OpenRoads Designer User Manual

U.S. Department of Transportation
Federal Highway Administration

Chapter 25

DRAINAGE AND UTILITIES TOOLS

2021 R2 Update 10 – Version 10.10.21.04 – WorkSpace 10.10.V
January 2023

Chapter 25 Drainage and Utilities Tools

This chapter covers Drainage and Utilities tools, which are used to model and analyze stormwater, wastewater, water distribution, and dry utility networks.

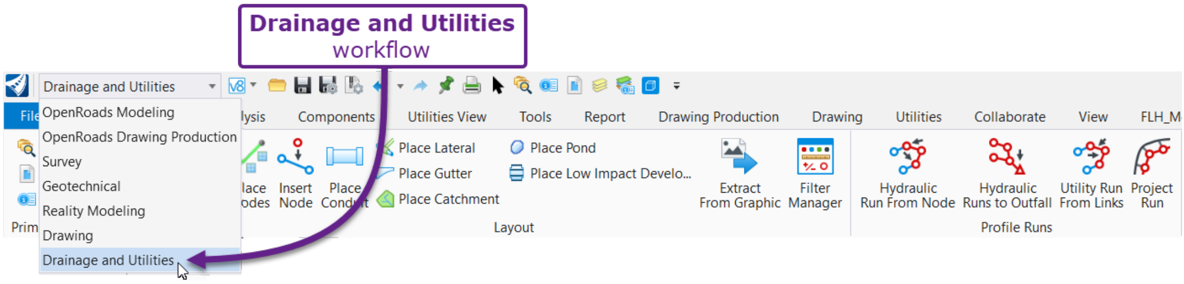
The focus of this chapter is modeling and analyzing stormwater networks, specifically culvert and storm sewer systems.

Using Drainage and Utilities tools, drainage areas can be delineated and analyzed with a design storm to produce a peak flow rate. The peak flow rate can be routed into a culvert or storm sewer network for hydraulic analysis.

TABLE OF CONTENTS

25A – INTRODUCTION TO DRAINAGE AND UTILITIES TOOL

Drainage and Utilities tools are used to model and analyze stormwater, wastewater, water distribution, and dry utility (i.e., Fiber Optic, Gas, and Electrical) networks. These tools are found in the Ribbon, under the **Drainage and Utilities** workflow:



The Drainage and Utilities tools can be used for following purposes:

- **Culvert Modeling and Analysis:** The Culvert modeling capabilities is similar to HY-8. The culvert span, material, slope, and inlet/out configurations can be modeled to analyze the culvert's capacity and determine road overtopping flows. Each culvert can be assigned an upstream catchment area (drainage area) to automatically calculate flow into the inlet. In analysis, a hydraulic profile for the culvert can be generated.
- **Stormwater Network Modeling, Analysis, and Automated Design:** A network of catch basins, inlets and storm pipe can be modeled and analyzed for capacity. Each inlet can be assigned a catchment area to automatically calculate flow into the inlet. In analysis, a hydraulic profile for the storm pipe runs can be generated. Optionally, a Design Scenario can be run to automatically design components of the stormwater network. For example, pipe sizes can be automatically sized to optionally accommodate the design storm.
- **Hydrologic Basin Modeling and Peak Discharge (Q) Calculations:** A drainage basin can be modeled to determine the peak discharge flowrate for different storm events. To utilize hydrologic analysis, an IDF Curve must be obtained for the project site.
- **Wastewater Network Modeling:** The layout of wastewater networks and infrastructure can be modeled. Each lateral or node can be assigned a load to analyze the capacity of the network.
- **Water Distribution Network Modeling:** The layout water distribution networks and infrastructure can be modeled. In analysis, head losses and pressure can be calculated.
- **Dry Utility Network Modeling:** The layout out gas, communications (i.e., fiber optic), and electrical lines and infrastructure can be modeled. For sheet production purposes, profiles can be generated from the dry utility network.

NOTE: This chapter focuses on steady-state analysis of drainage systems. A steady-state analysis does NOT consider the distribution of flow over time. In steady-state analysis, a single peak flowrate is computed and analyzed for each structure and conduit in the drainage system.

Detention basins and infiltrations systems (i.e., bioswales and permeable pavers) is NOT covered in this chapter because these features require an unsteady-state analysis. Detention basin and infiltration systems analysis can be performed in ORD but requires a subscription to the paid add-on software: CivilStorm.

WARNING: The calculation engine for the Drainage and Utilities tools differs from other frequently used software such as HY-8, HEC-HMS, and HEC-RAS. Results may differ depending on which software is used. For large structures and critical crossings, results generated from the ORD Drainage and Utility tools should be scrutinized for acceptability.

WARNING: The FLH WorkSpace contains limited resources for Drainage and Utilities modeling. Basic structures and conduit types are included in the FLH WorkSpace. However, structures specific to local agencies may NOT be found in the FLH WorkSpace.

WARNING: The Drainage and Utilities tools should ONLY be used after the layout and invert elevations of the drainage system is manually designed by the User. Drainage system layout should be drawn with MicroStation or ORD Tools.

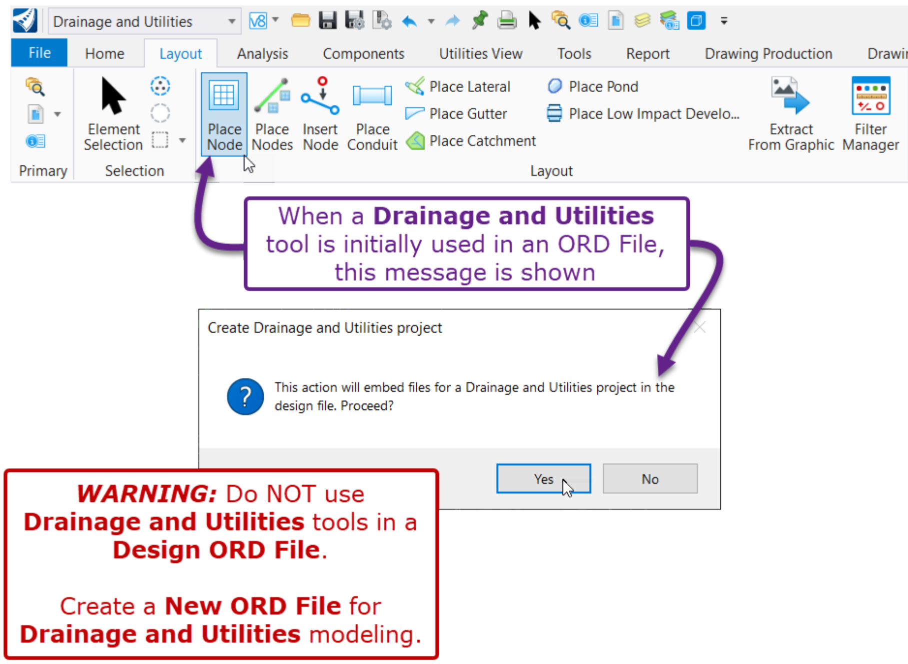
Ensure the layout of the drainage network is constructable. Ensure culvert and conduits meet minimum cover requirements. Ensure skewed culverts have sufficient length so that the inlet/outlet ends are NOT buried by the road embankment.

The Drainage and Utilities tools are intended for hydraulic analysis of a drainage system. The hydraulic analysis may inform to appropriate pipe sizes and capacities. However, it is the User’s responsibility to ensure the layout is constructable and meets FLH and project stakeholder’s standards.

25A.1 Drainage and Utilities Basics

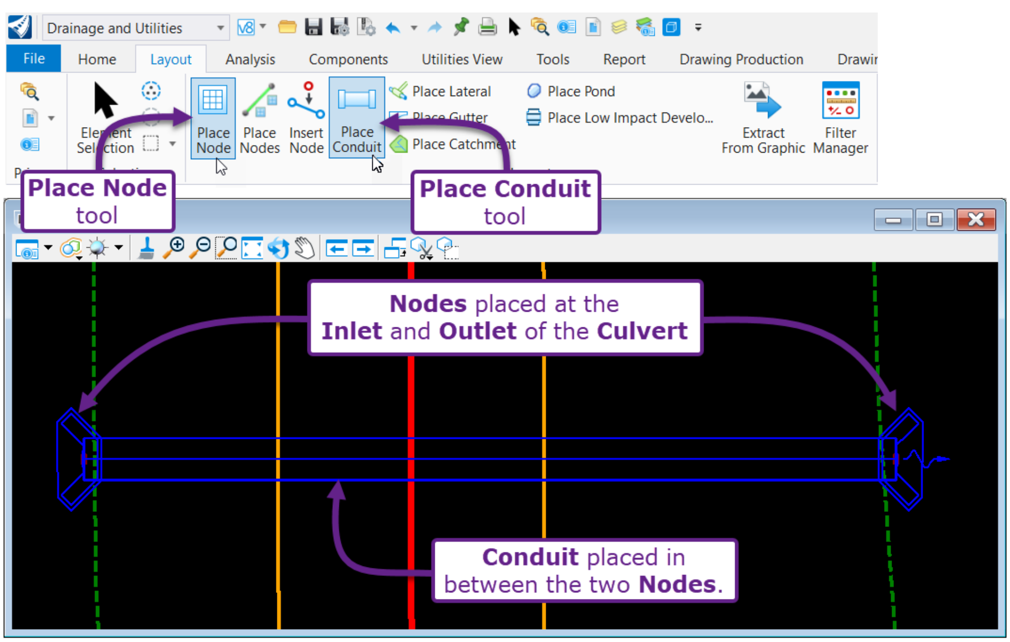
When a Drainage and Utilities tool is first used, a Project File is automatically created and embedded into the ORD File.

WARNING: When using the Drainage and Utilities tool always start with a new ORD File. Do NOT use the Drainage and Utilities tools in a Design ORD File, such as the Corridor ORD File. **Placing Drainage and Utilities elements in a Corridor ORD File may corrupt file.**

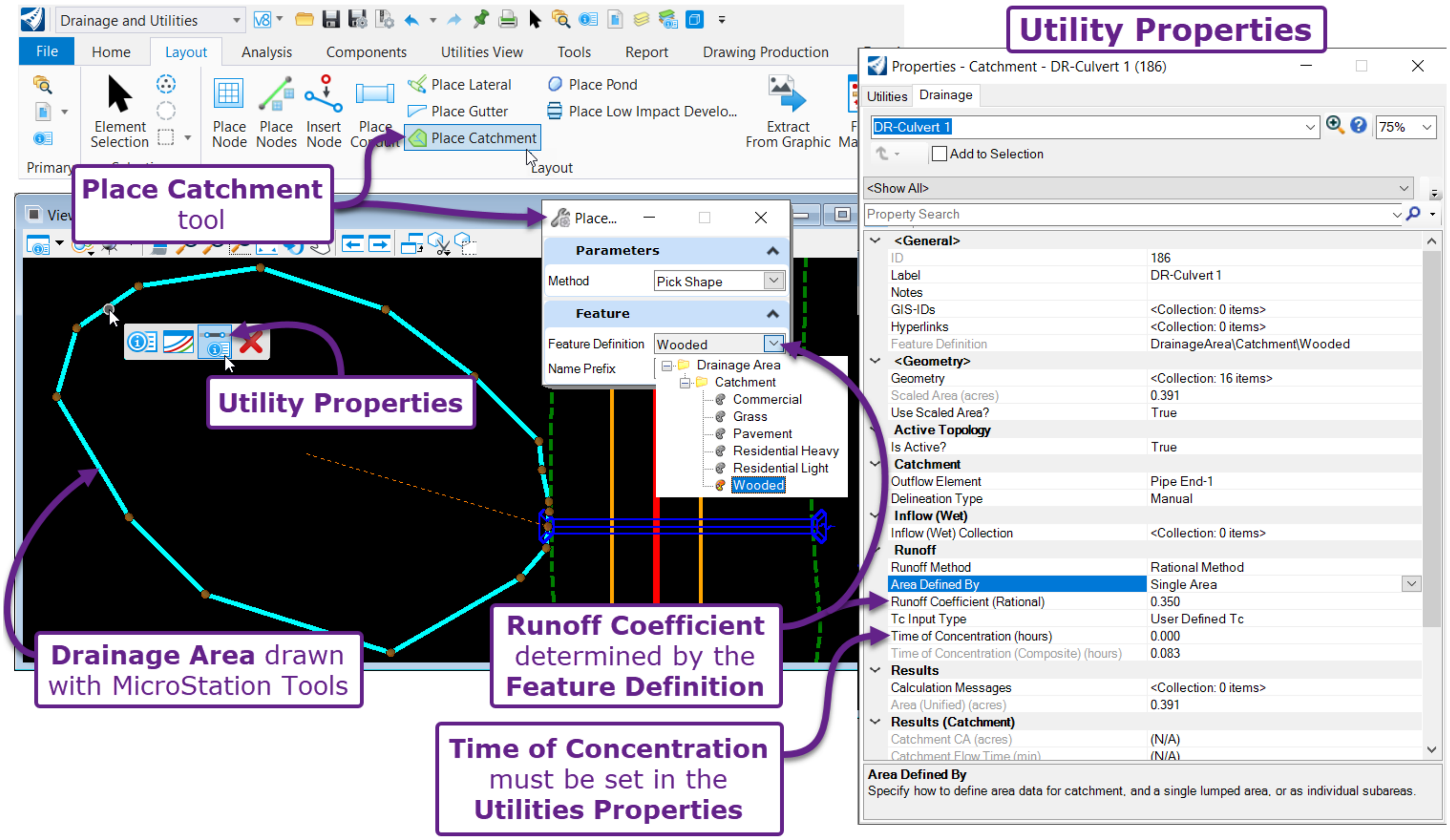


Drainage and Utility networks are primarily modeled with **Nodes** and **Conduits** elements. Nodes represent inlets, outlets, openings, outfalls, structures, manholes, and other important locations in the network. Conduits are placed in between Nodes to represent pipes, channels, and other means of hydraulic conveyance. A Conduit must be placed between two Nodes.

For example, a simple culvert is modeled with Nodes placed at the inlet and outlet locations. A Conduit is placed between the inlet and outlet Nodes to model the culvert material and geometry.



A **Catchment** represents a drainage area and is used to calculate runoff flowrates generated from the drainage area. A Catchment must be assigned to a Node. The flow generated by a catchment is routed to the Node. In the example shown below, a Catchment is assigned to the inlet Node of the Culvert.



TIP: A Catchment can be drawn with MicroStation Tools (i.e., *Place Smart Line* tool) to create a closed shape. When using the *Place Catchment* tool, use the **Pick Shape** method to select the closed shape.

NOTE: Without a paid add-on software, only the Rational Method can be used for hydrologic analysis of Catchments. To calculate runoff using the Rational Method, a Time of Concentration and Runoff Coefficient must be assigned to the Catchment element in the Utility Properties menu. Also, an IDF curve must be obtained and imported into ORD, before the analysis is performed. Importing IDF curves into ORD is shown in *25C – Create Storm Data and Scenarios for Catchments*.

25A.2 The Utility Properties Menu

All Drainage and Utilities elements (i.e., Nodes, Conduits, Catchments, Gutters, Ponds) have a set of hydraulic options that are shown in the *Utility Properties* menu.

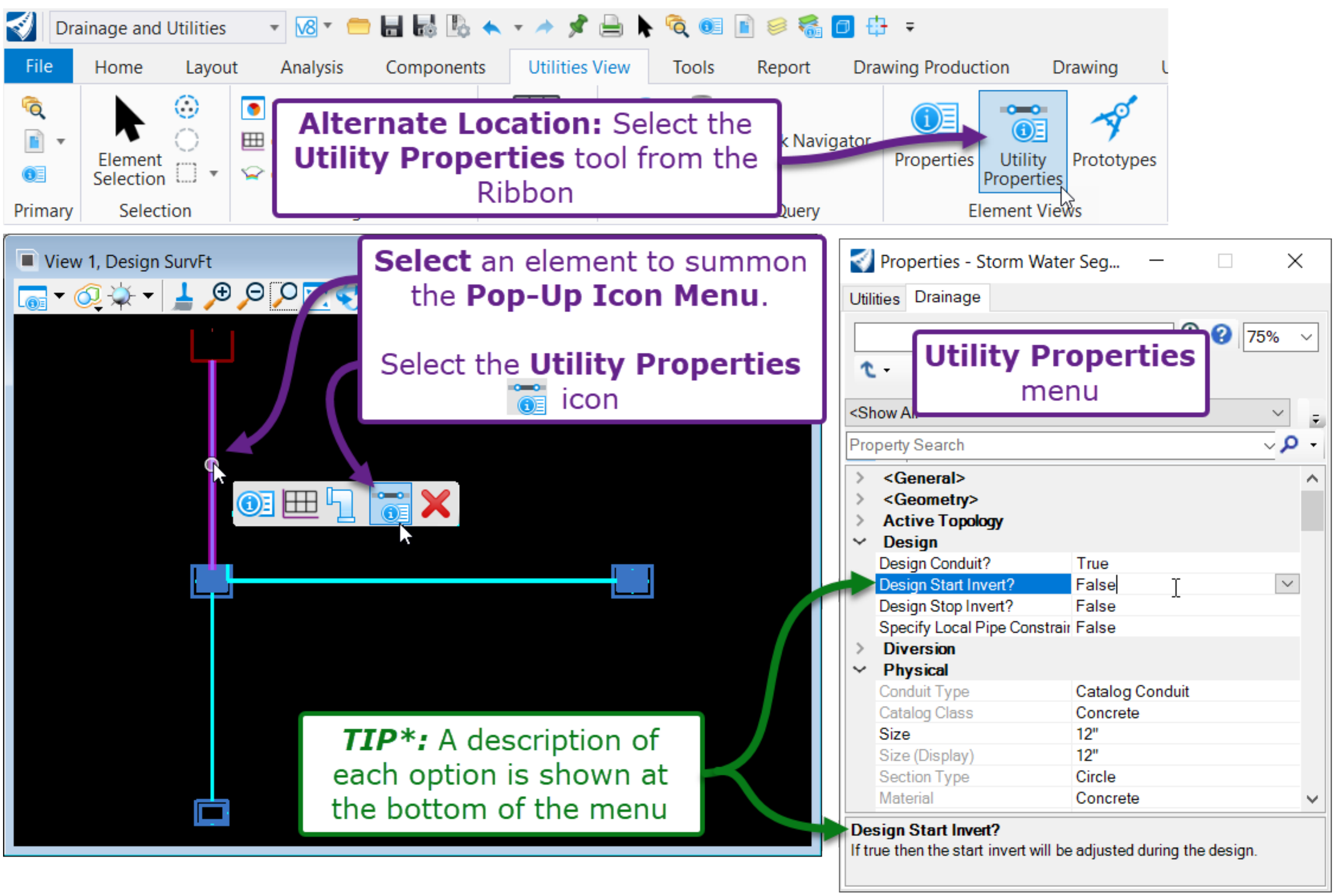
The Utility Properties governs all aspects of the analysis of utility networks. When performing Drainage and Utilities modeling and analysis, always keep the Utility Properties menu open.

NOTE: Settings shown in the conventional Properties box do NOT affect the hydraulic analysis of the element.

The *Utility Properties* menu can be opened by selecting a Drainage and Utilities element and summoning the Pop-Up Icon Menu. Accessing the Pop-Up Icon Menu is shown in *1A.2.c Pop-Up Icon Menu*.

Alternatively, the Utility Property menu can be accessed from the Ribbon in the following location:

Drainage and Utilities → Utilities View → Element Views



TIP*: When an option is selected (highlighted), a brief description and the effects of an option is shown at the bottom of the menu.

25A.3 General Workflow for Drainage and Utilities Modeling and Analysis

The flow charts below show the general procedure for modeling and analyzing drainage networks. These procedures are also relevant to wastewater, water-distribution, and dry utility networks.

WARNING: Create a new ORD File for placement of Drainage and Utilities elements. Do NOT use Drainage and Utilities tools in a Design ORD File (i.e., the Corridor ORD File).

Workflow #1: Model the Drainage Network with Nodes, Conduits, and Catchments:

Layout the Drainage Network with ORD Tool or MicroStation Tools

The Drainage and Utilities tools should ONLY be used after the layout and invert elevations of the drainage system is manually designed by the User.

Ensure the layout of the drainage network is constructable. Ensure culvert and conduits meet minimum cover requirements. Ensure skewed culverts have sufficient length so that the inlet/outlet ends are NOT buried by the road embankment.

Place Nodes Elements

Node elements represent culvert end treatments (FES and wingwalls), inlet catch basins, outfalls and other structures.

25B.1 Create the Inlet and Outlet Nodes

Place Conduit Elements Between Nodes

Conduits represent pipes and open-channels. Place a Conduit element between each Node.

25B.2 Create the Conduit (Pipe)

Place Catchment Elements (Drainage Area) [OPTIONAL]

Catchments represent the drainage area that flows into a Node. Trace the perimeter of a drainage area with a MicroStation Element and convert it to a Catchment.

Alternatively, if the flow into a Node is known, it is unnecessary to model Catchment areas and import an IDF curve into ORD (Workflow #2).

25B.4 Create the Catchments (Drainage Areas)

Edit Node, Conduit, and Catchment Settings in the Utility Properties

The Utility Properties menu controls hydraulic settings for Nodes, Conduits, and Catchments.

25B.3 Edit Node and Conduit Hydraulic Settings in the Utility Properties

If the flow into a Node is known, then set the known flowrate in the Utility Properties menu.

25B.3.a Set a Known Flow to the Inlet Node.

Workflow #2: Import an IDF Curve into the ORD Software and Setup Scenarios:

If using Catchments to generate rainfall runoff flowrates, then an IDF Curve must be imported into the ORD software. After importing the IDF Curve, a Scenario is created to specify the return event to be used in the analysis. See *25C – Create Storm Data and Scenarios for Catchments*.

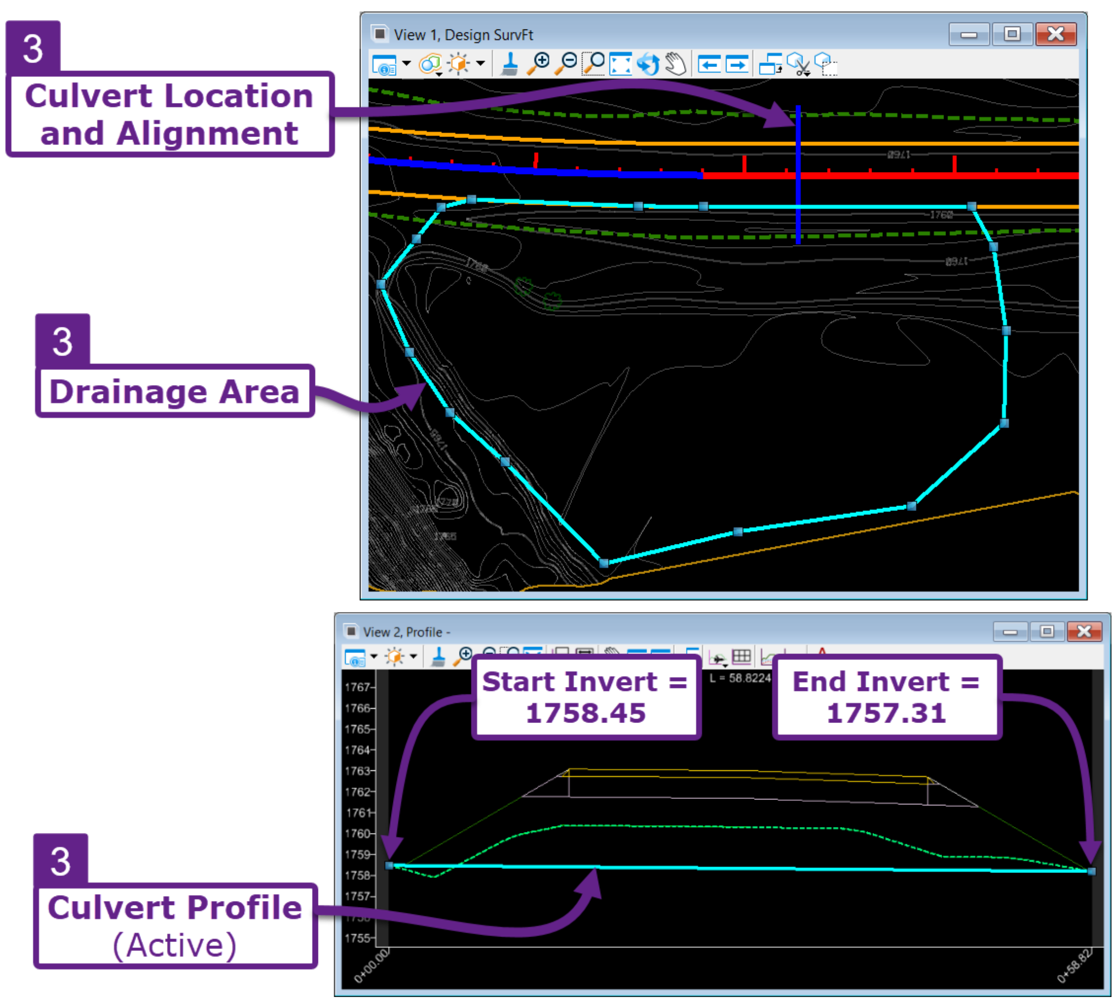
Workflow #3: Run an Analysis of the Drainage Network: Using the *Compute Center* tool, the IDF Curve and return event Scenario is run to generate a flow rate from the Catchment element. The Catchment flow rates are routed into the downstream Nodes and the hydraulics of the drainage network is analyzed. See *25D – Run an Analysis Scenario with the Compute Center tool*.

Workflow #4: Review Results and Generate Hydraulic Profiles: After running the analysis, review the hydraulic results and generate hydraulic profiles. If necessary, edit the layout and/or Utility Properties for elements in the drainage network and re-run the analysis. See *25E – Results: Creating Reports, Tables, and Profiles*.

25B – MODEL A CULVERT – BASIC WORKFLOW

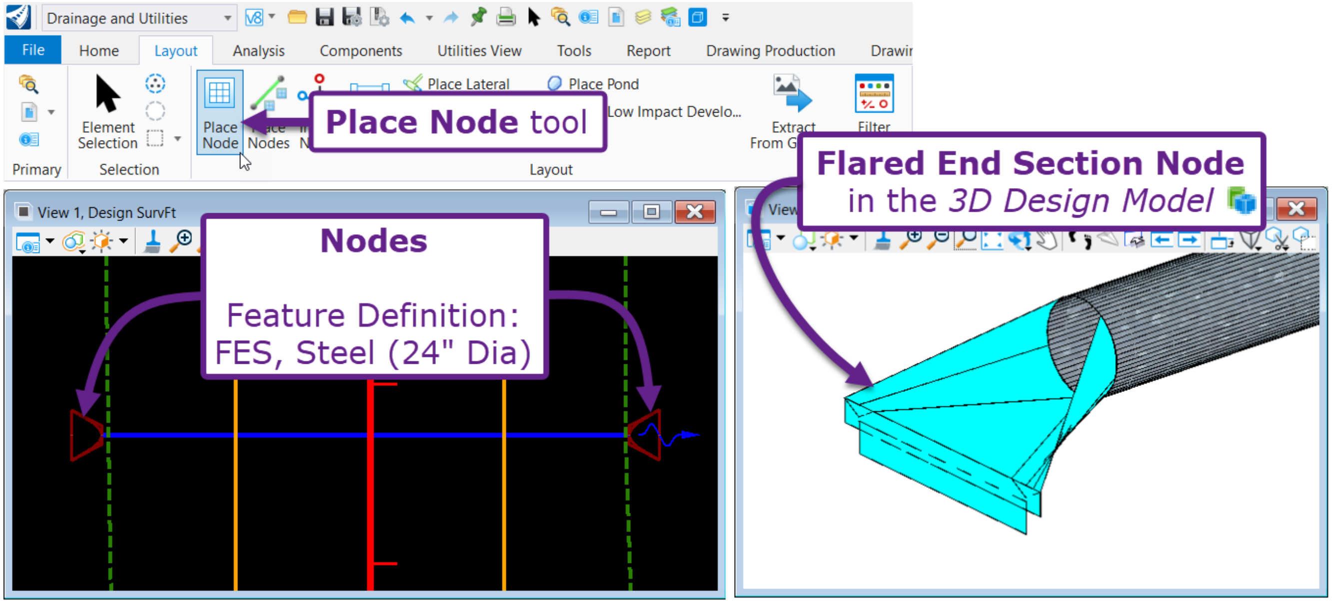
In this workflow, a simple culvert is modeled. The culvert network consists of a two Nodes, a Conduit, and a Catchment.

1	Create a new ORD File. ***WARNING:*** Do NOT place Drainage and Utilities elements in a Design ORD File (i.e., the Corridor ORD File). Use a **2D Seed File** when creating the new ORD File. For more information on Seed Files, see *3B.3 The Seed File*. In the new ORD File, **reference** in the Survey ORD File and all relevant Design ORD Files.
2	Activate the Existing Ground Terrain Model.
3	The culvert layout should be designed before placing Drainage and Utilities elements. The inlet and outlet Nodes will be placed at the end points of the Culvert Centerline Alignment. A Culvert Profile should be designed to determine the invert elevations for the inlet and outlet Nodes. Use ORD Geometry Tools to draw the Culvert Centerline Alignment and Culvert Profile. ***Activate* the Culvert Profile.** Use the *Place SmartLine* tool to draw the Drainage Area shape. The Drainage area Shape will be directly selected to create the Catchment element. ***NOTE:*** If the peak flowrate into a Culvert is already known, then creating a Catchment is unnecessary. The known flow can be directly inputted into the upstream inlet Node, which is shown in *25B.3.a Set a Known Flow to the Inlet Node*.

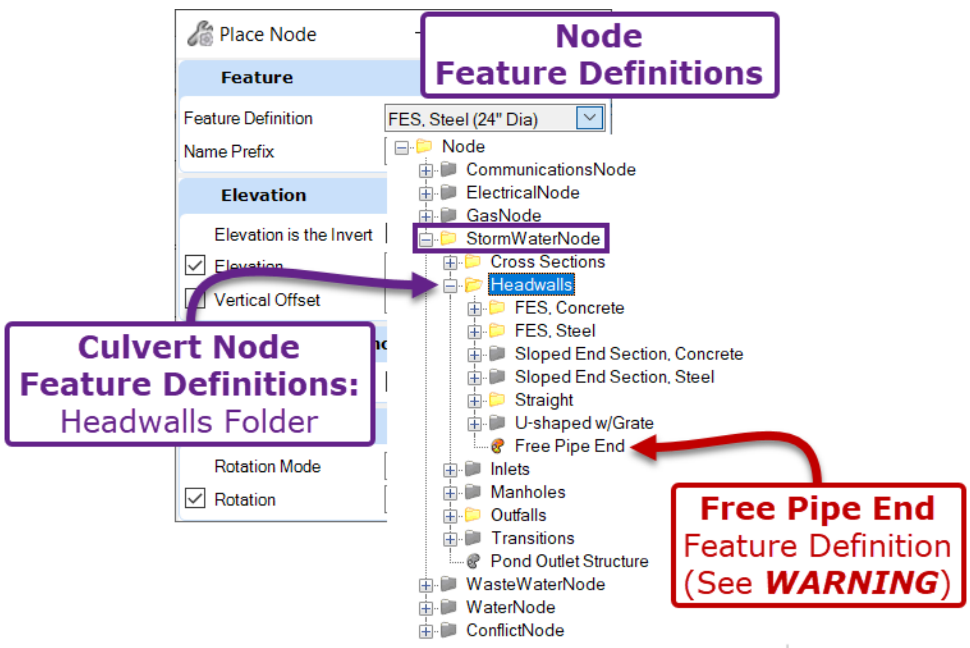


25B.1 Create the Inlet and Outlet Nodes

Nodes must be placed at the inlet and outlet location of the culvert.



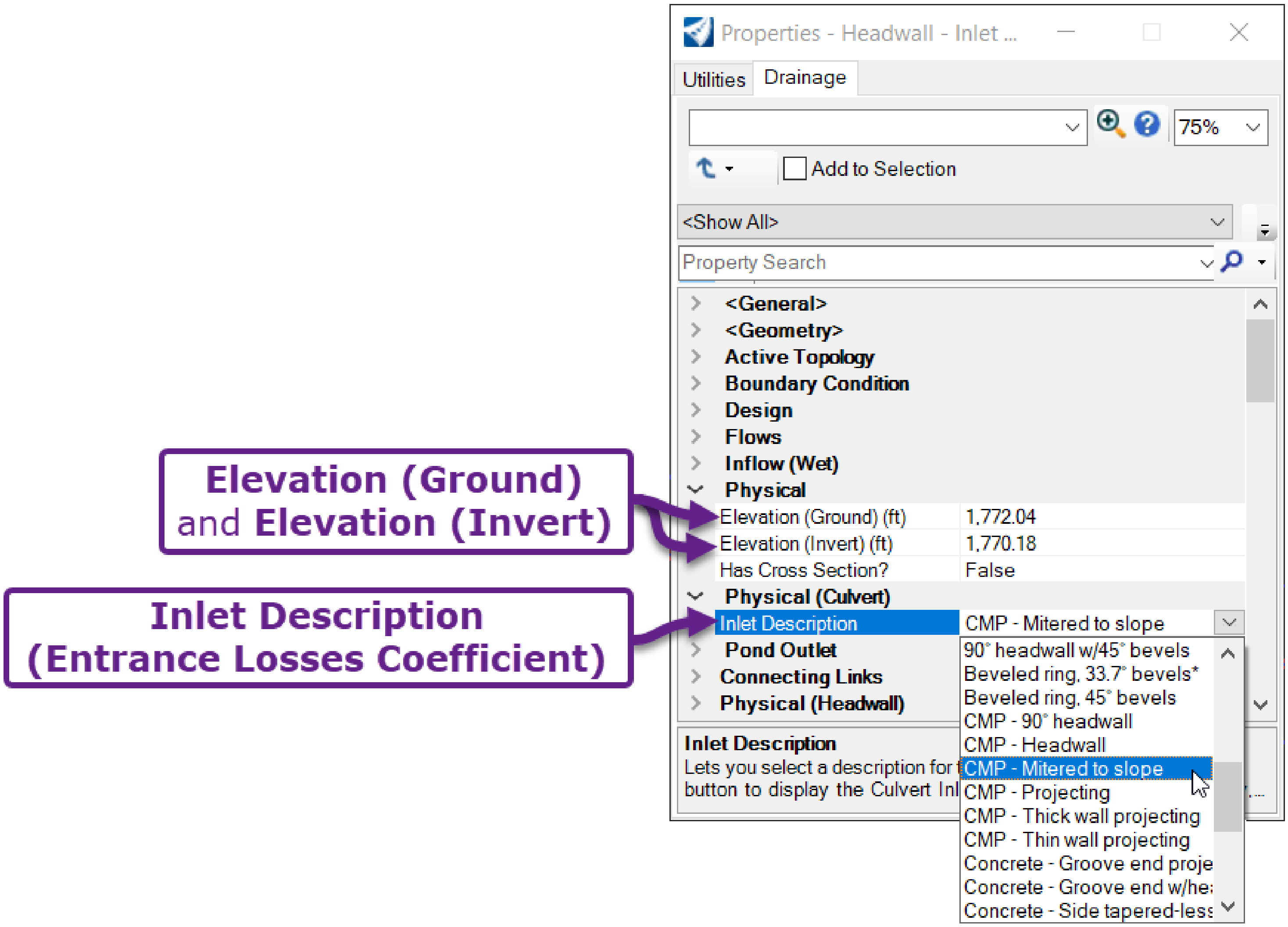
The Node **Feature Definition** represents the end treatment for the culvert. End treatments available in the Node Feature Definition library include headwalls, flared end sections (FES), and sloped end sections. Node Feature Definitions used in culvert modeling are found in the **StormWaterNode** → **Headwall** folder.



WARNING - Free Pipe End Feature Definition: Do NOT use the "Free Pipe End" Feature Definition. The "Free Pipe End" Feature Definition is intended for culverts that do NOT have a FES or headwall fixed to the inlet/outlet ends. However, this Feature Definition is broken and results in error when the analysis is run.

To model a free end culvert, use a Feature Definition from the "FES, Steel" folder. Select a "FES, Steel" Feature Definition that corresponds with the diameter of the culvert (i.e., "FES, Steel (24" Dia)"). After placement, enter the Node's Utility Properties and change the Entrance Loss Coefficient, K_e, to correspond with a free end culvert. The Entrance Loss Coefficient is set by the **Inlet Description** property, which is shown on the next page.

HYDRAULIC EFFECTS OF NODES DISCUSSION: Primarily, the Node Feature Definition sets the graphical appearance of the Node in the *2D Design Model* and *3D Design Model*. The graphical appearance of the Node has NO effect on the hydraulic analysis of the Node. The hydraulic analysis of a Node is dependent on two properties found in the Utilities Properties menu:



Inlet Description (Entrance Loss Coefficient, K_e): After placing a Node, the "Inlet Description" must be set in the Utility Properties. The "Inlet Description" determines the Entrance Loss Coefficient, K_e, for the culvert. By default, the "Inlet Description" is set to <None>, which results in an error when the analysis is ran. Select an "Inlet Description" that corresponds with the culvert end treatment.

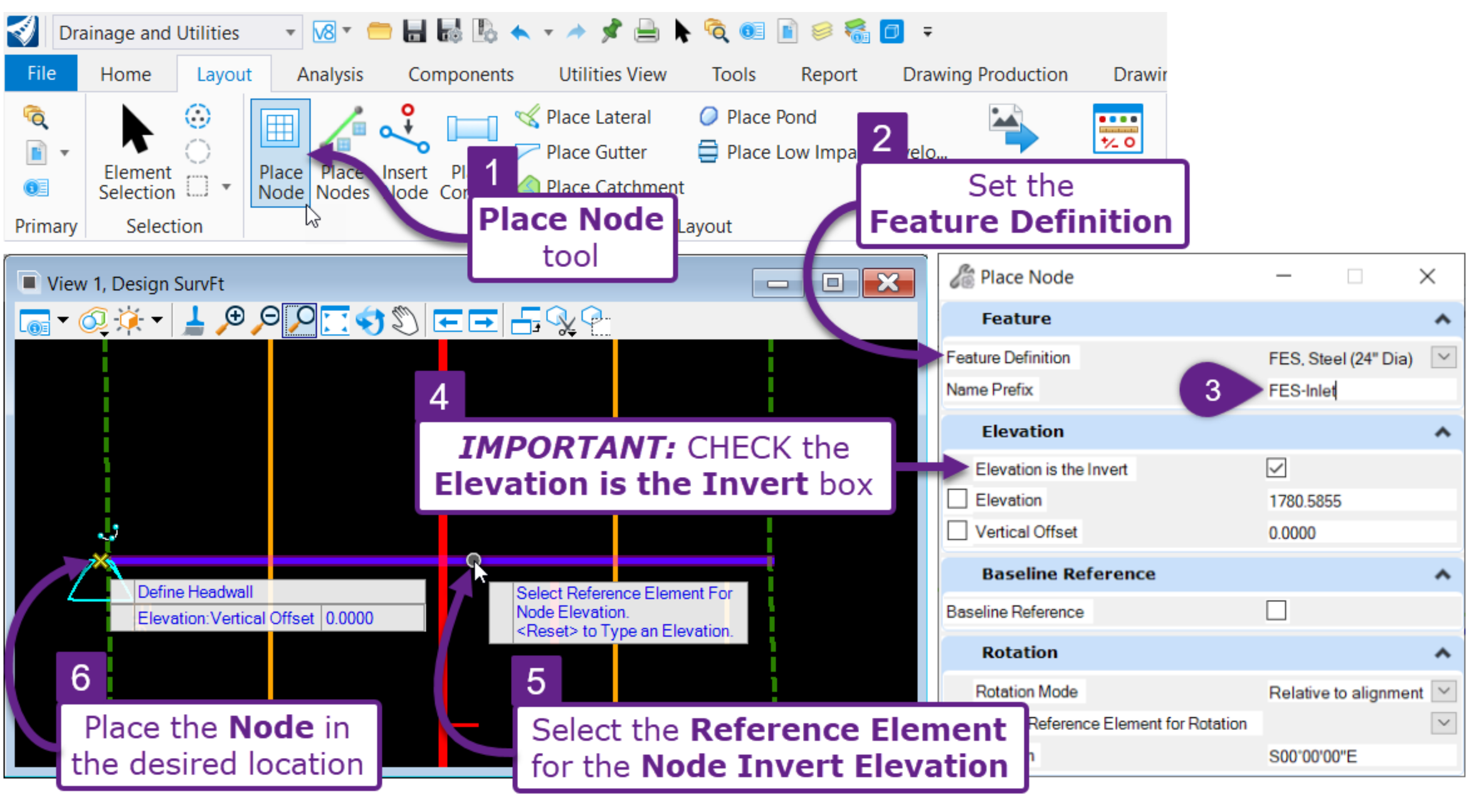
The procedure for setting the "Inlet Description" is shown in *25B.3 Edit Node and Conduit Hydraulic Settings in the Utility Properties.*

Hydraulic Opening Height: The "Elevation (Ground)" and "Elevation (Invert)" values determine the hydraulic height for the opening of the Node. The "Elevation (Invert)" value is determined in placement of the Node. The "Elevation (Ground)" setting corresponds with the pipe crown elevation and is automatically set based on the Node Feature Definition.

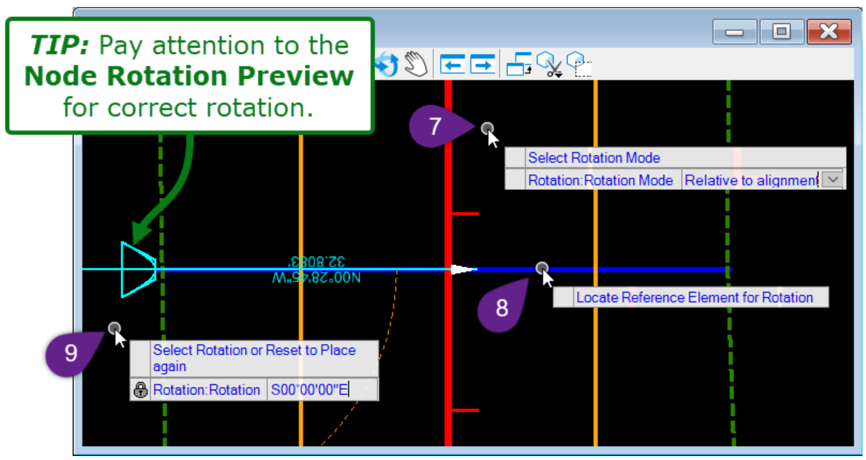
> ***WARNING - Free Pipe End Feature Definition:*** The "Free Pipe End" Feature Definition is broken because the "Elevation (Ground)" and "Elevation (Start)" are set to the same value, which results in a 0-foot Node opening and causes an error in analysis of the culvert. These values are linked, which makes it impossible to create a non-zero hydraulic opening height.

The procedure for placing a Node is as follows:

1	From the Ribbon, select the *Place Node* tool: [***Drainage and Utilities*** → ***Layout*** → ***Layout***].
2	In the *Dialogue Box*, set the **Feature Definition**. Select a Feature Definition from the **StormWaterNode** → **Headwall** drop-down. The Feature Definition must correlate with the culvert diameter. For example, use the "FLH - FES, Steel (24" Dia)" Feature Definition. to place a 24" diameter steel flared end section for use with a 24" diameter CMP culvert.
3	In the *Dialogue Box*, assign the Node a name in the **Name Prefix** box.
4	In the *Dialogue Box*, CHECK the **Elevation is Invert** box. ***WARNING:*** This box must be CHECKED when placing culvert Nodes. If UNCHECKED, then the top of the Node opening is placed at the elevation specified in the next step.
5	*Prompt: Select Reference Element for Node Elevation. <Reset> to Type an Elevation* – In this step, a Terrain Model or the Culvert Alignment can be selected to automatically set the Node elevation. In this example, the Culvert Alignment is selected to automatically pull the elevation from its ***Active* Profile**. ***NOTE:*** The Culvert Alignment and Profile were previously-created with **ORD Geometry Tools**. Alternatively, right-click (reset) to manually set the Node elevation in the next step.
6	*Prompt: Define Headwall* – Place the Node in the desired location. In this case the Node is placed at the end point of the Culvert Alignment (Reference Element).

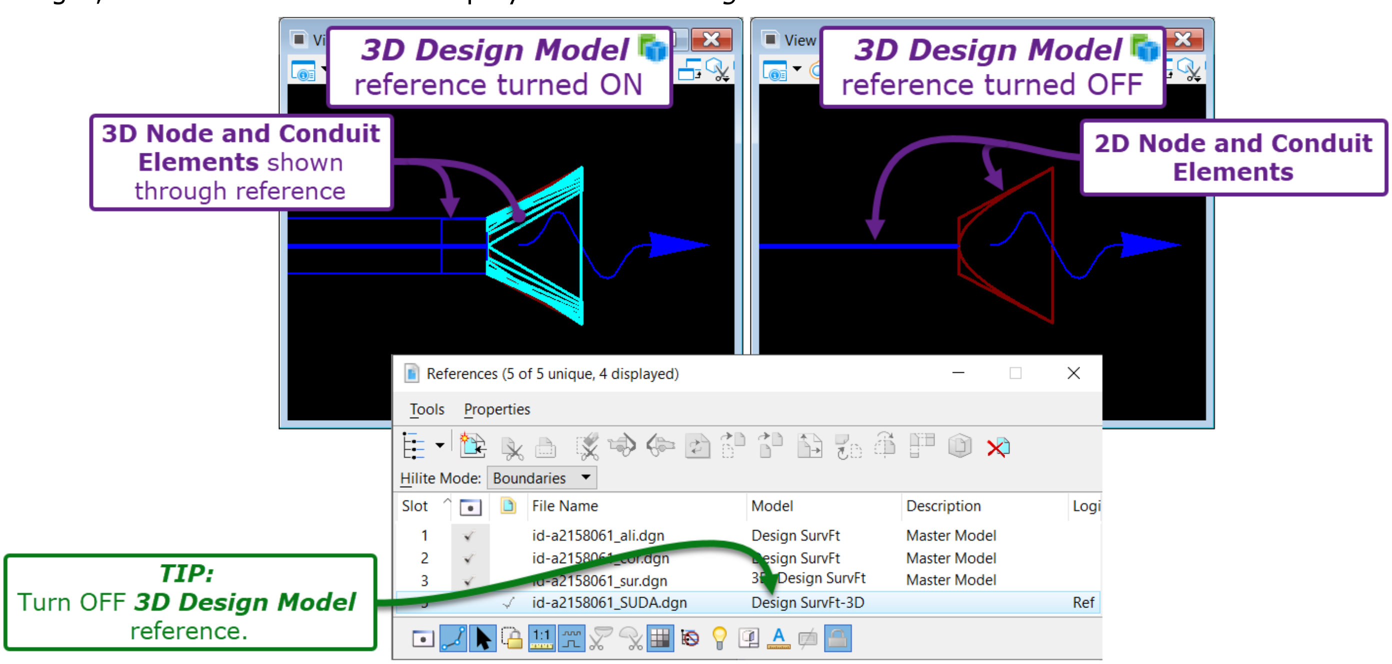


7	*Prompt: Select Rotation Mode* - If **Relative to Alignment** is selected, then a Reference Element is selected for rotation. If **Absolute** is selected, then rotation is relative to the true north direction. In this case, the **Relative to Alignment** option is selected, to rotate the Node inline with the Culvert Alignment.
8	*Prompt: Locate Reference Element for Rotation* – Select the Reference Element. In this case, the Culvert Alignment is selected.
9	*Prompt: Select Rotation or Reset to Place again* – Key-in the desired rotation angle. ***TIP:*** Key in either S00°00’00”E or N00°00’00”W to rotate the Node inline with the Culvert Alignment. If the required angle for the inlet is S00°00’00”E, then set the outlet to N00°00’00”W. After placement of the Node, the angle can be adjusted by selecting the Node element.



Repeat steps 1-9 to create a Node for the culvert outlet.

TIP: When a Node is created, graphical elements are placed in both the *2D Design Model* and *3D Design Model*. When working in the *2D Design Model* the graphics for the 3D element will overlap with the 2D element if the reference display of the *3D Design Model* is ON. In the References manager, turn OFF the reference display of the *3D Design Model*.



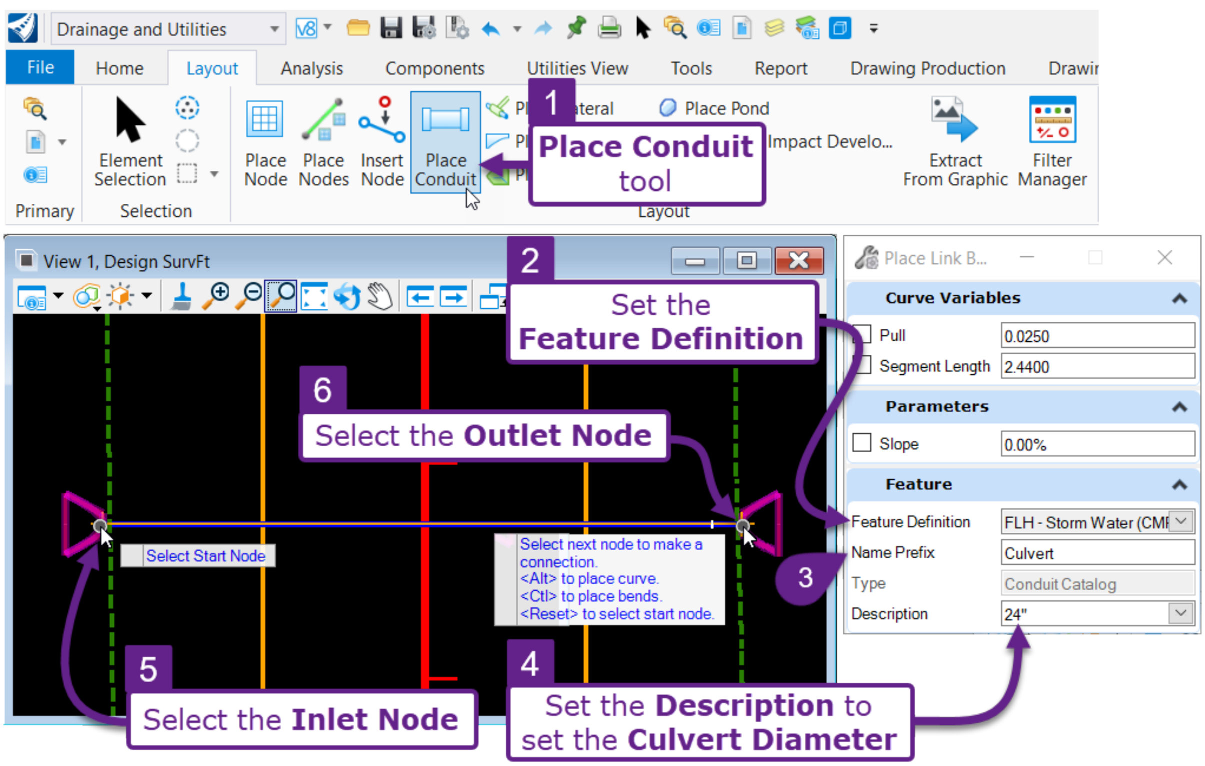
25B.2 Create the Conduit (Pipe)

The *Place Conduit* tool is used to create the Culvert pipe.

The Conduit **Feature Definition** represents the culvert material and pipe shape (i.e., circular, pipe-arch, elliptical. Culvert Feature Definitions are found in the **StormWater** folder.

The **Description** sets the diameter or dimensions of the pipe.

1	From the Ribbon, select the *Place Conduit* tool: [***Drainage and Utilities*** → ***Layout*** → ***Layout***].
2	In the *Dialogue Box*, set the **Feature Definition**. Select a Feature Definition from the **StormWater** drop-down.
3	In the *Dialogue Box*, use the **Description** drop-down to set the **Pipe Size**.
4	In the *Dialogue Box*, assign the Conduit a name in the **Name Prefix** box.
5	*Prompt: Select Start Node* – Select the **Inlet Node**. ***WARNING:*** To ensure the culvert is flowing in the correct direction, the inlet must be selected as the Start Node.
6	*Prompt: Select next node to make a connection* – Select the **Outlet Node**.



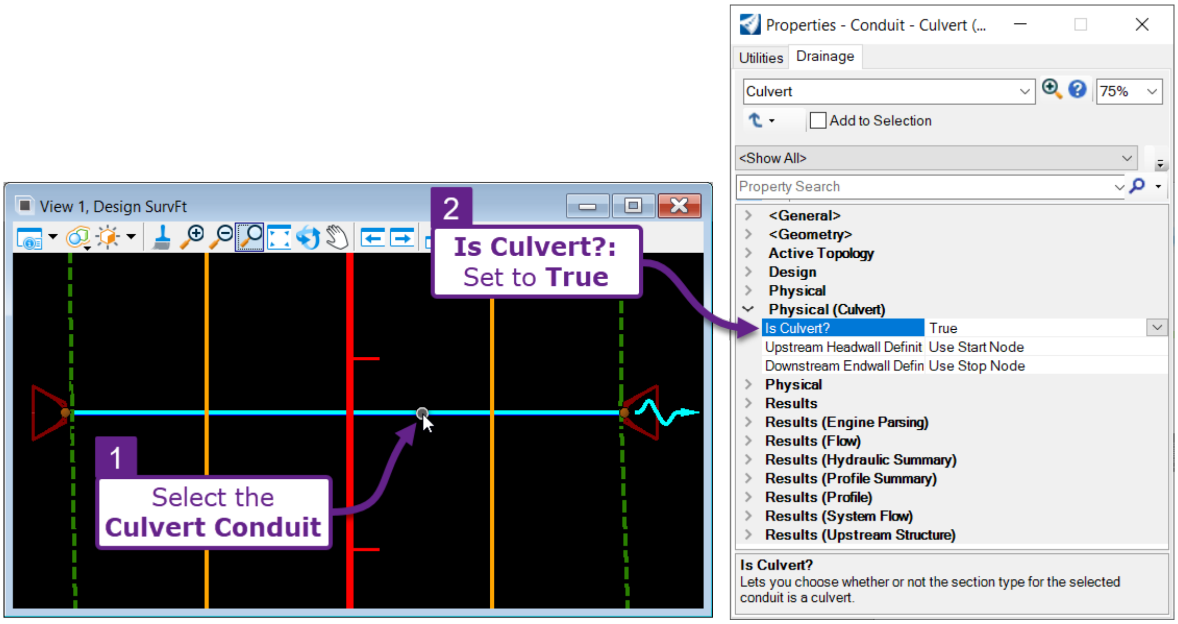
WARNING: In the *Dialogue Box*, do NOT CHECK the **Pull**, **Segment Length**, or **Slope** boxes. The Pull and Segment Length settings are used to add vertexes and curves in the Conduit.

The Slope setting is used to manually specify the slope for the Conduit. If the Slope box is UNCHECKED (preferred), then the Conduit slope is automatically set by the invert elevations of the Nodes.

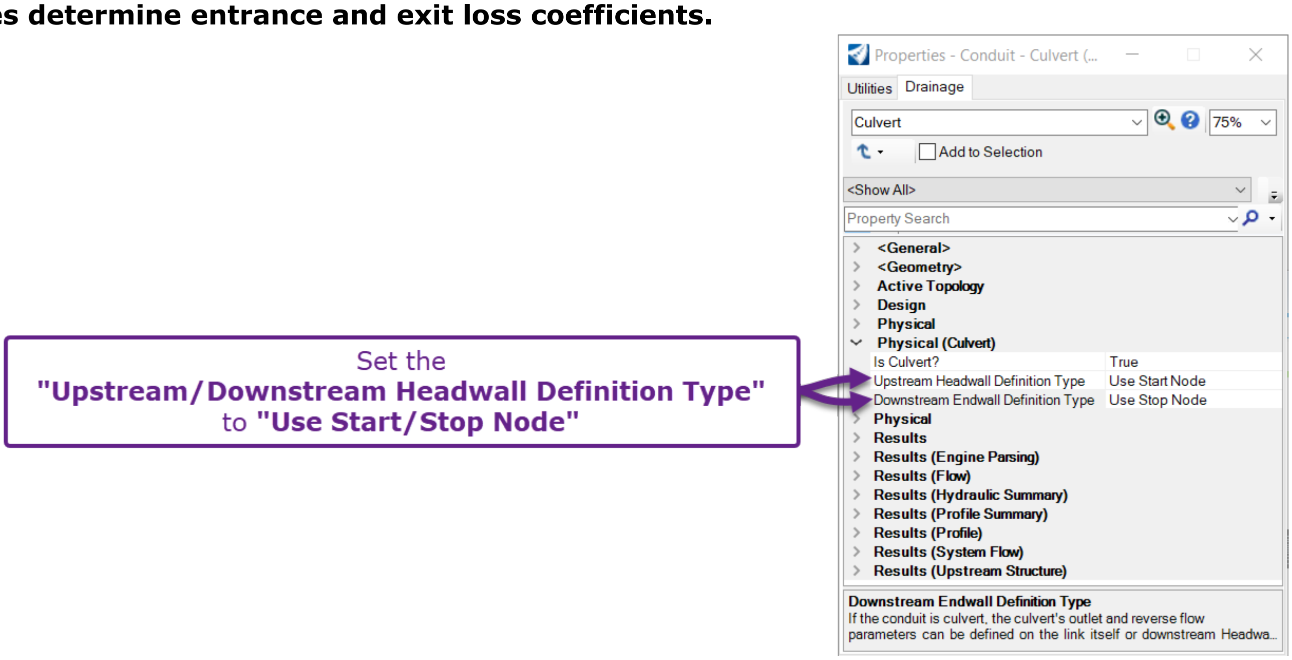
25B.3 Edit Node and Conduit Hydraulic Settings in the Utility Properties

Settings found in the Utility Properties menu control the hydraulic analysis. Opening the Utility Properties is shown in *25A.2 The Utilities Properties Menu*. Before a successful analysis can be run, the following Utility Properties settings must be edited for the Nodes and the Conduit:

Conduit Utility Property - Set the "Is Culvert" property to True: By default, the "Is Culvert?" property is set to False, which results in an incorrect analysis. This property must be set to True to correctly analyze the headwater at the inlet of the culvert. This property is located under the **Physical (Culvert)** drop-down.

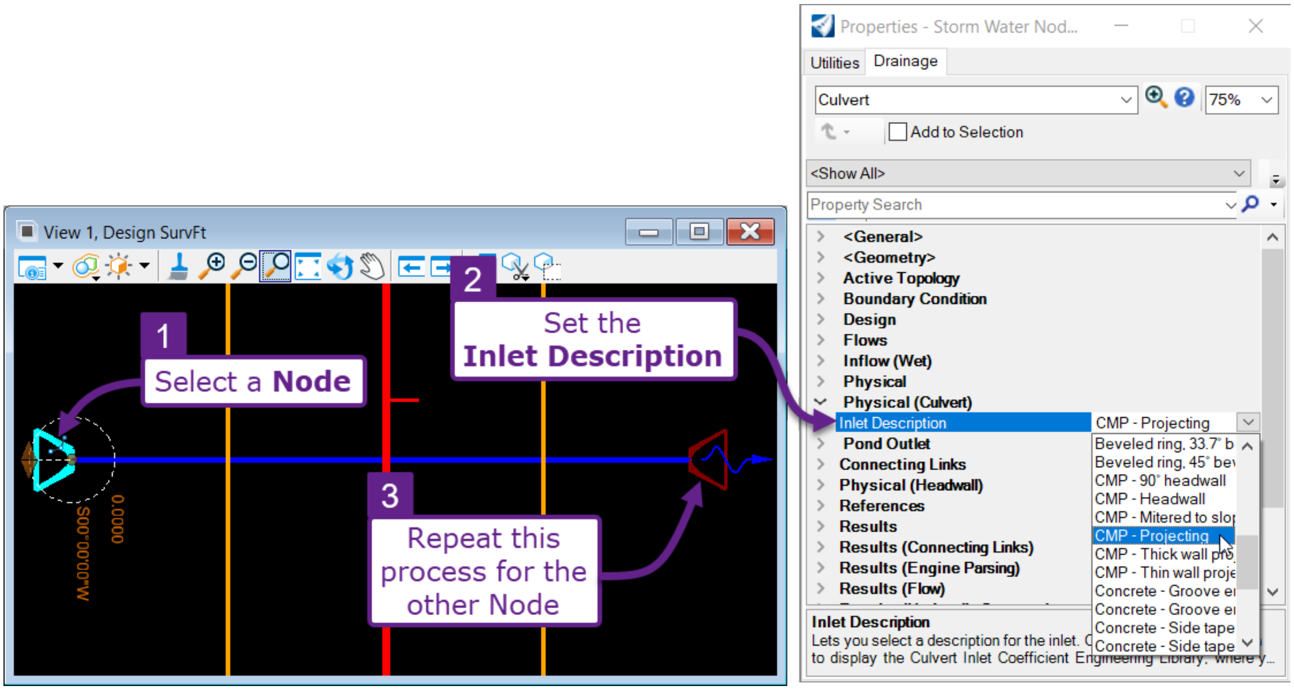


After the "Is Culvert?" property is set to True, options become available for the "Upstream/Downstream Headwall Definition". The "Headwall Definition" options should be set to "Use Start Node" and "Use Stop Node". **When set to "Use Start/Stop Node" the hydraulic properties for the inlet and outlet Nodes determine entrance and exit loss coefficients.**



NOTE: If this option is set to "Use Conduit" (default option), then a "Culvert Headwall" must be specified to set the entrance and exit loss coefficients for the culvert. The hydraulic properties of the Nodes are entirely ignored when set to "Use Conduit".

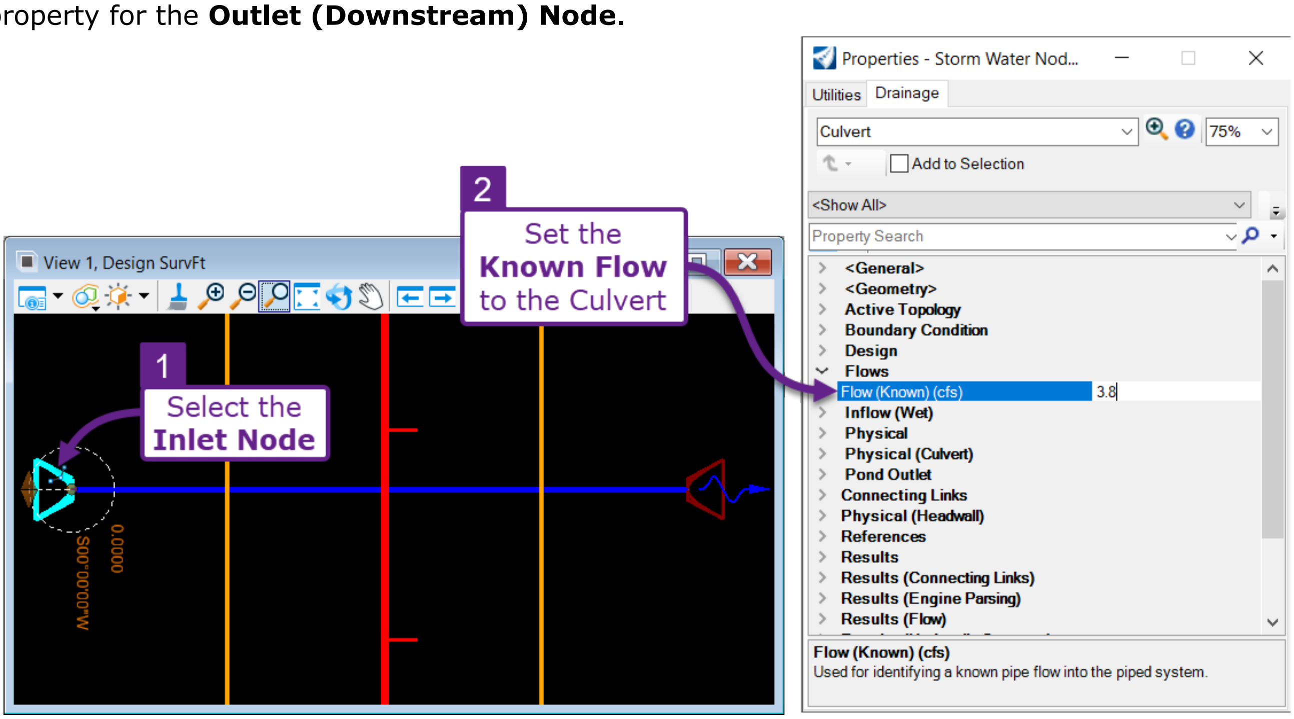
Node Utility Property - Set the "Inlet Description" property: Both the inlet and outlet Nodes must have an "Inlet Description" set. The "Inlet Description" determines the Entrance and Exit Loss Coefficients for the culvert. Select an option that corresponds with the end treatment for the culvert (i.e., headwall configuration, CMP - Projecting, CMP - Mitered to Slope). If the "Inlet Description" is NOT set, then the analysis results in an error. This property is located under the **Physical (Culvert)** drop-down.



25B.3.a Set a Known Flow to the Inlet Node

If the flowrate going into a culvert or inlet is known, then it can be set in the **Node** properties. This property must be set if Catchment elements are NOT used. This property is located under the **Flows** drop-down.

NOTE: The flowrate can only be set in the **Inlet (Upstream) Node** properties. It is NOT possible to set this property for the **Outlet (Downstream) Node**.



25B.4 Create the Catchments (Drainage Areas)

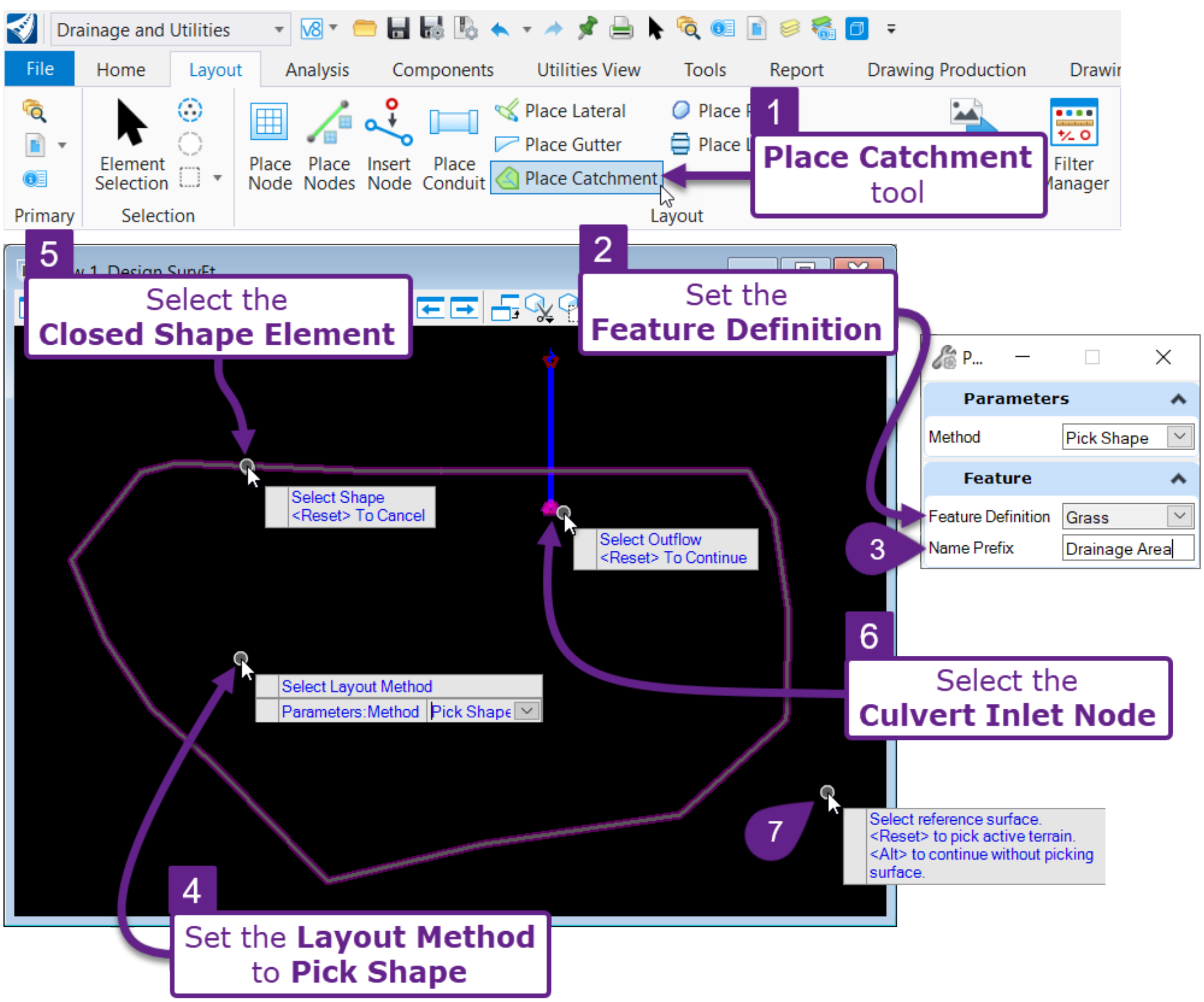
The *Place Catchment* tool is used to convert a closed shape element into a Catchment element. A Catchment represents the drainage area. Using the Rational Method*, a Catchment produces a peak flow rate. A Catchment must be assigned a downstream Inlet Node. All flow generated from the Catchment is routed into the assigned Inlet Node.

NOTE*: Without and add-on subscription to Civil Storm, only the Rational Method is available for the hydrologic analysis of Catchments.

When using the Rational Method, the hydrologic analysis of the Catchment is governed by three properties:

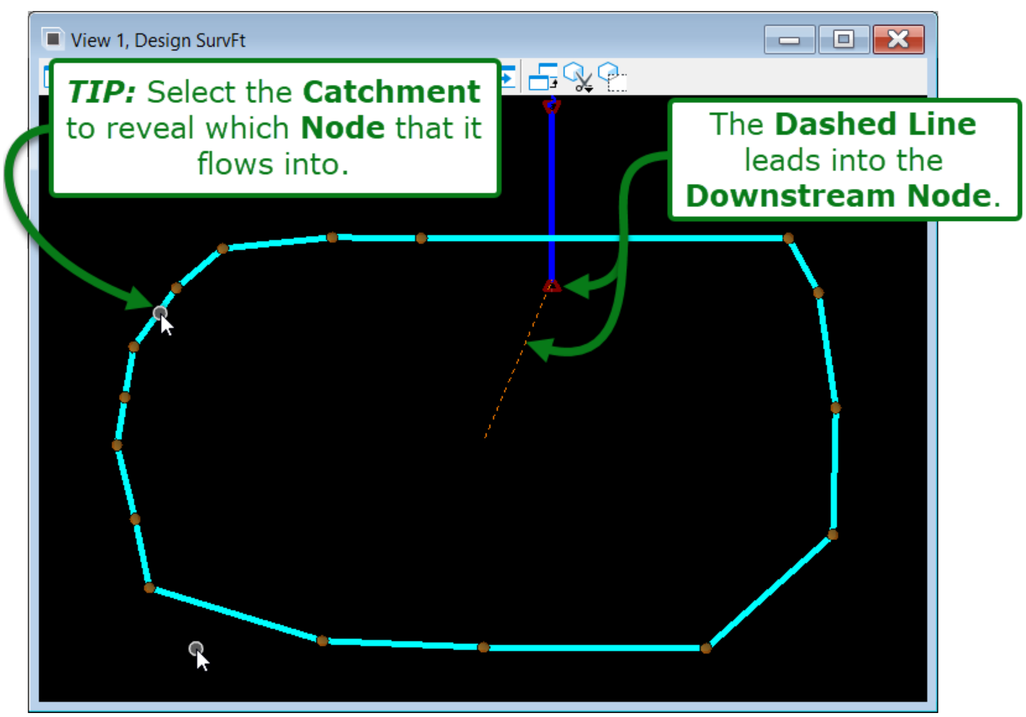
- **Catchment Area:** Determined by the geometry of the closed shape element used to create the Catchment.
- **Runoff Coefficient:** Initially, the runoff coefficient is set by the Feature Definition selected in creation of the Catchment. However, this value can be edited in the Utility Properties menu.
- **Time of Concentration:** The time of concentration must be calculated manually and is set in the Utility Properties menu.

After creating a Catchment element, Storm Data and Scenarios must be programed to set an IDF curve and select a return event. The Time of Concentration is referenced by the IDF curve to calculate a rainfall intensity for the Catchment area. Programming Storm Data is shown in *25C – Create Storm Data and Scenarios for Catchments*.



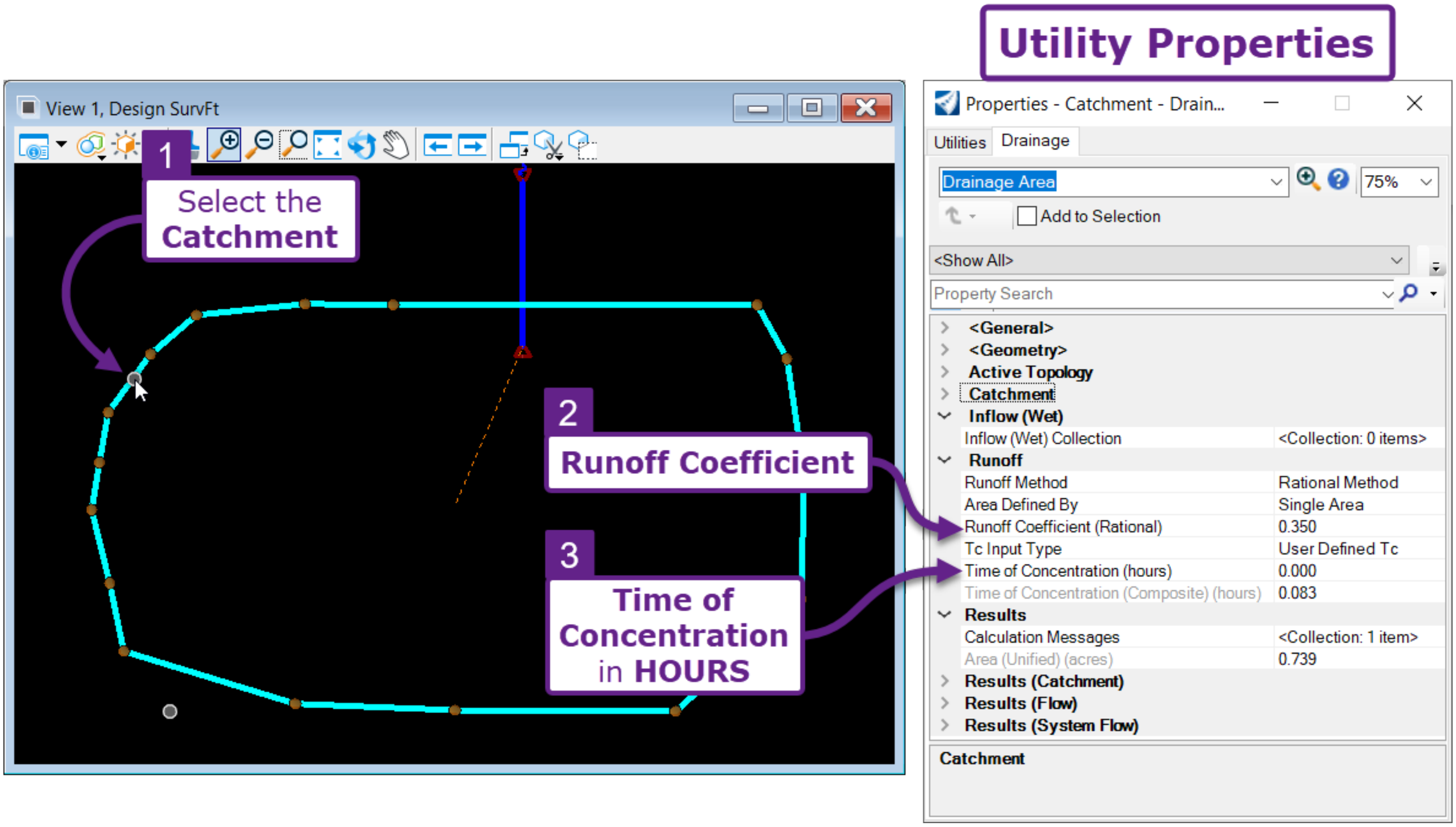
1	From the Ribbon, select the *Place Catchment* tool: [***Drainage and Utilities*** → ***Layout*** → ***Layout***].
2	In the *Dialogue Box*, set the **Feature Definition** that corresponds with the surface condition of the drainage area. The Feature Definition initially sets the runoff coefficient for the Catchment. However, in the Utility Properties, the runoff coefficient can be changed to any value.
3	In the *Dialogue Box*, assign the Catchment a name in the **Name Prefix** box.
4	*Prompt: Select Layout Method* – Select the **Pick Shape** method to select a closed shape element that represents the drainage area. ***WARNING:*** The closed shape element must be directly placed in the current ORD File. The closed shape element CANNOT be found in a reference ORD File. If the closed shape element is in a reference ORD File, use the *Copy* tool to transfer the closed shape element into the current ORD File.
5	*Prompt: Select Outflow* – The "outflow" represents the Node that the Catchment area will flow into. In this case, the inlet Node of the culvert is selected.
6	*Prompt: Select reference surface. <Reset> to pick active terrain. <Alt> to continue without picking surface*: In this step, a Terrain Model is selected to drape the Catchment onto. Draping the Catchment onto a Terrain Model has NO effect on the hydrologic analysis of the Catchment. The sole purpose of the draping effect is to place the Catchment at the appropriate elevation when viewed in the *3D Design Model*. Right-Click (reset) to use the active Terrain Model for draping purposes. Alternatively, select the desired Terrain Model or press the ALT key to forgo the 3D draping effect and place the catchment at an elevation of 0 feet.

TIP: After creating the Catchment, select it to show which Node the Catchment flows into.

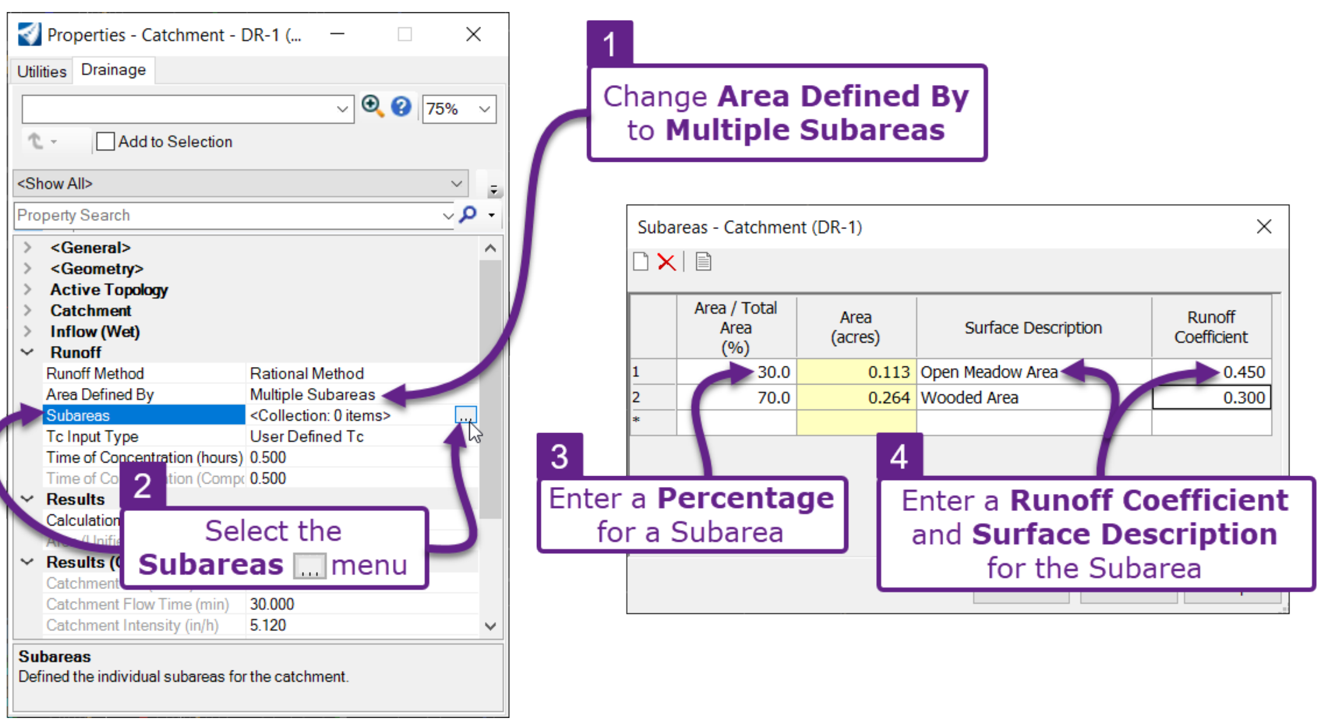


After creating the Catchment, the Time of Concentration must be set and the Runoff Coefficient should be verified or edited. The Time of Concentration and Runoff Coefficient are located in the Utility Properties for the Catchment, under the **Runoff** drop-down.

WARNING: The units for Time of Concentration is hours. If the Time of Concentration was manually calculated in minutes, then it must be converted to hours.

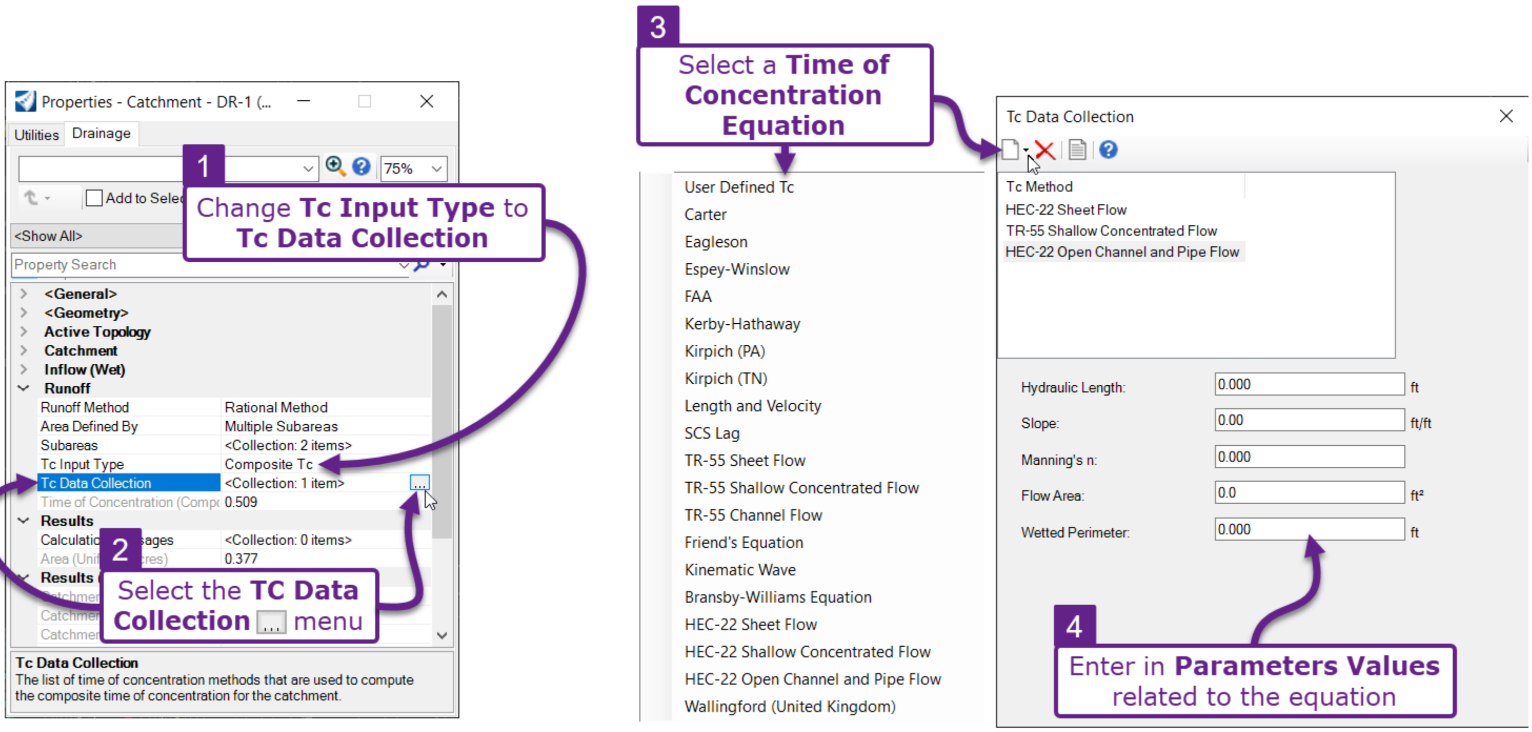


Runoff Coefficient Tips: The Catchment area can be assigned a composite Runoff Coefficient by changing the **Area Defined By** setting to "Multiple Subareas".



Time of Concentration Tips: By changing the **Tc Input Type** to "Composite Tc", there are a plethora of equations for calculating time of concentration. Hydraulic parameters, such as hydraulic length and slope, must be known to utilize the time of concentration equations.

In the **TC Data Collection** menu, expand the drop down to view equations available for calculating time of concentration. Multiple equations can be used to form composite time of concentration that includes both sheet flow and concentrated flow. For example, use the HEC-22 Sheet Flow and the HEC-22 Open Channel and Pipe Flow equations to calculate of time of concentration value that reflects both sheet flow and channel flow.



25C – CREATE STORM DATA AND SCENARIOS FOR CATCHMENTS

Storm Data is needed to calculate the rainfall runoff and peak flowrates for the Catchment areas. Storm Data is typically in the form of an Intensity Duration Frequency curve (IDF). The IDF curve must be obtained by the User and should correspond with the project location. IDF Curves must be compiled into CSV format for import into the ORD software.

NOTE: It is NOT necessary to import a IDF Curve and setup Storm Data if the peak flowrate for a Node is already known. If the peak flowrate for a drainage area was calculated in a separate software or program (i.e., StreamStats or HEC-HMS), then it can be directly assigned to a Node, which is shown in *25B.3.a Set a Known Flow to the Inlet Node*. If the flow to a Node is known, then proceed to *25D – Run an Analysis Scenario with the Compute Center tool*.

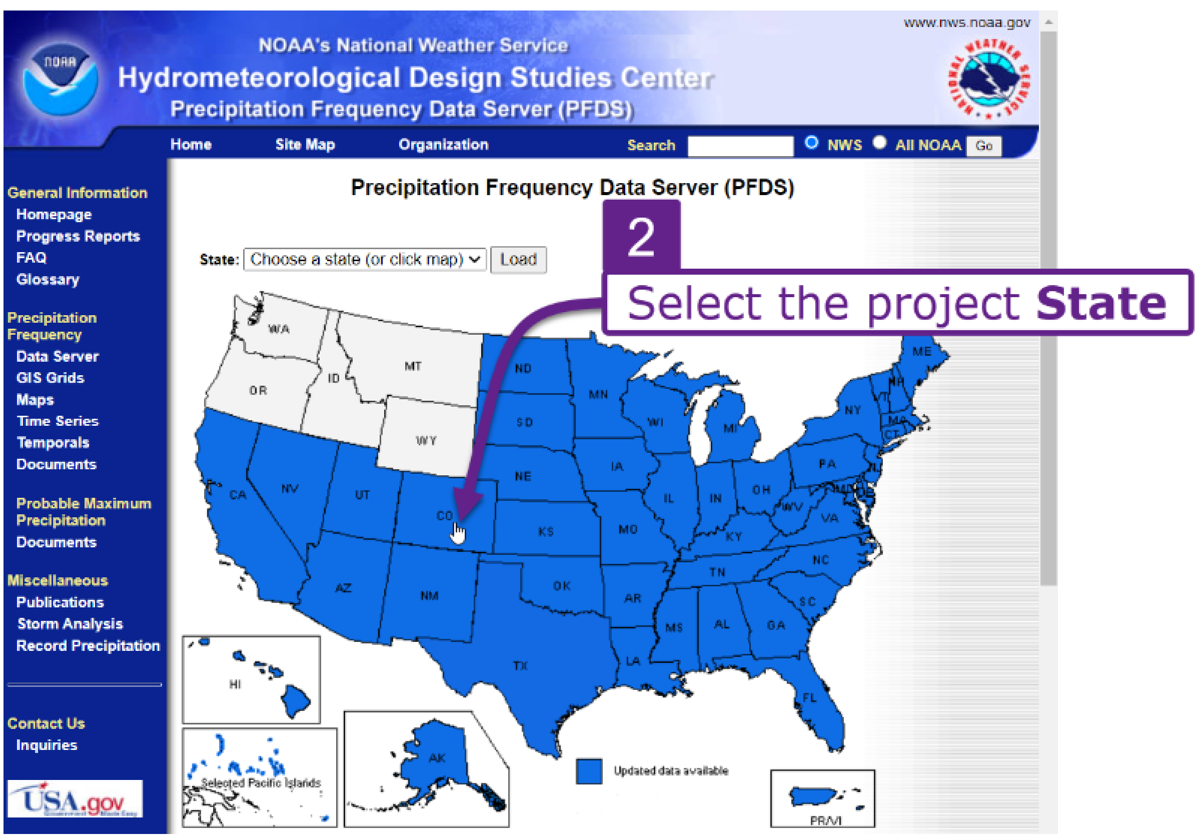
25C.1 Obtain and Set Up an IDF Curve File (CSV)

IDF Curves can be obtained in CSV file type format from the National Oceanic and Atmospheric Administration (NOAA). The Precipitation Frequency Data Server (PFDS) provides IDF Curves for most areas of the United States:

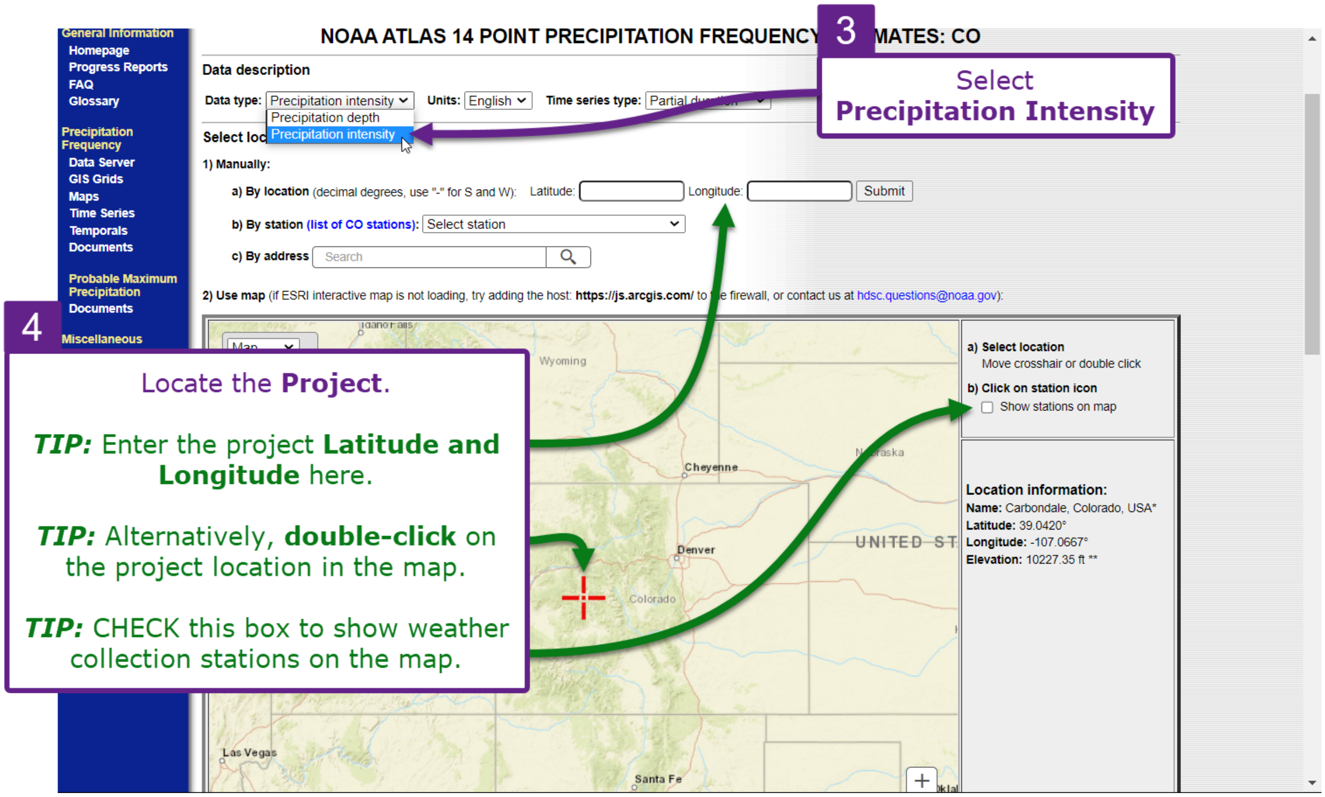
NOAA Precipitation Frequency Data Server: https://hdsc.nws.noaa.gov/hdsc/pfds/

NOTE: As shown below, NOAA data is NOT available in Washington, Oregon, Idaho, Montana, and Oregon. For projects in these states, refer to the FLH Drainage Technical Guidance Manual or local/state agency resources to obtain an IDF curve.

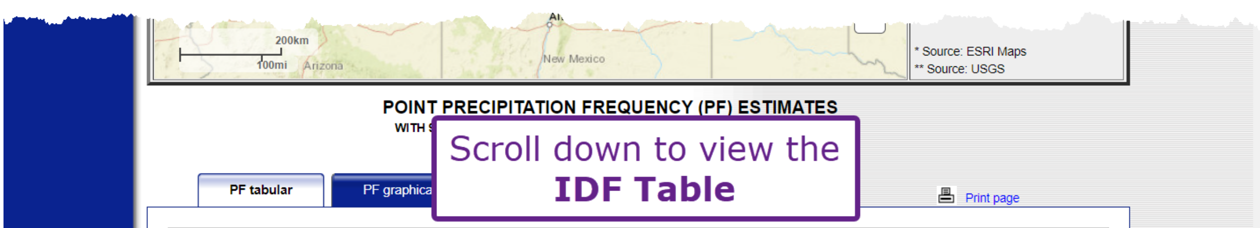
1	Access the NOAA Precipitation Frequency Data Server using the hyperlink shown above.
2	Select the project State.



3	Set the **Data Type** to **Precipitation Intensity**.
4	Select the project location.



5	At the bottom of the webpage, push the **Submit** button to generate a CSV file for the IDF Table.



PDS-based precipitation frequency estimates with 90% confidence intervals (in inches/hour)[1]

Duration	Average recurrence interval (years)									
	1	2	5	10	25	50	100	200	500	1000
5-min	1.81 (1.44-2.29)	2.56 (2.03-3.23)	3.71 (2.94-4.70)	4.61 (3.62-5.88)	5.78 (4.32-7.62)	6.62 (4.85-8.93)	7.42 (5.23-10.3)	8.17 (5.50-11.8)	9.07 (5.87-13.7)	9.70 (6.13-15.0)
10-min	1.33 (1.06-1.67)	1.87 (1.49-2.36)	2.71 (2.15-3.44)	3.38 (2.66-4.31)	4.24 (3.17-5.57)	4.85 (3.55-6.53)	5.43 (3.83-7.57)	5.98 (4.03-8.65)	6.64 (4.29-10.0)	7.10 (4.49-11.0)
15-min	1.08 (0.860-1.36)	1.52 (1.21-1.92)	2.21 (1.75-2.80)	2.74 (2.16-3.50)	3.44 (2.57-4.53)	3.94 (2.88-5.31)	4.42 (3.11-6.16)	4.86 (3.28-7.04)	5.40 (3.49-8.13)	5.77 (3.65-8.96)
30-min	0.694 (0.552-0.876)	0.992 (0.788-1.25)	1.45 (1.15-1.84)	1.80 (1.42-2.29)	2.24 (1.67-2.93)	2.54 (1.86-3.41)	2.82 (1.98-3.91)	3.07 (2.06-4.42)	3.36 (2.16-5.03)	3.54 (2.24-5.49)
60-min	0.454 (0.361-0.573)	0.614 (0.488-0.776)	0.860 (0.680-1.09)	1.05 (0.824-1.34)	1.28 (0.955-1.68)	1.44 (1.05-1.94)	1.59 (1.12-2.21)	1.72 (1.16-2.48)	1.87 (1.21-2.80)	1.96 (1.24-3.05)
2-hr	0.280 (0.225-0.351)	0.366 (0.294-0.459)	0.498 (0.397-0.625)	0.598 (0.474-0.756)	0.722 (0.543-0.936)	0.808 (0.596-1.07)	0.885 (0.629-1.22)	0.954 (0.648-1.36)	1.03 (0.672-1.53)	1.08 (0.690-1.66)
3-hr	0.220 (0.177-0.274)	0.273 (0.220-0.341)	0.355 (0.284-0.444)	0.417 (0.333-0.525)	0.496 (0.376-0.641)	0.551 (0.410-0.730)	0.601 (0.431-0.823)	0.647 (0.443-0.920)	0.700 (0.460-1.03)	0.734 (0.472-1.12)
6-hr	0.151 (0.123-0.187)	0.170 (0.138-0.211)	0.202 (0.163-0.250)	0.229 (0.184-0.285)	0.267 (0.207-0.347)	0.296 (0.224-0.394)	0.327 (0.238-0.449)	0.359 (0.250-0.511)	0.401 (0.269-0.594)	0.435 (0.282-0.657)
12-hr	0.104 (0.086-0.128)	0.109 (0.089-0.134)	0.120 (0.098-0.148)	0.133 (0.108-0.164)	0.155 (0.124-0.205)	0.176 (0.136-0.236)	0.200 (0.149-0.278)	0.228 (0.163-0.328)	0.270 (0.185-0.402)	0.307 (0.201-0.458)
24-hr	0.069 (0.057-0.084)	0.073 (0.061-0.089)	0.083 (0.068-0.101)	0.093 (0.076-0.115)	0.111 (0.090-0.146)	0.128 (0.100-0.170)	0.1[illegible] (0.110-[illegible])	0.169 ([illegible].121-0.240)	0.201 (0.139-0.296)	0.229 (0.152-0.338)

[1] Precipitation frequency (PF) estimates in this table are based on frequency analysis of partial duration series (PDS).

Numbers in parenthesis are PF estimates at lower and upper bounds of the 90% confidence interval. The probability that precipit[illegible] recurrence interval) will be greater than the upper bound (or less than the lower bound) is 5%. Esti[illegible] at upper bounds are no[illegible] estimates and may be higher than currently valid PMP values.

Please refer to NOAA Atlas 14 document for more information.

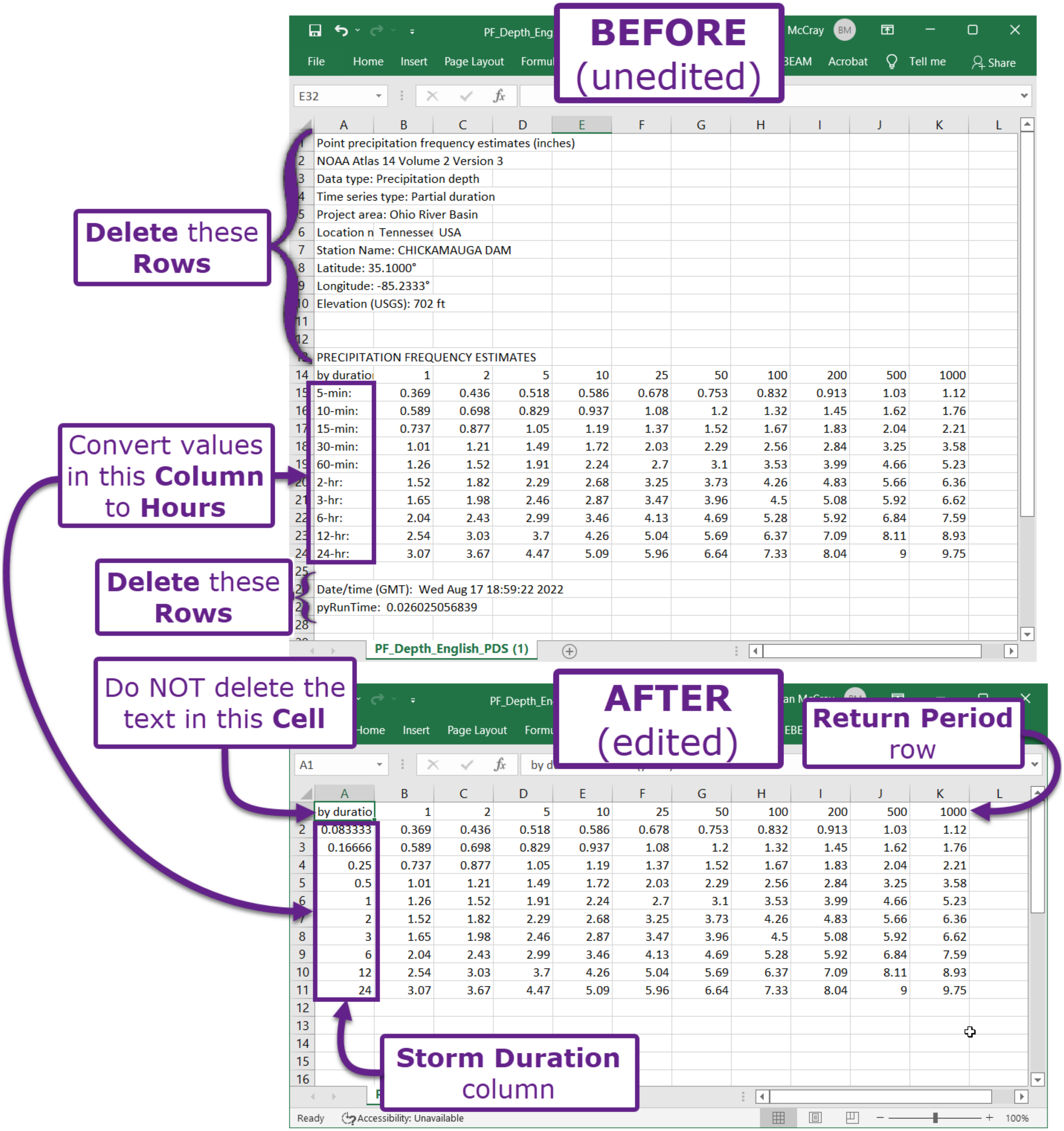
Estimates from the table in CSV format: Precipitation frequency estimates Submit

5

Push the **Submit** button to generate the **IDF CSV File**.

Main Link Categories:
Home | OWP

The CSV File must be edited and appropriately formatted in Microsoft Excel before importing to the ORD software. The **After (edited)** spread sheets shows the correct formatting for import into ORD.



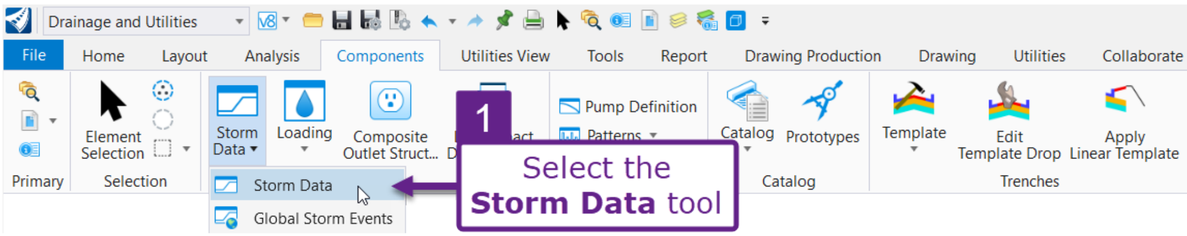
TIP: If compiling the IDF Curve data from a new Excel spread sheet, use the **Save-As** command to save the spread sheet in .CSV file format. Conventional Microsoft Excel files (.xlsx) CANNOT be imported into OpenRoads Designer.

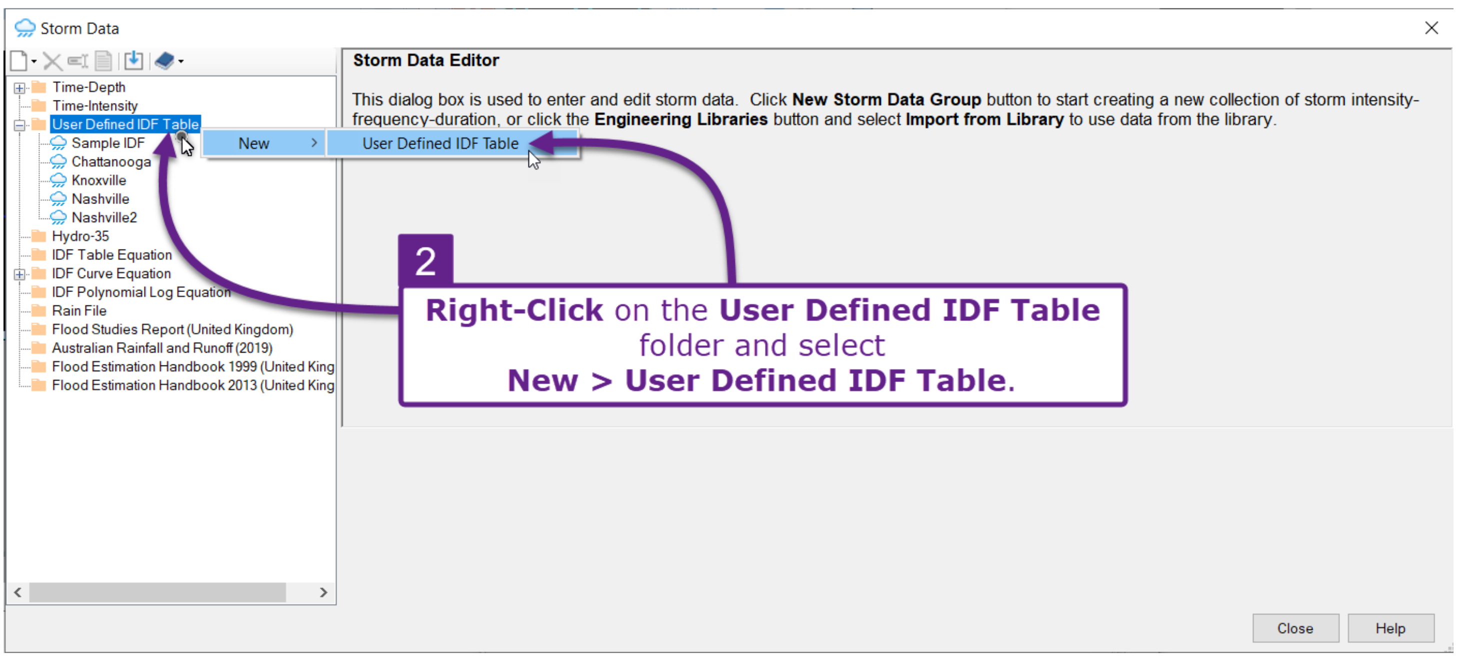
25C.2 Import the IDF Curve File (CSV) into the Storm Data Editor

The IDF Curve File (CSV) is imported into the OpenRoads software with the *Storm Data* tool.

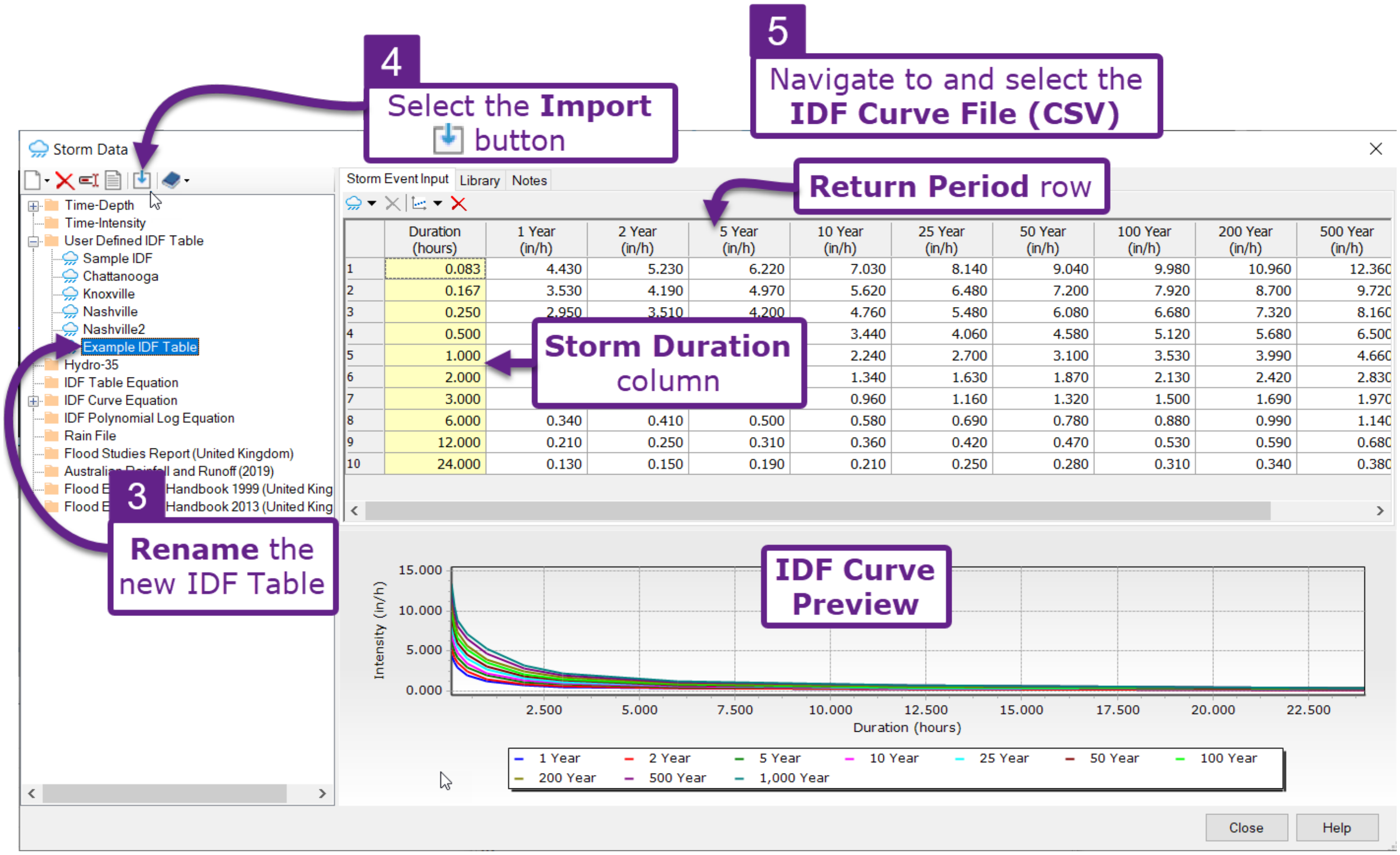
WARNING: Close the Microsoft Excel CSV file before importing into ORD. An error message is shown if the IDF Curve File is currently opened in Microsoft Excel.

1	From the Ribbon, select the *Storm Data* tool: [***Drainage and Utilities*** → ***Components*** → ***Common***].
2	On the left-side of the menu, right-click on the **User Defined IDF Table** folder and select: **New → User Defined IDF Table**.





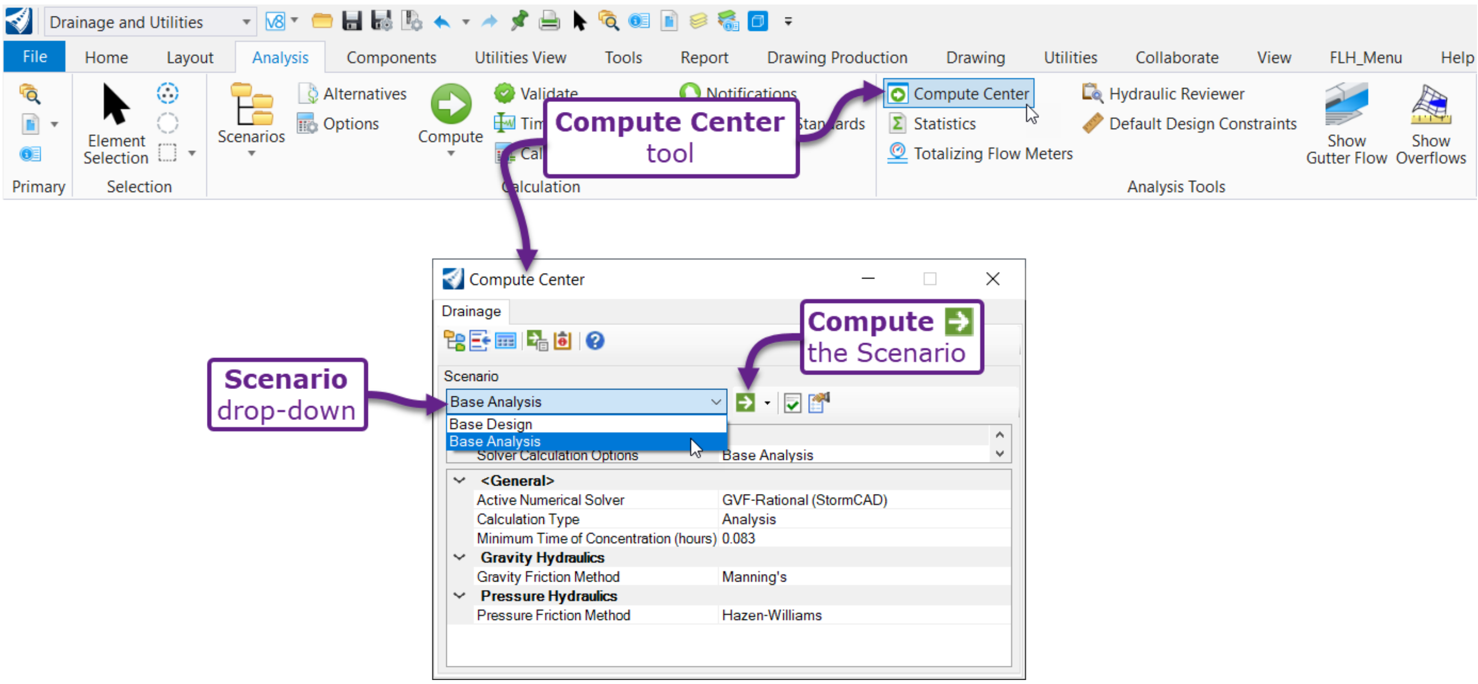
3	On the left-side of the menu, appropriately rename the IDF Table.
4	Select the **Import** button
5	Navigate to the file location and select the **IDF Curve File (CSV)**.



25C.3 Setup Scenarios

A Scenario is a collection of settings used in the analysis of the drainage network. The IDF Curve must be assigned to the "Base" Scenario. A *Child Scenario* must be created for each storm return event to be run.

Scenarios are selected in the *Computation Center*, prior to running the analysis. For more information on the *Computation Center*, see *25D - Run an Analysis Scenario with the Compute Center tool*.



There are two types of Scenarios: **Design** and **Analysis**.

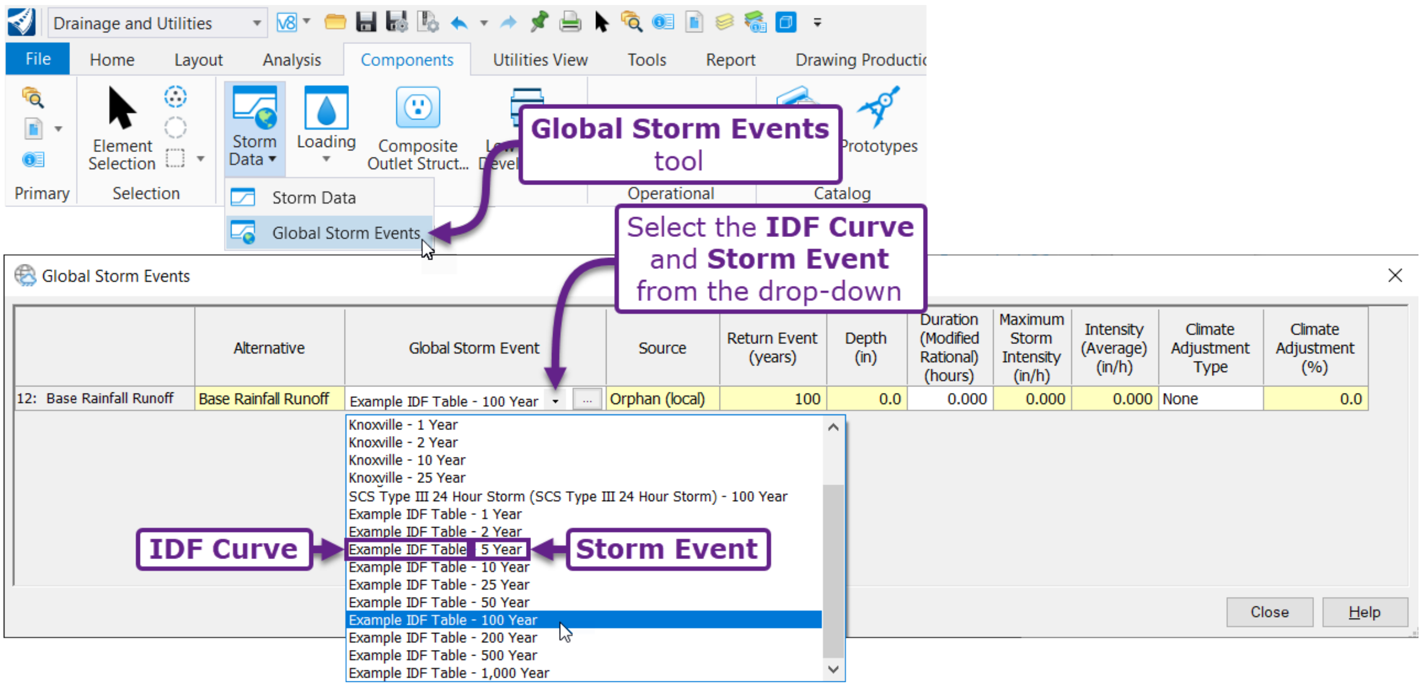
Design: If a Design Scenario is selected, then Conduits and Nodes are automatically resized or repositioned to optimally accommodate peak flowrates. Optionally, a Design Scenario can readjust Conduit inverts to achieve a minimum velocity or minimum cover over the pipe.

> ***WARNING:*** A Design Scenario should NOT be used for culvert analysis. Design Scenarios can be used for stormwater networks, but the resulting re-designed elements should be scrutinized.

Analysis: When an Analysis Scenario is selected, Conduits and Node elements will NOT automatically resize. The current configuration of elements will be analyzed and surcharge will be shown if present.

The appropriate IDF Curve and storm event must be assigned to the "Base" Scenario. This is accomplished with the *Global Storm Events* tool. This tool is found in the Ribbon in the following location:

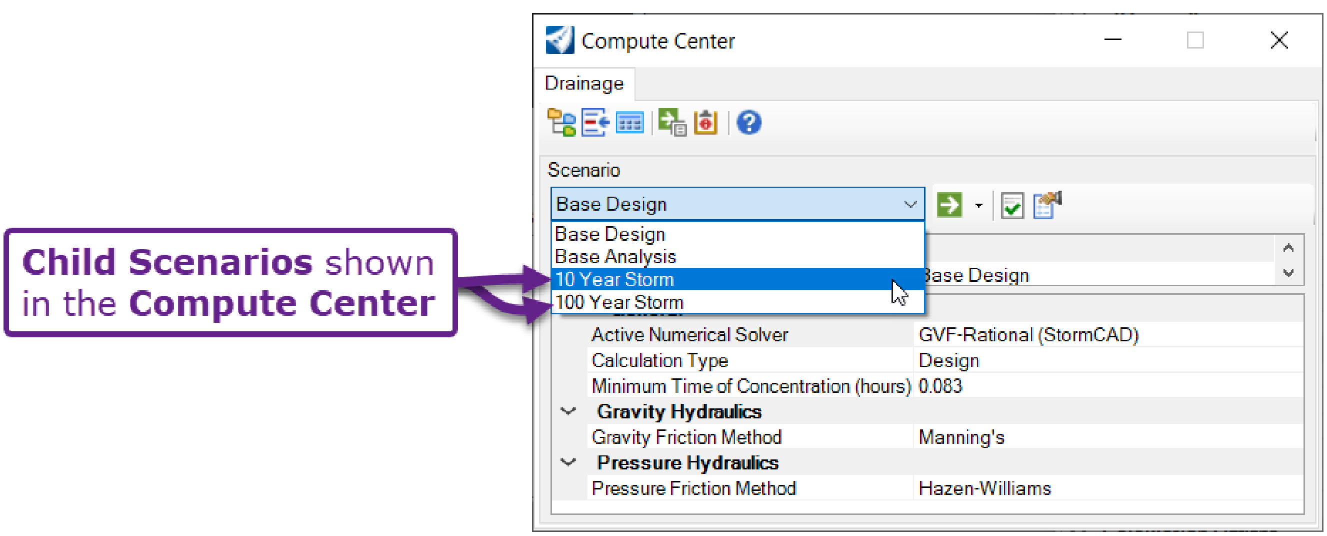
Drainage and Utilities workflow → **Components** tab → **Common** group



Both the "Base Design" and "Base Analysis" scenarios will use the selected *Global Storm Event* (IDF Curve and Storm Event) when the analysis is executed in the *Compute Center*.

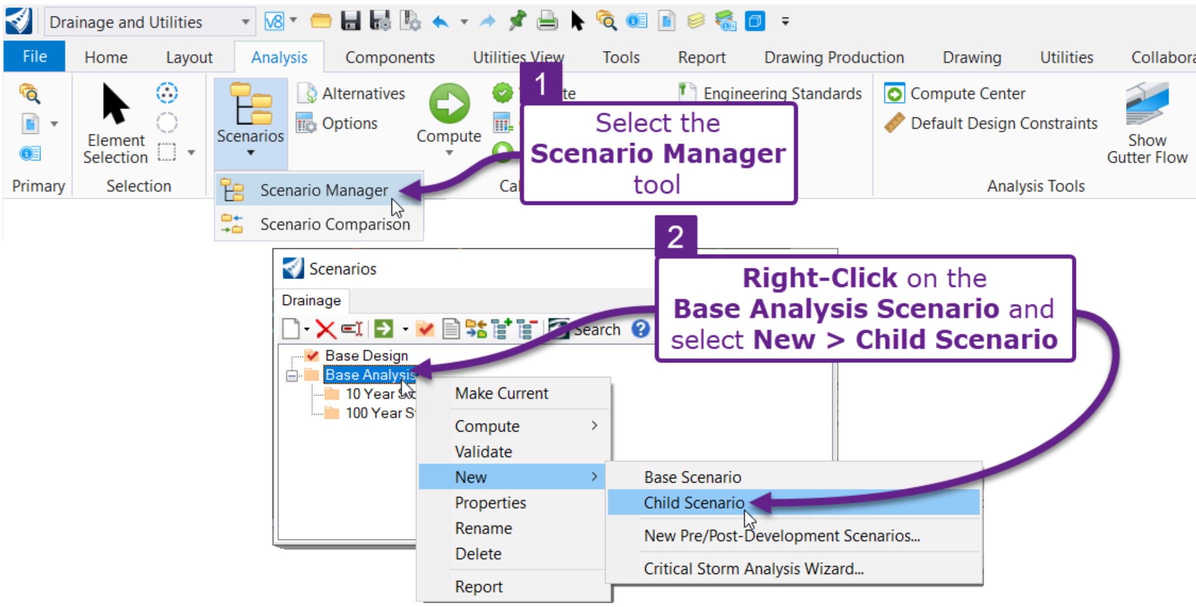
25C.4 Setup a Child Scenario for each Storm Event

To analyze multiple storm return events, **Child Scenarios** can be created and then selected in the *Compute Center*. Each Child Scenario correspond with a different for each return event. For example, a Child Scenario can be setup for both the 10-year and 100-year storm events.



Child Scenarios are created with the *Scenario Manager* tool. When initially created, the Child Scenario will contain the same computation settings as the Base Scenario used in creation. The *Scenario Manager* tool is found in the Ribbon in the following location:

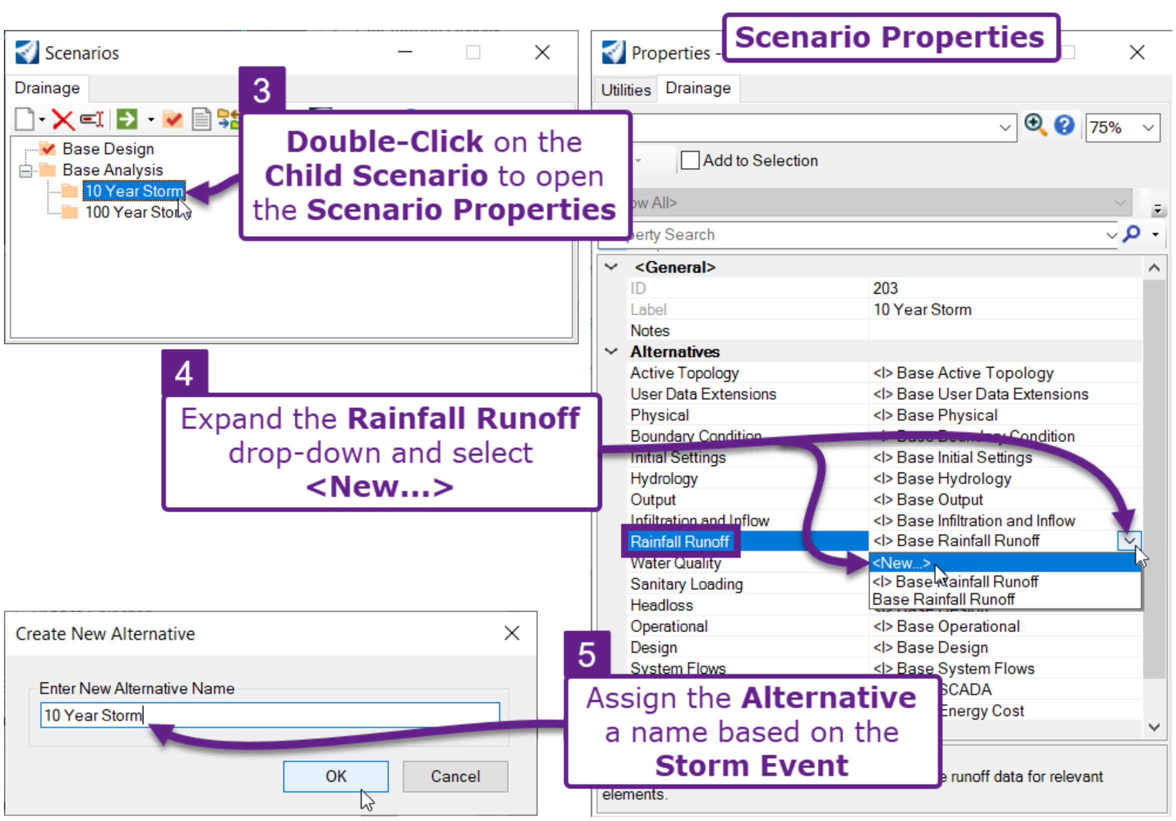
Drainage and Utilities workflow → **Analysis** tab → **Calculation** group



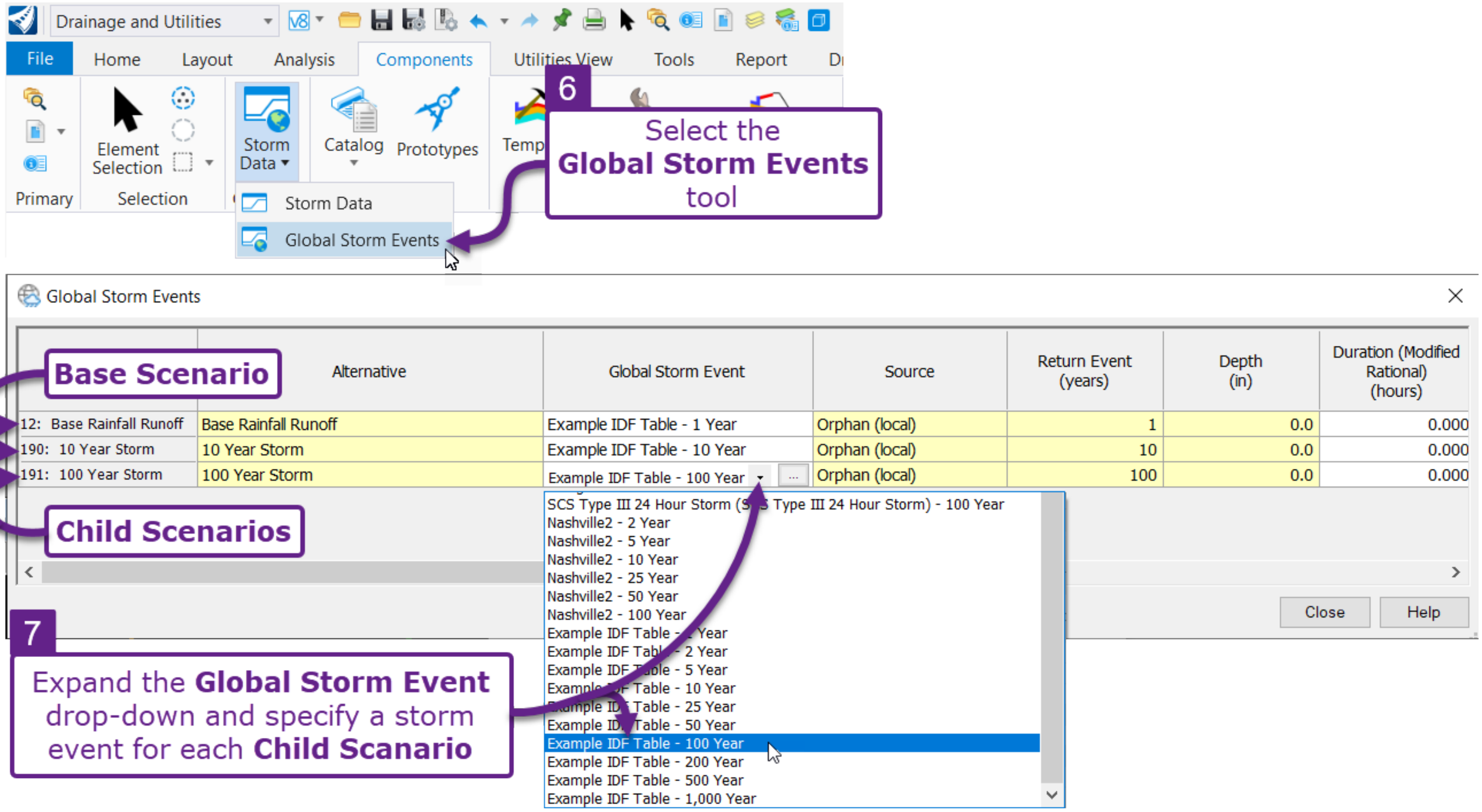
TIP: Name the Child Scenario to correspond with the storm event to be analyzed (i.e., "10 Year Storm").

After creating the Child Scenario, double-click on it to access the Scenario Properties. In the Scenario Properties, an Alternative must be created for the "Rainfall Runoff" setting.

NOTE: If unedited, the Child Scenario will inherit all calculation settings from the Base Scenario. Settings that are inherited from the Base Scenario are marked with a "<I>" prefix. An "Alternative" is a setting that differs from the inherited Base Scenario settings.



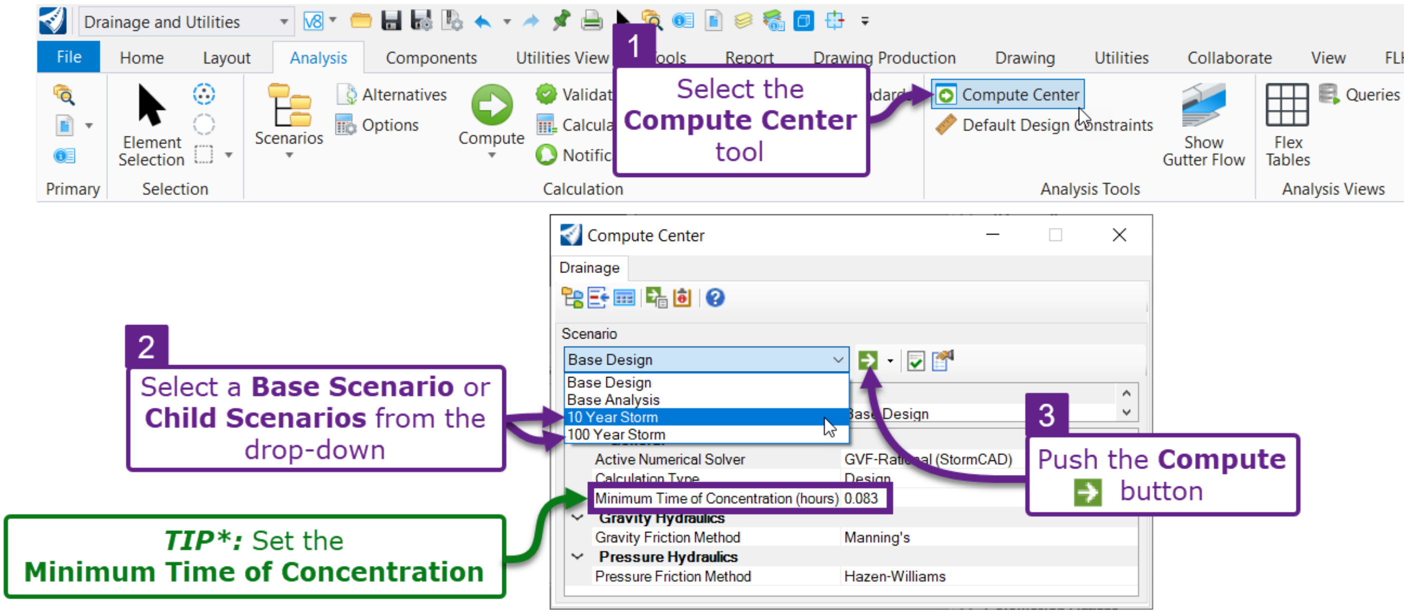
After a new Alterative is created, it must be assigned the appropriate storm event with the *Global Storm Event* tool.



25D – RUN AN ANALYSIS SCENARIO WITH THE COMPUTE CENTER TOOL

The *Compute Center* tool is used to select a Scenario and run the analysis of the drainage network. This tool is found in the Ribbon in the following location:

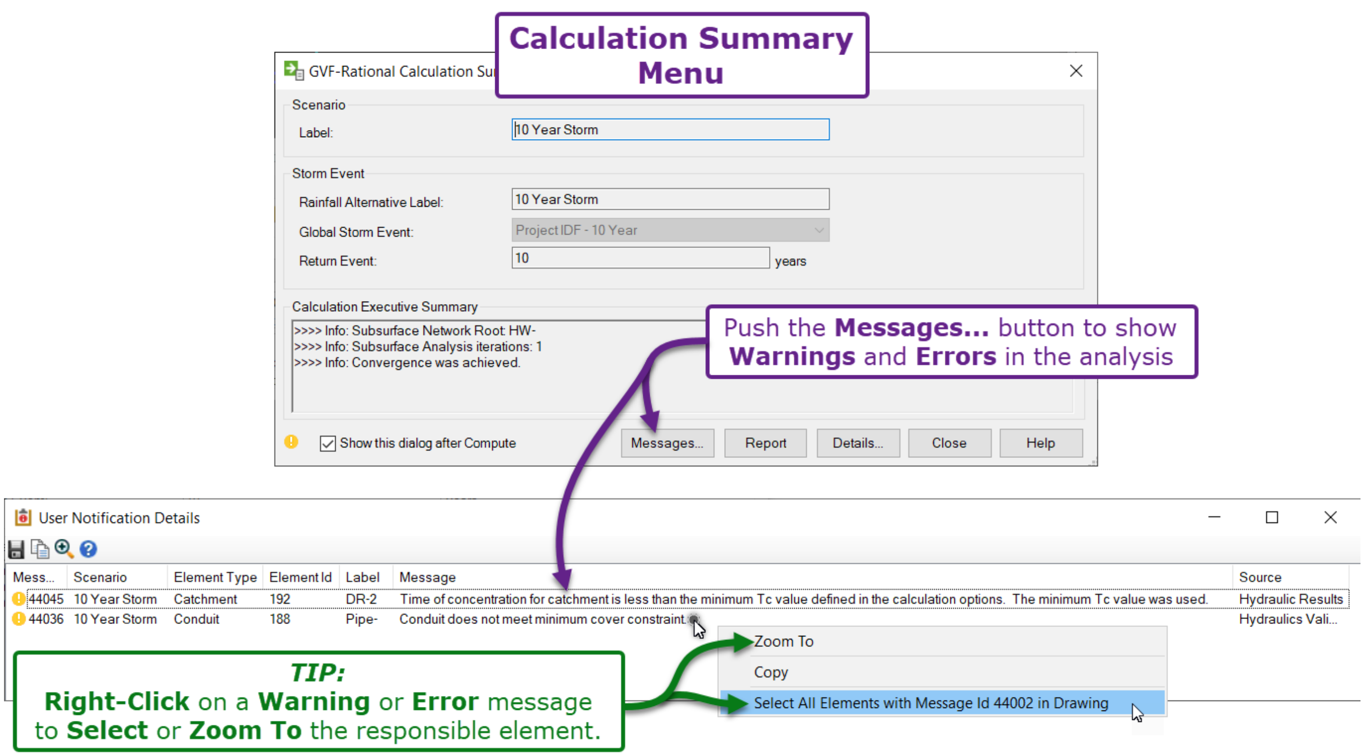
Drainage and Utilities workflow → **Analysis** tab → **Analysis Tools** group



NOTE: The difference between a **Design** and **Analysis** scenario is discussed in *25C.3 Setup Scenarios*. A Design Scenario may resize and reposition Conduits and Nodes.

After running the analysis, the Calculation Summary Menu is shown. Push the **Messages…** button to check Warnings and Errors relating to the analysis.

IMPORTANT: If an Error message is displayed, then the analysis was unsuccessful. All Errors must be rectified to run a successful analysis.



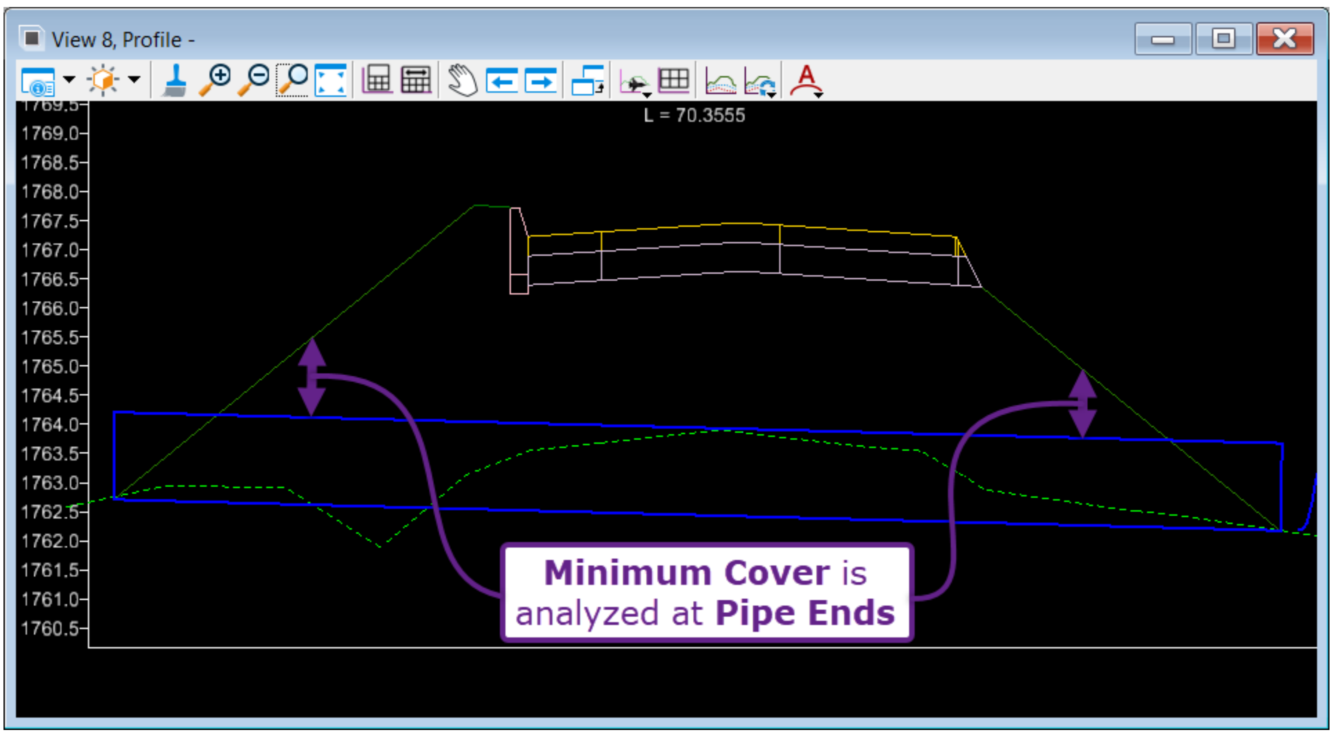
Warning messages may be found, even if the analysis is successful. However, all Warnings messages should be examined for acceptability. Typically, a few common Warning messages are shown.

Common Warning #1: Time of concentration for catchment is less than the minimum Tc value defined in the calculation options. The minimum Tc value was used.

This Warning message appears when the Tc value for a Catchment (drainage area) element is less than the minimum allowable value shown in the *Compute Center*. See the ***TIP*:*** on the previous page to set the minimum Tc value for the analysis in the *Compute Center*

Common Warning #2: Conduit does not meet minimum cover constraint.

All Conduit (pipes) are analyzed for minimum cover. The software analyzes the distance between the ***Active* Terrain Model** and the pipe crown to measure minimum cover. This message will likely be shown even if minimum cover is achieved because the software analyzes minimum cover along the entire length of the pipe. As shown in the graphic below, areas near the pipe ends are analyzed for minimum cover. In practical design, minimum cover must be achieved under the driving surface. Typically, the pipe ends are NOT analyzed for minimum cover.



BEST PRACTICE: Minimum cover should be manually designed and checked by the User. Do NOT rely on the Warning messages.

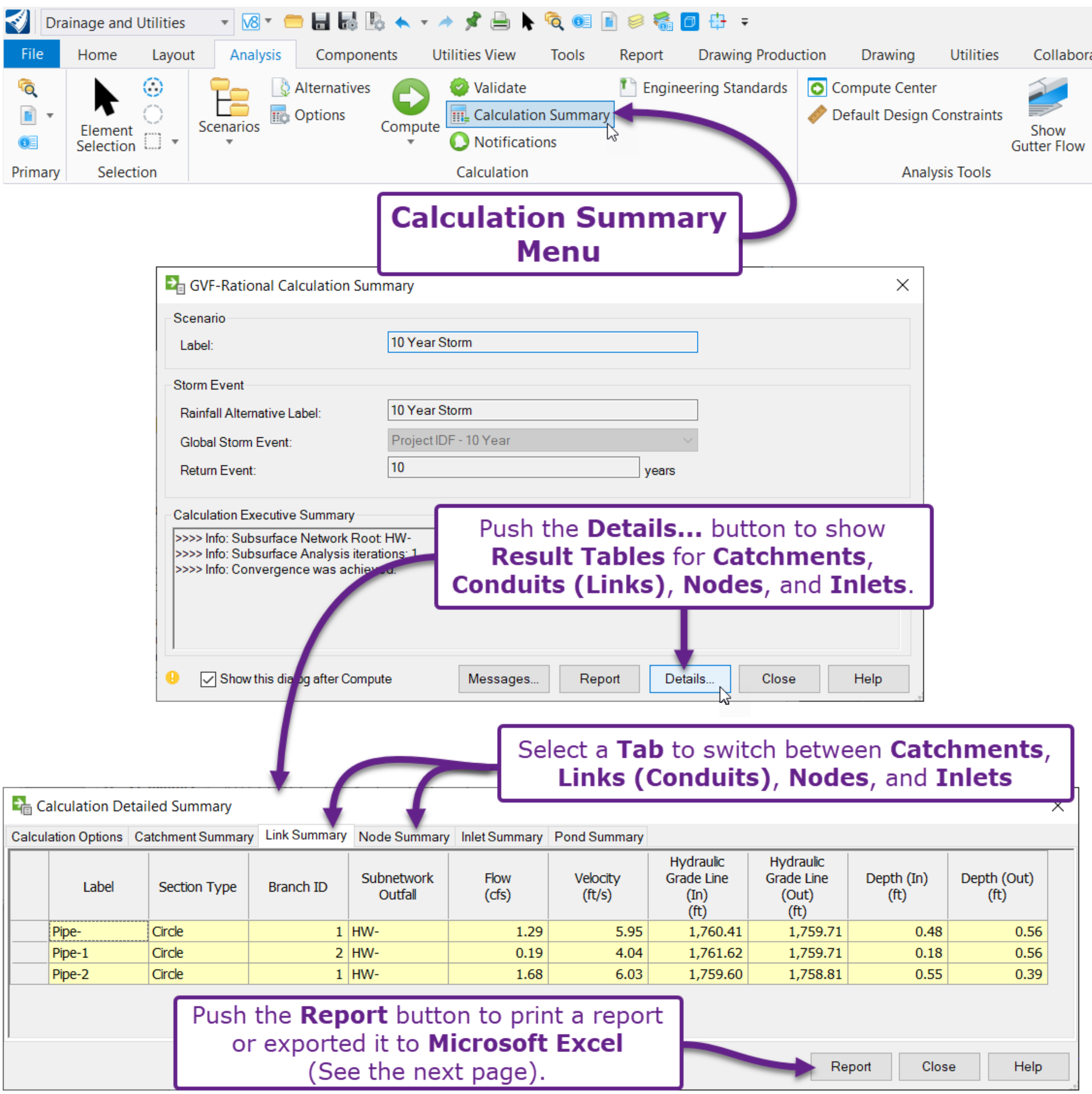
25E – RESULTS: CREATING REPORTS, TABLES, AND PROFILES

After a Design or Analysis Scenario is run in the *Compute Center*, there are many graphical and tabular options for reviewing the results.

25E.1 Generate a Results Table and Export to Microsoft Excel

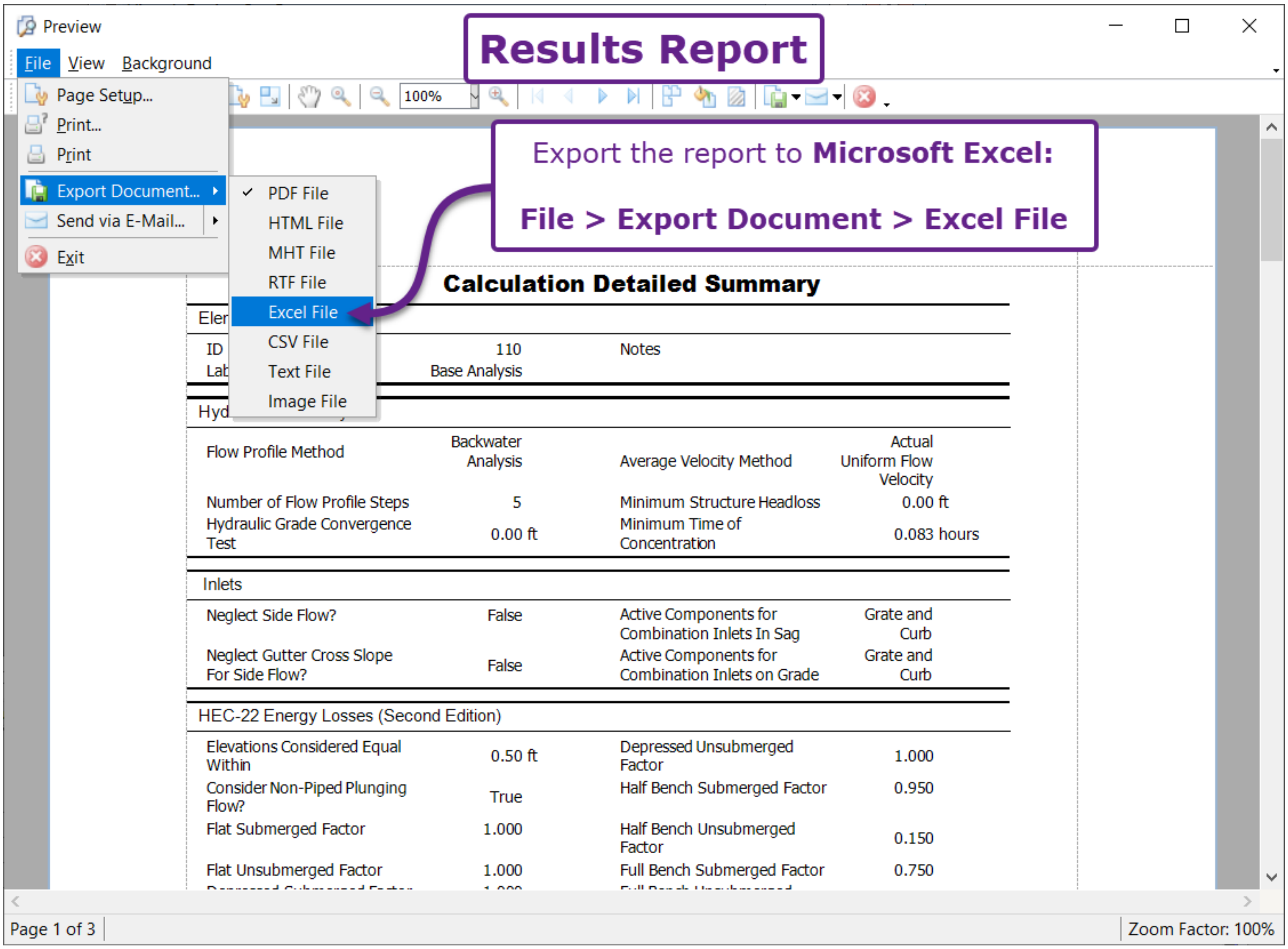
Directly after an analysis is run, the Calculation Summary Menu is shown. Push the **Details…** button to view summary tables for the analysis.

NOTE: If the analysis has already been run, reopen the Calculation Summary Menu with the *Calculation Summary* tool. [**Drainage and Utilities** workflow → **Analysis** tab → **Calculation** group]



As shown on the previous page, push the **Report** button to generate a results report. The results report contains all information provided in the Details Table.

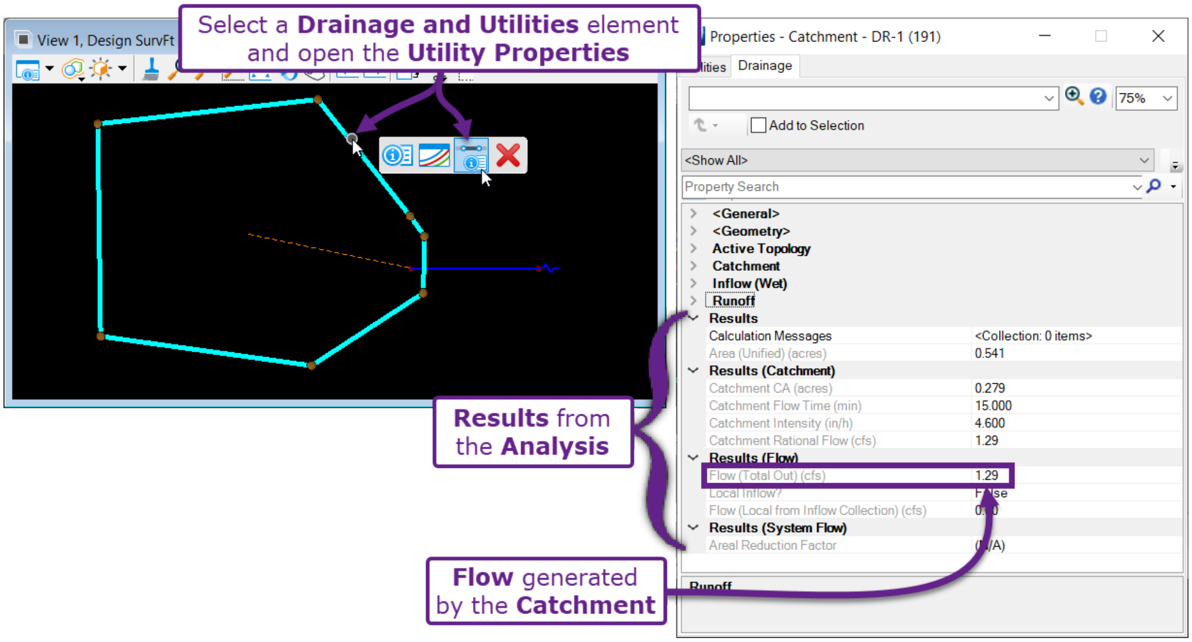
For additional formatting, the results report can be exported to Microsoft Excel.



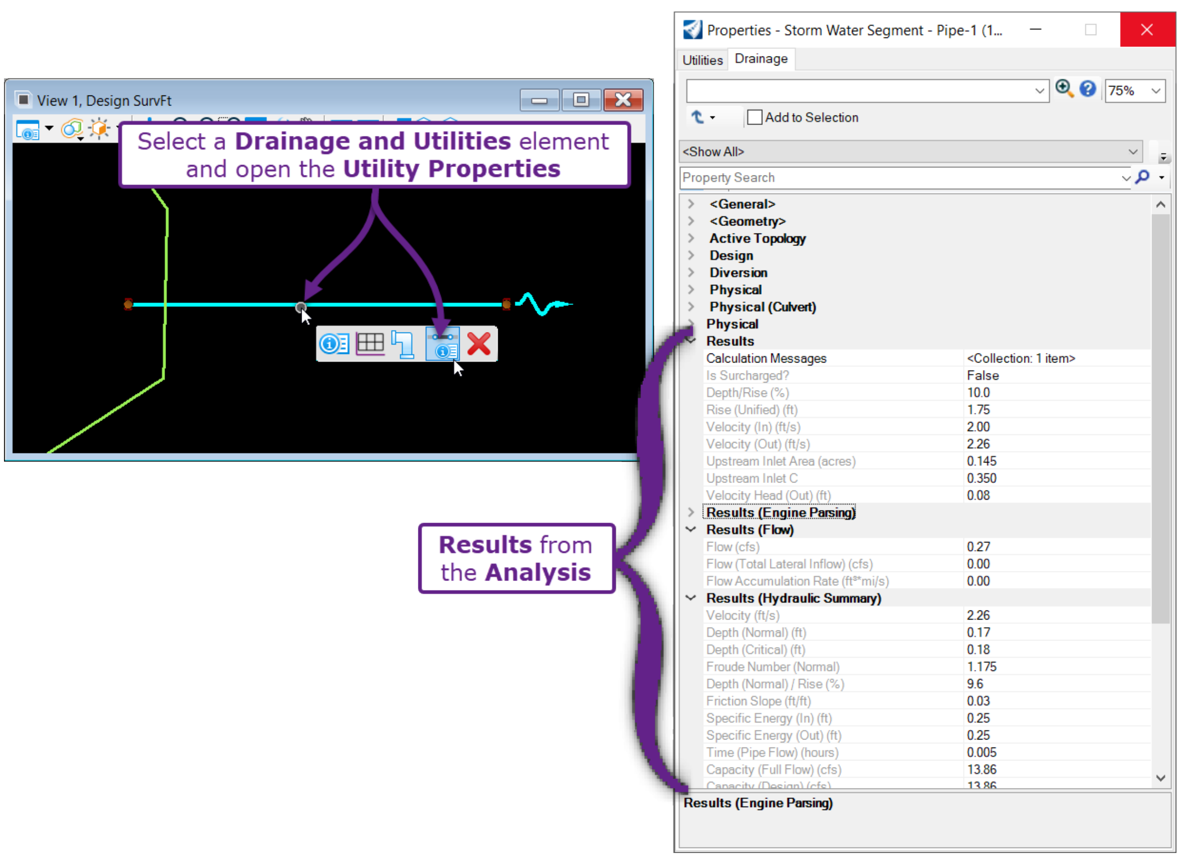
25E.2 Results in the Utility Properties for each Element

After the analysis is run, select a Drainage and Utilities element and open the Utility Properties. All results pertaining to the selected element are shown under the Results drop-downs in the Utility Properties.

For example, select a Catchment element to view the peak runoff flow generated by the drainage area.



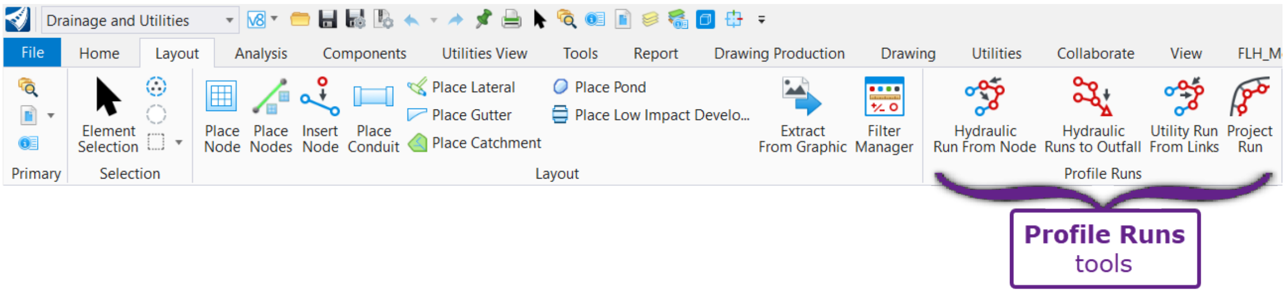
Select a Conduit element to view a variety of results, such as velocity, depth, and Froude Number

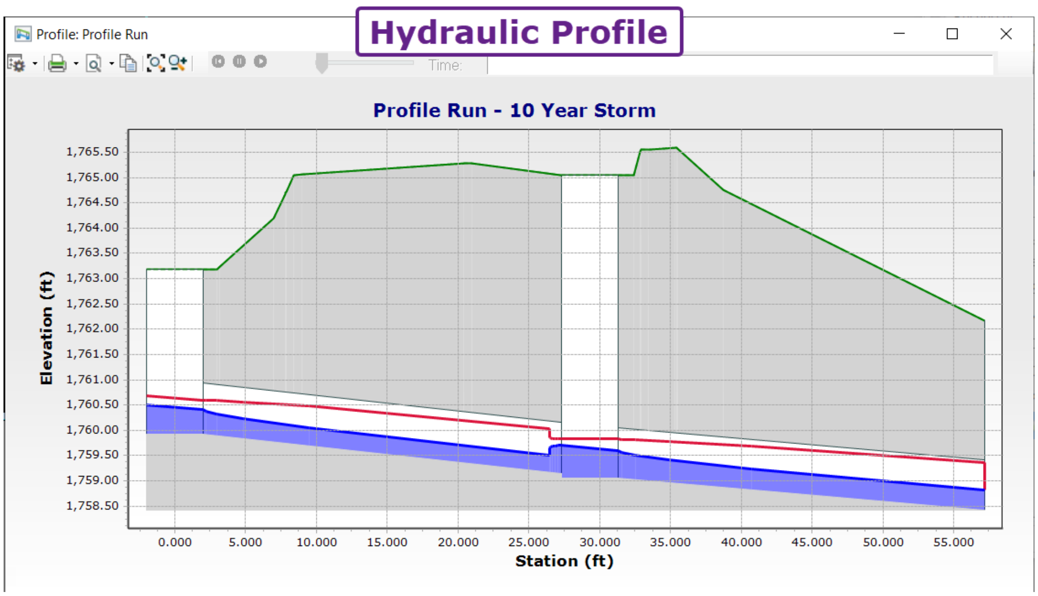


NOTE: In the Utilities Properties menu, results are shown in grey because they CANNOT be directly edited.

25E.3 Create Hydraulic Profiles

The **Profile Runs** tools are used to create hydraulic profiles for a culvert or storm sewer network.





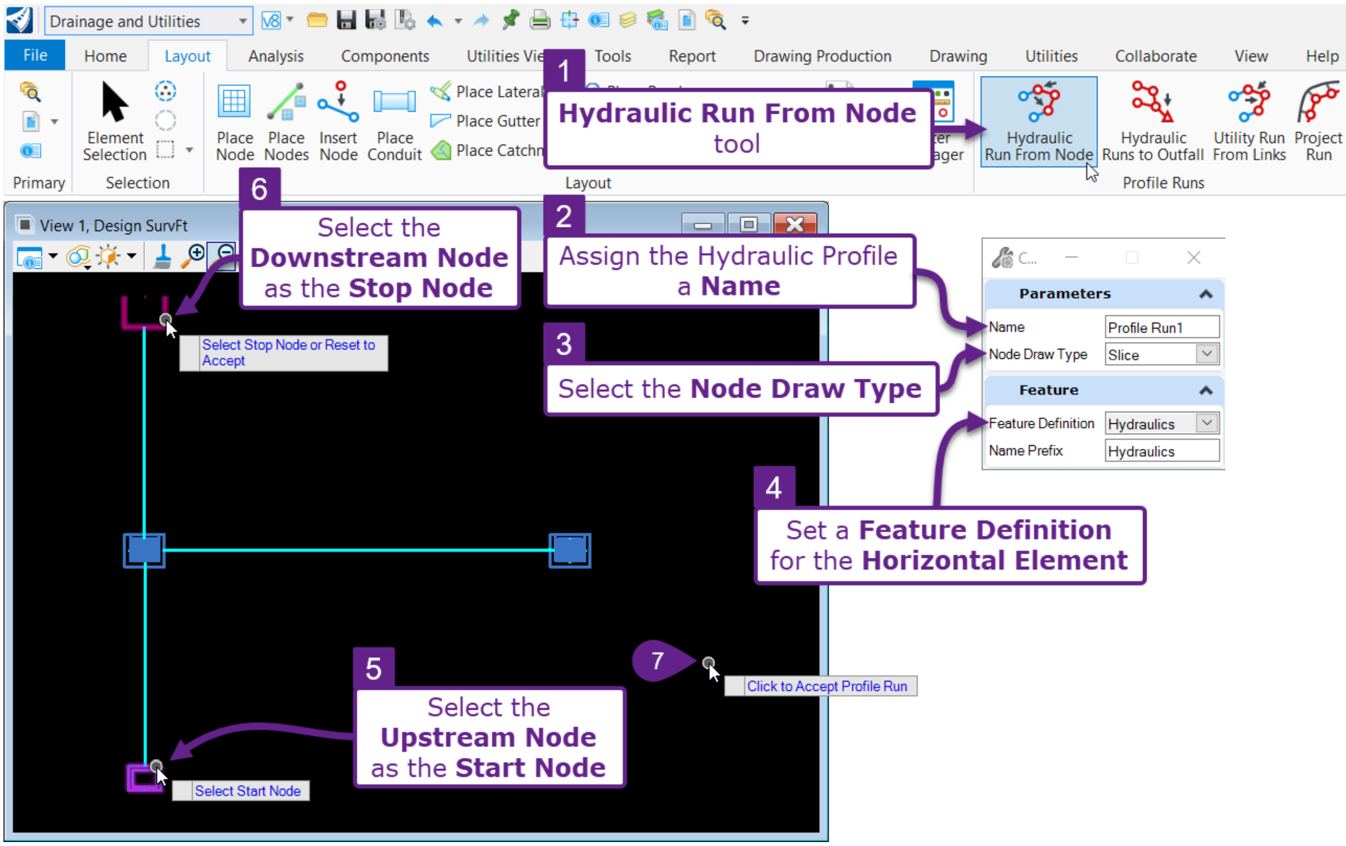
In summary, these tools all accomplish the same task. The primary difference between these tools is how Nodes or Conduits elements are selected to determine the extents of the profile. ***EXCEPTION:*** The *Project Run* tool does NOT create a hydraulic profile. Instead, this tool projects a previously-created hydraulic profile onto an adjacent or parallel alignment.

Profile Runs Tools		
Tool:		**Description:**
Hydraulic Run From Node	Hydraulic Run From Node	Creates a Profile Run between any two selected Nodes. All Conduits and Nodes between the two selected Nodes are shown in the profile. ***NOTE:*** This tool is only compatible with stormwater and wastewater networks.
Hydraulic Runs to Outfall	Hydraulic Runs to Outfall	Creates a Profile Run that begins at the downstream Outfall and extends to the furthest upstream Node. ***NOTE:*** This tool is only compatible with stormwater and wastewater networks.
Utility Run From Links	Utility Run From Links	Creates a Profile Run by selecting Conduit(s), instead of Nodes. ***NOTE:*** This tool is compatible with electric, communication, and water distribution lines.
Project Run	Project Run	After a Profile Run is created with any of the three tools shown above, it can be projected onto a parallel alignment. For example, a storm sewer network that is placed along the curb line can be projected onto the Centerline of Road Alignment and viewed in the *Profile Model*.

25E.3.a Hydraulic Run From Node tool

The *Hydraulic Run From Node* tool is used to create a hydraulic profile for a storm sewer network or culvert.

NOTE: After this tool is used, the resulting hydraulic profile can be viewed from a *Profile Model* perspective or generated from the Explorer.

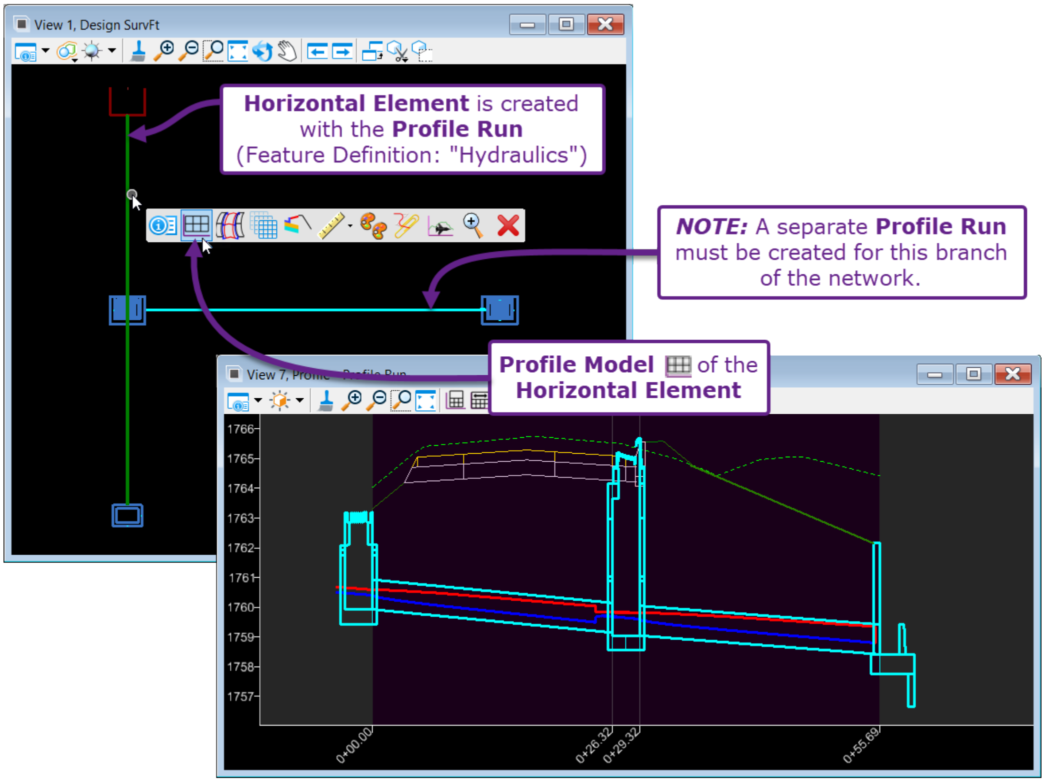


1	From the Ribbon, select the *Hydraulic Run From Node* tool: [***Drainage and Utilities*** → ***Layout*** → ***Profile Runs***].
2	In the *Dialogue Box*, assign the Profile Run a **Name**. The Name will help to identify and locate the Profile Run in the Explorer.
3	In the *Dialogue Box*, select the **Node Draw Type**. The Node Draw Type determines how Nodes are shown in the *Profile Model* of the resulting horizontal element. Select **Box** to show the Nodes as rectangular. Select **Slice** to show an actual representation of the structure and inlet.
4	In the *Dialogue Box*, select a **Feature Definition** for the horizontal element to be created. ***BEST PRACTICE:*** Use the "Hydraulics" Feature Definition, which is found under the "Misc" drop-down. ***NOTE:*** This tool creates a horizontal element that represents the horizontal extents of the hydraulic profile.
5	*Prompt: Select Start Node* – Select the furthest upstream Node element. If the outfall (downstream) Node is selected, then the hydraulic profile will be reversed.
6	*Prompt: Select Stop Node or Reset to Accept* – Select the furthest downstream Node or the outfall Node. After this step, the horizontal extents of the hydraulic profile is previewed with an orange line.

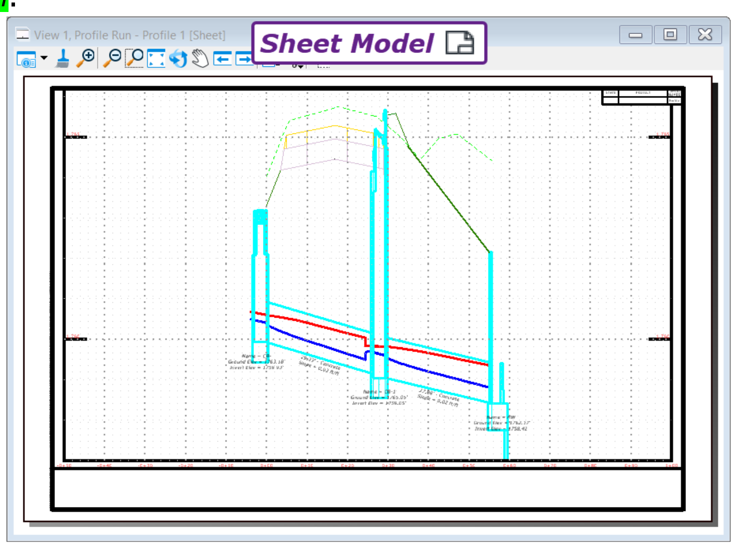
25E.3.b Hydraulic Profile in the Profile Model

As mentioned in step 4 on the previous page, the Profile Run tools creates a horizontal element that represent the extents of the profile. The hydraulic profile is shown in the *Profile Model* f the horizontal element.

TIP: If the *Profile Model* does NOT show the Finished Grade or Existing Ground Terrain Models, then use the *Profile from Surface* tool to show these graphics. Use the *Create 3D Cut* tool to show the Corridor or other proposed modeling graphics. See *7F.1.d Show Multiple Terrain Models in a Profile Model* and *7F.1.e Show Corridor and 3D Elements in a Profile Model with the Create 3D Cut tool*.



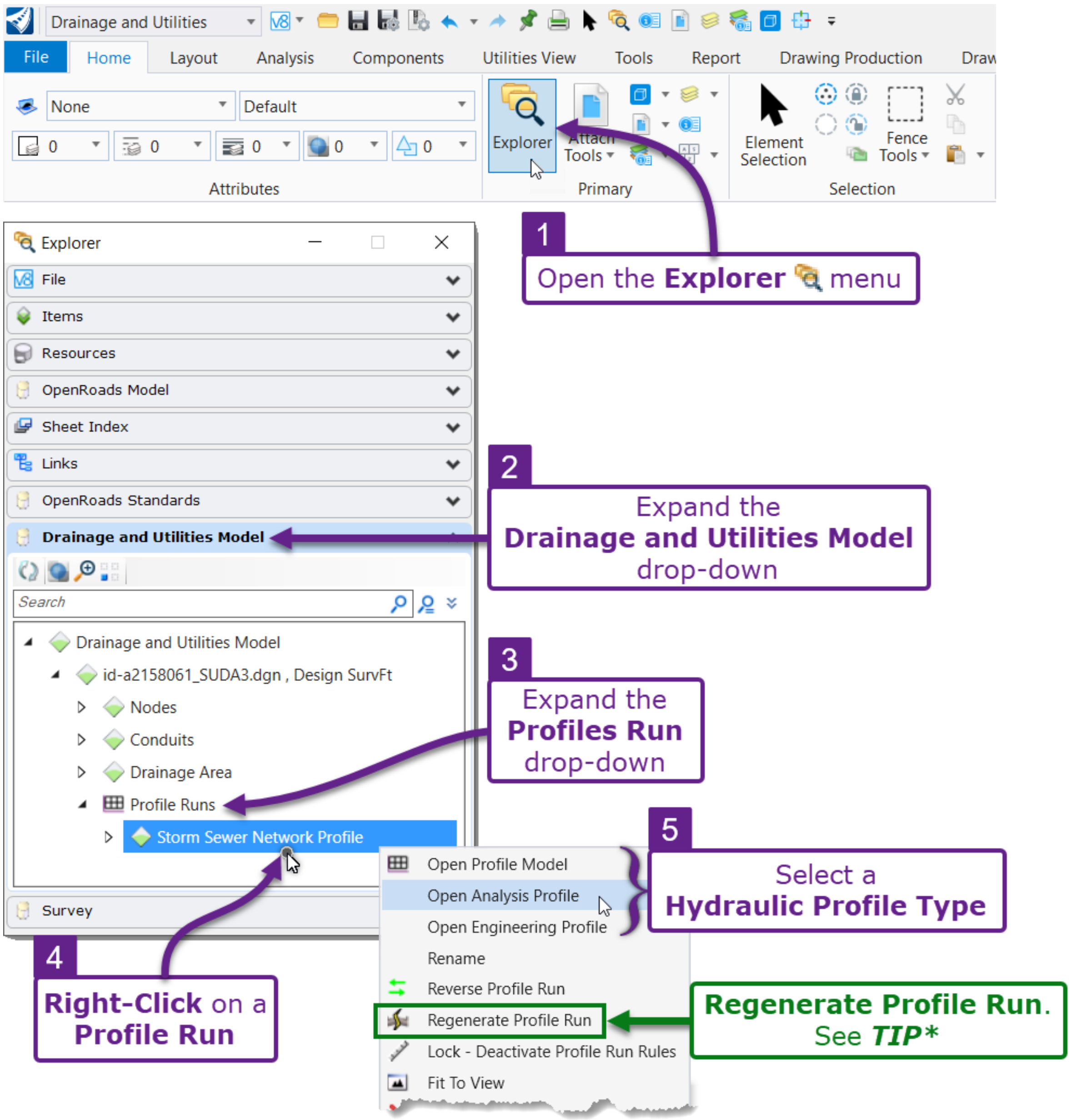
TIP: A plan sheet can be created from the *Profile Model* using a Named Boundary, Drawing Model, and Sheet Model. The process for creating a plan sheet from a *Profile Model* is shown in *Chapter 14 - Plan Sheet Production*.



TIP: The horizontal element can be used in conjunction with the *Project Run* tool to project the hydraulic profile onto a parallel element (i.e., the centerline of road alignment).

25E.3.c Hydraulic Profiles in the Explorer

After using a Profile Run tool (i.e., the Hydraulic Run From Node tool), hydraulic profiles are generated from the Explorer menu. Profile Runs are located under the **Drainage and Utilities Model** drop-down in the Explorer.

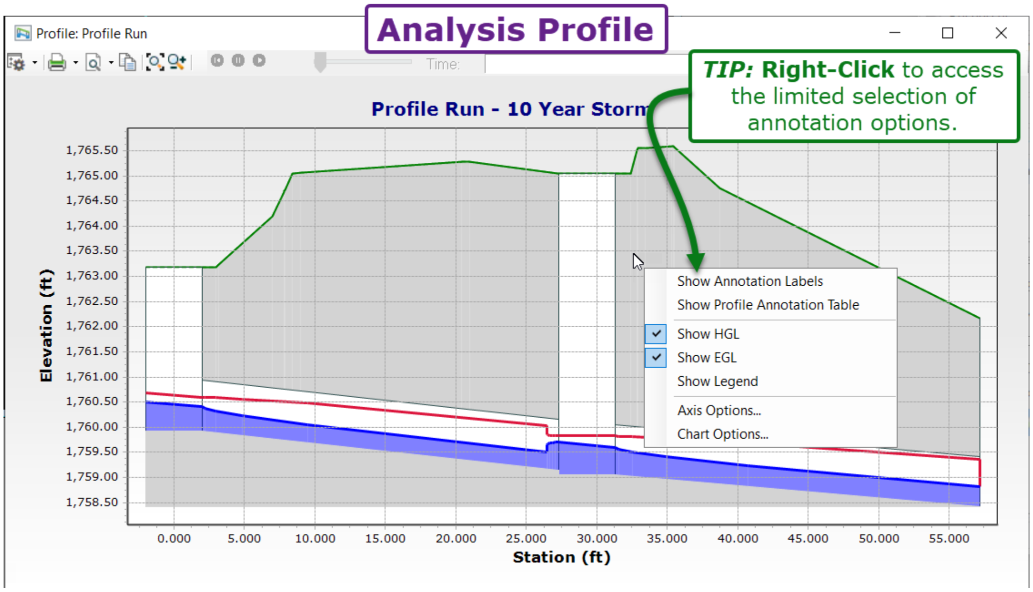


TIP*: If the hydraulic analysis is re-run in the *Compute Center*, then the **Regenerate Profile Run** tool must be used to update the hydraulic profile to reflect the most recent analysis.

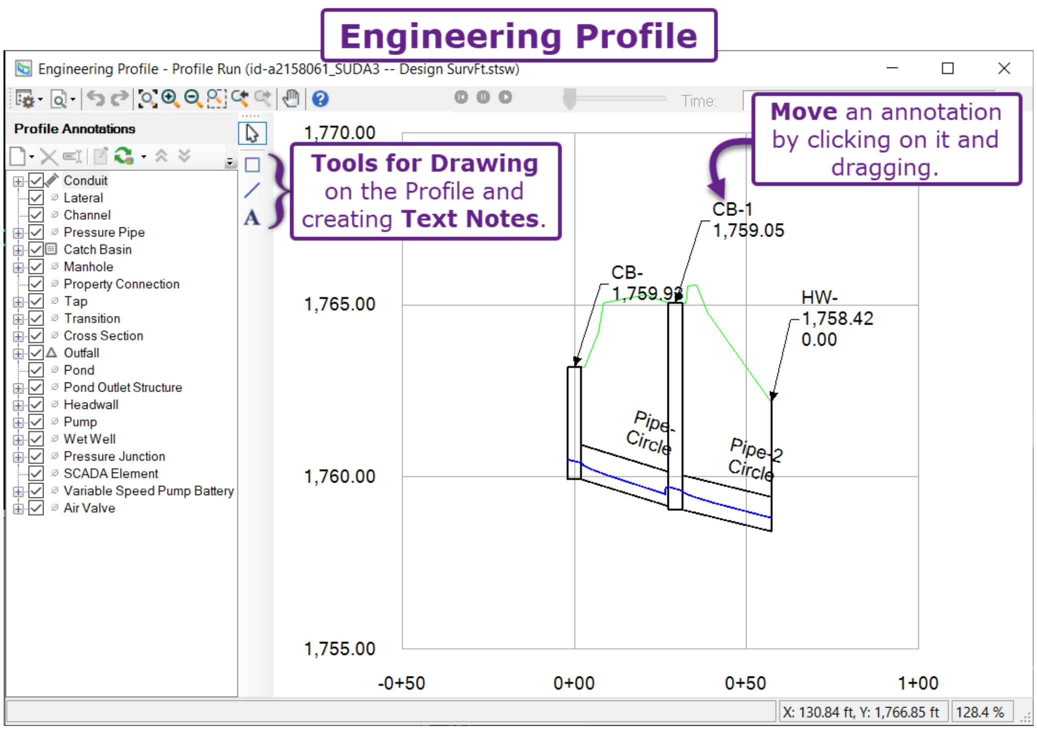
As shown on the previous page, there are three types of hydraulic profiles: the Profile Model, the Analysis Profile, and the Engineering Profile.

NOTE: The Profile Model is discussed in *25E.3.b. Hydraulic Profile in the Profile Model*. The Profile Model can be used to create a Sheet Model, which can then be annotated, labeled, and printed.

Analysis Profile: The Analysis Profile is intended for quick viewing of the hydraulic profile. The Analysis Profile has very limited options for annotation and labeling.



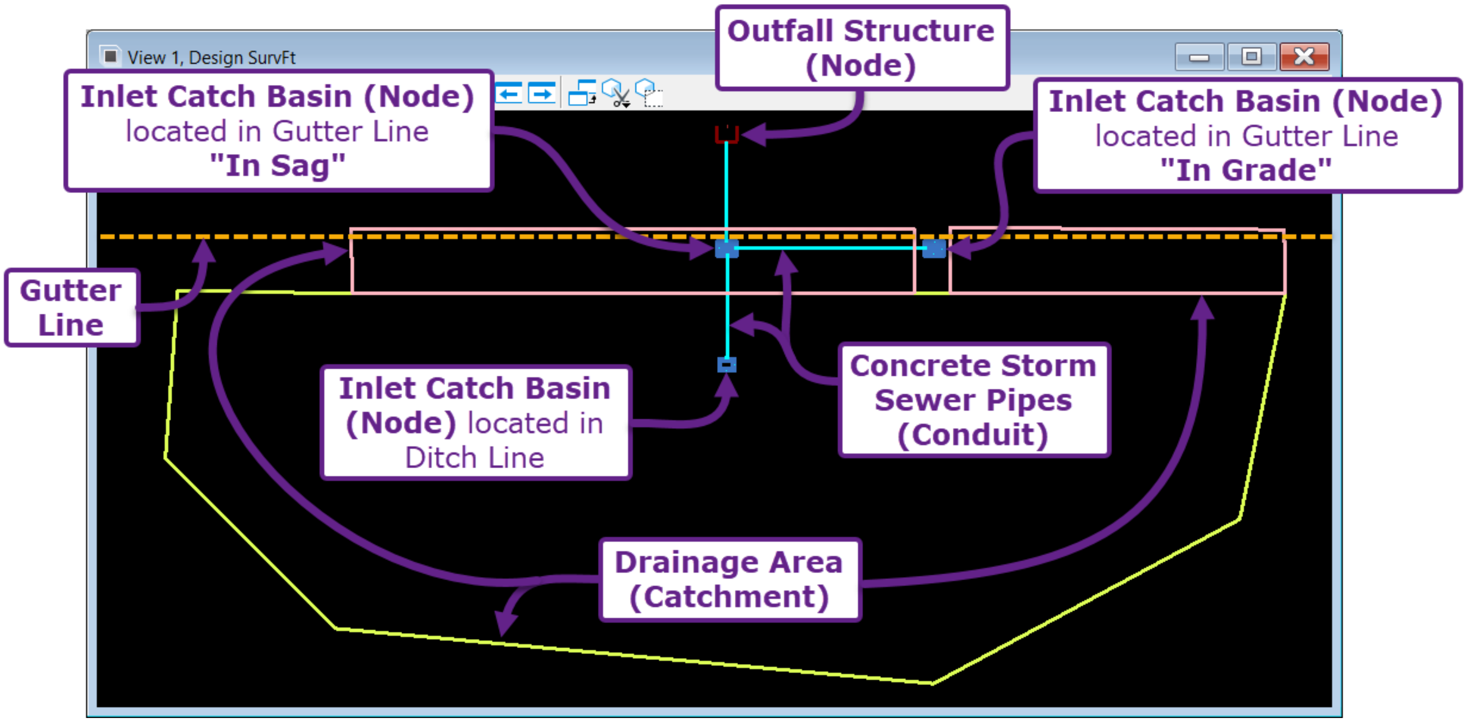
Engineering Profile: When initially opened, the Engineering Profile contains a few automatically-created annotations. Custom annotations can be created for the Engineering Profile. Also, the vertical exaggeration of the profile grid can be edited.



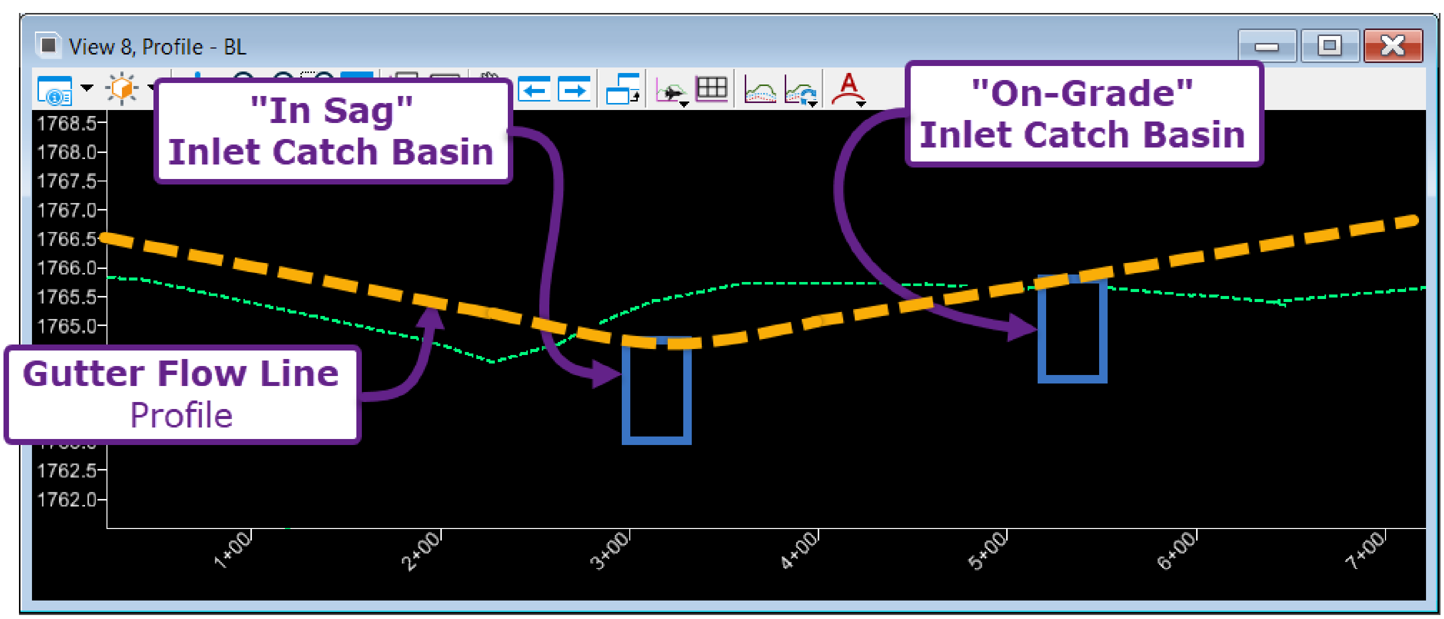
NOTE: The Ground Profile shown in the Engineering and Analysis Profiles is drawn from the *active* Terrain Model. Create a Finished Ground Terrain Model and *activate* it for display in the hydraulic profile graphics. See *Chapter 22 – Proposed Terrain Model Creation*.

25F – MODEL INLETS, CATCH BASINS, AND CURBS

In this workflow a storm sewer network is modeled and designed. The network consists of inlet catch basins (Node elements), storm sewer pipes (Conduit elements), an outfall structure (Node element), and a Gutter element. Drainage areas are assigned to each inlet catch basin using Catchment elements.



In this example workflow, there are two inlet Nodes located on a curb line (dashed orange line). The inlet Node located on the right-side is "On-Grade". The inlet Node located on the left-side is "In Sag". If the "On-Grade" inlet CANNOT accept all runoff generated by the Catchment area, then a portion of the runoff flow is bypassed along the Gutter to the "In Sag" inlet. The "In Sag" inlet is located at a vertical low-point and CANNOT bypass flow. If the runoff flow exceeds the "In Sag" inlet capacity, then water will pond up above the "In Sag" inlet. The "On-Grade" inlet will NOT pond water. Specifying whether an inlet catch basin is "On-Grade" or "In Sag" is accomplished in the Utility Properties, which is shown in *25F.1.a Inlet Catch Basin Node Utility Properties*.



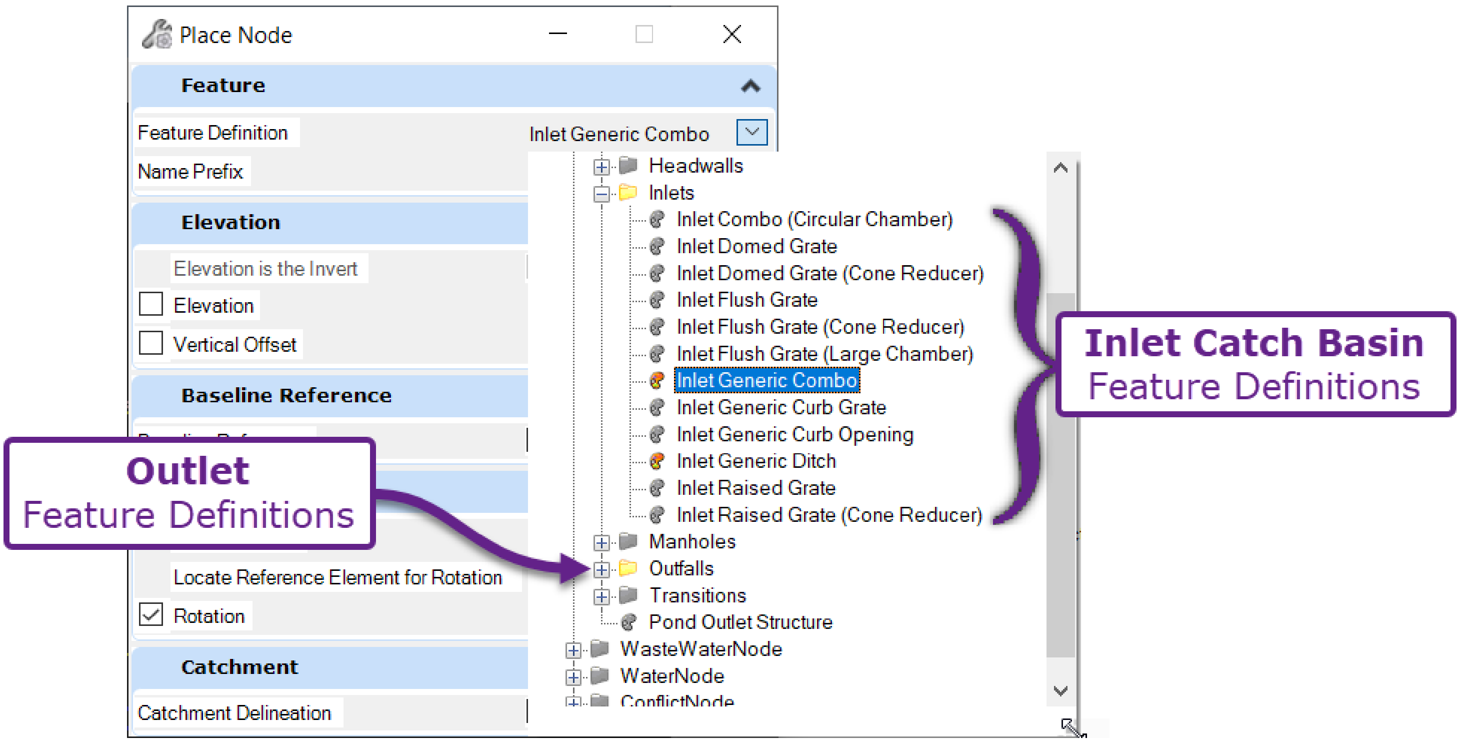
Automated Design for Pipe Conduit Sizes: OpenRoads is capable of automatically choosing and optimizing the diameter of Conduit placed between the Nodes. Using the *Alternatives* tool, design criteria can be programmed for the automatic selection of Conduit sizes. For example, a 10-year storm **Design Scenario** can be run and minimum Conduit sizes will be selected to accommodate the storm event. Then, a 50-year storm **Analysis Scenario** can be run to analyze how the pipe sizes perform in a larger storm event. The Conduit sizes will NOT automatically change in an Analysis Scenario.

Programming and running a Design Scenario is shown in *25F.6 Program the Design Scenario*.

25F.1 Place Catch Basin Nodes and Set Utility Properties

The *Place Nodes* tool is used to create the catch basin Nodes. The placement of Nodes is discussed in *25B.1 Create the Inlet and Outlet Nodes*.

The Node Feature Definition determines which type of structure is placed. For placing inlet catch basins, use Feature Definitions from the **StormWaterNode** → **Inlets** drop-down. folder. The downstream Outlet Node use a Feature Definition from the **StormWaterNode** → **Outlet** folder.



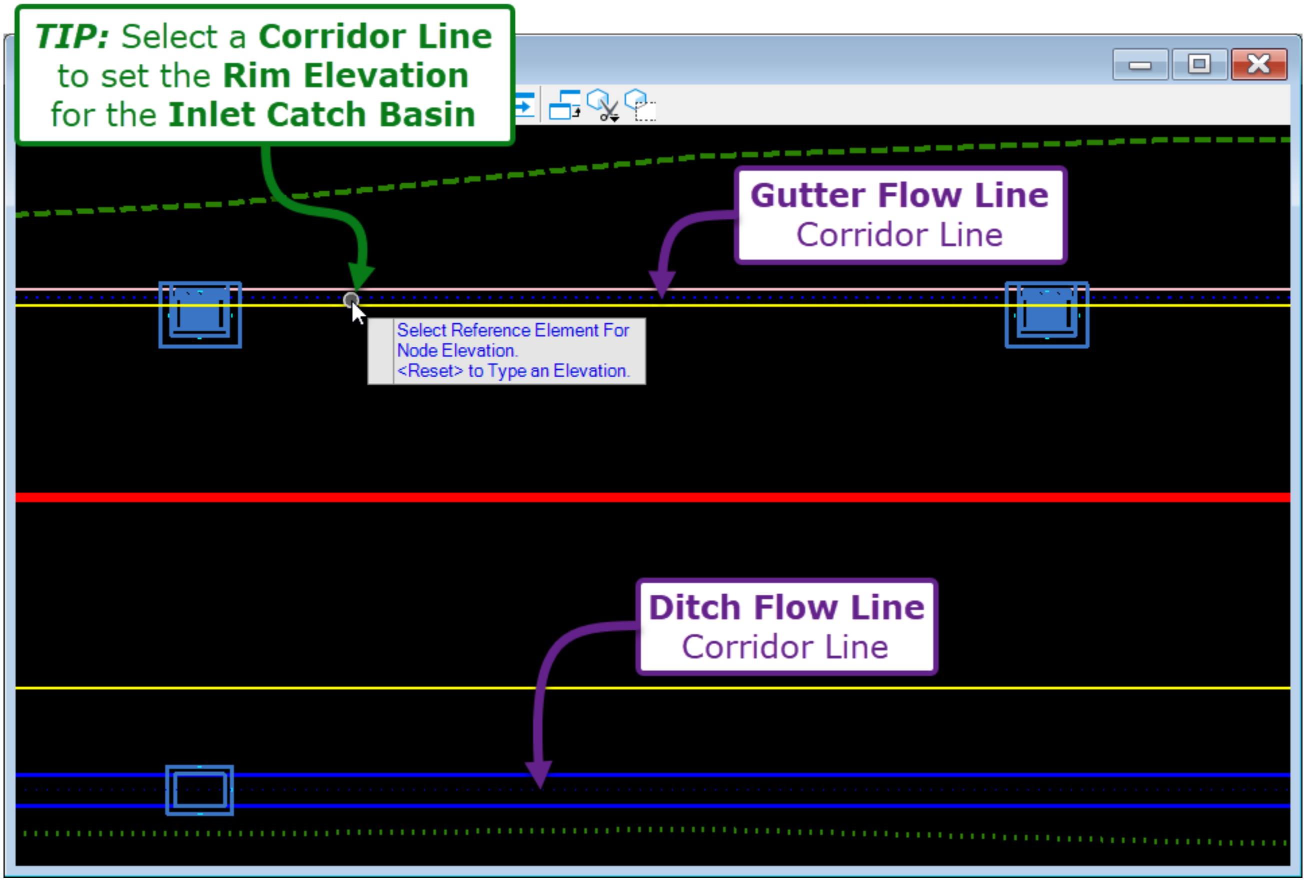
NOTE: The Node Feature Definition library is limited to only a few catch basin and inlet types. When selecting a Feature Definition, choose the appropriate inlet configuration and catch basin structure shape. The grate opening size for an inlet can be customized after placement of the Node. Similarly, the dimensions and invert (bottom) elevation for the catch basin portion of the Node can be customized.

For example, the “Inlet Combo (Circular Chamber)” Feature Definition contains a gutter grate and a curb opening inlet configuration with a circular catch basin structure shape. The “Inlet Generic Combo” Feature Definition contains a gutter grate and curb opening inlet, but with a rectangular catch basin structure shape.

The “Inlet Generic Ditch” Feature Definition is intended for placement in a Ditch. Feature Definitions identified with “… (Cone Reducer)” contain a cone placed atop a circular catch basin structure.

TIP: In placement of the Node, the prompt "*Select Reference Element for Node Elevation*" is shown to set the **Rim Elevation** for the inlet catch basin. In this step, a Terrain Model, an Alignment (must contain an *Active* Profile), or a Corridor Line can be selected. Alternatively, an elevation can be manually specified.

In this scenario, the Gutter Flow line (generated by the Corridor) is selected for the elevation placement of the gutter inlet Nodes. For the ditch inlet Node, the Ditch Flow line generated by the Corridor is selected.

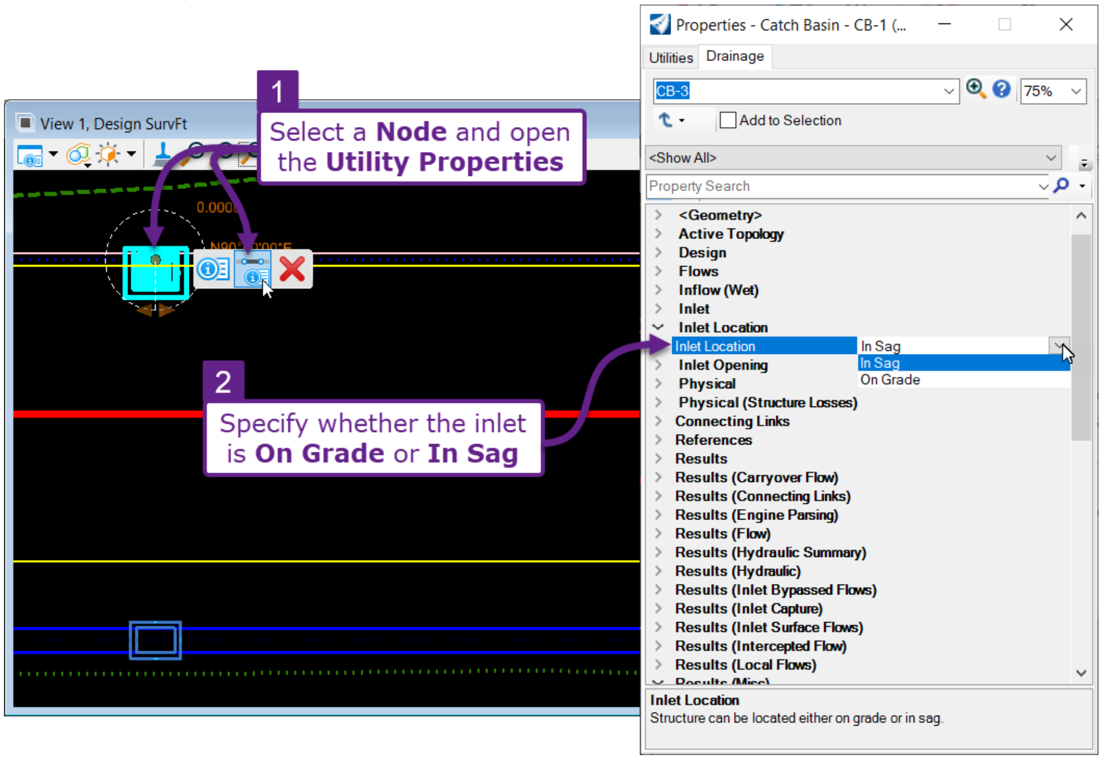


Rotation of Node elements: The rotation of the inlet catch basin nodes should be in line with the gutter line, ditch line, or whatever feature they correspond with. Use the **Relative to Alignment** Rotation Method to ensure that the Node is in line with the desired feature. This procedure is shown in *25B.1 Create the Inlet and Outlet Nodes*.

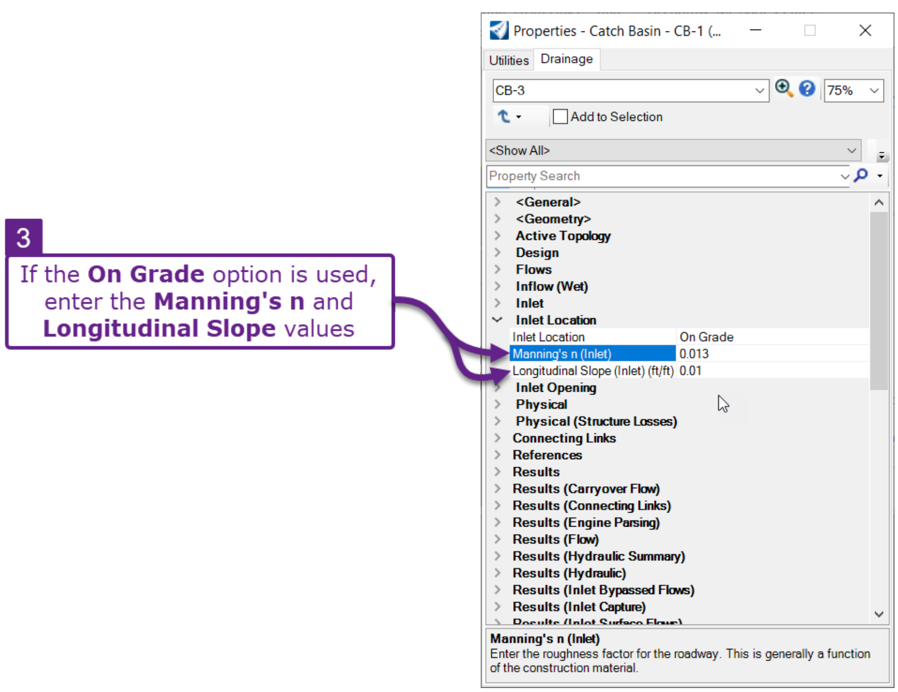
25F.1.a Inlet Catch Basin Node Utility Properties

After placing the Nodes, the Utility Properties need to be edited to set the hydraulic properties for the inlet catch basins. The following properties should be examined before running the hydraulic analysis.

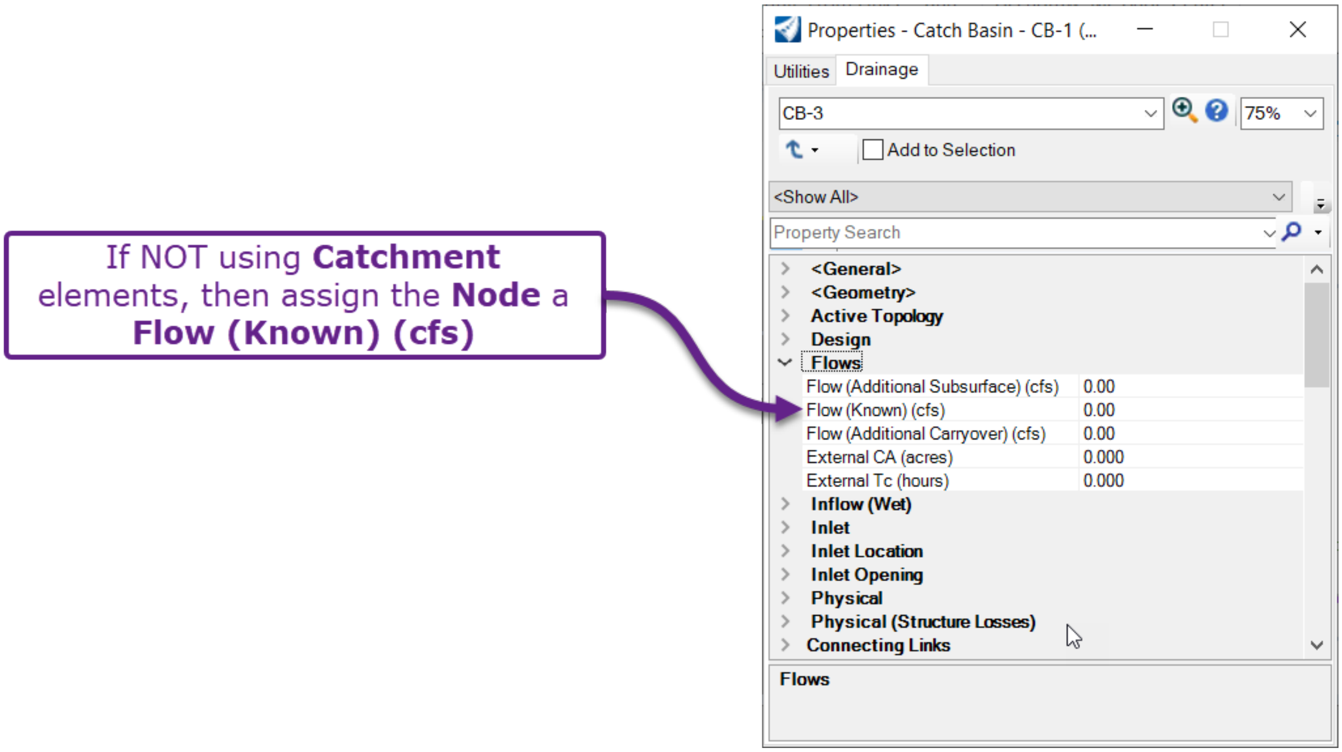
Inlet Location - "On Grade" vs "In Sag": By default, all inlet catch basin Nodes are assigned to the "On Grade" for the **Inlet Location** property. If the inlet catch basin is located at a vertical low point, then the assign it to the "In Sag" option.



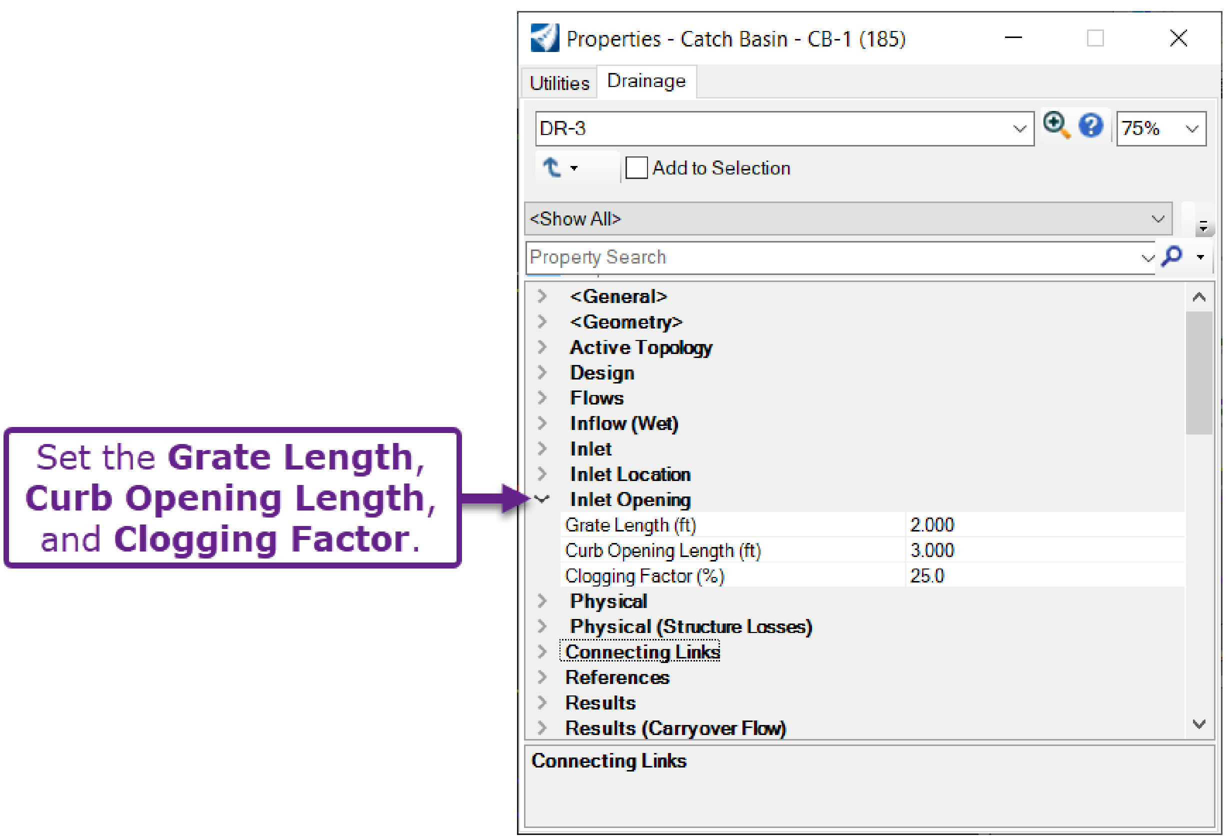
If the "On Grade" option is set, then the slope and the Manning's n value for the gutter line or roadway must be specified. The **Longitudinal Slope (Inlet) (ft/ft)** property corresponds with the slope of the curb flow line. The **Manning's n (inlet)** property corresponds with the roughness value for the curb or road.



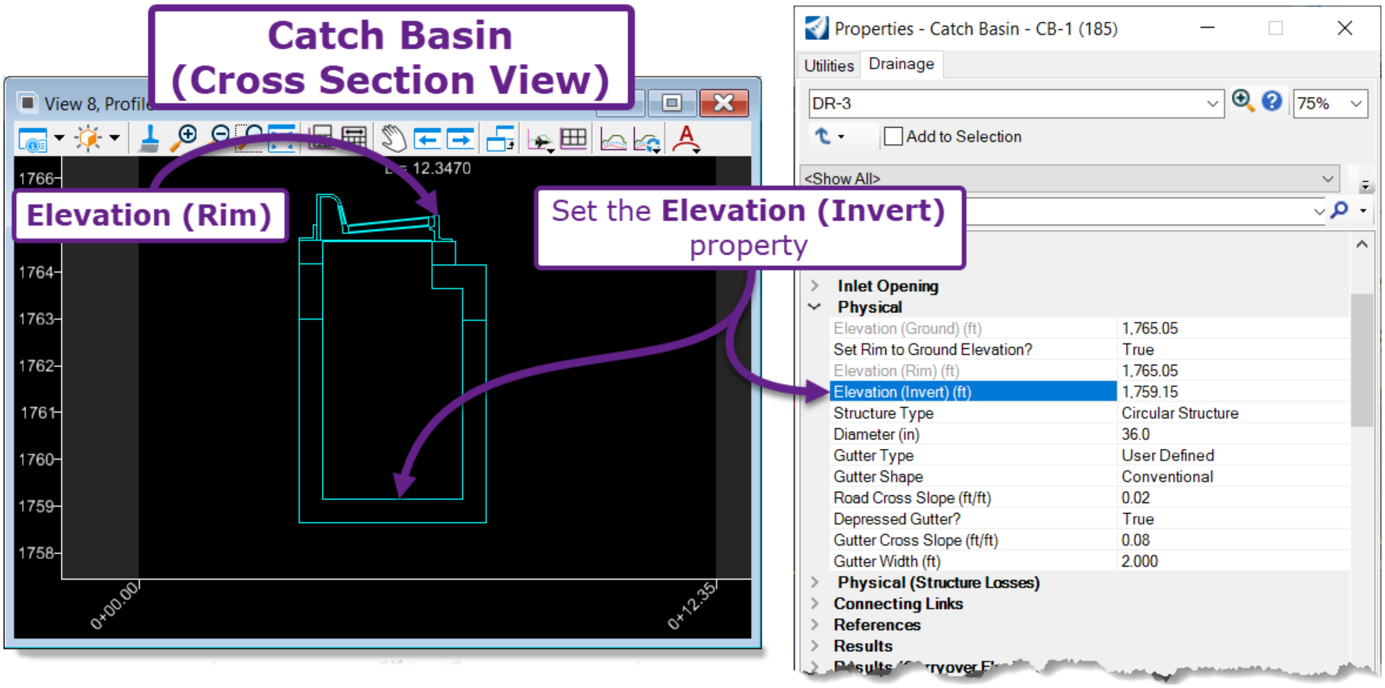
Specify a Known Flow to the Node (if NOT using a Catchment): Typically, Catchment elements are created to represent the drainage area contributing to an inlet catch basin Node. However, Catchment elements are NOT necessary if the Flow to the Node is known. If NOT using a Catchment, then enter a value into the **Flow (Known) (cfs)** property, located under the **Flows** drop-down.



Inlet Opening – Set the Inlet Grate Geometry and Clogging Factor: The properties under the **Inlet Opening** drop-down determine the dimensions for the inlet grate and curb opening. The **Clogging Factor** determines the efficiency of the grate and curb opening.

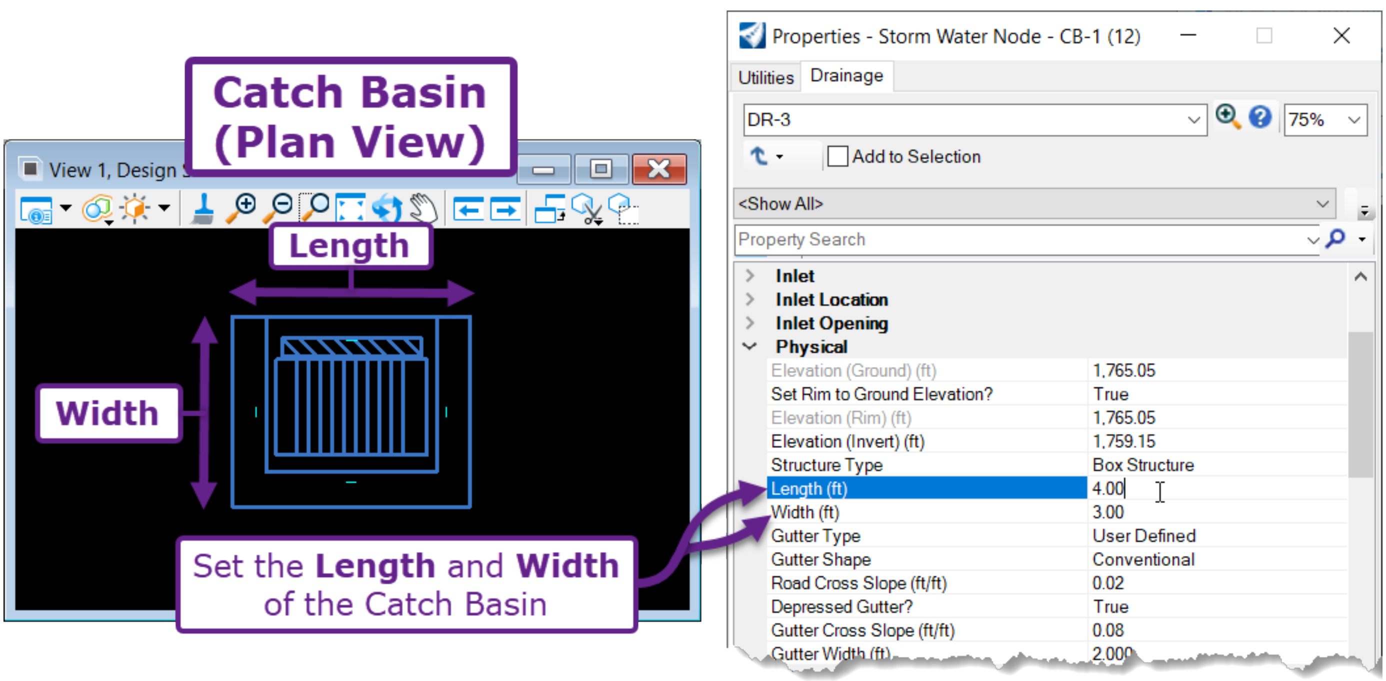


Physical – Set the Invert of the Catch Basin Structure: The overall height of the catch basin structure is determined by the **Elevation (Invert) (ft)** and the **Elevation (Rim) (ft)** properties found under the **Physical** drop-down. The Elevation (Invert) property corresponds with the bottom surface of the inside of the catch basin structure. The Elevation (Rim) property is NOT editable because it is set in the initial placement of the Node, using a Reference Element is selected.



NOTE: In later steps, Conduit elements are created between the catch basin Nodes. Initially, the Conduit inverts are automatically set to the Node invert elevation. After placement, the Conduit inverts can be manually edited to lay above the Node invert elevation. Editing Conduit inverts is shown in *25F.3.a Set the Conduit Invert Elevations in the Utility Properties*.

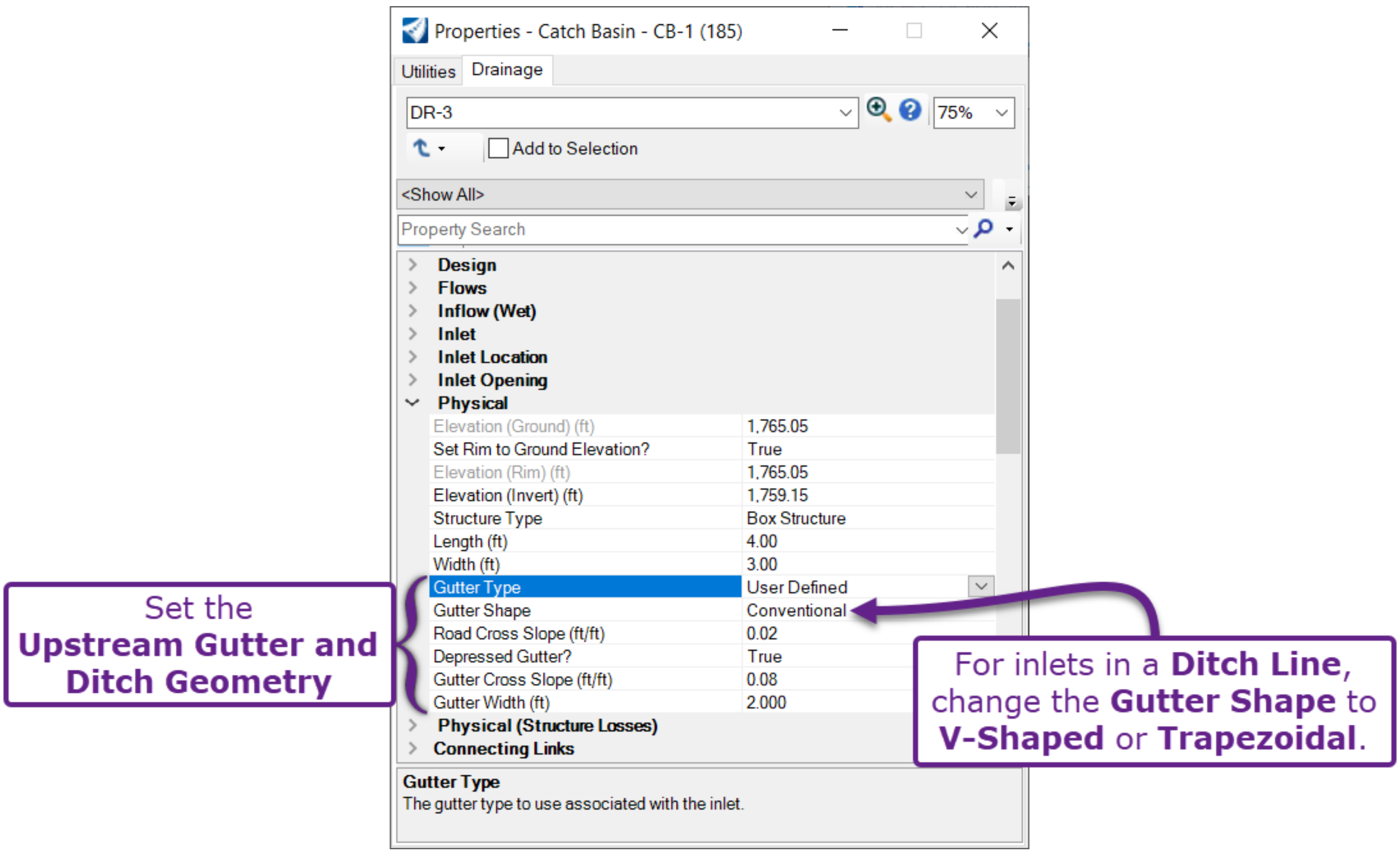
Physical – Set the Catch Basin Geometry: The Catch Basin dimensions are determined by the **Length (ft)** and **Width (ft)** properties found under the **Physical** drop-down.



If the Catch Basin is a circular structure, then the **Diameter (ft)** property is shown.

NOTE: Changing **Utility Properties** does NOT affect the graphical representation of the Node element in the *2D Design Model* and *3D Design Model*. However, the graphical representation has NO affect on the hydraulic analysis of the Node. The hydraulic analysis ONLY uses Utility Properties values to analyze the Node. The graphical representation is inconsequential.

Physical – Set the Upstream Gutter and Ditch Geometry: The upstream gutter or ditch geometry affect the hydraulics of water entering the inlet catch basin Node. Set the properties shown below to match the geometry of the gutter or ditch leading into the Node.



If the Node is in a ditch, then change the **Gutter Shape** to either "V-Shaped" or "Trapezoidal". When the Gutter Shape is changed, the geometry properties listed below it will change to reflect ditch geometry,

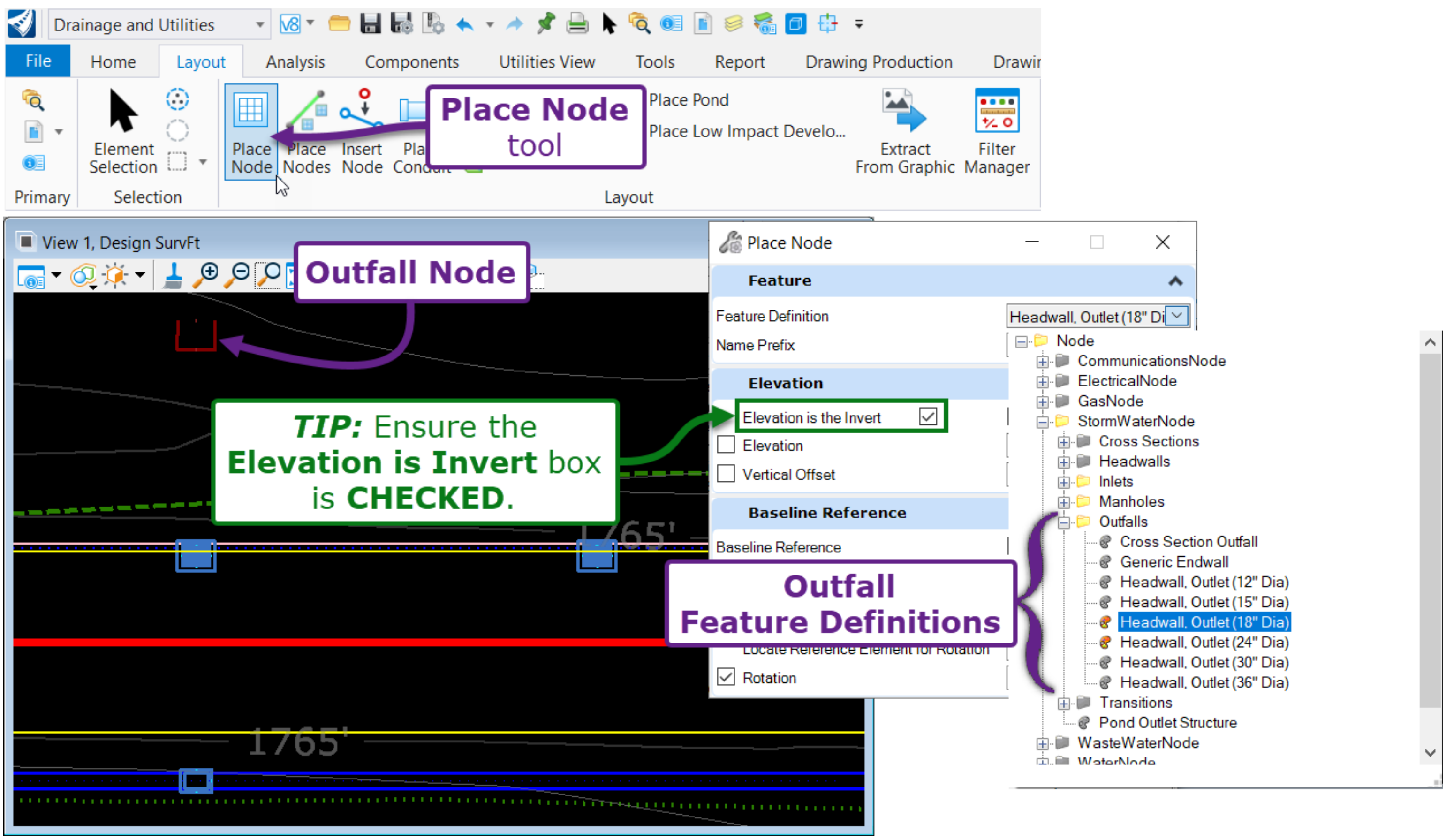
25F.2 Place the Outfall Node

An Outfall Node needs to be placed to designate a discharge location for the catch basin network.

Use the *Place Node* tool to create the Outfall Node. Use a Feature Definition under the **StormWaterNode** → **Outfalls** drop-down.

TIP: When placing the Outfall Node, ensure the **Elevation is Invert** box is CHECKED. If this box is UNCHECKED, then the Outfall Node is placed relative to the pipe crown (top) elevation.

TIP: In placement of the Node, the prompt "*Select Reference Element for Node Elevation*" is shown. Select a Terrain Model, Alignment, or Corridor Line to set the invert elevation of the Outfall Node.



NOTE: Typically, there are NO adjustments that need to be made in the Utility Properties for the Outfall Node.

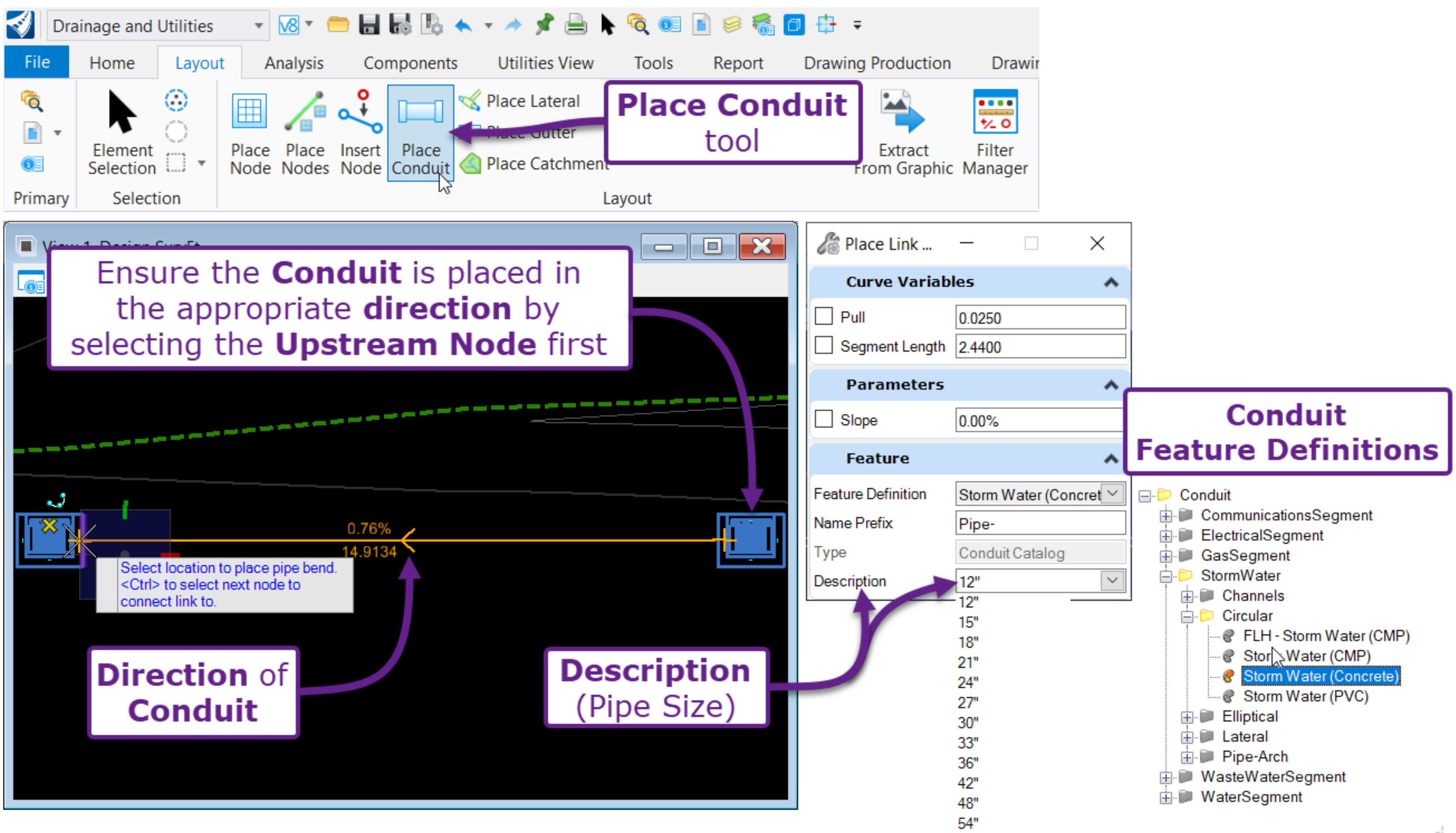
25F.3 Place Conduit and Set Utility Properties

The *Place Conduits* tool is used to create storm sewer pipes in between the Nodes. The placement of Conduits is discussed in *25B.2 Create Conduit (Pipe)*.

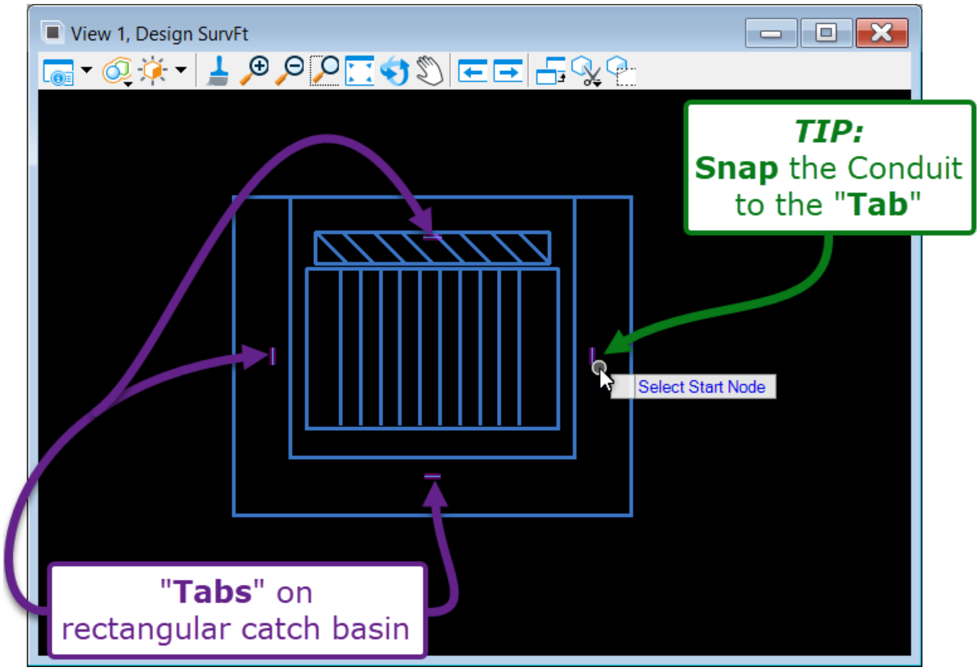
The Conduit Feature Definition determines the pipe material and shape. Typically, circular concrete pipes are used in catch basin networks. Use Feature Definitions found under the **StormWater** → **Circular** drop-down.

In the Dialogue Box, the Description determines the initial diameter of the pipe. The diameter can be changed in the Utility Properties after placement. Alternatively, if a Design Scenario is run, the pipe diameter will automatically adjust in size to optimally accommodate the storm event.

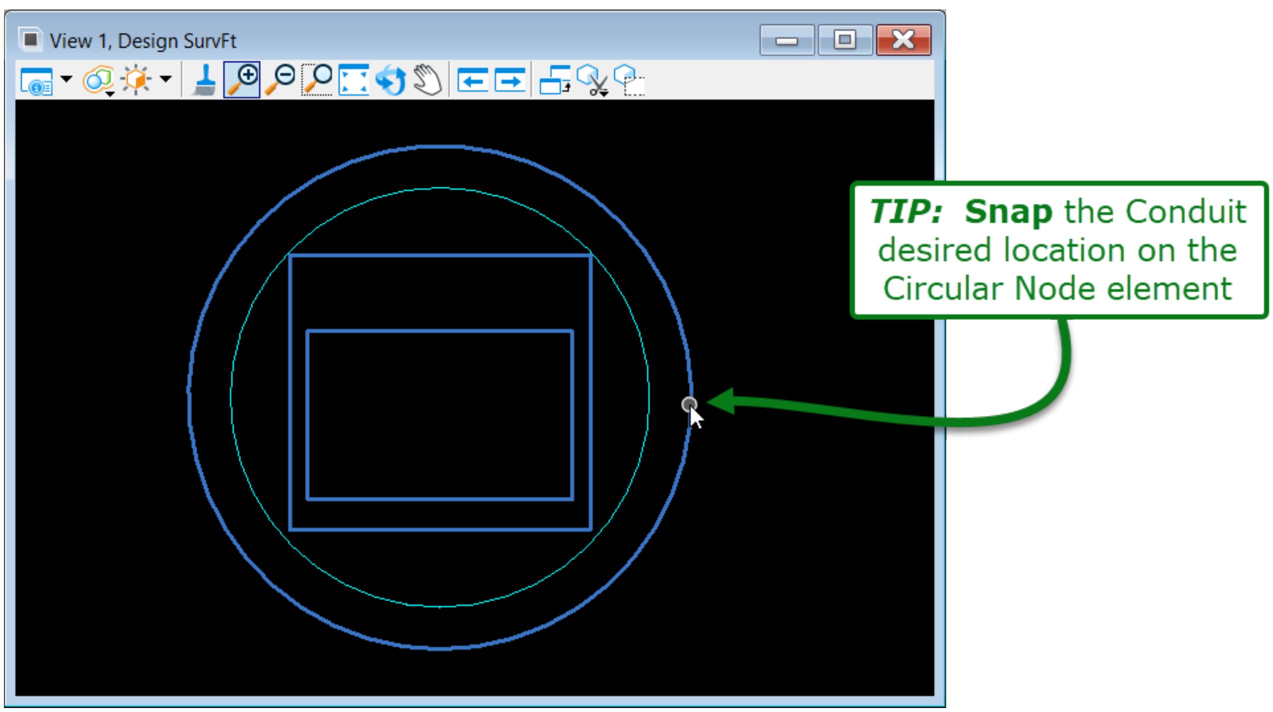
NOTE: The Conduit element must be aligned upstream to downstream. Select the upstream Node first to ensure the Conduit is aligned in the appropriate direction.



TIP: When selecting a Node for Conduit placement, the location that is clicked on determines the location and angle which the Conduit connects to the catch basin. Most rectangular Nodes have a "tab" on each side. When placing the Conduit, snap to the "tab" when selecting the Node.



When connecting a Conduit to a circular catch basin Node, click on the approximate location, along the perimeter of the Node, which the pipe should connect at.

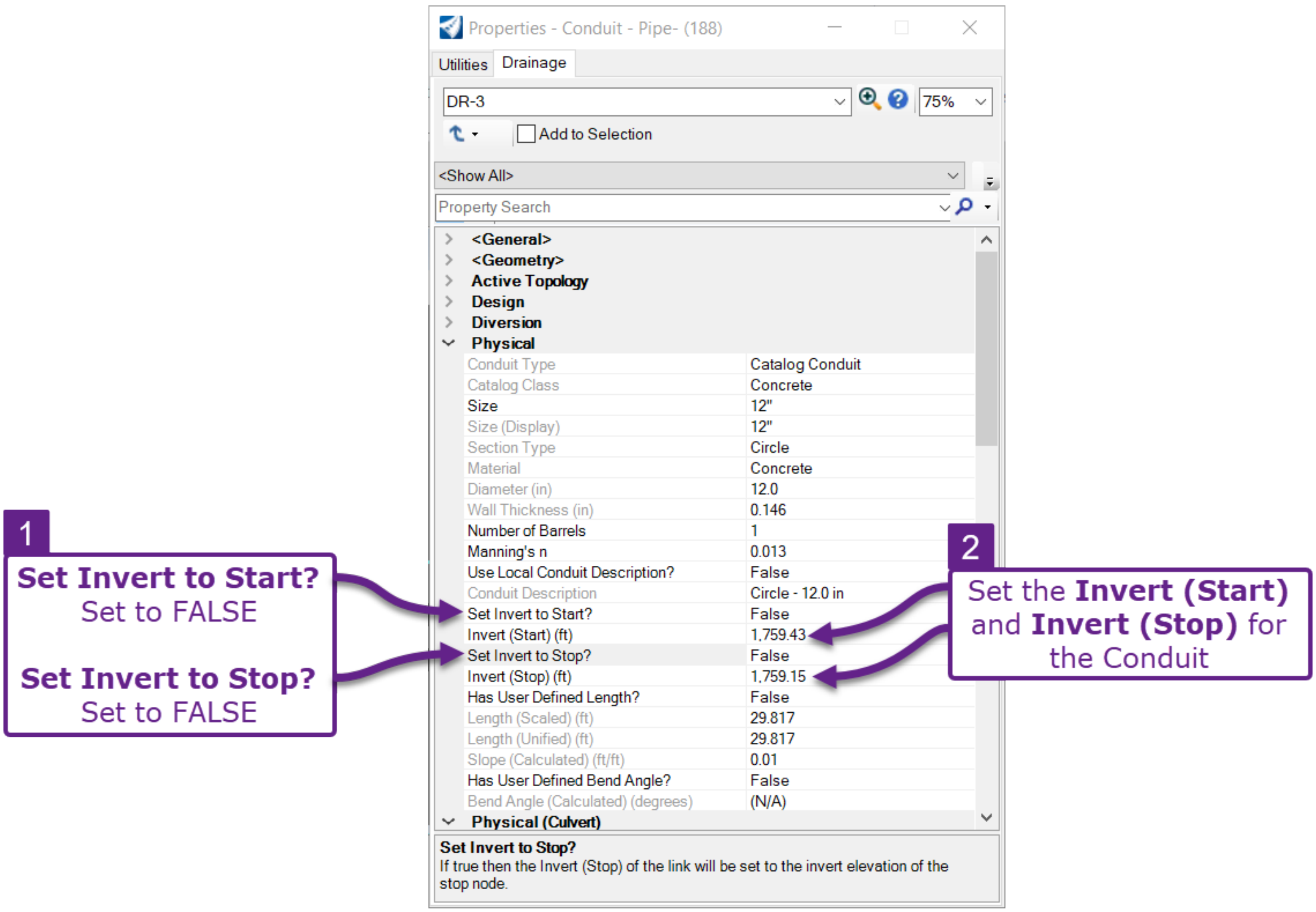


25F.3.a Set the Conduit Invert Elevations in the Utility Properties

When initially placed, the Conduit invert elevations are automatically set to match the Node invert elevation.

The Conduit invert elevations may be manually changed in the Utility Properties. To do so, the **Set Invert to Start** and **Set Invert to Stop** properties must be changed to FALSE. If the Set Invert to Start/Stop properties are set to TRUE, then the Node invert elevation will automatically set the Conduit invert elevation.

Conduit invert properties are found under the **Physical** drop-down.

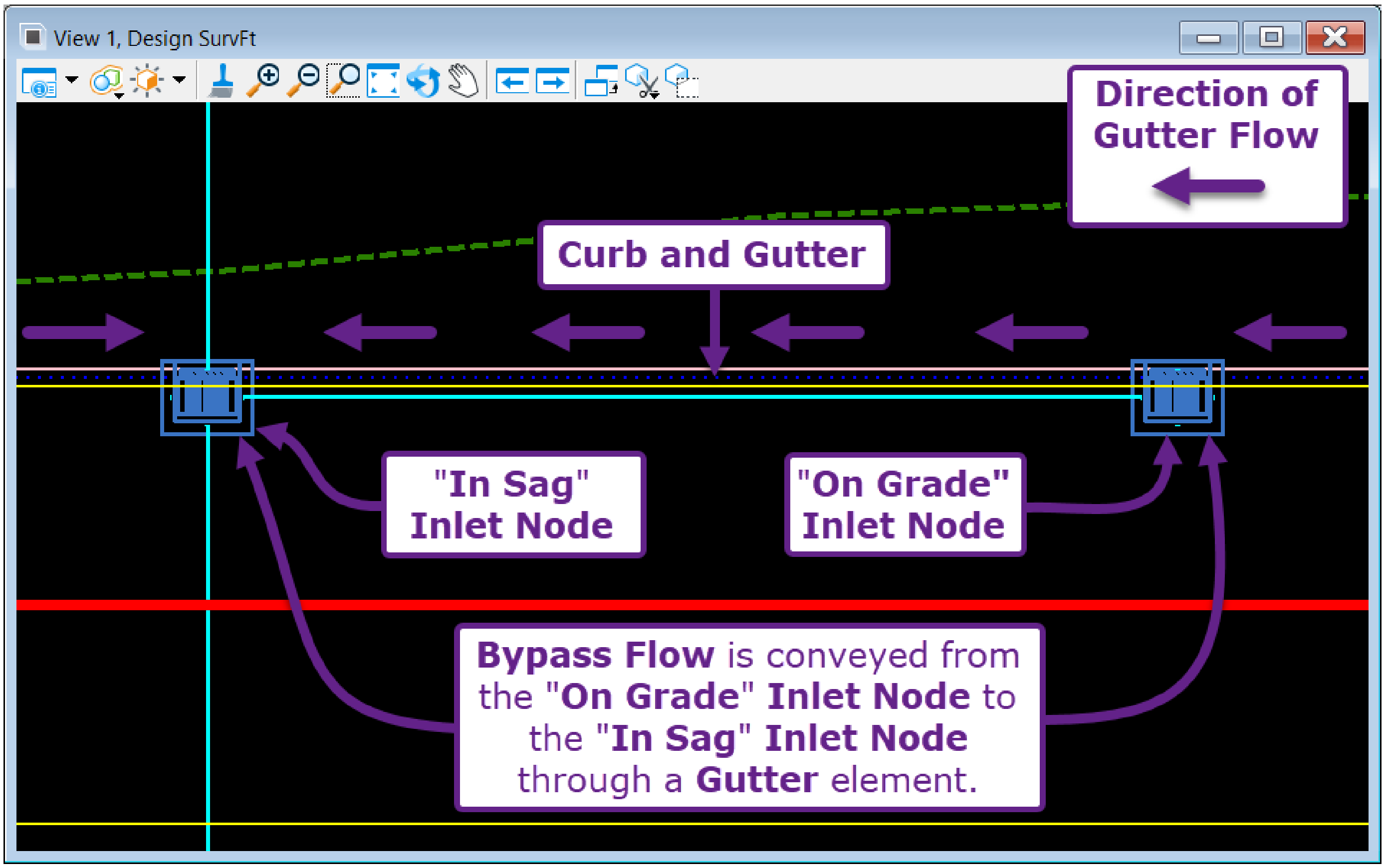


TIP: Changing the invert elevation for a catch basin Node is shown in *25F.1.a Inlet Catch Basin Node Utility Properties*. The invert elevation for the Conduit must be greater or equal to the Node invert elevation.

25F.4 Place Gutters between Inlet Nodes

If an inlet Node CANNOT accommodate the full runoff flow, a Gutter element can be placed to bypass a portion of the flow to a downstream inlet Node.

In this example, a Gutter element must be placed between the "On Grade" inlet and the "In Sag" inlet. Both inlet Nodes are positioned on the same curb and gutter line. In theory, any flow that is NOT captured by the "On Grade" inlet is conveyed through the gutter to the "In Sag" inlet. Since the "In Sag" inlet is located in a vertical low point, water will pond and surcharge above its rim elevation.



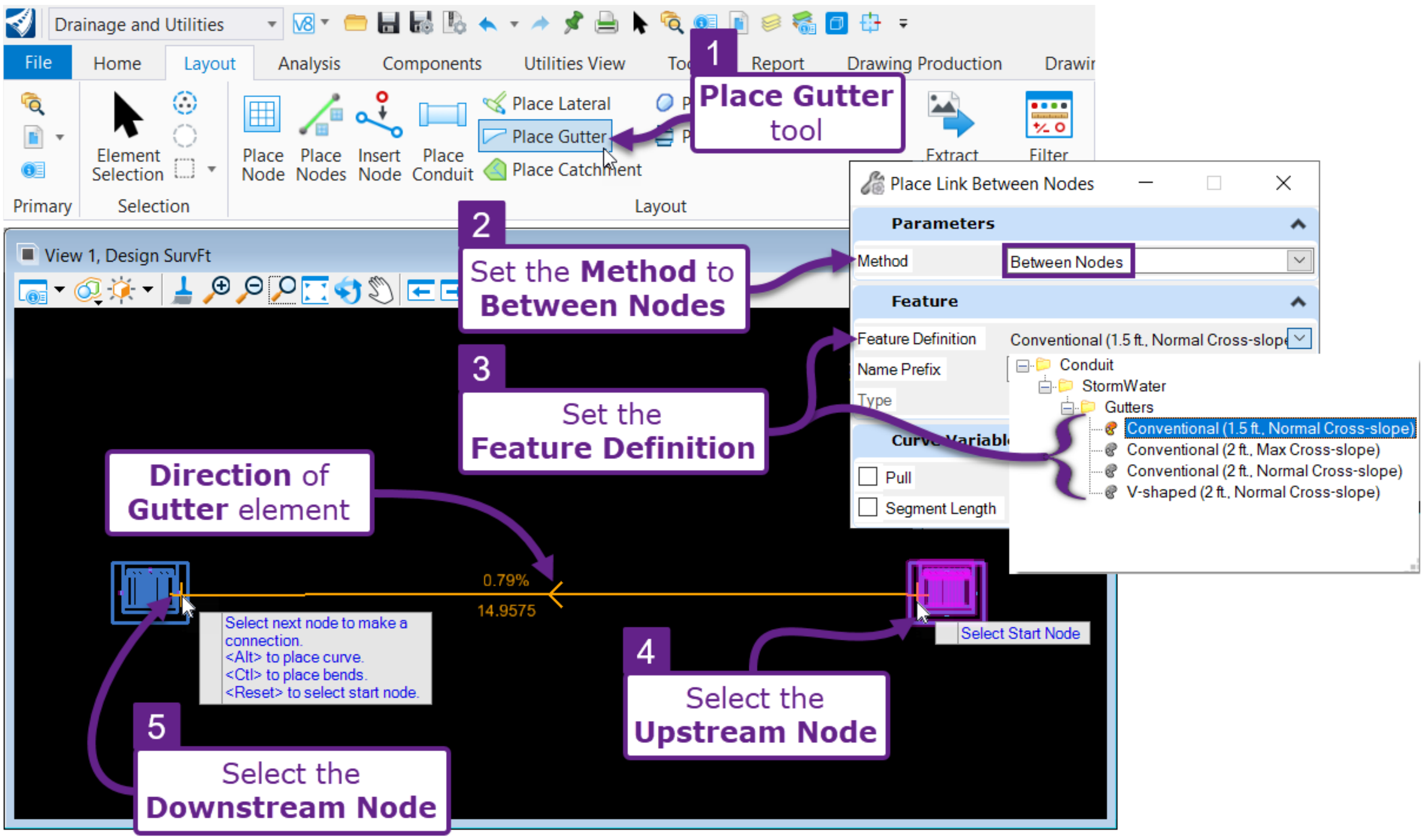
The *Place Gutter* tool creates a Gutter element between two Nodes on the same curb and gutter line.

The Gutter **Feature Definition** represents the initial geometry and shape for the gutter. However, after placement, the gutter geometry can be changed in the Utility Properties menu.

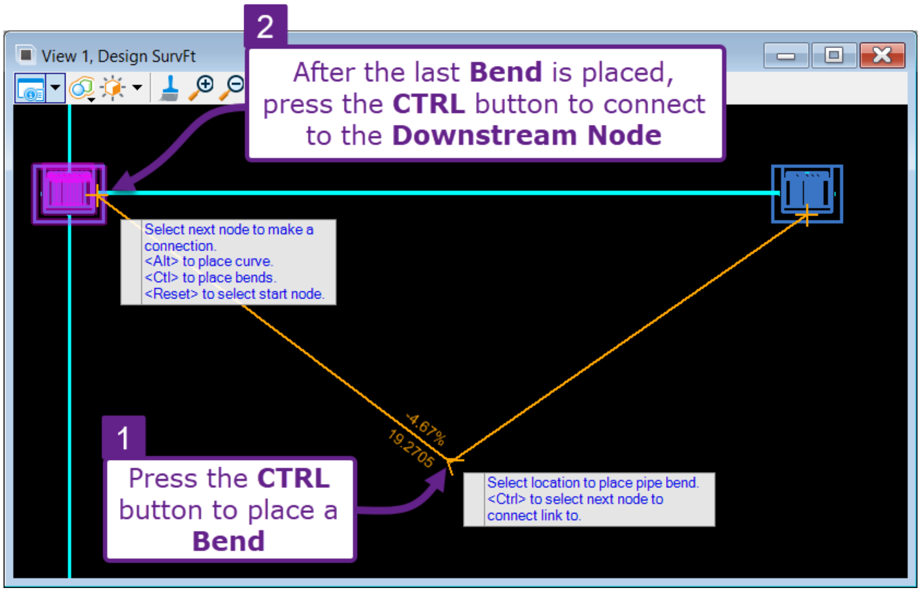
The **Method** determines if the Gutter element is drawn automatically or manually:

> If the **Between Nodes** method is selected, then the Upstream and Downstream Nodes are manually selected. If Bends or Curves are NOT placed, then the Gutter element will be drawn as a straight line between the selected Nodes. Bends or Curves need to be manually placed if the gutter line wraps around a radius or contains a deflection point between the selected Nodes.
>
> The **Trace Slope** method uses the Finished Grade Terrain Model to automatically draw the Gutter element.

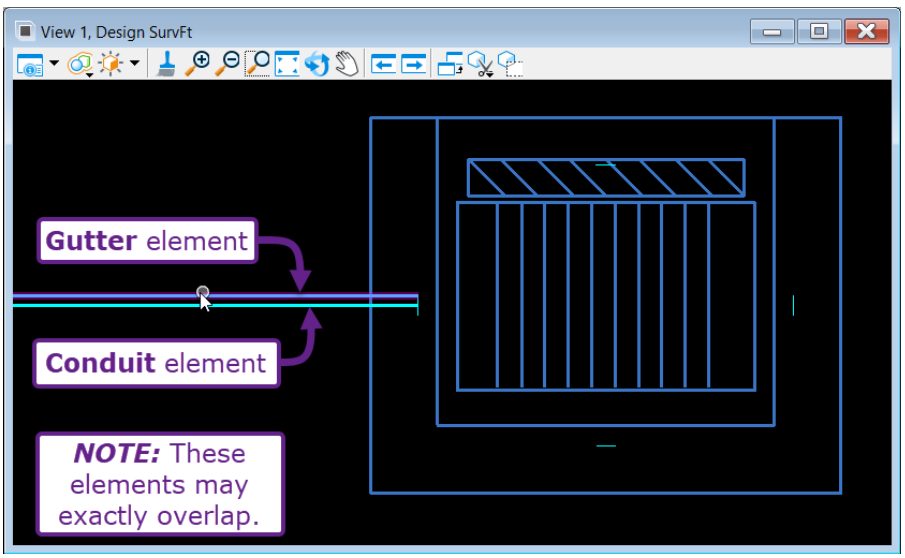
WARNING: The **Trace Slope** method is currently unsupported by the FLH WorkSpace. Use the **Between Nodes** method when creating Gutter elements.



1	From the Ribbon, select the *Place Gutter* tool: [***Drainage and Utilities*** → ***Layout*** → ***Layout***].
2	In the *Dialogue Box*, set the **Method** to **Between Nodes**.
3	In the *Dialogue Box*, set the **Feature Definition** to match the gutter size and shape.
4	*Prompt: Select Start Node* – Select the **Upstream Node**. ***WARNING:*** If the **Downstream Node** is selected first, then the Gutter will be aligned in the wrong direction, which results in an error when the hydraulic analysis is run.
5	*Prompt: Select next node to make a connection. <Alt> to place curve. <Ctrl> to place bends. <Reset> to select start node.* Select the **Downstream Node** to create a straight Gutter element. ***TIP:*** Before selecting the **Downstream Node**, place a Bend or Curve in the gutter by pressing CTRL or ALT. When the CTRL or ALT key is pressed, the next location clicked on will determine the placement of the Bend or Curve. After placing all desired Bends and/or Curves, press the CTRL or ALT key again to the select the Downstream Node. ***NOTE:*** A single Curve can be fitted between two Nodes. However, a combination of straight segments and curves is NOT possible for a single Gutter element. When wrapping gutters around a radius, use the Bend option to place a plethora of straight segments that generally fit to the radius.



After creation, the Gutter element appears as a narrow line. If their alignments coincided, it may be difficult to distinguish between the Conduit and Gutter elements. Zoom in on the Gutter element to locate it.

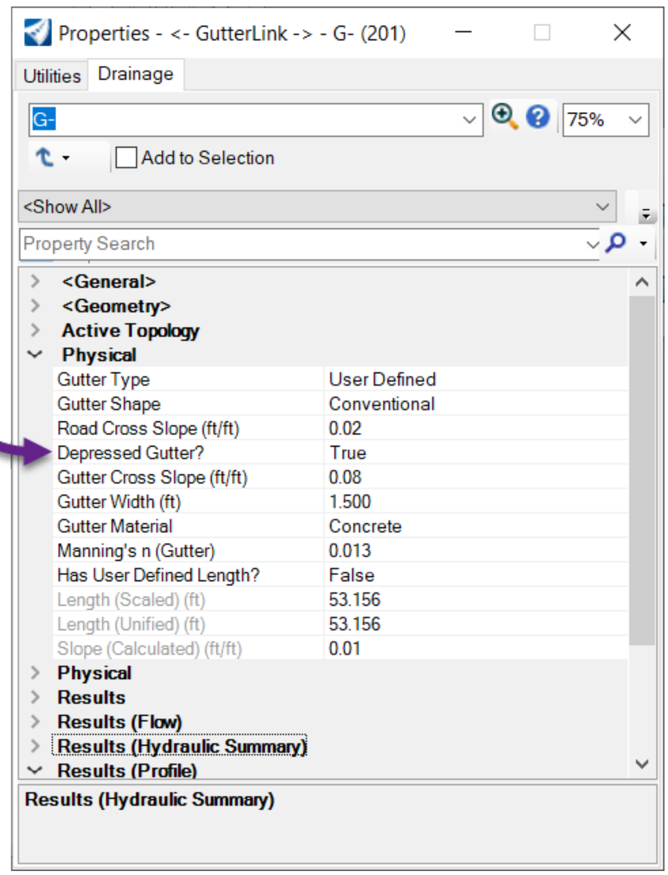


After creation, open the Utility Properties to modify the Gutter geometry under the **Physical** drop-down.

Modify or verify the following settings:

- **Road Cross Slope (ft/ft) –** Determines the transverse slope of the adjacent roadway.
- **Depressed Gutter? –** If set to FALSE, then the gutter and adjacent roadway are placed at the same slope (as determined by the Road Cross Slope (ft/ft) property). If set to TRUE, then the Gutter Width is placed at a steeper slope than the adjacent roadway.
- **Gutter Cross Slope (ft/ft) –** Determines the transverse slope of the gutter section. This setting is only shown if "Depressed Gutters?" is set to TRUE.
- **Gutter Width (ft) –** Determines the width of the gutter pan (i.e., from flowline to edge of asphalt). This setting is only shown if "Depressed Gutters?" is set to TRUE.
- **Manning's n (Gutter) –** Determines the roughness value for the gutter.

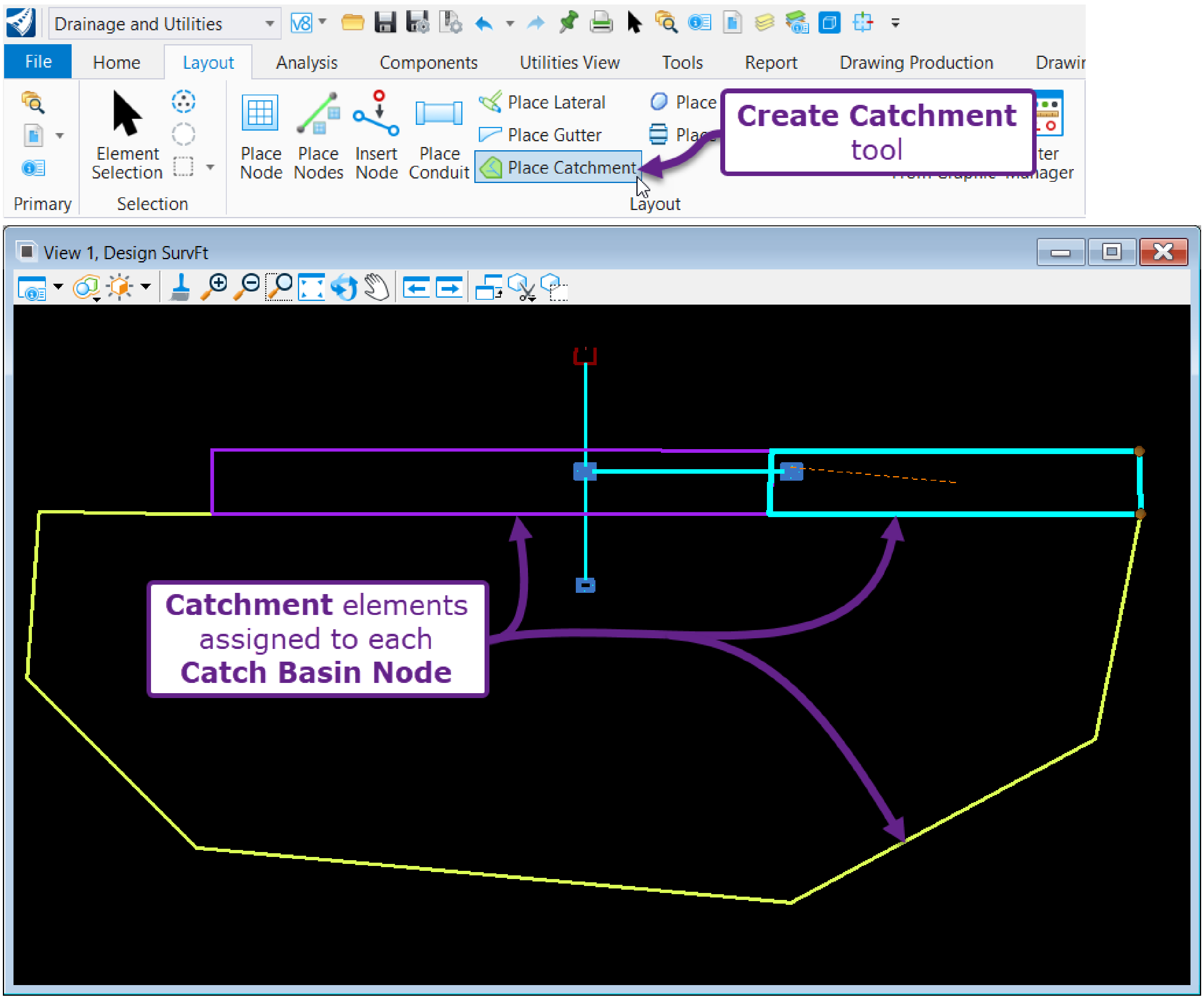
If the **Depressed Gutter?** option is changed to **False**, then the **Gutter Width** and **Gutter Cross Slope** options disappear.



25F.5 Create Catchments for each Inlet Catch Basin Node

To generate runoff flow rates, each catch basin Node should have a Catchment (drainage area) assigned to it. The creation of Catchment elements is discussed in *25B.4 Create the Catchment (Drainage Area)*.

ALTERNATIVELY: If the flow into a catch basin Node is known, then the Node can be assigned flow through the "Flow (Known) (cfs)" property (found in the Node Utility Properties under the Flows drop-down). If the flow is known, then a Node does NOT require a corresponding Catchment element. See *25B.3.a Set a Known Flow to the Inlet Node*.



25F.6 Program the Design Scenario

This section overviews the process for programming a Design Scenario.

> ***NOTE:*** If the desire is to run an Analysis Scenario, then proceed to *25C – Create Storm Data and Scenarios for Catchments* and *25D – Run an Analysis Scenario with the Compute Center tool*.

Every Node, Conduit, and Inlet has a set of Design Constraints properties that can be toggled ON/OFF or adjusted to calibrate the automatic design. There are many components of the storm network that are eligible for automatic design.

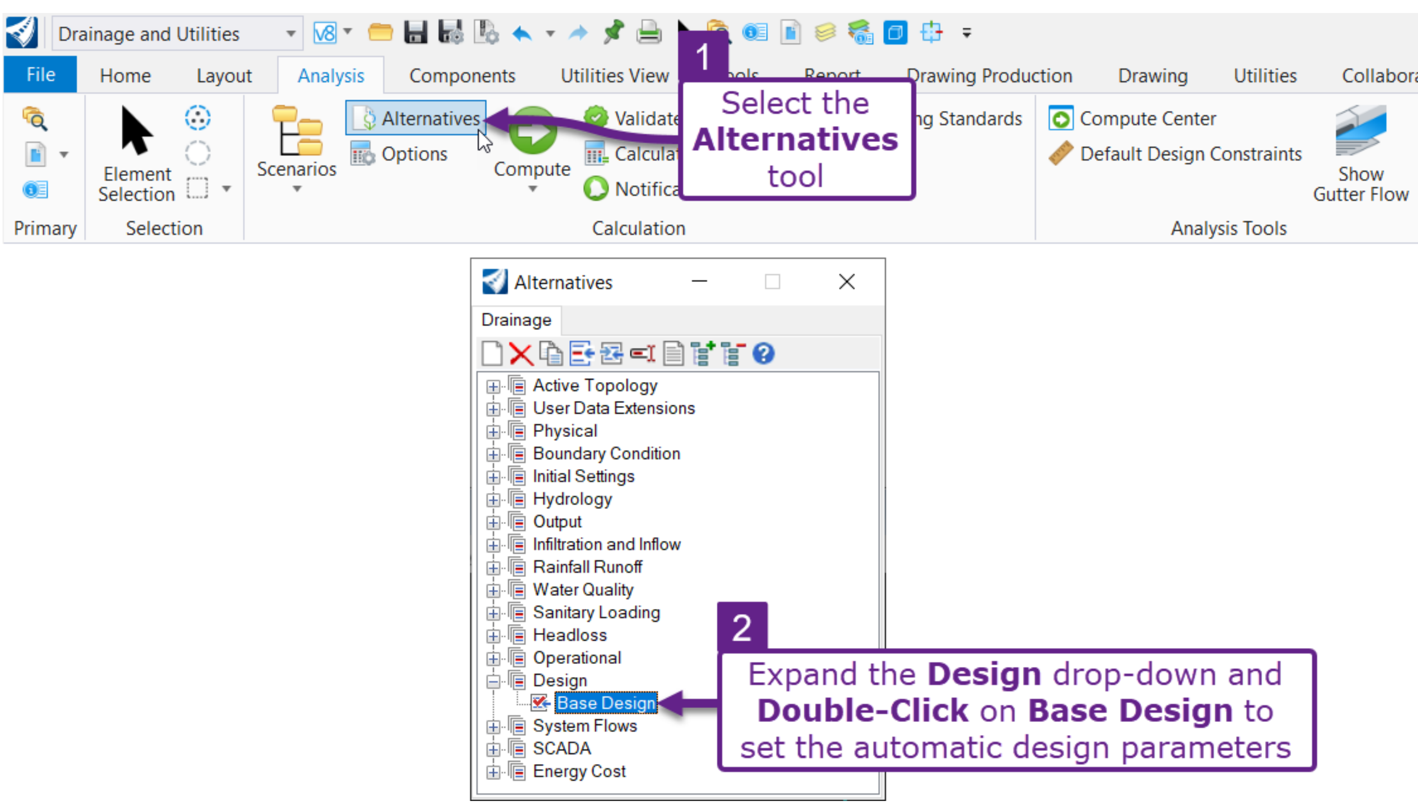
Some examples of automatic design that can be performed include:

- The grate opening dimensions of an inlet can be automatically sized to accommodate the flow.
- The invert elevation of a catch basin Node can be altered to accommodate a larger pipe or achieve minimum slope/velocity on a pipe.
- The size of a pipe can be increased or decreased to optimally accommodate the design storm. Similarly, the maximum percentage of flow through the pipe can be specified for redundant capacity (i.e., flow through the pipe should never exceed 75% of the pipe capacity).

BEST PRACTICE: There is a wide variety of options for automatic design. However, the User is ultimately responsible for producing an acceptable design. It is recommended that the automatic design functionality is limited to the resizing of pipes to achieve capacity. Automatic design of pipe inverts and grate opening dimensions is discouraged.

Programming the Design Scenario is accomplished through the *Alternatives* tool. The "Base Design" alternative must be customized to set the parameters for the automatic design. The *Alternatives* tool is found in the Ribbon in the following location:

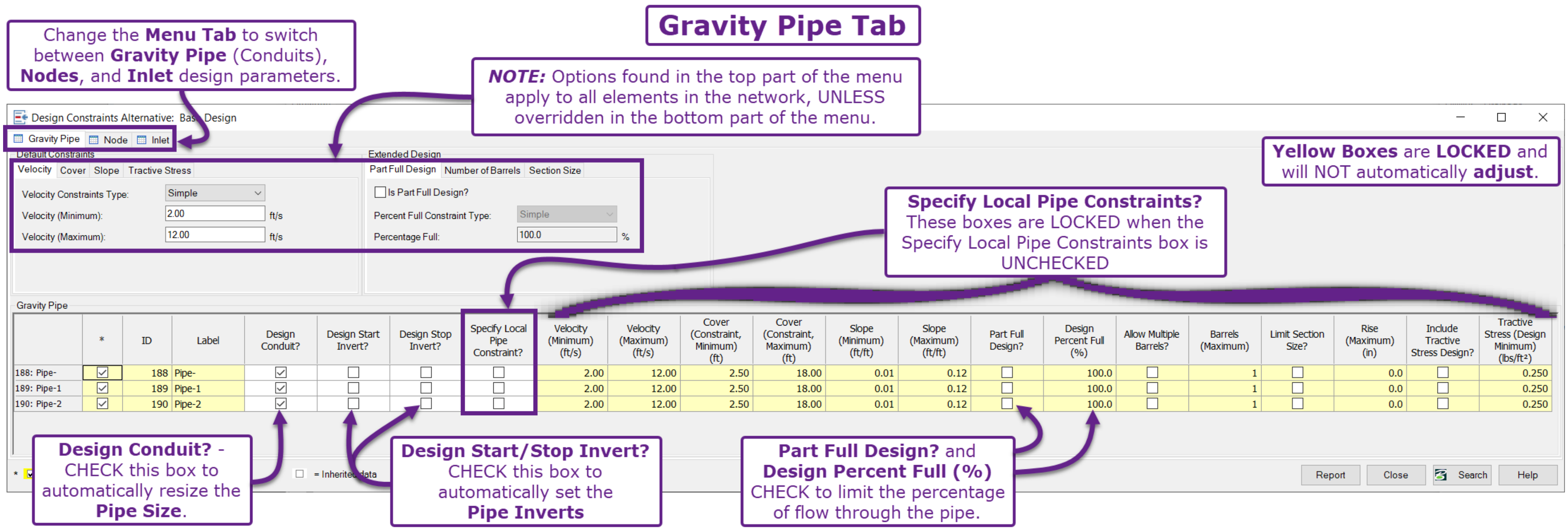
Drainage and Utilities workflow → **Analysis** tab → **Calculation** group



After double clicking on the "Base Design" option in the Alternatives menu, the Design Constraints menu is shown. All elements used in the stormwater network are listed in this menu. The tabs in the upper-left corner of the menu organize the stormwater network by element type: Gravity Pipe (Conduit elements), Node, and Inlet.

Gravity Pipe Tab: The graphic below shows the menu when the Gravity Pipe tab is selected. All Conduit elements in the stormwater network are listed at the bottom of the menu.

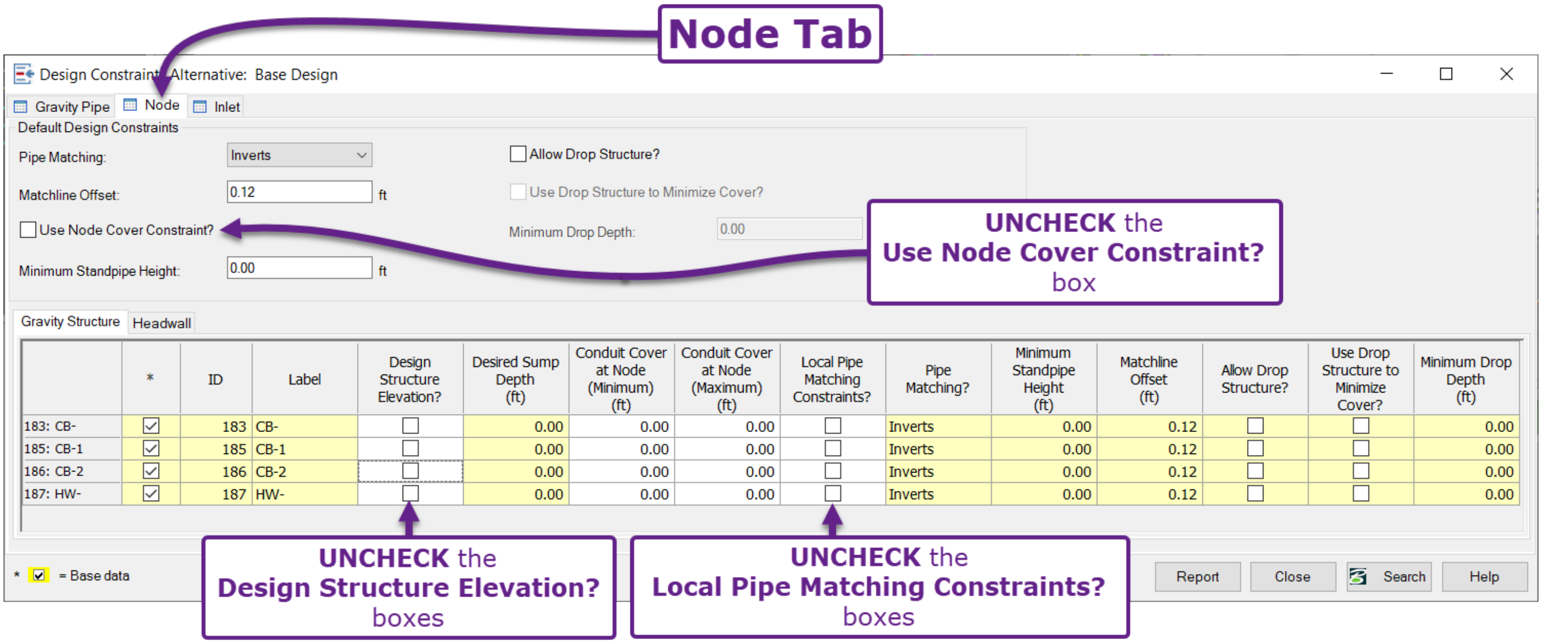
BEST PRACTICE: CHECK the **Design Conduit?** box for all Conduits that are to be automatically resized. If UNCHECKED, then the Conduit will NOT be automatically resized when the Design Scenario is run. UNCHECK the **Design Start/Stop Invert?** boxes. If these boxes are checked, then the inverts of the conduits will be automatically adjusted to achieve minimum/maximum slope and minimum velocity. However, the inverts of a Conduit should typically be manually designed by the User. UNCHECK the **Specify Local Pipe Constraints?** box. If checked, the Conduits will adjust in size or invert elevation to meet velocity, cover, slope, and other constraints.



	*	ID	Label	Design Conduit?	Design Start Invert?	Design Stop Invert?	Specify Local Pipe Constraint?	Velocity (Minimum) (ft/s)	Velocity (Maximum) (ft/s)	Cover (Constraint, Minimum) (ft)	Cover (Constraint, Maximum) (ft)	Slope (Minimum) (ft/ft)	Slope (Maximum) (ft/ft)	Part Full Design?	Design Percent Full (%)	Allow Multiple Barrels?	Barrels (Maximum)	Limit Section Size?	Rise (Maximum) (in)	Include Tractive Stress Design?	Tractive Stress (Design Minimum) (lbs/ft²)
188: Pipe-	☑	188	Pipe-	☑	☐	☐	☐	2.00	12.00	2.50	18.00	0.01	0.12	☐	100.0	☐	1	☐	0.0	☐	0.250
189: Pipe-1	☑	189	Pipe-1	☑	☐	☐	☐	2.00	12.00	2.50	18.00	0.01	0.12	☐	100.0	☐	1	☐	0.0	☐	0.250
190: Pipe-2	☑	190	Pipe-2	☑	☐	☐	☐	2.00	12.00	2.50	18.00	0.01	0.12	☐	100.0	☐	1	☐	0.0	☐	0.250

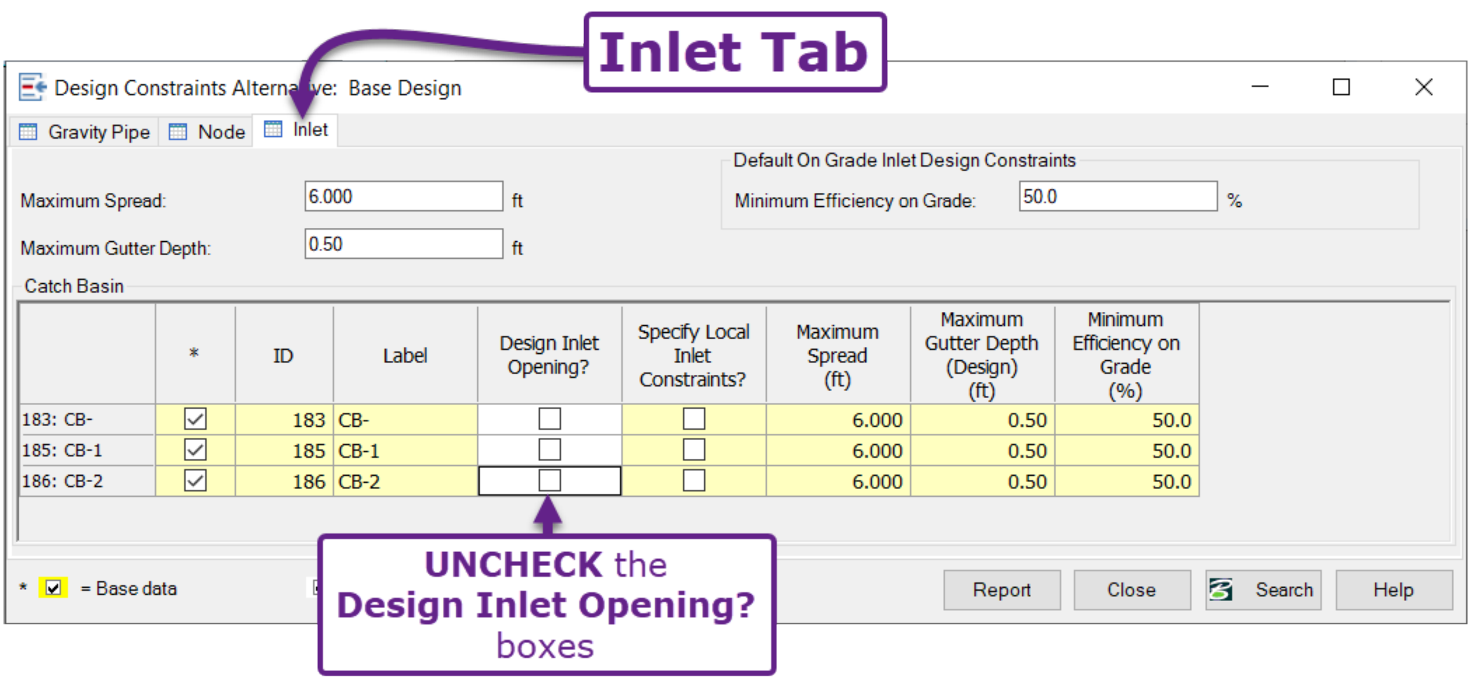
Node Tab: The Node tab is used to automatically design the catch basin configuration of the Node. These parameters can be used to automatically set the invert of the catch basin structure and adjust Conduit invert elevations. However, it is recommended that invert elevations are manually set and analyzed by the User.

BEST PRACTICE: UNCHECK the **Design Structure Elevation?** and **Local Pipe Matching Constraints** boxes. When these boxes are UNCHECKED, then the catch basin are NOT automatically designed.



Inlet Tab: The Inlet tab is used to the automatically resize the opening size of the Inlet portion of the catch basin Node. However, inlet grates are typically bought/provided in standard sizes and should not be automatically resized to a custom configuration.

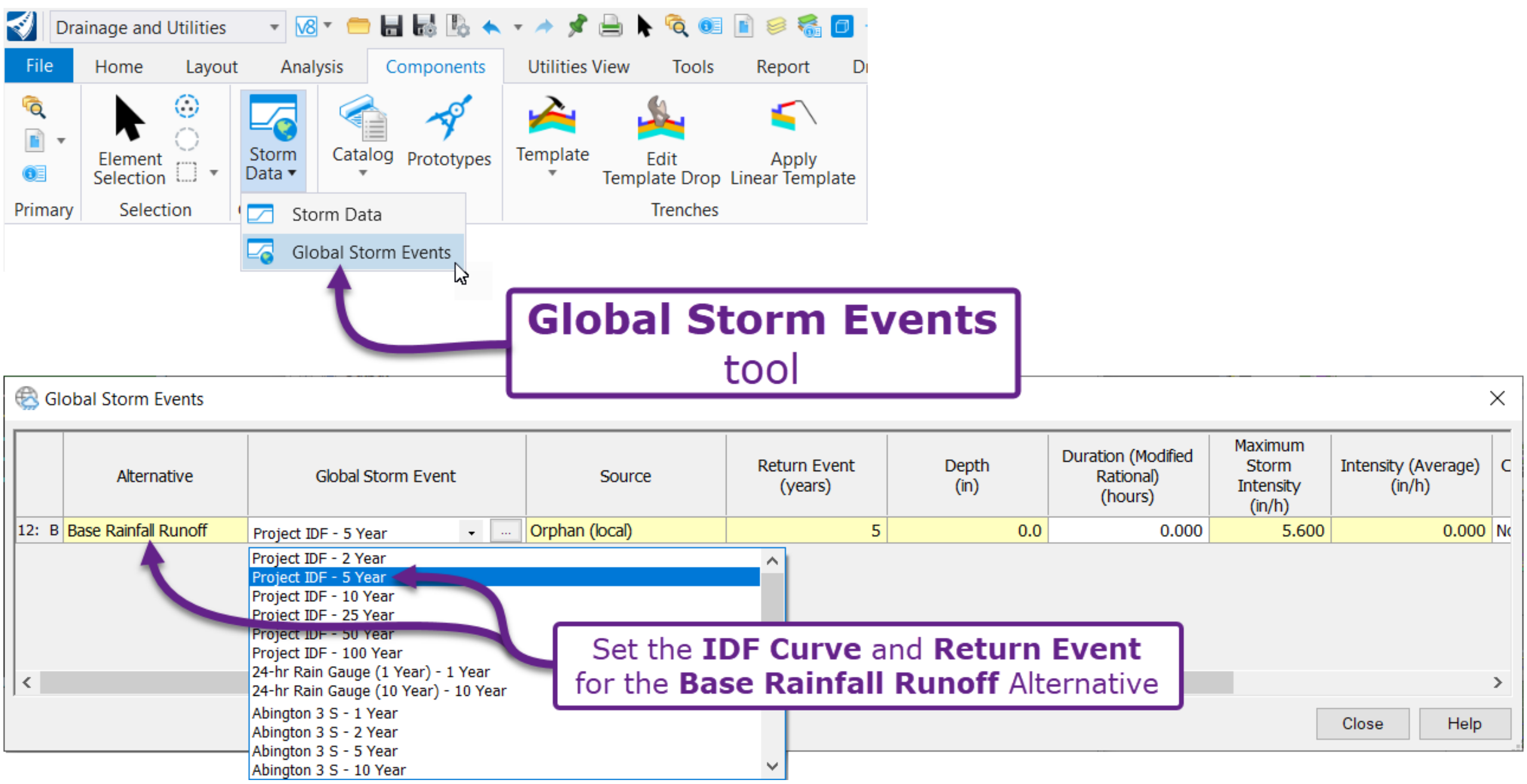
BEST PRACTICE: UNCHECK the **Design Inlet Opening?** boxes. When UNCHECKED, the inlet opening is NOT automatically designed.



25F.7 Import the Storm Data for the Design Scenario

Before running the Design Scenario, the appropriate IDF curve must be imported and the return event must be set:

1	Obtain an IDF curve and set it up in the appropriate CSV format. See *25C.1 Obtain and Setup and IDF Curve File (CSV)*.
2	Use the *Storm Data* tool to import the IDF curve file (CSV) into ORD. See *25C.2 Import the IDF Curve File (CSV) into OpenRoads*.
3	Open the *Global Storm Events* tool. [***Drainage and Utilities*** → ***Components*** → ***Common***]. Set the "Base Rainfall Runoff" Alternative to the appropriate IDF Curve and Return Event.



After the "Base Rainfall Runoff" Alternative is set, the Design Scenario can be run in the *Compute Center* tool menu.

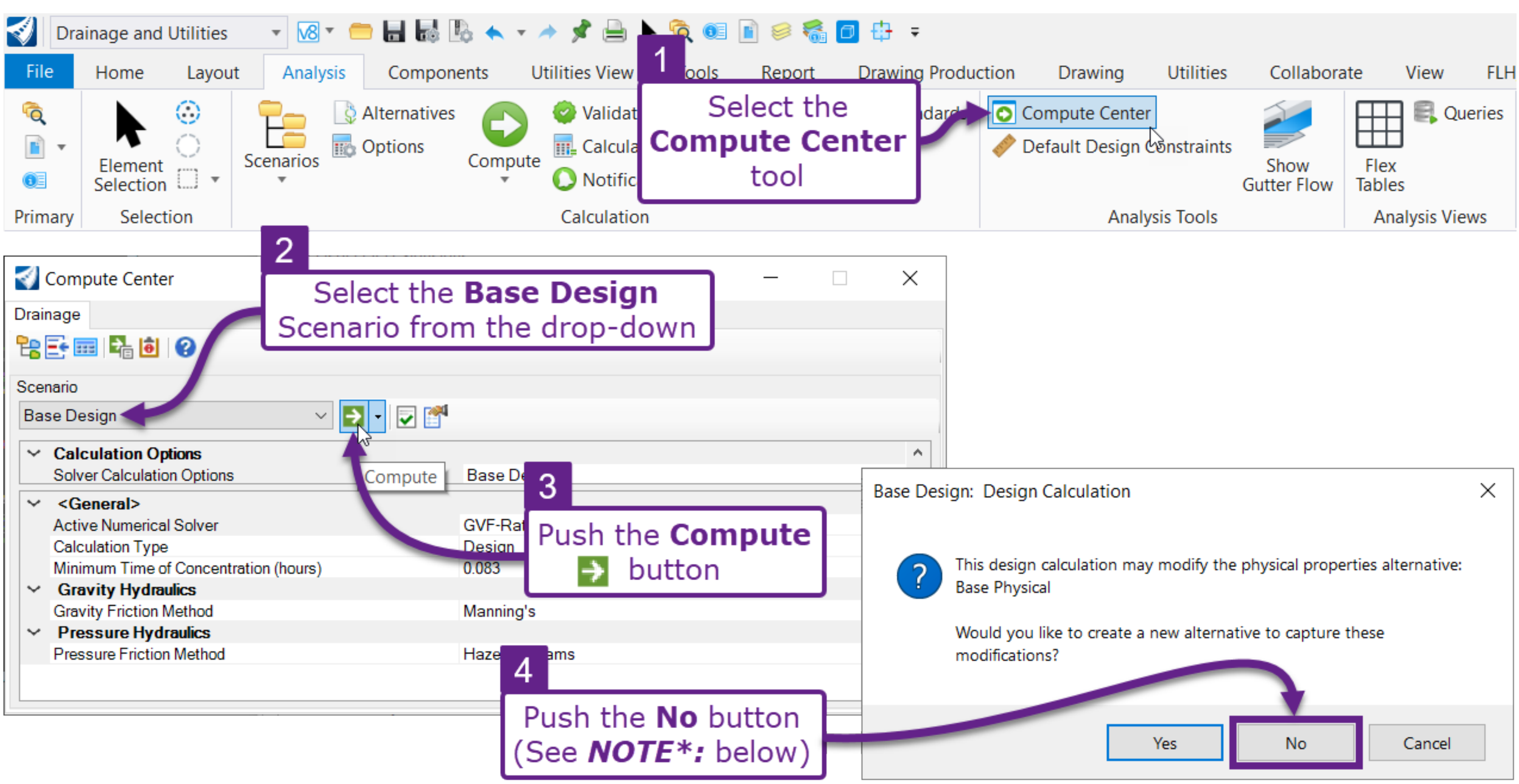
25F.8 Run the Design Scenario in the Compute Center

The *Compute Center* tool is used to select a Scenario and perform the hydraulic calculations.

BEST PRACTICE: Before running the Design Scenario, create a copy or perform a **Save-As** of the ORD File. After the Design Scenario is run, pipes will be automatically resized. It is recommended that the original storm sewer network configuration is archived for future reference.

TIP: After running the Scenario, review the stormwater network with the tools shown in *25E – Results: Creating Reports, Tables, and Profiles*.

1	From the Ribbon, select the *Compute Center* tool: [***Drainage and Utilities*** → ***Analysis*** → ***Analysis Tools***].
2	From the Scenario drop-down, select the **Base Design** option.
3	Push the **Compute** button to run the Design Scenario.
4	When a Design Scenario is run, a message box is shown and asks "Would you like to create a new alternative to capture these modifications?". Push the **No** button. If the Yes button is pushed, then the original storm sewer network is preserved and a duplicate is created. The duplicate network will contain resized and redesigned elements (if necessary). If the No button is pushed, the original storm sewer network is directly resized/redesigned. Push the No button to ensure the ORD File only contains a single storm sewer network. As mentioned in the ***BEST PRACTICE*** above, create a save-as of the ORD File to archive the original configuration.



25G – UTILITY MODELING

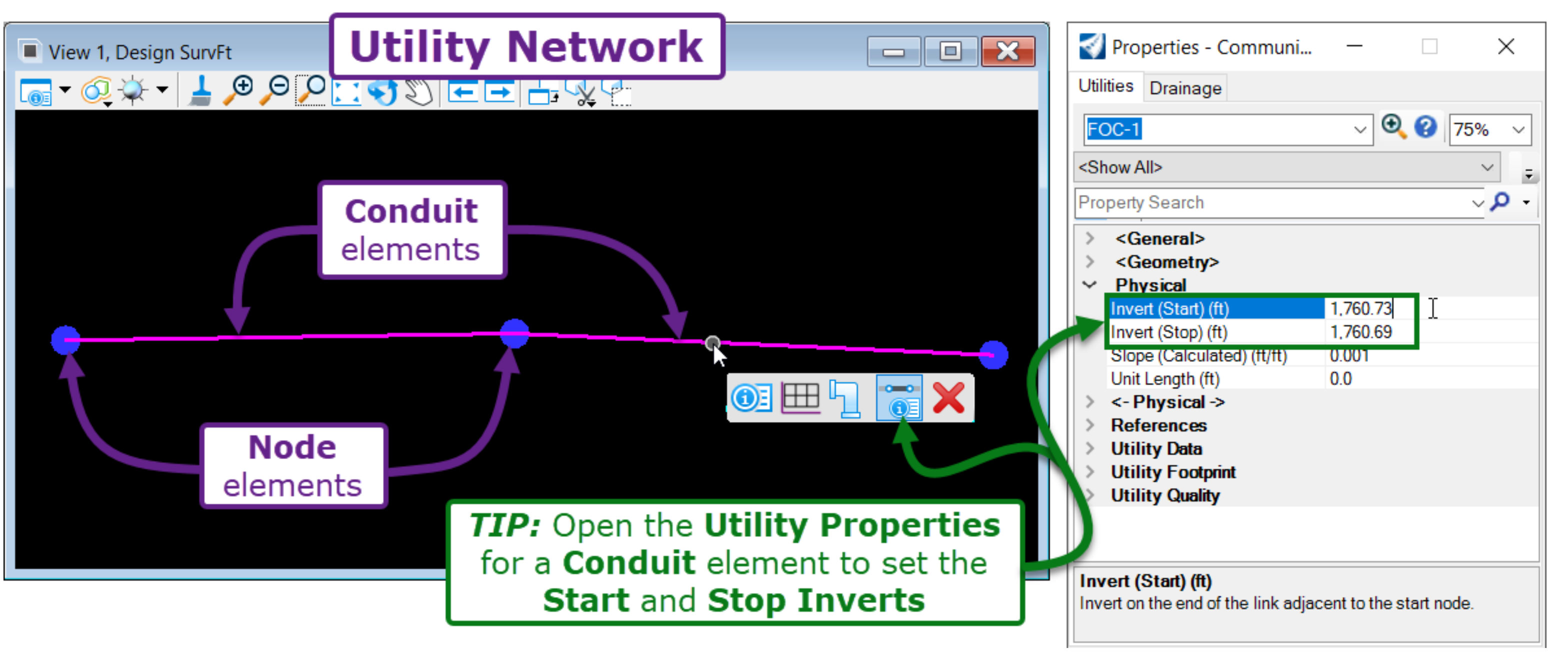
Proposed utility lines, conduits, and facilities can be modeled using the Drainage and Utilities tools. Placing utility lines and facilities is accomplished with Nodes and Conduits; in the same process used for stormwater drainage networks. See *25B.1 Create the Inlet and Outlet Node* and *25B.2 Create the Conduit (Pipe)*.

Modeling utility lines with Drainage and Utilities tools is **beneficial** for the following reasons:

- Utility lines and conduits modeled with Drainage and Utilities tools are displayed in Cross Section and Profile views. Modeling utility lines can help to identify conflicts by scrolling through Cross Sections or along the Profile.
- Utility Nodes are shown in 3D, Cross Section, and Profile views. For example, a fire hydrant, water valve box, electrical transformer, or utility pole can be quickly modeled using a Node element.
- Using the *Project Run* tool, they hydraulic profile of a utility run can be projected into an adjacent or parallel alignments, such as the Centerline of Road Alignment.
- The *Utility Run From Links* tool generates a profile of the utility line that shows Node locations.

Modeling utility lines with Drainage and Utilities tool can be **difficult** for the following reasons:

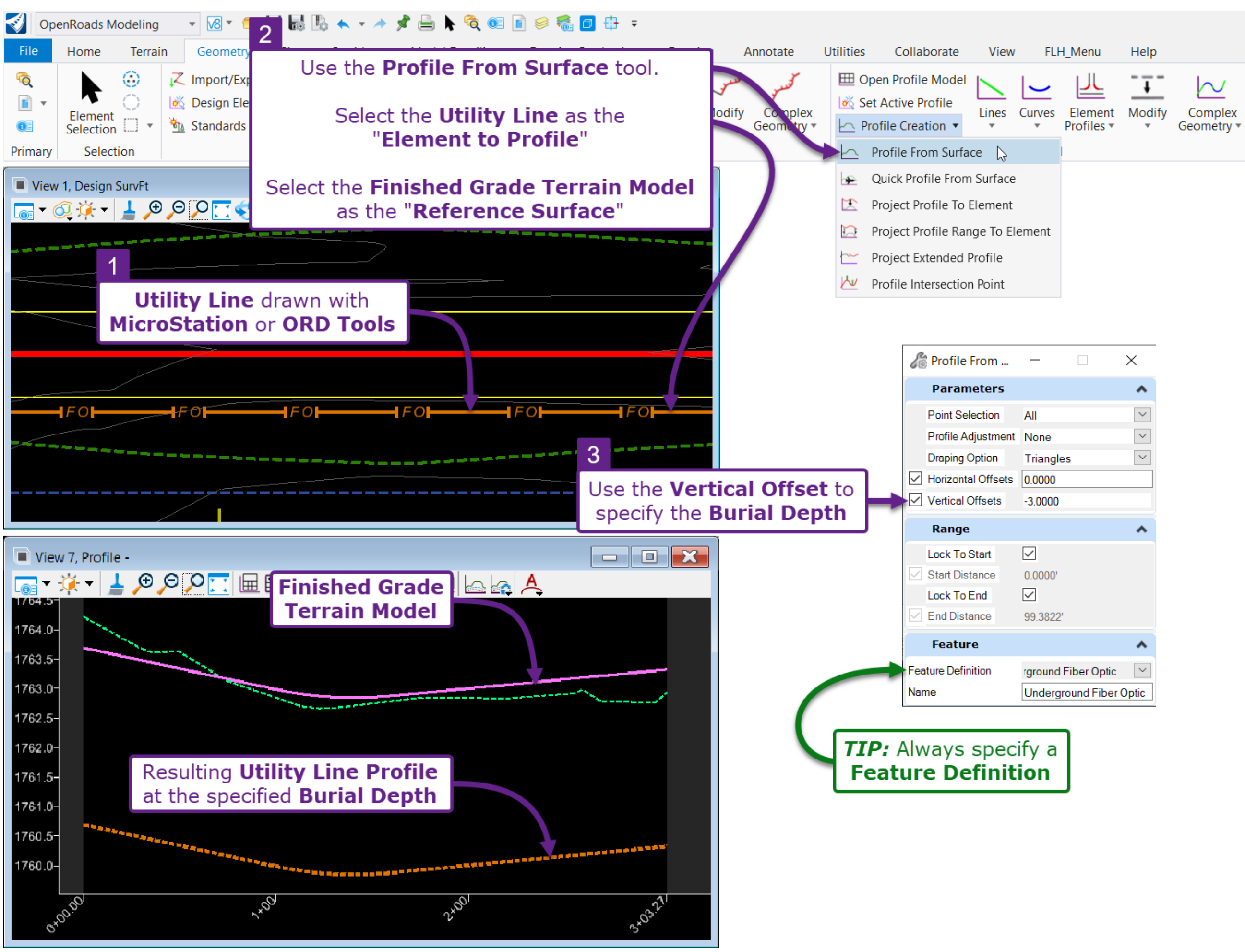
- A utility Conduit element must be placed between two Node elements. Bends (horizontal deflection points) can be placed in the Conduit element. However, a custom profile CANNOT be drawn for a Conduit. The profile of a Conduit is determined by the start and end Node elevation. Vertical deflection points or a custom profile CANNOT be drawn for a utility Conduit. Direct bury cables and flexible conduits are difficult to model because these types of utilities tend to follow the finished grade surface (i.e., buried 3 feet beneath finished grade). An alternative procedure for modeling direct bury cables is shown on the next page.
- Many Node types in the FLH WorkSpace are currently in development and their 2D and 3D graphics do NOT accurately represent the Node. For example, the Electric Transformer and Electric Meter node types are inaccurately depicted as manholes. Similarly, all Communications (Fiber Optic) nodes are inaccurately depicted as manholes.



Alternative to Utility Modeling with Drainage and Utilities tools: Instead of using Drainage and Utilities tools, draw utility lines with MicroStation tools (i.e., *Place SmartLine*) or ORD tools. With conventional drawing tools, a custom Profile can be drawn for the utility line.

Alternatively, a utility line can be drawn to follow the surface of a Proposed Terrain Model at a specified burial depth, using the process shown below:

1	To create the horizontal geometry for a utility line, offset the Edge of Pavement corridor line using the *Single Offset From Element* tool. See *7D.3.a Single Offset From Element*.
2	Use the *Profile From Surface* tool to create a profile that follows the surface of a Terrain Model, at a specified burial depth (Vertical Offset). See *7F.4.a Profile From Surface*. Select the horizontal utility line as the "Element to Profile". Select the Finished Grade Terrain Model as the "Reference Surface". ***TIP:*** To create a Finished Grade Terrain Model from a Corridor, see *Chapter 22 – Proposed Terrain Model Creation*.
3	In the *Dialogue Box* for the *Profile From Surface* tool, set the **Vertical Offset** to the desired burial depth.



A Linear Template can be assigned to the utility line to create a 3D model for display in the 3D Design Model, Cross Section, and Profile views. The process of creating a utility Linear Template is similar to the processes shown in *11D.6.c Create the Culvert Template* and *11D.6.d Apply the Linear Template*.