State of Florida
Department of Transportation

FDOT Civil 3D Drainage Stormwater & Sewer Analysis (SSA)

User Training Guide

April, 2021

PRODUCTION SUPPORT OFFICE | CADD
TALLAHASSEE, FLORIDA
http://www.fdot.gov/cadd/

FDOT Civil 3D Drainage

Stormwater & Sewer Analysis (SSA)

Description

This is a 2-day training course to include:

- Exporting Pipe Network Data from Civil 3D in a Hydraflow Storm Sewers (STM) format.
- Network Layout in SSA
- Displaying a CAD File as a Background
- Network Layout in SSA
- Running an Analysis on the Network
- Displaying Analysis Results
- Displaying a Profile Plot of a Pipe Run
- Editing Nodes and Links
- Exporting a STM File from SSA
- Importing a STM File in Civil 3D
- Viewing SSA Produced Values in Civil 3D Network Parts
- Displaying HGL and EGL in a Profile View in Civil 3D

Objectives

The student will have a basic understanding of using Civil 3D Pipe Network Data in SSA workflow.

Audience

This course is intended for Drainage Engineers, EIs, CAD Designers.

Prerequisites

Participants need to have a basic understanding of AutoCAD and Civil 3D. Participant is required to complete: 2018-2019 FDOT Civil 3D Drainage course

Duration:

16 Hours

Professional Credit Hours:

16 PDHs

Note PDH Credit will only be available with Instructor lead or Computer Based Training (CBT) thru Learning Curve.

TABLE OF CONTENTS

1 GETTING STARTED

DESCRIPTION

This course was developed for FDOT Drainage Designers using AutoCAD Civil 3D 20XX, the FDOT Civil 3D 20XX State Kit, and Autodesk's Storm and Sanitary Analysis (SSA) software. This course provides a complete workflow for taking a AutoCAD Civil 3D Pipe Network into Autodesk's SSA software, analyzing the network performance, editing pipes and structures, and returning the revised network to AutoCAD Civil 3D. This section provides some basic information to get started using Autodesk's SSA software.

OBJECTIVES

This course will train the user on how to use Autodesk AutoCAD Civil 3D, FDOT Civil 3D 20XX State Kit, and Autodesk's SSA software for the purpose of:

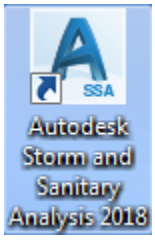
- Exporting Pipe Network Data from Civil 3D in a Hydraflow Storm Sewers (STM) format.
- Network Layout in SSA
- Displaying a CAD File as a Background
- Network Layout in SSA
- Running an Analysis on the Network
- Displaying Analysis Results
- Displaying a Profile Plot of a Pipe Run
- Editing Nodes and Links
- Exporting a STM File from SSA
- Importing a STM File in Civil 3D
- Viewing SSA Produced Values in Civil 3D Network Parts
- Displaying HGL and EGL in a Profile View in Civil 3D
- User Interface Basics
- Plan View Features

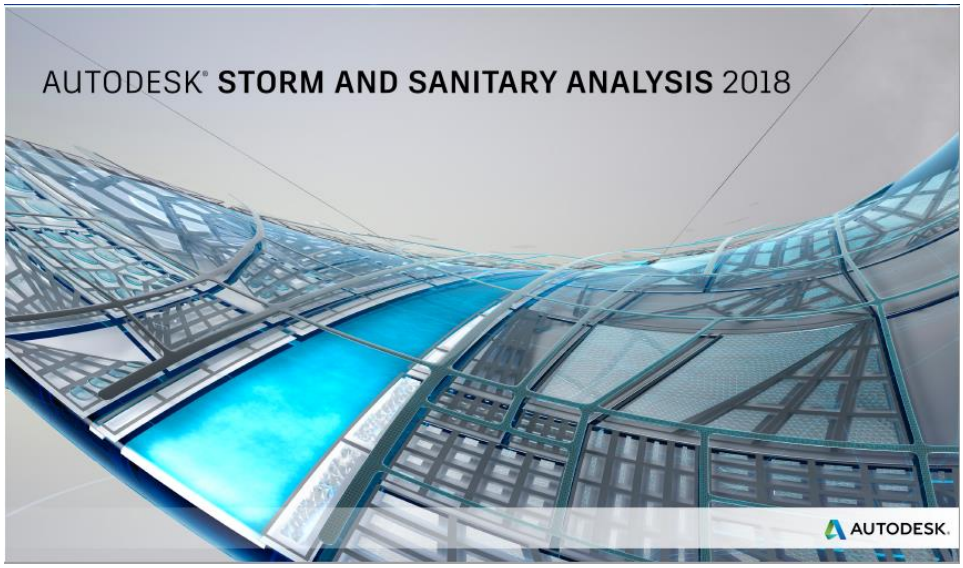
COURSE SUPPORTING FILES

The exercises for each module are independent of one another and can be used without having to complete the exercises in previous modules. Typically, a dwg file or spf file is provided as the starting data for your exercise. All files used in this course should be unzipped to C:\Civil 3D 20XX Projects\.

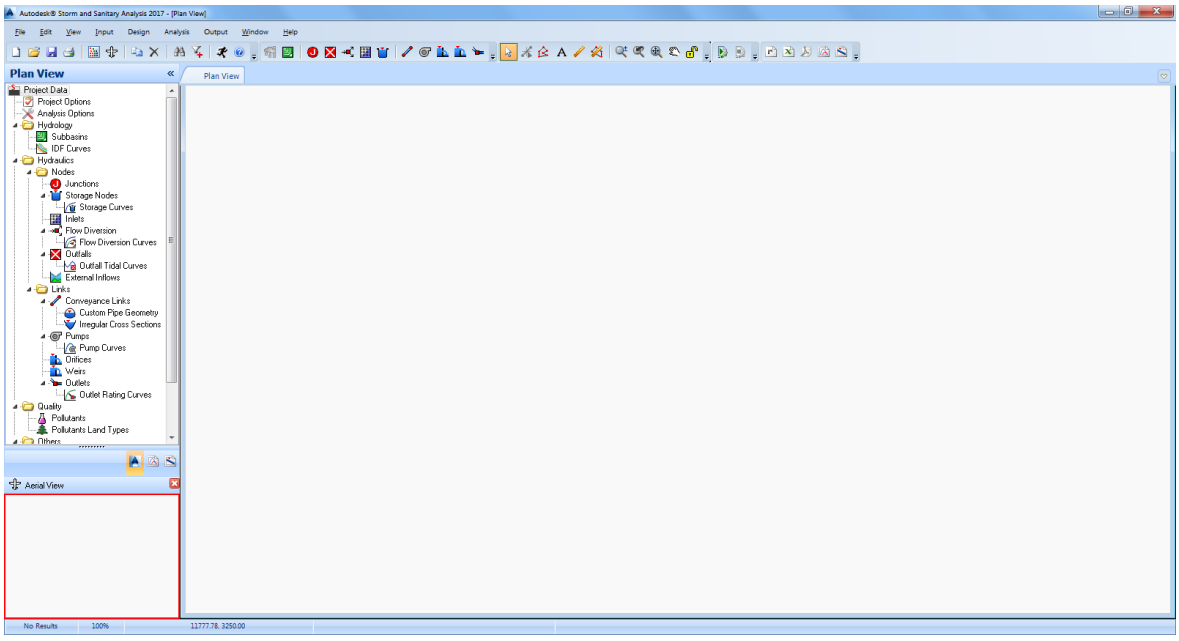
STARTING AUTODESK'S SSA SOFTWARE

From your desktop double click on the the SSA 20XX shortcut icon. A screen will appear to let you know that Autodesk SSA software is loading.



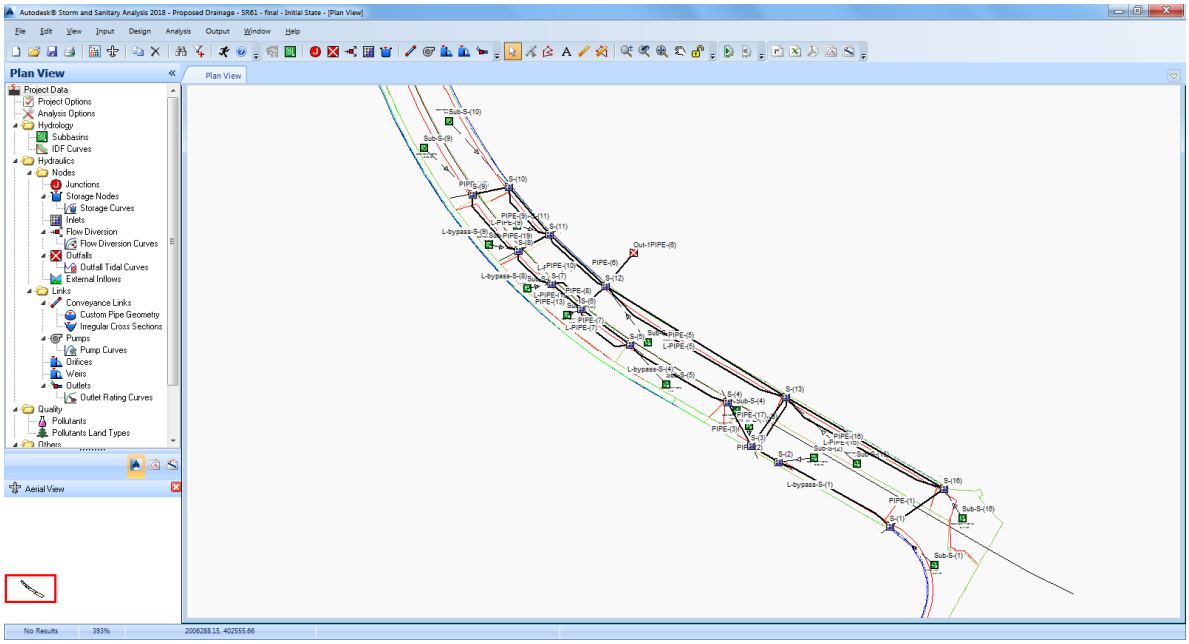


Once the Autodesk SSA software has loaded, a screen should display as shown below.

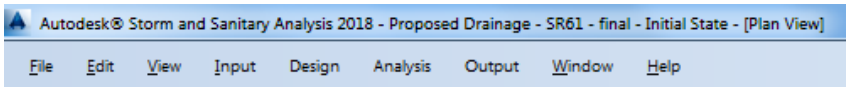


USER INTERFACE BASICS

The Autodesk SSA user interface is shown below. The user interface consists of the components: *Menu Bar*, *Toolbars*, *Plan View*, *Data Tree*, *View Tabs*, and *Status Bar*.

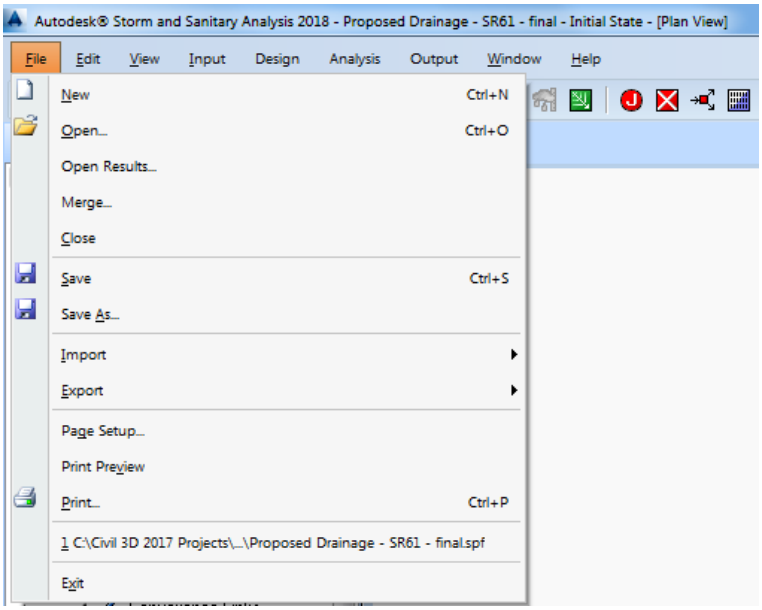


MENU BAR

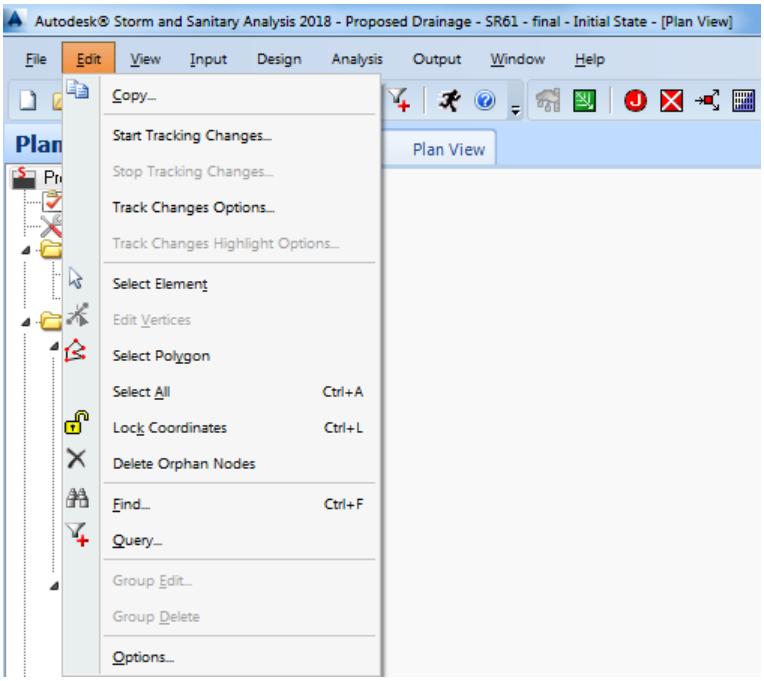


The menus are grouped by command type:

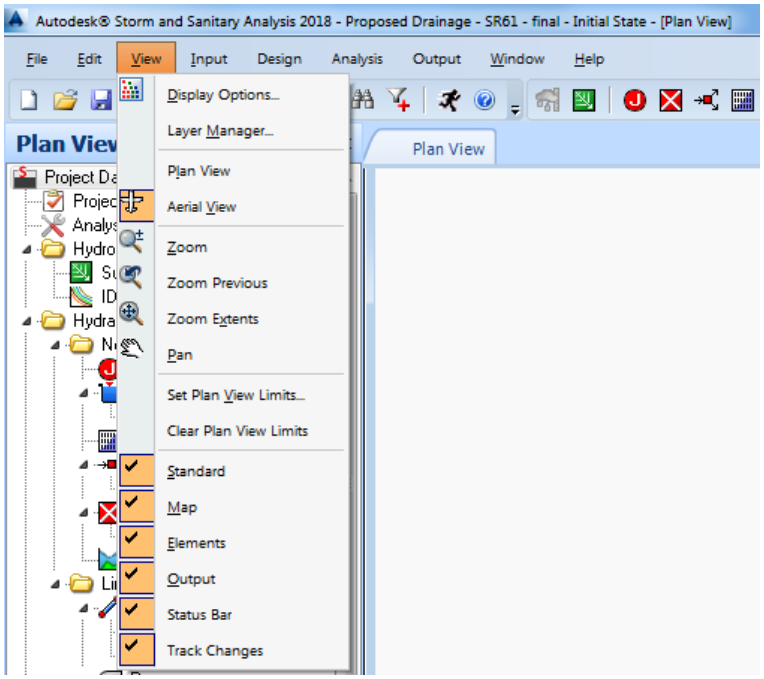
- **File** Commands for opening and saving data files; importing geo-referenced orthophotos; importing and exporting AutoCAD drawings, AutoCAD Hydraflow files, LandXML files, GIS shape files, EPA SWMM input files, and XPSWMM input files; and for printing.



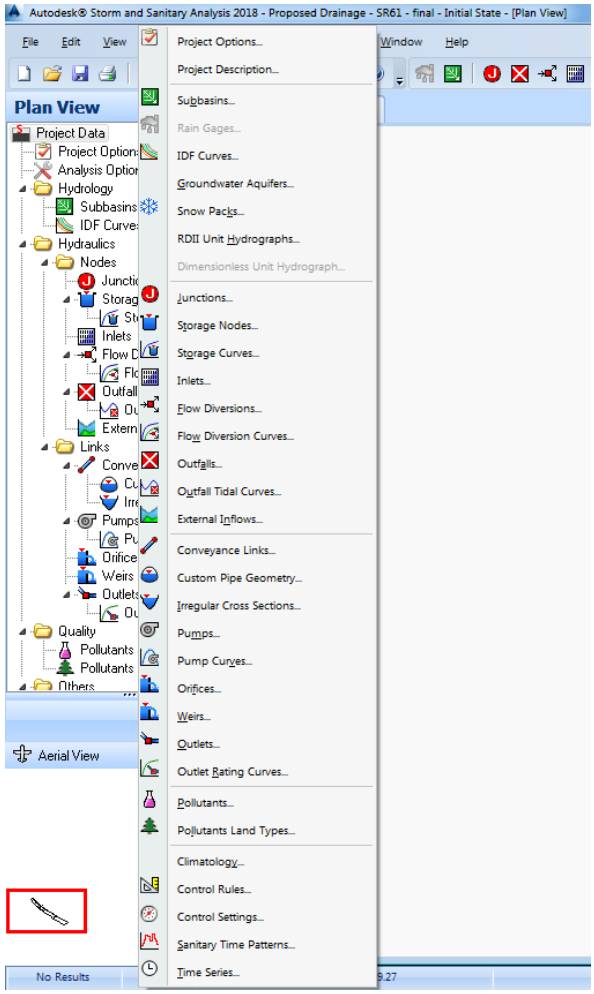
- **Edit** Commands for editing and copying.



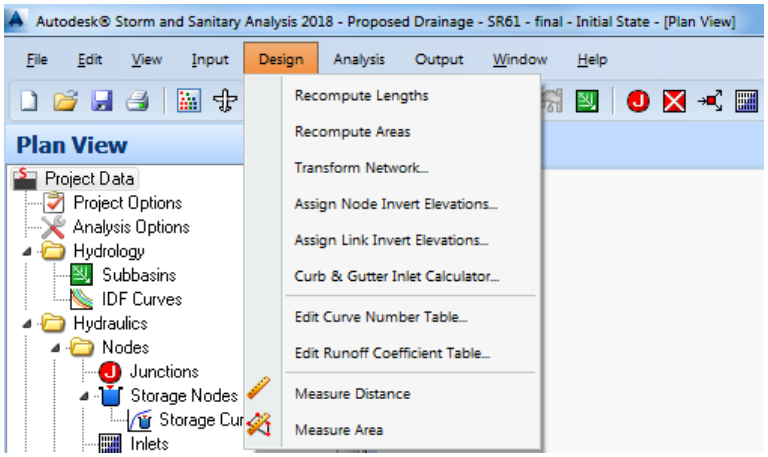
- **View** Commands for viewing, configuring Plan View display options and displaying toolbars.



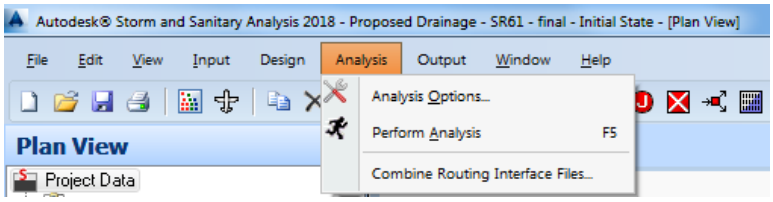
- **Input** Access to all drainage network element dialog boxes that define the model input data.



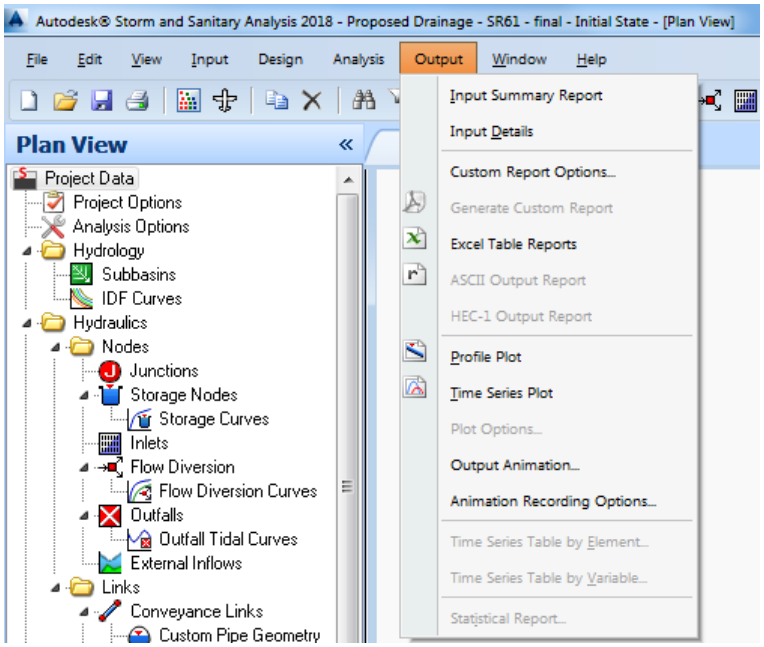
- **Design** Commands for performing design functions on the network model.



- **Analysis** Commands for defining analysis parameters and for performing the network model analysis.

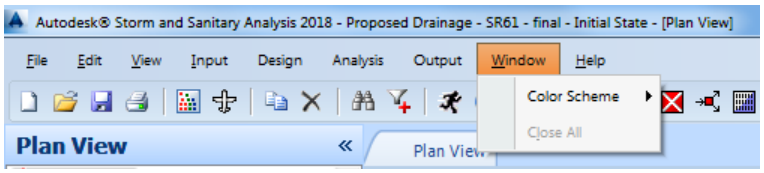


- **Output** Commands for displaying the network model analysis results as graphical plots and reports.

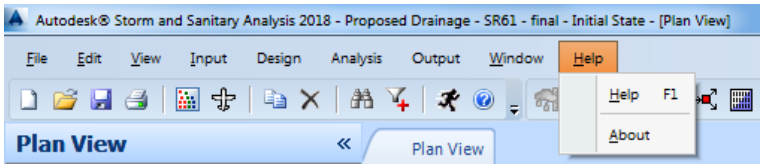


  - *Time Series Plot* Commands for the analysis results time series plot, if this plot is being displayed. Otherwise, this menu is not available.

- **Window** Commands for arranging and selecting windows within theapplication workspace.

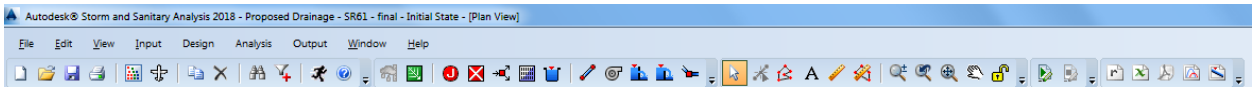


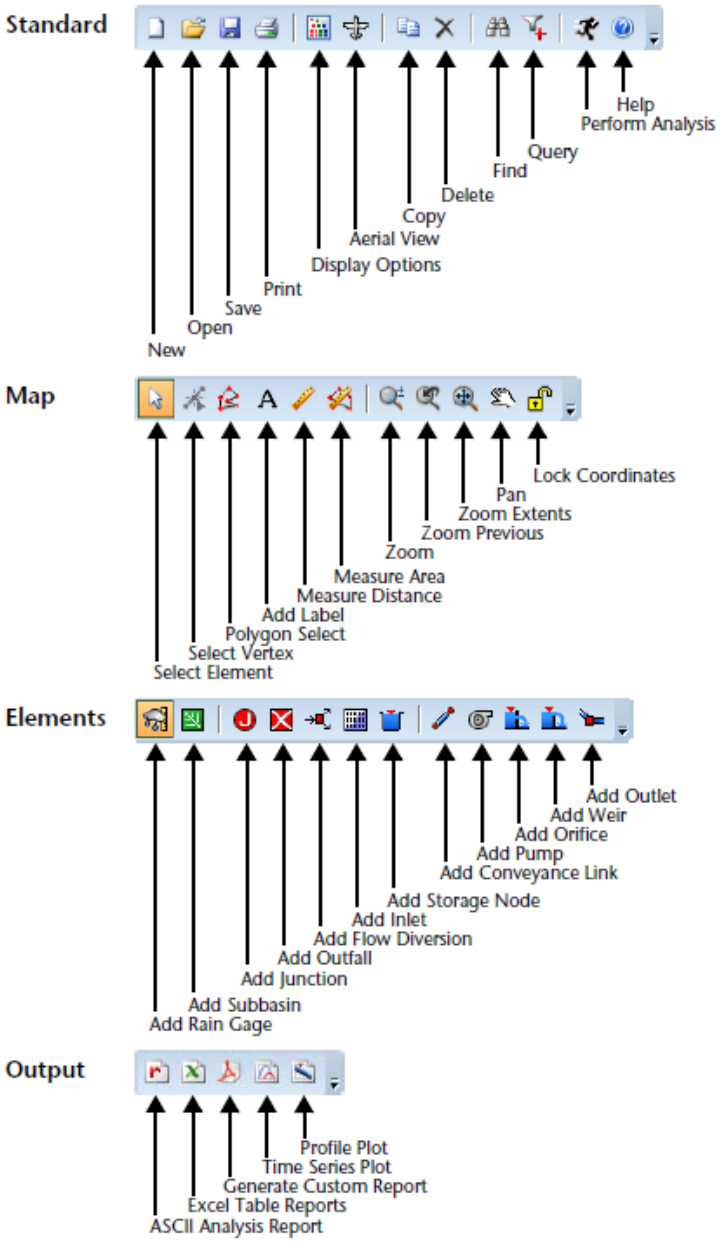
- **Help** Commands for getting help.



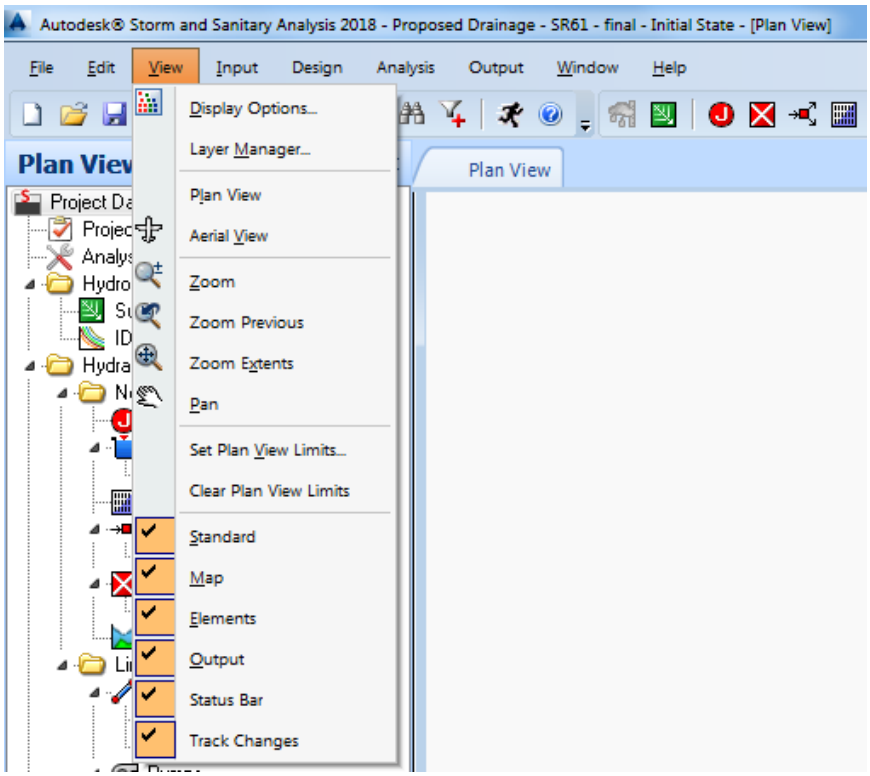
TOOLBARS

The software provides numerous toolbars that give you quick access to many commands and features. If you forget what a particular toolbar button accomplishes, point the cursor at the toolbar button. After you pause over the button, a tooltip will be displayed providing a description of what the button performs.





All toolbars are docked underneath the menu bar. To display a toolbar, select the **VIEW** menu and the appropriate toolbar. As shown in the following figure, a check mark will appear displayed adjacent to the toolbar to indicate that it is visible



MAP TOOLBAR

- **Select Element Tool** - The **SELECT ELEMENT** tool is used to select network elements in the Plan View. The following can be done with the **SELECT ELEMENT** tool:
  - ✓ Double-clicking a network element will display the appropriate element dialog box with the selected element current, allowing you to edit its properties. For example, double-clicking a junction will display the Junctions dialog box with the selected junction current.
  - ✓ Network elements (e.g., subbasins, junctions, rain gages, etc.) can be dragged to new locations. Select the element and then drag the element to the desired location. To lock the elements in place so that they cannot be accidentally moved, select **EDIT ➤ LOCK COORDINATES** and a check mark will show up to **signify that the elements (and corresponding vertices) are locked in place.**
  - ✓ Network elements can be deleted. Select the element and then press Delete. Note that the software will confirm the delete command before it is performed. Note that you can disable the delete confirmation check in the Options dialog
  - ✓ Network elements can be converted to a different type of element. Select the element and then right-click. From the displayed context menu, select **CONVERT TO**. Then, select the element type to convert to. Only data that is common to both element types will be preserved after the element is converted.

- **Edit Vertices Tool** - The **EDIT VERTICES** tool is used to grip edit subbasin, channel, and pipe vertex points in the Plan View. To grip edit a network element vertex, the following can be done:

  1. Select the subbasin, channel, or pipe to edit using the SELECT ELEMENT tool.
  2. Change to Vertex Editing mode by clicking the EDIT VERTICES tool, selecting EDIT ➤ EDIT VERTICES, or right-clicking the element and choosing EDIT VERTICES from the displayed context menu.
  3. The mouse pointer will change into an arrow tip, and any existing vertices on the selected network element will be displayed as small squares. The currently selected vertex will be displayed as a filled square. To select a particular vertex, click it.
  4. To add a new vertex, right-click and select ADD VERTEX from the displayed context menu or press Insert on the keyboard.
  5. To delete the currently selected vertex, right-click and select DELETE VERTEX from the displayed context menu or press Delete from the keyboard.
  6. To move a vertex to another location, select the vertex and drag it to its new location.
  7. While in Vertex Editing mode, you can begin editing the vertices for other network elements by clicking an element. To leave Vertex Editing mode, right-click and select QUIT EDITING from the displayed context menu or select another tool from the toolbar.

- **Select Polygon Tool** - The **SELECT POLYGON** tool is used to select a group of elements in the Plan View for editing or deleting. The following can be done with the **SELECT POLYGON** tool:

  1. To select a group of network elements, click the SELECT POLYGON tool or choose EDIT ➤ SELECT POLYGON.
  2. Draw a polygon around the area of interest on the Plan View by clicking for each point of the polygon.
  3. Close the polygon selection by either double-clicking or pressing Enter.

- **Measure Distance Tool** - The **MEASURE DISTANCE** tool is used to measure a distance, such as overland flow length or subbasin equivalent width from the Plan View. Measuring a distance can be done in the following manner:

  1. Click the MEASURE DISTANCE tool or choose DESIGN ➤ MEASURE DISTANCE.
  2. On the Plan View, click the mouse to draw a line to be measured. Click to define each vertex of the line.
  3. While creating the line, it is not unusual to make a mistake digitizing by clicking at the wrong location.
  4. Press the Backspace key to delete the last segment. Alternatively, right-click and select DELETE LAST SEGMENT from the displayed context menu.
  5. Press the Esc key to cancel the command. Alternatively, right-click and select CANCEL from the displayed context menu.
  6. Double-click or press Enter to complete the line being measured. Alternatively, right-click and select DONE from the displayed context menu. The software then displays the length of the line.

Note Pressing Esc while measuring cancels the command.

- **Measure Area Tool** - The **MEASURE AREA** tool is used to measure an area, such as soil type area or land use area from the Plan View. Measuring an area can be done in the following sequence:

  1. Click the MEASURE AREA tool or choose DESIGN ➤ MEASURE AREA.
  2. On the Plan View, click the mouse to draw a polygon outline of the area being measured. Click to define each vertex of the polygon.
  3. While creating a polygon boundary, it is not unusual to make a mistake digitizing by clicking at the wrong location.
  4. Press the Backspace key to delete the last segment. Alternatively, right-click and select DELETE LAST SEGMENT from the displayed context menu.
  5. Press the Esc key to cancel the command. Alternatively, right-click and select CANCEL from the displayed context menu.
  6. Double-click or press Enter to complete the polygon being measured. Alternatively, right-click and select DONE from the displayed context menu. The software will automatically close the polygon and display the area of the polygon.

Note Pressing Esc while defining the polygon cancels the command.

- **Zoom Tool** - The **ZOOM** tool is used to zoom in and out of the viewing area of the Plan View, Profile Plot, and Time Series Plot. Zooming can be done with the following sequence:

  - ✓ With the **ZOOM** tool, clicking the view zooms the viewing area in around the point by a factor of two. Holding down the Shift key while clicking causes the view to zoom out.
  - ✓ With the **ZOOM** tool, a rectangle can be dragged around a portion of the viewing area to zoom in to that region.
  - ✓ If your mouse has a scroll wheel, you can scroll the wheel to zoom in and out. This is especially handy when you have another tool active and you do not want to switch tools.

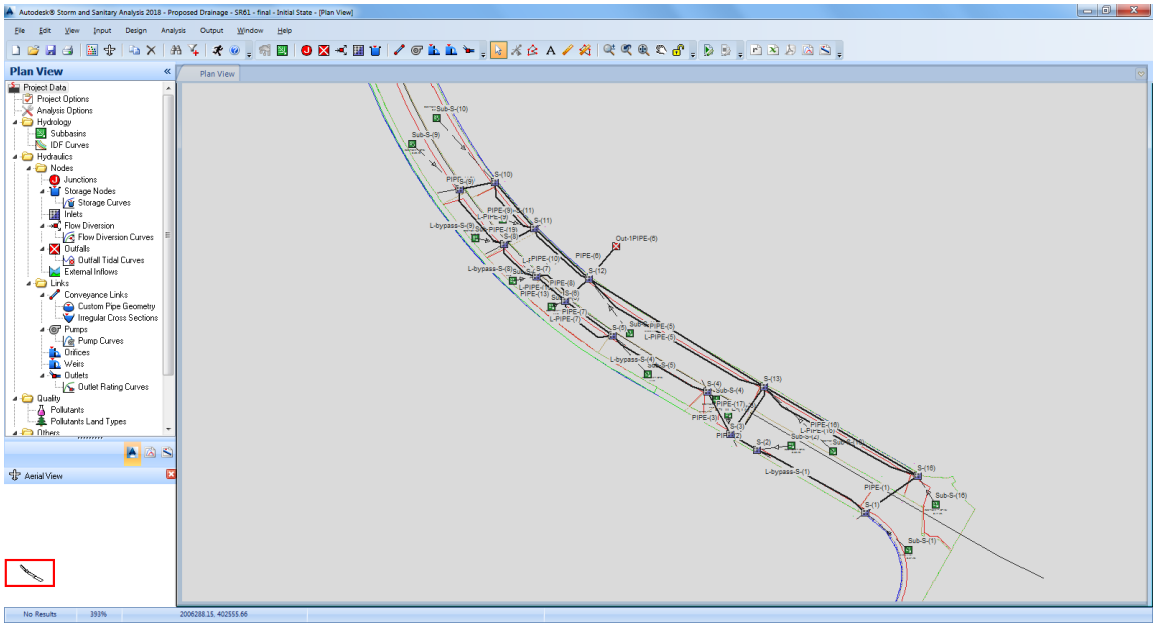
- **Zoom Previous Tool** - The ZOOM PREVIOUS tool is used to display the previous view.
- **Zoom Extents Tool** - The **ZOOM EXTENTS** tool is used to zoom out to the full extents of the Plan View, Profile Plot, and Time Series Plot.
- **Pan Tool** - The **PAN** tool is used to pan the viewing area of the Plan View, Profile Plot, and Time Series Plot. Panning can be performed in the following manner:

  - ✓ With the **PAN** tool, holding down the left mouse button while dragging moves the viewing area.
  - ✓ If your mouse has a scroll wheel (or a middle button), hold it down and drag to pan the viewing area. This is especially handy when you have another tool active and you do not want to switch tools.

- **Lock Coordinates** - The **LOCK COORDINATES** icon is used to lock and unlock the network elements from being moved within the Plan View, preventing the accidental movement of elements.

PLAN VIEW

The Plan View, as shown in the following figure, provides a layout view (or top view) of the stormwater or wastewater network system. The individual elements that make up the network are displayed. The Plan View also allows you to graphically layout the drainage network system.



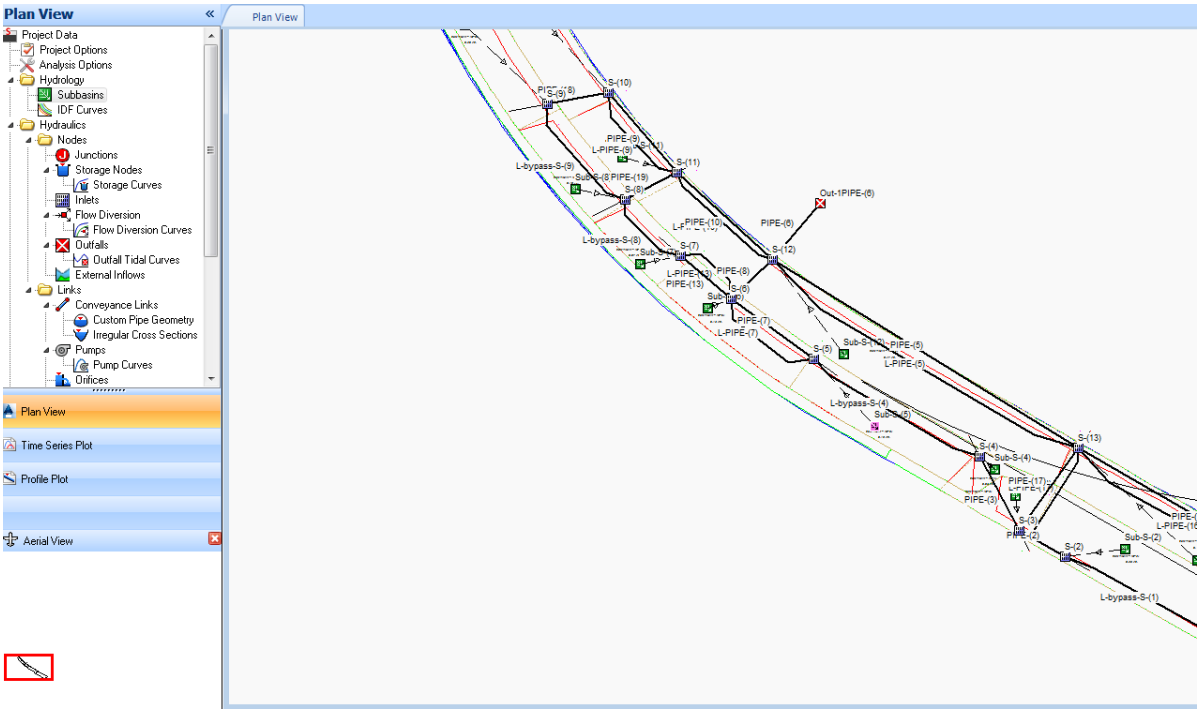
Key items of the Plan View include:

- The location of drainage network elements and the distances between them do not have to conform to the actual physical scale; they can represent a schematic diagram of the network.
- Elements can have their properties, such as flooding depth at junction nodes or flow velocity in channels and pipes, displayed using different colors. This color coding can be controlled using the displayed legend.
- New network elements can be added directly to the Plan View and existing elements can be selected for editing, deleting, and repositioning.
- Background images, such as geo-referenced TIFF aerial orthophoto images and maps, can be displayed as a background for reference.
- CAD drawing files, such as a street drawing, can be imported and displayed as a background for reference.
- The displayed drainage network can be zoomed into and panned from one position to another.
- Junction nodes, channels, and pipes can be displayed at different sizes to indicate a particular property. Flow directional arrows can be displayed on channels and pipes to indicate the direction of flow from the analysis results.
- Element ID labels and numerical property values can be displayed adjacent to network elements.
- The Plan View can be printed, copied to the Microsoft Windows clipboard for pasting into a Word document, or exported as an Autodesk® AutoCAD® drawing file for report generation.

AERIAL VIEW

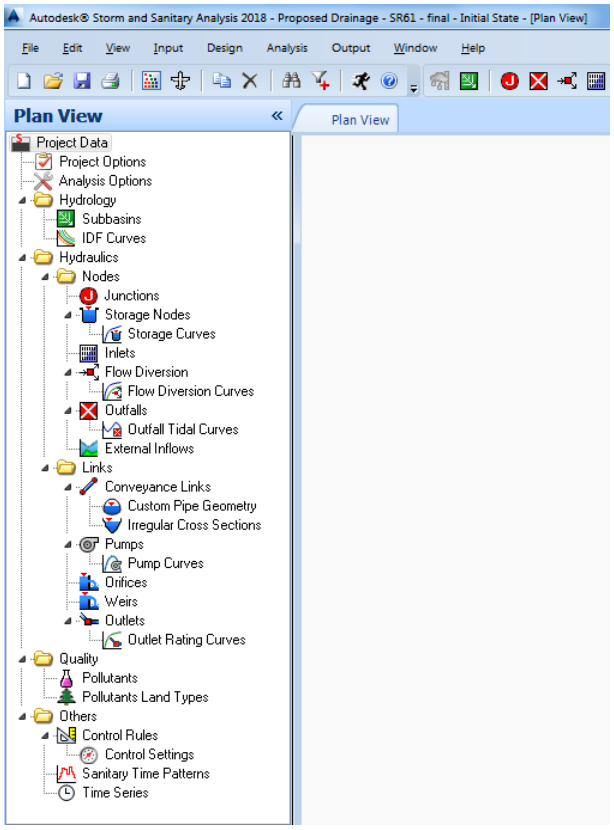
The Aerial View, as shown at the bottom of the Navigation Pane as in the following figure, displays the entire network and identifies the current view with a rectangular view box. You can use the Aerial View to change the view in the Plan View window quickly by dragging the view box. As you drag the view box to another location, the display in the Plan View will be redrawn accordingly.

To display the Aerial View, select **VIEW ➤ AERIAL VIEW** or click the **AERIAL VIEW** icon on the standard toolbar. To hide the Aerial View, select **VIEW ➤ AERIAL VIEW** or click the **AERIAL VIEW** icon again.



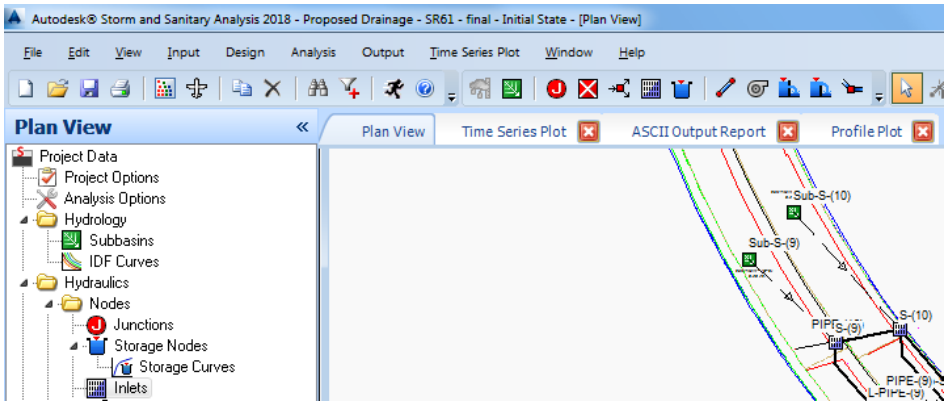
DATA TREE

The Data Tree, as shown below, provides access to all of the data elements contained in a project. Select the **EXPAND** ⊞ and **COLLAPSE** ⊟ icons to see the hierarchical representation of the data associated with a modeling project. The contents of the data tree vary, depending on what data is defined.



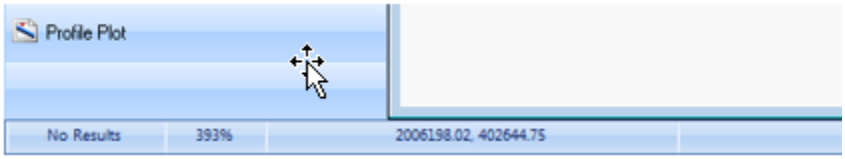
VIEW TABS

The view tabs at the top of the screen allow you to quickly move from one view to another by clicking the tab of your choice. As shown in the following figure, the view tab of the active view has a foreground color, tabs for inactive views have a background color. To close a tab, click the ☒ symbol on the tab. The view tabs at the top of the screen allow you to quickly move from one view to another.

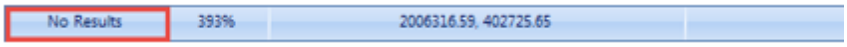


STATUS BAR

The Status Bar, as shown below, appears at the bottom of the application, and provides information about the network model or the task you are working on.

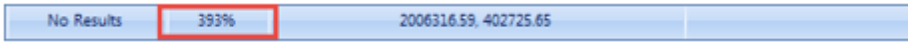


- **Run Status** This section of the status bar indicates whether the simulation results are available.Three different states can be shown:



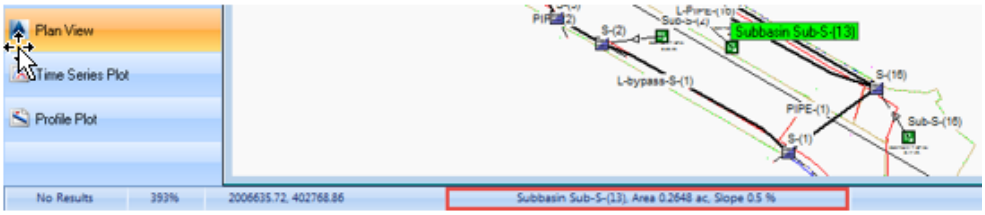
  - ✓ *No Results* Analysis results are not available. Re-run the simulation to get the results.
  - ✓ *Results Complete* Analysis results available.
  - ✓ *Results Differ* Analysis results are available, but may be invalid because the model data has been modified. Re-run the simulation to get the results.

- **Zoom Level** This section of the status bar indicates what the current zoom level is for the Plan View. A value of 100% indicates that the Plan View is zoomed to the extent of the model.



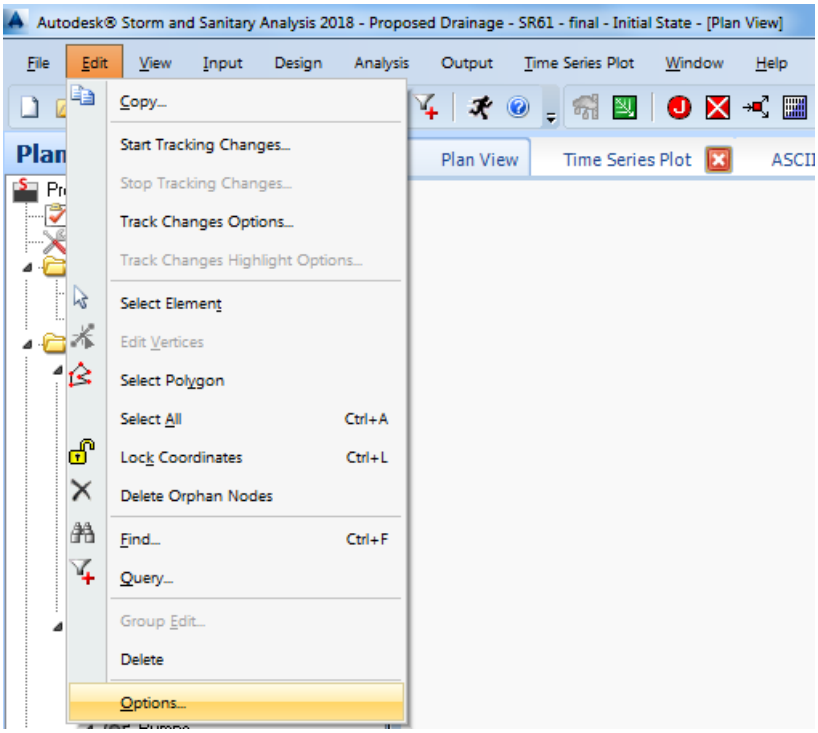
- **XY Coordinates** This section indicates the current coordinates of the mouse pointer.

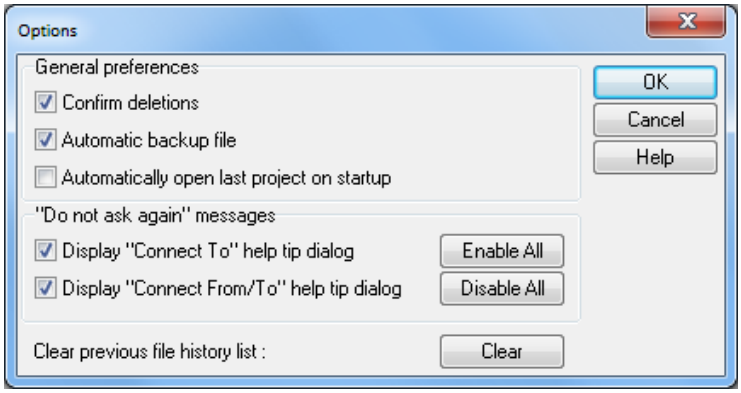
- **Element Information** As shown in the following figure, this section details information regarding the element directly underneath the mouse pointer.



PROGRAM OPTIONS

The program configurations are defined by the Options dialog box, as shown in the following figures. Select **EDIT ➤ OPTIONS** to display the Options dialog box.





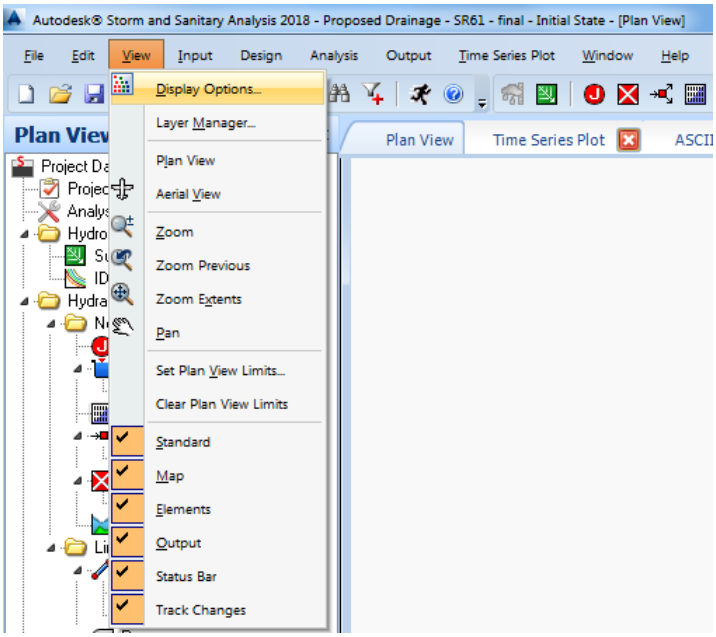
The Options dialog box controls the program configuration options. The following configuration settings can be specified in the Options dialog box.

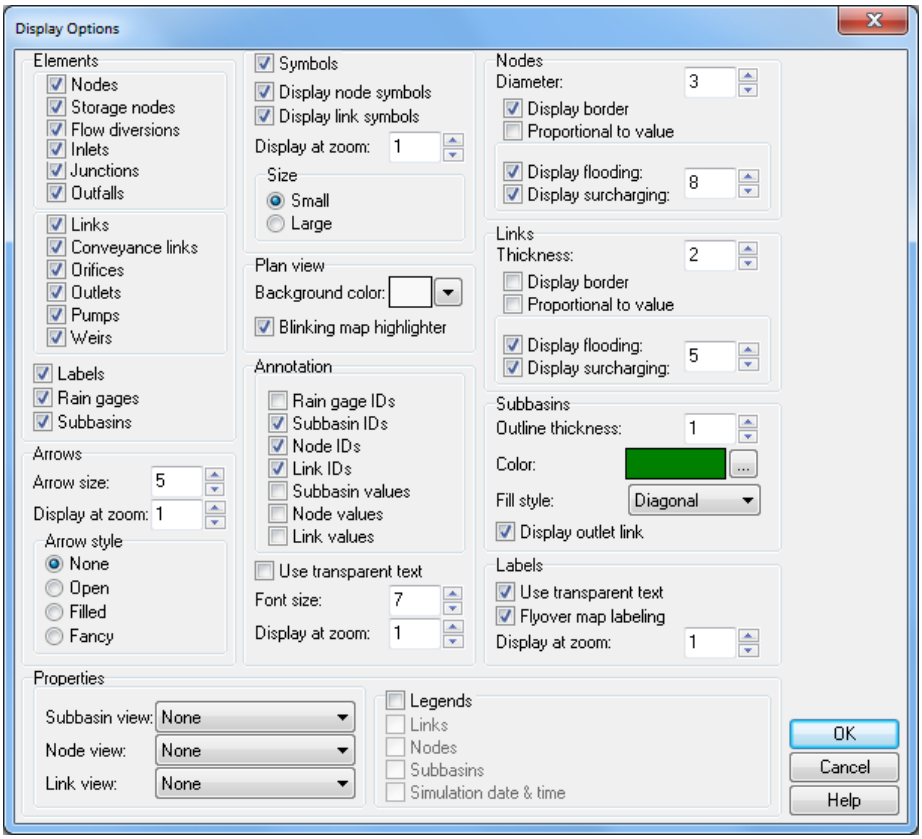
- **Confirm Deletions** This check box causes a confirmation dialog box to be displayed before deleting any network element.
- **Automatic Backup File** This check box causes the software to automatically save a backup copy of a newly opened project. The default backup file extension is .BCK so as not to overwrite an Autodesk® AutoCAD® Civil 3D® or Autodesk® AutoCAD® Map 3D drawing file backup.
- **Automatic Open Last Project on Startup** This check box causes the software to automatically load the last saved project upon program startup.
- "**Do not ask again" Messages** These check boxes control whether the Help Tip dialog boxes will be displayed.
- **Clear Previous File History List** Clicking Clear causes the list of most recently opened projects to be cleared from the File Menu.

DISPLAY OPTIONS

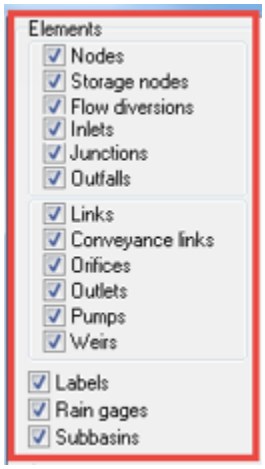
The Plan View display can be modified using the Display Options dialog box, as shown in the following figure. To show the Display Options dialog box:

- Select VIEW ➤ DISPLAY OPTIONS
- Click the DISPLAY OPTIONS icon
- Right-click the Plan View to display the context menu and then select DISPLAY OPTIONS





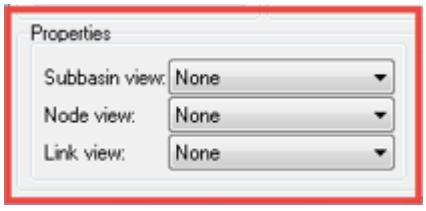
- **Elements** - The Elements section controls the display of network elements. Select the check box adjacent to each element type to control the display of that element type.

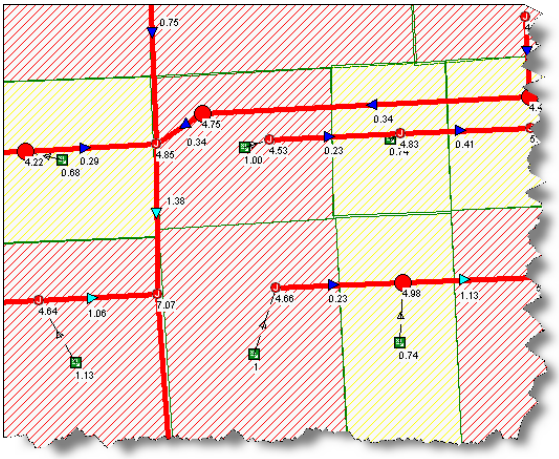


- **Arrows** - The Arrows section controls how element directional and flow arrows are displayed on the Plan View. Note that flow arrows will only be displayed after an analysis has been performed, and a computed output parameter has been selected for display. Otherwise, element directional arrows will point from the starting node to the ending node.

  - ✓ **Arrow Size**: This spin control sets the element directional and flow arrow size.
  - ✓ **Display at Zoom**: This spin control defines the minimum zoom ratio at which element directional and flow arrows should be displayed. Element directional and flow arrows will be hidden at zoom ratios smaller than this.
  - ✓ **Arrow Style**: This radio group defines the shape of element directional and flow arrow to display. Select **NONE** to hide the arrows.

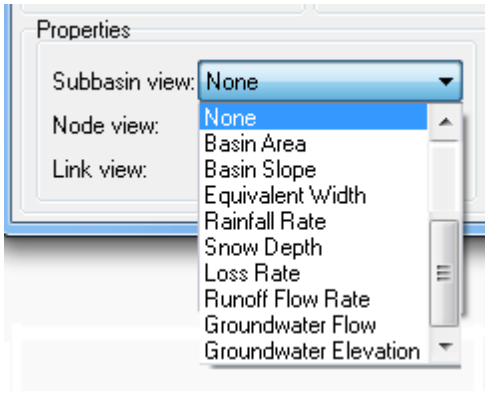
- **Properties** - The Properties section defines how subbasins, nodes, and links are color-coded based upon the selected input or output property. From the element drop-down list, select the property to be displayed as a color-coded attribute. The Plan View then displays the elements similar to the following figure.





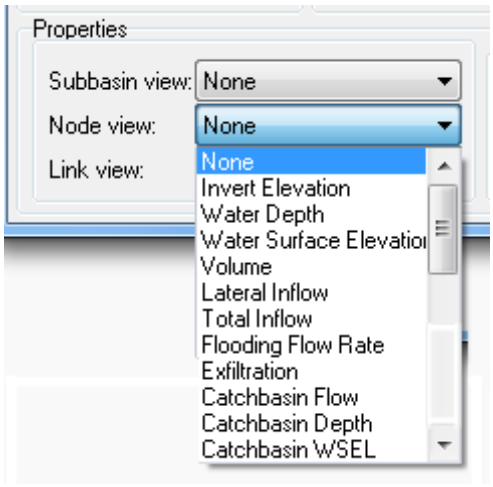
The following element properties are available for selection. Note that element output properties (i.e., Total Inflow, etc.) are only available after the analysis has been performed. Select **NONE** if you do not want to have any elements to be colored based upon a property.

✓ **Subbasins View**



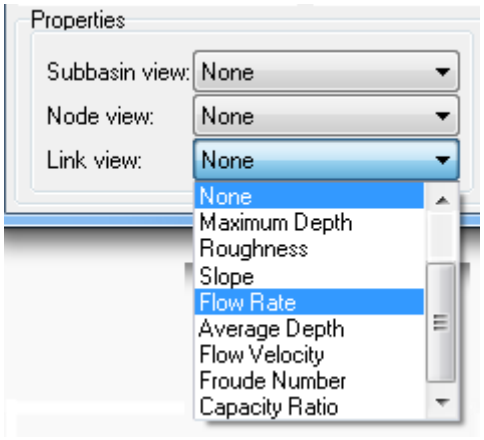
- *None*
- *Basin Area* Drainage area (acres or hectares)
- *Basin Slope* (ft/ft or m/m)
- *Equivalent Width* *(ft or m)*
- *Rainfall Rate* (in/hr or mm/hr)
- *Snow Depth* *(in or mm)*
- *Loss Rate* Infiltration + evaporation (in/hr or mm/hr)
- *Runoff Flow Rate* *(cfs or cms)*
- *Groundwater Flow* Groundwater flow into drainage network (cfs or cms)
- *Groundwater Elevation* *(ft or m)*

✓ **Node View**



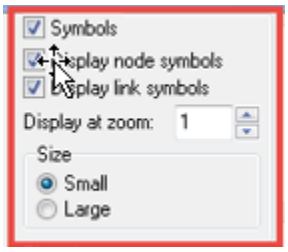
- *None*
- *Invert Elevation* (ft or m)
- *Water Depth* Water depth above node invert (ft or m)
- *Water Surface Elevation* (ft or m)
- *Volume* Water volume held in storage (including ponded water, ft^3 or m^3)
- *Lateral Inflow* Runoff + all other external inflows (cfs or cms)
- *Total Inflow* Lateral inflow + upstream inflows (cfs or cms)
- *Flooding Flow Rate* Surface flooding (inflows lost from the system
- *Exfiltration*
- *Catchbasin Flow*
- *Catchbasin Depth*
- *Catchbasin WSEL*

✓ **Link View**



- *None*
- *Maximum Depth* Maximum water depth (ft or m)
- *Roughness*
- *Slope* (ft/ft or m/m)
- *Flow Rate* (cfs or cms)
- *Average Depth* Average water depth (ft or m)
- *Flow Velocity* (ft/sec or m/sec)
- *Froude Number*
- *Capacity Ratio* Ratio of Depth to Full Depth

- **Symbols -** The Symbols section determines which element types are represented with special symbols on the Plan View.



✓ **Symbols**: If checked, then simple circles will be used.

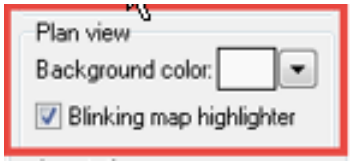
✓ **Display Node Symbols**: If checked, then special node symbols will be used. Otherwise, simple circles will be used.

✓ **Display Link Symbols**: If checked, then special link symbols will be used.

✓ **Display at Zoom:** This spin control defines the minimum zoom ratio at which symbols should be displayed. Symbols will be hidden at zoom ratios smaller than this.

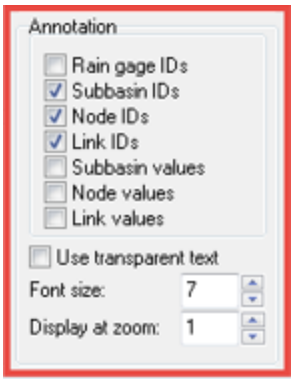
✓ **Size**: This radio group allows you to specify the symbol size to use.

- **Plan View** - These options affect the general display options of the Plan View.



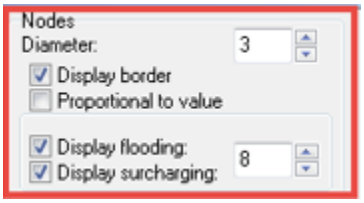
  - ✓ **Background Color**: The drop-down entry allows you to select the background color for the Plan View.
  - ✓ **Blinking Map Highlighter:** This check box controls whether the selected element on the Plan View is to blink.

- **Annotation** - The Annotation section controls what element IDs and specified values are to be displayed on the Plan View.



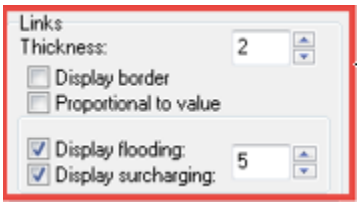
  - ✓ **Rain gage IDs**: If checked, then Rain gage IDs will display.
  - ✓ **Subbasin IDs:** If checked, then Subbasin IDs will display.
  - ✓ **Node IDs:** If checked, then Node IDs will display.
  - ✓ **Link IDs:** If checked, then LinkIDs will display.
  - ✓ **Subbasin values:** If checked, then Subbasin values will display.
  - ✓ **Node values:** If checked, then Node values will display.
  - ✓ **Link values:** If checked, then Link values will display.
  - ✓ **Use Transparent Text:** This check box will display the labels with a transparent background. Otherwise, an opaque background will be used.
  - ✓ **Font Size:** This spin control sets the annotation font size.
  - ✓ **Display at Zoom:** This spin control defines the minimum zoom ratio at which labels should be displayed. Labels will be hidden at zoom ratios smaller than this.

- **Nodes** - The Nodes section controls how nodes are displayed on the Plan View.



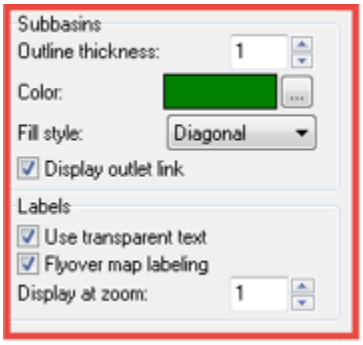
  - ✓ **Diameter:** When mapping input data or output results on to the nodes (e.g., invert elevation, water depth, total inflow, etc.), this spin control defines the default node diameter (or width) in pixels. When not mapping input data or output results on to the nodes, the nodes are displayed as their element icon.
  - ✓ **Display Border:** When mapping input data or output results on to the nodes, this check box defines if a border should be drawn around each node. This is recommended for light-colored backgrounds.
  - ✓ **Proportional to Value:** When mapping input data or output results on to the nodes, this check box specifies if the node diameter should increase as the viewed parameter increases in value.
  - ✓ **Display Flooding:** After the analysis is complete, this check box defines if a flooded node should be displayed on the Plan View in the color of blue. The adjacent spin control regulates the size of the displayed flooded and surcharged nodes.
  - ✓ **Display Surcharging:** After the analysis is complete, this check box defines if a surcharged node (bolted manhole cover) should be displayed on the Plan View in the color of red. The adjacent spin control regulates the size of the displayed flooded and surcharged nodes.

- **Links** - The Links section controls how links are displayed on the Plan View.



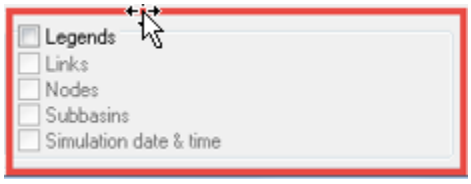
  - ✓ **Thickness:** When mapping input data or output results on to the links (i.e., diameter, flow rate, etc.), this spin control defines the link thickness in pixels.
  - ✓ **Display Border**: When mapping input data or output results on to the links, this check box defines if a black border should be drawn around each link. This is recommended for light-colored backgrounds
  - ✓ **Proportional to Value**: When mapping input data or output results on to the links, this check box specifies if the link thickness should increase as the viewed parameter increases in value.
  - ✓ **Display Flooding**: After the analysis is complete, this check box defines if a flooding link should be displayed on the Plan View in the color of blue. The adjacent spin control regulates the width of the displayed flooded and surcharged links.
  - ✓ **Display Surcharging**: After the analysis is complete, this check box defines if a surcharged pipe should be displayed on the Plan View in the color of red. The adjacent spin control regulates the width of the displayed flooded and surcharged links.

- **Subbasins** - The Subbasins section controls how subbasin areas are displayed on the Plan View.

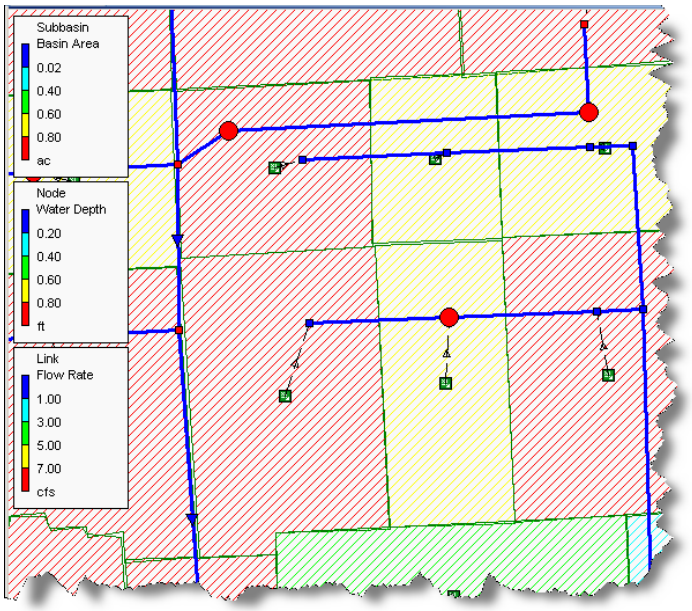


  - ✓ **Outline Thickness**: This spin control defines the thickness of the line used to draw the subbasin boundary. Specify a thickness of zero if no boundary is to be displayed.
  - ✓ **Color**: This color panel allows you to select the color to display the subbasin as. click the [...] browse button to display a color selection dialog box, which is used to change the color for the subbasins.
  - ✓ **Fill Style**: This radio button group specifies the style used to fill the interior of the subbasin area.
  - ✓ **Display Outlet Link**: This check box denotes whether a dashed line is to be drawn between the subbasin centroid and the subbasin's outlet node (or outlet subbasin).
  - ✓ **Labels**: The Labels section controls how labels are displayed on the Plan View.
  - ✓ **Use Transparent Text**: This check box will display the labels with a transparent background. Otherwise, an opaque background will be used.
  - ✓ **Flyover Map Labeling**: This check box will cause the element ID label and the value of the selected property to be displayed adjacent to the element whenever the mouse is placed over an element on the Plan View.
  - ✓ **Display at Zoom**: This spin control defines the minimum zoom ratio at which annotation should be displayed. Annotation will be hidden at zoom ratios smaller than this.

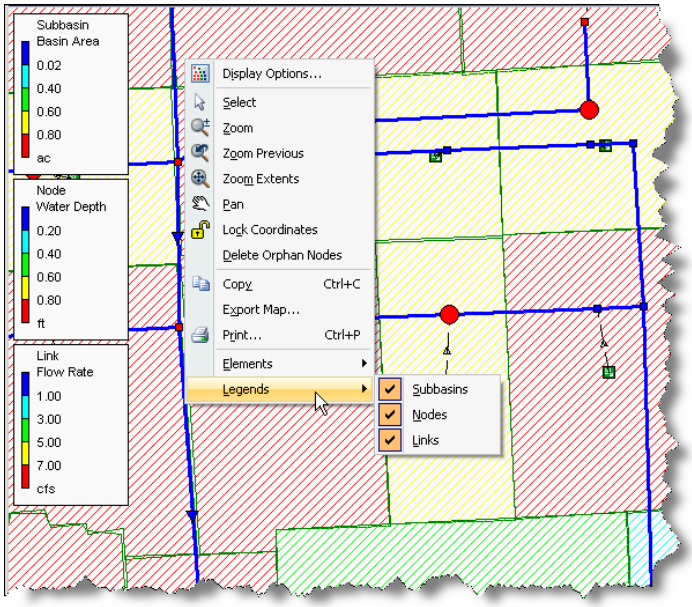
- **Legends** - The Legends section defines which element property legend should be displayed on the Plan View, as shown in the following figure. The legends display colors that are associated with a range of values for the element property being viewed.



Separate legends exist for subbasins, nodes, and links. If a particular element property is set to **NONE**, then that element's legend will not be displayed. Also, an option is available for displaying the date and clock time of the simulation period being viewed on the map.

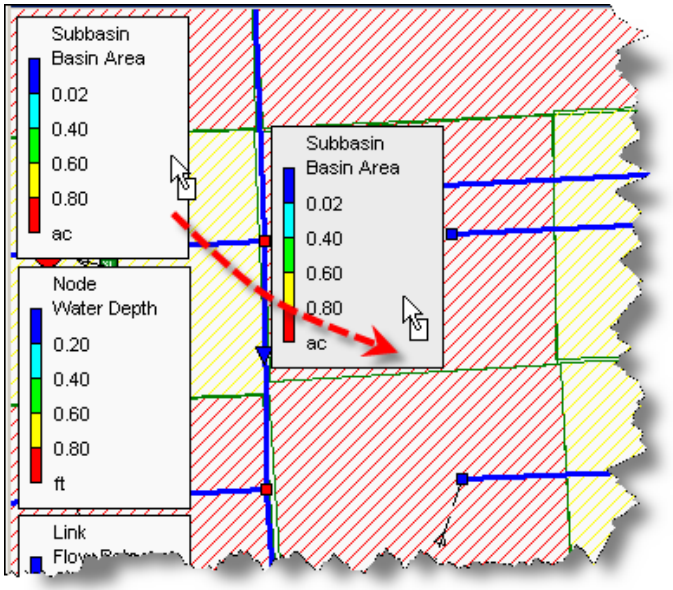


To turn off a legend that is displayed in the Plan View, double-click it. To re-display a legend that was turned off, right-click the Plan View. The software will display a context menu, which allows you to turn on the display of the legend.

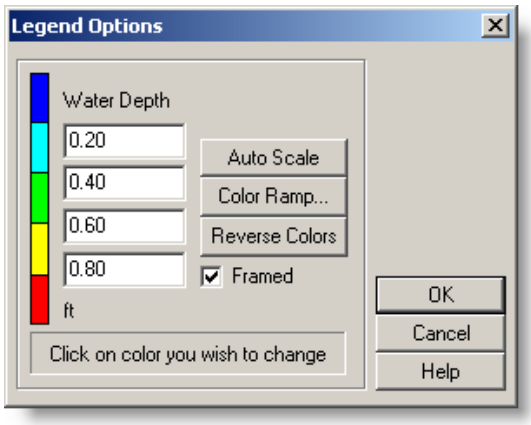


To re-display a legend that was turned off, right-click the Plan View and select the legend(s) to turn on or off

To move a legend to another location on the Plan View, click the legend with the left mouse button and drag the legend to its new location and then release the mouse.



To edit a legend, right-click the legend. The software then displays the Legends Options dialog box, as shown in the following figure.

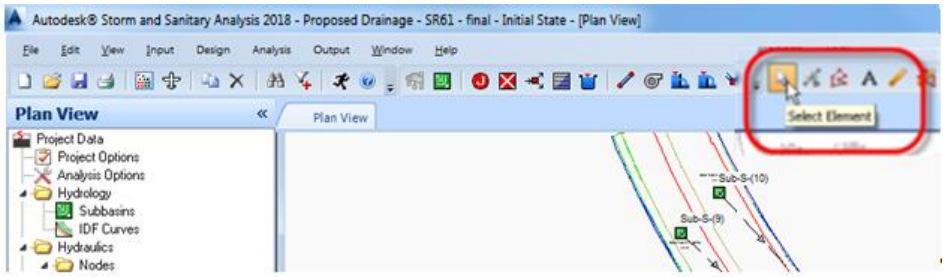


The Legend Options dialog box allows you to define a set of numerical ranges to which different colors are assigned for viewing a particular parameter for the elements on the Plan View. The following options are available:

- ✓ Water Depth values, entered in increasing downward order, are used to define the color ranges. All four edit fields do not need values specified.
- ✓ To change a color, click its color band in the dialog box. The Color dialog box will be displayed, allowing you to select a new color.
- ✓ Click the Auto Scale button to automatically assign ranges based on the minimum and maximum values attained by the parameter being displayed for the current time period.
- ✓ Click the Color Ramp button to select from a list of built-in color schemes.
- ✓ Click the Reverse Colors button to reverse the ordering of the current selection of colors (the color in the lowest range becomes that of the highest range and vice versa).
- ✓ Select the **FRAMED** check box if you want a frame drawn around the legend.

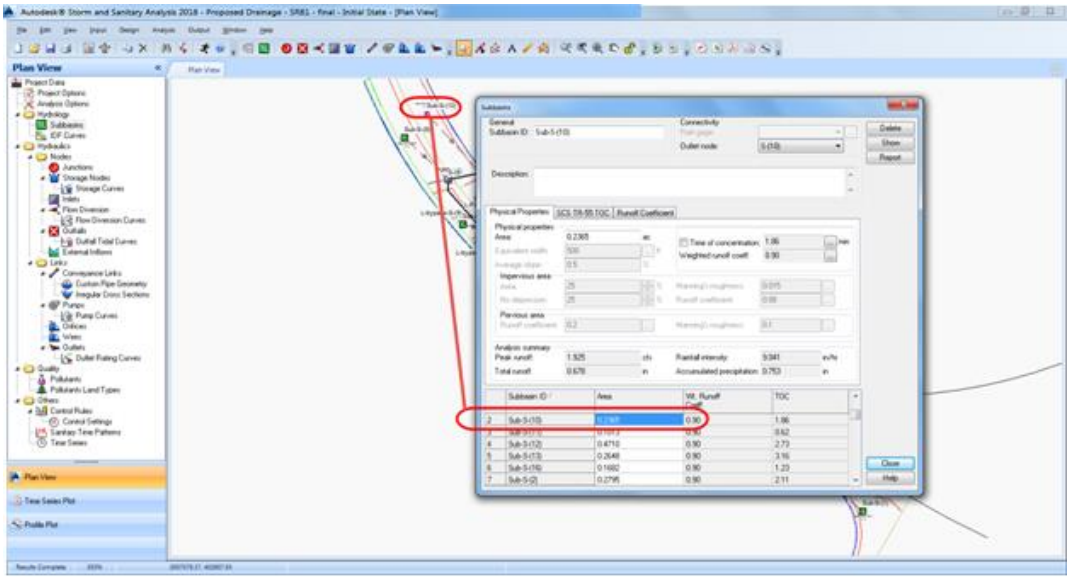
INPUT DATA DIALOG BOXES

Note that the software uses interactive dialog boxes for editing network input data. Double-click an element from the Plan View using the **SELECT ELEMENT** tool as shown in the following figure. The appropriate element dialog box is then displayed. Select the Select Element tool and then double-click a network element to display the corresponding element dialog box.



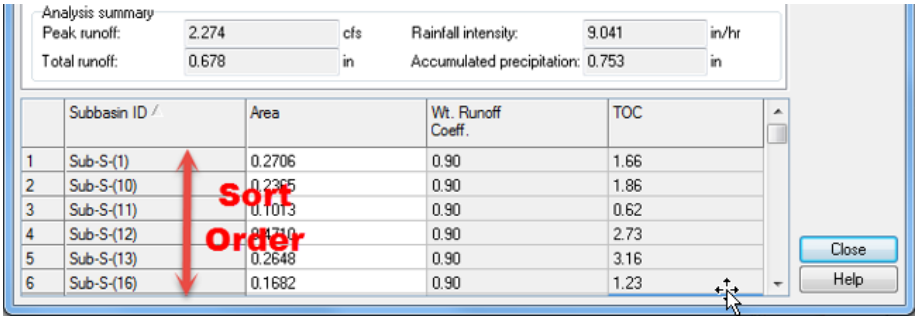
The network element dialog boxes are modeless, allowing you to keep them displayed while selecting other elements in the network from the Plan View.

For example, to examine manhole rim elevations, you can keep the Junctions dialog box displayed and select different junctions from the Plan View. Upon selecting a new junction, the software automatically updates the input (and output) data regarding the selected element, as shown in the following figure.



In most network element dialog boxes, the network element data is displayed in a table, allowing you to easily browse, edit, copy, and paste data. In addition, clicking the column header at the top of the table allows you to sort the data in descending (or if clicked twice, ascending) order, as shown in the following figure.

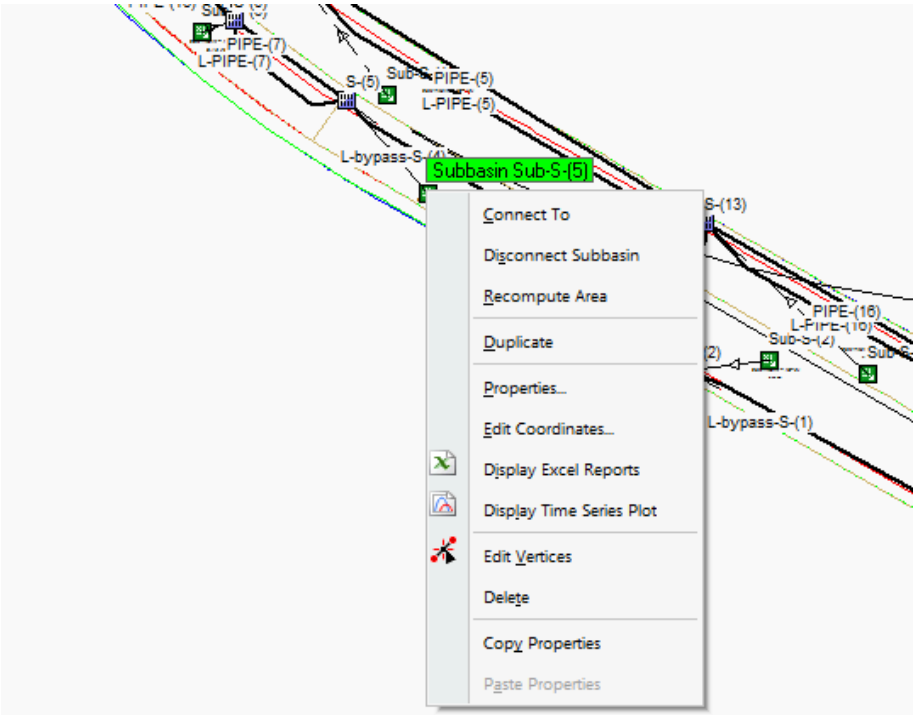
For example, using this feature allows you to quickly find "outliers" in the defined element data where there may have been a data input blunder. Sorting the column allows you to see if there are any elements with unusual values or perhaps a missing value.



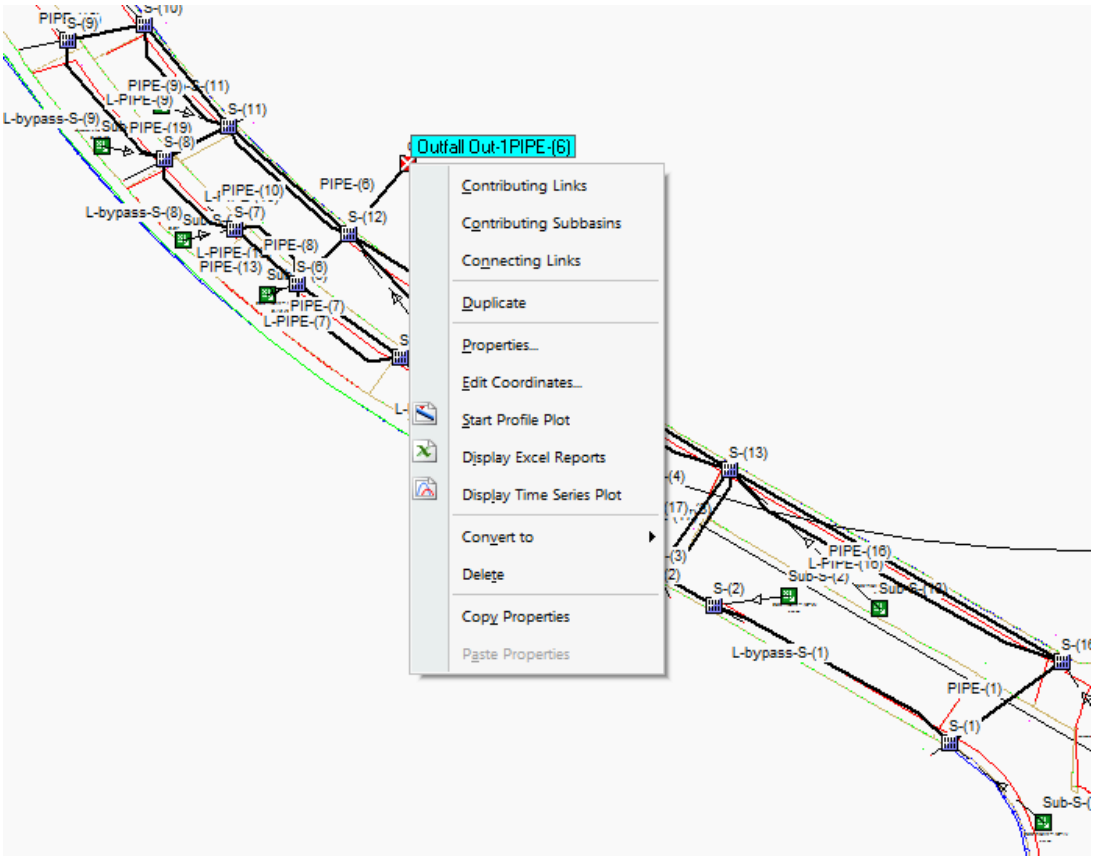
Multiple network element dialog boxes can be displayed, if desired. However, the computer monitor may become somewhat cluttered with dialog boxes. If you have dual monitors at your computer, you can grab the element dialog boxes and drag them to the other monitor allowing you to more effectively edit the network. Once completed with the data input for a particular network element, click the Close button.

RIGHT-CLICK CONTEXT MENU

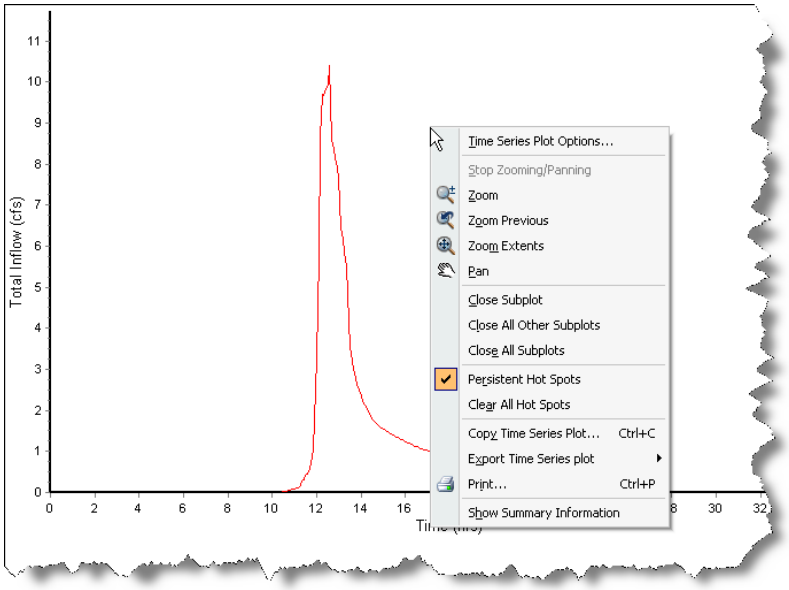
To provide you additional ease at using the software, the right-click context menu option is used extensively throughout the software. To display the right-click context menu, click using the right mouse button. An example of the right-click context menu for the Plan View is shown in the following figure.



The available options shown on the right-click context menu can change, based upon the context of what you are pointing at. For example, when pointing the cursor at a Subbasinl on the Plan View, only those commands corresponding to Subbasins are presented in the displayed right-click context menu. Moving the cursor away from any network elements, only commands corresponding to the Plan View are presented in the right-click context menu.



The right-click context menu is available throughout the software, from the Plan View, Data Tree, Time Series Plot, Profile Plot, etc.



[PAGE INTENTIONALLY LEFT BLANK]

2 SSA & CIVIL 3D WORKFLOW INTRODUCTION

DESCRIPTION

This chapter introduces the SSA and Civil 3D Workflows and the steps required to prepare Civil 3D Content for use in SSA.

OBJECTIVES

In this chapter, you will learn about : In this chapter, you will learn about:

- Network Layout in SSA
- FDOT Design Criteria used in this Manual
- Migration Requirements from Civil 3D to Storm & Sanitary Analysis.

CHAPTER SETUP

Run the Chapter 2 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX** Projects folder.

Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

OVERVIEW OF NETWORK LAYOUT IN SSA

When laying out a drainage network you have two strategies to choose from. The first, start in SSA with Catchment Areas and manually add nodes and links. This approach may be more appropriate when designing a schematic network that is not to scale. The second, start in Civil 3D with a Pipe Network then export to the Hydraflow Storm Sewers STM format file and import the STM file to SSA. The example in this training guide uses the second strategy.

The workflow is outlined below.

- Define the default options and element properties to use in the analysis model.
- Import Pipes and Structures in the STM file from Civil 3D
- Edit the properties of the elements that make up the system.
    - ✓ Connect Subbasins to Inlets
    - ✓ Convert Junctions to Inlets where necessary. Use Civil 3D Migration Settings to .
- Define the analysis options.
- Run the analysis.
- View the analysis results.
- Adjust the design
- Rerun the analysis

DESIGN CRITERIA FROM THE FDOT DRAINAGE MANUAL

➢ *TOC Minimum is 10 min.*

FDOT Drainage Manual, 3.5.1 Time of Concentration: Minimum allowable time of concentration is 10 minutes.

➢ *Pipe Slope and Minimum Velocity*

FDOT Drainage Manual, 3.6.1 Pipe Slopes: Use a physical slope that will produce a velocity of at least 2.5 feet per second (fps) and no greater than 15 fps when the storm drain is flowing full.

For pressure flow storm drain systems, the minimum physical slope is 0.1 percent.

➢ *Pipe Size and Length*

FDOT Drainage Manual, 3.10.1 Pipe Size and Length: The minimum pipe size for trunk lines and laterals is 18 inches. The minimum pipe diameter for all proposed exfiltration trench pipes (French drain) is 24 inches.

The 18-inch minimum pipe size does not apply to connections from external, private stormwater management facilities. The pipe size for these connections is the size required to convey the Chapter 14-86, F.A.C. or other authoritative permitted discharge limitations.

➢ *Using SSA to Specify Invert Elevations*

SSA Help File:

- Invert Elevations or Offsets: The software allows you to work in either elevation or depth mode. Working in elevation mode causes all input data to be entered as elevations (e.g., pipe inlet invert elevation). Working in depth mode causes some input data to be entered as a depth offset from the element invert (e.g., pipe inlet invert offset). Elevation is the default mode. Note that this is controlled by the entry ELEVATION TYPE in the Project Options dialog box, General tab.
- Inflow and Outflow Pipe Invert Elevations: Backwater surcharging can occur where smaller diameter pipes connect to larger diameter pipes and when the pipes have the same invert elevation. This typically happens along a main line sewer as the pipe size increases downstream and at connections of tributary and main line sewers. To reduce the potential for surcharging and backwatering, the following two options are generally used:
    - Crown (top of pipe) elevation of the smaller upstream pipe is matched to the crown elevation of the larger downstream pipe
    - Crown elevation of the smaller upstream pipe is above the crown elevation of the larger downstream pipe by the amount of loss in the access hole (this practice is often referred to as hanging the pipe on the hydraulic gradeline)

REFERENCE MATERIAL

- See document(s): GUID-53DEA840-95F1-4727-ACBE-6DACC3B14BBA-htm.html - Moving Pipe Network Data Between AutoCAD Civil 3D and Storm Sewers at https://knowledge.autodesk.com/
- Refer to the online Autodesk Help for more details on data migration.

REQUIREMENTS FOR MIGRATING CIVIL 3D DATA TO SSA

Subbasins (aka Catchments in Civil 3D), structures, and pipes are collectively referred to as Elements in SSA. In this guide we will use terminology used in SSA. As you can see this terminology is sometimes different than that used in Civil 3D.

Catchments, structures, and pipes in Civil 3D must be configured correctly in order for the successful transfer of data to SSA. This section covers what conditions must be met for each Element type in order to transfer data.

Note Failing to provide any one of these requirements will cause the structure to not export to SSA using the Edit in SSA command. Since there are numerous ways to fail using the Edit in SSA command and no error trapping to help you find and correct deficiencies we have provided a different workflow that can more predictably produce desired results.

Hydraflow Storm Sewers file format has a STM file extension. We will use this format to transfer data to and from SSA. Only the pipe network data (i.e., manhole structures, storm drain inlets, and pipes), subbasin data that contributes runoff to junctions and storm drain inlets, and Rational Method Intensity-Duration-Frequency (IDF) data is transferred using the STM file format. Time series rainfall data, detention ponds, and other specialty hydrology data is not transferred.The SSA Help file provides greater detail on using the Hydraflow Storm Sewers file format to transfer data between Civil 3D and SSA.

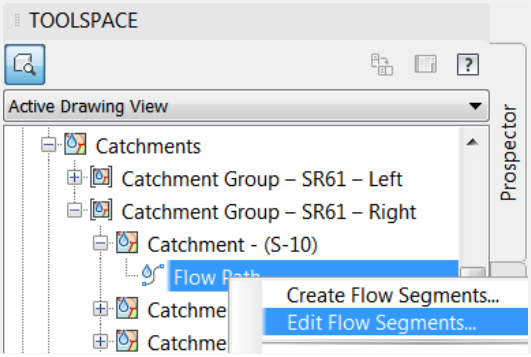
When creating the STM file you will need the catchments with the TOC computed from the Flow Path. An alternative is to create parcels from polylines and export to LandXML. The downside to Parcels is that they do not have any TOC info. Parcels have the benefit of automating labels and creation tables for catchment areas which are not a strength of catchments. Calculating TOC and assigning an inlet structure are the benefits to creating catchment objects and this method will be used in the exercises in this document. Use the Hydraflow STM format for output of Pipe Network structures, pipes, catchments with calculated TOC for use in SSA.

- **Catchments**
  - Flow Path Slope must have a non-zero, non-negative value.
  - Time of Concentration must be greater than zero.

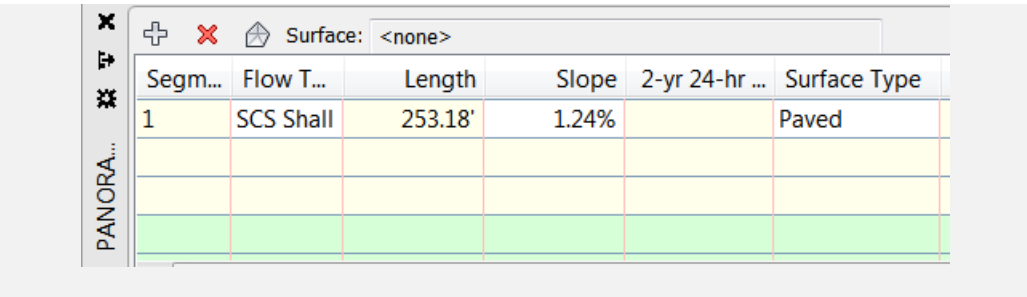
Note If Flow Path Slope is zero, the Time of Concentration is set to zero.

If you have selected the Time of Concentration Method for the Catchment Properties as TR-55 and did not provide the corresponding slopes to the Flow Segments (i.e. slopes for the Sheet Flow, Shallow Concentrated Flow and Channel flow segments) as the default value assigned by Civil 3D is zero slope.

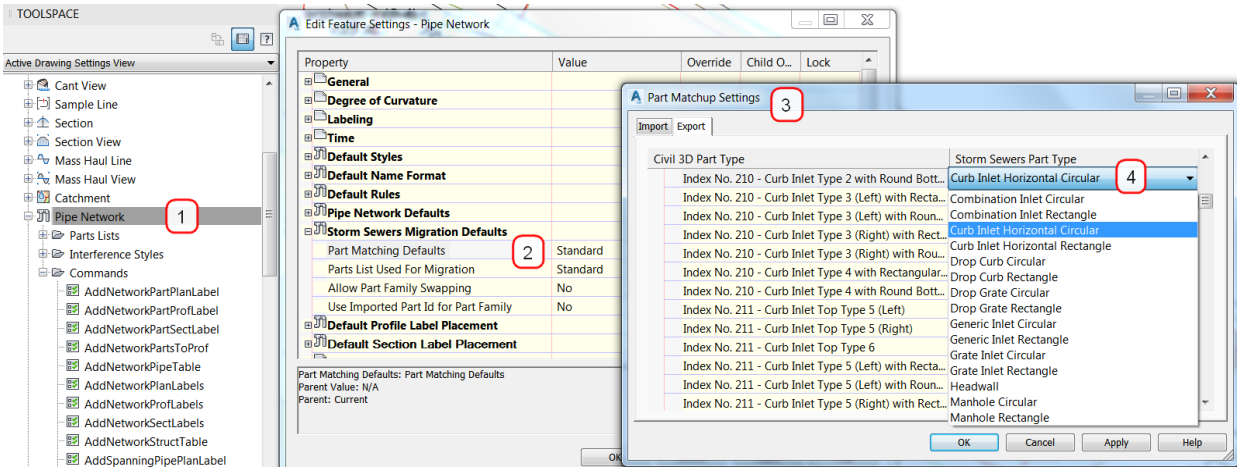
The Flow Path has no Time of Concentration as the slope of the flow line is zero. A Time of Concentration of zero does not export since there is nothing to evaluate.



Note Hence, you may want to expand the Catchments from the Prospector tab. Right click the Flow Path and select Edit Flow Segments to modify the slopes to a non-zero value. Make sure the Slope is not zero or negative. Set the Surface Type to Paved for the examples in this Manual.



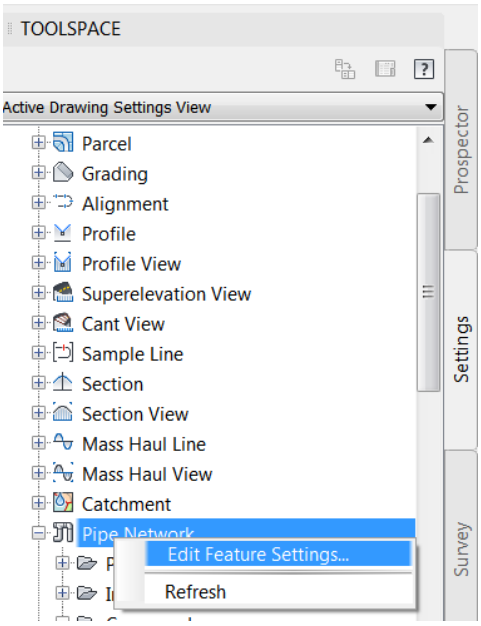
- Catchment must have a Structure assigned and set to Inlets in Storm Sewers Migration Defaults.



- **Structures**
  - Structure must have an outgoing pipe connected.
- **Pipes**
  - Pipes without an upstream structure will be assigned a null structure in SSA. The term Line is used in SSA to mean a pipe and an upstream structure.

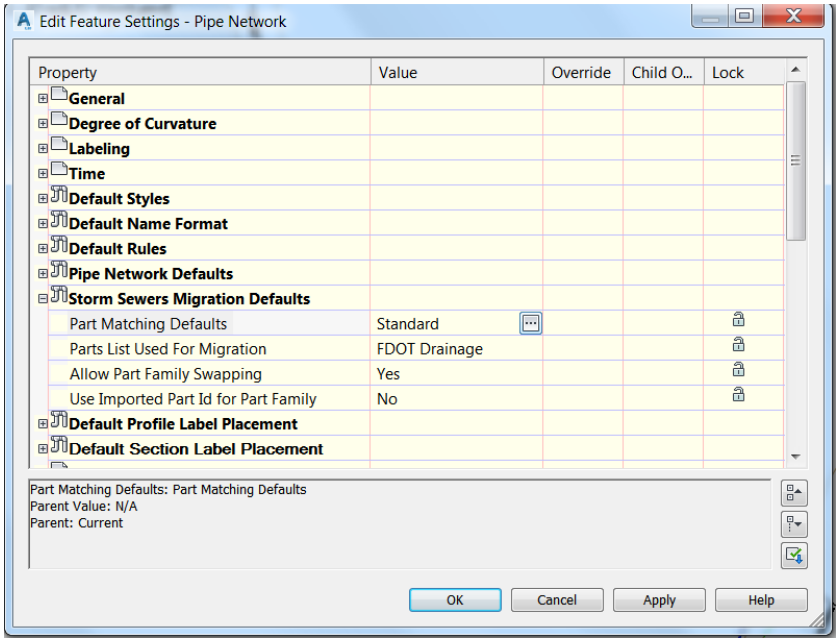
Exercise 2.1 Set Civil 3D Storm Sewers Migration Defaults

Assign structures used in the design as either an inlet or manhole. When imported to SSA you can save some work in SSA by having the correct structure type pre-defined in the Part Matchup Settings dialog.

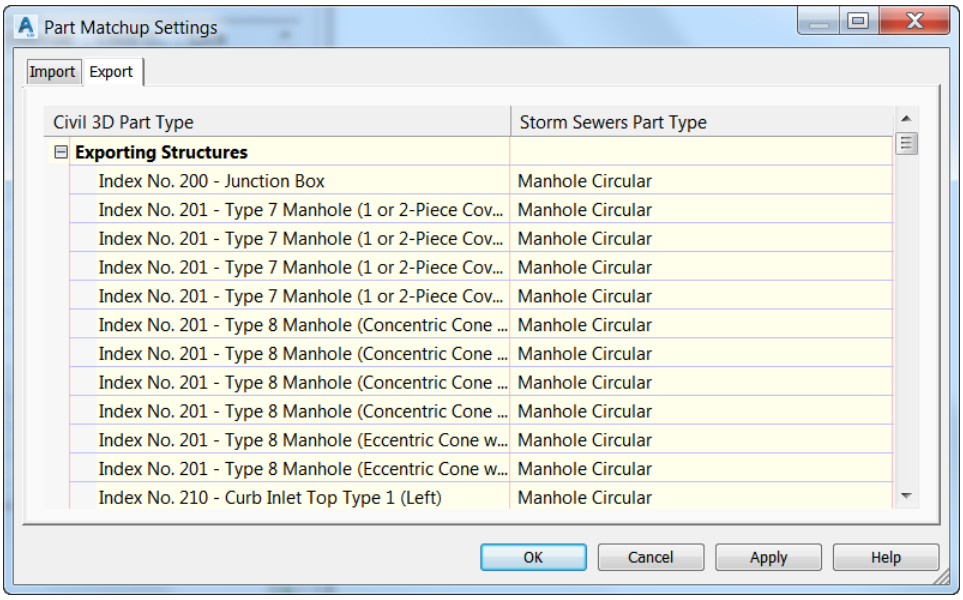


1. Start Civil 3D by double clicking the 

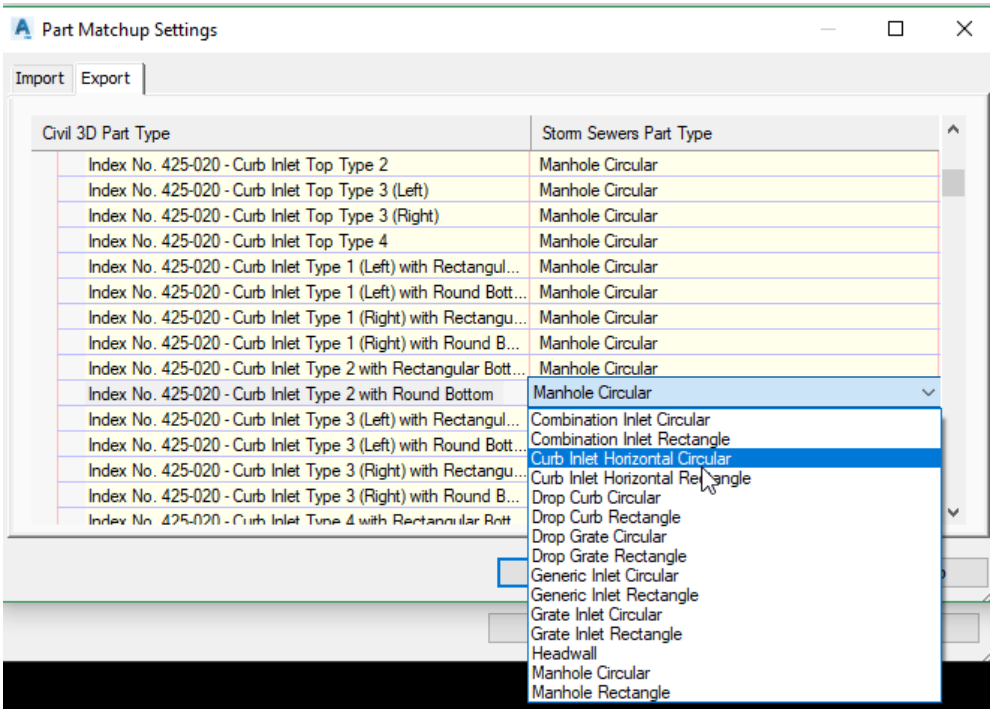
shortcut from your desktop.
2. Open the *DRPRRD05.dwg* file found in the C:\Civil 3D 20XX Projects\22049555201\Drainage folder of the project.
3. In the Toolspace on the *Settings* (tab) scroll down to *Pipe Network* right click and select **Edit Feature Settings**.
4. Scroll down and expand *Storm Sewers Migration Defaults* and find **Part Matching Defaults**.



5. In the Edit Feature Settings - Pipe Network dialog expand *Storm Sewers Migration Defaults*. Edit the Part Matching Defaults by picking the **ellipsis** button to open the Part Matchup Settings dialog.



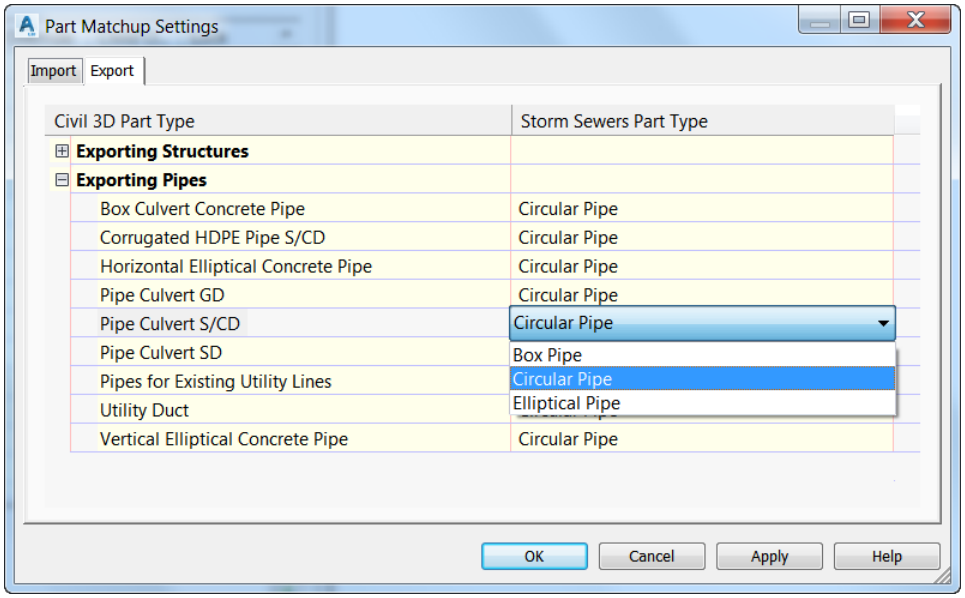
6. Select the *Export* tab and expand the **Exporting Structures** collection.



7. Scroll down to the *Civil 3D Part Type Index No. 425-020 - Curb Inlet Type 2 with Round Bottom*. Click the dropdown list in the *Storm Sewers Part Type* column and select **Curb Inlet Horizontal Circular**. You only need to assign the correct part type to the structures and pipes actually used.

Note You have a limited part list for the return trip from SSA to Civil 3D so putting effort into assigning multiple variations of the same part in your design in Civil 3D at this stage of the workflow is not recommended.

An outfall structure will be assigned in SSA therefore it has been omitted in the workflow up to this point.



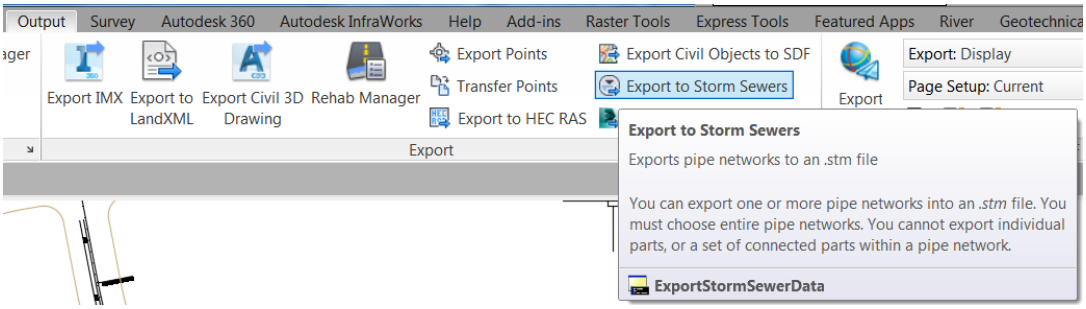
8. Scroll down to the *Exporting Pipes* collection to verify the *Civil 3D Part Type Pipe Culvert S/CD* is set to **Storm Sewers Part Type Circular Pipe**.

Note The Storm Sewers Part Type has a limited selection for Pipes therefore it is not recommended to spend time in Civil 3D assigning variations to pipe parts. Those variations will not survive the round trip from Civil 3D to SSA and back to Civil 3D. The only reason to assign variations is to keep a record of your original choices in a backup drawing file so you can manually compare it to the imported pipes from SSA.

9. Click the **Apply** button. Click the **OK** button to close the Part Matchup Settings dialog.

10. **Save** the *DRPRRD05.dwg*

Exercise 2.2 Export Pipe Network to a Storm Sewers File (.STM)



1. Continue using the *DRPRRD05.dwg* file from Exercise 2.1. From the *Output* tab on the ribbon in the *Export* panel, select the **Export to Storm Sewers** command to create an *STM* file.
2. In the C:\Civil 3D 20XX\Projects\22049555201\Drainage folder create an folder named **ssa** .
3. Create the file ***Proposed Drainage - SR61.stm*** file in the ssa folder: C:\Civil 3D 20XX\Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61.stm
4. **Save** and **Close** the *DRPRRD05.dwg*, and then **Close** Civil 3D.

3 SETUP IN SSA

DESCRIPTION

When you create a new project in SSA the first step is to configure the Options to fit the project's requirements. To do this you will make changes in the Project Options dialog, Analysis Options dialog.

OBJECTIVES

In this chapter, you will learn about:

- SCS TR-55
- Rain Gauges
- Rainfall Designer
- IDF Curve Zones
- Analysis Options

CHAPTER SETUP

Run the Chapter 3 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX** Projects folder.

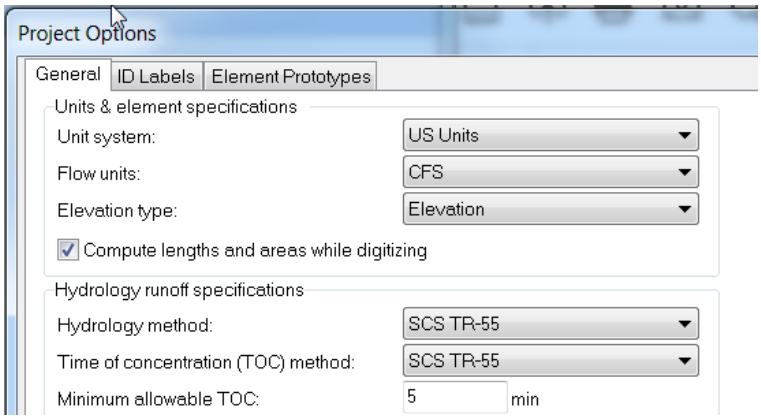
Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual

ABOUT HYDROLOGY METHODS

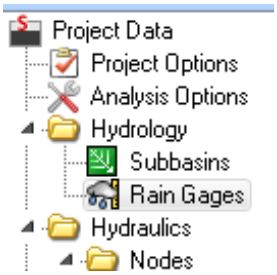
In SSA the input specifications for each Hydrology method are different. If you choose the SCS TR-55 method you are required to specify surface characteristics that affect time of concentration for each subbasin. Rain gauge data is required to specify the quantity of water produced by the storm event used in your analysis. If you choose Rational method you will use the FDOT provided IDF Curves to specify the rainfall intensity of a storm type for the location of your site.

This section of the training guide provides a brief overview on how SSA is configured to use subbasins and rainfall data sources for the SCS TR-55 method and the Rational method. The example we use to perform an Analysis of the network later in this training guide will use the Rational method.

SCS TR-55 METHOD



When the SCS TR-55 Method is selected in the Project Options the Data Tree displays Subbasins and Rain Gages nodes. Subbasins are where you can setup weighted curve numbers.



RAIN GAGES

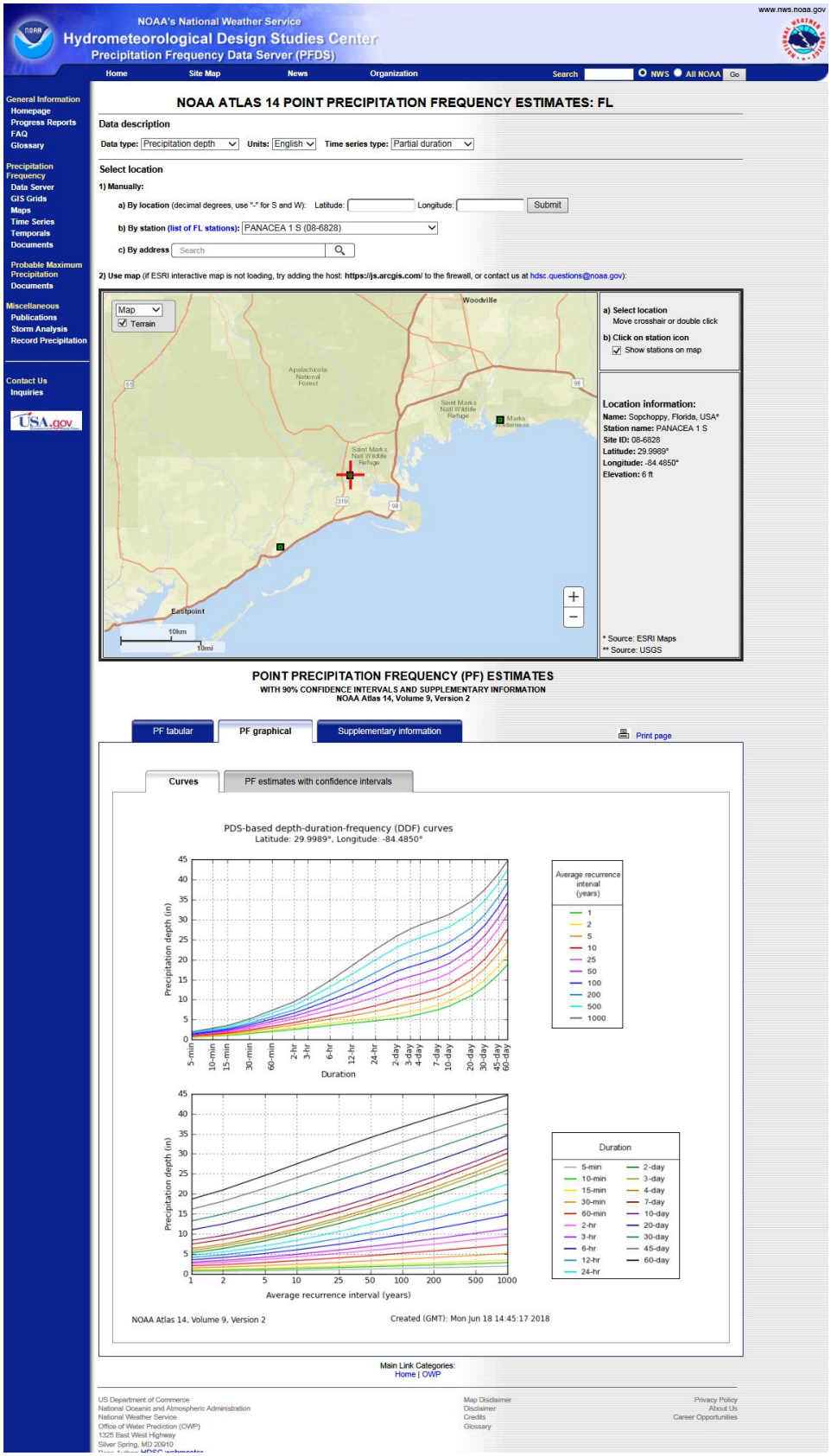
Rain gages supply rainfall data for one or more subbasins in the study area. The rainfall data can either be a user-defined time series or described in an external file. Several different popular rainfall file formats are supported, as well as user-defined formats.

The principal input data of a rain gage include:

- Rainfall data type (e.g., intensity, volume, or cumulative volume)
- Recording time interval (e.g., hourly, 15-minute, etc.)
- Source of rainfall data (input time series or external file)
- Name of rainfall data source

FDOT Drainage Manual makes reference to NOAA as a source for Rainfall Data:

https://hdsc.nws.noaa.gov/hdsc/pfds/pfds_map_cont.html?bkmrk=fl

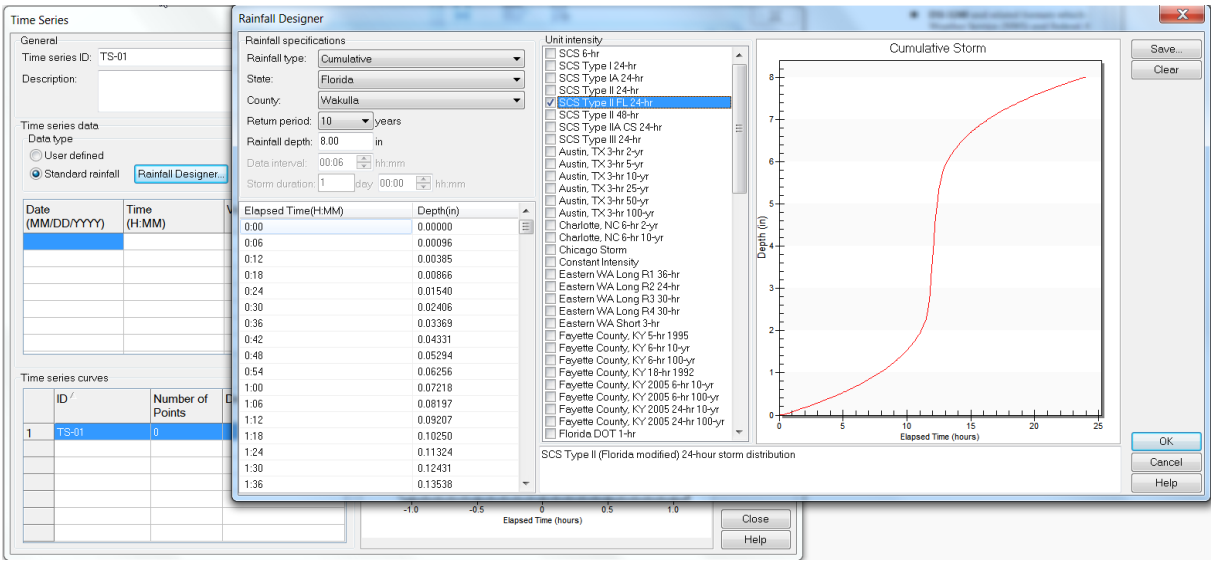


RAINFALL DESIGNER

SSA includes a Rainfall Designer which allows you to select any location within the USA and it will provide the design rainfall for the specified storm frequency. Alternatively, a user-defined rainfall can be specified. Then the appropriate storm distribution is selected and the design storm is then created. Multiple design storms can be created.

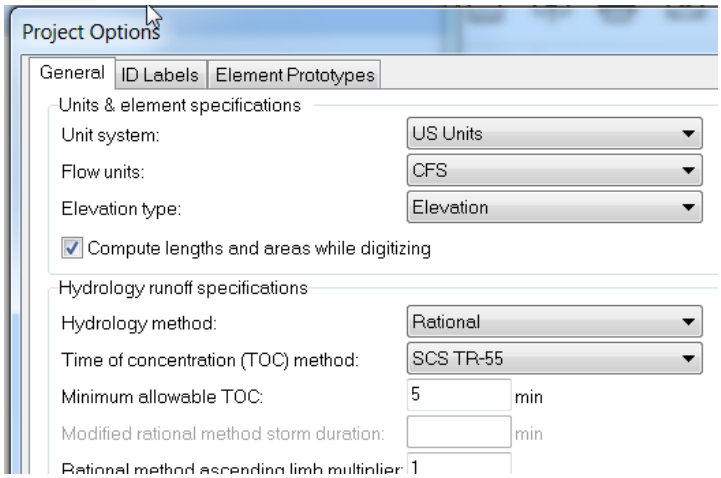
The Rainfall Designer provides the following capabilities:

- Automatically determines design rainfall (based upon study location) for 1, 2, 5, 10, 25, 50, and 100 year return frequencies
- Defines any storm duration
- Multiple storm events can be created
- Numerous storm distributions, including SCS, Huff, Eastern Washington, Florida, Chicago Storm, Hurricane Hazel, etc.

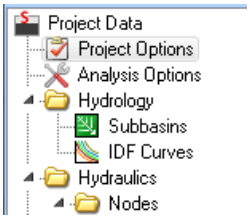


Example screenshot of Rainfall Designer. To access, click the Rainfall Designer button from the Time Series dialog box.

RATIONAL METHOD



When the Rational Method is selected in the Project Options the Data Tree displays Subbasins and IDF Curves nodes as shown below.



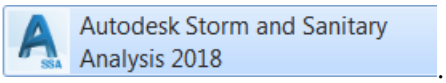
RAINFALL IDF CURVES ZONES

Note State Kit location of IDF Curves C:\FDOT20XX.C3D\Data\SSA\IDF Curve\.

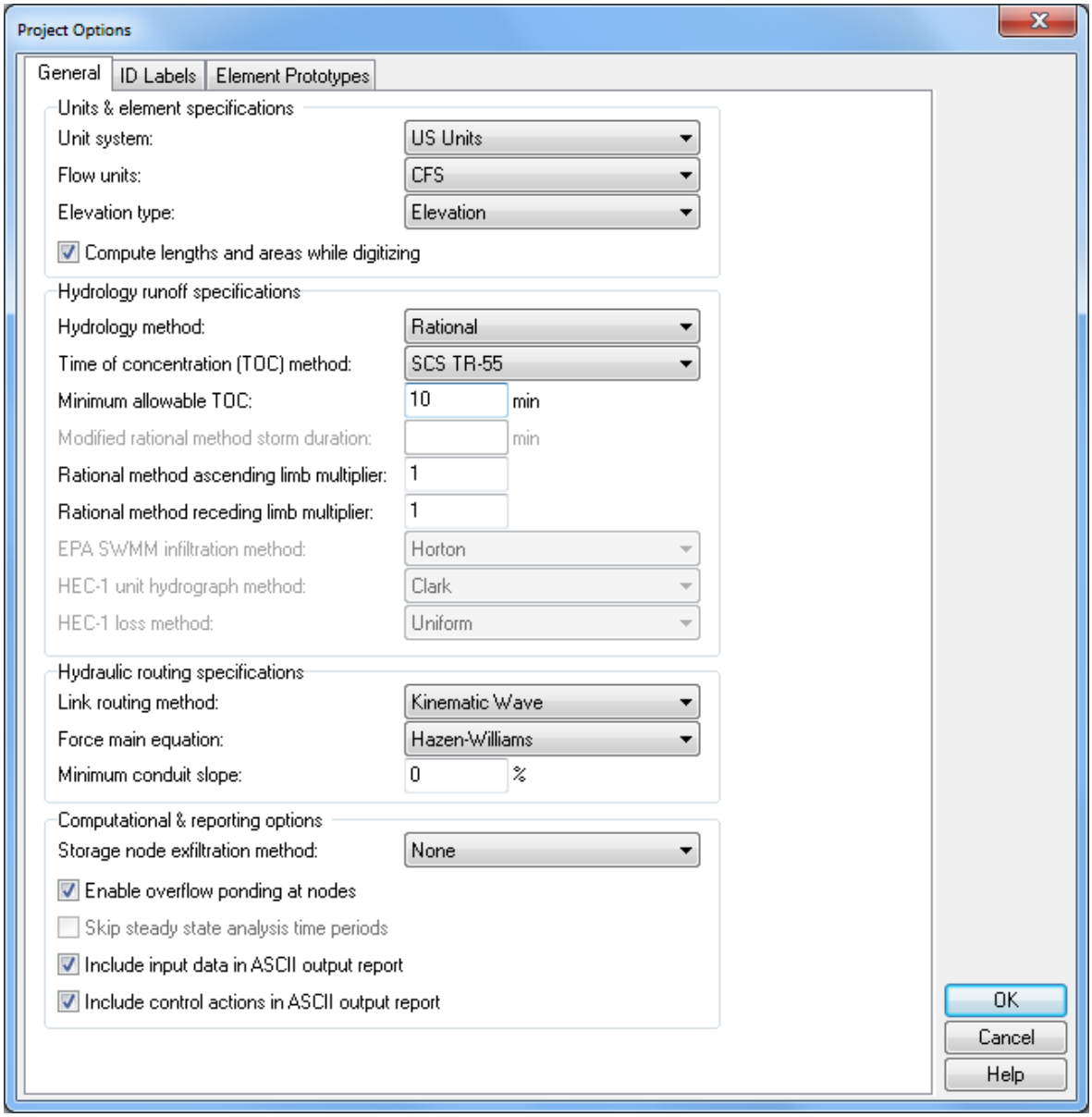
FDOT Drainage Manual Figure B-1 shown below is a map of Florida broken into zones.

http://www.fdot.gov/roadway/Drainage/files/IDFCurves.pdf

Exercise 3.1 Project Options

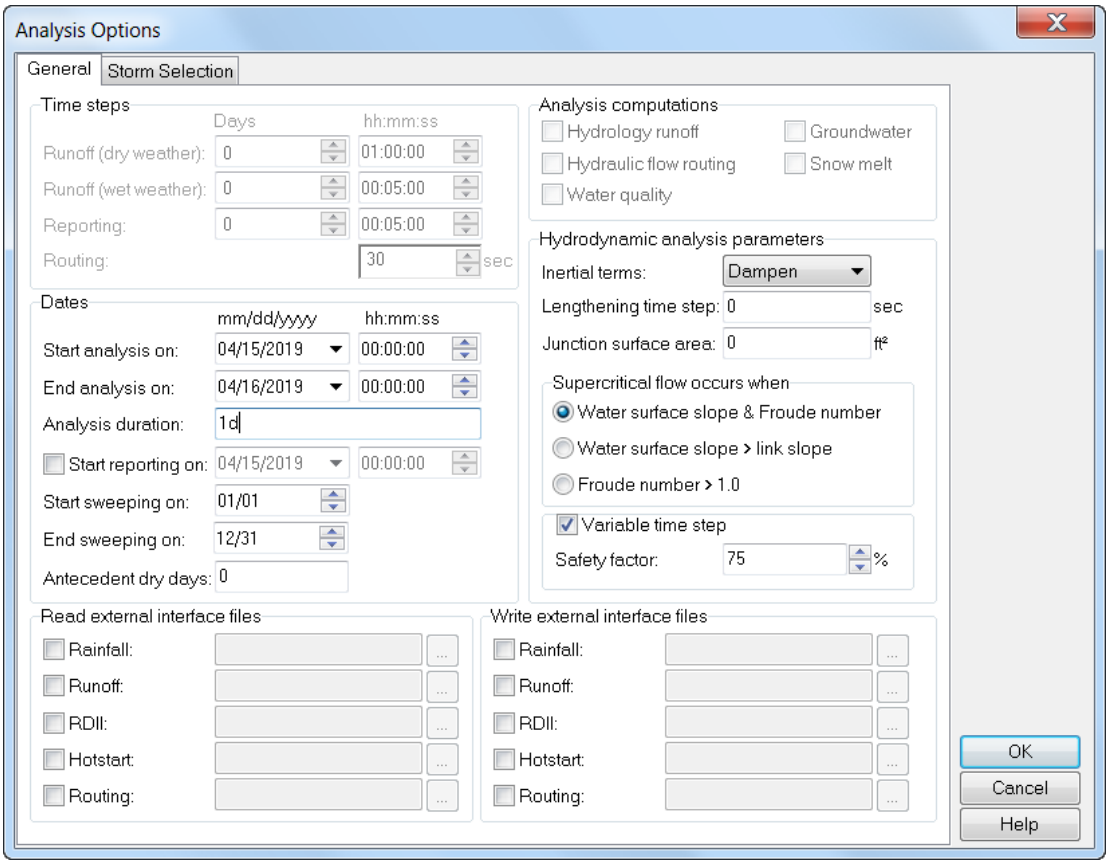
1. Start SSA double clicking the desktop shortcut 

.
2. Double click the *Project Options node* in the Data Tree to open the Project Options dialog.



3. Make the following changes in the *General* tab of the Project Options dialog:
   a. *Hydrology runoff* specifications
      - ✓ Set the *Hydrology method* to **Rational.**
      - ✓ Set the *Time of concentration (TOC) method* to **SCS TR-55.**
      - ✓ Set the *Minimum allowable TOC* to **10 min.**
   b. *Hydraulic routing* specifications
      - ✓ Set the *Link routing method* to **Kinematic Wave.**
4. Leave all other settings in the *General* tab unchanged. Click **OK**.

Exercise 3.2 Analysis Options



1. Double click the **Analysis Options node** in the data tree to open the Analysis Options dialog.
2. On the *General* tab set the *duration* for a **24hr** period by changing the following options:
   a. Under *Dates* change the *Start analysis on*: to **today's date**.
   b. Under *Dates* change the *End anaysis on*: to **tomorrow's date**.
   c. The *Analysis duration* should now display **1d** indicating one day (24hrs).

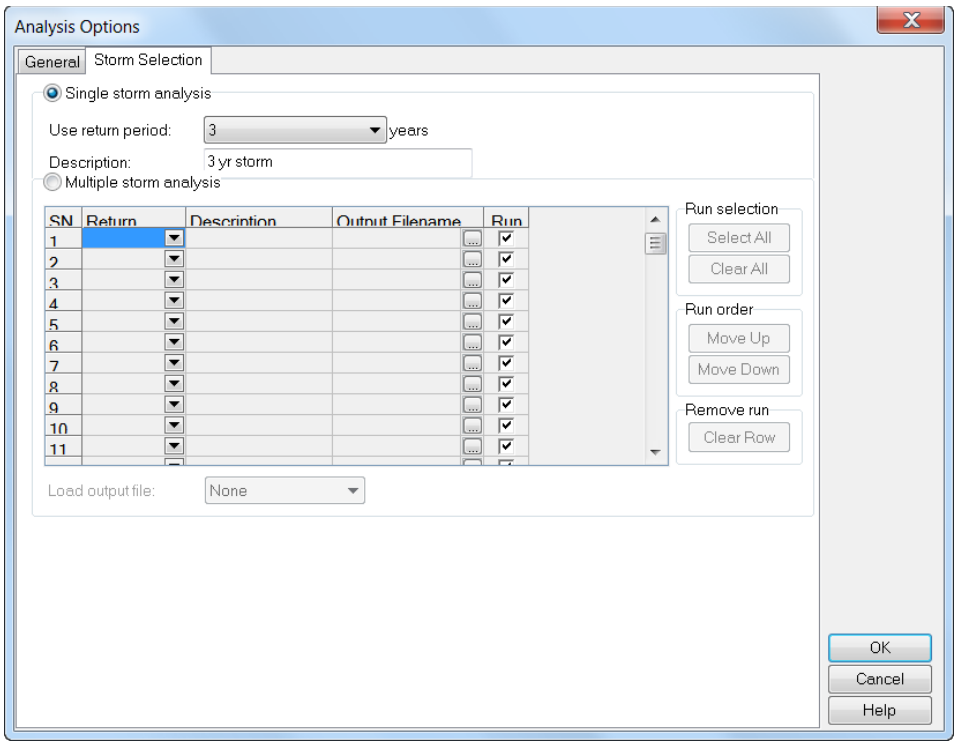
Note In the FDOT Drainage Manual Table 3.1 a 3-year frequency is specified for General design.

Table 3.1: Design Storm Frequencies of Storm Drain Systems

TYPE STORM DRAIN	FREQUENCY
General design	3-year
• General design work that involves replacement of a roadside ditch with a pipe system by extending side drain pipes • General design on work to Interstate Facilities	10-year
• Interstate Facilities for which roadway runoff would have no outlet other than a storm drain system, such as in a sag inlet or cut section • Outlets of systems requiring pumping stations	50-year

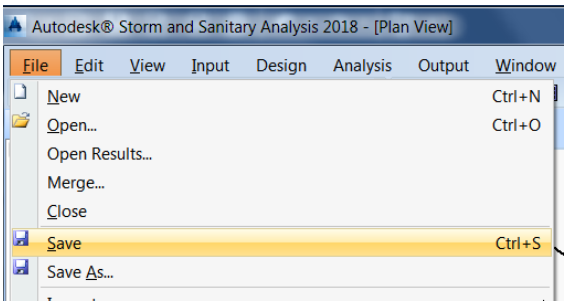
3. In the *Storm Selection* tab of the Analysis Options dialog, set the radio button to **Single storm analysis** and the *Use return period*: to **3 years** using the dropdown list.

Note In this example you are using the Rational Method enabling the Single storm analysis option in this dialog to specify the storm intensity for analysis.

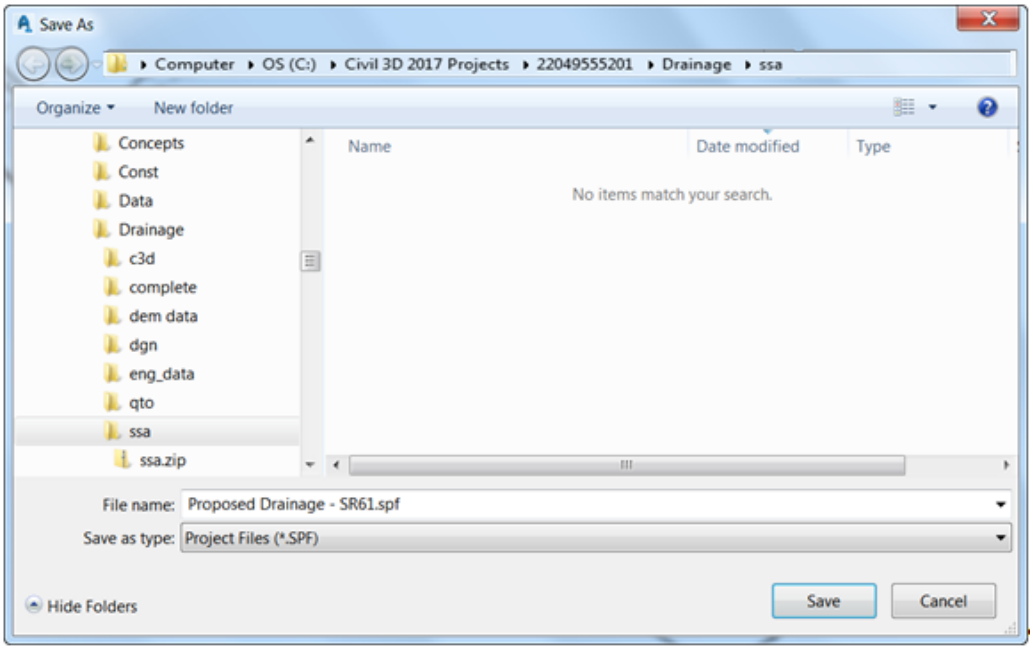


4. Make no other changes and click **OK** to dismiss the Analysis Options dialog.

5. **Save** the Project



6. From the **File** drop down menu and select the **Save As** command.



7. From the Save As dialog box, browse to the **ssa** folder under the *Drainage* folder and type the f*ile name* **Proposed Drainage - SR61.spf:**

   C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61.spf

8. **Close** SSA.

[THIS PAGE INTENTIONALLY LEFT BLANK]

4 IMPORT HYDRAFLOW STORM SEWER STM FILE

DESCRIPTION

This chapter introduces the process of importing Hydraflow Storm Sewers STM files

OBJECTIVES

In this chapter, you will learn about:

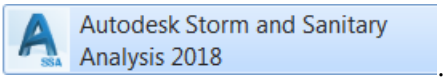
- Importing STM files
- Reviewing Log files
- Saving the Project

CHAPTER SETUP

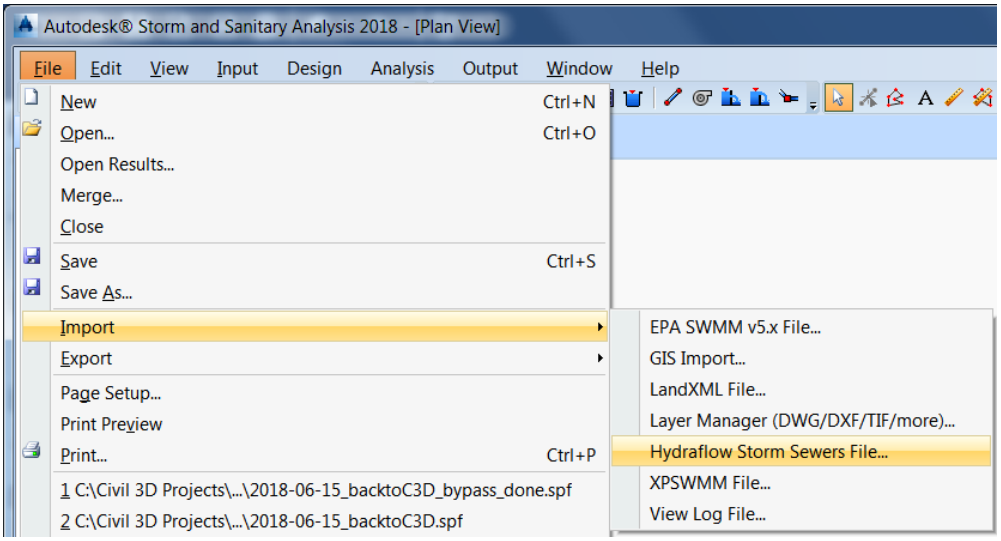
Run the Chapter 4 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX** Projects folder.

Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

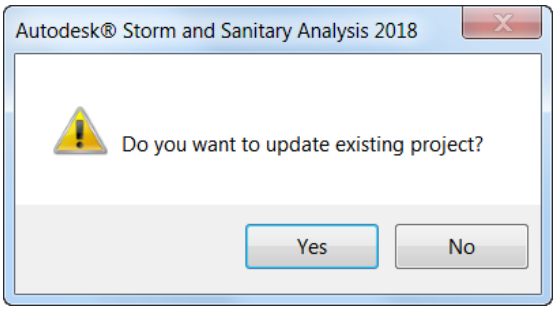
Exercise 4.1 Import .STM File

1. Start SSA by double clicking the desktop shortcut 

.
2. Start SSA and open the project file *C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61.spf.*
3. Click the **File** drop-down menu, hover over *Import*, select **Hydraflow Storm Sewers File**



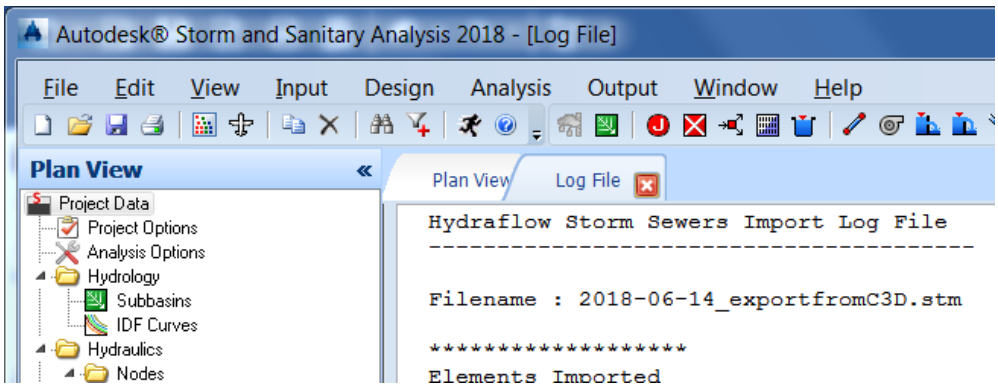
4. *Do you want to update existing project?* Click the **Yes** button if this box appears.



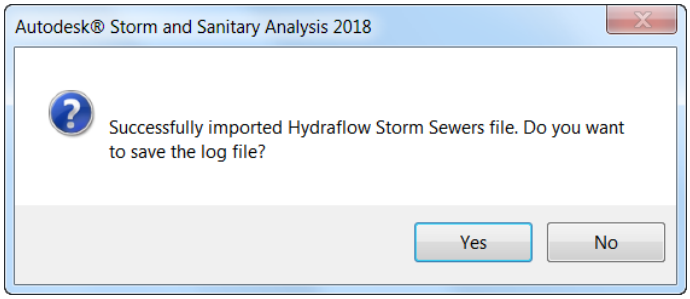
Note The current project has been preconfigured for Project Options and Analysis Options. Rather then having to redo those settings answer **Yes** to update the existing project.

5. Browse to the **ssa** folder under the C:\Civil 3D 20XX Projects\22049555201\Drainage folder and click on the *file name* **Proposed Drainage - SR61.stm**

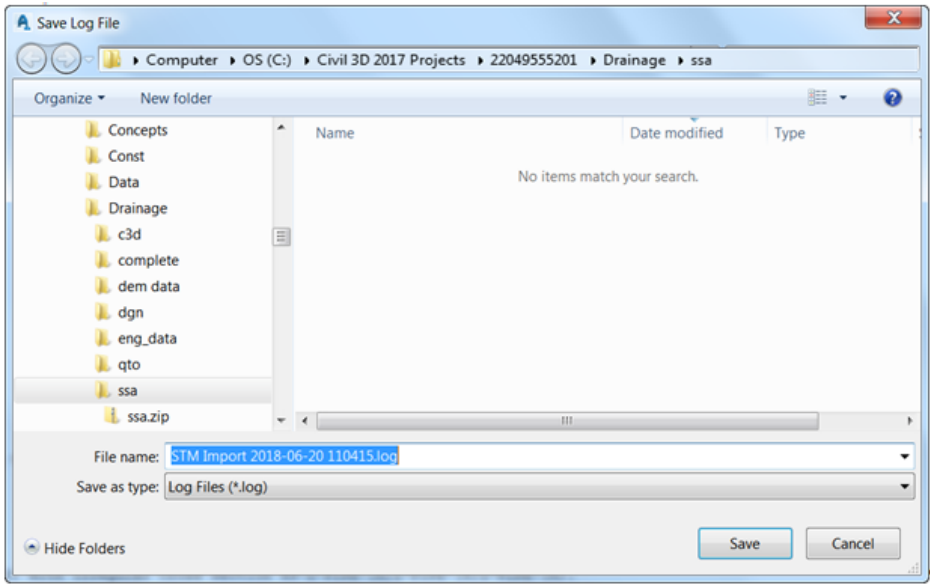
6. Review *Log File*



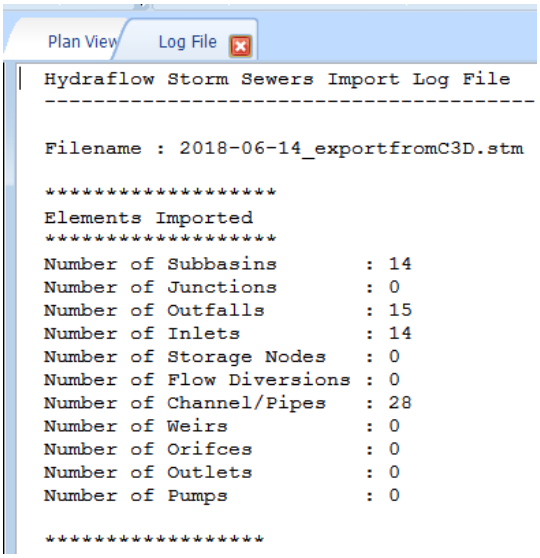
a. The *Log File* tab displays the results of the imported STM file.



b. Click the **Yes** button. There is no other place to access this information at a later time other than the saved log file so it is highly recommended that you save the log file.

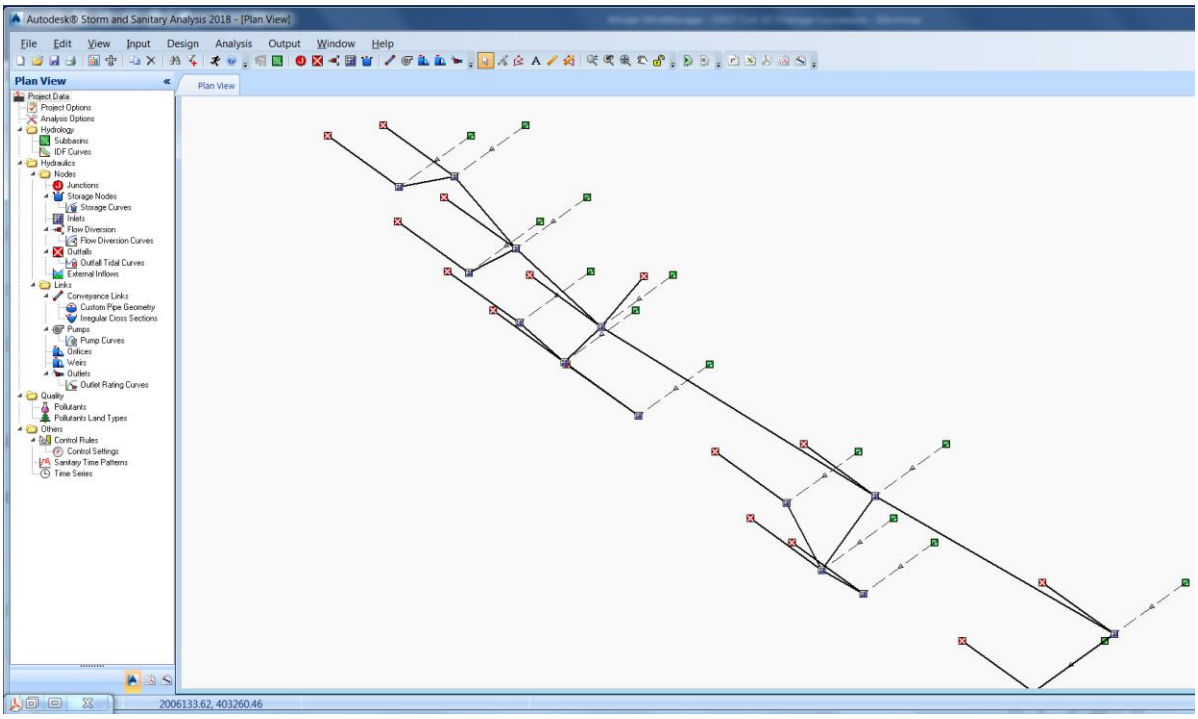


c. The *Save Log File* displays with the suggested file name formatted with today's date and time. Browse to the *ssa* folder under the *Drainage* folder of the project and click the **Save** button without changing the default name.

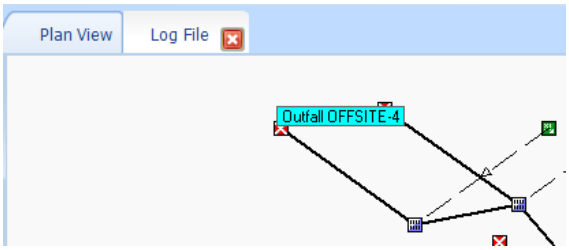


Note The *Log File* tab displays the log file from the STM File import process. It is important to review the *Elements Imported* section to make sure all the Subbasins were imported. The pipes are twice as many as you designed in Civil 3D since a bypass link is added for each inlet. Later in this training guide you will connect bypass links with the appropriate inlets replacing the bypass links and outfalls shown in the log file.

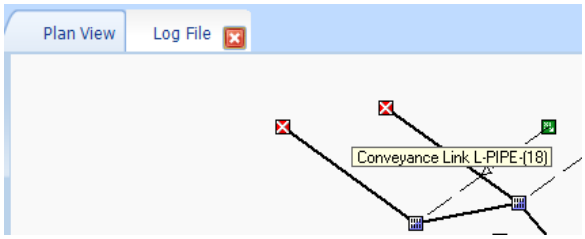
7. **Review** the *Plan View* tab.



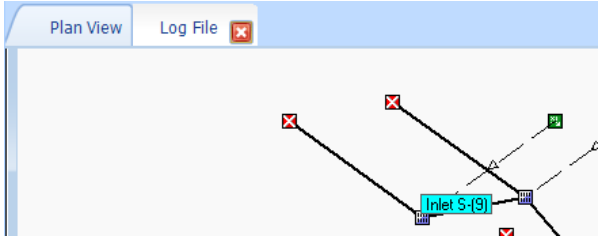
a. The *Plan View* tab displays the graphical representation of nodes and links from the imported STM file.



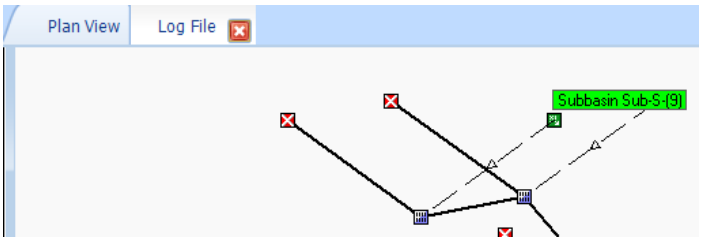
b. Hover over a *red node* to display the *Node ID* in its tool tip. The red node represents an *Outfall*.



c. Hover over the link connected to the *Outfall* to display the *Link ID* of the conveyance link. The *conveyance link* connected to the Outfall represents the bypass link connecting the Inlet to the Outfall which was automatically created by the Import STM process.

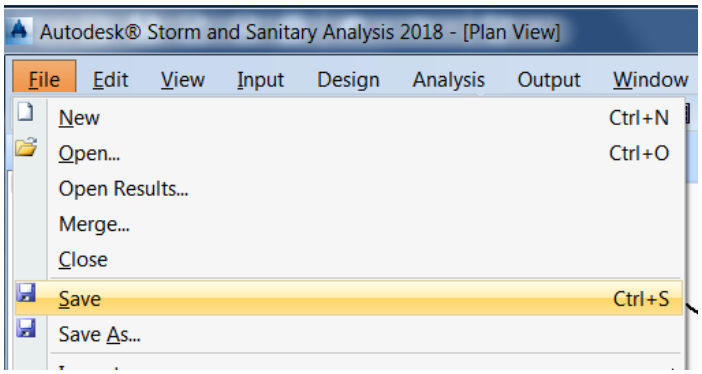


d. Hover over the *gray node* to display the *Node ID* of the Inlet.

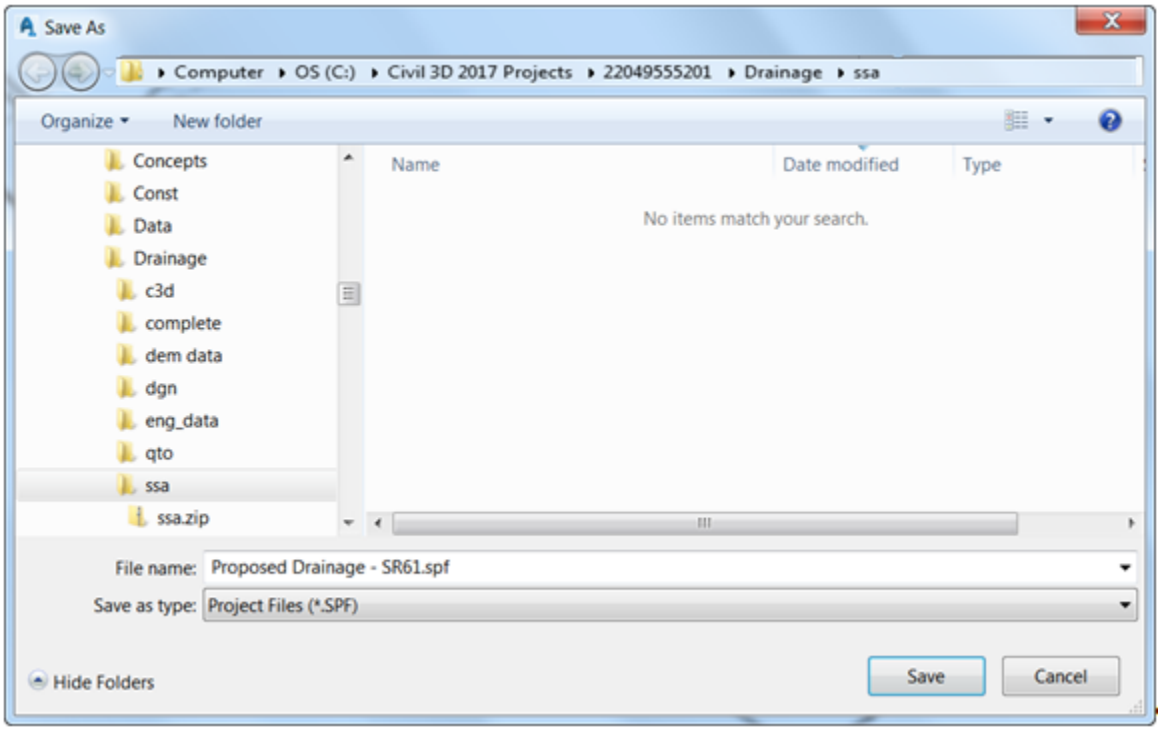


e. Hover over the *green node* to display the *Subbasin ID* connected to the Inlet.

8. **Save** the Project.



a. Click the **File** drop down menu and select the **Save As** command.



b. The Save As dialog box displays. Browse to the **ssa** folder under the *Drainage* folder and type the *file name* ***Proposed Drainage - SR61.spf.***

C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61.spf

9. **Close** SSA

[THIS PAGE INTENTIONALLY LEFT BLANK]

5 DISPLAY CAD FILE & ELEMENT LABELS

DESCRIPTION

This chapter introduces how to display a CAD file and element labels in SSA

As you work in SSA you will find that keeping organized and aware of how elements are connected is very important to working efficiently and avoiding mistakes. In this chapter you will display your Civil 3D drawing as a background for your network in SSA. You will also learn how to display ID labels for nodes and links. Then you will move the subbasin nodes inside the catchment polygon areas.

OBJECTIVES

In this chapter, you will learn about:

- Assigning a Background Drawing
- Displaying Node and Link ID Labels
- Move Subbasin Nodes next to Catchment Labels

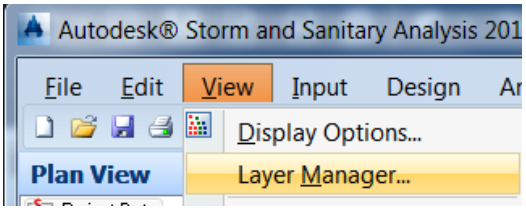
CHAPTER SETUP

Run the Chapter 5 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX** Projects folder.

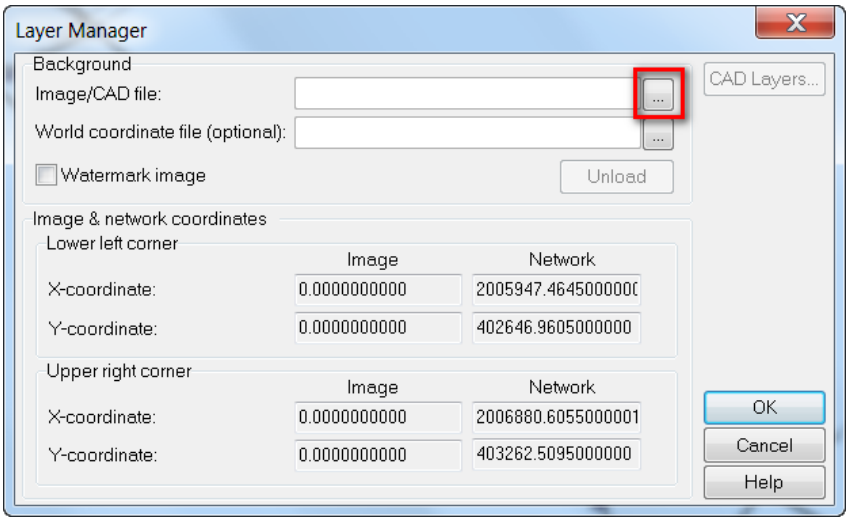
Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

Exercise 5.1 Assign Background Drawing

1. Start SSA and open the project file *C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61.spf.*
2. Click the **View** pull-down menu and choose **Layer Manager**.

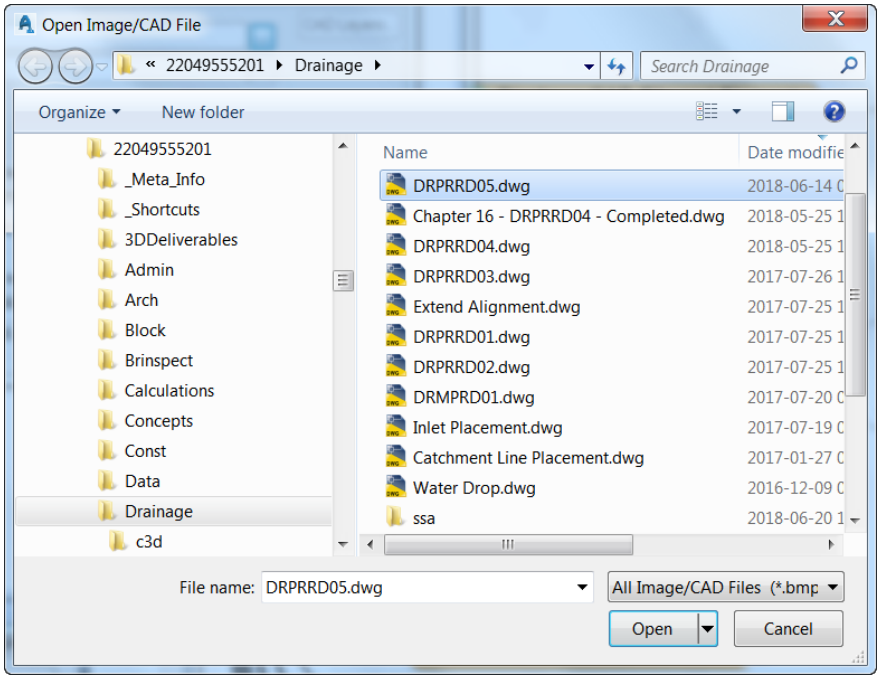


3. In the Layer Manager dialog you will see the *Background Image/CAD file*: field is empty. Click the ***ellipsis*** button (as indicated by the red box) to browse for a CAD file.

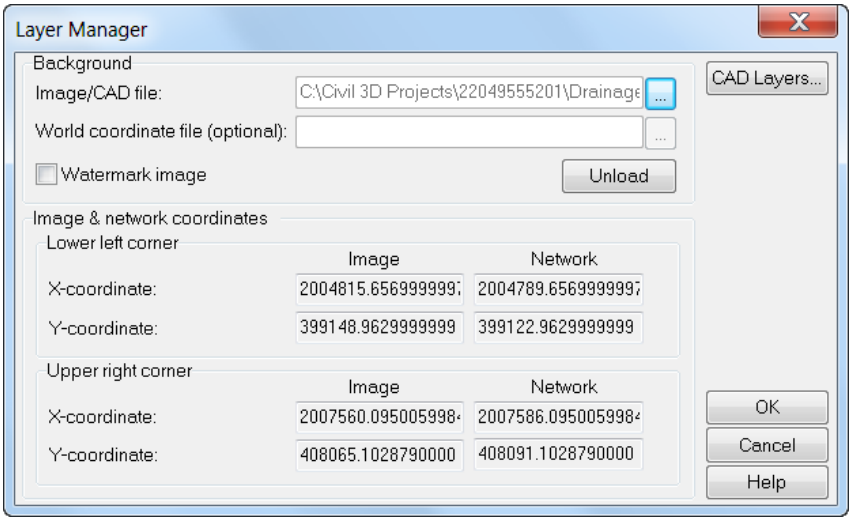


a. Browse to the **Drainage** folder: C:\Civil 3D 20XX Projects\22049555201\Drainage.

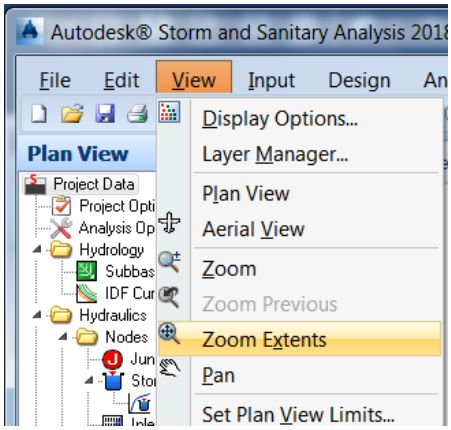
b. Select the *DRPRRD05.dwg* file and Click **OPEN**.



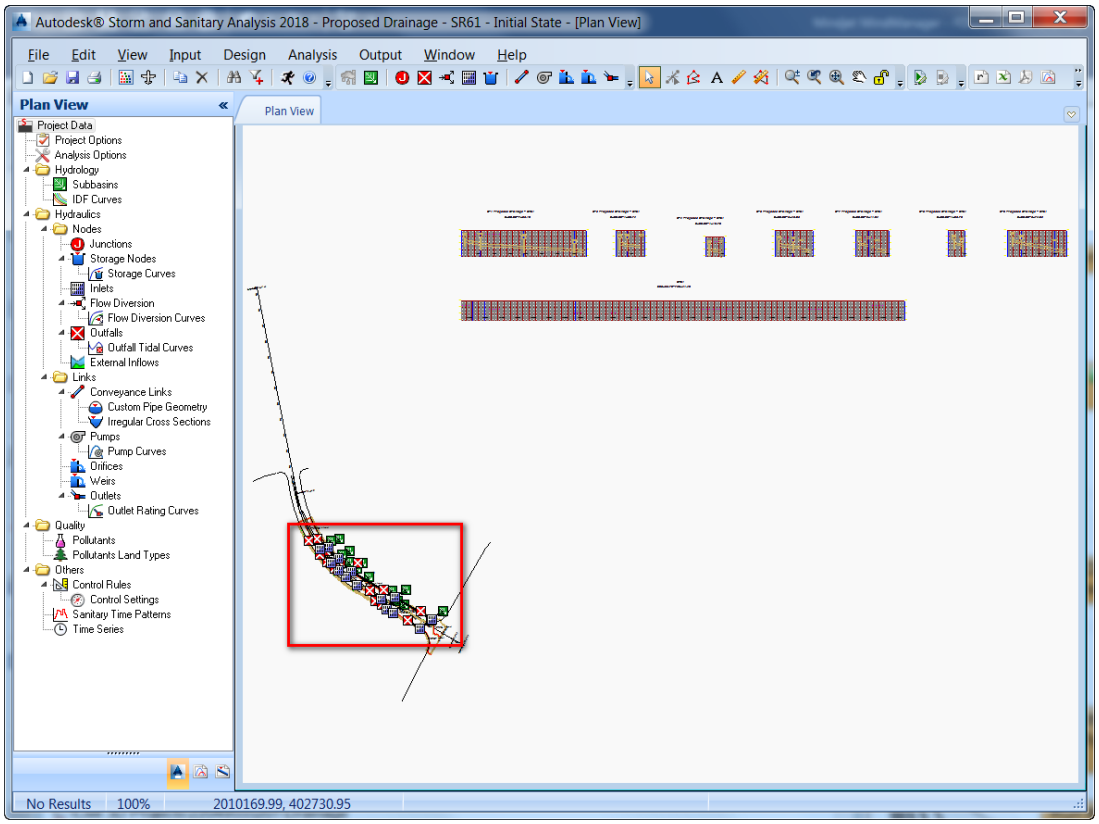
4. The *Background Image/CAD file* field displays the path and file name you just selected. Click **OK** to dismiss the Layer Manager dialog.



5. From the *View* pull-down menu and choose **Zoom Extents**.

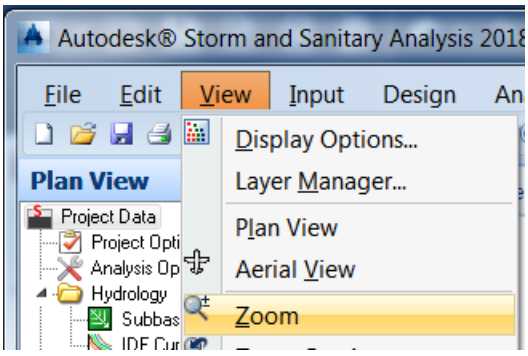


6. The *Plan View* tab displays the CAD file background and the design elements. **Zoom** into the area as indicated by the red box.



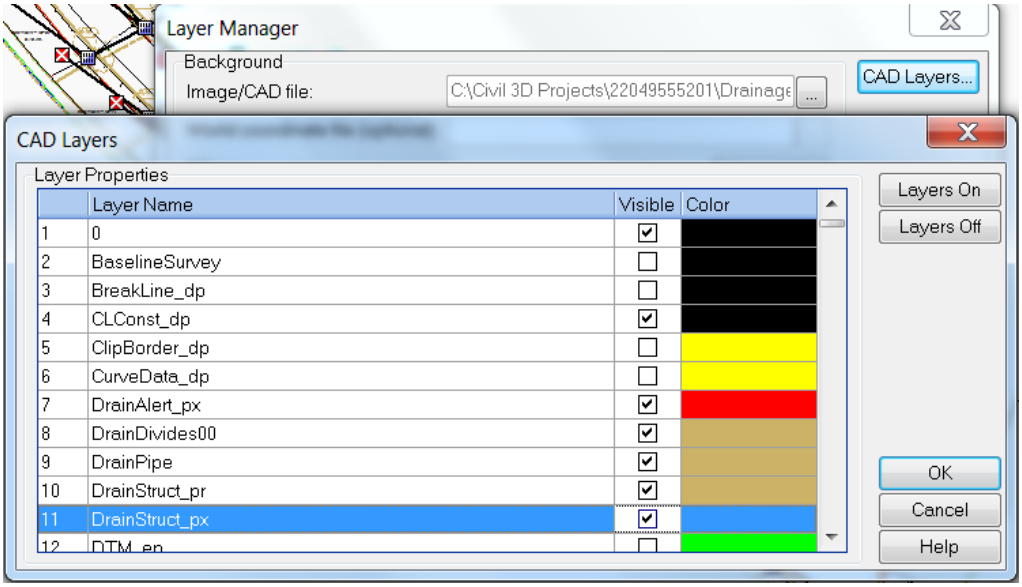
7. Click the **View** pull-down menu and select the **Zoom** command. There is no red box on your screen. The red box was placed on the screen capture as a point of reference. Click on the canvas in the approximate location of the upper left and lower right corners of the red box.

Note Being familiar with the zoom command in AutoCAD is helpful since the SSA Zoom command behaves similar to the Zoom Window command in AutoCAD.

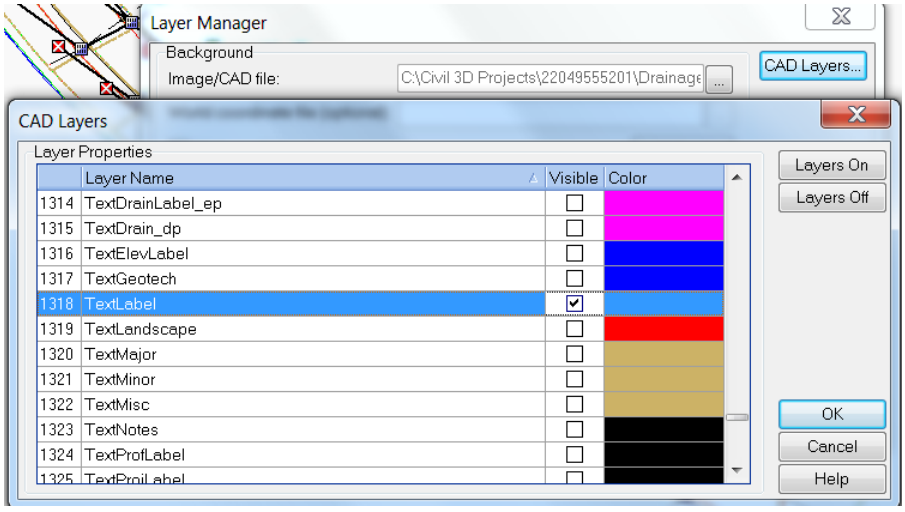


8. Your display should now look similar to this screen capture. Click the **ESC** key to terminate the zoom command.

9. Click the **View** pull-down menu and open the Layer Manager dialog box again. Click the **CAD Layers** button in the upper right corner of the dialog to open the CAD Layers dialog.



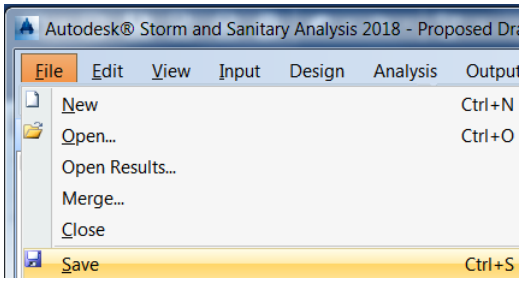
10. Click the **Layers Off** button. Check the boxes in the *Visible* column as indicated below.



11. Additionally, scroll down to find the **TextLabel** layer and check the box.You may need to click the *Layer Name* column heading to sort the list alphabetically. Click **OK** twice.

12. **Zoom in**. You should now see the nodes and links on top of the catchment areas from the background CAD file.

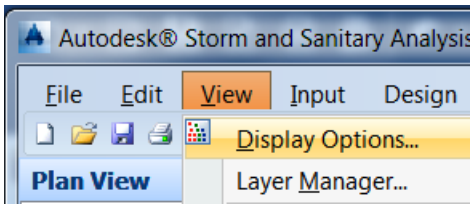
13. Select the **Save** command under the *File* pull-down menu to save your SSA project.



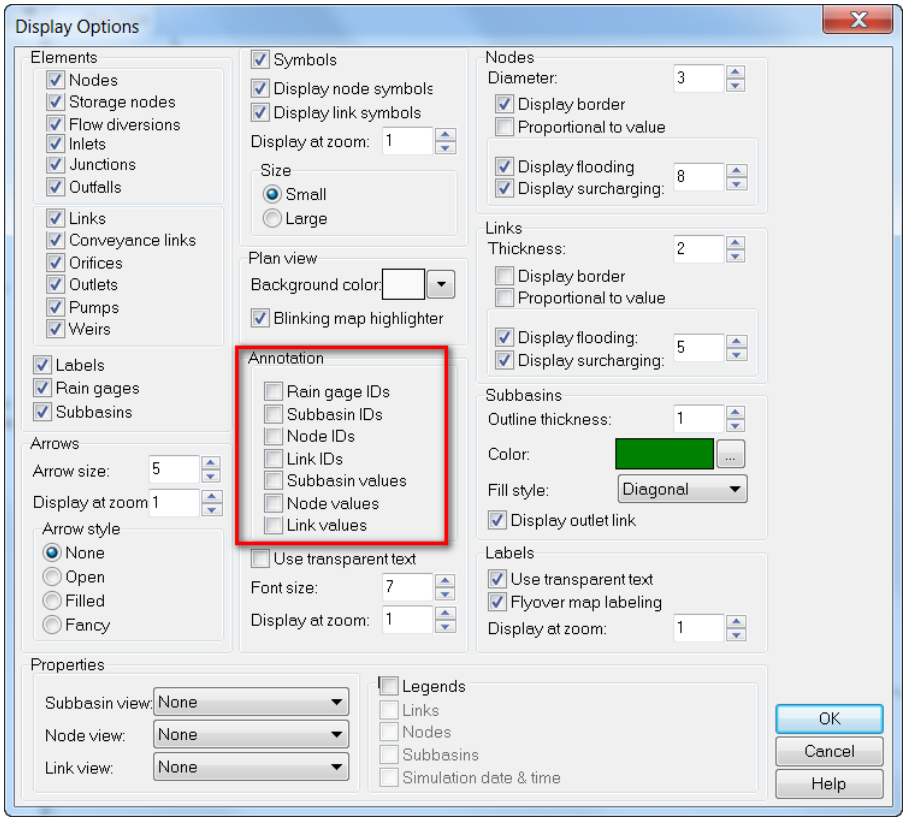
Exercise 5.2 Display Node and Link ID Labels

Nodes and Links have IDs that can be displayed as labels in the Plan View tab.

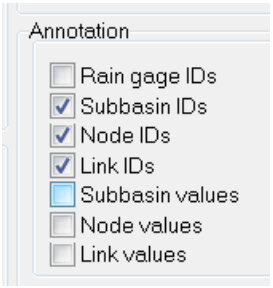
1. Click the *View* pull-down menu and select the **Display Options** command.



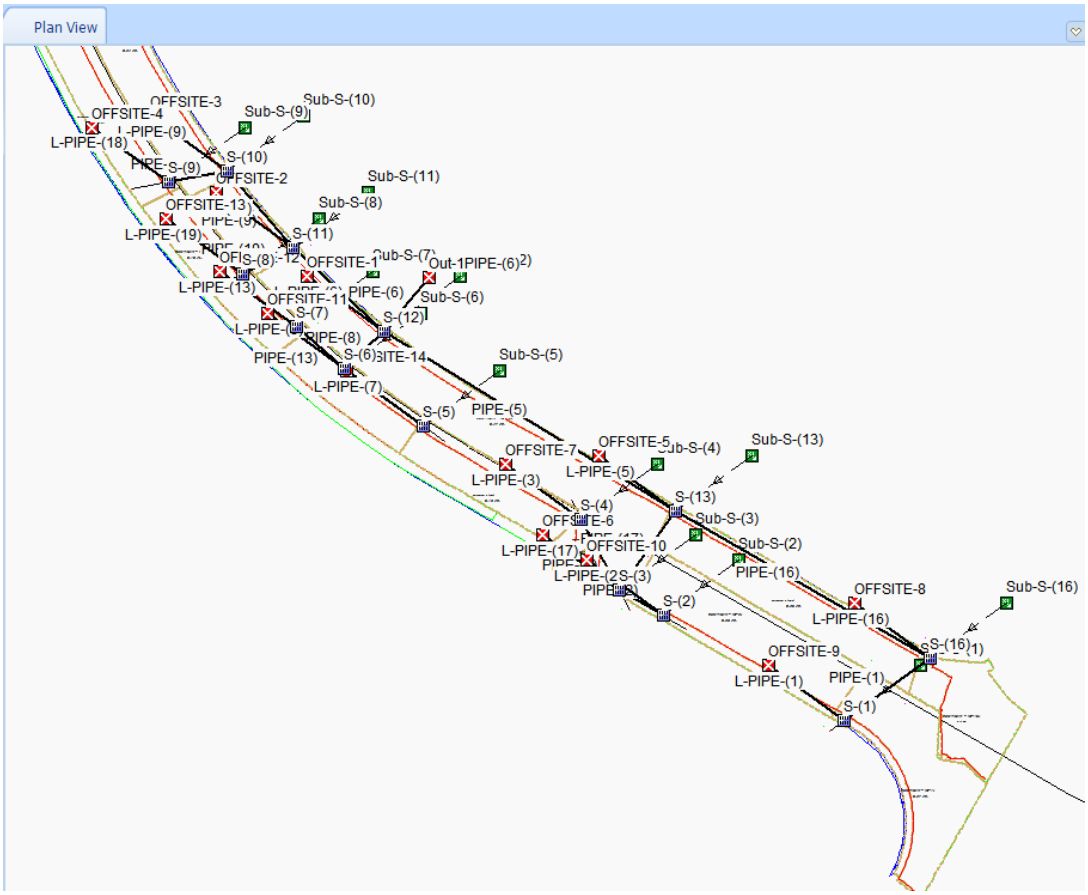
2. The Display Options dialog box displays. Find the *Annotation* section of this dialog.



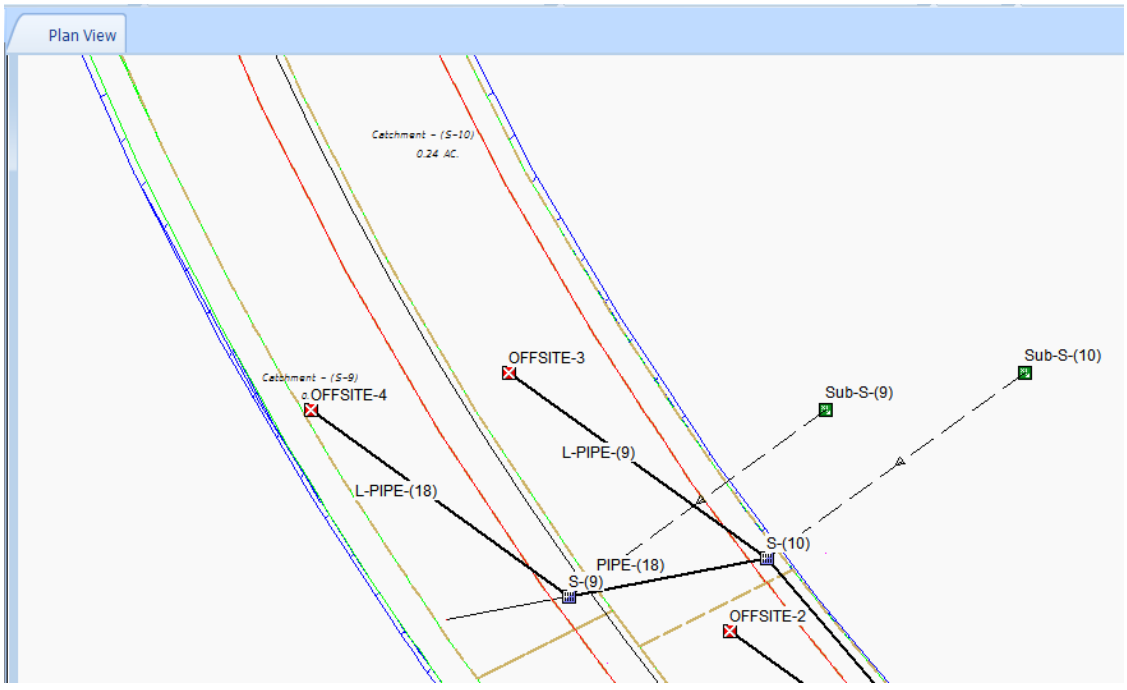
3. Check the box next to **Subbasin IDs, Node IDs, Link IDs**. Click **OK** to dismiss the Display Options dialog.



4. The labels overlap at this zoom scale. Try **zooming** in to where the labels do not overlap.



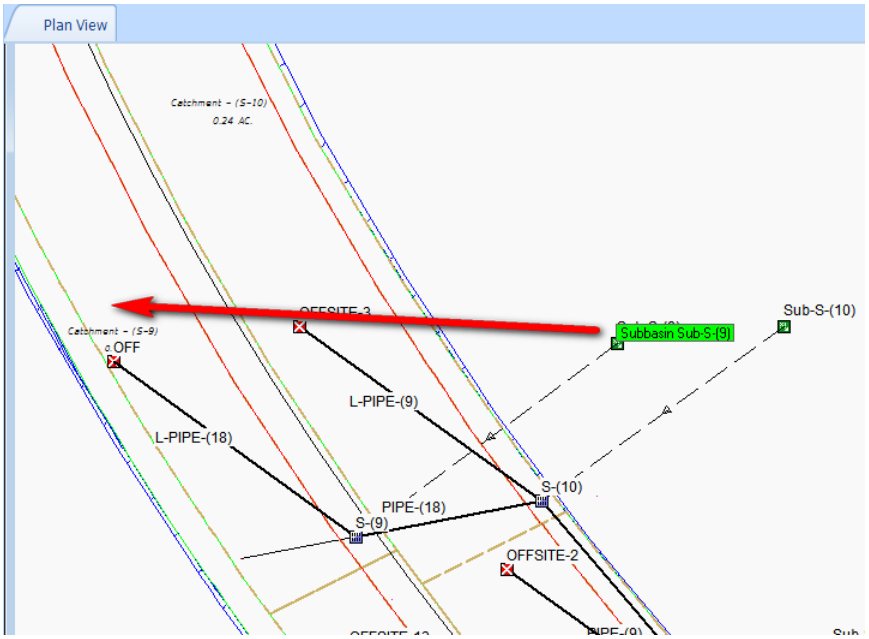
5. Use the wheel on the mouse to pan and **zoom** into the northern most Inlets **S-(9)** and **S-(10)**. Now the *Catchment areas* with their labels displays as background from the CAD file. Click the **Save** button.



Exercise 5.3 Move Subbasin Nodes Next to Catchment Labels

You can move nodes in SSA by clicking and dragging them. In this section you will move the subbasin nodes next to the corresponding Catchment label displayed by the background CAD file.

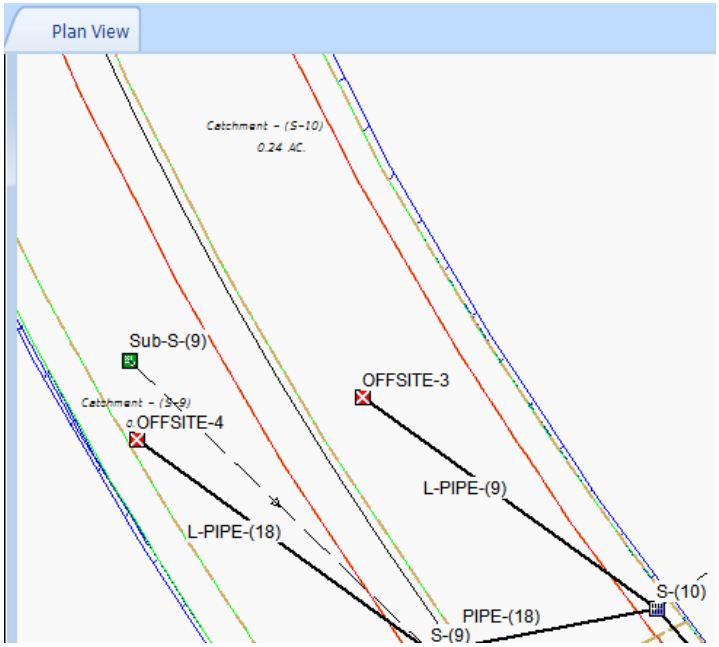
1. When you hover over *Subbasin Sub-S-(9)* a tool tip appears with a green background. The tool tip displays the *type of node (Subbasin)* and the *Node ID (Sub-S-(9))*. Left click the **green square** representing the subbasin node and left mouse drag it next to the label *Catchment - (S-9)* as indicated by the red arrow.



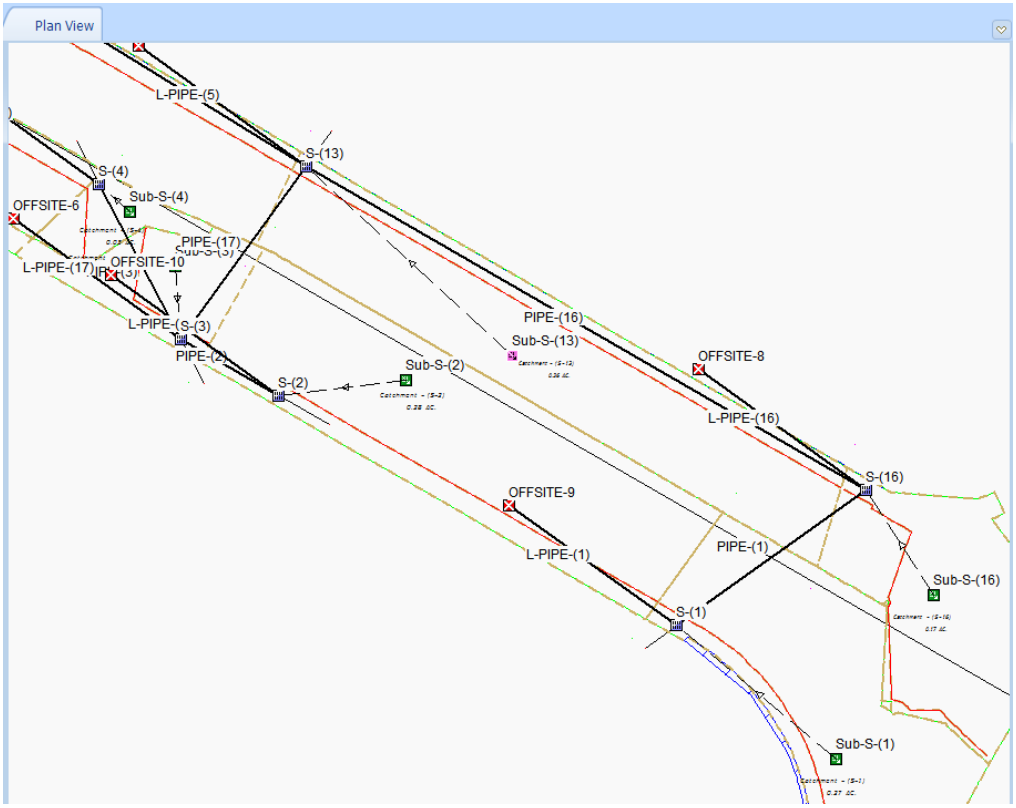
2. While still selected the subbasin node flashes green and purple. Click **ESC** to deselect the subbasin.

Note Why move the node? The reasons are mostly cosmetic in nature. You are organizing the content on the canvas so it appears to be in the correct location. The position does not effect the analysis.

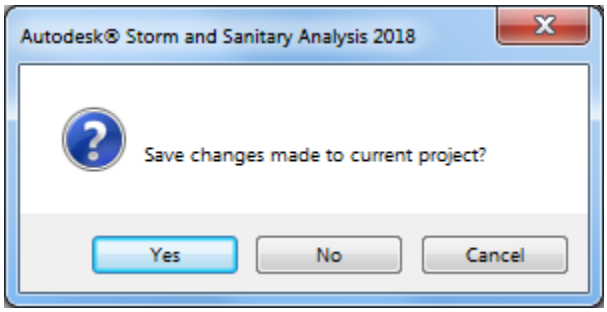
Continue moving the remaining subbasin nodes next to their corresponding Catchment label to complete the exercise.



3. This screen capture is an example of what the moved positions of the subbasin nodes might look like when you are done. Click the **Save** button to save the SSA project.



4. **Close** SSA.



5. **Save** changes made to current project? Click the **Yes** button if this dialog appears.

Note If you wish to skip this exercise I have saved a copy of the SSA Project with the completed task. Proposed Drainage - SR61 - moved subbasin nodes.spf in C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa.

6 DRAW BYPASS LINKS FOR INLETS SET ON GRADE

DESCRIPTION

This chapter you'll work in a methodical fashion to create bypass links, implement a naming convention for the bypass links, and delete the Outfall nodes created by the Import STM file process.

The suggested naming convention used in this task is not a requirement just a helpful suggestion for keeping organized with minimal effort by utilizing names automatically created by the software.

This method of creating bypass links and deleting Outfall nodes can help you pick up where you left off if you are interrupted while creating bypass links. The remaining Outfalls will indicate which bypass links have not been renamed.

OBJECTIVES

In this chapter, you will learn about:

- Drawing Conveyance links
- Deleting Outfall Nodes
- Connecting Inlets
- Replacing Bypass Link names

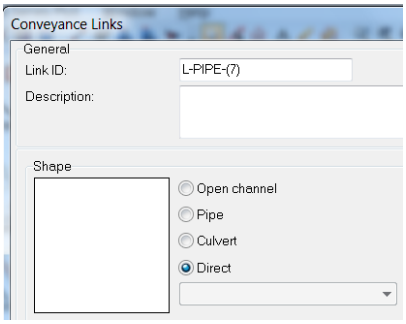
CHAPTER SETUP

Run the Chapter 6 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX** Projects folder.

Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

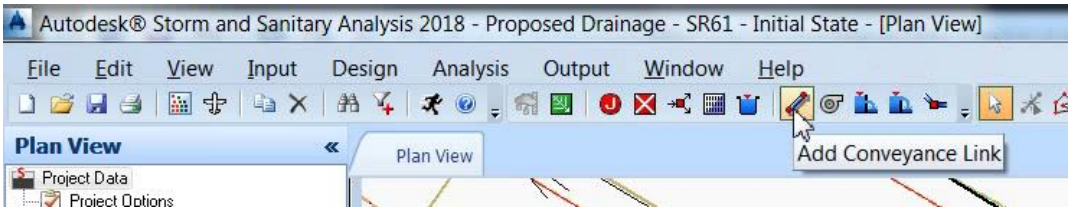
Exercise 6.1 Draw the Conveyance Link from Upstream to Downstream Inlets

Inlets on grade do not capture all the runoff. Some runoff will continue along the gutter of the roadway to the next downstream inlet. This gutter flow is call bypass flow. To account for this behavior, you must hydraulically connect upstream and downstream inlets with a bypass link. A bypass link is a conveyance link just like a pipe is a conveyance link.



For all bypass links in this Manual set the *Shape* to **Direct** in the Conveyance Links dialog as shown in the above example. In our examples using the Direct setting will simplify our workflow.

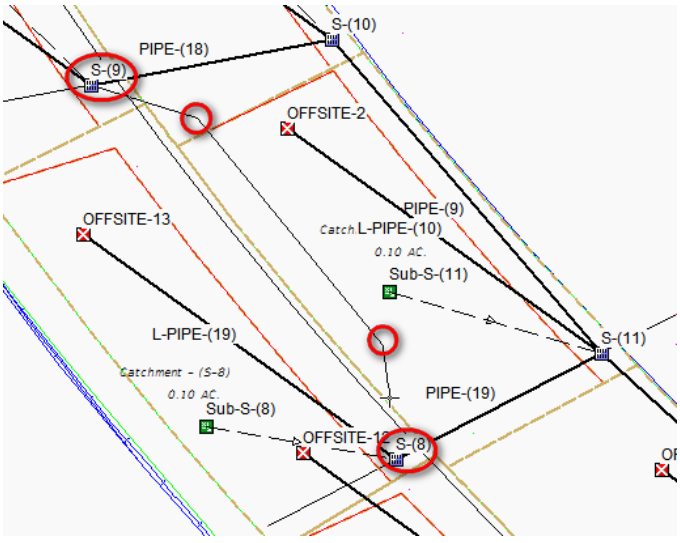
1. Start SSA and open C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61.spf
2. From the *Elements* toolbar use the **Add Conveyance Link** command.



3. Draw the vertices of the bypass link so they are clearly visible separately from the pipe.

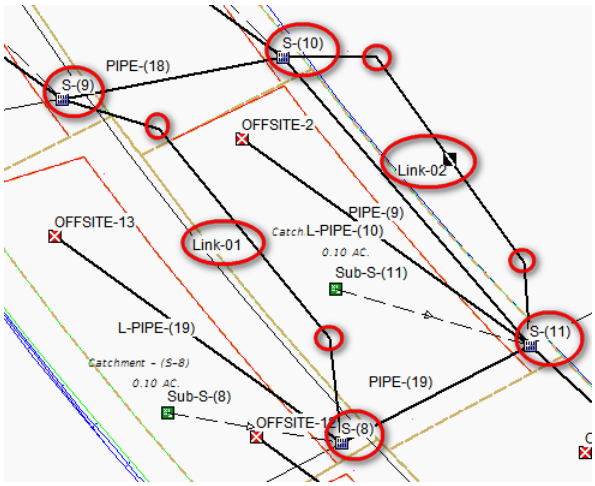
Note It is not necessary to have the bypass link directly on top of the pipe. The purpose is to connect an upstream inlet to the appropriate downstream inlet receiving the bypass flow. Keep in mind the downstream inlet for bypass flow may be different than the downstream node of the pipe. The image below shows the pick points represented by red circles. Bypass flow from S-(9) flows along the median gutter to S-(8). The from and to points are all that matter to SSA not the position of the link vertices.

Allow the software to automatically assign the Link ID. You will rename the links in a later step.

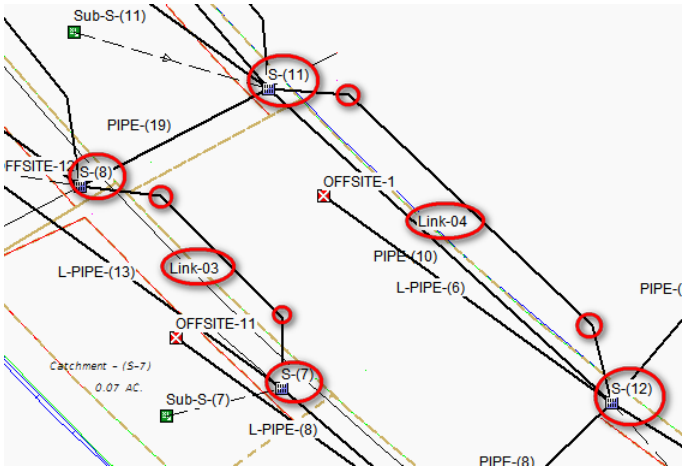


4. Draw bypass links for each of the remaining inlets from upstream to downstream using the **Add Conveyance Link** command. Use the screen captures below as a guide for each bypass link.

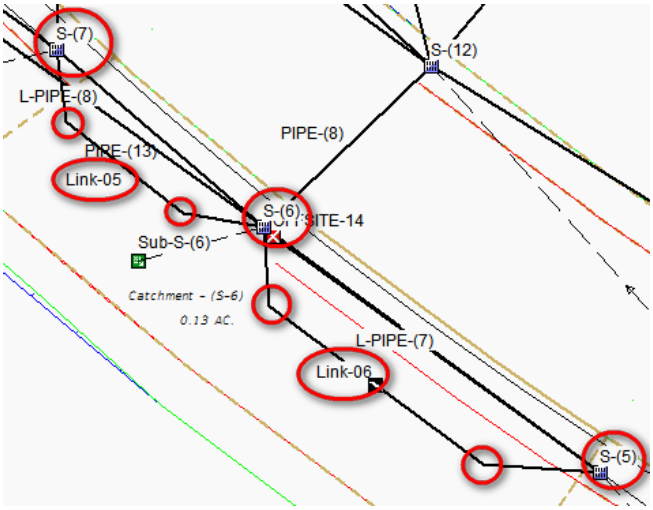
   a. S-(9) to S-(8) and S-(10) to S-(11)



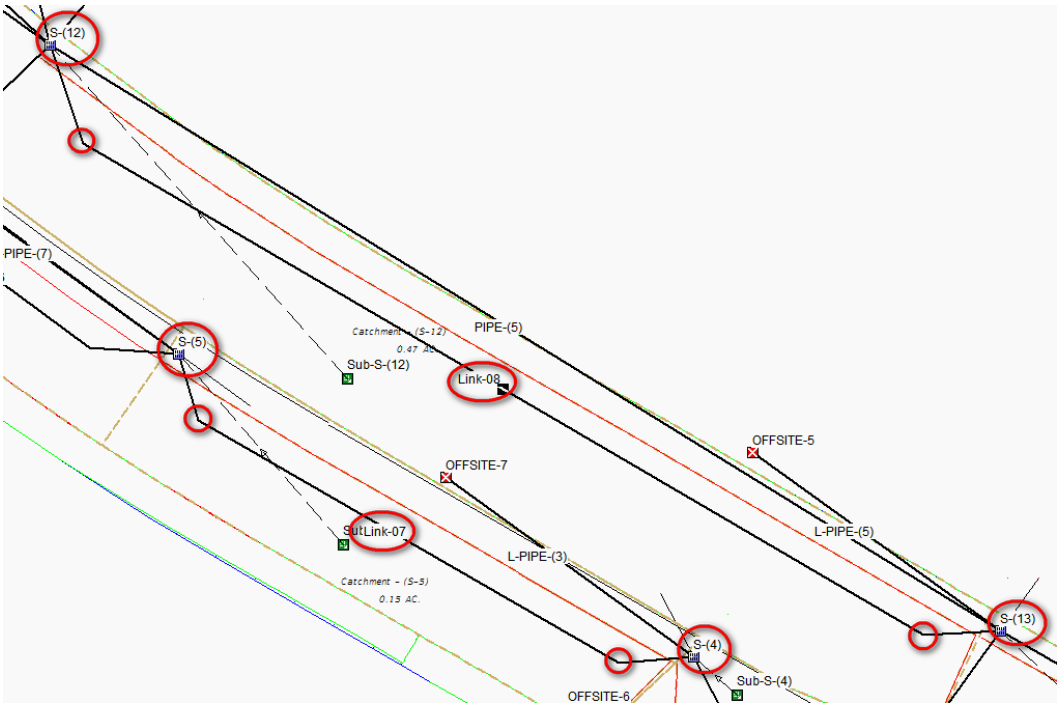
   b. S-(8) to S-(7) and S-(11) to S-(12)



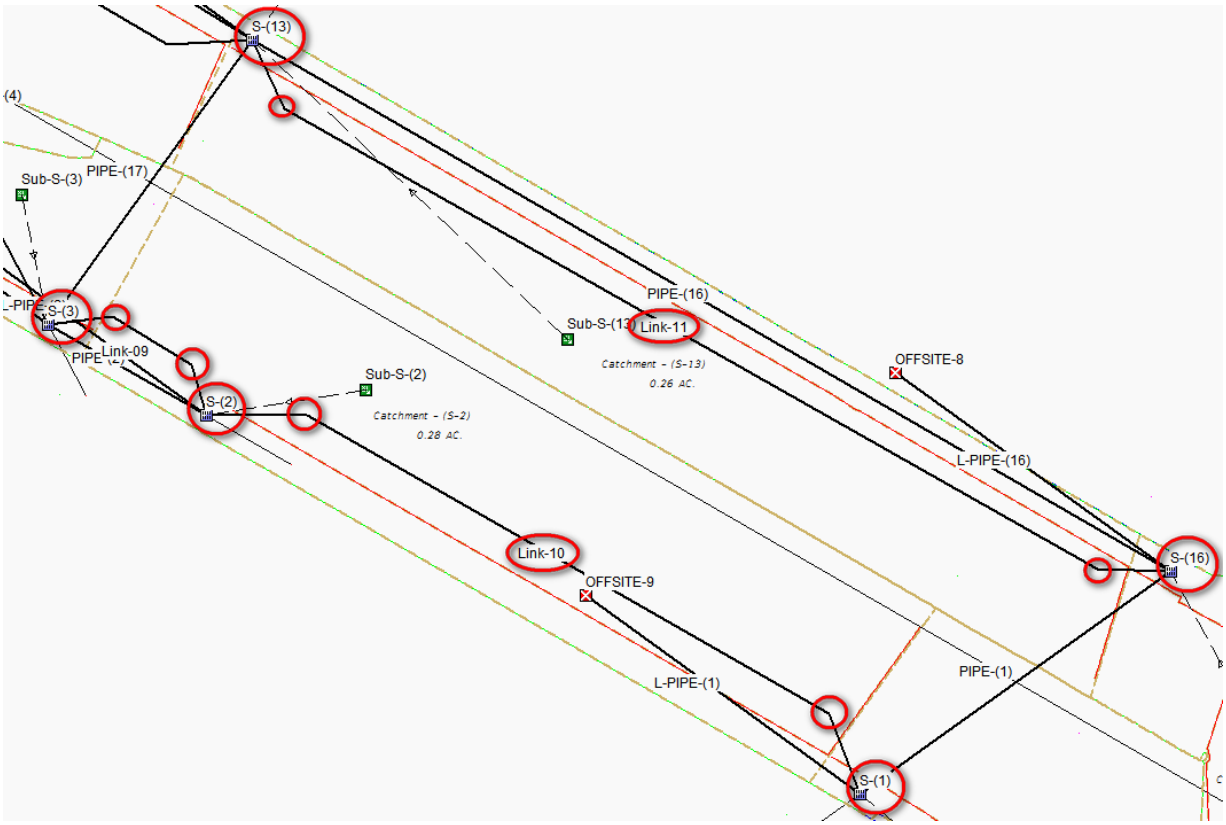
   c. S-(7) to S-(6) and S-(5) to S-(6)



d. S-(4) to S-(5) and S-(13) to S-(12)



e. S-(3) to S-(2) and S-(1) to S-(2) and S-(16) to S-(13)



5. **Save** the SSA project

DELETE THE OUTFALL NODES REPLACED BY THE BYPASS LINKS.

When the .stm file is imported the bypass link initially assigned to the Inlet goes to an Outfall node labeled OFFSITE-xx. You will replace this outfall and link with the bypass link created in a previous exercise. The bypass link hydraulically connects the upstream inlet to the downstream inlet.

Note You can save some editing if you assign the outfall link name to the newly created bypass link connected to the downstream inlet. You can avoid having to edit the Inlet properties to re-assign the bypass link if you rename the bypass link using the same name of the link currently connected to the outfall. After deleting the outfall node the connected conveyance link is also deleted. Duplicate names are not allowed in SSA so you must delete the link before reusing its name.

USE A NAMING CONVENTION FOR BYPASS LINKS TO REDUCE CONFUSION

Prefix the pipe name with L- for the bypass link name when the bypass link connects the same nodes as the pipe. For example: Pipe name is Pipe-(5) therefore the bypass link name is L-Pipe-(5)

When the bypass link does not follow the pipe path use the upstream structure name prefixed with L-bypass-. For example: Bypass link connecting S-(9) to S-(8) is named L-bypass-S-(9).

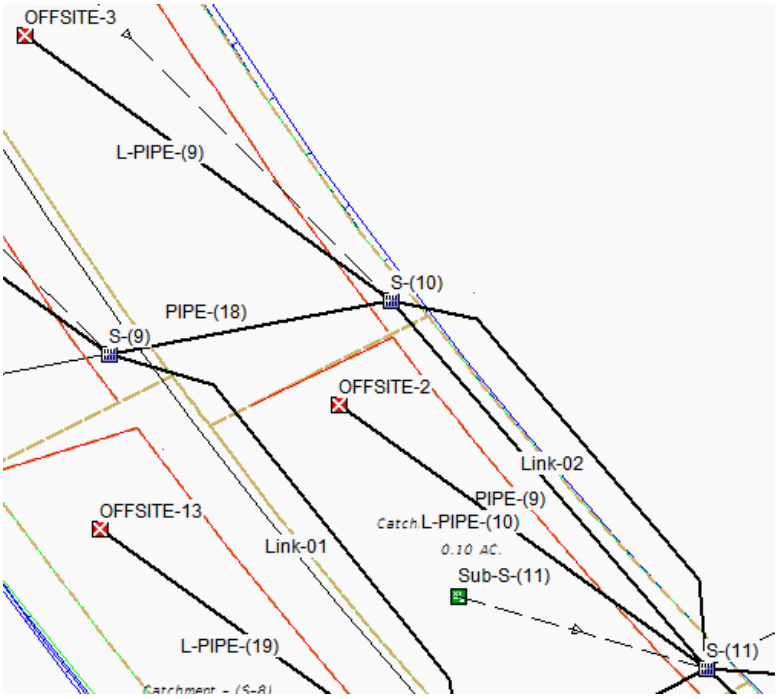
Having similar names for the Pipe (conveyance link) and the bypass link can be a great help in keeping organized and aware of how parts of the network are connected. Graphically it is easy to see the relationship but looking at names in a table in a dialog box is not so easy to keep track of those relationships.

If the clipboard copy/paste sequence is not your preferred method of making the edits just keep in mind the similar naming formats when you manually rename the conveyance links. This naming convention is not a requirement just a helpful suggestion for keeping organized minimizing effort by utilizing names automatically created by the software.

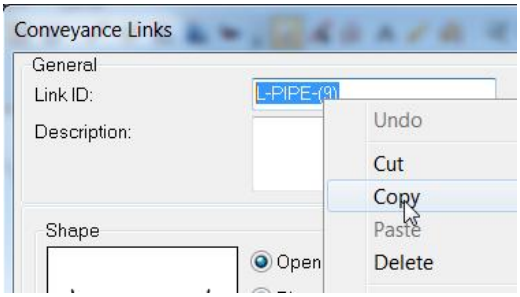
Exercise 6.2 Replace Bypass Link Name with Outfall Conveyance Link Name

Link-02 is drawn from S-(10) to S-(11) as is PIPE-(9). Notice the name of conveyance link L-PIPE-(9) to outfall OFFSITE-3 is similar to the name of the pipe PIPE-(9). You can copy the link name L-PIPE-(9) to the clipboard before deleting the outfall OFFSITE-3. Finally edit Link-02 and paste the name L-PIPE-(9) replacing the name Link-02. Now use the step by step process below to experience this workflow.

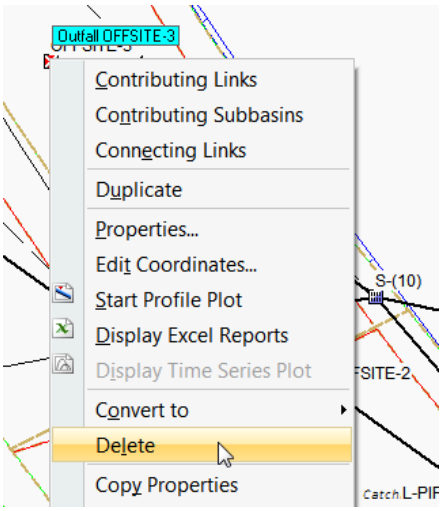
1. Double click **L-PIPE-(9)** to open the Conveyance Links dialog.



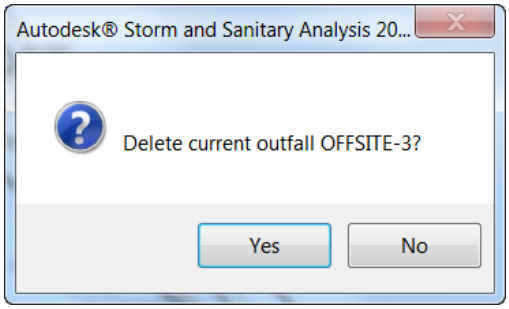
2. **Highlight** and **Copy** the *Link ID L-PIPE-(9)* to the Windows clipboard. **Close** the Conveyance Links dialog. Click the **Close** button.



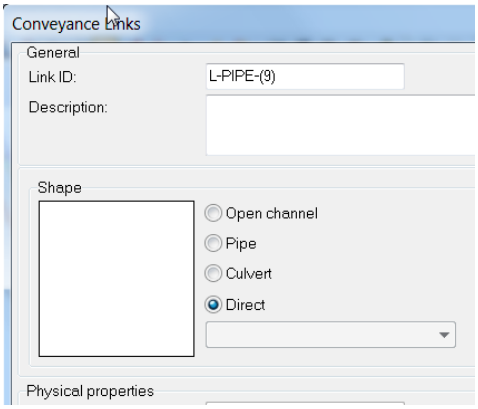
3. Select **outfall OFFSITE-3**, right click then select **Delete**.



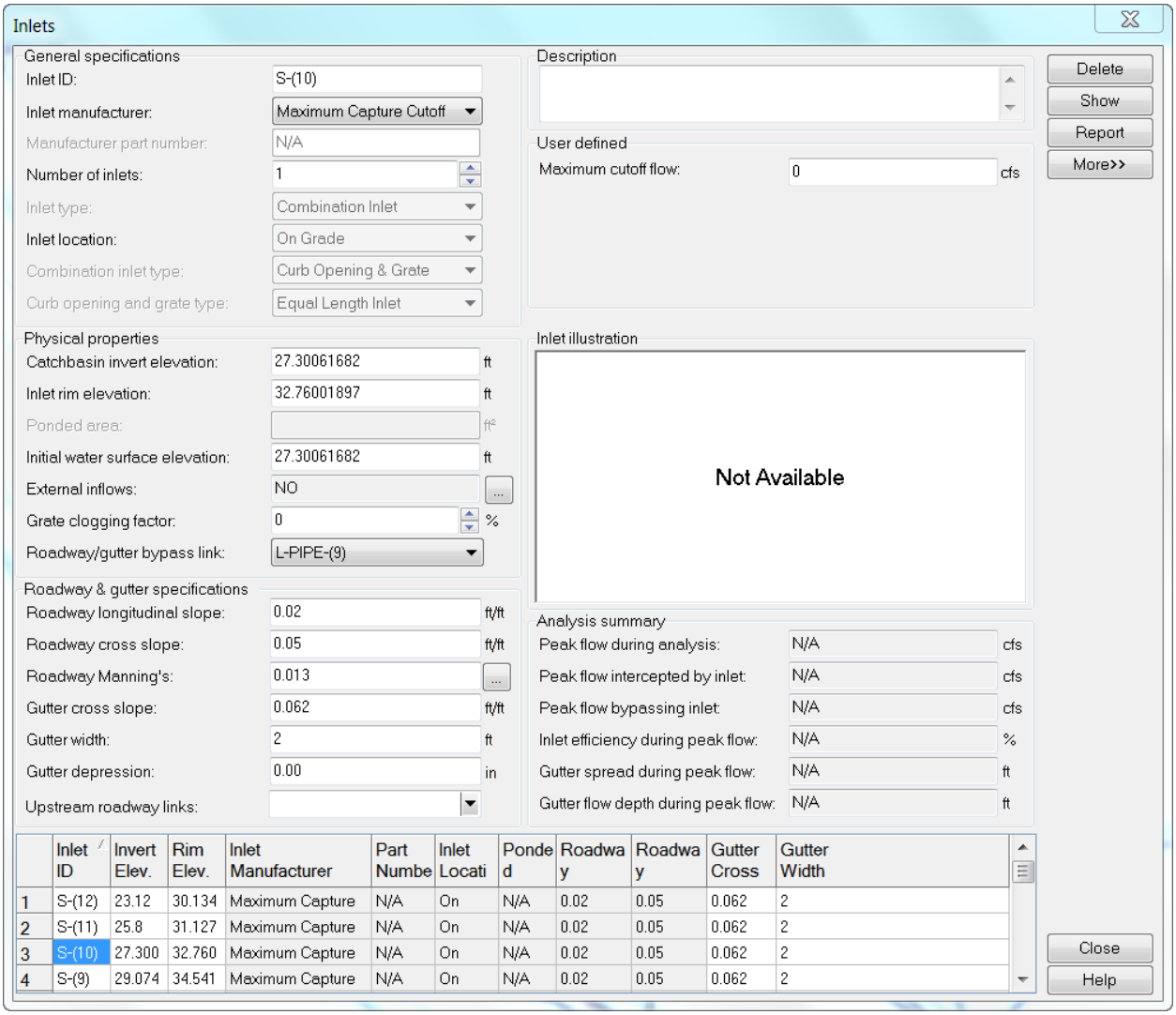
4. From the dialog, click **Yes** to delete *OFFSITE-3*.



5. Double click the **Link-02** *conveyance link* to open the Conveyance Links dialog.



6. **Highlight** and **Paste** the clipboard contents in the *Link ID* field to replace *Link-02* with *L-PIPE-(9)*.
7. Change the *Shape* to **Direct**. Click the **Close** button.
8. Double click the **upstream node S-(10)** to open the Inlets dialog.



9. Verify the *Roadway/gutter bypass link* is set to **L-PIPE-(9).** Click the **Close** button.

Note This workflow can help you pick up where you left off if you are interrupted while creating bypass links. The remaining Outfalls will indicate which bypass links have not been renamed.

10. Use a *naming convention* for bypass links to reduce confusion

Name Format	When to use the format
L-PIPE-(xx)	Prefix the pipe name with L- for the bypass link name when the bypass link connects the same nodes as the pipe. For example: Pipe name is Pipe-(5) therefore the bypass link name is L-Pipe-(5)
L-bypass-S-(xx)	When the bypass link does not follow the pipe path use the upstream structure name prefixed with L-bypass-. For example: Bypass link connecting S-(9) to S-(8) is named L-bypass-S-(9).

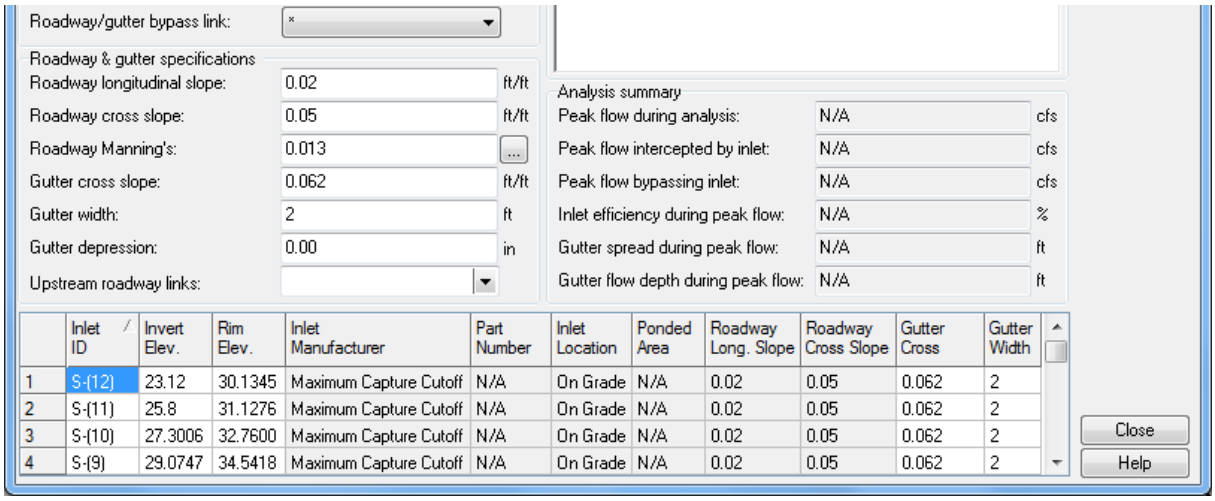
Note Having similar names for the Pipe (conveyance link) and the bypass link can be a great help in keeping organized and aware of how parts of the network are connected. This naming convention is not a requirement just a helpful suggestion for keeping organized minimizing effort by utilizing names automatically created by the software.

	ID	From Node	To Node	Shape
1	L-PIPE-(10)	S-(11)	S-(12)	Direct
2	L-PIPE-(13)	S-(7)	S-(6)	Direct
3	L-PIPE-(16)	S-(16)	S-(13)	Direct
4	L-PIPE-(2)	S-(3)	S-(2)	Direct
5	L-PIPE-(5)	S-(13)	S-(12)	Direct
6	L-PIPE-(7)	S-(5)	S-(6)	Direct
7	L-PIPE-(9)	S-(10)	S-(11)	Direct
8	L-bypass-S-(1)	S-(1)	S-(2)	Direct
9	L-bypass-S-(4)	S-(4)	S-(5)	Direct
10	L-bypass-S-(8)	S-(8)	S-(7)	Direct
11	L-bypass-S-(9)	S-(9)	S-(8)	Direct

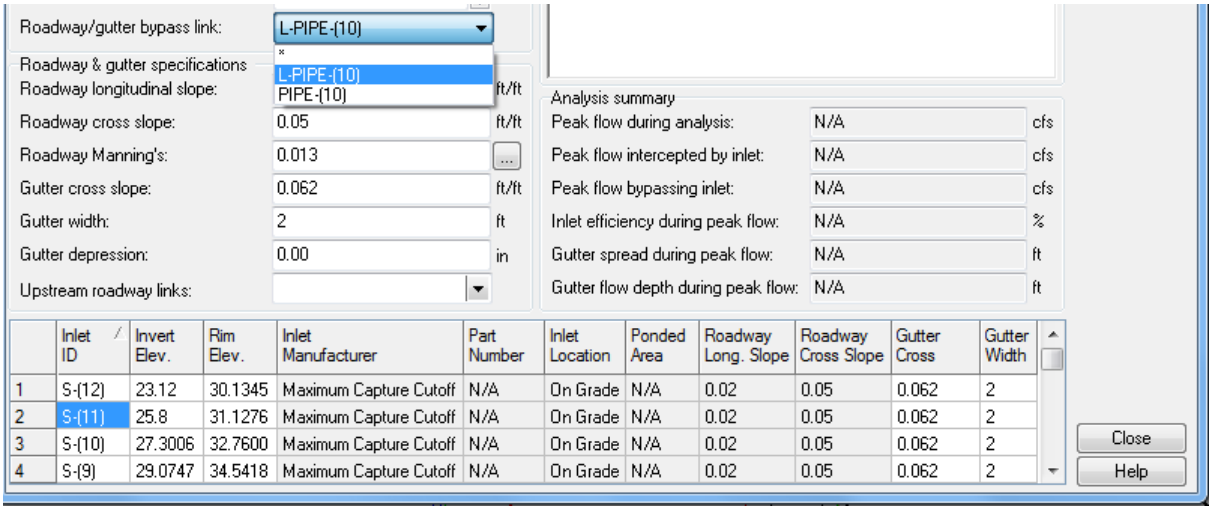
11. Continue deleting the *Outfall nodes* one at a time as you **rename** each *bypass link*. The only *outfall* in this pipe network is at *Pipe-(6)* which flows out from *structure S-(12)*. All other *Outfall nodes* should be deleted, **OFFSITE-4, OFFSITE-1**, and **OFFSITE-6**. The final list of *Link IDs* is shown above.

Note Double click any link to open the Conveyance Links dialog. At the bottom of the dialog you will find the list of Link IDs

12. Double click **inlet S-(9)** to open the Inlets dialog. Use the scroll bar on the right side of the table at the bottom of the dialog to scroll to the top of the list. Verify the *upstream node* has the correct bypass link assigned.



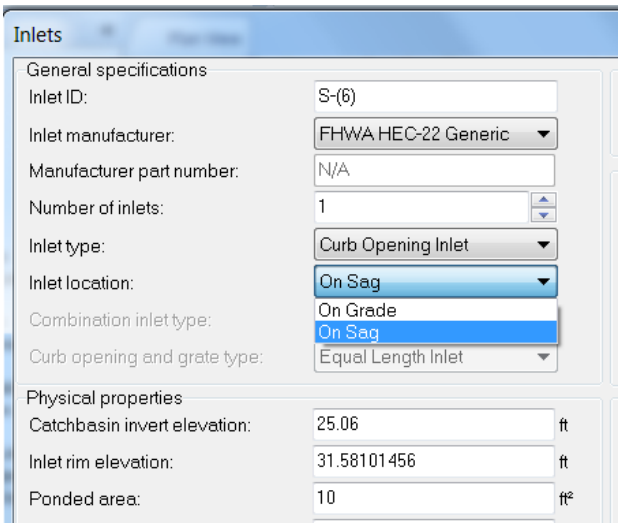
Note S-(12) is SAG inlet and has no bypass link.



13. Stay in the Inlets dialog, scroll through the list one at a time, review the *Inlet ID* and assign *Roadway/gutter bypass link*. The results should match the list below:

- ✓ S-(11) is assigned bypass link L-PIPE-(10).
- ✓ S-(10) is assigned bypass link L-PIPE-(9).
- ✓ S-(9) is assigned bypass link L-bypass-S-(9).
- ✓ S-(13) is assigned bypass link L-PIPE-(5).
- ✓ S-(3) is assigned bypass link L-PIPE-(2).
- ✓ S-(4) is assigned bypass link L-bypass-S-(4).
- ✓ S-(16) is assigned bypass link L-PIPE-(16).
- ✓ S-(1) is assigned bypass link L-bypass-S-(1).
- ✓ S-(2) is SAG inlet and has no bypass link.
- ✓ S-(6) is SAG inlet and has no bypass link.
- ✓ S-(7) is assigned bypass link L-PIPE-(13).
- ✓ S-(8) is assigned bypass link L-bypass-S-(8).
- ✓ S-(7) is assigned bypass link L-PIPE-(13).
- ✓ S-(5) is assigned bypass link L-PIPE-(7).

14. Change the *Inlet location* to **On Sag** for S-(6), and S-(2). Click the **Close** button.



15. **Save** the SSA project and **Close** SSA.

7 RUN ANALYSIS FOR ASCII OUTPUT REPORT

DESCRIPTION

The drainage network is defined and ready to run an analysis for a storm event. This chapter describes how to specify options to be used in the analysis and how to run the analysis.

OBJECTIVES

In this chapter, you will learn about:

- Setup Analysis for Rational Method
- Check Analysis Options
- Specify the IDF Curves
- Running an Analysis
- Typical Errors to Resolve

CHAPTER SETUP

Run the Chapter 7 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX** Projects folder.

Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

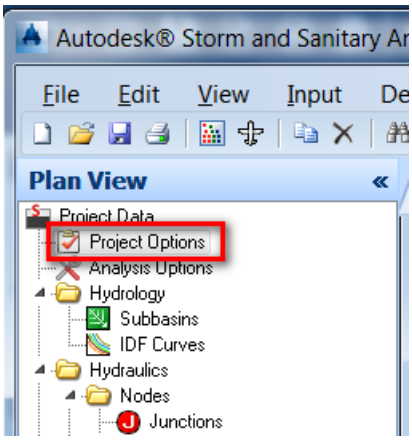
FROM THE SSA HELP FILE

Select ANALYSIS > PERFORM ANALYSIS or click the PERFORM ANALYSIS icon from the Standard toolbar. Before the software begins the simulation, the built-in Model Checker reviews the defined input data for any omissions or potential problems with the model data. If it encounters an error with the input data, it will explain what is wrong and how to correct it. The Model Checker can be thought of as an expert modeler, pointing out any errors contained within the model.

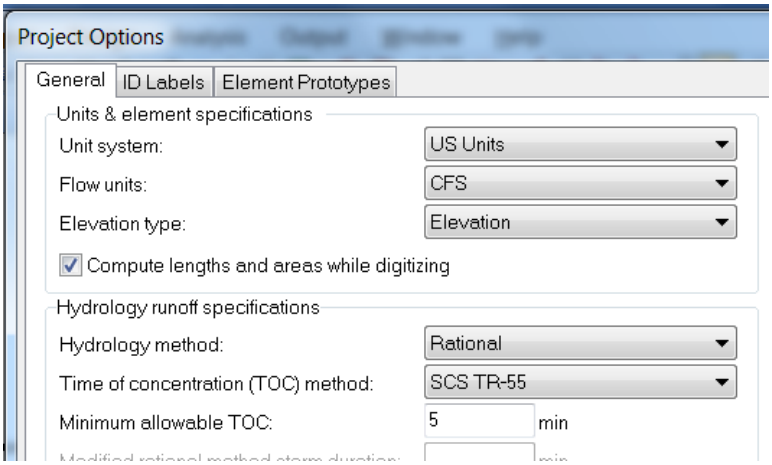
Exercise 7.1 Setup Analysis

➢ *Setup Hydrology Method in the Project Options*

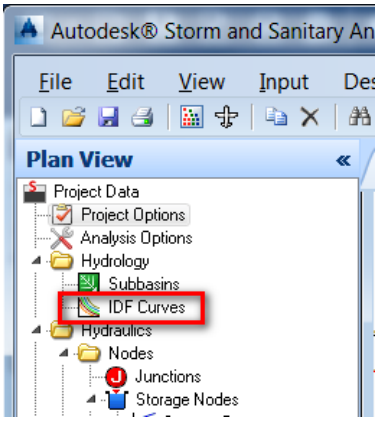
1. Start **SSA** and open the project file C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61.spf.
2. Double click the **Project Options** under *Project Data* in the *Data Tree* to open the Project Options dialog.



3. Set the *Hydrology method* to **Rational**. Set the *Minimum allowable TOC* to **10 min**. Click **OK**.

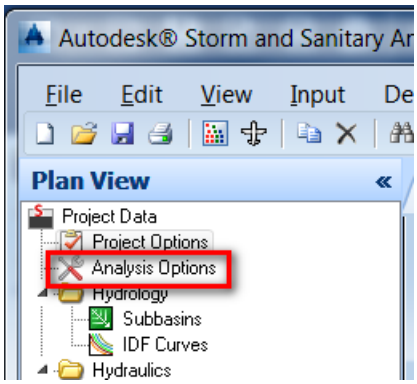


4. When the *Rational Method* is selected in the *Project Options*, the Data Tree displays **Subbasins** and **IDF Curves nodes** under the *Hydrology* branch.

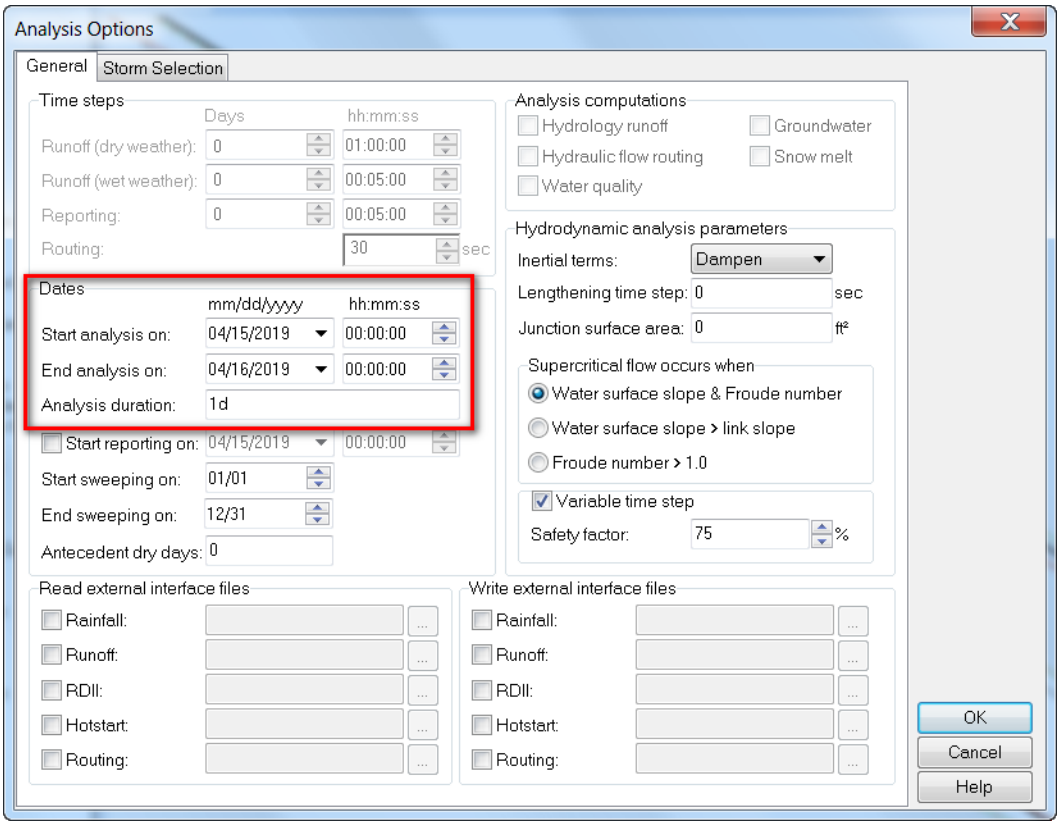


➢ *Check Analysis Options*

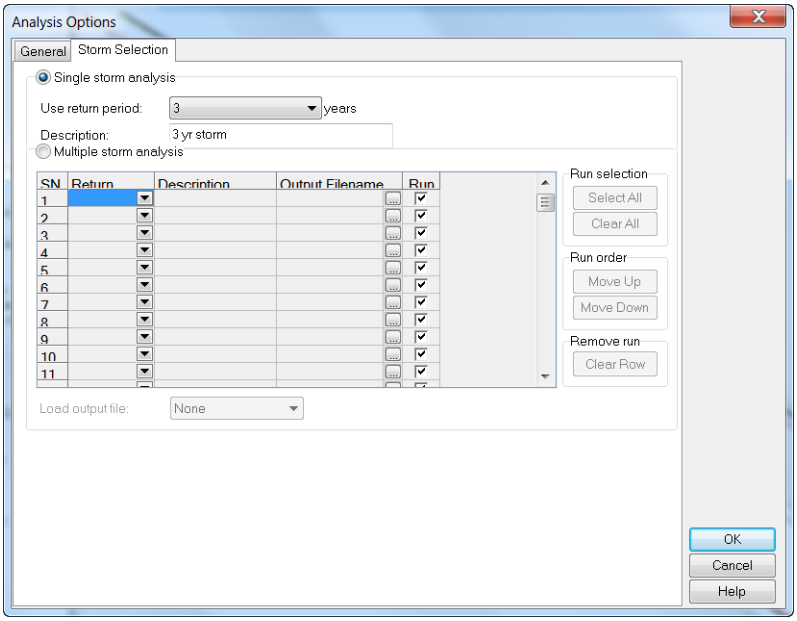
1. Double click the **Analysis Options** under *Project Data* in the *Data Tree* to open the Analysis Options dialog.



2. On the *General* tab check your *Dates area* to make sure you have **1 day** for the *Analysis duration*. Click **OK**.



3. Check the *Storm Selection* tab to make sure you have specified a *Single storm analysis* and set the *Use return period* to **3 years**. The *Description* you enter will appear in the *Analysis output*. Click **OK**.



➢ *Specify the IDF Curves for the Project*

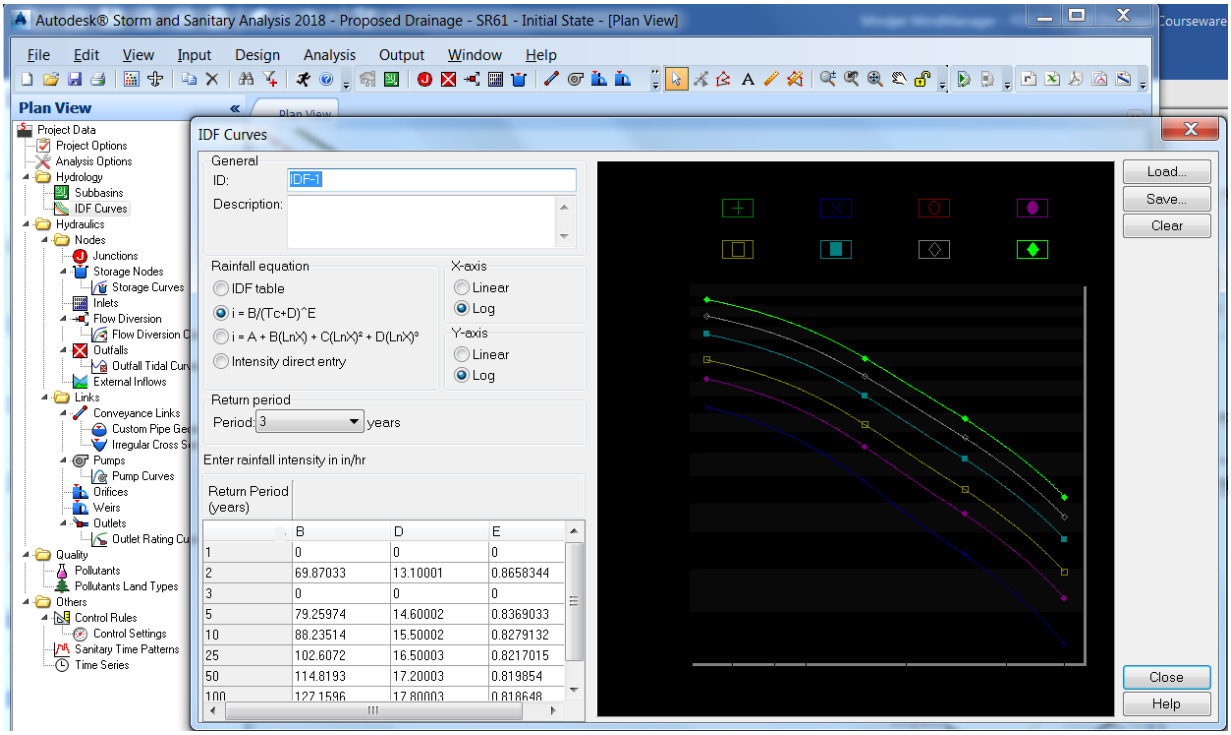
The Rational Method uses IDF Curve data instead of rain gage data. See page 418 in the SSA Help File, Chapter 9, Subbasin Element Data, IDF Curves for more a more detailed explanation of IDF Curves.

The FDOT Civil 3D State Kit provides IDF Curves files in a SSA compatible format which can be found in the following folder. C:\FDOT20XX.C3D\Data\SSA\IDF Curve\

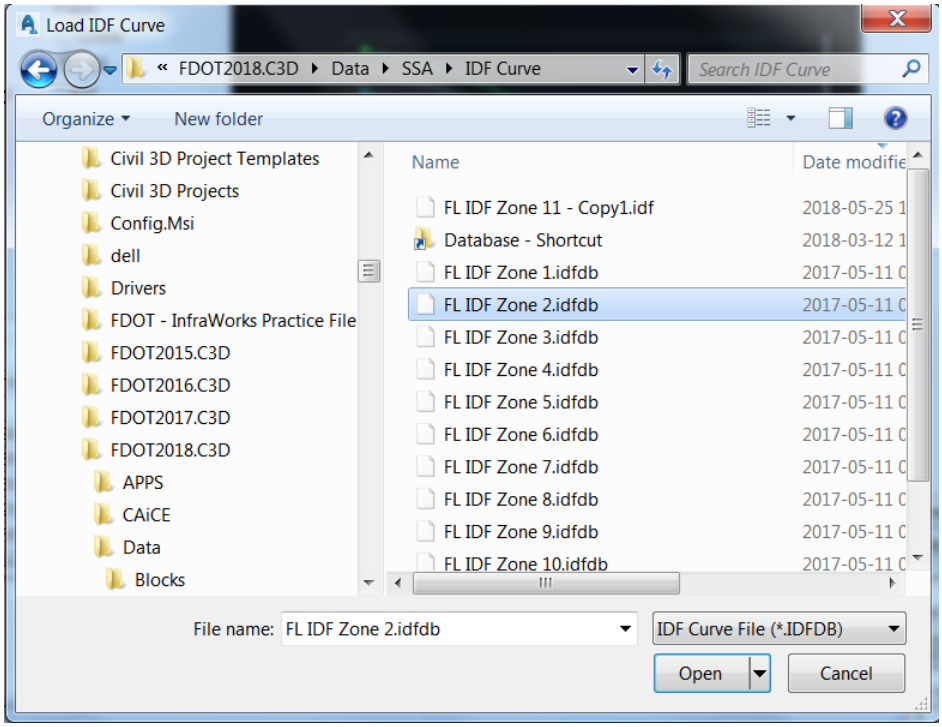
The IDF Curve Data Files are organized by Zones. The FDOT Drainage Manual Figure B-1 shows the IDF Curve Zones on a map of Florida: http://www.fdot.gov/roadway/Drainage/files/IDFCurves.pdf.

The map below shows our project in Wakulla County is located in Zone 2.

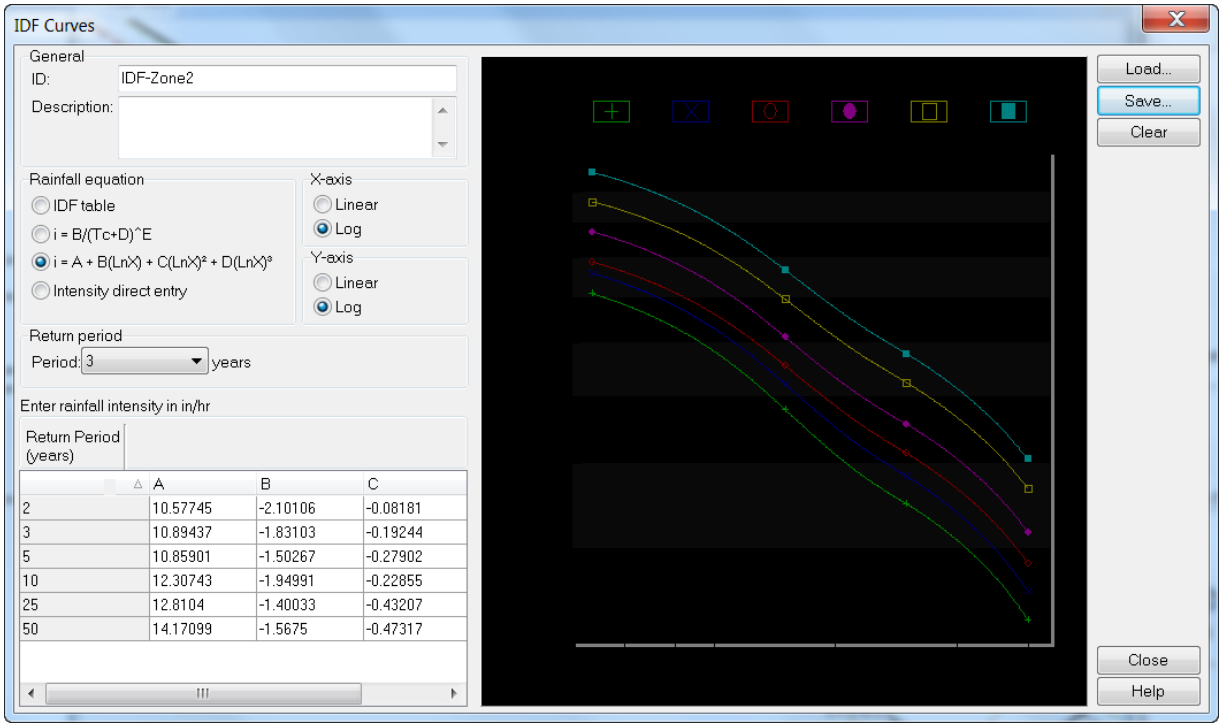
1. Double click the **IDF Curves** under *Project Data* in the Data Tree to open the IDF Curves dialog. Click the **Load** button in the upper right corner of this dialog.



2. Using the Load IDF Curves dialog, browse to the folder C:\FDOT20XX.C3D\Data\SSA\IDF Curve\ and select the file *FL IDF Zone 2.idfdb*. Click the **Open** button to return to the IDF Curves dialog.

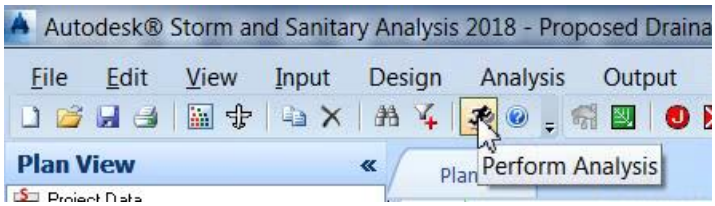


3. Change the *ID* to **IDF-Zone2**. Set the *Period* to **3 years**. Leave all other settings as shown below. Click the **Close** button in the lower right of the dialog.

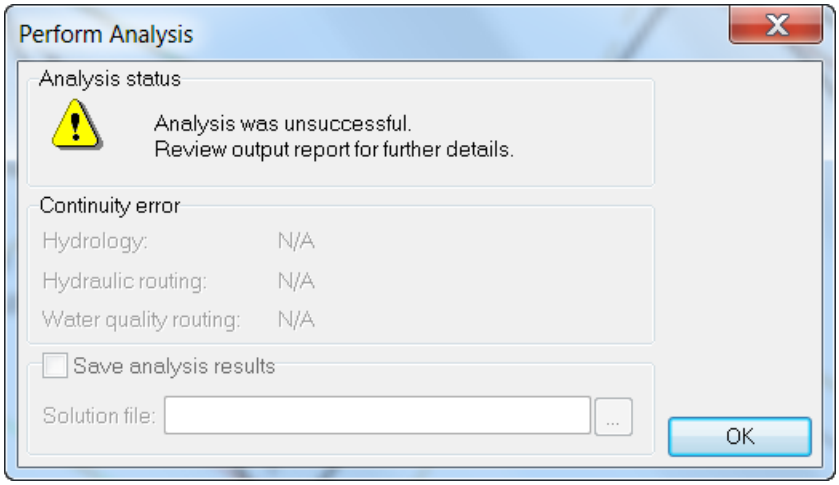


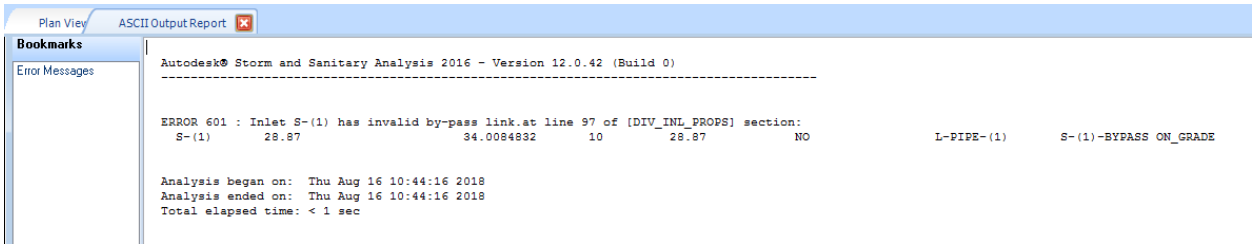
Exercise 7.2 Run Analysis

1. From the *Standard* toolbar, select the **Perform Analysis** command.



2. If there are errors in the Perform Analysis dialog the *Anaysis status* will display a message *Analysis was unsuccessful*. Click **OK** and the *ASCII Output Report* tab displays.

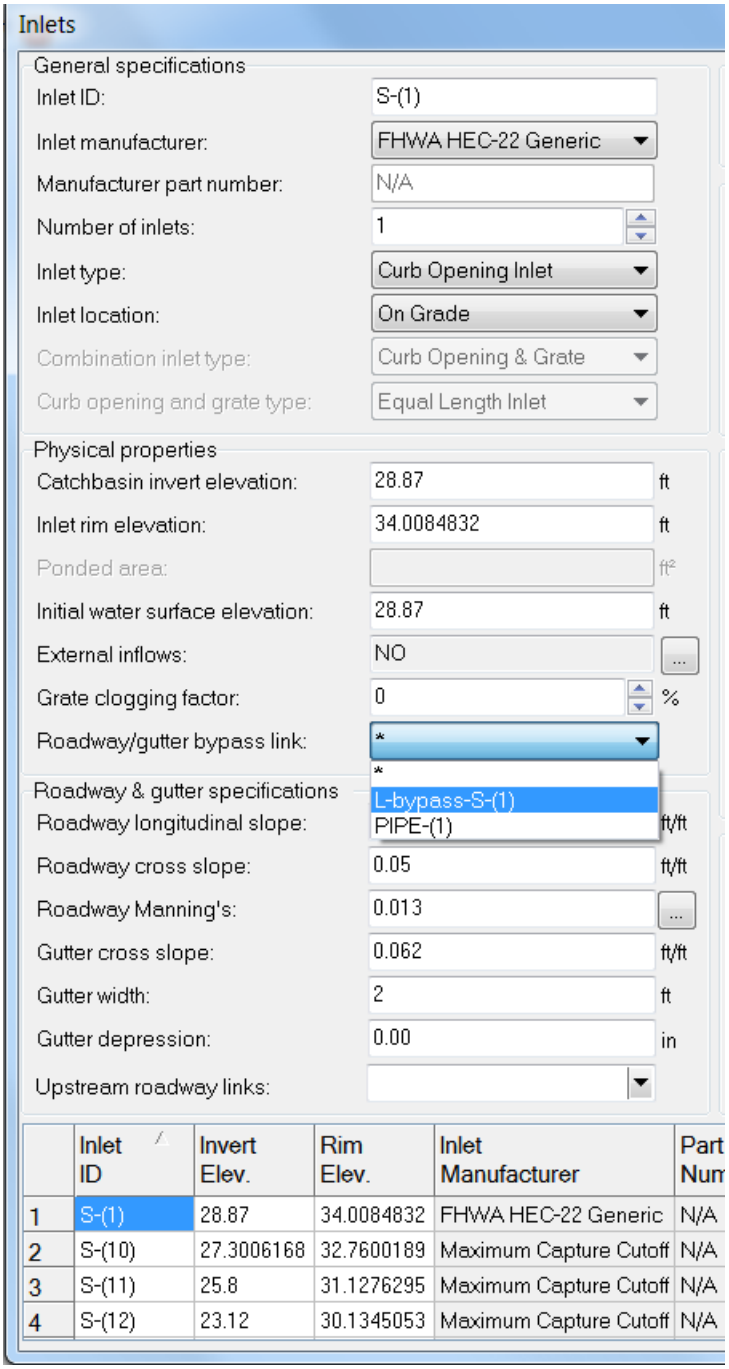




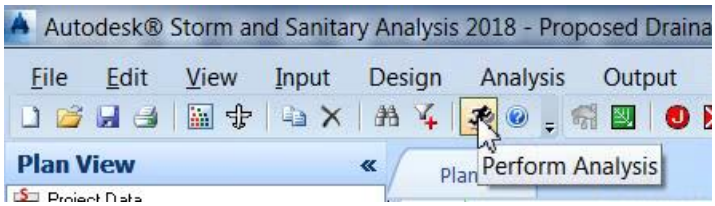
Note ERROR 601: Inlet *nnn* has invalid by-pass link.

The BYPASS LINK entry defines the ID of the link that receives any flow that bypasses the storm drain inlet. If the inlet location is defined as located ON SAG, then this entry is grayed out since there is no bypass link for sag inlets. The provided drop-down list provides a listing of links (e.g., channels, pipes, pumps, orifices, weirs, or outlets), allowing you to select the bypass link.

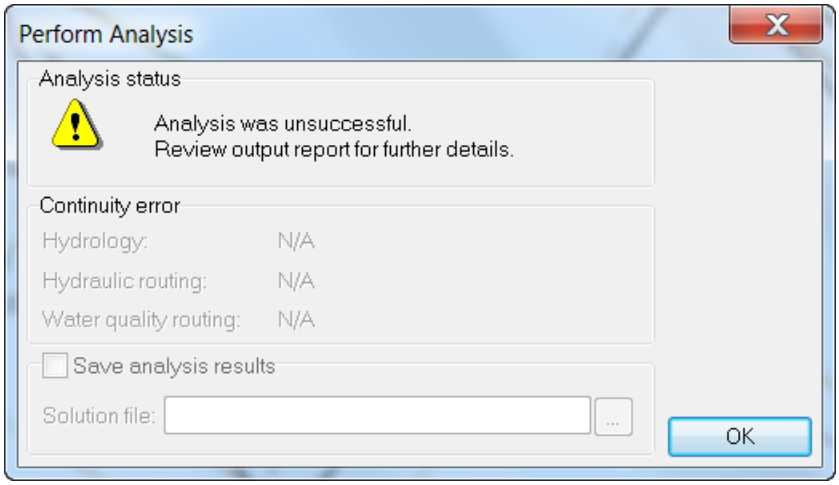
3. Double click **S-(1)** to open the Inlets dialog. The bypass link is not assigned. Set the *Roadway/gutter bypass link* to **L-bypass-S-(1).** Click **Close**.

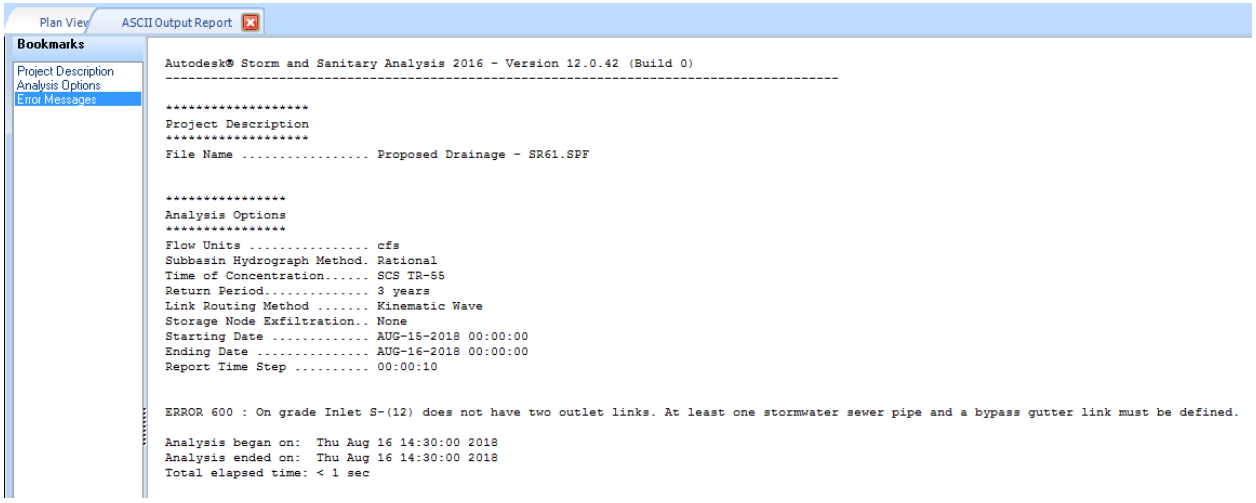


4. **Rerun Analysis**. From the *Standard* toolbar, Select the **Perform Analysis** command.



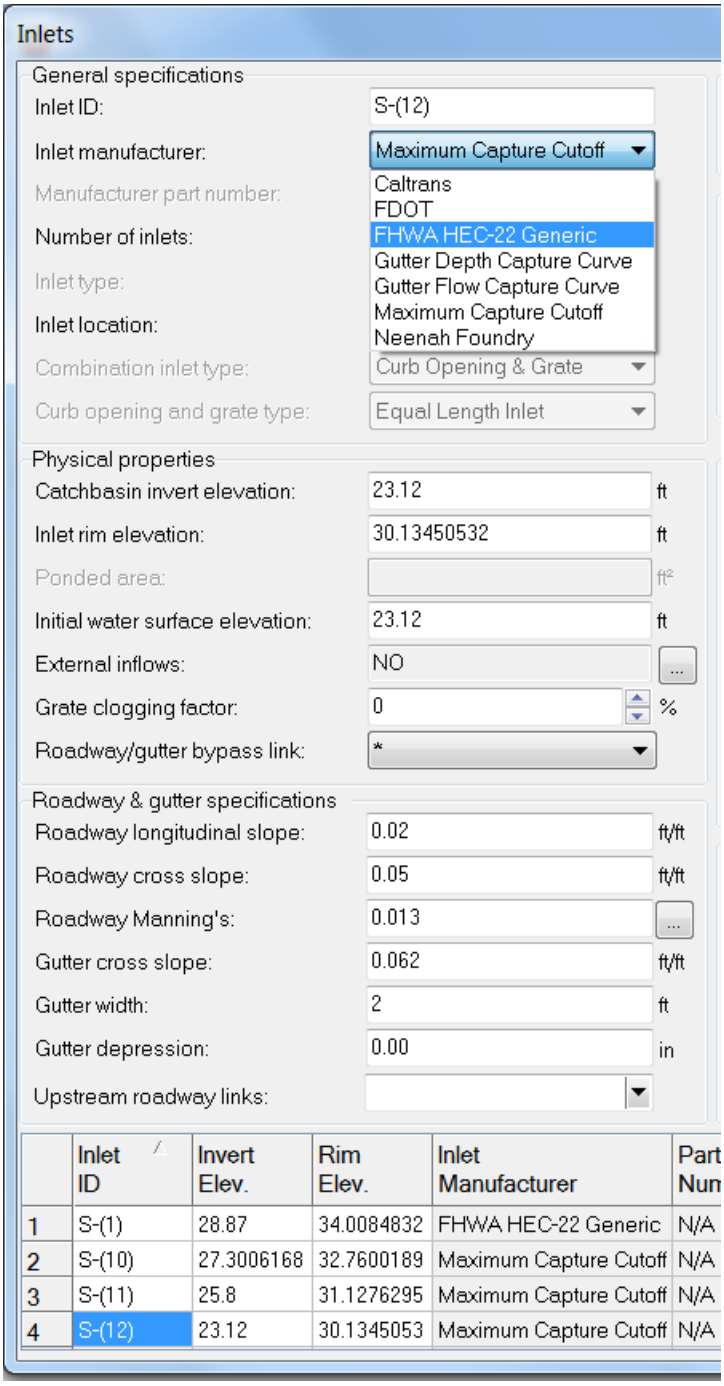
5. If there are errors in the Perform Analysis dialog the *Anaysis status* will display a message *Analysis was unsuccessful*. Click **OK** and the *ASCII Output Report* tab displays.



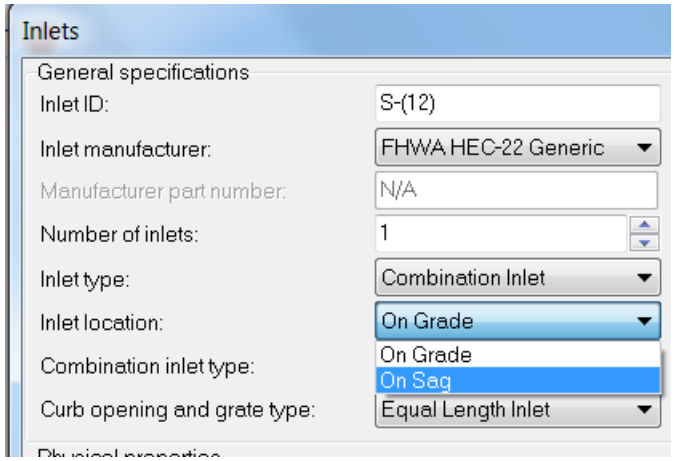


Note ERROR 600: On grade Inlet nnn does not have two outlet links. At least one stormwater sewer pipe and a bypass gutter link must be defined.

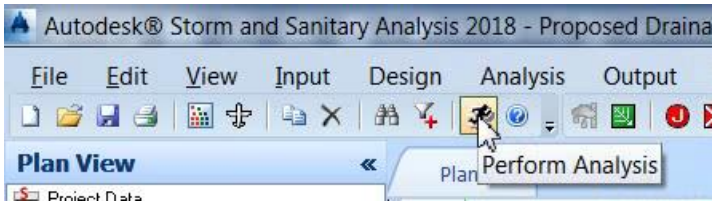
6. Double click on **S-(12)** to open the Inlets dialog. The *Inlet location* shows **On Grade**. **S-(12)** is a *sag Inlet*. Sag Inlets don't require 2 outlets as indicated by ERROR 600. The *Inlet Location* drop down list is grayed out.



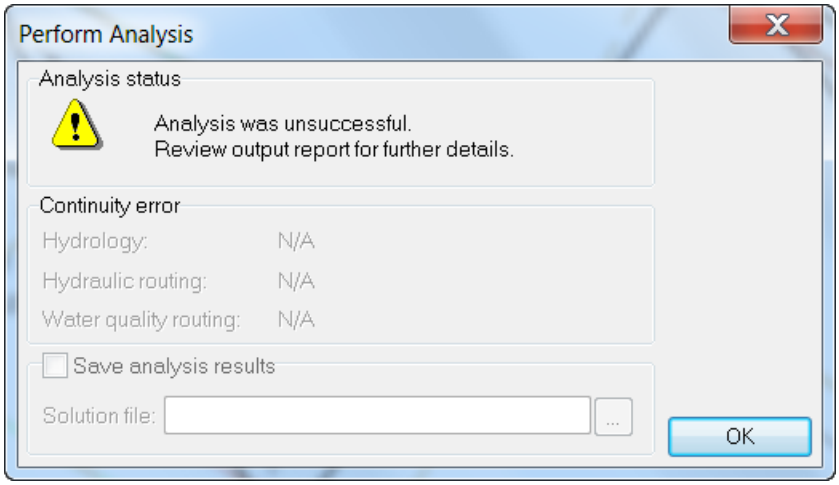
7. To correct this problem, you must change the *Inlet manufacturer*. Choose **FHWA HEC-22 Generic** from the Inlet dialog. Set the *Inlet Location* to **On Sag**.



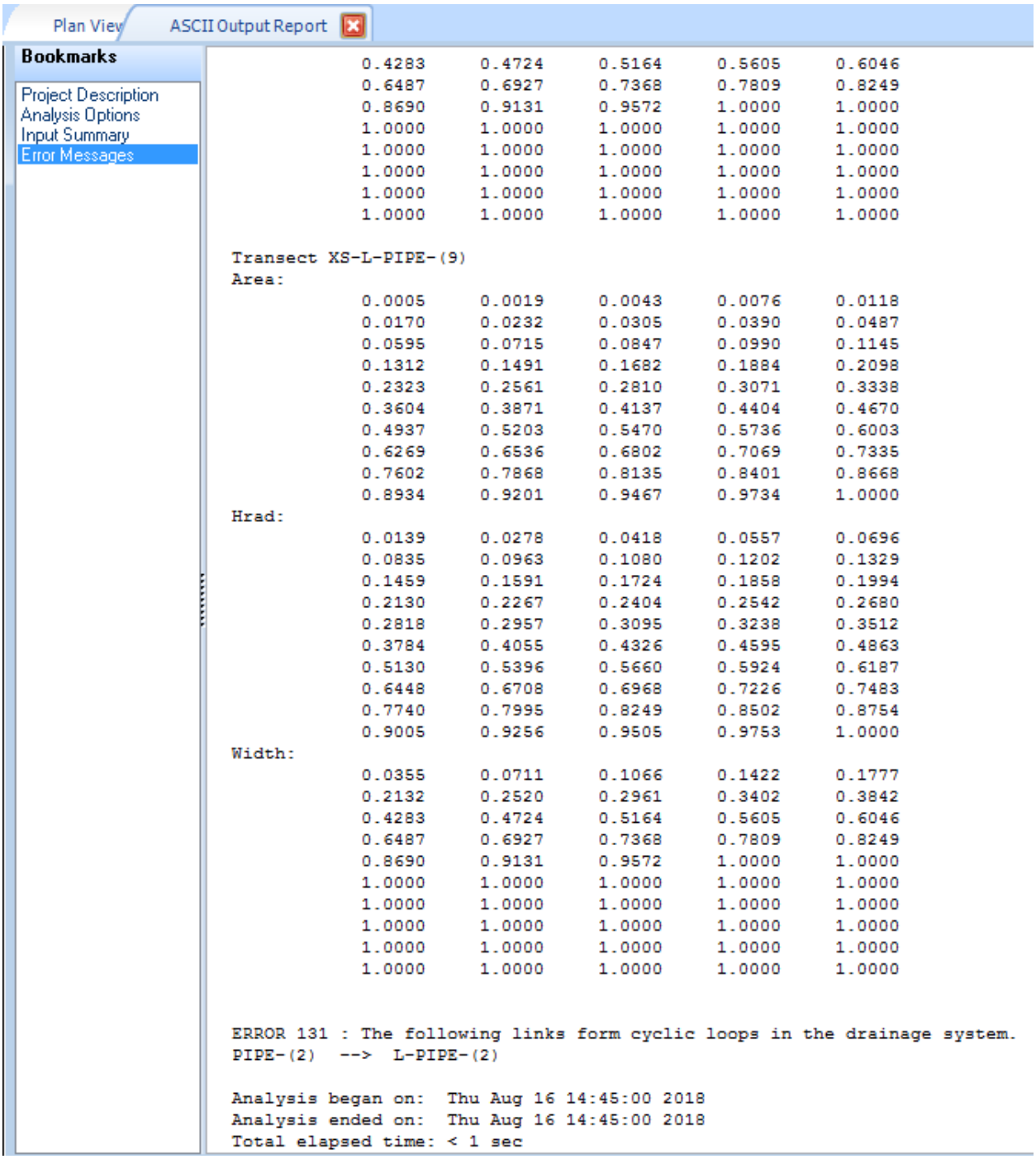
8. **Rerun Analysis**. From the *Standard* toolbar, select the **Perform Analysis** command.



9. If there are errors in the Perform Analysis dialog the *Anaysis status* will display a message *Analysis was unsuccessful*. Click **OK** and the *ASCII Output Report* tab displays.



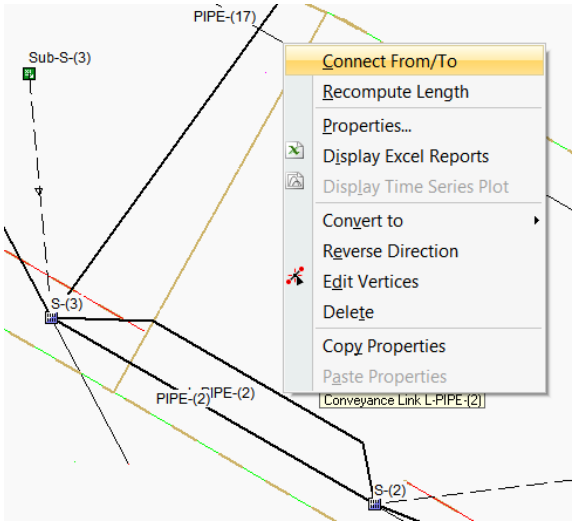
10. Click the **Error Messages** link in the *Bookmarks* panel to navigate to the bottom of the report.



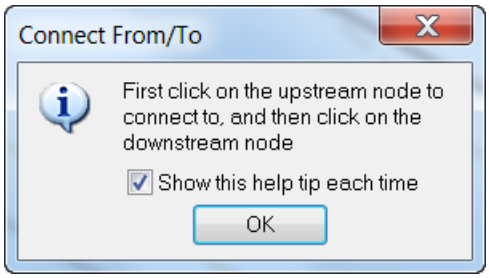
Note ERROR 131: The following links form cyclic loops in the drainage system.The Steady and Kinematic Wave flow routing methods cannot be applied to systems where a cyclic loop exists (i.e., a directed path along a set of links that begins and ends at the same node). The same is true for Hydrodynamic routing when water quality analysis is performed.

In this case the pavement gutter is bypassing S-(3) and flowing toward S-(2) (which is a sag inlet) but the pipe connected to S-(2) flows to S-(3). The design is correct but the internal logic in SSA views the bypass link to be pointing to an upstream node which it deems invalid. Ultimately the bypass flow from S-(3) will go downstream to S-(13) and we must configure the bypass link for S-(3) to agree with this logic.

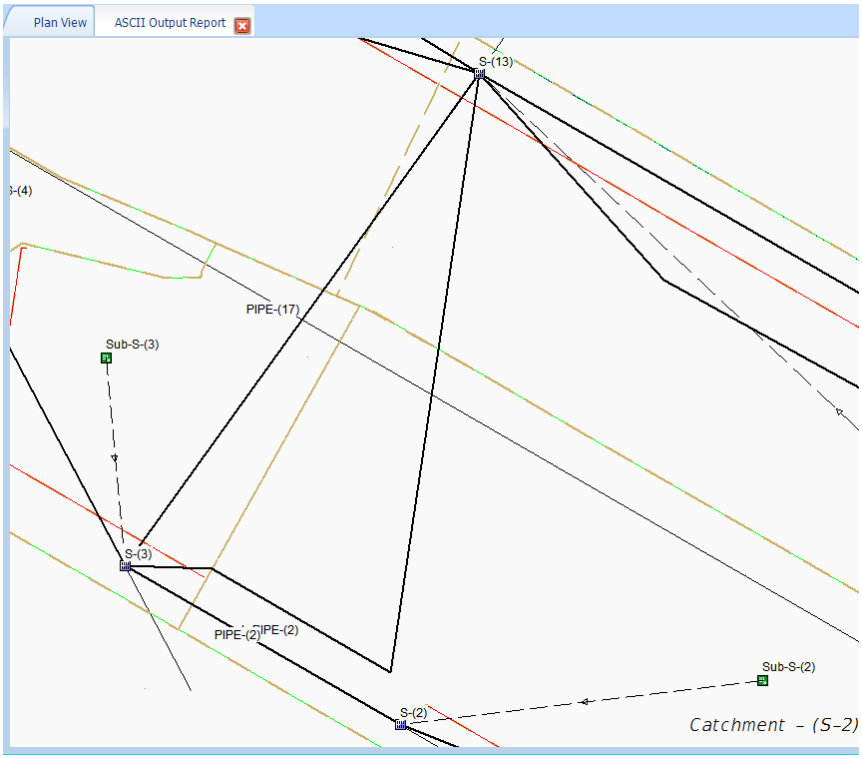
11. Click **L-PIPE-(2)** to select it then right click **L-PIPE-(2)** to display the *popup* menu, then select the **Connect From/To** command.



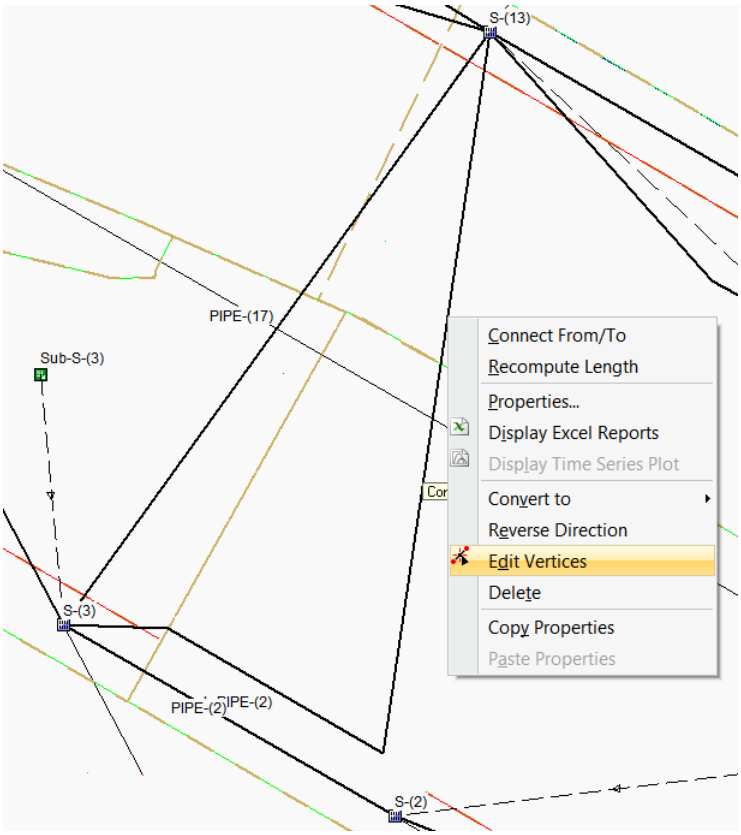
12. The Connect From/To box opens and provides instructions on how to use the command. Click **OK**.



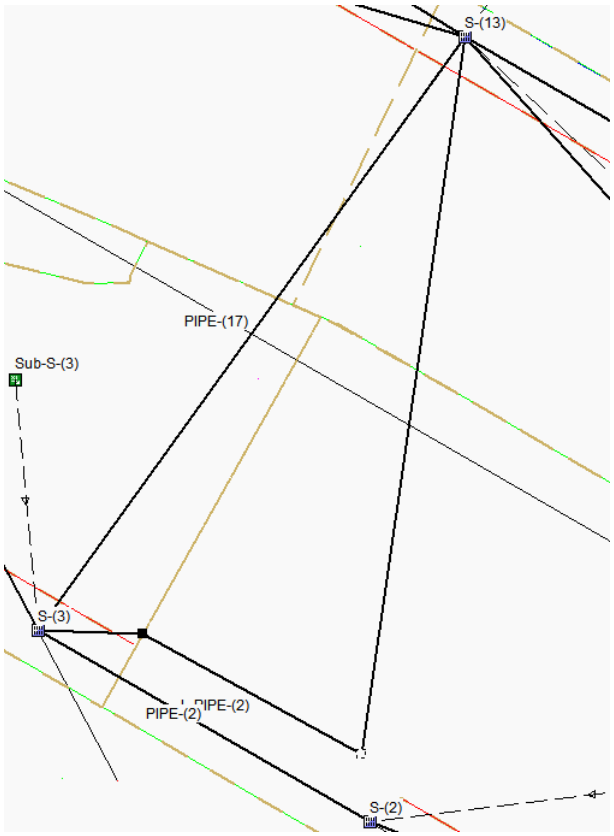
13. Click the **S-(3) node** first then click the **S-(13) node**. The bypass link is now hydraulically connecting the correct inlets but the vertices need to be adjusted to cleanup the display.



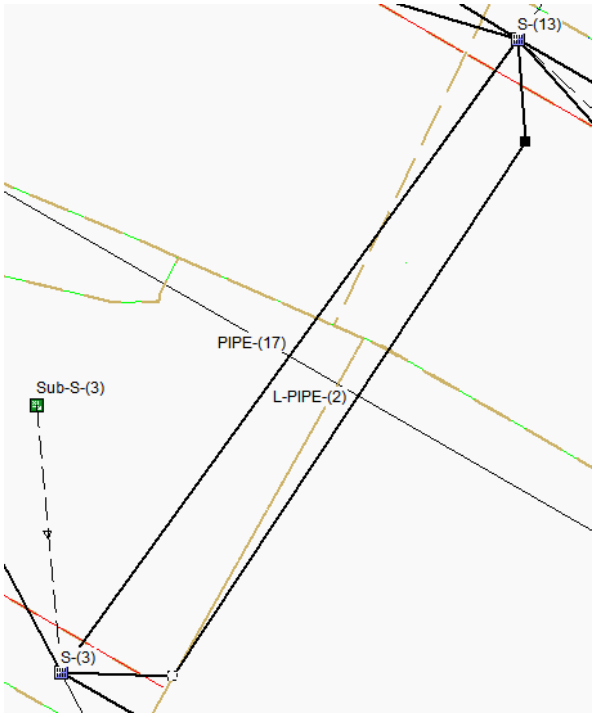
14. Right click **L-PIPE-(2)** to display the popup menu and choose **Edit Vertices**.



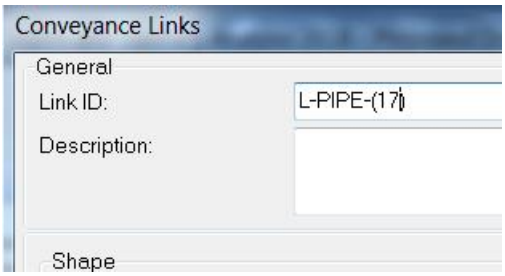
15. A *black node* appears at the current vertex. A *white node* appears at all other vertices.



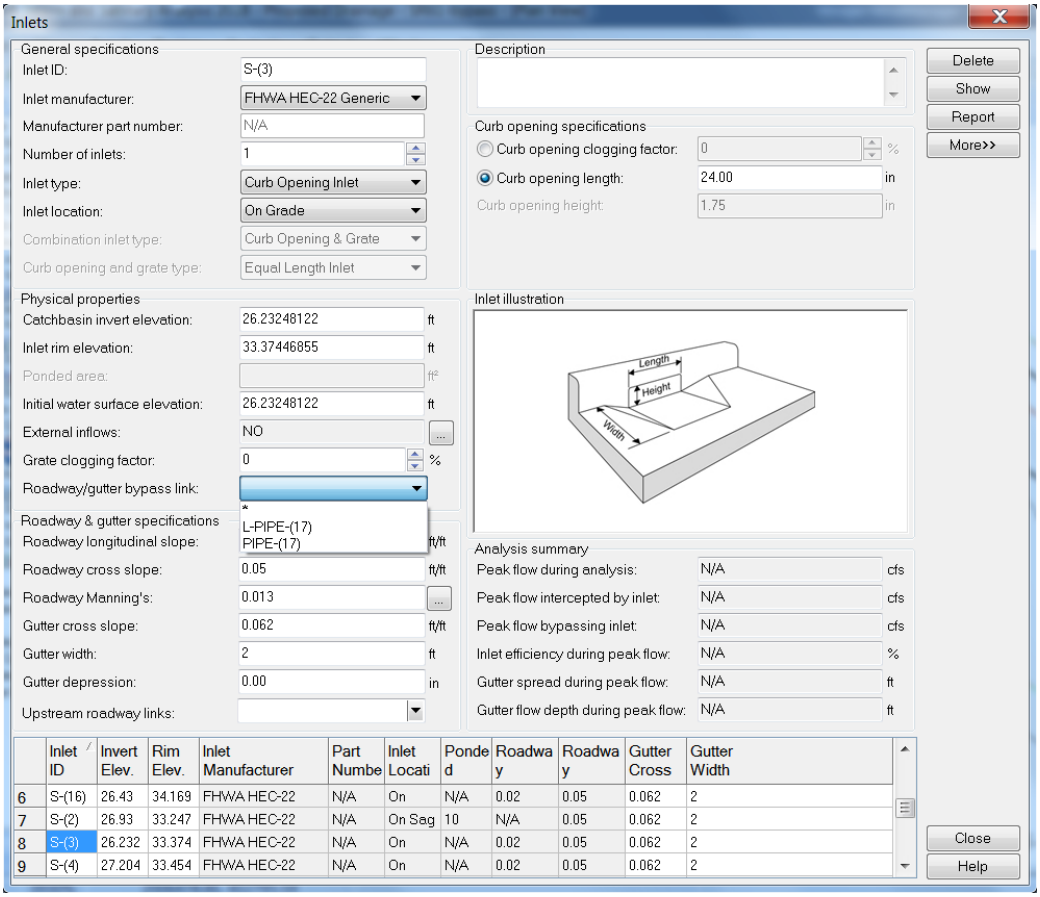
16. Click the **white vertex** nearest S-(2) and drag it toward S-(13) and place the vertex so the bypass geometry resembles other bypass links. Press **ESC** to finish the command.



17. **Rename** the *bypass link L-PIPE-(2)* to **L-PIPE-(17)**. Select the bypass link. Double click the node to open the Conveyance Links dialog and edit the *Link ID*. Click the **Close** button.



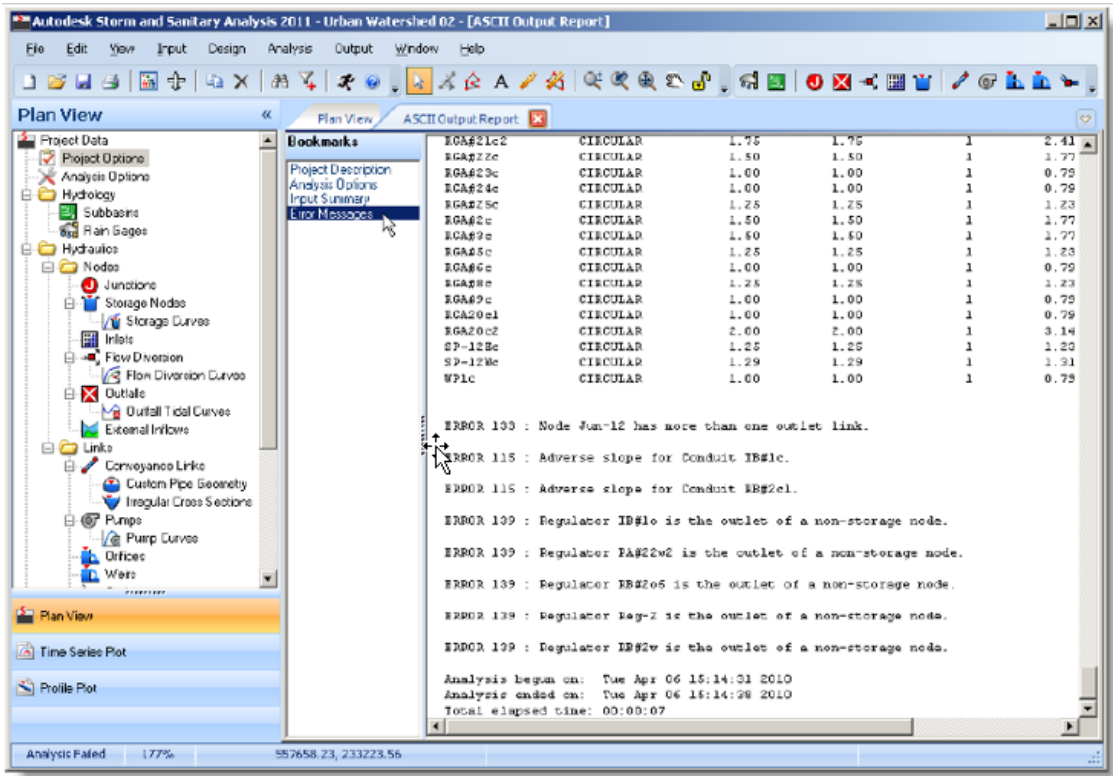
18. Since you renamed the bypass link the previous name is still assigned to S-(3). Double click **S-(3) node** to open the Inlets dialog. Click the *Roadway/gutter bypass* link drop-down list and select **L-PIPE-(17)**. Click the **Close** button. You will finish the analysis in Exercise 7.3.



TYPICAL ERRORS TO RESOLVE.

- Add Missing Bypass Links
- Correcting Flow Direction Errors
- Correcting Adverse Slopes

An ASCII Output Report is available after each analysis, as shown in the following figure. Select OUTPUT ASCII OUTPUT REPORT or click the ASCII OUTPUT REPORT icon from the Output toolbar to view the report of the analysis model run.

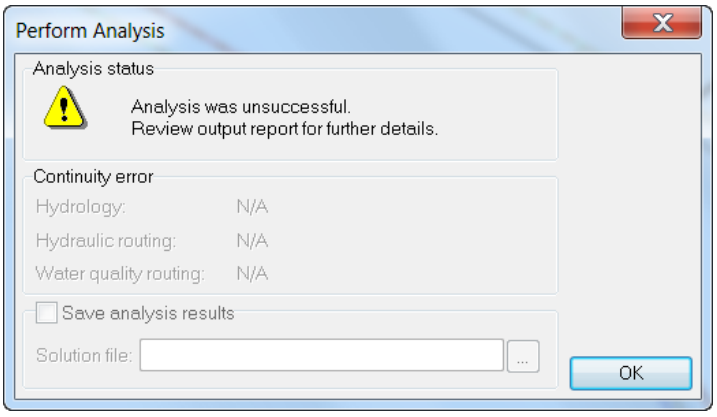


The ASCII Output Report will include an error statement, code and description of the problem encountered. For Example: ERROR 138 – JUN-32 has initial depth greater then maximum depth.

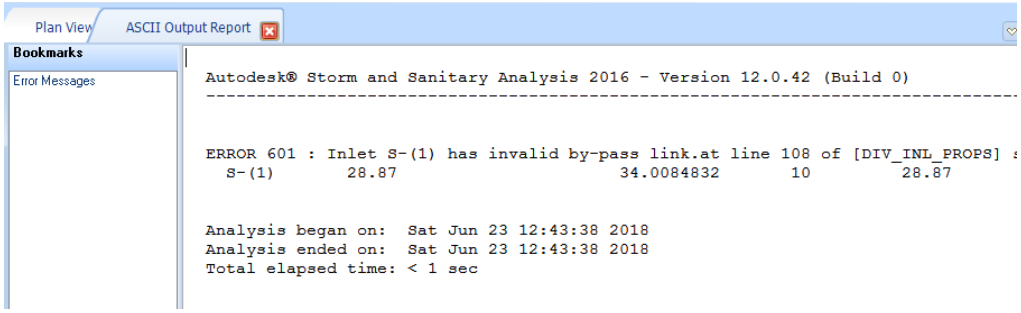
Note See Page 96 Chapter 4 Network Analysis in the SSA Help file to find a list of the Error Codes with an explanation of each one.

ADD MISSING BYPASS LINK

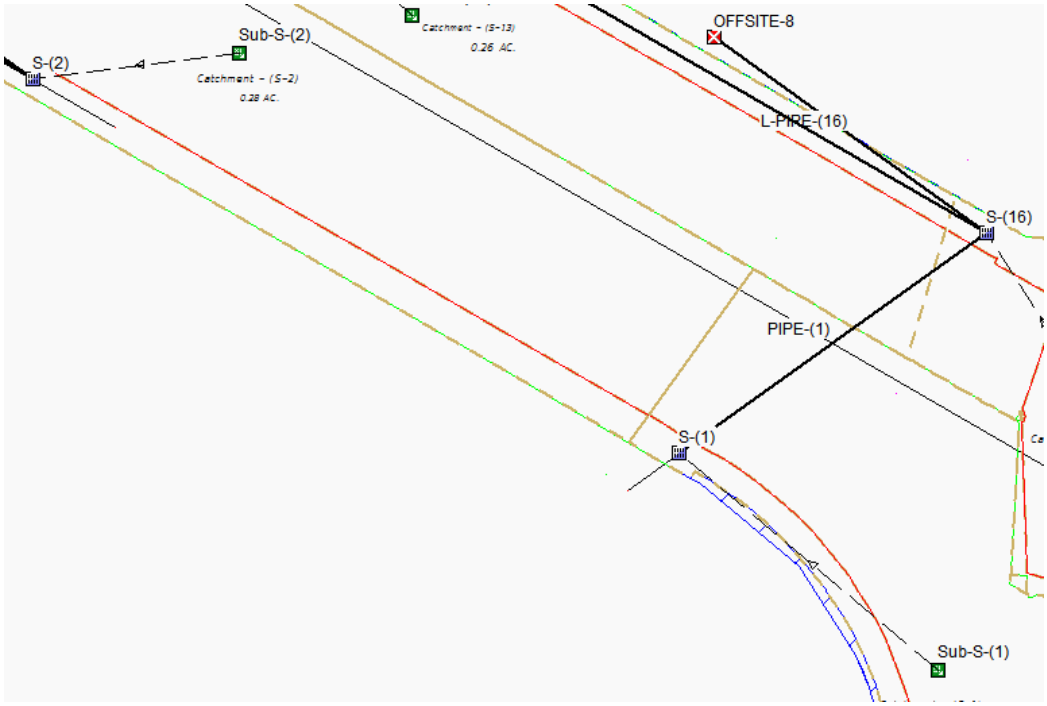
This is an example where I intentionally deleted the by-pass link then ran the Analysis to generate the error. This example has no steps for the user to perform. It is intended to illustrate a situation which you will encounter and need to be aware of its remedy.



When you get a message **Analysis was unsuccessful** click **OK** and review the output report displayed on the *ASCII Output Report* tab.

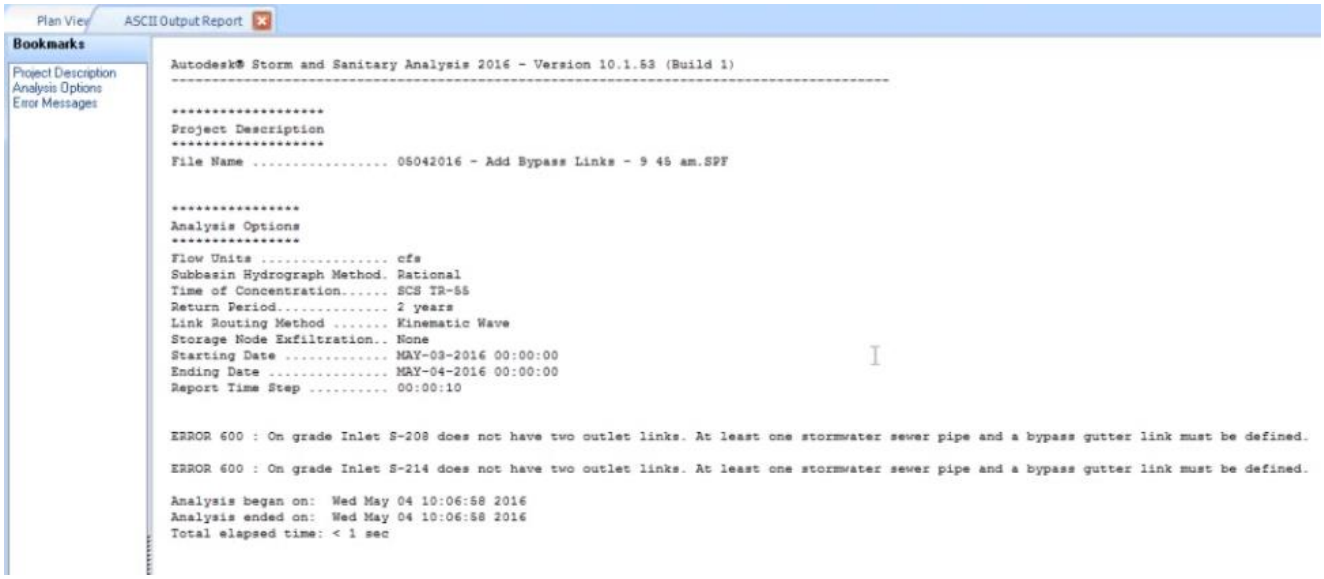


The error shown above indicates the by-pass link specified for inlet S-(1) is not valid.



To troubleshoot this error, pan and zoom to Inlet S-(1) on the *Plan View* tab. Pipe-(1) connects S-(1) to S-(16) but there is no bypass link to connect S-(1) to S-(2). You have an example in a previous section showing you how to create a bypass link.

Below is another report showing a more verbose explanation of Error 600.

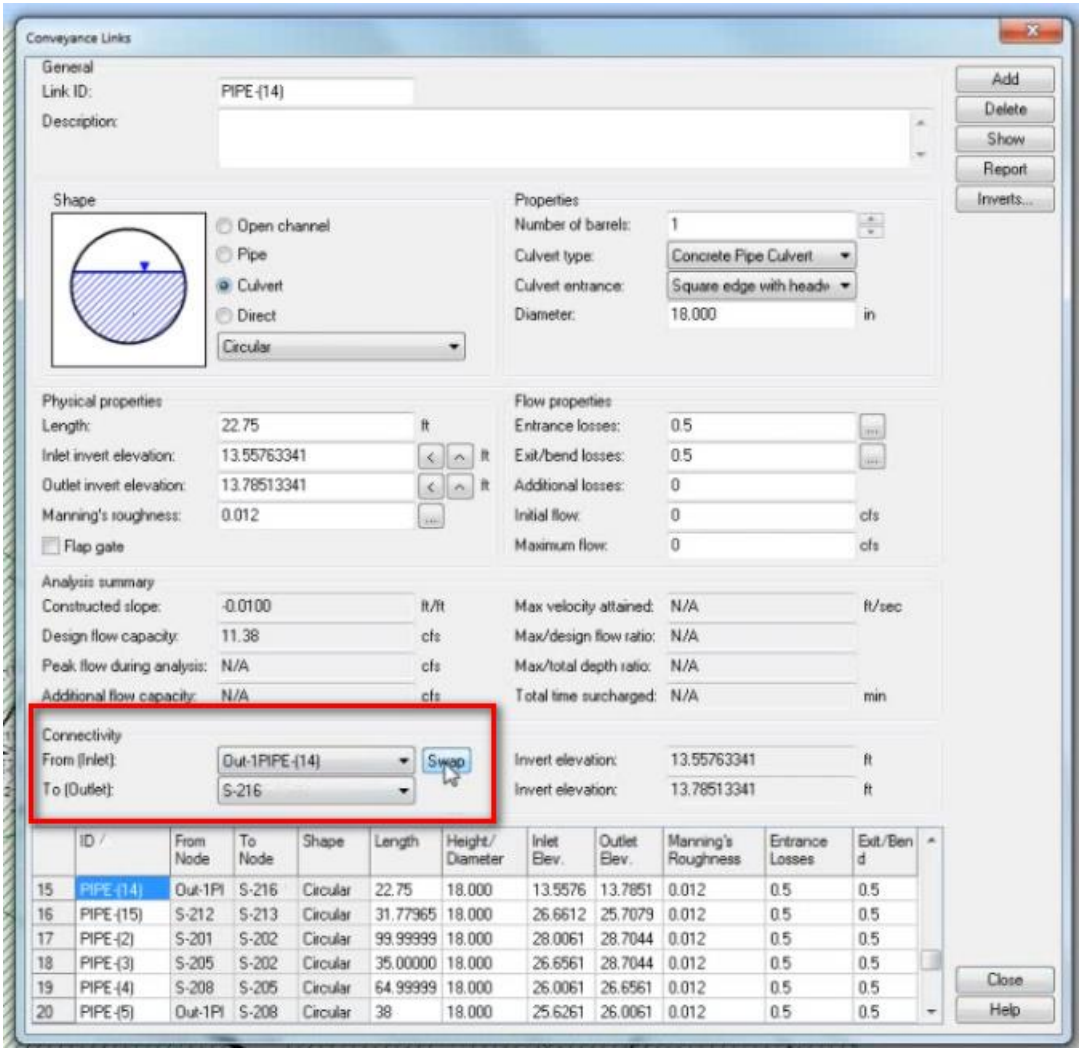


Note ERROR 600 : On grade Inlet S-208 does not have two outlet links. At least one stormwater sewer pipe and one gutter bypass link must be defined.

It is easy to view the errors within the ASCII Output Report and understand what needs to be done next. In this case we need to draw a bypass link and connect it to the upstream inlet. Correcting errors is an iterative process but produces the necessary configuration to complete the analysis without errors.

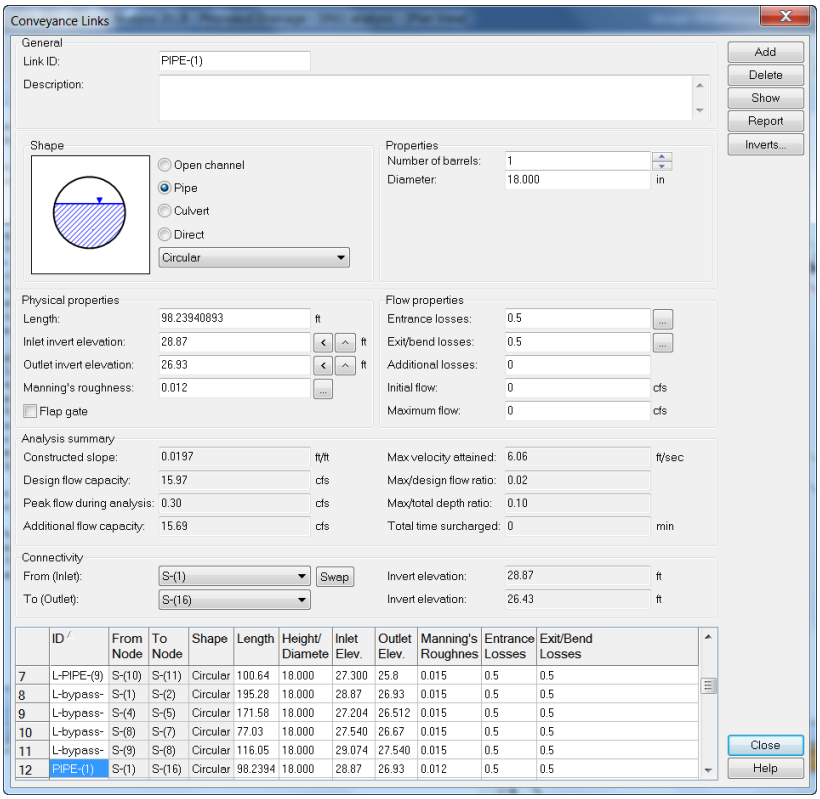
CORRECTING FLOW DIRECTION ERRORS

Swap the Upstream and Downstream nodes to correct flow direction errors.

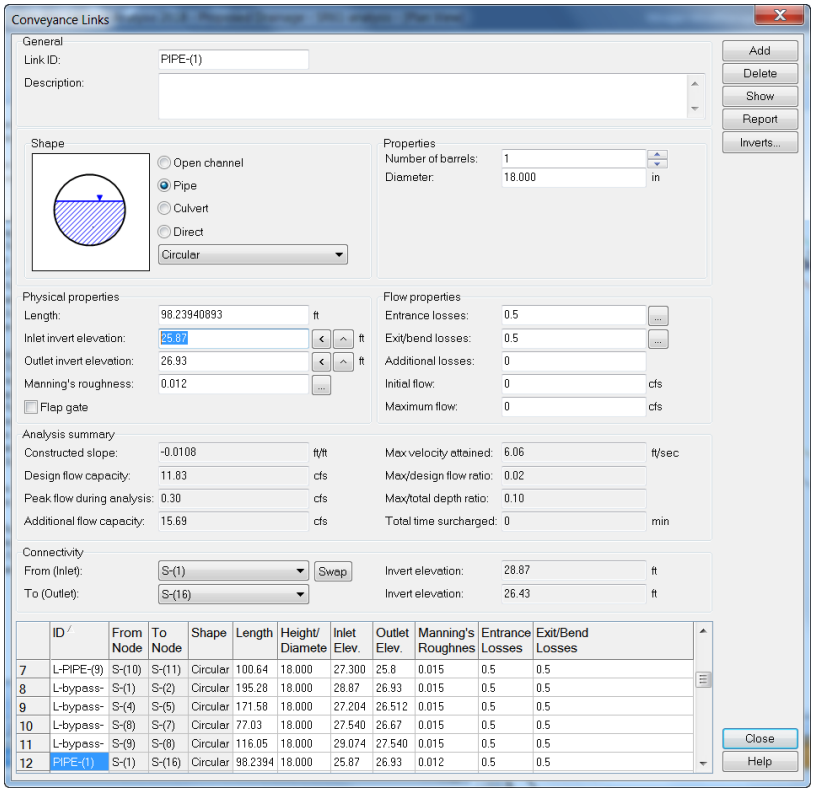


CORRECTING ADVERSE SLOPES

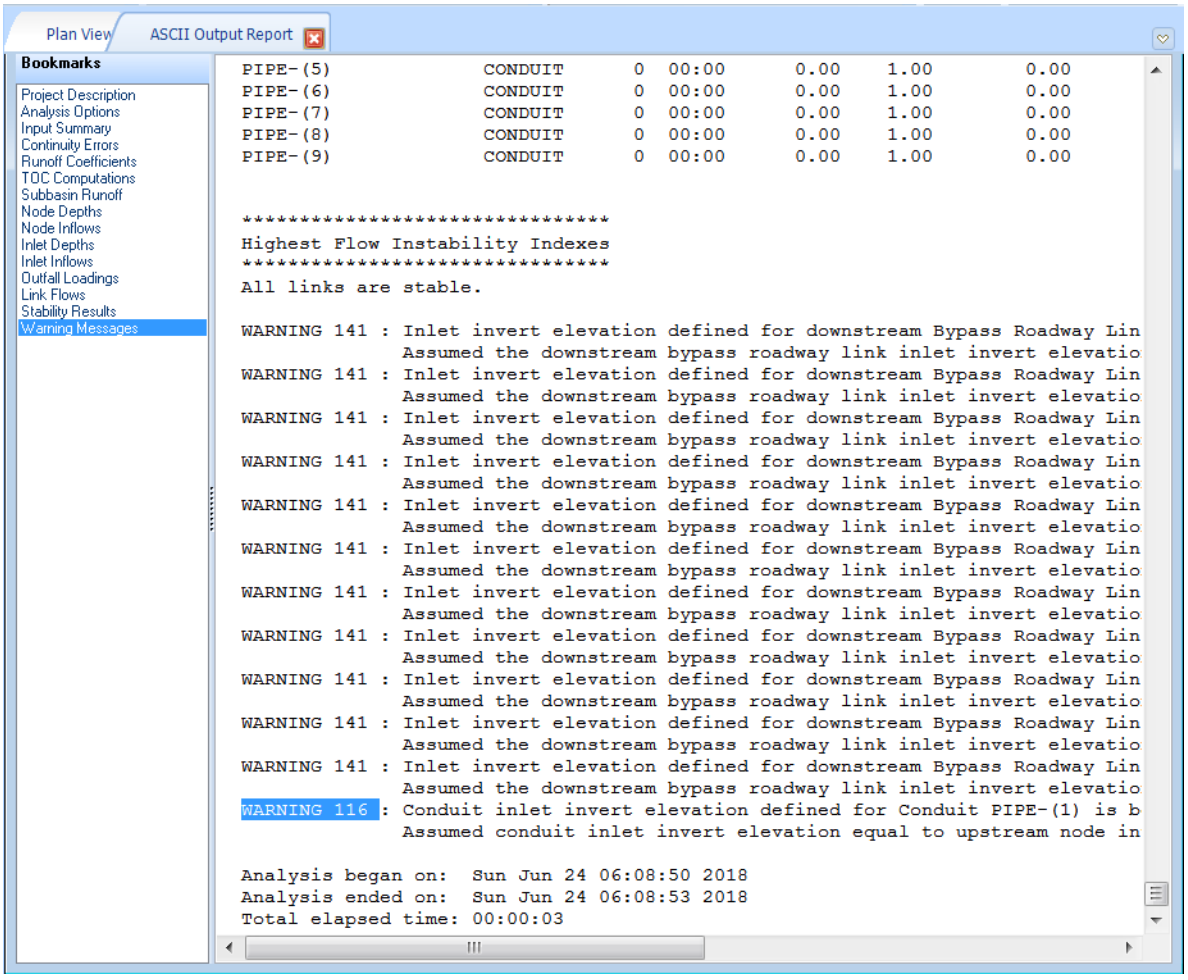
The Conveyance Links dialog shows the Inlet invert elevation at 28.87.



In this example I edited the inlet invert elevation to lower it below the Outlet invert elevation (now at 25.87).



When the Analysis is run you can see the warning in the ASCII Output Report.



Note WARNING 116 : Conduit inlet invert elevation defined for Conduit PIPE-(1) is below upstream node invert elevation. Assumed conduit inlet invert elevation equal to upstream node invert elevation.

The warning indicates a situation where the upstream invert elevation is lower than the downstream invert elevation. To correct the problem open the Conveyance Links dialog for PIPE-(1) and edit the inverts to produce a positive slope value.

The profile view can also help you choose an invert elevation that works without causing further errors upstream or downstream. Profile Views will be covered in a later chapter.

Note SSA Help File page 227 Chapter7 Network Element Data, Channel, Pipe, & Culvert Links

ADVERSE SLOPE

For both Steady Flow or Kinematic Wave routing, all channels, pipes, and culverts must have positive slopes (i.e., the outlet invert must be below the inlet invert). The software will check for this condition when it performs the analysis and will report this as a problem.

If you have incorrectly defined the inlet and outlet nodes for a link, you can easily correct this. Select the reversed link in the Conveyance Links dialog box and click the Swap button. The software will reverse the direction of the link so that the outlet node becomes the inlet node and the inlet node becomes the outlet node. Alternatively, select the link from the Plan View, right-click to display the context menu, and select REVERSE DIRECTION.

However, if a channel, pipe, or culvert does have an adverse slope (i.e., negative slope), where the outlet elevation is higher than the inlet elevation, reversing the direction of the link will not solve this issue. Networks with adverse sloped links can only be analyzed with Hydrodynamic routing.

SURCHARGED PIPES AND OSCILLATIONS

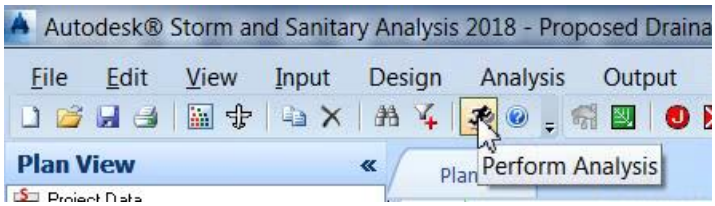
Note SSA Help File pg 228 Chapter 7 Network Element Data

If the upstream end of a pipe surcharges, then a head adjustment is performed by the routing engine at the upstream connecting node. Because this head adjustment is an approximation, the computed head at the upstream node can sometimes have a tendency to "bounce" up and down (or oscillate) when the pipe first surcharges. This bouncing can sometimes cause the analysis results to become unstable; therefore, a transition function is automatically used to smooth the changeover of head computations.

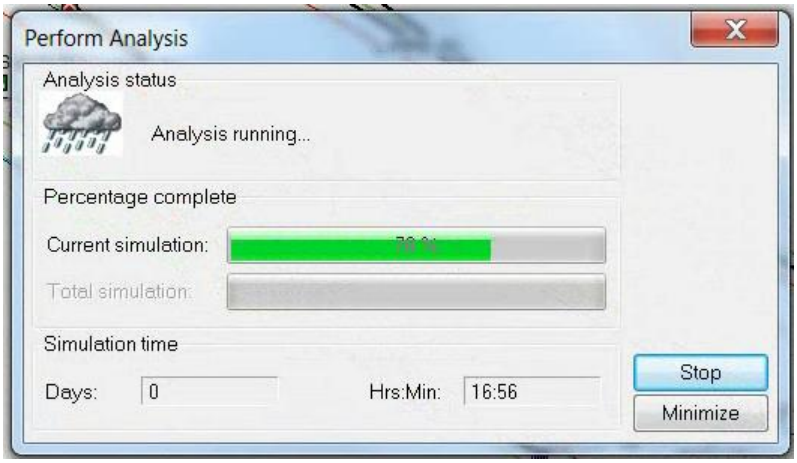
If you find that the oscillation continues at the upstream node while the connected downstream pipe is surcharged, then define a PONDED AREA at the downstream connecting node. This can sometimes eliminate the oscillation at the upstream node and produce a more stable model.

Exercise 7.3 Re-Run Analysis

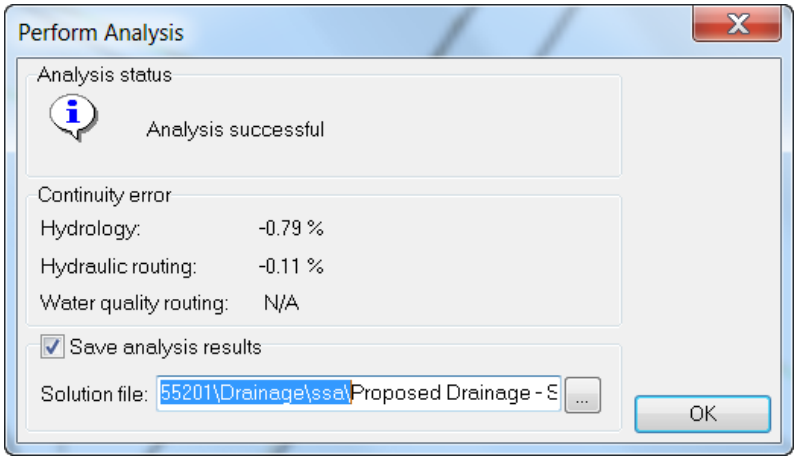
1. From the *Standard* toolbar, Click the **Perform Analysis** command.



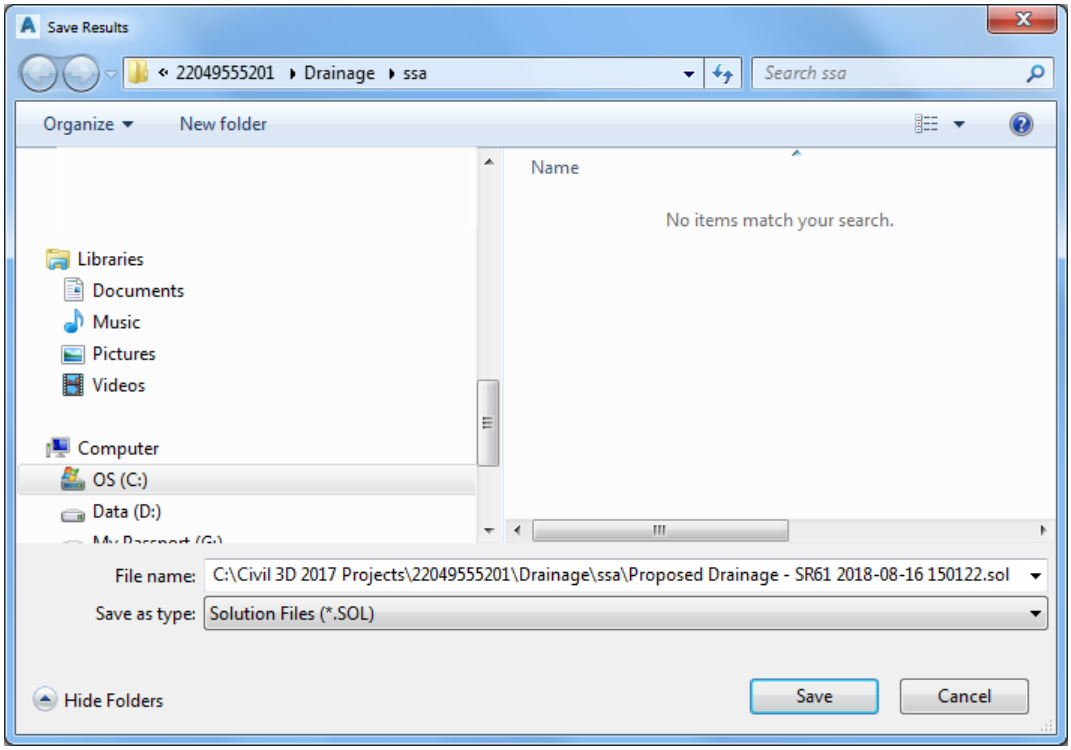
2. If there are no errors the green progress bars appear in the Perform Analysis dialog.



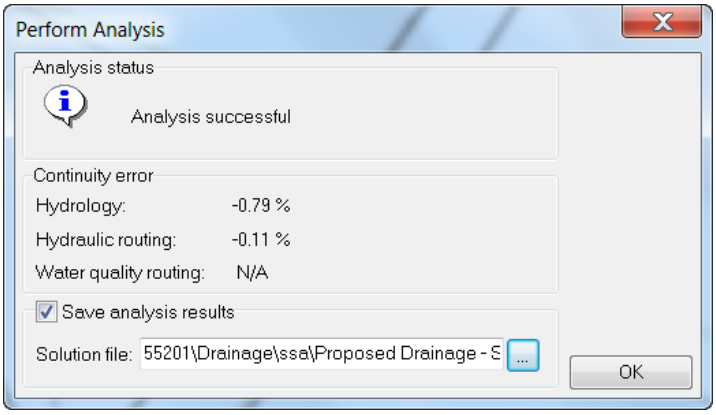
3. When the analysis is complete the Analysis status will display *Analysis successful* message. Click the checkbox **Save analysis results**.



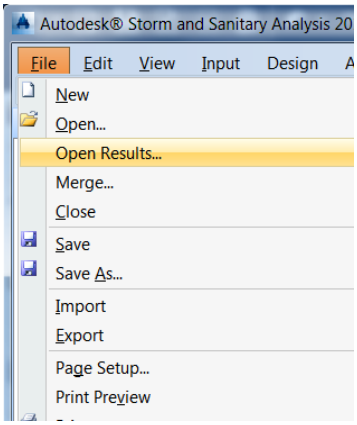
4. Browse to the folder *C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa* by clicking the **ellipsis** button. Keep the default name and click the **Save** button to return to the Perform Analysis dialog.



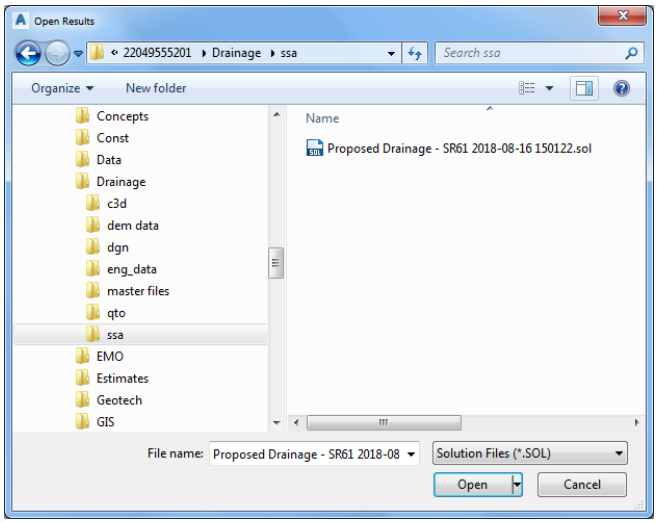
5. The *Solution* file allows you to **Save** the analysis results to a **.sol** file for later recall. Click **OK**.



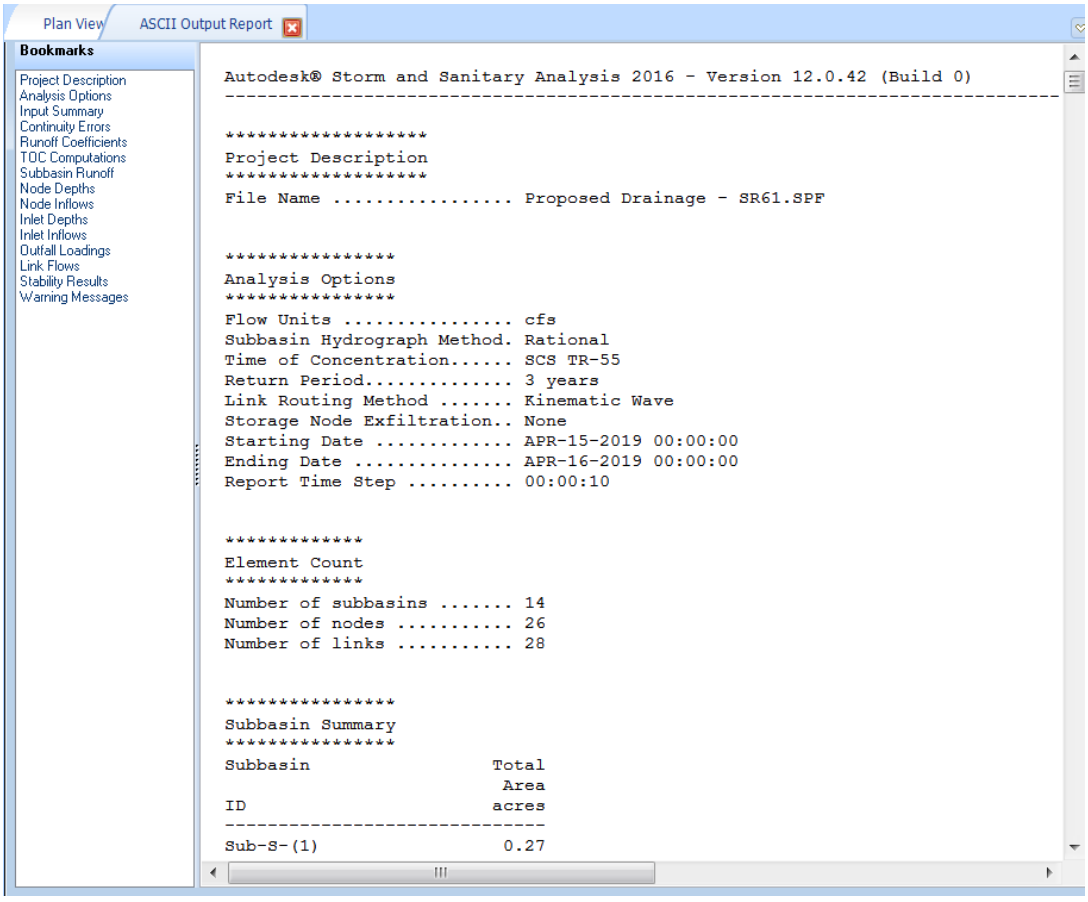
6. To review the *Solution file* click the **File** pull-down menu and select the **Open Results**... command.



7. Select the file you just saved. Click the **Open** button.

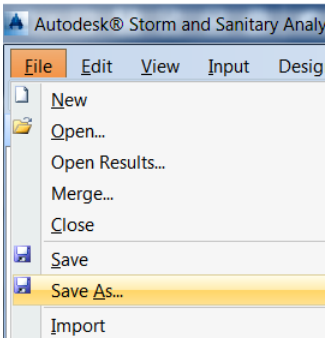


8. The output report on the *ASCII Output Report* tab displays the contents of the .**sol** file. The bookmarks on the left provide a convenient way to navigate to your area of interests within the report.



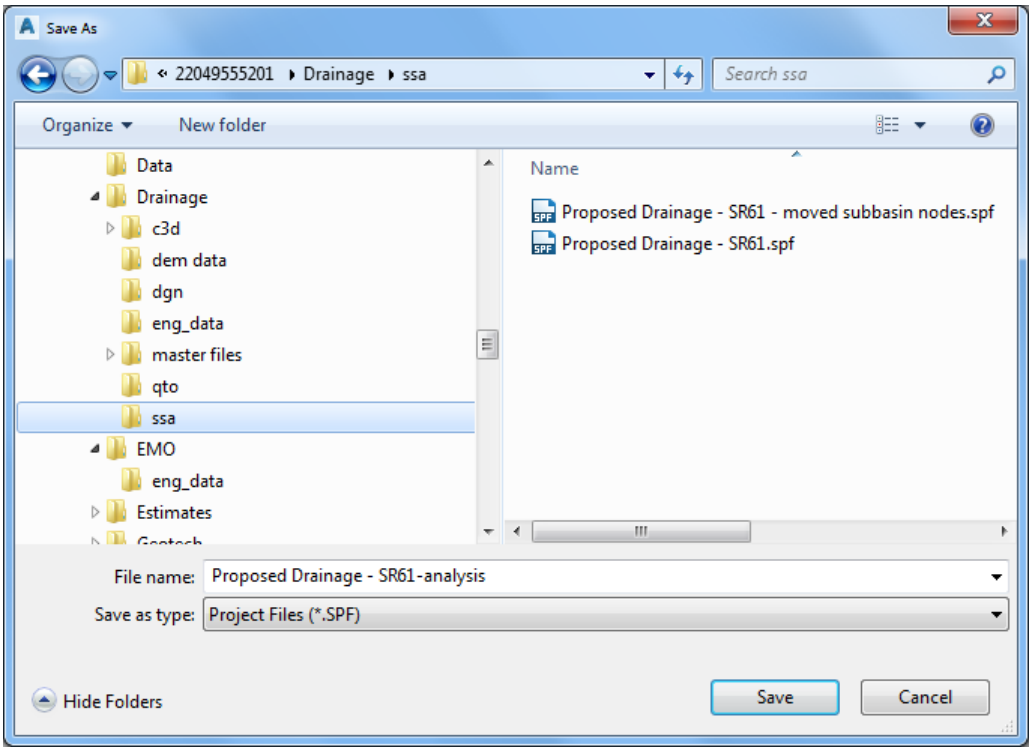
Note If the ASCII Output Report tab does not display, from the Output Toolbar Click the ASCII Analysis Report command.

9. Click the **File** drop-down menu and select **Save As**.



10. Save the file using the following name and path:

    C:\Civil 3D Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61-analysis.spf



11. **Close** SSA.

8 SHOW PROFILE PLOT

DESCRIPTION

This chapter demonstrates the process of showing a profile plot

OBJECTIVES

In this chapter, you will learn about:

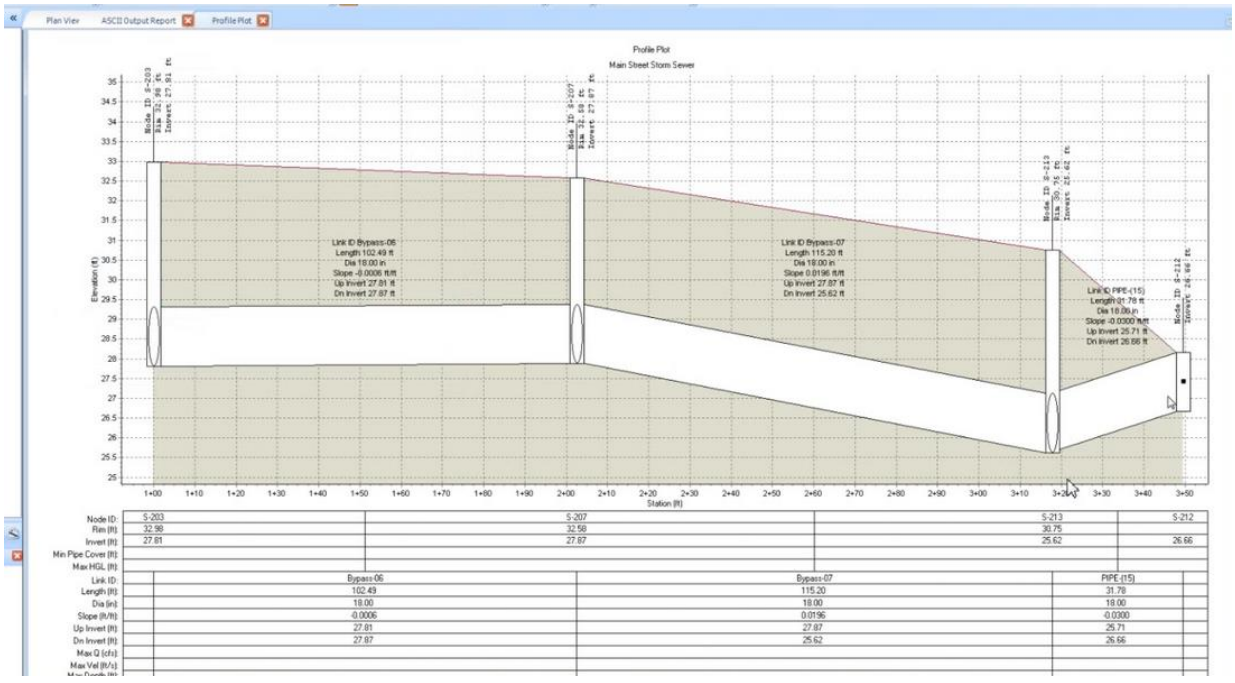
- Profile Plot Path Configuration
- Show Plot
- Displaying Water Levels
- Animating the analysis
- Creating an Excel Table Report

CHAPTER SETUP

Run the Chapter 8 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX Projects** folder.

Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

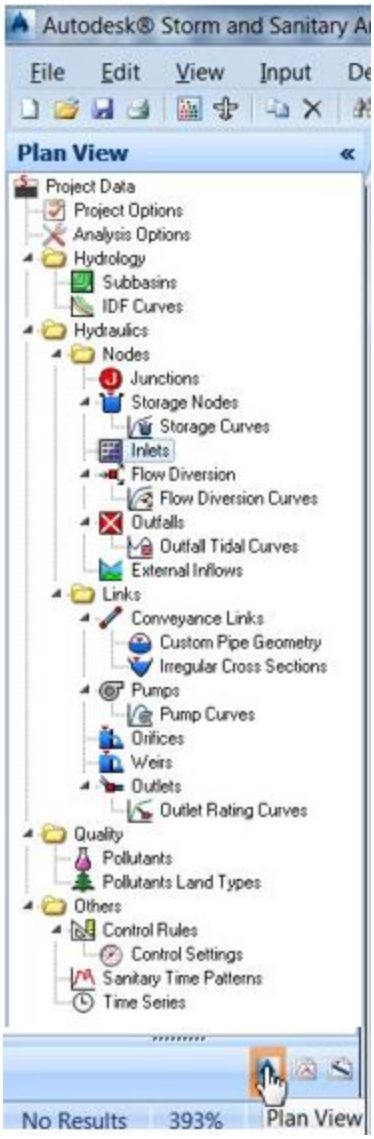
The Profile Plot makes is it easy to see if pipes are sloped in the wrong direction. You can edit the pipes and nodes by double clicking and viewing the corresponding dialog box to edit values then return to the Profile Plot to see the updated results.



DATA TREE TOOLBAR

The Data Tree has 3 buttons in a toolbar at the bottom. Each button changes the display of the Data Tree.

The left button the **Plan View** as displayed below.



The middle button is the **Time Series Plot.**

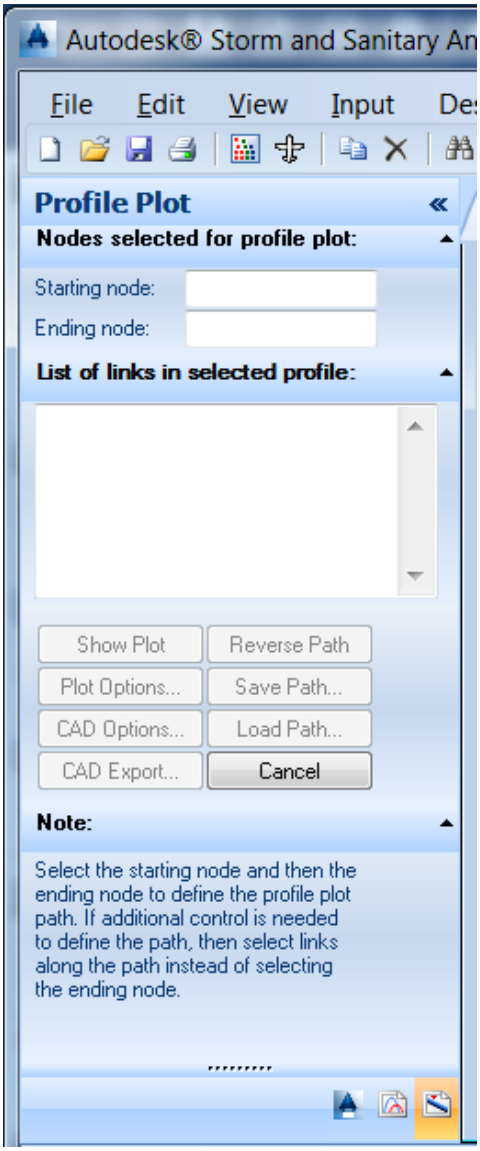
The right button is the **Profile Plot**.

Exercise 8.1 Profile Plot Path Configuration

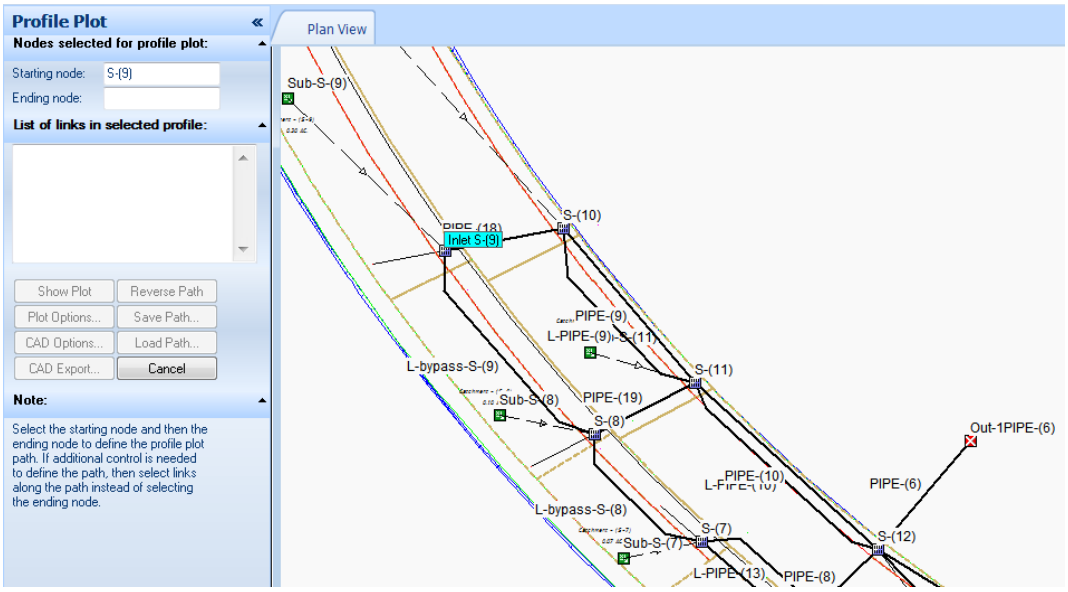
Use the Profile Plot window (a docked dialog panel) to configure the Profile Plot path to display the links and structures starting at S-(9) and connecting S-(10), S-(11), S-(12), and ending at outlet Out-1PIPE-(6).

➢ *Setup Profile Plot*

1. Start SSA and open the project file *C:\Civil 3D Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61-analysis.spf.*
2. Click the **Profile Plot** button to display the Profile Plot window.

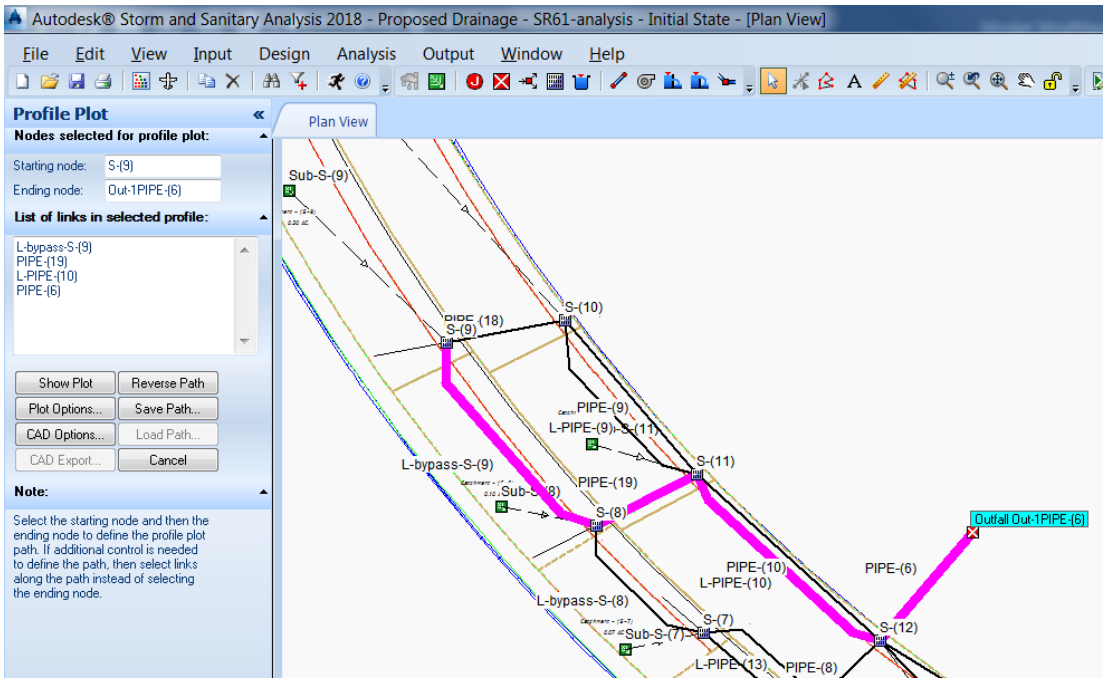


3. At the top in the *Nodes selected for profile plot* area, specify the *Starting node* and *Ending node* by first clicking the blank field next to the *Starting node*. Your cursor is now focused inside the field allowing you to select a structure in the *Plan View* and have that *Structure ID* placed in the *Starting node* field. Select **Inlet S-(9)**. The Structure ID S-(9) displays in the *Starting node* field.

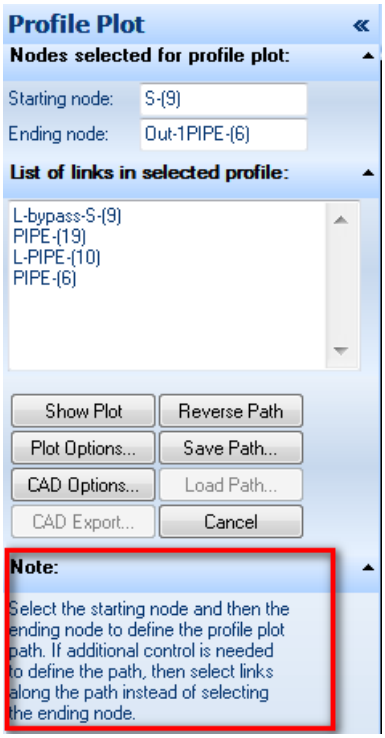


4. Select the blank field next to *Ending node*. Select the **Outfall Out-1PIPE-(6)**. The *outfall ID Outfall Out-1PIPE-(6)* now displays in the *Ending node* field.

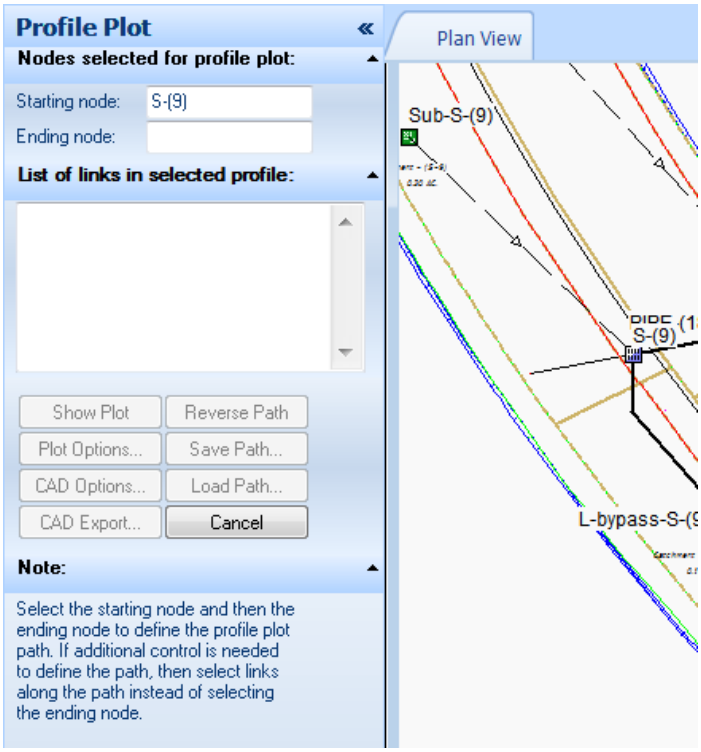
Note In the Plan View notice the elements connecting S-(9) to Out-1PIPE-(6) are highlighted in magenta. In the List of links in selected profile you see the list of conveyance link IDs matching what is highlighted in the Plan View. The software found a series of links connected the starting and ending structures but that is not connecting the desired structures.



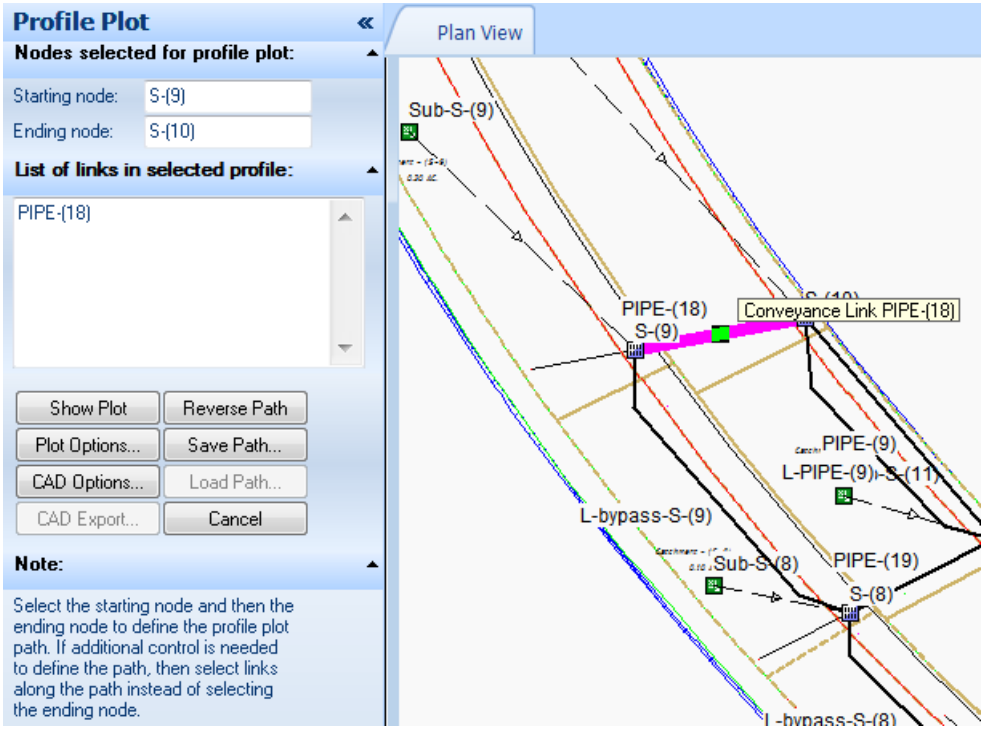
5. Read the *Note* in the Profile Plot. "…If additional control is needed to define the path, then select links along the path instead of selecting the ending node." There are no editing options built into this window. To start over you must click the **Cancel** button, then click the **Profile Plot** button.



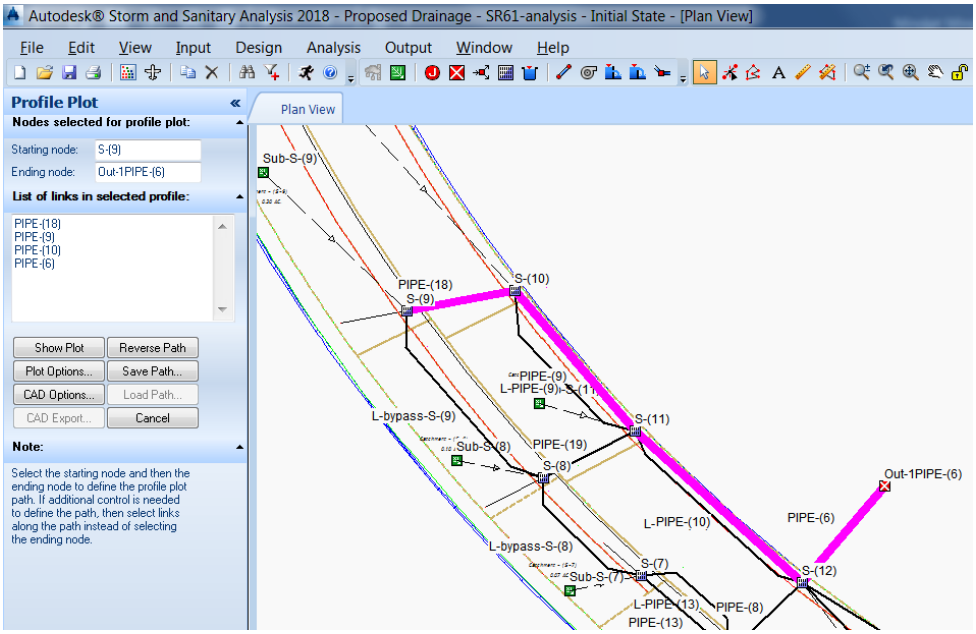
6. Click inside the blank field next to the *Starting node*. Select inlet **S-(9)**.



7. Click the *Conveyance Link* **PIPE-(18)** to begin defining the path to the outfall.

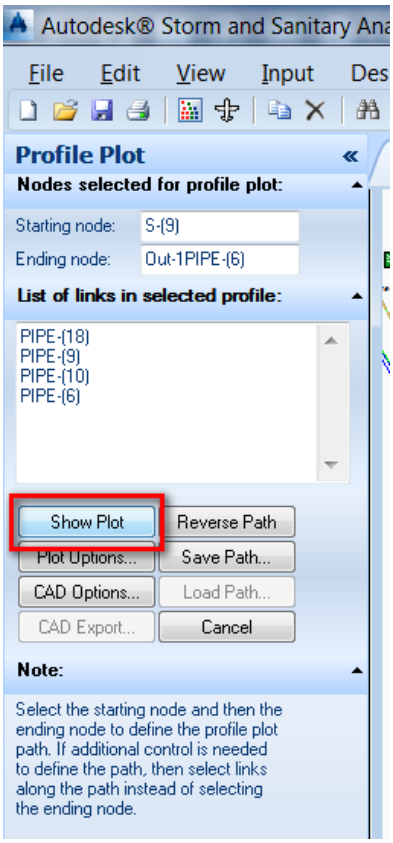


8. Continue by picking **PIPE-(9)**, **PIPE-(10)**, and **PIPE-(6).**

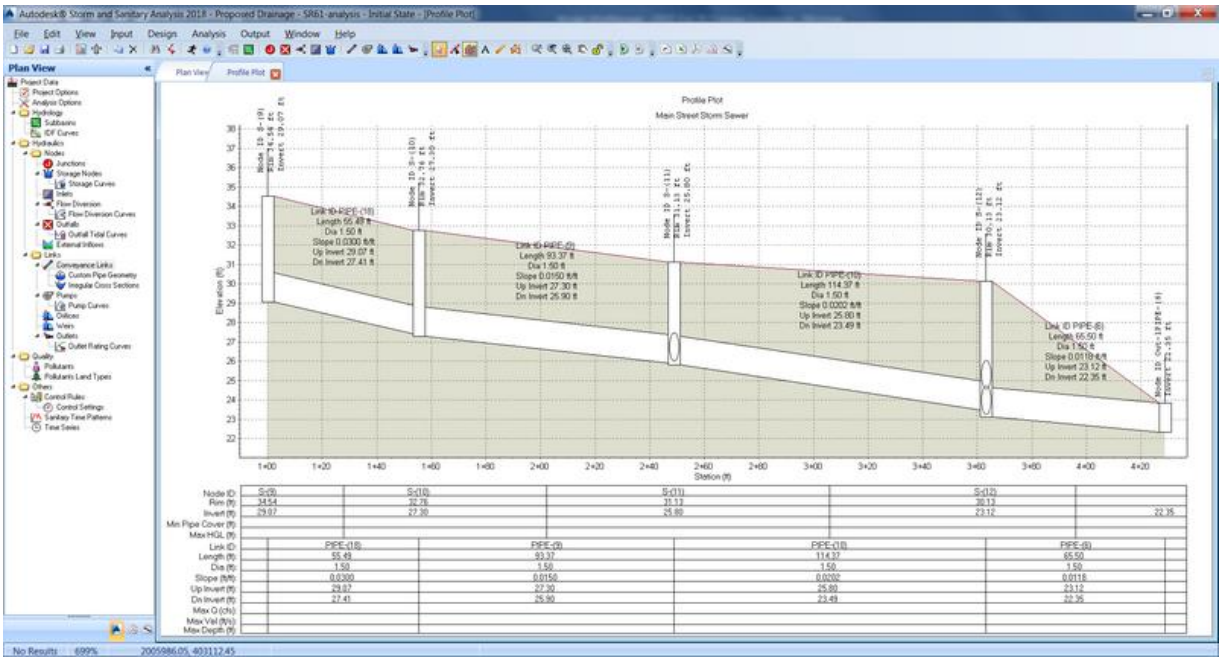


➢ *Show Plot*

1. Click the **Show Plot** button.



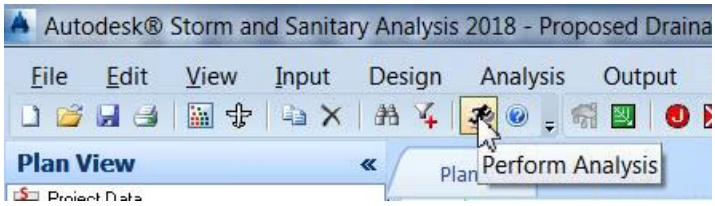
2. The *profile* **from S-(9) to outfall Out-1PIPE-(6)** displays.in the *Profile Plot* tab.



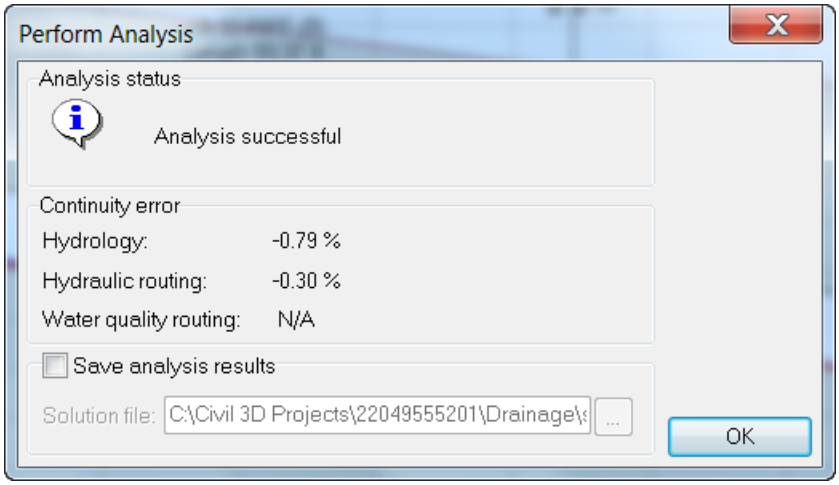
Exercise 8.2 Display Water Levels in Pipes

The water levels and EGL and HGL plots are displayed in the Profile Plot after running the Analysis.

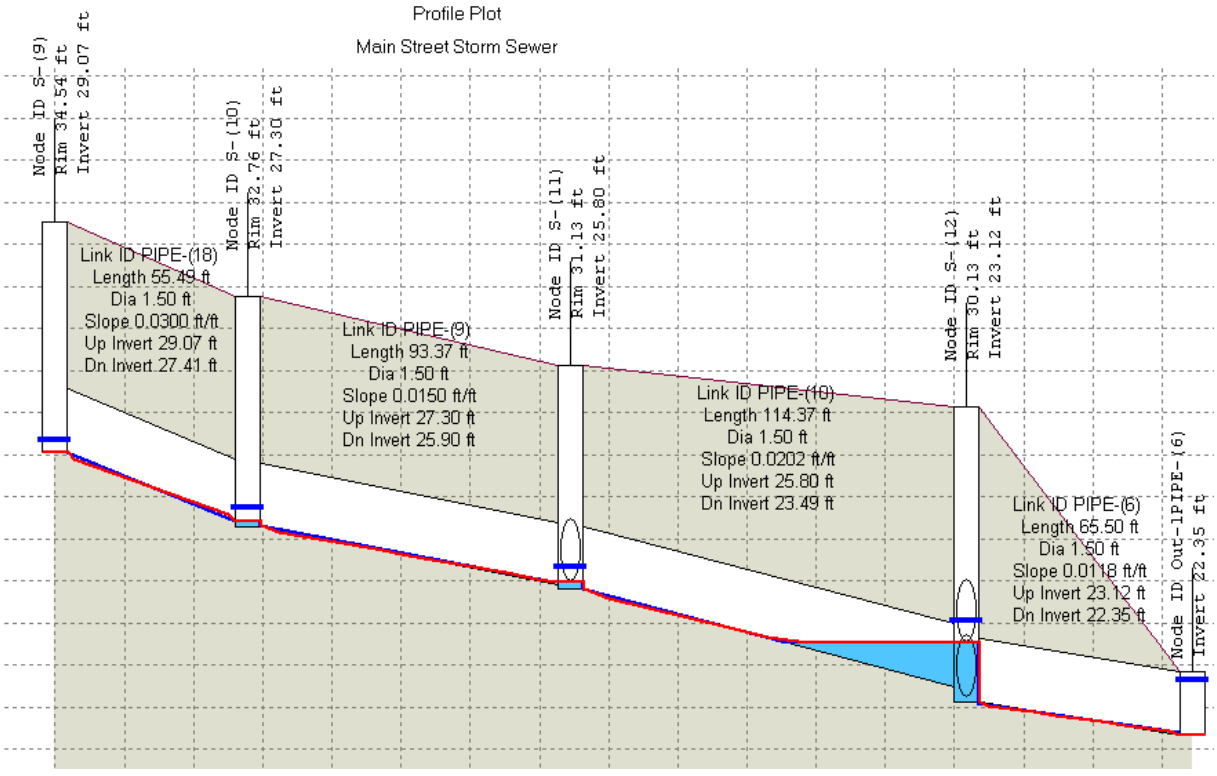
1. From the *Standard* toolbar, select the **Perform Analysis** command.



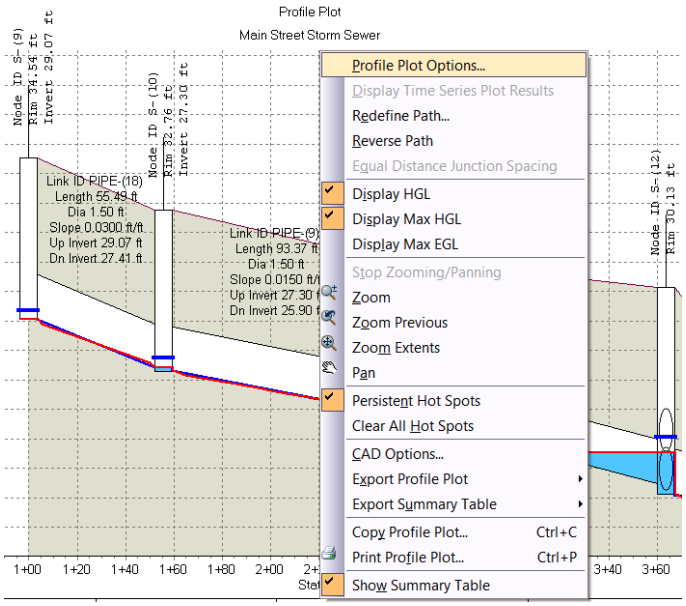
2. Clear the **Save analysis results** check box, and then Click **OK**.



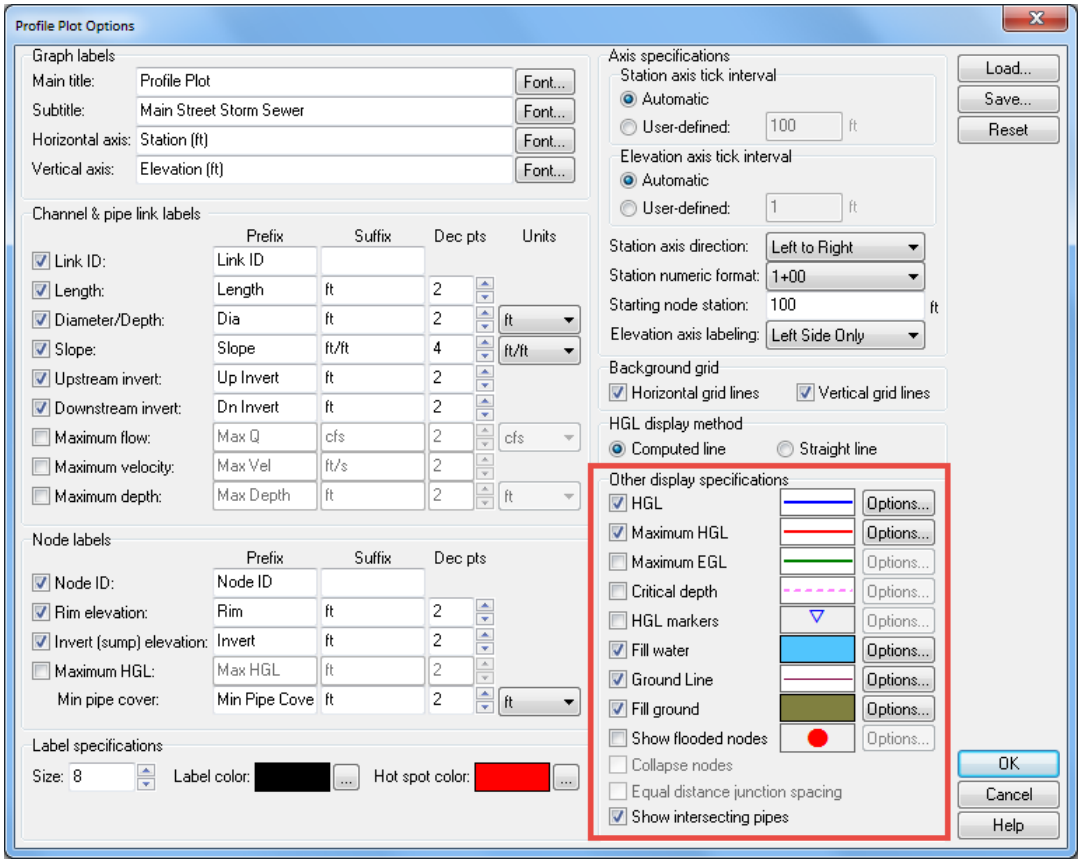
3. The *water level, HGL,* and *EGL* displays after completing the *Analysis* command. You can roll the wheel on the mouse to change the horizontal scale of the Profile Plot.



4. Right click in the **Profile Plot area** to display the *context* menu. Select the **Profile Plot Options** command.



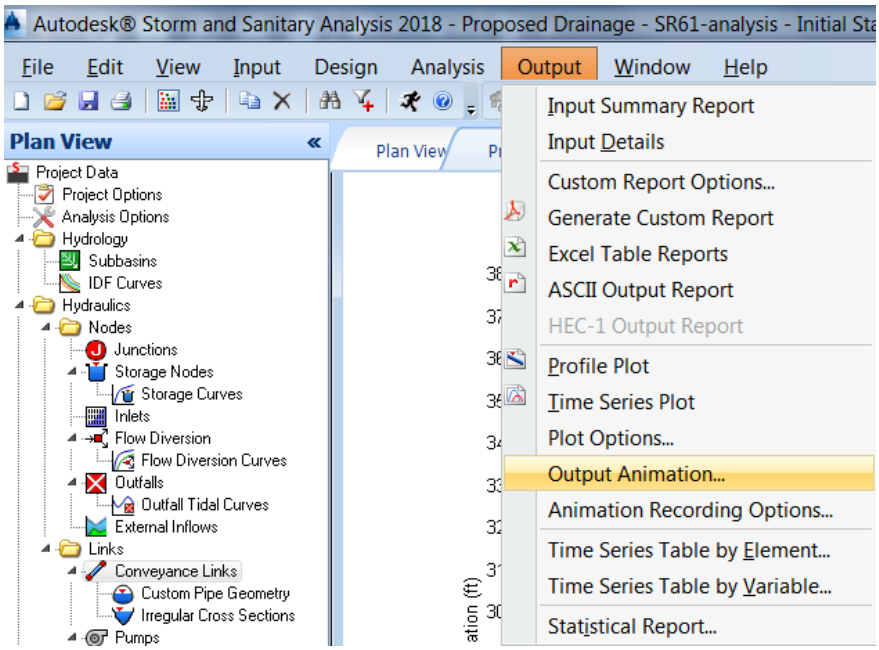
5. In the *Other display specifications* area of the Profile Plot Options dialog you can see the colors specified for *HGL*, *Maximum HGL*, and *Fill water*. The check box next to each element *enables* or *disables* display in the *Profile Plot*. Click **OK** to close the Profile Options dialog.



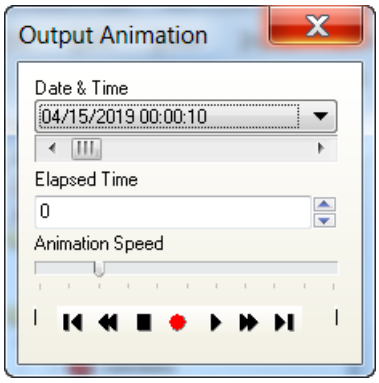
Exercise 8.3 Animate Storm Event in the Profile Plot

Note For additional information see the SSA Help file and refer to page 118 Chapter 5 Display analysis Results Output Animation.

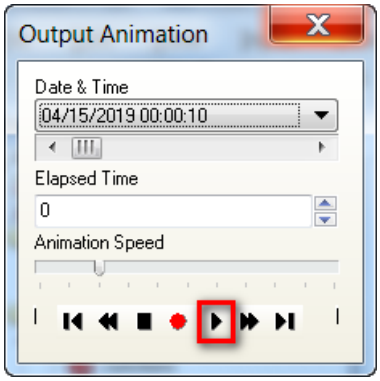
1. Click the **Output** drop-down menu and select the **Output Animation** command.

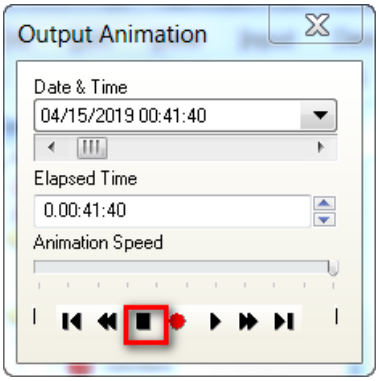


2. The animation controls are on the Output Animation dialog box. The *Date & Time* are for the current analysis run. Slide the animation speed all the way to the right.

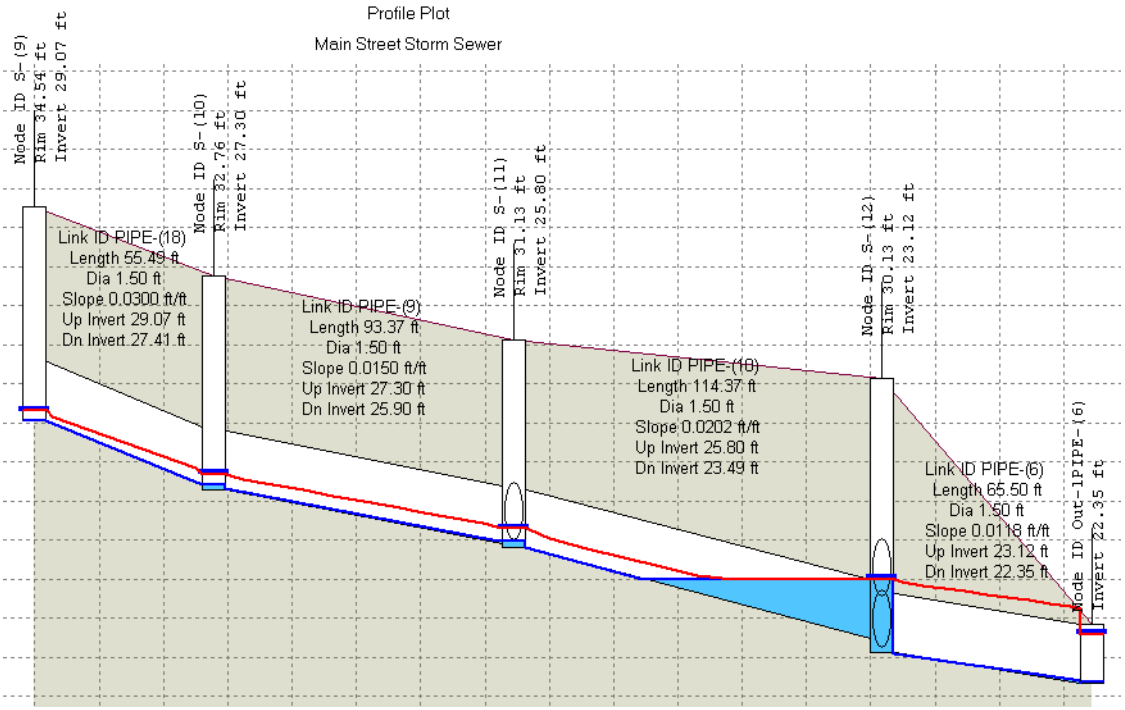


3. Click the **Play** button to start the animation. Click the **Stop** button after the elapsed time has reached 40 minutes.

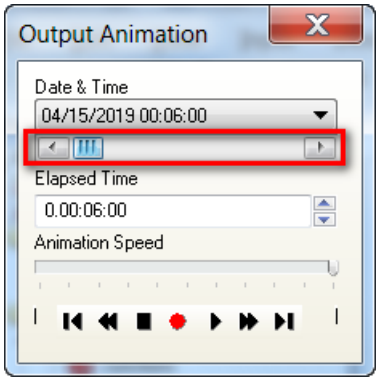




4. As the time progressed on the Output Animation dialog, the *Profile Plot* displayed changes in the HGL, Maximum HGL, and water level.



5. Using the *Slider Control* and the *Left* and *Right Arrow* buttons at either end are a convenient way to navigate to the desired date and time. Click the **X** to close the Output Animation dialog.



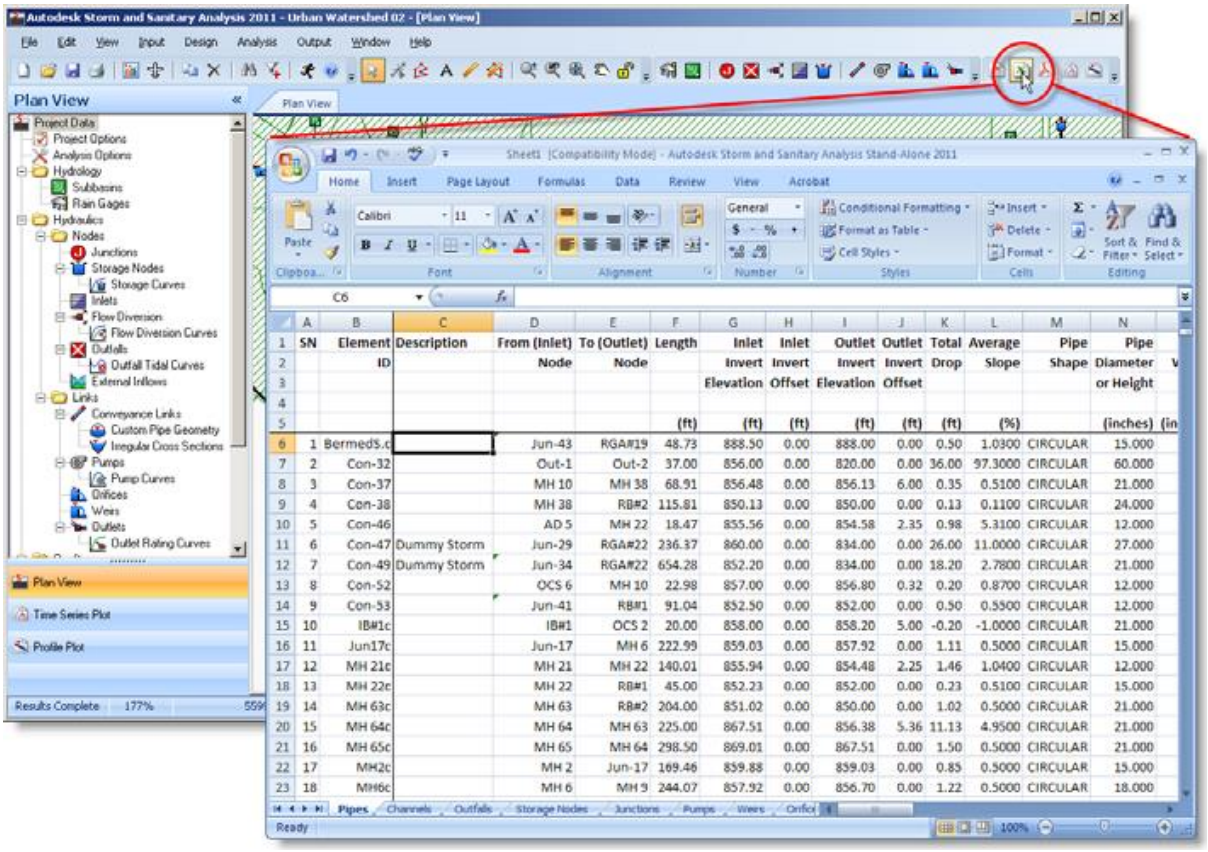
6. **Close** SSA and **Save** changes to the project.

EXCEL TABLE REPORTS

Note Run the Chapter 11 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX** Projects folder.

Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

As shown in the following figure, from the Output toolbar, select OUTPUT EXCEL TABLE REPORTS or click the EXCEL TABLE REPORTS icon from the Output toolbar. The software will automatically startup Excel and display the output reports.



	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	SN	Element	Description	From (Inlet)	To (Outlet)	Length	Inlet	Inlet	Outlet	Outlet	Total	Average	Pipe	Pipe
2		ID		Node	Node		Invert	Invert	Invert	Invert	Drop	Slope	Shape	Diameter
3							Elevation	Offset	Elevation	Offset				or Height
4														
5						(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(%)		(inches)
6	1	Bermed5.c		Jun-43	RGA#19	48.73	888.50	0.00	888.00	0.00	0.50	1.0300	CIRCULAR	15.000
7	2	Con-32		Out-1	Out-2	37.00	856.00	0.00	820.00	0.00	36.00	97.3000	CIRCULAR	60.000
8	3	Con-37		MH 10	MH 38	68.91	856.48	0.00	856.13	6.00	0.35	0.5100	CIRCULAR	21.000
9	4	Con-38		MH 38	RB#2	115.81	850.13	0.00	850.00	0.00	0.13	0.1100	CIRCULAR	24.000
10	5	Con-46		AD 5	MH 22	18.47	855.56	0.00	854.58	2.35	0.98	5.3100	CIRCULAR	12.000
11	6	Con-47	Dummy Storm	Jun-29	RGA#22	236.37	860.00	0.00	834.00	0.00	26.00	11.0000	CIRCULAR	27.000
12	7	Con-49	Dummy Storm	Jun-34	RGA#22	654.28	852.20	0.00	834.00	0.00	18.20	2.7800	CIRCULAR	21.000
13	8	Con-52		OCS 6	MH 10	22.98	857.00	0.00	856.80	0.32	0.20	0.8700	CIRCULAR	12.000
14	9	Con-53		Jun-41	RB#1	91.04	852.50	0.00	852.00	0.00	0.50	0.5500	CIRCULAR	12.000
15	10	IB#1c		IB#1	OCS 2	20.00	858.00	0.00	858.20	5.00	-0.20	-1.0000	CIRCULAR	21.000
16	11	Jun17c		Jun-17	MH 6	222.99	859.03	0.00	857.92	0.00	1.11	0.5000	CIRCULAR	15.000
17	12	MH 21c		MH 21	MH 22	140.01	855.94	0.00	854.48	2.25	1.46	1.0400	CIRCULAR	12.000
18	13	MH 22c		MH 22	RB#1	45.00	852.23	0.00	852.00	0.00	0.23	0.5100	CIRCULAR	15.000
19	14	MH 63c		MH 63	RB#2	204.00	851.02	0.00	850.00	0.00	1.02	0.5000	CIRCULAR	21.000
20	15	MH 64c		MH 64	MH 63	225.00	867.51	0.00	856.38	5.36	11.13	4.9500	CIRCULAR	21.000
21	16	MH 65c		MH 65	MH 64	298.50	869.01	0.00	867.51	0.00	1.50	0.5000	CIRCULAR	21.000
22	17	MH2c		MH 2	Jun-17	169.46	859.88	0.00	859.03	0.00	0.85	0.5000	CIRCULAR	15.000
23	18	MH6c		MH 6	MH 3	244.07	857.92	0.00	856.70	0.00	1.22	0.5000	CIRCULAR	18.000

Exercise 8.4 Create an Excel Table Report

1. Start SSA and open the project file *C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61-analysis.spf.*

2. On the far right of the *Output* toolbar, select the **Excel Table Reports** command.

3. Excel opens and displays the available reports organized by tabs in a single workbook. Click the **X** button to **Close** Excel and don't save changes.

	A	B	C	D	E	F	G	H	I	J	K	L
1	SN	Element	Description	From (Inlet)	To (Outlet)	Length	Inlet	Inlet	Outlet	Outlet	Total	Average
2		ID		Node	Node		Invert	Invert	Invert	Invert	Drop	Slope
3							Elevation	Offset	Elevation	Offset		
4												
5						(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(%)
6	1	L-bypass-S-(4)		S-(4)	S-(5)	171.58	27.20	0.00	26.51	0.00	0.69	0.4000
7	2	L-bypass-S-(8)		S-(8)	S-(7)	77.03	27.54	0.00	26.67	0.00	0.87	1.1300
8	3	L-bypass-S-(9)		S-(9)	S-(8)	116.05	29.07	0.00	27.54	0.00	1.53	1.3200
9	4	L-PIPE-(1)		S-(1)	S-(2)	195.28	28.87	0.00	26.93	0.00	1.94	0.9900
10	5	L-PIPE-(10)		S-(11)	S-(12)	116.70	25.80	0.00	23.12	0.00	2.68	2.3000
11	6	L-PIPE-(13)		S-(7)	S-(6)	69.06	26.67	0.00	25.06	0.00	1.61	2.3300
12	7	L-PIPE-(16)		S-(16)	S-(13)	273.69	26.43	0.00	24.81	0.00	1.62	0.5900
13	8	L-PIPE-(17)		S-(3)	S-(13)	49.89	26.23	0.00	24.81	0.00	1.42	2.8500
14	9	L-PIPE-(5)		S-(13)	S-(12)	319.62	24.81	0.00	23.12	0.00	1.69	0.5300
15	10	L-PIPE-(7)		S-(5)	S-(6)	102.37	26.51	0.00	25.06	0.00	1.45	1.4200
16	11	L-PIPE-(9)		S-(10)	S-(11)	100.64	27.30	0.00	25.80	0.00	1.50	1.4900
17	12	PIPE-(1)		S-(1)	S-(16)	98.24	28.87	0.00	26.93	0.50	1.94	1.9700
18	13	PIPE-(10)		S-(11)	S-(12)	114.37	25.80	0.00	23.49	0.37	2.31	2.0200
19	14	PIPE-(13)		S-(7)	S-(6)	59.05	26.67	0.00	26.08	1.02	0.59	1.0000

Pipes | Outfalls | Inlets | Subbasins

4. **Close** SSA and **Save** changes to the project.

[THIS PAGE IS INTENTIONALLY LEFT BLANK]

9 IDENTIFY SURCHARGED PIPES

DESCRIPTION

This chapter shows how to identify surcharged Pipes. The user interface in SSA highlights elements and values in dialog boxes in red when a surcharged condition exists.

OBJECTIVES

In this chapter, you will learn about:

- Changing Frequency
- Run Analysis
- Identifying surcharged elements

CHAPTER SETUP

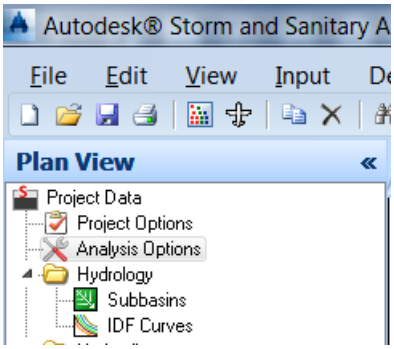
Run the Chapter 9 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX Projects** folder.

Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

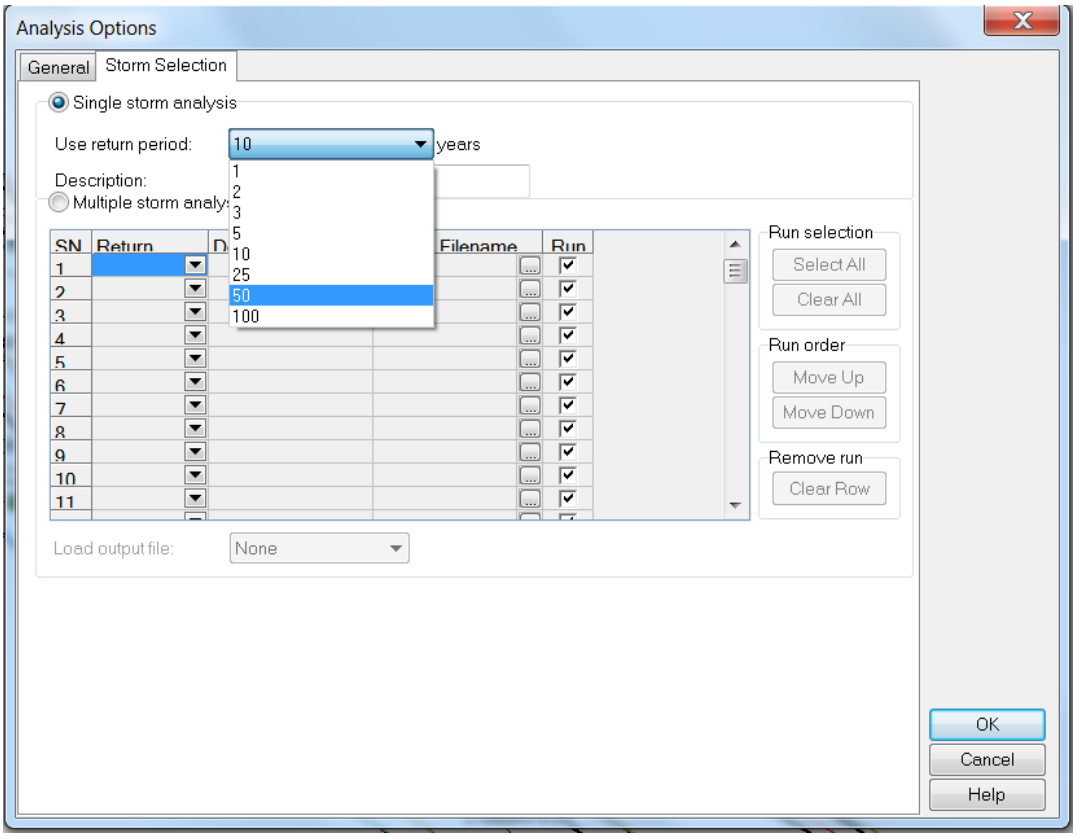
Exercise 9.1 Identify Surcharged Pipes

➢ Change the Frequency

1. Start SSA and open the project file *C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61-analysis.spf*.
2. Double click **Analysis Options** to open the Analysis Options dialog.

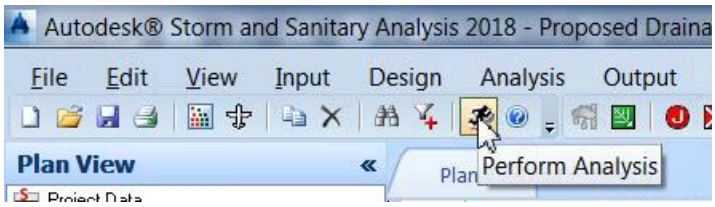


3. Click the **Storm Selection** tab. Change the *Use return period* drop-down list to **50.** Click the **OK** button.

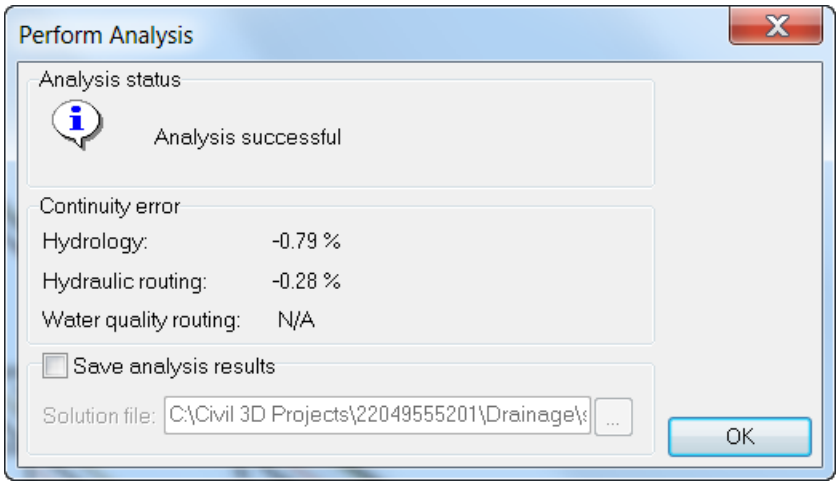


➢ *Run Analysis*

1. From the *Standard* toolbar, select the **Perform Analysis** command.

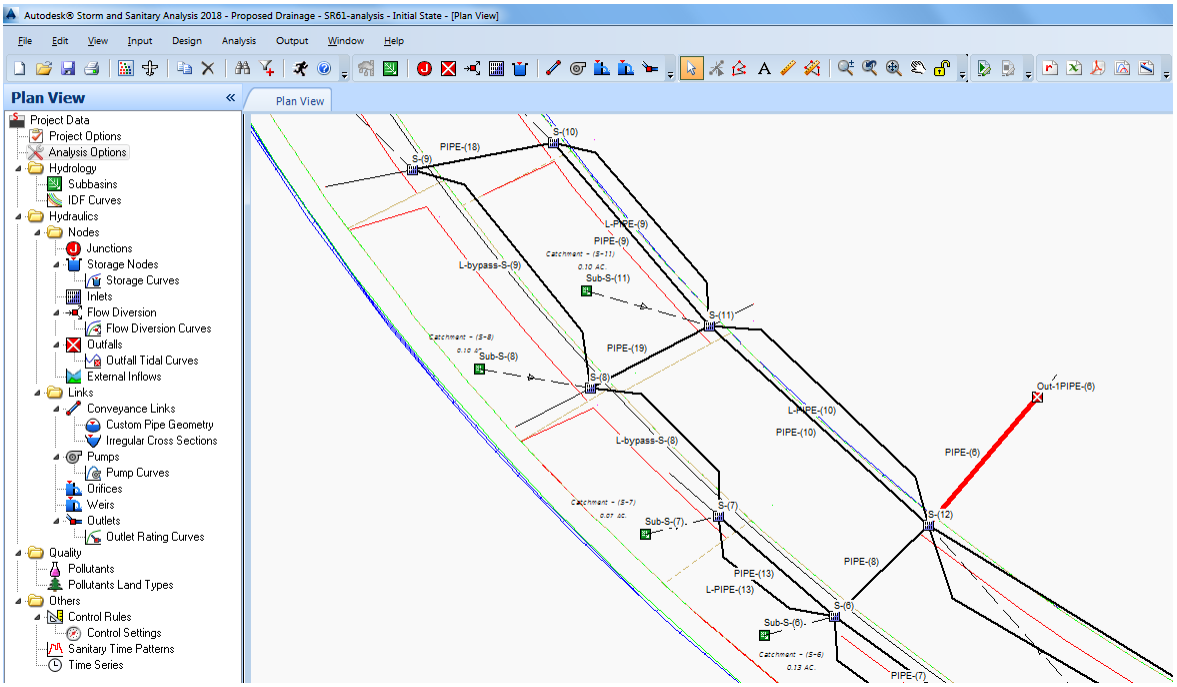


2. **Clear** the *Save analysis results* check box, and then click **OK**.



➢ *Display the Plan View Tab*

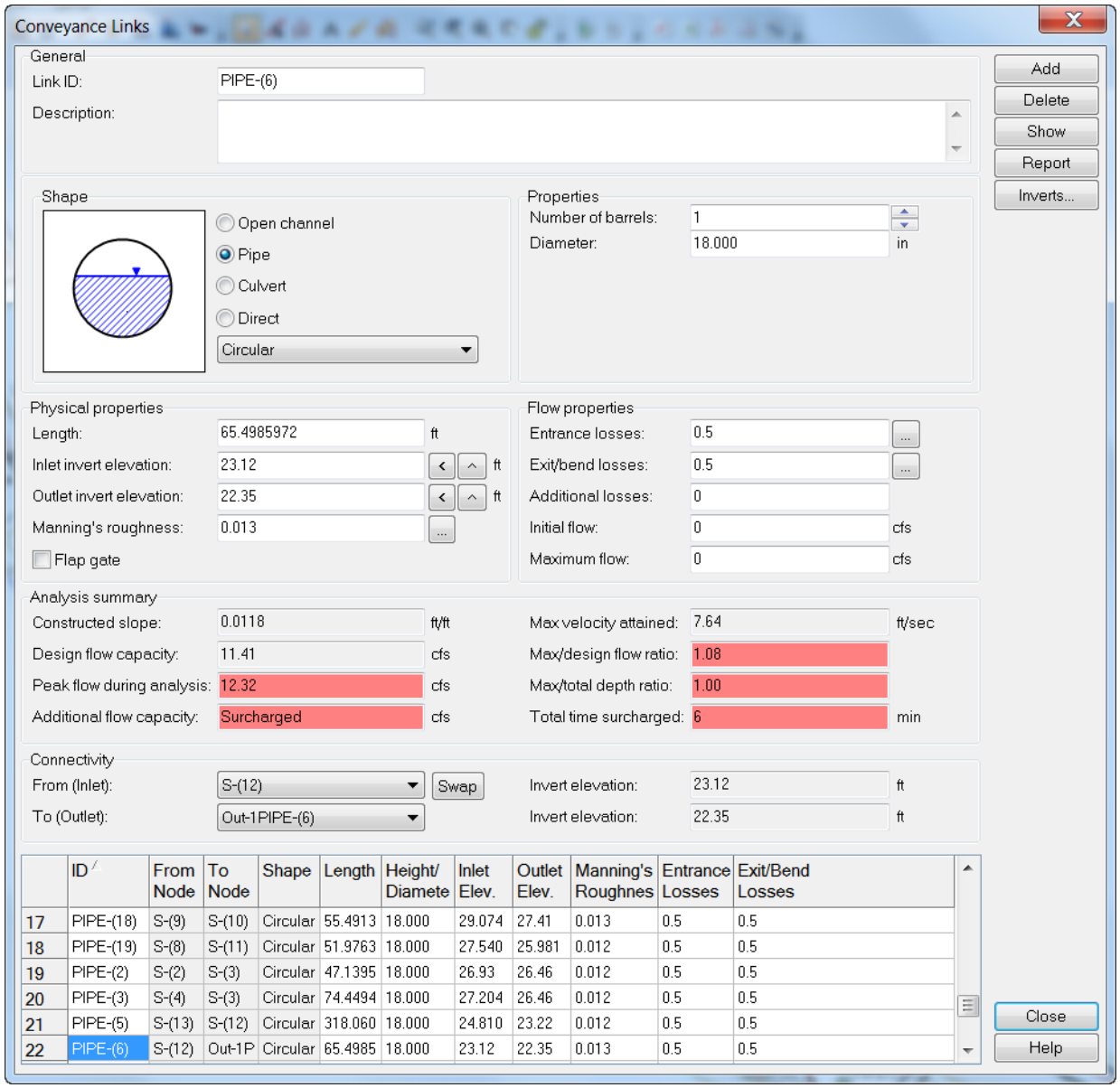
1. Select the **Plan View** tab, notice *PIPE-(6)* is highlighted in red. This indicates the pipe is *surcharged*.



➢ *Display PIPE-(6) in Conveyance Links Dialog*

The Conveyance Links dialog box provides a section titled ANALYSIS SUMMARY that provides a brief summary of the simulation results for the selected channel, pipe, or culvert link, as shown in the following figure.

1. Double click **PIPE-(6)** in *Plan View* to display the Conveyance Links dialog. The fields with the red background indicate the values that are *surcharged*.

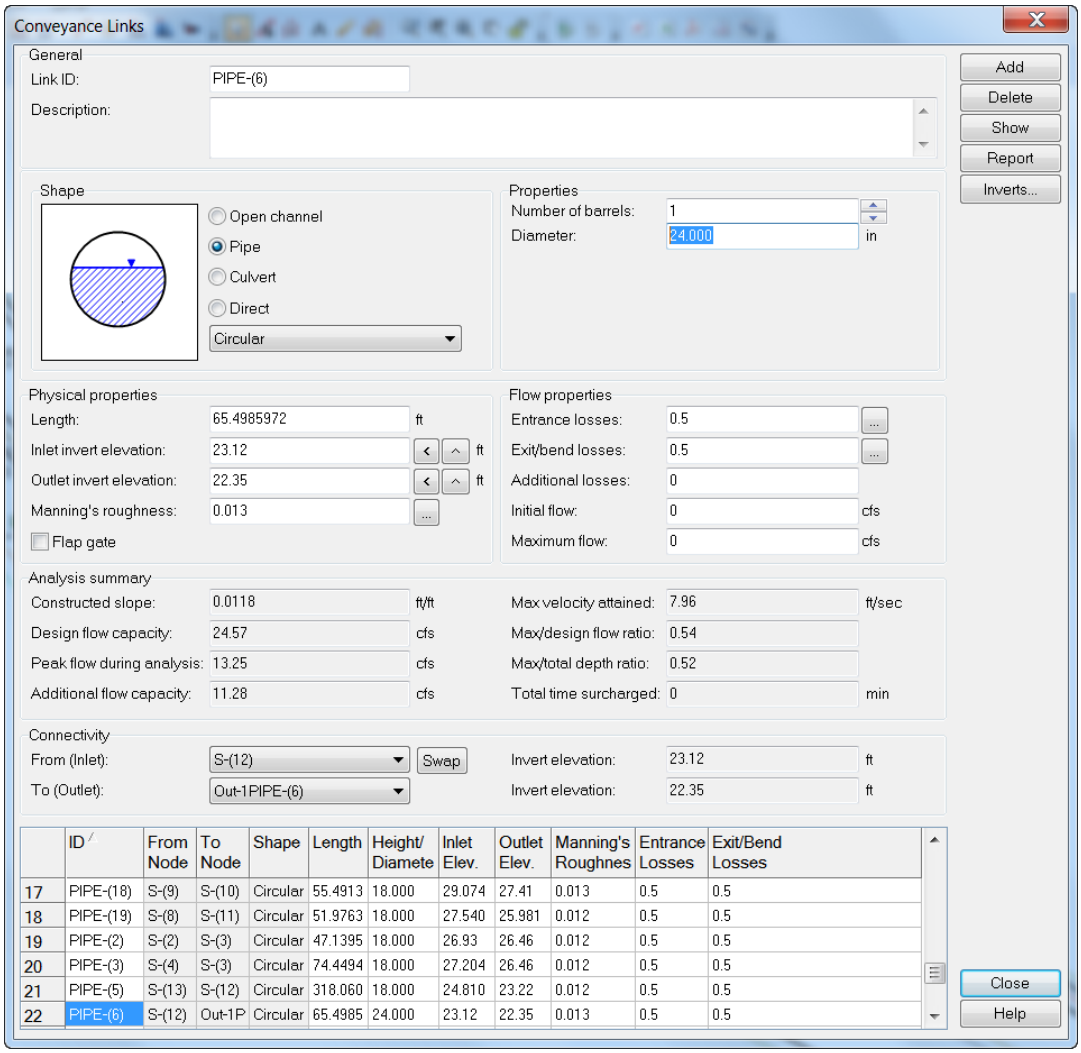


Note From the SSA Help File, Page 118 Chapter 5, Display Analysis Results, Input Dialog Boxes

For link elements that are experiencing surcharging, or node elements that are experiencing flooding, the analysis summary output fields will change the background color to red in order to help you find those elements that should be reviewed for adequacy.

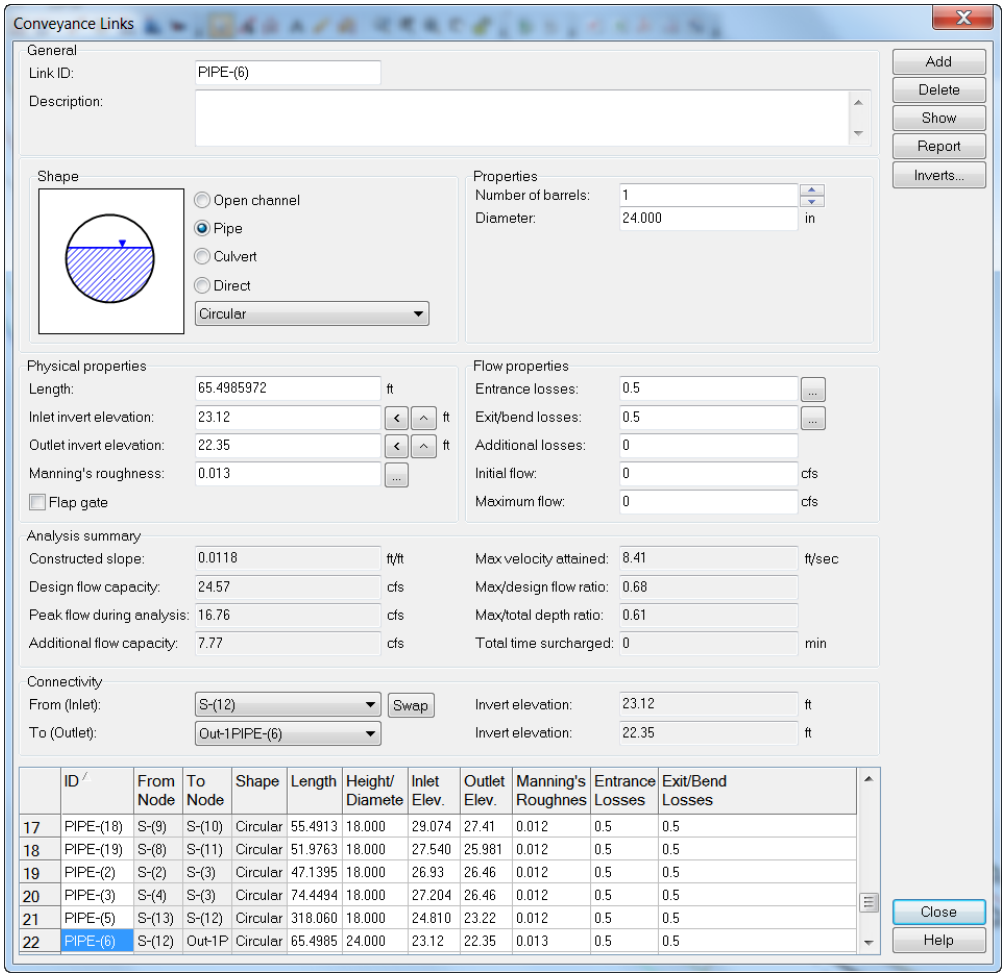
For storm drain inlets, in the Inlet dialog box Analysis Summary section the software assumes that if more than ½ of the roadway lane is covered with stormwater from gutter spread, that the inlet capacity is not sufficient.

2. Change the *PIPE-(6) diameter* to **24"** and **re-run** the *Analysis* to resolve the surcharge condition.

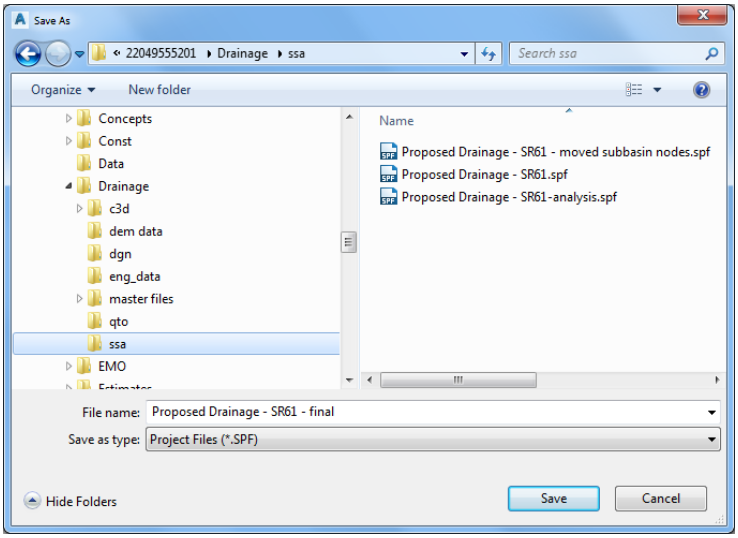


	ID	From Node	To Node	Shape	Length	Height/ Diamete	Inlet Elev.	Outlet Elev.	Manning's Roughnes	Entrance Losses	Exit/Bend Losses
17	PIPE-(18)	S-(9)	S-(10)	Circular	55.4913	18.000	29.074	27.41	0.013	0.5	0.5
18	PIPE-(19)	S-(8)	S-(11)	Circular	51.9763	18.000	27.540	25.981	0.012	0.5	0.5
19	PIPE-(2)	S-(2)	S-(3)	Circular	47.1395	18.000	26.93	26.46	0.012	0.5	0.5
20	PIPE-(3)	S-(4)	S-(3)	Circular	74.4494	18.000	27.204	26.46	0.012	0.5	0.5
21	PIPE-(5)	S-(13)	S-(12)	Circular	318.060	18.000	24.810	23.22	0.012	0.5	0.5
22	PIPE-(6)	S-(12)	Out-1P	Circular	65.4985	24.000	23.12	22.35	0.013	0.5	0.5

3. Double click **PIPE-(6)** in *Plan View* to view the results the Conveyance Links dialog. (Using the **50yr** *Return Frequency*). Click **Close**.



4. Use the **Save As** command to **Save** the SSA project to a new *file name* **Proposed Drainage - SR61 - final.spf.**



5. **Close** SSA and **Save** changes to the project.

10 EXPORT/ IMPORT STM FILE FROM SSA TO CIVIL 3D

DESCRIPTION

This chapter will review the process of exporting a STM file from SSA and the process of moving the pipe network to Civil 3D.

OBJECTIVES

In this chapter, you will learn about:

- Exporting an STM file from SSA
- Set Civil 3D Storm Sewers Migration Defaults
- Import a STM File into Civil 3D

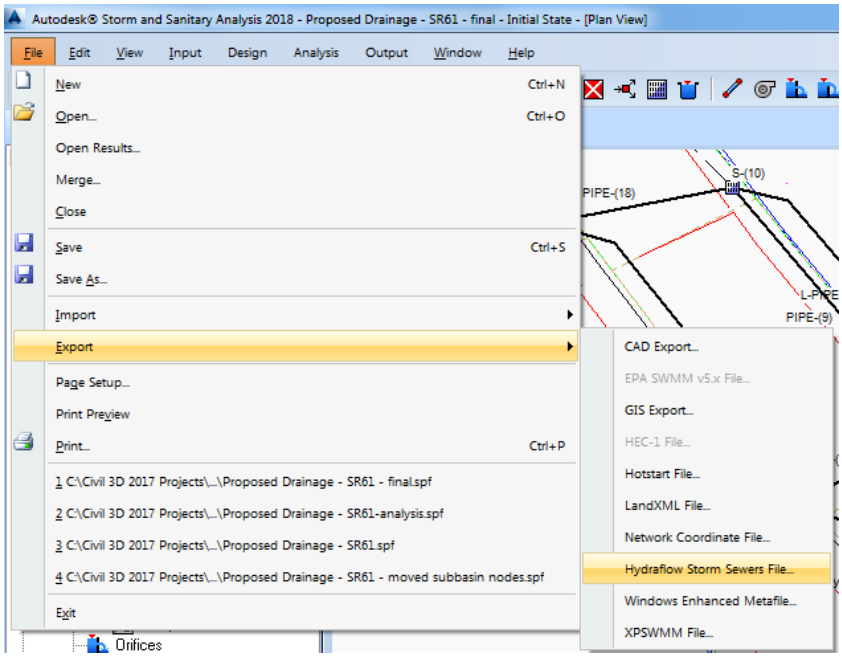
CHAPTER SETUP

Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

Exercise 10.1 Export to a STM File From SSA

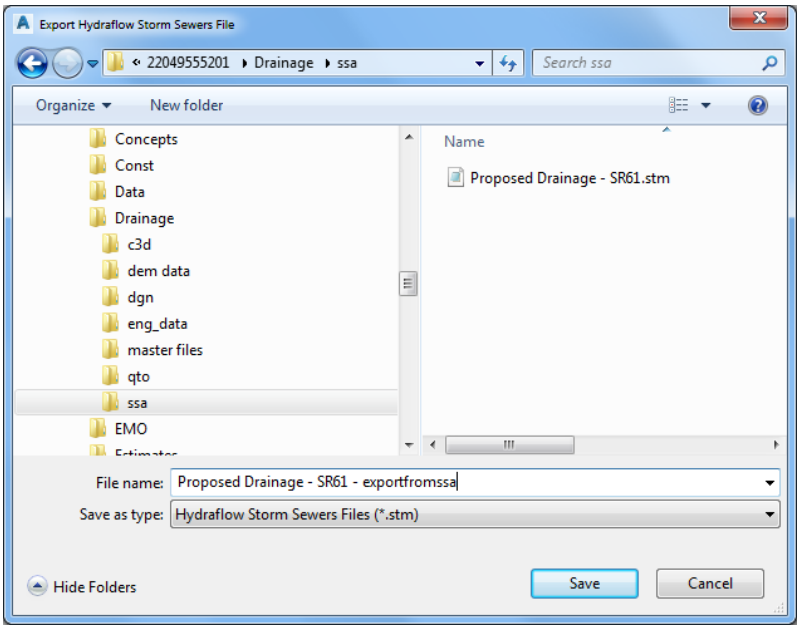
Run the Chapter 10 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX Projects** folder.

1. Start SSA and open the project file *C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa\Proposed Drainage - SR61-final.spf.*
2. Click the **File** pull-down menu, hover over *Export* and select **Hydraflow Storm Sewers File.**

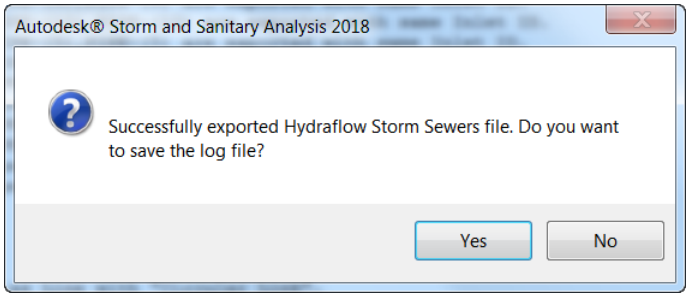


3. Browse to the *ssa* folder under the *Drainage* folder and create a new file:

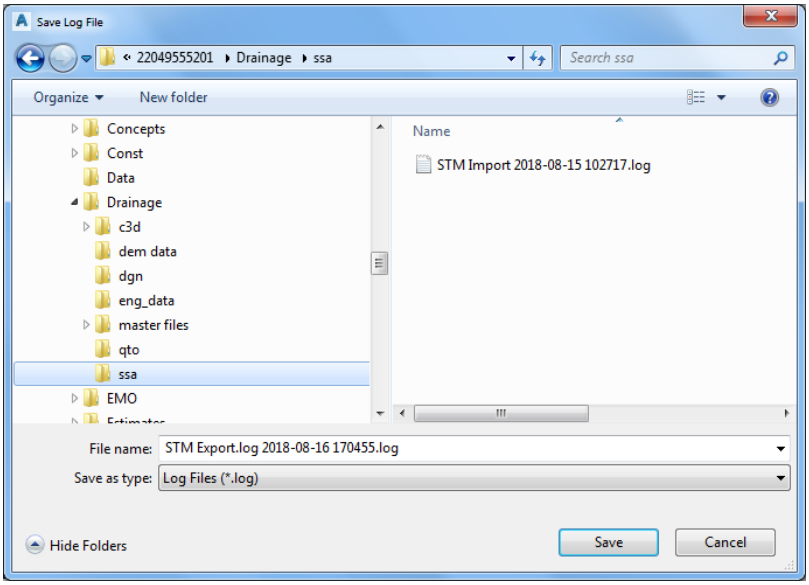
   *C:\Civil 3D 20XX Projects\22049555201\Drainage\ssa**Proposed Drainage - SR61 - exportfromssa.stm.***



4. Click the **Yes** button.



5. Use the default *file name* created by the program. Click the **Save** button.



6. **View** the *log* file on the *Log File* tab.

Plan View | Log File

```
Hydraflow Storm Sewers Export Log File
--------------------------------------

Filename : Proposed Drainage - SR61 - exportfromssa.stm

*******************
Elements Exported
*******************
Number of Subbasins        : 14
Number of Outfalls         : 1
Number of Channel/Pipes    : 25
Number of Inlets           : 14

*********
Summary
*********
Node S-(1) have 2 outlet links. So links L-bypass-S-(1),PIPE-(1) are exported with same Inlet ID.
Node S-(10) have 2 outlet links. So links L-PIPE-(9),PIPE-(9) are exported with same Inlet ID.
Node S-(11) have 2 outlet links. So links L-PIPE-(10),PIPE-(10) are exported with same Inlet ID.
Node S-(13) have 2 outlet links. So links L-PIPE-(5),PIPE-(5) are exported with same Inlet ID.
Node S-(16) have 2 outlet links. So links L-PIPE-(16),PIPE-(16) are exported with same Inlet ID.
Node S-(3) have 2 outlet links. So links L-PIPE-(17,PIPE-(17) are exported with same Inlet ID.
Node S-(4) have 2 outlet links. So links L-bypass-S-(4),PIPE-(3) are exported with same Inlet ID.
Node S-(5) have 2 outlet links. So links L-PIPE-(7),PIPE-(7) are exported with same Inlet ID.
Node S-(7) have 2 outlet links. So links L-PIPE-(13),PIPE-(13) are exported with same Inlet ID.
Node S-(8) have 2 outlet links. So links L-bypass-S-(8),PIPE-(19) are exported with same Inlet ID.
Node S-(9) have 2 outlet links. So links L-bypass-S-(9),PIPE-(18) are exported with same Inlet ID.

--------------
Link Section
--------------
Link L-bypass-S-(1) of type "Direct" exported as Line with "Circular Link".
Link L-bypass-S-(4) of type "Direct" exported as Line with "Circular Link".
Link L-bypass-S-(8) of type "Direct" exported as Line with "Circular Link".
Link L-bypass-S-(9) of type "Direct" exported as Line with "Circular Link".
Link L-PIPE-(13) of type "Direct" exported as Line with "Circular Link".
Link L-PIPE-(16) of type "Direct" exported as Line with "Circular Link".
Link L-PIPE-(17 of type "Direct" exported as Line with "Circular Link".
Link L-PIPE-(5) of type "Direct" exported as Line with "Circular Link".
Link L-PIPE-(7) of type "Direct" exported as Line with "Circular Link".
Link L-PIPE-(9) of type "Direct" exported as Line with "Circular Link".

END OF FILE
```

7. **Close** SSA and **save** changes to the project.

PIPE NETWORKS

Pipe networks imported from the Hydraflow Storm Sewers STM file format to Civil 3D can contain hydraulic property information, such as hydraulic grade lines (HGL) and energy grade lines (EGL) which is part of the reason why we use SSA, for better analysis of our design.

The following values are calculated in the Storm Sewers Extension version 2010 and beyond, but are not found in older versions of .stm files (prior to 2010):

- Pipes
  - Energy Grade Line Up
  - Energy Grade Line Down
  - Flowrate
- Structures
  - Energy Grade Line
  - Known Capacity

In the Storm Sewers Extension, the list of pipe sizes is hard coded and cannot be customized. This list does not distinguish between inner and outer pipe diameter, pipe material, or pipe class, as does AutoCAD Civil 3D.

MATCHING NETWORK NAMES & PART NAMES

When you import the .stm file, if any of the pipe network names in the .stm file match pipe network names in the AutoCAD Civil 3D drawing, you must choose to either create new pipe networks, or update existing pipe networks. Rules are not applied to pipes or structures when importing pipe networks from either LandXML or from the Hydraflow Storm Sewers STM file format.

As a beneficial workflow we will Save As the existing drawing with a new name so we can preserve the content that existed prior to editing in SSA and compare that to the revised Pipe Network data that returns from SSA to Civil 3D, overwriting the existing data that matches the part names of incoming data.

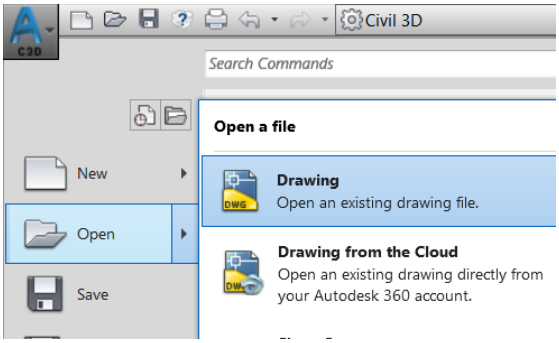
Note During the creation of this training manual, an MR2 update to the FDOT Civil 3D 2018 Kit was released, This MR2 release contains new FDOT Pipe and Structure databases. We have created a new DRPRRD06.dwg file which contains these updates.

Exercise 10.2 Open and Save As the DRPRRD06.DWG File

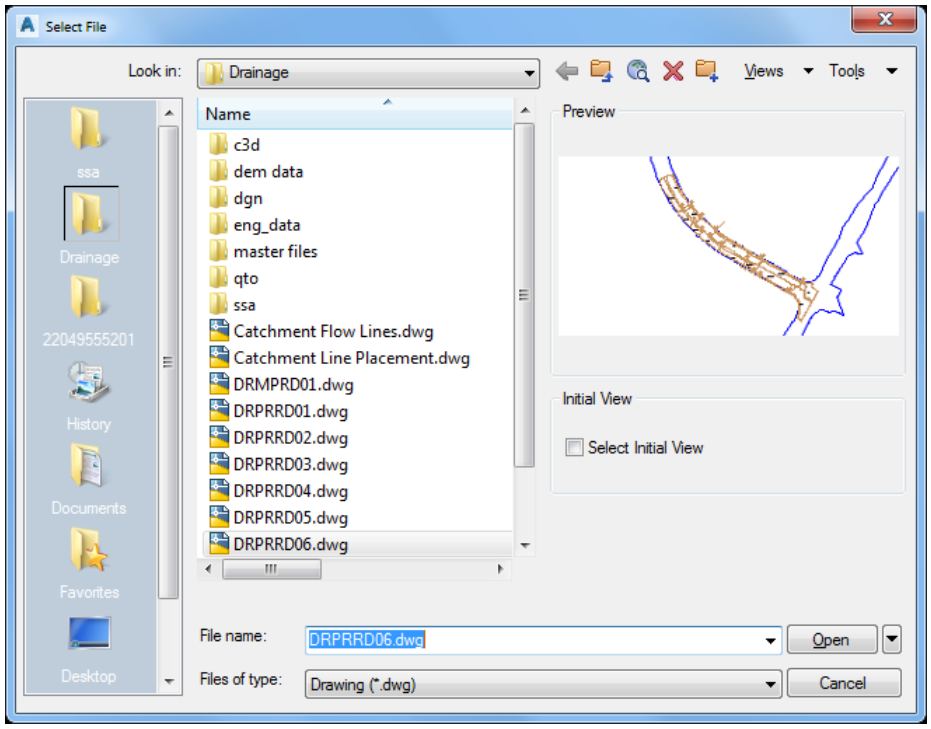
Run the Chapter 11 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX Projects** folder.

1. Start Civil 3D by double clicking the 

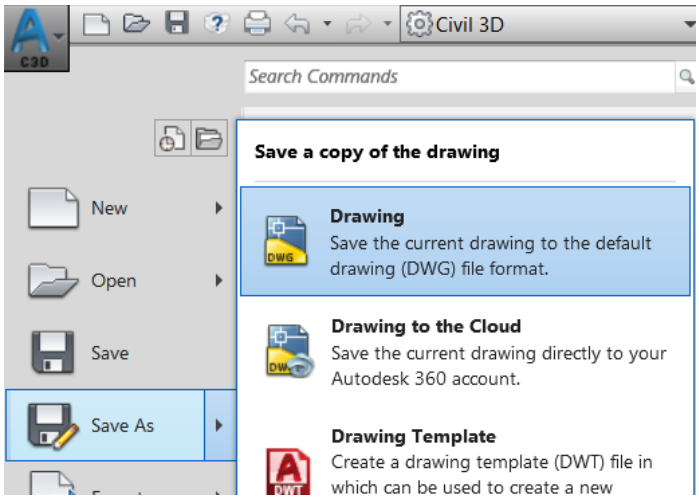
shortcut from your desktop.
2. Click the **Application** menu, then hover over *Open* and select **Drawing**.



3. Browse to the *C:\Civil 3D 20XX Projects\22049555201\Drainage* folder and select ***DRPRRD06.dwg.*** Click the **Open** button.

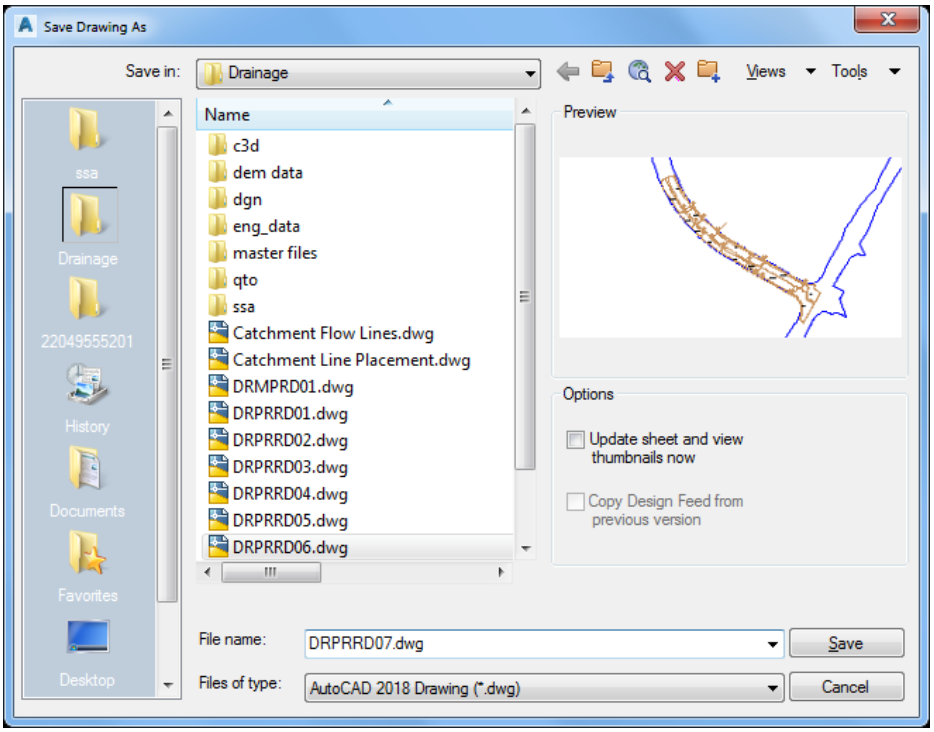


4. Click the **Application** menu and hover over *Save As* then select **Drawing**.



5. Browse to the *C:\Civil 3D 20XX Projects\22049555201\Drainage* folder and type ***DRPRRD07.dwg*** in the *File name* field. Click the **Save** button to create:

   C:\Civil 3D 20XX Projects\22049555201\Drainage\DRPRRD07.dwg.

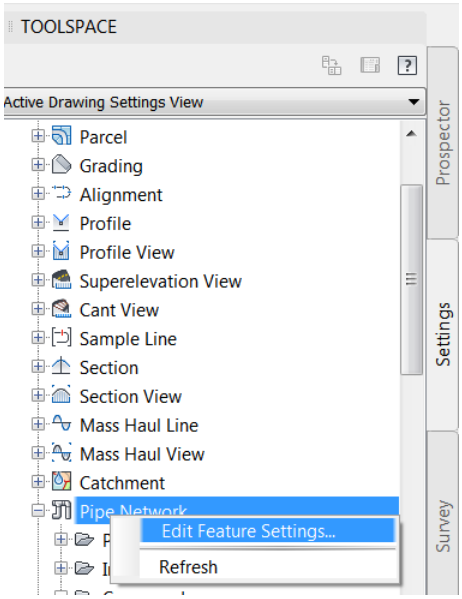


Exercise 10.3 Set Civil 3D Storm Sewers Migration Defaults

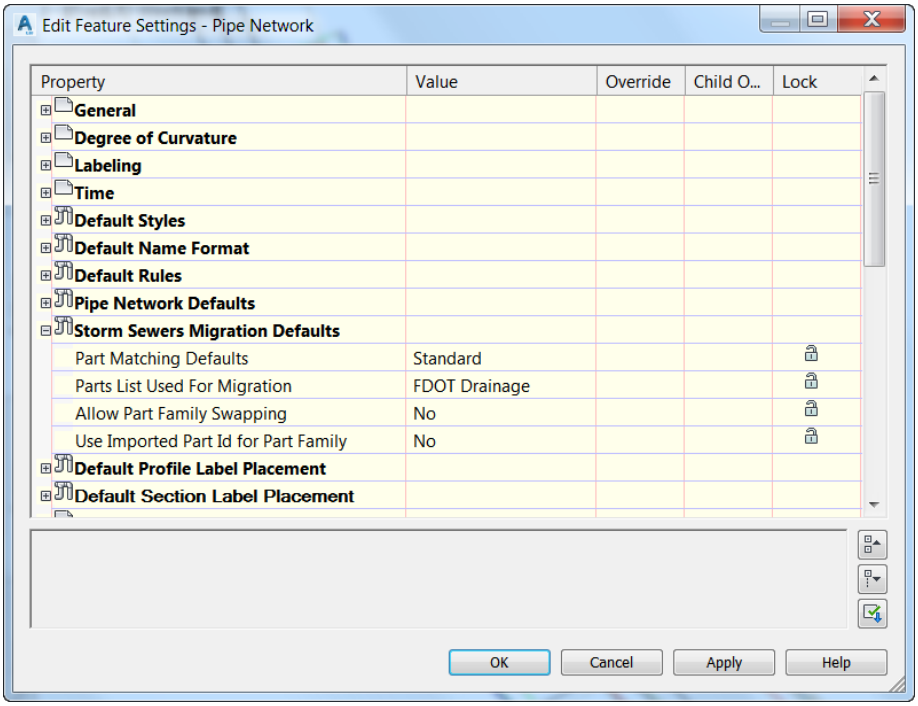
Pipe networks imported from the Storm Sewers Extension can contain hydraulic property information, such as hydraulic grade lines (HGL) and energy grade lines (EGL) and more.

➢ *Edit Feature Settings*

1. In the Toolspace select the **Settings** (tab). Right click on *Pipe Network* and select **Edit Feature Settings**.



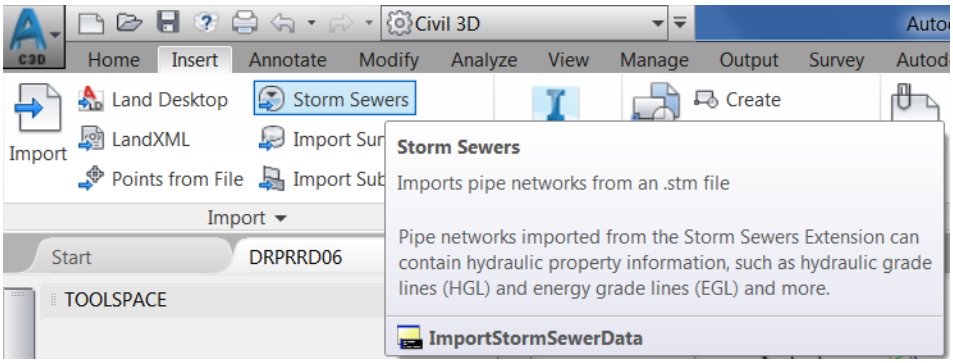
2. Set the *Allow Part Family Swapping* to **No** and the *Use Imported Part Id for Part Family* to **No**, then click **OK**.



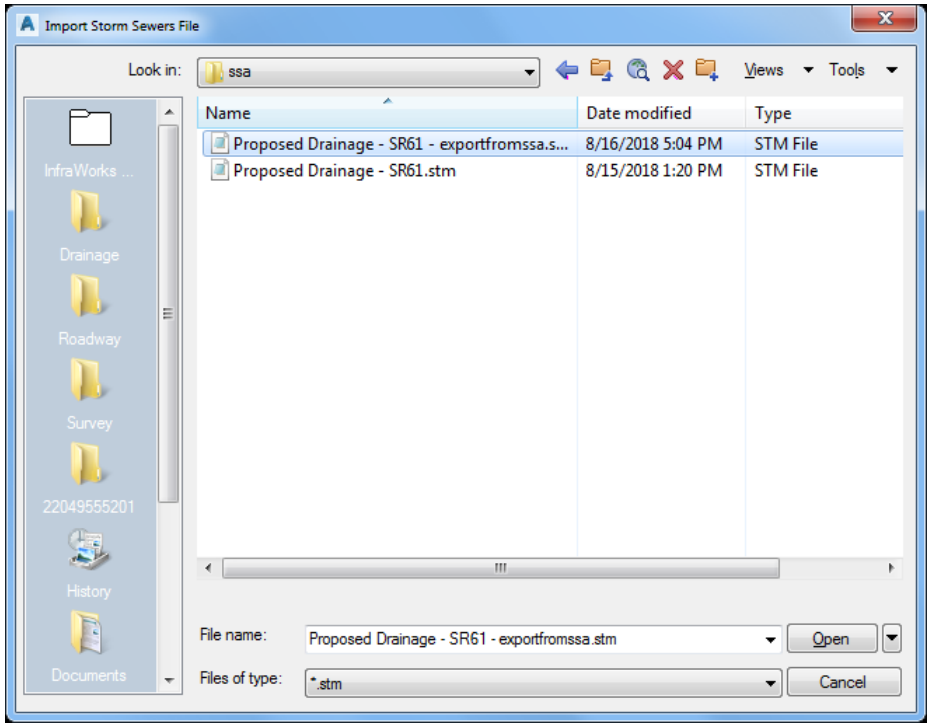
Note Part Family Swapping is disabled with these settings so you can update the existing structures and pipes in Civil 3D.

➢ *Import STM File in Civil 3D*

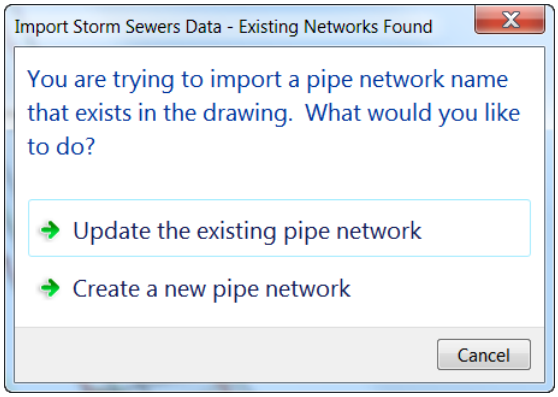
1. From the *Insert tab* > *Import* select the **Storm Sewers** command.



2. Browse to the *ssa* folder under the *C:\Civil 3D Projects\22049555201\Drainage* folder and select the file ***Proposed Drainage - SR61 - exportfromssa.stm***.

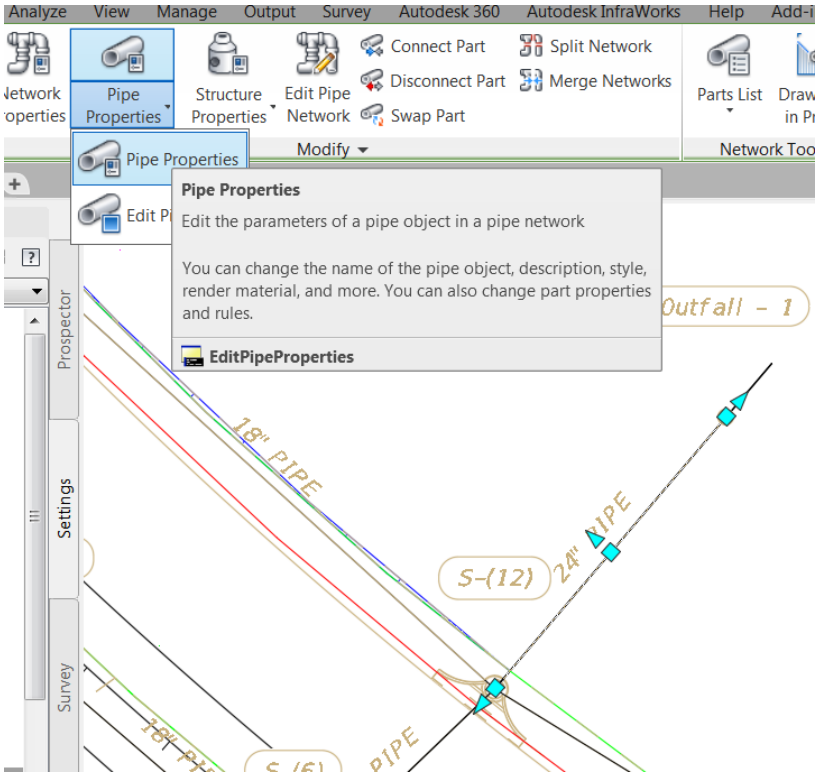


3. Click **Update the existing pipe network.**

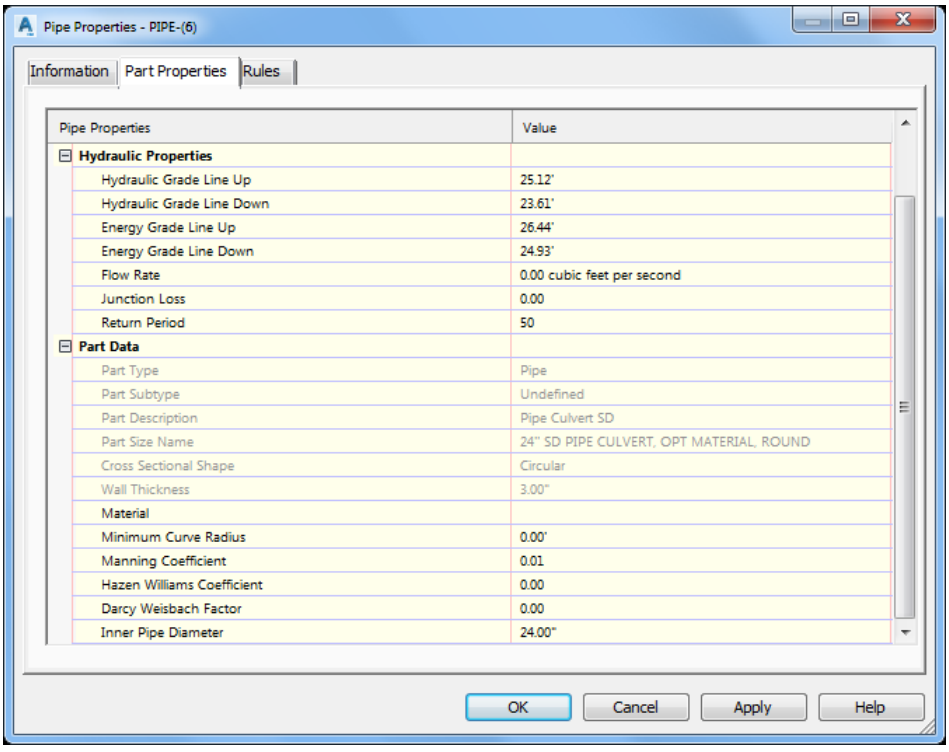


4. No errors display in the *Event Viewer*.

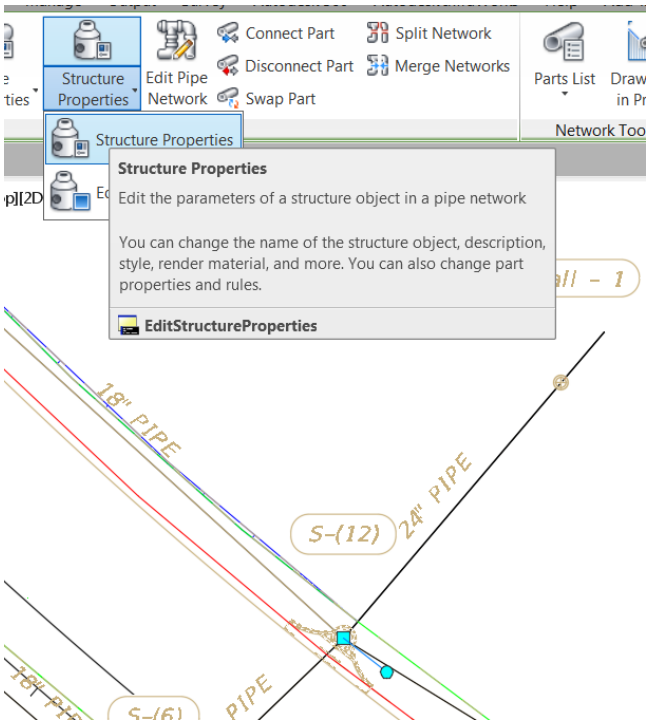
5. Grip PIPE-(6) and select the Pipe Properties command in the ribbon.



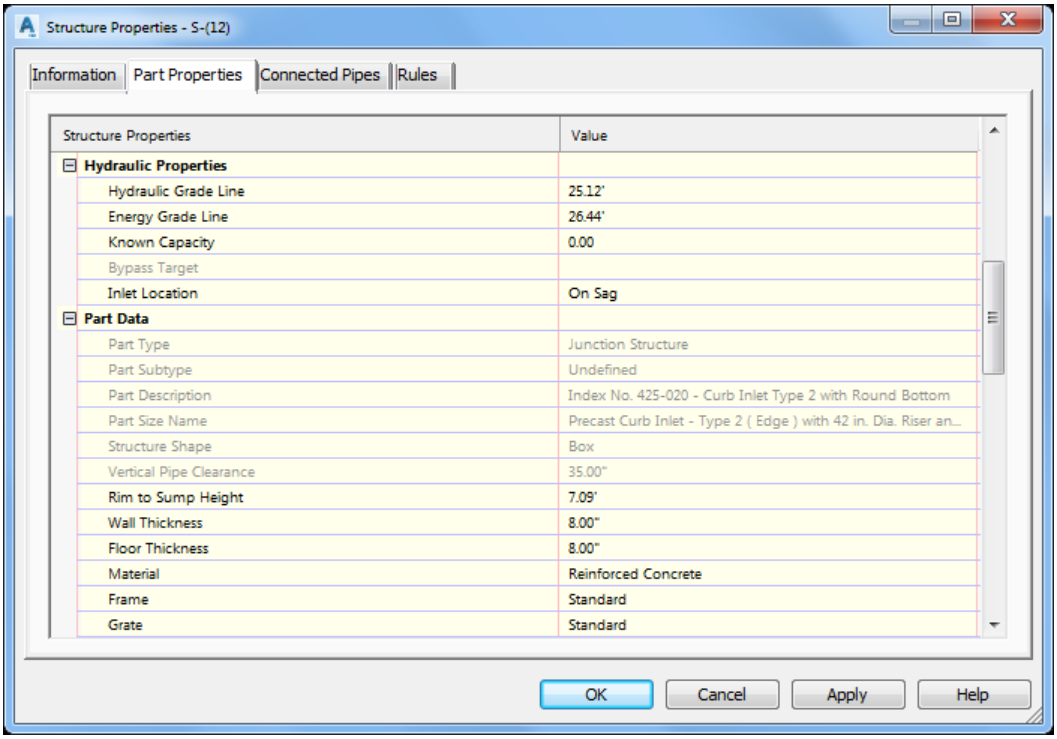
6. Open the Pipe Properties for PIPE-(6). The Hydraulic Properties have been updated. The Inner Pipe Diameter is now 24" Click OK to close the dialog. Press ESC to clear the grips on PIPE-(6).



7. Grip Inlet S-(12) and Click Structure Properties from the ribbon.



8. On the *Part Properties* tab scroll down to the *Hydraulic Properties*. Notice the *Hydraulic Grade Line* and *Energy Grade Line* are now populated with calculated values imported from SSA. Click **OK**. Click **ESC** to clear the grips.



9. **Save** the file and **Close** Civil 3D

11 Viewing HGL and EGL in Profile View

DESCRIPTION

This chapter covers the steps to view both HGL and EGL in the Profile View

OBJECTIVES

In this chapter, you will learn about:

- Edit Pipe Styles
- View Pipes in the Profile View

CHAPTER SETUP

Run the Chapter 12 - 22049555201.exe file and restore all files to the **C:\Civil 3D 20XX** Projects folder.

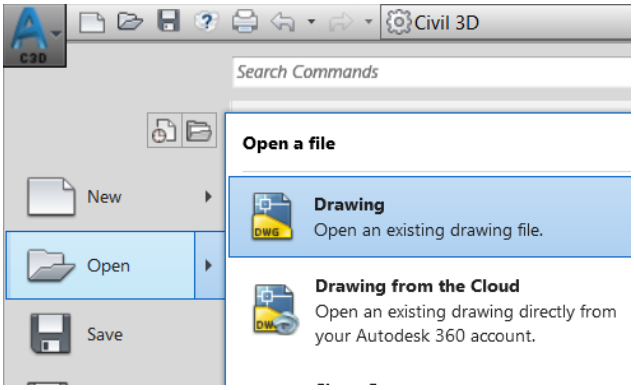
Do not change the location in which the Practice Files are installed. Doing so can cause errors when completing the exercises in this training manual.

Exercise 11.1 View Pipes in the Profile

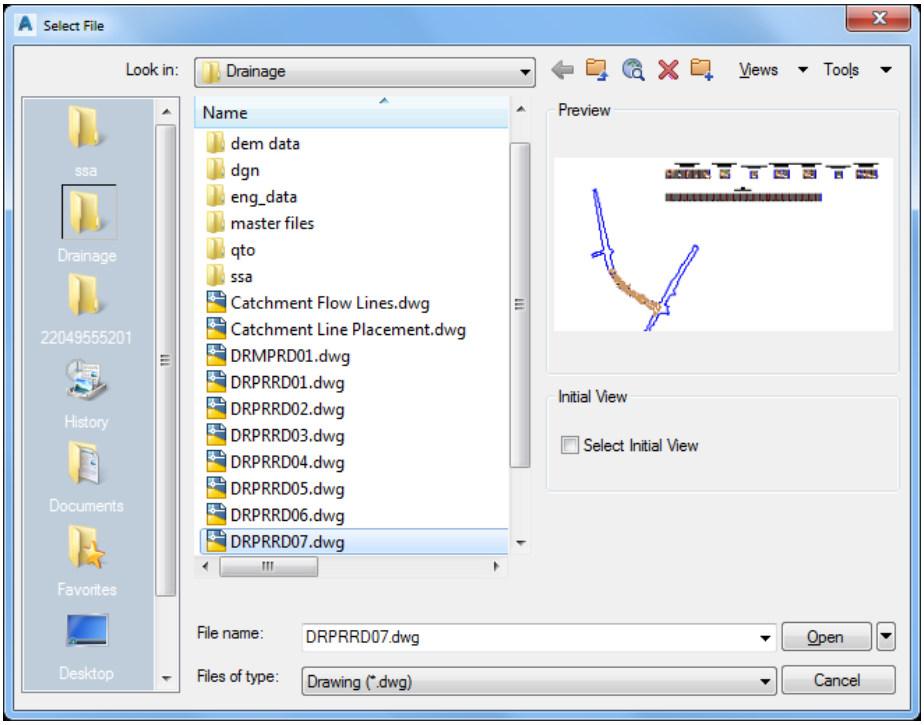
➢ *Edit the Pipe Style*

1. Start Civil 3D by double clicking the 

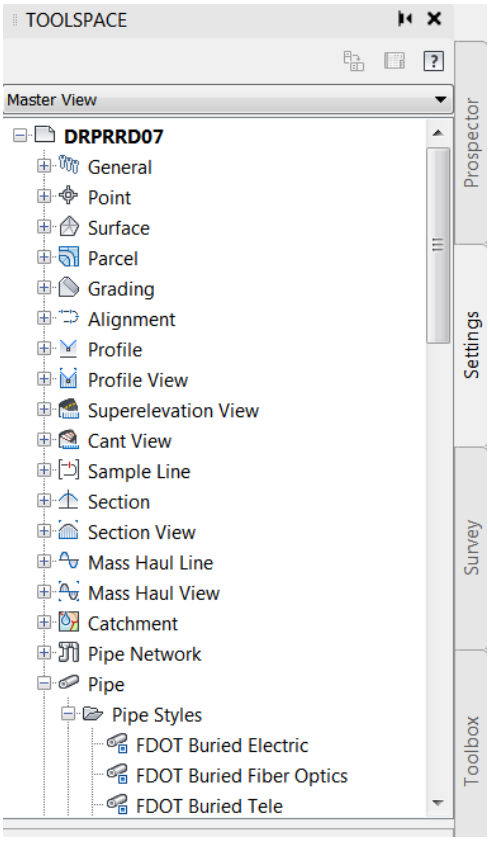
shortcut from your desktop.
2. Click the **Application** menu then hover over *Open* and select **Drawing**.



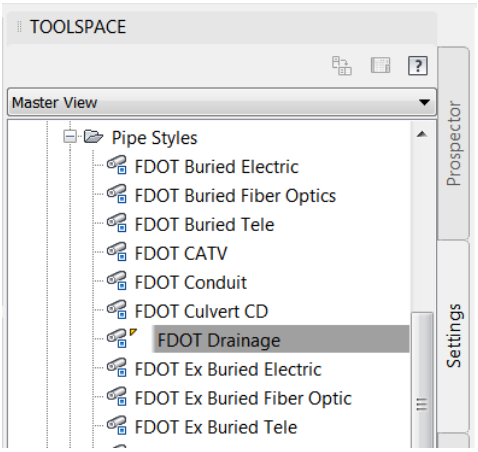
3. Browse to the C:\Civil 3D 20XX Projects\22049555201\Drainage folder and select the DRPRRD07.dwg. Click the Open button.



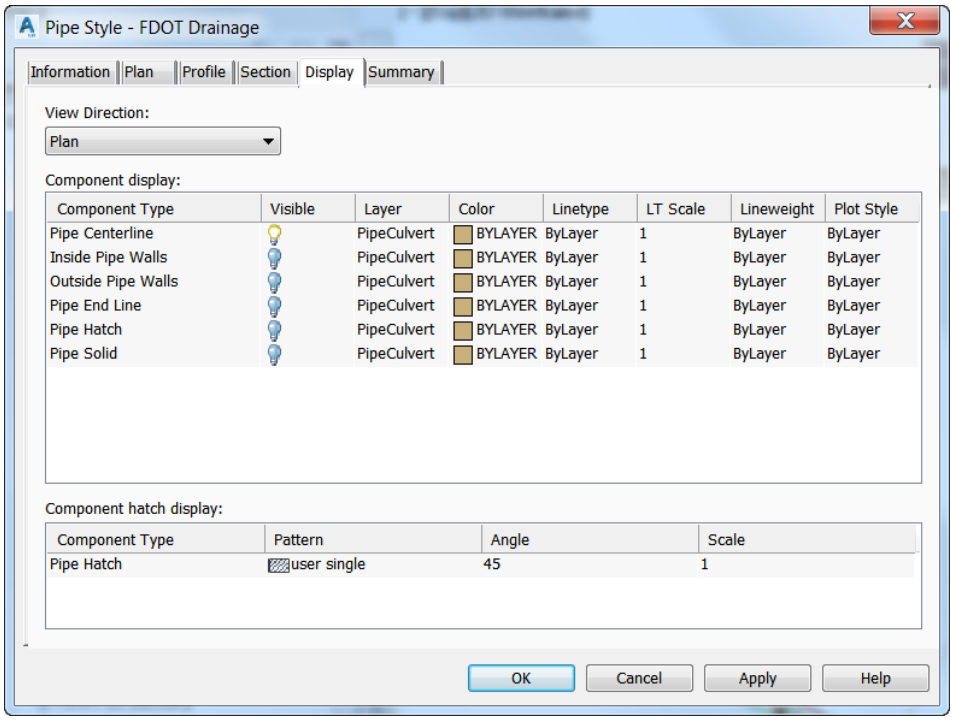
4. **Edit** the *pipe display style* for **HGL** and **EGL**. In Toolspace, on the *Settings* tab, expand the *Pipe* collection, then expand the *Pipe Styles* collection.



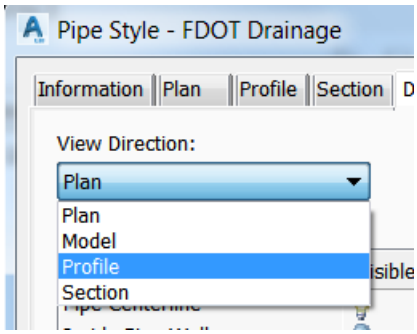
5. Double-click the *style* **FDOT Drainage** to edit it.



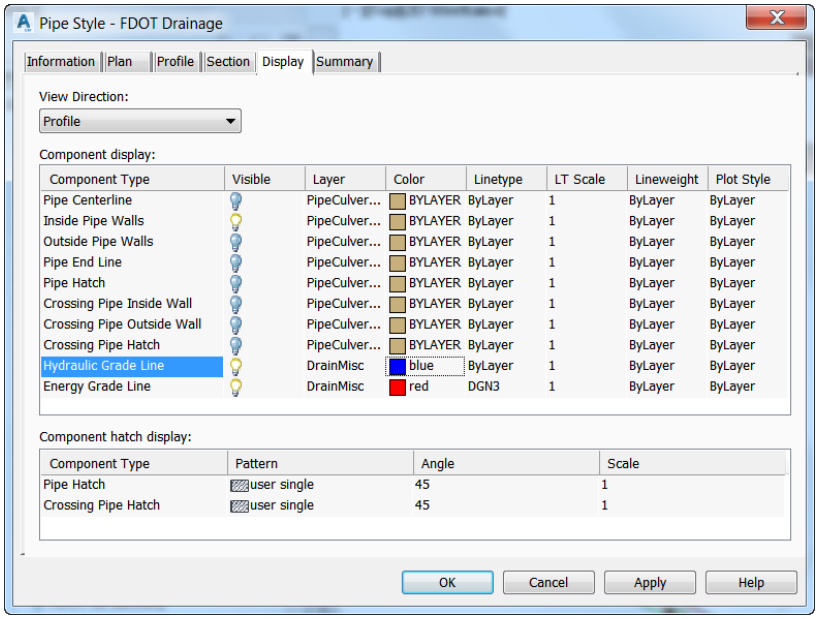
6. Select the **Display** tab in the Pipe Style dialog box.



7. Select **Profile** in the *View Direction* drop-down.

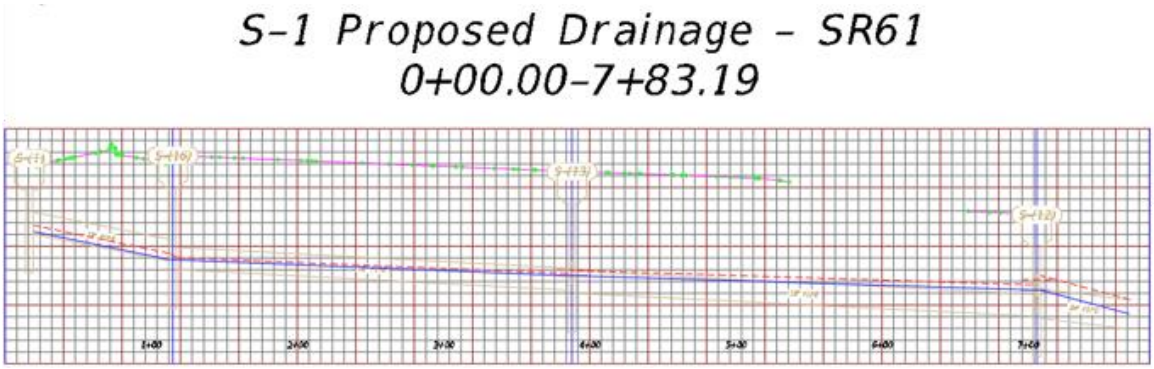


8. For the *Hydraulic Grade Line* and *Energy Grade Line* components, turn **On** *Visibility*, and set other display characteristics, such as *color* and *layer*, as shown. Click the **Apply** and **OK** buttons.

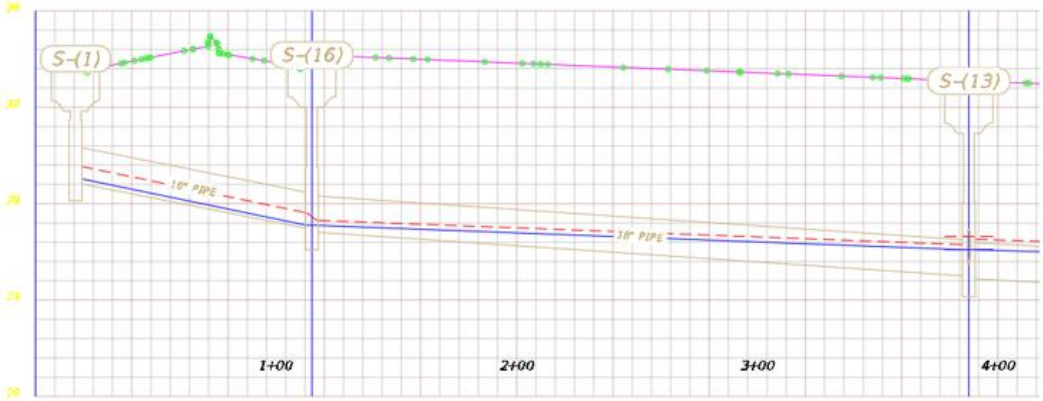


➢ *View Pipes*

1. **Zoom** to the *S-1 Profile View*. Notice the blue line inside the pipe walls represents the *hydraulic grade line*, the red dashed line represents the *energy grade line*.



Note You may wish to fade the grid lines in the profile view. To do this set the layer transparency to 70 for the layers GridMaj_dp and GridMin_dp.



2. **Save** the file and **Close** Civil 3D