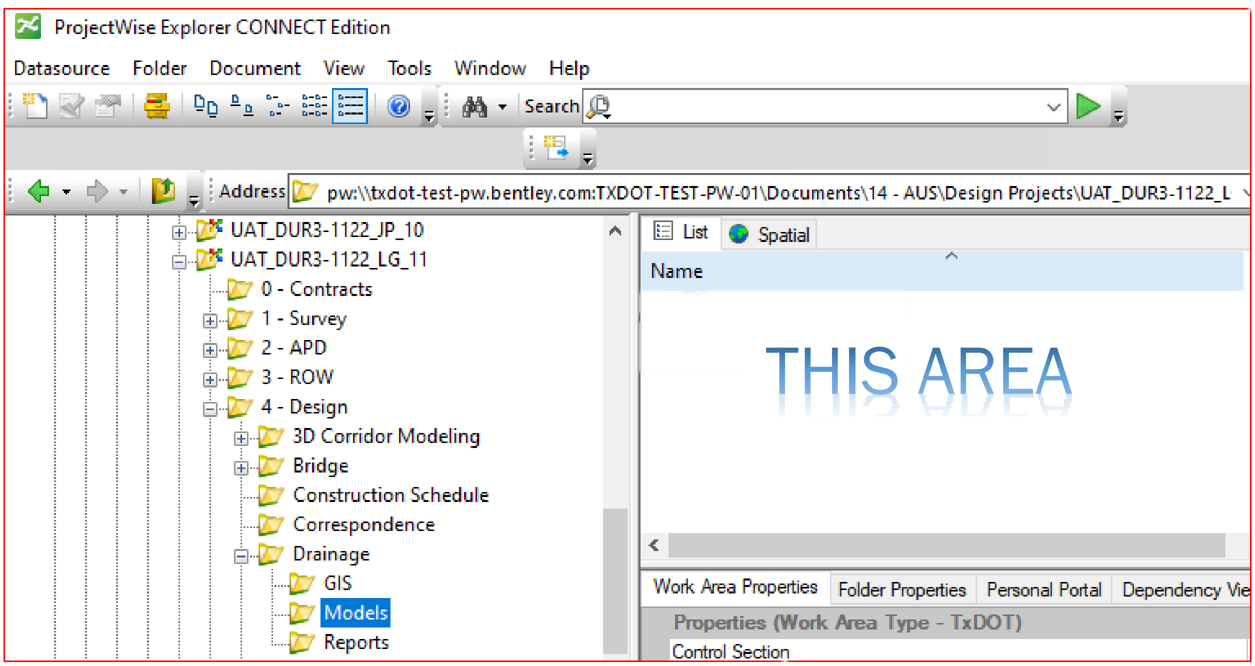
2. Create a new Drainage Design

In this case we will be creating a design dgn file in the “Drainage” folder.

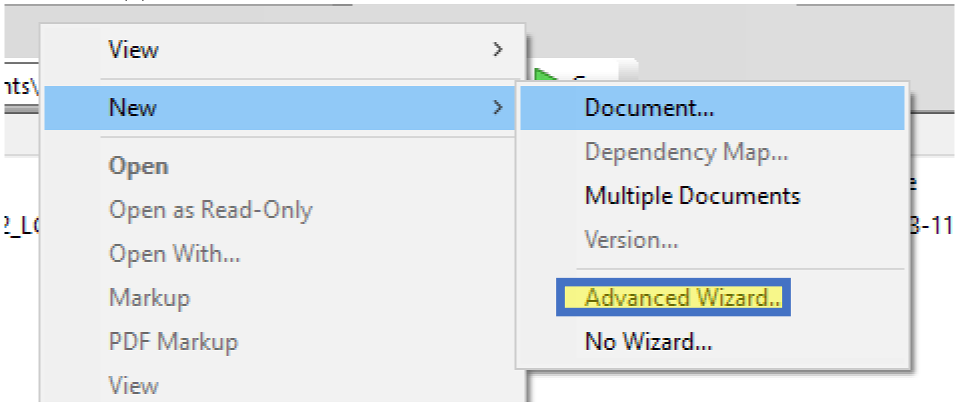


(a) click on the folder you wish to create the file in. For our example it is the models Folder.

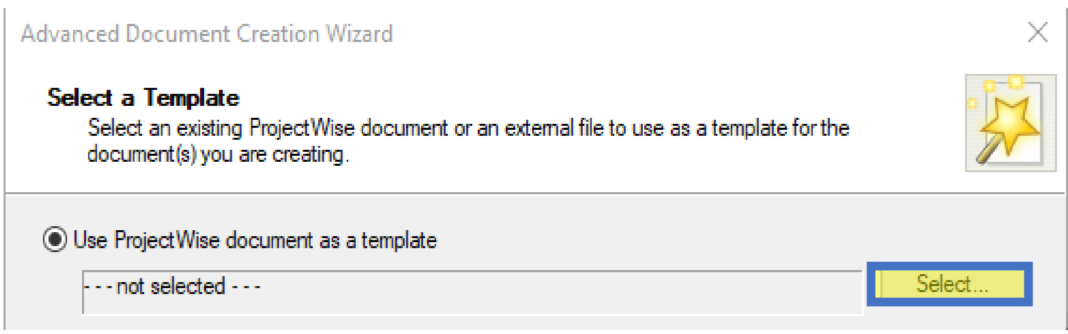
(b) Right click on the area to the right of te folder structure



(c) Select new and Advanced Wizard:

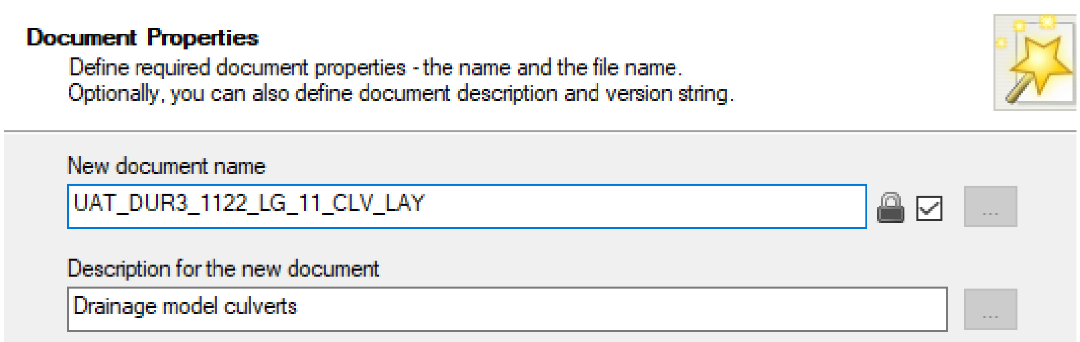


- Click next.
- Select your target folder > Drainage/Model.
- Click next.
- Select" Use ProjectWise document as a template" and click on Select.



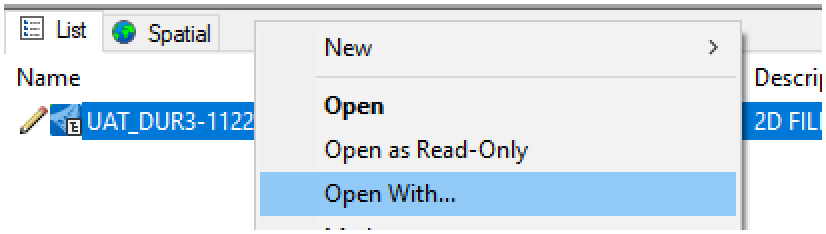
Browse to the location of your seed files:
pw:\\your district folder 01\Documents_Standards\Configuration\Organization-Civil\TXDOT\Seed\
Select the right seed for your file type; in this case, TxDOT_DesignDeed2d.dgn.

- Click Next.
- Select your new document name and description.



- Click Next on the next few screens.
- Right click on the dgn and select open with

- Open your newly created file with ORD.



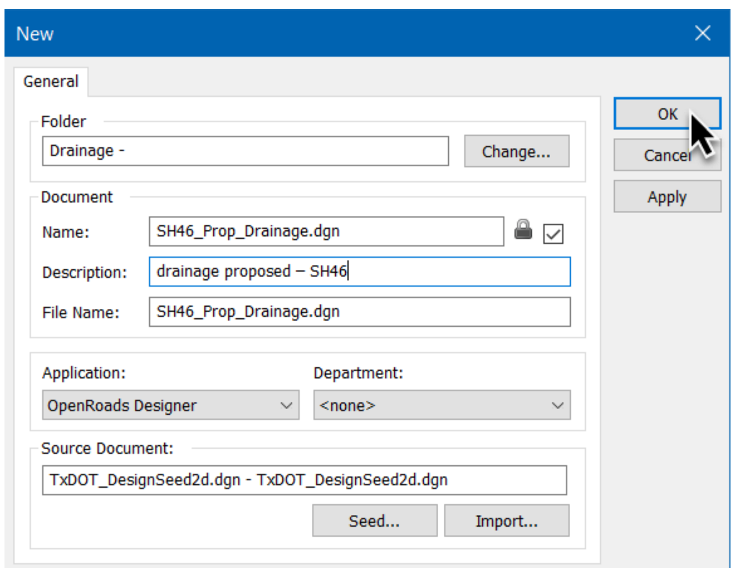
Follow the same procedure to create more files from PW. If you have a file already open in ORD, you can follow the steps below:

(a) Select your working folder:
1) In ORD, navigate to File > New > Select No Wizard > OK.
2) Select Change... > navigate to the Project Folder > 4 - Design > Drainage > Select OK.

(b) Under Document fill in the name and description fields:
3) Name: drainage file name, for this example: SH46_Prop_Drainage.
4) Check the lock box to the right so the name and file name are the same.
5) Description: drainage proposed – SH46.

(c) Select the appropriate seed.
6) Under Source Document: select (Select Seed > Select TxDOT_DesignSeed2d.dgn > Select Open > Select Yes on the Ready Only Dialog).

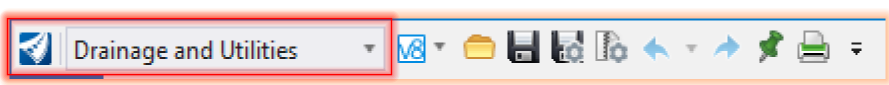
The information in your window should look as below:



(d) Select OK to create the new file in ORD. Check In the Starter file.

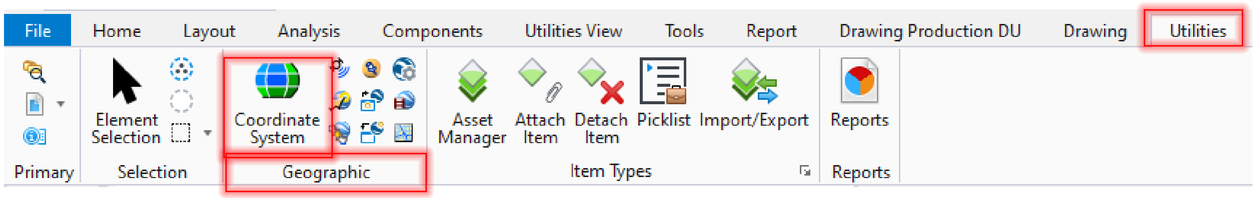
3. Set up your geographic coordinate system

Once a design file (dgn) has been set up (see steps above), verify your project location and survey controls. ***You must select the proper state plane coordinates prior to attaching design files.***



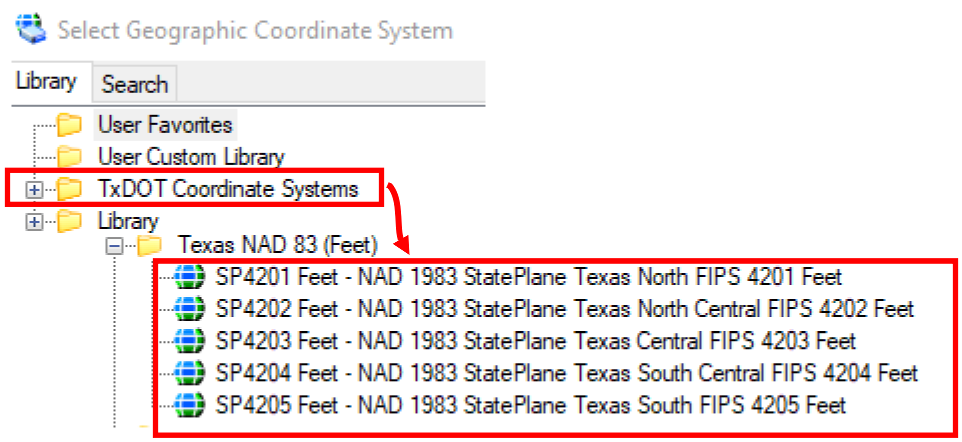
Navigate to the Drainage and Utilities workflow >utilities ribbon.

Under Geographic group, select Coordinate System.



From the Geographic Coordinate System window, select>From Library.

From Select Geographic Coordinate System, select TxDOT Coordinate Systems.



Select the right state plane coordinates. (You will find this information in your survey file.)

Survey files usually are provided in grid coordinates. To adjust to surface, you will need to apply a conversion factor. You can do this in a number of ways:

First Method

1) When selecting your coordinate system instead of using from library, highlighted below, you can select from file, circled in red:

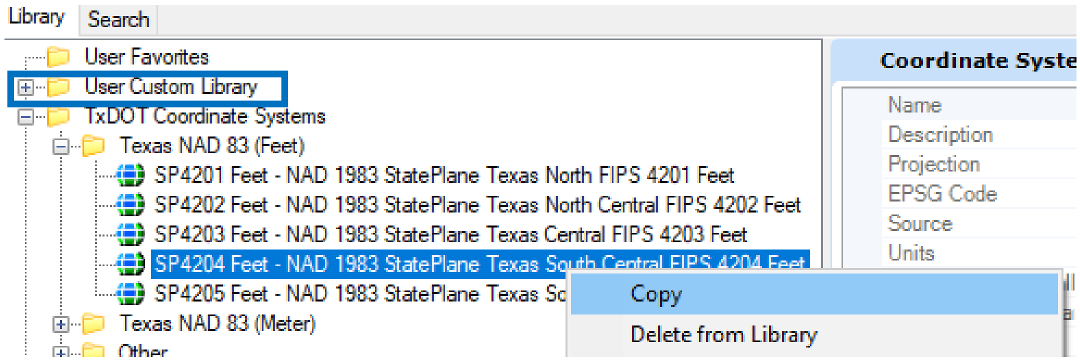
2) The survey terrain file that is provided by the surveyor, should have the coordinate system with adjustment.
3) The file will usually be found on the Projectwise folder assigned to the project pw:\\server\01\Documents\your district\Design Projects\your project number\1 - Survey\Terrain Survey\survey file for your project.dgn
4) This will assign the correct coordinate system to your file.

Second Method:

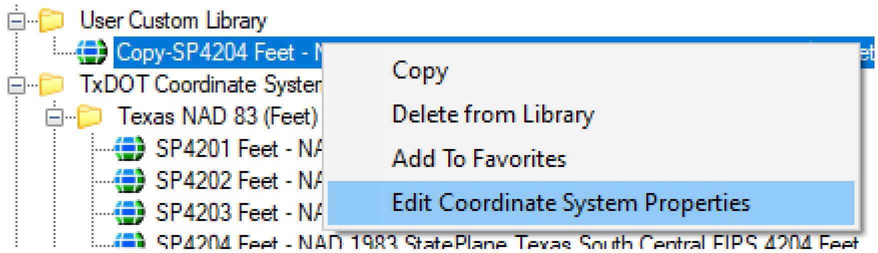
If a terrain file is not available with the coordinates adjusted, and you know your conversion factors, you can adjust one of the existing ones to use as your coordinate system:

1) From the geographic coordinate system window, select>From Library.

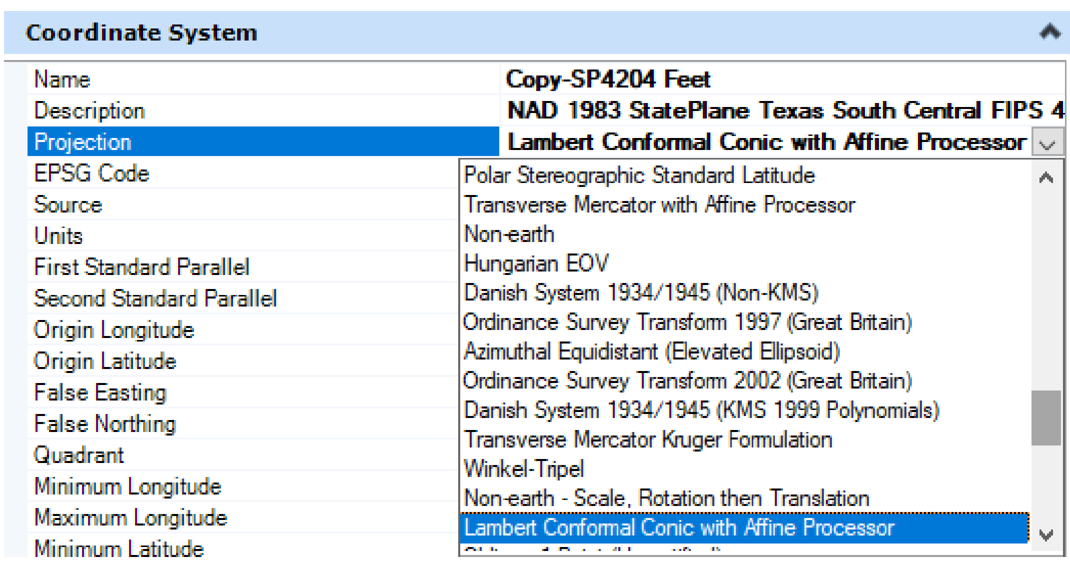
Right click in the coordinate system, select copy, and paste it in the user custom library folder
Open the User custom library folder, right click on the coordinate system created, and select Edit



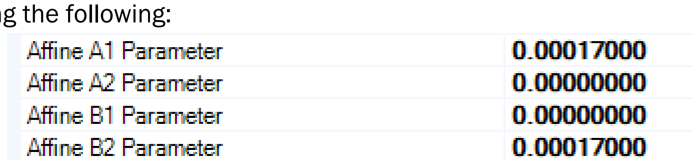
Coordinate System Properties:



A new window will pop up:
Change the projection to Lambert Conformal Conic with Affine Processor



Update the Affine A1 and B2 Parameters to the conversion factor for your county. In this example, we are using the following:

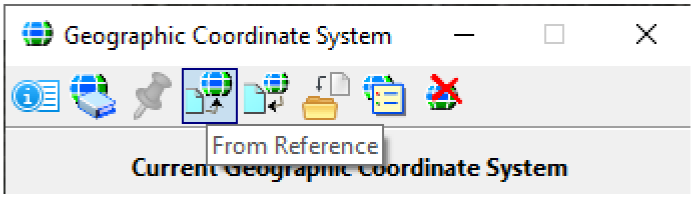


Click OK when done. Select your user custom file and click OK. Your newly created adjusted coordinate system is now applied to your file.

<u>Third Method:</u>

you can use an attachment reference to assign the coordinate system to your file. This method implies that you have already attached a reference file that the coordinate system you will use for you project attach. If you do not know how to attach a reference file to a DGN we will briefly explain it in the following step. "**3. Attach reference files.**" Once your reference file is attached do the following:

2) Select From Reference from your Geographic Coordinate System window as shown below.



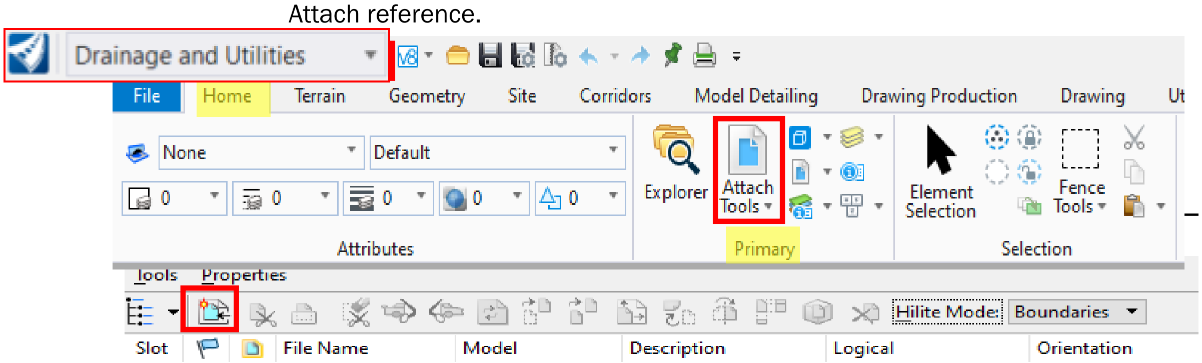
Select the reference that has the coordinate system you want to use. The coordinate system from that file will be applied to the active file.

4. Attach reference files

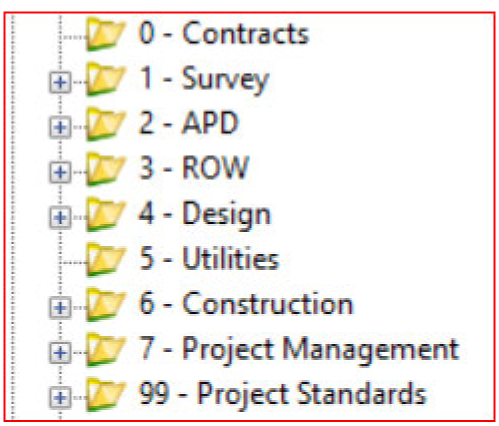
You will need to attach all survey, roadway geometry (alignment and planimetric files), existing and proposed terrains, proposed roadway corridors, and bridges. All files should be attached with No Nesting.

To attach the files:

(a) Navigate to Drainage and Utilities > Home ribbon > Primary Group > Attach Tools > Attach reference.



Under your project folder you will find the following structure:

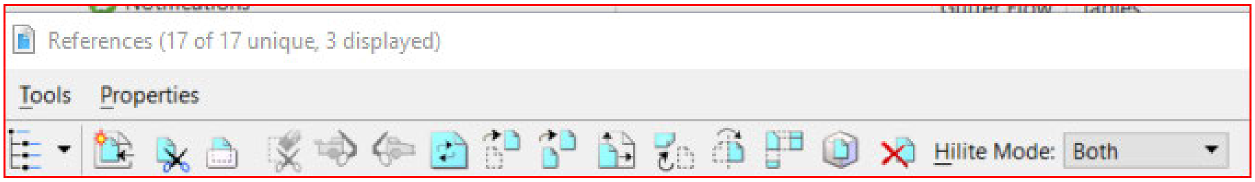


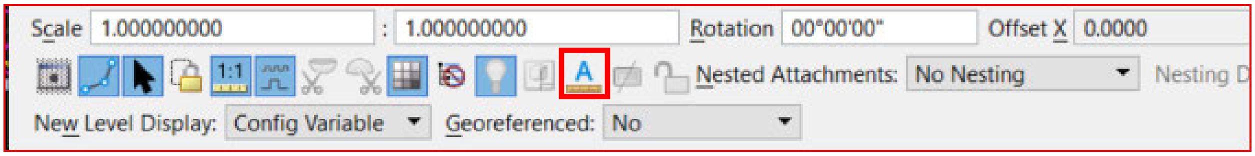
Here is a list of reference fles you will need and the folder where you will typically find them:

Folder 1- survey> Terrain survey>

- From this folder you will attach your planimetrics and Existing terrain files. Make sure your annotation scale is turned off for this files. You will see the option at the bottom of your reference window:

All other references can keep the annotation scale on.





Folder 4 - Design\3D Corridor Modeling\Terrain and Surfaces\

- Attach all proposed terrain files associated with your project.

Folder 4 - Design\3D Corridor Modeling\Geometry\

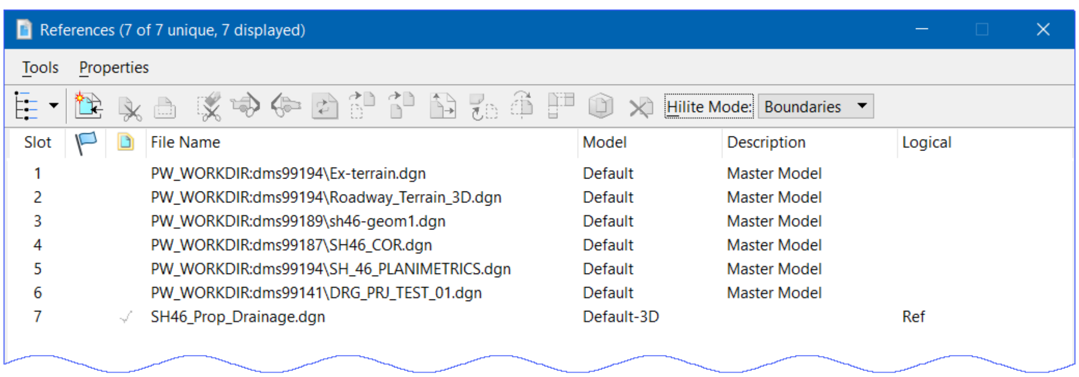
- Geometry files will contain your roadway alignment and Profile information. Attach all geometry files from this folder.

Folder 4 - Design\3D Corridor Modeling\Corridors\

- This folder contains the roadway 3D corridor files. Attach all corridors files from this folder

Folder 4 - Design\Bridge\

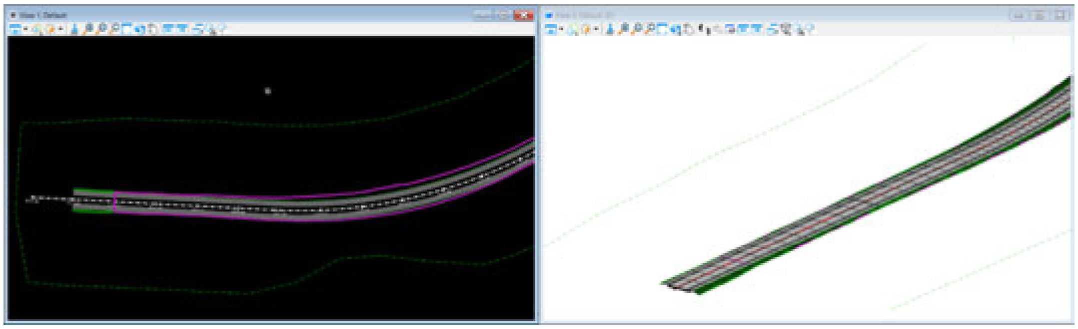
- If your project has any bridges this is where you will find the bridge files to attach.



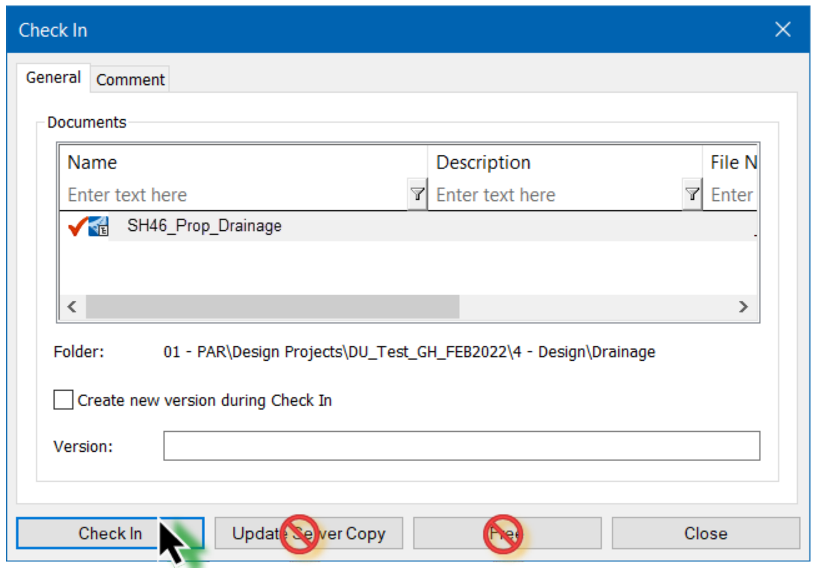
Fit view

Set the Terrain Active. On the prior step we attached both existing and proposed terrains. A proposed terrain must be created/merged from your proposed corridor and existing terrain. This is typically completed by the roadway designer. Select a terrain (existing or proposed) to set active. For proposed drainage system design, you will typically need a proposed terrain. Also, as we will see later, we will need a proposed terrain for automated DA delineation.

By setting the Terrain Active, ORD will create a Default 3D Model. See in the reference dialog, <u>DO NOT DELETE</u> the referenced 3D model, this will corrupt your file. As shown below the 2D model is shown on the left and the 3D model on the right. There are several preset view options you can access by clicking in the active view. Right click in the active view, a context menu will open click on the upper option "**View Control**" Several view options are available. The option shown below is "**2 views Plan and 3D**"



Before we move on to our next section here are some important recommendations:

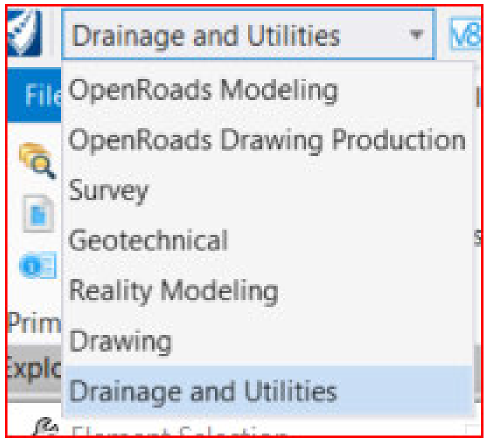


- <u>Do Your Design Following/Using the Drainage and Utility Tools.</u>
- <u>Save your work often and/or make up a back-up file. Use File > Save As within ORD.</u>
- <u>Free or update server copy may cause your drainage file to lose connection to the database and corrupt your file. ALWAYS CHECK-IN THE FILE(S) TO PROJECTWISE. DO NOT USE FREE OR UPDATE SERVER COPY.</u>

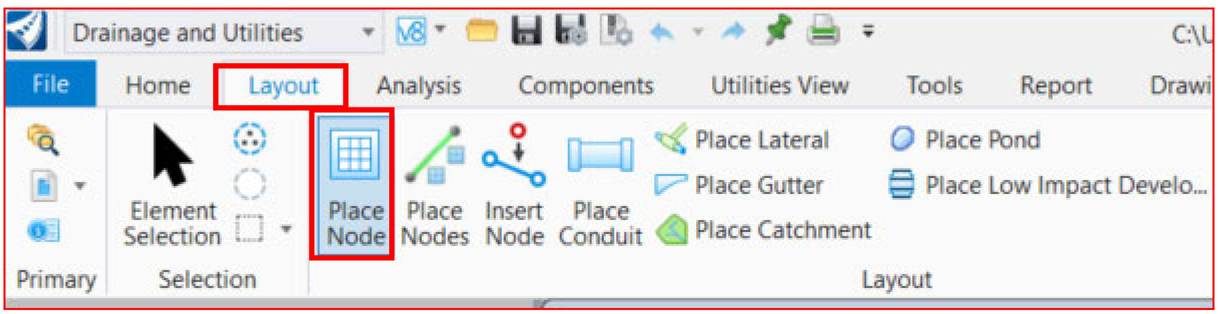
Drainage system layout

In this section we will get started by building an example storm sewer network.

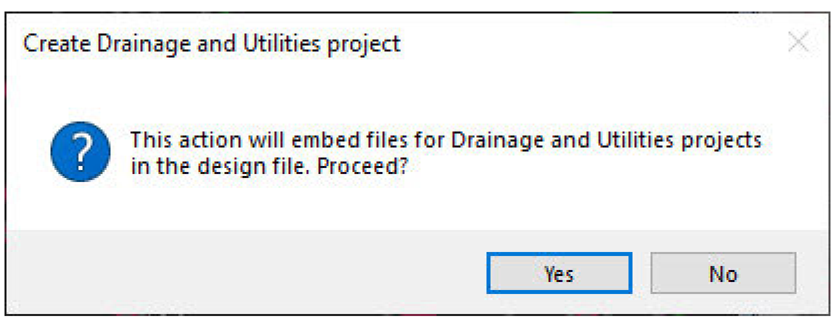
(a) In the Drainage and Utilities Workflow, select the Layout Tab.



(b) from the Layout group, select the Place Node tool.



(c) The following question dialog will pop up.



(d) Click Yes. This will create your utility model. The utility model is part of the active DGN file. It is recommended that you create this in its own file, for example, the Drainage design file. Do not use your corridor or geometry files to create your utility models. Once this is created, **IT CANNOT BE UNDONE.**

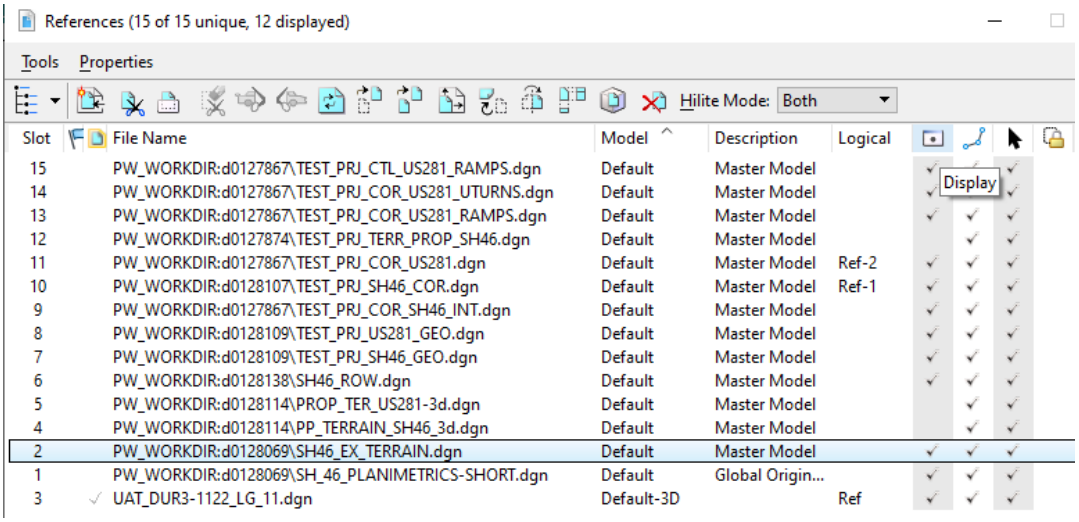
There are a number of drainage settings and software settings that automatically are added to the design file when the Utility Model is created. Some examples of information added would be:

- Rainfall Runoff Data (Storm Events). the Texas ebd curves are embedded in the workspace for selection).
- Hydraulic information used in calculations (Runoff Methods, Friction Loss Methods, Pipe Flow Calculation Methods, etc.)
- Design constraints (Slope, Velocity, Cover).

- Pipe, Inlet, Culvert, etc. catalogs available for placement and calculations.

Note: OpenRoads is 3D, but the default location for geometry and structures is the 2D Model. Do not place structures in the 3D Model. Place all drainage structures in the 2D Model. OpenRoads will manage the 3D portion automatically.

1. Place an Outfall



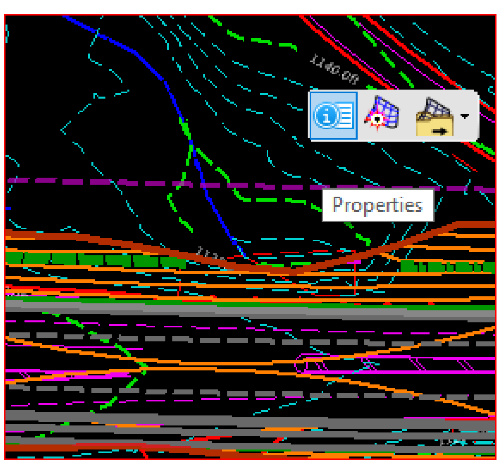
To start the design of a drainage system an outfall location must be decided. The outfall must meet certain design criteria to be practical. Your network cannot cross drainage basin boundaries, so your outfall must be in an area that is inside the overall basin and must ensure there is enough drop to meet the TxDOTs minimum 3fps design constraint.

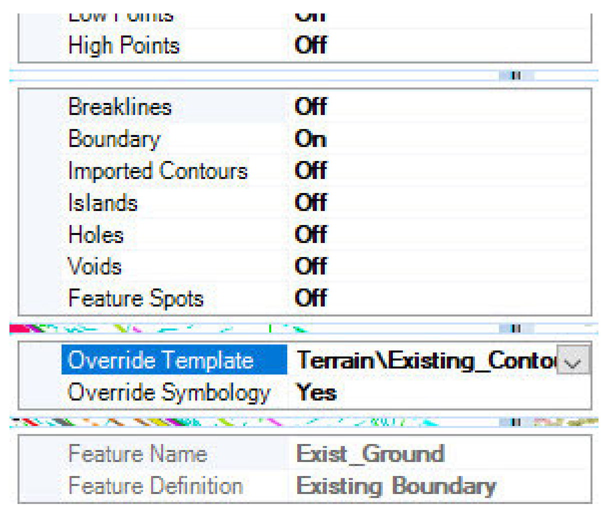
In this section, we will place an outfall for our system.

(a) Turn on the Existing Terrain Contours if not already on. Select the **Reference File** Tool.

(b) Highlight the **Existing Terrain DGN** and toggle on the **Display**.

(c) Using the **Element Selection** tool, select the *Existing Ground* terrain boundary, then **Hover** to access the **Element Properties.**

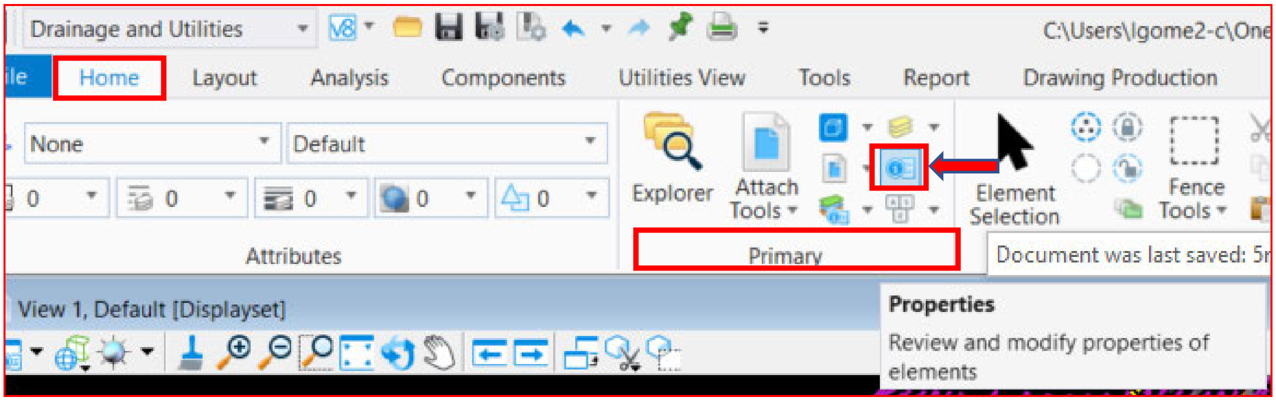




Change the *Override Symbology* from *No* to Yes and set the *Override Template* to Terrain\Existing_Contours.

The steps above sow you how to access the quick properties box. Alternatively, if you have the Main properties window open, you can access the main properties just by selecting an element. To open the properties windows and Dock it to the left, do the following:

- From the home tab primary tools select properties:

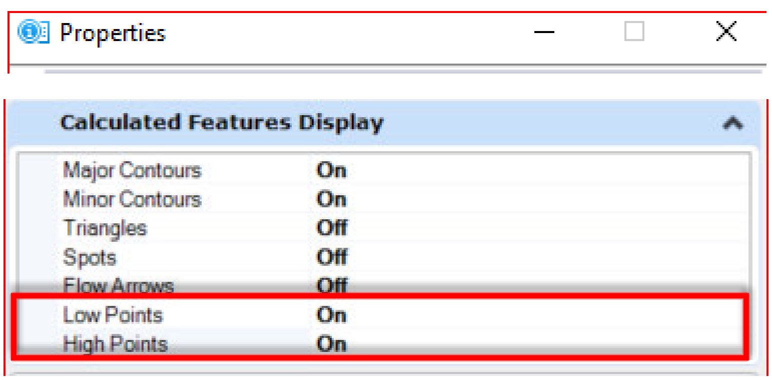


- Select the element. In this case the terrain boundary. The properties window will open.
- To dock it drag it to the left of the screen.

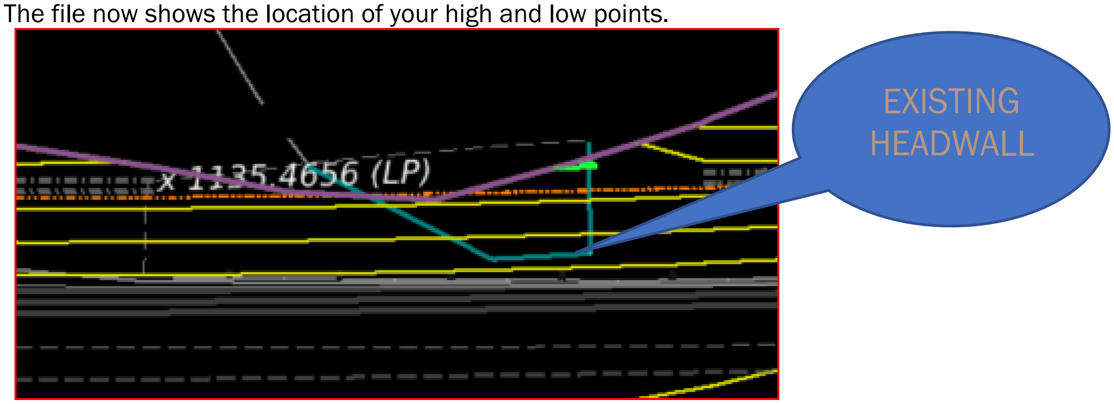
Now you will see element properties on the left just by selecting an item.

(d) Outfalls are usually located at or near low points. We can use the terrain element properties to determine and mark the location of this points.

(e) From the **element properties** window navigate to the **calculated features Display** and togle low points and high points on as shown below:

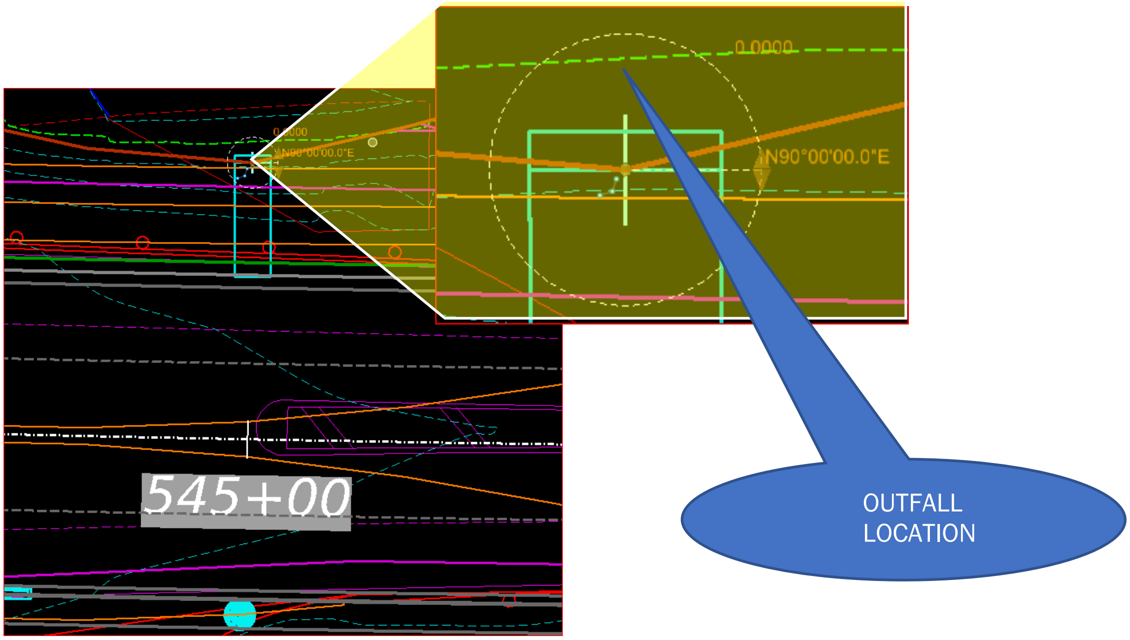


(f) The file now shows the location of your high and low points.



This, in combination with topo features that show the location of major crossings or culverts, will allow you to select an appropiate outfall with ease. In this case our low point is near an existing culvert crossing. We will use it as our outfall

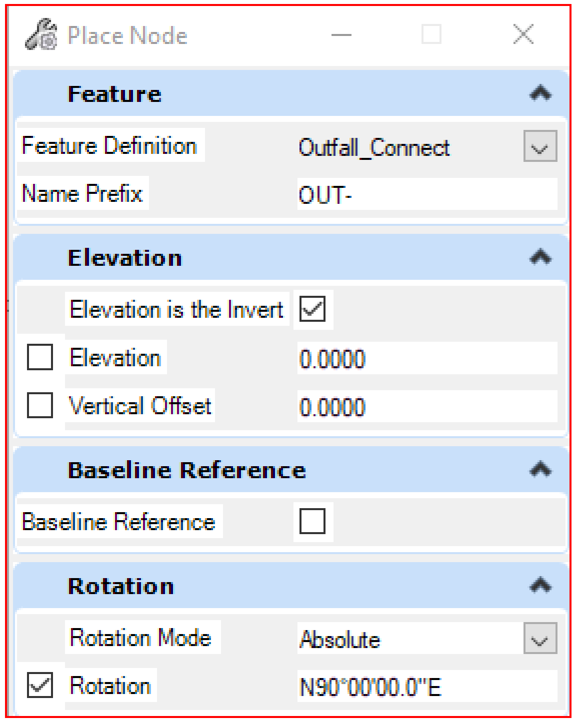
(g) Zoom to the outfall location, as shown in the image.



In this case, we will place the outlet on the North side of the roadway at the construction limit tie slope shown by the red line. The TxDOT workspace uses a generic outfall represented by an "X" that will be used as your outfall for calculation purposes. Your outfall conduit should tie into this element.

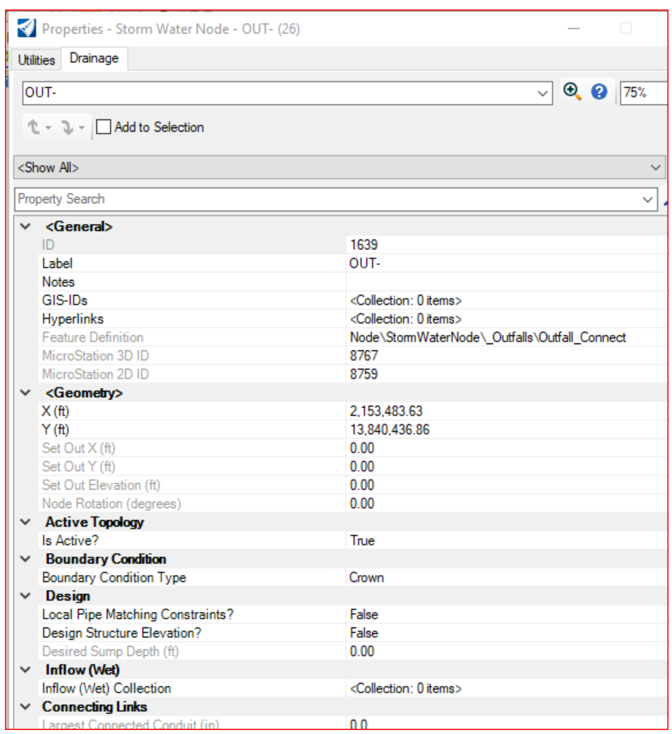
a) Select the **Place Node** tool.

1) Set the Feature Definition to Outfall_Connect. You can find it under the Node > StormWaterNodes > Outfalls category.
2) Set the *Name Prefix* to OUT – (your system or network name here)
3) Select Reference Element for Node Elevation by Selecting a contour line being displayed from the terrain surface, **Existing Ground**.
4) Toggle On the *Vertical Offset* option and set it to 0 (The Elevation is being read at the invert of the outfall in the cell).
5) *Define Outfall* location with a data point. Place along the red element that represents the *Tie Slope*.
6) Set the *Rotation Mode* to **Absolute**.



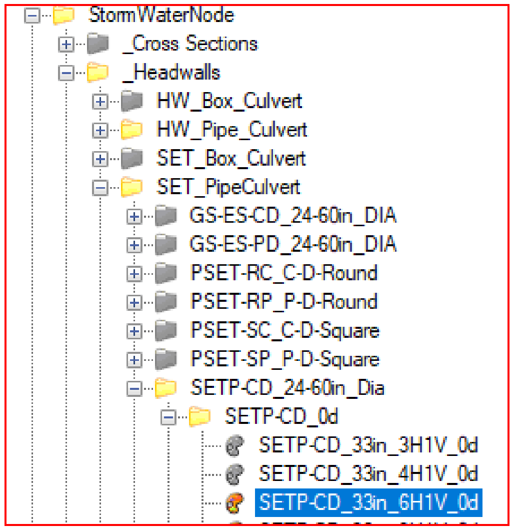
7) *Data Point* to place the outlet.

b) Review the Properties and the Utility Properties of the Outfall.



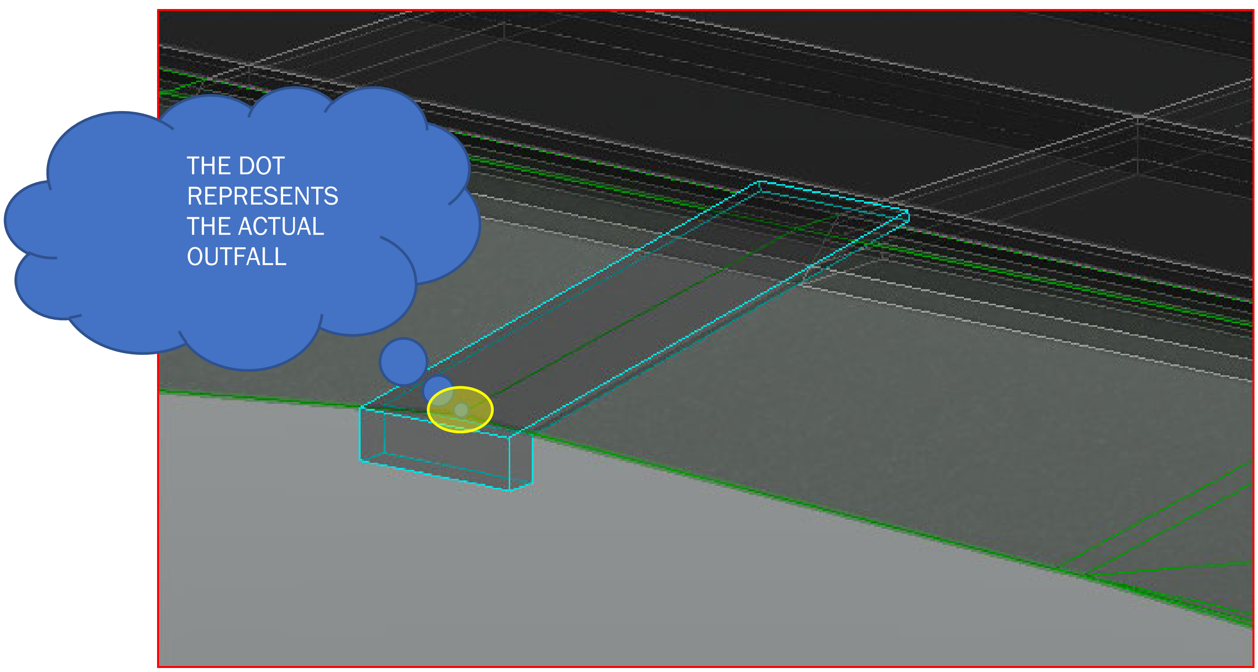
c) Close the two Properties Dialogs.

The “Outfall_Connect” node we placed in the step above contains all the hdyraulic properties of a storm sewer network outfall. Therefore for graphical and visual purposes, a headwall will be added. This time we will select feature definition **SETP-CD_33in_6H1V_0d** from the drop-down



menu.Follow steps 5 and 6 above.

d) Click in **View 2** to make it active. Select *Primary > References* and turn on the display for **your roadway corridor file.** Zoom in to the outfall. Your view may vary depending on your Display Style setting.



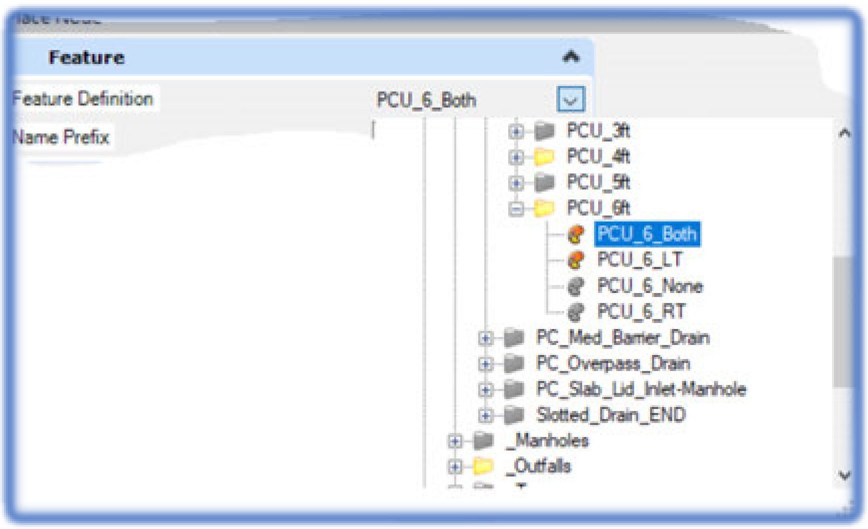
2. Place Drainage Node Without Automatic Catchment

We will now place the rest of our structures

(a) Zoom into the location in the model where you want to place your structure.

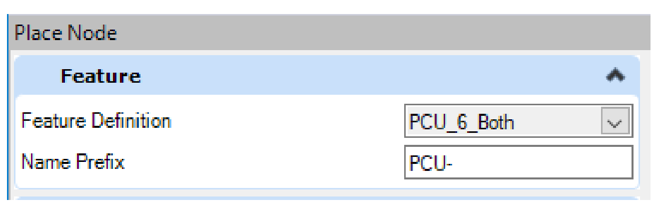
(b) click the **Place Node** tool again. This is under Layout> Place Node tool.

(c) In the next several steps we will Follow the context menu prompts to place a structure.

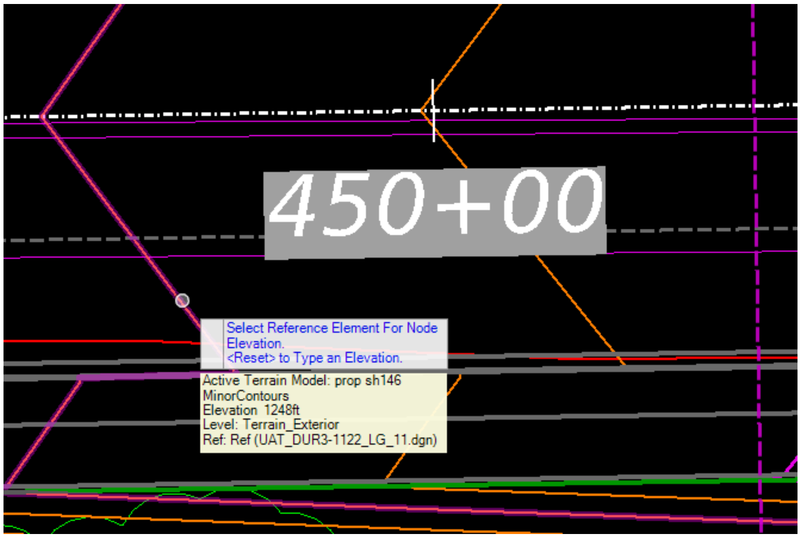


(d) From the Place Node window, set the Feature Definition to your structure type:

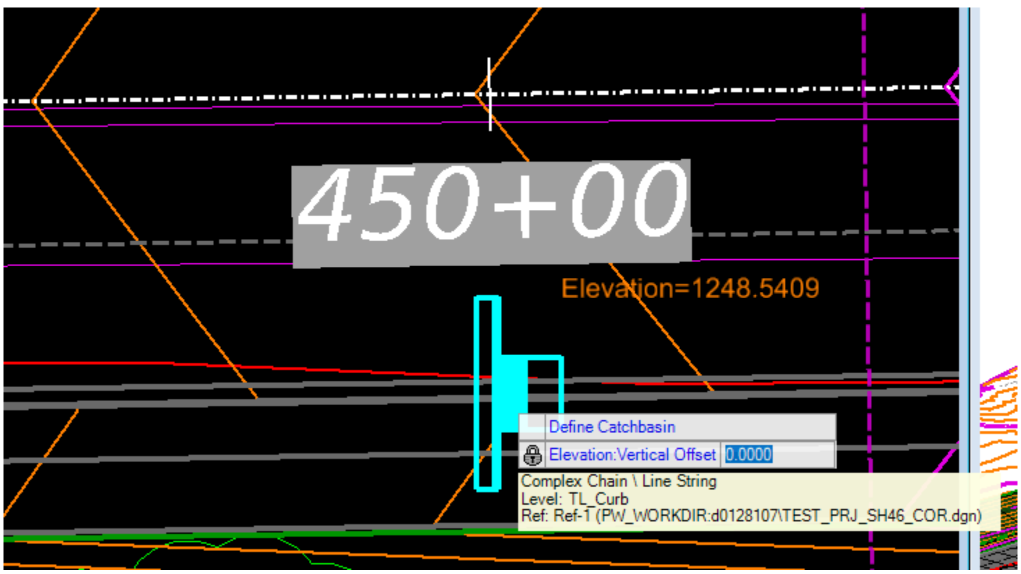
(e) Set the name Prefix – a predetermined name based on the structure beign placed will be generated. You can change the name if you like.



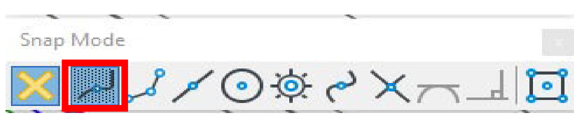
(f) You will be prompted to select Reference Element for Node Elevation.



- You can select a contour line being displayed from the Proposed terrain surface, or the proposed corridor.
- You can select the Reference Element in the 2D view or the 3D view.
  - In the 2D view, *View 1*, the plan view cell for the inlet, is attached to your cursor.



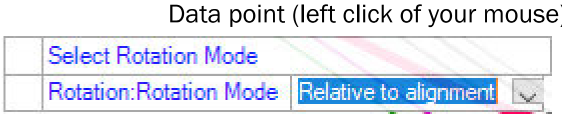
(g) Set the Snap Mode to Near Snap Point.



(h) Locate the Feature Curb_Face_FL_R. This represents the flow line in the corridor. Make sure you snap to it so you can read the logintudinal slope.

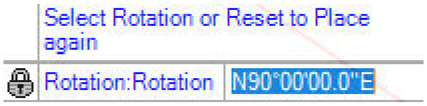
(i) Set the Vertical Offset Data point in View 1 to accept the Vertical Offset of 0.00.

(j) Set the Rotation Mode to Relative to Alignment.
Data point (left click of your mouse) in View 1 to accept that mode.



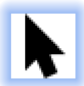
(k) Set the Locate Reference Element for Rotation. Click on the Centerline of the Roadway. (project alignment or PGL).

(l) For the Rotation setting, you will key in N 90 E and click the Tab key or Enter key to lock the rotation. The “E” or “W” direction will depend on which side of the roadway

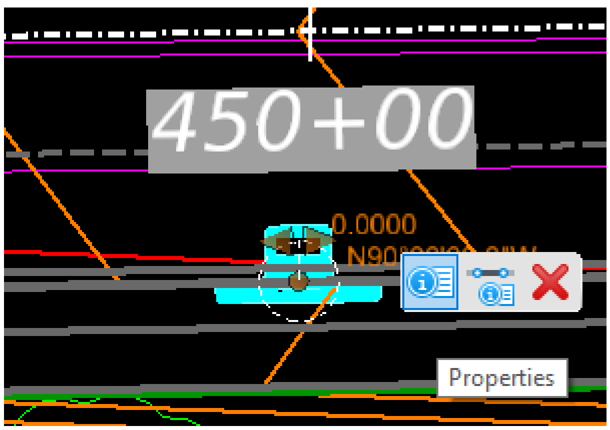


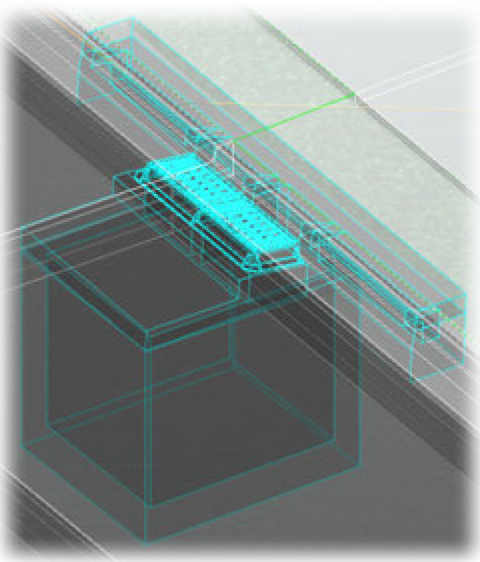
your inlet is on

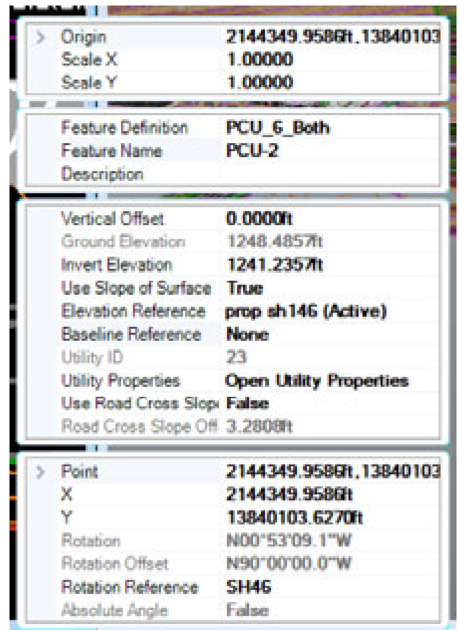
(m) Make Sure the Catchment > Catchment Delineation is unchecked and then Data Point(left click) in View 1.



(n) Select the Element Selection Tool. Select the newly placed inlet and then hover over it, and select the Properties.



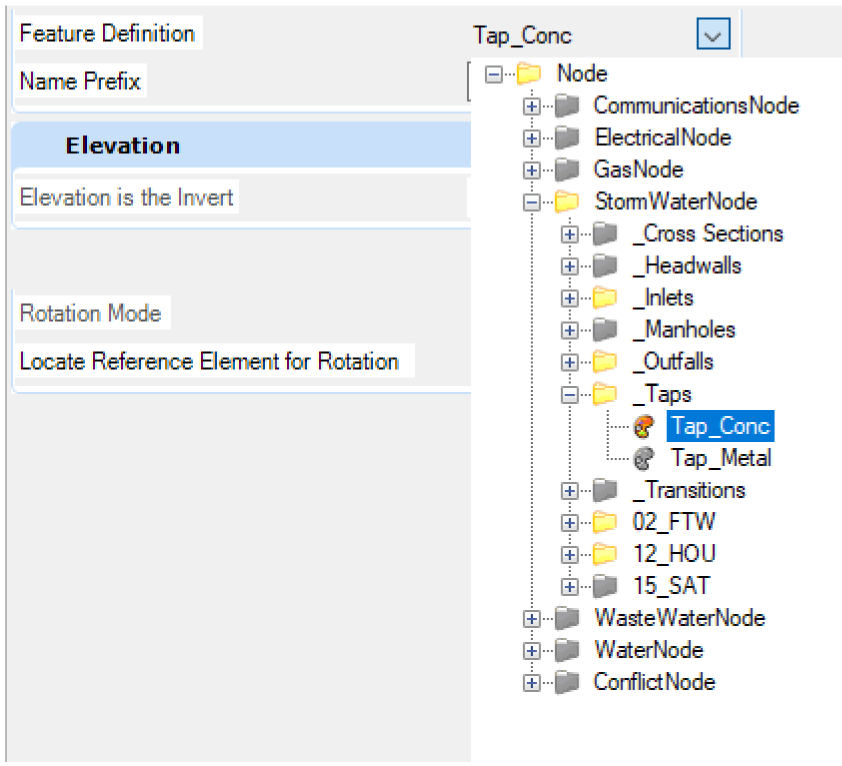




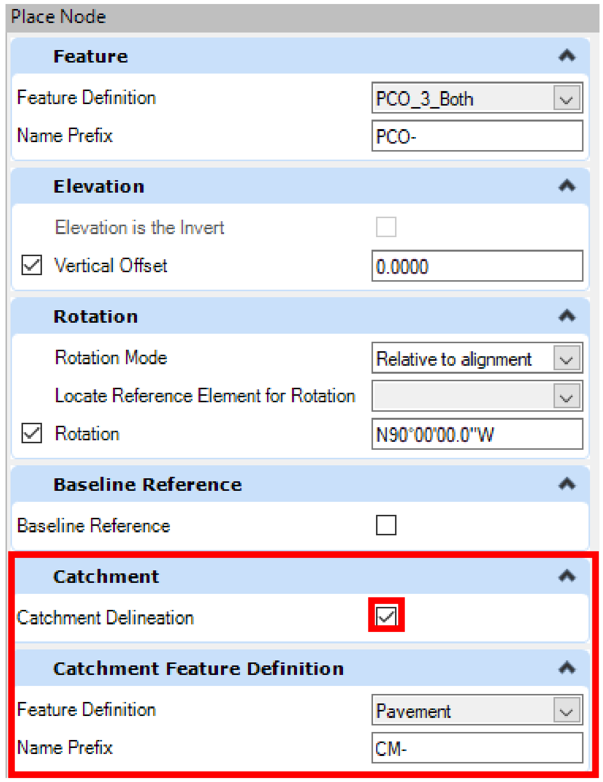
3. Place Drainage Node With Automatic Catchment

DU has the capability to automatically calculate and create drainage areas for inlet structures. DU must have a Proposed Terrain file to complete this task. See DES750 for instruction on how to create a proposed terrain. The roadway designer on the project can also create the proposed terrain.

(a) Follow the same procedure above, except this time, turn on catchment delineation. We will select a different node for this example. Drainage structures in the TxDOT workspace are cataloged as follows:



(b) Select the inlet type to use. Select Catchment Delineation and select your Catchment Feature Definition as shown below.



(c) Place your node as in the exercise above. Make sure your proposed terrain is selected as your reference elevation for catchment delineation.

The area will be automatically delineated and assigned to the node.

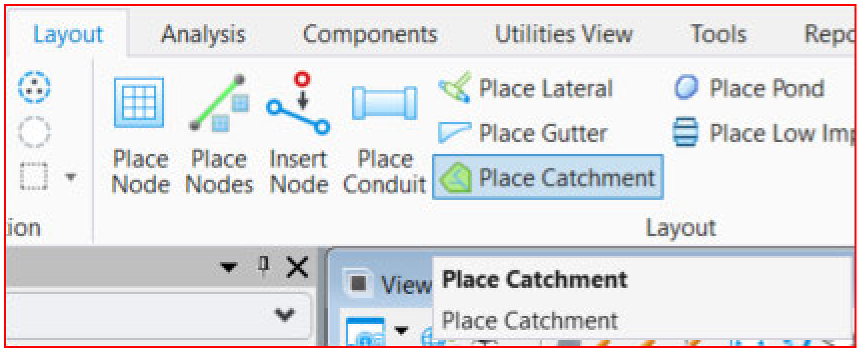
You can also assign a predefined area to a structure or trace the drainage area and assign it to the structure. We will show how to do this next.

4. Assigning a predefined area to a structure – picking your catchment.

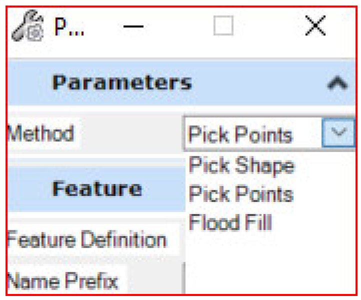
If you chose to trace your own drainage areas, they can be assigned to the inlets with the following workflow:

You can set your drainage area file in advance and reference to your draiwing, but in order to assign areas to your structures, areas must be copied to the active Dgn file.

(a) select the Place catchment tool form the Layout Tab

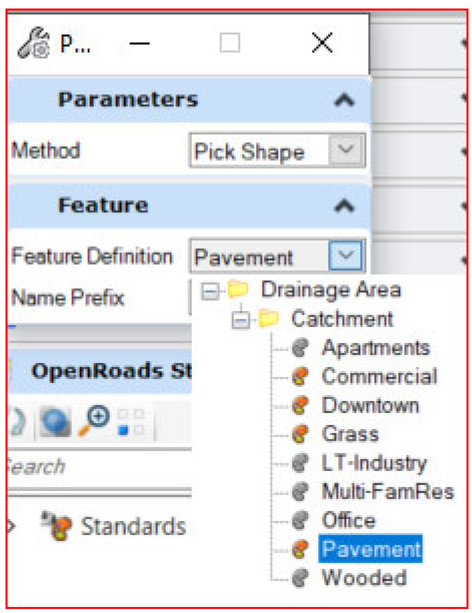


(b) Select your layout method. You have 3 options available as shown below:



(c) If you have preset shapes, select pick shape.

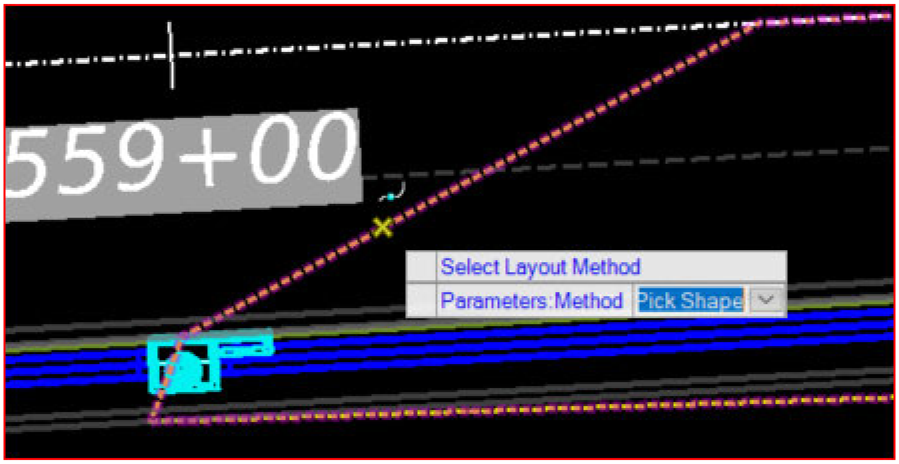
(d) If you want to trace your area select pick points.



(e) Select your feature definition from the list shown above

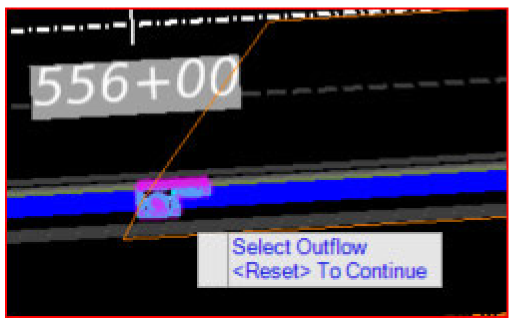
(f) For this example we selected pick shape for the method and Pavement as our Feature definition.

(g) Click on the area you wish to assign to your inlet

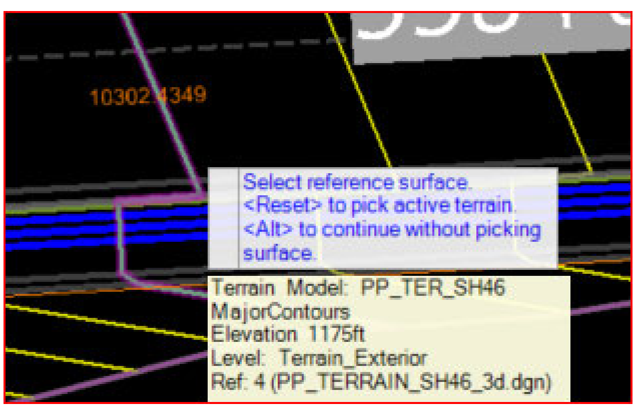


(h) Data point to accept the selection

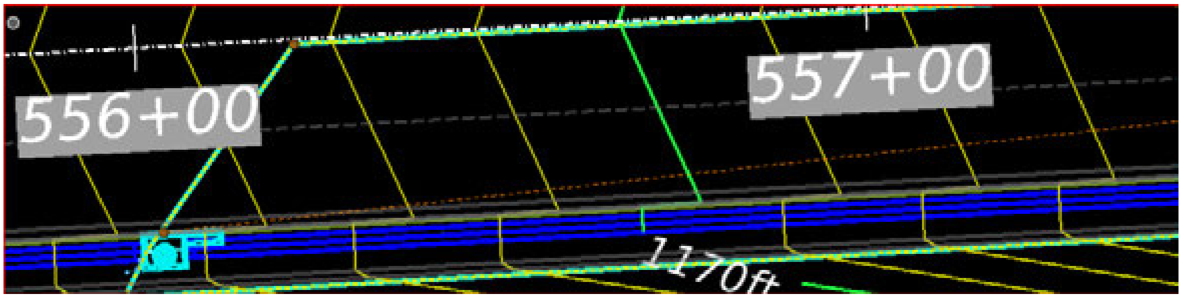
(i) Select the outflow (your structure)



(j) Select the active terrain. Your corridor or your proposed tin can be selected.



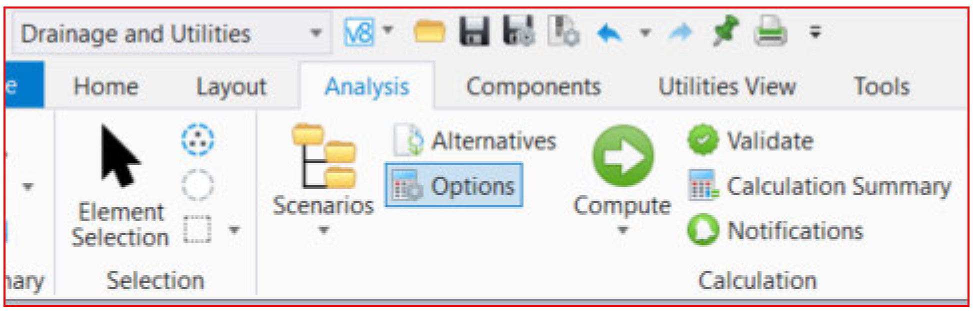
(k) your area is now assigned to your structure



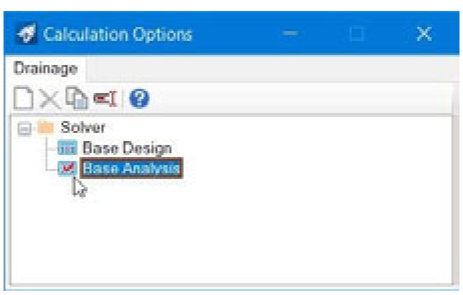
if you do not have predefined drainage area you can alternativly use the Pick points method and trace your area. Once your area is traced follow the promts above to assign it to the catch basin.

5. Setting the Minimum Time of Concentration

(a) *From the analysis tab, Click Analysis > Calculations > Options.*



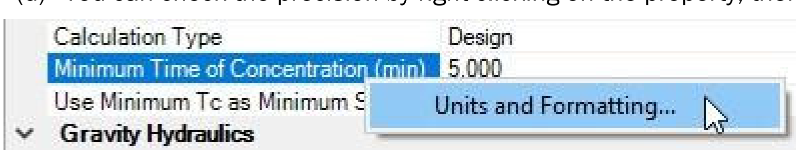
(b) the following window will popup:



(c) Double click the **Base Design** and check the *Properties.*

This is where the default value is set for the **Minimum Time of Concentration** for your catchments. This value is used if it has not been set on the individual catchment areas. The default value should be 10 minutes. If it is not set, go ahead, and change it here.

(d) You can check the precision by right clicking on the property, then clicking Units and



Formatting.

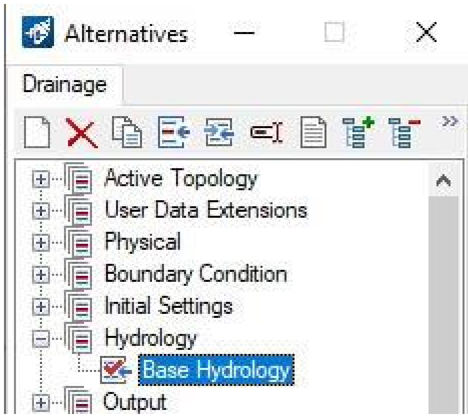
(e) Close the Properties dialog.

(f) Close the Calculation Options.

6. Setting the Time of Concentration for the Catchment Areas

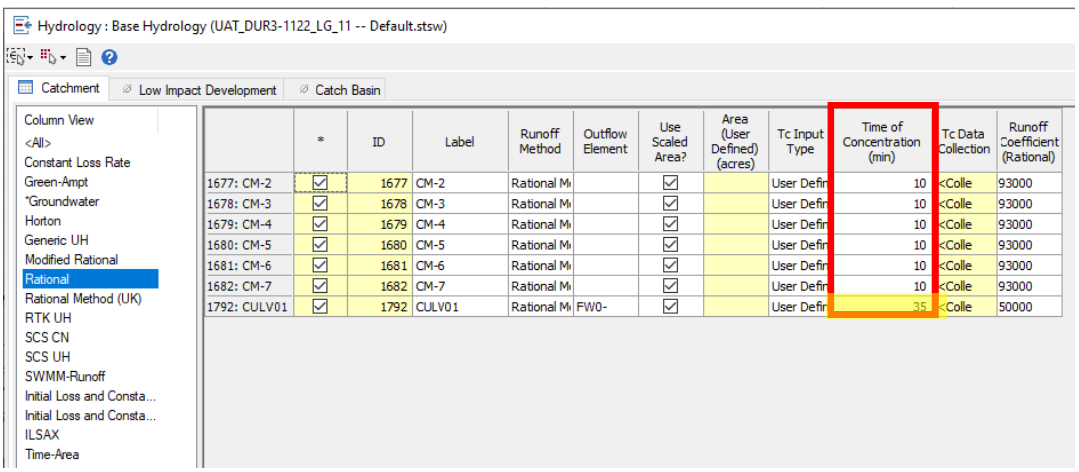
(a) On the Analysis > Calculations ribbon, click Alternatives.

In the *Alternatives* dialog, find the **Hydrology > Base Hydrology** Alternative.



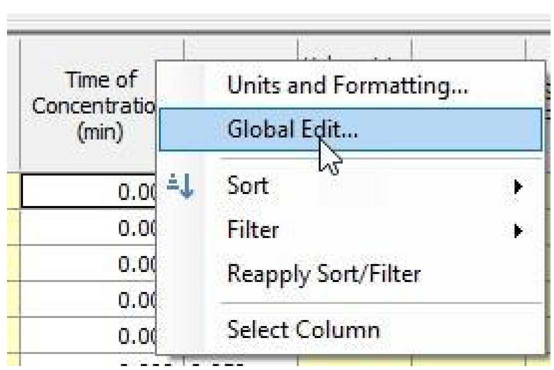
(b) Double Click on **Base Hydrology** to view the Hydrology.

(c) Select the **Catchment** tab.

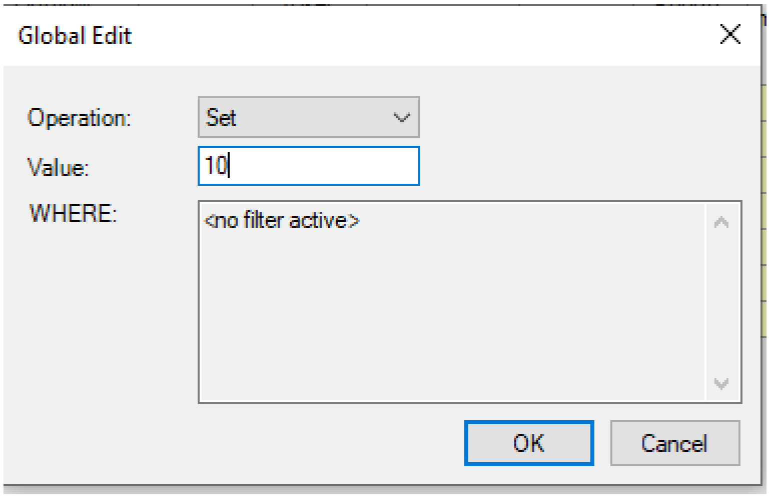


(d) Time of Concentration values can be set here for individual or all catchments.

(e) Right click on the **Time of Concentration (min)** column heading and select *Global Edit.*



(f) Set the value to *10* and click **OK.**

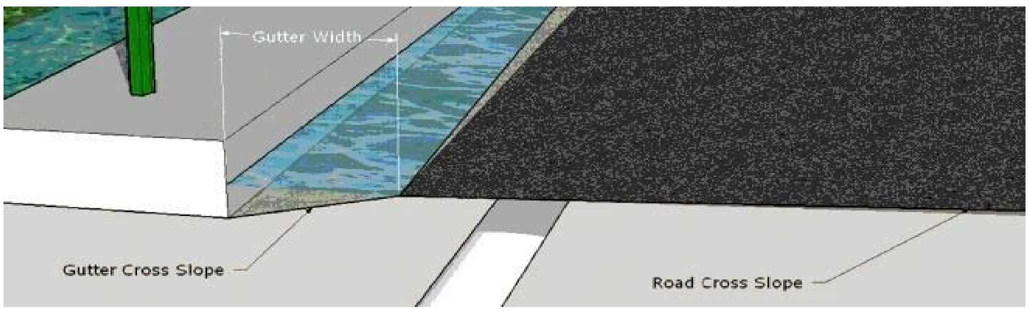


The values are now updated. Remember that individual modifications can be made if required.

(g) Close the dialog.

7. Placing Gutters

Placing gutters is necessary to assign by-pass flow in case your structure cannot capture all flow from the assigned drainage area.



1) Methods of Placing a Gutter

The software has two methods that can be used to place a gutter:

1. Using Trace Slope - Only asks you to select the start catch basin.
2. Between Nodes - Asks you to select the two catch basins that are to be linked by the gutter.

The Using Trace Slope method has two prerequisites:

- The catch basins were both placed using the same terrain model or mesh as the Elevation Reference.
- The catch basins are placed and rotated correctly within the gutter.

When you select the Start node, the software uses this as the starting location for a downstream flow trace, along with the terrain model or mesh element, to find the next catch basin downstream. If the Start node was not placed using a terrain model or mesh as the Elevation Reference, then you will not be able to select it.

The rotation of the catch basin is important if the prototype used by the gutter feature definition is set up so that cross-sections are created along the gutter. In this situation, the rotation of the catch basin should be perpendicular to the gutter flow line, so that a cross-section taken through the catch basin, along the bearing of the rotation, will include the shape of the gutter.

2) the Gutter shape

TxDOT has one gutter shape that will be available in DU, which is a conventional Gutter shape.

The prototype for the gutter feature definition has been predefined in the **DU Feature definitions and element templates DGNLIB** file.

We will now show the workflow to place the gutters.

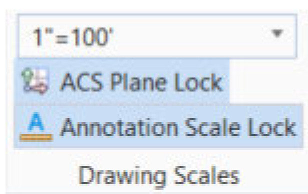
3) Placing a Gutter – Trace Slope Method

(a)  Select Layout > Layout > Place Gutter.

(b) When the tool is started, there is a check made to determine if **Analytic View** is turned on.

(c) Select **Yes** to toggle on the Analytic View because hydraulic gutter definitions are visible only by way of the Analytic View.

Note No labels are shown in the default view. This is a scale issue as our design files are set to 1" =100'.



(d) On the *Drawing Production* ribbon, select **Drawing** Scales > Annotation Scale. Select the drop down and pick *Full Scale 1:1.* The text is now visible.

The display shows useful information, but it may be overpopulated for placing gutters.

(e) In the top left corner of *View 1*, Select **View Attributes**.

(f) Ensure that the *Select Product* property is set to *Drainage*.

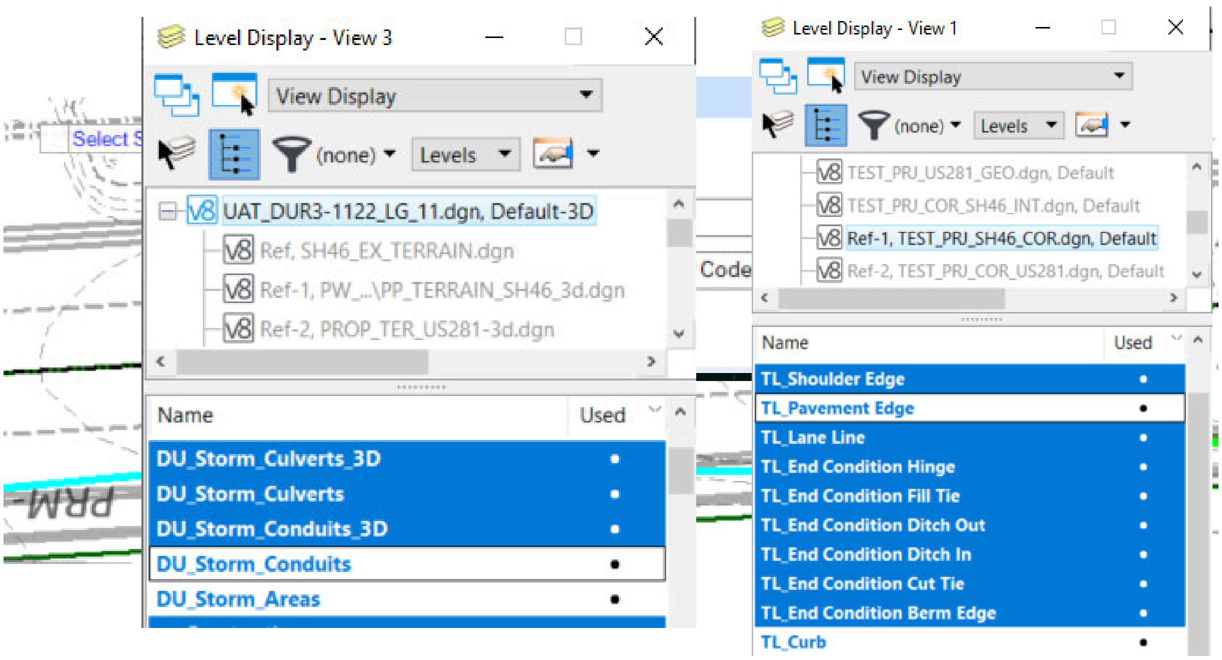
(g) Change the Symbology Definition from <default> to Pipe Size Color Code.

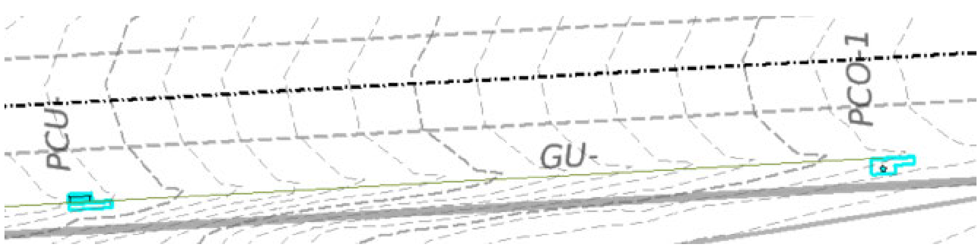
You can change back to *<default>* or other definitions later.

Note that **Analytic Symbology** can be turned on and off in **View Attributes** (remember you will not see the gutters if it is turned off).

(h) Go to **Home > Primary > Level Display**, click on **the Drainage Design DGN** and turn off the levels of *DU_Storm_Areas* and *DU_Storm_Conduits*.

After that, click on **the Roadway Corridor Reference** and turn off *TL_Curb, TL_Pavement Edge,* as these levels will overlap the newly placed Gutters.

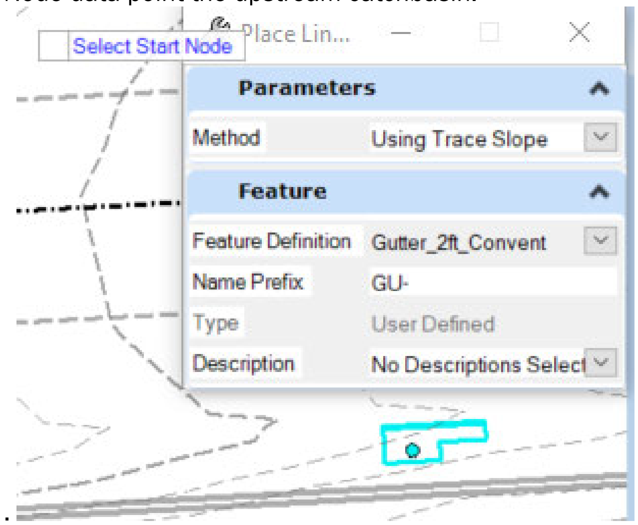




(i) Set the *Method* to **Using Trace Slope.** The path of the gutter will be calculated by tracing a slope downhill along the Elevation Reference surface.

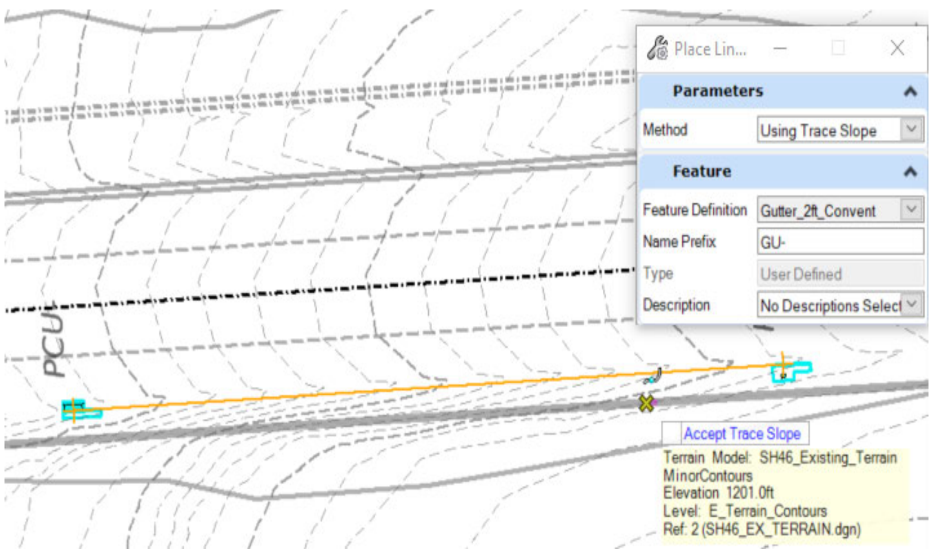
(j) Set the Feature Definition to Conduit > StormWater > Gutters > Gutter_2ft_Convent.

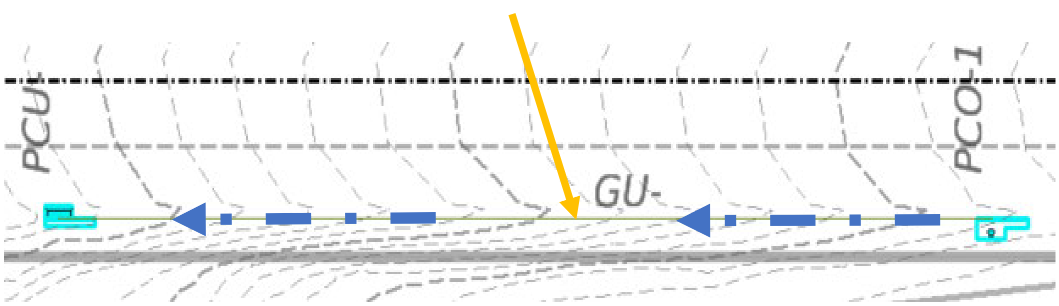
(k) The Name Prefix is set automatically to GU-. Following the prompts, at Select Start Node data point the upstream catchbasin.



The path of the downstream flow trace to the downstream catch basin is shown in orange.

Data point to Accept Trace Slope.





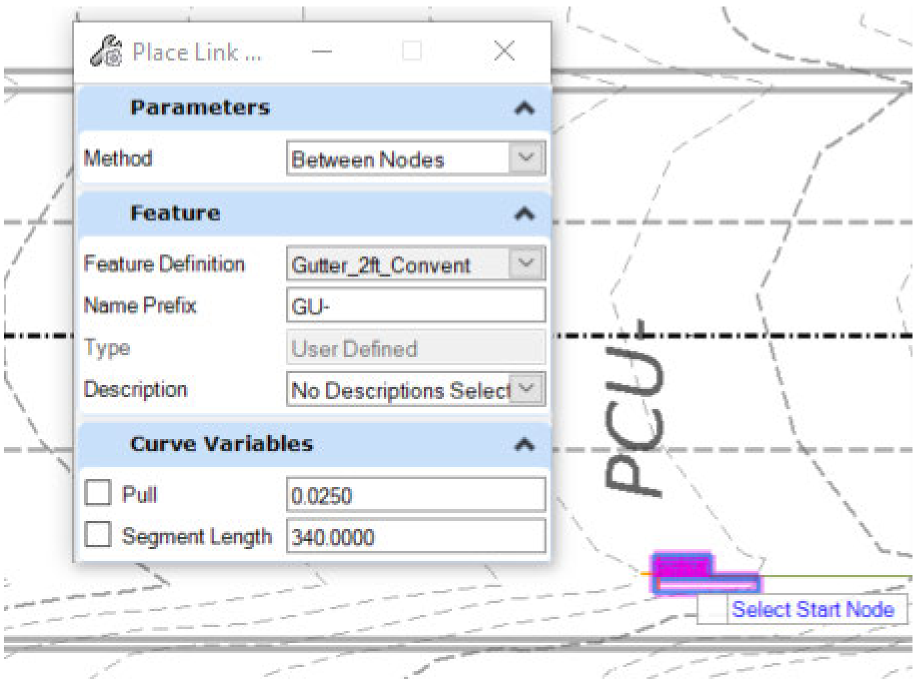
4) Between Nodes Method

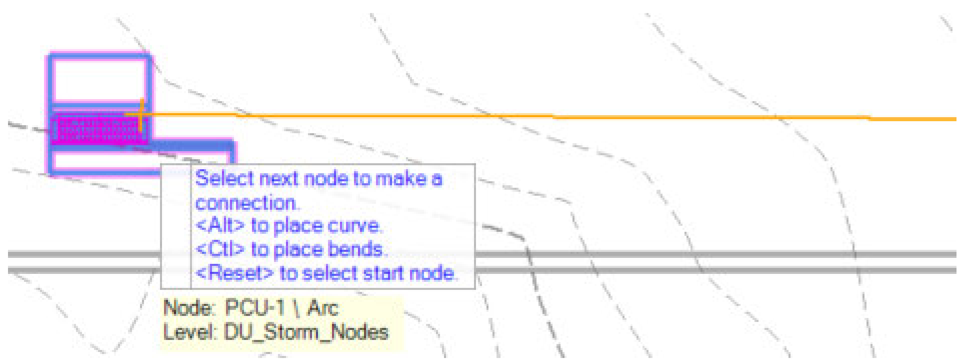
(a) You can also use the **Between Nodes Method** to place your gutters.

(b) Change the *Method* to **Between Nodes.** Both catch basins need to be selected in this method.
Note: The TxDOT workspace only has 1 feature definition option for gutters.

- This feature definition will use the same gutter cross-section as the Stop (downstream) Node.

Following the prompts, at *Select Start Node* data point your PC catch basin.





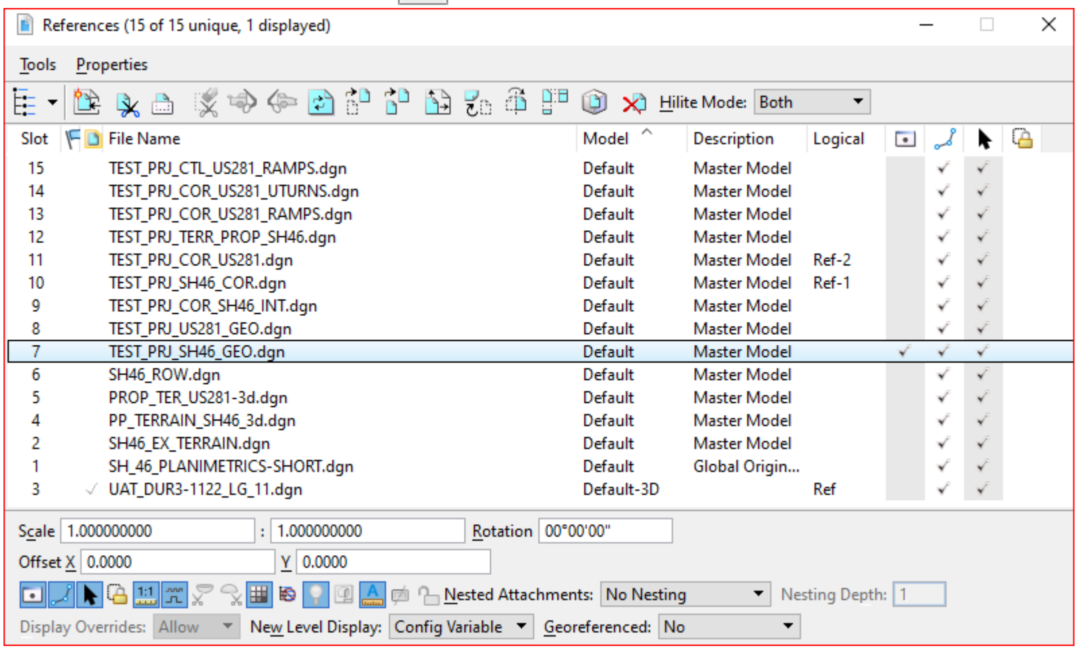
Note that, when you are using the **Between Nodes** method, you should pick a suitable location on the proper side of the catch basin, relative to the direction of the gutter.

8. Place Conduits and inserting nodes

1) `Place Conduits

In this section, we will cover creating drainage pipes to connect our previously placed nodes. We will begin at the *downstream end* of our **network system**. When placing the pipes, we will always ensure to place the **pipe** beginning at the *upstream node* and ending at the *downstream node*.

(a) First, we will turn off the display of several reference files to simplify our view. Select the **Reference File** tool.



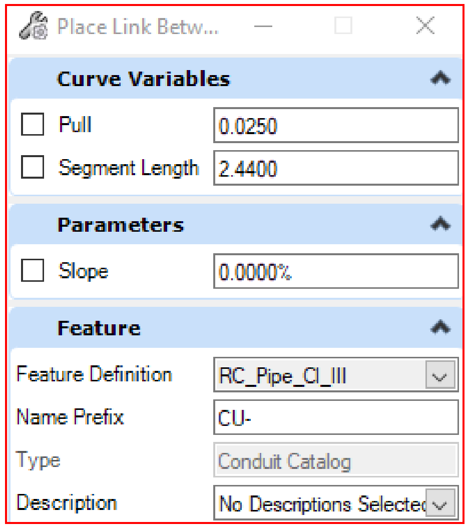
(b) Toggle off the **Display** of every Reference file in **View 1** except the **GEOMETRY OR ALIGNMENT FILE**.

(c) Close the Reference File dialog.

Note: In a prior step we turn off the pipe display in the 2D view to be able to see our gutters. Go ahead and turn the **DU Storm Conduits** level back on.

(d) Select **Drainage and Utilities** workflow, then **Layout > Layout > Place Conduit**.

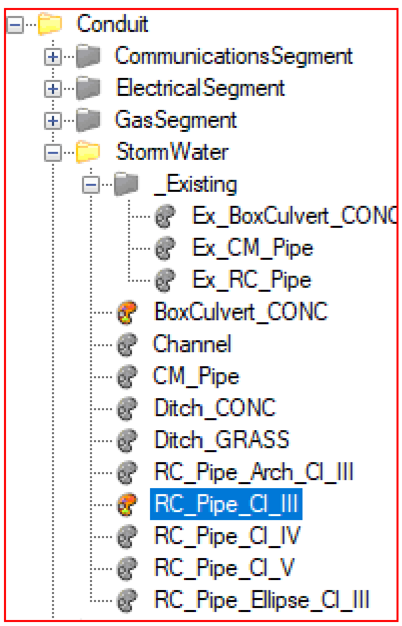
(e) The Place Link Between Nodes dialog will appear.



(f) We will set the Feature Definition.

1) Set the Feature Definition to RC_Pipe_Cl_III found in the Conduit > StormWater >.

The StormWater conduit feature definition holds all pipe, culvert, and ditch types available for design, including Existing, as illustrated below



(g) Set the *Name Prefix.* A standard name has been set, but it can be changed to your preferences.

(h) Select 18” for the *Description.*

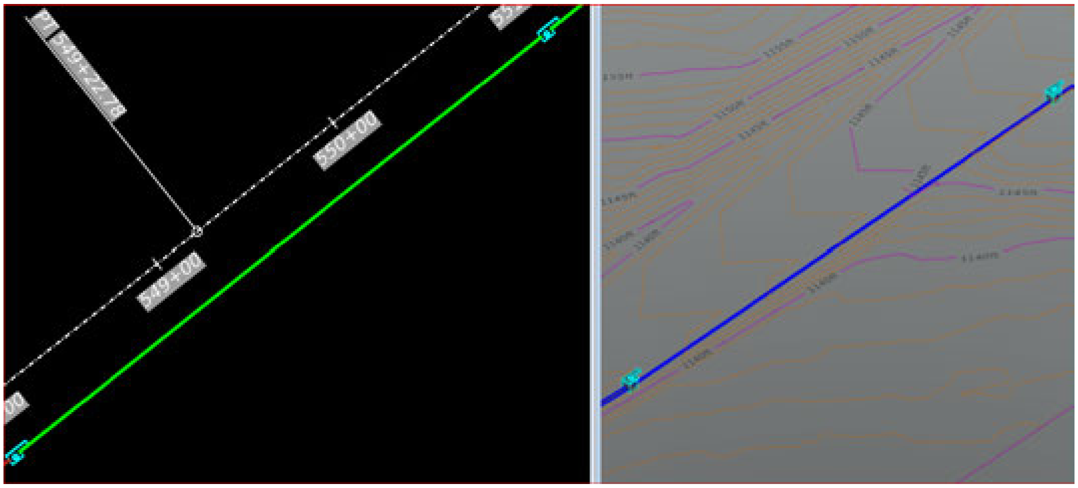
(i) Toggle Off *Civil AccuDraw* if it is still on.

(a) The first prompt is to select the **Start Node**.

(b) Next you are prompted to select the next node to make a connection.

Note: Always place conduits from the upstream node to the downstream node.

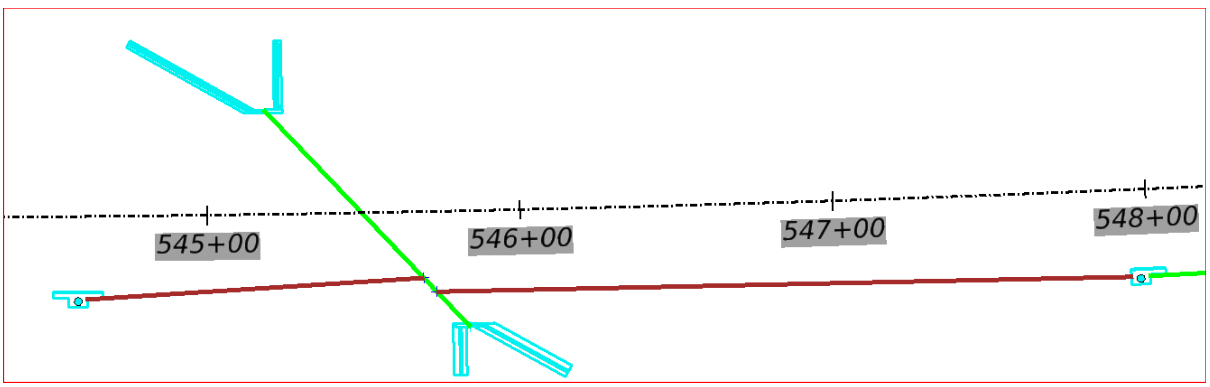
(c) The pipe is placed in the plan view and modeled in the 3D View.



We will continue in the Place Conduit tool and connect the rest of our system.

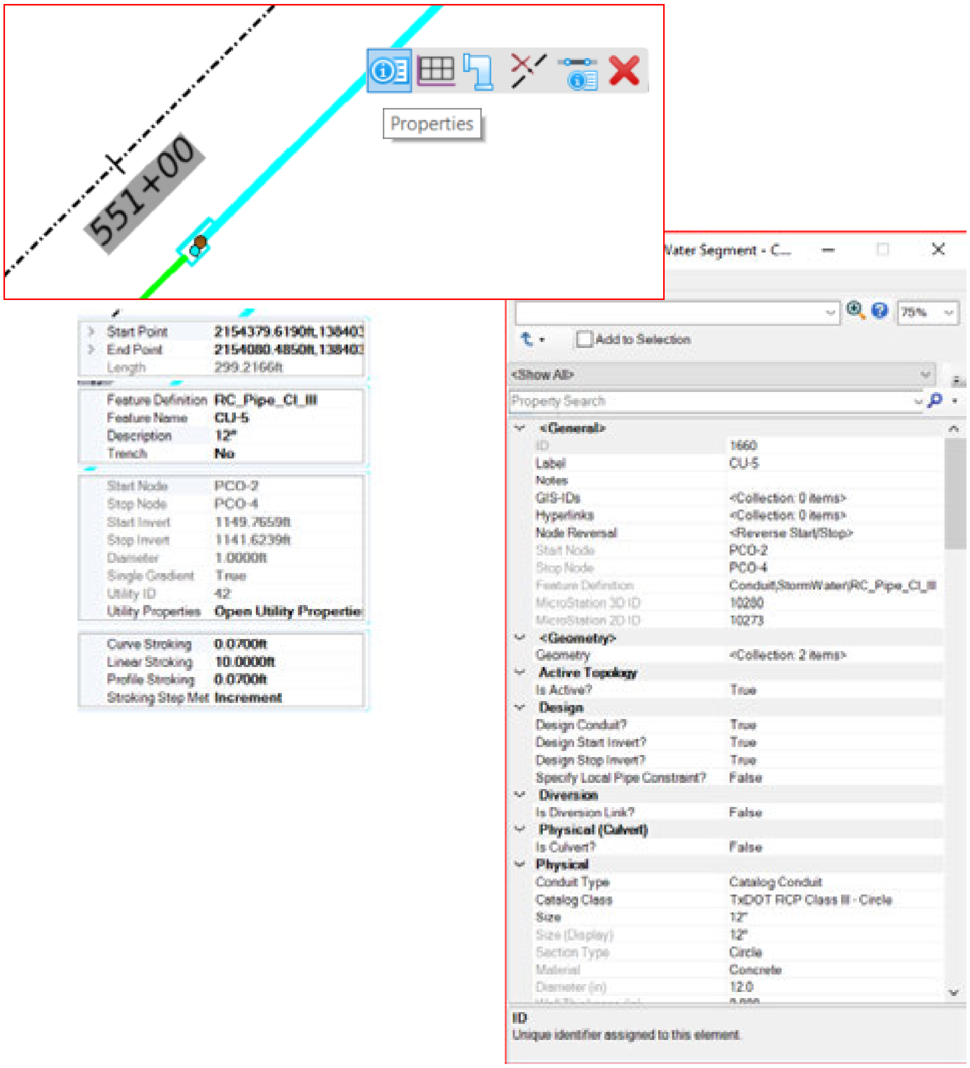
(a) Select the Place Conduit tool from the Drainage and Utilities Workflow > Layout > Layout.

(b) The sketch below shows how the conduit and node system will look in the plan view.



Note: Remember to place each pipe from the very first upstream inlet in your network and work your way downstream towards the outfall.

(c) Using the **Element Selection** tool, select one of the pipes you just placed and **Hover** over it to access the **Properties** Dialog and the **Utility Properties** dialog from the *Context Menu*.



(d) **Close** the Properties and Utility Properties dialogs.

2) Insert nodes

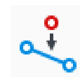
In this section, we will insert a manhole node into our Pipe.

(a) In **View 1**, Zoom so that you can see **one of the conduits**. We will place a manhole in the middle of this pipe.

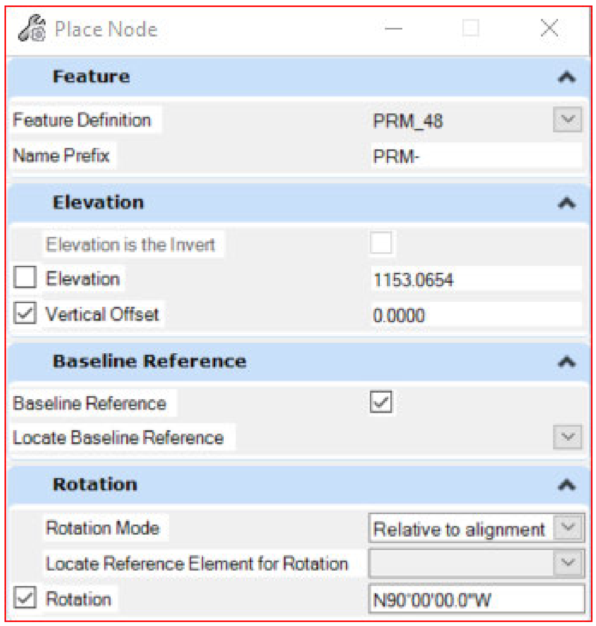
(b) Click in **View 2** to make it active.

(c) Pan or zoom so that you can see **the conduit** in that view as well.

(d) Make **View 1** active again by selecting the *view title bar.*

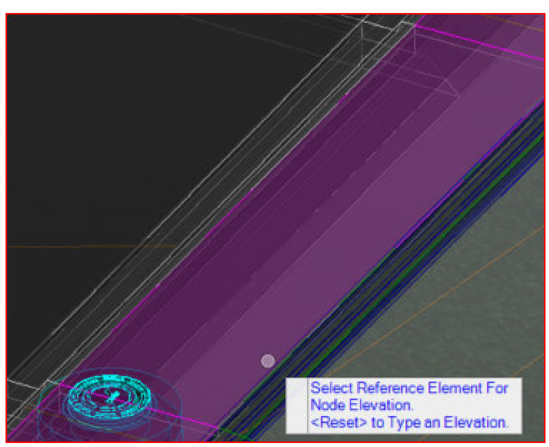
(e) Select the **Insert Node** tool from the **Drainage and Utilities** Workflow  **> Layout Tab > Layout.**

(f) We will preset the tool settings for the *Insert Node* tool.
1) Toggle **Off** Vertical Offset.
2) Toggle **On** Split Conduit.
3) Set the *Feature Definition* to Node > StormWaterNode > Manholes > PRM_48. The *Name Prefix* is preset to **PRM-.**

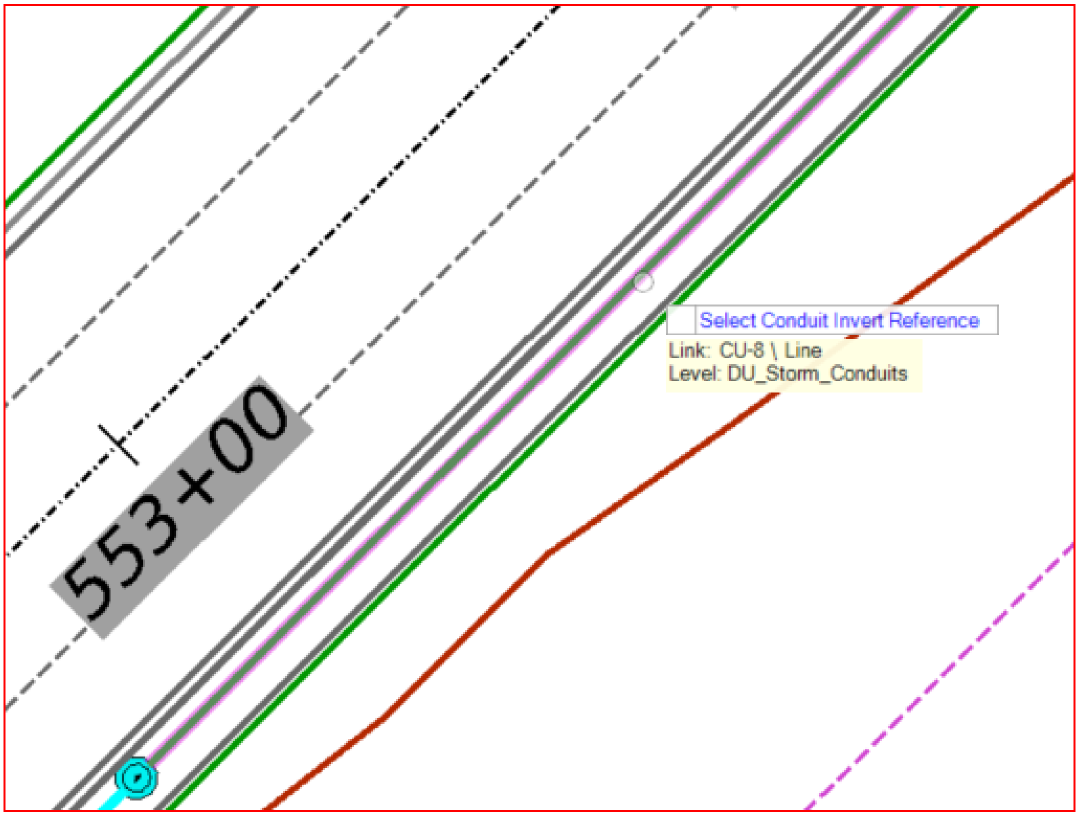


(g) The tool prompts you to Select Reference Element for Node Elevation.

(h) In **View 2**, the 3D view, select the proposed terrain.



(i) Select Conduit Invert Reference element, select the Conduit in the 2D view.



(j) Toggle Split Conduit to **Yes** by toggling the option on in the tool settings. So all we need to do is *Data Point* to accept *Yes*.

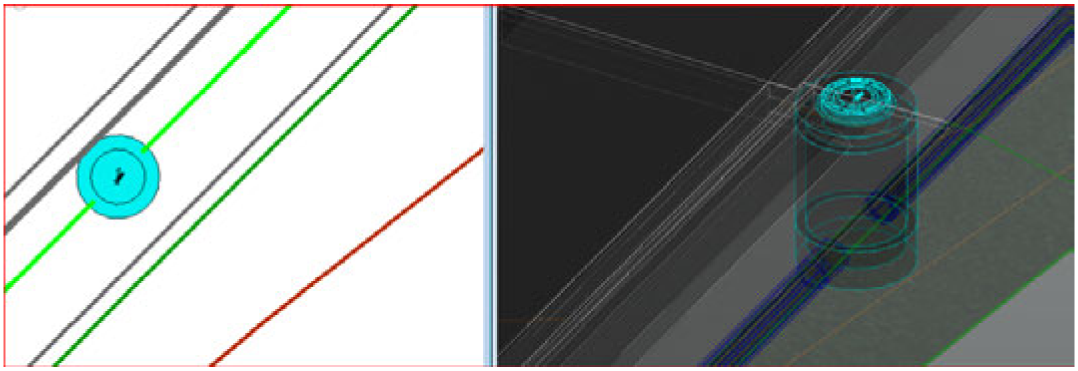
Now you will need to select the location where to insert the manhole. Pick a location and *Data Point*. Location is not critical.
Set the *Rotation Mode* to **Relative to Alignment** and *Data Point*.

(a) Locate Reference Element for Rotation.

(b) Click on the Centerline of the Roadway,

(c) For the Rotation setting, you will key in **N 90 E** and click the **Tab** key or **Enter** key to lock the rotation. Then data point in **View 1**.

(d) Zoom to the Manhole in **View 1.**



You can see from the view above that there will be some clean-up. You can move the two pipe ends to connect where you would like around the manhole, and you can rename the pipes if needed. Check that your connection points at the up and downstream inlets are attached to the correct side as well.

Note: To move the pipe, you would select it, then click on the Reconnect Link graphic handle and move it to the desired connection point.

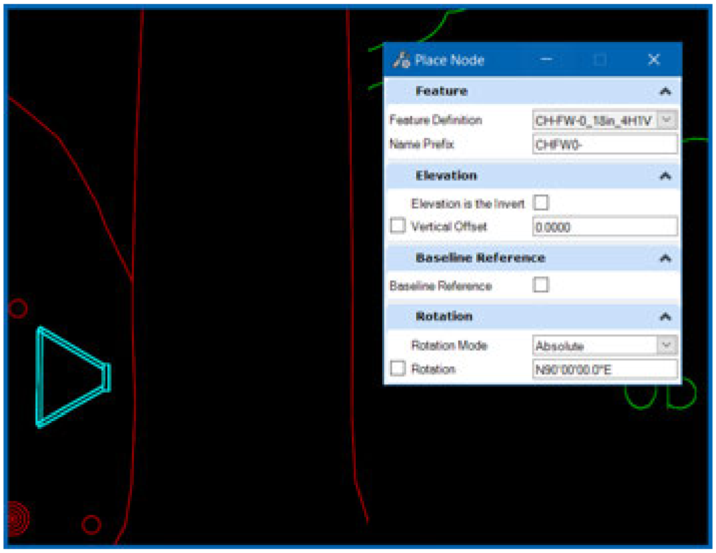
Before continuing, left click in **View 1**. Using the *References* tool, check that the **Corridor** reference file is turned **On**.

There are special workflows for placing POD inlets and creating lateral connections to an outfall. Also, we recognize the need to tie proposed drainage systems to existing pipes or structures. These particular cases will be addressed in more detail in the **SPECIAL WORKFLOWS SECTION OF THIS MANUAL.**

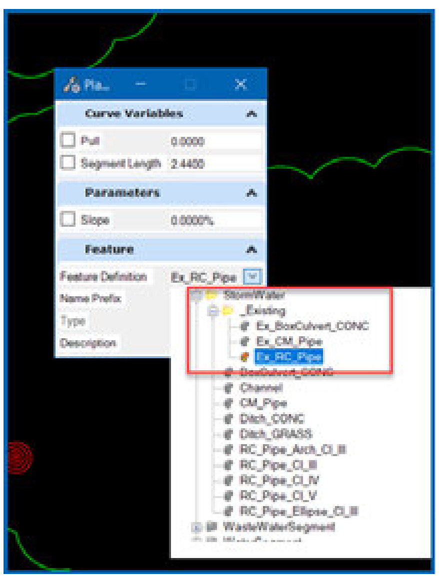
9. Drainage system layout - special workflows

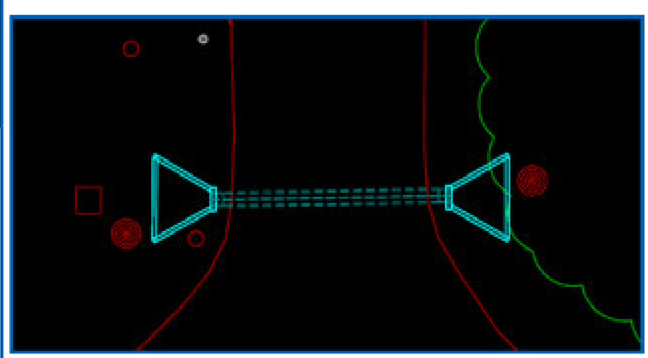
1) Extend Existing Pipe Using a Transition Node

(a) Place the existing structures using the Place Node command. This can be any node. Use the proposed nodes. User can change the line style in their file.



(b) Place the existing conduit using the Place Conduit command. Scroll to StormWater > Existing >. Use any of the 3 available conduits and select the size of the existing conduit, place from Start Node to Stop Node.

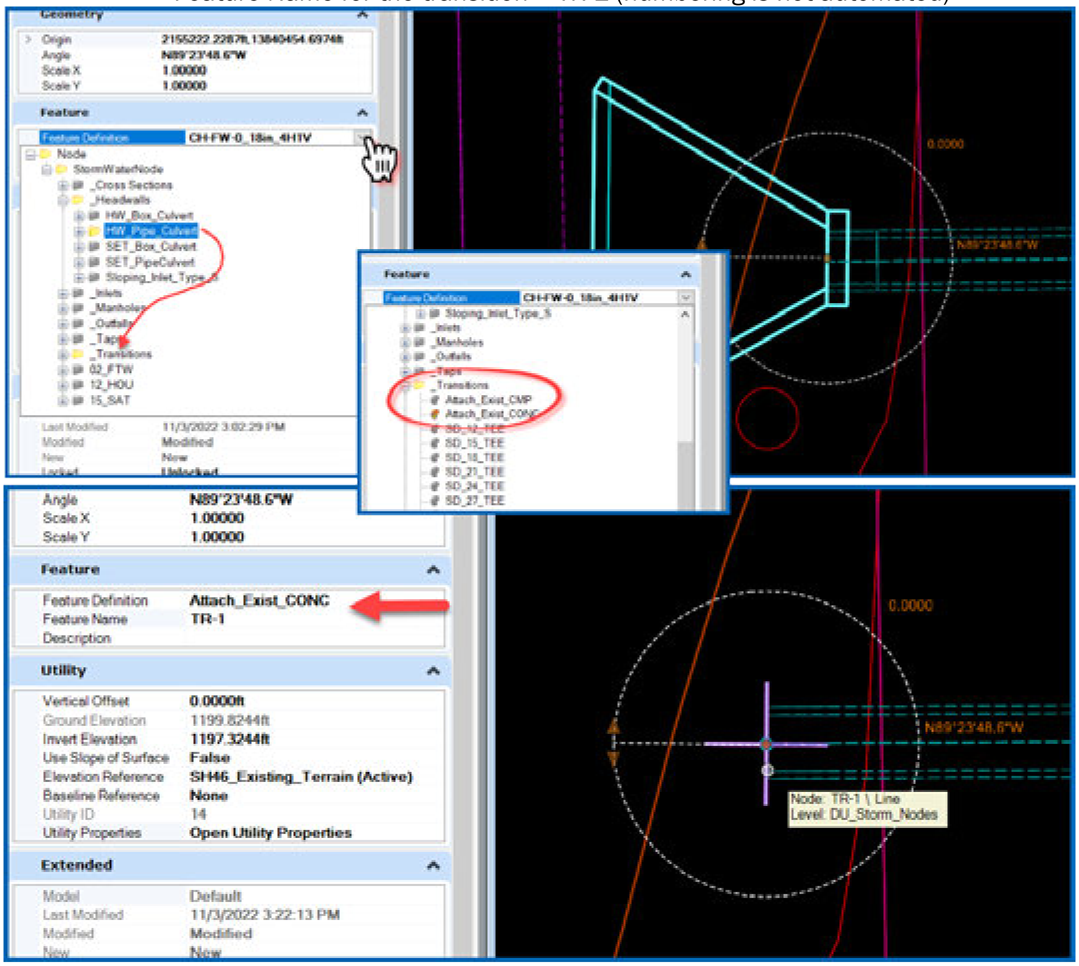




(c) Run your calculations for the "Existing System."

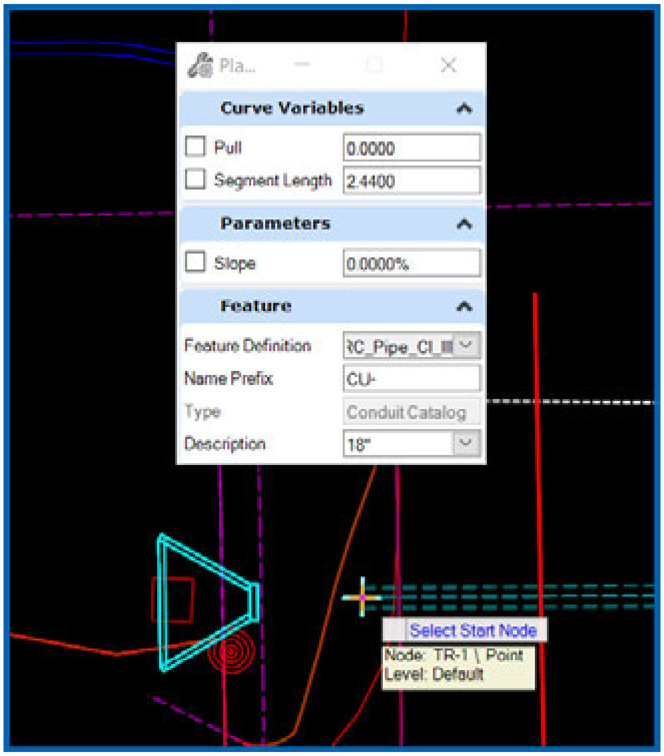
(d) Change extend the conduit with a proposed conduit. Remember the Node if deleted will also delete the conduit. To keep the conduit, we need to switch the feature definition.

(e) Change the Feature Definition for the Existing Node. In the Node Properties > Feature Definition > use the pull down. Click on Transitions >. Pick Attach_Exist_CMP or Attach_Exist_CONC. This will change your structure node to a Transition node. Type in a Feature Name for the transition > TR-1 (numbering is not automated)

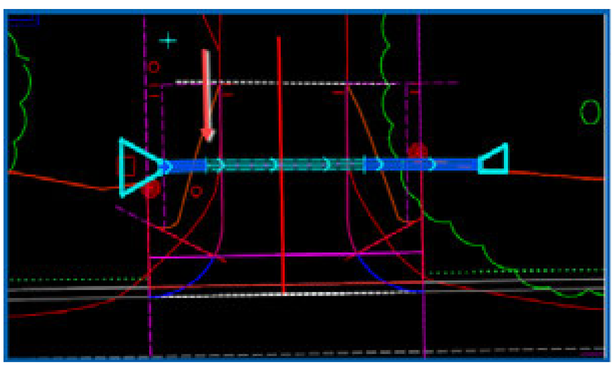


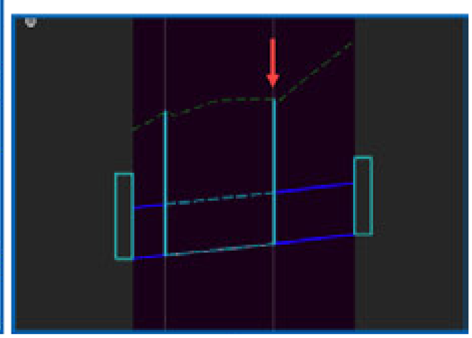
(f) Use the Place Node command to place the proposed structure node to the location needed.

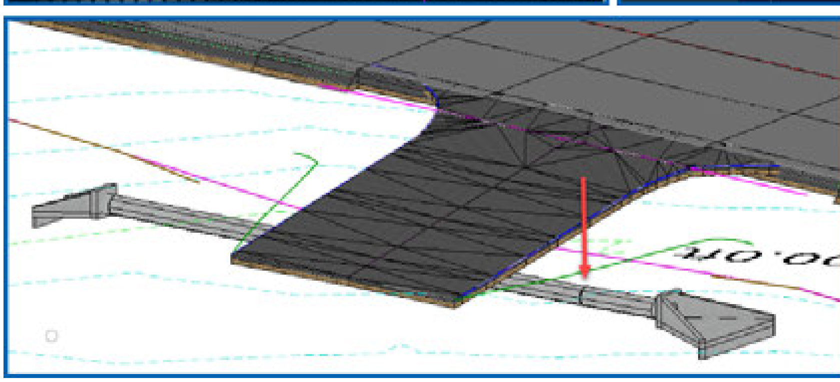
(g) Use the Place Conduit command to place the proposed conduit from the transition node (previous existing node) and the proposed structure node. Start at the highest elevation to the lower elevation.



(h) Finished. Calculate the proposed condition.



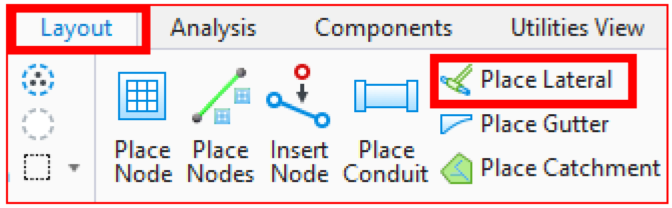




2) Place POD inlets and lateral connections

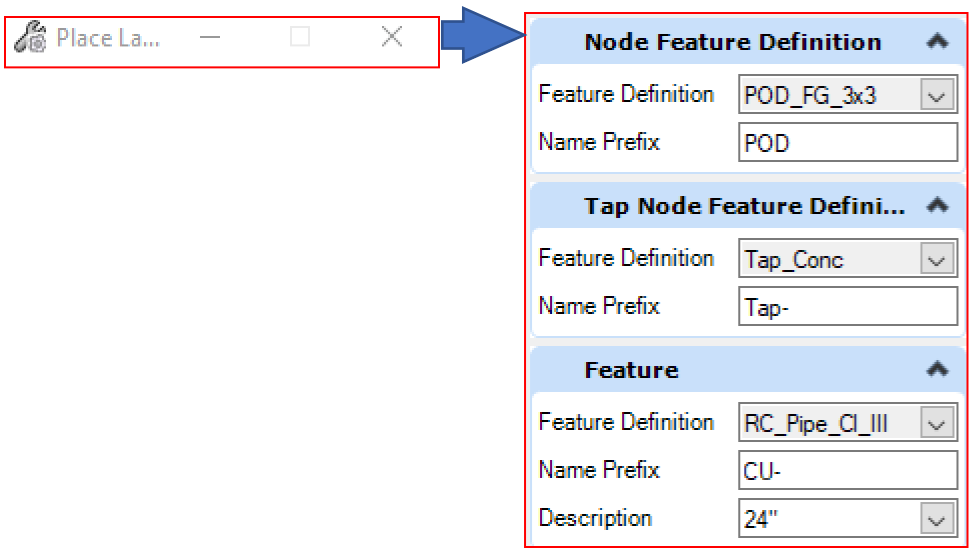
(a) Select layout tab.

(b) Select Place lateral tool.

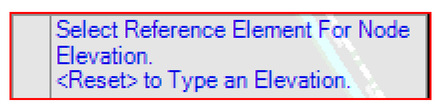


1. On the place lateral window select:

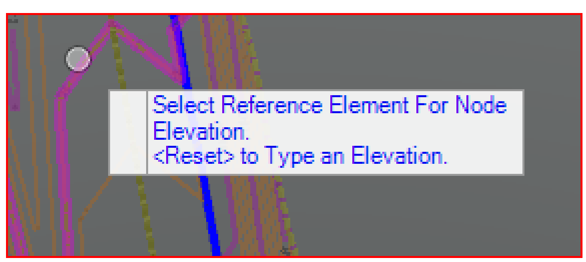
- node feature definition – your inlet or structure that will tap into the trunkline
- Tap node Feature definition – your transiton tap node
- And feature – size, type, and material of the pipe or culvert that will tap into the trunkline



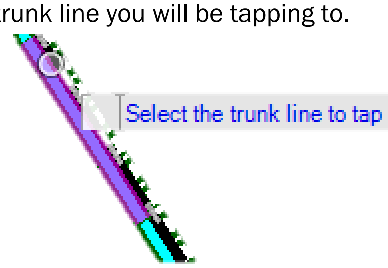
(c) follow the context menu prompts at the end of your cursor:



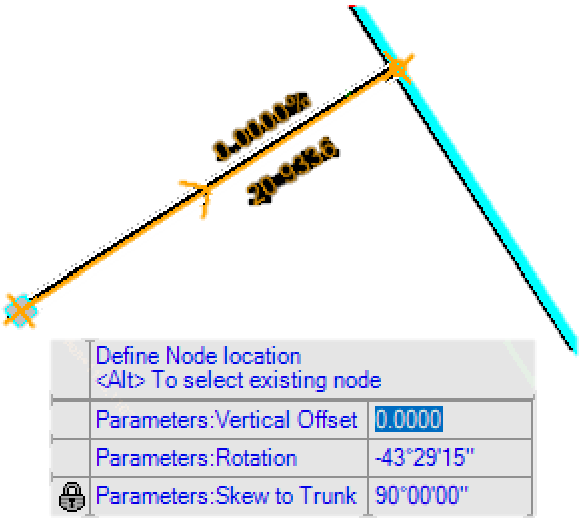
1. select your proposed terrain contours



2. Select the trunk line you will be tapping to.

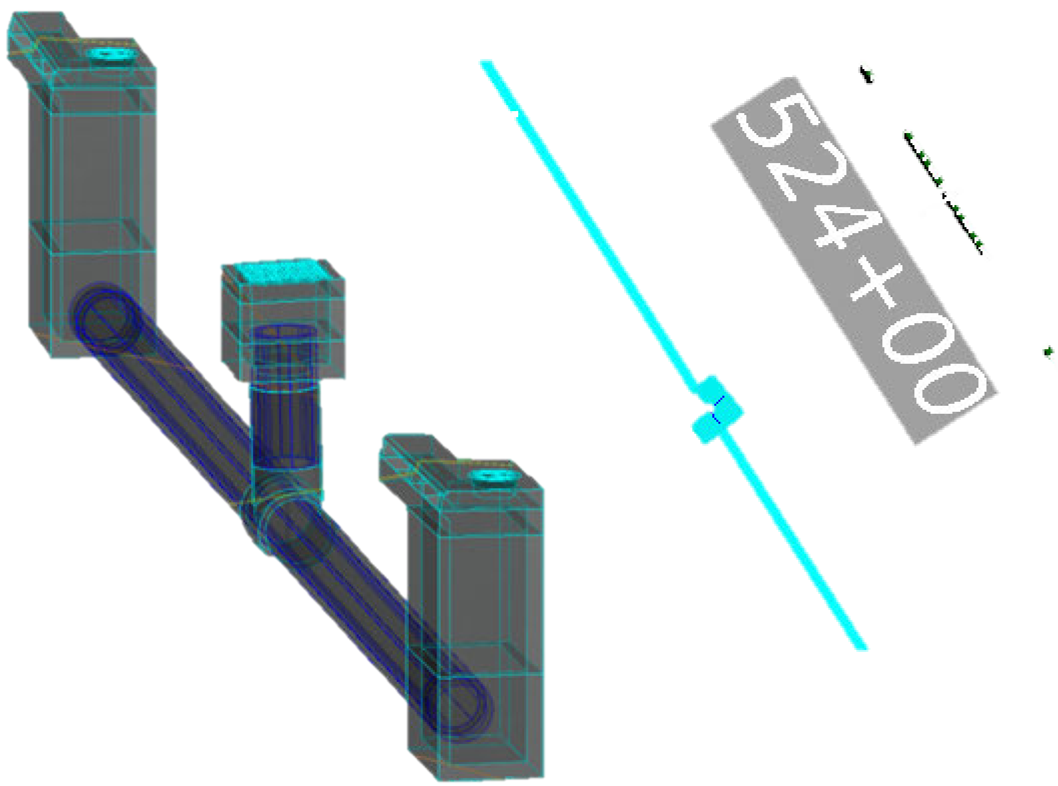


3. You will now see your structure attached at the end of your cursor and you can define your parameters



- In this case the lateral is 90 degrees oor perpendicular to the trunk line and rotated 43 degrees. You can change this to 0 degrees so the structure lines up with the trunkline. If your connecting pipe is coming at a skew, you can select the required skew angle.

4. PODs will attach to the trunkline from the top. So move your inlet as close as possible to you trunkline without touching it. Data point or right click to accept



Once your tap node is created, you can add the drainage area manually and assign it to the node or input the added flow. You can change the location or rotate the inlet like any other structure.

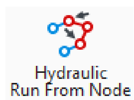
All PODs or lateral connections will need to be done this way.

10. Create Profile

In this section we will create a Profile Run. This profile will be used to aid in our design and to develop our plan profile sheets. We will discuss this in more detail under the **Plan production** section of this manual.

(a) Select one of the contours in **View 2** that represent the terrain and hover until you get the context menu. Then select **Set as Active Terrain Model.**

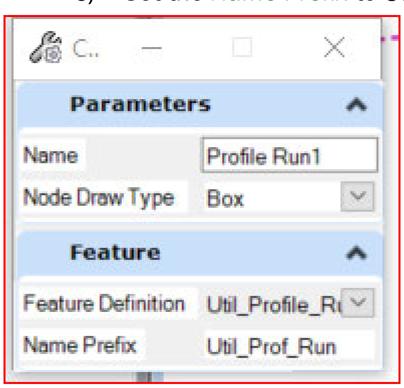
(b) Select the Hydraulic Utilities > Layout >

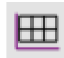


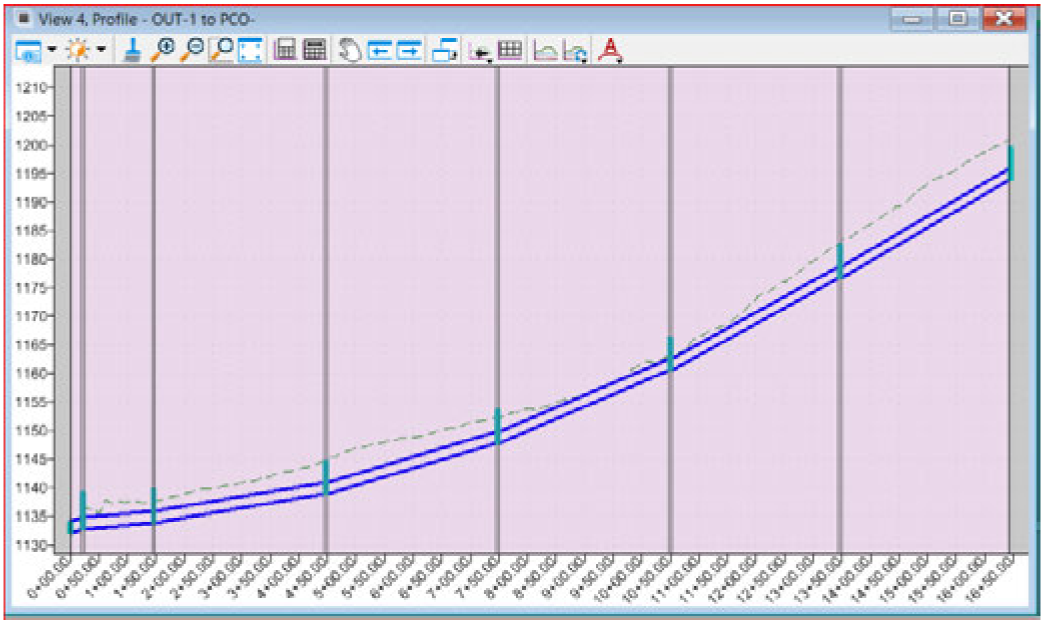
Run from Node tool located in the workflow Drainage and Profile Runs.

1) Set the Name to your preference. You can change the name later.
2) Set the *Feature Definition* to Linear > Miscellaneous > Util_Profile_Run.

3) Set the *Name Prefix* to **Util_Prof_Run.**



(c) Select the  **Open Profile Model** Icon and *Data Point* in *View 4* to place the profile.

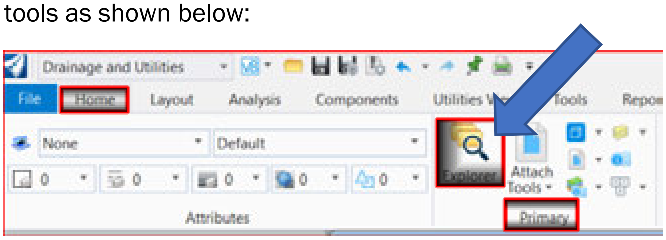


Note how the soffits match.

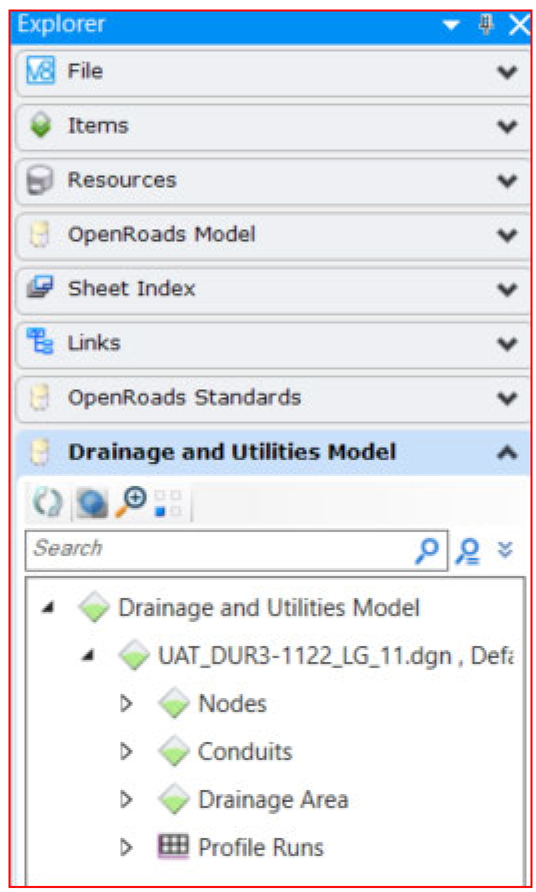
Up to this point we have showed you how to:

- create your drainage project, place your outfall,
- place structures with and without automatic catchment delineation,
- add gutters,
- add drainage areas and assign them to your structures,
- connect your structures with conduits
- add structures between conduits.
- Create profiles.

All the elements created on your model are classified by type and can be viewed in the project explorer under the drainage and utility Model tab as shown on the right. The project explorer can be found on the home tab under the Primary tools as shown below:



We will now show you how to set your rainfall parameters, compute your network and view the results.



Compute the System

In this section we will compute the system. No adjustments will be made. The software will simply compute the flows and provide you with a summary of the results for your review.

There are some settings you will have to go through to set up your storm event and location for your rainfall parameters. We will explain that in detail and show the location of the rainfall libs and how to add it to your scenarios before running the analysis

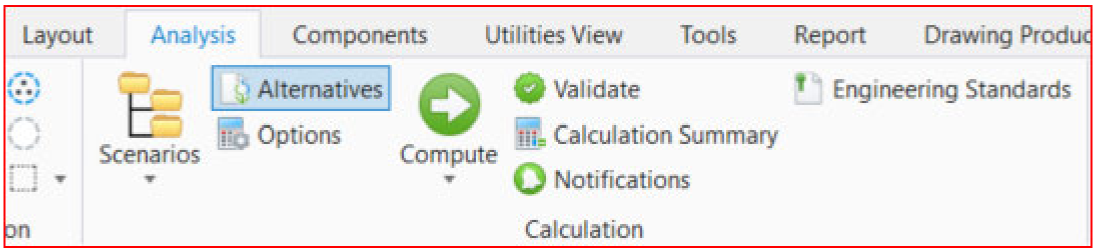
1. Setting up the storm event

(a) Leave the *Profile View* open.

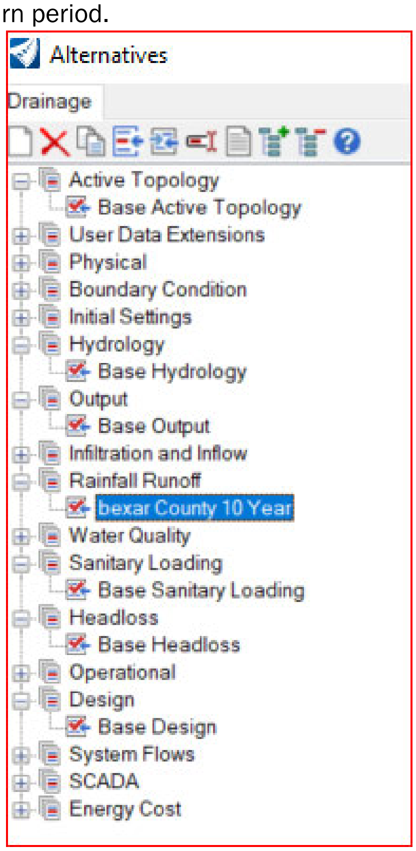
(b) Before running any computation, we need to set our storm event.

1) Select the workflow Drainage and Utilities, then select the Analysis Tab.

2) Select Alternatives from the calculations group.

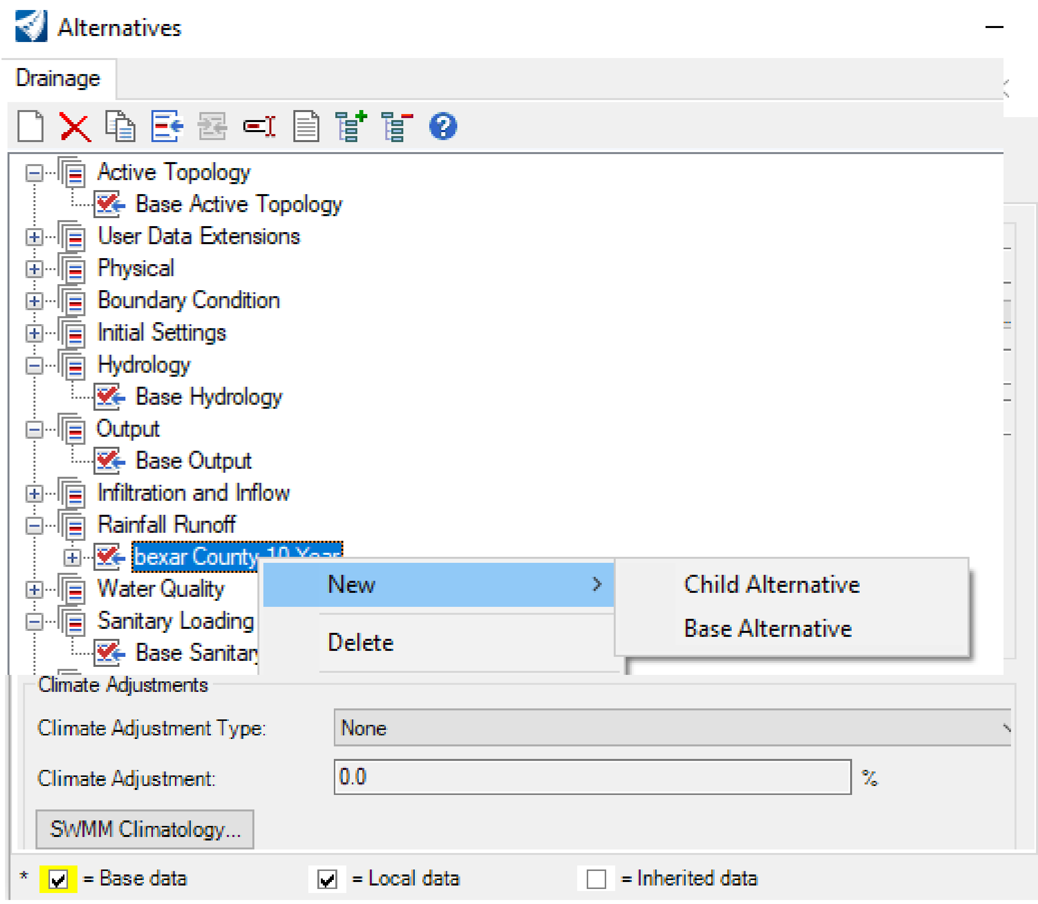


The alternatives window will open. Select the rainfall runoff and set the name to your desired location and return period.



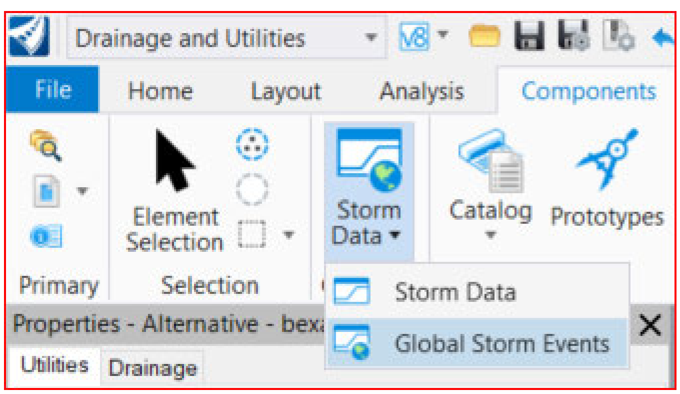
You can also create an additional alternative and set your preferences.

1) Right click on the Rainfall runoff and select the desired option. You can create a separate base alternative or a child of the existing. For this example, a child will be created.

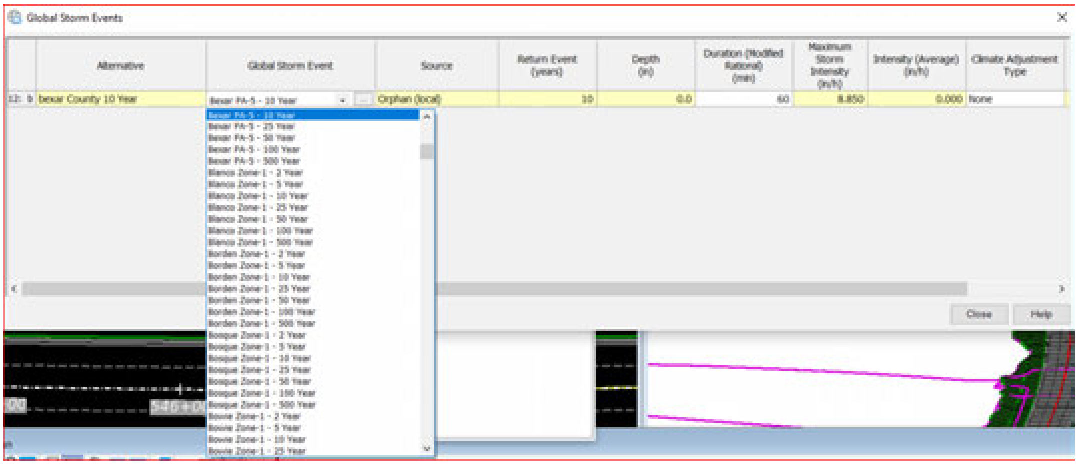


2) Double click on the rainfall runoff (Bexar County 10 year) and select the storm event.
3) Close the alternatives window.

You also could Select **Storm Data** from the **components tab** and select **Global Storm events** from the drop-down menu.

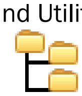


You will see your created alternatives listed. This way you can set up all your preferences for each scenario at one location.
Click on the global storm event column and select the storm event and location from the drop-down menu.
Close the global storm event window

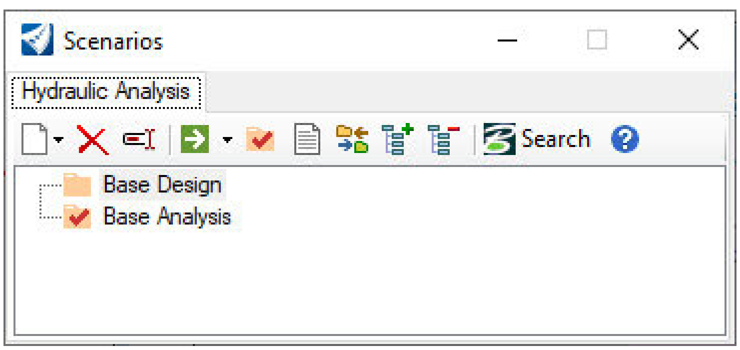


2. Run the Analysis Scenario – Compute your System

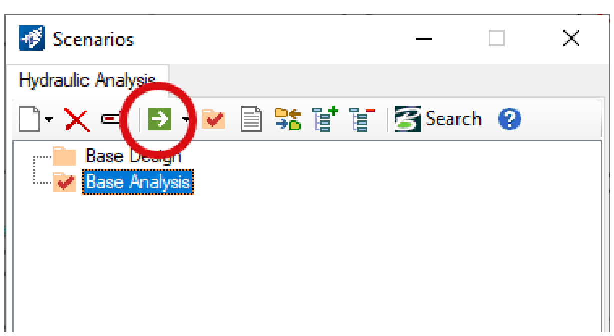
(a) Select the workflow Drainage and Utilities, then select the Analysis Tab.

(b) From the Calculation Group,  select the Scenarios Tool.

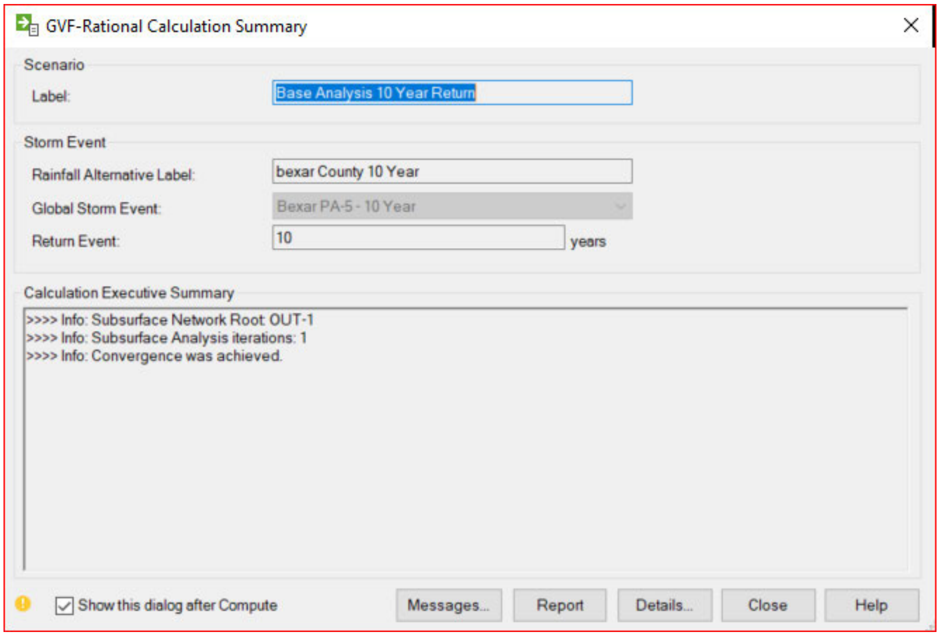
(c) Select the **Base Analysis** option. Right click and select properties



On the **Scenarios** dialog, click the green *Compute Scenario*  icon. Do not select Compute icon on the ribbon menu.

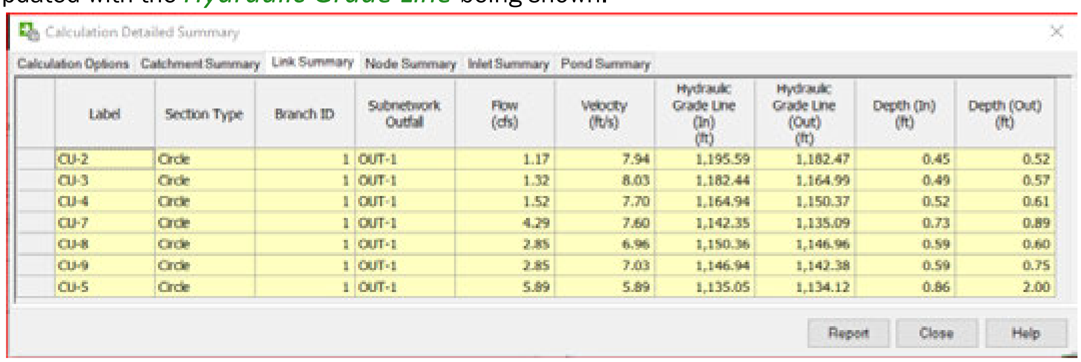


The hydrology and hydraulics calculations are carried out, and after a few moments, the Calculation Summary dialog above will open.

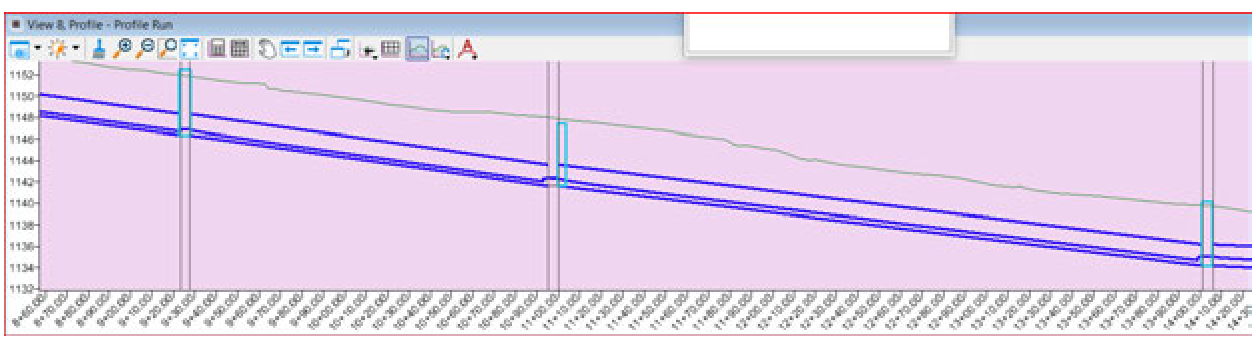


Note: *Convergence was achieved* is good news! Also, note that the dialog shows the Storm Event Properties.

(d) You can click the *Details* button to review the calculations. You also will notice that the profile is updated with the *Hydraulic Grade Line* being shown.



Label	Section Type	Branch ID	Subnetwork Outfall	Flow (cfs)	Velocity (ft/s)	Hydraulic Grade Line (In) (ft)	Hydraulic Grade Line (Out) (ft)	Depth (In) (ft)	Depth (Out) (ft)
CO-2	Circle	1	OUT-1	1.17	7.94	1,195.59	1,182.47	0.45	0.52
CO-3	Circle	1	OUT-1	1.32	8.03	1,182.44	1,164.99	0.49	0.57
CO-4	Circle	1	OUT-1	1.52	7.70	1,164.94	1,150.37	0.52	0.61
CO-7	Circle	1	OUT-1	4.29	7.60	1,142.35	1,135.09	0.73	0.89
CO-8	Circle	1	OUT-1	2.85	6.96	1,150.36	1,146.96	0.59	0.60
CO-9	Circle	1	OUT-1	2.85	7.03	1,146.94	1,142.38	0.59	0.75
CO-5	Circle	1	OUT-1	5.89	5.89	1,135.05	1,134.12	0.86	2.00



(e) Close the *Calculation Summary* dialog when finished.

We will now let the software design the system. But first lets discuss a few differences to take into account when laying out and analyzing the system as opposed to letting the sofware design it.

3. Invert Values during Layout and Design

During *layout*, inverts are set via these parameters:

- Nodes' inverts are the Ground Elevation (Terrain) minus the Height of the Node (either two cell elevation points or the Default Height field - depends).
- Pipe inverts are set at the adjacent nodes' invert elevation.
- Ditch inverts are set at the adjacent nodes' invert elevation.

During a *design calculation*, object inverts may be adjusted by these parameters:

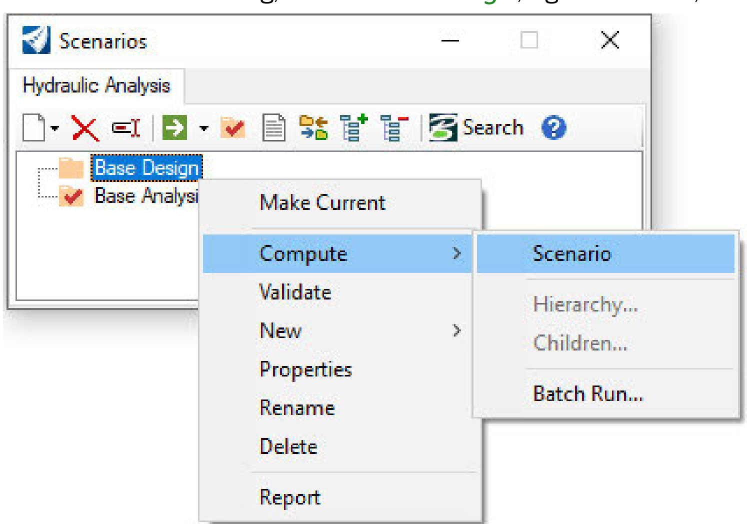
- Pipes will try to meet minimum cover, minimum slope, and minimum velocity criteria.
- These criteria are set by the Default Design Constraints and are assigned to each conduit at Layout.
- Each conduit end will lower or raise the adjacent nodes invert with them (excluding Outfalls).
- Conduit Channels behave like pipes: They try to meet minimum cover, etc. during design.
- Channels from Node do not get adjusted during Design.
- Outfall Inverts do not change during Design.
- Cross Section Nodes do not change during Design.

4. Let the Software Design the System

In this section, we will adjust a few settings and allow the software to design the system for us. We will allow the software to adjust Pipe Sizes and set Pipe Inverts for best system performance.

Note: When in design mode, the inverts cannot be manually adjusted/changed. To change invert elevations designer must switch to analysis mode.

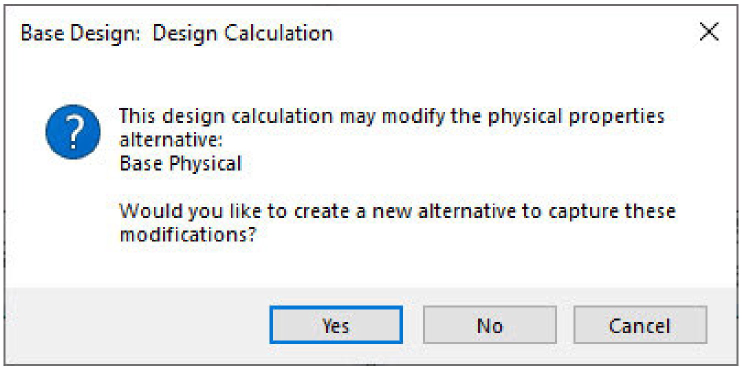
(a) In the **Scenarios** dialog, select *Base Design*, right click on it, and select *Compute > Scenario*.



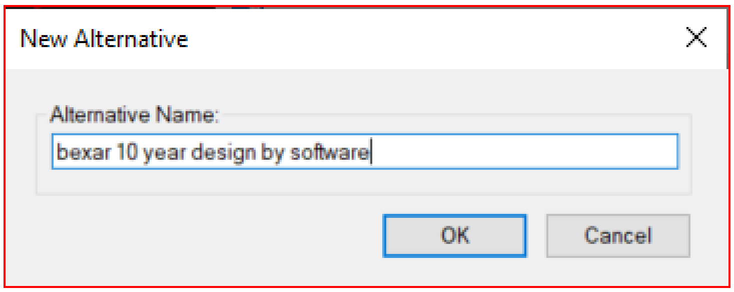
The software will begin to compute the scenario. The software recognizes that in a Design Calculation, Pipe Sizes, Invert Elevations, and other physical properties will change from your original layout.

The software stops the process to allow you to save these new sizes and elevations under a separate name of your choosing. Doing so makes it quite easy for you to return to your original data if you wish.

(b) *Click* the **Yes** button.

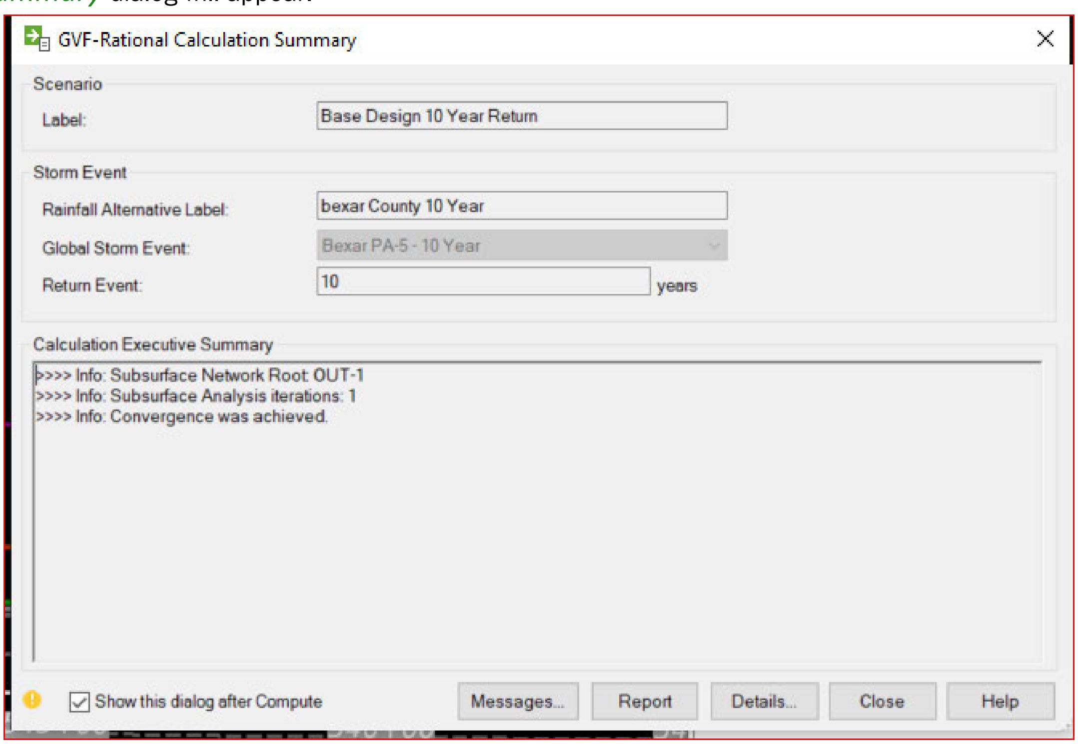


(c) Type in a name for the Physical properties alternative.
For example, key in Bexar 10 year Designed By Software.



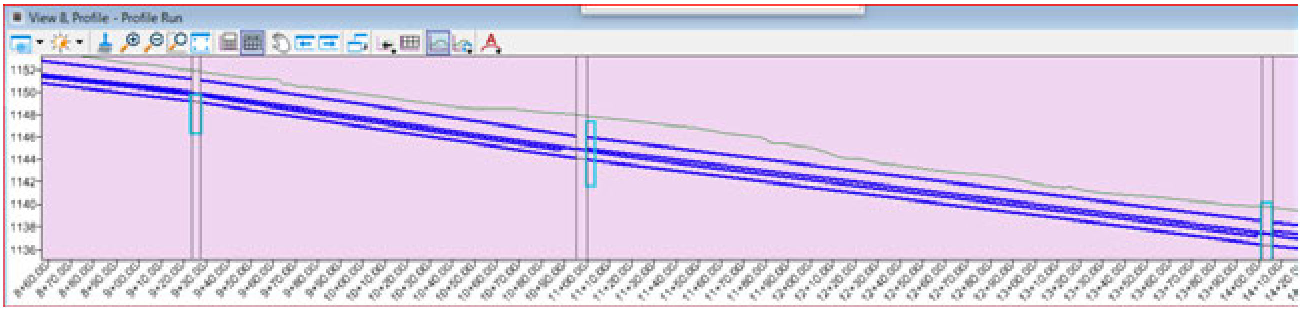
(d) Click OK

(e) The software continues to compute and optimize the design, and eventually the *Calculation Summary* dialog will appear.



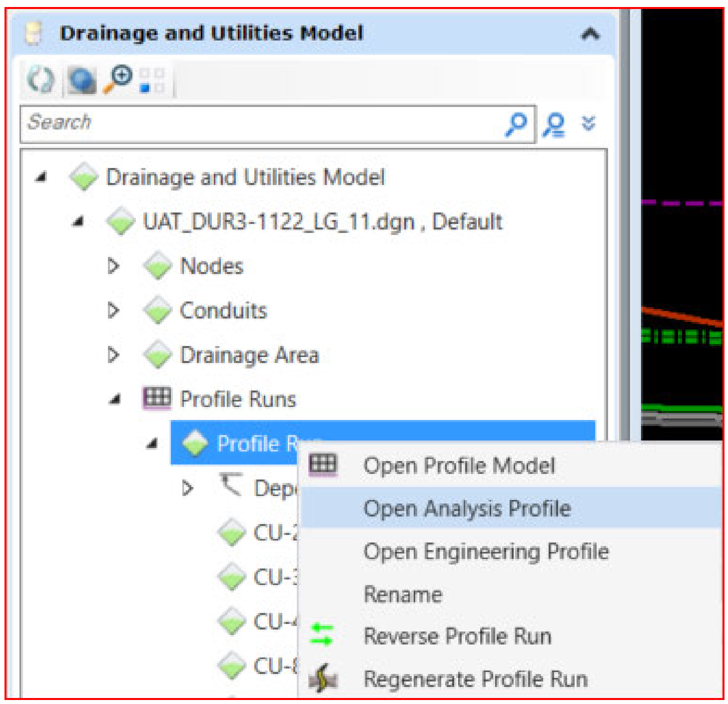
(f) Click the **Details** button to review the *Link Summary*.

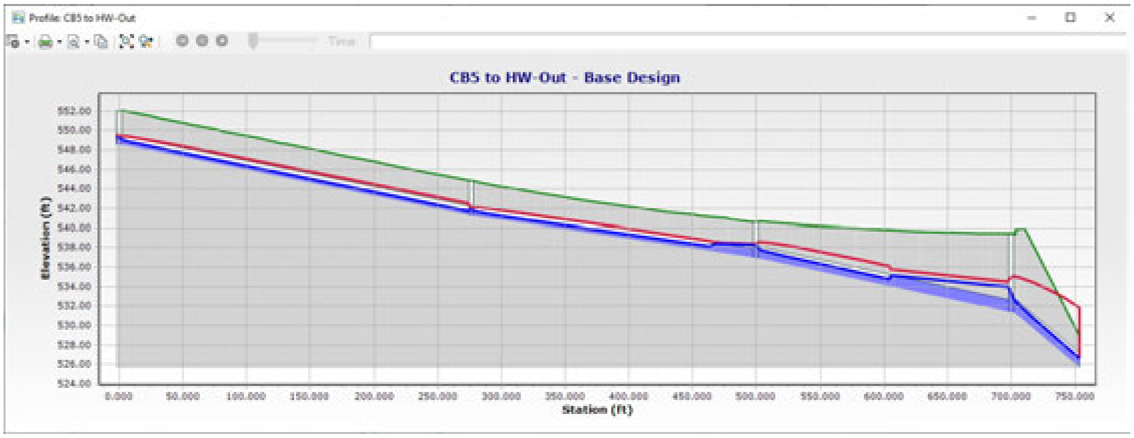
(g) Close the dialogs when finished.
The Profile View should still be open and has been updated to the current design scenario.



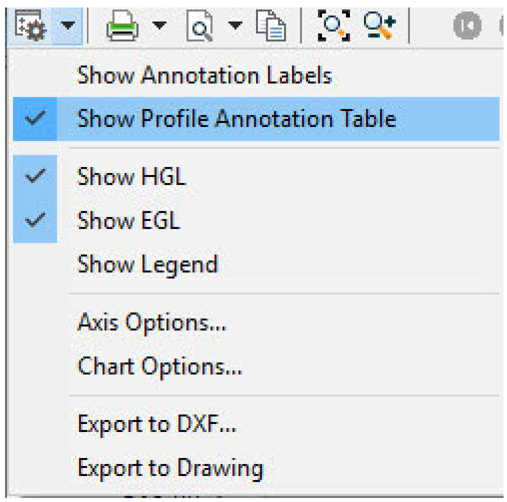
(h) Open the Explorer window and expand the Drainage and Utilities Model Tab.

(i) Expand the **Profile Runs** and *Right click* on the **Profile Run** profile and select **Open Analysis Profile**. This profile will display in its own window.

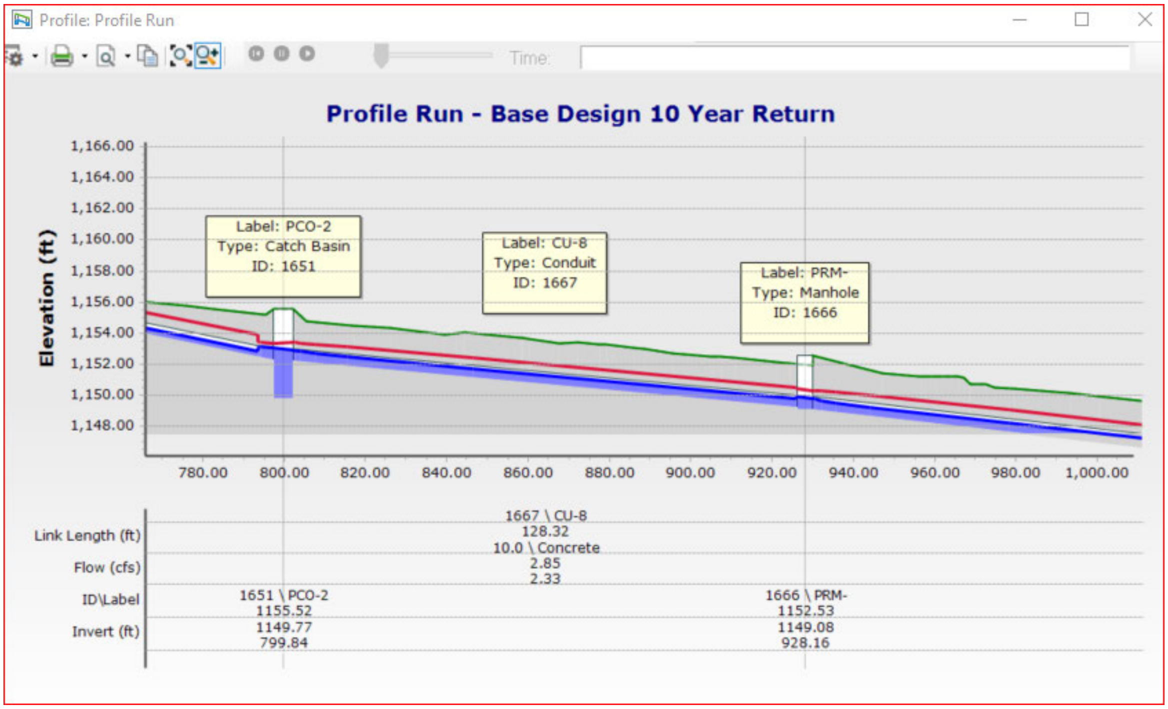




(j) Click the *Down Arrow*  next to the **Chart setting** icon in the upper left corner of the Profile window.
Select **Show Profile Annotation Table**



(k) Pipe information is shownalong the bottom of the Profile view window.



(l) **Close** the Profile analysis window when finished.

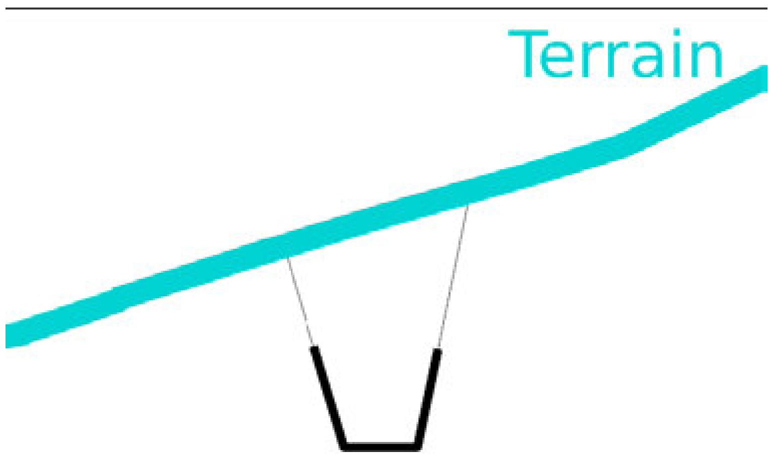
In the next section we will discuss how to create a ditch network and how to add and design a cross culvert.

Placing a Ditch and Culvert Network

We will now show the workflow to place a ditch and culvert network. Workflows are like the ones used to create a pipe and inlet network so we will focus only on the minor difference to set up your ditch system.

While Ditch systems can be analyzed using ORD DU, we will use this workflow only as a graphical tool to display ditch elements in profiles. Separate capacity calculation must be run to design ditches utilizing the same method used prior to ORD DU. The TxDOT workspace supplies a limited number of ditch sections but allows the designer the ability to create as many ditch sections as needed. This is discussed later under the **Place the ditch** workflow. The method used by the TxDOT workspace assumes no ditches exist in the terrain.

- **Terrain without Ditches** (or *Invert Below Grade*).
- The ditch bottom or even the ditch itself is NOT reflected in the terrain model.
- The terrain does not model the ditch. A primary example is a new proposed ditch to be placed on existing ground.



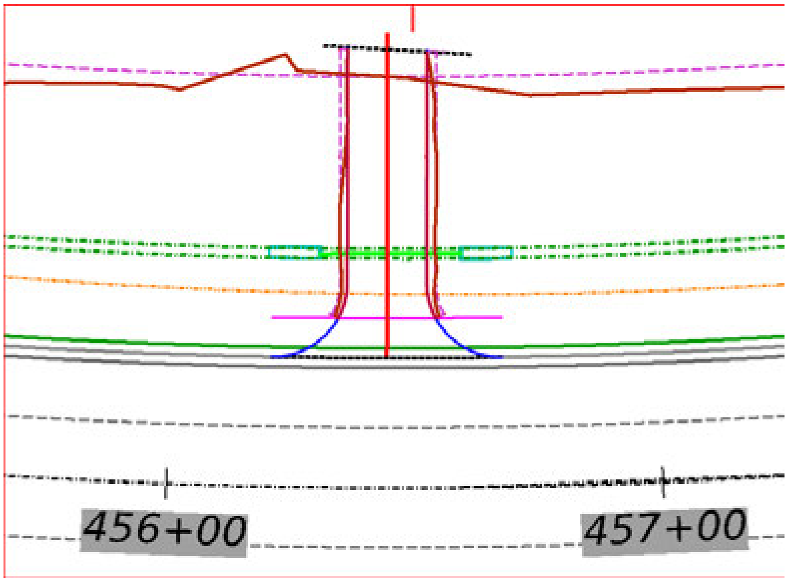
Note: While the focus is on the ditch, the inverts are dictated by the upstream and downstream Nodes (Endwalls, Cross Section Nodes, etc.).

1. Place the Headwall and Endwall

We will be placing a pipe under a driveway near the property line.

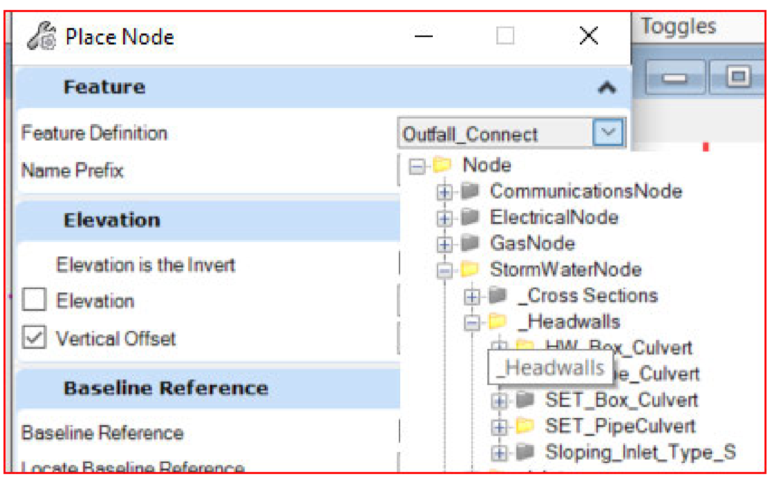
Later we will place ditches upstream and downstream of the driveway.

All Conduits must attach to Upstream and Downstream Nodes. The nodes must be placed prior to placing the culvert. Given the upstream and downstream ditches, the right nodes are SETs. We will place them now.



Note that the software has a Hydraulic category for Headwalls. "Endwalls" and Headwalls are identical as far as the software is concerned.

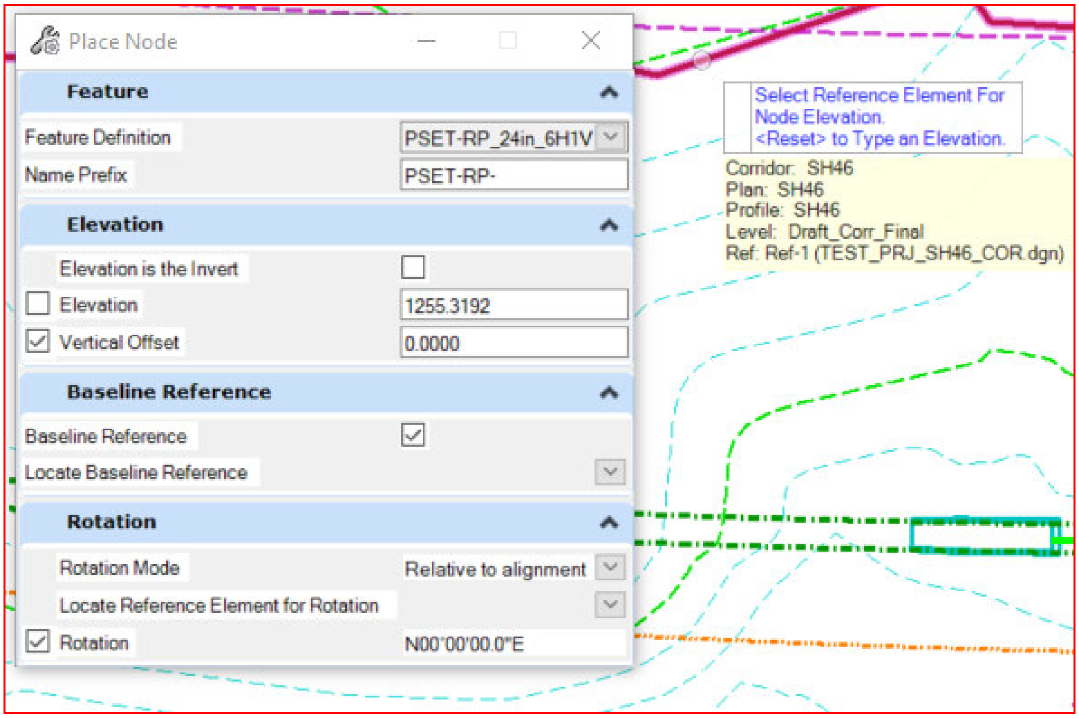
(a) Select the *Layout* > *Layout* > **Place Node** tool. Feature Definitions are how we choose the specific object type. The TxDOT Workspace has a variety of Headwalls and SETs under *Node >StormWaterNode >***Headwalls.**



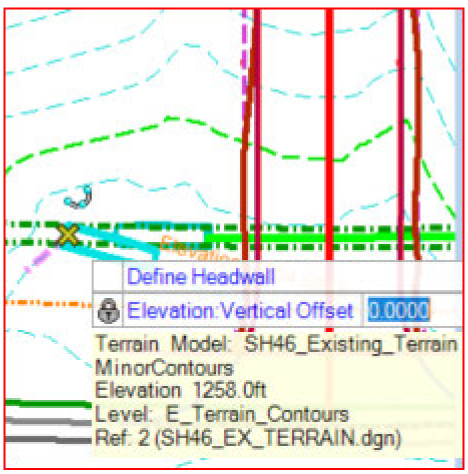
(b) In the *Place Node* dialog,

1) Set the *Vertical Offset* to **0.0**. Ensure the box is checked.
2) Set the *Rotation Mode* to **Absolute**, **clear** the *Rotation* check box.
3) Click the Feature Definition list.
   Feel free to see how the workspace Headwalls are organized.
4) Select Node > StormWaterNode > Headwalls > PSET-RP_P-D_Round >**select the SET**.

(c) At the Select Reference Element for Node Elevation prompt, select your corridor.

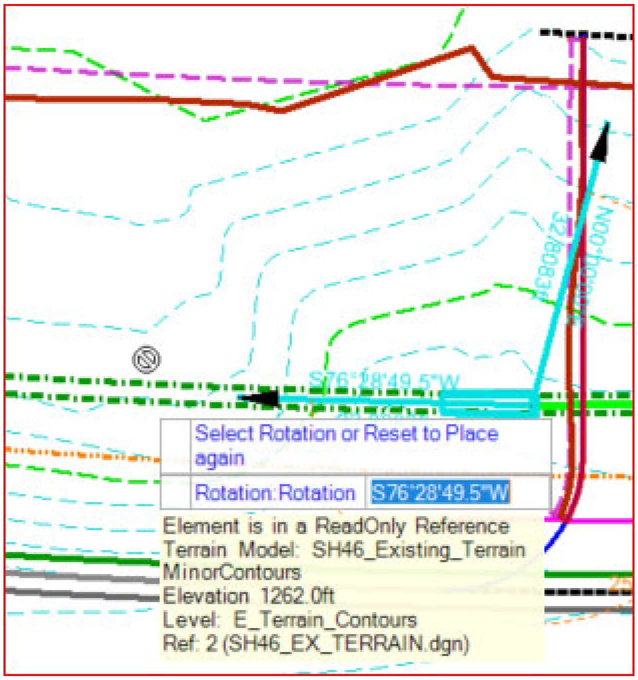


(d) At the **Define Headwall** prompt, ensure that the Vertical Offset is zero.



(e) Find a DW on your Roadway file and data point to locate the SET.

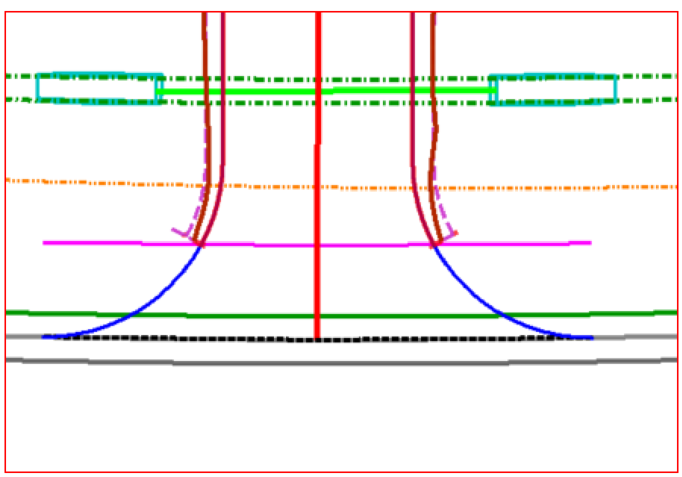
(f) Place a **datapoint** to confirm the *Rotation Mode* as *Relative to Alignment* (or press the Down Arrow key to change it to Absolute).



(g) At the *Select Rotation or Reset to Place again* prompt, drag the cursor to rotate the headwall along the path.
Note: If there is a Lock icon to the left of the Rotation field, click the **End** key to free the rotation. Rotation does not need to be exact.

(h) Place a datapoint at the intended rotation.This places the Headwall and the command loops to the Define Headwall prompt.

(i) Place another Headwall on the other side of the driveway, similar offset, similar orientation.



2. Place the Pipe

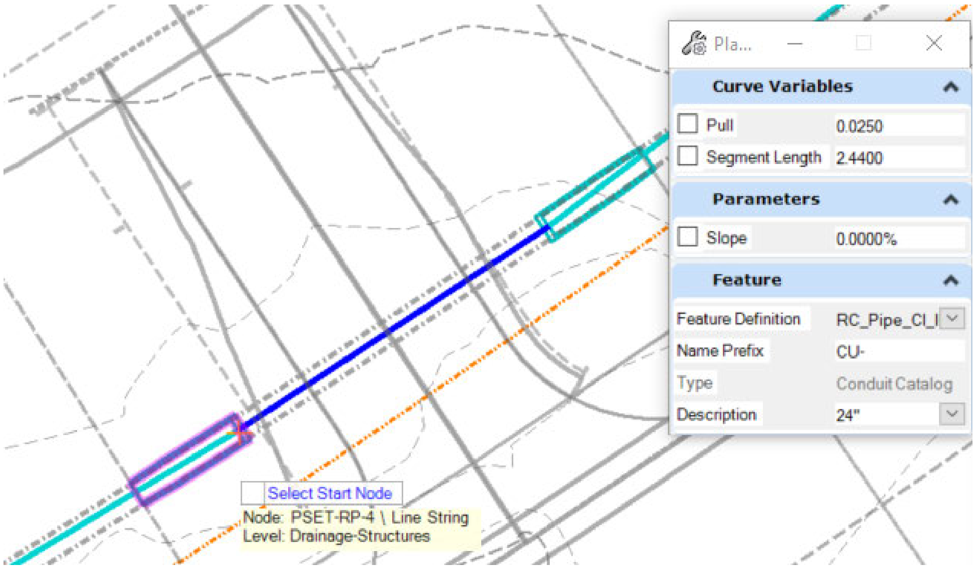
Place the pipe between the headwalls. The difference between a “pipe” and a “culvert” is the hydraulic method. The difference between a software “pipe” and a software “culvert” is a field called “Is Culvert?” which triggers the proper calculations.

You can set up Feature Definitions for Culverts by setting the Prototype's Is Culvert? field to True. By convention, however, all the Workspace pipes are Pipes rather than Culverts. Unless you change a Pipe to a Culvert, the calculation will be Pipe Calculation. We will show the culvert workflow in more detail later.

(a) Click the **Place Conduit**  tool from the Drainage and Utilities Workflow >Layout > Layout.

(b) In the Place Link Between Nodes dialog, set the Feature Definition to Conduit > StormWater> RC_pipe_Cl_III.

(c) For *Description*, select **24".**



(d) At the *Select Start Node* prompt, click on the upper headwall.

(e) At the *Select next node to make a connection* prompt, click on the lower headwall. The command loops.

(f) **Reset** to exit the tool.
Notice that you were not prompted for invert elevations. Pipe elevations are based on the inverts of the nodes and the Cover Depth as defined in the Default Design Constraints.

The Pipe Inverts match the SET Inverts.

Note: Although SETs are under the Headwall category, SET cells do not have a top locator point, so when placing SET structures, your ground elevation is the invert elevation, or the FL of your ditch.

3. Place the Outfall

All Conduits - pipes, ditches, etc. - need Nodes at both ends. Now we will place a downstream node at the Outfall location.

There is no necessary *physical* difference between a Headwall/Endwall and an Outfall. There are some subtle differences in the *hydraulic* properties of a Headwall/Endwall vs Outfall. An outfall can accept a wider variety of hydraulic Boundary Conditions and must be used at the end of your system. They are useful when routing ponds, for example.

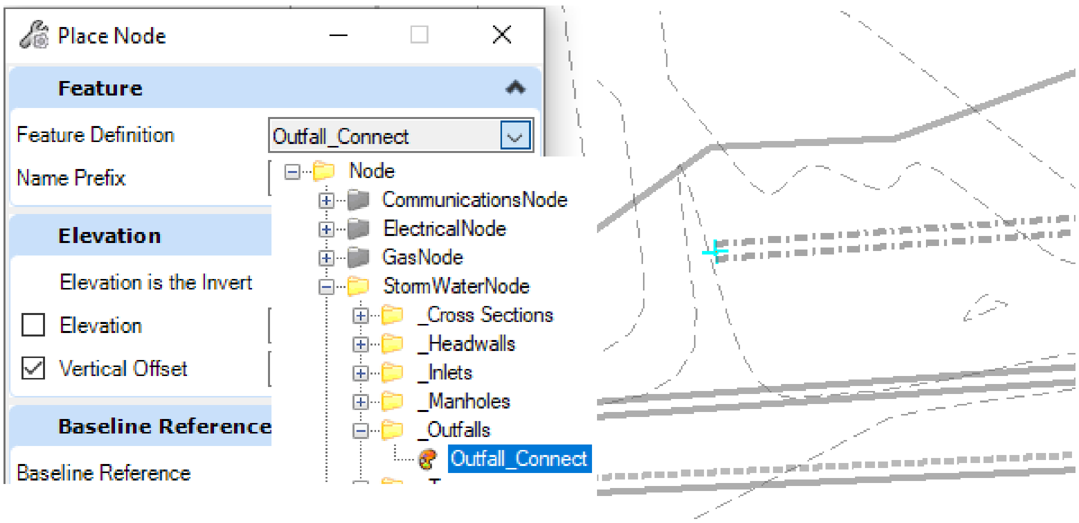
A given physical object may be used as either a Headwall/Endwall Feature or as an Outfall Feature, or both.

Due to how they link to different Hydraulic Prototypes, you will want to be clear in your workspace which type of object it is hydraulically:

The TxDOT Workspace has one generic Outfall under its own Outfalls heading. Objects under this heading use the Outfall Hydraulic Prototype. This manual has a special workflow to place an outfall and to connect the outfall structure. It is explained in more detail under the **SPECIAL DU WORKFLOWS SEGMENT.**

(a) Click Drainage and Utilities > Layout > Layout > Place Node.

(b) In the Place Node dialog, set the Feature Definition to Node > StormWaterNode > Outfalls > Outfall_Connect.



(c) At the *Select Reference Element for Elevation* prompt, pick any visible terrain element (a nearby contour).

(d) Define Outfall > Vertical Offset > **0.**

(e) At the *Define Outfall* prompt, place the Outfall Node at your outfall location. The primary variable here will be the elevation, which will affect the pipe slope. We'll explore and adjust this later.

(f) You then are prompted to *Select Rotation Mode*. Place a datapoint **< D >** to confirm **Absolute**.

(g) At the *Select Rotation or Reset to place again* prompt, place a datapoint **< D > i**n the ditch so that outfall is rotated in line with the pipe.

4. Place the Ditch

Now that we have an upstream node and a downstream node, we can place a ditch.

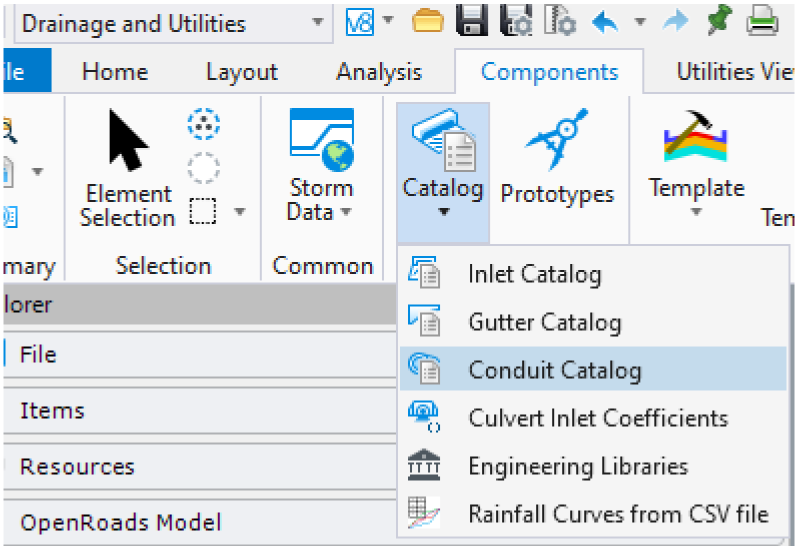
Note that a fundamental property of a ditch is its Cross Section Geometry. Ditch Cross Sections can be Rectangular, V-shaped, Trapezoidal - or any shape via Irregular or User Defined shapes.

When laying out Ditches, be aware of the shape you pick. Clearly named Feature Definitions are important.

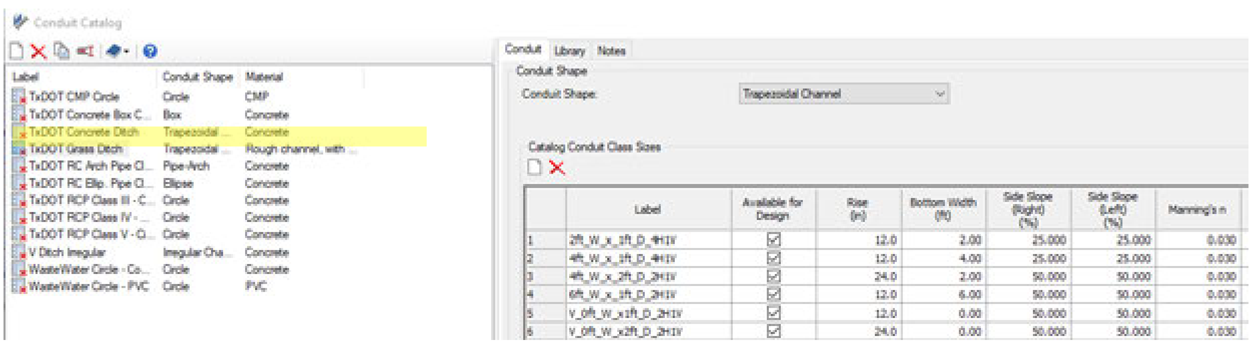
Only a few ditches Feature Definitions are provided in the TxDOT library. It is easy to add more cross sections as needed. We will now show you how.

(a) To review the available items catalog and add new items;
   1) Click Drainage and Utilities >Components > Catalog >

2) Select Catalog>Conduit Catalog from the drop down.

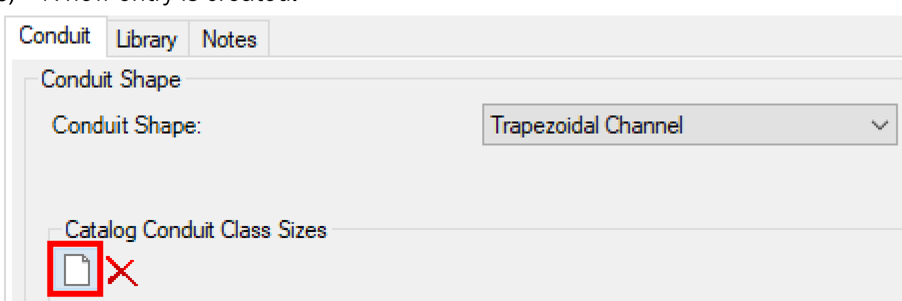


3) Select the *TxDOT Grass Ditch* label.



4) A list of available ditch sizes is provided. To add a new ditch to the catalog, select the conduit shape, and click on new under the Catalog of Conduits Class Sizes heading.

5) A new entry is created.



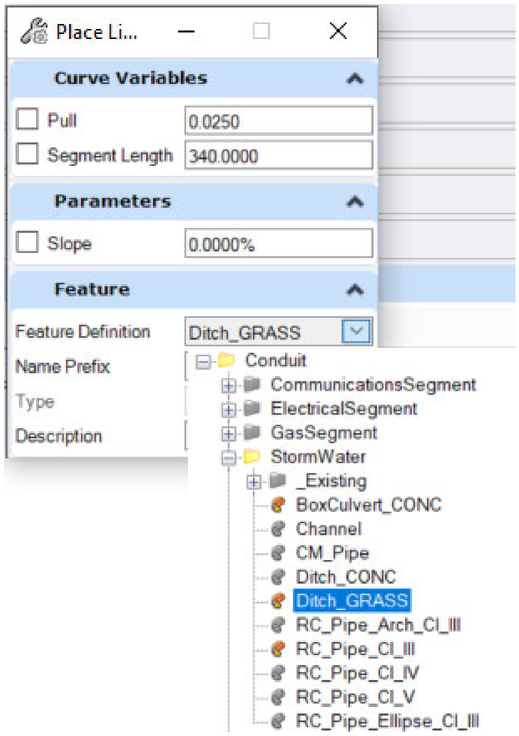
6) Select the label (for example: 2ft_W_x3ft_D_6H1VL 4H1VR).

7) Check the available for design box and set all the physical properties of your ditch.

8) Close the conduit catalog window.

(b) We will now place the newly created ditch section:

1) Click Drainage and Utilities > Layout > Layout > **Place Conduit**. Set the Feature Definition to Conduit > StormWater > Ditches **>Ditch (Grass)**.



2) Click on the Description list. This defines the Cross Section geometry.
Feature Definitions that are linked to a Catalog - like this one - will list their available sizes. You will notice our new added ditch is now available.
3) Click 2ft_W_x3ft_D_6H1VL 4H1VR.
This naming convention indicates the width, the height, and the side slopes.
4) At the *Select Start Node* prompt, click the upstream node.

Hint: You do not have to be zoomed in tight. The software will accept only graphics right for ditches: Headwalls or Cross Section Nodes.

5) At the *Select next node to make a connection* prompt, click the downstream node.
The ditch is placed, and the command loops.
6) Select **< Esc >** to exit.

5. Break the Ditch

Our ditch nodes are far apart, and the straight line between them diverges from the property boundary. You can place a bend in any conduit during layout or afterwards, but they are horizontal bends only:

- During placement, prior to picking the stop node, you can click the **<Ctrl>** key to place vertices. Click **<Ctrl>** again to pick the stop node.
- After placing a ditch, you can use **Bend Link Segment** from the ditch's context menu.

These have a single slope and single "hydraulic fields" (a single flow - Q, velocity, depth, etc.).

If you need a grade break or need to add flow, you will need to insert a node. This will break the conduit/ditch into two, each with full database properties.

For our ditch, we will insert an intermediate node. This way, we can follow the ditch from the roadway model more closely and have a different slope as we approach the outfall.

"Structure-less" Ditches

Ditches are different from pipes: Pipes always are physically quantifiable objects.

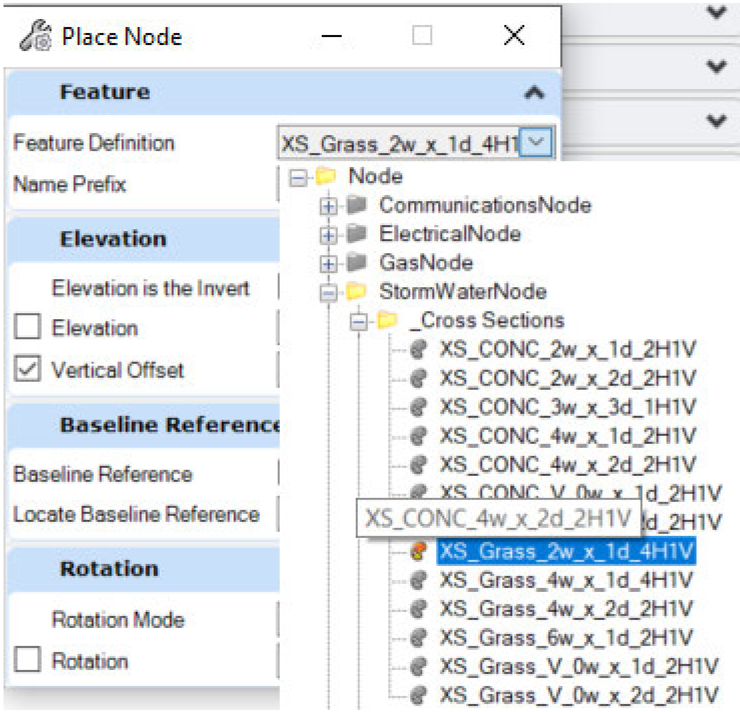
Ditches can be physical objects, such as a concrete-lined trapezoidal ditch. Natural ditches and new "dirt ditches" are not physical objects that show up on plans and in quantities.

Cross Section Nodes allow you to place start/end nodes and intermediate nodes without adding "physical objects." You get the ability to:

- Store x,y, z coordinates in the model,
- Accept added flow, and
- Change the ditch cross section.

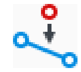
Our ditch is grass. Adding a bend is simply a matter of turning the bulldozer: no physical object at the bend is needed. A Cross Section Node is perfect for our needs.

The image below shows the workspace's Ditch Cross Section nodes. Note that the Feature Definition names include the Material and Cross Section shape.

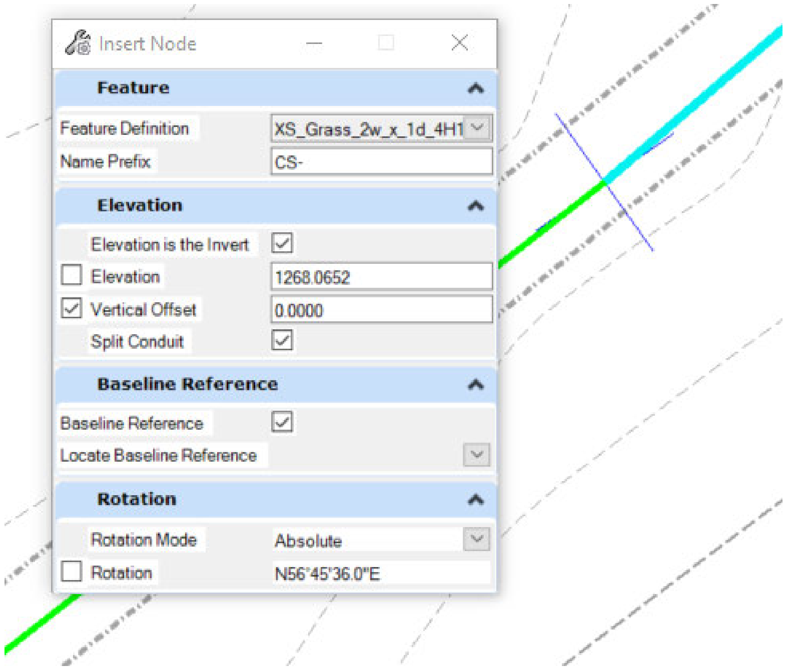


Note: A Ditch's shape is NOT bound by the Cross Section Node. We will discuss this later.

(c) Let's break our ditch into two by Inserting a Cross Section Node.



1) Click Drainage and Utilities > Layout > Layout > **Insert Node**.
2) Set the Feature Definition to Node > StormWaterNode >. XS_Grass_2w_x_1d_4H1V.



3) At the *Select Reference Element for Node Elevation* prompt, pick the roadway corridor boundary.
4) At the Select Conduit Invert Reference prompt, click on the ditch.
5) At the Split Conduit prompt, indicate Yes.
6) At the Define StormWater Node prompt, ensure that the Vertical Offset reads zero, then locate the node with a datapoint <D >.
7) The break location is located in the ditch. The node can be moved freely after it is placed.
8) You are then prompted to Select Rotation Mode. Place a datapoint < D > to confirm Absolute.
9) At the Select Rotation or Reset to place again prompt, place a datapoint < D > along the ditch.
10) Ensure the ditch node is at right angles to the ditch.
11) The Ground Elevation is the terrain. The Invert Elevation is the Ground Elevation, which is the same in this case.
12) Once your ditch is in place, use the Layout profile runs under the layout workflows to create your profile.
13) Use the Place catchment workflow to add catch basins to your ditch.

Note: There is a natural tendency to want to assign the drainage area to the node at the downstream end of a ditch. It drains in that direction, right? This is good for inlets, but not for ditches.

If you do that, then *none* of the flow gets into the ditch for calculations. Attach the Catchment to the upstream node of the ditch. This way, you get all the flow in the ditch for maximum flow, depth, and velocity values.

In reality, you get incremental interception of flow as you move from the upstream node of each ditch to the downstream node. If you need a more exact and "granular" flow design result, you can insert additional intermediate nodes and use smaller catchments to cover the areas. All these values decrease as you move upstream from the downstream end of the ditch.

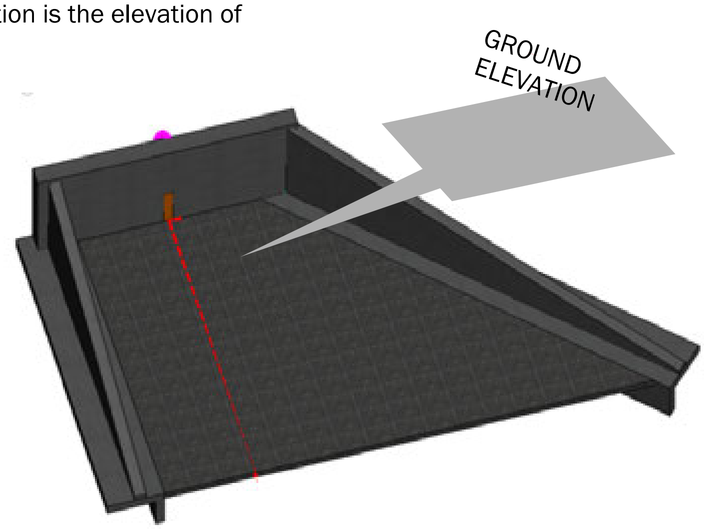
(d) Follow the analysis workflows for scenarios and alternatives to analyze and design your ditch.

6. Place and Design a Cross Culvert

In this section we will show how to place and design a culvert.

Note: The Elevation Reference – the terrain - is assigned to what is labeled in the Properties dialog as the Ground Elevation.

To be precise, for all Headwall cells, the Ground Elevation is the elevation of the top Locator Point of the cell.



The Vertical Offset set in the dialog/Heads Up Display is the difference between the Ground Elevation Point and the Elevation Reference (the Terrain).

The Invert Elevation is this Ground Elevation minus the height of the Headwall, either:

- The Default Height of Headwall - as defined in the headwall's Prototype, OR
- The Elevation Difference between the top and bott m elevation indicators in the Headwall 3D Cell definition. Th is explained in detail in other documentation.

For example, if the Height of the Culvert Headwall used is and the Terrain is at 100 ft, the Invert Elevation would be 97 ft.

Make the Pipe a Culvert

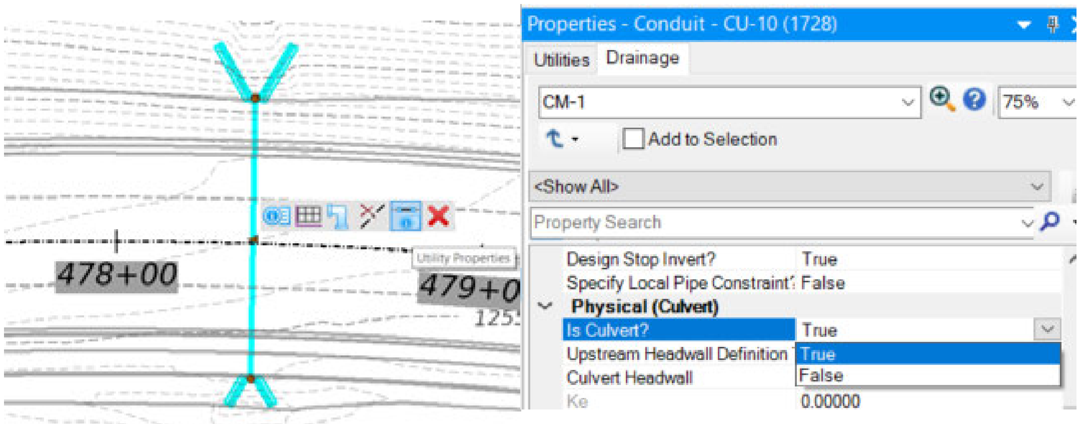
We placed a pipe. If we do nothing, it will be calcula n is fine for most applications. It is likely that the flow through it does not require "Culv calculations." For this example, we will place two Headwalls and a pipe and set it as a culvert for calculations.

There are two parts to Culvert Calculations:

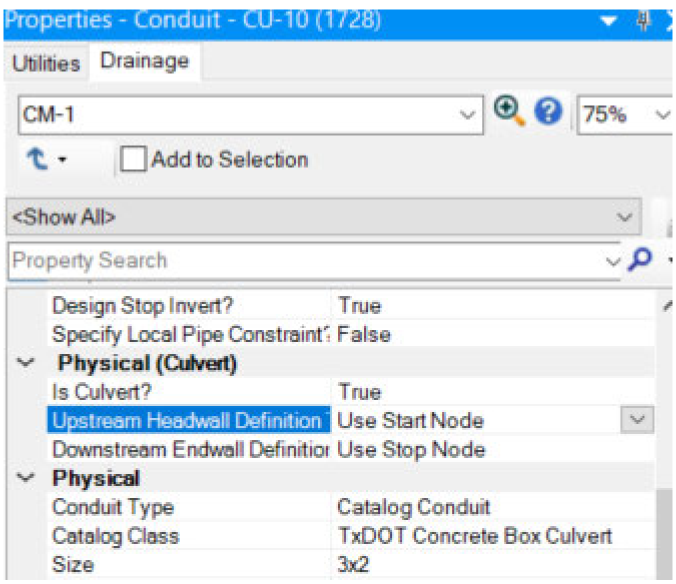
- Set *Is Culvert?* to **True**
- Ensure the proper Culvert Coefficients for the Calculations.

(a) Select the pipe, hover, and open *Utility Properties* from the context menu.

(b) In the *Drainage* tab, find the property for *Is Culvert* and change to **True**. This opens other fields.



Culvert Calculations need coefficients describing the hydraulic properties of the ends of the culvert. An Upstream Headwall Definition and a Downstream Headwall Definition are needed.



The options for each are

- *Use Conduit* - Define the values here in the dialog or
- *Use Node* - The attached Headwall has coefficients defined in it. Selecting this option reads those coefficients.

(c) TxDOT Workspace Headwalls have coefficients defined. Set both fields to **Use Node**.

(d) You will now set your area, for the inet headwall, and run the analysis or design scenario as needed as shown in the **Compute the system** section.

We have so far show you how to create and analyze or design your drainage system. In the next few sections, we will discuss valuable information that will aid in your design process.

Any object can be "locked" via the Design Alternative settings (see next section).

Controlling Object Design

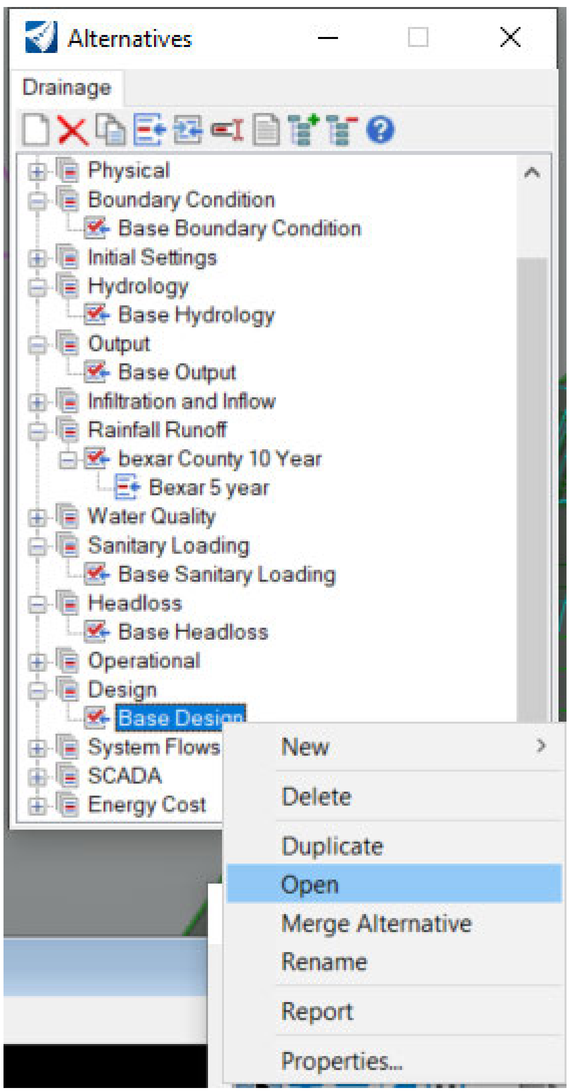
One sure way to ensure that no object properties get adjusted unintentionally is to perform an *Analysis Compute Option*. An Analysis computation routes the flow through the system without changing any of the physical object properties.

Sometimes we want to have the software size only *some* of the objects, rather than *all* the objects. During placement, an object's database fields are populated from the Feature Definition, the Hydraulic Prototype, and the *Default Design Constraints* (Drainage and Utilities > Analysis > Analysis Tools > Default Design Constraints).

You can view each object's data record using the *Utility Properties* dialog.

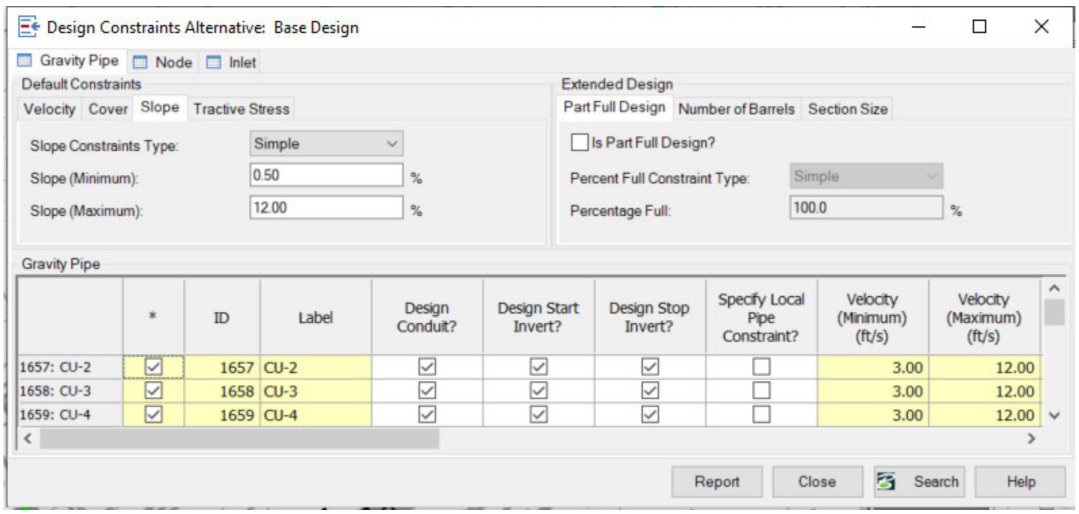
Alternatives are logically organized based on purpose. This helps reviewing and editing multiple objects targeting specific characteristics. This will be explained in more detail later

Design Alternatives group objects and properties related to design behavior. To open the files Design Alternatives, click *Drainage and Utilities* > *Analysis* > *Calculation* > **Alternatives**.



A Conduit Ditch's Feature Definition can set a Local Design Constraint to override the global Default Design Constraints. You can also set after objects are placed in the Constraints in the Design Alternatives dialog.

Design Conduit allows for automatic sizing of closed conduits only. Open Channels are not sized automatically.



Design Start or *Stop Invert* allows the software to change the invert elevations. Clearing the check box will "lock" the elevation.

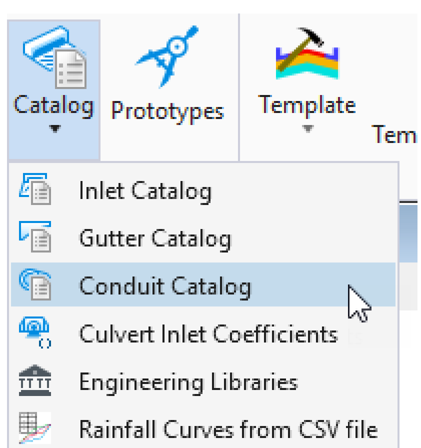
Note that no *Channels from Node* are shown here; they are "Channel Type" elements. This dialog shows "Conduit Channels." Neither are "designed" by the software - the software does not size them.

Please refer to the workflow on Pipe Networks for detailed Scenario Management and Computation Instructions and Troubleshooting.

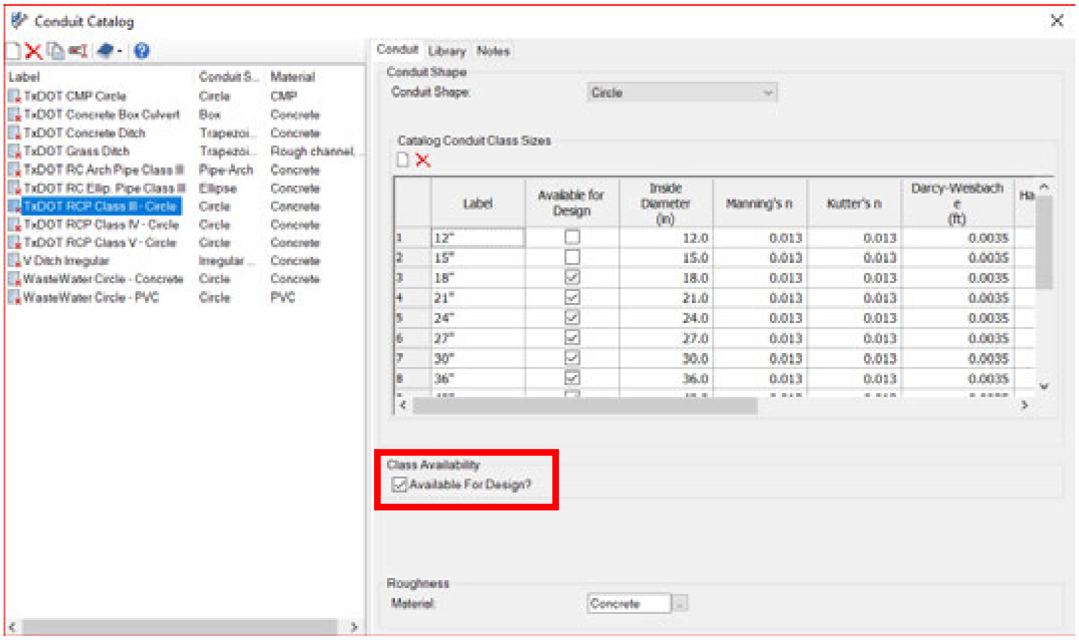
1. Pipe Sizes available for Design

(a) In the workflow **Drainage and Utilities** switch to the **Components** Tab.

(b) In the *Catalog* Group click the down arrow under the *Catalog* icon and select **Conduit Catalog**.



(c) The *Conduit Catalog* will open, and we will select **TxDOT RCP Class III - circle** under the *Label* column.



(d) In the *Available for Design* column, notice the options for **12" and 15"** sizes are *toggled* **off.**

There is a second **Available for Design** check box, further down the dialog, in a frame called **Class Availability**.

Clearing this check box would mean that the complete class would not be considered when the software calculates the size of conduit that is needed. This might be right if you have a class that is purely for some existing conduits, which are not available to use anymore.

NOTE: While all pipe sizes are available for use, the minimum pipe size for TxDOT design is 24". For details, please refer to the TxDOT Hydraulic design manual. 18" may be used in some cases with the engineer's approval.

(e) **Close** the Conduit Catalog.

2. Conduit Properties

On the **Tools** ribbon, select **General Tools > Update Descriptions**. This will update the conduit descriptions, which by default come from the prototype so they reflect the conduit diameters that you chose when placing the conduits. These descriptions are visible in the **Utility Properties** of each conduit.

(a) Click OK on the message which confirms that the update is complete.

(b) Select a conduit and open **Utility Properties.**

(c) **Note** the **Physical** Set Invert to Start? and Set Invert to Stop? Are both set to True? *True* means that the invert elevations of the conduit will be set to the invert elevation of the connected nodes.

The Conduit Description is also shown in the **Physical** category.

The default values for these properties were set in the **Prototype** that the conduit feature definition uses.

Understanding Scenarios

- There are hundreds of properties involved in hydraulic design and calculations.
- Regulatory agencies often require analysis on multiple variations of constraints (such as multiple storm events).
- The engineering design process itself often requires evaluation of alternative solutions.

Having a logical and manageable system to manage and compare the variations in the hundreds of properties is essential to ensure optimum design. This is what the effective use of Scenarios offers you.

Drainage and Utilities groups comparable properties into function-based groups called Alternatives. Calculations are then performed on a bundle of Alternatives, which are grouped together in Scenarios. A Scenario also controls how the calculations are performed, in the Calculation Options.

- *Property* – A property is any stored characteristic of a model element. Examples include:
  - A single numeric quantity such as a pipe's Diameter, Length, or Roughness.
  - A Yes/No toggle such as *Design Conduit?*
  - A value from a list such as *Design* or *Analysis* for Calculation Type

- *Alternative* – An Alternative is a logically organized set of properties. Examples include:
  - *Physical* Alternative – groups physical data for the network's elements, such as elevations, sizes, and roughness coefficients.
  - *Design* Alternative – groups engineering criteria that will be applied during calculations such as velocity limits and other settings that may or may not be applied during calculations, such as adjusting pipe diameters and inverts.
  - *Rainfall Runoff* Alternative – allows different storm events to be used in calculations.

- *Calculation Options* – a Calculation Option contains properties that control how to 'solve' the hydraulics and hydrology of the drainage system. Examples include:
  - *Calculation Type* – whether the system is to be analyzed (does not change elevations or sizes) or designed (hydraulic optimization of elevations or sizes)
  - *Active Numerical Solver* – the methods used for the calculations, such as 'Rational Method,' and the settings that the method needs.

- *Scenario* – a Scenario has a set of Alternatives, and the settings that control how the hydraulics and hydrology are calculated. This "bundling" of Alternatives lets you easily generate system conditions that mix and match groups of data that have been previously defined. Note that Scenarios do not actually hold any attribute data – the referenced alternatives hold the data.

There is always a current scenario. It specifies the current alternatives. The current alternatives are where your data is stored.

You can use multiple Scenarios to calculate multiple "What If?" situations in a single project file. You can try several designs and compare the results or analyze an existing system using several different input alternatives and compare the results.

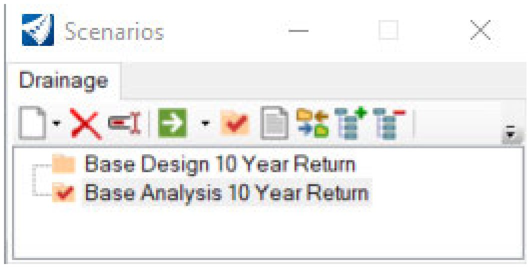
Scenarios and Alternatives can "inherit" properties from other Scenarios and Alternatives. These Parent-Child relationships are a critical tool in easily managing variations from otherwise global properties.

When creating a new project, Scenarios and Alternatives are copied from the Hydraulic Seed File. This is a DGN Library, which is specified in the WorkSpace or WorkSet configuration.

1. Scenario Manager

The Scenario Manager is the primary interface for creating, editing, and managing an unlimited number of Scenarios.

There must be at least one Scenario. Additional Alternatives and Scenarios are easily created to handle any design requirements.



Examples include:

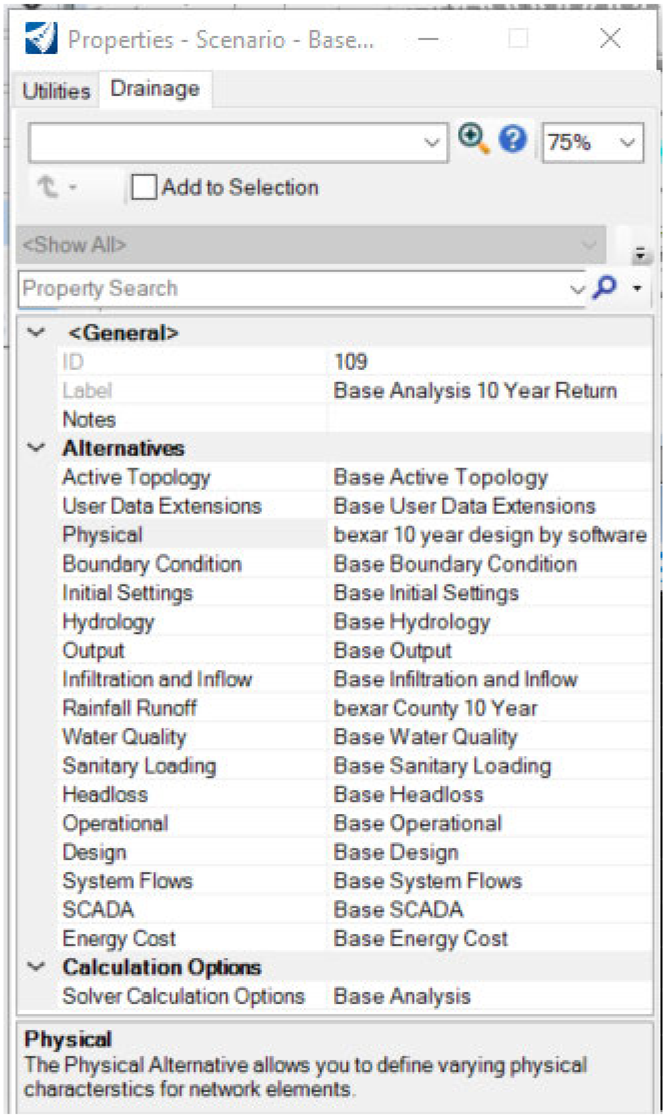
- Competing physical layouts may be managed by different Physical Alternatives.
- Different Storm Frequencies may be managed by different Rainfall Runoff Alternatives.

There are two types of Scenarios: Base Scenarios and Child Scenarios.

- Base Scenarios have all your working data. When you start a new project, you begin with a default Base Scenario. As you enter data and calculate your model, you are working with this default Base Scenario and the alternatives it references.
- When a Child Scenario is created, it inherits its data from its Parent. More precisely, its Alternatives and Calculation Options are links to the Alternatives and Calculations Options of its parent.

2. `Scenario Properties

As stated above, Scenarios are simply a list of Alternatives and a Calculation Options definition. Viewing, changing, and managing the different alternatives associated with a scenario is done through the Properties dialog. To view the scenario properties, double-click on the scenario, or right-click on the scenario and choose Properties.



A list of the alternatives associated with the scenario will be displayed in the Properties dialog.

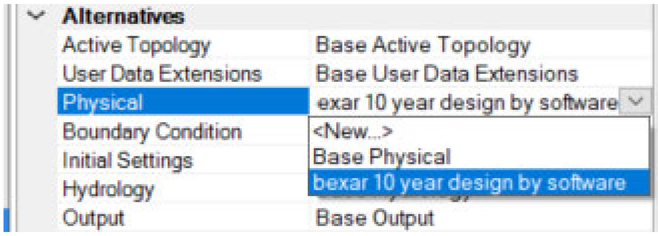
If you have created a new Base Scenario, all the Alternatives will be the same as in the "copied"

Scenario. They will be copied, but they will not keep any further relationship to the original Scenario's Alternatives. Changes to the original Scenario do NOT propagate to the new Base Scenario.

If you created a Child Scenario, the scenario initially would inherit all the alternatives from the parent scenario. In this case, you will see the "I" next to the name of the alternative. Changes to any of the settings in the original scenario – a change in the Rainfall Runoff Alternative, for example – automatically gets propagated to the new Child Scenario. They're linked.

If you manually pick an alternative without the "I," then the Child Scenario will no longer inherit any changes in alternatives made in the parent scenario.

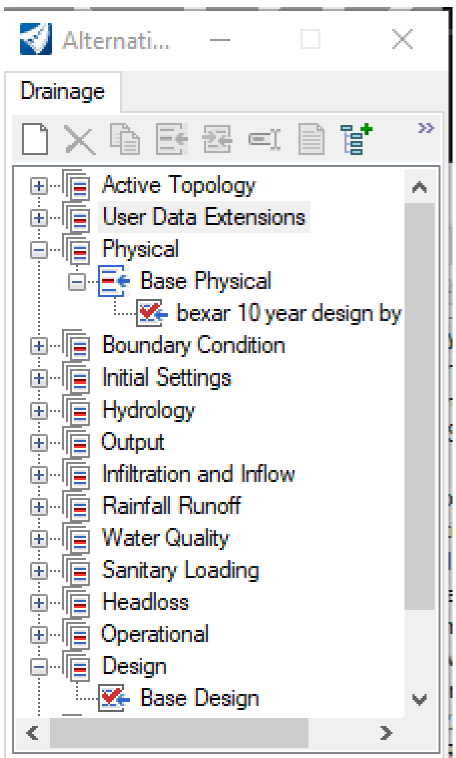
To change any alternative for a scenario, click the pulldown beside the scenario name and select the alternative.



If you have not yet created an alternative for the scenario, you can create a new alternative here as well, by selecting the “<New...>” item. You will be prompted to enter the name for the new alternative – after which the new alternative will be selected for the scenario and listed in the Alternatives Manager.

3. Alternatives Manager

The Alternative Manager lets you create, view, and edit the alternatives that make up the project scenarios. The dialog box consists of a pane that displays folders for each of the alternative types which can be expanded to display all alternatives for that type and a toolbar.



As with Scenarios, there are two kinds of Alternatives: Base Alternatives and Child Alternatives. Base Alternatives hold local data for all elements in your system. Child Alternatives inherit data from Base Alternatives or even other Child Alternatives. The data within a Child Alternative consists of data inherited from its parent and the data altered specifically by you (local data).

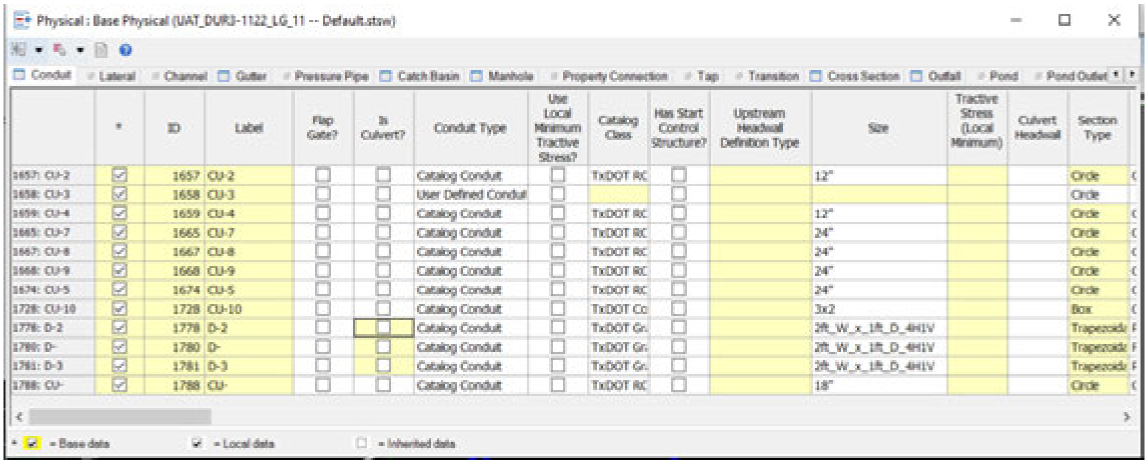
Remember that all data inherited from the Base Alternative is changed when the Base Alternative changes. Only local data specific to a Child Alternative remains unchanged.

4. Editing Alternatives

To edit an alternative, expand the tree so that all the alternatives for a given type are listed.

There are several ways to open an alternative:

- You can double-click on the alternative.
- You can also highlight the alternative and select the Open icon.
- Finally, you can right-click the alternative and select Open. This will open a new dialog.



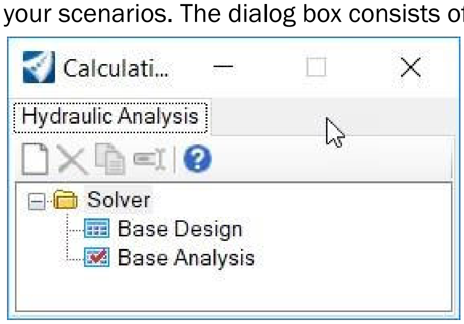
Each alternative will have different properties. Any column that is shown as white is a field that can be edited. Columns in yellow cannot be edited from the alternative, but in some cases, may be editable from other places in the model, such as the FlexTables or Properties.

The first column in any alternative editor has a check box indicating the records that have been changed in this alternative. If the box is checked, the record on that line has been changed, and the data is local, or specific, to this alternative. If the box is not checked, it means that the record on that line is inherited from its higher-level parent alternative. Inherited records are dynamic. If the record is changed in the parent, the change is reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

Changes made in the graphics, Properties, and FlexTables will automatically make changes to the values in the appropriate active Alternatives.

5. Calculation Options

The Calculation Options Manager lets you create, view, and edit the calculation options available for your scenarios. The dialog box consists of a pane that displays the calculation options created.



To edit the calculation options, double-click on the one you want to edit. This will display the properties of the calculation options in the Properties dialog.

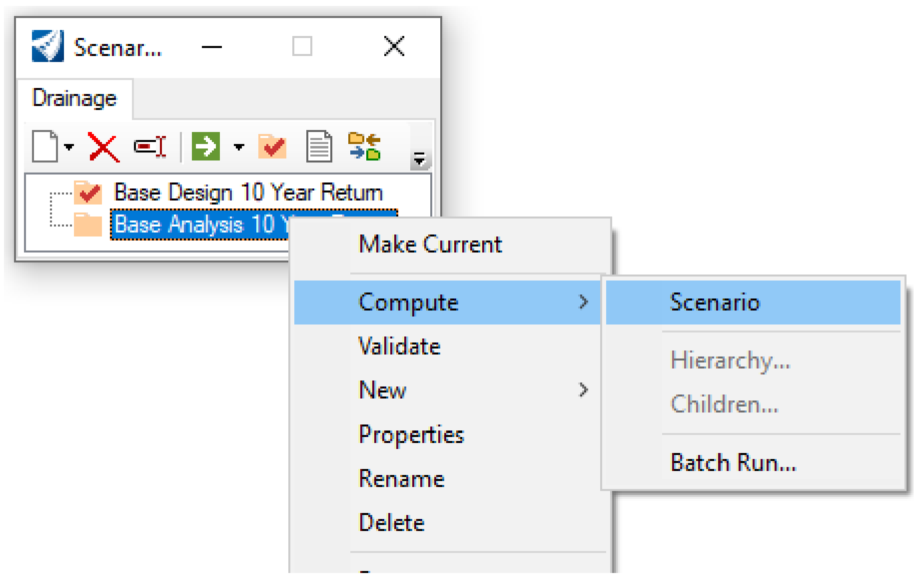
The parent/child function is not available for calculation options. New calculation options can be created by clicking the **New** icon.

6. Computing Scenarios

There are several places where you can compute a scenario:

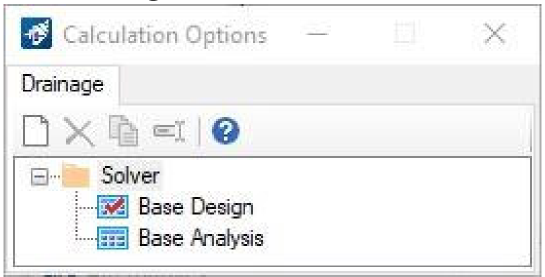
- The Compute icon in Analysis > Calculation - this computes the current scenario. Use this if you are confident that you know which scenario this is.

- The Compute Centre – this lets you choose the scenario to compute and shows you the most important settings that it contains. Use this when you are getting familiar with scenarios, as a straightforward way to check their settings.
- The Scenarios Manager – this lets you choose the scenario to compute and to compute multiple scenarios in a hierarchy or by selecting a batch. You will use this more when you are more proficient with the software and are using more scenarios.



Checking the Calculation Options

By default, the Gradually Varied Flow (GVF) calculations will be used to 'solve' the hydraulics of a storm drainage network.



(a) On the **Analysis** ribbon, select *Calculation >* **Options**. 

(b) **Note:** The calculation options that you see here have been copied in from the DGN library.

Note: The red tick  in the icon for the *Base Analysis* solver icon tells you that this is the active calculation option.

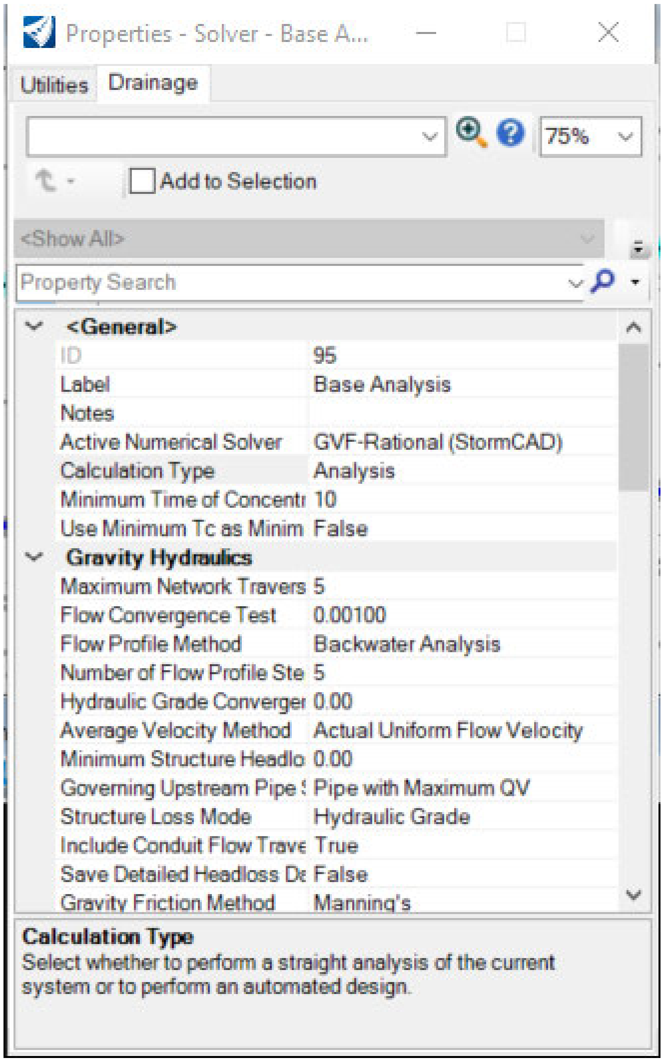
The most important property of a Scenario is the *Calculation Option:*

- An *Analysis* performs calculations but does NOT change pipe sizes and levels.
- A *Design* performs calculations and MAY change pipe sizes and elevations.

(c) Double click **Base Analysis** in the *Calculation Options* dialog.

Note that because you are editing the active Calculation Options, they will automatically be used by the active Scenario.

(d) The settings for *Base Analysis* are displayed in the *Properties* dialog. Your Calculation Options should be the same as the picture below.



Notes:

- The *Active Numerical Solver* is set to *GVF-Rational (StormCAD)* for Gradually Varied Flow using the Rational method.
- The *Calculation Type* is set to *Analysis.* This will not change the pipe sizes, levels etc. – none of the physical data.
- The *Flow Profile Model* is set to *Backwater Analysis*. This is the preferred option for computing a system, because the gradually varied flow algorithms it uses are more rigorous and generate solutions that more closely reflect reality.
- The *Minimum Time of Concentration* is set to 10 minutes. Any catchment with no Time of Concentration set, or one that is less than this, will be adjusted by the calculations to use this value.
- *Use Minimum Time of Calculation as Minimum System Time* applies when a pipe at the top of a branch has no flow.

(e) Scroll further down to *Calculation Options.*

(f) Locate the Use Rational Method Frequency Factors property in the Rational Method category.

(g) Check that the Use Rational Method Frequency Factors to False.

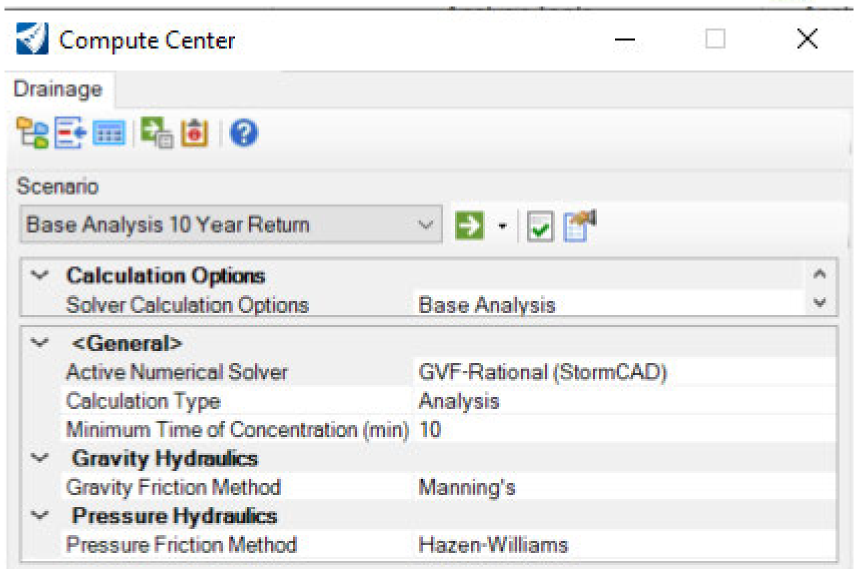
(h) Locate the Allow Runoff Coefficient to Exceed 1.0 property in the Rational Method category.

(i) Check that this is set to False.

This setting does not allow the product of the *Runoff Coefficient* and the *Frequency Factor* to exceed *1.0.* Checking the Active Scenario

1. *Computing the Scenario*

(a) From the **Analysis** ribbon, select *Analysis Tools* > **Compute Center.**



The toolstrip at the top of the Compute Center lets you access several Managers and tools to help you use scenarios. It always displays the current scenario when you open the dialog, but you can change the scenario to compute if you need to.

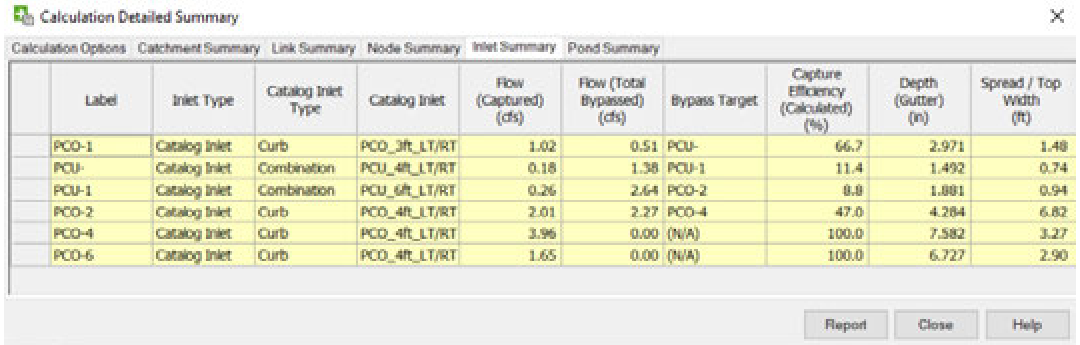
Review the other settings in the *Compute Center* dialog.

Note: Settings for the *Calculation Type* are set to **Analysis.** Remember - this means that the physical properties of the drainage network will not be changed. A *Calculation Type* that is set to **Design** may well cause the physical properties -- such as invert levels and pipe diameters -- to change.

(b) Click **Compute.** This computes the current scenario. On the GVF-Rational Calculation Summary panel, click on Details

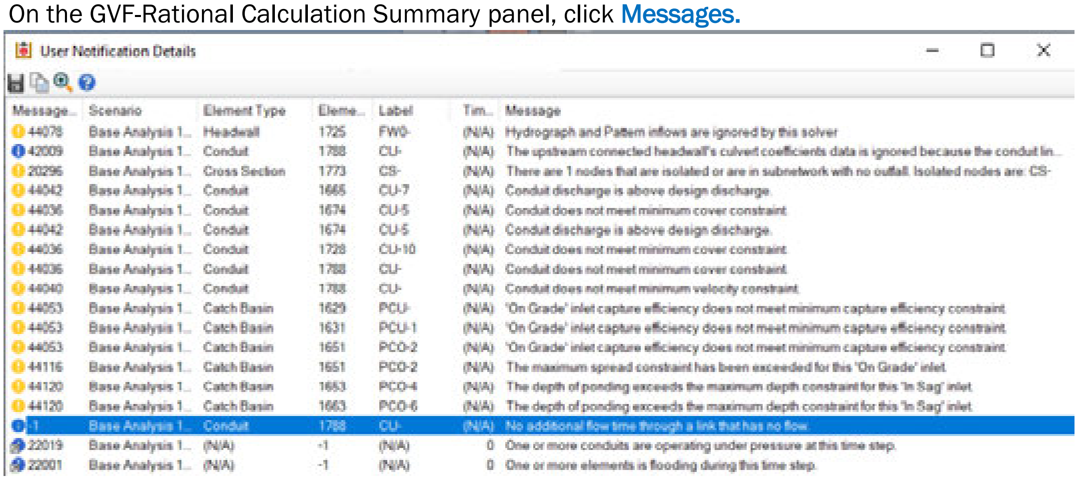
(c) Click the *Inlet Summary* tab and review the data.
You can see the *Capture Efficiency* of each catch basin, and the *Spread / Top Width*, so you can easily find any big issues.

The *Catchment Summary* tab also is a useful one to check at this stage, as it could reveal an issue with the storm data.



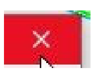
Label	Inlet Type	Catalog Inlet Type	Catalog Inlet	Flow (Captured) (cfs)	Flow (Total Bypassed) (cfs)	Bypass Target	Capture Efficiency (Calculated) (%)	Depth (Gutter) (in)	Spread / Top Width (ft)
PCO-1	Catalog Inlet	Curb	PCO_3ft_LT/RT	1.02	0.51	PCU-	66.7	2.971	1.48
PCU-	Catalog Inlet	Combination	PCU_4ft_LT/RT	0.18	1.38	PCU-1	11.4	1.492	0.74
PCU-1	Catalog Inlet	Combination	PCU_6ft_LT/RT	0.26	2.64	PCO-2	8.8	1.881	0.94
PCO-2	Catalog Inlet	Curb	PCO_4ft_LT/RT	2.01	2.27	PCO-4	47.0	4.284	6.82
PCO-4	Catalog Inlet	Curb	PCO_4ft_LT/RT	3.96	0.00	(N/A)	100.0	7.582	3.27
PCO-6	Catalog Inlet	Curb	PCO_4ft_LT/RT	1.65	0.00	(N/A)	100.0	6.727	2.90

(d) Close the *Details* dialog.
On the GVF-Rational Calculation Summary panel, click **Messages.**



Message...	Scenario	Element Type	Eleme...	Label	Tim...	Message
44078	Base Analysis 1...	Headwall	1725	FW0-	(N/A)	Hydrograph and Pattern inflows are ignored by this solver
42009	Base Analysis 1...	Conduit	1788	CU-	(N/A)	The upstream connected headwall's culvert coefficients data is ignored because the conduit lin...
20296	Base Analysis 1...	Cross Section	1773	CS-	(N/A)	There are 1 nodes that are isolated or are in subnetwork with no outfall. Isolated nodes are: CS-
44042	Base Analysis 1...	Conduit	1665	CU-7	(N/A)	Conduit discharge is above design discharge.
44036	Base Analysis 1...	Conduit	1674	CU-5	(N/A)	Conduit does not meet minimum cover constraint
44042	Base Analysis 1...	Conduit	1674	CU-5	(N/A)	Conduit discharge is above design discharge.
44036	Base Analysis 1...	Conduit	1728	CU-10	(N/A)	Conduit does not meet minimum cover constraint
44036	Base Analysis 1...	Conduit	1788	CU-	(N/A)	Conduit does not meet minimum cover constraint
44040	Base Analysis 1...	Conduit	1788	CU-	(N/A)	Conduit does not meet minimum velocity constraint
44053	Base Analysis 1...	Catch Basin	1629	PCU-	(N/A)	'On Grade' inlet capture efficiency does not meet minimum capture efficiency constraint
44053	Base Analysis 1...	Catch Basin	1631	PCU-1	(N/A)	'On Grade' inlet capture efficiency does not meet minimum capture efficiency constraint
44053	Base Analysis 1...	Catch Basin	1651	PCO-2	(N/A)	'On Grade' inlet capture efficiency does not meet minimum capture efficiency constraint
44116	Base Analysis 1...	Catch Basin	1651	PCO-2	(N/A)	The maximum spread constraint has been exceeded for this 'On Grade' inlet
44120	Base Analysis 1...	Catch Basin	1653	PCO-4	(N/A)	The depth of ponding exceeds the maximum depth constraint for this 'In Sag' inlet
44120	Base Analysis 1...	Catch Basin	1663	PCO-6	(N/A)	The depth of ponding exceeds the maximum depth constraint for this 'In Sag' inlet
-1	Base Analysis 1...	Conduit	1788	CU-	(N/A)	No additional flow time through a link that has no flow.
22019	Base Analysis 1...	(N/A)	-1	(N/A)	0	One or more conduits are operating under pressure at this time step.
22001	Base Analysis 1...	(N/A)	-1	(N/A)	0	One or more elements is flooding during this time step.

Review the messages.

(e)  Close the *User Notifications* dialog.

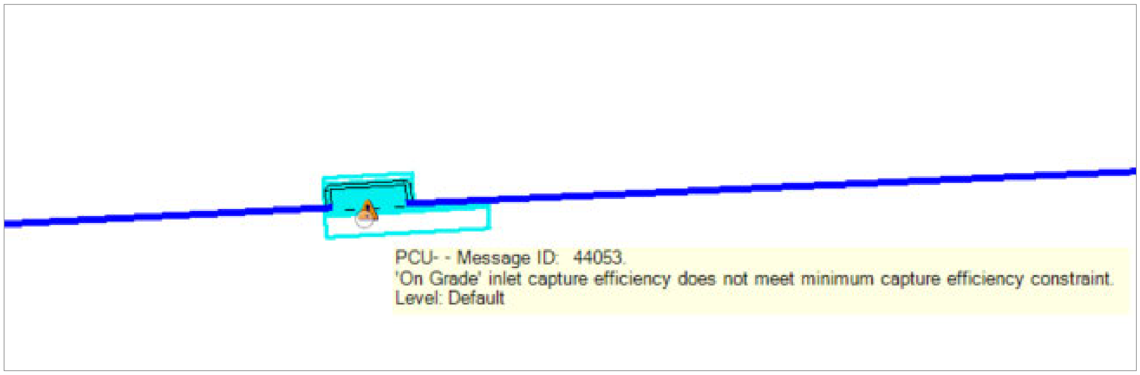
If this is the first time that you have used the User Notifications dialog, then the *Civil Message Center* dialog will now be displayed. The content of this will be like what you've just seen in the User Notifications, but this dialog also displays MicroStation and other ORD messages.

The buttons across the top of the dialog act as toggles, so you can click them to toggle the display of the errors, warnings, and information messages on and off. This also toggles on and off the glyphs in the graphics.

(f) Close the *Civil Message Center* dialog – if it is open.

(g) Close the GVF-Rational Calculation Summary and the Compute Center dialogs.

(h) *User Notifications* are now also shown as warning glyphs in the **Default** view.



If you ever want to review the information shown in the Details dialog, you do not need to compute the scenario again – you can simply click **Analysis > Calculation > Calculation Summary.**

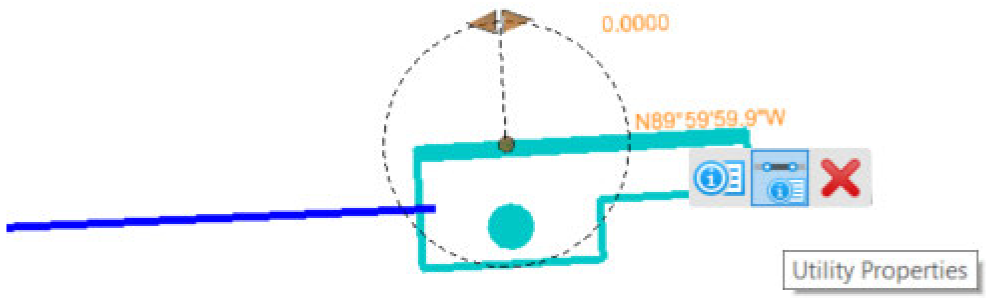
You can also review the Notifications at any time – if you have already computed the scenario.

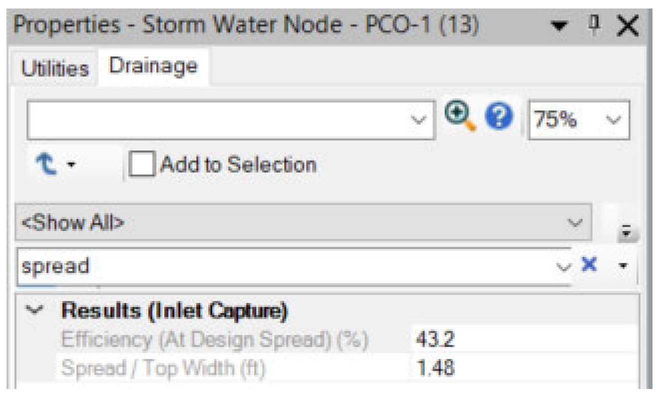
Click **Analysis > Calculations > Notifications.** The *User Notifications Manager* displays warnings and error messages

Note that if you click **Messages** on the *Calculation Summary* dialog to review the User Notifications, Zoom To is not available. This is because the software is still within the process of calculating the scenario – although the calculations have finished, the process has not – so user interaction is prevented. When you click **Notifications** on the **Analysis** ribbon, you are not within the calculation process, so Zoom To is available. The results that you see are those that were calculated when you last computed the scenario.

2. *Checking Hydraulic Properties*

(a) Select a catch basin and view the **Utility Properties.** There are quite a lot of result fields, so if you want to look at the spread width in the gutter, just type in the word *'spread'* in *Property Search*.





(b) You can see that the inlet is operating at *43%* of its efficiency.

(c) You can also see the width of flow in the gutter is approximately **1.48'**.

(d) Select one of the catchments

(e) Remove the *'spread'* text from the **Property Search** if you typed it in.
You can see the results for the catchment, including how much flow is coming off it.

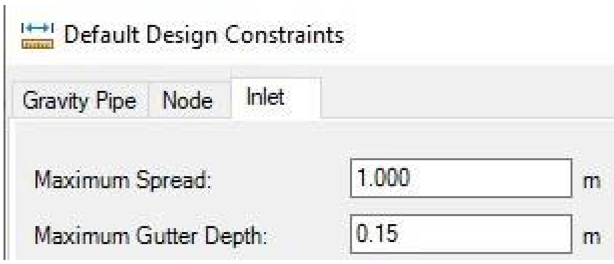
1) Now click on one of your catchments to review properties. Grayed out fields cannot be ediited from the properties window. All other properties can be changed individually or globally.

3. *Checking the Default Design Constraints*

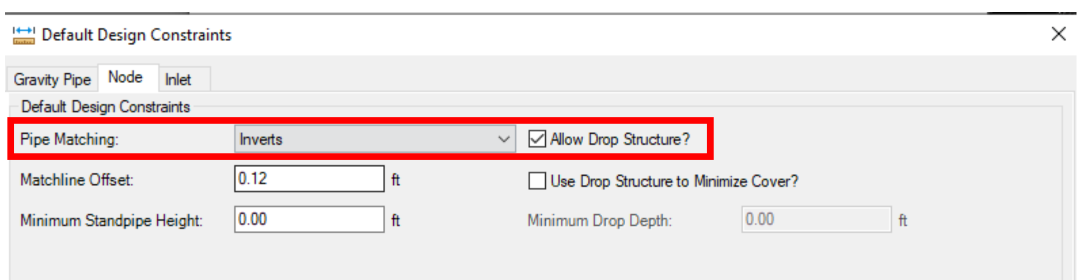
When you have computed the system previously, you have used the default settings for the spread width and depth. These values can be adjusted in the default design constraints.

(a) *On the Analysis ribbon, click Analysis*  *Tools > Default Design*  *Constraints.*

(b) *Click on all tabs and review the default design constraints. You can make changes here to set your design constraints according to your project needs and standards. Most design constraint defaults have been set to standard values, but you may need to make some changes according to your project needs.*



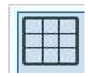
(c) *For example, if you want your structure to allow drop structure because your grade difference is higher than your allowed conduit slopes, click on the* ***Node*** *Tab.*

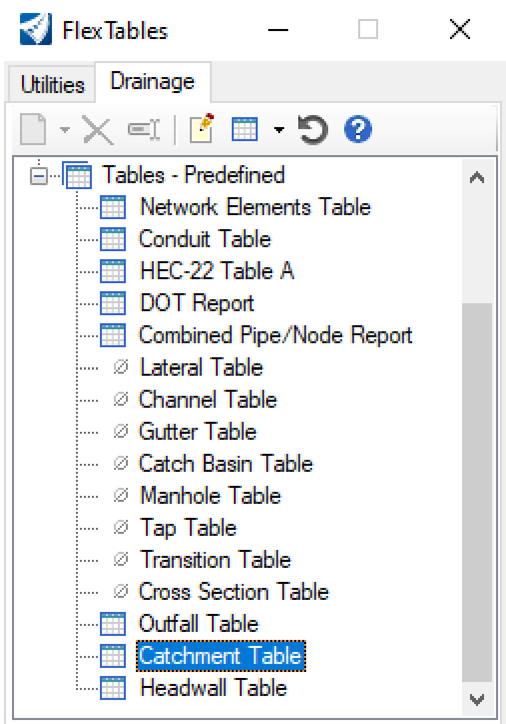


This will allow pipe inverts to be at different elevations. If you want to minimize depth of cover instead, you can uncheck the allow drop structure or check the box right below.

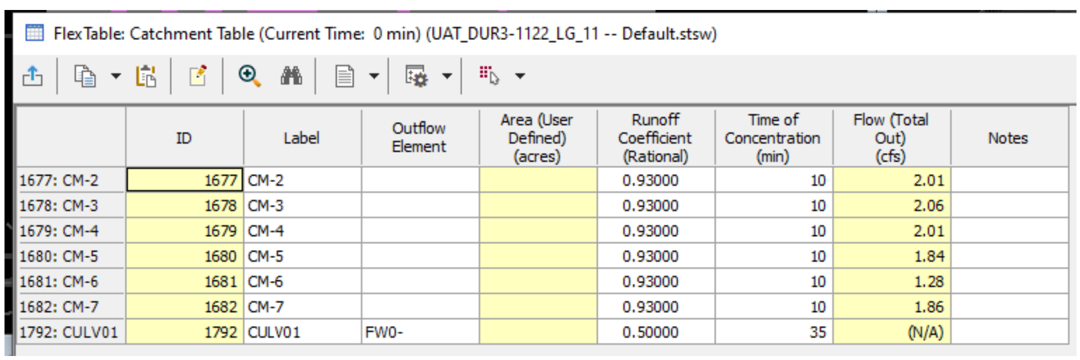
(d) *Close the dialog.*

4. Reviewing the Time of Concentration in a FlexTable

(a) Select Analysis > Analysis Views > FlexTables. 



(b) From the Drainage tab, double click Tables Predefined > Catchment Table.



FlexTable: Catchment Table (Current Time: 0 min) (UAT_DUR3-1122_LG_11 -- Default.stsw)

	ID	Label	Outflow Element	Area (User Defined) (acres)	Runoff Coefficient (Rational)	Time of Concentration (min)	Flow (Total Out) (cfs)	Notes
1677: CM-2	1677	CM-2			0.93000	10	2.01	
1678: CM-3	1678	CM-3			0.93000	10	2.06	
1679: CM-4	1679	CM-4			0.93000	10	2.01	
1680: CM-5	1680	CM-5			0.93000	10	1.84	
1681: CM-6	1681	CM-6			0.93000	10	1.28	
1682: CM-7	1682	CM-7			0.93000	10	1.86	
1792: CULV01	1792	CULV01	FW0-		0.50000	35	(N/A)	

The modified **Base Hydrology** for the *Catchments* can be confirmed here.

Note that the modification also could have been made in this dialog. Why have two dialogs – *Alternatives* and *FlexTables* – that both allow modifications? FlexTables always show values from the current scenario, so it is important that you check this before you start making changes. By using the *Alternatives* dialog, you are specifically choosing the Alternative to edit. Both workflows will achieve the same result.

(c) Close the *FlexTables* dialog.

(d) Close the Notifications dialog.

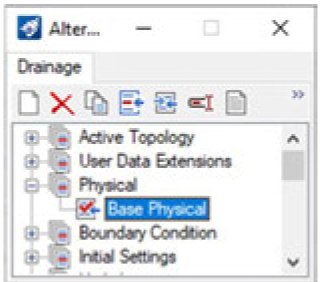
(e) Save Settings.

5. Best Practice for Storing the Physical Properties

Pipe sizes, invert elevations, etc., are stored in the current Physical Alternative. Values in the current Physical Alternative are overwritten every time you compute, but you can create as many alternatives as you like.

(a) On the Analysis > Calculation ribbon, click Alternatives.

(b) In the *Alternatives* dialog, locate the **Physical > Base Physical** Alternative.



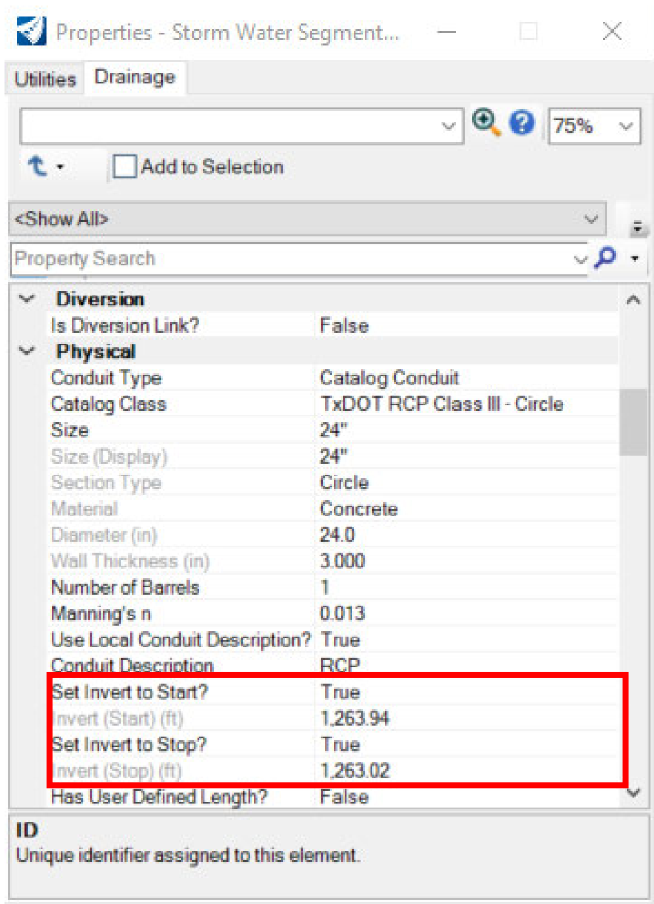
Note the red tick, showing you that this is the current Alternative – the one that is used by the current Scenario.

The name of the physical alternative can be changed, and additional alternatives can be created. Using sensible names is a good practice to adopt. Multiple alternatives can be created throughout a design workflow, so it makes sense to use sensible names for an audit trail.

6. Controlling Pipe Invert Elevations and Depth of Cover

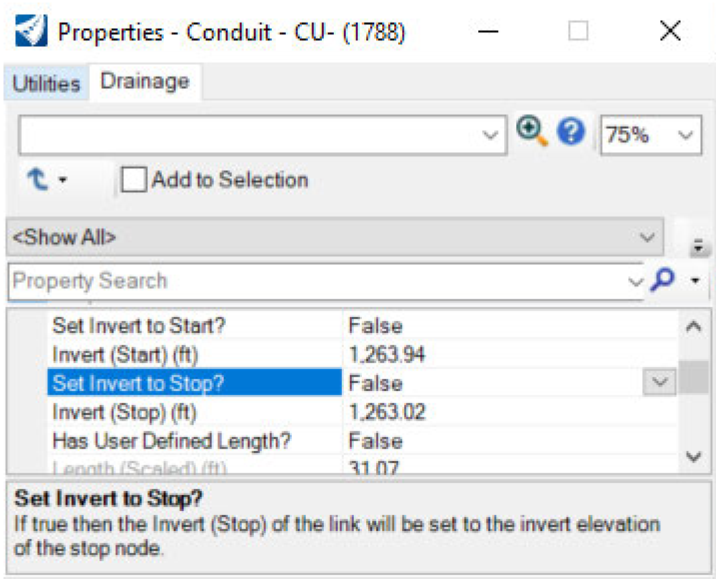
When setting up your system, pipe inverts are set to the invert elevation of the connecting nodes.

(a) Click on a conduit in your default view and open the Utility Properties



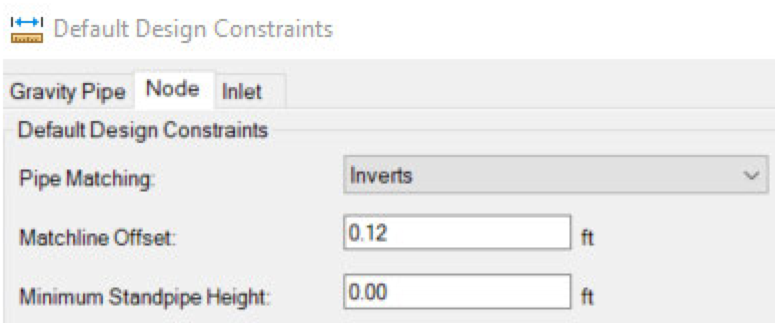
(b) Notice the set invert to start? and set invert to stop?, are set to true and the invert start and stop fields are grayed out.

(c) When you allow the software to compute the Design Scenario, these conduit properties are changed so design constraints for *pipe matching* can be honored.



Design of the conduits controls their invert elevations, and the connected node will update to follow the conduit.

Another setting in *Default Design Constraints* controls how elevations of conduits should change as they pass through a node. It is customary practice to have some change in elevations between the incoming and outgoing conduits to increase the hydraulic efficiency of the node. This is controlled on the Node tab.



In this case, the Default Design Constraints are telling the software that there should be a **0.12 ft** drop between the incoming and outgoing conduit invert elevations.

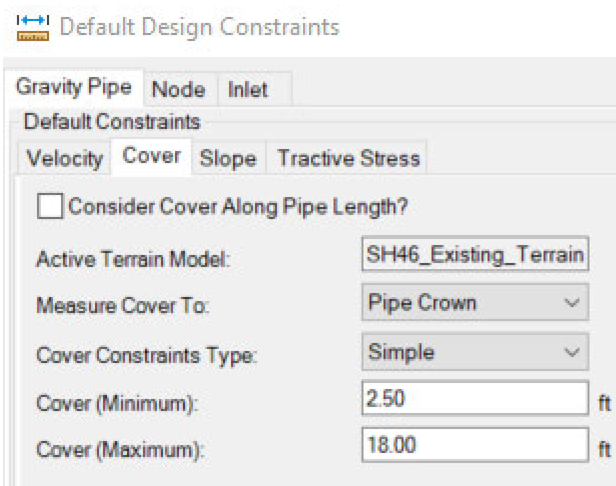
7. Controlling how the Default Depth of Cover is Applied

The depth of cover to each conduit is another factor that needs to be considered during the design process.

(a) Select a conduit, then click *Open Profile Model.*

(b) Open *View 4* and place the profile model in that view.
Do you want to consider the depth of cover along the conduit, or just at the nodes?

(c) From the Analysis ribbon, select Analysis Tools > Default Design Constraints. 

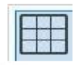


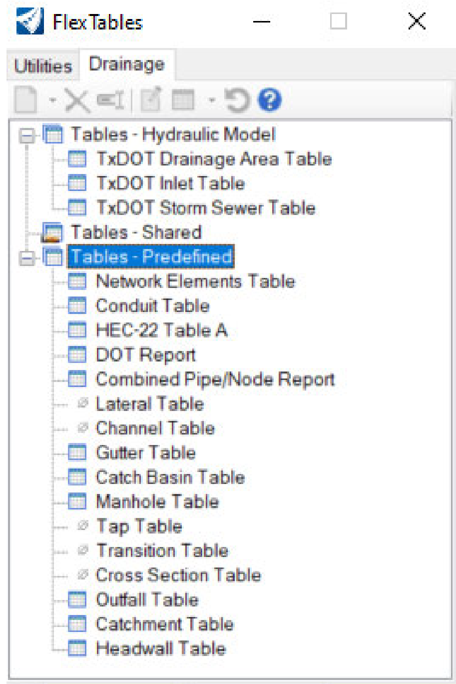
The **Gravity Pipe** tab allows you to enter default constraints to be used for design of pipes when performing a calculation run in design mode. The **Cover** tab has a check box – *Consider Cover Along Pipe Length?*

- If it is *unchecked,* the depth of cover is only checked at the nodes.
- If it is *checked,* the depth of cover is checked along the pipe length.

Reviewing the Changes

It is possible to select various conduits in your system and review their properties. However, it is easier to review and evaluate your system globally using the flex tables.

(a) Click Analysis>Analysis view>Flex tables 



You can Select from the list of predefined tables, create a project-specific Flex Table, edit its content, and create a report from that.

1. Reviewing the Available FlexTables

(a) Click the *Utilities* tab.

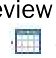
(b) Review the FlexTables available under *Tables – Predefined.*
Note the two FlexTables – *Storm Water Nodes* and *Storm Water Conduits.* These two FlexTables contain data for all types of storm drainage nodes and conduits, but they do not include any hydraulic data, and catchment areas are not included.

Note that FlexTables for other types of utility – Communications, Electrical, etc. -- also are available.

(c) Click the *Drainage* tab.

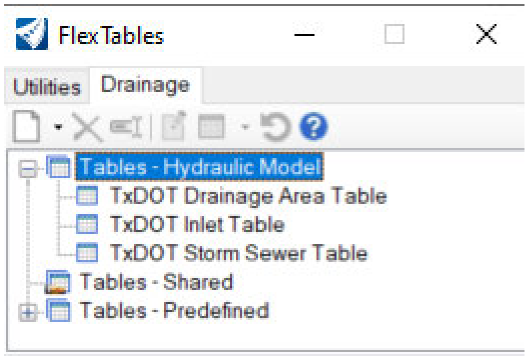
(d) Review the *FlexTables* that are available under *Tables – Predefined.*
Note that there is no concept of a utility type on the Drainage tab because all the FlexTables available on this tab refer specifically, and only, to storm and sanitary drainage data. While storm drainage data is available in both tabs, if it is accessed from the Drainage tab, then the information available is richer, because it includes the hydraulic data.

(e) Review the two icons that are used for each of the *FlexTables*:

-  This icon means that data for this element type is in this design file.
-  This icon means that this design file does not contain data for this element type.

For example, this design file contains conduits and catch basins, but it does not contain laterals or taps.

(f) If it isn't already expanded, expand **Tables – Hydraulic Model.**



- **Note** that there are several *FlexTables* listed here. These were automatically copied into this design file when the Utilities project was created, because they existed as Drainage *FlexTables* in the Drainage and Utilities DGN library. The same rule applies to Utilities *FlexTables* too.
- Also, **note** *Tables – Shared*. There will not be any *FlexTables* listed under here – unless you have previously set some up on this computer.

You have seen that there are three folders that are available at the top level. Their functionality differs, as follows:

- *Predefined FlexTables* are shipped with the software and act as examples. You cannot create new FlexTables under Predefined
- *Shared FlexTables* are available in every Utilities project that is opened on this computer.
- *Hydraulic Model FlexTables* are only available in this Utilities project. They are stored in the design file.

If you want to create FlexTables that anyone who works on the same design file can use, then *Hydraulic Model FlexTables* are an effective way to accomplish this.

2. Copying a Predefined FlexTable

There are two ways to create a FlexTable:

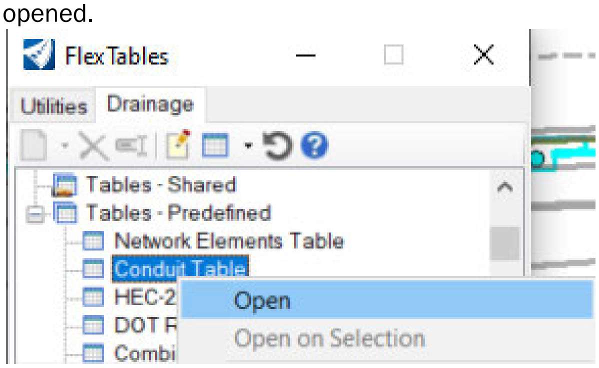
- Copy an existing FlexTable.
- Create a new FlexTable from scratch.

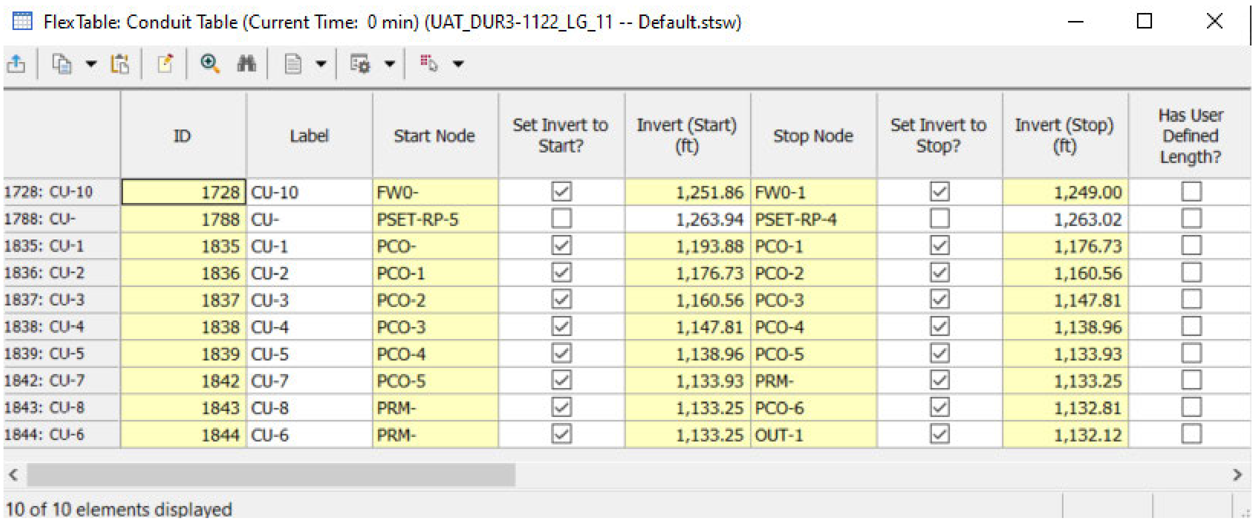
The first method of creating a FlexTable – by copying an existing table – is useful if the existing FlexTable contains similar content to what you want. The first step is to see what the existing FlexTable contains.

(a) In the *FlexTables* dialog, on the *Drainage* tab, scroll down so that you can see **Tables – Predefined.**

(b) Under Tables – Predefined, locate Conduit Table.

(c) Right-click on **Conduit Table**, then click **Open,** or double click the table. The Conduits FlexTable is opened.





FlexTable: Conduit Table (Current Time: 0 min) (UAT_DUR3-1122_LG_11 -- Default.stsw)

	ID	Label	Start Node	Set Invert to Start?	Invert (Start) (ft)	Stop Node	Set Invert to Stop?	Invert (Stop) (ft)	Has User Defined Length?
1728: CU-10	1728	CU-10	FW0-	☑	1,251.86	FW0-1	☑	1,249.00	☐
1788: CU-	1788	CU-	PSET-RP-5	☐	1,263.94	PSET-RP-4	☐	1,263.02	☐
1835: CU-1	1835	CU-1	PCO-	☑	1,193.88	PCO-1	☑	1,176.73	☐
1836: CU-2	1836	CU-2	PCO-1	☑	1,176.73	PCO-2	☑	1,160.56	☐
1837: CU-3	1837	CU-3	PCO-2	☑	1,160.56	PCO-3	☑	1,147.81	☐
1838: CU-4	1838	CU-4	PCO-3	☑	1,147.81	PCO-4	☑	1,138.96	☐
1839: CU-5	1839	CU-5	PCO-4	☑	1,138.96	PCO-5	☑	1,133.93	☐
1842: CU-7	1842	CU-7	PCO-5	☑	1,133.93	PRM-	☑	1,133.25	☐
1843: CU-8	1843	CU-8	PRM-	☑	1,133.25	PCO-6	☑	1,132.81	☐
1844: CU-6	1844	CU-6	PRM-	☑	1,133.25	OUT-1	☑	1,132.12	☐

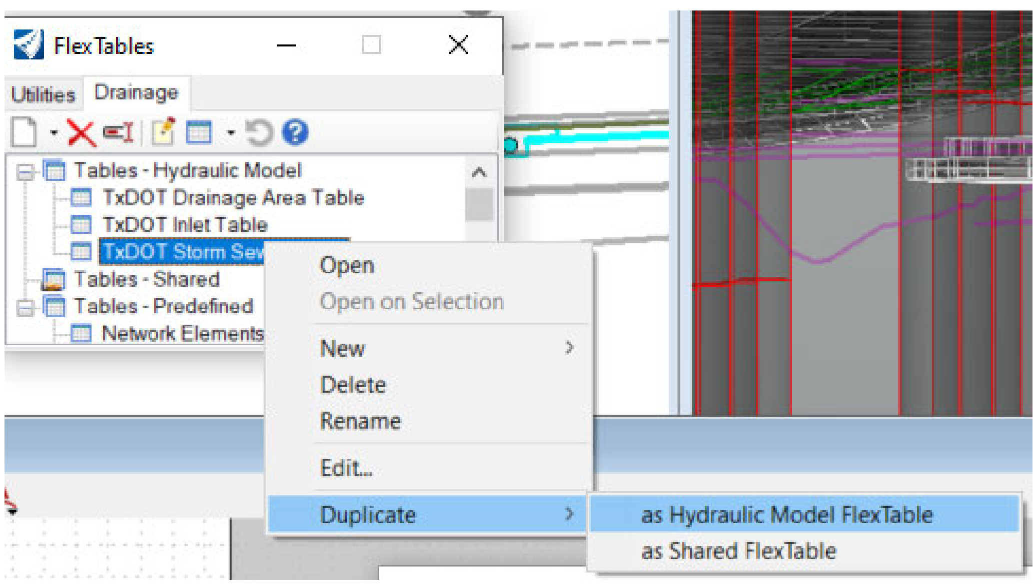
10 of 10 elements displayed

(d) Use the horizontal scrollbar to review its content.

(e) Close *FlexTable* Dialog

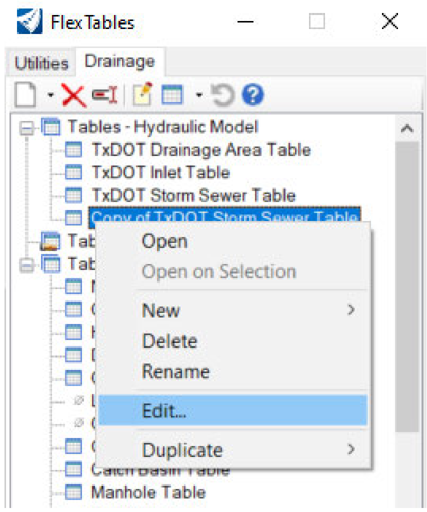
You will see several columns, including information on invert elevations, slope, size, etc.
TxDOT workspace provides 3 Hydraulic model tables to use. However, these tables may not contain all the information your district requires for reporting your calculations.
You can duplicate the predefine Hydraulic model table and add the information you need or remove information.

(f) Right click on TxDOT Drainage Storm Sewer Table, then click *Duplicate*> As Hydraulic Model Flex Table.

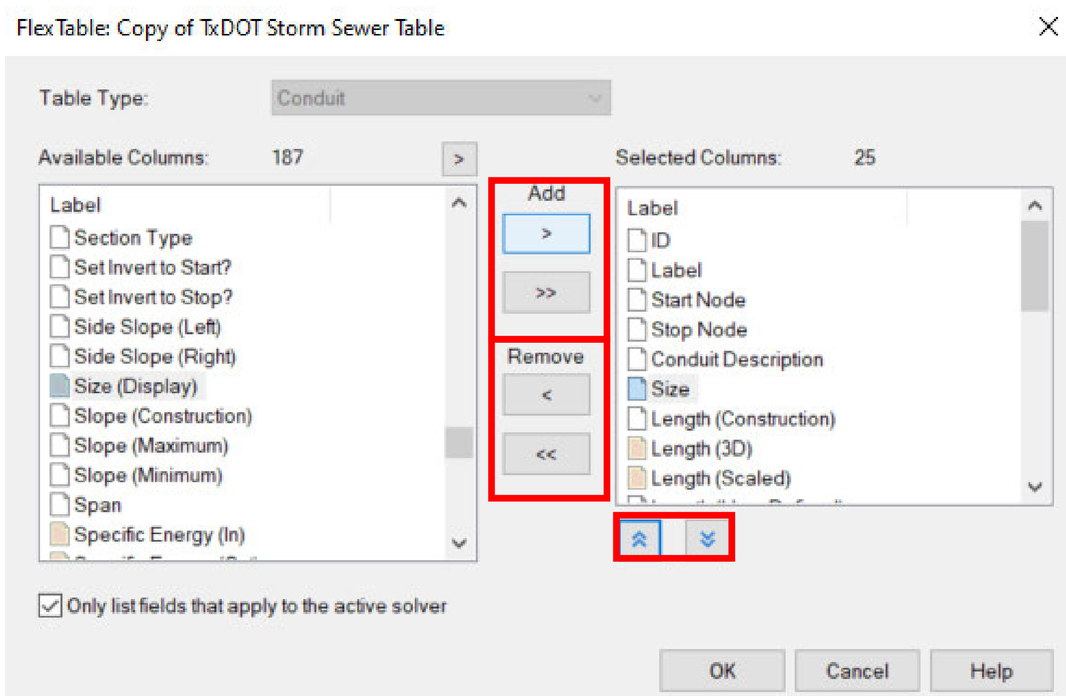


(g) Look at the FlexTables that are listed under *Tables- Hydraulic Model*. You will see that they now include Copy of TxDOT Storm Sewer Table.

(h) Right click on Copy of TxDOT Storm Sewer Table. Then click Edit.



(i) While there are a few special FlexTables that let you combine nodes and conduits in a single FlexTable – the Network Elements FlexTable is one example – FlexTables normally only show data for a single element type.



Note the *Table Type* shown at the top of the dialog. This is a read-only field, which says *Conduit* in this case. This dialog lets you change the content of the FlexTable. The list on the left-hand side shows the available columns, and the list on the right-hand side shows the selected columns.

You can add or remove columns from the table and select the placing of new added columns using the up or down arrows. In this case, a column for the size was added.
You can rename your table.

(j) You Right-click on Copy of TxDOT Storm Sewer Table, then click Rename.

(k) Type TxDOT (your district) Storm Sewer Table.

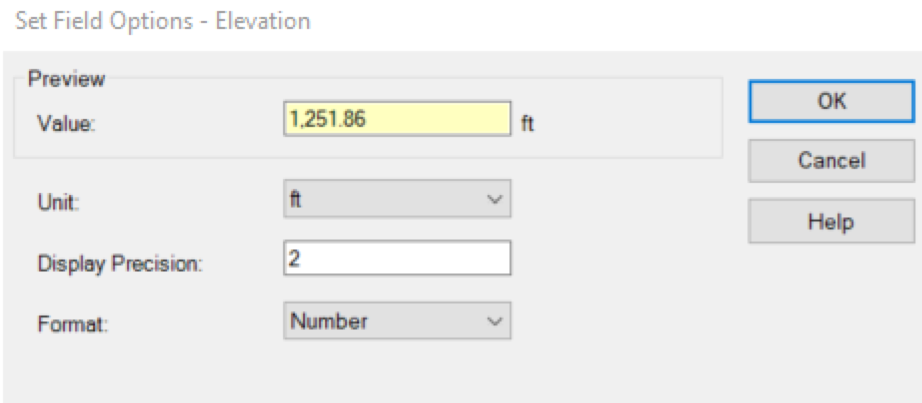
3. Formatting the Conduit FlexTable

To format the flex tables

(a) Right Click on the column you wish to format.

(b) Select Units and Formatting.

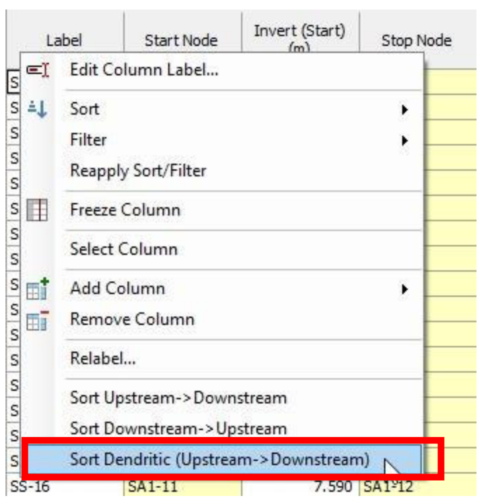
(c) Change your setting in the dialog window.



Note that all columns with the elevation formatter will share the same format. If you change the units or display precision, it will change it for all columns that report an elevation.

(d) Right-click on the title text of the *Label* column.

(e) Click on Sort Dendritic (Upstream > Downstream).

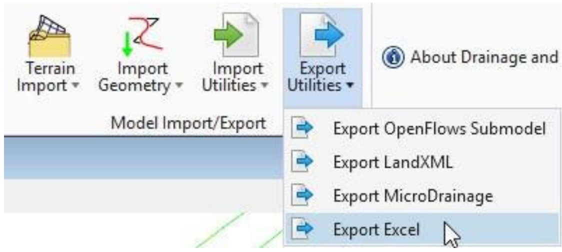


This sorts the order of the conduits so they are listed in dendritic order. All branches of your system that come to a single node from upstream to downstream will be together.

You can also edit the column label. Open the TxDOT Drainage area Table. You will notice that the intensity and Run-Off columns do not have a return year. To update the values, simply click on the Edit Column label of the menu on the right and update your return year.

4. Producing a Reports from FlexTable

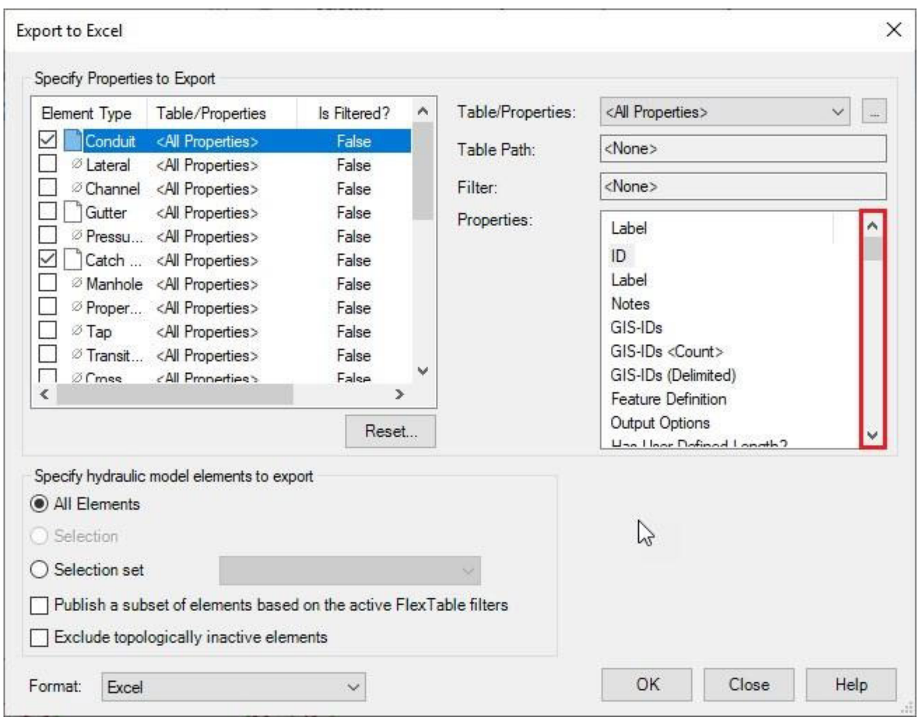
(a) On the *Home* ribbon, *click Export Utilities* > **Export Excel.** 



By default, Conduit is the selected Element Type in Specify Properties to Export. The column to the right of that, labeled Table/Properties, says <All Properties>, and the Is Filtered? column says False.

These settings show that, if you click OK now, this tool will export all the properties for every element type that exists in the model. This can amount to a lot of properties, and – depending on the purpose of the report – are probably many more than you need.

(b) Use the vertical scroll bar on the right-hand side of the dialog to review the available properties for a conduit.

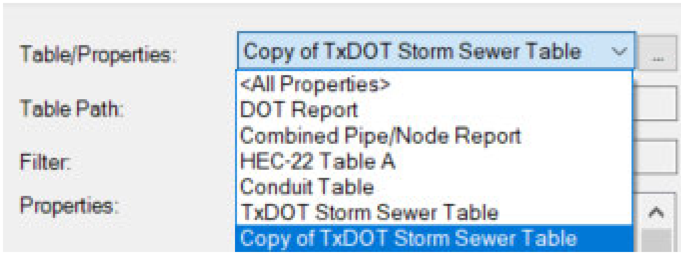


(c) Note the controls above the vertical scroll bar:
- Table/Properties
- Table Path
- Filter

These controls can be used to control the content included for each element type.

(d) Ensure that *Conduit* is the selected row in Specify Properties to Export and that the check box for it is checked on.

(e) Click in the **<All Properties>** field on the right-hand side of the dialog.
(f) The drop-down list includes all FlexTables which are valid for the Conduit element type.

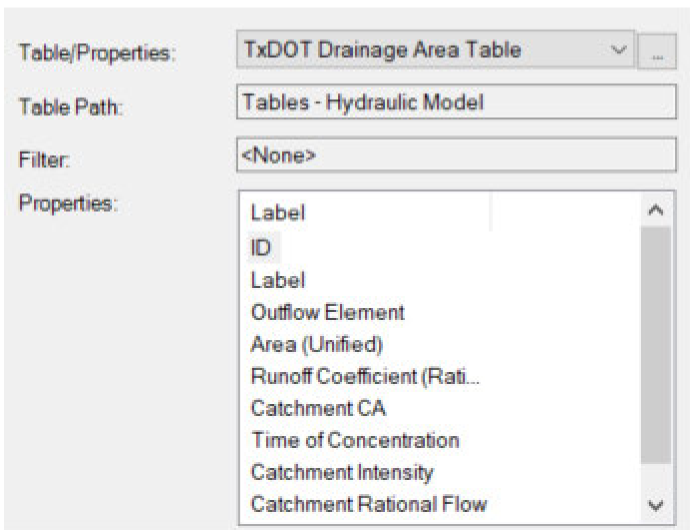


Note that you can select FlexTables from any location – Predefined, Shared, and Hydraulic Model.

(g) Click on TxDOT Storm Sewer Table.
The dialog tells you that this FlexTable is stored under Tables – Hydraulic Model, so you know that it is stored in this design file.

Note that the Filter control says *<None>* because the FlexTable you have selected does not have a filter.

(h) Use the vertical scroll bar to review the list of properties that now are included for conduits. It's a lot less.

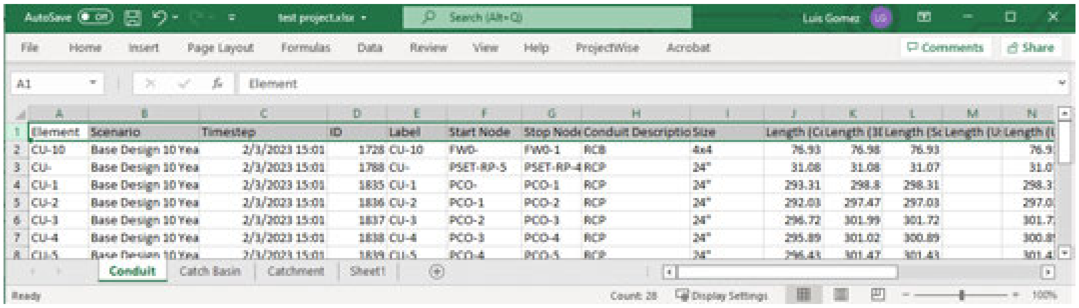


Your dialog should be the same as the picture above.

Use the same procedure to select your catch Basin (TxDOT Inlet Table) and Catchment (TxDOT Drainage Area Table).

(i) Click **OK** to create the spreadsheet.
(j) In the *Export to Excel* dialog, type in file name *Composite Report.*
(k) Click the **Save** button.

(l) If you have Excel (or a viewer for Excel files) on your computer, you can locate the file using Windows Explorer, and open it.



The spreadsheet contains a sheet for each selected element type. If you review row 1 on each sheet, you will see the columns that you selected in the FlexTables. Note that the Scenario and Timestep columns are added automatically.

5. Techniques for Controlling the Elements to Include

The Export to Excel dialog gives you several ways to control which elements are included in the report. This section will briefly discuss each of these methods.

- Using a Filter
- Using a Selection Set
- Excluding Topologically Inactive Elements

(a) Using a Filter

FlexTables can have a filter applied to any included column, so you could, for example, only include the elements that use a particular feature definition or have a certain size range. You can either create a filter directly in the FlexTable, or you can create a Query first, then use it to filter the FlexTable.

(b) Using a Selection Set

Selection sets may be a more flexible technique to use than filters, for several reasons:

- A comprehensive set of predefined Queries is included in the software.
- It is easy to build your own query, using any available fields and operators (And, Or <, >, etc.).
- Queries can be saved and reused as needed.

The basic workflow is to:

1) Click Queries on the Utilities View ribbon.
2) Identify the queries to use, or create your own.
3) Click Selection Sets on the Utilities View ribbon.
4) Select the Queries to use in the Selection Set.
5) Select the Selection Set in the Export to Excel tool.

An example of where this functionality could be useful is if you have more than one network in your design file, but you want to create a separate report for each network. Because each element in a drainage network has a property for the Subnetwork Outfall, you could use this to create queries for each one.

(c) Excluding Topologically Inactive Elements
Each drainage element has an “Is Active?” property, which controls whether it is active or not in the Physical alternative of a scenario. This offers another method that is useful when there are multiple networks in one design file.

The basic workflow is to:

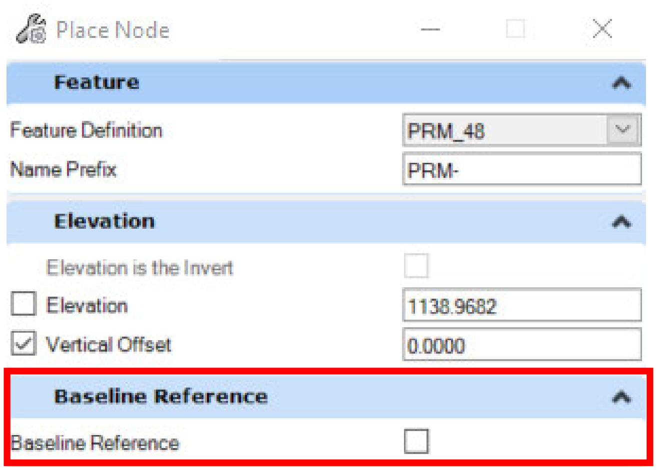
1) Create a Physical alternative for each network.
2) Create a Scenario for each network.
3) Select the appropriate Physical alternative for each Scenario.
4) Make one of the Scenarios current. This will make all elements not in it inactive.
5) Check the Exclude topologically inactive elements in the Export to Excel tool.

Conversely you can create active topology alternatives to exclude elements from your flex table by turning off the “is Active” toggle

6. Assigning a Station and Offset Reference to Nodes

The first step to include information on the stations and offsets in a report is to assign the relevant linear feature, such as an alignment, to the nodes. The station and offset values are calculated by raising a normal from the node location point to the linear feature.

If during your design process, you do not use a baseline as your reference alignment (the baseline reference is unchecked), it is possible to assign station offset to your element.



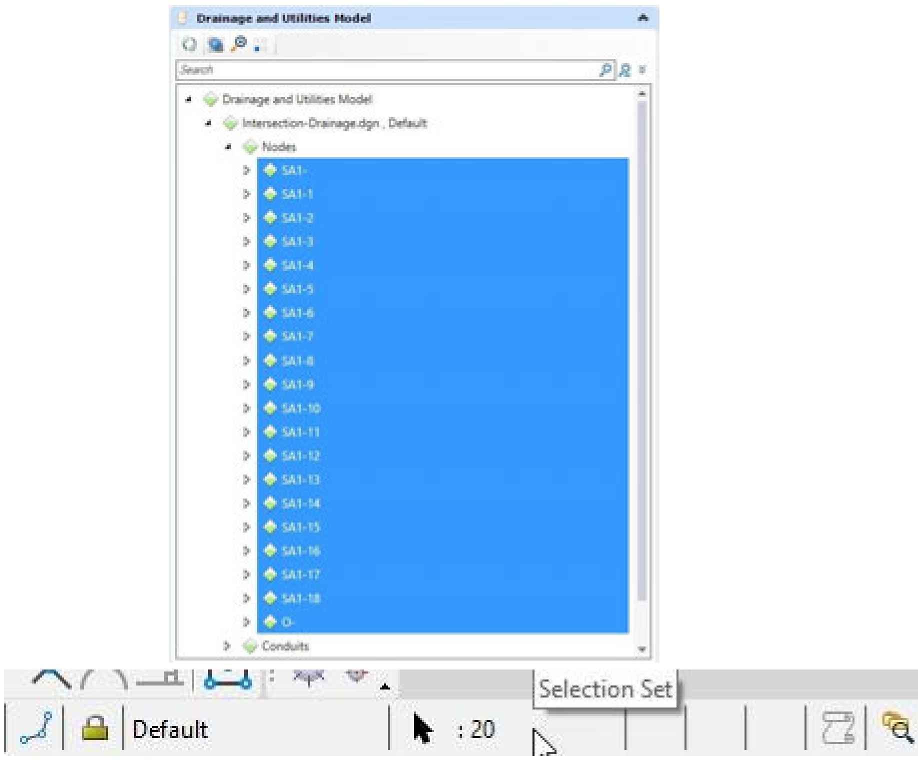
(a) Open the **Explorer** window. 

(b)  Open the **Properties** window.

(c) In the **Explorer** window, expand the *Drainage and Utilities Model.*

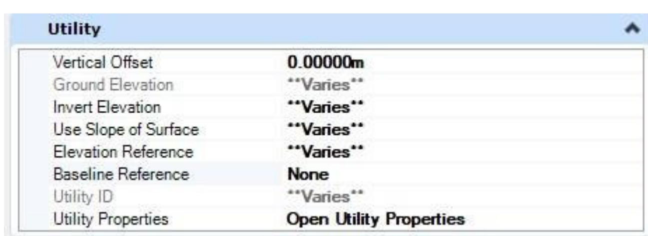
(d) Continue to expand until you see all the Nodes *(Intersection_Drainage.dgn > Nodes).*

(e) **Highlight** all the nodes, that are not linked to an alignment baseline or centerline to add them to a Selection Set. To do this, select the first node, press the Shift key, then select the last node. You can also use the Control key to add individual nodes to the selection. There are other tools, such as Queries and FlexTables, that also can be used to create a Selection Set.



(f) Confirm that the Selection Set contains all the nodes you need by looking along the bottom of the product window – where toolbars like AccuSnap often are docked. The number of elements selected is shown here.

(g) In the **Properties** window, locate the *Baseline Reference* property in the *Utility* section. It is currently set to *None*.

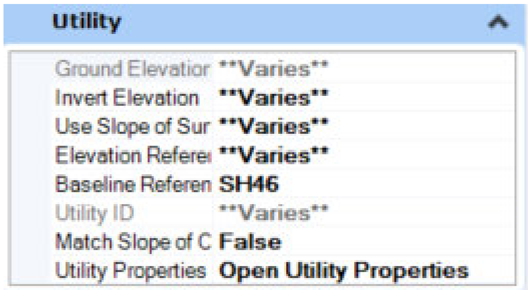


(h) Click in the property field, and click on the browse icon (three dots) on the right side of the property field.

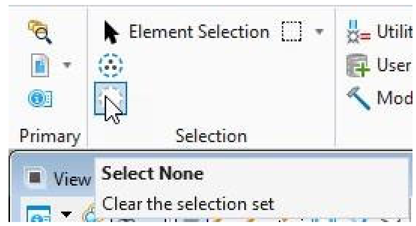
(i) You are prompted to **Select Reference**.

(j) Select the centerline or baseline of your choosing. To avoid picking other graphics, such as the catchment areas, it is easiest to window to the top of View 1.
This will set the *Station/Offset Reference* for all the selected nodes.

(k) Confirm that the Utility Properties now show *SH46* in the *Station/Offset Reference* property.

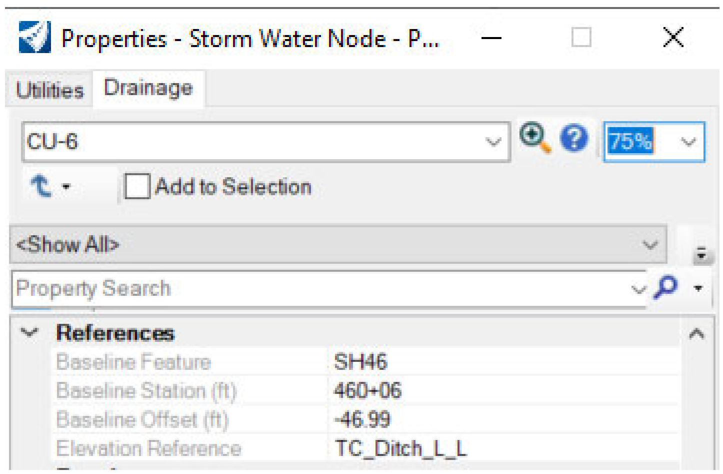


(l) Click the **Select None** icon to clear the Selection Set. This can be done by clicking in an area of View 1 that does not contain any graphics.

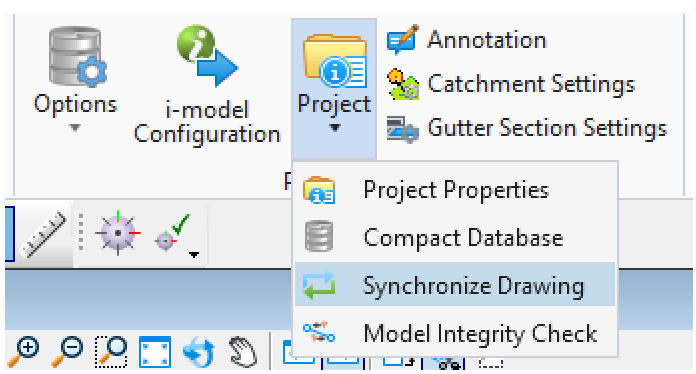


(m) Select a node and open the *Utility Properties*.

(n) Confirm that the *Baseline Station* and *Baseline Offset* properties have values. These values will update automatically if a node is moved.



Note that these values are shown in the *Utility Properties* dialog, but not the *Properties* dialog. They are listed here so that they can be included in FlexTables. If the Baseline REF is set but is not showing up in the FlexTables, you can run a “Synchronize Drawing” command. This should trigger those missing values to be populated appropriately. (Drainage & Utilities > Tools > Project Tools Group >



7. Embedding the Spreadsheet in the Design File

Your hydraulic data will need to be reported in hydraulic data sheets in your plans. You can achieve this by embedding the content of a spreadsheet in the design file. You have just learned how you can create spreadsheets that include the content you need, in the layout that you like. Runoff computations, inlet computations, and storm sewer computations can be extracted from the FlexTables.

(a) On the analysis tab click on FlexTables.

(b) Double click on the desired table to open.

(c) Click the **Export To File** icon to export the file to .csv format.

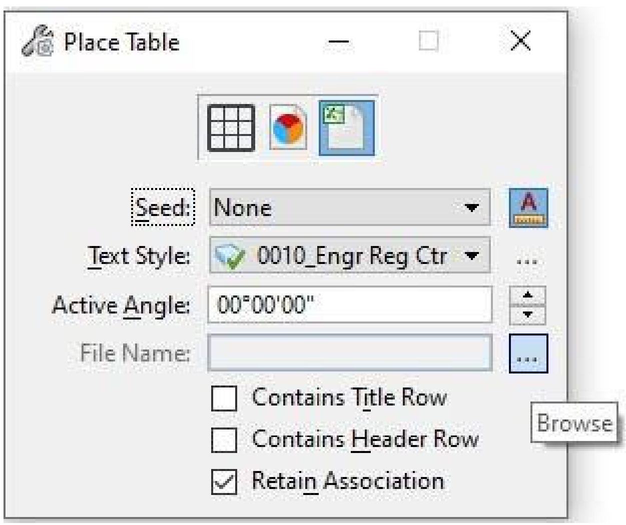
(d) Set the **Save as type** to *Comma Delimited File* (.csv).

(e) Name the file **Runoff Computations.csv.** (This is an example of one of your choices.)

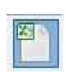
(f) Click Save.

(g) **Close** the *Flex Tables* dialog.

(h) On the Drawing Production ribbon, click *Tables* > Place Tables.



(i) Change the Text Style so the table is a reasonable size
- In Imperial, **Arial .0083 Ctr Ctr** may be an excellent choice
- In metric, *Arial 2.5mm Ctr Ctr* may be a desirable choice

(j) Click the **From**  **file** icon.

(k) Select the **Browse** icon.

(l) Change the file type to **Comma-separated (*.csv)**

(m) Browse to and select the **My Catch Basin Table.csv** file that you created in the last section. 7. Click **Open**. After a few seconds, the table will be attached to your cursor.

(n) Place Table dialog opens. Just select **OK.**

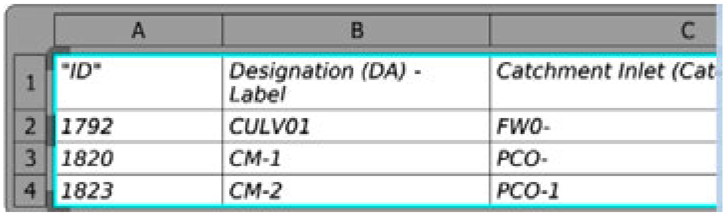
(o) Left click in *View 1* to place the table into the design file.

(p) Using the **Element Selection** tool, select the table.

Note that the table changes its appearance to rows and columns like a spreadsheet.

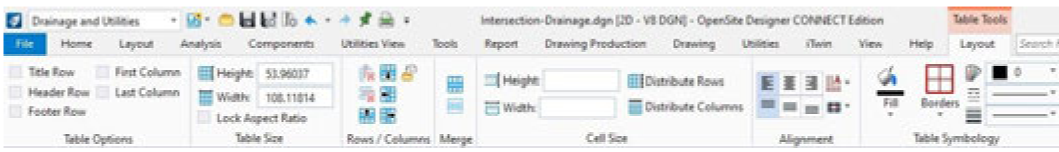
Because the *Retain Association* option is checked on, the table is graphically linked to the .csv file.

When tables are created from a file using this tool, they become very much like a spreadsheet.



	A	B	C
1	"ID"	Designation (DA) - Label	Catchment Inlet (Cat
2	1792	CULV01	FW0-
3	1820	CM-1	PCO-
4	1823	CM-2	PCO-1

Also, note that the ribbon menu changes and gives you access to common spreadsheet tools that can be used with the table.



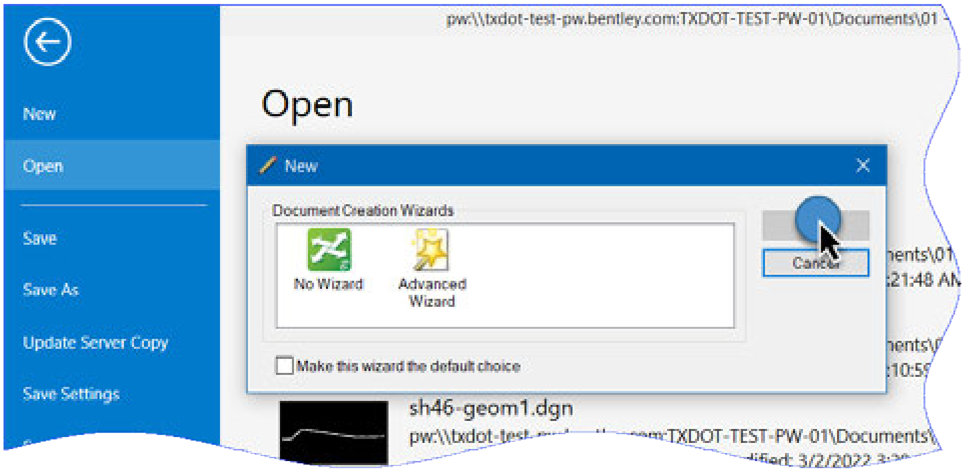
Left click anywhere in *View 1* to de-select the table.

Note: TxDOT will continue to use Axiom as the preferred method to embed tables in your plan set. Any resource files needed to set up your drainage table preferences will be posted to the external ORD site https://www.txdot.gov/business/resources/design-tools-training/OpenRoads-designer.html

Plans Production sheet cutting

1. Create Drainage Container File

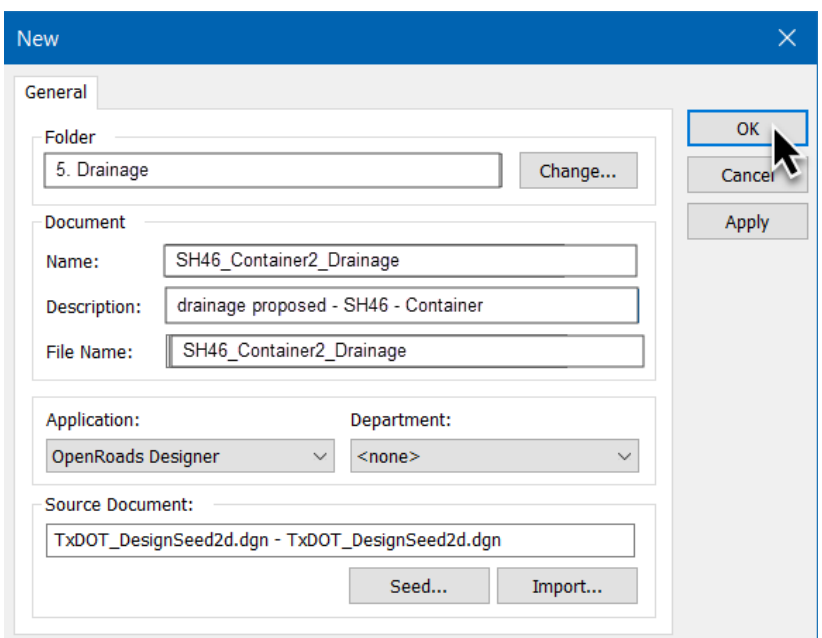
(a) In ORD navigate to File > New > Select No Wizard > OK.



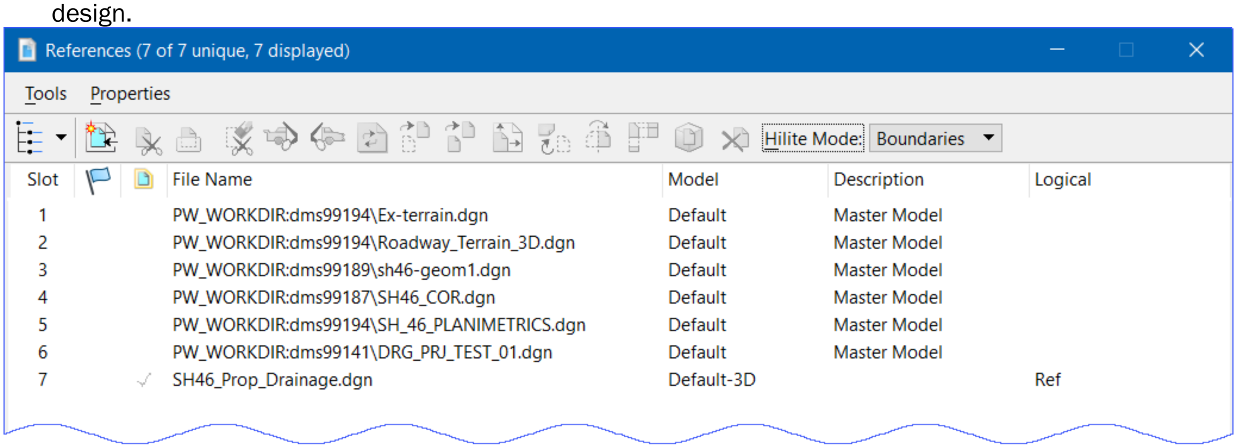
Folder: Select Change... > navigate to the Project Folder > 4 - Design > Plan Set > 5. Drainage > Select OK.

Document:
Name: Drainage container file name. Example: SH46_Container_Drainage
Description: Drainage proposed – SH46 - Container
Source Document: TxDOT_DesignSeed2d.dgn - TxDOT_DesignSeed2d.dgn
(Select Seed > Select TxDOT_DesignSeed2d.dgn > Select Open > Select Yes on the Ready Only Dialog)

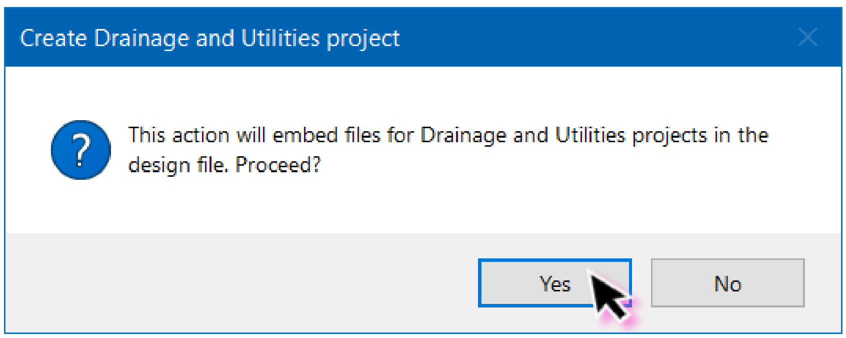
(b) Select OK to create the New file in ORD.



(c) Attach **the reference files** with no nesting such as terrain files, geometry, corridor, drainage design.



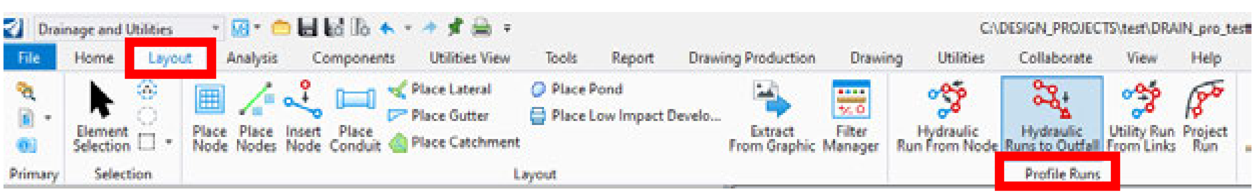
(d) **Activate the Drainage and Utility Project**
Within the file > select a node > hover and select Utility Properties >. Click Yes to activate the Drainage and Utility Project.



This will enable you to view and label the drainage and utilities properties of the drainage structures and conduits.

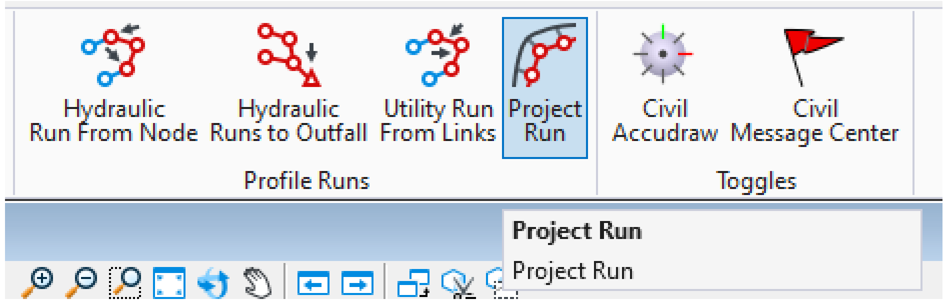
2. Set up the Profile View

We will now Create a projected Profile run for drainage profiles and P&Ps
a) Select layout tab.
b) Under profile runs, select hydraulic runs to outfall.



c) Follow the context menu prompts to create your profile run.

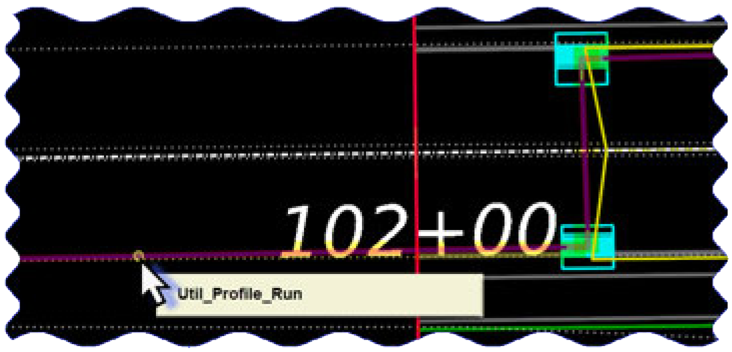
d) Once created, use the “Project Run” tool to project the profile run to the selected baseline.



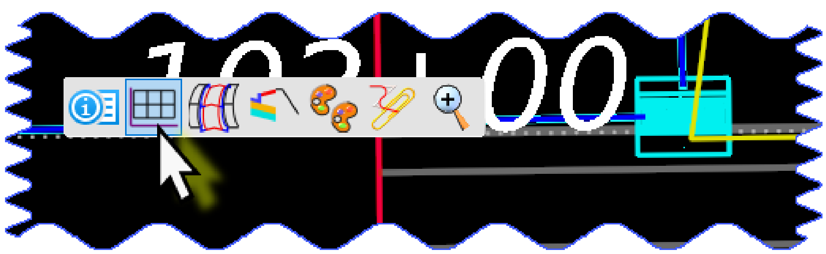
(e) Follow the plan production workflow for profile and P&P sheet creation with the Utility Profile Run shown.

A profile view of the utility profile needs to be set up in the container file to be able to create Profile sheet(s) of the drainage system.

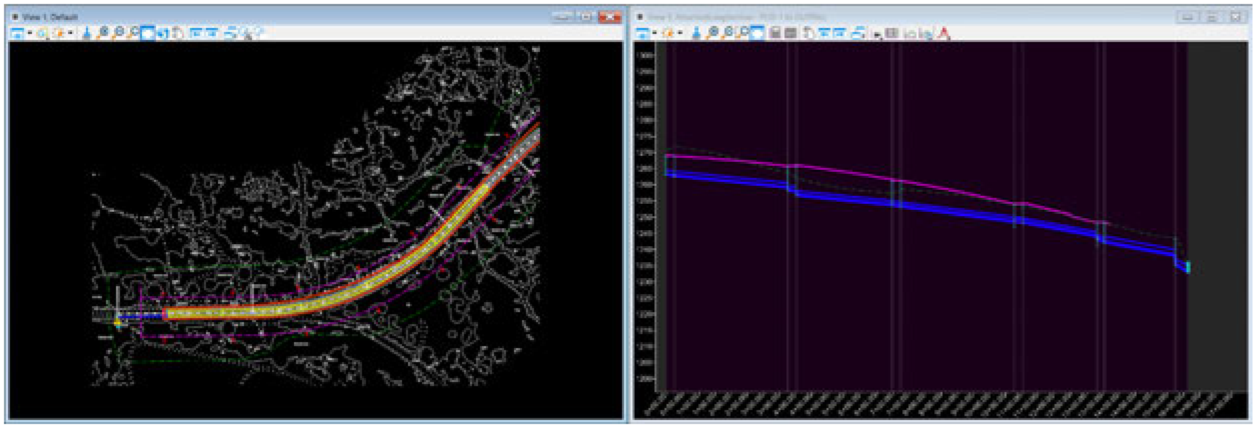
(f) In the plan view, using the Element Selection tool select the utility profile run in the Default view.



(g) Hover, and wait for the head-up menu to appear. From there, select the Open Profile view, open another view, and click within for the profile run to display. Fit view.



(h) Arrange View 1, Default and the Profile View (**BL OR CENTERLINE PROFILE WITH PROJECTED DRAINAGE PROFILE)**



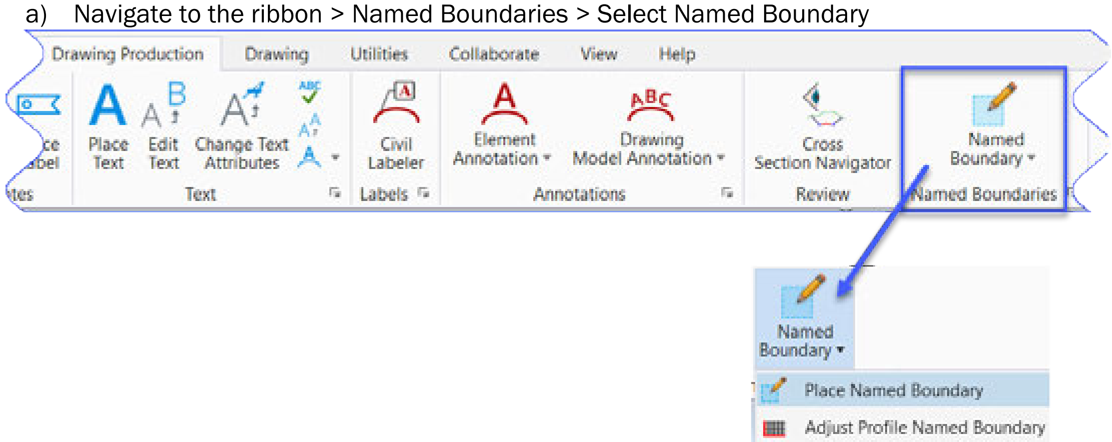
(i) Save Settings.

We are now ready to create the different boundary types to cut our sheets.

3. To create Plan sheets for Drainage area maps or layouts

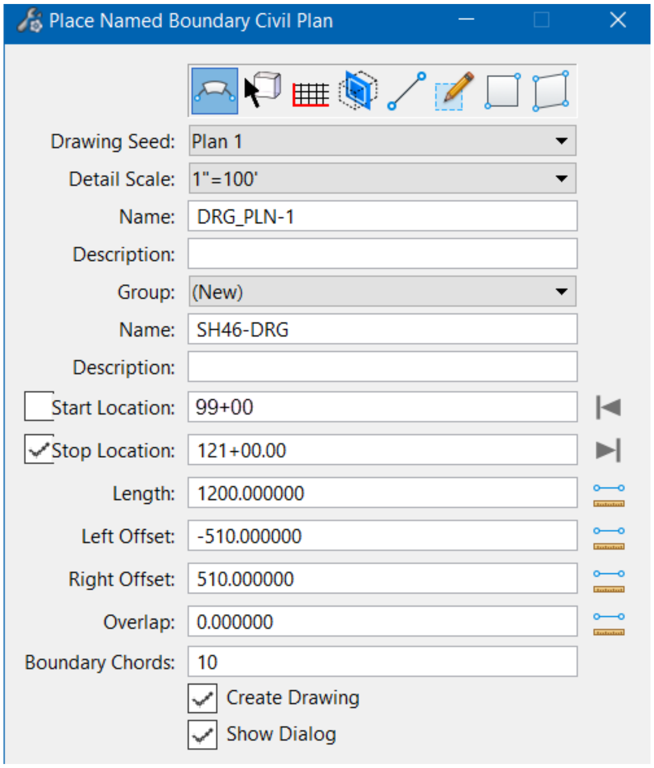
1. Create Plan Named Boundaries for Plan Sheets

a) Navigate to the ribbon > Named Boundaries > Select Named Boundary

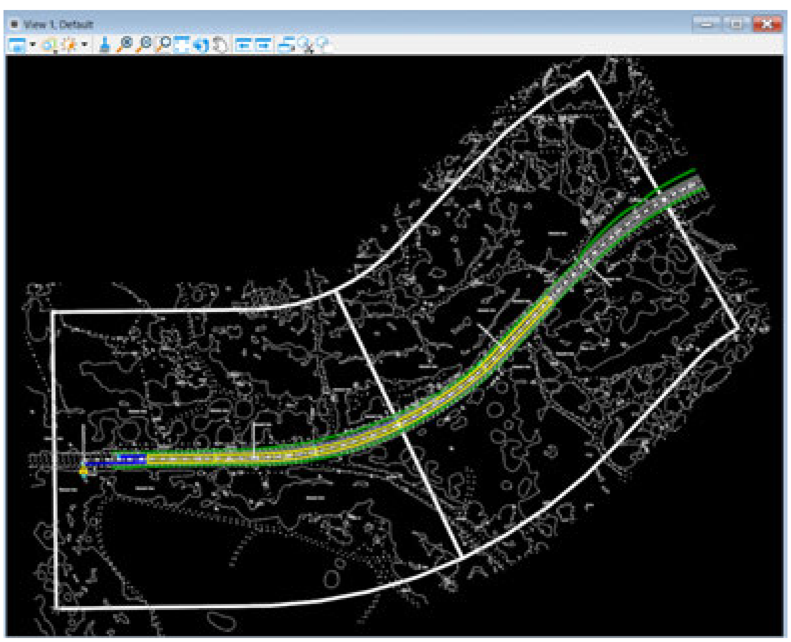


b) In the Place Named Boundary dialog box, select Civil Plan mode.
c) Navigate to Drawing Seed > select "Plan 1" and set the Detail Scale: 1" = 100. Once this seed is selected, other fields in the Place Named Boundary dialog box will be auto populated with default values configured in the seed file.
d) Follow the prompt in the bottom left corner of the screen.
e) Place Named Boundary Civil Plan > Identify Path Element > In the 2D view,
f) select your project's Geometry baseline.
g) Populate the Place Named Boundary Civil Plan dialog box as needed to show all drainage on plan sheets.

h) Move the cursor anywhere in the Plan view and follow the prompt on the bottom left corner of the screen > Left-Click through the prompts to complete.



i) The sheet boundaries are placed.



The Create Drawing dialog box will open after the sheet boundaries are placed.

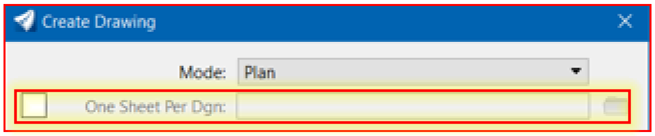
2. Create Plan Sheets

The new automated sheet creation process produces a pair of design file models for each “cut” sheet in the Drawing Model and Sheet Model.

Populate the Create Drawing dialog box with the following:

(a) At the top select Mode -**Plan**

There is a togle box to select whether you want to create a sheet per DGN. If your project is large and you have many sheets it is recommended to turn this toggle on and select the location and name of your DGN sheet files.



(b) Drawing Model Name:

(i) Change the Model Name field to SH46-DRG-1.

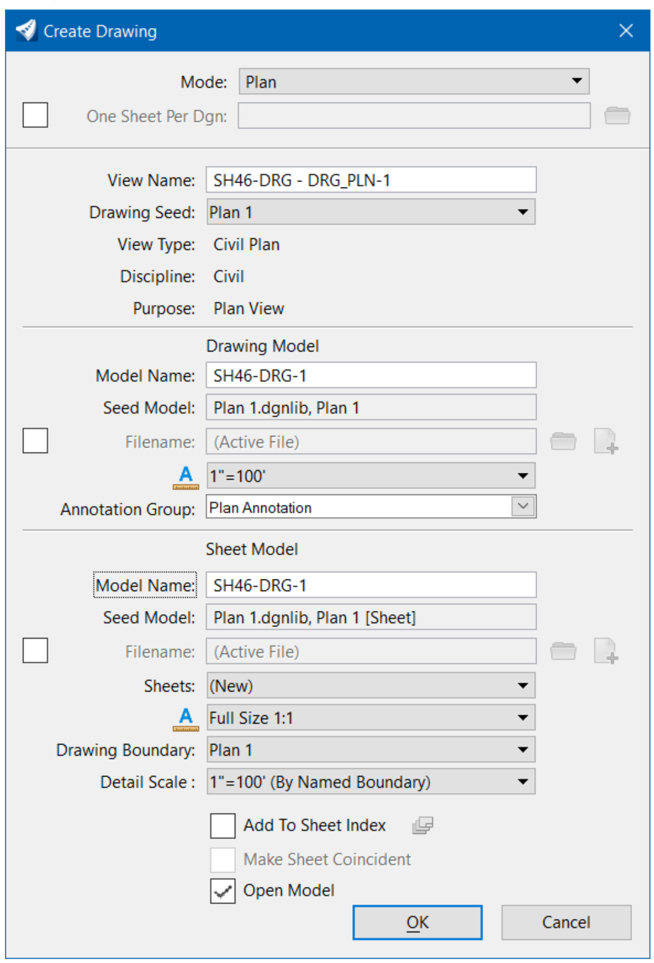
(ii) Verify the Drawing Model Annotation Scale is 1”=100’.

(c) Sheet Model:

(i) Change the Model Name field to SH46-DRG-1.

(ii) Verify the Sheet Model Annotation Scale is Full Size 1:1 and the Detail Scale is 1”=100’.

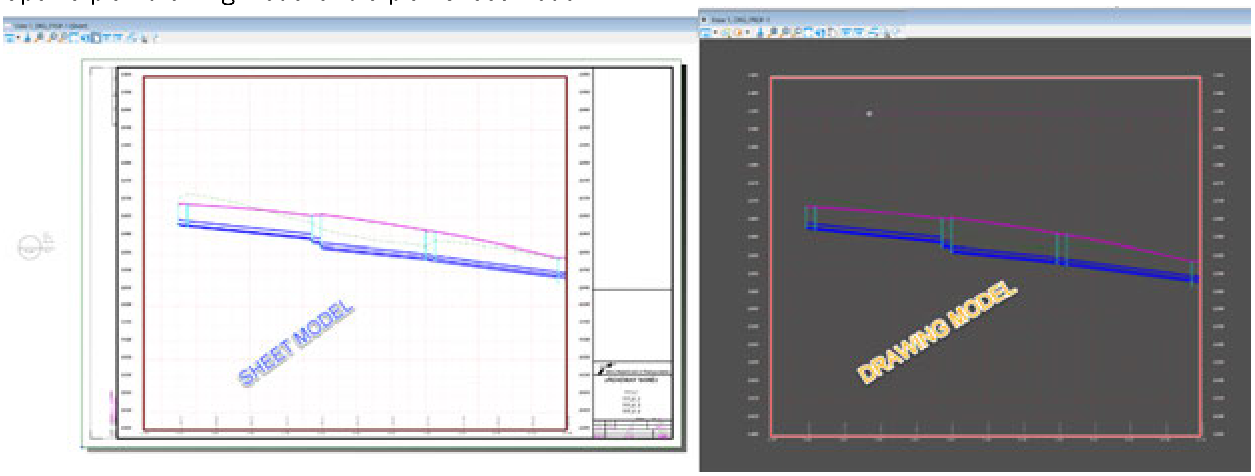
(iii) Leave the auto-populated values in the rest of the fields.Enable Add to Sheet Index -> select a folder from sheet index -> 05_Drainage (if you want to add to the Sheet Index)



(d) > Select OK

3. View the plan set

Open a plan drawing model and a plan sheet model.



4. Creating the Profile Sheets

1. Creating the Profile Named Boundaries for Profile Sheets

If the elevation difference in your profile is too large you will not be able to create plan and profile sheets. We will now show you how to create profile sheets.

(a) Navigate to ribbon > Named Boundaries > select Named Boundary.

(b) In the Place Named Boundary dialog box > select the Civil Profile mode.

(c) Navigate to Drawing Seed > select Profile 1.

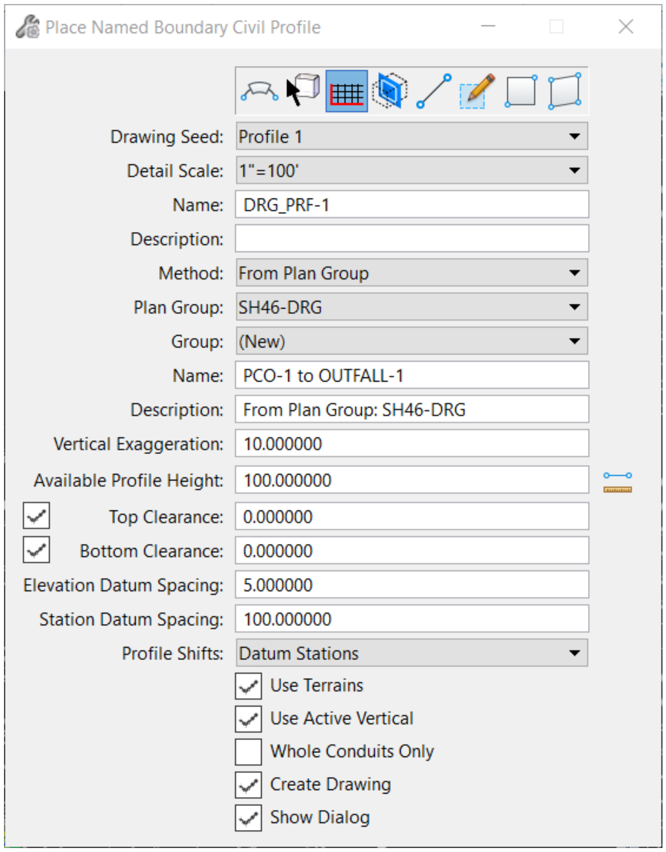
(d) Verify the Detail Scale is set to 1”=100.

(e) Once this seed is selected, other fields in the Place Named Boundary dialog box will auto-populate with default values configured in the seed file.

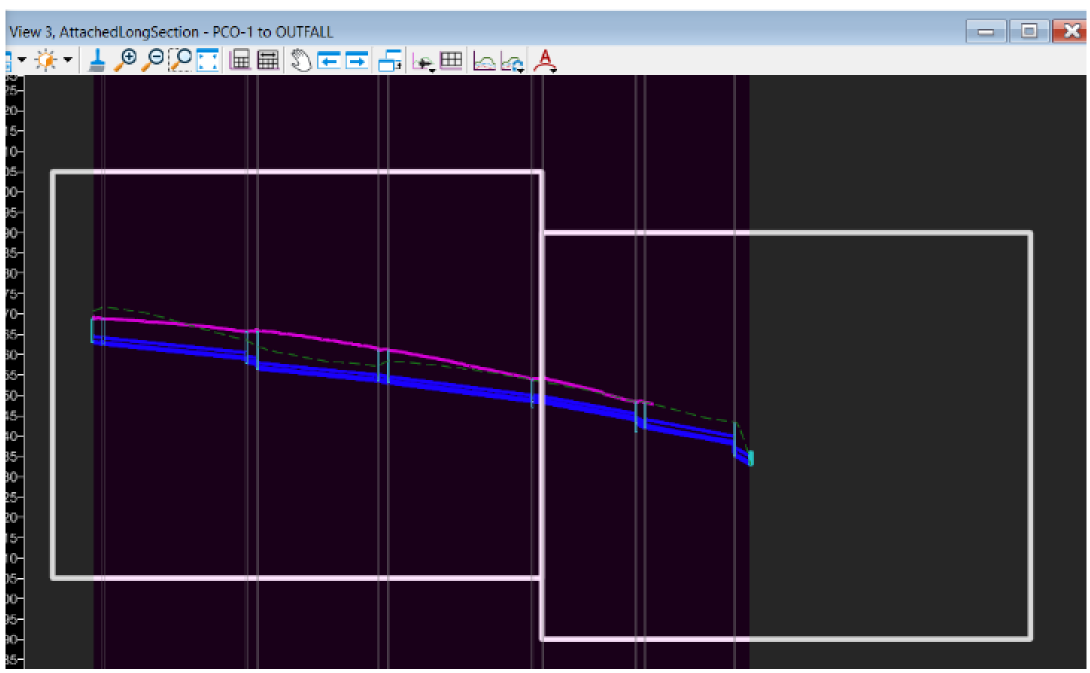
(f) Follow the prompt in the bottom left corner of the screen.

(g) Place Named Boundary Civil Profile > Identify Profile View > Left click in the profile view.

(h) Populate the Place Named Boundary Civil Plan dialog box as shown below.



(i) Move the cursor anywhere in the profile view and follow the prompt on the bottom left corner of the screen > Left Click through the prompts to place the profile boundaries.



The Create Drawing dialog box will open after the profile boundaries are placed.

2. Creating the Profile sheets.

(a) Populate the Create Drawing dialog with the following.

(b) Drawing Model:

(c) Change the Name field to DRG_PRF-1.

(d) Verify the Drawing Model Annotation Scale is 1”=100’.

(e) Sheet Model:

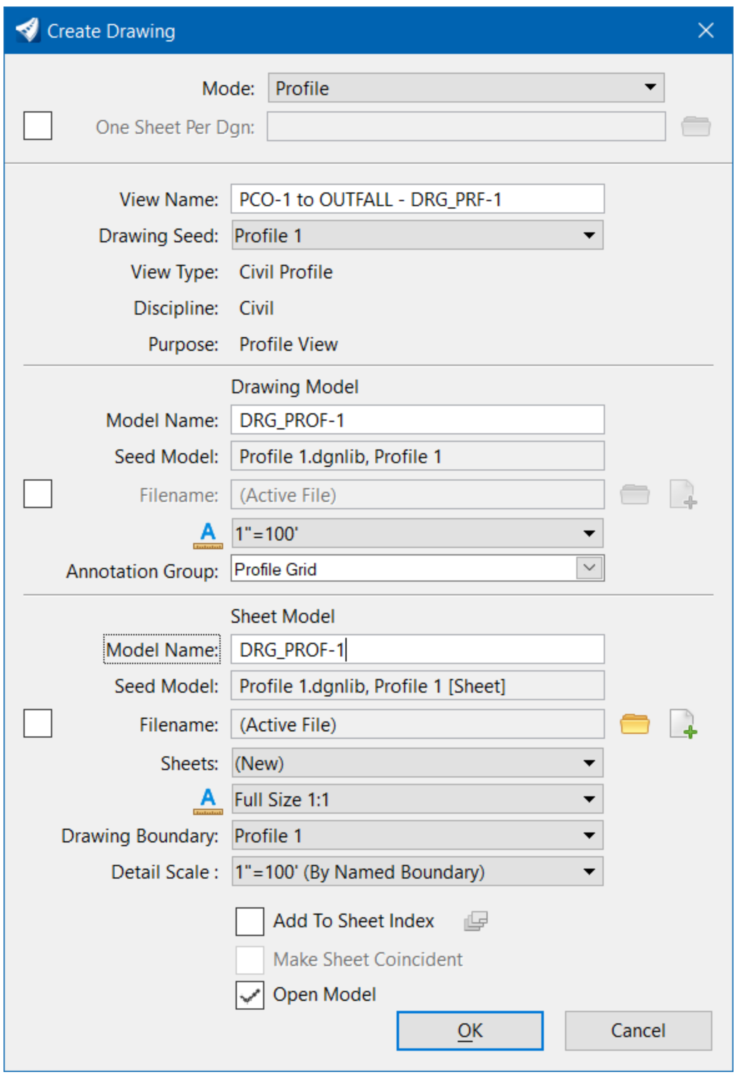
(f) Change the Model Name field to DRG_PRF-1.

(g) Verify the Sheet Model Annotation Scale is Full Size 1:1 and the Detail Scale is 1”=100’.

(h) Leave the auto-populated values in the rest of the fields.

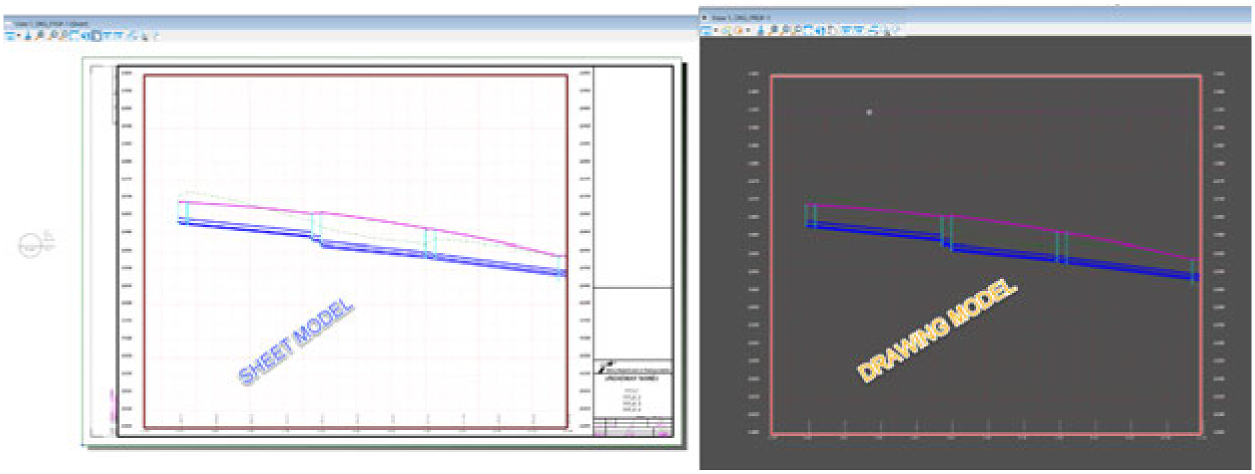
(i) Enable Add to Sheet Index -> select a folder from sheet index -> 05_Drainage (if you want to add to the Sheet Index).

(j) > Select OK.



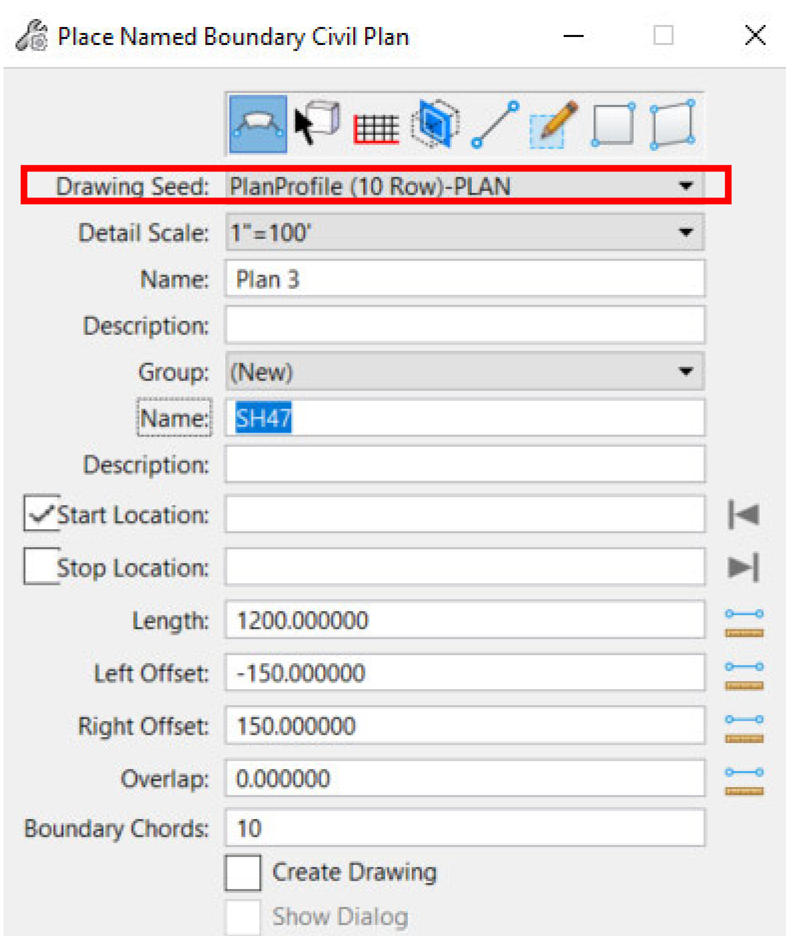
(k) Review the plan set

Open a profile drawing model and a profile sheet model.



5. Create a Plan Profile sheet

Follow the steps for creating the plan named Boundary but use the Plan Profile seed.



Do the same for the profile boundary but change the *Method* to **From Plan Group.**

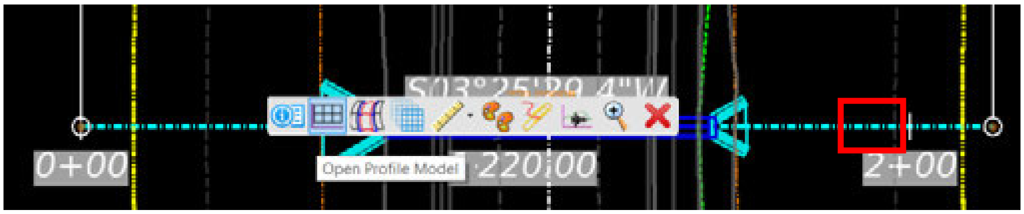
Follow the workflow for creating the sheets.

6. Create Cross Culvert Layout sheets,

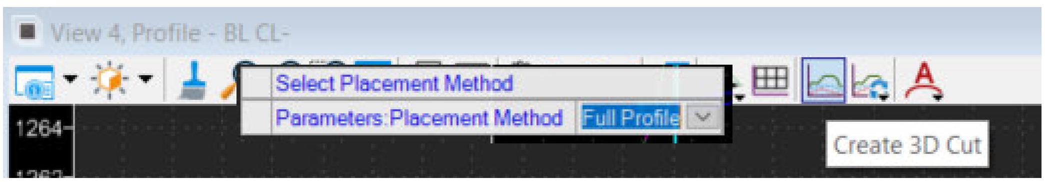
A culvert Layout sheet is a P&P sheet cut at the location of your proposed cross culvert.

To set up your file, create an alignment at the CL of your culvert and extend just pass your Right of Way. Give your CL a sensible name, you can classify your culverts by number as they appear up station of your CL alignment.

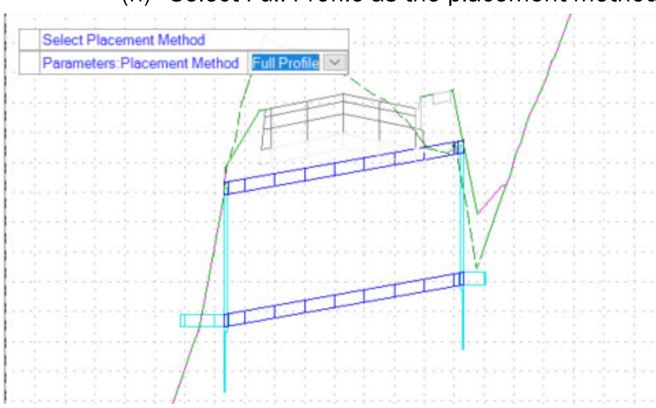
(l) Select the alignment and open the profile Model:



(m) In the profile view, select the Create 3D Cut Tool from the menu bar.



(n) Select Full Profile as the placement method.

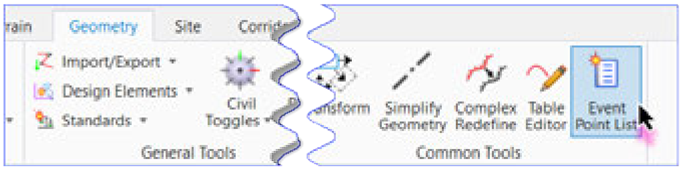


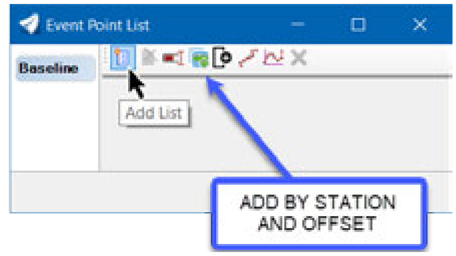
(o) Your profile view should adjust to show the proposed 3D elements.

(p) Follow the workflow for creating Plan and profile sheets.
NOTE: Make sure to select the appropriate scale for your drawing.

7. Cross section of the drainage crossings on roadways – Lateral sheets.

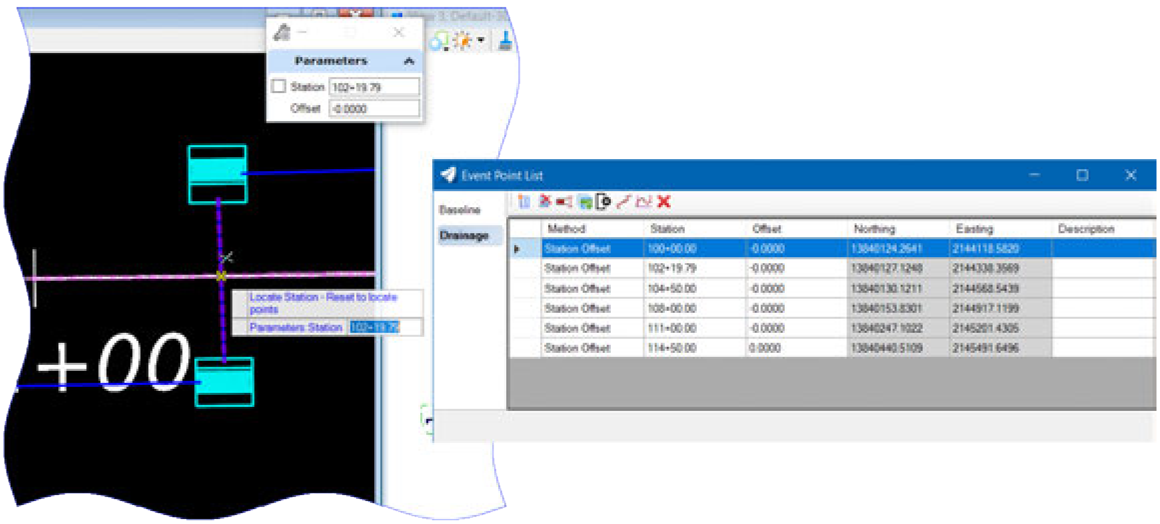
Use an event point list. This will allow you to create cross sections at each crossing when cutting cross sections.



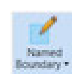


From the Geometry ribbon, select Common Tools group >. Select Event Point List and follow the prompts: Locate Alignment > Select your baseline geometry. The Event Point List box opens > Create a new list > Click the Add List icon, in the New Event Point List > Enter the Name: Drainage > Select OK.

Select the Add by Station and Offset icon, follow the prompts, and add the station and offset along the alignment where there is a drainage crossing.



1. Create Cross Section Named Boundaries

Navigate to ribbon > Named Boundaries group > Select Named Boundary 

In the Place Named Boundary dialog box, select the Civil Cross Section mode.

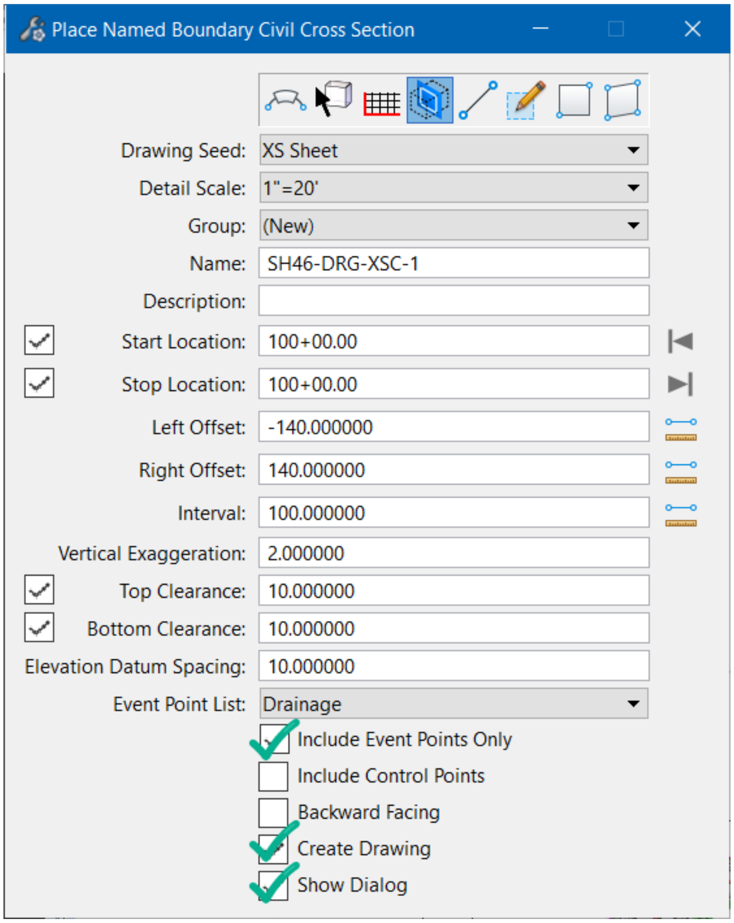
(a) Navigate to Drawing Seed > Select "XS Sheet." After this seed is selected, other fields in the Place

(b) Named Boundary dialog box will be auto populated with default values configured in the seed file.

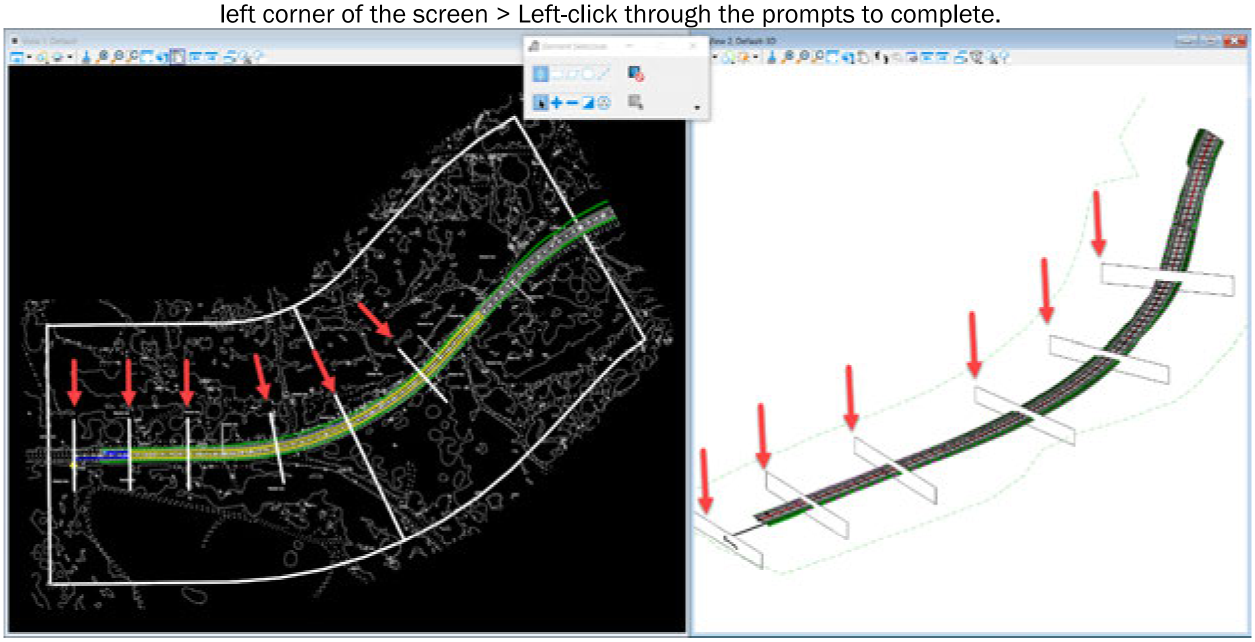
(c) Follow the prompt in the bottom left corner of the screen.

(d) Place Named Boundary Civil Cross Section > Identify Path Element – In the 2D view, select the baseline.

(e) Populate the Place Named Boundary Civil Cross Section dialog box as shown below.



(f) Move the cursor anywhere in the Plan view and follow the prompt on the bottom left corner of the screen > Left-click through the prompts to complete.



The Create Drawing dialog box will open after the sheet boundaries are placed.

2. Create Cross Section Sheets

The fields are auto populated based on the Drawing Seed file selected in the earlier section XS Sheet.

(a) Drawing Model

(b) Change the Model Name field to DRG_XSC-1.

(c) Verify that the Drawing Model Annotation Scale is set to 1” = 20’. **Note:** This is the prefered scale, but it may vary according to the size of your section

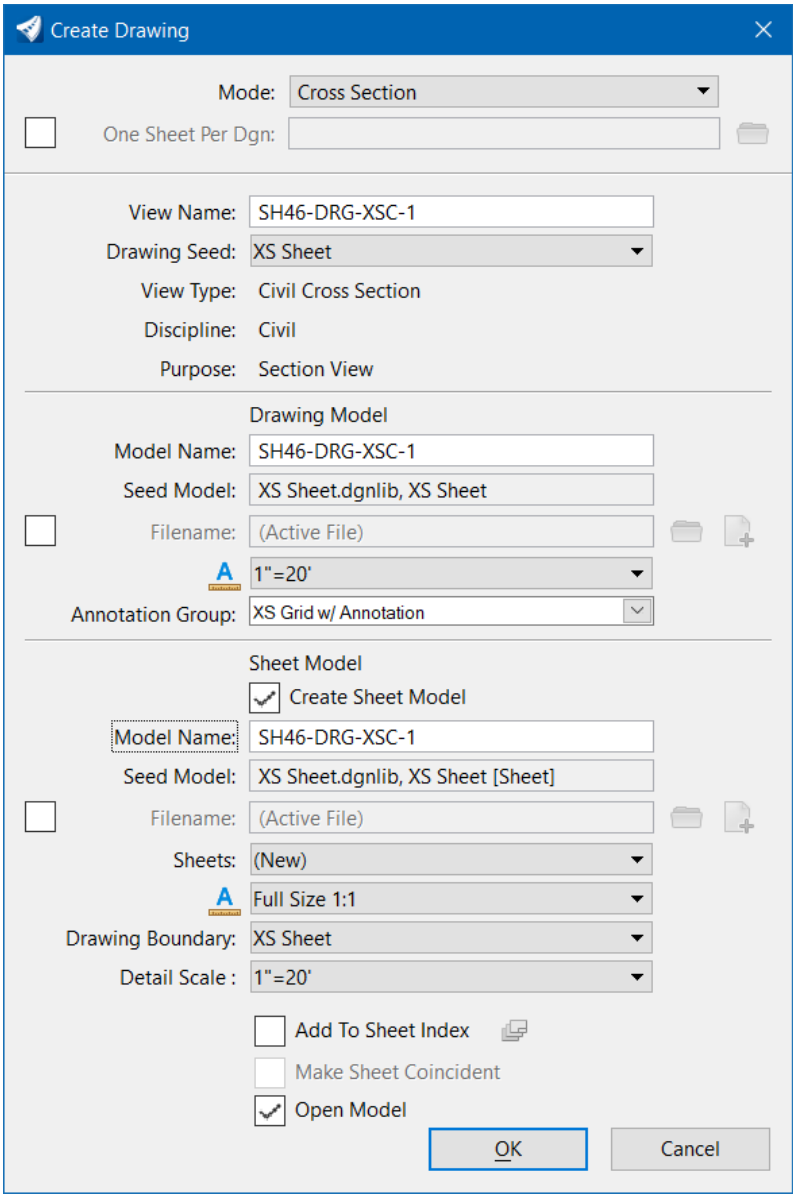
(d) Sheet Model

(e) Verify the Sheet Model Annotation Scale is Full Size 1:1 and the Detail Scale is 1” = 20’.

(f) Leave the auto populated values in the rest of the fields.

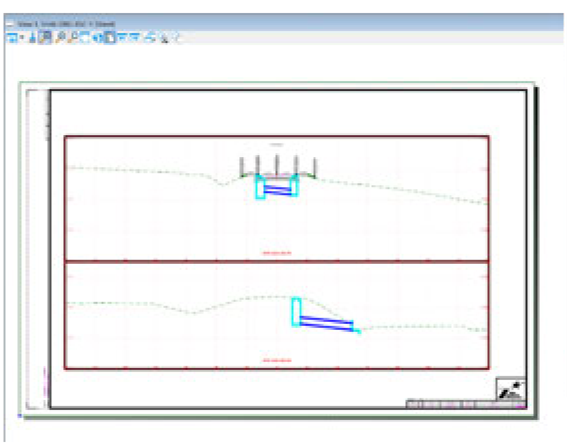
(g) Enable Add to Sheet Index > Select a folder from sheet index > 05_Drainage (if you want to add to the Sheet Index)

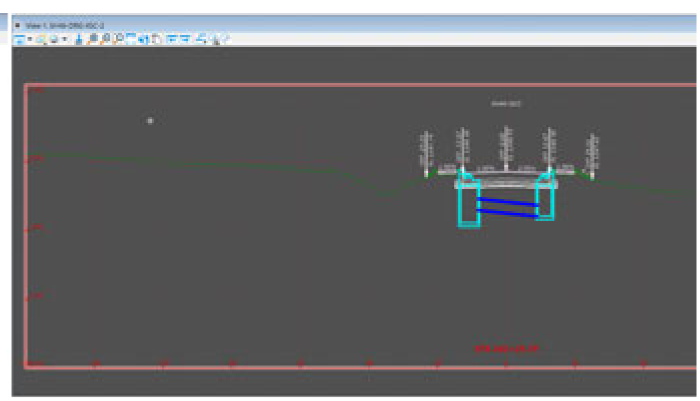
(h) > Select OK.



3. Review the cross section set

Open a cross section drawing model and a cross section sheet model.

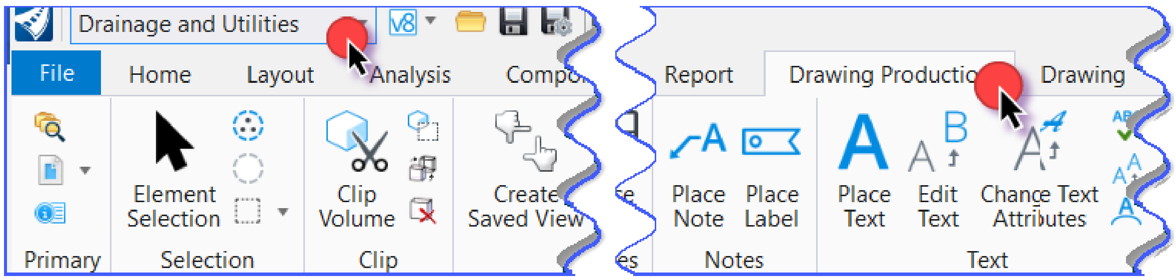


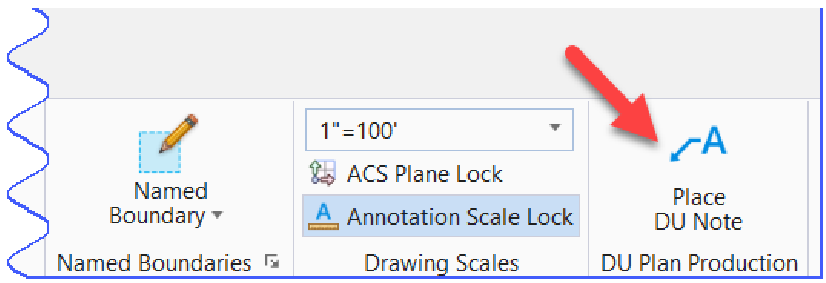


Plans Production Annotating the Drainage Plan set

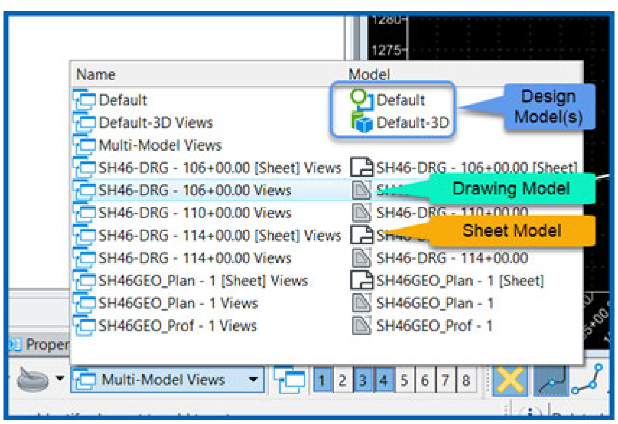
With this current version of TxDOT ORD DU workspace, the annotation is not yet automated, but we have created a tool in the Drainage and Utilities ribbon to help with it. Improvements will be coming soon.

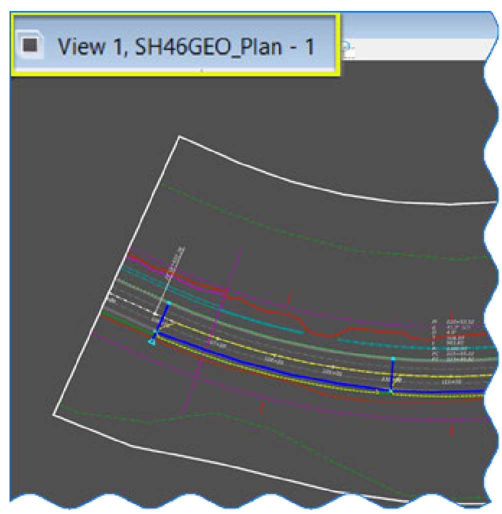
Workflow: Drainage and Utilities; Ribbon Tab: Drawing Production DU; Ribbon Group: DU Plan Production





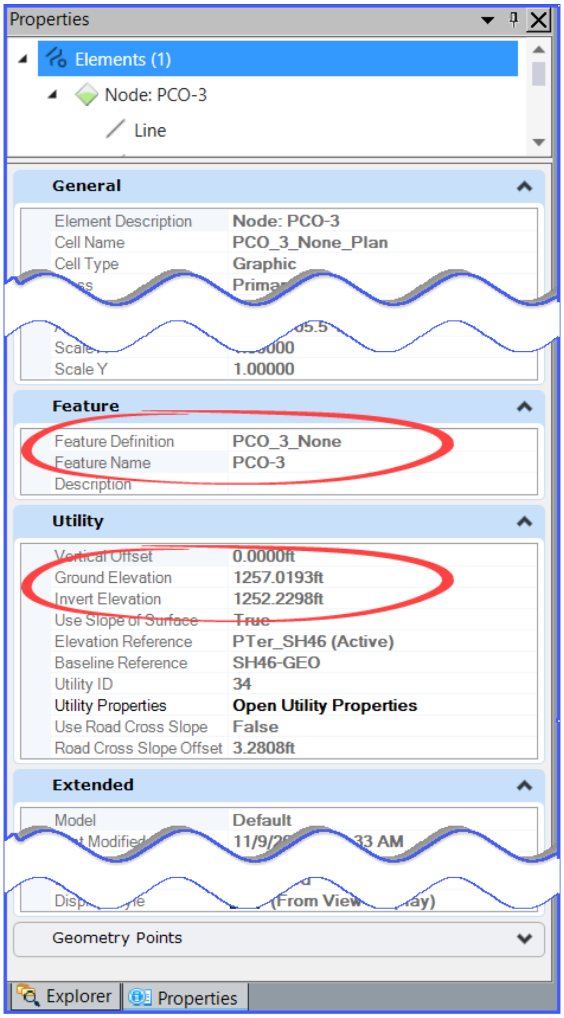
Annotation should be in the drawing model of your plan production file. Open your plan view to the drawing model. Open a Plan Drawing Model.

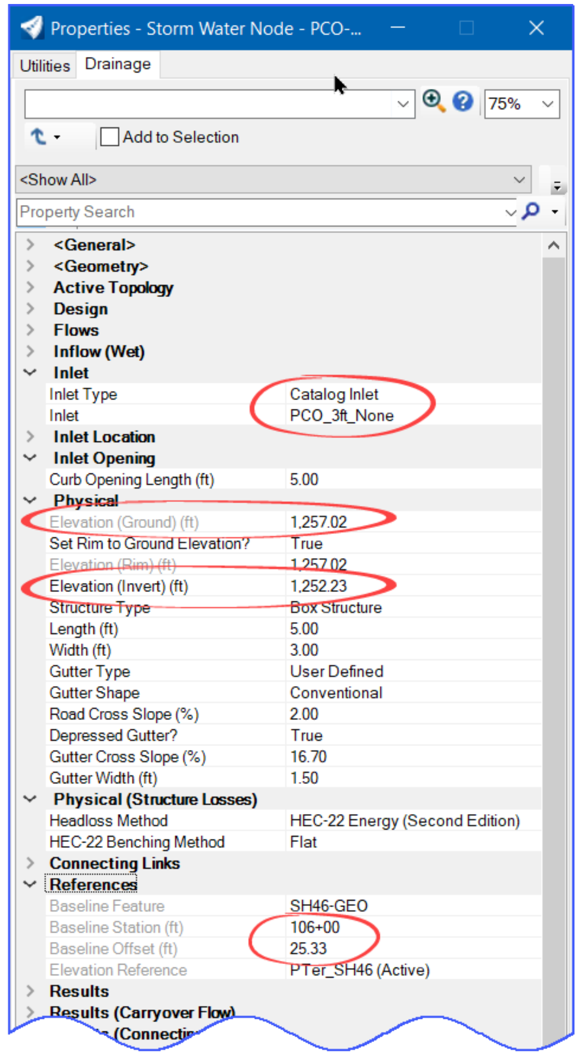




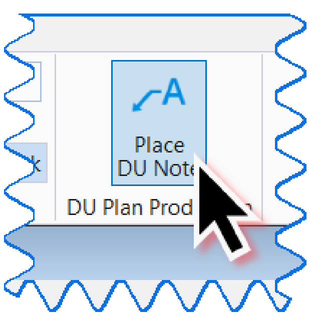
Use the Element Selection tool to highlight the drainage item you want to annotate.

Click on the structure: Inlet, in the properties box all the information needed to label this structure is there. Also open the Utility Properties if more information is needed (Station and Offset). From here you will copy and paste into the Text Editor the information needed for the drainage structure/Conduit.

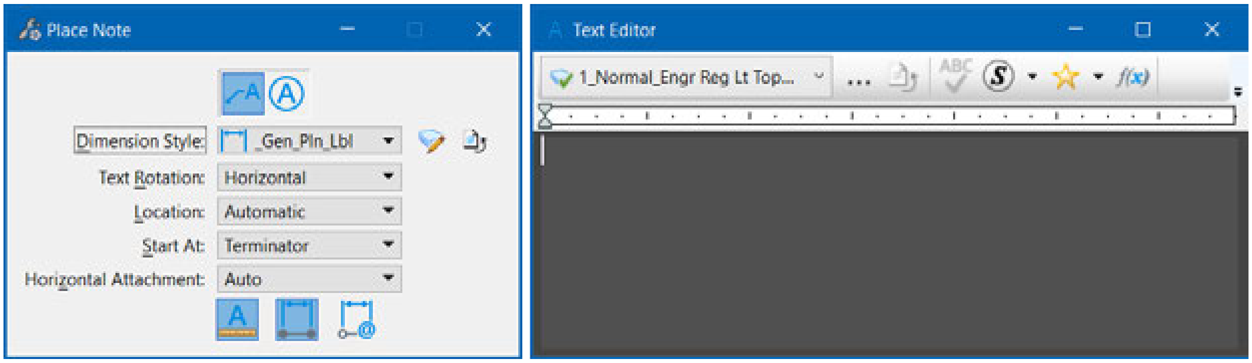




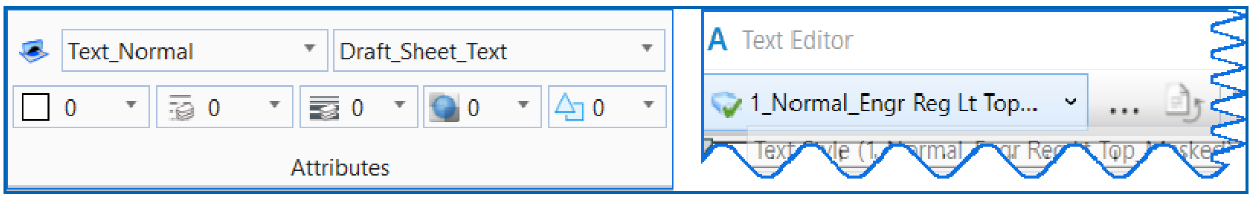
Select the Place DU Note command.



This will start the command to place a label/note in your drawing model. The "Place Note" toolbox should appear along with the Text Editor box.

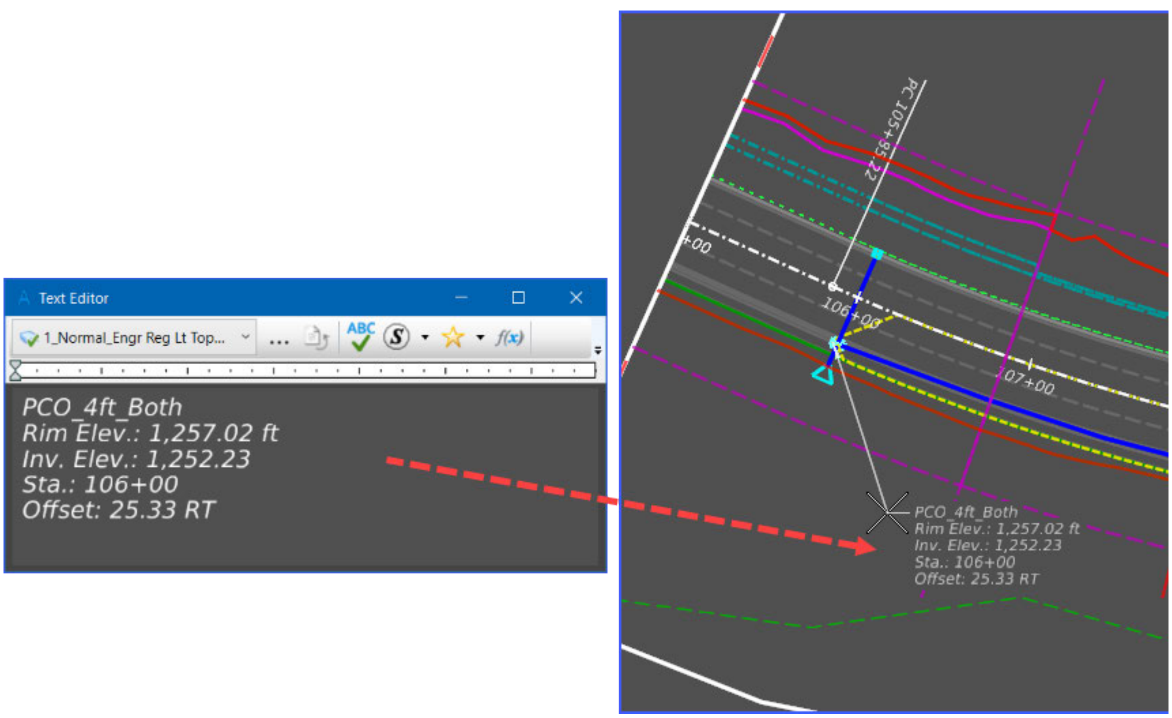


The command will pick the correct element template (attributes), text style, and dimension style.

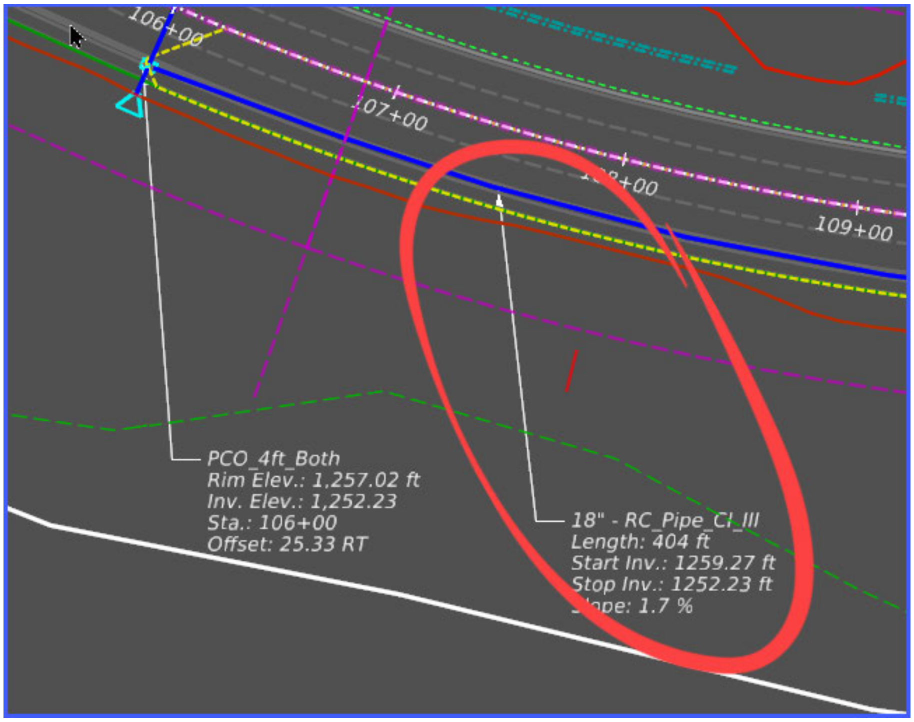


Ready to fill in the drainage information for the Place Note command.

In the properties box, select what you want to copy and paste, highlight, right-click, copy, move to Text Editor, right click, paste. Continue until all text needed has been added to the Text Editor window. Next click in the plan view on the structure once more, click again to where you want the text to be placed. Repeat as needed for all drainage items.



Do the same with the conduits, using Element Selection tool, select the conduit, click on the “Place DU Note” command, copy, and paste from the properties items as needed, select the conduit once more, place the text.



To label/annotate the catchment area (drainage area) you will change the “Place DU Note” tool from ‘Place Note’ to ‘Place Callout.’ As before, using the Element Selection tool, click on the drainage area for the properties, copy and paste, and place the callout.