VDOT
GEOPAK Drainage
Training Manual

Training Manual 2004 Edition

TRN007630-1/0002

BENTLEY
www.bentley.com

Trademarks

AccuDraw, Bentley, the “B” Bentley logo, MDL, MicroStation and SmartLine are registered trademarks; PopSet and Raster Manager are trademarks; Bentley SELECT is a service mark of Bentley Systems, Incorporated or Bentley Software, Inc.

Java and all Java-based trademarks and logos are trademarks or registered trademarks of Sun Microsystems, Inc. in the U.S. and other countries.

GEOPAK is a registered trademark of GEOPAK Corporation.

Adobe, the Adobe logo, Acrobat, the Acrobat logo, Distiller, Exchange, and PostScript are trademarks of Adobe Systems Incorporated.

Windows, Microsoft and Visual Basic are registered trademarks of Microsoft Corporation.
AutoCAD is a registered trademark of Autodesk, Inc.
Other brands and product names are the trademarks of their respective owners.

Patents

United States Patent Nos. 5,815,415; 5,784,068 and 6,199,125.

Copyrights

The information and the software discussed in this document are subject to change without notice and should not be considered commitments by GEOPAK Corporation or Bentley Systems, Incorporated. GEOPAK Corporation and Bentley Systems, Incorporated assume no responsibility for any error that may appear in this document.

This class utilizes the 2004 Edition of Bentley GEOPAK Civil Engineering Suite.

TABLE OF CONTENTS

Lab 18: Importing Stream Geometric Data to HecRas

Ditch Design

Lab 19: Ditch Design

Digital Terrain Model

Introduction

Objectives	Learn how to display and analyze DTMs for use in roadway design.

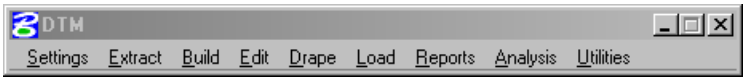
A **Digital Terrain Model (DTM)** represents the topography of a project in the form of a triangulated network. The DTM can be drawn in a 2D or 3D file, and then rotated to see the existing surface of the project area.

Digital Terrain Models can be generated from various sources including MicroStation elements, survey data, photogrammetry data, **GEOPAK** cross-sections, and geometry data.

Triangulation is a mathematical process applied to ground points and vertices along longitudinal features to create planar surfaces. The result of triangulation is the creation of triangles connecting these points. These triangles are included in the GEOPAK TIN file from which existing ground profiles and existing ground cross sections can be generated.

Accessing

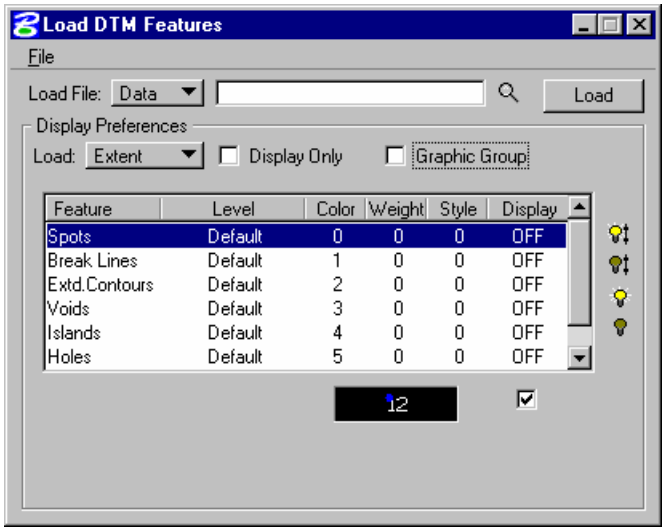
Clicking the **Existing Ground** button from the Road Project: Road1.prj dialog box within Project Manager invokes the tool frame shown to the right. All of the DTM tools can be accessed from the tool frame or from the DTM menu that can be accessed from the top/left icon on the tool frame.



Load DTM Features

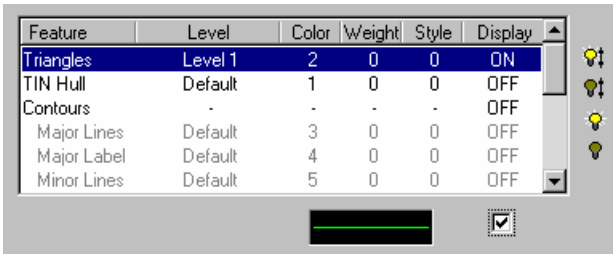
Tool	
Menu Bar	**Load > DTM Feature**

Load is the process by which we can visualize the DTM data, TIN model, lattice model, and contours. By selecting the Load DTM Feature icon, the following dialog box appears.



The user can choose to load the DTM data (**.dat**), the TIN file (**TIN**), or the Lattice file (**.lat**). Each of these files can be loaded for the model extents, within a fence, or selected view.

Activating the **Display Only** toggle enables the user to view the elements without writing them to the MicroStation file (temporary display). Conversely, deactivating the toggle writes the DTM features to the MicroStation design file at the defined symbology. These elements can be placed as a graphic group when **Display Only** is deactivated.



 Activates the display of all Features.

 Turns off the Display of all Features.

 Turns on only the selected Feature. This can also be accomplished by activating the toggle (below the list box) or double-clicking on an item that is turned off within the list box.

 Will turn off only the selected item. This can also be accomplished by activating the toggle (below the list box) or double-clicking on an item that is turned on within the list box.

Analysis Tools

Tools	
Menu Bar	**Analysis > Height** **Analysis > Profile** **Analysis > Volumes** **Analysis > Elevation Differences** **Analysis > Slope Area** **Analysis > Themes** **Analysis > Drainage Tools** **Analysis > Visibility** **Analysis > Trace Slope Path** **Analysis > DTM Camera** **Analysis > Trench Volumes**

The **Analysis** tools allow the user to visually analyze the digital terrain model utilizing numerous tools as a profile analysis, thematic analysis, drainage flow patterns, and visual portions on the model from any given location.

Height	To determine the height and other associated data dynamically based on user-defined data points within the model.
Profile	View a profile based on a user defined MicroStation element.
Volumes	To compute the volume between two TIN models, the volume between a TIN model and a plane, or the cut and fill totals between two TIN models while applying a shrinkage/swell factors.
Elevation Differences	Will display the elevation difference or the amount of cut and fill between two TIN models, or a TIN model and a plane of constant elevation.
Slope Area	The Slope Area tool displays the horizontal area and actual slope area (area following the terrain of the Model).
Themes	Displays the digital terrain model based on different user definable themes such as elevation ranges, slope percentage, slope degree, or aspect.
Drainage	Displays and analyzes drainage patterns within a TIN model. Tools include delineating watersheds, drawing flow arrows, determining upstream and downstream traces, finding high and low points, and ridge and sump lines.
Visibility	Based on a user-defined point of origin, GEOPAK visually displays which triangles can and cannot be seen, or what is visible between two points.
Trace Slope Path	Traces a path along a TIN file using a user specified slope and method.
DTM Camera	Allows a 3d visualization camera for the DTM
Trench Volumes	Compute the excavation volume needed along the drainage pipes.

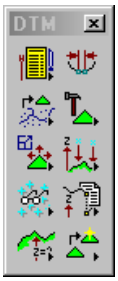
Lab 1: Digital Terrain Modeling

Accessing DTM Tools

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB1_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

Step 3. Select the **DTM Tools** frame from the **Applications > GEOPAK Road > DTM Tools** pull down menu.



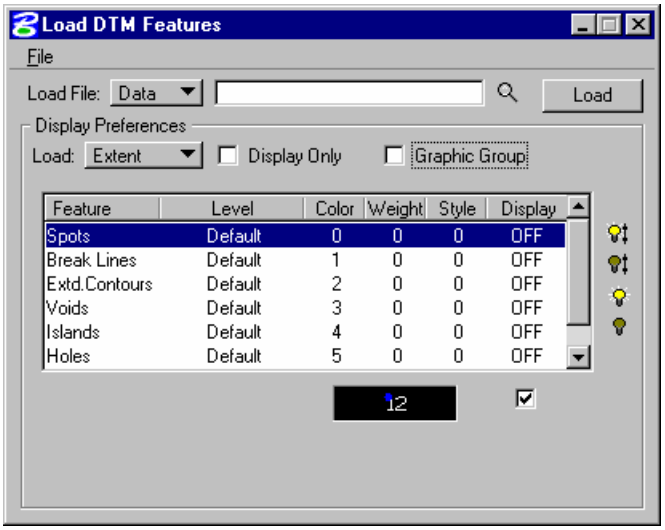
Load DTM Features

Step 1. Select the Load DTM Features icon from the tool frame to invoke the following dialog box.

Step 2. Change the Load File option to TIN.

Step 3. Select the TIN file 'SURVEY.TIN' by clicking the Files button.

Step 4. Enable the toggles for Graphic Group but leave the Display Only disabled.



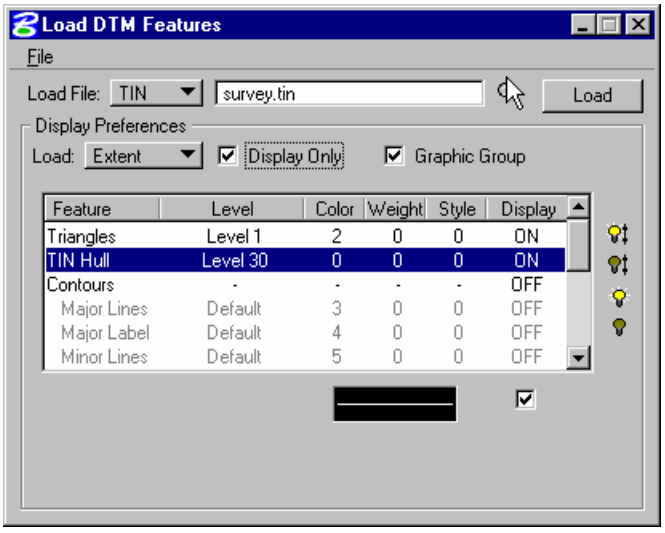
Step 5. Select the **Triangles** Feature by highlighting in the list box. Notice that initially the display is set to OFF. You can control the display of an item three ways:

Use the "lightbulb" icons to the right of the dialog box

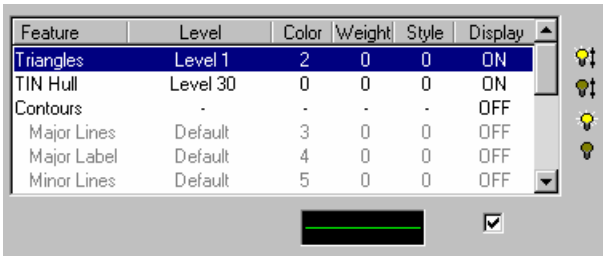
Double-click on an item

Use the "display" toggle in the lower right corner of the dialog box

Set the Triangles display to ON. At this point your dialog box should resemble the following:



Step 6. Set the display symbology for the Triangles as indicated below:



Step 7. Select the **Tin Hull** (the boundary of the surface model) feature. Activate the Display of the feature and set the symbology as follows; Level = Level 30, Color = 0, Weight = 0, Style = 0.

Step 8. Click **Load** and review the results.

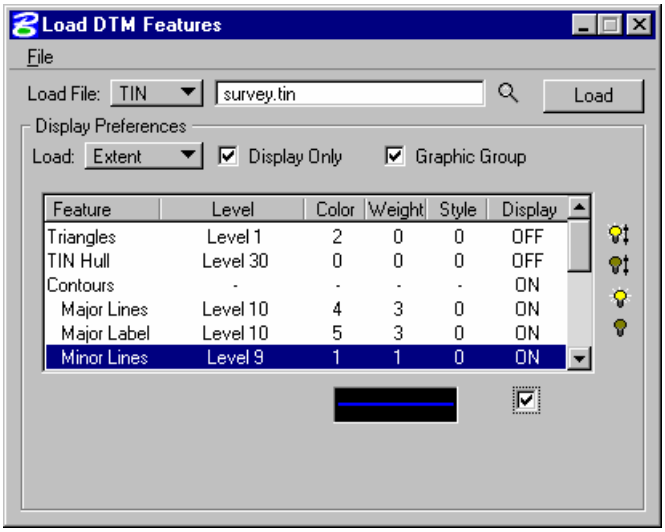
Step 9. When complete, toggle OFF the triangles and tin hull.

Step 10. Select the **Contours** feature. Activate the **Display** Feature.

Verify that **On** is set in the Display field.

Step 11. Complete the bottom of the dialog box as indicated below. To complete the Minimum and Maximum Z, click the **Read** button, which will review the active TIN file and determine these values.

Enable the **Display Only** toggle since we only want to "preview" our contours but not actually write them to the design file.



Step 12. Activate the Display of the Contour **Lines** and **Labels** and set the symbology as indicated below:

	Level	**Color**	**Weight**	**Style**	**Font**	**TH / TW**	**Distance**
Major Lines	Level 10	4	3	0			
Major Labels	Level 10	5	3	0	0	TH=3, TW=3	200
Minor Lines	Level 9	1	1	0			

Step 13. Click the **Load** button to initiate the process.

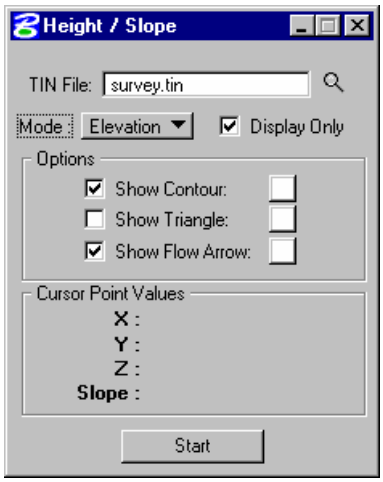
Step 14. Close the Load DTM Features dialog box.

Analysis Tools

We will review several analysis tools in the following exercises. These include the height, themes, profiles tools and drainage tools.

Height Tool

Step 1. Select the **Height** icon from the tool box to invoke the following dialog box.

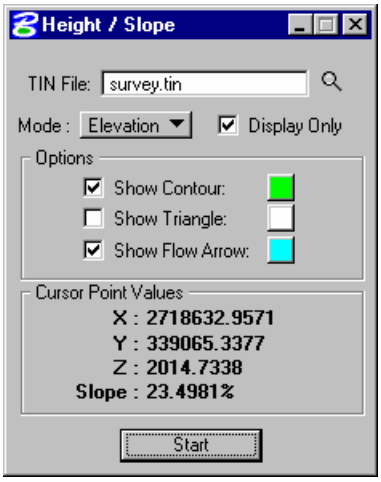


Step 2. Activate the Show Contour and Show Flow Arrow toggles.

The color may be altered to your choice of colors from the active color table on the Height dialog box. The weight of the contour line and arrow are controlled by the active MicroStation settings.

Step 3. Click the Start button and scan the surface model with the cursor to display the values for xyz and the slope on the DTM at the cursor point.

Step 4. Close the Height dialog box.



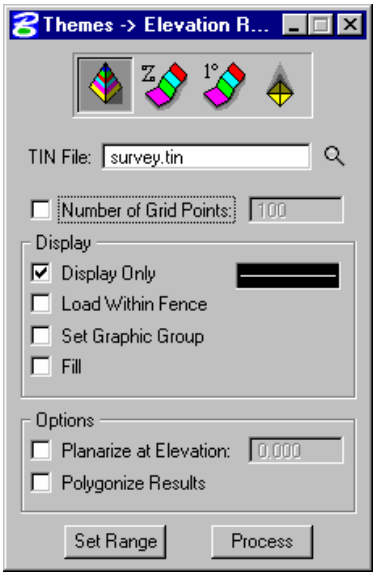
Themes Tool

Step 1. Select the **Themes** icon from the Analysis tool box to invoke the dialog box depicted below.

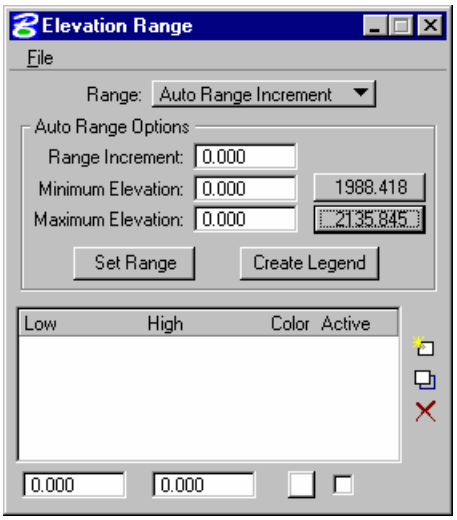
Step 2. Ensure that all graphic elements are fit in View 1.

Step 3. We will determine the Elevation Range for the **TIN File** 'SURVEY.TIN'. (This is the first icon on the dialog box).

Step 4. Activate the **Display Only** toggle.



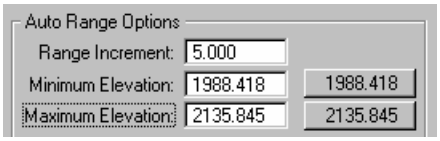
Step 5. Next, click the **Set Range** button to access the following dialog box.



As shown on the right of the **Minimum** and **Maximum** Elevation Fields, the values for the active TIN elevations are listed. By clicking these buttons individually, the elevation fields will automatically be populated.

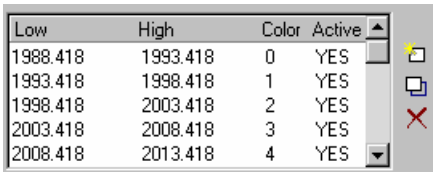
Step 6. Set the Range option to **Auto Range Increment**.

Step 7. Set the Auto Range Options as shown below:



Step 8. Click the **Set Range** button to populate the list box on the bottom portion of the dialog box.

Selection of any list of elevations will populate the fields at the bottom.



Utilizing the buttons on the right side of the list box will permit additional ranges to be included in the list, modifications to be made to elevation ranges (or the color display of this range), or deleting a particular elevation range.

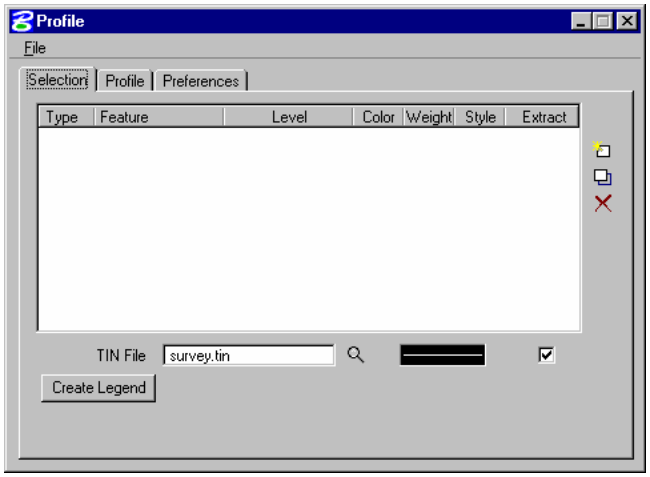
Step 9. Close the Elevation Range dialog box.

Step 10. Click the **Process** button to initiate the display of the Elevation Ranges for the surface model.

Step 11. Close the Themes dialog box.

Profile Tool

Step 1. Select the Profile icon from the Analysis tool box to invoke the following dialog box.

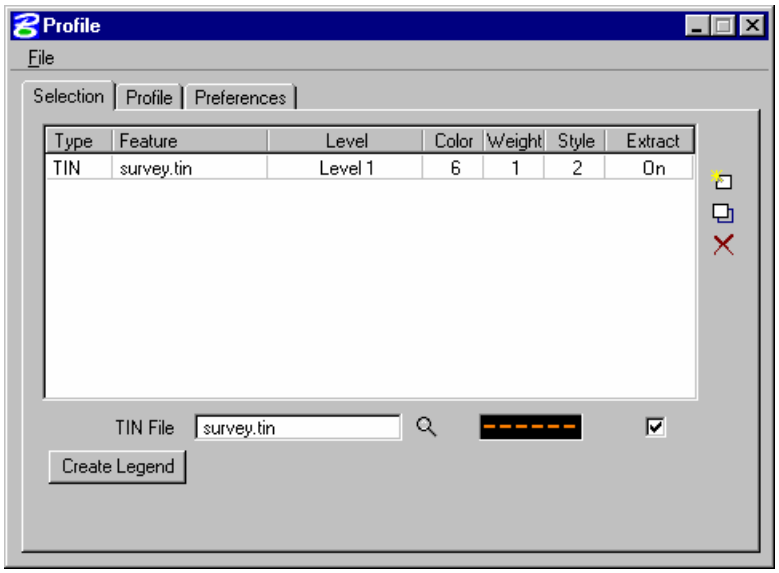


Step 2. Use the **Selection** tab to identify your surface model (TIN File) which you will generate a profile from, and the symbology for your profile.

Step 3. Select the Tin File "**SURVEY.TIN**" by using the **Select Files** icon.

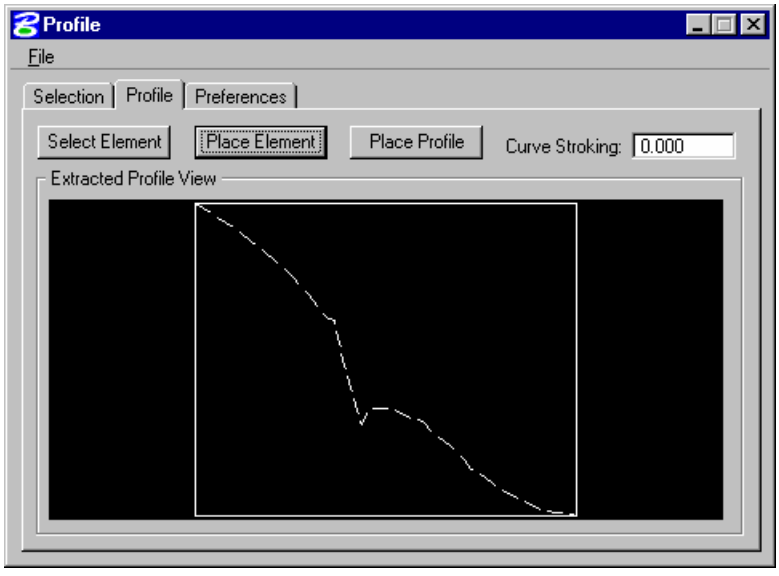
Step 4. Double-click on the symbology box (field to the right of the **Files** icon) to set the symbology representing the ground surface profile. You may choose any symbology you wish. Once the symbology has been set, click **OK**.

Step 5. Click the **Add** icon (top icon next to the list box) to add the feature to the collection box.



Step 6. Select the **Profile** Tab.

Step 7. Click the **Place Element** button to cut your actual profile at any desired area by just placing two data points (i.e. a line) or a series of data points (i.e. a line-string) across your surface model at any location.

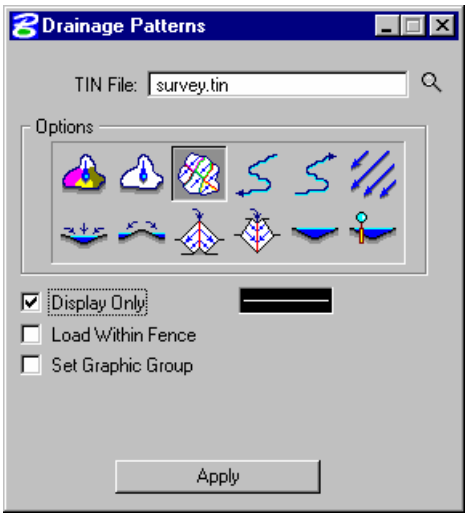


Step 8. You can use the **Preferences** tab to set up a grid for your profile if desired.

Step 9. Close the Profile dialog box.

Drainage Tools

Step 1. Select the Drainage Tools icon from the Analysis tool box to invoke the following dialog box.

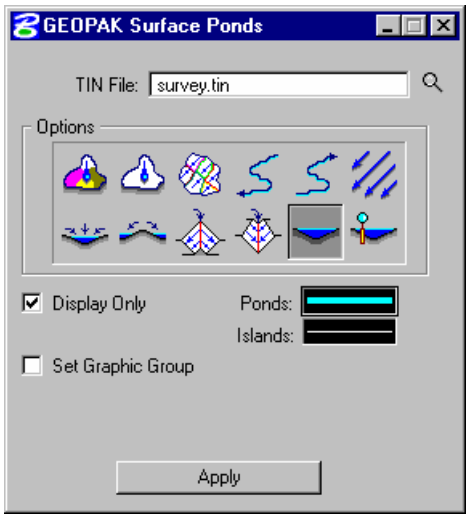


Step 2. Select the Drainage Patterns icon.

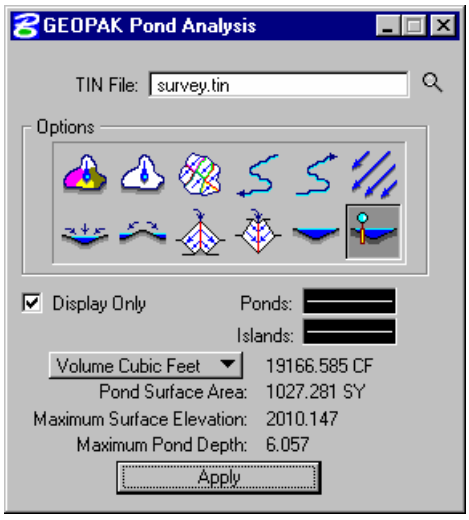
Fill out the dialog as shown.

Click the **Apply** button. The drainage flow lines are drawn into the design file.

Step 3. Select the GEOPAK Pond Analysis icon, and click the **Apply** button.

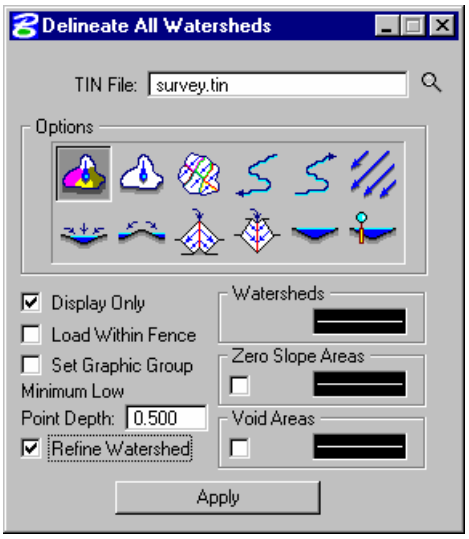


Step 4. Select GEOPAK Pond Analysis dialog. Press **Apply** and data point inside the predelineated pond at station 206+00. The volume is calculated as shown below.



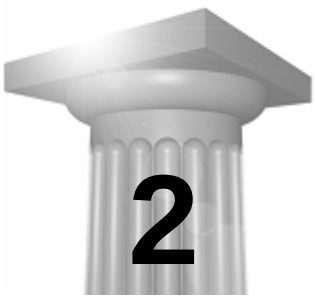
Step 5. Select the Delineate Watersheds icon.

Fill out the dialog as shown.



Click the **Apply** button. The watersheds will be drawn into the design file. Make sure the MicroStation View Attribute 'Fill' is toggled on. (Settings > View Attributes)

Step 6. Exit MicroStation.



2 Getting Started

Objective:	Introduction to GEOPAK Drainage.

Overview

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

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

Inlet design and analysis capabilities include Curb, Slotted Drain, and Grate inlets both on grade and in a sump. Inlet designs may proceed through a user specified library of standard inlet sizes to determine the most suitable design for a given set of user prescribed constraints. Inlets may also be designed or analyzed with by pass flows from one inlet to another, including by pass flows between inlets of different networks.

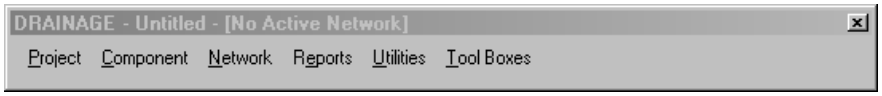
Most standard pipe configurations may be designed and analyzed including arch, box, circular, elliptical and pipe arches. A library of available sizes in most common materials (Aluminum, Concrete, Steel, and Plastic) is provided for the designer to select sizes or simply specify types for GEOPAK Drainage to select the appropriate size. Pipe design selections may be optimized for both size and depth of cover. Trapezoidal ditches may also be designed or analyzed anywhere within a storm drain network. The pipe and ditch hydraulics include backwater curve computations and junction loss options.

GEOPAK Drainage File (GDF)

The first component is the GEOPAK Drainage File (*.gdf). This binary file contains all the hydraulic information about the drainage system. Each GEOPAK Drainage Project contains Preferences, which include hydrologic and hydraulic computation options, visualization symbologies, and project file references to the Drainage Library and GEOPAK GPK files. Spatial data, connectivity and hydraulic properties for each drainage feature are stored in this external file. Sufficient data is maintained to rebuild the graphical representation of the Drainage Project with the exception of certain features, which are defined by MicroStation elements (i.e. Drainage Areas).

Main Menu Bar

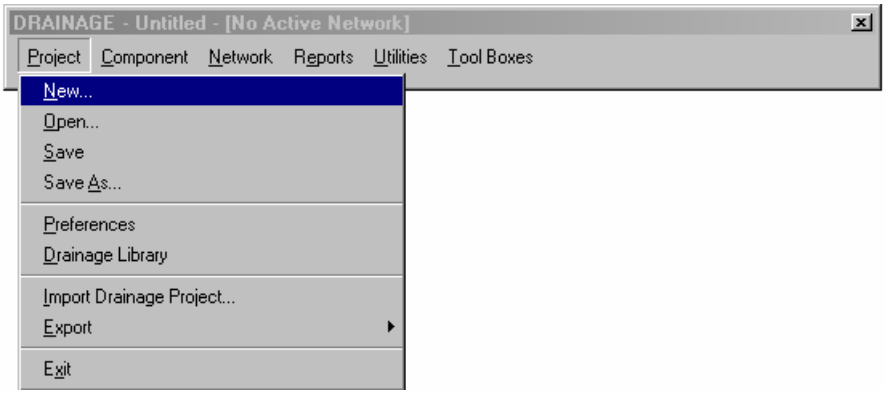
GEOPAK Drainage is invoked from within a two dimensional MicroStation graphics file. At the Applications pull down, select GEOPAK Drainage, which accesses the software and displays the main GEOPAK Drainage menu bar as depicted below.



Each menu selection accesses GEOPAK Drainage project information, feature placement and tools necessary to complete a GEOPAK Drainage project. Tool frames and toolboxes are also supported for all functions and are accessed via the Tools pull down.

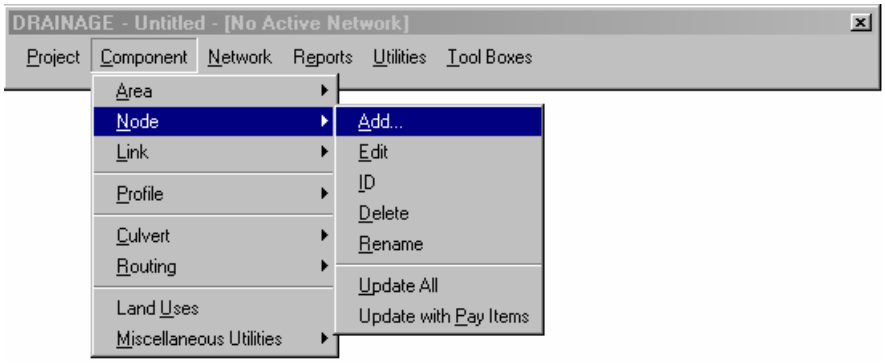
Project Menu Selections

The Project Menu selections are utilized for creating new GEOPAK Drainage projects, opening existing projects, saving projects, establishing the project Preferences, editing the Drainage Library, importing and exporting between GEOPAK Drainage Files (gdf).



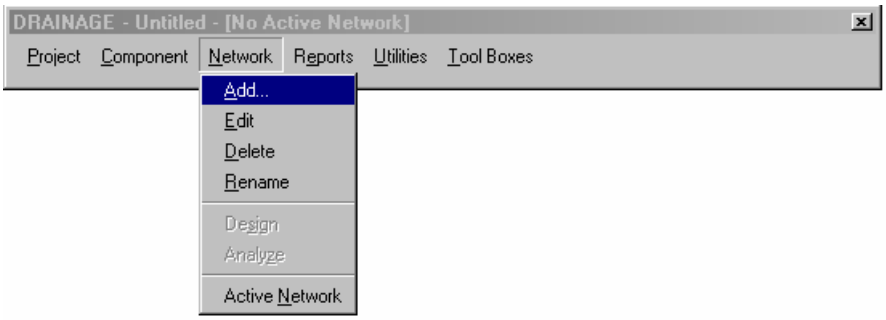
Component Menu Selections

The Component menu selections provide the mechanism to add, edit and delete the various elements, which comprise a drainage system including Areas, Nodes, Links, Reaches, and Land Uses. Each of these tools invokes a dialog wherein the specific component information can be added or edited.



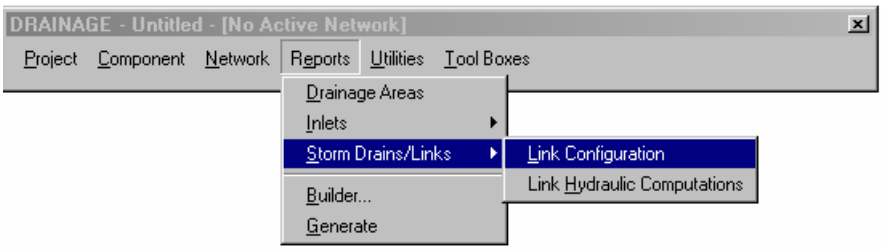
Network Menu Selections

The Network menu selections provide tools to add, edit, delete, and manipulate a drainage Network. While work is being done on a particular Network, it can be identified as the Active Network, which dictates which features are contained in the calculations and provides quick access to the components of a specific Network. Once a Network is added to the project and set as the Active Network, the Design and Analyze options perform their respective hydraulic procedures on the Network.



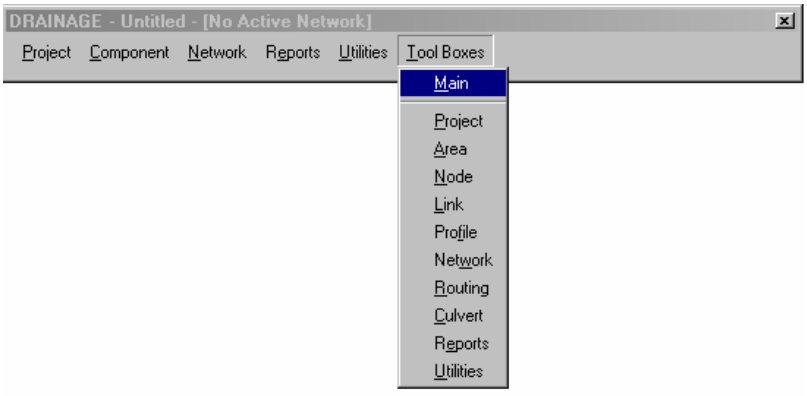
Reports Menu Selections

The Reports menu selections provide access to the predefined hydrologic and hydraulic reports available and to the Custom Report Builder and Generator.



Tools Menu Selections

The Tools menu selections provide access to all Tool Boxes, Navigator, Labeler, Culvert, Routing and DTM Drainage Tools.



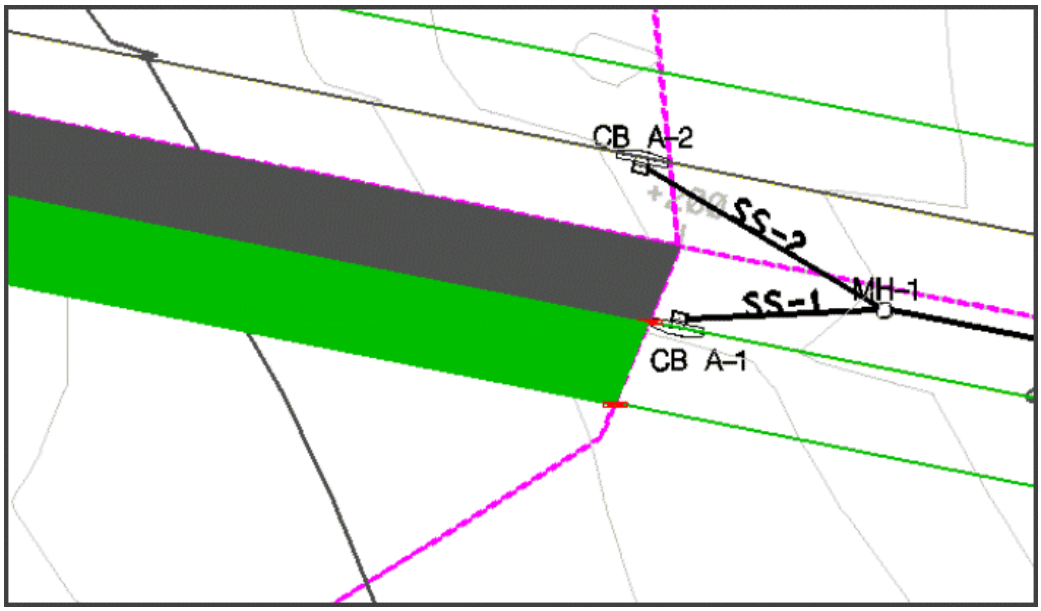
Project Workflow

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

Roadway alignments, vertical profiles, and digital terrain models may be used throughout GEOPAK Drainage to provide pertinent information to the drainage design. All drainage components feature interactive graphical placement tools for easy definition of the drainage system.

Each of these components (inlets, areas, and pipes) is composed of two basic types of information:

- Spatial information describing its location, shape and connectivity.
- Hydraulic and Hydrologic information describing its properties, conventions and other associated attributes.



Drainage Components

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

- Nodes: A node (inlets, manholes, etc.) is a point with a user-defined location. The location may be in Cartesian coordinates (x,y) or in curvilinear coordinates (station, offset).
- Links: A Link represents a linear feature depicting a path connecting two nodes, traversing upstream to downstream. The path may be straight line or curvilinear (along a graphic element).
- Networks: A network is a system of interconnected nodes and links that form a system through which water can flow to a single outlet node. A drainage project accommodates any number of Networks.

Other associated components in GEOPAK Drainage include:

Areas: A drainage area can be represented by a closed boundary or simply keyed-in (acres or hectares). All flows from a single drainage area are tributary to a single Node. There is a ***one to one***

correspondence between a node and an area. Therefore areas and nodes share the same name (ID). A drainage area may contain multiple subareas representing homogeneous features such as soil types and land uses (“C” values), thereby allowing composite "C" value calculations.

Reaches: A reach represents a path connecting any two nodes in a Network. Reaches provide the visualization of profiles between any two nodes in a drainage network.

LAB 2: Getting Started

File Information

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB2_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

The following files are utilized for this project. The directory path for the sample project is **c:\data\geo\VDOT\drain1**

Drainage Design File	h17682.dgn
Reference File (land use file)	h17682da.dgn
Reference File (proposed roadway file)	d17682des.dgn
Reference File (existing topography)	s17682.dgn
English Drainage Library	vdot.dlb
English Cell Library	drainage.cel
GPK Alignment File	job101.gpk
GEOPAK Drainage File (to be created)	h17682.gdf
Existing Triangulated Irregular Network file (TIN)	survey.tin
Proposed Triangulated Irregular Network file (TIN)	proposed.tin

Invoke GEOPAK Drainage

Step 1. Access GEOPAK Drainage from MicroStation's pull-down menu: **Applications > Geopak Drainage > Drainage**.

Step 2. Create a new drainage project: **Drainage > Project > New**.

Step 3. Name the new project : **h17682.gdf.**

Step 4. Exit MicroStation.

3 Drainage Library

Objective:	Review the Drainage Library.
Goal:	Review the various options contained within the Drainage Library.

Drainage Library

The Drainage Library is used to store data and standards which may be shared by different projects and designers. Each GEOPAK Drainage project simply accesses items within the library for use on the specific project. The library contains five major categories:

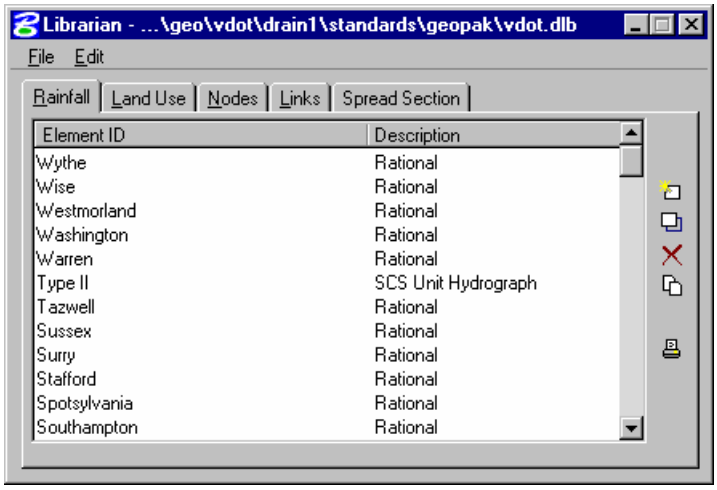
- Rainfall Data
- Land Use Runoff Coefficients
- Nodes (Inlets, Junctions, etc.)
- Links (Pipes)
- Spread Sections

Note: The Library is unit specific, i.e., Metric or English.

The Drainage Library is invoked by selecting *Tools* > *Tool Boxes* > *Main*, then identifying the **Drainage Library** tool, (second column, top box) as depicted to the right.

Alternately, selecting *Project* > *Drainage Library*, from the MicroStation Menu bar, can also invoke the Drainage Library.

The main Drainage Library dialog is displayed on the screen as depicted below.

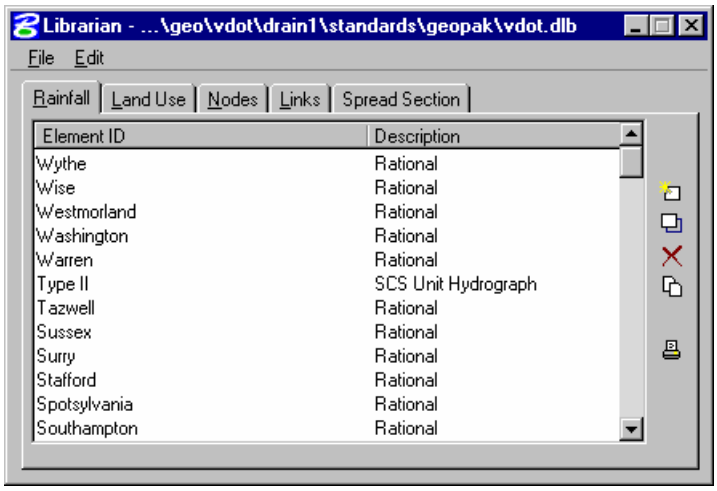


Five tabs are supported for the major categories within the Drainage Library:

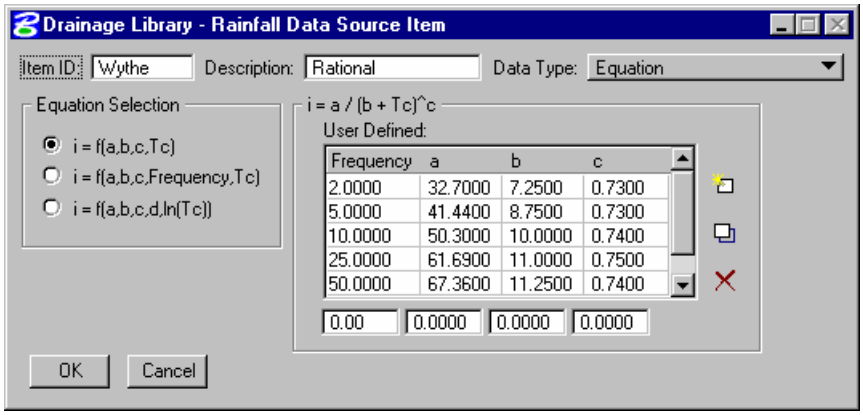
- Rainfall
- Land Use
- Nodes
- Links
- Spread Section

Rainfall Items

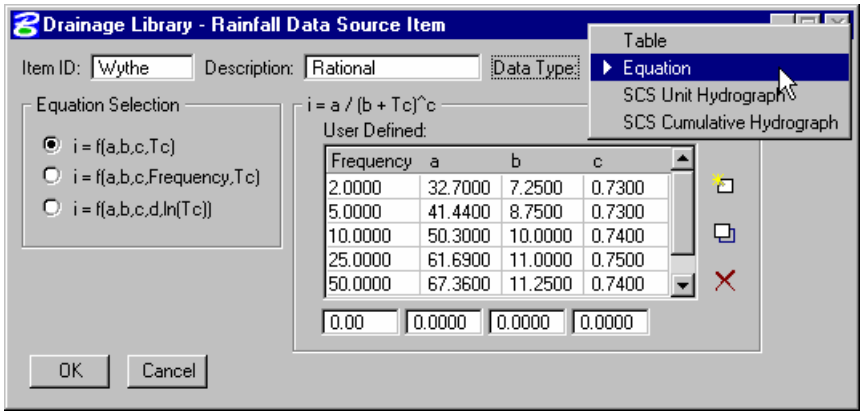
Within the Drainage Library (beneath the Rainfall tab) are the Rainfall Items, which store all the rainfall data sources to be used on subsequent GEOPAK Drainage Projects. GEOPAK Drainage supports rainfall sources in the form of intensity duration frequency (IDF) tables, two SCS methods, or as coefficients for three different preset intensity duration equation formats.



When an Item is modified or added, the Rainfall Data Source Item dialog is invoked as depicted below.

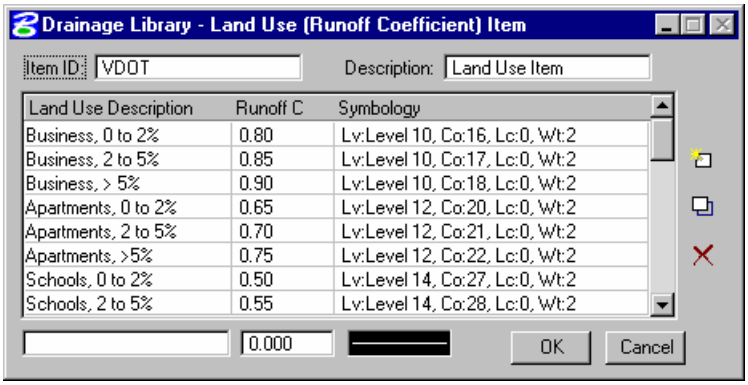


Four Rainfall Data Types are supported as depicted in the exploded view below. Selection of each option dynamically changes the dialog to reflect the selection.



Land Use Items

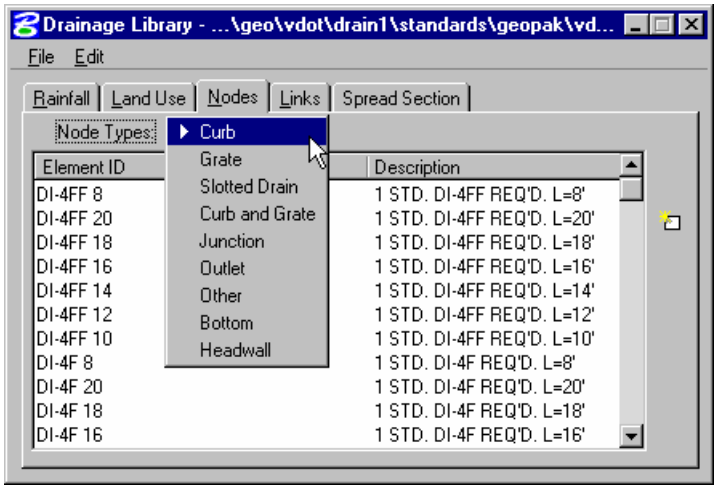
Land Use Items take advantage of the drainage area delineation tools in GEOPAK Drainage and are used to store runoff coefficients and corresponding graphic symbology for various land uses contained in Drainage projects. Under the Land Uses tab, the designer may create as many Items as desired. The Land Use Item dialog is depicted below.



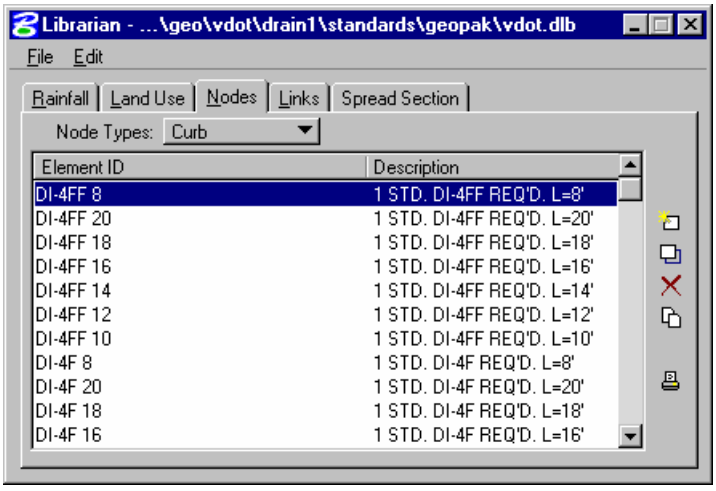
Item ID	Land Use Item Identification to recall the item from within a GEOPAK Drainage project. (Maximum of 32 alphanumeric characters).
Description	Description of the Land Use Item. (Maximum of 32 alphanumeric characters).
Land Use Description	Description of each Land Use. (Maximum 32 alphanumeric characters).
Runoff C	Runoff coefficient for the Rational Method (C Value).
Level, Color, Style, Weight	Specified element parameters which uniquely identify the various land use descriptions.
Edit Buttons:	To add to the list box, simply key in the required values in the edit fields directly below the list box, then press the **Add (+)** button to the right.
Add (+)	
Delete (-)	To delete a line from the list box, simply highlight the line to be deleted and press the **Delete (-)** button.
Modify (/)	Modify a line by highlighting the desired line in the list box; make the desired changes in the edit fields directly below the list box, then press the **Modify (/)** button.
OK / Cancel	Action buttons which also close the dialog

Node Items

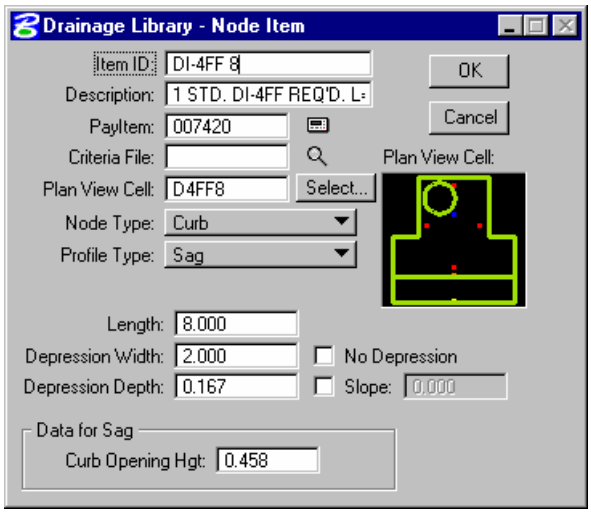
Node Library Items define the type of Node structures used on subsequent GEOPAK Drainage projects. The Nodes contain standard inlet configurations for Grate, Curb, and Slotted Drain inlet as well as Junctions (i.e., manholes), Outlets, and Other generic Nodes (i.e., grade breaks) as may be required for various Drainage projects. The description, plan view representation, and dimension information for each Node item is entered only once in the library and then referenced by each project.



Under each Node Types are various stored Node configurations. A typical listing may appear as follows:

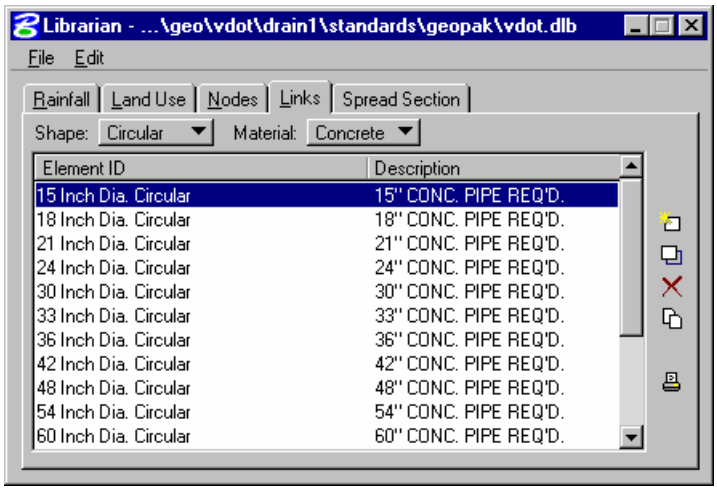


All Node items have certain characteristics in common. The typical Node item dialog and the common data requirements are shown below.

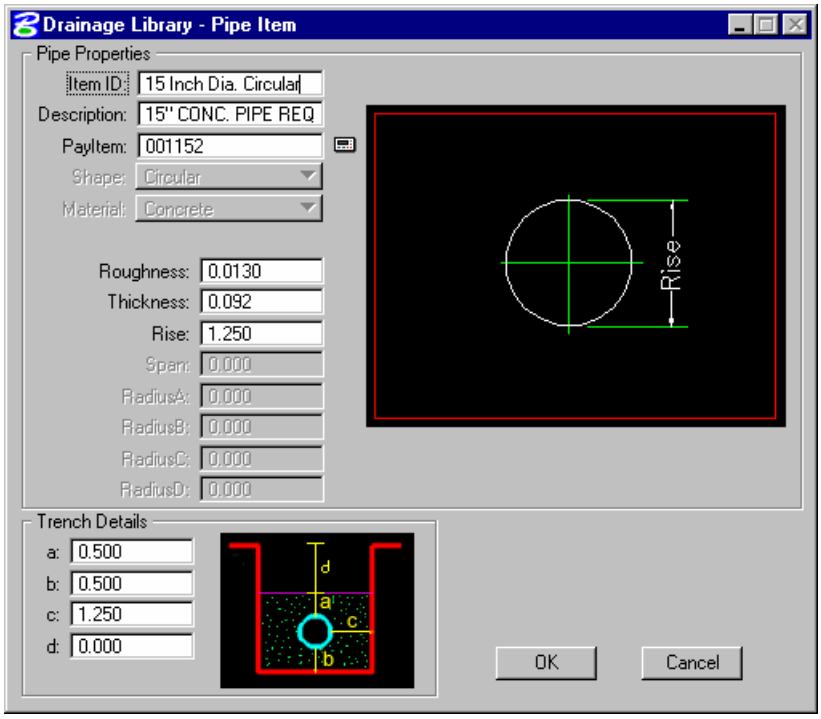


Link Items

The Link Items are used to store all the pipe configurations that may be used on a GEOPAK Drainage project and is depicted below.



The *Links Tab* is the source of shapes, materials, dimensions, and hydraulic properties of these features. When designing these components, the items serve as the means to specify design candidates. If pipe items are deleted, these items will no longer be available for use on a project using the specific library.



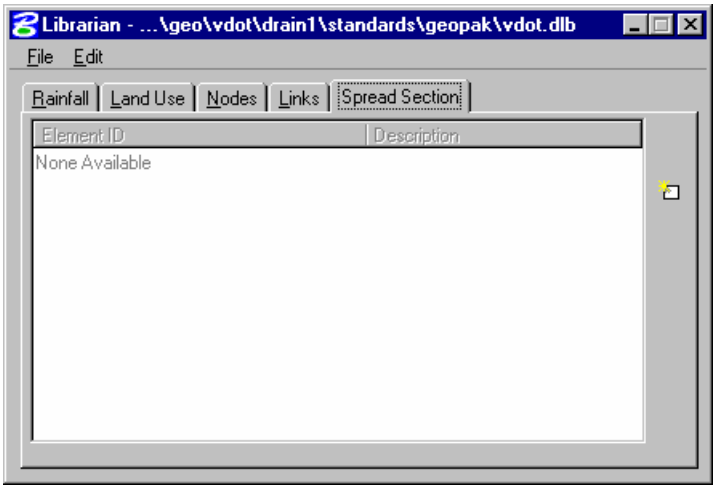
The Links are categorized by three types of properties for each pipe:

- Shape
- Material
- Type or corrugations

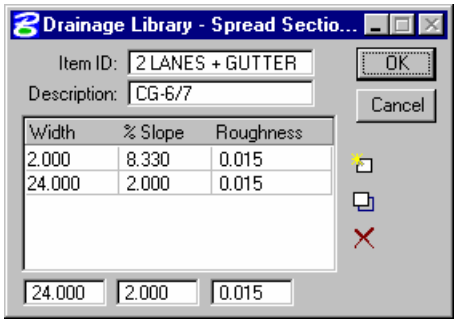
Each pipe contains the specific pipe sizes and default roughness coefficients for each shape, and material combination. There are English and Metric libraries provided with GEOPAK Drainage, which contain virtually every standard pipe configuration.

Spread Section

The Spread Section tab is utilized to store roadway pavement spread information. When selected, the dialog dynamically changes as depicted below.



When double clicking on an entry in the list box, the dialog depicted below opens.



Input is available for the Spread Width, Slope, and Manning's N value.

LAB 3: Drainage Library

Review Drainage Library

The *Drainage Library* is used to store hydraulic, hydrologic, and construction standards, which may be shared by different projects and designers. Each GEOPAK Drainage project accesses items from the *Drainage Library* for use on the specific project.

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB3_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the Geopak Drainage pull down menu.

Step 5. Select **Project > Drainage Library**. The library stored in the Preferences will be opened by default.

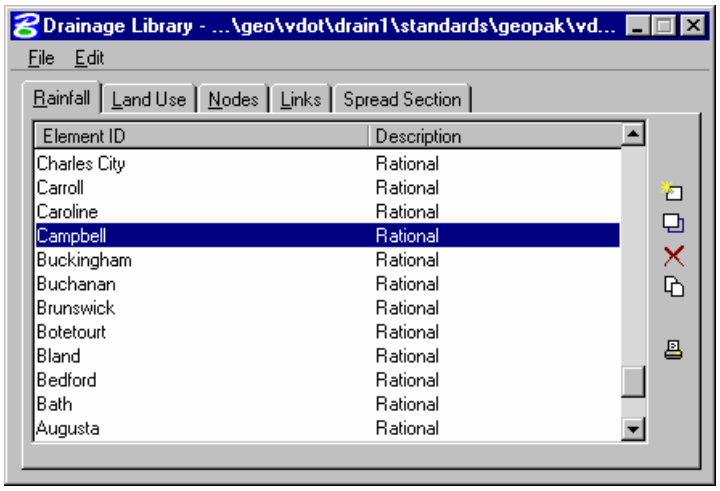
The Drainage Library currently contains five (5) tabs as indicated below:

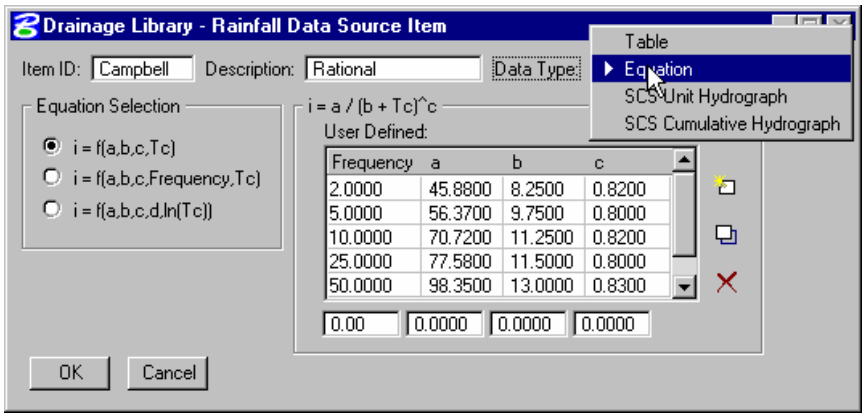
- Rainfall: Rainfall Data Source
- Land Uses: Land Uses and their corresponding “C” values and symbology
- Nodes: Inlets, Junctions, Manholes, Outlets, etc.
- Links: Circular Pipes, Elliptical Pipes, Pipe-Arch pipes, Boxes, etc.
- Spread Section: Inventory of varying Spread Cross Sections

Rainfall Tab:

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

Step 1. Select the **Rainfall tab**, highlight Campbell, and select **Modify:**



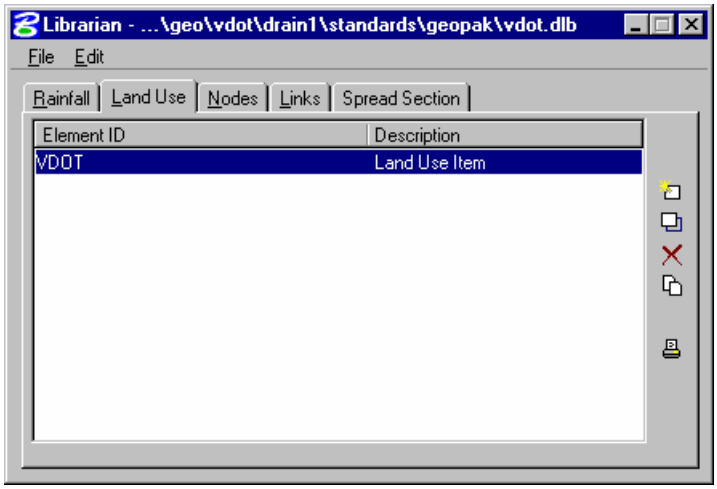


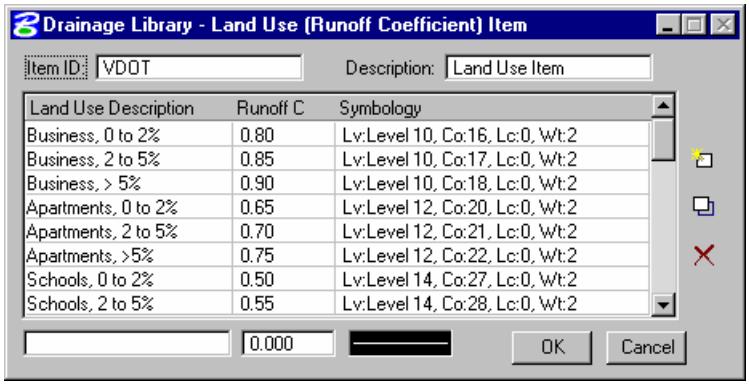
Review the various Data Types of Rainfall accepted.

Land Use Tab:

The Land Use Tab is used to store runoff coefficients ("C" values) and corresponding graphic symbology for each land use. Land Uses can then be delineated automatically using the selected symbology.

Step 1. Select the **Land Use tab**, highlight the VDOT Land Use item and select **Modify:**



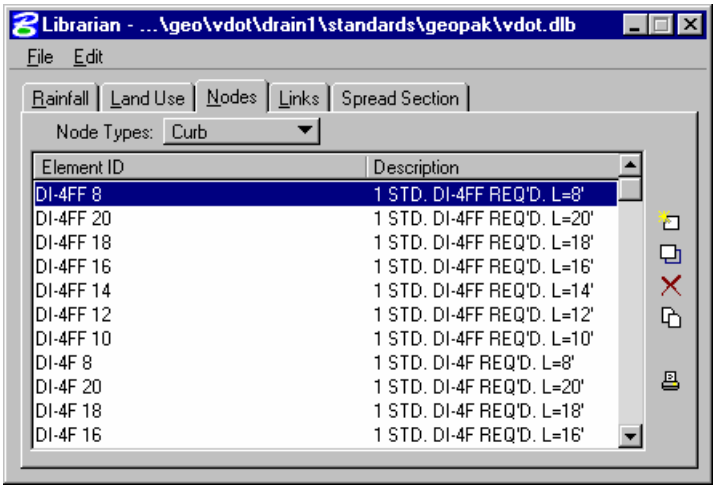


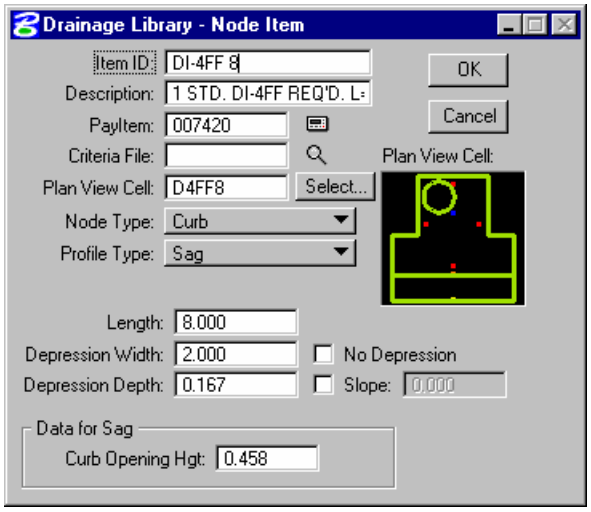
Review the various land uses and their associated symbology.

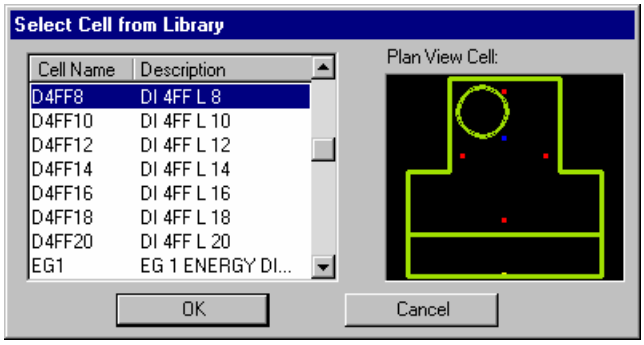
Nodes Tab:

The Nodes Tab contains standard configurations for Grates, Curbs and Slotted Drain inlets, as well as Junctions, Outlets and Other Nodes. The description, plan view representation, and dimensional information are stored for each node.

Step 1. Select the **Node tab**, highlight the first curb inlet, and select **Modify**:







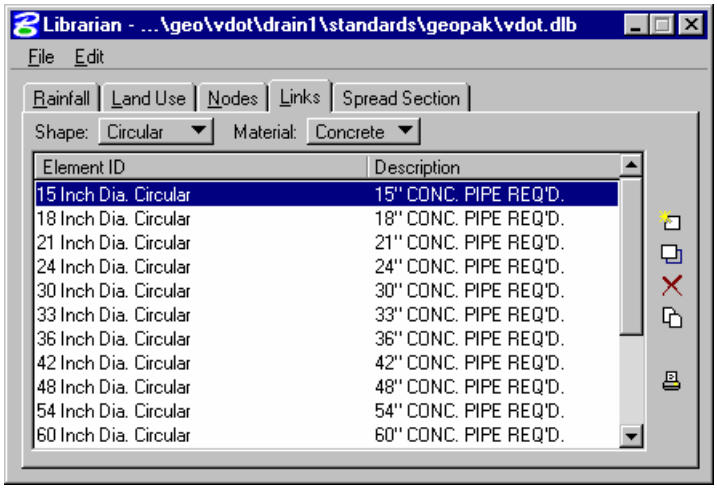
Review the various geometric inputs required for the Nodes.

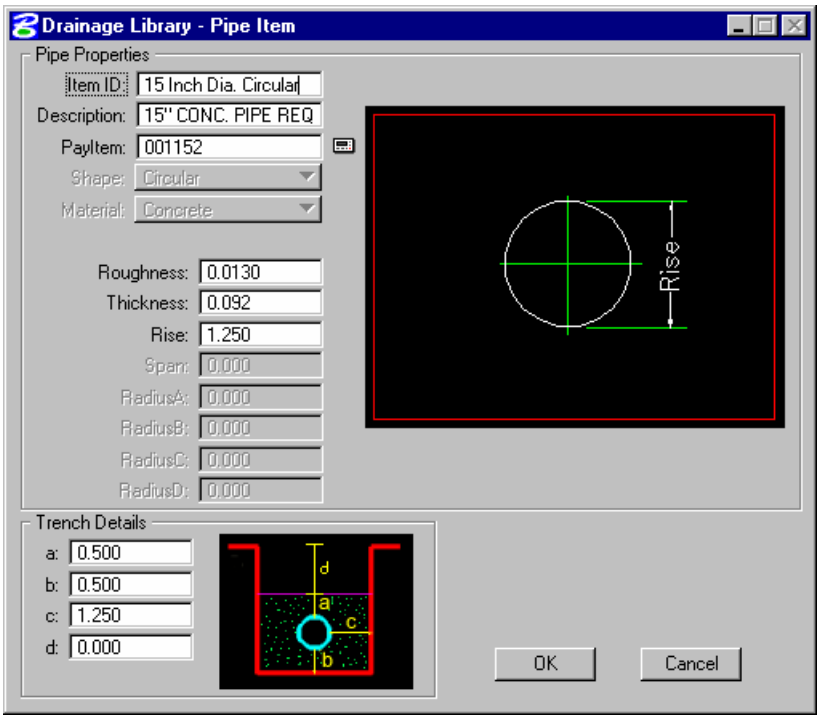
Links Tab:

The *Links Tab* contains all pipes to be used on the Drainage project. Each pipe is categorized by three properties: Shape, Material, and Type (corrugation); and contains information regarding specific pipe geometry, default roughness coefficient, and material combination.

Step 1. Select the **Links tab**, and switch the Shape to Circular and Material to Concrete.

Highlight the second Circular Concrete pipe, and select **Modify:**



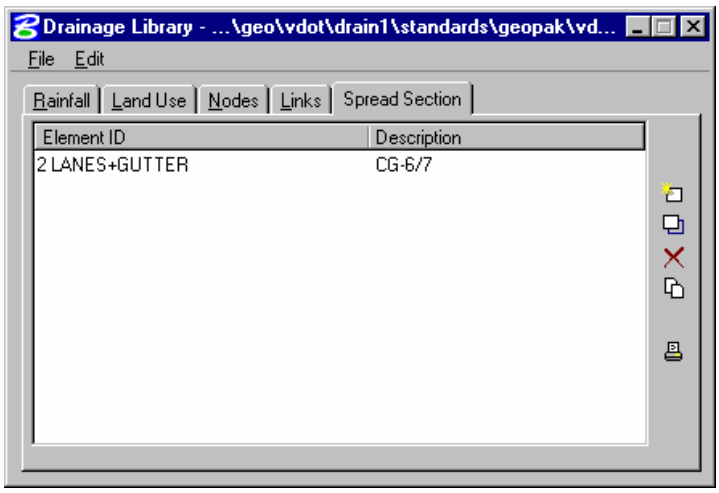


Review the various geometric inputs required for the Links.

Spread Section:

The *Spread Section* stores standard cross sections (for roads, gutters, etc.) to be used on GEOPAK Drainage Projects.

Step 1. Select the **Spread Section tab**, highlight the 2 LANES+GUTTER section, and select **Modify:**



Review the spread cross-section characteristics for each spread item.

Step 2. Exit MicroStation.

4 Project Preferences

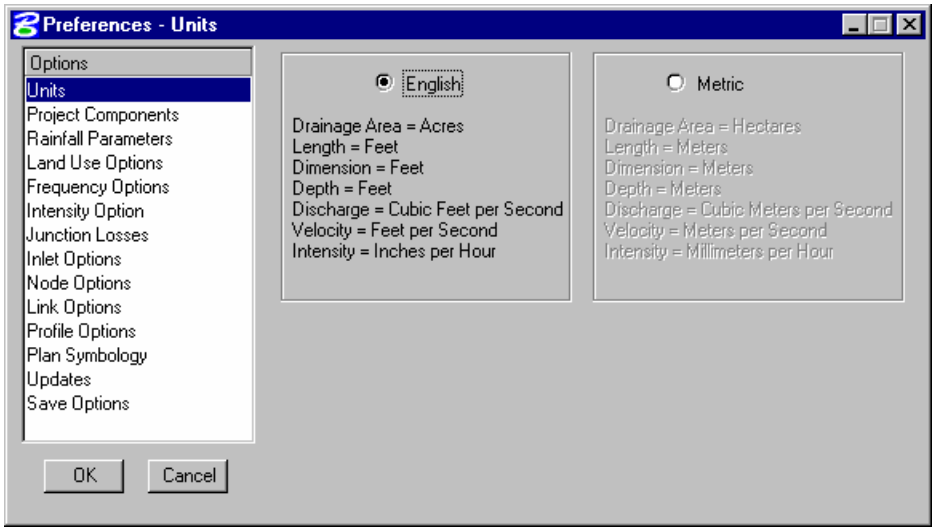
Objective:	Review the Project Preferences.

Introduction

GEOPAK Drainage supports a wide array of user defined Preferences which enables the designer to set project specific options, or a large organization to set parameters to maintain standards. Each GEOPAK Drainage project contains a set of Preferences and they remain with the project. Computation options in the Preferences may be changed easily and the systems completely redesigned utilizing the new computation options. The Preferences are invoked via *Project* > *Preferences.*

The Drainage Preferences can also be invoked by selecting *Tools* > *Tool Boxes* > *Main*, then identifying the **Drainage Preferences** tool, (first column, top box).

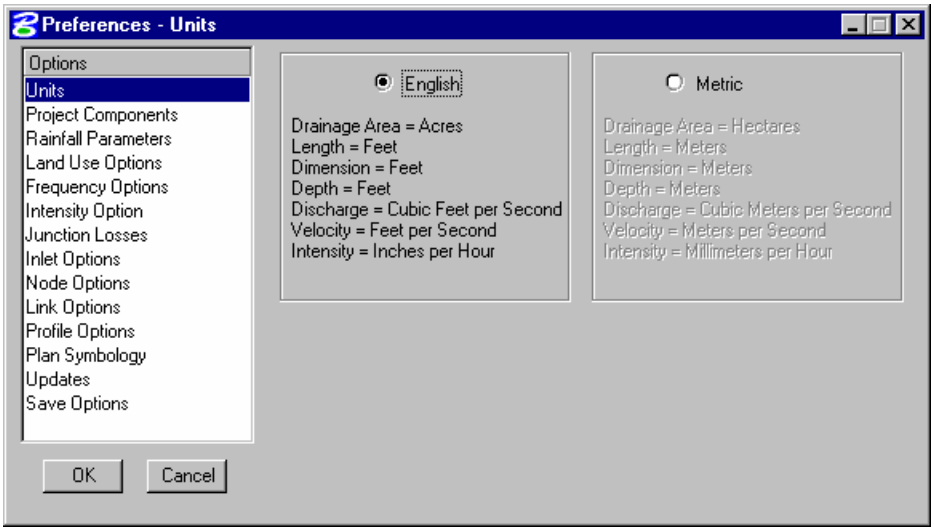
When invoked, the dialog depicted below is displayed.



The general Options are displayed in a list box in the left side of the dialog, while the right side displays parameters for the highlighted Option. As different Options are selected in the list box, the right side of the dialog will change dynamically to reflect the parameters for highlighted Option.

Units

The first Preference is Units, which establishes the input and output units for the current project. Two supported options are English and Metric, as depicted in the dialog below.



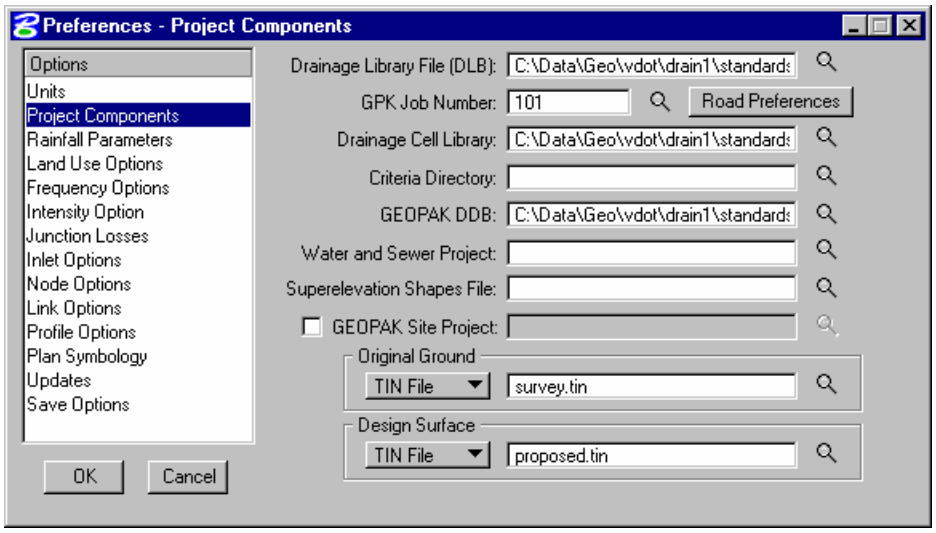
Within each group box is a listing of the units utilized for GEOPAK Drainage. These are display only and cannot be changed.

Note: Projects with mixed units are not supported within GEOPAK. There are no other unit specifications within the project. The data input is expected to be in the proper units (English and Metric). As fields are entered in the dialogs the units required will be displayed in the MicroStation Command Prompt.

Coordinate geometry databases (GPK files) must also be in the specified units when created within GEOPAK ROAD. For additional information on this topic, refer to the *GEOPAK Online help > Coordinate Geometry.*

Project Components

The Project Components, as depicted in the dialog below, defines the location of all files utilized by the project.



Fields in this dialog are described in the table below.

Drainage Library File (DLB)

Reference to the Drainage Library file containing the Rainfall Parameters, Land Use Options, Node Library items, Link Library items and spread sections. This file can be located within the working directory, in which case, no path need be specified. However, a more prudent option is placing the Drainage Library File on a central server or directory, so that all hydraulics designers can access the same database. This ensures standardization and minimal maintenance. The file name (and optionally, the path) may be supplied. Pressing the **Files** button will invoke the MicroStation File Manager, wherein the path and database file may be selected.

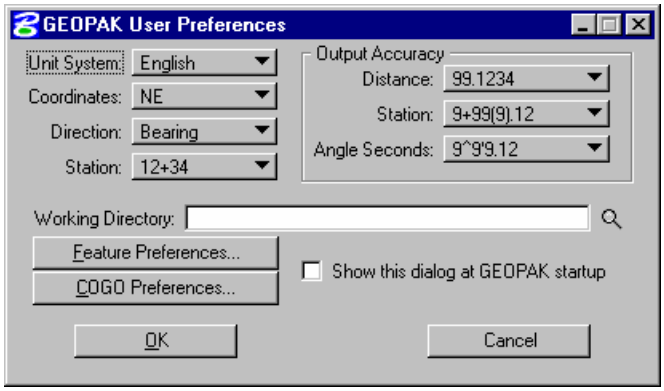
GPK Job Number

(optional)

GEOPAK coordinate geometry database which contains all alignment and profile information for the project. Creation and manipulation of this binary file must be accomplished from within GEOPAK ROAD. For further information, refer to the GEOPAK ROAD User Reference Manual, part II. The file name of the database is job*.gpk, where * is a one to three alphanumeric name unique to each project. It is this name that should be typed into the GPK Job Number field. In lieu of typing the name, pressing the **Select** button will invoke a File Manager wherein the GPK file may be selected.

Road Preferences

Invokes the GEOPAK User Preferences dialog as depicted below.



The working directory, wherein the GEOPAK coordinate geometry database is located, is specified within this dialog. The directory may also contain other project files, which is desirable; however, this is not required by GEOPAK Drainage.

Drainage Cell Library

Cell library wherein cells utilized for Nodes contained in the Drainage Library are located.

Site Modeler File

To leverage data from a GEOPAK Site project, activate the toggle, then select the directory and GEOPAK gsf file.

Criteria Directory

Define the directory wherein the criteria to draw nodes, links and other drainage data onto cross sections is stored.

Automatically Save Drainage Updates

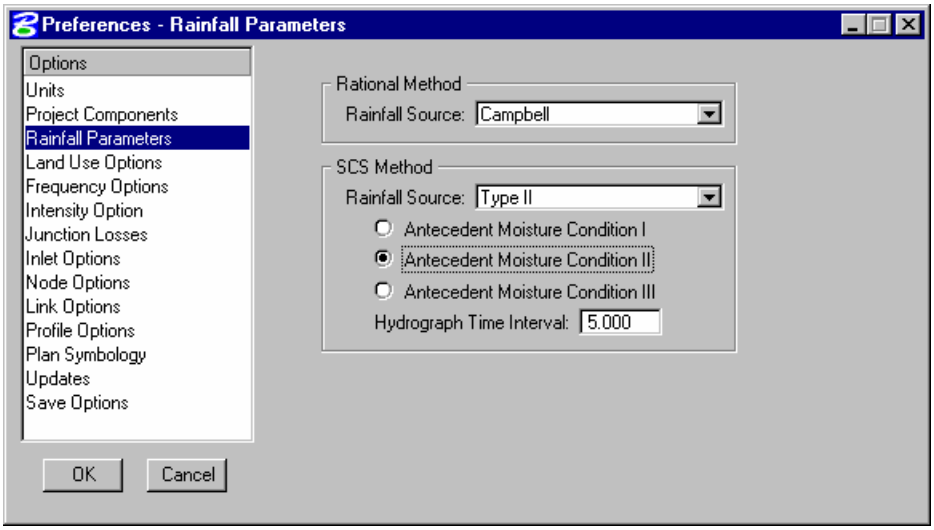
When activated, the GEOPAK drainage file (*.gdf) is automatically saved after each update.

Superelevation Shapes

Optionally, GEOPAK superelevation shapes created in GEOPAK Road can be leveraged into drainage. Specify the path and file name by manual entry or pressing the **Files** button to select from the dialog. In order to utilize and extract superelevation data from the shape file, the file must be referenced into the active design file.

Rainfall Parameters

The Rainfall Parameters dialog establishes the rainfall parameters for computing intensities and discharges for the current project. When selected from the Options list, the dialog depicted below is displayed.

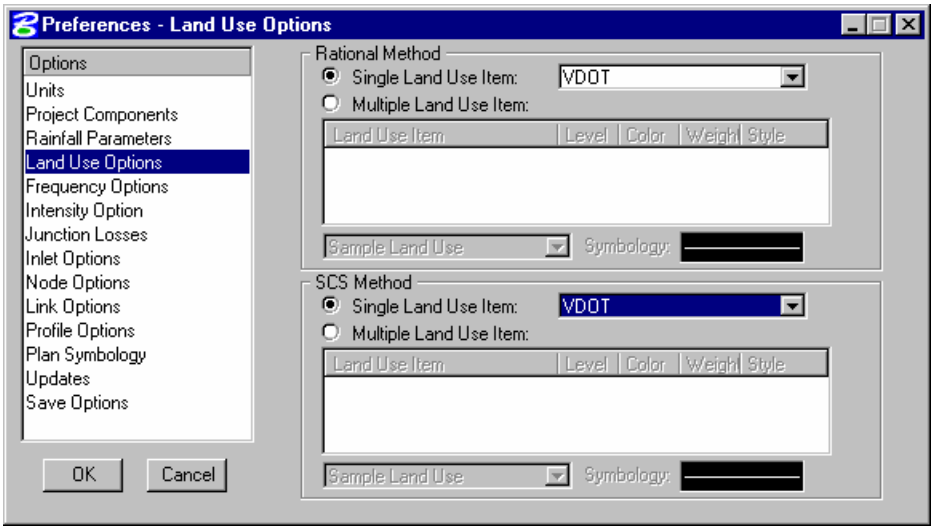


Fields in this dialog are described in the table below.

Rational Method Rainfall Source	The Rainfall Data Source for computing intensities and discharges, which is stored within the Drainage Library File, therefore it may only be selected here.
SCS Method: Rainfall Source	The Rainfall Data Source for computing discharges, which is stored within the Drainage Library File. Therefore, it may only be selected here. The Antecedent Moisture Condition I is the lowest runoff potential (dry soil), while Antecedent Moisture Condition II is the average condition, while Antecedent Moisture Condition III is the highest runoff potential (saturated soil).
Hydrograph Time Interval	Specified in terms of minutes.

Land Use Options

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



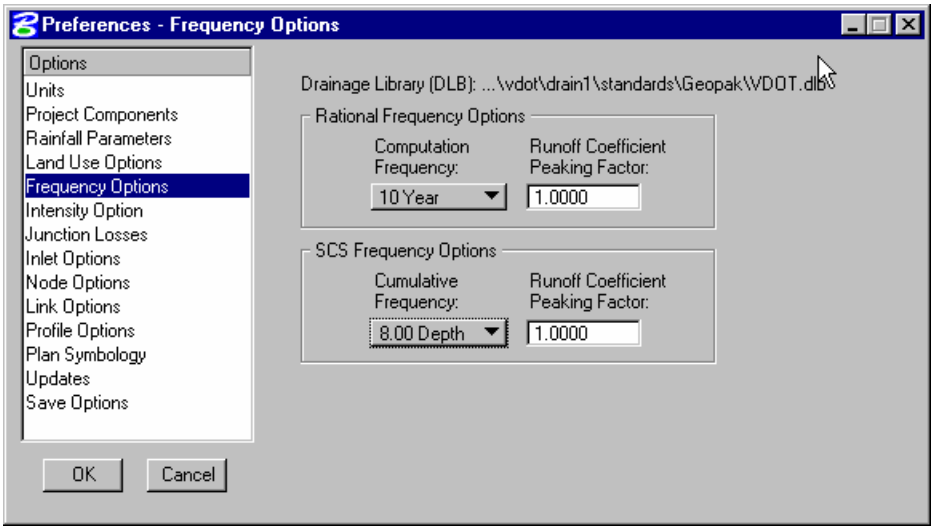
Fields in this dialog are described in the table below. Different Single and Multiple Land Use Items can be defined for the Rational and SCS Method.

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

Multiple Land Use Item — The land use sources from the Drainage Library containing the runoff coefficients and symbology for land use delineation. To select, utilize the down arrow in the list box and select the desired option. The item selected must be associated with a symbology that matches the symbology of each of the main land use shapes in the design file.

Frequency Options

The Frequency Options dialog establishes the computation frequency and runoff peaking factors for discharge computations for the current project. When selected from the Options list, the dialog depicted below is displayed.



Fields in this dialog are described in the table below.

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

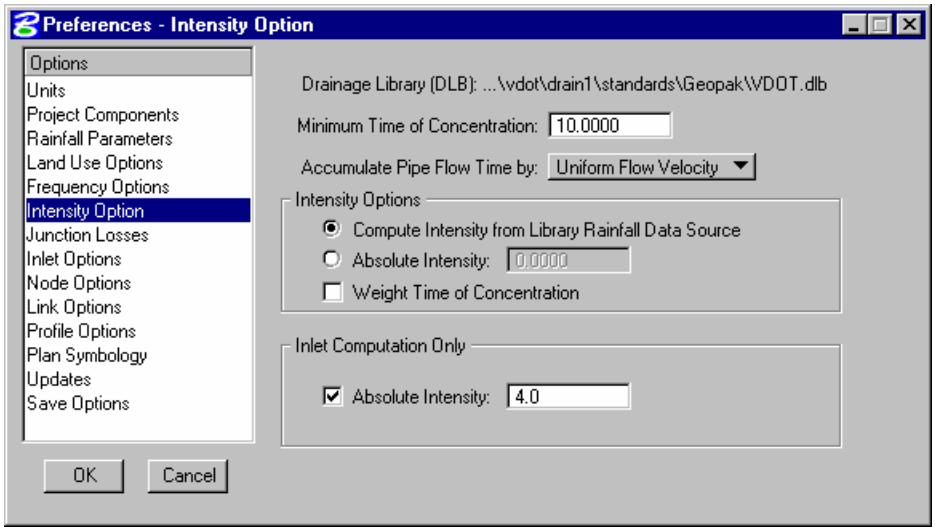
Computation Frequency — The displayed frequencies are selected from the available entries in the Rainfall Parameters chosen from the library. These fields cannot be modified except through the Drainage Library dialog.

Cumulative Frequency — The displayed depths are selected from the available entries in the Rainfall Parameters chosen from the library. These fields cannot be modified except through the Drainage Library dialog.

Peaking Factors — Runoff coefficient peaking factors to be applied in the runoff computations. These factors are used to adjust runoff coefficients for differing frequency rather than entering multiple set of coefficients for each frequency.

Intensity Option

The Intensity Options dialog establishes the intensity computation options for the current project. When selected from the Options list, the dialog depicted below is displayed.



Fields in this dialog are described in the table below.

Drainage Library File (DLB)

The current library file, as specified in the Project Component option, is displayed. It cannot be changed from this dialog, only in the Project Components.

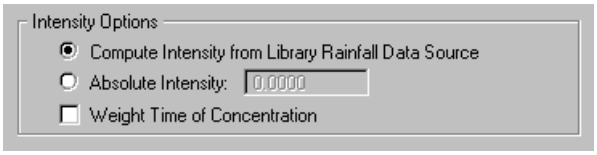
Minimum Time of Concentration

Minimum Time of Concentration to use in discharge computations. This is used throughout the computations as the minimum. In pipe hydraulic computations this value will be used until the actual time of concentration within the systems exceeds this value. Specified in terms of minutes.

Accumulate Pipe Flow Time by:
Uniform Flow Velocity
Full Flow Velocity
Interactive Velocity

The Accumulate Pipe Flow options adjust the travel time through the pipe. Options include Uniform Flow Velocity and Full Flow Velocity. If the iterative velocity is selected, GEOPAK designs the pipe for three iterations, so the travel time is based on the actual velocity for the three trials.

Intensity Options group box



Intensity options which allow for specification of an absolute (constant) intensity or computed intensities to use throughout the network discharge computation procedures.

Compute Intensity from Library Rainfall Data Source

Utilizes the values established and computed from the Rainfall Parameters (either default equation or table) from within the Drainage Library File.

Absolute Intensity

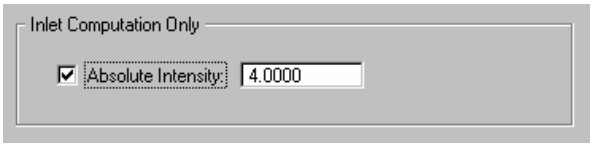
A constant intensity that overrides the Drainage Library File and utilizes the specified value for all discharge computations. It is specified in

terms of in/hr or mm/hr depending on the project units.

Weight Time of Concentration

When utilizing combining different types and times of concentration, activating this toggle weights the times of concentration such that the discharge will not decrease.

Inlet Computation Only
(group box)

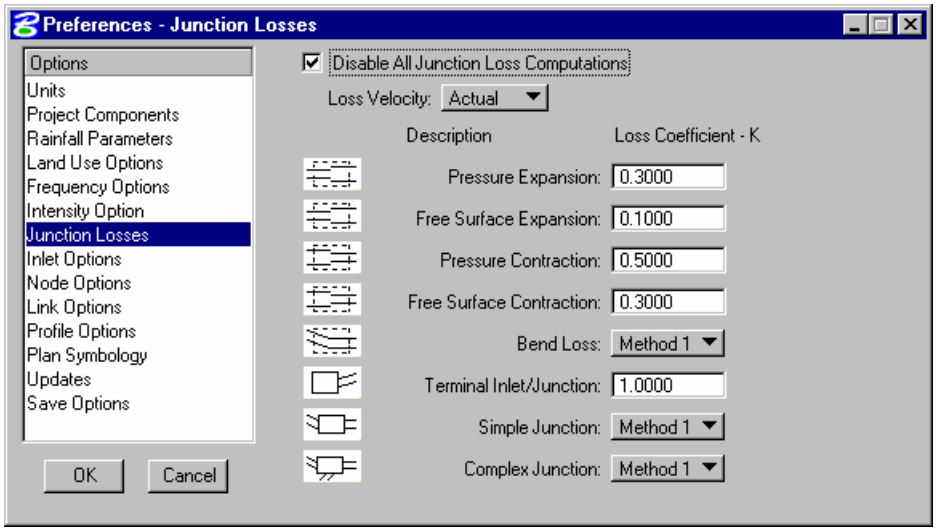


Absolute Intensity

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

Junction Losses

The Junction Losses dialog establishes the junction loss equations and coefficients utilized in Link hydraulic calculations for the current project. When selected from the Options list, the dialog depicted below is displayed.



Fields in this dialog are described in the table below. Unique Loss Coefficients (K) can be specified in the fields to the right of each Description.

Disable All Junction Loss Computations

If the toggle is active, no Junction Loss computations are calculated. Note if a single or small number of Junction Losses are not desired, an option is provided later in the design to accomplish this. The toggle here applies to the entire project.

Loss Velocity

Losses can be computed on Actual or Full Flow.

Pressure Expansion

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

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

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

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

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

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

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

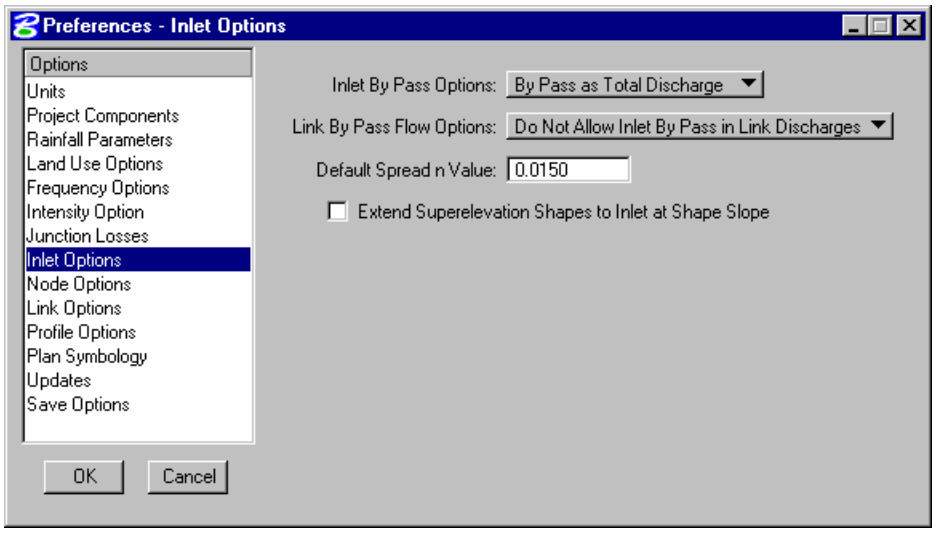
Simple Junction — Specify the methodology for computing junction losses at simple junction (one pipe into the junction and one pipe out).

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

For a detailed discussion of the specific hydraulic algorithms, see the technical reference section of the On-Line Help.

Inlet Options

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



Fields in this dialog are described in the table below.

Inlet By Pass Options

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

Link By Pass Options

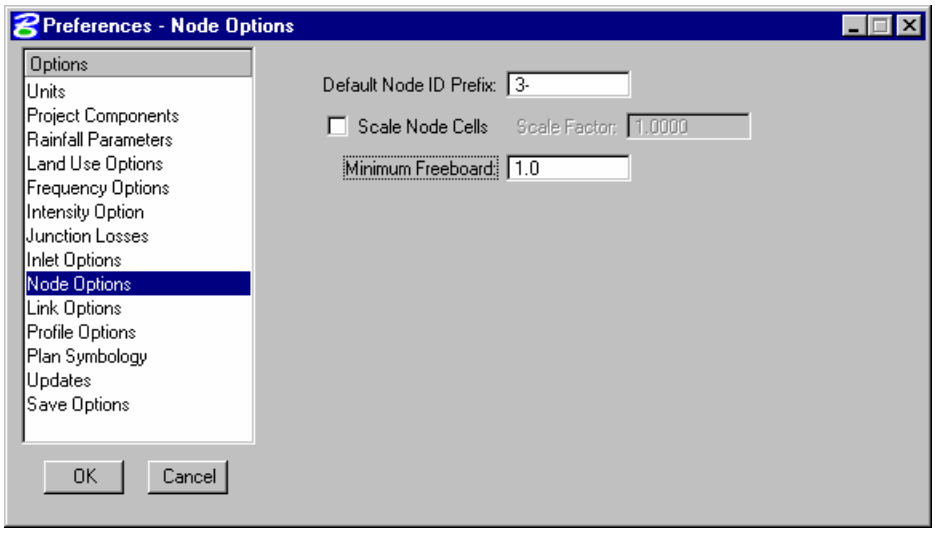
This option indicates the method by which Inlet By Pass flows are accounted in the Link discharge computations**. Do Not Allow Inlet By Pass in Link Discharges** will prevent the system from bypassing discharges from one inlet to the next when Link discharges are computed. The total discharge that reaches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

Default Spread N Value

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

Node Options

The Node Options establish the default naming prefix for Node entries and option for automatic link updates. When selected from the Options list, the dialog depicted below is displayed.

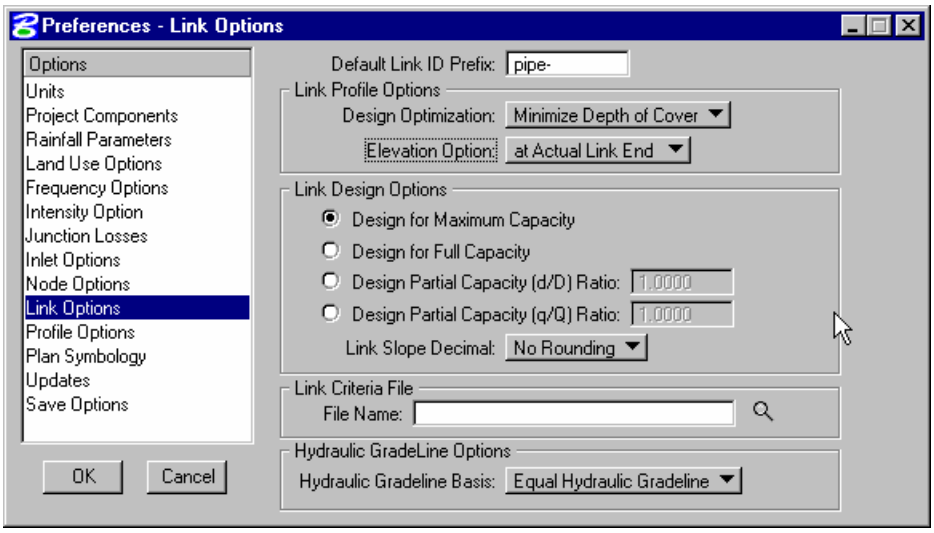


Fields in this dialog are described in the table below.

Default Node ID Prefix	This prefix will be added to the beginning of the each defined Node element automatically and then numerically sequenced by one. For example, the first Node stored would obtain default ID in this case of “CB A – 1” followed by “CB A – 2”, “CB A – 3”, etc. for subsequent Nodes.
Automatic Link Updates On Node Relocations	When activated, moving any Node will automatically update the connected Links.
Scale Node Cells / Scale Factor	When active, the node cells are scaled based on the specified scale factor. In this way, the node cell can utilize standard node dimensions, but the cells scaled in plan view.

Link Options

The Link Options dialog establishes the default Link design constraints. When selected from the Options list, the dialog depicted below is displayed.

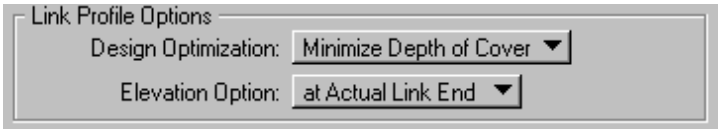


Fields in this dialog are described in the table below.

Default Link ID Prefix

This prefix, similar to the one found in the Node Options, is added to the beginning of the each defined Link element automatically and then numerically sequenced by one. For example, the first Link stored would obtain default ID in this case of “SS – 1” followed by “SS – 2”, “SS – 3”, etc. for subsequent Links.

Link Profile Options



A pipe profile envelope is developed which represents the minimum soffit elevation and maximum invert elevations for each network. This envelope is derived from an evaluation of all the constraints of the system; minimum and maximum depth, minimum and maximum pipe rise, minimum and maximum slope, and any elevations held through the system. The Minimize Pipe Size option will use the entire envelope to size pipes. If a smaller pipe will fit within the envelope at a steeper slope then it will be selected as the candidate of choice. If using the entire envelope does not result in smaller pipes then the top of the envelope will be used and the appropriate pipe fitting this slope configuration will be selected. The Minimize Depth of Cover will size pipes based on the top of the envelope, minimum soffit elevation and the pipe fitting this slope configuration will be selected. One Design Optimization method per project may be selected, however, the option may be changed anytime during the design process, the results quantified, and the effects of the Design Optimization reviewed.

Minimize Pipe Size

GEOPAK Drainage bases the design on minimizing pipe size, even though this may encourage deeper excavation.

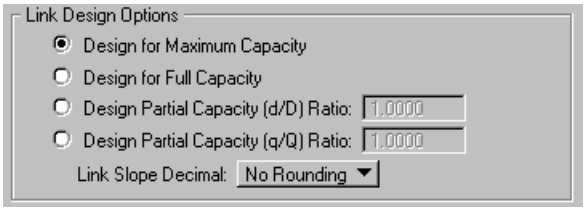
Minimize Depth of Cover

GEOPAK Drainage designs the system with minimal excavation, but utilizes larger pipe sizes.

Elevation Option

Two options are supported: at Hydraulic Center and at Actual Link End. This is reflected in the Link dialog in the condition option.

Link Design Options



Only one option may be selected for a project, however, this may be changed during the design process.

Design for Maximum Capacity

This option sizes pipe using maximum capacity. This typically occurs at some percentage less than full.

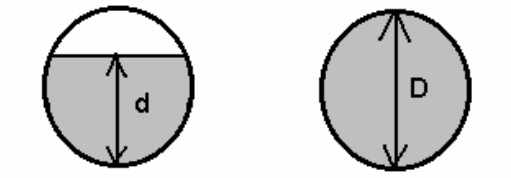
Design for Full Capacity

This option sizes pipe using full flow (100%) by depth. If a pipe is designed, the smallest size corresponding to a capacity greater than the discharge will be selected.

Design Partial Capacity

(d/D) Ratio

The ratio of the diameter of the pipe capacity versus the diameter of the total pipe to be used as the maximum capacity in pipe sizing as depicted in the graphic below. Note the value is a ratio (in decimal format) which must be equal to or smaller than one. If the value is equal to one, the pipe is designed for full capacity.



Design Partial Capacity

(q/Q) Ratio

The ratio of the pipe capacity versus discharge to be used in computing maximum capacity in pipe sizing algorithms. For example if the value is set to 0.5 then all pipes designed will have a capacity of twice the discharge and if set to 2.0 the pipes would have half the capacity of the discharge

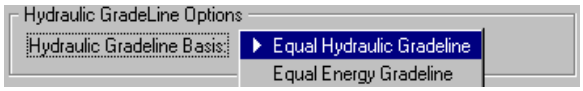
Link Slope Decimal

Options range from none to four. This limits the number of decimal places reported for most link slopes.

Link Criteria File

Name of criteria file which draws links onto cross sections.

Hydraulic Gradeline Options



Equal Hydraulic Gradeline

This option for computing hydraulic gradelines assumes that the starting downstream hydraulic gradeline of one Link is equal to the upstream hydraulic gradeline (plus junction losses if applicable) of the downstream Link.

Equal Energy Gradeline

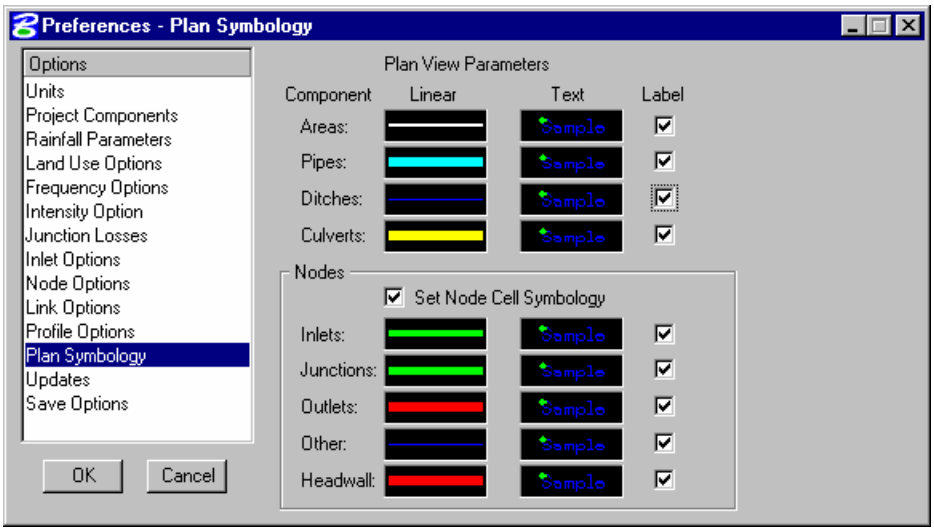
This option for computing hydraulic gradelines assumes that the starting downstream energy gradeline of one Link is equal to the upstream energy gradeline (plus junction losses if applicable) of the

downstream Link. The hydraulic gradeline will then proceed from the energy gradeline less the velocity head in the pipe. If the energy value in the downstream Link does not exist (too low) in the upstream pipe, the minimum specific energy and critical depth are used at the downstream end of the Link.

Design Symbology

The Design Symbology dialog establishes the element symbology for drawing drainage features during the design process.

When selected from the Options list, the dialogs depicted below are displayed, depending on the Type selection.



Fields in this dialog are described in the table below.

Type	Selection of the Type, either Plan or Profile, dynamically changes the dialog to reflect either Plan or Profile Components.
Component	The supported components are depicted in the graphic above. Unique parameters may be defined for each component or the same parameters may be utilized.
Level	Any valid MicroStation level may be specified wherein the component will be placed.
Color	The color picker may be utilized to select the desired color for each component.
Text Size	For the Plan type, labels will be placed at the specified text size. No text options are supported at this time for the Profile type all profile labels are placed at the current MicroStation Text Settings.
Label	For Plan type components, labels will be placed if the Label toggle is activated.
Style or Symbol	Any line style may be utilized, but custom line styles have to be utilized via the Design and Computation (DDB) tool. In addition, for Node elements a

default plan view cell may be entered under Symbol used in lieu of the cells for each Node in the Drainage Library when the DLB file can't be retrieved.

Weight Any valid MicroStation weight may be specified for each component.

LAB 4: Project Preferences

Project Preferences

The Project Preferences control the graphic and computational options of the drainage system. The Project Preferences may be changed at any time and the system can then be redesigned or analyzed utilizing the new preferences.

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB4_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

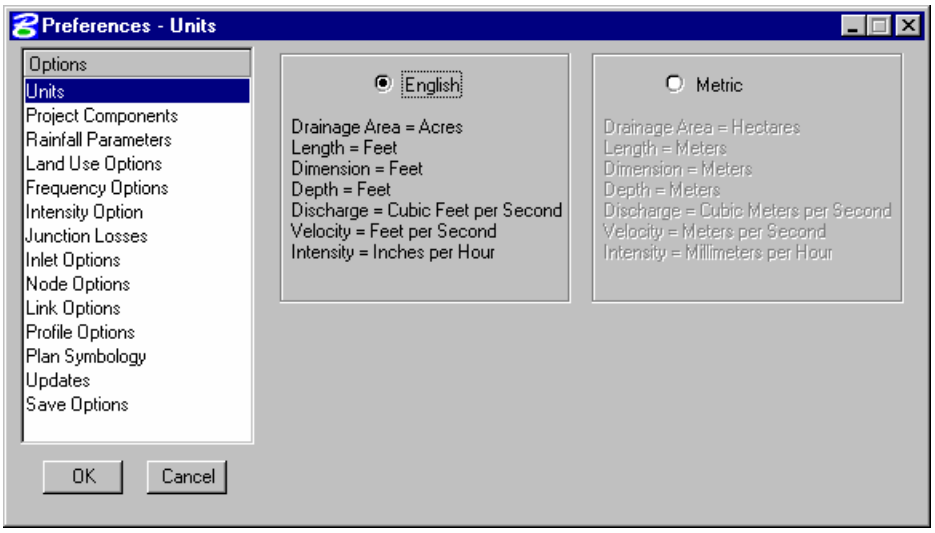
Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Drainage > Project > Open** from the MicroStation menu.

Step 5. Select **Drainage > Project > Preferences**.

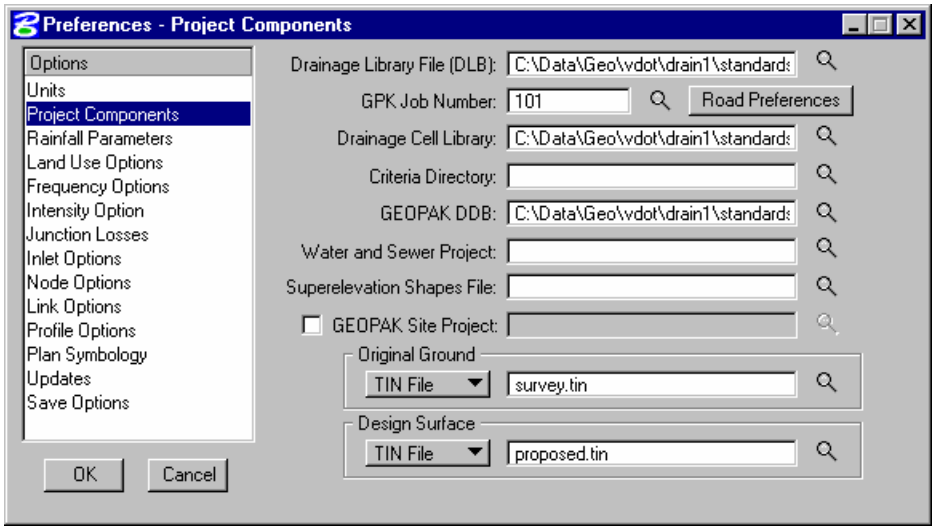
Note: Establish the Preferences by selecting each option in the column and defining the various values.

Step 6. **Units:**



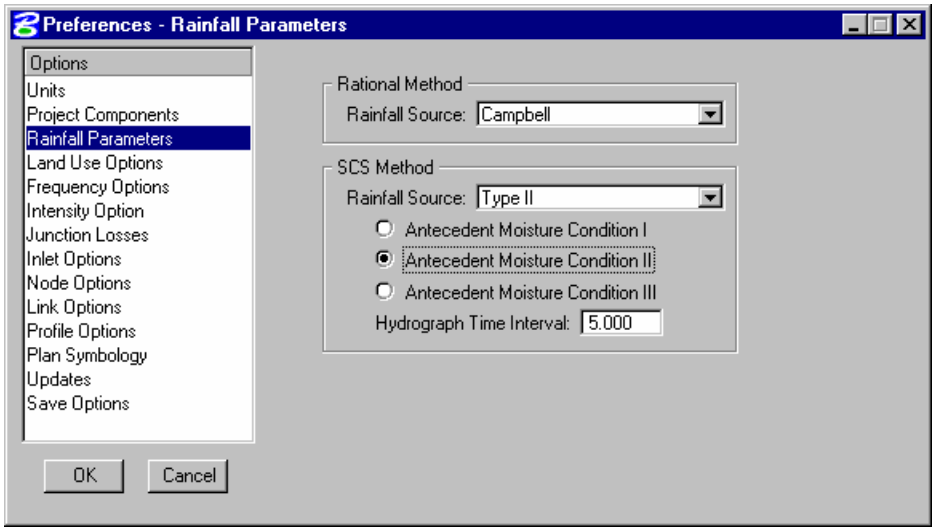
Select **English** units.

Step 7. Project Components:



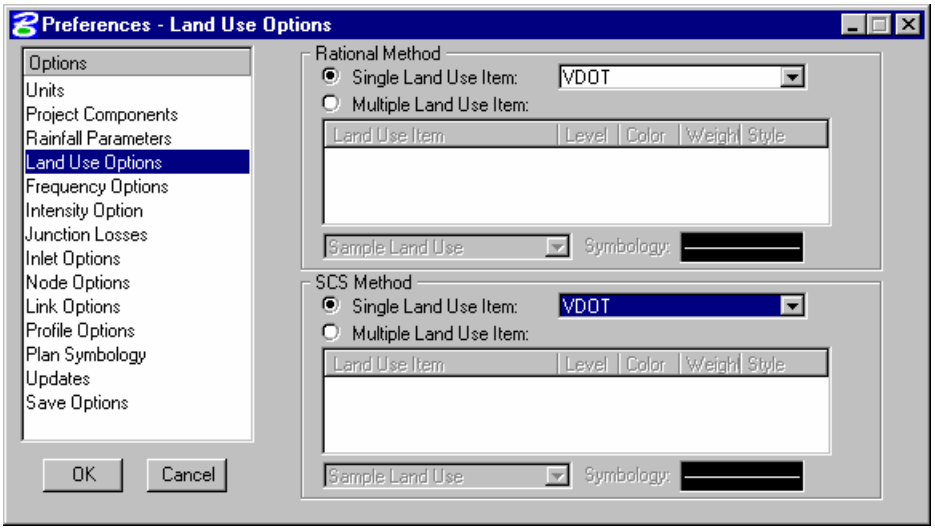
Drainage Library File (DLB)	`VDOT.dlb`
GPK Job Number	`Job101.gpk`
Drainage Cell Library	`drainage.cel`
Geopak DDB	`Vdotenglish.ddb`

Step 8. Rainfall Parameters:



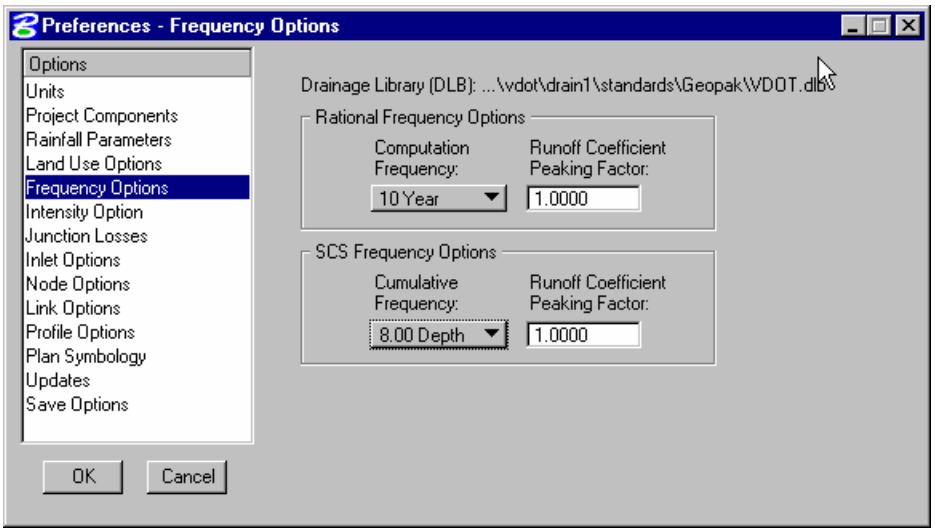
Rational Method	Campbell
SCS Method	Type II
Antecedent Moisture Condition II	ON
Hydrograph Time Interval	5.0

Step 9. Land Use Options:



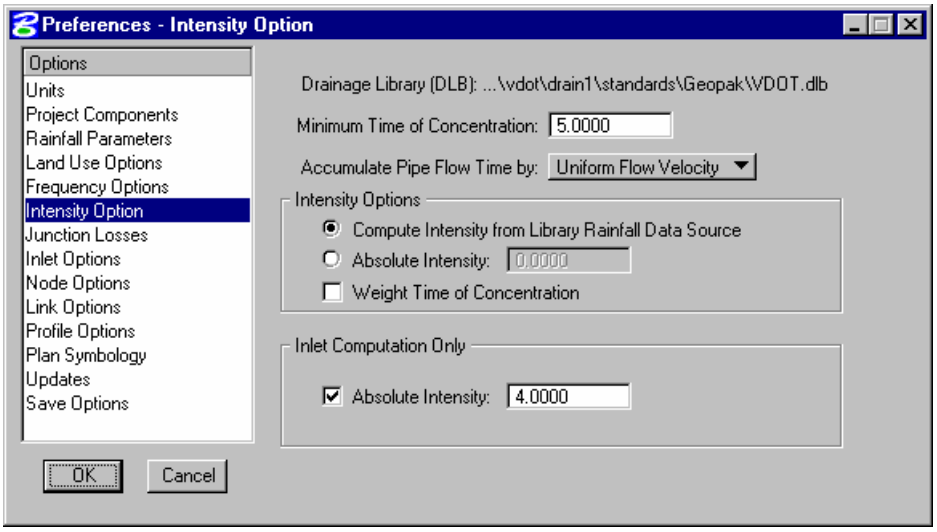
Set the Rational Method Single Land Use to **VDOT**.

Step 10. Frequency Options:



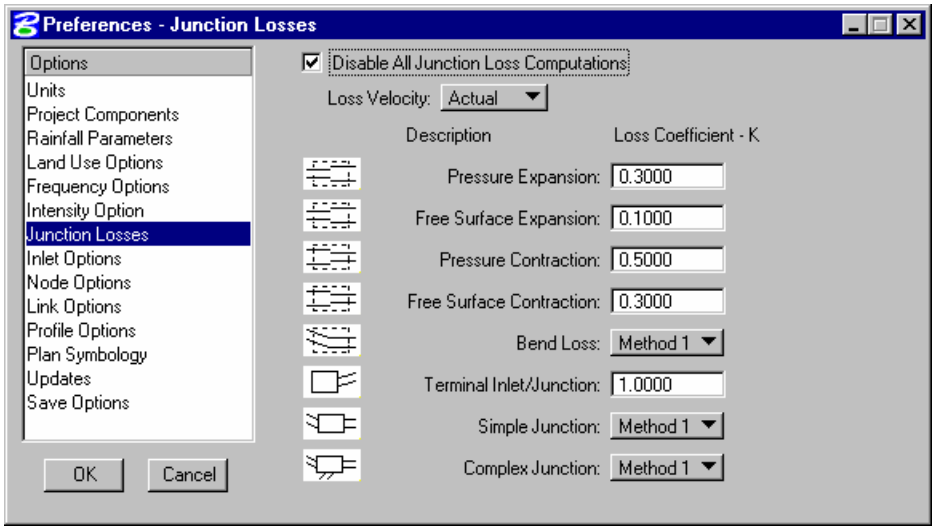
Set Rational Frequency Options to 10 Year with a 1.0 Runoff Coefficient Peaking Factor and the Cumulative Frequency to 8 inches for the SCS Method.

Step 11. Intensity Options:



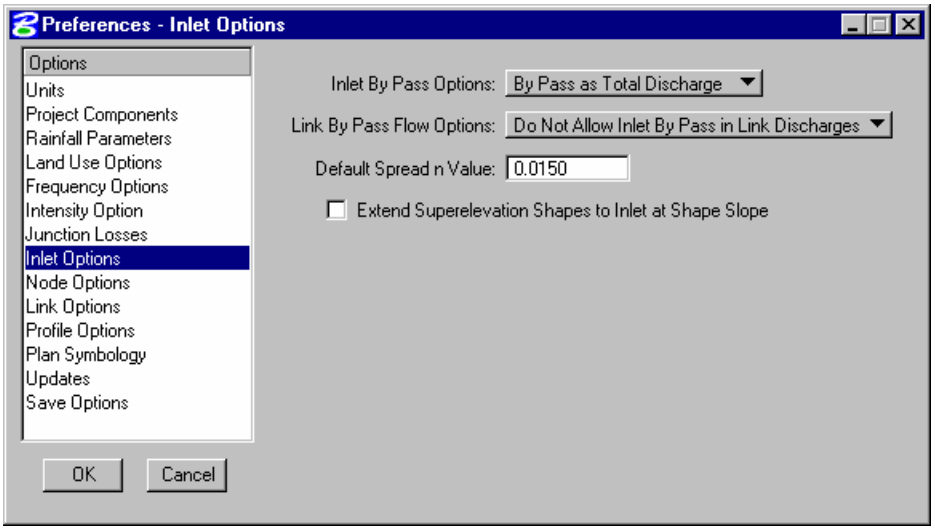
Minimum Time of Concentration	5 minutes
Accumulate Pipe Flow Time by	Uniform Flow Velocity
Intensity Options	Use Compute Intensity from Library Rainfall Data Source
Inlet Computation Only	Toggle *ON* , 4.0

Step 12. Junction Losses :



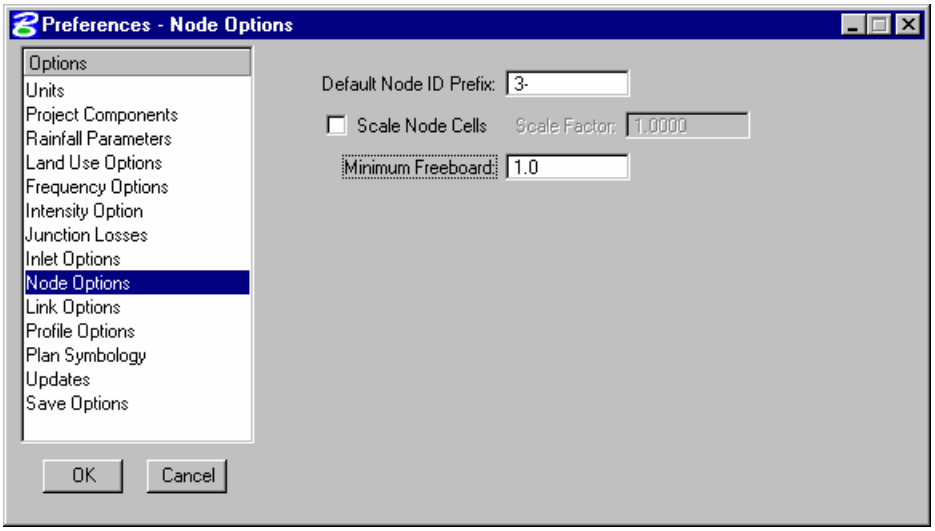
Toggle *ON* Disable All Junction Losses Computations.

Step 13. Inlet Options:



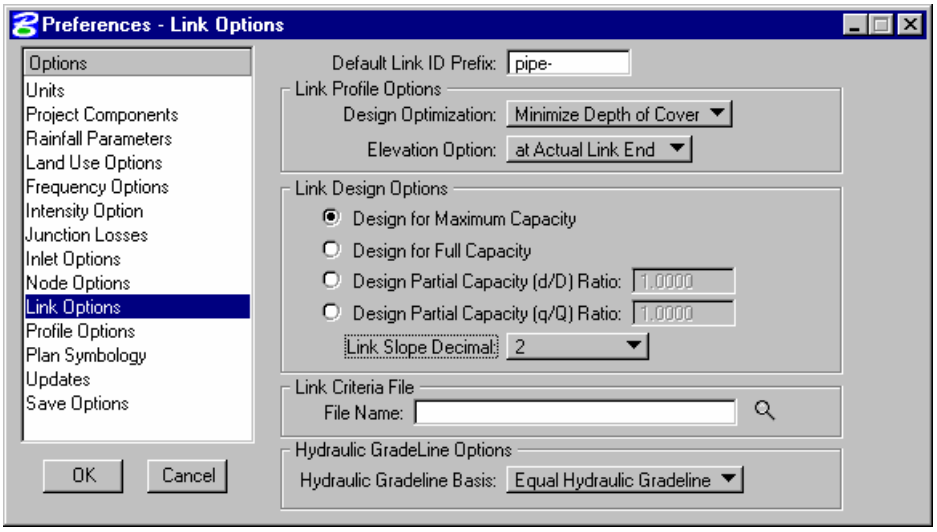
Inlet By Pass Options	By Pass as Total Discharge
Link By Pass Flow Options	Do Not Allow Inlet By Pass in Link Discharge
Default Spread n Value	0.015

Step 14. Node Options:



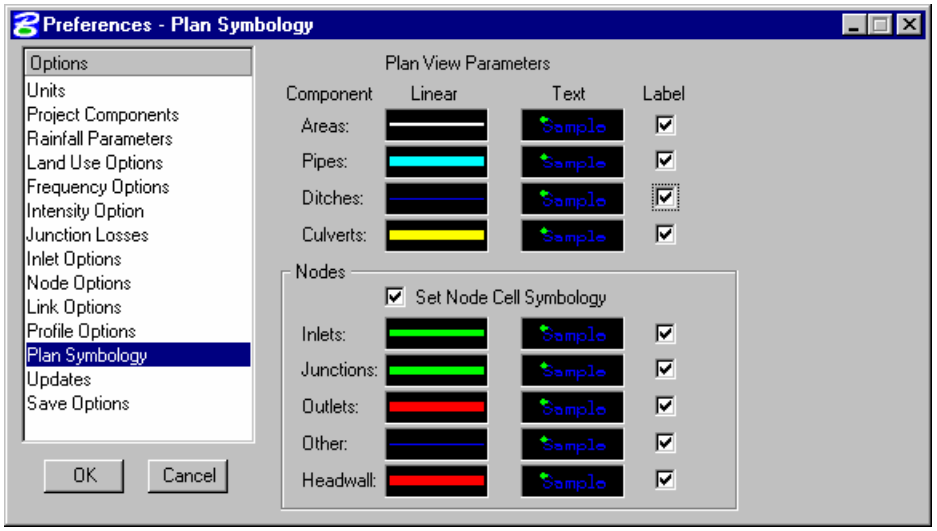
Default Node ID Prefix	3-
Scale Node Cells	Toggle *OFF*
Minimum Freeboard	1.00

Step 15. Link Options:



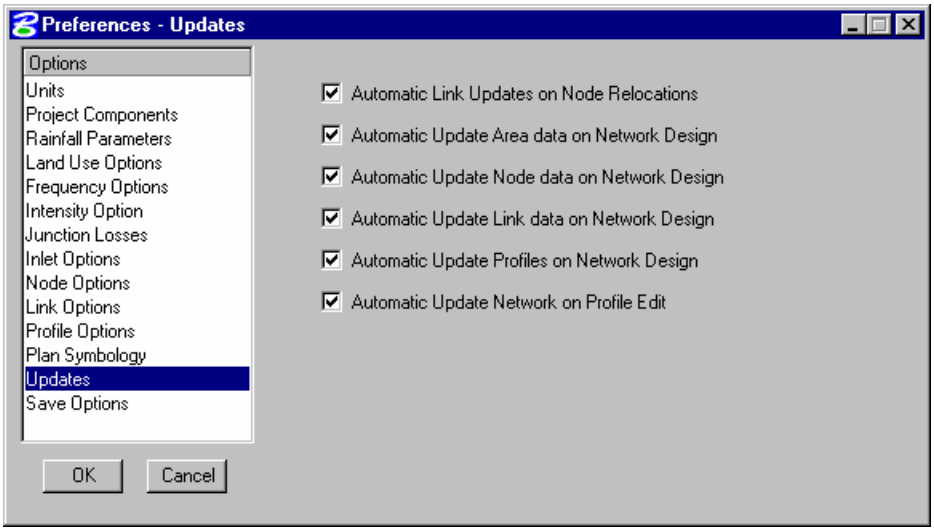
Default Link ID Prefix	pipe-
Design Optimization	Minimize Depth of Cover
Elevation Option	at Actual Link End
Link Design Options	Design for Maximum Capacity
Link Slope Decimal	2
Hydraulic Gradeline Options	Equal Hydraulic Gradeline

Step 16. Plan Symbology:

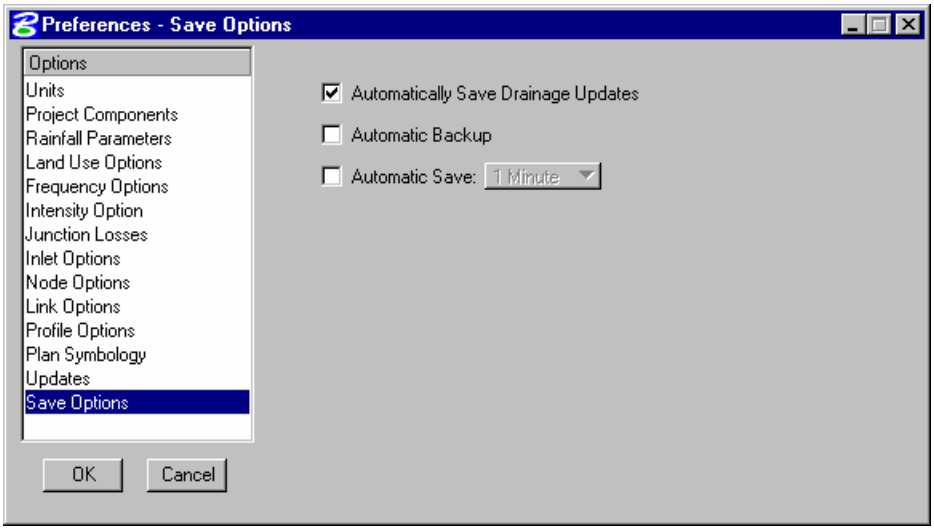


Utilize the plan symbology shown above for this project. Establish the symbology of future projects according to your standard conventions.

Step 17. **Updates:**



Step 18. **Save Options:**



Step 19. Press the **OK** button to accept the Project Preferences.

Note: The Preferences may be changed at any time during the course of the project.

Step 20. Exit MicroStation.

5 Drainage Areas and Nodes

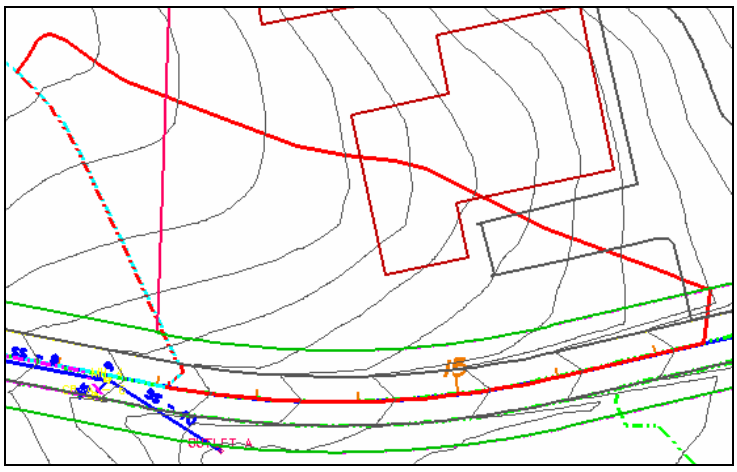
Objective:	Review Drainage design procedures.

Introduction

Drainage Areas in GEOPAK Drainage may be used to simply compute peak discharges or to attach computed discharge values to Nodes within a GEOPAK Drainage Project. The physical drainage area boundaries may be delineated using a digital terrain model, simply drawn with MicroStation, or just keyed in as a total area value.

The Drainage Library is an integral part of defining and computing discharges for Drainage Areas. With graphical definition of the area boundary, runoff coefficients may be automatically computed with the use of Land Use Items from a Drainage Library. Intensity values for the peak discharge computations are also computed with the use of Rainfall Items from the Drainage Library.

Graphical definition of Drainage Areas through the DTM Drainage Tools or MicroStation is accomplished through the creation of MicroStation closed shapes and subsequent selection of the shape during the Drainage Area creation process.



In order to assign Drainage Areas and the resulting peak discharges to Nodes of a storm drain network, the Drainage Area ID must match that of the Node.

Several options are supported to add, edit and delete Areas and are invoked via the Component pull down on the main menu bar.

Alternately, the Area tools are invoked by selecting *Drainage* > *Tool Boxes* > *Area*, then identifying the desired tool from the toolbox as depicted below.

Tools from left to right are:

- Add Drainage Area
- Modify Drainage Area
- Delete Drainage Area
- Update All Drainage Areas
- Drainage Area Report

Delineating Subareas

In order to delineate and add subareas for a particular Area, select the **Delineate Subarea** option within the Computations group box of the Drainage Area Definition dialog to invoke the Subarea Definition Tool. Subareas are defined as smaller components within a drainage area that exhibit different land uses or runoff coefficients.

- The Subarea Definition tool is utilized to define the subareas within the current Drainage Area. The subareas may be delineated from the land use boundaries automatically or entered into the table by the designer.

Introduction to Nodes

Nodes in GEOPAK Drainage are used to define structure points within a drainage Network. Nodes create all the Inlets, Junctions, and Outlets in the Network and provide for the connectivity of the Link system. Nodes are also used to indicate physical changes in Link sizes or slopes. Links cannot change size or slope, other than at Nodes. Eight categories of Nodes may be placed in a drainage project and are as follows:

- **Curb** – used for Curb opening Inlets with or without pipe confluences.
- **Grate** – used for Grate opening Inlets with or without pipe confluences.
- **Slotted Drain** – used for Slotted Drain Inlets with or without pipe confluences.
- **Junction** – used for any confluence of pipes, structure location (that is not an Inlet), pipe size or slope change, or flow addition that does not require an Inlet.
- **Other** - used for any miscellaneous points in a system such as a confluence of pipes, structure location (that is not an Inlet), pipe size or slope change, or flow addition that does not require an Inlet. They are also used between links that are ditches.
- **Outlet** – used for the Outlet, or outfall, of a storm drain Network.
- **Bottom** – used for the bottom portion of an inlet or node structure.
- **Headwall** – for culvert tool only.

The Drainage Library is an integral part of defining Node properties. The Node types, names, descriptions, physical dimensions, hydraulic properties, and plan view representation are all defined within the Drainage Library and merely referenced and selected out of the library.

Several options are supported to define, edit and delete Nodes and are invoked via the Component pull down on the main menu bar.

Alternately, the Node tools are invoked by selecting *Drainage* > *Tool Boxes* > *Node*, then identifying the desired tool from the toolbox as depicted below.

Tools from left to right are:

- Add Drainage Node
- Modify Drainage Node
- Delete Drainage Node
- Update All Nodes
- Update Nodes With Pay Items
- Drainage Node Report – Sump
- Drainage Node Report – On Grade

LAB 5: Drainage Design

Introduction

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB5_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

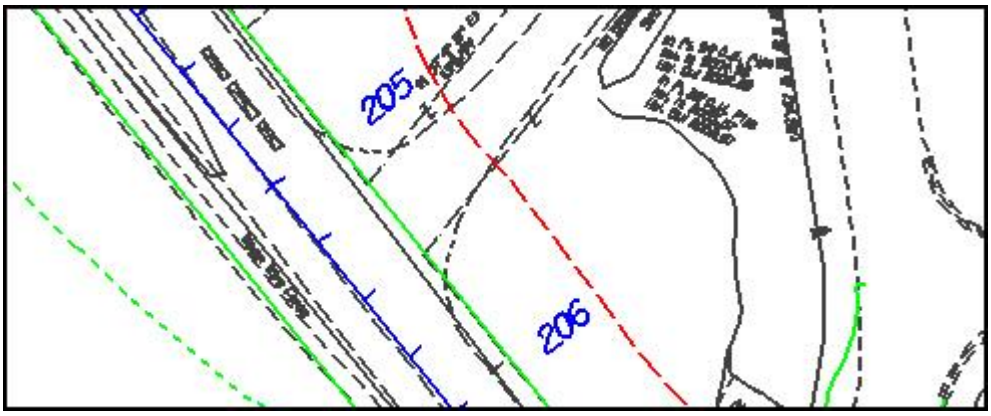
Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the GEOPAK Drainage pull down menu.

The proposed drainage design will accommodate drainage runoff from the portion of the roadway as shown below:

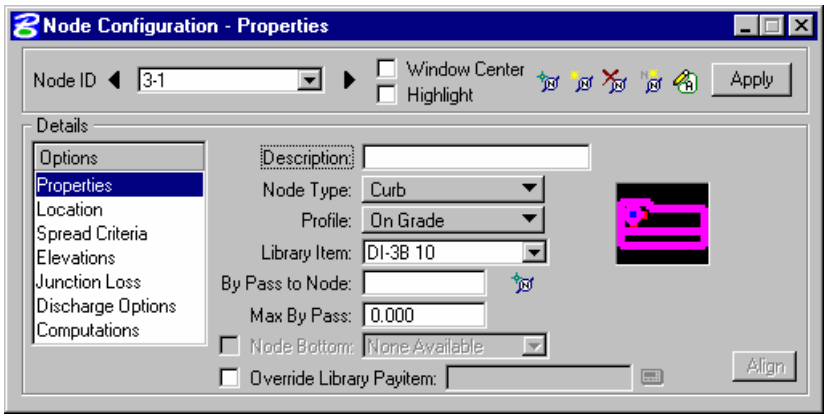
Design Inlet 3 – 1

Step 1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 205+86; offset –26.



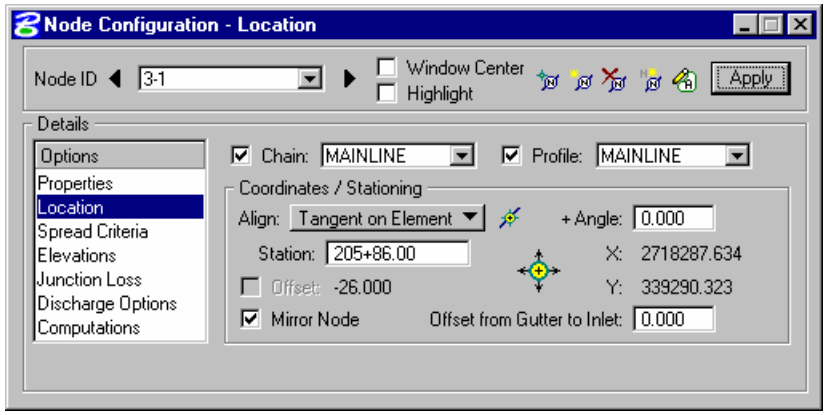
Step 2. Select from the Drainage menu: **Component > Node > Add.**

Step 3. Highlight **Properties** in the Options group box on the Node Configuration dialog. Select the Node to be used from the Drainage Library and assign its properties as depicted below:



Node ID	3-1
Node Type	Curb
Profile	On Grade
Library Item	DI-3B 10

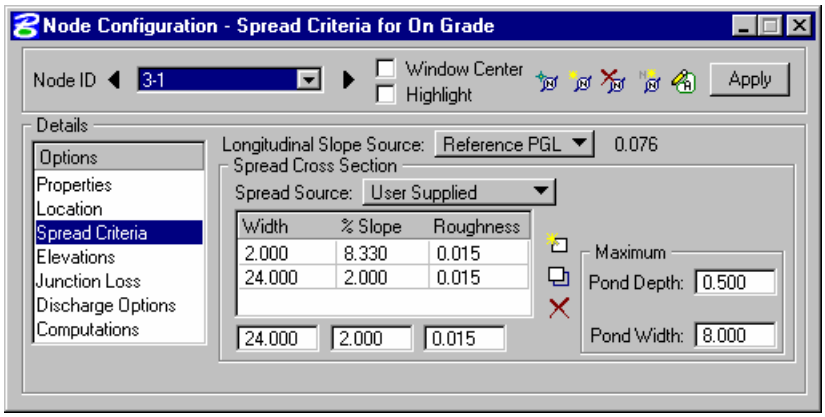
Step 4. Highlight **Location** to describe the inlet's location in the design plane:



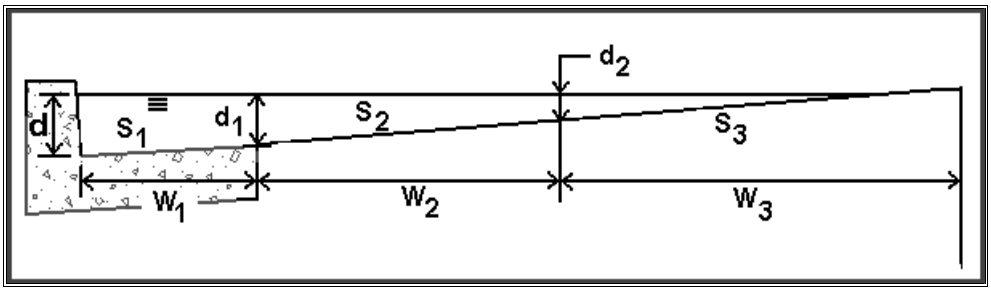
Reference Chain	**Select** Reference chain: MAINLINE
Reference PGL	**Select** Reference PGL: MAINLINE
Align	Tangent on Element
Station	205+86
Offset	-26.00 feet

Press the **Enter** key to automatically place the inlet at the specified location.

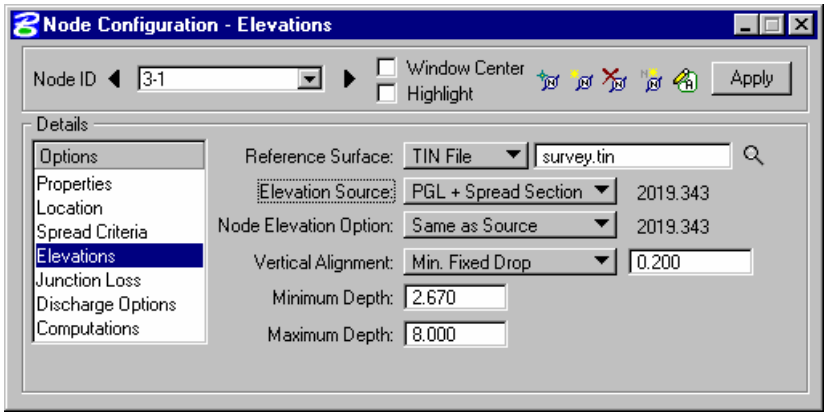
Step 5. Highlight **Spread Criteria** to describe the roadway cross sectional characteristics directly in front of the inlet. These values will be utilized to calculate inlet capacity and resulting by-pass flow:



Longitudinal Slope Source	Reference PGL
Spread Cross Section	User Supplied
Max Ponded Depth	0.50 feet
Max Ponded Width	8.00 feet

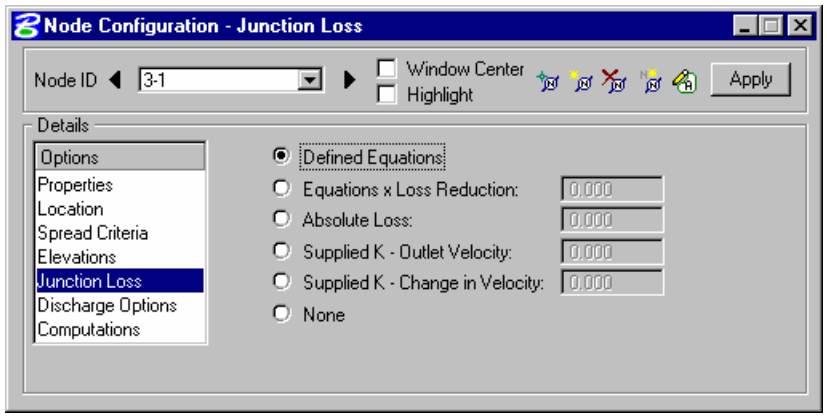


Step 6. Highlight **Elevations** to assign the inlets vertical elevation and vertical pipe alignment:

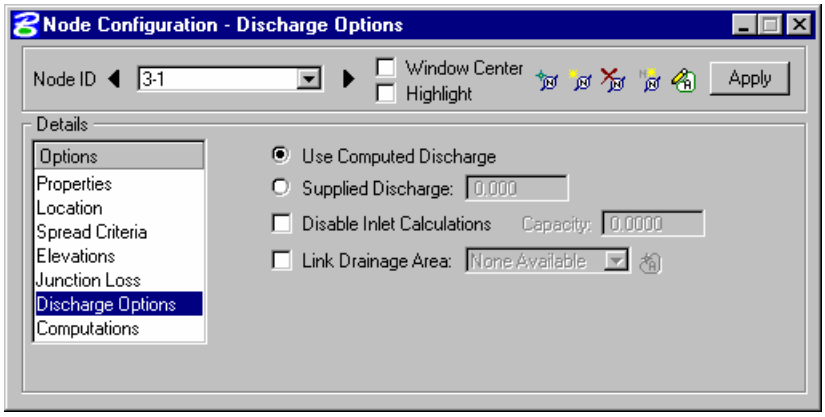


Reference Surface	Survey.tin
Elevation Source	PGL + Spread Section
Node Elevation Option	Same as Source
Vertical Alignment Preference	Min. Fixed Drop = 0.2
Minimum Depth	2.67 feet
Maximum Depth	8.00 feet

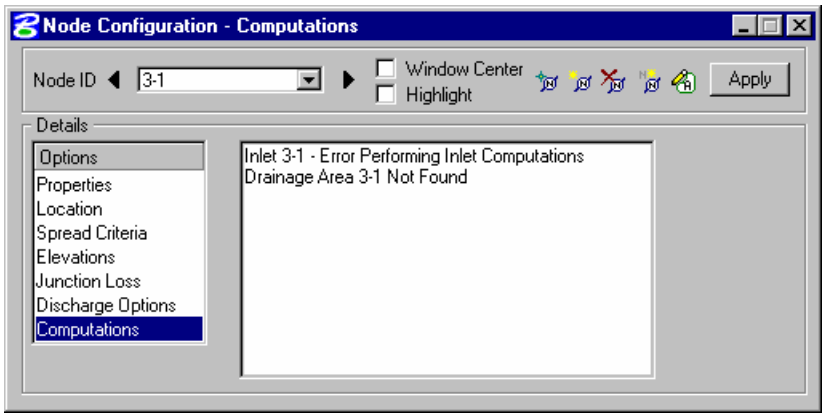
Step 7. Highlight **Junction Losses** to use **Defined Equations** (defaults to Preferences > Junction Loss Settings):



Step 8. Highlight **Discharge Options** to specify the source of the discharge contributing to this inlet. Toggle *ON* **Use Computed Discharge.**



Step 9. Highlight **Computations** to verify the inlet's hydraulic computations:



Note: The Drainage Area for this node won't be added until the next step. Therefore, the Computations for the node can't be completed until a discharge is known.

Step 10. Click **Apply** to add this Node to the project.

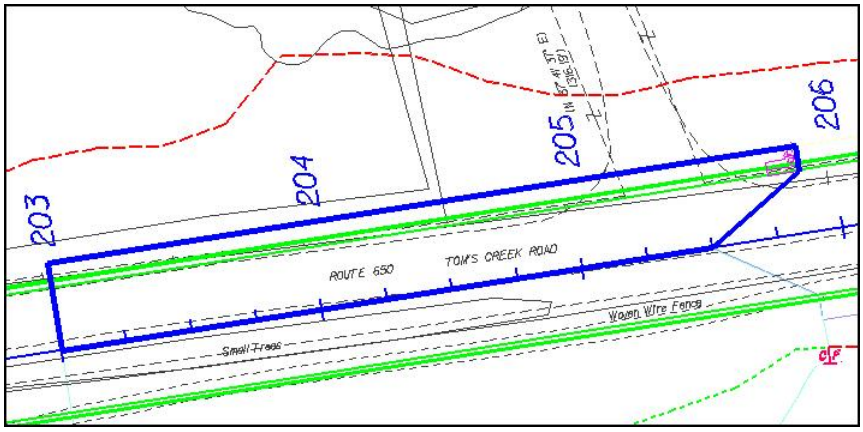
Step 11. Press the **Edit Area** icon to invoke the Drainage Area dialog.

Step 12. Select **Yes** to the Alert dialog box, since no area for the inlet is created yet.

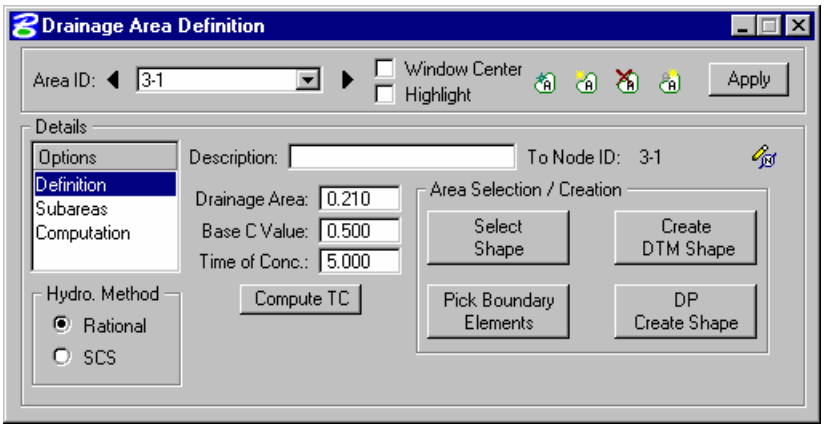
Design Drainage Area 3 – 1

The physical Drainage Area boundary may be delineated using a digital terrain model, simply drawn with MicroStation, or just keyed-in as a total area value (in units of acres or hectares). After the drainage area is delineated using MicroStation's drawing tools, runoff coefficients can be automatically computed with the use of *Land Use Items* from the Drainage Library.

The basin areas were predefined and were drawn in lv = Level 63. This closed shape will be selected to become the drainage area delineation.



Step 1. Turn on Level 63 in MicroStation and follow the instructions in the table below:



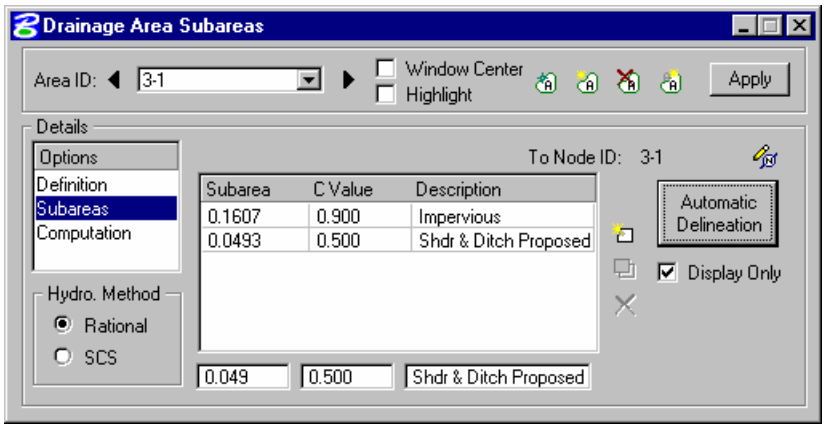
Drainage Area ID	3-1
Base C	0.5
Time of Concentration	5.00

Click **Select Shape** and select the MS shape for this area. Drainage area value is computed.

Step 2. Highlight **Subareas**.

Step 3. Toggle on **Display Only** and then Press the **Automatic Delineation** button.

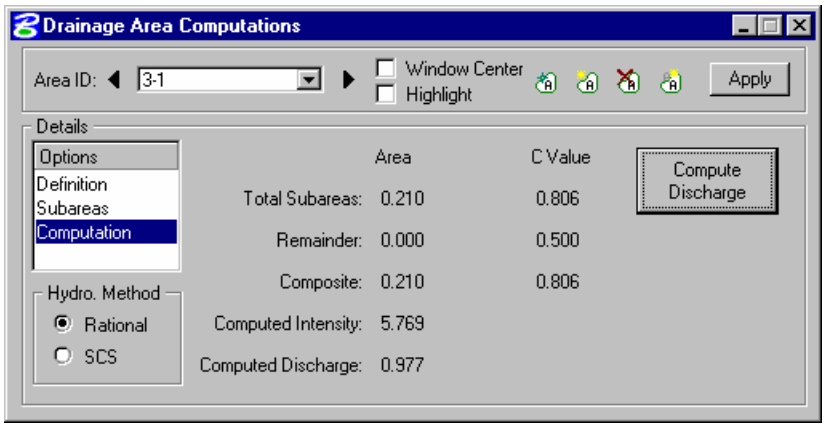
The file is scanned for closed shapes matching the Land Use symbology specified in the Drainage Library (Land Use Tab).



Step 4. Press the **Apply** button to apply the land uses (and their "C" values) to the Drainage Area.

Alternately, this table may be populated manually using the key-in fields and the **add**, **modify** and **delete** item buttons to the right of the table.

Step 5. Highlight **Computation** and press the **Compute Discharge** button to obtain the drainage area's computations.



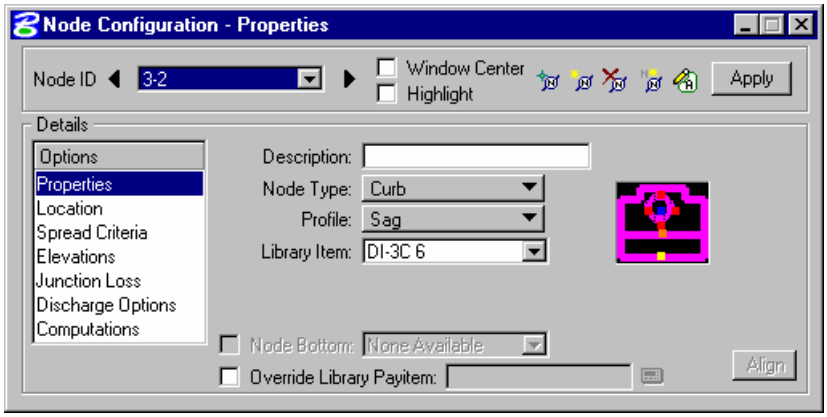
Verify the Computations; then press **Apply** to add the Area to the Project. Review the Computations in the Node Configuration dialog.

Design Inlet 3 – 2

Step 1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 206+36; offset -26.00.

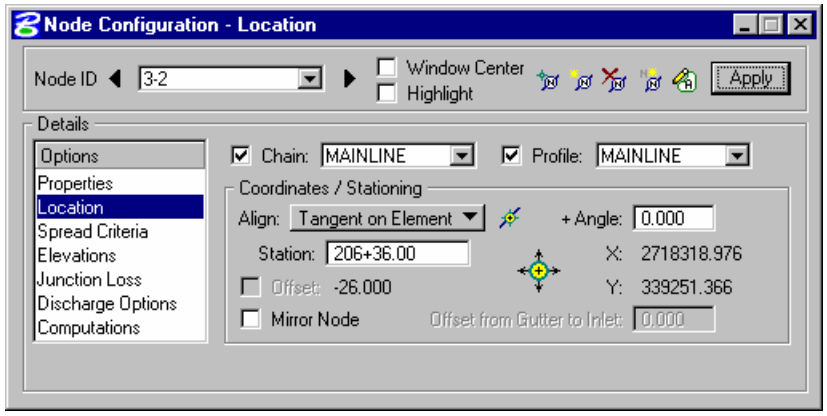
Step 2. Select from the Drainage menu: **Component > Node > Add.**

Step 3. Highlight **Properties.** Verify the Node Properties are defaulted from the previous Node.



Node ID	3-2
Node Type	Curb
Profile	Sag
Library Item	DI-3C 6

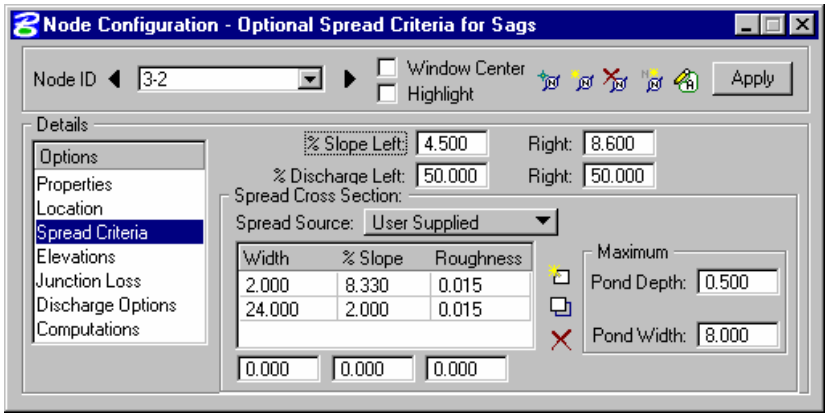
Step 4. Highlight **Location.** All Reference information is defaulted from the previous Node (3-1) such that only the new Location is required.



Station	206+36
Offset	-26

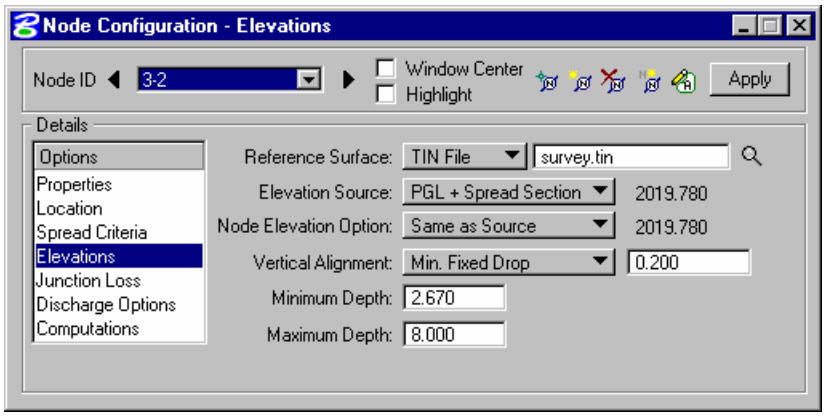
Press **Enter** to locate the inlet in the dgn file.

Step 5. Highlight **Spread Criteria.** Set required information to describe the flow to the inlet.



Longitudinal % Slope Left	4.50%
Longitudinal % Slope Right	8.60%
% Discharge Left	50 %
% Discharge Right	50%

Step 6. Highlight **Elevations.** Set the required information to describe the inlet's elevation:



Note: Elevation Source, Node Elevation Option, vertical Alignment, Minimum and Maximum Depth should be filled out based on the previous node 3-1.

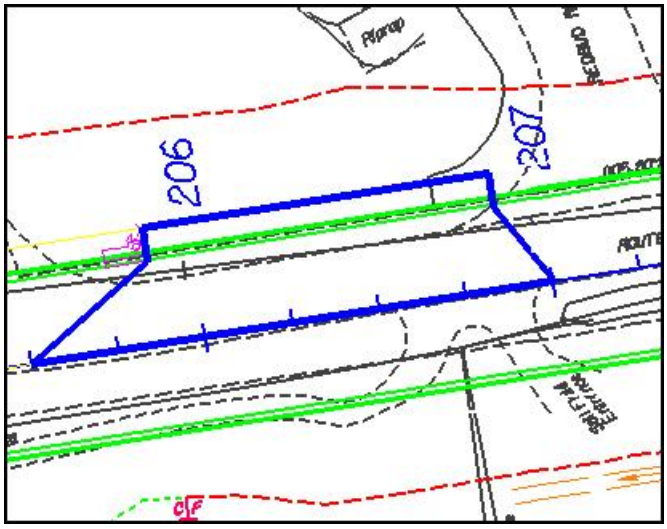
Step 7. Press the **Apply** button to include this node in the Drainage Project.

Step 8. Press the **Edit Area** icon to associate this inlet with its Drainage Area.

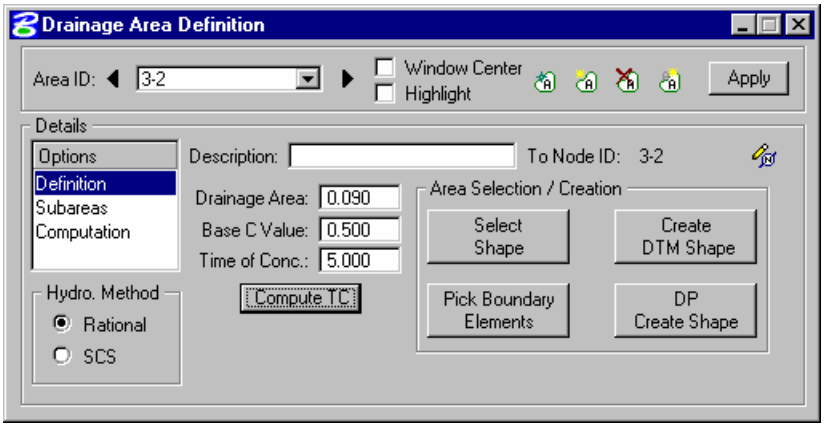
Step 9. Select **Yes** to the Alert dialog box, since no area for the inlet is created yet.

Create Drainage Area 3 – 2

This closed shape in lv = Level 63 will be selected to become the drainage area delineation.



Step 1. Follow the instructions in the table below.



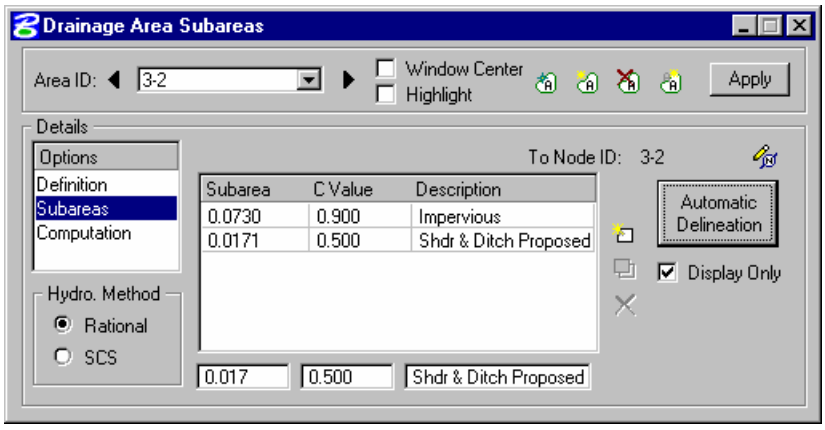
Drainage Area ID	3-2
Base C	0.5
Time of Concentration	5.00

Click **Select Shape** and select the MS shape for this area. Drainage area value is computed.

Step 2. Highlight **Subareas**.

Step 3. Press the **Automatic Delineation** button.

The file is scanned for closed shapes matching the Land Use symbology specified in the Drainage Library (Land Use Tab).

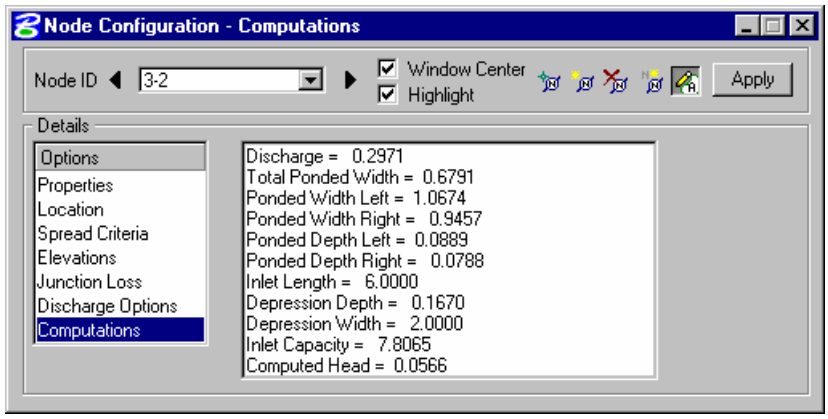


Step 4. Press the **Apply** button to apply the land uses (and their "C" values) to the Drainage Area.

Step 5. Highlight **Computation** and press the **Compute Discharge** button to obtain the drainage area's computations.

Step 6. Verify the Computations; then press **Apply** to add the Area to the Project.

Step 7. Review the computations in the Node Configuration dialog.

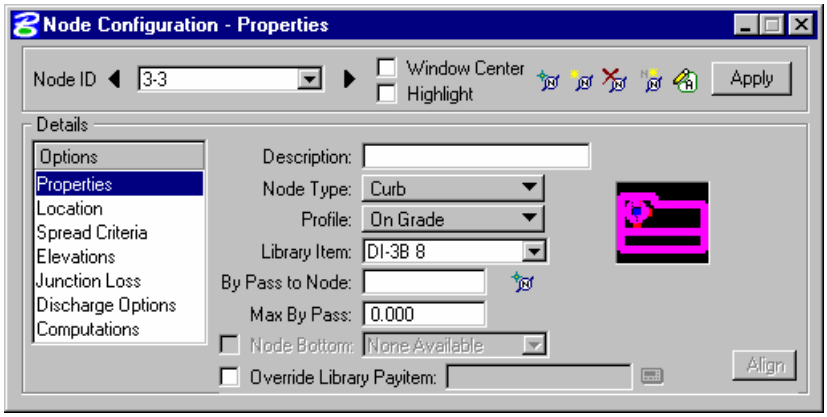


Design Inlet 3 – 3

Step 1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 206+86 offset -26.00.

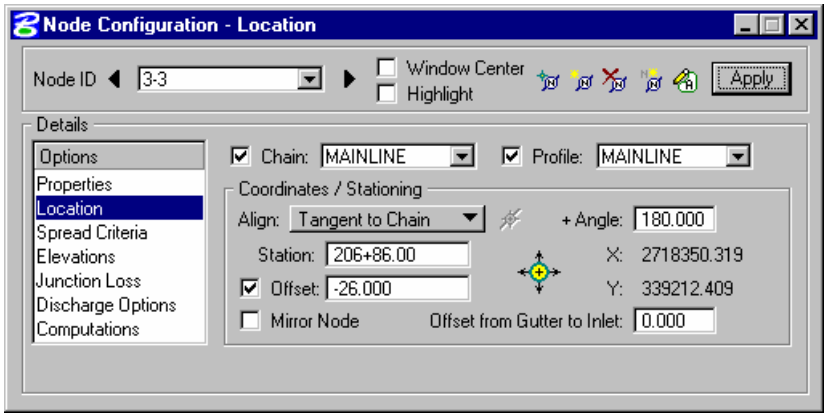
Step 2. Select from the Drainage menu: **Component > Node > Add.**

Step 3. Highlight **Properties** to select the Node to be used from the Drainage Library and assign its properties as depicted below:



Node ID	3-3
Node Type	Curb
Profile	On Grade
Library Item	DI-3B 8

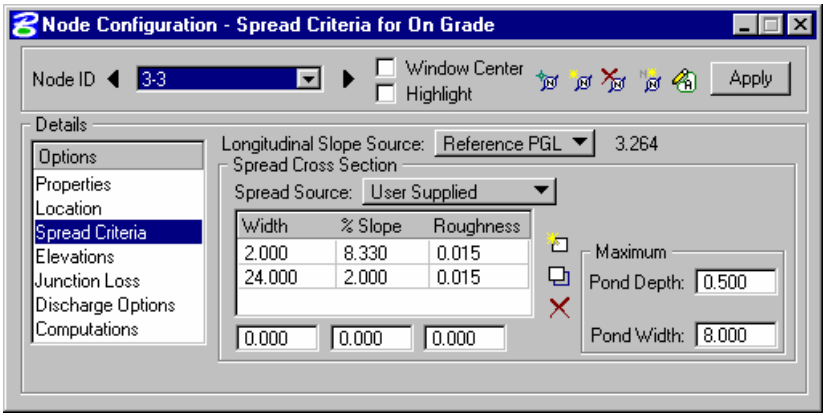
Step 4. Highlight **Location.** Describe the inlet's location in the design plane.



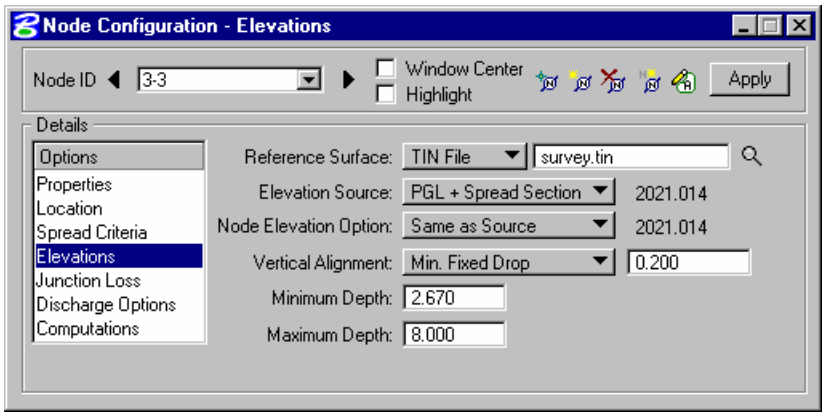
Station	206+86
Offset	-26

Press **Enter** to automatically place the inlet at the specified location.

Step 5. Highlight **Spread Criteria.** Set the required information to describe the flow to the inlet.



Step 6. Highlight **Elevations.** Set the required information to describe the inlet's elevation:



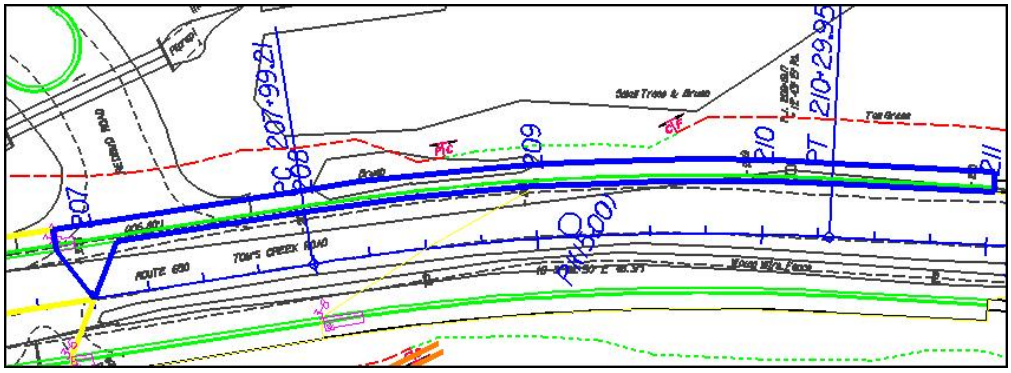
Step 7. Press the **Apply** button to include this node in the Drainage Project.

Step 8. Press the **Edit Area** icon to associate this inlet with its Drainage Area.

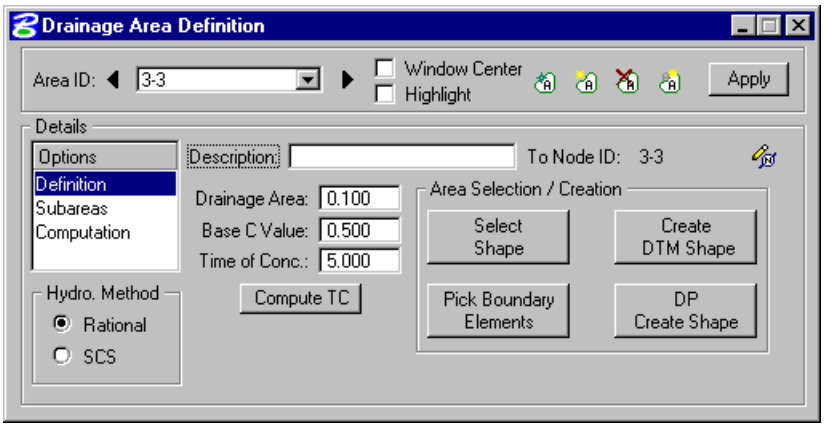
Step 9. Select **Yes** to the Alert dialog box, since no area for the inlet is created yet.

Create Drainage Area 3 – 3

This closed shape in lv = Level 63 will be selected to become the drainage area delineation.



Step 1. Follow the instructions below.



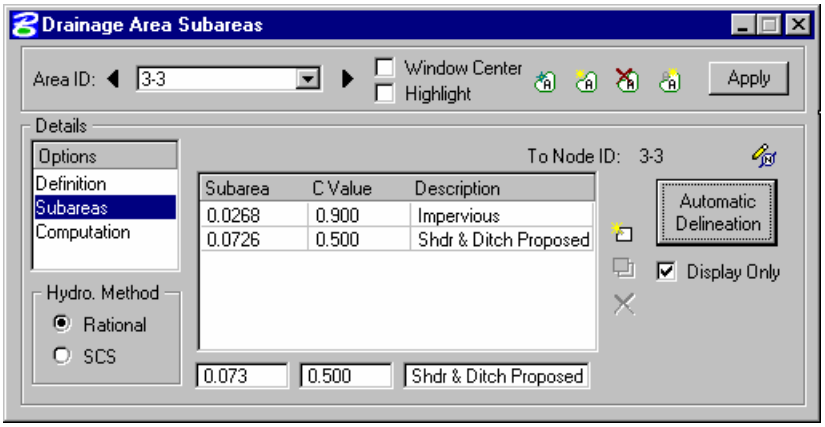
Drainage Area ID	3-3
Base C	0.5
Time of Concentration	5.00

Click **Select Shape** and select the MS shape for this area. Drainage area value is computed.

Step 2. Highlight **Subareas**.

Step 3. Press the **Automatic Delineation** button.

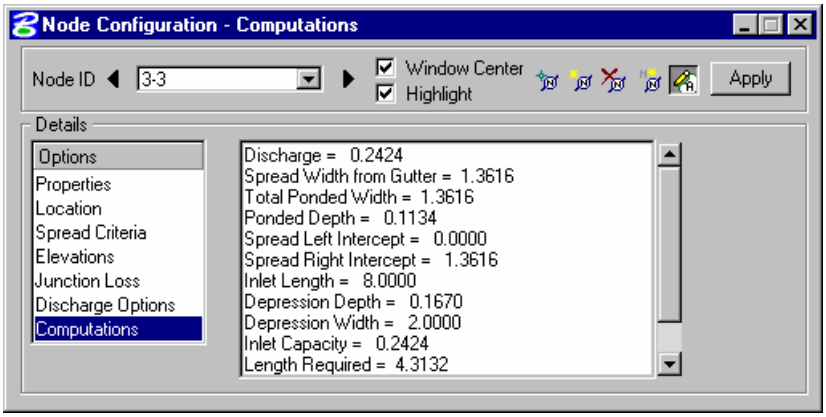
The file is scanned for closed shapes matching the Land Use symbology specified in the Drainage Library (Land Use Tab).



Step 4. Highlight **Computation** to obtain the drainage area's computations.

Step 5. Verify the Computations; then Press **Apply** to add the Area to the Project

Step 6. Review the computations in the Node Configuration dialog.

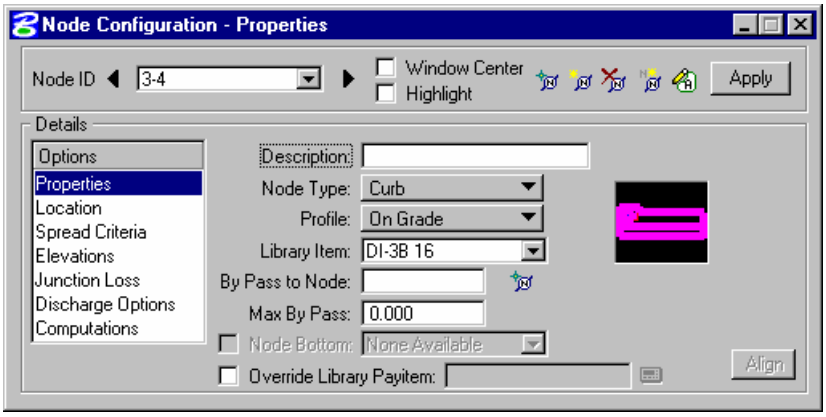


Design Inlet 3 – 4

Step 1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 205+86 offset 26.00.

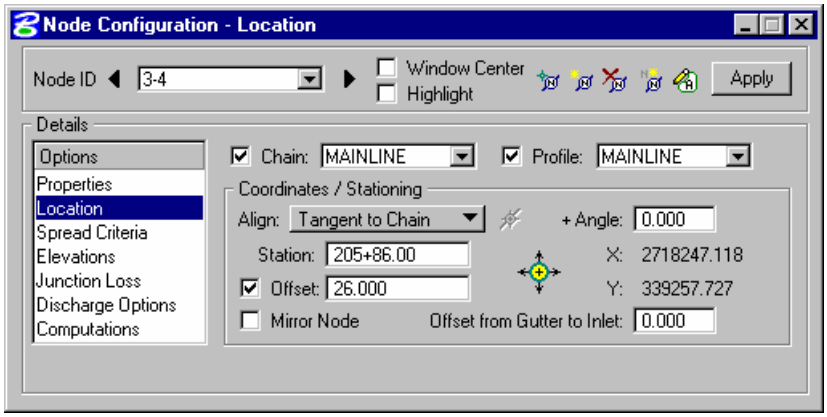
Step 2. Select from the Drainage menu: **Component > Node > Add.**

Step 3. Highlight **Properties.** Select the Node to be used from the Drainage Library and assign its properties as depicted below:



Node ID	3-4
Node Type	Curb
Profile	On Grade
Library Item	DI-3B 16

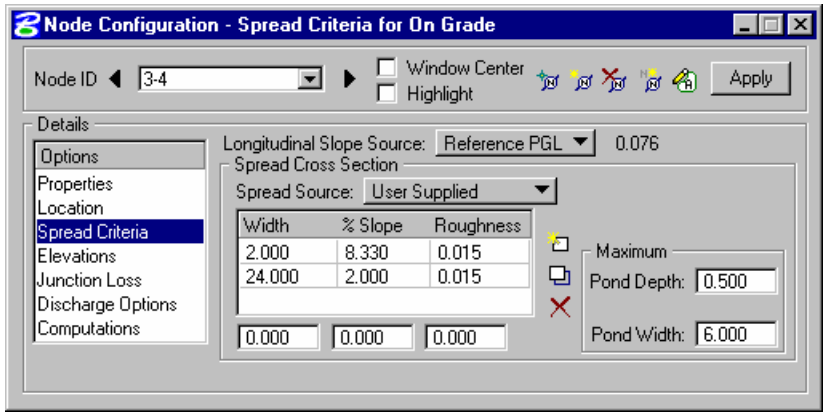
Step 4. Highlight **Location.** Describe the inlet's location in the design plane.



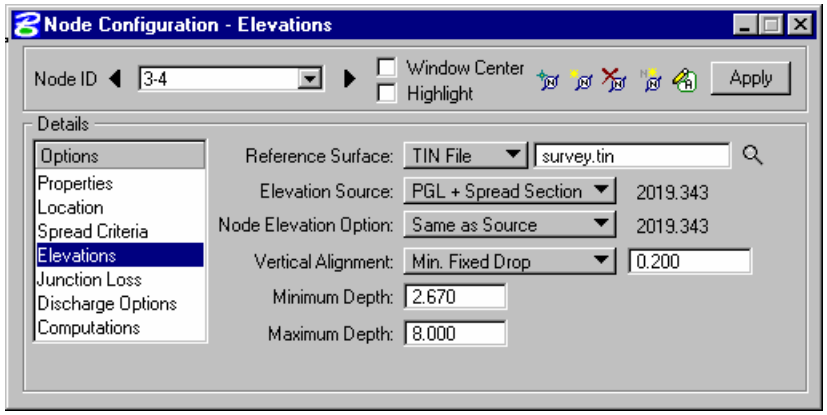
Station	205+86
Offset	26
Angle	180

Press **Enter** to automatically place the inlet at the specified location.

Step 5. Highlight **Spread Criteria.** Set required information to describe the flow to the inlet.



Step 6. Highlight **Elevations.** Set the required information to describe the inlet's elevation:



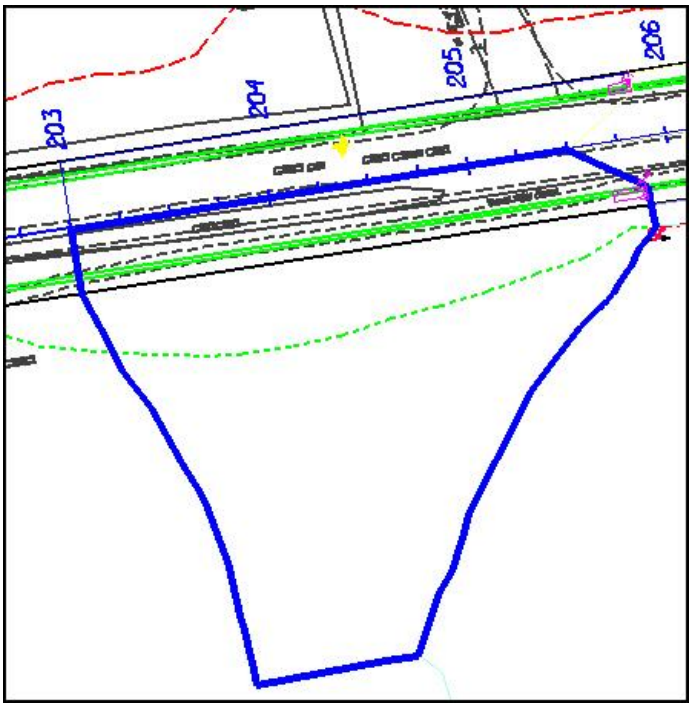
Step 7. Press the **Apply** button to include this node in the Drainage Project.

Step 8. Press the **Edit Area** icon to associate this inlet with its Drainage Area.

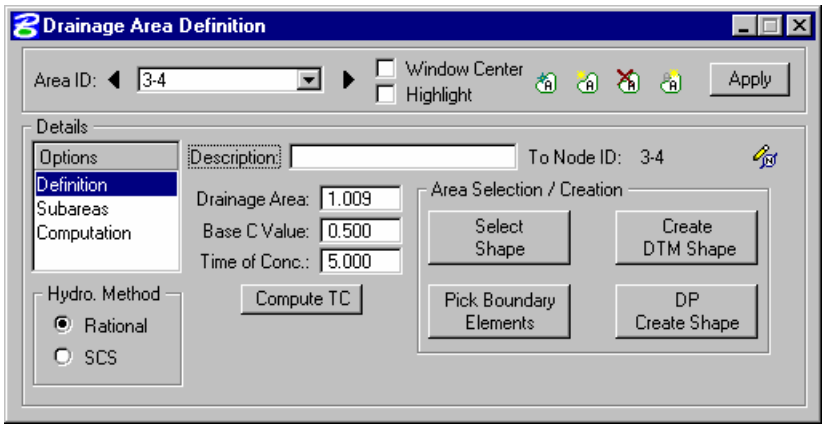
Step 9. Select **Yes** to the Alert dialog box, since no area for the inlet.

Create Drainage Area 3 – 4

This closed shape in lv = Level 63 will be selected to become the drainage area delineation.



Step 1. Follow the instructions below.



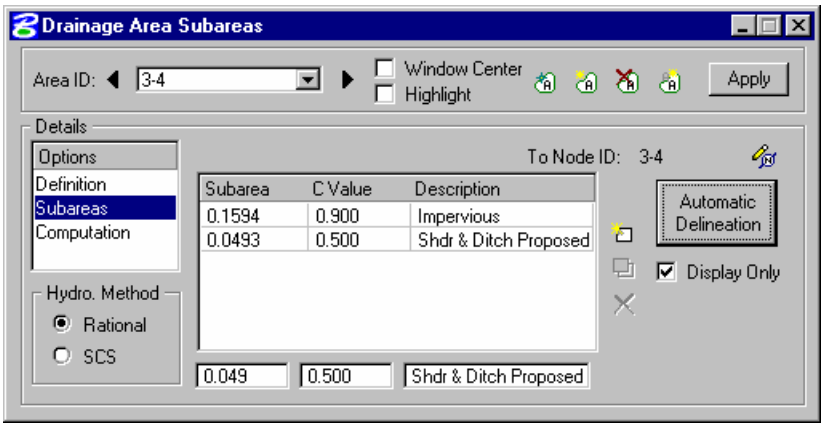
Drainage Area ID	3-4
Base C	0.5
Time of Concentration	5.00

Click **Select Shape** and select the MS shape for this area. Drainage area value is computed.

Step 2. Highlight **Subareas**.

Step 3. Press the **Automatic Delineation** button.

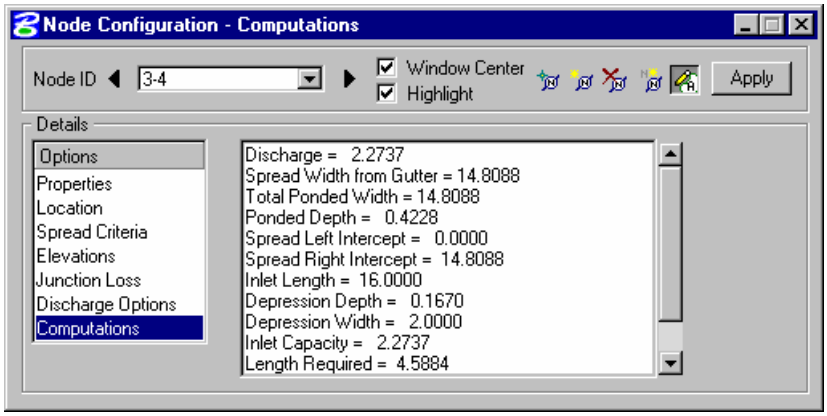
The file is scanned for closed shapes matching the Land Use symbology specified in the Drainage Library (Land Use Tab).



Step 4. Highlight **Computation** and press the **Compute Discharge** button to obtain the drainage area's computations.

Step 5. Verify the Computations; then press **Apply** to add the Area to the Project.

Step 6. Review the computations in the Node Configuration dialog.

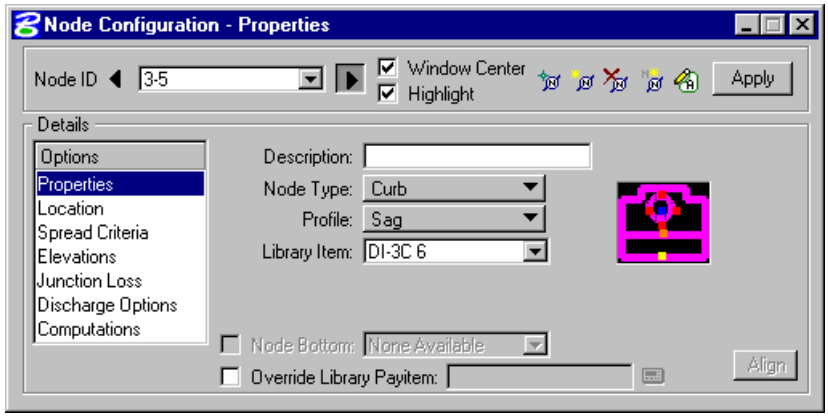


Design Inlet 3 – 5

Step 1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 206+36; offset 26.00.

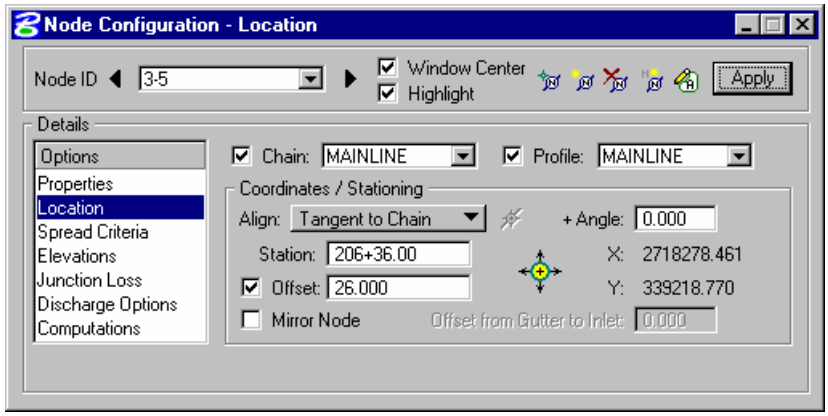
Step 2. Select from the Drainage menu: **Component > Node > Add.**

Step 3. Highlight **Properties.** Verify the Node Properties are defaulted from the previous node such we have to change a few properties.



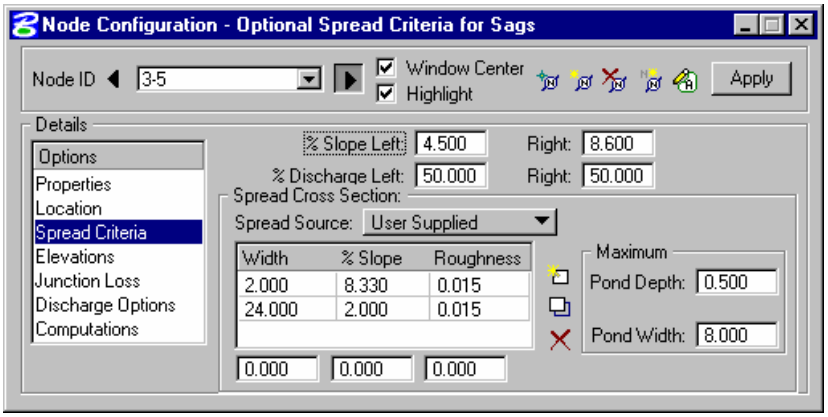
Node ID	3-5
Node Type	Curb
Profile	Sag
Library Item	DI-3C 6

Step 4. Highlight **Location.** All Reference information is defaulted from the previous Node (3-4) such that only the new Location is required:



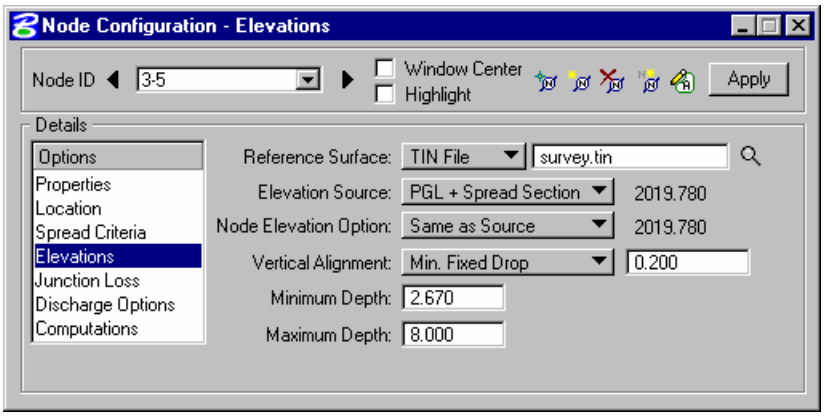
Station	206+36
Offset	26

Step 5. Highlight **Spread Criteria.** Set required information to describe the flow to the inlet.



Longitudinal % Slope Left	4.50%
Longitudinal % Slope Right	8.60%
% Discharge Left	50 %
% Discharge Right	50%

Step 6. Highlight **Elevations.** Set the required information to describe the inlet's elevation:



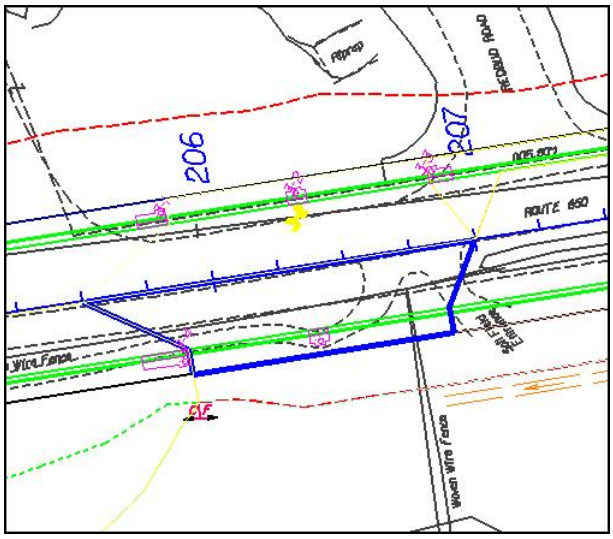
Step 7. Press the **Apply** button to include this node in the Drainage Project.

Step 8. Press the **Edit Area** icon to associate this inlet with its Drainage Area.

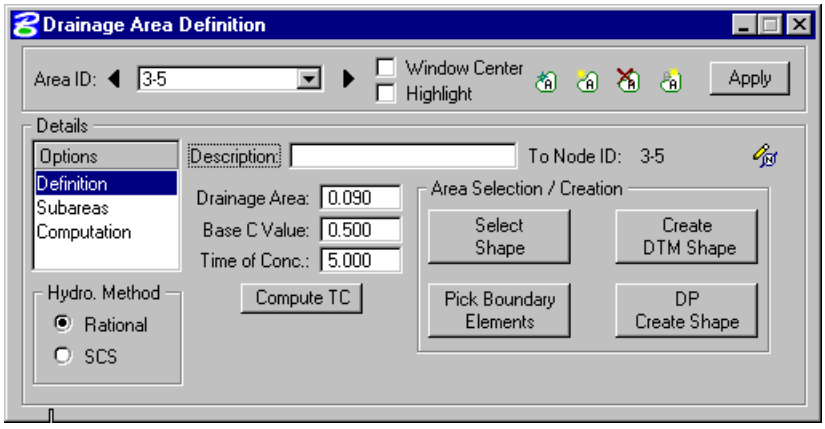
Step 9. Select **Yes** to the Alert dialog box, since no area for the inlet.

Create Drainage Area 3 – 5

This closed shape in lv = Level 63 will be selected to become the drainage area delineation.



Step 1. Follow the instructions in the table below.



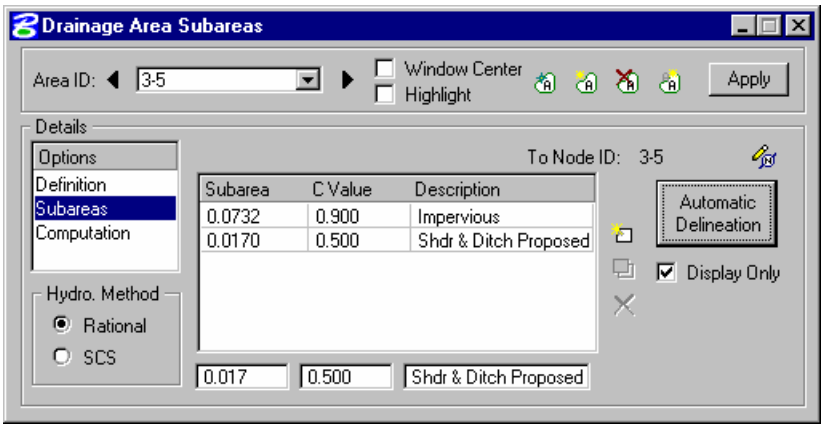
Drainage Area ID	3-5
Base C	0.5
Time of Concentration	5.00

Click **Select Shape** and select the MS shape for this area. Drainage area value is computed.

Step 2. Highlight **Subareas**.

Step 3. Press the **Automatic Delineation** button.

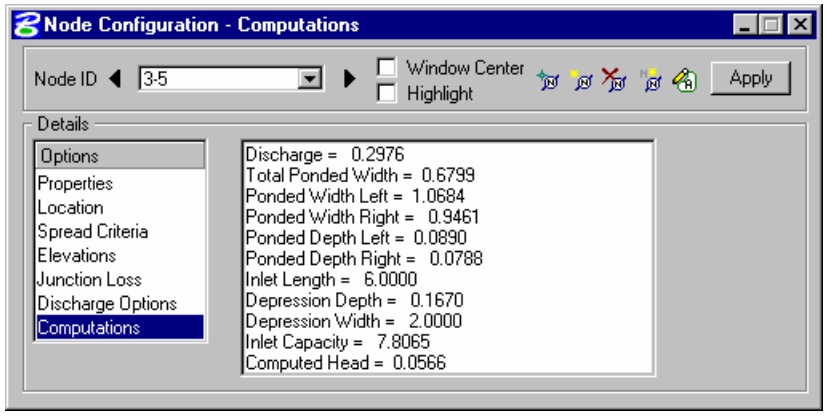
The file is scanned for closed shapes matching the Land Use symbology specified in the Drainage Library (Land Use Tab).



Step 4. Highlight **Computation** and press the **Compute Discharge** button to obtain the drainage area's computations.

Step 5. Verify the Computations; then press **Apply** to add the Area to the Project.

Step 6. Review the computations in the Node Configuration dialog.

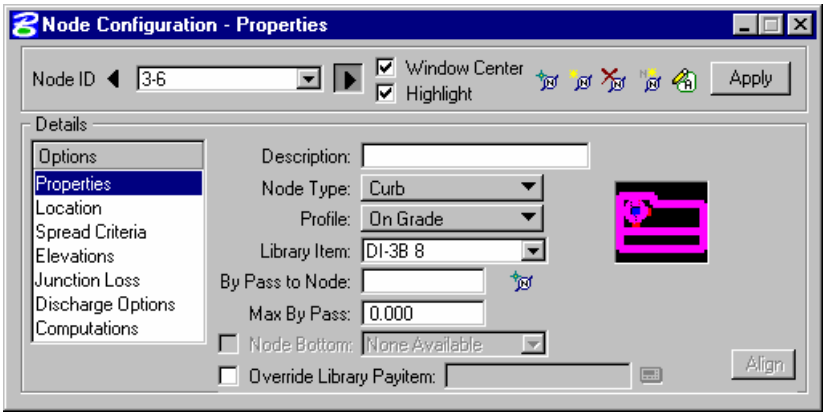


Design Inlet 3 – 6

Step 1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 206+86 offset 26.00.

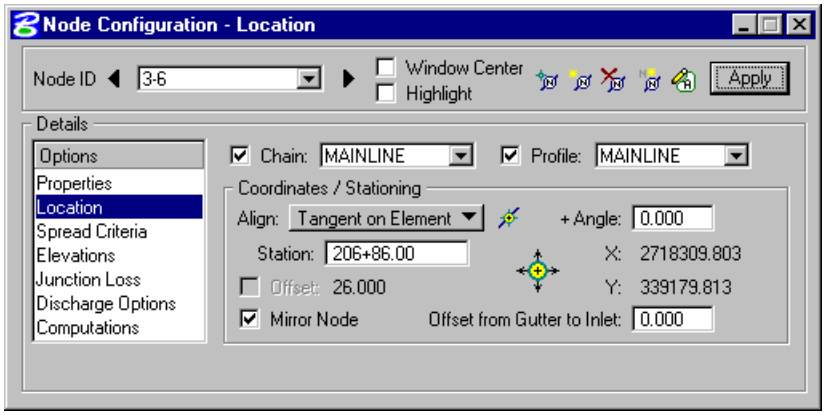
Step 2. Select from the Drainage menu: **Component > Node > Add.**

Step 3. Highlight **Properties.** Select the Node to be used from the Drainage Library and assign its properties as depicted below:



Node ID	3-6
Node Type	Curb
Profile	On Grade
Library Item	DI-3B 8

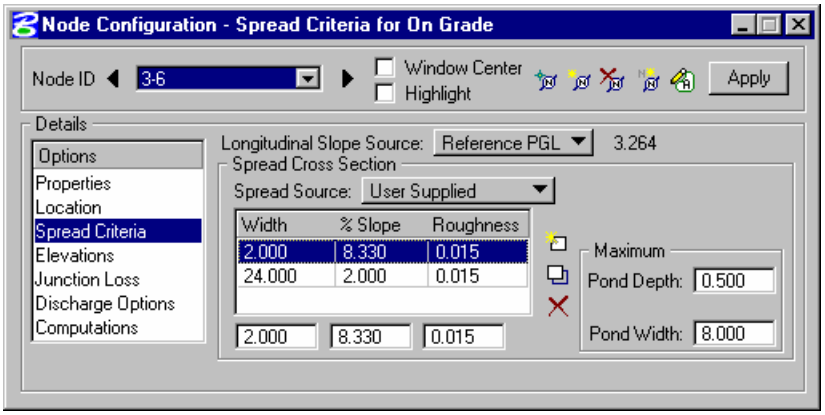
Step 4. Highlight **Location.** Describe the inlet's location in the design plane.



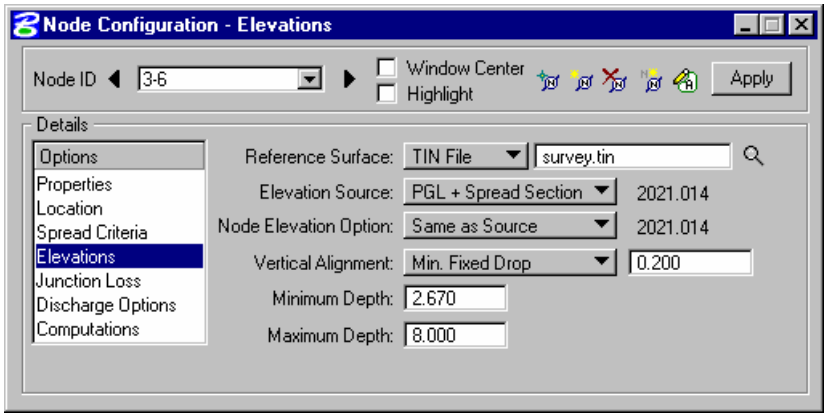
Station	206+86
Offset	26

Press **Enter** to automatically place the inlet at the specified location.

Step 5. Highlight **Spread Criteria.** Set required information to describe the flow to the inlet.



Step 6. Highlight **Elevations.** Set the required information to describe the inlet's elevation:



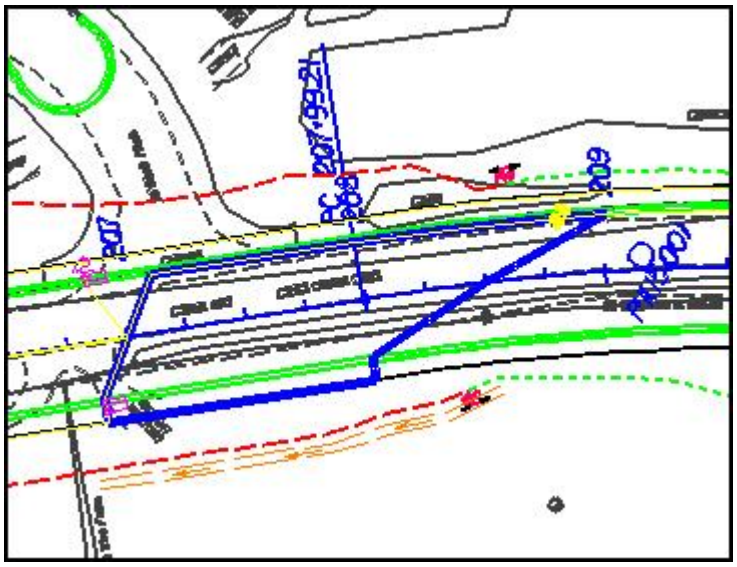
Step 7. Press the **Apply** button to include this node in the Drainage Project.

Step 8. Press the **Edit Area** icon to associate this inlet with its Drainage Area.

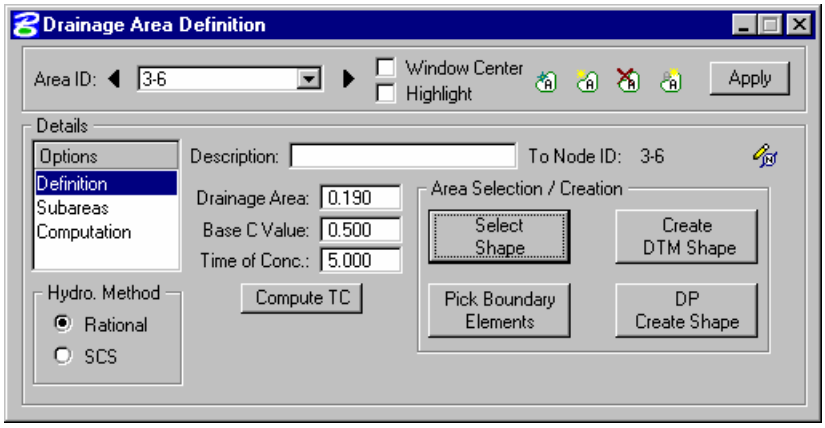
Step 9. Select **Yes** to the Alert dialog box, since no area for the inlet.

Create Drainage Area 3 – 6

This closed shape in lv = Level 63 will be selected to become the drainage area delineation.



Step 1. Follow the instructions below.

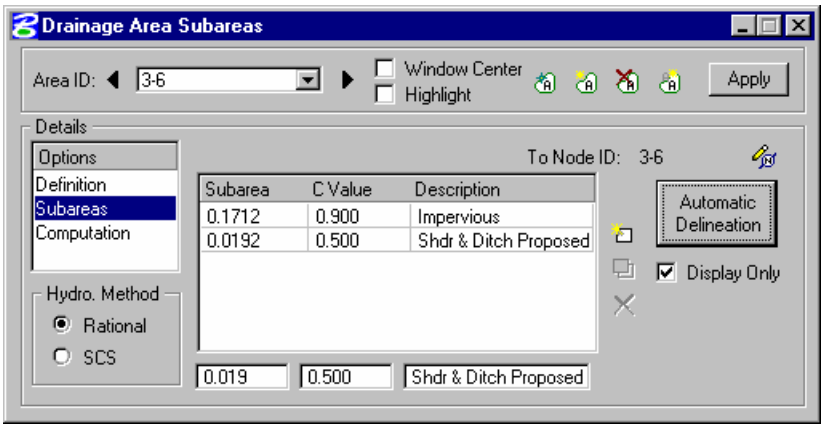


Drainage Area ID	3-6
Base C	0.5
Time of Concentration	5.00

Click **Select Shape** and select the MS shape for this area. Drainage area value is computed.

Step 2. Highlight **Subareas**.

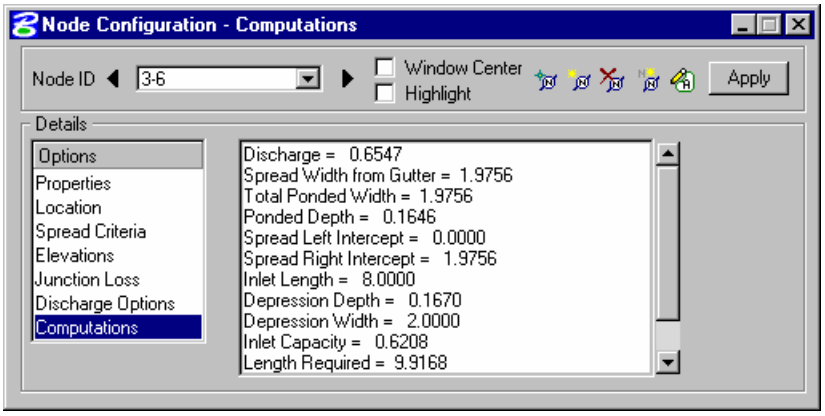
Step 3. Press the **Automatic Delineation** button. The file is scanned for closed shapes matching the Land Use symbology specified in the Drainage Library (Land Use Tab).



Step 4. Highlight **Computation** and press the **Compute Discharge** button to obtain the drainage area's computations.

Step 5. Verify the Computations; then press **Apply** to add the Area to the Project.

Step 6. Review the computations in the Node Configuration dialog.

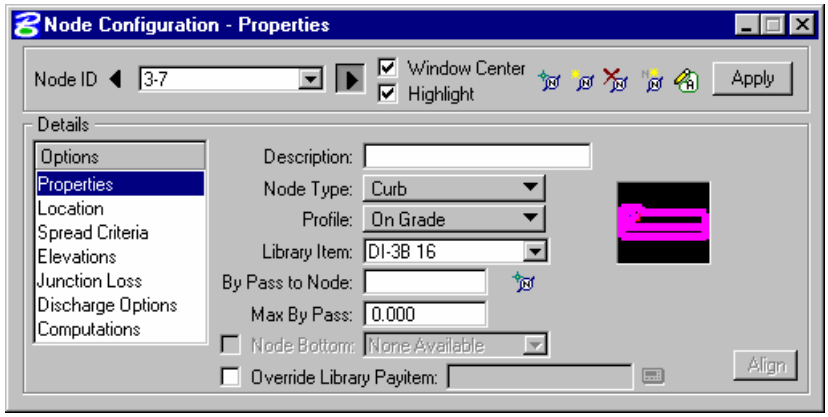


Design Inlet 3 – 7

Step 1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 208+00 offset 26.00.

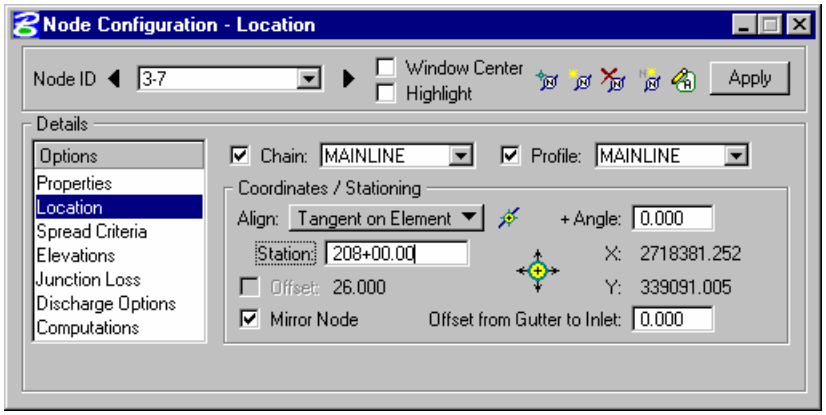
Step 2. Select from the Drainage menu: **Component > Node > Add.**

Step 3. Highlight **Properties.** Select the Node to be used from the Drainage Library and assign its properties as depicted below:



Node ID	3-7
Node Type	Curb
Profile	On Grade
Library Item	DI-3B 16

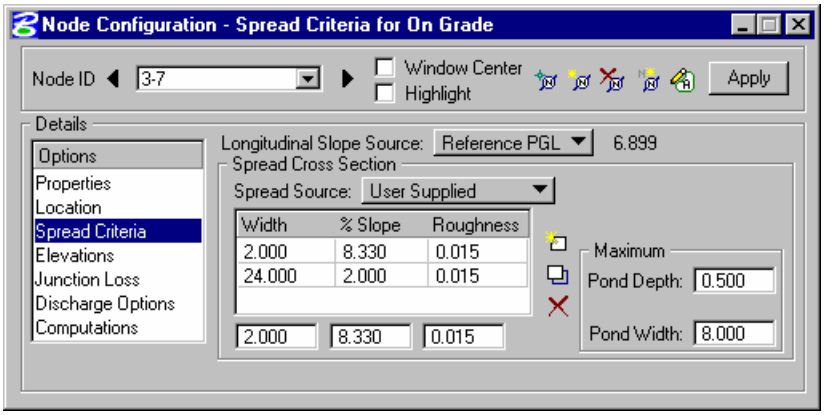
Step 4. Highlight **Location.** Describe the inlet's location in the design plane.



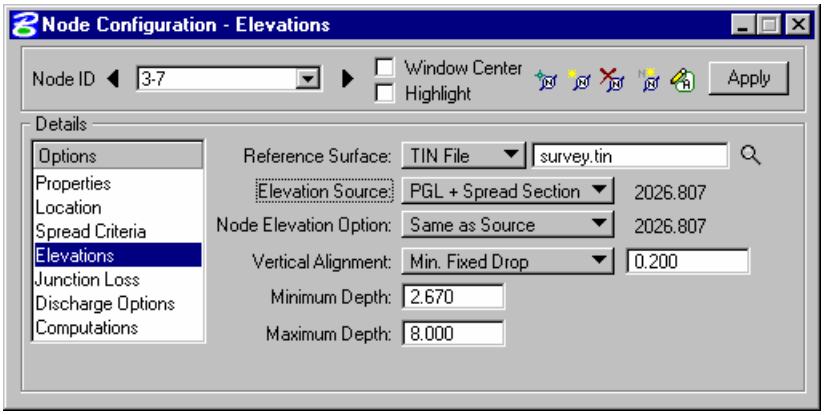
Station	208+00
Offset	26
Angle	0

Press **Enter** to automatically place the inlet at the specified location.

Step 5. Highlight **Spread Criteria.** Set required information to describe the flow to the inlet.



Step 6. Highlight **Elevations.** Set the required information to describe the inlet's elevation:



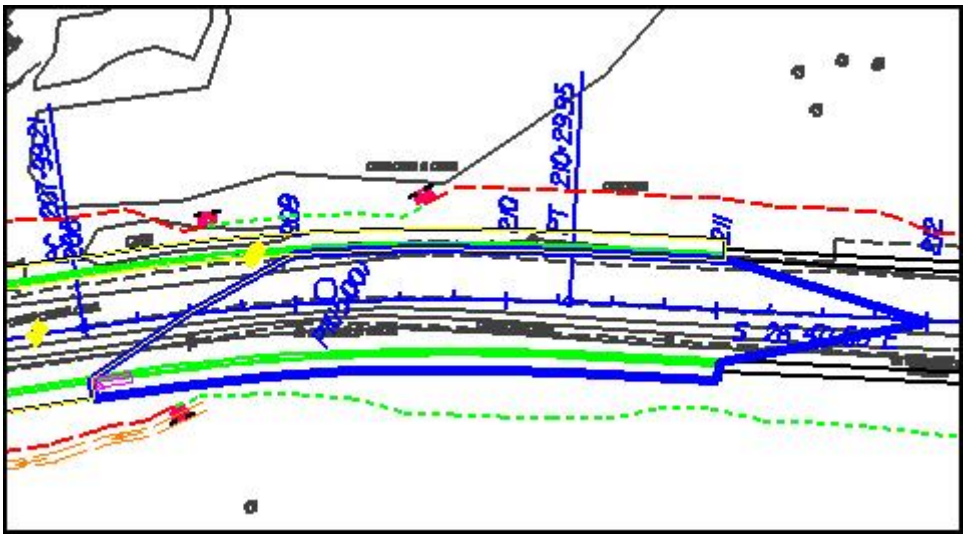
Step 7. Press the **Apply** button to include this node in the Drainage Project.

Step 8. Press the **Edit Area** icon to associate this inlet with its Drainage Area.

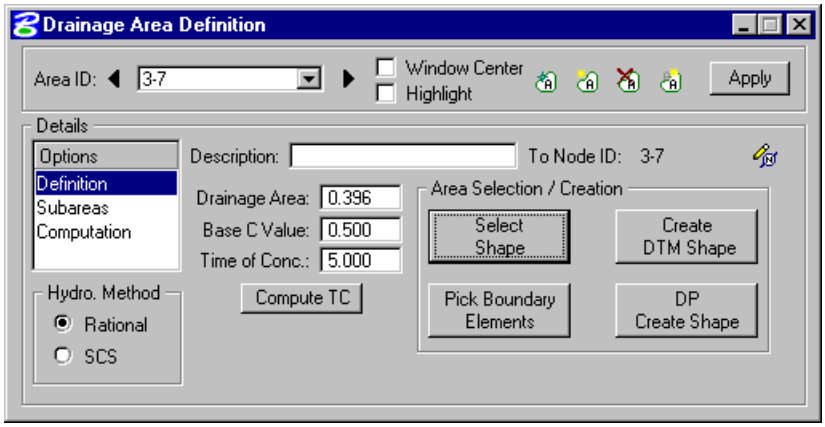
Step 9. Select **Yes** to the Alert dialog box, since no area for the inlet.

Create Drainage Area 3 – 7

This closed shape in lv = Level 63 will be selected to become the drainage area delineation.



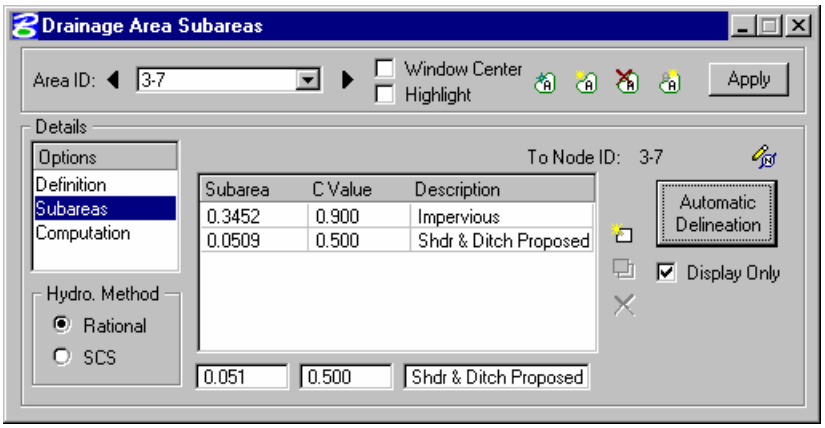
Step 1. Follow the instructions below.



Drainage Area ID	3-7
Base C	0.5
Time of Concentration	5.00

Step 2. Highlight **Subareas**.

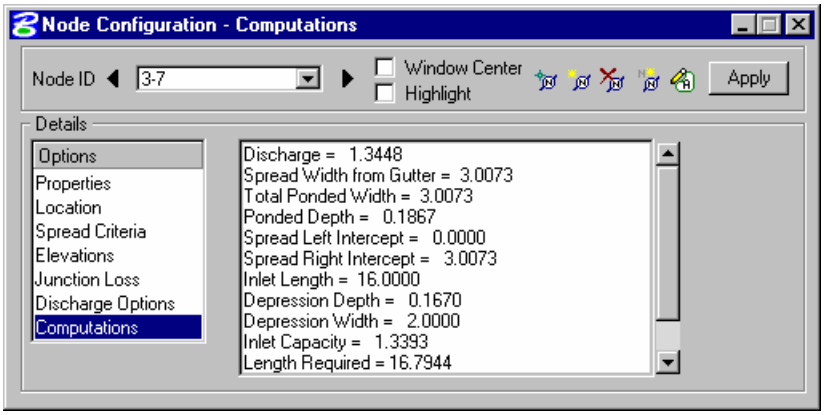
Step 3. Press the **Automatic Delineation** button. The file is scanned for closed shapes matching the Land Use symbology specified in the Drainage Library (Land Use Tab).



Step 4. Highlight **Computation** and press the **Compute Discharge** button to obtain the drainage area's computations.

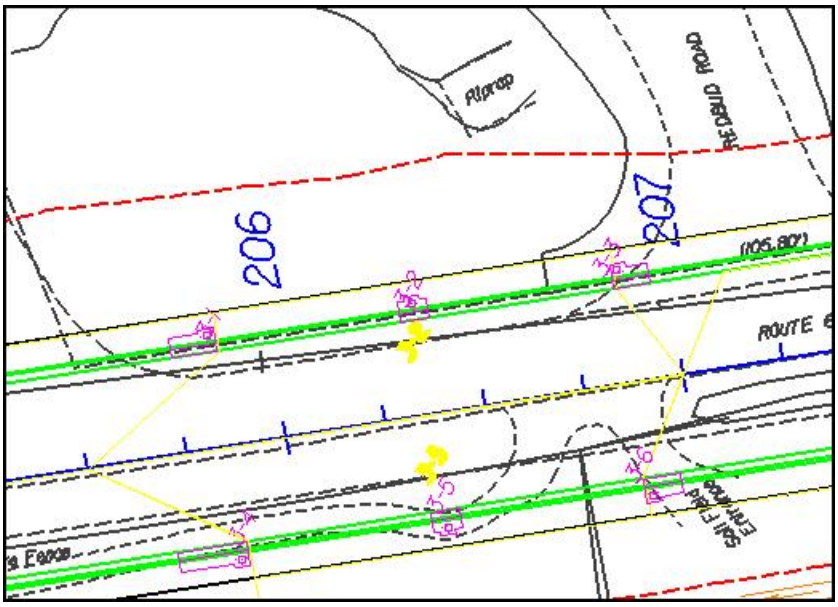
Step 5. Verify the Computations; then press **Apply** to add the Area to the Project.

Step 6. Review the computations in the Node Configuration dialog.



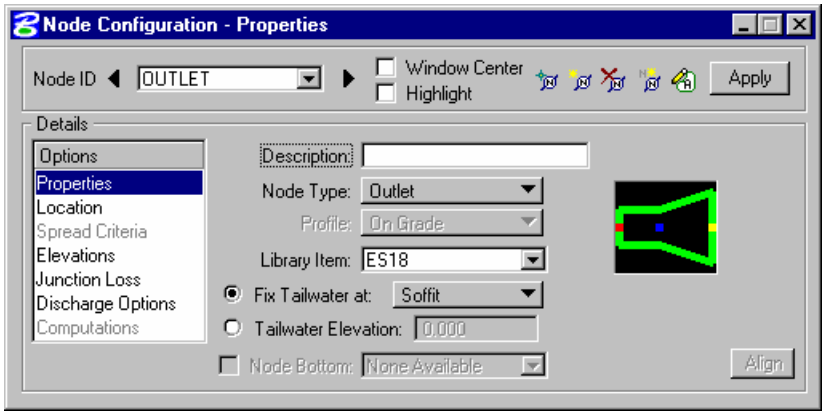
Design Node: Outlet

Step 1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 206+36; approximate offset -42.00:



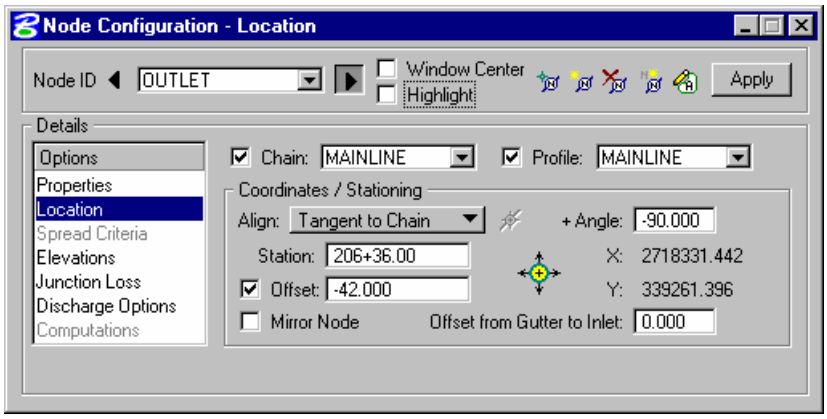
Step 2. Select from the Drainage menu: **Component > Node > Add.**

Step 3. Highlight **Properties.** Utilize the Standard *Outlet* for this node:



Node ID	Outlet
Description	(Optional)
Node Type	Outlet
Library Item	ES18
Fix Tailwater	Soffit

Step 4. Highlight **Location.** All Reference information is defaulted from the previous Node such that only the new Location is required:

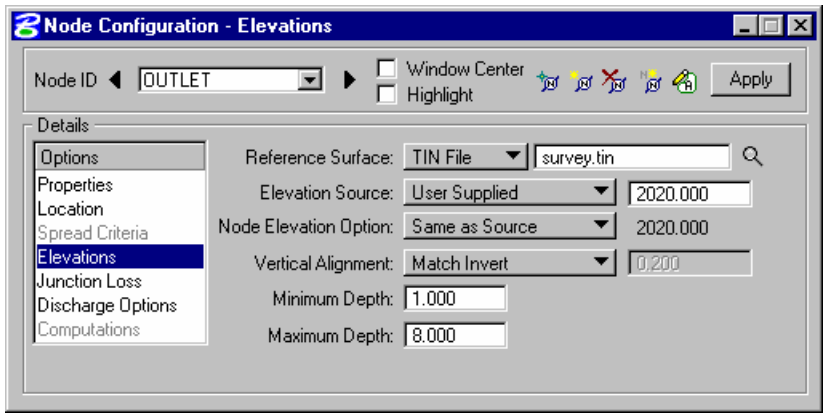


Station	206+36
Offset	-42
Angle	-90

Press **Enter** to automatically place the outlet at the specified location.

Step 5. Highlight **Elevations.** Set the Outlet's elevation using the User Supplied option.

Note: Since we are discharging into a ditch, the elevation should be the elevation of the top of the ditch. During the design of the conveyance system, we will adjust and hold the discharge elevation at the invert of the pipe.



Step 6. Press the **Apply** button to include this node in the Drainage Project.

Step 7. Exit MicroStation.

6 Conveyance System

Objective:	Review Drainage Links

Introduction

In GEOPAK Drainage, Links connect and convey runoff from the various Nodes within a Network to the Outlet, and may consist of pipes, boxes, or ditches. A multitude of options for sizing, and profiling Links are supported.

A Link represents a linear feature depicting a path connecting two Nodes. The path (upstream to downstream) need not be indicated because GEOPAK Drainage will determine the direction of flow purely based on connectivity. The path may be a straight line, line string, curvilinear, or a combination and series of linear MicroStation elements.

The Drainage Library is an integral part of designing and analyzing Links. The library contains all the standard pipe configurations based on shape, material, and, if applicable, corrugations which may be used in a Drainage project. Link library items selected to specify pipe sizes or the shape and material properties within the library may be used to establish the suitable candidates for design.

Alternately, the Link tools are invoked by selecting *Drainage* > *Tool Boxes* > *Link*, then identifying the desired tool from the tool box as depicted below.

Tools from left to right are:

- Add Drainage Link
- Modify Drainage Link
- Delete Drainage Link
- Update All Links
- Update All Links with Pay Items
- Drainage Link Report – Configuration
- Drainage Link Report – Computation

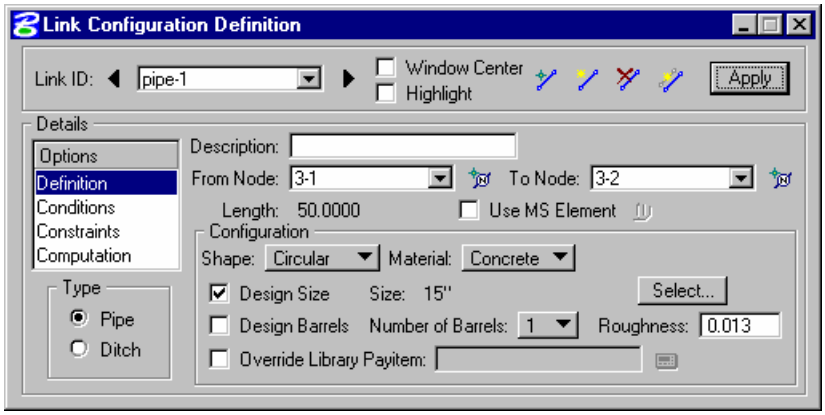
In addition, the dialog can be invoked via the **Navigator**.

Five options are supported under the Link pull down as described in the table below.

Add	Utilized to add a Link to the current GEOPAK Drainage project.
Edit	Utilized to select and edit any previously defined and stored Link within the current project. Note the Link Edit dialog is identical to the Link Add dialog, however, when the Link is identified, all associated data is displayed.
Delete	Utilized to select and delete the specified Link and associated data.

Update All Links Updates all link graphics.

Update All Links with Pay Items Updates all link graphics with the designated pay items.



Link options and types are shown in the Option list box on the left side of the dialog. The corresponding fields and items appear on the right. The four options are reviewed below.

Definition Defines the Link's path, connectivity, shape, material, roughness coefficient, size, and design options.

Conditions Defines or reviews the elevation condition for the Link including minimum depth, soffit, invert, and slope data. It also includes the profiling options for holding certain values constant.

Constraints Defines the Link constraints including the minimum and maximum rise (size), slope and velocity used in Link design.

Computation Reviews the Link hydraulic computations

When all the options have been reviewed and appropriate information supplied, pressing the **Add** or **Update** button will update the Drainage database.

Lab 6: Links

Link Design

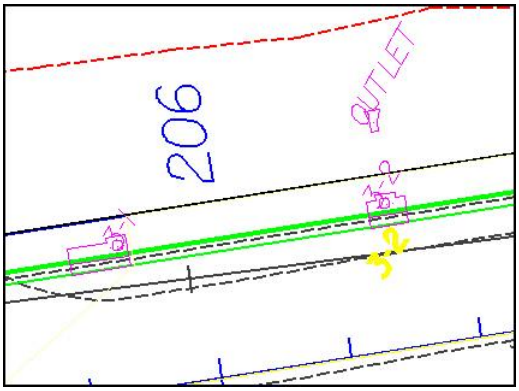
Step 1. Execute **c:\data\geo\VDOT\drain1\LAB6_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the Geopak Drainage pull down menu.

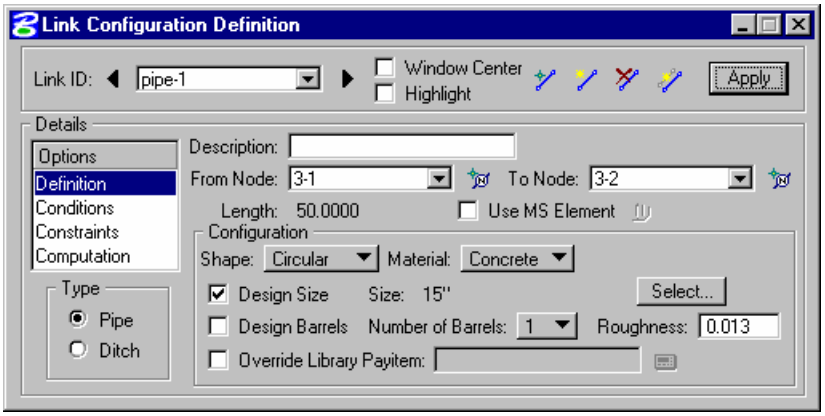
Step 5 Visually determine the tentative location of the first storm sewer pipe. This link will connect Nodes 3-1 and 3-2.



Step 6. Select from the Drainage Menu : **Component > Link > Add**.

Step 7. Highlight **Definition** to Set the pipe's spatial characteristics including From and To Nodes ID's, pipe shape, material, library items, etc.

Graphically select the Nodes by clicking the **ID** button for each one. Link 1 traverses From Node 3-1 To Node 3-2:

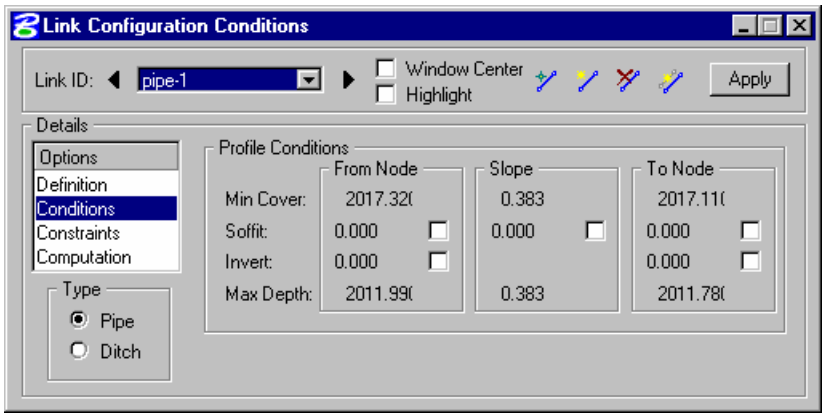


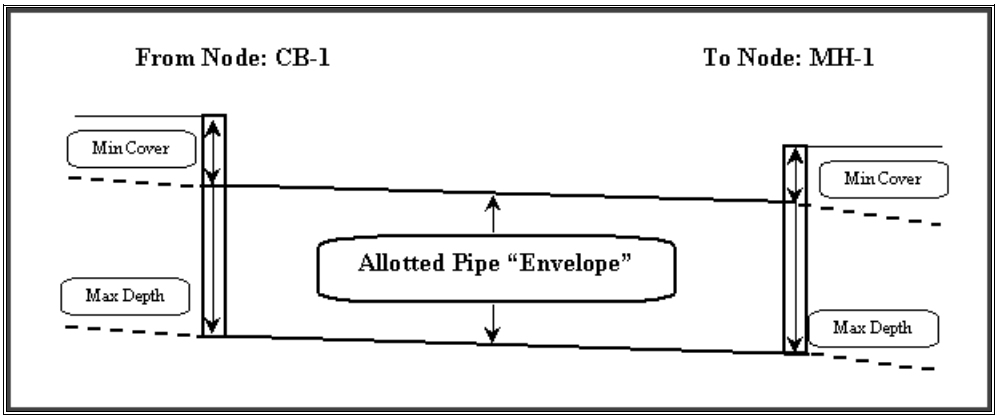
Shape	Circular

Material	Concrete
Design Size	Toggle ON
Design Barrels	Toggle OFF & select 1
Manning's n	0.013

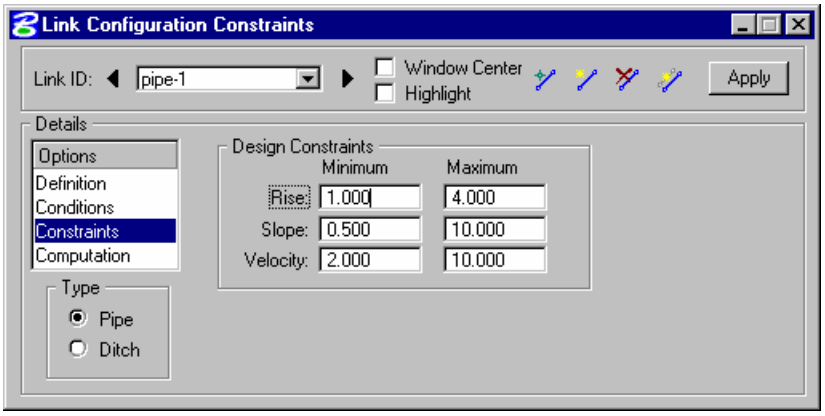
Step 8. Highlight **Conditions**. The elevations shown are based on the *From-Node* and *To-Node* elevation minus the min/max depth, as specified in the Node Definition Dialog Box for Nodes 3-1 and 3-2 respectively.

In this case, no entries are necessary; Geopak Drainage will design all the profiles for this project.



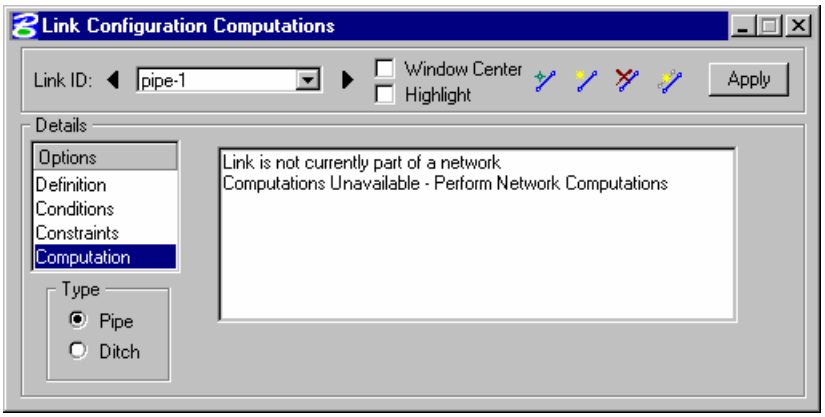


Step 9. Highlight **Constraints.** Establish the min/max design criteria for Links as follows:



Rise min/max	1.0 / 4.0 (feet)
% Slope min/max	0.50 / 10.00 (%)
Velocity min/max	2.00 / 10.00 (fps)

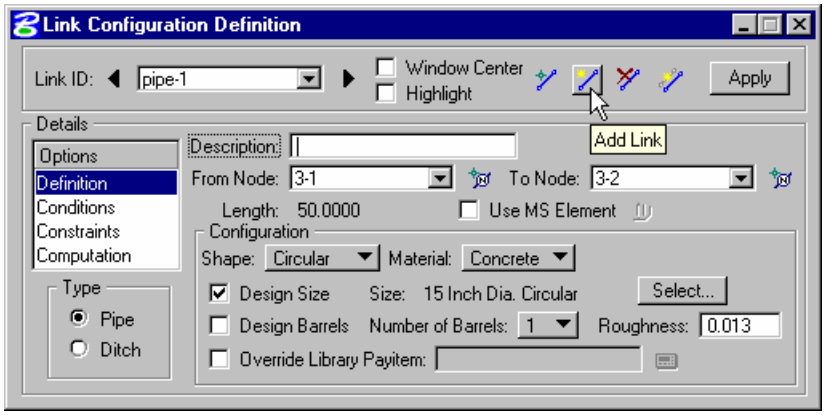
Step 10. Highlight **Computations** to display the computed hydraulic properties of the Link.



Note: Link hydraulics are not available for review until a Network has been established and designed or analyzed (next exercise) successfully. Check back here for computations after the Network has been added and designed or analyzed.

Step 11. Press **Apply** to incorporate the link to the project.

Step 12. Add the remainder of the link conveyance system from the Link Configuration dialog.



As Links are added, most dialog values default from the previous Link with the exception of the node and elevation information. Add Links between all of the following Nodes:

Link PIPE-2 Traverses From Node 3-3 To Node 3-2
Link PIPE-3 Traverses From Node 3-4 To Node 3-5
Link PIPE-4 Traverses From Node 3-7 To Node 3-6
Link PIPE-5 Traverses From Node 3-6 To Node 3-5
Link PIPE-6 Traverses From Node 3-5 To Node 3-2
Link PIPE-7 Traverses From Node 3-2 To Node OUTLET and hold the discharge elevation of the pipe at el. 2013.00

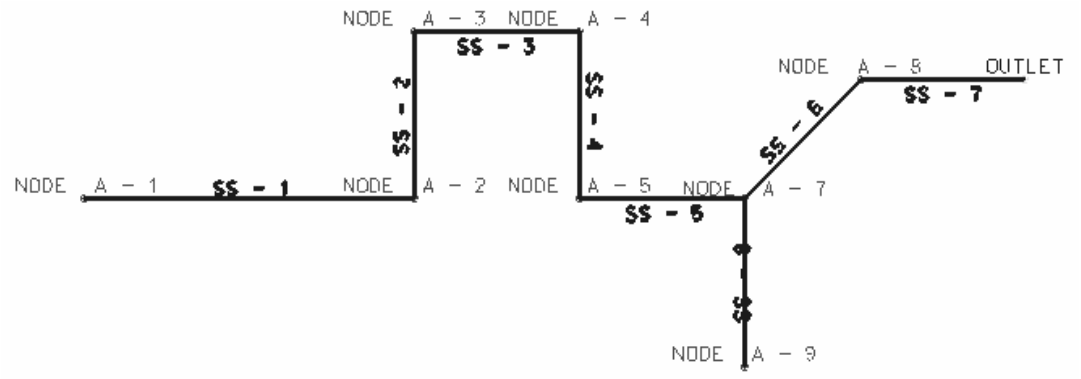
Step 13. Exit MicroStation.

7 Networks

Objective:	Review Network Computations.

Introduction

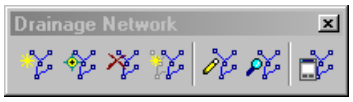
A GEOPAK Drainage Network is defined as a series an interconnected Nodes and Links draining to a single outlet. GEOPAK Drainage can maintain multiple Networks in a single project.



The Network computations serve as the final calculation process in the design or analysis of a storm drain system. Drainage Areas and Inlets may be computed individually and are not dependent on any type of Network topology. Pipes and Ditches, however, are dependent on the connectivity and Network characteristic and therefore, require a Network be defined and successfully built, in order to complete the hydraulic computations on these features.

Several options are supported to add, edit, and delete Networks and are invoked via the Network pull down on the menu bar.

Alternately, the Network tools are invoked by selecting **Drainage > Tool Boxes > Network**, then identifying the desired tool from the tool box as depicted below.



Network tools from left to right are:

- Add Drainage Network
- Modify Drainage Network
- Delete Drainage Network
- Set Active Network
- Design Network
- Analyze Network

Six options are supported under the Network pull down:

Add - Initially utilized to define the Network and associated data. When invoked, the Network Configuration dialog is blank.

Edit - Utilized to edit any previously defined Network. When selected, the Select Network dialog is invoked, wherein the desired Network is highlighted. Note the Network Edit dialog is identical to the Network Add dialog, however all associated data is displayed.

Delete - Utilized to delete the specified Network and associated data.

Design - This mode enables when an Active Network is defined. It's a shortcut to the Design procedure without having to invoke the Network dialog.

Analyze - This mode enables when an active network is defined. It's a shortcut to the Analyze procedure without having to invoke the Network dialog.

Active Network - GEOPAK Drainage uses an Active Network as a shortcut and organization tool. Many of the reporting, query, and navigation tools support the use of the Active Network to limit the information viewed to the system currently under design (the Active Network). When selected, the Active Network dialog is invoked, wherein a previously defined Network may be specified as active. The current Active Network is displayed on the main menu bar for quick reference.

LAB 7: Networks

Network Design

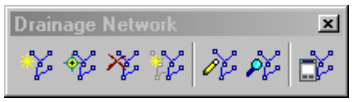
Step 1. Execute **c:\data\geo\VDOT\drain1\LAB7_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

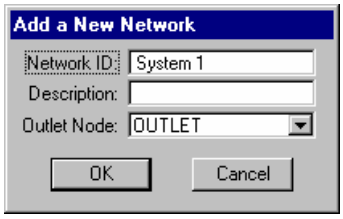
Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the Drainage pull down menu.

Step 5. Select the *Add Drainage Network* tool from **Drainage > Network > Add** or from the Network tool palette.

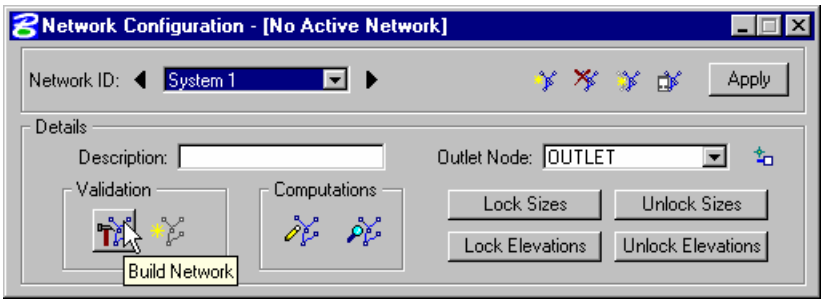


Step 6. In the *Add a New Network*, enter the following information:



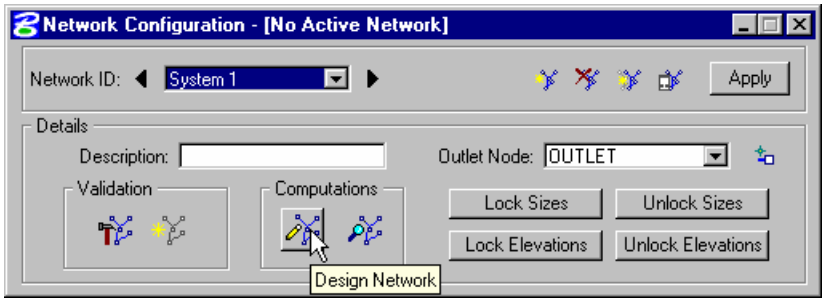
Network ID:	SYSTEM1
Description:	(Optional)
Outlet Node	Click the **ID** button and select the OUTLET

Step 7. Click the **Build Network** button. This feature verifies the nodal topology and link connectivity. The **Highlight Network** feature highlights all components (areas, inlets, pipes, etc.) connected to the active Network.



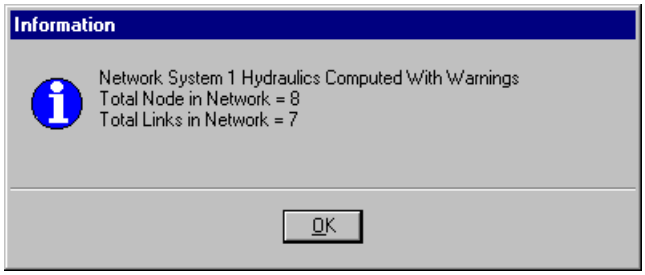
Step 8. Click the **Apply** button. Network "SYSTEM1" has been added to the project.

Step 9. Click the **Design** button. This command initiates the hydraulic design of the components contained in the Network.



Step 10. Accept the Update Graphics messages to update all node and link data if required (if any graphic or hydraulic information is missing from the components, this procedure will update the information):

Step 11. Then GEOPAK will design the network and issue the pertinent Warning messages, if necessary.



Step 12. Set the Active Network to *System1*, so we can review some of the provided reports. **Drainage > Network > Active Network**.

Step 13. We can now review some of the Geopak provided reports. **Drainage > Reports > Storm Drain/Links.**

Step 14. Exit MicroStation.

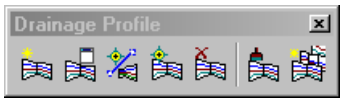
8 Profiles

Objective:	Review Profile Computations.

Introduction

A Profile is a path between two nodes, spanning one or more links. Profiles allow users to define a path running in any direction (upstream or downstream) in a drainage Network and visualize profiles (ground, pipes, depth of cover...) along that path. All that is needed to define a profile is the identification of the From Node and To Node, and GEOPAK Drainage automatically traverses the Network and finds the Links connecting the two Nodes.

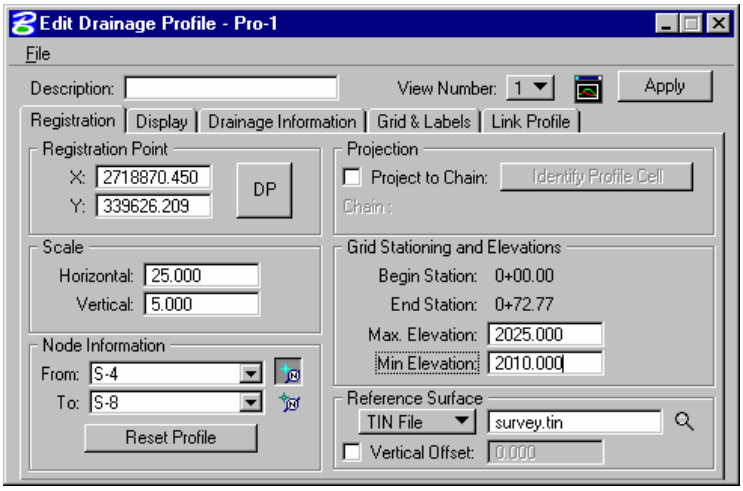
Several options are supported to add, edit, and delete Profiles and are invoked via the Component pull down on the Drainage menu. Alternately, the Profile tools are invoked by selecting *Tools* > *Tool Boxes* > *Profiles*, then identifying the desired tool from the tool box as depicted below.



The Drainage Profiles tool box contains seven tools (from left to right). In addition, the tools can be invoked via the **Navigator.**

- Add Drainage Profile : Utilized to add a Profile to the current GEOPAK Drainage project.
- Drainage Profile List: List of profiles in the current drainage project
- ID Drainage Profile: Utilized to select and edit any previously defined and stored Profile within the current project.
- Modify Drainage Profile: Utilized to modify the display properties of a plotted profile.
- Delete Drainage Profile: Utilized to select and delete the specified Profile and associated data.
- Update All Drainage Profiles
- Auto Create Profile: The Auto Create Profile tool draws a series of profiles based on the specified network.

When the Add, Edit, or Delete options are selected, the dialog depicted below is displayed.



The dialog is comprised of the Description area and five tabs. The fields detailed in the table below are independent of which tab is selected.

Profile ID	Identification of the Profile is displayed in the Dialog Title bar. (Maximum of 16 alphanumeric characters.)
View Number	Select the view in which the profile is to be drawn.
Center Profile	When clicked, the profile is centered in the specified view.
Description	Description of the Profile. (maximum of 32 alphanumeric characters.) This field is always at the top of the dialog, above the tabs.
Apply	Applies the current Profile information in the project.

The Edit Drainage Profile dialog under the Registration tab is the one that control the display of the profile. Its setup is reviewed in the table below.

Registration Point:	X coordinate corresponding to the specified Station, Elevation. Y coordinate corresponding to the specified Station, Elevation.
DP	Identify x, y coordinates using a data point. Click, then identify and accept the desired point on the screen.
Scale:	Horizontal and vertical scales. All profiles are drawn to true dimensions in the horizontal, so the scales are used to determine the distortion factor of the vertical.
Node Information:	Node used as the starting point of the profile. This is populated using the From Node in the New Profile dialog.
Reset Profile	This will reset the profile display to the current setting of the dialog.
Projection	If the Profile is to be projected onto a chain in order to utilize its stationing, the toggle is activated and chain parameters specified.
Grid Stationing and	If Chain Projection is not used, the Stationing at the From Node is set to

Elevations:	0+00 and is assigned to the Registration Point. If Chain Projection is used, the Station matches the Begin Station in the Projection group box. The ending station is automatically computed based on the length of the profile and the Beginning Station.
Reference Surface	Surface utilized to draw the ground profile above the Profile. A site model/object may also be specified. If blank, individual Node elevations are utilized.

LAB 8: Profiles

Profile Design

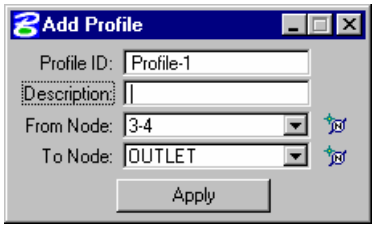
Step 1. Execute **c:\data\geo\VDOT\drain1\LAB8_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

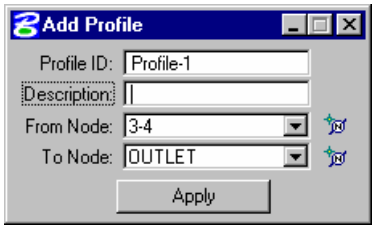
Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the GEOPAK Drainage pull down menu and make *System1* the Active Network using **Network > Active Network.**

Step 5. Select the Add Drainage Profile tool, **Drainage: Component > Profile > Add.**



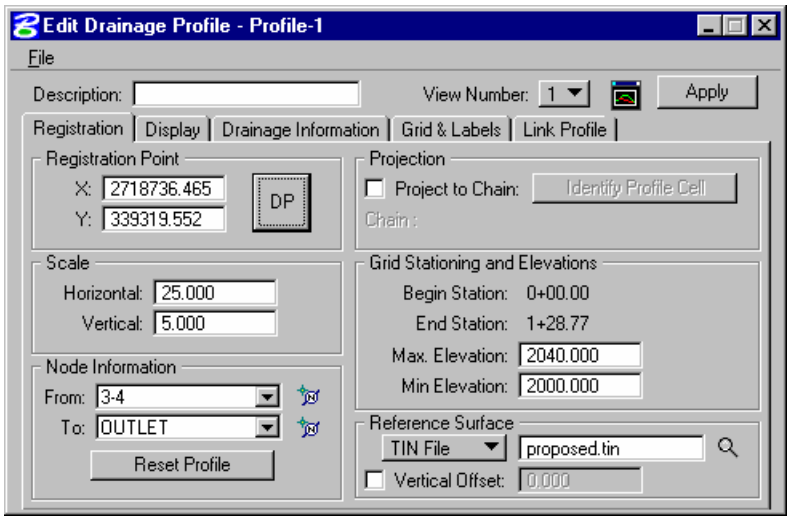
Step 6. Complete the dialog box information as follows:



Profile ID:	Profile-1
Description:	Optional
From Node:	Click the **ID** button and select 3-4.
To Node:	Click the **ID** button and select OUTLET.

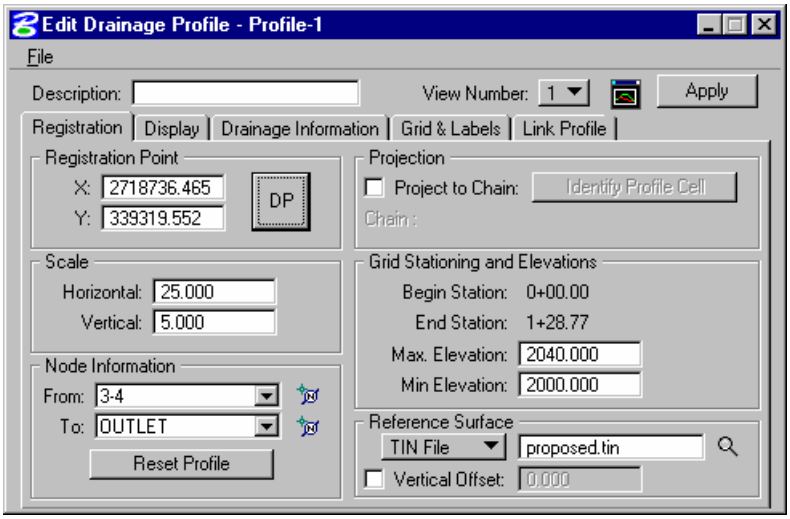
Step 7. Click **Apply.**

Step 8. Enter the profile information.



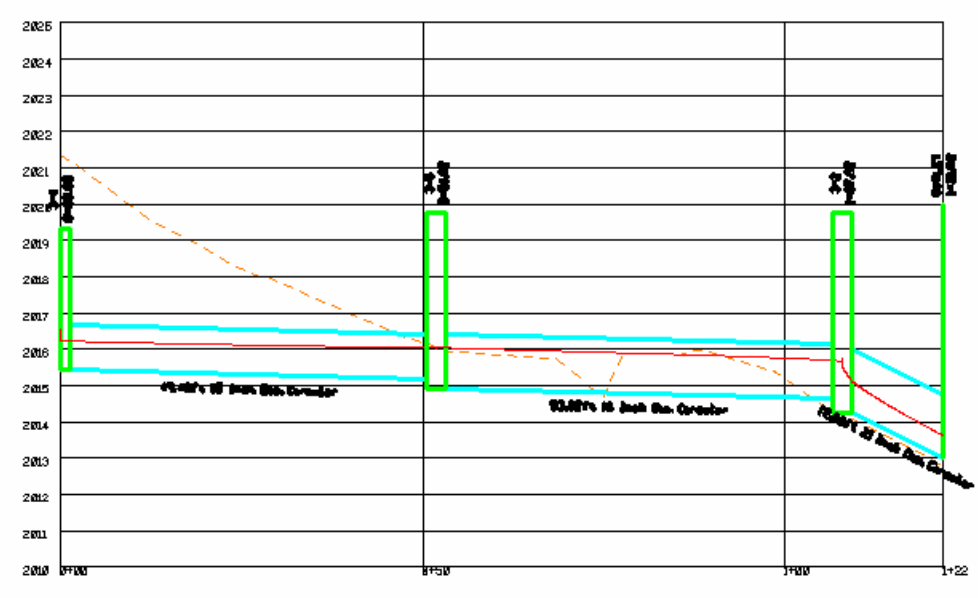
Reference Surface:	Select *proposed.tin*
Min. Elevation:	2010.00 (The profile datum)
Max. Elevation	2025.00
Horizontal Scale:	25.0
Vertical Scale:	5.0

Step 9. Click DP and select a data point in the MicroStation design plane. This is the location where the profile will be drawn.



Step 10. Select **File > Open** and select the preferences file : ***Drain_Profile.ppf*** *located* on ***c:\data\geo\vdot\drain1\standards\Geopak***

Step 11. Click **Apply**. Geopak Drainage draws the profile.



Step 12. Exit MicroStation.

9 Navigator/Global Editor

Objective:	Review the Navigator.
Goal:	Review the Navigator.

Introduction

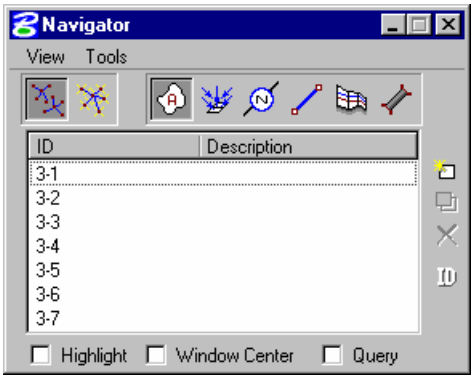
The Navigator tool is an easy method of maneuvering through the components of a GEOPAK Drainage Project. It provides a means to identify, add, edit, delete, and query the components of the current project. The Navigator also provides graphical maneuvering tools to move through the design file and view the components with highlighting and window centering functions.

In addition, the Navigator provides a Global Editor to edit parameters of the project components on a system wide basis. A tool is also provided to update the graphic display of any or all of the components in the Navigator List.

Using the Navigator

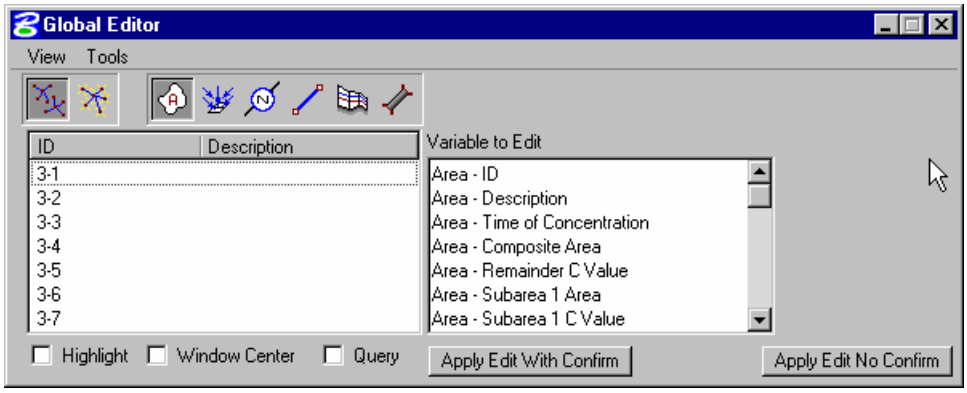
The Navigator is accessed via *Tools* > *Navigator* from the main menu bar.

Alternately, the Navigator tool can be invoked by selecting *Drainage* > *Tool Boxes* > *Main.* When selected, the dialog depicted below is invoked.



Global Editor

The Global Editor is used to edit the input and parameters of the project system on a system wide basis. When the *View* > *Global Editor* is selected from the Navigator tool bar, the dialog expands as depicted below.



LAB 9: Navigator/Global Editor

Navigating/Global Editing

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB9_V8.EXE**.

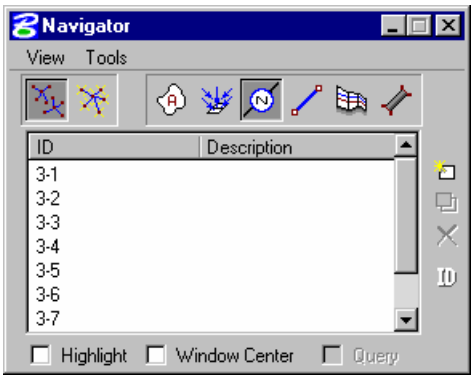
Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

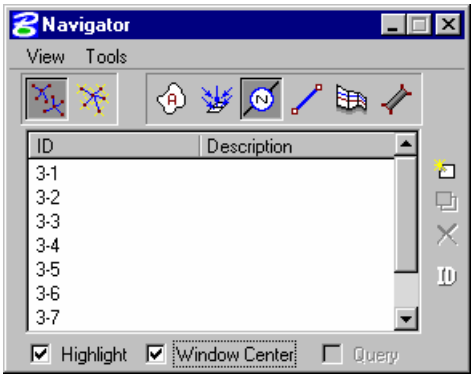
Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the GEOPAK Drainage pull down menu and make *System1* the Active Network using **Network > Active Network.**

Step 5. Select from the Drainage Menu **Drainage > Utilities > Navigator**.

Step 6. The Navigator opens defaulted to *All Networks*.

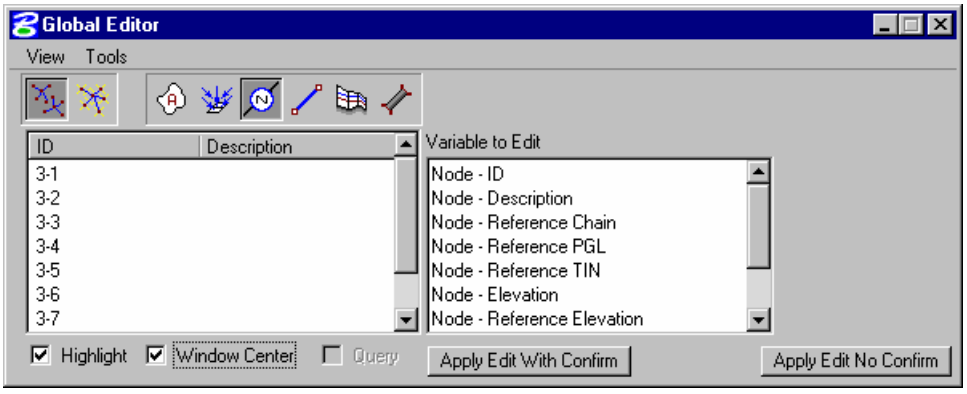


Step 7. Select the Drainage **Nodes** button on the *Navigator*:



Step 8. Toggle ON the **Highlight** and **Window Center** tools and click on the nodes in succession for easy navigation of any Drainage component.

Step 9. Expand the Navigator from the Navigator's *View* pull-down menu to the *Global Editor:*



Step 10. Select all the Node components. In the Variable to Edit list box, select *Node – Minimum Depth*, supply a New Value of 3.0 feet, click the **Apply Edit No Confirm** button to change all values.

Step 11. Redesign it with these changes in effect. ***Drainage > Network > Edit*** and select the design network icon.

Step 12. The previously plotted profile will be updated automatically.

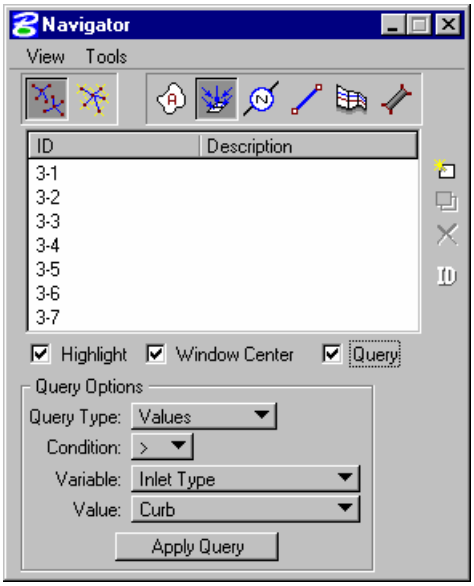
Step 13. Exit MicroStation.

10 Querying

Objective:	Review the Navigator's Query mode.

Querying From the Navigator

The Navigator provides a Query Tool to specify a subset of the current components based on a query of values or constraints. The Query Tool is only enabled for Areas, Inlets, and Pipes for which numerous computed values and design constraints are available for the query. When the Query toggle is activated, GEOPAK Drainage compiles a list for the selected component based on a designer's specified condition(s). The dialog depicted below illustrates the invocation of the query feature.



A query can be one of two types; Values or Constraints. A Value query will perform a search for components whose input or computed value for a specified variable meets the specified Condition and Value. A Constraint query will perform a search for components whose computed value for a design constraint meets the Condition**.**

LAB 10: Query

Query Mode

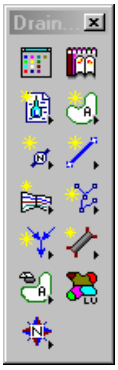
Step 1. Execute **c:\data\geo\VDOT\drain1\LAB10_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

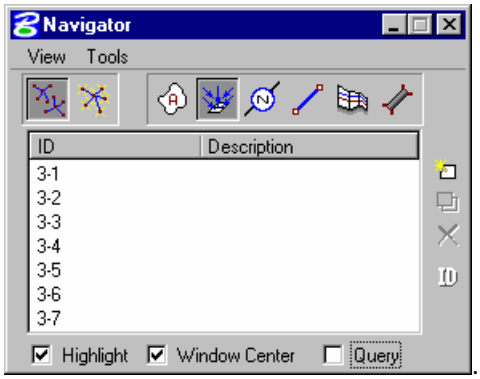
Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the Geopak Drainage pull down menu. and make *System1* the Active Network using **Network > Active Network.**

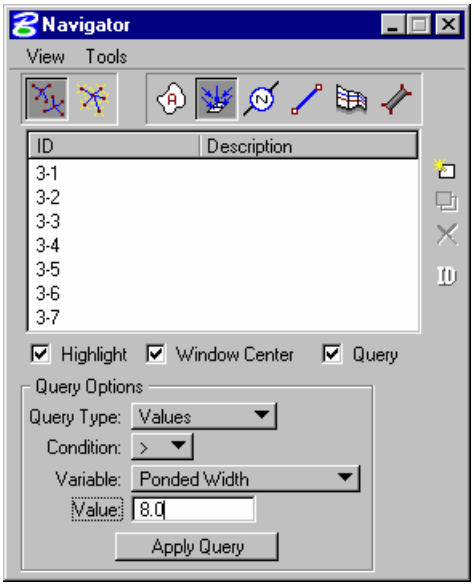
Step 5. Select from the Drainage Menu : **Utilities > Navigator**



Step 6. Set the Active Component Type to Inlet and toggle the **Query** option



Step 7. Use the Query tool to determine which inlets have exceeded a Ponded Width of 8 ft.



Query Type:	Values
Condition:	>
Variable:	Ponded Width
Value:	8.00

The components remaining in the navigator meet the conditions of the query.

Step 8. Follow the same procedures to determine the following:

Which pipes have a velocity greater than 3 fps? _____

Which pipes have exceeded their maximum diameter? _____

Which inlets have a ponded width greater than 6 feet? _____

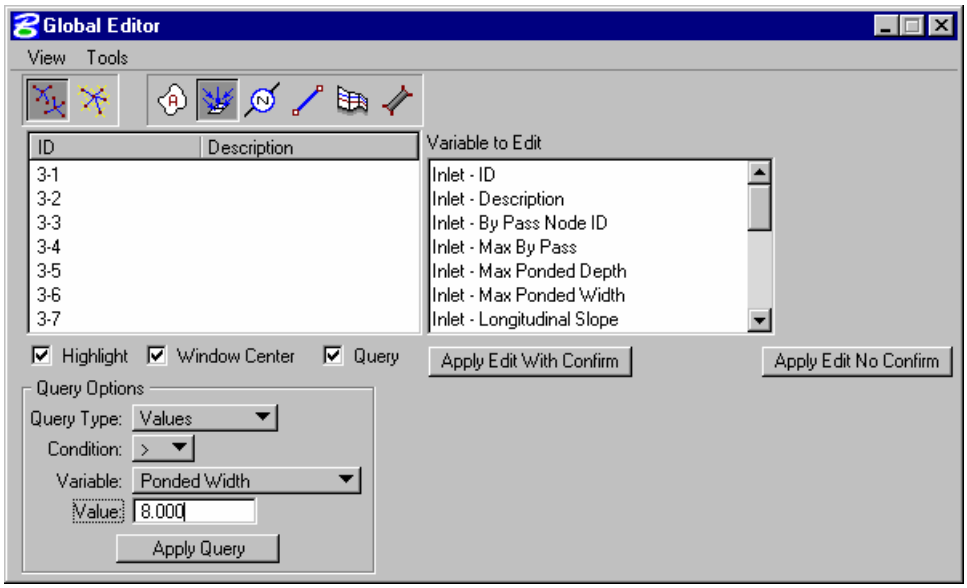
Step 9. Exit MicroStation.

11 System Modification and Analysis

Objective:	Review System Modification.

Global Editor

The Global Editor is used to edit the input and parameters of the project system on a system wide basis. When the *View* > *Global Editor* is selected from the Navigator tool bar, the dialog expands as depicted below.



The left side of the dialog is identical to the Navigator, hence, will not be discussed again.

LAB 11: System Modification and Analysis

Adding Existing Structures

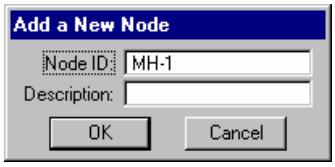
Step 1. Execute **c:\data\geo\VDOT\drain1\LAB11_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

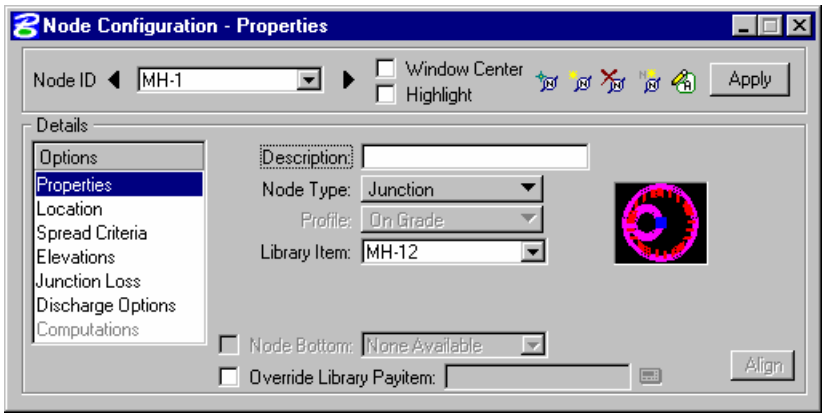
Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the Geopak Drainage pull down menu, and make *System1* the Active Network using **Network > Active Network**

Step 5. Add a manhole and incorporate it into the network. Select from the Drainage Menu: **Component > Node > Add**. Add Node ID: MH-1.



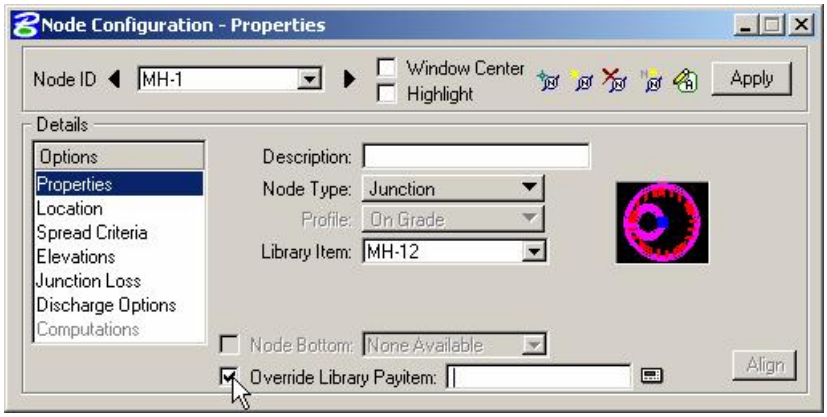
Step 6. Highlight **Properties.** Select the node to be used from the Drainage Library and assign its properties as depicted below:



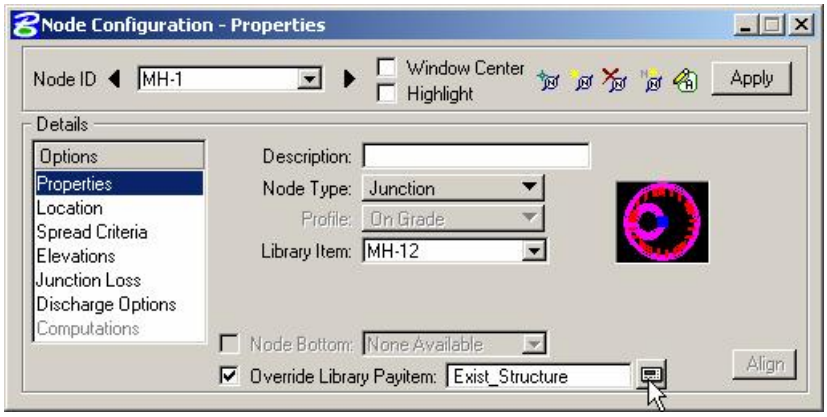
Node Type	Junction
Library Item	MH-12
Node Bottom	Toggle *OFF*

Note: This manhole is an existing drainage structure. Therefore, we have to override the payitem definition set in the VDOT drainage library.

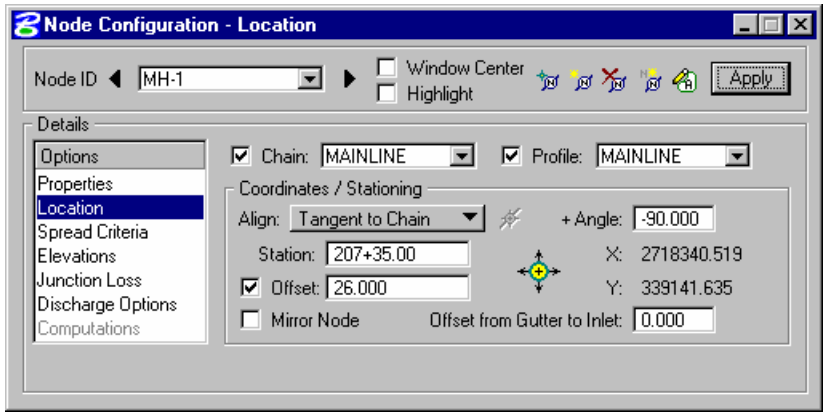
Step 7. Toggle **On** the Override Library Item box.



Step 8. Click on the calculator icon and select **Road Design > Drop Inlets > Exist_Structure** from the Design and Computation Manager.



Step 9. **Location >** Describe the manhole location in the design plane:

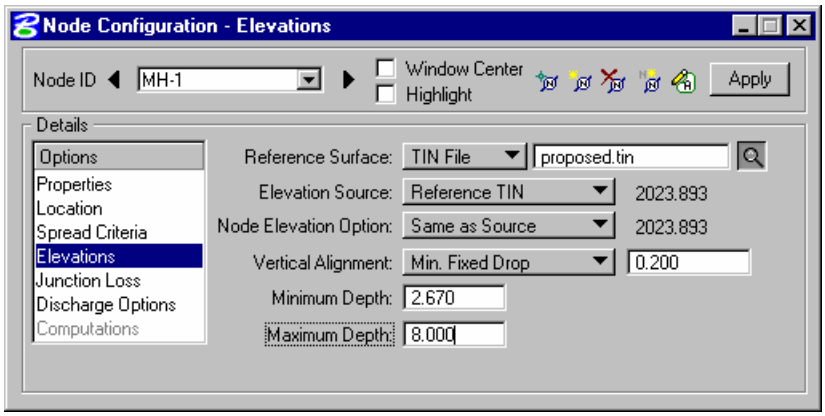


Reference Chain	**Select** Reference chain: MAINLINE
Reference PGL	**Select** Reference PGL: MAINLINE
Align	Tangent to Chain
Station	207+35

Offset	26.00 feet
Angle	-90

Press the **Enter** key to automatically place the inlet at the specified location.

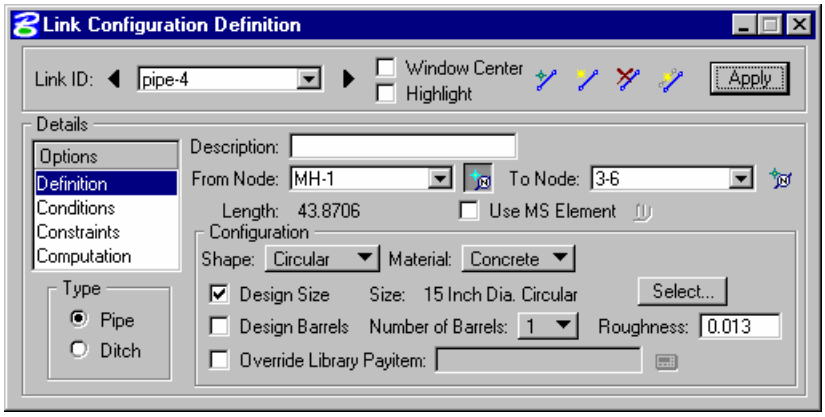
Step 10. Set the top elevation of the manhole. We will use the **proposed.tin** file to calculate the manhole elevation.



Step 11. Press **Apply** to add the manhole to the project.

Step 12. Select from the Drainage Menu: **Component > Link > Edit**.

Step 13. Select PIPE-4 and reconnect it from MH-1 to 3-6.



Step 14. Click **Apply**.

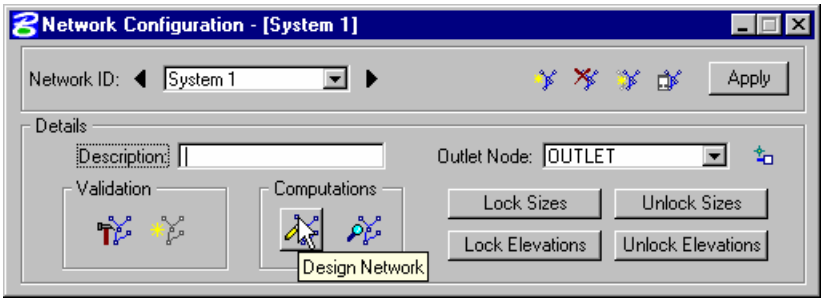
Step 15. Add a new pipe connecting 3-7 and MH-1. The new pipe will use the same conditions and constraints of the previously stored pipes.

System Design Update

Step 1. Select **Drainage > Network > Edit.** Edit *System1*

Step 2. Rebuild the Network and click **Apply** to update the Network.

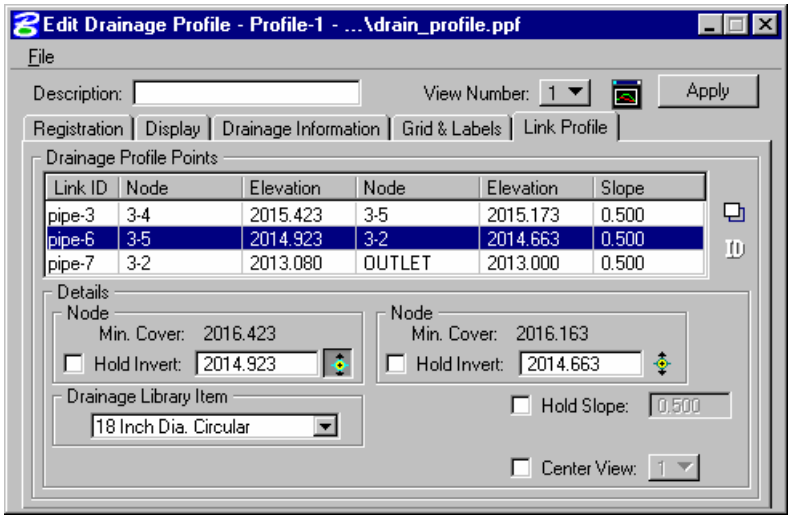
Step 3. Perform a system re-design.



Dynamically Editing Profiles

Step 1. Select the Edit Profile List tool from Drainage menu: **Component > Profile > Edit List**.

Step 2. Select the previously created profile: Profile-1.

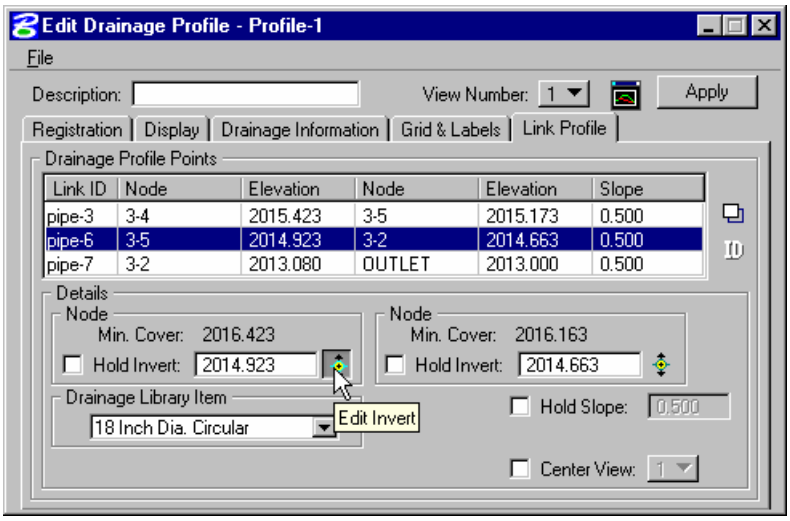


Step 3. Click the **Center Profile** icon, so you can visualize profile Profile-1 in View 1.

Step 4. Select the **Link Profile** tab.

Step 5. Click **ID.** Select and accept the link you want to modify from the profile view (pipe-6). This will make pipe-6 the active link.

Step 6. In the Details section, click **Edit Invert** in the left **Node** group.



Step 7. Move your cursor into the profile view. There is a circle around the downstream invert at node 3-5.

Step 8. Move the cursor into the circle and drag the invert down. Link pipe-6 will dynamically follow your cursor.

Step 9. Data point to define a new invert location in the Profile.

Step 10. Click **Apply**.

Step 11. Click **NO** to the Alert dialog box.

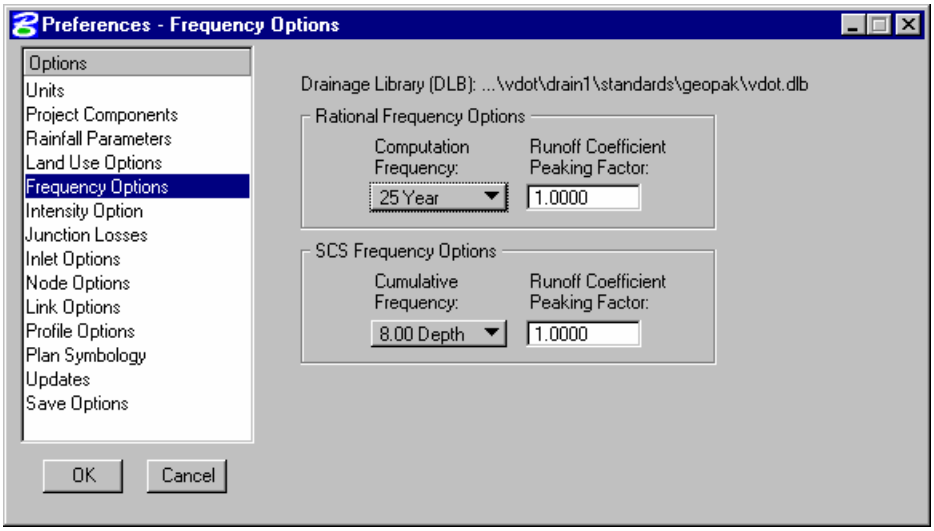
Note: GEOPAK Drainage redesigns your system aligning the pipes following the Vertical Alignment options (Match Soffit, Match Inverts, Allow Drop Manhole, etc.) of your Node configuration.

Step 12. Close the **Edit Drainage Profile** dialog.

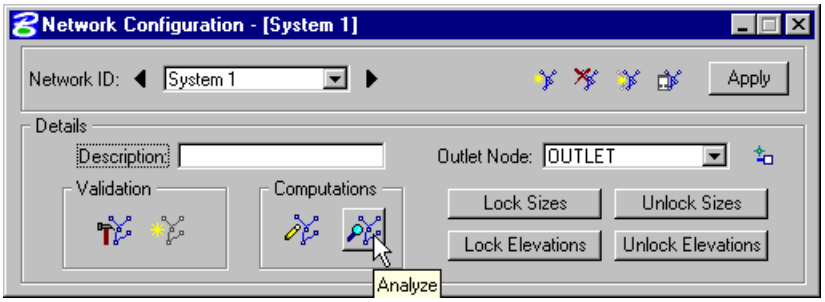
System Analysis

Step 1. Once the system has been designed, pipes sizes and invert elevations are set. If we need to run the system thru a different storm for checking or analysis purposes, change the frequency in the **Drainage > Project > Preferences.**

Step 2. Select the 25 year storm under **Frequency Option**s. Press **OK.**



Step 3. Select **Drainage > Network > Analyze**. GEOPAK will run the storm to the preset pipe sizes and elevations.



Step 4. Review the changes in the Hydraulic Grade Line in Profile-1 and change the Frequency back to the 10 year storm.

Step 5. Exit MicroStation.

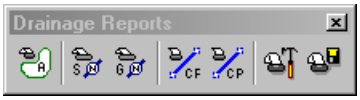
12 Reports

Objective:	Review Report features.

Introduction

GEOPAK Drainage provides a number of tools to create and generate both custom reports and quick, standardized reports. The standardized reports for Drainage Areas and Inlets are updated dynamically as any of the components contained in the report are updated, so the impact of the changes can be reviewed on the entire system. The Storm Drains/Links report dynamically tracks the condition of the data and provides notification whether the data is current in the report. These summaries are dependent on the Network computations being performed successfully and whenever the Network is designed or analyzed, these report summaries are updated automatically.

All report functions are invoked by selecting *Drainage* > *Tool Boxes* > *Reports*, then identifying the desired tool from the tool box as depicted below.



Tools from left to right are:

- Drainage Area Report
- Drainage Node Report – Sump
- Drainage Node Report – On Grade
- Drainage Link Report – Configuration
- Drainage Link Report – Computations
- Drainage Report Builder
- Drainage Generate Report

Of these reports, Drainage Areas, Inlets, and Storm Drains / Links are standardized reports, while the Builder and Generate menu selections are utilized for the customized reporting capabilities. Both standardized reports and customized reports can be viewed and subsequently, hard copy output can be produced.

Several useful features are common in most of the standardized reports at the bottom of the dialog as depicted in the fragment below.

Network When initially activated, all components within the project are reported. The option is All Networks. By selecting Active Network, only those components in the active network are reported. Note in order to use this option, an Active Network must be defined prior to selecting the option. If an Active Network is not defined, no warning message is displayed and the list box is not updated.

Edit (/) To edit an element directly from the report summary, highlight the desired element in the list box, then press the Edit (/) button, invoking the specified dialog in an update mode. When the desired updates have been made, press the Update button which closes the dialog, returns to the report dialog and automatically updates the report with the edited information.

Highlight When the toggle is activated, double click on a line in the list box. The MicroStation highlight color is applied to the specified item in the list for easy visual reference. Note the screen maintains the relative window, i.e., it does not perform any Zoom In or Zoom Out commands. Note the Window Center and Highlight toggles may be active simultaneously.

Window Center When the toggle is activated, double click on a line in the list box. The highlighted item in the list is window centered on the screen. Note the screen maintains the relative window, i.e., it does not perform any Zoom In or Zoom Out commands. Note the Window Center and Highlight toggles may be active simultaneously.

ASCII File The data may be written to an ASCII file. Manually type in the optional path and name of the file, or utilize the Files button to select from Create Area Summary ASCII File dialog. Note the default extension is *.sum.

LAB 12: Reports

Create Customized Reports

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB12_V8.EXE**.

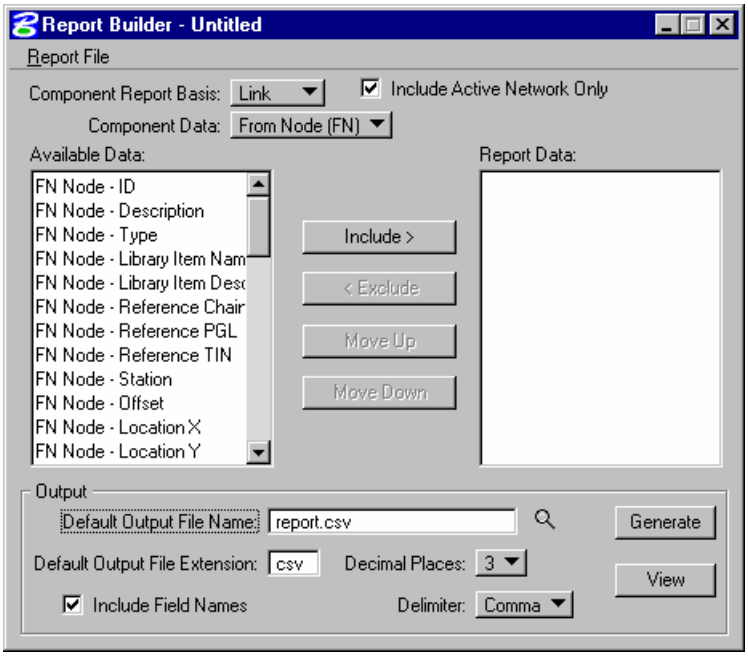
Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Drainage > Project > Open** from the MicroStation menu and make *System1* the Active Network using **Network > Active Network**

Step 5. Select from the Drainage Menu: **Reports > Builder**

Step 6. Set the Component Report Basis to *Link***.** Set the Component Data to *From Node:*



Step 7. Select the first few variables from the Available Data column and click the **Include** button to add them in the Report Data column.

Step 8. Type in the Default Output File Name and Extension.

Step 9. Select the number of Decimal Places: 3

Step 10. Select the Delimiter to *Comma*. This sets the output format to a comma separated values file. (Each variable will be separated by a comma for easy spreadsheet importation)

Step 11. Save this report format for future use from **Report File > Save As >** *: `test.drf`.*

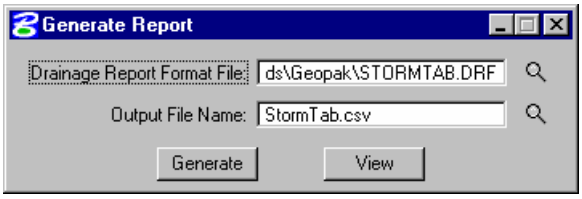
Step 12. The **Generate** button commences the report creation process.

Step 13. Alternately, the **View** button produces an on screen display of the data.

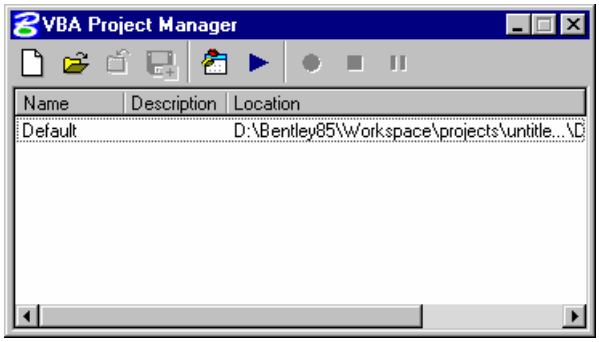
Create VDOT Storm Calculation Sheet

Step 1. Select from the Drainage Menu: **Reports > Generate.**

Step 2. Select the Drainage Report Format File: **Stormtab.drf** located in the …\standards\geopak directory. Press **Generate**.

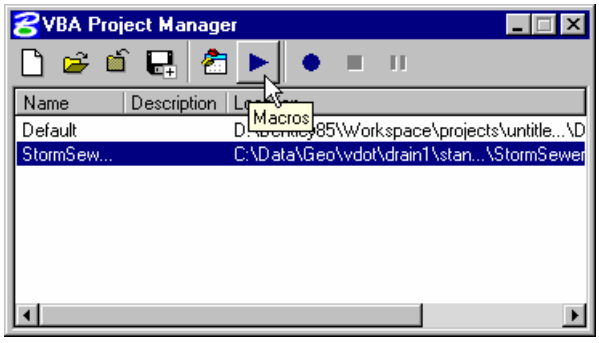


Step 3. Select **Utilities > Macro > Project Manager** from the MicroStation Menu.

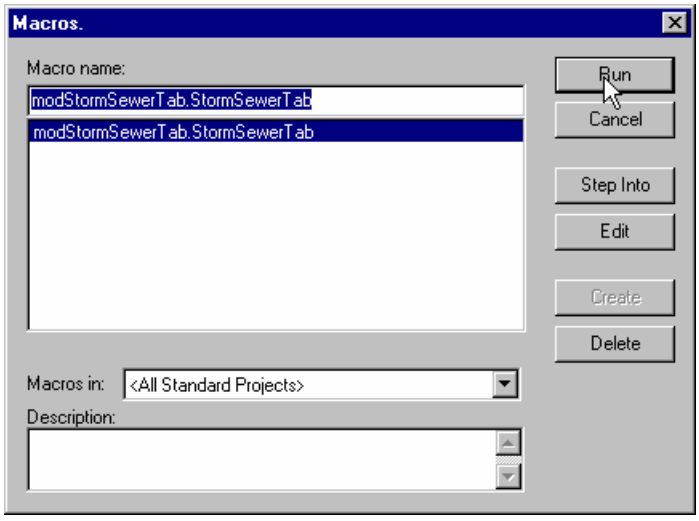


Step 4. Load the project: **StormSewerTab.mvba** located in the ….\standards\geopak\ directory.

Step 5. Select the previous loaded macro and press the **Macros** button.

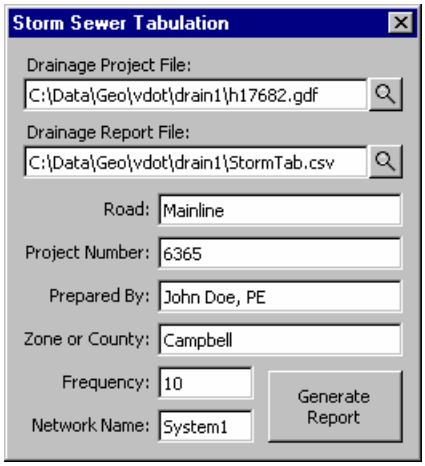


Step 6. Press the **Run** button.



Note: The previous steps could be replaced by typing the following command in the MicroStation key-in browser: "**vba load ; vba run**" and loading the StormSewer application.

Step 7. Set the drainage project and drainage report file in the Storm Sewer Tabulation dialog.



Step 8. Press **Generate Report**.

Step 9. Provide a name for the new storm tabulation Excel spreadsheet: **Storm.xls**

Step 10. Open Storm.xls with Microsoft Excel. Review the Stormtab and Spread worksheet.

STORM SEWER TABULATION FORM

Road: Mainline | Units: X Metric (SI) / US Customary | Sheet: _____ of
Project #: 8365 | Prepared by: A. M | Checked by:

LOCATION OF UPPER END			STRUCTURE NO.	NAME OF STRUCTURE	TYPE OF STRUCTURE	LENGTH (ft. or m.)	DRAINAGE AREA (ac. or ha.)			CUMULATIVE TIME OF CONCENTRATION (min)	TIME OF FLOW IN SECTION (min)	INTENSITY (in/hr or mm/hr)	TOTAL (C*A)	TOTAL RUNOFF (cfs or cms)	INLET ELEVATION (ft. or m.)	HYDRAULIC GRADELINE ELEVATION			DIAMETER (ft. or m.)	HYDRAULIC SLOPE (%)	UNIFORM VELOCITY (fps or mps)	ACTUAL CAPACITY (cfs or cms)
							BASE									CROWN ELEVATION						
CHAIN NAME							COMPOSITE									FLOWLINE ELEVATION						
STATION	OFFSET DISTANCE (ft. or m.)	SIDE	UPPER LOWER				C VALUE	AREA	SUB-TOTAL (C*A)							UPPER END (ft. or m.)	LOWER END (ft. or m.)	FALL (ft. or m)		ACTUAL SLOPE (%)	CRITICAL VELOCITY (fps or mps)	
MAINLINE			3-1	DI-3B 10	Curb	54.77	0.500	0.000	0.000	5.57	0.30	6.10	0.169	1.03	2019.99	2016.70	2016.22	0.48	1.25	0.956%	3.02	4.91
205+86.00	-26	Lt.	3-2				0.806	0.210	0.169							2017.32	2017.07	0.25		0.50%	3.08	
																2016.07	2015.82	0.25				
MAINLINE			3-3	DI-3B 8	Curb	54.77	0.500	0.000	0.000	5.57	0.23	6.10	0.061	0.37	2021.01	2017.72	2016.02	1.70	1.25	3.390%	3.92	10.9
206+86.00	-26	Lt.	3-2				0.607	0.100	0.061							2018.34	2017.11	1.23		2.47%	2.35	
																2017.09	2015.86	1.23				
MAINLINE			3-4	DI-3B 16	Curb	54.77	0.500	0.801	0.400	5.37	0.22	6.10	0.568	3.47	2019.34	2016.57	2016.08	0.49	1.25	0.976%	4.09	4.91
205+86.00	26	Rt.	3-5				0.563	1.009	0.568							2016.67	2016.42	0.25		0.50%	4.50	
																2015.42	2015.17	0.25				
MAINLINE			MH-1	MH-12	Junction	51.77				5.24	0.10	6.10	0.336	2.05	2023.89	2020.62	2017.40	3.21	1.25	6.837%	8.73	16.5
207+35.00	26	Rt.	3-6				0.000	0.000	0.000							2021.02	2018.34	2.68		5.70%	3.75	
																2019.77	2017.09	2.68				
MAINLINE			3-6	DI-3B 8	Curb	54.77	0.500	0.000	0.000	5.37	0.13	6.10	0.500	3.05	2021.01	2017.69	2016.36	1.34	1.25	2.674%	6.78	9.99
206+86.00	26	Rt.	3-5				0.860	0.190	0.164							2018.14	2017.11	1.03		2.07%	4.29	
																2016.89	2015.86	1.03				
MAINLINE			3-5	DI-3C 6	Curb	56.00	0.500	0.000	0.000	5.57	0.19	6.10	1.143	6.97	2019.78	2016.08	2015.69	0.40	1.5	0.765%	4.82	7.9
206+36.00	26	Rt.	3-2				0.825	0.090	0.074							2016.42	2016.16	0.26		0.50%	5.43	
																2014.92	2014.66	0.26				
MAINLINE			3-2	DI-3C 6	Curb	18.00	0.500	0.000	0.000	5.62	0.06	6.10	1.447	8.82	2019.78	2014.80	2014.75	0.05	1.75	0.306%	5.17	12.0
206+36.00	-26	Lt.	OUTLET				0.824	0.090	0.074							2014.83	2014.75	0.08		0.50%	5.52	
																2013.08	2013.00	0.08				
MAINLINE			3-7	DI-3B 16	Curb	67.75	0.500	0.000	0.000	5.14	0.14	6.10	0.336	2.05	2026.81	2023.72	2020.30	3.42	1.25	5.427%	8.10	14.9
208+00.00	26	Rt.	MH-1				0.849	0.396	0.336							2024.14	2021.22	2.91		4.63%	3.75	
																2022.89	2019.97	2.91				

Step 11. Exit MicroStation.

13 Plan View Labeling

Objective:	Review Plan View Labeling for drainage data

The Labeling automates the composition and placement of plan labels onto drawings. This interactive tool permits the creation of very simple to complex labels using many of the following features:

- Numerous Computed Text Inserts permit standardized computed values to be easily incorporated into labels. These standardized computed values are based on the type of element to be labeled and change as the user selects different elements. Types of elements available for computed inserts include points, lines, arcs, complex graphic elements, COGO features, and drainage features.
- User Text Inserts facilitate the simple inclusion of user defined text strings into the label. These User Inserts can be customized to include frequently used labeling terms and be saved for continual use.
- Standard shapes can be placed around labels and to accommodate different plan label formats.
- Standard leader lines and leader line terminators can also be utilized to create the desired label formats.
- Frequently utilized labels can be stored as Label Styles for subsequent recall. The complete label, including computed text inserts, user inserts, shapes, and leaders, are all stored within the Style.
- Tools to edit, move, and extract labels are also provided to make manipulation and plan changes easier.
- Enhanced ASCII is supported by use of the (\) symbol.
- Styles are supported, whereby often used labels and associated settings can be saved off and recalled in subsequent sessions.
- Updating options provide for efficient changing as the project dictates changes in the design.
-

While the labeler can be used for generic purposes, several of the fields are set up for specific applications. The specialty modes include:

- Plan View
- Profile
- Cross Section
- Drainage
- Site
- Water and Sewer

Under each mode, the types of elements available to label and the associated text inserts change. Other than this change, the remaining functionality operates exactly the same under each mode.

LAB 13: Plan View Labeling

Plan View Labeling

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB13_V8.EXE**.

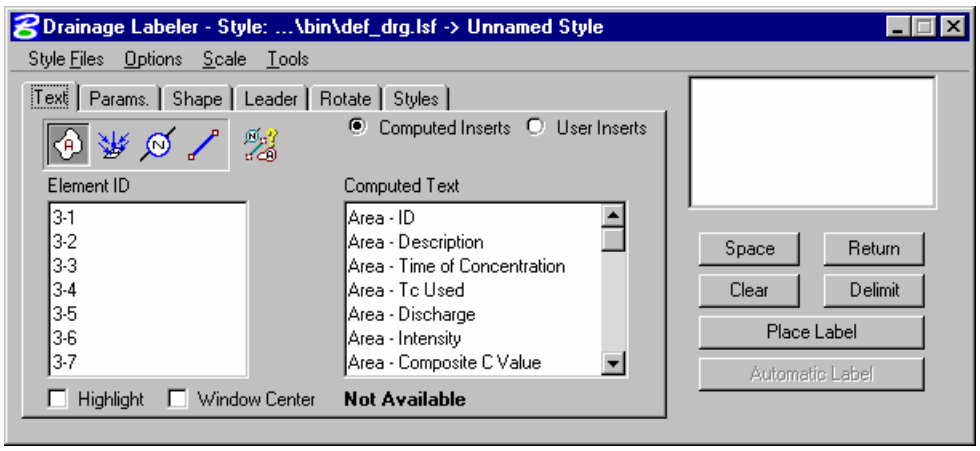
Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the GEOPAK Drainage pull down menu and make *System1* the Active Network using **Network > Active Network.**

Step 5. To open the Drainage Labeler, select from the Drainage down menu, **Utilities > Labeler:**

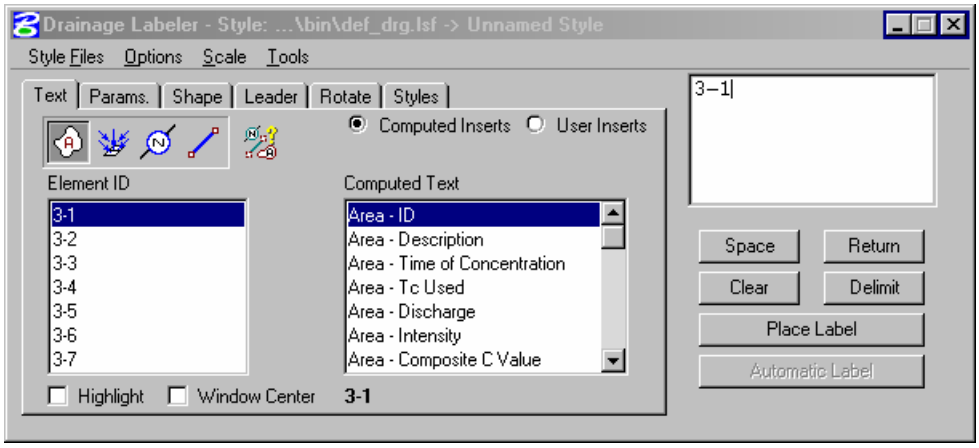
Step 6. Select the **Text Tab** and set the *Component* to "Area":



Step 7. Select the first *Element ID:* "*3-1*"

Step 8. Press the Computed Text: "Area – ID" to determine the Value. Double click the Computed Text: Area – ID to add it to the current label.

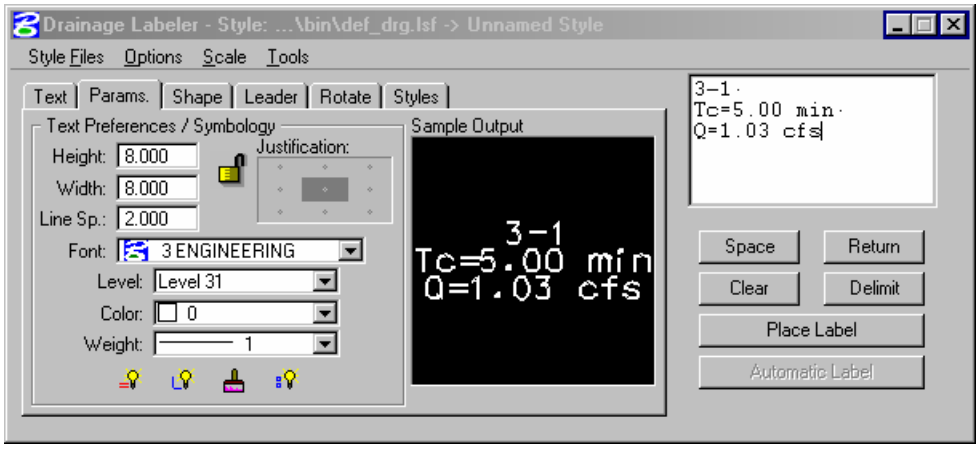
Step 9. Press **Return** to forward to the next line of label text.



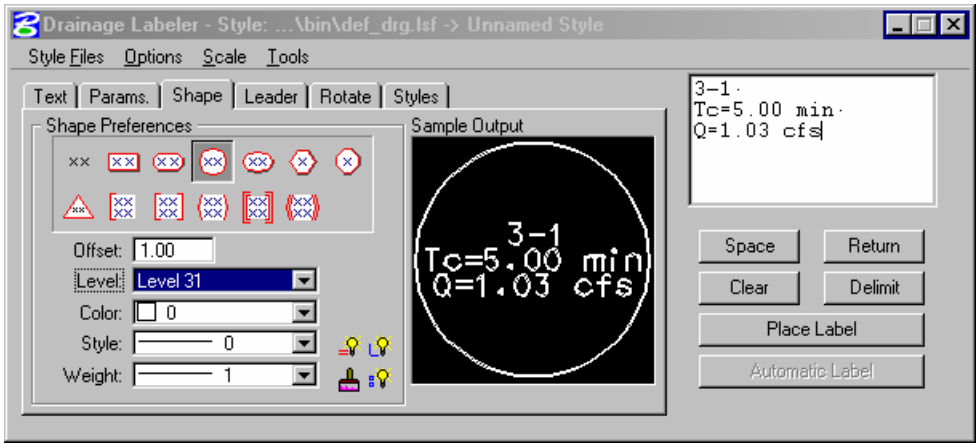
Step 10. Use the following *Computed Text* variables for the Area Label:

Area ID:	3-1; automatic text
Time of Concentration:	Type the prefix and suffix: **Tc** = **min**
Discharge:	Type the prefix and suffix **Q** = **cfs**

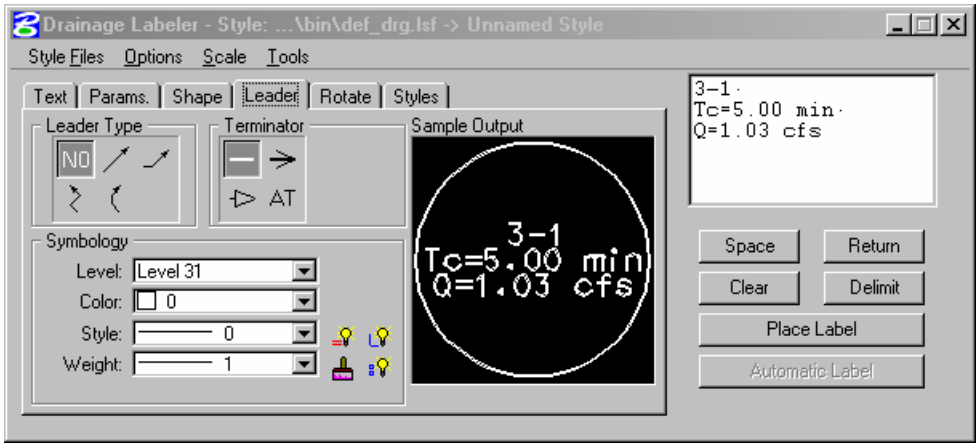
Step 11. Move to the **Params Tab** and set the *Text Preferences/Symbology* as shown below:



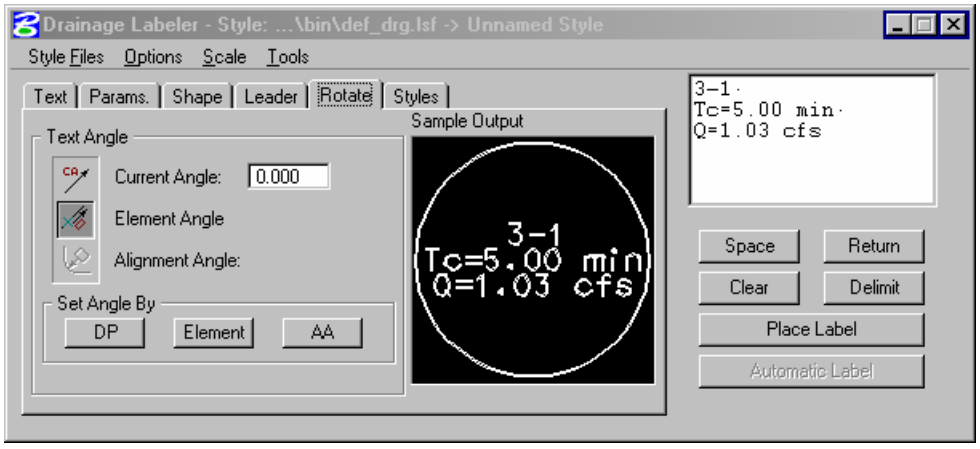
Step 12. Move to the **Shape tab** and set the *Shape Preferences* and *Symbology* as shown below:



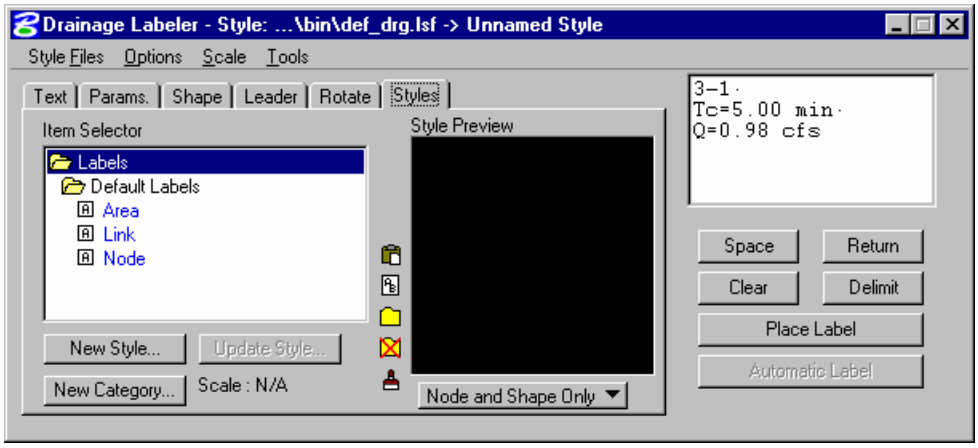
Step 13. Move to the **Leader tab** and set the *Leader Type, Terminator*, and *Symbology* as shown below:



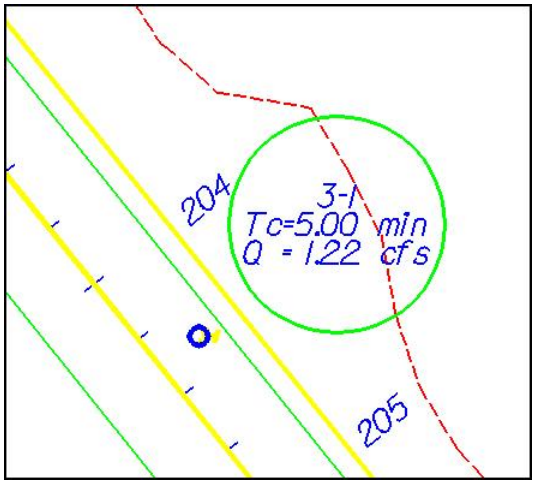
Step 14. Move to the **Rotate tab** and select the *Element Angle* option:



Step 15. Select the **Styles tab**. Initiate placement of the Area Label by pressing the **Place Label** button.



Step 16. Place the Label as shown below:



Step 17. Create a new Style by pressing **New Style**, provide a style name and press **OK.**

Step 18. Save the style by selecting **Style Files > Save**.

Step 19. Move to the *Text* tab and select the second Area: **3-2**. Note that all Computed text is updated for the new area. Place the remainder of the Area labels.

Step 20. Using the same procedures outlined above, place all the node labels.

Step 21. Repeat the process for the link labels and save the link style.

Step 22. Exit MicroStation.

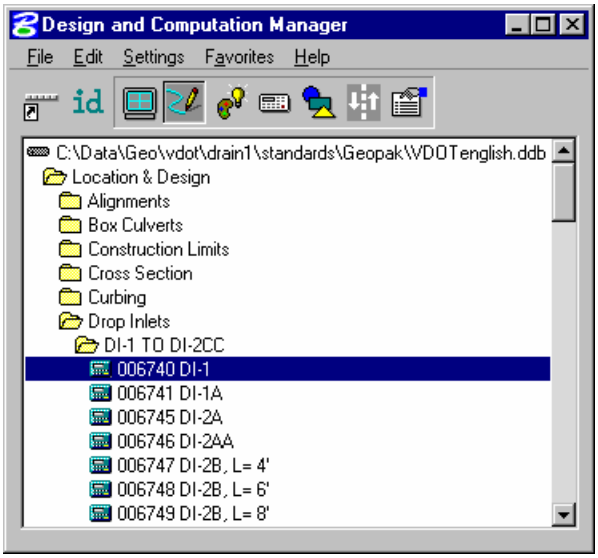
14 Automated Quantities

Objectives	Understand the computational modes of the D&C Manager for calculating automated quantities.

Introduction

The Design and Computation Manager (D&C Manager) is a tool that enables the user to standardize graphics elements for drafting and pay item quantities.

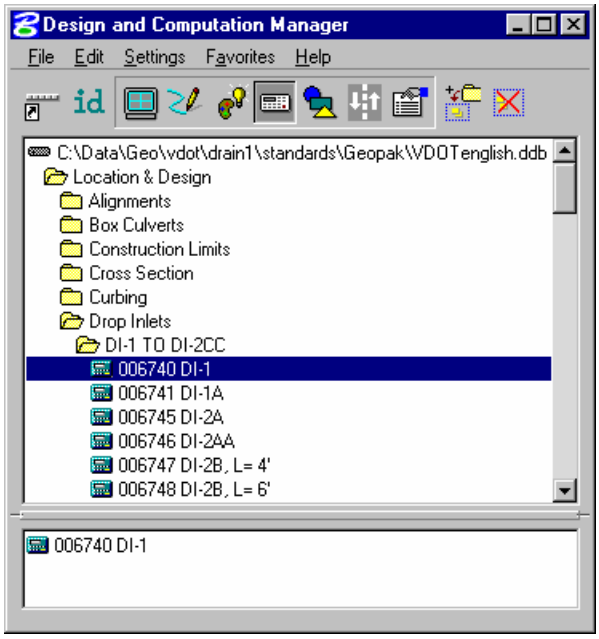
A D&C Manager Pay Item can be used for Node and Link items within the Drainage Library for use in performing quantity takeoffs and applying D&C symbology. Pressing the Select button within the specific Drainage Library Node or Link item invokes the Design and Computation Manager, in order to select the desired Pay Item.

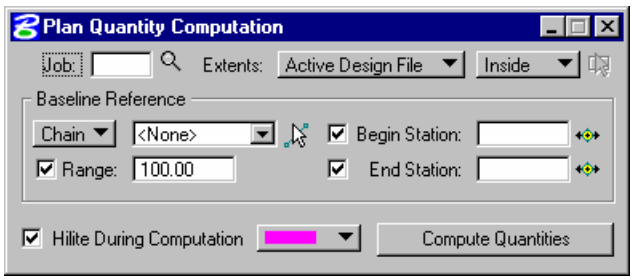


Operational Modes: Compute

Several operational modes are supported within the Design and Computation Manager; one of them is the Compute mode.

Through the Compute mode of the Design and Computation Manager, GEOPAK offers tools that make the completion of this task much quicker and easier. In addition, since the software utilizes the plan view construction drawings to compile quantities, discrepancies between the drawings and the tabulated quantities are non-existent. Chosen items within either the *View* or a *Fence* are calculated.





Once the MicroStation plan view design file is accessed, the first step is to invoke the Design and Computation Manager and click the shortcut button at the top of the dialog box to Compute.

The dialog box changes dynamically to reflect the compute operation.

The Design and Computation Manager dialog box is resizable. Within the dialog box, the sash (long, thin bar between list boxes) enables the user to adjust the partition between the list and collection boxes while the overall dialog box size remains constant. Simply place the cursor over the sash and dynamically move.

In addition to dynamically adjusting the main dialog box to reflect the compute operation, the auxiliary dialog box is invoked as depicted below. Note if the auxiliary dialog box was already invoked from another operation, it dynamically changes to reflect the Compute operation.

The next step is the selection of the correct item from the hierarchy. GEOPAK computes quantities only for those hierarchical items that are configured for the computation of quantities.

Two selection methods can be employed. The first method is to select an entire category. GEOPAK computes quantities for every item found in the selected category as well as any child categories. This is accomplished by double clicking through the groups until the desired category is contained in the Content list box. Single click onto the desired category and click the **Add to Collection** button at the top of the dialog box.

The second method involves the selection of individual features. This is accomplished by double clicking through the categories until the desired item appears in the hierarchy list box. When seen in the hierarchy list box, a single click onto the desired item and subsequent click of the **Add to Collection** button places the item in the Collection box.

Several output formats are available as described in the table below.

Item Report	Quantities Summary lists pay items, descriptions, units and total quantities for located elements. File is in ASCII format.
Comp Book	A more detailed report that lists not only quantity summaries, but also geometric properties such as plan view coordinates and station/offsets for located elements. File is in ASCII format.
Item Tables	Contains the same information as the Item Report, but formatted in tabular form.
CSV by Item	Quantities Summary lists pay items, descriptions, units and total quantities for located elements. Format is in CSV format.
SDF by Element	A more detailed report that lists not only quantity summaries, but also geometric properties such as plan view coordinates and station/offsets for located elements. Format is CSV format.
DBMS	Very detailed information including calculated and rounded quantities, geometric properties, pay item numbers, descriptions, station / offset values, etc. The format is the selected database (i.e., Microsoft Access, Oracle, SQL Server, and dbase.
Table	Places the report into the active design file.

An option button located on the left side of the Compute auxiliary dialog box dictates the Output. Options that apply to various Output types are detailed in the table below

View / Fence	If the **View** toggle is selected, only selected items that can be seen in MicroStation view one will be computed. If the view includes area outside of the **Range**, the **Range** will override. If the **Fence** toggle is selected, a fence must be placed, and all specified graphical features, which are inside both the fence, and the **Range** will be tabulated. The Fence mode is sensitive to the MicroStation Inside, Overlap and Clip modes.
File / Preview	If set to File, GEOPAK creates the file whose file format is based on the Output type and whose name is defined in the key-in field to the right of the File option. The name can be manually entered or selected via the Files (folder) icon. The file can be created or appended to an existing file. If desired, a full path may be specified, otherwise, the report will be found in the current directory. If the option is set to Preview, the file is not created and the output is displayed in a GEOPAK window.
Description	Any description keyed within this field will be printed at the top of the report. There is a maximum of 48 characters supported for this field.
Display	These short cut icons are functionally identical to the Display mode in the main dialog box and are discussed in the next section.
Highlight in Computation	When activated, all MicroStation elements utilized in computations are highlighted.
Compute	Commences the computation process.

Job Number	GEOPAK Coordinate geometry database (GPK) file wherein the specified chain is stored.
Chain	Requires a minimum of one chain, which must be stored in the specified job number.
Select	When selected, the Chain Selector dialog box appears, wherein the user can select the desired chain.
Range	Left and right offset limits (in master units) from the chain where the quantities will be computed. Any specified item located outside of the limits is not included in the report
Phase	Quantities can be separated into various phases, as a means to group various pay items. Default phases include: Design, Preliminary, and Final. Note the user can add new phases, but must be aware of naming limitations of downstream applications (i.e., Transport links in Quantities Manager.)
Run	The user may also runs as a method of grouping various quantities and / or pay items.

LAB 14: Automated Quantities

Updating Cadd Symbology

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB14_V8.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

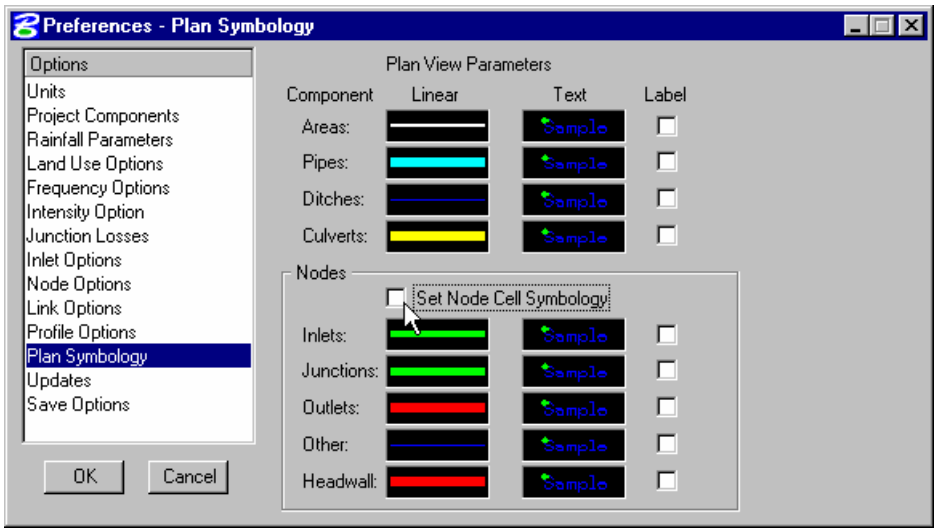
Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 4. Access the drainage project **h17682.gdf** and make *System 1* the Active Network using **Network > Active Network.**

Step 5. GEOPAK Drainage can track the component quantities within the Drainage project.

Select **Drainage > Project > Preferences > Plan Symbology**.

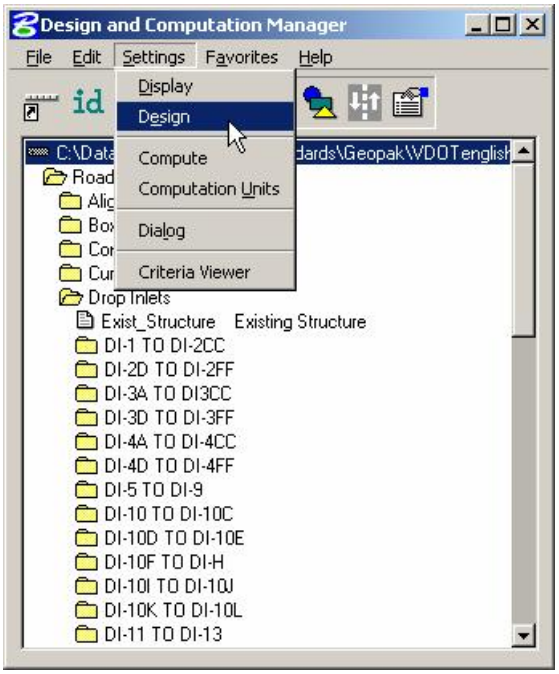
Disable the **Set Node Cell Symbology** and press **OK.**



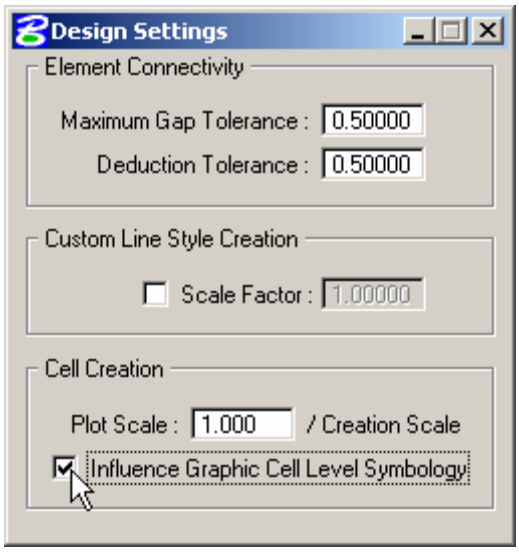
Step 6. Open the Design&Computation Manager from **Applications > Geopak Road > Design&Computation Manager**.

Step 7. In the Design&Computation Manager, do **File > Open** and load the **VDOTenglish.ddb** database.

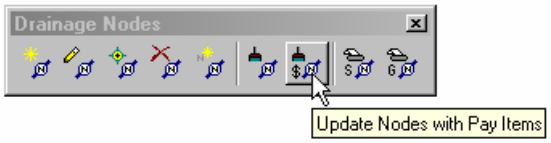
Step 8. Select **Settings > Design** in order to change the updating properties for the drainage structure cells.



Step 9. Toggle **ON** the Influence Graphic Cell Symbology option. Then, close the dialog box.

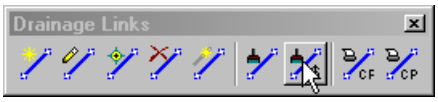


Step 10. Select from the Drainage menu*:* **Components > Nodes > Update With Pay Items**.



Note: Notice the existing manhole being change to a MicroStation Construction Class element.

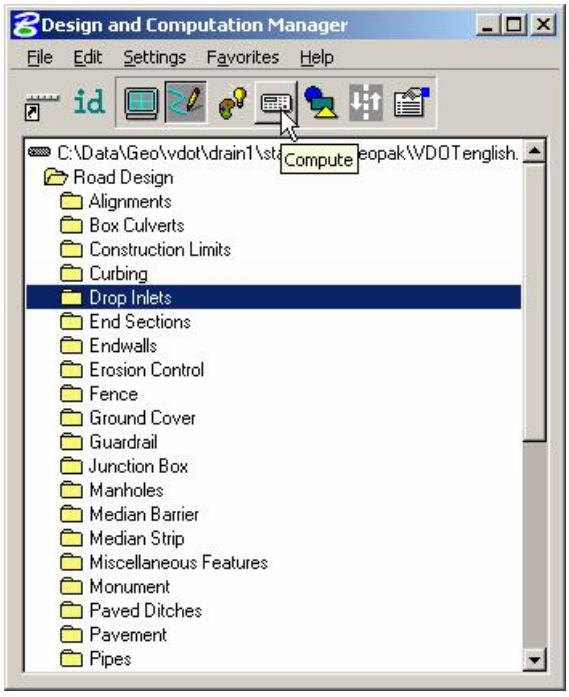
Step 11. Select from the Drainage menu*:* **Components > Link > Update With Pay Items**. GEOPAK will use the Design and Computation Manager ddb file to update the symbology of the drainage pipes.



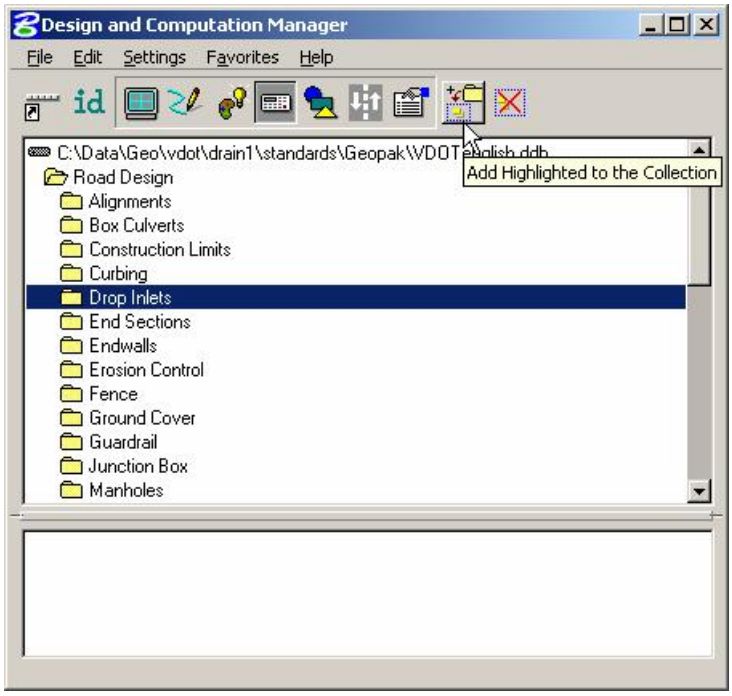
Automated Quantities

Step 1. Utilize the MicroStation command **Window Area** to view the entire Drainage Network in view 1.

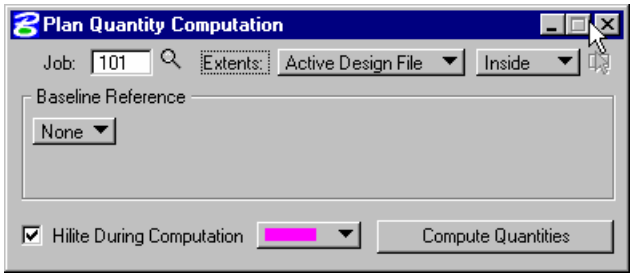
Step 2. In the Design and Computation Manager, set the Mode to **Compute** by pressing the 'calculator' icon button.



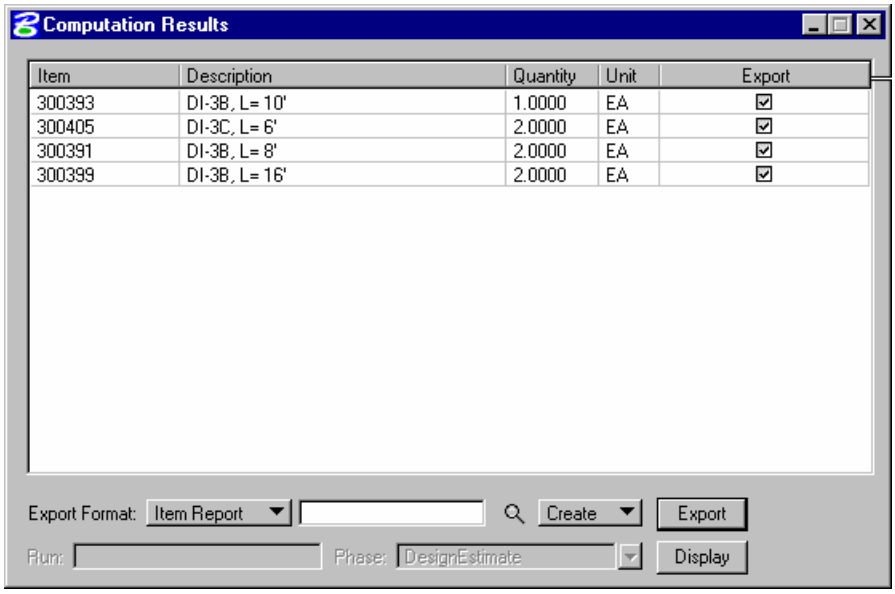
Step 3. Highlight the Drop Inlets Category under Location&Design, then press the **Add to Collection** button on the Design & Computation Manager toolbar.



Step 4. Set the Compute dialog as shown below.



Step 5. Press the **Compute Quantities** button to start the calculation.



Step 6. Exit MicroStation.

15 Culverts

Objective:	Review Culverts.
Goal:	Review the necessary data input required for the design of Culverts.

Introduction

GEOPAK Culvert is a powerful tool for the design and analysis of culvert structures. Located within GEOPAK Drainage, the Culvert tool can leverage data from a coordinate geometry database (GPK), a site database file (GSF), the drainage database file (GDF) or a binary TIN file. This enables the user to utilize roadway and / or storm sewer data, both existing and proposed.

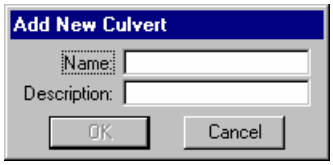
Invoking the Culvert Tool

The Culvert tool can be accessed by selecting *Component* > *Culvert* from the main Drainage menu.

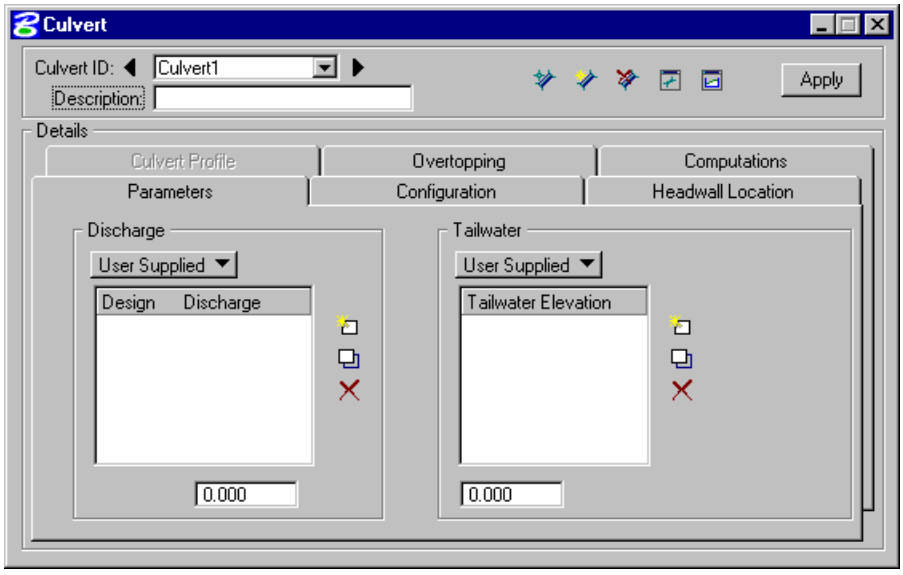
Previously stored culverts are displayed in the list box for easy selection and modification. Four icons are supported (from left to right):

- **Add Culvert** – invokes the add culvert dialog.
- **Modify Culvert** – invokes the culvert dialog populated with data for the specified culvert in the list box.
- **Identify Culvert** – when pressed and a previously stored culvert is selected on the screen, the culvert is displayed in the list box.
- **Delete Culvert** – deletes the culvert in the list box.

When the **Add Culvert** button is selected, the dialog depicted below is displayed. A prerequisite to storing culverts is a headwall must be stored within the Node options in the specified Drainage Library.



The Culvert dialog is used for adding, editing, or deleting Culverts.



The dialog consists of three fields at the top, and six tabs. The fields at the top remain regardless of which tab is selected. As each tab is selected, the dialog changes dynamically to reflect the selected tab. Only the culvert ID and description are constant while manipulating tabs. These tabs include:

- *Parameters*: Defines the design **Discharge** and **Tailwater** as well as the Discharges and Tailwater to be used for analysis.
- *Configuration*: Defines the properties of the culvert including the culvert shape, material, size, number of barrels, roughness coefficients, and entrance type.
- *Headwall Location*: Defines the location and reference information for the culvert.
- *Overtopping*: Defines parameters used in the calculation of overtopping flow.
- *Computations*: Provides for any combination of culvert, tailwater, and/or overtopping calculations and allows for output to an ASCII file.
- *Culvert Profile* - Not available in the current version.

The minimum requirements for storing a culvert are completion of the front three tabs: Parameters, Headwall Locations, and Configurations.

LAB 15: Culverts

Culvert Placement in Plan View

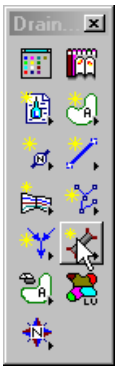
Step 1. Execute **c:\data\geo\VDOT\drain1\LAB15.EXE**.

Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

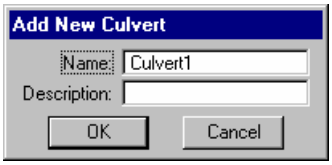
Step 4. Access the drainage project **h17682.gdf** by selecting **Drainage > Project > Open**.

Step 5. Select **Drainage > Tool Boxes > Main**. Drag the **Culvert icon** from the Drainage tool palette.

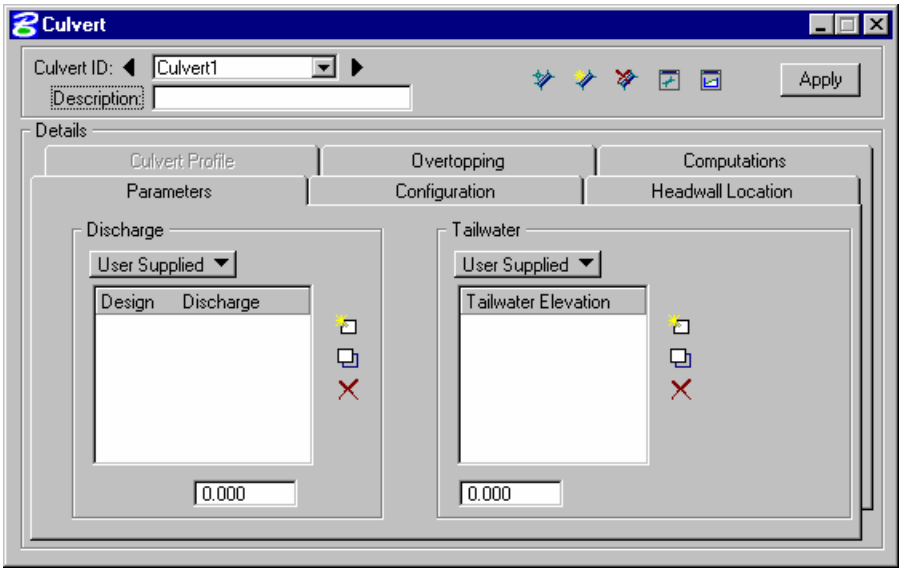


Step 6. The culvert toolbar will be invoked. Click the **Add Culvert** button.

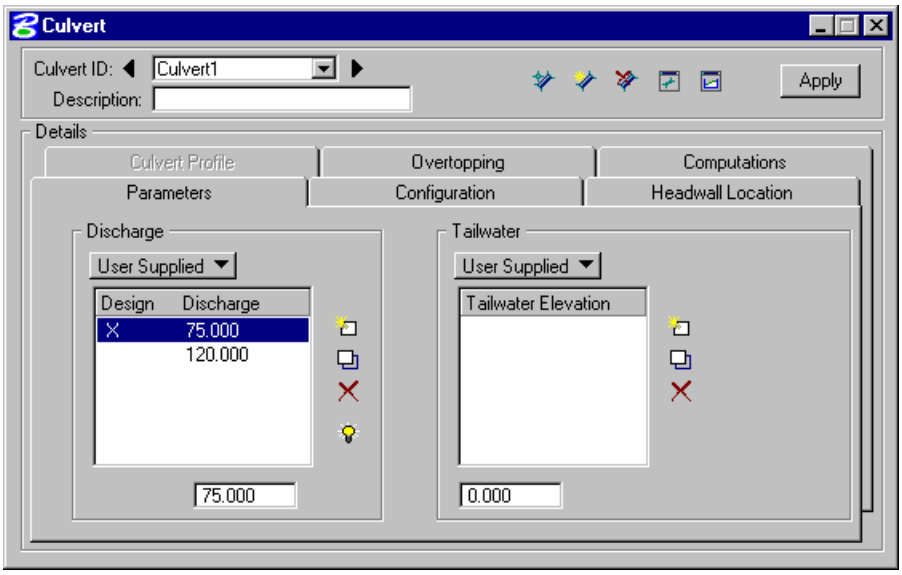
Step 7. Enter the Culvert Name as: Culvert 1 and Click **OK**.



Step 8. The *Culvert* dialog will open as seen below:

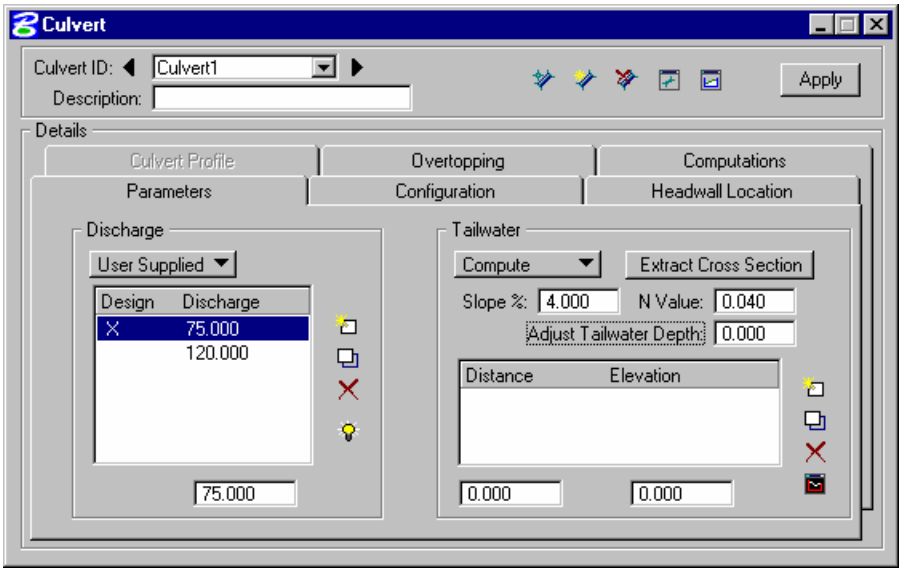


Step 9. Enter the culvert discharges.



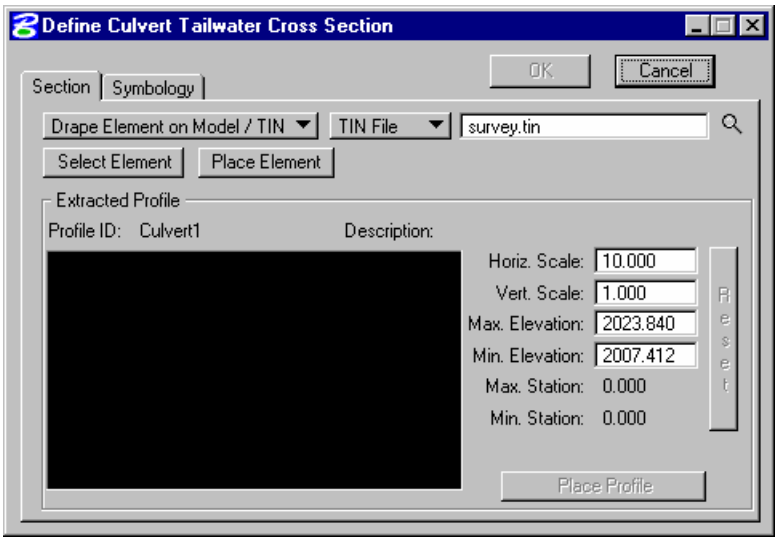
Discharge:	Keyin 75 cfs and click the **Add List Item** button. Keyin 120 cfs and click the **Add List Item** button.
Design Discharge:	Double click next to the 75 cfs within the design column.

Step 10. Define the *Tailwater*:

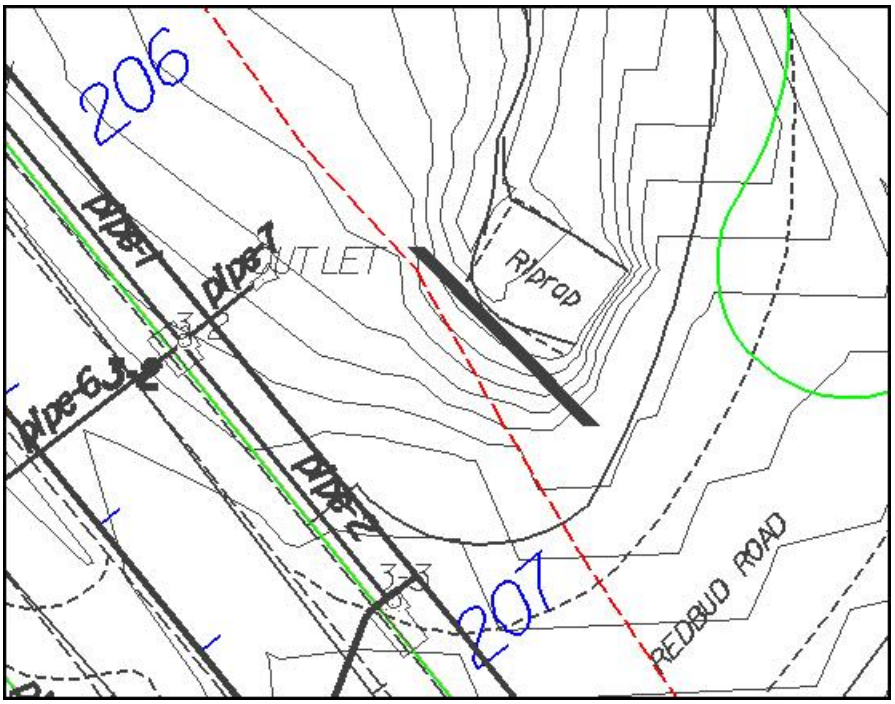


Slope:	4.0 %
N Value:	0.040

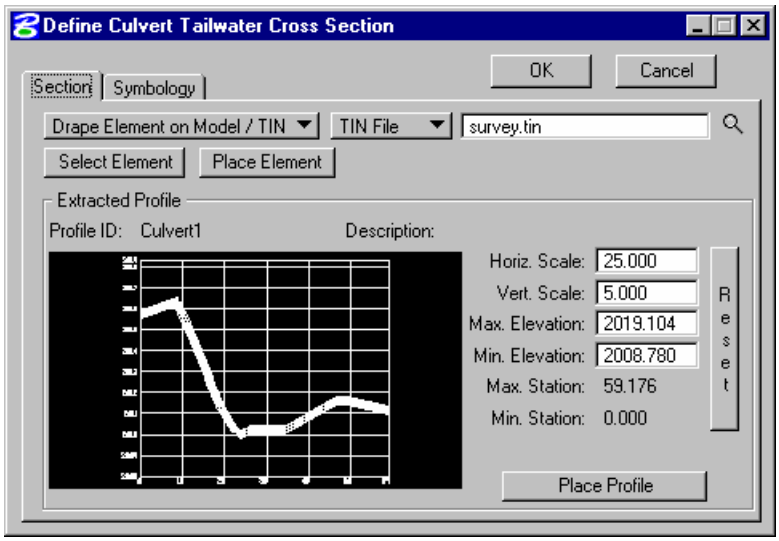
Step 11. Click the **Extract Cross Section** button. Set the *Drape Element on Model/TIN* and set to *TIN File.* Then select `survey.tin`



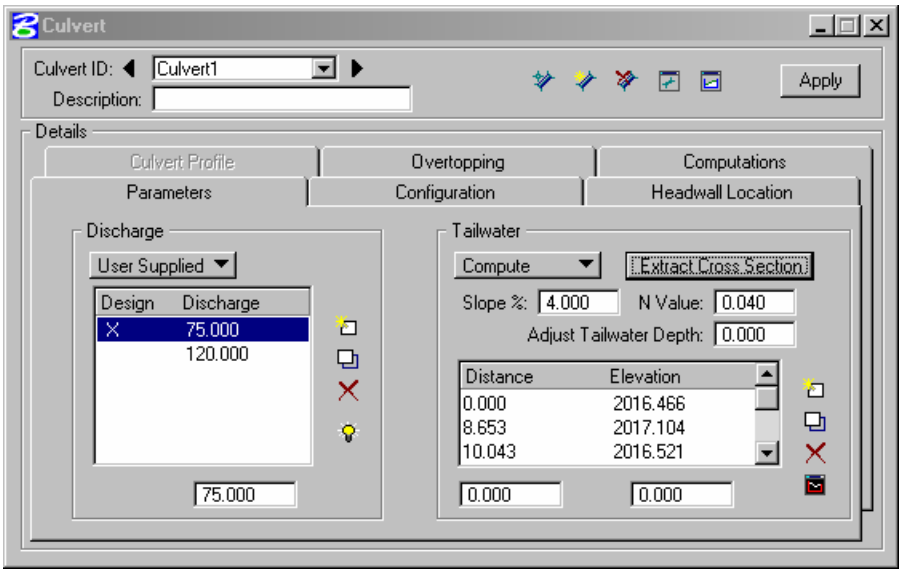
Step 12. Click the **Place Element** button to locate the postion of the tailwater profile to extract. Place a line in a location similar to that shown below:



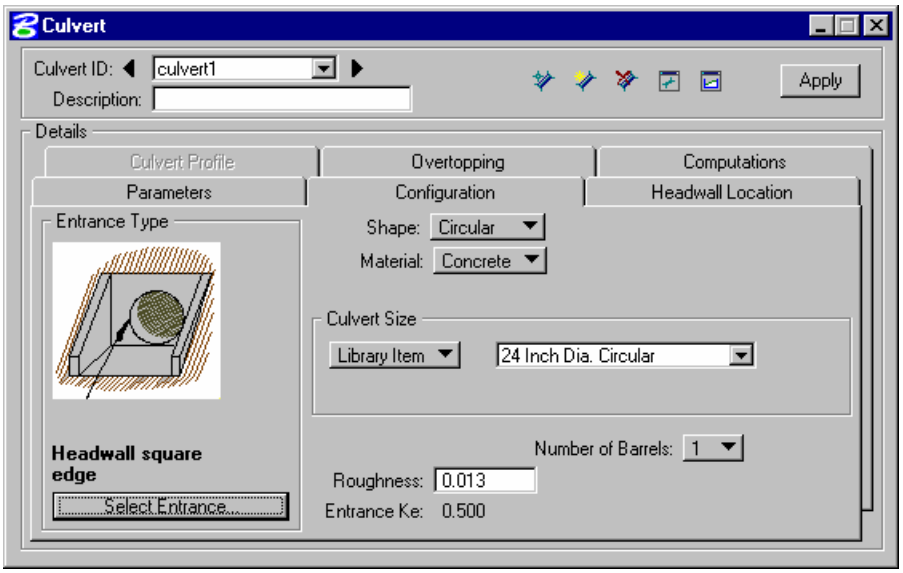
Step 13. The *Define Culvert Tailwater Cross Section* dialog will now contain the profile along the element placed representing the channel cross section at this location.



Step 14. Click the **OK** button and the main culvert dialog will open again. The coordinates of the profile will be populated.



Step 15. Activate the **Configuration tab** to define the type of *Culvert*.

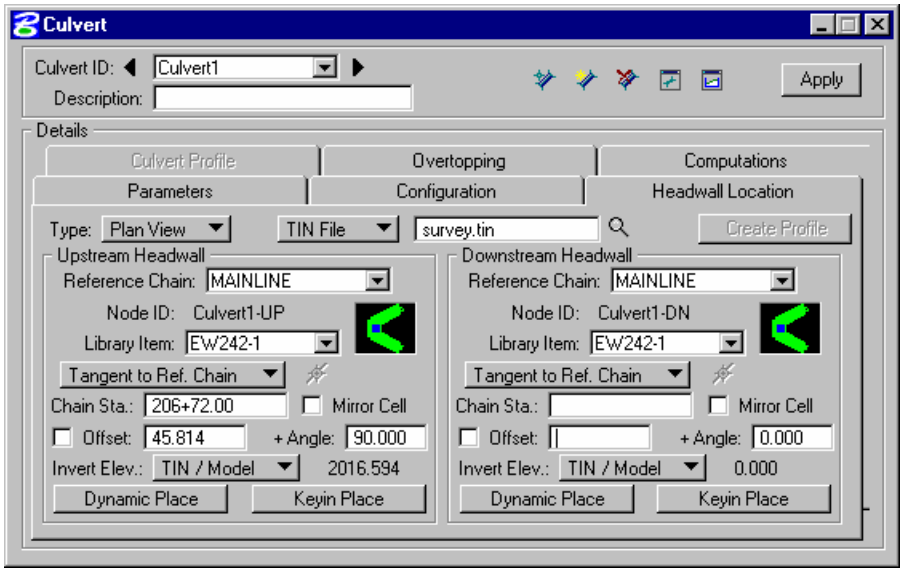


Shape:	Circular
Material:	Concrete
Library Item:	24 inch Dia. Circular
Manning's	0.013
Entrance Type:	Headwall Square Edge

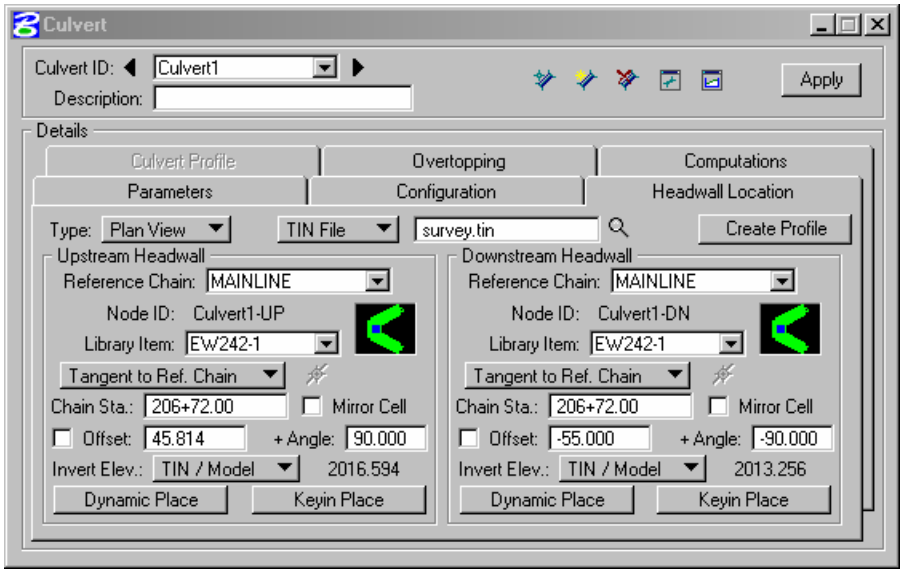
Step 16. Activate the **Headwall Location tab** to define the location of the upstream and downstream headwalls (nodes).

Type:	Plan View
Reference Chain:	Mainline
TIN File:	Survey.tin

Step 17. Locate the Upstream Node by clicking the **Dynamic Place** button and setting the upstream node at a location similar to that shown below:

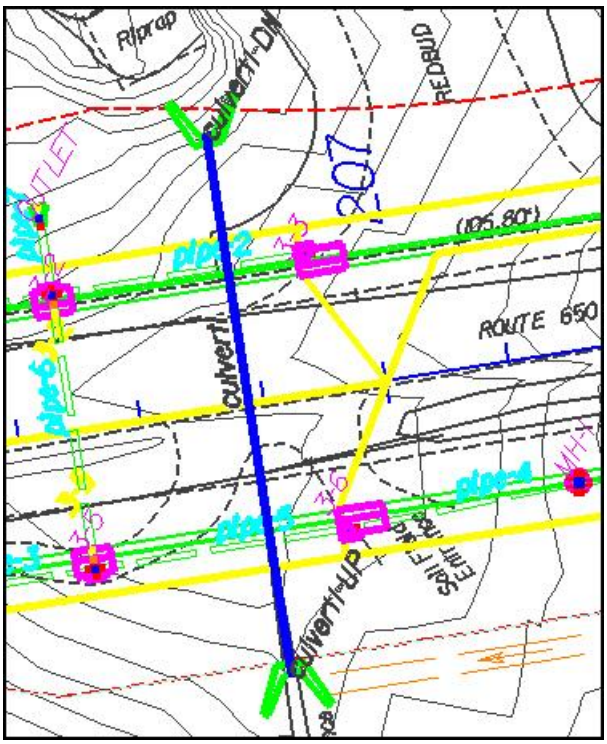


Step 18. Locate the Downstream Node by clicking **Dynamic Place** under the Downstream Node group.

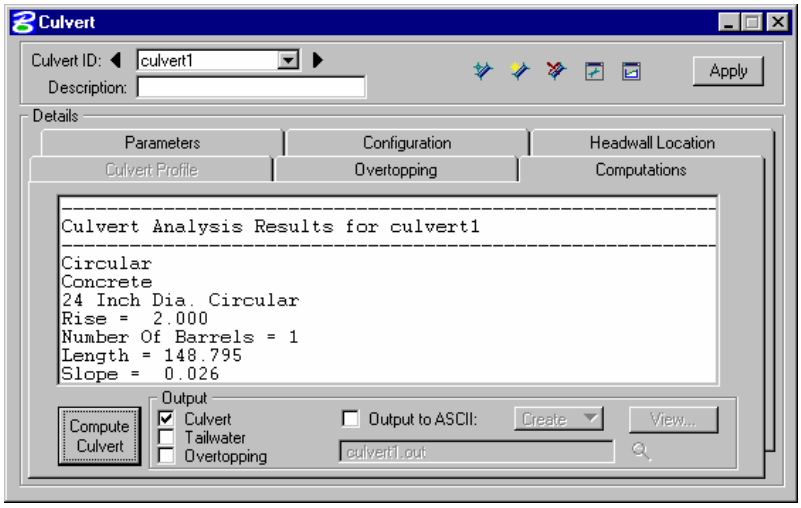


Step 19. Set the Elevations for both the Upstream and Downstream Nodes by toggling the Elevation source to *TIN/Model*. This will extract the elevations at the headwall locations with information from the TIN file.

Step 20. Once the Nodes have been located and the elevations set, the Culvert can be added to the project. Click the **Apply** button and the culvert will be drawn to the symbology in the Preferences and labeled.

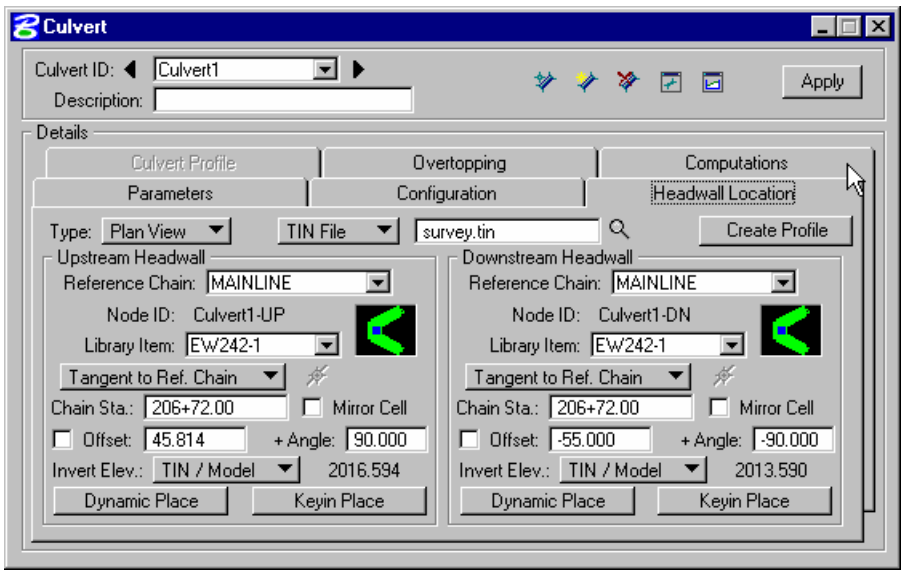


Step 21. The information to this point is enough to check the culvert computations. Activate the **Computations tab**. Toggle on the option to view the *Culvert* calculations. Click the **Compute Culvert** button to perform the calculations.

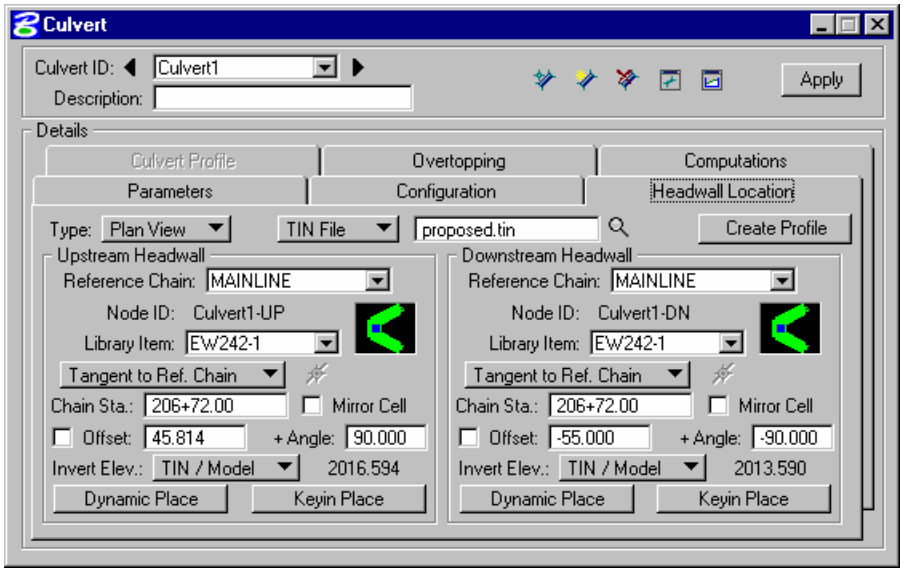


Culvert Adjustment in Profile View

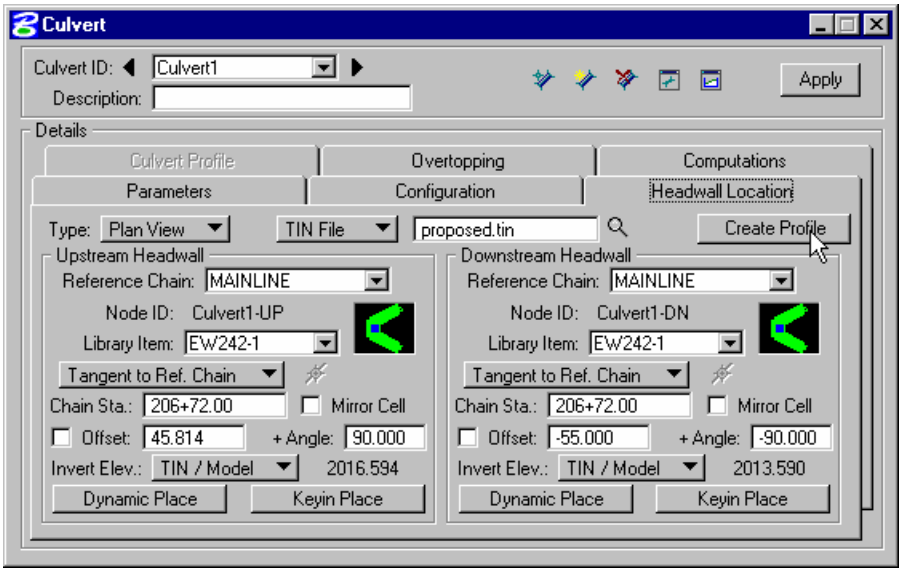
Step 1. Select the **Headwall Location tab**.



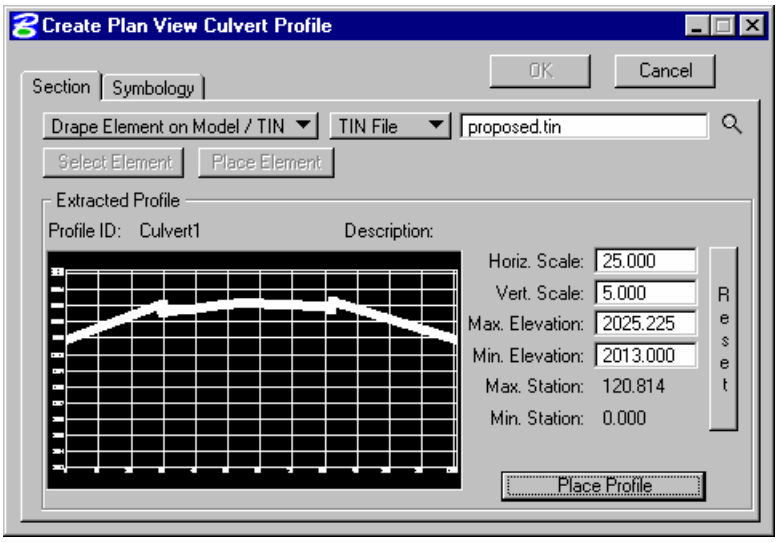
Step 2. Press the **Files** button and select : **Proposed.tin**. This is a proposed dtm file created by the Roadway Designers. It incorporates the existing ground data plus the designed cross sections of the project.



Step 3. Press the **Create Profile** button.



Step 4. Adjust the profile station, elevation and symbology.

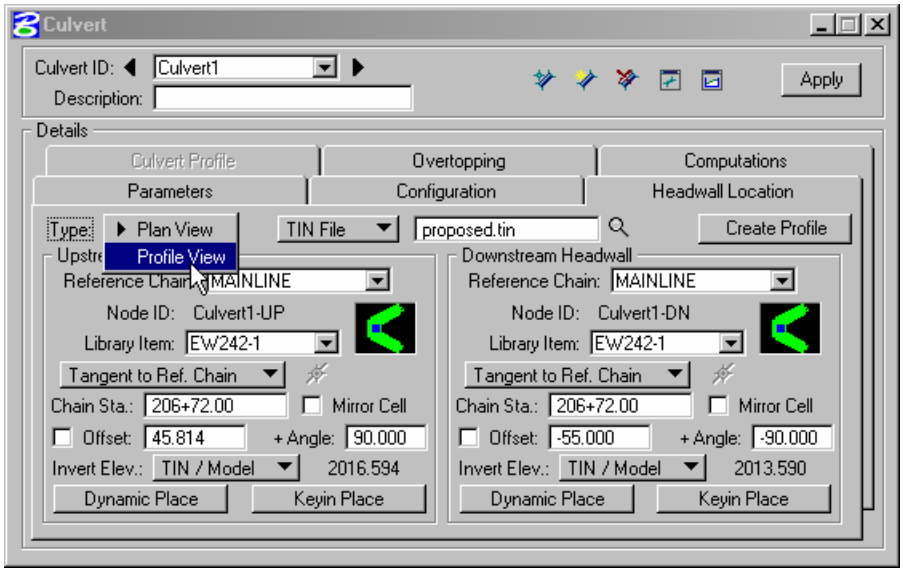


Step 5. Press the **Place Profile** button and place the profile in the dgn file.

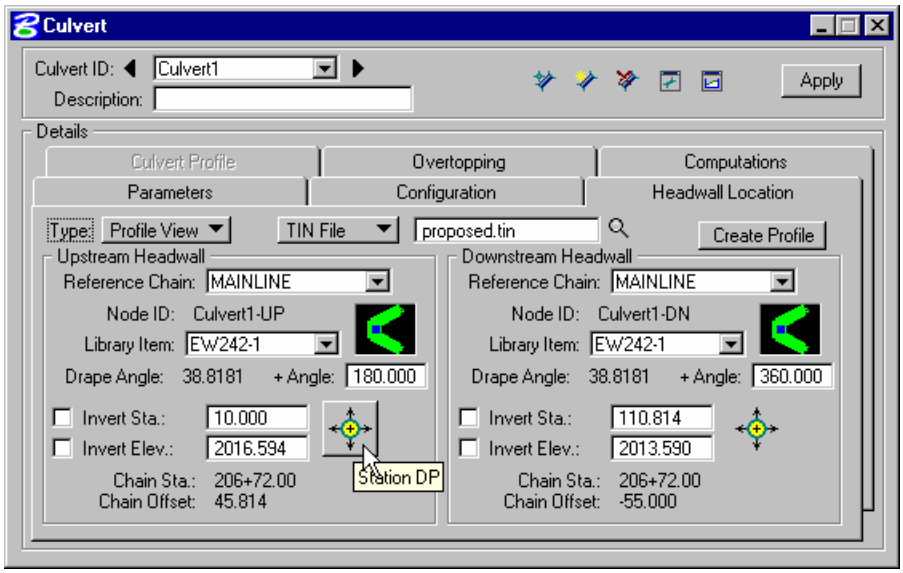
Step 6. Press **OK** when done.

Step 7. Press **Apply** to show the culvert in Profile View.

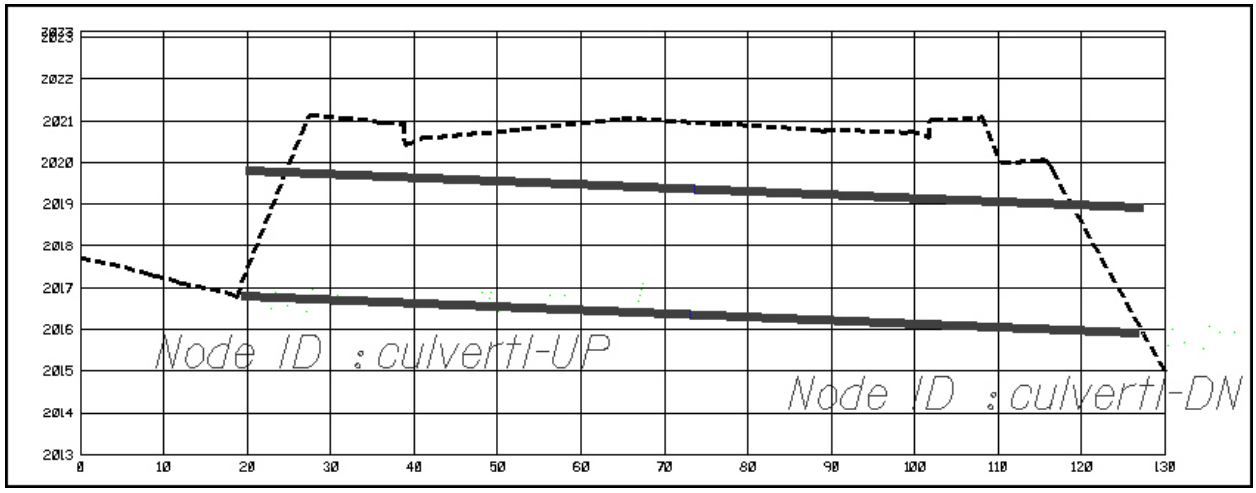
Step 8. Change the type of culvert placement to **Profile View**, and press OK at the prompt.



Step 9. Using the **Place** button adjust the location of the upstream and downstream headwall. The location of the headwall could be done dynamically by moving your cursor close to the circle marking the inverts of the culvert in profile view.



Step 10. When done press the **Apply** button to reset the location of the culvert in profile and plan view.



Step 11. Press the **Computations** tab and update the calculations.

Step 12. Exit MicroStation.

16 Special Ditch Profiles

Objectives	Review the concept of how to create a special ditch profile for proper drainage in GEOPAK and review the tools to automate this process.

Introduction

The creation of a special ditch profiles requires the usage of different tools within the GEOPAK software. The designer should be able to review the proposed cross sections provided, extract an existing ditch profile and then adjust it to create a proposed ditch profile.

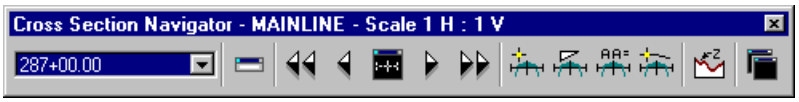
We will explain the major tools inside GEOPAK used to automate this process.

Cross Section Navigator

Project Manager	Not accessible from within Project Manager
Tools 	
Menu Bar Application	GEOPAK Road > Cross Sections > Navigator

The Cross Section Navigator tool is used to view and traverse between cross sections. It can also be used to draw cross section information.

When the Navigator is invoked, the following dialog box appears.



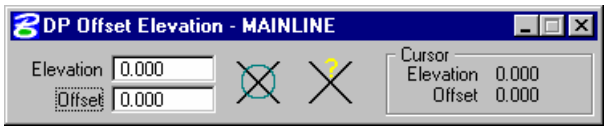
The user can scan through the cross sections by either choosing the station from the drop down list, or by using the First Section, Previous Section, Next Section, or Last Section icons (arrowheads). The Reset Navigator icon window centers the current station to the view.

The **Open View Control Dialog Box** enables the user to open several windows to view different portions of the cross section at the same time. The user can view the whole cross section in view 1, the left side in view 2, and the right side in view 3, etc.

Cross section elements can be added or modified using MicroStation tools or GEOPAK cross section drawing tools, as detailed in the table below.

DP Offset Elevation

Data points at a given offset/elevation, or find the offset/elevation of the cursor location.



DP Delta Distance Slope

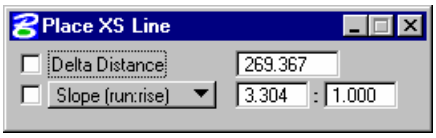
Draws a line at a given horizontal distance and slope.

XS Active Angle Tool

Sets the active angle to the given value. If a MicroStation tool is used with the active angle option, this value is used.

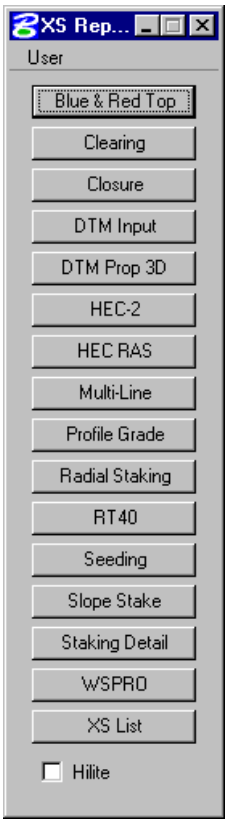
Draw XS Line

Draws a cross section line. The length and/or slope can be specified.



XS Reports

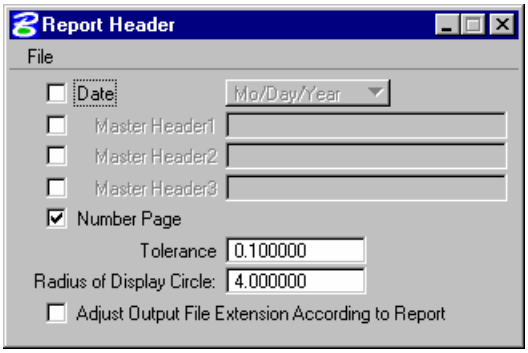
Objectives	Create various cross section reports: - for plan use (design elements and quantities) - as input for other programs and/or applications of GEOPAK - for construction layouts. Create and draw construction limits in the plan view file.
Project Manager	Reports and XS Quantities Limits of Construction
Tools  	



The GEOPAK Cross Section Report Utility can extract up to sixteen different reports from original and design cross-sections. For each report generated, the user must set the parameters of the existing and/or design cross sections. GEOPAK also provides an option to make custom headers for each of the reports via the **User** pull down menu.

Custom Header

From the XS Report dialog box, select **User > Preferences.** A Report Header dialog box will appear with all options ghosted out. To activate the individual fields simply toggle on the box next to the desired field. Once you have completed the dialog box, the information will be saved as an .hdr file. This allows for the creation of a separate header for each type of report. The tolerance field determines the maximum gap allowed between cross section elements.



Blue and Red Top

Based on the dialog box settings, GEOPAK determines the offset and elevation of a slope and its breakpoints. Blue refers to the top of pavement and Red is the top of subgrade. The user must determine this by indicating text and level, color, weight and style for each surface.

Clearing

The **Clearing Report** is useful for obtaining clearing and grubbing quantities. For each station, GEOPAK will list the clearing distance on each side of the chain and the width of any exception. You can obtain the results in the appropriate units. Toggle boxes for Cut Slope Rounding, Additional Clearing in Cut and Fill, and Minimum Clearing Width are provided for increased control over the output.

GEOPAK can also generate quantity sub-totals based on the value specified in **Sub Every**.

To use the **Except Width** option, you must have an existing ASCII file that includes the Beginning and Ending Station and Exception Width.

Once everything is set, you can output the information to an ASCII file

Closure

The **Closure Report** provides information on the intersection point between the user-defined proposed finish grade and existing ground. In addition to the ASCII report, the designer may instruct GEOPAK to close any gap either by drawing a vertical line between the endpoint of the proposed finish element and the existing ground or extending the slope of the last proposed element to intersect existing ground. The procedure will not extend existing ground. The **Closure Report** can be accessed within any MicroStation cross section file by selecting Closure from the main XS Reports dialog box.

DTM Input

This process generates XYZ coordinates from cross section elements and places this information into an ASCII file for use by the DTM portion of GEOPAK. To use this dialog box simply enter the .gpk job number, chain name and station range. GEOPAK will read the cross section elements based on level, weight, color and style.

DTM Proposed 3D

This report is similar to DTM Input except that you can set both original and proposed cross sections at the same time. This report also differs in that it makes break lines across the cross sections.

HEC – 2

This process reads cross section elements and formats the information in an ASCII text file suitable for use in the HEC-2 hydraulic program.

HEC RAS

This process reads cross section elements and formats the information in an ASCII text file suitable for use in the HEC RAS hydraulic program.

Multi-Line

This report is useful in creating cross-sections for staged construction. Begin by entering the job number, chain name and station limits. Primary cross section element parameters must be completed before secondary element parameters. This is important due to the order in which GEOPAK reads the information. Once all the parameters have been entered, the new cross sections may be drawn to the design file or you may choose the display only option. An ASCII text file will be generated.

Profile Grade

The **Profile Grade Repor**t is one of the most versatile reports available. It prints existing ground and design grade elevations and low point elevations for each cross section. Additionally, this report has the ability to search either for the low points or any text string that you specify and create horizontal and vertical alignments and store them directly into the .GPK. Horizontal alignments created from this report will have no curves.

Radial Staking

The **Radial Staking Report** is a specialized report created for the U.S. Federal Highway Administration (FHWA).

RT 40

The **RT 40 Report** produces RDS based RT40 data. To use this dialog box simply fill in the job chain name, stationing range and the parameters of the cross section elements you wish to use.

Seeding

Other than the usual entries, the user must enter the parameters of the elements to be seeded.

This dialog box includes slope and subtotal options as well as a way to limit the number of segments read (**By-Pass Segments**). The user may also establish additional seeding specifications (**Additional Distance**).

Once all of the settings are complete, the report will produce seed or sod quantities written to an ASCII output file for use in plan quantities.

Slope Stake

The **Slope Stake Report** is a special format report developed for the FHWA. This report generates offsets, elevations and superelevation information for each cross section. To generate this report fill in the usual cross section parameters plus Subgrade and Hub Staking information. When complete, push Apply and the report is written into an ASCII file.

Staking Detail

The **Staking Detail Report** determines the tie down point between the proposed finished grade and the existing ground. GEOPAK will list the right and left offset, elevation, and slope of the finish grade and superelevation rate for each cross section. To create this report, fill in the project information and desired cross section elements' parameters. Once complete, you have the choice between two formats, a FHWA ASCII report or a Montana DOT report (includes ditch elevations).

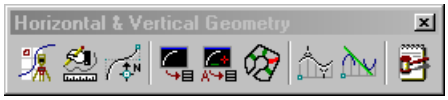
WSPRO

This report takes the cross section elements and turns them into an ASCII file for use as input in the WSPRO hydraulic analysis program.

XS List

This report creates a listing of elevations and offsets for each cross section element according to user-defined parameters. You have the option of creating either an original cross section list or a design cross section list. These reports are very similar to RDS cross section lists.

Vertical Alignment Generator

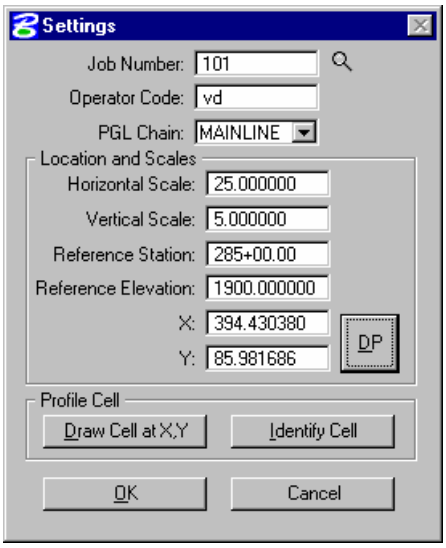
Objectives	To create and store vertical alignments using the **Vertical Alignment Generator**.
Project Manager	Vertical Alignment
Tools  	

GEOPAK has a variety of tools for generating and storing a proposed profile. Those tools include:

- The **Vertical Alignment Generator (VPI Based)** is a GEOPAK tool that can graphically create and modify proposed design profiles or modify an existing ground profile. The designer can generate a profile based on VPI data, design speeds, stopping sight distances, and K values. These operations may be accomplished through a dialog box and/or by dynamic manipulation of graphic elements.
- A profile may also be created with Coordinate Geometry (COGO) input (key in commands). In addition, there are VC commands that compute and display vertical geometry and the parameters associated to aid in the development and generation of vertical alignments used in conjunction with Store Profile commands. VC commands define parabolas for various combinations of stations, elevations, grades, slopes and the rate of vertical curvature (K value), and also define the tangents between two parabolas or between a parabola and a point.

Accessing

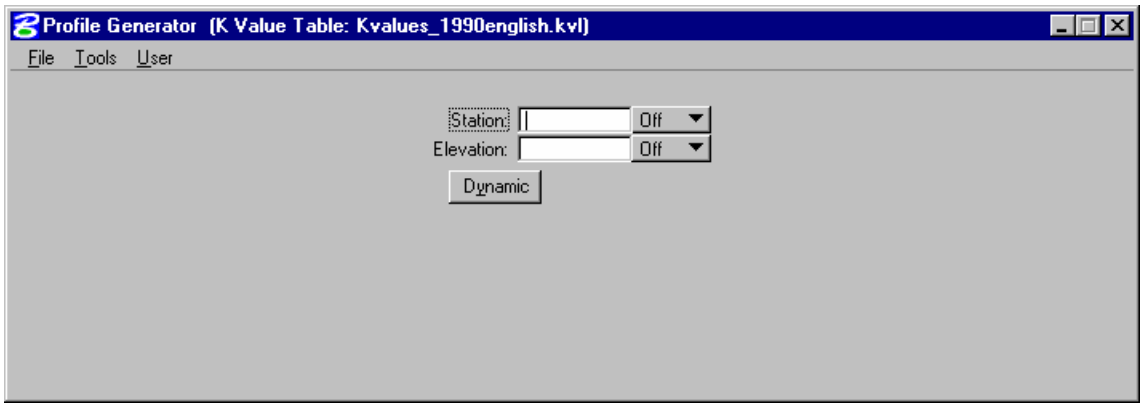
When selecting the Vertical Alignment generator, the first dialog box that appears is labeled **Settings**, as depicted here. The entries in this box set the parameters and define the location within the design file where the profile components are to be displayed. All fields must be completed before the design process can begin. Once the **OK** button is pushed, the Vertical Profile Generator dialog box appears.

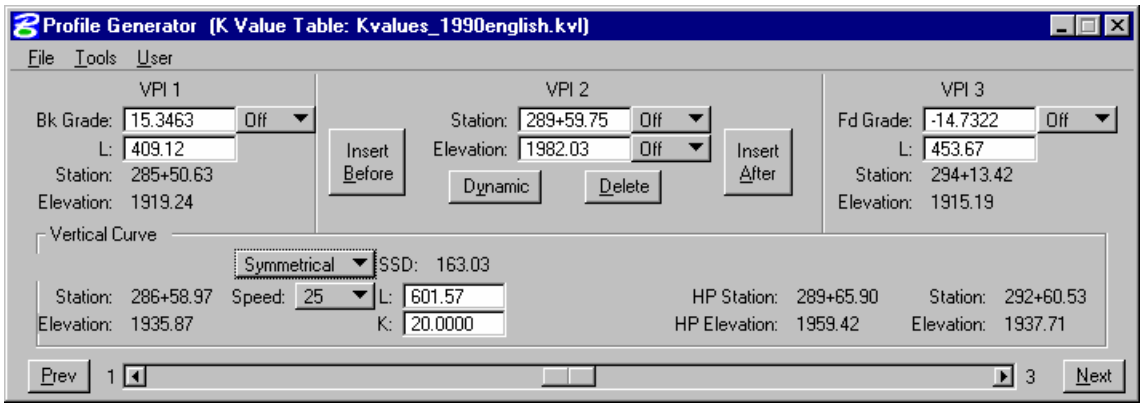


Note: This tool also utilizes the cell that was drawn in the previous exercise by using the **Identify Cell** button.

Using the Vertical Alignment Generator

This tool enables a user to load a previously stored profile or create a new profile. You will notice the dynamic changes in the dialog box throughout the vertical alignment design.





Various design parameters must be defined prior to designing a new profile; which are discussed in the options supported under the three menu options; **File**, **Tools**, and **User**.

File

Clear - clears the profile display from MicroStation graphics and removes all VPI's from the dialog box.

Draw - write the graphic elements of the profile to the MicroStation file.

Load - retrieves a previously stored profile from the coordinate geometry database (.GPK)

Save - stores a new profile or updates (redefines) a previously stored profile under the same name.

Save As - is used to store the profile or to save a modified profile under a different name.

Exit - ends the process.

Tools

Critical Points - Vertical curves may also be defined by one or two critical points – i.e. the curve will pass through these points If mathematically solvable, the vertical curve will be drawn and the design speed display adjusted to fit the current parameters.

Issue Data Point - Permits the user to type in stations and elevations, issue a data point that can be part of a MicroStation place line, place a cell or perform other generic operations. This is useful in displaying visual references within the profile that need to be considered in design of the vertical profile.

User

Preferences - sets the rounding parameters for each of the items listed in the dialog box. Also contains a toggle that will force the dialog box to window center the view on the current VPI.

K Values - is a table of stopping sight distance K-values for crest and sag conditions for various design speeds. These values are based on the AASHTO Green Book.

Settings - recalls the **Settings** dialog box that first appeared upon initializing Vertical Layout.

Creating A New Profile

Note: The enter/tab key must be used to ensure values are accepted.

A new profile can be created with the following steps:

Step 1. Place the first VPI using one of the four supported options.

- Type station and elevation of the VPI into the appropriate fields in the Profile Generator dialog box.
- Enter station of VPI as precision input (type in value), and allow the elevation to be defined through dynamic cursor placement on screen.

- Elevation is defined via precision input, and the station is defined through dynamic cursor placement on screen.

- Both values for the VPI can be established dynamically on screen by clicking on the **Dynamic** button and placing a DP in the view at the desired station and elevation.

Step 2. Define ahead (or back tangent)

Station, elevation, grade and length parameters may be defined via precision input, dynamic manipulation or a combination of both.

Step 3. Define remaining VPI's and Grades

A repetition of Step 2 with an option to insert VPI's between two existing VPI' s.

Step 4. Define Vertical Curves

Simply define the design speed from the Speed option button and GEOPAK will reference the K-value table and draw the vertical curve. If a *curve overlap* occurs, an overlap message will be displayed in the dialog box along with the overlap length. Length of Curve or K-value may also be keyed in.

Step 5. Adjusting Curve Lengths

The vertical curve can be modified by directly keying in either the K-value, curve length or defining the design speed in the dialog box. You will see the values computed automatically adjust to reflect the results of any modifications.

Step 6. Save the Profile

Click **File > Save As** to name and save the newly created profile. The names of profiles are any 1-9 alphanumeric characters.

Precision Placement Options

Options available for creating or modifying vertical curves, VPI's and grade lines:

- **OFF** - Values change.
- **INC** (Increment) - Ensures that the designated profile parameter will be adjusted as defined in the Preferences dialog box.
- **LCK** (Locked) - Forces selected operations to maintain the designated profile parameters.

LAB 16: Special Ditch Profiles

Reviewing the Roadway Cross Sections

Step 1. Execute **c:\data\geo\VDOT\drain1\LAB16_V8.EXE**.

Step 2. Open **c:\data\geo\vdot\drain1\d17682xsmainline.dgn**

Step 3. Select the Cross Section Navigator tool from **Applications > GEOPAK Road > Cross Sections > Navigator** or from the Road tools palette:

Step 4. Use the Cross Section Navigator to browse and check the elevations of the ditch at the right side of the alignment.

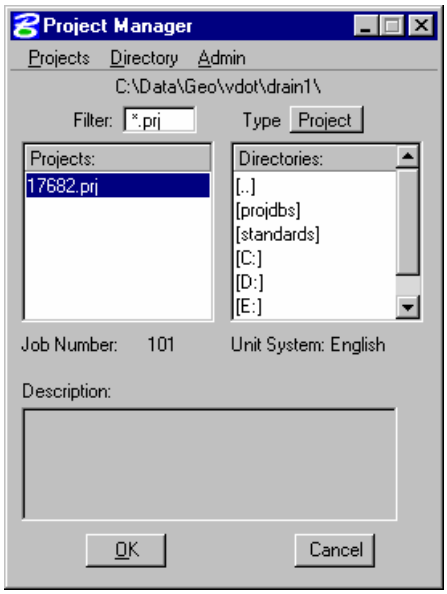
Step 5. Exit the Cross Section Navigator when done.

Extracting a Ditch Profile from Cross Sections

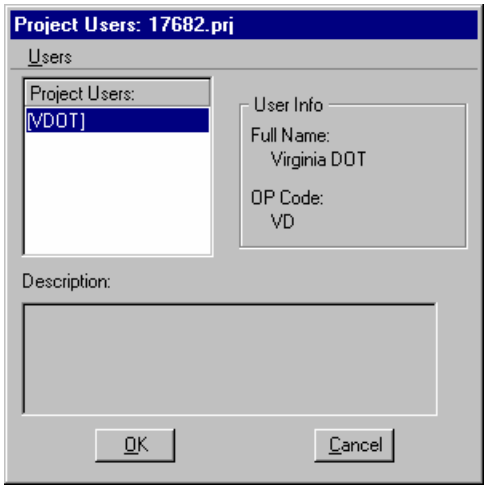
Step 1. Invoke the Road Tools tool frame from the MicroStation menu bar by selecting **Applications > GEOPAK Road > GEOPAK Road Tools.**

Step 2. Access Project Manager using the Project Manager icon from the Road Tools tool frame.

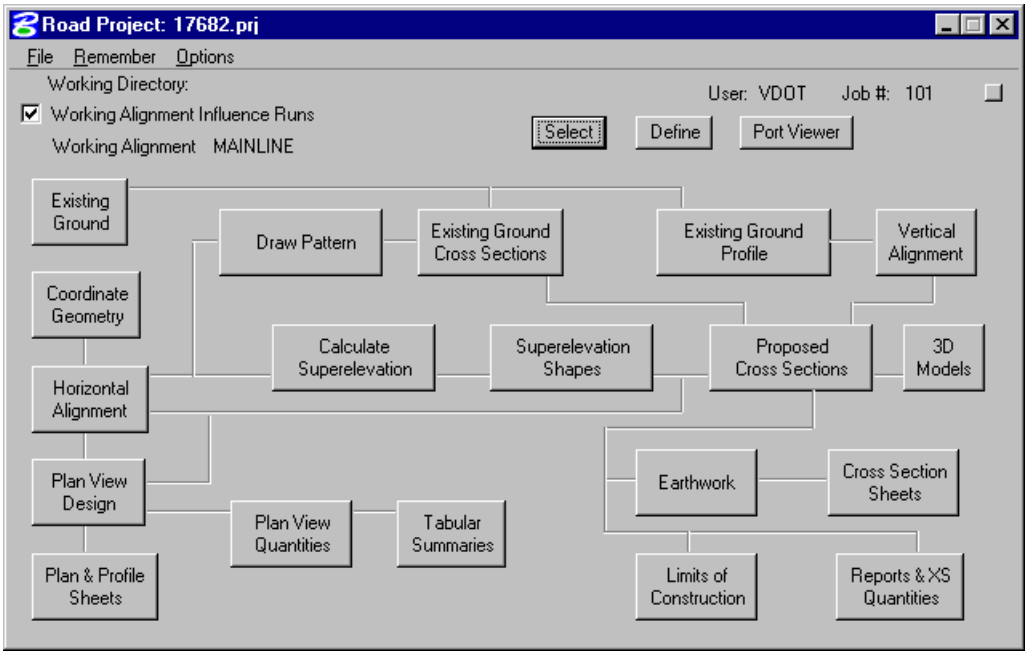
Step 3. Select the Project: d17682.prj located in c:\data\geo\vdot\drain1 and click **OK**



Step 4. Select the proper User and click **OK**.

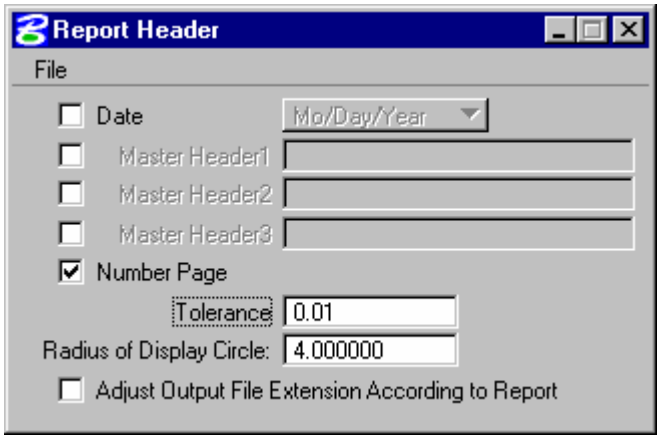


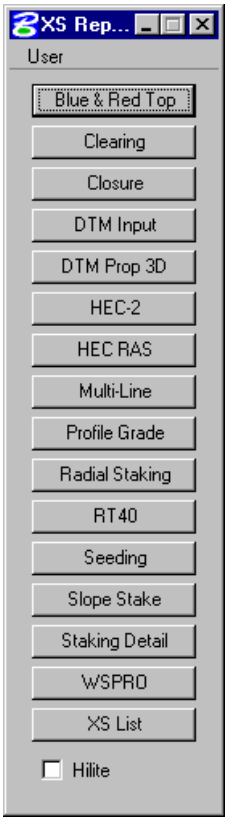
Step 5. You will now see the main workflow dialog box.



Step 6. Select **Reports and XS Quantities**:

Step 7. From the XS Reports dialog box, select the **User > Preferences** dialog box.





Set the tolerance to **0.01**.

Dismiss the **Report Header** dialog box.

Step 8. From the **XS Reports** dialog box, select the **Profile Grade** report.

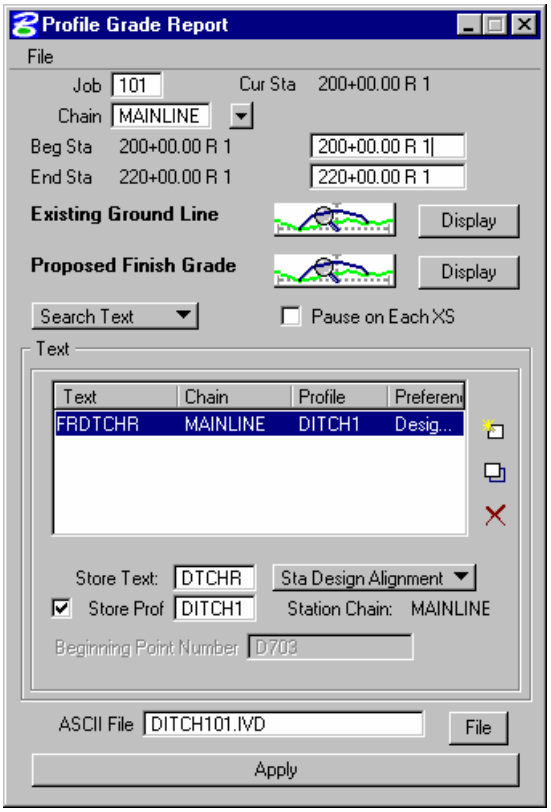
We are going to use this report to create a Profile along the bottom of the right side ditch.

Step 9. Fill out the **Profile Grade** dialog box as shown.

Select the proper symbology for the Existing Ground line and the Proposed Finish Grade.

Existing Ground Line : Level1

Proposed Finish Grade : Level2,Level3,Level4,Level5



To add the Text Search information to the collection box, push the **ADD** button. (Store Text: **FRDTCHR**). This text has been previously written in the cross section DGN file at the bottom of the ditch.

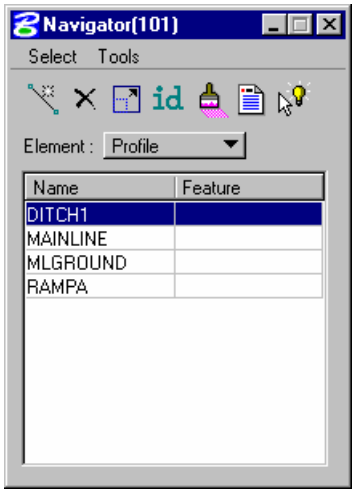
Step 10. Click **Apply** to start processing. Then, dismiss the dialog box.

Review the Ditch Profile

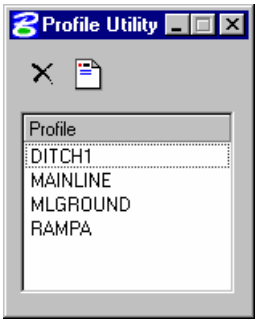
Step 1. Access **Coordinate Geometry** from the Project Manager dialog workflow.

Step 2. Access then Coordinate Geometry Navigator using **Tools > Navigator**.

Step 3. Use the **Profile** option to see a list of the stored profiles.



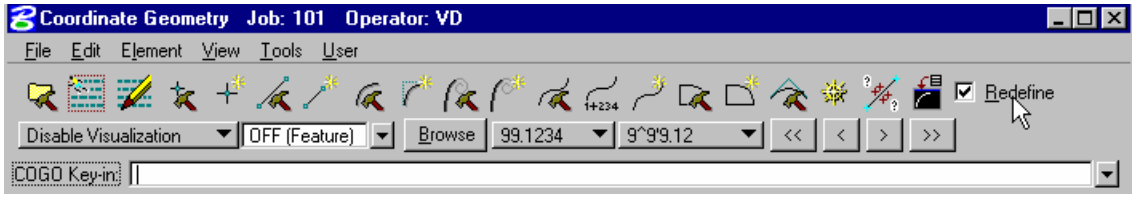
Step 4. Select **Element > Profile > Utility** from the Coordinate Geometry menu to invoke the list print dialog box.



Step 5. Select the profile “DITCH1” and press **Print.**

Step 6. Review the profile information in the COGO output display window.

Step 7. Check the Redefine box in the Coordinate Geometry dialog. This will allow us to overwrite information into the geometry database (job101.gpk).



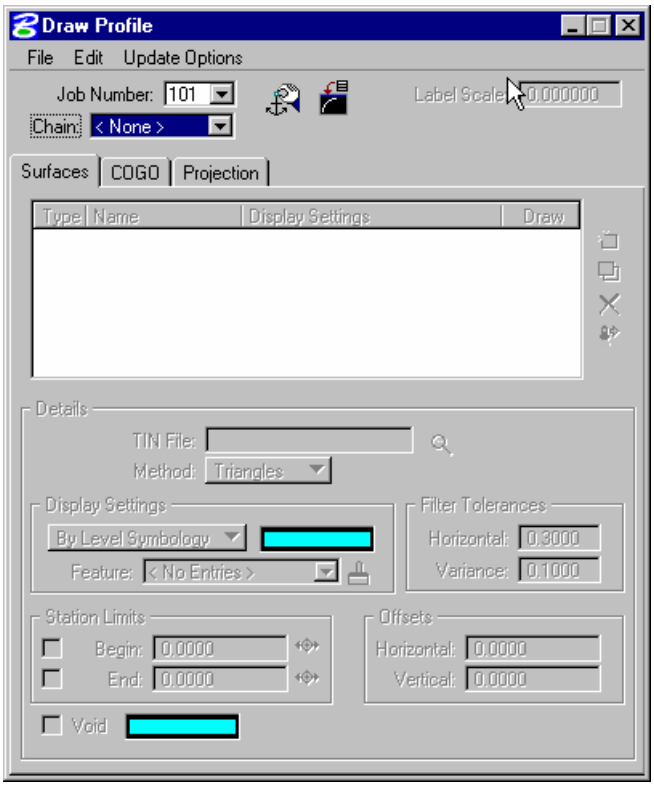
Step 8. Close the Coordinate Geometry dialog box . When prompted to save the COGO session select NO.

Draw the Ditch Profile

Step 1. Open **c:\data\geo\vdot\drain1\h17682.dgn**

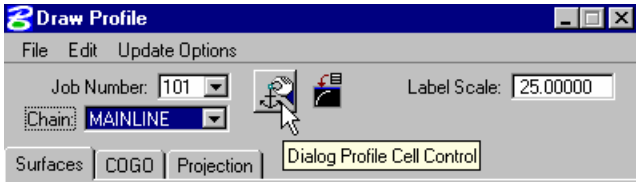
Step 2. Select the **Draw Profile** tool from the Road Tools palette or **Applications > Geopak Road > Plans Preparation > Draw Profiles.**

The dialog box below will open.

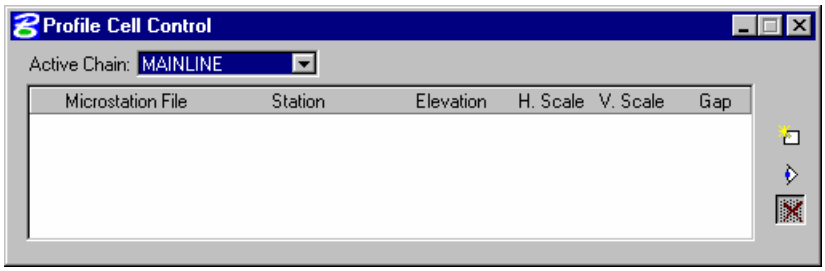


Step 3. Select chain **MAINLINE**.

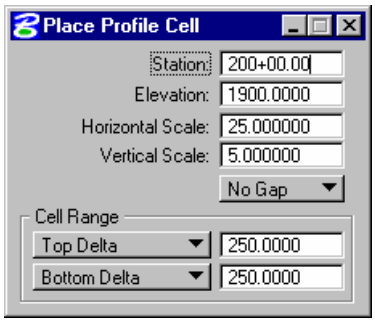
Step 4. Click the Dialog Profile Cell Control icon (to the right of Job Number).



Set the **Active Chain** to MAINLINE.



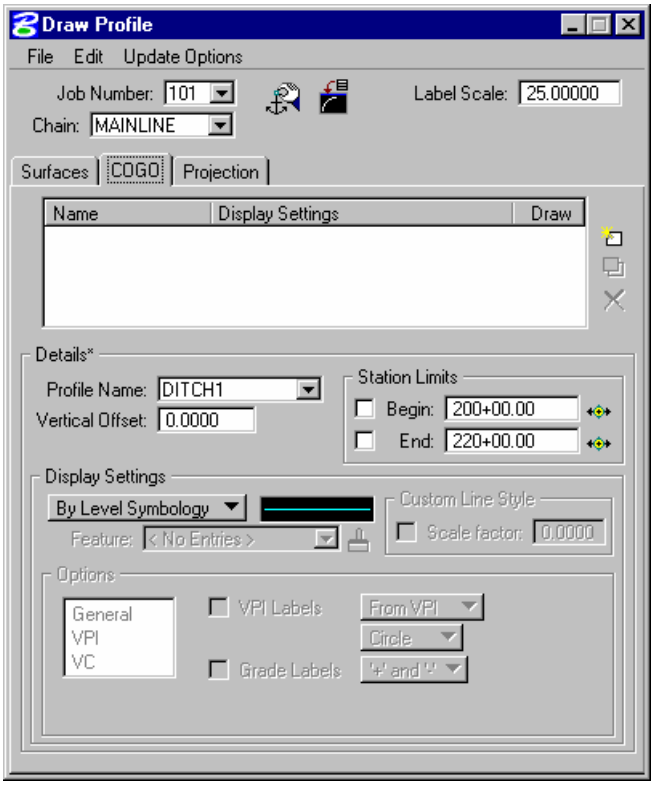
Step 5. Click the **Place Profile Cell** icon on the right side of the dialog box. This will open the Place Profile Cell dialog box.



Step 6. Fill the dialog box out as shown above and then **DP** somewhere in the DGN file in order to place the profile cell into the design file.

Step 7. Close the **Profile Cell Control** dialog box.

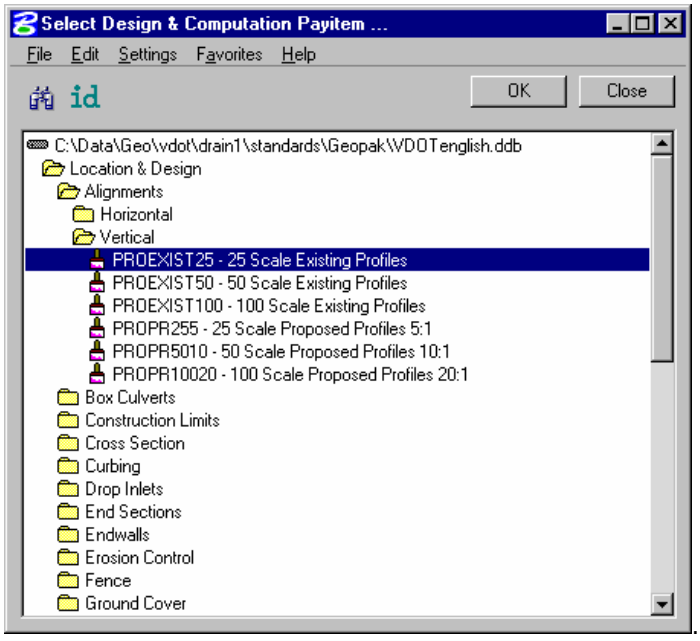
Step 8. Click the **COGO** tab and set the **Label Scale** to **25**.



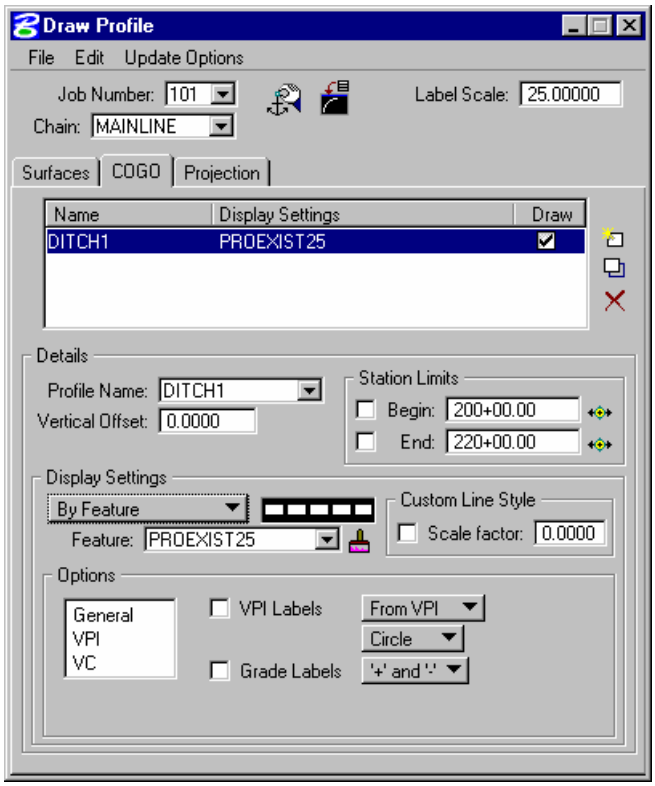
Step 9. Select Profile Name **DITCH1** from the drop down list in the Details area.

Step 10. Under the **Display Settings** set the option to **By Feature**.

Step 11. Click on the paintbrush icon and select the feature shown below. Then click **OK**.



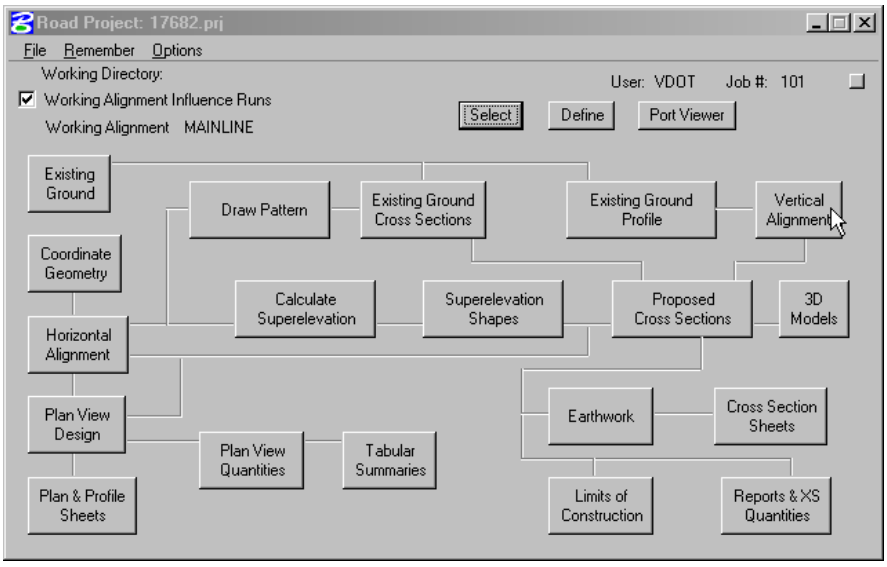
Step 12. Click the **Add COGO Profile Settings** icon to draw the profile. This will draw the ditch profile with the proper symbology.



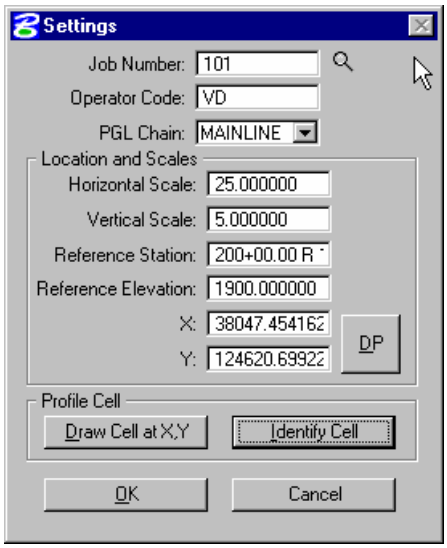
Step 13. Dismiss the Draw Profile dialog box.

Create a Special Ditch Profile

Step 1. Select the **Vertical Alignment** button from the Road Project: **17682**.prj workflow dialog

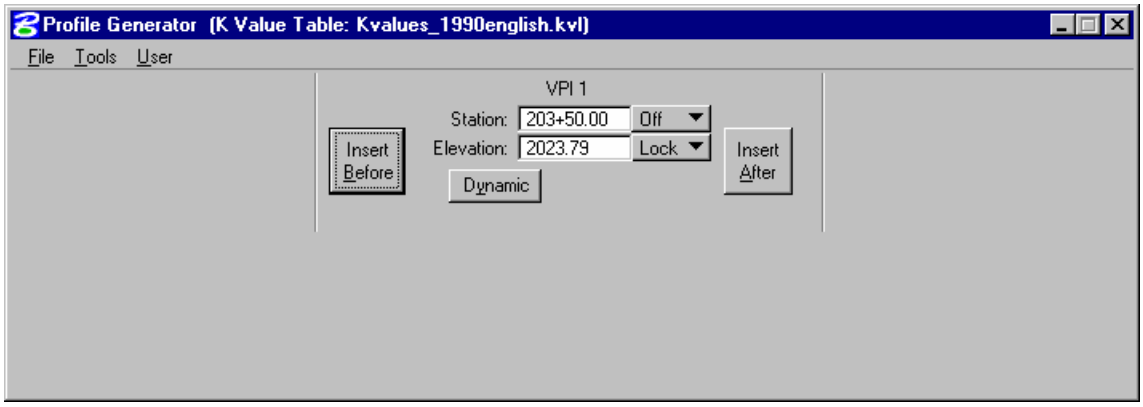


Step 2. Adjust the Settings for the proposed ditch profile by pressing the **Identify Cell** button and selecting the profile cell in the DGN file.



Step 3. Press the **OK** button to commence development of the special ditch profile.

Step 4. We are going to develop a special ditch profile between stations 203+50 and 206+00. Using the Dynamic button try to locate the first VPI. Be sure to press Enter after each entry is made.



Step 5. Press the **Insert After** button and complete the VPI Station and Elevation of the subsequent VPI's as detailed in the table below.

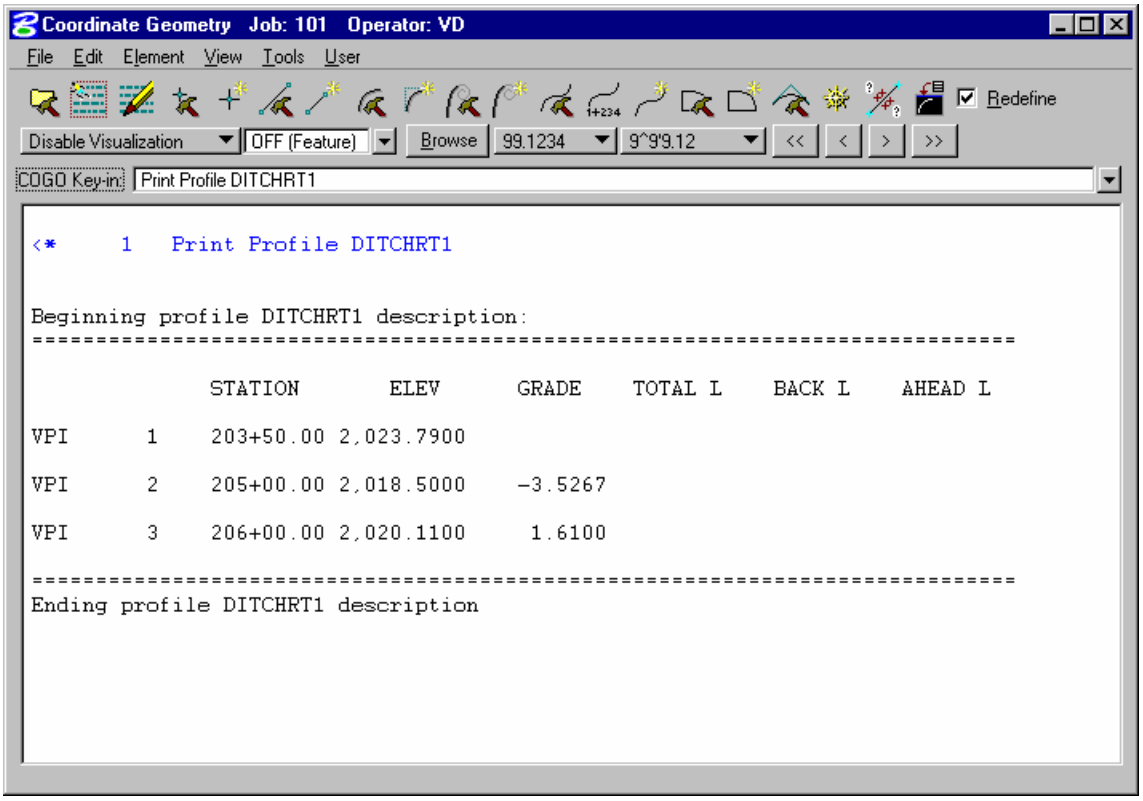
VPI	Station	Elevation
2	205+00	2018.50
3	206+00	2020.11

.

Step 6. When completed, select **File > Save As** from the Profile Generator to save the profile DITCHRT1.

Step 7. Exit the Profile Generator. When prompted to save your profile, answer **No** since we did that in the previous step.

Step 8. Open COGO and use the **Element > Profile > Utility > List/Print** tool to see a listing of information about the profile. The **Element > Profile > Elevations** tool will provide elevations at any station or increment of stations desired.



Step 9. Exit the Coordinate Geometry dialog box. Select NO when prompted to save the session.

Step 10. Now, Roadway designers should be able to incorporate this new profile when they update their cross sections runs.

Step11. Exit MicroStation.

17 Pond Design with GEOPAK Site Modeler

Objective:	Provide the user with a general understanding of GEOPAK Site Modeler functionality. Gain a general understanding of Site Models, Site Objects and Site Element interaction, associations, and terminology
Goal:	Setup a Geopak Site Modeler Project and associated Project Preferences.
Prerequisites:	Completion of an existing ground Digital Terrain Model.
Additional References:	GEOPAK Site Online Help – Modeler

Introduction

The GEOPAK Site Modeler enables fast, dynamic development of site models and easy management of the many changes that occur on site projects. It captures design intent as users work and provides immediate visual feedback. The software allows users to perform engineering modeling within a Digital Terrain Model (DTM)—without merging or extracting graphics into the DTM. Many traditionally cumbersome processes are reduced to one step: for example, you can dynamically move a building pad in a single drag and drop step—without measurement, clipping, merging, and placement steps.

The Site Modeler offers unprecedented flexibility in interactively working with site designs without the limitations of traditional iterative steps. No cumbersome triangle or point editing is necessary. The software maintains existing ground and provides full control over defining elevations and side slopes and balancing cut-and-fill. Analytical tools are included to afford powerful evaluation of site designs.

GEOPAK Site integrates digital terrain modeling with interactive 3D site design. The software allows you to incorporate design features in the model while maintaining existing ground. Move building pads. Change pond elevations. Extend retaining walls. Trim curbs. Resize berms. The Site Modeler automatically regenerates the DTM, yet retains full integrity of the models and the original DTM. There is no need for merging or extracting graphics into the terrain model. Easy on-the-fly functions let you change elevations and side slopes and balance cut-and-fill. You can add features such as contours to the model as needed to define your design. Tools are supported for evaluating the site design and producing drawings and site models.

Site Modeler Project Components

A GEOPAK Site Project is comprised of three components: Elements, Objects, and Models.

GEOPAK Site Elements

Elements are any MicroStation graphical element assigned an elevation and DTM Feature (breakline, boundary, contour, etc.) with GEOPAK Site tools. They can be placed into a 2D or 3D design file utilizing any generic MicroStation command, except Make or Drop Complex Chain. Once the elements are drawn, elevations are assigned with GEOPAK Site tools. At any time, they can be moved, copied, or otherwise manipulated. Once a group of elements is drawn to the designer's satisfaction, they can be defined as an object.

Elements can be modified at any time during the site design process. To move the location of an object in the X-Y plane, utilize any MicroStation command, except Create Complex Chain.

Let's review several types of modifications, their impact on element elevation(s) and their results on the site project. If using the Site Element Association feature, the elements elevations will be recreated per their original definition. If not using the associations then the following will apply to the elevations of the site elements.

Note: All modifications support selection sets and fence operations.

Extend / Shorten Line	Use generic MicroStation commands to shorten or lengthen. The Site Modeler holds the elevations of the original element.
Insert Vertex	Elevation of new vertices is interpolated between adjacent vertices.
Move Element	Maintains all vertices at the original elevations.
Scale	Maintains the elevations at all vertices.
Rotate	Maintains the elevations at all vertices.
Mirror	Maintains the elevations. If there are duplicate vertices due to the mirroring, the second elevation is utilized.
Delete Line	Removed from the DTM object.

GEOPAK Site Objects

Site Objects are collections of Site Elements that are grouped together into Objects for the purposes of side slope definition, model merging, volume quantity options, and logical geometric components. Site Elements can be added to Objects as they are created or at any point in the design process. Site elements can also be removed and included in more than one Site Object. Examples include:

- Parking lots
- Buildings
- Ponds

Each object can contain an unlimited number of elements. In addition, it has other site attributes and capabilities that should be considered during object creation:

- Side slopes - defines the interrelationship between the object and the model. Cut and fill side slopes are automatically generated around the extent or boundary of each Object in the model. The boundary is automatically determined from the extent of the elements contained in the object. Side Slopes can be as basic as a single cut or fill slope from the edge of a building to existing ground, or intersecting slopes between two objects, i.e., a building and a parking lot.
- Quantity depth - In order to do excavation quantities, an object may have an associated depth. For example, a building may have a quantity depth of two master units, which represents the amount of granular material required beneath the building. These quantities can be easily changed, and quickly computed.
- A specific set of visualization settings can be applied to each object to control the display symbology of each.
- A Site Model is created by merging each of the triangulated Objects specified in the Model. The designer determines which slopes control the merging process by means of a FIFO list.

The first object, also referred to as the Base Object is the starting surface. Each object is then merged "on the fly" in the order listed. To change the controlling slopes for intersecting, simply change the order of the list.

- Site Objects can be raised and lowered in their entirety to easily evaluate design alternatives.
- The Objects layout and the geometry of the member elements should be evaluated. The boundary of Objects is determined from the extent of the elements and the objects triangulated representation will include all elements. Therefore, specific attention to the grouping of elements based on vicinity and shape of the desired objects in the model should be made. It is recommended to create more objects than fewer as it provides more control and more flexibility in the design and evaluation of the final model.

Objects can be modified at any time during the site design process. The following are methods of modification:

- Add / remove elements from the object
- Modify Object Slopes
- Raise or lower the object

GEOPAK Site Models

A model is comprised of an unlimited number of objects, one base object (i.e., existing ground), and a “First In – First Out” (FIFO) list which dictates the merging order. Note the designer does not have to merge, as it's done automatically "on the fly." As a new object is added to the model, its resultant slopes, contours, etc., can be displayed on the screen.

Models can be modified at any time during the site design process. This can be accomplished in three methods:

- Add or remove objects from the model
- Change the base object, i.e., update existing ground terrain data
- Changing the FIFO list

The workflow involved in using the Site Modeler is designed to expedite the creation of digital terrain models while providing enough flexibility to easily accommodate design changes. The following outlines getting started with the Site Modeler.

- Establish base design planimetrics in 2D or 3D MicroStation design file. This includes design features that define the DTM (e.g., centerlines, curbs, berms, ponds, property line etc.).
- Obtain design information such as original ground or survey information that serves as the basis of the initial design.
- Review the geometric layout of design features and determine a preliminary concept for the type and configuration of Site Objects to be used in the design.
- Start Site Modeler New Project Wizard.
- Create an empty Model.
- Import the base design information into the Model.
- Select a location to begin the DTM design.
- Create a new Site Object that fits the starting location.
- Begin defining the elevations of the Site Elements and adding them to the Active Object.
- Continue to create Site Objects, Site Elements as needed.
- Evaluate and analyze the Site Model and adjust Site Elements, Site Objects and Model.

Site Modeler Main Menu

GEOPAK Site is invoked from within a MicroStation design file. If the Project Wizard dialog is bypassed (by activation of the Don't Show this dialog at Startup), the GEOPAK Site Modeler main menu bar (along with the Active site Object Control) is automatically displayed. An alternate method of invoking the menu bar is completion (without pressing the Cancel button) of the Project Wizard.

Each menu selection accesses GEOPAK Site project information, feature placement and tools necessary to complete a GEOPAK Site project. Tool frames and tool boxes are also supported for all functions and are accessed via the Tool boxes pull down. The current Site working directory and Project File are displayed in the header of the menu bar.

Project Menu Options

The Project Menu selections are utilized for creating new GEOPAK Site projects, opening existing projects, saving projects, establishing the project Preferences, Importing and Exporting data in and out of the project file, and exiting the Modeler.

When working on a project, frequent use of the File > Save pull down is important. Any changes made in the project are not recorded in the GEOPAK Site File (GSF) file unless a File > Save or File > Save As operation is executed. Hence, any power interruption or other malfunction will result in loss of data that can be avoided through the judicious use of the File > Save tool.

Each GEOPAK Site project contains a set of Preferences which control the project file components and visual symbology which are accessed through this menu selection.

GEOPAK Site Modeler supports two operations to import data into a site project:

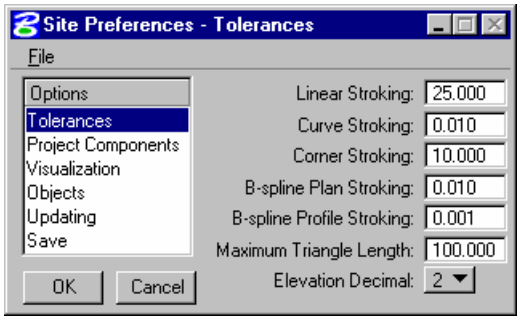
- Data
- 3D to Object
- In addition, it supports three operations to export data from a site project:
- Visualization to DGN (Contours, Triangles, etc)
- Model / Object (Creates DAT & TIN Files)
- To COGO (Creates Points in the COGO Database)
- Export to COGO Profile (Creates a Profile in COGO)

To exit GEOPAK Site, the designer may utilize the File > Exit tool on the Project pull down. When exiting, GEOPAK Site Modeler prompts for saving the current project.

Project Preferences

GEOPAK Site Modeler supports a wide array of user defined Preferences which enables the designer to set project specific options, or a larger organization to set parameters to maintain standards. Each GEOPAK Site project contains a set of Preferences and they remain with the project.

The general groupings of Options are displayed in a list box in the left side of the dialog, while the right side displays parameters for the highlighted Option. As different options are selected in the list box, the right side of the dialog changes dynamically to reflect the parameters for highlighted Option.



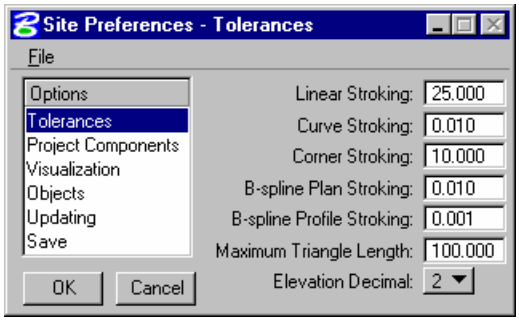
The general groupings of Options are displayed in a list box in the left side of the dialog, while the right side displays parameters for the highlighted Option. As different options are selected in the list box, the right side of the dialog changes dynamically to reflect the parameters for highlighted Option.

These Options and very brief descriptions are detailed below.

Tolerances	Tolerance and stroking options used in the creation of Site Elements.
Visualization	Variety of element symbology and display options
Updating	Toggle for automatic updating
Objects	Standard List of Object Types and default values for Object parameters
Save	Automatic saving and backup options

On the right side of the dialog, the **OK** and **Cancel** buttons commence closing the dialog and storing the preferences, if required.

Tolerances



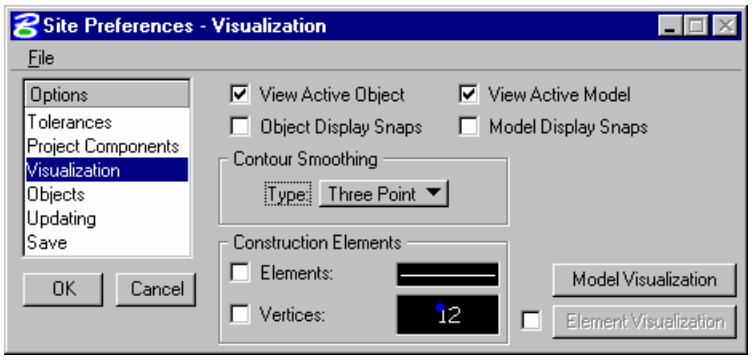
Linear Stroking	The Linear Stroking option densifies portions of linear elements and is utilized to interpolate new spots from the linear elements.
Curve Stroking	The Curve Stroking feature densifies the curved portions of the break lines. Curved break lines in GEOPAK are handled by segmenting the curve into small chord segments. The length of the chord segments can be determined by setting the Curve Stroking.
Maximum Triangle Length	External triangles whose external edge is longer in length than the Maximum Triangle Length user specified distance are dissolved.

Elevation Decimal	The Elevation Decimal controls the number of decimal places displayed on the elevation labels for points within the objects and models.

Note: Stroking is the process of automatically inserting points along Site Elements by interpolating new points from the linear and curved sections of the data.

Visualization

A variety of user-defined parameters are supported, to provide the designer with maximum flexibility in Visualization. These parameters are briefly detailed below.



View Active Object — When activated, the current object listed in the Active Site Object Control dialog is displayed. This object will maintain the display settings during view updates and changes. All visualization settings are derived from the definition of the specific object that is active.

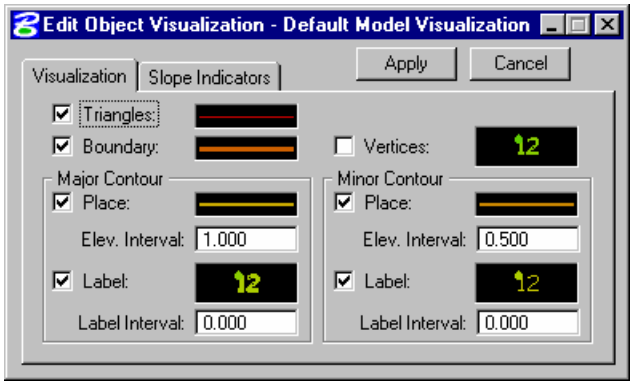
View Active Model — When activated, the current model listed in the Active Site Object Control dialog is displayed. This model will maintain the display settings during view updates and changes. All visualization settings are derived from the definition of the specific model that is active. If the mode is inactive, no models are displayed as depicted in the example below, where only the MicroStation elements are displayed.

Smooth Contours — When active, an algorithm is employed to smooth contours.

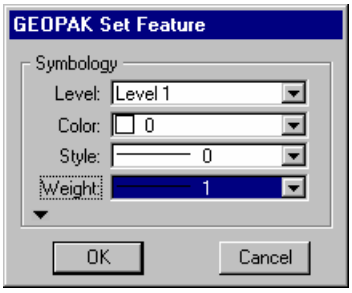
Construction Elements — The element symbology for Site Elements created in the modeler that is not contained in a Site Object. It may be desirable to establish Site Elements and their elevations purely as a reference or to construct other elements. These Site Elements are not needed in the Site Objects themselves and are considered Construction Elements. Site Elements contained in Site Objects are displayed according to the object visualization of the object to which they belong. The Construction Elements are displayed using these settings in the preferences.

Model Visualization

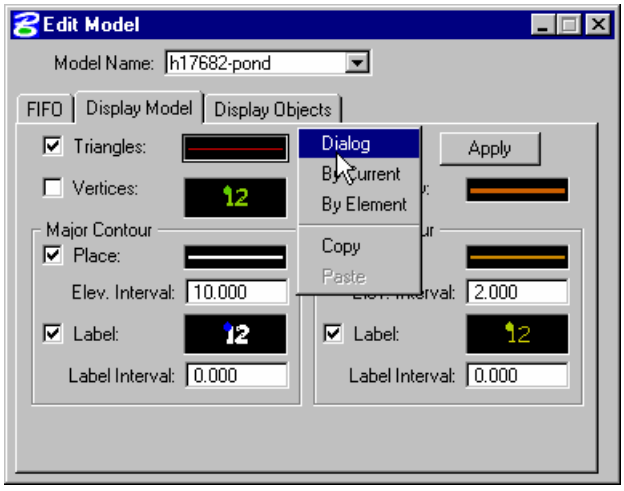
When the **Model Visualization** button is pressed, the dialog below is invoked.



The Visualization tab contains specifications for all features that are available for display in Site Models including triangles, boundaries, and vertices. These settings will be applied to all models created subsequent to setting these. All symbology controls, the sample line and text graphics in this case, can be set by double clicking the sample and using the Set Feature dialog shown below.

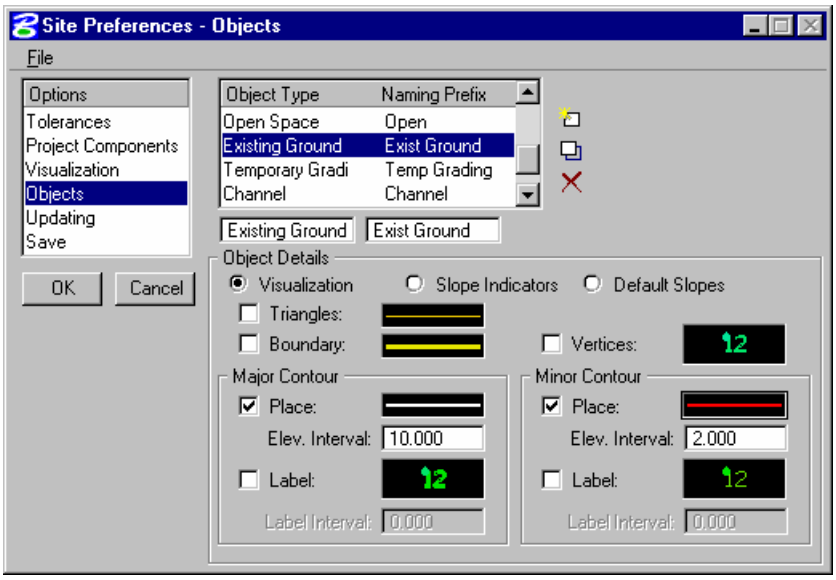


Additional options can be invoked by clicking the right mouse button over these sample controls.



Objects

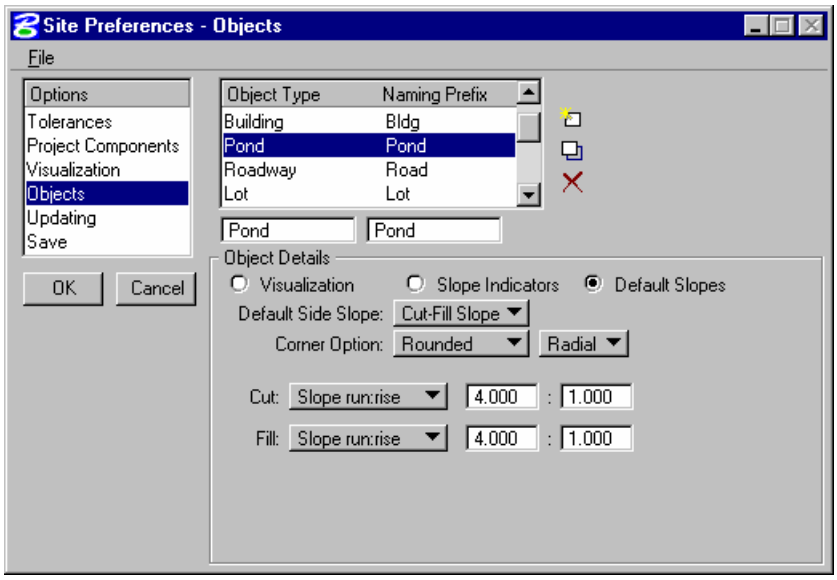
When the Objects option is selected, the dialog changes as reflected below. This option is utilized for creation, deletion, and manipulation of the library of objects. The information includes both the Object Type and Naming Prefix.



This list box displays all Object Types within the active project. Default types which are initially displayed include: Parking, Building, Pond, Roadway, Lot, etc. These Object Types assist in organizing and categorizing design features, controlling standard symbology, and establishing defaults for Objects created.

The portion of the dialog titled Object Details allows the user to set Visualization of the highlighted Object Type. Each Object Type may have its own settings, providing maximum flexibility to the user to display only specified parameters for the particular Object Type. Note its similarity to the Default Model Visualization dialog.

When an Object Type is highlighted and the Object Details is set to Default Slopes, the dialog depicted below is displayed.



As new Objects of a particular type are created they will obtain the default settings. Note that an Object slopes can be modified at any time, these defaults are just applied when the object is created and remain in effect until the slopes are modified while editing the Object. Slopes are applied around the boundary of the entire object and are generated as the intersection from the Object boundary to the Model. Three Default Side Slope options are supported:

- No Slopes
- Cut - Fill Slope
- Cut-Fill Table

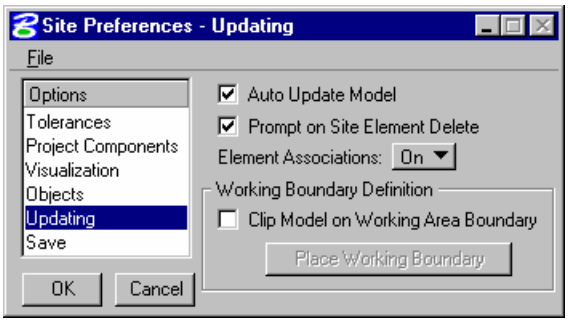
Corner Options can be specified to generate either Rounded or Straight slopes around corners of Objects and can be projected Radial or Planar. Supported options for Cut and Fill specifications are:

- Slope run:rise
- Slope rise:run
- Slope %
- Slope Unit / Unit (ft. / ft. or meter/meter)

Simply specify the desired slope.

Updating

The Updating dialog is depicted below.



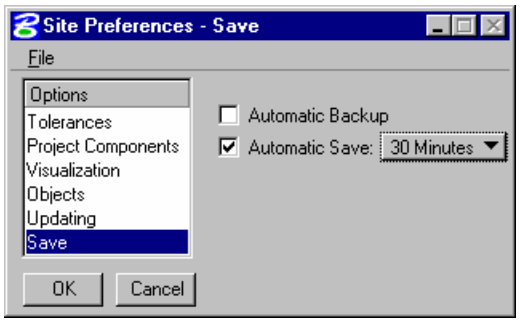
When the Auto Update Model is active, the model is updated whenever a change is made to an element, object, or the FIFO of the Active model. As Models get larger, it may not be necessary to update the Model on every single edit. Deactivating this speeds up the modifications that are being performed, but must either be activated or the Model itself processed to view any changes incurred.

Element Association refers to the relationship elements maintain with the original method by which they were defined. Elements defined by other elements / object / models can always maintain the same dependency on the items referenced in their creation.

When activated (On), it enables the associations. All subsequent element modifications that occur to the dependent elements or referenced identity utilize the original association when computing the impact of the modifications. Turning the toggle off (OFF) disables the associations.

Save

The Save dialog is depicted below.



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

The Automatic Save Option saves the GEOPAK Site project file (.GSF) to the disk at the user specified interval. Automatic Save intervals include 1, 2, 5, 10, & 30 minutes.

Model Menu

The Model menu selections provide the mechanism to add, edit and delete the various elements which comprise a Model, including Display parameters, merging order (FIFO), and Display options. Each of these tools invokes a dialog wherein the specific model information can be added or edited, or entire models deleted.

Edit Model

The Model editing options provide the mechanism to change the list of Objects contained in the model and modify the display setting of the model and its objects. When invoked, the dialog depicted below is displayed.

FIFO Functionality

The FIFO, or "First in First out," contains the list of all the Site Objects contained in the model and the order in which the Objects are to be merged. This Object list is processed from the top to bottom, starting with the Base Object any time a change is made to a Site Object or Site Element contained in the Model. The Site Preferences contain an option to disable this automatic updating should this immediate feedback be unnecessary or grow time consuming as the model grows. Since Objects can easily be added and removed from the FIFO list, certain time savings can be achieved by removing Objects not needed for the current design process and can be added back at the end or at such time as their impact is needed.

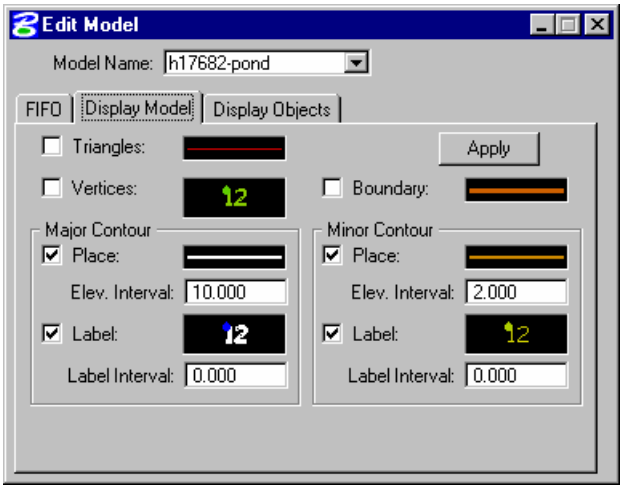
The process used in the creation of the final triangulated Site Model is as follows:

- The Base Object starts as the initial state of the Model.

- The first Object in the FIFO list is retrieved.
- The boundary of the Site Object is determined from the extent of all the Site Elements contained in it.
- The Side slopes associated with that Object are applied around the boundary down or up to the Model depending on the cut fill situation.
- This Object is then merged into the Model.
- This new state of the Model with the first Object and its side slopes merged into the base Object becomes the initial sate of the model for the next Object in the FIFO list and the process is repeated for all the objects in the list.

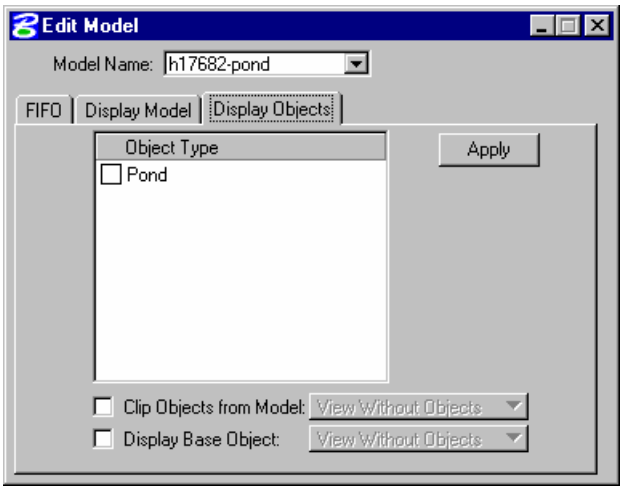
Display Model Tab

The Display Model Tab is utilized to control the visualization settings of the entire model. The dialog is depicted below and contains specifications for features displayed within the Modeler including triangles, boundaries, and vertices. When the toggle to the left of the feature is active, the element is displayed. When inactive, it is not drawn. To the right of each feature is the level and element symbology.



Display Objects Tab

When the Display Objects Tab is selected, the dialog depicted below is displayed.



Object Type — List box containing all current Object Types in the current model. If the eye icon is displayed to the left of the Object Type, then all objects of the specified type are displayed with their visualization settings.

Clip Objects from Model — When activated, this clips every object out of the model prior to display. It is good for showing proposed vs. existing.

Display Base Object — This toggle indicates whether the Base Object should be displayed with its visualization settings.

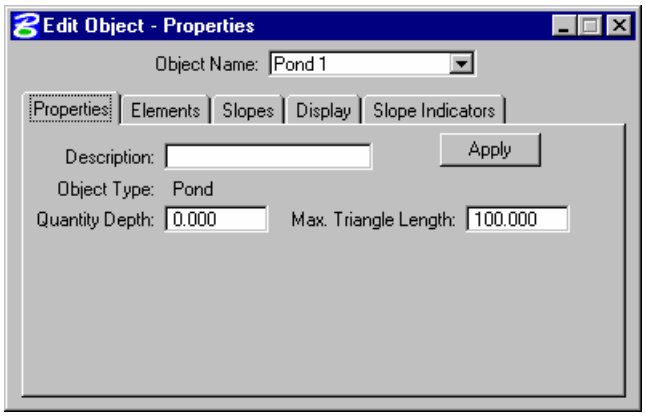
Apply — Commences the display procedure, which reacts to any display changes made.

Objects Menu

The Object menu selections provide tools to Add, Edit, Copy, Move, Raise, Lower, or Delete an Object.

Edit Object

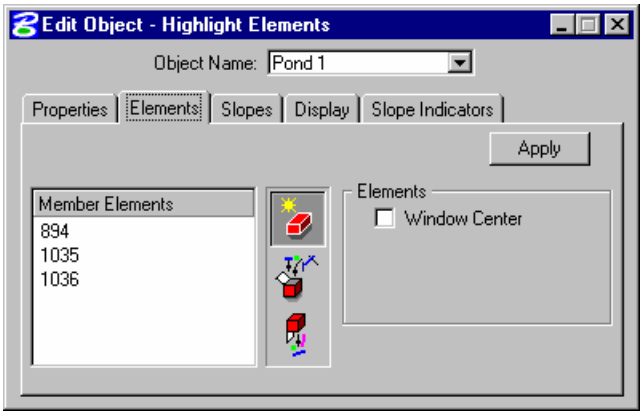
The object editing options provide the mechanism to modify an Object's properties, side slopes, and display settings.



Object Properties

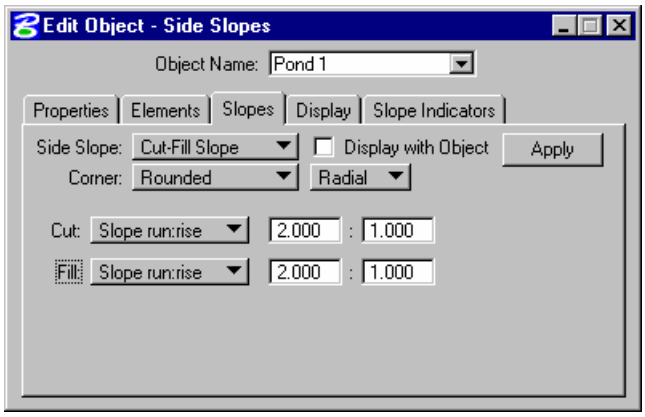
Under the Properties Tab the Object Name for the object to be edited is displayed in the dialog. A description can be entered if desired. Quantity Depth is the vertical depth beneath an Object where the earthwork volume is adjusted to compensate in the total volume. Note that this depth is applied to the limits of the Object's elements and is optional when Volumes are computed.

Three tools are supported within the Elements group box. Highlight and/or window center Site elements within the Active Site Object. Add previously created Site Elements to the Active Site Object and Remove Site Elements from the Active Site Object.



Object Slopes Tab

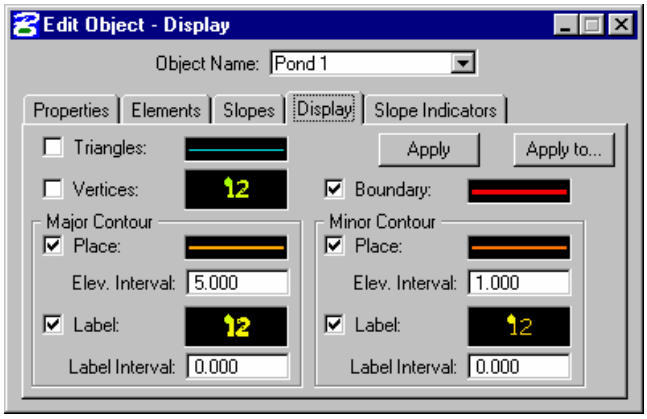
The Slopes options dictate the type of side slopes generated from the outer extent or boundary edge of the Object. These slopes are applied when the Object is placed into a Model and computed from the Object to whatever the state of the Model when it is merged. When the Display with Object toggle is activated, GEOPAK includes the display of slopes in the Object symbology.



There are four slopes options available: No Slopes, Cut - Fill Slope, Cut-Fill Table, Dynamic Slopes. Corner Options can be specified to generate either Rounded or Straight slopes around corners of Objects.

Display Options Tab

The Edit Object Display Tab is utilized to control the visualization settings of the Active Object displayed in the dialog. The dialog is depicted below and contains specifications for features displayed within the Modeler including triangles, boundaries, and vertices. When the toggle to the left of the feature is active, the element is displayed. When inactive, it is not drawn. To the right of each feature is the level and element symbology.



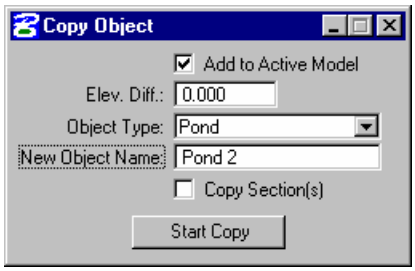
Slope Indicators Tab

The Slope Indicators Tab is used to set the display preferences of Object Slopes Indicators.

Copy Object

The copy tool is useful when the object is comprised of numerous elements which need to be copied.

To copy an object, simply key in the Elev Diff (0 if there is none), then press the **Start Move / Copy** button and identify the object to be copied. A second data point defines the revised location.



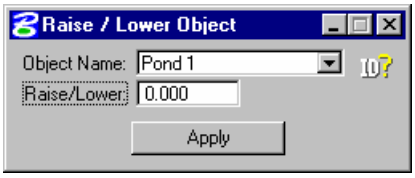
Note: When the Make Copy toggle is not activated, the dialog dynamically changes to a Move Object dialog.

Move Object

To move an object, simply key in the **Elev Diff** (0 if there is none), then press the **Start Move / Copy** button and identify the object to be moved. A second data point defines the revised location.

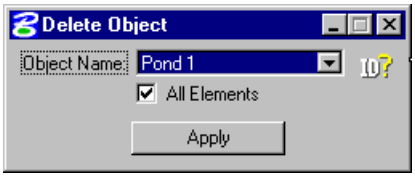
Raise/Lower Object

Raise/Lower Object tool can be used to change the elevation of an entire object by raising or lowering it. Note the value is in terms of master units, i.e., feet or meters.



Delete Object

The Delete Object tool deletes the entire object from the project file and (if desired) all the Site elements that are part of the Object.



Elements Menu

The fundamental components in the GEOPAK Site Modeler, Site Elements are simply MicroStation graphics (2D or 3D) that have been assigned elevations using one of these Site Element tools.

GEOPAK Site Modeler supports a wide variety of tools that when combined with generic MicroStation commands create and modify elements. These operations include:

- Creating / editing elements
- Changing element feature types

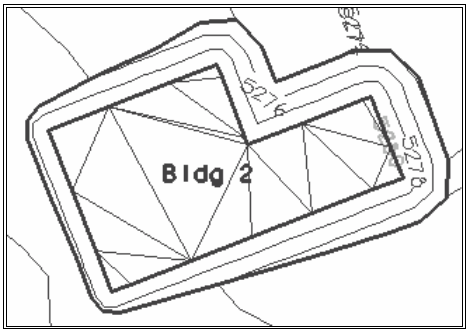
- Change Element Associations
- Raising/Lowering of elements
- Copy parallel existing elements
- Modifying elements
- Deleting element Z
- 3D elements
- Composite Sections
- Edit Profile
- Obtaining and modifying element information

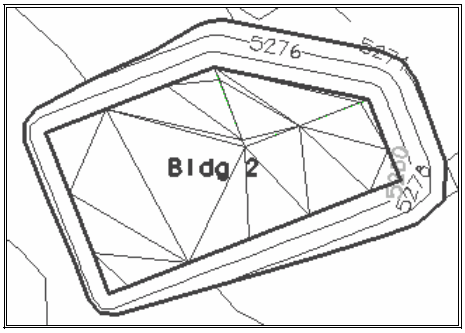
The 3D Element is active only within MicroStation 3D design files.

Element Feature Types

All Site Elements require a **Feature Type** to describe the way in which to process and interpret the features within the terrain model.

A **Boundary** is used to constrain the external boundary of the object or model. For example, an L-shaped building can be defined as a boundary, so that no triangles are created within the internal corner. If no boundary elements exist within an object, the limits of the object would result from the convex area of all the elements contained in the object.





Feature types can be modified at any point in the design to create the desired results. It is often desirable to leave features as break lines until the object design is near completion and then the desired elements can be specified as boundaries.

Break Lines designate linear features such as edges of buildings, parking lots, and other pavement. The generated triangles never cross a break line, rather the edges of the triangles are coincident with a break line.

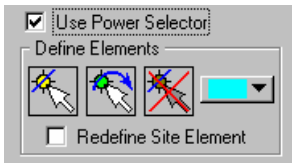
A **Contour** is an element of constant elevation. Triangles can intersect and / or cross over contours.

A **Void** delineates an area of no data or obscured area and is defined in a series of points forming a closed element.

A **Hole** is extremely useful when the base object surface (i.e., existing ground) is desired within an object. For example, an area of existing within a building footprint is to be landscaped and remain at the original ground elevation, while surrounded by a building pad.

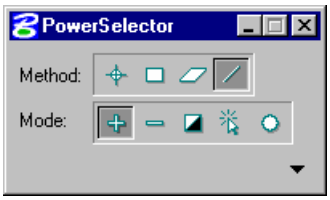
Selection Sets

Many of the Element tools utilize selection sets, therefore, a generic discussion is warranted into the operation and use of selection sets. Two options are supported: the MicroStation Power Selector or GEOPAK selection tools within each dialog.



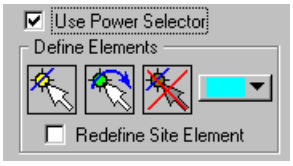
Use Power Selector

When active, the MicroStation Power Selector (as depicted below) is utilized. To invoke, press the Select Elements button in the Element Selection group box.



GEOPAK Selection Tools

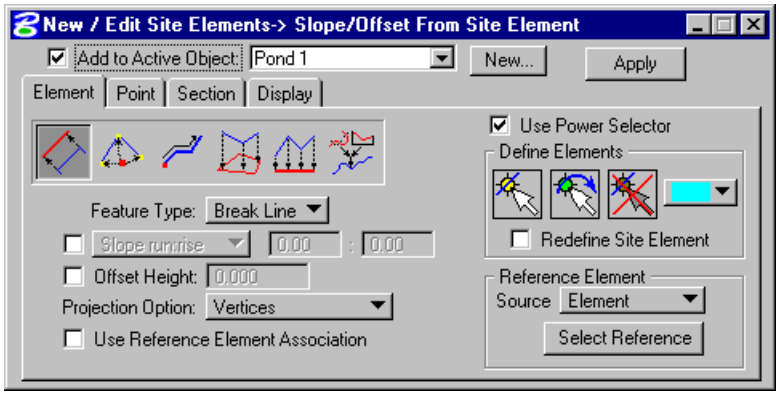
Selection commands can be utilized without the Power Selector. The leftmost button is **Select Elements**. Simply press the button, then select the desired elements. To select multiple elements, depress the <Control> button on the keyboard while selecting. To view the highlighted elements, rather than the "handles," a change in the preferences is needed. To change, select the *MicroStation Workspace > Preferences*. Then select Input and activate the option Highlight Selected Elements.



After utilizing numerous MicroStation commands, the highlighted elements may not be highlighted. To display previously selected elements, press the **Reselect Elements** button (center of the three) and the highlighted elements are displayed again in the specified color. To remove all selected elements from the selection set, press the Reset Selection Set button (rightmost of the three). The color of the selected elements is set with the color picker on the far right side of the group box. Note that Element Selection must be invoked through these selection icons. While the Site Modeler uses a selection set, it must be started through the use of these commands not the MicroStation Selection set commands.

New/Edit Site Elements

One of the primary tools within Elements is the New / Edit Elements tool. When invoked, the dialog depicted below is displayed.



The dialog contains five main options: Element, Point, Section, Side Slope and Display. When a tab is selected, the dialog dynamically changes to reflect the selection. The fields below are displayed regardless of which tab is selected.

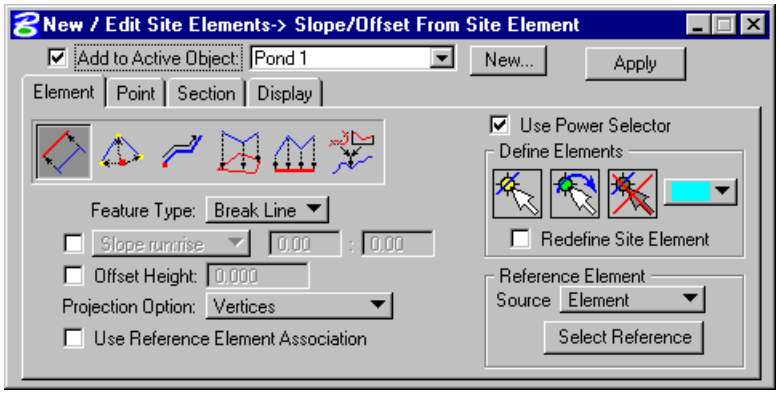
Add to Active Object — When toggled, created and/or edited elements are automatically added to the active object. Refer to the Active Object Control toolbar for the current active object.

Current Object — Located between the Add to Active Object and the Apply button, the active Object can be selected.

Apply — Commences the processing. This includes redrawing of the elements, inclusion into the active object, and if the active object is part of a model, reprocessing of the model.

Element Tab Options

The element options provide for the primary mechanism for the creation and redefinition of Site Elements. When the Element option is selected, the dialog depicted below is displayed.



Six tools to assign or modify elevations are provided. As each option is selected, the title bar as well as the left side of the dialog dynamically change to reflect the selection. These options include:

Slope/Offset from Site Elements

The Slope/Offset from Site Element option is used for assigning elevation to Site Elements by placing the elements at an optional slope and offset height from existing Site Elements. In calculating the elevation from the existing Site Element, a projection is performed from the new Site Elements to the reference based on the minimum distance from the element to the existing reference element.

Radial From Existing Points

The Radial From Existing Point option is useful for establishing the elevation of elements relative to points within another model or object. The tool uses a reference data point elevation and an optional slope and offset height to compute the elevation for the new element vertices.

Along Element From Existing Point

The Along Element from Existing Point tool assigns the elevations along an element at a constant slope and/or offset height from a data point. The data point will be projected to the element and the distance along the element from this point to the vertices will be used in conjunction with the slope and offset height to compute the elevation of the new element vertices.

Drape on Model / Object

The Drape on Model/Object tool is used to establish the elevation of Site Elements relative to the elevations contained in a model or object. It is useful to set the elevation of elements relative to existing ground or other object.

Constant Elevation

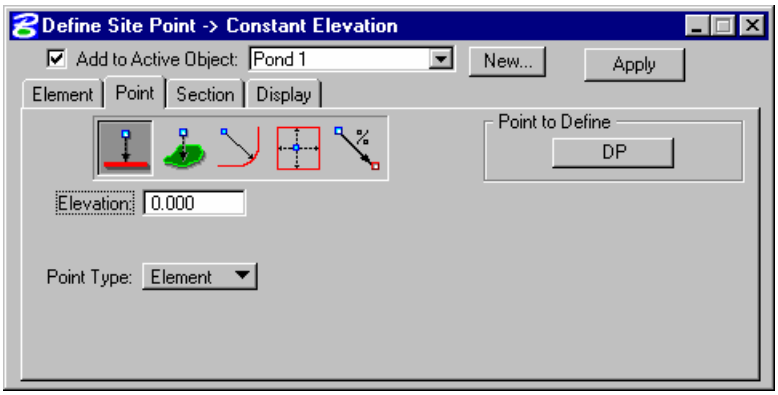
Placing elements at a constant elevation may be utilized when the desired elevation of an element is known. It can be used to set the elevation of a building or even place a contour into an object. If the computed contours within an Object do not exactly match the desired results, simply draw the contour wanted and then place in the Object as a Contour Feature at a constant elevation.

Alignment

The alignment tool is used to create Site Elements from a GEOPAK horizontal and vertical alignment. The alignments must have previously been created using one of the GEOPAK Coordinate Geometry features or the Horizontal and Vertical Alignment Generators. Once a Site Element is created from an alignment, the curbs, edges of pavement, etc., can be created using one of the other Site Element Creation tools. Typically this would be the Slope/Offset from Site Element.

Point Tab Options

When the Point Tab is selected within the New/Edit Site Element dialog, the dialog changes as displayed below.

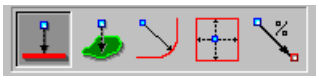


Point elements can be created and added to Objects to refine the surfaces or edit the vertices of existing Site Elements. When placing points, the Site Modeler can detect when an existing Site Element is snapped to and rather than placing a new point in the Object, the snapped vertex will be

changed to the new elevation defined in the Point options. This provides for "pull and tug" editing of Site Elements on a vertex by vertex basis. If no existing Site Element is detected, a MS point graphic element will be added to the design file and the Object.

If a point is snapped to a Site Element, but not to a natural vertex of the graphic element, the Site Modeler will actually insert the point along the Site element. This point is now considered dependent on the site Element and as the element is modified or moved, so will the point.

Five options are supported within the Point tab.



As each option is selected, the title bar as well as the left side of the dialog dynamically change to reflect the selection. These options include:

Constant Elevation

Used to add a point or edit a vertex of an existing Site element at a constant elevation.

Drape on Model / Object

The Drape on Model / Object option is useful for placing or editing points and placing them at an elevation relative the elevation in a Model or Object.

Slope from Site Element

The Slope from Site Element option is used for placing points at a given slope from a Site Element.

Min. / Max. Elevation at Slope From Elements

The Min. / Max Elevation at Slope from Elements option is used for placing points at the minimum or maximum elevation computed from a set of Site Elements. It can be used to insure a low point or high point amongst elements.

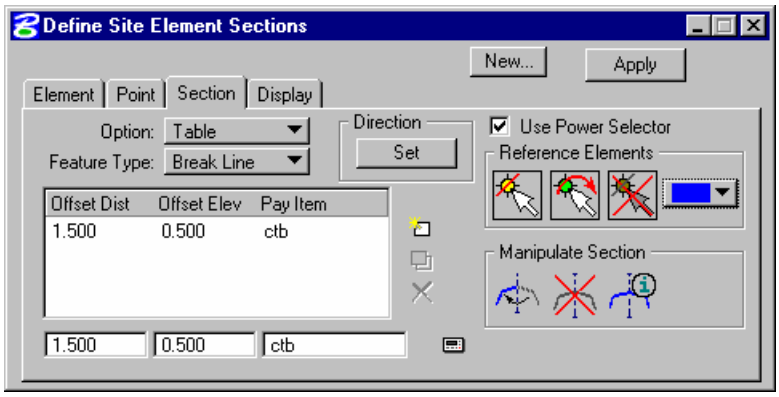
Slope from Point to New Point

The Slope from Point to New Point option is used to create a point at a given slope to an existing point in an object or model.

Section Tab Option

Sections are special Site Elements that are created and attached to existing Site Elements that define a constant vertical and horizontal offset from the existing element(s). It can be used to attach curb and gutter sections to the edge of pavement, retaining walls or benching sections.

Section Elements is another feature where the MS graphics are actually created by the Site Modeler in the active MS symbology. Section elements cannot be directly edited, they are locked and connected to the elements on which they are placed. As the Site Elements containing the section are modified, moved, deleted, or elevations reassigned, the Section Elements are automatically updated accordingly.



Provisions for assigning sections to multiple Site Elements at once, even disconnected Site Elements, are available. This capability is provided by gathering all the Site Elements selected for section placement and assigning the Direction or which side to place the section element. The Site Modeler may not be able to logically determine a direction given very complex sets of Site Elements. Suitable messages are provided in this case and a smaller selection set of Site Elements should be used.

Creating Sections

The following is the procedure for creating Section Elements.

- Select the Reference Elements to place the section along. It is recommended that the Power Selector be used in Single element select mode. If the elements are chosen in an order which would easily accommodate chaining the elements together the resulting sections will be simplified.
- Determine if a cell or table of horizontal and vertical offset is desired and fill tables.
- Set the Direction – this will indicate the side of all the elements for which to place the Section Elements.
- The Site Modeler attempts to chain all the elements together and then determine the side based on a single data point. If the element selection does not easily facilitate this process, it may be necessary to hit any key and reselect a smaller set of elements.
- Once the site modeler has chained the elements together, a dynamic display will indicate which direction would correspond to positive offsets.
- Set the Feature type and press **Apply**.
- The Section Elements are created and added to all the Objects that contain the Site Elements for which they were placed.

Manipulating Sections

The Manipulate Section has three tools, as detailed in the table below.

Reverse Section

This button flips all sections currently selected to the other side of the Site Elements that they are attached.

Remove Section

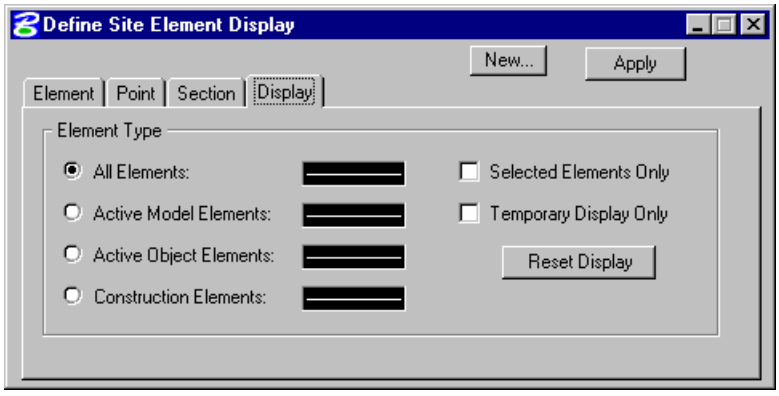
This button removes the sections from the selected Site Elements. Since Section Elements are locked, this is the only way to remove them from graphics.

Query Section

This button queries a Site Element and populates the offset table with the values from the Site Element.

Display Tab Options

When the Display Tab is selected, the dialog depicted below is displayed.



Element symbology is supported in this dialog for the following elements:

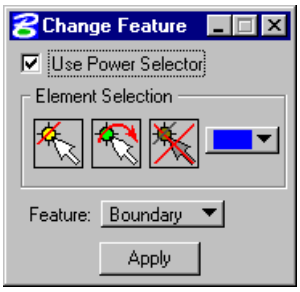
- All Elements
- Active Model Elements
- Active Object Elements
- Construction Elements
- Activate the desired elements option. Options are also supported for:
- Selected Elements Only
- Temporary Elements Only

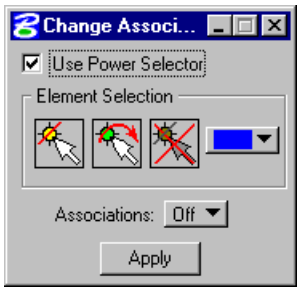
Both of these toggles may be active simultaneously. The Reset Display button redisplays the Elements within the selected group in the desired symbology.

Change Site Element Feature & Associations

These two element tools can be used to change the feature type of a previously defined site element or group of site elements as well a turn on or off the site element associations.

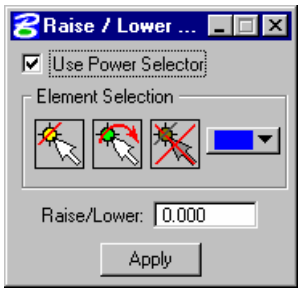
The dialogs below will appear when their respective tool is selected.





Raise/Lower Site Elements

This tool is utilized to change the elevation(s) of an existing element.



Modify Site Elements

A variety of tools are supported to modify existing site elements.

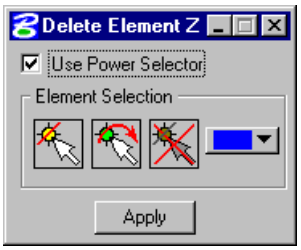
The tools (from left to right) are:

- Site Element Extend Line
- Site Element Extend Arc
- Site Element Extend to Intersect
- Site Elements Extend Both to Intersect
- Site Elements Fillet
- Site Elements Chamfer

All tools function identically to their generic MicroStation counterparts, except the Extend to Intersect and Extend Both To Intersect. These two commands have elevation options to be considered. The Modify Element Tools are only available within a 2D MicroStation Design File.

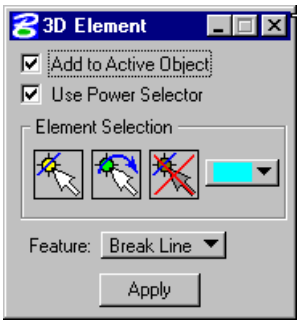
Delete Site Elements

This tool is used to delete a Site element that is associated with a MicroStation Element.



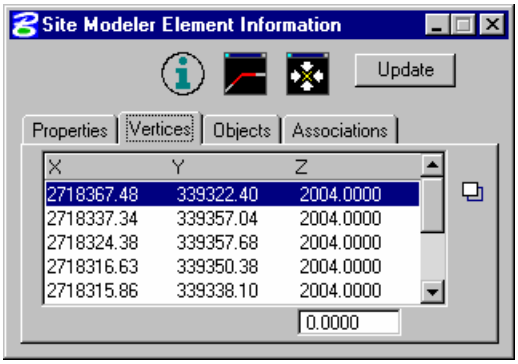
3D Element

Elements placed at the correct X, Y, Z coordinates within a MicroStation 3D design file can be utilized within this dialog. When invoked, the dialog depicted below is displayed.



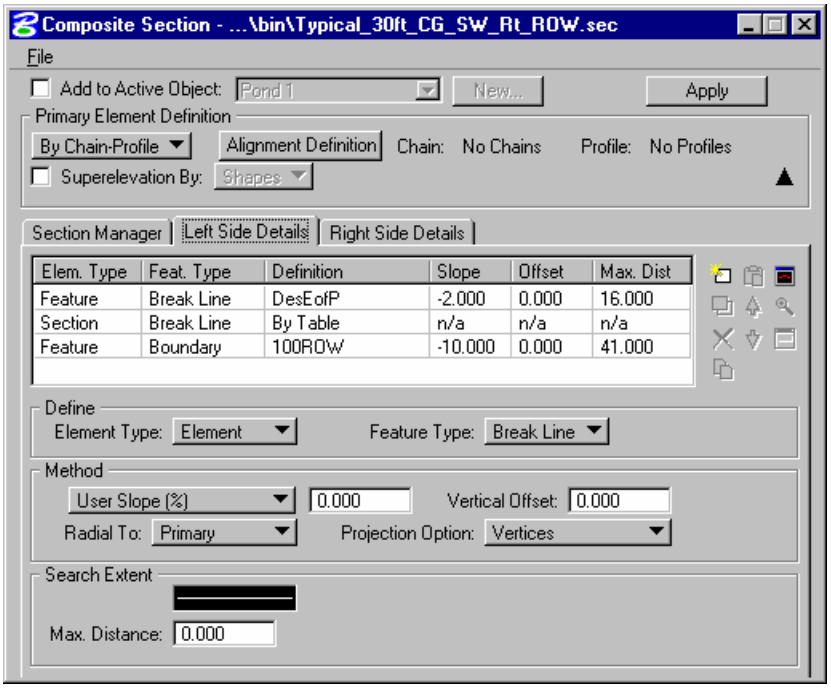
Site Element Information

The Information Tool can be used to display coordinate information, Feature type, stroking information and Member Objects and Site Element Association information. Most importantly however, the Information Tools can be used to quickly change element information when needed. The dialog below shows one option of the Information Tool. Each vertices of the Site element can be modified to achieve the desired result.



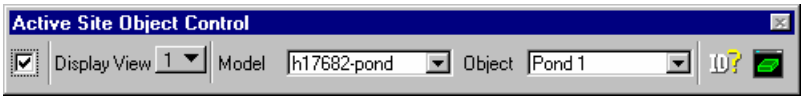
Composite Section Tool

The Composite Section Tool is used to create multiple Site Elements which will be grouped into a Site Object. This tool can be useful for creating Roadways, Channels, Levees, etc.



Active Site Object Control Toolbar

Upon opening an existing GEOPAK Site Project or creation of a new project, the Active Site Object Toolbar is displayed as shown below.



The tool bar is resizable and dockable and remains open throughout the entire Site Modeler session.

Display View — When the toggle to the left of Display View is activated, the Active Model and/or Active Object are displayed, based on their respective visualization settings.

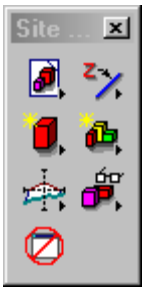
Model — The active model is displayed. Selection of any other model from the list changes the active model. When the model is selected, all Objects which belong to the model are listed in the Object listing to the right. When <All Objects> is selected, all objects in the project are listed. When <Orphan Objects> is selected, all objects which currently do not belong to a model are listed.

Object — The Object list changes dynamically based on the Model selection.

ID?  — Pressing the **ID** button and then identifying any Site Modeler object highlights the selected element and invokes the Select Object dialog depicted below.

Center Window On Object — After setting the Object list to the desired Object, then pressing the Window Center on Object button, the Site Modeler window centers (and fits the object) to the Display View.

Main Toolbar



The **Tools** menu selections provide access to all **Tool Boxes**. The Main Tools tool frame is depicted below and accesses the other tool boxes.

Each icon in the tool frame (except single tools) is a tool box that one can "tear off" to become a "tool box." The individual tool boxes (except those which have single tools) can be docked and resized. There are seven tool boxes, with the following titles (order down first column, then down second column):

- Project tools
- Object tools
- Composite Section
- Exit
- Elements tools
- Model tools
- Analysis tools

If the function of an icon is not apparent to the user, position the cursor on the icon. A detailed description is displayed in the status bar and a tool tip (flyover) appears.

Lab 17: Pond Design

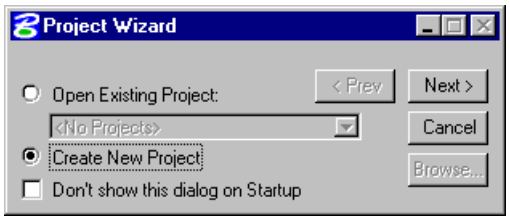
Creating a Site Modeler Project

Step 1. Execute **C:\data\geo\VDOT\drain1\LAB17_V8.EXE**.

Step 2. Open the MicroStation File **C:\Data\Geo\VDOT\Drain1\H17682.dgn**.

Step 3. Access the **Project Wizard** from the **Applications > GEOPAK Site > Site Modeler > Site Modeling** pull down menu or from the **Site Modeler** Icon on the GEOPAK Site Tool Frame shown.

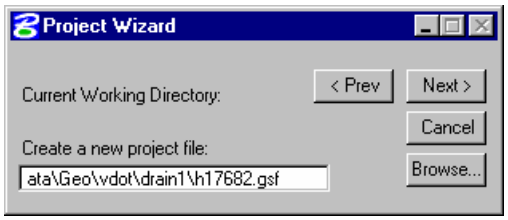
Step 4. Select the **Create New Project** button and then press the **Next>** button to continue.



Step 5. The Project Wizard dialog will appear filled out as shown.

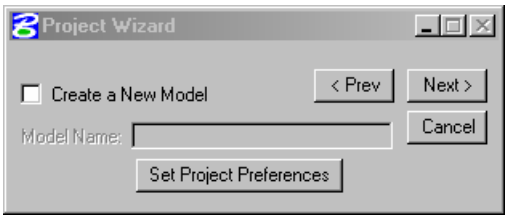
Create a New Project file: `H17682.gsf`

Press the **Next>** button to continue.

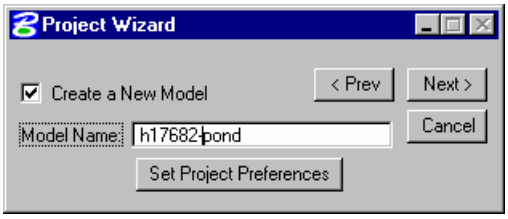


Defining the Project Preferences

Step 1. Continuing with the **Project Wizard** dialog as shown.

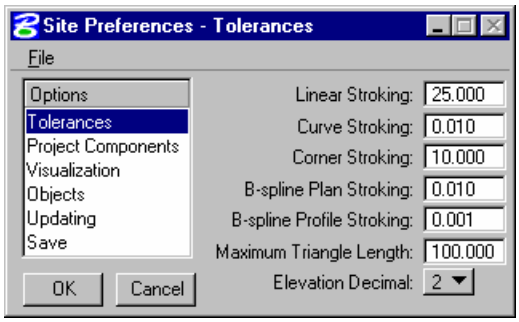


Fill out the **Project Wizard** dialog as shown below. When complete, click the **Set Project Preference** button.



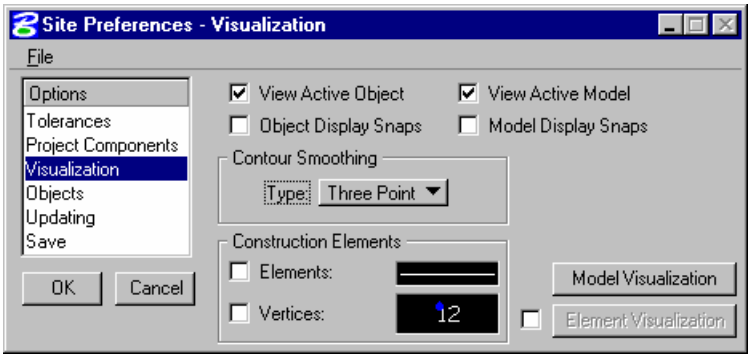
Create a New Model:	Toggle On
Model Name:	H17682-pond

Step 2. The **Site Preferences** dialog will appear. The **Tolerances Option** Item is highlighted. Set each field as shown.



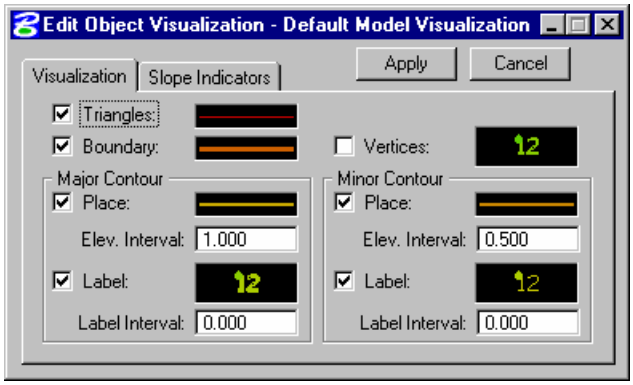
Linear Stroking	25.000
Curve Stroking:	0.010
Maximum Triangle Length:	100.00
Elevation Decimal:	2

Step 3. Highlight the **Visualization** item.

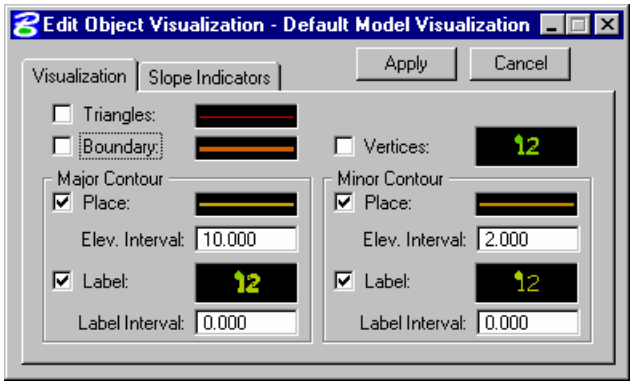


Toggle on the **View Active Object**, **View Active Model**, and **Smooth Contours** options.

Step 4. Click the **Model Visualization** button in the lower right corner of the **Site Preferences** dialog. The **Edit Object Visualization** dialog will appear.

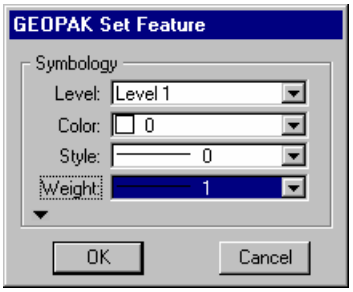


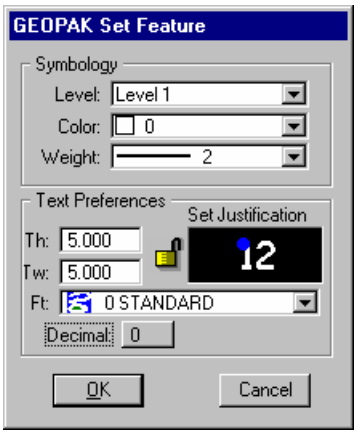
Step 5. Fill out the dialog as shown.



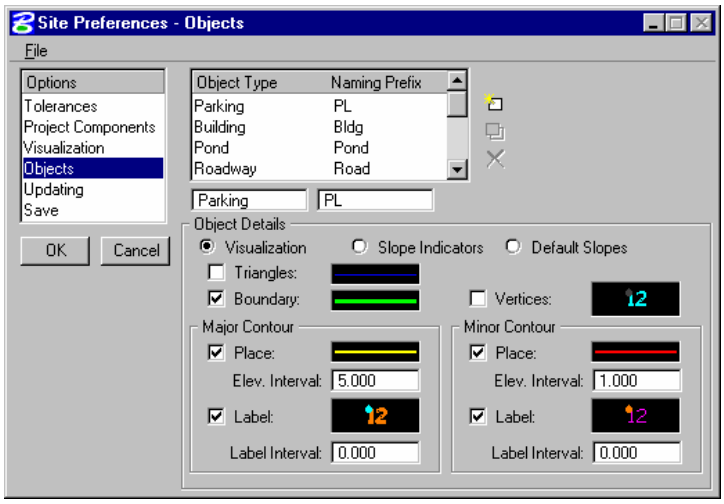
To change the symbology of each item, just double click on the sample graphic. The **GEOPAK Set Feature** dialog will appear. Make the appropriate changes and then Click the **OK** button. When finished with all items click the **Apply** button at the top of the dialog.

Below are two samples of the **GEOPAK Set Feature** dialog.

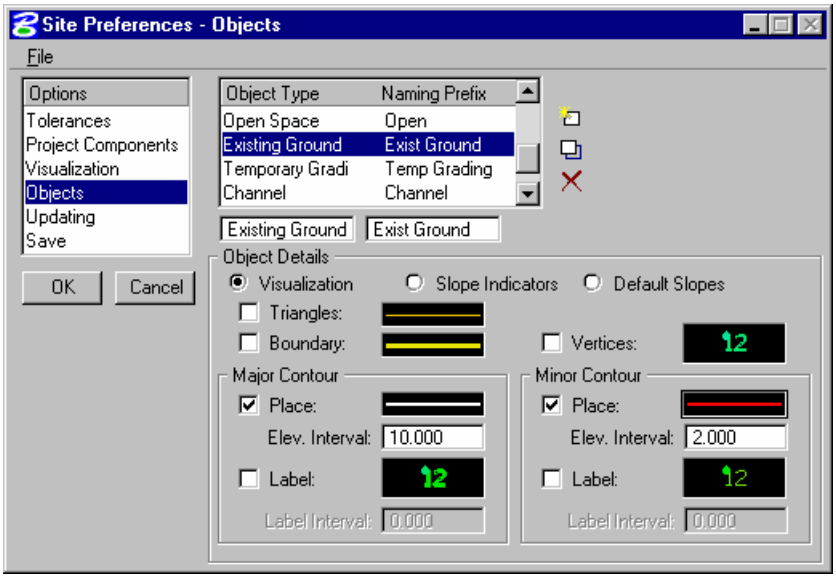




Step 6. Highlight the **Objects** item from the **GEOPAK Site Preferences** dialog. The dialog changes as depicted below.

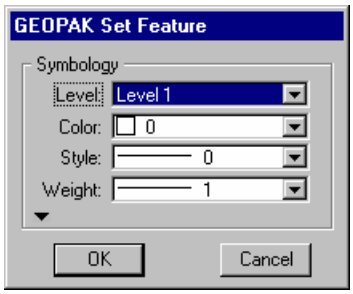


Highlight the **Existing Ground Object** Type and set the Visualization preferences as shown.



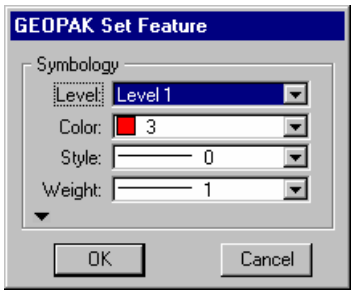
See the **GEOPAK Set feature** sample dialogs.

Major Contour:



Elev Interval: 10.000

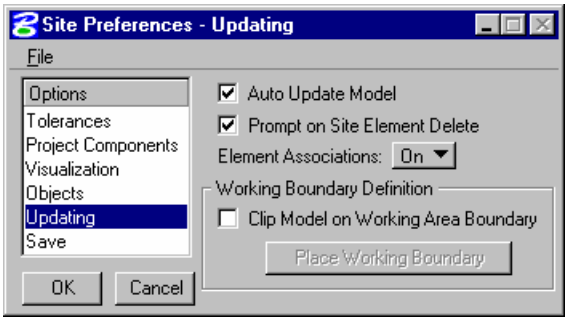
Minor Contour:



Elev Interval: 2.000

Note: To change the symbology of each item, just double click on the sample graphic. Once the appropriate changes have been made click the **Modify** Icon to save your changes. The *Modify* Icon is located next to the *Object Type/Naming Prefix* List.

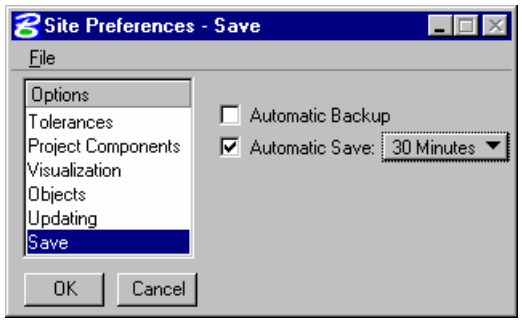
Step 7. Highlight the **Updating** Option Item as shown below.



Step 8. Highlight the **Save** Option Item as shown.

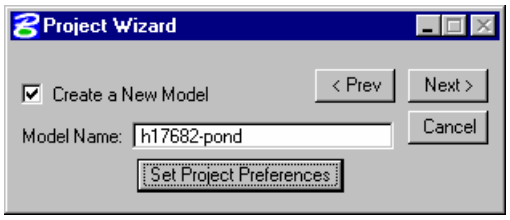
Toggle On the **Automatic Save** option and set the interval to **30 `Minutes`**.

Click the **OK** button to save the preferences and go back to the **Project Wizard** dialog.



Completing Project Set-up

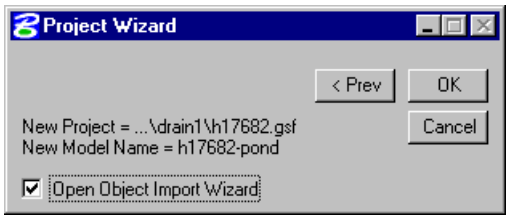
Step 1. Once back at the Project Wizard dialog, press the **Next>** button.



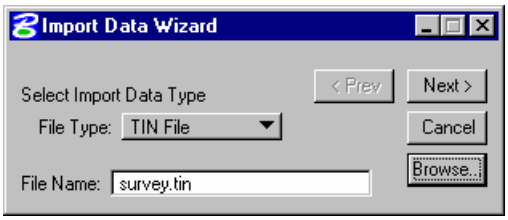
Step 2. The New Model Name and Project Name should appear as shown.

Toggle On the **Open Object Import Wizard** option.

Click the **OK** button when finished.



Step 3. We will now begin with the **Import Data Wizard**.



Select the **TIN File** as **the Import Data Type** option as shown in the dialog.

Fill out the **File Name** field as shown or use the **Browse** button to select the Tin file:

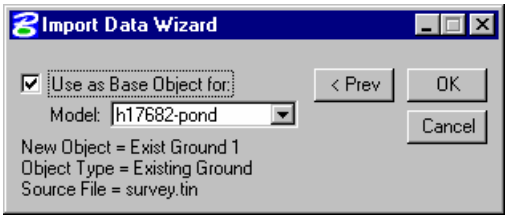
C:\Data\Geo\VDOT\Drain1\Survey.tin.

Press **Next>** when done.

Step 4. Fill out the **Import Data Wizard** dialog below as shown. By selecting the **Ground Object Type,** the **Enter New Object Name** field is automatically filled in.

Click the **Next>** button when finished.

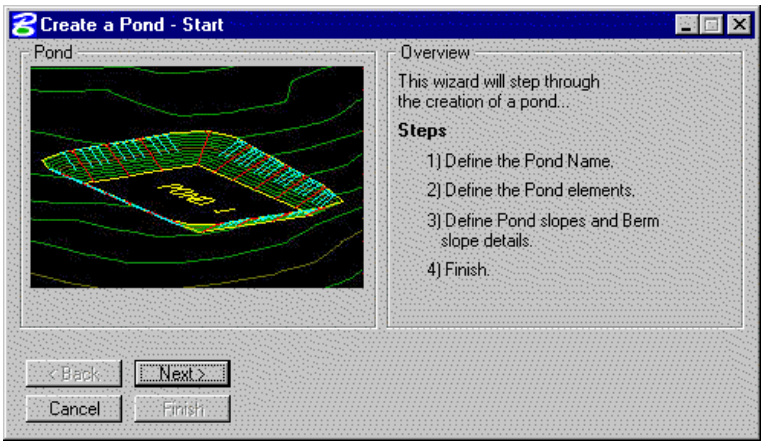
Step 5. The final step in the **Import Data Wizard** is to set the object Exist **Ground 1** as the **Base Object** for the **Model H17682-Pond**.



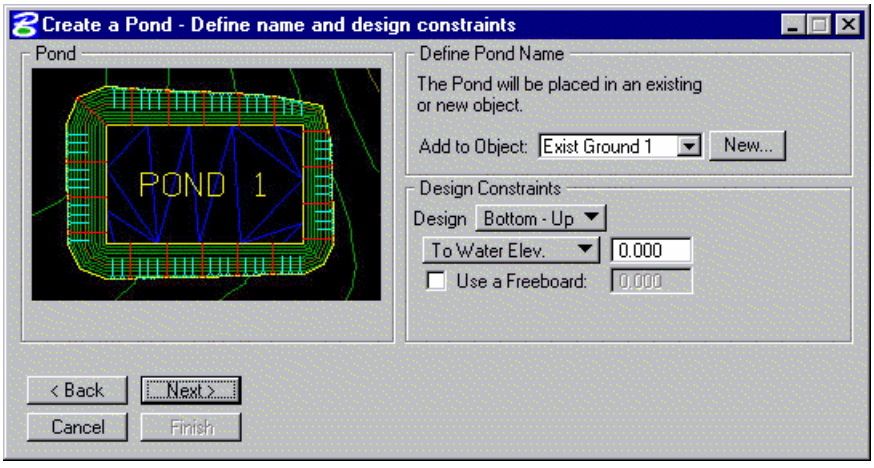
Click the **OK** button to initiate the drawing of the **Active Model, H17682-Pond**, and the **Active Object**, Exist **Ground 1**, into View 1 of MicroStation.

Creating a Pond Object

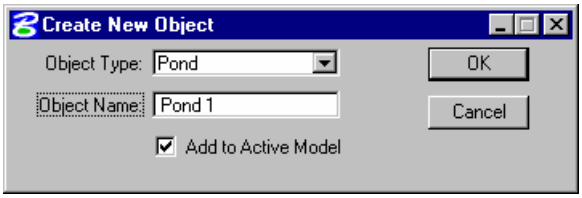
Step 1. Access **Modeler > Tools > Pond Design** in the MicroStation menu.



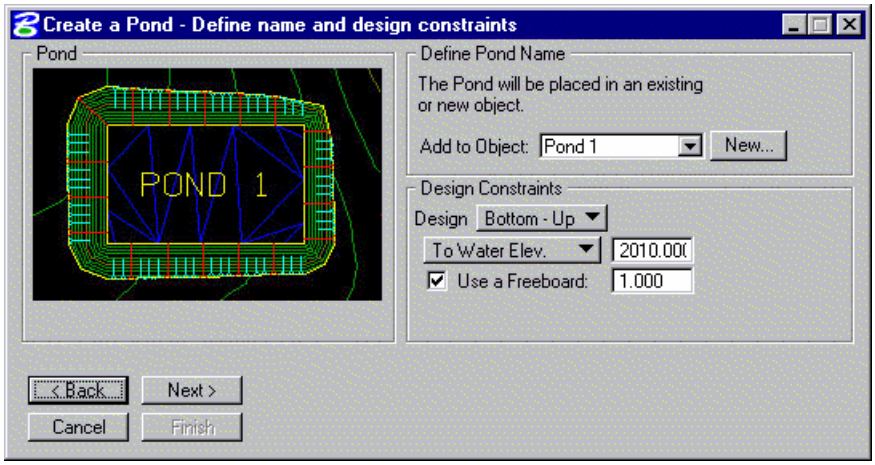
Step 2. Click **Next>.**



Step 3. Click **New** to create a new Pond object.

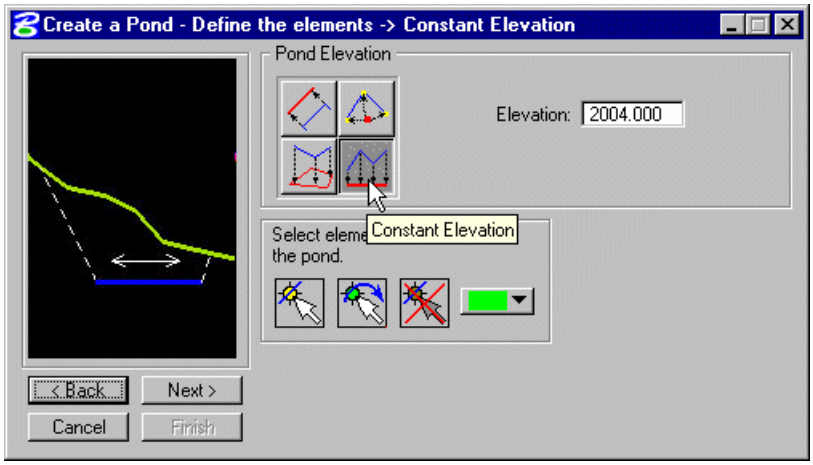


Step 4. Define the design constraints as shown below.

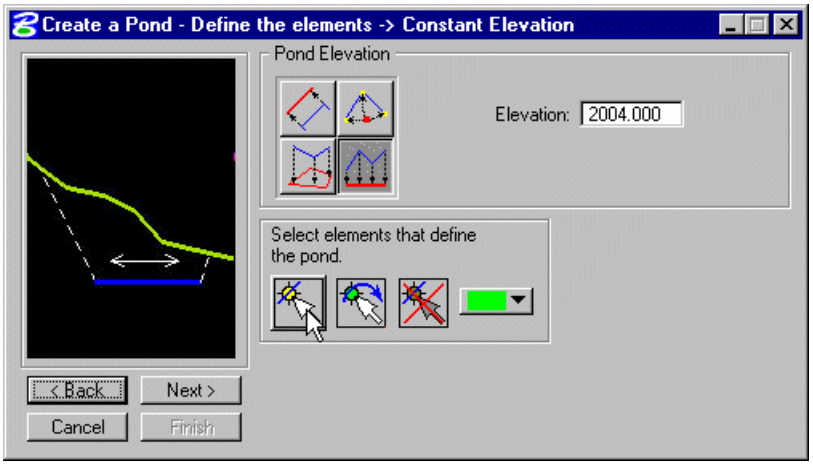


Design:	Bottom-Up
To Water Elevation:	2010.00
Use a Freeboard:	1.00

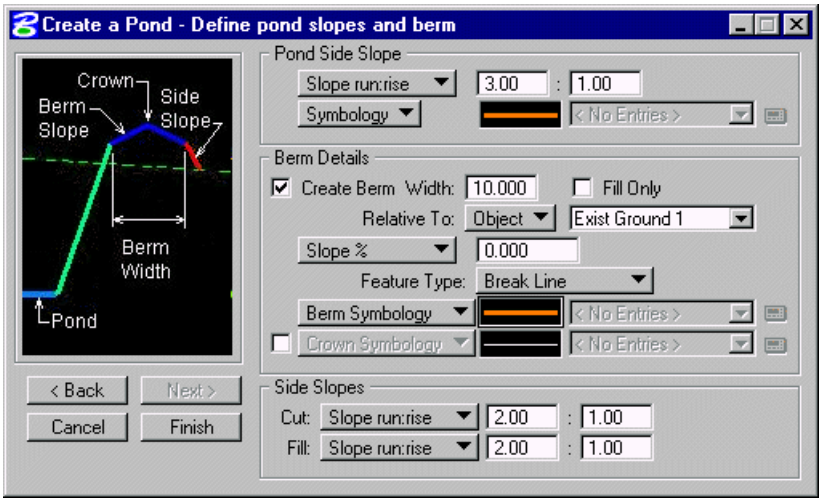
Step 5. Click **Next>** and setup the pond bottom at a Constant Elevation of 2004.0.



Step 6. Select the MicroStation element that represents the pond bottom, and click **Next>.**



Step 7. Setup the Pond Side Slopes, Berm and Side Slopes as shown in the dialog below.

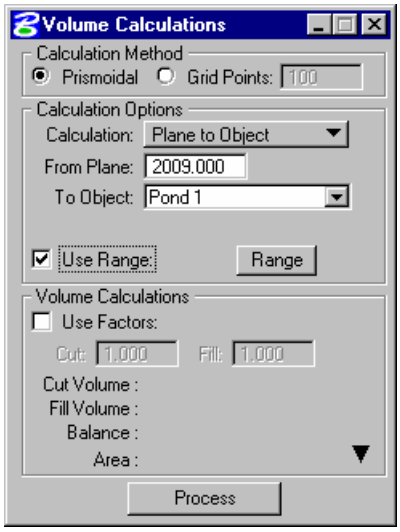


Step 8. Click **Finish** to build the pond.

Analyzing the Pond Volume

Step 1. Access the **GEOPAK Volume Calculations** Tool from the **Modeler > Analysis > Volumes** pull down menu or from the icon shown.

Step 2. Fill out the dialog as shown.



Calculation Method:	`Prismoidal`
Calculation:	`Plane to Object`
From Plane:	`2009.000`
To Object:	`Pond 1`
Use Range:	`Toggle On`

Step 3. Click the **Range** button.

The **Volume Elevation Range** dialog is depicted below.

Set the **Auto Range Options** portion of the dialog as shown.

Range: `Auto Range Increment`

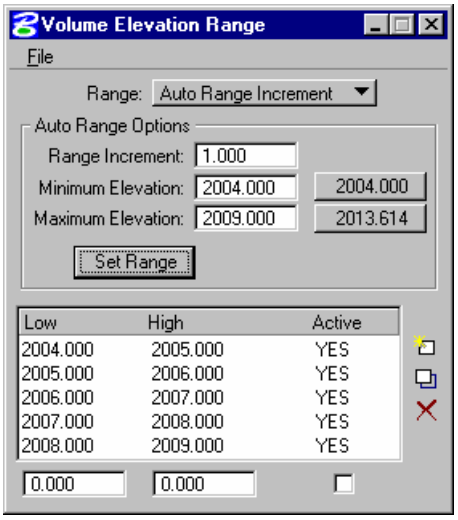
Range Increment: `1.000`

Minimum Elevation: `2004.000`

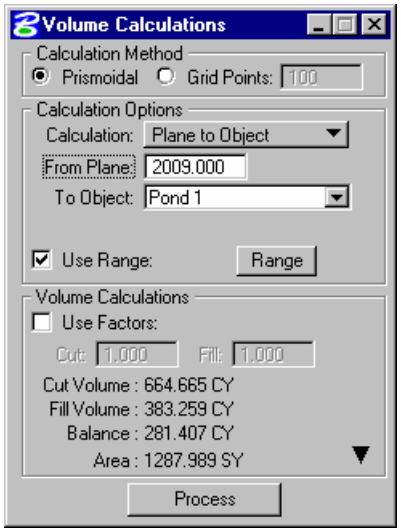
Maximum Elevation: `2009.000`

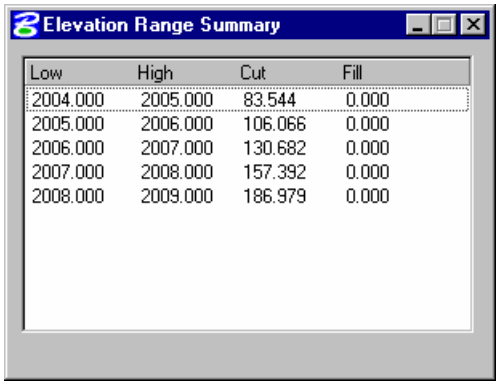
Click the **Set Range** button to fill out the bottom portion of the dialog.

Close the dialog when you are finished.



Step 4. Click the **Process** button on the **GEOPAK Volumes Calculation** dialog. The following report is generated. Close the dialog boxes when done.





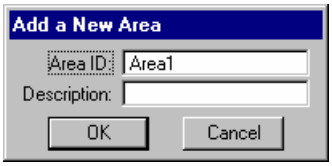
Computing Time of Concentration

During this section we will compute the time of concentration for a drainage area using the Seely formula under the Rational Method.

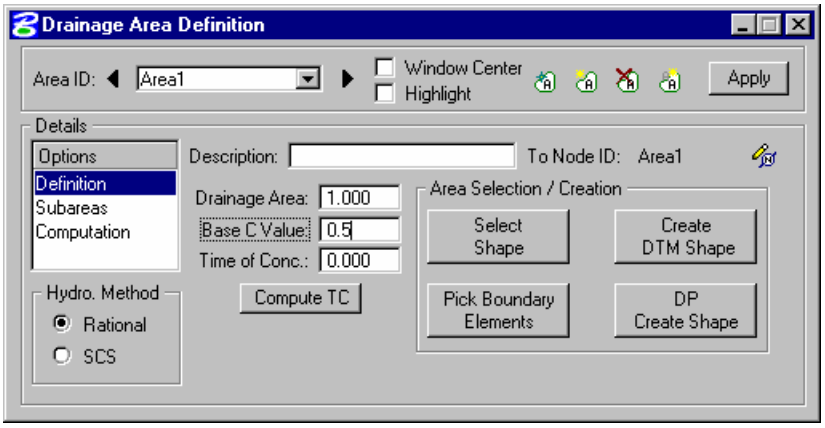
Step 1. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

Step 2. Access the drainage project **h17682.gdf** by selecting **Drainage > Project > Open** from the GEOPAK Drainage pull down menu.

Step 3. Select **Add** from the **Drainage > Components > Area** pull down menu in GEOPAK Drainage.



Step 4. Fill out the Drainage Area Definition dialog as shown.



Drainage Area ID:	Area1
Description:	Drainage Area for Routing
Hydrologic Method:	Rational
Drainage Area:	1.00
Base C Value:	0.5

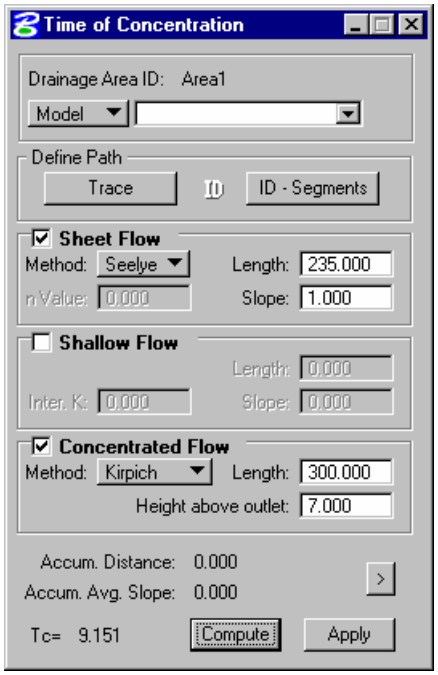
Click the **Apply** button when finished.

Step 5. Click the **Compute TC** button located in the middle of the Drainage Area Definition dialog.

The Drainage Area Definition dialog will temporarily minimize while we work on the Time of Concentration.

We will manually enter the hydraulic characteristics of sheet flow and concentrated flow.

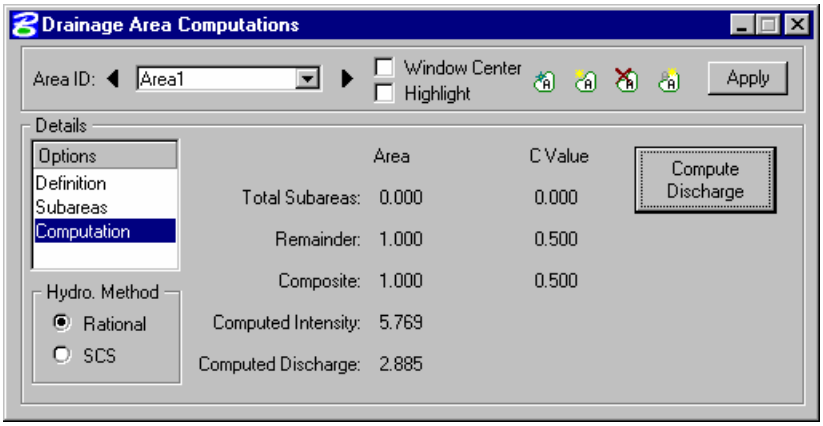
Fill out the dialog as shown.



Sheet Flow:	Toggle On
Method:	Seelye
Length:	235.00
Slope:	1.00
Concentrated Flow:	Toggle On
Method:	Kirpich
Length:	300.00
Height above outlet:	7.00

Click the **Compute** button and then click **Apply** to return to the **Drainage Area Definition** dialog.

Step 6. Highlight **Compute** in the **Drainage Area Definition** dialog and press the **Compute Discharge** button.

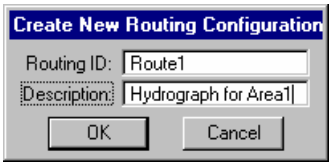


Step 7. Click **Apply** and close the **Drainage Area Definition** dialog.

GEOPAK Routing (Create the Runoff Hydrograph)

During this section we will create a runoff hydrograph according using the Rational Method for the previously created drainage area.

Step 1. Select **Add** from the **Drainage > Components > Routing**. Create a new routing configuration.

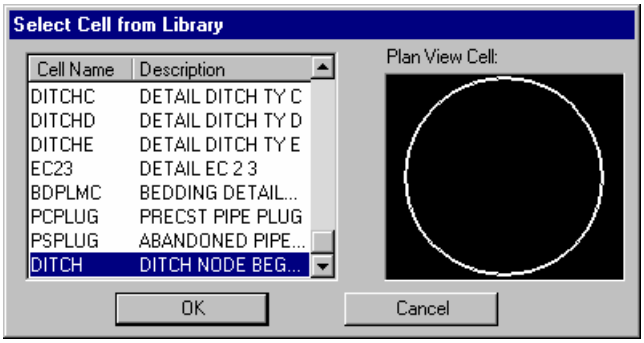


Step 2. Select the **Runoff** icon (1st icon located in the top middle of the dialog) and highlight the **Definition** item in the Options list.

Select 'Area1' as the **Area ID.**

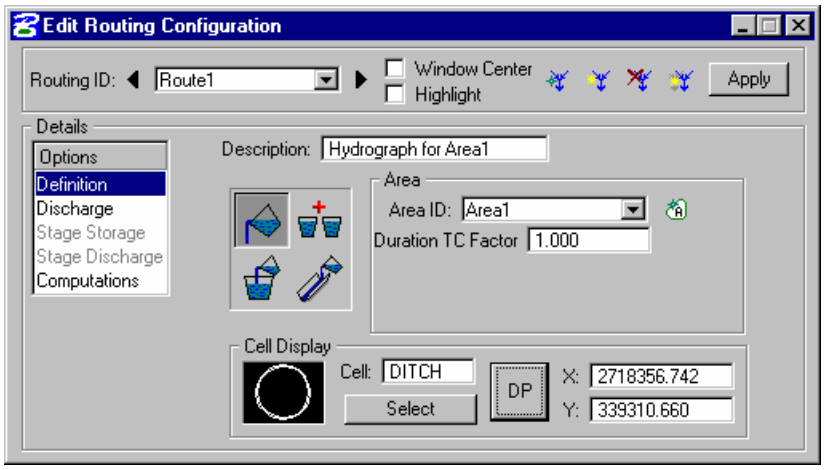
Step 3. Keyin the storm **Duration TC Factor** = 1.

Step 4. Click the **Select** button and select the Ditch cell.



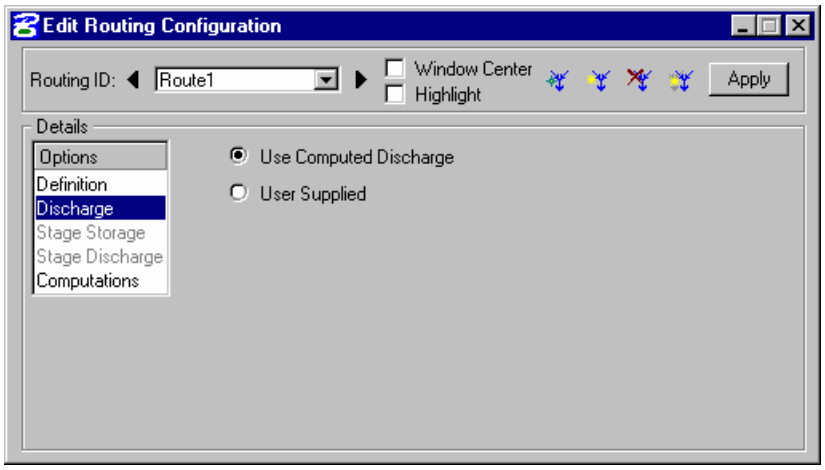
Click **OK** to continue.

Step 5. Click the **DP** button and data point in MicroStation view window 1 in the area where the pond is located.



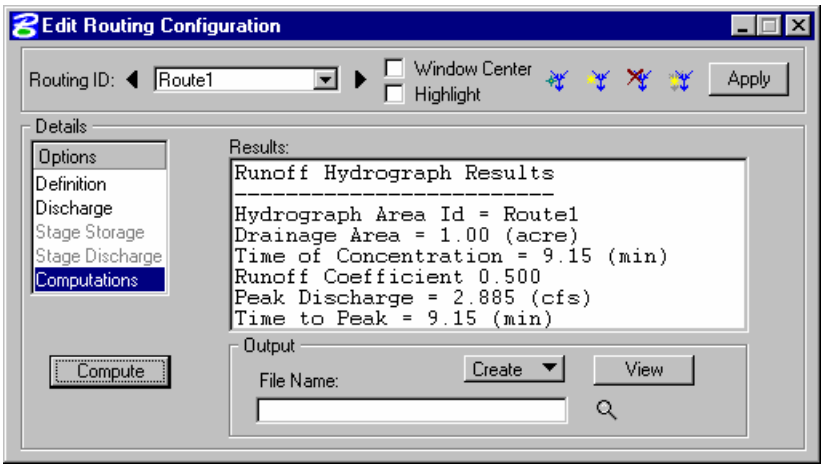
This will place a cell that graphically represents the routing configuration in MicroStation.

Step 6. Highlight the **Discharge** item in the Options list.



Select the **Use Computed Discharge** radio button.

Step 7. Highlight the **Computations** list item in the Options list.



Click the **Compute** button.

Step 8. Click **Apply** to save this configuration. Review the results.

GEOPAK Routing (Create a Reservoir Routing)

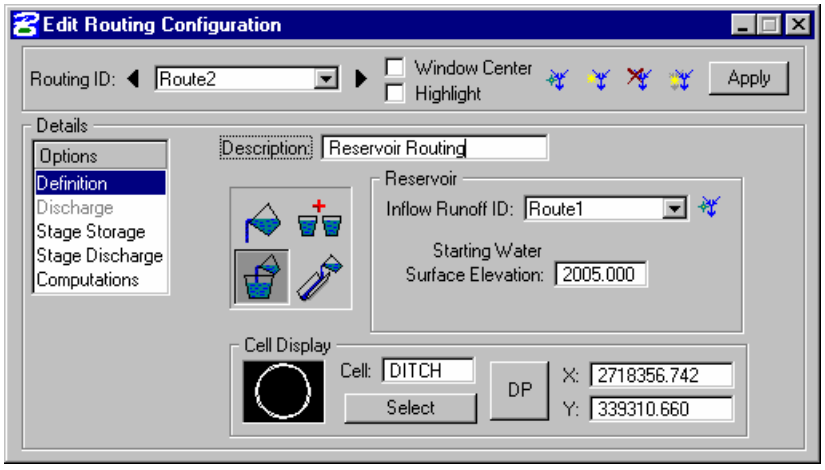
In this section we will create a reservoir (pond) routing configuration which we will use to route our runoff hydrograph for Area 1.

Step 1. Click the **Add Routing** button on the Edit Routing Configuration dialog.

Fill out the dialog as shown.

Click **OK** to continue.

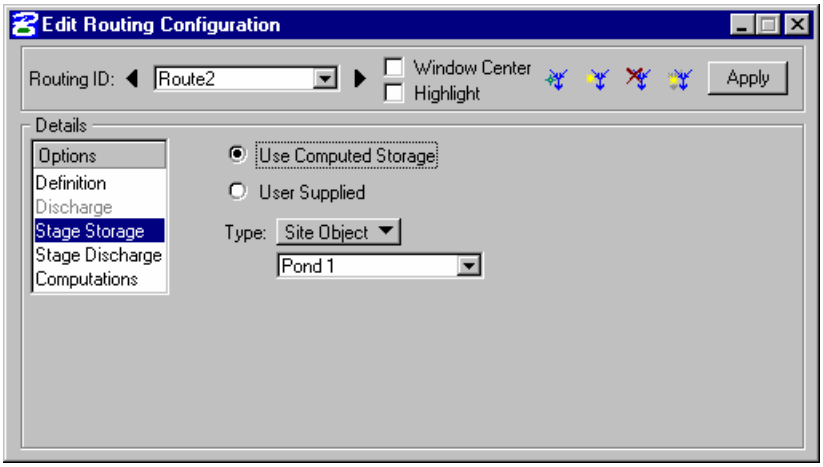
Step 2. Select the **Reservoir** icon and highlight the **Definition** list item from the Options list.



Select **Route1** for the Inflow Runoff ID and set the Starting Water Surface Elevation at **2005.00**.

Step 3. Click **DP** and data point for the location of the orifice.

Step 4. Highlight the **Stage Storage** list item from the Options list.



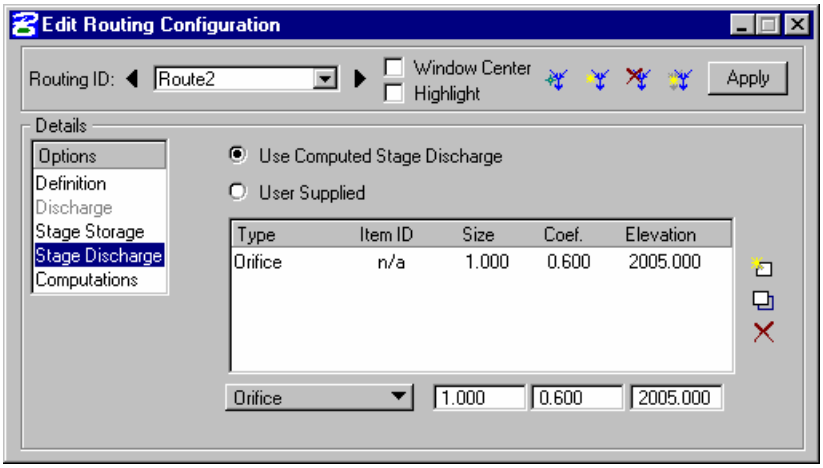
Select **Use Computed Storage** radio button.

Set the **Type** option to **Site Object** and select **Pond 1**.

Note: We were able to select a Site Object because in Geopak Drainage under, **Drainage > Project > Preferences > Project Components** we have previously included the name of the Site Modeler project: H17682.gsf.

Step 5. Highlight the **Stage Discharge** list item from the Options list.

Set the type, size, coef., and elevation options as shown.



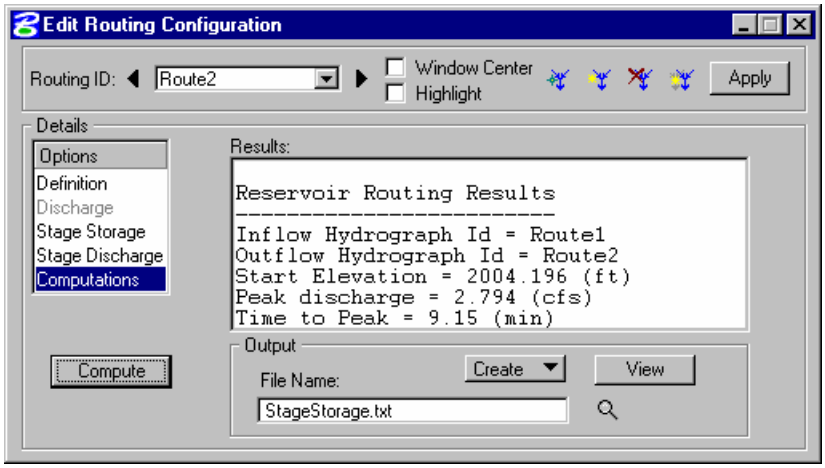
Type:	Orifice
Size:	1.00
Coef:	0.60
Elevation:	2005.00

Step 6. Highlight the **Computations** list item from the Options list.

Click the **Apply** button.

Create an output file called StageStorage.txt

Click the **Compute** button.



Step 7. Click the **View** button to review the Routing calculations.

18 Importing Stream Geometric Data to HEC-RAS

Objective:	Provide the user with a general understanding of GEOPAK Cross Section Reports and their capability to generate geometric information for the HecRas program.
Goal:	Create stream cross sections and import them to the HecRas software.
Prerequisites:	Completion of an existing ground Digital Terrain Model.
Additional References:	GEOPAK Cross Sections Online Help

Introduction

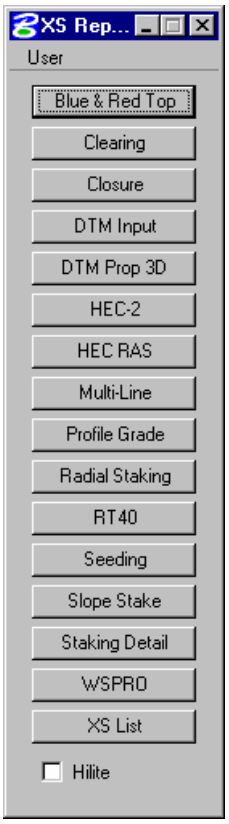
The creation of a stream or river cross sections requires the usage of different tools within the Geopak software.

The Geopak Cross Section Report tool enables us to extract information from the stream cross sections in a format accepted by the HecRas software.

The designer should be able to place a basic alignment along the stream and generate existing ground cross sections. These cross sections will be used in the Geopak Cross Section Report to format the data accordingly to HecRas Gis format.

XS Reports

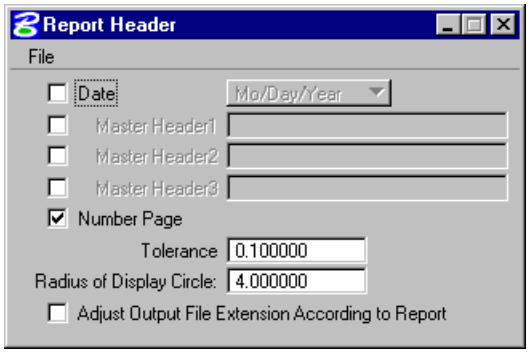
Objectives	Create various cross section reports: - for plan use (design elements and quantities) - as input for other programs and/or applications of GEOPAK - for construction layouts. Create and draw construction limits in the plan view file.
Project Manager	Reports and XS Quantities Limits of Construction
Tools 	
Menu Bar Application	GEOPAK Road > Cross Sections > Cross Section Reports GEOPAK Road > Cross Sections > Construction Limits



The GEOPAK Cross Section Report Utility can extract up to sixteen different reports from original and design cross-sections. For each report generated, the user must set the parameters of the existing and/or design cross sections. GEOPAK also provides an option to make custom headers for each of the reports via the **User** pull down menu.

Custom Header

From the XS Report dialog box, select **User > Preferences.** A Report Header dialog box will appear with all options ghosted out. To activate the individual fields simply toggle on the box next to the desired field. Once you have completed the dialog box, the information will be saved as an .hdr file. This allows for the creation of a separate header for each type of report. The tolerance field determines the maximum gap allowed between cross section elements.



HEC – 2

This process reads cross section elements and formats the information in an ASCII text file suitable for use in the HEC-2 hydraulic program.

HEC RAS

This process reads cross section elements and formats the information in an ASCII text file suitable for use in the HEC RAS hydraulic program.

Lab 18: Importing Stream Geometric Data to HecRas

Creating a Stream Baseline

Step 1. Execute **C:\data\geo\VDOT\drain1\LAB18_V8.EXE**.

Step 2. Open the MicroStation File **C:\Data\Geo\VDOT\Drain1\Survey_river.dgn**.

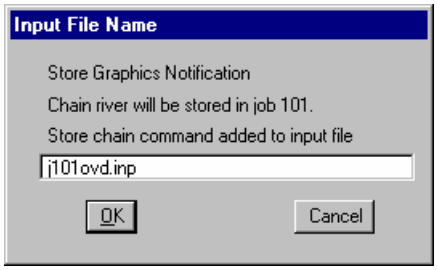
Step 3 Access **Store Graphics** from the **Applications > GEOPAK Road > Geometry > Store Graphics** pull down menu or from the **Store Graphics** icon on the GEOPAK Road Tool Frame shown

Step 4. Enter the information as shown in the dialog below.

Step 5. Press **ID Element** and select each of the lines that represent the alignment of the river stream until the **Store** button activates.

Step 6. Press the **Store** button.

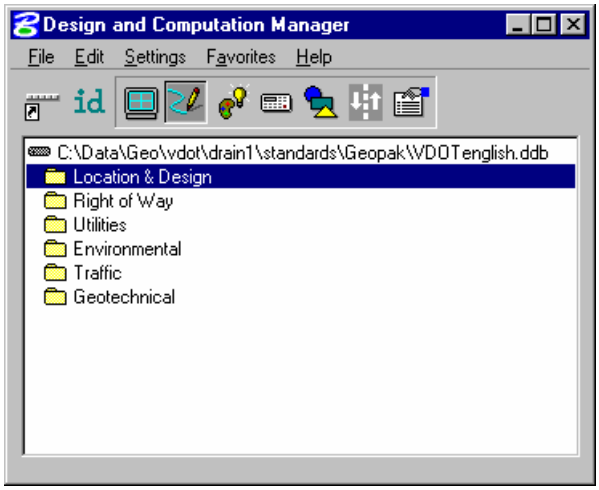
Step 7. Press **OK** at the Store Graphics notification dialog. The alignment will be saved into the geometric database and a copy of the generated commands will be written into the ASCII file j101ovd.inp.



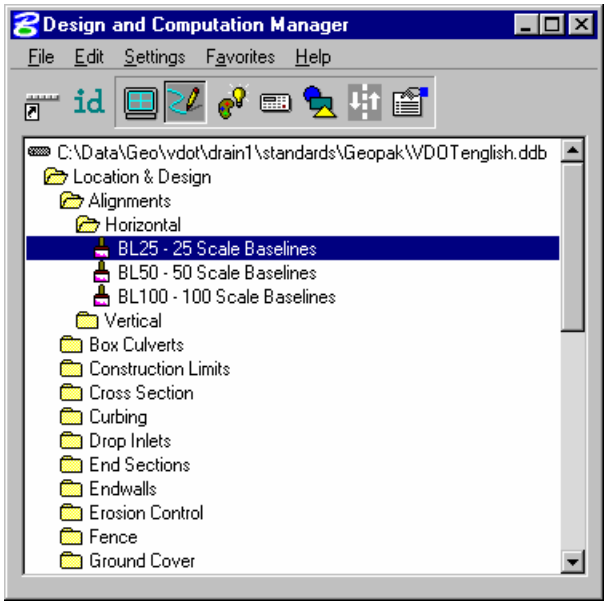
Step 8. Close the Store Graphics dialog and delete the MicroStation lines that represented the stream alignment.

Drawing Alignments

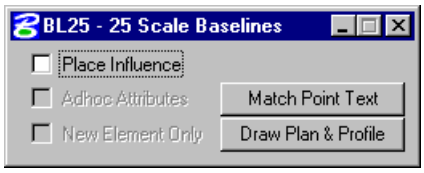
Step 1. Access the Design and Computation Manager from the **Applications > GEOPAK Road > Design & Computation Manager** pull down menu. A secondary dialog box will appear to be used in conjunction with the Design and Computation Manager dialog box.



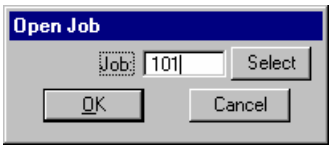
Step 2. Select the item shown in the dialog box below.



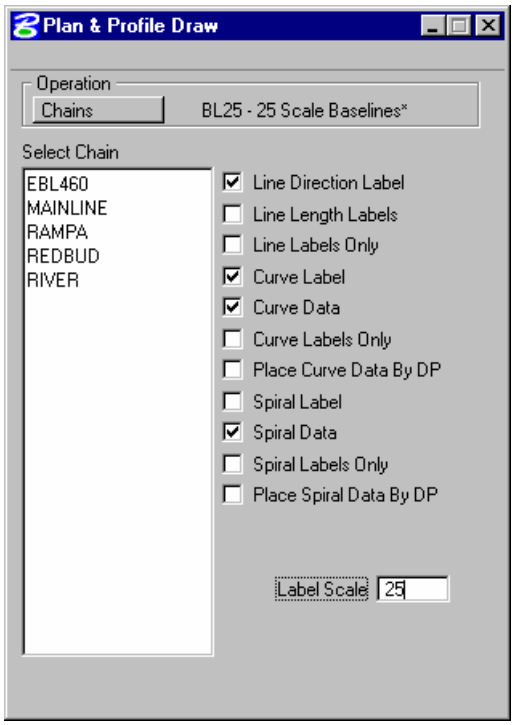
Step 3. Next, select the **Draw Plan and Profile** button from the secondary dialog box:



Step 4. Enter the job number: 101 at the dialog and press **OK**.



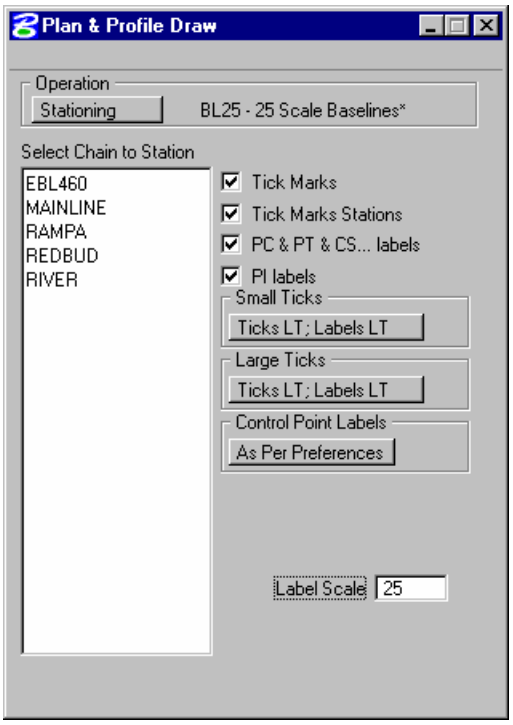
Step 5. Select the **Chains** operation. Each of the chains will be displayed in the list box as shown. Notice that the options that are to be drawn with the chain are already activated. Set the Label Scale to **25**.



Step 6. To draw the chain, click on the name of the chain: **River** in the list box. Only click once!! Each time you ID a chain it will be plotted into the design file.

Step 7. Change the Operation to **Stationing** and modify the dialog settings as shown in the image to the right.

Annotate the River alignment by clicking the chain name in the list box.

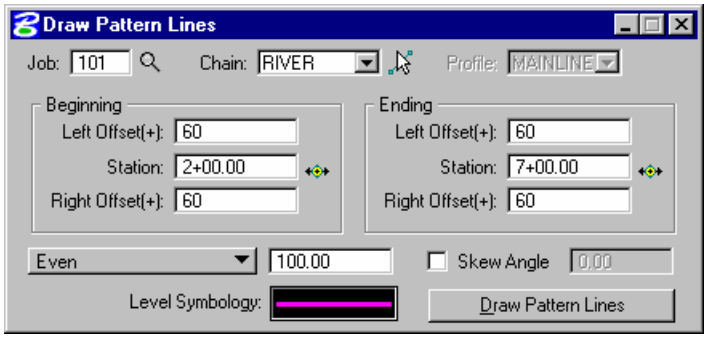


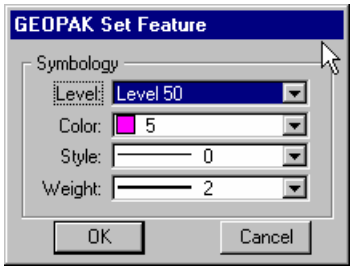
Step 8. Exit the Plan and Profile Draw dialog box and close the D&C Manager

Existing Ground Cross Sections: Drawing Pattern Lines

Step 1. Access **Draw Patterns** from the **Applications > GEOPAK Road > Cross Sections > Draw Patterns by Station Range** pull down menu.

Step 2. Populate the dialog box as depicted below.





Step 3. Click **Apply** to initiate the plotting of the patterns into the design file.

Step 4. Using the same MicroStation symbology place two more parallel lines at each side of the bridge that crosses the stream.

Step 5. Delete the pattern line at station 5+00. For Hec-Ras purposes, a cross section there is not needed.

Step 6. Close the Draw Pattern Lines dialog box.

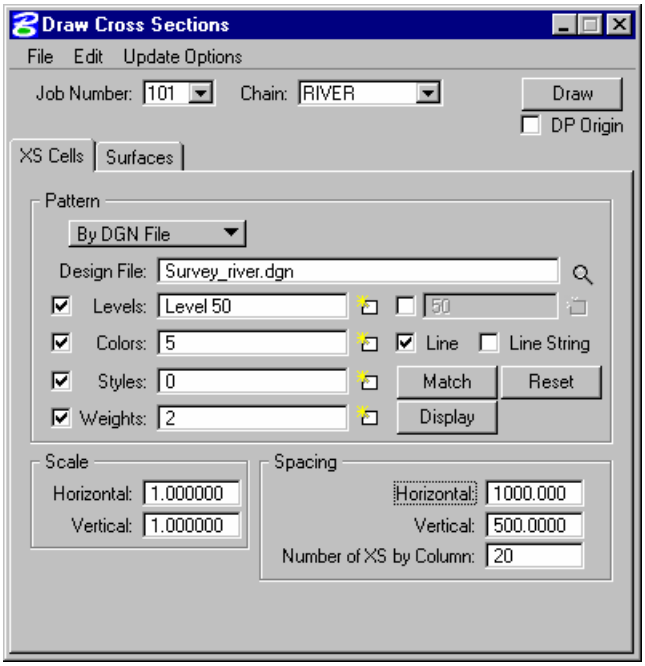
Existing Ground Cross Sections: Generating the Stream Cross Sections.

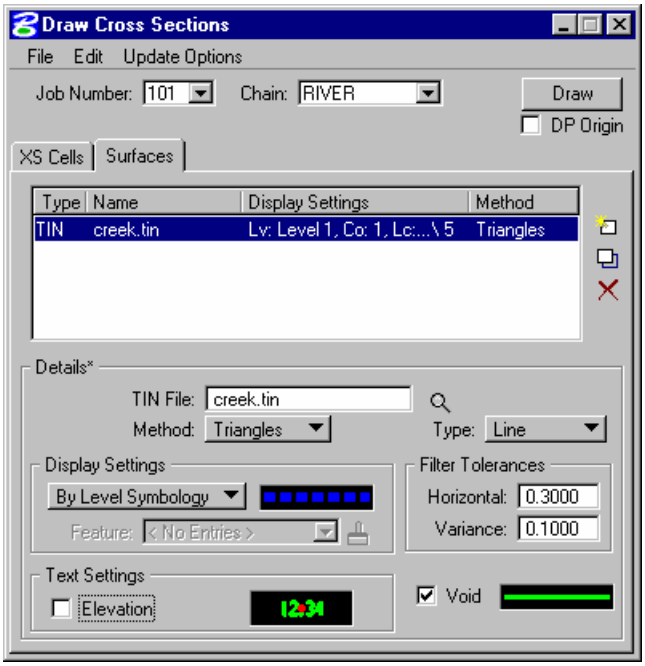
Step 1. Open the MicroStation file **c:\data\geo\VDOT\drain1\xsriver.dgn**

Step 2. Access **Draw Cross Sections from Surfaces** from the **Applications > GEOPAK Road > Cross Sections** pull down menu.

Step 3. Generate the existing ground cross-sections by populating the dialog box as shown on the following dialogs.

Note: The **Display Settings** for the existing ground line are set to level 1, color 1, style 2, weight 5.





Step 4. Press **Draw** to generate the cross sections.

Step 5. Exit the dialog box.

Review Cross Sections

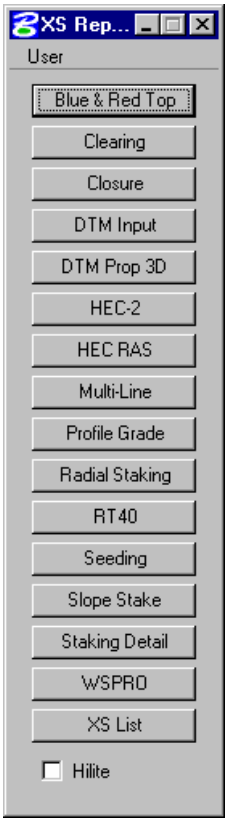
Step 1. Select the **Cross Section Navigator** tool from the **Applications > GEOPAK Road > Cross Sections > Navigator** pull down menu.

Step 2. Use the **Cross Section Navigator** to browse and check your existing cross sections.

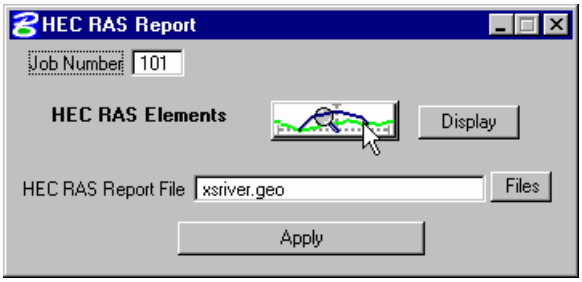
Step 3. Close the XS Navigator dialog box when done.

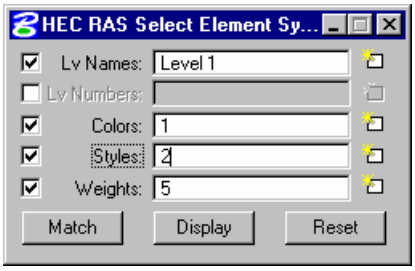
Exporting to Hec-Ras

Step 1. Access the **HEC-RAS** report from the **Applications > GEOPAK Road > Cross Sections > Reports** pull down menu.



Step 2. Populate the dialog box as shown below.



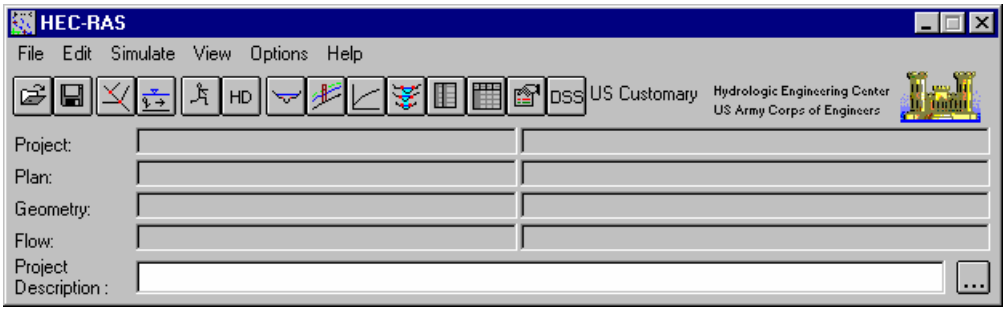


Step 3. Press **Apply** to generate the Hec-Ras cross section geometry file.

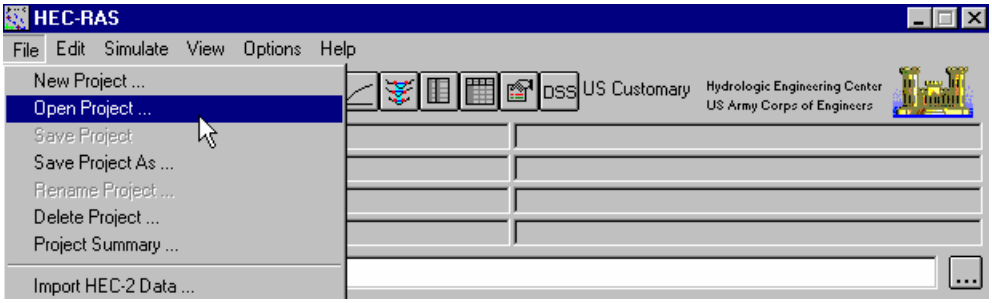
Step 4. Exit MicroStation.

Importing Cross Section Data into Hec-Ras

Step 1. Activate Hec-Ras from **c:\data\geo\vdot\drain1\hecras\ras.exe.**

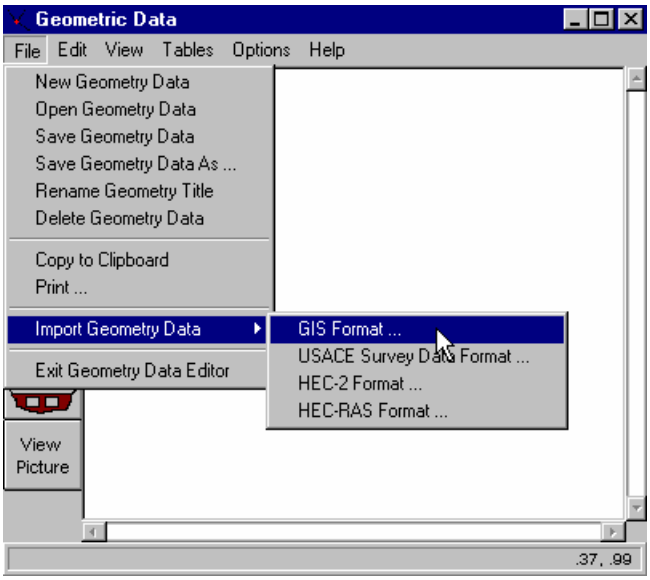


Step 2. Access the included Hec-Ras project: **xsriver.prj** located in the **c:\data\geo\vdot\drain1** directory.



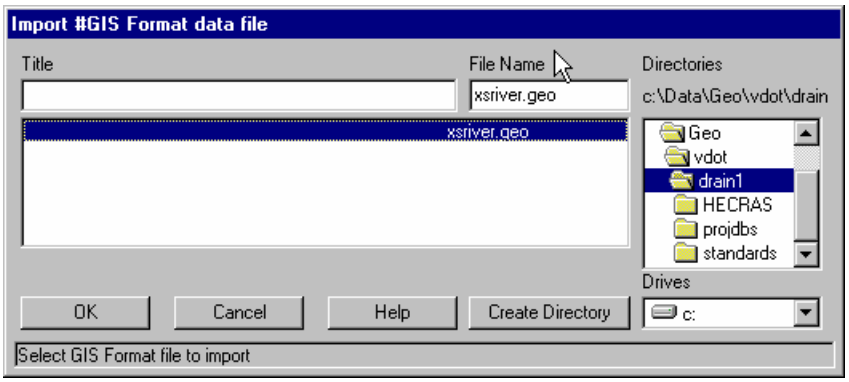
Step 3. Access the Geometric data module of Hec-Ras by selecting **Edit > Geometric Data** from the Hec-Ras pull down menu.

Step 4. Select **File > Import Geometry Data > GIS Format** from the Geometric Data dialog.

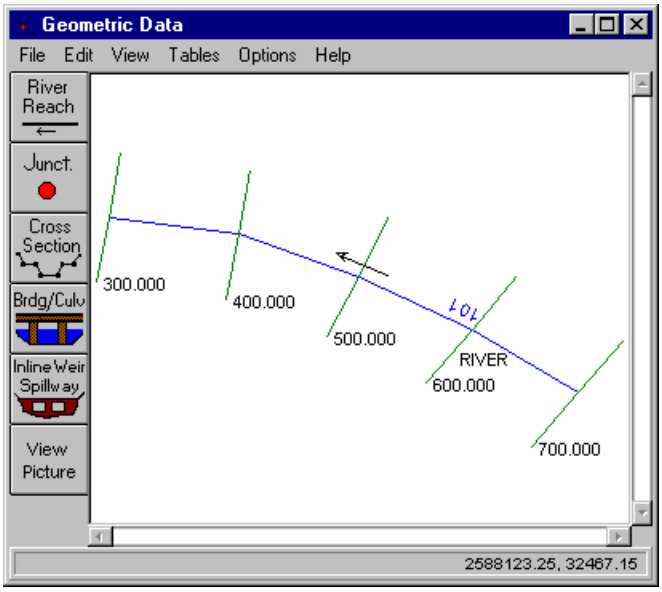


Step 5. Select the file we created from the Geopak cross sections:

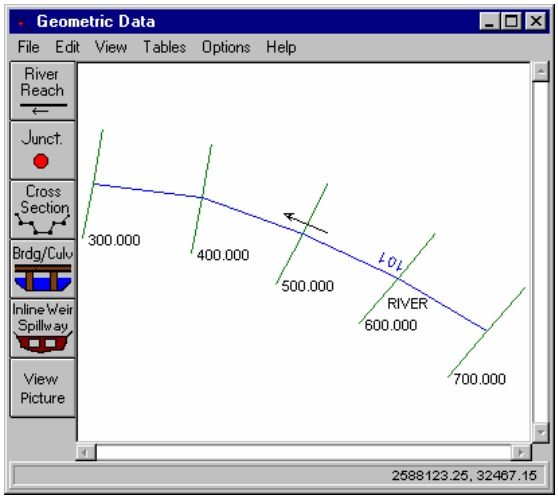
c:\data\geo\vdot\drain1\xsriver.geo

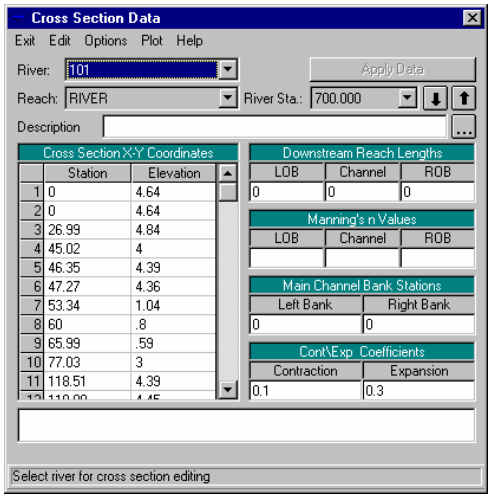


Step 6. Review the alignment and layout of the imported cross section data.

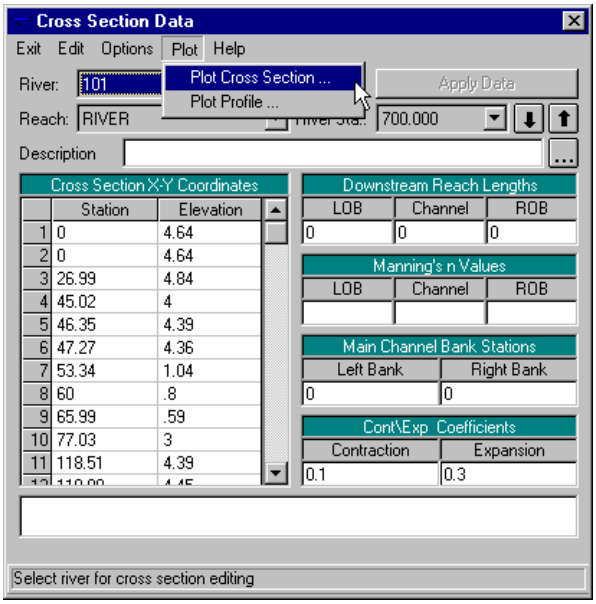


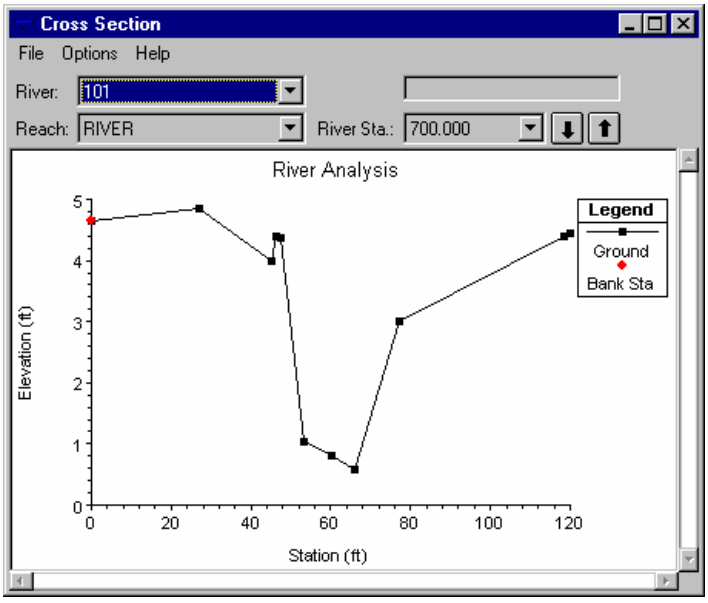
Step 7. Review the generated cross sections by pressing the **Cross Section** button in the Geometric Data dialog box.





Step 8. Review the cross sections graphically by selecting **Plot > Plot Cross Sections** in the Cross Section Data dialog box.





Step 9. Save your project data and exit Hec-Ras

19 Ditch Design

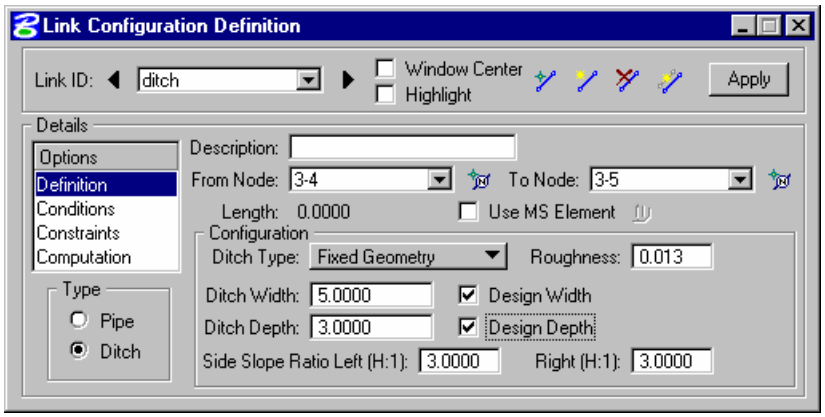
Objective:	Provide the user with a general understanding of how to design and analyze ditches in GEOPAK Drainage.
Goal:	Create a ditch system using Geopak Drainage.

Introduction

The creation of a ditch system in Geopak Drainage is similar to the construction of a storm sewer system, with the difference that instead of specifying a pipe for the link connection, the user needs to select the ditch option.

Geopak Drainage allows the user to specify a fixed ditch geometry or a ditch extracted from a reference surface.

Link/Ditch Configuration



Link options and types are shown in the Option list box on the left side of the dialog. The corresponding fields and items appear on the right. The four options are reviewed below.

Definition Defines the Link's path, connectivity, shape, material, roughness coefficient, size, and design options.

Conditions Defines or reviews the elevation condition for the Link including minimum depth, soffit, invert, and slope data. It also includes the profiling options for holding certain values constant.

Constraints Defines the Link constraints including the minimum and maximum rise (size), slope and velocity used in Link design.

Computation Reviews the Link hydraulic computations

Lab 19: Ditch Design

Introduction

Step 1. Execute **C:\data\geo\VDOT\drain1\LAB19_V8.EXE**.

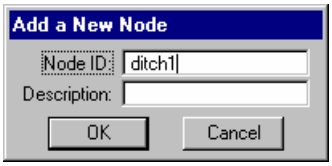
Step 2. Open the MicroStation file **c:\data\geo\VDOT\drain1\h17682.dgn**.

Step 3. Select **Drainage** from the **Applications > GEOPAK Drainage > Drainage** pull down menu.

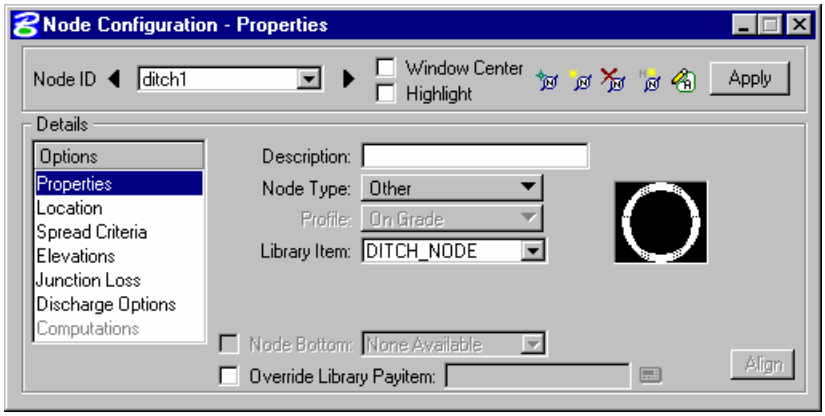
Step 4. Access the drainage project **h17682.gdf** by selecting **Project > Open** from the Geopak Drainage pull down menu.

Design Ditch Node: ditch1

Step 1. Select from the Drainage Menu: **Component > Node > Add.**

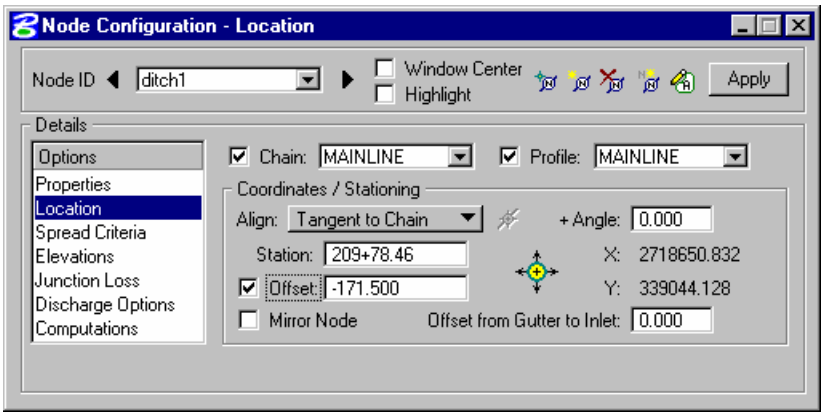


Step 2. Highlight **Properties** in the Options group box on the Node Configuration dialog. Select the Node to be used from the Drainage Library and assign its properties as depicted below:



Node ID	Ditch1
Node Type	Other
Library Item	Ditch_Node

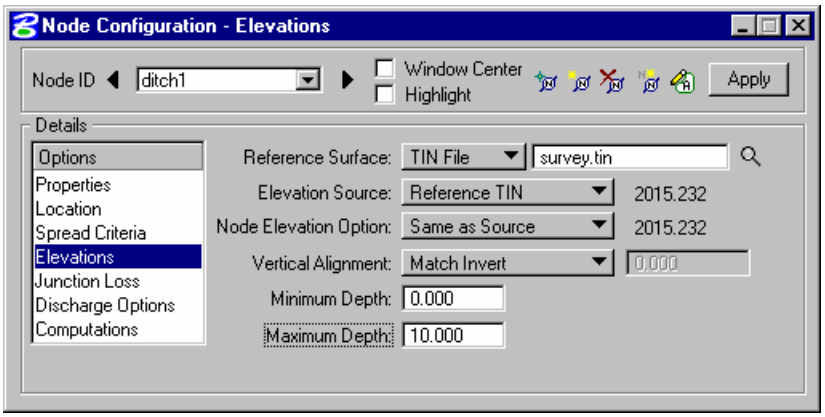
Step 3. Highlight **Location** to describe the inlet's location in the design plane:



Reference Chain	**Select** Reference chain: MAINLINE
Reference PGL	**Select** Reference PGL: MAINLINE
Align	Tangent to Ref. Chain
Station	209+78.46
Offset	-171.50

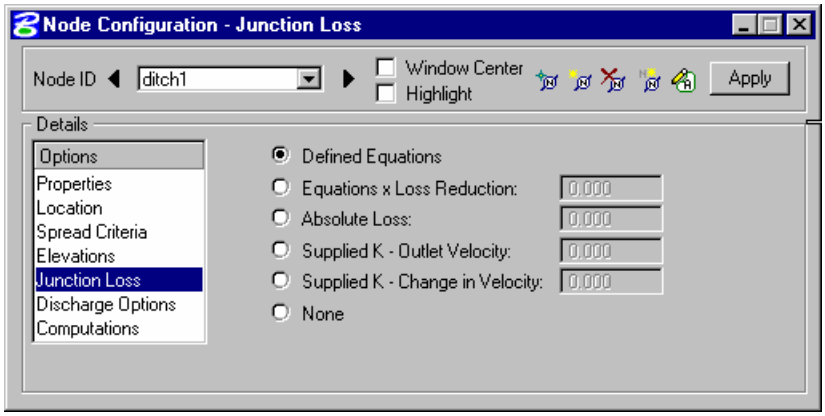
Press **Enter** to automatically place the node at the specified location.

Step 4. Highlight **Elevations** to assign the node an elevation and vertical alignment of the ditch node.

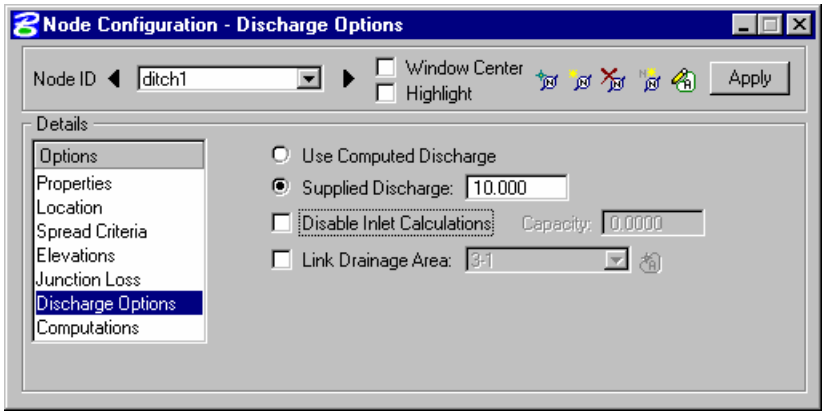


Elevation Source	Reference TIN
Node Elevation Option	Same as Source
Vertical Alignment Preference	Match Invert
Minimum Depth	0 feet
Maximum Depth	10.00 feet

Step 5. Highlight **Junction Losses** to use **Defined Equations** (defaults to Preference > Junction Loss Settings).



Step 6. Highlight **Discharge Options** to specify the discharge getting into the ditch. The actual flow was previously calculated. Therefore, keyin 10 cfs at the **Supplied Discharge** field and activate the toggle.

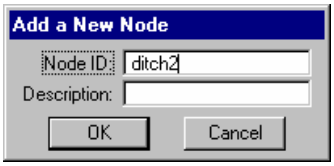


Step 7. Click **Apply** to add this node to the project.

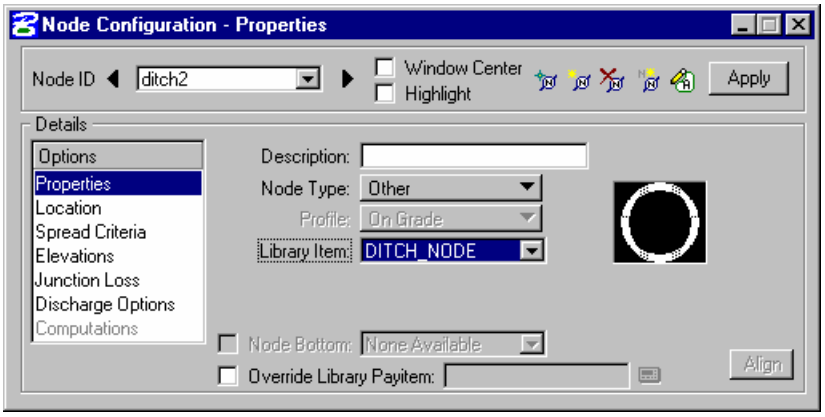
Step 8. Close the Node Configuration dialog.

Design Ditch Node: ditch2

Step 1. Select from the Drainage Menu: **Component > Node > Add.**

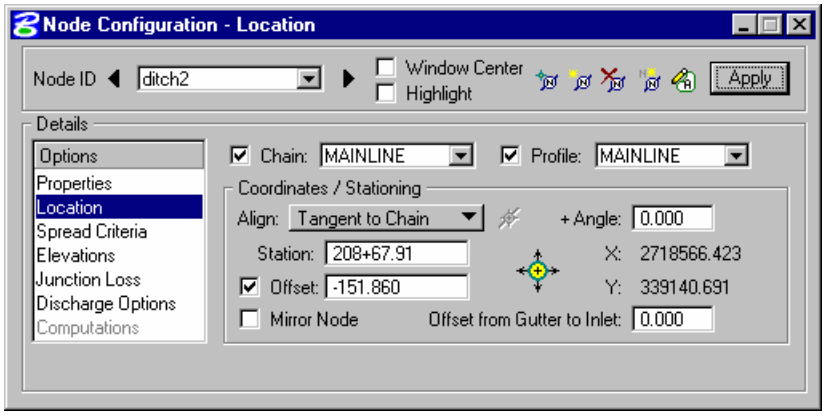


Step 2. Highlight **Properties** in the Options group box on the Node Configuration dialog. Select the Node to be used from the Drainage Library and assign its properties as depicted below:



Node ID	Ditch2
Node Type	Other
Library Item	Ditch_Node

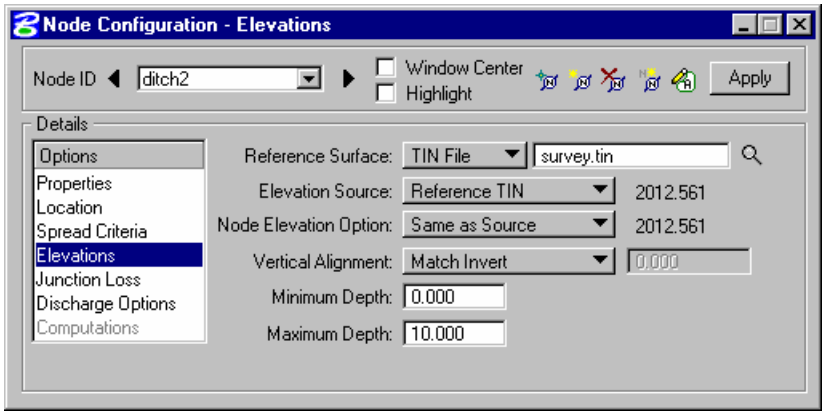
Step 3. Highlight **Location** to describe the inlet's location in the design plane:



Reference Chain	**Select** Reference chain: MAINLINE
Reference PGL	**Select** Reference PGL: MAINLINE
Align	Tangent to Ref. Chain
Station	208+67.91
Offset	-151.86

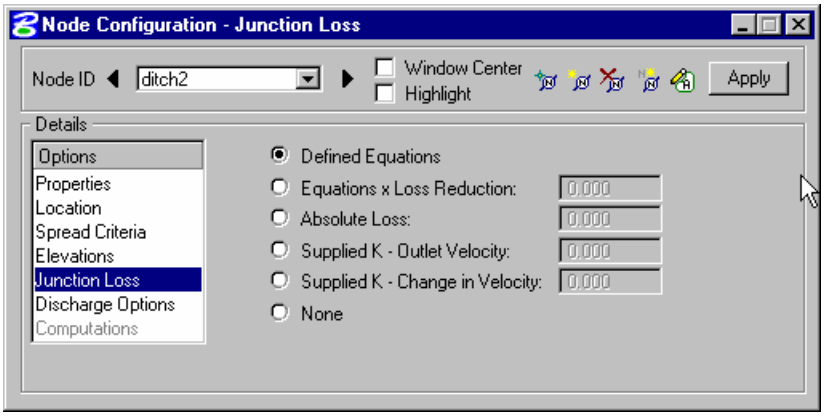
Press **Enter** to automatically place the node at the specified location.

Step 4. Highlight **Elevations** to assign the node an elevation and vertical alignment of the ditch node.

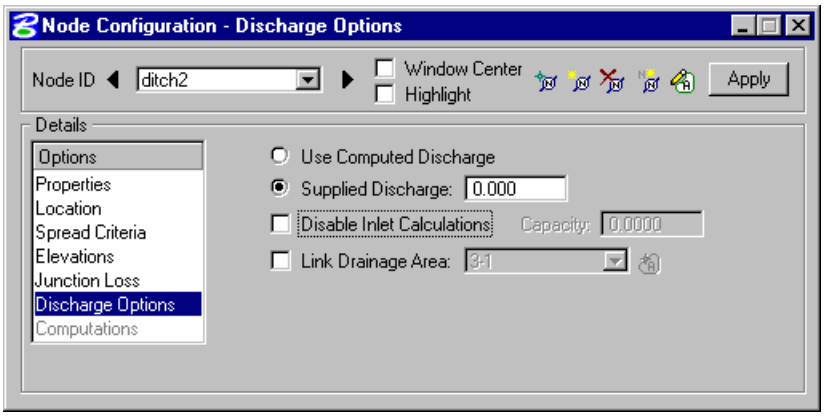


Elevation Source	Reference TIN
Node Elevation Option	Same as Source
Vertical Alignment Preference	Match Invert
Minimum Depth	0 feet
Maximum Depth	10.00 feet

Step 5. Highlight **Junction Losses** to use **Defined Equations** (defaults to Preference > Junction Loss Settings).



Step 6. Highlight **Discharge Options** to specify the discharge getting into the ditch. This node will not receive any direct discharge other than the one coming from the previous node. Therefore keyin 0 cfs at the **Supplied Discharge** field and activate the toggle.

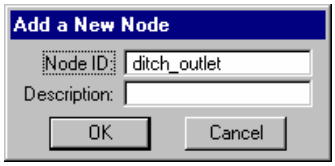


Step 7. Click **Apply** to add this node to the project.

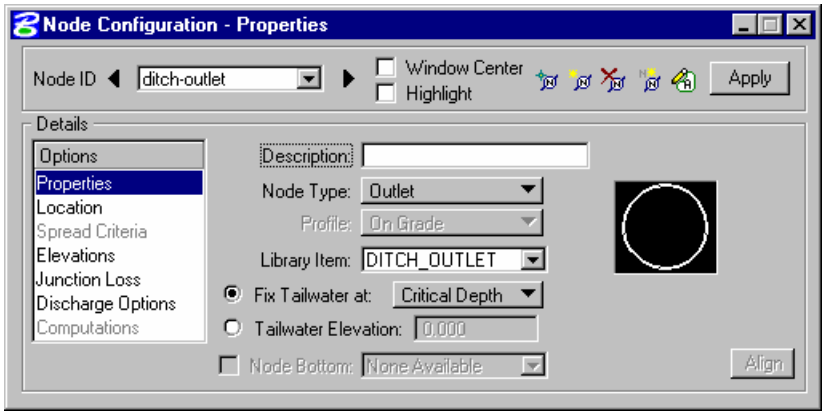
Step 8. Close the Node Configuration dialog.

Design Ditch Outlet Node: ditch-outlet

Step 1. Select from the Drainage Menu: **Component > Node > Add.**

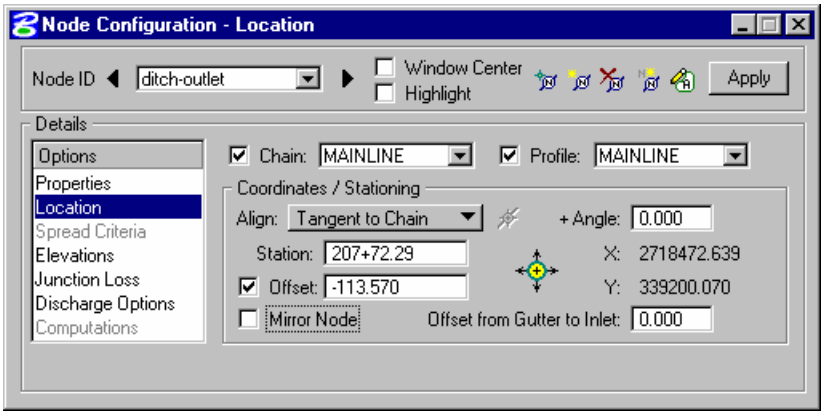


Step 2. Highlight **Properties** in the Options group box on the Node Configuration dialog. Select the Node to be used from the Drainage Library and assign its properties as depicted below:



Node ID	Ditch_outlet
Node Type	Outlet
Library Item	Ditch_Outlet
Fix Tailwater	Critical Depth

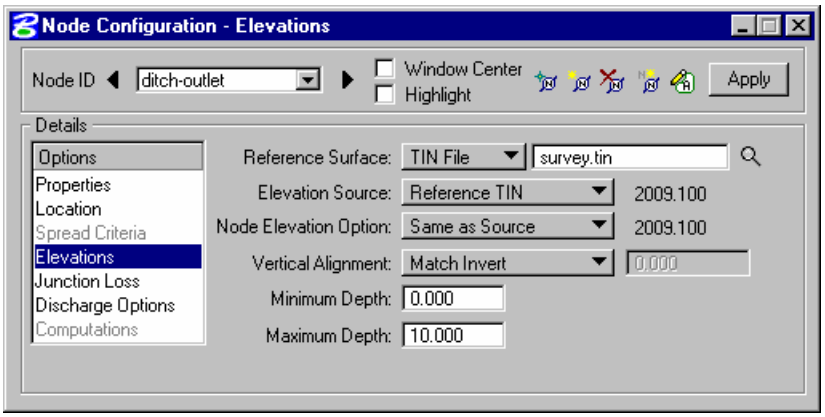
Step 3. Highlight **Location** to describe the inlet's location in the design plane:



Reference Chain	**Select** Reference chain: MAINLINE
Reference PGL	**Select** Reference PGL: MAINLINE
Align	Tangent to Ref. Chain
Station	207+72.29
Offset	-113.57

Press **Enter** to automatically place the node at the specified location.

Step 4. Highlight **Elevations** to assign the node an elevation and vertical alignment of the ditch outlet node.



Elevation Source	Reference TIN
Node Elevation Option	Same as Reference
Vertical Alignment Preference	Match Invert
Minimum Depth	0 feet
Maximum Depth	10.00 feet

Step 5. Click **Apply** to add this node to the project.

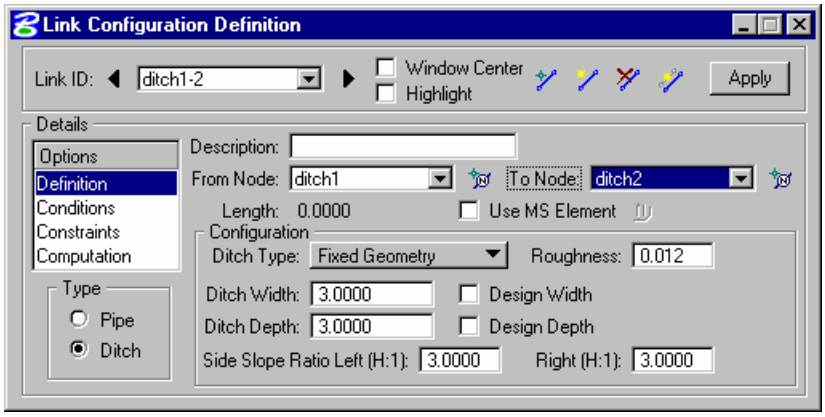
Step 6. Close the Node Configuration dialog.

Design Ditch Links

Step 1. Select from the Drainage Menu : **Component > Link > Add**.

Step 2. Highlight **Definition** to Set the ditch spatial characteristics including From and To Nodes ID's, and geometry.

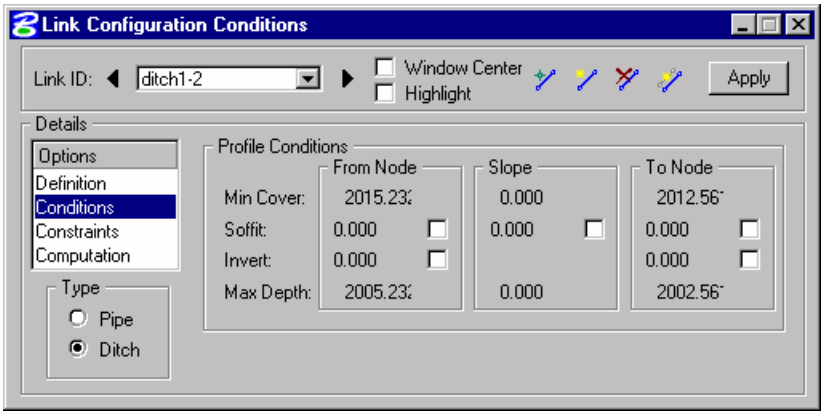
Graphically select the Nodes by clicking the **ID** button for each and identifying the Nodes. Dtich1-2 traverses From Node ditch1 to Node ditch2



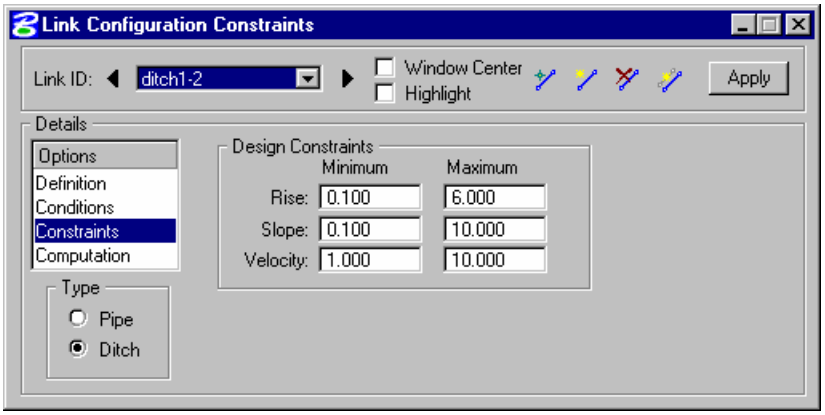
Type	Ditch
Geometry	Fixed Geometry
Manning's n	0.012
Ditch Width	3.00
Design Width	Toggle OFF
Ditch Depth	3.00
Design Depth	Toggle OFF
Side Slope Ratio	3:1 Left and 3:1 Right

Step 3. Highlight **Conditions.** The elevations shown are based on the *From-Node* and *To-Node* elevation minus the min/max depth, as specified in the Node Definition Dialog Box for Nodes ditch1 and ditch2 respectively.

In this case, no entries are necessary; GEOPAK Drainage will design all the profiles for this project.

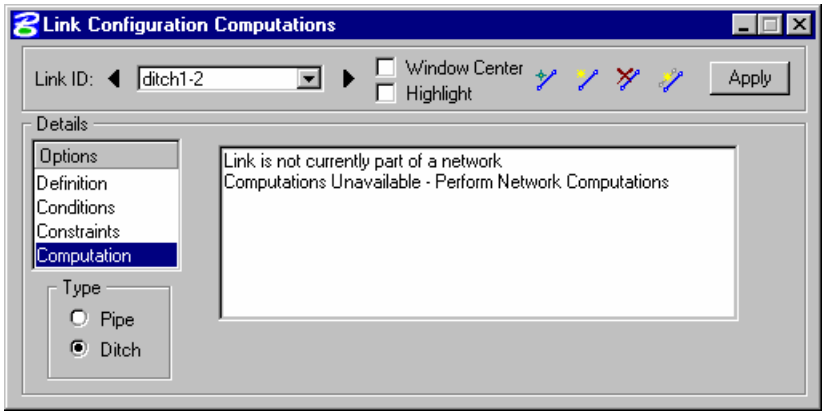


Step 4. Highlight **Constraints.** Establish the min/max design criteria for Links as follows:



Rise min/max	0.1 / 6.0 (feet)
% Slope min/max	0.10 / 10.00 (%)
Velocity min/max	1.00 / 10.00 (fps)

Step 5. Highlight **Computations** to display the computed hydraulic properties of the ditch.



Note: Ditch hydraulics are not available for review until a Network has been established and designed or analyzed successfully. Check back here for computations after the Network has been added and designed or analyzed.

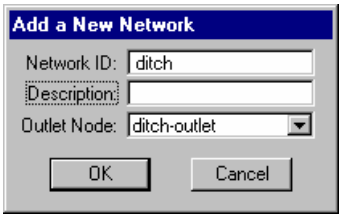
Step 6. Press **Apply** to incorporate the link to the project.

Step 7. Add the remainder of the ditch system from Node ditch2 to Node ditch-outlet using the Drainage Menu **Component > Link > Add**.

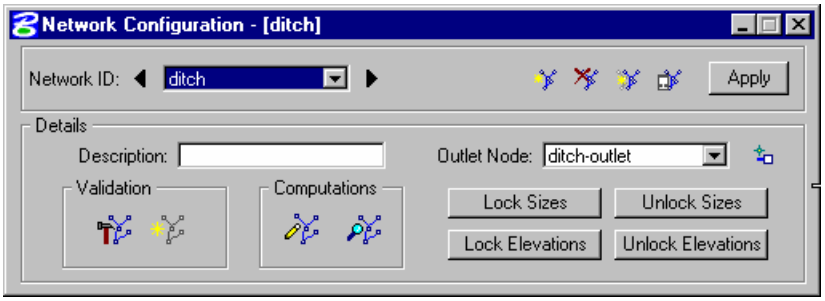
Step 8. After adding the last portion of the ditch, close the Link configuration dialog.

Ditch Network Design

Step 1. Select **Network > Add** from the Drainage menu.

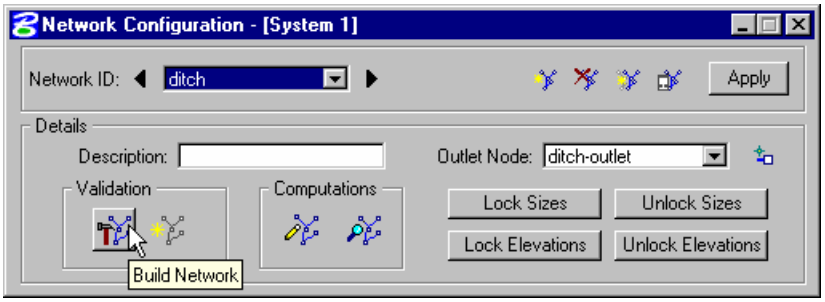


Step 2. In the Network Configuration dialog, enter the following information:



Network ID:	DITCH
Description:	(Optional)
Outlet Node	Click the **ID** button and select the DITCH-OUTLET node.

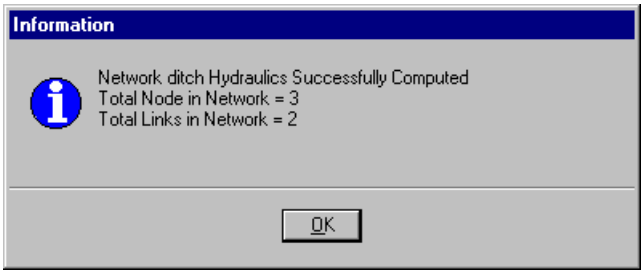
Step 3. Click the **Build Network** button. This feature verifies the nodal topology and link connectivity. The **Highlight Network** feature highlights all components (ditch nodes and ditch links) connected to the active Network.



Step 4. Click the **Apply** button. Network "DITCH" has been added to the project.

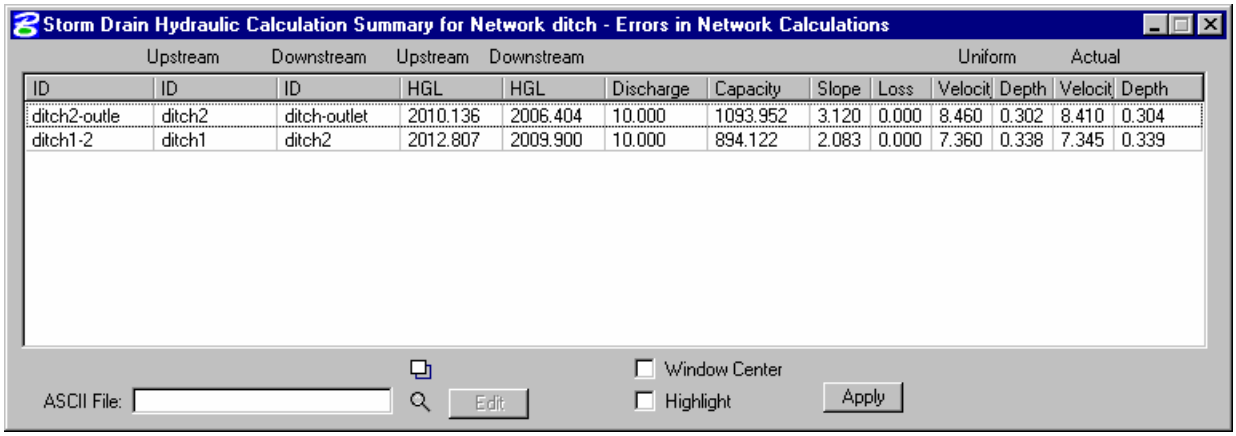
Step 5. Click the **Design** button. This command initiates the hydraulic design of the components contained in the Network.

Step 6. Then GEOPAK will design the network and issue the pertinent Warning messages, if necessary.



Step 7. Set the Active Network to *Ditch* so we can review some of the provided reports.

Step 8. We can now review some of the GEOPAK provided reports. **Drainage > Reports > Storm Drain Links**.



Step 9. Exit MicroStation.