CONNECTICUT DEPARTMENT OF TRANSPORTATION

DIGITAL DESIGN ENVIRONMENT GUIDE

CONNECT EDITION

Volume 5 – OpenRoads Designer Drainage and Utility Modeling

Published Date: March 12, 2025

Table of Contents

Course Overview

The engineer will need a fundamental understanding of drainage design and the Rational Method that is used throughout OpenRoads Designer Drainage and Utilities.

This platform includes storm drainage design and analysis as well as utilities engineering with capabilities to include conflict detection and non-drainage attribution. OpenRoads .Designer Drainage and Utilities 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 designer's responsibility to adhere to the CTDOT Drainage Manual, CTDOT Hydraulics & Drainage Directives and Standards.

When possible, drainage components have been set-up to follow CTDOT standards and methods. The engineer is responsible to verify drainage items properties.

Skills Taught

- Obtain NOAA Atlas Data for Point Precipitation Frequency (PF) Estimates
- Node and Pipe layout
- Storm data input
- Preform and review computations

Introduction

Drainage and Utilities 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. The data ***is in*** the graphics within the design file (*.dgn file). The design file is part of a "federated" dataset. Design files are segregated by purpose > different engineering object types or scopes are in their own file. The project is assimilated through the dynamic use of reference files.

Use Drainage and Utilities to:

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

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

Tools

This guide will not document each tool that is available in the OpenRoads Designer **Drainage and Utilities** interface. See the Bentley Online Help for commands not detailed in this document.

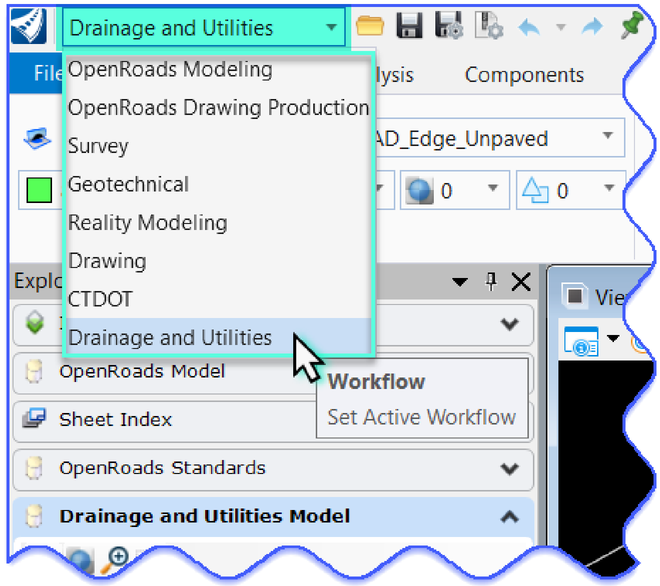


Figure 1 Drainage and Utilities Workflow

Home Tab

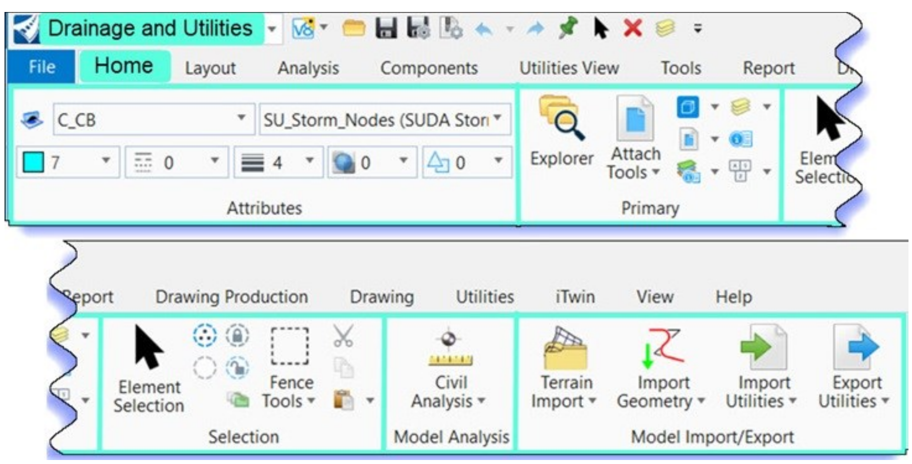


Figure 2 Drainage and Utilities Home Tab

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.

Civil Analysis – gives the user the option to analyze various 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 anyone point within the active terrain. The Trace Slope tool should be used in its separate file and referenced into the drainage file as needed.

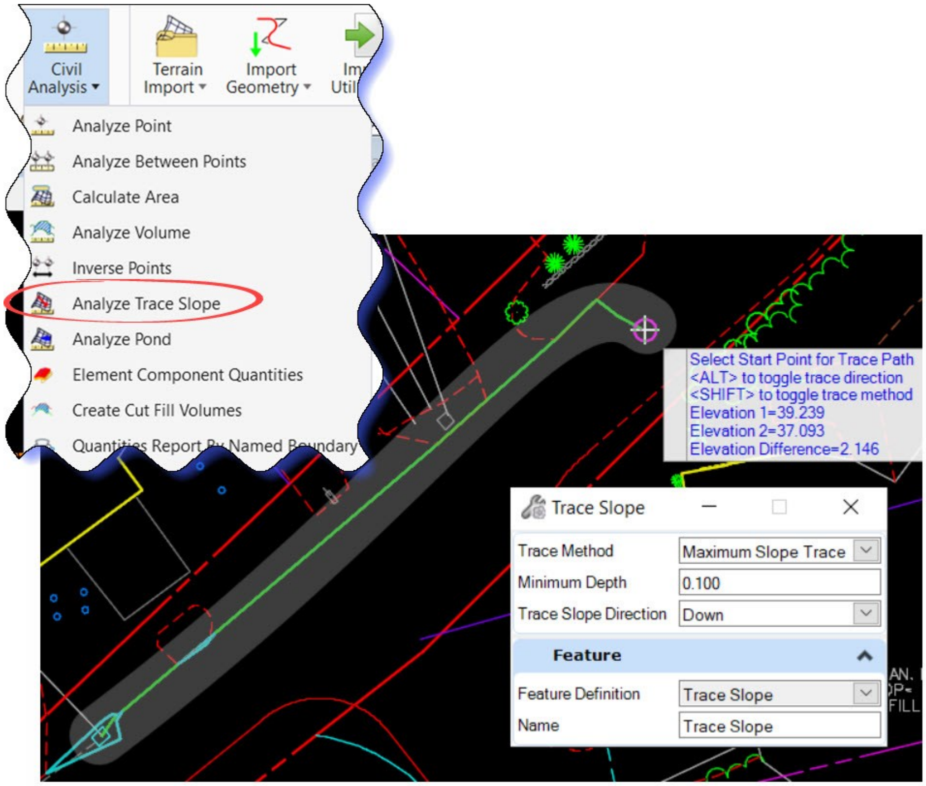


Figure 3 Analyze Trace Slope

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, i.e. InRoads and Export Utilities to various other software, see Bentley Online help.

Export Utilities – export drainage data 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.

Layout Tab

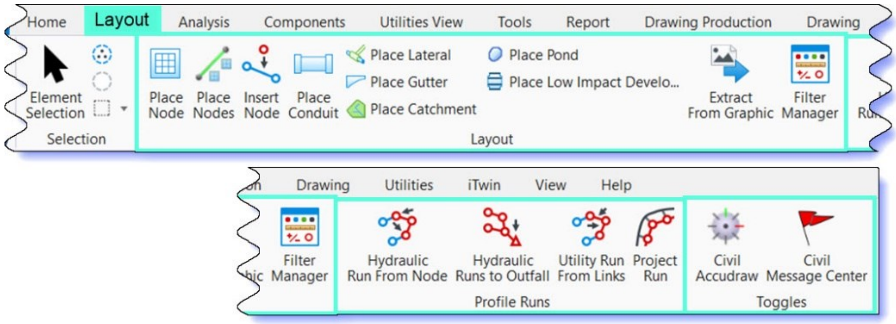


Figure 4 Drainage and Utilities Layout Tab

The **Layout** tab allow the user to place utility structures and connect the structures with conduits. 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.

Profile Runs - 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.

Analysis Tab

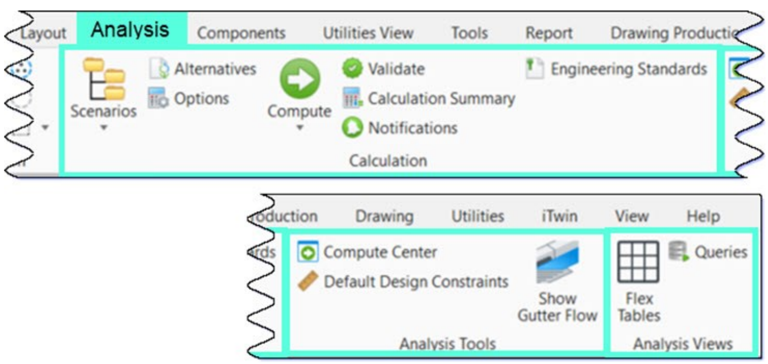


Figure 5 Drainage and Utilities Analysis Tab

The **Analysis** tab set calculation parameters, standards, constraints, and scenarios for drainage items.

Calculation - this is the center for drainage calculations.

Analysis Tools - holds the Compute Center (one tool box for access to calculations), Default Design Constraints are set to CTDOT standards and should be reviewed for project specific constraints (i.e. max. spread, conduit slope), Show Gutter Flow will show the path of any overflows from catch basins.

Analysis Views - houses the flex tables – CTDOT specific flex tables for hydraulic analysis have been established, queries can be used to report on specific item

Components Tab

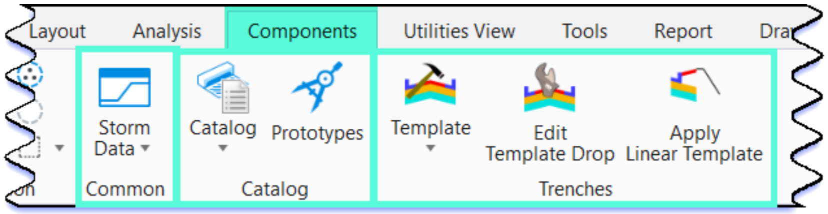


Figure 6 Drainage and Utilities Components Tab

The **Components** tab are used to establish project specific storm data, drainage properties and settings.

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 CTDOT Standards and can be updated to project specific data if needed.

Trenches - access to template tools – edit template drop – apply linear template for trenches used in the design.

Utilities View Tab

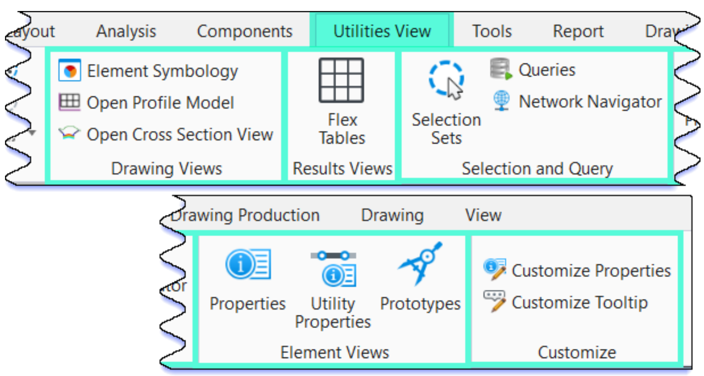


Figure 7 Drainage and Utilities Utilities Viewer Tab

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

Drawing Views - element symbology allows control 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 – CTDOT specific Flex Tables for hydraulic analysis have been established 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 CTDOT 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.

Tools Tab

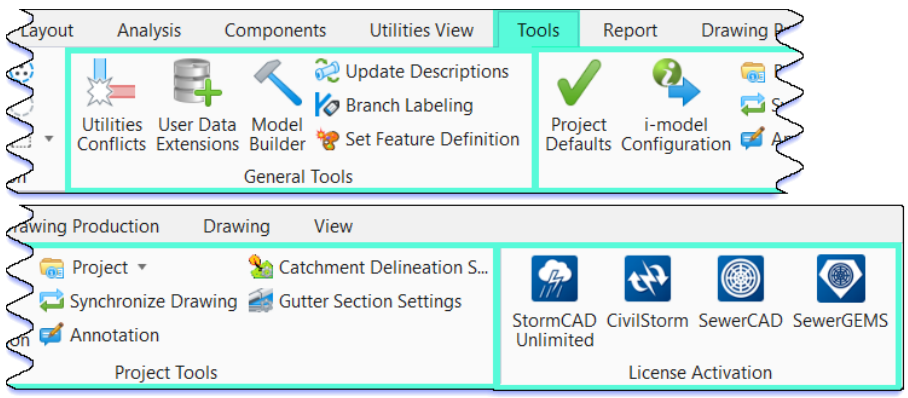


Figure 8 Drainage and Utilities Tools Tab

The **Tools** tab is divided into general tools and project tools.

Tools - utilities conflicts allows detection of physical clashes and clearances between elements; user data extensions allows 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, synchronize drawing and annotation tool to label the drainage items.

License Activation - StormCAD Unlimited.

Report Tab

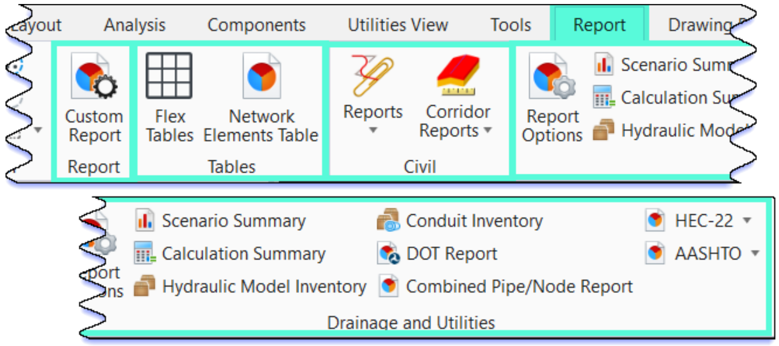


Figure 9 Drainage and Utilities Report Tab

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.

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.

Drawing Production tab

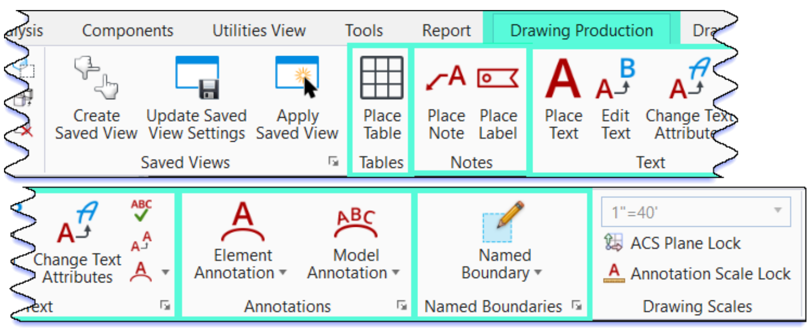


Figure 10 Drainage and Utilities Drawing Production Tab

The **Drawing Production** tab is divided into tables, notes, text, annotation and named boundaries, explanations are found in detail under Volume 13.

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

Workspace Files

Feature Definitions

…CT_Configuration\Organization-BIM_CT_Civil Standards\Dgnlib\Feature Definitions\

- CV_DU_Ex_Features_Levels_ElemTemp.dgnlib
- CV_DU_Pr_Features_Levels_ElemTemp.dgnlib

Cells

…CT_Configuration\Organization-BIM_CT_Civil Standards\Cell\

- CV_DU_Ex_Ends_Walls.cel
- CV_DU_Pr_Ends_Walls.cel
- CV_DU_Utilities.cel
- CV_DU_Ex_CB_MH.cel
- CV_DU_Pr_CB_MH.cel

Inlet Library

…CT_Configuration\Organization-BIM_CT_Civil Standards\Data\

- CV_SubsurfaceUtilitites_Inlets.xml

These files will be updated as needed by AEC Applications to incorporate changes in the CTDOT standards. Should the designer find an item changed or was added (example: new wing wall) please bring this to the attention of AEC Applications.

Exercise 1 - Getting Started

The designer should compile the information and material necessary to complete a drainage design. The most commonly needed items for a drainage design (existing or proposed) are *storm data* and *terrain data* for the project.

1.1 NOAA ATLAS 14 Data

The Department's Hydraulic and Drainage Unit advises the designer to use the *NOAA Atlas 14 data*, which can be accessed through the NOAA ATLAS 14 POINT PRECIPITATION FREQUENCY ESTIMATES webpage in conjunction with the **CTDOT Drainage Manual**.

Follow the direction on the **NOAA Atlas 14** webpage and directions given by **CTDOT Hydraulics & Drainage unit** on how to develop the Point Precipitation Frequency (PF) Estimates, which can be downloaded in CSV-format and opened with Excel. The data in the PF tabular format will be used for Storm Data to create a **User Defined IDF Table**.

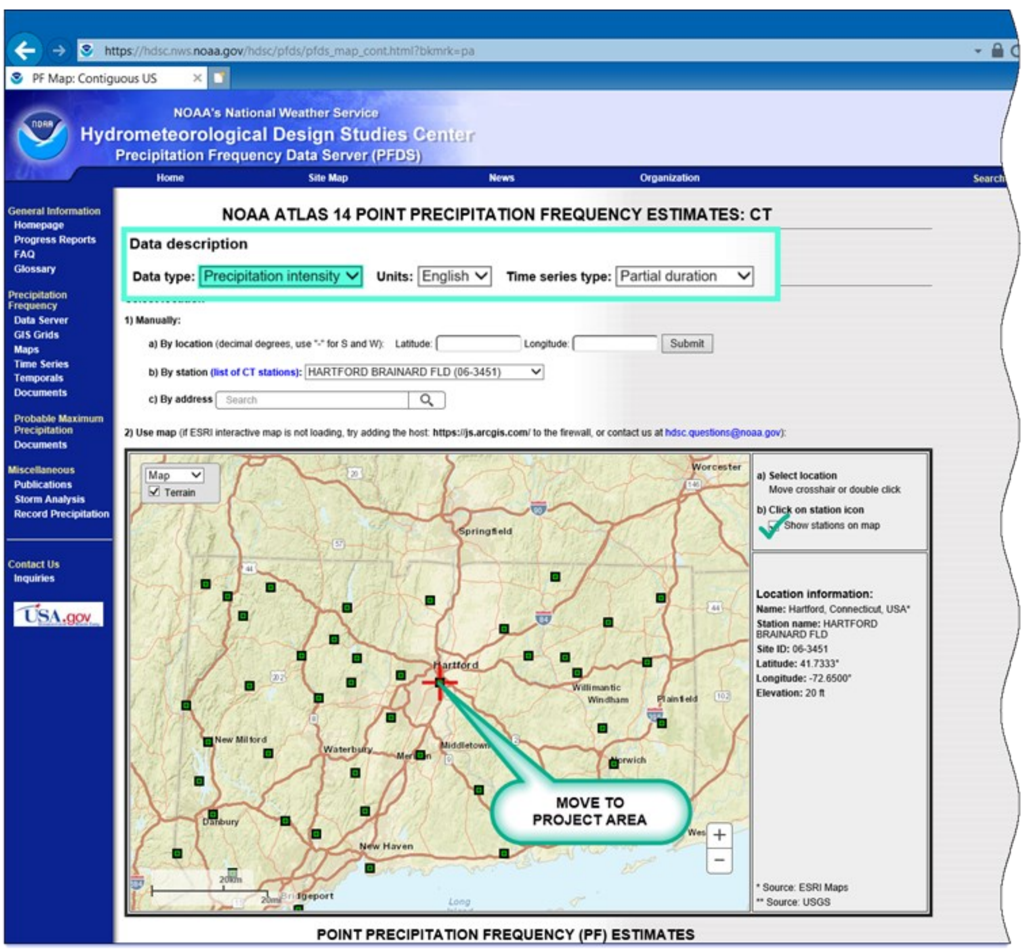
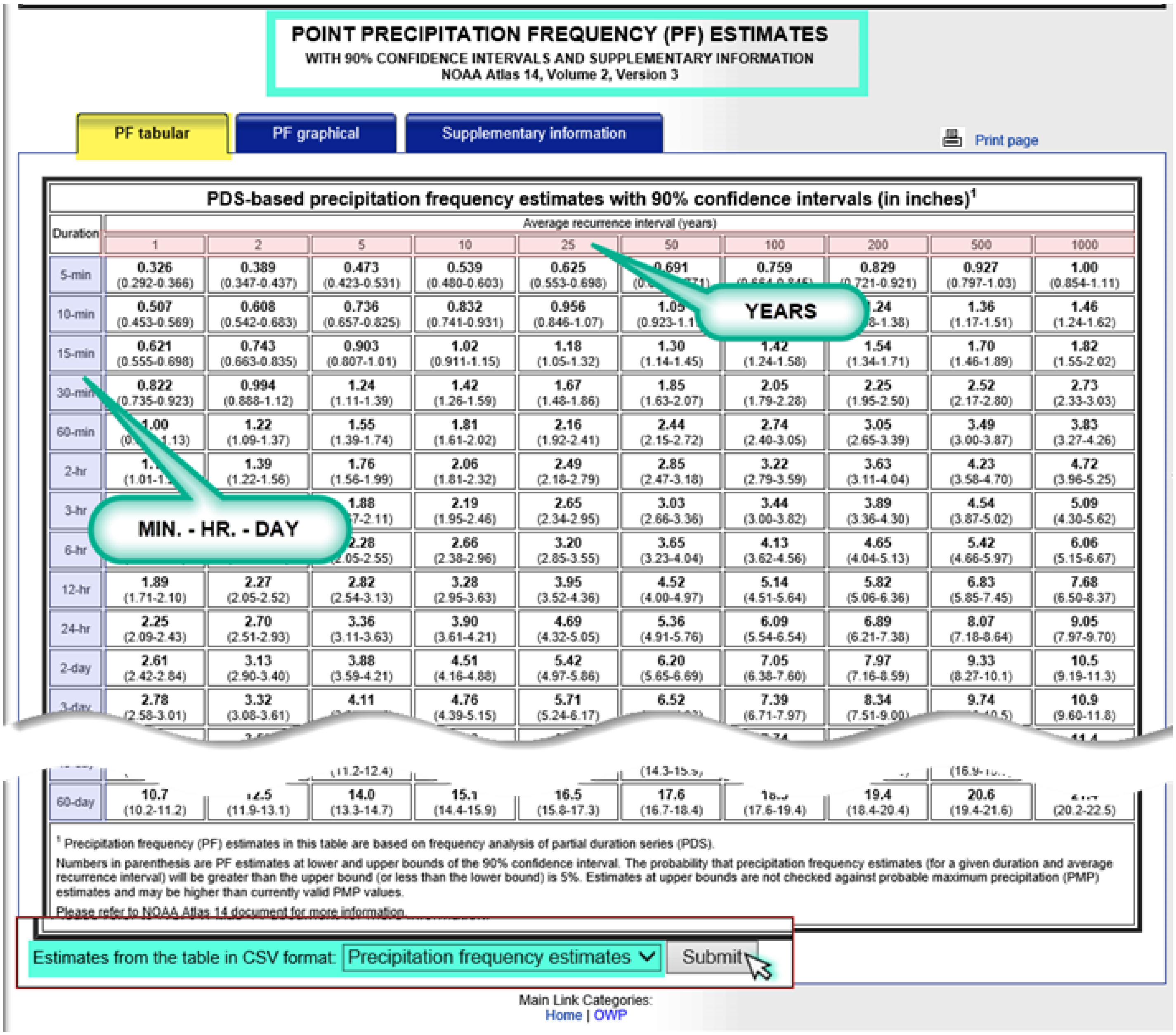


Figure 11 NOAA Atlas 14

Scroll down the webpage, making sure CSV format is set to: Precipitation Frequency Estimates and click **Submit**.



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

Duration	Average recurrence interval (years)									
	1	2	5	10	25	50	100	200	500	1000
5-min	0.326 (0.292-0.366)	0.389 (0.347-0.437)	0.473 (0.423-0.531)	0.539 (0.480-0.603)	0.625 (0.553-0.698)	0.691 (0.[illegible]71)	0.759 ([illegible])	0.829 (0.721-0.921)	0.927 (0.797-1.03)	1.00 (0.854-1.11)
10-min	0.507 (0.453-0.569)	0.608 (0.542-0.683)	0.736 (0.657-0.825)	0.832 (0.741-0.931)	0.956 (0.846-1.07)	1.05 (0.923-1.1[illegible])	[illegible]	[illegible].24 ([illegible]8-1.38)	1.36 (1.17-1.51)	1.46 (1.24-1.62)
15-min	0.621 (0.555-0.698)	0.743 (0.663-0.835)	0.903 (0.807-1.01)	1.02 (0.911-1.15)	1.18 (1.05-1.32)	1.30 (1.14-1.45)	1.42 (1.24-1.58)	1.54 (1.34-1.71)	1.70 (1.46-1.89)	1.82 (1.55-2.02)
30-min	0.822 (0.735-0.923)	0.994 (0.888-1.12)	1.24 (1.11-1.39)	1.42 (1.26-1.59)	1.67 (1.48-1.86)	1.85 (1.63-2.07)	2.05 (1.79-2.28)	2.25 (1.95-2.50)	2.52 (2.17-2.80)	2.73 (2.33-3.03)
60-min	1.00 (0.[illegible]-1.13)	1.22 (1.09-1.37)	1.55 (1.39-1.74)	1.81 (1.61-2.02)	2.16 (1.92-2.41)	2.44 (2.15-2.72)	2.74 (2.40-3.05)	3.05 (2.65-3.39)	3.49 (3.00-3.87)	3.83 (3.27-4.26)
2-hr	1.[illegible] (1.01-1.[illegible])	1.39 (1.22-1.56)	1.76 (1.56-1.99)	2.06 (1.81-2.32)	2.49 (2.18-2.79)	2.85 (2.47-3.18)	3.22 (2.79-3.59)	3.63 (3.11-4.04)	4.23 (3.58-4.70)	4.72 (3.96-5.25)
3-hr	[illegible]	[illegible]	1.88 ([illegible]7-2.11)	2.19 (1.95-2.46)	2.65 (2.34-2.95)	3.03 (2.66-3.36)	3.44 (3.00-3.82)	3.89 (3.36-4.30)	4.54 (3.87-5.02)	5.09 (4.30-5.62)
6-hr	[illegible]	[illegible]	2.28 (2.05-2.55)	2.66 (2.38-2.96)	3.20 (2.85-3.55)	3.65 (3.23-4.04)	4.13 (3.62-4.56)	4.65 (4.04-5.13)	5.42 (4.66-5.97)	6.06 (5.15-6.67)
12-hr	1.89 (1.71-2.10)	2.27 (2.05-2.52)	2.82 (2.54-3.13)	3.28 (2.95-3.63)	3.95 (3.52-4.36)	4.52 (4.00-4.97)	5.14 (4.51-5.64)	5.82 (5.06-6.36)	6.83 (5.85-7.45)	7.68 (6.50-8.37)
24-hr	2.25 (2.09-2.43)	2.70 (2.51-2.93)	3.36 (3.11-3.63)	3.90 (3.61-4.21)	4.69 (4.32-5.05)	5.36 (4.91-5.76)	6.09 (5.54-6.54)	6.89 (6.21-7.38)	8.07 (7.18-8.64)	9.05 (7.97-9.70)
2-day	2.61 (2.42-2.84)	3.13 (2.90-3.40)	3.88 (3.59-4.21)	4.51 (4.16-4.88)	5.42 (4.97-5.86)	6.20 (5.65-6.69)	7.05 (6.38-7.60)	7.97 (7.16-8.59)	9.33 (8.27-10.1)	10.5 (9.19-11.3)
3-day	2.78 (2.58-3.01)	3.32 (3.08-3.61)	4.11	4.76 (4.39-5.15)	5.71 (5.24-6.17)	6.52	7.39 (6.71-7.97)	8.34 (7.51-9.00)	9.74	10.9 (9.60-11.8)
60-day	10.7 (10.2-11.2)	[illegible]2.5 (11.9-13.1)	14.0 (13.3-14.7)	15.1 (14.4-15.9)	16.5 (15.8-17.3)	17.6 (16.7-18.4)	[illegible] (17.6-19.4)	19.4 (18.4-20.4)	20.6 (19.4-21.6)	[illegible] (20.2-22.5)

[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 precipitation frequency estimates (for a given duration and average recurrence interval) will be greater than the upper bound (or less than the lower bound) is 5%. Estimates at upper bounds are not checked against probable maximum precipitation (PMP) estimates and may be higher than currently valid PMP values.

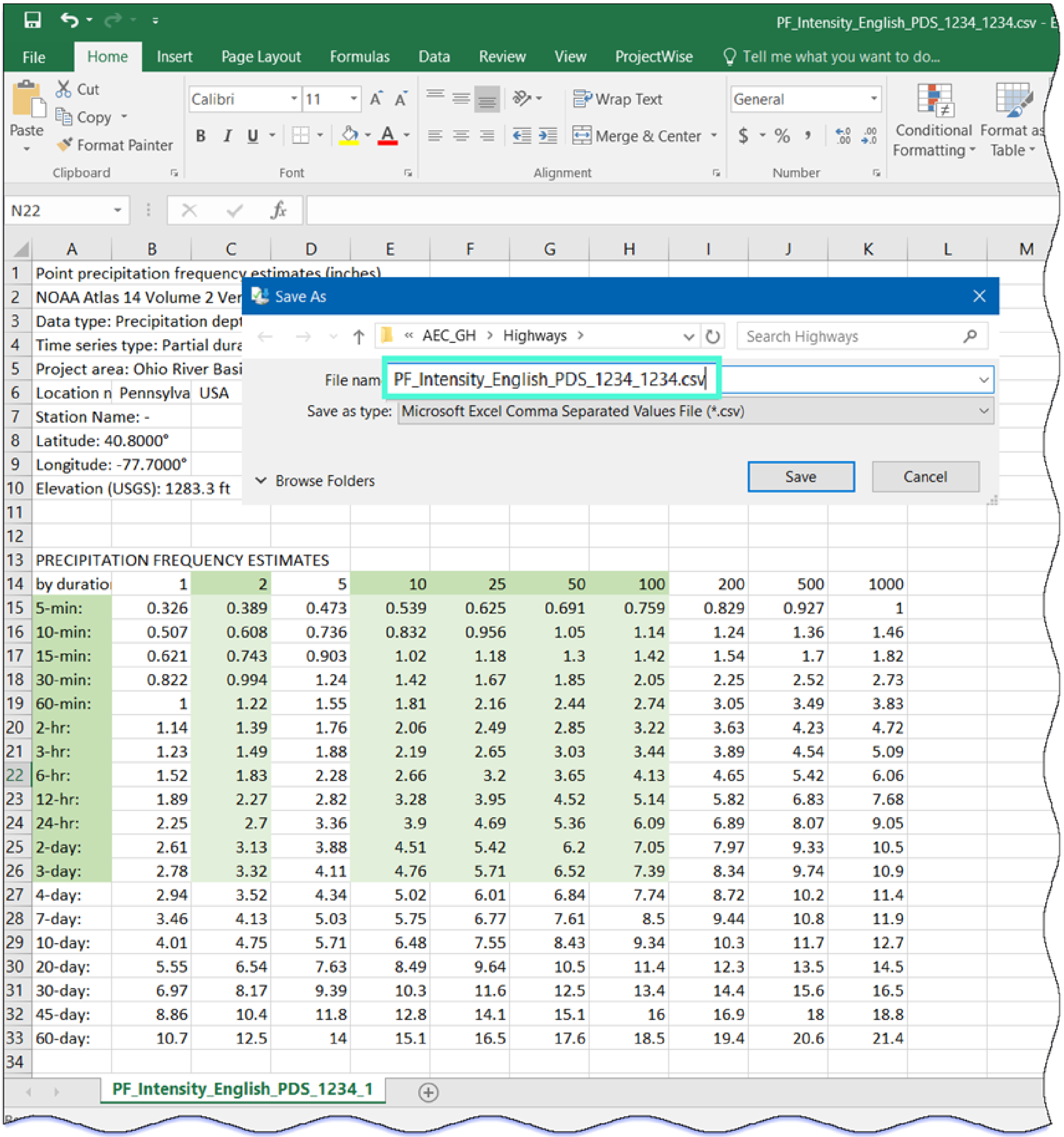
Please refer to NOAA Atlas 14 document for more information.

Figure 12 Precipitation Frequency Estimates

Either select **Open**, **Save** or **Save As**, (save to Project directory).

Figure 13 Save, Open, Cancel Buttons

The Excel file will be used to create an **IDF-Table** for the projects storm data. The options available to create an IDF-table will be covered later in this manual.



Point precipitation frequency estimates (inches)
NOAA Atlas 14 Volume 2 Ver
Data type: Precipitation dept
Time series type: Partial dura
Project area: Ohio River Basi
Location n Pennsylva USA
Station Name: -
Latitude: 40.8000°
Longitude: -77.7000°
Elevation (USGS): 1283.3 ft

PRECIPITATION FREQUENCY ESTIMATES

by duratio	1	2	5	10	25	50	100	200	500	1000
5-min:	0.326	0.389	0.473	0.539	0.625	0.691	0.759	0.829	0.927	1
10-min:	0.507	0.608	0.736	0.832	0.956	1.05	1.14	1.24	1.36	1.46
15-min:	0.621	0.743	0.903	1.02	1.18	1.3	1.42	1.54	1.7	1.82
30-min:	0.822	0.994	1.24	1.42	1.67	1.85	2.05	2.25	2.52	2.73
60-min:	1	1.22	1.55	1.81	2.16	2.44	2.74	3.05	3.49	3.83
2-hr:	1.14	1.39	1.76	2.06	2.49	2.85	3.22	3.63	4.23	4.72
3-hr:	1.23	1.49	1.88	2.19	2.65	3.03	3.44	3.89	4.54	5.09
6-hr:	1.52	1.83	2.28	2.66	3.2	3.65	4.13	4.65	5.42	6.06
12-hr:	1.89	2.27	2.82	3.28	3.95	4.52	5.14	5.82	6.83	7.68
24-hr:	2.25	2.7	3.36	3.9	4.69	5.36	6.09	6.89	8.07	9.05
2-day:	2.61	3.13	3.88	4.51	5.42	6.2	7.05	7.97	9.33	10.5
3-day:	2.78	3.32	4.11	4.76	5.71	6.52	7.39	8.34	9.74	10.9
4-day:	2.94	3.52	4.34	5.02	6.01	6.84	7.74	8.72	10.2	11.4
7-day:	3.46	4.13	5.03	5.75	6.77	7.61	8.5	9.44	10.8	11.9
10-day:	4.01	4.75	5.71	6.48	7.55	8.43	9.34	10.3	11.7	12.7
20-day:	5.55	6.54	7.63	8.49	9.64	10.5	11.4	12.3	13.5	14.5
30-day:	6.97	8.17	9.39	10.3	11.6	12.5	13.4	14.4	15.6	16.5
45-day:	8.86	10.4	11.8	12.8	14.1	15.1	16	16.9	18	18.8
60-day:	10.7	12.5	14	15.1	16.5	17.6	18.5	19.4	20.6	21.4

Figure 14 Excel file IDF-Table

1.2 Terrain Data

Drainage and Utilities uses the terrain model referenced into a DGN file to display the storm drainage and other utilities in 2D and 3D models. Drainage structures and conduits are fully integrated in graphic and database functions. The data **is in** the graphics within the design file (*.dgn file).

Drainage and Utilities can use the elevations from the active terrain, but Drainage and Utilities can only work with **one terrain**; this can make it necessary to merge terrains such as existing, LIDAR, point clouds and/or proposed into one terrain. The designer should create a specific "drainage terrain" for use with the drainage layout and calculations.

To merge two or more terrains, add Lidar data the user should visit Bentley Help - Terrain Edit.

This "drainage terrain" is in its own MicroStation DGN file and is referenced into the "drainage" MicroStation DGN file.

1.3 Create a Drainage Model (DGN File)

Ensure OpenRoads Designer Connect Edition is set to the correct Workspace and Workset. Create a new *dgn file (2D) exclusively for your storm drainage layout, calculations, and utility conflicts.

1. From the **New** dialog box, ***browse*** to the proper discipline folder, enter the file using Civil Design CAD File Naming Conventions DD_DC_PROJ_NUMB_Description.

   Example: ***HW_CB_1234_1234_ProposedDrainage.dgn***

2. Change the workflow to: **Drainage and Utilities** (upper left corner), open the **Drainage and Utilities Model** within the Explorer dialog box.

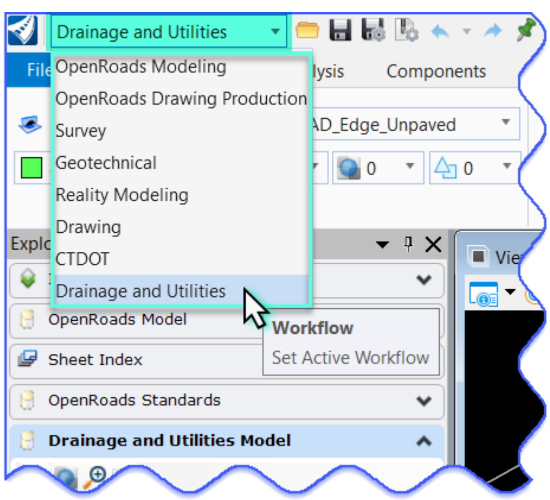


Figure 15 Drainage and Utilities Workflow

3. From the **Home** tab use the ***Attach Tools - References*** to attach the survey, terrain, aerials, and design files as needed. ***Turn OFF*** Levels as needed from the individual files.

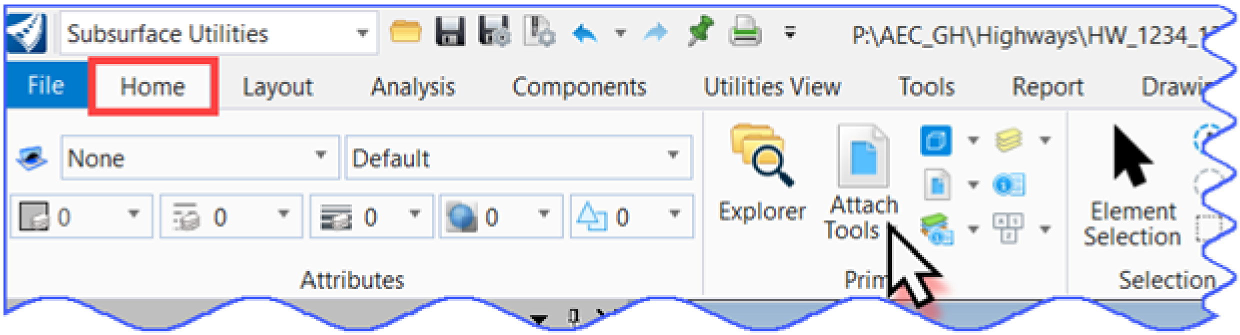


Figure 16 Attach Tools - References

4. Set the **Active Terrain**. Using the ***Element Selection tool*** click on the boundary line for the terrain, hover over it, select the tool ***Set As Active Terrain Model***, this will make the terrain active and will be used for placing the drainage structures.

5. In the ***2D view (Default)*** override the Symbology and ***turn off*** triangles. Open the ***Default 3D view*** and set-up as needed to your preference.

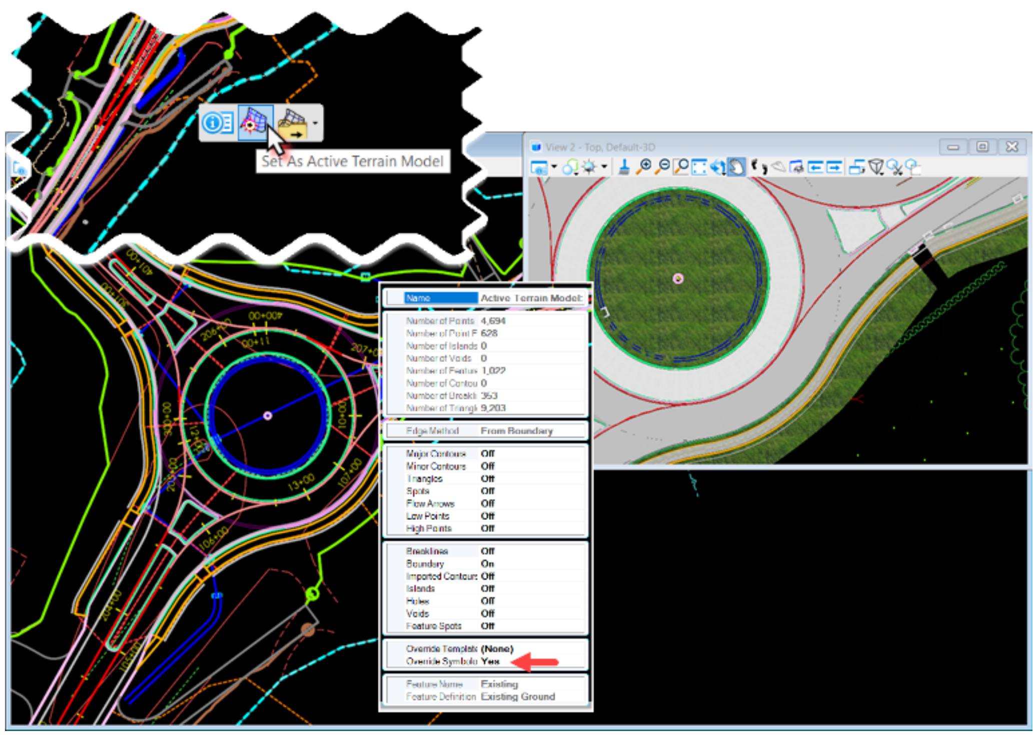


Figure 17 Set As Active Terrain Model

6. **Save settings**.

7. The **Drainage and Utilities Hydraulic Engine** needs to be activated. From the **Layout** tab, select the ***Place Node*** command. This has to be done once per DGN file to activate the

utility model and to create the hydraulic project. **Only one hydraulic project per dgn file**, but multiple systems can be place and calculated.

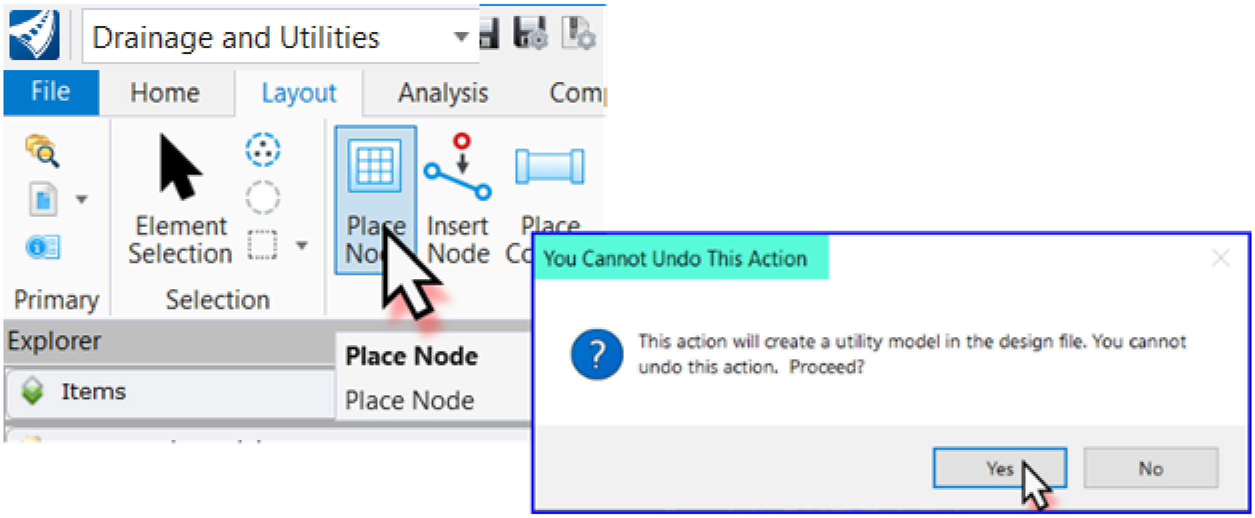


Figure 18 Utility Model Creatin Warning Box

8. The **You Cannot Undo This Action** message box will open, ***click Yes***. This will open and prepare the Drainage and Utilities project in your DGN file, some context menus will pop-up and disappear, wait until completed and your cursor comes back. Now you are ready to place your structures, conduits, catchment areas and do your calculations.

1.4 Set Defaults and Constraints

Some project specific properties, defaults and constraints need to be reviewed and set for each project, where possible CTDOT standards have been set.

1.4.1 Options

1. Click on the **Tools** tab and **select Options - Drainage**. The **Options** box opens.
2. **Hydraulic Model** tab:
    - Pipe Length – Use 3D Length? is unchecked.
    - ***Units*** tab: Default Unit System for New Project is set to: ***US Customary***
    - make sure all units are set correctly. Such as rainfall intensity is set to the intervals as shown on the NOAA PPF Estimates.
3. Click **OK**.
4. Click on the **Analysis** tab and select Default Design Constraints.

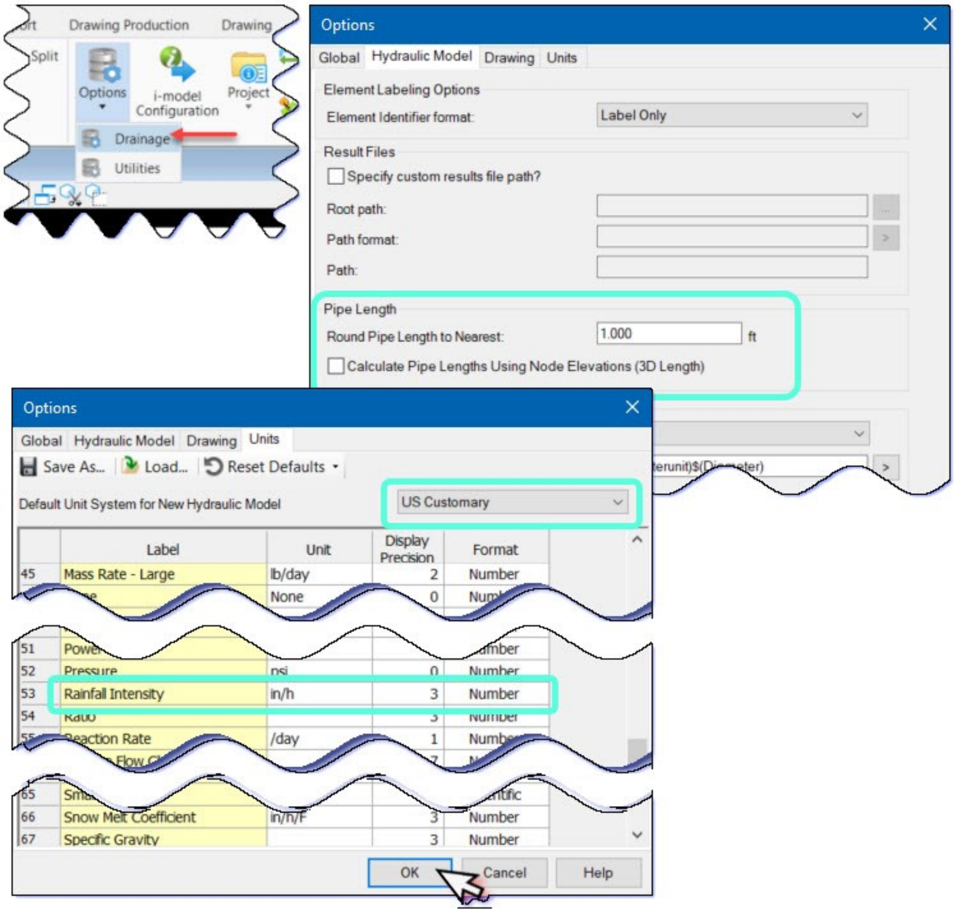


Figure 19 select Options - Drainage

The constraints have been set to follow the *CTDOT Drainage Manual* where possible. If you want Drainage and Utilities to design your drainage system, it is especially important to review, edit and make changes to follow design standards for the individual project, such as *maximum spread*. The **Drainage and Utilities Help** menu should be used for each specific explanation of the **Default Design Constraints** topic.

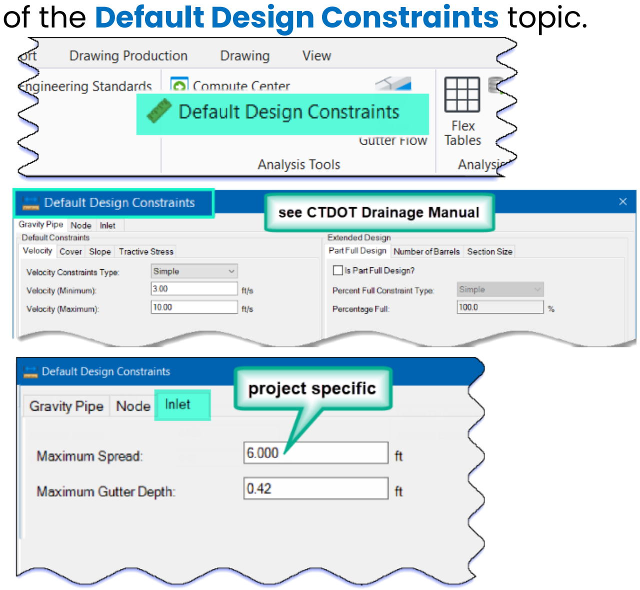


Figure 20 Default Design Constraints

1.4.2 Prototypes

1. Click on **Components > Catalog > Prototypes**.

Prototypes allow for entering default values for elements in a drainage network and are used to ***define the hydraulic properties of features***. A prototype works in cooperation with element templates and feature definitions to define the characteristics of a drainage feature. The most widely used conduits, gutters, catch basins, manholes, cross sections, catchments, and headwalls etc. have been set to **CTDOT Standards**.

Some properties are project specific and should be edited as an individual item within the items Hydraulic Analysis. It is important the engineer reviews and edits these to fit project needs. See the Bentley Help menu for further explanation of the prototype settings.

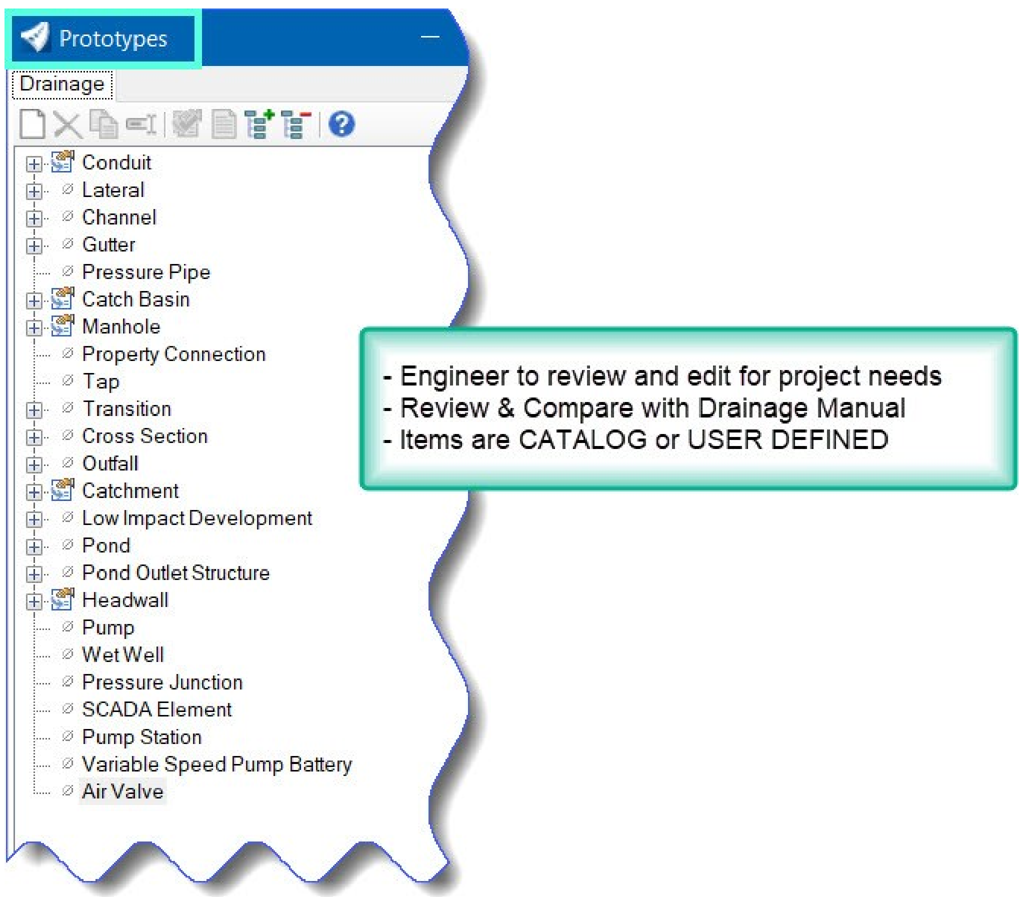


Figure 21 List of Prototypes

Exercise 2 – Laying Out Drainage

2.1 Nodes

2.1.1 Place Catch Basin

1. In **View Attributes** *turn off* **Fill** to only see the outline of structures (catch basin, etc.), this will help when placing nodes and conduits.
2. Select the **Place Node** command to lay out your drainage structures: catch basins, endwalls, etc.

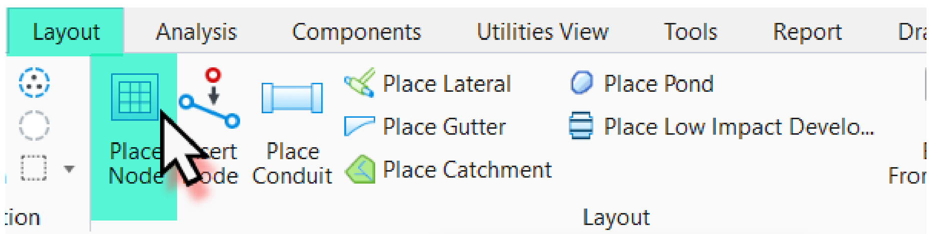


Figure 22 Place Node

3. Click on the reference element for node elevation (active terrain or edge of road) or click reset to type a node elevation, type in 0.00 for vertical offset, this can be changed later for the gutter depression of Type "C" catch basins. Rotate as needed to place.

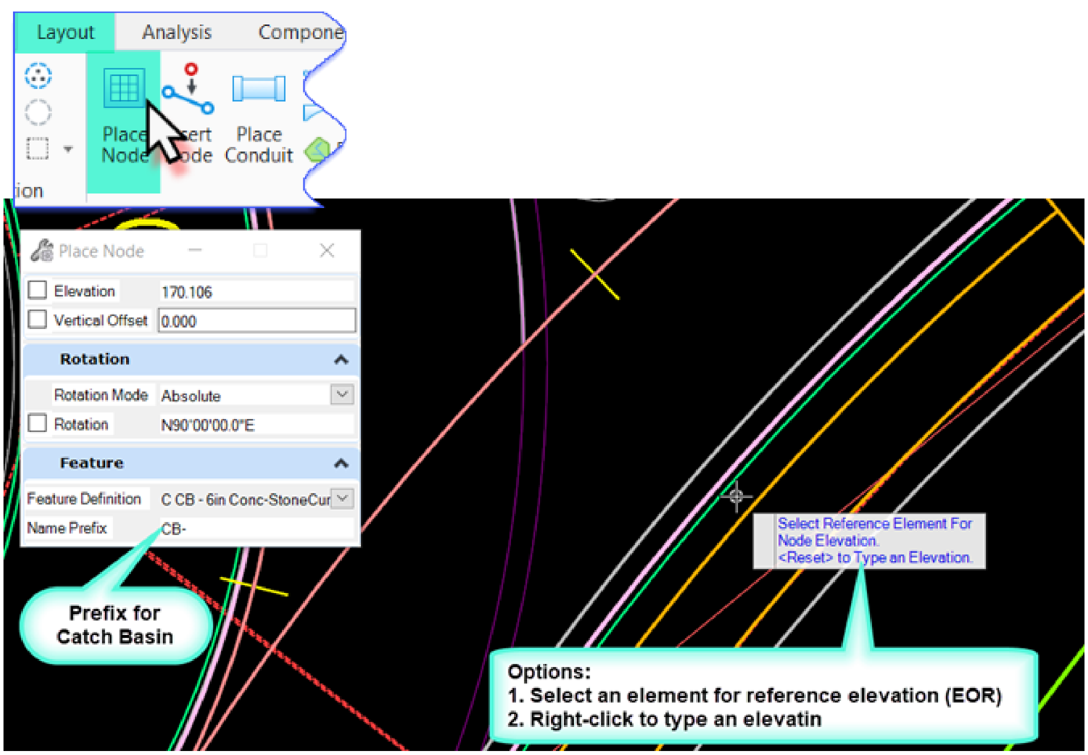


Figure 23 Place Node Settings

4. Hover over the node to enable the context toolbox: properties, utility properties and delete. The node can be moved and rotated.
5. Continue placing more structures (catch basins, end structures or manholes). After you placed the last structure, right click once more, and select the Element Selection tool to end the Place Node command.

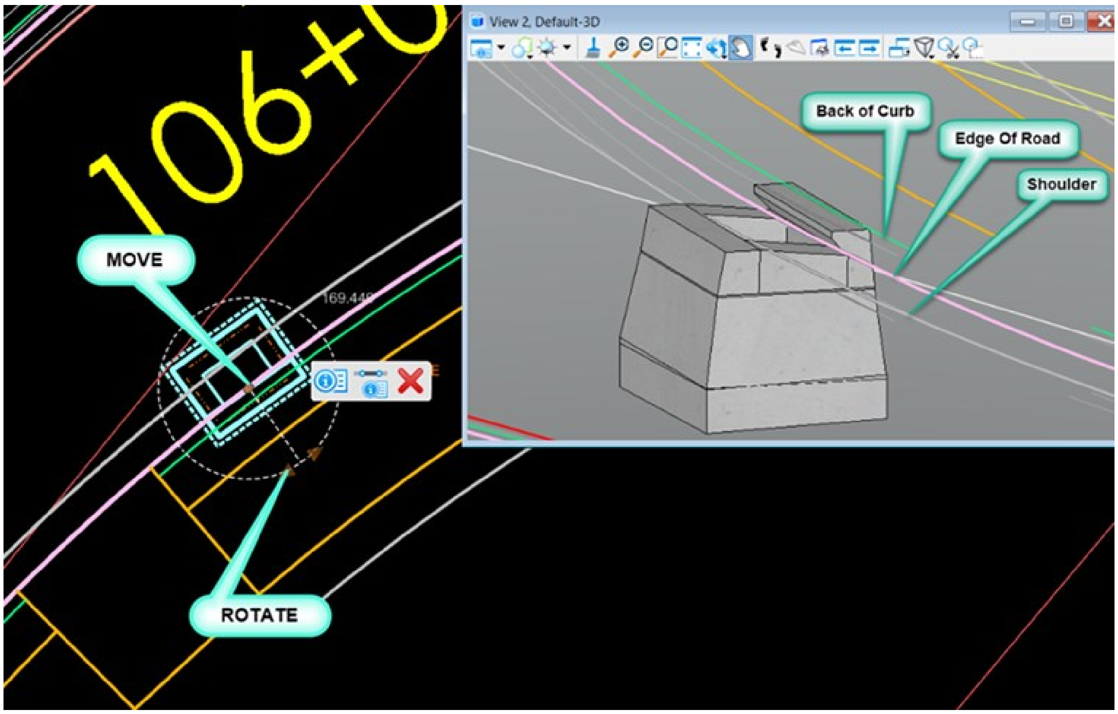


Figure 24 Catch Basins Placement Criteria

2.1.2 Place Culvert End

1. Select the **Place Node** command to place a culvert end.
2. Select the **Feature Definition** (example: 24" RCCE) and follow the pop-up prompts. The Name Prefix is **RCCE-** by default.
3. Click on the reference element for node elevation (active terrain or edge of road) or click reset to type a node elevation, type in 0.00 for vertical offset. Rotate as needed to place.

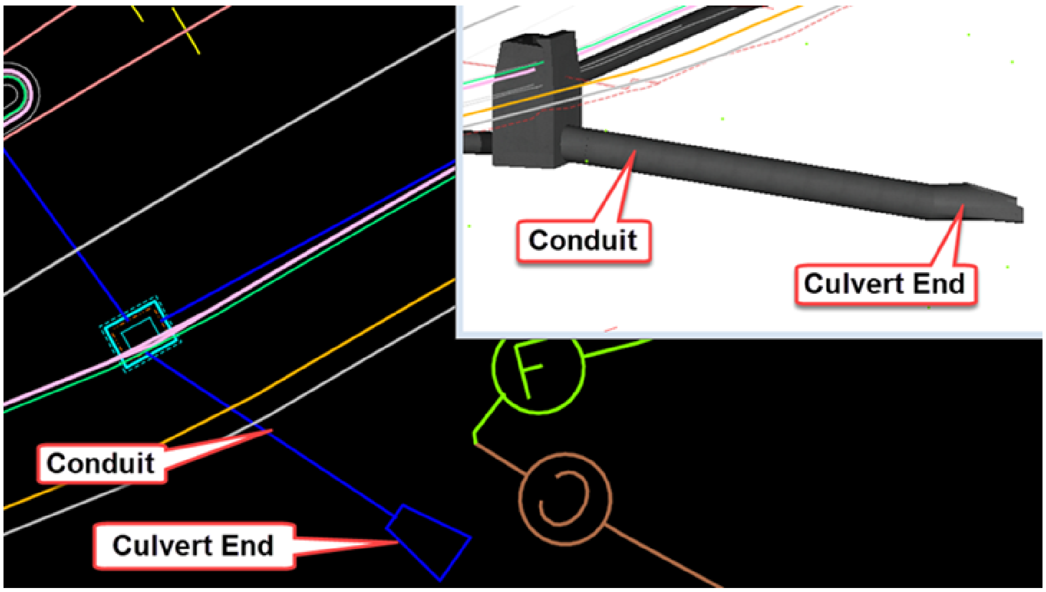


Figure 25 Culvert End Placement Criteria

4. After the conduit is placed click and hover over the culvert end for the context menu, click on properties.

5. In the properties box, change the following:
   - Use Slope of Surface = ***False***
   - Match Slope of Conduit = ***True***

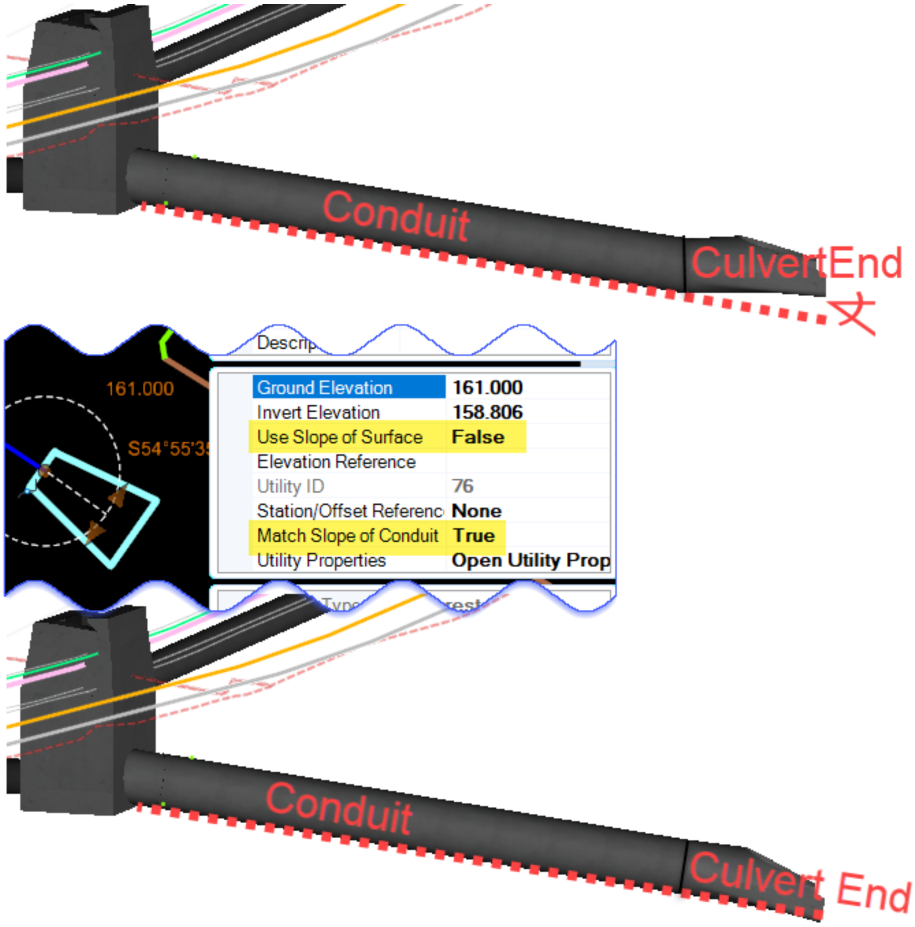


Figure 26 Culvert End Properties Update

2.1.3 Feature Definitions for Drainage Nodes

1. To view the available Drainage nodes expand the **OpenRoads Standards** area in Explorer. Click ***Standards > Libraries > Feature Definitions***.
2. Select either ***Feature Definition (CV_DU pr…)*** or ***(CV_DU Ex…)***
3. Select ***Drainage and Utilities > Node > Catch Basins*** and review the options.
4. Browse the other listed folders.

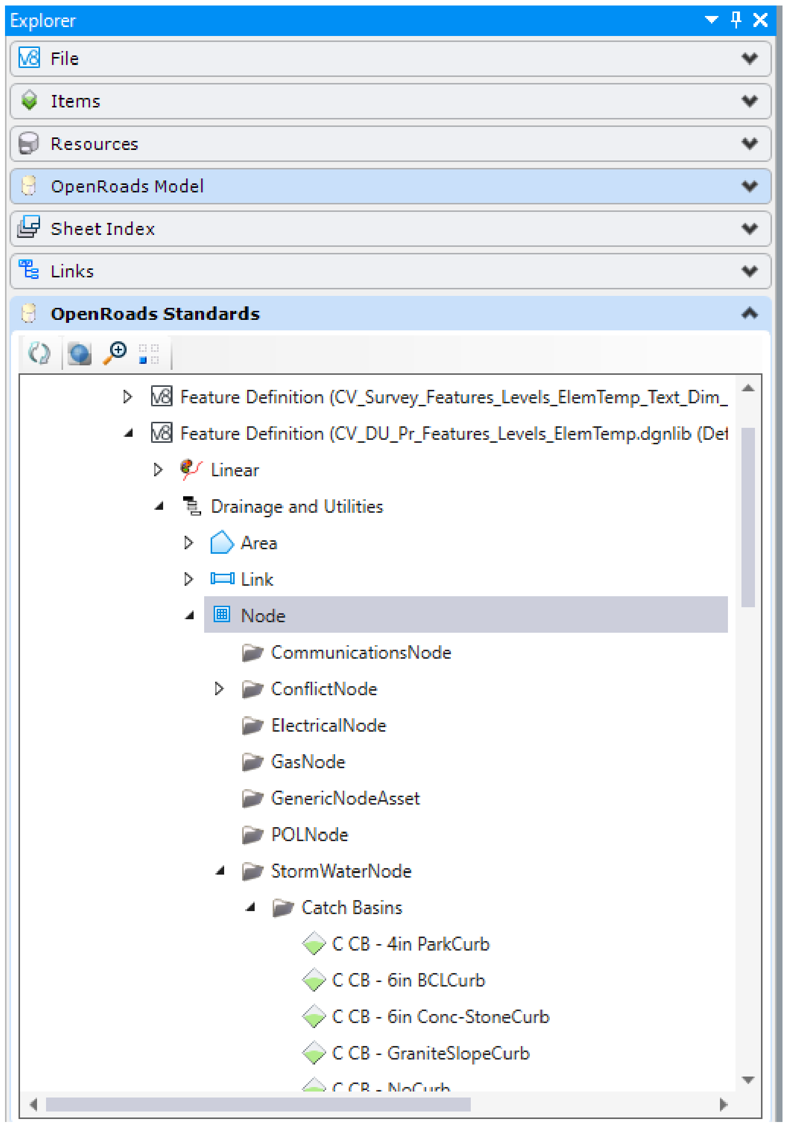


Figure 27 Available Drainage Nodes

2.1.4 Inlet Catalog

All inlet nodes (structures) available for design are stored in the **Inlet Catalog**. This catalog has been set-up for all catch basin types and other structures according to CTDOT standards.

To view the inlet catalog: Click the ***Components*** tab – ***Catalog*** – ***Inlet Catalog***

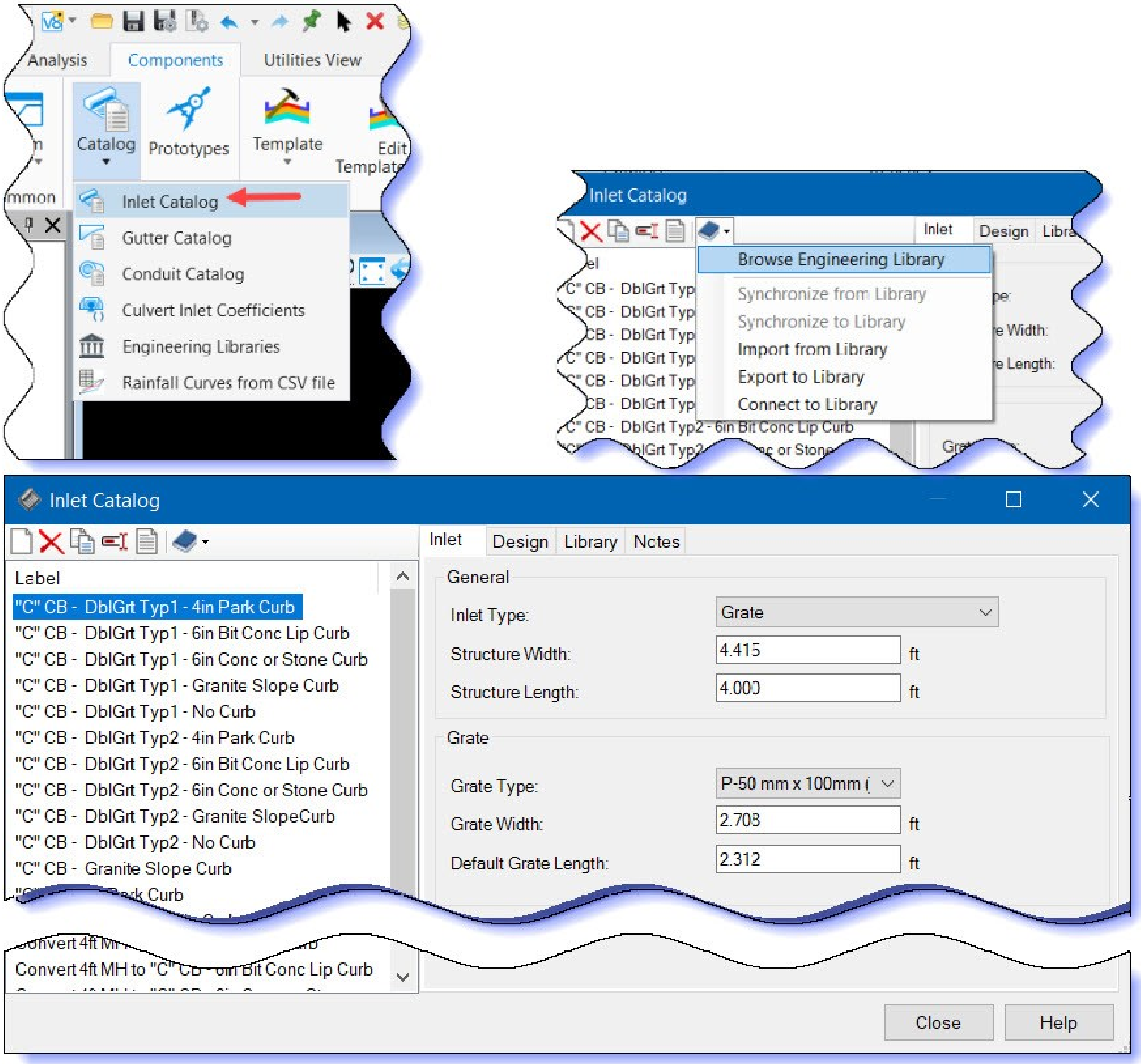


Figure 28 Inlet Catalog

2.2 Conduit

2.2.1 Place Conduit

Conduits are placed between nodes such as catch basins, manholes and head- and end walls.

Note: *If a structure is deleted the associated conduit connecting the node is also deleted.*

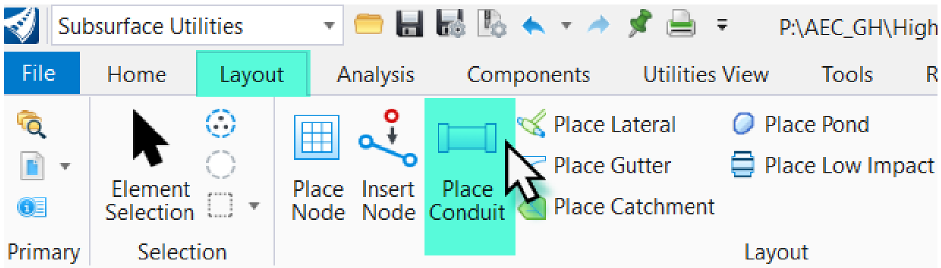


Figure 29 Place Conduit

1. Select the **Place Conduit** command, choose the **Feature Definition** for the pipe you want to place (example: RC Pipe). The Name Prefix is **SW-** (Storm Water) by default. The Type: is greyed out – this comes from the conduit catalog already attached, for **Description** click on the down arrow to select the pipe size (example: 15").
2. Follow the prompts: Select ***Start Node***, in the drawing hover your courser over/close to the first inlet (node i.e. catch basin), it will snap to the node where the conduit can connect, this is called the ***Connection Region***. **Accept** (left click).

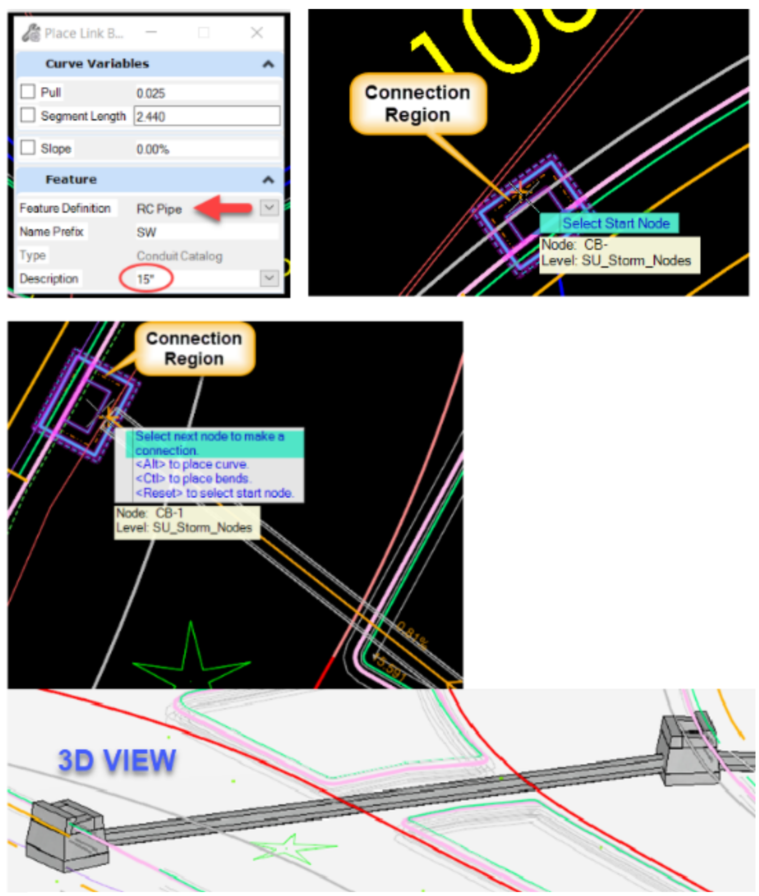


Figure 30 Place Conduit Criteria

3. Select next node to make a connection. Again, hover your cursor close to the inlet to connect the conduit to (second node i.e. catch basin). **Accept** (left click).
4. Continue to place conduits to connect the drainage structures from the first inlet to the outlet for the system. Select the Element Selection tool to end the place conduit command.

2.2.2 Conduit Catalog

Go to ***Components - Conduit Catalog*** to see available conduits. In the Conduit Catalog the designer is able to set conduits ***Available for Design*** for the drainage design.

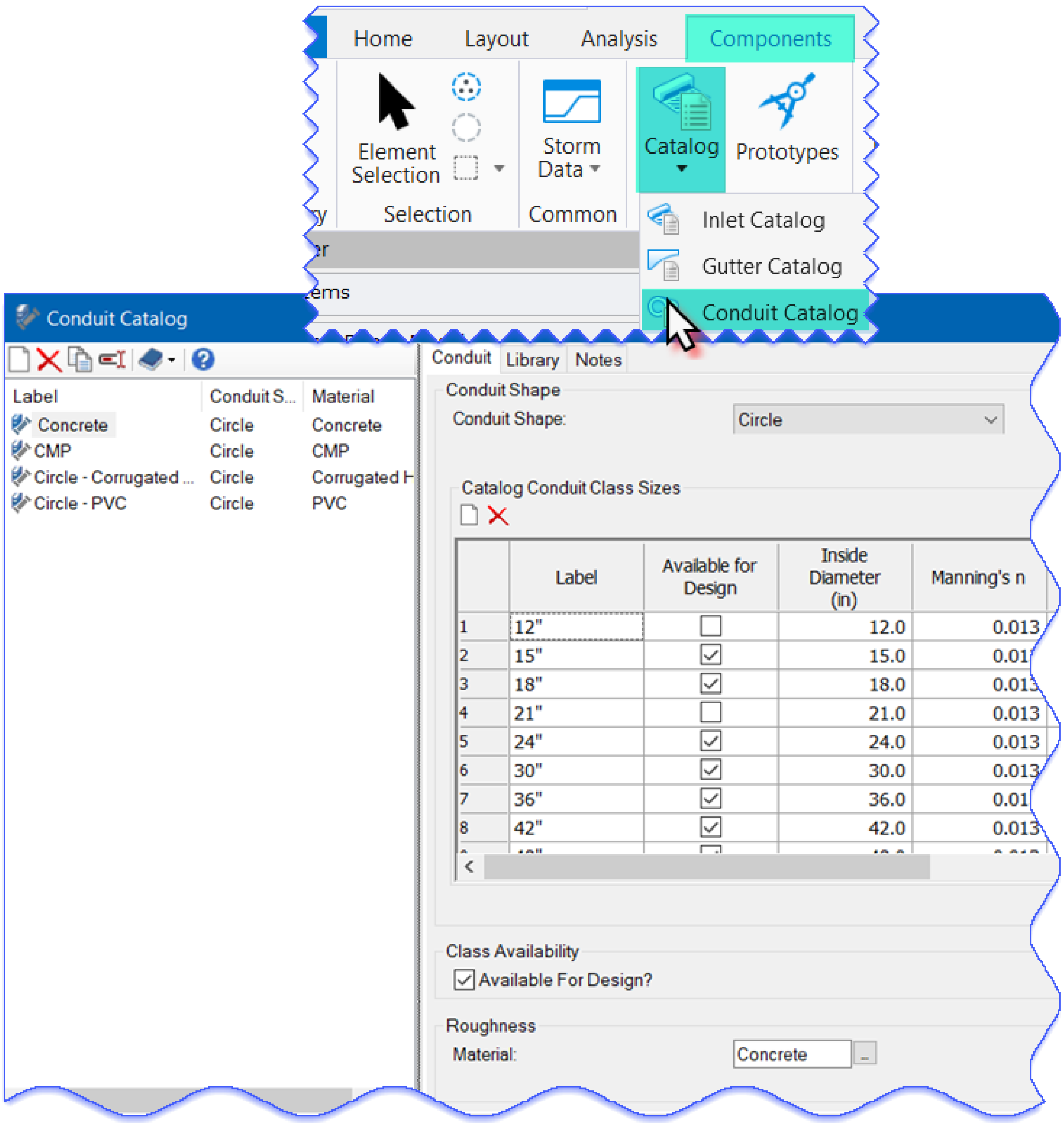


Figure 31 Conduit Catalog

2.3 Input Invert Elevations

After completing the layout for the drainage structures and conduits you will enter the invert elevations for all structures and conduits, Drainage and Utilities sets conduit inverts to the bottom of the structures by default.

Drainage and Utilities has the capability to calculate the pipe inverts when you use the *Base Design Scenario* in the Hydraulic Analysis. The Default Design Constraints need to be set for the project as required. It is recommended the designer reviews the *Bentley Learning path* for Hydraulic Analysis and Design in OpenRoads Designer.

By selecting a placed utility feature, some feature properties can be edited, rotated and/or moved in the graphic (just as with other OpenRoads tools), it shows the label and level. Also, the context toolbox displays and offers two options to input/edit utility information:

Properties - lists the CADD properties such as level, color, feature, feature definition etc.; and

Utility Properties- lists every available hydraulic property including user customized properties.

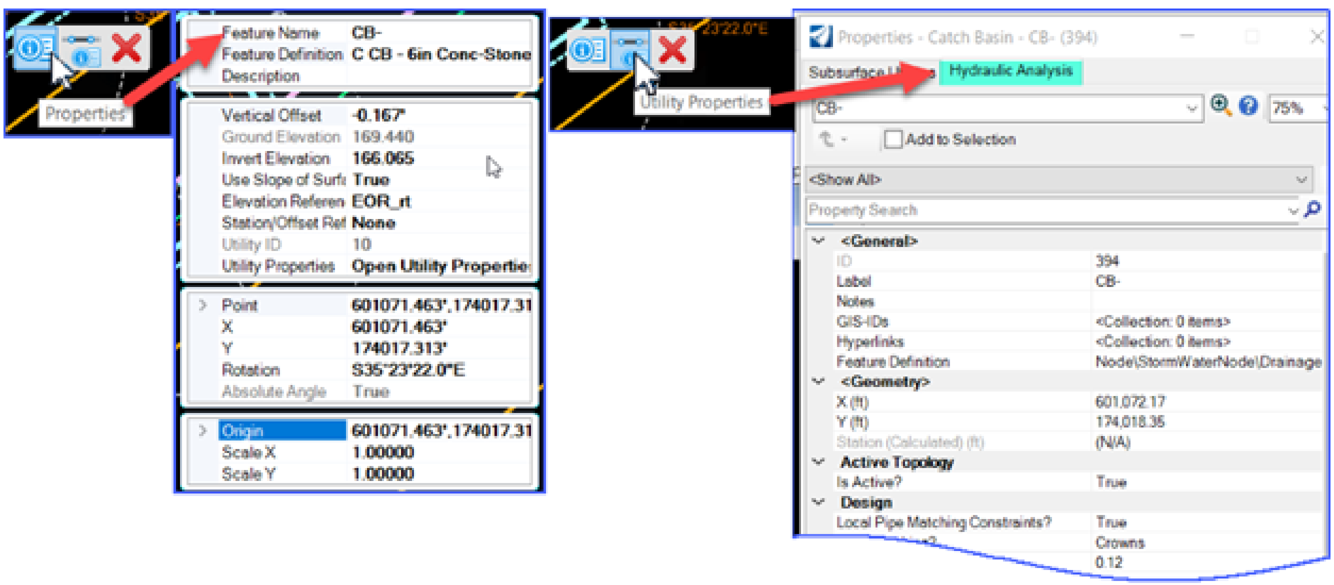


Figure 32 Properties vs Utility Properties

Drainage and Utilities tools represent the merger of two diverse technologies. OpenRoads Designer builds the 3D models of the utilities, but Bentley's Haestad technology (StormCAD) executes the analytic modeling for hydraulic and utility engineering. We will concentrate on Utilities Properties.

2.3.1.1 Node

Click and ***hover over*** a node (catch basin or manhole) from the context toolbox select **Utility Properties**, click on the **Hydraulic Analysis** tab.

Check and edit the physical properties for the node:

- Elevation (Ground) = Terrain elevation at node
- Set Rim to Ground Elevation? False/True
- Elevation (Rim) = Top of Grate elevation
- Elevation (Invert) = bottom elevation of structure
- Update and edit Inlet properties for the node:
- Inlet Type: set to Catalog Inlet (CV_SubsurfaceUtilities_Inlet.xml)
- Inlet: click the down arrow and select from the available structures
- Other items edit can be edited later

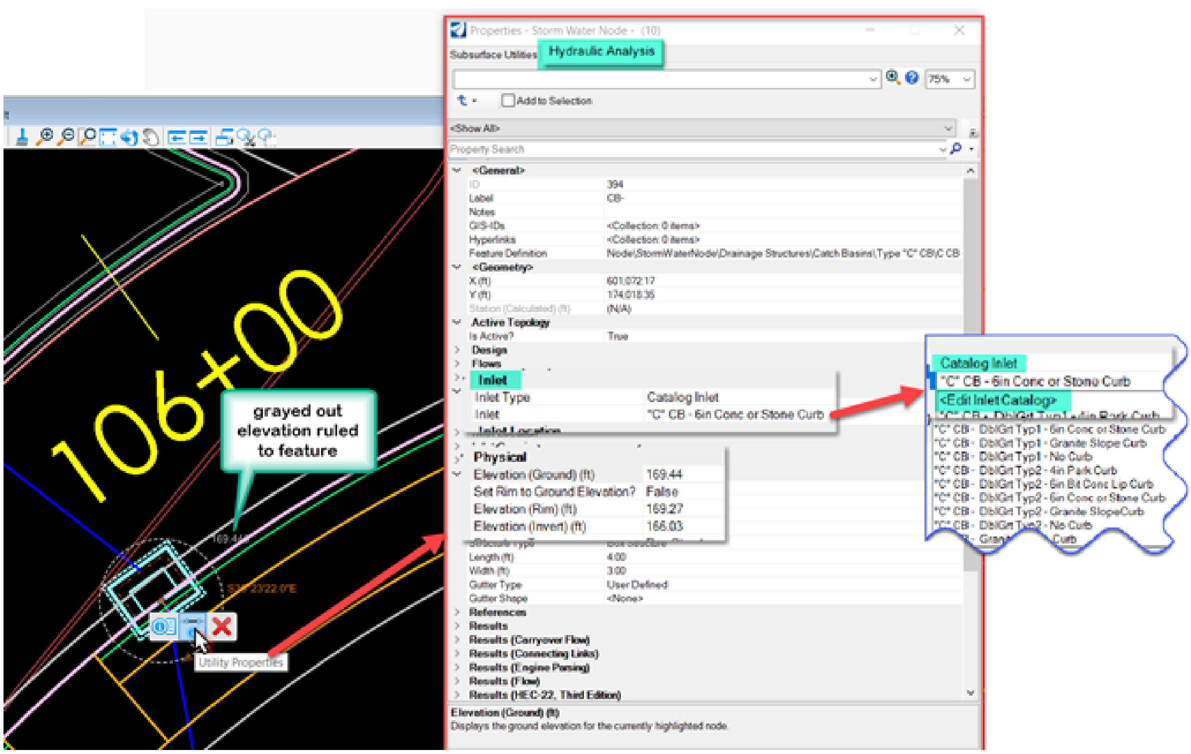


Figure 33 Utility Properties, Hydraulic Analysis tab

The Inlet catalog: **CV_SubsurfaceUtilities_Inlet.xml** has been set-up for CTDOT catch basin standards and will be updated as needed.The catalog should be available automatically when starting the drainage engine, but is also available here:

...CT_Configuration|Organization-BIM|_CT_Civil Standards|Data|CV_SubsurfaceUtilties_Inlet.xlm

2.3.1.2 Conduit

Click and ***hover over*** a conduit, in the context toolbox select **Utility Properties**, click on the **Hydraulic Analysis** tab.

Check and edit as needed the Physical Properties for the conduit:

- Conduit Type: Catalog Conduit
- Catalog Class: click the down arrow and select: Concrete, PVC, CMP
- Size: click down arrow and select size
- Invert (Start): enter elevation
- Invert (Stop): enter elevation
- Has user Defined Length? True (default) or False
- Length (user defined) enter length
- Other items will be edited later

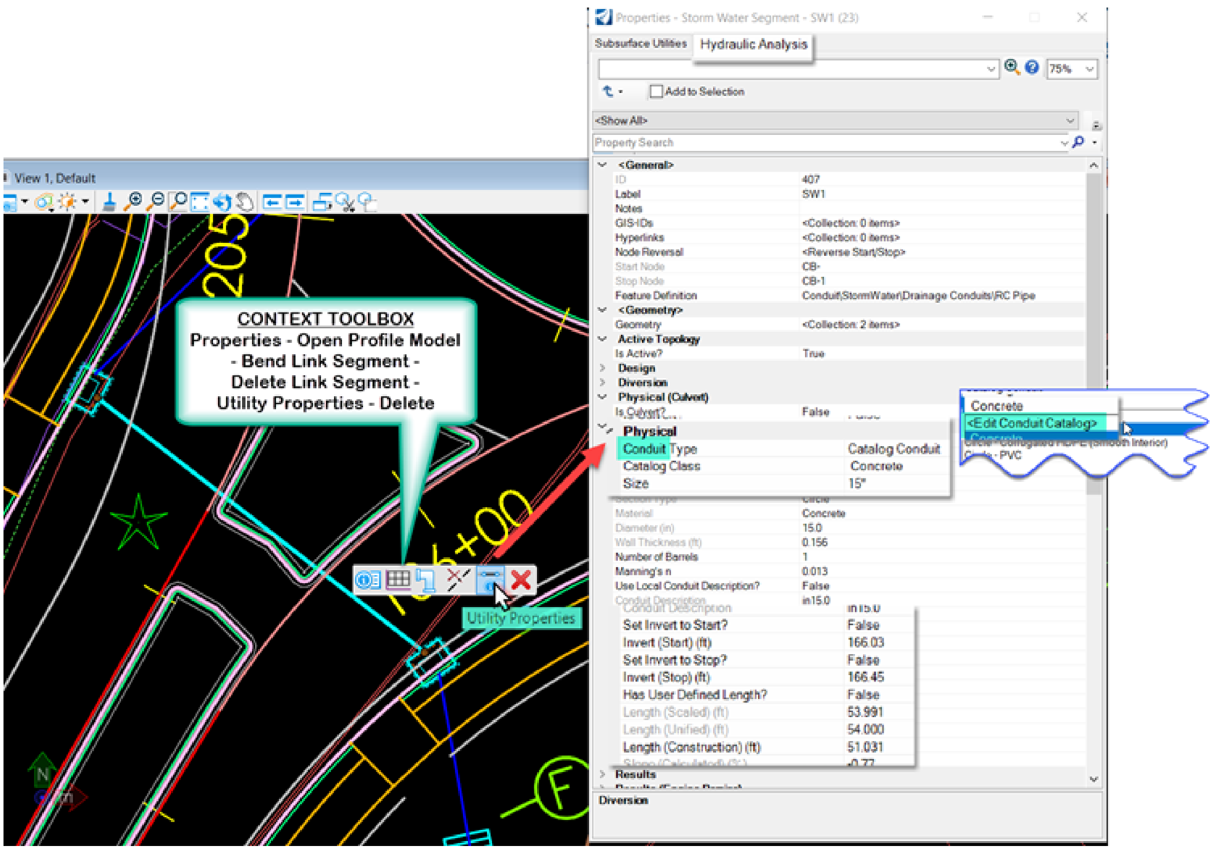


Figure 34 Utility Properties, Hydraulic Analysis tab

Most used conduit classes have been added to the catalog, if others are needed those can be added:

Click ***Catalog – Conduit Catalog – Import from Library***

Select the conduit to add from the available **Conduits Library.xml**.

2.4 Define Drainage Areas

Drainage areas are defined using existing and proposed terrains, which can be merged with terrains coming from various other sources such as LIDAR.

There are several options to place catchment areas (see Bentley Help):

- Place Catchments by Picking Shapes
- Place Catchments by Picking Points
- Place Catchments by Flood Fill

Catchment properties include catchment area, runoff coefficient, time of concentration types, and time of concentration.

2.4.1 Catchment Area

1. From the **Layout Tools** select **Place Catchment** and select the Method that best serves your purpose: example ***Pick Points***
2. From the ***Feature Definition*** pick the land use most appropriate for the catchment area: example ***Pavement***

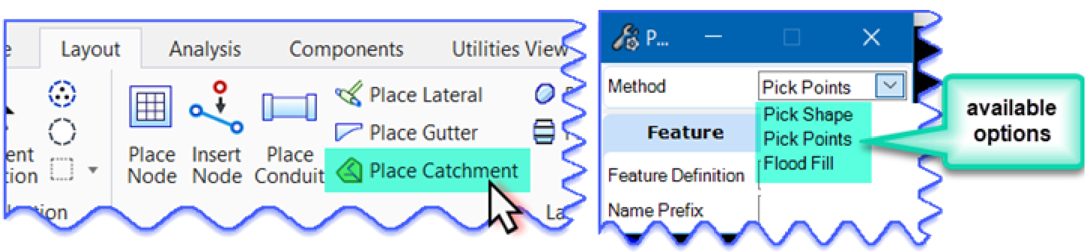


Figure 35 Place Catchment TOOL

3. For each land use prototypes have been created that follow the CTDOT Drainage Manual. The designer should adjust the Runoff Coefficient (Rational) for the project if needed in the Hydraulic Analysis tab for the catchment area.
4. Follow the command prompts: ***Select First Vertex***, ***Select Next Vertex***, ***(Reset) To Complete***, ***Select Outflow***, and ***Select Reference Surface***.

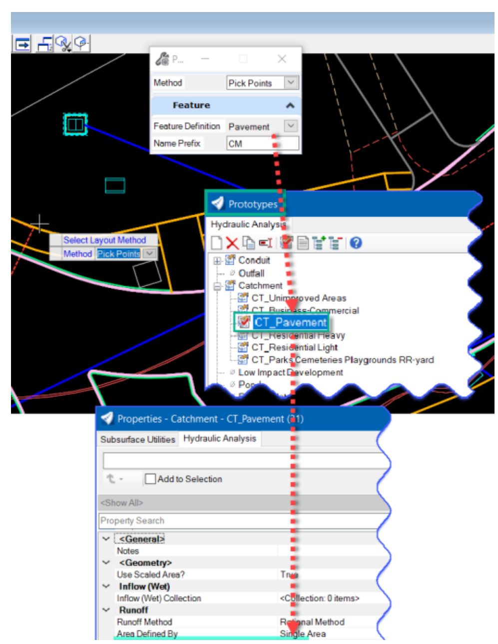
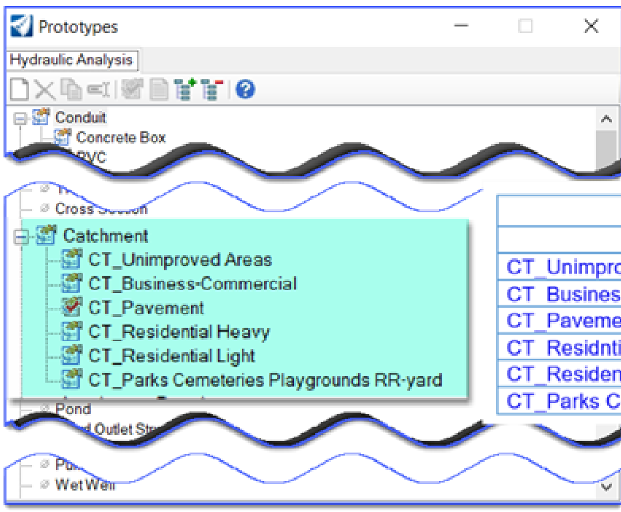


Figure 36 Select Prototype

2.4.2 Runoff Coefficient

Runoff Coefficients have been defined as prototypes following the CTDOT Drainage Manual. They are composites from Chapter 6 – Runoff Coefficient – Table 6-4 Recommended Coefficient of Runoff Values for Various Selected Land Uses and Table 6-5 Coefficients for Composite Runoff Analysis. More Catchment types can be added and/or the coefficients can be changed specific to the project need.



RATIONAL METHOD	
Catchment Area	Runoff Coefficent
CT_Unimproved Areas	0.2
CT_Business-Commercial	0.75
CT_Pavement	0.9
CT_Residntial Heavy	0.6
CT_Residential Light	0.4
CT_Parks Cemeteries Playgrounds RR-yard	0.35

Figure 37 Runoff Coefficients List

2.4.3 Time of Concentration

Time of Concentration (Tc) is defined as the time required for a particle of runoff to flow from the hydraulically most distant point to the outlet or design point (catch basin etc.). Factors affecting the Tc are length of flow, slope of flow path and roughness of flow path.

Please see the CTDOT Drainage Manual - Chapter 6 for more detail.

Tc Input Types:

- *User Defined Tc* -where the user inputs minutes for Tc or
- *Composite Tc* -where the user inputs a data collection of Tc's (Tc Data Collection)

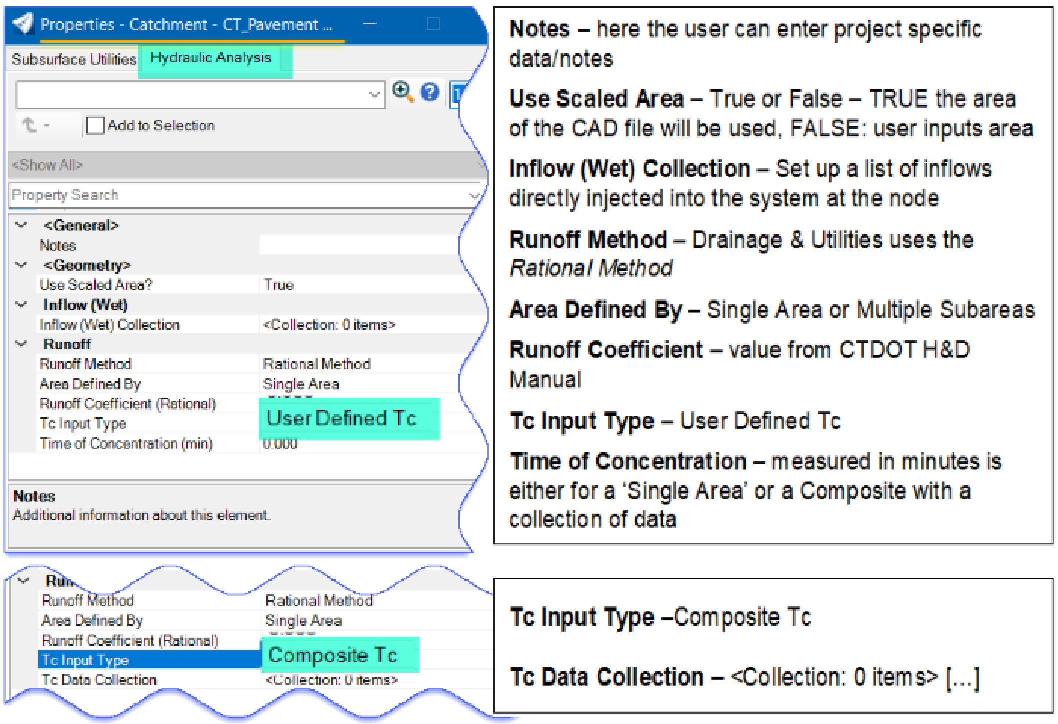


Figure 38 Time of Concentration Information

2.4.4 Time of Concentration using Tc Data Collection

This follows the CTDOT Drainage Manual by using the **TR-55 Sheet Flow**, **Shallow Concentrated Flow** and **Channel Flow** Tc Data Collection method.

SHEET FLOW

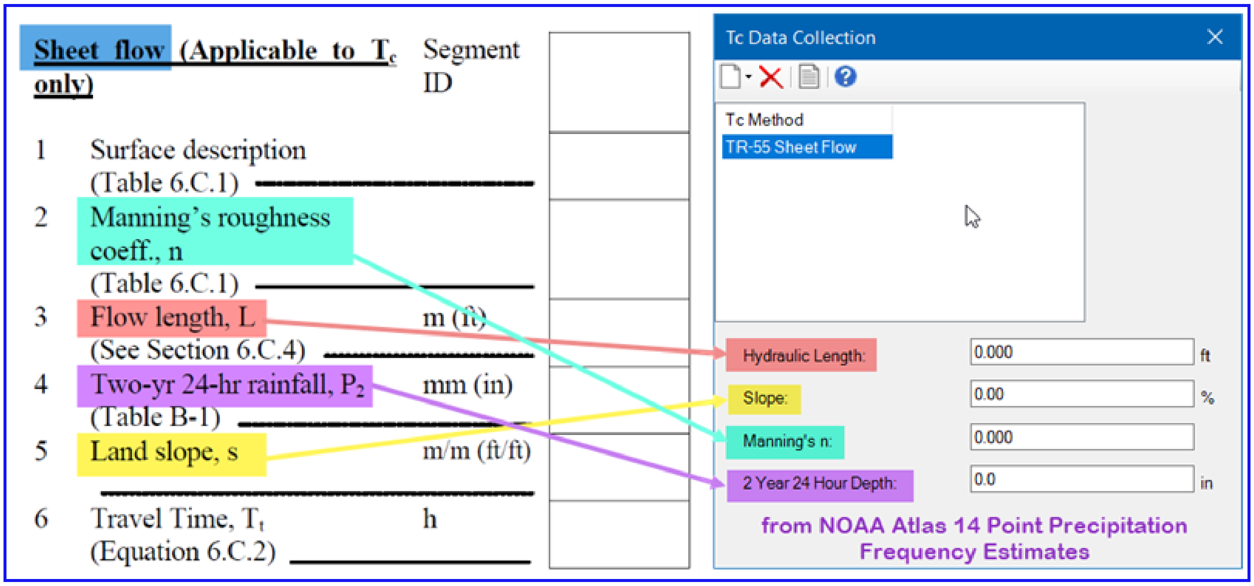


Figure 39 Sheet Flow

SHALLOW FLOW

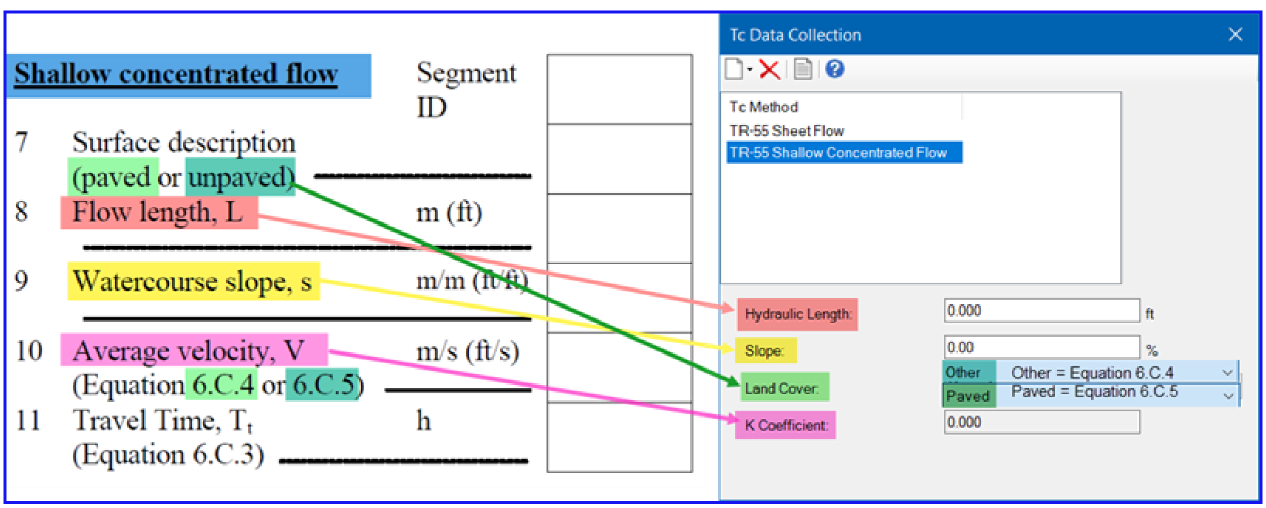


Figure 40 Shallow Flow

CHANNEL FLOW

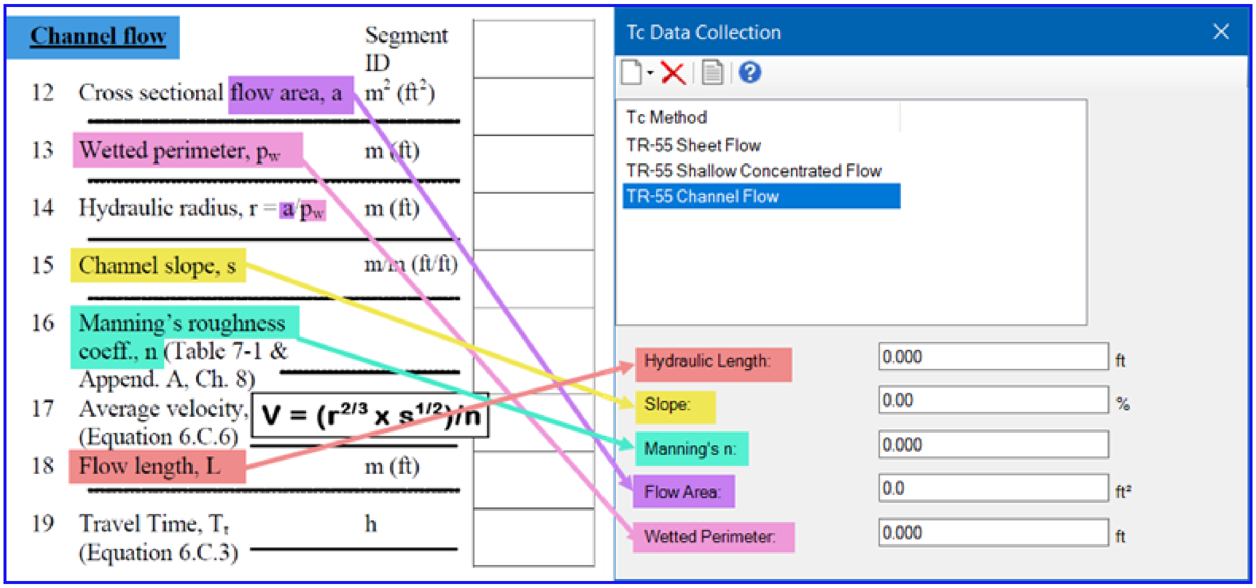


Figure 41 Channel Flow

2.5 Placing a Gutter

Gutters are the method by which a catch basin bypass flow is assigned to a downhill catch basin (inlet). Gutters only represent the hydraulic relationship between the inlets and are not accurate 2D and 3D models.

To display and add gutters to a drainage system:

- ***Turn Off*** Annotation Scale Lock (it is usually turned on by default): ***Drawing Production tab – Drawing Scales tools – Annotation Scale Lock***
- Change the display in the 2D view to use **Analytic Symbology.** In View 1, Default (2D view) click on ***View Attributes***, ***check off 'Text'*** under Presentation, then expand the ***Analytic Symbology*** (if not expanded) and ***check on "Use Analytic Symbology"***. This will turn on "labels" in the plan view (2D view only). If the labels do not show, refresh the view.

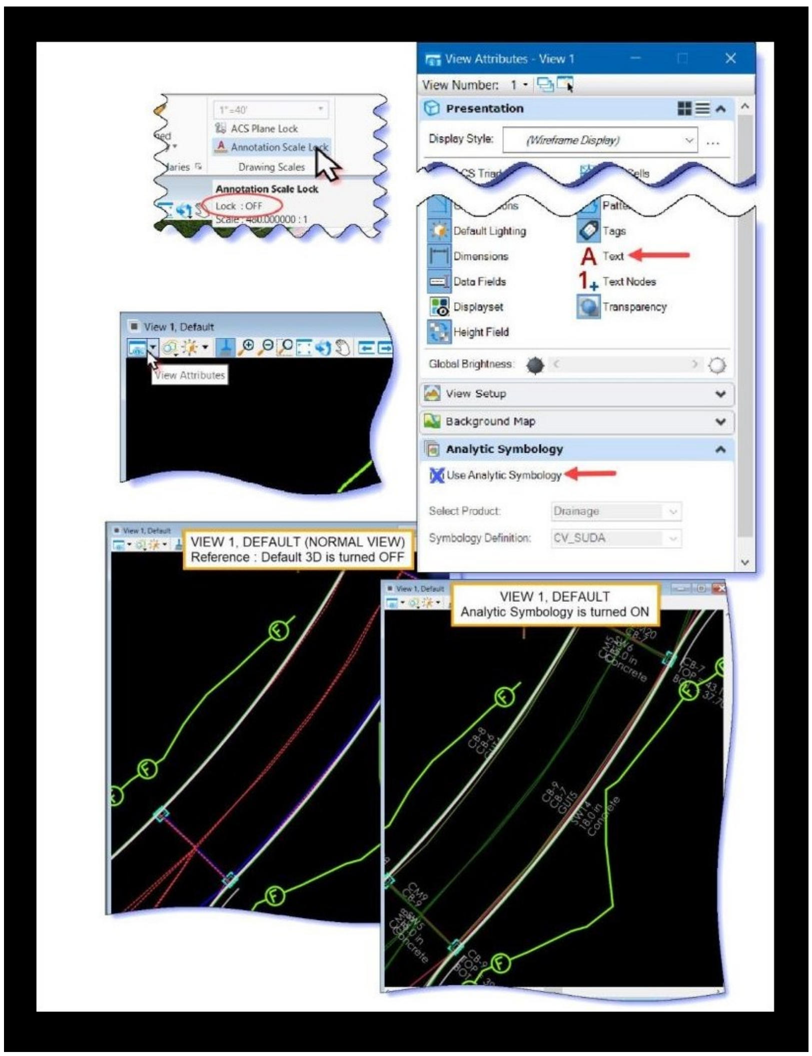


Figure 42 Place a Gutter Setup

There are four (4) gutters available that correspond to the **CTDOT Drainage Manual, Chapter 11 – Storm Drainage Systems, 11.9 Gutter Flow Calculations**:

- 11.9.3 - Uniform Cross Slope Procedure and
- 11.9.3 - Full-Super Uniform Cross Slope Gutter
- 11.9.4 - Composite Gutter Sections Procedure and
- 11.9.5 - V-Type Gutter Sections Procedures.

The engineer (user) has to decide which gutter best fits their project and edit as needed.

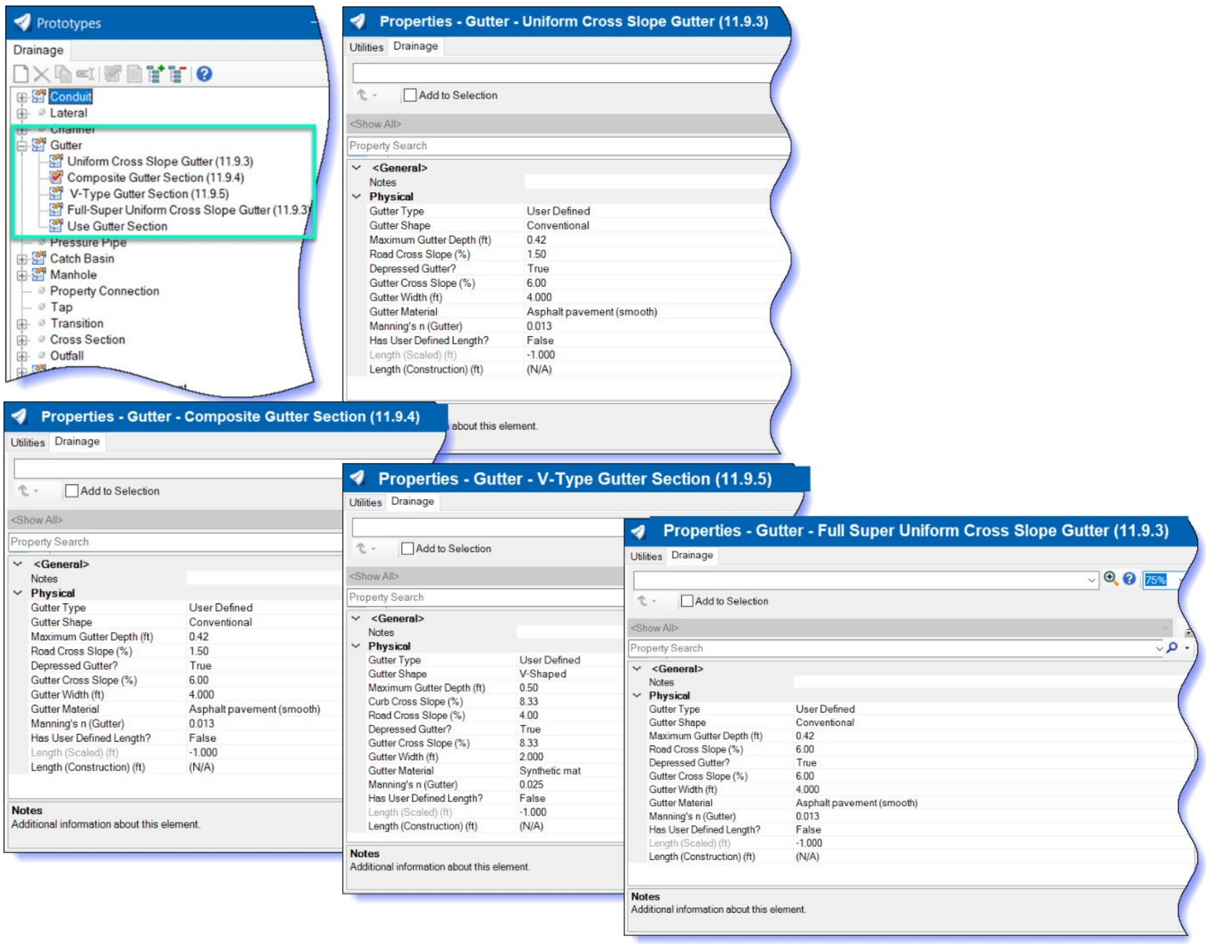


Figure 43 Gutter Properties

1. From the **Layout** tab, select the **Place Gutter** tool.
2. From the ***Feature Definition*** select the Gutter needed for your project.
3. Example: **Uniform Cross Slope** Gutter for proposed, **Uniform Cross Slope eGutter** for existing.

4. Follow the prompts: ***Select Start Node***.

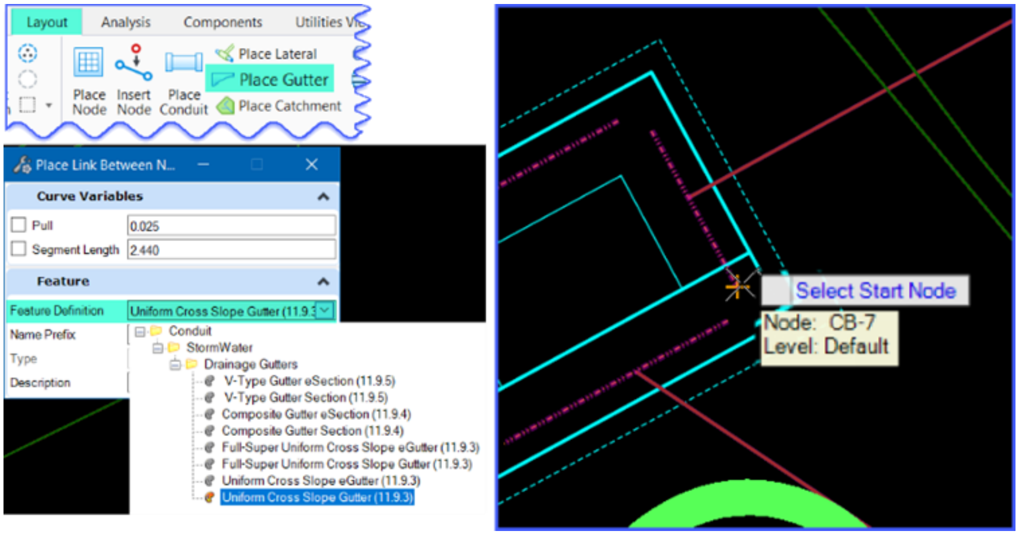


Figure 44 Place Gutter Criteria

5. ***Select next node to make connection***. <Alt> to place curve. <Ctrl> to place bends. <Reset> to select start node. Example select the next node.

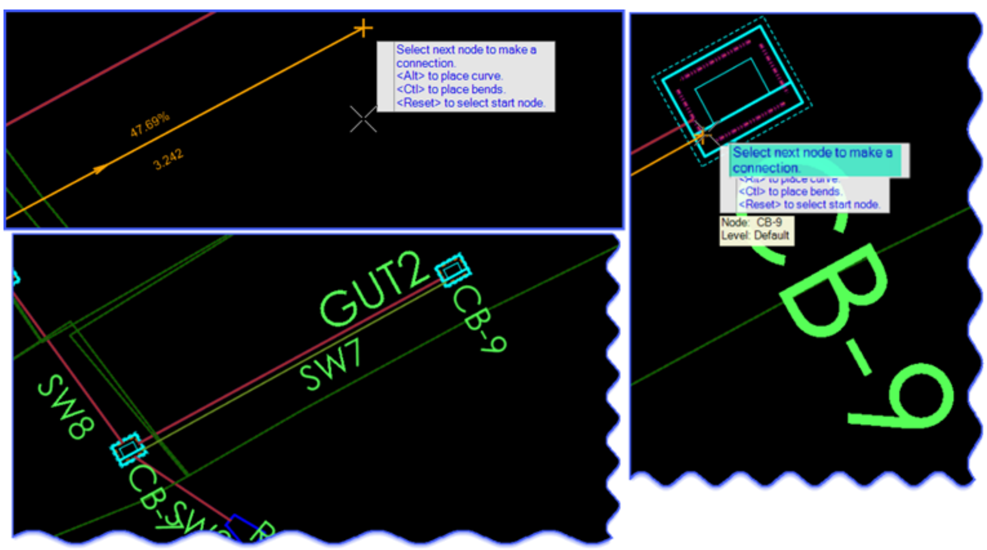


Figure 45 Place Gutter Criteria

6. After placing the gutters ***turn off*** the analytic symbology under view attributes.

2.6 Creating Storm Data

Storm Data is a single rainfall curve that represents one rainfall event for a given recurrence interval. Once the storm data is created it can be used globally by selecting Global Storm Events.

1. Click on the **Components** tab and click on the **Storm Data** tool. Several methods for specifying storm data are available, the ***User Defined IDF Table*** will be discussed and used.
2. Right-Click on ***User Defined IDF Table***, ***New***, ***User Defined IDF Table***.
3. Right-Click to rename the new User Defined IDF Table; example ***IDF_Table_Prj_No_XXX_YYYY***

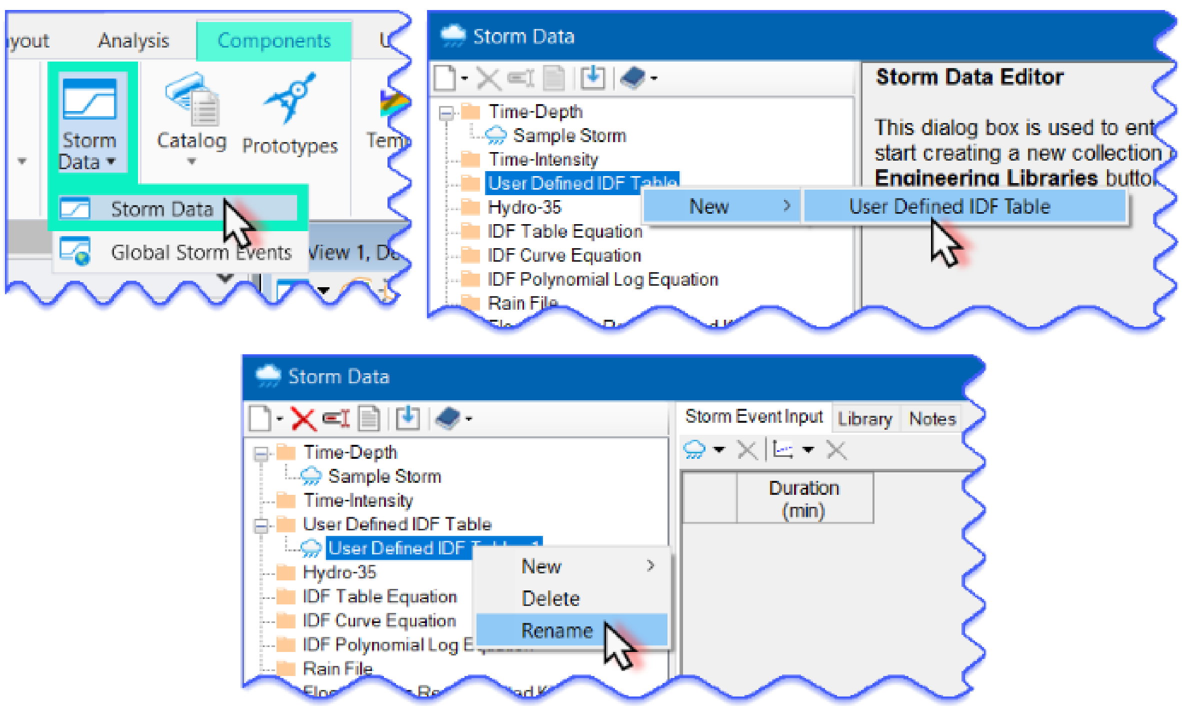


Figure 46 Strom Data Dialog Box

There are two options for the precipitation frequency estimates to be imported or used: **Importing an IDF Table from a File** or **Creating a New IDF Table using the Dialog**.

2.6.1 Create IDF-Table by importing CSV-file

- Use Excel to open the PF_Intensity_English_PDS.csv file saved from NOAA Atlas 14.
- You need to edit the file to only show the columns for durations and the rows for minutes.
- It is recommended to edit further by adding/interpolating the data for more minutes, follow Hydraulic and Drainage recommendations.
- To import an IDF table the data has to be in either the CSV-format or in a TXT-format.

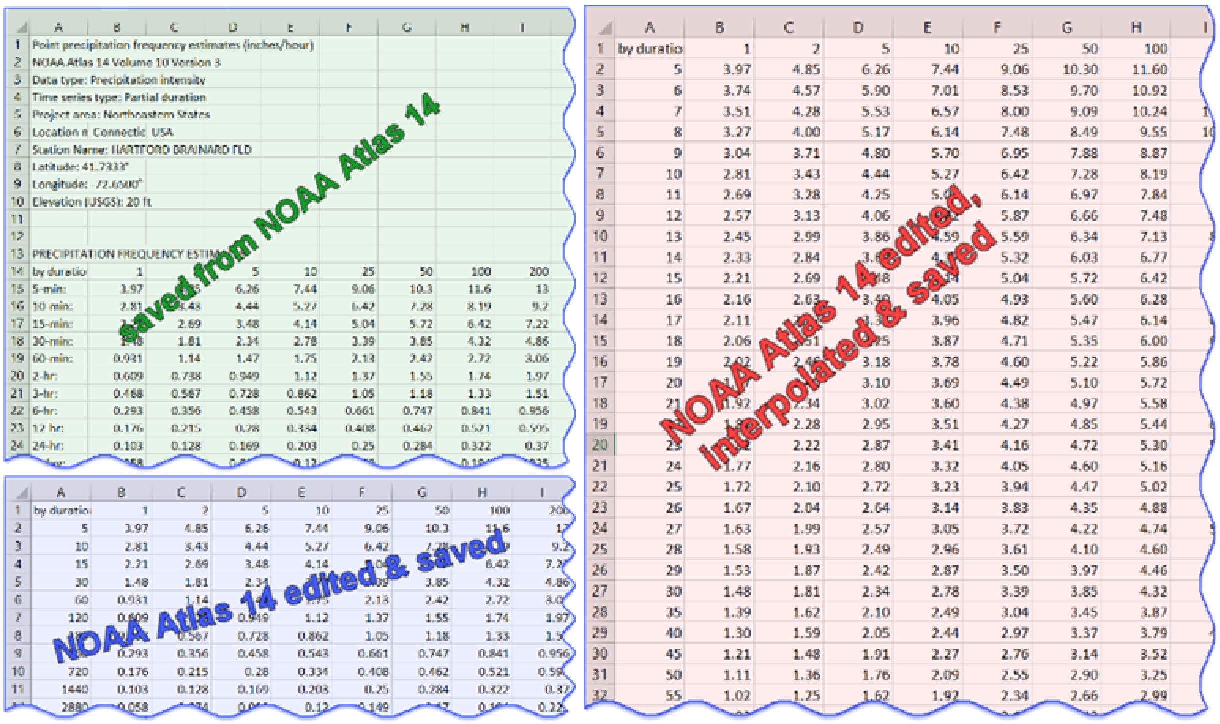


Figure 47 Create IDF-Table by importing CSV-file

1. In the Storm Data box ***select the user Defined IDF Table*** (IDF_Table_Prj_No_XXX_YYYY) and click on the **Import** icon. A Warning message will come up (Importing data will clear all ….) click **Yes**.

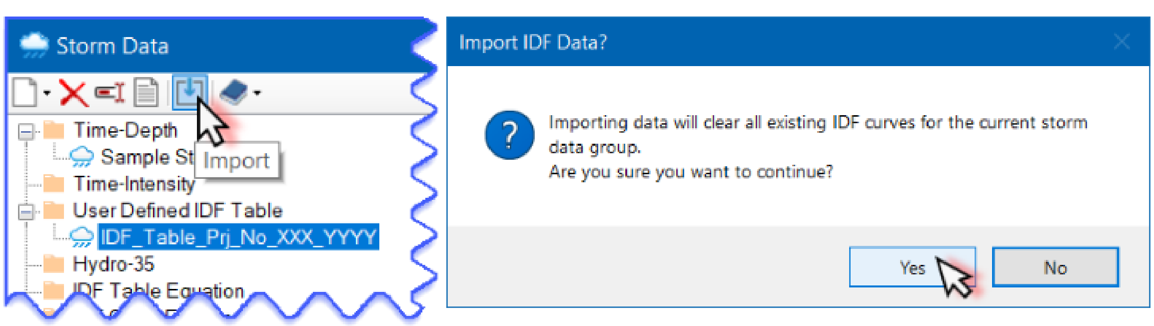


Figure 48 select the user Defined IDF Table

2. In the Explorer select and open the ***NOAA Atlas 14*** (edited & interpolated) ***CSV-file***. Imported data will be populated in the Storm Data window.

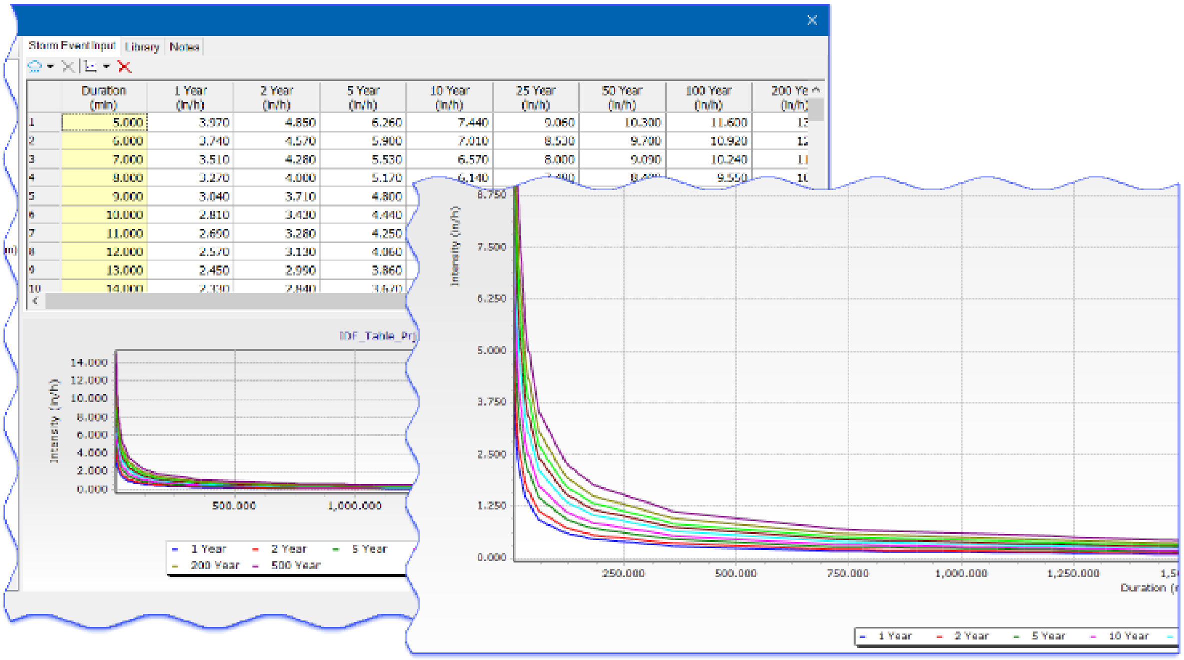


Figure 49 NOAA Atlas 14 CSV-file

3. **Close** the Storm Data box.

4. Select the Tools tab, click ***Options***, ***Drainage*** – this will open the ***Options*** box.

5. In the Options box click on the ***Units*** tab. Scroll down to the ***Rainfall Intensity*** and **ensure** that it matches the units in the IDF file. Click **OK**.

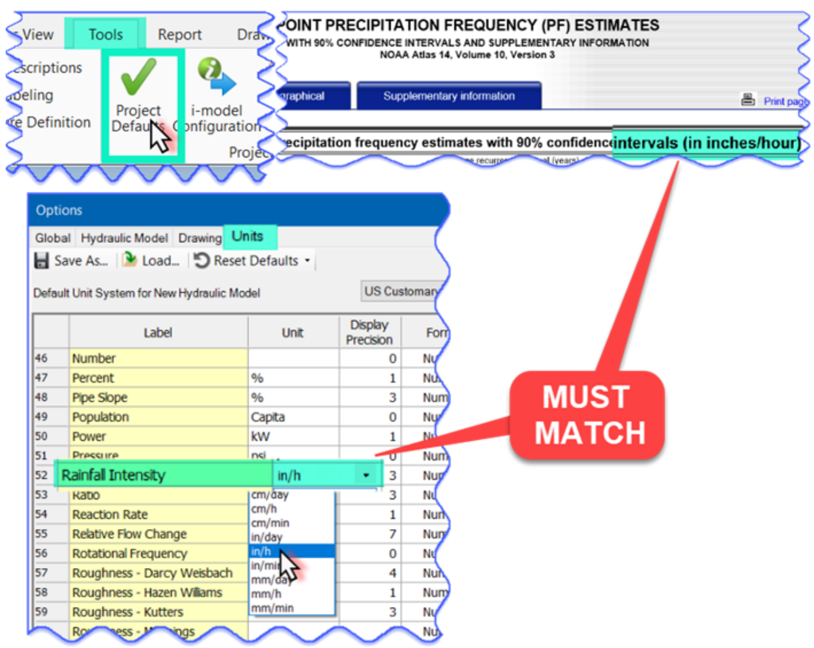


Figure 50 Drainage Options Match Intensity Units

2.6.2 Create IDF-Table Adding Return Periods and Durations (quick & dirty)

Note:

Merriam-Webster's Definition of quick and dirty: Expedient and effective, but not without flaws or unwanted side effects.

Figure 51 Definition of Quick and Dirty

Repeat steps 1 through 3 to create an IDF-Table. Instead of importing a CSV-file, we are adding data into the Storm Event Input.

- In the Storm Data box under the ***Storm Event Input*** tab, click on the down arrow for ***Add Range…***
- ***Add the years needed*** for your project; example: 2 enter, 5 enter, and so on for: 10, 25, 50, 100 years.
- *Click* on the down arrow for ***Add Duration*** and ***Add Range…*** Enter the data from the saved PF_Intensity_English_PDS.csv file.

Start in the first column, ***enter the data***, ***tab*** to the next column, and ***enter the data*** and so on.

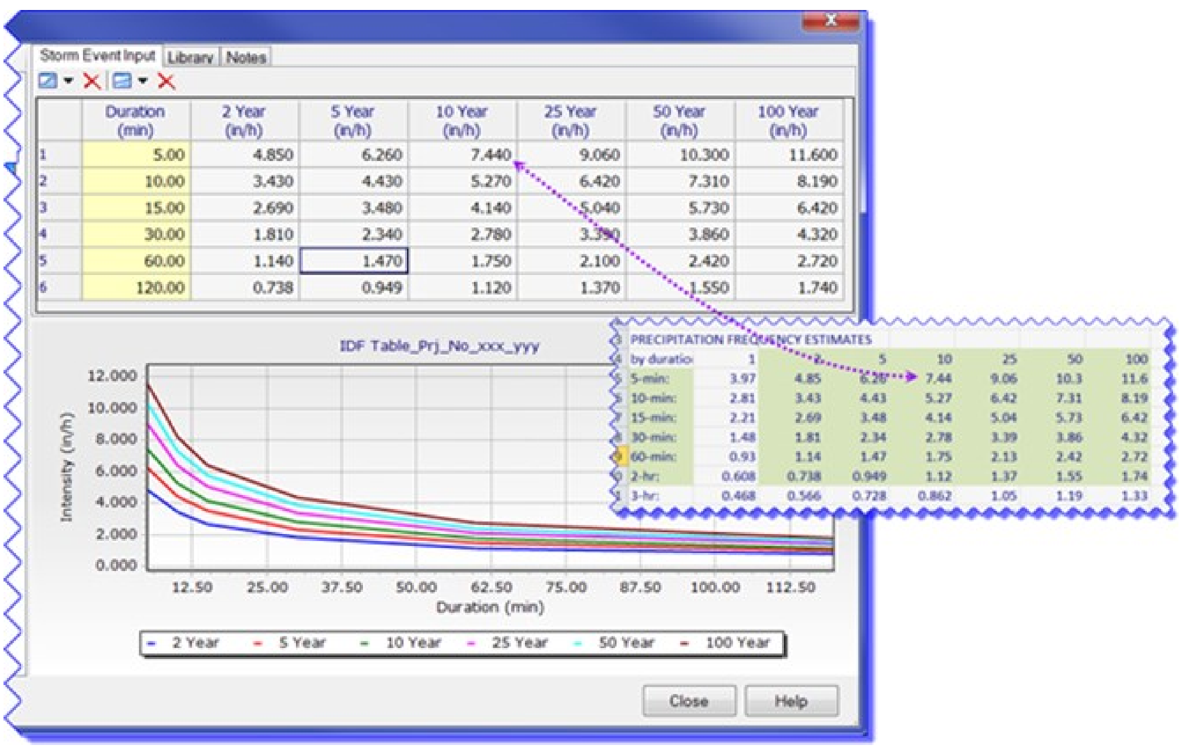


Figure 52 Storm Event Input

Exercise 3 - Drainage Computation

3.1 Setting Global Storm Events and Alternatives

Drainage and Utilities calculations are run on groups of settings called **Scenarios**, which have distinct subgroups of settings focused on aspects of drainage design or analysis. The subgroups are called **Alternatives** and include scopes as Physical Alternative, Design Option Alternatives and **Rainfall Runoff Alternatives** (focus here). Each *Rainfall Runoff Alternative* points to a **Storm Event** selected from the Storm Data collection (the IDF-Table imported previously).

Storm Events used for analysis and design of storm drainage systems are usually:

- 2-year for water handling
- 10-year for the drainage system and
- 25-year and/or 50-year for sag condition

(See Table 11-2 Pavement Drainage Design Criteria, Chapter 11 – CTDOT Drainage Manual)

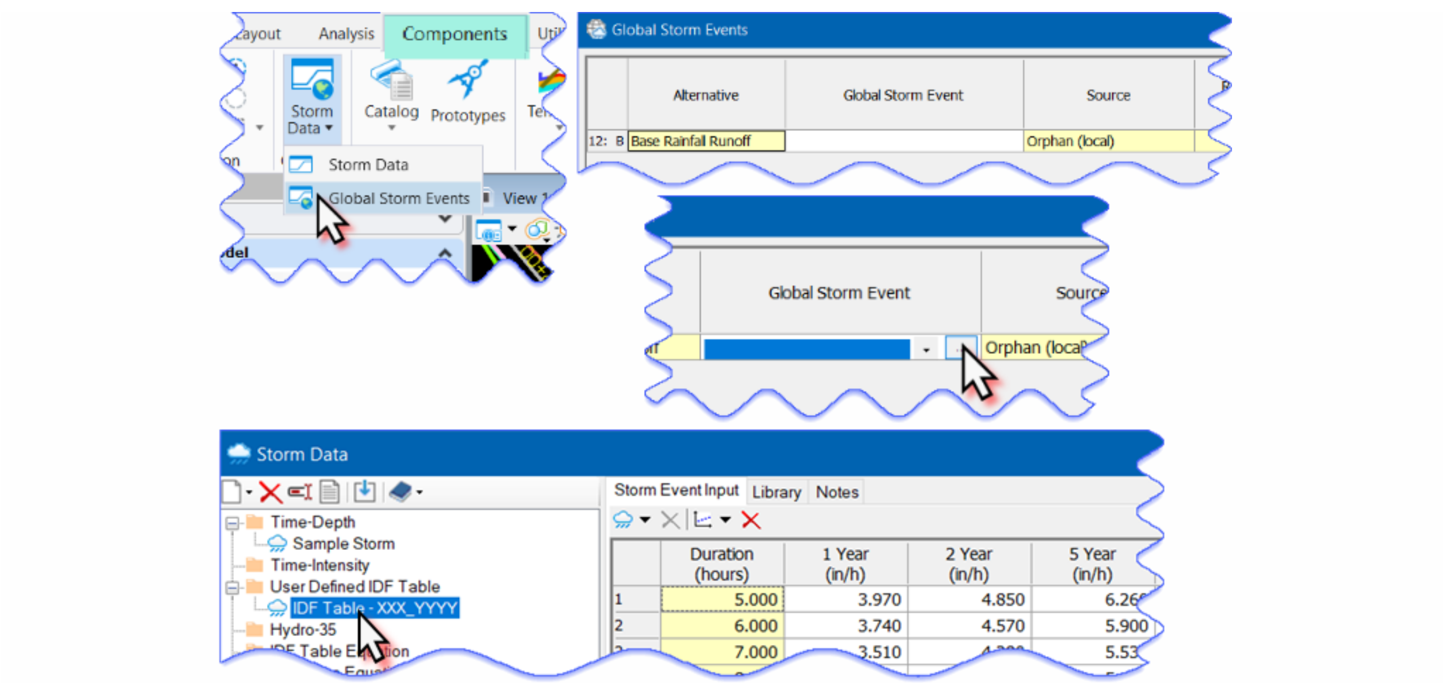


Figure 53 Storm Events

1. Select the **Components** tab, click on **Storm Data**, click on **Global Storm Events**. You should see one Alternative - *Base Rainfall Runoff*.

2. Click inside the ***Global Storm Event*** box for Base Rainfall Runoff.

3. Click on the ellipsis (…) and select the ***IDF_Table_Prj_No_XXX_YYYY*** created previously and the year you would want as your base rainfall runoff and click ***Close***.
4. ***Close*** the Global Storm Events box.
5. Select the **Analysis** tab, here tools are combined to calculate and report on the drainage scenarios.
6. The *Analysis* tab is split into three (3) sections: ***Calculation***, ***Analysis Tools*** and ***Analysis Views***. Most tools in the *Calculation* section are also available in the **Compute Center** box from the *Analysis Tools* section.

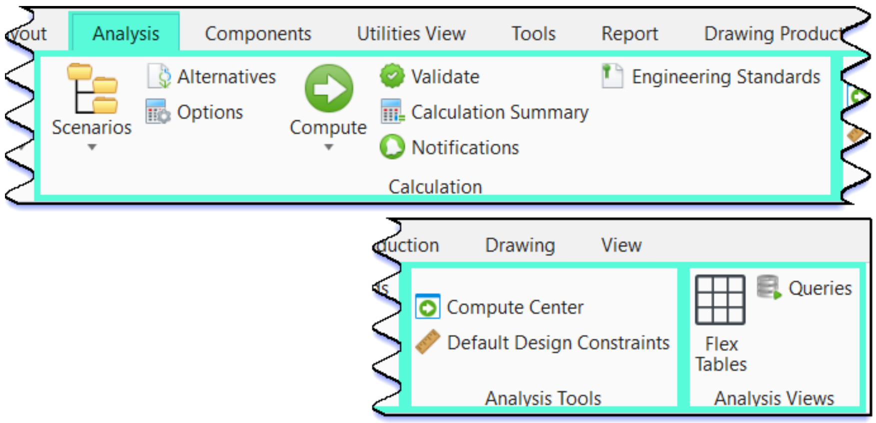


Figure 54 Analysis Tools - Compute Center

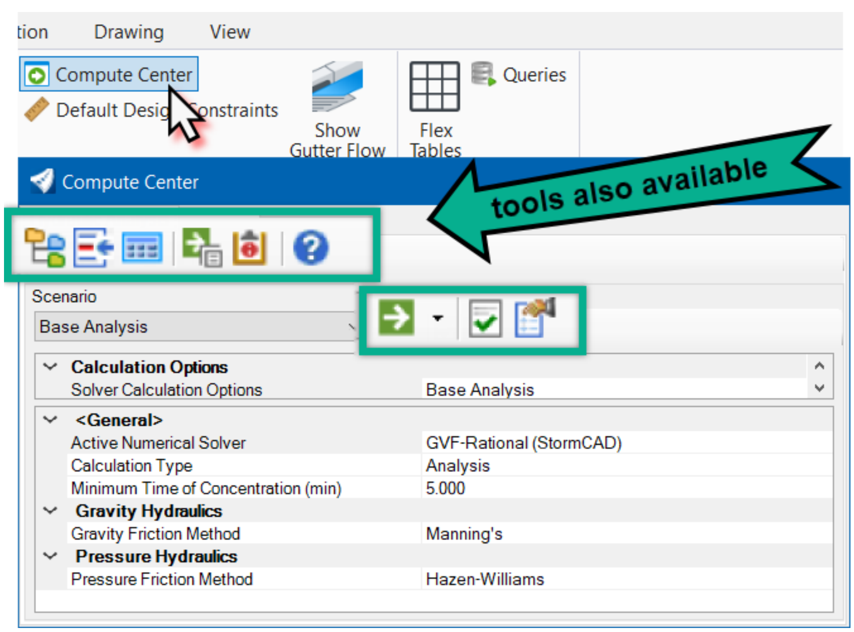


Figure 55 Compute Center tools

Select the **Scenarios** icon, there are two Scenarios: ***Base Design*** and ***Base Analysis***.

- Base Design will resize pipes, change inverts to *Default Design Constraints.*
- Base Analysis will not resize pipes or change inverts but will check if pipes and inverts conform to *Default Design Constraints*. Any discrepancies will be shown in '*User Notifications'* after verified or computed.

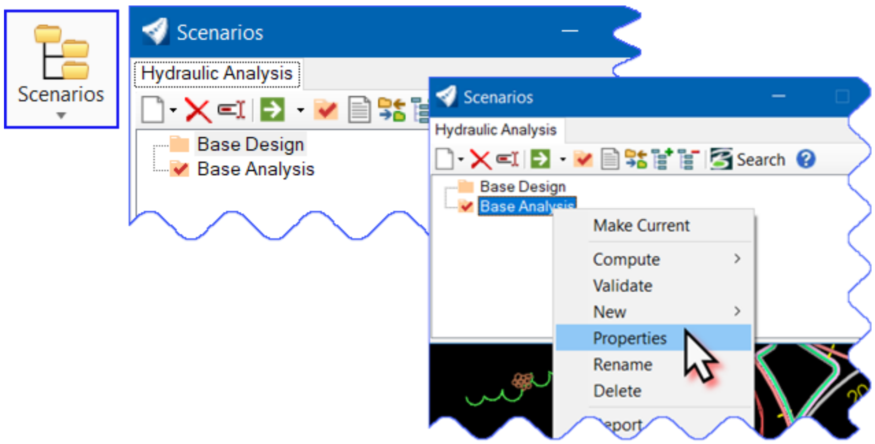


Figure 56 Scenarios Properties

1. Click on the **Scenarios** icon for the Scenarios box to open.
2. Select the **Base Analysis, right-click** and click on **Properties**, click on the ***Hydraulic Analysis*** tab in properties.

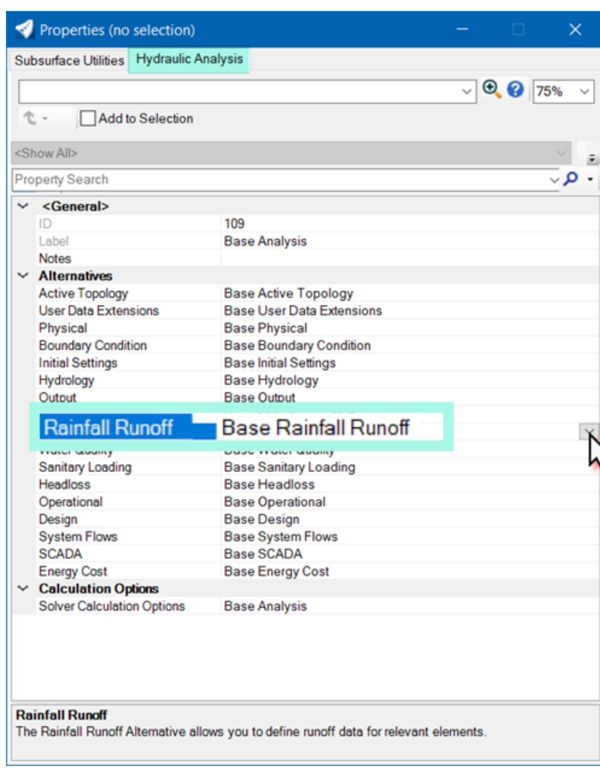


Figure 57 Hydraulic Analysis Rainfall Runoff setting

3. Click on ***Rainfall Runoff***, click on the ***down arrow*** next to Base Rainfall Runoff and click ***<New…>***.

4. In the **Create New Alternative** box enter a new rainfall alternative for your Project: ***Base Analysis - 2-yr Storm***, click **OK**.

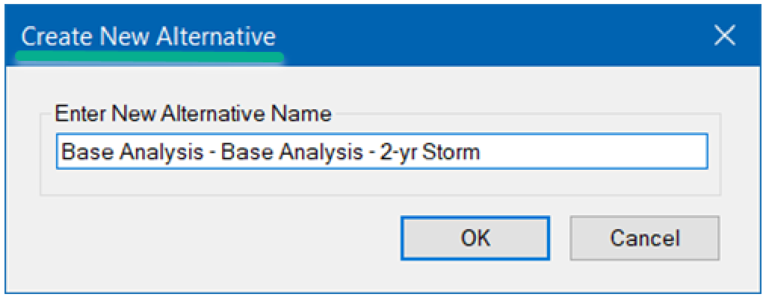


Figure 58 Create New Alternative

5. Select ***<New…>*** again add more Rainfall Alternatives as needed:

- Base Analysis – 10-yr Storm
- Base Analysis – 25-yr Storm
- Base Analysis – 50-yr Storm

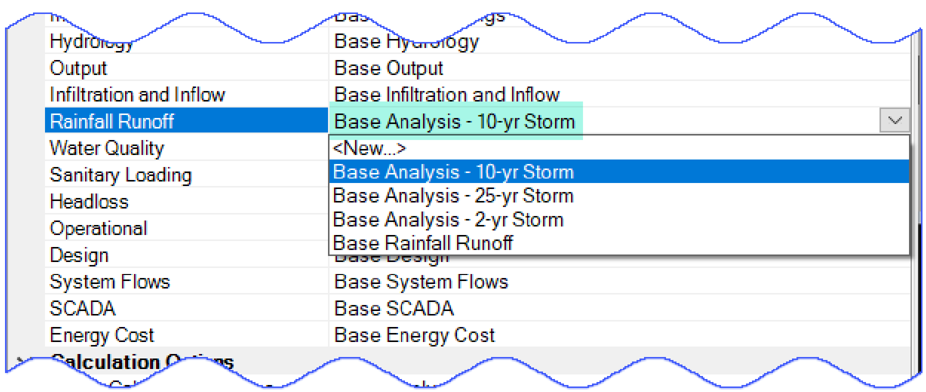
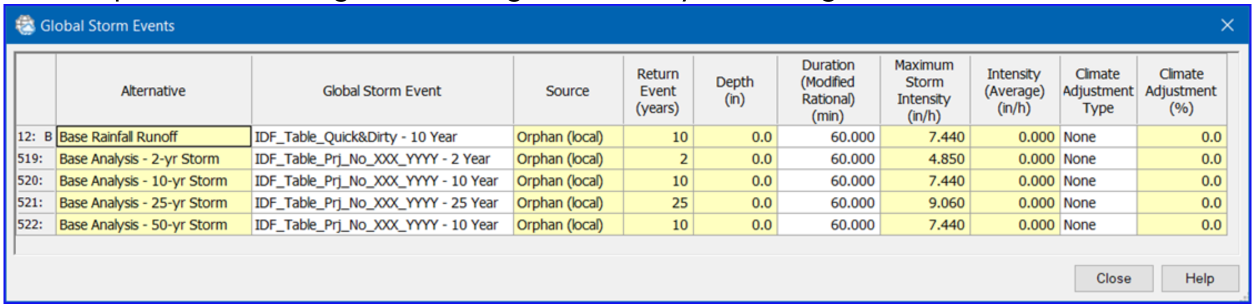


Figure 59 Create New Alternative Properties

6. Select the ***Base Analysis*** you want to run, example: ***Base Analysis - 10-yr Storm***. ***Close*** the properties box. ***Close*** the Scenarios box.

7. Select the ***Components*** tab, click on ***Global Storm Events***. There are now additional alternatives.

8. Change the ***Global Storm Event*** to the projects corresponding event storms (2-yr, 10-yr, 25-yr and 50-yr). For ***Base Analysis – 2-yr Storm*** click on down arrow and ***select: IDF Table_Prj_No_XXX_YYYY – 2 Year.*** Repeat for each Base Analysis and Storm.



	Alternative	Global Storm Event	Source	Return Event (years)	Depth (in)	Duration (Modified Rational) (min)	Maximum Storm Intensity (in/h)	Intensity (Average) (in/h)	Climate Adjustment Type	Climate Adjustment (%)
12: B	Base Rainfall Runoff	IDF_Table_Quick&Dirty - 10 Year	Orphan (local)	10	0.0	60.000	7.440	0.000	None	0.0
519:	Base Analysis - 2-yr Storm	IDF_Table_Prj_No_XXX_YYYY - 2 Year	Orphan (local)	2	0.0	60.000	4.850	0.000	None	0.0
520:	Base Analysis - 10-yr Storm	IDF_Table_Prj_No_XXX_YYYY - 10 Year	Orphan (local)	10	0.0	60.000	7.440	0.000	None	0.0
521:	Base Analysis - 25-yr Storm	IDF_Table_Prj_No_XXX_YYYY - 25 Year	Orphan (local)	25	0.0	60.000	9.060	0.000	None	0.0
522:	Base Analysis - 50-yr Storm	IDF_Table_Prj_No_XXX_YYYY - 10 Year	Orphan (local)	10	0.0	60.000	7.440	0.000	None	0.0

Figure 60 Global Storm Event

9. *Select* the ***Analysis*** tab and select the ***Scenarios*** icon. Right-click on ***Base Design*** and select ***Properties***. Click on the ***Hydraulic Analysis*** tab.
10. Change the ***Rainfall Runoff*** to the ***Base Analysis*** as previously. Example: ***Base Analysis – 10-yr storm***. ***Close*** the properties box.

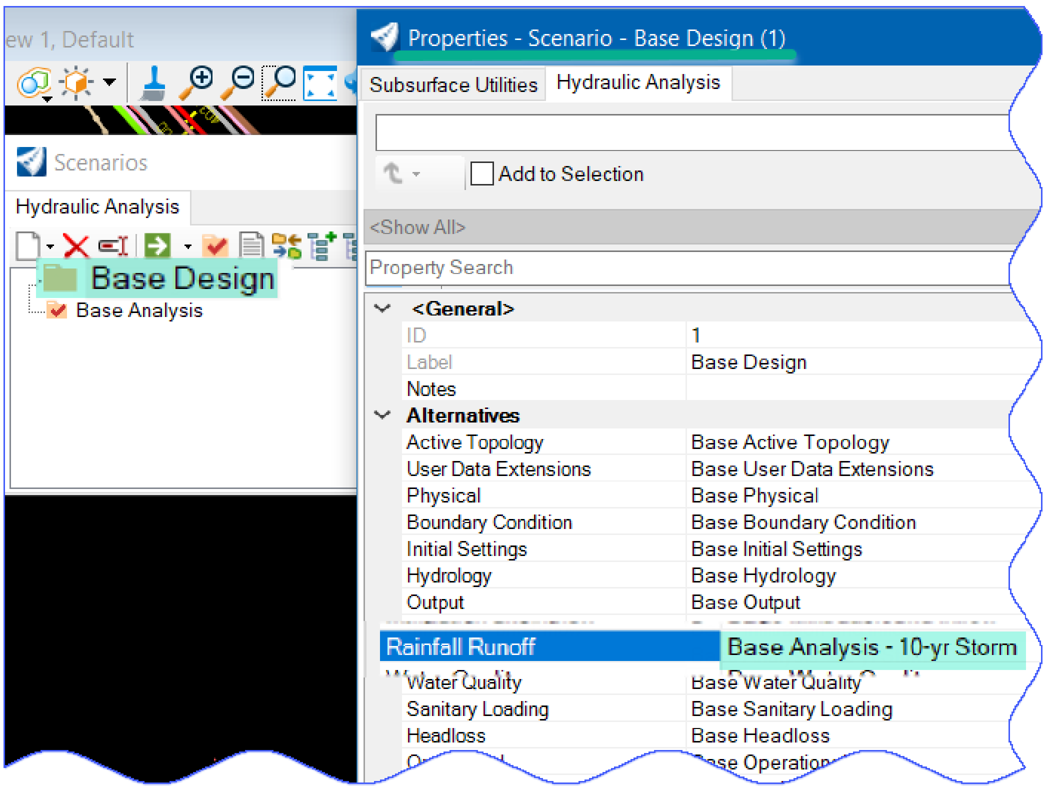


Figure 61 Properties – Scenario – Base Design

11. This is the minimum preparation needed to **Compute** the storm drainage system(s) for a project.

3.2 Validate

The **Validate** command is in the ***Analysis*** tab – ***Calculation*** section and within the ***Compute Center*** menu box. The **Validate** command will run a diagnostic check on the network data (drainage system) and alert you to possible problems that could be encountered during the calculations. This is a manual validation and checks only for input data errors.

The **Compute** command is the automatic validation that Drainage and Utilities runs when it is initiated, and it will check for input data errors and for network connectivity errors as well as many other things beyond the manual validation checks. It is a good idea to first check (validate) the drainage system for any input data errors or missing items/data especially when there is a large system. This may need to be done multiple times, because some errors may be hidden after the initial validation.

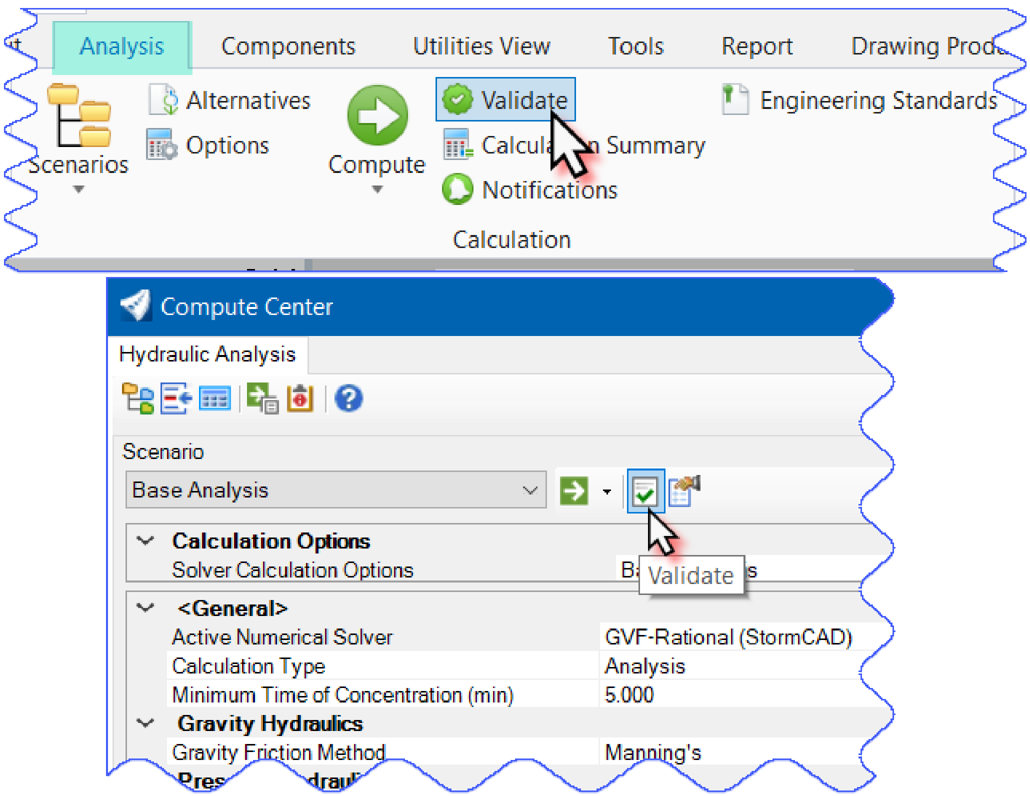


Figure 62 Validate

Click on the ***Validate*** command, you will/may get a message for errors or warnings: Click ***OK***.

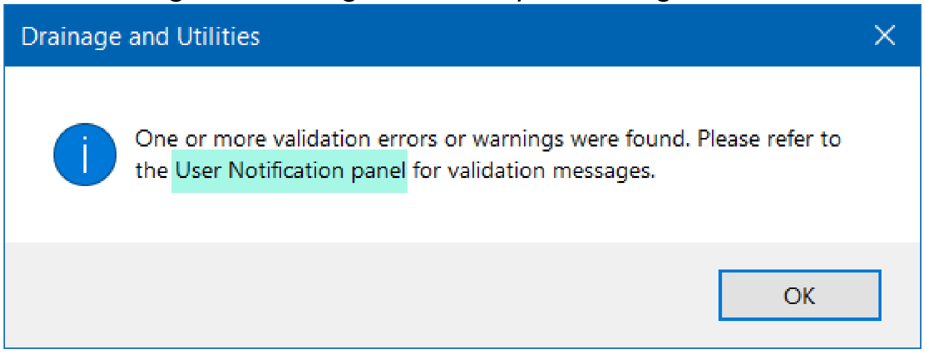


Figure 63 Validate Warning

After the validation is complete you will probably see two different kinds of markers in your 2D view window (usually View 1, Default).

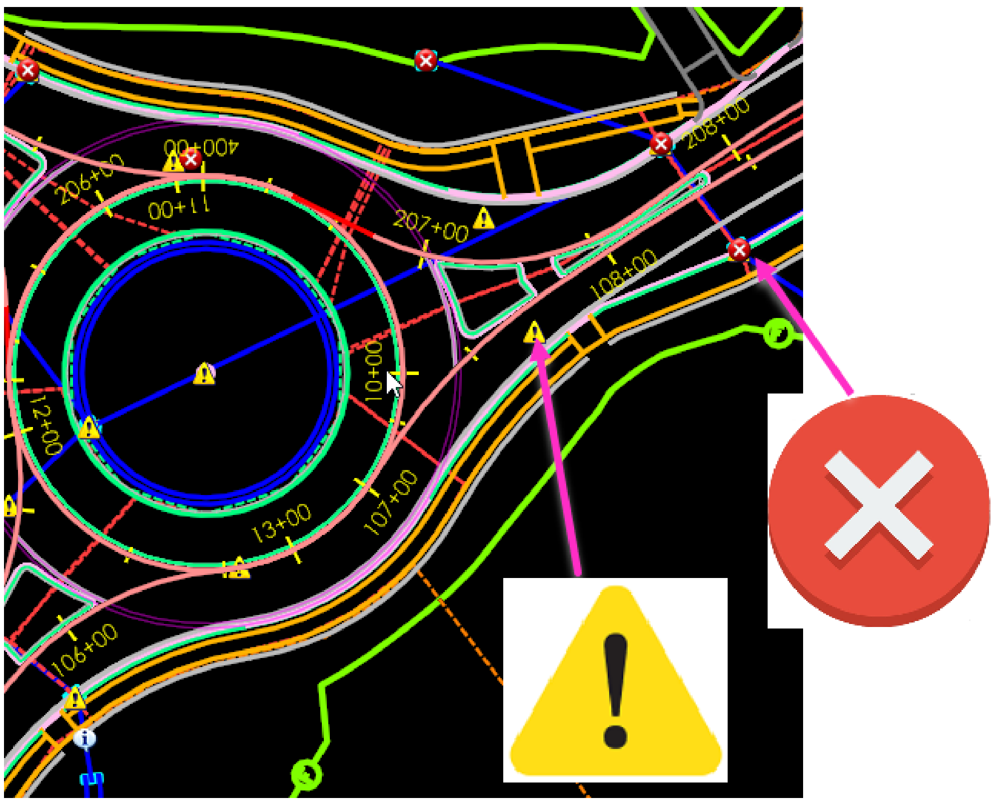


Figure 64 Validate Symbols

Click on ***Notifications*** or the ***User Notification*** icon in the Compute Center.

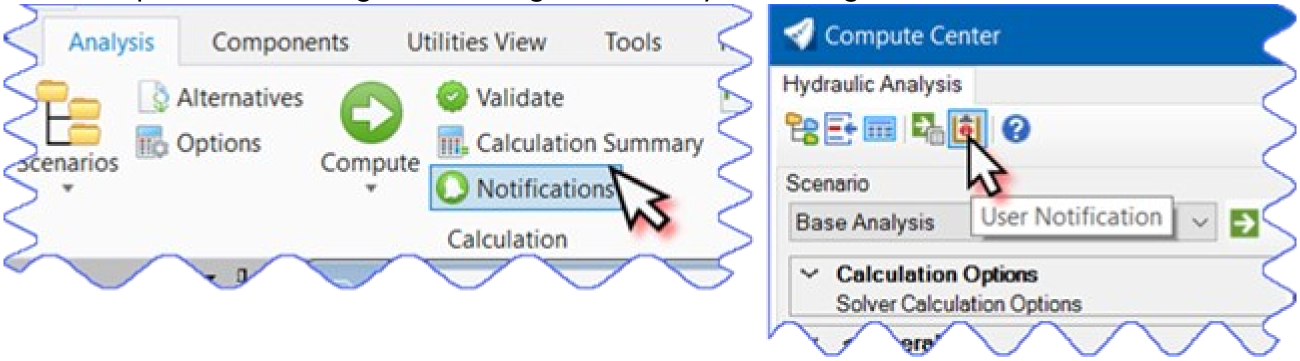


Figure 65 User Notification

The notifications window will come up displaying a list of warnings generated during the validation process. There are two types:

Red error **must** be corrected before the model (system) can be computed.

Yellow **does not** prevent successful calculation of the model, but element does not meet certain criteria.

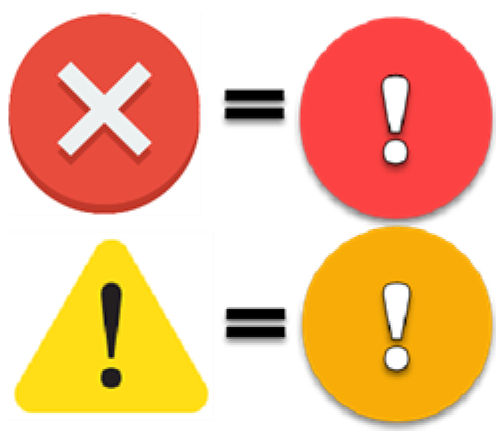


Figure 66 User Notification Symbols

In the user Notification box, you will see messages with the **RED** and/or **YELLOW** icon as well as a toolbox (see below).

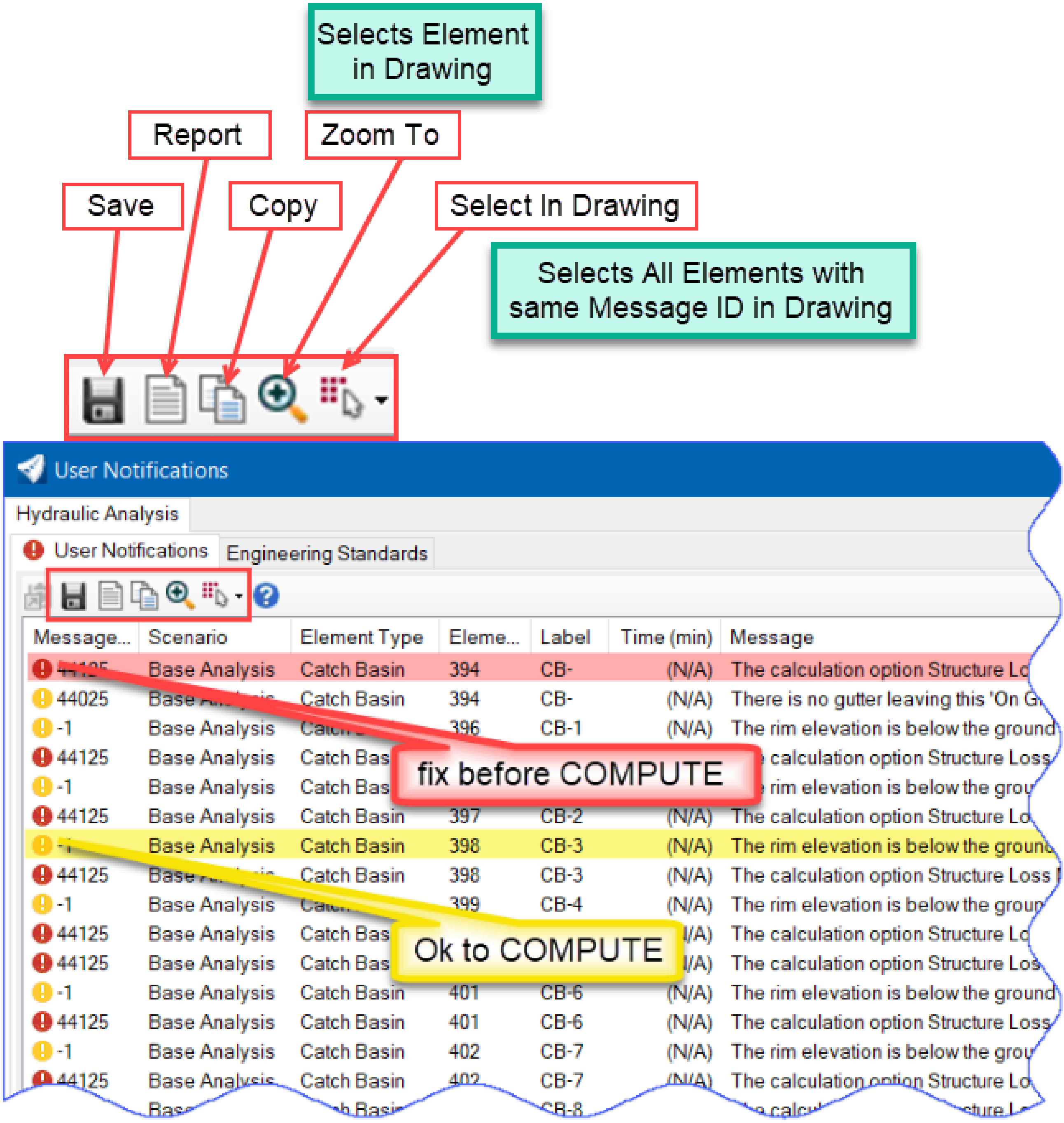


Figure 67 User Notification List

Example of User Notifications:

- Message: The calculation option Structure Loss Mode must be set to EGL in order to use the HEC-22 Energy (Third edition) Headloss Method.
- Meaning: Inlet calculations use the Headloss Method: HEC-22 Energy (3rd Edition) to use this, the Structure Loss mode must be set to the Energy Grade Line (EGL) in the Calculation Options.

- Solution: Analysis tab, Options, Right-Click on Base Analysis, Properties. In the properties box, select Structure Loss Mode and select Energy Grade.

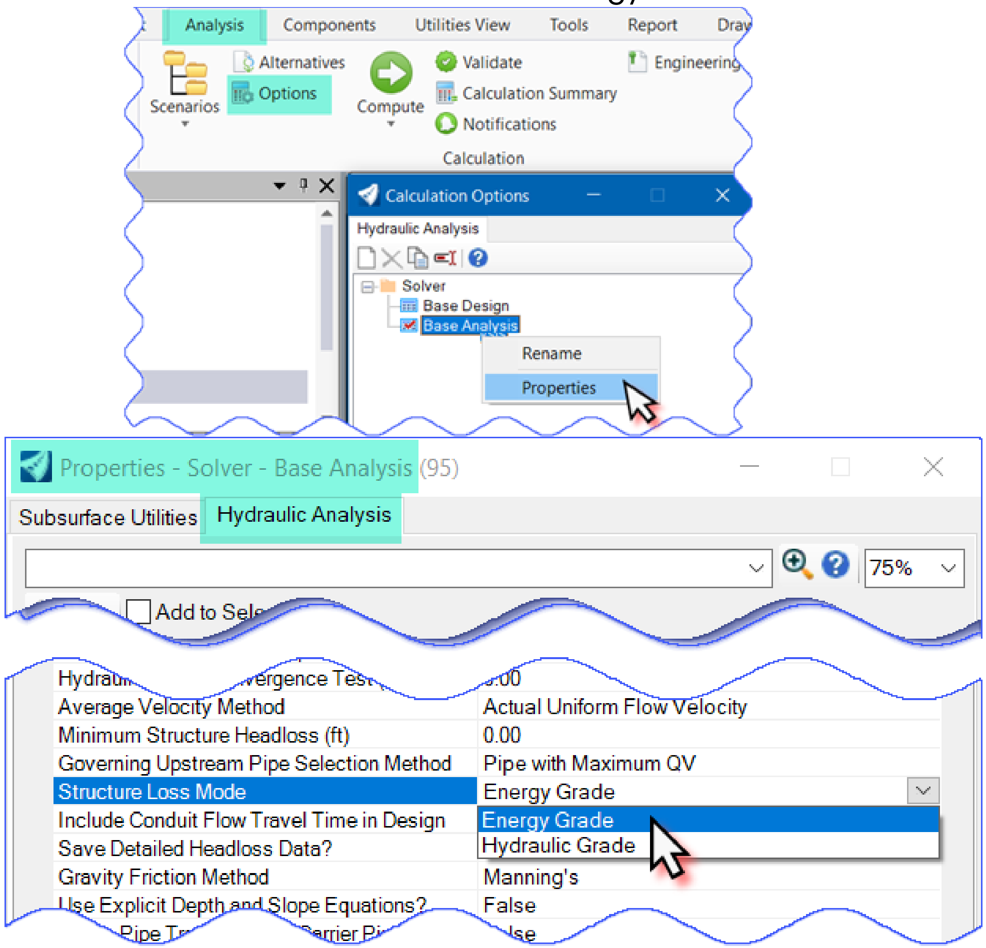


Figure 68 Example of User Notifications

Some more examples:

Message — Meaning — ***Solution***:

There is no gutter leaving this 'On Grade' catch basin. Bypassed flow is directed to the subnetwork outfall. — The catch basin doesn't have a gutter attached. — **Open the Analytical View and place a gutter to direct the flow to next catch basin or system. If going to overland flow (no curbing) you can disregard.**

Conduit does not meet minimum cover constraint. — The min. cover that is set in Default Design Constraints is not met. — ***Change the inverts, or size or add barrel if possible, or ignore for calculations and use Class V RCP for callouts and estimating.***

Structure and pipe inverts do not agree with selected benching type. — Benching method is set to HEC-22 Benching Method = Flat in accordance with the CTDOT Drainage Manual. — ***You can disregard this message.***

3.3 Computation and Results

3.3.1 Compute

You are ready to compute after all **RED** issues are resolved and all **YELLOW** issues are modified or considered.

1. Click the ***Compute*** icon. Drainage and Utilities will run the calculations, you will see some windows pop-up, wait until the ***GVF-Rational Calculation Summary*** box opens.

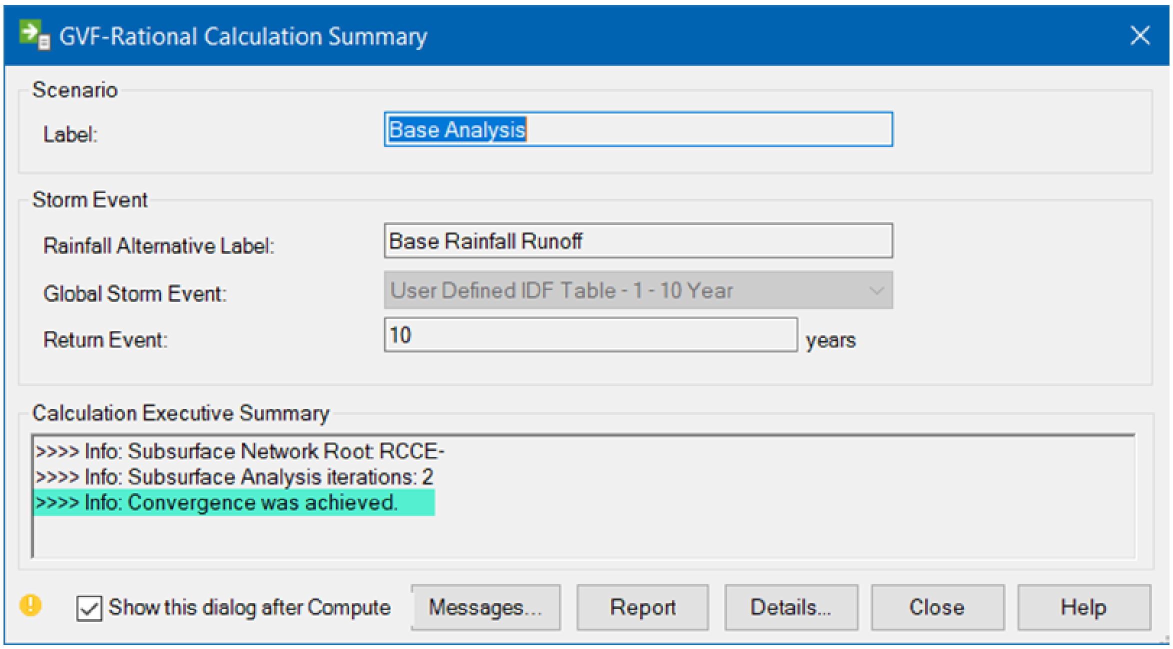


Figure 69 GVF-Rational Calculation Summary

2. Calculations were successful if you see: ***Convergence was achieved*** in the Calculation Executive Summary box.
3. Click on ***Messages…*** to see the newest User Notifications, if there is an error, the calculations will stop at that point, you will have to solve the error and rerun calculations.
4. Click on ***Report*** and for the **Scenario Summary** preview, this can be printed.
5. Click ***Details…*** the ***Calculation Detailed Summary*** box will open showing:
   - Calculation Options
   - Catchment Summary
   - Link Summary
   - Node Summary
   - Inlet Summary
   - Pond Summary

6. Click on **Report** to get one combined report to ***review/print/save***. For individual and more complex reports use ***Flex Tables***.

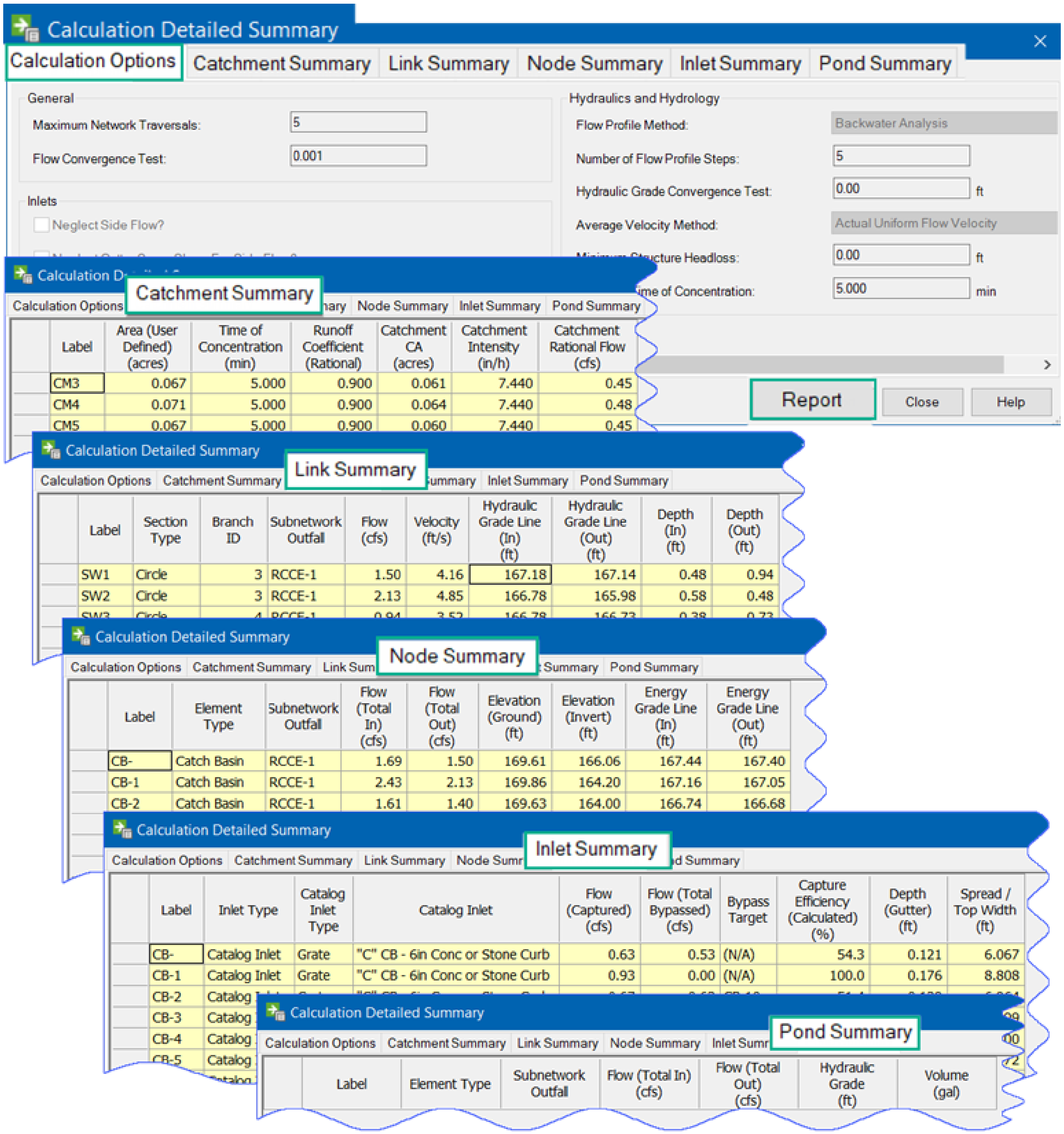


Figure 70 Calculation Detailed Summary

3.3.2 Flex Tables

Drainage and Utilities shows the engineering data in tabular format called **Flex Tables**. The most common flex tables have been established as **CTDOT Flex Tables** have been formatted to follow the [CTDOT Drainage Manual](). Any flex table can be edited to best fit the project. The designer should use the drainage manual as a guide to determine the information needed to represent the drainage system.

Click the ***Flex Tables*** icon in the ***Analysis Views*** section. In the ***Flex Tables*** box click on the ***Hydraulic Analysis***t ab. Tables are characterized into **Hydraulic Model — Shared — Predefined**.

Drainage and Utilities has 'predefined tables' set-up that can be edited as needed. ***Expand*** the Tables – ***Hydraulic Model*** folder, here ***CTDOT Flex Tables*** are available for use and are formatted to show all information and data required. Should you have to edit to reflect information/data needed for the drainage computations, they still can be edited as needed specific to the project. ***Copy*** the ***flex table*** and ***rename*** project specific.

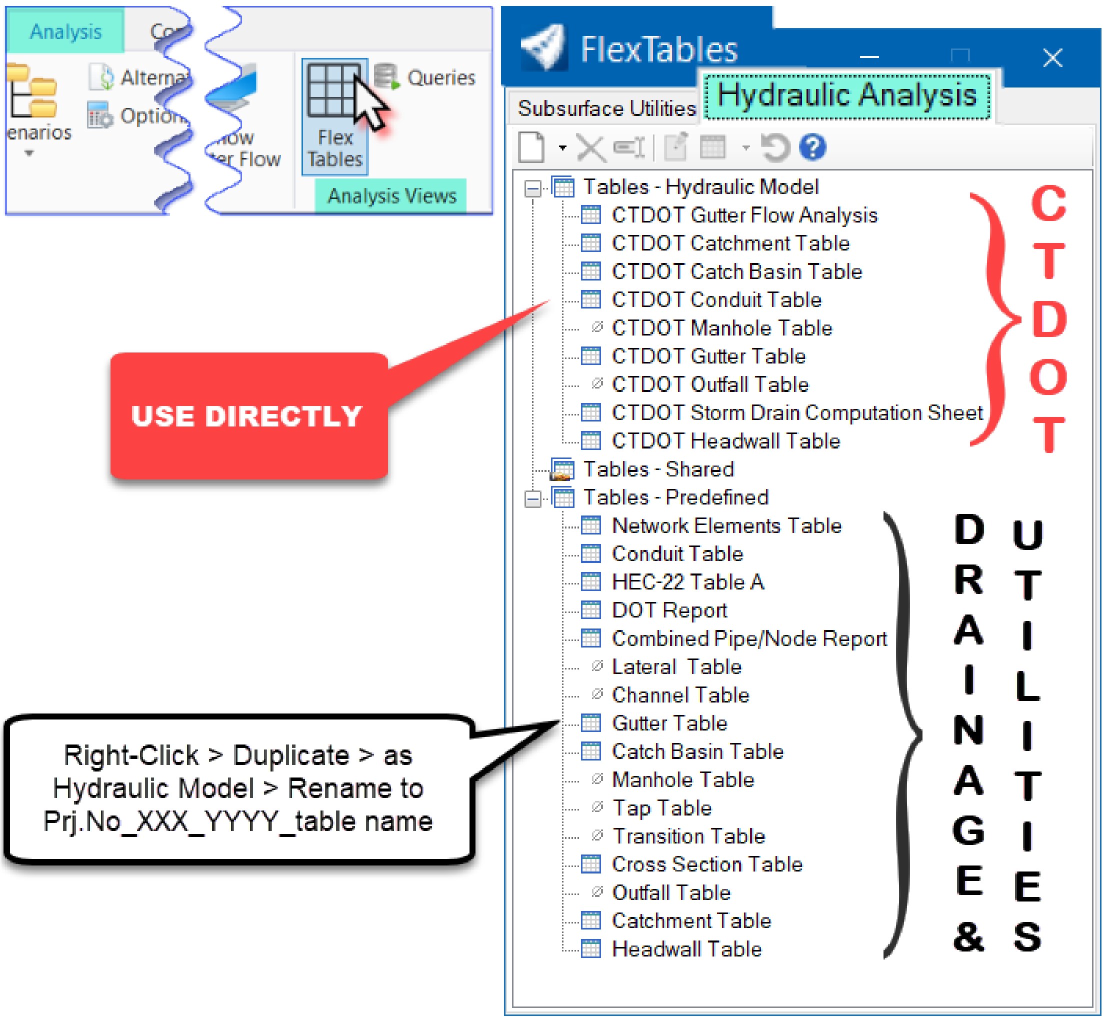


Figure 71 Flex Tables

CTDOT Gutter Flow Analysis, ***CTDOT Gutter Flow Analysis at Low Point*** and ***CTDOT Strom Drain Computation Sheet*** follows the computation sheets of the CTDOT Drainage Manual Chapter 11.

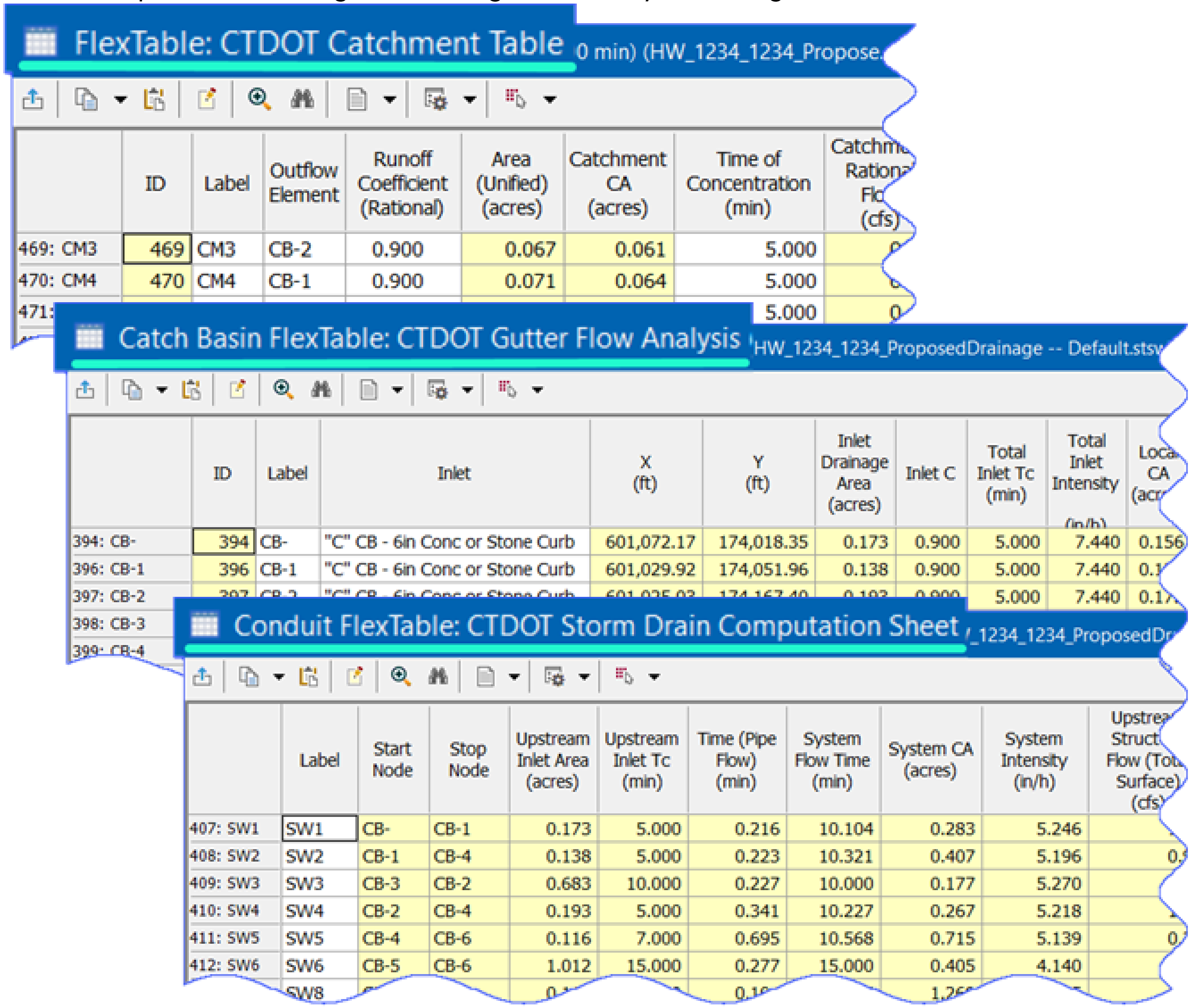


Figure 72 Flex Tables

Double-Click on a flex table to open it. Each table has various tools such as: export, copy, paste, edit, zoom to, report, options and select in drawing.

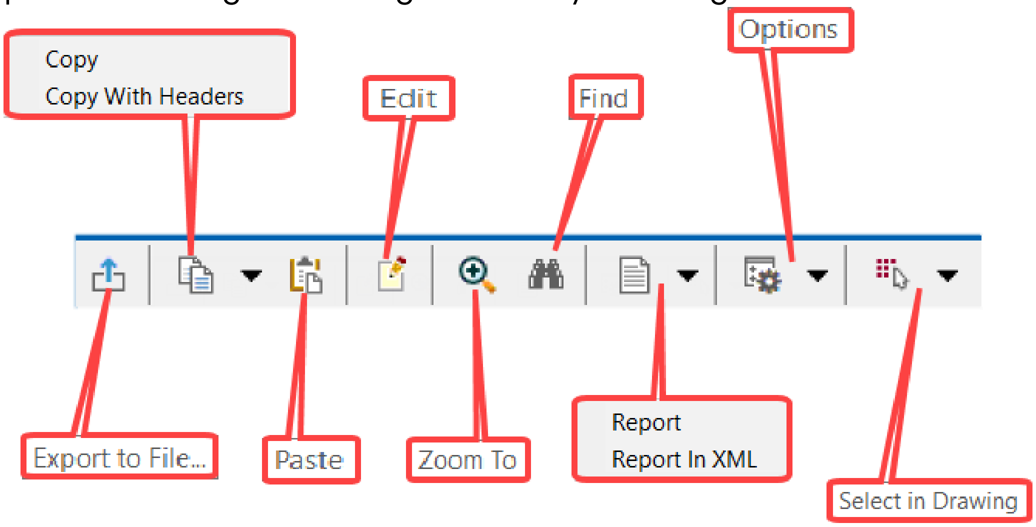


Figure 73 Flex Tables Tools

Export to File... — creates a text or csv file to work within Excel or shape file if needed.

Edit... — allows columns to be added or removed from the flex table. Example: ***scroll*** in the ***Available Columns to X***, click on ***Add***. Now X is added to the Selected Columns side and click ***OK***. X column is added to the catch basin flex table.

Zoom To... **—** ***highlight*** a row and click on ***Zoom To***, the active view will zoom in to the item/row selected and is highlighted.

Report... **—** creates a report that can be printed. Paper size and page set-up can be edited in the report page.

Options... **—** gives the ability to create selection sets.

Create Selection Set — can be used to filter various items. As example to split up the drainage data into left and right side of the roadway.

- Within the catch basin flex table ***select the catch basins for the left side of the roadway*** (highlight all left CBs using the CTRL key).
- Click on the down arrow next to Options and select ***Create Selection Set...,*** type in selection set name: ***RteABC-LT***. Click ***OK***.
- ***Repeat*** these steps for the catch basins on the right side of the roadway.
- Click on the ***Utilities View*** tab and click on ***Selections Sets***, click on ***Hydraulic Analysis*** tab. Here all Selection Sets for the project are stored.

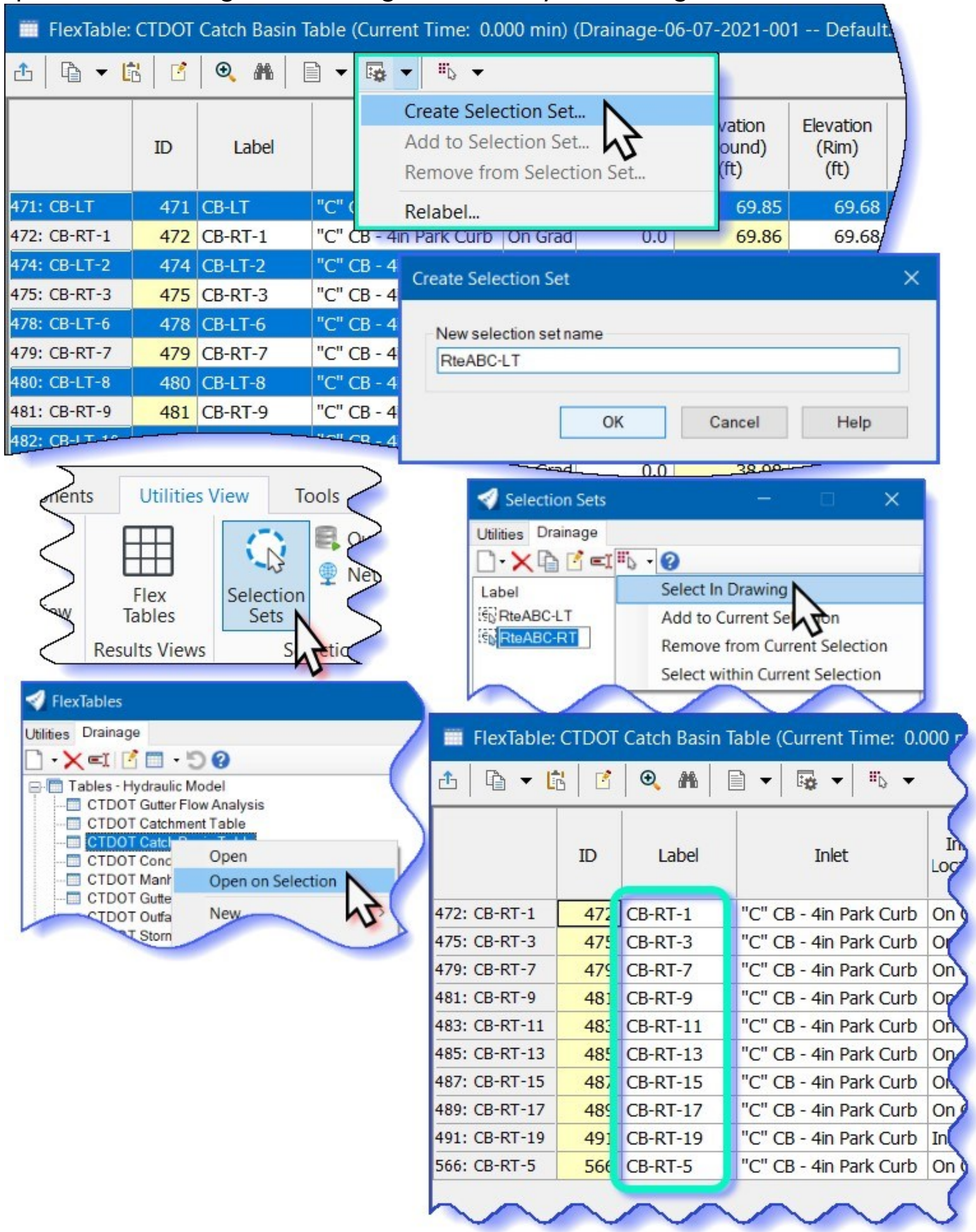


Figure 74 Flex Tables

3.3.3 Profile for Drainage

3.3.3.1 Single Pipe Profile

You can view the profile of any single pipe between two drainage structures by ***right-clicking on the pipe*** and ***hovering over*** for the pop-up menu to come up and ***picking*** the ***Open Profile Model*** bottom, it will ask to ***Select or Open View*** to place the Profile of the pipe.

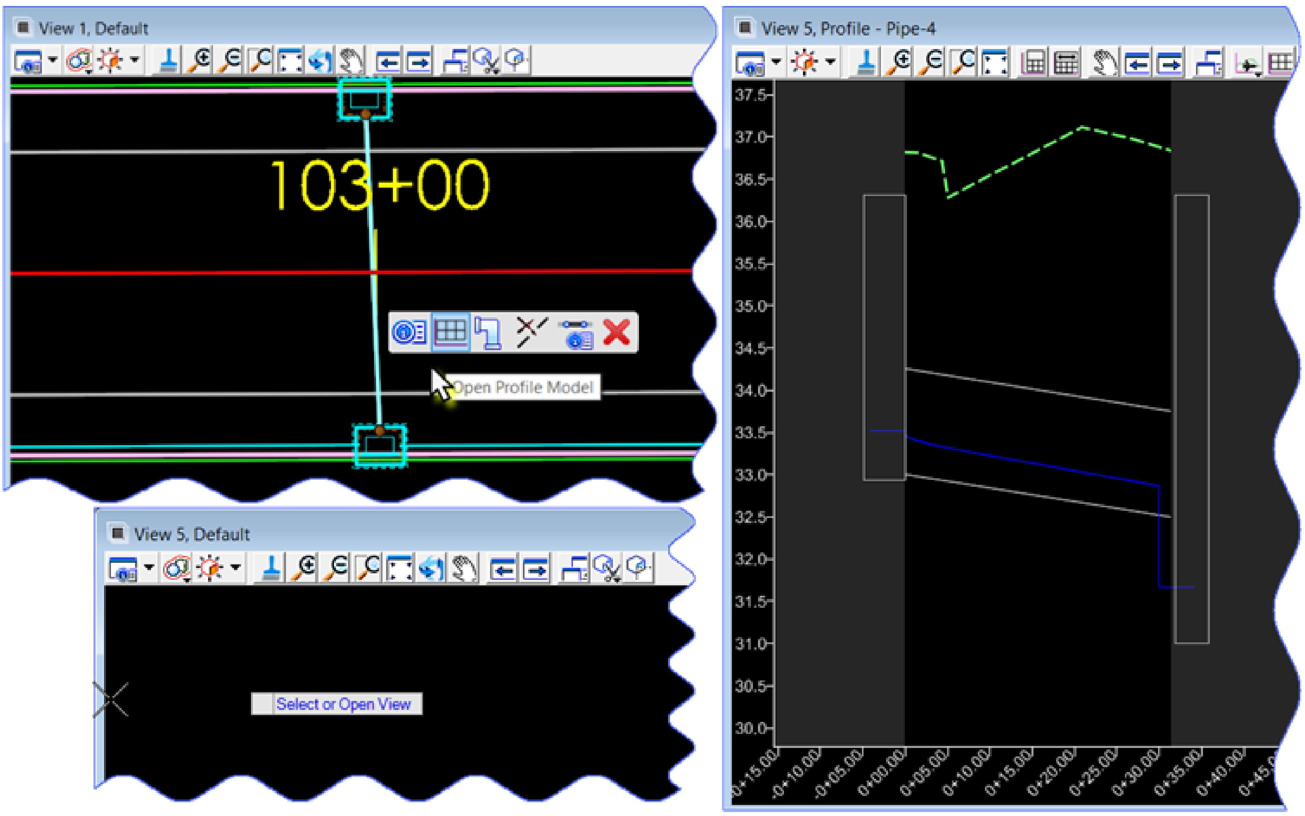


Figure 75 Single Pipe Profile

3.3.3.2 Profile Runs

Profile Runs show hydraulic information such as Hydraulic Grade Line (HGL) and/or Energy Grade Line (EGL), show the depth of cover, and show whether slopes of conduits are in the same direction and magnitude as the road design.

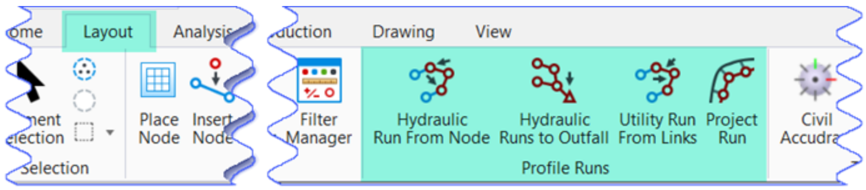


Figure 76 Profile Runs Tools

Hydraulic Run From Node: will create one profile run from a selected node to the outfall, or between two selected nodes.

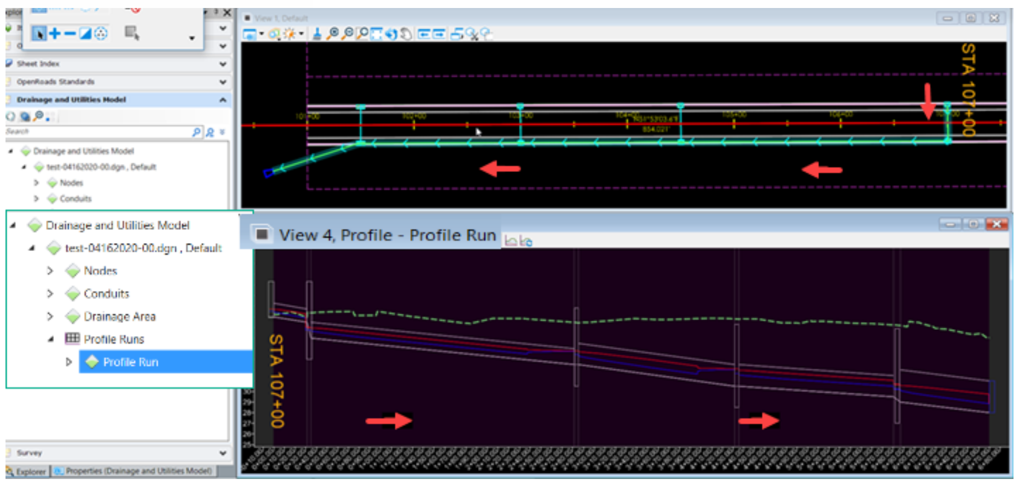


Figure 77 Hydraulic Run From Node

Hydraulic Run To Outfall: will create a profile run for every path, from farthest upstream node to the outfall. This tool will create profile runs for each run to the system outfall. When completed there will be 2 or more profile runs depending on the system.

Figure 78 Hydraulic Run To Outfall

Utility Run From Links: will create a profile run from selected links, for any type of utility (storm, communication, sanitary, electric, water). The links must be consecutive without gaps between them.

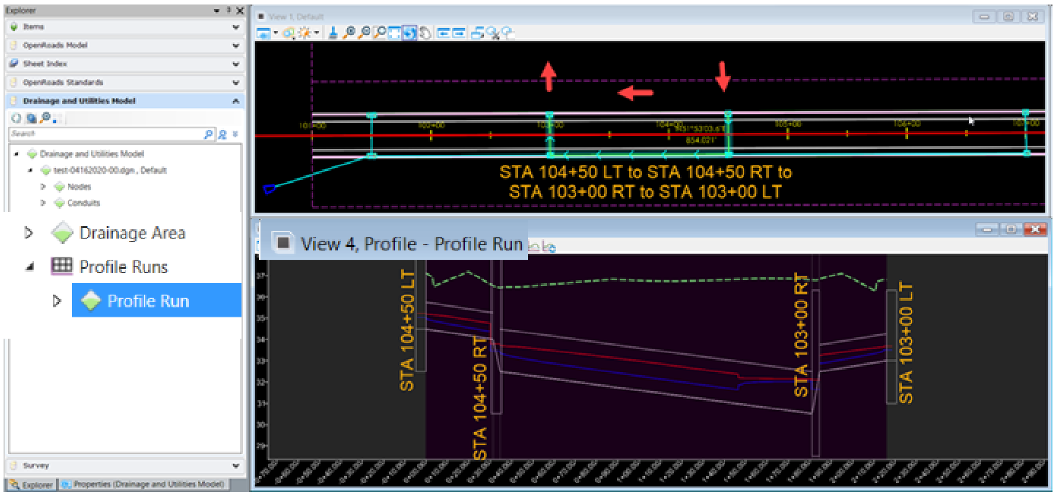


Figure 79 Utility Run From Links

Project Run: This will project a previously created profile run onto a linear element such as a centerline or another profile run. Great tool to show the drainage structures and pipes on the road's centerline profile (vertical alignment).

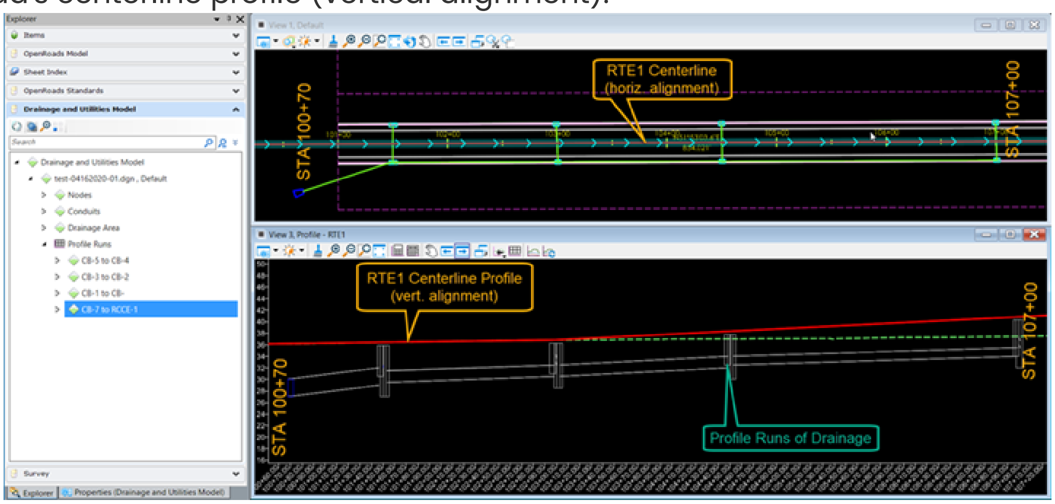


Figure 80 Project Run

In the ***Explorer***, expand ***Profile Runs*** (click on down arrow); the newly created profile runs are now available to view. ***Right-click*** on the run to see all available options to view/edit the profile data.

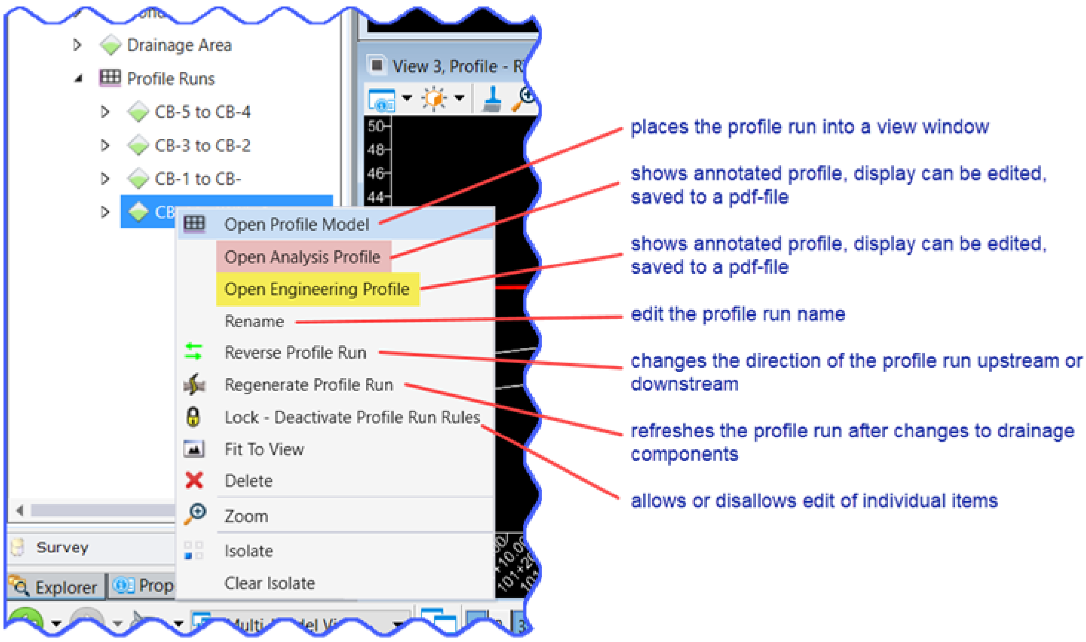


Figure 81 Profile Runs

Open Profile Model – ***right-click*** on a Profile Run, ***select Open Profile Model*** – ***click in the view window*** to place the profile run.

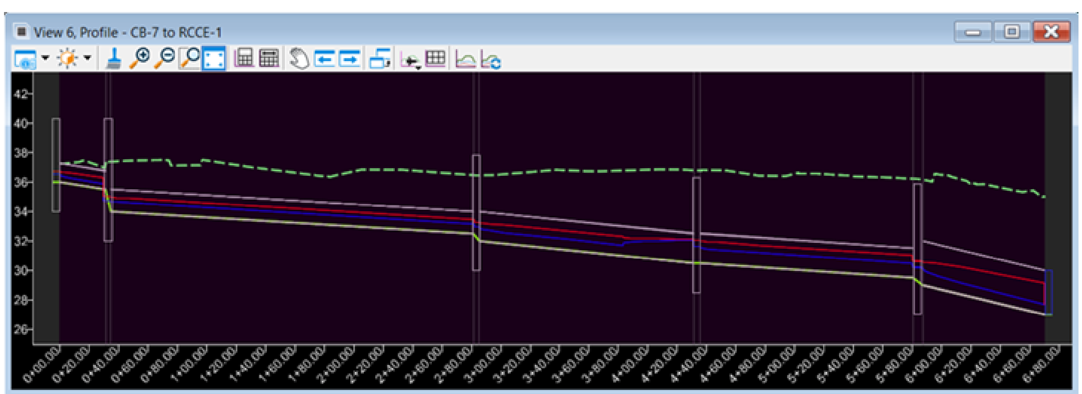


Figure 82 Profile Model

Open Analysis Profile - ***right-click*** on Profile Run, ***select Open Analysis Profile*** - a window opens with the analysis profile. This can be modified to include Hydraulic Grade Line (HGL), Energy Grade Line (EGL), labels and table annotation and legend. The file can be exported to a drawing or can be printed.

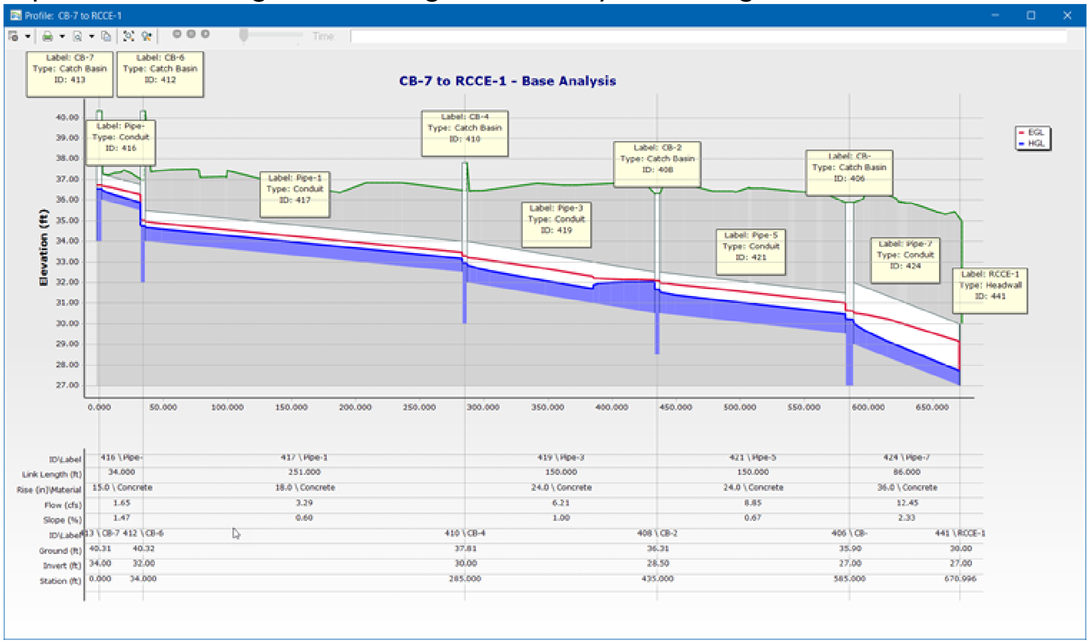


Figure 83 Analysis Profile

Saved as a pdf-file

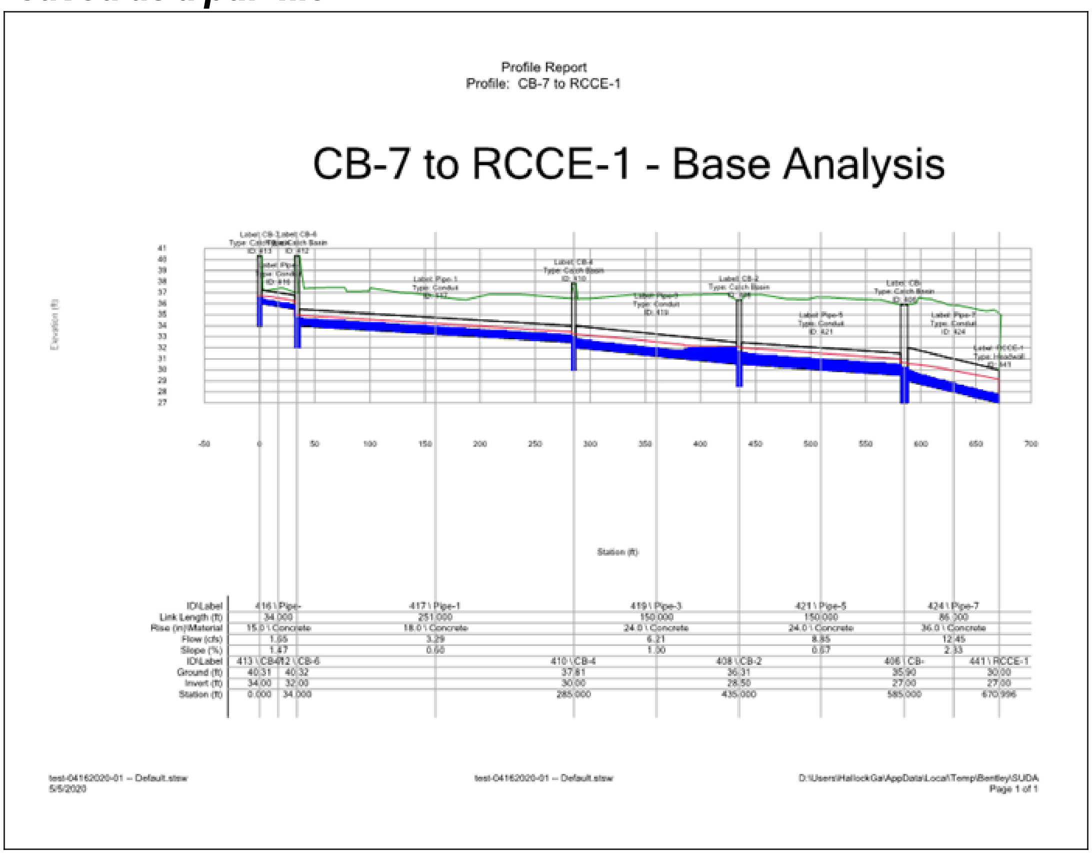


Figure 84 Saved as a pdf-file

Open Engineering Profile – ***right-click*** on Profile Run, ***select Open Engineering Profile*** - a window opens with the engineering profile. This can be modified to include inverts, slopes, types, length and more. The file can be exported to a drawing or can be printed and exported to a document.

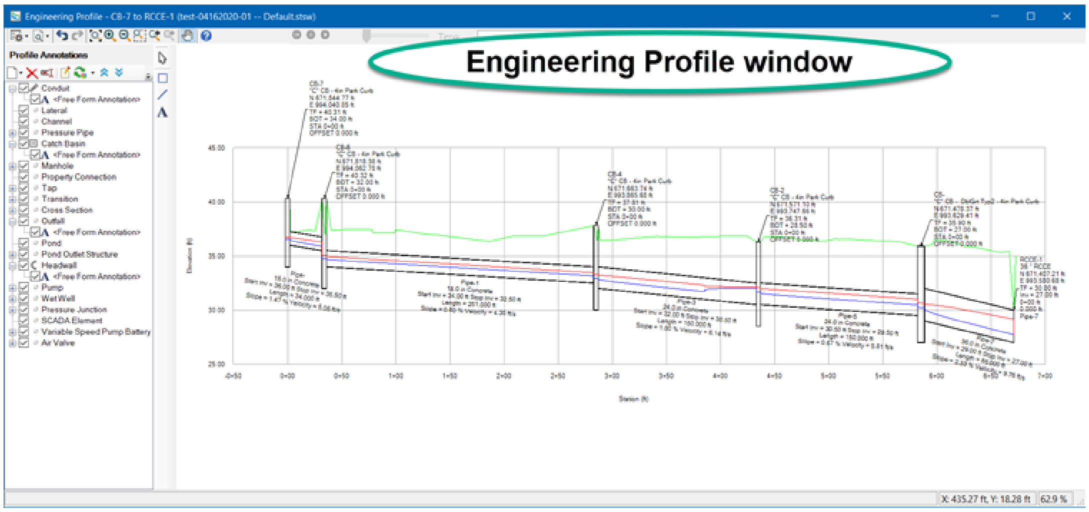


Figure 85 Engineering Profile

Saved as a pdf-file

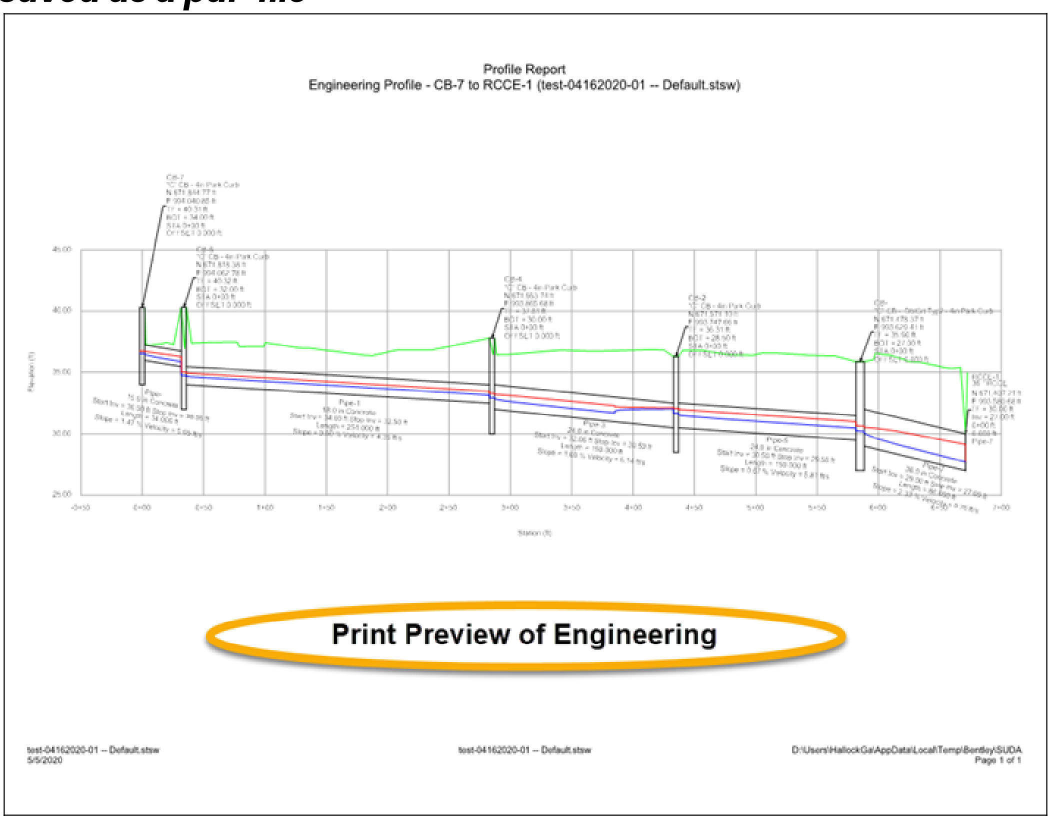


Figure 86 Saved as a pdf-file