SCDOT Specifications and Support Manual for GEOPAK Drainage

Table of Contents

Introduction

On April 17, 2000, the Hydraulic Engineering Section of the South Carolina Department of Transportation adopted GEOPAK Drainage as our primary storm sewer design package. Due to the many specifications that need to be set to run the program, the best plan was to write a manual to aid everyone using the software. It is not the purpose of this manual to provide you every piece of information needed to run the program. Additional resources are available through online help and training manuals.

Most of our other programs are much simpler to set up, so when misinformation is entered it can be easily found and corrected. GEOPAK has many settings that can cause the program to crash that you could spend several hours looking for that one incorrect specification. This manual was created to help avoid such errors and to set standards for naming conventions, levels, colors, etc. so our information would be easily transferable to the Roadway Designer. Never before has another office worked so closely with output from one of our programs. This is why it is critical that we are consistent with documentation so others can read our information.

This manual ***does not*** supersede the Requirements for Hydraulic Design Studies of the Department. **Careful** attention should be paid to those requirements and every effort made to adhere to them.

SCDOT still possesses other storm sewer design packages, and there may be a time when one of these methods is the best solution for a particular situation. The Hydraulic Designer will make that determination. If you must use another program, please notify the Roadway Designer immediately to inform them that you will be submitting red lined plans for their use. Some information like riprap pads and outfall improvements will still need to be red lined. We will not be required to forward any MicroStation files with notes.

GEOPAK Drainage, like all other hydraulic programs, gives you the best design and analysis based on the input. Engineering judgment must be used to evaluate the output that the program produces. You must check all data carefully and make determinations on the validity of the answers based on your knowledge of hydrology and hydraulics.

Do not be surprised to find that the program overrides some of your specifications. The program has the ability to change your Preferences and Constraints in order to design the system. The Preferences and Constraints are just guidelines to give the program a starting point to work with; therefore, it is imperative that you check all of your output for validity.

All specifications required to run the program will be added to this manual at the appropriate time. You will also be receiving updates for the storm sewer part as they are published.

NOTE TO CONSULTANTS: Consultants are to follow the standards set forth in this manual and the Requirements for Hydraulic Design Studies. The only procedural change that differs for consultants is discussed under Downloading & Organizing Files and Uploading Files. The consultants will obtain Road Design's files and send their data back to the Hydraulic Engineer, not the Roadway Designer, via electronic means as designated by SCDOT. All other guidelines are to be adhered to as stated.

Chapter 1: Data and Organization

1. Drives

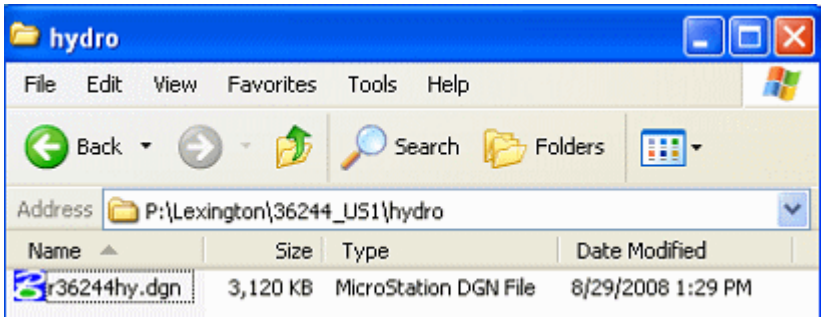
Network Locations and Workspaces

Preconstruction Data Folder Structure

P: \\nts\hq\precon

The hydraulic, roadway, geotechnical, and structural project files are consolidated by county on the network drive designated as the **"P"** drive. This drive serves as a central location for hydraulic, roadway, geotechnical, and structural CADD and data files that are project related and are sorted by county and Project ID number. New projects as well as current projects are stored at this location. \\nts\hq\precon

Example of a hydro folder located in Lexington County on the "P" drive:

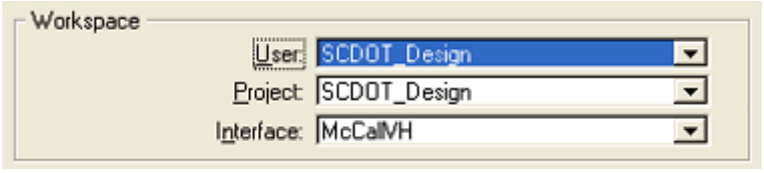


MicroStation and Geopak Workspaces

MicroStation and Geopak uses a custom workspace that is available to every CADD user. The workspace *automatically* controls access within MicroStation to the most current cell libraries, reference border sheets, drainage files, etc.

Network path
\\nts\hq\CaddStandards\SCDOT-Bentley\Standards\SCDOT_Design\

When entering MicroStation, the User and Project should be set to SCDOT_Design with the Interface set to the NTS user name:



Drainage Library (DLB)

The current version is: SCDOT_english_2018_Alt.dlb

\\nts\hq\CaddStandards\SCDOT-Bentley\Standards\SCDOT_Design\Geofiles\Drainage

Cell Library (CEL)

\\nts\hq\CaddStandards\SCDOT-Bentley\Standards\SCDOT_Design\MSfiles\cell\RoadV8.cel

GEOPAK (DDB)

\\nts\hq\CaddStandards\SCDOT-Bentley\Standards\SCDOT_Design\Geofiles\Database\scdot_engV8.ddb

2. Files

File Type:	GPK
Type	Binary
Acronym	GPK = **G**eo**P**a**K**
Use	Stationing, Profiling and Reporting

Through GEOPAK Drainage we will read the *.GPK file that the Roadway Designer produces to obtain the pgl and chain files for placing drainage nodes. The Roadway Designer will provide the names for the centerline profile and chain, all side road profiles and chains, and any curb grade profiles. From these files GEOPAK Drainage will automatically read curb grades and identify the centerline stationing of mainline and side roads. The *.GPK file is extremely valuable. The *.GPK file will usually be named job###.gpk. The ### will be the last 3 digits of the Project ID number.

File Type:	PP.DGN
Type	Binary
Acronym	DGN = **D**esi**GN**
Use	Plan View

The pp.dgn file is the plan view of the project. It is shown exactly as surveyed. The pp.dgn file will usually be named r#####pp.dgn. The ##### is the Project ID number. The .dgn is the designation for all MicroStation design files.

File Type:	PF.DGN
Type	Binary
Acronym	DGN = **D**esi**GN**
Use	Plan & Profile View

The pf.dgn file is the plan and profile view of the project as it appears on the plan sheets. There will be more than one of these files for every project unless the project is very small. You will not need this file for GEOPAK Drainage. You may need to open these files to get tie equalities or look at some of Roadway Designer's notes.
The pf.dgn file can be named many different things. Some squads name them by the Project ID number, i.e. 12345pf.dgn, 1234apf.dgn, 1234bpf.dgn… Some squads name them by the road, i.e. sc100pf.dgn, sc100apf.dgn… The Roadway Designer will provide the names for these files.

File Type:	HY.DGN
Type	Binary
Acronym	DGN = **D**esi**GN**
Use	MicroStation file for Geopak Drainage design

This is the MicroStation file that you will do all of your design in when running GEOPAK Drainage. You will need to create this file. It will be named using the same conventions as the pp.dgn file. Basically we will reference Roadway Designer's pp.dgn file into our created hy.dgn file. This will allow us to view the road, but our design will be in another design file. If the Roadway Designer changes the pp.dgn file, it will not affect our design. This will be discussed in the Downloading & Organizing Files section.

File Type:	*.NEW
Type	Binary
Acronym	NEW = **NEW**
Use	Survey shots

The .new file contains a list of all surveyed points. In the past, the Hydraulic Engineer used this file to determine pipe elevations from plans. The .new is usually named a series of numbers with part of those numbers being the Project ID number. Any additional surveys submitted for the same project will follow the same naming convention but will add an A, B, C, etc. to the file name to be sent to the Roadway Designer. You will not need this file for GEOPAK Drainage.

File Type:	*.CEL
Type	Binary
Acronym	CEL = **CELL**
Use	Used for placing Node Cells

The Cell Library is used for placing Drainage Nodes into the HY.DGN file.

File Type:	*.DDB
Type	Binary
Acronym	DDB = **D**esign **D**ata**B**ase
Use	Used for computing quantities

The Design & Computation Manager database file can be used to compute drainage quantities.

File Type:	*.GDF
Type	Binary
Acronym	GDF = **G**eopak **D**rainage **F**ile
Use	Stores the drainage components

The .gdf is short for **G**eopak **D**rainage **F**ile. We will produce this file when we create a design in GEOPAK Drainage. It contains all of our preferences, nodes, links, networks, profiles and output. This file references our Drainage Library, cell library, and the Roadway Designer's GPK File. NOTE: The GDF file is not a MicroStation file, but MicroStation uses this file to plot our basins and pipes. This is a file that you will upload for the Roadway Designer to aid them in labeling the plans. Name your .gdf files according to the naming convention discussed in the New Project section. The naming convention must be followed for consistency. You will only have one .gdf file per project. Even if you have multiple roads on one set of plans and you need to design storm sewers for all of them, you will still have only one .gdf file. Normal in-house projects have one designer assigned to that project, but consultant and large in-house rush projects will have several people working on them. If multiple people work on a project at the same time, then there will be multiple .gdf files. All .gdf files will need to be merged before they are sent to the Roadway Designer for labeling.

File Type:	*.DPF
Type	Binary
Acronym	DPF = **D**rainage **P**reference **F**ile
Use	Loading Project Preferences

The .DPF file references all necessary standard files needed when setting up a .GDF. These files are named according to the rainfall station that the file references. Once settings are loaded, they will be saved in the *.GDF.

File Type:	*.DRF
Type	Binary
Acronym	DRF = **D**rainage **R**eport **F**ormat
Use	Standard Report format for the output CSV files

The DRF file is the standard Drainage Report Format file created in-house that contains the information format that is required for our standard reports. When you run these files they will automatically generate the CSV files that contain the output data that you will cut and paste into our standard output files. The .drf files that you will need to run have been preset and are on the server. This will be discussed in the Reports section. These files are areastandard.drf, nodestandard.drf, linkstandard.drf, nodestandardroad.drf, linkstandardroad.drf, which need to be run for each storm sewer system, and nodesum.drf and linksum.drf, which are not storm sewer specific. The last two files mentioned are the summation of all the nodes and links, respectively, for the entire project.

File Type:	*.CSV
Type	ASCII
Acronym	CSV = **C**omma **S**eparated **V**alues
Use	EXCEL™ file

The CSV file is the output file that you will make through GEOPAK Drainage that is readable inside of EXCEL™. It will only contain the information that the DRF file requested to be obtained. There will be multiple files for every network (storm sewer system); three for our records and two to send to the Roadway Designer. This will be discussed in the Output section.

File Type:	*.DLB
Type	Binary
Acronym	DLB = **D**rainage **L**i**B**rary File
Use	Geopak Drainage Library file

The DLB File is the **D**rainage **L**ibrary **F**ile. You will download a copy of this file into your project directory. This file is discussed in the Drainage Library section.

This file will not email as is. You must place it into a zip file first. Right-click the file and select Send To->Compressed (zipped) folder

File Type:	*.TIN
Type	Binary
Acronym	TIN = **T**riangulated **I**rregular **N**etwork
Use	Stores 3d surfaces

The TIN File is the **T**riangulated **I**rregular **N**etwork file and is used to represent the existing or proposed 3D surface.

3. Exchange of Information

Per Instructional Bulletin No. 2005-8, you should receive the following information from the Roadway Designer to begin a design. The submittals from the Roadway Designer should be checked to ensure that all the information was received by the Hydraulic Engineer. The Hydraulic Engineer should also check to ensure that submittals to the Roadway Designer are complete.

Information to be Obtained from the Roadway Designer for the Hydraulic Engineer

Hard Copy

1. Cross Sections to scale on half size sheets
   a. Mainline
   b. Side roads
   c. Outfall ditches
2. Plan Sheets to scale on half size sheets
   a. Centerline final grades for mainline
   b. Final grades for side roads
   c. All outfall ditch surveys
   d. Limits of construction line
   e. All existing survey pipe recommendations

Electronic Copy

1. Project **.GPK** file
2. Project **PP.DGN** file
3. Project **PF.DGN** file
4. Project **DX.DGN** & **FX.DGN** files
5. Project **.NEW** file
6. Excel file containing survey data, including control points
7. Centerline and top of curb pgl's.
8. Project curb grades

Information to be Provided for the Roadway Designer from the Hydraulic Engineer

Hard Copy

1. Plan sheets stating which existing pipes are to be abandoned or retained.

Electronic Copy

1. Project **.GDF** file
2. Project **.DLB** Drainage Library file (if modified for project exceptions)
3. Project **HY.DGN** file
4. List of file locations and file names of requested information emailed to the Roadway Designer
5. Proposed ditch cross sections
6. Alternate Pipe spreadsheet
7. Erosion Control Data Sheet
8. Additional erosion control BMP recommendations as appropriate

Explanation of common plan sheet items -- On the plan sheets, the limits of construction line is the cut/fill line. Also in the plan sheets, you will find the survey recommendations to abandon or retain a pipe.

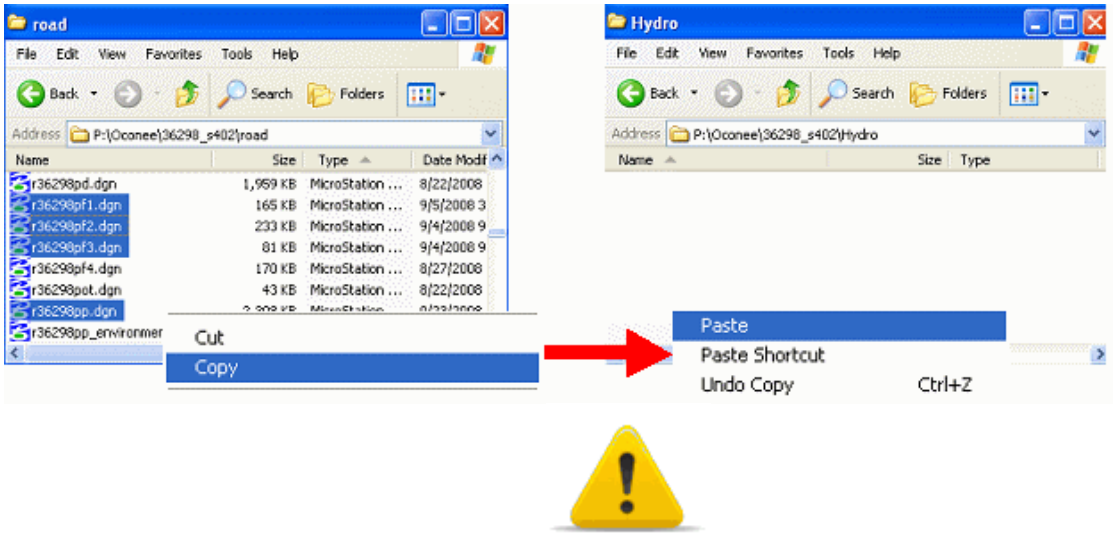
4. Downloading & Organizing

The Roadway Designer will email you a **Hydraulic Information spreadsheet** containing a list of CADD and survey files that you will need to copy from the project road folder to the hydro folder located on the "P" drive.

A	B	C	D	E	F
	ROADWAY DESIGN DATA FOR HYDRAULIC DESIGN				
	DATE:	**2/6/2015**			
DESIGN GROUP:	RPG-1/R2				
PROJECT NO:	39168				
COUNTY:	Jasper				
ROAD/ROUTE NO.:	US 17				
PROJECT DESCRIPTION:	From: Georgia State Line To: SC 315				
ADDITIONAL NOTES:	Beginning of project ties to a bridge Georgia is constructing and the end of the project ties to project id 0039603 (US 17 at grade RR crossing) that will be constructed first.				
		FILE INFORMATION			
	SERVER	**DESIGN GROUP**		**COUNTY**	**PROJECT NO.**
FILES LOCATED:	nts\hq\precon	RPG-1/R2		Jasper	39168_US17
'.GPK' FILE(S):	job811.GPK			12811.TIN	
'PP' FILE(S):	r811app.dgn				
'PF' FILE(S)	r811pf1.dgn	ofd1pf.dgn		ofd5pf.dgn	
	r811pf2.dgn	ofd2pf.dgn			
	r811pf3.dgn	ofd3pf.dgn			
	r811sc315pf.dgn	ofd4pf.dgn			
'.NEW' FILE(S):	12811A.NEW				
DX/FX' FILE(S):	r811adx.dgn	r811cfx.dgn			
	r811afx.dgn	sc315dx.dgn		typical-324sections.dgn	
	r811bdx.dgn	sc315fx.dgn			
	r811bfx.dgn	ofd1 To ofd5dx.dgn			
	r811cdx.dgn	ofd1 To ofd5fx.dgn			

CHAIN NAME	ROPOSED PROFI	DESCRIPTION
US17A1	US17A1FP	
US17A2		
2NB24	NBFP	
2SB24	SBFP	
SBTRAN	US17A1FP	
NBTRAN	US17A1FP	
SBTRAN2	SBTRAN2	
NBTRAN2	NBTRAN2	
SC315	SC315FP	
OFD1	OFD1 & OFD1FL	
OFD2	OFD2 &OFD2FLOW	

Navigate to the project directory on the "P" drive under the **road** subfolder. Highlight the files listed in the index file by using the ctrl + click method to select multiple files at one time. Right click and select copy. Navigate to the **hydro** subfolder under the same project directory and right click and select paste.

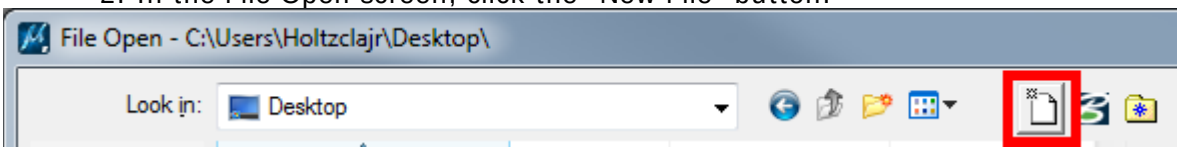


Be certain to **copy**, not **cut**, the files from the road folder into the hydro folder. Files should remain intact in the road folder at all times.

For consultants, the Roadway Designer will provide the spreadsheet and deliverables electronically to the consultant.

5. Hydraulic Engineering Files

- You will need to download the english.DLB Drainage Library file from \\nts\hq\CaddStandards\SCDOT-Bentley\Standards\SCDOT_Design\Geofiles\Drainage into the project directory on the network.
- You will need to create the hydrology file *#####hy.dgn (refer to pp.dgn for symbol significance). This can be done as follows:
    1. Open Microstation.
    2. In the File Open screen, click the "New File" button.



    3. Enter the name of the new hydrology file in the File Name box at the bottom of the window.
    4. Check the Seed box below File Name. It should be directed to \\nts\hq\CaddStandards\SCDOT-Bentley\STANDARDS\SCDOT_Design\MSfiles\seed\seed2d.dgn. If not, enter that address in the box or browse to that directory and select seed2d.dgn.
    5. Press Save to return to the File Open menu, then select the new hydrology file and press Open.
- Next, you need to reference in the pp.dgn file as follows:
    1. Go to **File>Reference**.

2. Go to **Tools>Attach…** Choose the pp.dgn file in your project directory. When it asks for a logical name just enter a 1.
3. In the Reference window beside the pp.dgn that you just entered, put a √ under the **Snap** column. This will allow you to snap to the reference file, but not delete anything.

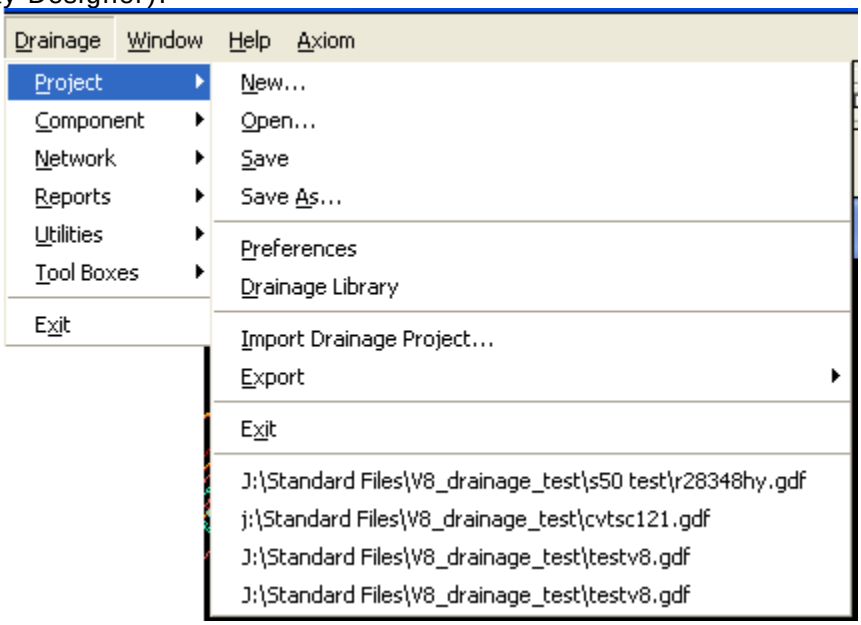
It should appear as if you are looking at the plan view of the project. The greatest difference now is that you cannot manipulate anything currently on your screen. It is purely for viewing. Now the hy.dgn file is ready to use through GEOPAK Drainage.

Chapter 2: Initial Settings

1. Start a New Project

To start a new Drainage Project:

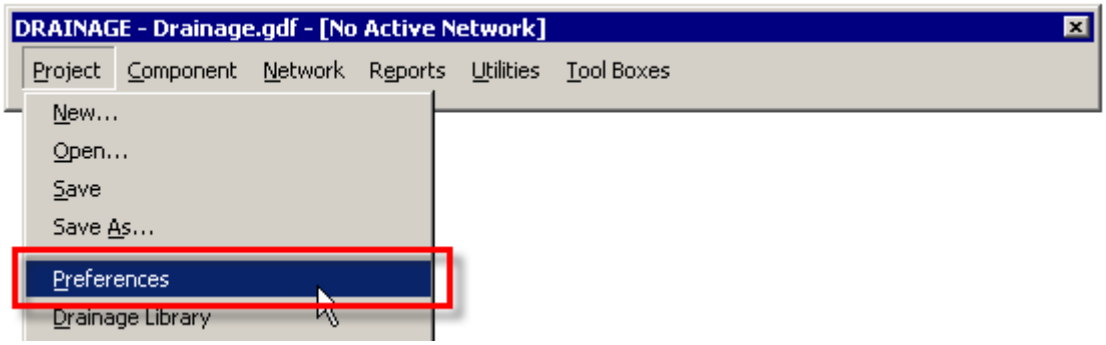
1. Open your hy.dgn MicroStation file.
2. From MicroStation, select Applications>GEOPAK DRAINAGE>Drainage (if GEOPAK is not activated select Applications>GEOPAK>Activate GEOPAK).
3. You should now have a new menu item by Applications titled Drainage.
4. Selecting Drainage>Project>New… Name the file after the road being improved, i.e. SC100.GDF (this is the GDF file that will contain all of your work and be sent to Roadway Designer).



GEOPAK Drainage always starts in an untitled project; it does not remember or automatically reopen a GDF that was previously worked in. The .GDF file must be reopened manually each time you open the hy.dgn. Select Project > Open each time you start to edit or continue working on a project.

2. Preferences

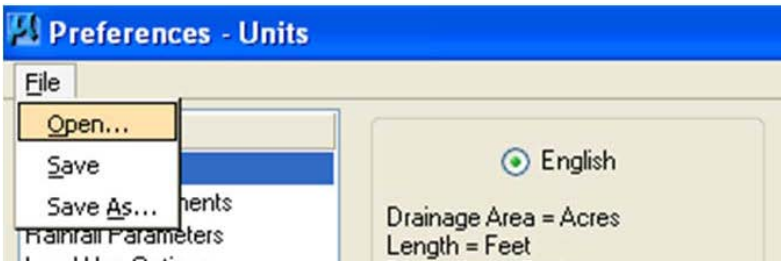
Set the Preferences as necessary at any time during project design or analysis, from the main menu bar select *Project* > *Preferences:*



To establish the Preferences of a new project, navigate in Windows Explorer to

\\nts\hq\precon-general\Hydraulics\Standard Files_GEOPAK\DPF Files

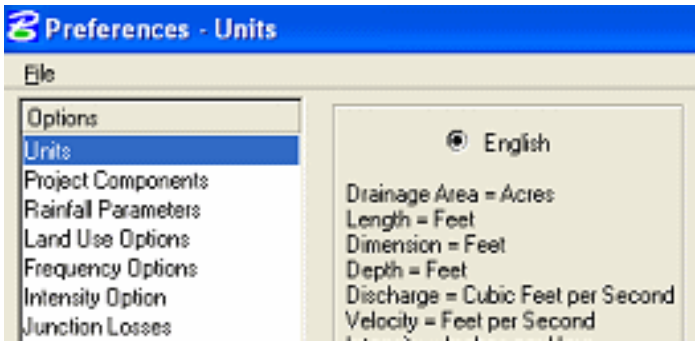
Select the appropriate *.DPF corresponding to your project and copy and paste it to your working directory. In Geopak Drainage, go to Preferences, File>Open and open the copied .DPF.



This will load the most up to date versions of all SCDOT preference files and save them to the created GDF when prompted.

3. Units

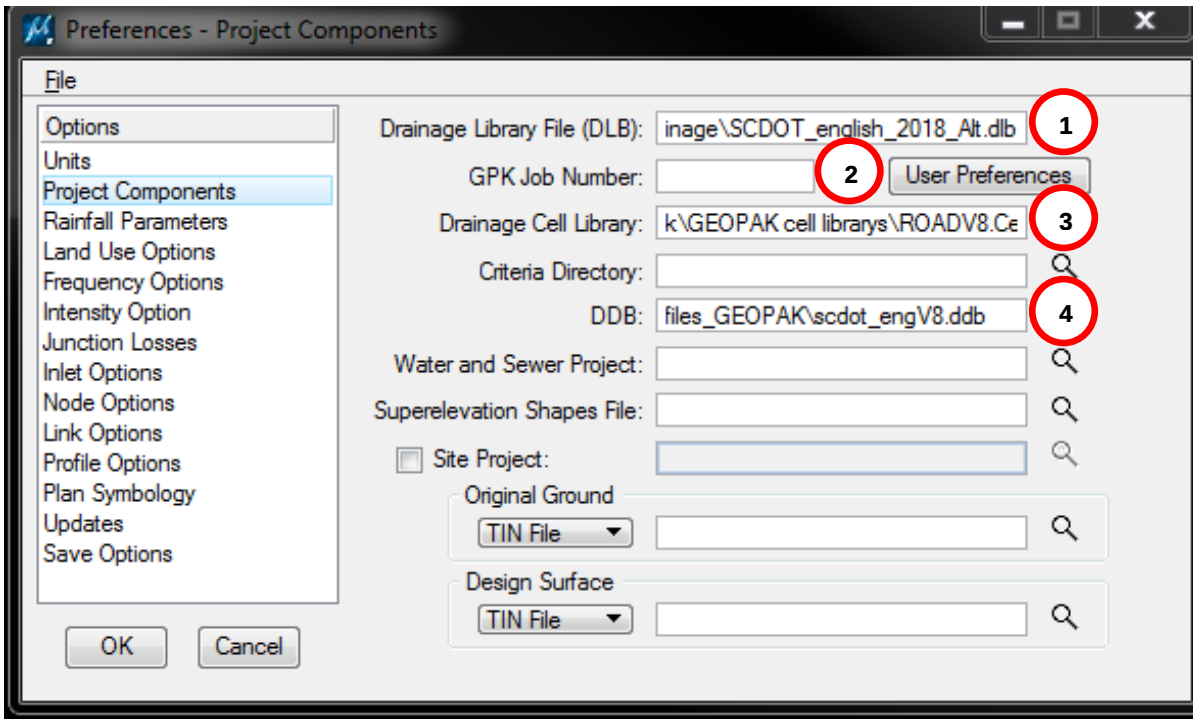
Set the Project Units to English:



4. Project Components

Set the Project Components:

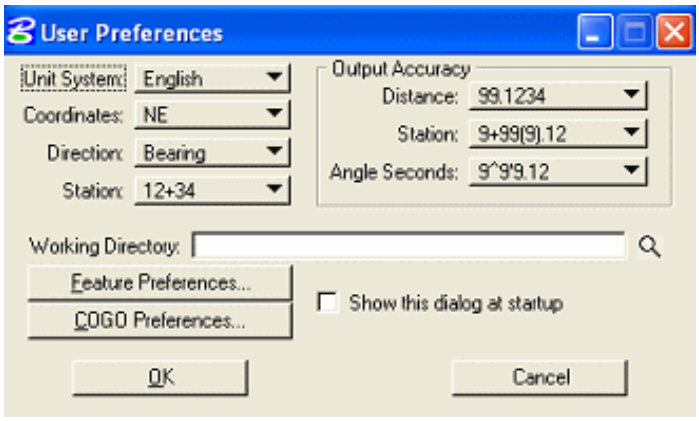
The DLB, CEL, and DDB can all be loaded from the *.DPF. The .GPK should be obtained from the Roadway group and placed in the same folder as the hy.dgn and .GDF files.



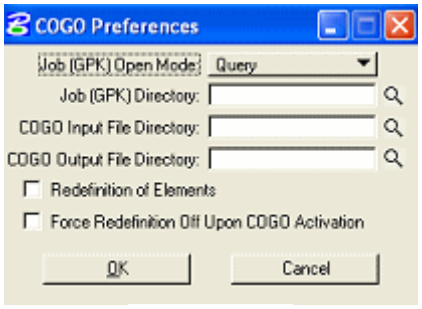
1	**DLB**	Click on the magnifying glass button and select the DLB file which was copied to the hydro project directory.
2	**GPK**	Click the magnifying glass button and select the GPK Job Number. You need to select your project directory and choose the file job###.gpk; there should only be one .gpk file. When you choose OK only the 3-digit number that was in the file name appears.
3	**CEL**	Click on the magnifying glass button and select the cell file located in the workspace folder. Select roadv8.cel.
4	**DDB**	Click on the magnifying glass button and select the DDB file located in the workspace location. Select the scdot_engV8.ddb.

Leave all the other file locations blank.

Under the Road Preferences, be sure that the box for Working Directory is empty. This will ensure that your GEOPAK iterations write to the current working project directory – the hydro folder.



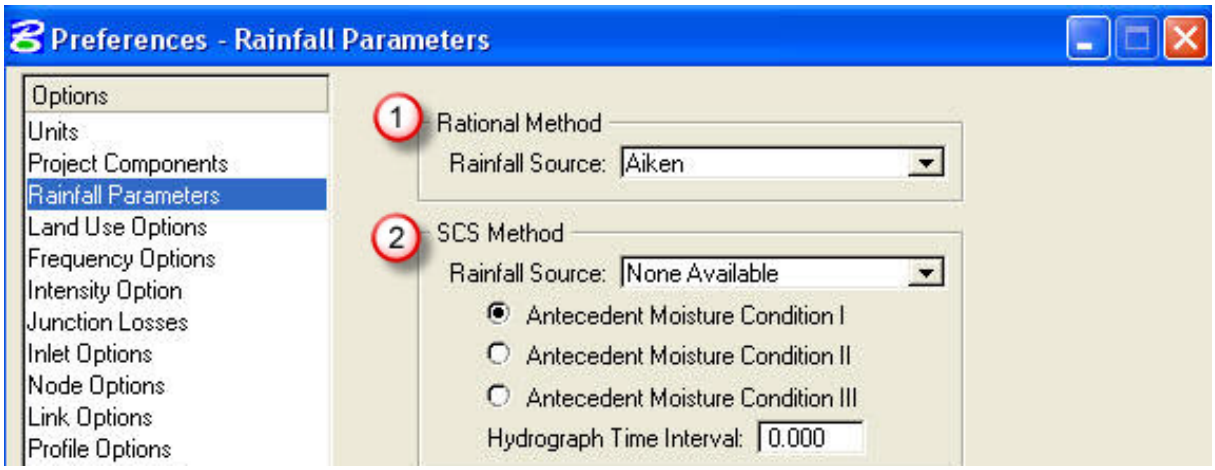
Press the COGO Preferences button and ensure that all three directories are empty also.



If the Working Directory box is not empty, you risk overwriting the GPK in the roadway folder with the new hydrology GPK.

5. Rainfall Parameters

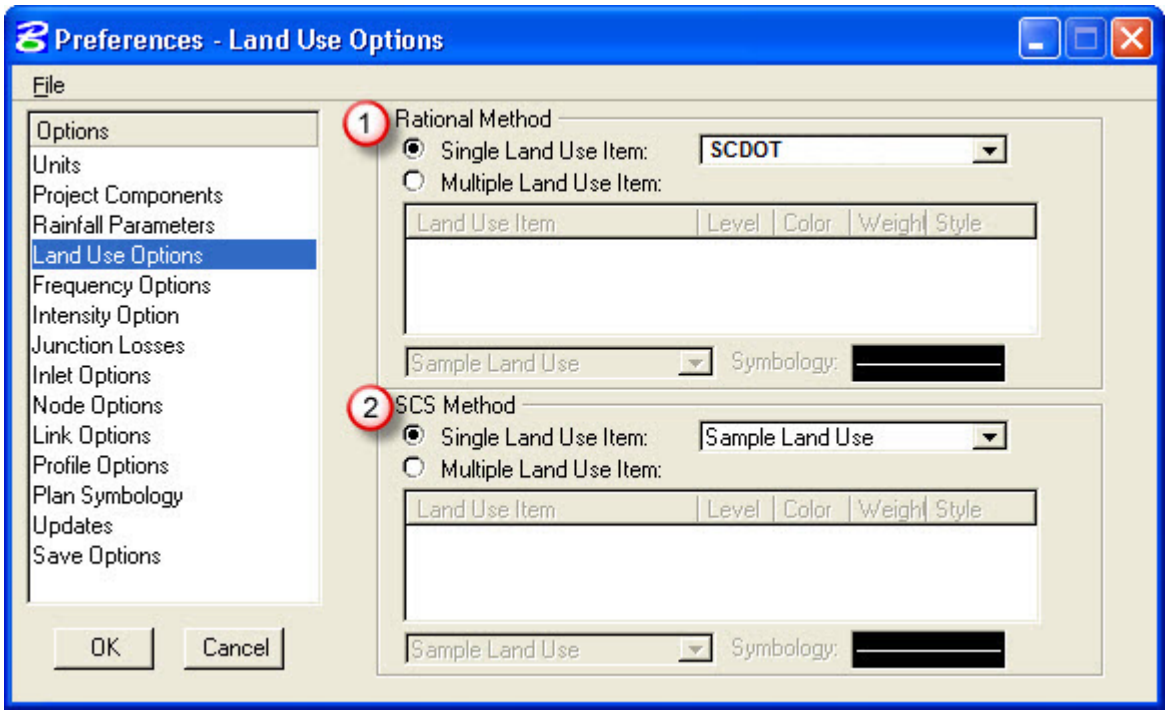
Set the Rainfall Parameters:



1	Rational	Choose the down arrow under Rational Method and a list of counties from South Carolina appears from the Drainage Library DLB file (choose the county that is closest to your site).
2	SCS	Ignore the SCS Method for now.

6. Land Use Options

Set the Land Use Options:



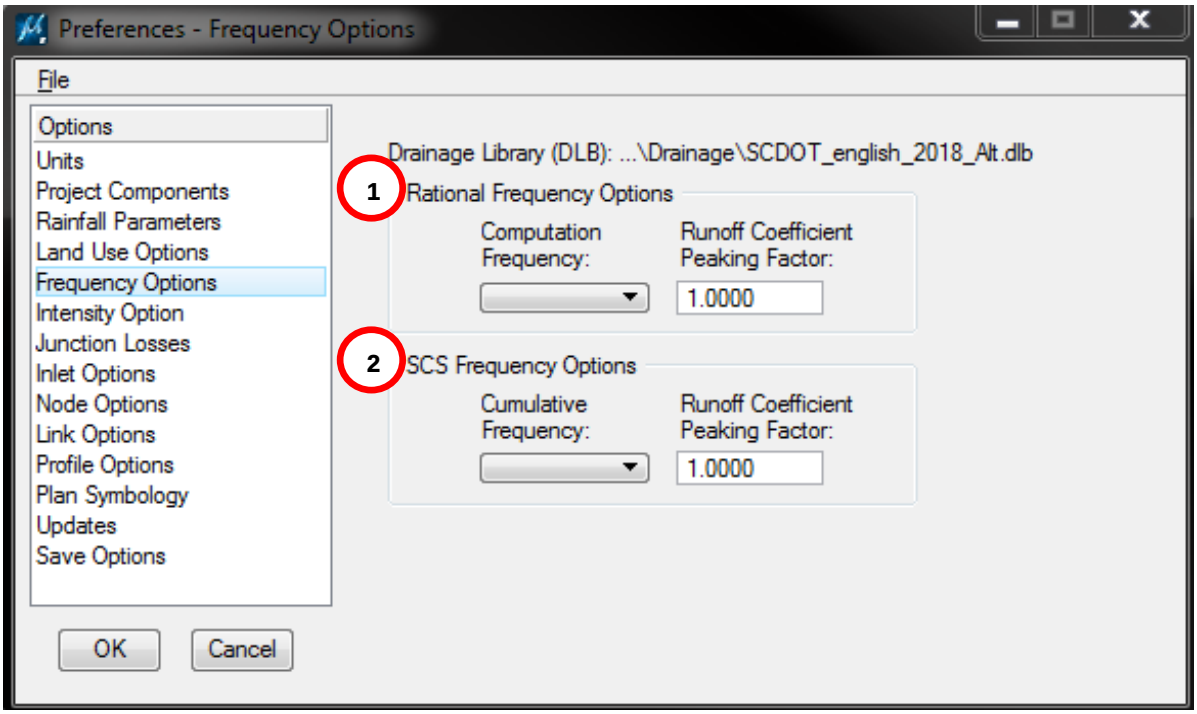
1	Rational	The land uses are the "C" values associated with the different land uses and slopes throughout the state. If you have a design exception for a project, you will need to copy the Drainage Library (DLB) to your hydro project directory. You will then be able to input specific "C" values into the Drainage Library. Be sure to include this file in the Electronic Files to the Roadway Designer if you make modifications.
2	SCS	Ignore the SCS Method for now.

For details of the SCDOT Land Use values, see Chapter 2, Section 19 and Appendix A of this manual.

For instructions on using Land Use see Chapter 4, Section 2 & 3 of this manual.

7. Frequency Options

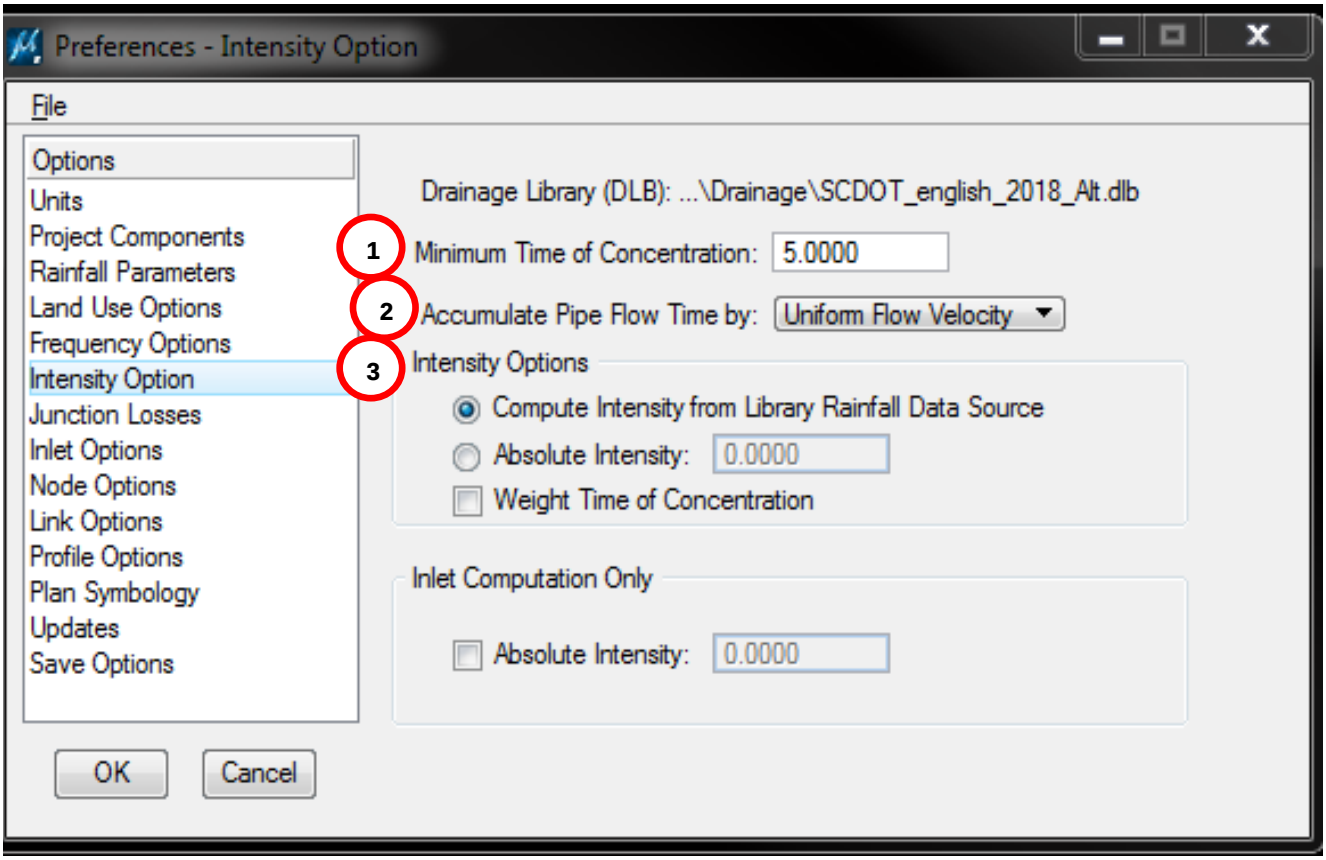
Set the Frequency Options:



1	Rational	Our basic storm sewer design is for a 10-year storm, but if cross lines are involved you will need to evaluate the storm sewer for a 25 or 50 year storm event. The Drainage Library has the values listed in it for the 2, 10, 25, 50 and 100-year storms. You can only run one storm at a time, so initially choose the 10-year storm under Rational Frequency Options and input 1.0 as the peaking factor. If you need to run the other storms, simply choose that storm and input the correct peaking factor from the requirements for Hydraulic Studies.
2	SCS	Ignore the SCS Method for now.

8. Intensity Option

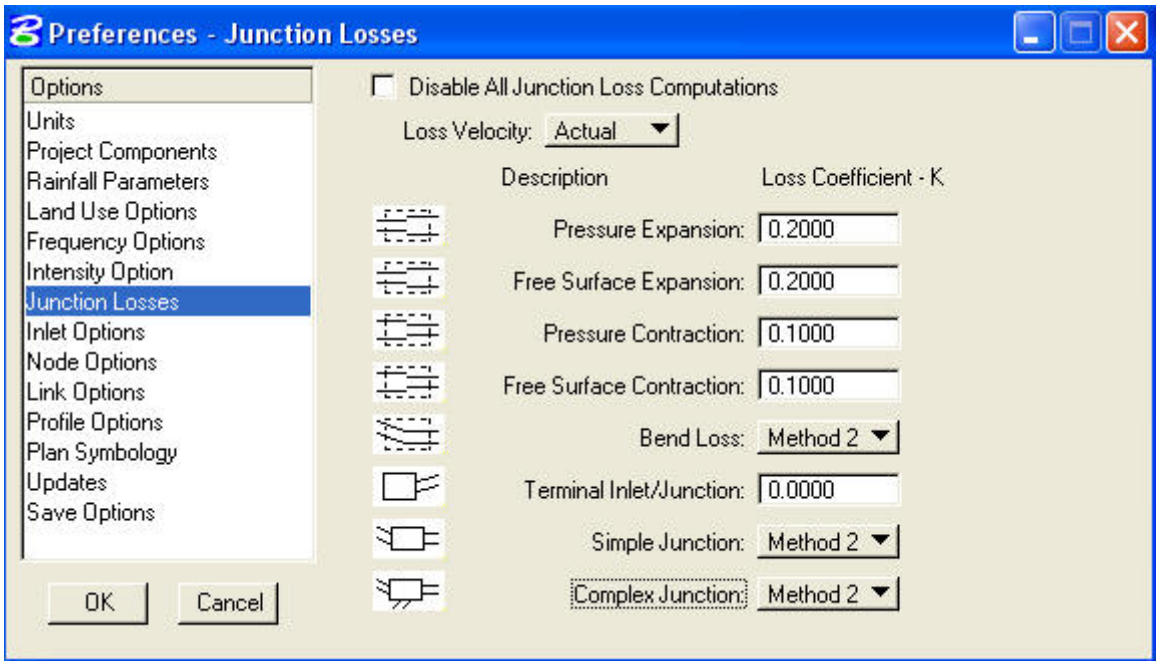
Set the Intensity Option:



1	Min Tc	Set the "Minimum Time of Concentration" to 5.0 minutes.
2	Accumulate Time by:	Under "Accumulate Pipe Flow Time by" select "Uniform Flow Velocity".
3	Intensity Options:	Always toggle **ON** the option to "Compute Intensity from Library Rainfall Data Source".

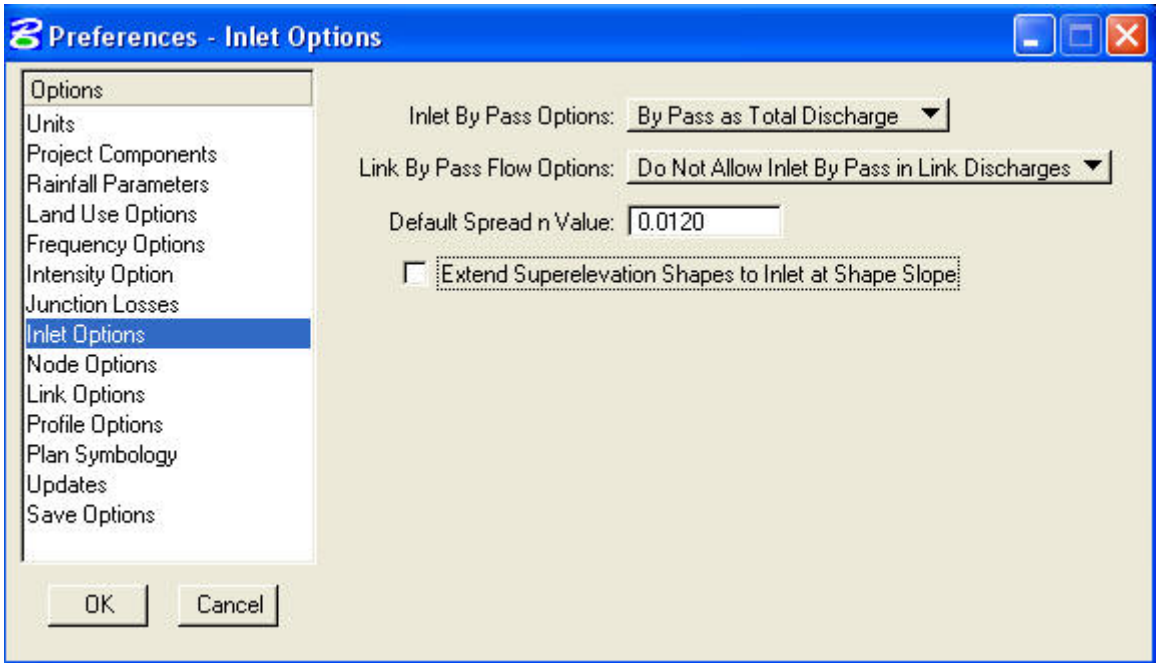
9. Junction Losses

Set the Junction Losses (use the values shown below for the Junction Losses):



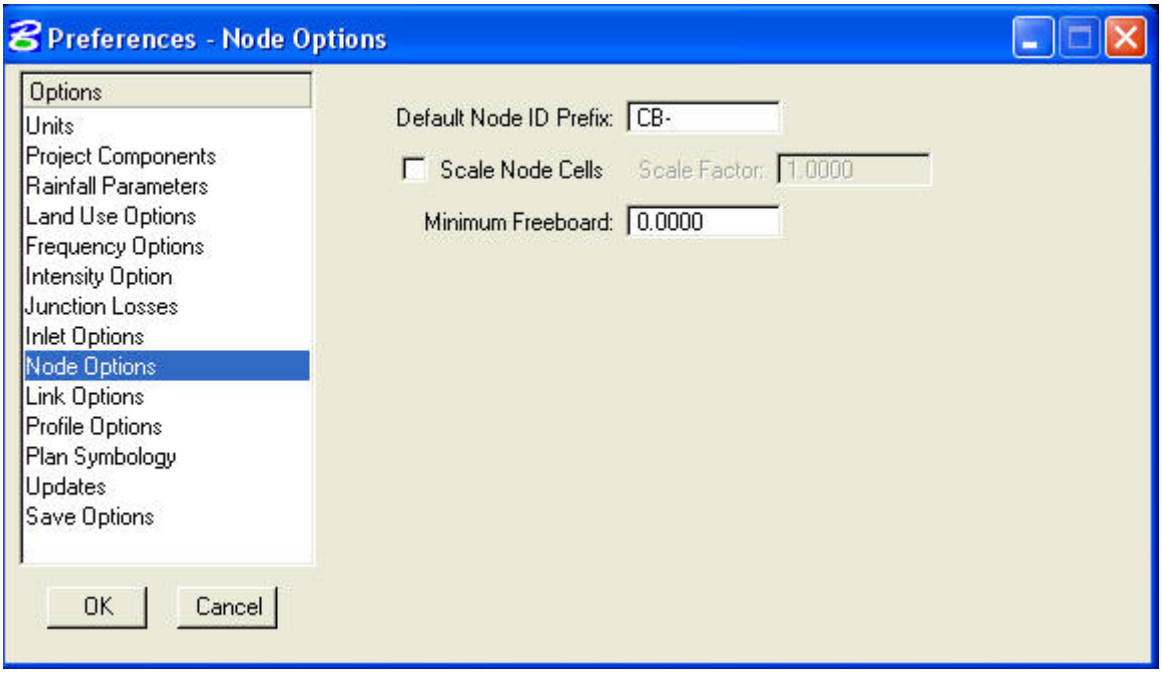
10. Inlet Options

Set the Inlet Options (use the values shown below for the Inlet Options):



11. Node Options

Set the Node Options:

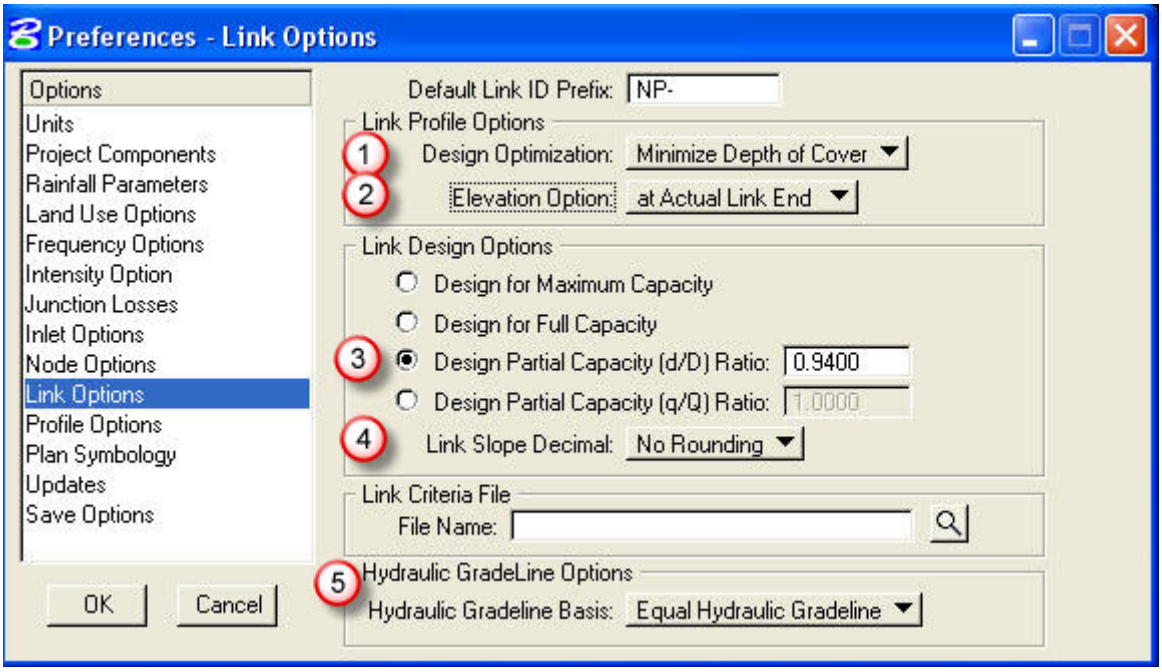


The Default Node ID Prefix will be different depending on the type of node being placed. Refer to the Nodes section and input the most commonly used prefix.

We need to be consistent with our naming conventions so we can easily read each other's data and the Roadway Designer will not be confused by different symbology.

12. Link Options

Set the Link Options:



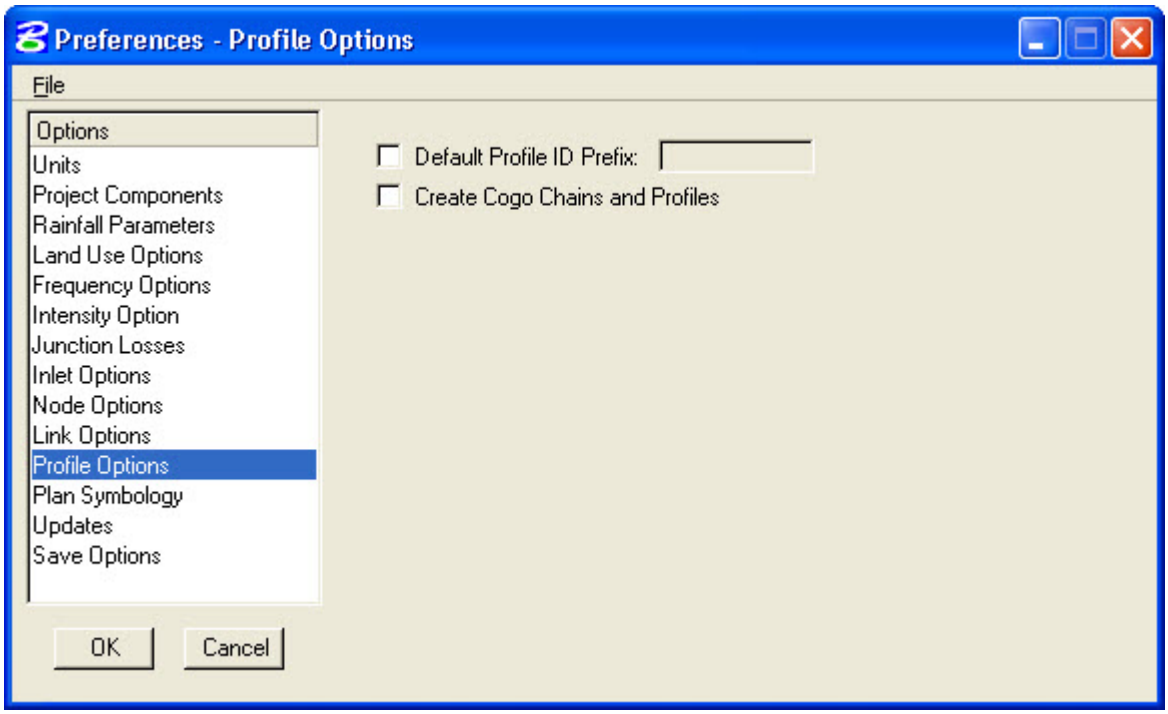
1	Design Optimization	Set "Design Optimization" to "Minimize Depth of Cover".
2	Elevation Option	Set "Elevation Option" to "at Actual Link End".
3	Link Design Option	Set "Link Design Options" to "Design Partial Capacity (d/D)" and set to 0.94 (i.e 94% full).
4	Link Slope Decimal	Set "Link Slope Decimal" to "No Rounding".
5	Hydraulic Gradeline	Set "Hydraulic Gradeline Basis" to "Equal Hydraulic Gradeline".

The Default Link ID Prefix will be different depending on the type of link being placed. Refer to the Links section and input the most commonly used prefix.

We need to be consistent with our naming conventions so we can easily read each other's data and the Roadway Designer will not be confused by different symbology.

13. Profile Options

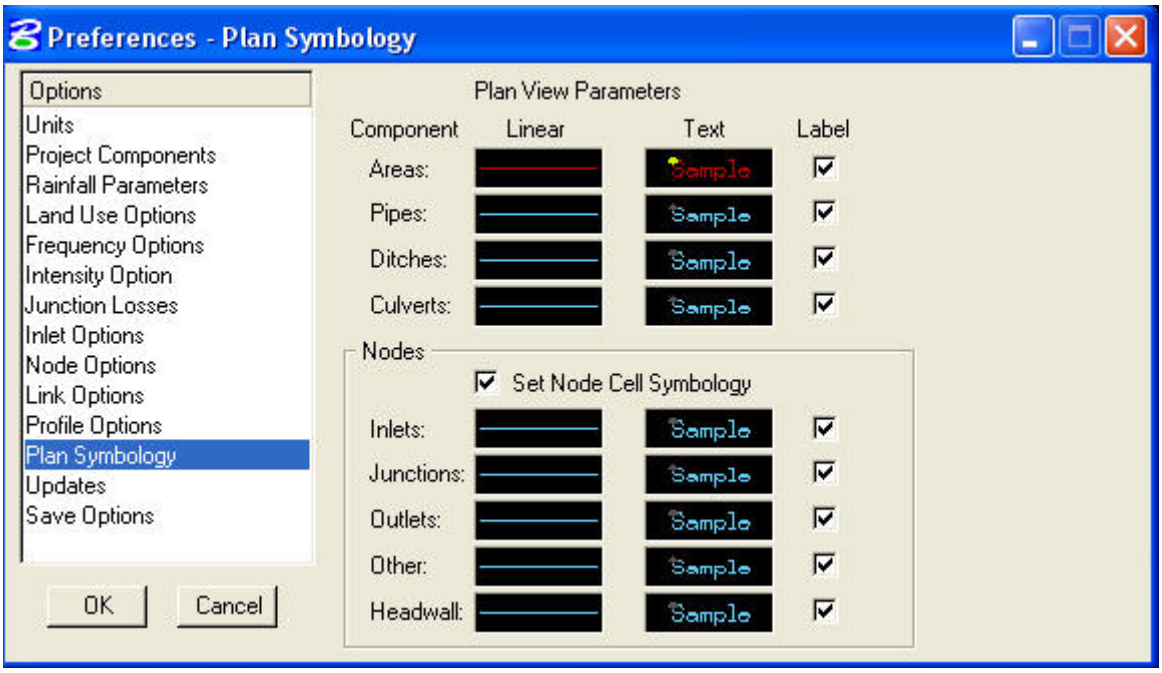
Set the Profile Options:



You can choose to have a prefix for all profiles that you create. For now we do not plan to create any COGO chains or profiles.

14. Plan Symbology

Set the Plan Symbology: We need to make sure our design is on levels that the Roadway Designer will not use. This way when they bring in our design there will be no confusion as to whose information is on what level. The labels will share the same attributes as its component. Make sure the following conventions are followed for levels, weight, line style, and colors for consistency:



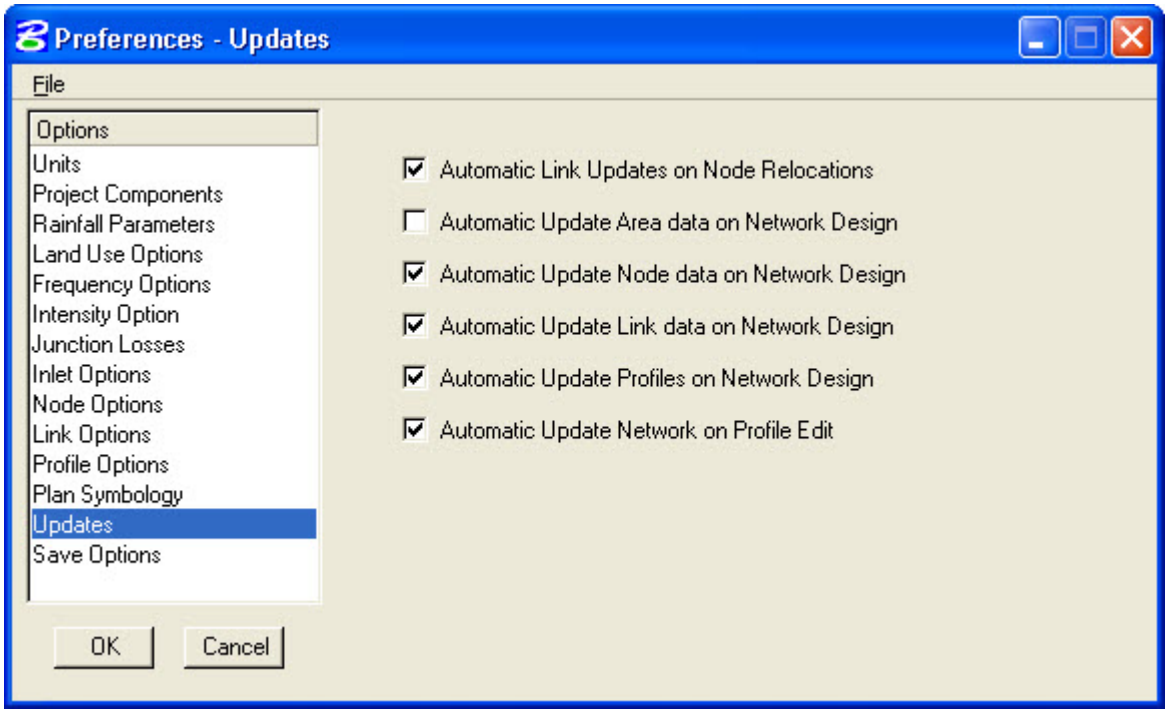
Use the following symbology:

Attribute	Areas	Nodes	Pipes	Ditches
Level	RD_HY_Area_SS	RD_PD_DR_Inlet	RD_PD_DR_Pipe	RD_PD_DR_FlowCtrl
Color	3	106	106	106
Style	0	0	0	0
Weight	0	0	8	0

In Plan Symbology, you can copy settings from one black box to another by right-clicking on a box and selecting Copy, then right-clicking on another and selecting Paste.

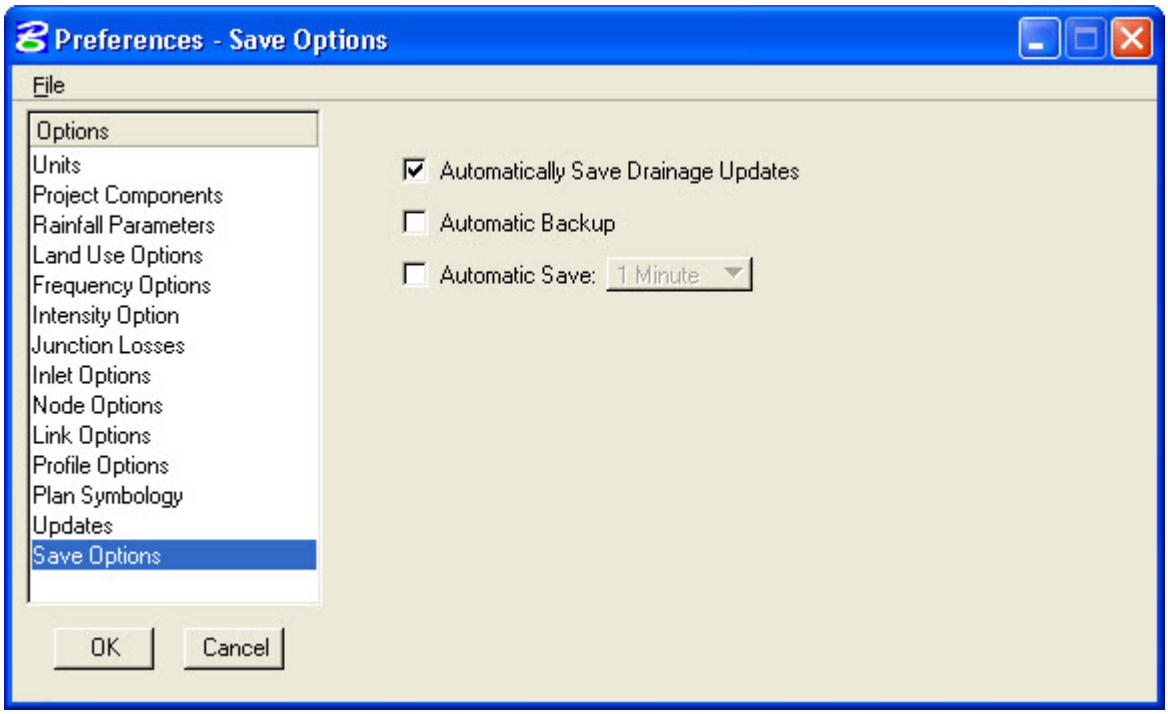
15. Updates

Set the Updates Options (use the settings below):



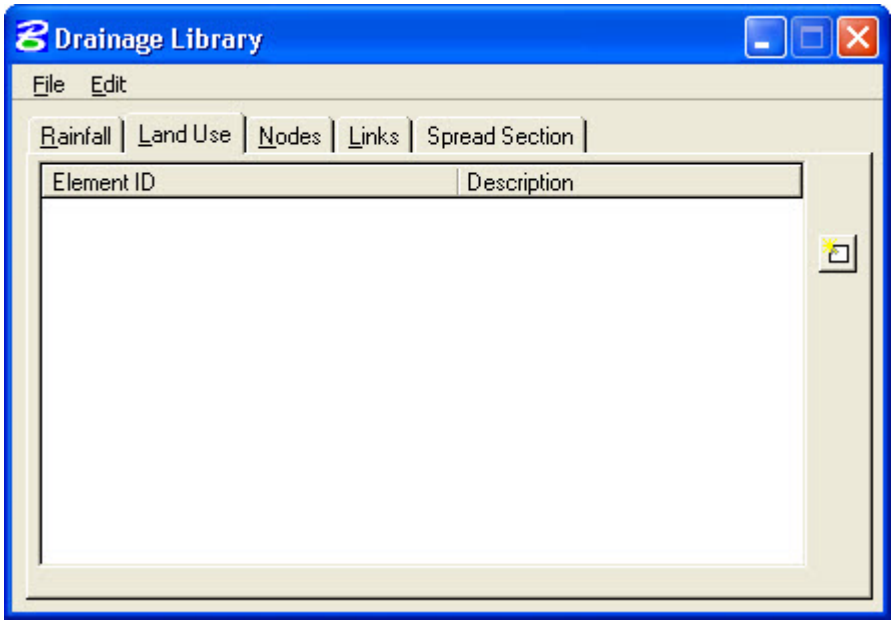
16. Save Options

Set the Save Options (use the settings shown below):



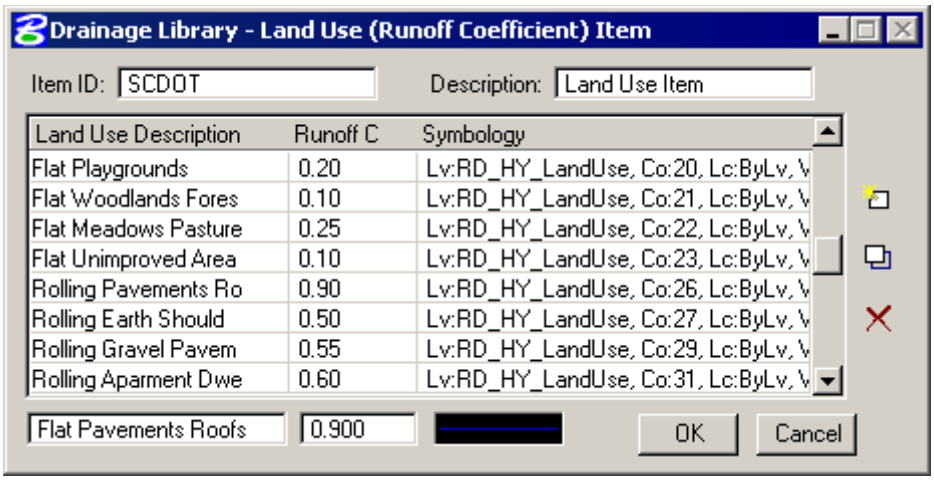
17. Drainage Library

Select Drainage>Project>Drainage Library. Make sure that the Drainage Library DLB File for your active GDF File is open. If not, then go to File and choose the correct one. There are five tabs – Rainfall, Land Use, Nodes, Links, Spread Section. Rainfall, Nodes, Links, and Spread Section have been preset and need not be modified.



18. Land Uses

Use the Land Use Tab to view the land uses on your project. The following are the most commonly used Land Uses.

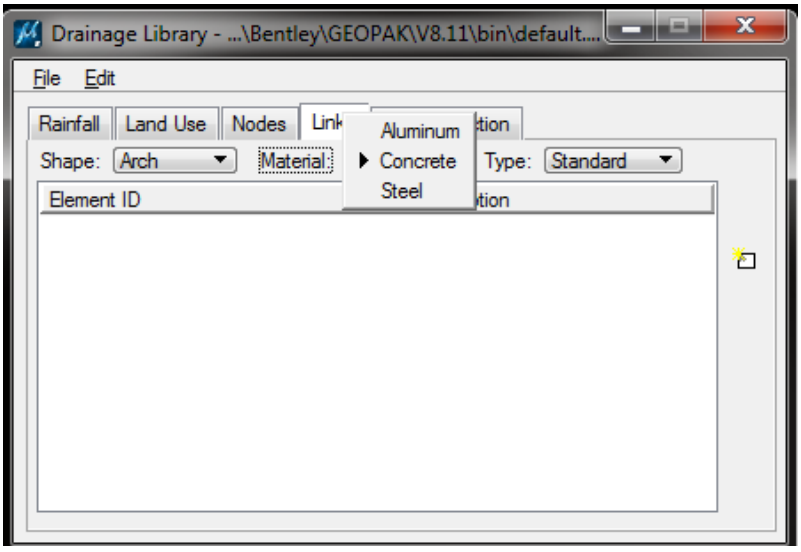


Entering drainage areas will be discussed in the Areas section.

19. Links

Circular Links can be the following material types: Aluminum, Concrete, Plastic (HDPE), or Steel.

In this menu, the “Steel” setting contains the information for all smooth wall (non-corrugated) pipes including Concrete and Plastic. The “Concrete” and “Plastic” settings are not used – pipes of those materials should be set to “Steel”. Corrugated Wall pipes are stored in the “Aluminum” material setting.



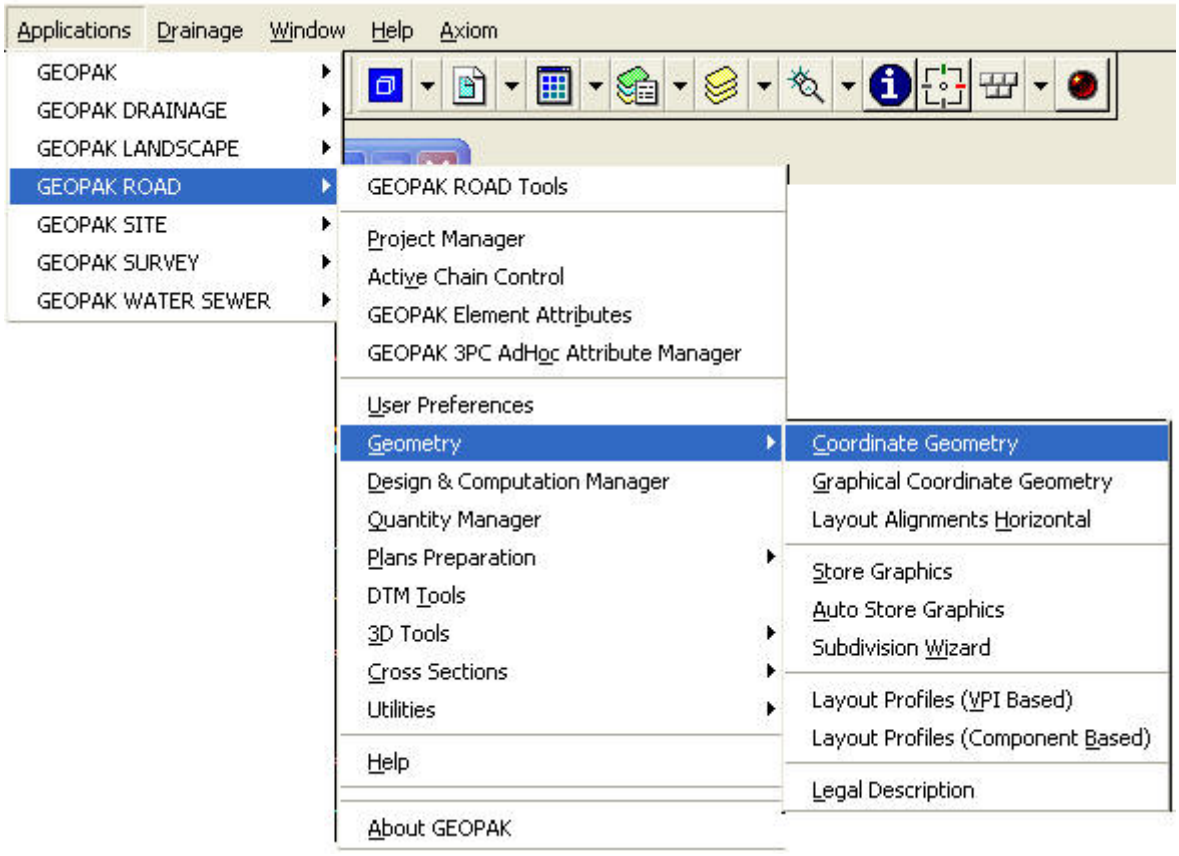
Chapter 3: Nodes

A node represents any new or existing catch basin, drop inlet, junction, open-end pipe or outlet. Each has a cell symbol through GEOPAK that represents that particular node. All nodes appear large in the Drainage Library, but some intentionally plot as a dot and are only visible when you zoom in closely. Outlets, pipe ends, and existing structures do not need a visible mark. Outlets and pipe ends are defined by the line drawn for the pipe. Existing structures are already defined by the Roadway Designer and we want to avoid an overlap of cells. All nodes will have a visible label when placed, and you can always select a node by its label.

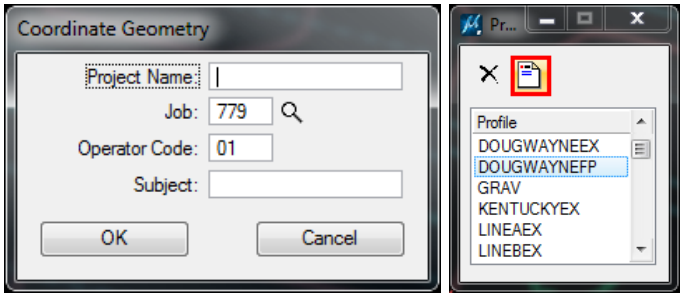
Note that GEOPAK Drainage does not automatically space your basins; you must use our spacing charts. To aid in finding low and high points, use the COGO tool.

1. COGO

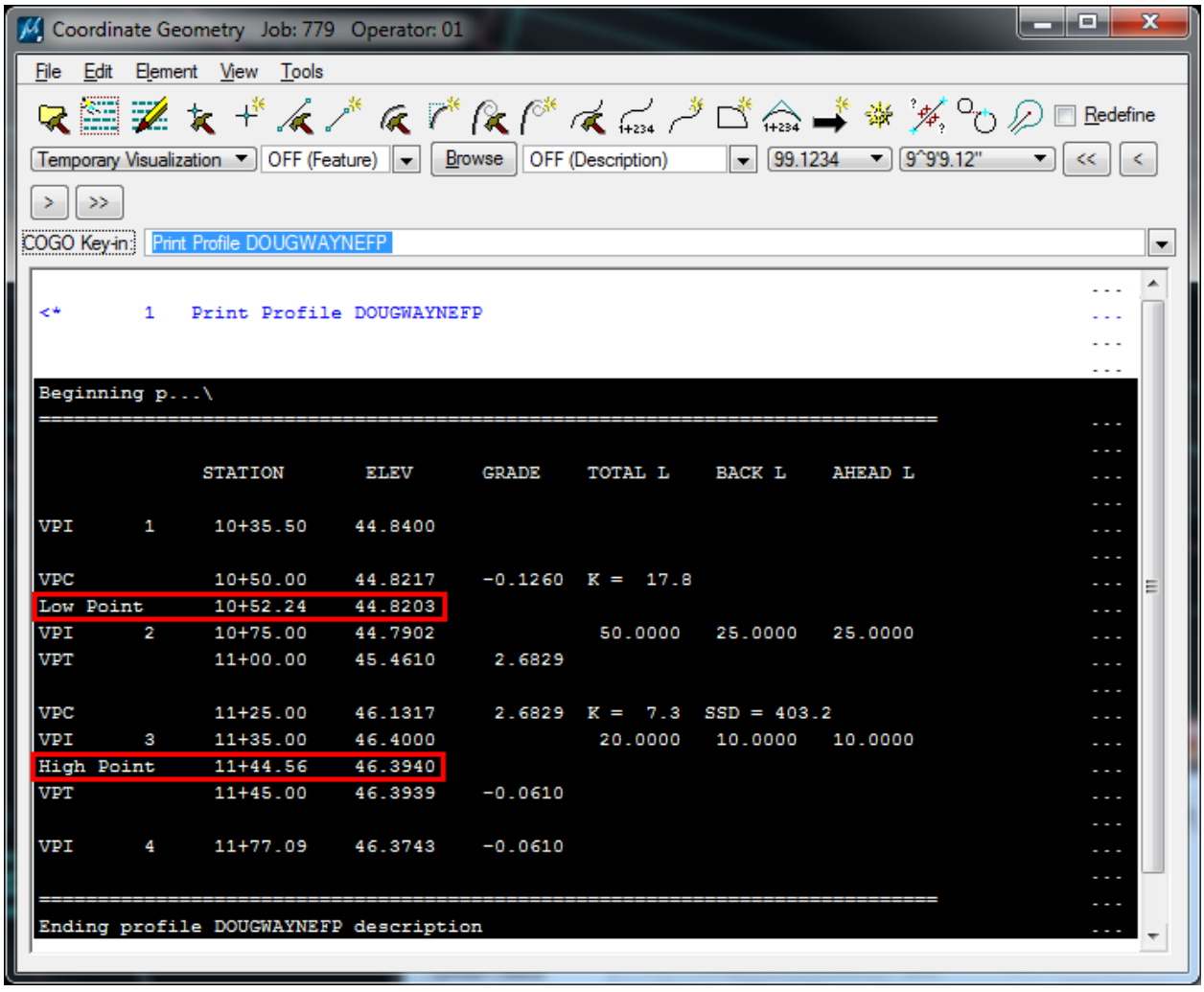
In December 1998, we were advised to use COGO to locate high and low points on the centerline profile of the road. You can utilize this information to space your basins.



The program is under GEOPAK Road instead of GEOPAK Drainage. To open this go to **Applications>GEOPAK Road>Geometry>Coordinate Geometry**. In the Coordinate Geometry box choose the Magnifying Glass button by **Job Number** and select the GPK file. The **Operator Code** is the county number. Do not fill in the other boxes.



Open **Element>Profile>Utility**. Obtain the centerline profile and left and right top of curb profiles from the spreadsheet the Roadway Designer emailed, choose one from the list and select the **Print** button. The information will appear in the window. All low and high points for that profile will be marked accordingly. Repeat the process for the two remaining profiles.

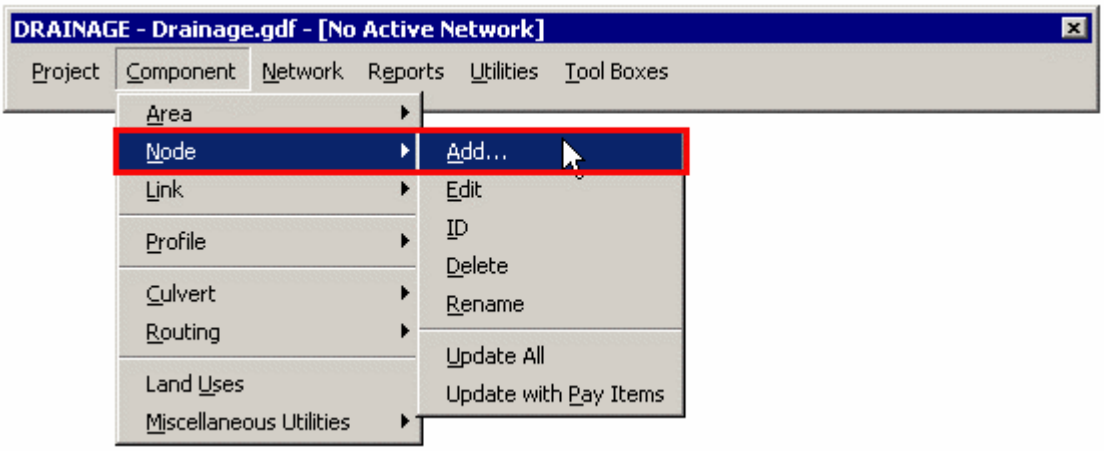


To print, highlight the data, then right click, then choose **Copy**. Paste the information into a text file or word processor so you can remove unnecessary information before printing as normal for text documents.

2. New Nodes

It is best to place all of your nodes for a system before adding links; then if you need to add more nodes you can do so without having to move links.

To add nodes go to ***Drainage>Component>Node>Add***. The "Add a New Node" dialog will open for each Node asking for a Node name prefix and number. The nodes number automatically by increasing linearly from 1 to 10000.



Notice in the **Node ID** that it is using the default that you set in Preferences. You may need to change this. The following chart will aid you in the naming conventions:

New Node:

Node Type	ID Prefix
CB Type 16,17,18,1,9,12,14,15, 25	CB
DI	DI
Tee Joints	T
MH, JB	J
Dummy Nodes	DN
Outlet	OP
Water Quality Structure	WQS
Bends (15,30,45,90)	B

Wye	W
Multi-grate	MG
Rebuild Existing Structure	RB

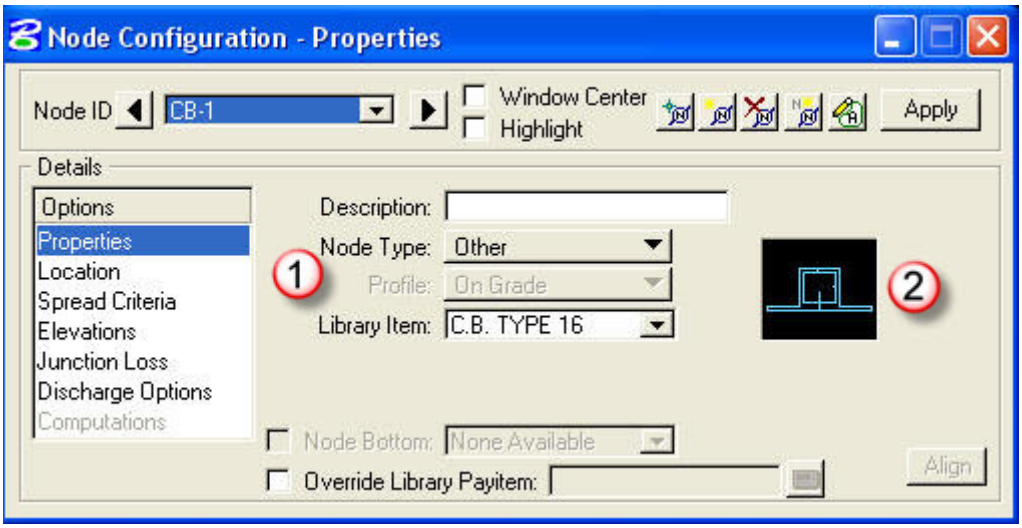
Existing Node:

Node Type	ID Prefix
All CB, DI, Tee Joints, MH, JB, Dummies*, Outlets, Water Quality Structure, Bends, Wye	EX

* A dummy node defines pipe ends. It can also represent existing structures that do not have cells; however, cells do exist for existing manholes, drop inlets, and catch basins.

3. Properties

Set the Properties of the Node (the Description field is optional):

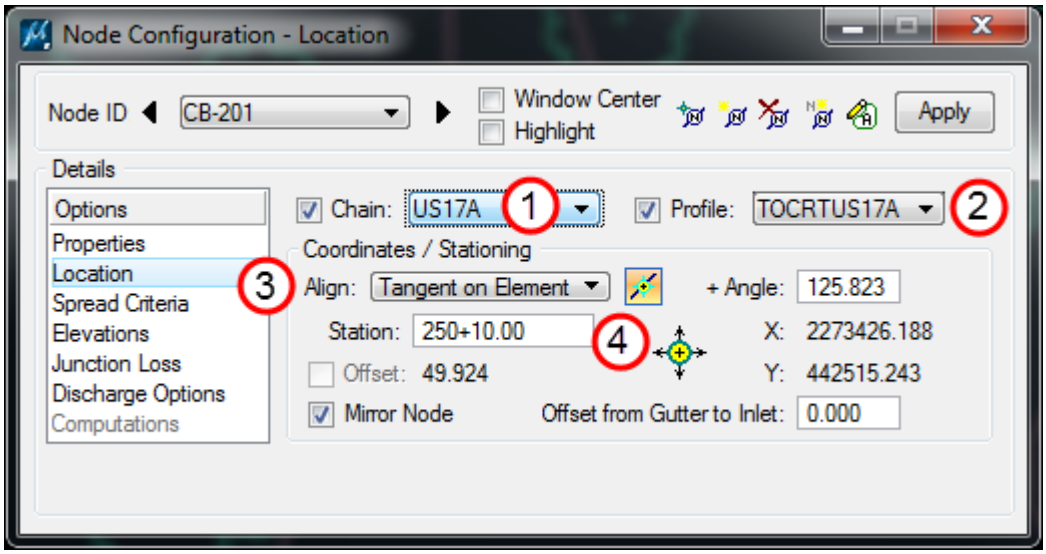


1	Node Type	Choose the appropriate "Node Type", "Profile", and "Library Item" to be placed from the Drainage Library
2	Preview	Ensure the correct cell is shown in the preview window

If placing a Tee or a Bend, initially choose the Tee or Bend that has the description “generic”. After the design is finalized go back and reselect the Library Item to the correct size of Tee or Bend. Make sure you adjust the inverts at the Tees if you have a different trunk line pipe size. For Tees, the trunk line is listed first and the stub is second. If you are entering the outlet node, this is where you would define the tailwater.

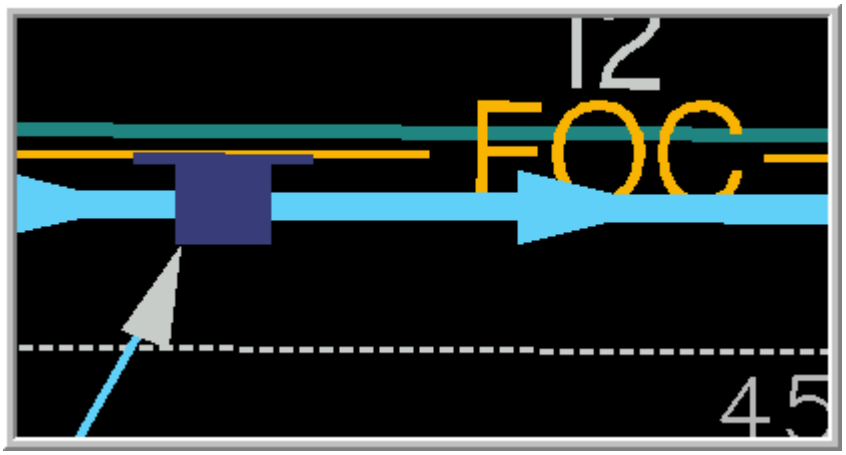
4. Location

Set the Location of the Node:

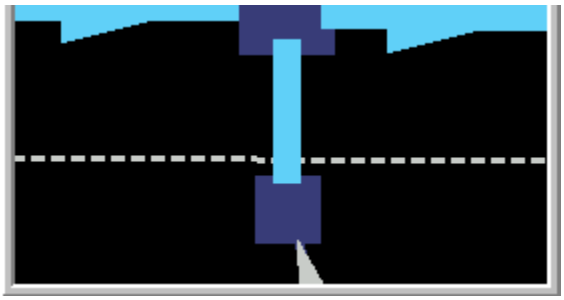


1	Chain	Use the pulldown arrow to select the correct chain name. This information comes from the GPK File that you downloaded, and the Roadway Designer will list in the spreadsheet what chain represents which road. If you are placing a node on a side road make sure to use the corresponding side road chain name.
2	Profile	Use the pulldown arrow to choose the correct profile name. Once again this information will come from the GPK File, and the Roadway Designer will list in the spreadsheet the names of the left and right top of curb grades. You will use top of curb grades to define the starting elevation of the catch basin for all basins on curb.
3	Alignment	Set the **Align** to **Tangent on MS Element** for all nodes that will be flush with the Face of Curb (FOC). Then click the **Select** button by Tangent on MS Element and select the FOC line which will also automatically set the offset distance.
4	Placement	Either enter the Station value or click the large **PD** button to dynamically place the node cell

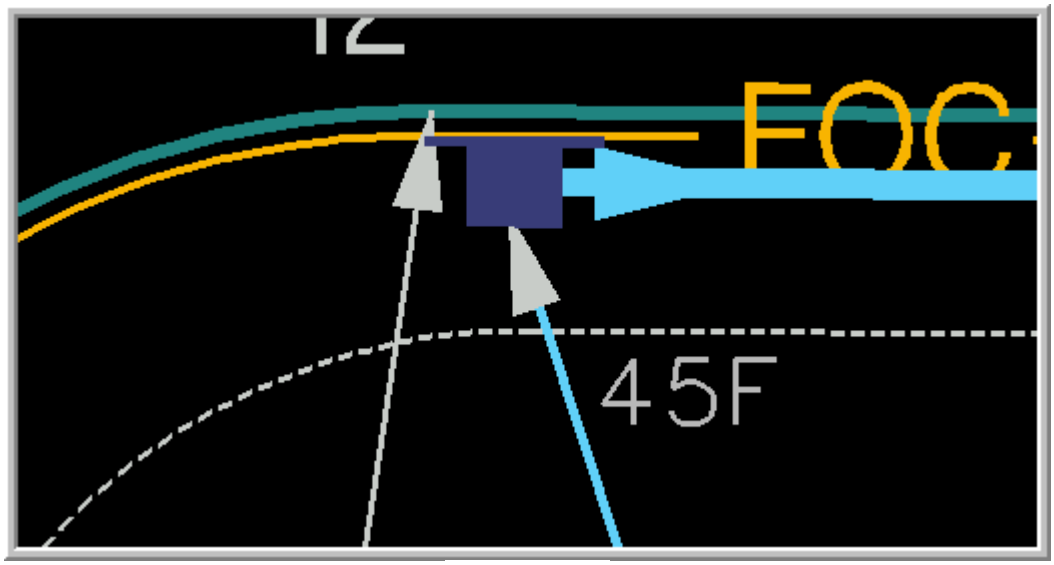
To be consistent with the Roadway Designer the following picture represents where the cell should actually be placed when on the curb. If you place nodes as instructed previously, the cells will match perfectly with the FOC as shown below:



The image below exhibits how to place a drop inlet against the fill. Offset the inlet approximately one foot from the toe of fill. You don't need to measure the one foot distance, just get close; the important thing is to provide a little space between the inlet and the slope. This avoids cutting into the slope to install the drain and reduces sediment flow into the storm system.



The image below exhibits how to place a cell at a radius. Place the beginning of the throat at the radius point allowing the entire inlet to be constructed along the tangent section outside of the radius.



We need every cell in its proper location because the Roadway Designer cannot move our cells around; this would violate our design and they will have trouble with labeling. Please be conscientious with every cell location. Once you get a feel for it, it will not take but a second to get the cells located correctly.

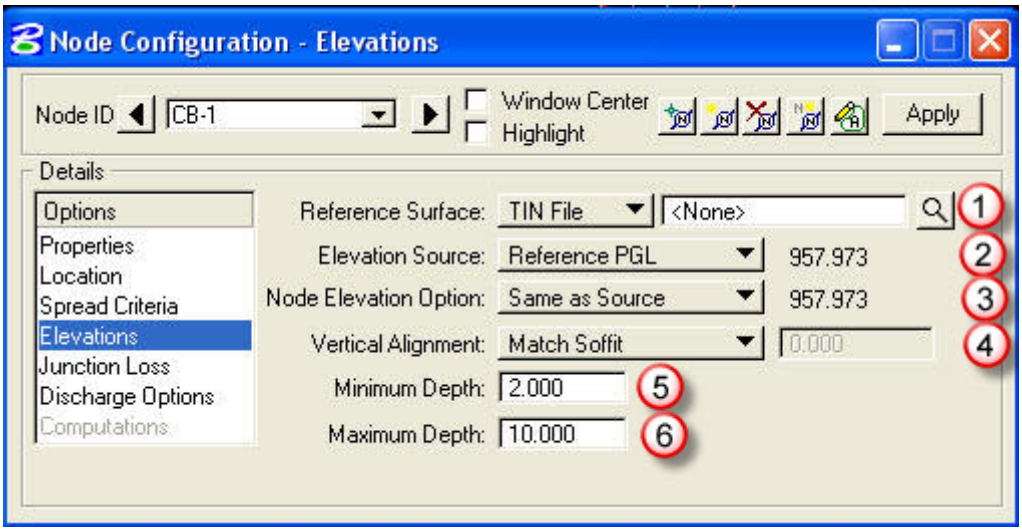
5. Spread Criteria

N/A for SCDOT Drainage design.

SCDOT does not enter any information in this window. An error message may appear due to spread. This message should be ignored.

6. Elevations

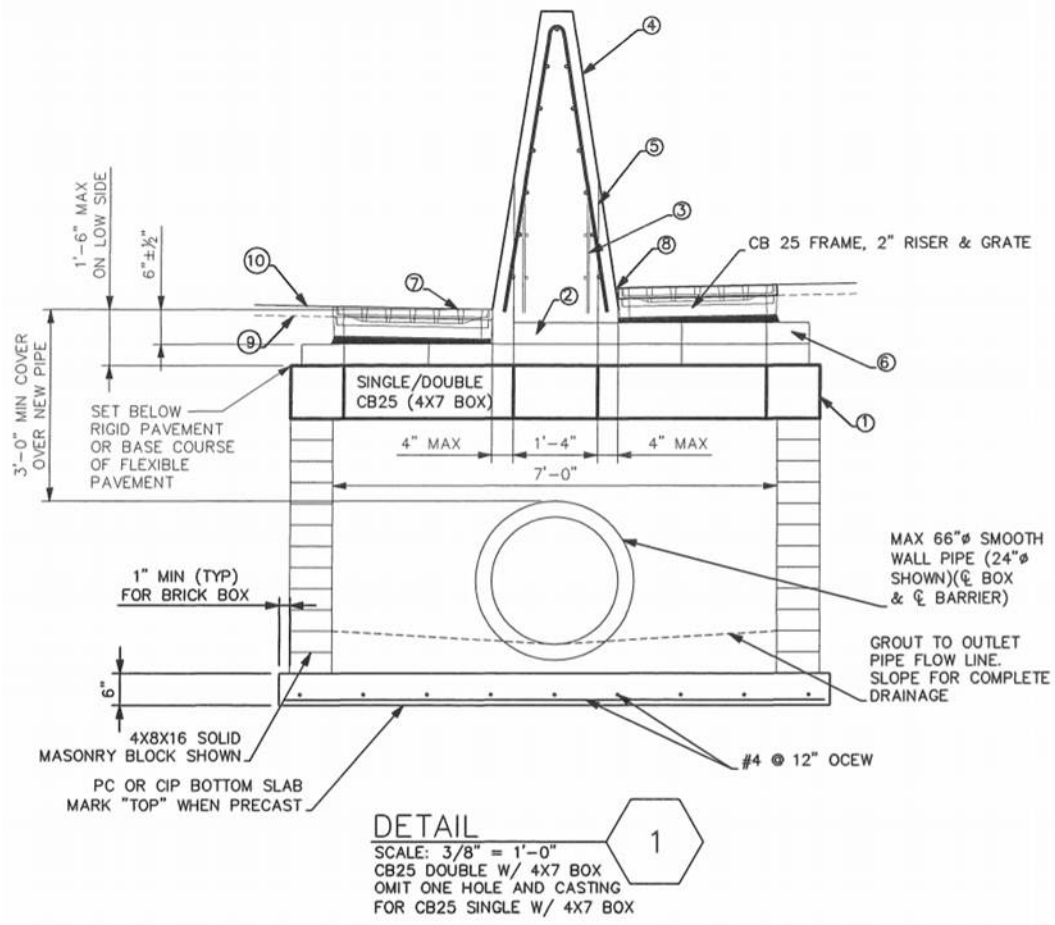
Set the Elevation of the Node:



1	Reference Surface	If using a TIN file enter it here, otherwise leave blank
2	Elevation Source	Select "Reference PGL"; an elevation will automatically appear (this is the top of curb elevation and it is read from the Profile you entered previously in the Location option). If the cell is not on the curb, you will need to choose "User Supplied" and enter the elevation. For a drop inlet enter the ground elevation. The outlet is discussed in detail under Networks. A stub out or open end pipe elevation will be determined following the same procedure used for the outlet.
3	Node Elev. Option	Set to "Same as Source";
4	Vertical Align	Start with "Match Soffit" but change as needed to represent drop manholes, to match flowlines with other systems, etc.
5	Min. Depth	The Min Depth varies per Node type (see table below)
6	Max. Depth	Enter 10'. If you get a warning message, you have exceeded your maximum depth; in this case special consideration will be required with the road designer.

NODE	Min. Depth	
	From TOC	From Ground
Water Quality Structure	-	0.0’
CB 1, CB 15, CB 16, DI 115	2.0’	-
CB 17, CB 18	3.0’	-
DI 24x24, DI 24x36, DI 112	-	1.0’
CB 9	-	2.0’
MH, CB 25	-	3.0’
Outlet, Dummy	0.0’	0.0’

For a double CB25, the minimum cover should be measured from the lower pavement elevation as shown below. This works for grade separations of up to 18”.



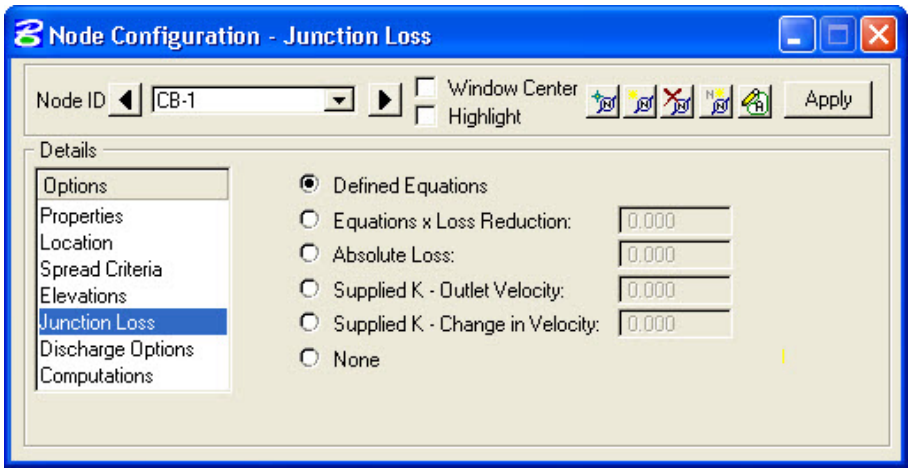
For grade separations larger than 18", the designer should only use single CB25s. If drainage is needed on both sides of the barrier/wall,

- Design parallel drainage on both sides of the wall.
- Minimum cover is 3' from lower grade for pipe on lower side.
- Minimum cover is 3' from upper grade for pipe on higher side.
- Stagger Stations of catch basins at least 100' (only box on one side is allowed in any given cross section)
- See Standard Drawings for Concrete Gravity Retaining Walls for additional restrictions (when available).

For walls taller than 6', the hydraulic designer should work with the Structural Engineer to identify maximum box depth on the high side of the wall so the box and pipe do not conflict with the retaining wall footing.

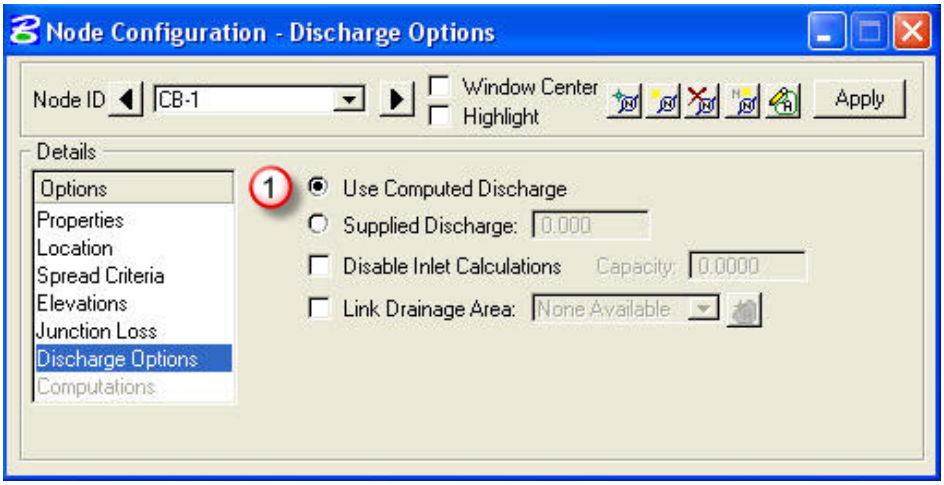
7. Junction Loss

Set the Junction Losses (use the Defined Equations option as shown below):



8. Discharge Options

Set the Discharge into the Node:

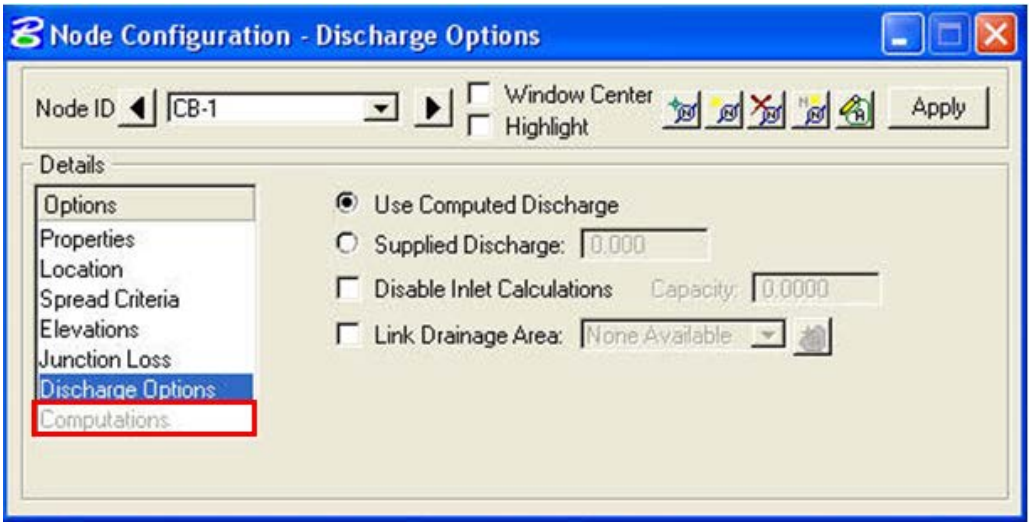


1	Discharge	Set to "Use Computed Discharge" for almost all situations so the discharge from the associated area is directed towards the node.

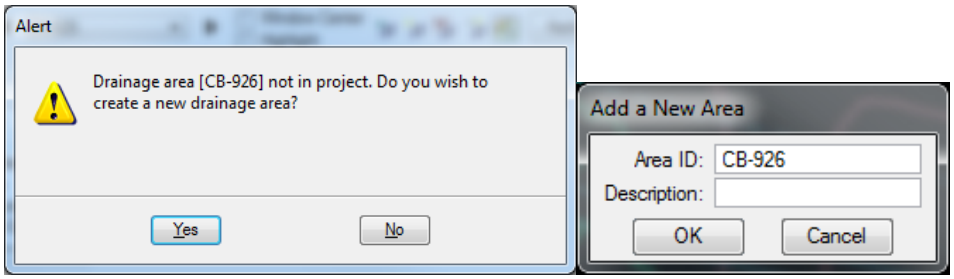
Some drop inlets and cross line pipes require a “Supplied Discharge", in which case enter the discharge value in the key-in field.

9. Computations

Note that the Computations Option is greyed out. Since SCDOT does not let GEOPAK calculate the spread, this option will ***not*** be available. Otherwise, view the computations.



The Edit Area button (to the left of the Apply button in the above picture) opens the **Drainage Area Definition** window. Use this to create an associated drainage area if your node collects water discharged from one. Junctions, outlets, and water quality structures cannot collect water. When you select this button it will tell you that this area does not yet exist and ask if you want to create it. Answer "yes". The **Add a New Area** box will appear. The **Area ID** is whatever you named the node; do not change this. You can add a description if you desire, but it will only be used to describe the area and not the node.



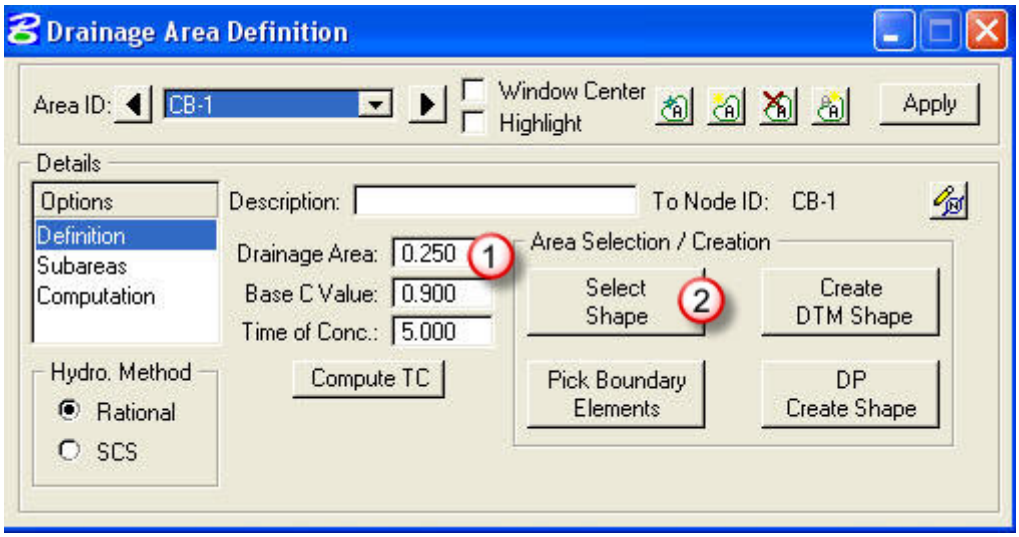
Chapter 4: Areas

See the individual Area options below.

SCDOT uses the "Rational Method" only.

1. Definition

Define the Drainage Area:



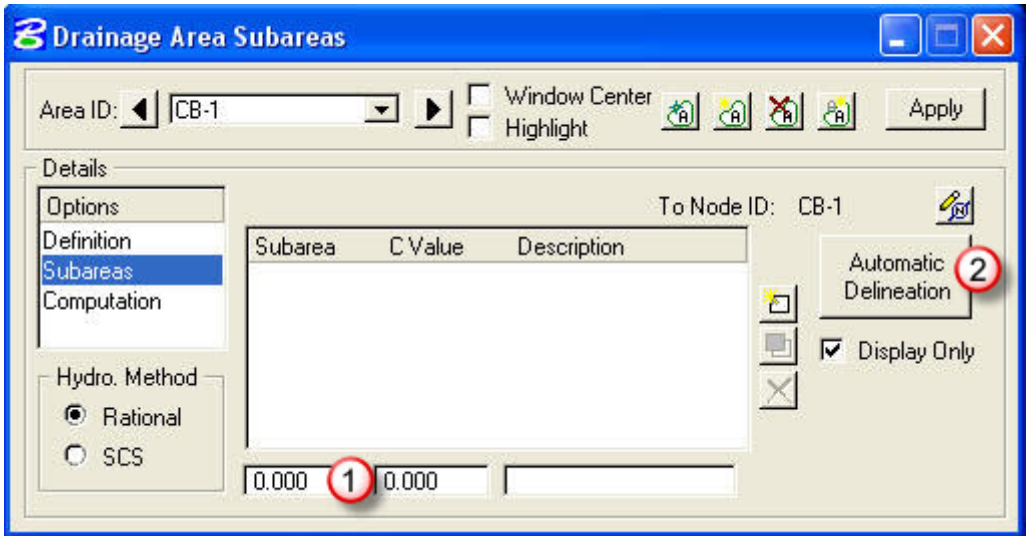
There are four methods of inputting areas for a basin; use one of the two below, or for more info see the Bentley Geopak Drainage Manual.

1	Drainage Area	Just key-in the acreage in the key-in field
2	Select Shape	Draw a MicroStation closed shape, then click the Select Shape button, identify and accept the shape, and the acreage key-in field will automatically populate with the acreage value.

Enter the **Base C Value**, and the **Time of Concentration**. Disregard the **Compute Tc** button.

2. Subareas

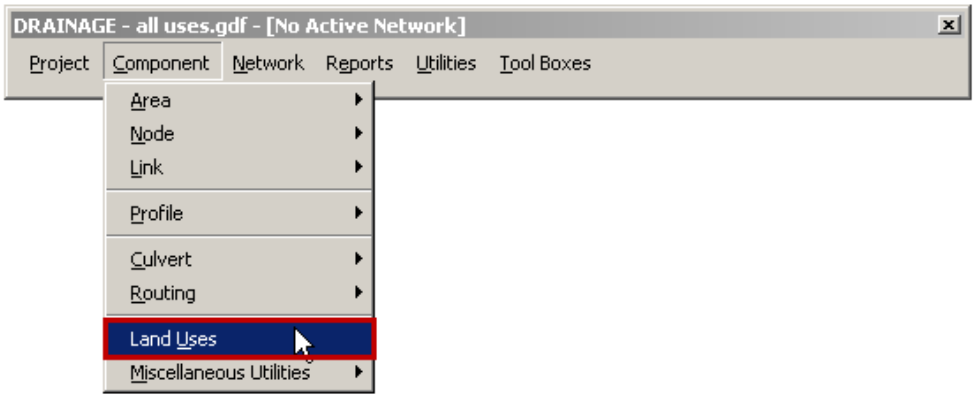
Set the Drainage Area's subareas options. This process assigns new C values to pieces of the overall Drainage Area set in the Definition option and provides a final composite C value for the total area. Any remaining area not explicitly defined in this section is left as the C value set in the Definition option.

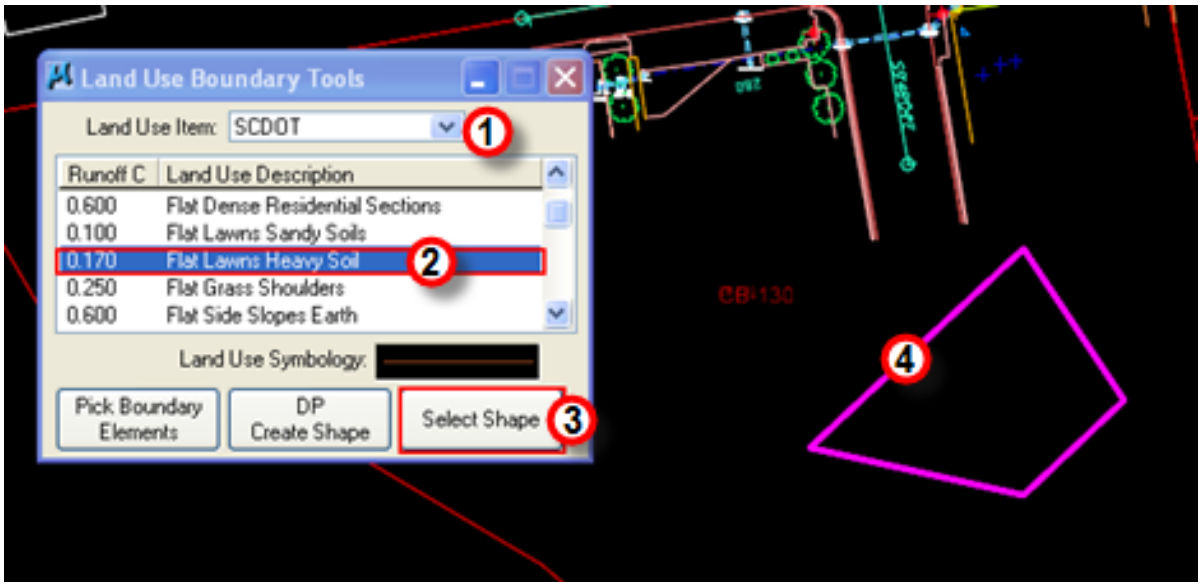


There are three methods of inputting subareas into a basin; use one of the two below, or for more info see the Bentley Geopak Drainage Manual.

1	Manually	Just key-in the subarea acreage into the first key-in field, the "c" values in the second, and any desired description in the third, and use the add/modify/delete buttons to effect the entries in the list
2	Automatically	Click the **Automatic Delineation** button, which initiates a scanning of the DGN file and reference files within the main drainage area for closed shapes matching the symbology of the Library Land Uses as selected in the Preference Land Use Option, and automatically populates the Subarea Table.

Subareas can be drawn on the appropriate symbology for or have the symbology assigned after drawing. To assign land uses to existing, closed shapes in the design file, select Component > Land Uses:



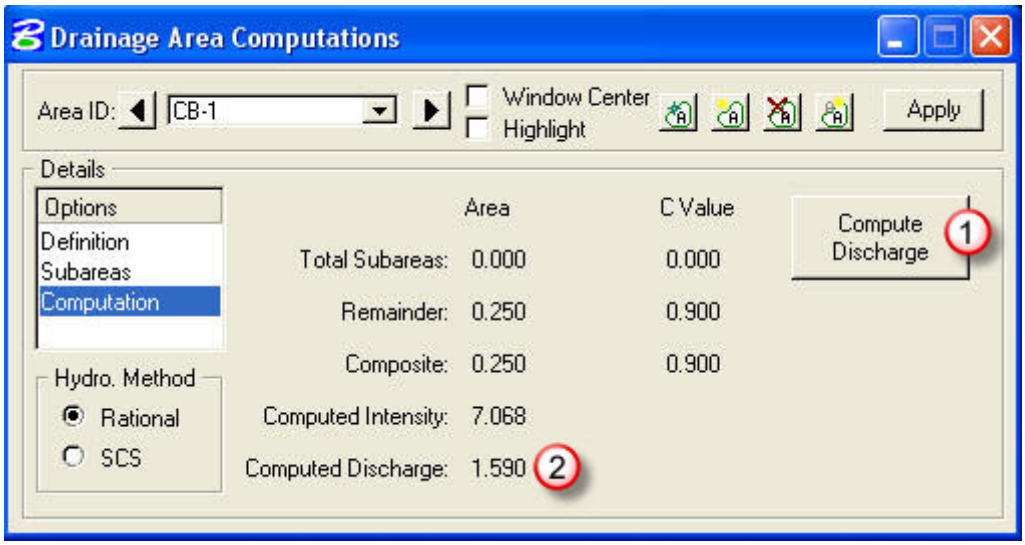


1	Land Use Item	Select from the available Drainage Library Land Use Items
2	Table entry	Highlight the Land Use Description you want to designate
3	Select Shape	Click the **Select Shape** button.
4	MS select the shape	Data point on the drawn shape, and again to accept, which will modify the shape's symbology to that of the designated land use.

Once the closed shapes are designated the correct symbology above, use the **Automatic Delineation** button in the Subareas tab to delineate them.

3. Computation

Check the Computation option, as shown below:

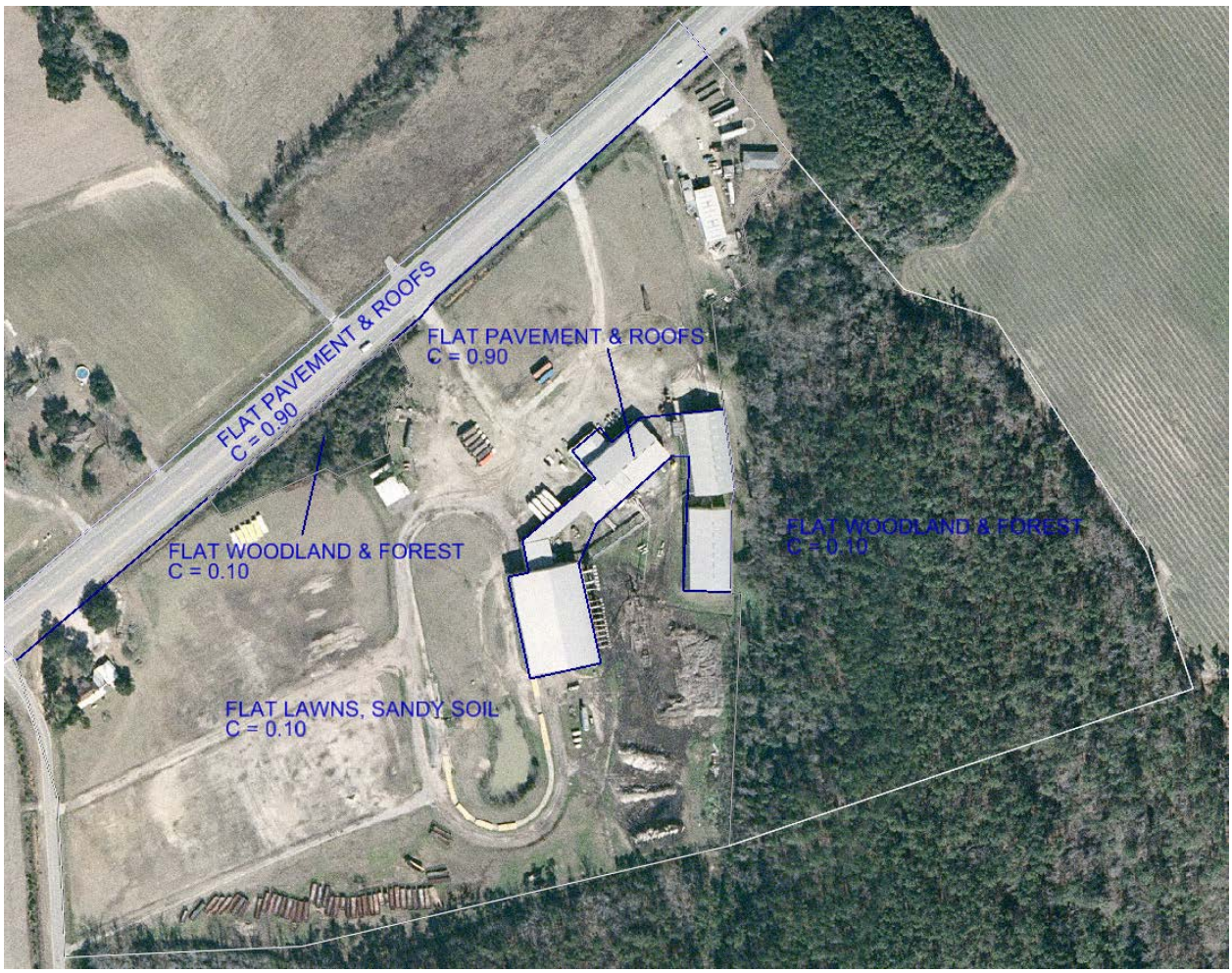


1	Compute Discharge	Click the **Compute Discharge** button to initiate the computations
2	Results	Verify the Drainage Area computation results in the *read-only* dialog

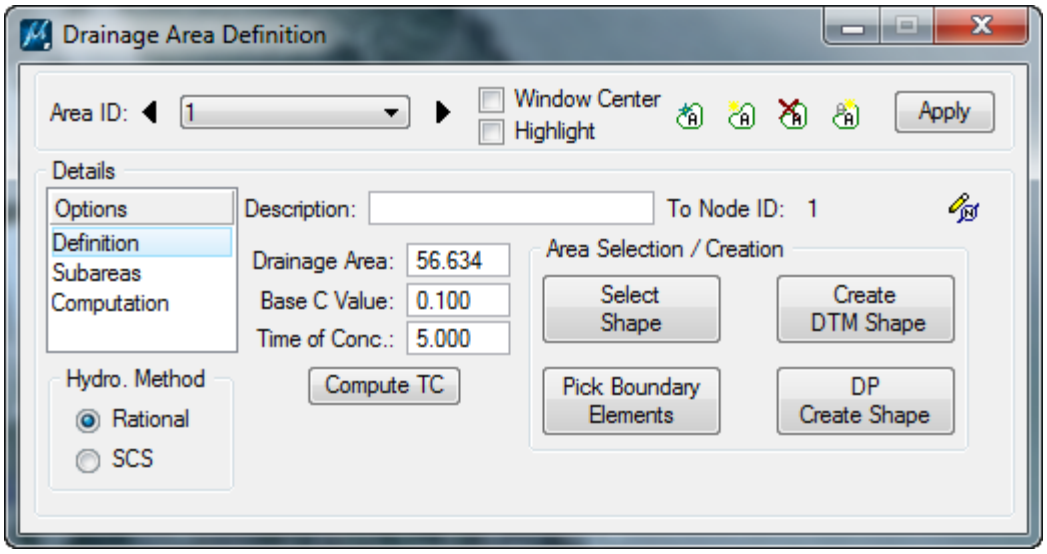
This is just the discharge value for this individual node, and ***not*** the discharge in the pipe.

Example:

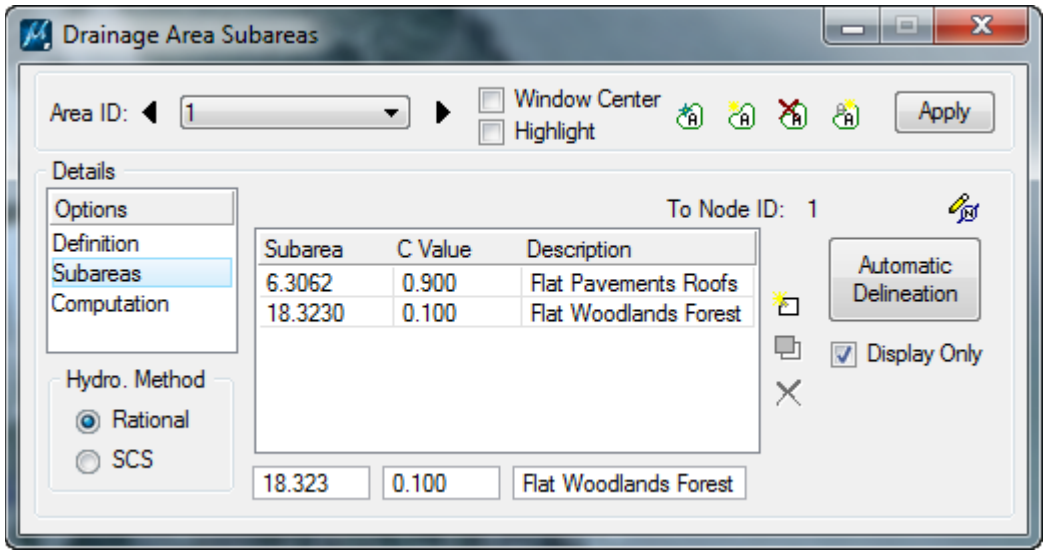
The image below shows a hypothetical drainage basin divided into sub-areas. The majority of the basin is open land. The basin also contains buildings, paved roadway, and wooded areas. First, the overall basin must be delineated using the MicroStation Place Smartline tool and the primary land-use for the basin determined – in this case, the land use of the majority of the basin is Flat Lawns, Sandy Soil. The sub-areas with different land uses from the primary area must also be delineated and the smartlines set to the appropriate colors and level using the Land Use Boundary Tool as described in Chapter 4, Section 2.



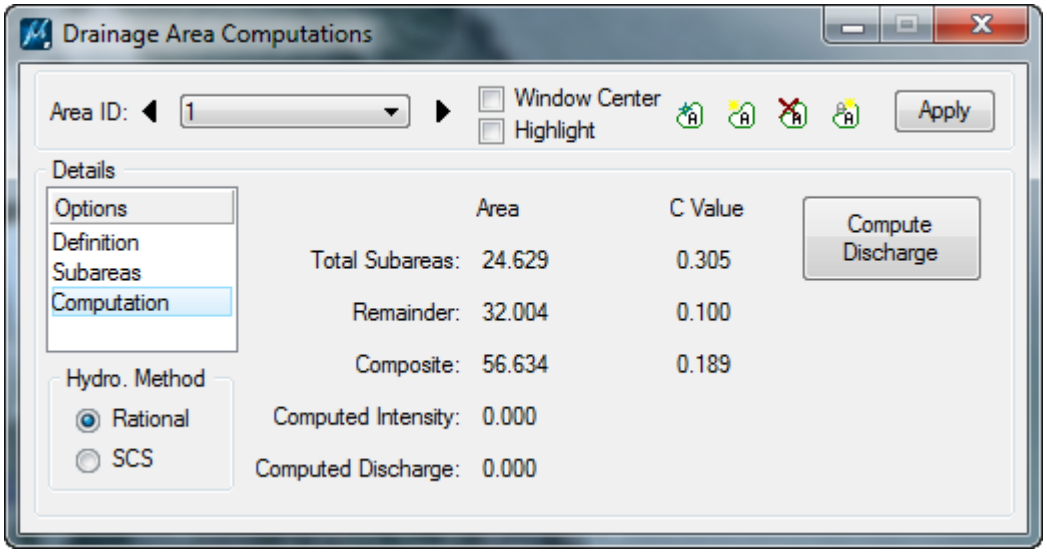
Open the Drainage Areas window (Drainage -> Components -> Areas -> Add) and create a drainage area to represent the basin. In the Definition submenu, click the Select Shape button and choose the shape delineating the overall basin. The Drainage Area, Base C Value, and Time of Concentration forms will be filled in automatically. Change the Base C Value to 0.10, as the majority of the basin is Flat Lawns, Sandy Soil.



On the Subareas submenu, click the Automatic Delineation button. The subareas list will be populated with the total area associated with each land use.



The Computations submenu shows the composite area representing the entire basin and its C-value. When the drainage area is linked to a node, Geopak will use these values to calculate the discharge into the storm system.

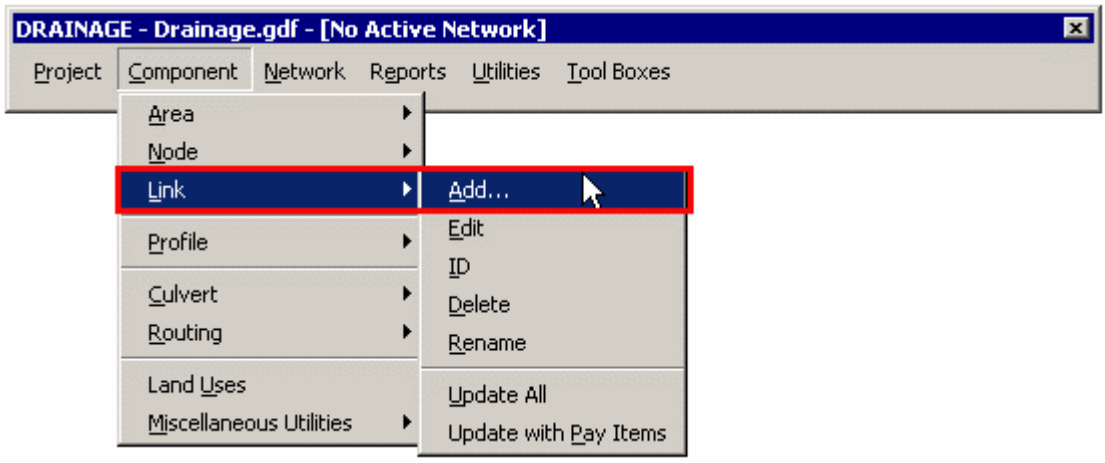


Chapter 5: Links

Links can be pipes or ditches. At SCDOT, the Geopak Drainage ditches are used for analyses only and not for auto-designing.

There must be two nodes of any type already placed in order to define a link.

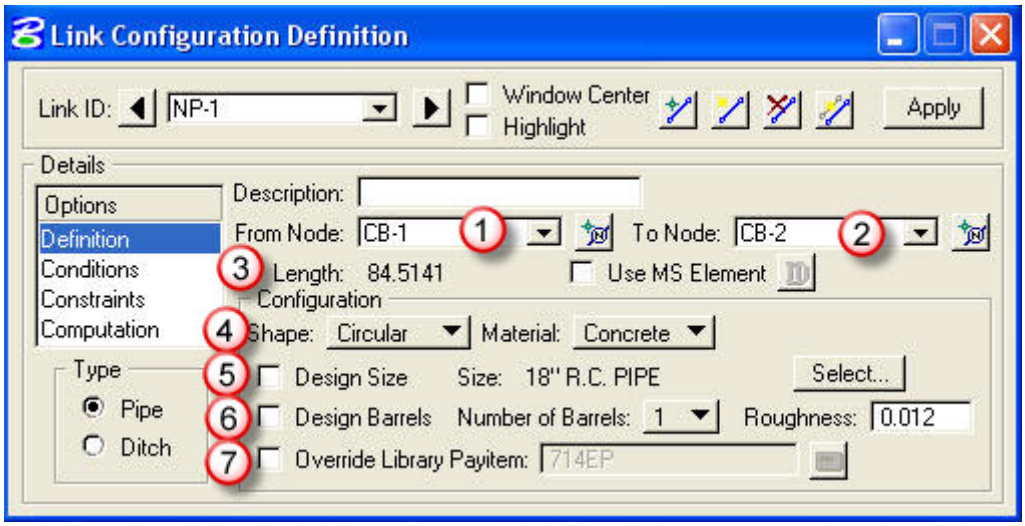
To add a link, go to **Drainage>Component>Link>Add**. Select a link name as stored in the **Preferences** for the prefixes shown below (the numbering will be auto-sequenced):



New Pipe	**NP**
Relay Pipe	**RP**
Existing Pipe	**EP**
New Ditch	**ND**
Existing Ditch	**ED**

1. Definition

Set the Link Definition option as shown below (Description field is optional):



1	From Node	Select the *Up*stream Node
2	To Node	Select the *Down*stream Node
3	Length	The length is automatically displayed as the distance between the From and To Nodes
4	Shape/ Material	Set the Shape to "Circular" and the Material to the appropriate type from the Drainage Library.
5	Design Size	Toggle ON for new pipe (NP); or toggle OFF for existing pipe (EP or RP) and click Select to pick the size from the Drainage Library
6	Design Barrels	Toggle OFF. Set to 1 for new pipe (NP), or to the number of existing barrels for existing pipe (EP or RP).
7	Override Library PayItem	If your link is an existing pipe (EP) then check the box beside **Override Library Payitem** and type **714EP** in the space provided. This will prevent the Roadway Designer from overriding the existing pipe line style with the new pipe line style.

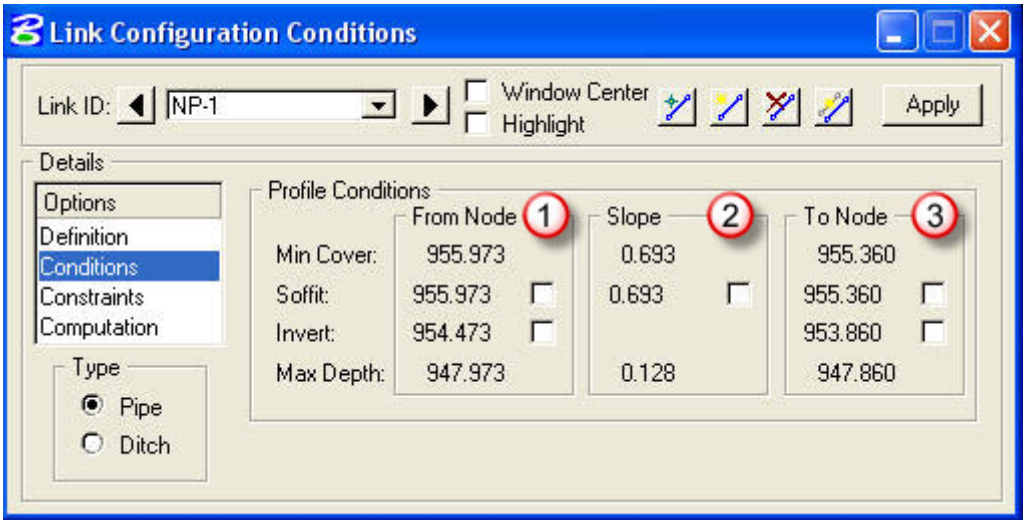
*__Note__: Refer to SCDOT Standard Drawings for fill height limitations for specified pipe. Where possible, keep pipe between the minimum and maximum fill height appropriate for each material. Keeping cover over pipe between 3' and 10' for most installations will allow one of each alternate pipe available to the contractor.

Work with road designer when establishing pipe depth, particularly when pipe will be very shallow (<3' cover) or very deep (>10' cover). Shallow pipe may also present complications with drainage structures (see section V.D.4). Road designer will determine required class, gage, or type based on fill heights listed in SCDOT Standard Drawings.

Pipe Type	Standard Drawing #
RCP	714-205-01, 714-205-02
SRAP	714-605-01, 714-605-02
HDPE	714-705-01, 714-705-02
CAAP	714-810-01, 714-810-02

2. Conditions

Set the Conditions of the link (for the **initial** design only, ***do not*** hold any of the conditions. If you use existing pipes, then they will need to be held):

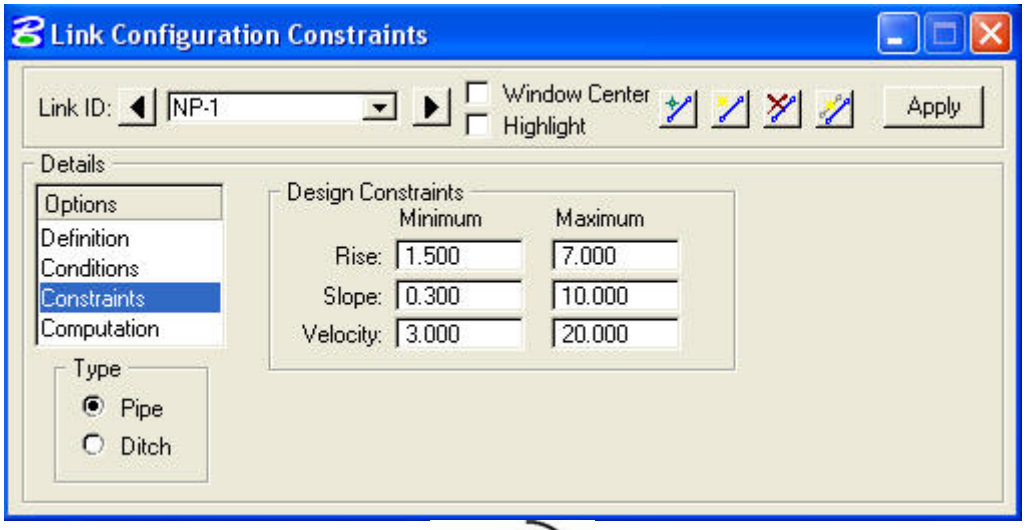


1	From Node	After the initial design, and only if necessary, toggle ON to set and hold the Invert or Soffit of the pipe. It is **better** to hold the Soffit and **do not** hold both.
2	Slope	After the initial design, and only if necessary, toggle ON to set and hold the pipe slope.
3	To Node	After the initial design, and only if necessary, toggle ON to set and hold the Invert or Soffit of the pipe, it is **better** to hold the Soffit and **do not** hold both.

The values for **Min Cover** and **Max Depth** come from the Node elevations.
You can also make adjustments through the **Profile** window.
Use this window to define the downstream outlet pipe elevation; as discussed in **Networks.**

3. Constraints

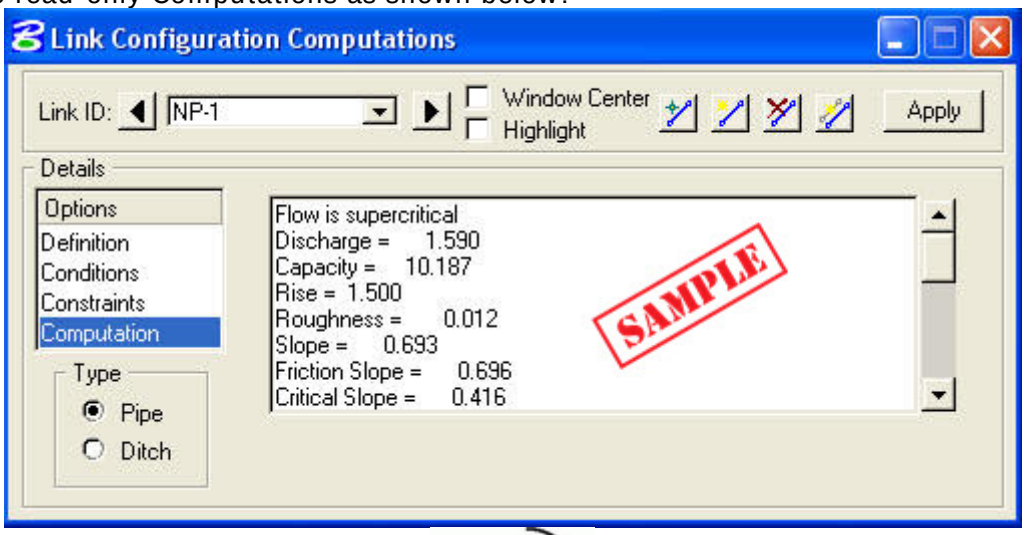
Set the Link Constraints (**use** the values shown below for English units):



These values should **always** be used for the initial design; you may have to change the minimum slope or min/max pipe sizes later, but ***only*** for projects with design exceptions.

4. Computation

View the read-only Computations as shown below:



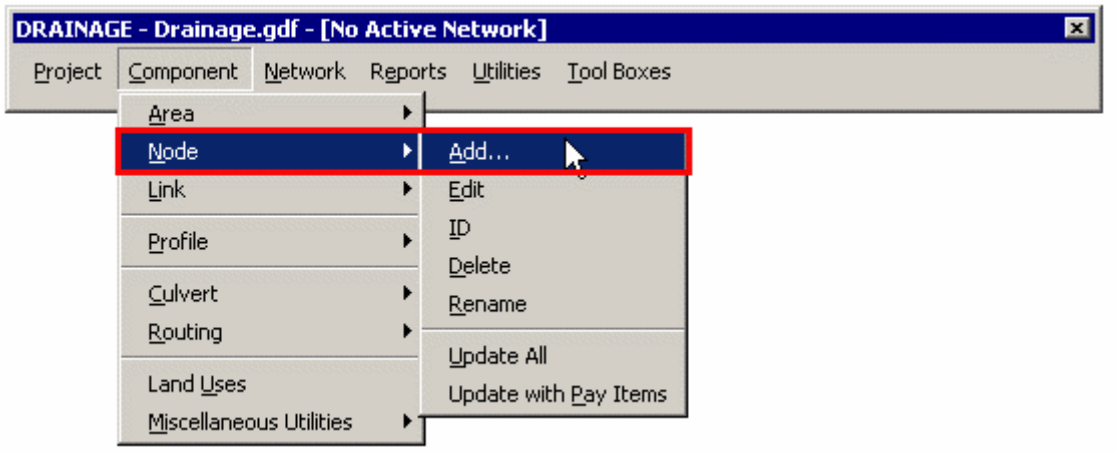
After you add the network, and then perform a design, computations and errors for this individual pipe appear in this window.

Chapter 6: Non-Network Pipes

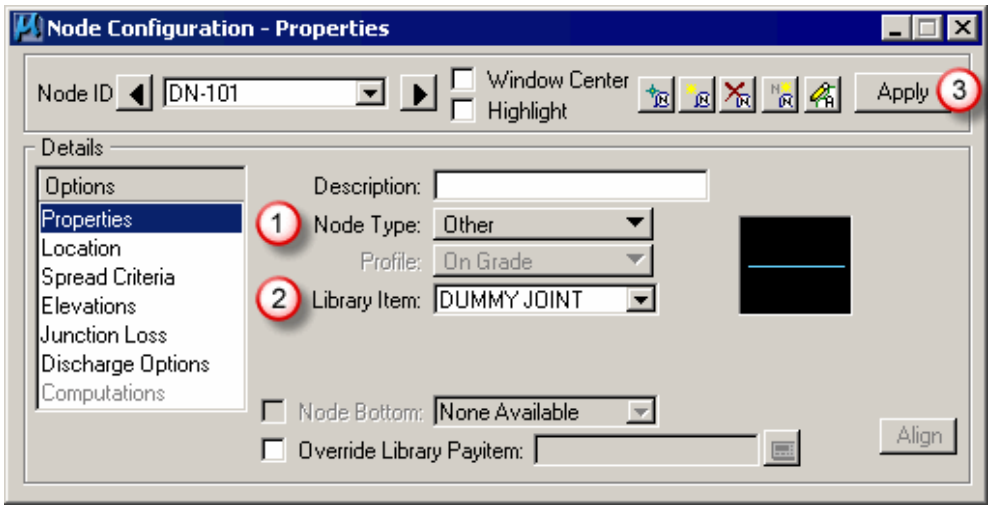
The following procedures are used to draw, label and report on drive pipes or cross pipes.

1. Add Nodes

To add nodes go to ***Drainage>Component>Node>Add***. The "Add a New Node" dialog will open for each Node asking for a Node name prefix and number. The number automatically increases for each node up to 10000. These instructions will place the upstream node for the pipe. Repeat them for the downstream node when you are through.

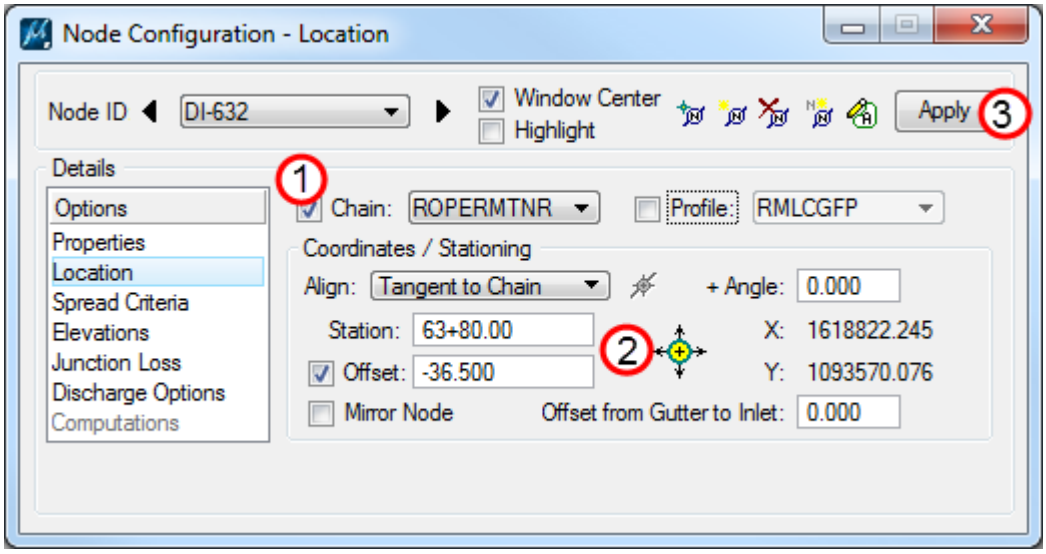


Set the Properties Options:

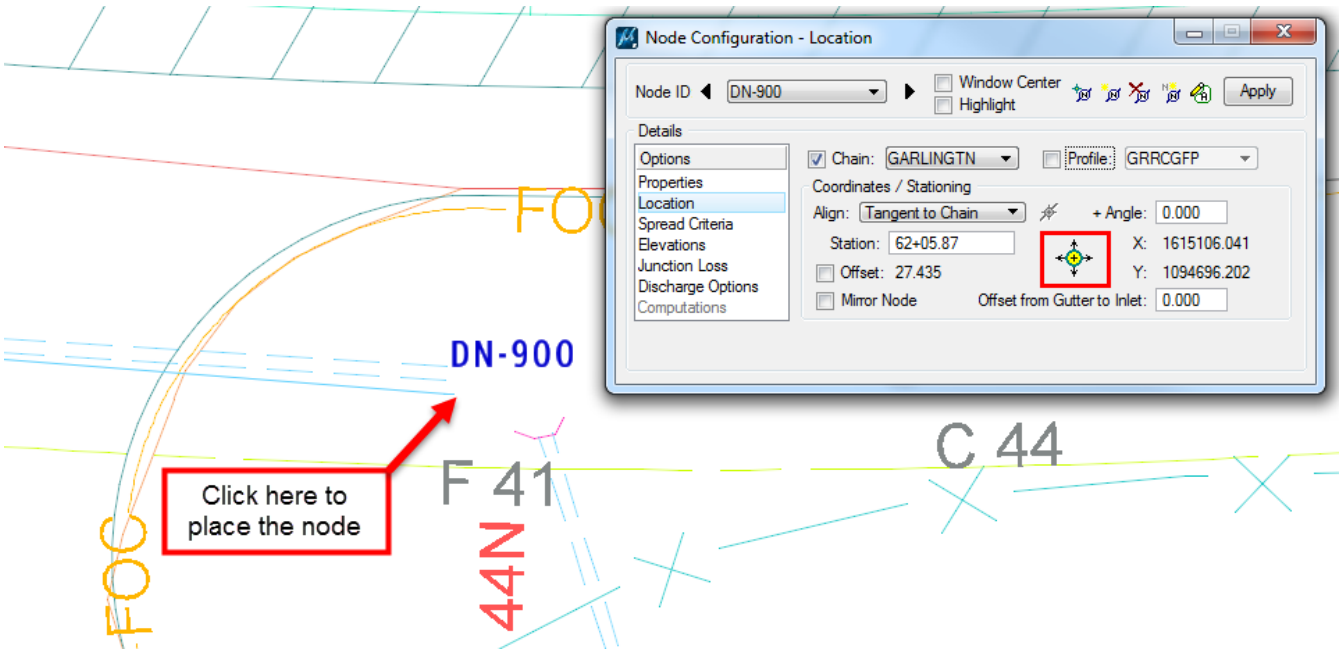


1	Node Type	Set to "Other"
2	Library Item	Set to "DUMMY JOINT"
3	Apply	Click the Apply button to accept the Properties.

Set the Location Option:



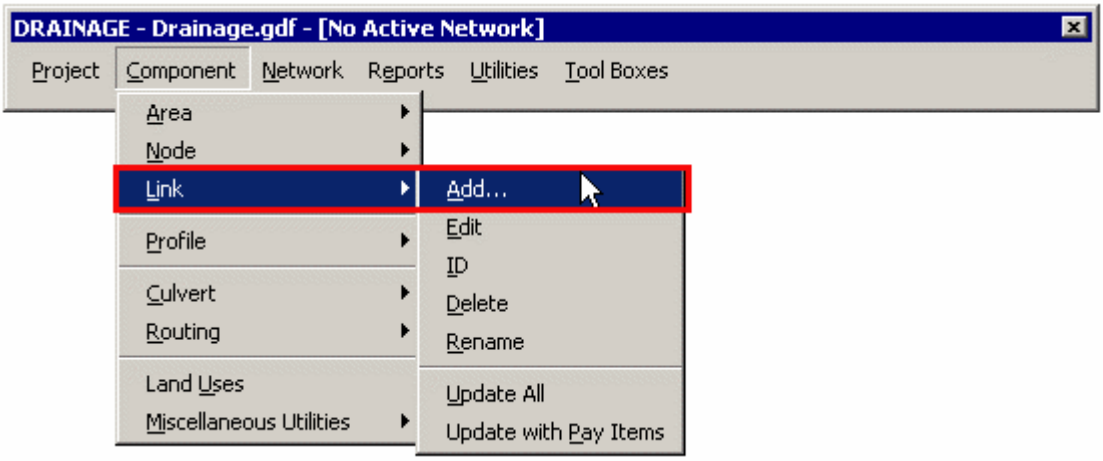
1	Chain	Use the pulldown arrow to select the correct chain name. This information comes from the **GPK File**. If you are placing a node on a side road make sure to use the corresponding side road chain name. Turn off the curb profile option for dummy nodes.
2	Station DP	Click the **Station DP** button, move the cursor to place the upstream dummy joint at the upstream entrance of the driveway pipe.
3	Apply	Click the Apply button to accept the Location.



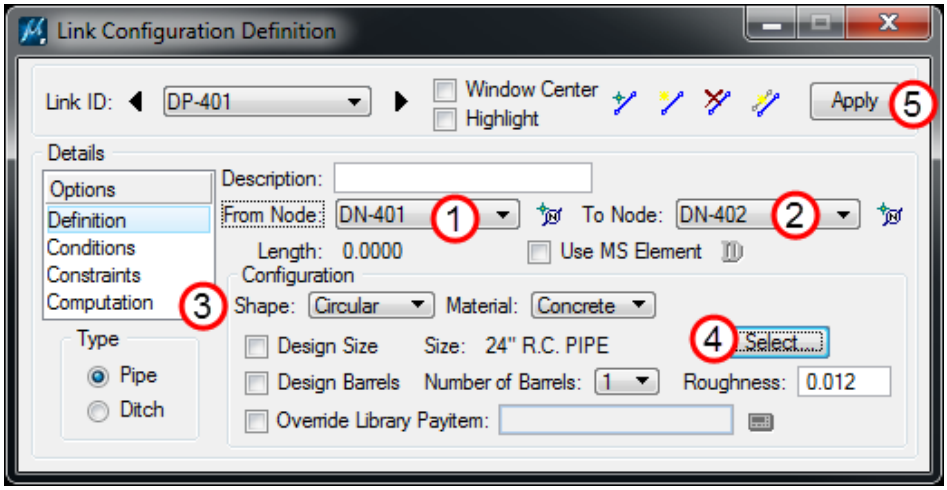
Repeat this **Node>Add** procedure for the downstream pipe location, so you'll have 2 Nodes for each Link.

2. Add Link

To add a link, select **Drainage>Component>Link>Add** and give it a name:



Set the Definition Option:



1	From Node	Select the *Up*stream Node

2	To Node	Select the *Down*stream Node
3	Shape/ Material	Set the Shape to "Circular" and the Material to the appropriate type from the Drainage Library
4	Design Size	Toggle OFF, and click **Select** to pick the size from the Drainage Library
5	Apply	Click Apply to accept the Pipe Definition.

Continue adding Nodes and Links for the rest of the driveway or cross pipes.

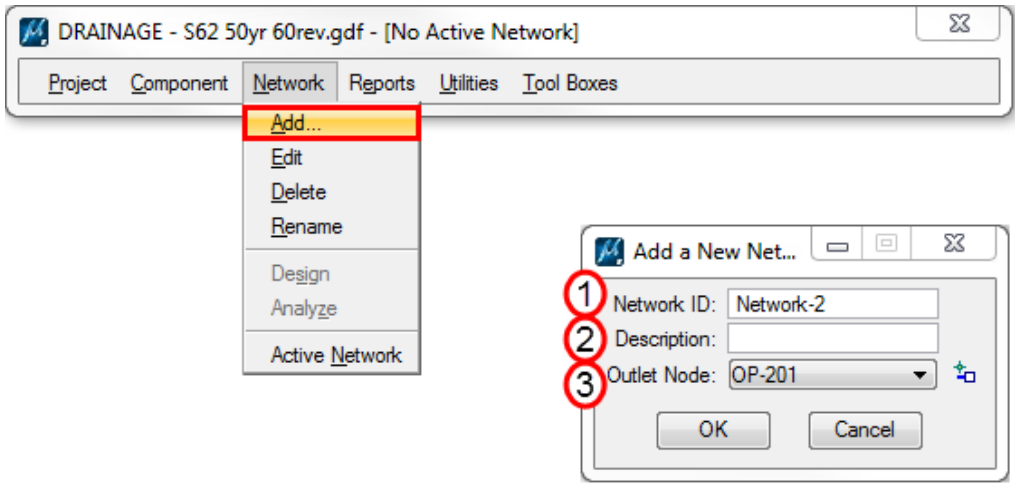
3. Label Plans

The plans are now ready to be labeled using the **Drainage Labeler**.

Since only a few Node and Link variables were set, only certain label variables will be available for driveway or cross pipes.

Chapter 7: Networks

The network includes all areas, nodes and links connected to a single outlet node. GEOPAK Drainage can handle any number of networks in one GDF file.
To add a network select **Drainage>Network>Add**.

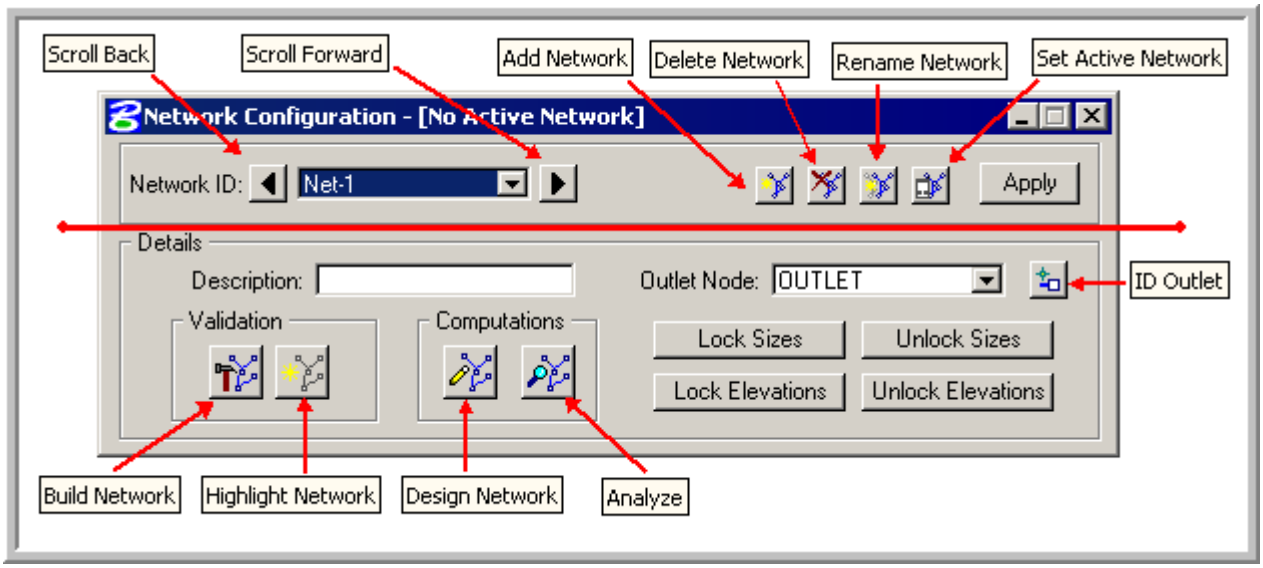


1	Network ID	Name the network based on the names of its Outlet. A network ending at Outlet OP-201 would be Network-2.
2	Description	If there are multiple roads on a project put the road name in the description that matches the outfall station.
3	Outlet Node	Select the Outlet Node connected to the rest of the components contributing flow to this outlet.
Click **OK** to add the Network to the GDF File, else **Cancel**.		

If changes are made to the components the network will need to be redesigned or reanalyzed.

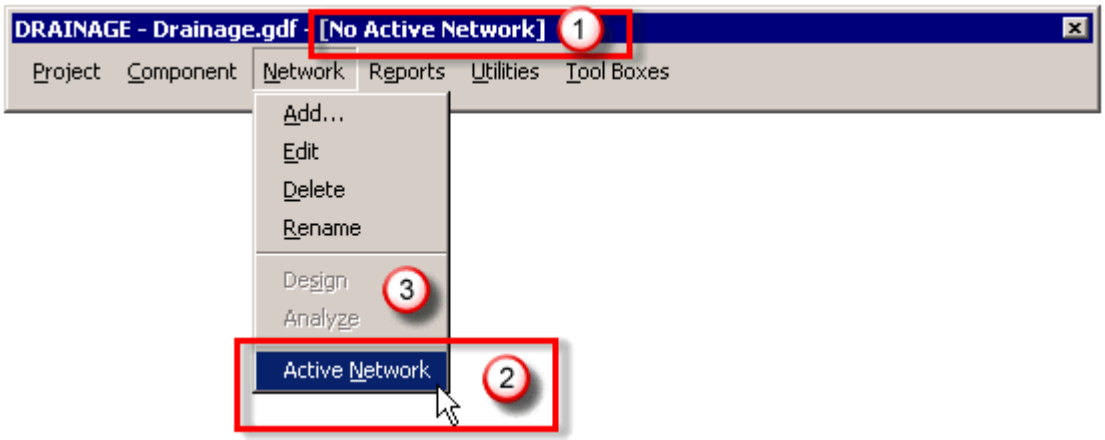
1. The Network Dialog

The Network Configuration dialog is split as shown below; top half for navigating and bottom half (Details) for input of data and computations:



2. Activating Networks

A Network should be activated prior to a Design or Analysis; select *Network* > *Active Network*:



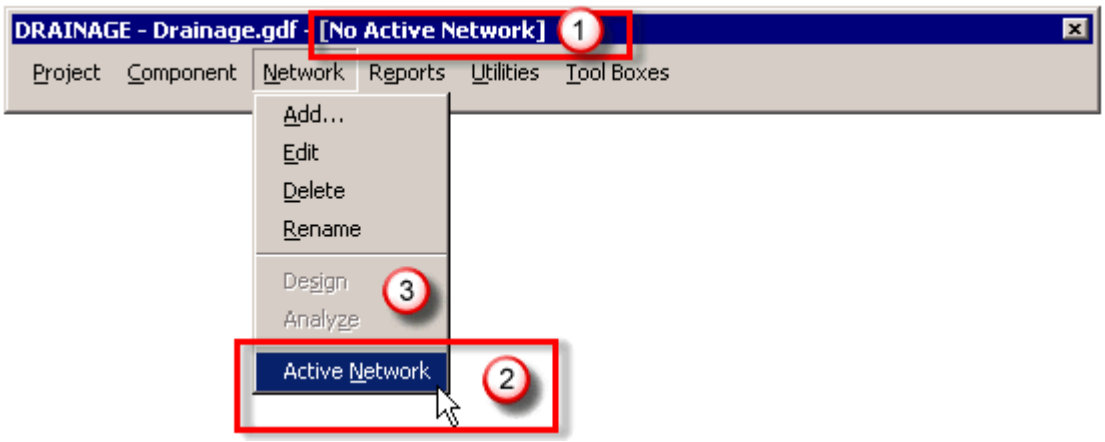
1	Active Network Indicator	This field will display the Active Network
2	Active Network	Select a Network to make Active
3	Design/Analyze	See Design or Analyze

3. Designing Networks

Designing a Network computes the hydrologic and hydraulic parameters of the Network, and ***automatically sets*** any unheld node or link elevations.

There are two (2) methods to Design a Network:

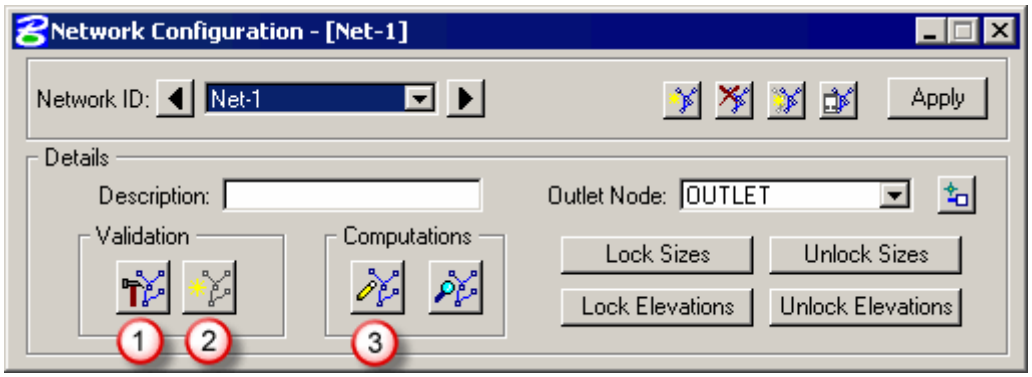
Method 1: From the main menu bar, select *Network* > *Active Network*:



1	Active Network Indicator	This field will display the Active Network
2	Active Network	Select the Active Network on which to perform the design.
3	Design	Click to Design the activated network.

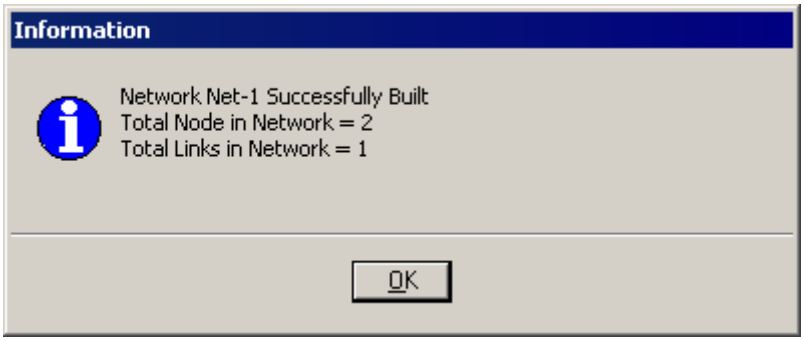
Method 2: Select Network>Edit:

Use the workflow below:



1	Build Network	Click **Build Network** to build the component connectivity contributing flow to the Outlet Node

A "Successfully Built" message should ensue, otherwise see the **errors** section.



2	Highlight Network	*(Optional)* Click **Highlight Network** to MicroStation-highlight all the components contributing flow to the Outlet Node.
3	Design Network	Click **Design Network** to design the components of the GDF project.

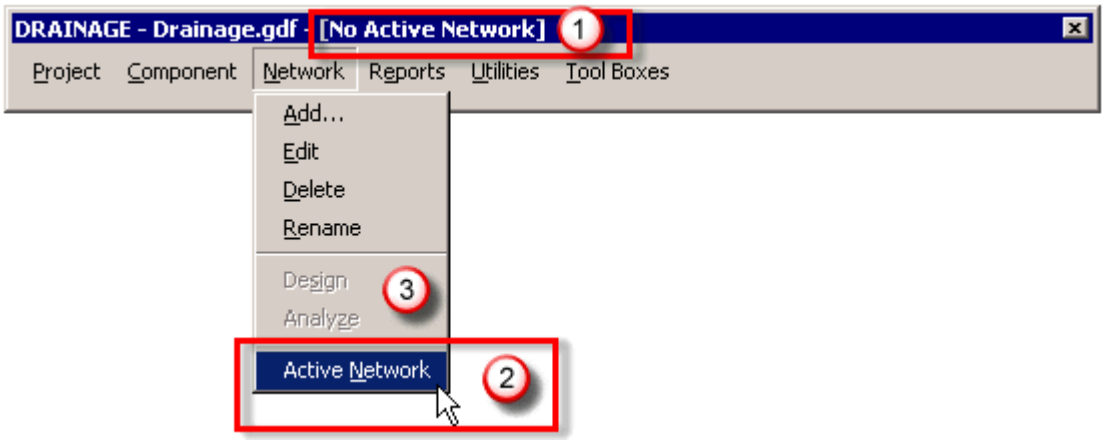
A "Successfully Computed" message should ensue, otherwise see the errors section. Drainage Warning/Error messages are reported for all components in the GDF project and are stored in the drgmsg.txt file in the Working Directory.

4. Analyzing Networks

Analyzing a Network computes the hydrologic and hydraulic parameters of the Network, but does ***not*** set any unheld node or link elevations and is useful for determining effects of various storm events on existing networks.

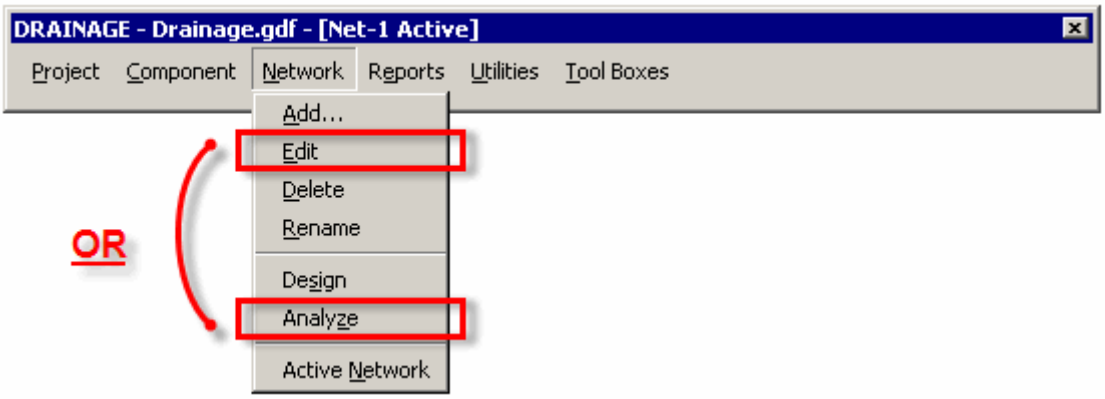
There are two (2) methods to Analyze a Network:

Method 1: From the main menu bar, select *Network > Active Network:*

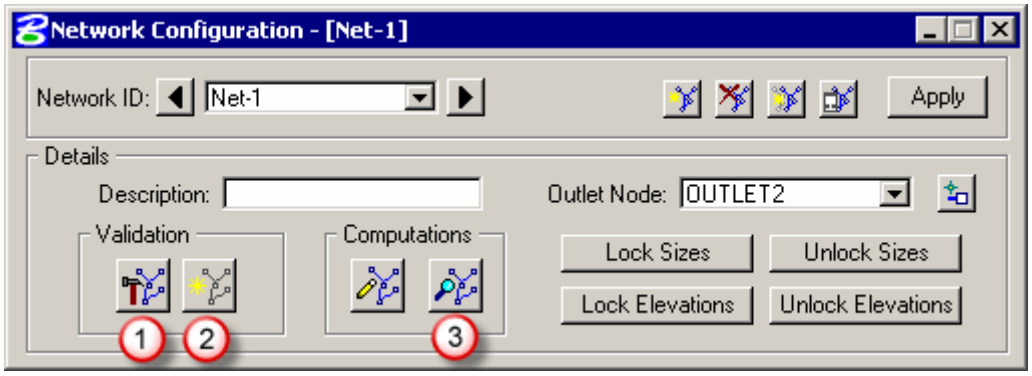


1	Active Network Indicator	This field will display the Active Network
2	Active Network	Select the Active Network on which to perform the design.
3	Analyze	Click to Analyze the activated network.

Method 2: Select Network > Edit:

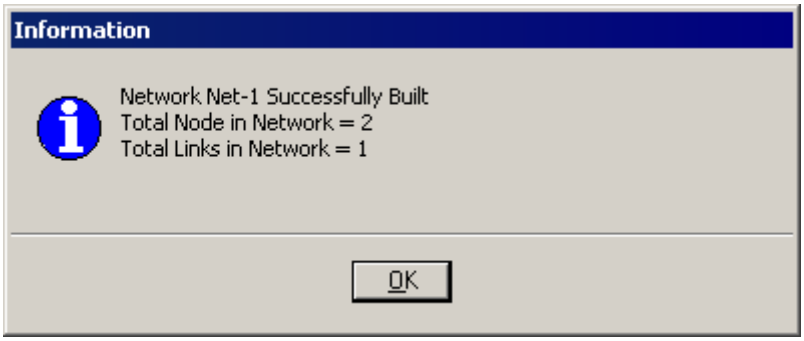


Follow the workflow below:



1	Build Network	Click **Build Network** to build the component connectivity contributing flow to the Outlet Node

A "Successfully Built" message should ensue, otherwise see the errors section.

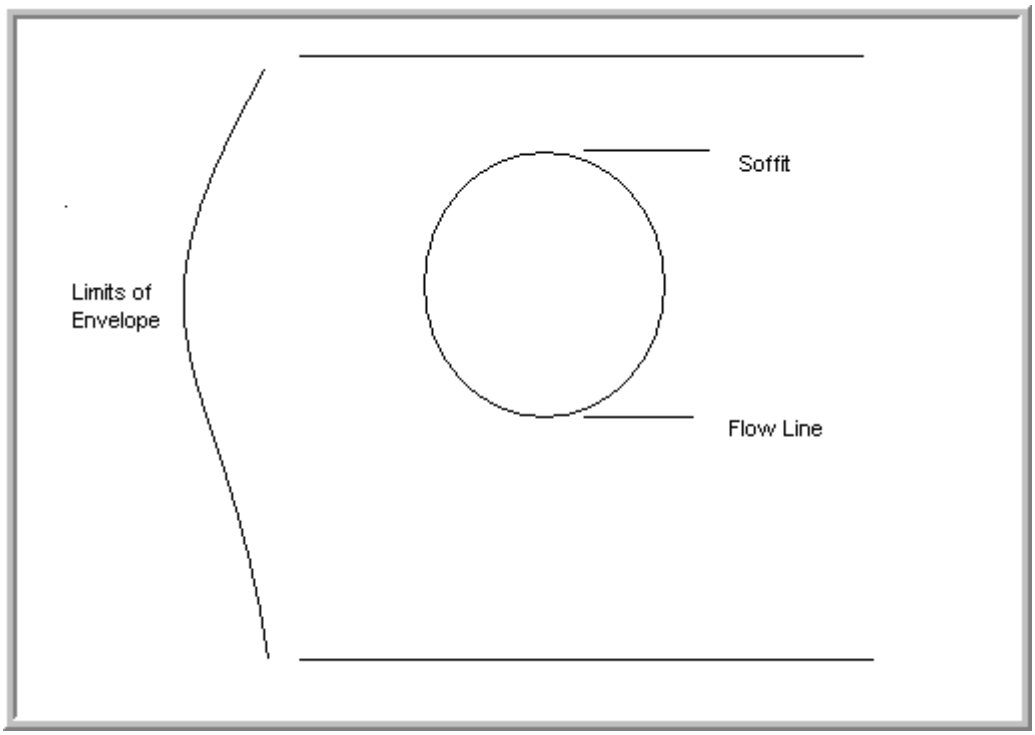


2	Highlight Network	*(Optional)* Click **Highlight Network** to MicroStation-highlight all the components contributing flow to the Outlet Node
3	Analyze Network	Click **Analyze Network** to analyze the components of the GDF project

A "Successfully Computed" message should ensue, otherwise see the errors section. Drainage Warning/Error messages are reported for all components in the GDF project and are stored in the drgmsg.txt file in the Working Directory.

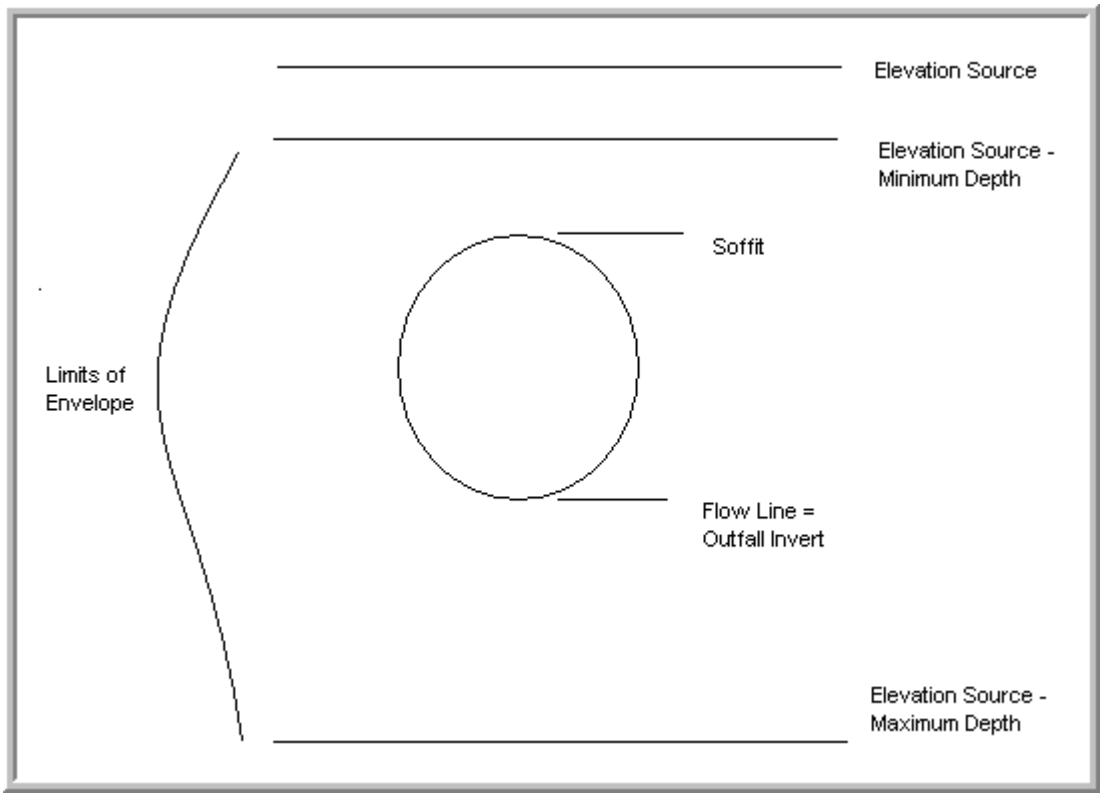
5. Outlet Definition

Several references have been made to defining the outlet in this document. The outlet is the only node where you have a set elevation that you must match. You want the pipe invert equal to the outfall invert. Below is the full explanation on how to do this. GEOPAK Drainage wants an envelope defined for every node. This envelope is a vertical range that Drainage can use to place the pipe:



The pipe, regardless of its size, will fit within these limits that you set. Each node has an envelope, but the outlet envelope is much harder to define.

In the node box under **Elevation**, the **Elevation Source** and **Minimum/Maximum Depth** defines the envelope (Figure 2). The program takes the **Elevation Source** and subtracts the **Minimum Depth**. This is the top of the envelope and the highest elevation where the soffit of the pipe can be located.

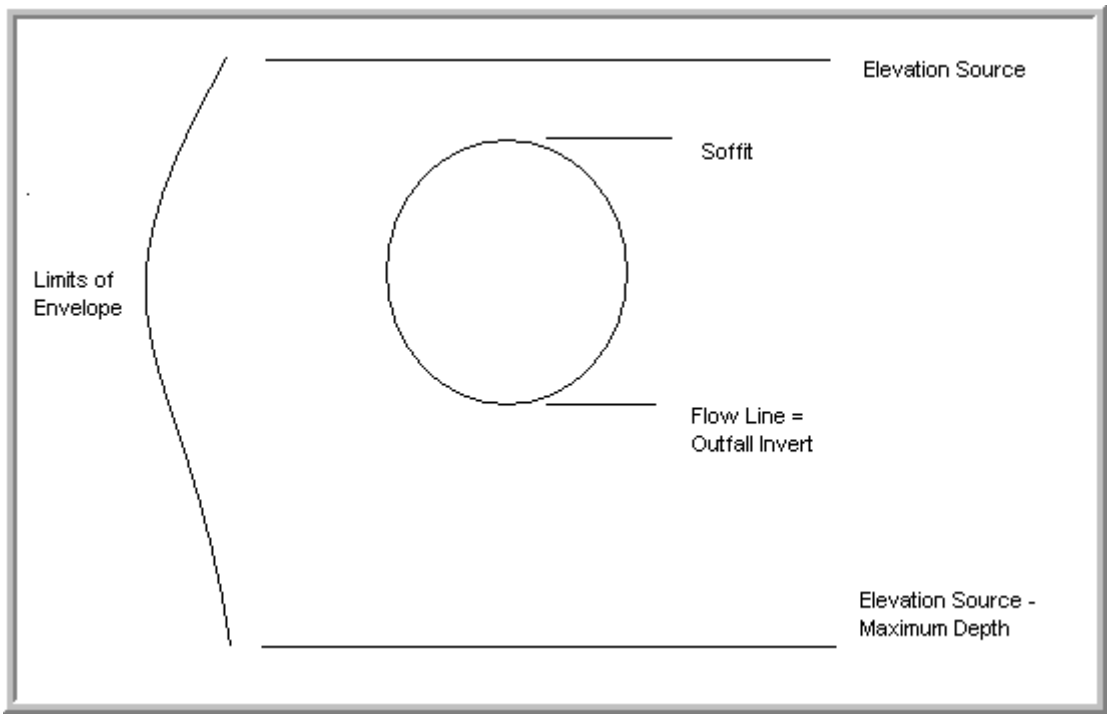


The program then takes the **Elevation Source** and subtracts the **Maximum Depth**. This is the bottom of the envelope and the lowest elevation where the flow line of the pipe can be located. This process is called building the envelope.

For all nodes but the outlet, the **Elevation Source** is easily defined as either the top of curb or ground elevation. The outlet, however, does not have a set elevation above the soffit. The pipe is out of the ground, and the ground elevation is at the bottom of the pipe. The top of the envelope has to be defined with an assumed elevation.

The following steps will aid you in building the envelope for the Outlet:

1. Open the Node window for the outlet.
2. Put 0.00 as the **Minimum Depth** in the **Node** window under **Elevations**. This means the program will not have to subtract anything to get the top of the envelope.
3. Put what you normally would for the **Maximum Depth**. This will define the bottom of the envelope. This will be below the known flow line elevation, but this will not cause a problem.



4. Assume an **Elevation Source**. Most of the time you have a good idea of the outlet pipe diameter by looking at the area draining to your system. If you have 800' of road drainage, an 18" pipe will probably do. In this case you would add 1.5' to the outfall invert or flow line and enter this as the **Elevation Source**.
5. Open the Links box for the last link of our system.
6. Take the elevation that you calculated for **Elevation Source** and enter that value as the downstream soffit elevation in the **Conditions** window.

Run the Design process for the network and check the invert elevation of your outlet pipe. If you guessed a pipe size too large, then your pipe is not flush with your outfall invert. Go back to Links>Constraints and change the soffit elevation. If you guessed a 24" pipe and it designed 18", subtract 0.5' from your original input.

If you guessed a pipe size too small, then your pipe invert is underground. Go back and change your Elevation Source under Node>Elevations. Use this value under Links>Constraints as your new soffit elevation. If you guessed an 18" pipe and it designed a 24", then add 0.5' to your original input.

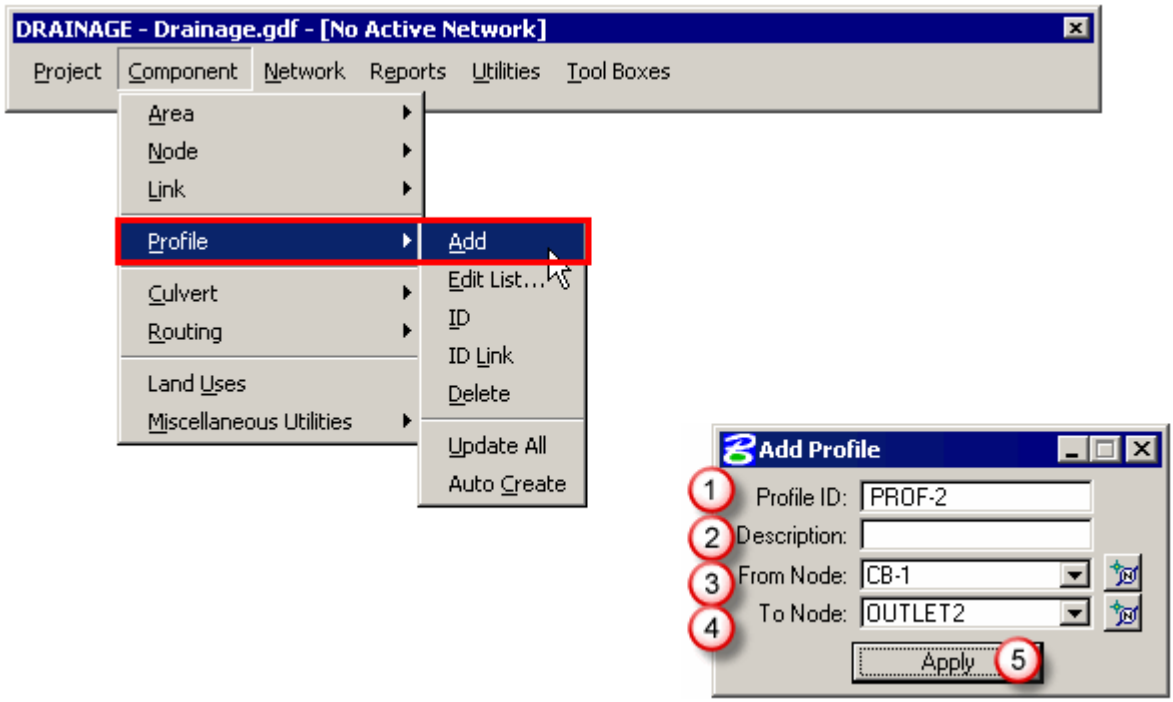
It is possible to have the limits of the envelope above and below the pipe with the pipe not coinciding with either limit.

Chapter 8: Profiles

GEOPAK Drainage can draw the entire profile of a storm sewer very quickly and is an easy way to determine if you are surcharging nodes.

Remember this information is used for evaluation purposes of our storm sewers and ***is not*** the information that the Roadway Designer will be expected to read.

To add and draw a profile, select **Drainage>Component>Profile>Add**:



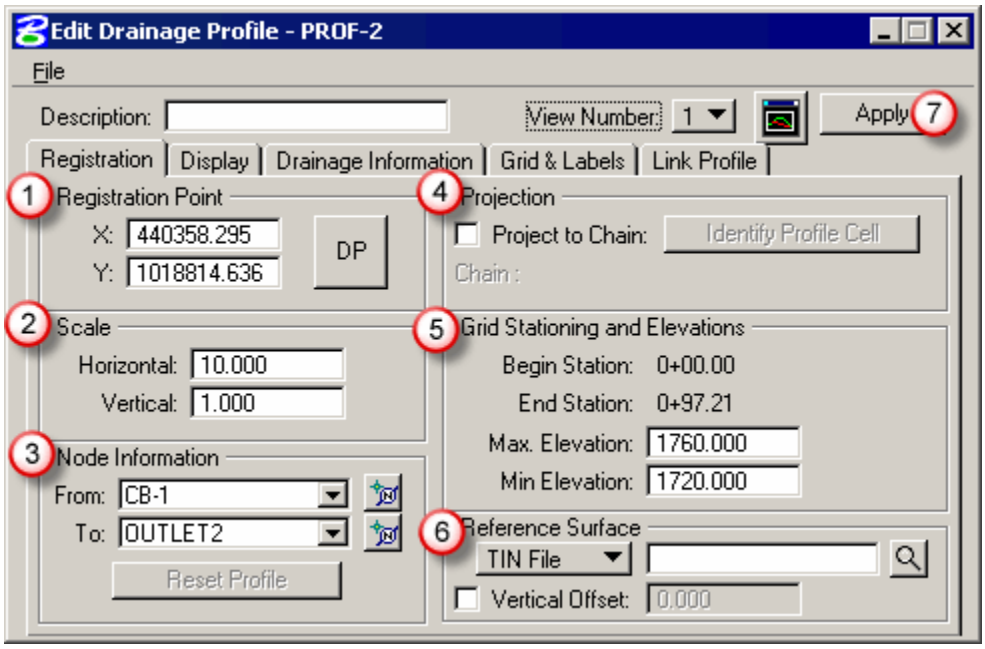
1	Profile ID	Key-in the Profile name (best not to use spaces)
2	Description	*(Optional)*
3	From Node	Leftmost Node on grid and 0+00 station.
4	To Node	Rightmost Node on grid and last station.
5	Click Apply to add the Profile.	

A profile is for a single run of pipe in between any two connected nodes. You ***cannot*** draw a profile for parallel pipes at once; therefore, draw multiple profiles for a storm sewer system with dual trunk lines.

As the profile is drawn to scale, you can measure distances in between lines. These profiles can also be printed through IPLOT.

1. Registration

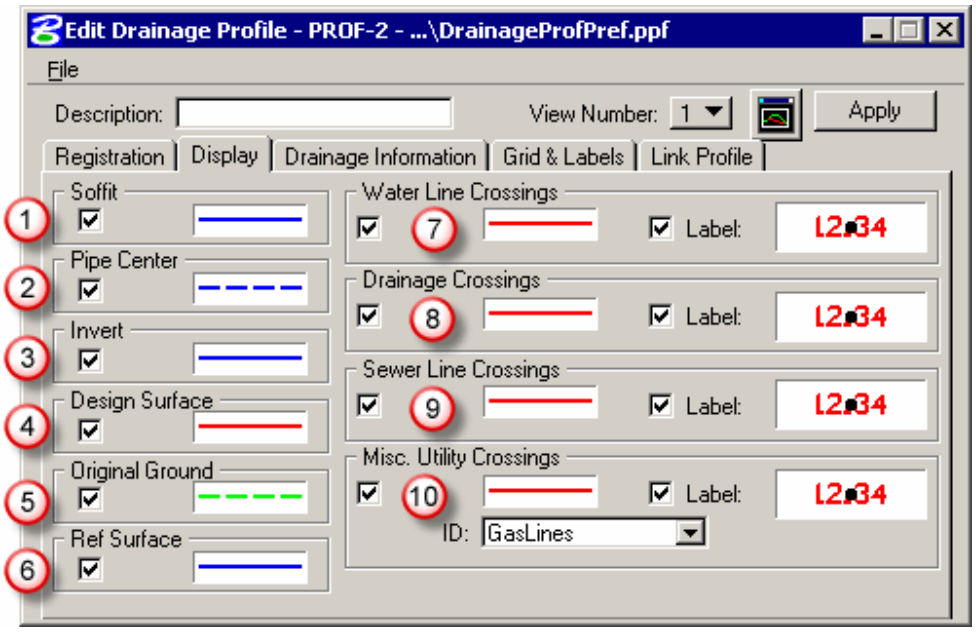
The Registration Point determines the location, scale and elevation of the profile:



1	Registration Point	Set the lower left point (X,Y) on the grid. Click **DP** and click a point in the drawing to place the grid or enter the coordinates manually.
2	Scale	Set the Horizontal & Vertical Scale.
3	Node Information	**From Node** = left-hand on grid and **0+00** station; **To Node** = right-hand on grid and ending station.
4	Projection	n/a
5	Stations& Elevations	Stationing is *read-only*. Set the Max. and Min. Elevations to encompass all profiles and surfaces.
6	Reference Surface	n/a
7	Click Apply to draw the profile in the DGN file.	

2. Display

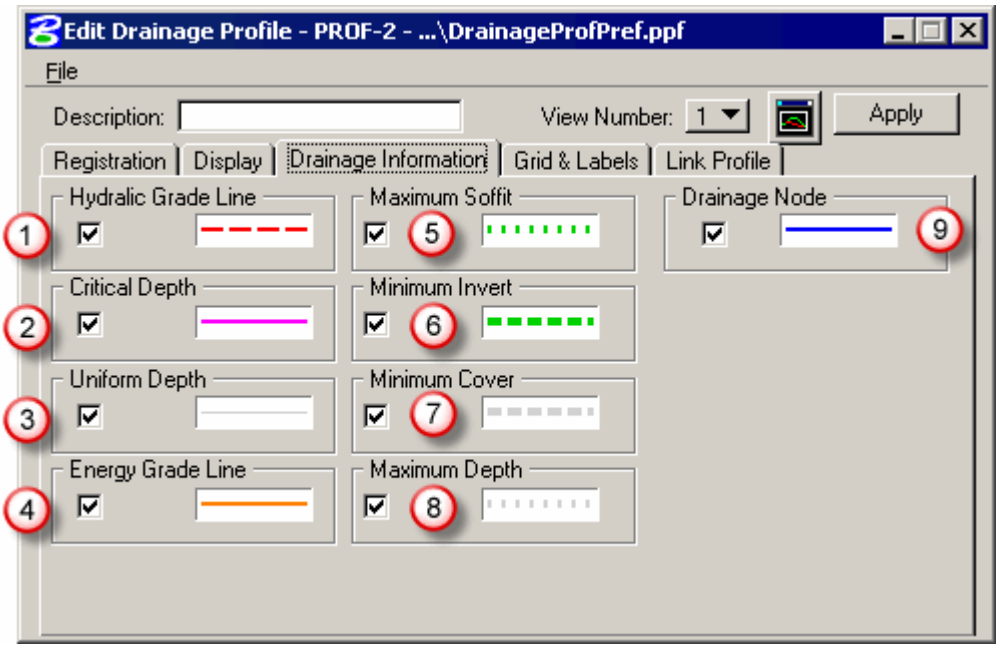
Set the Profile's display. Make sure they are all on Level RD_HY_Gpkprofile, but choose whatever color and weight you want.



1	Soffit	Top of Pipe
2	Pipe Center	Center of Pipe
3	Invert	Bottom of Pipe (flowline)
4	Design Surface	Proposed Finished Grade (TIN or Model) located in the Preferences
5	Original Ground	Existing Ground (TIN or Model) located in the Preferences
6	Ref Surface	(Optional) 3rd surface from the Registration Tab
7	Water Line Crossings	n/a
8	Drainage Crossings	Toggle ON to show the drainage crossings
9	Sewer Line Crossings	n/a
10	Misc. Utility Crossings	n/a
Click Apply to modify the Profile.		

3. Drainage Information

Select the drainage information to be displayed in the profile view. Make sure they are all on Level RD_HY_Gpkprofile but choose whatever color and weight you want.



1	Hydraulic Grade Line	Auto-generated Hydraulic Grade Line
2	Critical Depth	Auto-generated Critical Depth
3	Uniform Depth	Auto-generated Uniform Depth
4	Energy Grade Line	Auto-generated Energy Grade Line
5	Maximum Soffit	Maximum elevation of top of pipe
6	Minimum Invert	Minimum elevation of bottom of pipe
7	Minimum Cover	Minimum cover on top of pipe
8	Maximum Depth	Maximum depth of bottom of pipe
9	Drainage Node	Node display symbology in profile view
Click Apply to modify the Profile.		

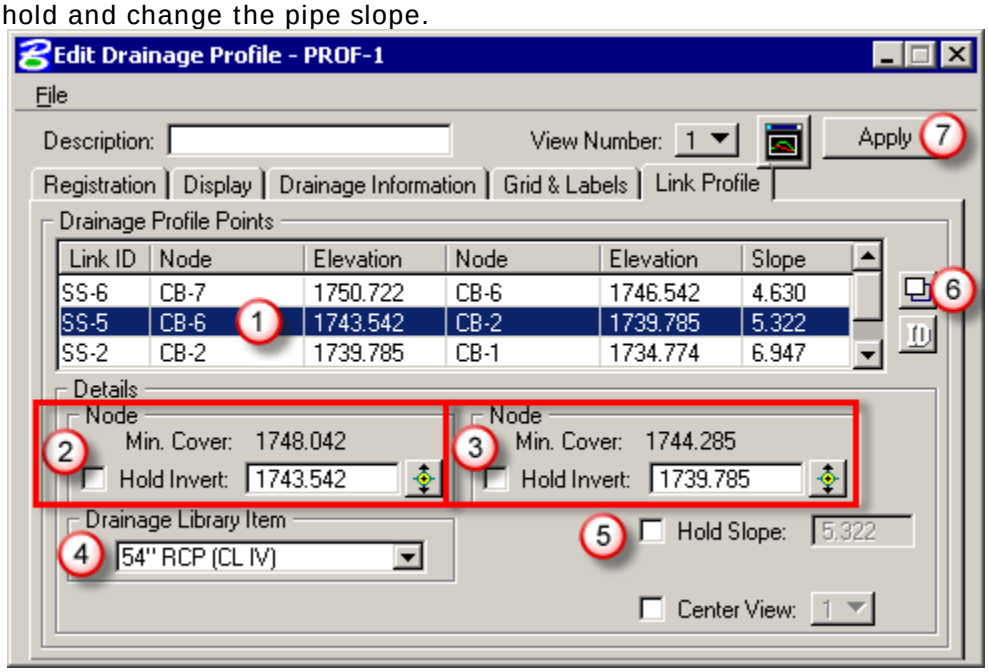
4. Grids & Labels

Set each option on an individual basis.

Pick from the options and choose whatever looks best to you. You can easily clutter the profile when choosing too many options; so make sure the information chosen will look good printed.

5. Link Profile

This is a powerful tool allowing you to modify the pipe invert elevations and slopes. In the top portion you will see all the pipes in the profile. Highlight a pipe and its invert elevations appear under Details>Node. From there you can manually change the invert and hold it or you can click the Edit Invert button and move the invert on the profile. You also have the option to hold and change the pipe slope.



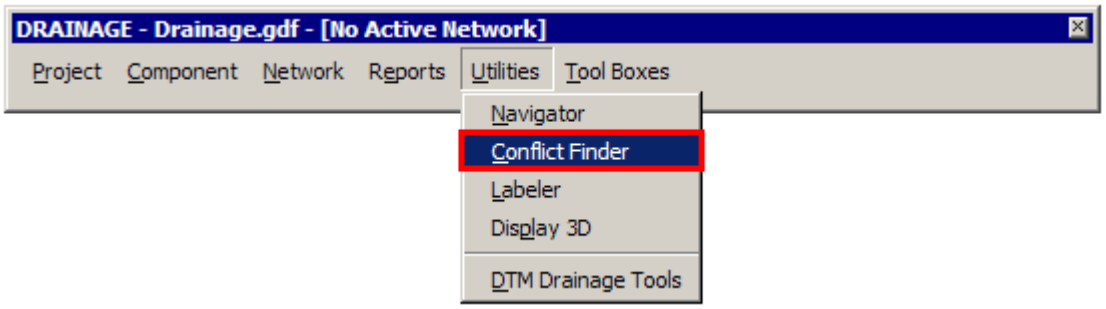
1	Link ID	Highlight the Link ID in the list
2	Node	Use the button to dynamically adjust the vertical invert in profile view; or key-in the "Hold Invert" elevation then toggle ON "Hold Invert".

3	Node	Use the button to dynamically adjust the vertical invert in profile view; or key-in the "Hold Invert" elevation then toggle ON "Hold Invert".
4	Drainage Library Item	n/a
5	Hold Slope	Toggle ON and key-in a slope value.
6	Modify	Click the Modify button to change the Link.
7	Click Apply to accept the revision, redesign the Network, and redraw the Profile.	

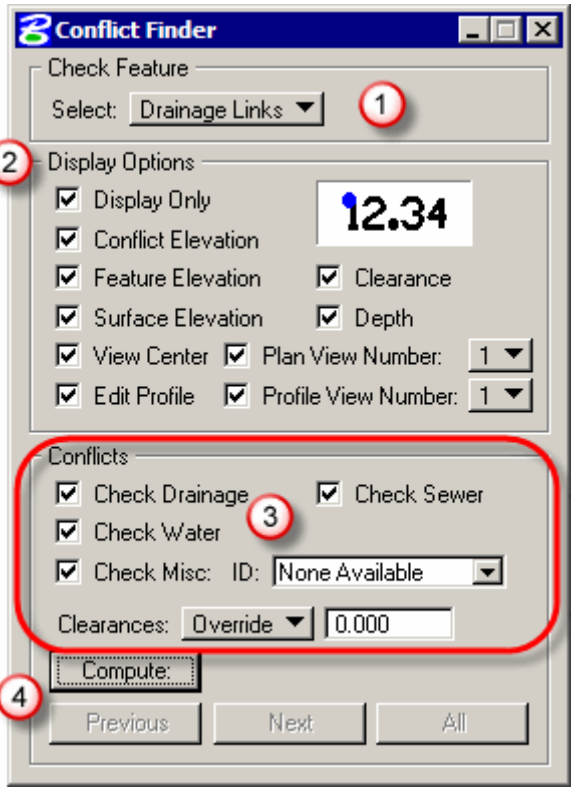
If a change is made, the Network should be redesigned or reanalyzed.

6. Conflict Finder

Use Conflict Finder to check for vertical piping conflicts. From the main menu bar, select *Utilities* > *Conflict Finder*:



To assess conflicts, set the Conflict Finder using the workflow shown below:



1	Check Feature	
	Select: Single Feature	Click the **ID** button and *data point on* a single Drainage Link to check.
OR	Select: Drainage Links	Checks all Drainage Links for all crossings.

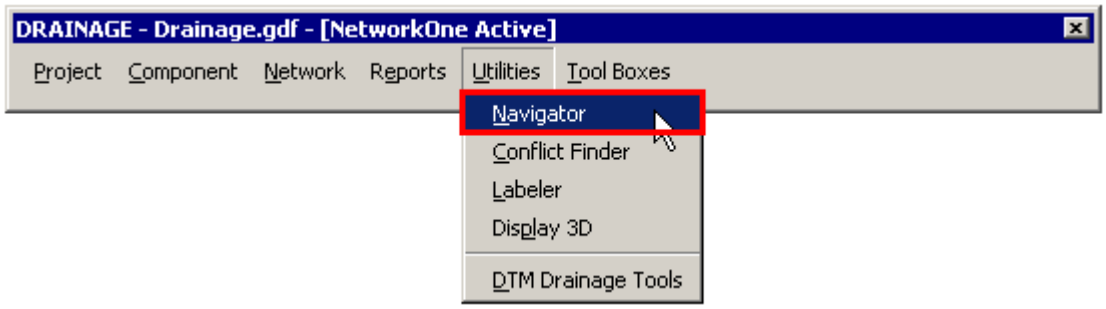
2	**Display Options** *(All Optional)*	
	12.34	Double-click to activate the control and set the symbology for the remaining display options.
	☑ Display Only	Views the conflict data information in temporary display mode.
	☑ Conflict Elevation	Displays the vertical elevation of the utility crossing in profile view.
	☑ Feature Elevation	Displays the vertical elevation of the main crossing in profile view.
	☑ Surface Elevation	Displays the vertical elevation of the proposed surface you stored in the preferences.
	☑ View Center	Centers the selected MS window on the conflict in plan view and/or profile views.
	☑ Edit Profile	When toggled ON, and Conflicts are found, the Drainage Profile Edit dialog opens to enable on-the-fly conflict resolution.
	☑ Clearance	Displays the vertical clearance (feet or meters) between the crossings.
	☑ Depth	Displays the vertical depth (feet or meters) at the crossing location.
	☑ Plan View Number: 1	Centers the MS plan view on the crossing.
	☑ Profile View Number: 2	Centers the MS profile view on the crossing.
3	**Conflicts Options**	
OR	☑ Check Drainage	Checks for Drainage pipe crossings.
OR	☑ Check Sewer	Checks for Sewer pipe crossings stored in the Drainage Preferences option.
OR	☑ Check Water	Checks for Water pipe crossings stored in the Drainage Preferences option.
OR	☑ Check Misc: ID: Gas Line	Checks for Misc. Utility pipe crossings.
4	**Compute Options**	
	Compute: 1 of 2 Conflicts	Click **Compute**, the # of Conflicts found will be displayed.
	Clearances: Override 10.000	Use **Default** for Sewer clearance value you stored in the sewer line dialogs or the water line dialog, or use **Override** and supply a vertical clearance in (feet or meters).

OR	Previous	Displays the **Previous** crossing location
OR	Next	Displays the **Next** crossing location
OR	All	Displays **All** conflicts

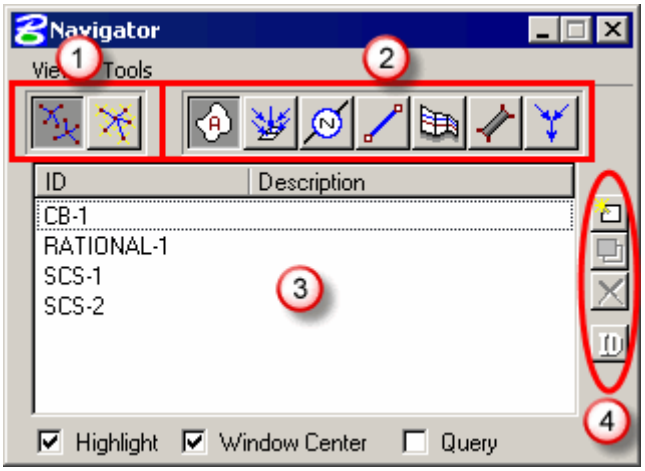
Profiles need to be drawn into the DGN file before using this tool.
To remedy conflicts, toggle **ON** "Edit Profile" and use the Link Pipe Profile tab to modify vertical elevations as needed.

Chapter 9: Navigator

The Navigator is a powerful tool to aid you in maneuvering through a project. You can also add, edit, delete and identify all drainage components from the Navigator:



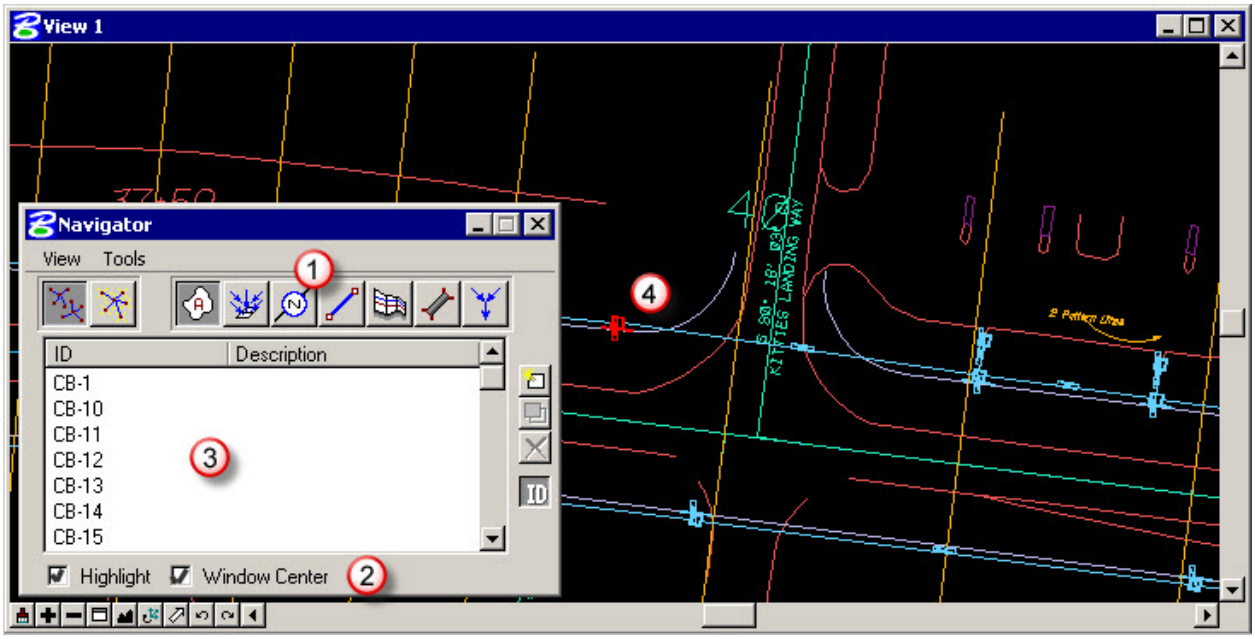
The Navigator has the following toolsets:



1	Networks	Choose to navigate through "All Networks" or only the "Active Network".
2	Components	Options include: Areas, Inlets, Nodes, Links, Profiles, Culverts and Routings, displayed left to right.
3	ID List	The list of components available by the selection in bullet #2 above.
4	Action buttons	Click to add / modify / delete / or ID any component for modifications.

1. Navigating

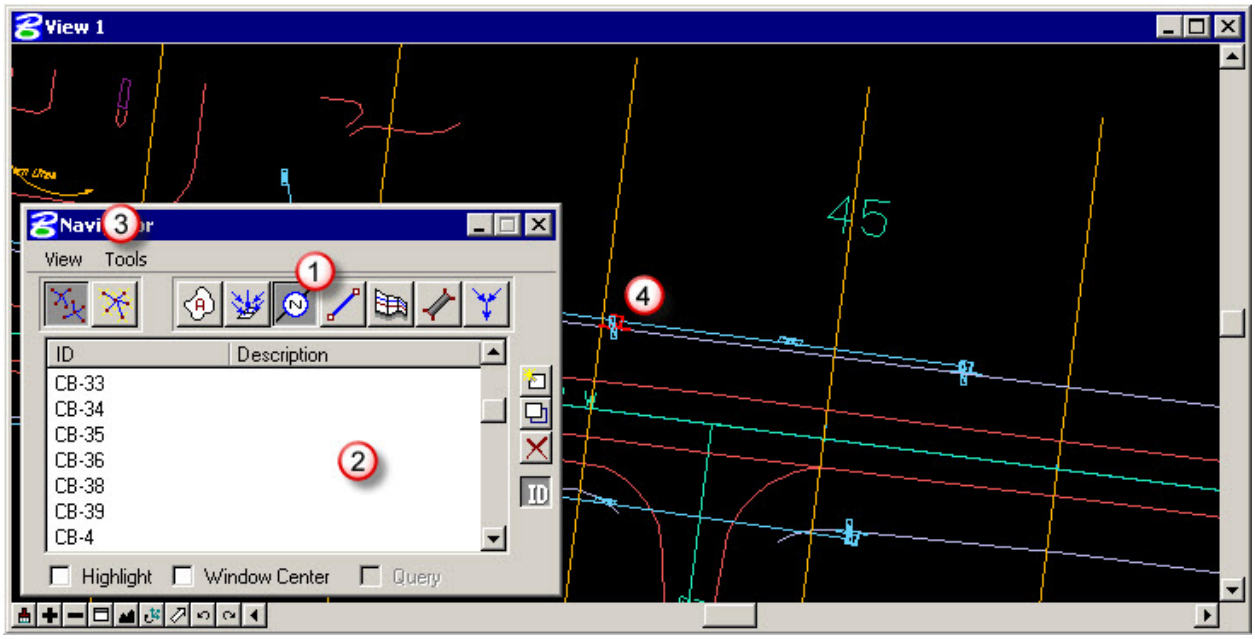
The Navigator window allows you to click through the lists of drainage objects to identify and modify the components you are interested in.



1	Type	Select the component type of interest – only that type will be shown in the components list.
2	Toggles	Toggle ON so the MicroStation view will highlight and center on a drainage component when it is selected.
3	Component	Click any component in the list.
4	View	The Microstation view updates to window center and highlight the selected drainage component.
5	Repeat as necessary to navigate the components	

2. Updating Graphics

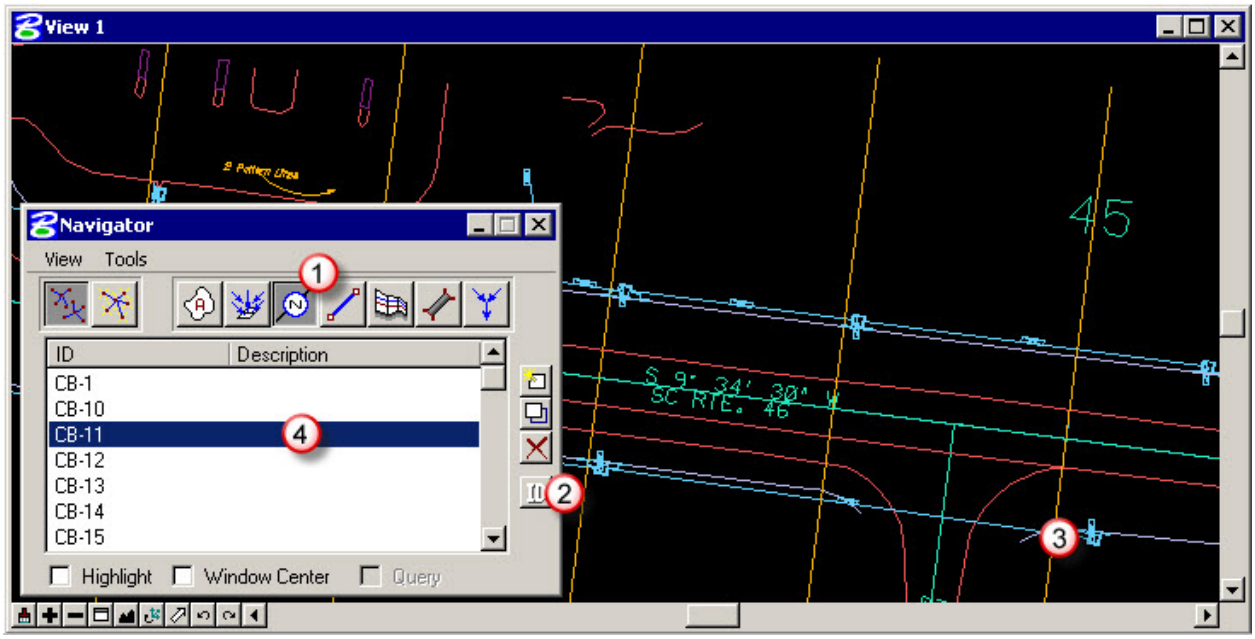
After editing the drainage components, you must update their graphics in the Microstation file.



1	Type	Select the component type of interest.
2	Selection	Select all (or any you wish) components in the list for graphical updating.
3	Tools	Select Tools > Update Graphics to commence the procedure.
4	View	The Microstation view updates with the revised graphics.
5	Repeat as necessary to update the graphics on the components.	

3. Identifying Items

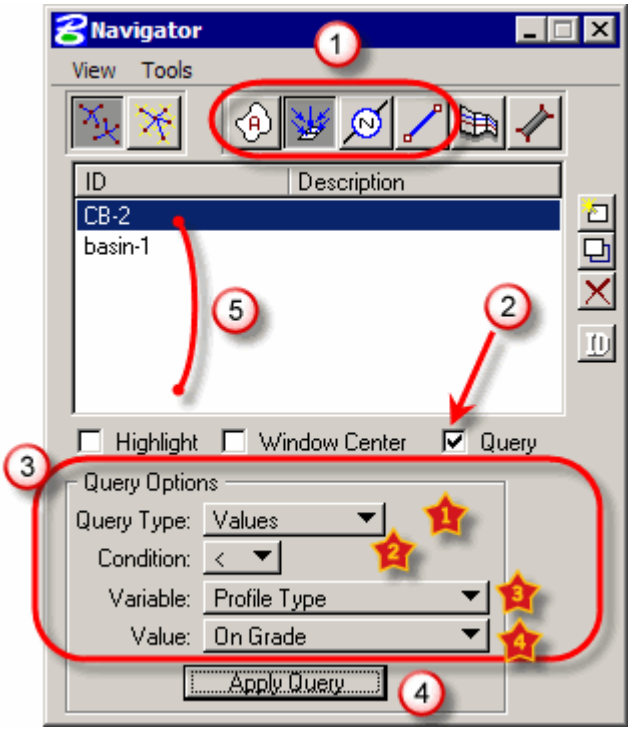
Navigator can also be used to identify a drainage component from its graphic in the Microstation file.



1	Type	Select the component type of interest.
2	ID	Click the ID button
3	Select	Select any component in the DGN file
4	Navigator	Notice the specific component is located and highlighted in the Navigator window
5	Repeat as necessary to identify the drainage components.	

4. Query Mode

Query Mode allows you to set several requirements and search the project for any drainage components that meet them. Use the workflow below to utilize the Query Mode:



1	Select the Component Type	
		Use Query Mode to search for Areas
OR		Use Query Mode to search for Inlets
OR		Use Query Mode to search for Links

2	Toggle ON "Query" mode

3	Query Options	
1	Query: Values	A **Value** is usually one of two types: "alpha" (i.e. pipe material) or "numeric" (i.e. pipe slope).
OR	Query: Constraints	A **Constraint** is usually one of two types: "min" (i.e. minimum depth) or "maximum" (i.e. maximum depth).

2	>	Condition = "greater than"
OR	=	Condition = "equals"
OR	<	Condition = "less than"
3	Variable	If the Query Type is set to 'value', a variable may be available.
OR	Constraint	If the Query Type is set to 'constraint', a constraint may be available.
4	Value	Value mode is usually one of two types "alpha" (i.e. curb) or "numeric" (i.e. curb length)

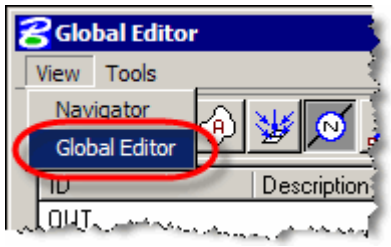
4	**Apply Query**	
	Apply Query	Click **Apply Query** to initiate the computations

5	**Navigator Window**	
	Query Applied	The components ***remaining*** in the Main Navigator Window meet the results of the applied Query.

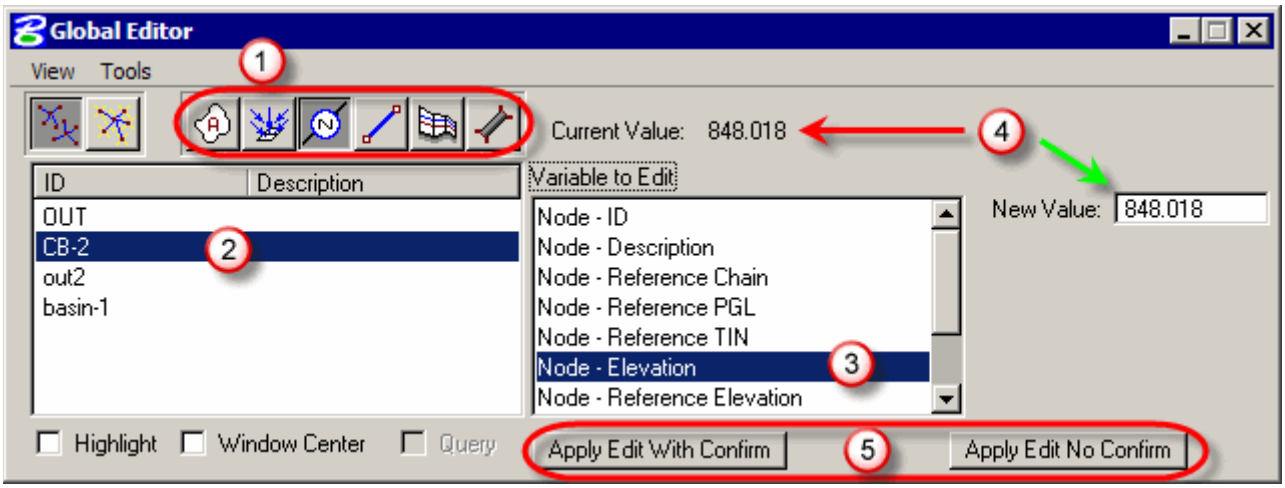
After a Query is applied, the Navigator Window will ***continue*** to display only the results of the Query, until it is reopened, or another Component Type is selected.

5. Global Editor Mode

The Global Editor mode expands the Navigator to let you edit components directly from the Navigator window. From the Navigator's View pull-down menu, select Global Editor:



Use the workflow below for Globally Editing components:



1	Select the Component Type:	
	Components	Only **Areas**, **Inlets**, **Nodes** or **Links** are available for Global Edits.

2	Select the Component(s):	
	Keyboard Input	Highlight (use keyboard shortcuts) single, multiple, or all components on which to perform the Global Edit(s).

3	Variable to Edit	
	Variable to Edit	Select any one of the available **Variables** to Edit.

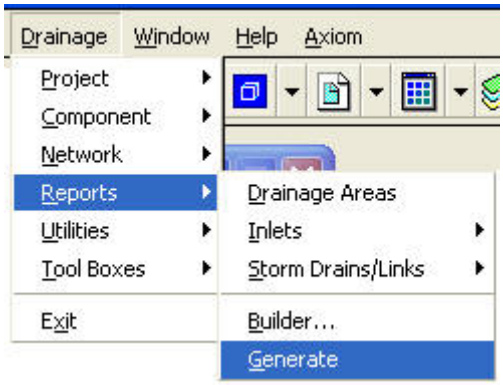
4	Current / New Value	
Current Value	Current Value	Displays the current value of the selected variable in the step above.
New Value	New Value: 849.983	**Key-in** the new value of the Variable.
Overwrite OR Find/Replace	Overwrite Text: New Text: existing.tin Find and Replace: Find Text: existing.tin Replace Text: proposed.tin	Either "Overwrite Text" with "New Text", **OR** Use "Find Text" / "Replace Text" feature.

5	Apply Edits	
	Apply Edit with Confirm	Click to make the changes without any confirmation warning messages.
OR	Apply Edit No Confirm	Click to make the changes with a confirmation warning message for ***each*** component selected in bullet #2 above.

Use the same workflow above (for any available variables) for Areas, Inlets, Nodes, or Links.

Chapter 10: Reports

Under the Drainage>Reports menu, the three top options are summary reports that GEOPAK Drainage automatically generates. To print these reports you must open one of the three reports, press Alt-PrtScn, then open Paint and press Ctrl V or open Microsoft Photo Editor and choose Edit>Paste as New Image. You can print directly from either program. To generate a specific report you must open the **Report Builder**. This is where you can generate custom reports for system evaluation.



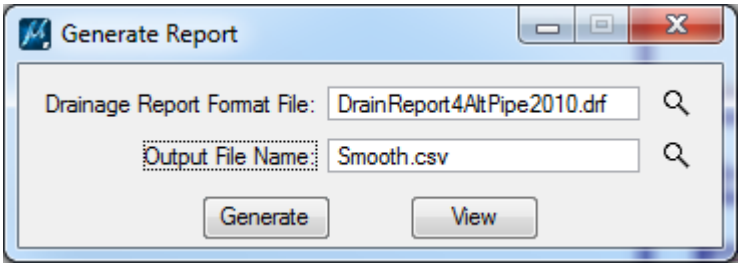
1. Alternate Pipe

The alternate pipe spreadsheet is used to organize drainage network data for inclusion in plan sets. The spreadsheet itself can be found on the **Preconstruction intranet webpage** under "Design Tools" listed as **Alternate Pipe Editor**. The information for generating the necessary reports is located in the **DrainReport4AltPipe2010.drf** file in **\\nts\hq\precon-general\Hydraulics\Standard_Files_GEOPAK_2009\GEOPAK_Output\Output_drf_files**.

2. Generate

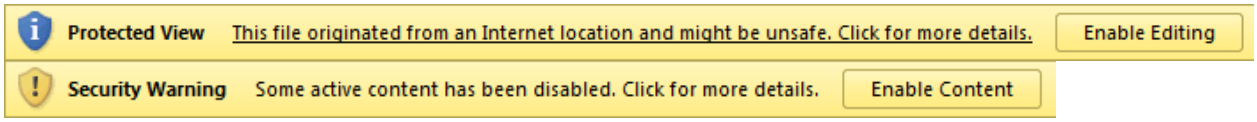
To export the drainage system data from Geopak, select **Drainage>Reports>Generate**. Make sure you have an active network selected. By **Drainage Report Format File** choose the preset **DrainReport4AltPipe2010.drf**.

By **Output File Name** choose your working directory and name the file whatever you desire, but change the extension to csv. Select **Generate** and you will receive a message stating the file has been generated.

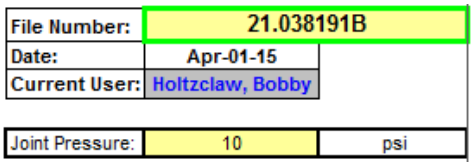


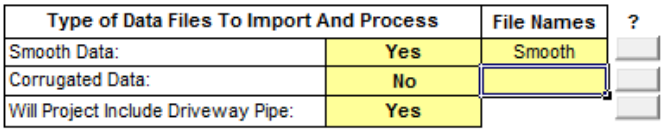
3. AltPipe Spreadsheet

Copy the AlternatePipeEditor spreadsheet from the Preconstruction intranet webpage to your working directory and open it. You will need to authorize it to use Macros and possibly to confirm that files from the internet are allowed.



Fill in the File Number on the Input tab of the spreadsheet. The Joint Pressure cell should be 13 psi for coastal regions and 10 psi for all others unless otherwise indicated for the project in question. Set Smooth Data to Yes and Corrugated Data to No. If Driveway Pipes are to be included in the final report, set that field to Yes. Then enter the name of the .drf file created in the previous section next to the Smooth Data field under File Names.





Press the "Import Geopak Data" button to fill in the network data from the .CSV file. The leftmost column of data, System ID, must be filled in manually. For each Link, enter the point number of the outlet of its system. The SPT "N" values must be entered manually by the geotech. Once both columns have been filled out, press the "Generate Hydro Output Tables" button and enter a file name to create the final reports.

4. Standard Interoffice

The information for our standard interoffice reports and Roadway Designer reports has been preset in a set of DRF files found on **\\nts\hq\precon-general\Hydraulics\Standard_Files_GEOPAK_2009\GEOPAK_Output\Output_drf_files**: - nodestandard.drf, linkstandard.drf, areastandard.drf, nodestandardroad.drf, linkstandardroad.drf, nodesum.drf, linksum.drf. The .drf files were developed under Report Builder and their contents are listed in section 4.

To create our final standard interoffice reports is no simple task. Basically you must generate a set of CSV files from a set of DRF files, and then paste the information in the .csv files in a .xls file for the final report. In the end, you will only have five .xls files for the whole project regardless of the number of storm sewers you design.

To create the CSV files, follow the same instructions from section II, substituting the appropriate .drf files as needed. Linksum.csv and nodesum.csv will only need to be generated once, and it does not matter which storm sewer is active since these files are not storm sewer specific. You must generate the remaining five CSV files for every network (storm sewer system) you designed. These files are specific to each particular storm sewer.

5. Final Reports

Download our standard output XLS files from **\\nts\hq\precon-general\Hydraulics\Standard_Files_GEOPAK_2009\GEOPAK_Output\ Blank Files** to your project directory. The CSV files (linksum.csv and nodesum.csv) will be pasted in the linkstandardroad.xls and nodestandardroad.xls, respectively. We will discuss this in the next section. Now we will discuss the storm sewer specific files. The storm sewer specific files are the CSV files you created from the DRF files (nodestandardroad.drf, linkstandardroad.drf, nodestandard.drf, linkstandard.drf, and areastandard.drf.) You should have five .csv files for each storm sewer. The following is the process for creating our standard output files, for the individual storm sewers, from the CSV files.

1. Open the appropriate .xls file for the .csv file that you already have open– nodestandard.xls, linkstandard.xls, areastandard.xls, nodestandardroad.xls, linkstandardroad.xls. You will alter this file to reflect your information.
2. Add headings to the first sheet in rows 1 and 2. View the files under **\\nts\hq\precon-general\Hydraulics\Standard Files\GEOPAK Output\Sample output** to see exactly how the reports should look.
3. Select **Edit>Move or Copy Sheet…** and copy as many sheets as you have networks/storm sewer systems.
4. Change the sheet name to match the network name, i.e. System 12+00. Each sheet within the workbook will represent one system.
5. Highlight the data in the .csv file.
6. Right click and select **Copy**.
7. Go to the standard file (.xls) you just modified. Activate the appropriate sheet for the .csv file you are copying.
8. Click on the first cell under the table heading in the A column.
9. Right click and select **Paste Special…**
10. Choose **Values**
11. Choose **OK**. The font and alignment should be automatically set.

The files with road at the end (linkstandardroad.xls and nodestandardroad.xls) are to be sent to Roadway Designer. The other files are for your in-house report. Be sure to check that the max Design Capacity of 94% has not been violated. Eventually we will add a line in the report to give you such information.

Summary Report Files

We need to paste the linksum.csv and nodesum.csv files into the linkstandardroad.xls and nodestandardroad.xls files, respectively.

1. Open one of the sum.csv files.
2. Go to **Data>Sort…**.
3. Click the down arrow under **Sort by** and choose **Item**.
4. Choose **OK**.
5. Open the appropriate .xls for the appropriate sum.csv file you have open.
6. Select the sheet labeled Summary Report.
7. Add headings to the first two rows. View the files under **\\nts\hq\precon-general\Hydraulics\Standard Files\GEOPAK Output\Sample output** to see exactly how the reports should look.
8. Highlight the data in the .csv file.
9. Right Click and select **Copy**.
10. Go to the standard file (.xls) you just modified. Activate the Summary Report sheet for the .csv file you are copying.

12. Click on the first cell under the table heading in the A column.
13. Right click and select **Paste Special…**
14. Choose **Values**
15. Choose **OK**. The font and alignment should be automatically set.

Lastly, we need to run a macro to sum the pipes and basins. Two summary tables will be generated, one in the nodestandardroad.xls and one in the linkstandardroad.xls.

1. Open either nodstandardroad.xls or linkstandardroad.xls and activate the Summary Report sheet.
2. Open **\\nts\hq\precon-general\Hydraulics\Standard Files\macros\drainage_report.xls**. This is the macro file.
3. A message box will appear. Choose **Enable Macros**. It will appear as if nothing has happened.
4. Go to **Tools>Macro>Macros…**.
5. Choose **drainage_report.xls!Link_cal.Link_cal** if you are in linkstandardroad.xls and **drainage_report.xls!Node_cal.Node_cal** if you are in nodestandardroad.xls.
6. Choose **Run**.
7. You will get two messages. Click **OK** for both. The summary table will appear just below your inputted data.
8. Repeat these steps to generate the other summary table.

6. Interoffice Report

The following information is preset in our DRF files.

Our interoffice report will contain the following information:

- Node Report

File name: nodestandard.drf

Node Information:
1. I.D.
2. Description
3. Type
4. Library Item Name
5. Reference PGL
6. Station
7. Offset
8. Reference Elevation
9. Elevation
10. Depth
11. Junction Loss
12. Tc Used
13. Cumulative Tc
14. Cumulative Discharge
15. Cumulative Area
16. Cumulative C Value

17. Cumulative Intensity

- Area Report

File name: areastandard.drf

Area Information:

1. I.D.
2. Tc Used
3. Discharge
4. Intensity
5. Composite C Value
6. Composite Area
7. Total Subarea C Value
8. Total Subarea
9. Remainder C Value
10. Remainder Area

- Link Report

File name: linkstandard.drf

Link Information:

1. I.D.
2. Upstream Node
3. Downstream Node
4. Shape
5. Material
6. Number of Barrels
7. Rise
8. Actual Length
9. Slope
10. Discharge
11. Capacity
12. Uniform Depth
13. Uniform Velocity
14. Soffit Upstream
15. HGL Upstream
16. Soffit Downstream
17. HGL Downstream
18. Invert Upstream
19. Invert Downstream
20. Actual Velocity Upstream
21. Actual Velocity Downstream
22. Actual Depth Upstream
23. Actual Depth Downstream

7. Roadway Report

The following information is preset in our DRF files.

Road Design's report will contain the following information:

- Link Report

File name: linkstandardroad.drf

Link Information:

1. I.D.
2. Library Item
3. Type
4. Material
5. Shape
6. Upstream Node
7. Downstream Node
8. Actual Length
9. Slope
10. Rise
11. Pay Item
12. Invert Upstream
13. Invert Downstream

- Node Report

File name: nodestandardroad.drf

Node Information:

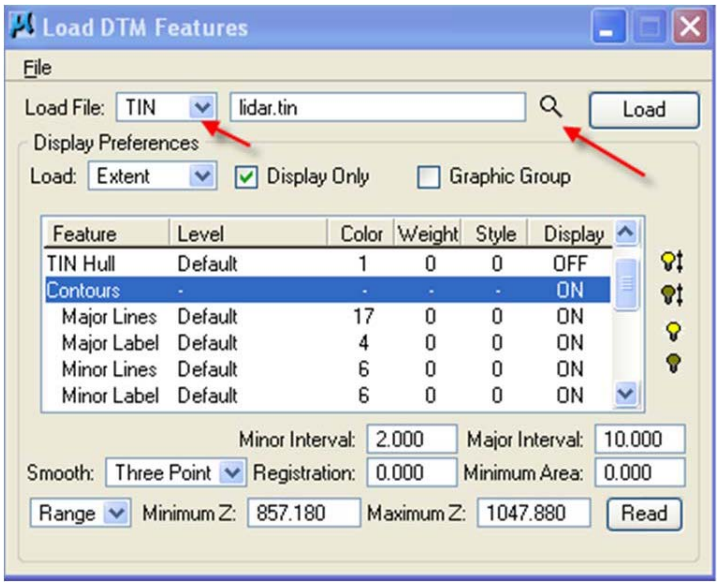
1. I.D.
2. Library Item Name
3. Reference PGL
4. Station
5. Offset
6. Elevation
7. Depth
8. Pay Item

Chapter 11: Microstation Drafting

Add completed networks and surface data to the Microstation plans, profiles, and cross-section sheets.

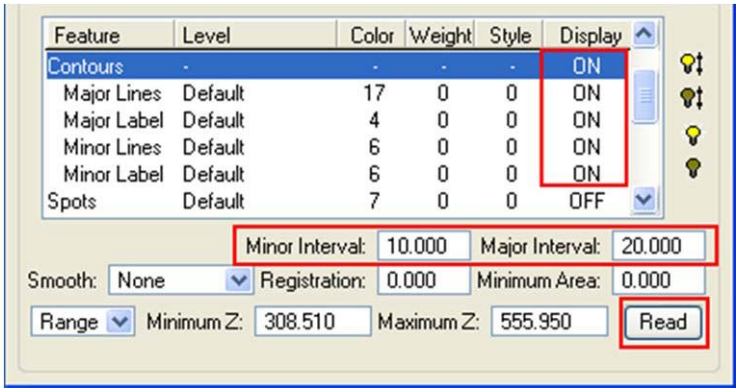
1. TIN Files – Draw Contours

From DTM Tools->Load->DTM Feature choose your *.TIN file to be loaded.

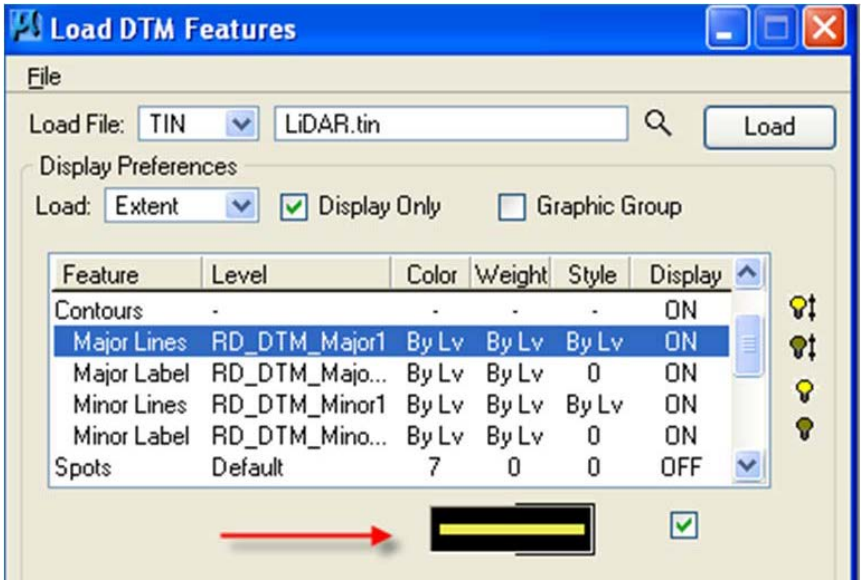


From the Load DTM Features dialogue set up the Display preferences. In the Load dropdown, choose whether you would like to load the entire .TIN or just specific locations. If just for reference purposes choose display only. This temporarily loads the DTM but will be removed at any change of the window (i.e. zoom in/out).

Highlight Contours and double click to turn on the display. With Contours highlighted set the values desired for Minor and Major intervals. Also, select read to set the MIN/MAX contour lines inside of the tin to be displayed.



Select the Major Lines and double click the black box to set up the features of the Major lines.

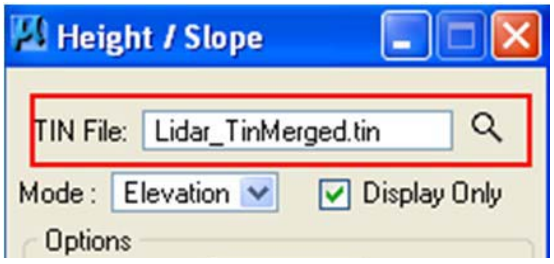


This opens up the Set Feature dialogue. Set up the Symbology as shown below, changing the Level to correspond to the feature for Major Label, Minor Lines, and Minor Labels.

Once all settings have been established choose Load to display contours.

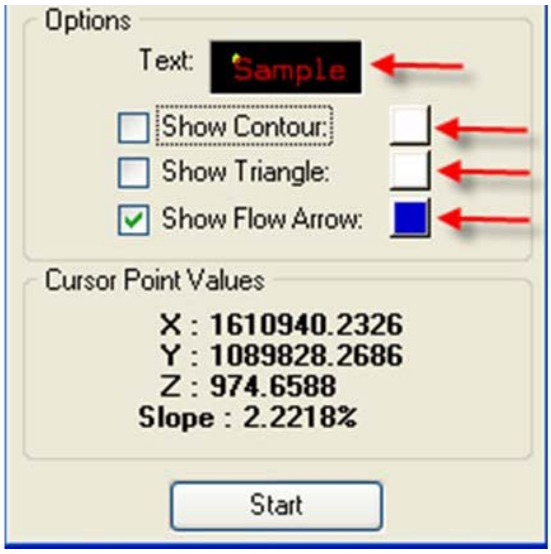
2. TIN Files – Height/Slope Analysis

From DTM Tools->Analysis->Height/Slope, load your *.TIN file to be analyzed.

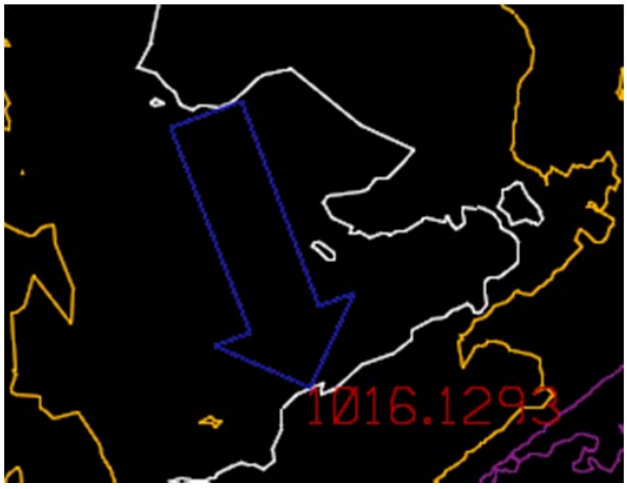


After a *.tin file has been selected the user should choose the mode for analyzing the the*.tin, choosing from Elevation or Slope from the Mode drop down box.

With Elevation selected, choose the Display Only check box to find spot elevations for reference purposes only. To adjust the text for elevations double click the text box to load the text dialogue box. Within elevation mode the user can choose to show the contour associated with the elevation, show the triangle associated with the slope, or show the arrow denoting the direction of water flow. The colors for each of these can be changed by double clicking the box to the right.



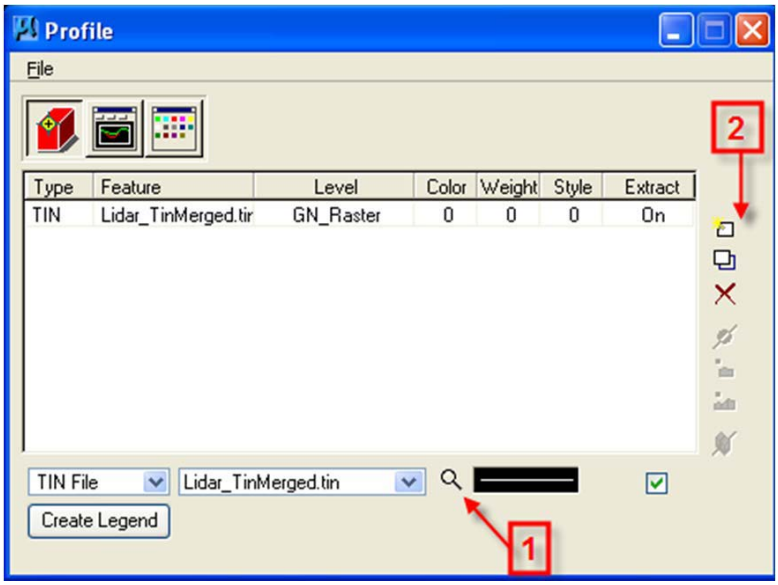
Select Start to begin. Move cursor throughout the *.tin file to display elevations and the other options selected. If display only was checked, any data point will be removed if any change is made to the view (i.e. zoom in/out)



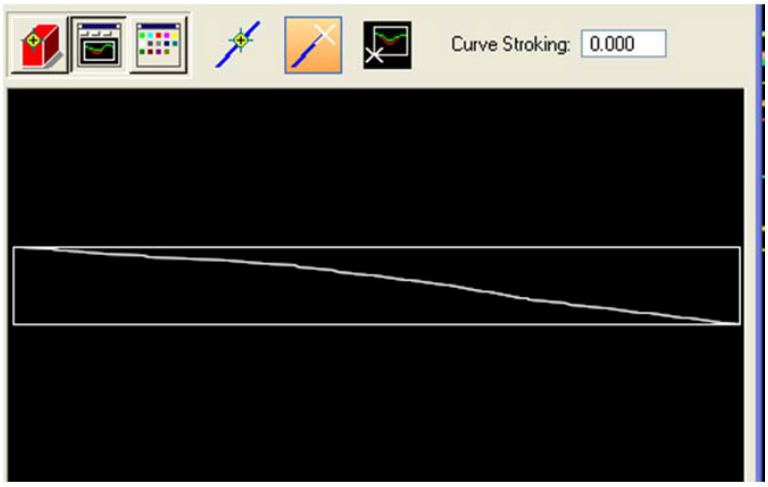
With Slope selected you will have the same option of changing the text and what will be displayed. Two options are available for measuring the slope: using a triangle and the two point method. Triangle slope changes dynamically with the cursor, while point method is found by selecting two points.

3. TIN Files – Creating Profiles

From DTM Tools->Analysis->Profile load your *.TIN file to be analyzed, and select “add list item”



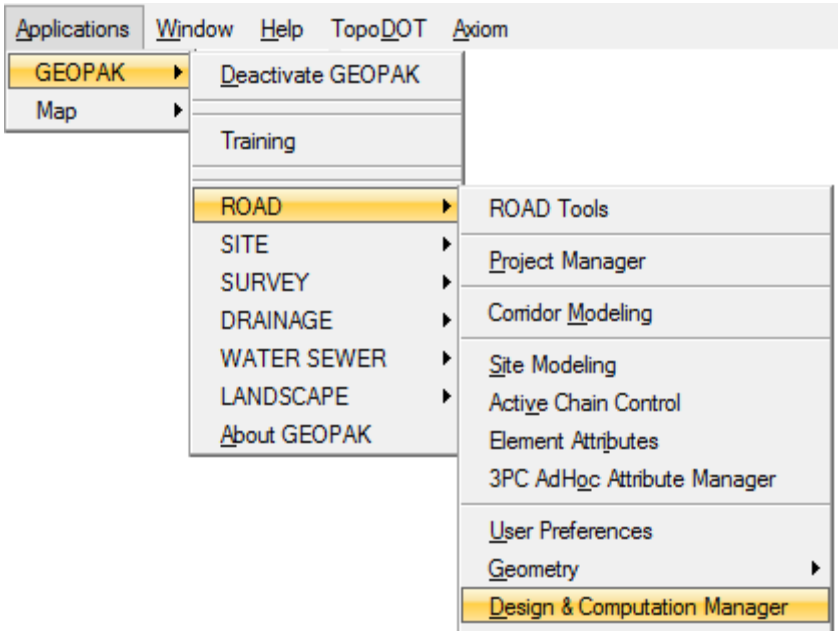
Select the Create Profile  button, choose the option of place profile element  and place two data points in the area you want the profile. This will display the profile in the dialog for review.



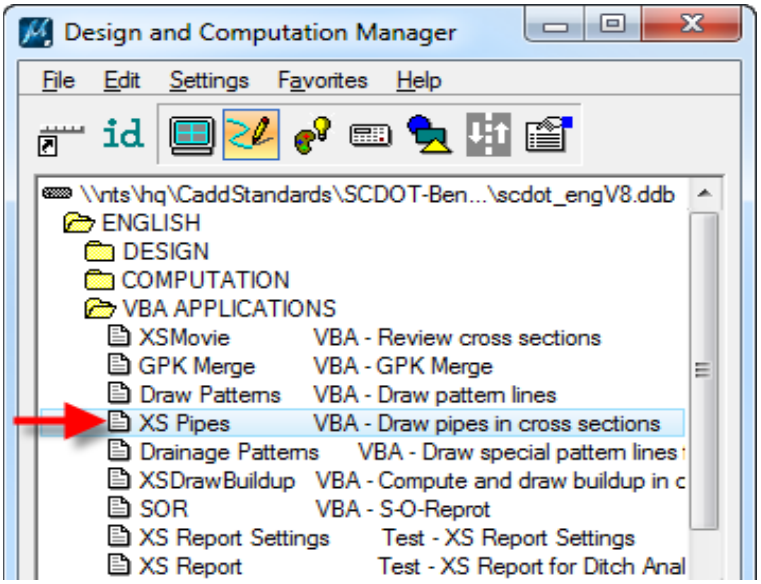
If satisfied with the profile previewed choose the third option to place the profile into the drawing.  To place the profile data, click anywhere within the *.dgn.

4. Cross Section Files – Draw Side Line Pipes

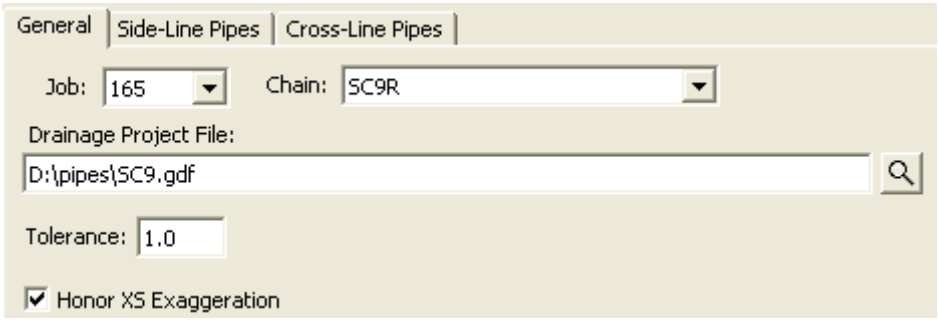
Open the *dx.dgn design file and select
Applications -> GEOPAK -> ROAD -> Design & Computation Manager from the menu bar.



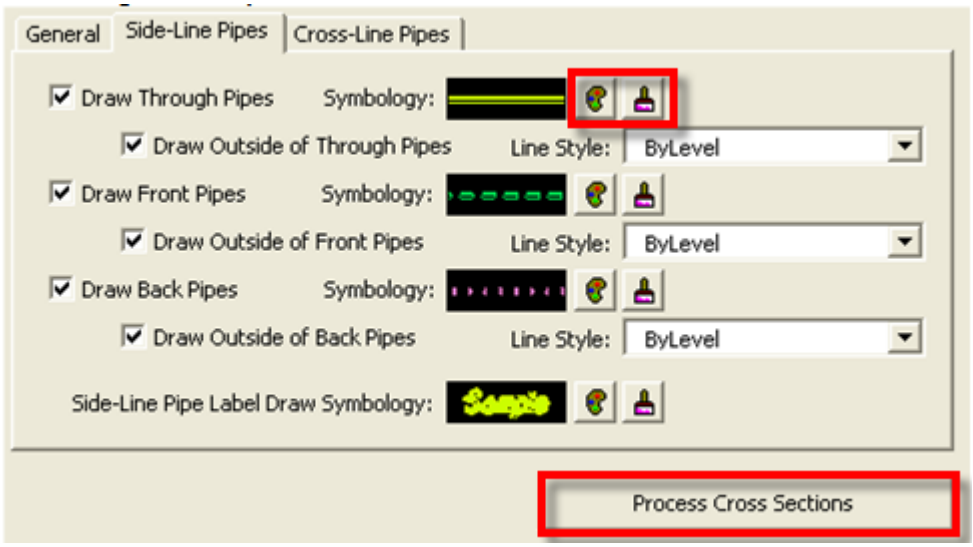
Select *English -> VBA Applications -> XS Pipes* and double click to start the macro.



On the **General** tab, select the Job and Chain, then browse to select the *.gdf file in the project folder. Check on the Honor XS Exaggeration. (This toggle controls the size of the pipe drawn on the cross sections in case they are drawn 10:5 vs. 5:5).



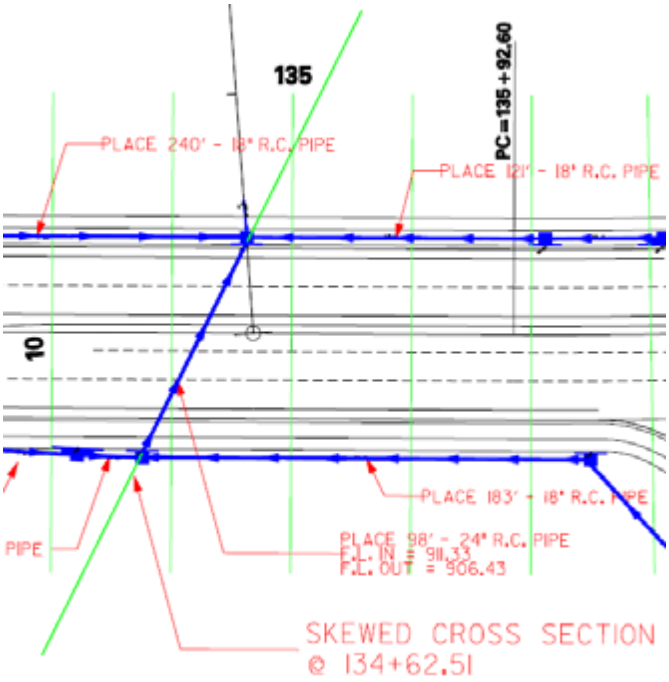
On the **Side-Line Pipes** tab, accept the defaults and press the **Process Cross Sections** button. (You can change the symbology by clicking the appropriate buttons prior to processing as well.)



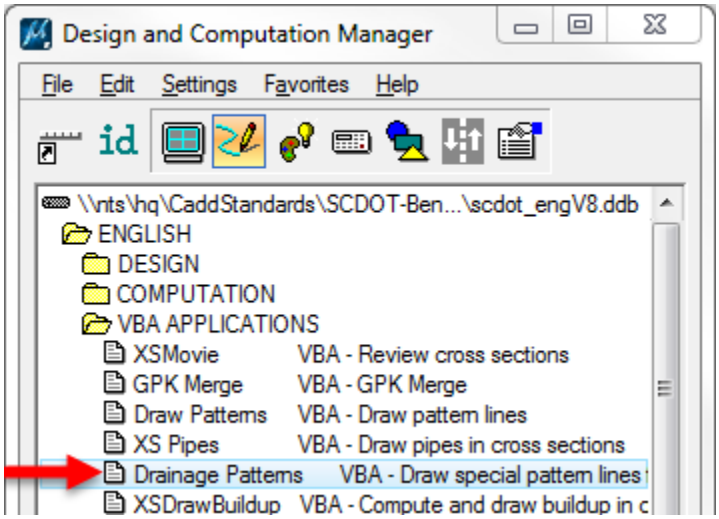
Press the **Begin** button and wait for the program to finish. Select **Yes** to save a *.csv pipe report file. Pipe drawings in cross sections **should be reviewed by roadway and hydraulic engineers** after processing.

5. Cross Section Files – Draw Cross Line Pipes

Open a plan view .dgn file showing the cross line pipes. Draw drainage pattern lines along each cross line pipe using MicroStation "draw line" command. Pattern lines should be set to one of the RD_PD_PatLn# levels and should extend a minimum of 20 feet beyond each end of the cross line pipe or to the construction limits.

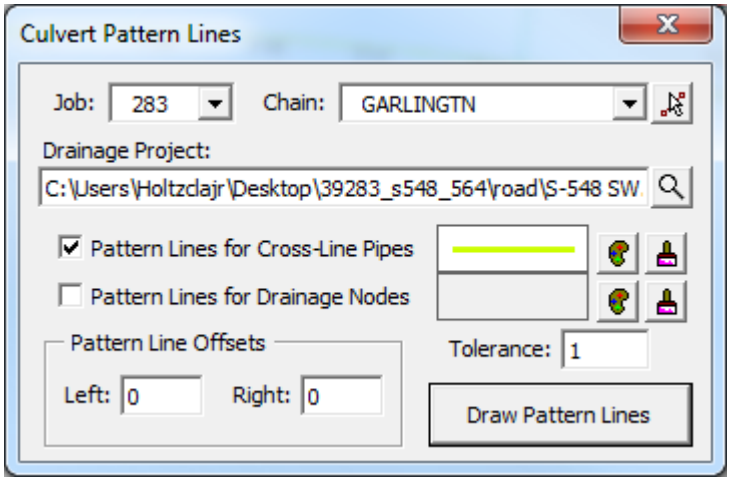


Alternatively, open the D&C Manager (*Applications -> GEOPAK -> ROAD -> Design & Computation Manager* on the menu bar), select *English -> VBA Applications -> Drainage Patterns*, and double click to start the macro.



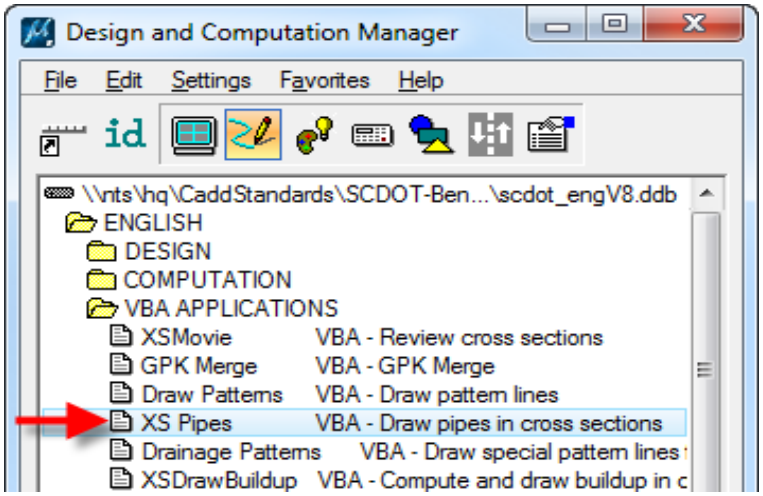
In the Culvert Pattern Lines window, enter the Job number of the project and select the chain of the road being crossed. Click the magnifying glass next to the "Drainage Project"

space and browse to the .GDF file for the project. Check the box marked "Pattern Lines for Cross-Line Pipes" and enter 0 for the Pattern Line Offsets



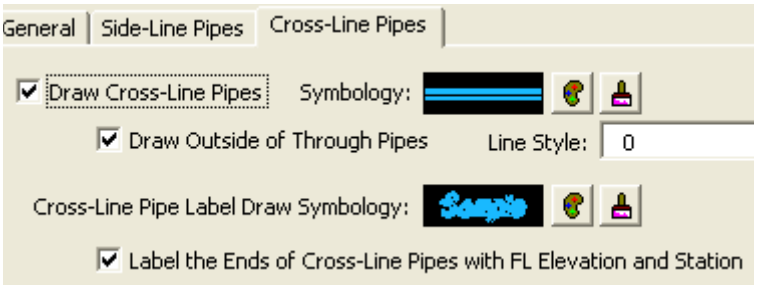
Open or create a *xp.dgn file for the project. Cut existing and proposed cross sections using the new drainage pattern lines. Cross sections will be skewed.

Open the D&C Manager, select *English -> VBA Applications -> XS Pipes* and double click to start the macro.



On the **General** tab, select the Job, Chain and browse to select the *.gdf file. Check on the Honor XS Exaggeration. On the Side-Line Pipes tab, turn off all of the checks.

On the **Cross-Line Pipes** tab, check on all of the boxes. You can accept the defaults for symbology or click the appropriate buttons to change them.

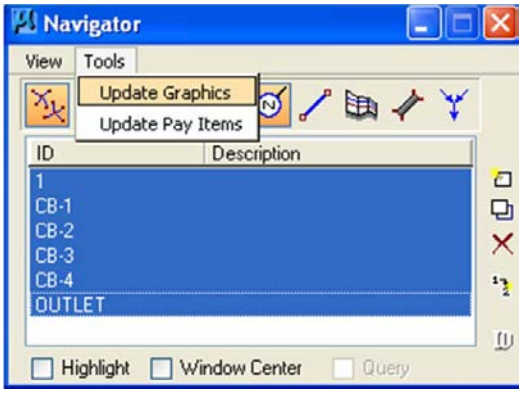


Press the **Process Cross Sections** button. Press the **Begin** button and wait for the program to finish. Select **Yes** to save a *.csv pipe report file. Pipe drawings in cross sections **should be reviewed by roadway** and hydraulic engineers after processing.

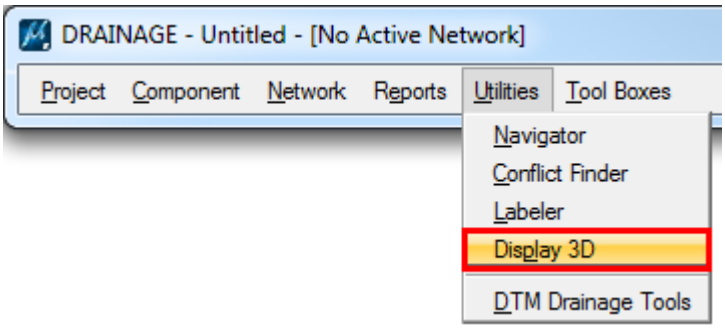
6. Displaying 3-D Networks

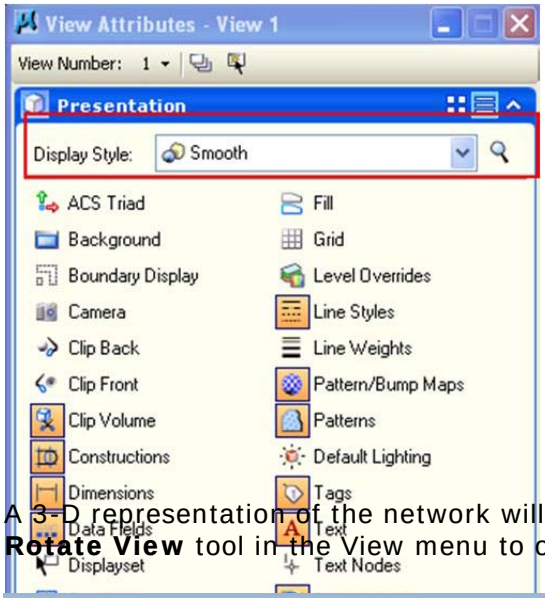
Create a new 3-D .dgn. Start GEOPAK Drainage and open the .gdf containing the network. Open the GEOPAK Drainage Preferences menu and go to the Project Components tab. Ensure that the Drainage Cell Library being used is the most up-to-date RoadV8.cel dated 8/24/2011 or later. If not, load the *.dpf file associated with the project and check again.

Update all Nodes and Links via Navigator.

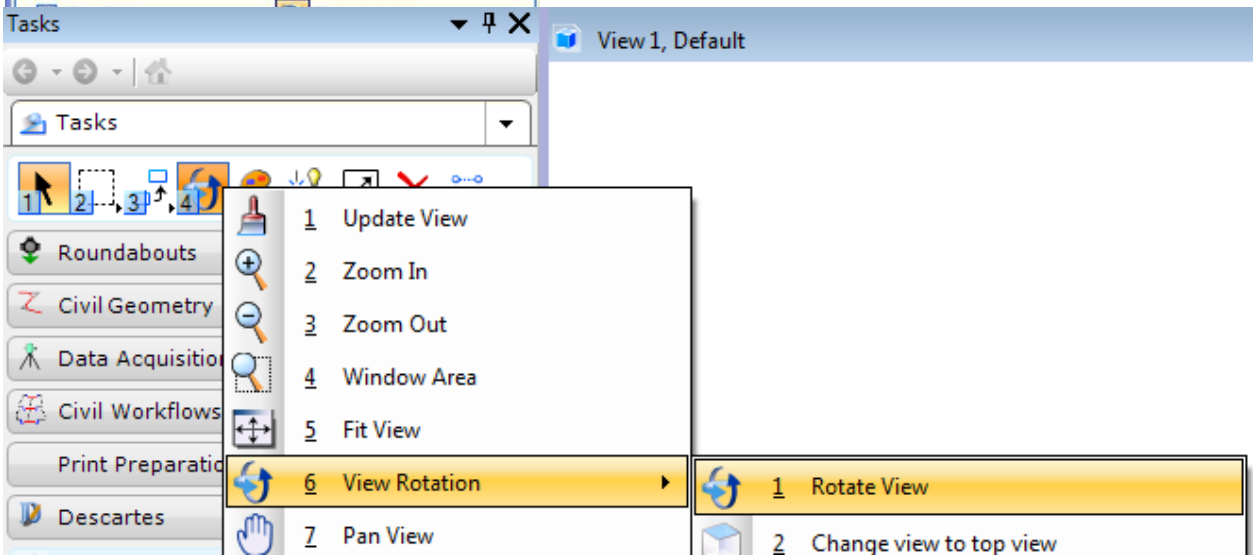


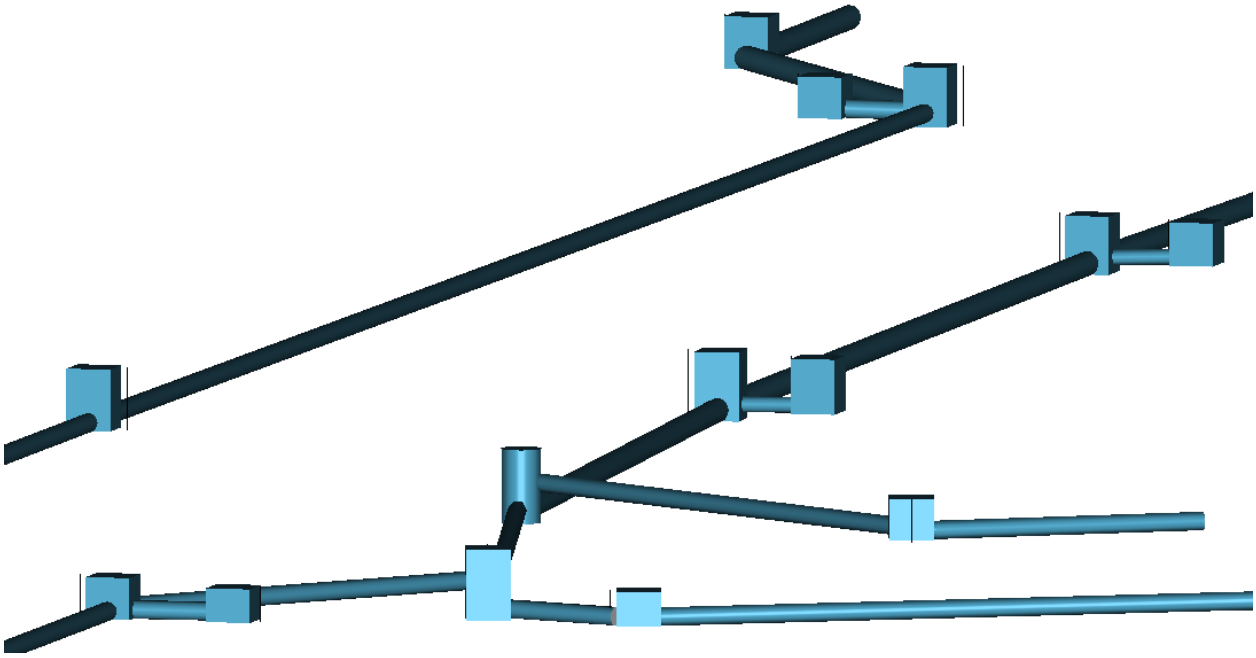
Activate the 3D display using Utilities -> Display 3D in the Drainage menu. Change the Display Style in View Attributes from “Wireframe” to “Smooth.”





A 3-D representation of the network will be drawn above the existing 2-D version. Use the **Rotate View** tool in the View menu to observe the network from other angles.





A 3-D view can reveal errors in the network design, such as incorrect structure and pipe elevations. Examining this view for pieces that appear out of place or incorrectly shaped can be much quicker than comparing the data associated with each pipe and node in text format using the Navigator tool.

Chapter 12: Uploading Files

After you complete your design, you must put all pertinent information about your project on the server for the Roadway Designer to access. All the information that you must supply to them is listed under Exchange of Information. Below is the form that you will email to the Roadway Designer explaining where the files are located and what you named them.

This will be the only thing that you will send them electronically. They will access everything else electronically from our server. Make sure you send them a hardcopy of the abandon/retain notes.

ROAD DESIGN DATA FOR HYDRAULIC DESIGN
DATE: 8/18/2008

DESIGN GROUP:	RPG 4 - UPSTATE
PIN NO.:	36298
COUNTY:	OCONEE
ROAD/ROUTE NO.:	S-402 / SHEEPFARM RD.
PROJECT DESCRIPTION:	WIDEN (SHEEPFARM RD.) EXISTING 2 LANES TO 5 LANE CURB & GUTTER WITH BIKE LANES AND SIDEWALKS ON BOTH SIDES. CONNECT SHEEPFARM ROAD TO SC 28 (BLUE RIDGE BLVD.) WITH NEW LOCATION STARTING AT S-135 (BOUNTYLAND ROAD).
ADDITIONAL NOTES:	S-135 WILL BE WIDENED TO 5 LANE CURB & GUTTER AT THE INTERSECTION WITH SHEEPFARM ROAD. IT WILL THEN TIE BACK TO THE EXISTING PAVEMENT. STONEBROOK DRIVE WILL HAVE A VALLEY GUTTER. ALL OTHER SIDE ROADS WILL HAVE DITCH SECTIONS.

FILE INFORMATION

	SERVER	DESIGN GROUP	COUNTY	PIN
FILES LOCATED:	NTS/HQ/Precon/	RPG 4	OCONEE	36298
'.GPK' FILE(S):	JOB298.GPK			
'PP' FILE(S):	R36298PP.DGN			
'PF' FILE(S)	R36298PF1.DGN		S-402 (SHTS. 6 - 13)	
	R36298PF2.DGN		SIDE ROADS (SHTS. 14 - 23)	
	R36298PF3.DGN		SIDE ROADS (SHTS. 24 - 26)	
'.NEW' FILE(S):	36298.NEW			
	36298A.NEW			

CHAIN NAME	PROPOSED PROFILE	DESCRIPTION
S402REL	S402RFP	CENTER LINE GRADE S-402 RELOCATION
US76		CENTER LINE US 76
BROOKLANER	BROOKLANERFP	CENTER LINE GRADE BROOK LN. REL.
STONEBROOKR	STONEBROOKRFP	CENTER LINE GRADE STONEBROOK DR. REL.
S135REL	S135RFP	CENTER LINE GRADE S-135 RELOCATION
SPRINGWOOD		CENTER LINE EAST SPRINGWOOD DR.
OCONEER1	OCONEER1FP	CL GRADE OCONEE ESTATES REL. PART 1
OCONEER2	OCONEER2FP	CL GRADE OCONEE ESTATES REL. PART 2
ALBERTSR	ALBERTSRFP	CENTER LINE GRADE ALBERT'S RD. REL.
PAULGILLR	PAULGILLRFP	CENTER LINE GRADE PAUL GILLISON RD. REL.
SC28		CENTER LINE SC 28

TANGLEWOOD		CENTER LINE TANGLEWOOD DR.
STREAM1		CENTER LINE OF STREAM #1
STREAM2		CENTER LINE OF STREAM #2
STREAM3		CENTER LINE OF STREAM #3
STREAM4		CENTER LINE OF STREAM #4
STREAM5		CENTER LINE OF STREAM #5
STREAM6		CENTER LINE OF STREAM #6

CHAIN NAME	CROSS SECTIONS	DESCRIPTION
S402REL	S402REL_DX_SCALED.DGN	STA. 10+00.00 - 114+64.38
US76	US76DX.DGN	STA. 10+50.00 - 20+00.00
BROOKLANER	BROOKLANERDX_SCALED.DGN	STA. 10+35.87 - 16+27.98
STONEBROOKR	STONEBROOKRDX_SCALED.DGN	STA. 10+35.92 - 15+94.48
S135REL	S135RDX_SCALED.DGN	STA. 10+00.00 - 42+79.57
SPRINGWOOD	SPRINGWOODDX.DGN	STA. 10+12.71 - 15+57.93
OCONEER1	OCONEER1DX_SCALED.DGN	STA. 10+30.38 - 16+00.00
OCONEER2	OCONEER2DX_SCALED.DGN	STA. 36+00.00 - 41+26.31
ALBERTSR	ALBERTSRDX_SCALED.DGN	STA. 7+00.00 - 12+22.35
PAULGILLR	PAULGILLRDX_SCALED.DGN	STA. 15+02.15 - 19+01.89
SC28	SC28DX.DGN	STA. 648+00.00 - 660+00.00
TANGLEWOOD		

If you have revisions to your GDF file and must resubmit files to Roadway Designer, resend them the email and state what changes were made. Be specific and name the pipes and nodes that were affected. If a large section of roadway was affected then give road stations that define the limits of those changes.

Consultants will still need to submit all files to the Hydraulic Engineer by a CD or FTP for the Roadway Designer's use and a hard copy of the plans for our review. Labels will still be required for all pipes and basins, but they are to be very basic. The Roadway Designer will do the final labels themselves for both Right-of-Way and Construction plans. Also include the spreadsheet on the previous page in the submittal and the Hydraulic Engineer will forward it to the Roadway Designer.

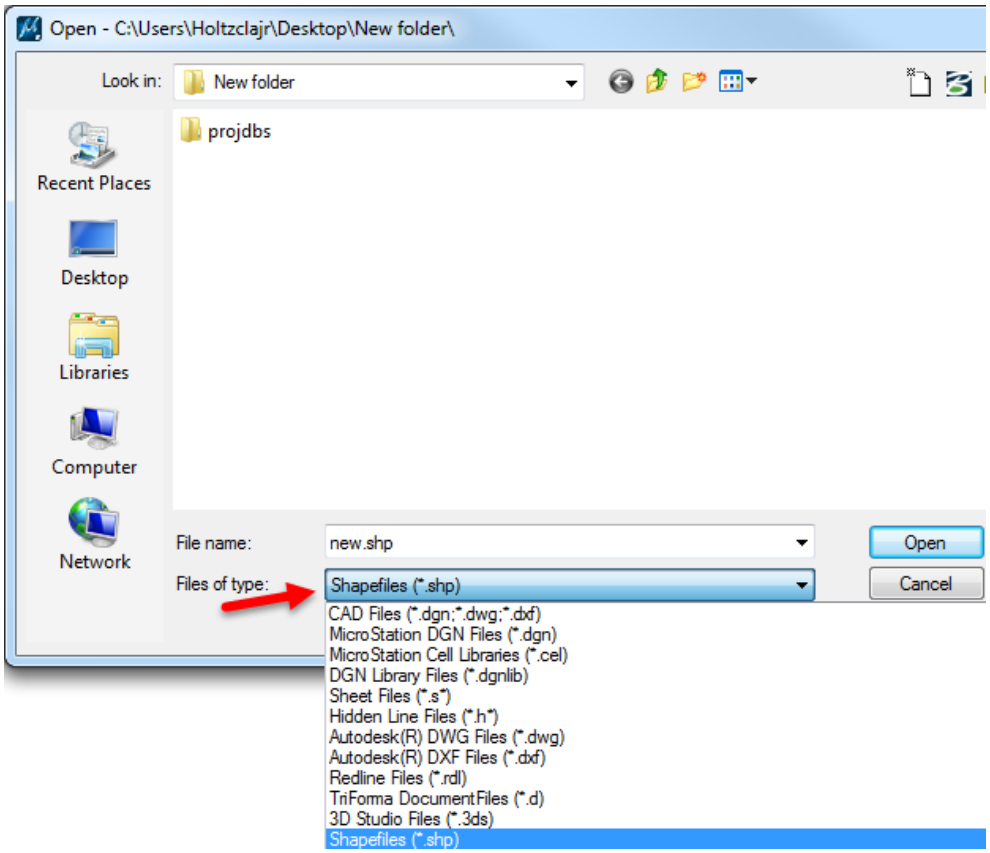
Chapter 13 - Import/Export

This section covers the import & export of shapefiles, XPSWMM, HEC-RAS, Google Earth, DEM files and LiDAR.

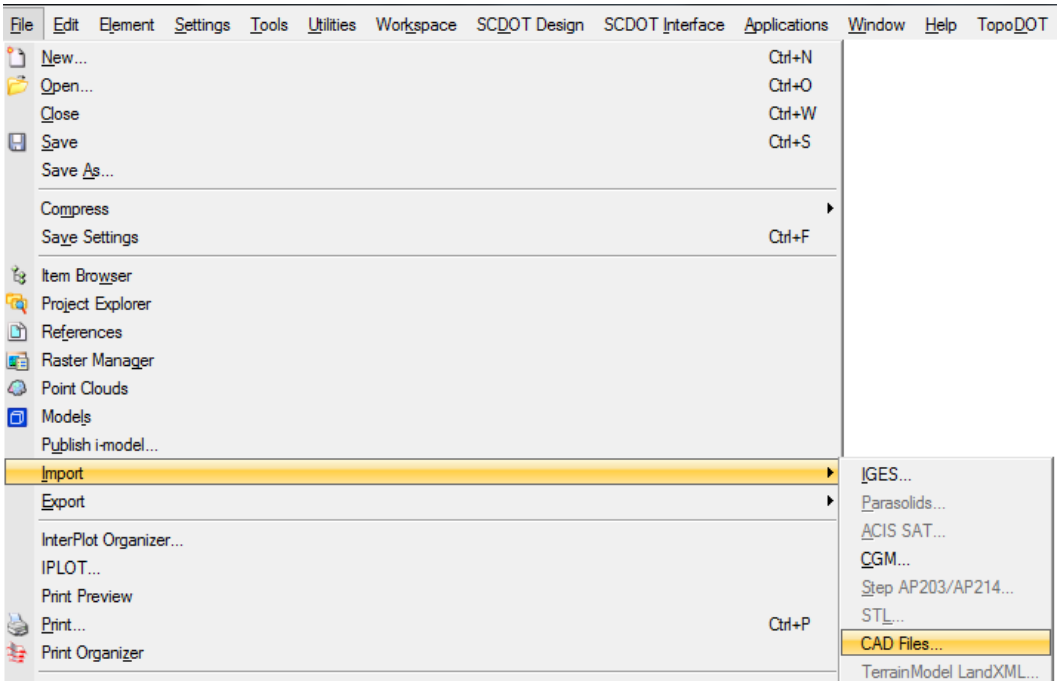
1. Shapefiles

Shapefiles (.SHP) can be opened as a MicroStation file and saved as a .DGN, imported into an existing drawing, or referenced within a drawing.

Opening a shapefile in MicroStation works the same as opening a .DGN file. Go to File->Open in the menu bar. In the Open File screen, change the file type (located at the bottom of the screen) from CAD Files (.DGN) to Shapefiles (.SHP), then select the desired shapefile and press open. The file will be opened as Read-Only, but it can then be saved normally as a .DGN file and edited from there.



To Import a shapefile into an existing document, use the File->Import->CAD Files command in the main menu.

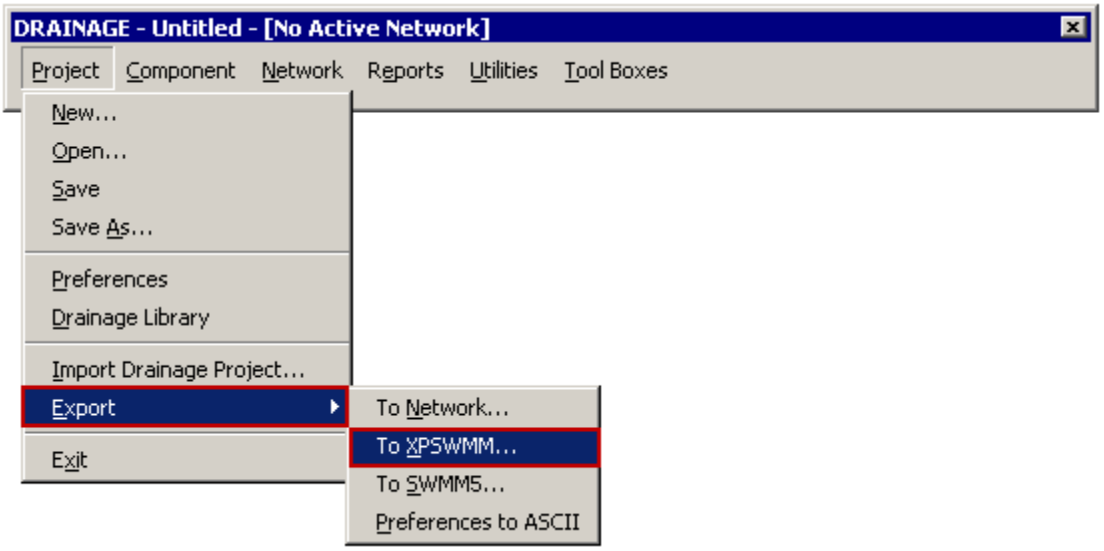


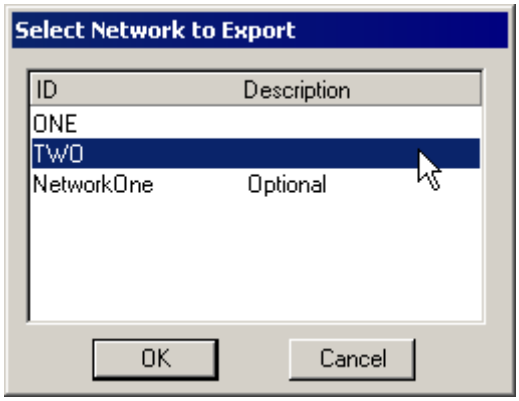
In the import menu, set the file type to Shapefiles exactly as per the process for opening a shapefile, select the file in question, and press Open.

To attach a Shapefile as a reference, follow the same process as for attaching another Microstation drawing as a reference. When browsing to the desired file, set the file type to Shapefile as per the previous two processes, then select the shapefile and press Open.

2. XPSWMM

To Export any Network of the current Drainage GDF file:



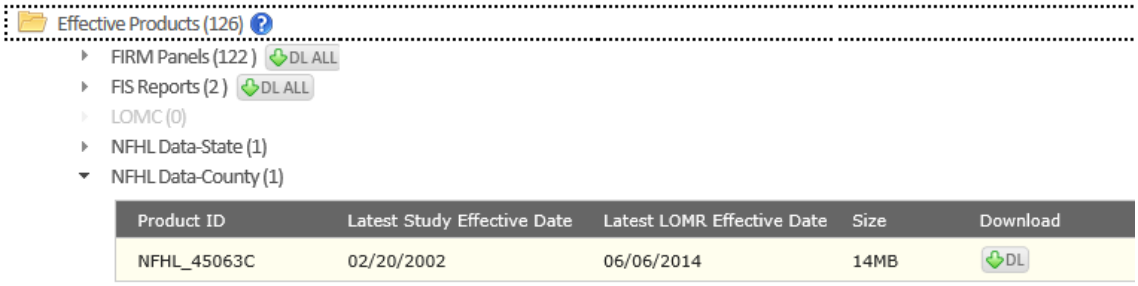


The Network is now available for import into XPSWMM.

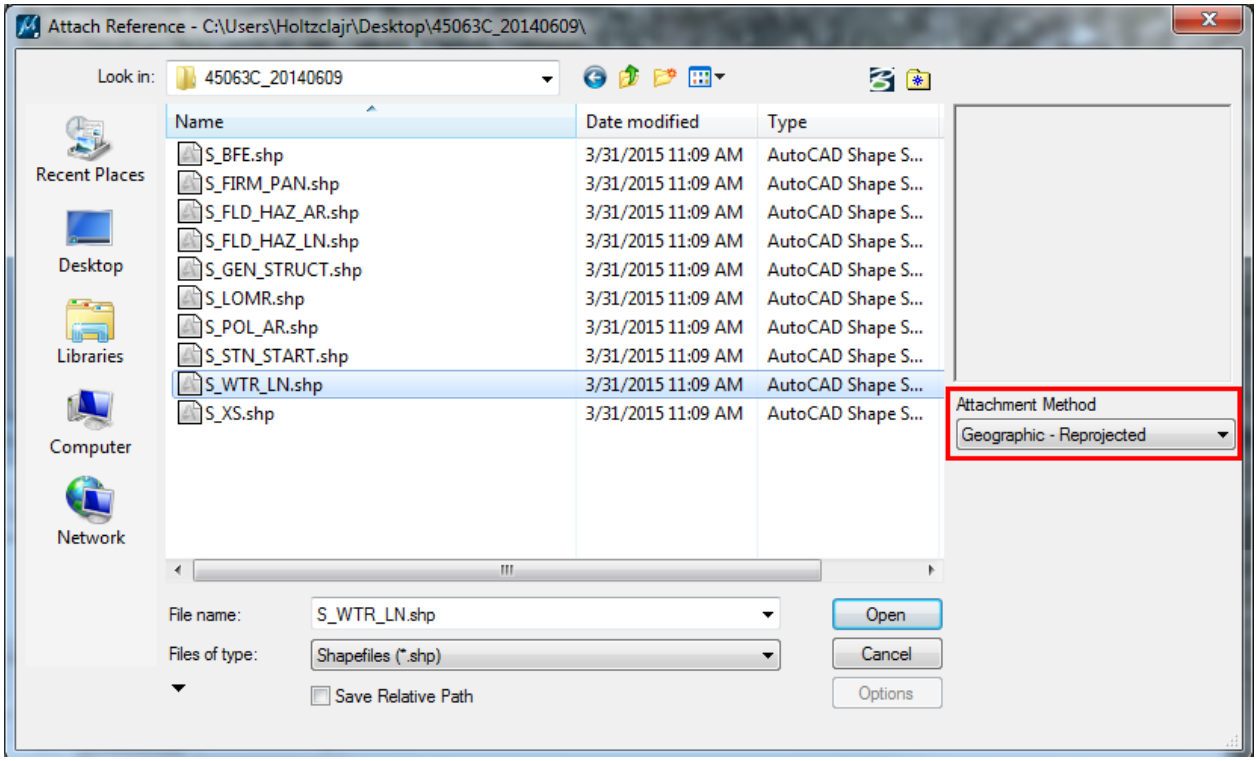
3. HEC-RAS

Microstation surfaces can be used to produce cross-section data importable into HEC-RAS models. To do so, you will create a chain representing the stream, cut cross-sections along it from a surface file, and then export those cross-sections as a file readable by HEC-RAS.

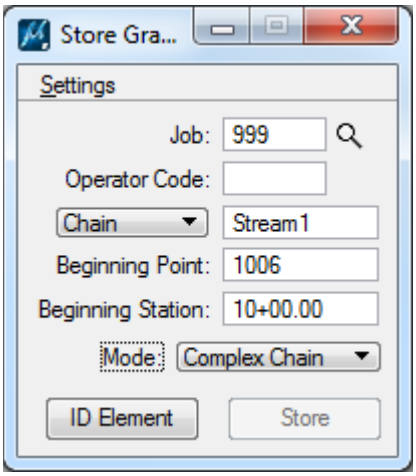
Streams may be delineated in the project survey. However, preparing a new chain for the stream based on the FEMA blue-line data will allow closer coordination with FEMA and other state and federal authorities that base their data on FEMA models. The FEMA stream delineation can be found in the S_WTR_LN.shp ShapeFile in the National Flood Hazard Layer (NFHL) data available in SCDOT files and on the FEMA website.



Open the r#####hy.dgn file for the project. Open the References menu and Select Attach, then browse to the shapefile containing the stream line. For the Attachment Method, select "Geographic – Relocated", then press Open.

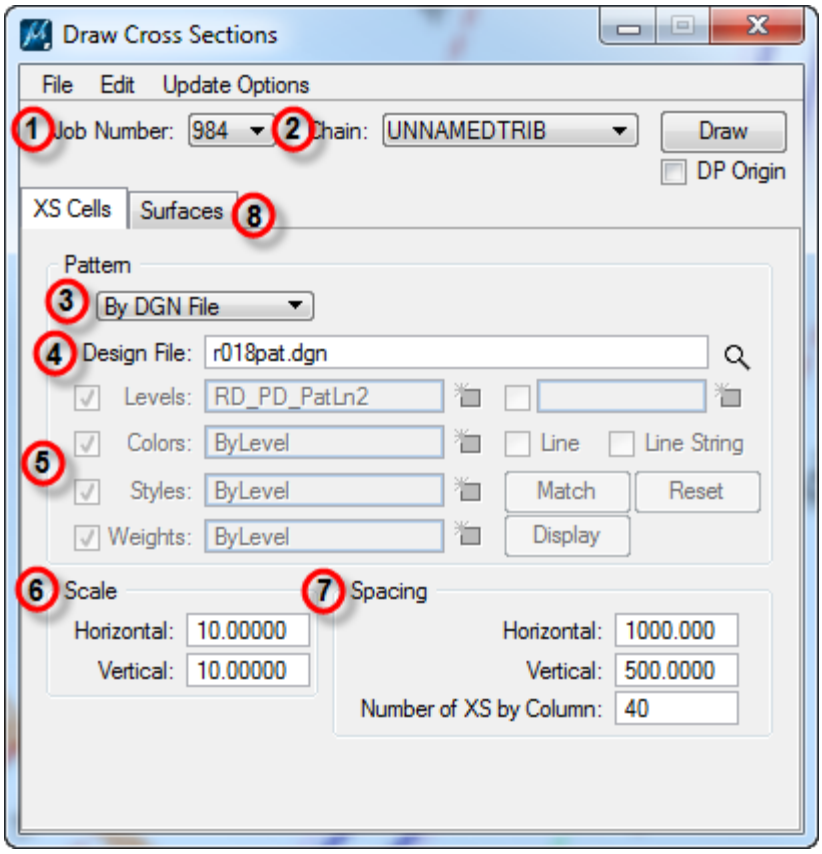


Use the Smartline tool to draw a new line for the stream, beginning at the downstream end, and use *Geopak -> Road -> Geometry -> Store Graphics* to turn it into a chain. Select the GPK file for the project and name the Stream. The Beginning Point and Beginning Station entries can be left as they are. Click ID Element, click on the line drawn for the stream, then click Store.

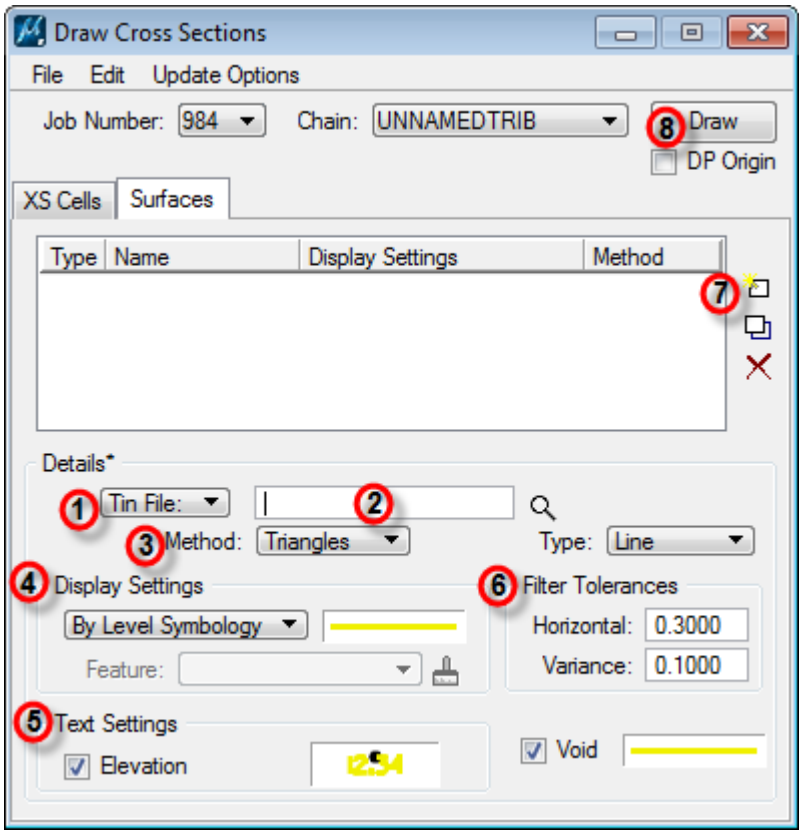


Using Smartlines, draw the pattern lines for the required cross-sections along the stream. The pattern lines should be drawn on one of the RD_PD_PatLn levels. The pattern lines should be perpendicular to the chain within the streambed, then turn to perpendicular with the topographic contours in the overbank areas to either side. Place pattern lines as needed to clearly define the stream bed cross-section and any curves in its flowline, as well as any and all notable features in the overbank area such as steep slopes, ditches, and structures. Features that directly impact the waterway such as bridges or connections with other streams should be clearly defined using as many cross-sections as necessary.

Open *Geopak -> Road -> Cross Sections -> Draw Cross Sections From Surfaces*.

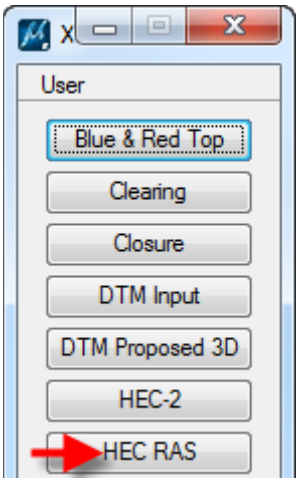


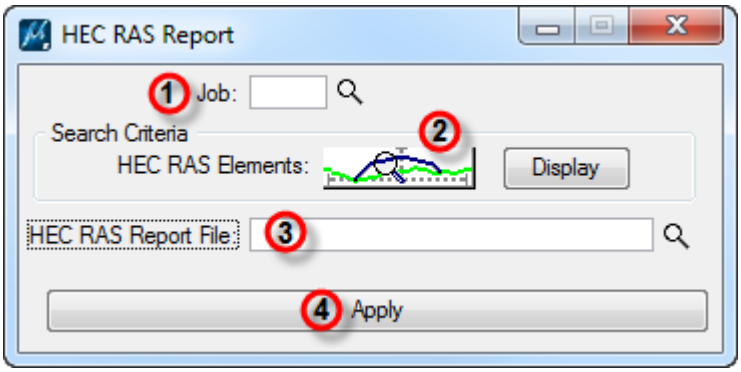
1	Job Number	Select the .gpk file for the project
2	Chain	Select the chain representing the stream
3	Pattern	Select “By DGN File” to use the pattern lines in the current file.
4	Design File	Select the file for the cross sections to be drawn in. It may be the current file.
5	Pattern Line Symbology	Select the Level, Color, Style, and Weights of the pattern lines for the cross sections.
6	Scale	Set the horizontal and vertical scale for the cross section views
7	Spacing	Set the spacing at which the cross section views with be drawn
8	Surfaces Tab	Go to the Surfaces tab to enter data about the topographic surface.



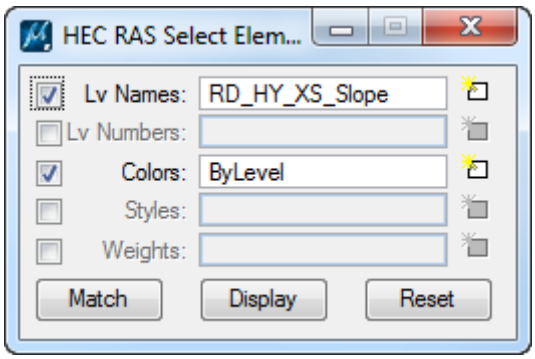
1	File Type	Select the file type of the topographic surface, usually a .Tin file.
2	Surface	Browse to the location of the topographic surface file and select it
3	Method	Select **Triangles**
4	Display Settings	If desired, set new symbology for the cross-section line work.
5	Text Settings	If desired, set new symbology for the cross-section elevations.
6	Filter Tolerances	Do not change.
7	Add Surface	Click the top icon to add the topographic surface to the system
8	Draw	Click the **Draw** button to draw the cross sections

Go to the new cross sections you just drew. Open *Geopak -> Road -> Cross Sections -> Reports* and click the **HEC-RAS** button.



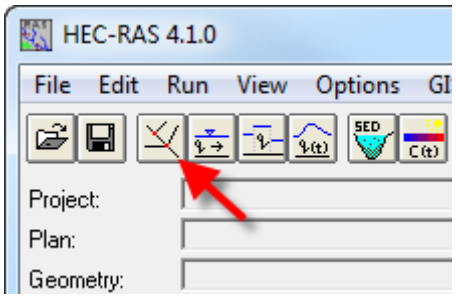


1	Job	Select the .GPK file for the project
2	HEC RAS Elements	Click the image to go to the HEC RAS Select Elements Window – see below.
3	HEC RAS Report File	After completing the HEC RAS Select Elements window, enter or browse to the file location and name of the report to be created
4	Apply	Click Apply to create the export file

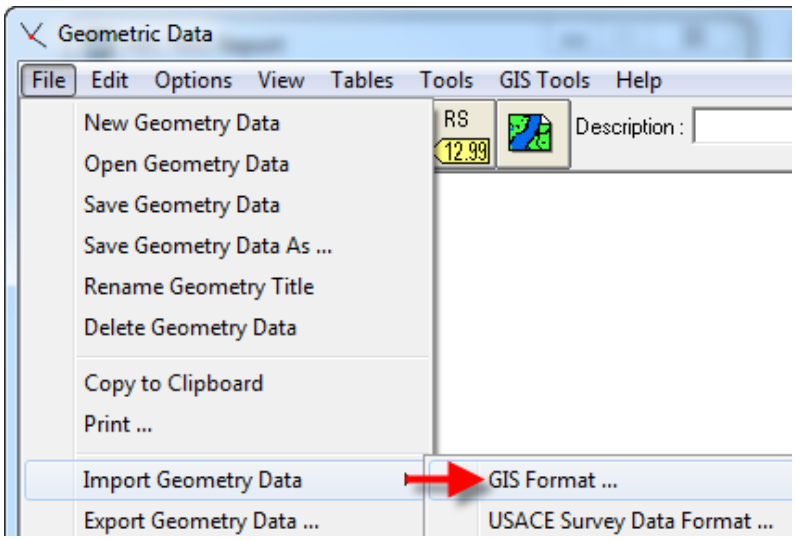


Click **Reset** to clear the fields, then make sure that only **Lv Names** and **Colors** are turned on using the check boxes. Click **Display**, then **Match** to begin inputting data. Click on the surface lines in the cross sections. As level names and colors are filled in, any lines matching those combinations of level and color will be highlighted to show they are being included in the report. Make certain that all surface lines in the cross sections are highlighted before continuing. Then close the **Select Elements** window, enter the HEC RAS Report File name and location and click **Apply** in the **HEC RAS Report** window.

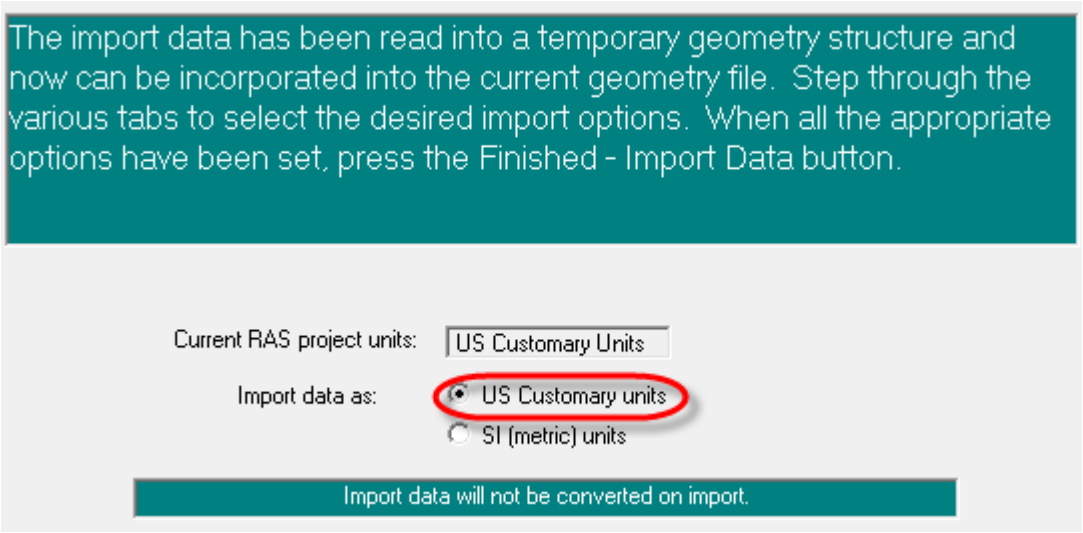
Open HEC RAS. Create a new project or open an existing one, then open the Geometric Data screen.



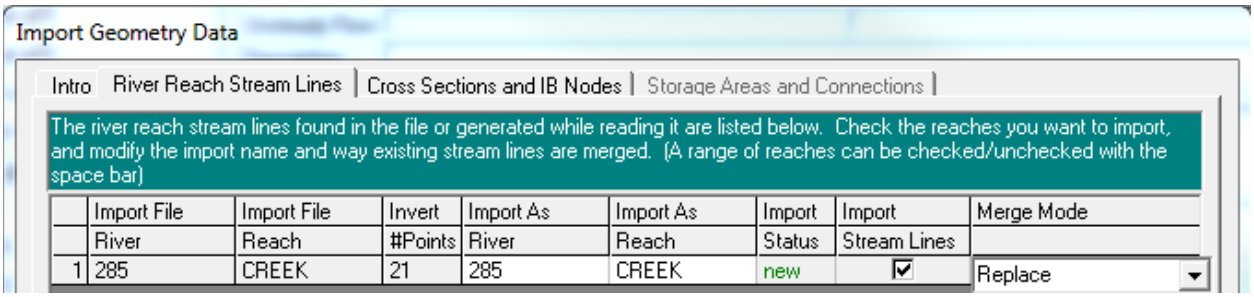
Select *File -> Import Geometry Data -> GIS Format* and browse to the report created in Microstation.



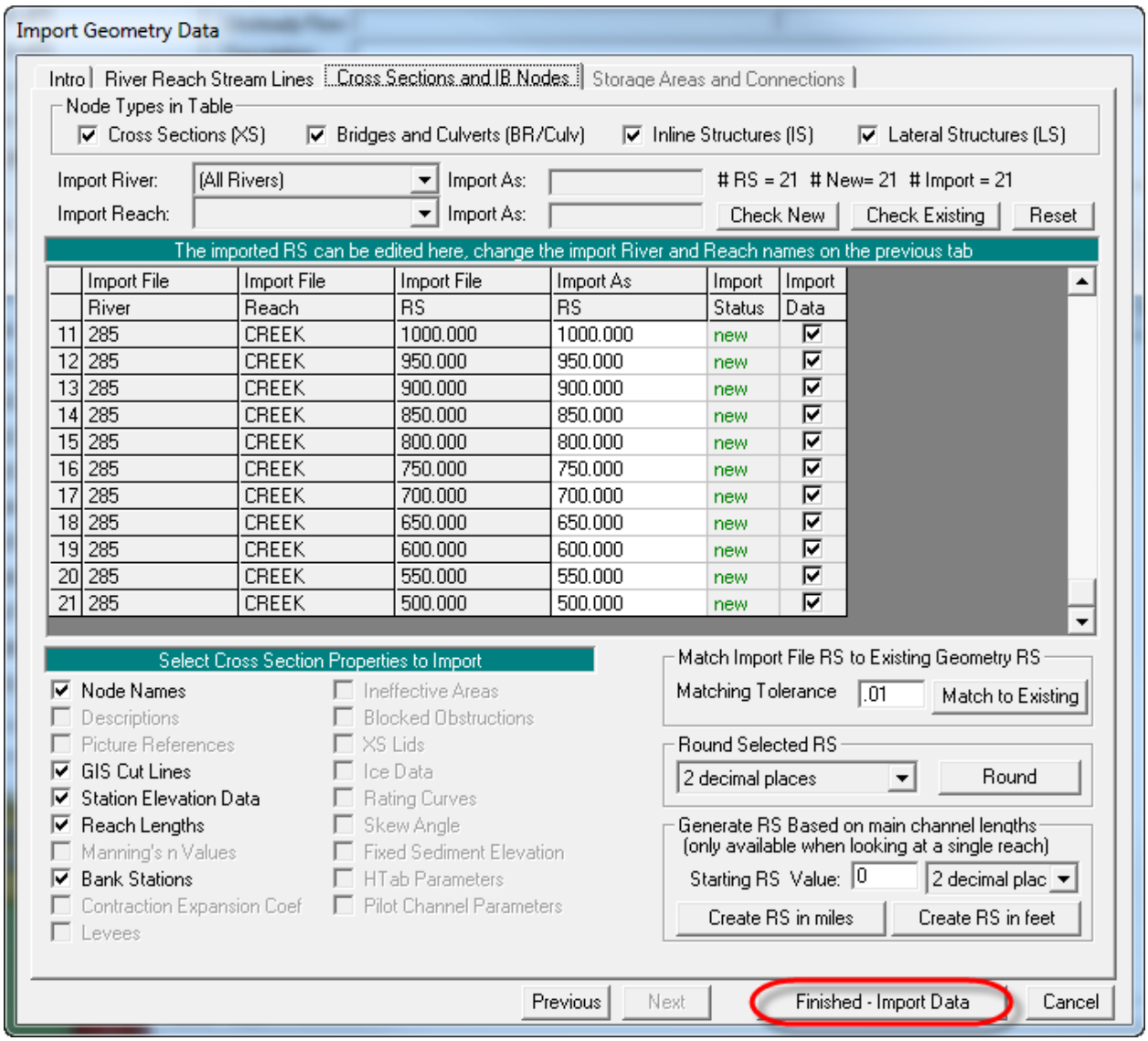
On the **Import Geometry Data** screen, **Intro** tab, make sure US Customary Units is selected.



On the **River Reach Stream Lines** tab, confirm that the data shown is correct, that stream lines are being imported, and that the merge mode is set to “Replace”.



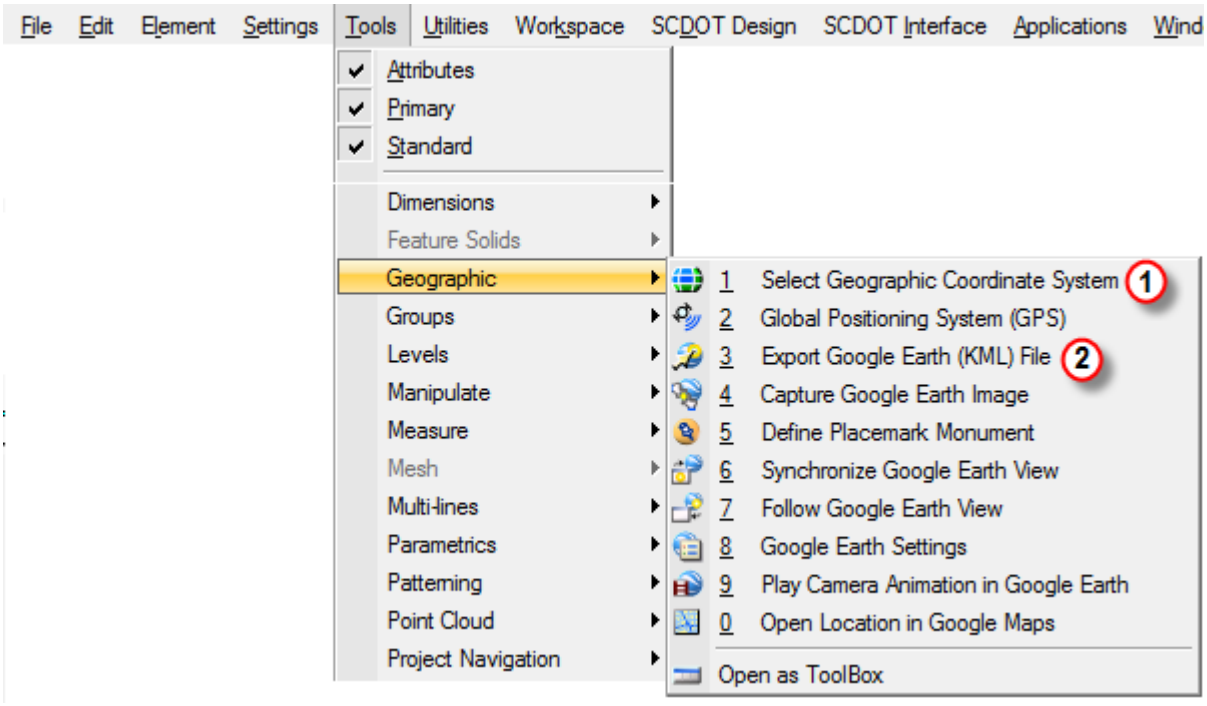
On the **Cross Sections and IB Nodes** tab, confirm that the data shown is correct before pressing the **Finished – Import Data** button.



4. Google Earth

- Exporting DGN Files In SPCS

Exporting to Google Earth from a Microstation .DGN file set to the State Plane Coordinate System is very simple. SCDOT's Microstation seed files are automatically set to the SPCS, as are the majority of new surveys. Older files may use modified or local coordinate systems.



1	Coord. System	Check or set the coordinate system.
2	Export KML	Export the entire Microstation file as a .KML file for Google Earth.

- Exporting DGN Files In Local Coordinates

Exporting to Google Earth from a file set to local or adjusted coordinates requires a 3-step procedure:

1. Use Google Earth to add & save a Placemark
2. Use MicroStation to Define the Google Earth Placemark Monument
3. Use MicroStation to Export the Google Earth File

- Add/Save Placemark

Use Google Earth to add & save the Placemark:

1. Zoom to your desired location in Google Earth.
2. From Google Earth, select Add > Placemark (enter the name; verify the latitude & longitude are correct).
3. Right-click on the Placemark, select "Save Place As" and save as the KML or KMZ (the Z means zipped format and is thus a smaller file) to a folder on your local computer or server.

- Define Placemark

In MicroStation, select Tools>Google Earth:

1. Zoom to your desired location in MicroStation (corresponding to step 1 in add placemark).
2. In MicroStation's Google Earth toolbar, click "Define Google Earth Placemark Monument", and snap to the location in step #1 above.
3. In MicroStation's "Select Monument Placemark File" window, select the saved placemark file (from step 3 in add placemark).

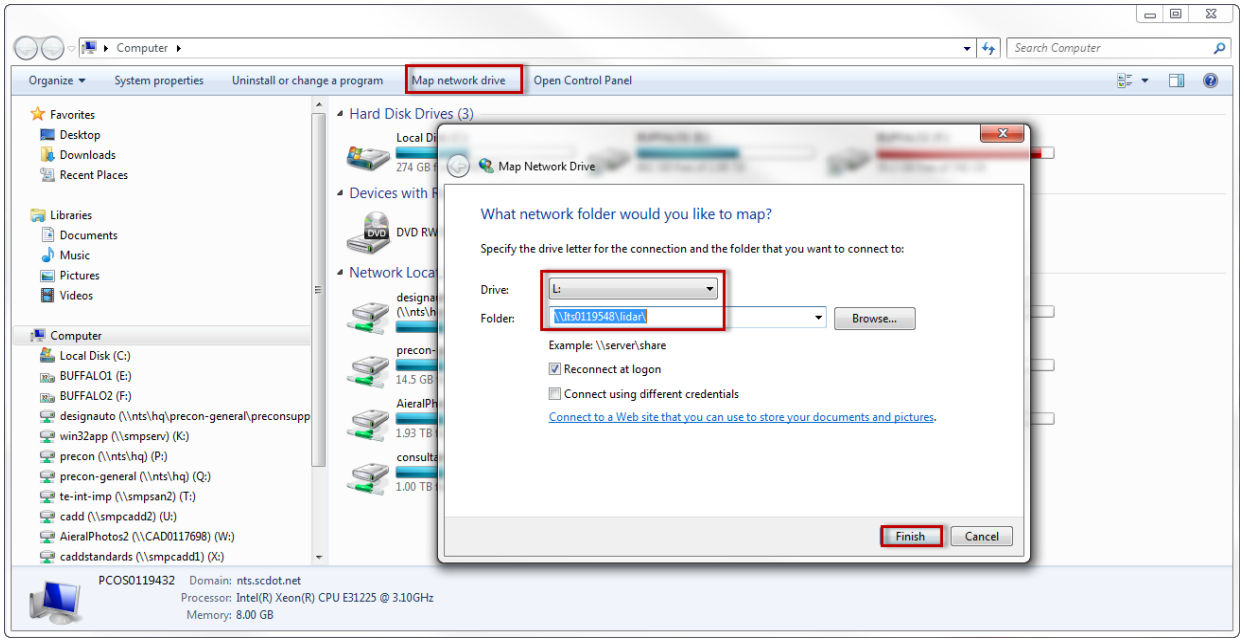
- Export KML

In MicroStation, select Tools>Google Earth (if not already open):

1. In MicroStation's Google Earth toolbar, click "Export Google Earth (KML) File" and key-in the name of the (KML or KMZ) file to export to Google Earth.
2. After clicking OK, Google Earth should automatically open (if not open already) and the cad elements will be exported into the Google Earth view at the proper location.

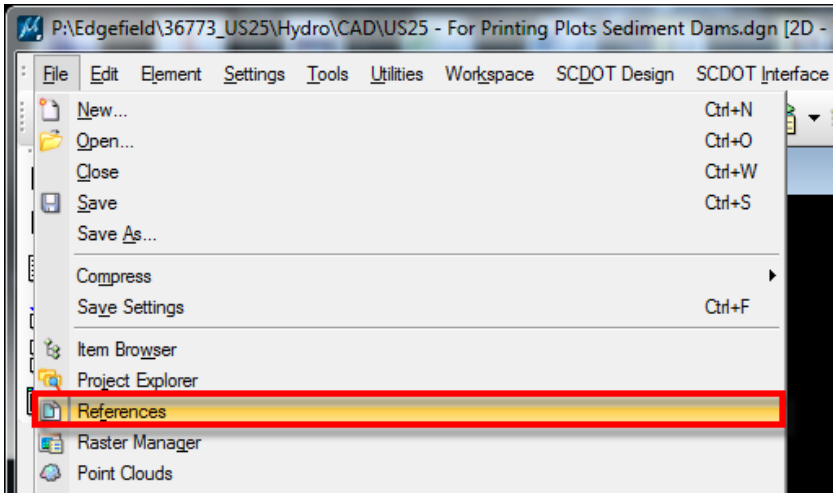
5. LiDAR

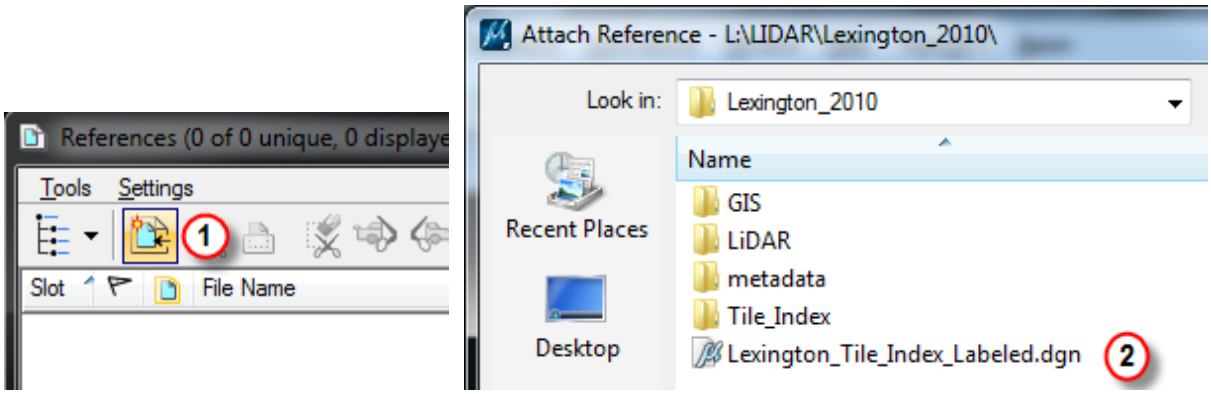
To create a .tin surface file from SCDOT's LiDAR data records, you will need access to the L: network drive (\\nts\hq\scdot-apps\scdot-lidar).

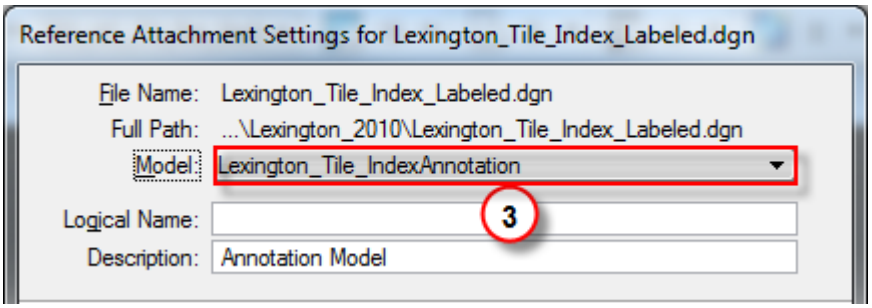


- Identify Required LiDAR Tiles

In Microstation, select the menu item File > References, then Tools > Attach.

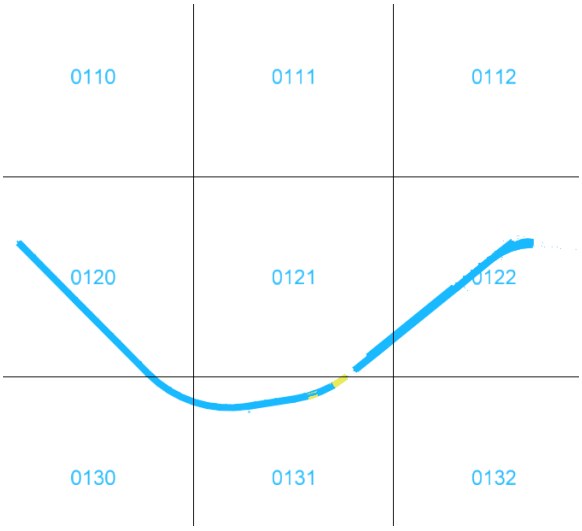






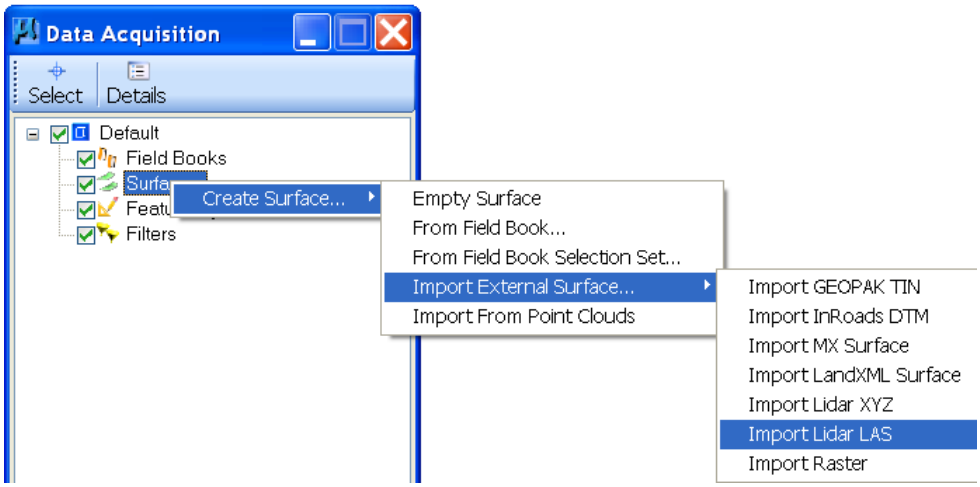
1	Attach	Navigate to L:\LIDAR and choose the appropriate county folder.
2	Tile Index	Select the Tile_Index file for the county.
3	Model	Select the County Annotation model in the Attachment Settings window.

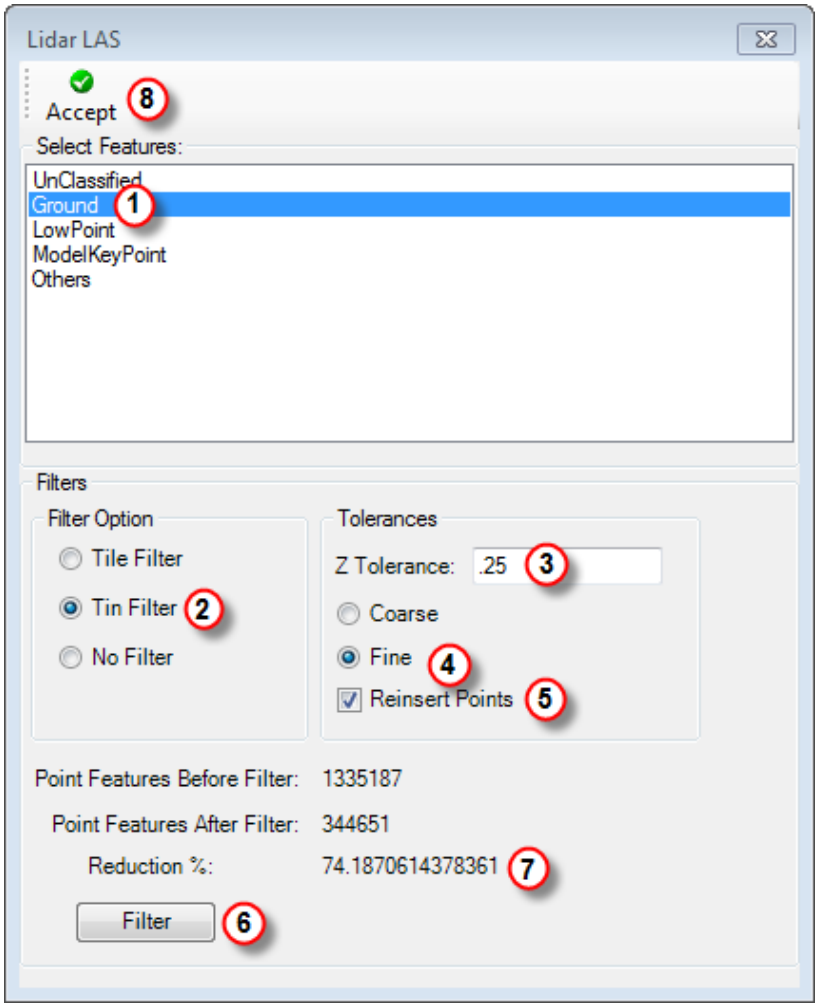
Check the numbered squares that correspond to the project location to determine which LiDAR tiles to import.



- Importing LiDAR Tiles

Activate Data Acquisitions (*Tools* > *Data Acquisitions*), right-click on Surfaces, and create a new surface by importing the first of the required LiDAR tiles.



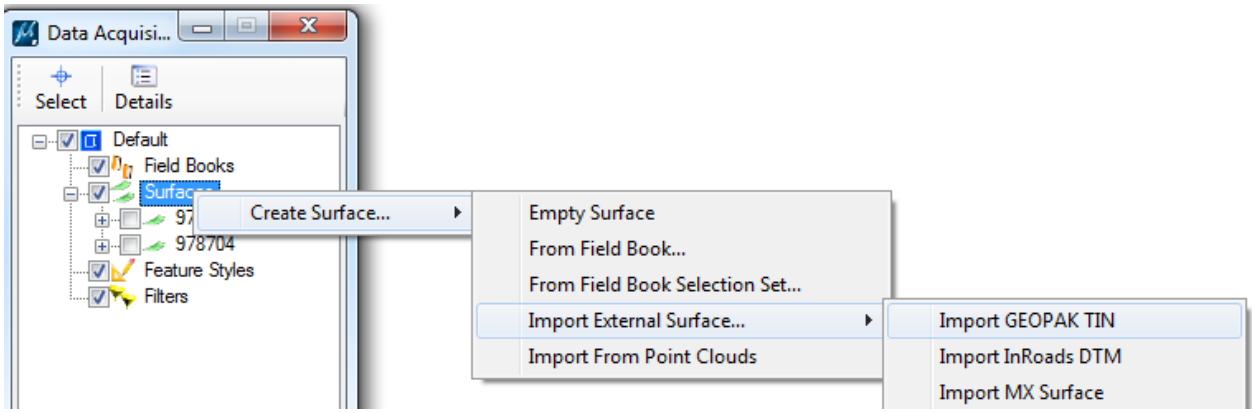


1	Select Features	Import only points listed as “Ground”
2	Filters	Tin Filter
3	Z-Tolerance	0.05 to 0.25, depending on site (see item 7)
4	Points Density	Fine
5	Reinsert Points	Toggle ON to maximize surface detail
6	Filter	Calculates Reduction % for proposed filter settings.
7	Reduction %	Shows reduction in number of points after proposed filtering. Adjust Z-Tolerance to achieve 60%-75% Reduction
8	Accept	Filters and imports the LiDAR surface using these settings.

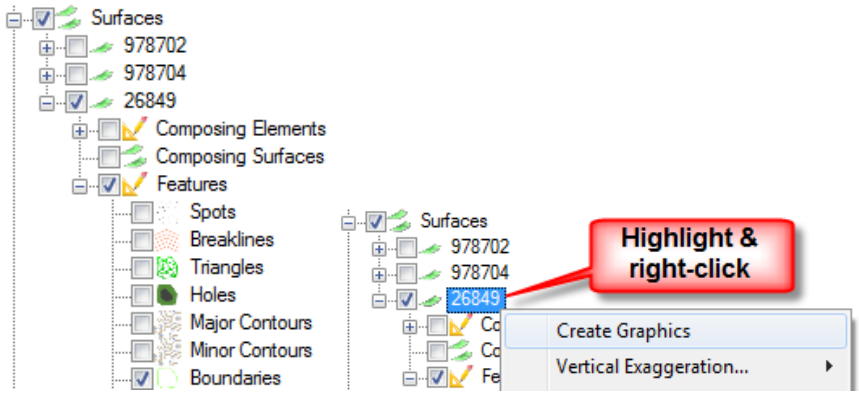
Repeat these steps for all required LiDAR tiles.

- Merge Tiles and Survey

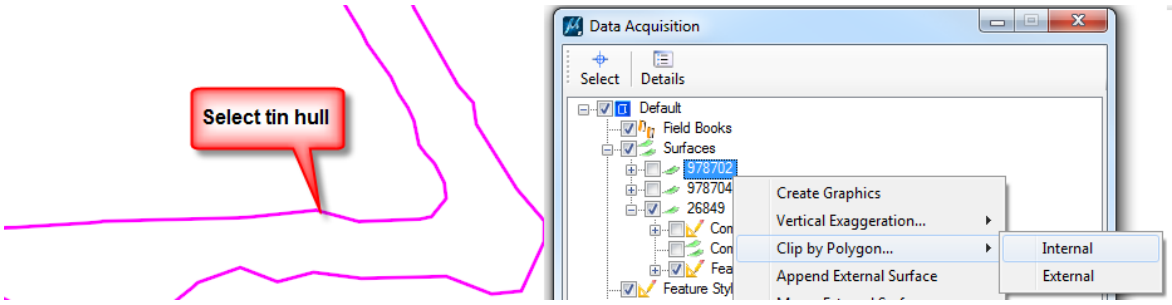
Locate and copy the project survey to the project directory. Import the survey as a Geopak .TIN file.



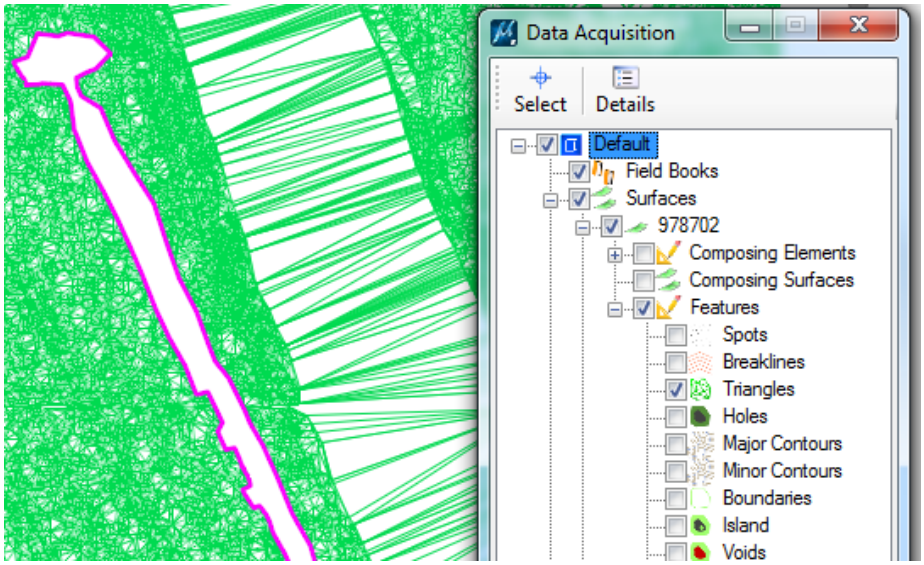
Expand the data tree for the project .tin until Features is expanded. Click to turn on the Boundaries. Highlight the surface for the project. Right click and select Create Graphics. (This draws the tin hull into the design file.)



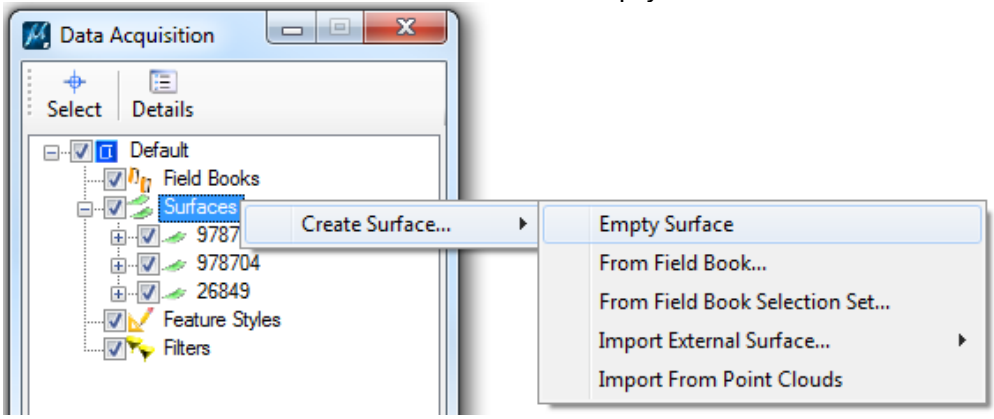
Use the MicroStation selection tool to select the tin hull. Right click on the first Lidar .las tile and select Clip Polygon > Internal. Repeat right clicking on the remaining Lidar .las tiles and select Clip Polygon > Internal for each one.



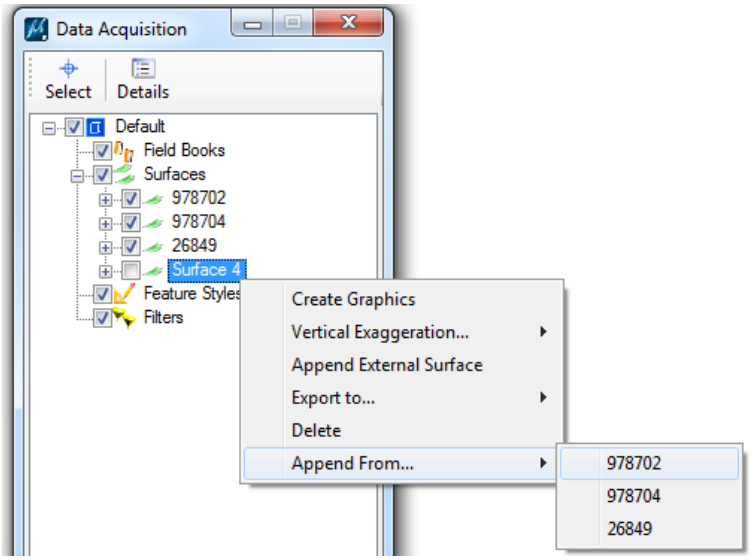
Expand the data trees for each of the LAS Surfaces and turn on Features -- Triangles to verify that the tin hull has been clipped from each of the tiles. (You can then turn the display back off).



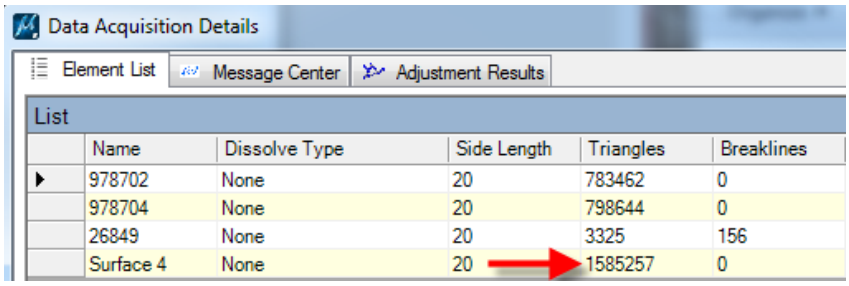
Right click on Surfaces and select Create Surface > Empty Surface.



On the newly created Surface, right click and select Append From > ###### (select your first .las surface in the list). Repeat until all of the las surfaces in the list have been added. (Do not add the project surface in this step).

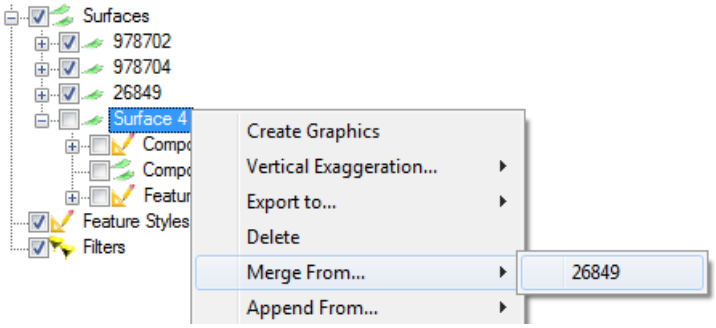


Pay attention to the # of Triangles in the Details panel. Occasionally it will bomb out and reset to 0 and you will have to delete the surface and start over again.



Name	Dissolve Type	Side Length	Triangles	Breaklines
978702	None	20	783462	0
978704	None	20	798644	0
26849	None	20	3325	156
Surface 4	None	20	1585257	0

Right click on the new project surface, select Merge From, and select the project surface.



Right click on the project surface and select Export to > Geopak Tin. Save the file.

Appendix A - C Value Table

The highlighted values are the most commonly used "C" values and are stored in the Drainage Library:

RUNOFF FACTORS FOR RATIONAL METHOD			
	Flat	**Rolling**	**Hilly**
	0% - 2%	**2% - 10%**	**Over 10%**
Pavements & Roofs	0.90	0.90	0.90
Earth shoulders	0.50	0.50	0.50
Drives & Walks	0.75	0.80	0.85
Gravel Pavements	0.50	0.55	0.60
City Business Areas	0.80	0.85	0.85
Unpaved Road, Sandy Soils	0.34	0.45	0.59
Unpaved Road, Silty Soils	0.35	0.47	0.61
Unpaved Road, Clay Soils	0.40	0.53	0.69
Apartment Dwelling Areas	0.50	0.60	0.70
Suburban, Normal Residential	0.45	0.50	0.55
Dense Residential Sections	0.60	0.65	0.70
Lawns, Sandy Soils	0.10	0.15	0.20
Lawns, Heavy Soils	0.17	0.22	0.35
Grass Shoulders	0.25	0.25	0.25
Side Slopes, Earth	0.60	0.60	0.60
Side Slopes, Turf	0.30	0.30	0.30
Median Areas, Turf	0.25	0.30	0.30
Cultivated Land, Clay & Loam	0.50	0.55	0.60
Cultivated Land, Sand & Gravel	0.25	0.30	0.35
Industrial Areas, Light	0.50	0.70	0.80
Industrial Areas, Heavy	0.60	0.80	0.90
Parks & Cemeteries	0.10	0.15	0.25
Playgrounds	0.20	0.25	0.30
Woodland & Forest	0.10	0.15	0.20
Meadows & Pasture Land	0.25	0.30	0.35
Unimproved Areas	0.10	0.20	0.30
Rail Yards	0.25	0.30	NA
Expressways & Freeways *	0.60*	0.70*	0.75*
* The designer can also calculate weighted ‘C’ values for expressways and freeways using the values in the table for pavement, side slopes and planted medians.			

Appendix B - Troubleshooting

Here are some common problems that are easy to correct. Check here before panicking and thinking that the program has gone haywire.

If your problem is not listed below, call your coordinator for help.

Cannot find Cell Library	Make sure that all of your drives are connected. You may have to copy the cell library and put it in your project directory. Make sure to update the project components.
Cannot find Land Use	Say OK and ignore
No active network selected	Got to Network>Active Network
Basins appear on screen but not in program	Make sure that you have the correct project open and pertinent levels on
Discharge is equal to 0 0	Check to be sure that the rainfall data matches the frequency option under Preferences
Flow line of downstream pipe is higher than flow line of upstream pipe	Make sure that you have the correct pipe sizes listed for all existing pipe or make sure design is selected new pipe
The nodes/links that I deleted are still on the screen	Go to Navigator and do a Graphic Update
Some nodes/links that I did not want deleted are not on my screen anymore but are in the Navigator	Go to Navigator and do a Graphic Update
When I hit Graphic Update only one nodes/link changed	Be sure to highlight everything that you want changed in the Navigator window
I have input 5 nodes/links - where are they?	Be sure that you are choosing Add after you enter all information and choosing Component > Node/Link > Add to create a new node/link. Also check to see if the level is on
I cannot see my labels	Check Preferences to see if you have labels checked. Do a Graphic Update. Check the label offset in Preferences.
The program keeps changing my set elevations/slopes	Check to see if your envelopes are large enough to accommodate the elevations you chose. Or you may need to make one of your nodes a drop box
I referenced my pp.dgn file, but I cannot see it on the screen	Go to File>Reference and make sure that there is an X under the Display column
The Macro for running reports didn't ask me to enable and it will not let me run nodesum and linksum reports.	In EXCEL go to Tools>Macro>Security. Make sure the security level is set to medium.

It keeps changing my held elevations and saying that I have exceeded the min deep at the basin	You may need to change that node to allow drop manholes instead of match soffit elevations. Your held elevations are making it impossible for your inflow and outflow pipes to have matching soffits. The program changes the elevations to match the specifications that you have set
Design/Analyze is grayed out	Got to Network Active Network and choose the network you want to run
When I try to move a piece of text in the profile the whole profile moves	Go to Settings/Locks/Graphic Group
It will not update my nodes. It says cannot find the cell library.	Download the cell library to your project directory and select it again in Preferences.