The ASAD© hydraulics engine was developed from the procedure detailed in Chapters 10 and 12 in Volume 2A of the *FDOT Drainage Manual, 1987*. The geometry and hydraulics calculations have been broken into different modules. The first four of these modules performs the calculations necessary to analyze and define the node locations and hydrologic basin areas, set the flow lines, and compute the volumes and velocities for each node and reach of the system. The remaining analysis tools check for utility conflicts, and perform a cost estimate for each system.

4.1 COMPUTE STORM SEWER GEOMETRY AND DRAINAGE AREAS

The first step in the computations, is the Compute Storm Sewer Geometry and Drainage Areas dialog (See figure 4.1). This routine affects only those nodes and reaches in the Active Systems (see figure 1.2, item #15). It contains five separate options for calculation.

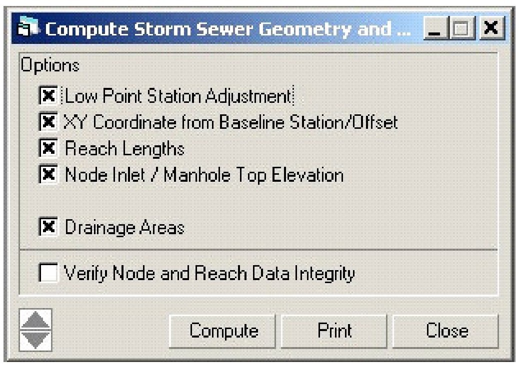

Figure 4.1

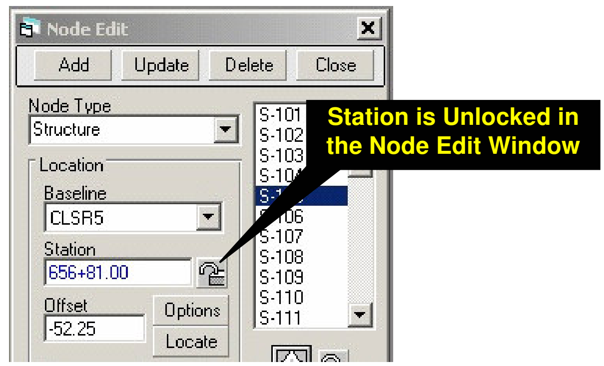

Figure 4.2

Low Point Station Adjustment: This checks and adjusts (to the profile low point) all unlocked nodes (see figure 4.2) that are within 50' of a low point in the profile.

XY Coordinate from Baseline Station/Offset: This option translates the station/offset node locations into a rectangular x,y (i.e. East & North) coordinate system. This option is also responsible for determining the structure inlet rotation angle and orientation.

Reach Lengths: the Reach Length option calculates the pipe or ditch length between two nodes. When the reach is a pipe, then both 'Quantity' and 'Hydraulic' lengths will be generated. This routine uses the North and East coordinates of the two nodes to compute the length.

Node Inlet/Manhole Top Elevation: This option determines the elevation curb inlet elevation, grate elevation, or manhole top elevation. For this routine to run successfully, it must use the Station, Offset, Profile and Cross Slope fields from the node. Using the Station, ASAD first computes the Profile elevation at that station. Next, it computes, from that elevation, down across the Cross Slope to the Offset. Finally, it adjust, using the Offset Adjustment field in the structure definition, for the distance between the node's Offset and the point of connection between the curb inlet and the edge of pavement to arrive at the final Inlet Elevation value.

Drainage Areas: The drainage area option calculates the effective drainage areas if the input area is unlocked.

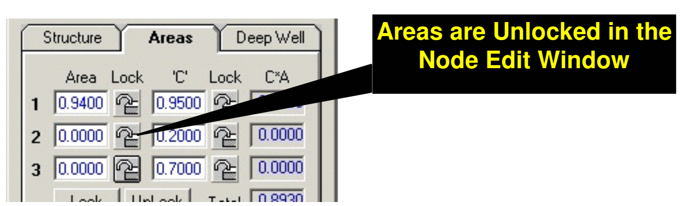

Figure 4.3

The cross widths and profile are used to determine the areas. This process may take as long as 30 seconds to complete depending on the number of nodes in the system and the number of lines in the cross widths detail.

Verify Node and Reach Data Integrity: The last check box verifies node and reach data integrity. This process checks the nodes and reaches for errors in the node and reach data. This option is available in most of the other calculation dialogs.

Note: If this process is run before other task have been completed, then some warnings about missing data will probably occur. For instance, running Verify Node and Reach Data Integrity before computing flow line elevations may result in a 'Missing flow data for reach …' warning for some reaches.

4.2 COMPUTE PIPE SIZES

This window is used to compute the pipe size needed to convey a computed Q through each reach. The Q is computed using the rational method Q=CIA. The C refers to the weighted C (unit less) value for the entire contributing area passing through the upstream node. The I is the intensity (inches per hour) based on the Time of Concentration (Tc) to the upstream node. The A is the area (acres) for the entire contributing area passing through the upstream node. Look at this sample:

Given: Area (A) = 4.38 acres
C = .83
Intensity (I) = 5.61 in/hr (based on a Tc of 13.31 minutes)

Q = 0.83 * 5.61 * 4.38 = 22.49 cfs (cubic feet per second)

Once the Q is computed for a reach, the process will cycle through the Pipe Selection List, as determined by the 'Use Pipes from Group' setting, from smallest to largest, until the condition selected in the 'Constraints:' section of the window is met.

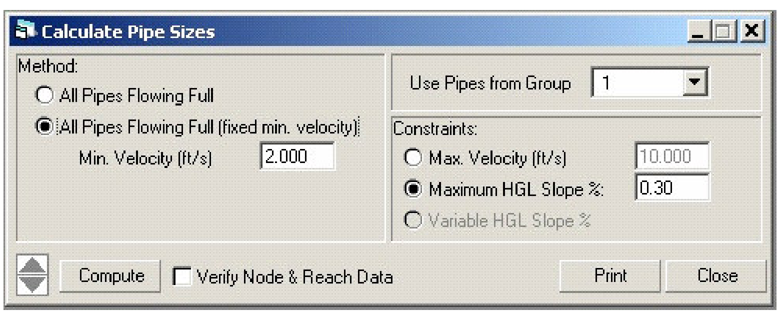

Figure 4.4

The Method: All Pipes Flowing Full or All Pipes Flowing Full (with a fixed minimum velocity): Initially the velocity is computed, using the full flow method, by dividing the cross sectional open area of the pipe into the flow (Q). For instance, a Q of 4.8cfs flowing full through a 24" pipe with an area of 3.14 sq.ft. (Pi * (1' ^2)) would result in a velocity of 1.53fps (4.8cfs / 3.14sqft).

All Pipes Flowing Full: By choosing the first option, 'All Pipes Flowing Full', the losses and therefore the hydraulic gradient through this pipe would be based on the 1.53fps velocity and the travel time (and the time of concentration (Tc)) to the next node would also be based on the 1.53fps velocity.

All Pipes Flowing Full (with a fixed minimum velocity): With the second option, 'Pipes Flowing Full (fixed minimum velocity)', the computed velocity of 1.53fps would be used to compute the losses and the hydraulic gradient through this pipe but the travel time (and the time of concentration (Tc)) to the next node would be based on the 2.00fps minimum velocity as prescribed by the **Min. Velocity** field. This method is more conservative approach to computing the Tc's and Q's. The shorter the Tc, the higher the Q, and therefore, the larger the pipe. Even though this method was established by FDOT in the mid 80's and continued into the late 90's, many private consultants today are unaware of this nuance in the calculations.

Use Pipes from Group: Use this to select which pipes (Pipe Selection List), with their own individual characteristics i.e. pipe size, shape, N value, cost, etc., to use in the pipe sizing. Typical grouping for pipes are:

Pipe Descriptions	Group
Round RCP 12" & 15"	10
Round RCP 18" thru 72"	1
Ellip. RCP All Sizes	2
Round BCCMP All Sizes	3

Figure 4.5

These groupings are maintained through the Group field in the Pipe Definition dialog.

Constraints:
Max. Velocity: This option sizes the pipe so as to not exceed the specified velocity. The lower the velocity, the larger the pipe diameter, and vice versa.
Maximum HGL Slope %: This option sizes the pipe so as to not exceed the specified hydraulic grade line (HGL) slope. The flatter the HGL, the larger the pipe diameter, and vice versa. Listed here is an example of how to use this routine.

A storm sewer system has 20 inlets (S-1 thru 2-20) that collect runoff and outfall through S-21. The tail water elevation (at the outfall) is 62.20. After reviewing the inlet elevations and the distances from the outfall, for the other 20 inlets, we determine the worst case, HGL-wise, is probably S-5. This inlet is at a low point (inlet elevation of 66.11) and is 1850' from the outfall. We now can compute an approximate maximum HGL slope that will work with our tail water and our distant low point.

Compute Max. HGL at Inlet:

```
Inlet Elevation:         66.11
1' freeboard at inlet:  -1.00
------------------------------
Max. HGL at Inlet:       65.11
```

Compute Vertical Loss as the difference between the Max HGL at Inlet and Tail water:

```
Max. HGL at Inlet:       65.11
Less Tail water:        -62.20
------------------------------
Vertical Loss:            2.91
```

Now compute the max HGL by dividing the horizontal distance (distance from inlet to outfall) into the vertical loss:

```
Vertical Loss:            2.91'
                         ------ = 0.001573 * 100 = .1573% or 0.16%
Horizontal Distance:     1,850'
```

Now we enter 0.16 into the 'Maximum HGL Slope %' field and press the compute button. The routine will then select pipe sizes that will result in the HGL for each pipe being approximately 0.16% which should provide us with a smooth HGL slope from the tail water up to our worst case inlet.

4.3 SET FLOW LINES

The set flow line window computes the upstream and downstream flow lines for each pipe if, in the Reach Edit window, the From and/or To flow line Locks are unlocked (see figure 4.6).

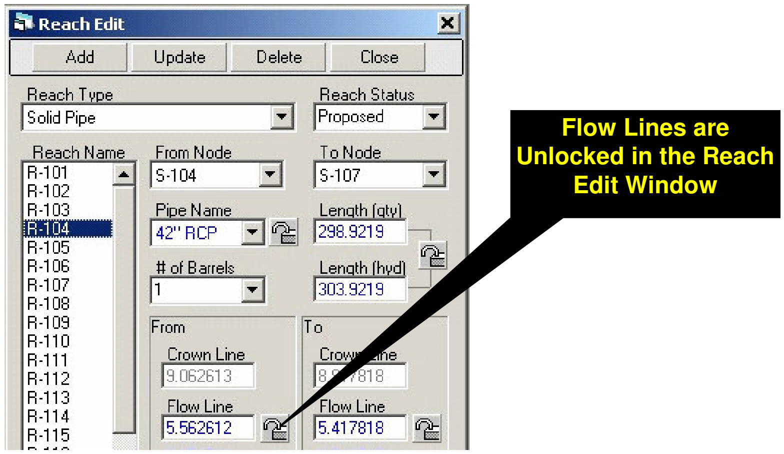

Figure 4.6

The Set Flow Lines window uses the following method for computing flow lines:
Starting with the node at the top of the system (the most upstream node of the system or a branch of the system), ASAD will query the structure for the 'Minimum Cover' value. Also, get the 'Pipe Height' and 'Wall Thickness' values from the pipe definition (for the outgoing pipe). Compute the maximum outer crown line elevation:

Max Outer Crown Line Elev = Inlet Elevation - Minimum Cover
Max Flow Line Elev = Max Outer Crown Line Elev - Wall Thickness - Pipe Height.

Now with the upstream flow line (Max Flow Line Elev) determined, ASAD, using the Pipe Slope selected by the user, will now compute the estimated downstream flow line at the next node. Once at the next node, ASAD will determine if 'Minimum Cover' is met and whether other incoming pipes meet the 'Minimum Cover' and will match all pipes at the node by either crown line or flow line. If the minimum cover is not met, then ASAD will lower the flow lines for the pipes while still matching flow lines or crown lines. It will then compute the downstream flow line elevation again and repeat the cycle until it gets to the final reach that connects to the outfall.

Note: The Compute Flow Lines routine does not compute the flow line elevation at the outfall. Use the Reach Edit window to set this flow line and lock it.

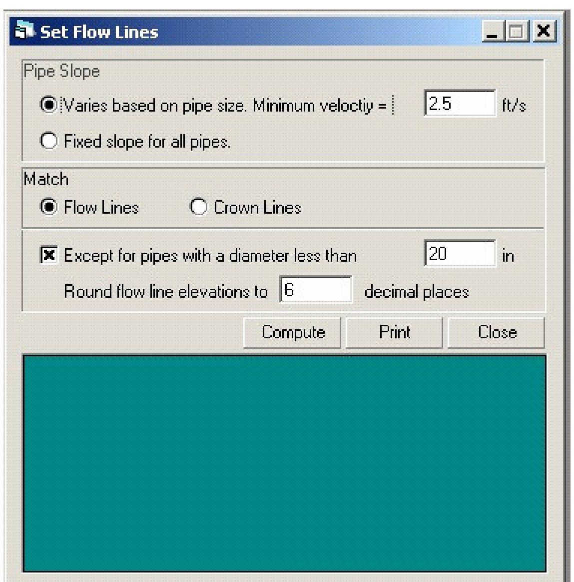

Figure 4.7

Pipe Slope:
Varies based on pipe size. Minimum velocity =: Use this option to allow ASAD to vary the slope of the pipe based on a minimum velocity (at pipe capacity). As pipe size increases, this slope becomes flatter and vice versa. This minimum velocity is important because it ensures enough slope, and therefore velocity, to flush dirt and debris out of the pipe. A default value of 2.5fps is standard on most FDOT projects.
Fixed slope for all pipes: Use this option to set the minimum pipe slope for all pipes in the active system to a user defined value.
Match: Flow Lines or Crown Lines: This option determines if ASAD is going to match pipe crown lines or flow lines.
Except for pipes with a diameter less than: Use this option to prevent the match flow line/crown line process from being performed on pipes with a diameter less than xx (typically 20" as seen in figure 4.7). This prevents stub pipes on one side of the road, which are at a minimum depth, from diving steeply to match the crown line or flow line of the much deeper trunk line on the other side of the road. If this option is checked and 20" is the limit, then all 15" and 18" pipes will slope, at the minimum slope, from the stub inlet to the truck line.
Round flow line elevations to: This option sets the number of decimal places to compute the flow lines to. This is only for the values as stored in the database. The reports and drawing tools, such as Storm Tabs and Draw Drainage Structure, will round (decimal places determined by the user) these values at the time they are drawn.
Note: It is best to leave the number of decimal places for flow lines set at 6.

4.4 COMPUTE HYDRAULICS AND GENERATE STORM TABS

To perform the hydraulics calculations, the designer must press the "Compute hydraulics" button (See figure 4.8). The message window will display any warnings and errors that may occur.

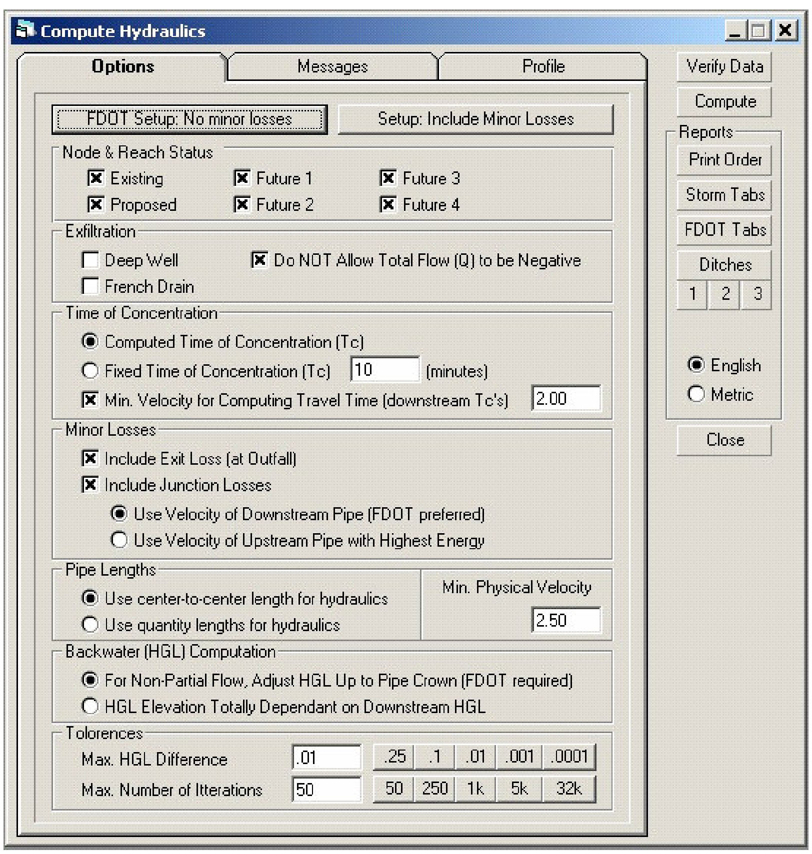

Figure 4.8

FDOT Setup: No Minor Losses (button): Sets the options to FDOT standard settings for storm sewer with no Deep Wells or French Drains. This setting **does not include** minor losses (junction losses at inlet & manholes plus exit loss at the outfall).
Setup: Include Minor Losses (button): Sets the options to FDOT standard settings for storm sewer with no Deep Wells or French Drains. This setting **includes** minor losses (junction losses at inlet & manholes plus exit loss at the outfall).
Node & Reach Status (check boxes): Using their status, select the nodes and reaches to include in the hydraulic calculations.

Exfiltration section
Deep Well(check box): Check this check box to include discharge into deep wells as part of the hydraulics calculations. Uncheck this box and deep wells will be ignored.
French Drain (check box): Check this check box to include discharge into French drain as part of the hydraulics calculations. Uncheck this box and French drains will be ignored.
Do Not Allow Total Flow to be Negative (check box): When there is discharge into deep wells and/or French drains, there is the possibility of having a negative flow (Q). This may or may not cause problems depending on where, in the system, the flow (Q) goes negative and where is resumes as positive. If an error occurs due to a negative flow, usually as a 'Divide by Zero' or 'Overflow' error, then 'check' this option and the flow (Q) will not be allowed to go negative.

Time of Concentration section
Computed Time of Concentration (option button): Select this option to compute the time of concentration in the usual manner.
Fixed Time of Concentration (option button and text box): Select this option fix the time of concentration (Tc), and therefore the rainfall intensity, to the value in the adjacent text box. This option is used to compute the 'worst-case scenario' hydraulics assuming the entire storm sewer if full of water at the beginning of the storm event. Basically, the first drop is the first drop out analogy.

Warning: Read this next section on 'Minimum Velocity for Computing Travel Time' VERY CAREFULLY. This setting could have significant ramifications on your design. If after reading, more clarification is needed, call tech support at (352) 383-4191.
Minimum Velocity for Computing Travel Time (downstream Tc's) (option button):
The travel time from one node to the next is a function of the length divided by the velocity. Under full flow conditions, the velocity is computed by dividing the cross sectional open area of the pipe into the flow (Q). For instance, a Q of 1.8cfs flowing full through a 24" pipe with an area of 3.14 sq.ft. (Pi * (1' ^2)) would result in a velocity of 0.57fps (1.8cfs / 3.14sqft). The computed velocity of 0.57fps would be used to compute the losses and the hydraulic gradient through this pipe. However, the travel time (and the time of concentration (Tc)) to the next node would depend on whether or not this option is check (ON) or unchecked (OFF).
If this option is unchecked (OFF), then travel time and Tc would be based on the 0.57fps velocity.
If this option were checked (ON) and the computed velocity (0.57fps) were less than the value in the adjacent text box (2.00fps in this case), then the travel time and the Tc would be based on the 2.00fps velocity.
Why is this important? **It could greatly change how the total flow (Q) is computed** and in-turn effect the hydraulic grade line. Take for instance the scenarios below and note the difference in Tc's and rainfall intensities.

Given:

```
IDF Curve                        Zone 5 Frequency 3yr
Initial Tc at node #1            10 minutes
'C' at node #1                   1.00
Rainfall intensity (I) at 10 min 6.82 in/hr
```

```
Area (A) at node #1                  0.26 acres
Computed Q=CIA at node #1            1.8 cfs
```

Scenario #1 (Use actual velocity):

```
Pipe Size                            24 in
Length of pipe to node #2            300 ft
Full flow velocity (actual)          0.57 fps
Travel time to node #2               8.77 minutes
Tc at node #2                        18.77 minutes (Initial Tc 10 min.
                                     at node #1 plus 8.77 minutes)
Intensity node #2 at 18.77 min       4.83 in/hr
```

Scenario #2 (Use a modified velocity of 2.00fps):

```
Pipe Size                            24 in
Length of pipe to node #2:           300 ft
Full flow velocity (modified)        2.00 fps
Travel time to node #2:              2.50 minutes
Tc at node #2:                       12.50 minutes (Initial Tc 10 min.
                                     at node 1 plus 2.50 minutes)
Intensity node #2 at 12.50 minutes:  5.75 in/hr
```

These two scenarios show a big difference in Tc (and intensity) when just evaluating one reach. This difference can become much larger when compounded over many reaches with low velocities. On the other hand, with most systems this difference may never show itself because the velocities are 2fps or faster.

To Use this Option or Not? This is a question best posed to the agency for which the design work is being done. Checking (ON) this option is a more conservative approach to computing the Tc's and Q's. The shorter the Tc, the higher the Q, and therefore, the larger the pipe. Even though this method was established by FDOT in the mid 80's and was widely known by engineers that computed storm tabs by hand, it appears this method did not get passed along to newer engineers. This is probably due to the dependence on computers and hydraulics software to perform all of the storm tab calculations.

Pipe Lengths section

Use center-to-center lengths for hydraulics (option button): Use the hydraulic length (center to center) when computing hydraulics. This is usually the option when NOT including minor losses.

Use quantity lengths for hydraulics (option button): Use the quantity length when computing hydraulics. Use this option when including minor losses.

Backwater (HGL) Computation section

For Non-Partial Flow, Adjust HGL Up to Pipe Crown (FDOT required) (option button): This option will artificially move the HGL up to the crown line, of an upstream reach, in certain circumstances. This is an FDOT requirement that typically yields a more conservative design. However, sometimes this artificial jump in the HGL downstream can adversely affect the HGL upstream, especially if full flow (pressure) conditions.

HGL Elevation Totally Dependant on Downstream HGL (option button): Using this option eliminates the artificial jump to crown line as described above. Typically you will want the use the above option 'Adjust HGL Up to Pipe Crown (FDOT required)' as a starting point and switch to this option only if needed.

Tolerances section

Max. HGL Difference: This value is used when fine tuning systems with multiple deep well and/or French drains (DW/FD). When first working with DW/FD, start with a setting of 0.25 for faster calculations. As the design is refined, change this value to .01 or .001. For standard storm sewers (no DW/FD) and ditches, use 0.01.

Max. Number of Iterations: In most standard storm sewers (no DW/FD) and ditches, ASAD will go through 3 to 5 iterations of calculations to bring the systems into 'balance'. For these type systems, set this value to 50. Computing the hydraulics for systems with multiple deep wells and/or French drains can be quite chore by hand and even taxing for ASAD. To bring one of these systems into balance may require 300, 400 or even a thousand iterations. Start your calculation process 250 or 1000. Note: the smaller the HGL difference (above), the higher the number if iterations needed to solve it.

What does this message? Maximum number of passes 50 reached without a stabilized HGL, Increase the 'Max. Number of Iterations' value and try again.
ASAD accumulates the flows from the top of the system to the bottom (outfall). Then using those flows, it computes the losses, and subsequently the hydraulic grade line (HGL), from the bottom (Outfall) to top of the system. During each of these 'cycles', each reach is determined to be full or partial flow. Usually the type of flow for a reach does not change from cycle to cycle. However, some reaches that are on the full flow/partial flow threshold, will toggle between full and partial on every other cycle thus causing the message above.
Solution? As it turns out, this problem is not really a problem after all. The HGL has been computed properly as well as the flows, velocities and losses. The Storm Tabs are correct. However, if you would like to make a change(s) to your system to eliminate this message here are some suggestions:

1) Look for a reach that has a 'Flow TOTAL' (Q) and a 'Capacity' that is nearly the same. Change an invert, on either pipe end, to change the slope of the pipe. Flattening the pipe will force it to be 'Full' flow. Steepening the pipe will force it to be 'Partial' flow.
2) Look for a reach that has an HGL that is close to the crown of the pipe. Follow the procedure in #1.
3) Try changing the slope of the pipe going to the outfall. This can sometimes be the culprit.
4) Other changes may include changing the tail water or storm event.

Verify Data (button): This button verifies node and reach data integrity. This process checks the nodes and reaches for errors in the node and reach data. This option is available in most of the other calculation dialogs.
Note: If this process is run before other task have been completed, then some warnings about missing data will probably occur. For instance, running Verify Node and Reach Data Integrity before computing flow line elevations may result in a 'Missing flow data for reach …' warning for some reaches.
Compute (button): Computes the storm tabs hydraulics and stores the results in the ASAD database.
Print Order (button): This button opens the Print Order window. This window allows the user to change the order in which the nodes are printed in the storm tabs report.
Storm Tabs (button): This button invokes the report generator which prints the storm tabs report. This button prints tabs in the old (prior to Fall 2006) FDOT format.
FDOT Tabs (button): This button invokes the report generator which prints the storm tabs report. This button prints tabs in the new (after Fall 2006) FDOT format.
Ditches, 1, 2, 3 (buttons): These buttons report hydraulic data specific to the ditches in the systems.

4.5 UTILITY CONFLICTS

The utility conflicts are determined be selecting the reaches the designer wants to analyze (See figure 4.9). The minimum clearance must be entered in the key-in field. The designer must then press the "Compute" button. The results of the analysis are displayed in the message window.

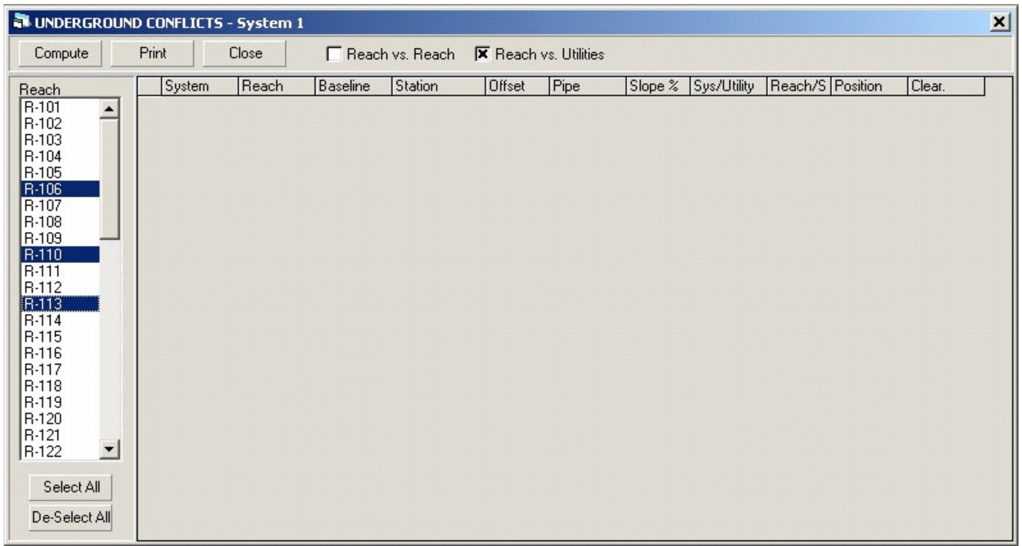

Figure 4.9

4.6 STORM SEWER COST ANALYSIS

The designer must select which information is to be included with the results of the cost analysis (See figure 4.10). Three selections are available; the Node/Reach details, the Node/Reach Sub-totals, and the Grand Totals. One of these options must be selected for the system to be analyzed. The cost analysis will be displayed in the message window at the bottom of the dialog.

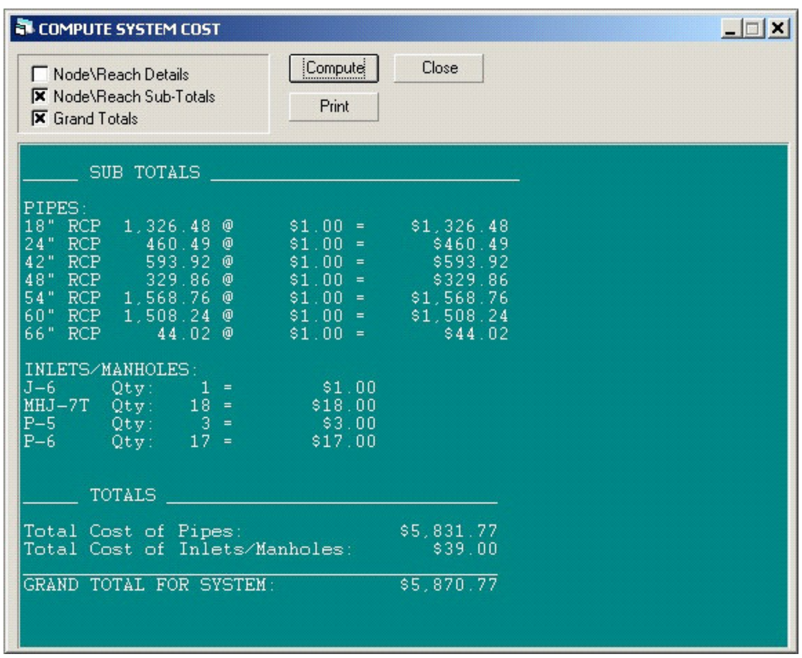

Figure 4.10

4.7 PROFILE ELEVATION INQUIRY

The designer must select the profile from the list in the left side of the dialog (See figure 4.11). Wither the increment or Hi/Low tab can be selected. If the inquiry is performed by increment, the beginning and ending stations as well as the increment must be supplied by the designer. When the inquiry is computed, the results are displayed in the message window at the bottom of the dialog.

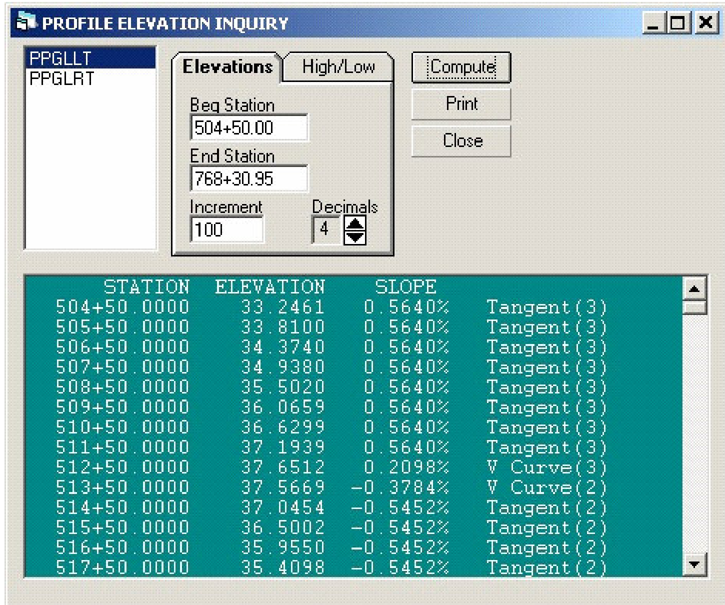

Figure 4.11

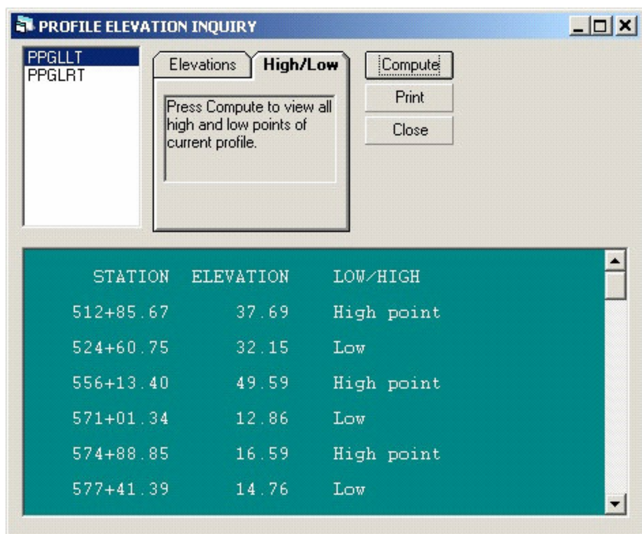

Figure 4.12