Chapter 1

Introduction

1.1 What is GEOPAK Drainage?

GEOPAK Drainage is a comprehensive system for designing and analyzing storm drain systems, which can leverage many roadway design features to create a seamless information exchange to the drainage design process. A GEOPAK Drainage project may contain multiple drainage networks; each comprised of any number of topologically connected drainage areas, inlets, pipes and ditches. The GEOPAK Drainage workflow closely mirrors conventional design processes allowing for the design of the surface collection system (i.e. drainage areas, inlets) then the design of the subsequent conveyance system.

GEOPAK Drainage is extremely flexible, in that the hydraulics designer can create and manipulate elements of the system, while simultaneously seeing the effects. Interactive dialogs and design visualization make the process easy to learn and efficiently produce results. Manipulations and redesign are accomplished quickly and easily, whether it's moving a single inlet or developing an entire network. At any time during the process, customized reports can be generated to provide hard copy outputs.

Roadway alignments, vertical profile gradelines, and digital terrain models created as part of the design process may be used throughout GEOPAK Drainage to provide pertinent information to drainage design. All drainage components feature interactive graphical placement tools for easy spatial definition of the drainage system. Visualization tools in GEOPAK Drainage for networks, drainage components, and computations all but eliminate the tedium of thumbing through pages of computer output to evaluate whether the system is functioning adequately. The visualization allows for immediate evaluation of the drainage system.

GEOPAK Drainage hydrologic and hydraulic capabilities include runoff computations, inlet design and analysis, and pipe and ditch design and analysis. All computations follow recommended methodologies in the FHWA publication “Drainage of Highway Pavements” as well as the procedures in the AASHTO Model Drainage Manual.

Runoff computations are performed using either the Rational or SCS method where rainfall parameters may be specified with common intensity equations, hydrographs, or by tabular intensity-duration data. Drainage area delineation tools allow easy creation of contributing areas and the graphical assignment of these to drainage features. The extremely tedious task of compiling different subareas and runoff coefficients is practically eliminated by GEOPAK Drainage’s graphical shape tool, allowing easy creation of drainage areas and land use boundaries. Once the boundaries are defined, the runoff coefficient and associated hydrologic parameters are computed automatically by delineating the land use zones inside of the drainage area. Any subsequent modifications to the boundaries will update the runoff coefficient via a single mouse click.

For GEOPAK Drainage support, please contact the CADD Support Center.

1.2 Objectives

- Understand the components of GEOPAK Drainage
- Understand the contents of the Drainage Library
- Learn how to set Project Preferences

1.3 Project Workflow

The GEOPAK Drainage workflow mirrors the conventional design process beginning with the design of the surface collection system (inlets, drainage areas) followed by the design of the conveyance system (subsurface pipes, channels).

Roadway alignments, vertical profile gradelines and digital terrain models may be used throughout GEOPAK Drainage to provide pertinent information to the drainage design. All drainage components feature interactive graphical placement tools for easy definition of the drainage system.
Each of these components (inlets, areas, pipes, and other objects) are composed of two basic types of information:

Spatial information describing its location, shape and connectivity.

Hydraulic and hydrologic information describing its properties, conventions and other associated attributes.

1.4 Components

GEOPAK Drainage organizes the components of a drainage system according to their spatial characteristics. Spatial information is stored as Nodes, Links and Networks.

Nodes: A node (inlets, manholes, etc.) is a point with a user-defined location. The location may be in Cartesian coordinates (x,y) or in curvilinear coordinates (station, offset). Nodes are used to model:

- Inlets (curb inlets, grate inlets, slotted drains)
- Junctions (manholes, grade breaks, bends)
- Outlets
- Others (user-defined)

Links: A Link represents a linear feature depicting a path connecting two nodes, traversing upstream to downstream. The path may be straight line or curvilinear (along a graphic element). Links are used to represent:

- Pipes (over +1,000 default pipes)
- Ditches (any shape – including automatic design capabilities)
- Channels (cross section based utilizing TIN files)

Networks: A network is a series of interconnected nodes and links that form a system through which water can flow to a single outlet node. A single project accommodates any number of Networks of varying types.

Other associated components in GEOPAK Drainage include:

Areas: A drainage area can be represented by a closed boundary or simply keyed-in (acres or hectares). All flows from a single drainage area are tributary to a single Node. Thus, there is a one to one correspondence between a node and an area. Therefore areas and nodes share the same name (ID). A drainage area may contain multiple subareas representing homogeneous features such as soil types and land uses (“C” values), thereby allowing composite "C" value calculations.

Profiles (Hydraulic)**:** A profile represents a path connecting any two nodes in a Network. Profiles provide the visualization of profiles between any two nodes in a drainage network.

1.5 Invoking GEOPAK Drainage

GEOPAK Drainage is part of the GEOPAK suite of civil tools and needs to run within the MicroStation environment. Follow the steps listed to access GEOPAK Drainage.

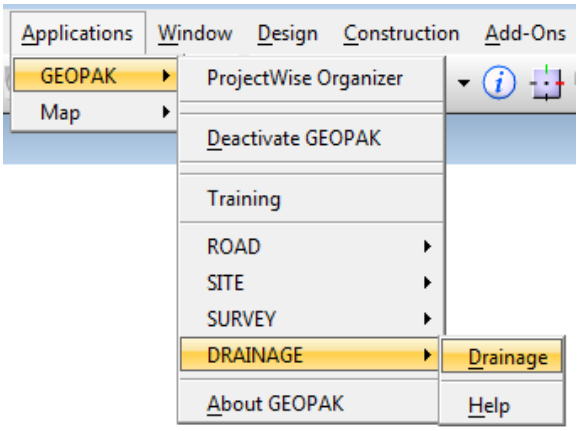
1. Launch ProjectWise Explorer and navigate to the following location:

District CADD\ Design\Cole#\J5EX501\Data

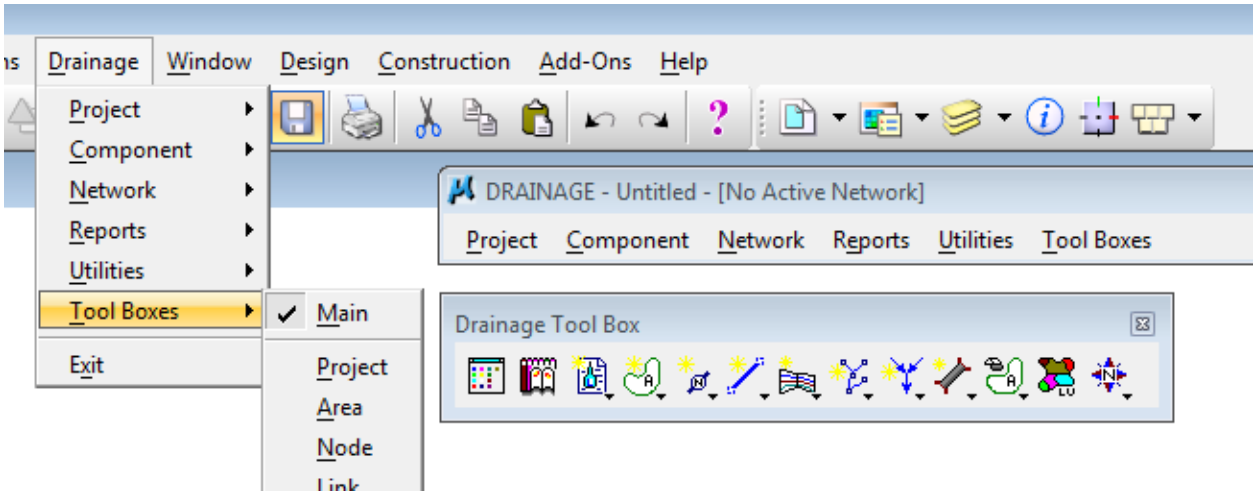
2. Find the following file and open it with MicroStation:

plan_J5EX501.dgn

3. From the Applications pull-down menu choose GEOPAK > Drainage



Once activated, a separate menu for Drainage will show up next to the Applications menu. All items in Drainage can be accessed through this main **Drainage** pull-down menu. You can also use the **Drainage Tool Box** by choosing the **Main** option under the **Tool Boxes** menu or use the **Drainage** menu bar which is displayed whenever Drainage is active:



Every time GEOPAK Drainage is activated it opens as an untitled project and is set to use the Bentley default preferences. In order to have the correct default MoDOT preferences you will want to change this by using one of the procedures outlined below.

Beginning a New Drainage Project - When you are starting a new project and have activated GEOPAK Drainage for the first time on that project:

1. Click the Project pull-down menu on the Drainage menu bar and select the Preferences option
2. Click the File pull-down menu on the Preferences dialog box and select the Open option
   - The Open Drainage Preferences dialog should pop up and the folder location should be set to your project data folder
3. Select MoDOT_Drainage.dpf from the list of available files and click the Open button in the lower right corner of the Open Drainage Preferences dialog box
   - The MoDOT_Drainage.dpf file simply changes all of the variables in the Drainage Preferences to reflect the suggested MoDOT settings
4. Click the OK button on the Preferences dialog box to apply the new preference settings to your drainage project
   - You will be prompted to save your drainage project at this point
   - This must be done before you can change preference settings
5. Choose the No Wizard option and click OK
6. In the Document section of the Save Drainage File dialog box change the Name and the File Name to the name you would like to use for your drainage project file
   - The system automatically fills in these fields using the name of the DGN file you are currently working in and giving it a GDF extension
   - The name you use much match in both the Name and the File Name fields
   - It is not necessary to change the Description but it can be helpful later on if the project is large and might require the use of more than one GDF file
7. Click the Save button on the Save Drainage File dialog box
8. Click the Yes button when the system prompts you to store the preference changes
9. Now you can open the Preferences again and make any changes necessary for your project situation then click OK to apply to your GDF

Opening an Existing Drainage Project - The GDF file stores the preferences that were assigned at the time the project was last saved and so if you have an existing project then you simply need to open your GDF file to restore your preferences and settings:

1. Click the Project pull-down menu on the Drainage menu bar and select the Open option
2. Select your existing GDF file from the list and click the Open button in the lower right corner of the Open Drainage File dialog box

Once you have done either of these processes the banner for your Drainage menu bar and the MicroStation window should display your drainage project name.

4. Follow the instructions provided on the previous page for beginning a new drainage project.
- **For step 4 we will make changes to the Preferences as we progress through the following pages and review the different options**
- **When you reach step 6 in the instructions use the following name for your file:**

drainage_J5EX501.gdf

A little information about the GEOPAK Drainage File (GDF):

The GEOPAK Drainage File or GDF stores the drainage project data in much the same way that the GPK file stores the data for GEOPAK Road.

A single GDF file can be used to store data for multiple networks (individual drainage systems). However, the GDF file format can become corrupted if pushed too far and it is recommended that if networks are large/complex that each network should be stored in a separate GDF file.

The GDF file can be corrupted if more than one user accesses it at one time, so the GDF file should be stored in ProjectWise and not on the T: drive.

As mentioned in the instructions given for opening an existing drainage project, the GDF file stores the preferences that are assigned when the project is saved and those preference settings will be restored when the GDF file is opened for use.

This page intentionally left blank

1.6 Review Project Preferences

The Project Preferences control the graphic and computational options of the drainage system. Each category of options contains variables that are set to MoDOT suggested standard settings. These settings can be manipulated by the user if needed during the design process.

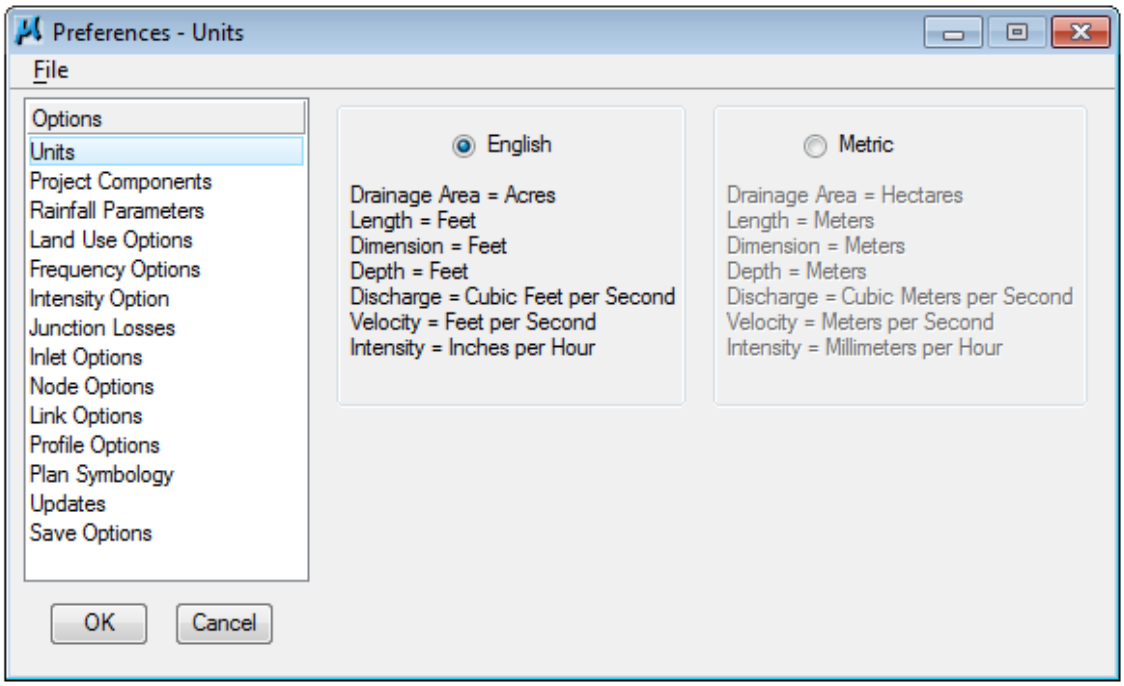
1. Click the Project drop-down menu and select Preferences.

For each of the Options on the Preferences dialog box we will be changing the settings to fit our project needs. The values we will be using are highlighted. Please make sure your preferences are set up in a similar manner.

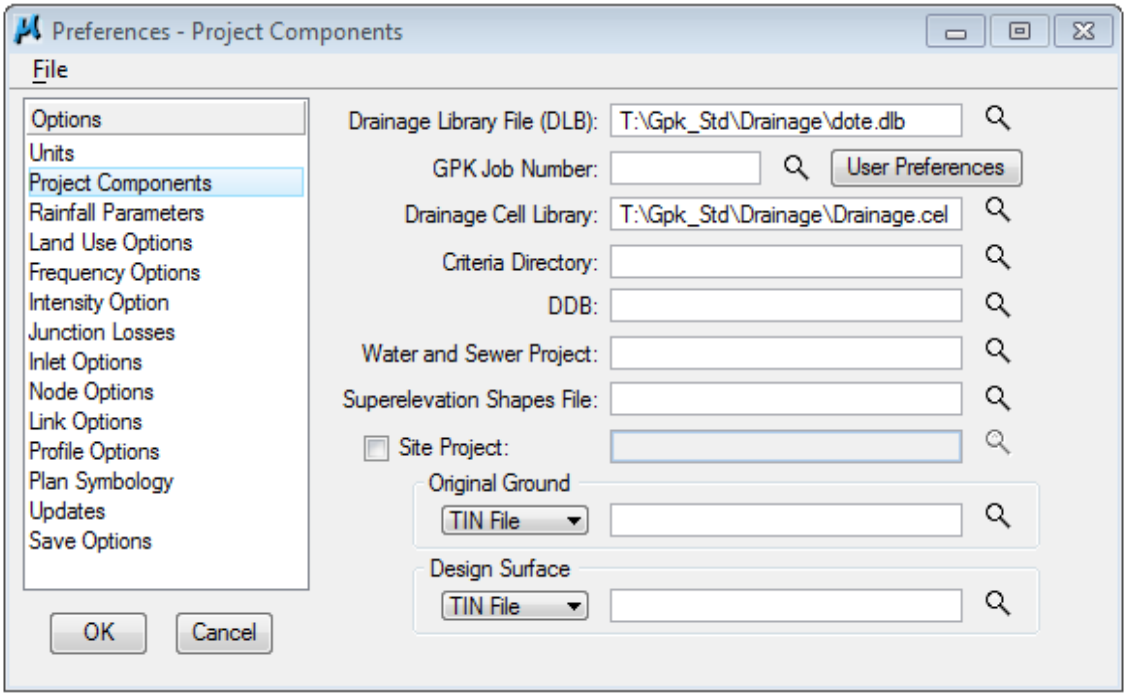
Units

This option defines what system of measure the project will use when performing calculations and generating output. It also provides the user with information about what units of measure are being used for specific input and output fields.

English



Project Components



Drainage Library File (DLB): **T:\Gpk_std\Drainage\dote.dlb**

GPK Job Number: **job###.gpk** (GPK file for the job you are currently working on)

When setting up the preferences for the first time on a Drainage project the user needs to set the path to the GPK:

1. Click the User Preferences button to the right of the magnifying glass
2. Click COGO Preferences button on the User Preferences dialog box
3. Set Job (GPK) Directory to the correct T: drive data folder for the current job
4. Click OK for the COGO Preferences and again for the User Preferences
5. Once this path is established the user can click the magnifying glass button and select the correct GPK file from the list.

Drainage Cell Library: **T:\Gpk_std\Drainage\drainage.cel**

Criteria Directory: MoDOT does not use criteria for Drainage.

DDB: MoDOT does not link Drainage to the GEOPAK D&C manager database at this time.

Water and Sewer Project: MoDOT does not use Water and Sewer projects.

Supereleveation Shapes File: This field can be pointed to the superelevation shapes file (DGN) allowing Drainage to read elevations, cross slopes, and grades for jobs with roadway sections that are superelevated
The superelevation shapes file must also be referenced into the DGN file you are using for your drainage project

Site Project: This field can be pointed to the GEOPAK Site File (GSF) for a Site project allowing Drainage to use data from a Site model or object

Original Ground: MoDOT does not typically assign an original ground surface

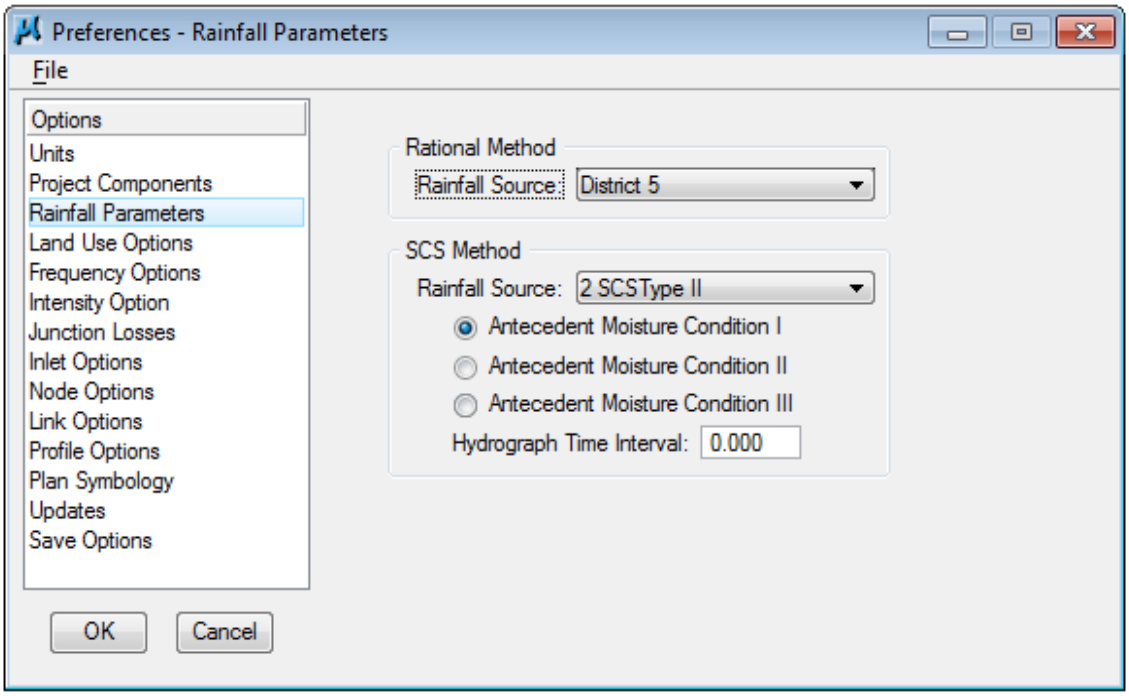
Design Surface: User can assign a final design (proposed) TIN file that will allow Drainage to read and use surface elevations when working with components

Rainfall Parameters

The rainfall Parameters option is used to select the intensity equation to be utilized on the final project. Both the Rational Method and the SCS Methods are available.

Section 749.4 of the EPG states that the Rational Method should be used on all watersheds less than 200 acres in size. For further guidance please refer to Category 749 of the EPG.

Select District 5 as the Rainfall Source under the Rational Method.



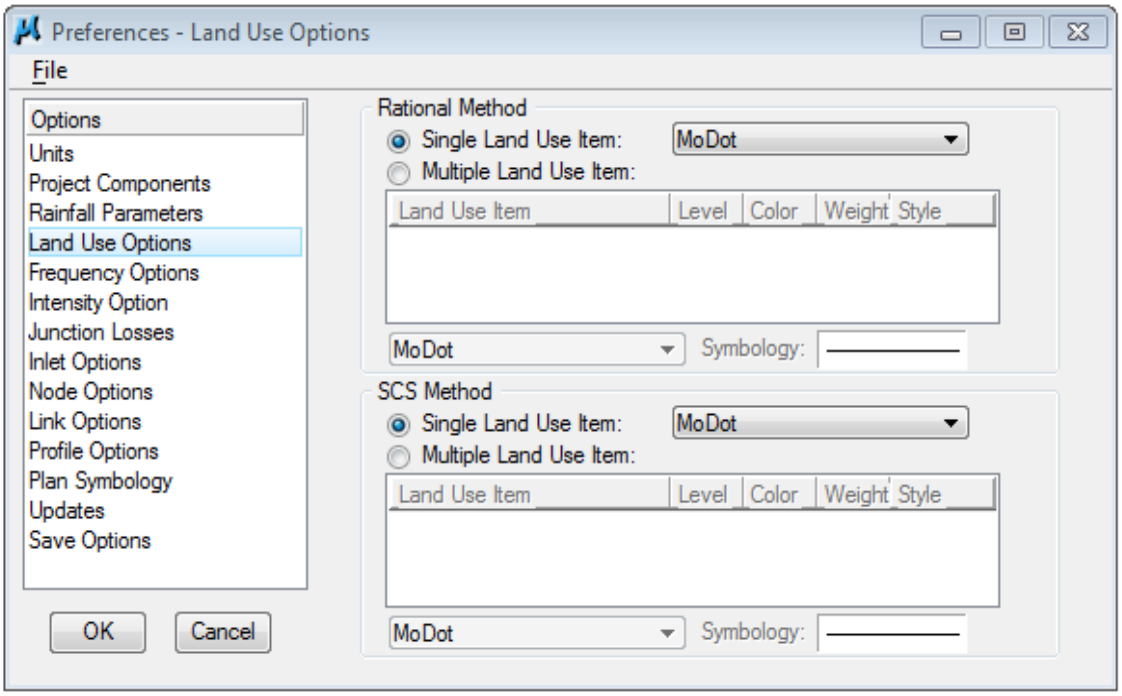
Rational Method Rainfall Source: The Rainfall Data Source for computing intensities and discharges, which is stored within the Drainage Library File, therefore it may only be selected here.

SCS Method: Rainfall Source is the data source for computing discharges, which is stored within the Drainage Library File - therefore, it may only be selected here. The Antecedent Moisture Condition I is the lowest runoff potential (dry soil), while Antecedent Moisture Condition II is the average condition, and Antecedent Moisture Condition III is the highest runoff potential (saturated soil).

Hydrograph Time Interval: Specified in terms of minutes.

Land Use Options

The Land Use dialog establishes the land use options used to delineate sub-areas and runoff coefficients for the current project. When selected from the Options list, the dialog depicted above is displayed.



Single Land Use Item The land use source from the Drainage Library containing the runoff coefficients and symbology for land use delineation may be selected from the list box.

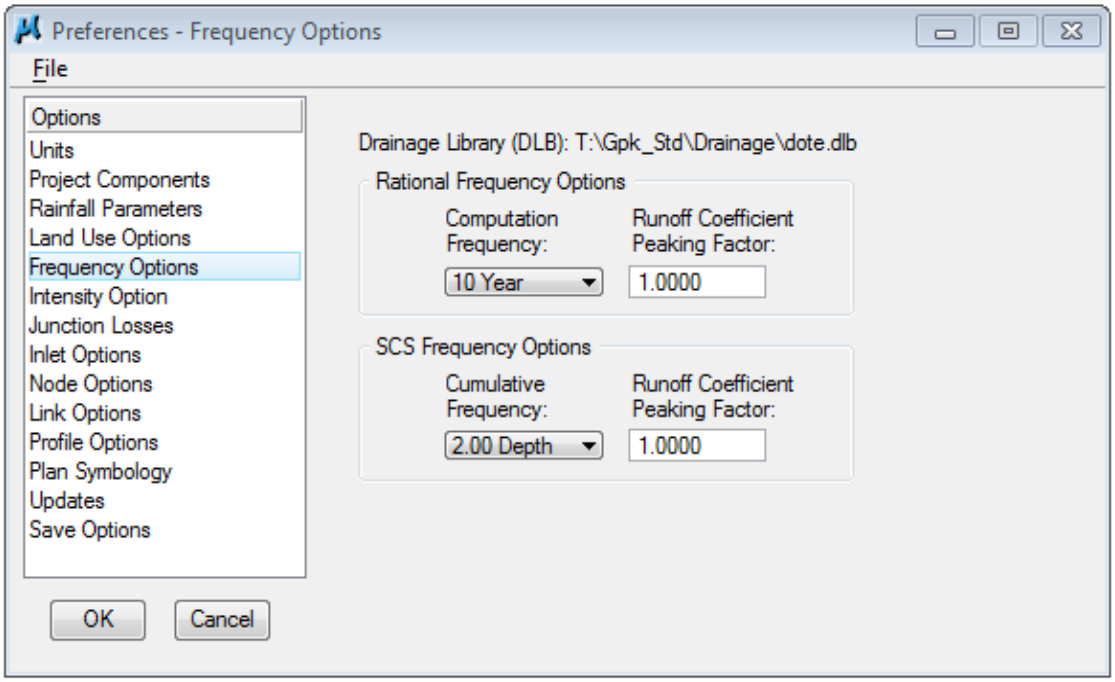
Select Single Land Use Item and make sure MoDot is set in the list box

- This will allow you to choose from all the predefined land use types and their corresponding C values contained in the MoDOT library when assigning land uses.

Multiple Land Use Item Allows the use of multiple categories if available in the specified Drainage Library. To select, utilize the down arrow in the list box and select the desired option. The item selected must be associated with a symbology that matches the symbology of each of the main land use shapes in the design file. **Our library does not contain multiple categories, so this option is not used at MoDOT.**

Frequency Options

The Frequency Options dialog establishes the computation frequency and runoff peaking factors for discharge computations for the current project.



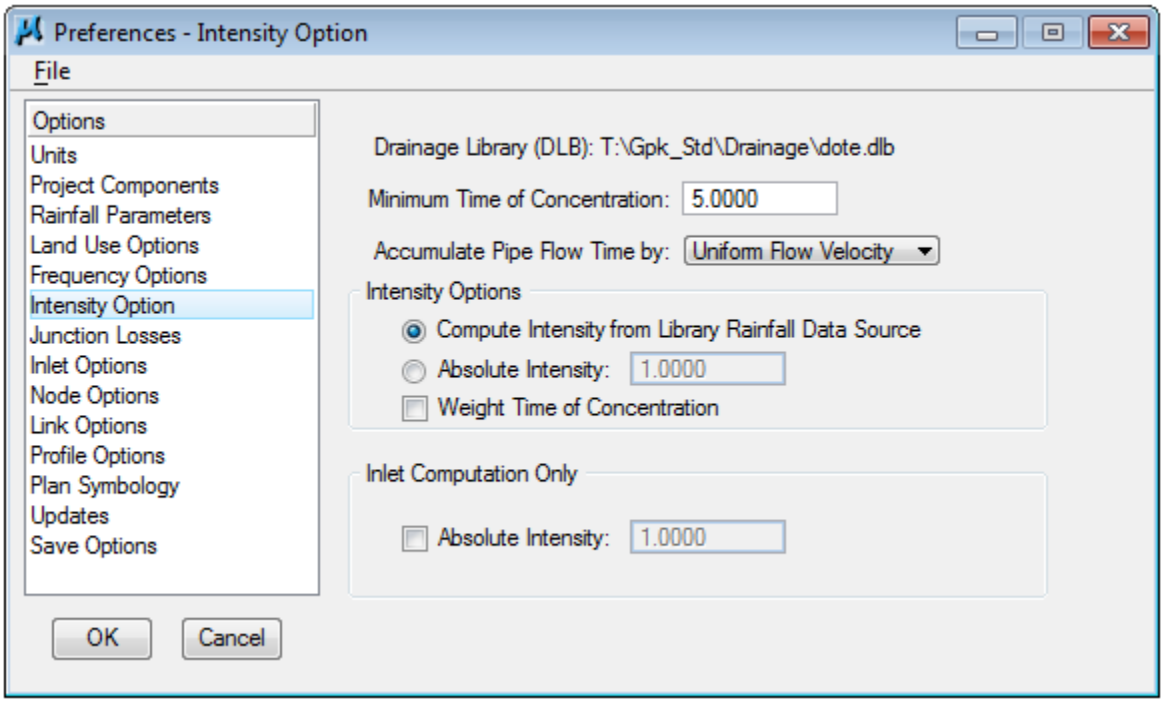
Drainage Library File (DLB) The current Library File as specified in the Project Component option is displayed. It cannot be changed from this dialog, only in the Project Components.

Computation Frequency The displayed frequencies are selected from the available entries in the Rainfall Parameters chosen from the library.

Cumulative Frequency The displayed depths are selected from the available entries in the Rainfall Parameters chosen from the library.

Peaking Factors These factors are used to adjust runoff coefficients for frequencies other than the 5-year or 10-year options. The frequency correction factor is 1.10 for the 25-year event, 1.20 for the 50-year event, and 1.25 for the 100-year event. However, the resulting runoff coefficient (original "C" multiplied by the frequency correction factor) may not be greater than 1.0 (EPG 749.5.2.3).

Intensity Options



Minimum Time of Concentration: 5.0000
The Minimum Time of Concentration is used for all drainage area computations and pipe travel time in discharge computations and is used throughout these computations as the minimum. In pipe hydraulic computations this value will be used until the actual time of concentration within the systems exceeds this value. Specified in terms of minutes.
See EPG Section 749.5.3 for Time of Concentration information.

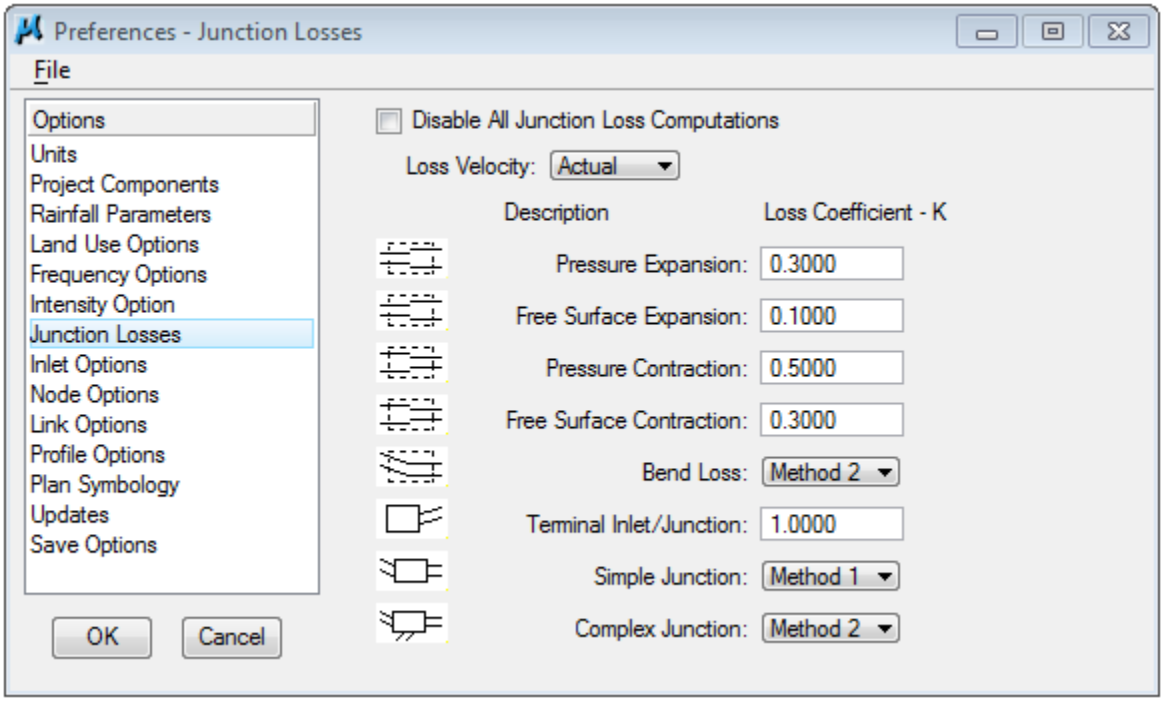
Accumulate Pipe Flow Time by: Uniform Flow Velocity
The Accumulate Pipe Flow options adjust the travel time through the pipe. Options include Uniform Flow Velocity (uses velocity associated with normal depth to establish travel time), Full Flow Velocity (uses velocity associated with full flow [V=Q/A] to establish travel time) , and Iterative Velocity (GEOPAK analyzes the network under three iterations, so velocity and travel time closely match actual flow conditions).

Intensity Options Compute Intensity from Library Rainfall Data Source
The Weight Time of Concentration is used when combining different types and times of concentration, activating this toggle weights the times of concentration such that the discharge will not decrease.

Inlet Computation Only The Absolute Intensity provides a constant intensity that overrides the Drainage Library File and utilizes the specified value for discharge computations required for inlet and spread calculations only. Network hydrologic discharges will be computed as specified in the Intensity Options. Note, however, for the value to be utilized, the toggle to the left of the Absolute Intensity within the Inlet Computation Only box must be activated or the value will be ignored. Specified in terms of in/hr or mm/hr depending on the project units.

Junction Losses Options

AASHTO default coefficients and Method 2 are used to compute junction losses.



Disable All Junction Loss Computations
If the toggle is active, no Junction Loss computations are calculated.

Note: If a single or small number of Junction Losses are not desired, an option is provided later in the design to accomplish this. The toggle here applies to the entire project.

Loss Velocity
Losses can be computed on Actual or Full Flow.

Pressure Expansion
Loss coefficient to be used in pressure (full flow) expansions within the system.

Free Surface Expansion
Loss coefficient to be used in free surface (partial flow) expansions within the system.

Pressure Contraction
Loss coefficient to be used in pressure (full flow) contractions within the system

Free Surface Contraction
Loss coefficient to be used in free surface (partial flow) contractions within the system.

Bend Loss
Specify the methodology and source of bend loss computations.

- Method 1 utilizes Modern Sewer Design, while Method 2 utilizes AASHTO methods.

Terminal Inlet / Junction
Loss coefficient to be used at terminal inlets or junctions (no upstream pipes) within the system.

Simple Junction
Specify the methodology for computing junction losses at simple junction (one pipe into the junction and one pipe out).

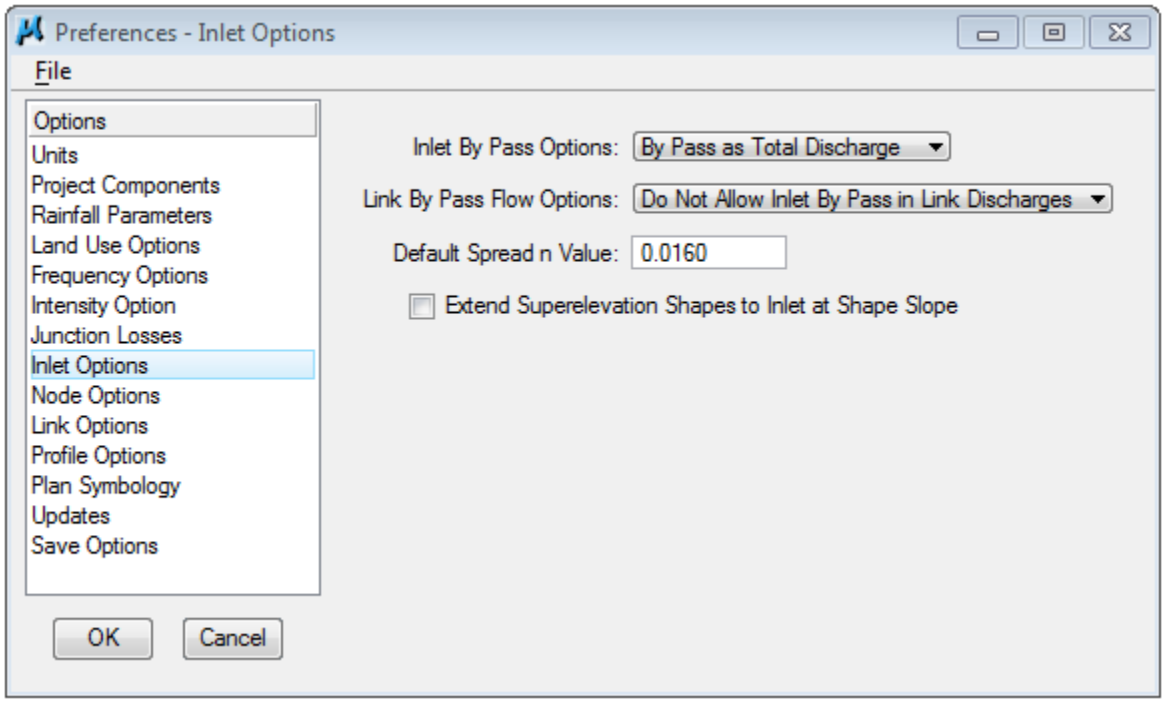
- Method 1 utilizes Modern Sewer Design, while Method 2 utilizes AASHTO methods.

Complex Junction
Specify the methodology for computing junction losses at complex junctions (more than one pipe in and only one pipe out).

- Method 1 utilizes Modern Sewer Design, while Method 2 utilizes AASHTO methods.

Inlet Options

The Inlet Options establish the default inlet variables to use in the Node Configuration Properties for each type of inlet.



Inlet By Pass Options: By Pass as Total Discharge

This option indicates the method by which Inlet by pass flows are accounted for in the system. **By Pass as Total Discharge** will account for by pass flows as the difference between the total discharge to the inlet and the inlet capacity. This difference, expressed in flow units, will be the total by pass from the inlet and can be added at a subsequent downstream inlet. **By Pass as C x Area Product**, on the other hand, is determined from the total bypass at the inlet as a product of runoff coefficient and drainage area that is contributing to the total bypass at the inlets computed intensity. This product of area and runoff coefficient will then be added to the composite runoff coefficient and area at the downstream inlet.

Link By Pass Flow Options: Do Not Allow Inlet By Pass in Link Discharges

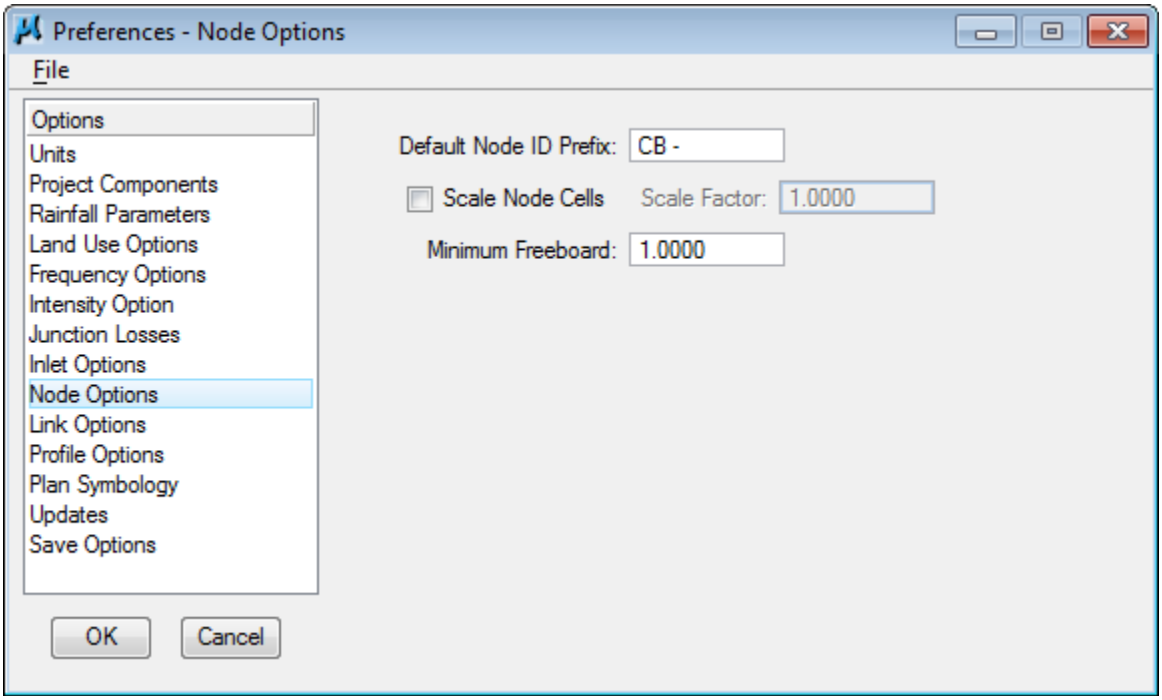
This option indicates the method by which Inlet By Pass flows are accounted in the Link discharge computations**. Do Not Allow Inlet By Pass in Link Discharges** will prevent the system from bypassing discharges from one inlet to the next when Link discharges are computed. The total discharge that reaches the inlet will be considered entering the Link at that point. This option in no way impacts the inlet computations, bypass flows will be reflected in these computations. **Allow Inlet By Pass in Link Discharges** will account for the discharge that bypasses the inlets when the Link discharges are computed. Only the discharge entering the inlet will be considered entering the Link at that location. Bypass flows will be directed to the appropriate inlet and the Link accordingly.

Default Spread N Value: 0.016

The default Manning's roughness coefficient for the spread hydraulic computations.

Node Options

Select the node-link graphical interface and node naming convention.



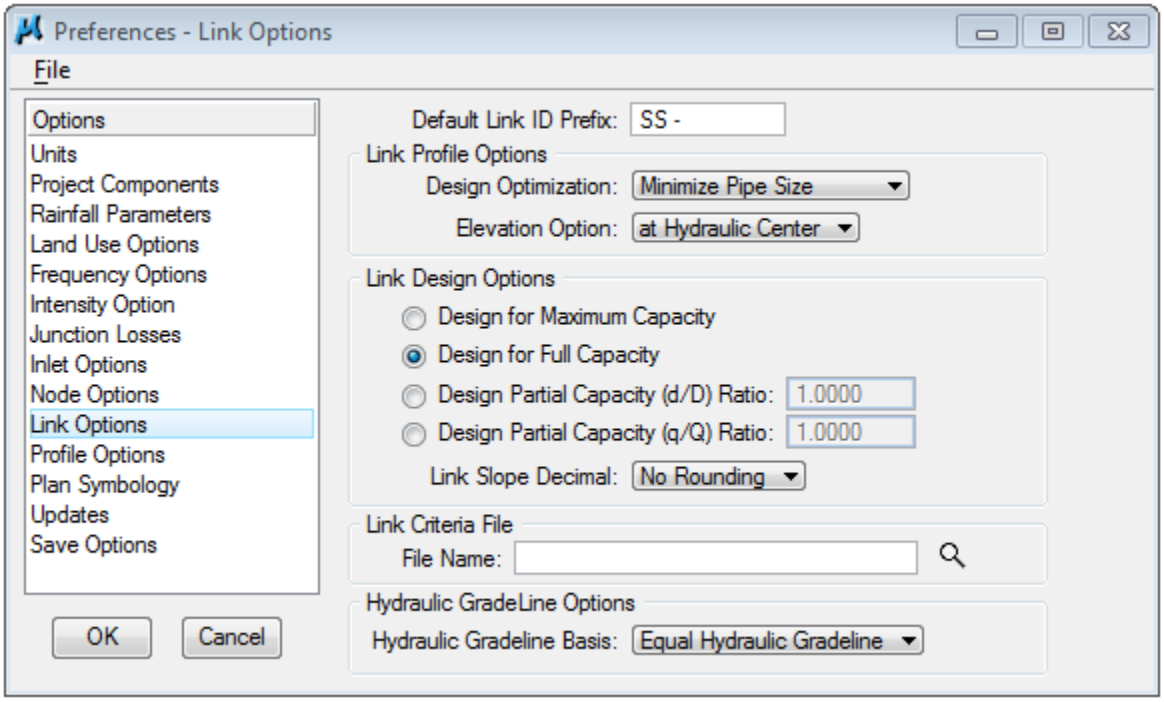
Default Node ID Prefix: CB –
All nodes will be identified as CB-# where # starts at 1and is numerically sequenced by 1 as additional nodes are added to the system. (Drainage Areas share the same ID's as Nodes.) Feel free to change the prefix. For large projects it may be beneficial to break up the project into different series: (A1, A2, …An) , (B1, B2, …Bn), etc.

Scale Node Cells should be toggled off. We design projects on a one to one scale. If it is on, just make sure it is at a Scale Factor of 1.0000

Minimum Freeboard: **1.0000** (see EPG Sec 750.4.4.7)
This option is used to maintain a certain amount of free space in the node from top of water surface to top of inlet. GEOPAK measures freeboard from the hydraulic grade line to inlet elevation.
Note: This option does not affect computations in any way. It is only used to trigger a warning (HGL Blowout) if the hydraulic grade line forces less than minimum freeboard.

Link Options

Set the preferred design options for the design of the conveyance system.



Default Link ID Prefix: SS –
All links will be identified as SS-# and start at 1 and numerically sequenced by 1 as additional links are added to the system. Feel free to change the prefix.

Design Optimization considers the pipe profile envelope created when nodes and links are placed and a minimum soffit and maximum invert are established.
Minimize Pipe Size uses the entire envelope to size pipes and will choose smaller pipe sizes with steeper slopes if possible. If the entire envelope does not result in a smaller pipe then the appropriate size pipe will be selected based on the top of the envelope.
Minimize Depth of Cover bases pipe sizing on the top of the envelope maintaining a minimum soffit elevation.

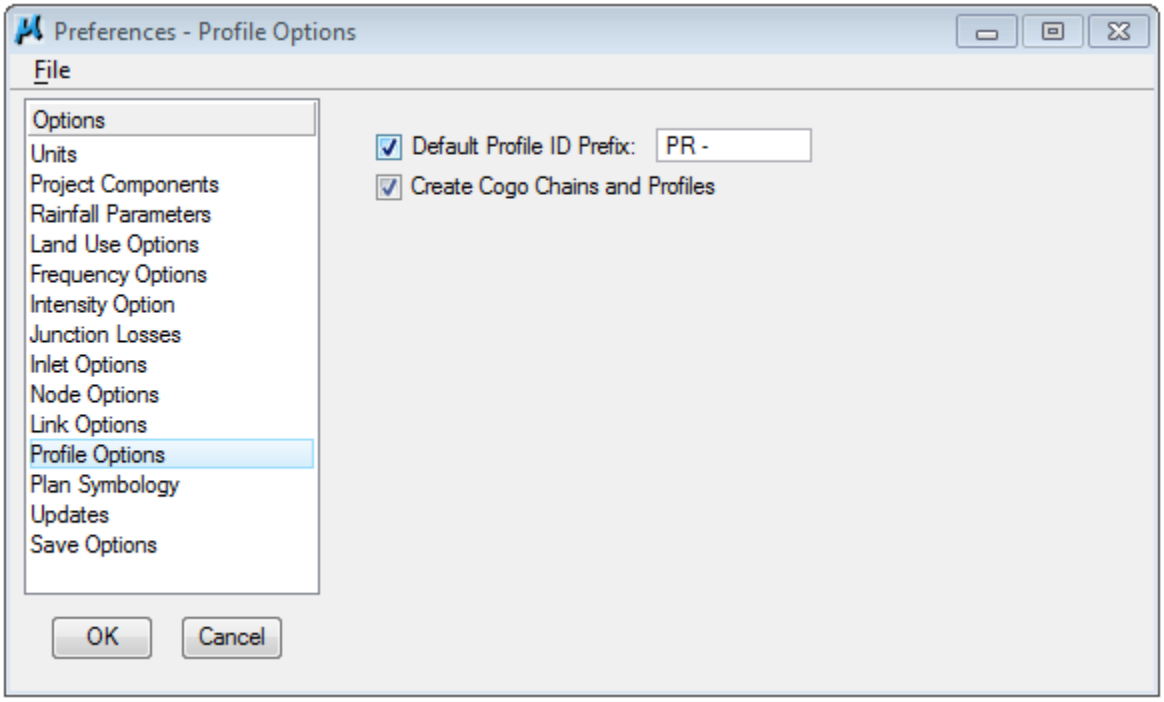
Elevation Option is used to establish the elevation that is reported for the ends of each link when a profile is generated. **Hydraulic Center** reports the elevation at the center of the node structure and **Actual End** reports the elevation at the actual end of each pipe.

Link Design Options: Design for Full Capacity is the preferred option and uses 100% flow to design the pipe size for capacity greater than the discharge.
Note: In comparison Design for Maximum Capacity assumes approximately 90% flow.

Hydraulic Gradeline Basis: Equal Hydraulic Gradeline is preferred and assumes that the starting lower end HGL of a link is equal to the upper end HGL of the downstream link.

Profile Options

GEOPAK Drainage allows users to store chains (hydraulic) and profiles (hydraulic) in COGO.



Default Profile ID Prefix: PR –
This prefix is added to the beginning of the each defined profile automatically and then numerically sequenced by one. For example, the first Profile stored would obtain default ID in this case of "PR–1" followed by "PR–2", "PR–3", etc. for subsequent Profiles. If the links are to be used for COGO chains or profiles, the length of this prefix should be within the parameters for chain and profile character limitations.

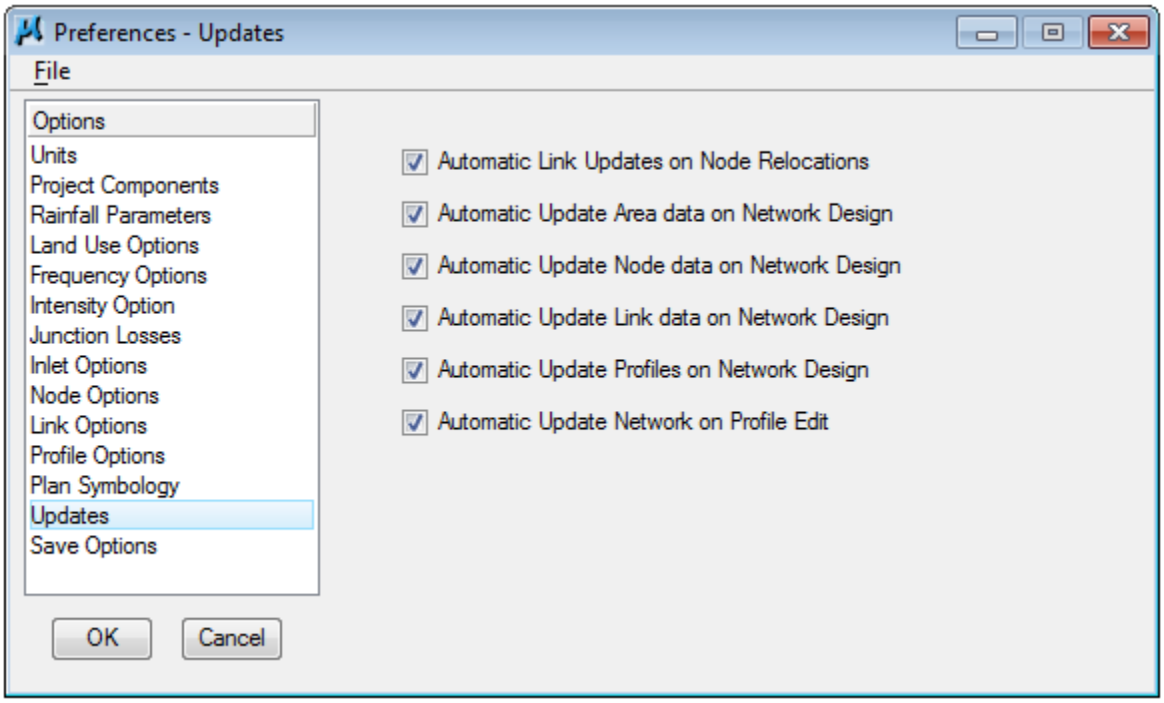
Create COGO Chains and Profiles
When toggled on, the links are stored in the coordinate geometry database (GPK) specified in the **Project Components**. The link name is used as both the chain and profile name. Stationing starts at 0+00.

Plan Symbology

The defined symbology is set to reflect the MoDOT CADD Standards. This symbology may be revised or manipulated at any time. To change the symbology for any item simply double-click that item and make the desired revision.

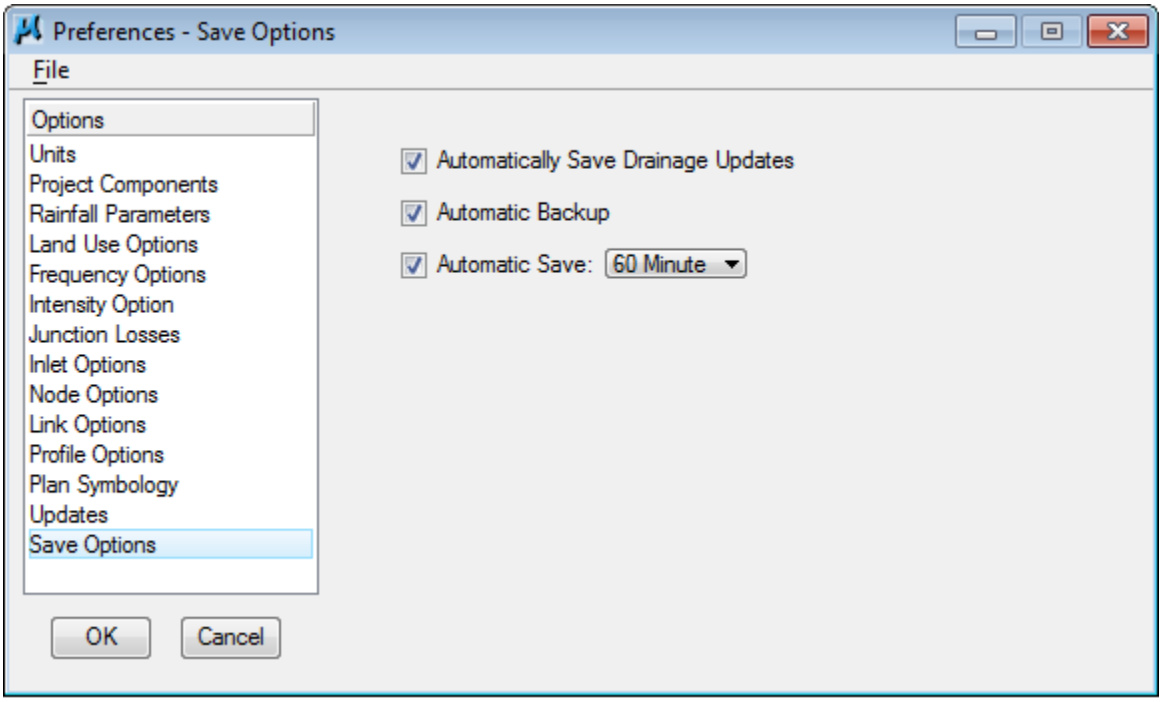
Updates

Choose to Automatically update the components in the current drainage file as changes are made to the system.



Save Options

Choose whether to Automatically save all drainage updates, save a backup or save at give intervals of time



Automatically Save Drainage Updates When activated, the Drainage project file (.gdf) is saved every time an Apply command is issued from any Drainage dialog.

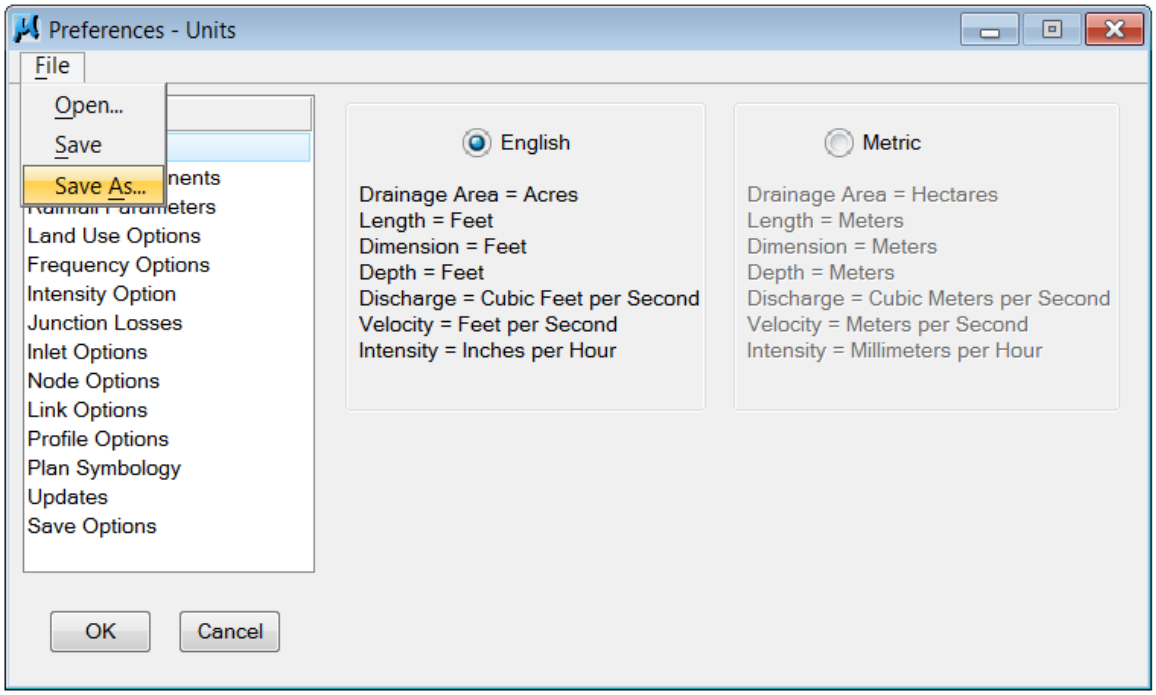
Automatic Backup When activated, a backup file is always created (in the project directory with a *.bak extension) when the project is opened.

Because of the corruption issues that we deal with from time to time on the GDF files, it is recommended that you enable the Automatic Backup option.

Automatic Save To automatically save, activate the toggle to the left of the option, then select the desired time interval.

Drainage Preferences may be changed at any time and the system can then be redesigned or analyzed utilizing the new settings.

There are times when you might want to set up different scenarios using different preference settings to see how a drainage system will act under different conditions. Preferences can be saved by going to the **File** pull-down menu on the **Preferences** dialog box and choosing the **Save As** option and saving the current preference settings as a new DPF (Drainage Preference File). These preferences can then be recalled anytime they're needed by using the **Open** option under the **File** pull-down menu.

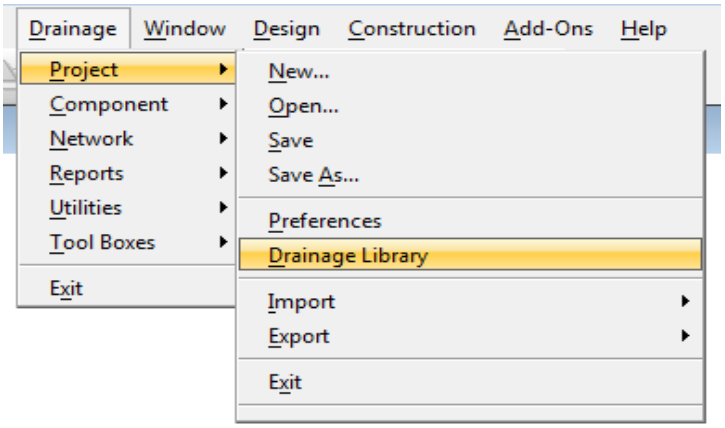


2. Click the OK button on the Preferences dialog box to enable the current settings and close the dialog.

1.7 Review Drainage Library

The Drainage Library is used to store data and standards that may be shared by different projects and designers. MoDOT supports one main library (**dote.dlb**). A metric library is also available as well as custom libraries if needed. These libraries are maintained at CADD Support, and should not be modified.

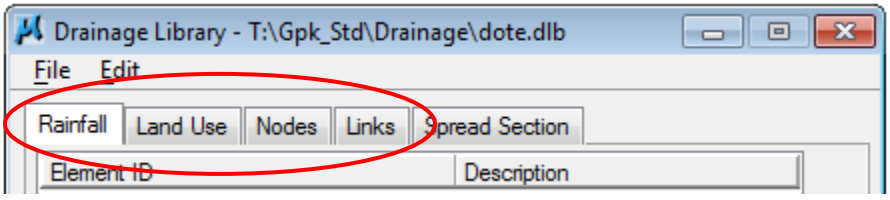
1. Click the Project drop-down menu and select Drainage Library.



When the Drainage Library dialog comes up, make sure the banner displays the path as **T:\Gpk_Std\Drainage\dote.dlb**

Note the first four tabs in the Drainage Library:

- **Rainfall**: Contains the Rainfall Data Sources
- **Land Uses**: Contains the specific Land Uses and their corresponding "C" value
- **Nodes**: Inlets, Junctions, Manholes, Outlets, etc.
- **Links**: Circular Pipes, Elliptical Pipes, Pipe-Arch pipes, Boxes, etc.

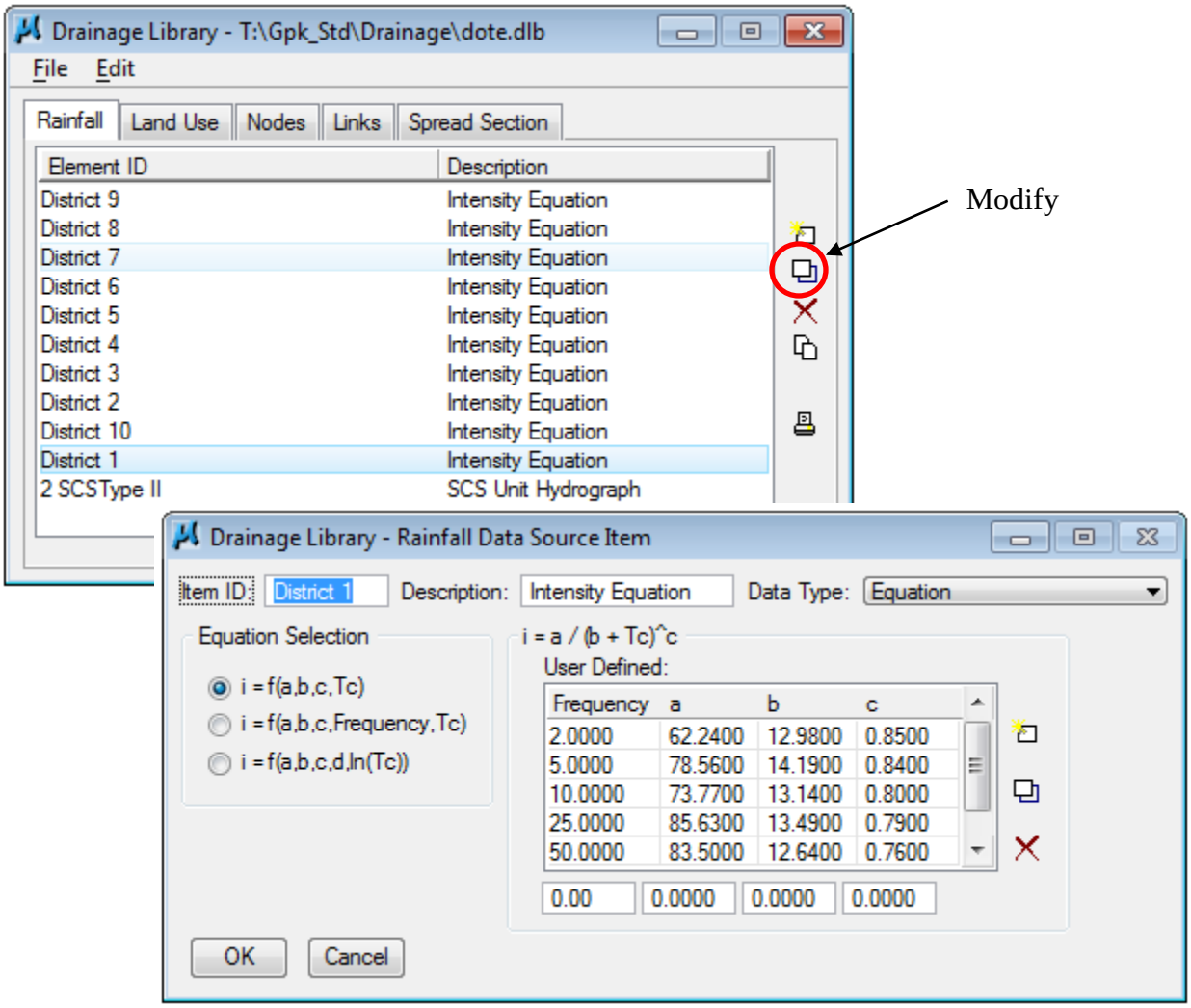


Rainfall Tab

Rainfall Items store all the rainfall data information to be used on GEOPAK Drainage Projects. GEOPAK Drainage supports rainfall sources in the form of intensity duration frequency (IDF) tables, or as coefficients for intensity-duration-equation formats. Also supported are the SCS, HYDRO-35 and TP 40 methods of rainfall.

This is the same information used to develop our IDF curves for each District. See the **EPG 749.5.4**.

To explore the data highlight any rainfall item, then right-click and select **Modify**. Or select the modify button on the right. Take a look at the defined information.
When you are finished reviewing the information select the **Cancel** button to exit the Rainfall Data Source Item dialog.



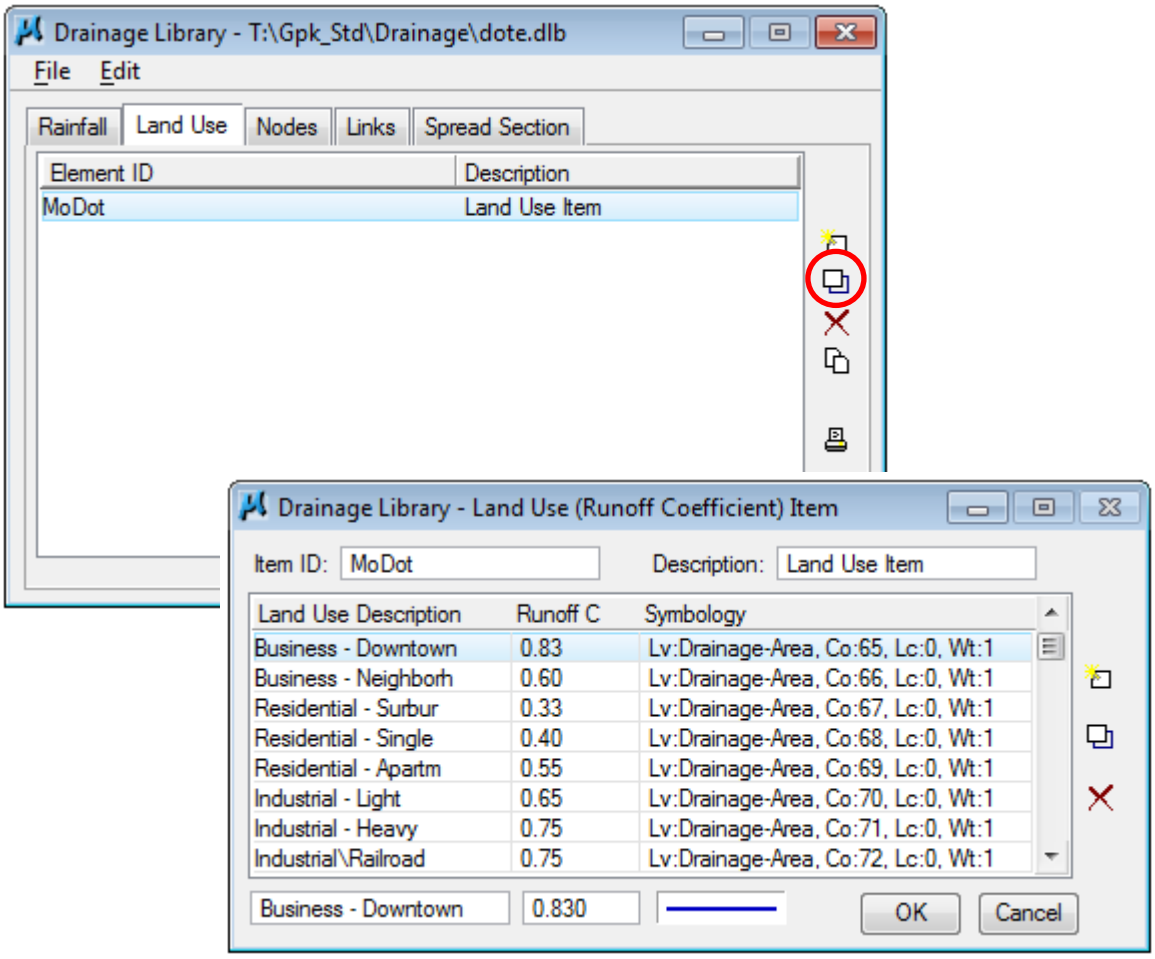
Land Use Tab

Land Uses are used to store runoff coefficients ("C" values) and corresponding symbology for each land use. The runoff coefficients are divided up into two groups: Rural and Urban. The rural coefficients are based on the topology and soil classification while the urban coefficients are defined based on the land use. See the **EPG 749.5.2**.

Land Uses can be automatically delineated with GEOPAK Drainage. The DGN file is scanned for the land use symbology from the Drainage Library and associates the symbology, as well as the graphical shape, with the "C" value as set in the library. The “C” values are then weighted according to their ratio of land area within the main area delineation, and a properly weighted “C” value is calculated for the land area under consideration. Alternately, the land uses can be put into the project manually.

To explore, highlight the **MoDOT** item, right-click and choose **Modify**
When done select **Cancel**.

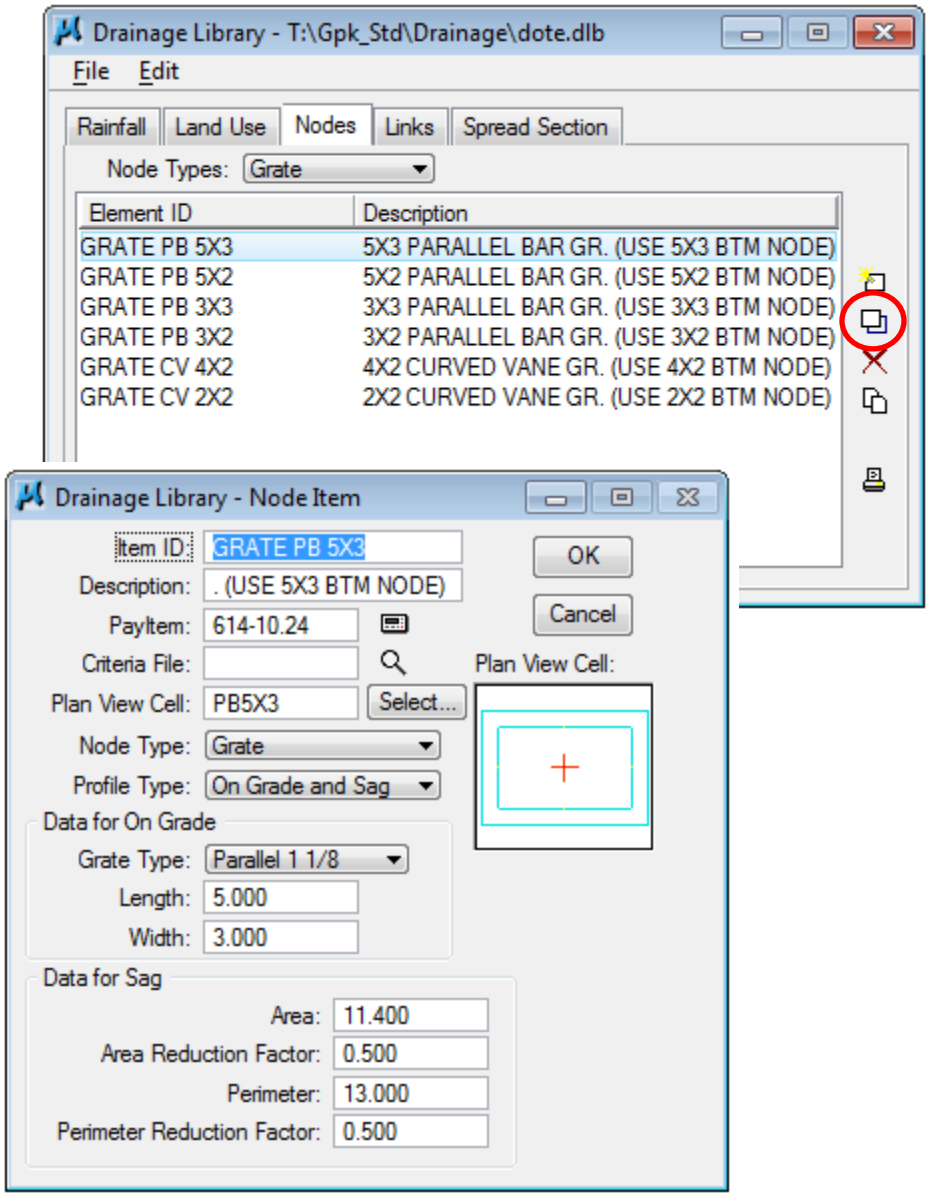
.



Nodes Tab

Nodes contain standard configurations for Grates, Curbs and Slotted Drain inlets, as well as Junctions, Outlets and Other Nodes. The description, plan view representation and dimensional information for each Node item is entered only once in the Drainage Library and then referenced by each project.

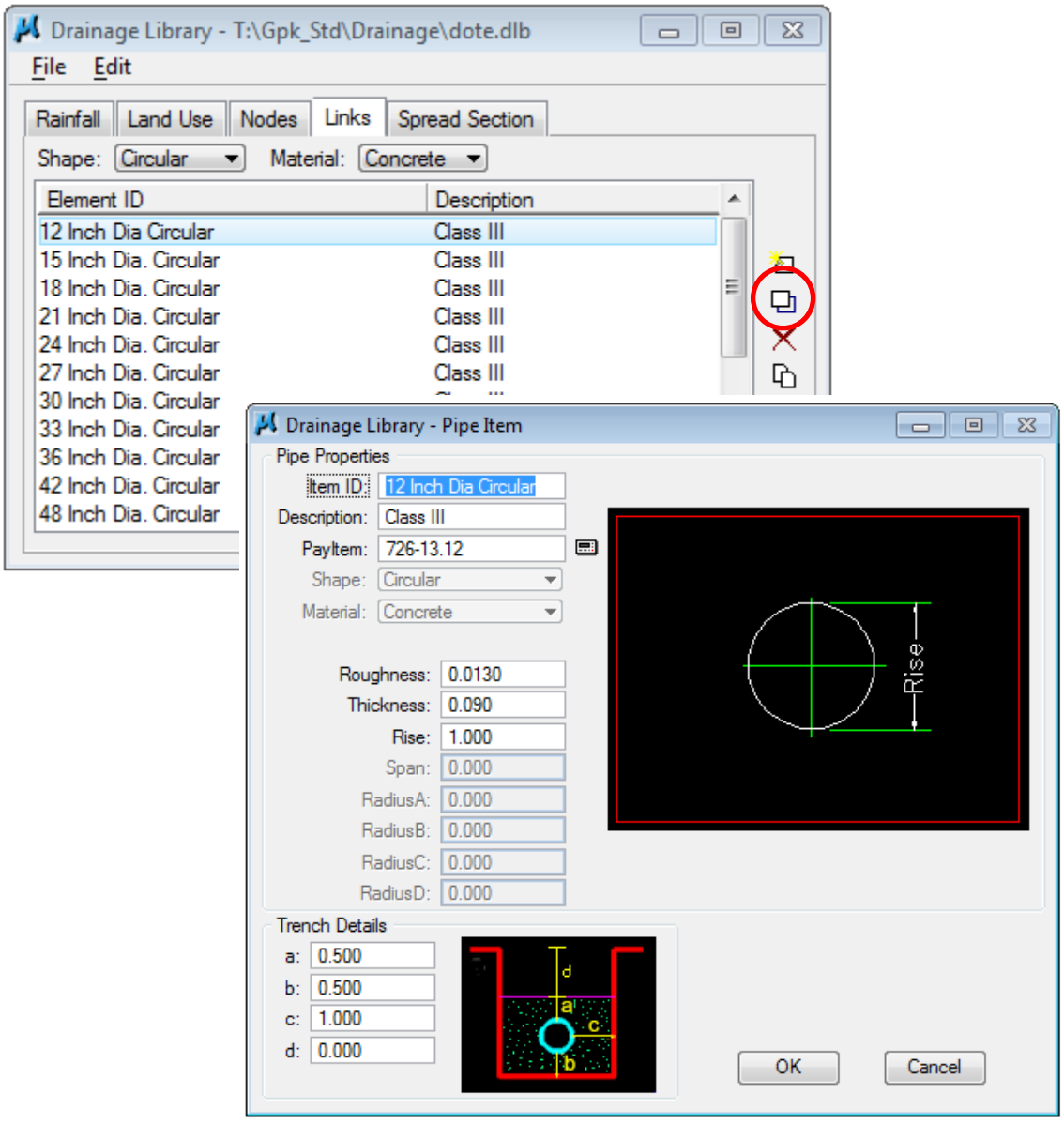
Choose the **Grate** option from Node Types, then select the first node and **Modify**.
You can review the cell information by pressing the **Select** button to the right of the Plan View Cell field.
When complete, close both the cell selector and the node item by choosing **Cancel**.



Links Tab

The Links are categorized by three properties for each pipe: Shape, Material, and Type (corrugation). Each pipe resides in its own dialog box that contains the specific pipe geometry, default roughness coefficient and material combination.

To explore the type of data common to the links, set the Shape to **Circular**, the Material to **Concrete**. Highlight the first pipe, then choose **Modify**.
When done select **Cancel**.



Spread Section Tab

The **Spread Section** tab is used to set up roadway section spread situations. MoDOT has not developed library items for this option.

More information will be given when setting Spread Criteria in the Node Configuration (Inlets) section of the class.

2. Exit out of the Drainage Library.

Chapter 2

Drainage Areas

2.1 Objectives

Introduce the designer to Drainage Areas and Land Uses in Geopak Drainage

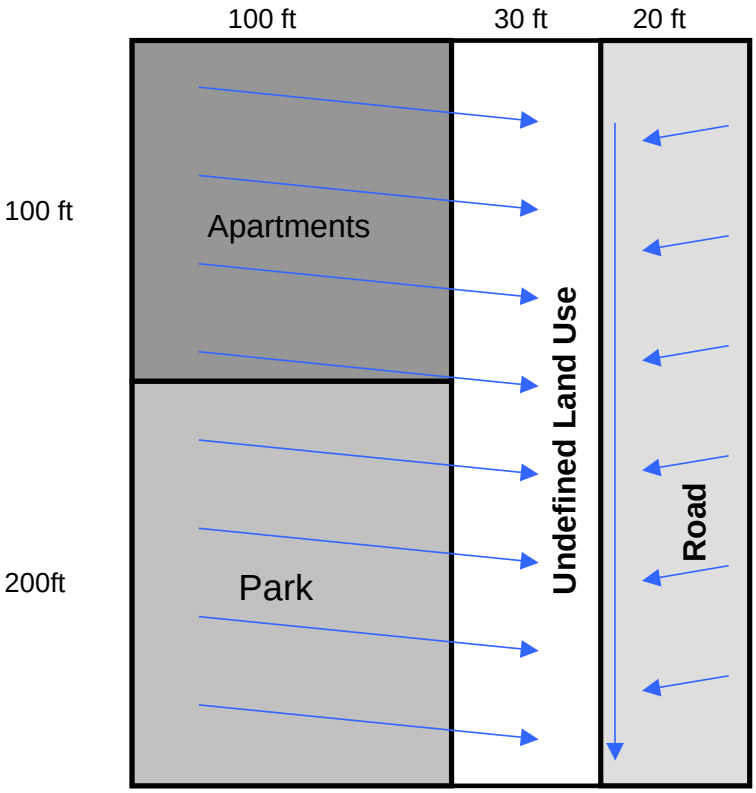
2.2 Overview

The term **drainage area** refers to a total extent of land that collects surface water and funnels that water to a low point that immediately impacts a roadway project. A drainage area can have just one land use type or might be made up of multiple land use types. A drainage area's land use types need to be defined in order to get the correct runoff value.

The **land use** types of a drainage area are determined by their surface type and functional use. These two factors are used to determine the amount of rain water that will be absorbed, how much will run off, and how quickly it will run off during a storm event.

For more information on drainage areas and land uses see Category 749

Establish land uses in the watershed which include an apartment complex, a park, and pavement as well as an area of undefined land use. All water drains to the gutter at the edge of the paved road and then to the downstream end of the watershed. The dimensions for the various land uses and the overall drainage area are given below. Assume a time of concentration of 10 minutes.

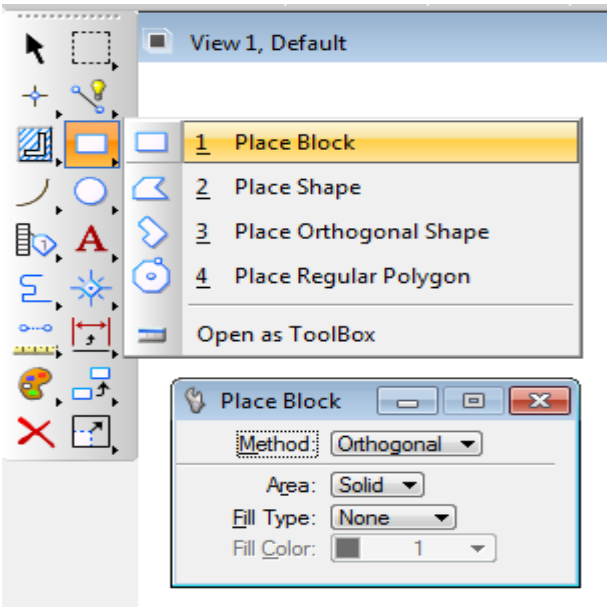


2.3 Basic Layout

Using the steps given below, draw Microstation shapes representing the overall drainage area and the areas that represent the individual land use types shown in the diagram above.

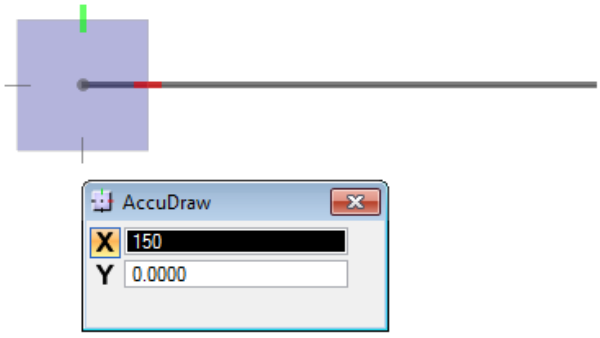
1. Make sure AccuDraw is activated

2. Select the Place Block tool

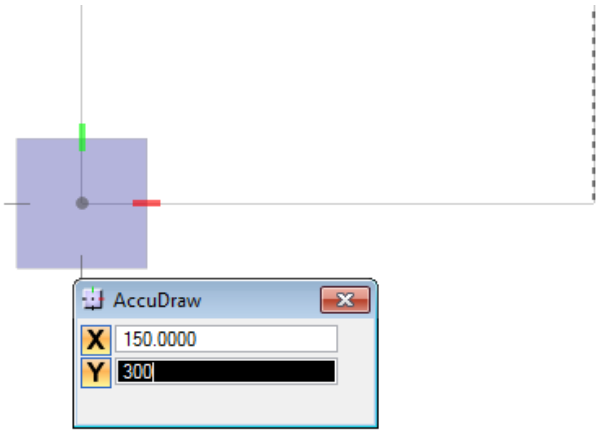


3. Data point at the location that will represent the bottom left corner of your drainage area

4. Move the mouse to the right (X direction) type in 150 to lock distance into AccuDraw



5. Now move the mouse up (Y direction) and type in 300 to lock in this distance, and hit Enter on the keyboard



You should now have the rectangle that forms the outside perimeter of the entire drainage area.

Now we can establish the land use areas that will provide runoff to our project.

6. Based on the supplied dimensions complete the areas that represent the apartments, park, undefined area, and road.

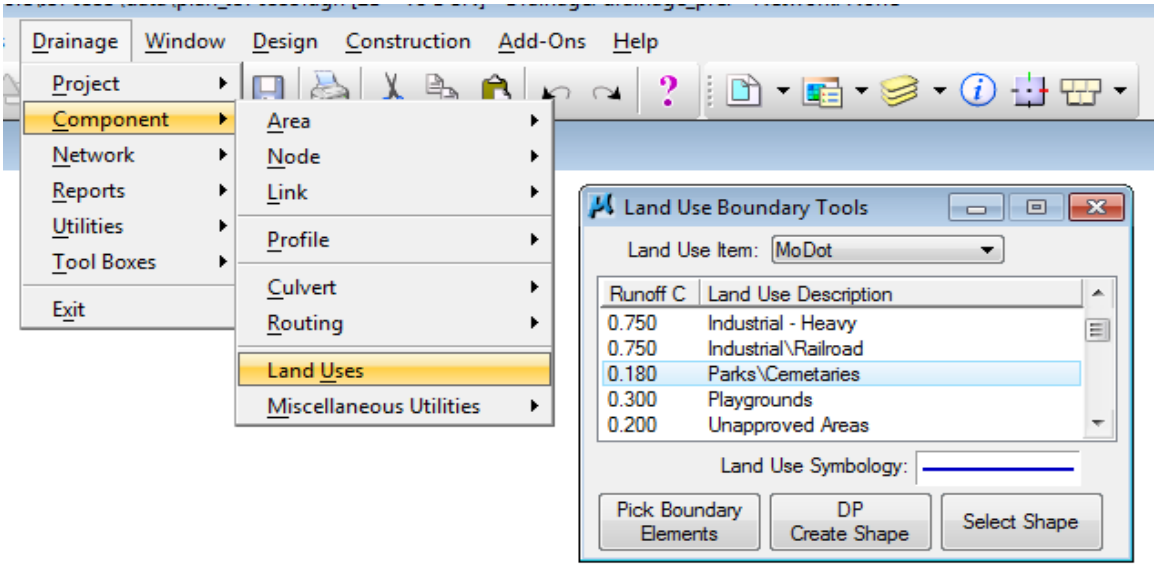
When finished, you should have drawn a total of 5 rectangles, 1 for the overall drainage area 4 more for each land use type inside the overall drainage area.

2.4 Define Landuses

1. From the Geopak Drainage file menu go to Component >Landuses

2. For the lower left hand area select Parks\Cemeteries

- **Click the Select Shape button**
- **Data point the edge of the lower left hand area to select the shape**
- **Data point a second time to accept the selection. The area should change to the assigned color in the drainage library (In this case dark blue)**

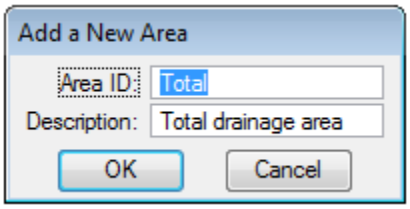


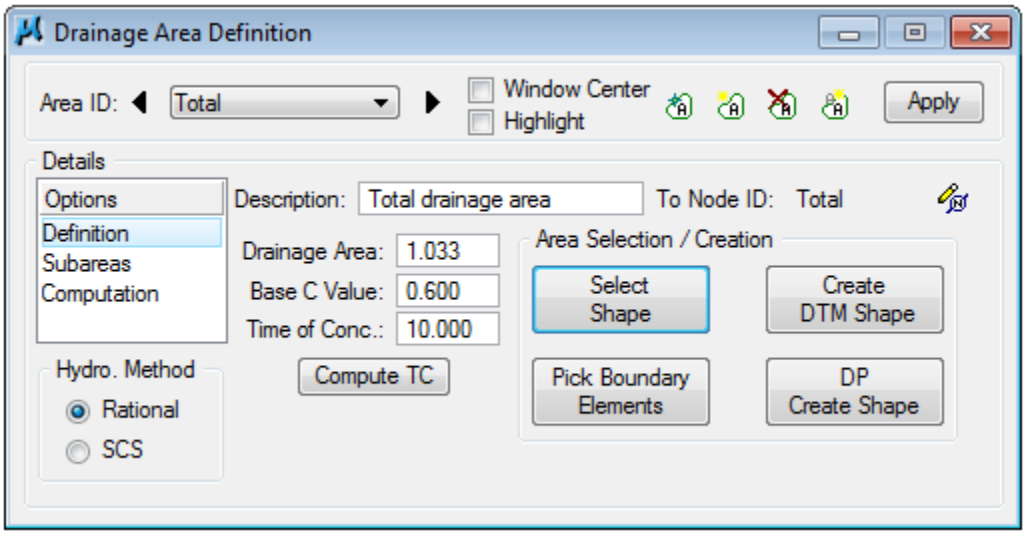
3. Based on the supplied descriptions assign the remaining land uses. When finished you should have 3 assigned land uses.

2.5 Define Drainage Areas

1. Go to Component >Area >Add

- **Notice the Drainage Area ID defaults to CB – 1. For this example change the ID to Total**
- **In the Area Definition region click the Select Shape button**
- **Data point on the rectangle that represents the entire drainage area**
- **Data point a second time to accept the selection (The drainage area field should now contain the area in acres)**





2. Enter in the time of concentration (Tc) as 10.000

If the time of concentration entered is below the minimum Tc set in the preferences, the program will default to the minimum to compute the peak discharge for the area.

3. Enter in the base C value as 0.6

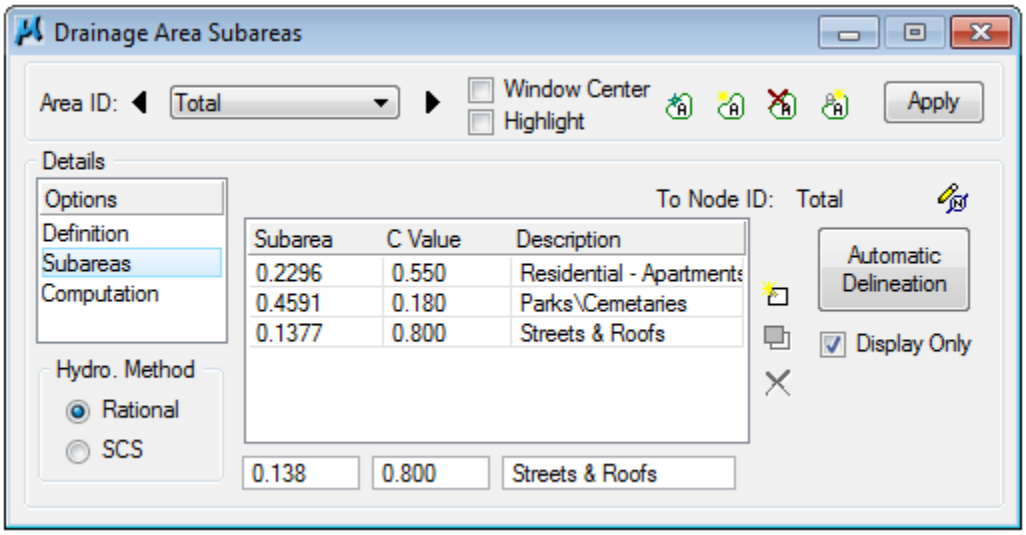
Any areas that do not have a land use associated with them will be assigned the base C value. In this example the area between the (Apartments – Park) and (Road) will be assigned a C value based of 0.6.

4. Select the Subareas option on the left side

5. Select the Automation Delineation button.

This will compute the area for any shapes with an assigned land use. Notice how the subareas are temporarily filled in the with the assigned land use color. (You may need to go to >Settings >View Attributes in the Microstation menu and toggle on Fill)

Also note that you could override the default C value for any land use by highlighting the row and selecting the modify button to the right of the screen.

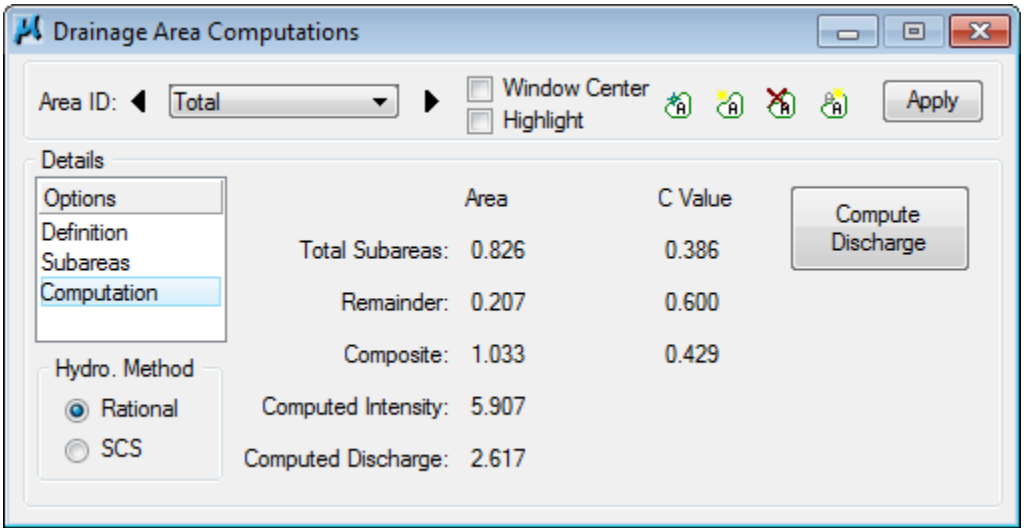


6. Select the Computation option

Notice that the remainder is the area of any regions that do not have an assigned land use.

7. Click the Compute Discharge button

Check the Composite C, Intensity, Total Area, and Discharge computed by Geopak.



8. Select the Apply button to add the Drainage Area ID to the drainage area

Chapter 3

Inlets

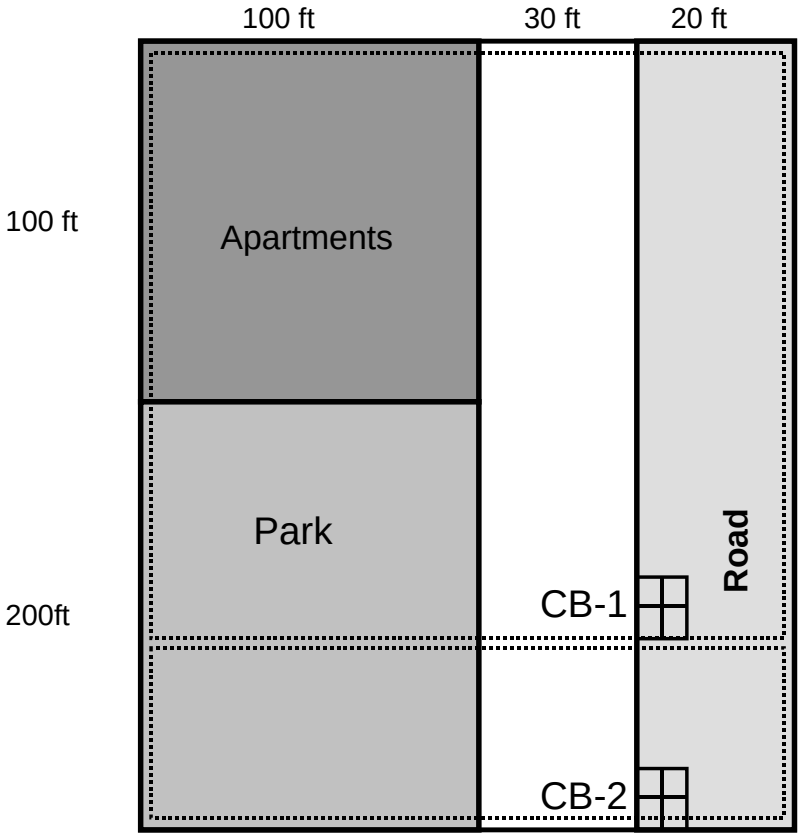
3.1 Objectives

Create and place stormwater inlets and associated drainage areas in GEOPAK Drainage

3.2 Overview

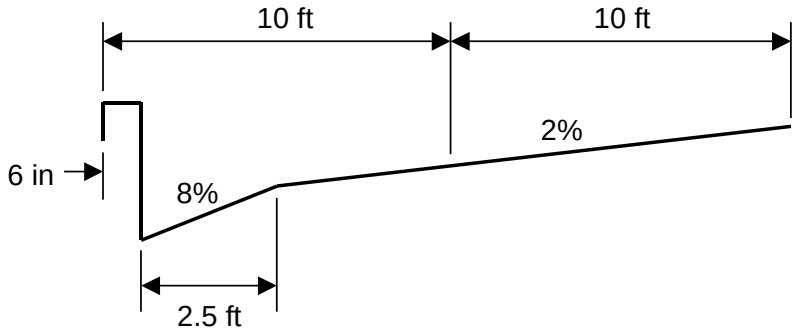
Establish inlets to collect run off from the drainage area that we created in Chapter 2. We will need to place 2 inlets in order to satisfy the ponded width (spread), ponded depth, and interception criteria. If we recall from Chapter 1 that there must be a 1 to 1 relationship between inlets and drainage areas, so the overall drainage area will have to be subdivided and each of these subareas will be associated with an individual inlet.

The first inlet (CB–1) is a 2'x2' curved vane grate and will be located 80 feet from the downstream end of the drainage area. The second inlet (CB–2) is a 2'x2' combo curved vane grate and curb opening and will be located at the downstream end of the drainage area. The typical spread section and vertical alignment of the roadway are shown in the figures on the next page.

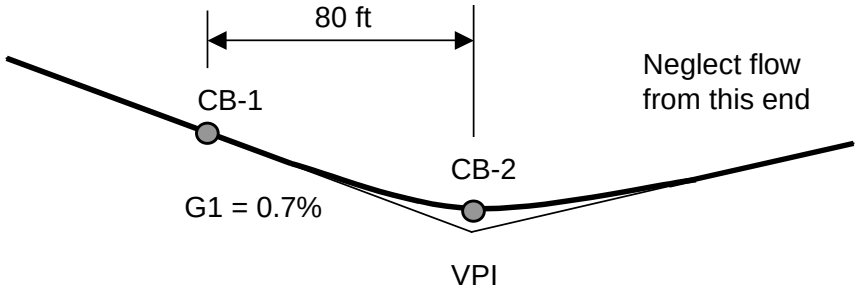


Inlet Label	Inlet Type	Tc (min)	Elevation (ft)
CB-1	2x2 CV Grate on Grade	5	100
CB-2	2x2 Combo CV Grate	6	99.5

Typical Spread Section



Vertical Alignment



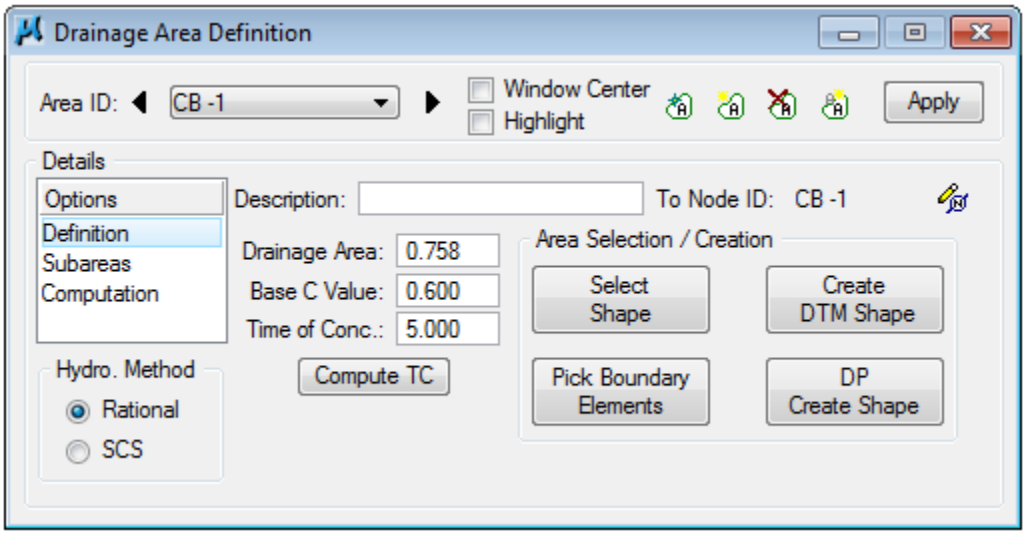
3.3 Define Subareas

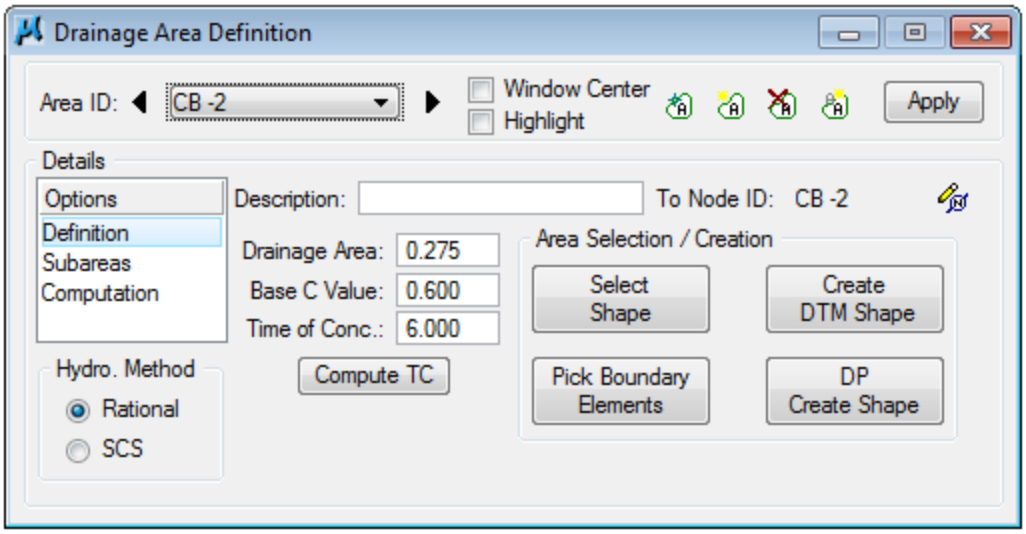
1. **Create MicroStation shapes for subareas CB-1 and CB-2 as defined in the illustration in 3-1. Area CB-1 is 150 ft wide and 220 ft long and area CB-2 is 150 ft wide and 80 ft long.**

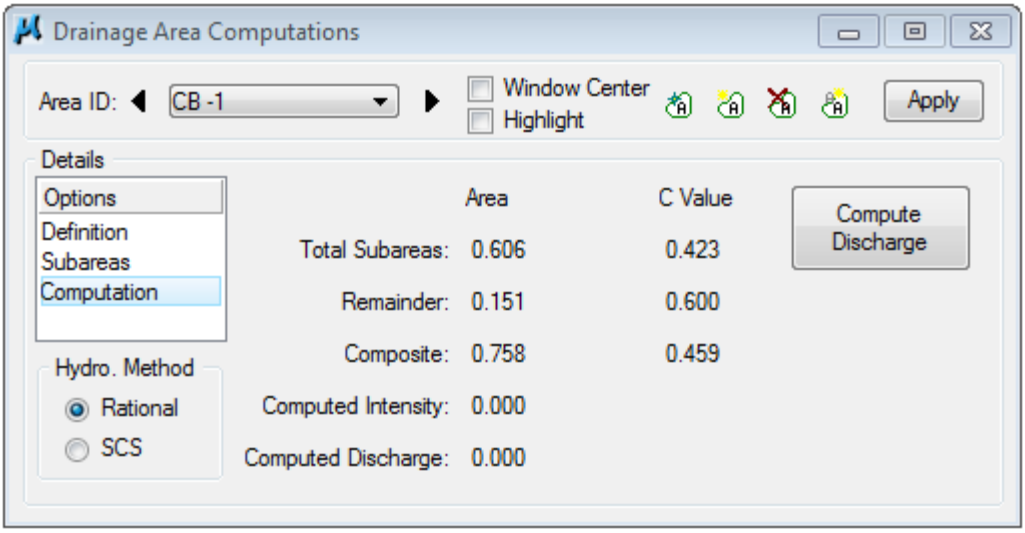
2. **Use the Drainage menu Component >Area >Add option to add each of the two drainage areas. Be sure to add the Base C Value and Time of Concentration for each drainage area and use the Delineate Subareas button to include the land uses previously defined. The Time of Concentration for CB-1 should be set to 5.0 and set to 6.0 for CB-2.**

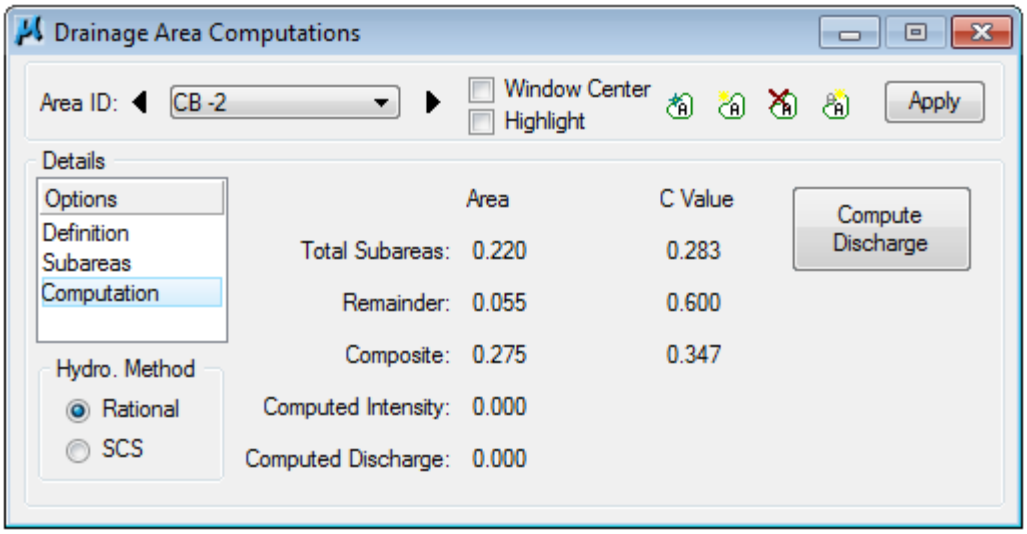
Make sure that you click the **Apply** button when you complete your inputs for both drainage areas to store the values.

Base C Value *The runoff coefficient (C value) for the drainage area. If subareas are to be delineated within the Area, this value will be applied to all subareas which have no associated land use.*




Drainage Area Definition
Area ID: CB -2
Window Center
Highlight
Apply
Details
Options
Definition
Subareas
Computation
Description:
To Node ID: CB -2
Drainage Area: 0.275
Base C Value: 0.600
Time of Conc.: 0.000
Compute TC
Area Selection / Creation
Select Shape
Create DTM Shape
Pick Boundary Elements
DP Create Shape
Hydro. Method
Rational
SCS

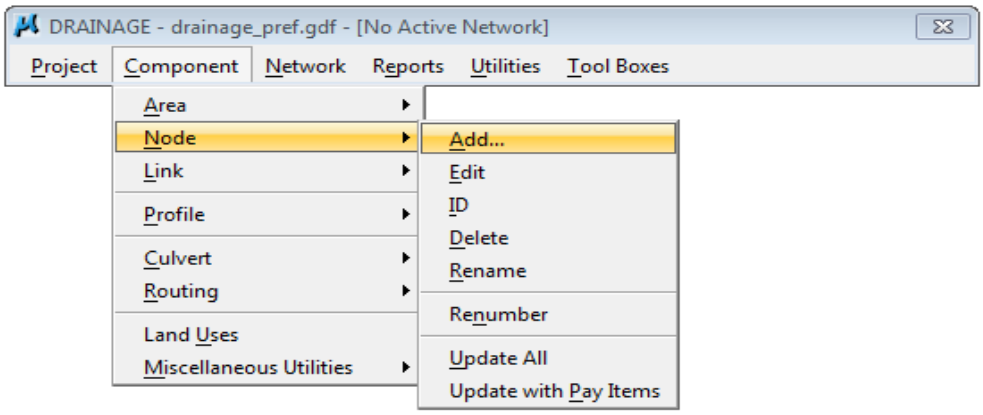

Drainage Area Computations
Area ID: CB -1
Window Center
Highlight
Apply
Details
Options
Definition
Subareas
Computation
Area
C Value
Compute Discharge
Total Subareas: 0.606 0.423
Remainder: 0.151 0.600
Composite: 0.758 0.459
Computed Intensity: 0.000
Computed Discharge: 0.000
Hydro. Method
Rational
SCS


Drainage Area Computations
Area ID: CB -2
Window Center
Highlight
Apply
Details
Options
Definition
Subareas
Computation
Area
C Value
Compute Discharge
Total Subareas: 0.220 0.283
Remainder: 0.055 0.600
Composite: 0.275 0.347
Computed Intensity: 0.000
Computed Discharge: 0.000
Hydro. Method
Rational
SCS

3.4 Define an inlet on grade

1. Go to Component >Node >Add . The node ID should appear as CB –1.

This is important to note because drainage areas and inlets are associated by name only. In other words, the discharge from drainage area CB-1 will flow into inlet CB-1. Because of this you may want to give inlet CB-1 a description like “Inlet for sub-area 1”.



Use the following information to create CB-1

Properties Option

Node Type: Grate

Profile: **On Grade** (Note that the hydraulic computations are different for inlets on grade and inlets at sags so it is important to select the proper profile.)

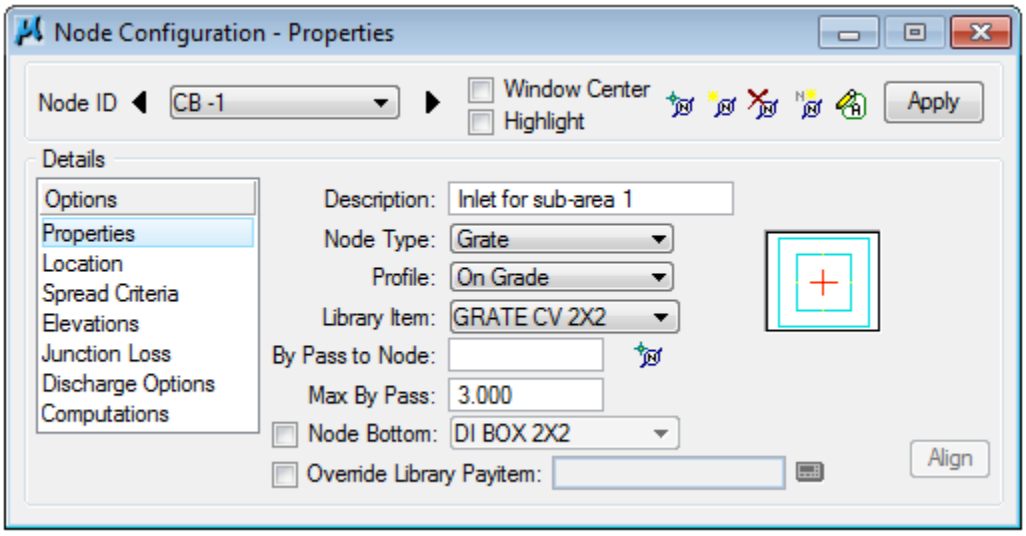
Library Item: Select **GRATE CV 2x2** for a 2’ x 2’ Curved Vane Grate

By Pass to Node allows us to identify the downstream inlet that will collect any runoff that is not caught at CB-1. This can be accomplished by selecting the ID button and then selecting either the label or cell for the downstream inlet. Alternately we can type in the node ID. We will complete this step after placing CB-2.

Max By Pass does not enter into any hydraulic computations, and is just used for querying the system. This will be discussed later in the course.

Node Bottom won’t be used at this time.

Override Library Payitem enables the user to select an inlet item other than the one specified from the defined drainage library. We will not use this option.


Node Configuration - Properties
Node ID
CB -1
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
Description:
Inlet for sub-area 1
Node Type:
Grate
Profile:
On Grade
Library Item:
GRATE CV 2X2
By Pass to Node:
Max By Pass:
3.000
Node Bottom:
DI BOX 2X2
Override Library Payitem:
Align

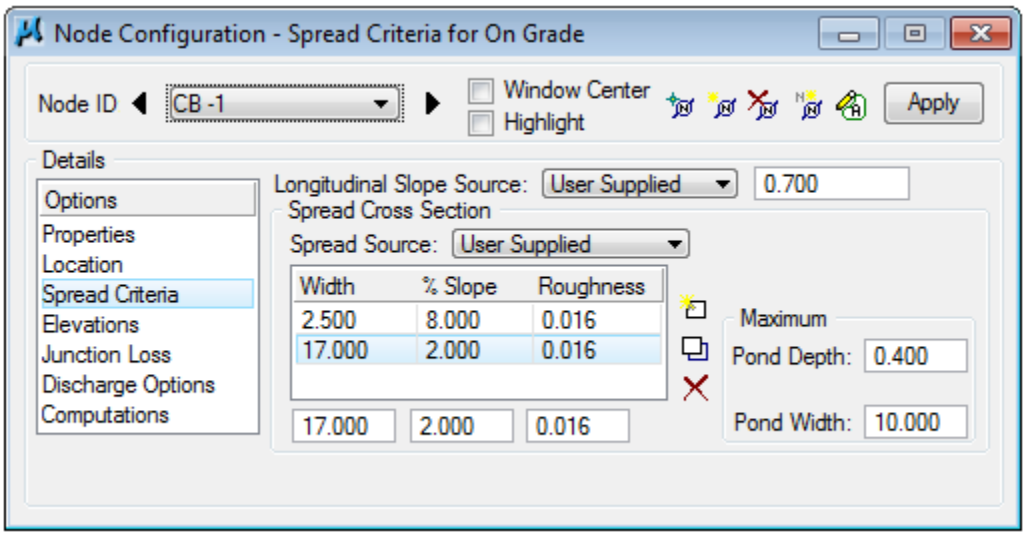
Spread Criteria Options

Longitudinal Slope Source: This is the actual grade along the vertical curve and not necessarily the tangent grade G1 or G2. For this example set the option to User Supplied and enter **0.7**

Spread Source: Enter the cross section of the roadway beginning at the curb face and working towards the crown of the roadway. For this example set the option to User Supplied

Max Ponded Depth is the maximum depth of water that can pond at the inlet. The ponded depth does not affect inlets at grade but does enter into the calculations for inlets at sags. Since we are using a 6" curb height, the maximum ponded depth is **0.4** ft.

Max Ponded Width is how much water will pond on the spread cross-section measured from the curb face. For this example enter **10** ft.



Elevations Options

Elevation Source: User Supplied - 100

Node Elevation Option: Same as Source

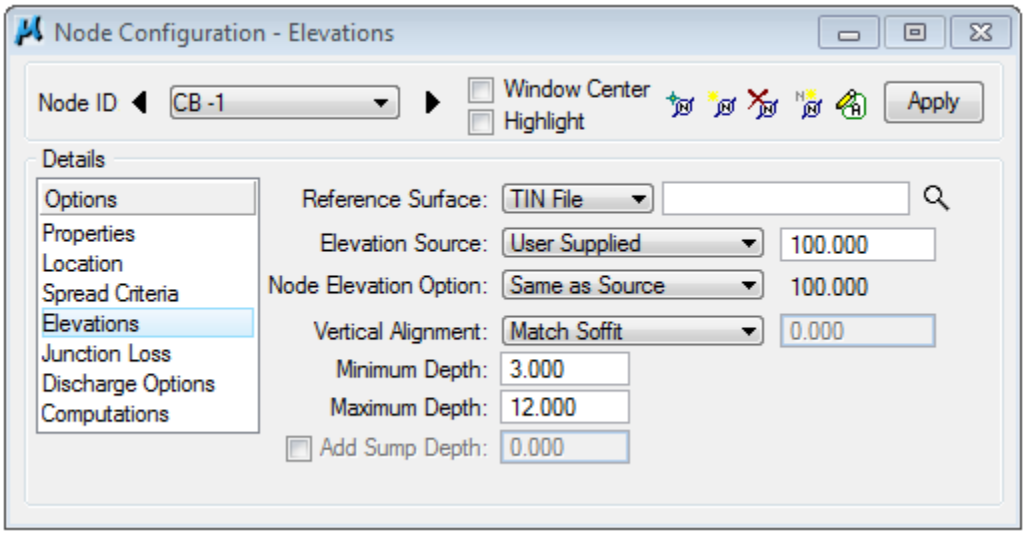
The next 3 settings deal with establishing pipe elevations

Vertical Alignment Preference: Match Soffit (top inside surface of the pipe)

Minimum Depth for this example is **3** ft. This is the upper limits of the design envelope, the soffit of the pipe cannot be placed above this line.

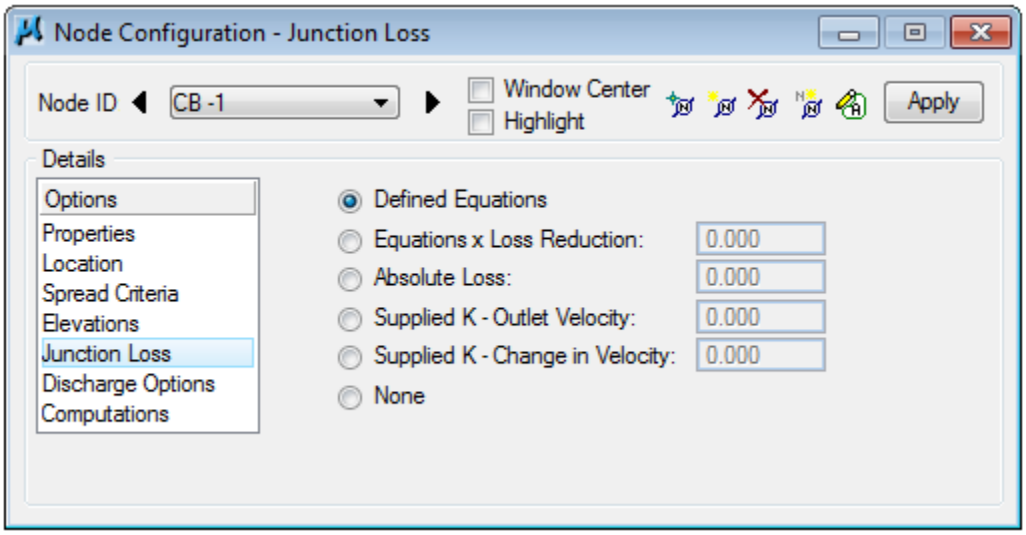
Maximum Depth for this example is **12** ft. This is the lower limit of the design envelope; the invert of the pipe cannot be placed below this line.

The design envelope created by the Minimum and Maximum Depth values are used by GEOPAK Drainage to set the absolute top and absolute bottom elevations for pipes to enter and exit the node. Additional information provided in the Chapter 3 Reference material.



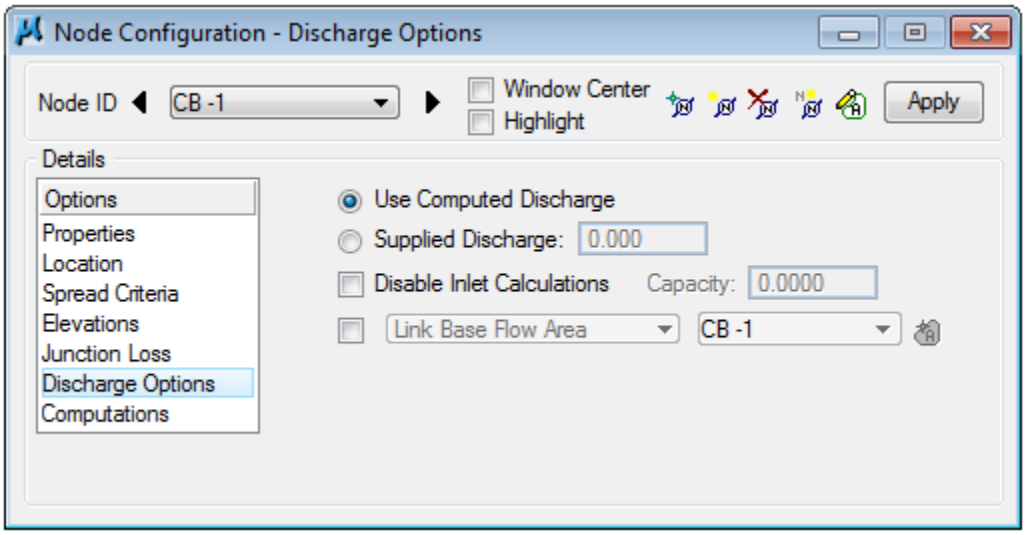
Junction Loss Options

Select Defined Equations



Discharge Options

Use Computed Discharge will associate drainage area CB-1 to inlet CB-1. Alternately you can specify a discharge.
Disable Inlet Calculations Capacity: When this toggle is not active, GEOPAK utilizes the inlet capacity calculations. To override these calculations, the user may activate the toggle and provide the desired capacity.
Link Base Flow Area/Discharge: When not activated, the drainage area identified for the node is utilized for calculations. When this toggle is activated, additional drainage areas may be defined. The area can be identified graphically by pressing the ID button and selecting the MicroStation shape or may be manually entered in the key-in field. Note the area is included in the pipe, but is not included in the inlet spread calculations.
A discharge amount may be specified as well by choosing that option from the drop-down box and inputing



Location Options

We are not utilizing data from Geopak Road in this example.

1. Set the Align setting to Tangent to Element

2. Select the MS Alignment Element button

3. Data point on the rectangle that represents the roadway.

Make sure AccuDraw is running

4. Select the Station DP button and snap (single middle mouse click) to the lower right hand corner of the roadway. *Do not left-click to accept the snap!*

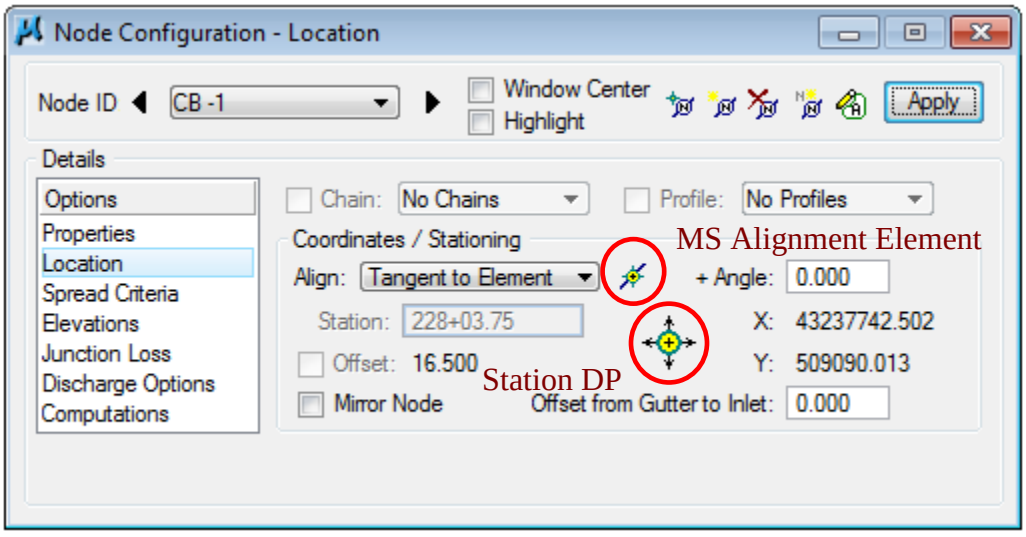
5. Select the letter O on the keyboard to set the origin.

6. Move the mouse in the negative X direction, enter 18.5
(20' pavement width, then subtract 6" curb and ½ the grate width)

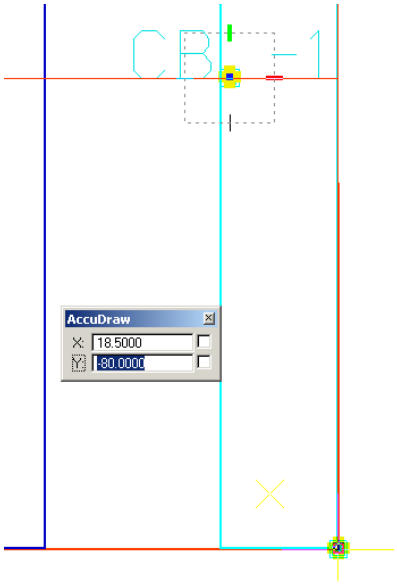
7. Move the mouse in the positive Y direction, enter 80, and data point to accept.

8. Finally select the Apply button.

The inlet is now saved to the *.gdf file, any changes to the inlet will now be updates.



After you are finished CB-1 should appear in this location:

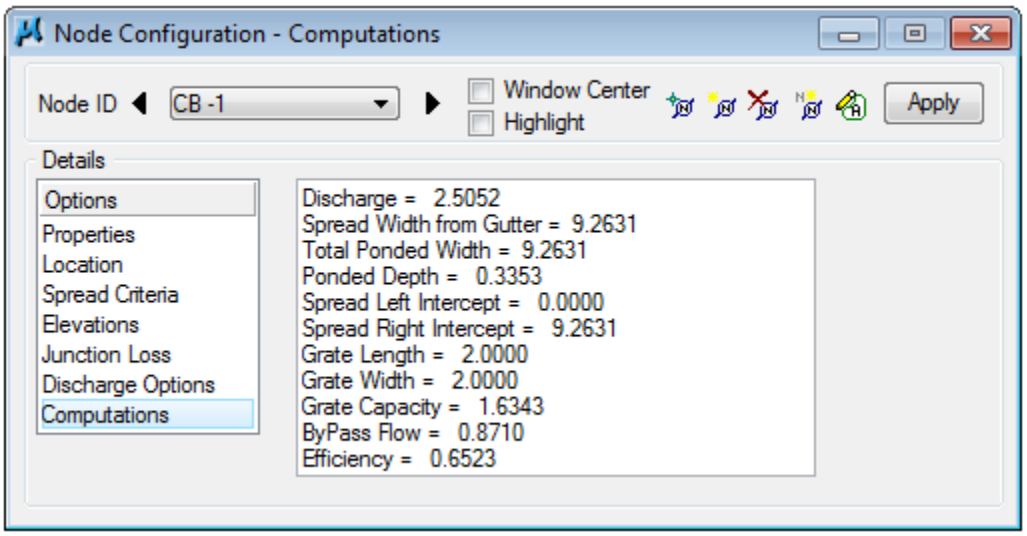


Computations Options

The Computations Option will show all the information pertaining to the new inlet based on the area associated with it.

In this example we can see that the Spread Width is 9.2631, which is less than the 10' that was allowed. The ponded depth is .3353, which is less than the .4' that was allowed. There is some bypass flow indicating that some flow is going to need to be intercepted downstream of this inlet.

We can see from these computations that this inlet already meets the criteria that we set for our roadway. However, if this inlet had not worked we would need to consider what steps we could take to fix any issues that came up. With this being the first inlet in our drainage project, the fix could be as simple as just establishing the rest of the system to see if it relieves the issues, but if the computations produced values that fell too far outside of our needs we may want to consider a larger inlet.



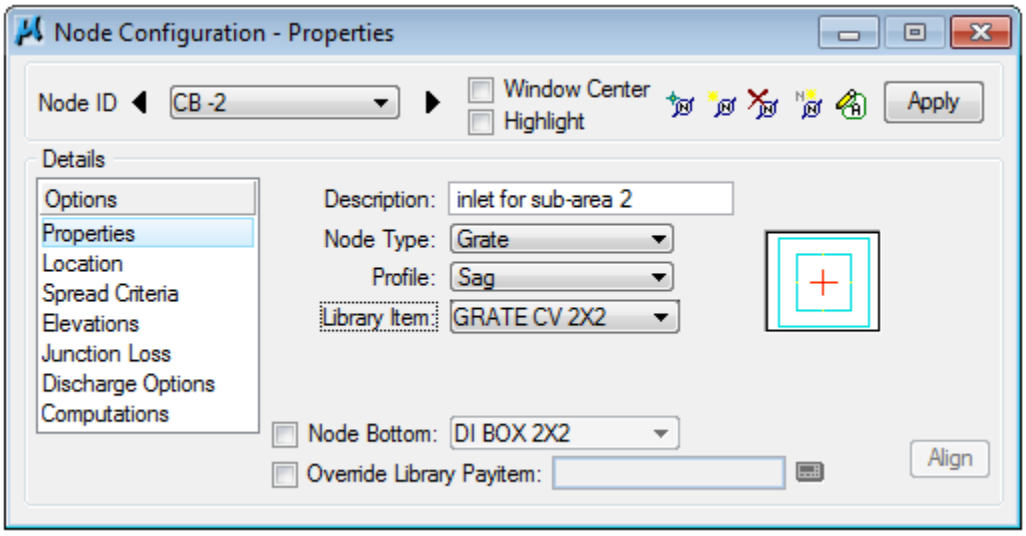
Make sure you have clicked the **Apply** button to save your work to CB-1 before moving on to creating CB-2!

3.5 Define an inlet at a sag location

Add inlet CB-2 at the sag location.

Properties Options

Use the options shown below in the dialog screenshot



Location Options

1. Set the Align setting to Tangent to Element

2. Select the MS Alignment Element button

3. Data point on the rectangle that represents the roadway.

Make sure AccuDraw is running

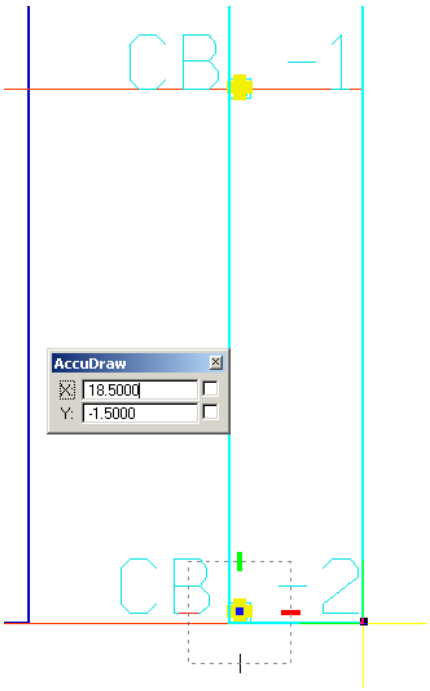
4. Select the Station DP button and snap (single middle mouse click) to the lower right hand corner of the roadway. *Do not left-click to accept the snap!*

5. Select the letter O on the keyboard to set the origin.

6. Move the mouse in the negative X direction, enter 18.5
(20’ pavement width, then subtract 6” curb and ½ the grate width)

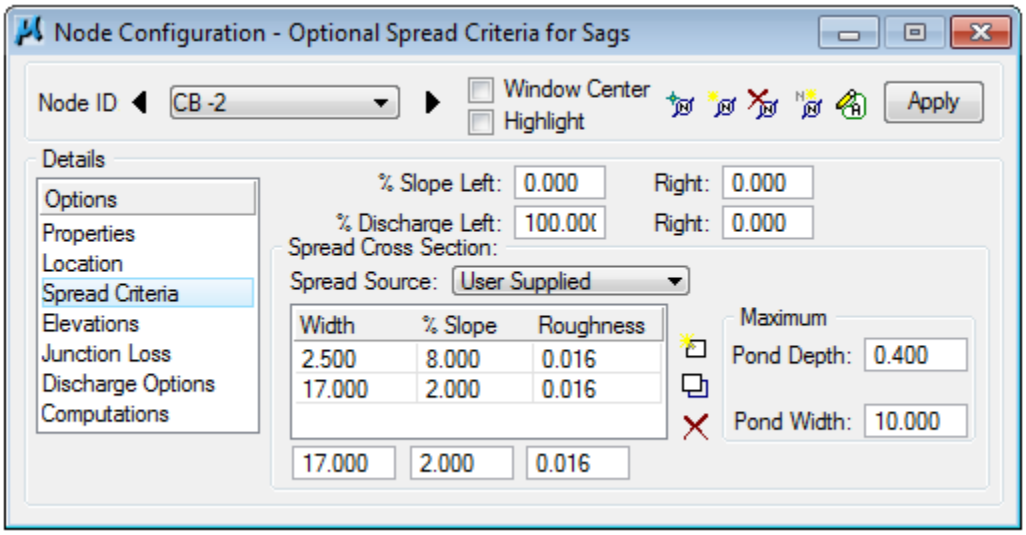
7. Move the mouse in the positive Y direction, enter 1.5, and data point to accept.

8. Finally select the Apply button.



Spread Criteria Options

Enter 0.0 for the slope left & right, and 100% of the discharge approaching from the left.
We are neglecting flow from the right hand side. A low point will always have flow approaching from both sides, the right side was neglected for simplicity.



Elevations Options

Enter the elevation as **99.5**

Discharge Options

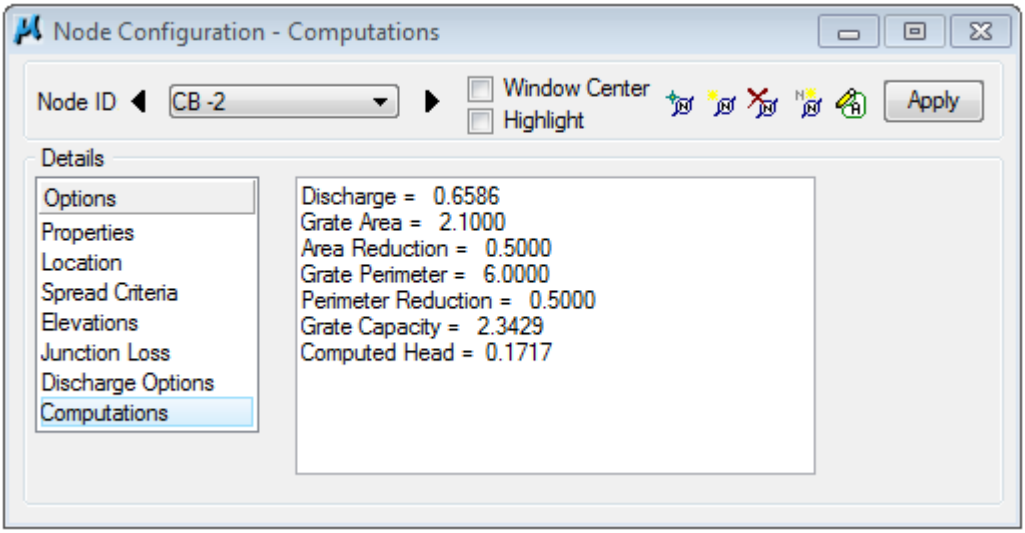
Remain the same

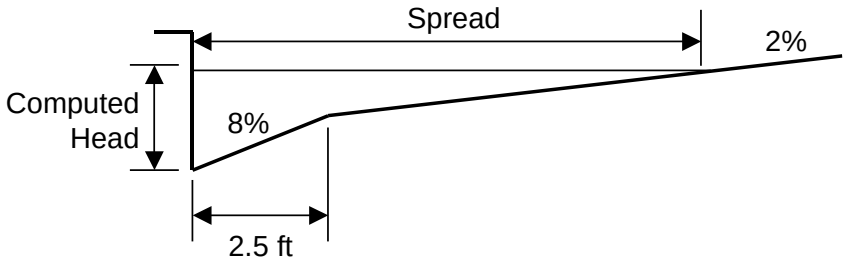
Computation Options

The spread used by MoDOT for design purposes at sag inlets is not contained in the computations window. You can compute the spread using simple geometry by taking the computed head (depth of flow in the gutter) and applying it to the spread section.

Spread can be generated using the Report Builder which is not covered in this class

.

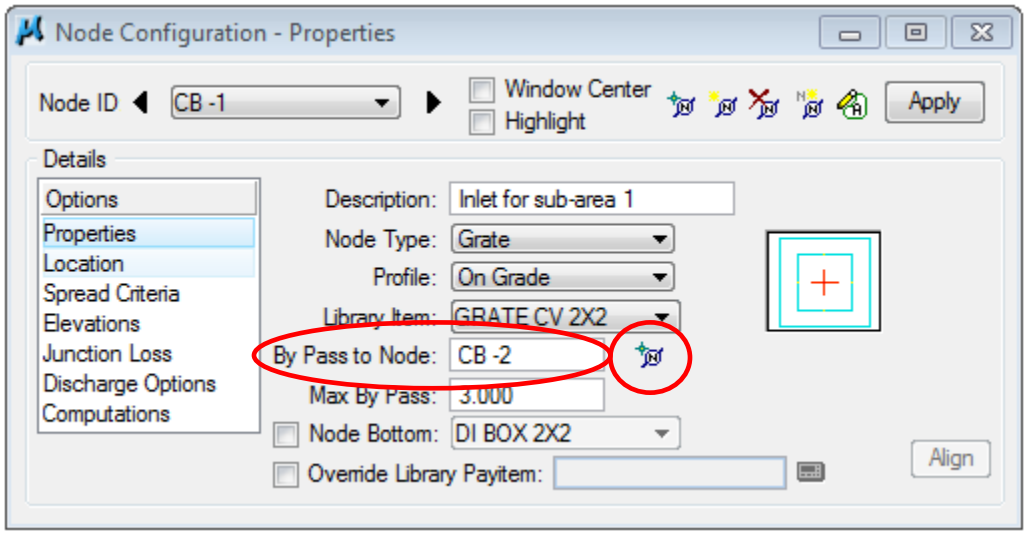




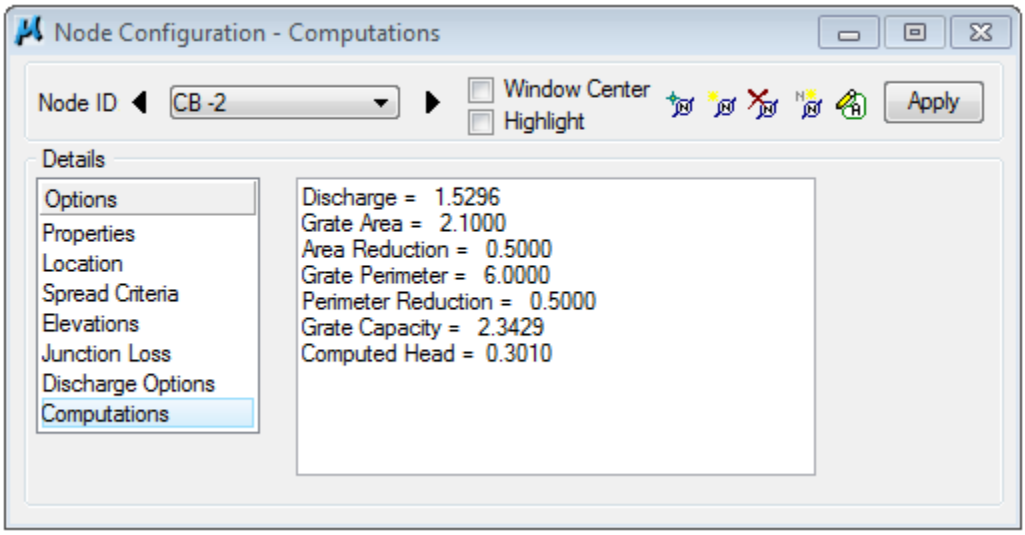
Set By Pass Flow

Now that we have a downstream inlet we can set it up to catch the water that isn't collected by our upstream inlet. To catch that flow, and make sure it is intercepted by the next inlet:

- Open the previous instance, in this case, CB – 1
- Click on the **Properties** Option
- Select the **ID By Pass Node** tool next to **By Pass to Node**
- Using the left mouse button, select the inlet downstream, in this case, **CB –2**



- Now open the CB – 2 and look at the computations again



Chapter 3

Inlets-Reference

Node Placement Options

Node: Properties

Node ID — GEOPAK will automatically number the nodes based on the order in which the nodes are entered. The nodes can be renumbered later in the design process. This is discussed in the Chapter about Design
Revisions.

Description (optional) — This may be used to better describe the node.

Node Type — This will show the different node types defined in the drainage library.

Profile — The designer needs to specify the profile condition of the particular node to be placed. On Grade or in Sag condition. This affects the input and calculations for spread.

Library Item — Depending of the Node Type selected above, the list box will only show the available nodes that match the type condition.

By Pass to Node — The designer can specify or ID the node that will take the by passed discharge from the active node.

Max By Pass — Defines the maximum allowable by pass flow from the inlet. No specific computations are adjusted by this value; it is used to merely query the system for by pass flows which exceed this value. Used for On Grade Inlets only. This value could be left at zero (default) if no By Pass to Node is specified.

Node Bottom — This allows the user to specify a box size to be used along with the type of inlet and enables GEOPAK to assign an item for quantity calculations.
MoDOT has not use this type of node in the past and currently the system is not yet set up to use the Node Bottom for quantity calculation purposes. The new drainage library contains Node Bottom items that can be used if desired.

Override Library Item — Each node in the drainage library is associated to a pay item or Item ID in the Design and Computation Manager database. This option will override this linkage, allowing the user to select their own Item ID. The Department uses that option to make the placed node an “existing” node for quantity computations.

Node: Location

Chain	A previously defined horizontal alignment stored in the "gpk" file. Activate the toggle to the left of the field and select the chain from the list.
Profile	A previously defined vertical profile stored in the "gpk" file. Activate the toggle to the left of the field and select the desired profile from the list.
Align: At Point	When this option is utilized, the node is placed at the active angle in the MicroStation file at the current or dynamic Node coordinates.
Align: Tangent to Chain	When this option is utilized, the node is dynamically moved along the Chain, and is rotated maintaining tangency to the chain. The designer can enter the station location or place the node dynamically.
Align: Tangent on Element	When this option is utilized, the **ID** button to the right of the option is activated and the designer is prompted to identify and accept a MicroStation element. The Node is placed tangent to and on the identified element projected from the current or dynamic Node coordinates. The designer can enter the station location or place the node dynamically.
Align: Tangent to Element	When this option is utilized, the **ID** button to the right of the option is activated and the designer is prompted to identify and accept a MicroStation element. The Node is placed tangent to the identified element at the current or dynamic Node coordinates.
Angle	An optional angle rotation could be applied to the node. A 180-degree angle is often needed to properly align the throat of curb inlets and is sometimes used in conjunction with the Mirror Node toggle.
Station and Offset	Nodes may be located by entering the station and offset in reference to the selected chain. If "Tangent on Element" is used, the offset is computed.
Mirror Node	When activated, the node cell is mirrored, in order that the same cell can be utilized to accommodate flow coming from either direction.

Offset from Gutter to Inlet

Inlet hydraulic equations assume the Inlet is at the edge of the computed spread (i.e. the Curb). If the Inlet is offset from the Curb, the discharge in the spread section will bypass the Inlet. This value, expressed in terms of feet or meters, is used to compute this bypass and adjust the actual flow to the Inlet. This may be used, for example, when a Grate Inlet is actually set some distance away from the curb of the road.
Therefore any flow calculated in that segment will not be added to the inlet and it will bypass to the next assigned one.

Node: Spread Criteria

User Supplied

The designer will enter the width and slopes across the spread section starting from the Inlet and continuing towards the edge of the section farthest away from the inlet.

Reference TIN

A spread cross section will be extracted at the inlet location from the TIN file defined in the Project Components item under the Drainage Project Preferences. The designer needs to be careful on what DTM is referenced since for most MoDOT projects the only TIN file generated is the existing ground DTM.

Library Item

A spread cross section will be assigned to the inlet using a Spread Section item defined in the Drainage Library. Once a Library Item has been selected, the spread source can be changed to User Supplied and the spread section defined by the Library Item modified to accommodate non-typical situations such as pavement width transitions.

Shape

A spread cross section will be extracted at the inlet location from the Superelevation Shapes dgn file defined in the Project Components item under the Drainage Project Preferences and shown in the Spread Criteria table.
For curb inlets in superelevation or superlevation transition, the designer can use this option or enter the spread source (cross section) as User Supplied.
If the superelevation shapes are not defined to the edge of pavement, the designer can take advantage of the toggle:”Extend Superelevation Shapes to Inlet at Shape Slope”, located in *Drainage > Project > Preferences* under the Inlet Options item. This will extend the last slope of the shape to the node location point defined in the structure.
Therefore, if the shape file is used to define the spread, be aware that the cross section starts at the edge of pavement creating a theoretical section that is shifted from the actual spread section by the width of the gutter. This is acceptable because the calculated spread width is the same regardless of the shift. The gutter depression cannot be modeled with this approach. To account for the gutter depression while using the shape option, the "Shape and Library Item" would need to be used.

Shape and Library Item A spread cross section will be assigned to the inlet combining the section extracted from the Superelevation Shapes dgn file and Library item from the Drainage Library. If the gutter section is made a library item, this option can be used to account for the gutter capacity.
To properly use this option, the superelevation shapes need to be defined to the edge of pavement line, and the ”Extend Superelevation Shapes to Inlet at Shape Slope” toggle must be off.

Maximum Ponded Depth The maximum depth allowed for ponding in the cross section (expressed in terms of feet or meters). This does not affect any computations, but will produce a warning if the computed depth is greater.
For sag inlets, this value is used to compute the capacity of the node.

Maximum Ponded Width The maximum width allowed for ponding in the cross section (expressed in terms of feet or meters). This does not affect any computations, but will produce a warning if the computed depth is greater.

Reference PGL Slope will be extracted from the profile defined in the Location item of the Node dialog box at the inlet location. The slope will update as the .gpk file is updated.

User Supplied Slope will be entered by the designer in percentage format while omitting the % sign.

Shape Slope will be extracted from the superelevation shapes dgn file defined in the Project Components item in the Drainage Project Preferences.
For inlets in superelevation, superelevation transition, or other transitions, the designer can use this option or enter the longitudinal slope as User Supplied.

For an inlet in sag conditions, the approaching longitudinal slopes have no effect on the calculation of the inlet capacity or spread at the inlet. The spread approaching a sag inlet needs to be calculated and the approaching longitudinal slopes affect those calculations. The fields unique to Spread Criteria for Sags relate to the spread along the approaching longitudinal slopes and are discussed below. The designer must specify a value for Maximum Ponded depth for sag inlets. Although this value is not needed to compute spread, it is used by GEOPAK to compute the capacity of sag inlets.

% Slope Left and Right — For defining the longitudinal slope to use in spread (ponded width) computations for the flow approaching from the right and left side of the inlets. The terminology (right and left) is somewhat arbitrary as long as the designer recalls which side is intended as left and right.

% Discharge Left and Right — Defines the percentage of the total discharge to the inlet to allocate to the left and right approach spread computations. The terminology right and left are somewhat arbitrary as long as the designer recalls which side is intended as left and right and is consistent with the definition of % Slope Left and Right.

Node: Elevations

Reference Surface Define the desired TIN File or a Model or Object within the site project specified in the Drainage Preferences – Project Components.

Elevation Source Defines the source of the Node elevation. With the source specified as something other than User Supplied, the Node elevation will
automatically update as the Node is moved. The options are:
Reference Surface: Computes the actual elevation from the X and Y coordinates of the structure from the Location dialog and the Reference TIN File or Site Model. This option may be useful for placing Ditch Bottom Inlets at the existing ground elevation depicted in a TIN file.
Reference PGL: Utilizes the Station from the Location dialog to ascertain the profile elevation on the Reference PGL. Note: This option will report the elevation directly at the station on the PGL.
PGL+Spread Section: Utilizes the PGL and information from the Spread Criteria cross section to compute the elevation. The elevation is ascertained along the PGL at the specified Station and then adjusted along the spread cross section width and slope segments to the end of the cross section. Although it says PGL "plus" spread section, the programs subtracts positive values from the PGL.

Node Elevation Options: The designer may adjust the inlet elevations up or down from the Elevation Source item. These options are:
Same as Source: matches the node elevation with elevation source.
Constant Offset: enables a plus or minus value to the elevation source selected. For example, if the reference profile is PGL+ Spread Section to the flow line of the gutter, a Constant Offset can be specified to have the node elevation be the edge of pavement (EOP) elevation. This is useful because the EOP is used for curb inlet elevations. A plus offset will add to the elevation value.
User Supplied: The user may specify an elevation.
For the outlet node elevation, the elevation is not a physical point on the structure. The elevation should be at least 1 foot above the tailwater to avoid a warning. In general, using the berm or ground elevation that covers the outlet structure is acceptable for the outlet node elevation.

Vertical Alignment Alignment preferences for incoming and outgoing pipes from the Node. As pipes are designed the elevations will be set according to the selected preference.

	Options available are: • Match Soffit: aligns all the pipes in the system by matching the inside top surface of each pipe. (Also referred to as the crown of the pipe) • Match Invert: aligns all the pipes in the systems by matching the bottom inside surface of each pipe. • Match Surface: aligns the pipes following the elevation of the water surface inside the pipes. • Allow Drop Manhole: allows for "drops" inside the node for the pipes arriving or existing the node. Not matching inverts or soffits will be performed. This option allows for greater flexibility. Since the design preference is to minimize excavation, the first pass during the design will align all the pipes at the soffit. As the designer needs to move the inverts of certain pipes due to various factors, e.g. utility conflicts, the change will only affect the selected pipe and others will remain at the previously set elevation. • Min. Fix Drop: sets a minimum drop in elevation between the pipes arriving and exiting the node.
Minimum Depth	Minimum depth expressed in master units (i.e., feet or meters) for the pipe to be buried (minimum cover) measured from the Node Elevation to the soffit (top inside) of the highest pipe. This value is used to establish the minimum cover elevation noted in the Link Configuration Conditions dialog box. For additional information see below.
Maximum Depth	Maximum depth expressed in master units (i.e., feet or meters), for the pipe to be buried measured from the Node Elevation to the invert of the lowest pipe. This value is used to establish the Maximum Depth elevation noted in the Link Configuration Conditions dialog box. For additional information see below.
Add Sump Depth	Extra depth to be allowed in the manhole structure after the maximum depth of the structure has been computed. This has no effect in the hydraulic calculations, but some designers add this extra depth for pollution control measures. MoDOT does not use this option

Minimum and Maximum Depth at Nodes Definition

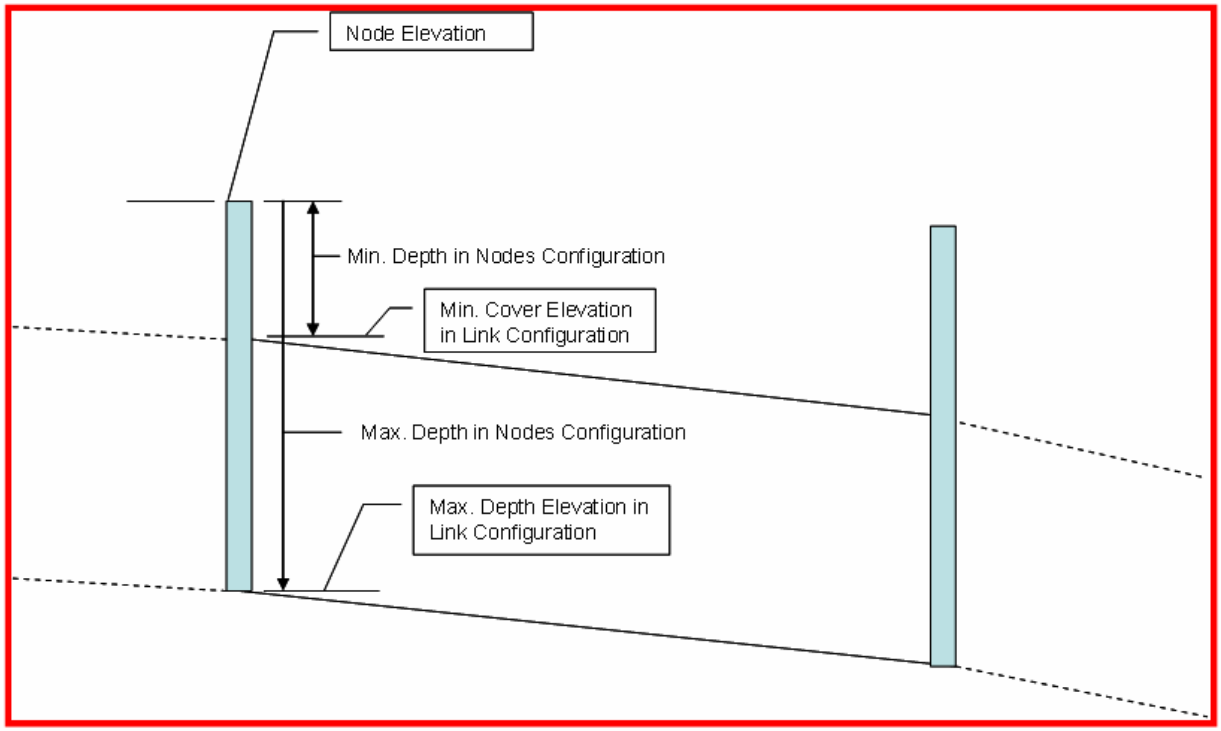
The values entered for Minimum and Maximum Depth define the vertical limits that the program uses to place the pipe. The Minimum Depth establishes the upper limit that pipes are designed in. The Maximum Depth establishes the lower limit.

The minimum depth of a pipe is often established by cover.

.

Consider the following for the Maximum Depth: The approximate elevation of the outlet node should be known while designing the storm drain. The outlet elevation may be controlled by the minimum pipe slope or by the depth of the pond at the outlet. The designer will be able to lock the elevation of the outlet pipe in the Link Configurations-Conditions dialog. Locking the elevation of the outlet will control the total drop through the system. This will limit the allowable pipe envelope that the pipes are designed within. Reasonable results should be obtained by setting the Maximum Depth for all nodes to be deeper than the outlet pipe (maybe use a tentative 20' depth). Again, GEOPAK will not fully use this depth to design the pipes if the outlet elevation is locked.

Regardless of what depths are used, the designer will still need to check for adequate depth, pipe slope, and hydraulic gradient clearance later in the design process.



Nodes: Discharge Options

Use Computed Discharge — Indicates using the computed discharge from a stored Drainage Area as the discharge to the Node.

Supplied Discharge — This option allows for a user supplied discharge, with disregard to any calculated value from a drainage area.

Disable Inlet Calculations Capacity — When toggled on, the designer may provide the desired inlet capacity overriding any GEOPAK Drainage calculated value.

Link Base Flow Area / Link Base Flow Discharge — When toggled on, an additional drainage area may be defined. The area can be identified graphically by clicking **ID** and selecting the MicroStation shape previously defined as a GEOPAK Drainage Area.
For the Link Base Flow Discharge. the designer can also enter a value for the extra discharge in the proper units defined in the Project Preferences (cubic feet per second or cubic meters per second).
This “extra” area or discharge is included in the pipe, but is not included in the inlet spread calculations.

Chapter 4

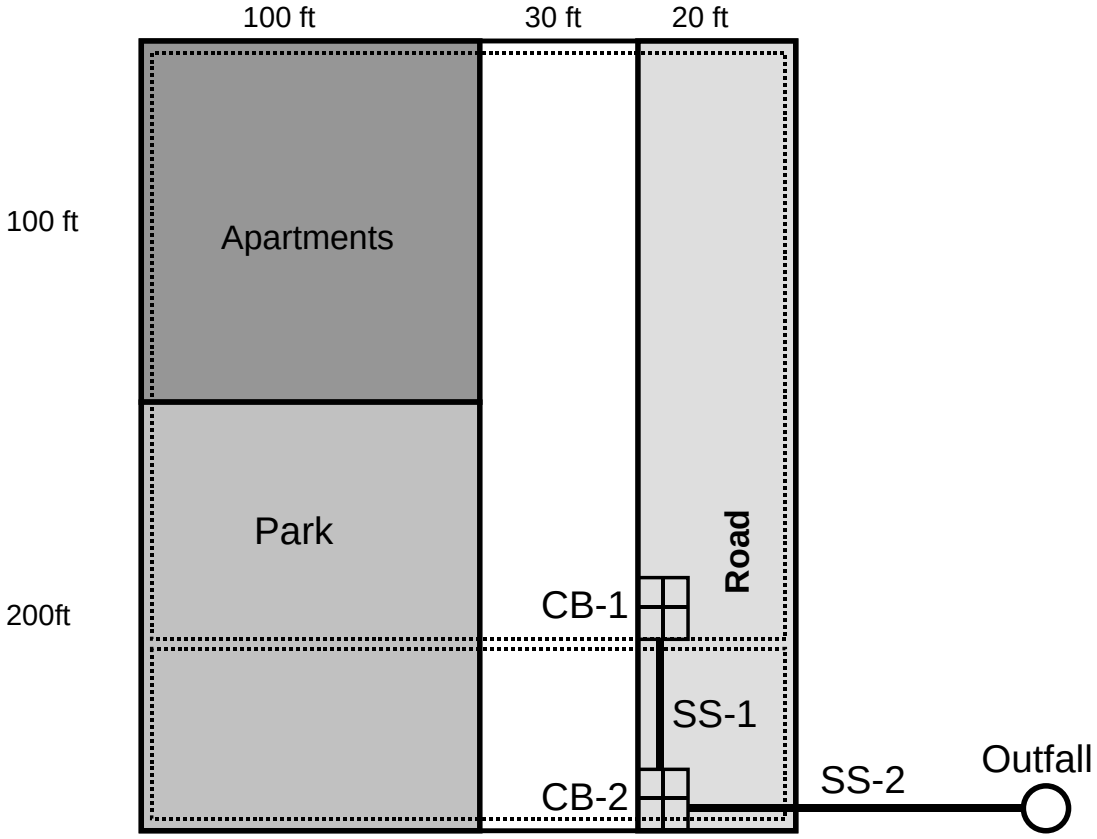
Storm Sewers

4.1 Objectives

Link established inlets with storm sewer pipes and create a drainage network in GEOPAK Drainage then design the network and view output.

4.2 Overview

Place an outfall approximately 100 ft to the left of inlet CB-2. Design pipe SS-1 between inlet CB-1 and CB-2. Design pipe SS-2 between inlet CB-2 and the outlet. Assume the outfall is at an elevation of 99.0 ft and the HGL begins at normal depth in the pipe.



4.3 General Pipe Design Criteria

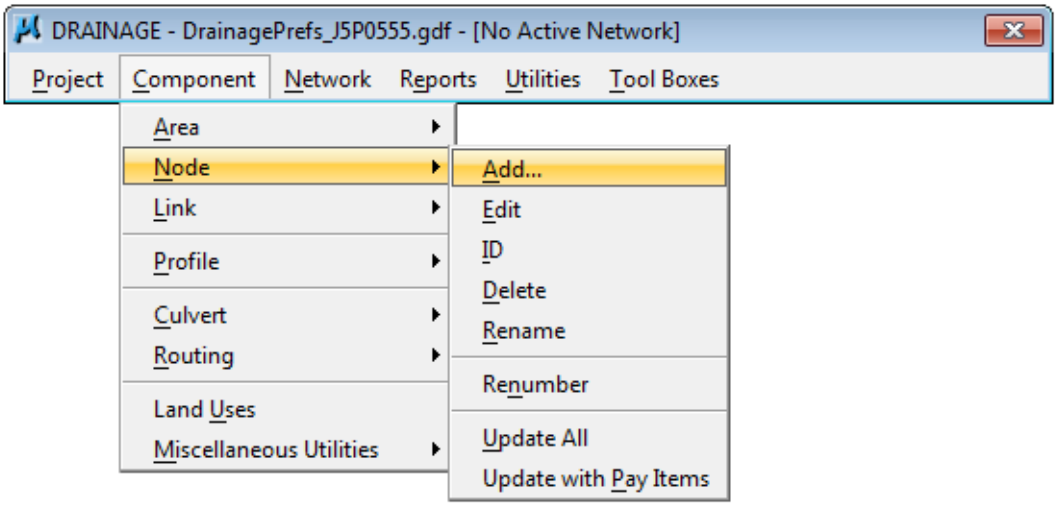
The basic design of the pipe is controlled by the parameters set in three locations:

Preferences >**Link Options**
Link Profile Options -Minimize Pipe Size or Minimize Depth of Cover
Link Design Options -Design for Full Capacity

Node Configuration >**Elevations** >**Vertical Alignment Preference**
Use **Match Soffit (**Top inside surface of the pipe)
Set the Minimum Depth and Maximum Depth. This determines the design envelope for the pipe design. The preferred minimum depth of cover is 3 ft; the absolute minimum is 1 ft below the pavement. EPG 750.4.2.1

Link Configuration >**Constraints**
As a rule of thumb use a minimum slope of 0.0103 and a minimum velocity of 3.0 ft/s. See EPG 750.4.2.5 for a table of minimum slopes to achieve a self-cleaning velocity when the pipe is flowing a quarter full. The maximum slope and velocity is up to the discretion of the designer.
Use a minimum rise (diameter) = 1'. This is the smallest pipe in the drainage library.
Use a maximum rise (diameter) = 9'. This is the largest pipe in the drainage library

4.4 Create an Outfall



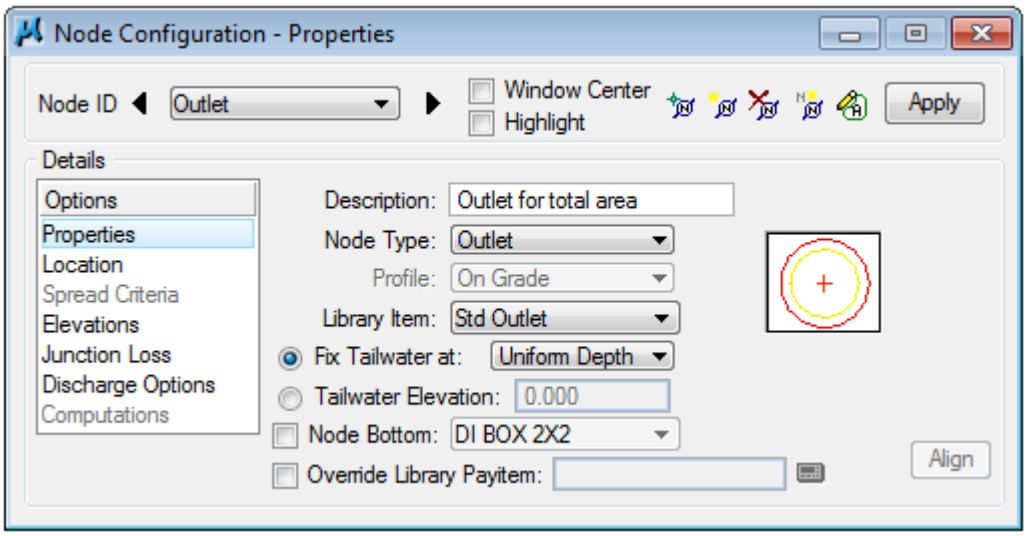
1. Go to Component >Node >Add

Name the node **OUTLET**

2. Configure the Outlet node according to the following specifications:

Properties Option

Node Type: **Outlet**
Library Item: **Std Outlet**
Fix Tailwater at: **Uniform Depth** for this example.
Note: For information on establishing tailwater for storm sewers see EPG 750.4.4.1

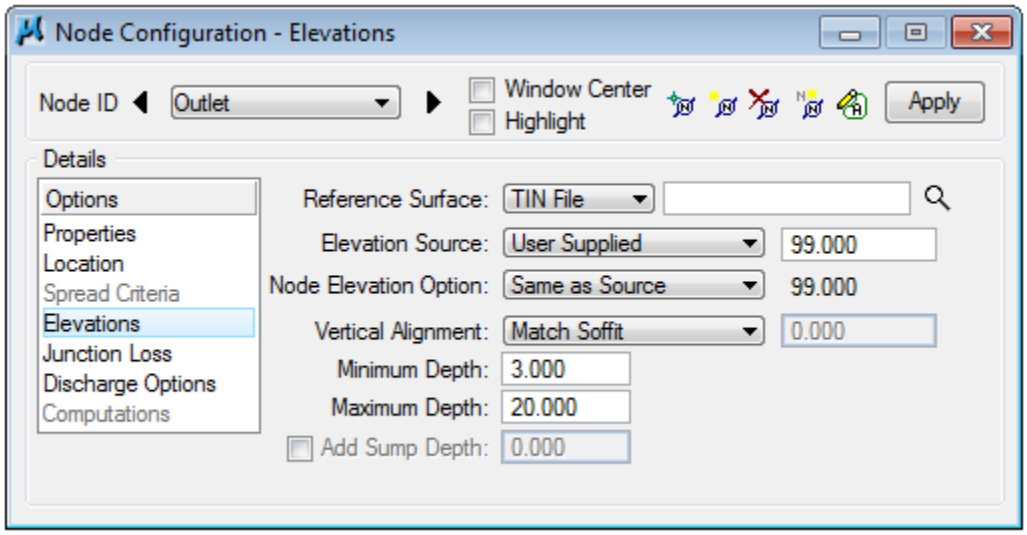


Location Option

Place the outfall approximately 100 ft to the right of CB-2, using the same approach used for placing inlets CB-1 and CB-2.

Elevations Option

Elevation Source: **User Supplied – 99**



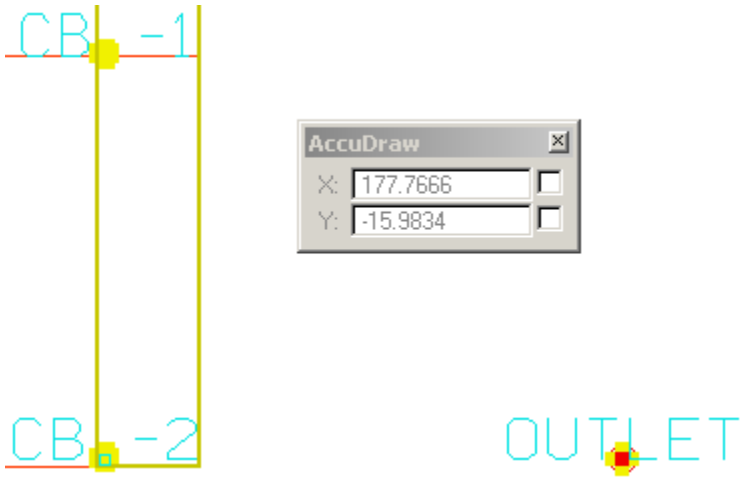
Junction Loss

No changes required.

Discharge Options

Select Use Computed Discharge

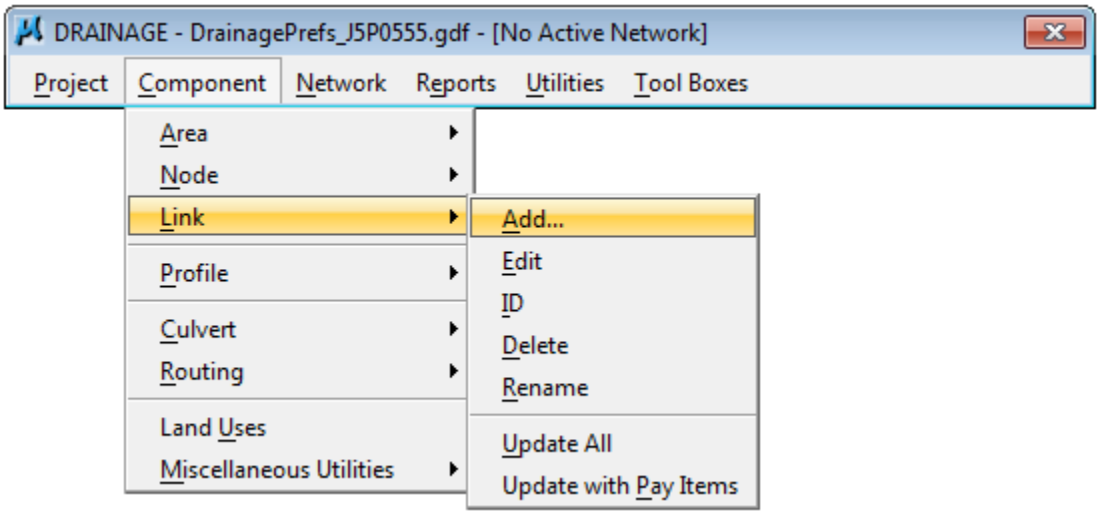
Finally select the Apply button



4.5 Add Pipes to the Storm Sewer Network

1. Go to Component >Link >Add.

The link ID should appear as SS-1.



2. Configure SS -1 by following the information provided below:

Definition Option

Set the From Node and the To Node fields by using the drop-down lists or by using the ID option and selecting the nodes graphically.

Shape: **Circular**

Material: **Concrete**

Toggle on **Design Size** and select any pipe from the drainage library.

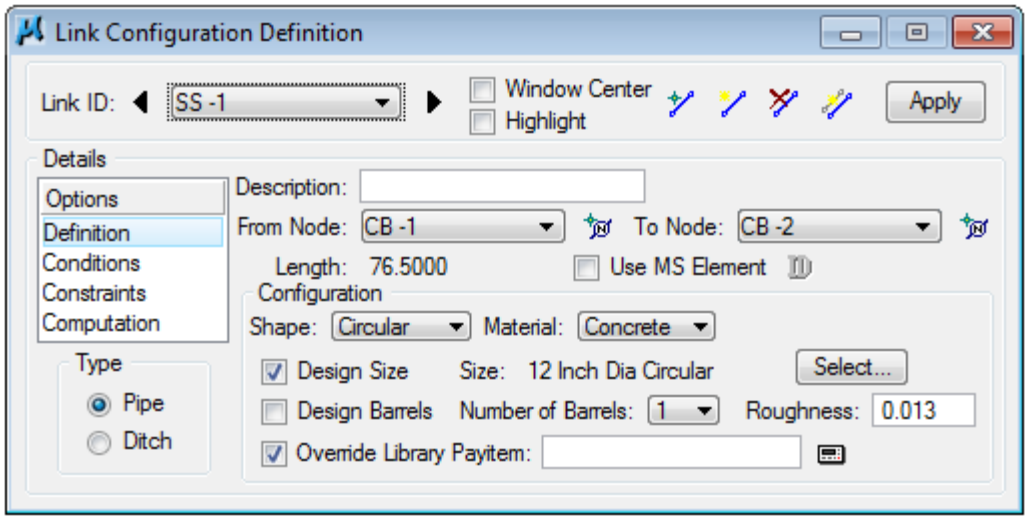
With the Design Size toggle enabled GEOPAK will run through the pipes contained in the drainage library starting from the smallest diameter until it finds one that will work for the supplied design parameters.

You can force a pipe diameter by un-toggling Design Size and selecting a pipe from the drainage library.

Number of Barrels: **1**

Check the **Override Library Payitem**

This will allow for a deviation from the standard pay item number.



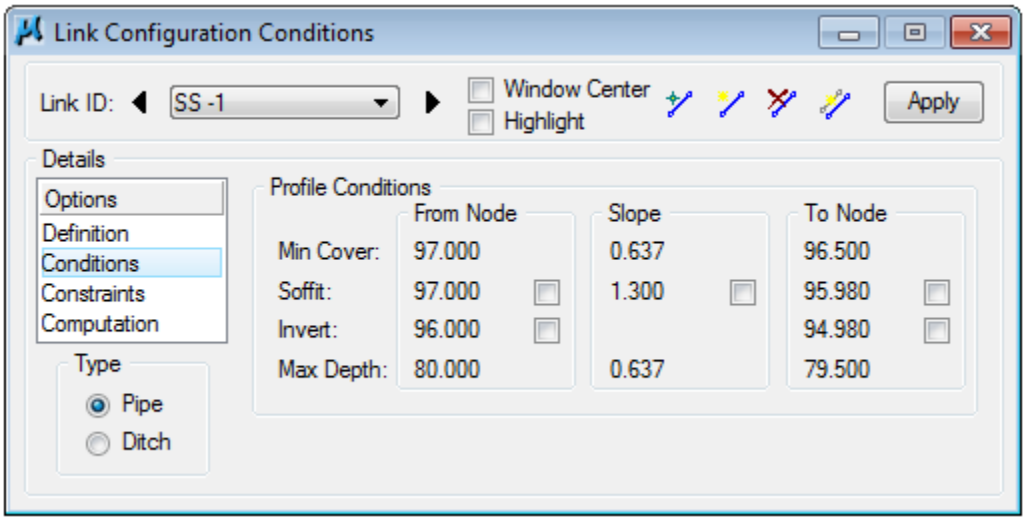
.

Conditions Options

The elevations corresponding to minimum cover and maximum depth are taken from the node configurations.

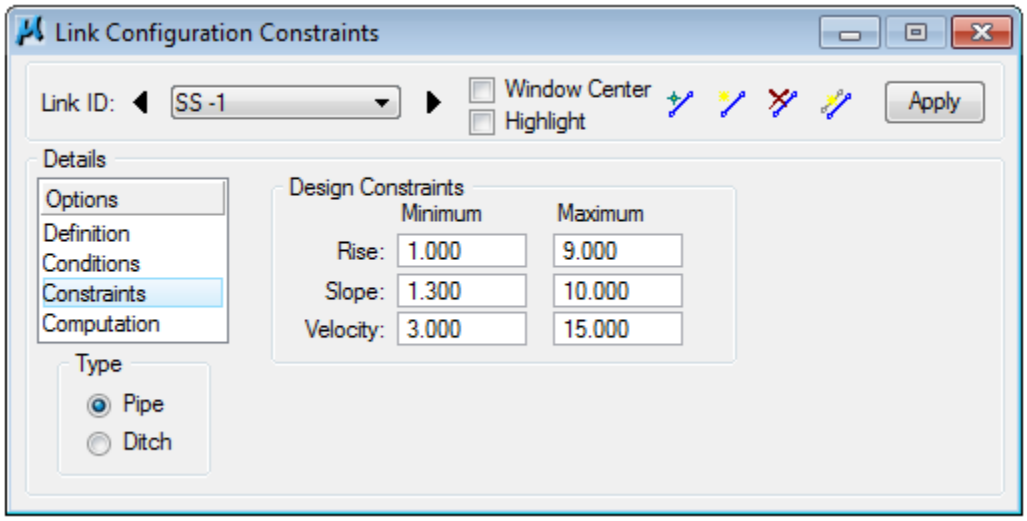
Note the soffit (top inside surface of pipe) and invert (bottom inside surface of pipe) are initially zero. After the pipe has been designed (or if a pipe diameter had been selected under the definitions option) these values will be populated.

Any two parameters can be fixed in this dialog, by selecting the toggle next to the field. For example you could force the soffit at node CB-1 to an elevation of 97 and a slope of 1.5%.



Constraints Option

Set the **minimum and maximum rise** (pipe diameter) the program should use. The program will design a pipe by choosing from the drainage library between these two extremes.
Set the **minimum and maximum pipe slopes**. Select a minimum slope steep enough to establish a self-cleaning velocity.
Set the **minimum and maximum velocity**. Select a self-cleaning velocity for the minimum.

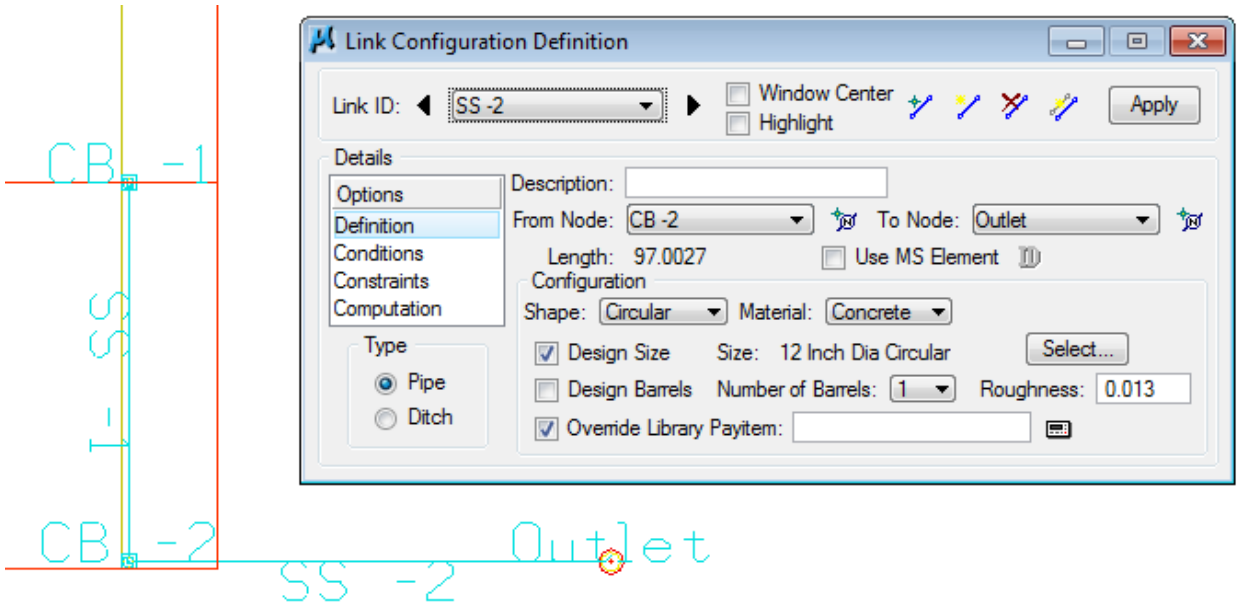


Computation Option

Note there will not be any computations available until after a network has been created.

3. Select the Apply button to store the pipe in the gdf file.

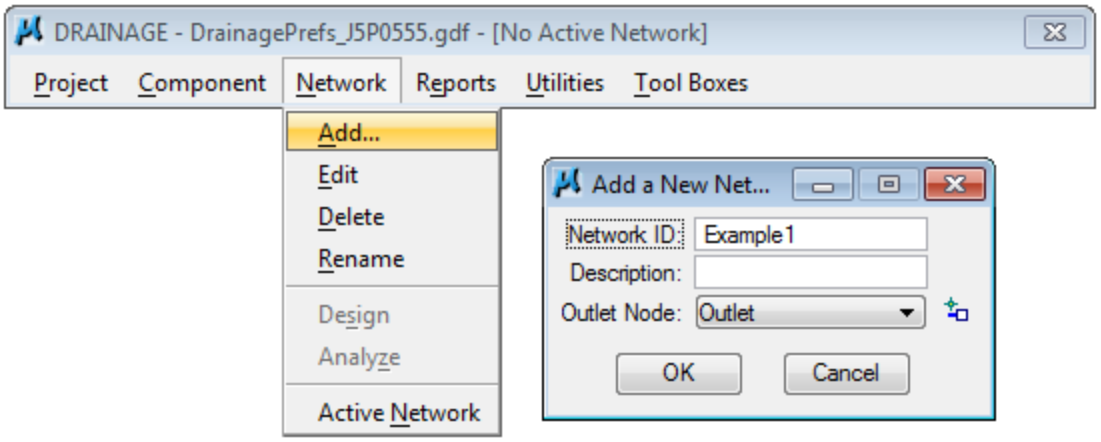
4. Add pipe SS-2 in a similar fashion.



4.6 Create a Network

1. Go to Drainage >Network >Add menu

2. Enter **Example 1** for the network id
3. Specify the **Outlet Node** as **Outlet**
4. Select **OK**

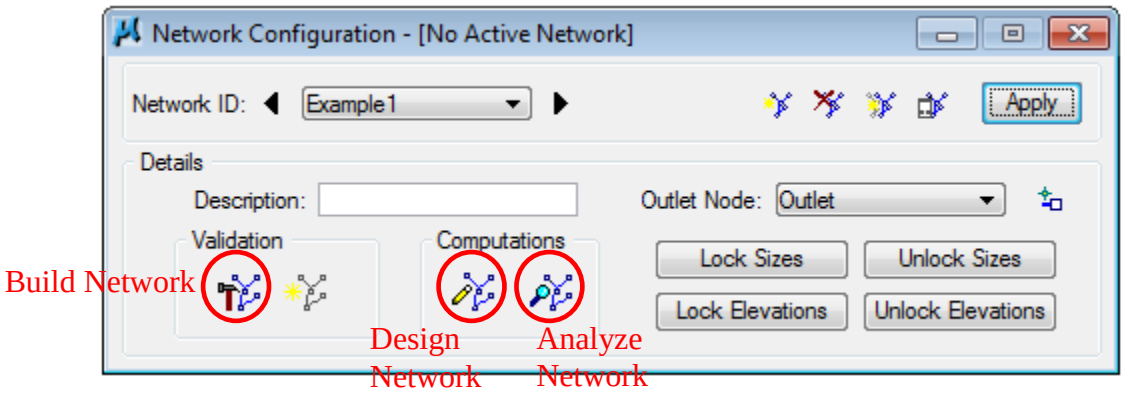


5. Select the **Build Network** button
   The Build Network function checks for connectivity and establishes the flow direction through all of the links. This function also validates all components for errors and unresolvable issues such as loops, duplicate links, no outlet, or multiple outlets.

6. Select the **Apply** button to place the network in the GDF file.

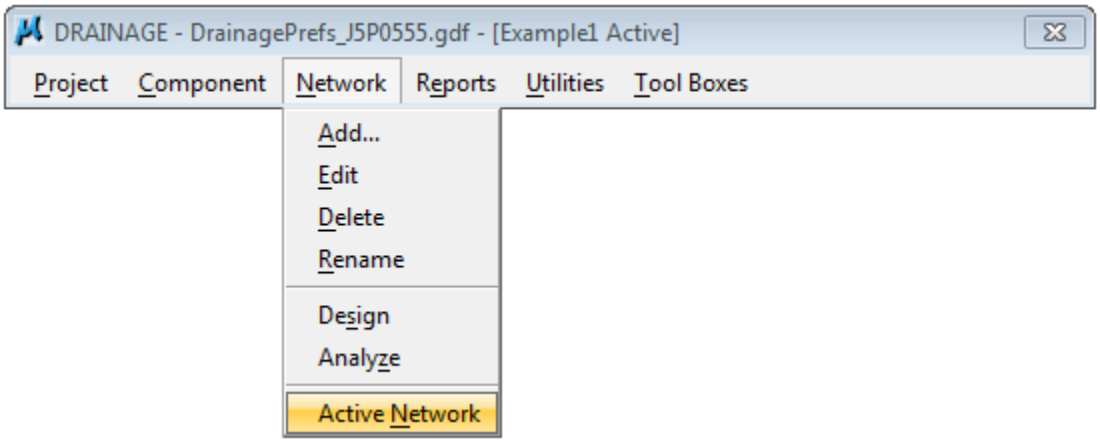
7. Select the **Design** button to design the network based on any design toggles that have been selected. (Do not click the **Apply** button after choosing **Design**)

   The **Analyze** button will ignore any design toggles that have been selected and simply performs a hydraulic analysis of the network.



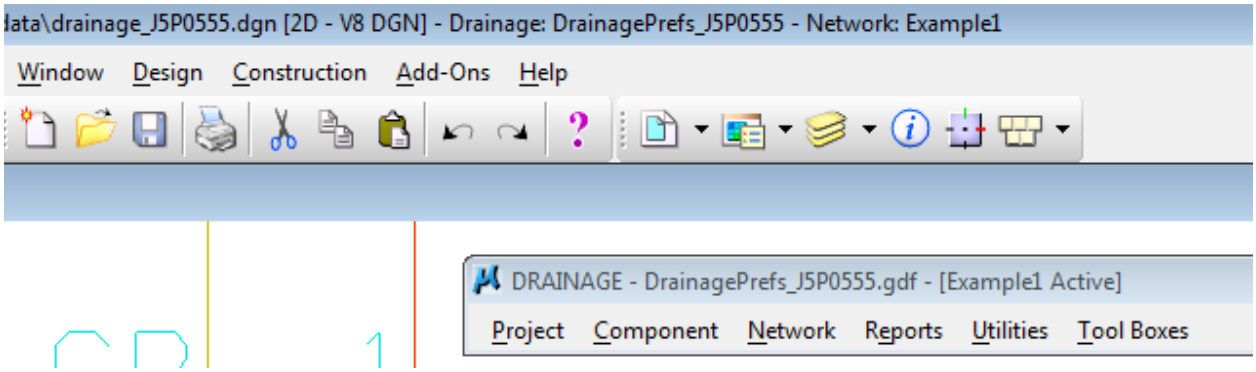
Lock Sizes will lock all sizes in the network before designing
Lock Elevations will lock all elevations in the network before designing

8. Now go to the Network drop-down menu on the Drainage menu bar and select Active Network



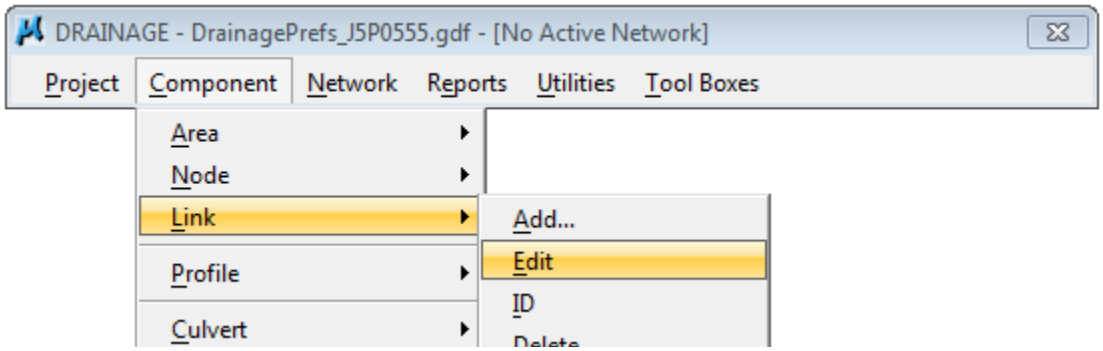
A drainage project can contain multiple networks and GEOPAK Drainage uses the Active Network feature to limit the amount of information processed by the application. This speeds up performance by focusing reporting, querying, and navigation tools on the current active network. It is a good practice to assign the network you are working on as the active network.

Note that **Example1** is now displayed in the banner as the active network.



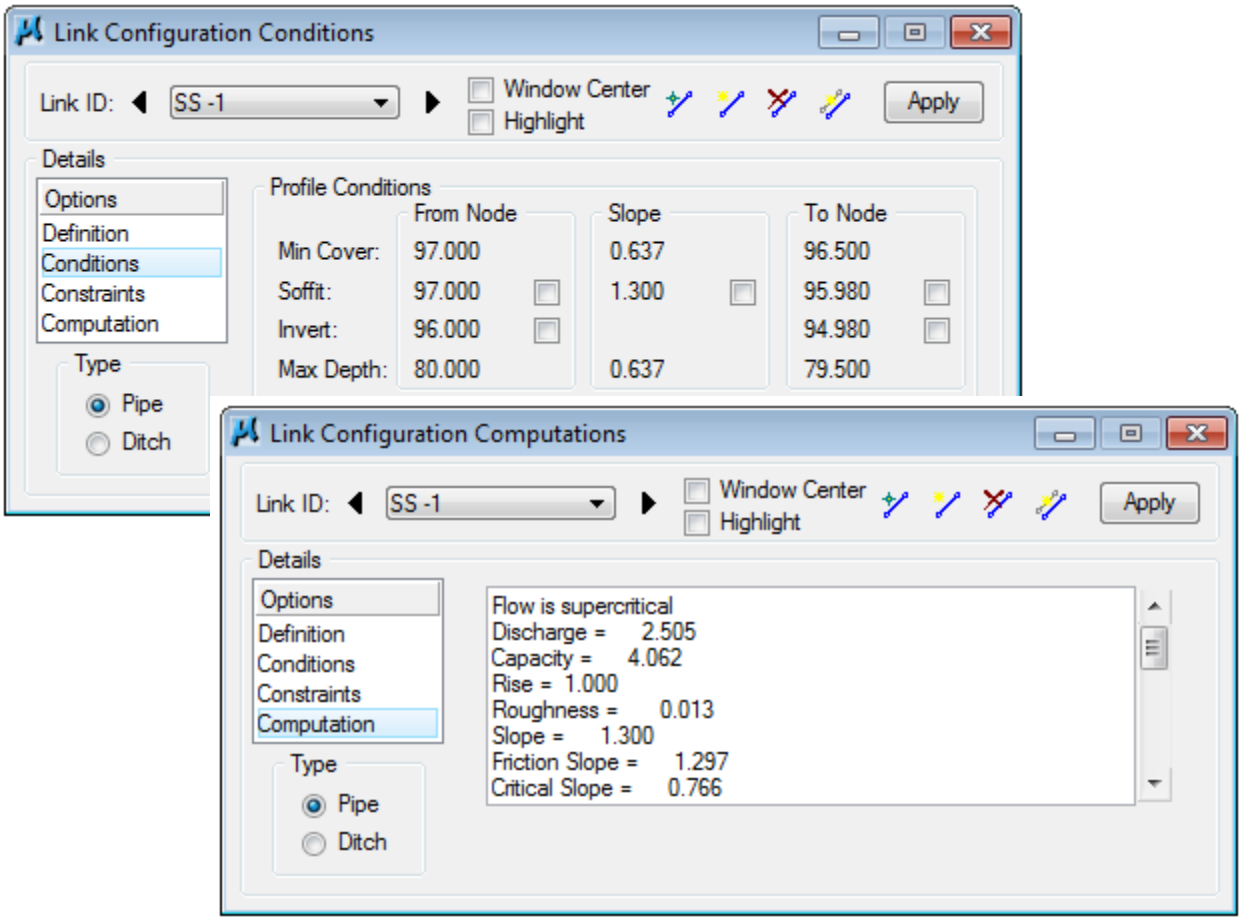
The computations for the pipes can now be viewed.

1. **Go to Component >Link >Edit**



2. **Data point on either the link or text for pipe SS-1**

Both the conditions and computations now have values.



Chapter 5

Advanced Storm Sewer Placement

5.1 Objectives

Create and place stormwater inlets and associated drainage areas in GEOPAK Drainage using advanced placement options
Use existing GEOPAK Road data in drainage design
Create a reach and view the storm sewer profile

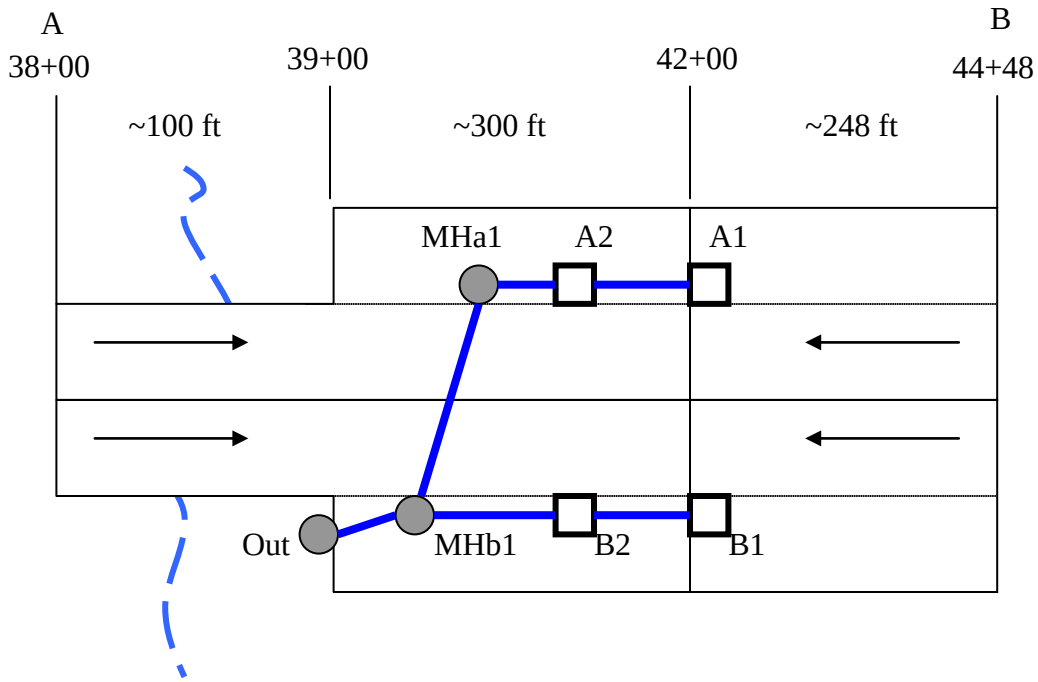
We will be introducing a few new tools and options in this exercise including the Navigator feature in Drainage.

5.2 Overview

GEOPAK Drainage allows the use of existing GEOPAK Road data in the design of drainage systems. Information such as topography, stationing and roadway cross-sections can be incorporated into GEOPAK Drainage, greatly simplifying the manual input of data.

Using existing GEOPAK Road data, we will construct a drainage system as shown below.

Simplified Plan View

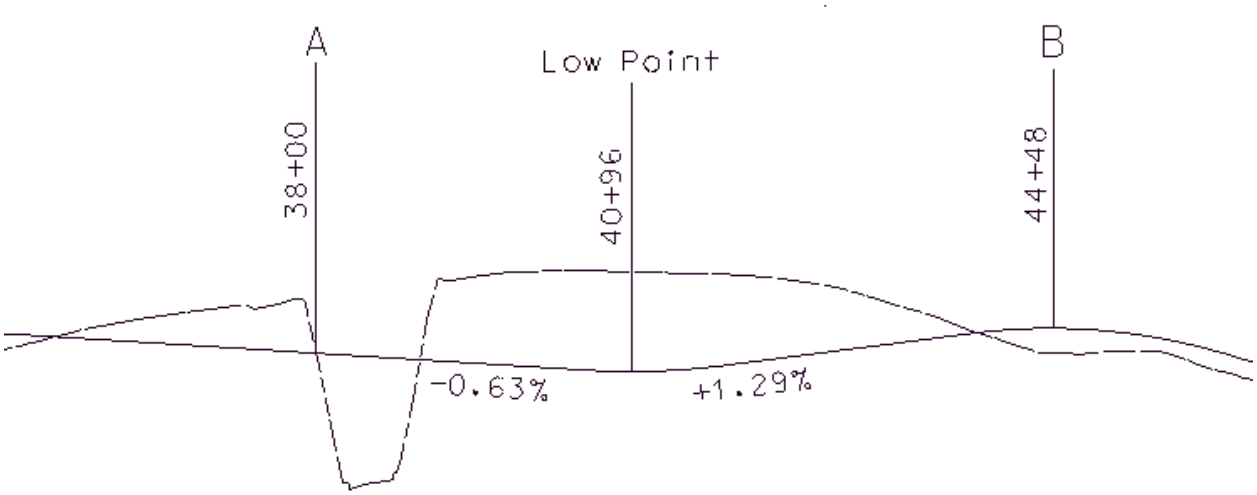


There is a 40 ft wide bridge structure from Sta 38+00 to Sta 39+00 that we will collect runoff from. From Sta 39+00 to Sta 44+48 we will be collecting runoff from centerline out to a distance of 75 feet.

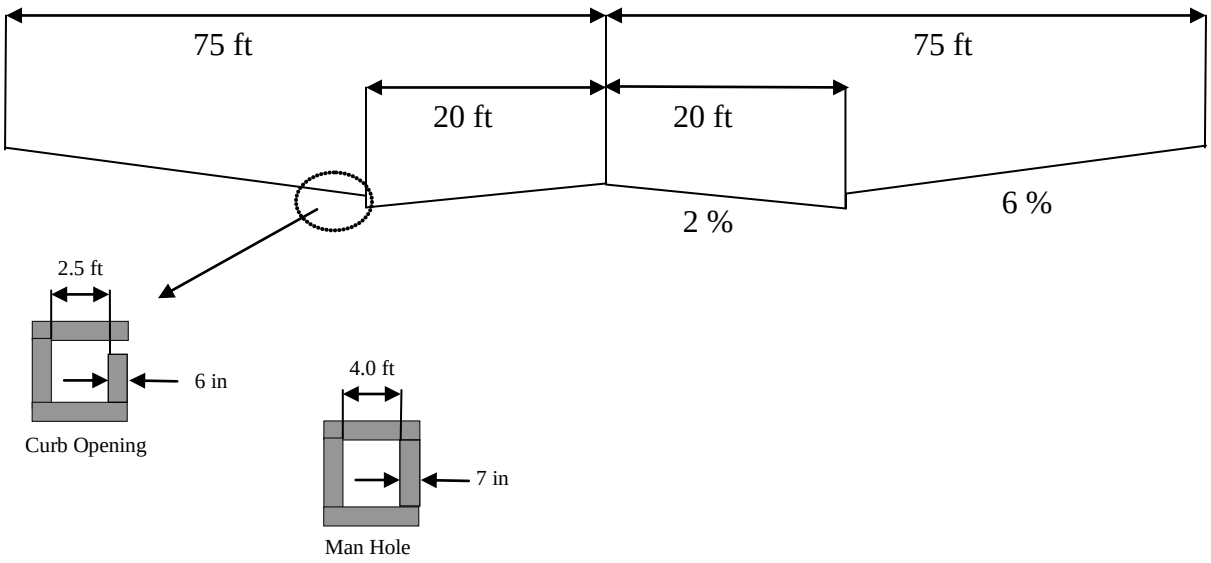
We will design for a **10-year** storm in **District 5**.

Additional geometry that we will use for constructing our system is shown below:

Profile View



Typical Section



5.3 Set Project Preferences

For this exercise we will be working with the Cole\J5EX502 project.

The first few steps will involve creating the files we will be using to construct our drainage system and setting up the project preferences.

1. Using ProjectWise, open the plan_J5EX502.dgn file from the project's data folder

2. Using the MicroStation File menu, choose the Save As option and save as a new file named drainage_J5EX502.dgn

- You can use the No Wizard option when that dialog pops up, and make sure you save your new file to the data directory of the project (it should default to the correct location)
- Make sure to Check-In the plan_J5EX502.dgn file when prompted

A file has already been set up that outlines the land uses for our project area. We simply need to make this file available to us by referencing it into our drainage plan.

3. Access the References dialog box and attach the landuse_J5EX502.dgn file (located in the project data directory) using the Coincident-World orientation

4. In order for Drainage to recognize the land use assignments the geometry must be in the active design file, so using the Merge to Master function merge the landuse_J5EX502.dgn file into the active file and then close the References dialog

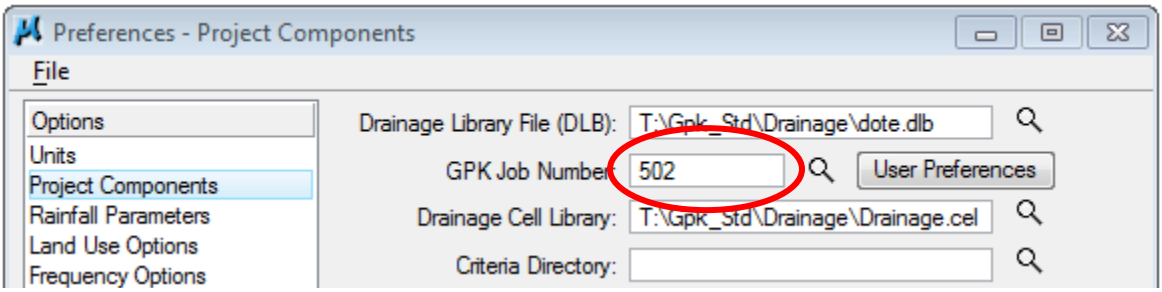
Now we need to set up our drainage preferences for the project.

5. Follow the instructions in Chapter 1 outlined on page 5 for Beginning a New Drainage Project

When you reach step 6 in the instructions name your drainage project **drainage_J5EX502.gdf**

6. Once your project is saved then open the Preferences dialog box again so we can make some changes necessary to fit our project

7. Select the Project Components Option. Associate job502.gpk with this drainage project. The gpk file is located at T:\de-proj\Cole\J5EX502\data.



Also make sure your working directory is set to **\J5EX502\data** in the correct ProjectWise location under the User Preferences

8. Select the Rainfall Parameters Option and choose District 5

9. Make sure the Frequency Option is set to 10 years

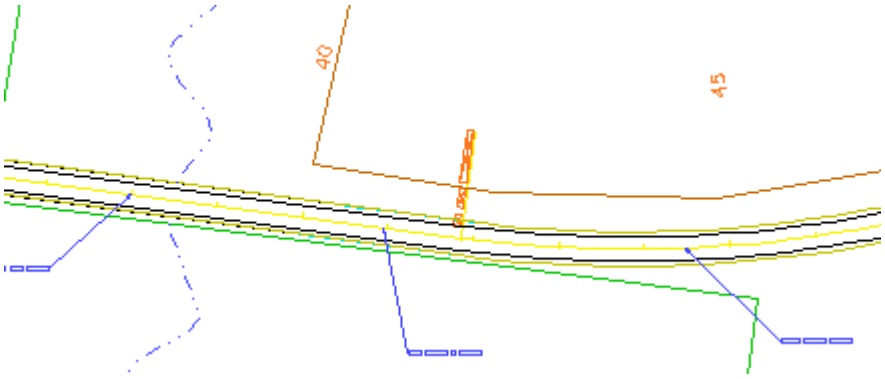
10. Select the Node Option and change the prefix from " CB – " to " A - "

11. Click OK on the Preference dialog and choose Yes when prompted to store changes

12. Under the Drainage Project menu options, choose the Save option

This will store the changes we made to our project GDF file

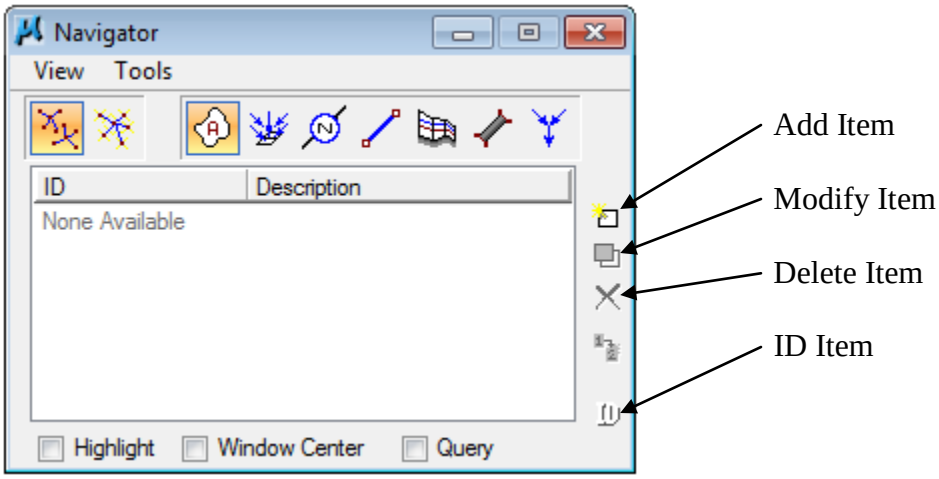
13. Zoom in on area between station 38+00 and station 45+00



5.4 Inlet Placement using Navigator

One of the methods for adding components to a project is with Drainge's Navigator. We will be working with the basic operation of the Navigator throughout the rest of this exercise.

1. **Access the Navigator by going to Utilities >Navigator**



 Displays components for all networks in the Drainage file

 Displays components for only the active network

 Displays the drainage areas

 Displays the inlets

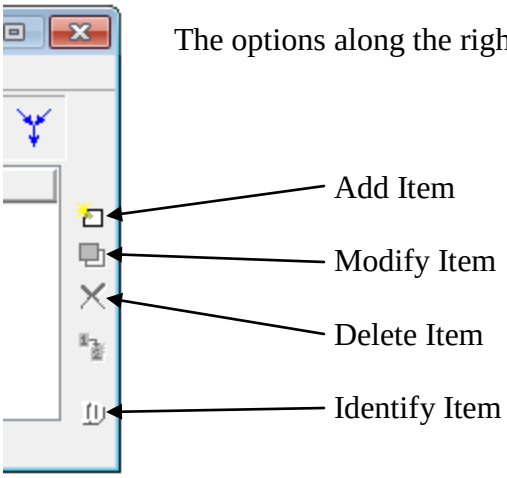
 Diplays the nodes (inlets will also be included)

 Displays the links (pipes or ditches)

 Displays the profiles (reaches)

 Displays the culverts (not covered in this class)

 Displays system routing (not covered in this class)



The options along the right side of the Navigator dialog allow you to:

2. Select the Inlet icon from Navigator and choose the Add Item icon on the right.

3. The inlet should automatically be assigned the name A-1 since we set that up in the Preferences, so click OK to begin assigning the properties of the inlet.

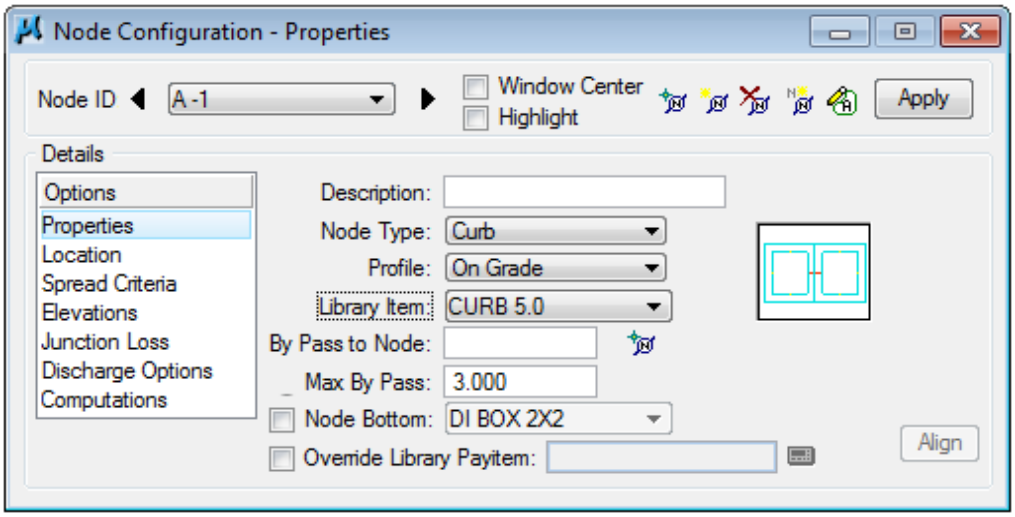
4. Now use the information provided for each option to construct the A-1 inlet.

Instruction 4 involves using the data and screenshots contained on the following 5 pages.

Properties

Node Type: **Curb**
Profile: **On Grade**
Library Item: **CURB 5**

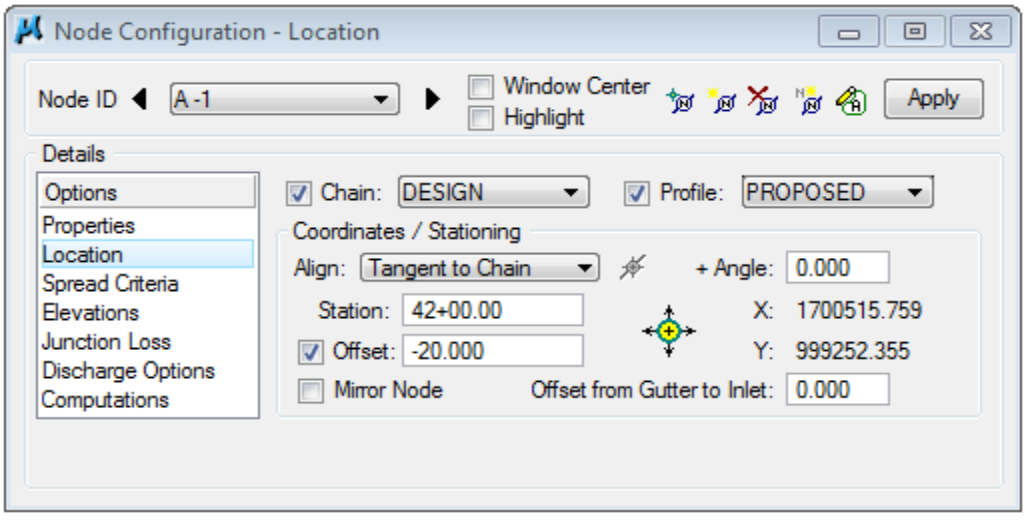
Click Apply in the upper right corner



Location

Reference Chain: **DESIGN**
Profile: **PROPOSED**
Align: Tangent to Chain
Station: **42+00**
Offset: **–20 ft**

Click **Apply**



Spread Criteria

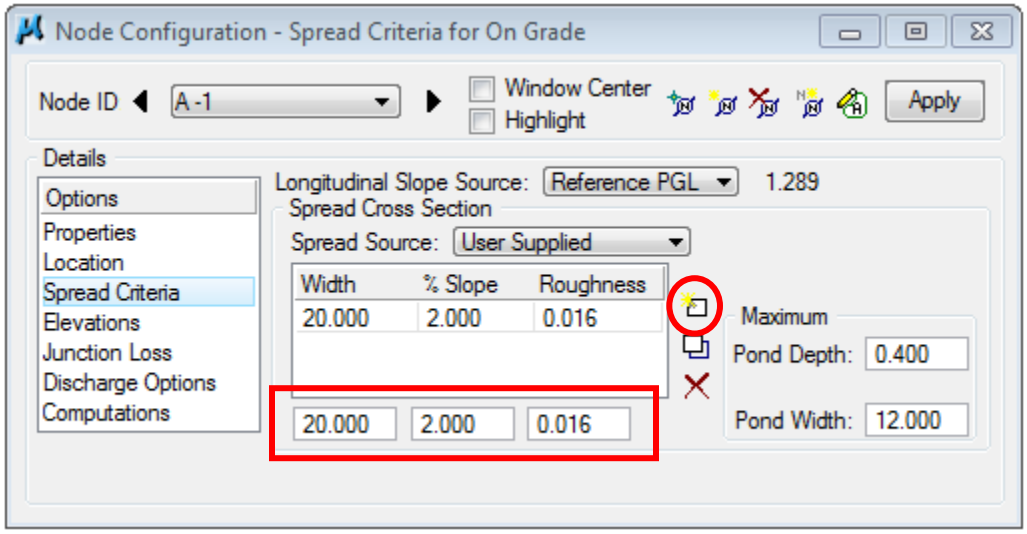
Longitudinal slope: **Reference PGL**

Enter the provided spread section data in the input fields highlighted below and then click the add item icon along the right side.
Width = **20**
Slope = **2%**
Manning's n = **0.016**

Maximum Ponded Depth: **0.4**
For inlets at grade this does not enter into the hydraulic computations. It is merely used for querying the system. The depth is measured at the inside of the curb face. For a 6" curb a maximum allowed depth is 0.4 ft.

Maximum Ponded Width (spread): **12**
This value does not enter into the hydraulic equations. It is merely used for querying the system and generating a warning if the value is exceeded.
Section 640.1.2.2 of the EPG covers Gutter Spread. The allowable spread is based on the functional classification of the roadway. The maximum allowed under any circumstances is 12 ft and so we will use that for our situation.

Click **Apply**



Elevations

Choose a TIN File by selecting the magnifying glass icon and selecting **J5EX502.tin** from the file listing in our data directory.

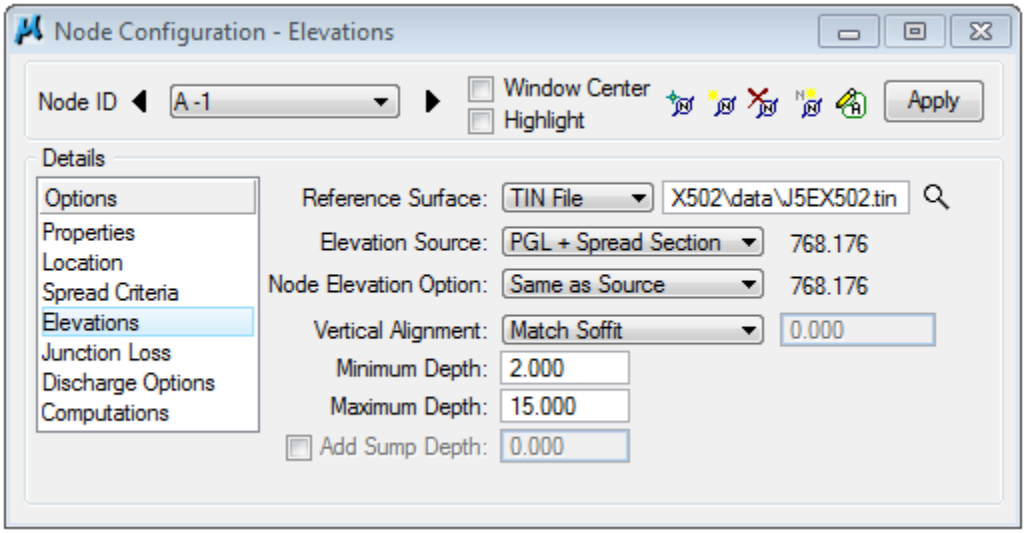
Elevation Source: **PGL+Spread Section**
The program will use the PGL elevation at the specified station in conjunction with the spread section to compute the elevation at the inlet.

Node Elevation Option: **Same as Source**
One of the options is to select constant offset. A possible application would be if a profile were defined along the top of curb. The elevation could be offset a distance of the curb height to achieve the inlet elevation.

Vertical Alignment Preference: **Match Soffit**
Another option that could be utilized is select match surface. Match surface will align the pipes based on the flow line.

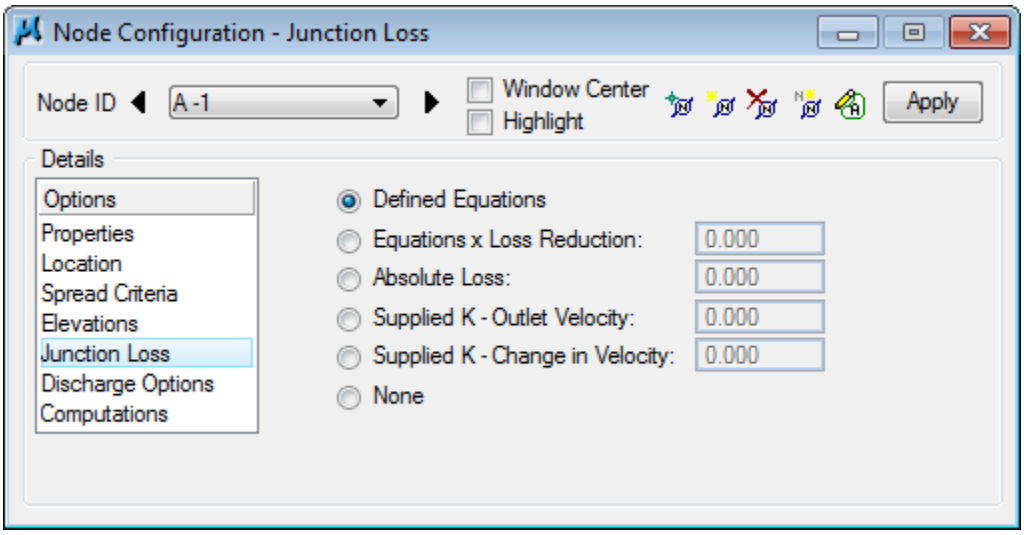
Minimum depth: **2.0**
Maximum depth: **15.0**
This will set the "design envelope" for your pipes.

Click **Apply**



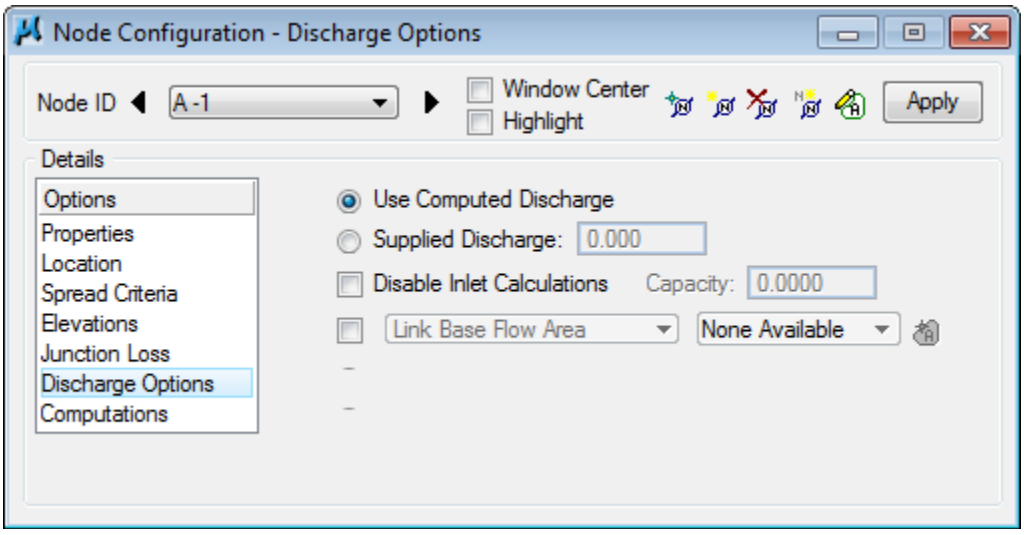
Junction Loss

Select Defined Equations



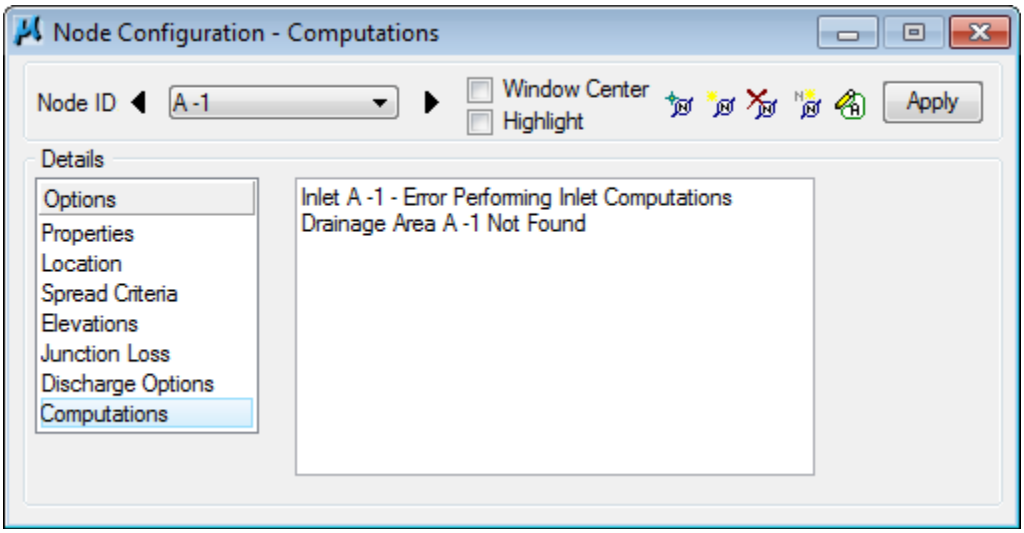
Discharge Options

Select Use Computed Discharge

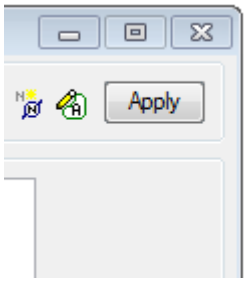


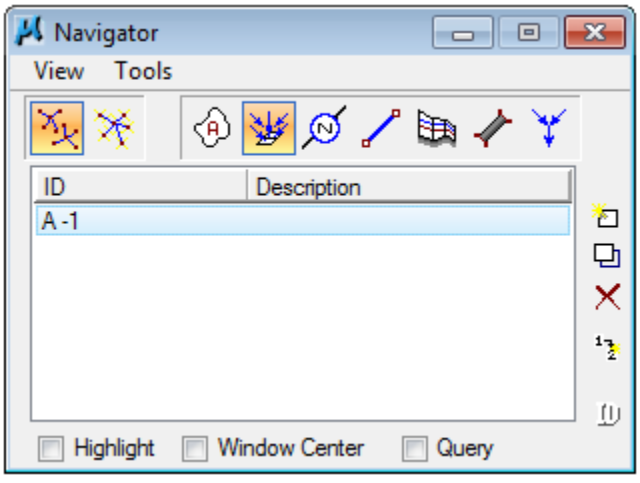
Computations

The computations window will be empty until drainage area A -1 has been added or a discharge has been supplied.



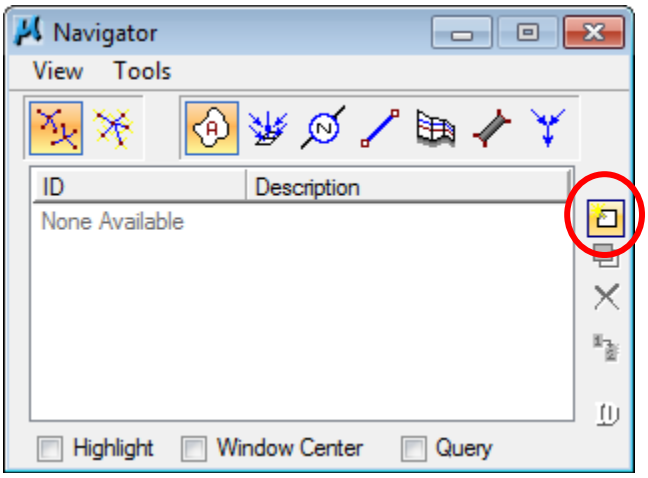
5. Finally click the Apply button to save the inlet to the drainage project. Note that after the inlet has been added it is now contained in the navigator.



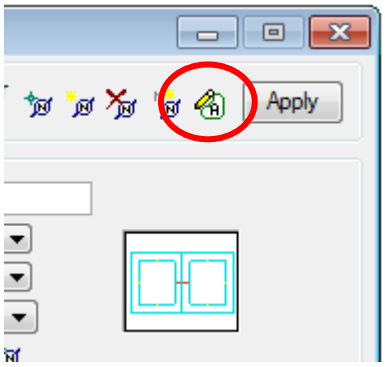


5.5 Area Placement

The area corresponding to inlet A-1 can be placed in several different ways. The first method is accomplished in a similar fashion as inlet A-1 was placed using the Navigator. The second method utilizes the **Edit Area** button within the Node Configuration dialog located in the upper right hand corner.



OR

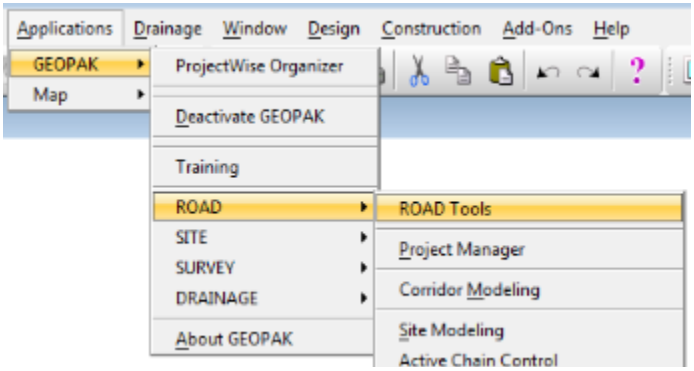


1. Select either of the options above to begin constructing the area

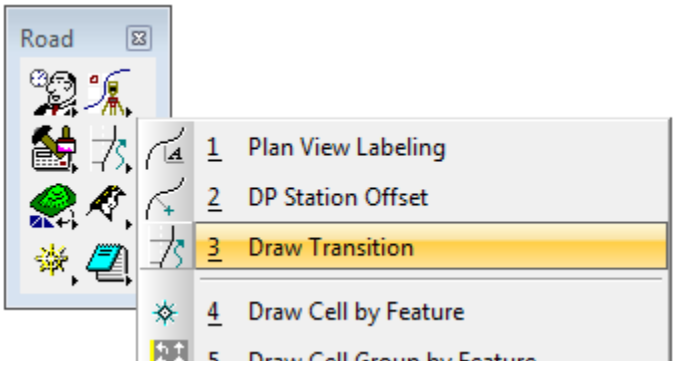
2. Assign the name for your area as A-1 and click OK

Now we need to draw the shape that we will use to establish our physical area boundary. This shape will extend from inlet A-1 to the high point and should have a width of 75 ft measured from the centerline. Utilizing GEOPAK Road tools like **Draw Transition** provides an efficient method of creating the desired shape.

Select the **ROAD Tools** option under GEOPAK >ROAD in the Applications Pull down menu

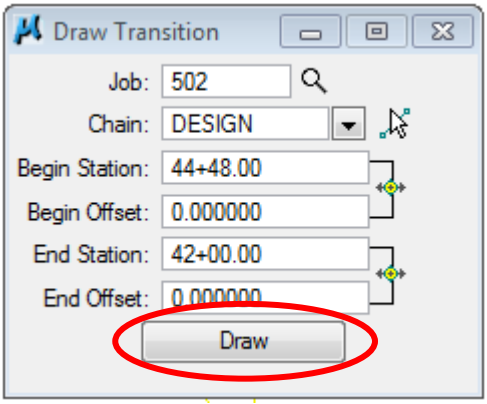


Select the Draw Transition tool from the Plans Preparation set of tools



Key in the job number **502** and select the **DESIGN** chain
Begin Station: **44+48.00**
The high point is located at 44+48.
End Station: **42+00.00**
The inlet at A –1 is located approximately at 42+00.
Begin & End Offset: **0.0000**
For the first line, we want to trace over the top of the centerline from these two locations, so keying in an offset of 0 will ensure this first line will fall directly on top of the existing centerline.

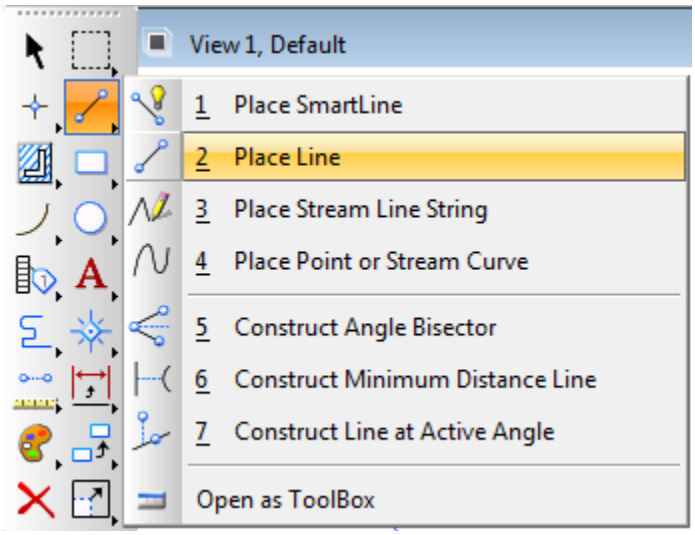
Click the **Draw** button



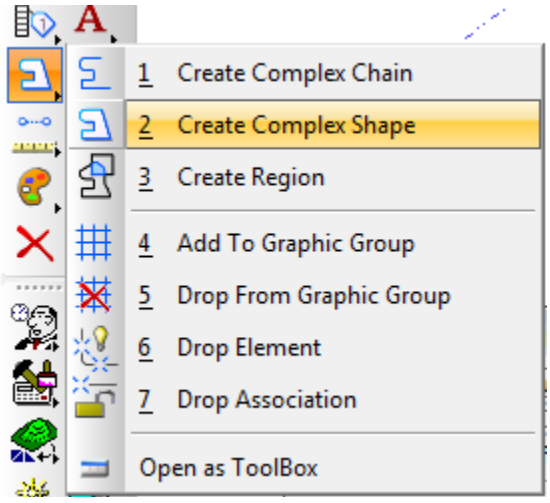
Now we will place the outer boundary of our shape.

With the Draw Transition tool, use the same settings except make the Beginning and Ending Offsets **-75** and click **Draw**

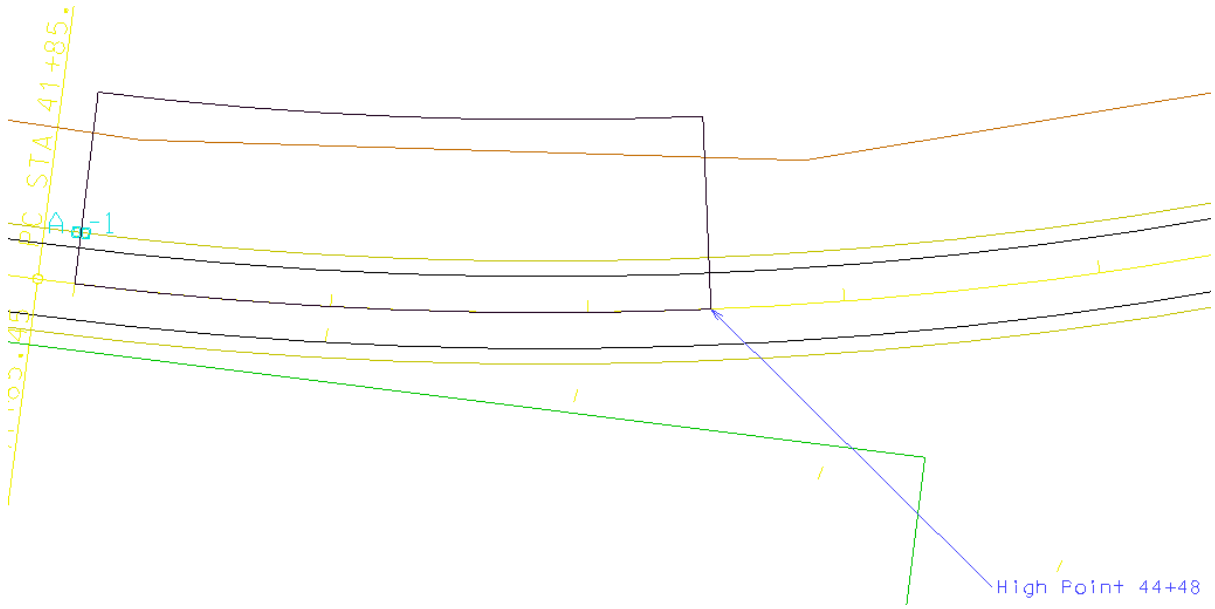
Now using the Place line tool, snap to connect the endpoints of the two chains created in the above steps.



Use the Create Complex Shape tool and select each of the lines and chains that will make up this area



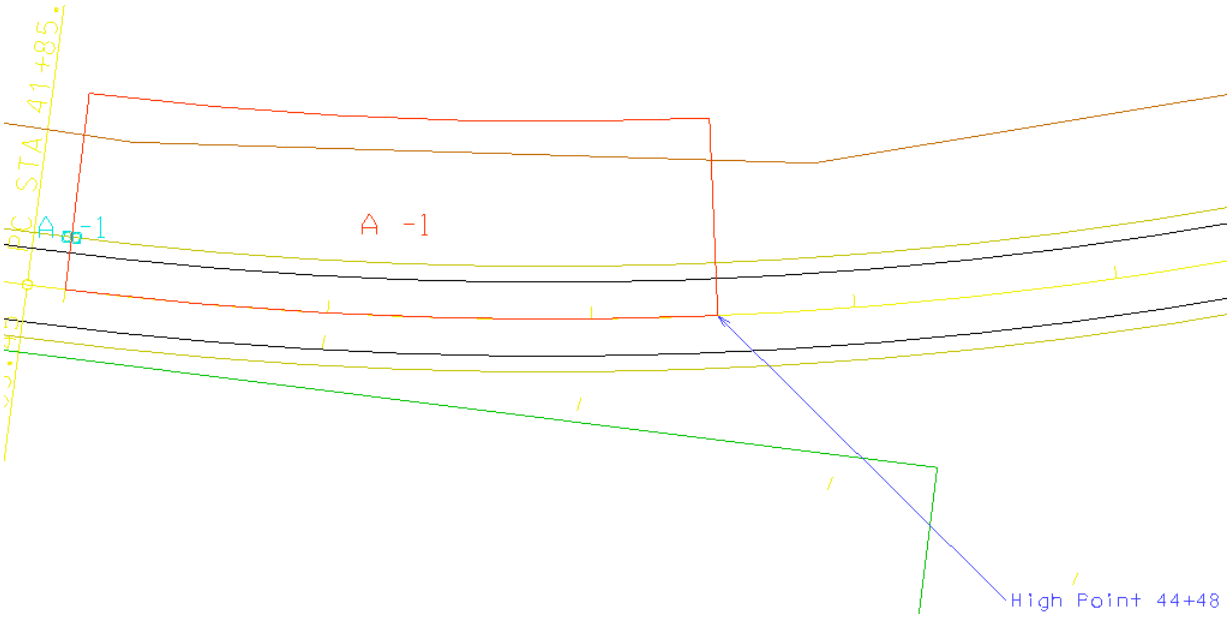
You can use either Manual or Automatic as the Method and you will have to Accept the boundary items once you have all of them selected so that MicroStation will establish the shape.



Now that we have our shape created, we can use it to define the boundary of our first drainage area. We will be using the Drainage Area Definition dialog for this.

3. Click the Select Shape button in the Drainage Area Definition dialog and data point on the drainage area and accept the selection

Once the shape has been accepted the Drainage Area field should be populated. This value will be in acres. The shape that we established in the previous steps should also change to reflect the symbology of a GEOPAK Drainage area and should be labeled with the area name.



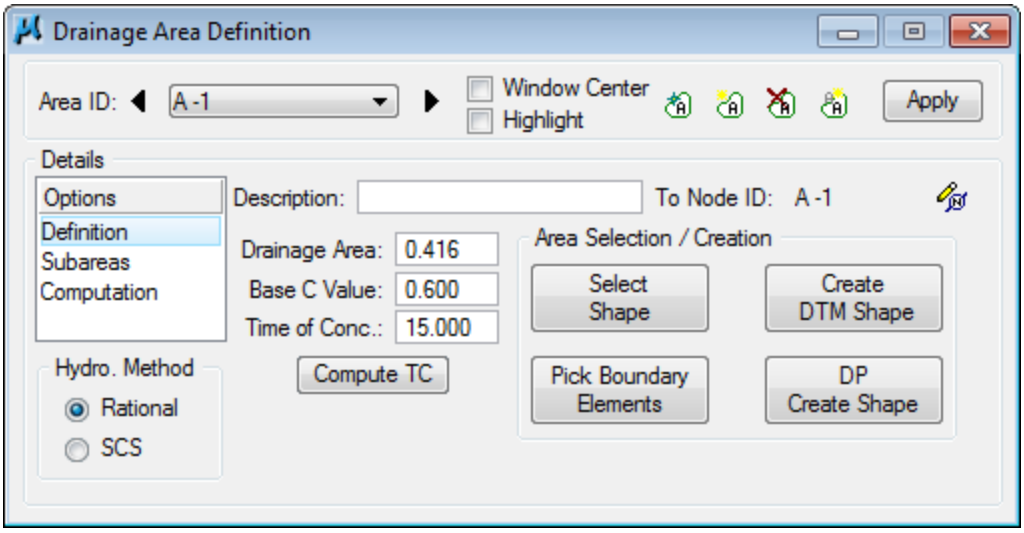
4. Enter the following values for the Definition option:

Base C value: **0.6**

This is the runoff coefficient used in any portions of the drainage area that do not have a pre-defined land use.

Time of Conc.: **15**

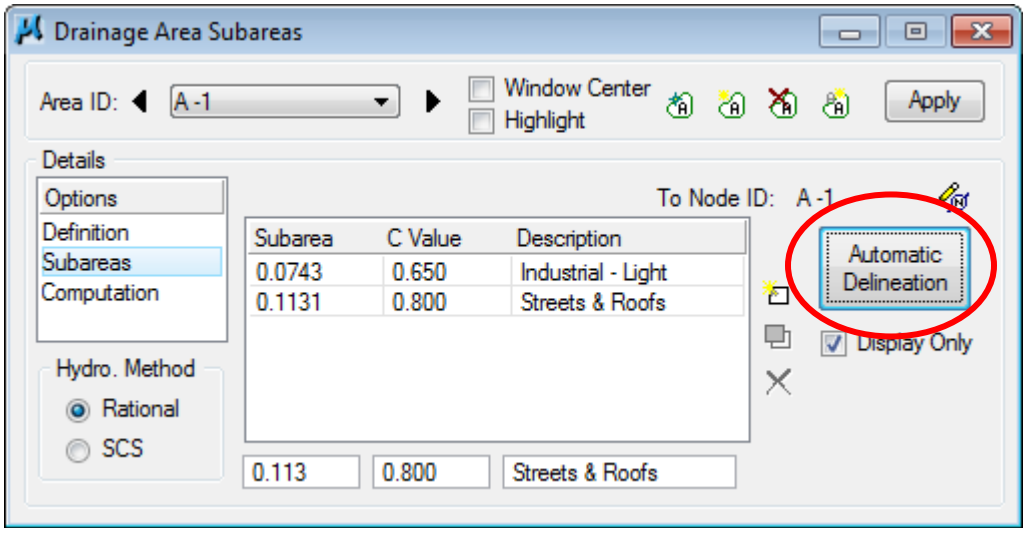
Section 749.5.3 of the EPG outlines Time of Concentration



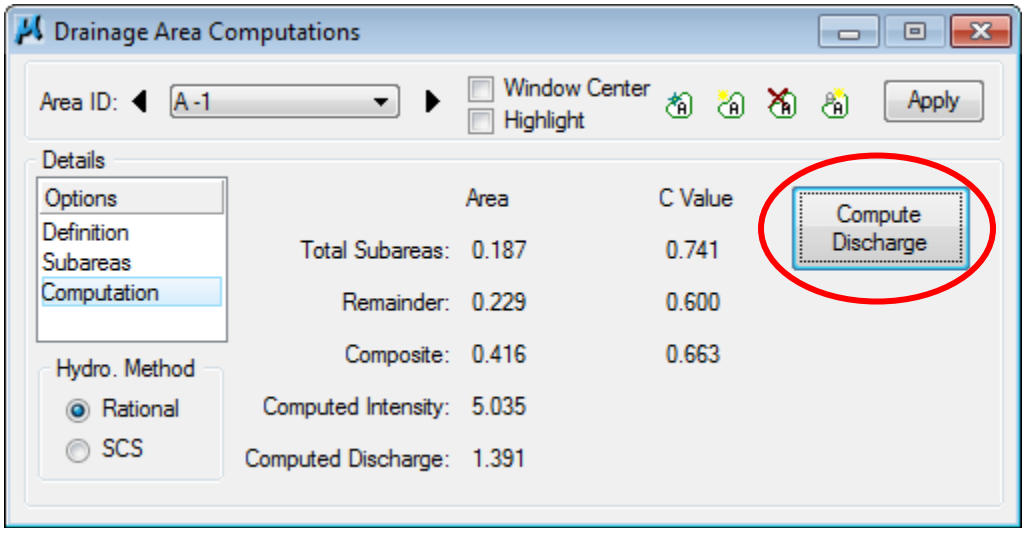
5. Select the Subareas option on the left of the dialog box and use the Automatic Delineation button to delineate the subareas based on the defined land uses

Note that the sub areas will become temporarily shaded based on the corresponding color contained in the drainage library.

If satisfied with the results select the **Apply** button.



6. Select the Computation option and then click the Compute Discharge button to generate intensity and discharge for the drainage area.

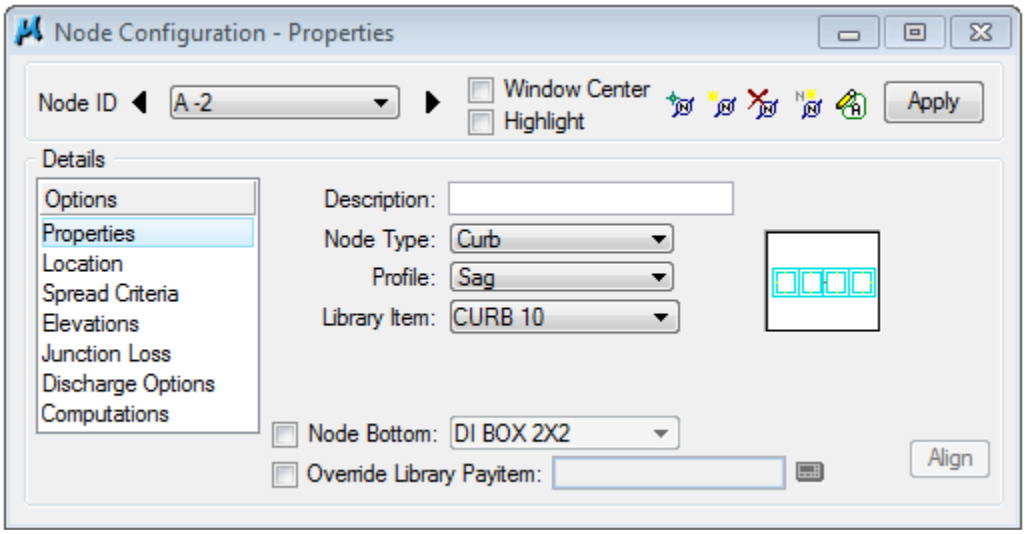


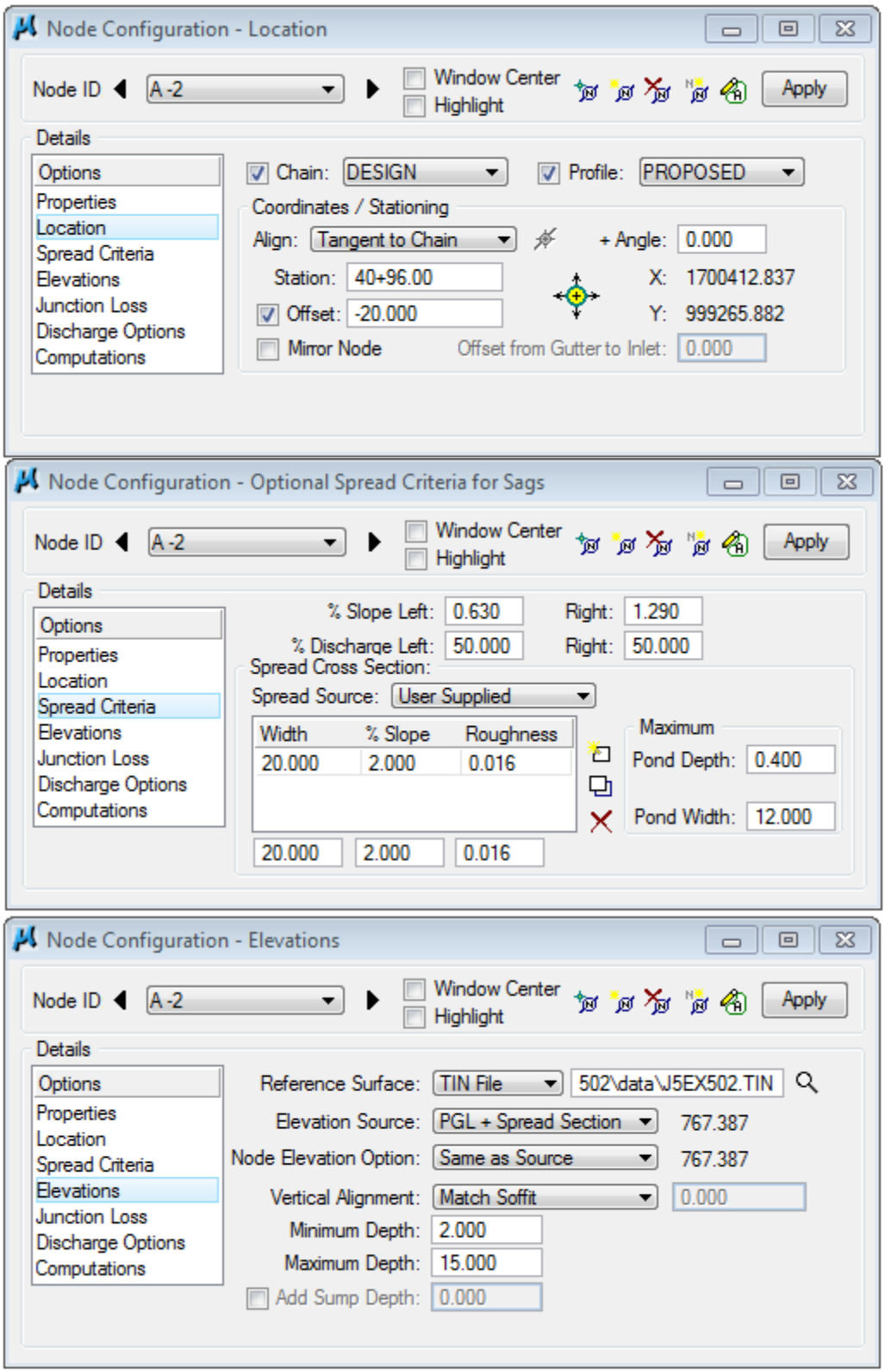
7. Finally select the Apply button to place Drainage Area A-1 in the drainage project

Note that after the area has been added the area is now contained in the navigator.

8. Based on the data supplied in the following screenshots and information, add the remaining nodes and areas.

Inlet A-2

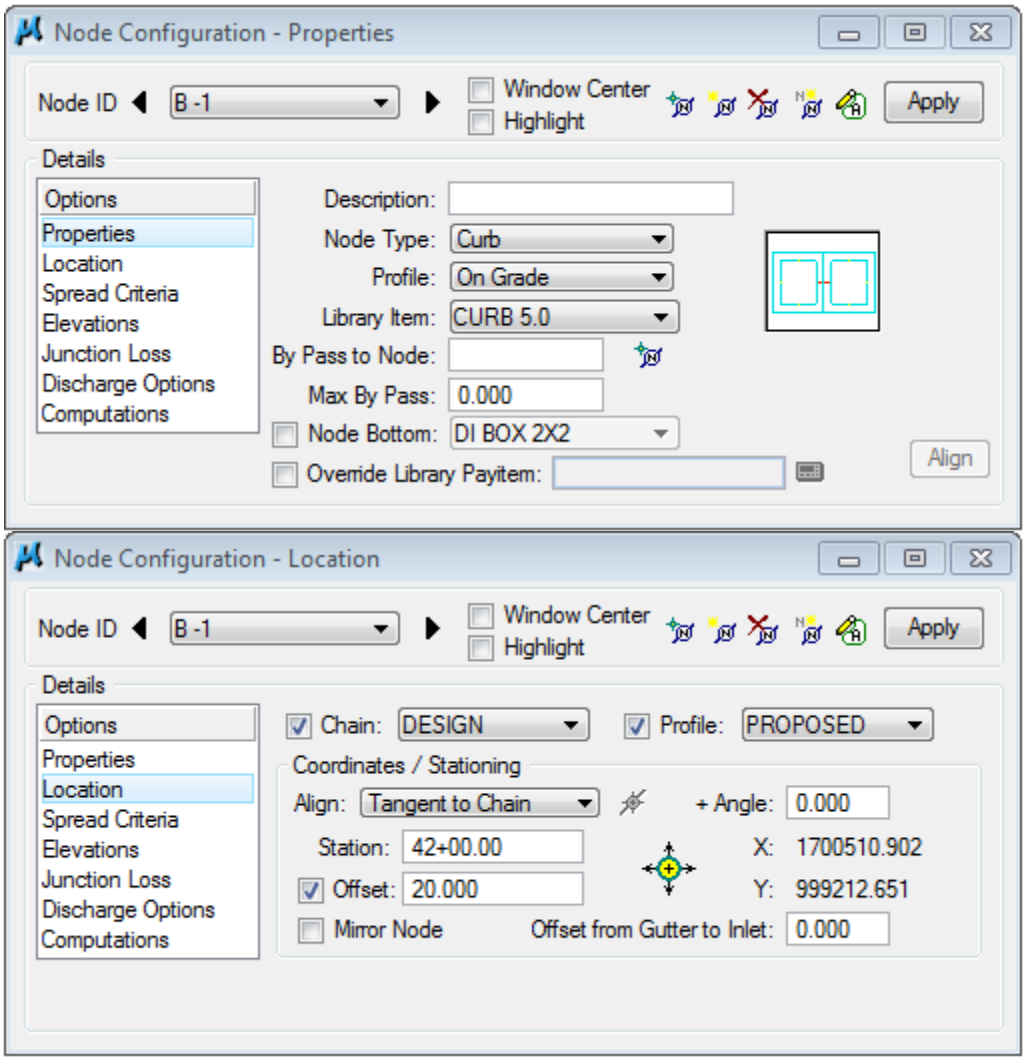


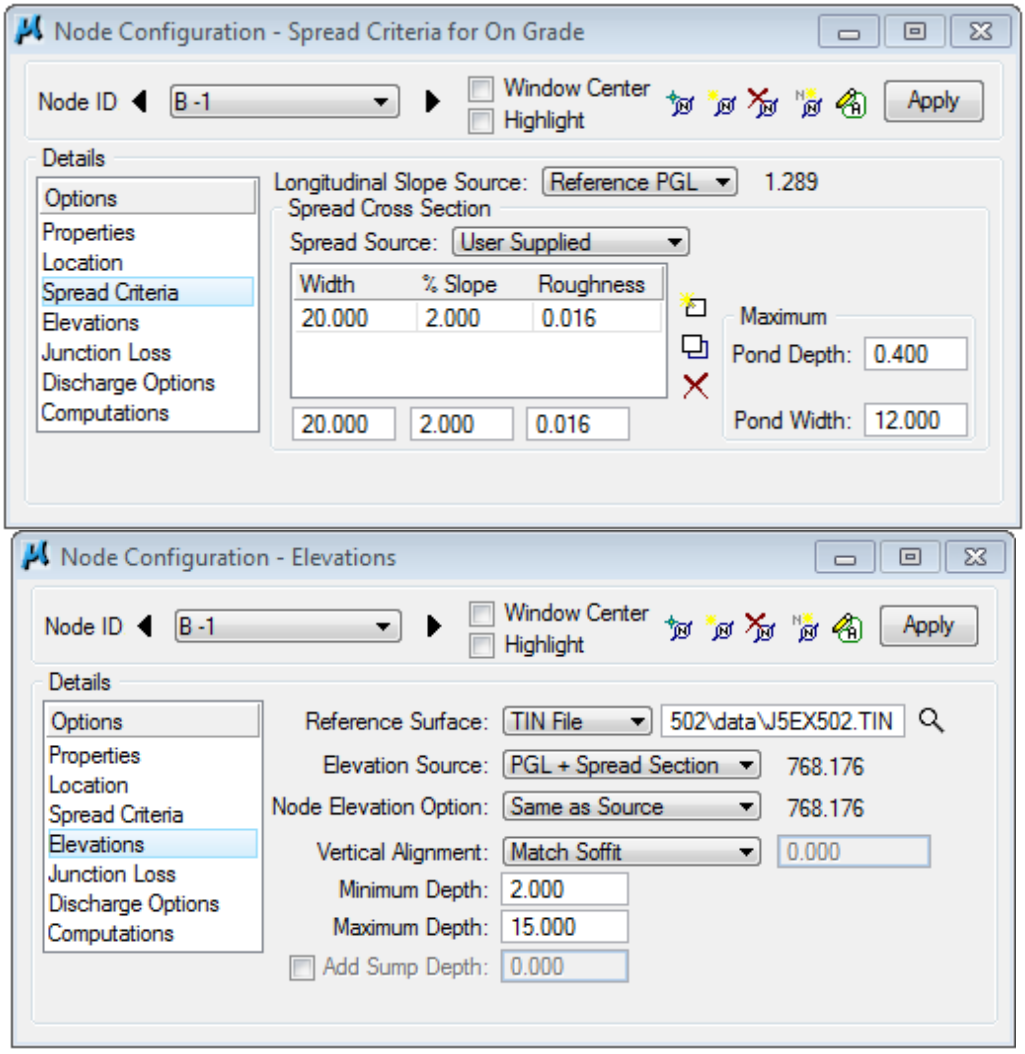

Node Configuration - Location
Node ID A -2
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
Chain: DESIGN
Profile: PROPOSED
Coordinates / Stationing
Align: Tangent to Chain
+ Angle: 0.000
Station: 40+96.00
Offset: -20.000
X: 1700412.837
Y: 999265.882
Mirror Node
Offset from Gutter to Inlet: 0.000
Node Configuration - Optional Spread Criteria for Sags
Node ID A -2
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
% Slope Left: 0.630
Right: 1.290
% Discharge Left: 50.000
Right: 50.000
Spread Cross Section:
Spread Source: User Supplied
Width
% Slope
Roughness
20.000
2.000
0.016
Maximum
Pond Depth: 0.400
Pond Width: 12.000
20.000
2.000
0.016
Node Configuration - Elevations
Node ID A -2
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
Reference Surface: TIN File
502\data\J5EX502.TIN
Elevation Source: PGL + Spread Section
767.387
Node Elevation Option: Same as Source
767.387
Vertical Alignment: Match Soffit
0.000
Minimum Depth: 2.000
Maximum Depth: 15.000
Add Sump Depth: 0.000

Once you've completed Inlet A-2, you can create and assign Area A-2 in a similar manner to what we used for Area A-1 using the Draw Transition tool in GEOPAK. *Remember that Area A-2 contains the bridge as well as the roadway section, so there will be a change in the width that occurs from Sta 38+00 to Sta. 39+00.

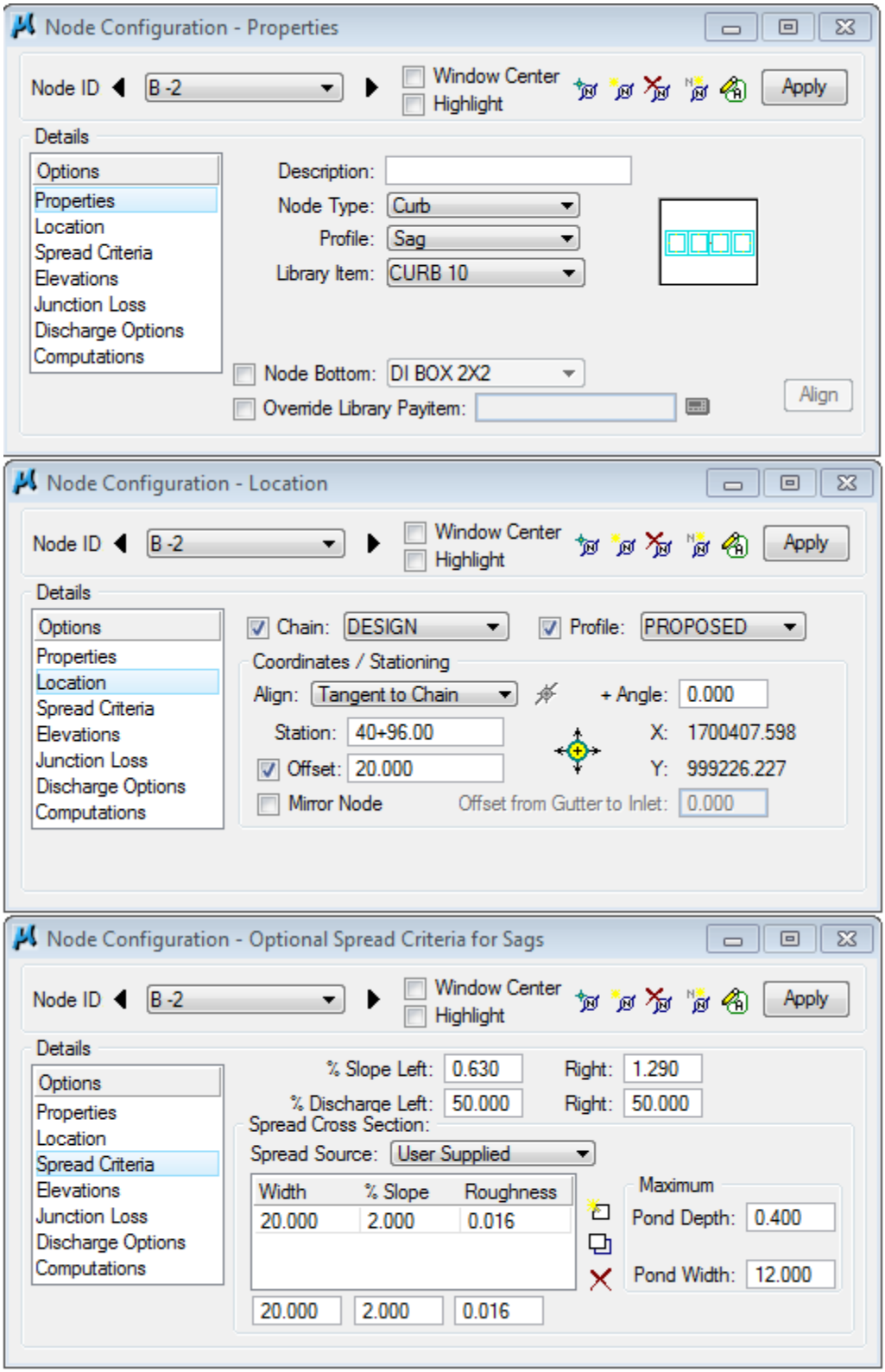
Then follow the same process for creating Areas B-1 and B-2. The only change is that we will use a Tc value of 12 for these areas.
*Area B-2 contains the bridge just like A-2 so there will be a similar offset in width.

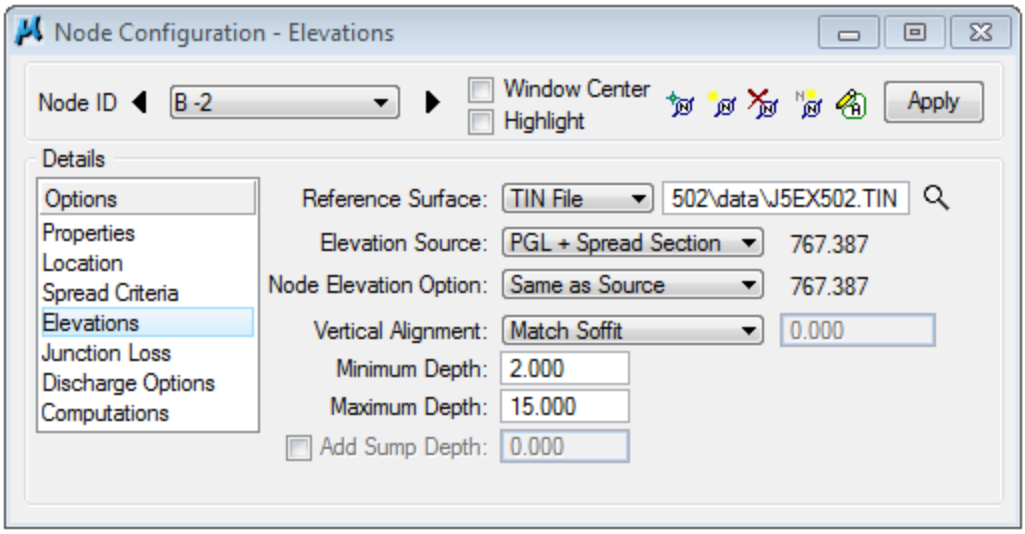
Inlet B-1




Node Configuration - Spread Criteria for On Grade
Node ID
B -1
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
Longitudinal Slope Source:
Reference PGL
1.289
Spread Cross Section
Spread Source:
User Supplied
Width
% Slope
Roughness
20.000
2.000
0.016
Maximum
Pond Depth:
0.400
Pond Width:
12.000
Node Configuration - Elevations
Node ID
B -1
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
Reference Surface:
TIN File
502\data\J5EX502.TIN
Elevation Source:
PGL + Spread Section
768.176
Node Elevation Option:
Same as Source
768.176
Vertical Alignment:
Match Soffit
0.000
Minimum Depth:
2.000
Maximum Depth:
15.000
Add Sump Depth:
0.000

Inlet B-2

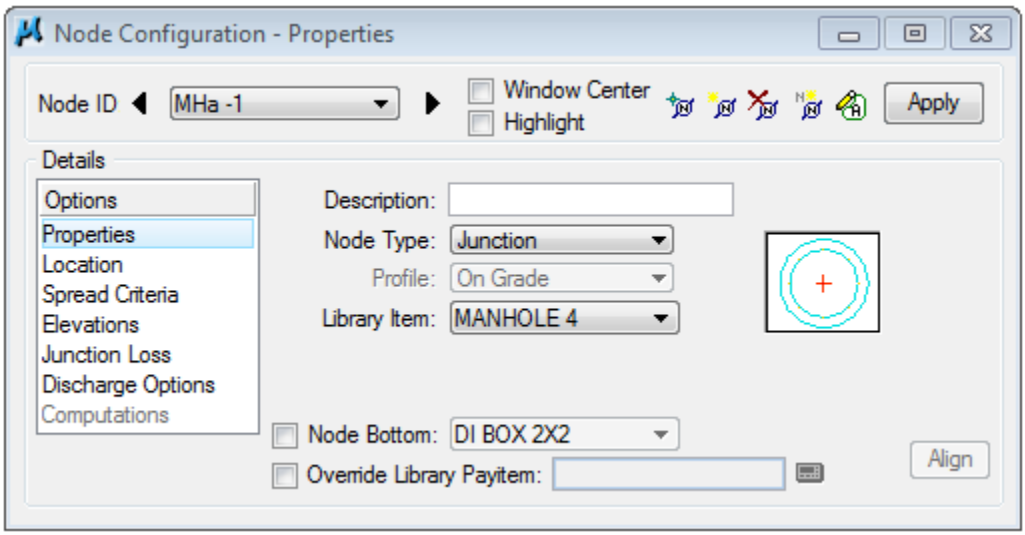

Node Configuration - Properties
Node ID
B -2
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
Description:
Node Type:
Curb
Profile:
Sag
Library Item:
CURB 10
Node Bottom:
DI BOX 2X2
Override Library Payitem:
Align
Node Configuration - Location
Node ID
B -2
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
Chain:
DESIGN
Profile:
PROPOSED
Coordinates / Stationing
Align:
Tangent to Chain
+ Angle:
0.000
Station:
40+96.00
X:
1700407.598
Offset:
20.000
Y:
999226.227
Mirror Node
Offset from Gutter to Inlet:
0.000
Node Configuration - Optional Spread Criteria for Sags
Node ID
B -2
Window Center
Highlight
Apply
Details
Options
Properties
Location
Spread Criteria
Elevations
Junction Loss
Discharge Options
Computations
% Slope Left:
0.630
Right:
1.290
% Discharge Left:
50.000
Right:
50.000
Spread Cross Section:
Spread Source:
User Supplied
Width
% Slope
Roughness
20.000
2.000
0.016
Maximum
Pond Depth:
0.400
Pond Width:
12.000
20.000
2.000
0.016

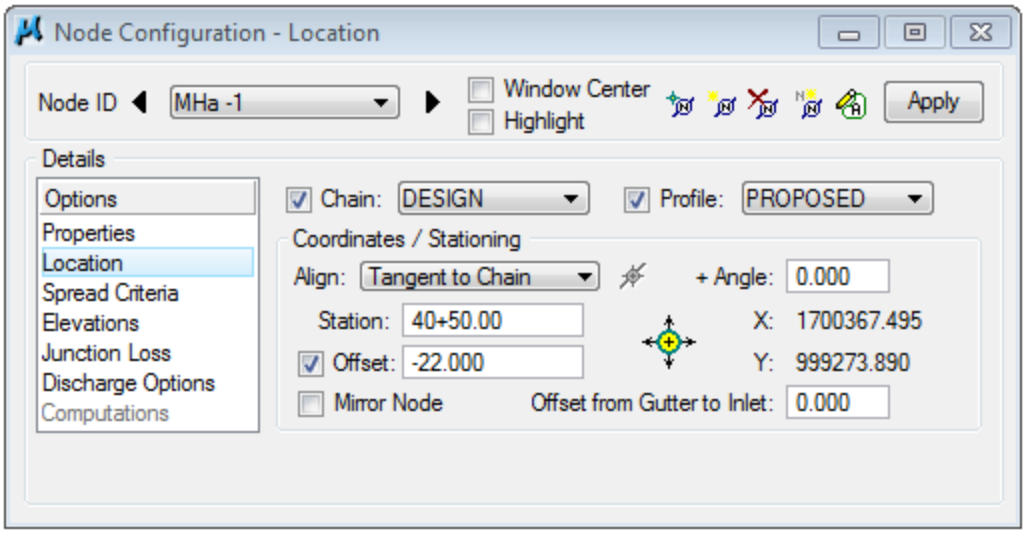


Once you've constructed your 4 inlets as outlined above, make sure you go back to A-1 and B-1 and assign your bypass flow to the downstream inlet.

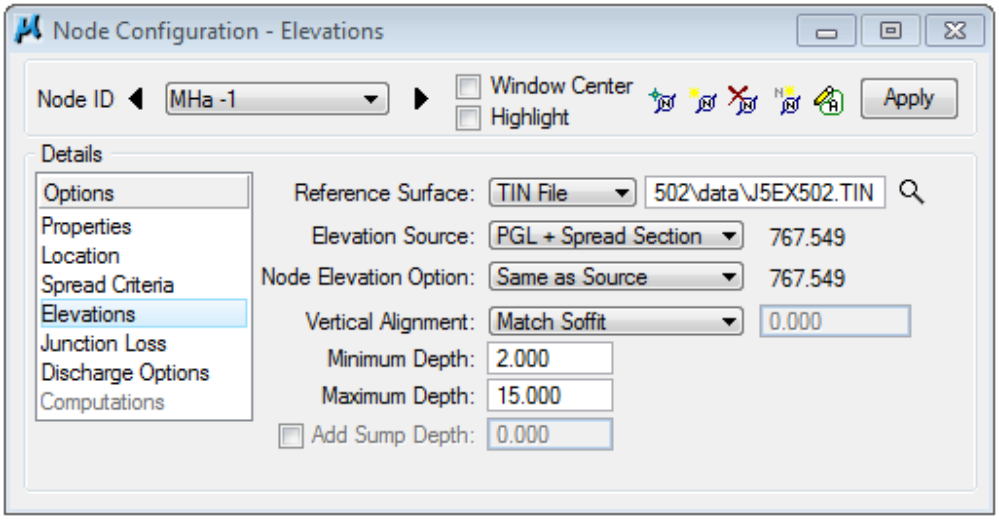
Now we'll place the manholes and the outlet for our system.

Node MHa-1

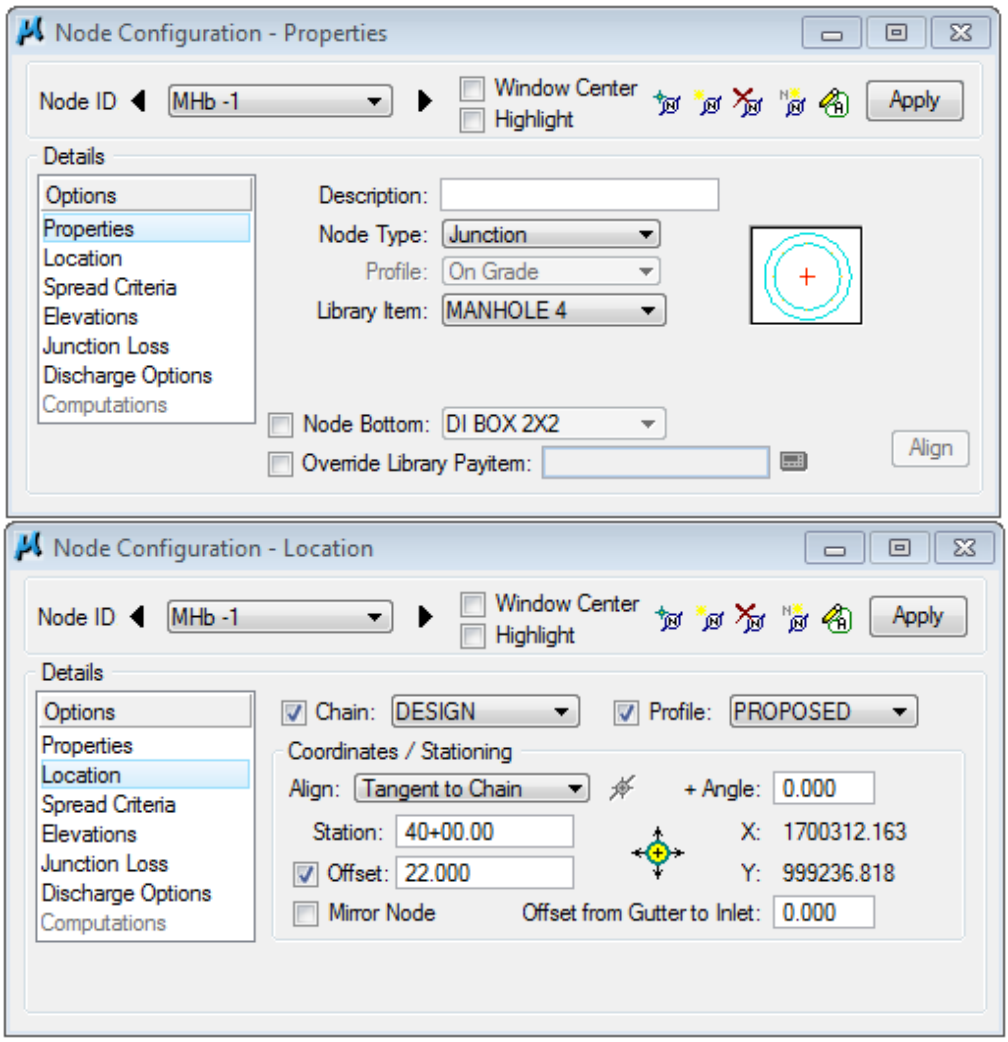




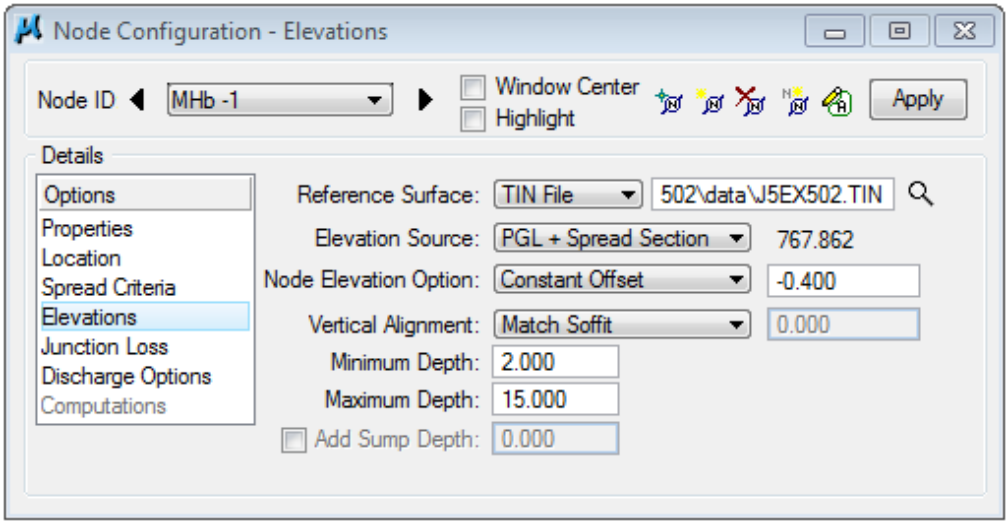
We will use the same Spread Criteria for MHa-1 that we've used for the inlets. This will allow us to use this data to set the Elevation Source by using the PGL + Spread Section option.



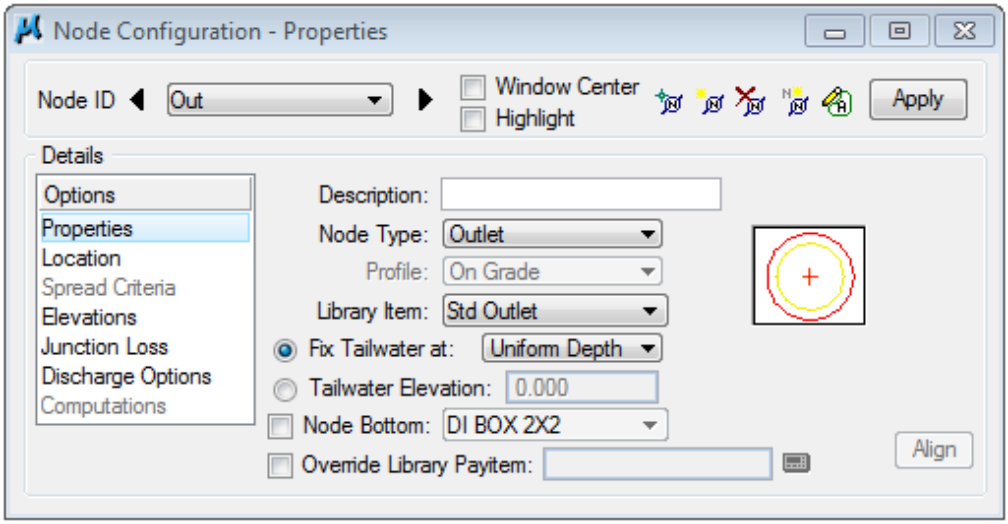
Node MHb-1



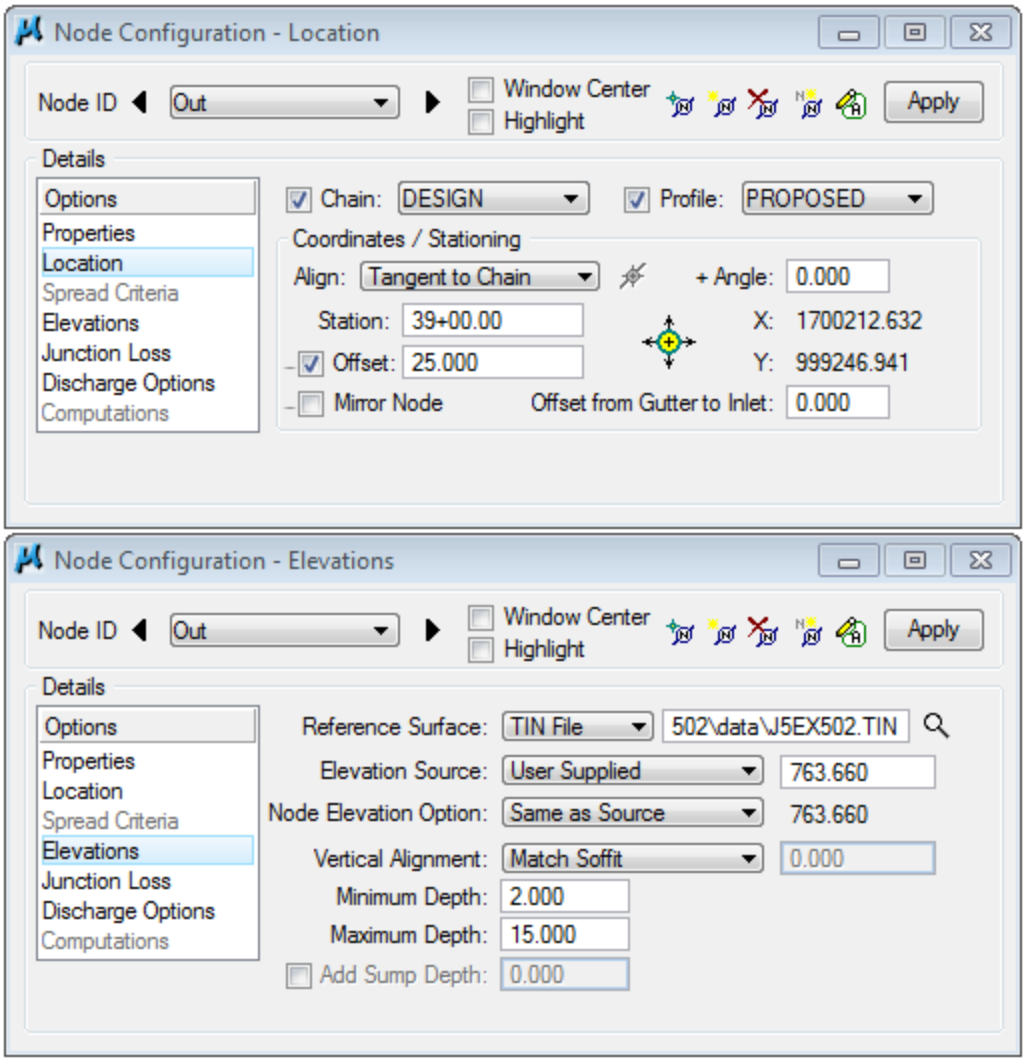
If we use the same Elevation Source and Node Elevation options that we used for the last manhole you'll notice that the elevation is higher than MHa-1. This won't necessarily create a problem since we can always adjust the elevation that the pipes enter MHb-1, but let's adjust the elevation for this node anyway. There are times when you will want to manually set the elevation of a node in order for your system to function properly or to achieve a desired outcome. Drainage offers a couple ways this can be done, we can choose User Supplied and input the elevation we want to set our node at or we can tie the elevation to our GPK referenced data and then adjust the final elevation by using the Constant Offset setting under Node Elevation Options.



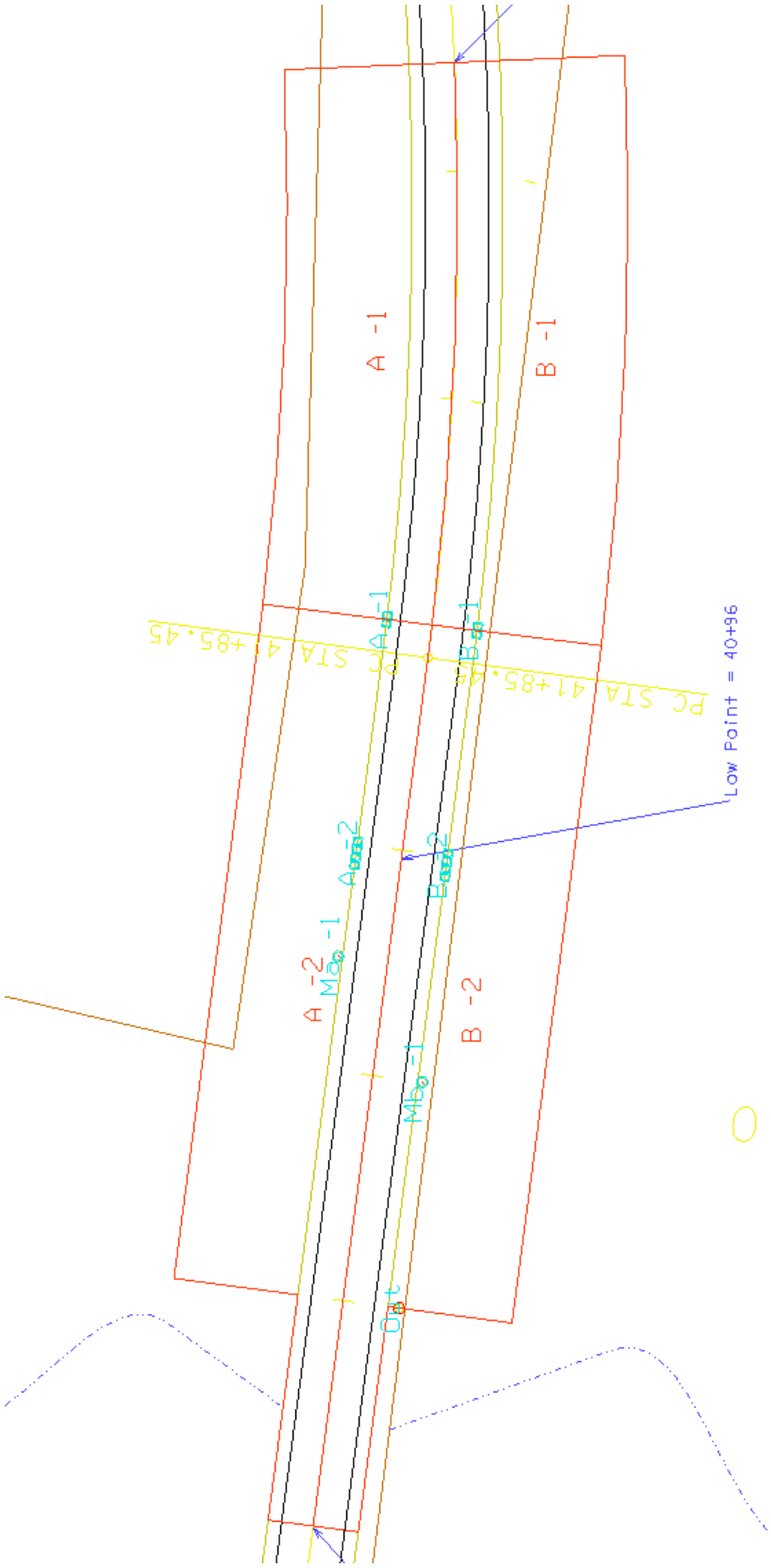
Node Out



We are going to assume a constant normal depth of flow in the channel or stream that we are draining our system into, so we will use the Fix Tailwater at option and use the Uniform Flow setting.



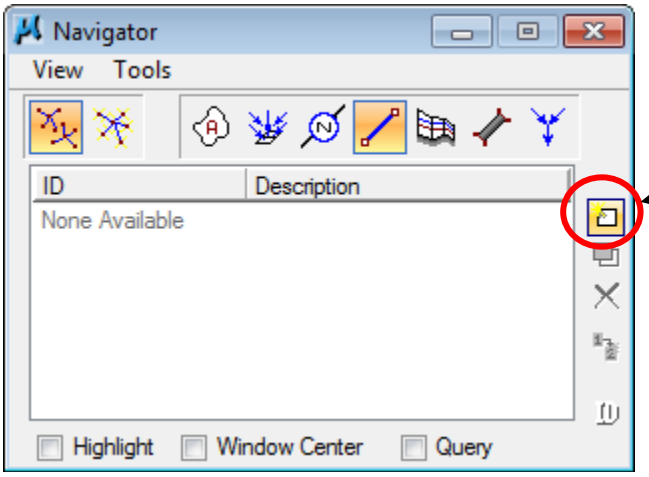
Upon completion you should have a drainage system resembling the screenshot on the next page.


A -1
B -1
PC STA 41+85.45
Low Point = 40+96
A -2
B -2
Out

5.6 Pipe Placement

Now we need to link our inlets and nodes together.

1. Using the Navigator, select the Link icon at the top and then select the Add Item button.



Select the **Add Item** Icon

You could also use the Component drop-down menu from Drainage and choose the Link >Add option to place your pipe. Another option is to launch the Drainage Main Tool Box and use the Link tool set.

We will be using the default naming convention for our pipes, so you can click OK when prompted to name SS -1 for your first link.

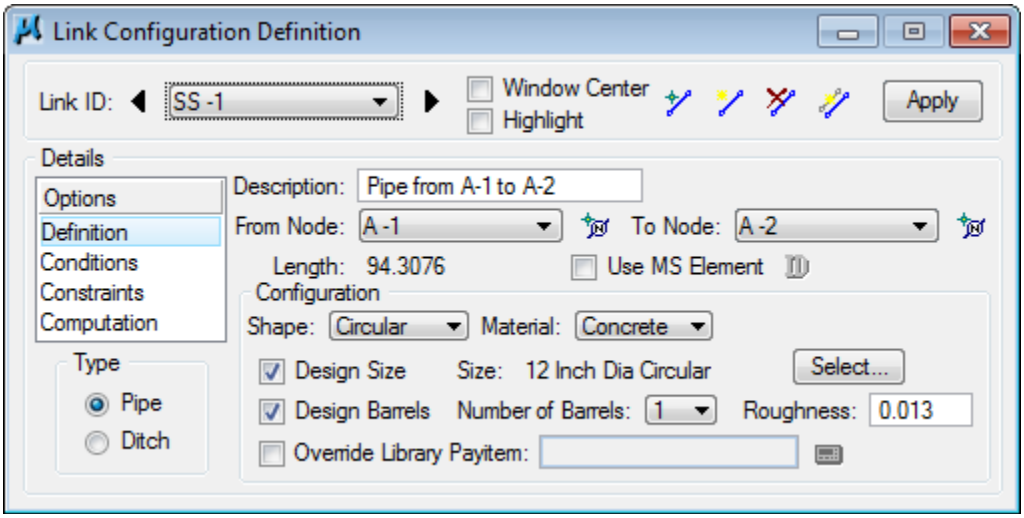
2. Now use the information outlined below to configure SS -1

Definition

Description: **Pipe from A-1 to A-2**

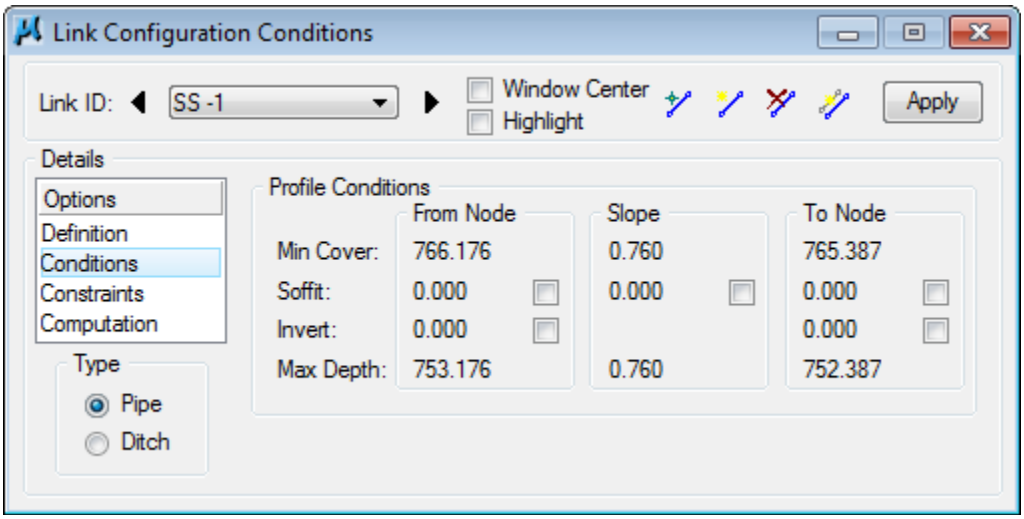
Use the drop-down lists to assign the From Node and To Node

Define the configuration:
Shape: **Circular**
Material: **Concrete**
Size: Select any pipe from the drainage library
Enable Design Size
Number of Barrels: **1**



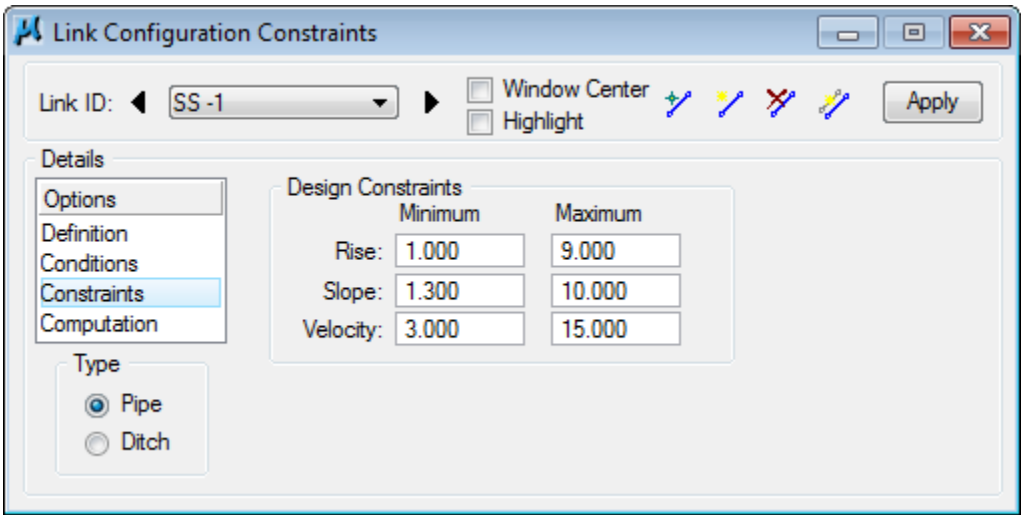
Conditions

The soffit and invert will not be populated until after the network has been created.

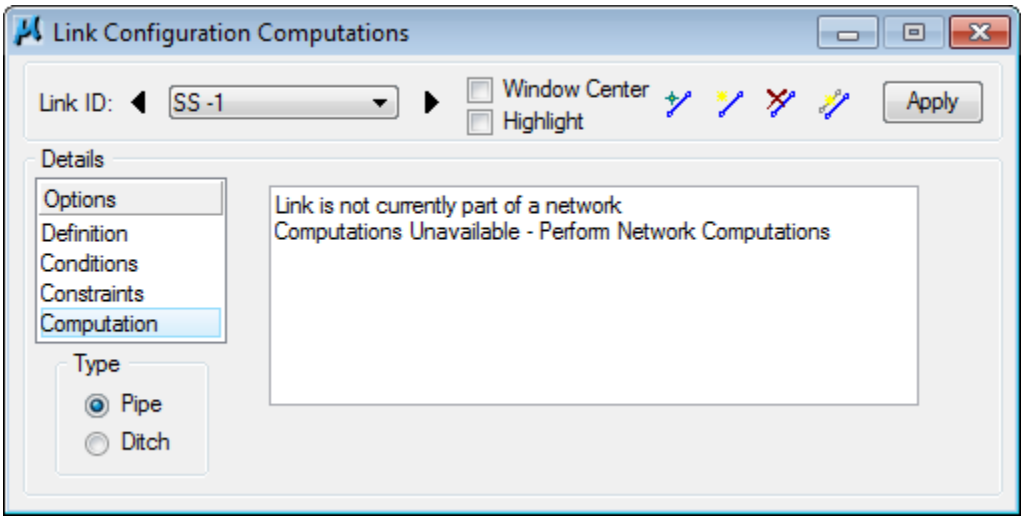


Constraints

Use the design constraints listed at the start of the exercise.



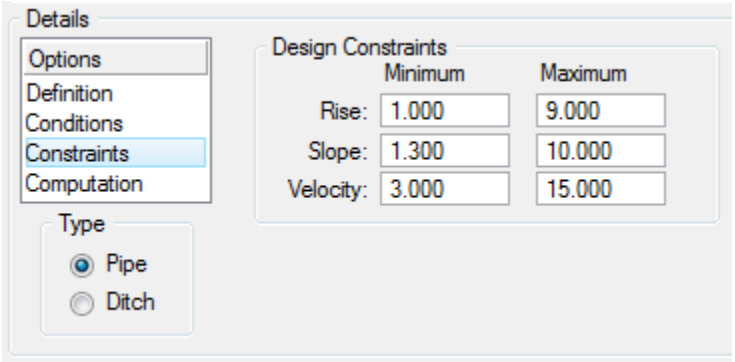
Computations
The network must be created before computations will be performed.



3. **Based on the supplied data add the remaining links.**

Pipe ID	From Node	To Node	Shape	Material
SS -1	A -1	A -2	Circular	Concrete
SS -2	A -2	MHa-1	Circular	Concrete
SS -3	MHa -1	MHb -1	Circular	Concrete
SS -4	MHb -1	Out	Circular	Concrete
SS -5	B -1	B -2	Circular	Concrete
SS -6	B -2	MHb -1	Circular	Concrete

The screenshot below should be used for all pipes when setting the Constraints



5.7 Network Creation

Now that we have our nodes and our links established, we need to build and design our network.

1. Using the Drainage menu bar go to Network and select the Add option

2. Name your network Network-1 and make sure the outlet node is designated as Out

Click OK

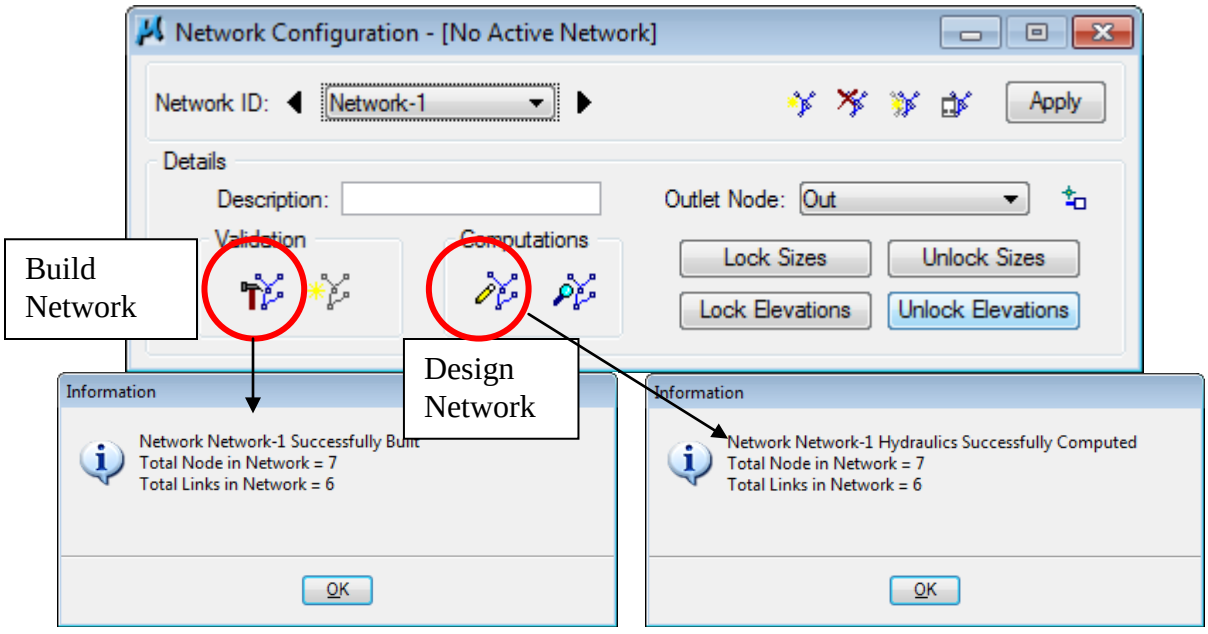
3. Select the Build Network option under Validation

Selecting the **Highlight Network** button will highlight your entire network to show you which components are recognized by GEOPAK as being part of your network

4. Select the Apply button to add the network to the drainage project

5. Select the Design Network option under Computations

The **Design Network** process accounts for any design options that were enabled during the component placement process (for example: we enabled the Design Size function under the Configuration options when we placed our pipes)



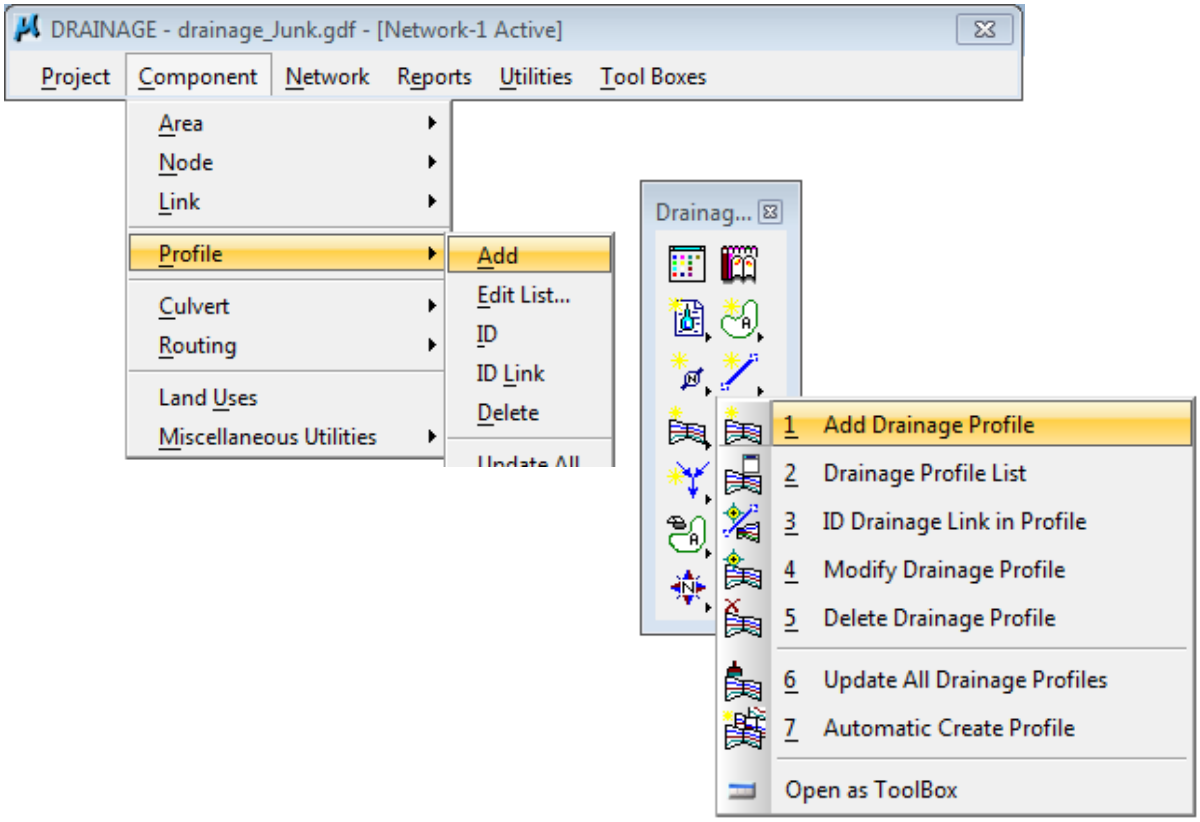
6. Now set the active network by going the Network menu on the Drainage menu bar and selecting the Active Network option then select Network-1 from the list of available networks

The banner should now display **Network-1 Active**

5.8 Profile/Reach Creation

Profiles are constructed in a path running in any direction (upstream or downstream) in a drainage Network and are utilized to visualize the network cross–section. A profile can also be utilized to construct a customized profile (including groundline, pipes, depth of cover, hydraulic gradeline, etc. according to what display options are checked during creation) along any path.

Just like most of our other components, we can use a number of different ways to add our profile. Adding a profile can be done by going to the Component drop-down menu on the Drainage menu bar or by using the Add tool from the Drainage Profile tool set.



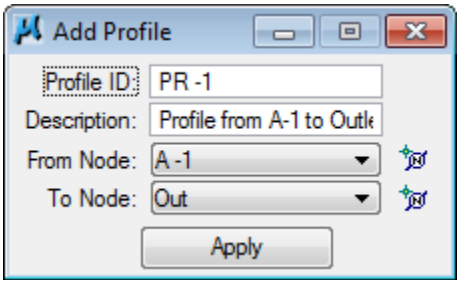
Let's use the **Navigator** to add our profile.

1. Select the Profile option along the top of the Navigator then choose the Add button on the right-hand side

2. Complete the Add Profile information as follows:

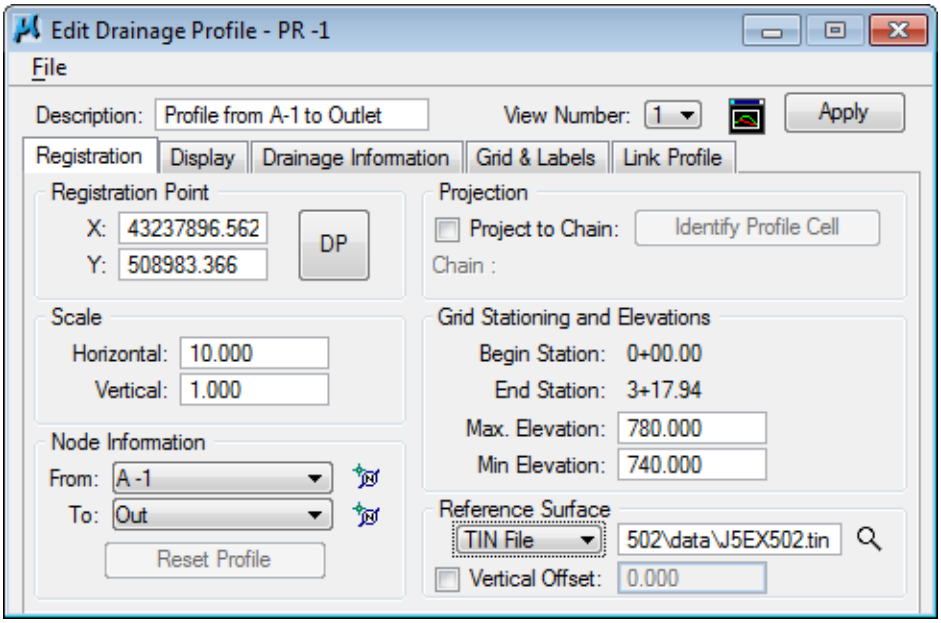
Profile ID: **PR -1**
Description: Profile from A-1 to Outlet
From Node: **A1** (Select from drop-down list or use ID option to select element)
To Node: **Out** (Select from drop-down list or use ID option to select element)

Click **Apply**



In the Registration tab for PR-1 use these settings:

Reference Surface: **TIN File**
Select **J5EX502.tin** (If left blank, individual Node elevations are utilized to construct the ground profile.)



3. Click the DP button in the Registration Point section and then data-point click a spot in your design where you want to place the profile

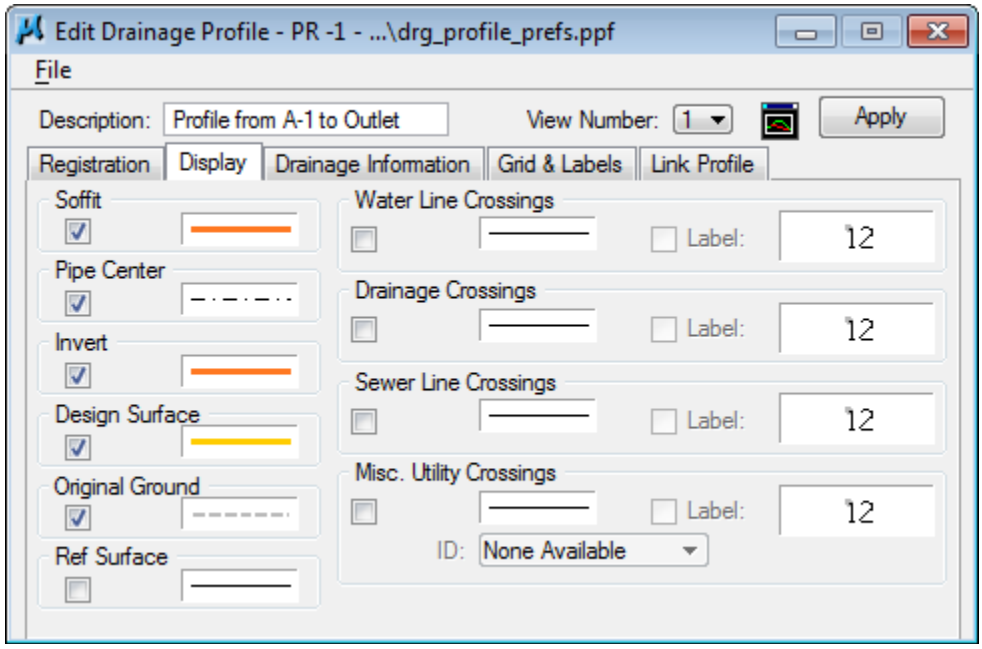
Now before we graphically generate our profile, we will want to set up some of the different options and parameters.

4. Choose the Display tab

At this point you could enable any available option and change the colors and levels as necessary by double-clicking on the symbology box for that item. However, for this example a preference file has been created and saved in our data directory as:

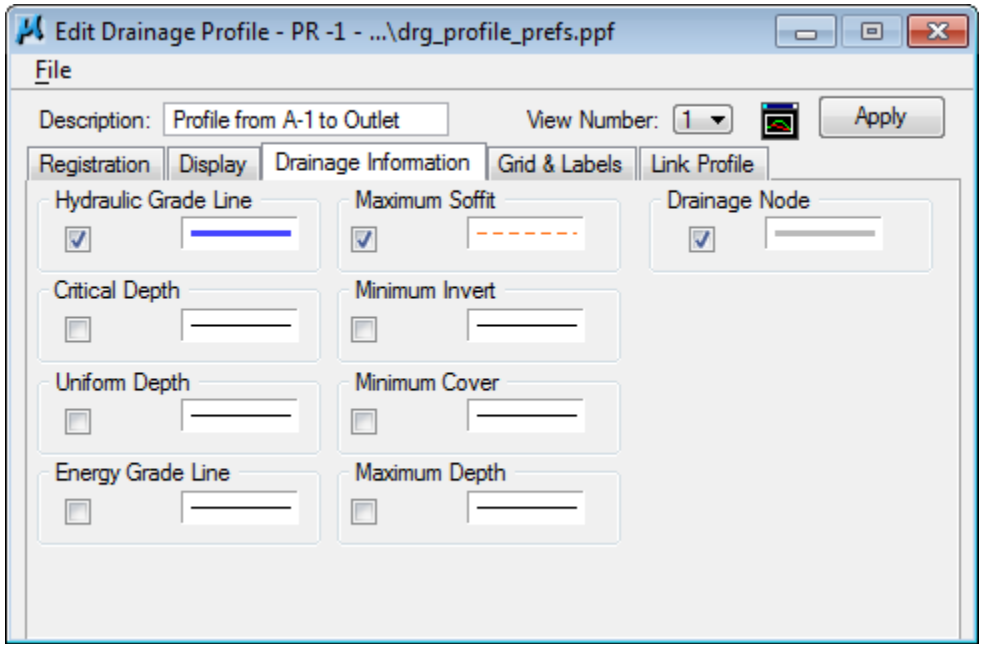
drg_profile_prefs.ppf

Use the File >Open option in the Profile dialog and choose this file to load the preferences for our profile



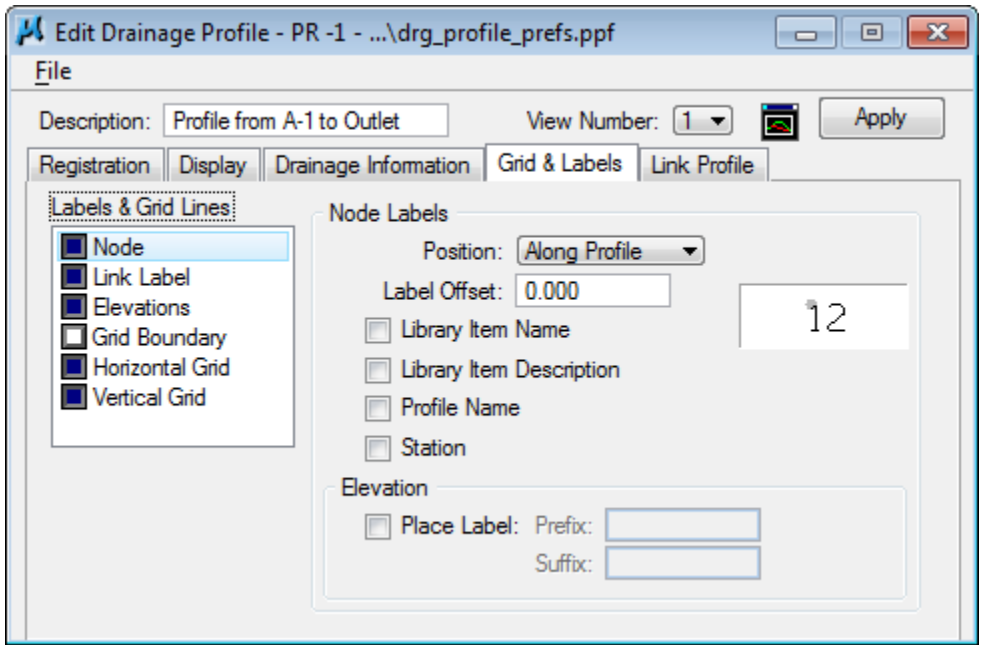
5. Choose the Drainage Information tab

This will allow you to check the items you wish to show on the profile. Loading the preference file in step 5 will check certain items and give them a color and level. You may still choose to show other items here.



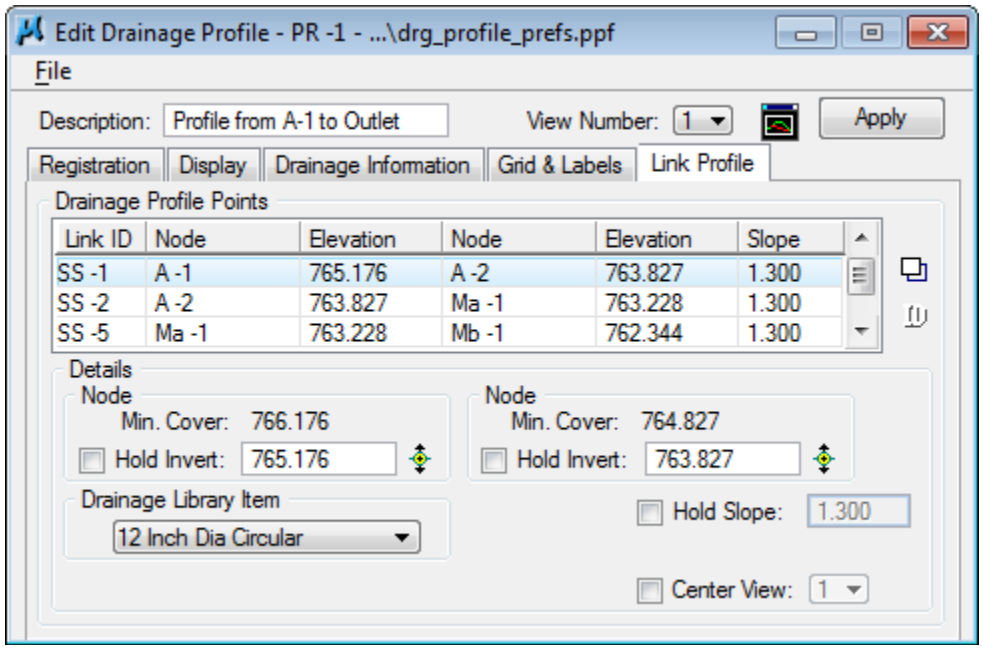
6. Choose the Grid & Labels tab

This tab allows the horizontal and vertical grids to be placed in the profile along with the appropriate text.



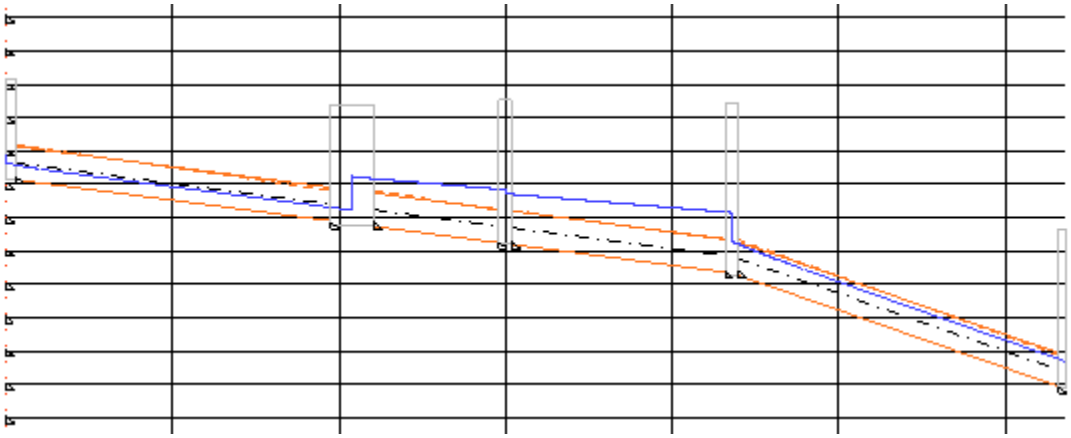
7. Choose the Link Profile tab

This tab displays the link configuration associated with a selected profile. You can gather information on elevations and even edit them in this tab. You can select elevations or key them in, and change the library item currently being used from one section of your network to another.



8. Finally select the Apply button to save the profile to drainage project.

Take a look at the generated profile.



Discuss options to affect the hydraulic jump.