3.1 DEFINING A DRAINAGE BASIN

The primary elements of the Drainage Basin in ASAD are:

- **Storm Event –** A set of data used to compute the time to the rainfall intensity (in/hr) relationship. This can be the Intensity-Duration-Frequency (IDF) tables which are already defined for Florida in the ASAD database. Another method for computing rainfall intensity is the User Define Time-Intensity tables.
- **Nodes –** A node is any 'Structure' or 'Ditch' that collects runoff and/or defines a position in the basin.
- **Reaches –** Reaches convey the runoff from one node to the next downstream node. Reaches can be 'Solid Pipes', 'Perforated Pipes in a Trench' (French drains), or 'Ditches'.
- **Outfall –** Each drainage system (basin) has **one and only one** outfall. It is the last node in the system and usually empties into a pond, lake, canal, etc. Sometimes an outfall node may be connected to some other existing storm sewer.

Note: Defining the drainage basin in ASAD is just part of the overall process. Review Appendix A: Step-By-Step User Flowchart for more details.

3.1.1 Adding, Deleting and Renaming a Storm Sewer System

The "Storm Sewer Add/Delete/Rename" dialog allows the designer to add or delete systems (see figure 3.1). A system must be added before nodes and reaches can be defined. Start this dialog from pulldown Edit>Storm Sewer System Add/Delete/Rename.

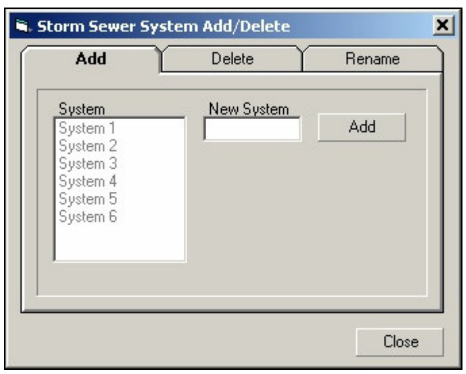


Figure 3.1

System list box: List the existing storm sewer systems.

New System: Enter the name of the new system you want to add. The new system name must conform to the ASAD object naming convention.

Add: After entering the new system name, click on the Add button to write it into the project database.

Note: Once a system has been added to the database, the system name will appear in the system list in the lower left corner of the ASAD status bar.

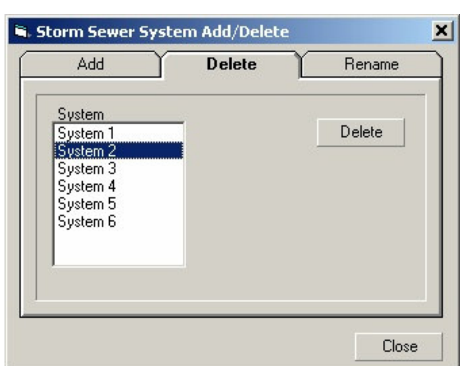


Figure 3.2

System list box: Click on the storm sewer system you want to delete.

Delete: Click on the Delete button to delete the selected storm sewer system.

Warning: Deleting the storm sewer system also deletes ALL NODES and REACHES that are in that system. However, it has no effect on structure or pipe definitions.

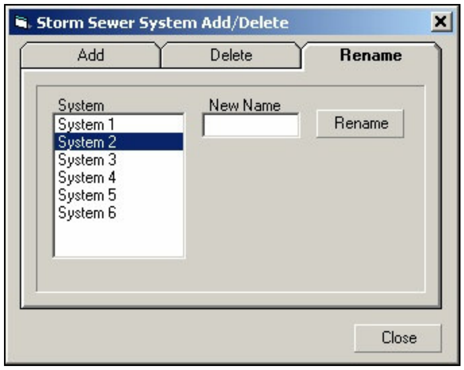


Figure 3.3

System list box: List the existing storm sewer systems. Select the system to be changed.

New Name: Enter the new name for the system. The new system name must conform to the ASAD object naming convention.

Rename: After entering the new system name, click on the Rename button to change the system name.

3.1.2 Outfall and Header Details

Once a system has been added, the next step is to enter the "SS System Header and Outfall Details" data (see figure 3.4). The outfall node is defined in this window. The Outfall tab includes the outfall node's number, baseline, station, offset, tail water elevation and exit loss coefficient. The storm event and runoff coefficients are entered on the Hydro tab. On the Designer tab, you can enter the designing engineer's, as well as the checker's, name. All data entered in this routine applies to the currently active system only. Start this dialog using pulldown Edit>SS Outfall and Header Details.

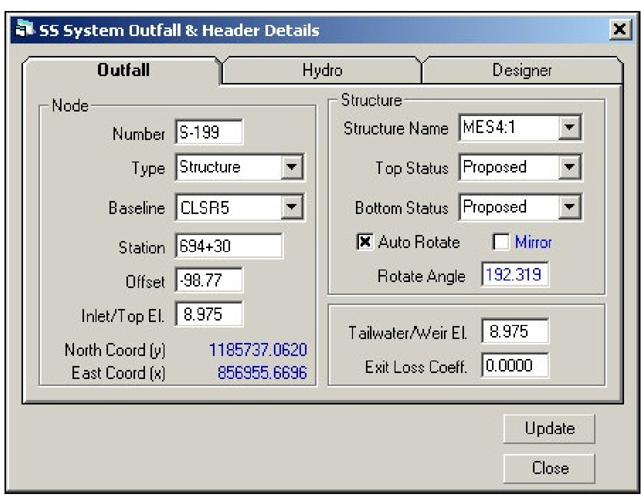


Figure 3.4

Number: The outfall node number. This name must conform to the ASAD object naming convention.

Note: The outfall is a special node and must be defined in this window and this window only. Do not define an outfall node in the general Node Edit window.

Type: Set the type of structure. Select 'Structure' or 'Ditch'.

Baseline: The baseline field (along with the node's station and offset) is used to compute the node's location, i.e. x and y coordinates. Baselines can be added, modified, and deleted using the Baseline Edit window.

Station: The station field (along with the offset) when applied to baseline geometry, is used for computing a node's location, i.e. x and y coordinates. The required format is xxxx+xx.xx (i.e. 327+28.12). This field is also printed on various reports including the storm tabulation report.

Offset: The offset field (along with the station) when applied to baseline geometry, is used for computing a node's location, i.e. x and y coordinates. Denote a left-of-centerline offset as negative(-), while the right-of-centerline offset is positive (plus sign not necessary). An offset directly on the centerline is 0.This field is also printed on various reports including the storm tabulation report.

Inlet/Top El.: This field is used when drawing the cross sectional view (drainage structures) of this node and only for inlets with tops (i.e. manholes, ditch bottom inlets). If this node is an endwall (MES, U-Endwall, etc) or is of type 'Ditch' then this field can be left blank (0).

Structure Name: The name of the structure. To review the structures go to pulldown Edit>Structure Definitions.

Top/Bottom Status: Set the top and/or bottom status of the structure with these two fields. Valid selections include Existing, Proposed, Future 1, Future 2, Future 3, and Future 4. For more information on structure status see Node Edit (see section 3.3)

Auto Rotate, Mirror & Rotate Angle: These three fields control the angle and orientation of the structure as drawn in plan view. For more information on these fields and their use, see Node Edit (section 3.3)

Tailwater Elevation: Water surface elevation for the water body the outfall node discharges to.

Exit Loss Coefficient: Enter an exit loss coefficient to compute the exit loss at the outfall. Normal values for this coefficient are 0.0 - 1.0. The system outfall Control Elevation = Tailwater Elevation + Exit Loss.

Update: Update saves the information on all 3 tabs back to the project database.

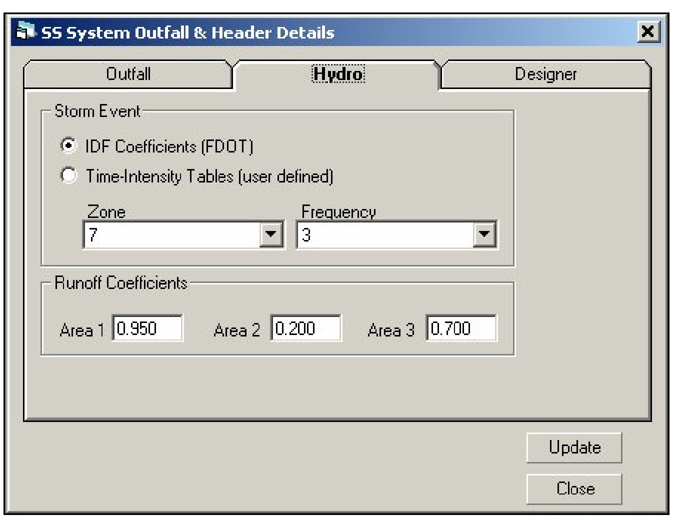


Figure 3.5

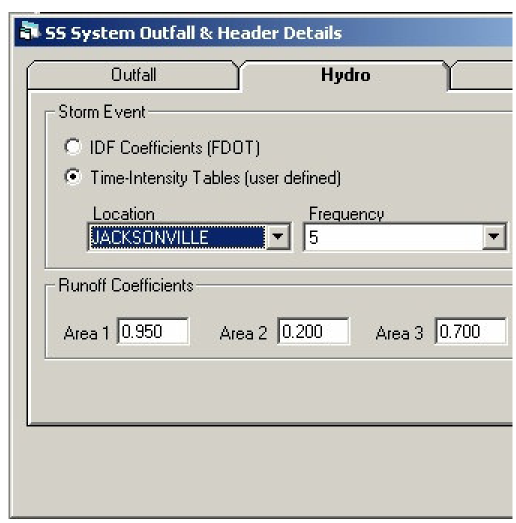


Figure 3.6

IDF Coefficients (FDOT) / Time-Intensity Tables (user defined): This toggle selects between using the FDOT IDF curves for rainfall intensity or a user defined time-intensity data table.

Zone & Frequency: The Zone and Frequency, as seen in figure 3.5, determine the IDF Curve

Coefficients to use when computing rainfall intensity during pipe sizing and storm tabulation calculations. The eleven Storm Zones which are geographic areas within the State of Florida are defined in Table 5-1 of the FDOT Drainage Manual. The hydrologic characteristics for each zone are graphed in “Rainfall Intensity-Duration-Frequency” curves in Tables 5-2 - 5-12. ASAD uses these tables to determine the rainfall intensity in units of depth / hour. The IDF coefficients used to generate the curves from a polynomial equation can be altered if the curves are adjusted. The coefficients are accessed in the “IDF Coefficients” dialog. The standard design frequencies are defined in the FDOT *Drainage Manual* in Section 10.3.1.

Location & Frequency: Figure 3.6 shows the screen when the Time-Intensity Tables (user defined) option is selected. Location is usually a city or county name and the Frequency is usually the return period in years. Both are defined using the Time Intensity Edit window which can be started using pulldown Edit>Storm Time-Intensity Table (user defined).

Area 1, 2 & 3: Runoff coefficients for Area 1, Area 2 and Area 3 are used to compute the total runoff flow (Q) created by the actual area 1, 2 & 3 which contribute to nodes in the current active system. Typically, area 1, 2, and 3 are assigned to be Onsite Impervious (pavement), Onsite Pervious (grass), and Offsite respectively. These values are the defaults for all nodes in the current system. However, each node, using the Node Edit window, has the ability to override these values.

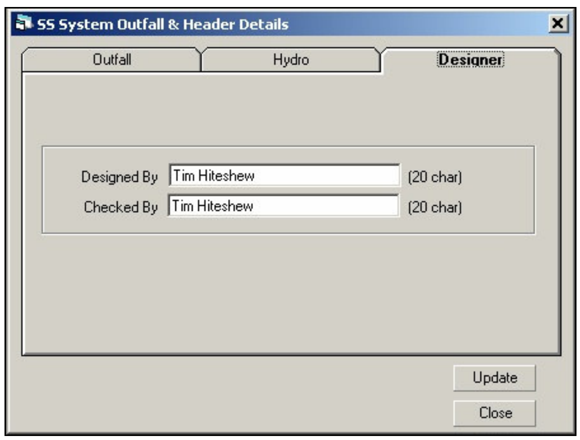


Figure 3.7

Designed By & Checked By: These two fields are used for documentation only. They will be printed on the storm tabulation report as well as other ASAD reports.

3.1.3 Dividing and Combining Systems

ASAD provides this tool for dividing a single storm sewer system into two separate systems (see figure 3.6) and combining two systems to make one (see figure 3.8).

To divide a system into two, by first, creating a new system and, second, by moving selected nodes (with their coincided reaches) into the new system.

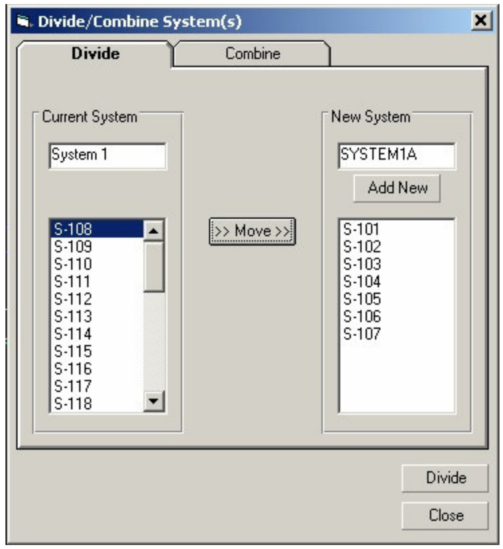


Figure 3.8

Add New: First, click on the Add New button to create a new storm sewer system. The new system name must conform to the ASAD object naming convention.

>> Move >>: The Move button will move the currently highlighted node from the Current System's list box to the New System's list box.

Current System: Select nodes in the Current System list box to be moved to the new system. Double clicking a node will automatically move it.

Apply: The apply button will start the separation routine. This routine will scan the list boxes and perform the node moving operation.

To combine a two systems, into one, simply select the 'From' system, then select the 'Combine to' system, and then press the 'Combine' button.

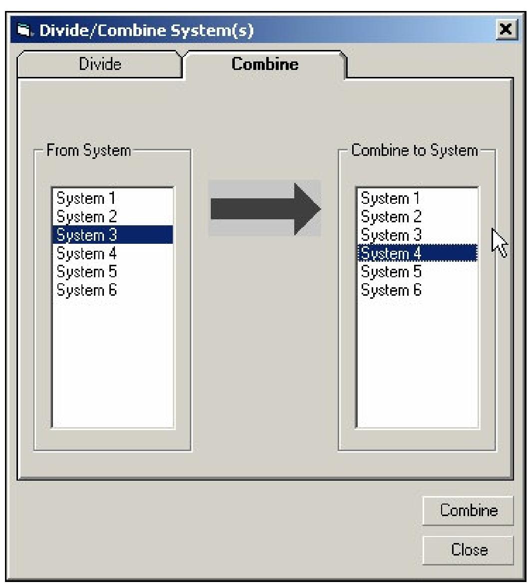


Figure 3.9

Once the process, dividing or combining, has completed, the user will need to reconnect some reach connections as well as review the outfall details.

3.2 NODE/REACH AUTOMATIC CREATION

ASAD provides a tool to allow the creation of multiple nodes at one time. This dialog window is the 'Node/Reach Automatic Creation'. This window is used to add multiple nodes and their outgoing reaches to the current active system quickly. Typically used during the initial entering of a node and reach data, it is a very fast way of creating 5, 10, 25, or as many nodes as you need. This routine can create Structure and Ditch type nodes. Type 'Structure' is shown in figure 3.10.

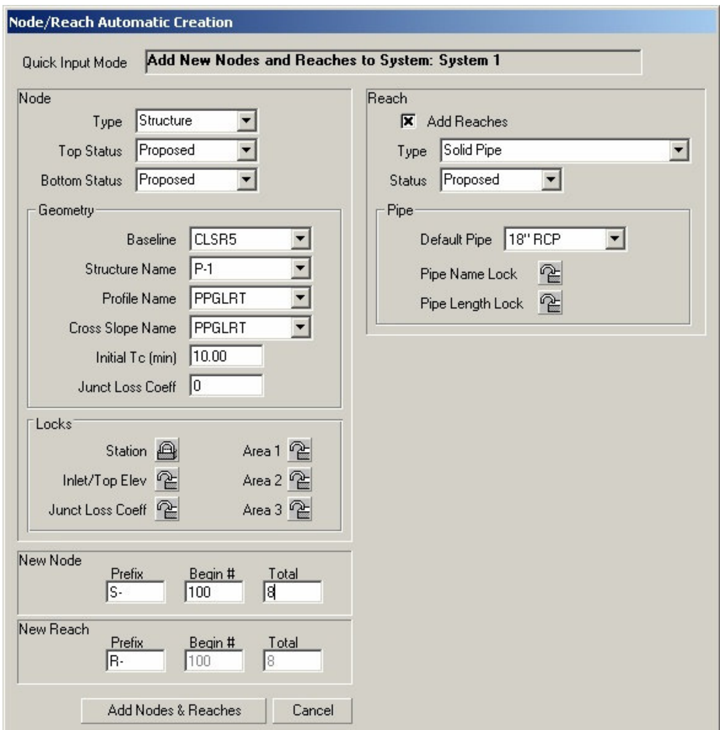


Figure 3.10

Type (Node): The node type must be either Structure or Ditch.

Top/Bottom Status (Node): Select the status for the structure top and bottom. Selections include: Existing, Proposed, and Future 1, 2, 3 & 4. This field is when computing quantities and drawing level/symbology

Baseline: The baseline field (along with the node's station and offset) is used to compute the node's location, i.e. x and y coordinates. Baselines can be added, modified, and deleted using the Baseline Edit window.

Structure Name: Select the type of drainage structure. New structures can be added to the list or existing structures can be modified using the Structure Definition window. By selecting a structure you are assigning to the node certain structure characteristics which are embedded in the structure definition.

Profile Name and Cross Slope Name: These pulldown boxes contain the associated profile and cross slope definitions for the current node. These two values are used by the 'Compute SS Geometry and Drainage Areas' window to compute: Low point station adjustment, node Mirror setting, Inlet/top/grate elevation, and onsite drainage areas

Initial Tc (min): Initial Time of Concentration (Tc). If left blank, then ASAD assumes 10 minutes for hydraulics calculations.

Junc Loss Coef: To compute a node's minor (junction) losses, a number greater than 0 should be entered into this field. The typical range for this coefficient is 0.0 (no loss) to 1.0. See the Engineer's Reference for coefficient details

 **Station Lock:** Set the station lock to 'Unlocked' only when you want the 'Compute SS Geometry and Drainage Areas' window to change the station value to the nearest low point station in the profile. Otherwise leave the station lock set to 'Locked'. The default is 'Locked'

 **Inlet/Top Elev Lock:** Setting this lock to 'Unlocked' allows you to automatically compute a nodes inlet elevation using the 'Compute SS Geometry and Drainage Areas' window. Locked or unlocked, you can always manually enter a value.

 **Area 1, 2 & 3 Lock:** If an area lock is Locked then its associated area can only be changed manually. Unlocking the lock will allow an area to be changed by the 'Compute Storm Sewer Geometry and Drainage Areas' window as well as changed manually.

Prefix (New Node & New Reach): Use these fields to create node names and reach names with a prefix. A typical prefix for a node is 'S-'. A typical prefix for a reach is 'R-'.

Begin # (New Node & New Reach): This field contains the starting number of the nodes and reaches to be created.
Total: The number of new nodes and reaches to create.
Add New Nodes & Reaches (button): This starts the process of creating new nodes and reaches. The node and reach names have the format Prefix + Begin #. After the first node and reach are created the node and numbers are incremented by one.

The top-right section of the Node/Reach Automatic Creation window (see figure 3.10) defines the characteristics of the reaches to be created. Figure 3.11 shows the options available for both reach types 'Solid Pipe' and 'Perforated Pipe w/Trench'. Figure 3.12 displays the fields to be defined when creating reaches of type 'Ditch'.

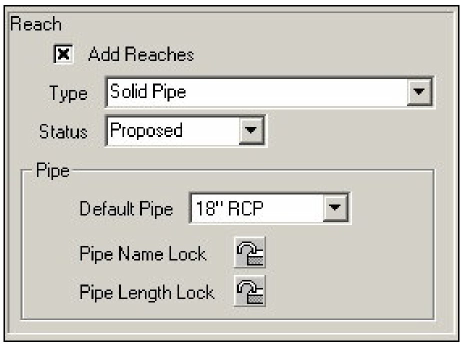


Figure 3.11

Add Reaches: This checkbox determine whether new reaches are created when the nodes are created. If this is off, then creation routine will create nodes only.
Type: Select the type of reaches to be created. Selections include: Solid Pipe, Perforated Pipe w/Trench (French Drain), and Ditch.
Status: Set the status of all of the reaches to be created. Selections include: Existing, Proposed, Future 1, 2, 3, or 4.
Default Pipe: Set the pipe that the new reaches will initially be defined with.
Pipe Name & Pipe Length Lock: Set the lock status for the reaches to be created.

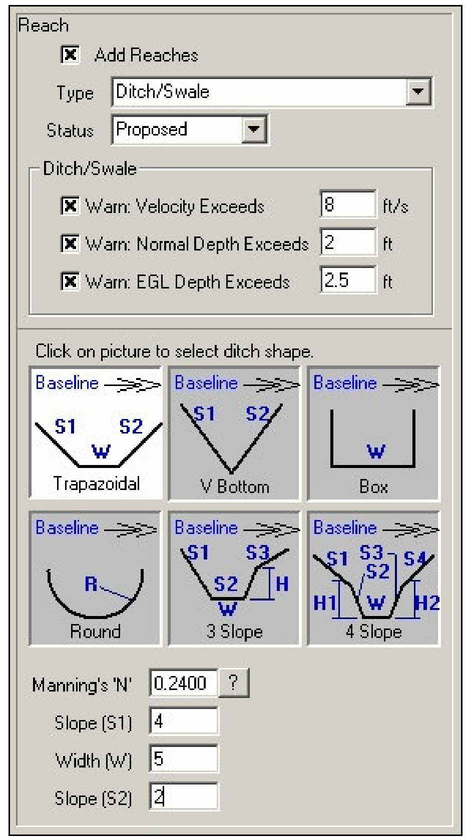


Figure 3.12

The three warning settings, in the 'Ditch/Swale' section of figure 3.12, are used by the Compute Hydraulics window. If, during the hydraulic computations, any of these three values are exceeded, a warning noting the problem will be displayed in the Results window of the Compute Hydraulics window.
Warn: Velocity Exceeds: The maximum velocity allowable in the ditch section.
Warn: Normal Depth Exceeds: The maximum normal depth of flow allowable in the ditch section.
Warn: EGL Depth Exceeds: The maximum energy grade line (normal depth + velocity head) depth allowable in the ditch section.

The bottom half of figure 3.12 defines the geometry of the ditches to be created. Note the location of the baseline in relation to the shape of the disk. This is important when the ditch is drawn in plan view. Click on the desired ditch shape and the appropriate data fields will be displayed.
Manning's 'N': Enter or select from a list, by clicking on the '?' button, Manning's 'N' value based on the ditch surface type.
Slope: Enter the cross slope run (horizontal distance) for a rise (vertical distance) of 1. For instance a value of 4 indicates a 4:1 slope (4' of run per 1' of rise). Typical values are 1, 2, 4, 6 and 10.
Width: Bottom width of the ditch.
Height: This value indicates a change in the cross slope at a Height above the ditch bottom.

3.3 NODE EDIT

A node is a structure or position which is used to connect reaches, define runoff areas, define elevations and structure characteristics. The node is the cornerstone in the layout of a drainage system. In previous versions of ASAD a node had to be a structure such as a manhole, junction box, curb inlet, gutter inlet, ditch bottom inlet, and endwall. ASADv3 will continue to use these structures and they will be referred to as type 'Structure'. However, ASADv3 can also include ditches in its design. To facilitate this ASAD now recognizes a second type of node called 'Ditch'. This type of node is used to position ditches as well as introduce overland flow to a ditch. As before all nodes in a system are attached by pipes and/or ditches (reaches) to other nodes and eventually to the outfall node. ASAD allows up to 1000 nodes per system.

3.3.1 Editing Nodes: Type 'Structure'

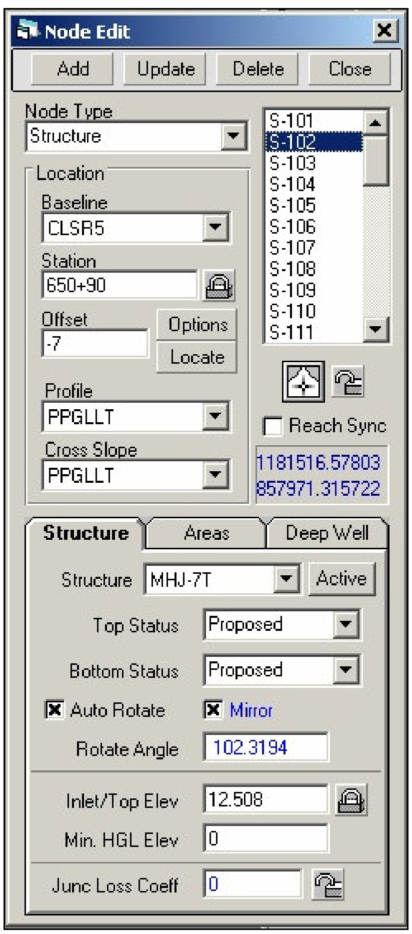


Figure 3.13

Node Type: The node type is either Structure or Ditch
Node List: The list on the right side of the window is the Node name list box. The Node name list box shows all nodes (except the outfall node) that are in the current active system. Many users use a prefix (S-) and a number when numbering their storm systems (i.e. S-231).
Add: Click on the Add button to create a new node. After entering the new node, you can fill in its data fields. The node name must conform to the ASAD object naming convention.
Update: After entering new data or modifying existing data, click on the update button to save it to the database.
Delete: Use the Delete button to delete individual nodes one at a time.
Note: To delete the entire system including outfall and header details, use Storm Sewer Add/Delete.

Baseline: The baseline field (along with the node's station and offset) is used to compute the node's location, i.e. x and y coordinates. Baselines can be added, modified, and deleted using the Baseline Edit window.
Station: The station field (along with the offset) when applied to baseline geometry, is used to compute the node's location, i.e. x and y coordinates. The required format is xxxx+xx.xx (i.e. 327+28.12). This field is also printed on various reports including the storm tabulation report.
 Set the **station lock** to 'Unlocked' only when you want the 'Compute SS Geometry and Drainage Areas' window to change the station value to the nearest low point station in the profile

Otherwise leave the station lock set to ‘Locked’. The default is ‘Locked’

Offset: The offset field (along with the station) when applied to baseline geometry, is used for computing a node’s location, i.e. x and y coordinates. Denote a left-of-centerline offset as negative(-), while the right-of-centerline offset is positive (plus sign not necessary). An offset directly on the centerline is 0.This field is also printed on various reports including the storm tabulation report.

Options & Locate: These two buttons work together to allow the user to populate the Station and Offset fields with the location of a data point selected in the CAD window. The Locate button works as a DataPoint button. Click on Locate, then click on the CAD window and the Station and Offset fields will be filled, based on the options setup. The Options button invokes the Locate Node – Options window (see figure 3.14). The options window allows the user to set how the data point will be interpreted.

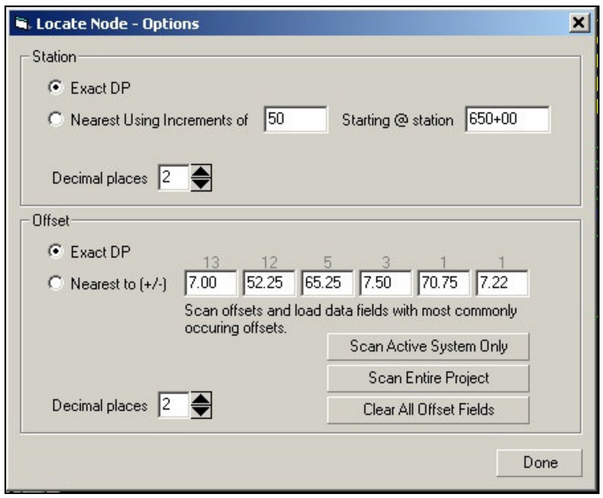


Figure 3.14

Exact DP (Station): When this option is selected, the Locate function returns the exact station to the Station field.

Nearest Using Increments of: This option tells the Locate function to return a value, to the station field, rounded to the nearest increment set from a starting station. In the example above, a user clicks on the CAD window at station 658+33.27 but the value is rounded up to 658+50.00. If the user had clicked at 658+23.04 then the station value would be rounded down to 658+00.00. This is useful, when first setting up a drainage system, to initially locate structures on even stations (every 100 or 50 feet). Just click close to where the structure should be and let ASAD round it to a neat and tidy number.

Decimal Places (Station and Offset): Controls the number of decimal places for each value returned to the Station and Offset.

Exact DP (Offset): When this option is selected, the Locate function returns the exact offset to the Offset field.

Nearest to (+/-): This option sets the rounding values for the offset. The six fields to the right contain desirable offset values that the Locate function will round to. When clicked, the datapoint location will be converted to an offset and then compared to the positive (+) and negative (-) number for each value listed. The routine will then select the closest offset and fill-in the Offset field in the Node Edit window with that value. This is useful in two different scenarios:

(1) When setting up a new system, the user may determine that the typical offset for many of their curb inlets is -52.25 and 52.25 for inlets on the left and right side outer edge of pavement. The user also knows that they have a ditch that will require ditch bottom inlets at an offset of -78.5. By entering in values of 52.52 and 78.5, they can now just click close to the location and ASAD will round it to the correct offset.

(2) A user wanting to change some node offsets or add new nodes to the same offsets, as similar nodes, can **Scan Active System Only** or **Scan Entire Project** to accumulate the most popular offsets. In the example above, the first field containing 7.00 indicates that offset occurs 13 times (note the light-grey #13 above the field) while the second field containing 52.52 had 12 occurrences and so-on. Now the user need only click close to 52.25 to have 52.25 be placed into the offset field.

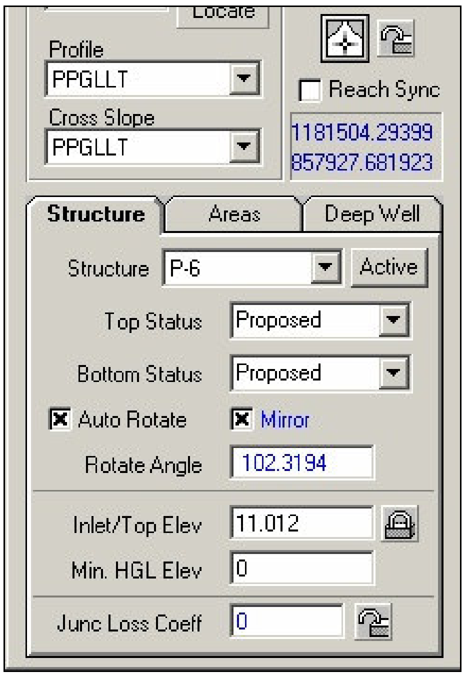


Figure 3.15


The leftmost of these two buttons is a node window center button. When the drawing window is open, pressing this button will relocate the drawing window, centering around the currently highlighted node in the node name list box. Press this button each time you want to center the window about a node. The rightmost button (actually a lock) is used to lock the window center button on, so that when anytime a node is selected from the node name list box, it is automatically displayed and centered in the drawing window.

Reach Sync: If the Reach Sync switch is on AND the reach edit window is open, then as you click from node to node in the list, the coincidental reach in the reach edit window will be displayed. This will let you view the Node/Reach combination by simply clicking on the node only.

Profile Name and Cross Slope Name: These pulldown boxes contain the associated profile and cross slope definitions for the current node. These two values are used by the 'Compute SS Geometry and Drainage Areas' window to compute: Low point station adjustment, node Mirror setting, Inlet/top/grate elevation, and onsite drainage areas

Structure Name: Select the type of drainage structure. New structures can be added to the list or existing structures can be modified using the Structure Definition window. By selecting a structure you are assigning to the node, certain structure characteristics which are embedded in the structure definition.

Top/Bottom Status: Select the status for the structure top and bottom. Selections include: Existing, Proposed, and Future 1, 2, 3 & 4. This field is for when computing quantities and drawing level/symbology.

Rotate Angle, Mirror, and the Auto Rotate switch: The rotate angle field is used by the Draw Plan View routine to display the inlets, manholes, ditch bottom inlets, etc., at the proper rotation angle in relation to baselines and other structures. Values range from 0-359.999

The mirror value (on or off) has an effect only on non-symmetrical inlet tops such as FDOT inlet tops type 1, 3, and 5. These inlets are intended to accept runoff flow from only one direction, unlike their cousins, types 2, 4, and 6, which accept flow from both directions. These non-symmetrical structures may be mirrored to show (in plan view) them accepting flow from the appropriate (upstream) direction.

If the auto rotate switch is ON, then both the rotate angle and mirror setting can be computed by the 'Compute SS Geometry and Drainage Areas' window. If the switch is off, then these values can only be changed by manually changing them.

Inlet/Top Elev: The field represents the inlet, grate, or manhole top elevation. In the case of an endwall or mitered end section, this field should be equal to or greater than the pipe crown line elevation. This field is very important for computations such as setting flow lines and computing HGL in to inlet clearance. This field is also used for the Node/Reach/HGL profile viewing on the Profile tab of the Compute/Print Storm Tabs window. If the adjacent lock is 'Unlocked', then this value can be changed by the 'Compute SS Geometry and Drainage Areas' window. If the lock is 'Locked', then this value can only be changed by manual entry.

Min. HGL Elev: Use this field to raise the Hydraulic Grade Line to a specified elevation at this node. Typically this is used to simulate a wall inside a structure which raises the HGL to produce more head on upstream French Drains.

Junc Loss Coef: To compute a node’s minor (junction) losses, a number greater than 0 should be entered into this field. The typical range for this coefficient is 0.0 (no loss) to 1.0. See the Engineer’s Reference for coefficient details.

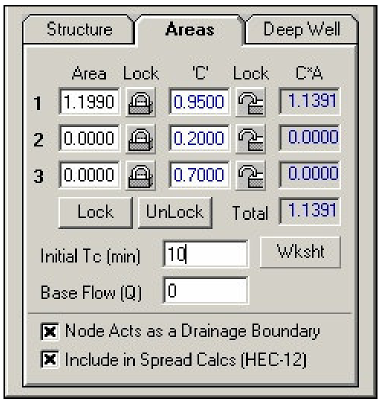


Figure 3.16

Area 1, 2 & 3 and Lock: Enter the runoff area collected by this node. Typically Area 1 represents onsite impervious (pavement), Area 2 is onsite pervious (grass), and Area 3 is offsite. If the lock is Locked, then the adjacent area can only be changed manually. Unlock the lock and the adjacent area can be changed by the ‘Compute Storm Sewer Geometry and Drainage Areas’ window as well as changed manually.

‘C’ and Lock: This value allows the user to override the system-wide defaults for runoff coefficients ‘C’. First, key in the desired ‘C’ value for each area to change. Then Lock the lock to the right of the ‘C’ column. If the lock remains Unlocked, then the ‘C’ value will revert back to the system-wide defaults as entered in the Storm Sewer Outfall and Header Details window.

Lock and Unlock buttons: These two buttons Lock or Unlock all six locks on this tab.

Initial Tc (min): Initial Time of Concentration (Tc). If left blank, then ASAD assumes 10 minutes for hydraulics calculations.

Wksht button: Clicking on the Wksht button will initiate the Tc Worksheet window. Upon returning from the worksheet window, the new Tc value may be placed automatically in the Initial Tc field. This worksheet is based on TR55 methodologies. See Time of Concentration Worksheet for more details.

Base Flow (Q): Base Flow is that flow that cannot be directly associated with an area. Some, but not all, possibilities include: (1) accepting an existing flow where only the flow rate is known, (2) where the flow is computed using a method other than the rational method, (3) where a node is accepting flow produced from under drain collection or other groundwater conditions.

Node Acts as a Drainage Boundary: This switch relays to the ‘Compute Drainage Areas’ routine of the ‘Compute SS Geometry and Drainage Areas’ window, whether or not this node is a drainage boundary. For instance, curb inlets usually indicate a drainage boundary, therefore in this case the switch should be ON. However, manholes that do not intercept flow and usually do not represent a drainage boundary, should be set to OFF.

Include in Spread Calcs (HEC-22): This switch is set to ON for those nodes that are determined be intercepting roadway runoff flows, such as curb inlets and gutter inlets. OFF is the appropriate setting for structures not intercepting roadway runoff, such as manholes, endwalls in ditches, ditch bottom inlets, etc..

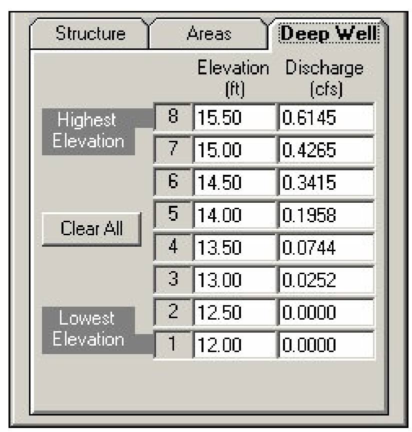


Figure 3.17

Elevation: Enter the elevation in feet above mean sea level.

Discharge: Enter the discharge rate in cubic feet per second (cfs)

3.3.2 Editing Nodes: Type 'Ditch'

The node edit window (see figure 3.18) for Ditch nodes is very similar that of Structure type nodes. Many of the fields are common to both types. To avoid being redundant, we have left the common fields out of this section. Refer back to Section 3.3.1 for their definition(s).

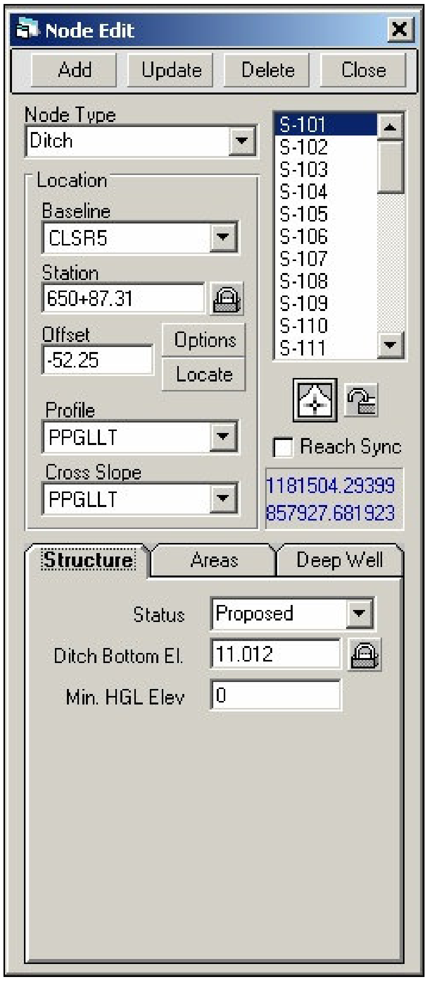


Figure 3.18

Status: Set the status for this node as: Existing, Proposed, Future 1, 2, 3, or 4. A node's status is used by ASAD's drawing routines to set the level/symbology based on the status.
Ditch Bottom El: Set the ditch bottom elevation using this field.
Min. HGL Elev: See Section 3.3.1

The Areas Tab for a Ditch node is the same as a Structure node (see figure 3.19).

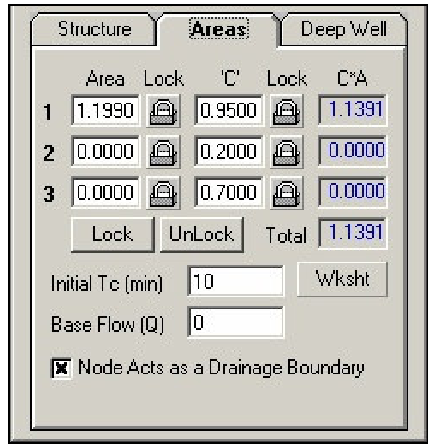


Figure 3.19

3.3.3 Defining Node Structure Characteristics

Each node is provided with drainage structure characteristics for geometry, size, elevation and hydraulic capacity in the "Structure" portion of the "Node Edit" dialog. The first characteristic that the user must define is the Structure Name. This determines the inlet type from the FDOT *Roadway and Traffic Standard Index, 200 Series*. The inlet top and structure bottom are selected from the pre-defined list of inlet types. Custom inlet types can be created. This topic will be discussed in detail in section 3.6.

The designer must also select the corresponding Profile Name (see section 2.2) and the Cross Slope Name (see section 2.3) from the previously defined list of each.

If a junction loss coefficient is deemed necessary by the designer for any specific node, it is entered in the key-in field provided and can be locked by the designer.

The Inlet/Top Elevation can be manually set by the designer if design considerations dictate so. This value can also be locked to prevent adjustment during the geometric computations.

3.3.4 Developing Hydrologic Basin Areas and Calculating Tc

Most nodes will have contributing area. These areas are defined as Area 1, 2, and/or 3. These values can be entered manually or automatically computed in the Compute Storm Sewer Geometry and Drainage Areas window assuming Cross Width data has been entered. The information entered in the "Edit Cross Widths" dialog (see figure 2.9) is not accurate when dealing with significant amounts of offsite runoff. The runoff basin areas can be adjusted in the "Areas" tab of the "Edit Nodes" dialog (see figure 3.20).

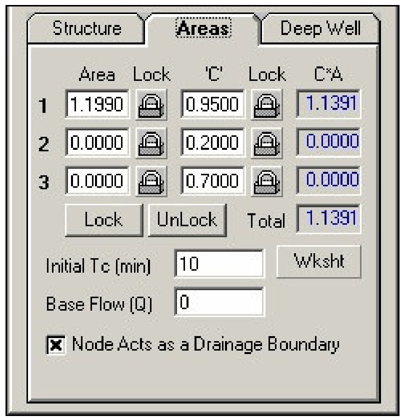


Figure 3.20

The next requirement for the node is the initial time of concentration (Tc). This value can either be entered as an estimate by the designer, or based on the TR-55 method for calculating Tc from the ASAD worksheet (see figure 3.21). In order to calculate Tc using the ASAD worksheet, the designer must supply the information requested by the worksheet. A library of Manning's 'N' values is furnished to the designer. This library can be utilized by clicking on the question mark icons. As the designer fills in the necessary information into each section of the worksheet, the Tc for each segment is calculated, the designer must press the "Accept" button which saves the values entered into the worksheet and also fills in the Tc in the "Edit Node" dialog, or "Cancel" to discard them.

The Tc worksheet has four tabs; Sheet Flow (see figure 3.21), Shallow Concentrated Flow (see figure 3.22), Channel Flow (see figures 3.23 and 3.24), and an Optional worksheet for additional segments (see figure 3.25). The Sheet Flow tab has two different methods for processing this T_t. The TR-55 and the Kinematics Wave functions are available to the designer.

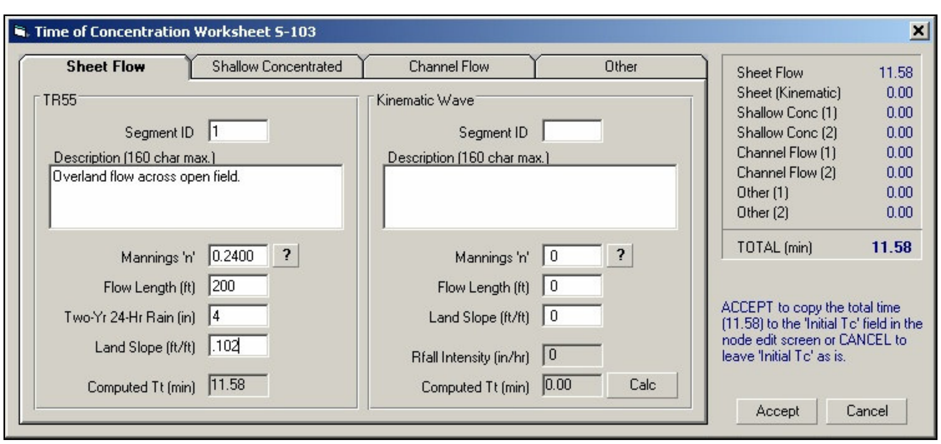


Figure 3.21

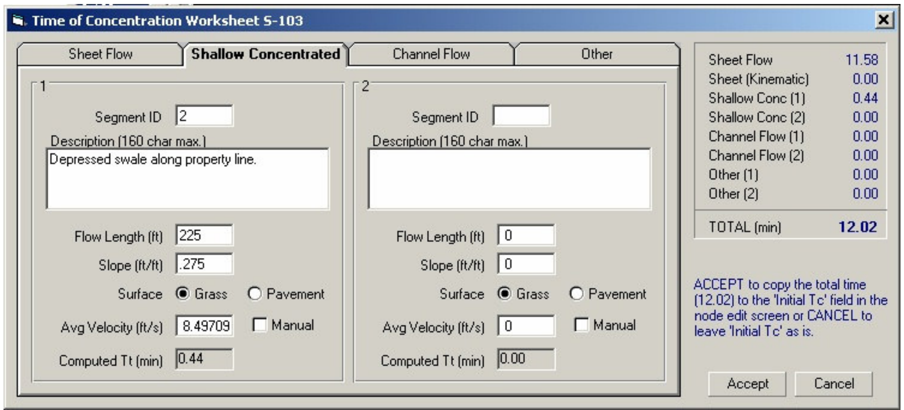


Figure 3.22

In the Channel Flow tab, the cross section area and wetted perimeter can be calculated by a channel geometry tool (see figure 3.24). Start the tool by clicking on the '?' button to the right of the 'XS Area' and 'Wet Perim' fields. This tool calculates the area and wetted perimeter based on the channel geometry and dimensions. The geometry is determined by the designer by selecting the geometry from one of three tabs in the dialog. The choices are trapezoidal, V-shaped, and rectangular channels. After the channel type is selected, the designer keys-in the dimensions of the channel. Once the necessary dimensions are entered the area and wetted perimeter are displayed. The designer must either "Accept" or "Cancel" the results.

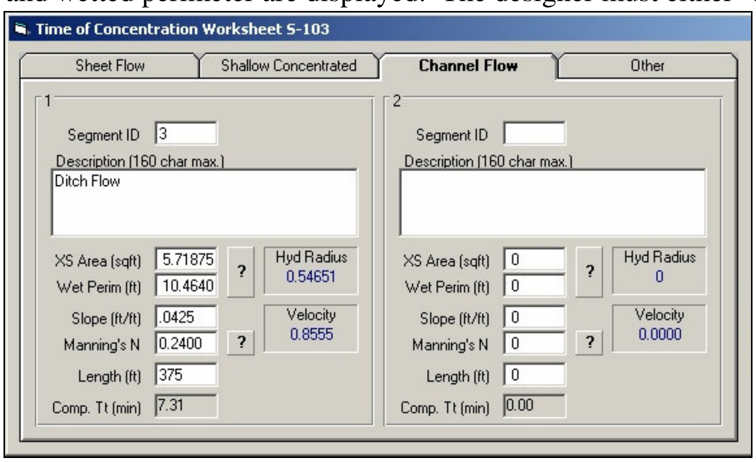


Figure 3.23

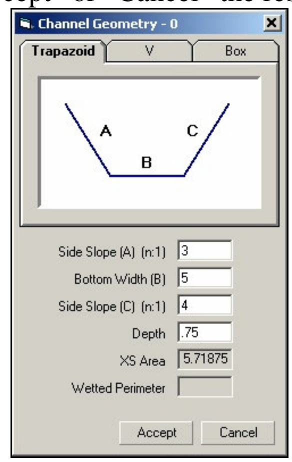


Figure 3.24

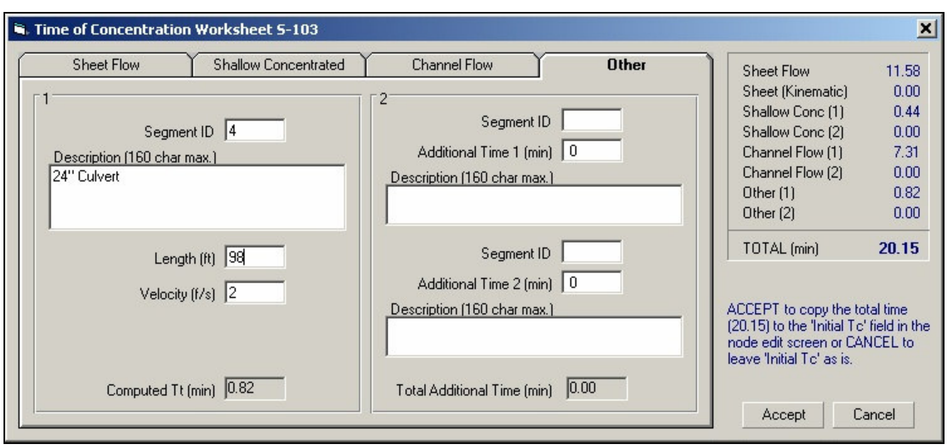


Figure 3.25

3.4 REACH EDIT

The reach provides the link between the nodes and ties the system together. There are 3 types of reaches: (1) Solid Pipe, (2) Perforated Pipe with Exfiltration Trench, and (3) Ditch/Swale. Some data fields are common to all three types. They are: Reach Name, Status, From Node, and To Node. All other fields are specific to their own reach type. The reach edit window changes size and appearance based on the reach type. The next four sections cover: (1) Editing Reaches – Data Common to All Types, (2) Editing Reaches – Solid Pipes, (3) Editing Reaches – Perforated Pipes w/Exfiltration Trench, (4) Editing Reaches – Ditches & Swales.

3.4.1 Editing Reaches – Data Common to All Types

Reaches are created by selecting the "Add" button located along the top of the dialog. The designer can then enter the reach name which will be displayed in the list box on the right side of the dialog. Select the desired reach to edit from the list.

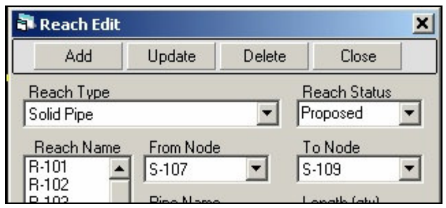


Figure 3.26

Reach Name: The reach name has to be a unique (used only once) name in the entire project database. Typically the reach name has a prefix of 'R-' and the numerical portion is very similar to the upstream node to which it is connected. For instance node S-231 would have an outgoing reach R-231. Since each node has only one outgoing reach, this naming convention has become standard in ASAD projects.

Note: Valid names in ASAD contain a maximum of eight alphanumeric characters.

Reach Type: There are 3 types of reaches: (1) Solid Pipe, (2) Perforated Pipe with Exfiltration Trench, and (3) Ditch/Swale.

Reach Status: Select from the pulldown list: Existing, Proposed, Future 1, Future 2, Future 3, Future 4. This field may be used by other windows to determine if the reach should be included in the summary of quantities or the level/symbology to draw in CAD. For example, setting a reach's status to 'Existing' may cause the reach to be drawn as a dashed line, while setting a different reach to 'Proposed' causes it to draw as a solid line.

From Node & To Node: The location of a reach is determined by the "From" and "To" nodes which were defined in the "Node Edit" dialog. The designer must choose the "From" and "To" nodes for the reaches. This process determines the location of all the pipes and how they are connected in the system. Since pipe does not, in most cases, bend; the designer might have to insert some manholes into the system. Manholes are treated just like any other node except they usually do not have any drainage area directly associated to them.

3.4.2 Editing Reaches – Solid Pipes

Selecting Reach Type: Solid Pipe will result in the screen below (see figure 3.27).

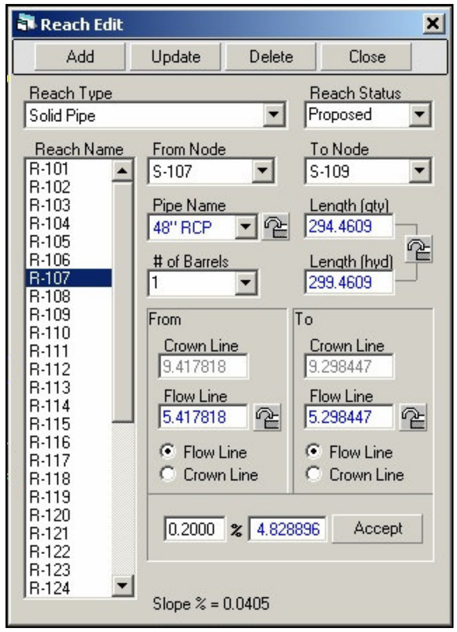


Figure 3.27

Pipe Name: The designer must select the initial pipe name (size) from the list of predefined pipes. This size will probably be adjusted when the hydraulic computations for sizing the pipes are performed (see section 4.2). The pipe lengths are also determined from the hydraulic computations (see section 4.1).

# of Barrels: The user may select from 1 to 10 pipes that make up this reach. These pipe(s) are assumed to be the same size (see Pipe Name) and the have the same flow line elevations at the From node and the To node respectively. When the hydraulics is calculated, the flow is divided evenly between the numbers of barrels. For instance, a total flow (Q) of 9.0cfs travels from the 'From' node to the 'To' node with each pipe carrying only 3cfs.

Length (hyd): This length is measured from the north and east coordinates of the upstream node to the north and east coordinates of the downstream node. This center-to-center length is known as the Hydraulic (hyd) length.

 Leave the Length lock to 'Unlocked' when you want the 'Compute SS Geometry and Drainage Areas' window to compute, and possibly change, both lengths. Otherwise set the length lock set to 'Locked'. The default is 'Unlocked'

Length (qty): This length starts with the Hydraulic length (see Length (hyd) above) and them subtracts some length from both ends based on the type of structure. The amount deducted is defined in the individual structure definitions (see section 3.6). This more realistically represents the actual length of pipe. Because of this, ASAD's quantity calculation (Cost Analysis window, Summary of Drainage Structures window, etc.) routines use the Quantity (qty) length.

Flow Lines & Crown Lines (From & To): The designer can select to key in or let ASAD compute the Flow Line or Crown Line elevation for the To or From ends of the reach. That boils down to a lot of options when it comes to defining pipe elevations.

Let's start with the first part: '…key in or let ASAD compute…'. Leave the lock set to 'Unlocked' (note the adjacent value will display in blue) and ASAD will compute the flow line for you when you run the 'Set Flow Line' window. Otherwise set the lock to 'Locked' (note the adjacent value will display in black) and manually enter the flow lines. The default is 'Unlocked'

Now let's move on to the '…compute the Flow Line or Crown Line elevation…' part. Choose from options 'Flow Line' and 'Crown Line' to select which value you want to key in or compute. By choosing

Flow Line, any changes in the pipe size will change the Crown Line. By choosing Crown Line, any changes in the pipe size will change the Flow Line.

Accept (button): Use this button in conjunction with the text box (black text) to the left of the button to compute a new downstream flow line based on the upstream flow line elevation, the slope keyed into the text box, and the quantity length. Key in the desired pipe slope into the text box (black text). Click on another reach in the reach list and then back to the original reach. A new 'tentative' elevation will be displayed in the other text box (blue text). Pressing the 'Accept' button will copy the 'tentative' elevation into the downstream flow line elevation box.

3.4.3 Editing Reaches – Perforated Pipes w/Exfiltration Trench

Selecting Reach Type: Perforated Pipes w/Exfiltration Trench will result in the screen below (see figure 3.28). The left half of this window is the same as reach type: Solid Pipe. Refer to section 3.4.2 for the definitions. The definitions for the fields in the right half of the window are listed below.

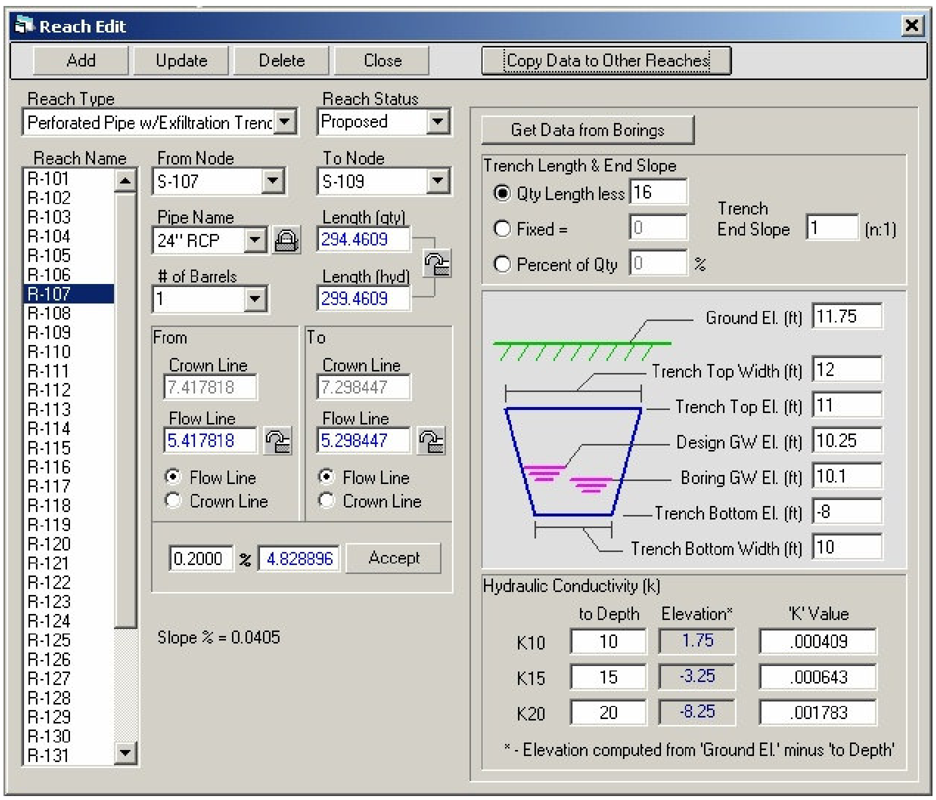


Figure 3.28

Copy Data to Other Reaches (button): This button invokes another window (see figure 3.29) that allows the user to select items to be copied and the destinations reaches. Completing the task will copy the specified data from the Source Reach to the Destination Reaches. This is useful when creating or changing multiple segments of a ditch or French drain that have similar characteristics.

Get Data from Borings (button): If you have used the 'Percolation Borings' window to compute K values, then you can quickly load the reach data directly from the percolation boring data.

Qty Length less: Define the French drain length as a function of the Quantity Length. This is the most often used option. In the scenario above (see figure 3.28) the French drain length would be the 294.46' (quantity length) less 16' (solid pipe 8' on both ends) or 278.46'. If the quantity length changes, the French drain length would automatically change.

Fixed (length): The user can key in the exact length of the French drain portion of the reach. The remainder of the pipe is assumed to be standard solid pipe.

Percent of Qty: Similar to the 'Qty Length less' option above, this option sets the French drain length as percentage of the quantity length. The remainder of the pipe is assumed to be standard solid pipe.
Trench End Slope: Define the slope of the trench at the beginning and the end of the French drain. For a 45 degree slope use 1 (1:1), for a steeper slope use .5 (1/2 :1), or for vertical end use 0 (0:1).
Ground El, Trench Top & Bottom Elevations and Widths: Refer to the diagram in the window above.
Design & Boring GW El. The Design Groundwater elevation is usually an estimated elevation based on historical or seasonal high groundwater elevations. The Boring Groundwater elevation is the observed elevation at the time the borings were done.
To Depth: Indicates the depth at which the corresponding 'K' values occur.
'K' Value: The user enters, or it is copied from the percolation borings, the rate at which water moves laterally into the soil.

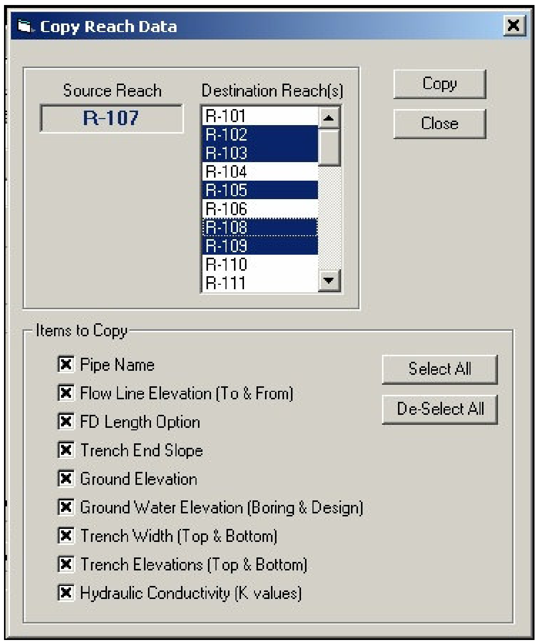


Figure 3.29

3.4.4 Editing Reaches – Ditches & Swales

Selecting Reach Type: Ditch/Swale will result in the screen below (see figure 3.30).

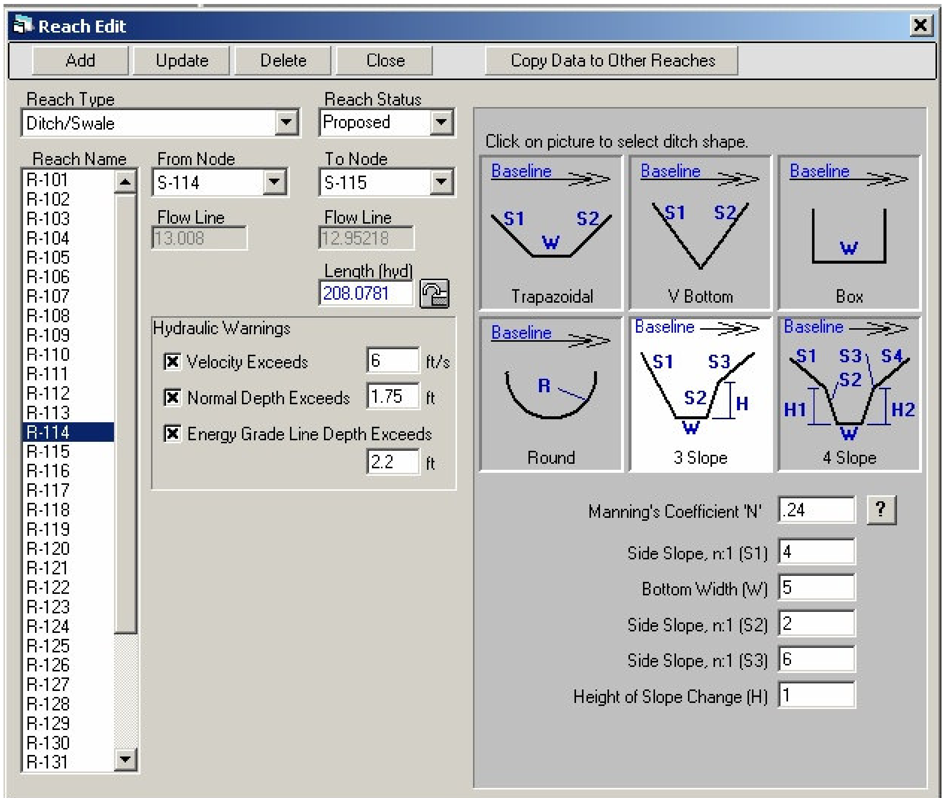


Figure 3.30

The three warning settings, in the 'Ditch/Swale' section of figure 3.30, are used by the Compute Hydraulics window. If, during the hydraulic computations, any of these three values are exceeded, a warning noting the problem will be displayed in the Results window of the Compute Hydraulics window.

Velocity Exceeds: The maximum desirable velocity through this section of ditch. If the velocity is too high, then there is a possibility of erosion of the ditch bottom and sides.

Normal Depth Exceeds: The maximum desirable depth of flow. If the flow is too deep, it may come over the banks of the ditch.

Energy Grade Line Depth Exceeds: This is the combination of the normal depth plus the velocity head. The user would usually want this value to be at or below the top of the ditch bank.

The bottom half of figure 3.30 defines the geometry of the ditches to be created. Note the location of the baseline in relation to the shape of the disk. This is important when the ditch is drawn in plan view. Click on the desired ditch shape and the appropriate data fields will be displayed.

Manning's Coefficient 'N': Friction coefficient for the ditch bottom and banks. Clicking on the '?' button opens up a menu (see figure 3.31) of Manning's settings to choose from or simply enter your own value.

Side Slope: Enter the cross slope run (horizontal distance) for a rise (vertical distance) of 1. For instance a value of 4 indicates a 4:1 slope (4' of run per 1' of rise). Typical values are 1, 2, 4, 6 and 10.

Width: Bottom width of the ditch.

Height: This value indicates a change in the cross slope at a Height above the ditch bottom

Figure 3.31

3.4.5 Percolation Borings

The Percolation Borings window (see figure 3.32) computes the 'K' values that are used in the French Drain calculations. The 'K' values are computed using the boring depth, groundwater elevation, and the rate of discharge into the ground. Florida DOT usually gets the boring information in its raw form and has to compute the 'K' values in-house. This window will do that computation for you. Conversely, most private Engineering firms doing design work for FDOT will receive the soil data from other Geotechnical Engineering firms with the 'K' values already computed. If you already have the 'K' values you can skip this section all together. The equations and terminology used in this window and the French Drain calculation window are based on publication:

SUBSURFACE DRAINAGE WITH FRENCH DRAINS
Florida Dept. of Transportation, District 6, Drainage Section
Prepared by: Ricardo Salazar, PE; Reinaldo Carvajal, PE; Jose A. Gonzalez, EI
Date Prepared: June 20, 1991

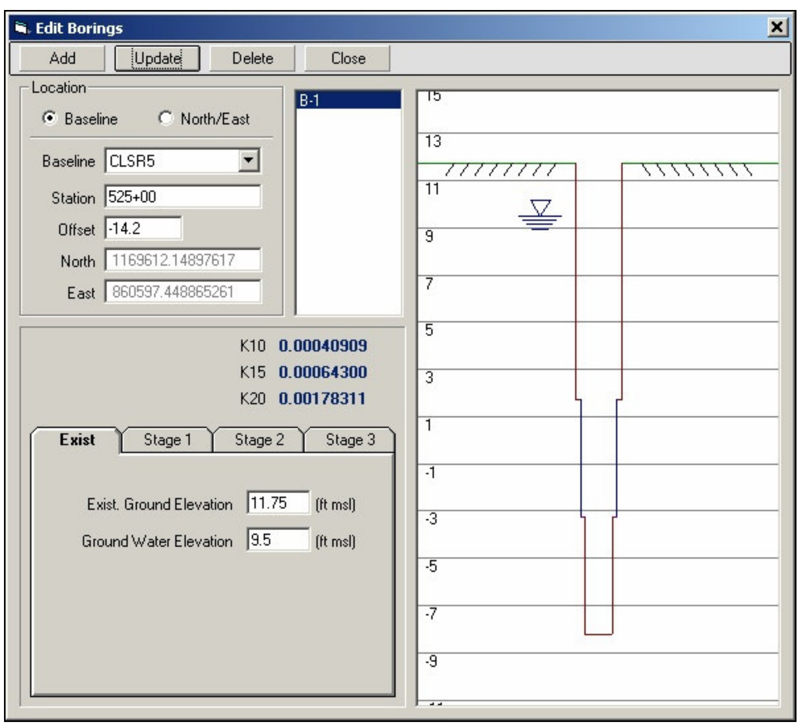


Figure 3.32

Add (button): Use the add button to create a new boring. Once it is created it will be placed in the list and you can enter the values.
Note: Valid names in ASAD contain a maximum of eight alphanumeric characters.
Update (button): Like all other screens in ASAD, simply clicking on a different item in the list will automatically write, to the database, any changes you made to the previous item's data. For this reason you are usually not required to click on 'Update'. However, if you make changes to an item and want to close the window without clicking on another item, them you must click 'Update' to force the update before closing the window. Of course you can click on 'Update' anytime to force an update.
Locate – Baseline or North/East: The location of the soil boring is stored in the North and East fields. These fields can be keyed in directly or they can be computed from the Baseline, Station and Offset.
Baseline: The baseline field (along with the boring's station and offset) is used to compute the boring's location, i.e. North and East coordinates. Baselines can be added, modified, and deleted using the Baseline Edit window.
Baseline: Use this pulldown to select the baseline adjacent to the boring.
Station: The station field (along with the offset) when applied to baseline geometry, is used to compute the boring's location, i.e. North and East coordinates. The required format is xxxx+xx.xx (i.e. 327+28.12).
Offset: The offset field (along with the station) when applied to baseline geometry, is used for computing a boring's location, i.e. North and East coordinates. Denote a left-of-centerline offset as negative(-), while the right-of-centerline offset is positive (plus sign not necessary). An offset directly on the centerline is 0.
North and East: The actual location, on the design plane, of the soil boring.
Exist. Ground Elevation: The existing ground elevation at the location of the boring. This elevation, like all ASAD elevations, is relative to Mean Sea Level (msl).
Ground Water Elevation: The observed actual groundwater at the location of the boring.

The three windows shown in figure 3.33 represent the characteristics of up to three boring depths. When boring to different depths, the shallowest depth must be in the Stage 1 tab, the next lower depth, if there is one, should be in the Stage 2 tab, and the deepest depth, if there is one, should be in the Stage 3 tab.

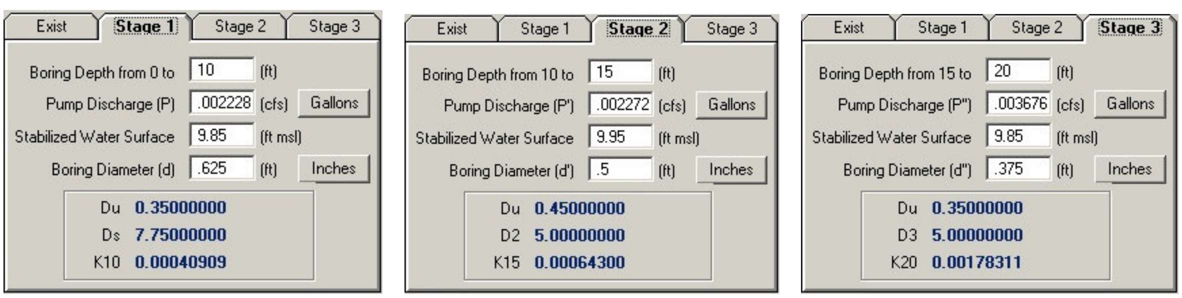


Figure 3.33

Tabs – Stage 1, 2 and 3
Boring Depth from <depth> to <lower depth>: The depth range for a particular stage.
Pump Discharge (P): Rate of discharge, of water into the boring. This value is defined in cubic feet per second (cfs).
Gallons (button): This button opens another window that will convert gallons into cubic feet per second (cfs).
Stabilized Water Surface: During the discharge into the boring hole, the water surface will stabilize at an elevation. This elevation will be above the Ground Water Elevation.
Boring Diameter (d): The boring diameter. Different stages may have different boring diameters.
Inches (button): Opens another window that will convert inches to feet.

3.5 NODE AND REACH RENUMBERING

Over the course of a large drainage project there will be inlets (nodes) inserted and deleted. Invariably there will be nodes with undesirable names or node names, in a sequence, that are not used. It is not unusual to see a system with nodes such as: S-121, S-122, S-122A, S-122B, S-123, S-126, S-127, etc. Even though this is ok during the design process, it makes for a less confusing set of construction plans if the nodes are renamed before final engineering plans are drawn. The System Node/Reach Renumber window (see figure 3.34) will renumber any number of nodes and reaches. The basic sequence for using this window is (1) sort the nodes, (2) fill node and reach number columns with new numbers, (3) check to see if the renumbering would create any duplicate numbers (4) if not then renumber nodes and reaches. In this section, node and reach names will also be referred to as node and reach numbers. They are the same thing.

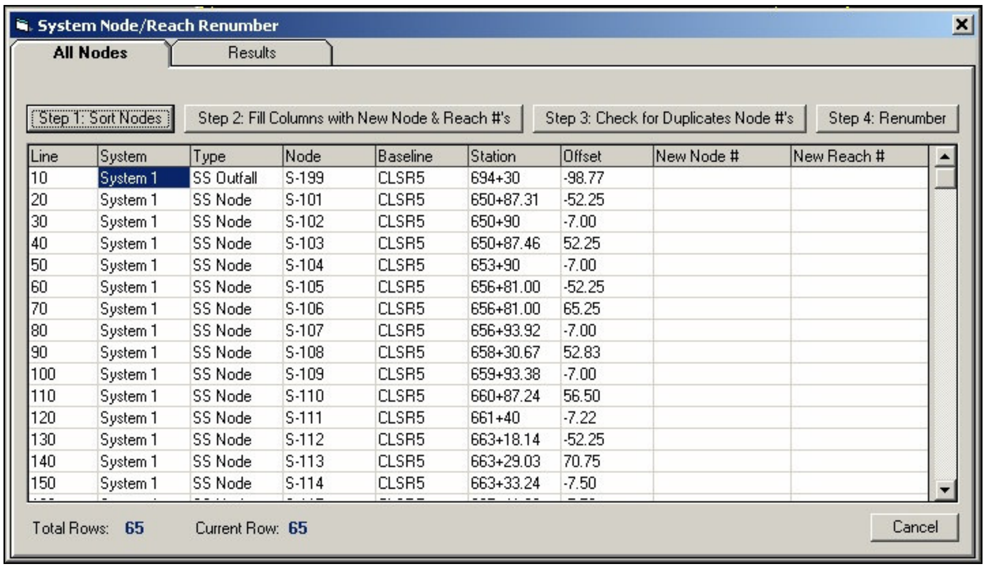


Figure 3.34

Cancel (button): At anytime prior to Step 4: Renumber, the user can press the Cancel button to close the window without making any changes.

Step 1: Sort Nodes (button): This step is optional. This button will open the window below (see figure 3.35). The user has many options and can sort the list of nodes by Baseline, Station, Offset, System, Type, Node name, and/or Line which is the leftmost column. Once the nodes have been sorted, continue to Step 2.

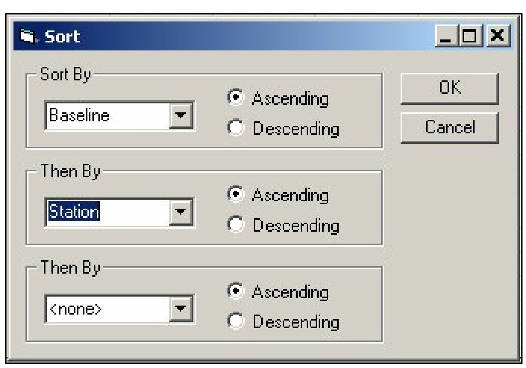


Figure 3.35

Note: Use the leftmost (Line) column for maximum flexibility in renumbering. If the desired order cannot be achieved using typical sort combinations, then the user can fine tune the sort order by keying in different Line values and sorting by Line.

Step 2: Fill Columns with New Node & Reach #'s (button): This step is optional. Before pushing this button to start the fill routine, click on the cell (in the New Node # column) where you wish to begin the renumbering. For this demonstration, the cell in row two has been selected. Now, push the 'Fill Columns…' button. The window below (see figure 3.36) will appear. Notice the 'Current Row' is 2.

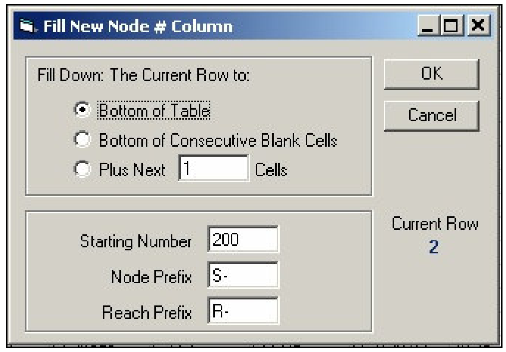


Figure 3.36

Bottom of Table: This option will fill the New Node # and New Reach # columns (see figure 3.34) from the Current Row 2, to the bottom of the table. The first number will be the Node Prefix plus the Starting Number. In this case, S-200 and R-200 will be placed in the New Node # and New Reach # columns of row 2, respectively. Then the number will be incremented by 1 and S-201 and R-201 will be placed in the New Node # and New Reach # columns of row 3, respectively. This procedure will continue to the bottom of the table.

Bottom of Consecutive Blanks Cells: This option is similar to the Bottom of Table option, except this option will stop filling in the node and reach numbers when it encounters a non-blank cell in the New Node # column.

Plus Next X Cells: This option is similar to the two options listed above except it will start at the current row and continue down the table for the next X number of rows.

Starting Number: This the numerical portion of the node and reach names.

Node & Reach Prefix: This is the prefix portion of the node and reach names. Typically these will be 'S-' and 'R-' for nodes and reaches, respectively. Leaving the prefixes blank will cause the fill routine to create node and reach names with the numerical portion only.

Note: As an alternative to using the Sort and/or the Fill routines listed above, the user can simply type in the desired Node Name and Reach Name in the appropriate cells and then run Steps 3 & 4.

Step 3: Check for Duplicate Node #'s: ASAD does not allow duplicate node names or reach names. Therefore, this routine is required to run before the renumbering procedure is started. The results window (see figure 3.37) shown below indicate No duplicates…Continue. Occasionally, you might have a result that looks like figure 3.38 below. It is indicating a duplicate node could occur but does not display the node name. This is always caused by a blank Node Name in an Outfall definition. Looking at figure 3.39, you will see that the node in the top row has no current 'Node' name. Use SS Outfall and Header Details to provide a node name, or delete the system if it is not being used.

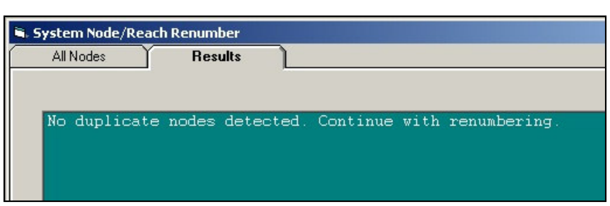


Figure 3.37

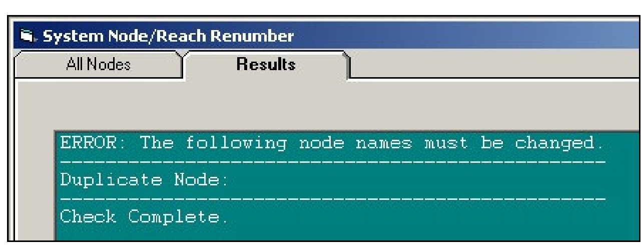


Figure 3.38

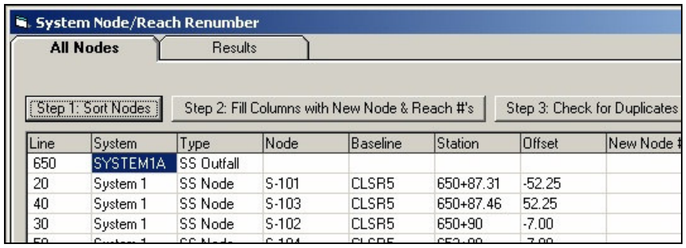


Figure 3.39

Step 4: Renumber: Now that the user has the New Node # and New Reach # cells filled, the renumbering process can be started by pressing the Step 4: Renumber button.

3.6 STRUCTURE DEFINITIONS

The ASAD Database, as created from the ASAD seed database (see section 1.2.1), contains a list of drainage structures. Most of these structures are Florida Department of Transportation (FDOT) structures and were defined using FDOT's 'ROADWAY AND TRAFFIC DESIGN STANDARDS' publication. If the user cannot find the exact structure or the exact version of the structure they are seeking, then the user can 'Add' a new structure and define all of its data from scratch or they can 'Duplicate' a similar structure and modify only the data fields that are different. More on this below.

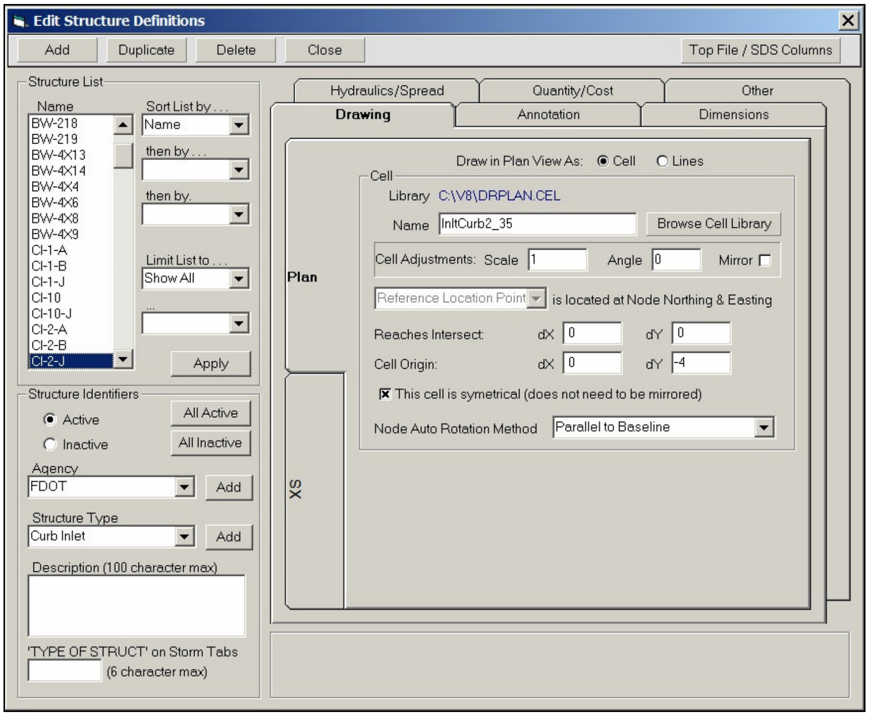


Figure 3.40

This section will discuss only the command buttons at the top of the window and the fields in the leftmost portion of this window. The tabs on the right side of the window are discussed in sections 3.6.1 through 3.6.6

Add and Delete (buttons): The "Add" and "Delete" buttons along the top of the dialog are used to create or remove a specific pipe definition.
Note: Valid names in ASAD contain a maximum of eight alphanumeric characters.
Warning: Deleting a pipe that is used by reaches may cause future 'Missing Pipe' errors. Edit these reaches and make appropriate changes to the 'Pipe Name' field.

Duplicate (button): You may want to create a new structure that is very similar to an existing structure and this button will do just that. It will make an exact copy of the original structure but with a different name that you assign. Why do this? The following scenario demonstrates a common use of the function:
You already have, in your database, a J-1 inlet (inlet type 1 with a J bottom) that has a 6'x6' bottom. But you need a J-1 with a 4'x8' bottom instead. You have two options. (1) you can change the bottom dimensions of inlet J-1 to be 4'x8'. However, you may already be using J-1 in other nodes in which case 4'x8' may not work for those nodes or (2) duplicate the existing structure J-1. Name the new structure J-1_4x8. Now edit structure J-1_4x8 and change to bottom dimensions to 4'x8'. Now you have two different J-1 type inlets in your inventory. Repeat this process again if you need yet another J-1 type inlet with other dimensions.

Sorting and Limiting the Structures Displayed in the List: The ASAD version 3 Seed database has many more structures defined than previous versions. Even though it's nice to have many different structures to choose from, the reality is, you will probably only use 10 to 20 different types on any given project. So if you are only using 10 to 20 structures, why wade through a list of 100 or even 200 different structures on this window (Edit Structures) and the Node Edit window? You don't have too. With ASAD v3, you can limit the number of structures and the order in which they are displayed.
Sort List by…: The sort list function can do a multi-level sort to change the order in which the structures are displayed in the list. Select one of the following fields to sort by: Name, Active, Agency, Type, or Description.
Then by …: then select from those same fields for the second layer sort criteria…
then by …: then select from those same fields again for the third layer sort criteria.
Limit List to…: Next you can select which structures to show in the list. The options are: Show All (default), Active, Inactive, Agency, and Type
Apply (button): Once the fields above have been selected, press the 'Apply' button to initiate the sort and limit process.
Active & Inactive (options): This sets a flag in the structure definition to either Active (default) or Inactive. This flag is used only for determining if a structure is displayed in a list or not. It has no other effect. For instance, a structure named 'C-1-B' is used in many nodes. Even though 'C-1-B' is flagged as 'Inactive', all operations (i.e. drawing, computing hydraulics, computing inlet elevations, etc) ignore the 'Inactive' flag and perform normally.
All Active & All Inactive (button): These buttons will change all structures in the database to either 'Active' or 'Inactive'.
Agency: This field is used to define the agency or organization that a structure is usually associated with. The Seed file is delivered with the agency field for most structures set to 'FDOT'. However, there are a few structures set to 'Jacksonville'. You can use the adjacent 'Add' button to create your own agency to assign a structure to.
Structure Type: The structure type generally groups types of structures by function. This field is usually used to limit the display list of structures. See figure 3.41 for the list Structure Types. You can use the adjacent 'Add' button to create you own structure type to assign a structure to.

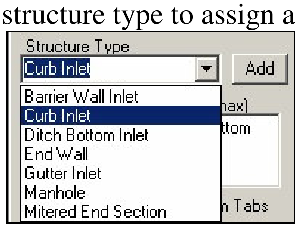


Figure 3.41

Description: Because the structure name is limited to 8 characters, only so much information can be conveyed in the name. The Description field was created to contain a very detailed description of the structure. With a maximum length of 100 characters, there is plenty of room for detail.
TYPE OF STRUCT on Storm Tabs: The structure name on the Storm Tabs may not clearly identify, to the reviewer, what kind of structure it is. The 'TYPE OF STRUCT on Storm Tabs' was added as a short but concise field that will clearly convey, on the Storm Tabs, the structure type. Typical values include: P-1, J-5, MH, MES, FES, DBI, etc. This field is limited to 6 characters.

3.6.1 Drawing (tab)

The drawing tab has two sub-tabs, 'Plan' and 'XS' (cross section). The first part will deal with the 'Plan' view data fields while the second part will deal with the 'XS' view.

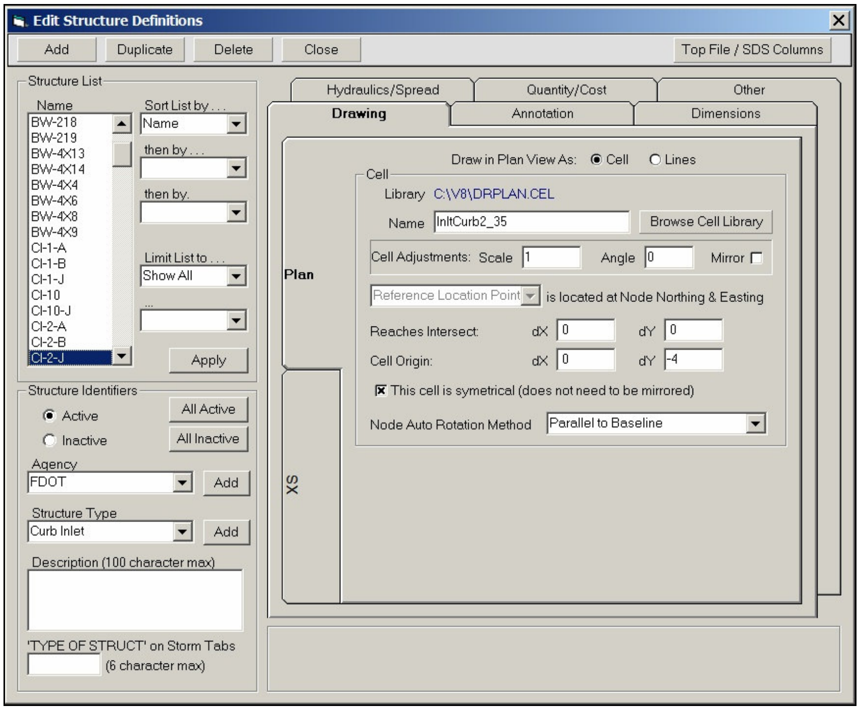


Figure 3.42

Draw in Plan View As: This determines if the structure as drawn in plan view will be a cell (new in version 3) or as individual lines (as in ASAD version 2).
Note: If 'Cell' is selected, then the path to the cell library must be defined in the 'File Locations' window which can be opened using the 'File>Top/Report/Cell Files' pulldown (see section 1.2.2).
Name: If it is drawn as a cell, then it needs a cell name.
Browse Cell Library (button): Use this function to view the attached cell library and pick a cell Name.
Scale: Set the scale the cell will draw at.
Angle: Set the rotation angle the cell will be drawn at.
Mirror: Toggle switch to draw the cell 'as is' or draw it mirrored.
Is located at Node Northing & Easting: This field indicates which point the North and East coordinate (from the Node data) is associated with. It is currently set the 'Location Reference Point' (LRP) and is not changeable by the user.
Reaches Intersect: If the reaches come together at a point other than the 'Location Reference Point' (LRP), then these values set the offset from the LRP. This point is referred to as the 'Pipe Intersect Point' (PIP).
Cell Origin: If the cell origin is a point other than the LRP, then these values indicate the offset from the LRP. This point is referred to as the 'Cell Origin Point' (COP).
Node Auto Rotation Method: This field determines how the structure is to be rotated, using the Compute Storm Sewer Geometry and Drainage Areas routine (see section 4.1), with respect to the baseline or pipe bearing. Set curb inlets, manholes, ditch bottom inlets, etc, to 'Parallel to Baseline'. Set mitered end sections, u-endwalls, flared end sections, etc, to 'Same Bearing as Outgoing Reach' or in other words, in-line with the pipe.

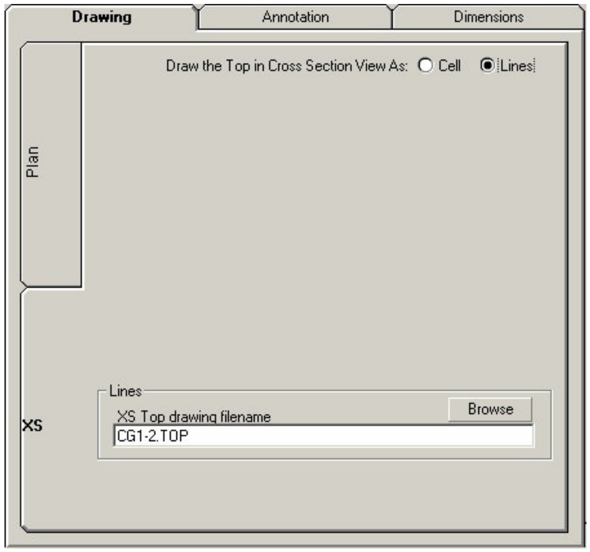
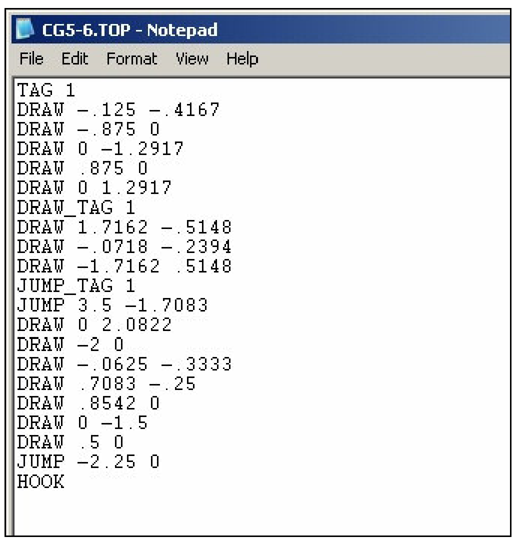


Figure 3.43

XS Top drawing filename: Use the browse button to locate and select a ‘Top’ file. These files are simple text files that contain drawing instructions for drawing the cross sectional view of structure tops. Shown here (see figure 3.44) is a sample of a top file used to draw an FDOT curb inlet type 5 and 6.




```
TAG 1
DRAW -.125 -.4167
DRAW -.875 0
DRAW 0 -1.2917
DRAW .875 0
DRAW 0 1.2917
DRAW_TAG 1
DRAW 1.7162 -.5148
DRAW -.0718 -.2394
DRAW -1.7162 .5148
JUMP_TAG 1
JUMP 3.5 -1.7083
DRAW 0 2.0822
DRAW -2 0
DRAW -.0625 -.3333
DRAW .7083 -.25
DRAW .8542 0
DRAW 0 -1.5
DRAW .5 0
JUMP -2.25 0
HOOK
```

Figure 3.44

3.6.2 Annotation (tab)

This tab is used to define the paragraph that will be placed when the drainage structure is drawn in the ‘Drainage Structure’ sheets.

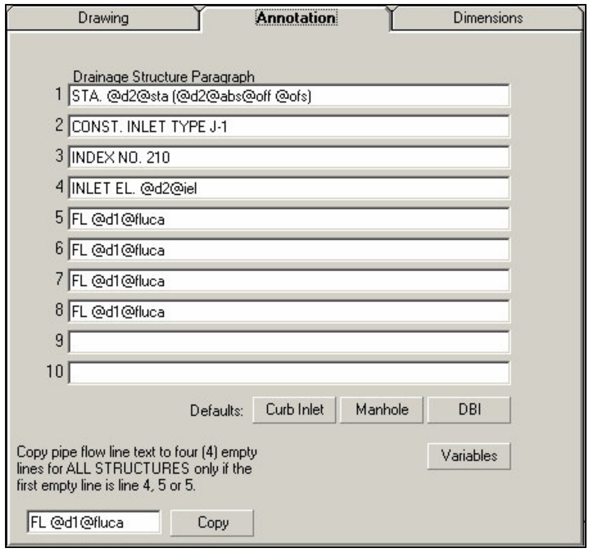


Figure 3.45

Drainage Structure Paragraph: Key in up to 10 lines of text, with variables, to be placed when the drainage structure is drawn in cross section. Note the 4 lines of text (5 to 8) for the flow line labels (FL @d1@fluca). This is needed to label the flow line elevations for the one outgoing pipe and the (up to) 3 incoming pipes.

Defaults 'Curb Inlet', 'Manhole', and 'DBI' (button): Clicking on one of these three buttons will automatically fill in the paragraph with a default write-up.

Copy (button): The copy button is used to fill in the Flow Line labeling for any structure that has none. The adjacent text box contains (FL @d1@fluca in this case) the text to be copied.

Variables (button): The variables (see figure 3.46) are text in the paragraph write-up that are replaced with the actual data when the paragraph is drawn in the drainage structure sheet. For instance, if a node had label text (FL @d1@fluca), it would place in the drawing with ***FL 32.6 BK,*** if the pipe going ahead of the node had a flow line elevation of 32.6123. Note the @d1 says whatever number follows, round it to 1 decimal place. Also note, the @fluca says replace with upper case ***AH, BK, LT*** or ***RT*** based on the direction of the pipe.

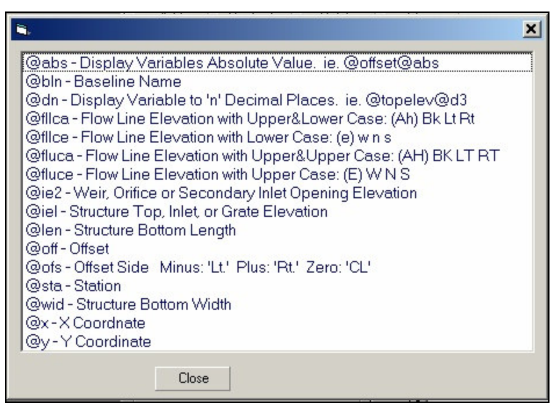


Figure 3.46

3.6.3 Dimensions (tab)

Based on the options selected in the 'Structure Top/Bottom Relationship' section of the window below (see figure 3.47), the bottom portion of the window will change to reflect the data required.

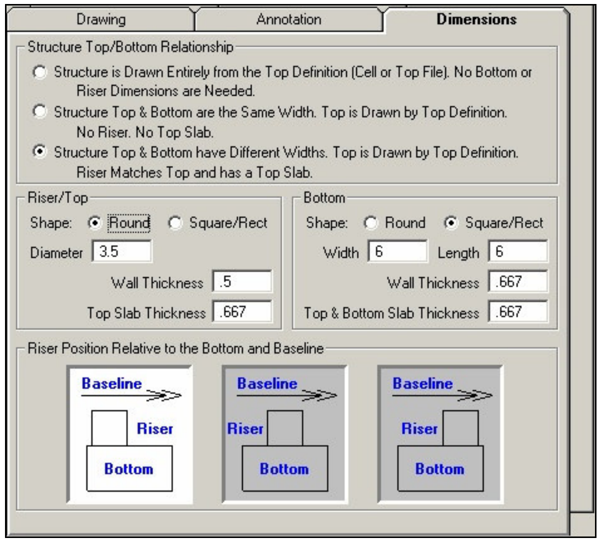


Figure 3.47

Structure Top/Bottom Relationship (options):

Structure is Drawn Entirely from the Top Definition (Cell or Top File). No Bottom or Riser Dimensions are Needed: Use this option when defining an endwall such as a mitered end section or u-endwall.

Structure Top & Bottom are the Same Width. Top is Drawn by Top Definition. No Riser. No Top Slab: Use this option when defining most P bottom structures such as P-1's or P-8's or DBI type D, or E, etc.

Structure Top & Bottom have Different Widths. Top is Drawn by Top Definition. Riser Matches Top and has a Top Slab: Use this option when defining most J bottom structures such as J-1's or J-8's or DBI type D with a J bottom, etc.

Riser/Top:

Shape (options): The shape, Round or Square/Rectangular, of the riser or top.

Diameter: If the shape is Round, then enter the inside diameter of the riser or top.

Width: If the shape is Square/Rectangular, then enter the inside width dimension of the riser or top as seen in the cross sectional (i.e. drainage structure) view.

Length: If the shape is Square/Rectangular, then enter the inside length dimension of the riser or top as seen from a profile view.

Wall Thickness: The thickness of the walls of riser or top.

Top Slab Thickness: The thickness, from top to bottom, of the slab that usually sits on top of a J Bottom.

Bottom:

Shape (options): The shape, Round or Square/Rectangular, of the bottom of the structure.

Diameter: If the shape is Round, then enter the inside diameter of the bottom of the structure.

Width: If the shape is Square/Rectangular, then enter the inside width dimension of the bottom of the structure as seen in the cross sectional (i.e. drainage structure) view.

Length: If the shape is Square/Rectangular, then enter the inside length dimension of the bottom of the structure as seen from a profile view.

Wall Thickness: The thickness of the walls of the bottom of the structure.

Top & Bottom Slab Thickness: The thickness of both the top and bottom slabs of the bottom.
Riser Position relative to the Bottom and Baseline: The J-type bottom of a structure can be oriented to extend in towards the adjacent baseline, extend outwards away from the adjacent baseline, or centered under the top and riser.

3.6.4 Hydraulics (tab)

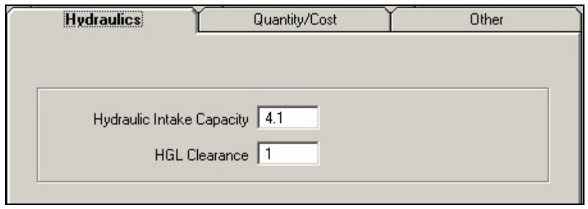


Figure 3.48

Hydraulic Intake Capacity: This value is the inlet capacity as found in the FDOT Standard Index from 1992. These values were not published after 1992. It is an estimated capacity that's used as a general guideline when determining inlet types to use. A warning will be generated by the Compute Hydraulics/Storm Tab Report window if this value is exceeded. The warning will be displayed in the 'Messages' tab of that window.
HGL Clearance: This value is the minimum distance desired between the Hydraulic Grade Line elevation and the inlet/top/grate elevation for the structure. Typically, this value is set to 1 foot for roadway drainage structures (i.e. curb inlets, barrier wall inlets, manholes) and 0 for ditch/overland drainage structures (i.e. ditch bottom inlets) If this value is violated during computations in the Compute Hydraulics/Storm Tab Report window, a warning will be displayed in the 'Messages' tab of that window.

3.6.5 Quantity/Cost (tab)

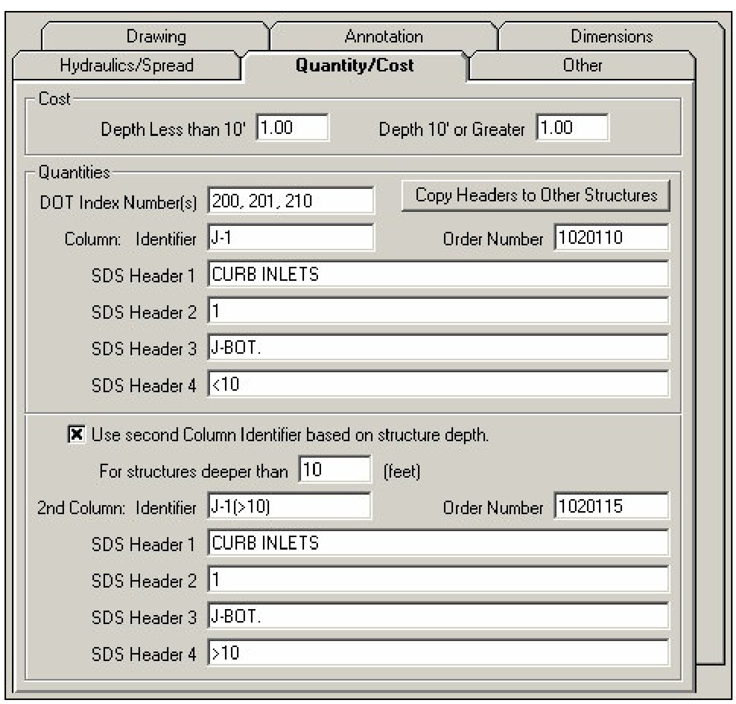


Figure 3.49

Cost (less than or greater than 10' deep): The cost field represents the construction cost of the structure for both, less than 10 feet deep and greater than 10 feet. This field is use by the Cost Analysis window to compute the storm sewer system total cost.

DOT Standard Index Number(s): This field contains the index number(s) that will appear in the 'Index Number' column of the Summary of Drainage Structures tables.

Column Identifier & 2nd Column Identifier: These fields identify which column (in the Summary of Drainage Structures tables) the structure belongs. For instance, you may have more than one J-1 structure defined (it may be necessary to have different bottom dimensions or riser positions) in your inventory of structures for a project. To place this entire group of slightly different J-1s into the same column, simply give them all the same column identifier.

The 2nd column identifier works the same as above, however, it provides the option to have deep structures separated into their own columns. This field is used only if the 'Use Second Column Identifier …' switch is ON and the depth of the structure (node inlet/top/grate elevation minus the lowest flow line elevation at node) is greater than the 'For structures deeper than' value.

Column Order Number (for 1st and 2nd column): This field should contain a number no greater than 2,147,483,647 (long integer data type) which defines this structure's position (column-wise) in the Summary of Drainage Structures table. The typical range of Column Order Numbers for structures is 1,000,000 to 5,000,000. For example, an inlet type 1 with a P-bottom (P-1) has a Column Order Numbers of 1020100 and 1020105 for the less than 10' and greater than 10', respectively.

Use second Column Identifier based on structure depth: This switch allows deep structures to be placed into a different column from the regular depth structures of the same type.

For structures deeper than: During the Summary of Drainage Structures table build routine, this value is compared to structure depth (node inlet/top/grate elevation minus the lowest flow line elevation at node). Shallower structures, not as deep as this value, will be placed in the 'Column Identifier' column, while deeper structures will go into the ' 2nd Column Identifier' column.

SDS Header 1-4: These fields provide the Column Header text for the Summary of Drainage Structures window (see section 5.2.4.4).

Copy Headers to Other Structures (button): This button opens the 'Copy SDS Data to Structures' window (see figure 3.50). Use this window to copy the 'Summary of Drainage Structures' header text to other similar structures.

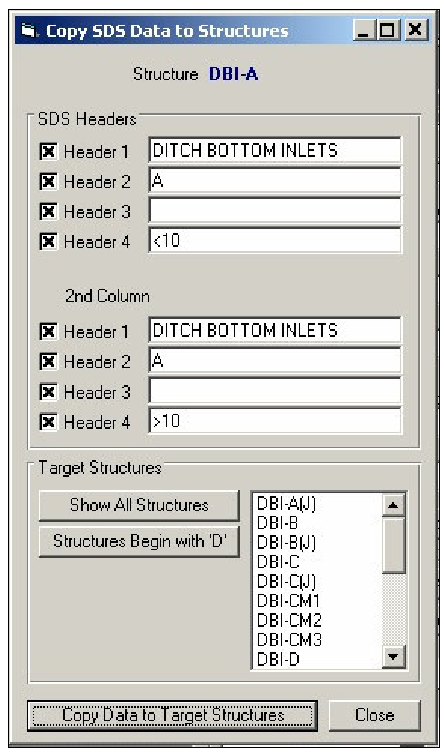


Figure 3.50

3.6.6 Other (tab)

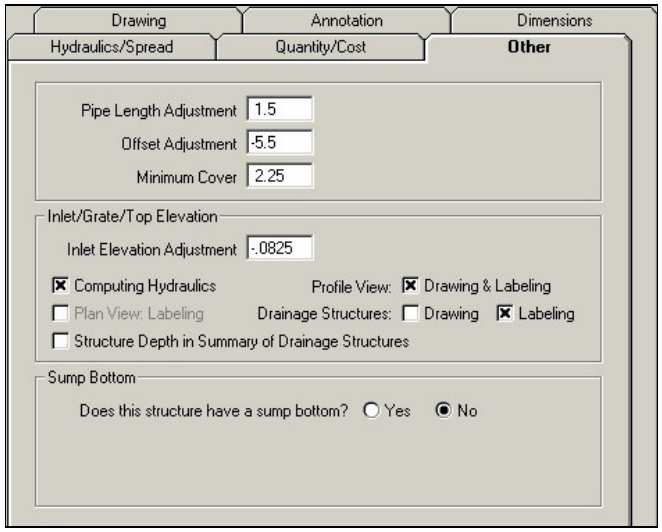


Figure 3.51

Pipe Length Adjust: Used for computing pipe length quantities, ASAD deducts this value from the actual center-to-center distance for each reach that connects to this structure. Valid 'Pipe Length Adjust' values are: greater than or equal to zero.

Offset Adjustment: The Offset Adjustment value is needed to accurately compute the inlet elevation of curb inlets. A node's offset is at the center of the structure, but the inlet elevation is at the edge-of-pavement (EOP). ASAD, therefore, determines the inlet elevation at the EOP by adding the offset adjustment to the node's offset and applying it to the profile elevation and cross slope distance. This value is usually a negative (-) number for any inlets where the inlet/top/grate location is not in the center of the structure such as curb inlets. The value is zero when the inlet/top/grate location is in the center of the structure such as in the case of manholes and ditch bottom inlets.

Minimum Cover: ASAD uses the Minimum Cover value, when setting flow line elevations, to ensure a minimum vertical distance between the inlet/top/grate elevation and the top (outside crown) of all pipes connected to this structure. This value is always a positive (+) number.

Inlet/Grate/Top Elevation (section): The inlet elevation shown in the Node Edit window is computed as the edge of pavement elevation (in the case of curb inlets). Some agencies prefer to report the inlet elevation as the 'Theoretical Grade Line' of the gutter adjacent to the inlet. This section describes how to make adjustments to the computed inlet elevation.

Inlet Elevation Adjustment: Adjust the inlet, grate, or top elevation by this amount. A negative number will result in a lower 'Inlet Elevation' and a positive number will result in a higher ''Inlet Elevation'.

Computing Hydraulics: This switch determines if the 'Inlet Elevation Adjustment' will be added to or subtracted from the Inlet Elevation found in the Node Edit window for use by the hydraulics (Storm Tabs) computations.

Profile View Drawing & Labeling: This switch determines if the 'Inlet Elevation Adjustment' will be added to or subtracted from the Inlet Elevation (as found in the Node Edit window) for drawing and annotation of storm sewer Profiles.

Drainage Structures: Drawing: This switch determines if the 'Inlet Elevation Adjustment' will be added to or subtracted from the Inlet Elevation (as found in the Node Edit window) for drawing the Drainage Structures. The *.top files, which control how the drainage structure tops are drawn, were created using the 'point-of-attachment' location as the starting point for drawing. For curb inlets, this is the edge of pavement elevation. For this reason, the 'Drainage Structures: Drawing' checkbox should be unchecked as seen in figure 3.51.

Drainage Structures: Labeling: This switch determines if the 'Inlet Elevation Adjustment' will be added to or subtracted from the Inlet Elevation (as found in the Node Edit window) when placing the drainage structure write-up paragraph. For those structures that required the 'theoretical gutter line' elevation, this checkbox should be checked as seen in figure 3.51.

Note: When using the 'theoretical gutter line' as the inlet elevation, it is normal to have the 'Drainage Structures: Drawing' = unchecked but the 'Drainage Structures: Labeling' = checked.

Structure Depth in Summary of Drainage Structures: This switch determines if the 'Inlet Elevation Adjustment' will be added to or subtracted from the Inlet Elevation (as found in the Node Edit window) when determining if a structure is less than or greater than 10 feet deep.

Does this structure have a sump bottom: Select Yes to include a sump bottom in the structure. If a sump bottom is used, the drainage structure will be drawn with the sump bottom and a weep hole.

Distance below the lowest invert elevation: This value is typically a minimum of 4 feet.

Weep Hole Diameter: A hole cut into the sumped bottom of a structure is called a weep hole. The hole allows water to drain out of the structure leaving the sediment behind. This weep hole typically has a diameter of 0.5' (6").

3.7 Pipe Definitions

The ASAD Database, as created from the ASAD seed database (see section 1.2.1), contains a list of pipes. Most of these pipes are Florida Department of Transportation (FDOT) pipes and were defined using FDOT's 'ROADWAY AND TRAFFIC DESIGN STANDARDS' publication. If the user cannot find the exact pipe they are seeking, then the user can 'Add' a new structure and define the pipe by filling in the data fields. Information about these fields is listed below.

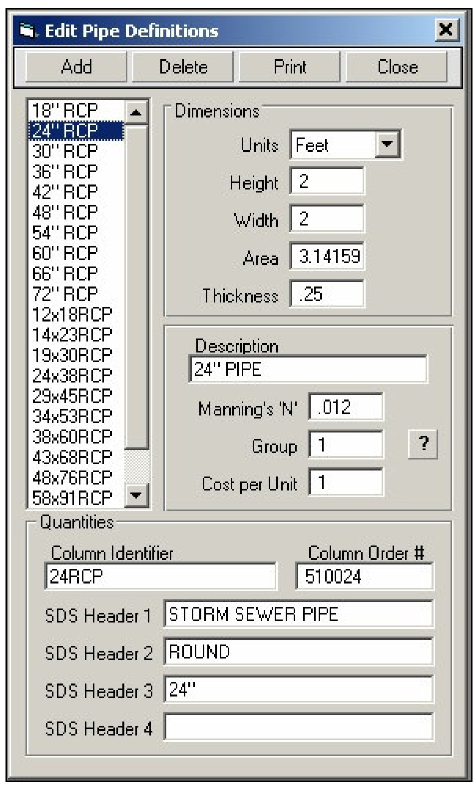


Figure 3.52

Add and Delete (buttons): The "Add" and "Delete" buttons along the top of the dialog are used to create or remove a specific pipe definition.

Note: Valid names in ASAD contain a maximum of eight alphanumeric characters.

Warning: Deleting a pipe that is used by reaches may cause future 'Missing Pipe' errors. Edit these reaches and make appropriate changes to the 'Pipe Name' field.

List: Select the current pipe to edit.
Units: This selector defines which units have been entered into the dimension fields.
Note: The "Units" selector will not convert the dimensions when switched between metric and English units.

Height: The inside height dimension of the pipe.
Width: The inside width dimension of the pipe.
Area: The opening cross sectional area of the pipe.
Thickness: The wall thickness of the pipe. In the case of a corrugated metal pipe, it would be the total distance from the inside of one ridge to the outer skin on an adjacent ridge.
Description: The description is a text string that is used to label the pipe when drawn onto the plans.
Column Identifier: This field is used by the Summary of Drainage Structure routine to distinguish between different pipes so as to sort the quantities of each pipe into the correct column.
Column Order #: The Column Order # is used in conjunction with the Column Identifier by the Summary of Drainage Structures (SDS) routine to place each pipes quantities, in the correct column order. For instance, the column order number for a 30" RCP pipe is 510030. The column order numbers for 24" and 36" RCPs are 510024 and 510036, respectively. When the SDS routine is run, the column containing the 30" RCP pipe quantities will automatically be place between the 24" and 36" pipe columns.
Group: This field is used by ASAD to "Group" pipes for automatic selection during the sizing routine (see section 4.2). The groups provide a way to prevent pipes of different materials (i.e. RCP, CMP, PVC, etc.) from being mixed during the pipe sizing routine. Also, it is a good way to leave out certain sizes of pipes during the pipe sizing routine. For instance, reinforced concrete pipe (RCP) is usually group 1. By changing the group number of 12" and 15" RCP pipes to any other unused group number such a 62, then running the pipe sizing routine and selecting 'Pipes from Group" 1, you have eliminated RCP pipes 12" and 15" from the selection list and left only RCP pipes 18", 24", 30". . . 72" to choose from.
Manning's 'N': This is the hydraulic roughness coefficient based on the type of material the pipe is constructed from.
Cost per Unit: The last descriptive field is the "Cost per Unit" which is simply the price in dollars per length of pipe.
SDS Header 1-4: These fields provide the Column Header text for the Summary of Drainage Structures window (see section 5.2.4.4).