VDOT Advanced GEOPAK Drainage

Student Course Guide

Bentley Institute

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.

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. 5,8.15,415 and 5,784,068 and 6,199,125.

Copyrights

©2000-2007 Bentley Systems, Incorporated.
MicroStation ©1998 Bentley Systems, Incorporated.
    IGDS file formats ©1981-1988 Intergraph Corporation.
    Intergraph Raster File Formats ©1993 Intergraph Corporation.
    Portions ©1992 – 1994 Summit Software Company.
    Portions ©1992 – 1997 Spotlight Graphics, Inc.
    Portions ©1993 – 1995 Criterion Software Ltd. and its licensors.
    Portions ©1992 – 1998 Sun MicroSystems, Inc.
    Portions ©Unigraphics Solutions, Inc.
    Icc ©1991 – 1995 by AT&T, Christopher W. Fraser, and David R. Hanson. All rights reserved.
    Portions ©1997 – 1999 HMR, Inc. All rights reserved.
    Portions ©1992 – 1997 STEP Tools, Inc.
    Sentry Spelling-Checker Engine ©1993 Wintertree Software Inc.
    Unpublished – rights reserved under the copyright laws of the United States and other countries. All rights reserved.
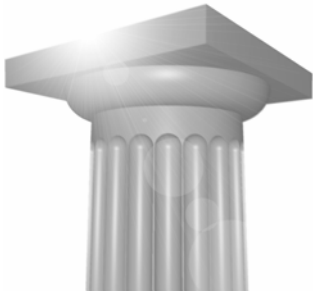
Table of Contents

**3   Starting Drainage projects using GEOPAK
            Road Project Manager   3-1**

1 Digital Terrain Modeling for Drainage Design and Analysis

OBJECTIVES

In this chapter, you will learn more about:

- Displaying DTM features such as contours and triangles.

- Create TIN files from Superelevation Shapes.

- Create TIN files from proposed cross sections.

INTRODUCTION

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. The tools associated with the DTM are not related to and are independent of GEOPAK Drainage.

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

The DTM tool frame can be accessed from the *MS Menu*: *Applications>Geopak Road>DTM Tools*. Then clicking on the DTM menu icon, the user can access the DTM tool bar depicted below.
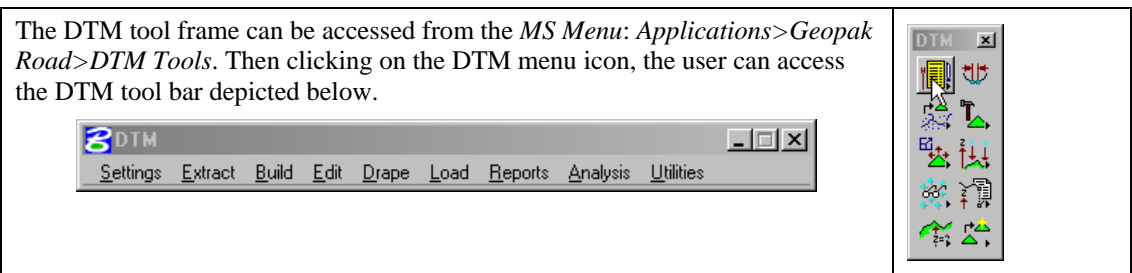
SETTINGS

When the Stroking Options tool is selected, the DTM Stroking Options dialog opens.
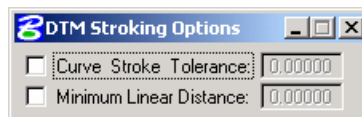
Two user-defined stroking values can be defined before graphics can be extracted to create a DTM. Stroking is the process of automatically adding shots to the DTM Input file by interpolating new shots

from the linear and curved sections of the data. If the source topography data is mapped in a 3D-design file, stroking may be applied. Stroking is not available if the topography data resides in a 2D-design file.

Curve Stroke Tolerance	The maximum distance between the arc and the chord used to approximate the arc in the DTM. Stroking is only applied to breaks and contours.
Minimum Linear Distance	If a linear segment is greater than the Minimum Linear Distance, points are interpolated and added to the segment such that the distance between the points in not greater than the Minimum Linear Distance.

EXTRACT GRAPHICS

The Extract Graphics tool reformats MicroStation elements into DTM input data.
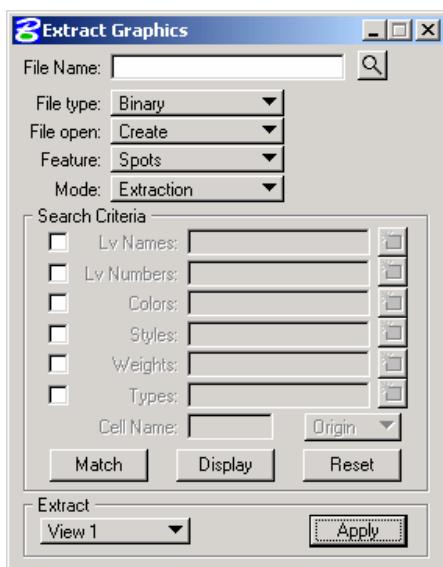
File Name	The name of the file to be created for storing the input data. The .dat extension is automatically assigned so you are not required to key this in; however, an alternate extension may be defined. If file already exists, it may be selected by clicking Files.
File Type	Format of the new file. Either format produces the same results. The difference between the two is ASCII files can be viewed and edited with any text editor while binary files process faster, but cannot be viewed.
	For ASCII file designation, you must define the number of decimal places for the data.
File Open	Indicates whether a file is being created (Create) or the user is appending data (Append) to an existing .dat file.
Feature Type	This notifies GEOPAK of the feature type that the extract graphics will put in the DTM.
	Spots – Random survey points. Can also be vertices of a Line or a Line String.
	Breaks – Designate linear features such as edges of pavement, ditch bottoms, ridges, etc. No triangle leg will cross a Break.
	Boundary – The external boundary of the digital terrain model – only per .dat file.
	Contours – For use in extracting digitized or otherwise imported contours.
	Void – Closed shapes representing an area with missing data or obscure areas. (i.e. ponds, headwalls, concrete pads, etc.). The void coordinates determine void lines, which are inserted as drape lines on the triangulated surface and do not alter the slope or elevations of the TIN.
	Island – Area within a void that contain data and should be included in the model.

Graphic Triangle – For use in extracting triangles from a TIN model that has been otherwise created or imported.

Drape Void – Same as Void, except the Void's vertex elevations are ignored and the triangulated model elevations at these vertex points are utilized.

Break Void – Same as Void, except the void elevations are included in the triangulated model at it's x y coordinates; therefore, the slope and elevations are modified on the TIN surface.

Mode   The Extraction mode calculates XYZ data directly from the coordinate values of 3D MicroStation elements. The Interpolation mode produces XYZ data by interpolating between spot elevations along linear MicroStation elements. This mode works in both 2D and 3D files wherein the elements are not represented with a defined Z value.

Search Criteria   The Select Criteria group box provides options to specify the features to be extracted. When a "check" is placed in the box next to Levels, the Select box is activated. Then click the Select button to invoke the Level Mask dialog wherein only those levels you want GEOPAK to search for when extracting data can be selected. If the Levels box is *not* turned on, GEOPAK searches all levels. The same procedure is utilized for other criteria selections. The three buttons located at the bottom of the Select Criteria group box Match, Display and Reset enable you to interactively define, highlight elements that match the search criteria or clear the search criteria respectively.

Extract   The Extract group box has several options for data extraction. Complex Chain reads those elements along adjoining MicroStation elements. Selection Set uses the MicroStation selection set tools to identify elements to extract. Fence extracts all elements within a fence boundary. View1, View2, View3, etc. extracts all selected elements displayed in the selected view.

BUILD TOOLS

Tools	
Menu Bar	**Build > Triangle** **Build > Lattice** **Build > Merge TINs** **Build > Clip TIN** **Build > Pad** **Build > Delta Surface**

Included under the Build pull down and icons are options for creating, manipulating and merging DTM models. This class addresses, in detail, *only* those options utilized in routine DTM operations.

BUILD TRIANGLES

Tool	
DTM Menu Bar	**Build > Triangle**

The Build Triangle tool processes the information stored in a DTM input file (**.**dat) to create a triangulated model (TIN). The file extension TIN (which is automatically assigned) represents a *triangulated irregular network*.

Data File	The DTM input file where the extracted topological features are stored. The file can also be selected via **Files**. If the full path is not given, GEOPAK utilizes the working directory to locate the file.
TIN File	The filename in which the DTM data will be stored in a binary format. The file can also be selected via **Files** if the file exists. If the full path is not given, GEOPAK utilizes the working directory to locate the file.
Dissolve Option	This option will allow the elimination of external triangles based on the selected option. Note that internal triangles are not affected. The three options are: **None** - no external triangles are deleted. **Sliver** - long, thin triangles based on a hard-coded formula within the software are deleted. **Side** - external triangles where a side is longer than the user specified length are deleted.
Process	Commences the TIN creation process.

ADDITIONAL BUILD OPTIONS

Tool	
Menu Bar	**Build >** *Additional Tools*

Several additional Build tools are supported, which are detailed in the following table.

Lattice	Creates a gridded mesh (**.lat**) of the triangulated data (TIN) to create a three dimensional visual display of the topography.
Merge TINs	Two triangulated models to be merged together as long as the boundaries of one model overlaps the other at a minimum of one point. This process creates a third model (**TIN**) from the combination of the two existing models.
Clip TIN	Creates a new model (**TIN**) from a portion of an existing model. The area is defined as internal or external to a user defined polygon.
Build Pad	Defines a pad (such as a building) and integrates the pad into the existing terrain with a variety of slope options.
Build Delta Tin	Creates a new model based on the difference between two other models or a model and elevation surface.

REPORTS

Tools	
Menu Bar	**Reports > Duplicate Points** **Reports > Crossing Features** **Reports > Tin Statistics** **Reports > Lattice Statistics**

Options under the **Reports** pull down and tool frame include a way to check for duplicate points or crossing breaklines, and the ability to generate statistics associated with a **TIN** or **Lattice** file. Each report is described in the following table.

Duplicate Points	Reports points with the same x and y coordinates
Crossing Features	Reports intersecting (same xy location, varying z) breaklines or contours.
Triangle Statistics Lattice Statistics	Displays a summary indicating the total count of each element type and minimum and maximum X, Y, Z ranges for the specified TIN or LAT file.

UTILITIES

Tools	
Menu Bar	**Utilities > Convert TIN** **Utilities > ASCII to Binary** **Utilities > Binary to ASCII** **Utilities > Check Triangulation** **Utilities > Metric to English** **Utilities > DTM** **Utilities > Import LandXML** **Utilities > Export LandXML** **Utilities > LIDAR XYZ Tools**

Tools under the **Utilities** pull down and tool frame include a way to check the validity of a triangulated file, converting triangulated files from previous versions of GEOPAK, converting the DTM data file between ASCII and binary format; as well as exporting the LATTICE information to the Trimble DTX format. They are detailed in the following table.

Convert TIN	Converts a pre-GEOPAK 98 triangulated file to the current format.
ASCII to Binary Binary to ASCII	Reformats the DTM data file (**.dat**).
Check Triangulation	Verifies the integrity of the triangulated file. A message appears indicating "Triangulation Valid" if the file structure is intact.
English to Metric Metric to English Metric to Imperial Custom	GEOPAK TIN files are neither English nor metric. English and metric preference settings within GEOPAK are for volume purposes and do not affect the TIN. This utility converts an entire TIN file with optional translation and rotation tools.
Export DTM	This tool utilizes a lattice file to generate a Trimble DTX file, Trimble TTM file, or Leica GSI file suitable for use in Trimble and Lecia field equipment.

VDOT GEOPAK Drainage © 2007 VDOT 1-5

Import LandXML	This tool is utilized to create a DAT file from a LandXML file.
Export LandXML	This tool is utilized to create a LandXML file from a TIN file.
LIDAR XYZ Tools	This tool set is used to process, filter, clip, compare, report, and display LIDAR data.

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 designer 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.
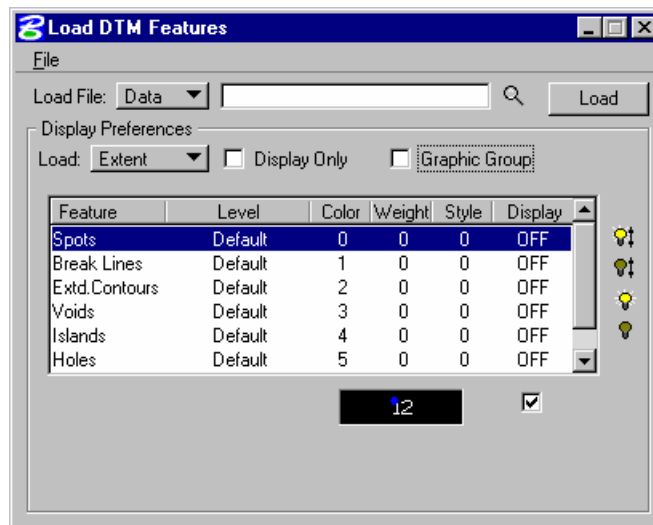
Activating the **Display Only** toggle enables you to view the elements without writing them to the MicroStation file (temporary display). Conversely, deactivating the toggle writes the DTM features to the MicroStation design file at the defined symbology. These elements can be placed as a graphic group when **Display Only** is deactivated.
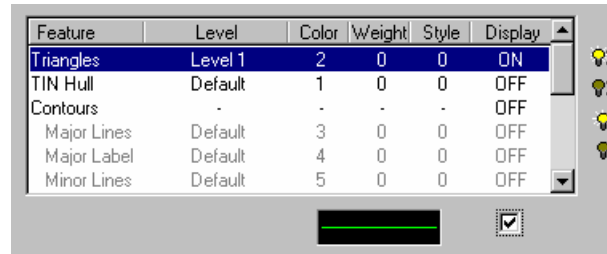
		Activates the display of all Features.
		Turns off the Display of all Features.
		Turns on only the selected Feature. This can also be accomplished by activating the toggle (below the list box) or double-clicking on an item that is turned off within the list box.
		Will turn off only the selected item. 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.

EDIT TOOLS

Tools	
Menu Bar	**Edit > Triangles** **Edit > Duplicate Points** **Edit > Crossing Break Lines** **Edit > Filter Vertices** **Edit > Join Linear Features** **Edit > Z Range Clip**

GEOPAK supports an extensive range of powerful editing tools, both pre- and post TIN generating.

Triangles	The triangulated model can be dynamically edited in terms of adding, deleting or moving spot elevations. Break lines can be added, draped or deleted.
Duplicate Points	Interactive editing of points in a DTM data file with the same X and Y coordinates within the data (DAT) file.
Crossing Break Lines	Interactive editing of crossing break lines and extracted contours within the data (DAT) file.
Filter Vertices	Removes vertices from break lines or extracted contours within the data (DAT) file when located within the specified Tolerance of another vertex.
Join Linear Features	The Join Linear Feature is useful when the source break line or contour line consists of several elements containing numerous vertices. In these cases, the end of one break line is the beginning of the next break line, so the software utilizes this as two separate break lines. During the Joining process, GEOPAK changes these from the beginning and end of individual elements to the beginning and subsequent shots of one element.
Z Range Clip	Deletes individual vertices from the data file (**.DAT**) based on a user defined elevation range.

DIGITAL TERRAIN MODEL (TIN)

Data File

Feature Types: Spots

Breaks

Voids

Contours

Boundaries

TRIANGULATION

(Build Triangles)

Triangulated Irregular Network

**Build Lattice
Build Pad**

Analysis

DISPLAY FEATURES

**Ground XS and
Profiles**
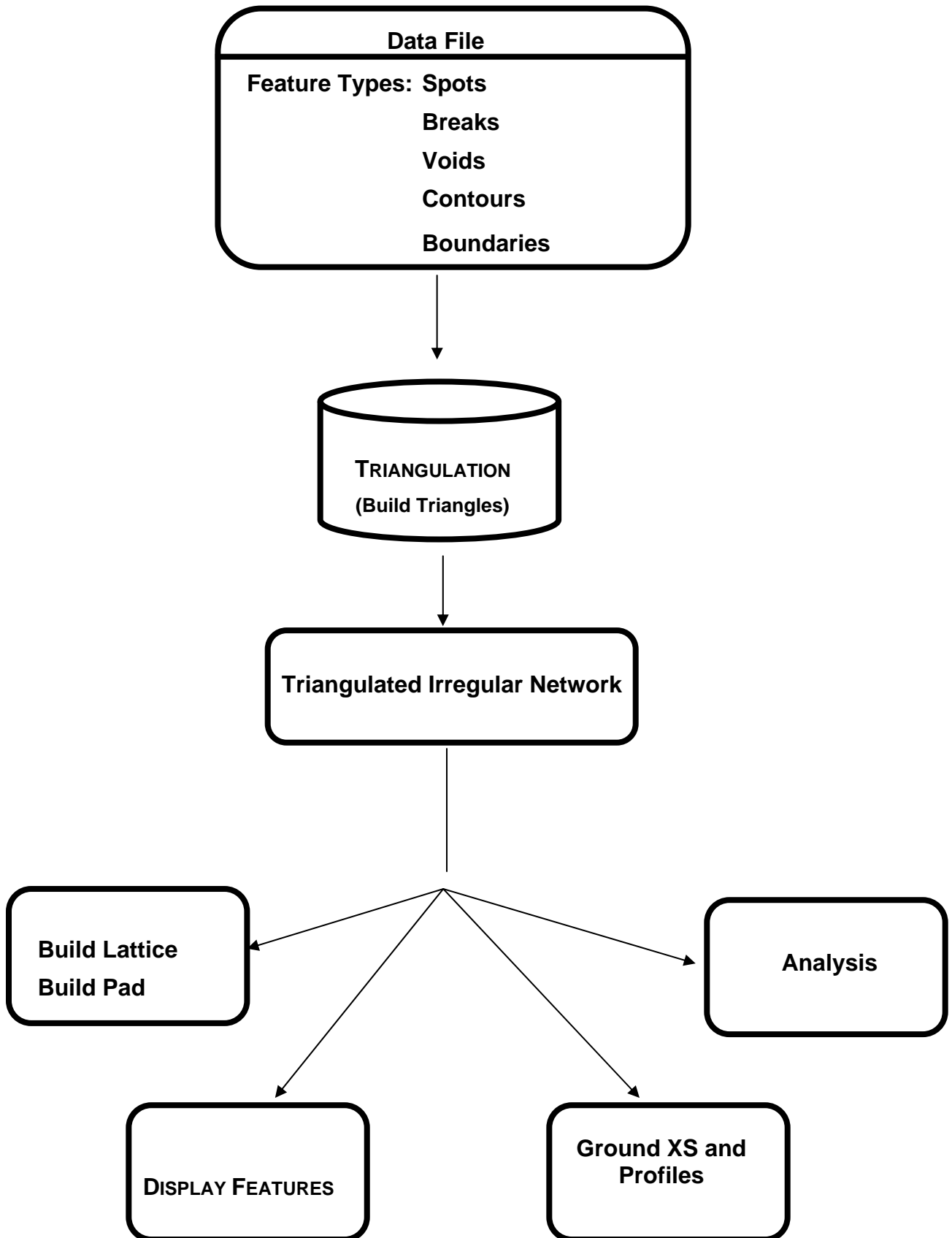
LAB EXERCISE: DIGITAL TERRAIN MODELING

The purpose of this exercise is to review the triangles of a tin file and to create contours from it.

> ### *Accessing DTM Tools*

1. Execute *c:\data\geo\vdot\drain1\LAB01_V8ad.EXE.*

2. Open the MicroStation file, *c:\data\geo\vdot\drain1\h17682.dgn*.

3. Access the GEOPAK DTM Tools frame. *(Applications > GEOPAK Road > DTM Tools)*

> ### *Load DTM Features*

1. Select the Load DTM Features icon from the tool frame.
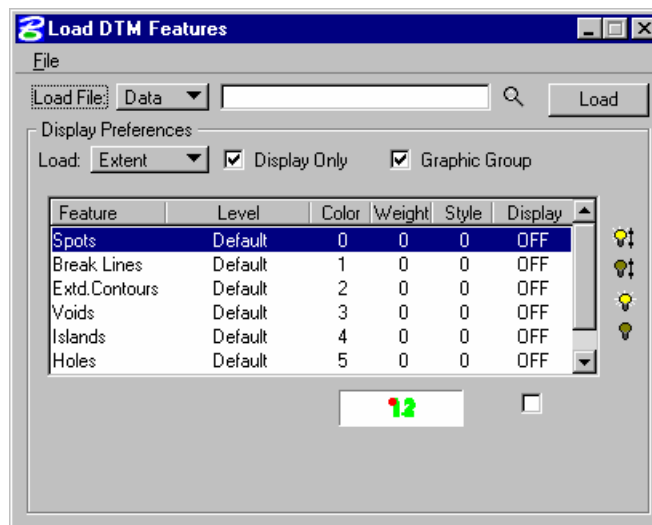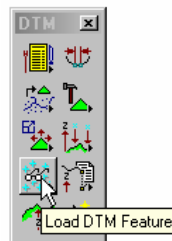
2. Change the Load File option to TIN.

3. Select the TIN file located in *c:\data\geo\vdot\drain1\survey.tin* by clicking the Files icon.
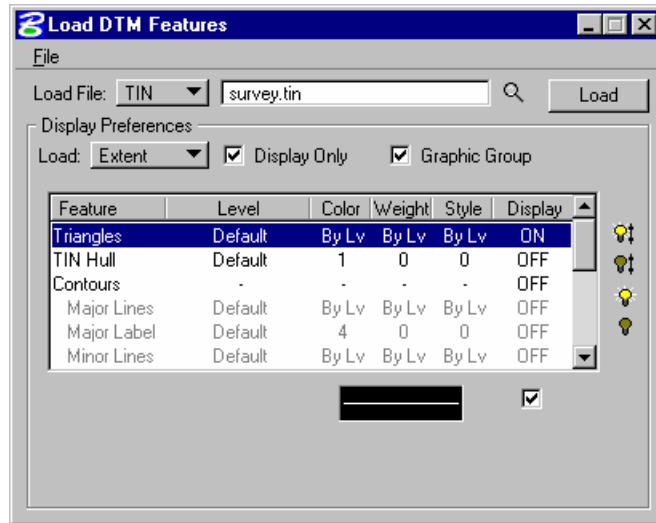
4. Enable the toggles for Graphic Group and the Display Only .

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:

                                                                       VDOT GEOPAK Drainage
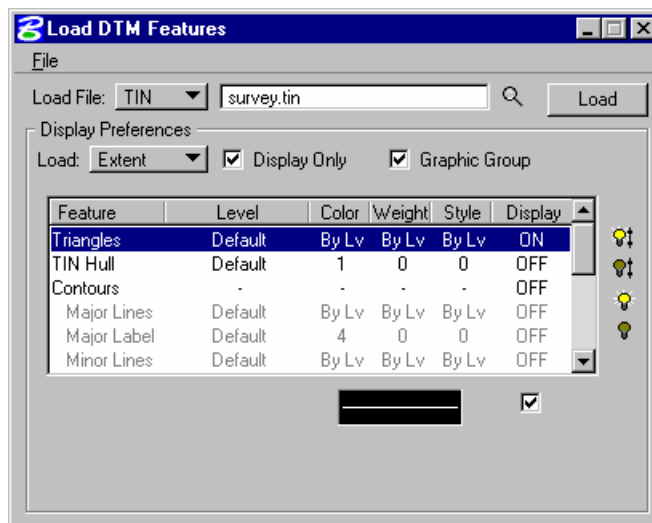
6. Set the display symbology for the Triangles as indicated below:
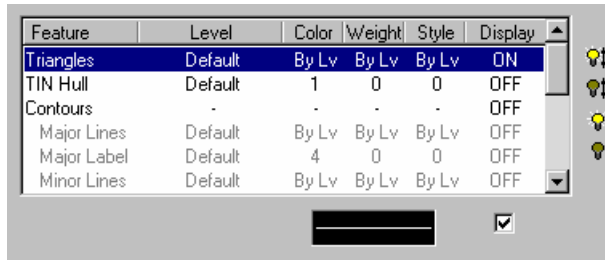
7. Click **Load** and review the results.

8. When complete, toggle OFF the triangles.

9. Select the **Contours** feature. Activate the **Display** Feature. Verify that **ON** is set in the Display field.

10. 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.
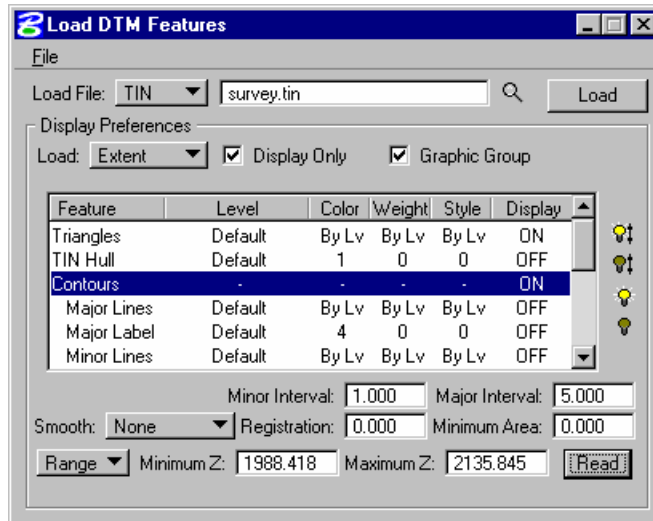
11. Enable the **Display Only** toggle since we only want to "preview" our contours but not actually write them to the design file.

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			

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

14. Close the Load DTM Features dialog box.

SUPERELEVATION SHAPES TO DTM TOOL

The Shape to DTM tool reads GEOPAK superelevation shapes and creates a DAT file suitable for use in the Build Triangles tool in the DTM tool frame. Building this proposed tin of the pavement surface helps the designer to better analyze the drainage flow due to superelevation conditions ("hidden low points", zero slope conditions, etc.)

This tool can be invoked from *Applications>Geopak Road>Cross Sections>Superelevation Shape Manager Tools.*
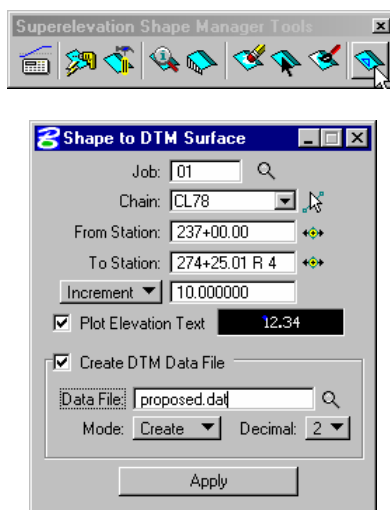
Job	Gpk file where the chains and profiles of the drawn shapes are stored.
Chain	Chain associated with the superelevation shape, i.e., shape cluster baseline defined when the shape was created.
From Station/To Station	Lower and upper stationing to be used as the limits of the DAT file.
Increment/Even	The interval of points used for the data file. When Increment is used, the value is added to the From Station iteratively until the To Station is reached. When Even is utilized, the points are taken at even values within the station range.
Plot Text Elevation	When turned on, text is placed at each point included within the data file.
Data File	Data file to be created or appended.
Mode	The dat file can be created or a previously created dat file can be utilized, in which case, the data created in this procedure is appended to the original data
Decimal	Number of decimal places used when creating the Data file.

Apply	Commences procedure.

LAB EXERCISE: SHAPES TO DTM

The purpose of this exercise is to create a proposed dtm from the superelavations shapes developed by the roadway designers.

➢ *Creating a Proposed DTM*

1. Attach the superelevation shapes dgn file: *c:\data\geo\vdot\drain1\d17682work.dgn* as a Microstation reference file. *(MS Menu: File > Reference)*

2. Open Project Manager (*MS Menu: Applications > GEOPAK Road > Project Manager)*

3. Select project:*17682.prj* located in *c:\data\geo\vdot\drain1*, and click **OK**.
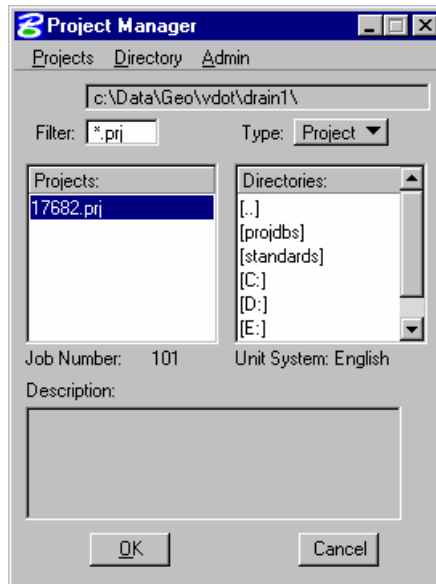
4. Access User: VDOT.
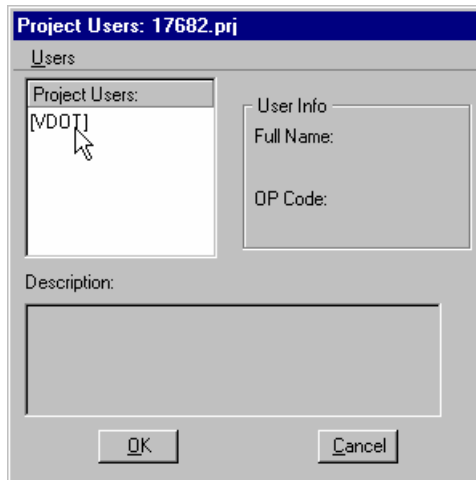
5. Click **OK**. The Road Project dialog will load. This will setup the proper directories for the location of the gpk file.
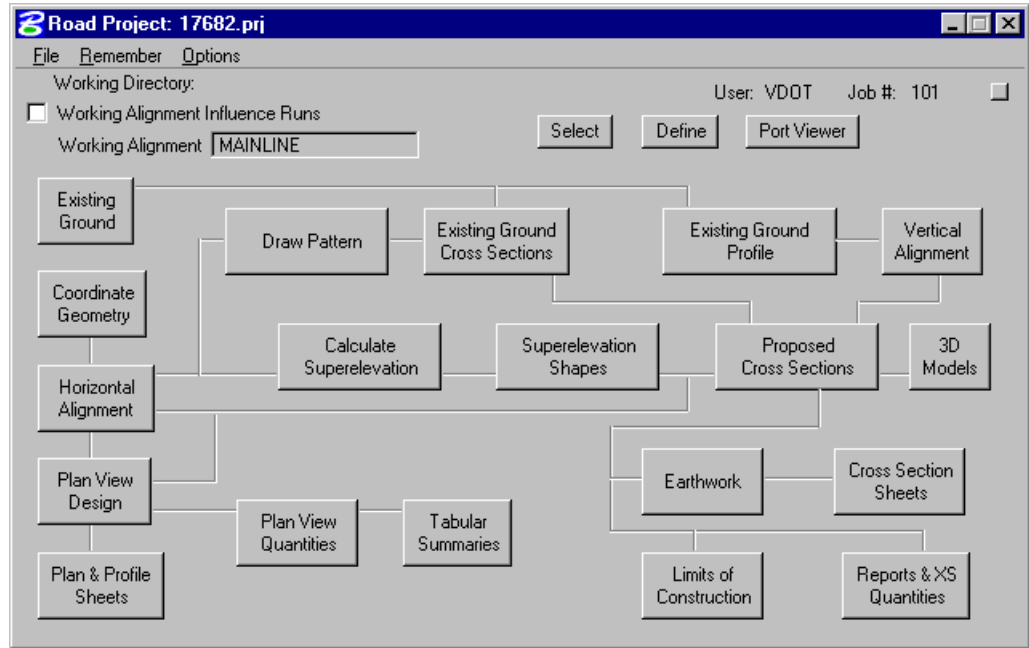
6. Minimize the Road Project dialog box.

7. Access the Superelevation Shapes Manager tools. *(Applications > GEOPAK Road >Cross Sections > Superelevation Shapes Manager Tools).*

8. Select the **Shape to DTM** icon.
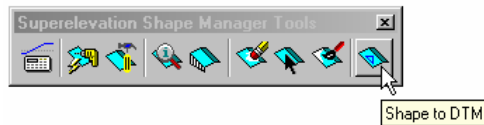
9. Populate the Shapes to DTM Surface dialog as shown below.
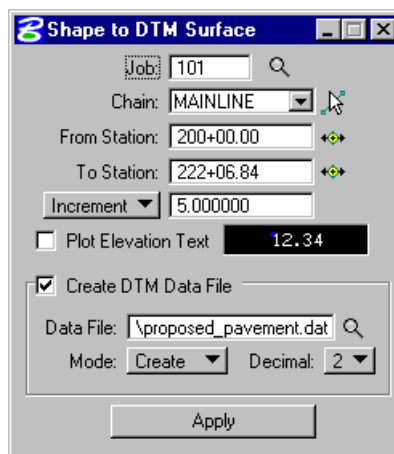
10. Click **Apply** to create the proposed_pavement.dat. This file will be created in your active working directory: *c:\data\geo\vdot\drain1*.

11. Access the GEOPAK DTM Tools frame. *(Applications > GEOPAK Road > DTM Tools)*

12. Click **OK** to accept the Run.

13. Select the Build Triangles icon from the tool frame.
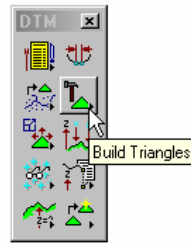
14. Populate the Build Triangles dialog as shown below.
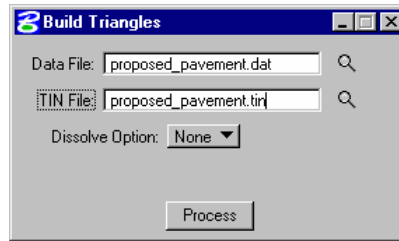
15. Click **Process** to create the proposed_pavement.tin DTM file.

16. Use the steps in the previous lab exercise to display the new triangles and contours along the pavement surface.

17. Exit Microstation.

PROPOSED CROSS SECTIONS TO DTM

The DTM Prop 3D tool locates specified proposed cross section elements and existing ground, and generates an x, y, z file of vertices for break lines suitable for use with GEOPAK's Digital Terrain Modeling. Each cross section generates one break line. In the hierarchy, the path follows (from left to right) existing ground until a proposed finish element is intersected. The proposed path is followed until it ties back to an existing ground element.

Job	Job which contains the cross section chain.
Chain	Chain utilized to generate the existing ground cross sections or the name on the XSCELL
Current Station	Defaults to the lowest stationed cross section in the file. During processing, the current cross section being reported is displayed
Begin Station / End Station	GEOPAK determines the beginning and ending cross section station within the MicroStation file and they are noted below the chain and placed in the key-in areas. If these defaults are utilized, the report is generated for the entire set of cross sections. However, if a report for a smaller range is required, the user may simply key in the required beginning and/or ending station.
Search Criteria: Existing Ground Line/Proposed Finish Grade	Refers to the Microstation symbology used to draw the existing ground line and any Microstation element that makes up the upper layer of the proposed cross section.
Pause on Each XS	When the toggle is activated, the first section is processed and stops. Each click of Apply processed another section.
ASCII File	Specify the name of the ASCII file to be generated. The designer has the option of keying in the file name, or clicking **Files**, invoking the Files dialog, wherein the designer may select the desired file. The file is placed in the current directory, unless a full path is specified. If the specified file name already exists, the

	file is overwritten with no warning message given.
Apply	Commences processing.

LAB EXERCISE: PROPOSED CROSS SECTIONS TO DTM

The purpose of this exercise is to create a proposed dtm from the roadway cross sections developed by the roadway designers. The newly created dtm will include the existing and proposed elements of the roadway merged together.

➢ *Creating a Proposed DTM*

1. Open the Microstation file *c:\data\geo\vdot\drain1\d17682xsmainline.dgn (MS Menu: File > Open)*

2. Open Project Manager (*MS Menu: Applications > GEOPAK Road > Project Manager)*

3. Select project:*17682.prj* located in *c:\data\geo\vdot\drain1*, and click OK.
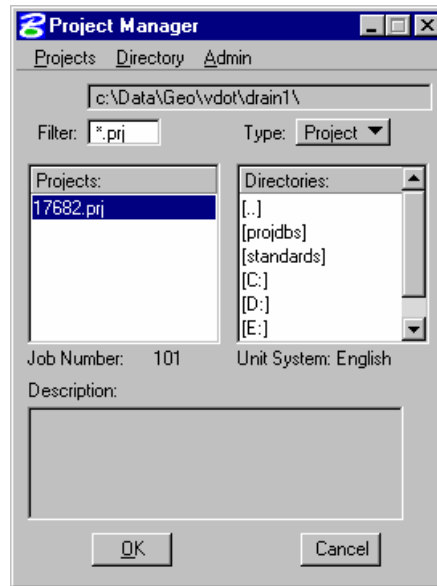

4. Access User: VDOT.
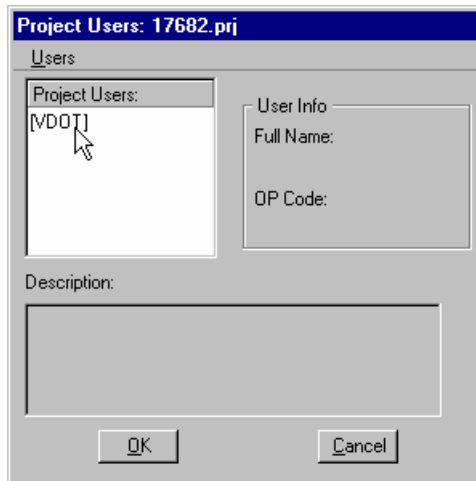
5. Click **OK**. The Road Project dialog will load. This will setup the proper directories for the location of the gpk file
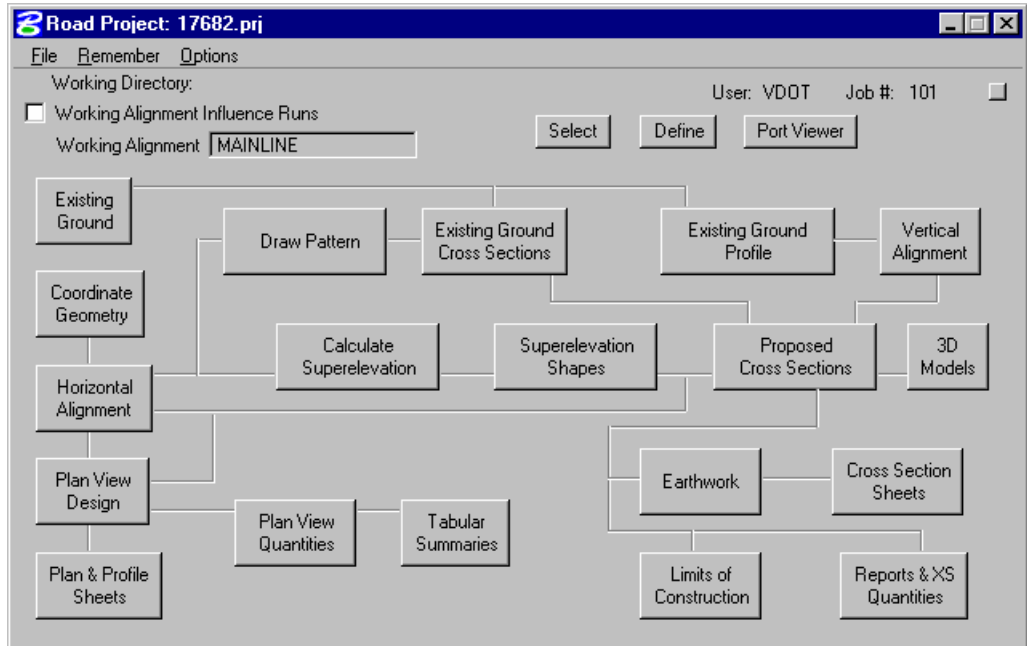
6. Click the **Report &XS** Quantities button

7. Access the DTM Proposed 3D report.
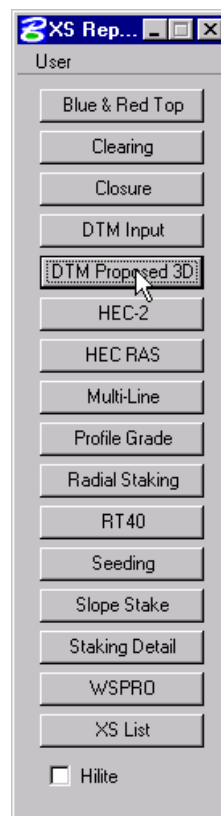
8. Setup the DTM Proposed 3D Report dialog as shown below.
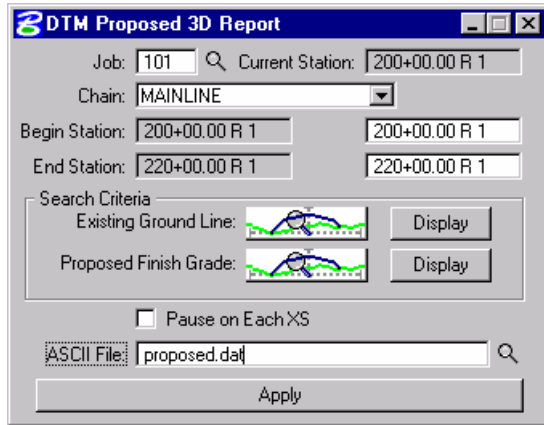
| Existing Ground Line:	Lv Name: Level 1
Proposed Finish Grade:	Lv Name: Level 2 thru Level 13
ASCII File	Proposed.dat

9. Click **Apply** to create the proposed.dat file. This file will be created in your active working directory : *c:\data\geo\vdot\drain1*.

10. Access the GEOPAK DTM Tools frame. *(Applications > GEOPAK Road > DTM Tools)*

11. Click **OK** to accept the run.

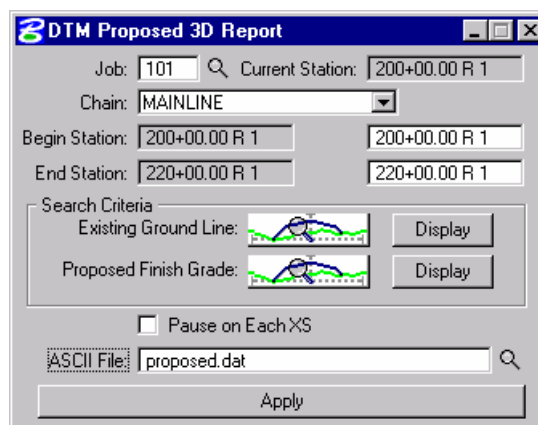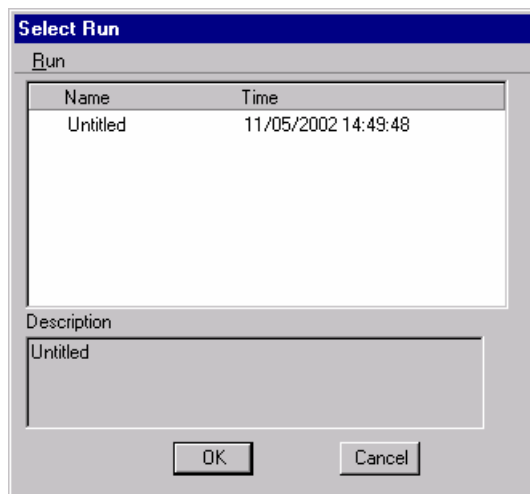12. Select the Build Triangles icon from the tool frame.
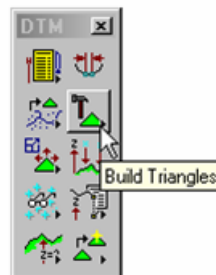
13. Populate the Build Trainagles dialog as shown below.
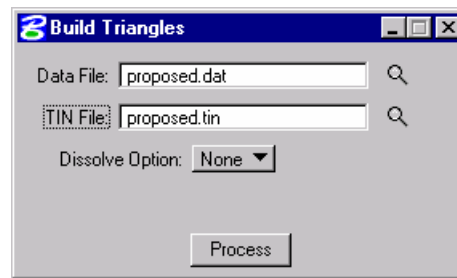
14. Click **Process** to create the proposed.tin file

15. Use the steps in previous lab exercises to display the new triangles and contours along the pavement surface.

16. Exit Microstation.

2   **Drainage Analysis**

OBJECTIVES

Learn how to use different Geopak DTM, shapes and VBA tools for the analysis and creation of drainage areas in a project.

INTRODUCTION

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. The tools associated with the DTM are not related to and are independent of GEOPAK Drainage.

ACCESSING

The DTM tool frame can be accessed from the *MS Menu*: *Applications>Geopak Road>DTM Tools*. Then clicking on the DTM menu icon, the user can access the DTM tool bar depicted below.

DTM

Settings   Extract   Build   Edit   Drape   Load   Reports   Analysis   Utilities

ANALYSIS TOOLS

Tools	Analysis
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 you 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.

ANALYSIS TOOLS: DRAINAGE

These are available tools within the dialog.
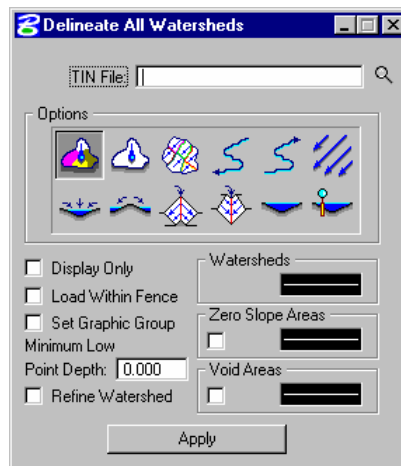
Delineate All Watersheds	Displays the watershed boundaries that exist within a DTM.

	A Watershed is defined by either a low point within the TIN or a low edge point along the TIN hull. MicroStation shapes may be created for each watershed contained in the TIN. Two important options to consider are: • **Minimum Low Point Depth**: A DTM has many low points in which drainage areas could be delineated, but many of depth are small dips in the TIN file in which water will keep flowing. GEOPAK determines each low point, then determines if water placed at the specified depth overflows these dips in the DTM and flows into adjacent triangles. If it does not flow into adjacent triangles, it is identified as a low point and the drainage area is delimited. The designer must use the appropriate criteria to set this value. • **Refine Watershed**: this option processes the entire TIN evaluating ridges, sumps, and low points and determines more precise flow boundaries within the triangles. It does not retriangulate, more specifically it insert lines within the TIN that actually represent flow divides.
Delineate Watershed	The Delineate Watershed tool further delineates watersheds at any location within the TIN. A data point representing the pour point of the watershed is indicated and the contributing watershed area is computed and delineated.
Drainage Patterns	Evaluates the flow paths contained in a region of the TIN, Model or Object. This tool performs a downstream trace from the centroid of each triangle contained in the region specified.
Downstream Trace	Trace tool delineates the flow path downstream from a given point in the TIN. The indicated path follows the steepest descent from the point through the TIN terminating at a low point or the edge of the TIN.
Upstream Trace	Delineates the flow path upstream from a given point in the TIN. The indicated path follows the steepest ascent from the point through the TIN terminating at a high point or the edge of the TIN.
Flow Arrows	Indicates the direction of flow within the triangles for a given region of the TIN.
Delineate Low Points	Locates all low points within a region of a TIN. A flow arrow is placed and the text "LP" is placed at the triangle vertex. Text placement utilizes the current MicroStation text parameters.
Delineate High Points	Locates all the high points within a region of a TIN. A flow arrow is placed and the text "HP" is placed at the triangle vertex. Text placement utilizes the current MicroStation text

	parameters.
Delineate Ridge Lines	Indicates the ridge lines within a region of a TIN. A ridge line is defined as a triangle edge where the flow on each side of the edge is away from the edge.
Delineate Sump Lines	Indicates the sump lines within a region of a TIN. A sump line is defined as a triangle edge where the flow on each side of the edge is towards the edge and is found in areas of concentrated flows such as streams and ditches.
Surface Ponds	Delineates the area(s) of ponded water within the specified TIN.
Pond Analysis	Traces a point downstream to a low point and fills it giving the volume, maximum depth, and maximum elevation. In addition, the pond delineation is graphically displayed.

LAB EXERCISE: ANALYSIS OF THE PROPOSED DTM

Accessing DTM Tools

1. Execute *c:\data\geo\VDOT\drain1\LAB02_V8ad.EXE.*

2. Open the MicroStation file, *c:\data\geo\VDOT\drain1\h17682da.dgn*.

3. Access the GEOPAK DTM Tools frame. *(Applications > GEOPAK Road > DTM Tools)*

Reading Elevations with the Height Tool

1. Access the DTM Menu tool bar.

2. Select the **Height** tool from the DTM Menu. *(DTM > Analysis > Height)*
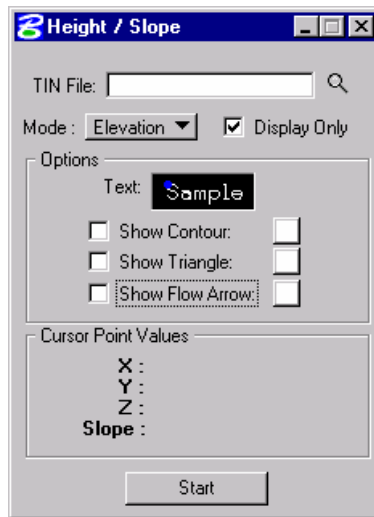
3. Select the proposed tin file: *proposed.tin* located in *c:\data\geo\VDOT\drain1*

4. 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.
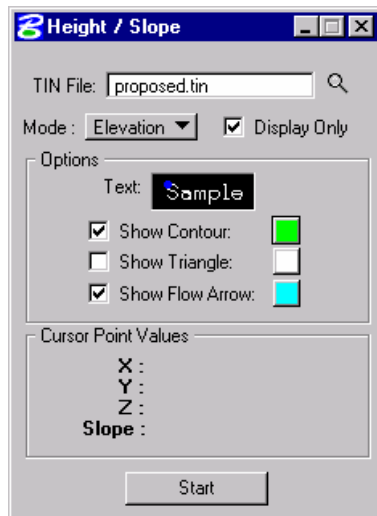
5. Click the **Start** button and scan the surface model with the cursor to display the values for xyz and the slope on the DTM at the cursor point.

6. Close the height/slope dialog.

Dynamic Proposed DTM Profiles

1. Select the **Profile** tool. from the DTM Menu**.** *(DTM > Analysis > Profile)*
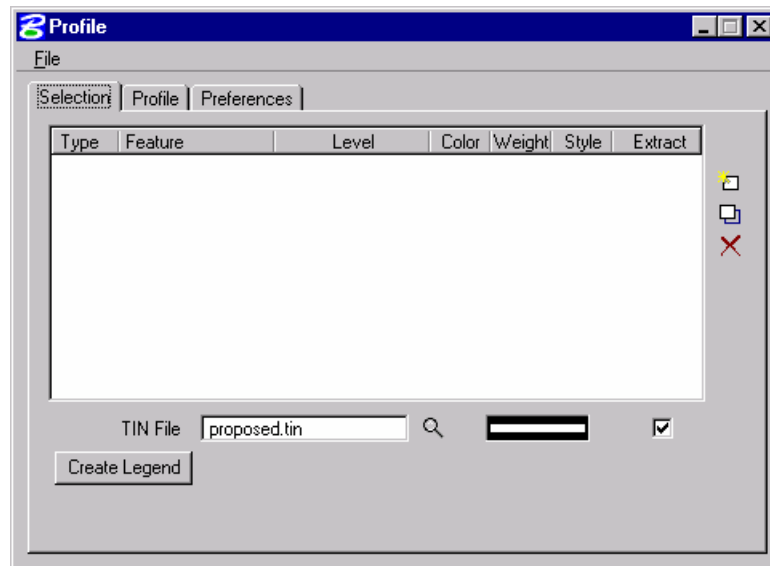
2. Use the **Selection** tab to identify your surface model (TIN File) which you will generate a profile from, and the symbology for your profile.

3. Select the Tin File :**proposed.tin** by using the **Select Files** icon.

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**.

5. Click the **Add** icon (top icon next to the list box) to add the feature to the collection box.
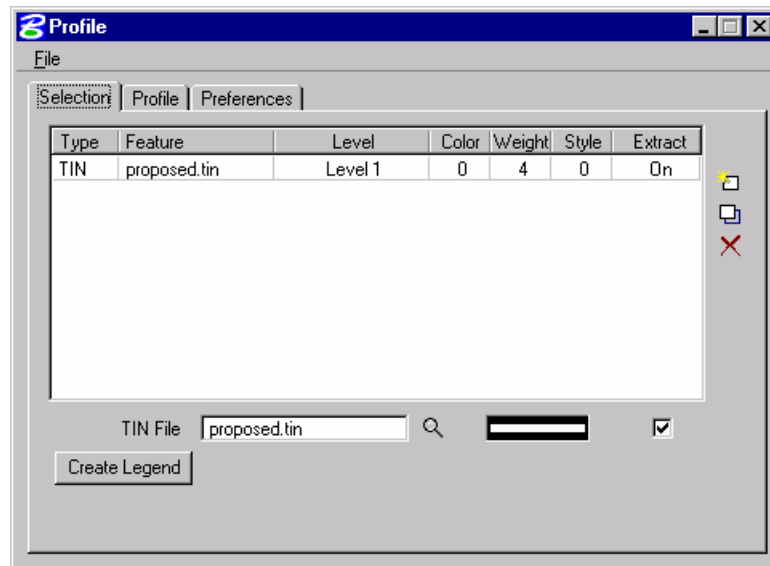
6. Select the **Profile** Tab.

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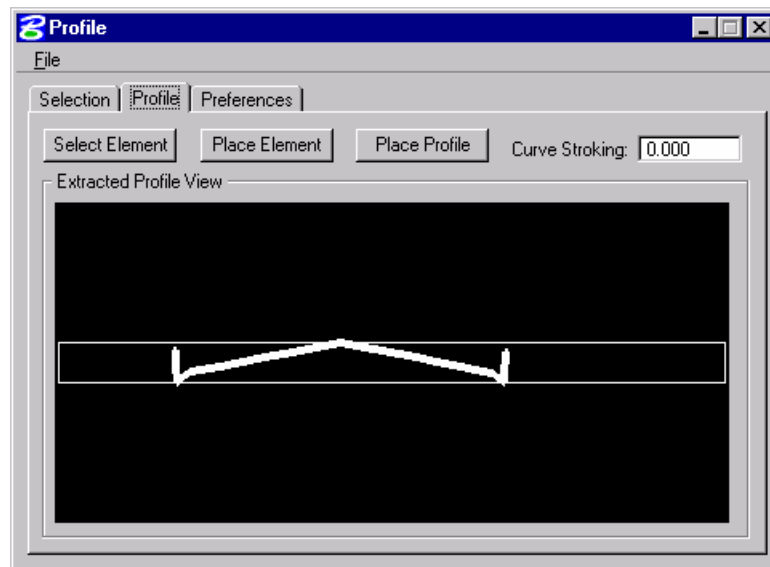
8. You can use the **Preferences** tab to set up a grid for your profile if desired.

9. Close the profile dialog.

Analyzing Drainage Flow Pattern

1. Select the Drainage Tools from the DTM Menu. *(DTM > Analysis > Drainage Tools)*

2. Select the Drainage Patterns icon and fill the dialog box as shown below.
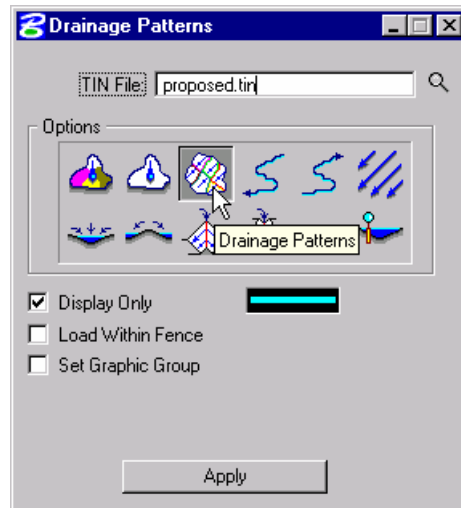
3. Click the **Apply** button. The drainage flow lines are drawn into the design file.

4. Close the drainage patterns dialog.

Locating Low Points

1. Select the **Delineate Low Points** icon and populate the dialog as shown below.
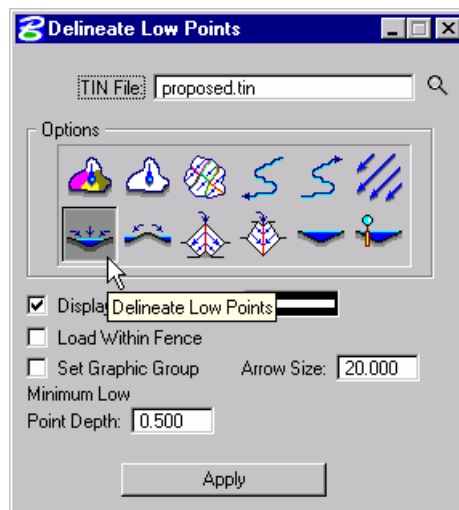
2. Click **Apply** to review the low points location in the Microstation file.

3. Close the delineate low points dialog.

Delineating Drainage Areas

1. Select the **Delineate All Watersheds** icon and populate the dialog as shown below.
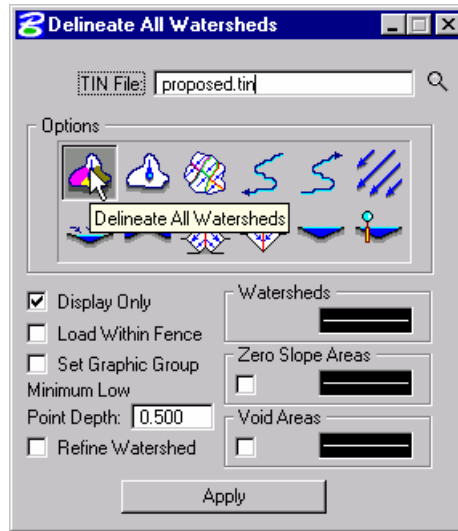
2. Click **Apply** and review the tentative watersheds for the pavement drainage.

3. Close the delineate all watersheds dialog when done.

Note   This process will define all the "mathematically" possible drainage watersheds. The designer will need to define the final drainage areas for the project.

Scouting for Pond Locations

1. Select the **Surface Ponds** icon.
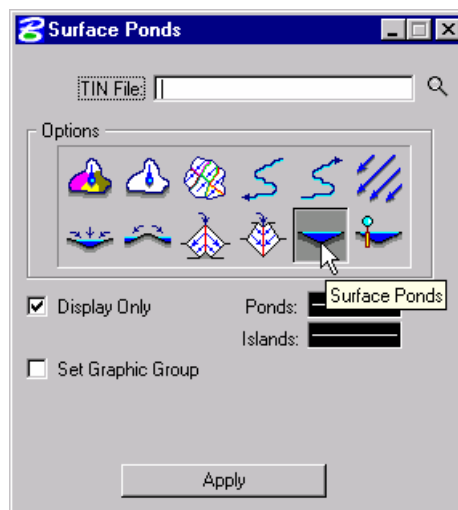
2. Select the Tin File: *survey.tin* located in *c:\data\geo\VDOT\drain1*. The proposed DTM does not cover the areas where possible pond locations area available. Setup the dialog as shown below.
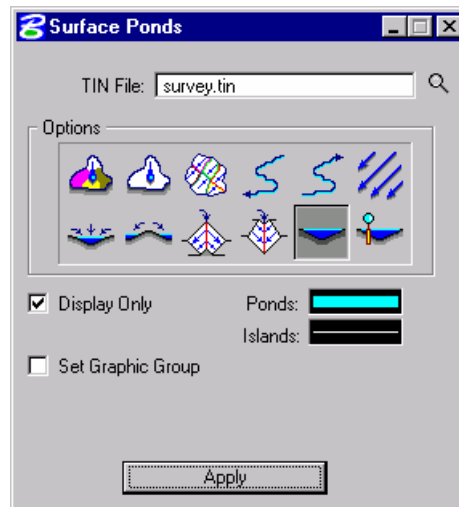
3. Click **Apply**. Geopak will display the possible areas where ponded runoff is located.

Analyzing the Pond Locations

1. Select the **Pond Analysis** icon.

2. Click **Apply**.

3. Select a point inside one the delineated possible pond locations.
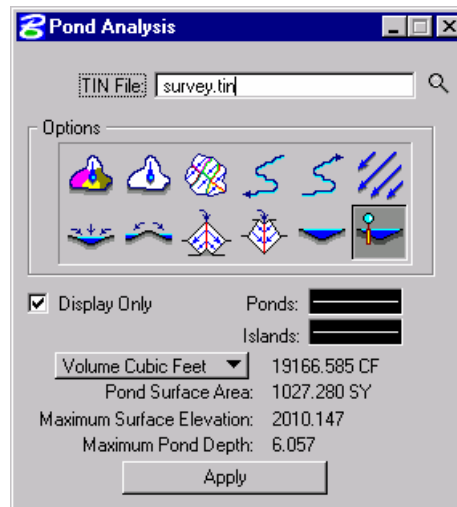
4. Following the previous steps, analyze other possible pond locations.

5. Close the pond analysis dialog when done.

LAB EXERCISE: PLOTTING PROFILE ELEVATIONS ALONG THE CENTERLINE

Loading th VBA Application

1. Load the *DrawProfInfoInPlan.mvba* macro located in
    *c:\data\geo\VDOT\drain1\standards\Geopak. (MS Menu: Utilities > Macro > Project
    Manager).*
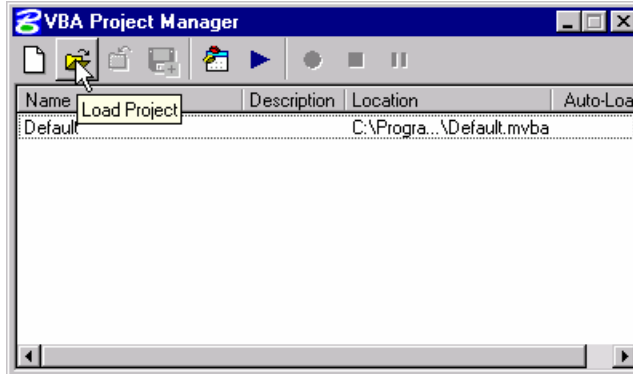
2. Highlight the VBA application and click on the Macros button.
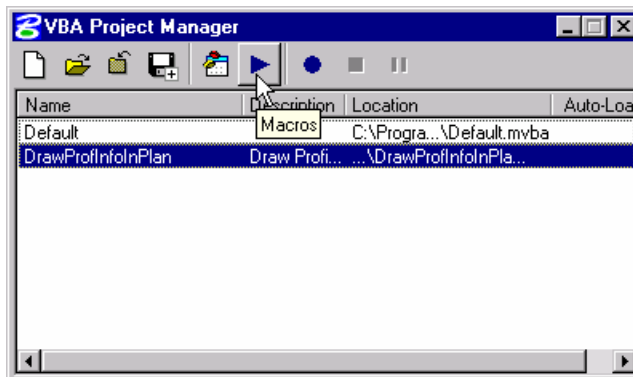
3. Execute the loaded macro by clicking on the **Run** button.
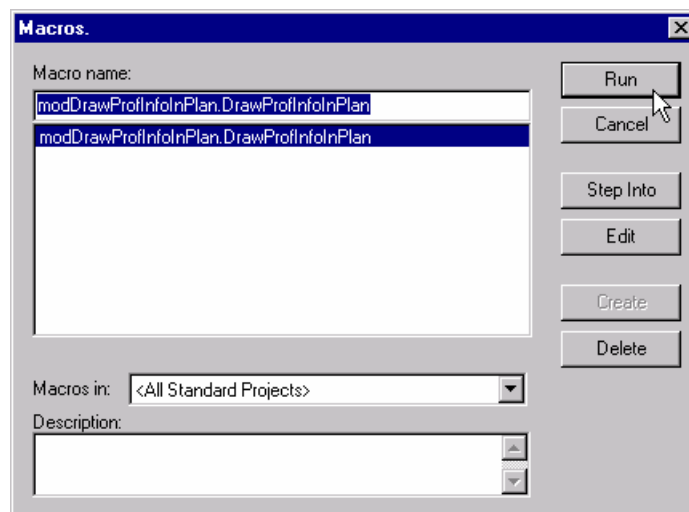
4. Setup the **Elevations** tab part of the dialog as shown below.
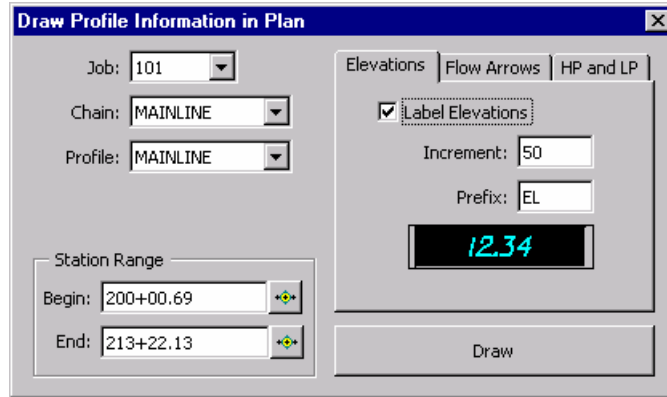
5. Double click on the text box to setup the text size and level symbology.
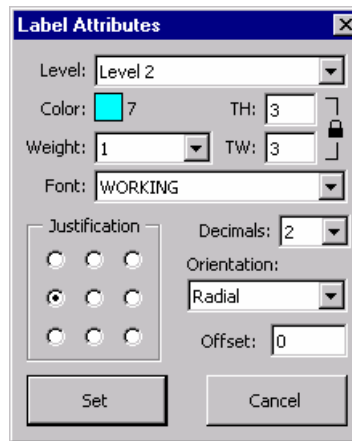
6. Click **Set** when done.

7. Select the **Flow Arrows** tab.

8. Attach the Microstation cell library that contains the flow arrow cell : *flow_arrow.cel* located in *c:\data\geo\VDOT\drain1\standards\Geopak. (Ms Menu: Element > Cells).*
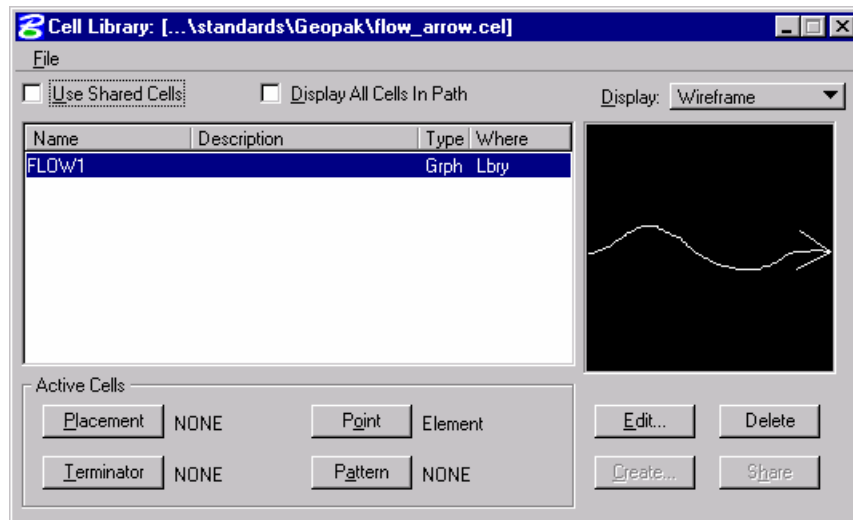
9. Close the Cell library dialog.

10. Setup the **Flow Arrows** part of the dialog as shown below.
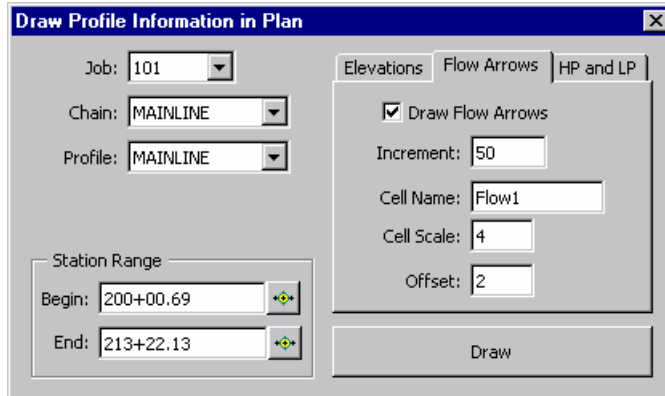
11. Select the **HP and LP** tab and setup the dialog as shown below.
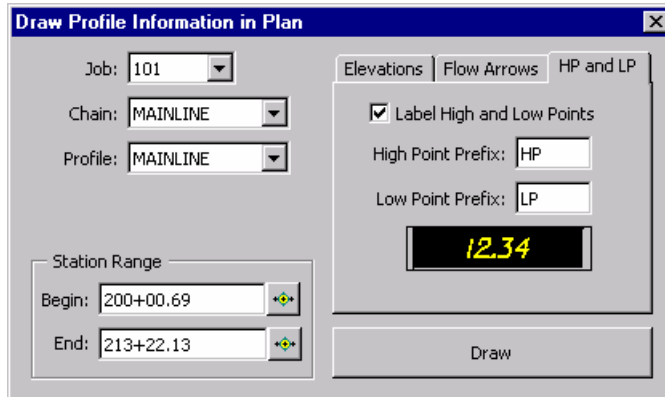
12. Double click on the text box to setup the text size and level symbology.

13. Click **Set** when done.

© 2007 VDOT VDOT GEOPAK Drainage
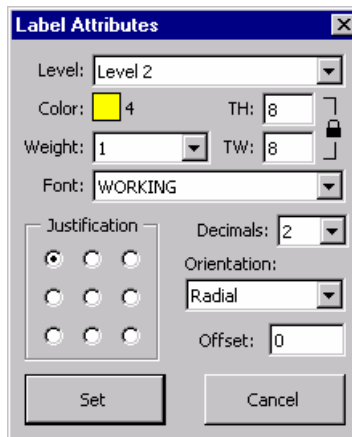
14. Click the **Draw** button. Geopak will place the profile elevations along the centerline and show the direction of flow. The low and high points of the profile are also labeled.
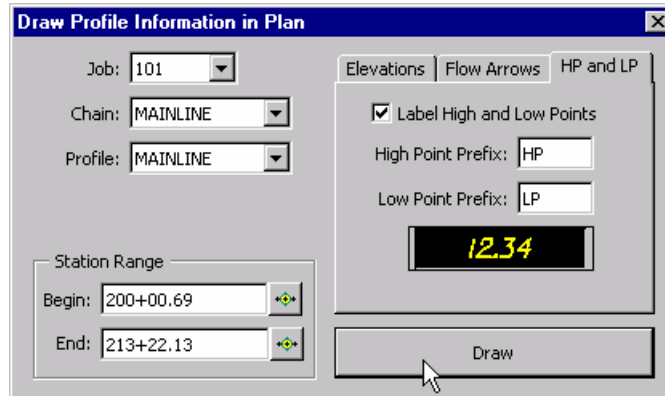
15. Close the dialog when done.

16. Exit Microstation.

LAB EXERCISE: SETTING UP LAND USE AREAS

Creating Land Use Shapes

1. Open the MicroStation file, *c:\data\geo\VDOT\drain1\h17682da.dgn*.

2. Create Microstation shapes depicting the land use areas of the project: pavement, grass, building, etc. *(MS Menu: Tools > Main > Polygons)*

Assigning Proper Land Use Symbology

1. Open Geopak Drainage. *(MS Menu: Applications > Geopak Drainage > Drainage).*

2. Open the Geopak Drainage Library tool. *(Drainage : Project > Drainage Library)*

3. Select the VDOT Drainage Library: *VDOT.dlb* located in
   c:\data\geo\VDOT\drain1\standards\Geopak. (Drainage Library: File > Open).
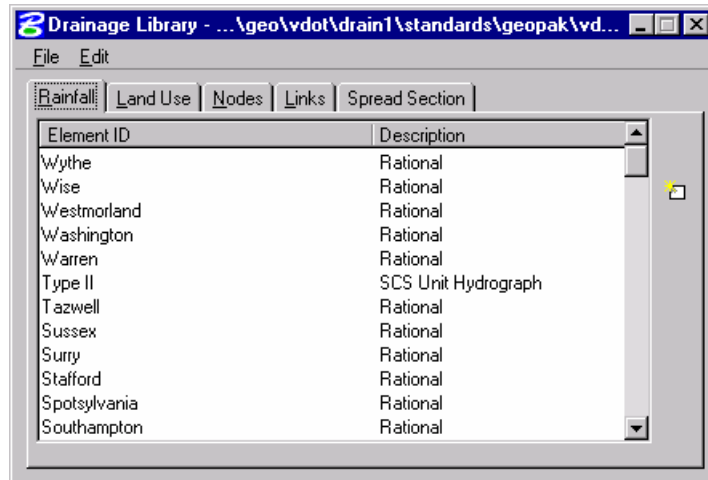
4. Close the Drainage Library dialog.

5. Select the Land Use tool *(Drainage: Component > Land Uses)*

© 2007 VDOT
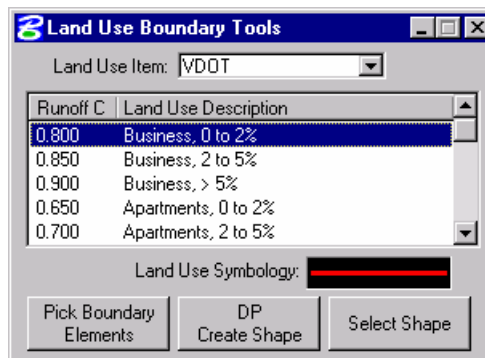
6. Select the Impervious land use.
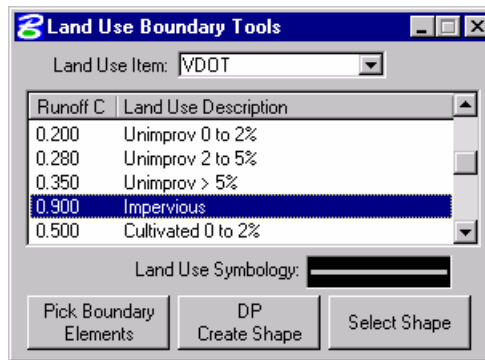
7. Click the **Select Shape** button.
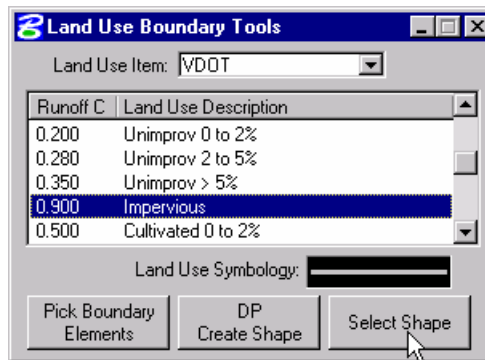
8. Select the previously placed Microstation shape that depicts the pavement portion of the project. A new level symbology will be assigned to the shape.

9. Repeat the process with all the shapes defined and assign the proper land use.

10. Close the dialog when done.

11. Exit Microstation.

This page left intentionally blank.

3 Starting a Drainage Project using GEOPAK Road Project Manager

OBJECTIVES

In this chapter, you will learn more about:

- Setting up a project using Project Manager.

- How to utilize Project Manager as a workflow guide

- How to access GEOPAK dialogs from the Project Manager

- How Geopak Project Manager can help organize you project directories and files

- Various file types used within GEOPAK Drainage.

- Creating a new GEOPAK Drainage Project file.

- The general process for using GEOPAK Drainage to design a storm drain system.

INTRODUCTION

Project Manager is a GEOPAK tool that associates a project with its respective coordinate geometry job number, users, working directories and project files. Project Manager provides an easy workflow system that keeps records of processes run throughout the design of a project.

ACCESSING PROJECT MANAGER

To access Project Manager, select Applications > GEOPAK Road > Project Manager or click the Project Manager icon. The adjacent dialog appears.

The current directory is displayed at the top of the dialog. This may be modified by traversing to a different directory in the Directories list box. Project files (.prj) are displayed in the projects list box.

The remainder of the Project Manager dialog displays information after a project has been selected from the Projects list box. At the bottom of the dialog are the OK and Cancel buttons. To exit Project Manager, click the Cancel button. To continue in the Project Manager process, click the OK button.
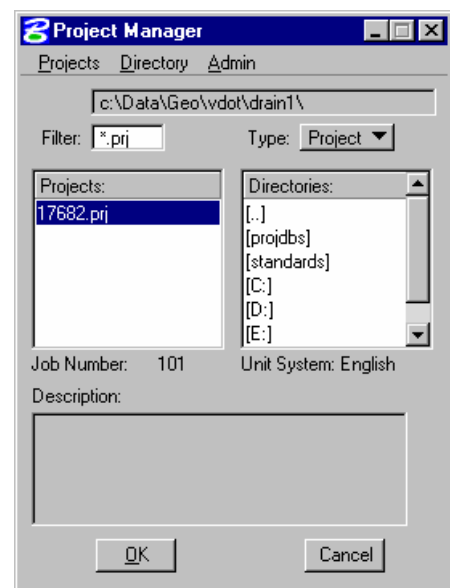
PROJECT MANAGER MENU BAR

There are three options on the Menu Bar: Projects, Directory, and Admin. Each of these choices has tools contained in the pull down.

PROJECT TOOLS

There are four options under the Projects pull down, New, Edit, Delete and Exit.

The Windows motif also keeps track of the last few processes that have been executed, which may be recalled as needed by selecting the desired process.

The **New** menu option is used to create a new project. The .prj file is stored in the directory shown in the Directory path at the top of the Project Manager dialog.
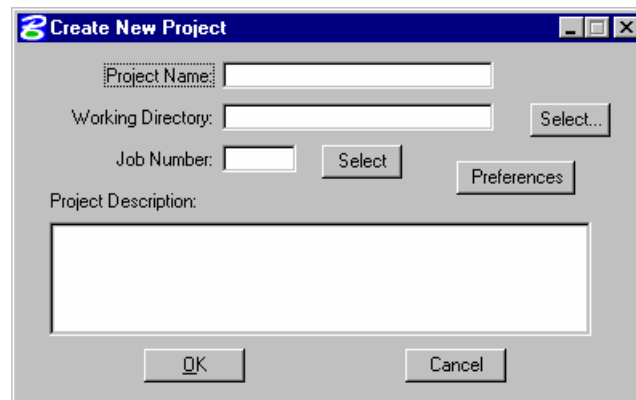
The Project Name can be any number of alphanumeric characters. The Working Directory specifies the location of the project data files and may be manually entered. In lieu of typing, click the Select button and select the appropriate directory.

> *Note* The **Working Directory** can be left blank if it is desired to have all of the MicroStation and GEOPAK files in a single directory.

The next field is for keying in the COGO Job Number or the Select button may be clicked and the appropriate COGO job number selected.

Next, click Preferences, invoking the Project Preferences dialog.
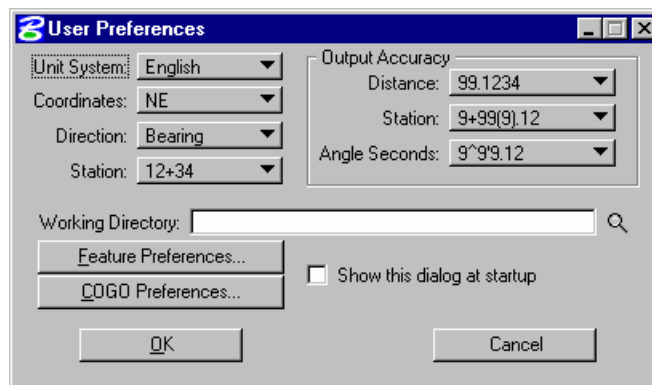
You may set the parameters for each project as well as the directories for the COGO job number, COGO input files and COGO output files. By setting these parameters for each project, you can open a

 VDOT GEOPAK Drainage

design file in one directory and then open another design file in a different directory and still access the correct .GPK file and also save COGO input and output files to the proper directory. Simply choose the project and all parameters are automatically set. After all the information is entered, click OK or, if you want to abort, click Cancel.

> *Note*   If these fields are left blank, the project working directory is used.

Shown below is a sample project that includes a project description, which may be keyed in at the bottom of the Create New Project dialog.
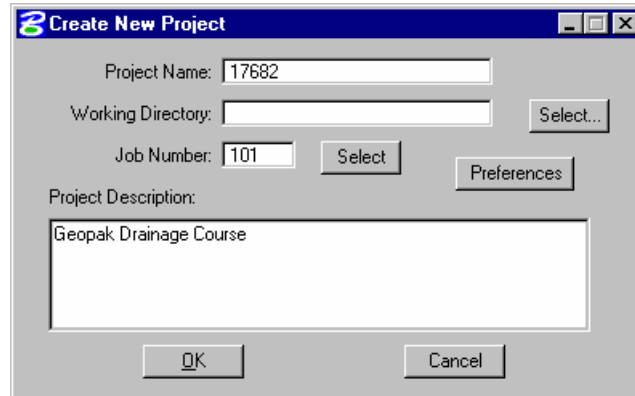
Use the Edit menu option to change any settings associated with the currently selected project.

Use the Delete menu option to delete any project that has been stored. Highlights the project in the Projects list box and select Delete.

Use the Exit menu option to close the Project Manager and write the settings to a resource file.

DIRECTORY TOOLS

There are two options under the Directory pull down, Create New Directory and Current Working Directory.

The Create New Directory option creates a new directory on the disk within the current directory. If the Current Working Directory option is chosen, the directory path in Project Manager is changed to that directory.

ADMINISTRATION TOOLS

Use the Administration menu option to set a password on a project, if needed.

PROJECT USERS DIALOG

Once a project is highlighted and OK is selected, the Project Users dialog appears.
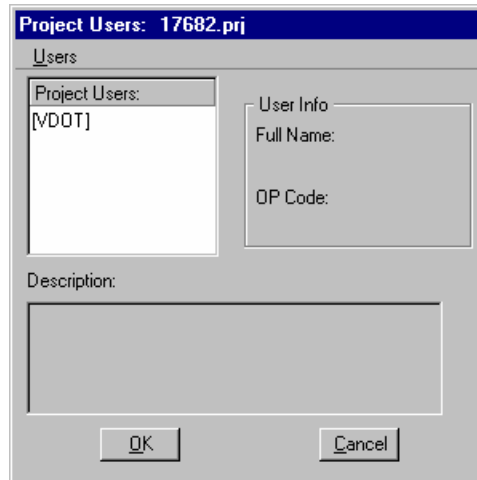
This dialog has three sections: Project Users, User Info, and Description. The Project Users list displays a list of users that have been created to work with any project that resides in the current projects home directory. The names shown in Project Users will be the user id's of the people working on that project as specified in the Start Job dialog. Within the User Info group box, the Full Name field further identifies the user, and displays the full name of the user that is currently selected. The OP Code field displays the GEOPAK Operator Code of the currently selected user. The GEOPAK Operator Code is used for all coordinate geometry operations during this session. The Operator Code, along with the Job Number is utilized whenever a COGO input or output file is created by the software. The Operator Code is typically the user's initials. The Description field displays the description of the currently selected user.

Four tools are supported on the Project Users pulldown: New, Edit, Delete, and Exit.

ROAD PROJECT DIALOG

After at least one user has been defined, click OK or double click on a project user, which opens the Project Manager dialog.
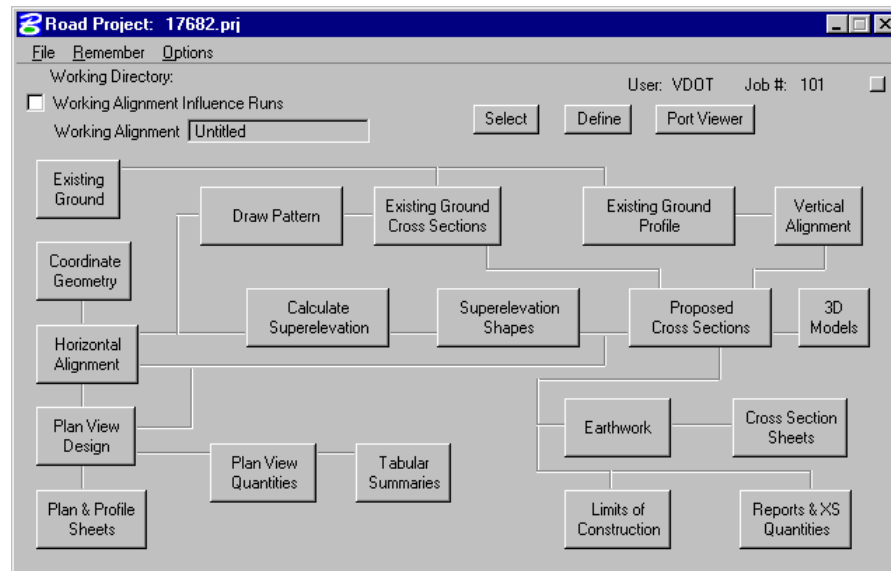
The top of the dialog displays the Working Directory, Working Alignment (if defined), User and GEOPAK Job Number. In addition, a toggle for Working Alignment Influence Runs is also supported. The bottom portion of the dialog displays the various processes supported during the design process. The small square in the upper right corner will condense the dialog as depicted in the graphic below.
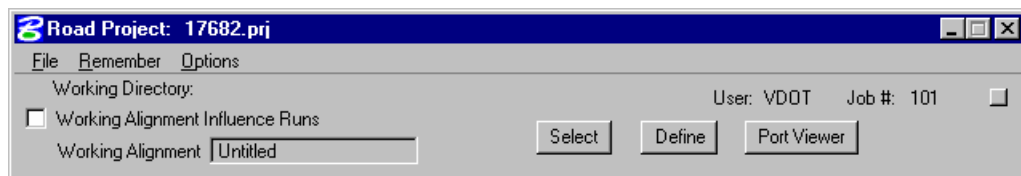
ROAD PROJECT DIALOG MENU BAR

There are two pull down menu bar options: File and Remember. When the File option is selected, Close and Exit are supported.

If you select the Close option, you are returned to the Applications dialog. If you select Exit, you are exited from the Project Manager.

When you select the Remember option is selected, you can instruct the software to remember the Project or User in subsequent sessions. For example, if both toggles are activated, and the Project Manager is completely closed (all dialogs), the invocation of the Project Manager immediately invokes the Road Project dialog (flow chart) and utilizes the project name, user name, etc., which were active when the Remember toggles were selected.

If you activate only the User toggle, you are returned to the Project dialog in later sessions. If you activate only the Project toggle, you are returned to the Project Users dialog in subsequent sessions. This option is useful when numerous users are working on one project.

When you enable the Auto Sink option, the Road Project flow chart sinks behind all open views when any tool is selected from the flow chart.

WORKING ALIGNMENT

The concept of a working alignment enables the designer to organize a project and to access project information without continually typing the required information. On a simple project, only one working alignment may be needed. On a more complicated project, an unlimited number of working alignments may be defined. With each working alignment, the designer can define associated data, i.e., name of cross section and plan view files, element symbology, etc. The designer can easily change from one working alignment to another by highlighting the desired alignment listed in the Select dialog. Once the working alignment is selected, all previously stored data is utilized in subsequent processing. Three tools relating to working alignments are located at the top of the Road Project dialog:

- Select

- Define

- Working Alignment Influence Runs (toggle on left side of dialog)
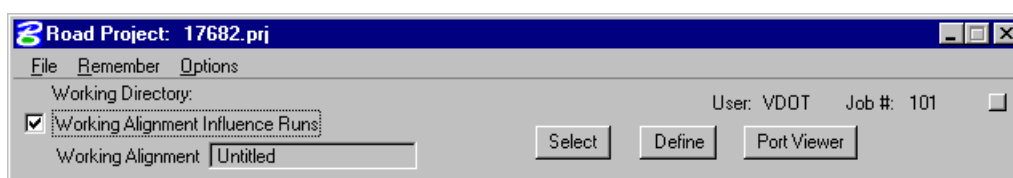
SELECT OPTION
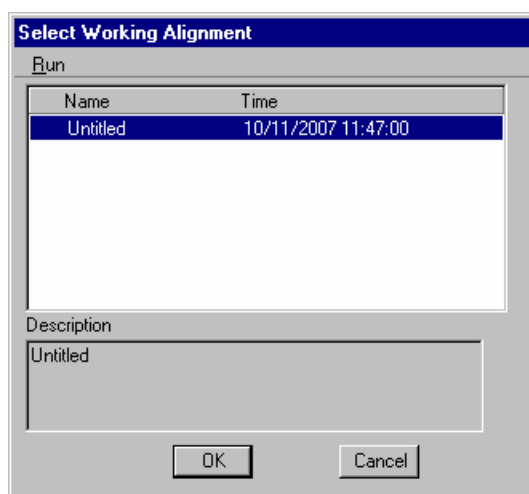
When you click the Select button, the Select Working Alignment dialog appears. If no working alignments have been defined, UNTITLED appears in the Run List box. If working alignments have been defined, they are listed with the last run time. The description of the working alignment can be seen in the bottom of the dialog when each Name is highlighted. To select a previously defined working alignment, highlight the run from the list, then click OK at the bottom of the dialog. Double clicking on the Name also selects a previous working alignment for subsequent processing. Clicking Cancel closes the Select Run dialog without any File selection. Several Run options are supported as detailed in the following table.

New	Create a new working alignment
Copy	Copy a run from an existing Project, a User in the same project or Run created by the current user.

Modify	Change the name of the existing working alignment.
Delete	Delete an existing working alignment.

DEFINE OPTION

Once a Run has been selected, clicking Define opens the Working Alignment Definition dialog.
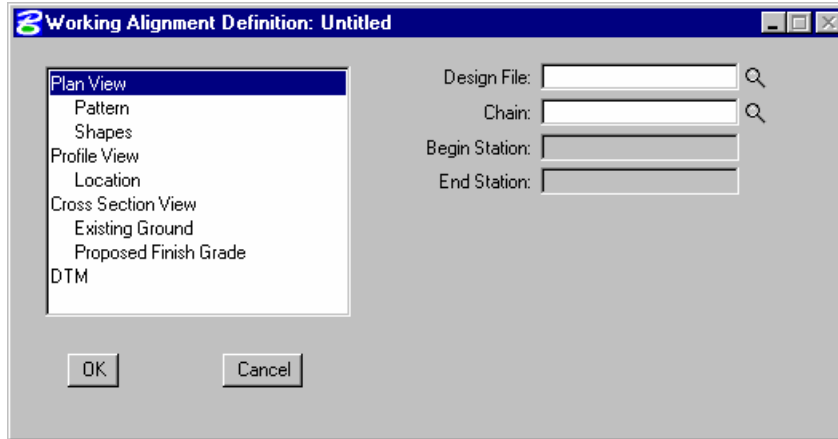
The information that can be associated with a working alignment is listed in the left portion of the dialog. As each option is chosen, the right side of the dialog will change to reflect the information needed as can be seen in the dialog above after Plan View was selected.

All information entered in these fields can be used in subsequent processes run from Project Manager. In the beginning of a project, much of this information will not be known, but as you go through the design process, it can be added to the working alignment definitions.

For example, as soon as a chain has been stored in COGO, you can enter that information in the Plan View fields on the right side of the dialog. For a more complete explanation of each option shown, please see *On-Line Help.*

PROJECT MANAGER PROCESS
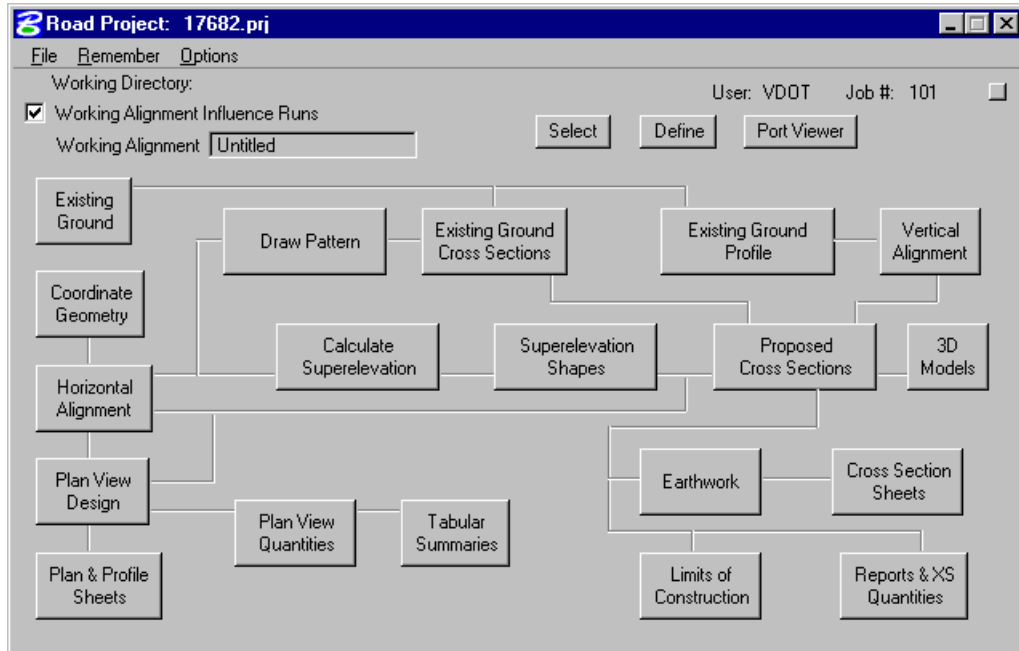
The primary Project Manager dialog is shown above. The advantage of utilizing the Project Manager rather than selecting functions directly from the Road menu is that pertinent information stored within the Project Manager is automatically displayed within the invoked dialog. Therefore, job numbers, chain names, stationing, file names, and data associated with the project do not have to be typed in each time a dialog is utilized. However, you have the option to change the fields.

Many of the Project Manager processes function identically to their corresponding dialog's invocation from the Road menu. Some processes invoke the Select Run dialog prior to invoking the actual dialog. The Select Run dialog enables you to set up different options to use in alternative design choices or different typicals.

LAB EXERCISE: PROJECT MANAGER

Create a GEOPAK Road Project

1. Execute *c:\data\geo\vdot\drain1\LAB03_V8ad.EXE*

2. Open the MicroStation file *C:\data\geo\vdot\drain1\h17682.dgn.*

3. Select the Road Tools tool frame *(Applications > GEOPAK Road > GEOPAK Road Tools).*

4. Select the **Project Manager** tool *(Applications > GEOPAK Road > Project Manager).*

5. Ensure that the project directory is set to *C:\data\geo\vdot\drain1\.*
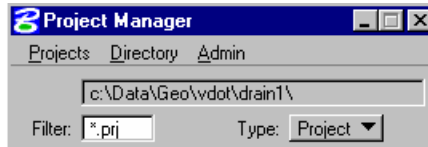
6. Select **Projects > New** and create a new project.
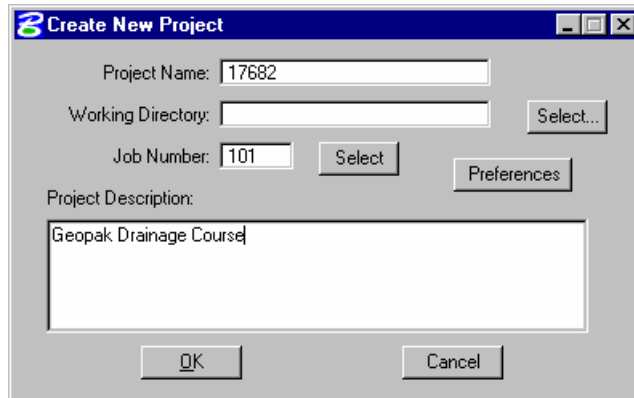
Project name:	17682
Job Number:	101
Project Description:	GEOPAK Drainage Course Project

7. Click **Preferences**. Ensure the dialog is populated as shown below.
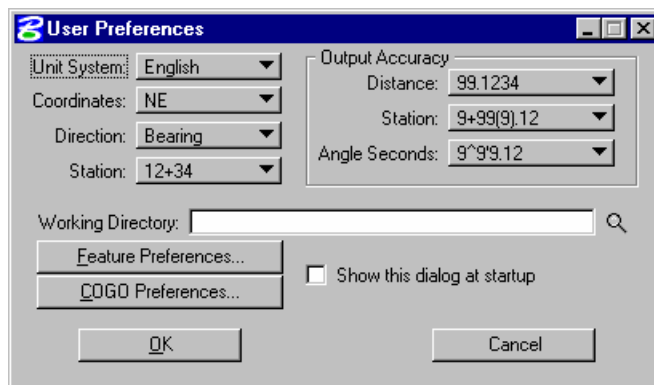
8. Click **OK**.

    After exiting the **Preferences** dialog, you will return to the **Create New Project** dialog.
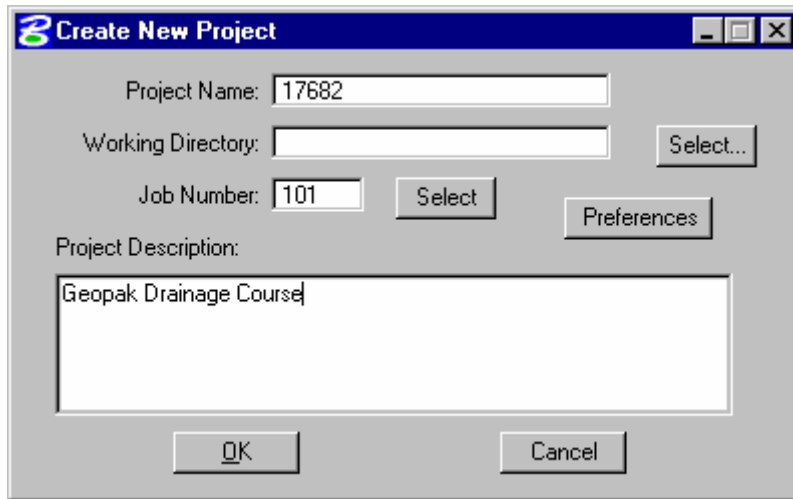
9. Click **OK**.

10. Click **OK**. An **Alert** message is displayed stating that Job 101 was not found in the Working Directory you have defined.

11. Click **Yes** to automatically create this job file. The main Project Manager dialog is now displayed and the project has been created and ready to be accessed.

Note   The previous two steps may not occur if the designer is using the gpk file previously created by the roadway engineers.

Create a User

1. Highlight **17682.prj**.

    To access the project, select the project by single clicking on name of the project in the dialog. The project is highlighted as shown in the following dialog with all pertinent information displayed.
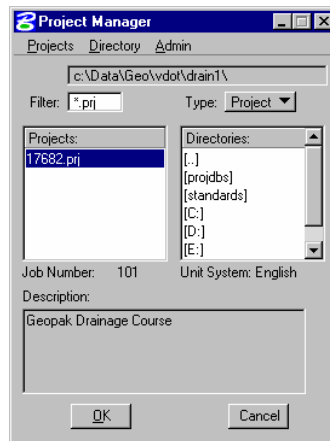
2. Click **OK**.

3. Select **Users > New**.
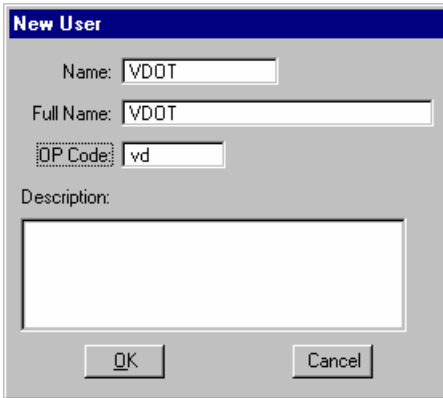
4. Enter the **New User** information.

Name	VDOT
Full Name	VDOT
OP Code	vd

5. Click **OK**. When prompted to define a password for this user, click **No**.

6. Double click the user. You now see the Main Workflow dialog.
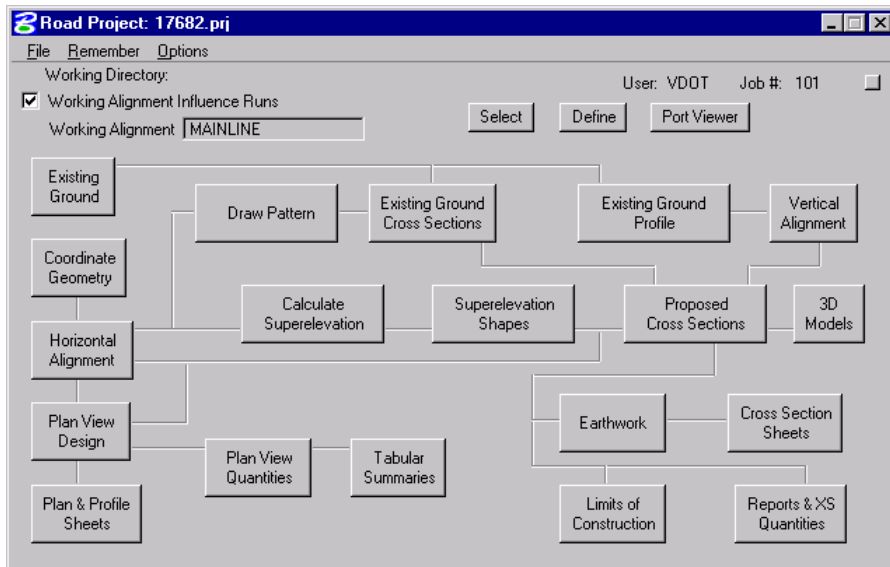
7. Use the **Remember** option to have Project Manager remember our Project and User.
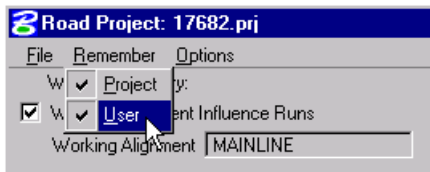
8. Exit MicroStation.

GEOPAK DRAINAGE

GEOPAK Drainage has four major components within each project: the GEOPAK Drainage File (*.gdf), MicroStation 2D design file, Drainage Library, and the MicroStation Cell Library. In contrast to the GEOPAK Drainage File and MicroStation graphics file, the Drainage Library and Cell Library are utilized for numerous projects, as they contain the Department's standards. The designer only has to select the desired items for each specific project.

File Type	Description
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.
MicroStation Design File (*.DGN)	This two or three dimensional graphics file is utilized for the visualization of the drainage project and definition of certain drainage features using MicroStation graphic elements. The only data stored within this file are attributes attached to the graphics for reference to the GEOPAK Drainage file (.gdf).
Drainage Library (*.DLB)	The Drainage Library is utilized for numerous projects, as it contains the standards for an entire organization. The Drainage Library contains the Rainfall Parameters, standard inlet types, standard pipes configurations, spread sections, and land use symbology tables. All of these items are merely referenced by each project to accommodate standardization and information sharing among projects.
Drainage Cell Library (*.CEL)	Drainage cells are the plan views of the Department's drainage structures such as inlets and manholes. The Cell Library contains the commonly used structures. The Nodes used in the Drainage Library refer to appropriate cells in the Cell Library.
Design and Computation Manager database (*.DDB)	The Design and Computation Manager database is used for the Department as a tool to expedite the drafting of plans using the VDOT CADD Standards. The D&C Manager also allows the designer to compute plan view quantities based on MicroStation level symbology.
Geometry Database (*.GPK)	The gpk file is a database that stores all the geometric information of the project. All alignments (chains), profiles, points, curves, spirals, parcels are saved into a single file per project. Then, Geopak Drainage can refer to the nodes location by station and offset from a particular chain and get or calculate elevations from a profile.

MicroStation Design File (*.DGN)

Drainage Project File (*.GDF)

Project Preferences

Project Drainage Features

Drainage Library File (*.DLB)

Rainfall Parameters

Land Use Symbology

Inlets

Pipes

Drainage Cell
Library (*.cel)

D & C Manager
Database (*.ddb)
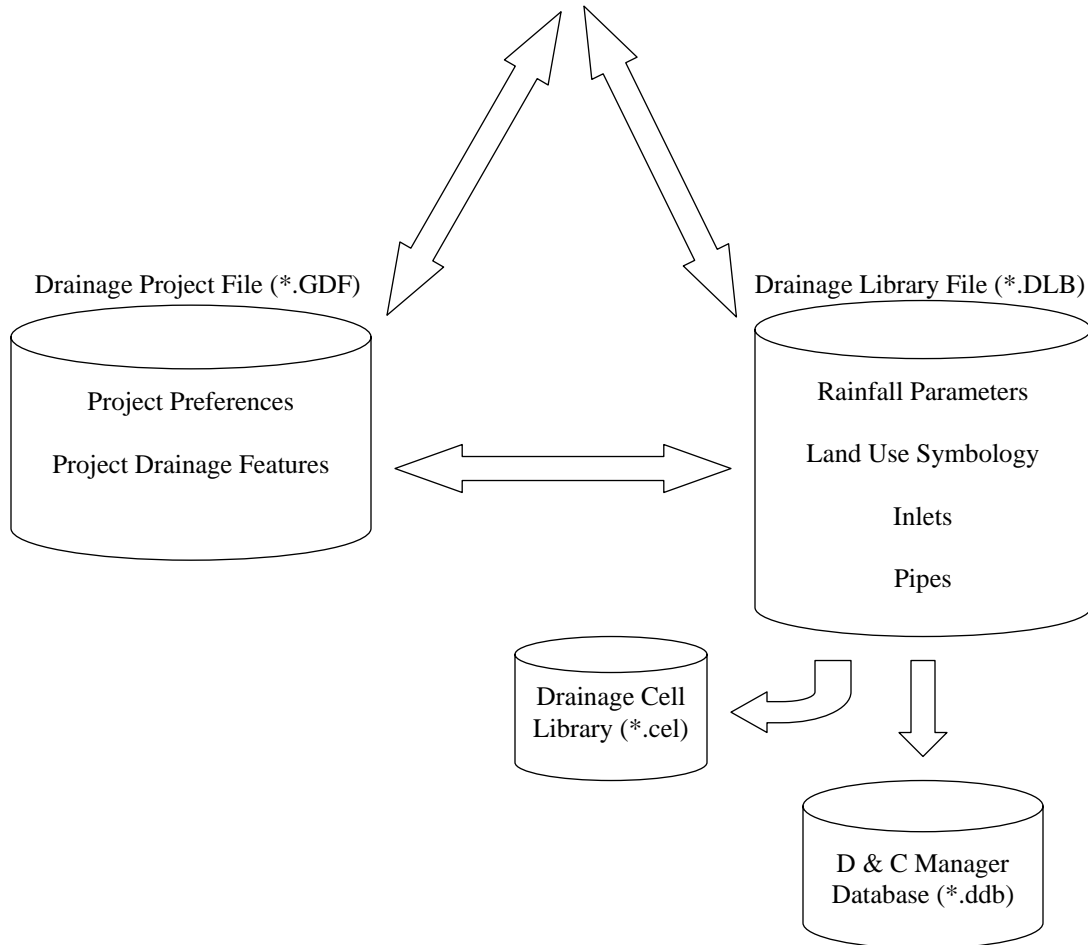
DRAINAGE COMPONENTS

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 sub areas representing homogeneous features such as soil types and land uses ("C" values), thereby allowing composite "C" value calculations.

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

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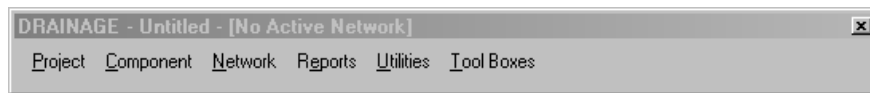
Each menu selection accesses GEOPAK Drainage project information, feature placement and tools necessary to complete a GEOPAK Drainage project. Tool frames and toolboxes are also supported for all functions and are accessed via the Tools pull down.

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).
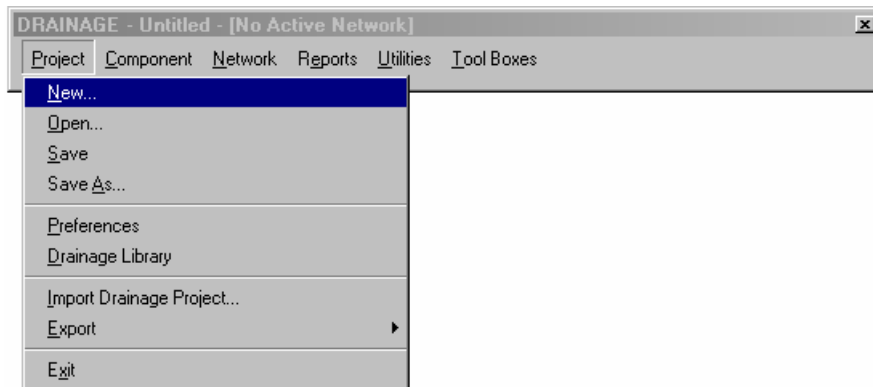
COMPONENT MENU SELECTIONS

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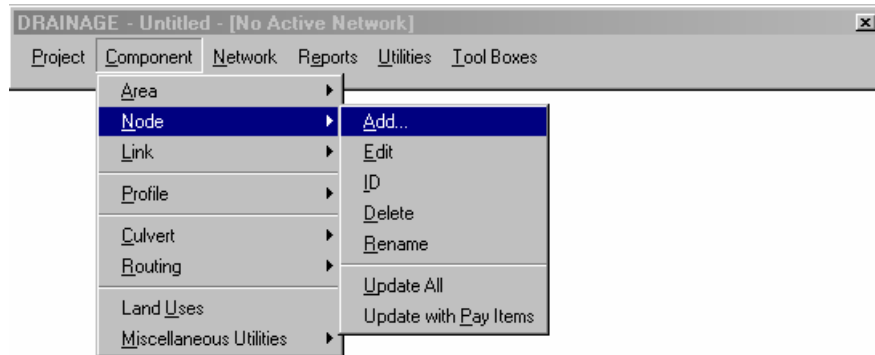
NETWORK MENU SELECTIONS

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.
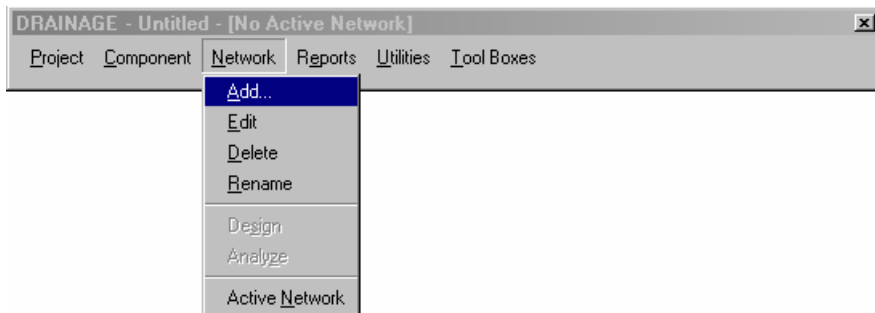
REPORTS MENU SELECTIONS

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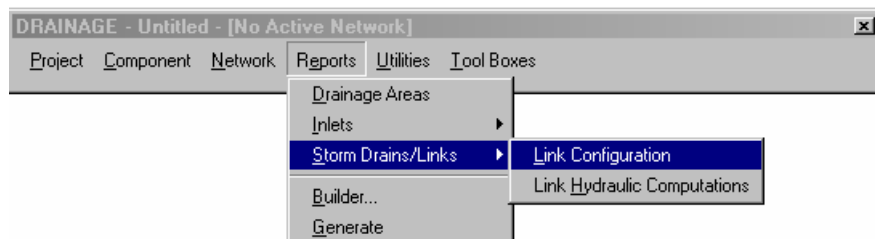
TOOLS MENU SELECTIONS

The Tools menu selections provide access to all Tool Boxes, Navigator, Labeler, Culvert, Routing and DTM Drainage Tools.
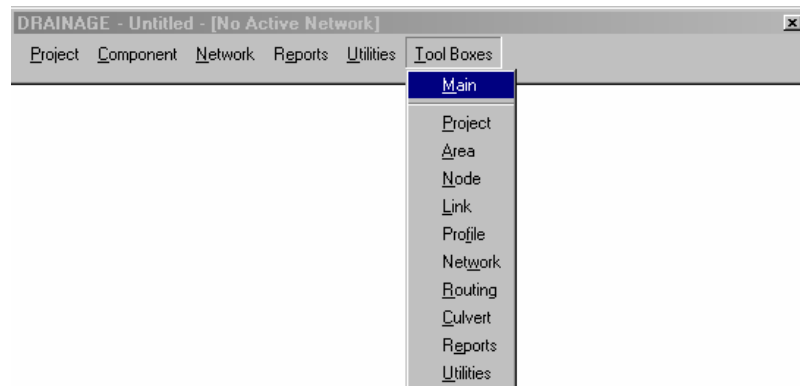
GENERAL DESIGN PROCESS FOR STORM DRAINS

The design of a storm drain system is usually an iterative process. Inlets locations and pipe sizes are selected. The designer then checks if the system meets the Department's criteria and changes pipe sizes and other items as needed. The need to check if the system meets criteria is not avoided by using GEOPAK Drainage. This document describes several ways that designers can use GEOPAK to check and identify where the system need to be changed. The following is a general outline for creating storm drain systems in GEOPAK. Details are provided in other chapters.

1. Start a New Project file (.gdf) by setting **Project Preferences**.

2. Select Inlet locations and types by adding **Nodes**.

3. Define the **Areas** to each node

4. **Link** the inlets together with pipe segments.

5. Identify the **Network** of pipes and inlets.

6. Use GEOPAK to **Design** the system.

7. Check if the design meets criteria. This can be done by several methods such as profiles, querying, warnings, or output reports.

8. Edit the system, re-compute the hydraulics and check if criteria are met.

9. Print the VDOT Storm Drain Tabulation Form.

> *Note*   It is recommended that the designer should create or use an already existing Geopak Road Project Manager file (*.prj). This will help locate other project files that Geopak Drainage may use (Design and Computation Manager ddb file, Coordinate Geometry gpk file, etc).

VDOT VBA APPLICATIONS

The Department has provided VBA applications for specific tasks during the design and plans production process of the drainage system. These VBA applications are accessed thru the MicroStation: Utilities>Macro menu. They are:

- Draw flow arrows along the roadway centerline analyzing the profile conditions.

- Draw Drainage Storm Sewer Tabs: creates an Excel spreadsheet for the VDOT Storm Tabulation form with information extracted from the GEOPAK Drainage gdf file.

- Sketch of Geopak Drainage Network: creates a jpg image with a sketch (not to scale) of the selected drainage network.

- Draw flow directional arrows along the pipes.

ACCESSING GEOPAK DRAINAGE

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 Drainage application into the MicroStation pull-down menu.

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.

GEOPAK Drainage always starts in an 'Untitled' project as noted in the MicroStation menu bar. Therefore it is necessary to always either create a new project or open an existing project prior to performing any other GEOPAK Drainage functions.

LAB EXERCISE: CREATING A NEW GEOPAK DRAINAGE PROJECT

> *Opening Road Project Manager*

1. Open the MicroStation file *c:\data\geo\vdot\drain1\h17682.dgn*.

2. Open Project Manager (*MS Menu: Applications > GEOPAK Road > Project Manager)*

3. Select project: *17682.prj* located in *c:\data\geo\vdot\drain1*, and click **OK**.
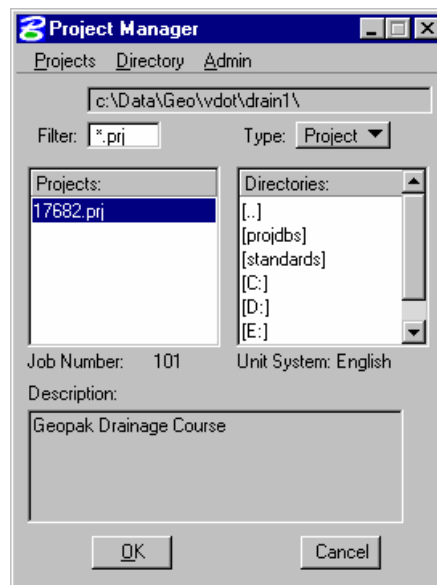
4. Access User: VDOT
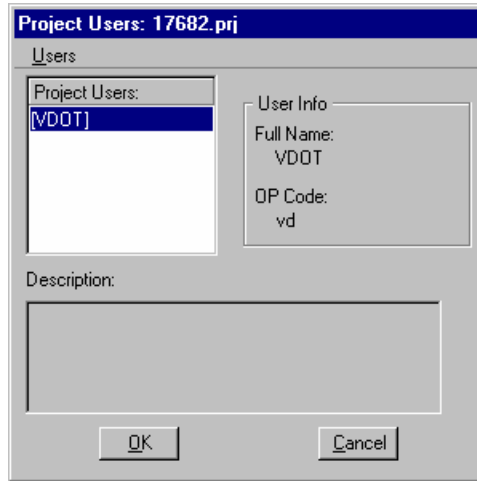
5. Click **OK.** The Road Project dialog will load. This will setup the proper directories for the location of the gpk file.
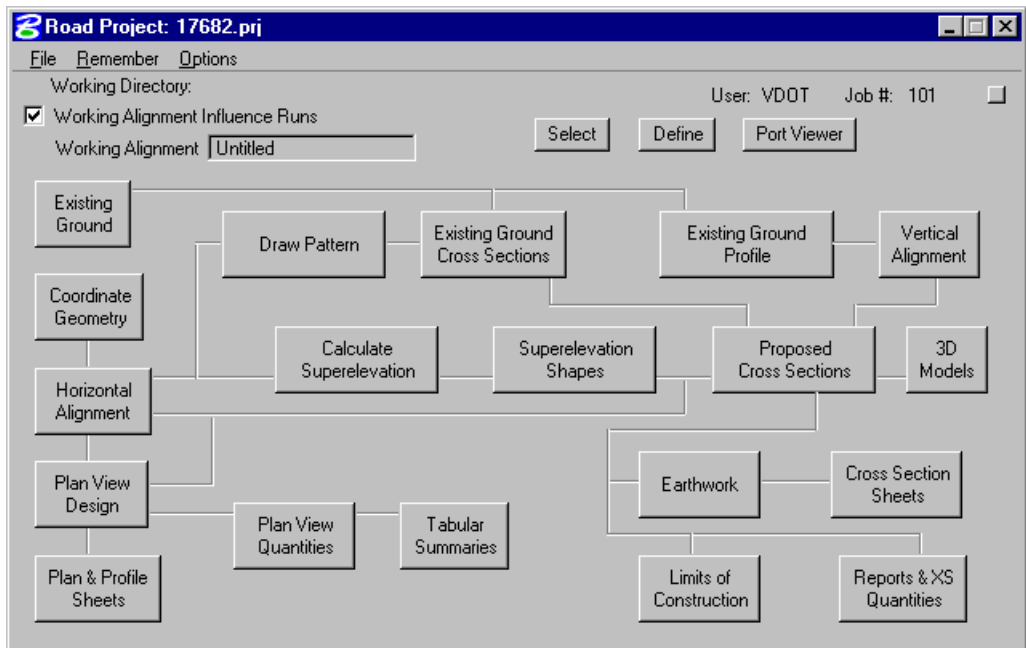
6. Minimize the Road Project dialog box.

> *Create a New Geopak Drainage Project*

1. Access GEOPAK Drainage *(MS Menu: Applications > GEOPAK Drainage > Drainage).*

   All items in Drainage can be accessed through this main GEOPAK Drainage menu. GEOPAK Drainage opens in an Untitled project file.

2. Create a new drainage project, *c:\data\geo\vdot\drain1\h17682.gdf (GEOPAK Drainage Menu: Project>New).*

   The following files are utilized for this project.

 VDOT GEOPAK Drainage

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
Reference File (service utilities)	su17682.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

3. Select the Main Drainage Tools tool frame. *(GEOPAK Drainage Menu: Tool Boxes > Main)*

The GEOPAK Drainage tool frame is similar to the MicroStation main tool frame, is 2x7, not resizable, but can be docked.

During the course you will become familiar with the various tools on this tool frame.

This page left blank intentionally.

4   GEOPAK Drainage Project Preferences

OBJECTIVES

The objective of this chapter is to 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* menu selection. The Drainage Preferences can also be invoked by selecting *Tool Boxes > Main*, then identifying the **Drainage Preferences** tool, (first column, top box).
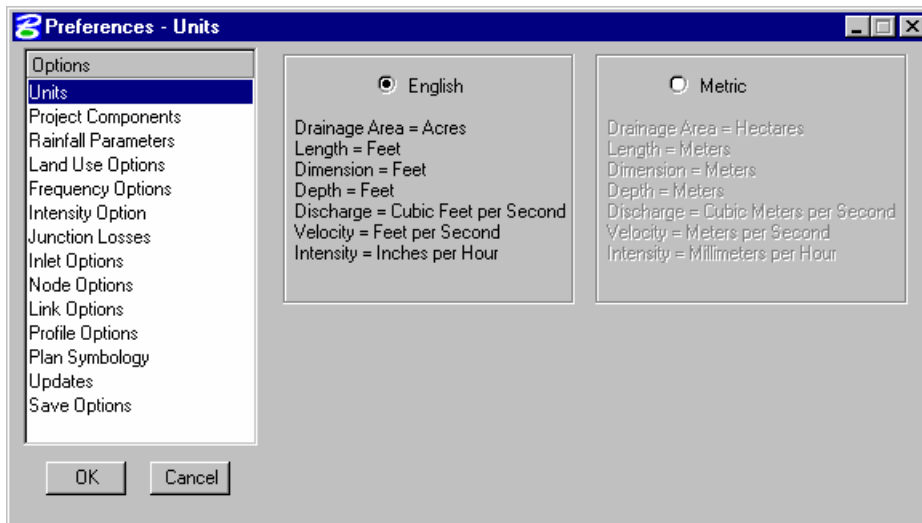
The general Options are displayed in a list box in the left side of the dialog, while the right side displays parameters for the highlighted Option. As different Options are selected in the list box, the right side of the dialog will change dynamically to reflect the parameters for highlighted Option.

UNITS

The first Preference is Units, which establishes the input and output units for the current project. Two supported options are English and Metric. FDOT supports only English units.
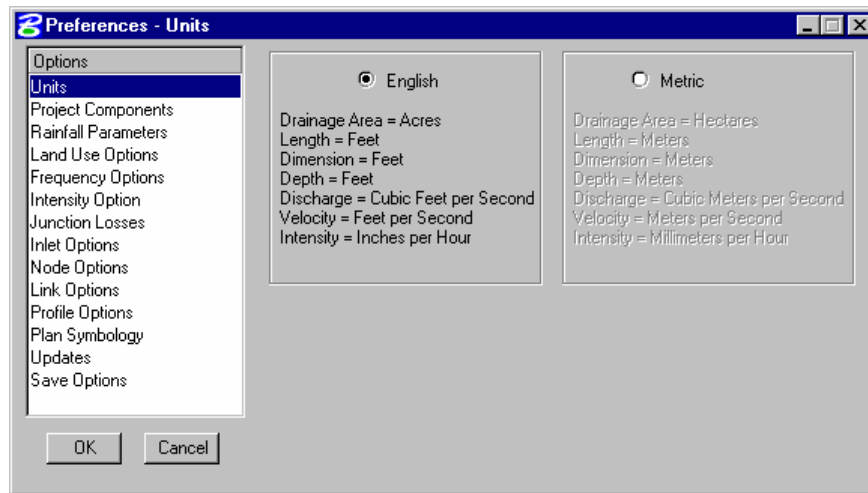
Within each group box is a listing of the units utilized for GEOPAK Drainage. These are display only and cannot be changed.
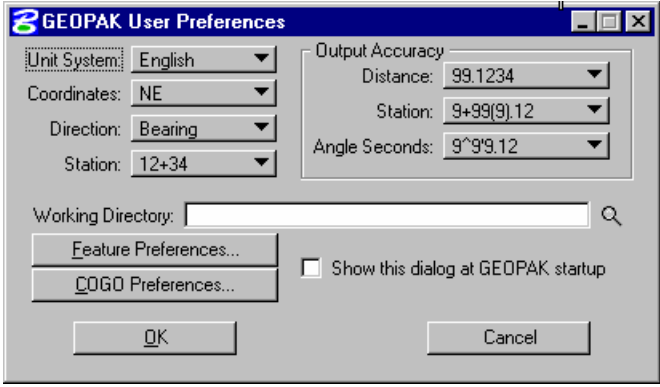
Projects with mixed units are not supported within GEOPAK. There are no other unit specifications within the project. The data input is expected to be in the proper units (English and Metric). As fields are entered in the dialogs the units required will be displayed in the MicroStation Command Prompt.

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 define the location of all files utilized by the project.

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. Clicking Files opens the MicroStation File Manager, wherein the path and database file may be selected. The drainage library is ***vdot.dlb***. The file is provided by VDOT.
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. 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, it may be selected by clicking Select. For VDOT project the job number is always 101, but for that particular reason, special attention must be given to the directory path where the gpk file of the project is located.
Road Preferences	 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. The designer should adjust the Output Accuracy group settings as desired, since they will not affect the precision and accuracy of the drainage calculation. They are just output values.
Drainage Cell Library	Cell library containing cells utilized for Nodes contained in the Drainage Library. The cell library is ***drainage.cel***. The file is provided by VDOT.
Criteria Directory	Directory wherein the criteria to draw nodes, links and other drainage data onto cross sections is stored. VDOT does not uses this functionality.
GEOPAK DDB	Defines the Design and Computation Manager which is utilized for matching pay items to the Drainage Library in order to draw nodes and links at the desired symbology and final quantities computation upon completion of the drainage

	design. The file is ***VDOTenglish.ddb.***
Water and Sewer Project	To leverage data from a Water and Sewer project, activate the toggle, then select the directory and GWS file. This is not a required element for the drainage design. VDOT has not standardized the usage of GEOPAK Water and Sewer.
Shape File	Optional, GEOPAK superelevation shapes created in GEOPAK Road can be leveraged into drainage. Specify the path and file name by manual entry via the browse icon. In order to utilize and extract superelevation shapes data from the superelevation shape file, the file must be referenced into the active design file.
Site Project	To leverage data from a Site project, activate the toggle, then select the directory and GSF file. A site project model or object may be used. This is not a required element for the drainage design. VDOT has not standardized the usage of GEOPAK Site.
Original Ground	The Model , Object or tin file to be used as existing ground. The Original Ground profile along the drainage pipes is drafted when drawing the profile of the drainage system.
Design Surface	The Model , Object or tin file to be used as another reference surface, in this case as a "proposed" surface. The Design Surface profile along the drainage pipes is drafted when drawing the profile of the drainage system.

RAINFALL PARAMETERS

The Rainfall Parameters dialog establishes the rainfall parameters for computing intensities and discharges for the current project.
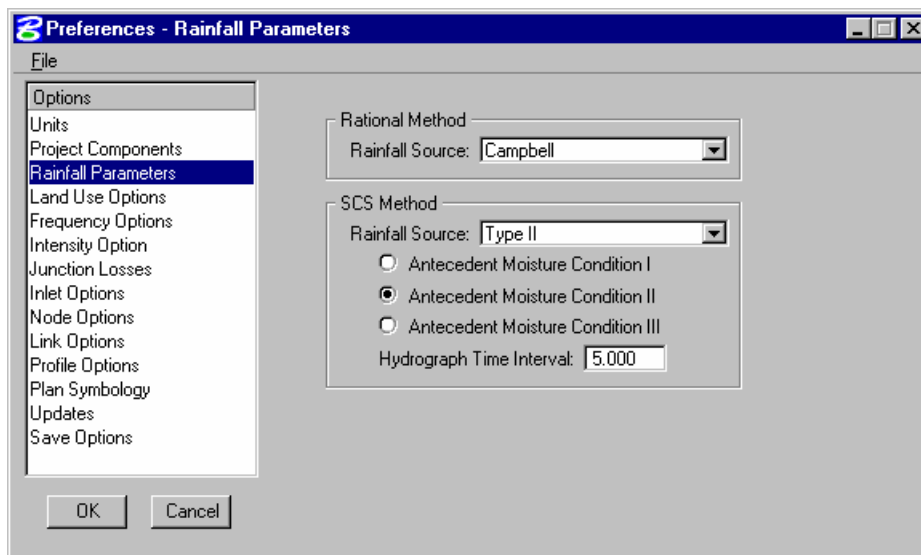
Rational Method Rainfall Source	The Rainfall Data Source for computing intensities and discharges, which is stored within the Drainage Library File, therefore it may only be selected here. Select the zone where the project is located. Refer to the FDOT Hydrology Handbook for the boundaries of the zones
SCS Method:	The Rainfall Data Source for computing discharges, which is stored within the

Rainfall Source	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 minutes.

LAND USE OPTIONS

The Land Use dialog establishes the land use options used to delineate sub areas and runoff coefficients for the current project.
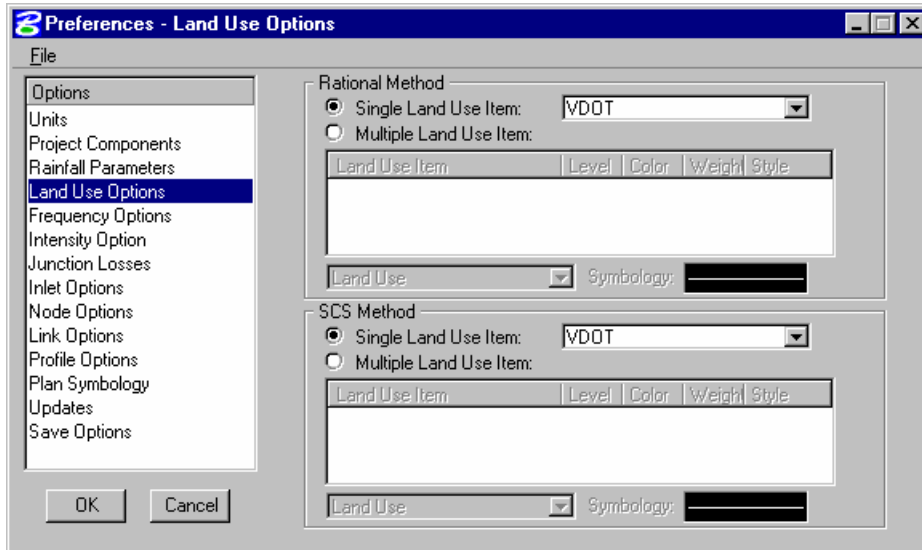
Different Single and Multiple Land Use Items can be defined for the Rational and SCS Method.

Since the Rational Method is the standard methodology used for VDOT Storm Drain projects, the designer should disregard the settings for the 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. The Department has only defined a Single Land Use Item in their Drainage Library. Therefore, this option would not be used by the designers.

FREQUENCY OPTIONS

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

Drainage Library File (DLB)	The current Library File as specified in the Project Component option, is displayed. It cannot be changed from this dialog, only in the Project Components.
Computation Frequency (Rational Method)	The displayed frequencies are selected from the available entries in the Rainfall Parameters chosen from the library. Select the appropriate frequency for the project. Refer to the VDOT Drainage Manual for further details. These fields cannot be modified except through the Drainage Library dialog.
Cumulative Frequency (SCS Method)	The displayed depths are selected from the available entries in the Rainfall Parameters chosen from the library. The SCS approach is not standard for VDOT storm drains, but it can be used for preliminary routing calculations. 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 OPTIONS

The Intensity Options dialog establishes the intensity computation options for the current project.

Drainage Library File (DLB)	The current library file, as specified in the Project Component option, is displayed. It cannot be changed from this dialog, only in the Project Components.
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. The VDOT Drainage Manual specifies a minimum of 5 minutes.
Accumulate Pipe Flow Time by: Uniform Flow Velocity **Full Flow Velocity** **Iterative Velocity**	This option controls how travel times through the pipes are determined. The following provides an overview. **VDOT uses the Uniform Flow Method.** When selected, the velocity associated with normal depth (uniform flow) establishes the travel time. If Full Flow Velocity is selected, the velocity associated with full flow velocity (V= Q/A) establishes the travel time. If the Iterative Velocity method is selected, GEOPAK makes three passes up and down the network performing runoff and hydraulic calculations. By making several passes, the velocity used to compute the travel time closely matches the actual flow conditions. For pipes flowing full, the full flow velocity (V= Q/A) establishes the travel time. For pipes flowing partially full, the average velocity through the pipe establishes the travel time. GEOPAK performs backwater curve (water surface profile) calculations so the average velocity is often different than normal depth (uniform flow) velocity.
Intensity Options Group	Intensity options which allow for specification of an absolute (constant) intensity or computed intensities to use throughout the network discharge computation procedures. Select **Compute Intensity from Library Rainfall Data Source** for VDOT project since the rainfall intensities are already predefined in the library.

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. Ignore this toggle for VDOT projects.
Inlet Computation Only: 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.
	VDOT performs most inlet and spread calculations on 4 in/hr.

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.
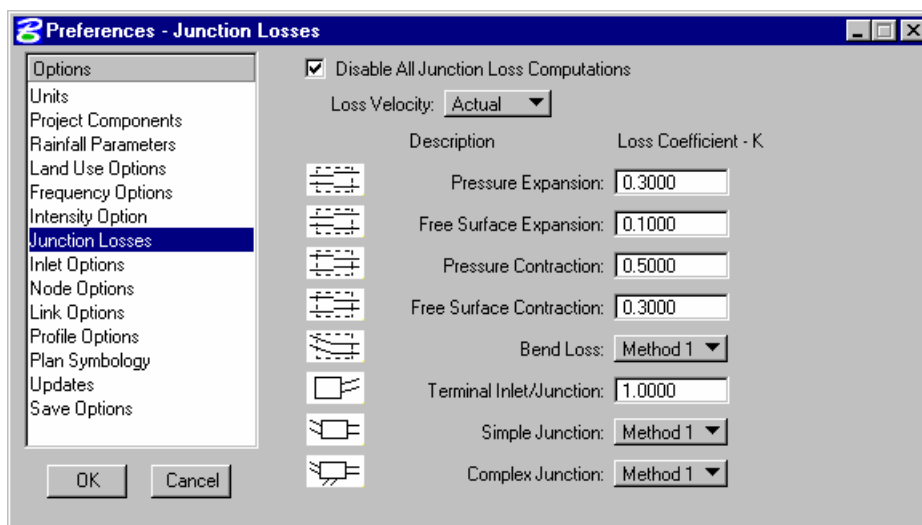
Unique Loss Coefficients (K) can be specified in the fields to the right of each Description, and apply to the whole project. On the other hand, designers can specify a particular Loss Coefficients for each junction in the "Node Configuration - Junction Loss" Dialog box discussed in the Nodes Chapter. However, if the Disable All Junction Loss Computations is activated, no losses will be computed for the entire project.

Disable All Junction Loss Computations	If the toggle is active, no Junction Loss computations are calculated, regardless of the option selected in the Nodes - Junction dialog. Note if a single or small number of Junction Losses are not desired, an option is provided in the Nodes - Junction dialog 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).

INLET OPTIONS

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

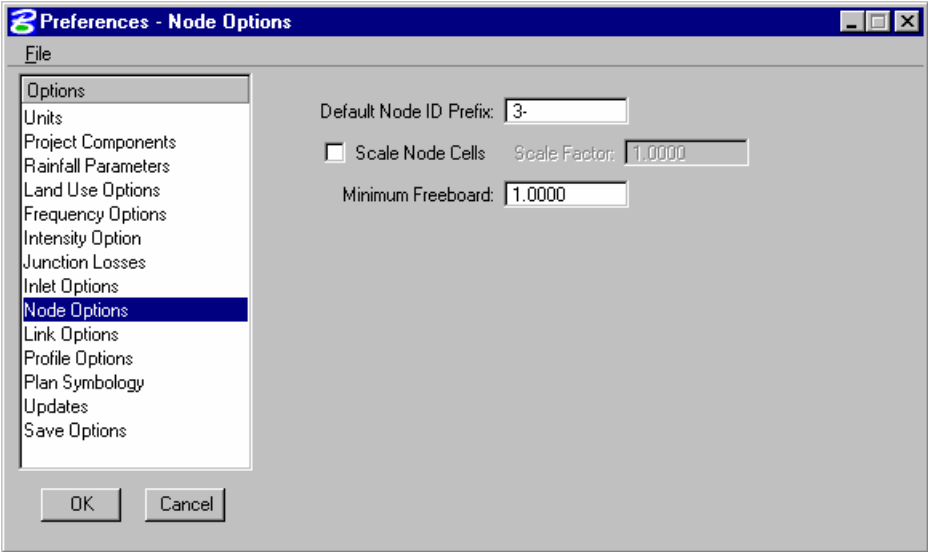|	
Inlet By Pass Options	This option indicates the method by which Inlet by pass flows are accounted for in the system. By Pass as Total Discharge will account for by pass flows as the difference between the total discharge to the inlet and the inlet capacity. This difference, expressed in flow units, will be the total by pass from the inlet and can be added at a subsequent downstream inlet. VDOT spread calculations have typically been performed using this approach. 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. VDOT uses this first approach. 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.
Extend Superelevation Shapes to Inlet at Shape Slope	This option extends the slope of the last superelevation shape to the drainage location point of the drainage node. E.g., sometimes roadway superelevation shapes are not created to the extend of the whole pavement width.

NODE OPTIONS

The Node Options establish the default naming prefix for Node entries and option for automatic link updates.

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 3– 1" followed by "3 – 2", "3– 3", etc. for subsequent Nodes.
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 are scaled in plan view. VDOT cells should be placed at 1.0, so inactivate the Scale Node Cells or set them to 1.0
Minimum Freeboard	This value does not affect the hydraulic computation in any way. It is used solely to trigger a warning message (HGL Blowout) if the hydraulic gradient causes less free board. GEOPAK measures freeboard from the hydraulic gradient to the inlet elevation.

LINK OPTIONS

The Link Options dialog establishes the default Link design constraints.

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 "pipe – 1" followed by "pipe – 2", "pipe– 3", etc. for subsequent Links.
Link Profile Options group box	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. 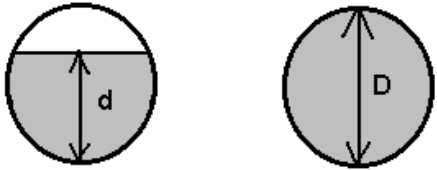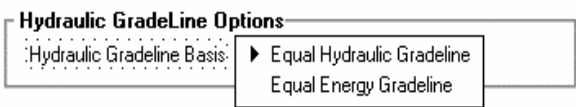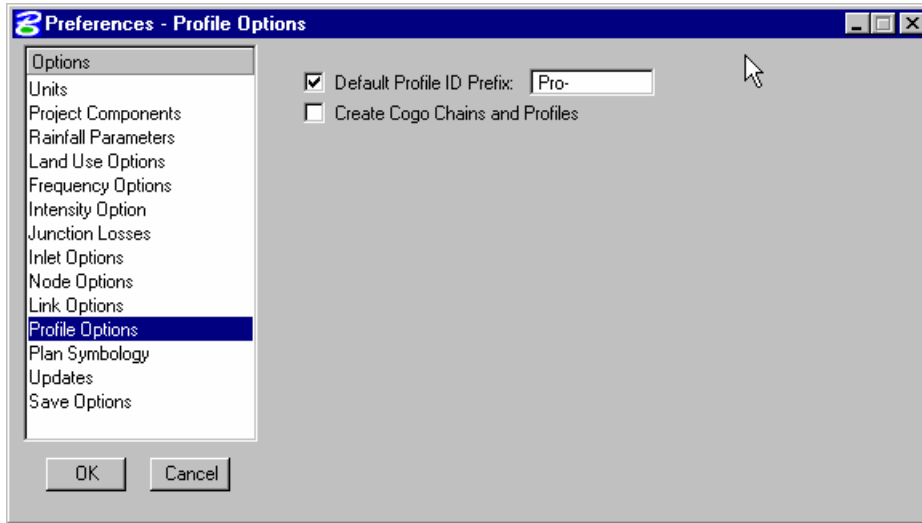These constraints are established later in the Link and Node dialogs. Refer to the Link and Node Chapters for additional information. The Minimize Pipe Size option will use the entire envelope to size pipes. If a smaller pipe will fit within the envelope at a steeper slope then it will be selected as 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	Drainage bases the design on minimizing pipe size, even though this may encourage deeper excavation.
Minimize Depth of Cover	Drainage designs the system with minimal excavation, but utilizes larger pipe sizes.
Elevation Option	This refers to the elevation reported for the ends of the links. Two options are supported: at Hydraulic Center and at Actual Link End. Hydraulic Center reports the elevation at the center of the node structure. Actual Link End reports the

	elevation at the inside of the structure wall. Use Actual Link End for VDOT projects. The option selected 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. It is not critical what is selected here, because the pipes will likely be adjusted to meet standards after GEOPAK initially designs the system. .
Design for Maximum Capacity	This option sizes pipe using maximum capacity. This typically occurs at some percentage (90%) 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. VDOT does not draw drainage structure cross sections as a standard. Therefore, this entry should be blank.
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. Use this for all VDOT projects.
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.

PROFILE OPTIONS

The Profile Options establishes the default naming convention for profiles along the links designed.

Default Profile ID Prefix	This prefix is added to the beginning of the each defined profile automatically and then numerically sequenced by one. For example, the first Profile stored would obtain default ID in this case of "PRO–1" followed by "PRO–2", "PRO–3", etc. for subsequent Profiles. If the links are to be used for COGO chains or profiles, the length of this prefix should be within the parameters for chain and profile character limitations.
Create COGO Chains and Profiles	When toggled on, the links are stored in the coordinate geometry database (GPK) specified in the Preferences - Project Components. The link name is used as both the chain and profile name. Stationing starts at 0+00. This is not applicable to VDOT projects.

PLAN SYMBOLOGY

The Plan Symbology dialog establishes the element symbology for drawing the drainage features during the design process.
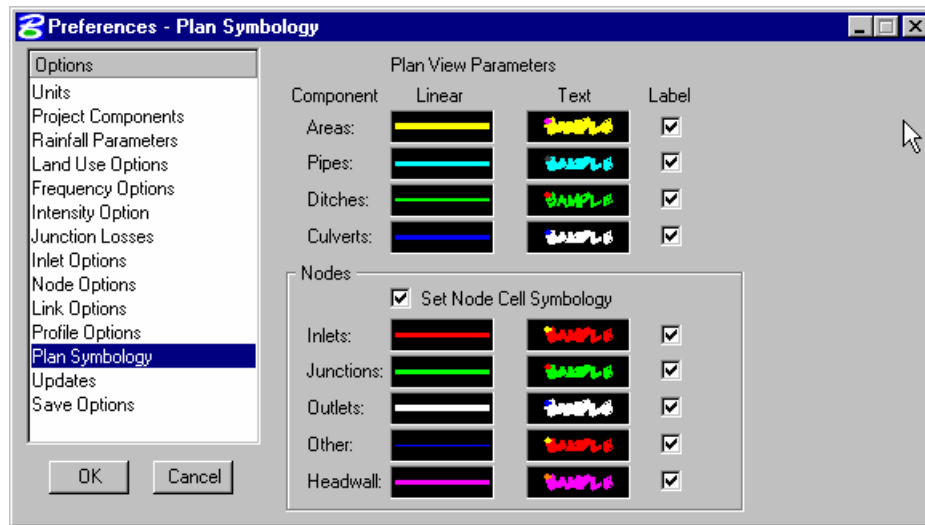
These are **"design process"** symbology, therefore it does not need to match VDOT CADD standards. When the design is completed, GEOPAK Drainage will update automatically all the CADD elements to be complied with VDOT standards using the Design and Computation Manager.

Linear	Any valid MicroStation symbology may be specified for drawing the component. To set the symbology, double click on the graphic, which opens the Set Feature dialog.
Text	Any valid MicroStation text parameters may be specified. To set the symbology, double click on the graphic, which opens the Set Feature dialog.
Label	GEOPAK Drainage will automatically label drainage areas, nodes and link to aid during the design process when the toggle is activated

UPDATES

The Updates dialog enables you to determine whether various data is updated during the design process. You may toggle on as many options as desired. When toggled on, the automatic updating is activated. If the option is not toggled on, no automatic updating occurs.
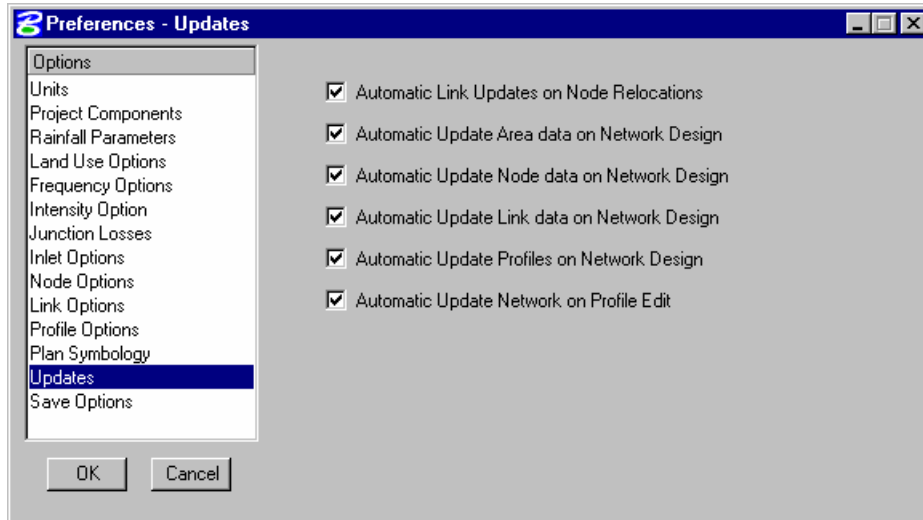
SAVE OPTIONS
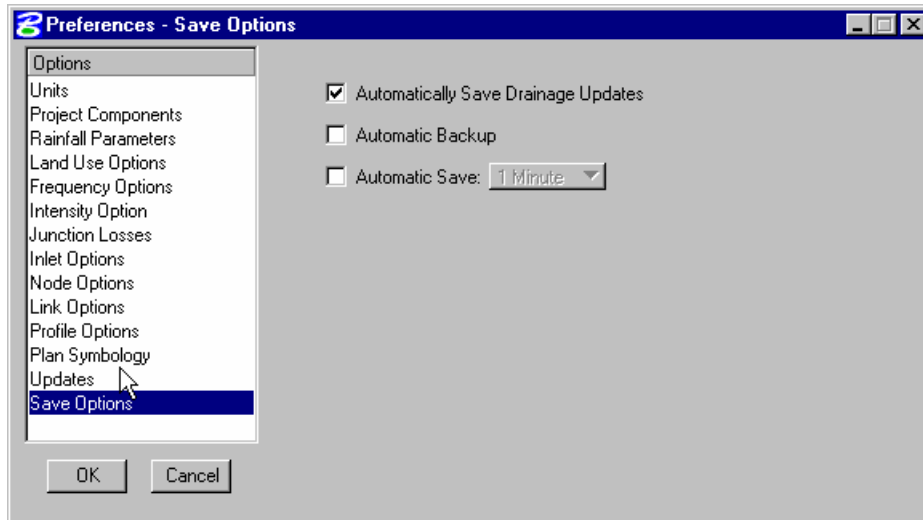
Automatically Save Drainage Updates	When activated, the Drainage project file (.gdf) is saved every time an Apply command is issued from any Drainage dialog.
Automatic Backup	When activated, a backup file is always created (in the project directory with a *.bak extension) when the project is opened.
Automatic Save	To automatically save, activate the toggle to the left of the option, then select the desired time interval.

PRINTING THE PROJECT PREFERENCES

Users may print all of the preferences by going to *Project > Export >Preferences to ASCII*.

LAB EXERCISE: 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.

> ### ➤ *Setup the Project Preferences*

1. Execute *c:\data\geo\vdot\drain1\LAB04_V8ad.EXE.*

2. Open the MicroStation file, *c:\data\geo\vdot\drain1\h17682.dgn.*

3. Open Project Manager (*MS Menu: Applications > GEOPAK Road > Project Manager)*

4. Select project:*17682.prj* located in *c:\data\geo\vdot\drain1*, and click OK.
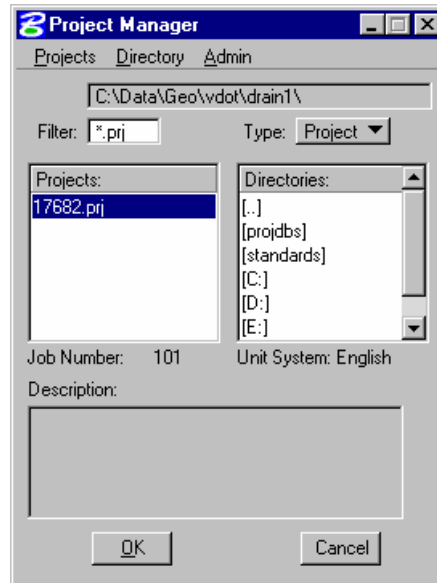
5. Access User: VDOT.
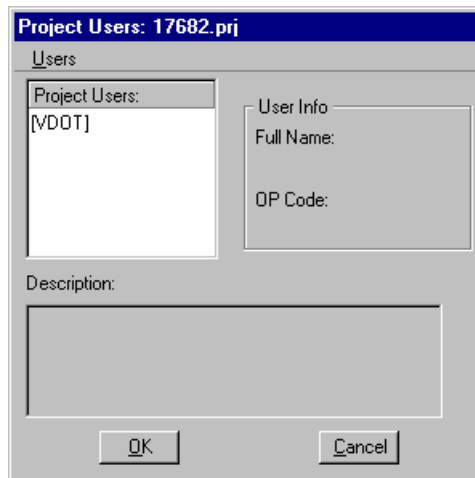
6. Click **OK**. The Road Project dialog will load. This will setup the proper directories for the location of the gpk file.
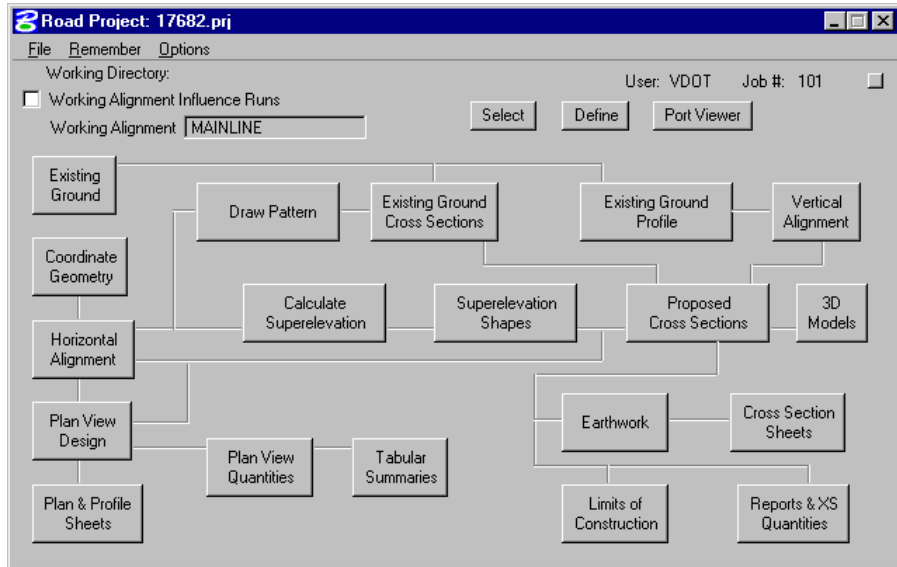
7. Minimize the Road Project dialog box.

8. Access GEOPAK Drainage *(MS Menu: Applications > GEOPAK Drainage > Drainage).*

9. Select the drainage project, *c:\data\geo\vdot\drain1\h17682.gdf. (Drainage Menu: Project > Open).*

10. Select the **Preferences** tool *(Drainage: Project > Preferences).*

    Establish the preferences by selecting each option in the columns and defining the various values.

11. Highlight **Units**.
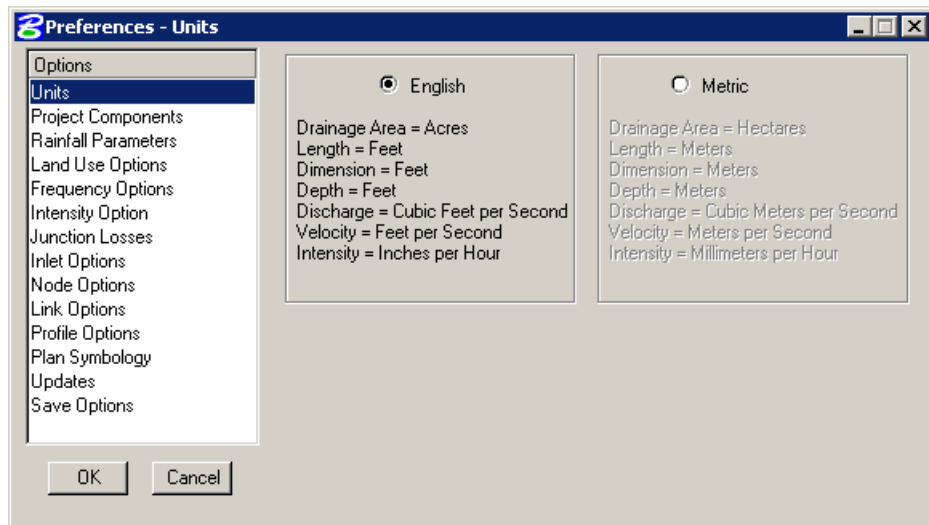
12. Click **Project Components**. Populate the dialog as shown below.
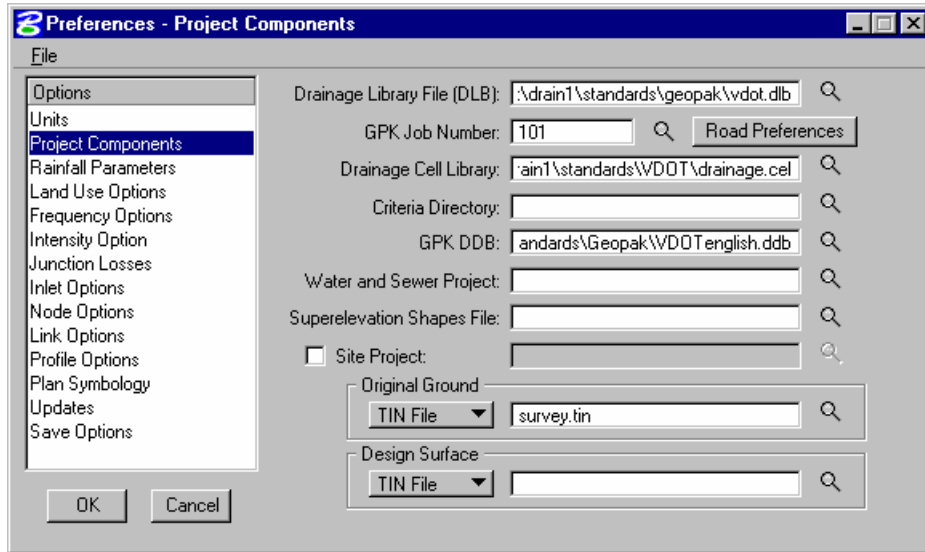
GPK Job Number	*Job101.gpk.*
Original ground	*survey.tin*

Note   Click the Road Preferences button to setup the proper location of the gpk file.

13. Click **Rainfall Parameters**.

We will use the Rational Method for the Storm Sewer design.
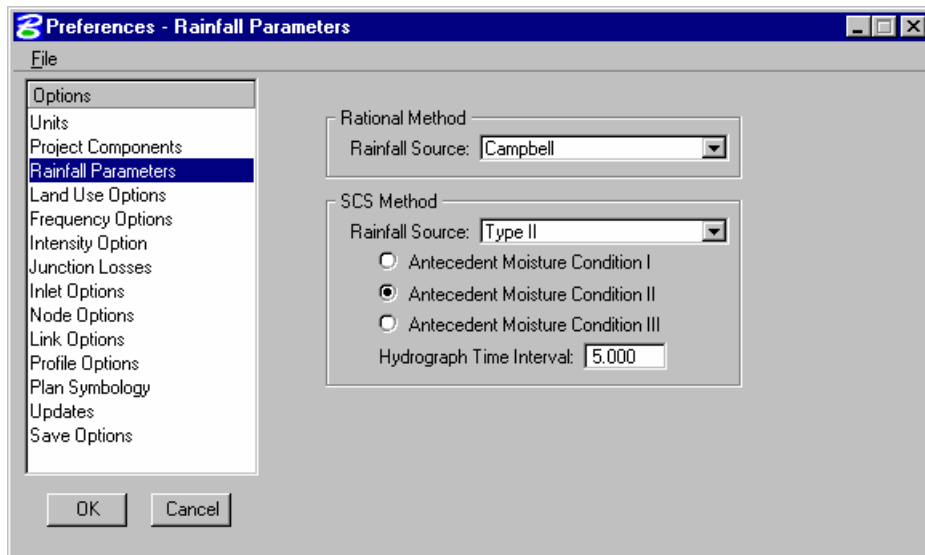
Rational Method	Campbell
SCS Method	Type II

14. Highlight **Land Use Options**.

15. Set the **Rational Method Single Land Use** to VDOT .
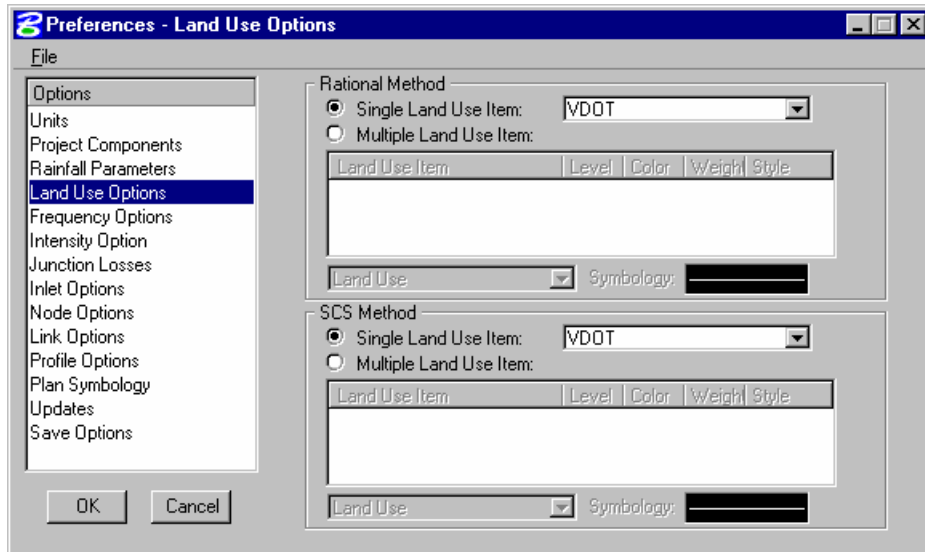
16. Highlight **Frequency Options**.

17. Set the Rational Frequency and the SCS Frequency options as shown below.
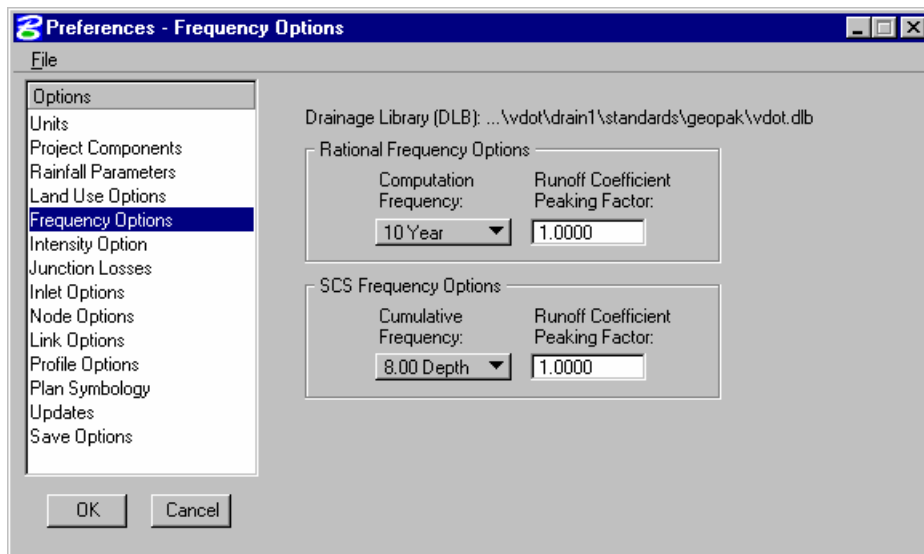
18. Highlight **Intensity Options**.
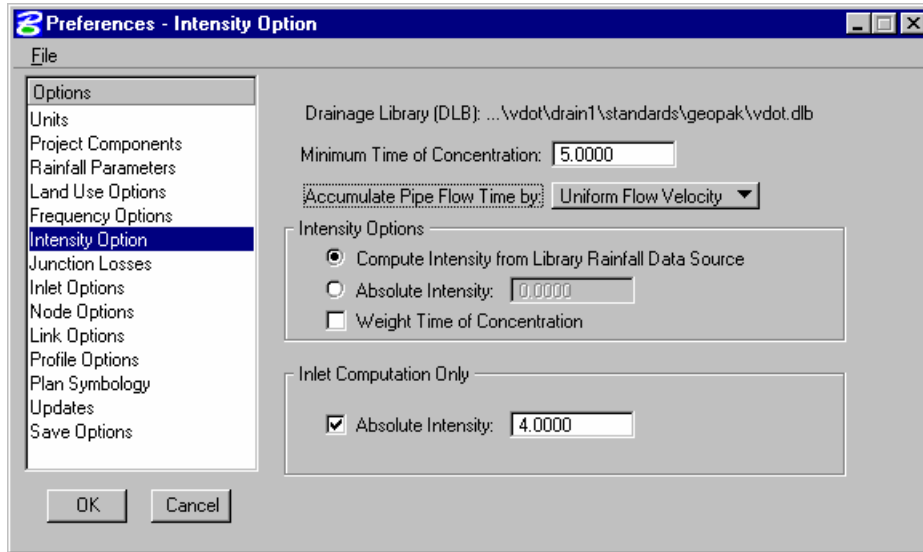
Minimum Time of Concentration	5 minutes
Accumulate Pipe Flow Time by	Uniform Velocity
Intensity Options	Use Compute Intensity from Library Rainfall Data Source
Inlet Computation Only	Toggle *ON* , 4.0

19. Highlight **Junction Losses**.

20. Toggle *ON* Disable All Junction Losses Computations.
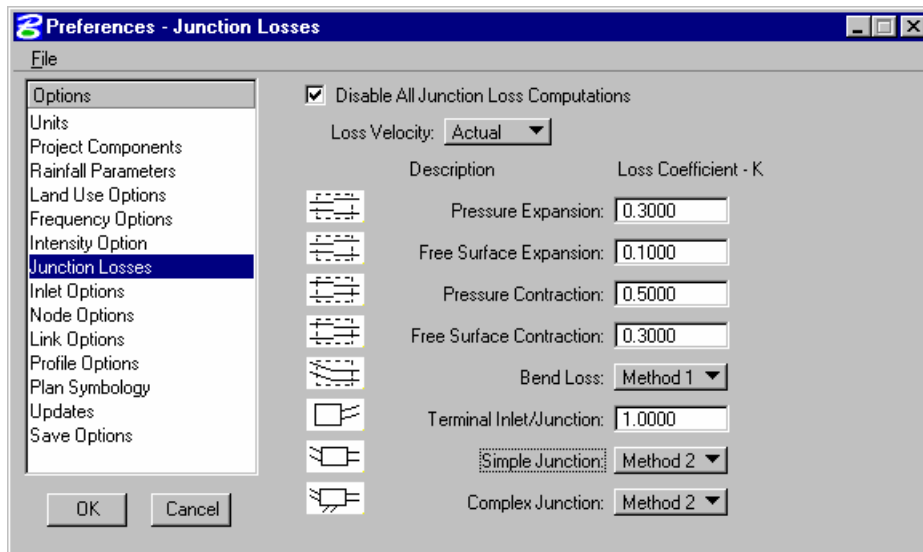
21. Highlight **Inlet Options**.
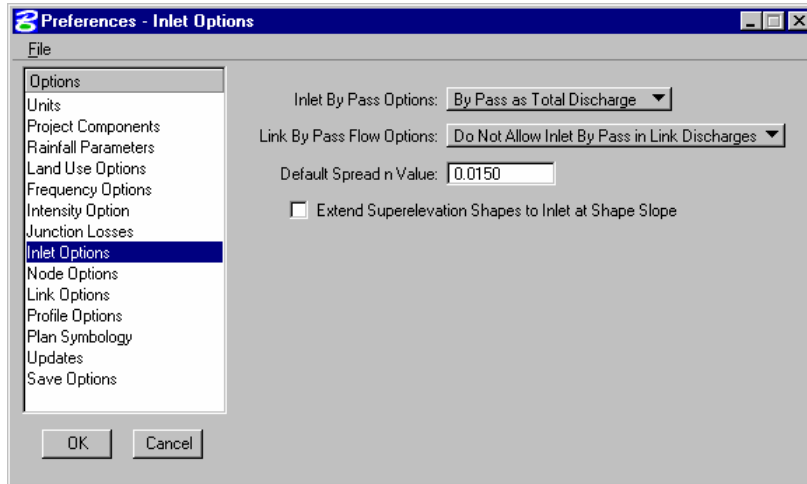
Inlet By Pass Options	By Pass as Total Discharge
Link By Pass Flow Options	Do Not Allow Inlet By Pass in Link Discharge
Default Spread n Value	0.015

22. Highlight **Node Options**.
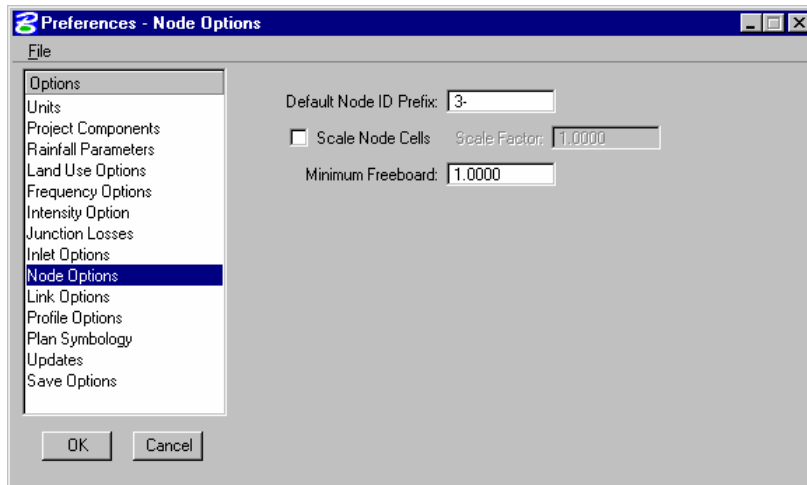
Default Node ID Prefix	3-
Scale Node Cells	Toggle *OFF*
Minimum Freeboard	1.00

LAB EXERCISE: PROJECT PREFERENCES

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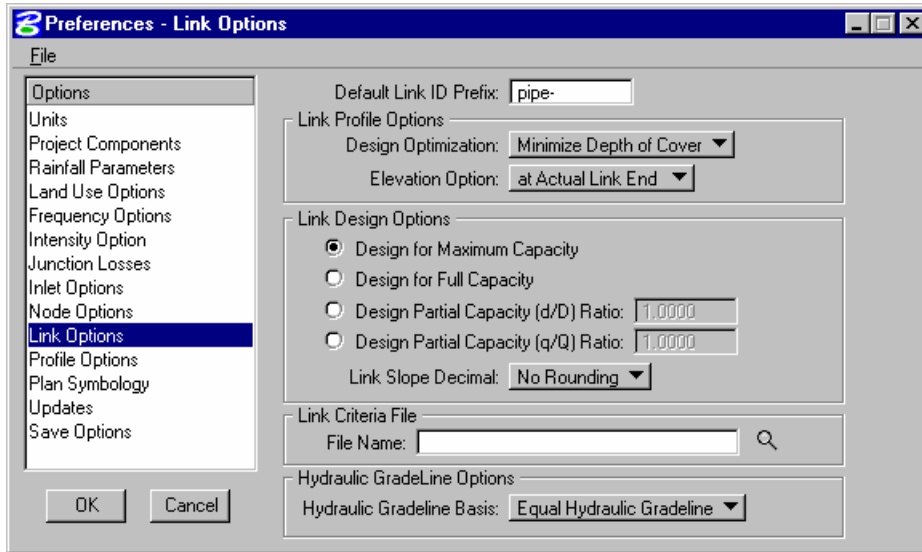
Default Link ID Prefix	pipe-
Design Optimization	Minimize Depth of Cover
Elevation Option	at Actual Link End
Link Design Options	Design for Maximum Capacity
Link Slope Decimal	2
Hydraulic Gradeline Options	Equal Hydraulic Gradeline

24. Highlight **Profile Options**.
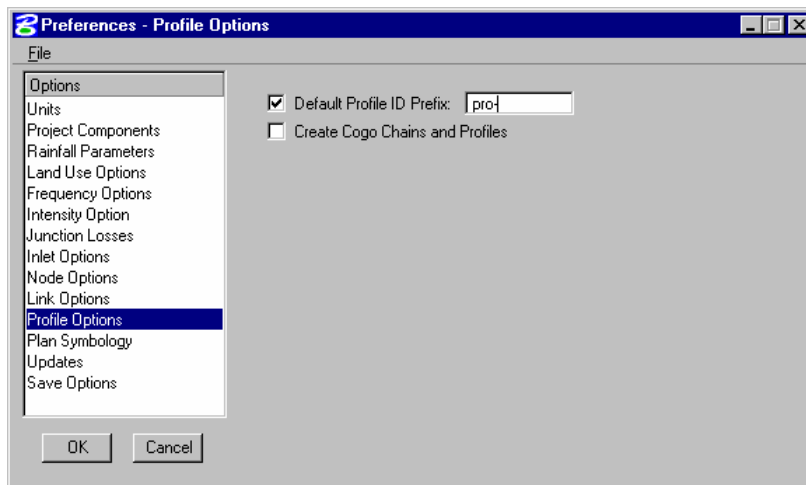
Default Profile ID Prefix	Pro-
Create COGO chains and profile	Toggle *OFF*

25. Highlight **Plan Symbology**.

26. Establish the symbology of the different project elements according to your standard conventions. The Preferences may be changed at any time during the course of the project.

.
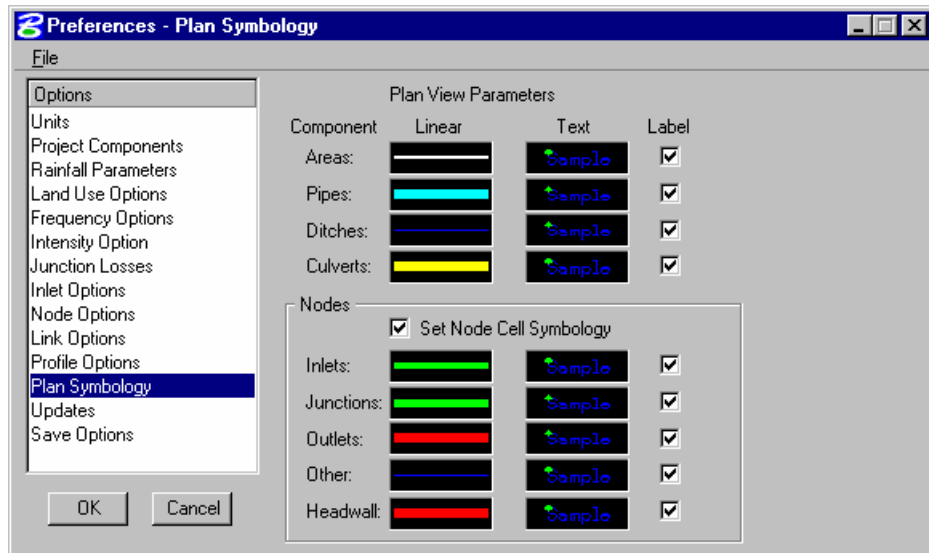
Note   These are "design process" symbology, therefore it does not need to match VDOT CADD standards. When the design is completed, GEOPAK Drainage will update automatically all the CADD elements to be complied with VDOT standards using the Design and Computation Manager.

27. Highlight **Updates**.

28. Toggle *ON* all the options, so all the individual project elements will be updated every time we recalculate our project.
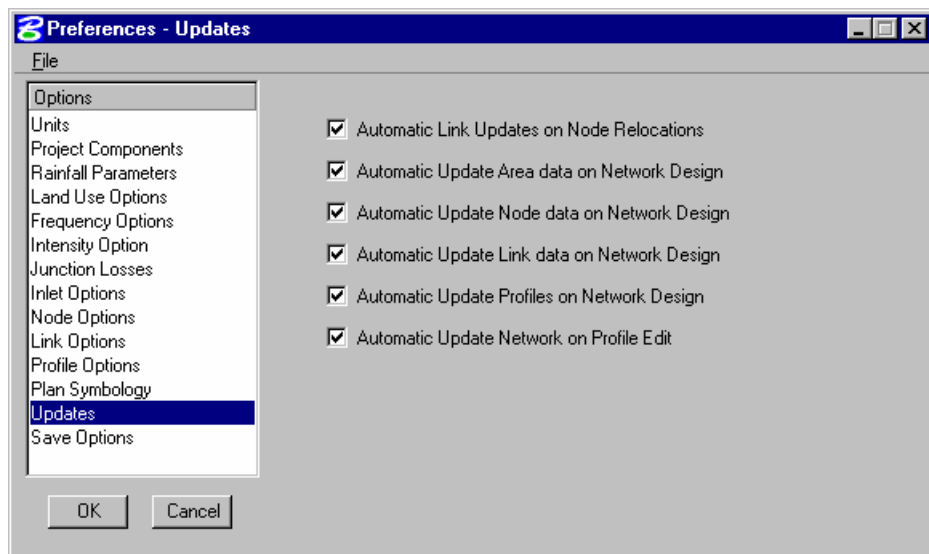
29. Highlight **Save Options**.

30. Toggle *ON* **Automatically Save Drainage Updates**.
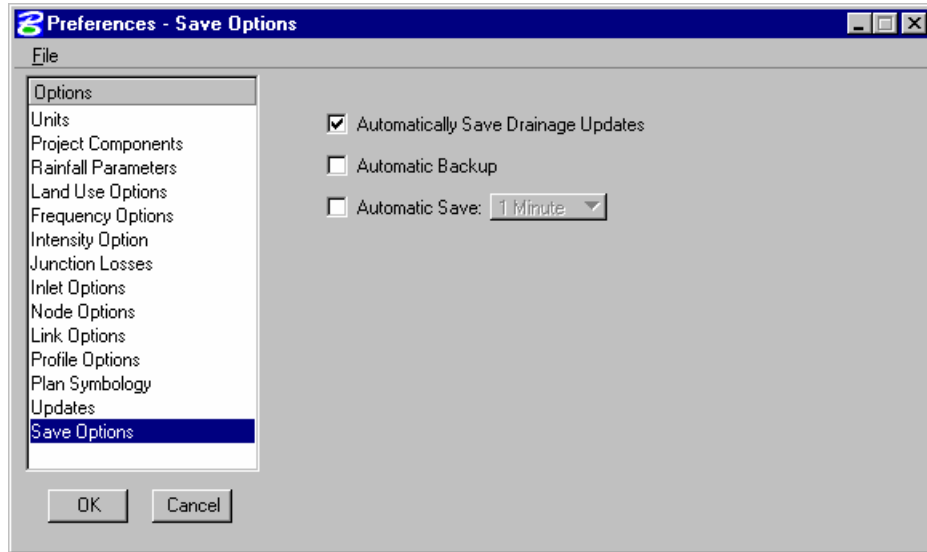
31. Click **OK** to accept the Project Preferences.

32. Create the drainage preferences report for the project. *(Drainage: Project > Export>Preferences to ASCII.*

33. Save the file as : *drainage_preferences_17682.drp*.

This page left intentionally blank.

5 VDOT Drainage Library

OBJECTIVES

In this chapter, we will learn more about the drainage library.

INTRODUCTION

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 VDOT CADD Support Section has created a drainage library (*vdot.dlb*) that will apply to most projects.

The provided drainage library could be modified in order to address the designer's specific needs. Caution should be used since these changes may affect existing and new projects.

The Drainage Library is invoked by selecting Project > Drainage Library from the Drainage pull down menu or the Drainage Library tool located on the Main Drainage tools tool frame.
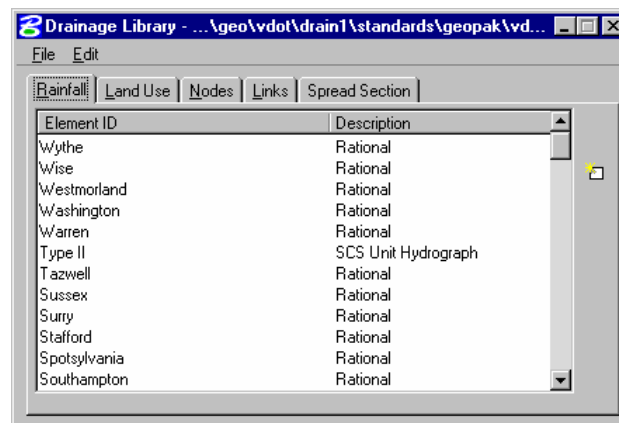
Five tabs are supported for the major categories within the Drainage Library:

- 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. Each curve is associated with a certain zone of the state. Refer to the VDOT Drainage Manual for more information.
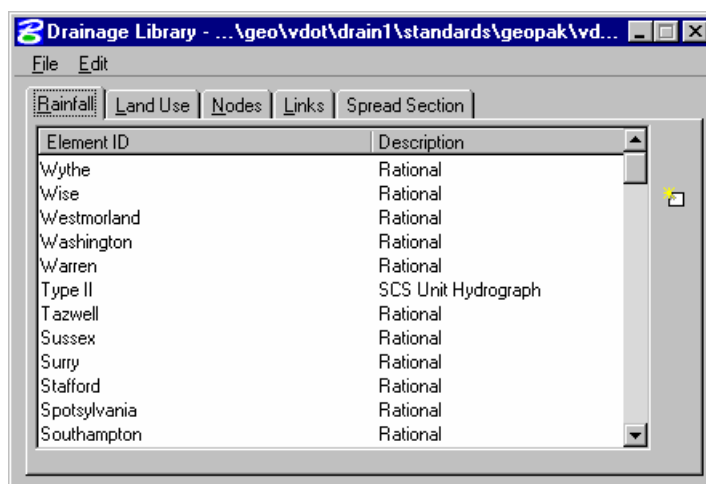
When an Item is modified or added, the dialog displays the Rainfall Data Source information.
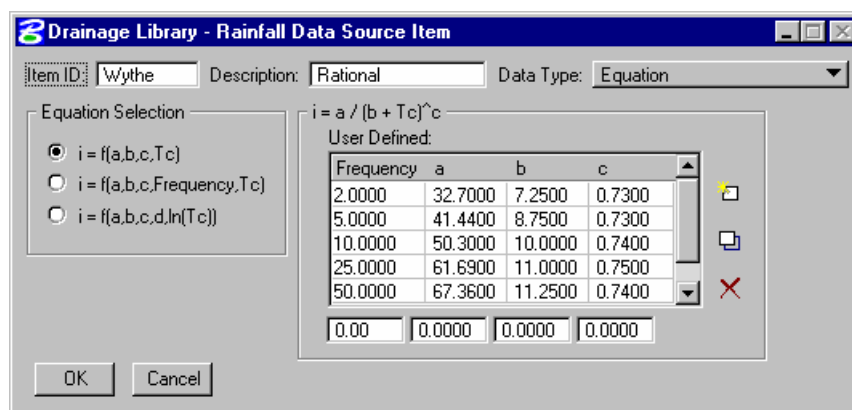
The dialog depicted above is divided into the following sections:

- **User Defined Frequencies (years):** User defined frequencies group box contain the return period frequencies in years for the intensity value columns directly below.

- **Duration (min):** Listed vertically in column one of the list box is the duration of the corresponding intensity values in the specific row.

- **Intensities (inch/hr o mm/hr) :** Comprises columns two through six in the table. They can be manually entered or calculated utilizing the three populating options. These intensity values correspond to the duration specified in the far left column and are in inches or mm depending on the units of the library.

User should not edit the rainfall source for typical rational method calculations on VDOT projects. If needed for special occasions, three other Rainfall Data types are supported; Table, SCS Unit Hydrograph, and SCS Cumulative Hydrograph. Selecting each of these dynamically changes the dialog to reflect the selection.

LAB EXERCISE: UNDERSTANDING THE DRAINAGE LIBRARY

> ### Review the Drainage Library: Rainfall Data

1. Open the Microstation file *c:\data\geo\vdot\drain1\h17682.dgn*

2. Select the Drainage Librarian tool. *(GEOPAK Drainage Menu: Project>Drainage Library).*

   At this point you may be looking at a default drainage library. We will now open our standard drainage library and review the various items stored in our library.

3. Select **File > Open**.

4. Open the Drainage Library *c:\data\geo\VDOT\drain1\standards\Geopak\vdot.dlb.*

5. Click the **Rainfall** Tab.

6. Double click *Campbell*.
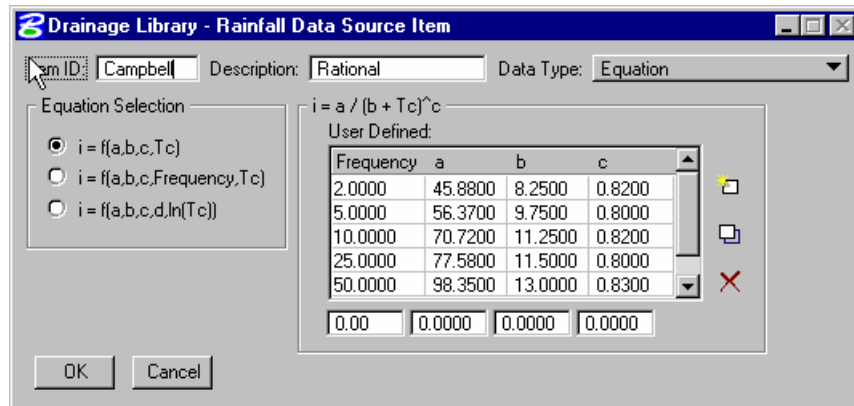
7. Review the data source item.

8. Click **Cancel**. Do not close the Library.

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 Department has included a basic land use item. This item is shown below and has its Item ID Land Use. The designer may use this item or create others. Land use items can have as few as one land use description.

The designer must draw MicroStation shapes matching the symbology below so GEOPAK Drainage can recognize and compute the different types of land use areas inside the drainage basins. These MicroStation shapes could be drawn in the active design file or in a referenced dgn file.
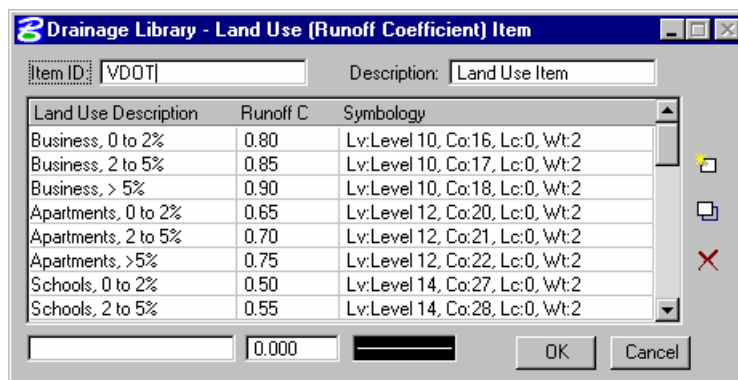
Item ID:	Land Use Item Identification to recall the item from within a GEOPAK Drainage project. (Maximum of 32 alphanumeric characters).
Description:	Description of the Land Use Item. (Maximum of 32 alphanumeric characters).
Land Use Description:	Description of each Land Use. (Maximum 32 alphanumeric characters).
Runoff C:	Runoff coefficient for the Rational Method (C Value).
Symbology	Specified element parameters which uniquely identify the various land use descriptions.
Edit Buttons:	Add - key in the required values in the edit fields directly below the list box, then click **Add**.
OK / Cancel	Action buttons which also close the dialog

   VDOT GEOPAK Drainage

> **Review the Drainage Library: Land Use (Runoff Coefficients)**

1. Continuing in the Drainage Library, click the Land Use tab.
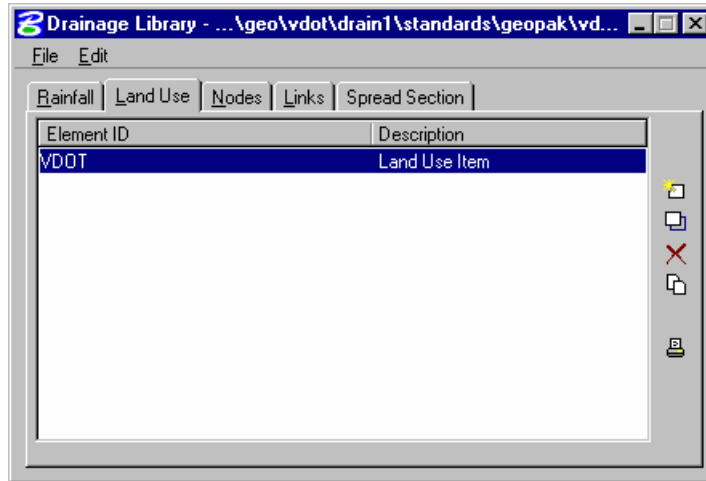
2. Double click the first land use item.

3. Review the different runoff coefficients for the land uses and associated MicroStation symbology.
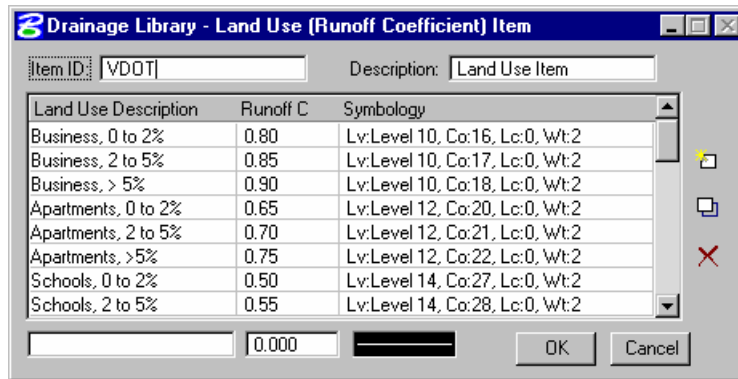
4. Click **Cancel**.

NODE ITEMS

Node Library Items define the type of Node structures used on subsequent GEOPAK Drainage projects. The Nodes contain the Department's standard inlet configurations for Ditch Bottom Inlets (grated), Curb Inlets 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 are included in the library and referenced by each project.

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

GEOPAK Drainage supports the following node types:

- Curb: This is used to depict a curb inlet. This type of inlet could be placed on a On-grade or Sag condition.

- Grate: This is used to describe a grate inlet. This type of inlet could be placed on a On-grade or Sag condition.

- Slotted Drain: This node type is used to enter slotted drain inlets.

- Junction: This node type is used to enter data for any type of drainage structure required or representing a confluence of pipes that is not considered an inlet or outlet. Typical uses are manholes, junction boxes or Wye connections.

- Outlet: It is used to enter data for outlets to a storm drain system. This is typically a pipe headwall or other structure to be placed at the outlet of a system.

- Bottom: It is used for the bottom portion of an inlet or node structure. VDOT does not use this type of node.

- Other: It is used to enter data for any generic Node that is considered to be an inlet, junction, or outlet. Typical uses could include pipe grade breaks, or just system connections where no computations or structures apply to the Node. Type Other is commonly used as the node type between ditch links.

- Headwall: It used to enter data for any node that is to be used as a headwall in conjunction with the culvert tool. It has no hydraulic properties since it is used only for graphical purposes.

All Node items have certain data that describe them. The typical Node item dialog and common data requirements are shown below. For further explanation of the dialog contents, please refer to the standard GEOPAK Drainage On-line Help.
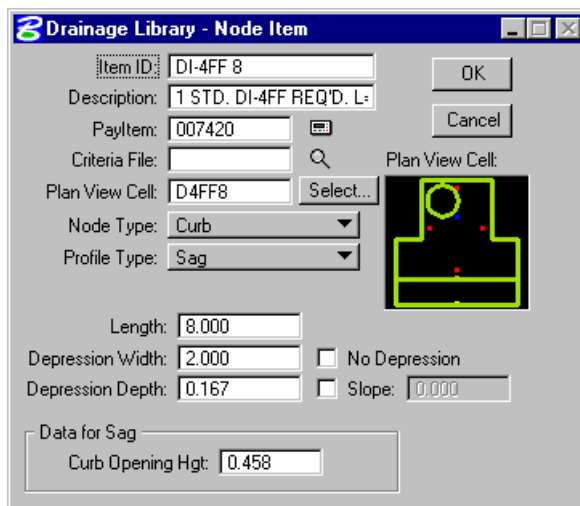
The bottom part of the dialog contains parameters used in the hydraulic computations of the inlet. These parameters are needed for typical FHWA HEC-12 or HEC-22 calculations.

A key element in the node definition is to properly configure the MicroStation cell representing the node. For that reason, GEOPAK Drainage supports the definition of certain key points within a cell that can be used to increase the flexibility of placing Nodes, connecting Links, and reporting on Node information.

- **Pipe Connection Points** – these points can be placed within a cell to indicate the points around the structure to which the pipes shall connect. GEOPAK Drainage will subsequently track the actual pipe length from this connection point to that of the other nodes. If omitted from the cell, the pipes will connect to the cell origin. For circular nodes, a construction class circle may be placed inside the cell to allow the pipes to connect at any location around the circumference. These points are defined by using a MicroStation Construction Class Point, Line Style 6. For circular nodes, a construction class circle with Line Style 6 may be placed in the cell indicating the pipe location around the circumference.

- **Hydraulic Center Point** – a single point may be placed in the cell representing the hydraulic center of the structure. All hydraulic calculations will be based on hydraulic length from one hydraulic center to the next. If omitted, the cell origin will be used as the basis for hydraulic length. These points are defined by using a MicroStation Construction Class Point, Line Style 7.

- **Node Elevation Point** – a single point may be placed in the cell representing the location for elevation definition on the Node. This point location will be used for subsequent extraction of the Node elevations from the DTM. If omitted, the origin of the cell will be used as the location of elevation sources. These points are defined by using a MicroStation Construction Class Point, Line Style 5.

- **Node Location Point** – a single point may be defined in the cell representing the preferred point to locate the Node. A typical example would be if the cell origin is in the middle of the

structure and the structure is usually offset from the curb. If placed at the cell origin, the Node would always need to be placed at the offset from the curb. If the location point is placed within the cell at the offset from the curb, the Node can subsequently be placed directly on the curb. These points are defined by using a MicroStation Construction Class Point, Line Style 4.

- **Node Alignment Point** – used for top and bottom of cell. VDOT does not configure these points.

LINK ITEMS

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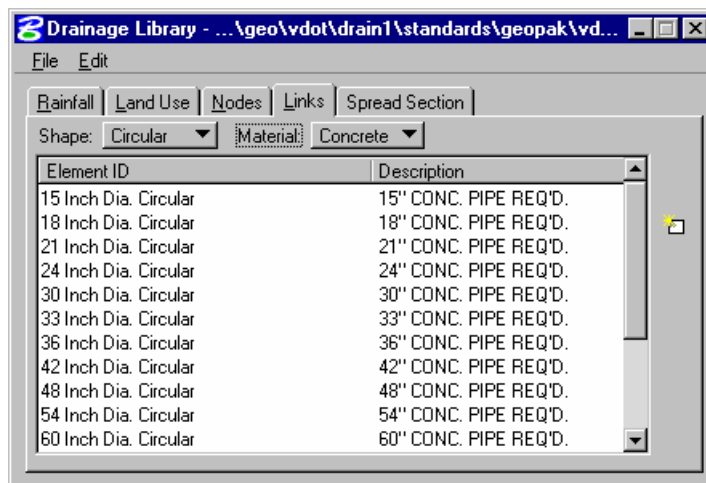
The *Links Tab* is the source of shapes, materials, dimensions, and hydraulic properties of these features. When designing these components, the items serve as the means to specify design candidates. If pipe items are deleted, these items will no longer be available for use on a project using the specific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

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

- Shape
- Material
- Type or corrugations

The VDOT's library contains shapes for box, circular, and elliptical only. Manning's n of 0.012 is applied to all of these and concrete is the only applicable material.

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.
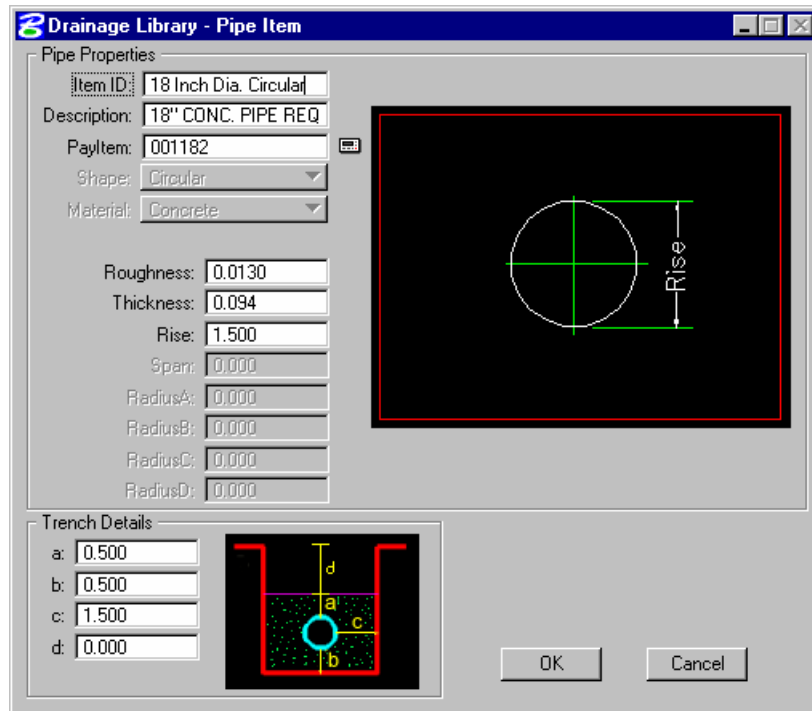
LINKAGE TO AUTOMATED QUANTITIES

One important feature of the drainage library, apart from their hydraulic utilization, is the ability to connect to the quantities database. This offers the designer the ability of computing final plan view quantities at the end of the design process.

The drainage library connects to the Design and Computation Manager database (VDOTenglish.ddb) thru the PayItem definition in the Node and Link item dialog box. The PayItem definition field in the drainage library should correspond to the ItemID element in the Design and Computation Manager. The method and units of computation (each, linear feet, etc) will be defined by the design database or VDOTenglish.ddb.
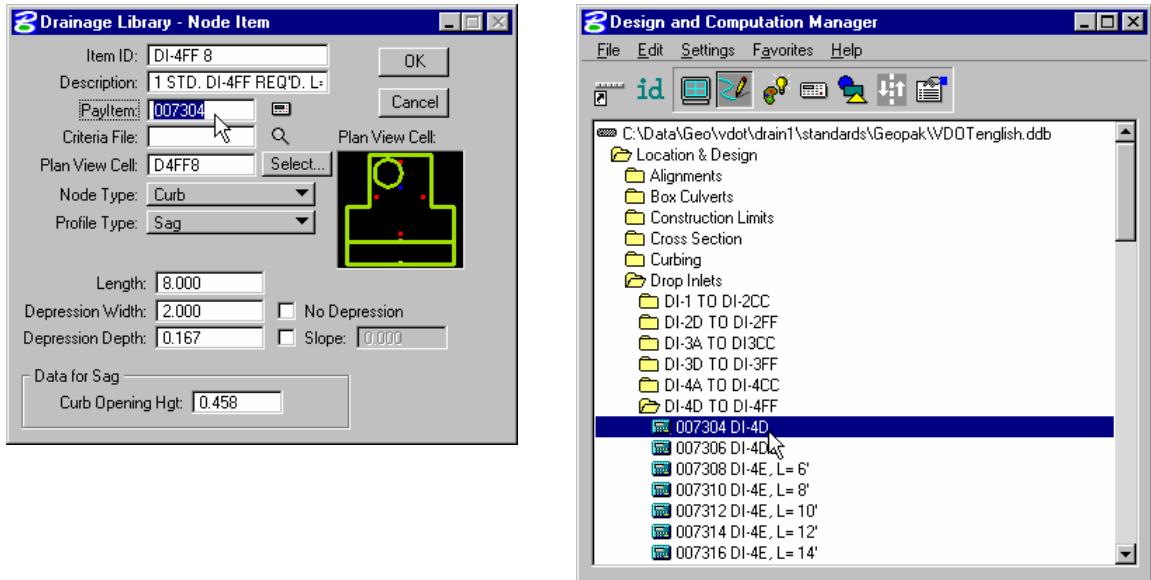
SPREAD SECTION

The Spread Section tab is utilized to store roadway pavement spread information.

© 2007 VDOT   VDOT GEOPAK Drainage
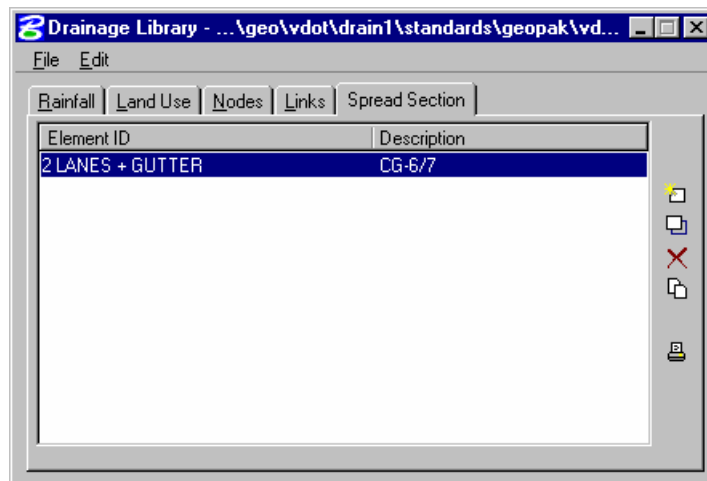
When double clicking on an entry in the list box, the Spread Section dialog opens.
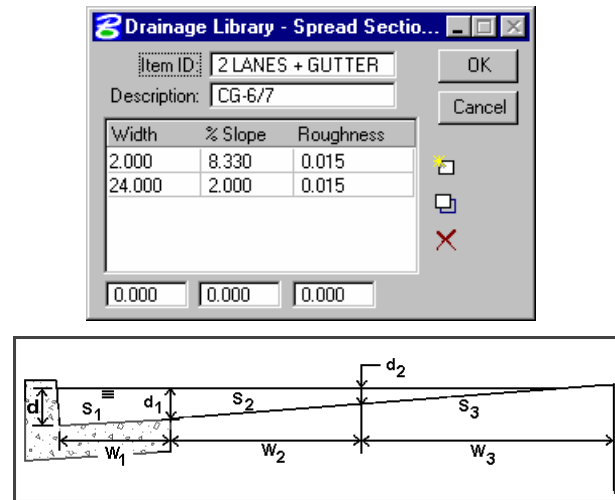
Input is available for the Spread Width, Slope, and Manning's N value. The Department's library contains several common spread sections. Note that some or all may not include the gutter. Users can create additional spread sections as needed.

> *Review the Drainage Library: Spread Section*

   1. Select the **Spread Section tab**, highlight the *2Lanes+Gutter* section, and select **Modify**:
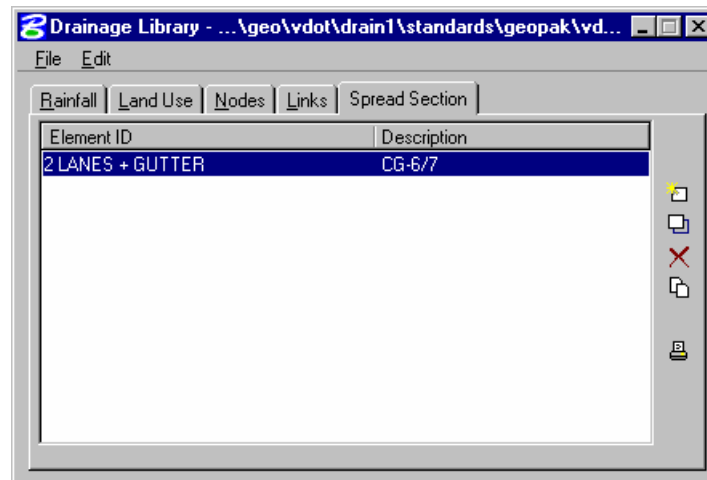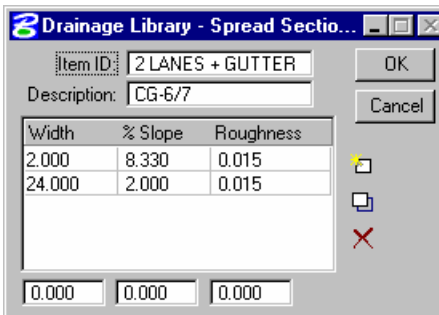
2. Review the spread cross section characteristics for each spread item.

3. Exit MicroStation.

6   Advance Storm Sewer Design

OBJECTIVES

The objective of this chapter is to review drainage design procedures on how to design a storm sewer system.

DRAINAGE AREAS: 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 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 computed using the FDOT Intensity-Duration-Frequency curves that are part of the Department's 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.
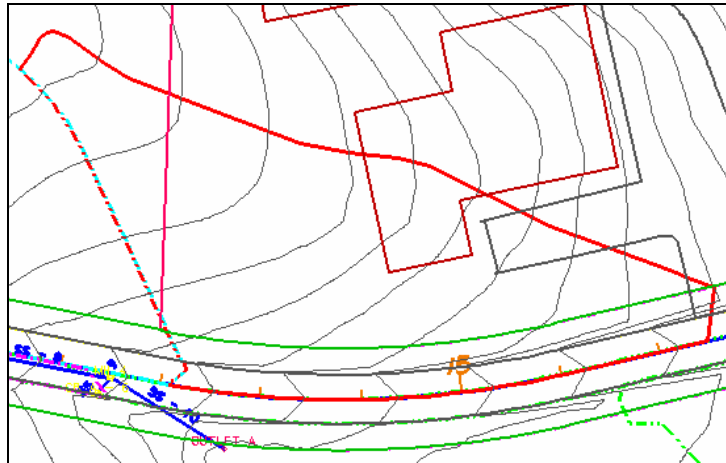
In order to assign Drainage Areas and the resulting peak discharges to Nodes of a storm drain network, the Drainage Area ID must match that of the Node. The best option to match the Node and the Area is thru *Component>Node*, then selecting the Edit Area button.

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

Tools from left to right are:

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

DRAINAGE AREA: DEFINITION

The Drainage Area Definition dialog is used for adding or editing Areas. The general properties of the drainage area are displayed in a list box on the left side of the dialog under the Options group, 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 the highlighted Option. When all the options have been reviewed and appropriate information supplied, pressing the **Apply** button will update the Drainage database
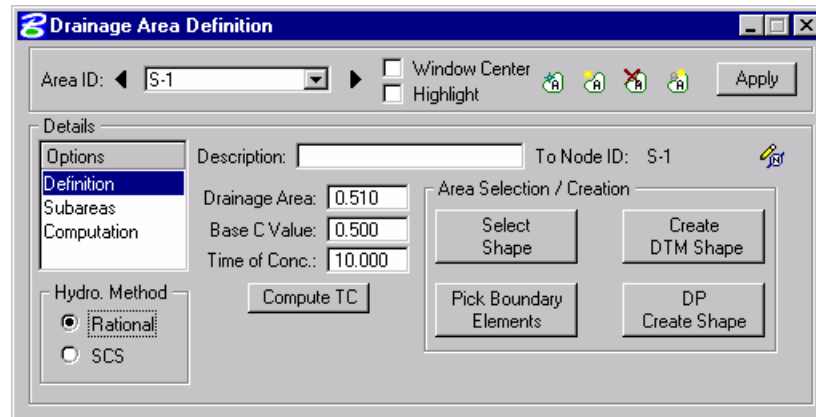
This dialog is divided into six different group boxes.

Description	This may be used to better describe the drainage area.
Hydrologic Method	Rational and SCS are supported. The VDOT Drainage Manual requires that the Rational method be used for storm drain design.
Drainage Area	Total drainage area can be entered here or the designer can select a valid MicroStation shape and the software will automatically compute the area of the shape. The area units are either Acres or Hectares, depending on the project units.
Base C Value	If no sub areas are delineated, the Base C value applies to the whole drainage

	area. If sub areas are delineated within the drainage area, this Base C value will be applied to all "remaining" sub areas that do not have their own C Value. If the Land Use Library items address all the possible land uses on the project, there would be no "remaining" sub areas for the Base C-value to apply to. See discussion of Sub areas below.
Time of Concentration	The overland Time of Concentration for the Drainage Area expressed in minutes. The minimum time of concentration entered under *Preferences>Intensity Options* will override smaller values if entered here.
Compute TC	Opens the Time of Concentration dialog box. This is a time of concentration calculator that separates the flow into Sheet, Shallow and Concentrated flow. Different formulas and methods are applied for each type of flow. The flow path could be keyed in by the designer or could be extracted from DTM data.
Area Selection / Creation:	Three methods of defining the area are supported: Select Shape, Pick Boundary Element or Create a DTM Shape. **Select Shape**: Select a previously drawn MicroStation shape, click **Select Shape**, then identify and accept the existing shape. These MicroStation shapes could be drawn in the active design file or be in a visualized reference dgn at the time of selecting the shape. **Pick Boundary Elements:** Click **Pick Boundary Elements**, then select all required MicroStation elements which comprise the boundary of the area. Note the vertices need not be trimmed but the elements must bound an enclosed region. Once the elements are selected for the bounding area, click **DP Create Shape**, and indicate a Data Point within the enclosed area. Upon successful creation of the shape, a Data Point will accept the shape and reset will undo the shape. If a solution cannot be found and numerous trials through the bounding elements are occurring, hitting any key will cancel the process. **Create DTM Shape:** Create a Drainage Area shape via the Delineate Watershed-Drainage Area Tool. *Drainage>Utilities>Drainage Area* Tools.

DRAINAGE AREA: SUB AREAS

In order to delineate and add sub areas for a particular Area, select the **Sub areas** item within the Drainage Area dialog. Sub areas are defined as smaller components within a drainage area that exhibit different land uses or runoff coefficients (C values).
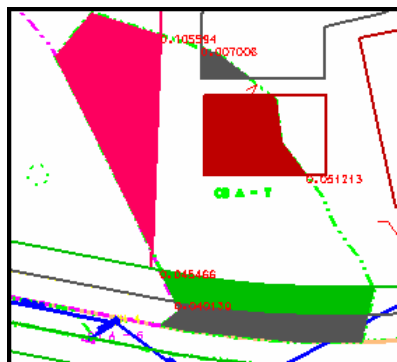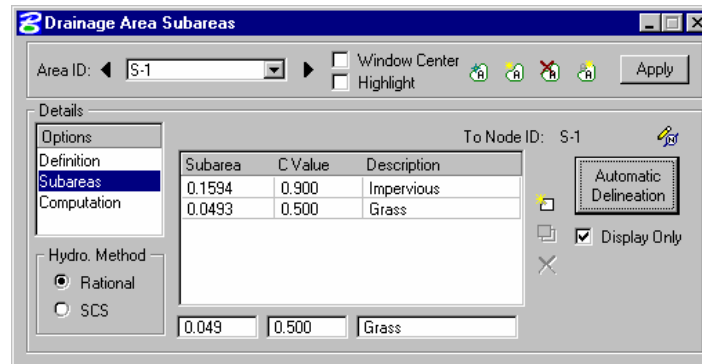
The Sub areas and C-Values may be entered in the table manually or Sub areas may be delineated from the land use boundaries automatically.

The Automatic Delineation button is used to define the sub areas within the current Drainage Area. In order for the software to delineate the land use areas automatically, MicroStation shapes need to be drawn into the active or reference dgn file following the symbology specified in the Department's Drainage Library under the tab Land Uses. The designer can also assign this MicroStation symbology by using the land use definition tool located in *Drainage>Components>Land Uses.* Upon calculation of the sub areas, GEOPAK will write the sub area values in the dgn file unless the Display Only check box is activated.

DRAINAGE AREA: COMPUTATION

The Computation option will calculate the discharge for the selected drainage area, using the computed weighted average or composite C value and the rainfall intensity associated to the entered or calculated time of concentration. The computed discharge shown does not reflect the absolute intensity, if the designer selected an absolute intensity for inlet computations in *Preferences>Intensity Options.*
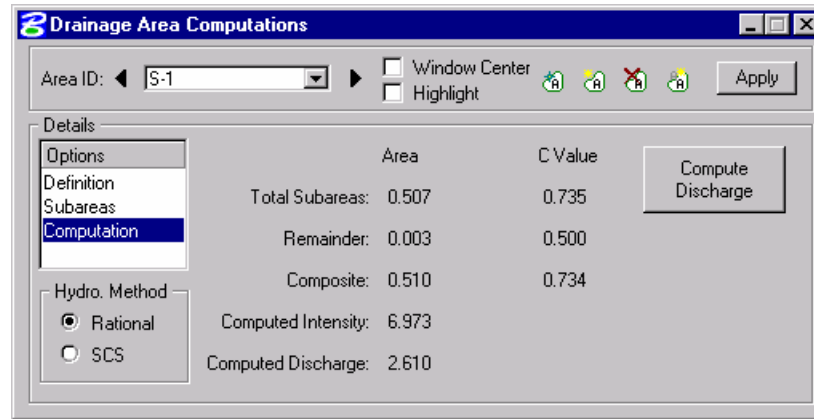
NODES: INTRODUCTION

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. Although GEOPAK will support more, the Department provides library items for six Node types as described below.

Node	Description
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. The Department has no Library items for slotted drain at this time (2007).
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. The Library items include junction boxes and manholes.
Other	The Department intends these to be hydraulic openings that accept flow into the system. These node types can also be 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. VDOT does not use this type of nodes.
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. One option is invoked by the Drainage pull downs *Components>Node* and another option is by selecting *Drainage > Tool Boxes > Node* ,then identifying the desired tool from the tool box as depicted below.

Tools from left to right are:

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

NODE: PROPERTIES

To define a node as a part of the drainage network, the Node Configuration dialog needs to be setup with the appropriate node data. The general Options are displayed in a list box on 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. When all the options have been reviewed and appropriate information supplied, pressing the **Apply** button will update the Drainage database.
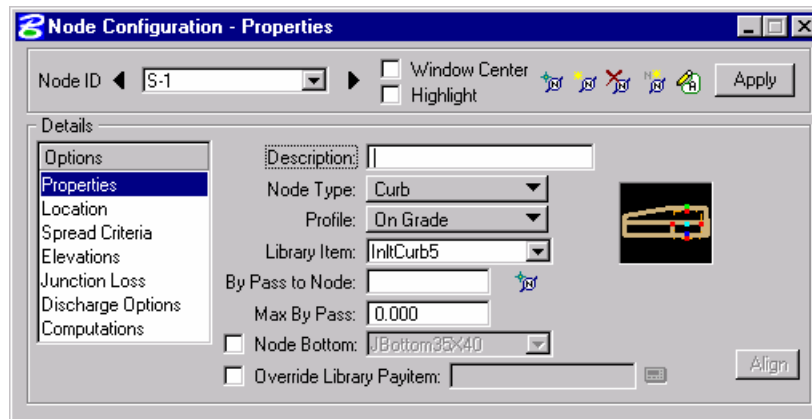
Node ID	GEOPAK will automatically number the nodes based on the order in which the nodes are entered. The nodes can be renumbered later in the design process. This is discussed in the Chapter about Design Revisions.
Description (optional)	This may be used to better describe the node.

|	
Node Type	This will show the different node types defined in the drainage library.
Profile	The designer needs to specify the profile condition of the particular node to be placed. On Grade or in Sag condition. This affects the input and calculations for spread.
Library Item	Depending of the Node Type selected above, the list box will only show the available nodes that match the type condition.
By Pass to Node	The designer can specify or ID the node that will take the by passed discharge from the active node.
Max By Pass	Defines the maximum allowable by pass flow from the inlet. No specific computations are adjusted by this value; it is used to merely query the system for by pass flows which exceed this value. Used for On Grade Inlets only. This value could be left at zero (default) if no By Pass to Node is specified.
Node Bottom	VDOT does not use this type of node.
Override Library Item	Each node in the drainage library is associated to a pay item or Item ID in the Design and Computation Manager database. This option will override this linkage, allowing the user to select their own Item ID. The Department uses that option to make the placed node an "existing" node for quantity computations.

NODE: LOCATION

The designer will place the drainage node into the dgn file using different locating options. If a geometric database (gpk) is used in the Project Components, the Chain and Profile options will be activated, therefore the nodes could be referenced to a station and offset from the selected alignment or chain, and the node elevation could also be computed from the profile selected. If no gpk file is available, the node will be placed using the coordinate information from the MicroStation file.
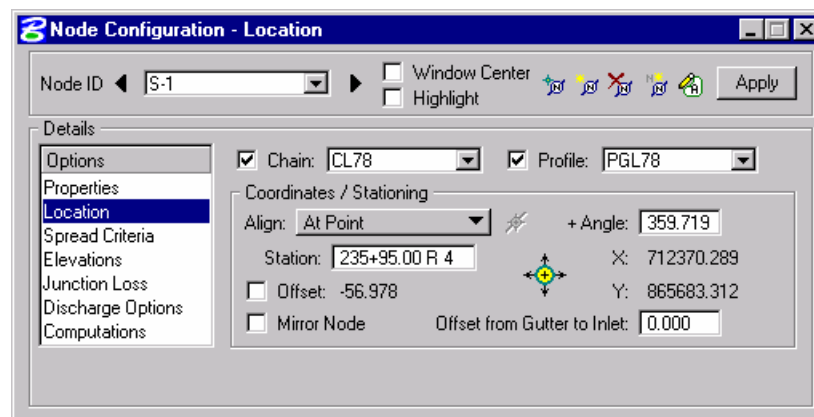

|	
Chain	A previously defined horizontal alignment stored in the "gpk" file. Activate the toggle to the left of the field and select the chain from the list

Profile	A previously defined vertical profile stored in the "gpk" file. Activate the toggle to the left of the field and select the desired profile from the list.
Align: At Point	When this option is utilized, the node is placed at the active angle in the MicroStation file at the current or dynamic Node coordinates.
Align: Tangent to Chain	When this option is utilized, the node is dynamically moved along the Chain, and is rotated maintaining tangency to the chain. The designer can enter the station location or place the node dynamically.
Align: Tangent on Element	When this option is utilized, the **ID** button to the right of the option is activated and the designer is prompted to identify and accept a MicroStation element. The Node is placed tangent to and on the identified element projected from the current or dynamic Node coordinates. The designer can enter the station location or place the node dynamically.
Align: Tangent to Element	When this option is utilized, the **ID** button to the right of the option is activated and the designer is prompted to identify and accept a MicroStation element. The Node is placed tangent to the identified element at the current or dynamic Node coordinates
Angle	An optional angle rotation could be applied to the node. A 180-degree angle is often needed to properly align the throat of curb inlets and is sometimes used in conjunction with the Mirror Node toggle.
Station and Offset	Nodes may be located by entering the station and offset in reference to the selected chain. If "Tangent on Element" is used, the offset is computed.
Mirror Node	When activated, the node cell is mirrored, in order that the same cell can be utilized to accommodate flow coming from either direction.
Offset from Gutter to Inlet	Inlet hydraulic equations assume the Inlet is at the edge of the computed spread (i.e. the Curb). If the Inlet is offset from the Curb, the discharge in the spread section will bypass the Inlet. This value, expressed in terms of feet or meters, is used to compute this bypass and adjust the actual flow to the Inlet. This may be used, for example, when a Grate Inlet is actually set some distance away from the curb of the road. Therefore any flow calculated in that segment will not be added to the inlet and it will bypass to the next assigned one. This value should remain zero for standard VDOT inlet applications.

NODE: SPREAD CRITERIA

Inlets utilize this option to compute the node elevation and spread calculations and for other nodes, it is used for elevations only. The dialog changes slightly depending on whether the Inlet is On Grade or in a Sag. The following are the common components of the Spread Criteria for sections in Sag or On Grade.
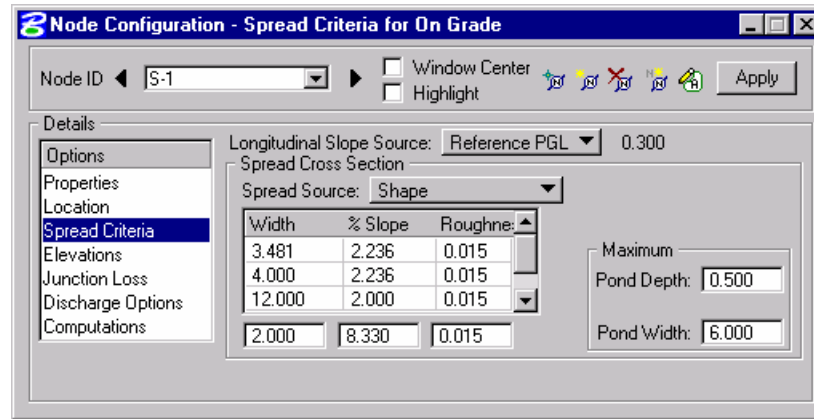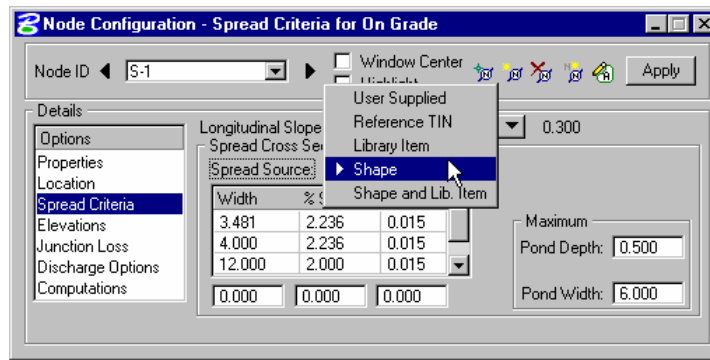

The Spread Cross Section Table in the box above is used to define the spread cross section including the gutter section and/or pavement section which may encompass the spread of flow approaching the Inlet. The top entry is at the outermost point of flow. In most situations, this entry is the section abutting the curb. GEOPAK drainage refers to this part of the cross section as Slope 1, Width 1, and Roughness 1. Adjacent parts of the cross section are identified as Slope 2, Slope 3 etc. as shown below. The cross section defined here may also be used to compute the inlet elevation from the PGL, so the sections should continue to the location of the PGL, if inlet elevations will be computed this way.
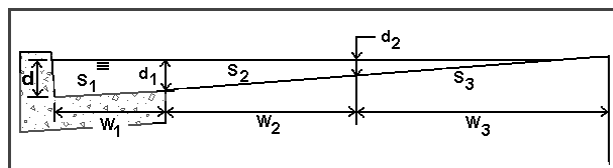
A spread section needs to be defined for every node, including junctions or outlets for the software to design or analyze the system, since the spread section could also be used to compute the structure elevation.

Note   The slopes entered are positive in value if "going up" from the inlet's point of view towards the centerline. Negative slopes are supported but not for the bottom (or last) one in the list since it does not convey flow to the node.

The Spread Cross Section Table could leverage information from the drainage library, the gpk file or the MicroStation Superelevation Shape file as described below.

User Supplied	The designer will enter the width and slopes across the spread section starting from the Inlet and continuing towards the edge of the section farthest away from the inlet.
Reference TIN	A spread cross section will be extracted at the inlet location from the TIN file defined in the Project Components item under the Drainage Project Preferences. The designer needs to be careful on what DTM is using since for most VDOT projects the only TIN file generated is the existing ground DTM.
Library Item	A spread cross section will be assigned to the inlet using a Spread Section item defined in the Drainage Library. Once a Library Item has been selected, the spread source can be changed to User Supplied and the spread section defined by the Library Item modified to accommodate non-typical situations such as pavement width transitions.
Shape	A spread cross section will be extracted at the inlet location from the Superelevation Shapes dgn file defined in the Project Components item under the Drainage Project Preferences and shown in the Spread Criteria table.

For curb inlets in superelevation or superlevation transition, the designer can use this option or enter the spread source (cross section) as User Supplied.

If the superelevation shapes are not defined to the edge of pavement, the designer can take advantage of the toggle:"Extend Superelevation Shapes to Inlet at Shape Slope", located in *Drainage > Project > Preferences* under the Inlet Options item. This will extend the last slope of the shape to the node location point defined in the structure.

Therefore, if the shape file is used to define the spread, be aware that the cross section starts at the edge of pavement creating a theoretical section that is shifted from the actual spread section by the width of the gutter. This is acceptable because the calculated spread width is the same regardless of the shift. The gutter depression cannot be modeled with this approach. To account for the gutter depression while using the shape option, the "Shape and Library Item" would need to be used |
| Shape and Library Item | A spread cross section will be assigned to the inlet combining the section extracted from the Superelevation Shapes dgn file and Library item from the Drainage Library. If the gutter section is made a library |

|	item, this option can be used to account for the gutter capacity.
	To properly use this option, the superelevation shapes need to be defined to the edge of pavement line, and the "Extend Superelevation Shapes to Inlet at Shape Slope" toggle must be off.

Note   Some options could be gray out if no active elements are present. E.g. roadway superelevation shapes files or TIN file not specified in the *Project>Preferences*.

To accurately depict the spread section for inlets along a special gutter grade, the designer can provide the cross section as User Supplied, or use the Superelevation Shapes, if needed to account for a special gutter grade.

The designer can specify threshold values for the spread at each node. When either of the following values is exceeded, GEOPAK Drainage will issue a Warning Message while designing the Network. After a networks is initially designed, it can be Queried for inlets that violate these values.

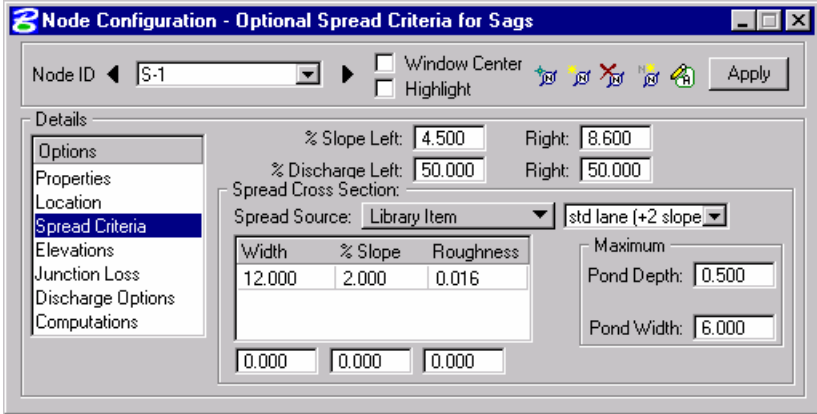| **Maximum Ponded Depth**	The maximum depth allowed for ponding in the cross section (expressed in terms of feet or meters). This does not affect any computations, but will produce a warning if the computed depth is greater.
	For sag inlets, this value is used to compute the capacity of the node.
Maximum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

The Longitudinal Slope for Inlets On Grade can be identified several ways as shown below.
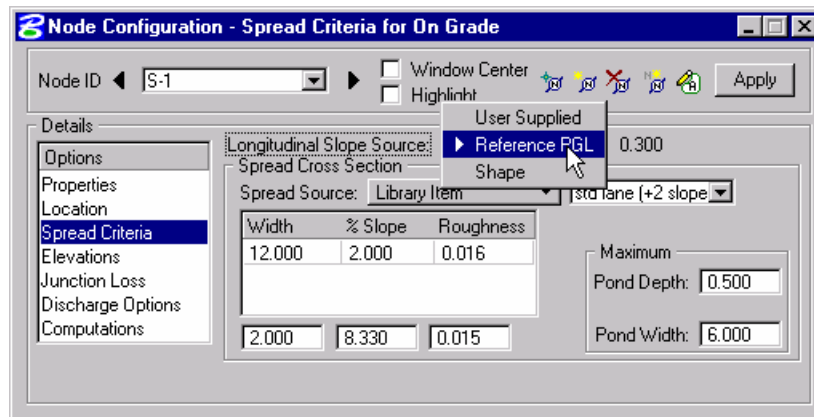
Reference PGL	Slope will be extracted from the profile defined in the Location item of the Node dialog box at the inlet location. The slope will update as the .gpk file is updated.
User Supplied	Slope will be entered by the designer in percentage format while omitting the % sign.
Shape	Slope will be extracted from the superelevation shapes dgn file defined in the Project Components item in the Drainage Project Preferences.

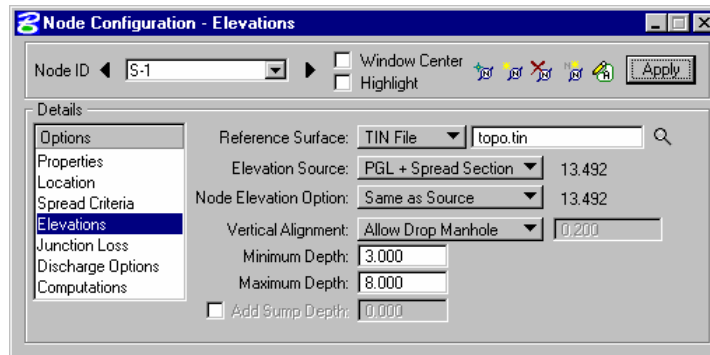|	For inlets in superelevation, superelevation transition, or other transitions, the designer can use this option or enter the longitudinal slope as User Supplied.

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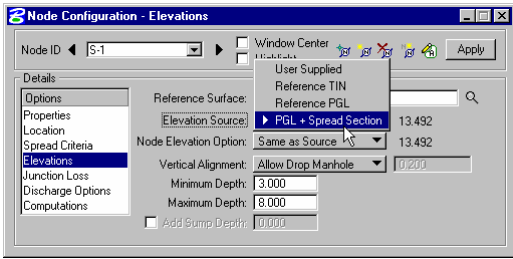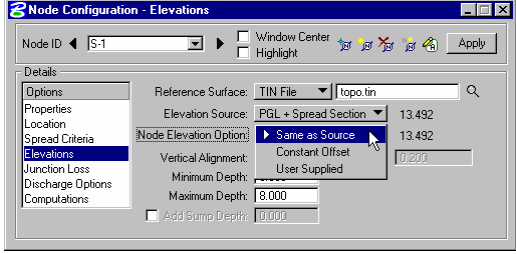

% Slope Left and Right	For defining the longitudinal slope to use in spread (ponded width) computations for the flow approaching from the right and left side of the inlets. The terminology (right and left) is somewhat arbitrary as long as the designer recalls which side is intended as left and right.
% Discharge Left and Right	Defines the percentage of the total discharge to the inlet to allocate to the left and right approach spread computations. The terminology right and left are somewhat arbitrary as long as the designer recalls which side is intended as left and right and is consistent with the definition of % Slope Left and Right

NODE: ELEVATIONS

The Node Configuration-Elevation dialog will assign the elevation of the drainage nodes from different sources, dictate the vertical matching point of the pipes entering and exiting the structure, and the minimum and maximum depth of the node.

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

Elevation Source item. These options are:

- Same as Source: matches the node elevation with elevation source.

- Constant Offset: enables a plus or minus value to the elevation source selected. For example, if the reference profile is PGL+ Spread Section to the flow line of the gutter, a Constant Offset can be specified to have the node elevation be the edge of pavement (EOP) elevation. This is useful because the EOP is used for curb inlet elevations. A plus offset will add to the elevation value.

- User Supplied: The user may specify an elevation.

For the outlet node elevation, the elevation is not a physical point on the structure. The elevation should be at least 1 foot above the tailwater to avoid a warning. In general, using the berm or ground elevation that covers the outlet structure is acceptable for the outlet node elevation.

| **Vertical Alignment** | Alignment preferences for incoming and outgoing pipes from the Node. As pipes are designed the elevations will be set according to the selected preference. Options available are: Match Soffit, Match Invert, Match Surface, Allow Drop Manhole and Min. Fix Drop. |

- Match Soffit: aligns all the pipes in the system by matching the top of each pipe or crown elevation.

- Match Invert: aligns all the pipes in the systems by matching the bottom of each pipe or invert elevation.

- Match Surface: aligns the pipes following the elevation of the water surface inside the pipes.

- Allow Drop Manhole: allows for "drops" inside the node for the pipes arriving or existing the node. Not matching inverts or soffits will be performed. This option allows for greater flexibility. Since the design preference is to minimize excavation, the first pass during the design will align all the pipes at the soffit. As the designer needs to move the inverts of certain pipes due to various factors, e.g. utility conflicts, the change will only affect the selected pipe and others will remain at the previously set elevation.

- Min. Fix Drop: sets a minimum drop in elevation between the

	pipes arriving and exiting the node. VDOT uses this option.
Minimum Depth	Minimum depth expressed in master units (i.e., feet or meters) for the pipe to be buried (minimum cover) measured from the Node Elevation to the soffit (top inside) of the highest pipe. This value is used to establish the minimum cover elevation noted in the Link Configuration Conditions dialog box. See the discussion below.
Maximum Depth	Maximum depth expressed in master units (i.e., feet or meters), for the pipe to be buried measured from the Node Elevation to the invert of the lowest pipe. This value is used to establish the Maximum Depth elevation noted in the Link Configuration Conditions dialog box. See the discussion below.
Add Sump Depth	.Extra depth to be allowed in the manhole structure after the maximum depth of the structure has been computed. This has no effect in the hydraulic calculations, but some designers add this extra depth for pollution control measures. VDOT does not use this option

The designer can use multiple combinations of the previous parameters in order to automate the computation of the node elevations. Some of the project conditions that affect inlet elevations and spread are superelevation, turn lanes, and through lane transitions.

GEOPAK has different ways to handle different project conditions. The options within GEOPAK that may affect calculations of inlet elevations and spread are listed below. It is a the designer discretion to use them depending on the structure and project conditions given.

Item	**Options**
1.- Shape Files: These are created by the roadway designer.	Created to edge of travel way
	Created to less than edge of travel way.
2.- Extend Shapes to Inlet: This option is in Preferences>Inlet Options. This affects the entire project because it is in Preferences.	Yes (activate toggle) if shapes are not created to the edge of travel way.
	No
3.- Longitudinal Slope Source: This option is in Node Configuration> Spread Criteria.	User Supplied
	Reference PGL
	Shape
4.- Spread Source: This option is in Node Configuration> Spread Criteria. This refers to the cross section of the spread area.	User Supplied
	Library Item
	Shape

	Shape + Library Item (this option is not applicable if the shapes are extended to the Inlet
4.- Elevation Source: This option is in Node Configuration> Elevations	Reference PGL
	PGL+Spread
5.- Node Elevation Option: This option is in Node Configuration> Elevations	Source
	Constant Offset
	User Supplied

MINIMUM AND MAXIMUM DEPTH AT NODES DEFINITION

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

The minimum depth of a pipe is often established by cover requirements specified in the VDOT Drainage Manual.

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

Regardless of what depths are used, the designer will still need to check for adequate depth, pipe slope, and hydraulic gradient clearance later in the design process.
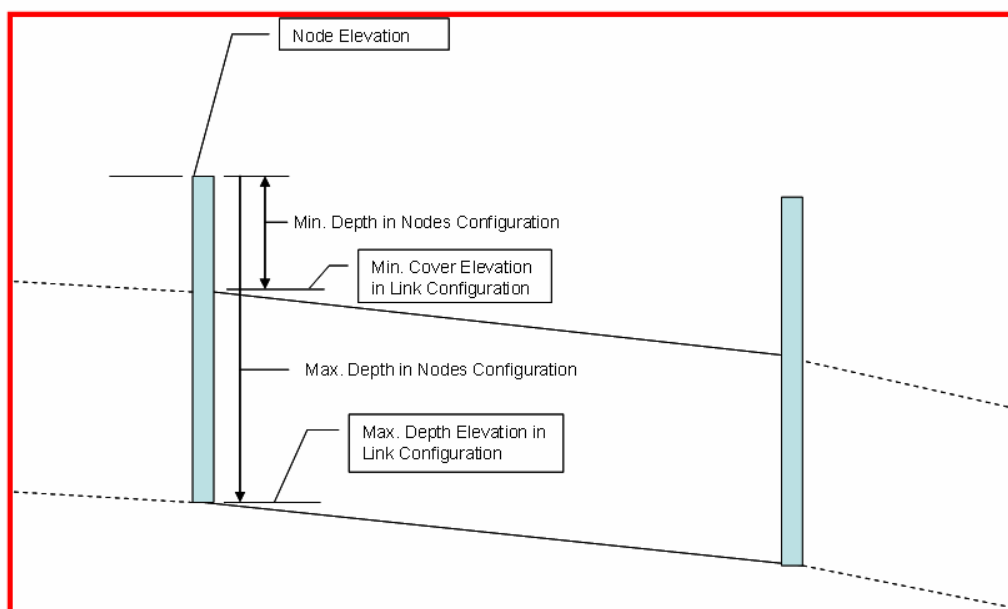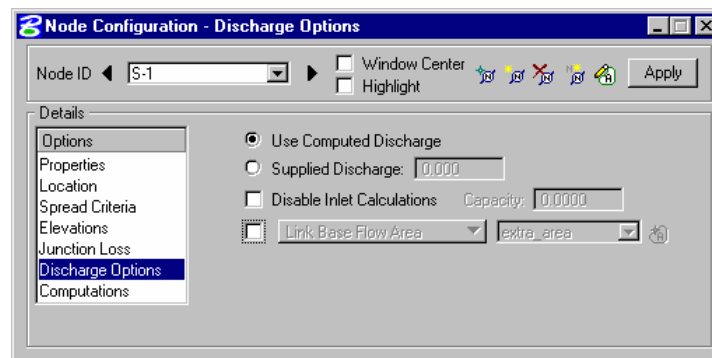
NODE: JUNCTION LOSS
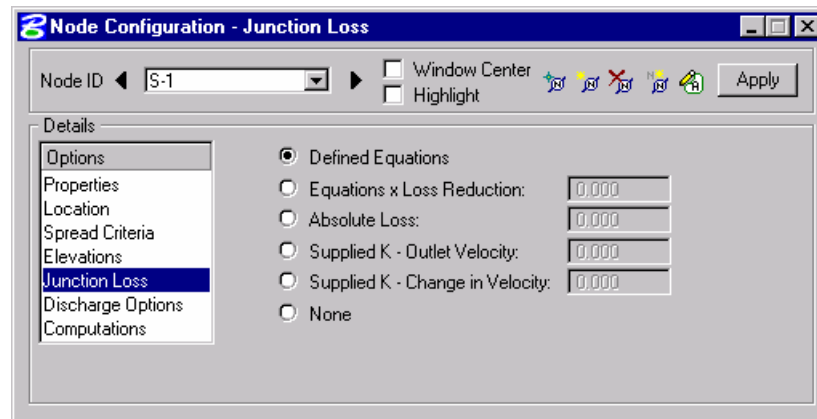
Within this dialog the designer has the option to enter specific values for the junction losses in the particular structure being analyzed. These settings will override the coefficients selected under the Drainage Project Preferences, but the toggle *Disable All Junction Loss Computation* must be **off.**

NODE: DISCHARGE OPTIONS

The Node Configuration-Discharge option is used to select type and possible amount of flow entering the drainage structure directly from its drainage area or from "offsite" locations.

The options are:

Use Computed Discharge	Indicates using the computed discharge from a stored Drainage Area as the discharge to the Node.
Supplied Discharge	This option allows for a user supplied discharge, with disregard to any calculated value from a drainage area.
Disable Inlet Calculations Capacity	When toggled on, the designer may provide the desired inlet capacity overriding any GEOPAK Drainage calculated value.
Link Base Flow Area / Link Base Flow Discharge	When toggled on, an additional drainage area may be defined. The area can be identified graphically by clicking **ID** and selecting the MicroStation shape previously defined as a GEOPAK Drainage Area. For the Link Base Flow Discharge. the designer can also enter a value for the extra discharge in the proper units defined in the Project

|	Preferences (cubic feet per second or cubic meters per second). This "extra" area or discharge is included in the pipe, but is not included in the inlet spread calculations.

NODE: COMPUTATIONS

Once all the information regarding a specific Node has been defined, clicking the Compute Option computes and displays all pertinent information in the list box. If sufficient information has not been provided, warning or error messages are displayed in the list field.
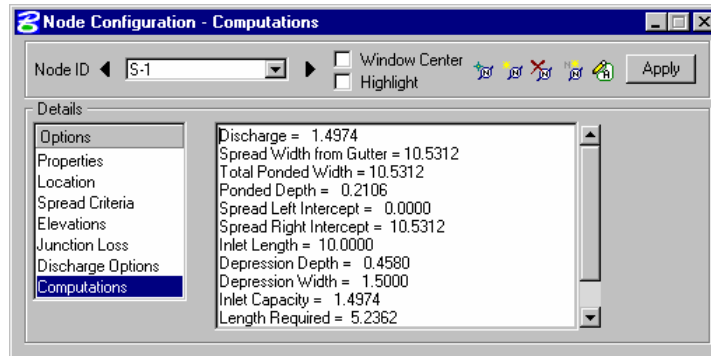

Note   The designer should be aware that the computations shown are the spread calculations. Therefore, the value of discharge may be different from the discharge calculated in the drainage area, since VDOT uses an absolute intensity of 4 inches/hour intensity for inlet computations. This discharge value also includes any bypass discharge added to the current node.

LAB EXERCISE: DRAINAGE DESIGN

> ### *Opening the Drainage Project*

1. Execute *c:\data\geo\vdot\drain1\LAB06_V8ad.EXE*

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

3. Open Project Manager (*MS Menu: Applications > GEOPAK Road > Project Manager)*

4. Select project: *17682.prj* located in *c:\data\geo\VDOT\drain1*, and click OK.
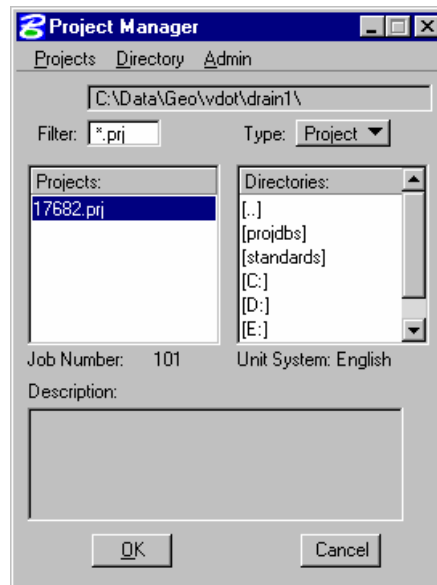
5. Access User: VDOT.
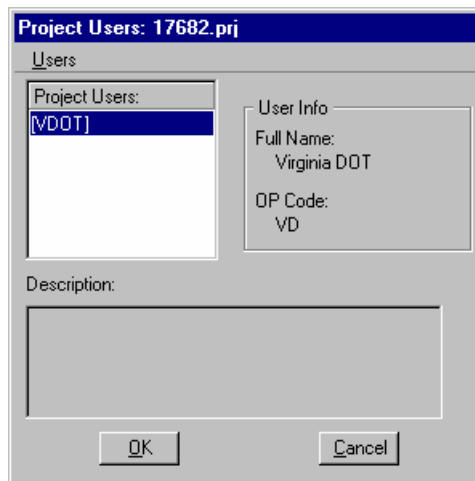
6. Click **OK.** The Road Project dialog will load. This will setup the proper directories for the location of the gpk file.
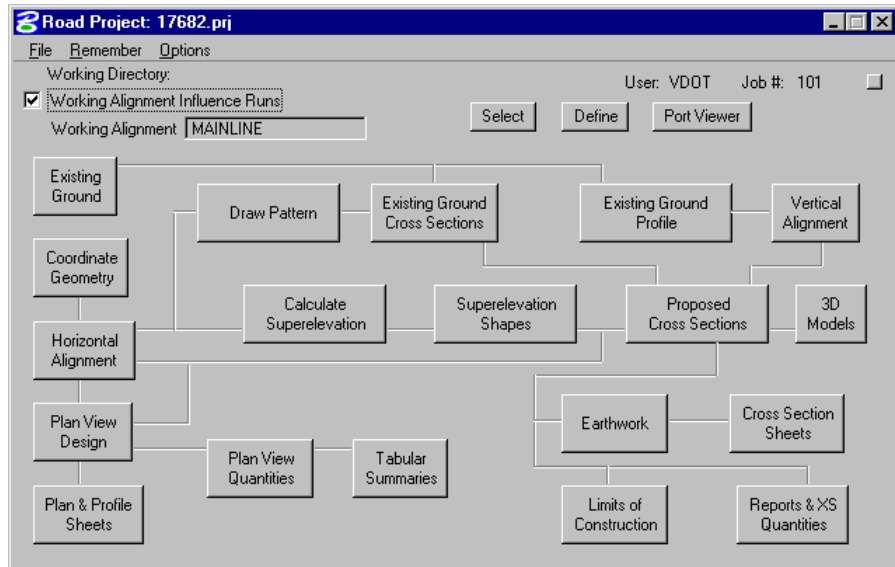
7. Minimize the Road Project dialog box.

8. Access GEOPAK Drainage *(MS Menu: Applications > GEOPAK Drainage > Drainage).*

9. Open the drainage project, *c:\data\geo\vdot\drain1\h17682.gdf (Drainage Menu: Project > Open).*

> *Define Properties and Locate Inlet 3-1*

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

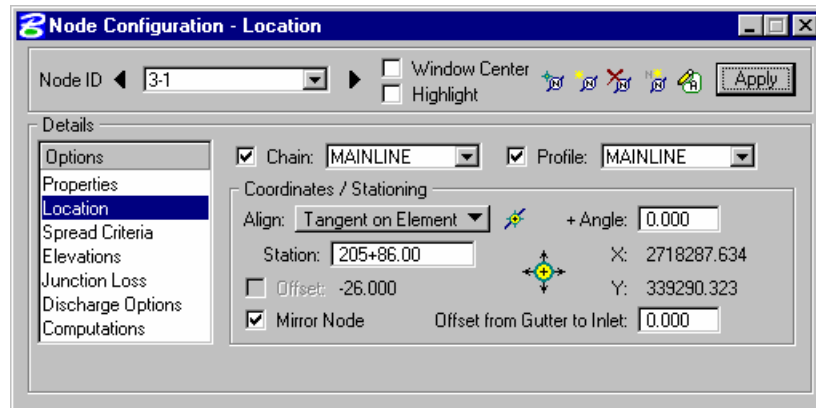2. Select the Add Drainage Node tool *(Drainage: Component > Node > Add)* and click **OK.**
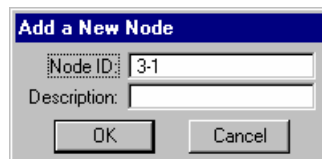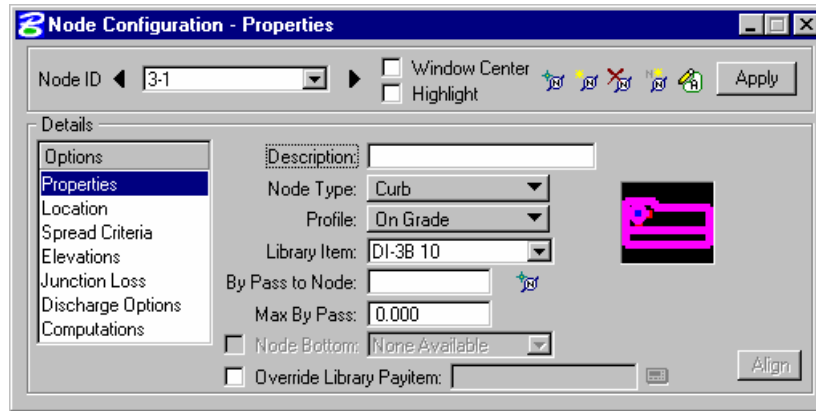
3. Highlight **Properties** in the Options section.

4. Select the Node to be used from the Drainage Library and assign its properties.

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

5. Highlight **Location**.

6. Define the inlet's location in the design plane.

| Chain	MAINLINE
Profile	MAINLINE
Align	Select Tangent on Element. Activate the Select MS Alignment Element button ✶ and select and accept the edge of pavement line. Activate the Station DP button ✛ and the node will dynamically move along the selected element. Using the left mouse button, place the node close to where it is desired. The station can be revised by entering a different station value.

Station	205+86
Offset	-26.00 feet

7. Click **Apply**.

    This automatically places the inlet at the specified location.

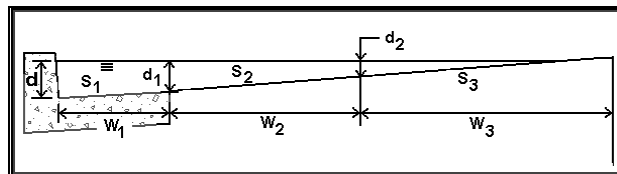> *Define Additional Inlet Information*

1. Highlight Spread Criteria.
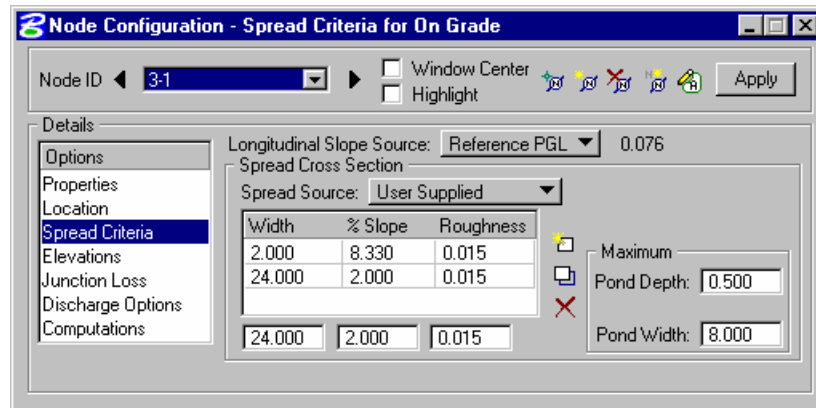
2. Define the roadway cross sectional characteristics directly in front of the inlet.

Longitudinal Slope Source	Reference PGL
Spread Source	User supplied
Max Ponded Depth	0.50
Max Ponded Width	8.00

These values will be utilized to calculate inlet capacity and resulting by-pass flow.
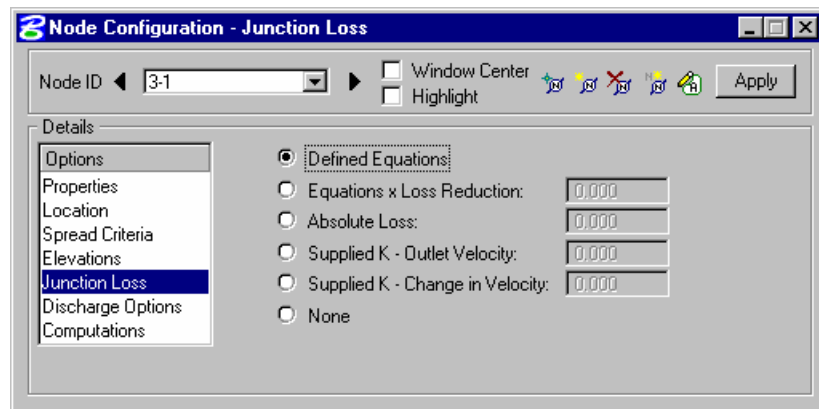
3. Highlight **Elevations**.
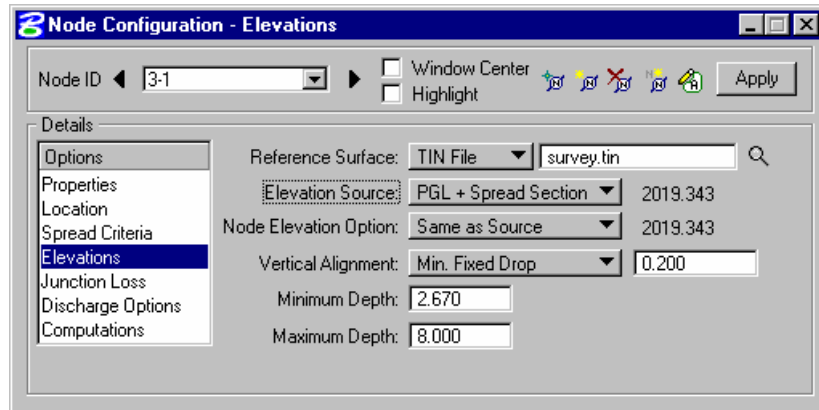
4. Assign the inlet's vertical elevation and vertical pipe alignment.

| 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	8.0 feet

5. Highlight **Junction Losses.**

6. Use **Defined Equations** (defaults to *Preferences > Junction Loss Settings*).

7. Highlight **Discharge Options**, and Toggle *ON* **Use Computed Discharge**
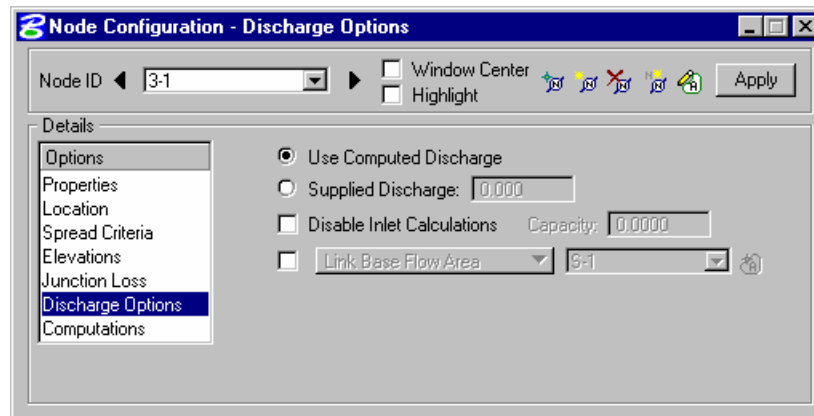
> *Complete Hydraulic Computations and Add Node to the Project*

1. Highlight **Computations** to verify the inlet's hydraulic computations.
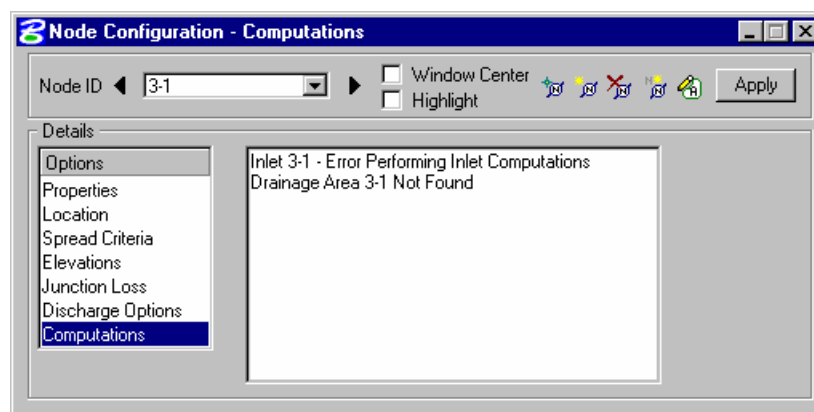
Note   The Drainage Area for this node won't be added until the next step. Therefore, the Computations for the node can't be completed until a discharge is known (Delineate Drainage Areas section).

2. Click **Apply** to add this Node to the project.

DELINEATING DRAINAGE AREAS

The physical drainage area boundary may be delineated using a digital terrain model, simply drawn with MicroStation, or 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.

> *Design Drainage Area for 3-1*

1. The basin limits were predefined and were drawn on the MicroStation Level: *63*. A closed shape will be created in the next steps in order to become the drainage area for each structure.

2. Turn On Level *63* in the MicroStation dgn file. (*MS Menu: Settings>Level>Display).*
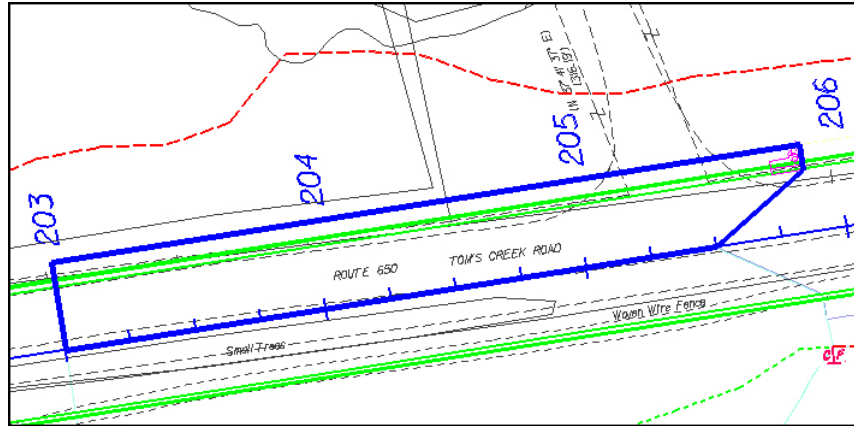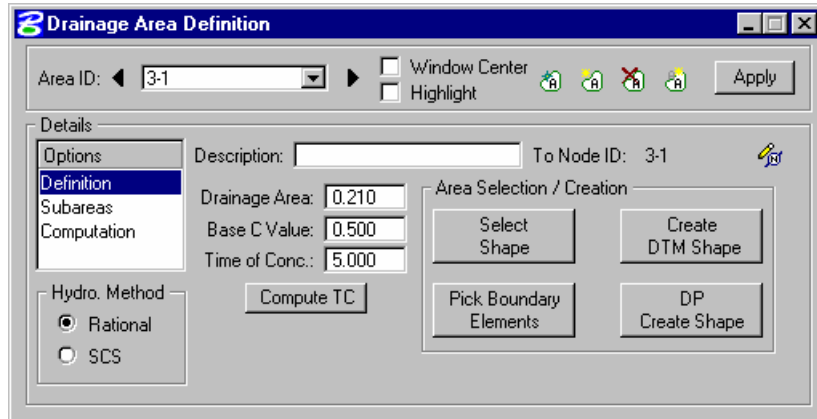

3. Still in the Node Configuration dialog, highlight **Properties**, and click **Edit Area** (to the left of the Apply button).
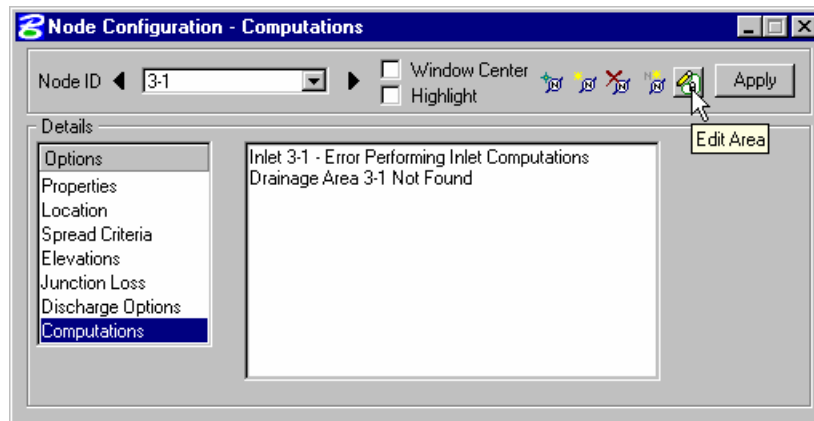
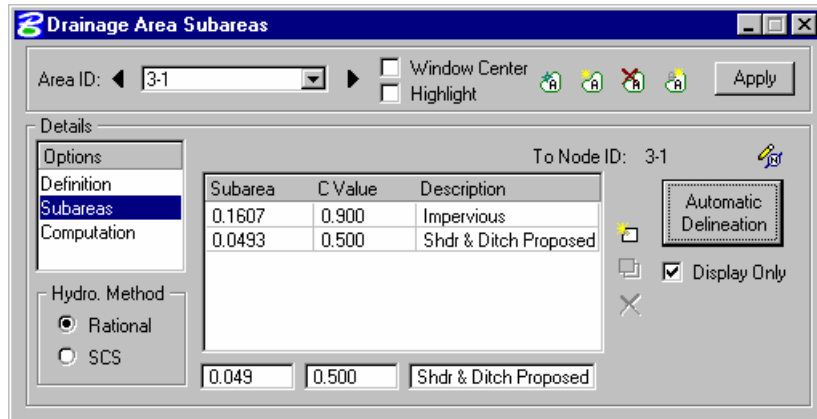4. Select **YES** at the prompt of creating a new drainage area and enter the Definition information.

Base C	0.50
Time of Concentration	5.0

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

6. Highlight **Subareas**.

7. Toggle on **Display Only.** Then, the subarea values will not be written in the dgn file.

8. Click **Automatic Delineation**.

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

9. Click **Apply**.

   This applies the land uses (and their "C" values) to the Drainage Area. Alternately, this table may be populated manually using the key-in fields and **add**, **modify** and **delete** icons to the right of the table.

10. Highlight **Computation**.
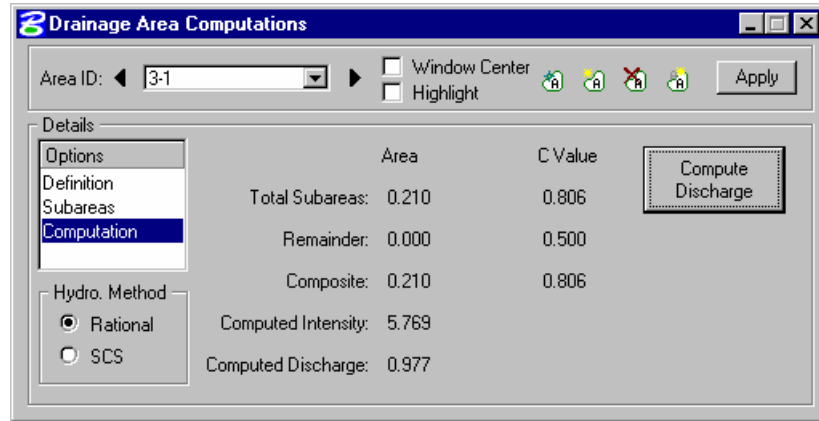
11. Click **Compute Discharge**.

12. Verify the computations.

13. Click **Apply** to add the Area to the Project.

14. Close the Drainage Area Computations dialog.

15. Return to the Node Configuration dialog, and highlight **Computations**.

16. Review and verify the computations.

17. Click **Apply** to update the node information.

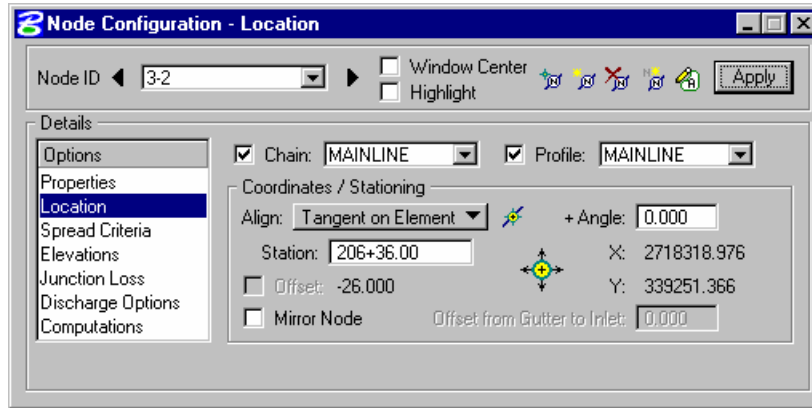### Define Properties and Locate Inlet 3-2

1. Visually determine (zoom to) the location of the Proposed Inlet Location: Station 206+36; offset –26.

2. Select the Add Drainage Node tool *(Drainage: Component > Node > Add)* and click **OK.**

3. Highlight **Properties** in the Options section.
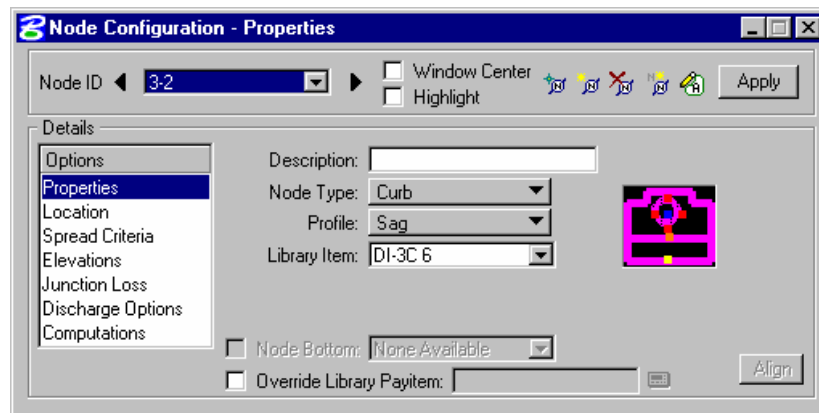
4. Select the Node to be used from the Drainage Library and assign its properties.

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

5. Highlight **Location**.

6. Define the inlet's location in the design plane.

Chain	MAINLINE
Profile	MAINLINE
Align	Select Tangent on Element. Activate the Select MS Alignment Element button ⚡ and select and accept the edge of pavement line. Activate the Station DP button ⬦ and the node will dynamically move along the selected element. Using the left mouse button, place the node close to where it is desired. The station can be revised by entering a different station value.
Station	206+36
Offset	-26.00 feet

7. Click **Apply**.

   This automatically places the inlet at the specified location.

8. Highlight **Spread Criteria**.
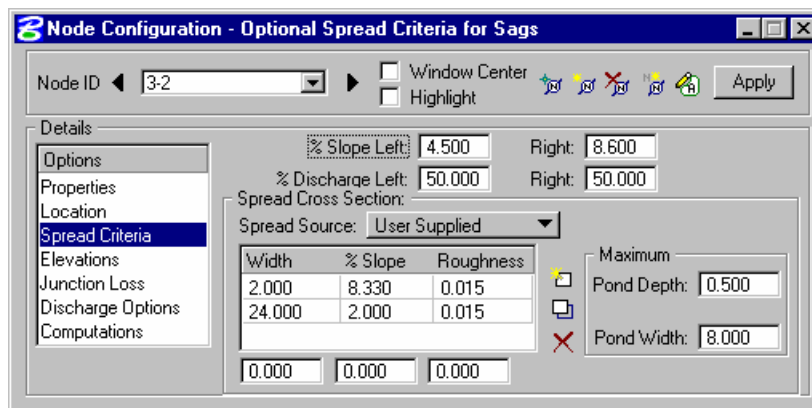
Longitudinal % Slope Left	4.50%
Longitudinal % Slope Right	8.60%
% Discharge Left	50 %
% Discharge Right	50%

10. Highlight **Elevations** and assign the inlet's vertical elevation and vertical pipe alignment

11. Highlight **Computations** to verify the inlet's hydraulic computations.

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

12. Click **Apply** to add this Node to the project.

13. Click the **Edit Area** icon to associate this inlet with its Drainage Area.

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

15. Enter the Definition information.
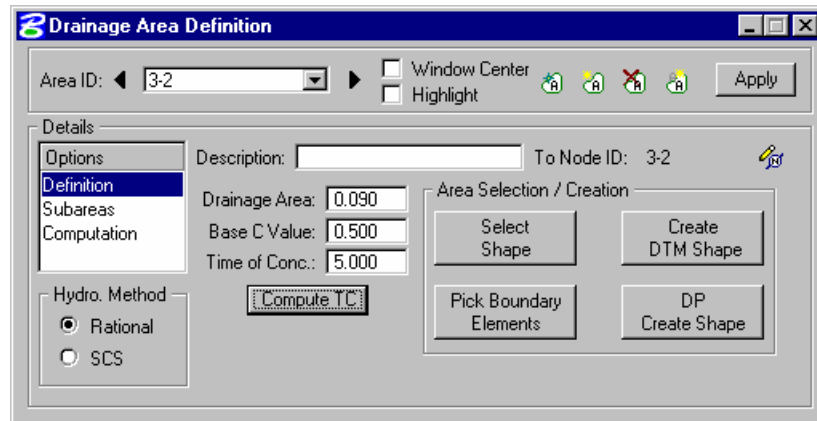
Base C	0.50
Time of Concentration	5.0

16. Click **Select Shape** and select the MS shape for this area. The drainage area value is computed.
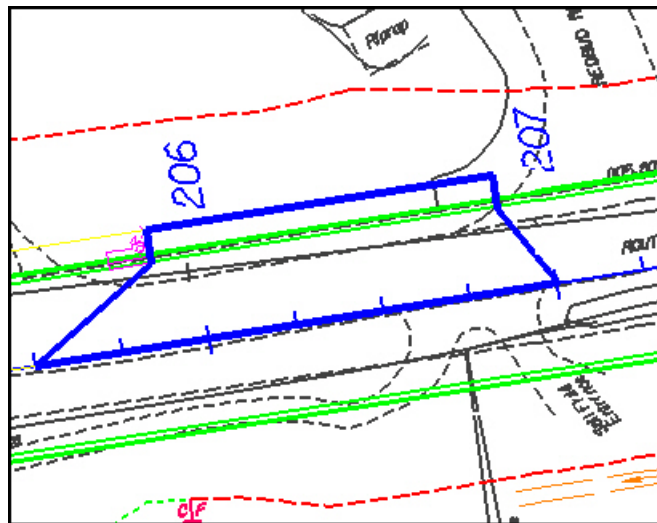
17. Highlight **Subareas**.

18. Toggle on **Display Only.** Then, the subarea values will not be written in the dgn file.

19. Click **Automatic Delineation**.

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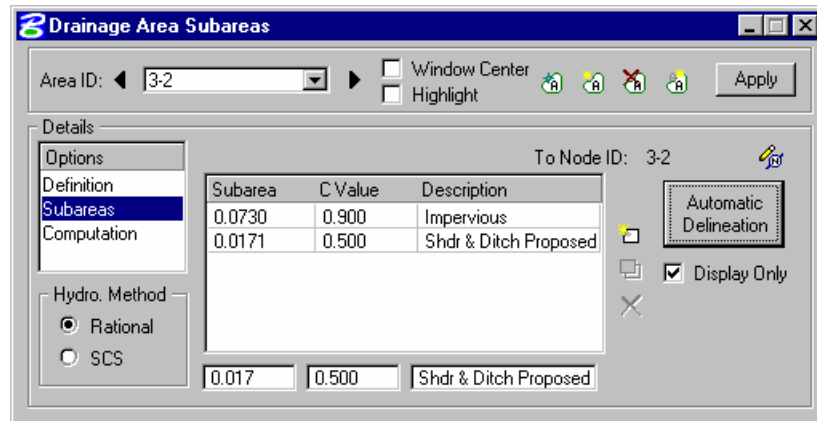
20. Click **Apply**.

    This applies the land uses (and their "C" values) to the Drainage Area. Alternately, this table may be populated manually using the key-in fields and **add**, **modify** and **delete** to the right of the table.

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

22. Click **Apply** to add the Area to the Project.

23. Close the Drainage Area Computations dialog.

24. Return to the Node Configuration dialog, and highlight **Computations**.
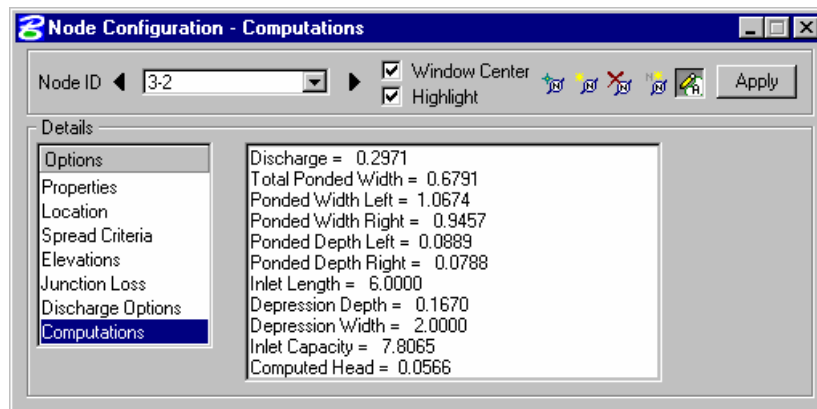
25. Review and verify the computations.

26. Click **Apply** to update the node information.

COMPLETE THE INLET LOCATIONS

1. Select from the Node Configuration dialog the **Add Node** icon.

2. Layout the inlets for the project using the same methodology as described above, and create the correspondent drainage areas.

Node ID	Type	Profile	Library Item	Station	Offset	Elevation
3-3	Curb	On Grade	DI-3B 8	206+86	-26.00	PGL+Spread
3-4	Curb	On Grade	DI-3B 16	205+86	26.00	PGL+Spread
3-5	Curb	Sag	DI-3C 6	206+36	26.00	PGL+Spread
3-6	Curb	On Grade	DI-3B 8	206+86	26.00	PGL+Spread
3-7	Curb	On Grade	DI-3B 16	208+00	26.00	PGL+Spread
Outlet	Outlet		ES18	206+36	-42.00	El. 2020.00

Note It is recommended to input the Node Elevation for an outlet node at least one foot higher than the tailwater elevation. This elevation is not used in the hydraulic calculations and does not represent a physical point on the outlet node. E.g. pond's top of berm elevation, ditch top of bank, etc.

3. Close the dialog box when done.

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.

Several options are supported to add, edit, and delete Links. One option is invoked by the Drainage pull down menu *Component>Link.* Another option is the tool box obtained by selecting *Tools > Tool Boxes > Link*, then identifying the desired tool from the tool box as depicted below.


Tools from left to right are:

- Add Drainage Link
- Edit Drainage Link
- ID Drainage Link

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

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. The Link Edit dialog is identical to the Link Add dialog, however, when the Link is identified, all associated data is displayed.
ID	Utilized to ID the link in the design file.
Delete	Utilized to select and delete the specified Link and associated data.
Rename	Rename the link name in the GEOPAK database.
Update All Links	Updates all link graphics in the design file.
Update All Links with Pay Items	Updates all link graphics with the designated pay items (specified in the Drainage Library) from the Design and Computation Manager database.
Drainage Link Report Configuration	Displays in table report the links configuration (size, inverts, and slopes) after the design is performed.
Drainage Link Report Computation	Displays in table report the links hydraulic computation (discharge, capacity, HGL) after the design is performed.

A third option is to access the Links the **Navigator** *(Utilities>Navigator).*
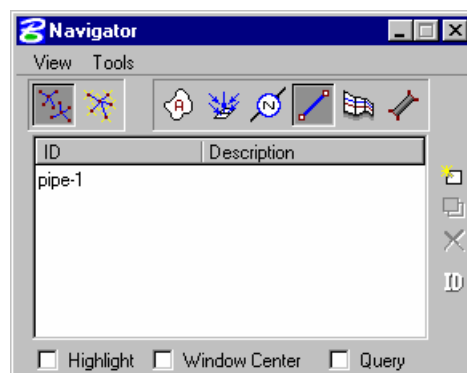
LINK CONFIGURATION DIALOG: INTRODUCTION

The Link Configuration dialog box is the one used to properly place a link (pipe or ditch) into the database.
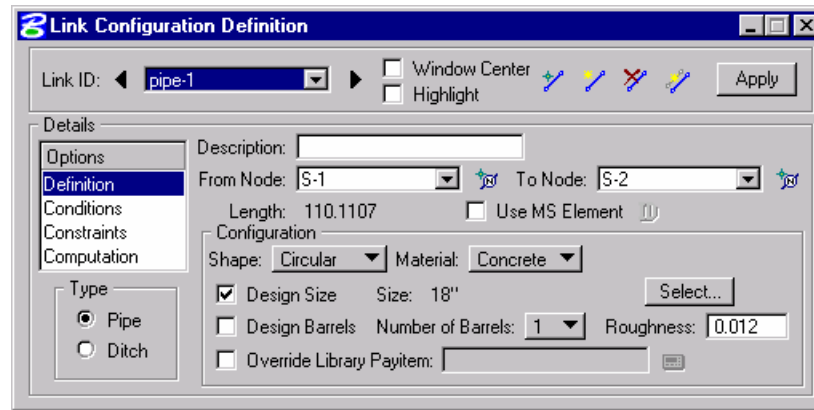

The Link options are displayed in a list box in the left side of the dialog, while the right side displays parameters for the highlighted Option.   The options are:

- Definition

- Conditions

- Constraints

- Computation

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

LINK CONFIGURATION DIALOG: DEFINITION

This option will define the Link's path, connectivity, shape, material, roughness coefficient, size, and design options.
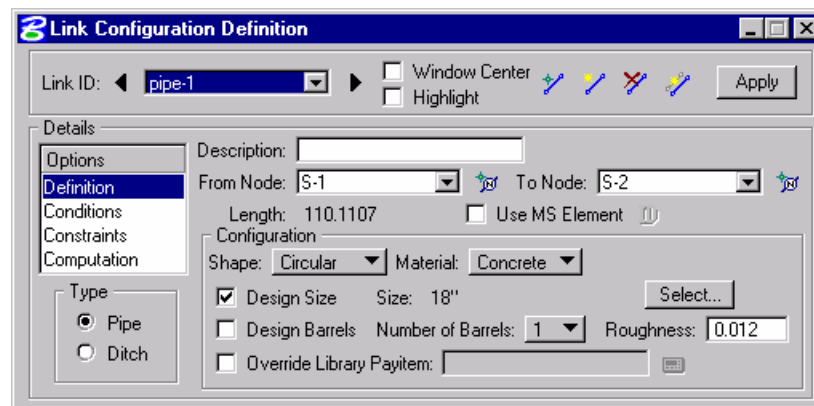
Description (optional)	Description of the Link. (Maximum of 32 alphanumeric characters.)
From Node	Identification of the Node from which the Link originates. The flow direction, upstream or downstream, need not be established. The

 VDOT GEOPAK Drainage

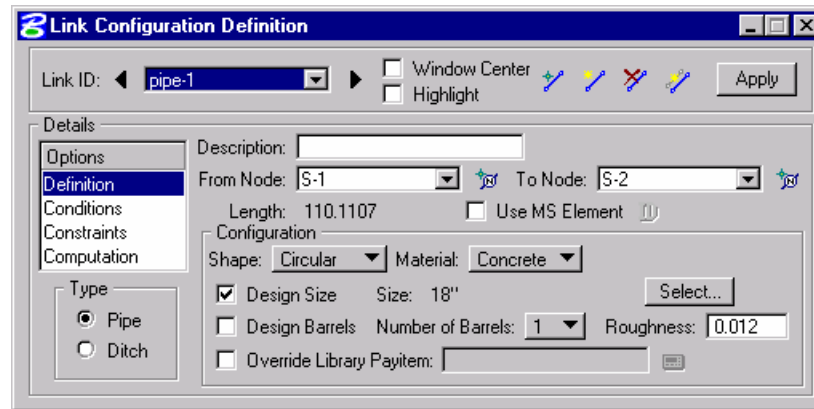|	
	direction is determined by the connectivity of the Network in which the Link resides. If the designer identifies a downstream node as a From Node, GEOPAK will reverse the From Node and To Node based on actual flow direction of the Network. GEOPAK does this when the network is built (See the Network Chapter). It is recommended to use the ID button to select the node since it will connect the link to the proper pipe connection points defined in the Department's Drainage Library.
To Node	Identification of the Node from which the Link originates. The flow direction, upstream or downstream, need not be established. The direction is determined by the connectivity of the Network in which the Link resides. It is recommended to use the ID button to select the node since it will connect the link to the proper pipe connection points defined in the Department's Drainage Library
ID	Identification of the Node from which the Link originates or ends. The flow direction, upstream or downstream, need not be established. The direction is determined by the connectivity of the Network in which the Link resides.
Length	Once the From Node and To Node are identified, GEOPAK computes the length of the Link based on distance between Nodes or the MicroStation element selected as the Link path. GEOPAK Drainage considers two lengths for a link: actual length and hydraulic length. • Actual Length: this is the distance from the pipe connection point of one node to the pipe connection point of the next node. This is often called the construction length of the pipe. • Hydraulic Length: this is the distance from the hydraulic point of one node to the hydraulic point of the next node. This length is used to perform the drainage calculations (travel time, HGL slopes, etc). The Department has defined the hydraulic point of every node at the center of the structure.
Use MS Element	To define and select the Link path with previously drawn MicroStation elements. Activate the toggle, press the Select button, then identify the element. This is only used when the designer needs to model a link with a bend without a node.
Type Group: Pipe or Ditch	Selection of this option dynamically changes the Configuration group box to reflect the selection. Links in GEOPAK Drainage could be a pipe or a ditch.

The Configuration group box will change depending upon designer's choice to use a pipe or ditch as a link between nodes.
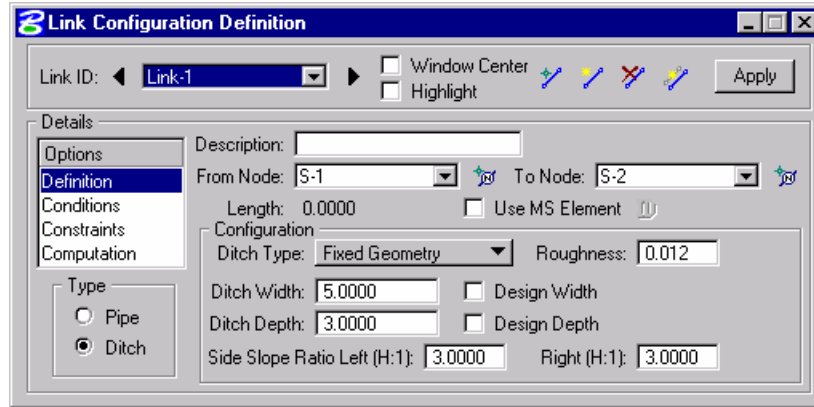
If a pipe is selected, then the Configuration Group box will allow the following:
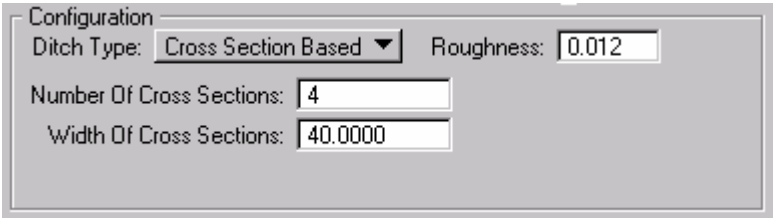

Shape	Option button for pipe shapes. The Department's Drainage Library contains only box, circular, and elliptical shapes.
Material	Option button for Pipe materials. The Department's Drainage Library has all the links stored as concrete material, so the designer must select concrete. This does not control the pipe material for construction. The pipe material constructed is controlled by the Department's Optional Pipe process.
Design Size	Toggle on to indicate pipe size to be determined by the software based on given design Configuration and Constraints. If the pipe is an existing one or the designer wants to select a specific pipe, then toggle off this option. Pressing the **Select** button invokes the Select Size dialog, wherein the desired size may be selected. Pipes listed in the dialog are based on the sizes in the Drainage Library for the specified Shape and Material configuration.
Design Barrels	Toggle on to indicate the number of barrels (multiple pipes) GEOPAK is to design based on given configuration and constraints. If the toggle is not activated, GEOPAK designs for one barrel. The designer can select from 2 to 10 barrels from node to node.
Manning's n	Manning's roughness coefficient. The default value is based on shape and material found in the Drainage Library. All of the current Library items have Manning's n = 0.012. This is consistent with the Drainage Manual standards for acceptable Storm Drain and Cross Drain materials. The designer can also override this value for each individual pipe.
Override Library Pay Item	Each pipe in the drainage library is associated to a pay item or Item ID in the Design and Computation Manager database. This option will override this linkage, allowing the user to select their own Item ID. The Department uses that option to make the placed pipe an "existing"

	pipe for quantity computations.

If a ditch option is selected, then the Configuration Group box will allow the following:


| Ditch Type	A ditch with fixed geometry or cross section based could be placed.
Ditch Width	Ditch bottom width in master units (i.e., feet or meters).
Ditch Depth	Ditch depth in master units (i.e., feet or meters).
Side Slope Ratio Left	Left side slope of the ditch.
Side Slope Ratio Right	Right side slope of the ditch
Manning's	Manning's roughness coefficient.
Number of Cross Sections	If the ditch type selected is cross section based: Number of cross sections to create along the path of the Link. A minimum of four sections is suggested to balance energy. The TIN file or Site Model to extract the cross sections is specified in the Reference Surface option of the Node Definition dialog.
Width of Cross Sections	Width of the cross sections created along the path of the Link. Cross sections will be centered along the Link.

LINK CONFIGURATION DIALOG: 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.

The Minimum Cover and Maximum Depth values on this dialog represent calculated elevations based on the values for the nodes connected to the link. The Minimum Cover elevation = Node Elevation - the Minimum Depth of the node. The Maximum Depth Elevation = Node Elevation – Maximum Depth of the node.
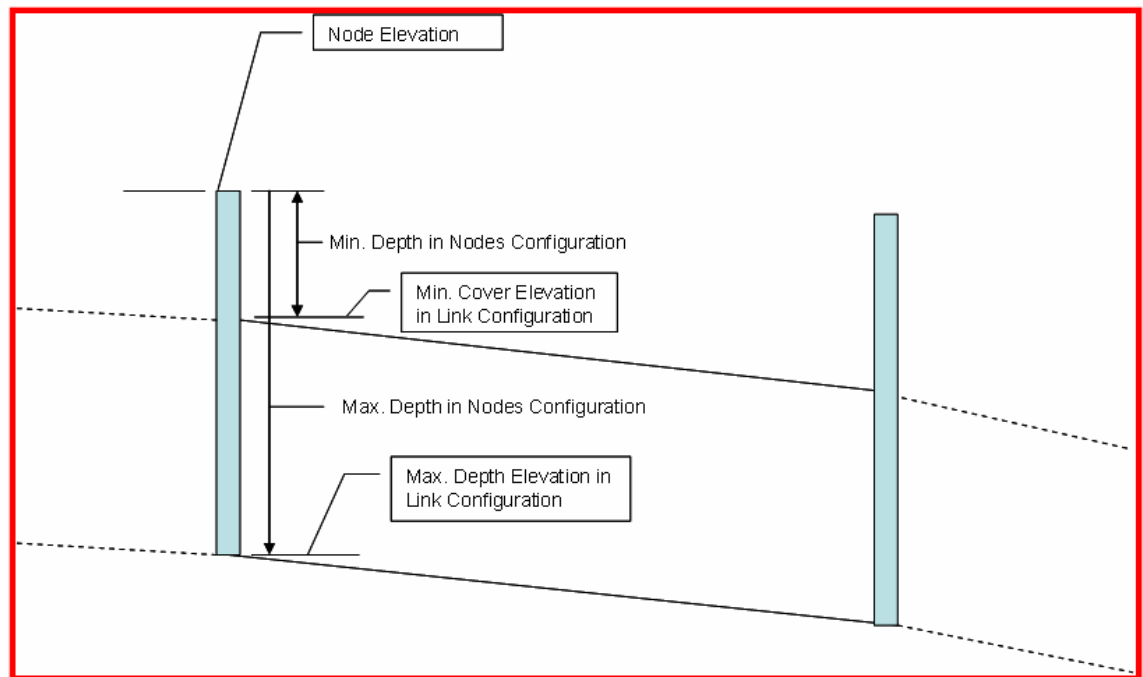
The soffit, invert and slope of the pipes are the calculated elevations after the Network is designed. The designer has the option to "lock" these elevations, therefore GEOPAK Drainage will not adjust them. To lock one of these items, activate the toggle and type the desired value in the window. Another option to lock the pipes elevations is thru the Edit Profile dialog box.
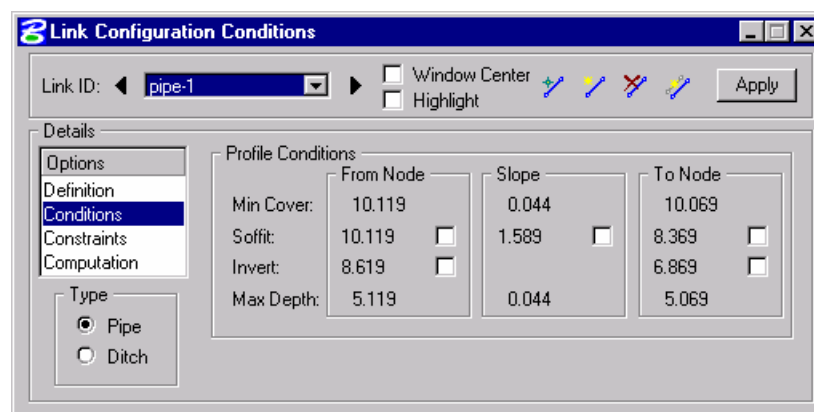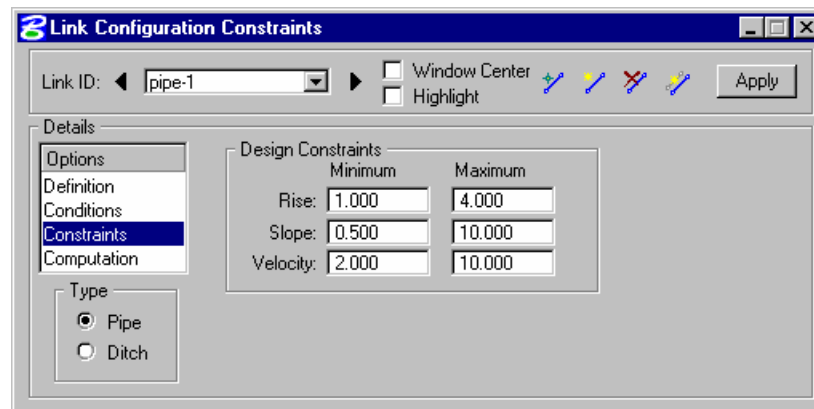

Note   GEOPAK Drainage cannot accommodate adverse pipe. If locked or held flow lines result in one of these conditions, GEOPAK will adjust the flow lines to fix the problem, thus giving the

> appearance that the Lock or Hold Invert option does not work. A Warning message will inform
> the designer of the changes made to the pipe elevations after the design is performed.
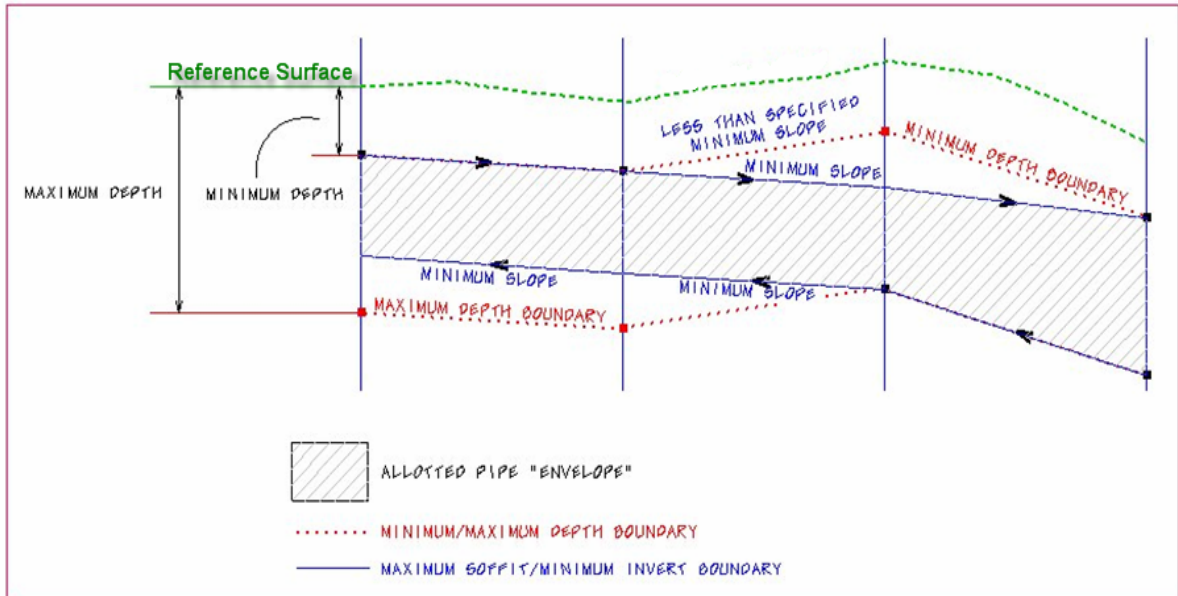
LINK CONFIGURATION DIALOG: CONSTRAINTS

The Constraints establish further design criteria for Links. GEOPAK Drainage utilizes the physical
constraints of minimum and maximum size and slope to determine a suitable pipe size.

GEOPAK Drainage will perform the following steps to determine the "allotted pipe envelope"

- Starting upstream, GEOPAK computes each link slope by connecting the nodes at the
  elevation set by the node elevation minus the minimum depth.

- If this slope is less than the minimum slope, GEOPAK Drainage sets the slope equal to the
  minimum slope. This will provide the upper limit of the envelope. Then, it continues
  downstream.

- To setup the bottom part of the "envelope" , it starts from the outlet elevation, move upstream
  and setup the minimum invert of the pipes by using node elevation minus the maximum depth
  values on each node.

- If the invert falls within the minimum and maximum slopes the link is placed at the calculated
  elevation. If not, the link is once again placed at the minimum specified slope from that node
  to the upstream end of the system.

- If during the construction of the envelope, any elevations that the user has held will be
  accommodated if physically possible and the envelope constrained to that elevation.
  However, if a held elevation of a pipe violates the minimum or maximum depth line, the
  elevation will not be held. With the minimum and maximum depth profiles computed,
  GEOPAK Drainage can then design a suitable pipe within this envelope that satisfies all the
 constraints possible.

The other constraints, minimum and maximum velocities are for querying purposes and evaluating the adequacy of the design.

|	
Minimum Rise	Constraint - Minimum rise (diameter, height) for pipe or ditch in master units (i.e. feet or meters). The Department's minimum round pipe size is 1.5' (18").
Maximum Rise	Constraint - Maximum rise (diameter, height) for pipe or ditch in master units (i.e. feet or meters). Unless the designer knows a constraint, it is recommended that this be set for a large pipe such as a 6' (72").
Minimum Slope	Constraint - Minimum physical slope for pipe or ditch expressed in percentage. The minimum pipe slope varies with the pipe size (0.15% for 18", 0.102% for 24" etc.) See the discussion in the Storm Drain Handbook. During the initial design, the pipe sizes are not known, so this constraint may need to be changed as the design progresses. In moderately sloped terrain, using 0.15% may be acceptable for all pipes, because the terrain will force the pipes to be steeper than that. In flat terrain, using 0.15% for all pipes may cause the downstream pipes to be too deep.
Maximum Slope	Constraint - Maximum physical slope for pipe or ditch expressed in percentage. The Department does not have a maximum slope constraint, so it is recommended that this be set high for all pipes.
Minimum Velocity	Query - Minimum velocity for pipe or ditch expressed in feet per second or meters per second. This value must be something greater than zero for the program to do the hydraulic calculations.
Maximum Velocity	Query - Maximum velocity for pipe or ditch expressed in feet per second or meters per second.

LINK CONFIGURATION DIALOG: COMPUTATION

The Computations option allows for evaluating the computed hydraulic properties of the Link. Link hydraulics are not available for review until a Network has been established containing the Link and the Network hydraulic computations have been successfully performed.
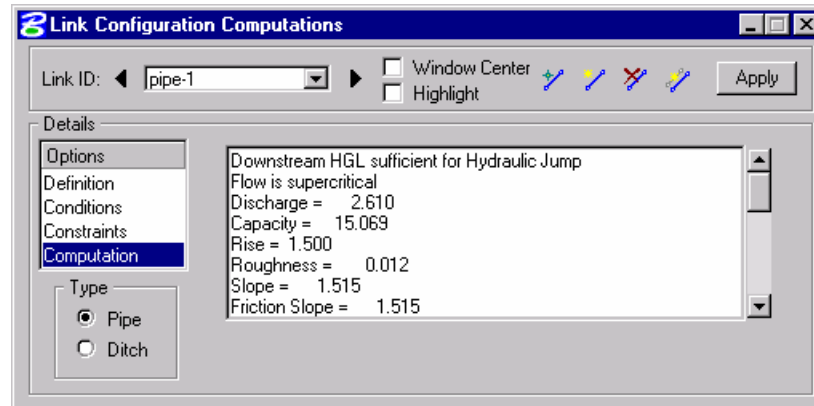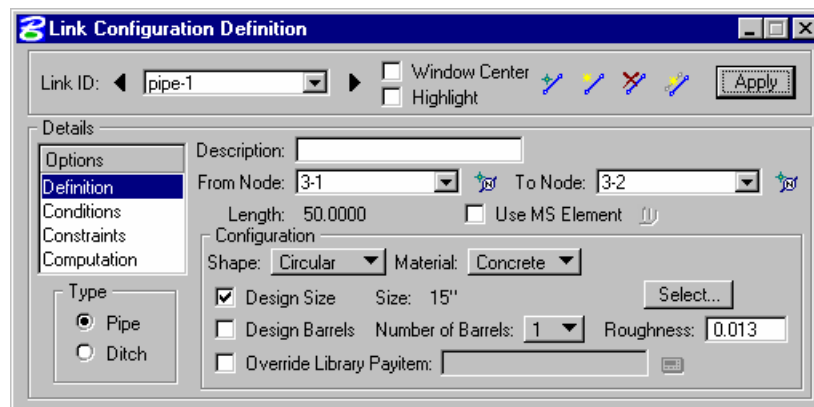

LAB EXERCISE: LINKS

> *Link Placement*

1. Select the Add Drainage Link tool *(Drainage: Component > Link > Add).*
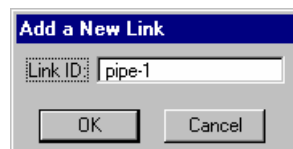
2. Click **OK**. We are using the default Link ID.

3. Highlight **Definition.** This sets the pipe's spatial characteristics including From and To Nodes ID's, pipe shape, material, library items, etc.

4. Graphically identify the From Node (3-1) by selecting the ID Button ⚐ and left clicking in the proper pipe connection point of the structure.

5. Accept the highlighted point by left clicking one more time.

6. Graphically identify the To Node (3-2) by selecting the ID Button ⚐ and left clicking in the proper pipe connection point of the structure

7. Accept highlighted point by left clicking one more time

8. Set the other pipes properties under the Definitions option.

| Shape	Circular
Material	Concrete
Design Size	Toggle ON
Design Barrels	Toggle ON and Select 1 for the Number of Barrels, then Toggle OFF.
Manning's n	0.013

5. 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; Drainage will design all the profiles for this project.
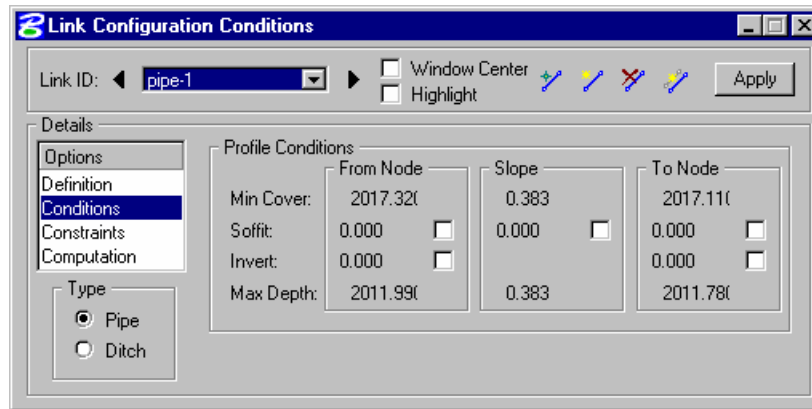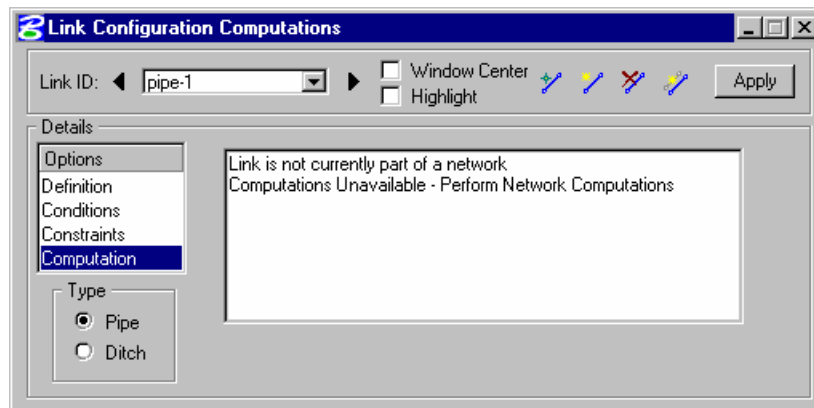
6. Highlight **Constraints**.
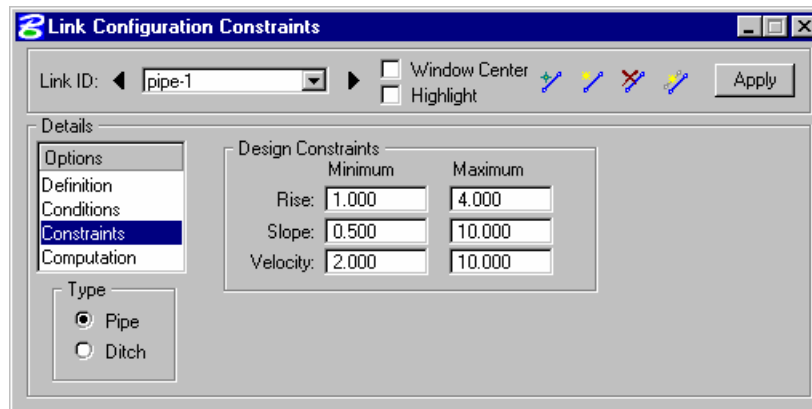
7. Establish the min/max design criteria for Links.

Rise min/max	1.0 / 4.0 (feet)
% Slope min/max	0.5 / 10.00 (%)
Velocity min/max	2.00 / 10.00 (fps)

8. Highlight **Computations**.

   This displays 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 here for computations after the Network has been added and designed or analyzed.

9. Click **Apply.** This incorporates the link to the project.

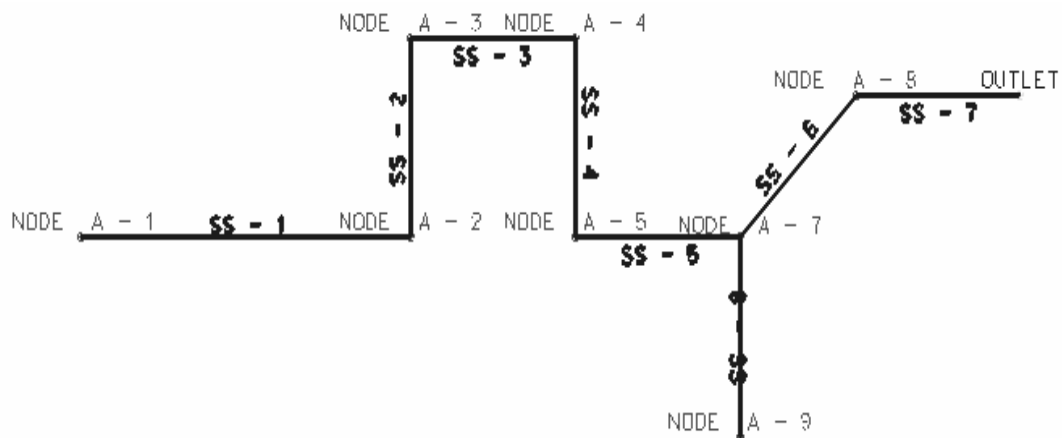10. Add the remainder of the link conveyance system by clicking **Add Link**.

Note   As Links are added, most dialog values default from the previous Link with the exception of the node and elevation information.

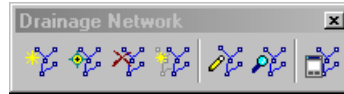| Link	Traverses From Node To Node
PIPE-2	3-3 To 3-2
PIPE-3	3-4 To 3-5
PIPE-4	3-7 To 3-6
PIPE-5	3-6 To 3-5
PIPE-6	3-5 To 3-2
PIPE-7	3-2 To Outlet and set the invert elevation at the Outlet node at el. 2013.00 using the Conditions option in the Link Configuration dialog box.

11. Close the dialog box when done.

NETWORKS: 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 main menu bar as depicted below. Alternately, the Network tools are invoked by selecting **Tools > Tool Boxes > Network**, then identifying the desired tool from the tool box.

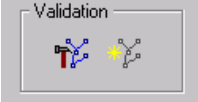Seven tools are supported in the Network tool box (from left to right).

Tools	Description
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.
Rename	Utilized to rename any previously defined Network.
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.
Set 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.

NETWORK CONFIGURATION DIALOG

It is within this dialog where a network is built and computed. The dialog box is divided into two major groups: Validation and Computations.
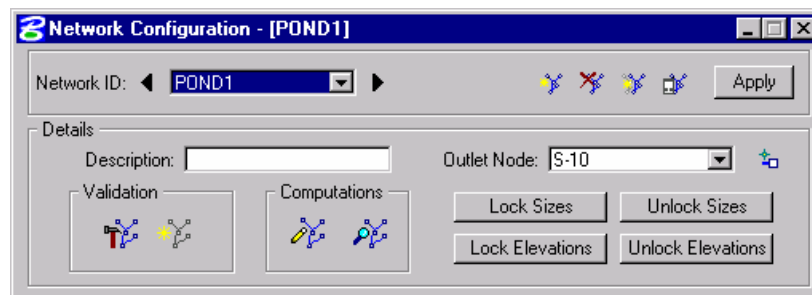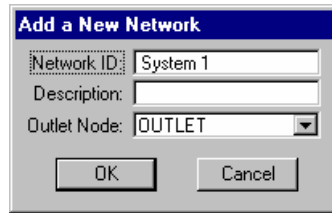

Validation: Build Network. Highlight Network.	**Build Network:** Builds the storm drain network starting from the Outlet Node. The purpose of building a Network is to check the topology and connectivity and determine whether this storm drain system is ready to

Validation	be analyzed or designed. It is at this time that all the flow directions through the Links are established. The Network is established by traversing from the Outlet Node upstream, splitting off into multiple paths at a fork, until terminating nodes are reached on all paths. For a gravity system, the topology must be a tree structure. This process validates all the interconnected components for errors and/or unexpected conditions like: no or multiple outlets identified, indeterminate flow path due to loops, duplicate links, or multiple downstream links from a single node. **Highlight Network:** Highlights all elements (**Areas**, **Nodes**, and **Links**) contained in the current Network for visual inspection of the system.
Computations: Design. Analyze Computations	**Design:** Performs the hydraulic design of all the components contained in the Network. It evaluates the current Network, and designs any features that were indicated as such. **Analyze:** Performs the hydraulic analysis of all components contained in the Network. It evaluates the current network, ignoring all design toggles and maintaining current feature sizes. This can be helpful when evaluating a system that has already been designed to see how it accommodates a different design storm. This is also helpful when the designer is changing the size of certain pipes to meet the hydraulic gradient clearance requirements. In this case the designer may want to change only one pipe while holding the size of the others.
Lock/Unlock Sizes	Lock or Unlock the sizes prior to computing. Be careful with this option since it will lock/unlock all pipes sizes, even the ones manually defined by the designer.
Lock/Unlock Elevations	Lock or Unlock the pipe invert elevations prior to computing. Be careful with this option since it will lock/unlock all pipe inverts, even the ones manually defined by the designer.

LAB EXERCISE: NETWORKS

> ### *Network Design*

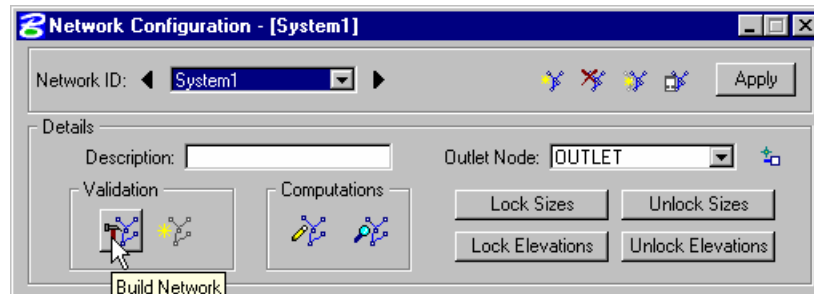1. Select the Add Drainage Network tool *(Drainage: Network > Add)*.

Add a New Network
Network ID: System 1
Description:
Outlet Node: OUTLET

OK   Cancel

2. Enter the network information.

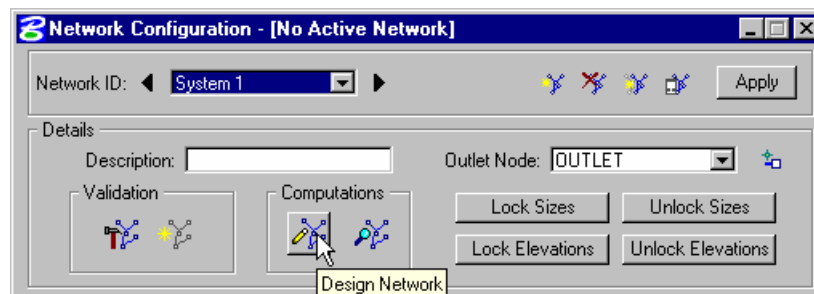| **Network ID:**	System1
Outlet Node	Outlet

3. Click **OK**.

4. In the Validation section, click **Build Network**.

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.

Network Configuration - [System1]
Network ID: System1   Apply
Details
Description: Outlet Node: OUTLET
Validation   Computations   Lock Sizes   Unlock Sizes
Lock Elevations   Unlock Elevations
Build Network

5. Click **Apply**. The Network SYSTEM1 has been added to the project.

6. In the Computations section, click **Design Network.**

GEOPAK Drainage designs the network and issues the pertinent Warning Messages, if necessary. These warning messages are also written in an ASCII file named: *drgmsg.txt* located in the GEOPAK's working directory.

Network Configuration - [No Active Network]
Network ID: System 1   Apply
Details
Description: Outlet Node: OUTLET
Validation   Computations   Lock Sizes   Unlock Sizes
Lock Elevations   Unlock Elevations
Design Network

7. Click **Set Active Network** (to the left of Apply).

8. Now we can review some of the provided reports of this network.

9. Select the Drainage Area Report tool *(Drainage: Reports > Drainage Area).* Review some of the available reports. This will be further discussed in the Reports Chapter

10. Close the dialog box when done.

DRAINAGE PROFILES: INTRODUCTION

A Profile is a path between two nodes, spanning one or more links. Profiles allow you 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.
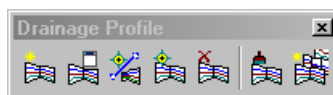
The Drainage Profiles tool box contains seven tools (from left to right). In addition, the tools can be invoked via the **Navigator.**

Tool	Description
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	Allows selecting a previously plotted profile from the screen and making it the active one.
Modify Drainage Profile	Utilized to select and edit any previously defined and stored Profile within the current project.
Delete Drainage Profile	Utilized to select and delete the specified Profile and associated data.
Update All Drainage Profiles	Recalculates and redraws all the profiles in the active gdf file.
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.
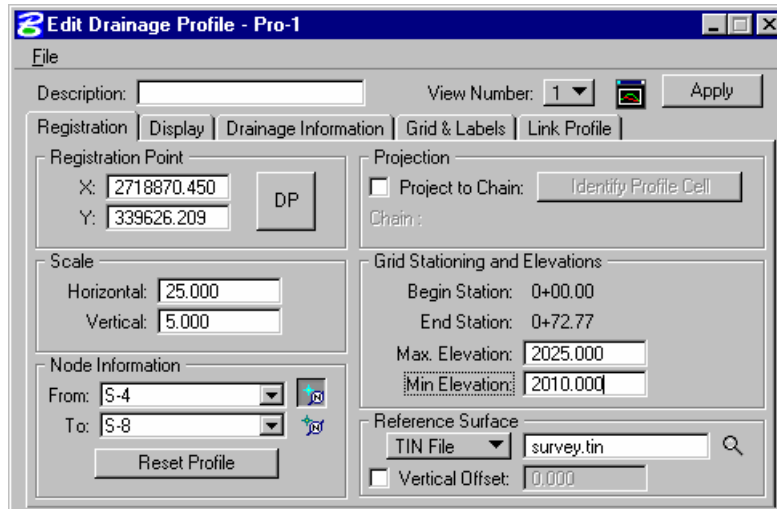
The dialog is comprised of the Description area and five tabs. The fields detailed in the table below are independent of which tab is selected.

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.

REGISTRATION

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. Be aware that profile plotting does not support rotated views. The vertical direction should always be due North.
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. Example: 10 Horizontal : 1 Vertical means that the vertical elements are distorted 10 times larger than the horizontal.
Node Information:	Node used as the starting point of the profile. This is populated using the From Node in the New Profile dialog. The From Node is on the left side of the profile and the To Node is on the right side.
Reset Profile	This will reset the profile display to the current setting of the dialog after the Project to Chain option is deactivated.
Project to Chain	If the Profile is to be projected onto a chain in order to utilize its stationing, the toggle is activated and chain parameters specified by selecting a profile cell previously plotted in the dgn file.
Grid Stationing and Elevations:	If Chain Projection is not used, the Stationing at the From Node is set to 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.

DISPLAY PROPERTIES

The Display Tab determines what information is drawn for each profile. Activate the toggles for the desired components of the profile drawing and set the desired element symbology. For the Water, Drainage, Sewer Line and Misc. Utilities Crossings, labels may be placed by activating the toggle to the left of label. A separate set of symbology is used for the textual label from the crossing elements.
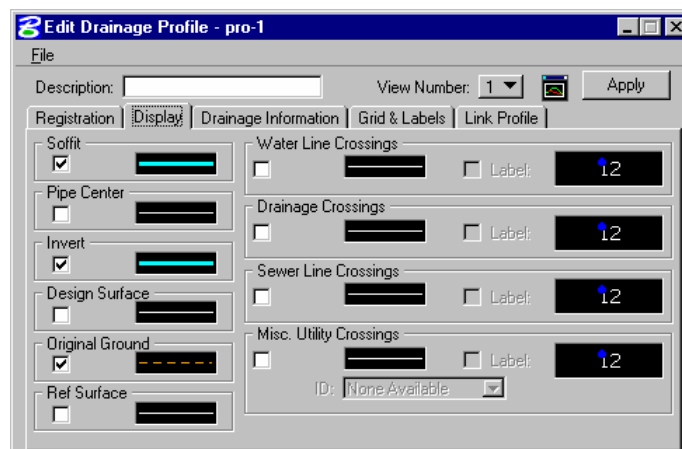
DRAINAGE INFORMATION

Other information from the drainage GDF file may be optionally drawn. This information is generated after the design or analysis of the network. Simply activate the toggle to the left of the desired drainage information, and set the symbology.
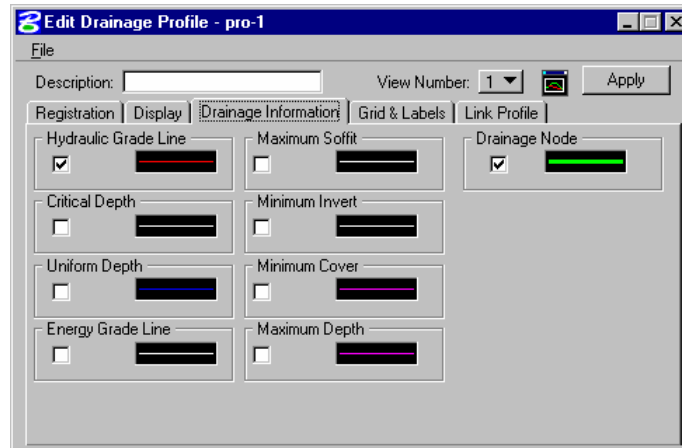
GRID AND LABELS

A variety of Grid and labels options are supported to enable the user to customize the profile drawings. Six Grid & Labels options are listed on the left side of the dialog. To activate an option, double click to fill the option box to the left. Double click again to turn off, which ghosts the right side of the dialog.
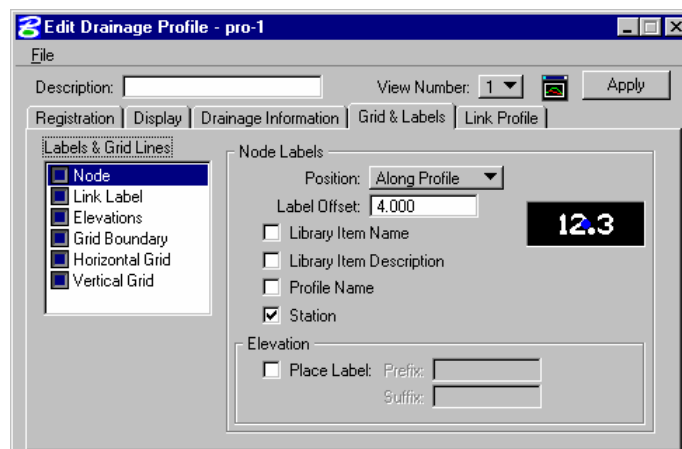
LINK PROFILE

In this tab, each node of the link profile can be reviewed and modified. Clicking on any Link ID in the list box populates the fields below. To modify, change the desired field(s), then click Modify (to the right of the list box.) To Hold an Invert, toggle on the button to the left of Hold Invert. The Slope can also be held by toggling on in the lower right corner. A pipe size could also be change selecting the Drainage Library Item option.

The user can also identify a Link ID, click ID then graphically identify one link of the profile. The associated line is highlighted in the list box.
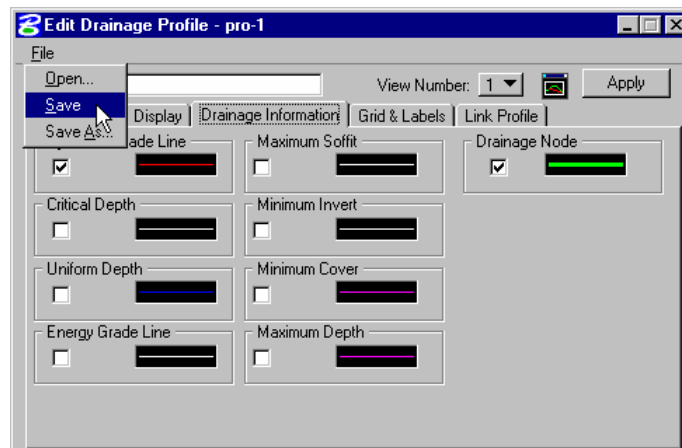
After performing the changes, click Apply to accept all modifications. Depending on your Update options under Project>Preferences, Geopak Drainage will redesign your network using the same design parameters and constraints specified under the Nodes and Links menus.
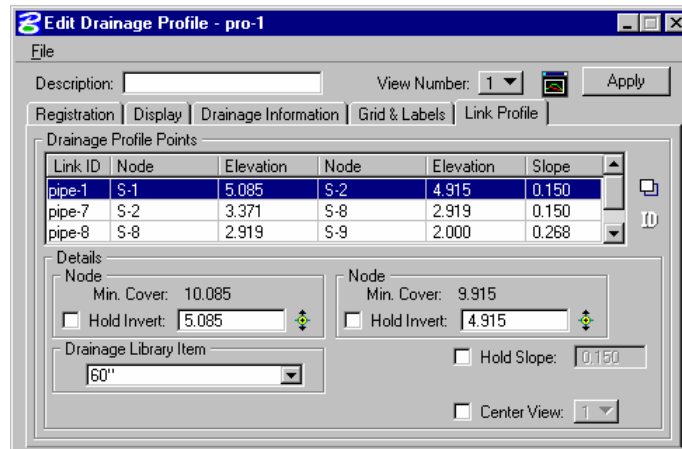
Note   Geopak Drainage cannot accommodate adverse pipe. If locked or held flow lines result in this condition, Geopak will adjust the flow lines to fix the problem, thus giving the appearance that the Lock or Hold Invert option does not work. A Warning message will inform the designer of the changes made to the pipe elevations after the design is performed.

PREFERENCES

The setup of the Edit Drainage Profile dialog box could be could be saved into a preferences file with default extension *.ppf*. This allows greater flexibility to the users since properties and display information of most profiles in the same project are very similar.

MISCELLANEOUS UTILITIES

Miscellaneous utilities can be stored within a drainage project and utilized within the utility conflict tool. Each group of utilities is stored as a group, then the group is referenced for conflicts. Therefore, it is prudent to group by location for subsequent location of conflicts, rather than grouping by utility type. For example, all utilities in an intersection, (i.e., gas, power, cable TV) can be placed within a single group, rather than all gas for an entire subdivision as a group.

This tool checks for conflicts in elevation but it does not address conflicts in the horizontal plane.

Miscellaneous utilities for a drainage project are defined thru Components>Miscellaneous Utilities>Add.


To properly define a utility location, the user needs to specify the horizontal and vertical location of the buried utilities in the Details section of the dialog box. The options available are:

Alignment	Element: specify the MicroStation symbology of the drawn utility. Chain: select the desired chain from the list of chain available in the gpk database
Elevation Source	Model: it can be selected from the list of all models in the current site project or GSF file. Object: it can be selected from the list of all models in the current site project or GSF file. Tin file: select the DTM or tin file of project.
Size	Defines the diameter of the utility in terms of master units, feet or meters.
Offset	Vertical distance (in master units) from the top of the utility to the specified model, TIN file, object or profile. Negative offsets indicate below the terrain, while positive offsets are above it.

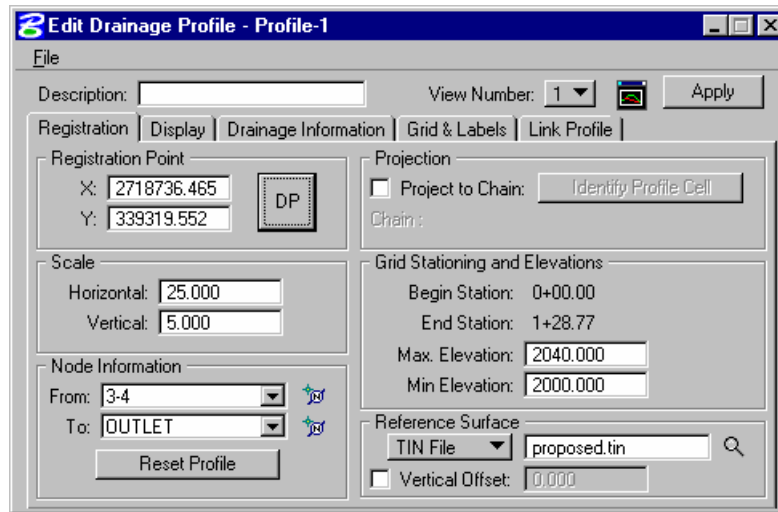## LAB EXERCISE: PROFILE DESIGN

> ### *Creating a Drainage Profile*

1. Select the Add Drainage Profile tool *(Drainage: Component > Profile > Add).*
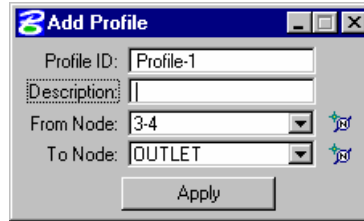
2. Enter the Add Profile information.

Profile ID:	Profile1
From Node:	Select 3-4
To Node:	Select Outlet

3. Click **Apply**.

4. Enter the Edit Drainage Profile information.

Min. Elevation:	2010.00
Max. Elevation:	2025.00
Horizontal Scale:	25
Vertical Scale:	5

5. Click **DP** and select a data point in the MicroStation design plane.

   This is the location where the profile is drawn.

6. Select **File > Open** and select *c:\data\geo\vdot\drain1\standards\Geopak\Drain_Profile.ppf.*

7. This loads the profile plotting preference file.

8. Click **Apply**. GEOPAK Drainage draws the profile.
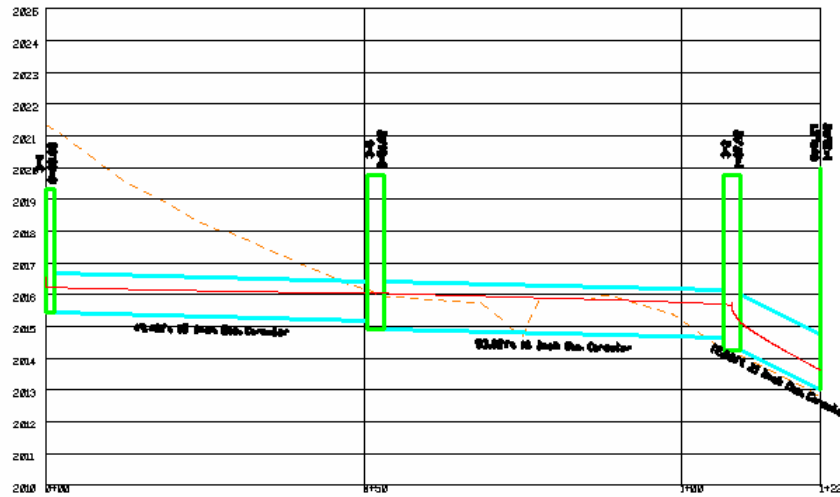
> **Dynamically Editing Profiles**

1. Select the **Link Profile** tab.

2. Click ID. Select and accept the link you want to modify from the profile view. (pipe-3). This will make pipe-3 the active link.

3. In the Details section, click **Edit Invert** in the left **Node** group.
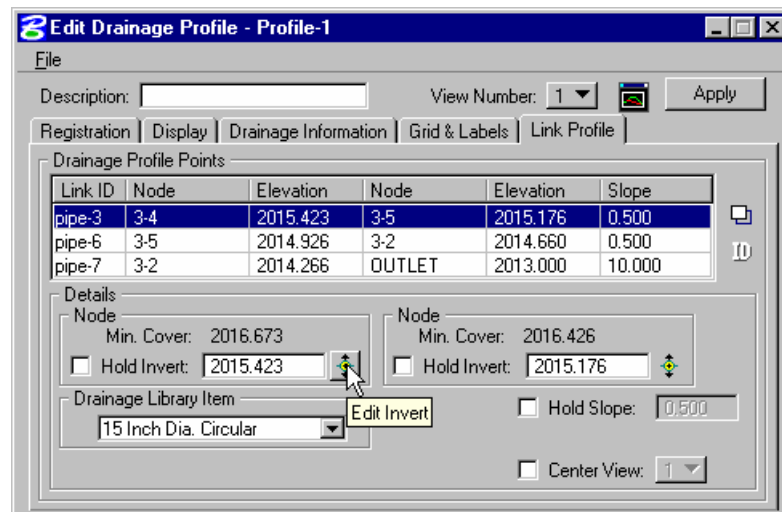
4. Move your cursor into the profile view. There is a circle around the downstream invert at node 3-4.

5. Move the cursor into the circle and drag the invert now. Link pipe-3 will dynamically follow your cursor.

6. Data point to define a new invert location in the profile.

7. Click **Apply.** For this example, the "Automatic Update Network" dialog was activated in the Preferences, so the network is automatically redesigned after clicking **Apply.**

8. Click **NO** to the alert dialog box if you do not want to display any warning or error message generated by the design process.

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. The inverts of the modified pipe get automatically locked but if there are other pipe sizes or inverts that you do not want to change, you must lock them in the Link Configuration Conditions dialog before changing an invert as shown above.

AUTOMATED PROFILE BUILDER

The Auto Create Profile tool draws a series of profiles based on the specified network.
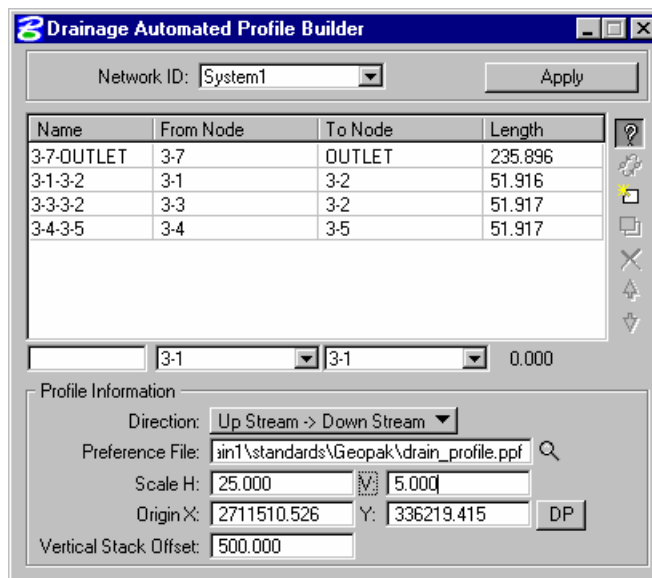
The setup of this dialog is similar to the individual profiles one. However, the automated process incorporates the need of specific action buttons.

?	**Query** - Click to populate the list box from the specified network.
	Reverse Nodes - Changes the From Node To Node and the To Node to the From Node.
	Add List Item - Enter the **Profile Name**, **From Node**, and To Node. Then click **Add**.
	Edit List Item - Highlight a line in the list box, make the desire changes. Then click Modify.
✕	**Delete List Item** - Highlight a line in the list box, then click Delete.
⬆	**Move Line Up** - Highlight a line, then click. Each click moves the line up one place in the list box.
⬇	**Move Line Down** - Highlight a line, then click. Each click moves the line down one place in the list box.

The profile information group controls the setup of the created profiles.

| Preference File	File created in the Edit Profile dialog, which contains the settings for drawing profiles.
Scale: H,V	Vertical and horizontal scale. The horizontal scale is drawn to true measurements, while the vertical scale is distorted.
Origin: X,Y	Coordinates of the lower left corner of the first profile.
Vertical Stack Offset	Distance between profile drawings.

LAB EXERCISE : AUTOMATED DRAINAGE PROFILES

> ### *Creating Automated Profiles for a Network*

1. Select the Auto-Create profiles tool. *(Drainage: Component > Profile > Auto_Create).*

2. Select **System1** as your Network ID.

3. Setup the Profile Information group in the Drainage Automated Profile Builder dialog as shown below.

   The Preference file is located in *\data\geo\vdot\drain1\standards\Geopak\Drain_Profile.ppf.*

   The Origin X and Y represent the coordinates in the MicroStation file where the profiles will start plotting. Clicking the **DP** button will populate the coordinates.
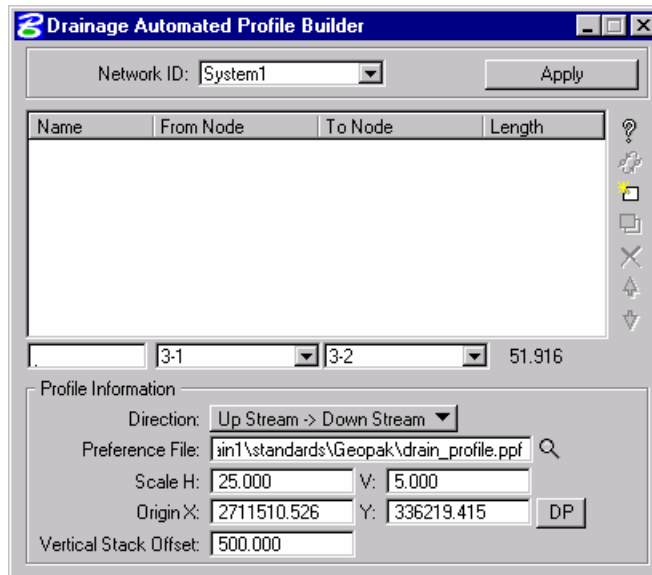
4. Select the **Query** icon. Geopak will create a series of possible profiles along the network.
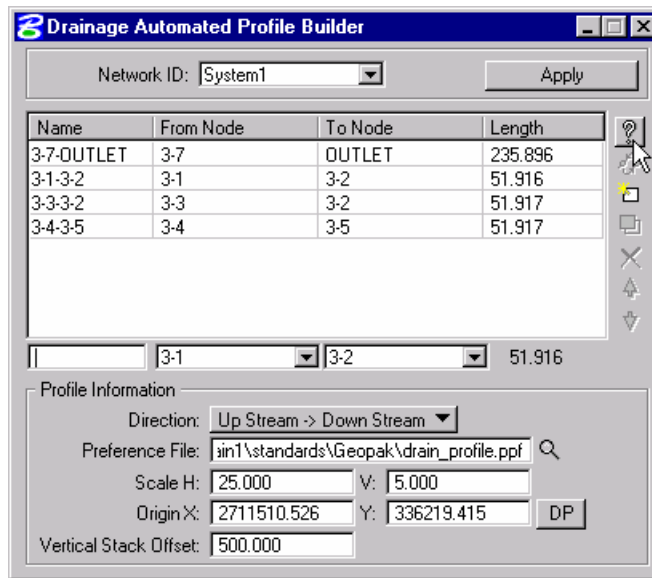
5. Click **Apply** to plot the profiles in the dgn file.

6. Close the dialog when done.

REPORTS: 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.
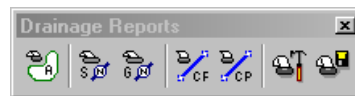

Tools from left to right are:

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.
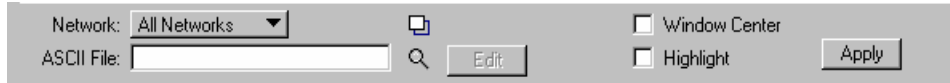
Network When initially activated, all components within the project are reported. The option is All Networks. By selecting Active Network, only those components in the active network are reported. Note in order to use this option, an Active Network must be defined prior to selecting the option. If an 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 EXERCISE: CREATE VDOT STORM CALCULATION SHEET
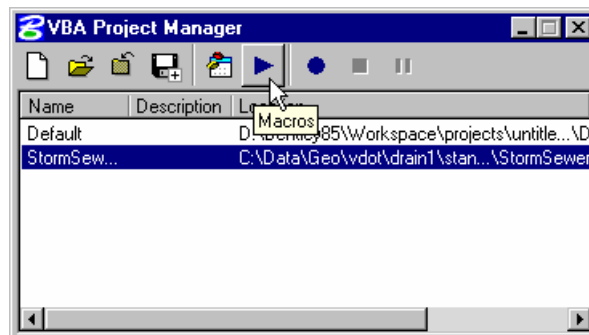
Saving the Drainage Project Preferences Report

1. Create the drainage preferences report for the project. *(Drainage: Project > Export>Preferences to ASCII.*

2. Save the file as : *drainage_preferences_17682.drp*.
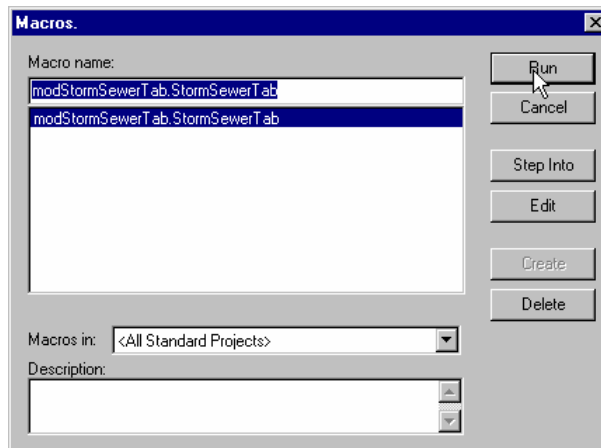
Creating the VDOT Storm Calculation Sheet

1. Activate the Macro tool in MicroStation. *(MS Menu: Utilities > Macro > Project Manager)*
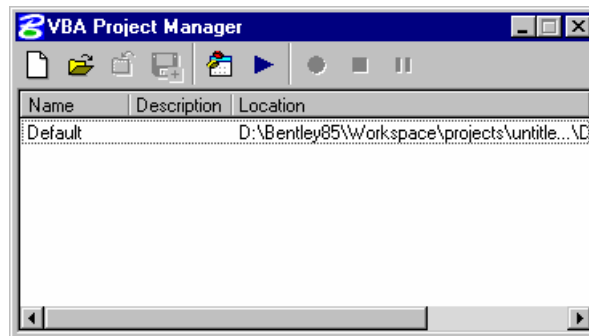
2. Load the project: **StormSewerTab.mvba** located in the *c:\data\geo\vdot\drain1\standards\Geopak* directory.

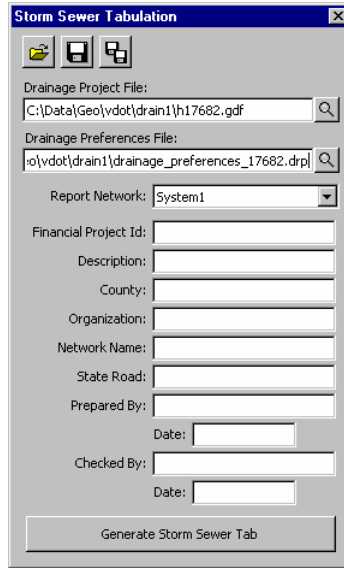3. Select the previous loaded macro and press the **Macros** button.

4. 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.

5. Set the drainage project , drainage preferences report file and network in the Storm Sewer Tabulation dialog.

Storm Sewer Tabulation

Drainage Project File:
C:\Data\Geo\vdot\drain1\h17682.gdf

Drainage Preferences File:
:o\vdot\drain1\drainage_preferences_17682.drp

Report Network: System1

Financial Project Id:

Description:

County:

Organization:

Network Name:

State Road:

Prepared By:

Date:

Checked By:

Date:

Generate Storm Sewer Tab

6. Press **Generate Storm Sewer Tab**

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

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

STORM SEWER TABULATION FORM

Road: **Mainline**
Project #: **8365**

Units: **X** Metric (SI) / US Customary

Sheet: _____ of
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)
CHAIN NAME							BASE / COMPOSITE									CROWN ELEVATION / 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	0.000	0.000	0.000	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				

9. Exit MicroStation.

This page left intentionally blank.

7 Service Utilities and Utility Conflicts

OBJECTIVES

- The objective of this chapter is to review the GEOPAK's functionality of locating possible utility conflicts in plan and profile view.

- Usage of the Conflict Finder tools and Miscellaneous Utilities.

INTRODUCTION

The **Conflict Finder** tool is an easy method of locating conflicts between components of a Drainage project and miscellaneous utilities. When a conflict is found, a textual label is displayed or drawn in the design file, based on user-defined parameters. Then, the designer has the option to fix these conflicts using the drainage profiling tools.
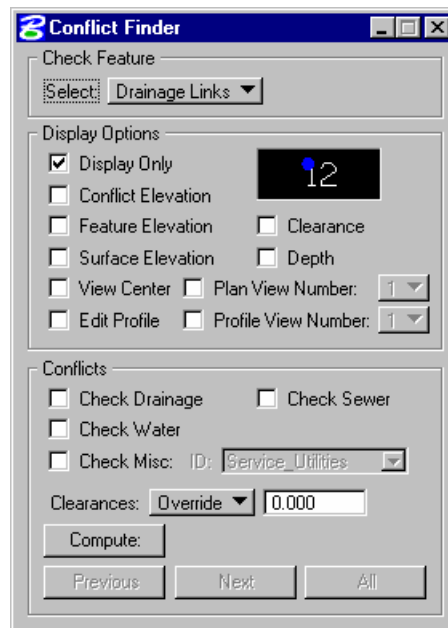
The Conflict Finder dialog is divided into three groups: Check Feature, Display Options and Conflicts.

CHECK FEATURE

Several types of conflict checking are supported: Single Feature, Water Lines, Sewer Lines, and Drainage Links. When Single Feature is utilized, the **ID Element** is available and used to graphically identify any current Water or Sewer line in the project. Once identified, GEOPAK checks the element against any other elements identified in the Conflicts group box.

DISPLAY OPTIONS

The Display Option group controls the way the possible conflicts will be displayed or drawn in the MicroStation dgn file. Several definitions are present:
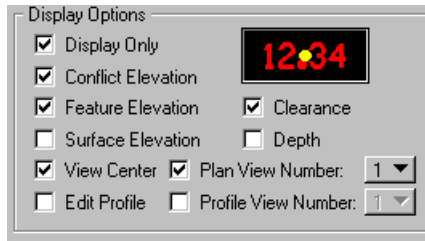
Display Only	The conflicts are labeled with display only, not written into the file. Therefore, they disappear when a View Control or Window command is issued. When not activated, the label is drawn into the design file.
Conflict Elevation	When the toggle is activated, the elevation of the conflicting feature is displayed. Note this corresponds to the feature defined in the Conflicts group box. When the toggle is not activated, no elevation is displayed.
Feature Elevation	When the toggle is activated, the Elevation of the feature is displayed. Note this corresponds to the feature(s) defined in the Check Feature group box. When the toggle is not activated, no elevation is displayed.
Surface Elevation	When the toggle is activated, the Elevation of the Design Surface at the X, Y location of the conflict is displayed. The Design Surface is defined in the Water and Sewer Preferences (Terrain). When the toggle is not activated, no elevation is displayed.
Clearance	When the toggle is activated, the elevation difference (in master units) between the Check Feature and the Conflict Feature is displayed. When the toggle is not activated, no clearance is displayed.
Depth	When the toggle is activated, the elevation difference (in master units) between the Surface Elevation and the Check Feature is displayed. When the toggle is not activated, no depth is displayed.
View Center	When activated, either the plan view or profile view may be centered in the specified view.
Plan View Number	When activated and a conflict is found and the View Center toggle is activated, the numbered view is utilized to window center about the plan view area of conflict.
Profile View Number	When activated and a conflict is found and the View Center toggle is activated, the numbered view is utilized to window center about the area of conflict in the profile drawing of the

	Check Feature.
Edit Profile	When activated and a conflict is found, and the Edit Profile dialog is activated, the numbered view is utilized to window center about the profile view area of conflict.

CONFLICTS

The Conflict group allows the designer to select what services or utilities to check. Possible conflicts could be checked against other drainage networks, water and sewer systems and user defined miscellaneous utilities.

Check Drainage	When the toggle is activated, all drainage links in the current Drainage project (gdf file) are checked for conflict with the Checked Feature(s). When the toggle is not activated, no conflicts are checked. The drainage project is specified in the Water and Sewer Preferences (Project Components).
Check Sewer	When the toggle is activated, all sewer links in the current Water and Sewer project are checked for conflict with the Checked Feature(s). When the toggle is not activated, no sewer conflicts are checked.
Check Water	When the toggle is activated, all water links in the current Water and Sewer project are checked for conflict with the Checked Feature(s). When the toggle is not activated, no water line conflicts are checked.
Check Misc	When the toggle is activated, all miscellaneous utilities in the specified group in the current Water and Sewer project are checked for conflict with the Checked Feature(s). The group is selected from the option list of all utility groups in the current project. When the toggle is not activated, no utility conflicts are checked.
Clearances	When set to Default, the clearance stored with each line is utilized. In this case, the key-in field is ghosted. However, this can be overridden by selecting Override and typing in the desired Clearances value, in master units.

When the dialog is populated with the desired settings, clicking **Compute** commences the process. If no conflicts are found, an Information message is displayed.

In addition, a note is displayed to the right of **Compute** in the Conflicts section.

If conflicts are found, the number is noted to the right of **Compute** in the Conflicts section

The **Prev** and **Next** can be utilized to move between conflicts, if more than one conflict has been found.

MISCELLANEOUS UTILITIES

Miscellaneous utilities can be stored within a drainage project and utilized within the utility conflict tool. Each group of utilities is stored as a group, then the group is referenced for conflicts. Therefore, it is prudent to group by location for subsequent location of conflicts, rather than grouping by utility type. For example, all utilities in an intersection, (i.e., gas, power, cable TV) can be placed within a single group, rather than all gas for an entire subdivision as a group.

This tool checks for conflicts in elevation but it does not address conflicts in the horizontal plane.

Miscellaneous utilities for a drainage project are defined thru Component>Miscellaneous Utilities>Add.

To properly define a utility location, the user needs to specify the horizontal and vertical location of the buried utilities in the Details section of the dialog box. The options available are:

Alignment	Element: specify the MicroStation symbology of the drawn utility. Chain: select the desired chain from the list of chain available in the gpk database
Elevation Source	Model: it can be selected from the list of all models in the current site project or GSF file. Object: it can be selected from the list of all models in the current site project or GSF file. Tin file: select the DTM or tin file of project.
Size	Defines the diameter of the utility in terms of master units, feet or meters.

Offset	Vertical distance (in master units) from the top of the utility to the specified model, TIN file, object or profile. Negative offsets indicate below the terrain, while positive offsets are above it.

To Edit any of the previously defined utilities, simply select the one to be edited. Once selected, the dialog is automatically populated with the data associated with the selected utility. Edit the data as desired, then click **Apply** to complete the editing. The only data which cannot be edited is the ID. To change the ID, delete the utility and use the Add Miscellaneous Utility tool with the new ID and associated information.

LAB EXERCISE: CHECKING FOR UTILITY CONFLICTS

> *Opening the Drainage Project*

1. Execute *c:\data\geo\vdot\drain1\LAB07_V8ad.EXE*

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

3. Open Project Manager (*MS Menu: Applications > GEOPAK Road > Project Manager)*

4. Select project: *17682.prj* located in *c:\data\geo\VDOT\drain1*, and click OK.
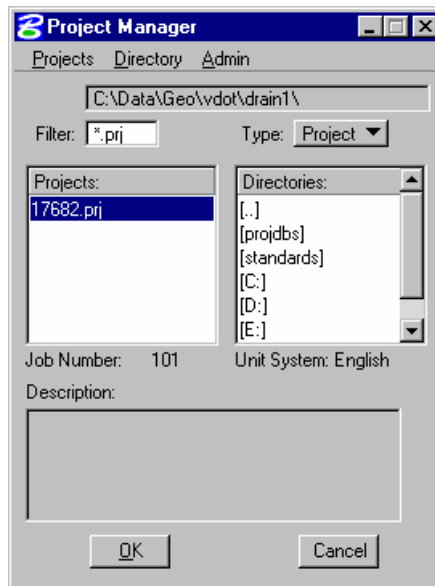
5. Access User: VDOT.
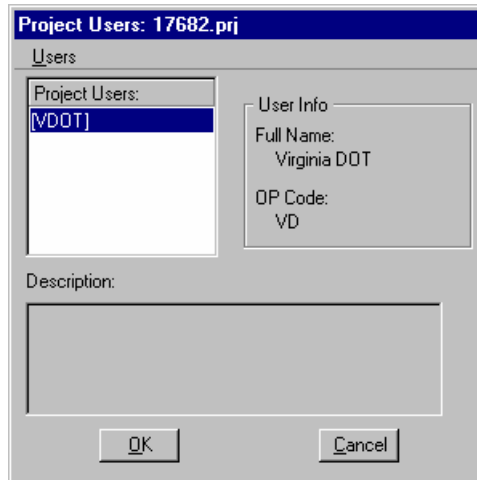
6. Click **OK.** The Road Project dialog will load. This will setup the proper directories for the location of the gpk file.
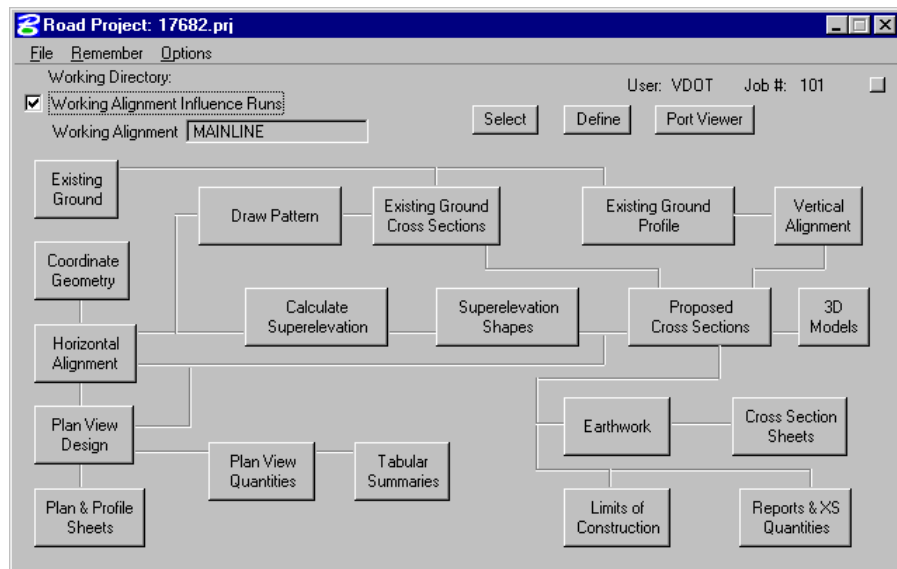
7. Minimize the Road Project dialog box.

8. Access GEOPAK Drainage *(MS Menu: Applications > GEOPAK Drainage > Drainage).*

9. Open the drainage project, *c:\data\geo\vdot\drain1\h17682.gdf (Drainage Menu: Project > Open).*

> *Adding Miscellaneous Utilities*

1. Select the Miscellaneous Utilities tool *(Drainage: Component>Miscellaneous Utilities>Add).*

2. Enter ID: Service_utilities. Click **OK** when finished.

3. Enter the electric line information.

Alignment	**Element**. Level=Level24 , Co=Yellow (4), Style=0, Weight=7
Elevation Source	**TIN File**. *C:\data\geo\vdot\drain1\survey.tin*
Size	2.0
Offset	-3.0

4. Click **Add List Item** (right side of the dialog).

5. Enter the water main data.

Alignment	**Elemen**t. Level=Level25 , Co=Purple(5) , Style=0, Weight=7
Elevation Source	**TIN File**. *C:\data\geo\vdot\drain1\survey.tin*
Size	2.5
Offset	-6.0

6. Click **Add List Item** (right side of the dialog).

7. Enter the telephone lines information.

Alignment	**Element**. Level=Level27 , Co=Brown(6) , Style=0, Weight=7
Elevation Source	**TIN File**. *C:\data\geo\vdot\drain1\survey.tin*
Size	0.5
Offset	-1.5

8. Click **Apply** when done entering all the utilities in the project.

9. In the Miscellaneous Utility dialog, select **File>Save As.** Save the file: **Service_utilities.utl** under *c:\ data\geo\vdot\drain1*
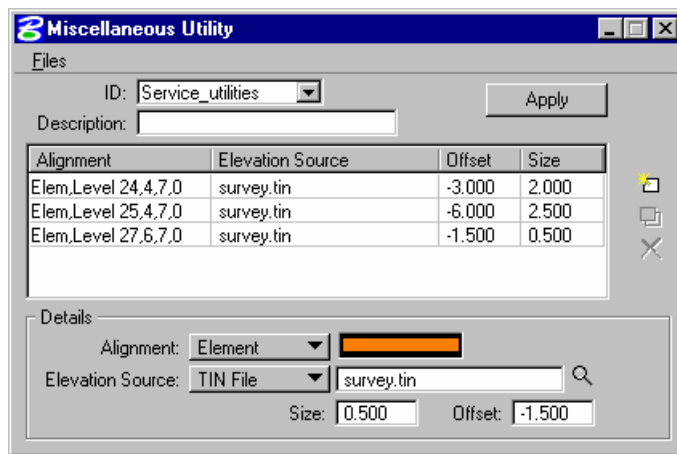
10. Close the Miscellaneous Utility dialog.

> **> *Checking for Utility Conflicts in Plan View***

1. Select the Conflict Finder tool. *(Drainage: Utilities>Conflict Finder).*

2. Populate the Conflict Finder dialog as shown below.
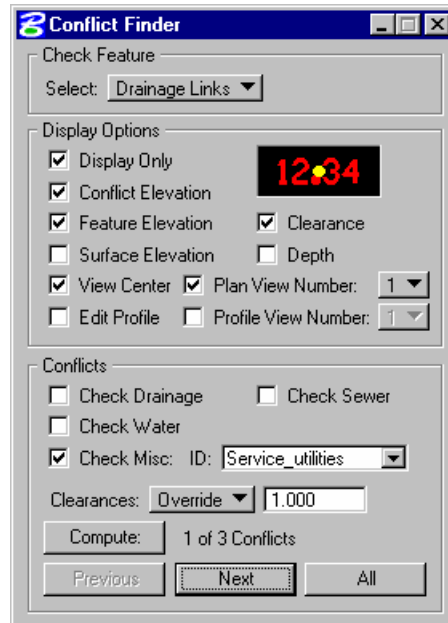
3. Double click inside the text display box to setup the plotting parameters.
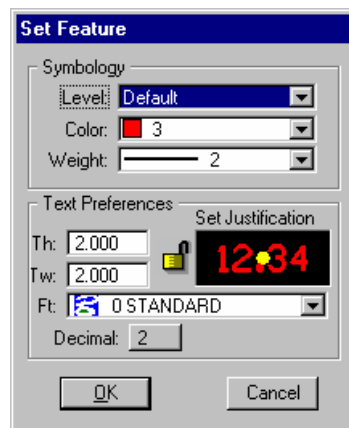
4. Click **OK** when done.

5. Click **Compute** in the Conflict Finder dialog to initiate the process.

6. Using the **Next** and **Previous** buttons verify the utility conflicts.
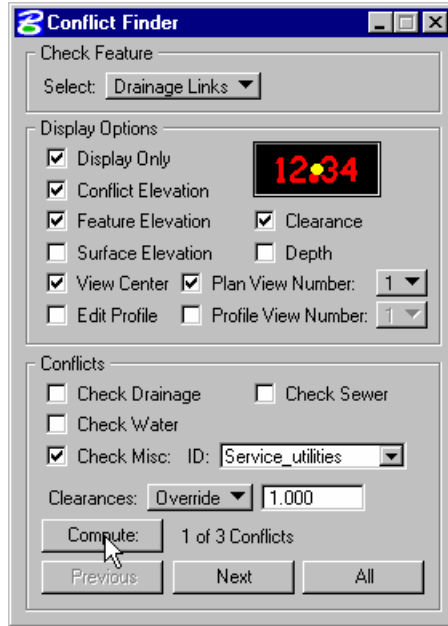
7. Close the dialog when done.

➢ *Checking for Utility Conflicts in Profile View*

1. Edit the previously created drainage profile. *(Drainage:Component>Profile>Edit List).*
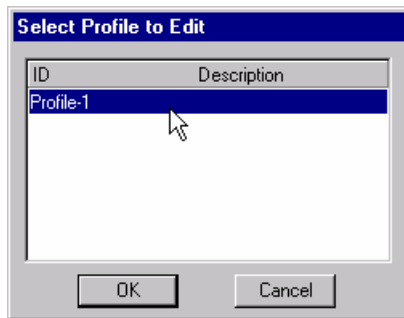
2. Center the profile in the MicroStation view by clicking the Center Profile icon.
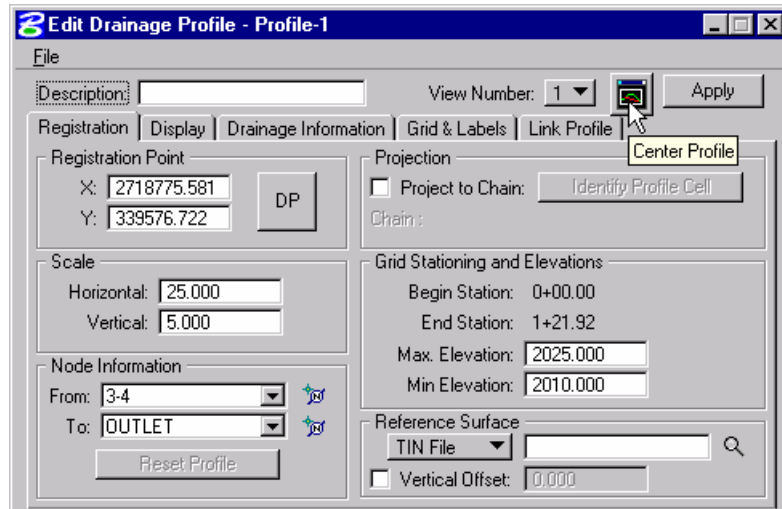
3. Select the Display Tab

4. Toggle ON the Misc.Utility Crossings group.
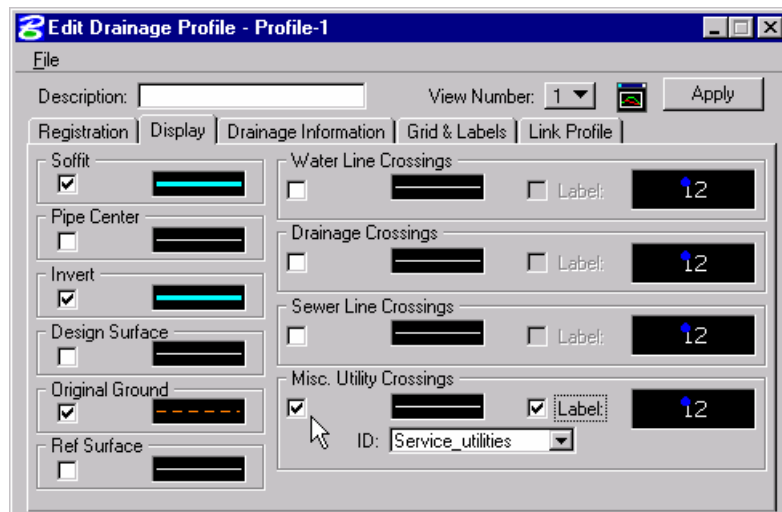
5. Click **Apply** to update the profile and show the utility crossings.
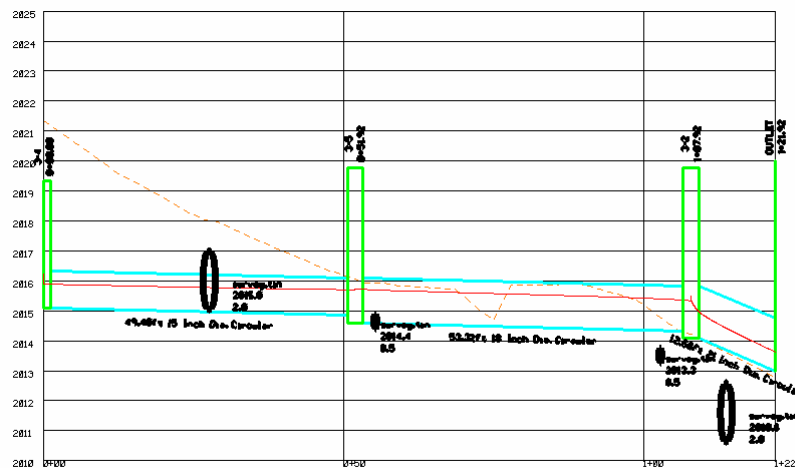
6. Select the Link Profile tab.

7. Modify the pipe inverts to clear the utility conflicts.
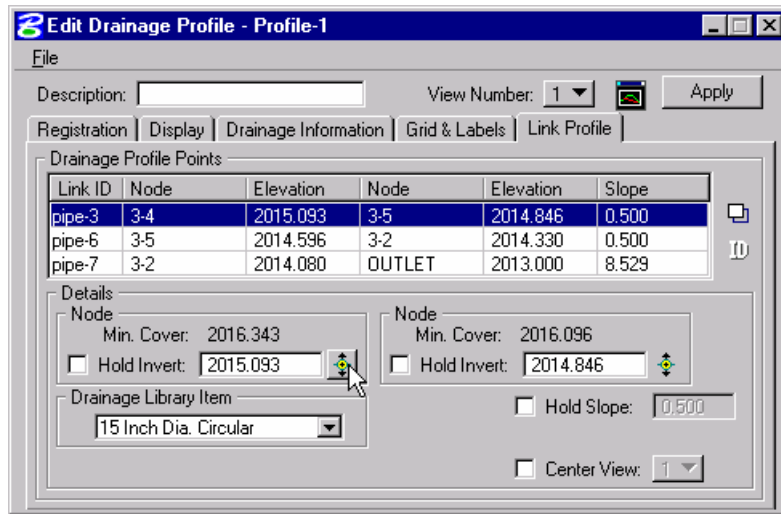
8. When done, click **Apply** to redesign the network with the new pipe inverts and/or sizes.

7. Review the new design and exit MicroStation when done.

This page left blank intentionally.

8   **Plans Preparation**

OBJECTIVES

- Review plans preparation procedures under GEOPAK Drainage.

- Renumber Nodes along the network system.

- Plan View Labeling of drainage structures and pipes.

- Plotting flow directional arrows along the pipes.

- Plotting a sketch of the drainage network.

- Compute Automated Quantities

- Create Summary Tables.

PLAN VIEW LABELING

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 you select 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.

RENUMBER NODES

GEOPAK Drainage provides an automatic tool to renumber all the nodes in the gdf. *(Drainage: Component > Nodes > Renumber).*

The renumbering tool will update not only the selected nodes, but all the pipes connected to them, drainage areas and profiles. Conflict resolution procedures are also available to the user prior the renumbering process.
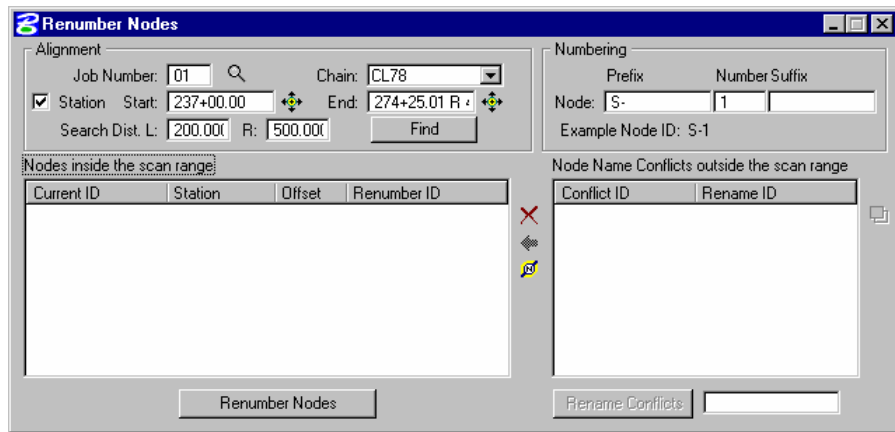
The Renumber Node dialog box is divided in four groups:

Alignment	Possible nodes for renumbering will be scanned along a specified chain and station range. The scan range is also limited by a search distance along the left and right side of the alignment. The **Find** button will initiate the scanning process.
Numbering	The new nodes will be renumbered or renamed according to the new prefix, number and suffix specified
Nodes Inside the Scan Range	The list box will show the encountered nodes along the alignment after the scanning process is finished.
Node Name Conflicts outside the scan range	This list box will be populated only if the new numbering of the nodes has a name conflict with an existing node in the system that was not selected by the FIND process. The designer will have to resolve this on an individual basis, renaming the conflict node and place it into the Nodes inside the Scan Range list box.

LAB EXERCISE: RENUMBERING NODES

➢ *Opening the Drainage Project*

1. Execute *c:\data\geo\vdot\drain1\LAB08_V8ad.EXE*

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

3. Access GEOPAK Drainage *(MS Menu: Applications > Drainage > Drainage)*.

4. Open the drainage project, *c:\data\geo\VDOT\drain1\h17682.gdf (Drainage: Project > Open)*

➢ *Renumbering the Nodes*

> *Warning* It is recommended that before renumbering, the designer should create a copy of the dgn and gdf file for the drainage project as a backup.

1. Select the Renumber tool *(Drainage:Component>Node>Renumber)*.

2. Fill out the Renumber Nodes dialog as shown below. We will renumber all the structures following the direction of the alignment Mainline, starting with structure 3-1.
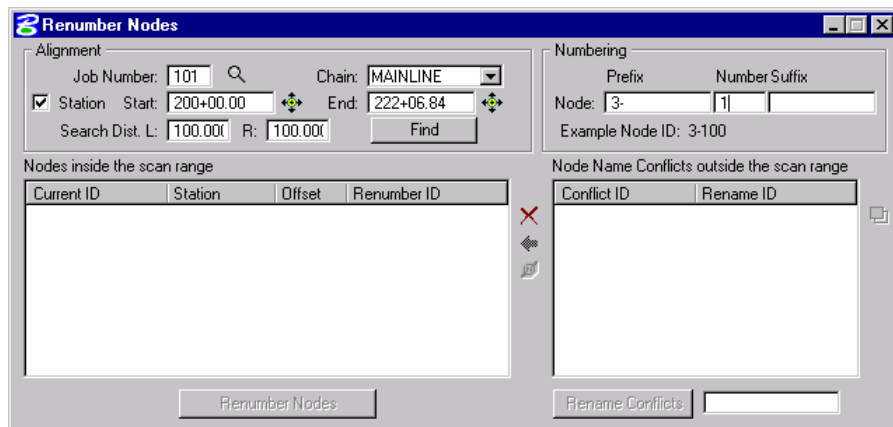
3. Click **Find**.
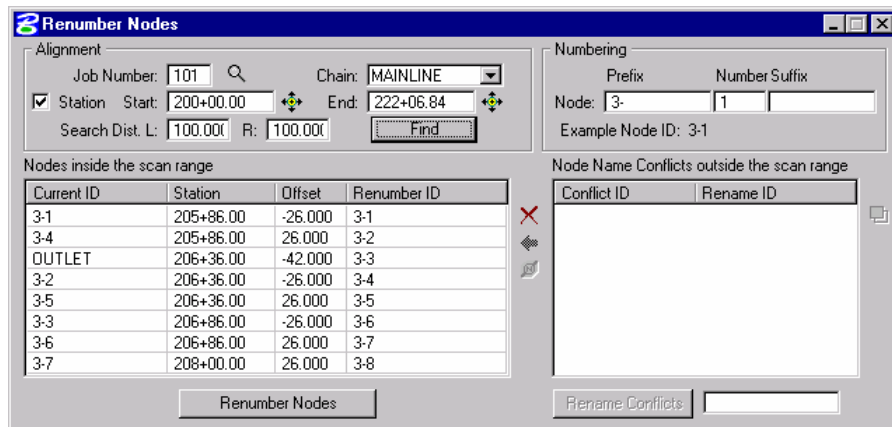
4. Click **Renumber Nodes**. GEOPAK will renumber all the drainage structures, update the links connections, and drainage areas, and accordingly to our previously setup project preferences it will update the calculations and profiles.

LAB EXERCISE: SKETCH OF THE GEOPAK DRAINAGE NETWORK

Creating a Sketch of the Network

1. Open VBA Project Manager dialog. *(MS Menu: Utilities > Macro > Project Manager).*

2. Load the macro: **DrainageNetworkSketch.mvba** located in the *c:\data\geo\VDOT\drain1\standards\geopak* directory.
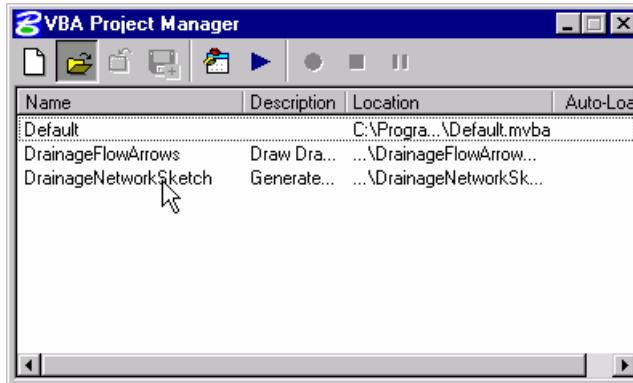
3. Select the previously loaded macro and press the **Macros** button.
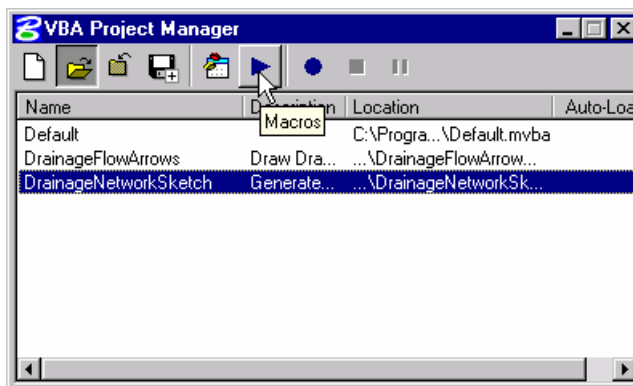
4. Click the **Run** button.
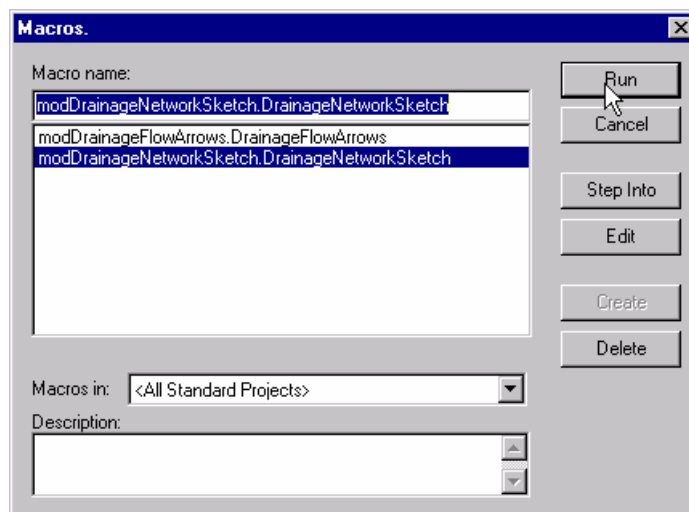
5. Set the drainage project and drainage network in the Drainage Network Sketch dialog.
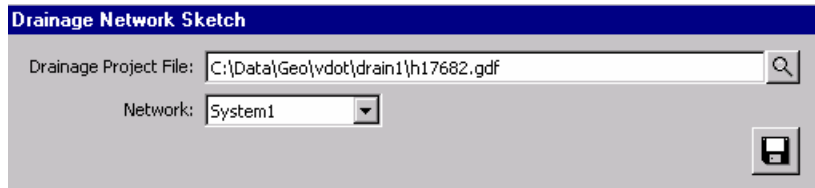
6. Upon selection, the sketch will be displayed in the dialog box.
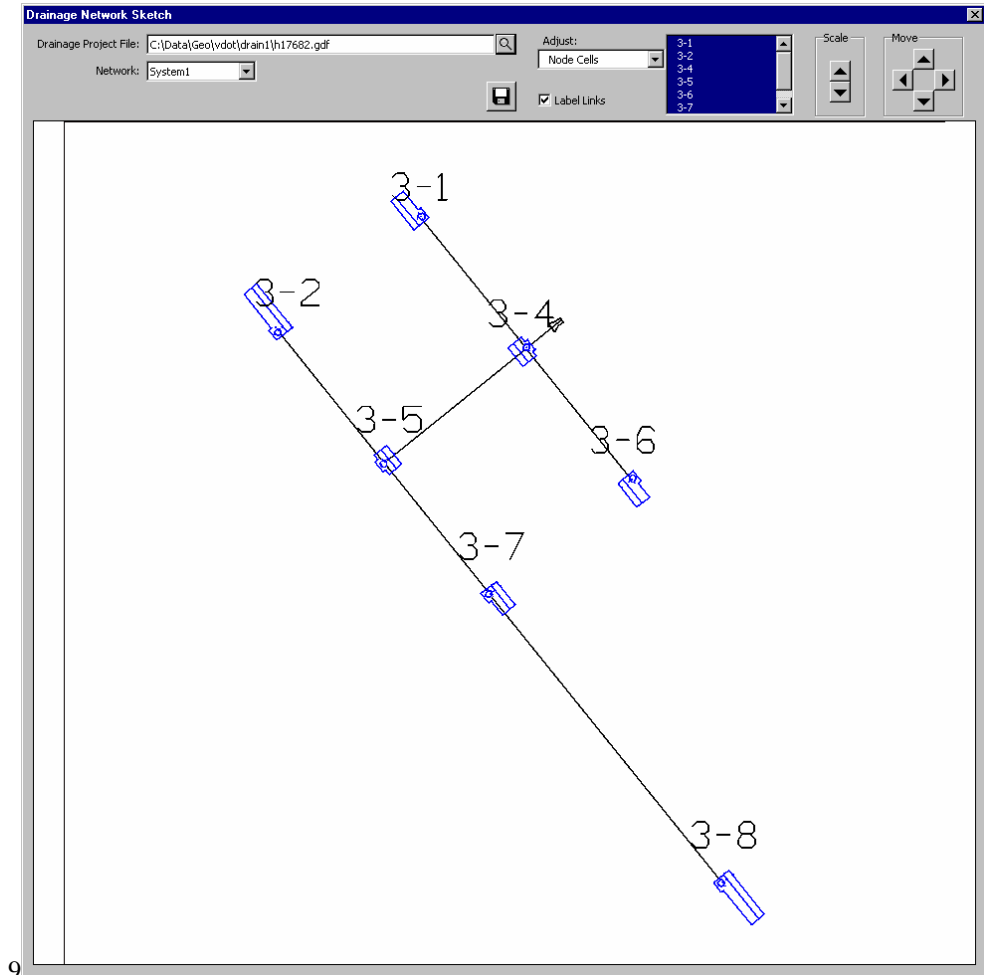
7. Using the Scale and Move buttons adjust the size of the node labels, node cells and link labels.
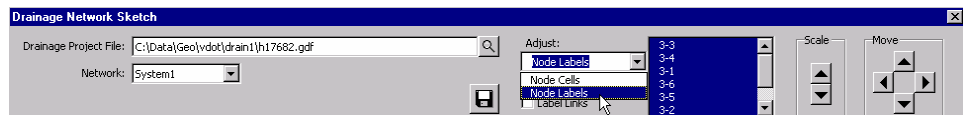
8. Click on the **Save** icon to store the network sketch as a JPG image file.
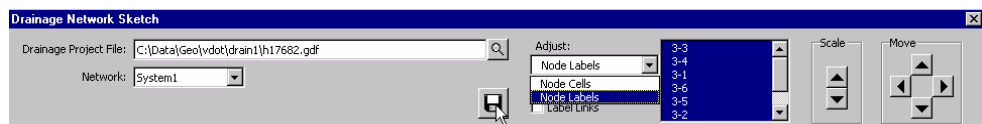
9. Close the dialog when done.

LAB EXERCISE: UPDATING THE DESIGN TO VDOT CADD STANDARDS

> ### *Updating the Drainage Design to VDOT CADD Symbology.*

1. Open the Drainage Preferences dialog. *(Drainage: Project > Preferences).*

2. Select the Plan Symbology item.

3. Toggle OFF the Labels option.

4. Disable the **Set Node Cell Symbology**.
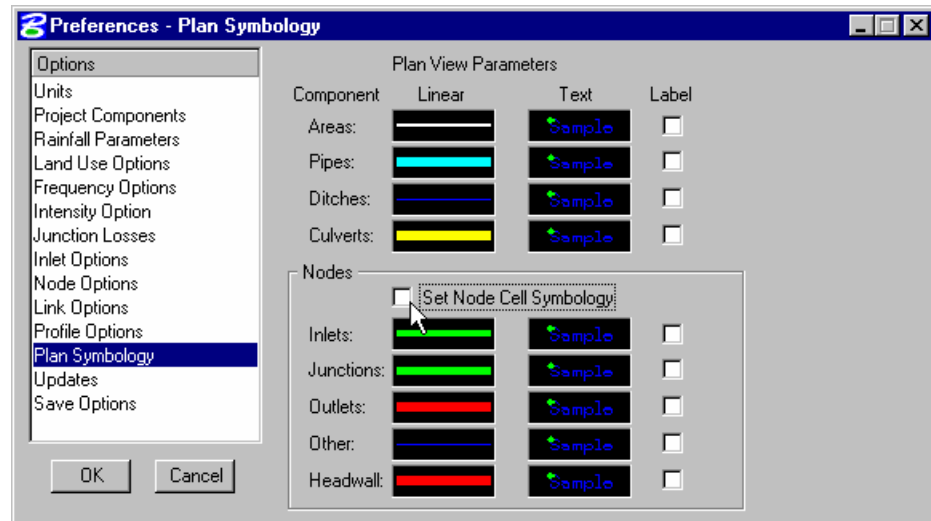
5. Click **OK** when done.

6. Open the Design & Computation Manager. *(Applications > GEOPAK Road > Design & Computation Manager)*

7. In the Design & Computation Manager, open the **VDOTenglish.ddb** database located in the *c:\data\geo\VDOT\drain1\standards\geopak* directory. *(Design and Computation Manager : File > Open)*
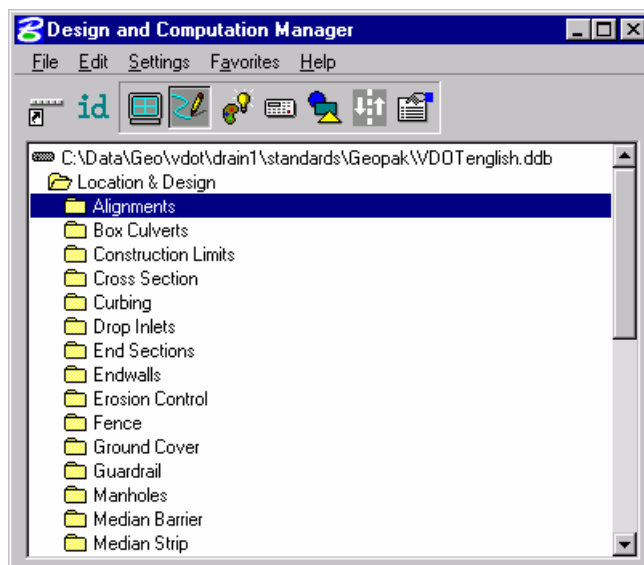
8. Select **Settings > Design** in order to change the updating properties for the drainage structure cells.
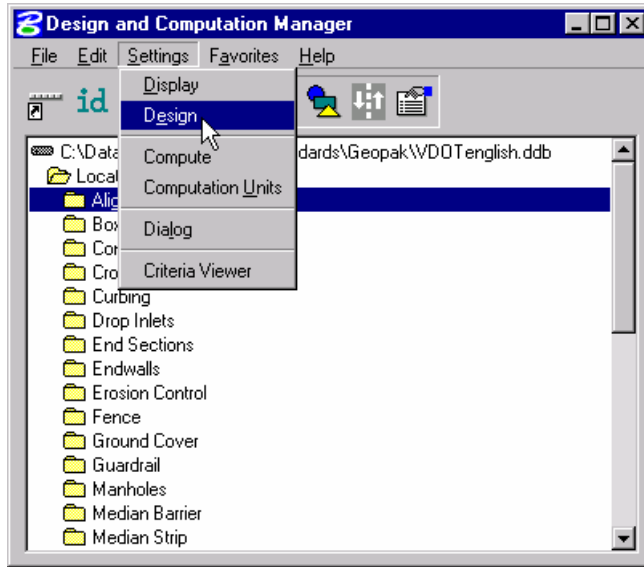
9. Toggle **ON** the Influence Graphic Cell Symbology option. Then, close the dialog box.
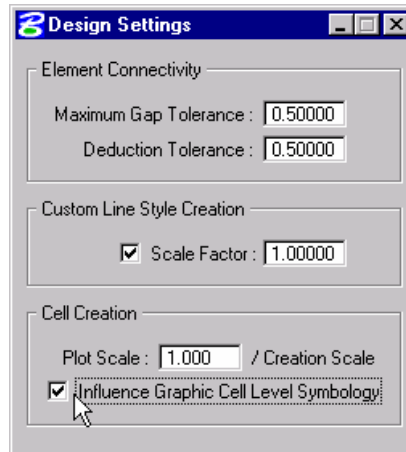
10. Execute the Area Update All option. *(Drainage: Component> Area > Update All).*GEOPAK Drainage will clear the previously place drainage area labels.

11. Execute the Nodes Update with PayItems option. *(Drainage: Component> Nodes > Update With Pay Items).* GEOPAK Drainage will update the drainage structures level symbology according to VDOT CADD Standards.

12. Execute the Links Update with PayItems option. *(Drainage: Component> Link > Update With Pay Items).* GEOPAK Drainage will update the drainage structures level symbology according to VDOT CADD Standards.

LAB EXERCISE: PLAN VIEW LABELING

> ### *Labeling Drainage Structures*

1. Open the Drainage Labeler. *(Drainage: Utilities > Labeler).*

2. Select the **Text Tab** and set the Component to **Node**.
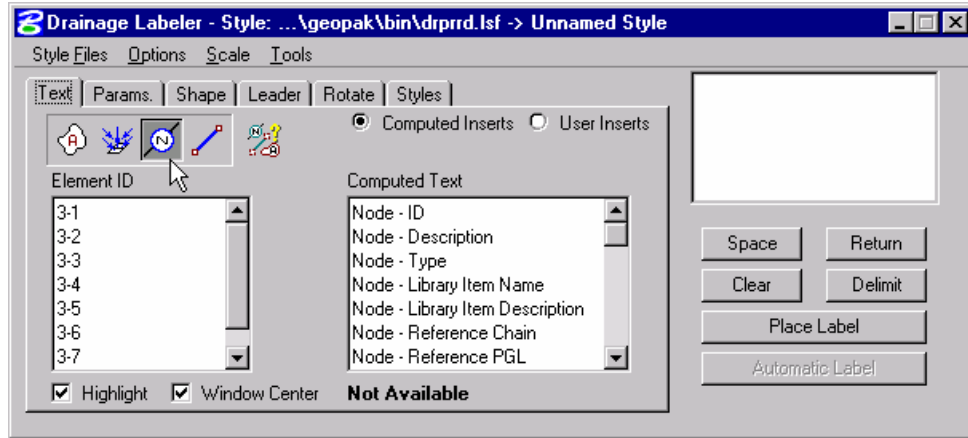
3. Select the first Element ID: **3-1**.

4. Double-click the Computed Text: **Node-ID** to add it to the Editor box.

5. Click on the **Parameters Tab** and set the Text Preferences/Symbology as shown below.
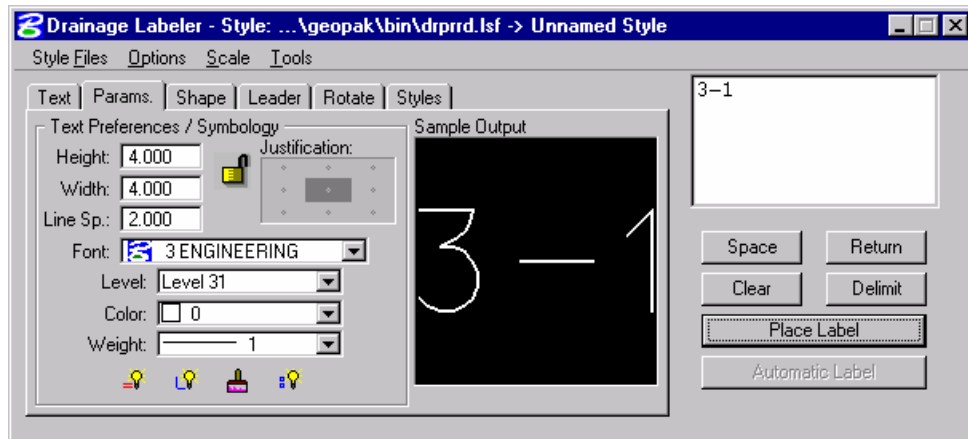
6. Move to the **Shape Tab**, and set the Shape Preferences and Symbology as shown below:
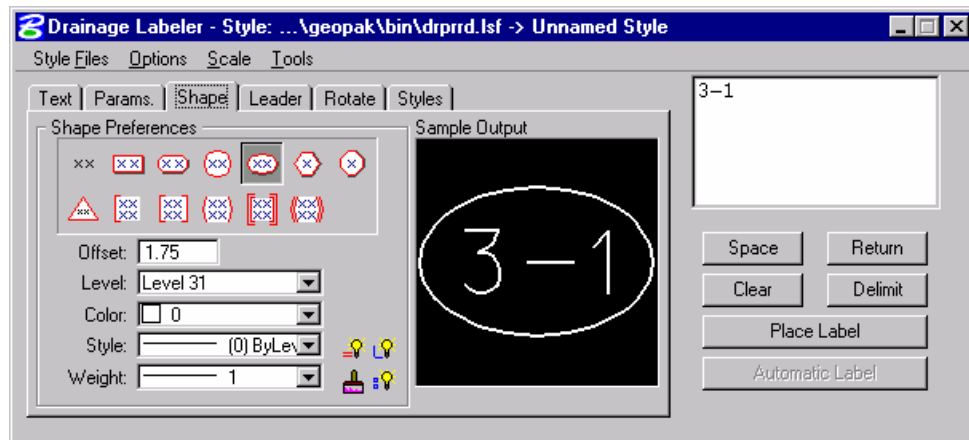
7. Move to the **Leader Tab** and set the Leader Type as shown below.
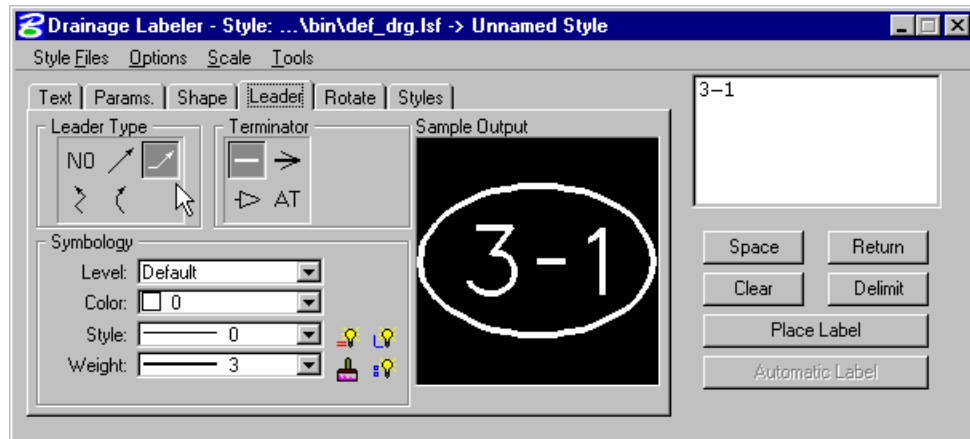
8. Move to the **Rotate Tab** and select the Current Angle option as shown below.
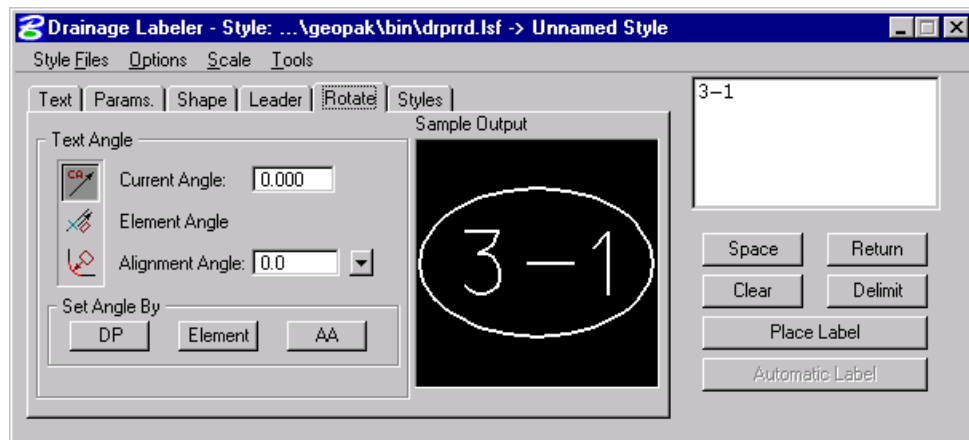
9. Place the label in the design file by clicking **Place Label**.
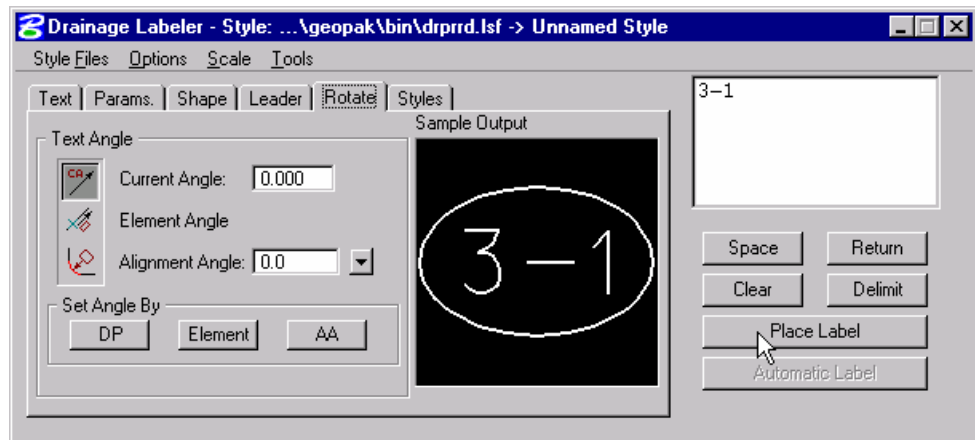
10. Create a new Style by moving to the **Styles Tab** and clicking on **New Style**. Provide a style name for the label, and press **OK**.
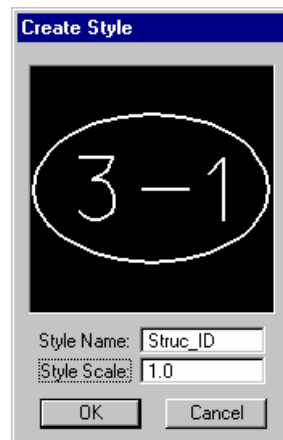
11. Save the Style by selecting *Style Files > Save*.

 VDOT GEOPAK Drainage
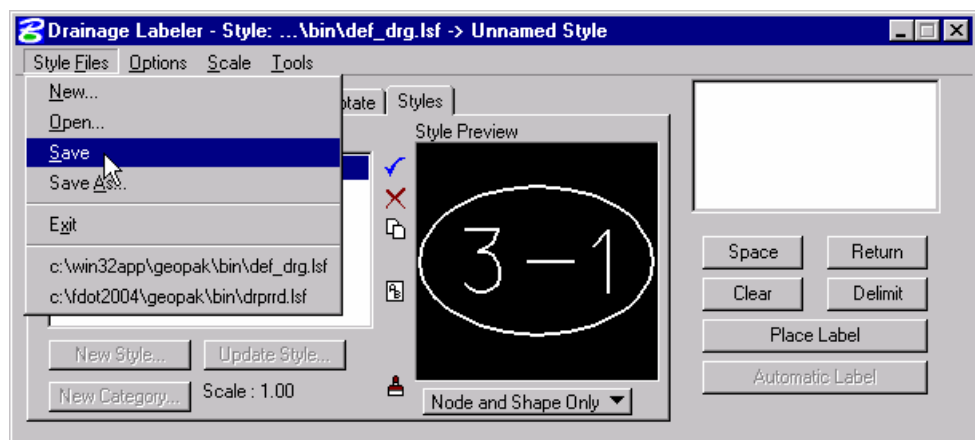
12. Move to the **Text Tab**, and select the second structure: **3-2**. By activating the **Highlight** and **Window Center** toggles, the Drainage Labeler will navigate to the correspondent drainage structure location in the dgn file.
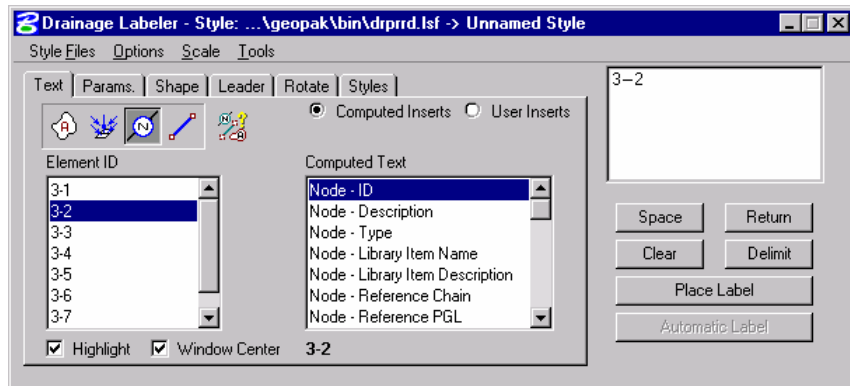
13. Label the remaining drainage structures.

> *Labeling Pipes*

1. After labeling the drainage structures, click on the **Clear** button in order to empty the Editor box.
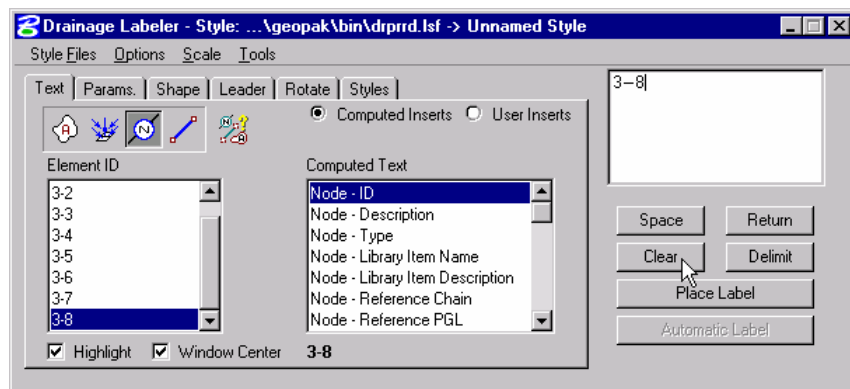
2. Select the **Text Tab** and set the Component to **Links**.
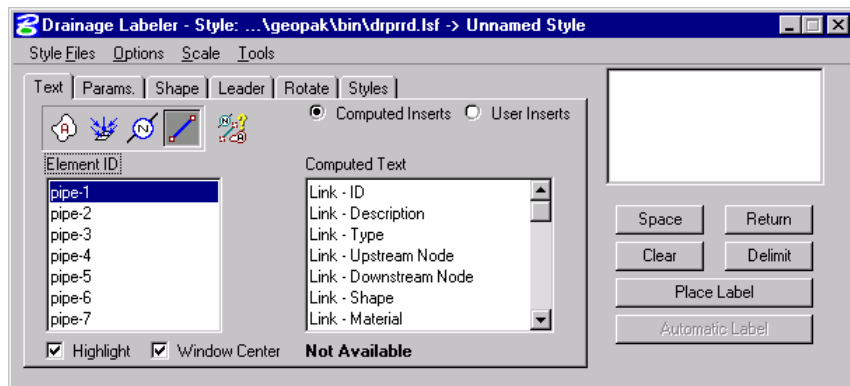
3. Select the first Element ID: **pipe-1**.

4. Compose the label by double-clicking into the Computed Text: **Link-Library Description**.
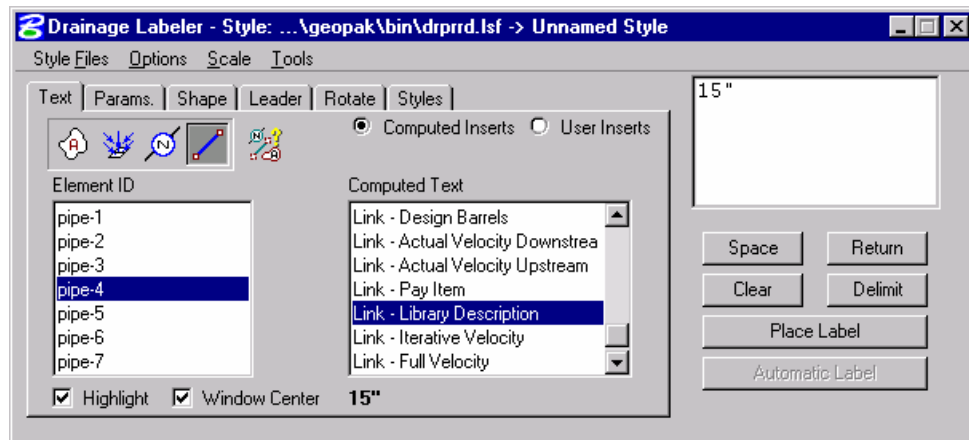
5. Click on the **Parameters Tab** and set the Text Preferences/Symbology as shown below.
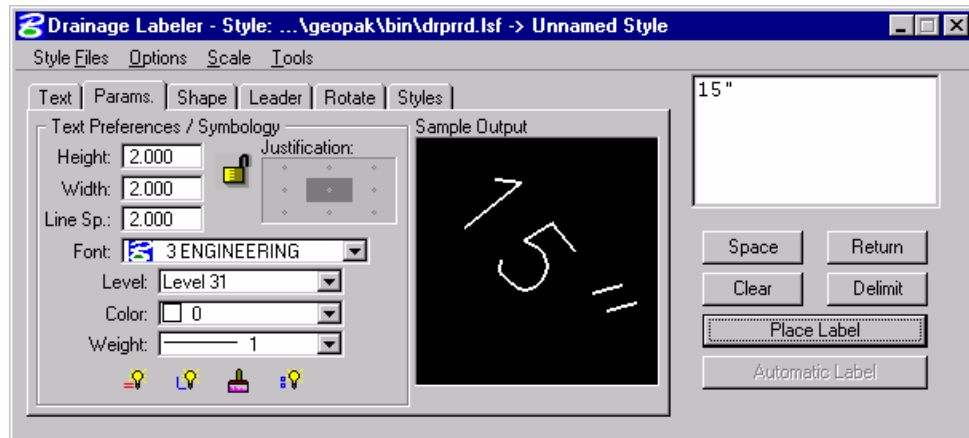
6. Move to the **Shape Tab**, and set the Shape Preferences and Symbology as shown below:
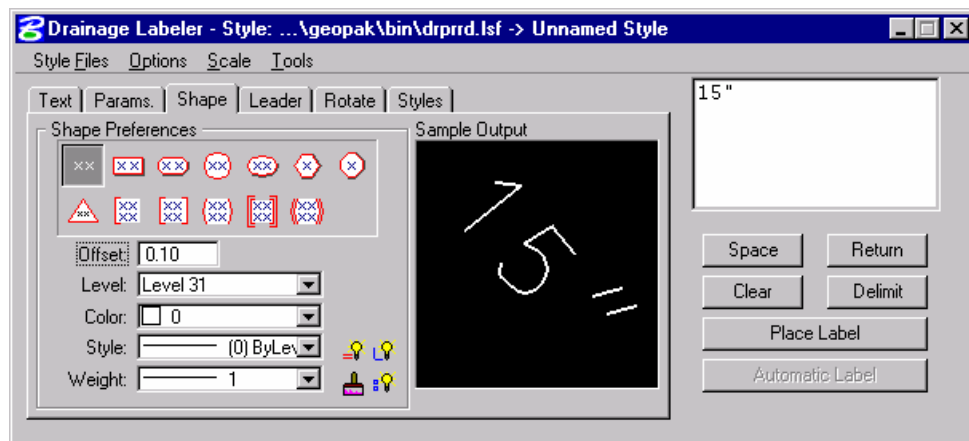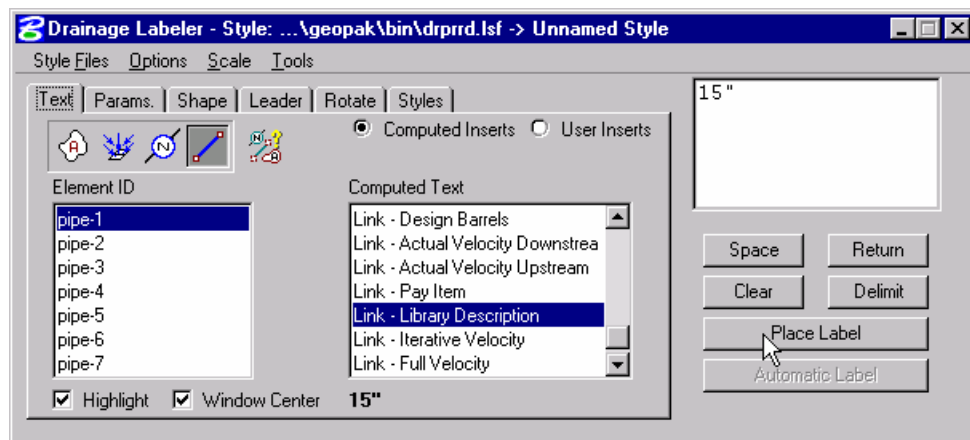
7. Move to the **Leader Tab** and set the Leader Type, Terminator and Symbology as shown below:
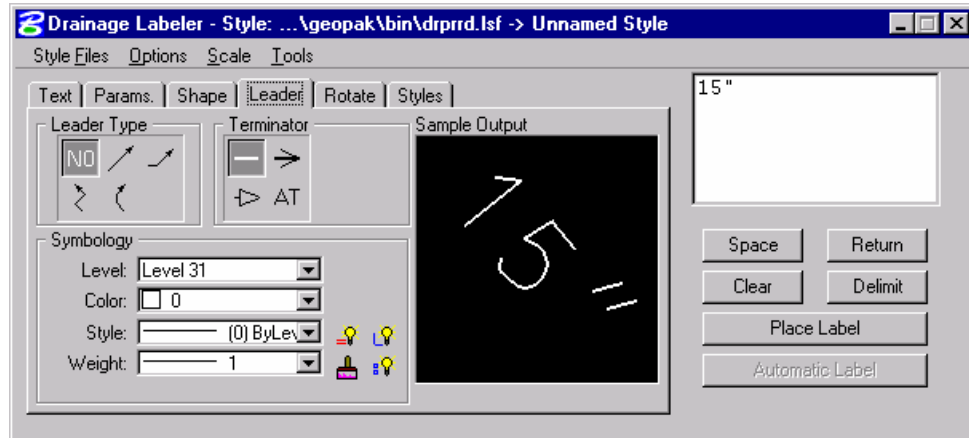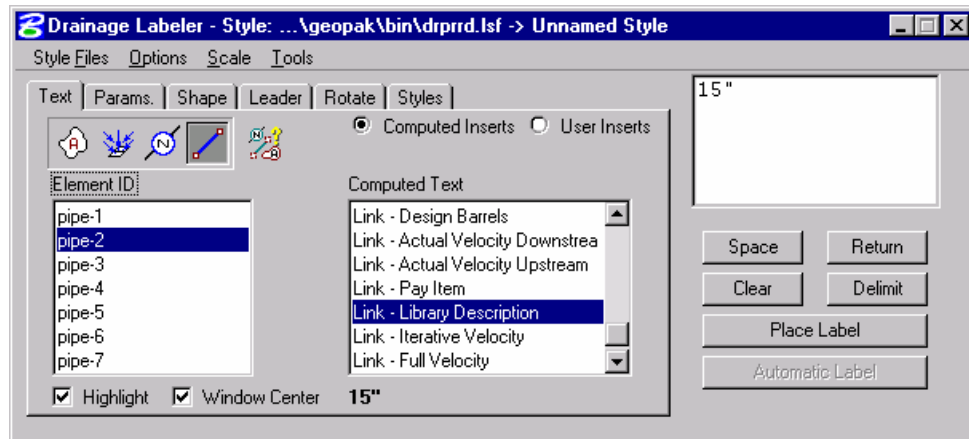

8. Move to the **Rotate Tab** and select the Current Angle option as shown below:
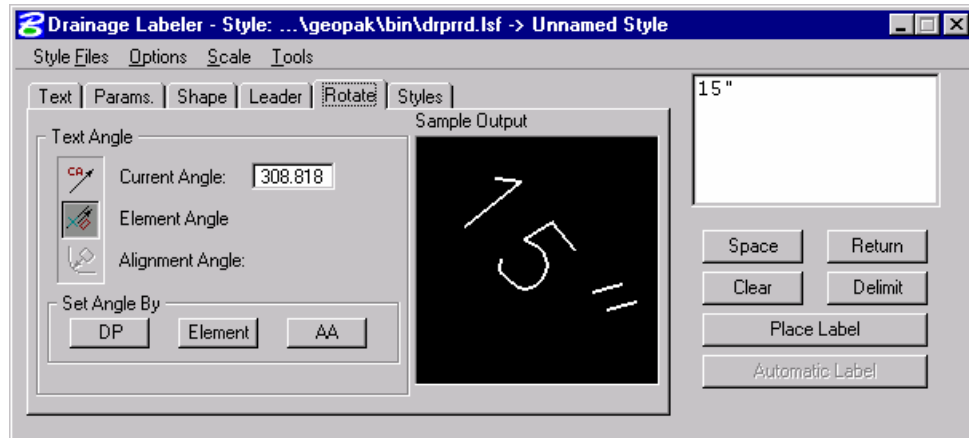
9. Place the label in the design file by clicking **Place Label**.

10. Move to the **Text Tab**, and select the second pipe: pipe-2. By activating the **Highlight** and **Window Center** toggles, the Drainage Labeler will navigate to the correspondent drainage pipe location in the dgn file.
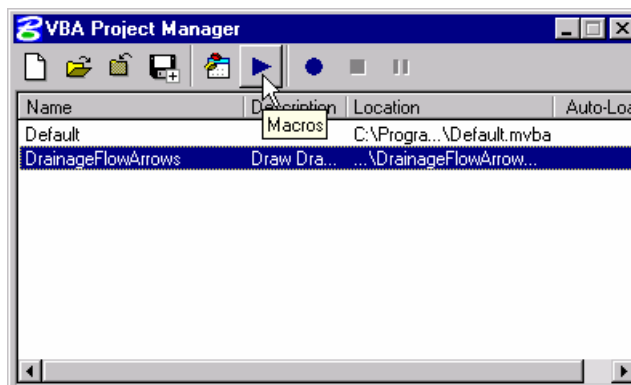

11. Label the remaining pipes and close the Drainage Labeler when done.
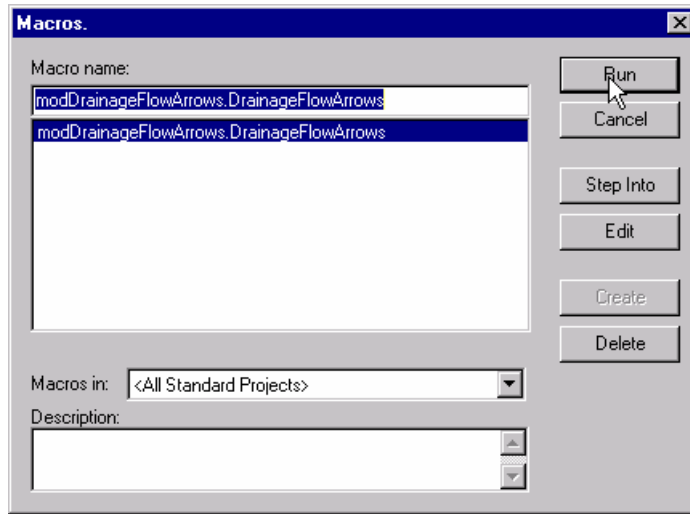
> *Plotting Flow Arrows along the Pipes*

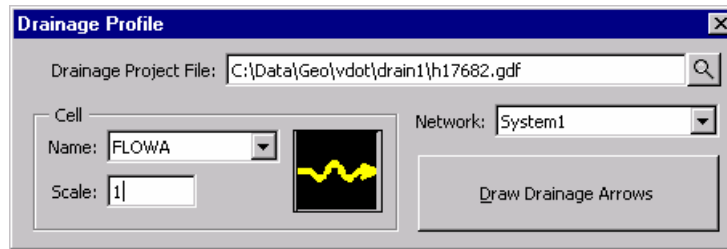1. Open VBA Project Manager dialog. *(MS Menu: Utilities > Macro > Project Manager).*
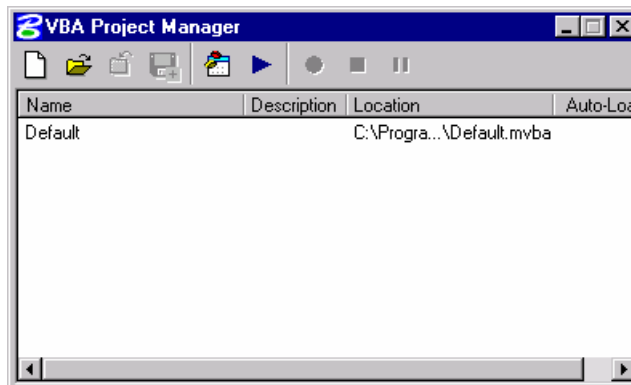
2. Load the macro: **DrawFlowArrow_along_pipes.mvba** located in the *c:\data\geo\VDOT\drain1\standards\geopak* directory.

3. Select the previously loaded macro and press the **Macros** button.

   VDOT GEOPAK Drainage

4. Click the **Run** button.

5. Setup the Drainage Profile dialog as shown below.

6. Click on the **Draw Drainage Arrows** button to plot the flow lines along the pipes.

7. Close the dialog when done.

AUTOMATED QUANTITIES

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.
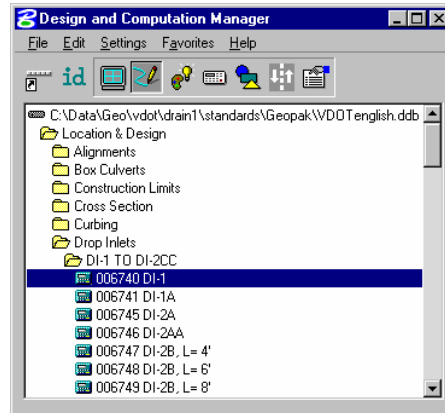

OPERATIONAL MODES: COMPUTE

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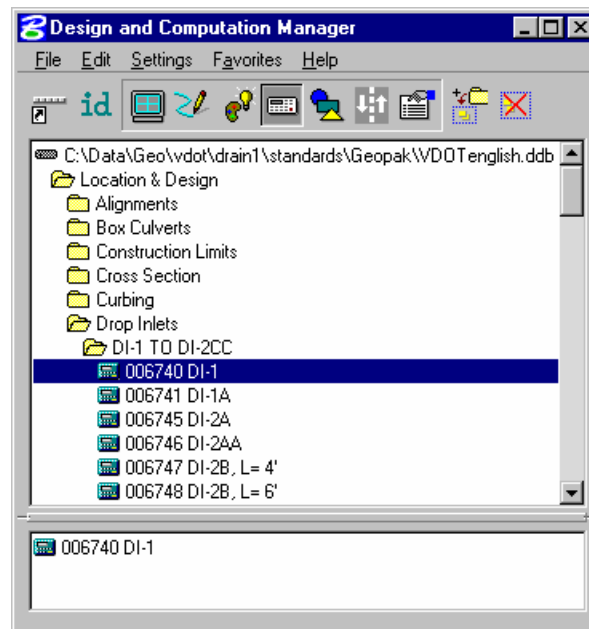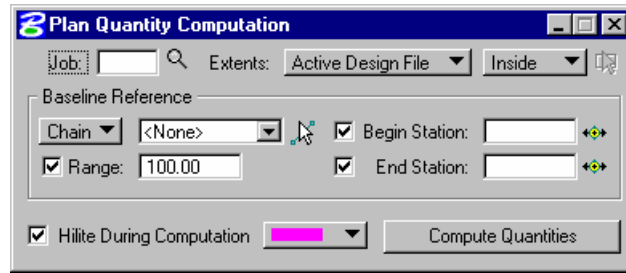
Once the MicroStation plan view design file is accessed, the first step is to invoke the Design and Computation Manager and click the shortcut button at the top of the dialog box to Compute.

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 EXERCISE: AUTOMATED PLAN VIEW QUANTITIES

### *Computing Automated Quantities*

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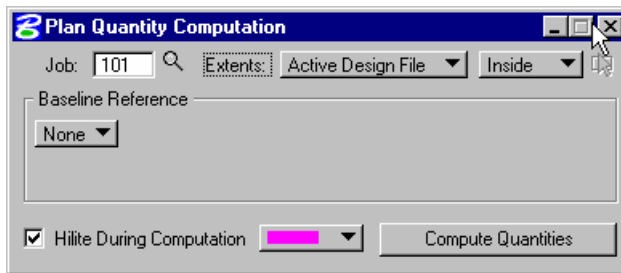

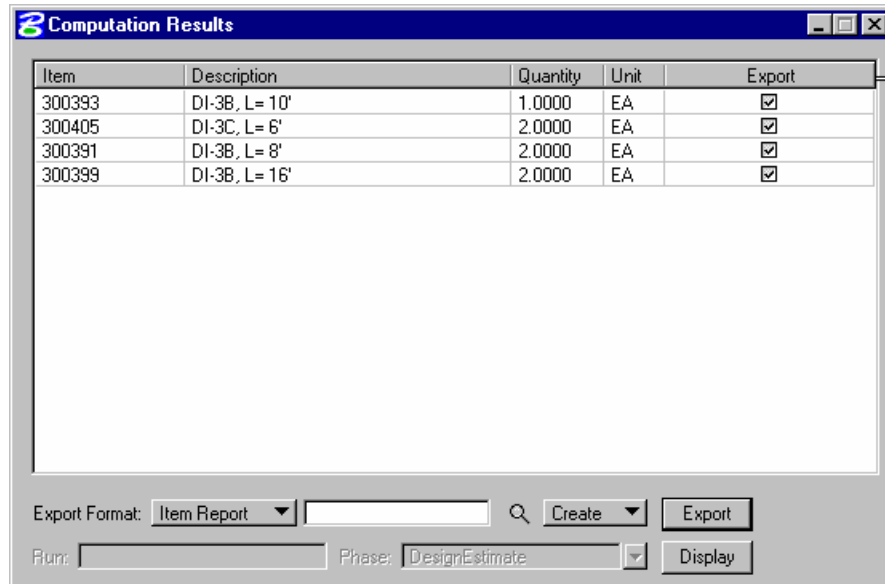2. In the Design and Computation Manager, set the Mode to **Compute** by pressing the 'calculator' icon button.
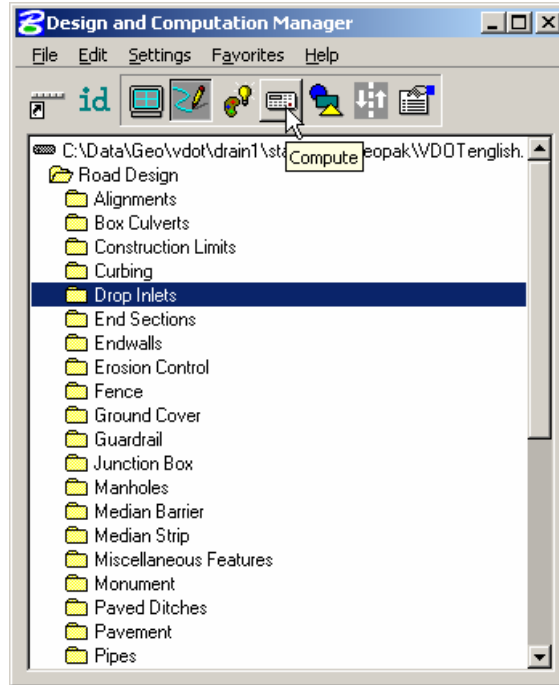
3. Highlight the Drop Inlets Category under Location&Design, then press the **Add to Collection** button on the Design & Computation Manager toolbar.
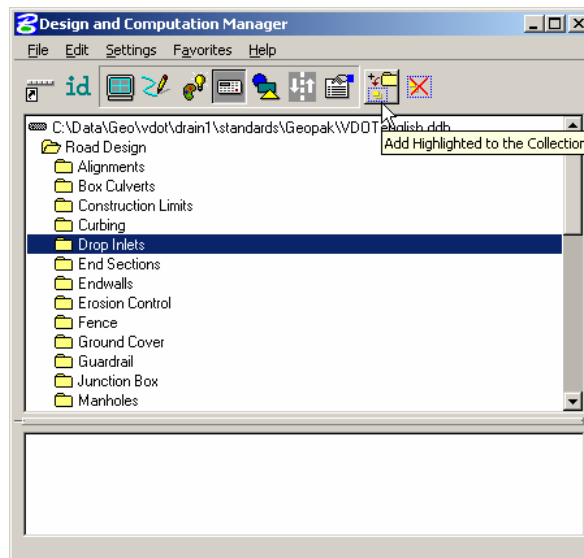
4. Set the Compute dialog as shown below.

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

6. Close the dialog when done.

7. Exit Microstation.

LAB EXERCISE: PLOTTING SUMMARY TABLES

### *Entering the Information in Microsoft Excel*

1. Open the Excel file, *c:\data\geo\VDOT\drain1\ Erosion Control Summary_master_data_entry.xls*

2. Enter the pertinent information for your project.

Warning   Do not modify any of the first two rows. They are part of the formatting of the Table operation.

3. When finished, delete the last row of the spreadsheet that contains the column labels.
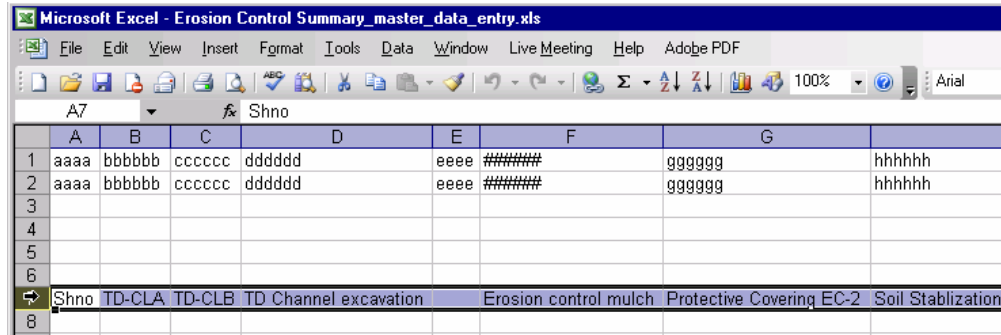
4. Click in the upper left corner of the spreadsheet to select all columns and rows.
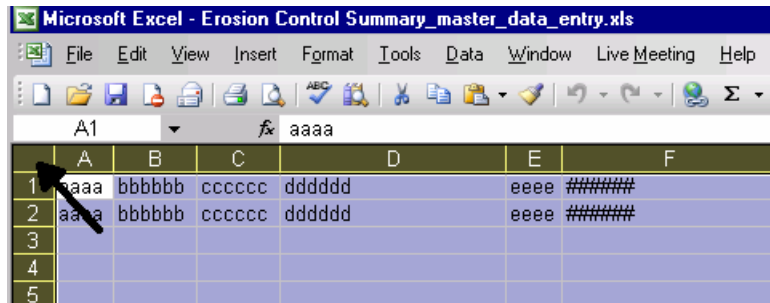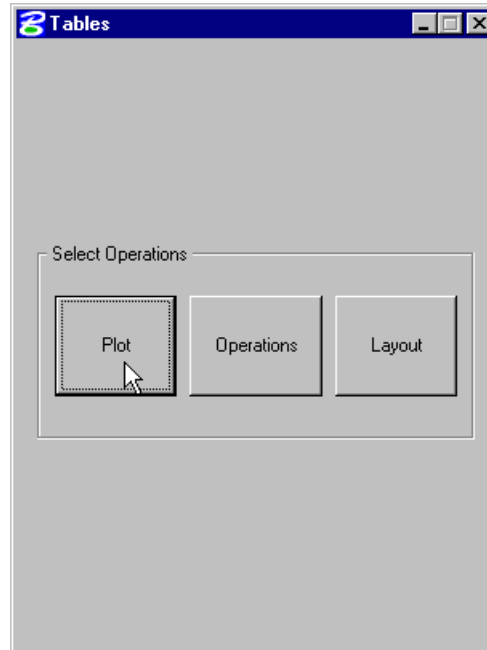

5. Format the cells using the AutoFit function. *(Excel: Format > Column > AutoFit Selection)*

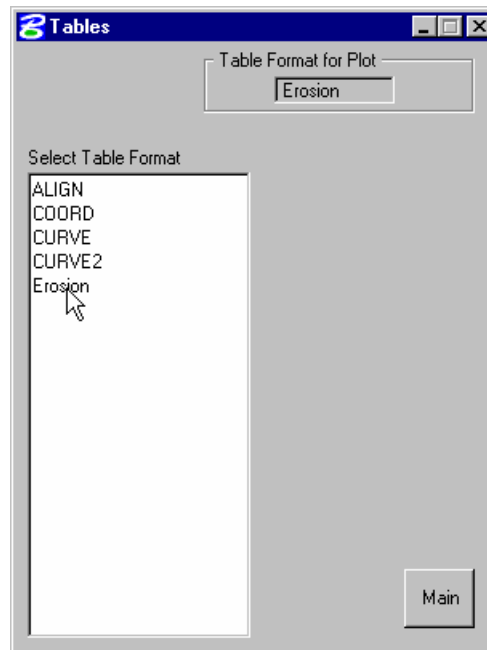6. Save the spreadsheet formatted as a space delimited file. *(Excel : File > Save As)*
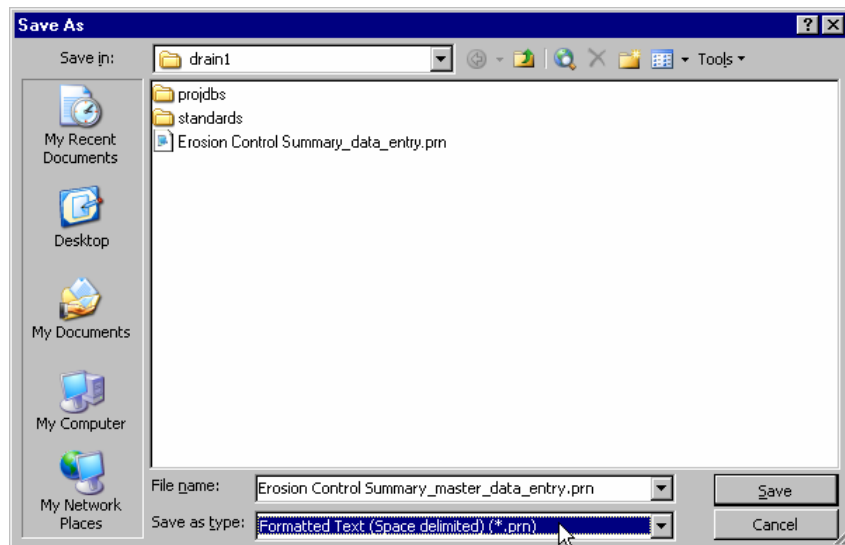
7. Close Microsoft Excel.

Plotting Tables

1. Open the MicroStation file, *c:\data\geo\VDOT\drain1\a5.dgn*

2. Open the Tables tool. *(Applications: GEOPAK Road > Plans Preparation > Tables)*

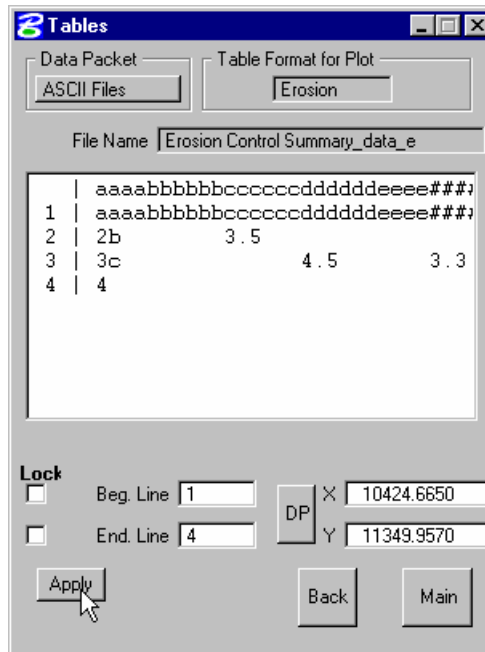3. Select the **Plot** option.

4. Select the Erosion table and load the file *c:\data\geo\VDOT\drain1\ Erosion Control Summary_data_entry.prn*.

5. Select the Begin and End line number you desire to plot.

6. Give a data point at the location indicated in the dgn file.

7. Click **Apply**. The table information will be plotted in the dgn file.

8. Close the dialog when done.

9. Exit MicroStation.

This page left intentionally blank.

9 Coordinate Geometry: Alignments and Profiles
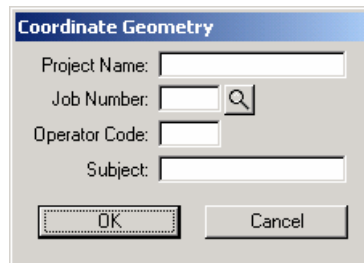
OBJECTIVES

In this chapter, you will learn more about: setting up and accessing the coordinate geometry database file and create alignments and profiles for a future ditch.

INTRODUCTION

The Coordinate Geometry database file (*.GPK file) is comprised of stored geometric elements such as points, lines, curves, spirals, chains, parcels and profiles. When COGO calculations are completed, the results are stored in the binary database file, which is then utilized throughout the design process with many other applications.

ACCESSING COGO

When Coordinate Geometry is started, the Start-Up dialog appears.

Project Name	Name displayed on reports. This is an optional entry, with a maximum of 60 alphanumeric characters. If Project Manager is active, this field is populated automatically.
Job Number	Identifies coordinate geometry database file and is a required field. The job number is a maximum of three alphanumeric characters. If Project Manager is active, this field populated automatically. This field is required.
Operator Code	Unique 2-character code which enables multiple users access to database file. The user's initials are suggested. This field is not required for single user access, but highly recommended. If Project Manager is active, this field is populated automatically.
Subject	Description of coordinate geometry work (could relate to the project) with a maximum of 48 alphanumeric characters. This field is optional.

After the dialog is populated and **OK** is clicked, the main coordinate geometry dialog is invoked.

COORDINATE GEOMETRY DIALOG
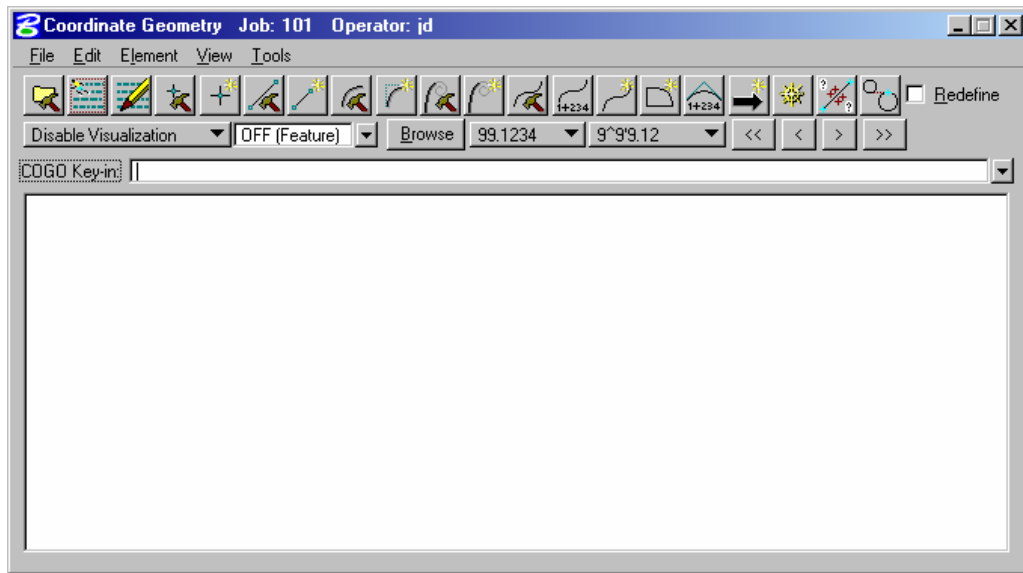
The coordinate geometry dialog box is made up of three separate display areas:

- Tool Bar consisting of drop down options and icons. The display of specific icons is customizable.

- COGO Key-in field enables manual entry of COGO commands. The drop down button allows for a history of commands to be reviewed.

- Output Display Window shows the results generated by the commands.

FILE COMMANDS
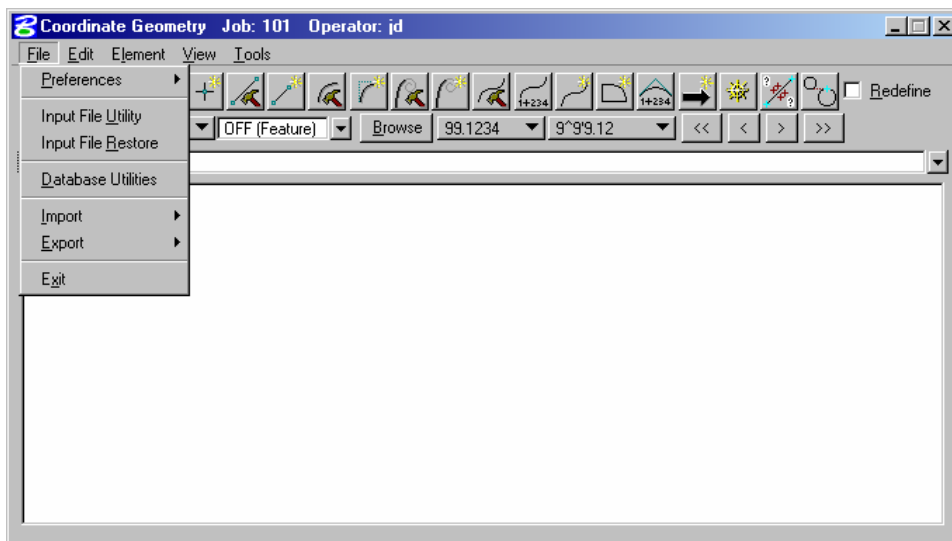
Preferences	Short-cut to COGO preferences and toggles for visualization coordinate readout and angle display.
Input File Utility	This option opens another dialog that contains group of file utilities that will be explained later.

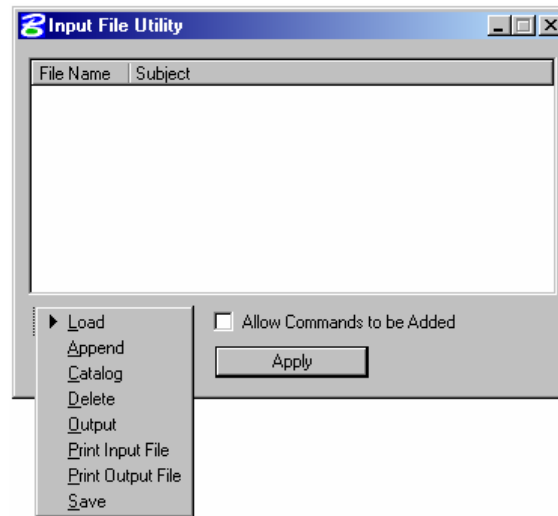| Input File Restore	Loads an ASCII file of GEOPAK commands into the display window. Note no processing occurs. The file must be named **j999ooc.inp**, where **999** is the current job number and "**oc"** is the operator code.
Database Utilities	Two tools are supported to validate, clean, and compress the COGO GPK database.
Import	Imports horizontal and vertical alignments and points from **ASCII Points**, **SDMS Alignments and Points**, **LandXML 1.0 Geometry**, **RDS**, and **VDOT PLT** formats into the GEOPAK .gpk file.
Export	Exports **ASCII points**, **SDMS Points**, **SDMS Alignments**, **LandXML 1.0 Geometry**, and **Alignments and Profiles** into formats for use in various data collectors.
Exit	Closes the COGO dialog box and ends the coordinate geometry session. A prompt to save the session appears. Yes saves the audit trail, No exits without saving, Cancel returns to the COGO session. Whether you pick Yes or No everything you did is still saved in the .gpk file.

USER PREFERENCES
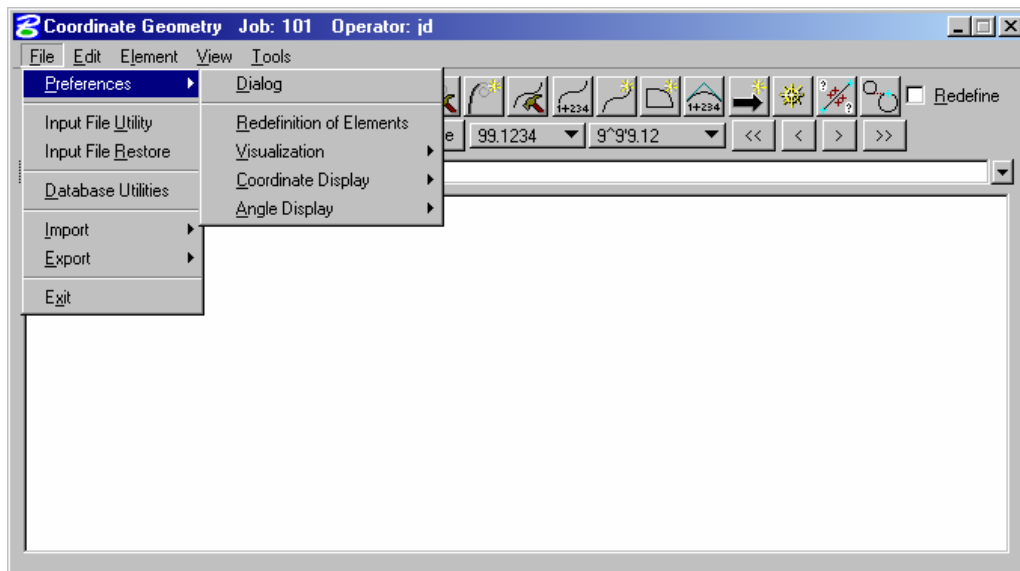
The User Preferences (as displayed in the exploded view above) are detailed in the following table.

| **Dialog**	Allows access to COGO Preferences dialog box.
Redefinition of Elements	Toggles the Redefine option on/off. If Redefine is on, COGO data can be redefined/overwritten.
Visualization	Enables the elements to be displayed in the MicroStation file permanently, temporarily, or not at all.
Coordinate Display	Toggles between displaying NE or XY coordinates.
Angle Display	Toggles between displaying Bearing or Azimuth.

INPUT FILE UTILITY

The File Utility option invokes the following dialog.

| Load	When selected, a menu appears listing all saved input files. *Highlight* a file then click **OK**. The input lines from the highlighted file are now displayed in the display window for viewing, editing, or processing.
Append	This command is for *input files* only. A new input file is created by copying the contents of an existing input file to the end of the current input file. The Save command must be used in order to store this new file.
Catalog	When selected, a menu appears listing all saved input files (under the current operator code) in the project directory. This is for reference only no action is taken.
Delete	When selected, a menu appears listing all saved input files. *Highlight* a file then click the **OK** button to remove this file from your project directory. Although the file is deleted, the elements stored as a result of processing the commands are not deleted.
Output	Writes an ASCII file of the current display for reviewing and / or printing. (Fname999.ooc, where **999** is the current job number and **oc** is the operator code).
Print Input File	Sends your input file to the printer.
Print Output File	Sends your output file to the printer.
Save	Saves the current audit trail to a file. (Fname999.ioc, where **999** is the current job number and **oc** is the operator code.).
Allow Commands to be Added	When this is toggled on, COGO commands are added to a previously loaded file.

EDIT COMMANDS
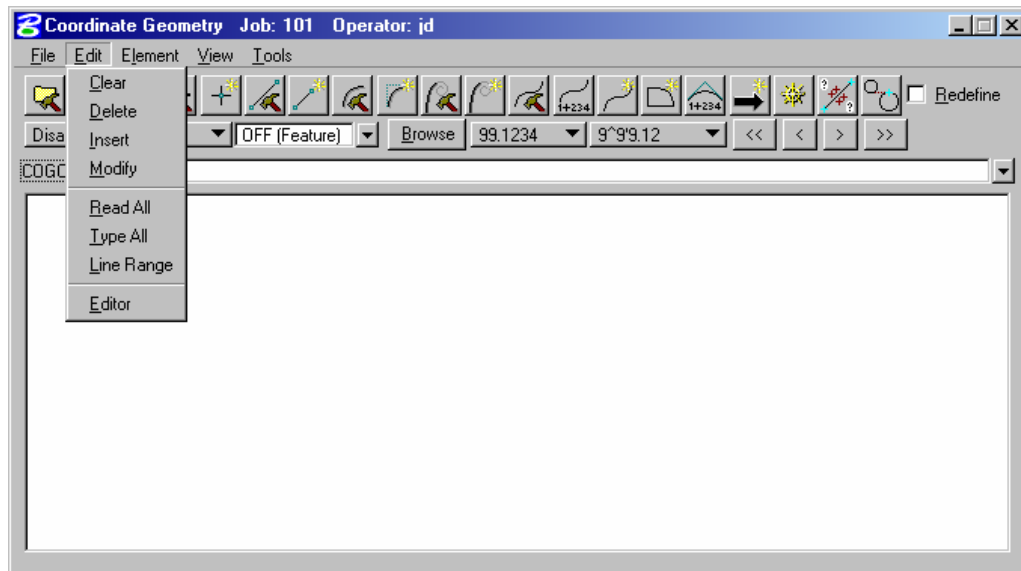
|	
Clear	Empties the memory of the current audit trail without saving and initializes the line numbers to begin a new sequence of commands. This does not clear the display window.
Delete	Deletes input commands in the input buffer by line number (or range of line numbers) and re-sequences the line numbers for the remaining commands.
Insert	Adds a command line to the current input buffer *before* a specified line number; subsequent command lines shift down and line numbering is automatically re-sequenced.
Modify	Changes a fragment in a command line. The modified command line is not processed until a Read command is performed.
Read All	Processes all lines in the audit trail.
Type All	Displays the contents of the audit trail.
Line Range	Opens the **Line Range** dialog where a range of lines can be entered that can either be read or displayed.
Editor	Opens the **COGO Command Editor**, which enables you to edit the current input file.

ELEMENT COMMANDS

The mainstay of GEOPAK Coordinate Geometry is the various element commands. Tools are supported for storing, stationing, adding additional data, deleting, etc. GEOPAK supports a wide variety of elements: points, curves, spirals, chains or alignments, parcels, and profiles. Each element is detailed on the following pages.

ELEMENT > POINT
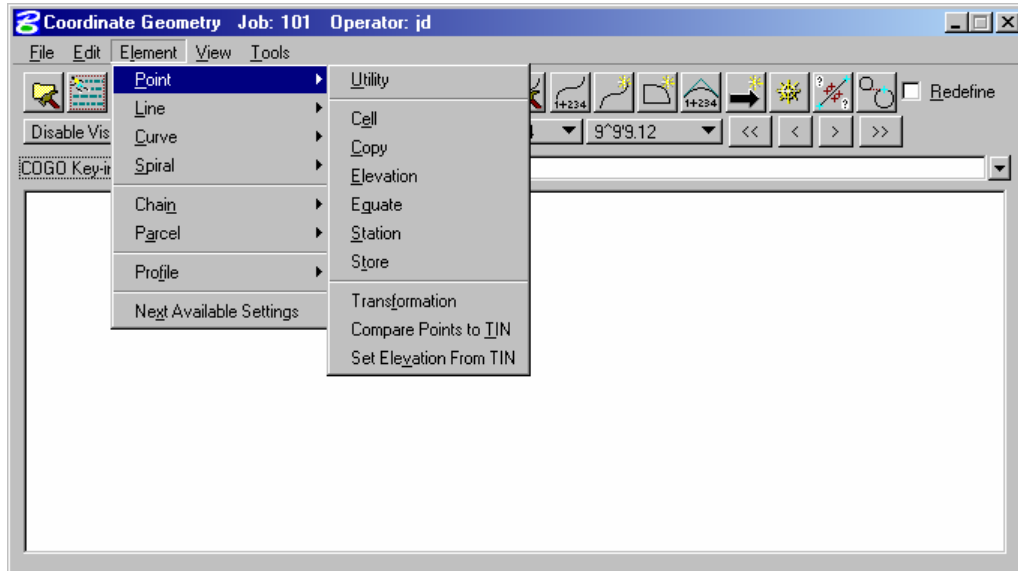
One of the most utilized sets of COGO tools are the point commands, which are detailed in the following table.

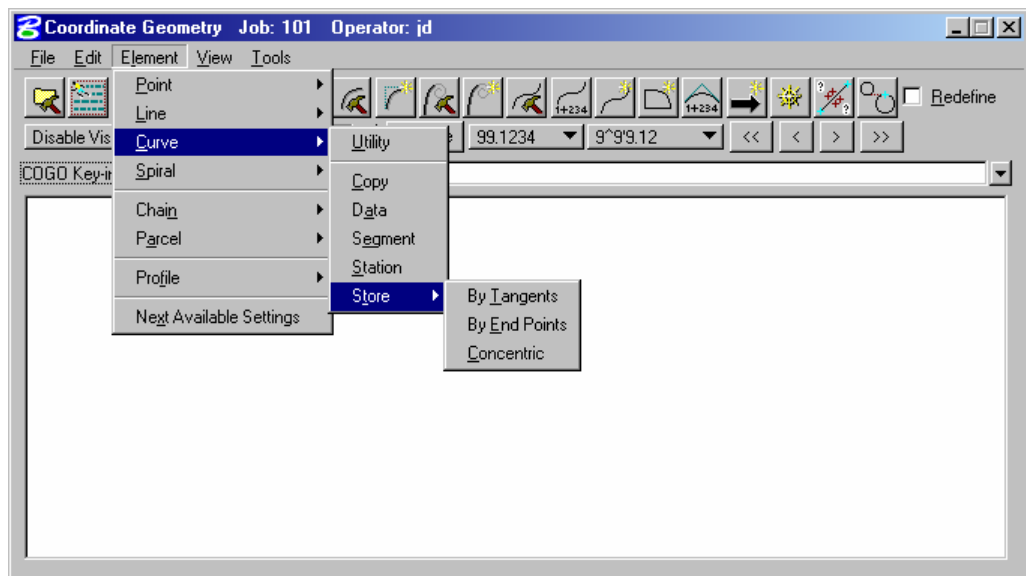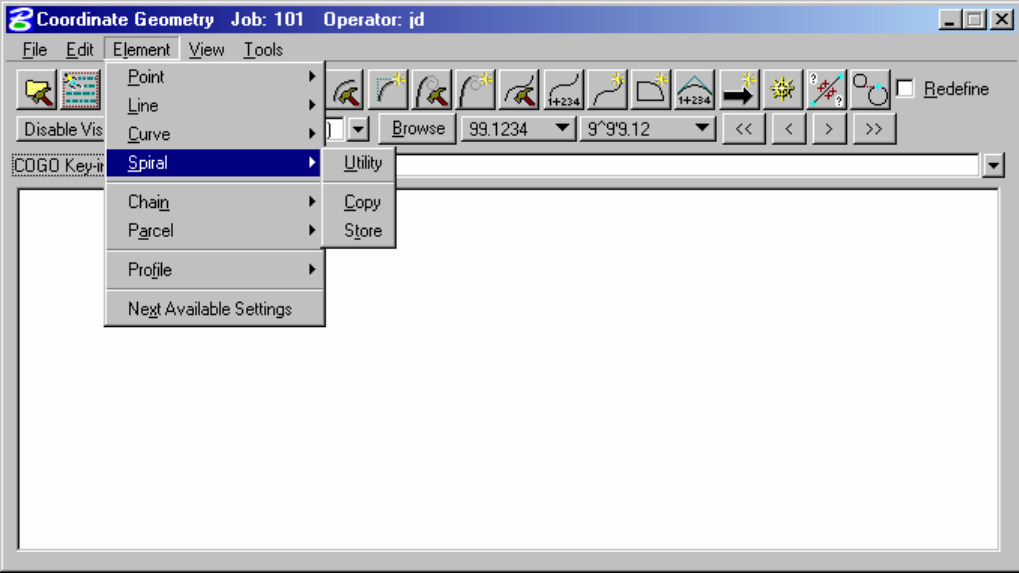|	
Utility	Opens the **Point Utility** dialog where the following options are available.
	Delete - Deletes a point number (or range of numbers) from the database file.
	Print - Displays the selected point numbers x, y, z coordinates, station value and other stored information in the display output window.
	Display - Visualizes the selected point numbers currently stored in the GPK file (Temporary Visualization or Permanent Visualization must be turned on).
Cell	Assigns a cell name to a previously stored point.
Copy	Copies points or a point range to a new point number or range within the same GEOPAK database file
Elevation	Assigns an elevation to a previously stored point
Equate	Stores a new point with the same values as a previously stored point
Station	Adds a station to a previously stored point.
Store	Stores a point by key-in coordinates or by selecting a location graphically. A station, elevation, point code, cell, feature, or description can be optionally stored with the point.
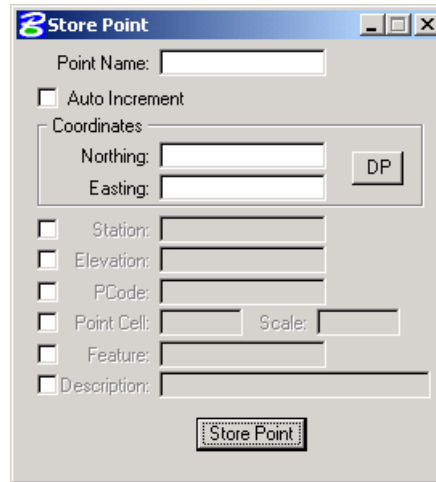
Transfor-mation	Performs coordinate transformations based on user-defined control pairings
Compare Points to TIN	Utilizing a set of COGO points (with elevations) compares the COGO elevation to the TIN elevation and completes statistical analysis for "goodness of fit."
Set Elevation from TIN	Utilizing this command, GEOPAK adds or updates an elevation to a previously defined point, based on a TIN model.

ELEMENT > CURVE

Utility	Opens the **Curve Utility** dialog where the following options are available.
	Reverse - Reverses a previously stored curves direction.
	Delete - Deletes a curve (or range of curves) from the database file.
	Print - Displays the selected curves x, y, z coordinates, station value and other stored information in the display output window.
	Visualize - Visualizes the selected curves currently stored in the GPK file (Temporary Visualization or Permanent Visualization must be turned on).

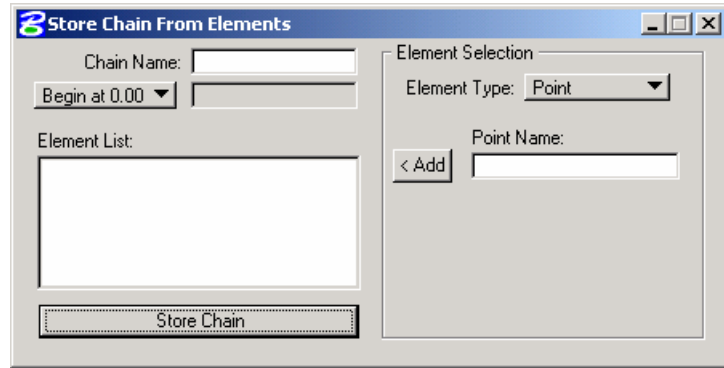|	
Copy	Copies a specified curve to another curve name.
Data	Calculates the geometric parameters of a curve, displaying values for Delta, Degree, Tangent, Length and Radius.
Segment	Defines new curves by dividing a stored curve into segments.
Station	By identifying a curve and a control point on the curve (PC, PI, or PT), a station value may be assigned.
Store	Provides various options for defining and storing curves such as **Store Curve By Tangents** as shown below.

ELEMENT > SPIRAL

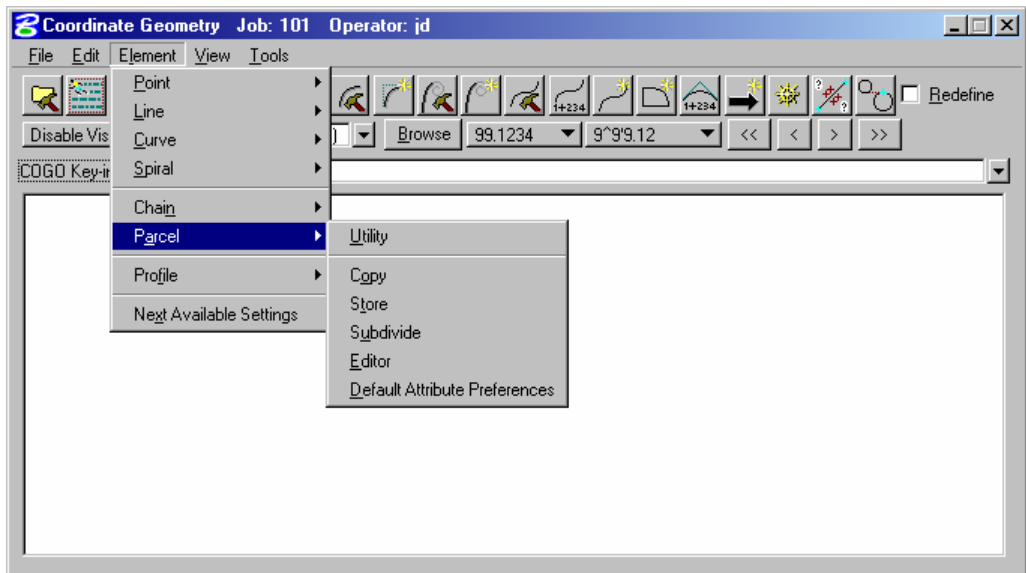|	
Utility	Opens the **Spiral Utility** dialog where the following options are available.
	Delete - Deletes a spiral (or range of spirals) from the database file.
	Print - Displays the selected spirals x, y, z coordinates, station value and other stored information in the display output window.
	Visualize - Visualizes the selected spirals currently stored in the GPK file (Temporary Visualization or Permanent Visualization must be turned on).
Copy	Copies a specified spiral to another spiral name.
Store	Provides various options for defining and storing spirals.

ELEMENT > CHAIN

|	
Utility	Opens the **Chain Utility** dialog where the following options are available.
	Area - Calculates the area of a closed chain.
	Delete - Deletes a chain (or range of chains) from the database file.
	Print - Displays the name of each element in the selected chains.
	Describe - Displays the alignment data of each element in the selected chains.
	Visualize - Visualizes the selected spirals currently stored in the GPK file (Temporary Visualization or Permanent Visualization must be turned on).
Layout Offset	Project points onto a chain or compute offset distance and direction between two chains at each control point or at user defined intervals of a specified station range.
Station	Provides a method for stationing or re-stationing a chain.
Station Equation	Provides a method for applying a station equation to a chain.
Store	Provides three options for storing a chain in the database file, **From Elements**, **Store Offset Chain**, and **Store Transition Chain**. The **Store Chain From Elements** dialog is shown below. (Chain name can be between 1-9 alphanumeric characters)

ELEMENT > PARCEL

Utility	Opens the **Parcel Utility** dialog where the following options are available.
	Delete - Deletes a parcel (or range of parcels) from the database file.
	Print - Displays the elements of composition, the area of tract stored, taken and remaining.
	Describe - Displays the elements of composition, the area of tract stored, taken and remaining and a point, bearing, distance description of the specified tract.
	Visualize - Visualizes the selected parcels currently stored in the GPK file (Temporary Visualization or Permanent Visualization must be turned on).
Copy	Copies a specified parcel to another parcel name.
Store	Stores a parcel by adding points, curves and spirals.
Subdivide	Divides a parcel into individual lots.
Editor	Edits a parcel.
Default Attribute Preferences	Defines a set of attributes that are assigned to each additional parcel that is stored.

PARCEL COMMANDS (MANUAL ENTRY)

Several parcel commands are supported only by manual entry and have no associated dialogs.

Store Taken	Store the portion of a parcel taken by entering point and curve names in either a clockwise or counterclockwise direction.
Store Easement	Store easements by entering point and curve names in either a clockwise or counterclockwise direction.
Own Parcel	Stores the names of the owners associated with previously stored parcels.
Make Legal	Creates a metes and bounds description and writes it to a user named text file. See also *Legal Description Editor* for the ability to create detailed and customizable legal descriptions.

ELEMENT > PROFILE
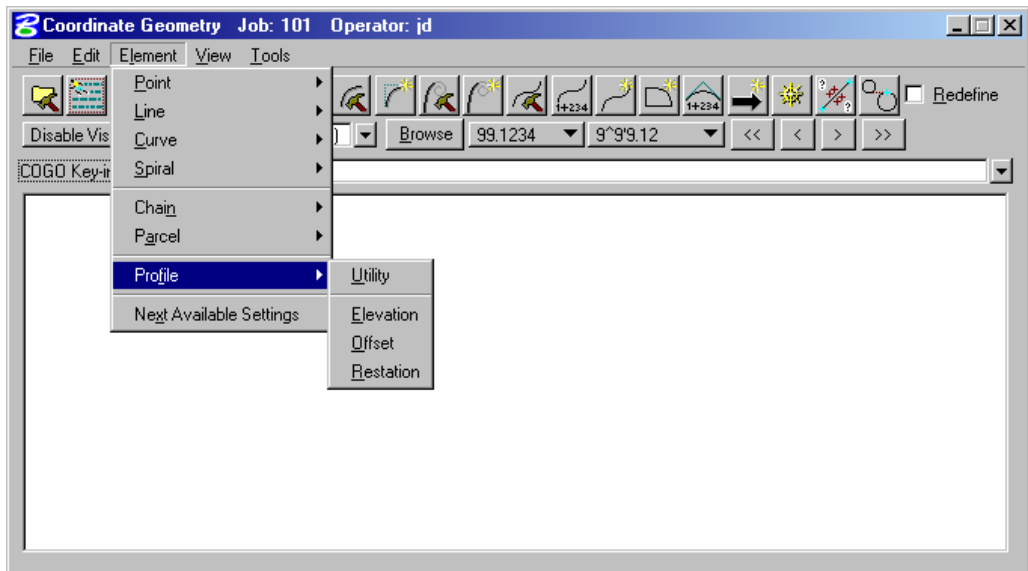
The Profile commands (as displayed in the exploded view above) are detailed in the following table. Additional commands are supported as keyin only commands.

|	
Utility	Opens the **Profile Utility** dialog where the following options are available.
	Delete - Deletes a profile (or range of profiles) from the database file.
	Print - Displays the data of the selected profile from the database file.
Elevation	Provides three options for reporting elevations along a selected profile, Station, Even Station, Incremental Stations.
Offset	Stores a new profile within a specified Station Range at a defined vertical offset from the original profile.
Restation	Creates a new profile with different stationing from a previously stored profile.

VIEW COMMANDS
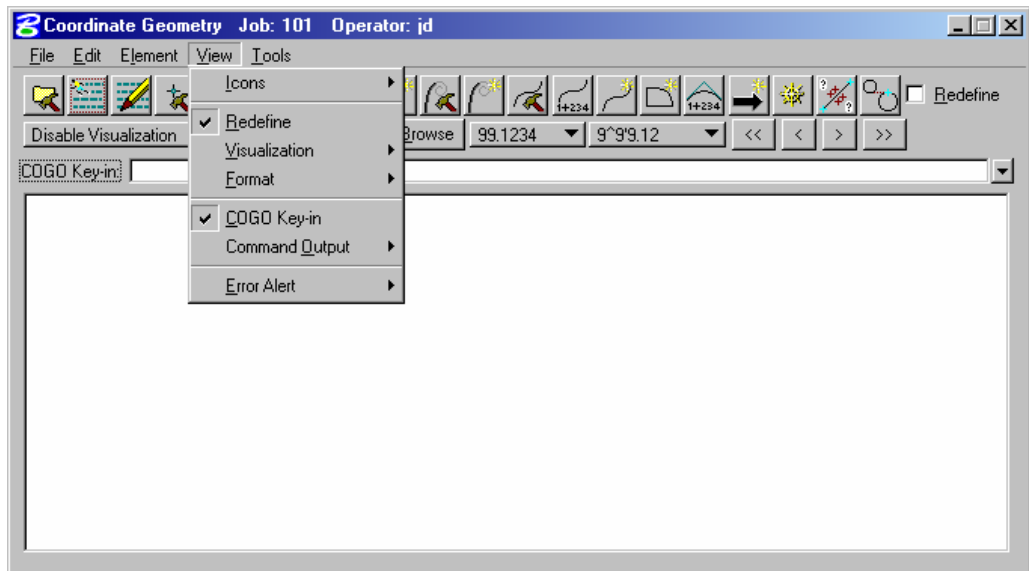
Icons	Enables you to customize which icons appear in the tool bar.
Redefine	Displays the Redefine box on the tool bar.
Visualization	Displays the visualization items on the tool bar.
Format	Displays the format items (i.e. number of decimals, station format, etc.) on the tool bar.
COGO Key-in	Displays the COGO Key-in box for entering commands in the dialog.
Command Output	Displays the Command Output Window in the dialog box, and controls options for the Command Output Window. The Command Output window can also be saved to a text file in lieu of using the File > Output command.
Error Alert	Enables you to activate a *beep* and/or restore a minimized COGO dialog when an error occurs.

TOOLS COMMANDS
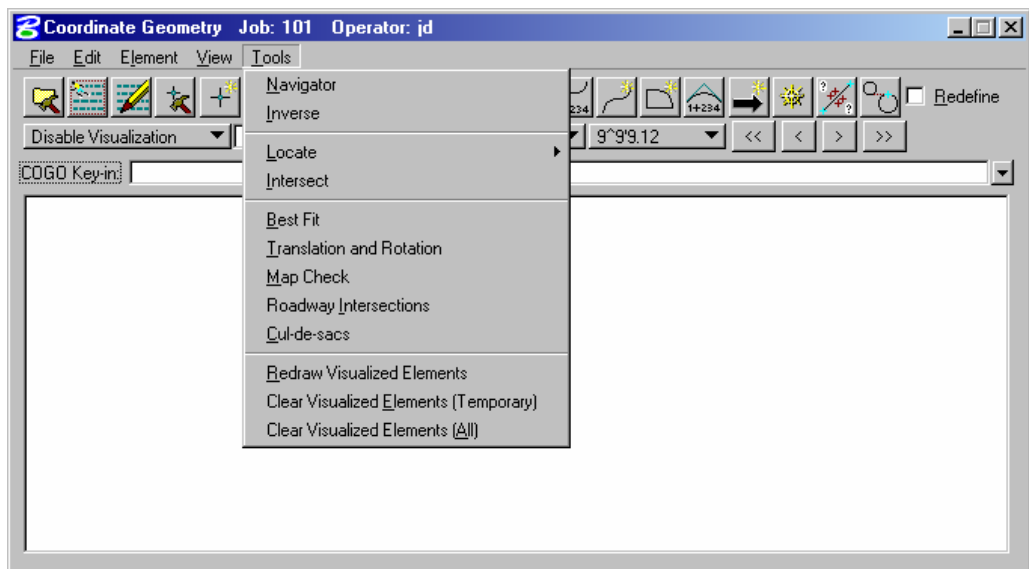
Navigator	Invokes the COGO Navigator.
Inverse	Calculates the distance and direction between points.
Locate	Stores a point or series of points not by specifying coordinates, but based on a previously stored point using several methods. To locate a point by distance and bearing, use the **Tools > Locate > Traverse** as shown in the dialog below.
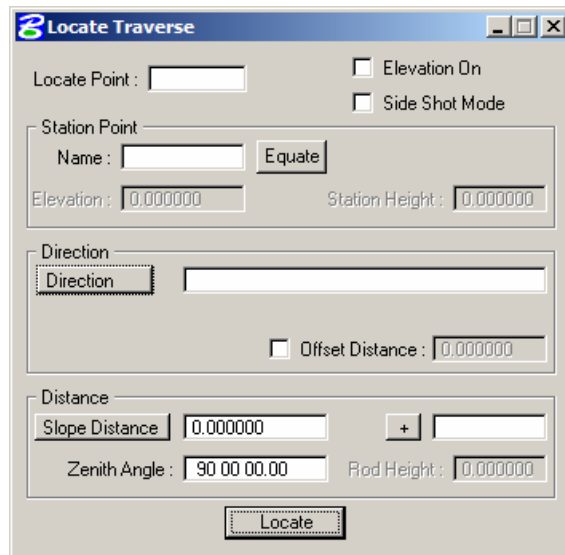
Intersect	Stores a point at the intersection of any two COGO elements.
Best Fit	Calculates a best-fit chain through a set of points.
Translation and Rotation	Moves, rotates, and scales a data set.
Map Check	Edits a parcel.
Roadway Intersections	Calculates data for the intersection of two COGO elements, usually chains.
Cul-de-sacs	Calculates data for a cul-de-sac.
Redraw Visualized Elements	Re-syncs the visualized COGO elements in the MicroStation file with the coordinate geometry data.
Clear Visualized Elements (Temporary)	Removes all temporary visualized elements from the view.
Clear Visualized Elements (All)	Clears all temporary and permanent visualized elements from the MicroStation file.

COGO NAVIGATOR

The COGO Navigator is a tool utilized to easily view and edit COGO data. Store, delete, edit, print elements, plus visualization and selection set commands are easily accessible via the Navigator. The Navigator can be accessed by the pull down menu **COGO > Tools > Navigator** or by the Navigator icon. The following dialog opens.
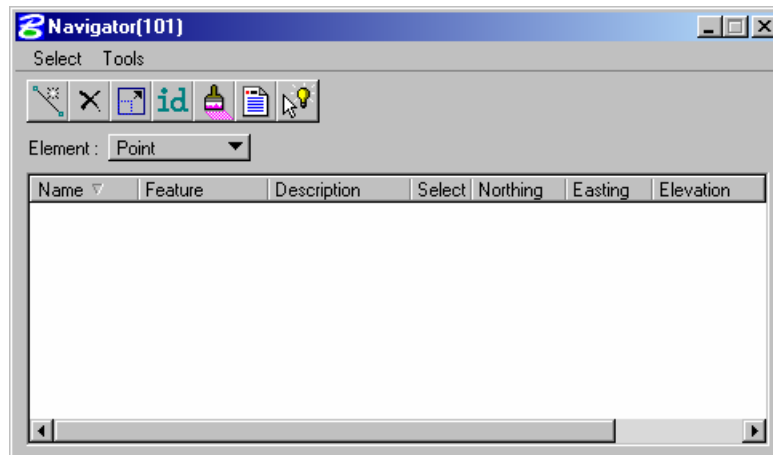
From the Navigator, points, curves, spirals, chains, survey chains, parcels, and profiles (all COGO elements contained in the coordinate geometry database file; *.gpk) can be added, deleted, modified, identified, visualized, printed, or selected.

NAVIGATOR > SELECT TOOLS

A variety of selection tools can be invoked via the Select pulldown, as depicted in the following exploded view.
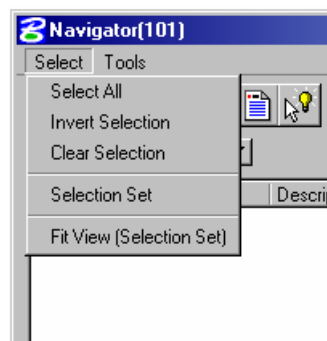
Select All	Selects all data items of a certain type. (i.e. all points)
Invert Selection	Selects all items not previously selected, and unselects all items previously selected.
Clear Selection	Unselects all items.
Selection Set	Allows you to create a selection set that meets particular criteria. Same as using the Selection Set icon
Fit View (Selection Set)	Fits the selected items to the active MicroStation window.

NAVIGATOR > TOOLS

A variety of tools can be invoked via the Tools pulldown, as depicted in the following exploded view. Many can also be invoked via the shortcut icons displayed at the top of the Navigator.
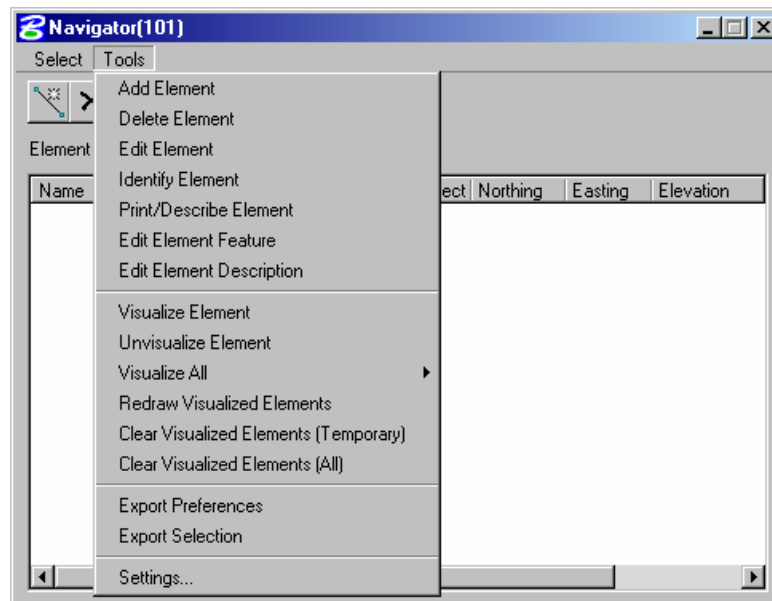
Add Element	Enables the selected type of element to be stored. When selected, the appropriate Store Element dialog is invoked.
Delete Element	Deletes the highlighted element(s).
Edit Element	Invokes the appropriate Store Element dialog, populated with the associated data of the selected element for editing.
Identify Element	When clicked and a COGO element is graphically selected, the element is highlighted in the display field. If the selected element is not the displayed element type (i.e., curve or point), the element type is also changed.
Print/Describe Element	Displays the selected element's data.
Edit Element Feature	Changes the feature code of the element.
Visualize Element	Displays the selected element(s) according to their feature code or default symbology.
Unvisualize Element	Removes the display of selected element(s) from the design file.
Visualize All	Displays all of the elements of a certain type, or all elements.
Redraw Visualized Elements	Re-syncs the coordinate geometry data with the data displayed in the MicroStation file.
Clear Visualized Elements (Temporary)	Clears only temporary visualized elements from the design file.

Clear Visualized Elements (All)	Clears all visualized elements from the design file.
Settings	Enables you to define certain actions and behaviors of the Navigator.
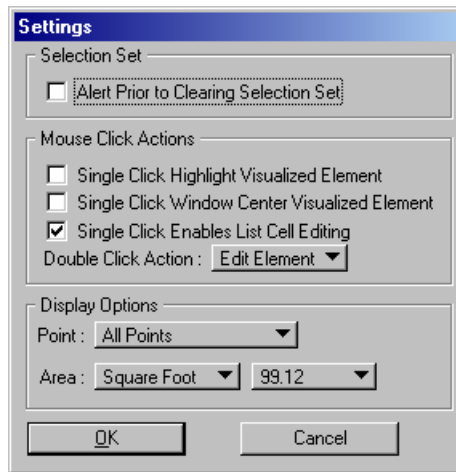
LAB EXERCISE: COORDINATE GEOMETRY
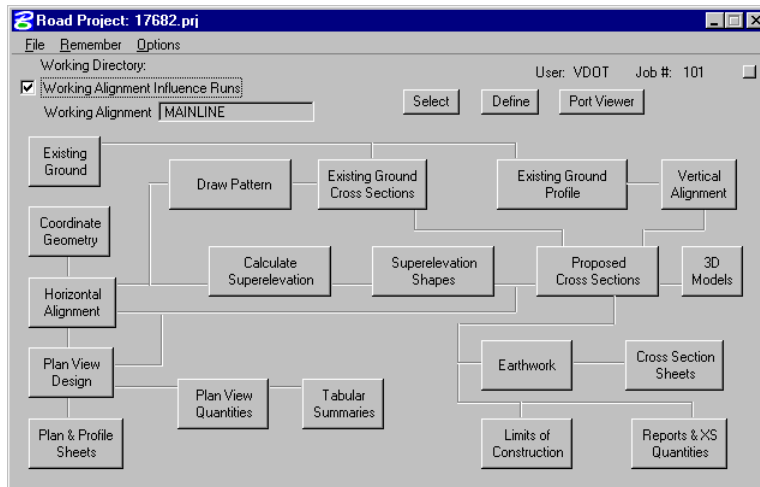
➢ *Opening the Road Project Manager*

1. Execute *c:\data\geo\vdot\drain1\LAB09_V8ad.EXE*

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

3. Open Project Manager (*MS Menu: Applications > GEOPAK Road > Project Manager)*

4. Select project: *17682.prj* located in *c:\data\geo\VDOT\drain1*, and click OK.
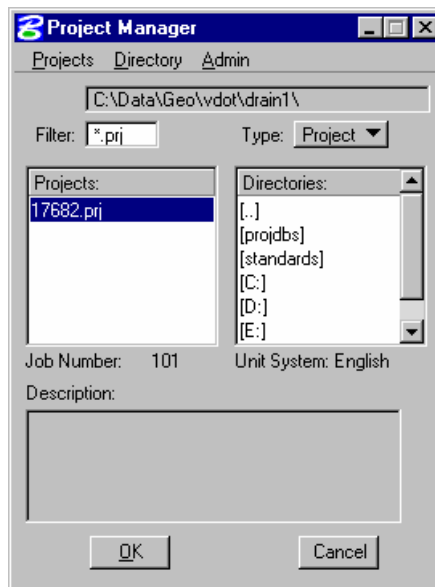
5.   Access User: VDOT.
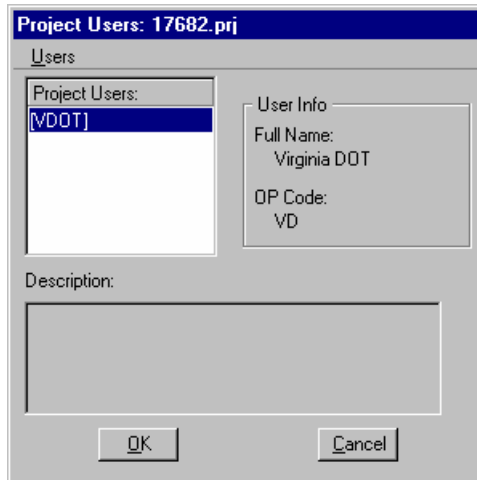
6.   Click **OK.** The Road Project dialog will load. This will setup the proper directories for the location of the gpk file.

7.   Click the Coordinate Geometry button.
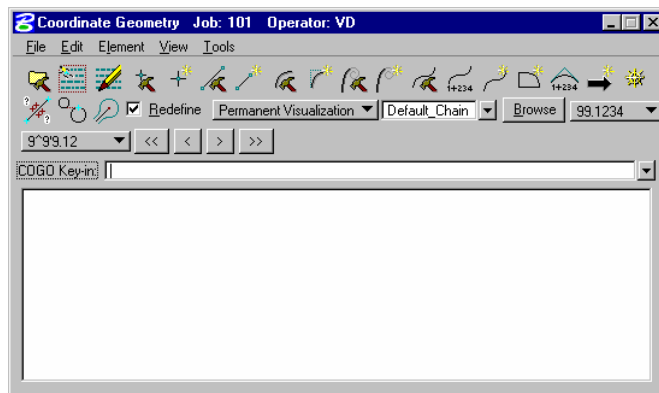
> ➢ *Storing COGO Points*

1. Select **Permanent Visualization** on the main COGO dialog.

2. Select the **Store Point** tool *(COGO: Element > Point > Store).*
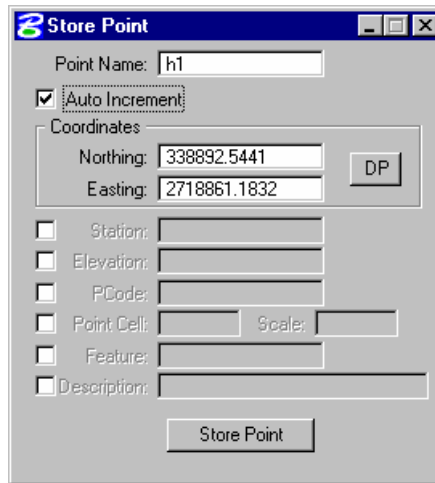
3. To store point **h1**, enter the point number and coordinates as indicated in the dialog shown above.

4. Activate the **Auto Increment** toggle. This will automatically increment the point number by **1** for subsequent **Store Point** commands.

5. Click **Store Point.** This stores the point into the COGO database (gpk).

6. Continue storing points utilizing the coordinates as shown below.

Point Number	Northing	Easting
H2	339044.1284	2718650.8320
H3	339140.6908	2718566.4231
H4	339200.0702	2718472.6387

7. Close the **Store Point** dialog.

> ➢ *Store Curves*

1. Access the **Store Curve By Tangents** dialog. *(COGO: Element > Curve > Store > By Tangents).*

2. To store the curve **Ditch1,** complete the dialog as shown below.
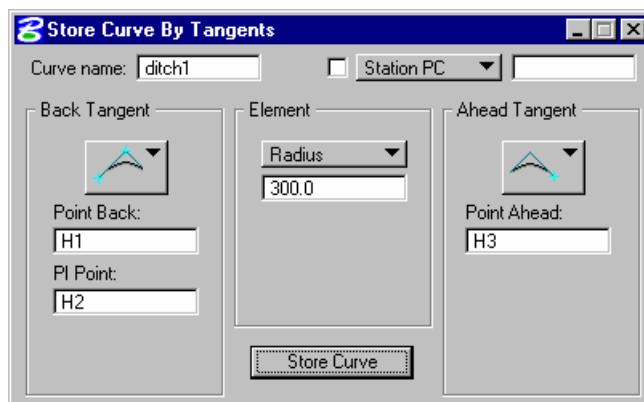
3. Click **Store Curve** to store curve **Ditch1** into the COGO database.

4. Store curve **Ditch2**, populating the dialog as shown below.
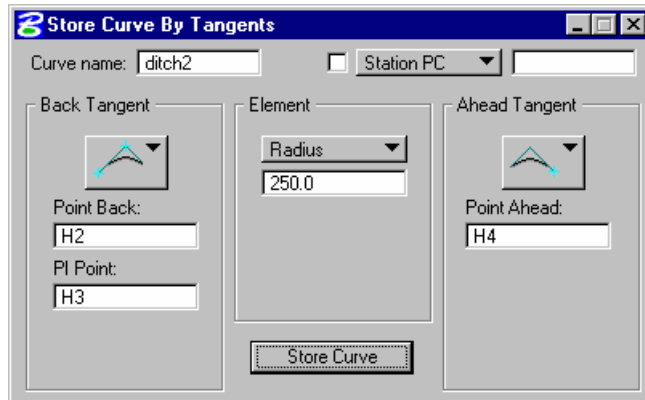
5. Click **Store Curve** to store curve **Ditch2** into the COGO database.

6. Close the **Store Curve By Tangents** dialog.

➤ *Storing the Ditch Alignment*

1. Access the **Store Chain From Elements** dialog. *(COGO: Element > Chain > Store > From Elements).*

2. Enter **Ditch** for the **Chain Name.**

3. Set the option button to **Begin Station** and enter **0+00**.

4. Build the list of COGO elements which will become a part of the chain. You can fill in the elements comprising the chain by keyin or by graphically selecting the visualized elements in the design file. Ensure that the elements are added to the list in the order that you want them to be stored as a chain.
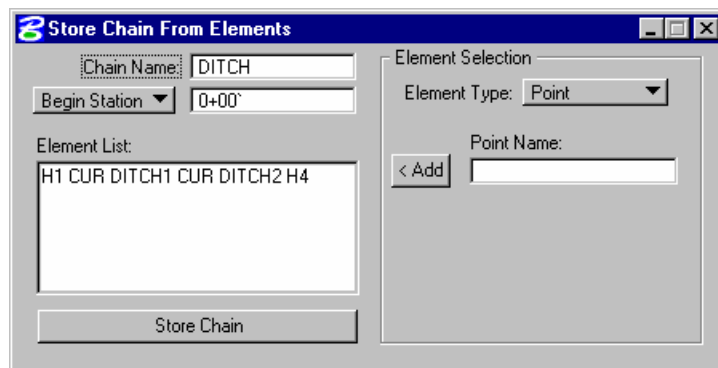
5. Click **Store Chain** to store chain **DITCH** into the COGO database.

6. Close the **Store Chain from Elements** dialog.

➤ *Describe the Ditch Alignment*

1. Access the **Cogo Navigator**. *(COGO: Tools > Navigator).*

2. Set the **Element** option to **Chain.**
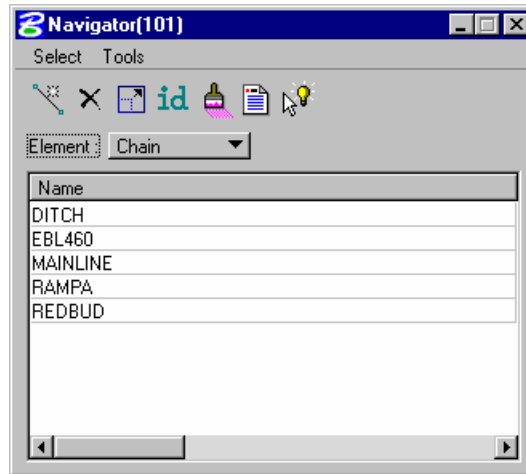
3. Highlight the chain DITCH

4. Select the **Print/Describe** command. *(COGO Navigator: Tools > Print/Describe Element).*

5. The chain geometry will be displayed in the main COGO window as shown below.
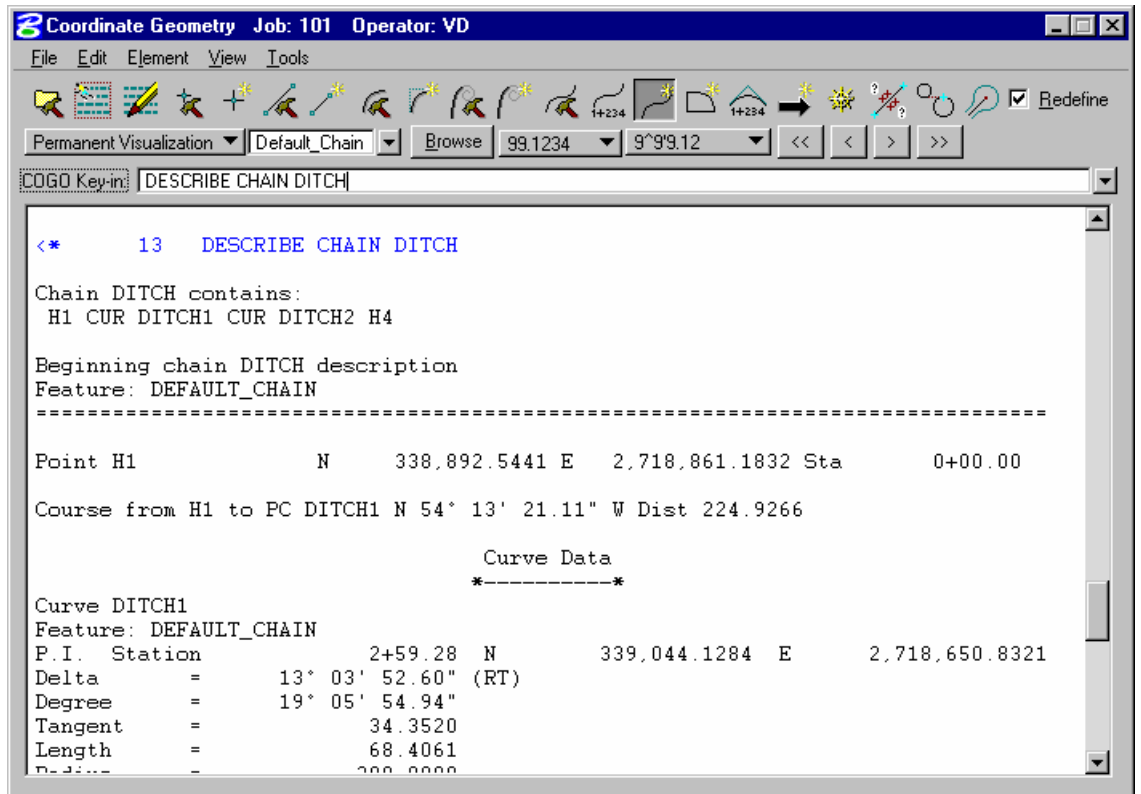
6. Close the **Navigator** dialog.

> ➤ ***Create Output Files***

1. Access the **Input File Utility** dialog. *(COGO: File > Input File Utility).*

2. Set the option button to **Output.**

3. Enter **Ditch** for the **Output File Name**.
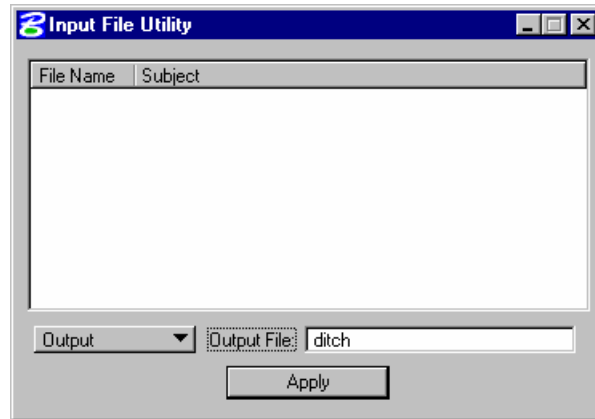
4. Click **Apply.**

5. An ASCII file named **ditch101.ovd** will be created in your project directory.

6. Close the **Coordinate Geometry** dialog box.

➢ *Drawing the Ditch Alignment*

1. Access the Design and Computation Manager *(Applications > GEOPAK Road > Design & Computation Manager)*. A secondary dialog box will appear to be used in conjunction with the Design and Computation Manager dialog box.
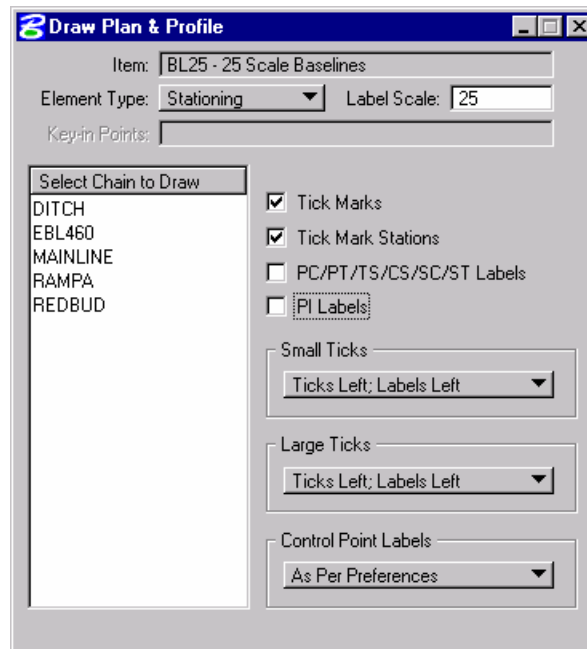
2. Select the item shown in the dialog box shown below.

3. Click on the **Draw Plan and Profile** button from the secondary dialog box.
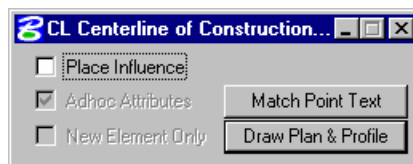
4. Select **Chains** operation as the Element Type. Each of the chains will be displayed in the list box as shown. Notice that the options that are to be drawn with the chain are already activated. Set the Label Scale to **25**.
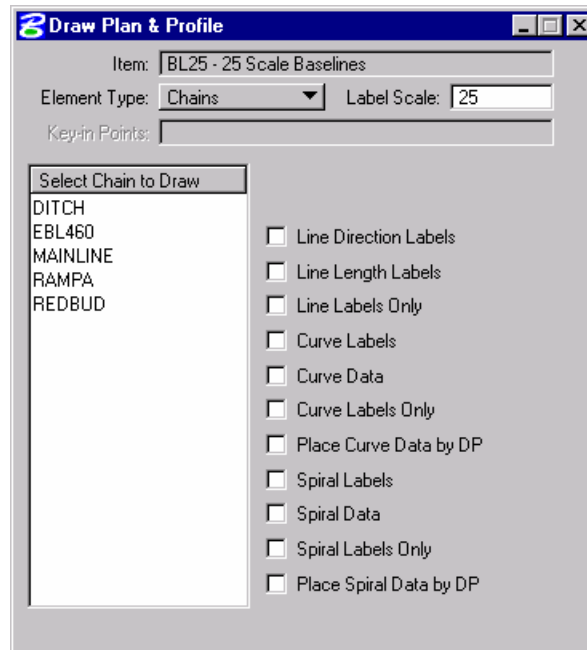
5. To draw the chain, click on the name of the chain: **Ditch** in the list box. Only click once!! Each time you ID a chain it will be plotted into the design file.

6. Change the Element Type to **Stationing** and modify the dialog settings as shown.

7. To draw the stationing for the chain, click on the name of the chain: **Ditch** in the list box.

8. Exit the Plan and Profile Draw dialog box and close the D&C Manager.

EXISTING GROUND PROFILES : DRAW PROFILE TOOL

The Draw Profile Tool is not supported by Project Manager. It can be accessed by selecting Applications > GEOPAK ROAD > Plans Preparation > Draw Profiles or from the GEOPAK ROAD Tools tool frame.

The generation of profiles must be invoked from within the 2D MicroStation design file where the profile is to be drawn.
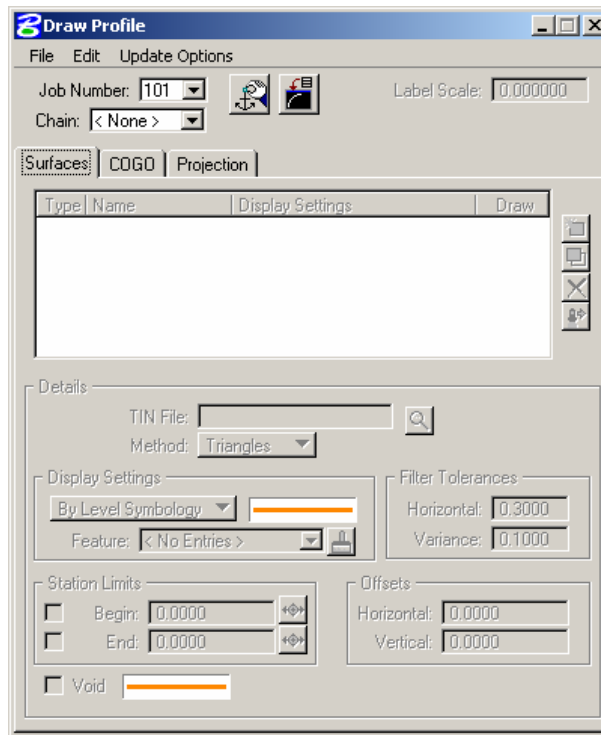
The initial entries that are required are the Job Number and Chain name. Once selected, click Dialog Profile Cell Control (to the right of the Job Number field).
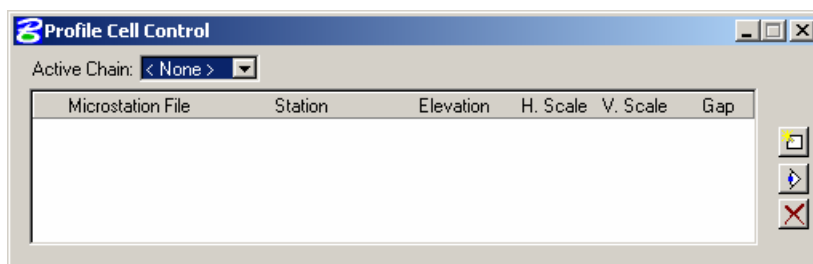
Select the Active Chain that the new or existing profile is to be stationed along. If no profile cell exists, click Place Profile Cell along the right side of the window to place a new cell. If multiple profile cells exist, highlight the desired cell and click Activate Profile Cell.

UPDATE OPTIONS

Four update options exist that allow you the control needed to tell the software what to do if a profile has been previously drawn in the file and needs updated. The options are toggled off/on by each selection of the option by clicking the **Update Options** pulldown. The check to the left of the option indicates the option is active. Only one option may be active at any given time, therefore, if an option

is selected, the previous active option is de-activated. The four options are detailed in the following table.

Delete Existing Elements and Redraw	When this option is activated, any profiles previously drawn with this tool are deleted and new ground lines are drawn.
Delete Non-Modified Elements and Redraw	When this option is activated, any profiles previously drawn with this tool and not modified with any MicroStation commands are deleted and new ground lines are drawn.
Draw on Top of Existing	When this option is activated, any previously drawn profiles are ignored and a new set is drawn, resulting in multiple copies of each profile.
Query	When activated, you are prompted each time the Draw button is pressed.

SURFACES TAB

The Surfaces tab defines the surfaces utilized as source data when drawing profiles. Multiple surfaces from a variety of sources can be drawn in a single processing. When the tab is selected and a profile cell is placed, the dialog dynamically changes as shown in the following graphic.
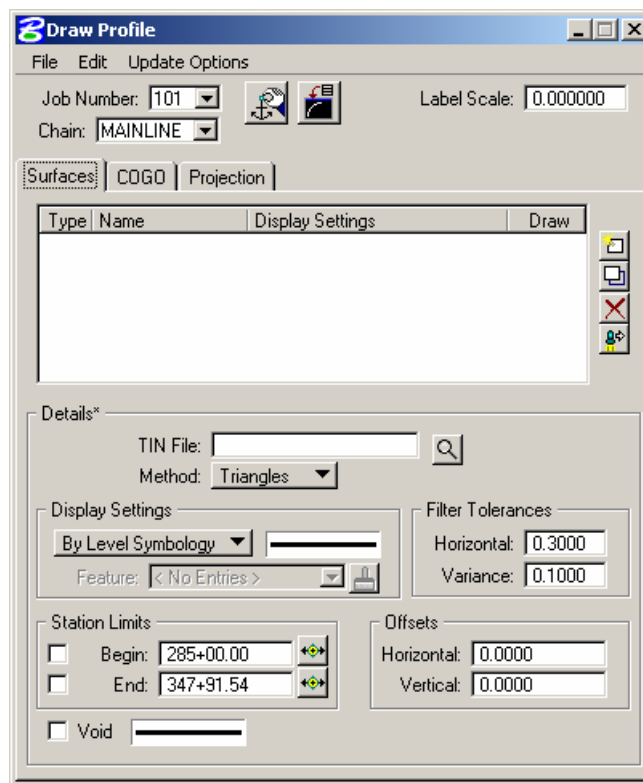
Each surface to be drawn must be added to the list box. This is accomplished via the action/edit buttons on the right side of the dialog box. Each surface draws a cross section line string or lines. When a line is added to the list box, the profile is drawn. Note: clicking on the draw toggle in the list box deletes or redraws the profile.

To add to the list box, simply select the source data and method, Display Settings, Tolerances and Offsets, and then add the profile to the list. Once a profile is drawn, it may be updated at any time by

using **Update Profile**. Refer to the **Update Options** section to review those options. Any profile may optionally be stored to the GPK file.

DISPLAY SETTINGS

The Display Settings group box specifies the element symbology of the cross section elements being generated. The symbology may be specified using Level Symbology or a feature selected from a D&C Manager database.

To use **By Level Symbology**, set the option to By Level Symbology, then double click on the graphic. This opens the Set Feature dialog box, wherein the symbology can be specified. Click OK when completed.

To use **By Feature**, set the option to By Feature. Next, press the Paintbrush icon, which invokes the current Design and Computation Manager. Select the desired item, and then press the OK button, which closes the Design and Computation Manager, and populates the Draw Profile Display Settings group box.

FILTER TOLERANCES

Both Horizontal and Variance filter tolerances are considered together for each pair of profile segments. The middle point is deleted if both segment lengths are less than the Horizontal filter tolerance while the projected distance between the mid-point and the chord between the two end points is less than the Variance tolerance.
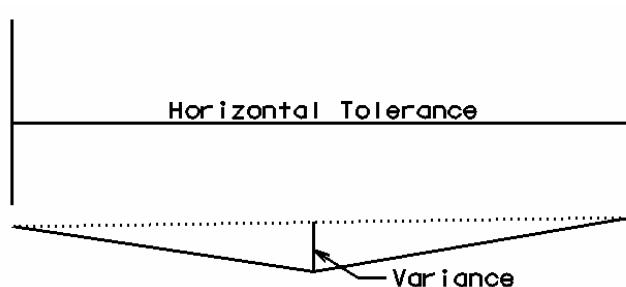

OFFSETS

Vertical or horizontal offsets may be specified in terms of master units (feet or meters). The horizontal offset is the distance offset from the chain. Once the horizontal location for the profile is determined, the data source is utilized to determine the profile. Any vertical offset is applied after the profile is generated from the source data.

STORE PROFILE TO COGO

When the profile is drawn, it may also be stored in COGO by pressing this button, which opens the dialog box depicted below left.
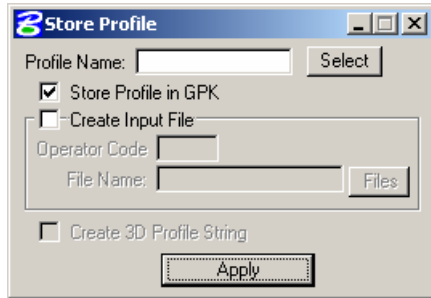
The profile may be stored, or just the input file may be created. If an input file is created, the Operator Code and File Name are required. If the dialog box is invoked while in a 3D file, you may toggle on the option to create a 3D profile string. The 3D profile string consists of the circles denoting every location where the design centerline intersects topographic elements. If the circles are not deleted at this time, they can be deleted at any time with a single application of the MicroStation Delete Element command.

COGO TAB

The COGO tab is utilized for drawing any profile that has been previously stored into the GPK file.
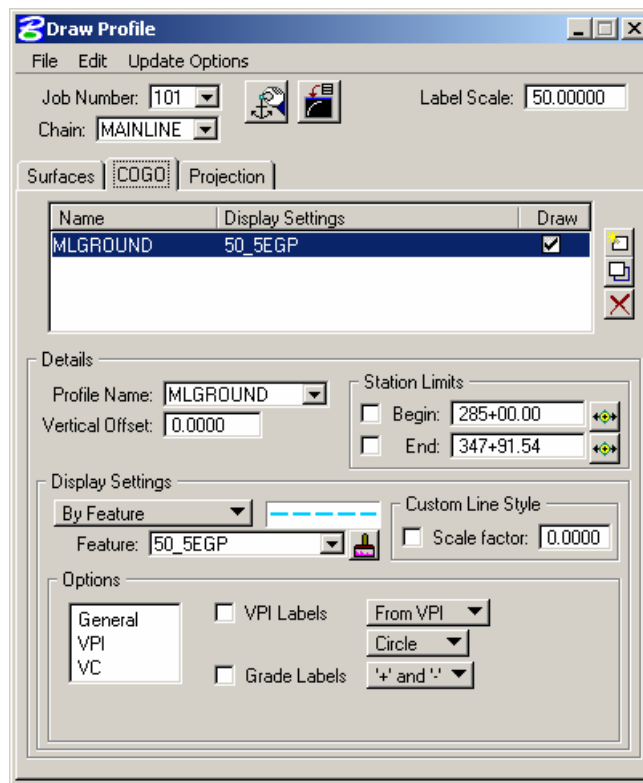
To draw a profile, first select the Profile Name to be drawn. Vertical Offset defaults to a value of 0, but can be set to any value. This instructs the software to draw the profile at a distance above or below the elevations in the GPK file. Station Limits may also be specified to have the software draw only a portion of the profile. If the Display Settings are set to By Feature and a feature is selected, the Options area of the dialog box is un-ghosted. This is where various labeling options for the profile can be specified. If a value is given for the Strip Grade Increment, elevations will be placed at that increment along the bottom of the profile.

PROFILE REPORT

To review a textual report listing stations and elevations of a ground profile, open the GEOPAK Coordinate Geometry dialog box. A couple of different methods can be used to create this listing for any profile.

By selecting **Element > Profile > Utility** then highlighting a specific profile name and clicking **Print**. The results of this command will be displayed in the Command Output window of the Coordinate Geometry dialog box. To obtain a hardcopy of this information, create an output file using the **File > File Utility > Output** command and supply a filename of 1-5 characters. Then use the **File > File Utility > Print Output File** command to print this to the default system printer.
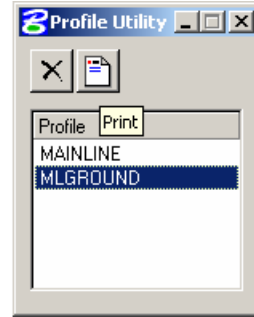
The second method utilizes the COGO Navigator tool to create the **Print Profile** command. The advantage to this method is that multiple profiles can be identified at once and printed to the Command Output window. Start the COGO Navigator by clicking **Tools > Navigator** or selecting the icon from the COGO dialog box. Change the **Element** to **Profile** then highlight the desired profile(s). Once selected, click the **Print/Describe Element** icon on the Navigator to process the profile(s).
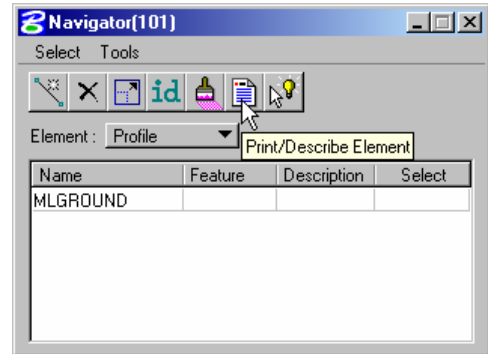
LAB EXERCISE: EXISTING GROUND PROFILES

➢ *Extract an Existing Ground Profile Along the Ditch Alignment*
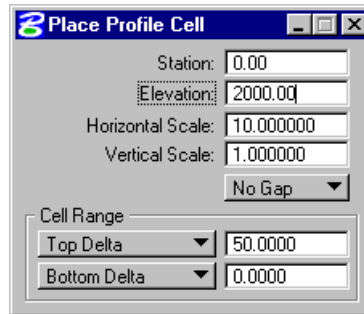
1. Select the **Draw Profile** Tool. *(MS Menu: Applications>GEOPAK ROAD>Plans Preparation>Draw Profiles).*
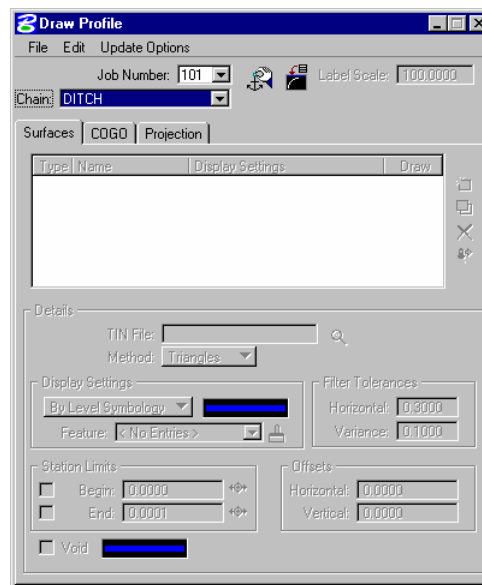
2. Select chain **DITCH**.

3. Click **Dialog Profile Cell Control** icon .

4. Set the **Active Chain** to **DITCH**

5. Click **Place Profile Cell** on the right side of the dialog and populate it as follows:

6. Locate the profile cell by giving a data point in a clear area of the design file.

7. From the Surfaces tab, select the **Browse TIN File** button in the **Details** area, then select the **SURVEY.TIN** file for the **TIN File** to cut the profile.

8. Set the **Method** to **Triangles**.

9. Set the option button to **By Feature** in the **Display Settings** area of the dialog.

10. Click the **Browser Feature** icon then navigate to the item shown below.
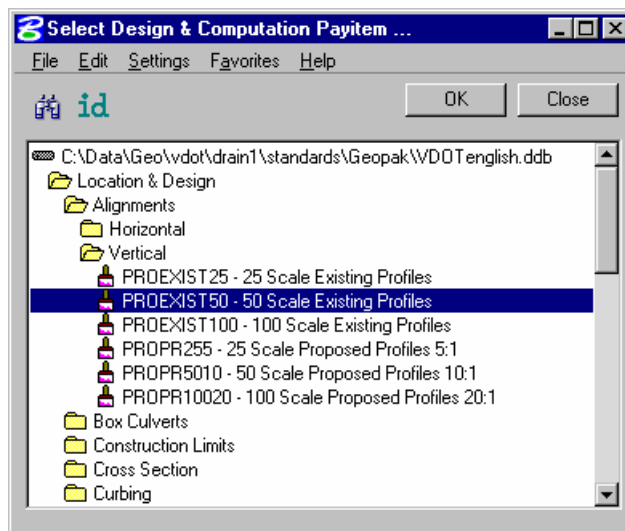
11. Click **OK.**

12. Click **Add Surface Settings** to add the profile to the list box as well as draw the profile into the MicroStation file.
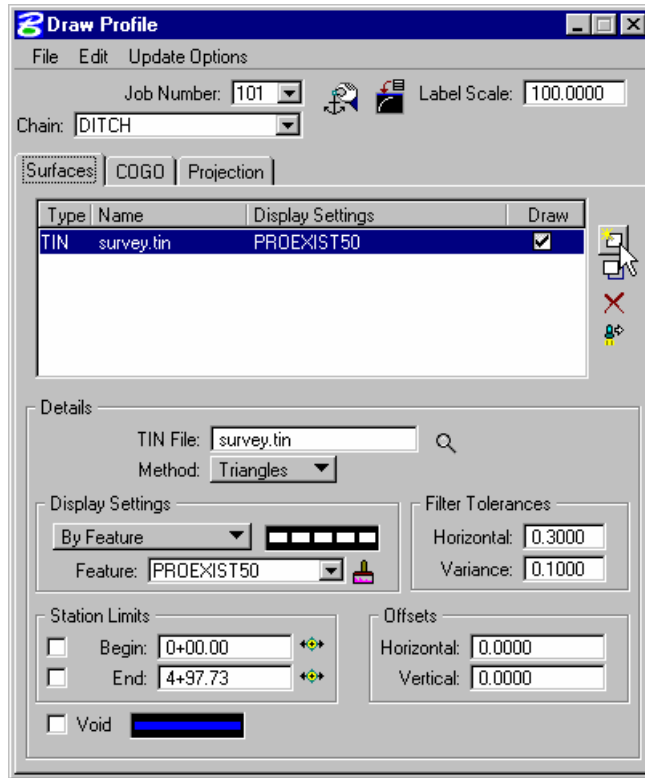
> ➢ *Store the Existing Ditch Profile into the COGO Database*

1. Click **Store Surface** in the **Draw Profile** dialog.

2. Populate the dialog as shown below.

3. Click **Apply** to store the EX_DITCH profile into the gpk database.

4. Close the **Store Profile** and **Draw Profile** dialogs when done.

PROPOSED PROFILES : VERTICAL ALIGNMENT GENERATOR

The Vertical Alignment Generator is a GEOPAK tool that can graphically create and modify proposed design profiles or modify an existing ground profile. These operations may be accomplished through a dialog box and/or by dynamic manipulation of graphic elements.

The Vertical Alignment Generator can be accessed by selecting Applications > GEOPAK ROAD > Geometry > Layout Profiles (VPI Based). It can also be invoked from Project Manager by clicking the Vertical Alignment button or by selecting the VPI Based Vertical Alignment Design Tools icon from the GEOPAK ROAD tool frame.

When selecting the Vertical Alignment generator, the first dialog 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 OK is clicked, the Vertical Profile Generator dialog box appears.
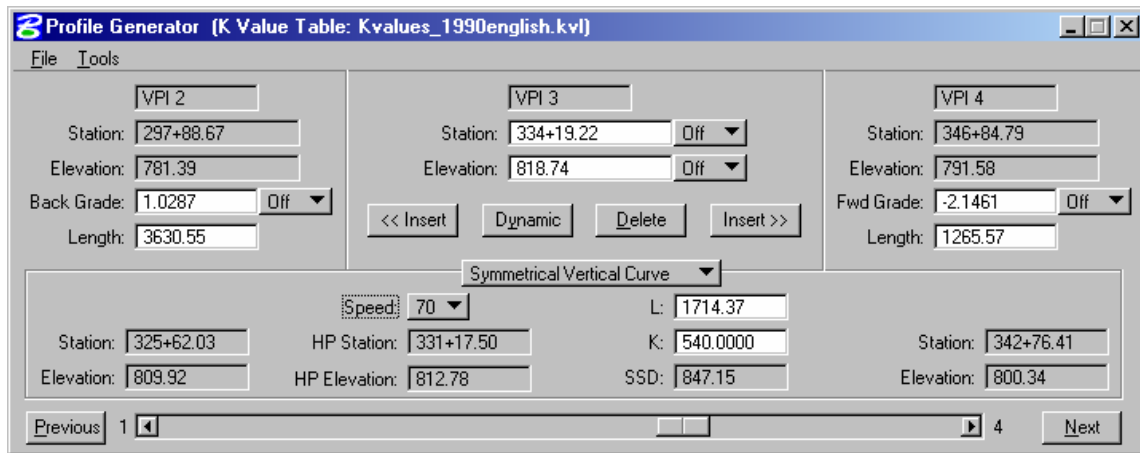
Note   This tool also utilizes the cell that was drawn in the previous exercise by using the **Identify Cell** button.
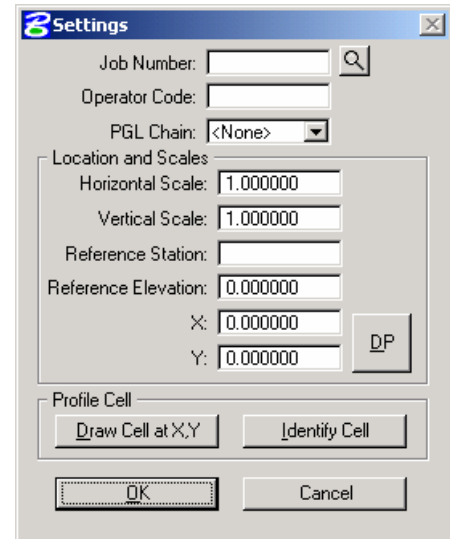
USING THE VERTICAL ALIGNMENT GENERATOR

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 throughout the vertical alignment design.
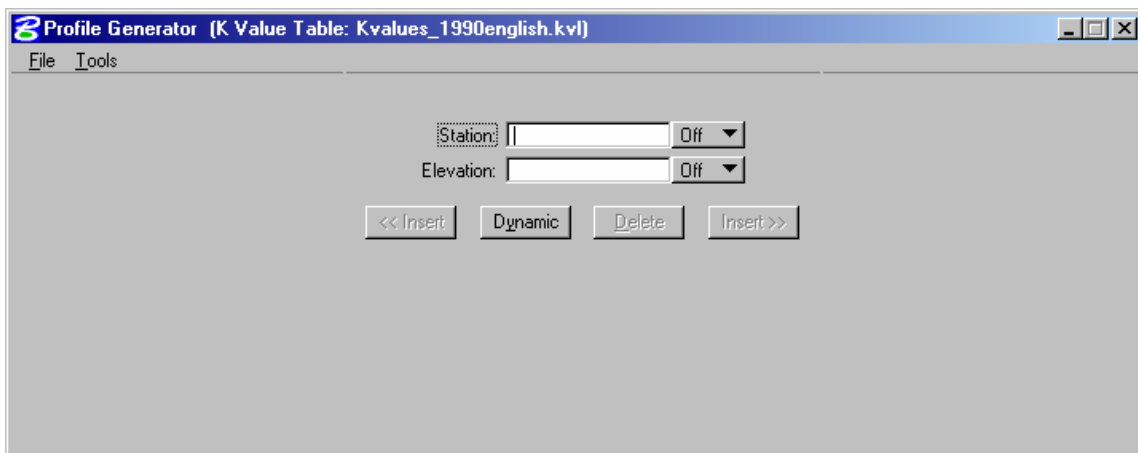
Various design parameters must be defined prior to designing a new profile; which are discussed in the options supported under the two menu options; **File** and **Tools**.

FILE

Preferences – Elevation, length, grade increment and vertical curve preferences.

Profile Cell Settings – contains the profile cell definition.

K Value Table – defines the stopping sight distance K-values for crest and sag conditions at various design speeds.

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

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

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

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

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

Exit - ends the process.

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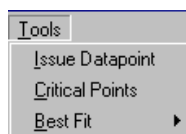
Issue Data Point - Permits you to type in stations and elevations, issue a data point that can be part of a MicroStation place line, place a cell or perform other generic operations. This is useful in displaying visual references within the profile that need to be considered in design of the vertical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

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.

Best Fit – Allows you to best fit a line, parabola (vertical curve), or profile through a series of existing VPI's.

CREATING A NEW PROFILE

> *Warning*   The enter/tab key must be used to ensure values are accepted.

A new profile can be created with the following steps:

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.

    - 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.

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.

3. Define remaining VPI's and Grades.

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

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.

5. Adjust 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.

6. Save the Profile.

    Select **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.

- **LCK** (Locked) - Forces selected operations to maintain the designated profile parameters.

LAB EXERCISE : VERTICAL ALIGNMENT GENERATOR

➢ *Creating a Proposed Ditch Profile*

1. Select the Vertical Alignment Generator dialog. *(MS Menu: Applications > GEOPAK Road > Geometry > Layout Profiles (VPI Based))*.

2. Click the **Identify Cell** button and select the previously plotted profile cell. This will automatically populate the **Settings** dialog as shown below.
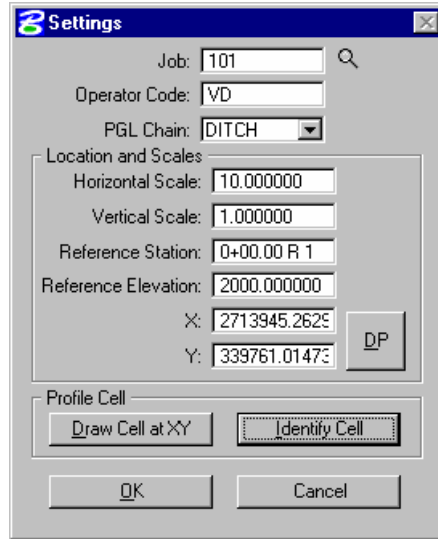
3. Click **OK**. This will invoke the **Profile Generator** dialog.

4. Provide the initial VPI Station = **0+00** and the initial VPI Elevation = **2033.50** as shown in the diagram below. Be sure to press Enter after each entry is made.
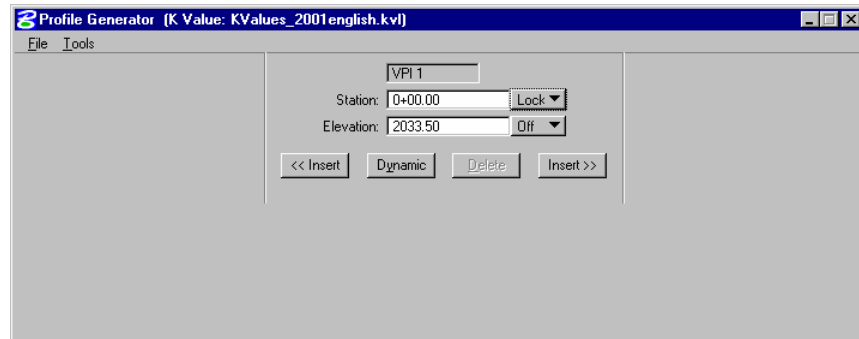
5. Click the **Insert** button and complete the VPI Station and Elevation of the subsequent VPIs as detailed in the table below.

VPI	Station	Elevation
2	2+20	2018.00
3	3+90	2012.05
4	4+95.13	2009.11

6. Click the **Previous** button to return to **VPI 2** as shown in the diagram below. Enter a 200 ft vertical curve.
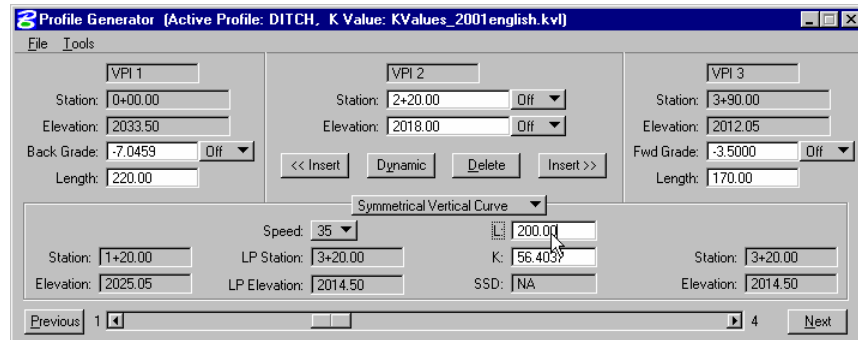
7. Click the **Next** button to move to **VPI 3**. Enter a 100 ft vertical curve.
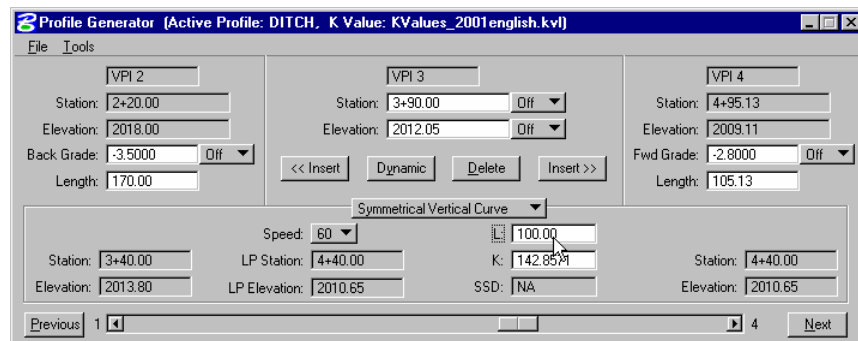
8. Select **File > Save Profile** to save the profile **DITCH**.

9. Close the **Profile Generator** dialog.

➢ *Drawing the Ditch Profile*

1. Select the **Draw Profile** tool. *(MS Menu: Applications>GEOPAK ROAD>Plans Preparation>Draw Profiles).*

2. Click the **COGO** tab then set the **Label Scale** to **50** in the upper right hand corner.

3. Select the **Profile Name** DITCH in the **Details** section.

4. Set the option button to **By Feature** in the **Display Settings** area of the dialog.

5. Click the **Browser Feature** icon then navigate to the item shown below.
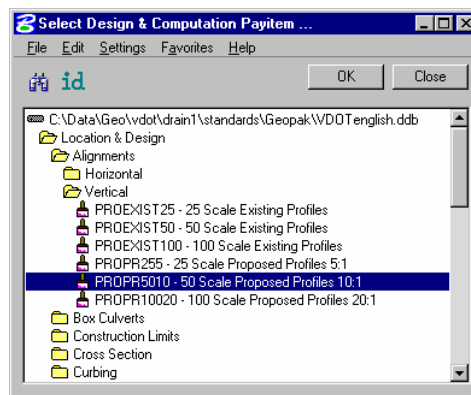
6. Click **OK**.

7. Click **Add COGO Profile Settings** to draw the profile.
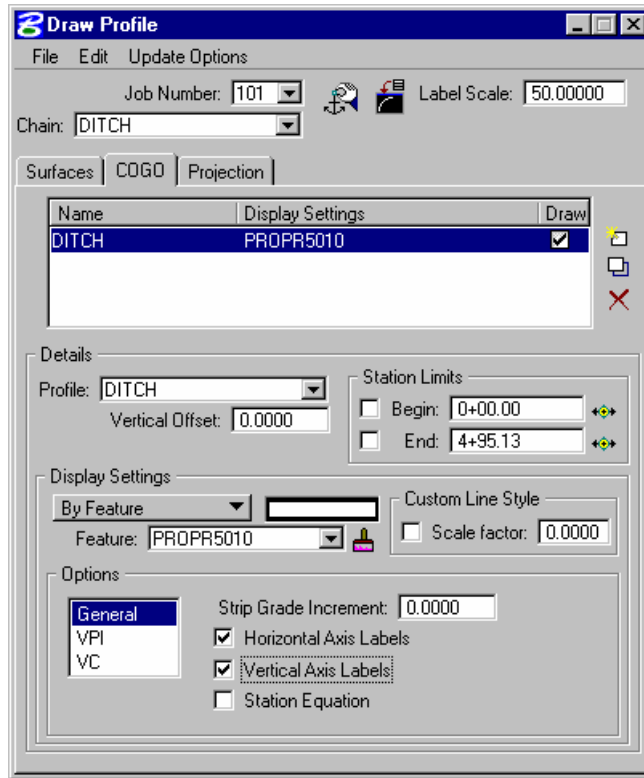
8. Review the proposed ditch profile. This will draw the proposed profile with the proper symbology in addition to labeling the stations.

9. Close the **Draw Profile** dialog.

10. Exit MicroStation.

This page left blank intentionally.

10 Stream Design: From GEOPAK to HEC-Ras

OBJECTIVES

Provide you with a general understanding of GEOPAK Cross Section Reports and their capability to generate geometric information for the HecRas program.

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.

LAB EXERCISE: IMPORTING STREAM GEOMETRIC DATA TO HEC-RAS

➤ *Creating a River Baseline*

1. Execute *c:\data\geo\vdot\drain1\LAB10_V8ad.EXE*

2. Open the MicroStation file, *C:\Data\Geo\VDOT\drain1\Survey_river.dgn*

3. Access **Store Graphics** *(MS Menu: Applications > GEOPAK Road > Geometry > Store Graphics)*

4. Enter the information as shown in the dialog below.

5. Press **ID Element** and select each of the lines that represent the alignment of the river stream until the **Store** button activates.

> *Note*   Remember to consider that for HEC-RAS the highest station value of a section is the upstream point of the channel. In our example, the river flows from south to north.

6. Click on the **Store** button.

7. Click **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 j101ojd.inp.

8. Close the Store Graphics dialog and delete the MicroStation lines that represented the stream alignment.

➤ *Drawing the River Alignment*

1. Access the Design and Computation Manager *(Applications > GEOPAK Road > Design & Computation Manager)*. A secondary dialog box will appear to be used in conjunction with the Design and Computation Manager dialog box.

2. Select the item shown in the dialog box shown below.

3. Click on the **Draw Plan and Profile** button from the secondary dialog box.
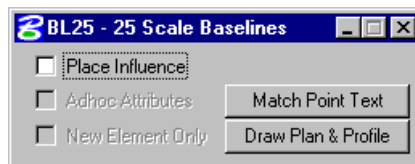
4. Select **Chains** operation as the Element Type. Each of the chains will be displayed in the list box as shown. Notice that the options that are to be drawn with the chain are already activated. Set the Label Scale to **25**.
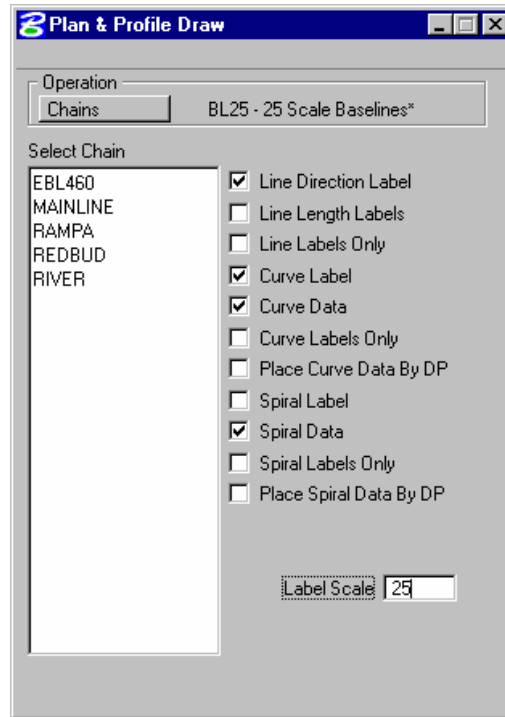
5. To draw the chain, click on the name of the chain: **River** in the list box. Only click once!! Each time you ID a chain it will be plotted into the design file.

6. Change the Element Type to **Stationing** and modify the dialog settings as shown.

7. To draw the stationing for the chain, click on the name of the chain: **River** in the list box.

8. Exit the Plan and Profile Draw dialog box and close the D&C Manager.

> **Existing Ground Cross Sections: Drawing Pattern Lines**

1. Access **Draw Patterns** *(Applications > GEOPAK Road > Cross Sections > Draw Patterns by Station Range).*
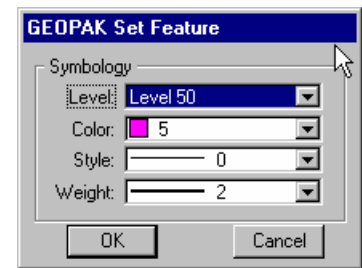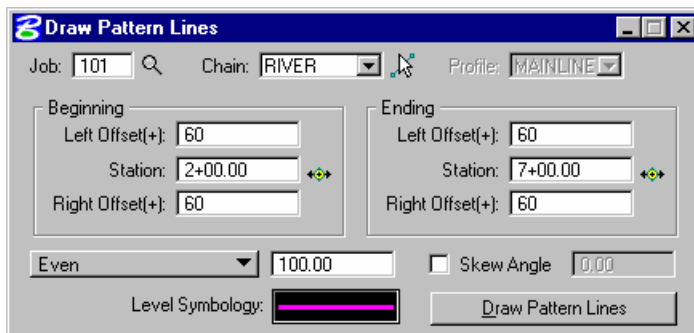
2. Click **Draw Pattern Lines** to initiate the plotting of the patterns into the design file.

3. Adjust the length of the pattern lines, using MicroStation commands, at both ends of the bridge just enough to cover the channel section.

Note   The designer has the option to add, delete or modify patterns lines manually using the Place Line command in MicroStation. GEOPAK identifies any line as a pattern line for cross section plotting by searching for user specified CADD symbology.

4. Close the Draw Pattern Lines dialog box.

> **Existing Ground Cross Sections: Generating the Stream Cross Sections**

1. Open MicroStation file *c:\data\geo\VDOT\drain1\xsriver.dgn*

2. Access **Draw Cross Sections from Surfaces** *(Applications > GEOPAK Road > Cross Sections).*

3. Select the job number: **101** and chain name: **RIVER** in the Draw Cross Sections dialog.

4. Generate the existing ground cross-sections by populating the dialog boxes as shown below.
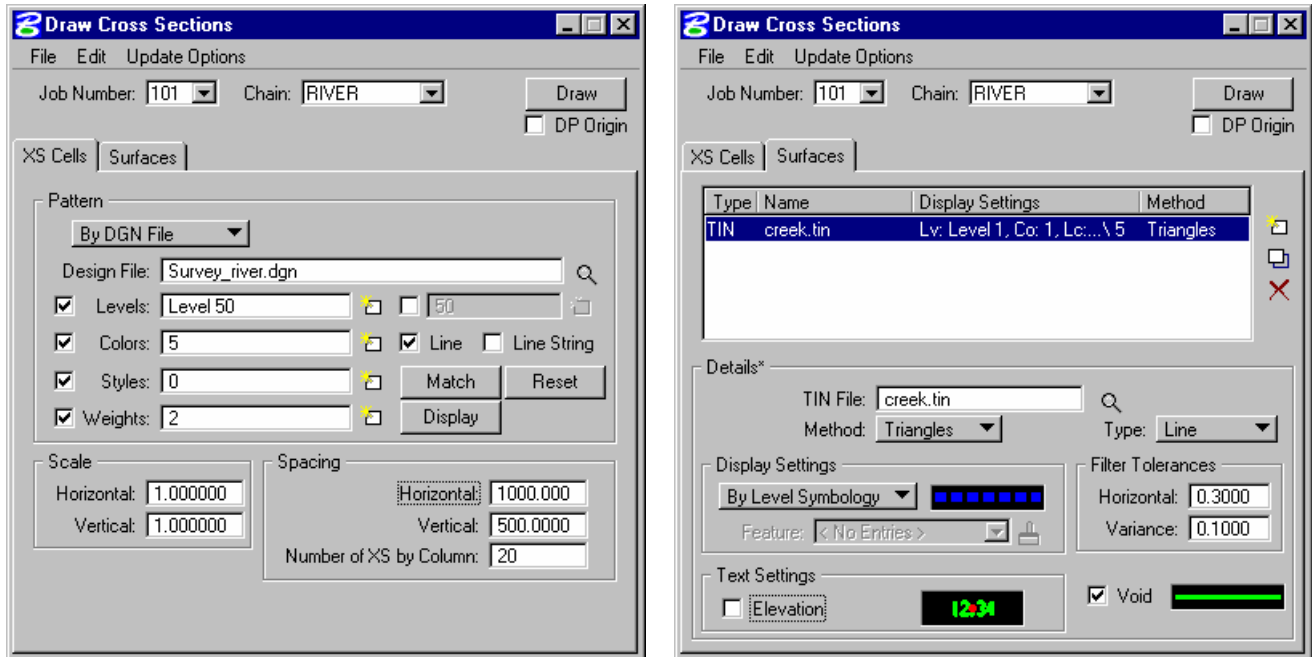
Note   The **Display Settings** for the existing ground line are set to level 1, color 1, style 2, weight 5
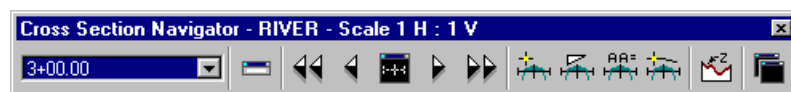
5. Click **DRAW** to generate the cross sections.
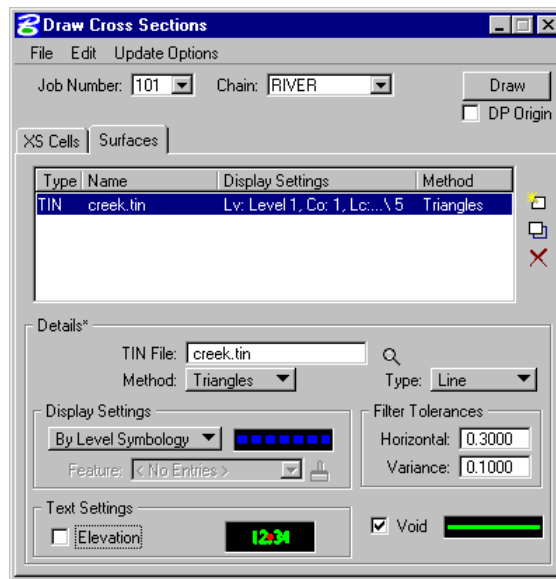
6. Exit the Draw Cross Sections dialog box.
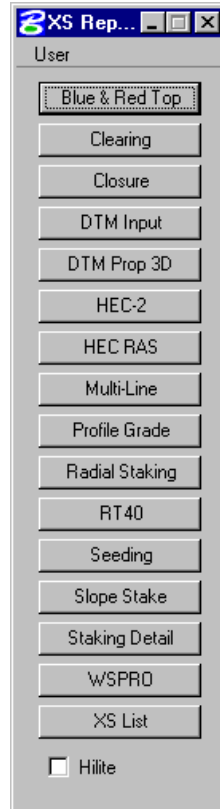
> *Review the Cross Sections*

1. Select the **Cross Section Navigator** tool. *(Applications > GEOPAK Road > Cross Sections > Navigator)*
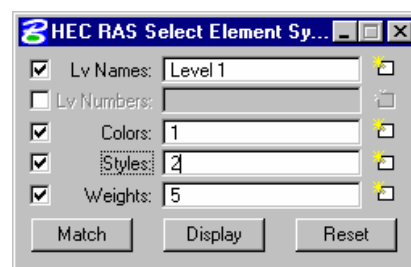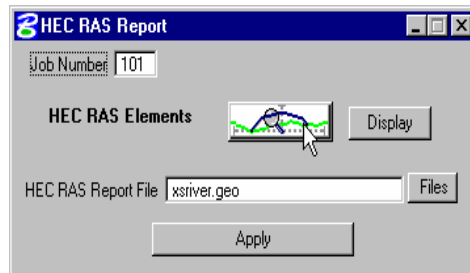
2. Use the **Cross Section Navigator** to browse and check your existing cross sections.

3. Close the XS Navigator dialog box

> **> *Exporting to Hec-Ras***

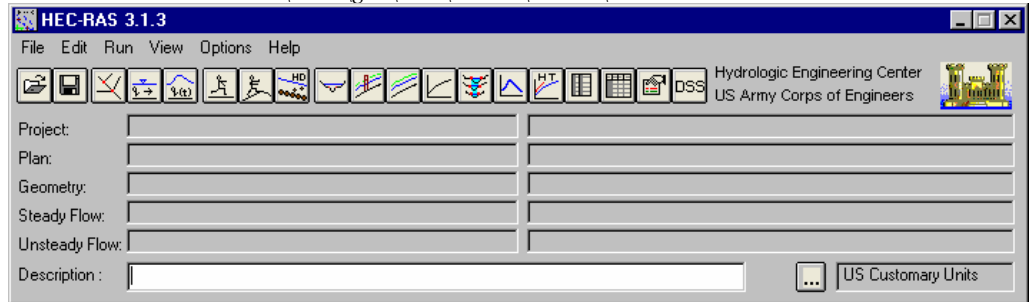1. Access the **HEC-RAS** report *(Applications > GEOPAK Road > Cross Sections > Reports)*

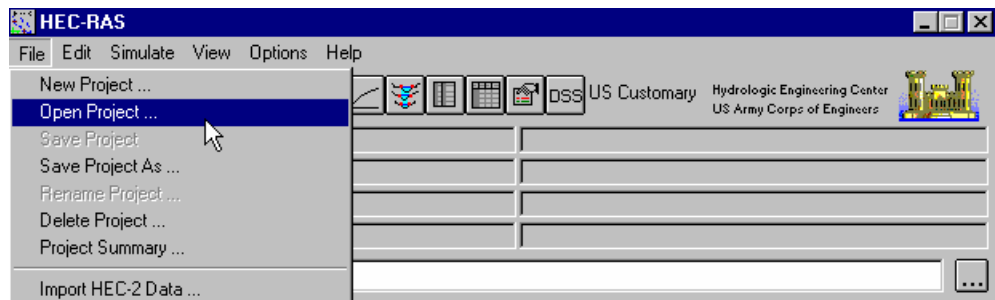2. Populate the dialog box as shown below.

3. Click **Apply** to generate the Hec-Ras cross section geometry file.

4. Exit MicroStation.

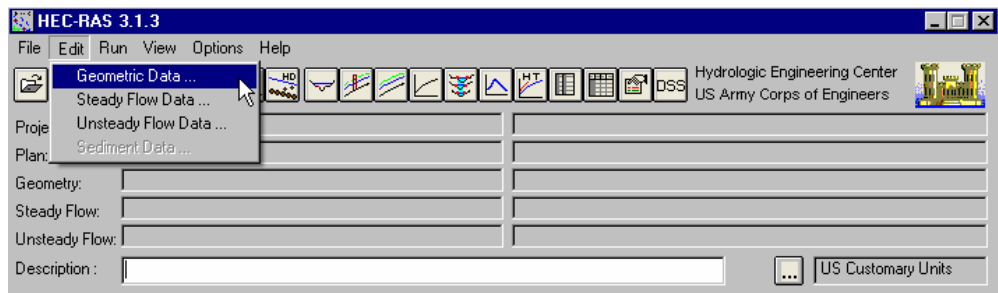> **>** ***Importing Cross Section Data into Hec-Ras***

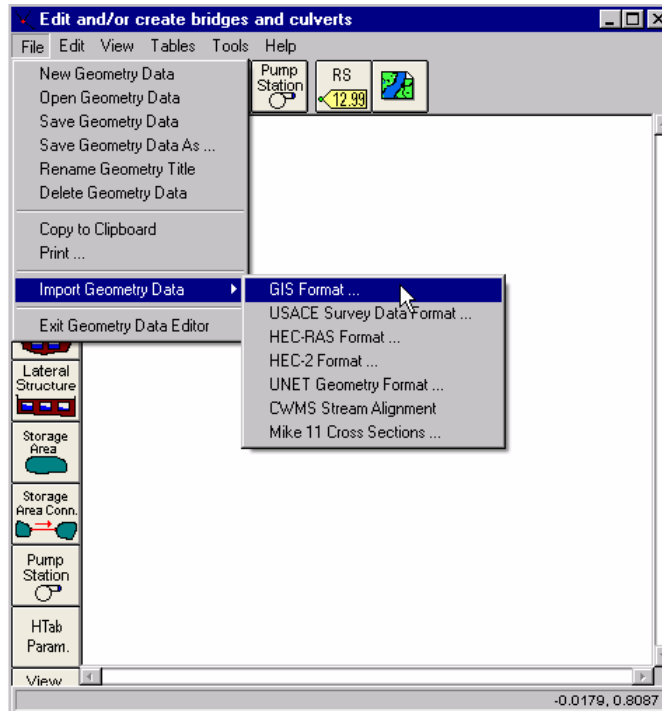1. Activate Hec-Ras from *c:\ data\geo\vdot\drain1\hecras\ras.exe.*

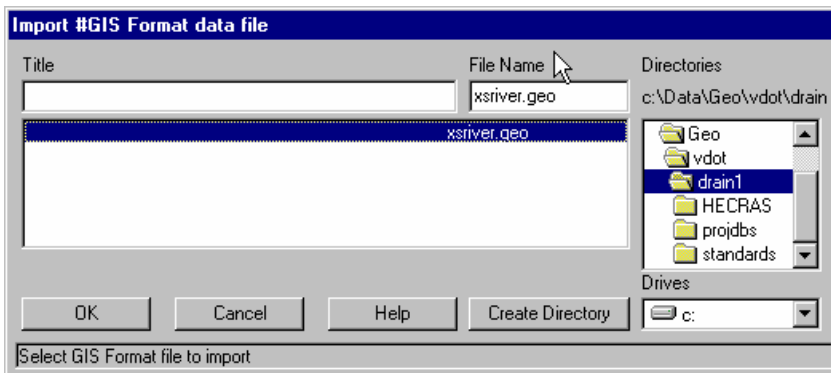2. Access the included Hec-Ras project: *xsriver.prj* located in the *c:\data\geo\vdot\drain1*

3. Access the Geometric data module of Hec-Ras by selecting **Edit > Geometric Data** from the Hec-Ras pull down menu.
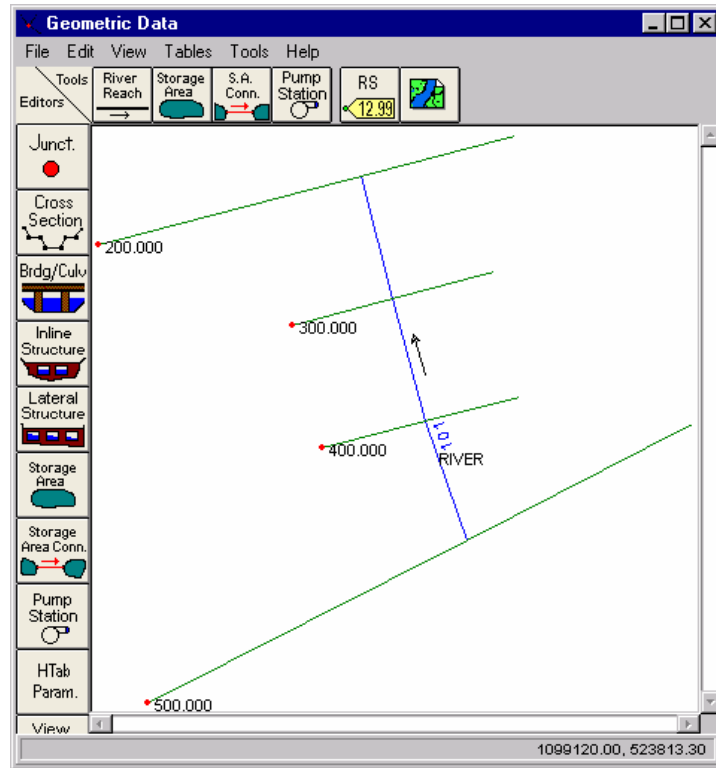
4. Import the cross section data generated by GEOPAK. *(File > Import Geometry Data > GIS Format)*

5. Select the file we created from the GEOPAK cross sections:

6. Review the alignment and layout of the imported cross section data.

7. Review the generated cross sections by clicking on the **Cross Section** button in the Geometric Data dialog box. Click on the **Expand XS Editor** button to see a graphical representation of the cross section data.
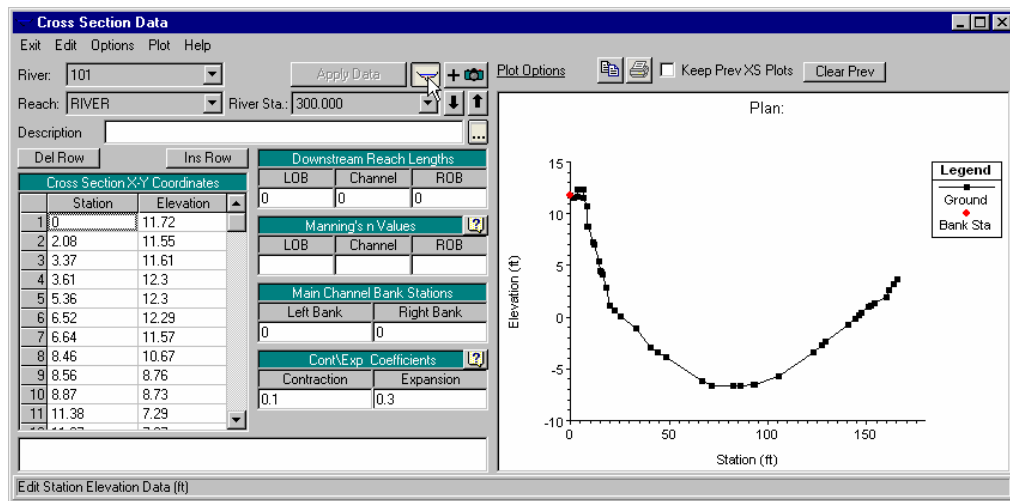
8. Save your project data and exit Hec-Ras.

This page left blank intentionally.

11 Site Design

OBJECTIVES

- 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.

- Design of a Pond and Channel

- Review Pond Routing Calculations.

- Review a procedure for Flood Plain Determination

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 performs 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, or 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 mode
- 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
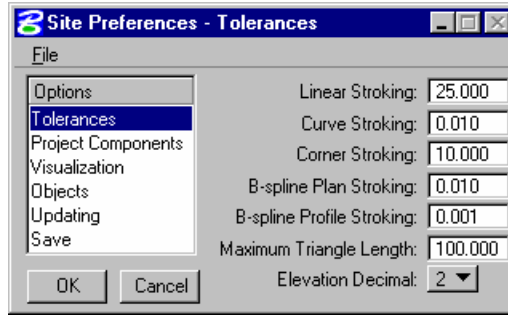

Option. As different options are selected in the list box, the right side of the dialog changes dynamically to reflect the parameters for highlighted Option.

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
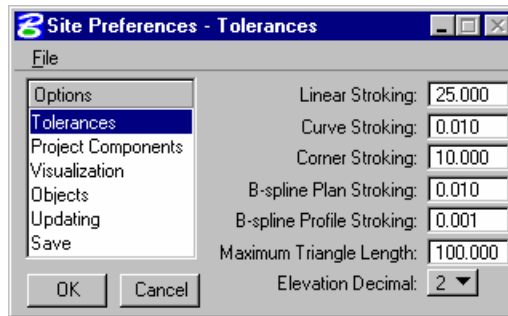
Linear Stroking	The Linear Stroking option densifies portions of linear elements and is utilized to interpolate new spots from the linear elements.
Curve Stroking	The Curve Stroking feature densifies the curved portions of the break lines. Curved break lines in GEOPAK are handled by segmenting the curve 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.
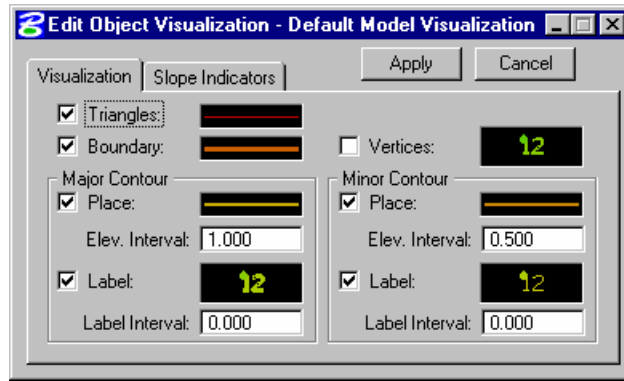
VISUALIZATION

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.
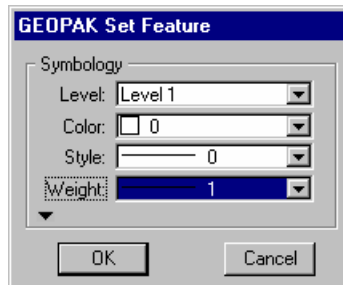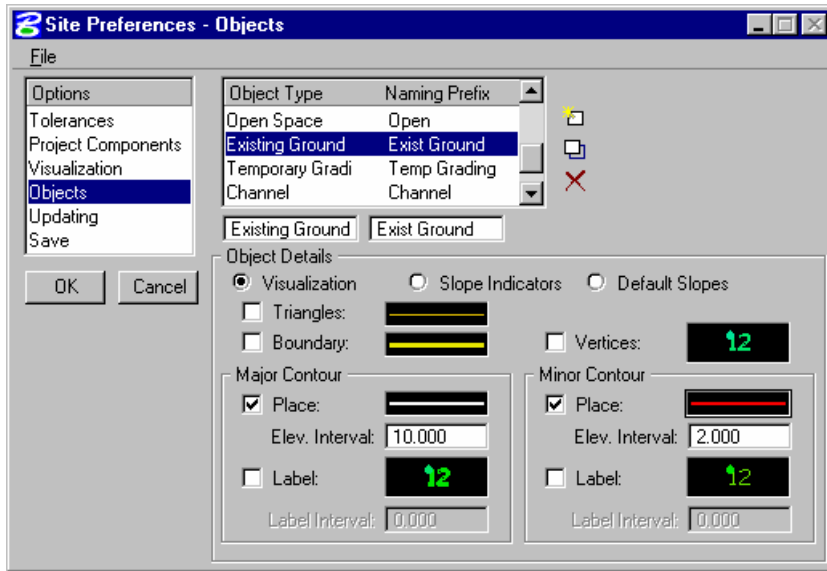

Additional options can be invoked by clicking the right mouse button over these sample controls.
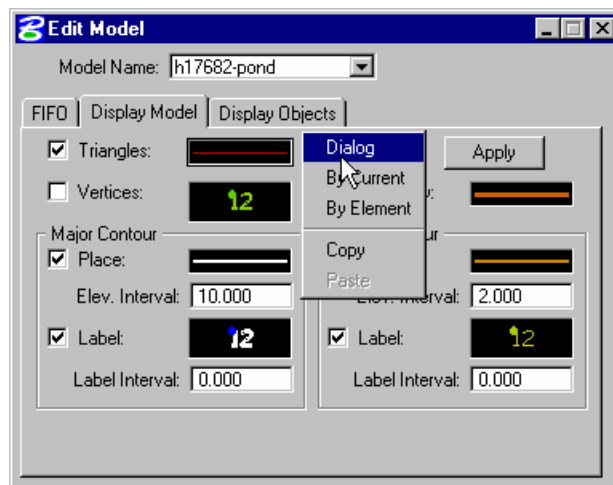
OBJECTS

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 you to set Visualization of the highlighted Object Type. Each Object Type may have its own settings, providing maximum flexibility for you 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.
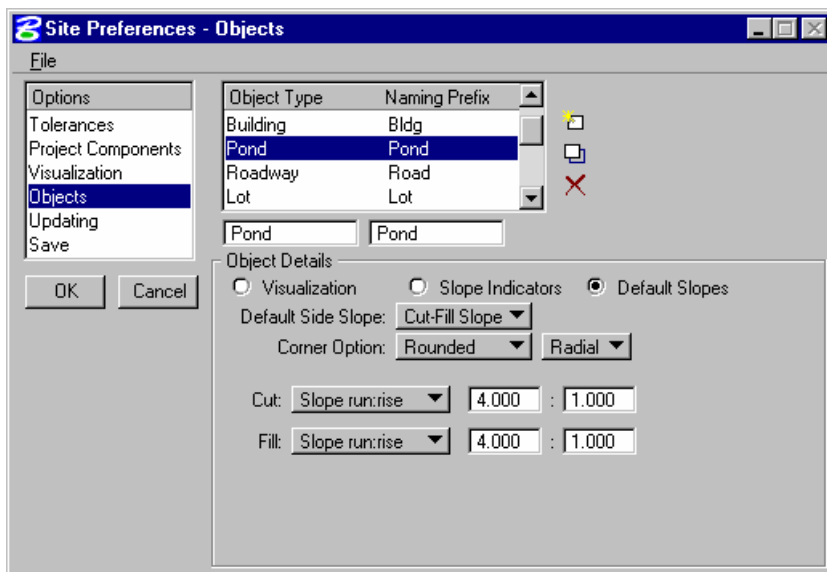
As new Objects of a particular type are created they will obtain the default settings. Note that an Object slopes can be modified at any time, these defaults are applied when the object is created and remain in effect until the slopes are modified while editing the Object. Slopes are applied around the

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.
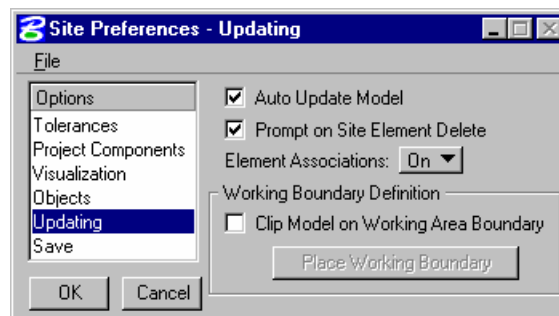
When the Auto Update Model is active, the model is updated whenever a change is made to an element, object, or the FIFO of the Active model. As Models get larger, it may not be necessary to update the Model on every single edit. Deactivating this speeds up the modifications that are being performed, 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. Clearing the toggle (OFF) disables the associations.

SAVE

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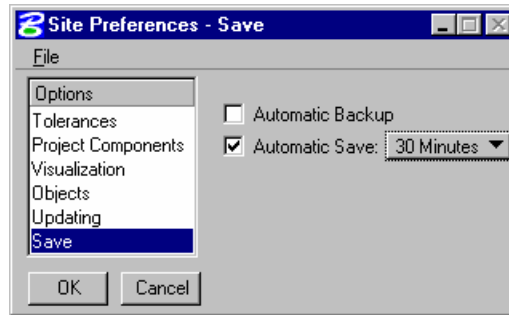
When the Automatic Backup option is activated, a backup file is always created (in the project directory with a *.bak extension) when the project is opened.

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.
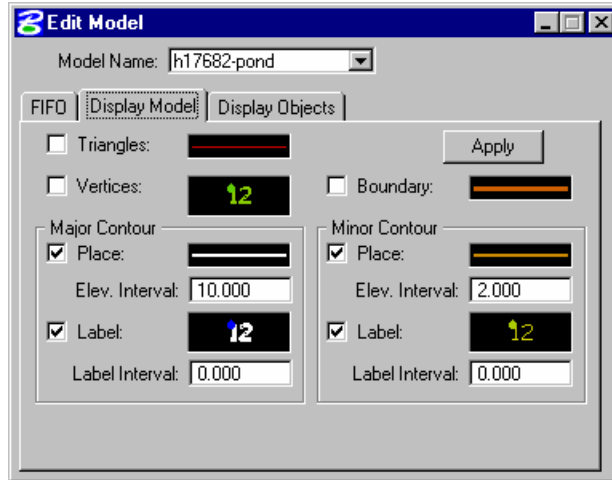
DISPLAY OBJECTS TAB

When the Display Objects Tab is selected, the dialog depicted below is displayed.
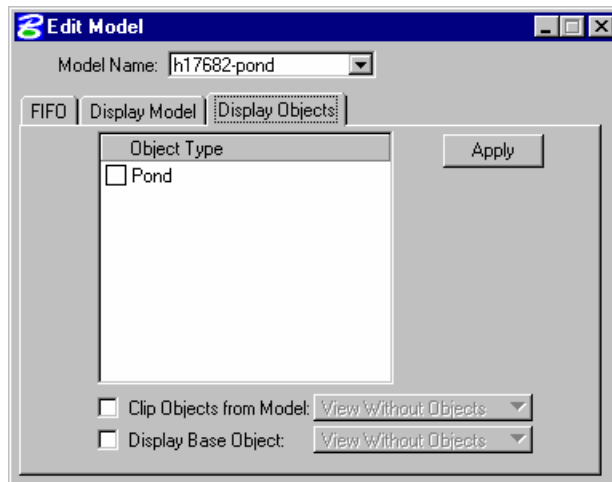
Object Type	List box containing all current Object Types in the current model. If the eye icon is displayed to the left of the Object Type, then all objects of the specified type are displayed with their visualization settings.
Clip Objects from	When activated, this clips every object out of the model prior to display.

Model	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.
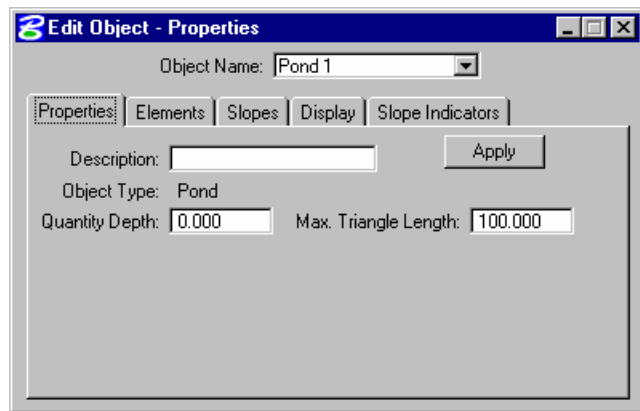
OBJECT PROPERTIES

Under the Properties Tab (shown above) 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.

OBJECT ELEMENTS TAB

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.
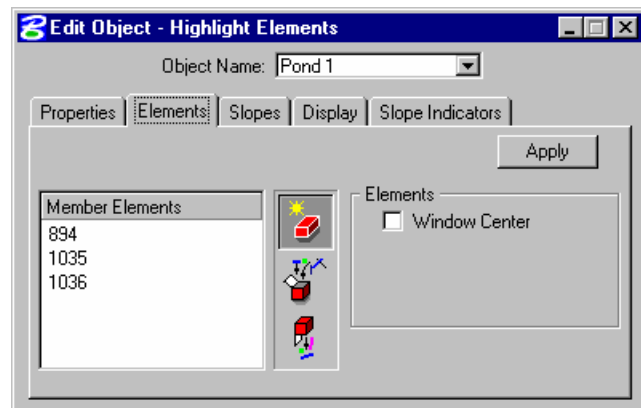
OBJECT SLOPES TAB

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.
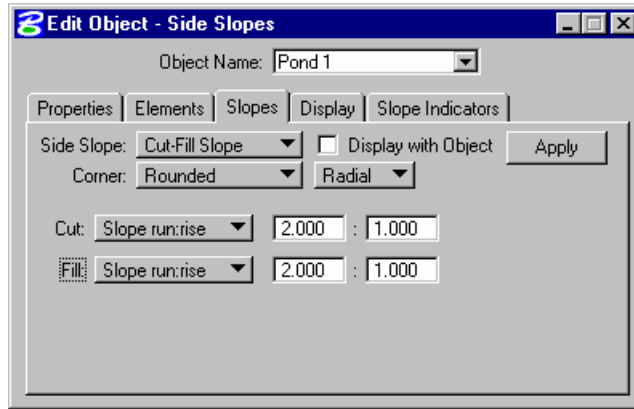
There are four slopes options available: No Slopes, Cut - Fill Slope, Cut-Fill Table, and Dynamic Slopes. Corner Options can be specified to generate either Rounded or Straight slopes around corners of Objects.

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.
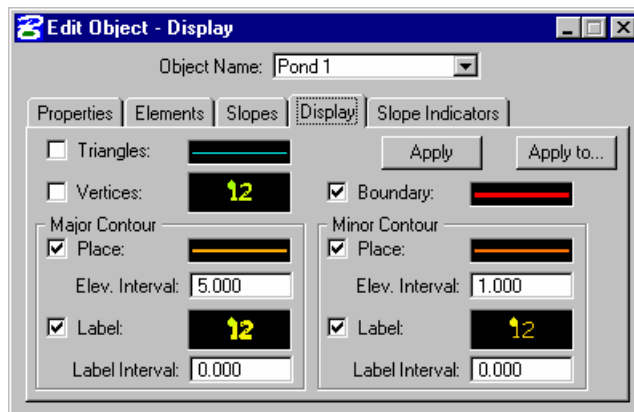
SLOPE INDICATORS TAB

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.
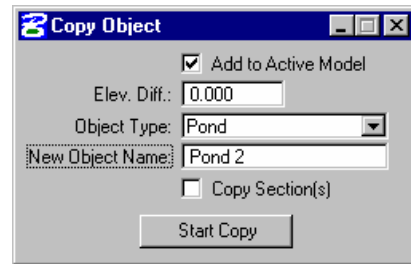
> *Note*   When the Make Copy toggle is not activated, the dialog dynamically changes to a Move Object dialog.

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.
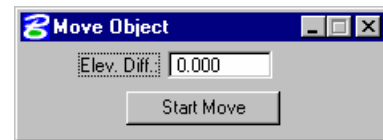
RAISE/LOWER OBJECT

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.
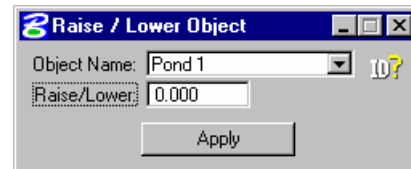
DELETE OBJECT

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.
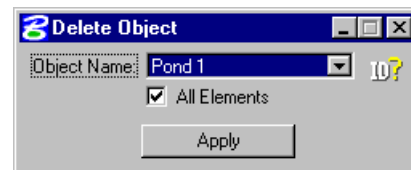
ELEMENTS MENU

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.
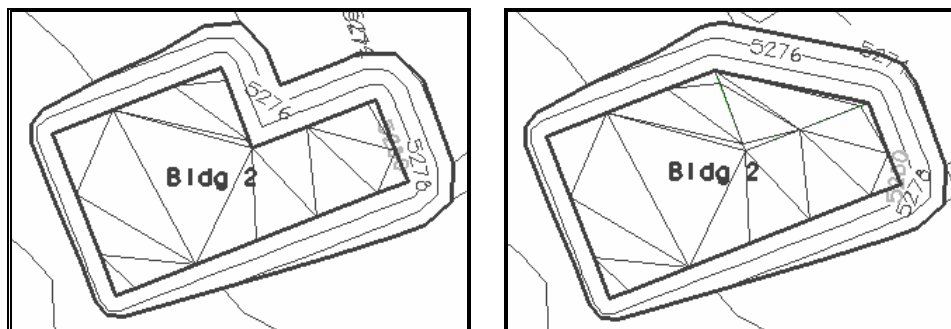
Feature types can be modified at any point in the design to create the desired results. It is often desirable to leave features as break lines until the object design is near completion and then the desired elements can be specified as boundaries.

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.
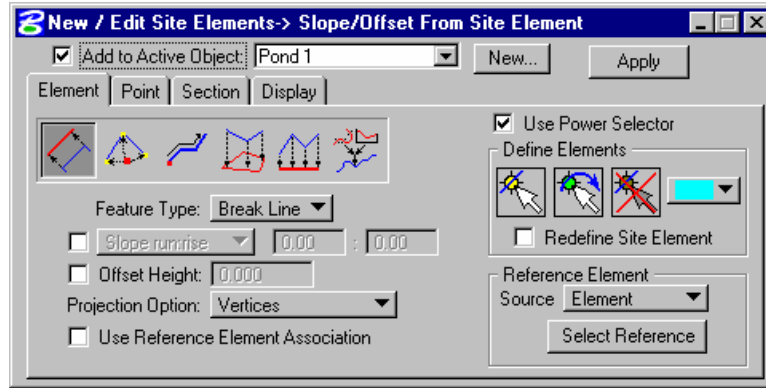
The dialog contains five main options: Element, Point, Section, Side Slope and Display. When a tab is selected, the dialog dynamically changes to reflect the selection. The fields below are displayed regardless of which tab is selected.

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.
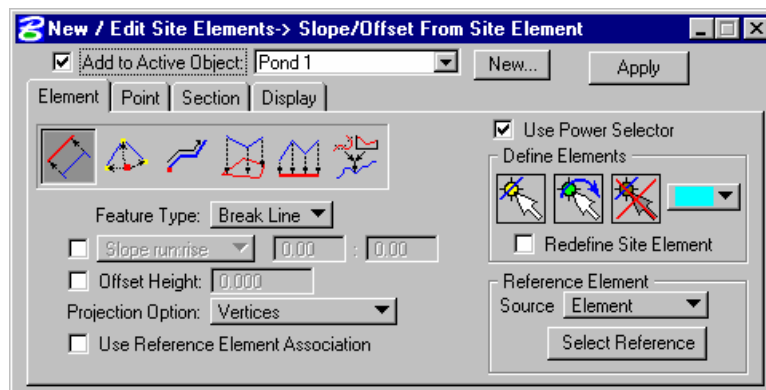
Six tools to assign or modify elevations are provided. As each option is selected, the title bar as well as the left side of the dialog dynamically changes to reflect the selection. These options include:

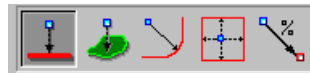|	Slope/Offset from Site Elements	The Slope/Offset from Site Element option is used for assigning elevation to Site Elements by placing the elements at an optional slope and offset height from existing Site Elements. In calculating the elevation from the existing Site Element, a projection is performed 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.
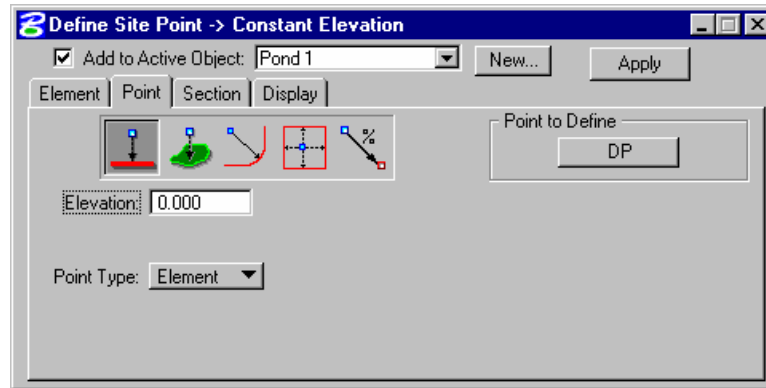
Point elements can be created and added to Objects to refine the surfaces or edit the vertices of existing Site Elements. When placing points, the Site Modeler can detect when an existing Site Element is snapped to and rather than placing a new point in the Object, the snapped vertex will be changed 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 changes 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 Point on Model / Object	The Drape on Model / Object option is useful for placing or editing points and placing them at an elevation relative to 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.
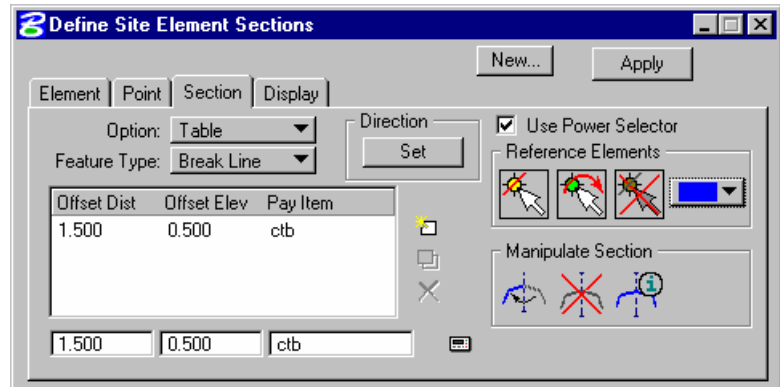
Provisions for assigning sections to multiple Site Elements at once, even disconnected Site Elements, are available. This capability is provided by gathering all the Site Elements selected for section placement and assigning the Direction or which side to place the section element. The Site Modeler may 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.

VDOT GEOPAK Drainage

	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.
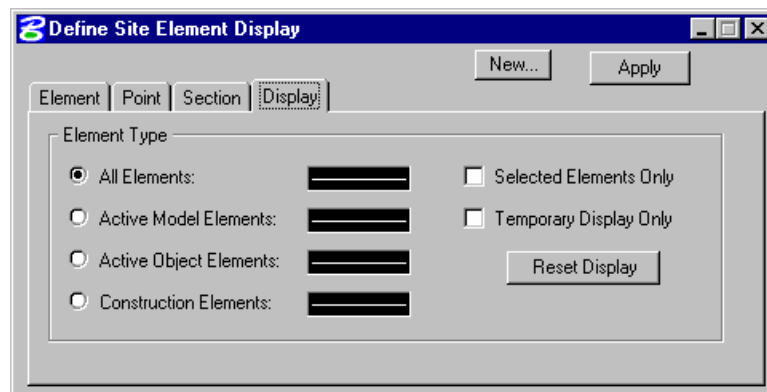
Element symbology is supported in this dialog for the following elements:

- 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.
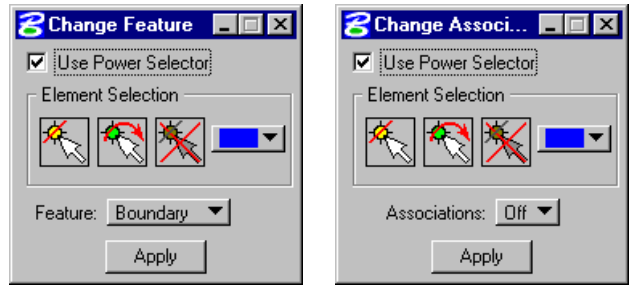
RAISE/LOWER SITE ELEMENTS

This tool is utilized to change the elevation(s) of an existing element.
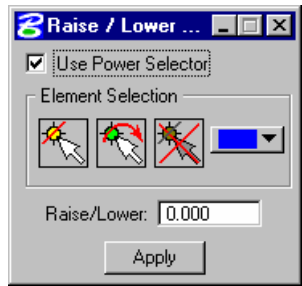
MODIFY SITE ELEMENTS

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.

       VDOT GEOPAK Drainage
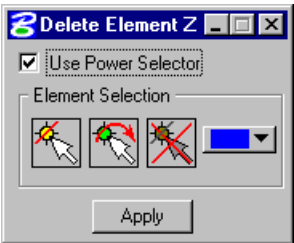
3D ELEMENT

Elements placed at the correct X, Y, and Z coordinates within a MicroStation 3D design file can be utilized within this dialog. When invoked, the dialog depicted below is displayed.
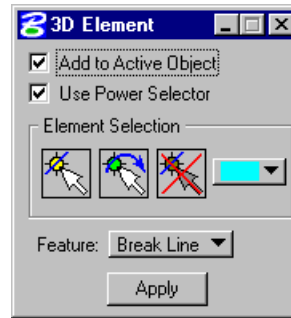
SITE ELEMENT INFORMATION

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.
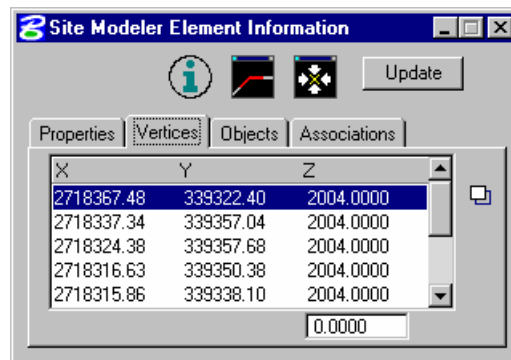
COMPOSITE SECTION TOOL

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.
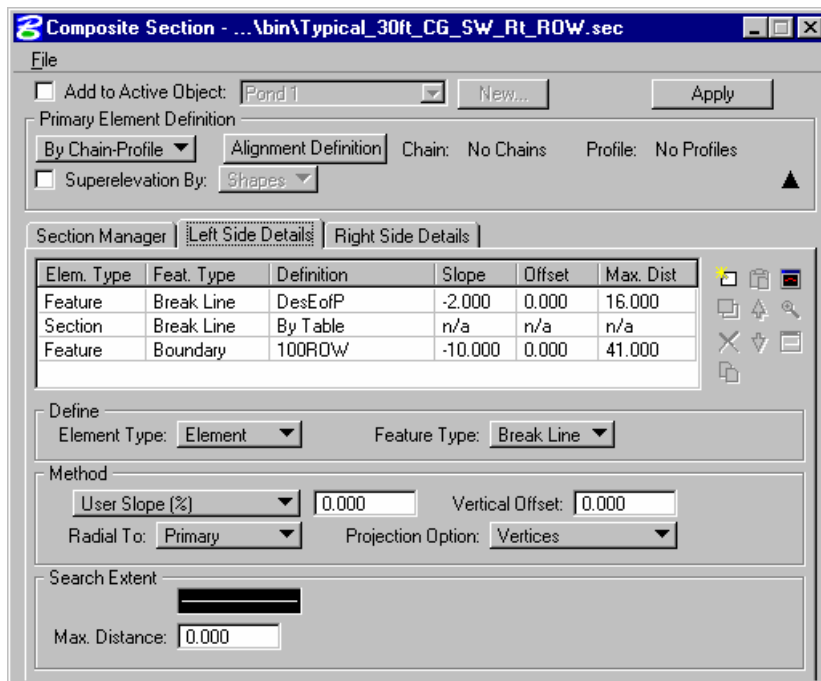
ACTIVE SITE OBJECT CONTROL TOOLBAR

Upon opening an existing GEOPAK Site Project or creation of a new project, the Active Site Object Toolbar is displayed as shown below.
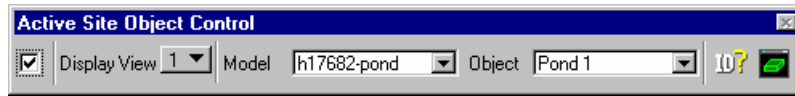
The toolbar is resizable, can be docked and remains open throughout the entire Site Modeler session.

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
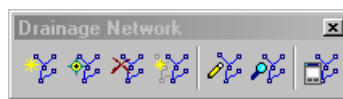- Exit
- Elements tools

- 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.

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

Seven tools are supported in the Network tool box (from left to right).

| Tools	Description
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.
Rename	Utilized to rename any previously defined Network.
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.
Set 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 EXERCISE: POND DESIGN

> ### *Creating a Site Modeler Project*

1. Execute c:*data**geo**VDOT**drain1**LAB11_V8ad.EXE*

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

3. Access the Project Wizard. *(Applications > GEOPAK Site > Site Modeler > Site Modeling)*

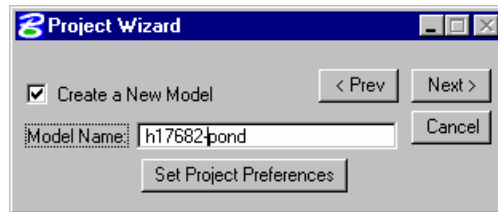4. Select the Create New Project button and then press the **Next>** button to continue.
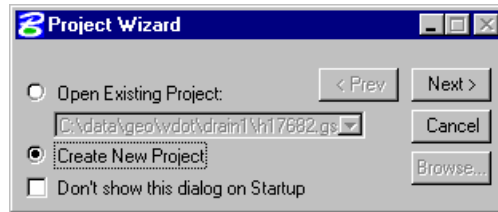
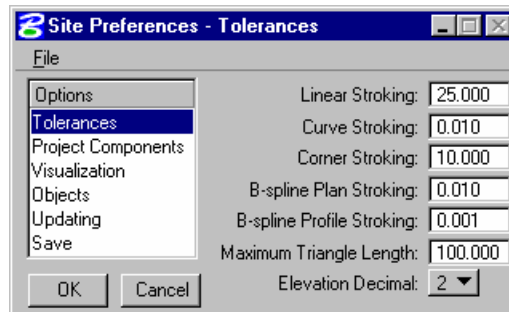5. The Project Wizard dialog will appear filled out as shown. Click the **Next>** button to continue.
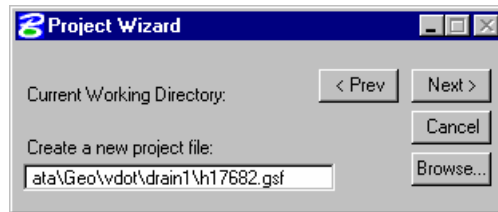
> ### *Defining the Project Preferences*

1. Create a new Model: **h17682-pond.**

2. Click **Set Project Preferences**, and populate the dialog as shown below.

3. Select the **Visualization** item and toggle on the **View Active Object** and **View Active Model**.
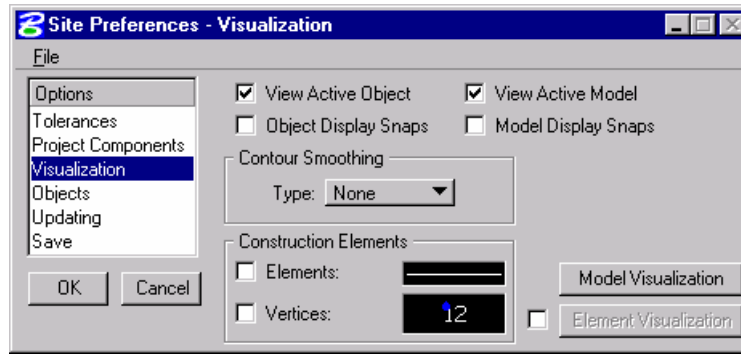
4. Click the **Model Visualization** button in the lower right corner of the **Site Preferences** dialog. The **Edit Object Visualization** dialog will appear.
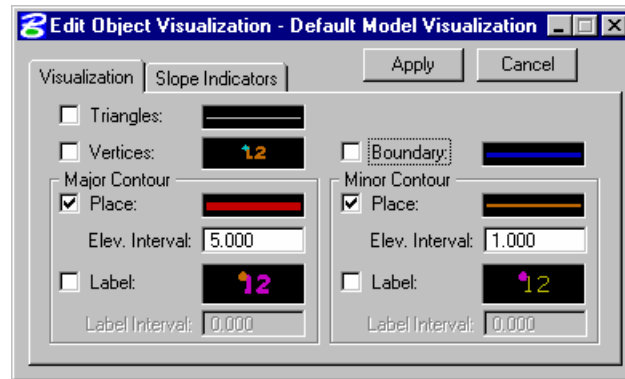
5. Click **Apply** to return to the Site Preferences dialog.

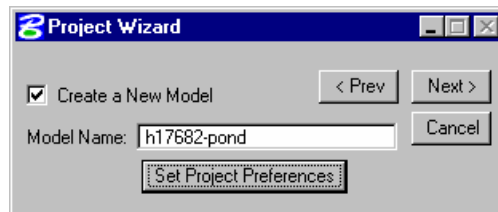6. Select the **Updating** option as shown below.
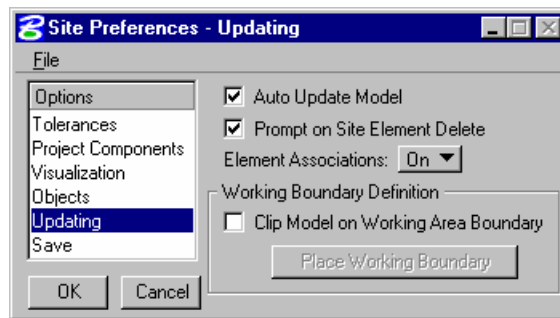
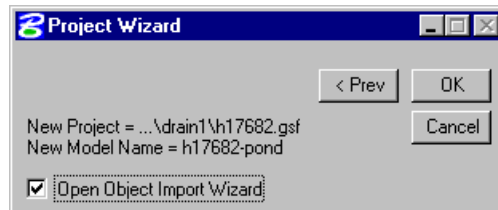7. Select the **Save** option and set the **Automatic Save** to 30 minutes.
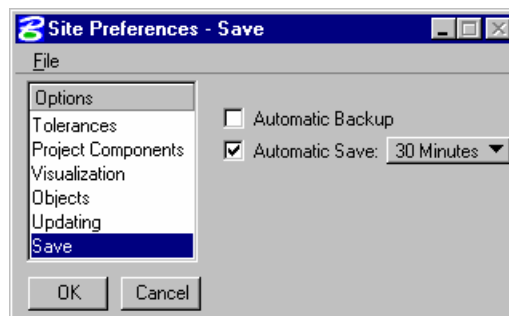
8. Click **OK**.

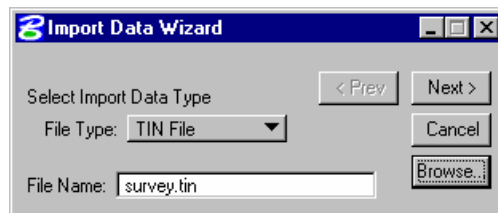> **> *Completing Project Setup***
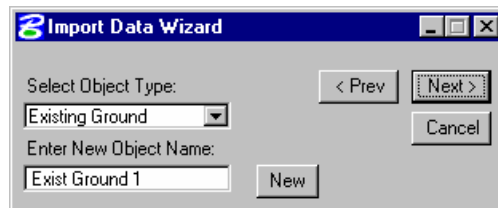
1. Click **Next>**.

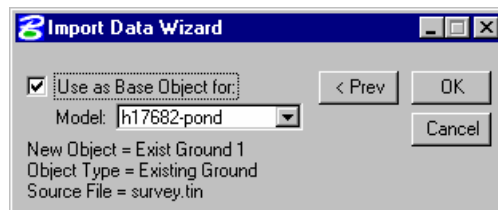2. Toggle On the Open Object Import Wizard and click **OK**.

3. Select Tin File in the Import Data Wizard, and **Browse** for the *survey.tin* file located in *c:\data\geo\VDOT\drain1*

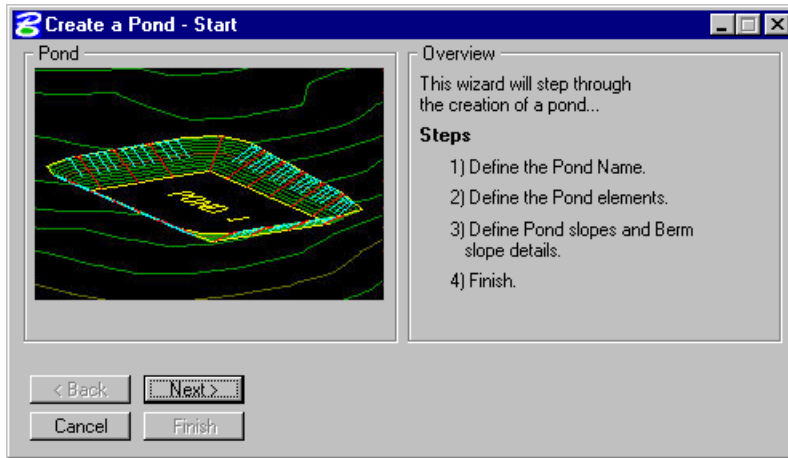4. Click **Next>** and select the Existing Ground object type.

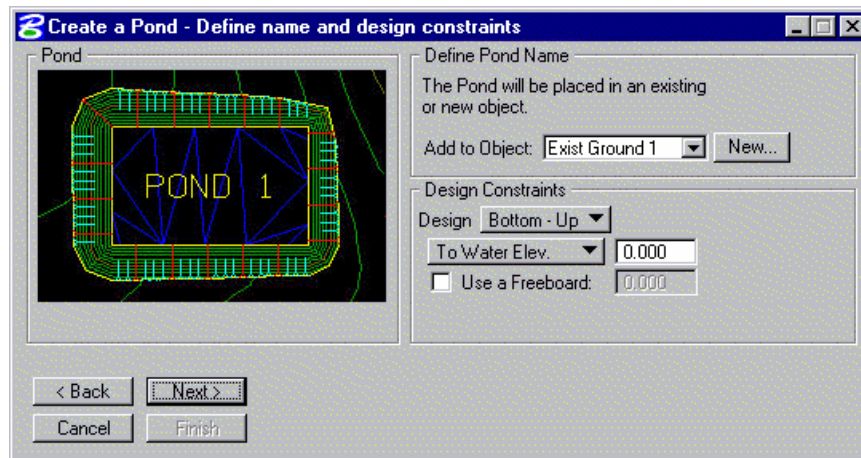5. Click **Next>** and setup the dialog as shown and click **OK**.
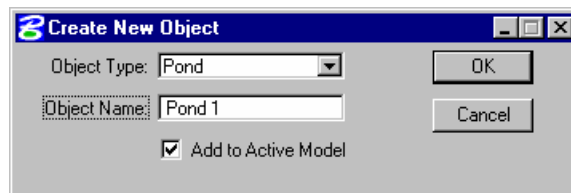
> *Creating the Pond*

1. Access the Pond Design tool. *(MS Menu: Modeler > Tools > Pond Design)*
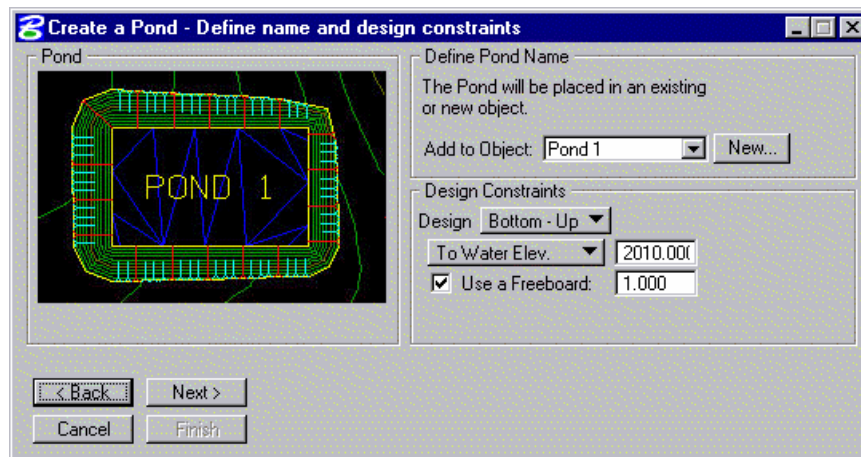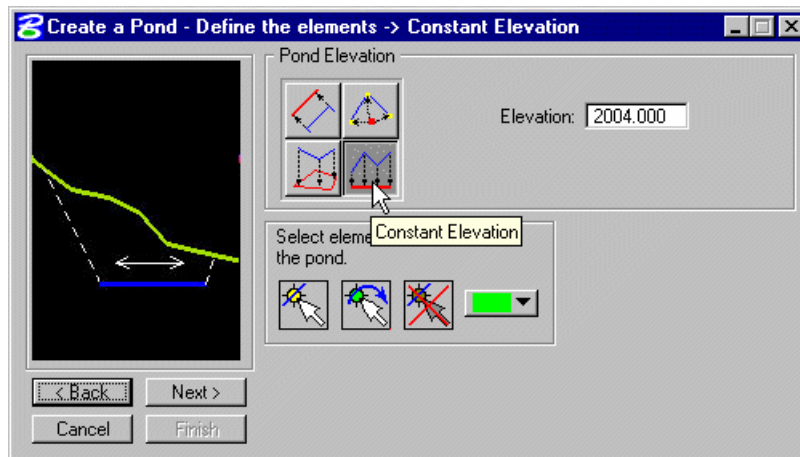

2. Click **Next>**.

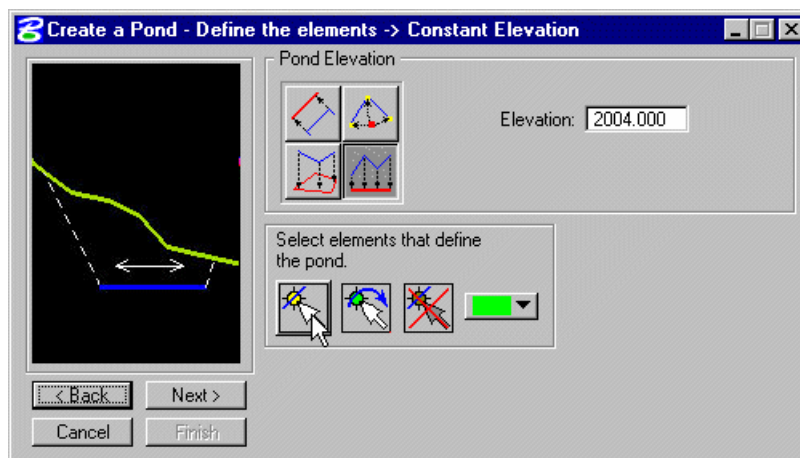3. Click **New** and create a new object Pond1. Click **OK** when done.

4. Setup the dialog as shown below.

5. Click **Next>**.

6. Setup the pond bottom at a constant elevation.

7. Select the MicroStation element representing the pond bottom.

8. Click **Next>**. Then, setup the Pond Side Slopes, Berm and Side Slopes as shown in the dialog below.
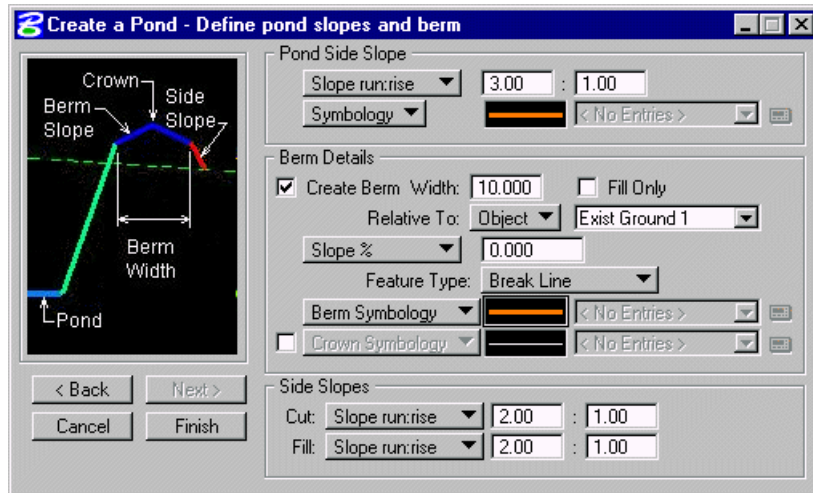
9. Click **Finish** to build the pond.

Note   The pond bottom could be modified using MicroStation commands like: move, modify and scale. This will recompute new elevations and adjust the slopes based on the original pond configuration.

> *Computing the Pond Volume*

1. Access the **GEOPAK Volume Calculations** tool. *(MS Menu > Modeler > Analysis > Volumes).*
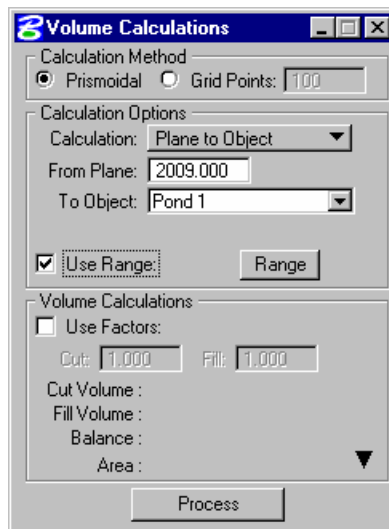
2. Click the **Range** button.

3. Set the **Volume Elevation Range** as shown below.
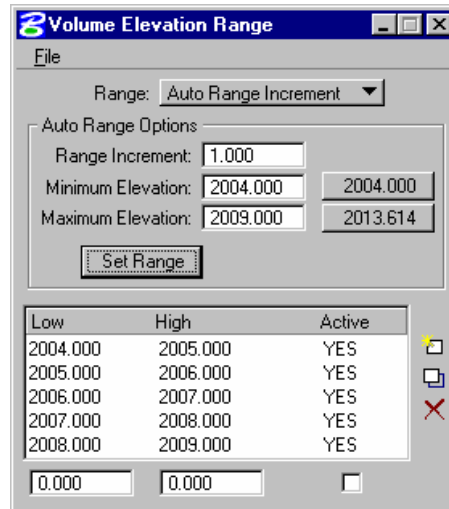
4. Click the **Process** button on the **Volumes Calculation** dialog.
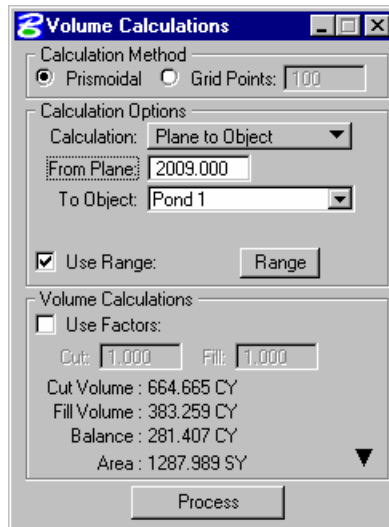
5. The following report is generated.

6. Close the Volume Calculations dialog when done.

LAB EXERCISE : POND VISUALIZATION FOR PLANS

Adjusting the Existing Ground Level Symbology

1. Select the Edit Object Display tool. *(MS Menu: Modeler > Object > Edit)*

2. Select the **Object Name :** Exist Ground 1

3. Click on the Display tab.
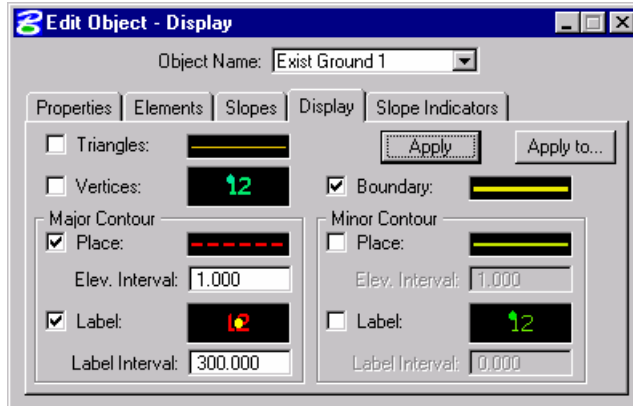
4. Toggle ON the Major Contour option and setup the symbology as shown below.
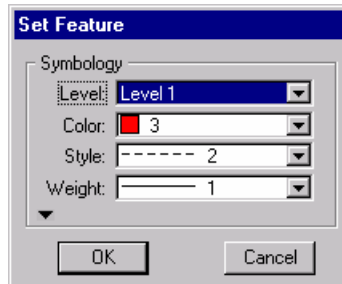
5. Toggle ON the Label option and setup the text parameters as shown below.
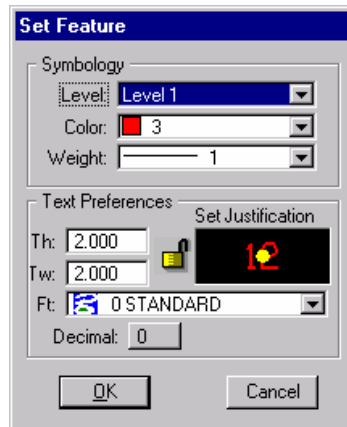
6. Setup the Elev. Interval and Label Interval spacing as shown below.
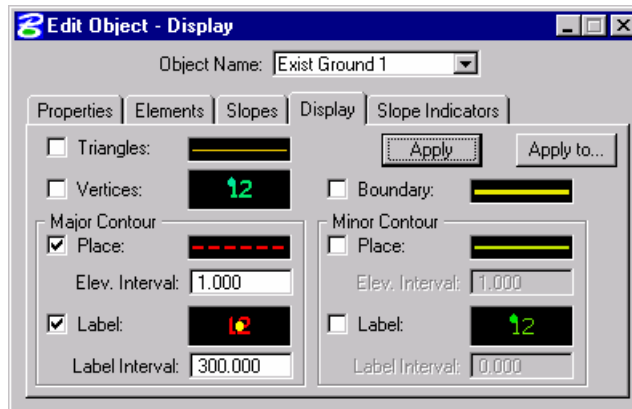
7. Click **Apply** when done. The Exist Ground 1 symbology in the dgn file will update.

8. Close the Edit Object dialog when done.

Setting Up the Model Display

1. Select the Edit Model tool. *(MS Menu: Modeler > Model > Edit)*

2. Click on the Display Model tab.
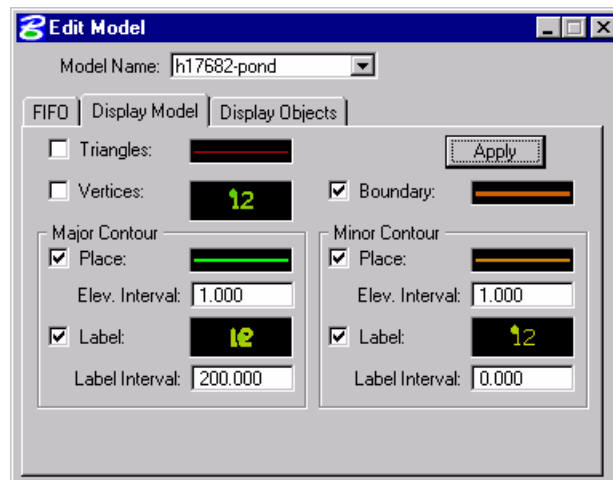
3. Toggle ON the Major Contour option and setup the symbology as shown below.
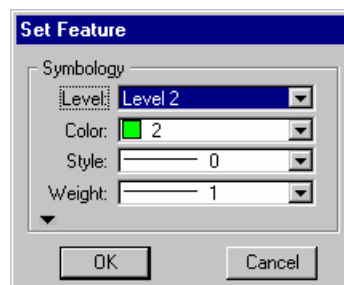
4. Toggle ON the Label option and setup the text parameters as shown below.
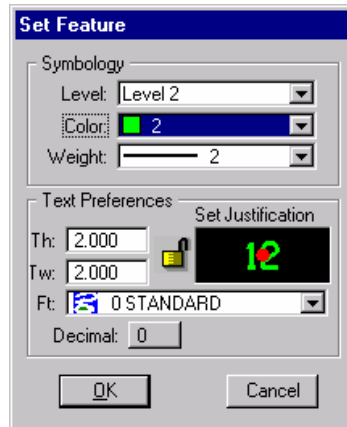
5. Setup the Elev. Interval and Label Interval spacing as shown below.
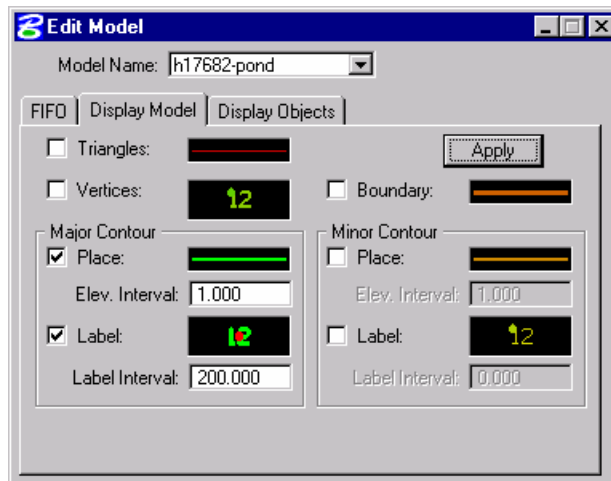
6. Select the Display Objects tab and setup the dialog as shown below.
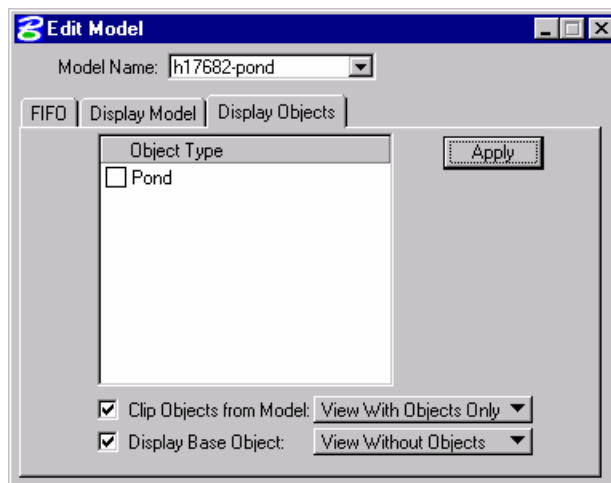
7. Click **Apply** when done. The Model display will update.

8. Close the dialog when done.

Writing the Model Display to DGN

1. Select the Object: Exist Ground 1 in the Active Site Object Control dialog as shown below.

2. Select the Export Visualization to DGN tool. *(MS Menu: Modeler > Project > Export > Visualization to DGN).*

3. Setup the dialog as shown below.
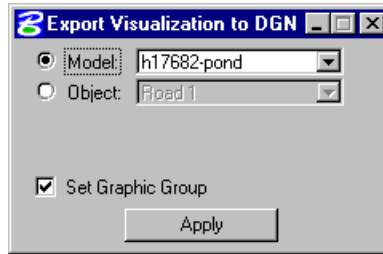
4. Click **Apply**.

5. Click **OK** to the Alert message. The pond contours will be written into the dgn file.
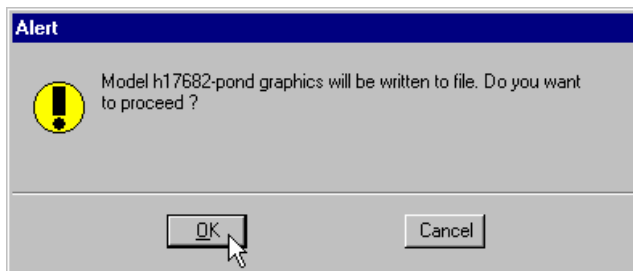
6. In the same Export Visualization to DGN dialog, select Object and select Exist Ground 1
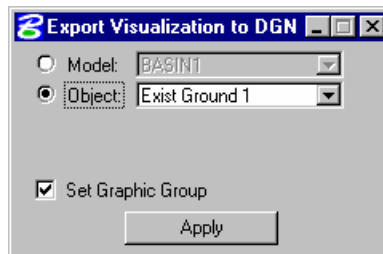
7. Click **Apply**.

8. Click **OK** to the Alert message. The existing ground contours will be written into the dgn file.

 VDOT GEOPAK Drainage
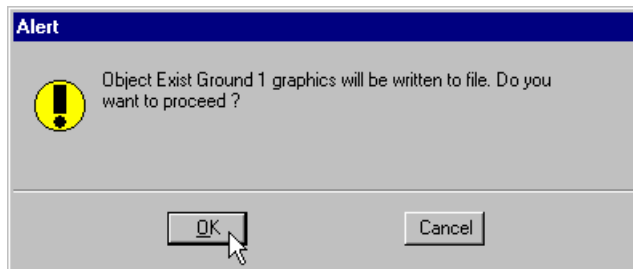
9. Toggle OFF the Display View option in the Active Site Object Control. We can verify that the final contours were written into the dgn file.

10. Save the Site project. *(MS Menu: Modeler>Project>Save).*

LAB EXERCISE: CHANNEL DESIGN

Creating the Channel Object

1. Toggle ON the Display View option in the Active Site Object Control.
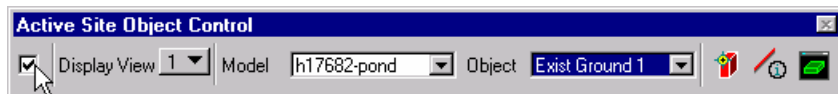
2. Access the Channel Design tool. *(MS Menu: Modeler > Tools > Channel Design)*
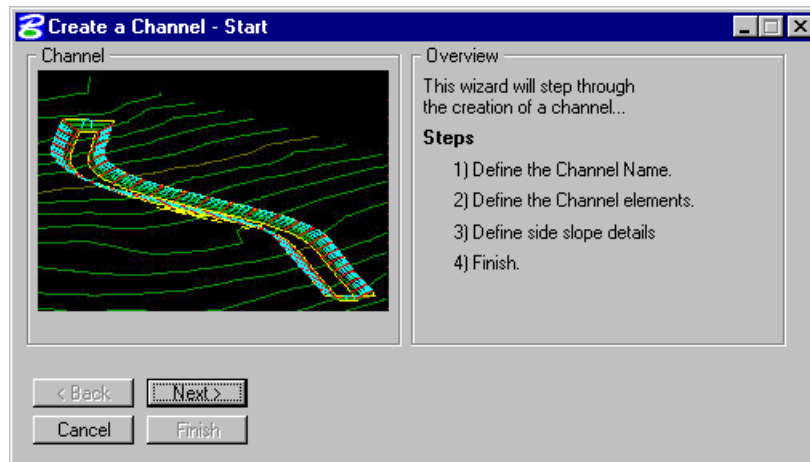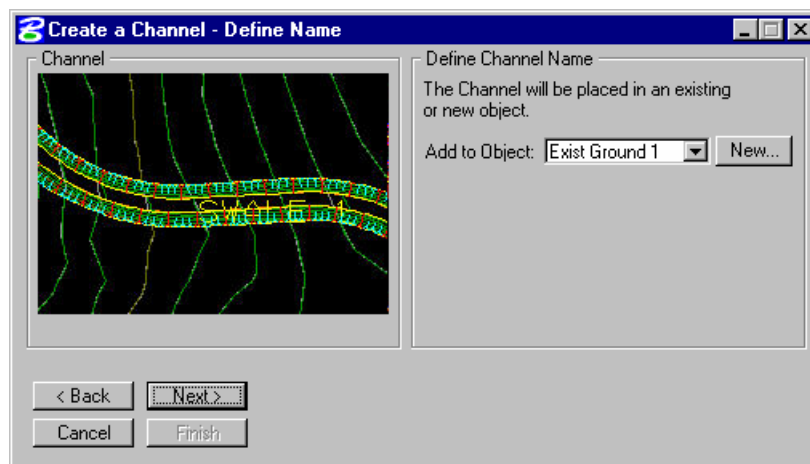
3. Click **Next>**

4. Click **New** and create a new object: Channel 1
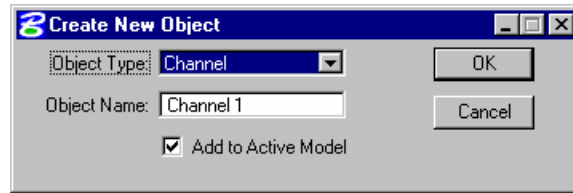
5. Click **OK** when done.
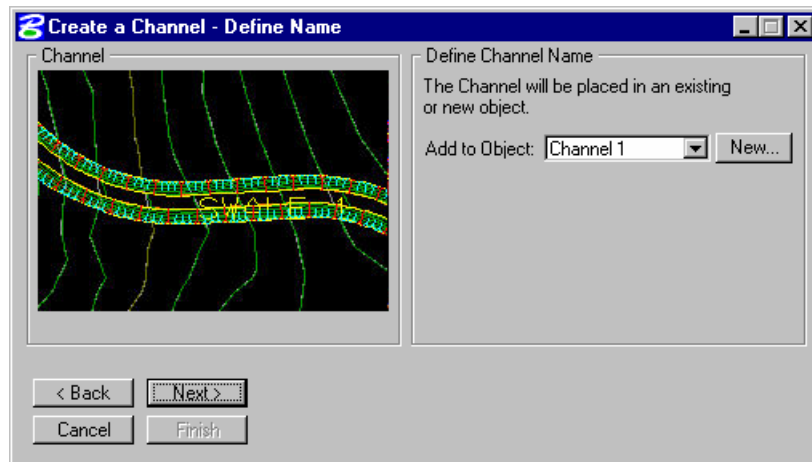
6. Click **Next>** and setup the dialog as shown below. We will use the alignment and profile created in a previous chapter.
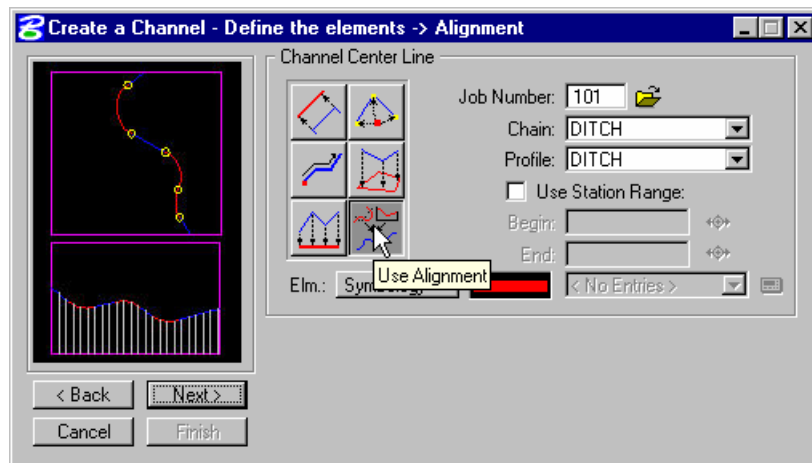
7. Click **Next>** and setup the dialog as shown below.
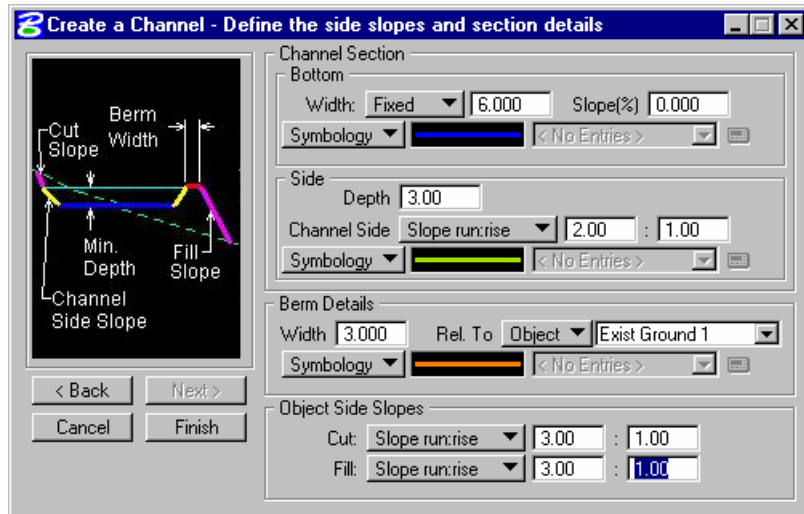
8. Click **Finish** to build the channel.

Reviewing the Channel Section

1. Access the profile tool. *(Modeler > Analysis > Profiles).*

2. Setup the dialog as shown below.
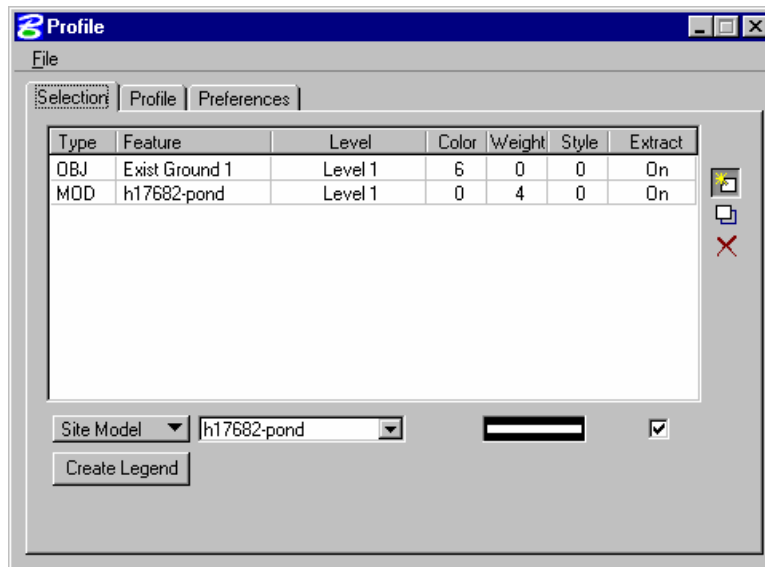
3. Select the **Profile** tab.

4. Click on the **Place Element** button and draw a MicroStation line across the channel section.
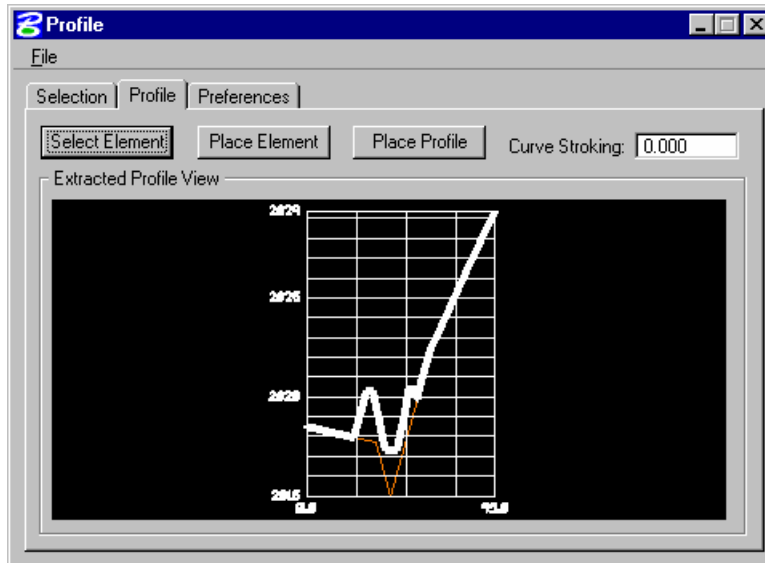
5. Reviewing the section, we can see that the ditch profile could be lowered to better follow the ground.

Adjusting the Ditch Profile

1. Open the Coordinate Geometry tool. *(Applications > Geopak Road > Geometry > Coordinate Geometry).*

2. Setup the dialog as shown below and click **OK** when done.
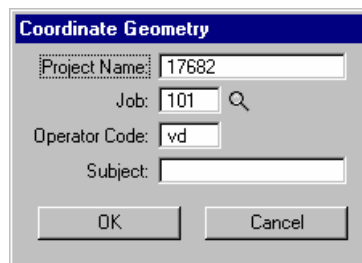
3. From the Coordinate Geometry tool, open the Offset Profile tool. *(Coordinate Geometry > Element > Profile > Offset).*
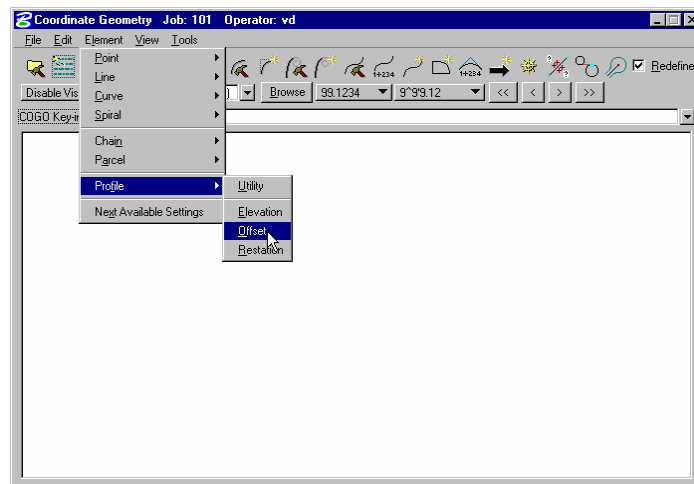
4. Setup the Offset Profile dialog as shown below. Click the **Add** icon when done.
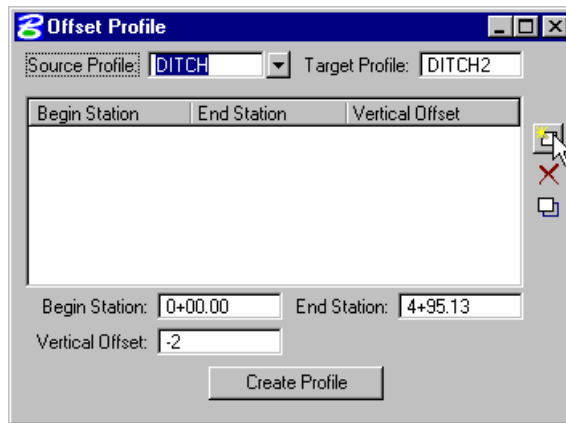
5. We will lower the Ditch profile 2 feet and create a new profile : Ditch2
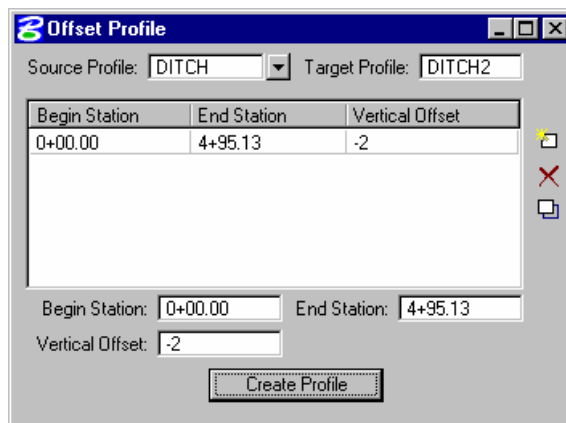
6. Click the **Create Profile** button. The new profile Ditch2 will be created and stored.
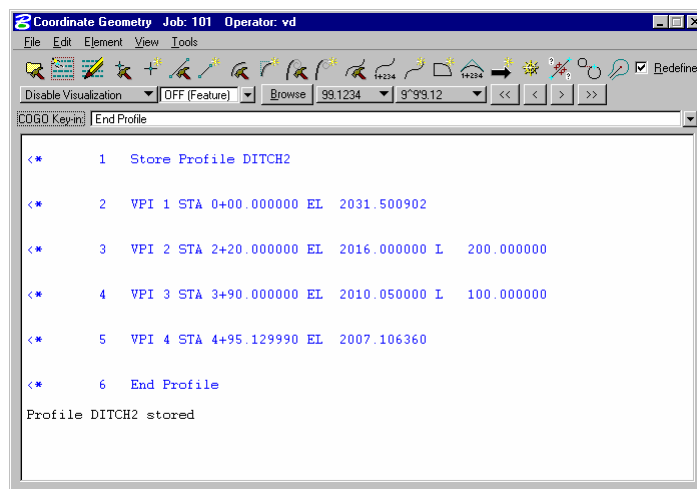
Updating the Channel Design with the New Profile

1. Select the Element Information tool. *(Modeler > Elements > Information).*

2. Click the **Select Element** icon and select the centerline of the ditch in the dgn file.
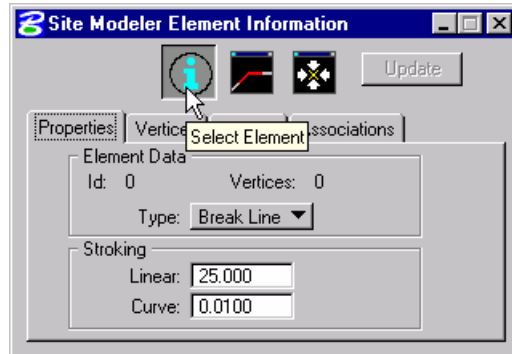
3. Select the **Associations tab** and switch the Options to Reference Associations
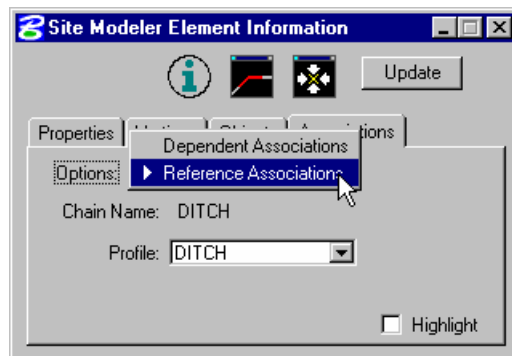
4. Change the Profile to Ditch2 and click **Update** when done.
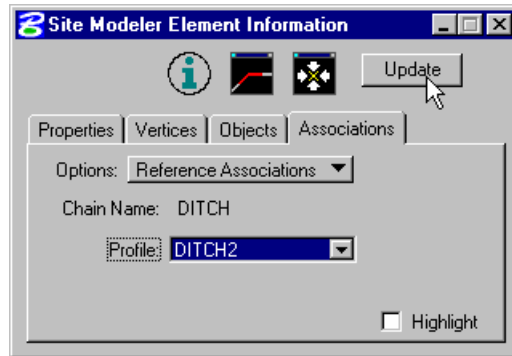
5. Click **Yes** on the Alert box in order to accept the changes.
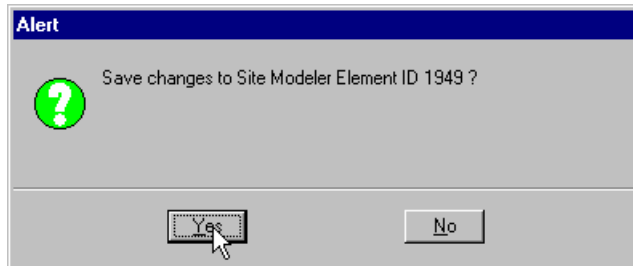
Reviewing the New Channel Section

1. Access the profile tool. *(Modeler > Analysis > Profiles).*

2. Setup the dialog as shown below.
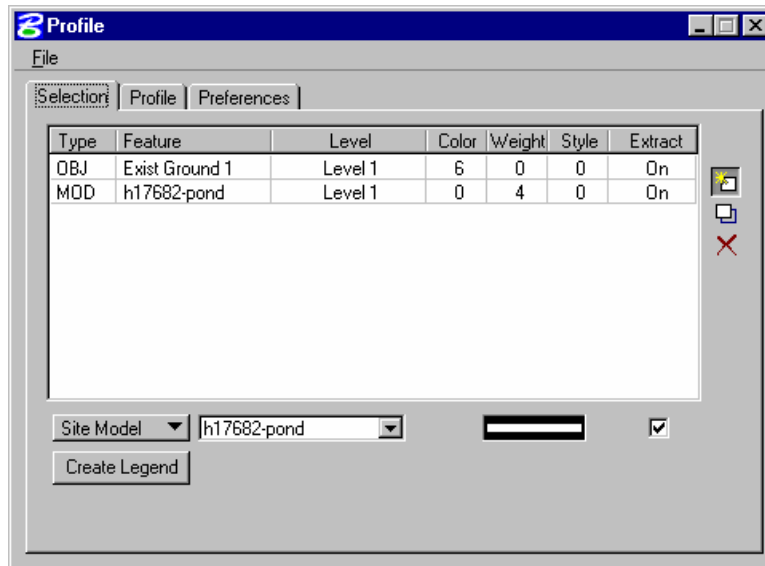
3. Select the **Profile** tab.

4. Click on the **Select Element** button and select the MicroStation line across the channel section.
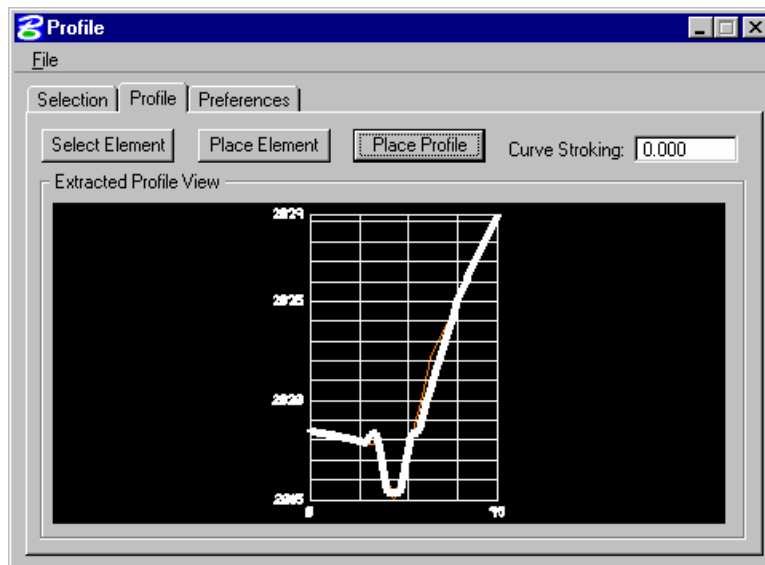
5. Click the **Place Profile** button to place the cross section of the channel in the design file.

6. Save the Site project. *(MS Menu:Modeler>Project>Save).*

7. Exit Modeler. *(MS Menu: Modeler > Project > Exit).*

LAB EXERCISE : POND ROUTING

> ### *Review GEOPAK Drainage Project Preferences*

1. Activate GEOPAK Drainage. *(MS Menu > Applications > GEOPAK Drainage > Drainage)*

2. Open the drainage project: *c:\data\geo\vdot\drain1\h17682.gdf. (Drainage > Project > Open)*

3. Select the **Preferences** tool *(Drainage > Project > Preferences)*

4. Click **Project Components**.

5. Activate the GEOPAK Site Project as shown below.
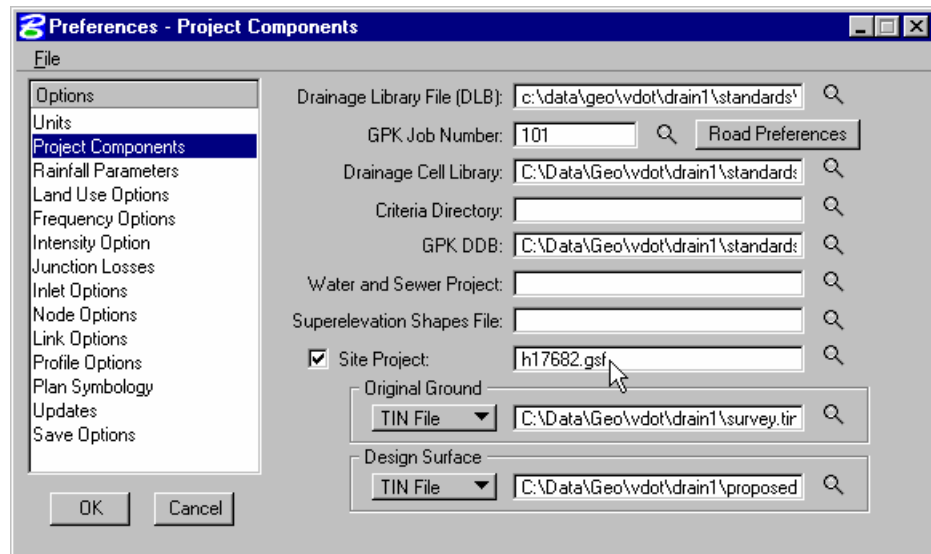
6. Click **OK** to save the changes in the drainage project.

> ### *Create a Drainage Area for Routing*

1. Create a new drainage area for the routing calculation. *(Drainage > Components > Area > Add)*
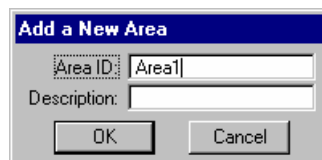
2. Setup the drainage area dialog as shown below.
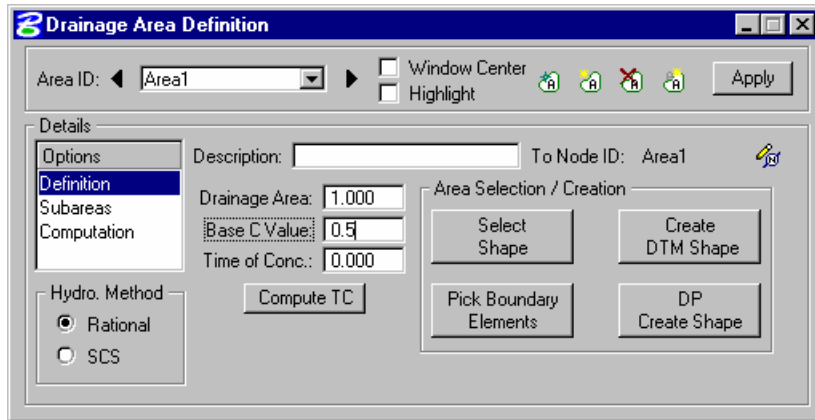
3. Click the **Compute TC** button. Setup the dialog as shown below.
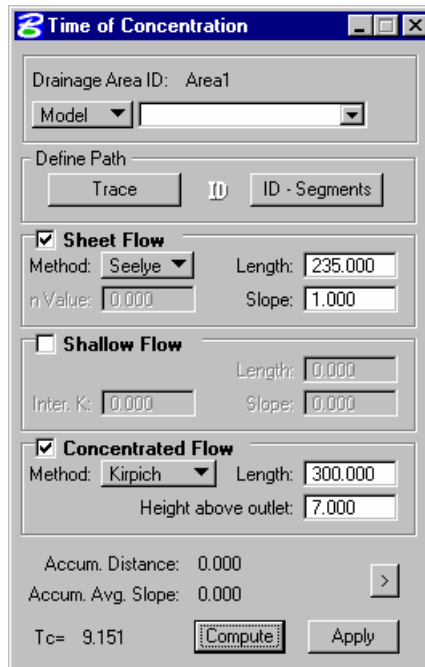
4. Click the **Compute** button and then, **Apply** to return to the Drainage Area Definition dialog.

5. Highlight **Compute** in the **Drainage Area Definition** dialog and press the **Compute Discharge** button.
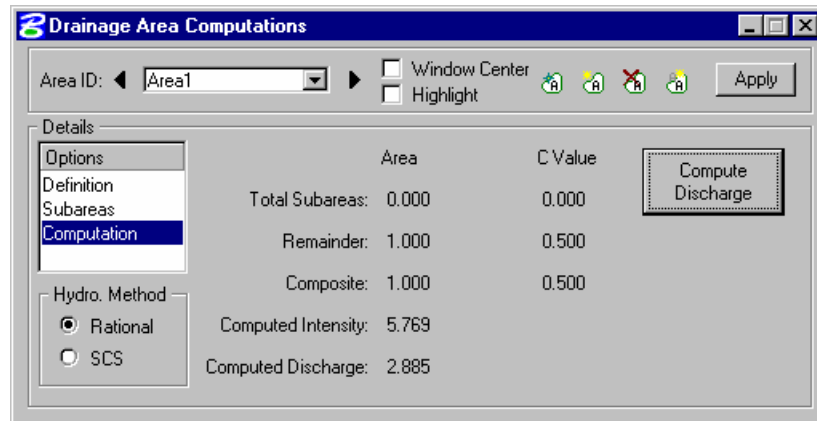
6. Click **Apply** to store the drainage area into the *h17682.gdf* project.

7. Close the Drainage Area Definition dialog.

> **Create a Runoff Hydrograph**

1. Create a new routing ID: Route1. *(Drainage > Components > Routing > Add)*.
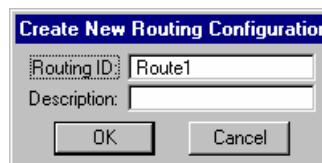
2. Select the **Runoff** icon (1st icon located in the top middle of the dialog) and highlight the **Definition** item in the Options list. Select *Area1* as the **Area ID**.
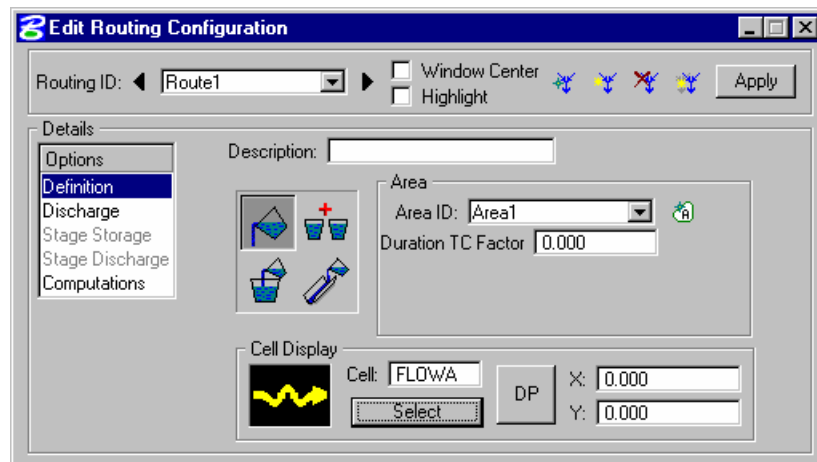
3. Keyin the storm **Duration TC Factor** = 1.

4. Click the **Select** button and select the any cell, since it is just used as a graphical marker, a procedure will be executed in the dgn file.
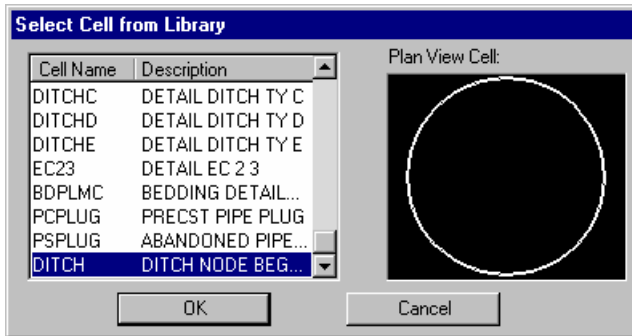
5. Click **OK** to continue.

6. 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.

7. Select the **Use Compute Discharge** radio button in the **Discharge** option.
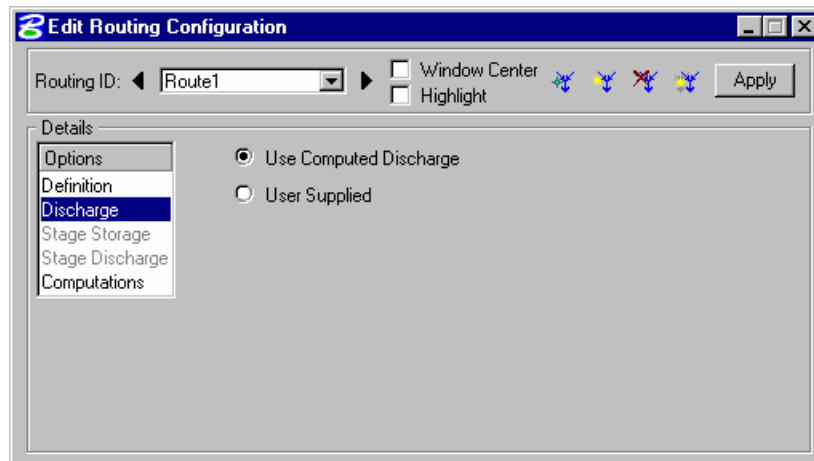
8. Select the **Computations** list item in the Options list.

9. Click the **Compute** button.
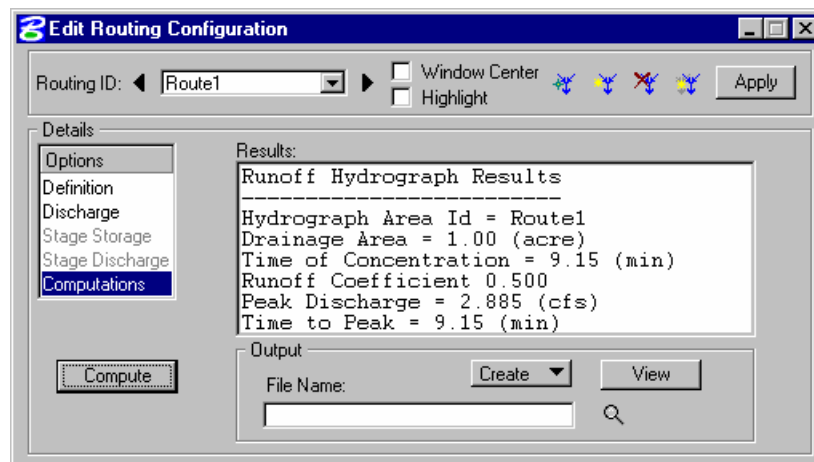
10. Click **Apply** to accept the calculations.

> **> *Create the Reservoir Routing***

1. Create a new routing ID: Route2. *(Drainage > Components > Routing > Add)*.

2. Select the **Reservoir** icon and highlight the **Definition** list item from the Options list.
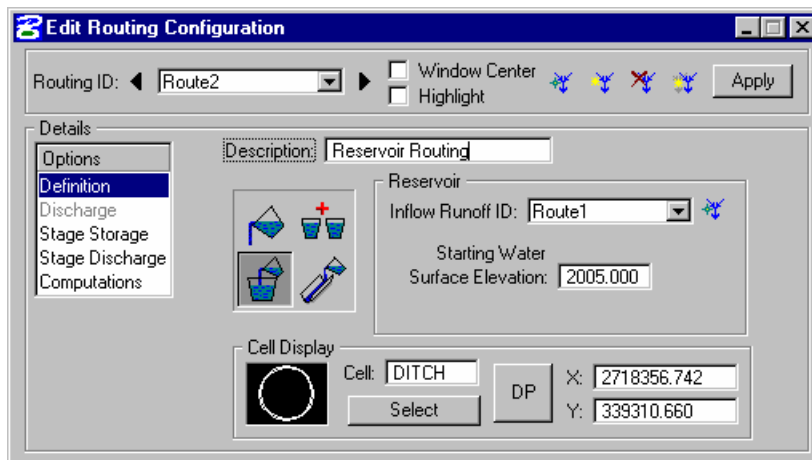
3. Select **Route1** for the Inflow Runoff ID and set the Starting Water Surface Elevation at **2005.00**.
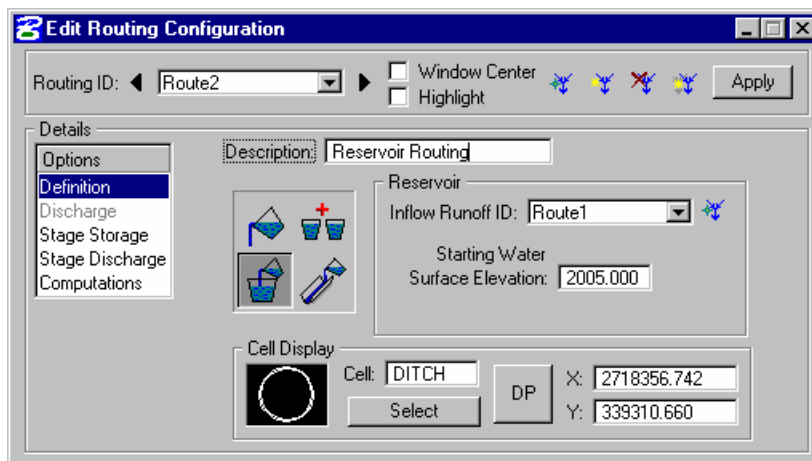
4. Click the **DP** button and data point in MicroStation view window 1 in the area where the control structure is located. This will place a cell that graphically represents the second routing configuration in MicroStation.

5. Select the **Stage Storage** list item from the Options list, and setup the dialog box as shown below.
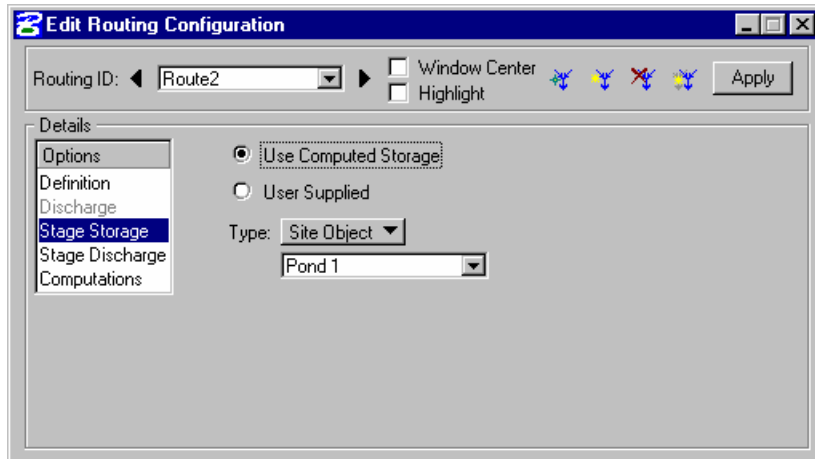
6. Select the **Stage Discharge** list item from the Option list. Set the type, size, coef., and elevation options as shown.
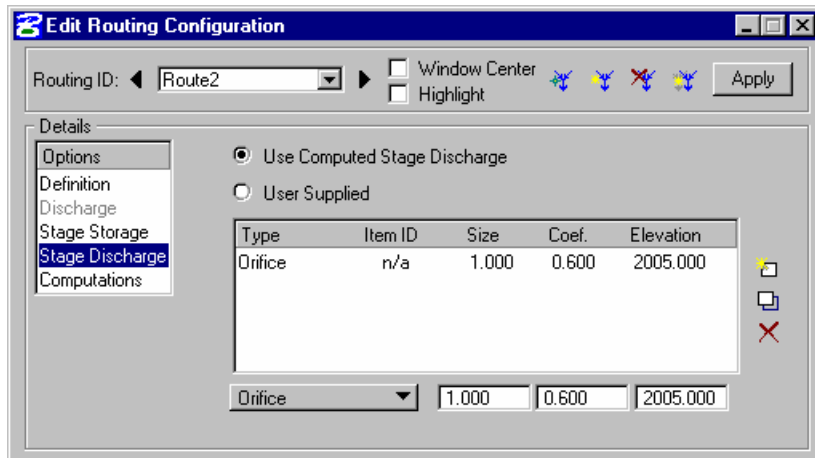
7. Select the **Computations** list item from the Options list.

8. Type *routing.txt* as a filename in the Output group box.

9. Click the **Compute** button to perform the routing calculations.
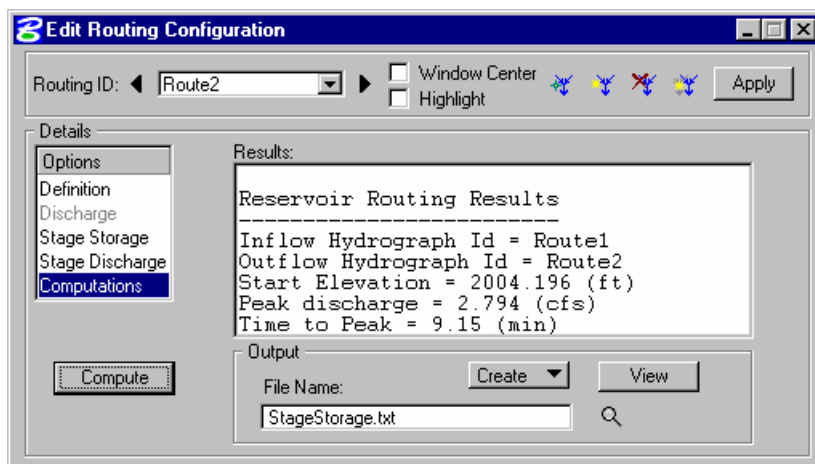
10. Click **Apply** to store the routing configuration.

11. Click **View** to review the calculations.

12. Close the dialog when done.

13. Exit MicroStation.

LAB EXERCISE : FLOOD PLAIN DETERMINATION

> *Determining the Water Surface Elevation*

1. Execute *c:\data\geo\vdot\drain1\LAB11_V8ad_Flood.EXE*

2. Open the MicroStation file, *c:\data\geo\VDOT\drain1\Survey_river.dgn.*

3. Activate Hec-Ras from *c:\ data\geo\vdot\drain1\hecras\ras.exe.*
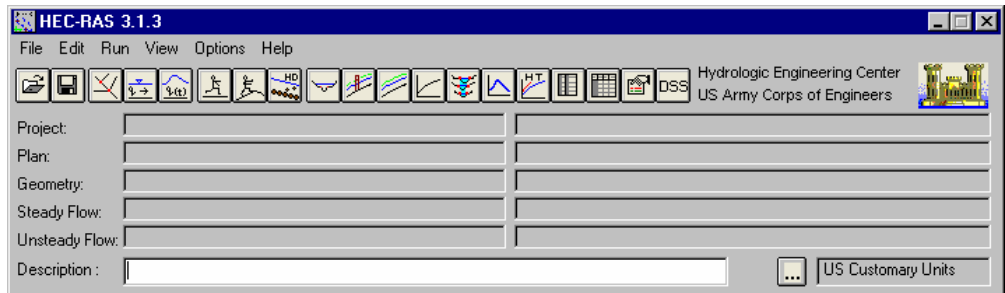
4. Access the included Hec-Ras project: *xsriver.prj* located in the *c:\data\geo\vdot\drain1*
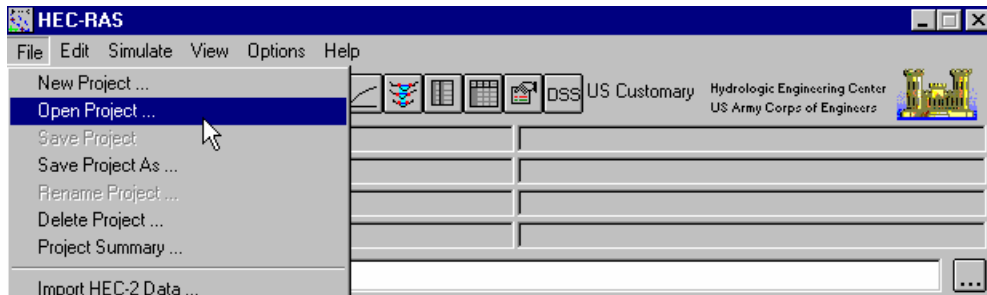
5. The project contains the geometric and hydraulic information of the channel shown in the dgn file.
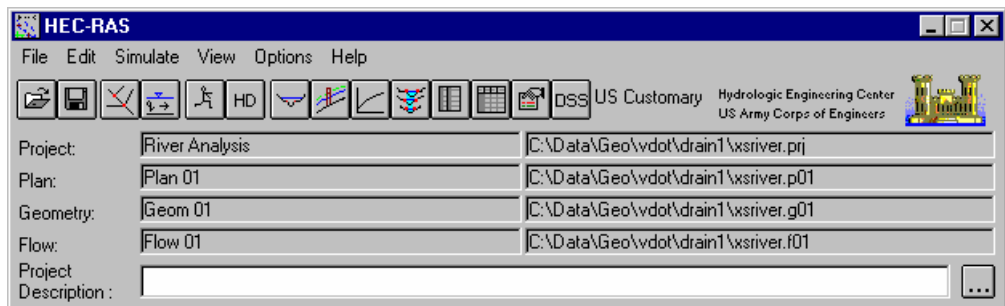
6. The geometry and station information has been updated according to the channel configuration. *(HEC_Ras: Edit > Geometric Data)*
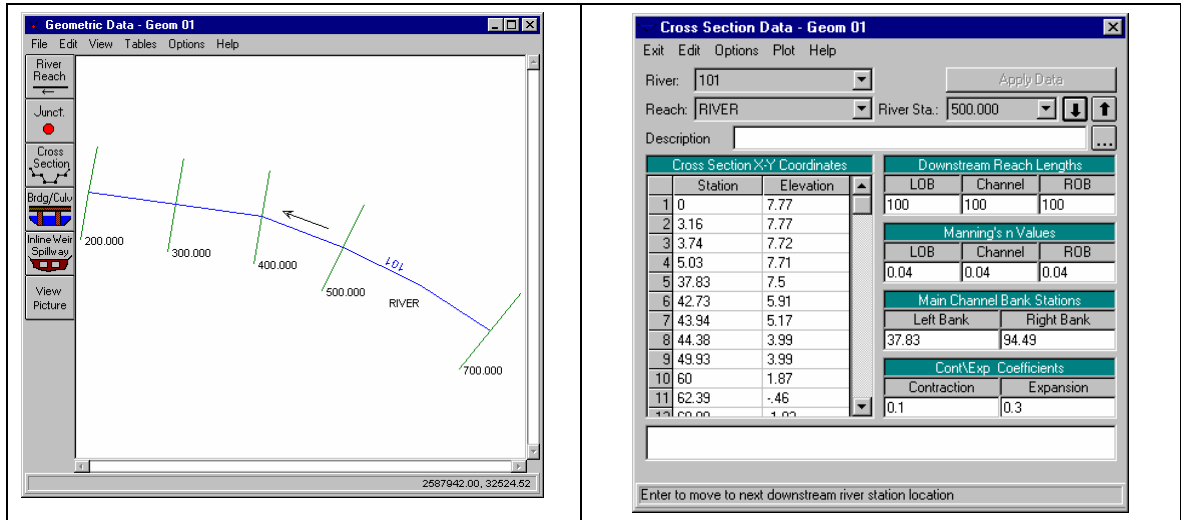
7. Review the Steady Flow Data information *(Hec-RAS: Edit>Steady Flow Data)*
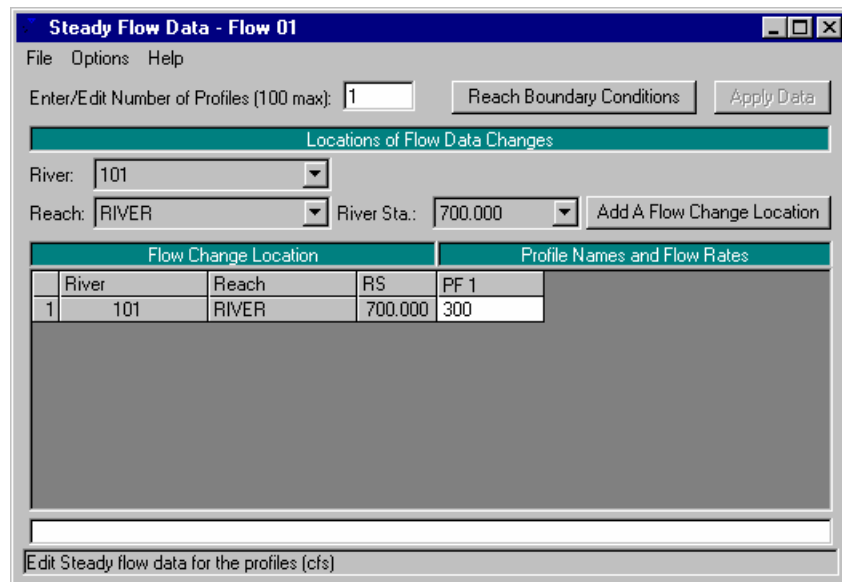
8. Perform the Steady Flow Analysis in order to get the new water surface elevation. *(Hec-RAS: Simulate>Steady Flow Analysis)*
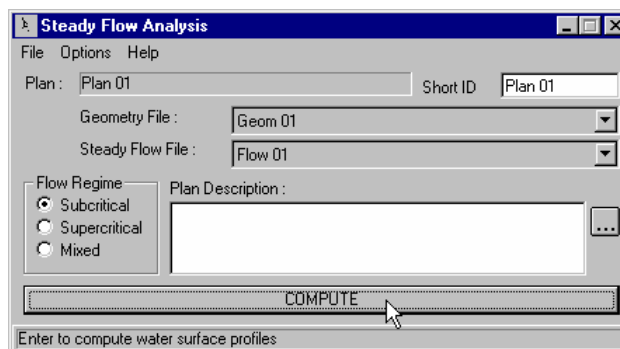
9. Review the water elevation in the channel thru the cross section display. *(Hec-RAS: View>Cross Section).*
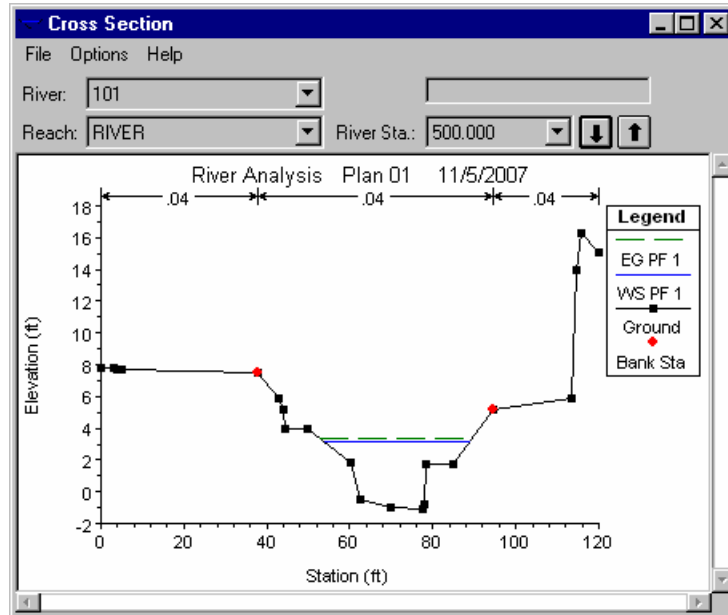
> ## *Creating a HEC-RAS Profile Table*

1. Create a profile table that will allow us to export the information needed. *(Hec-RAS: View>Profile Table).*
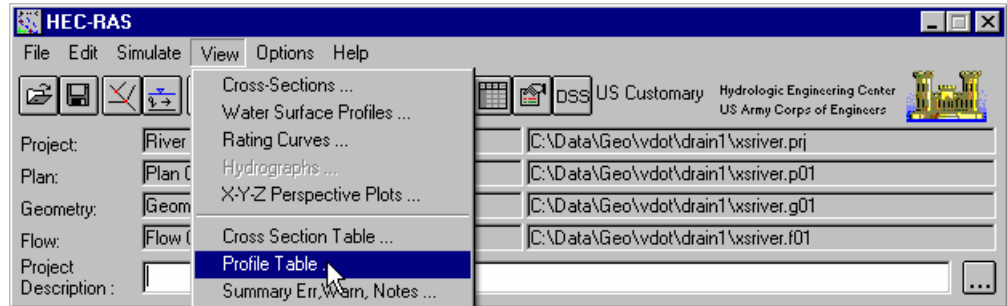
2. Define a new table *(Profile Output Table: Option > Define Table).*

3.   Click on the **Clear All Table Headings** button.
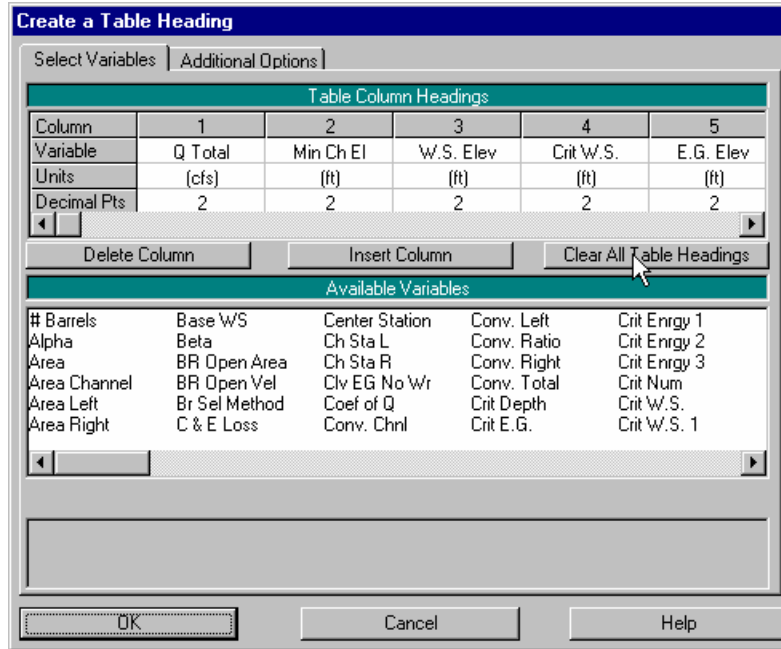
4.   Add the following variables in each column: **Center Station, Sta W.S. Lft, Sta W.S. Rgt** and **W.S. Elev.**
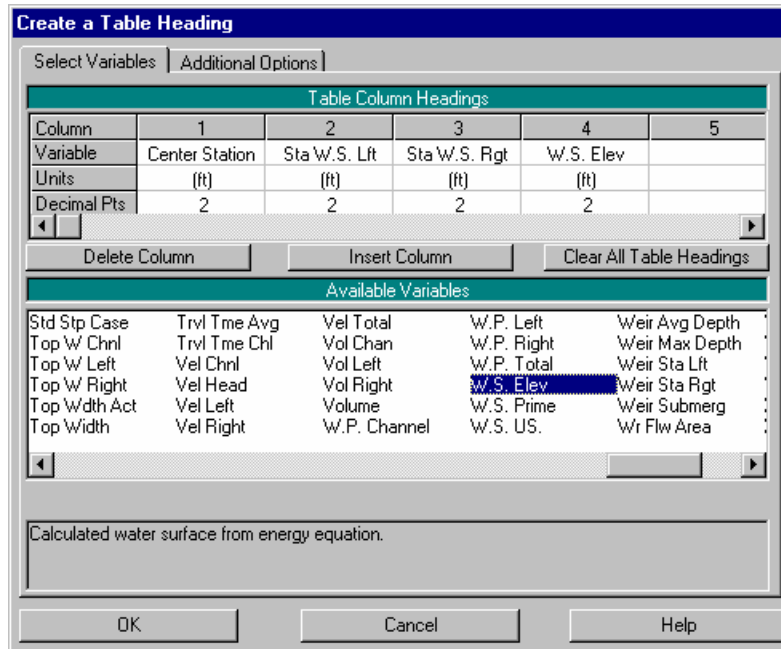
5.   Click **OK** when done.

6. Include the profile name in the newly created table. *(Profile Output Table: Options > Include Profile Name in Table)*

7. Select all the elements in the profile by clicking the upper left corner of the table.

8. Copy the elements to the clipboard. *(File>Copy to Clipboard)*

9. Open a new Microsoft Excel file and name it: *Flood_plain_table.xls*

10. Paste the contents of the clipboard into the spreadsheet. *(Excel: Edit > Paste)*
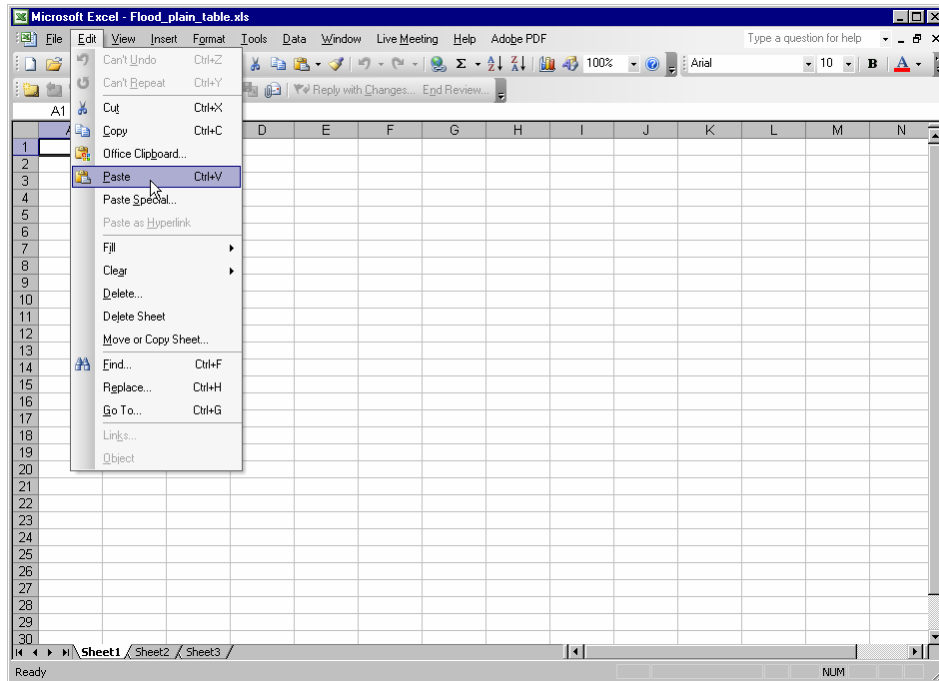
11. Save the spreadsheet *(Excel: File > Save)*

12. Exit Microsoft Excel.

Creating a DTM Surface from the Profile Table

1. In the MicroStation file: *Survey_river.dgn*, load the *VDOT_FloodPlainDTM* macro located in *c:\data\geo\VDOT\drain1 . (MS Menu: Utilites > Macro > Project Manager)*
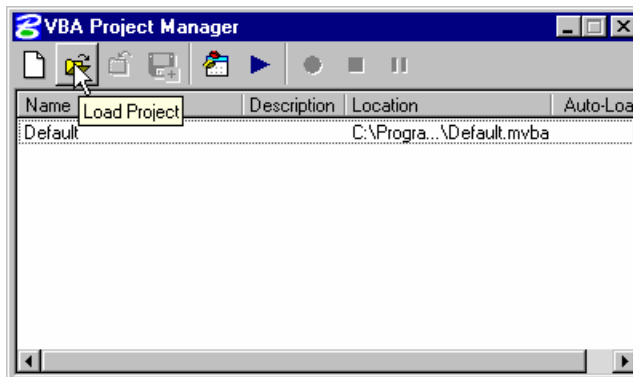
2. Highlight the VBA application and click on the Macros button.
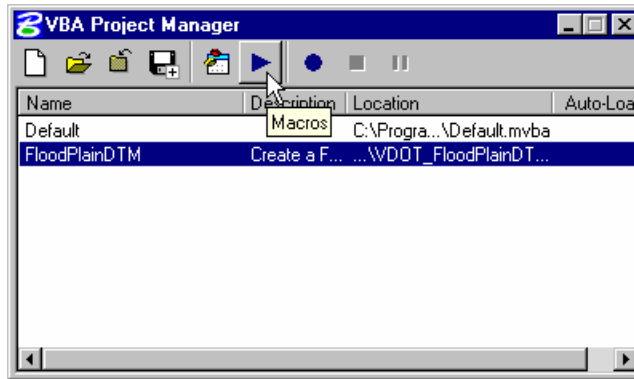
3. Execute the loaded macro by clicking on the **Run** button.
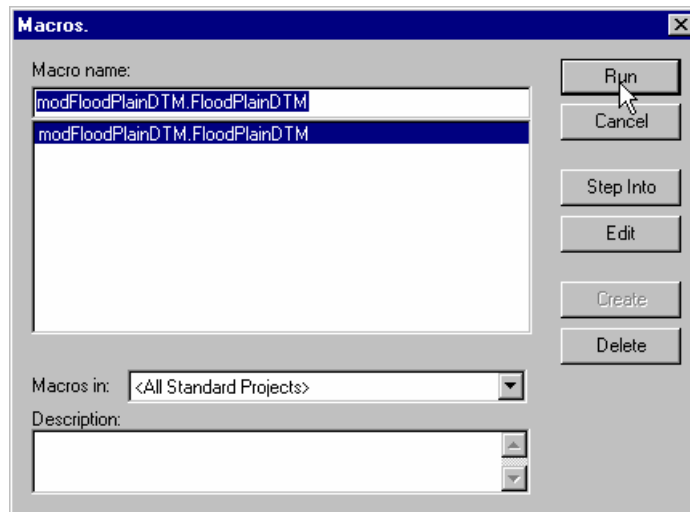
4. Setup the Flood Plain DTM dialog as shown below and generate a Flood Plain DTM. This will create a GEOPAK TIN file at the floodplain level.
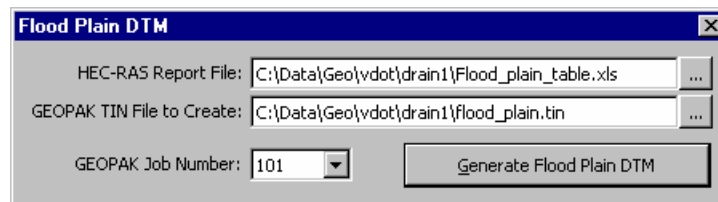
5. Close the dialog when done.

Plotting the Flood Plain Encroachment Limits in the Channel

1. Select the DTM Tools from GEOPAK Road. *(MS Menu: Applications > GEOPAK Road > DTM Tools)*

2. Select the DTM Menu icon.

3. Select the Elevation Differences dialog from the DTM Menu. *(DTM > Analysis > Elevation Differences).*

4. We will need to find where the floodplain dtm (flood_plain.tin) intersects at the same elevation into the existing ground dtm (creek.tin). Setup the dialog as shown below.
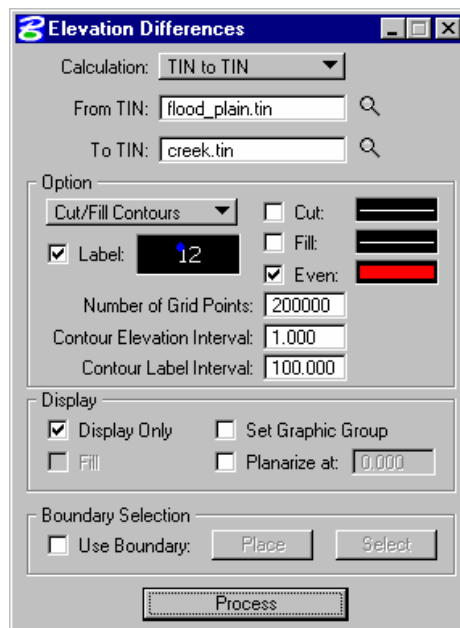
5. Click **Process**.

6. A series of lines will be plotted delineating the flood elevation as it is encroaches into the existing dtm.

7. Exit MicroStation.

This page left intentionally blank.

12 Technical Reference

OBJECTIVES

- The objective of this chapter is to review the computation methods utilized by GEOPAK Drainage.

THE RATIONAL METHOD

The Rational method is the most widely used formula to calculate storm water runoff. VDOT recommends the use of that for pipe sizing, inlet capacity and spread computations.

This method uses certain assumptions that make their use only suitable for areas of less than 200 acres.

These assumptions are:

- The entire drainage area contributes to the discharge

- Rainfall intensity, measured in inches per hour, is distributed evenly in the whole drainage area.

- Rainfall intensity is at a constant rate along the entire duration of the storm. The storm duration is at least equal to the time of concentration.

- Frequency of the computed flow is of the same frequency as that of the rainfall intensity. A 5 year rainfall intensity is set to produce a 5 year peak flow or discharge.

- Coefficient of runoff is the same for all storms and probabilities of recurrence.

The rational formula is given as:

$$Q = CIA$$

Q = computed peak discharge in cubic feet per second (cfs)

C = runoff coefficient (non dimensional value)

I   = intensity of storm in inches/hr.

A   = drainage area in acres

The formula is not dimensionally correct as it is a result of empirical studies.

Let's analyze in detail each component of the formula.

RUNOFF COEFFICIENT (C)

The runoff coefficient is a function of the ground cover and other hydrological abstractions.

This is a dimensionless number that represents the fraction of rainfall that remains on the surface of the ground.

The runoff coefficient relates the estimated peak discharge to a theoretical maximum of 100 percent of runoff.

VDOT provides a table of runoff coefficients that depends on the slope of the terrain being analyzed and the use of it.

In more practical terms we can summarize that for storm drain projects, we can divide our drainage area into pervious and impervious areas.

We can conclude that a pervious area is the one with a c value equal or less than 0.5. A typical pervious area could be woodlands, grassy terrain or residential areas. Then we can also say that an impervious area is the one with a c value greater than 0.5. Pavement areas or heavy commercial and industrial lots are a good representation of this type of drainage area.

Therefore, we can say that for storm drain projects the following assumption is valid:

C = 0.95 for pavement areas

C = 0.50 for grassy areas

In a drainage area, multiple types of surfaces could be present with different runoff coefficients. Then, a weighted runoff coefficient should be determined. The weighting is based on the area of each land and is found by the following formula:

$$C = \frac{\sum CA}{\sum A}$$

INTENSITY (I)

Intensity is usually represented at the rate of rainfall in inches over an hour period. Intensity computations in GEOPAK Drainage are based on one of several options for defining the Intensity-Duration-Frequency (IDF) relationship.

The selection of the intensity over a drainage area depends of the frequency of the storm to be analyzed and their duration, than in general we equal to the time of concentration of the area.

The intensity over a drainage area is calculated or extracted from widely develop I-D-F (intensity-duration-frequency) curves. VDOT also provides mathematical functions for each of the curves so they can be implemented in computer programs. GEOPAK Drainage uses a four parameter intensity curve with the time of concentration in logarithmic format to represent the intensity rainfall in Virginia.

$$I = a + b\ln(tc) + c\ln(tc)^2 + d\ln(tc)^3$$

I = rainfall intensity for frequency (inch/hr)

n = time of concentration (min)

a,b,c,d = empirical factors which are input for each frequency.
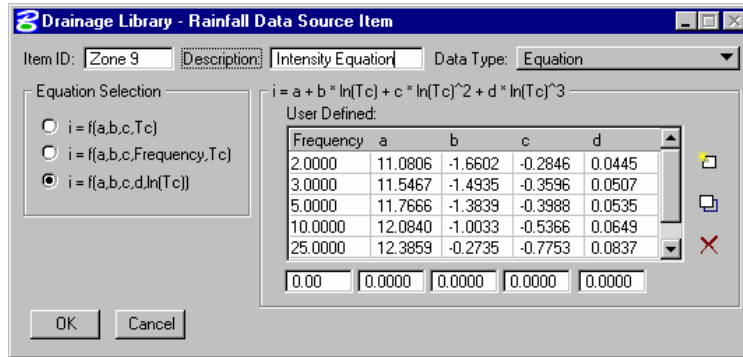
VDOT has also published I-D-F curves covering the State of Virginia for different frequencies under different rainfall zones.

In order to find the intensity for drainage area, we have to do the following.

1.- Localize our project inside one of the zones VDOT has divided the State.

2.- Select the proper I-D-F curve according to the selected zone.

3.- Enter the chart using the time of concentration as the storm duration and intersect the proper frequency curve.

4.- At the point of intersection, read the rainfall intensity in inches per hour.

TIME OF CONCENTRATION (TC)

Time of concentration is defined as the time required for a drop of rainfall to travel from the most remote point of a watershed to the point of analysis. GEOPAK Drainage time of concentration calculations are derived from FHWA HEC-22.

VDOT recommends a minimum time of concentration equal to 5 minutes despite a lower calculated time.

To calculate time of concentration, we have to recognize the different types of flow that occur inside the drainage basin and along the flow path. These are sheet flow, shallow concentrated flow and open channel or pipe flow. For each type of flow a myriad of formulas could be used; each one of them using their own assumptions. FHWA in their "Urban Drainage Design Manual" cites the following formulae for each types of flow.

Sheet Flow Travel Time: it is the starting type of flow in a drainage basin in which runoff has a uniform depth along a sloping surface. We can say that the limit is the first 400 ft. The formula is a version of the kinematic wave equation.

$$T_{sheet} = \frac{K_u}{I^{0.4}} \left(\frac{nL}{\sqrt{S}} \right)^{0.6}$$

T = sheet flow travel time (minutes)

n = Manning's roughness coefficient of the watershed.

L = flow length (ft)

I =   rainfall intensity (inches/hr)

S =   surface slope (ft/ft)

K_u = empirical coefficient equal to 0.933

Manning's Roughness Coefficient (n) for Overland Sheet Flow

Surface Description	n
Smooth Asphalt	0.011
Smooth Concrete	0.012
Ordinary concrete lining	0.013
Good wood	0.014
Brick with cement mortar	0.014
Vitrified clay	0.015
Cast iron	0.015
Corrugated metal pipe	0.024
Cement rubble surface	0.024
Fallow (no residue)	0.05
Cultivated soils:	
Residue cover < 20%	0.06
Residue cover > 20%	0.17
Range (natural)	0.13
Short grass prairie	0.15
Dense grasses	0.24
Bermuda grass	0.41

Woods (when selecting n, consider cover to a
height of about 30 mm. This is the only part of the
plant cover that will obstruct sheet flow.

| Light underbrush | 0.40 |
| Dense underbrush | 0.80 |

This is an iterative process since the intensity depends on the time of concentration. Therefore, the assumed time of concentration must be the same as the calculated one in order to stop the iterative process.

Shallow Concentrated Flow Travel Time: Up to a distance of 400 ft, sheet flow tends to concentrated in rills and then gullies of increasing proportions. First, we need to calculate the velocity in order to estimate the travel time.

$$V = K_u k S^{0.5}$$

V = velocity (ft/s)

K_u = coefficient of 3.28

k = intercept coefficient.

S = slope in percentage

Intercept Coefficients for Velocity vs. Slope Relationship

Land cover / Flow regime	k
Forest with heavy ground litter; hay meadow (overland flow)	0.076
Trash fallow or minimum tillage cultivation; contour or strip cropped; woodland (overland flow).	0.152
Short grass pasture (overland flow)	0.213
Cultivated straight row (overland flow).	0.274
Nearly bare and untilled (overland flow) alluvial fans in western mountain regions	0.305
Grassed waterway (shallow concentrated flow).	0.457
Unpaved (shallow concentrated flow.)	0.491
Paved area (shallow concentrated flow); small upland gullies	0.619

Open Channel and Pipe Flow Travel Time: It is the flow collected in channel and pipes. The Manning's equation can be used to estimate the average flow velocities in pipes and open channels.

$$V = \frac{1.49}{n} R^{2/3} S^{0.5}$$

V = velocity (ft/s)

n = Manning's runoff coefficient

R = hydraulic radius (flow area divided by the wetted perimeter)

S = slope (ft/ft)

Then the travel time could be calculated as:

$$T_{channel} = \frac{L}{60V}$$

Values of Manning Coefficient (n) for Channels and Pipes:

Conduit Material	Manning n
Closed Conduits:	
Asbestos – cement pipe	0.011-0.015
Brick	0.013-0.017
Cast Iron pipe	
Cement-lined & seal coat	0.011-0.015
Concrete (monolithic)	0.012-0.014
Corrugated-metal pipe (1/2 in x 2 ½ in corrugations)	
Plain	0.022-0.026
Paved Invert	0.018-0.022
Spun Asphalt Lined	0.011-0.015
Plastic pipe (smooth)	0.011-0.015
Vitrified Clay:	
Pipes	0.011-0.015
Liner Plates	0.013-

Open Channels

Lined Channels:

A. Asphalt	0.013-0.017
B. Brick	0.012-0.018
C. Concrete	0.011-0.020
D. Rubble or riprap	0.020-0.035
E. Vegetal	0.030-0.40

Excavated or dredged

Earth, straight, and uniform	0.02-0.030
Earth, winding, fairly uniform	0.025-0.040
Rock	0.030-0.045
Not maintained	0.050-0.14

Natural channels (minor streams, top width at flood stage < 100 ft.	
Fairly regular section	0.03-0.07
Irregular section with pools	0.04-0.10

Then, the total time of concentration for a watershed will be:

$$T_{concentration} = T_{sheet} + T_{shallow} + T_{channel}$$

STORM FREQUENCY

Frequency can be defined as the return period of a specific storm or their exceedance probability.

Exceedance probability is the probability that an event having a specific volume and duration will be exceeded in a specified time period. Return period is the average length of time between events having the same volume and duration.

Be careful, saying that a storm has a 50 year frequency does not mean that it happens every 50 years. The probability of a storm happening is inversely proportional of the return period. This means that a 50 year storm has a 2% probability of recurrence.

VDOT recommends the following design frequencies depending of the type of project:

Type of Project	Design Frequency
General Design	3 year
General design with ditch replacement and adding of side pipes.	10 year
General design work on interstates	10 year
Interstate facilities which only outlet is a storm drain system and for the outlet of systems requiring pumping stations.	50 year

INLET SPREAD COMPUTATIONS

The spread or ponded width of runoff at an inlet location is a geometric function of the cross sectional geometry, longitudinal pavement/gutter slope and discharge. A direct solution is not possible for complex cross sections with multiple breaks in the transverse slope so an iterative procedure derived from HEC-22 is employed.
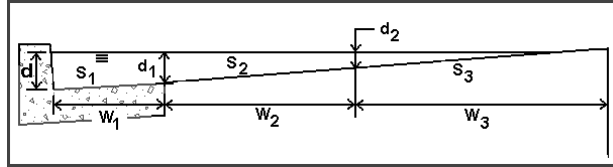
CURB INLETS ON-GRADE

The computation of on-grade curb opening inlets involves a determination spread or ponded width characteristics, computation of length required for total interception, and consideration of inlet efficiency.
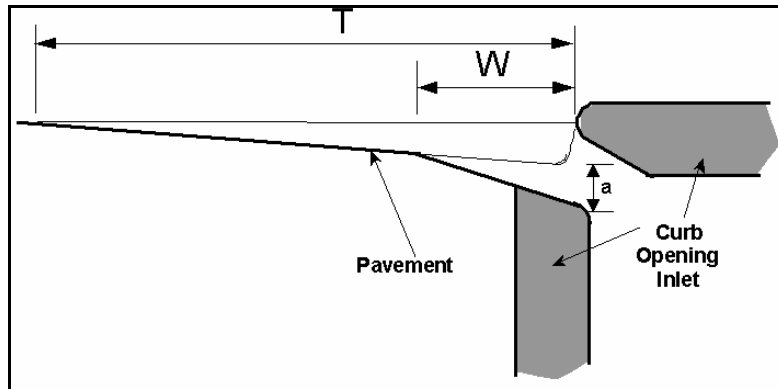
The hydraulics of Curb Inlets On Grade is derived from HEC-22. The applicable equations are as follows:

The length of curb inlet required for total interception using Equation:

$$L_r = 0.60 \, Q^{0.42} \, S^{0.3} \left(\frac{1}{n \, S_e} \right)^{0.6}$$

where:

L_r = length of curb inlet required (ft)

Q = flowrate in gutter(cfs)

S = longitudinal slope (ft/ft)

n = Manning's roughness coefficient

S_e = equivalent cross slope (ft/ft) for non depressed inlets

The equivalent cross slope (S_e) for a depressed curb-opening inlet using Equation:

$$S_e = S_x + \frac{a}{W} E_o$$

where:

S_e = equivalent cross slope (ft/ft)

S_X = cross slope of the road (ft/ft)

a = gutter depression depth (ft)

W = gutter depression width (ft)

E_O = ratio of depression flow to total flow

The Ratio of depression flow to total flow is determined using the following equation:

$$E_o = 1 - \left(1 - \frac{W}{T}\right)^{8/3}$$

where:

E_O = ratio of depression flow to total flow

Inlet interception capacity is then defined by:

$$Q_{it\ ercepted} = Q_{total} \times Efficieny$$

where:

Qi = intercepted discharge (cfs)

Qt = total inlet discharge (cfs)

Efficiency = inlet efficiency defined below

$$Efficiency = 1 - (1 - \frac{L}{L_r})^{1.8}$$

where:

L_r = length of curb inlet required for 100% interception (ft)

L = length of curb inlet provided (ft)

CURB INLETS IN SAG CALCULATIONS

The capacity of a curb inlet in a sag depends on the water depth at the curb opening, and the height of the curb opening.

The inlet operates as a weir to depths equal to the curb opening height and transition to orifice flow on higher depths. In the transition the capacity is based on the lesser of the computed weir and orifice capacity.

The head on the inlet is computed by adding the ponded depth (y) and the inlet depression (a) using the following Equation:

$$h = y + a$$

where:

h = head at inlet opening (ft)

y = ponded depth in gutter (ft)

a = inlet depression depth (ft)

The capacity of the inlet when operating under weir conditions is calculated with the equation:

$$Q = C_w (L + 1.8*W) h^{1.5}$$

where:

Q = total flow reaching inlet (cfs)

C_W = weir coefficient = 2.3

h = head at inlet opening (ft)

L = length of curb inlet opening (ft)

W = lateral width of inlet depression (ft)

The equation for interception capacity of a curb opening operating as an orifice follows:

$$Q = C_o h L \sqrt{2 g d_o}$$

where:

Q = total flow reaching inlet (ft^3/s)

C_o = orifice coefficient = 0.67

h = depth of opening (ft)

L = length of curb opening inlet (ft)

g = acceleration due to gravity = 32.2 ft/s^2

d_o = effective head at the centroid of the orifice (ft)

GRATE INLETS ON-GRADE CALCULATIONS

The capacity of a grate inlet on-grade depends on its geometry and the cross slope, longitudinal slope, total gutter flow, depth of flow, and pavement roughness. The design of a grate inlet on-grade involves an analysis of given grate dimensions to estimate the interception rate. The difference between the estimated interception rate and the total approach discharge equals the bypass rate. The following procedure is applicable to grate inlets on-grade:

The Ratio of discharge over the grate (Frontal Flow) to flow to total flow is determined using the following equation:

$$E_o = 1 - \left(1 - \frac{W}{T}\right)^{8/3}$$

where:

E_O = ratio of flow over the grate to total flow

The ratio of frontal flow intercepted to total frontal flow:

$$R_f = 1 - 0.3\left(V - V_o\right), \text{ if } V > V_o$$
$$R_f = 1.0, \text{ if } V < V_o$$

where:

R_f = ratio of frontal flow intercepted to total frontal flow

V = approach velocity of flow in gutter (ft/s)

V_o = minimum velocity that will cause splash over grate (ft/s)

The splash over velocity is of the form of

$$V_o = AL^N factor^{1-N}$$

where:

Vo = is splash over velocity

A = constant for different grate type

N = a power coefficient for different grate type

L = grate length

factor = 1 for English and .3048 for metric

Grate Type	Coefficient A	Coefficient N
P-1-7/8	5.74872	0.5038679
P-1-1/8	4.54822	0.5058875
Curved Vane	3.92812	0.5954068
45 Tilt Bar	3.35159	0.5926234
30 Tilt Bar	2.67291	0.7567052
P-1-7/8-4	3.01181	0.6454720
Reticuline	2.48235	0.7659749

The ratio of side flow intercepted to total side flow:

$$R_S = \left[1 + \frac{\left(0.083 \, z V^{18} \right)}{L^{23}} \right]^{-1}$$

where:

R_S = ratio of side flow intercepted to total flow

z = inverse of transverse slope

V = approach velocity of flow in gutter (ft/s)

L = length of grate (ft)

The efficiency of grate, E_f, using equation:

$$E_f = \left[R_f E_o + R_s \left(1 - E_o \right) \right]$$

The interception capacity of the grate, Q_i, using equation:

$$Q_i = E_f Q = Q \left[R_f E_o + R_s \left(1 - E_o \right) \right]$$

The bypass flow rate:

$$CO = Q - Q_i$$

GRATE INLETS IN SAG CALCULATIONS

A grate inlet in sag configuration operates in weir flow at low ponding depths. The transition to orifice flow begins as the ponded depth increases. The procedure for calculating the inlet capacity is as follows:

The capacity of a grate inlet operating as a weir is determined with the following equation:

$$Q_w = (1 - P_R)C_w P h^{1.5}$$

where:

Q_w = weir capacity of grate (cfs)

C_w = weir coefficient = 3.087

P = perimeter of the grate (ft)

h = allowable head on grate (ft)

P_R = perimeter reduction factor to account for clogging

Under orifice conditions, the grate area controls the capacity. The capacity of a grate inlet operating under orifice flow is computed equation:

$$Q_o = A * (1 - A_R)C_o \sqrt{2gh}$$

where:

Q_o = orifice capacity of grate (cfs)

C_o = orifice flow coefficient = 0.67

A = clear opening area (ft^2) of the grate (the total area available for flow)

g = acceleration due to gravity (32.2 ft/s^2)

h = allowable head on grate (ft)

A_R = area reduction factor to account for clogging

The capacity of a grated inlet in a sag is based on the minimum flow calculated from weir and orifice conditions. The figure below demonstrates the relationship between weir and orifice flow. If Q_o is greater than Q_w (to the left of the intersection in the figure), then the capacity would be that calculated with the weir equation. If, however, Q_o is less than Q_w (to the right of the intersection), then the capacity as determined with the orifice equation would be used.
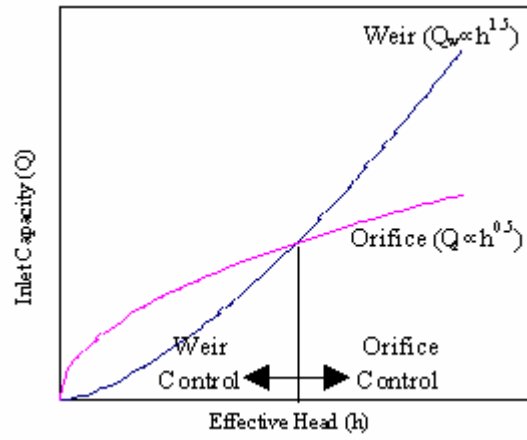
LINK HYDRAULIC COMPUTATIONS

Numerous algorithms are employed to perform the pipe and ditch hydraulic computations. The fundamental equations and terminology are presented here for reference purposes.

The most widely used formula for determining the hydraulic characteristics of storm drain networks is the Manning Formula, expressed by the following equation:

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

where:

V = mean rate of flow (ft/s)

R = the hydraulic radius (ft)

S = the slope of hydraulic gradeline (ft/ft)

n = Manning's roughness coefficient

The hydraulic radius (R) is defined as follows:

$$R = \frac{A}{W_P}$$

where:

W_P = wetted perimeter (ft)

A = cross sectional area of flow (A^2)

To satisfy continuity,

$$Q = A V$$

where:

Q = discharge (m^3/s)

Combining these equations gives the following equation:

$$Q = \frac{1}{n} A R^{2/3} S^{1/2}$$

Conveyance describes the geometric carrying capacity of a hydraulic conduit and is described by the following equation:

$$K = \frac{Q}{S^{\frac{1}{2}}} = \frac{1}{n} A R^{\frac{2}{3}}$$

Friction Slope is the slope (S) in Manning's formula required to convey a specified discharge under uniform flow conditions given a depth, roughness, and shape. Under uniform flow, depth and flow area are constant and the friction slope, the actual slope, and the energy slope are all equal.

$$S_f = (\frac{Q}{K})^2$$

LINK DESIGN CONSTRAINTS

The Constraints establish further design criteria for Links. GEOPAK Drainage utilizes the physical constraints of minimum and maximum size and slope to determine a suitable pipe size.

GEOPAK Drainage will perform the following steps to determine the "allotted pipe envelope"

- Starting upstream, GEOPAK computes each link slope by connecting the nodes at the elevation set by the node elevation minus the minimum depth.

- If this slope is less than the minimum slope, GEOPAK Drainage sets the slope equal to the minimum slope. This will provide the upper limit of the envelope. Then, it continues downstream.

- To setup the bottom part of the "envelope", it starts from the outlet elevation, move upstream and setup the minimum invert of the pipes by using node elevation minus the maximum depth values on each node.

- If the invert falls within the minimum and maximum slopes the link is placed at the calculated elevation. If not, the link is once again placed at the minimum specified slope from that node to the upstream end of the system.

- If during the construction of the envelope, any elevations that the user has held will be accommodated if physically possible and the envelope constrained to that elevation. However, if a held elevation of a pipe violates the minimum or maximum depth line, the elevation will not be held. With the minimum and maximum depth profiles computed, GEOPAK Drainage can then design a suitable pipe within this envelope that satisfies all the constraints possible.

The other constraints, minimum and maximum velocity are for querying purposes and evaluating the adequacy of the design.

Minimum Rise	Constraint - Minimum rise (diameter, height) for pipe or ditch in master units (i.e. feet or meters). The Department's minimum round pipe size is 1.5' (18").
Maximum Rise	Constraint - Maximum rise (diameter, height) for pipe or ditch in master units (i.e. feet or meters). Unless the designer knows a constraint, it is recommended that this be set for a large pipe such as a 6' (72").
Minimum Slope	Constraint - Minimum physical slope for pipe or ditch expressed in percentage. The minimum pipe slope varies with the pipe size (0.15% for 18", 0.102% for 24" etc.) See the discussion in the Storm Drain Handbook. During the initial design, the pipe sizes are not known, so this constraint may need to be changed as the design progresses. In moderately sloped terrain, using 0.15% may be acceptable for all pipes, because the terrain will force the pipes to be steeper than that. In flat terrain, using 0.15% for all pipes may cause the downstream pipes to be too deep.
Maximum Slope	Constraint - Maximum physical slope for pipe or ditch expressed in percentage. The Department does not have a maximum slope constraint, so it is recommended that this be set high for all pipes.
Minimum Velocity	Query - Minimum velocity for pipe or ditch expressed in feet per second or meters per second. This value must be something greater than zero for the program to do the hydraulic calculations.
Maximum Velocity	Query - Maximum velocity for pipe or ditch expressed in feet per second or meters per second.

HYDRAULIC GRADE LINE COMPUTATIONS

The methodology employed by GEOPAK Drainage to compute the water surface profiles through a storm drain network is typical of any open channel water surface procedure. A backwater analysis is performed through the system beginning at the most downstream point (outlet) and progressing upstream to the most remote nodes. GEOPAK Drainage will compute the hydraulic grade line using gradually varied flow analysis in free surface flow conditions and pressure flow computations under full flow conditions.

The resulting hydraulic gradeline represents the set of elevations to which the water would rise if open to atmospheric pressure (e.g., piezometer tubes) along a pipe run and can be used to evaluate the adequacy of the design and identify areas where flooding occurs.

HYDRAULIC GRADELINE

The hydraulic gradeline (HGL) procedure begins at the most downstream node (outlet) and proceeds upstream through each link in the same fashion. A starting HGL at this downstream point elevation must be defined. The procedure for developing the HGL through a link of the network from downstream node to upstream node is as follows:
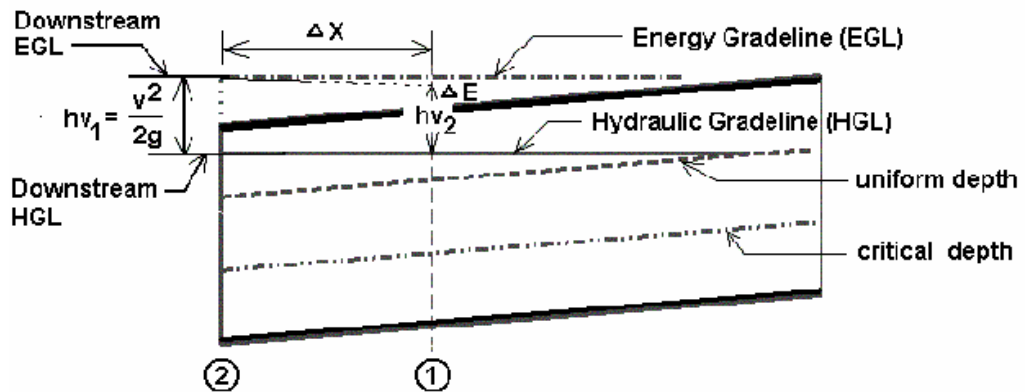
1. Beginning with the HGL at the downstream node (HGL_1), the Energy Gradeline (EGL_1) is computed from:

$$EGL = HGL + \frac{v^2}{2g}$$

where:

v = velocity at flow depth HGL (ft/s)

g = gravity (32.2)

HGL = elevation of hydraulic gradeline

Assuming a very small change in the energy and depth (y) compute the HGL_2 from $HGL_2 = HGL_1 + y$

2. Determine the distance along the pipe (x) to create the y loss due to friction = x = S_fy where S_f = $0.5(S_1 + S_2)$

3. This procedure of computing HGL_2 and the distance along the pipe proceeds until the end of the pipe has been reached yielding the HGL at the upstream end of the pipe.

4. The resulting upstream HGL is subsequently used on the next upstream pipe as its starting downstream HGL. GEOPAK Drainage can alternatively use the computed energy gradeline upstream as the starting downstream EGL on the next pipe upstream. This alternative method differs in that the EGL's upstream and downstream at a node are equal rather than the conventional assumption that the HGL's are equal.

5. If junction losses are desired they computed prior to progressing upstream and added to the hydraulic gradeline.

SPECIAL CONSIDERATIONS

- If the starting HGL is less than critical depth then critical depth will be assumed.

- If the HGL converges to equal the uniform depth the computations proceed to the upstream end at uniform depth

- Once the HGL reaches the soffit of the pipe full flow conditions begin.

- Hydraulically steep pipes where uniform depth is less than the critical depth are checked with a backwater profile to verify if the resulting upstream HGL drowns out critical depth at the upstream end. If it does than the backwater profile is accepted. If the backwater curve does not exceed the critical depth, then a forewater profile is generated for this supercritical condition. A forewater profile uses the same procedure as above but progresses from the upstream end towards the downstream end. It begins at critical depth at the upstream end and converges towards uniform depth as the calculations proceed downstream.

- Hydraulic jumps may occur when mixing supercritical and subcritical water surface profiles. GEOPAK Drainage does not compute the exact location of a jump but will check if downstream subcritical depths have sufficient energy to force a supercritical profile to its conjugate depth thereby satisfying the conditions required for a jump to occur. Message will be appear indicating the existence of a jump.

JUNCTION LOSS METHODOLOGIES

Five junction loss methodologies are discussed:

- Free surface transition losses

- Pressure flow transition losses

- Bend losses

- Terminal inlet / junction losses

- Complex junction losses

FREE SURFACE TRANSITION LOSSES

The energy losses may be expressed in terms of the kinetic energy at the between the incoming and outgoing pipes:

$$H_t = K_t \Delta (\frac{V^2}{2g})$$

where K_t is the transition Loss Coefficient.

$$H_t = K_t (\frac{V_2^2}{2g} - \frac{V_1^2}{2g})$$

for $V_2 > V_1$

$$H_t = K_t (\frac{V_1^2}{2g} - \frac{V_2^2}{2g})$$

for $V_1 > V_2$

where:

V_1 = upstream velocity

V_2 = downstream velocity

K_t = Loss Coefficient for expansion of contraction

PRESSURE FLOW TRANSITION LOSSES

The energy losses may be expressed in terms of the kinetic energy at the between the incoming and outgoing pipes:

$$H_t = K_t \left(\frac{(\Delta V)^2}{2g} \right)$$

where K_t is the transition Loss Coefficient.

$$H_t = K_t \left(\frac{(V_2 - V_1)^2}{2g} \right)$$

for $V_2 > V_1$

$$H_t = K_t \left(\frac{(V_1 - V_2)^2}{2g} \right)$$

for $V_1 > V_2$

where:

V_1 = upstream velocity

V_2 = downstream velocity

K_t = Loss Coefficient for expansion of contraction

BEND LOSSES

Method 1 (From Modern Sewer Design)

Bend losses may be estimated from the equation:

$$H_b = K_b \frac{V^2}{2g}$$

Where the K_b can be estimated from the following table.

K - Bend Loss Coefficient	Degree of Turn at Bend
0.10	20
0.32	40
0.64	60
1.06	80
1.32	90

Method 2 (From AASHTO)

Bend losses may be estimated from the equation:

$$H_b = K_b \frac{V^2}{2g}$$

Where the K_b can be estimated from the following table.

K - Bend Loss Coefficient	Degree of Turn at Bend
0.19	15
0.35	30
0.47	45
0.56	60
0.64	75
0.70	90

CURVE LOSSES

Losses for curved pipe segments may be estimated from the equation:

$$K_b = .25 \sqrt{\frac{\phi}{90}}$$

where:

= central angle of bend in degrees

TERMINAL INLET /JUNCTION LOSS

$$H_t = K_t \left(\frac{V^2}{2g} \right)$$

where:

V = Velocity at terminal end of junction

K_t = Loss Coefficient for terminal junction

COMPLEX JUNCTIONS

Manhole losses in many cases comprise a significant percentage of the overall losses within a sewer system. Losses at junctions are dependent upon flow characteristics, junction geometry and relative sewer diameters.

Method 1

Losses at junctions where one or more incoming laterals occur may be estimated by combining the laws of pressure plus momentum where H_j is equal to the junction losses.

$$H_j = K_j \frac{V^2}{2g}$$
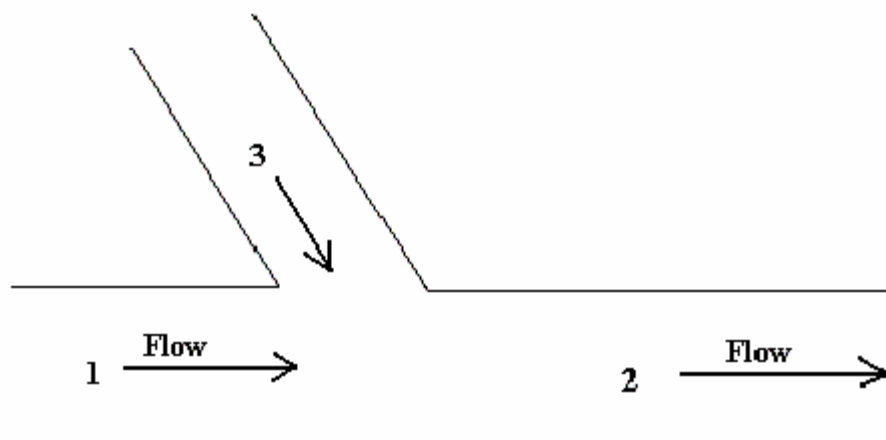
Using the laws of pressure plus momentum, the loss H_j can be estimated as follows:

$$(H_j + D_1 - D_2)\frac{(A_1 + A_2)}{2} = \frac{Q_2^{\,2}}{A_2\, g} - \frac{Q_1^{\,2}}{A_1 g} - \frac{Q_3^{\,2}}{A_3\, g}\cos \delta$$

Method 2

Losses at junctions where one or more incoming laterals occur utilize the principle of conservation of energy, involving position energy and momentum energy. The energy content of the inflows is equal to the energy content of the outflow plus any losses due to the collision and turbulence.
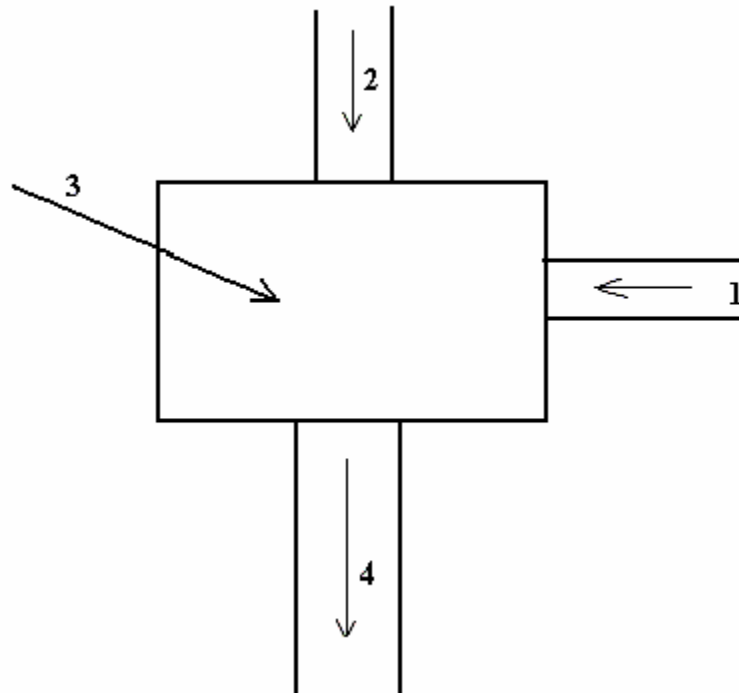
The loss H$_j$ can be estimated as follows:

$$H_j = \frac{Q_4 V_4^2 - Q_1 V_1^2 - Q_2 V_2^2 + K Q_1 V_1^2}{2 Q_4 g}$$

where:

Q = discharges

V = horizontal velocities

K = bend loss factor

GEOPAK DRAINAGE WARNING MESSAGES

PREFERENCES

- **Error Opening GPK file:** The directory path for the location of the project's geometric database (gpk file) is not properly defined in the Drainage project preferences.

- **Error Opening Drainage Library:** The directory path for the location of the project's drainage library (dlb file) is not properly defined in the Drainage project preferences.

- **Close All Open Library Item Dialogs Before Exiting Librarian**: All edit dialogs must be saved or cancelled before exiting the drainage library dialog.

- **Unable to load path data:** The directory paths for any of the project components are invalid or do not exist as defined in the Project preferences dialog.

- **Unable to find shape file in preferences:** The roadway superelevation shapes dgn file is not at the location specified in the Project preferences dialog.

AREAS

- **No element meets the boundary level/symbology criteria:** The dgn element symbology specified as a search criteria for the area definition does not match any of the elements in the dgn file.

- **Area Creation Failed or Aborted - Area Must be Enclosed**: A drainage area must be closed shape element in Microstation.

- **No Boundary Elements Selected**: A boundary or borderline Microstation elements to define a drainage area were not selected.

- **Sum of Subareas = ## Exceeds Total Drainage Area = ## Do You wish to Reset Total** ?: The sum of the subareas entered in the Area>Subareas option exceeds the total area defined in the Area>Definition dialog box. GEOPAK will adjust the total area as the total sum of the subareas.

NODE:

- **Error Retrieving Cell:** The Microstation cell library containing the drainage node cells is not attached to the dgn file, it is not found or it is not specified in MS_Celllist variable.

- **NO element identified:** User has not selected/identified the Microstation element where the drainage node is going to take as a reference for placement while using the Tangent to Element or Tangent on Element option in the Node>location dialog.

- **Unable to determine PGL Elevation:** The station where the node is placed does not exist along the profile specified as a reference.

- **Unable to compute TIN Elevation**: The node is placed outside the boundary of the TIN file specified or the TIN file does not exist.

- **Error Finding D&C Manager:** The D&C Manager ddb file is not found at the location specified in the Project Preferences.

- **Error Computing Inlets Calculations for Network**: GEOPAK Drainage was not able to compute any hydraulics for the specified inlet. No flow is getting to the inlet. This could be caused by a wrong spread criteria definition in which the first slope of the spread criteria cross section is diverting the flow out of the inlet (negative slope) or the user has not defined a drainage area or discharge into the node.

- **Error Opening ACBOOK:** The D&C Manager ddb file is not found at the location specified in the Project Preferences.

- **Error Accessing TIN or Model:** The TIN file or Site Model gsf file do not exist at the directory path specified in the Project Preferences.

- **Error in Inlet By Pass Designations - Loop Detected:** User is bypassing flow into the same node.

- **Error using shapes**: GEOPAK was unable to extract information from the roadway superelevation shapes defined in the Project preferences. This could be caused by wrong definitions inside the shapes (chain and profile), the shape dgn file does not exist in the path specified in the Project preferences or there are not shapes defined at the node location.

LINK:

- **Link placement ERROR:** Link was not placed and stored in the drainage database (gdf file). This could be caused by a wrong definition (from and to node are the same) or a Microstation element to define the link is not selected.

- **Cannot Hold the Slope and From and To Node Elevations:** User can not specify hold values for the inverts and slope of the link/pipes at the same time. User should specify the inverts and let the slope to be calculated by the software.

- **Cannot Hold From Node Soffit and Invert Elevations:** User can not specify values for the soffit and invert elevations in the same FROM node. User should specify either one of them and let the software calculate the remaining elevation with the link rise or pipe diameter.

- **Cannot Hold To Node Soffit and Invert Elevations:** User can not specify values for the soffit and invert elevations in the same TO node. User should specify either one of them and let the software calculate the remaining elevation with the link rise or pipe diameter.

- **Cannot Hold More than Two Conditions: User** can not specify hold values for the inverts and slope of the link/pipes at the same time.

- **Rise Exceeds Maximum allowed by Profile Envelope:** The selected or designed link for the segment is encroaching in the profile envelope defined by the elevations and minimum depths of the connected nodes.

- **Downstream HGL sufficient for Hydraulic Jump:** GEOPAK Drainage does not calculate the exact location of a hydraulic jump but it can warn the user of its possible occurrence.

- **Minimum Slope used for Positive Drainage:** Link slopes were adjusted to the minimum specified slope in order to assure gravity/positive flow. Adverse slope pipes are not allowed.

- **Min Depth Exceeded at Upstream or Downstream of Link: Link** rise is encroaching the limits defined by the node elevation minus the minimum depth.

NETWORK:

- **Error Building Network :** An invalid network has been defined. Multiple outlet nodes or "loop" connections are not allowed.

- **Network Calculations Containing Profile are not up to date, Please Perform Network Calculations:** Shown results may not be up to date since possible edit operations have occurred in the network.

- **An Active Network must be selected:** The Drainage>Network>Active Network has not been selected in the project. Some reporting options required this definition at the user request.

- **Network Computations are not up to date show results anyway** ?: Shown results may not be up to date since possible edit operations have occurred in the network.

- **Network Calculations involving link are not up to date, Please perform Network Computations: Shown** results may not be up to date since possible edit operations have occurred in the network.

- **Errors Preclude Computations:** GEOPAK Drainage was unable to calculate the network system. This could be caused by some zero values in the node or link definitions (no flow into the nodes, minimum velocities equal to zero) or no drainage areas defined for inlets.

- **HGL Blowout at upstream end: The** hydraulic grade line is encroaching the limits defined by the node elevation minus the minimum freeboard value (defined in the Project Preferences).

- **Commingling Runoff Methods not supported:** GEOPAK Drainage can compute a network either using the Rational or SCS method for the drainage areas definitions.

- **Error Building Network - More than One Outlet found:** An invalid network has been defined. Multiple outlet nodes are not supported.

- **Error Building Network - Loop or Multiple Downstream Links:** An invalid network has been defined. Loop or closed circuits connections are not supported.

- **Error Building Network - More than five Incoming Links for Node:** GEOPAK Drainage does not allow more that five link/pipes connections in a node.

PROFILES

- **Unsuccessfully projecting to chain __ from link __:** The chain extends/limits does not include the link location.

- **Error Accessing TIN or Model:** The TIN does not exist at the directory path specified in the Project Preferences.

- **Drainage Profile __ contains Node __:** The Drainage Profile dialog must be closed to edit this node: The Drainage Profile dialog must be closed before editing a node that is part of the active profile.

- **Drainage Profile __ contains Link __:**The Drainage Profile dialog must be closed to edit this link: The Drainage Profile dialog must be closed before editing a link that is part of the active profile.

NAVIGATOR

- **No Queries Available for Selected Type:** There are no query options for the drainage element type selected.

- **No Elements Meeting Query were found:** An specific query has returned no results.

ROUTING

- **Unable to load computation file:** The ASCII file specified to store the computations was not previously created while in Append mode.

GEOPAK REPORT BUILDER VARIABLES

AREA

Area-ID: Drainage area name/identification stored in the drainage database (gdf file)

Area – Description: Drainage area user provided description.

Area - Time of Concentration: User provided or calculated time of concentration.

Area - Tc Used: Time of concentration used in the calculation for the drainage area discharge.

Area – Discharge: Computed discharge for the drainage area.

Area – Intensity: Intensity rainfall used as computed from the time of concentration and rainfall library.

Area - Composite C Value: Calculated composite C coefficient for the Rational method.

Area - Composite Area: Calculated composite area value for the Rational method.

Area - Total Subarea C Value: Calculated composite C value for only the delineated subareas.

Area - Total Subarea: Sum of the delineated subareas only.

Area - Remainder C Value: C coefficient of the area not included into the delineated subareas.

Area - Remainder Area: Calculated value equal to Total Drainage Area minus the Total Subareas.

Area - Subarea 1…20 Area: Area value of the 1..20 delineated subarea.

Area - Subarea 1…20 C Value: C coefficient of the 1..20 delineated subarea.

Area - Subarea 1…20 Description: Description of the 1..20 delineated subarea.

NODE

Node-ID: Drainage node name/identification stored in the drainage database (gdf file)

Node – Description: Drainage node user provided description.

Node – Type: Node type as defined in the drainage library. E.g. curb, grate, junction, other, curb-grate, headwall.

Node - Library Item Name: Node name as it is stored in the drainage library.

Node - Library Item Description: Node description as it is stored in the drainage library.

Node - Reference Chain: GEOPAK chain used to locate the node by station.

Node - Reference PGL: GEOPAK profile used to compute the elevation and longitudinal slope for the node.

Node - Reference TIN: GEOPAK DTM file used to compute the elevation of the node.

Node - Station : Station value computed at the node location using the Node - Reference Chain.

Node - Offset : Offset or distance value computed from the Node - Reference Chain to the node location. A negative value will indicate the left side of the chain.

Node - Location X Node/ Node - Location Y Node : Coordinate location of the drainage node.

Node – Rotation Node: Angle value in which the node is rotated.

Node - Cumulative Tc : Accumulated time of concentration at the node location.

Node - Supplied Discharge: Applied flow into the node. A discharge is entered directly without a drainage area definition.

Node - Cumulative Supplied Q : Accumulated directly-entered discharge in the node.

Node – Freeboard : Distance computed as node elevation minus hydraulic grade line elevation.

Node - Cumulative Discharge : Accumulated discharge/flow in the node.

Node - Cumulative Area : Accumulated drainage area in the node.

Node - Cumulative C Value : Accumulated C coefficient in the node.

Node - Cumulative Intensity: Accumulated intensity in the node.

Node - Hyd Center X / Node - Hyd Center Y : Coordinate location of the node hydraulic center.

Node - Elev Point X / Node - Elev Point Y : Coordinate location of the node elevation point .

Node - Origin X / Node - Origin Y : Coordinate location of the node cell origin.

Node - Minimum Depth / Node - Maximum Depth: Distance measured from the node elevation point. These distances paired with the next connected node creates en envelope in which the links can be placed.

Node – Depth : Distance computed from the node elevation point minus the node bottom elevation.

Node - Minimum Invert Elevation : Minimum invert elevation computed from the links arriving or leaving the node.

Node - Maximum Rise In : Maximum link size arriving into the node.

Node - Maximum Rise Out: Maximum link size leaving the node.

Node - Minimum Invert In / Node - Minimum Invert Out : Minimum invert elevation of the links arriving or leaving the node.

Node - Bottom Name : Name as stored in the drainage library of the bottom structure attached to the node.

Node - Bottom Description : Description as stored in the drainage library of the bottom structure attached to the node.

Node - Pay Item : D&C Manager item ID attached to the node.

Node - Bottom X / Node - Bottom Y : Coordinate location of the bottom structure attached to the node.

Node - Bottom Station: Station value computed at the bottom of the structure attached to the node using the Node - Reference Chain.

Node - Bottom Offset: Offset or distance value computed from the Node - Reference Chain to the bottom of the structure attached to the node . A negative value will indicate the left side of the chain.

Node - Bottom Elevation: Elevation of the bottom of the structure attached to the node.

Node - Sump Depth: Extra depth specified by the user added after computing the node bottom elevation.

Node - Is Link Base Flow: Returns a value of -1 if a Baseflow (offsite drainage discharge) option is added to the node.

Node - Link Base Flow Type: Returns a value of 1 if the Baseflow (offsite drainage discharge) is entered as a numeric value. Returns 0 if the baseflow discharge is entered as a predefined drainage area.

Node - Link Base Flow Area: Returns the name or ID of the drainage area entered as baseflow. Returns N/A if not.

Node - Link Base Flow Discharge: Returns the numeric value entered as baseflow discharge. Returns N/A if not.

INLET

Inlet - ID: Drainage inlet name/identification stored in the drainage database (gdf file).

Inlet – Description: Drainage inlet user provided description.

Inlet - By Pass Node ID : Node name in which by pass flow is directed from the present inlet.

Inlet – Type: Inlet type as defined in the drainage library. E.g. curb, grate, curb-grate.

Inlet - Profile Type : Inlet placement profile condition. Either On Grade or Sag.

Inlet – Discharge : Inlet computed discharge from the drainage area or supplied discharge.

Inlet – Capacity : Capacity of the inlet based on their physical properties and placement conditions.

Inlet - Max By Pass: Maximum by pass flow/discharge assigned to the inlet as a query value. It is not used for design purposes.

Inlet - By Pass Flow: Computed flow that the present inlet will bypass due to lack of capacity.

Inlet - By Pass Flow Into: Inlet name in which by pass flow is directed from the present inlet.

Inlet - Curb Length: Curb length as defined in the drainage library.

Inlet - Curb Depression: Curb Depression as defined in the drainage library.

Inlet - Curb Height: Curb Height as defined in the drainage library.

Inlet - Curb Depression Width: Curb Depression Width as defined in the drainage library.

Inlet - Slot Length: Slot Length as defined in the drainage library.

Inlet - Slot Width: Slot Width as defined in the drainage library.

Inlet - Grate Type: Grate Type as defined in the drainage library.

Inlet - Grate Length: Grate Length as defined in the drainage library.

Inlet - Grate Width : Grate Width as defined in the drainage library.

Inlet - Grate Area: Grate Area as defined in the drainage library.

Inlet Grate Perimeter: Grate Perimeter as defined in the drainage library.

Inlet - Grate Clog Area: Grate Clog Area as defined in the drainage library.

Inlet - Grate Clog Per Inlet: Grate Clog Per Inlet as defined in the drainage library.

Inlet - Max Ponded Depth: Maximum allowed spread depth. This is used a query value and not for design purposes.

Inlet - Max Ponded Width: Maximum allowed spread width. This is used a query value and not for design purposes.

Inlet - Longitudinal Slope: Slope along the direction of stationing defined by the user or a GEOPAK profile.

Inlet - Computed Ponded Width: Computed spread width.

Inlet - Computed Ponded Depth: Computed spread depth.

Inlet - Right Spread Intercept: : Computed spread width at the right side of the inlet.

Inlet - Left Spread Intercept: Computed spread width at the left side of the inlet.

Inlet - Length Required: Computed inlet required in order to handle all the discharge.

Inlet - Spread N: Manning's coefficient for the spread section.

Inlet - Composite Spread Slope: Composite slope across the inlet.

Inlet - Inlet to Gutter Offset: Distance applied when the inlet is set at a certain offset of the section considered for computing spread width and depth.

Inlet - Spread Slope 1…5: Slope of the different segments of the Spread Criteria cross section. Section 1 is the closest to the inlet.

Inlet - Spread Width 1…5: Width of the different segments of the Spread Criteria cross section. Section 1 is the closest to the inlet.

LINKS

Link – ID: Drainage link name/identification stored in the drainage database (gdf file).

Link – Description: Drainage link user provided description.

Link – Type: Link type. Either pipe or ditch.

Link - Upstream Node: Upstream node ID of the connecting link.

Link - Downstream Node: Downstream node ID of the connecting link.

Link – Shape: Link shape as defined in the drainage library. E.g. Arch, box, circular, ellipse or pipe arch.

Link – Material: Link material as defined in the drainage library. E.g. aluminum, concrete, plastic, or steel.

Link - Corr – Type: Link corrugation or type as defined in the drainage library. E.g. Standards, low profile, high profile, horizontal, or vertical.

Link - Library Item: Link name as it is stored in the drainage library.

Link - Number of Barrels: Number of parallel pipes going from one node to another.

Link - Actual Length: Link length measured from the graphical ends of the element.

Link - Hydraulic Length: Link length measured from hydraulic center to hydraulic center of each node.

Link - Manning's N Value: Manning's coefficient of the link.

Link – Slope: Link slope in percentage.

Link – Rise: Link rise in feet or meters.

Link – Span: Link span in feet or meters.

Link – Discharge: Link discharge/flow.

Link - Uniform Depth: Flow uniform depth at the link.

Link - Uniform Velocity: Uniform velocity computed at the link.

Link - Critical Depth: Critical depth computed at the link.

Link - Critical Velocity: Critical Velocity computed at the link.

Link - Friction Slope: Friction slope computed at the link.

Link - Critical Slope: Computed slope caused by critical flow at the link.

Link - Actual Depth Downstream: Downstream runoff depth at the link.

Link - Actual Depth Upstream: Upstream runoff depth at the link.

Link – Capacity: Link capacity under the present physical conditions.

Link - HGL Downstream/ Link - HGL Upstream: Hydraulic grade elevation at the link ends.

Link - EGL Downstream/ Link - EGL Upstream: Energy grade line elevation at the link ends.

Link - Upstream Junction Loss: Junction loss value at the upper portion of the link.

Link - Soffit Upstream /Link - Soffit Downstream: Soffit or crown elevation at the link ends.

Link - Invert Upstream/Link - Invert Downstream: Invert elevation at the link ends.

Link - Minimum Rise: Minimum link rise used for design purposes.

Link - Maximum Rise: Maximum link rise used for design purposes.

Link - Minimum Slope/ Link - Maximum Slope: Minimum/maximum slope at which links would be placed.

Link - Minimum Velocity/Link - Maximum Velocity: Minimum/maximum velocity at which links would be placed.

Link - Design Size: Return a Yes/No value if GEOPAK Drainage will design the size of the links.

Link - Design Barrels: Return a Yes/No value if GEOPAK Drainage will consider parallel links from node to node.

Link - Actual Velocity Downstream/Link - Actual Velocity Upstream : Velocities computed at the link ends.

Link - Pay Item: ID of the Design and Computation Manager item associated with the link as defined in the drainage library.

Link - Library Description: Link description as it is stored in the drainage library.

Link - Iterative Velocity: Computed velocity at the link after 10 iterations.

Link - Full Velocity: Computed velocity assuming that the links are flowing full.

Link - Upstream Freeboard: Computed elevation from the upstream node minus the assigned freeboard to the node.

This page left blank intentionally.