

CES322351

## Analyze This—Exploring the New Analyze Gravity Networks in Civil 3D

Brian J. Hailey, P.E.  
GEI Consultants

### Learning Objectives

- Discover the methodology behind the Analyze Gravity Networks tool
- Learn how to prepare your drawing and pipe network to be taken into the Analyze Gravity Networks tool
- Learn how to adjust the settings and non-pipe-related values (such as rainfall)  
Learn how to get the results of the analysis back into your pipe network, as well as generate results for your drainage reports

### Description

Rain. What civil project doesn't have to deal in one way or another with water falling from the sky? If there's insufficient space to convey the water above ground, we need to take it underground in pipes. The Civil 3D workflows for designing our gravity storm sewers have been very disconnected. But with the release of the updated Analyze Gravity Networks in Civil 3D 2020, we have a much more integrated way of designing our storm systems. In this class you'll learn how to set up your drawing to run the analysis efficiently, you'll see what information needs to be entered into the analysis (such as rainfall data), and you'll discover how to get your pipe network to update with the results.

### Speaker

In 1998, armed with a Bachelor of Science in Civil Engineering from Colorado State University, I began my career working at a general civil engineering firm using Land Desktop LDT. Within a short time, I became the resident self-taught expert in LDT and AutoCAD and then learned AutoCAD Civil 3D upon its release. I then spent 13 years in the Autodesk reseller channel as a technical specialist delivering best-in-class training and support using AutoCAD Civil 3D, InfraWorks, and other Autodesk solutions. Currently, as CAD Manager and licensed professional engineer at GEI Consultants, I provide training and support to the engineers and designers on staff as well as implement new technologies to enable us to provide better designs on time and under budget. To that end, I am professionally certified in Autodesk software, have taught at Autodesk University ten times since 2007, am a member of the Autodesk Expert Elite program, and won the AUGI Top DAUG competition at AU 2011.

c3dplus@gmail.com

## Introduction

Rain, a civil engineer's job would be almost nothing if it wasn't for rain. Sure, we still have underground utilities and roads but so much of what we design must consider the water falling from the sky, landing on the ground, and flowing to the ocean or other destination.

The Analyze Gravity Network (AGN) tool was introduced in Civil 3D 2018 but the user interface was very limited. A good portion of the data had to be entered into the dialog box after running the tool and, all in all, it was just clunky. With the release of Civil 3D 2020, major improvements have been made to the AGN tool that make it much more user friendly.

## Methodology

The Analyze Gravity Network (AGN) tool is based primarily off the Federal Highway Administration Hydraulic Engineering Circular No. 22 or the FHWA HEC-22 ("HEC-22" for short). This is a publicly available publication that is widely used throughout the United States. The Federal Highway Administration has published this document to assist engineers in designing drainage infrastructure for roadway projects. HEC-22 uses the rational method to compute peak flows for the system. As a refresher, the rational method is  $Q=CIA$  where  $Q$  is the peak flow,  $C$  is the runoff coefficient of the contributory area,  $I$  is the intensity of the rainfall, and  $A$  is the contributory area.

## Limitations

HEC-22 has some limitation that you should be aware of. First, it was created with the roadway drainage in mind. An example of this is that inlets are assumed to be in curb and gutter. If you are using the Analyze Gravity Networks (AGN) for an area inlet in a parking lot, it will assume one side of the inlet has curb and gutter. This means that the peak flow into the inlet could be greater than what is calculated because the weir flow conditions are calculated based on flow not entering on one side (the side with the "curb").

Another limitation is that all the inlet capacities assume the inlets are rectangular in shape. If the grate for an inlet is circular, you must adjust the geometry of the inlet in AGN to match the properties of the actual inlet you are using.

Finally, there are several other limitations that are posted in the Civil 3D Help system. You can read more about those limitations [HERE](#).

## Prepare drawing and pipe network

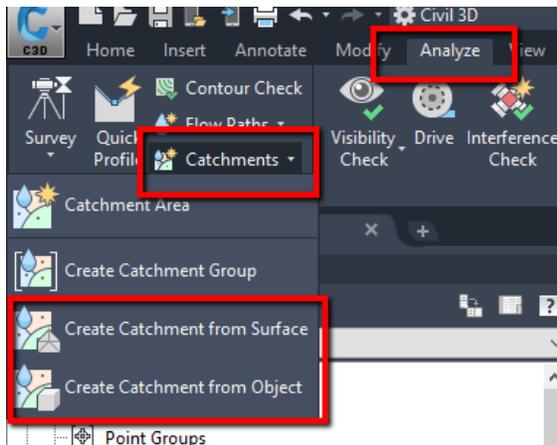
### Contributory Area

The drainage area that is contributory to an inlet can be defined a couple different ways. First, the catchment tools in Civil 3D can be used to create a catchment that can be linked to the structure. Using this method, you can have multiple catchments with different properties connecting to the same structure. Alternatively, you can simply add the contributory area directly to the structure within the structure properties.

### Catchments

There are two different methods for creating catchments: 1) You can manually draw in the catchment and create the catchment from the objects or 2) You can use the surface to define the catchment.

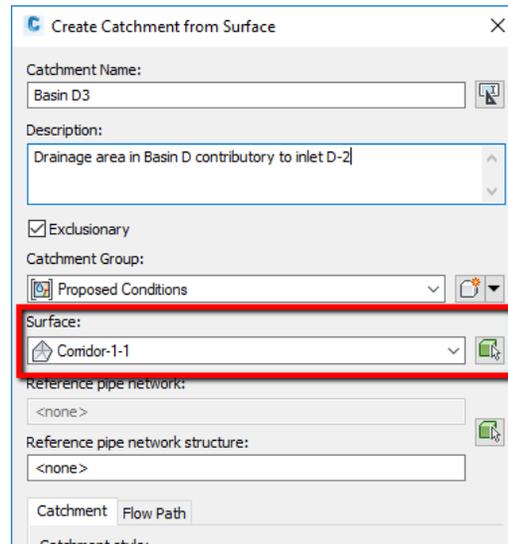
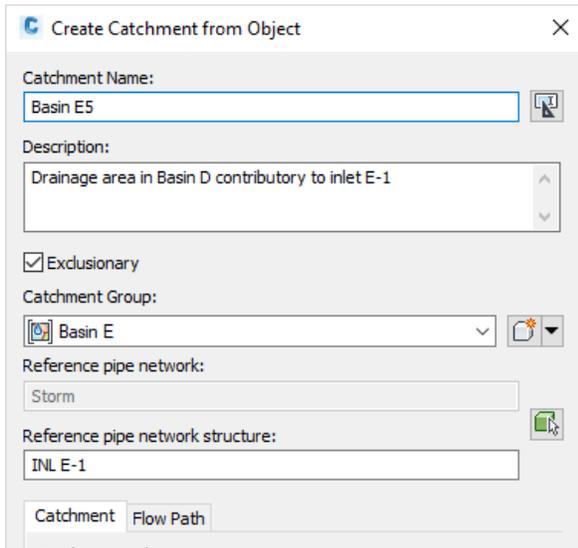
To create the catchment, on the Analyze tab on the ribbon, expand out Catchments on the Ground Data panel and choose either “Create Catchment from Object” or “Create Catchment from Surface”.



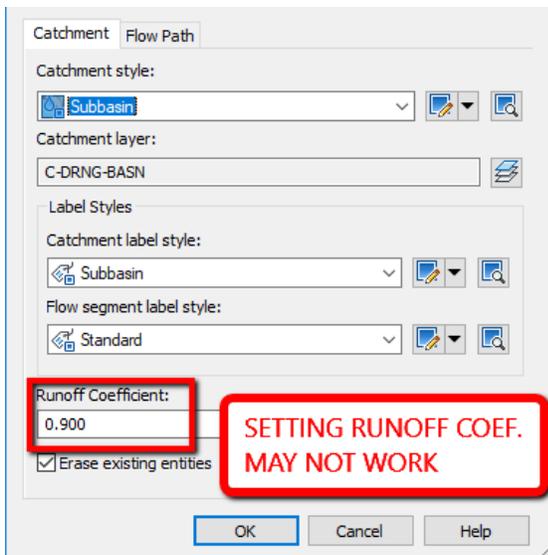
If creating the catchment from object, select the boundary of the catchment and you will be prompted to select the upstream end of the flow path. If you don't have a flow path, simply hit ESC to skip the flow path. After selecting the objects, the “Create Catchment from Object” dialog box is displayed.

Alternatively, to create a catchment from a surface, select “Create Catchment from Surface”. You will then be prompted to select the downstream point of the surface (typically an inlet but not necessarily).

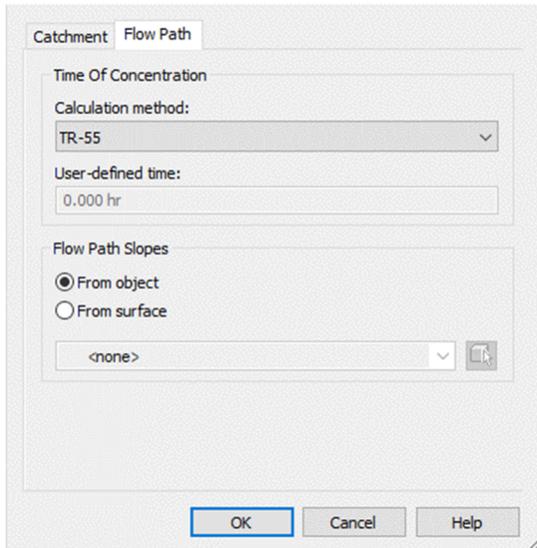
Both commands bring up essentially the same dialog box except the create from surface option allows you to choose which surface you will be using. In this dialog box, give the catchment a name and a description. You can also specify if the catchment is “Exclusionary”. If the option is turned on, the catchment will not overlap other catchments in the Catchment Group. If unchecked, catchment areas can overlap. Select the Catchment Group the catchment will go into. The Catchment Group is only important if using the exclusionary option. Catchment Groups can also be helpful for organizing your catchments. Finally, specify which structure in your pipe network the catchment is contributory to. This is critically important if you want this catchment data to be used in the Analyze Gravity Network tool.



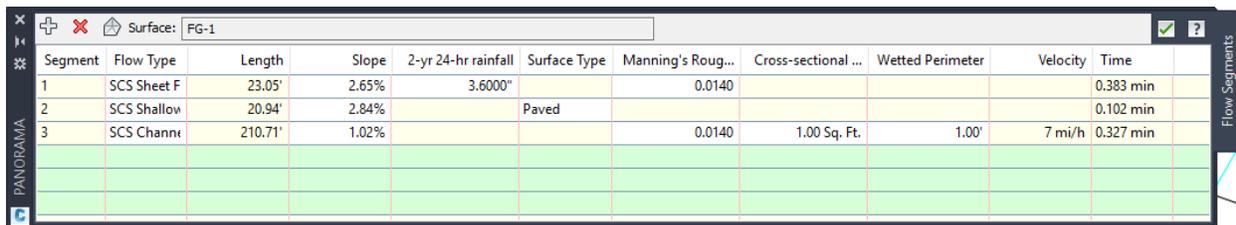
The lower portion of the dialog box has two tabs, Catchment and Flow Path. The Catchment tab allows you to specify the properties of the catchment itself, such as layer, style, and label style. There is also an option to specify the runoff Coefficient. If you are using the “Create Catchment from Surface” command, no matter what value is typed into this field, the catchment will always be created with a runoff coefficient of 0.5 To set the runoff coefficient, you’ll have to go into the catchment properties after it’s been created and set it there.



The Flow Path tab allows you to specify the settings for the Time of Concentration (TOC). There are only two options for the TOC calculations, TR-55 and User Defined. If the TR-55 method is used, you can specify the Flow Path Slopes from a surface or from the object itself (if the flow path was drawn as a 3D Polyline at elevations).



To calculate the TOC using the TR-55 method, select the catchment in the drawing, right click, and select “Edit Flow Paths” (this is not available on the ribbon for some reason).

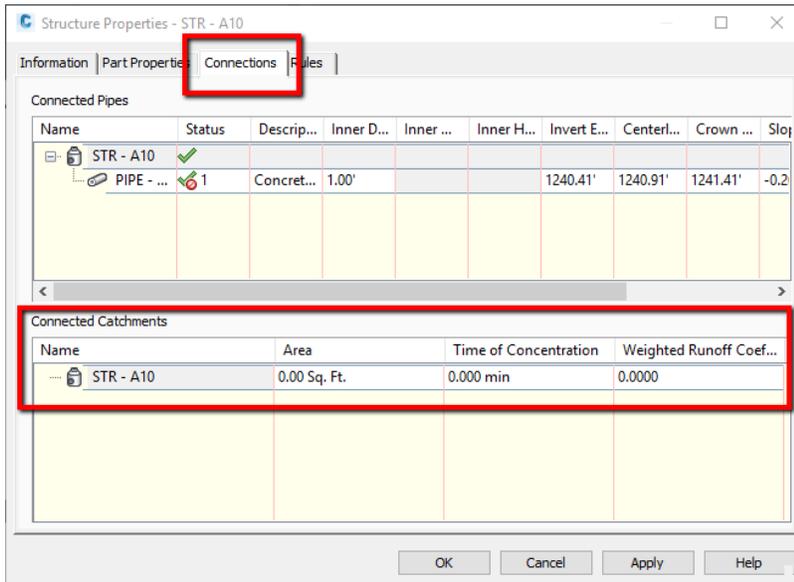


Segment	Flow Type	Length	Slope	2-yr 24-hr rainfall	Surface Type	Manning's Roug...	Cross-sectional ...	Wetted Perimeter	Velocity	Time
1	SCS Sheet F	23.05'	2.65%	3.6000"		0.0140				0.383 min
2	SCS Shallow	20.94'	2.84%		Paved					0.102 min
3	SCS Channn	210.71'	1.02%			0.0140	1.00 Sq. Ft.	1.00'	7 mi/h	0.327 min

To add a flow path segment, click the white plus (+) button in the top left of the panorama and then click in the drawing where you want the flow to transition from one type to another. Once all the flow segments have been created, set the Flow Type and then set the properties to match your design.

### Structure properties

An alternative to creating a catchment and then attaching that to a pipe network structure is to simply edit the properties of the structure itself. In the structure properties, on the Connections tab, simply enter the catchment information in the Connected Catchments section. You'll need to manually calculate the area, TOC, and Runoff Coefficient but in some situations, this can be a much simpler process than creating a catchment.



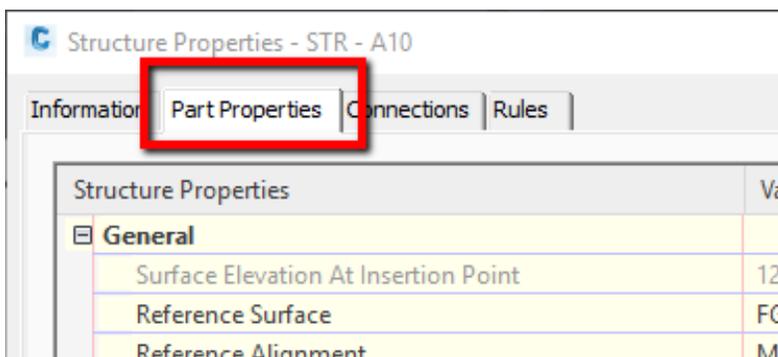
If the structure already has a catchment attached to it, you will not be able to edit the data or manually add an additional catchment. If catchments are used, they are the only thing that can be used. If catchments aren't used, you can adjust the data manually in the structure properties.

Connected Catchments

Name	Area	Time of Concentration	Weighted Runoff Coef...
STR - A10	17317.58 Sq. Ft.	10.000 min	0.5000
Catchment - (17)	17317.58 Sq. Ft.	10.000 min	0.5000

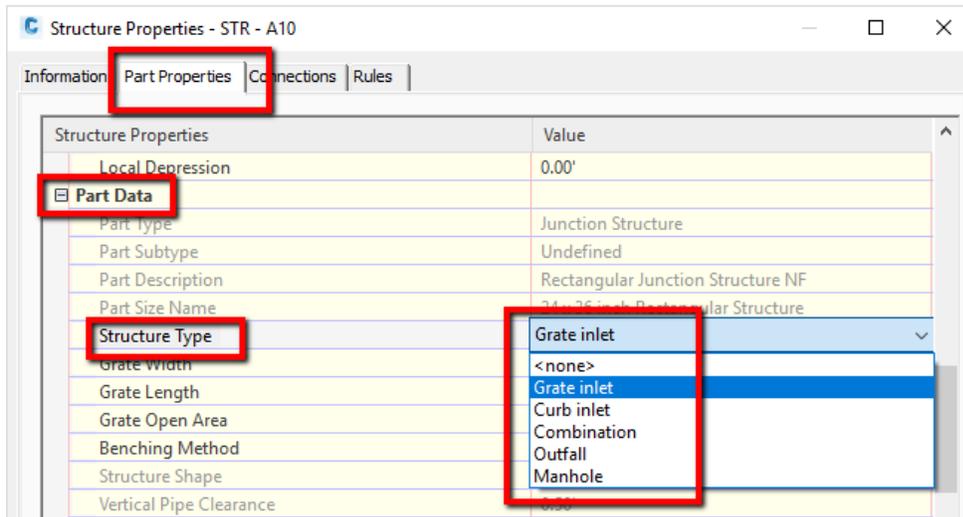
### Structure Properties

The catchment information can be provided via a catchment object or by editing the properties of the structure. There are additional properties that need to be specified within the properties. These will be set on the Part Properties tab of the Structure Properties. The settings that need to be specified include the structure type, the gutter details, and the default properties.



## Structure Type

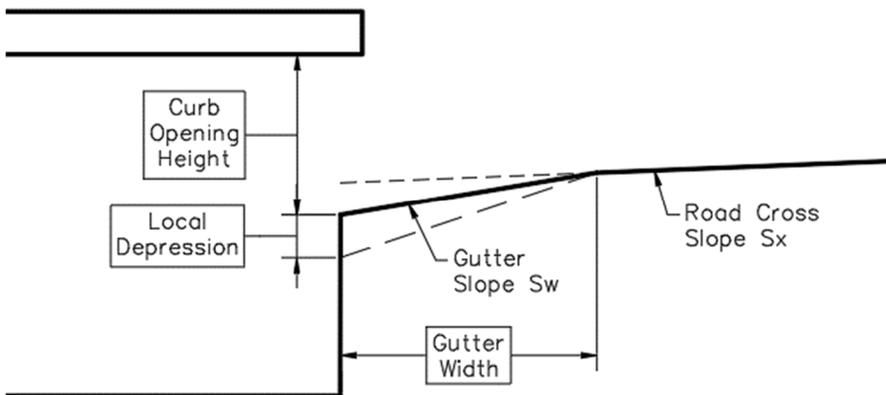
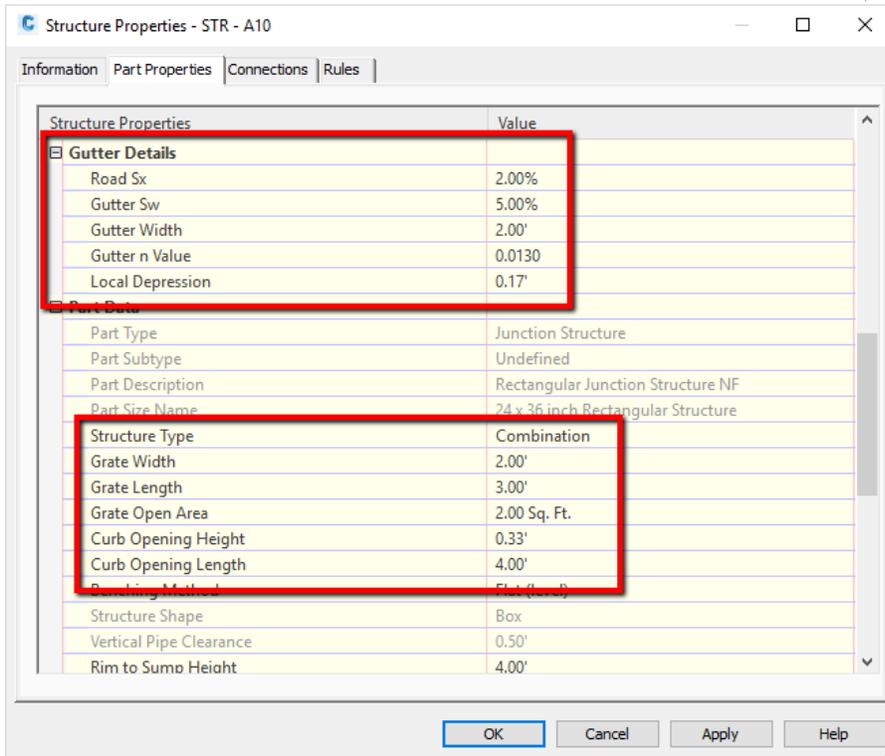
The Structure Type allows you to specify what type of structure this part represents. There are five options for the structure type, Grate Inlet, Curb Inlet, Combination Inlet, Outfall, and Manhole. The type of structure will be used if the inlet analysis option is selected when running the analysis.



The data available in the Structure Properties will change depending on the Structure Type chosen. For example, if a “Curb inlet” is chosen, the properties Curb Opening Height and Curb Opening Length will be displayed but, if a “Grate inlet” is chosen, these same properties will not be available.

## Inlet Details

If one of the inlet types is chosen for the Structure Type, the Gutter Details section will be available and additional properties in the Part Data section will also be available. As HEC-22 was developed for highway drainage, inlets are assumed to be in a curb and gutter. The geometry of the curb and gutter and the inlet openings as specified in the Gutter Details and Part Data will be used in the efficiency calculations of the inlet and will also be used to calculate much the flow will spread into the street.



### Hydraulic Properties

Structures also have hydraulic properties that can be assigned, most of which deal with inlets. For all structures, you can input the Hydraulic Grade Line (HGL) and the Energy Grade Line (EGL). The Known Capacity and Known Flow can be input as well. The Known Flow will be added to the flow calculated from the catchment information for that structure. As far as I know, the Known Capacity isn't used in the Analyze Gravity Networks tool. The HGL and EGL will be calculated when the Analyze Gravity Networks tool is run.

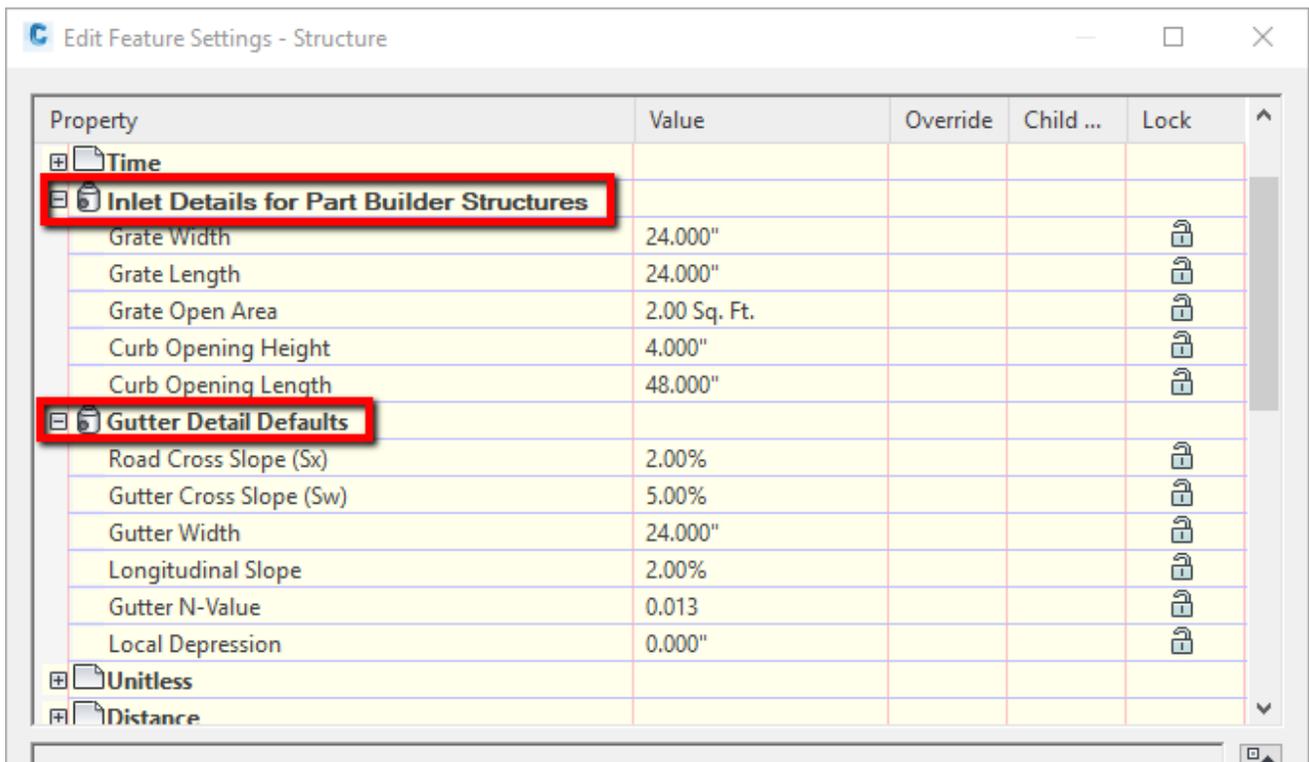
For inlets, you can also specify if the inlet is in a sag condition (all water that gets to the inlet eventually enters the inlet) or on grade (water can flow past the inlet and continue to another

inlet or offsite). If the inlet is on grade, you can specify which inlet the bypass flow continues to. Please note that the Analyze Gravity Network only analyzes one outfall at a time so if your bypass flow is contributory to another outfall, this tool will not account for that.

<b>Hydraulic Properties</b>	
Hydraulic Grade Line	1241.70'
Energy Grade Line	1241.70'
Known Capacity	0.0000
Known Flow	0.0000 cubic feet per second
Bypass Target	Offsite
Inlet Location	On Grade
<b>Gutter Details</b>	
Road Sx	On Grade
Gutter Sw	On Sag
Gutter Sx	<none>

### Default Properties

When creating a pipe network, the default properties for the inlets can be set in the feature settings for the structures. In the Settings tab of the Toolspace, right click on Structure and choose Edit Feature Settings... In the “Edit Feature Settings – Structure” dialog box, expand out “Inlet Details for Part Builder Structures” and “Gutter Detail Defaults”. If you work in an area where the typical gutter width is 18” and not 24”, changing this setting will save time when running the analysis.

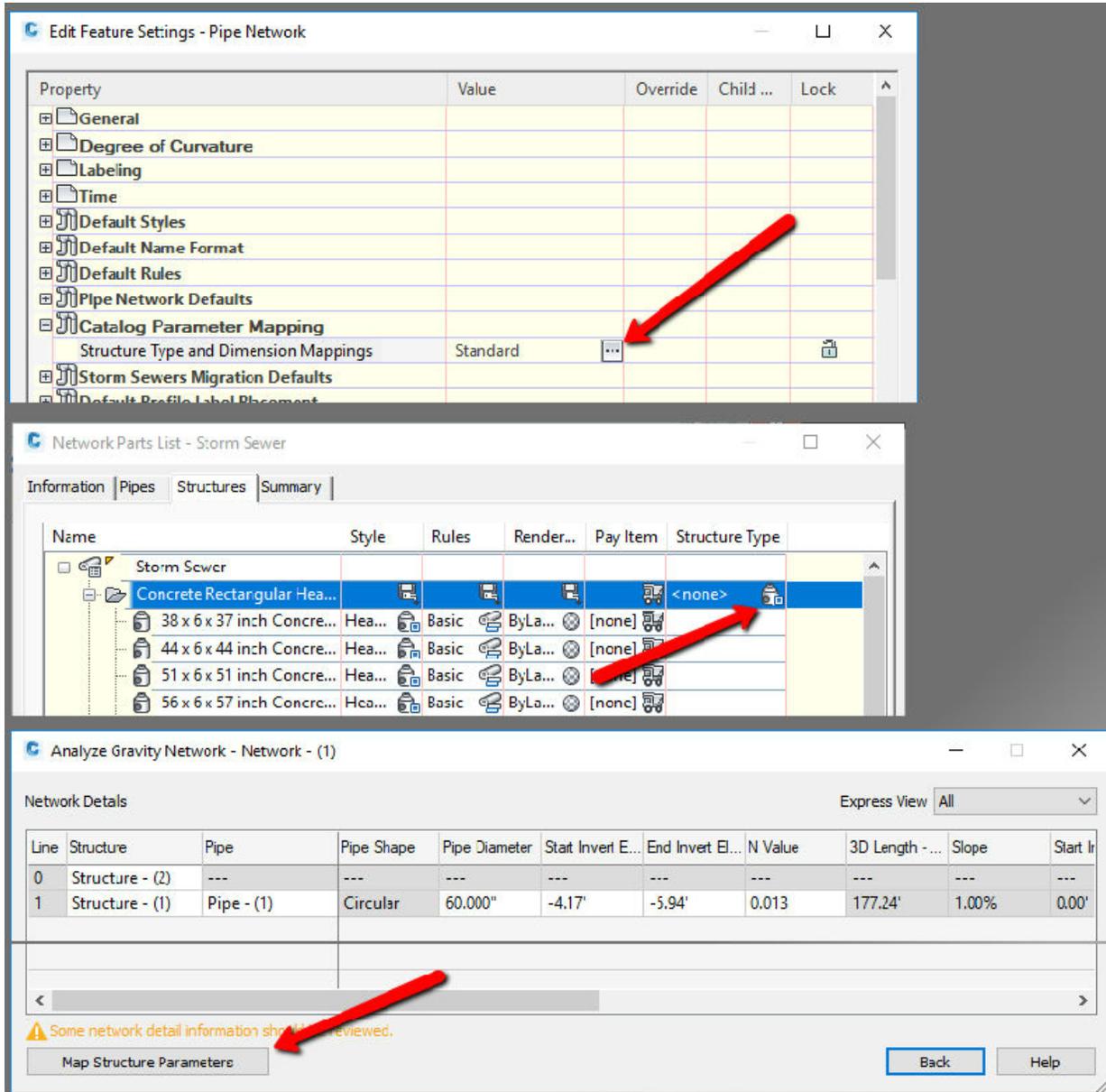


Property	Value	Override	Child ...	Lock
<b>Time</b>				
<b>Inlet Details for Part Builder Structures</b>				
Grate Width	24.000"			🔒
Grate Length	24.000"			🔒
Grate Open Area	2.00 Sq. Ft.			🔒
Curb Opening Height	4.000"			🔒
Curb Opening Length	48.000"			🔒
<b>Gutter Detail Defaults</b>				
Road Cross Slope (Sx)	2.00%			🔒
Gutter Cross Slope (Sw)	5.00%			🔒
Gutter Width	24.000"			🔒
Longitudinal Slope	2.00%			🔒
Gutter N-Value	0.013			🔒
Local Depression	0.000"			🔒
<b>Unitless</b>				
<b>Distance</b>				

### Map Structure Parameters

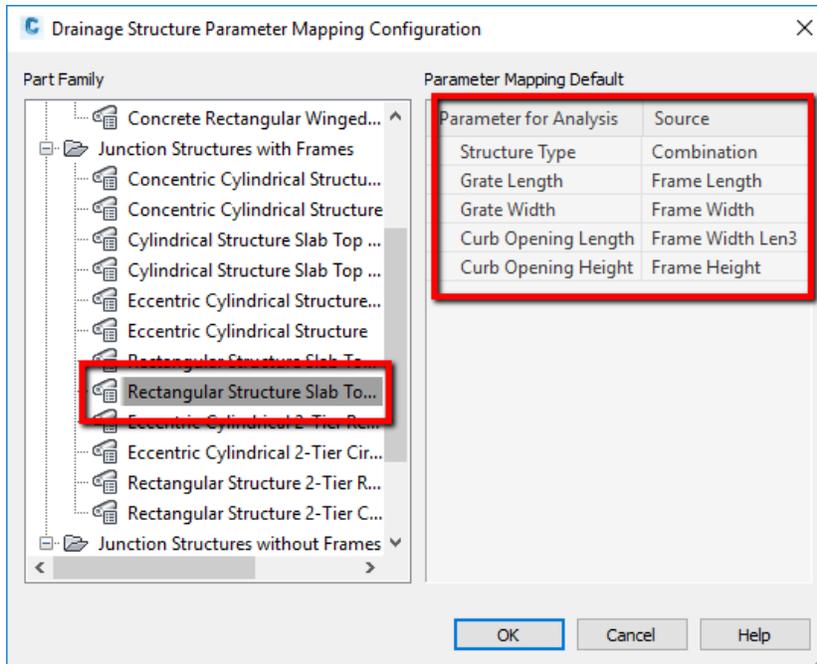
If you have a well-defined parts catalog and parts list, you can map the properties of the structure from part builder to the structure properties. This is done with the Map Structure

Parameters. There are multiple ways to access this tool: via the Analyze Gravity Networks dialog, via the parts list, or via the pipe network feature settings.



In the “Drainage Structure Parameter Mapping Configuration” dialog box, select the part family on the left and then set the structure type on the right. Once you’ve set the structure type, you can then map different parameters from the structure to the properties the Analyze Gravity Networks tool will use. For example, I’m using the structure “Rectangular Structure Slab Top Rectangular Frame” for my combination inlets. On the left, I select that part family, and on the right, I’ll set the Structure Type to “Combination”. I can then use different properties of the structure to map to the Grate Length, Grate Width, Curb Opening Length, and Curb Opening Height. Once set up, as the structures are added to the drawing, the Part Data in the structure

properties will be determined by the parameters of the part in the part list. If a part has the incorrect properties, swapping the part will set the properties as defined in the parts list.



### Profile Views

When the analysis has been completed, there is an option to preview the results. For this to work, you must have a profile view already created that is displaying the pipes. For complex systems, multiple profile views can be used but they must exist in the drawing prior to running the analysis tools

### Adjust settings and non-pipe-related values

In addition to the settings that are applied to the structures in the network, there are several settings that aren't related directly to the network that need to be set. In general, these include the rainfall data and the analysis settings.

### Rainfall Data

If you recall, the Analyze Gravity Network uses the Rational Method ( $Q=CIA$ ) to compute runoff. The intensity of the rainfall is integral to this method and must be specified prior to running the analysis.

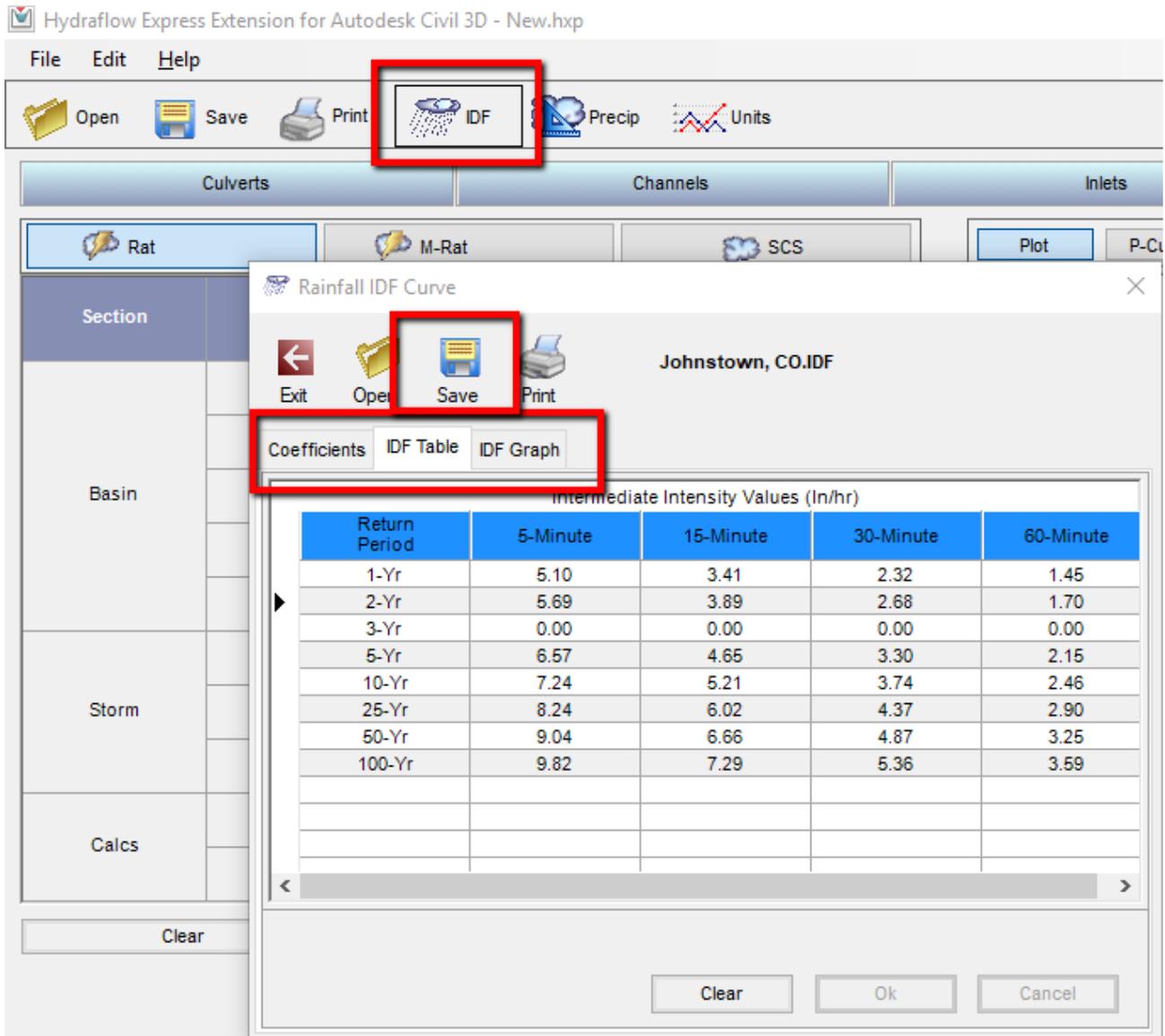
#### Format

The rainfall intensity data can be provided in three different formats, an H&H IDF Tabular file (\*.idf), a NOAA file (\*.csv), or an X-Degree file (\*.csv).

- H&H IDF Tabular file

If you are provided with rainfall intensity data, this is probably the best format to use for that data. It's the same IDF file type that is used in the Hydraflow programs (Storm Sewers, Hydrographs, and Express). To create this file, open one of these programs (Express is used for this example) and select the IDF button. Once in the tabular editor, you can enter the data in a few ways. If you were provided coefficients for the rainfall, you can enter them on the

Coefficients tab. There are two different equations that can be used for the coefficients, the FHA equation and a polynomial. If the only data you have is the intensities, durations, and frequencies, this data can be entered in the IDF Table tab. Once the data has been input, save the file and it can then be used in the analysis.



- NOAA file
- The National Oceanic and Atmospheric Administration (NOAA) has an excellent resource for determining the rainfall for just about any location in the United States. The website [https://hdsc.nws.noaa.gov/hdsc/pfds/pfds\\_map\\_cont.html](https://hdsc.nws.noaa.gov/hdsc/pfds/pfds_map_cont.html) provides an interactive map that allows you to specify the location of the study and to download the data. At the very top of the application, there is an option for "Data type:". **In order for this data to work with the Analyze Gravity Networks, it must be set to "Precipitation intensity"**.

### NOAA ATLAS 14 POINT PRECIPITATION FREQUENCY ESTIMATES: CO

**Data description**

Data type: Precipitation intensity Units: English Time series type: Partial duration

**Select location**

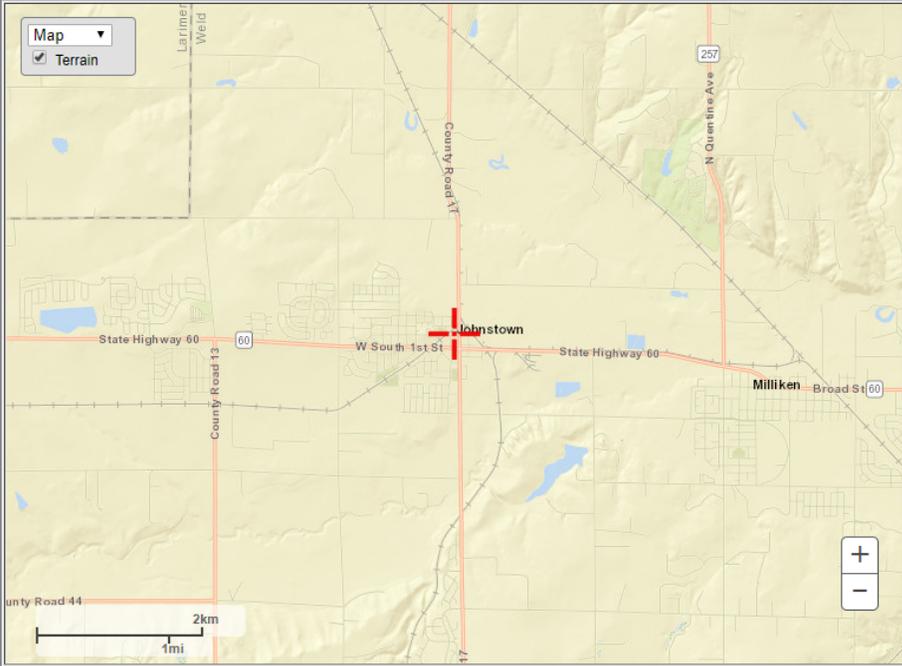
1) Manually:

a) By location (decimal degrees, use "-" for S and W): Latitude:  Longitude:

b) By station (list of CO stations):

c) By address

2) Use map (if ESRI interactive map is not loading, try adding the host: <https://js.arcgis.com/> to the firewall, or contact us at [hpsc.questions@noaa.gov](mailto:hpsc.questions@noaa.gov)):



**a) Select location**  
Move crosshair or double click

**b) Click on station icon**  
 Show stations on map

**Location information:**  
**Name:** Johnstown, Colorado, USA\*  
**Latitude:** 40.3361\*  
**Longitude:** -104.9068\*  
**Elevation:** 4823.7 ft \*\*

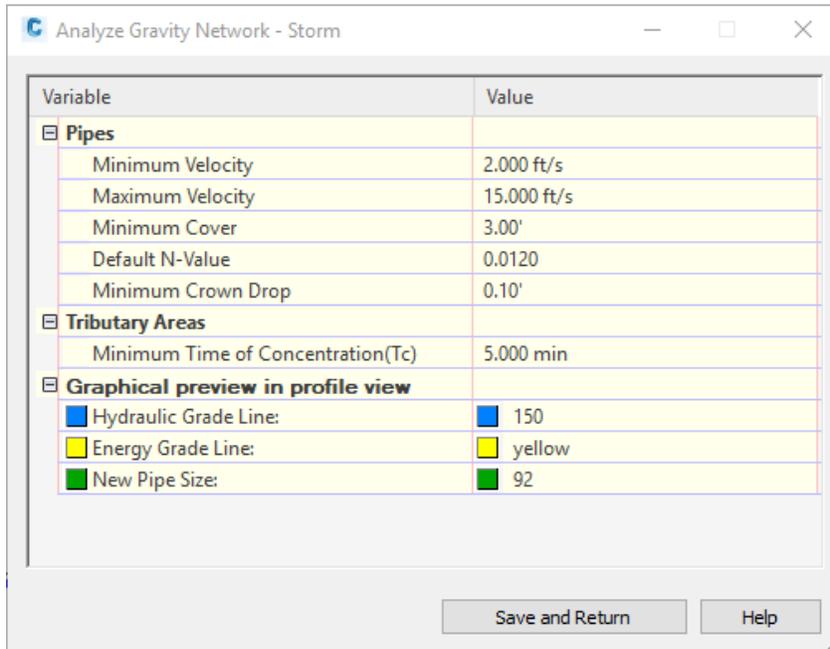
\* Source: ESRI Maps  
\*\* Source: USGS

- X-Degree file

The last of the data types that can be used in the Analyze Gravity Networks is the X-Degree file. There's very little information available on this file type. The InfraWorks help has a little bit more description on it (not much though). It shows it as the "Australian Rainfall & Runoff 6-degree polynomial data (.csv)".

### Analysis Settings

Last thing prior to running the analysis, there are a few settings that must be set. In the Analyze Gravity Network dialog box, click the "Settings" button. There are three different categories for the settings, Pipes, Tributary Areas, and Graphical preview in profile view.



### Pipes

In the pipes section, this is where the minimum and maximum velocities and the minimum cover is set. You can also set a default Manning's N-Value if the pipes don't already have that value assigned. A minimum drop through the structures can also be set here (as this is a storm analysis, the drop is assigned at the crown of the pipe, not the invert).

### Tributary Areas

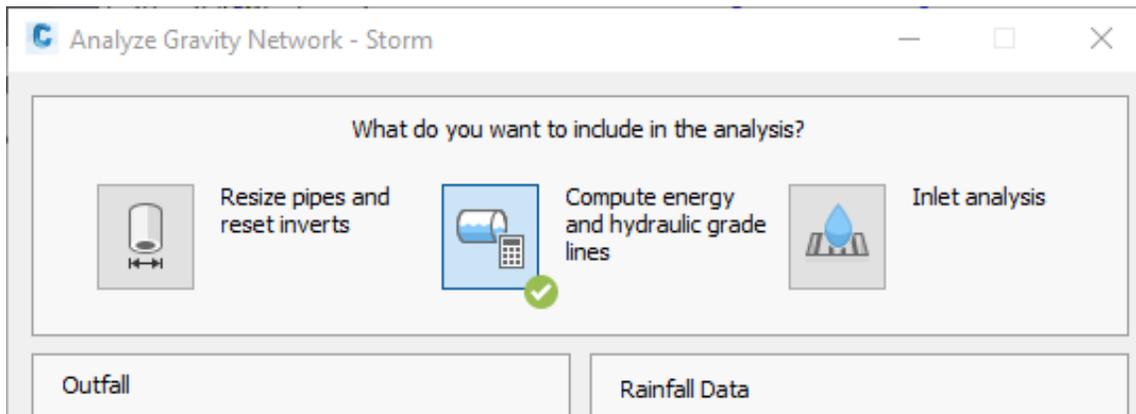
The tributary areas has only one setting, and that is a Minimum Time of Concentration. This prevents very small and/or steep catchments from having unrealistically large runoffs created from having a small time of concentration. A minimum ToC of five or ten minutes is very common.

### Graphical preview in profile view

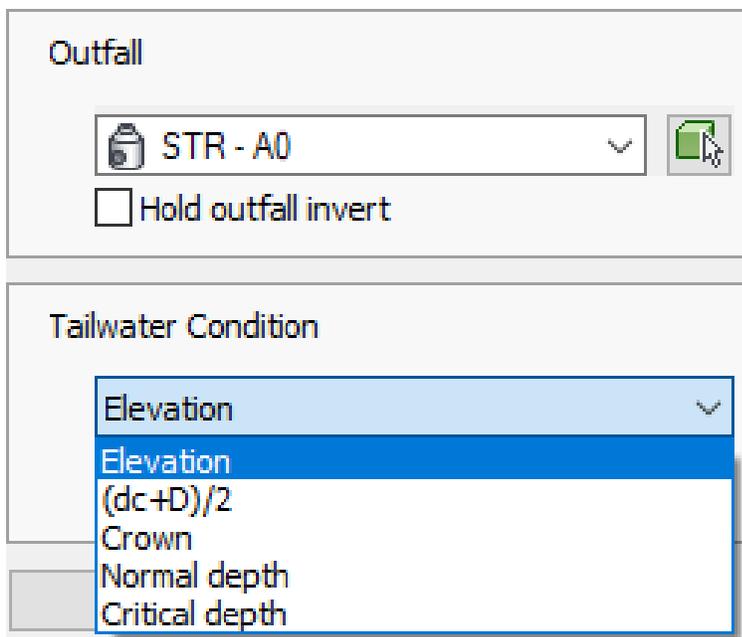
If your analysis is set to "Resize pipes and reset inverts", the resulting changes can be previewed in a profile view prior to committing to the changes. These settings allow you to specify the colors of the HGL, the EGL, and the adjusted pipes in the profile view preview.

## Run the Analysis

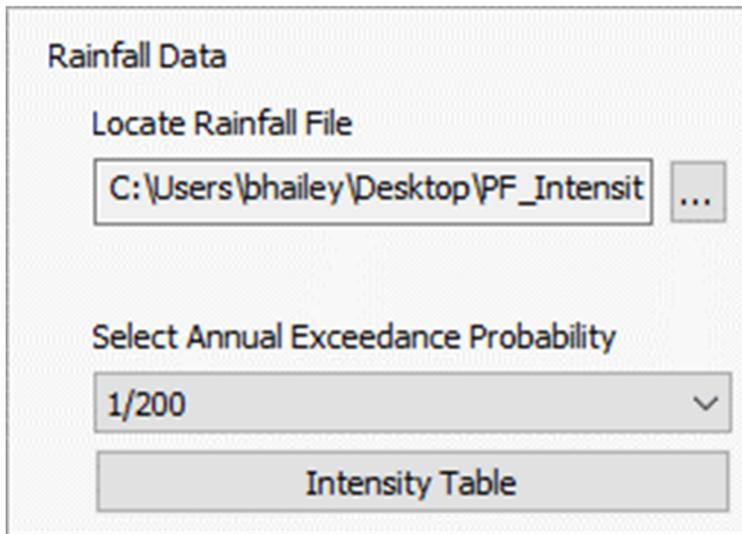
Once you're ready to run the analysis, on the Analyze tab of the ribbon, select "Analyze Gravity Network" (AGN). You will be prompted to select in the drawing the network you want to run the analysis on or you can press enter to select the network from a list. Once the network is selected, the Analyze Gravity Network dialog box is displayed. At the top of the dialog box, there are three options for the analysis, "Resize pipes and reset inverts", "Compute energy and hydraulic grade lines", and "Inlet analysis". If you just want to see how the pipes handle the flows, select just the middle option. If you want the AGN tool to size your pipes and set the inverts, select the first option, and if you want to include the inlet information (capture efficiency, bypass flow, etc.) select the third option. Each of these options can be toggled on and off as needed.



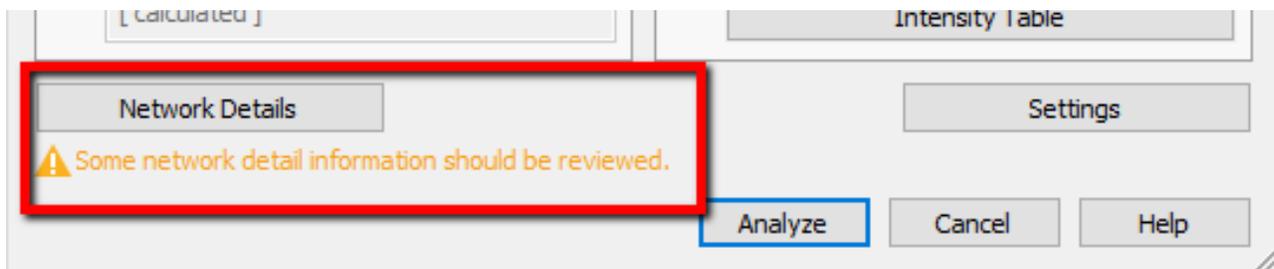
On the left side of the dialog box, is the outfall information. The Analyze Gravity Networks can only use one outfall at a time. When an outfall is selected, all the upstream pipes of that outfall will be analyzed. Any pipes connected to a different outfall will not be considered and the command will need to be run again for them. If the option “Resize pipes and reset inverts” is selected, you can manually set the outfall elevation and have it remain constant during the analysis by selecting, “Hold outfall invert”. Once the outfall has been selected, set the Tailwater Condition. There are five options for the tailwater: Elevations,  $(dc+D)/2$  (this is half the distance from the top of the pipe to critical depth), Crown, Normal depth, and Critical Depth. If the elevation option is chosen, enter the elevation in the text box below the tailwater condition (hidden by the combo box in this image).



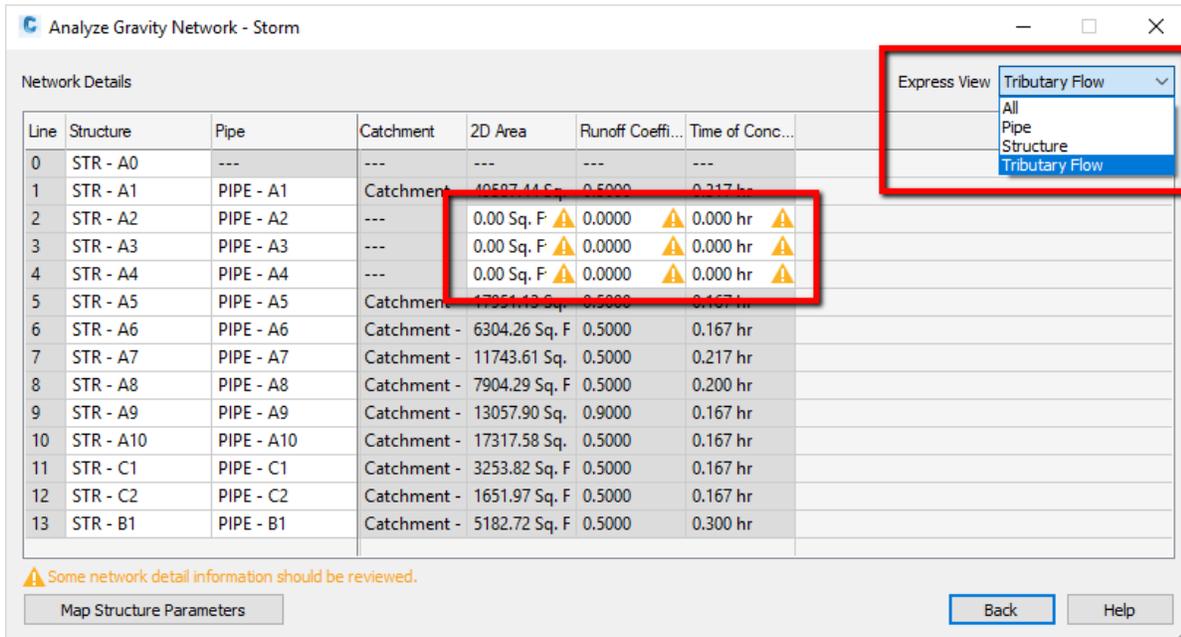
On the right side of the dialog box is the rainfall information. Select the rainfall file and then the exceedance probability (this is the same concept as the return interval). To see the rainfall information, select the “Intensity Table” button.



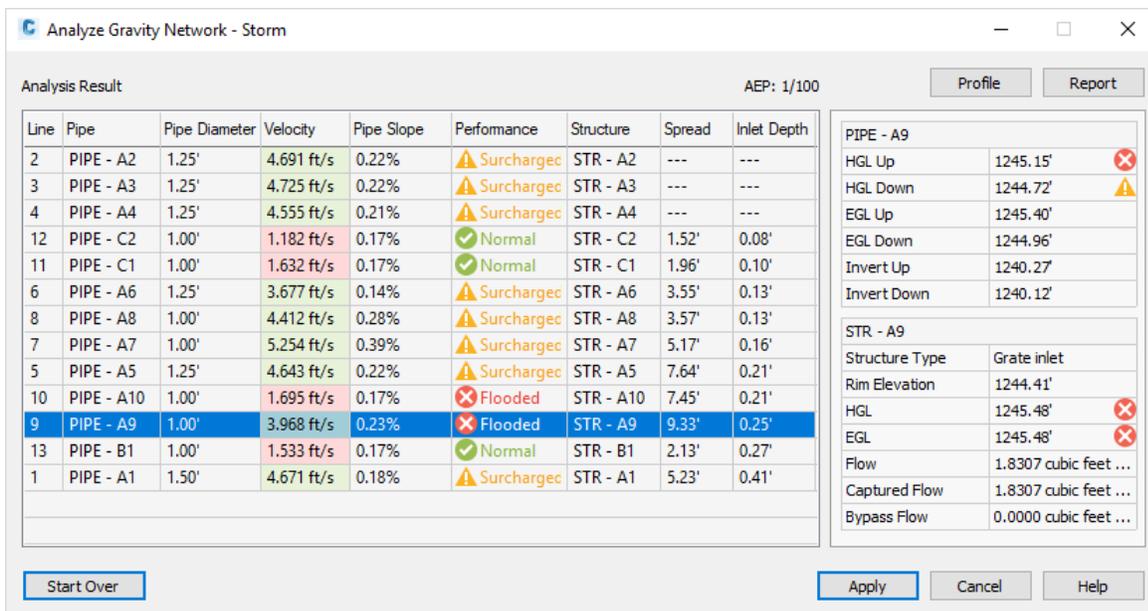
Often, the model will need some attention. Things might not be set correctly in the structure properties, data might be missing, or other issues may be present. In the lower left corner of the dialog box is the “Network Details” button. If there are issues with the model, there will be a note near this button to let you know something is up. Select the button and the Network Details button to bring up the Network Details.



In the top right corner of the Network Details dialog box, there are options for what data you would like to see in the dialog box. This can help simplify the editing process. For example, if you need to make some changes to the pipes in your drawing, change the Express View to “Pipe”. If there are any issues with the network, there will be yellow triangles with an exclamation point. Review these to ensure you have the correct data. Not all warnings need to be cleared. In the image below, the warnings are because there is no contributory flow to some of the structures. In this example, this is fine because these structures are manholes. Once you are satisfied with the state of the network, click the Back button to return to the main dialog box.



Back in the main AGN dialog box, it's time to run the analysis. Now that the network is setup and everything is ready, select the "Analyze" button at the bottom of the dialog box to run the analysis. Once the analysis has been run, the "Analysis Results" dialog box is displayed.



Here you will see how well the system has performed. In this example, I ran the 10-year storm and had the system reset the inverts and pipe sizes. I then ran it again with the 100-year storm but kept the 10-year inverts and sizes to see how well the system performed. As you can see, there are several pipes/structures that are flooding. If the flow is completely within the pipe, the performance will display as "Normal". If the pipe is flowing full, then the performance is "Surcharged". If the flow is above the rim elevation of one of the structures, then it will display as

“Flooded”. In this example, there are couple structures that are flooded so I’ll need to adjust some inverts or pipe sizes and run the analysis again. Fortunately, you can make these edits directly in the Analyze Gravity Network tool in the Network Details.

If you don’t like the results, AGN has not applied results to the network yet. You can simply close the dialog box and you’re back to where you were prior to running the analysis. Alternatively, if you would like to change some values in the network or analysis, you can select the “Start Over” button in the bottom left of the dialog.

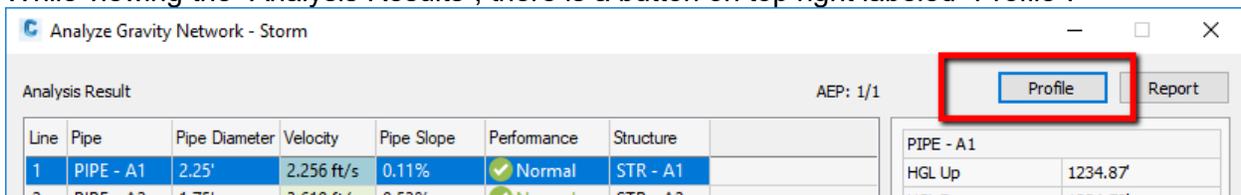
While viewing the results, you can see more detailed information about each entity. Select the row on the left side of the dialog box and the detailed information will be displayed on the right side of the dialog box. There are no editing capabilities in this dialog box, it simply shows the results of the analysis. If you need to make changes to the network or the analysis, select the “Start Over” button in the bottom left.

## Visualize Results and Reports

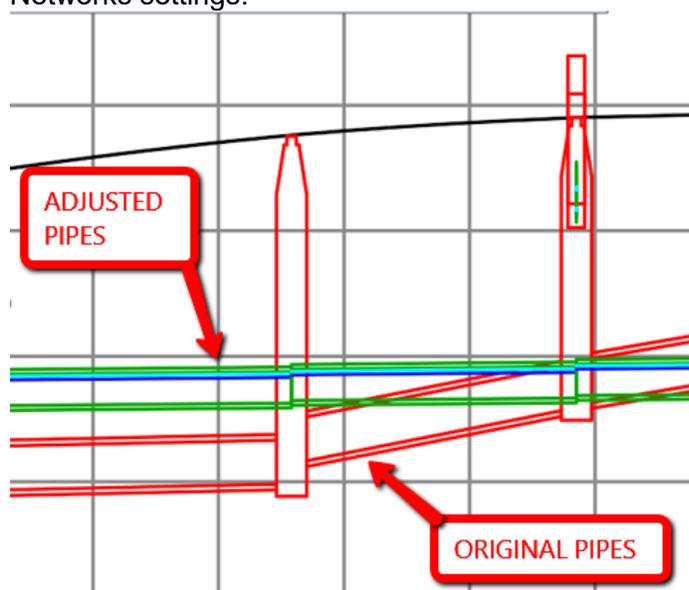
After running the analysis, you’ll want to display the results of the analysis as well as create reports.

### Visualize the Results in Profile

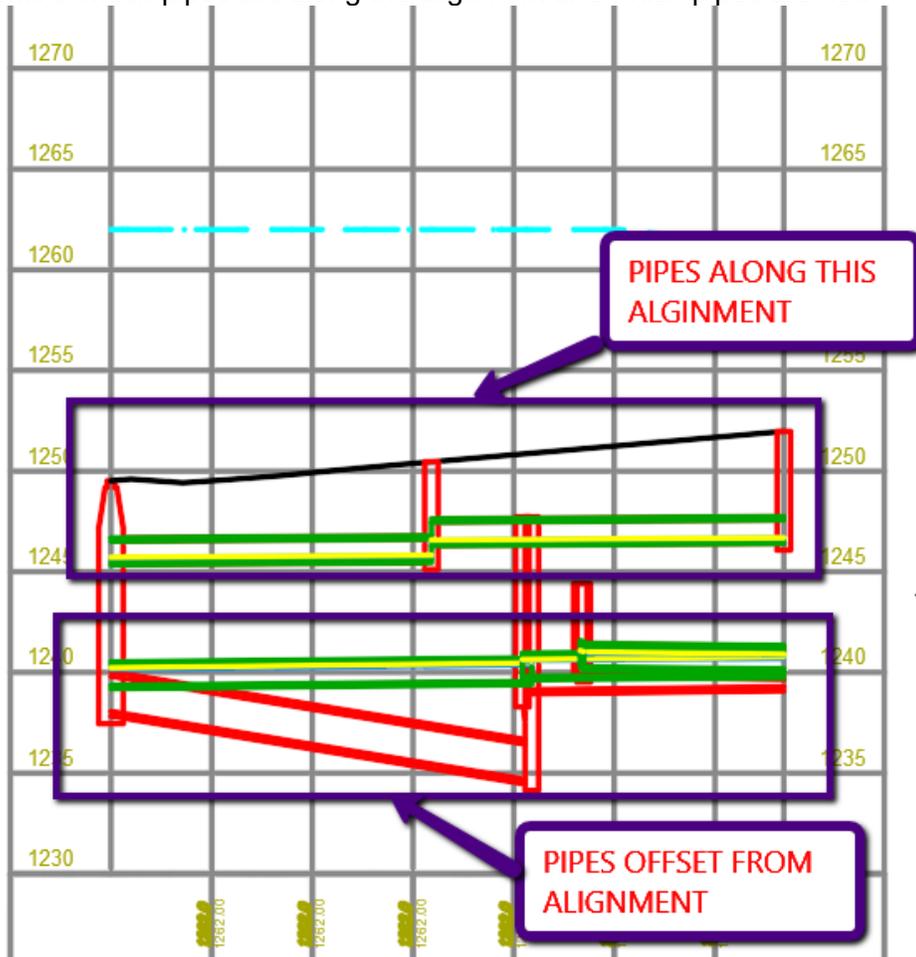
Once the analysis has been run, you can visualize the results in a profile view prior to committing the changes to the network. To do this, you must have a profile view in the drawing. While viewing the “Analysis Results”, there is a button on top right labeled “Profile”.



Select this button and then select the profile view you in which you want to preview the results. You can control the colors of the new pipe, the HGL, and the EGL in the Analyze Gravity Networks settings.



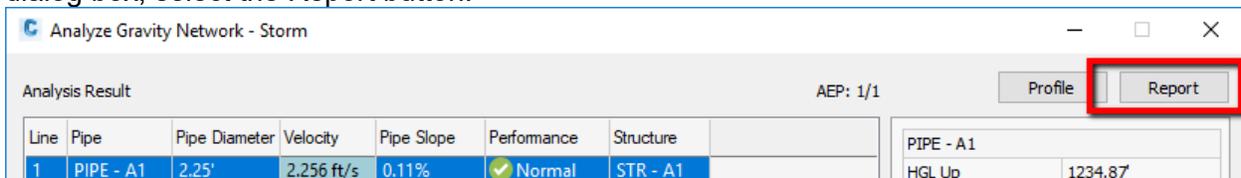
Take note that this does not just show the pipes that are within a certain distance along the alignment, it will show all pipes that have a valid station and offset to the alignment. Sometimes it will display more information than you need, and you'll need to use your best judgement to determine which pipes are along the alignment and which pipes are not.



Once you are satisfied with the results, click the Apply button in the Analysis Results. This will update your pipe network with the new invert elevations, sizes, and hydraulic properties.

### Creating Reports

The reporting tools in the Analyze Gravity Networks provide all the data from the analysis however, the formatting is currently very limited. To create the report, on the Analysis Results dialog box, select the Report button.



This will create a .csv file with all the data from the analysis. In my opinion, the formatting is limited. All the data is in one file. To get the data into a format that you would like to include in

another report (such as a drainage report) the data will need to be manipulated to meet your standards.

The report includes three sections; the first is the data for the structures, the second is the data for the pipes, and the third is the energy information for the system (although I'm not exactly sure on this). There is no legend in the .csv file that tells you what the different parameters represent but they do seem to match the symbols in the HEC-22 documentation (starting on page xxi).

A	B	C	D	E	F	G	H	I	J
#Line	Pipe	From	To	3D Length - Center to Center (ft)	Drainage Area Inc (sq. ft)	Drainage Area Total (sq. ft)	Runoff Coeff "C"	Area X "C" Inc (sq. ft)	Area X "C" To (sq. ft)
1	PIPE - A1	STR - A1	STR - A0	143.67	49587.44	133954.72	0.5	24793.72	7220C
2	PIPE - A2	STR - A2	STR - A1	167.98	0	79184.56	0	0	4481E
3	PIPE - A3	STR - A3	STR - A2	45.38	0	79184.56	0	0	4481E
4	PIPE - A4	STR - A4	STR - A3	122.55	0	74278.77	0	0	4236E
5	PIPE - A5	STR - A5	STR - A4	124.32	17951.13	74278.77	0.5	8975.56	4236E
6	PIPE - A6	STR - A6	STR - A5	24	6304.26	56327.64	0.5	3152.13	3338E
7	PIPE - A7	STR - A7	STR - A6	73.88	11743.61	50023.39	0.5	5871.81	3023A
8	PIPE - A8	STR - A8	STR - A7	24.06	7904.29	38279.77	0.5	3952.15	2436E
9	PIPE - A9	STR - A9	STR - A8	66.54	13057.9	30375.48	0.9	11752.11	2041
10	PIPE - A10	STR - A10	STR - A9	24	17317.58	17317.58	0.5	8658.79	865E
11	PIPE - C1	STR - C1	STR - A3	63.66	3253.82	4905.79	0.5	1626.91	24E
12	PIPE - C2	STR - C2	STR - C1	70	1651.97	1651.97	0.5	825.99	82E
13	PIPE - B1	STR - B1	STR - A1	25.27	5182.72	5182.72	0.5	2591.36	2591

#Line	Struct. ID	D (ft)	Q (cu. ft/sec)	L (ft)	V (ft/s)	d (ft)	dc (ft)	v^2/2g (ft)	EGLo (ft)
1	STR - A1	1	2.3889	143.67	3.457	1	0	0.19	123
2	STR - A2	1	1.5899	167.98	2.301	0.82	0.54	0.08	123E
3	STR - A3	1	1.6009	45.38	2.317	0.82	0.54	0.08	123E
4	STR - A4	1	1.5574	122.55	2.274	0.81	0.53	0.08	123E
5	STR - A5	1	1.6213	124.32	2.346	0.82	0.54	0.09	124C
6	STR - A6	1	1.288	24	2.229	0.69	0.48	0.08	124C
7	STR - A7	1	1.1954	73.88	2.197	0.65	0.46	0.08	124C
8	STR - A8	1	1.005	24.06	2.119	0.58	0.42	0.07	124C
9	STR - A9	1	0.9119	66.54	2.065	0.55	0.4	0.07	124
10	STR - A10	1	0.3868	24	1.642	0.34	0.26	0.04	1241
11	STR - C1	1	0.1096	63.66	1.118	0.18	0.14	0.02	124E
12	STR - C2	1	0.0369	70	0.777	0.11	0.08	0.01	124E
13	STR - B1	1	0.0877	25.27	1.051	0.16	0.12	0.02	123E

#Line	Struct. ID	Exit Ho (ft)	Hf (ft)	Hb (ft)	Hc (ft)	He (ft)	Hj (ft)	Total (ft)	Ei (ft)
1	STR - A1	0.19	0.55	0	0	0	0	0.55	
2	STR - A2	0	0.29	0	0	0	0	0.29	
3	STR - A3	0.03	0.08	0	0	0	0	0.08	C
4	STR - A4	0.03	0.2	0	0	0	0	0.2	C
5	STR - A5	0.03	0.22	0	0	0	0	0.22	C
6	STR - A6	0.02	0.04	0	0	0	0	0.04	C
7	STR - A7	0.02	0.12	0	0	0	0	0.12	C
8	STR - A8	0.01	0.04	0	0	0	0	0.04	

Keep in mind, this tool has undergone some major changes in the last release, hopefully in the near future, Autodesk will provide some better reporting mechanisms for us.

## Conclusion

If the pipes on your project aren't too complex and you're going to use the rational method to calculate the runoff for your project, go ahead and give the Analyze Gravity Network tools a chance. There's no need to export the data out to a different program, simply run the analysis right there in your drawing.