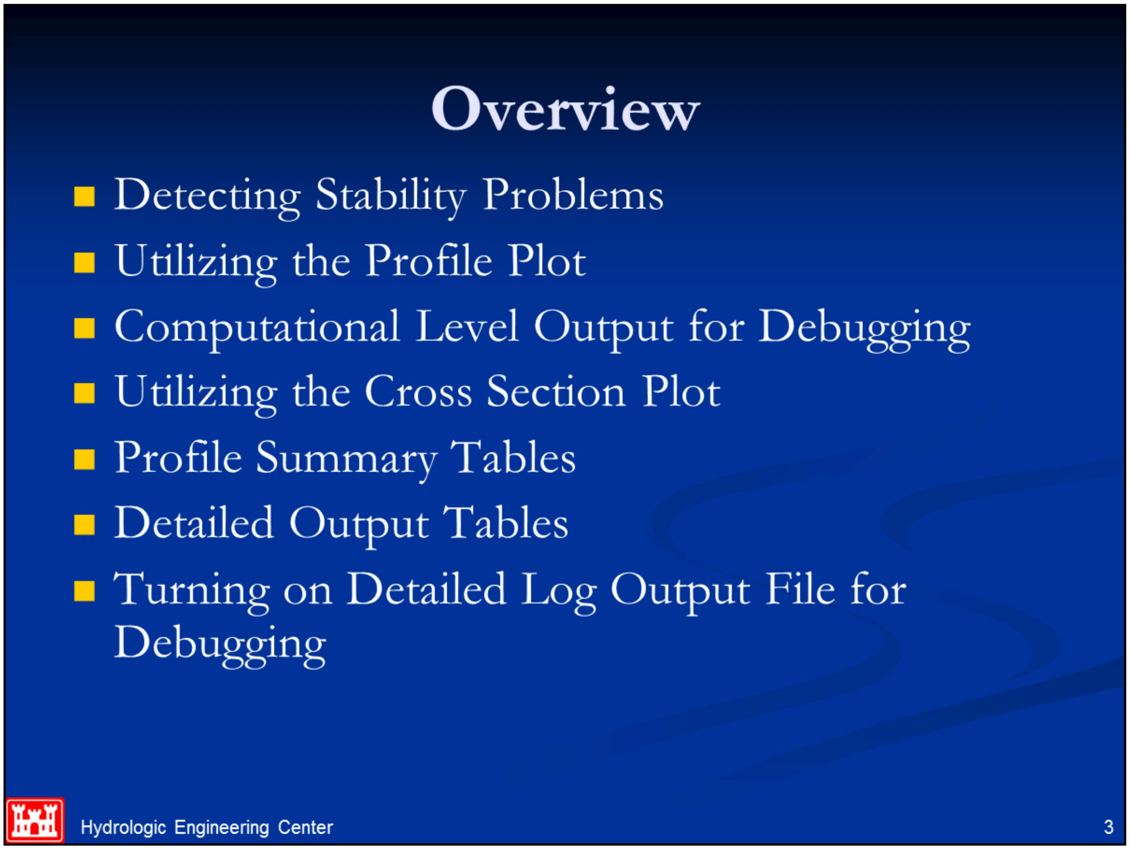
Finding and Fixing Model Stability Problems

Hydrologic Engineering Center 1

Objectives

- The Objectives of this lecture are to teach students how to detect, find, and fix model stability problems, using the available tools in HEC-RAS

Hydrologic Engineering Center 2


Overview
Detecting Stability Problems
Utilizing the Profile Plot
Computational Level Output for Debugging
Utilizing the Cross Section Plot
Profile Summary Tables
Detailed Output Tables
Turning on Detailed Log Output File for Debugging
Hydrologic Engineering Center
3

Detecting Stability Problems

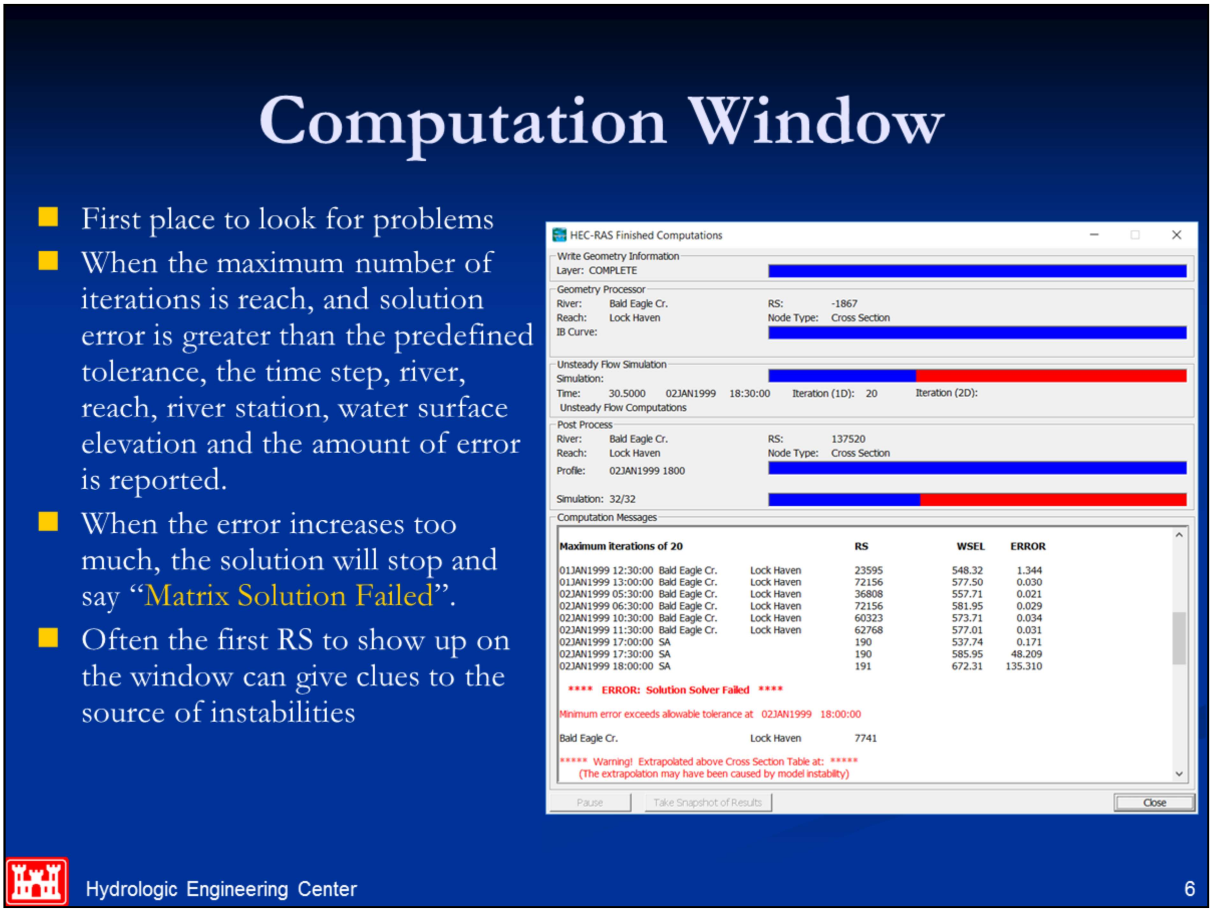
- How do you know you have a model stability problem?
  - Program completely blows up during run.
  - Program says matrix solution went completely unstable during the calculations.
  - Computed error in water surface calc is very large
  - Program goes to maximum number of iterations for several time steps in a row, with large errors.
  - Program has oscillations in the computed stage and flow hydrographs.

Hydrologic Engineering Center 4

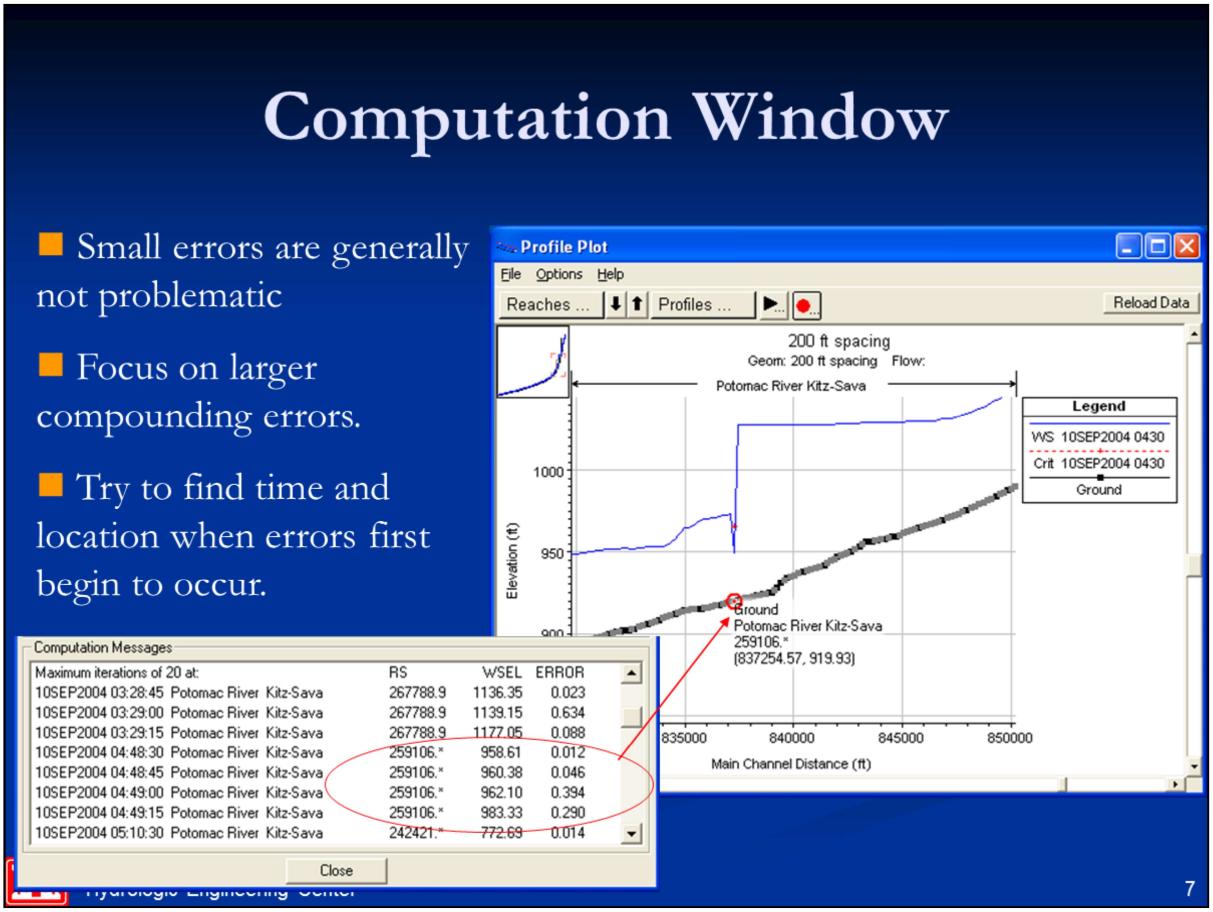
Detecting Stability Problems - Continued

- What do you do when this happens?
  - Note the simulation time and location from the computation window when the program either blew up or first started to go to the maximum number of iterations with large water surface errors.
  - Use the HEC-RAS Profile and Cross Section Plots as well as the Tabular Output to find the problem location and issue.
  - If you can not find the problem using the normal HEC-RAS output - Turn on the "Detailed Output for Debugging" option and re-run the program.
  - View the text file that contains the detailed log output of the computations. Locate the simulation output at the simulation time when the solution first started to go bad.
  - Find the river station locations that did not meet the solution tolerances. Then check the data in this general area.

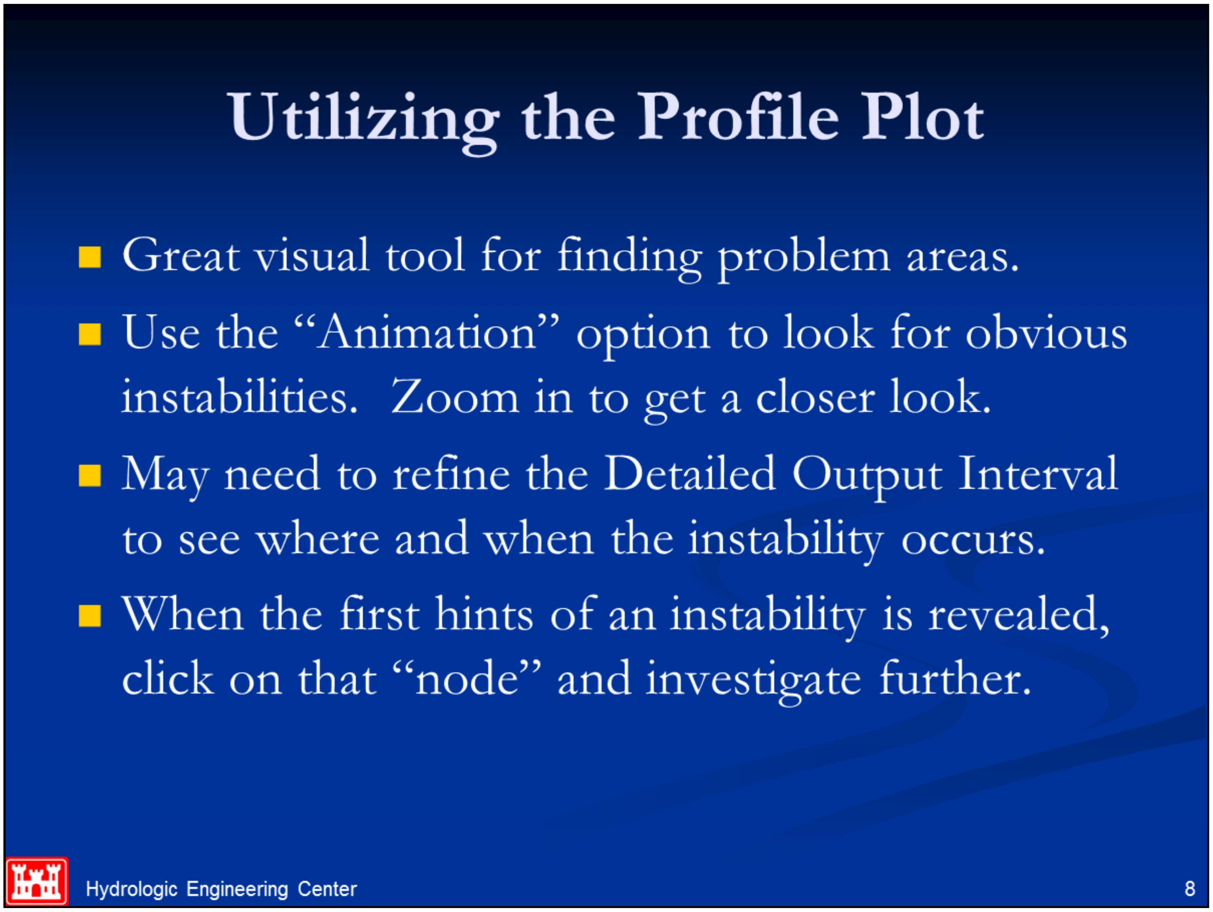
Hydrologic Engineering Center 5



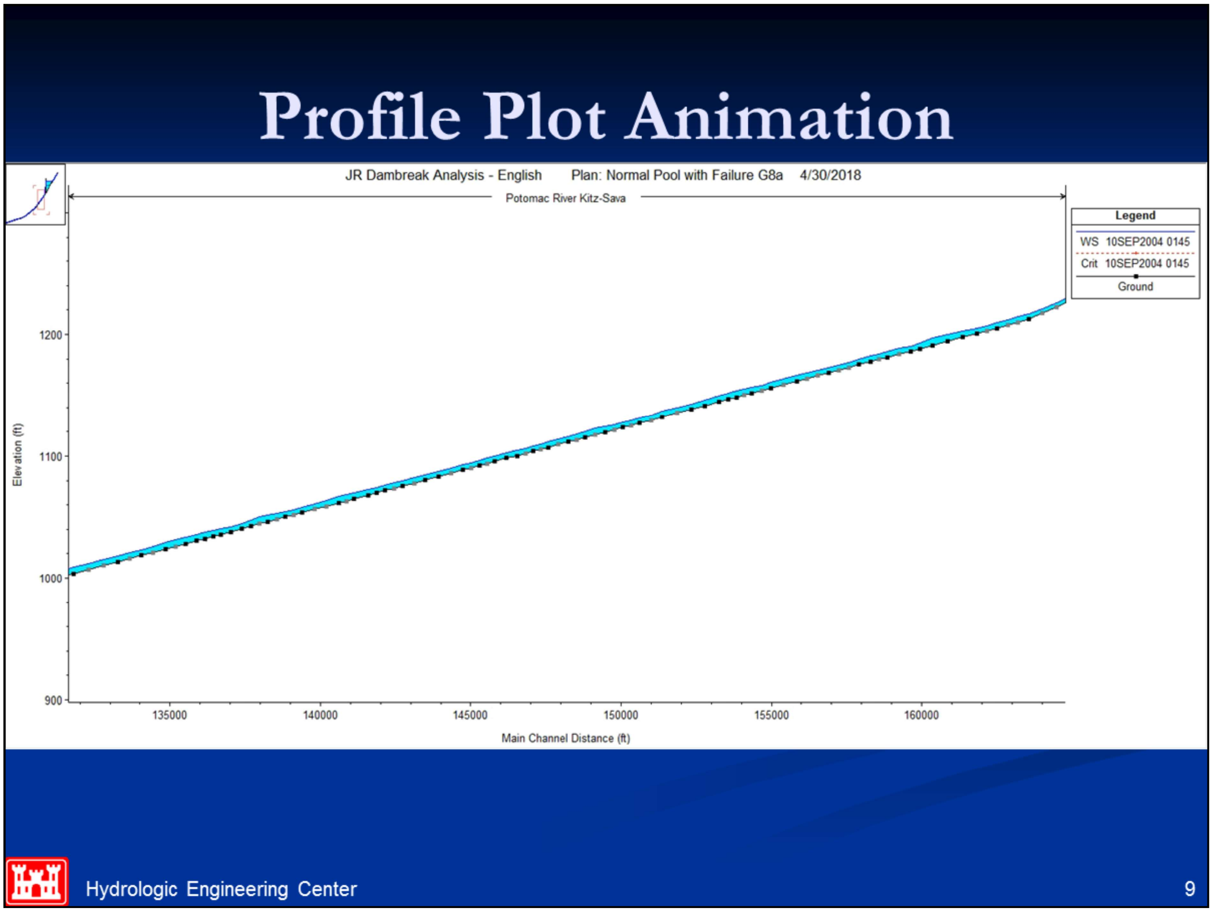
The first place to look for instabilities and errors is the Computations Window during and just after the simulation is run. The red progress bar indicates the model went unstable and could not complete the simulation. The Computation Messages window provides a running dialog of what is happening in the simulation at a given time step in a given location. This allows the user to watch errors propagate during the simulation. Once the simulation has crashed, don't close the Computations Window. Instead, scroll up through the messages and try to determine where the propagation of errors began, and at what time.



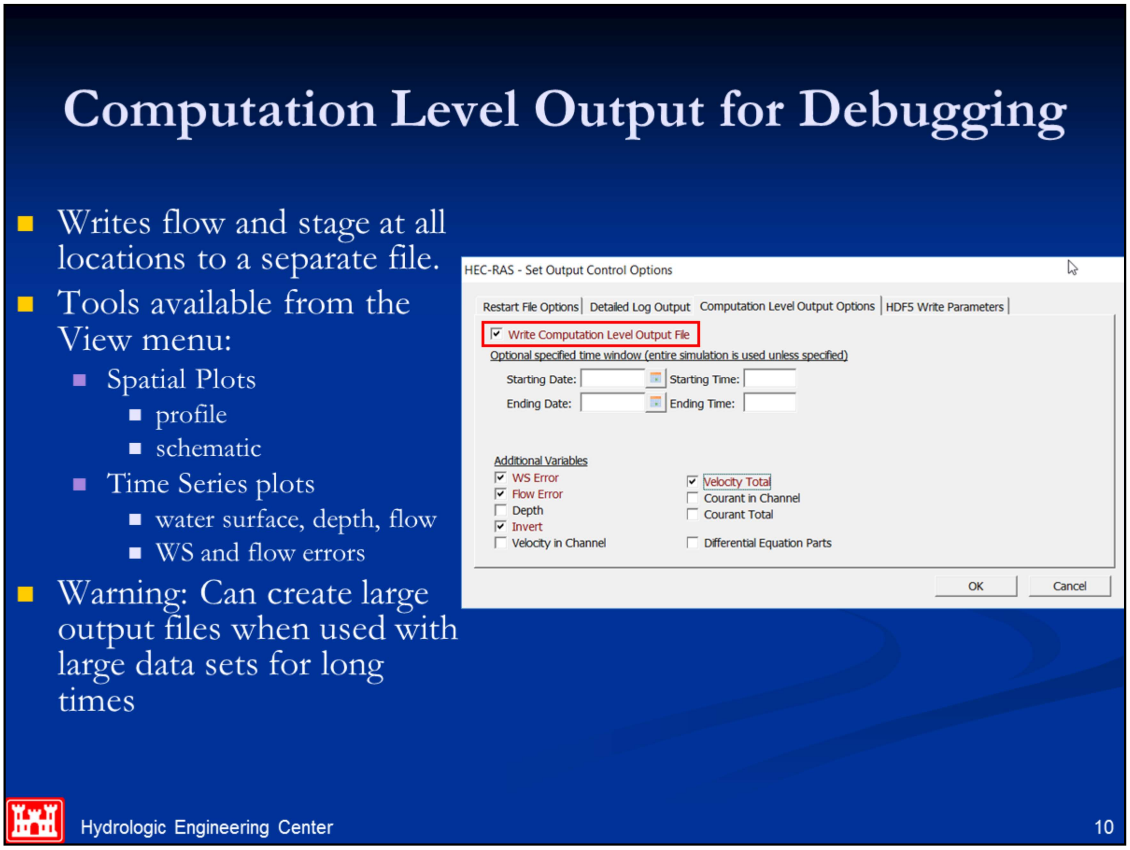
Sometimes the first error to occur is at the beginning of the simulation and is just a result of the model settling out after the transition from initial conditions to the first time step. Particularly if the error only occurs once for that given river station. It is better to focus on reoccurring errors or compounding errors first. The example on this slide shows a relatively small error at river station 259106* that grows to 0.4 ft in the next few time steps.



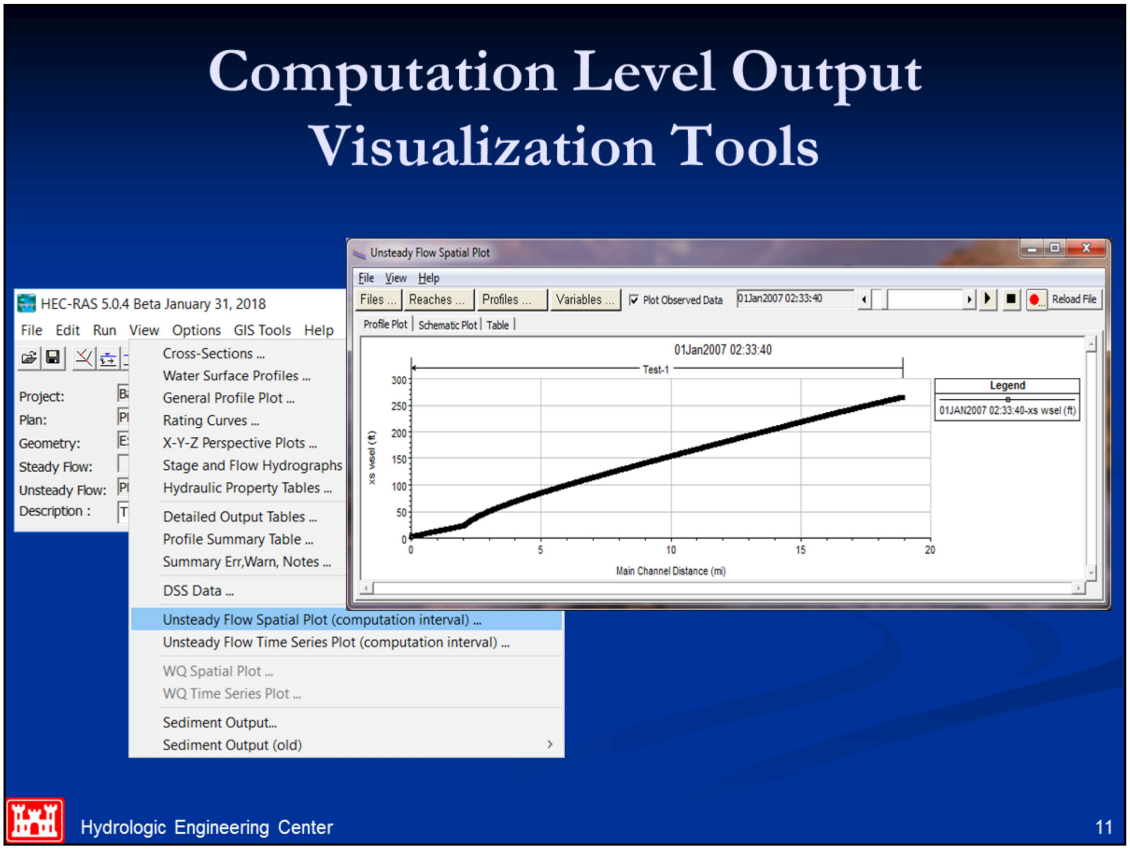
The profile plot is typically the first graphical tool to use to try to pinpoint instabilities. Obvious errors are shown distinctly in this plot and you can see what is going on in the entire reach at the same time. Stepping through each profile using the animation tool allows you to see changes over time, including the progression of the flood wave as well as propagation of errors. The profile output is taken from the detailed output file. Therefore, it is sometimes necessary to refine the detailed output interval to adequately see the beginning of instabilities. The profile plot allows the user to click on a given node to determine its river stationing. Find the node where the instability first occurs and investigate further.



The above slide was an animation of the profile plot, showing the progression of a model instability problem. The profile plot can assist you in locating where an instability is occurring and when. You may need to zoom in to get a closer look. You may also need to set the **Detailed Output Interval** to a smaller value and re-run the simulation in order to see what is happening at a finer time step increment.



When performing an unsteady flow analysis the user can optionally turn on the ability to view output at the computation interval level. This is accomplished by checking the box labeled **Computation Level Output** on the Unsteady Flow Analysis window (In the Computations Settings area on the window). When this option is selected an additional binary file containing output at the computation interval is written out. After the simulation the user can view computation level output by selecting either **Unsteady Flow Spatial Plot** or **Unsteady Flow Time Series Plot** from the **View** menu of the main HEC-RAS window.

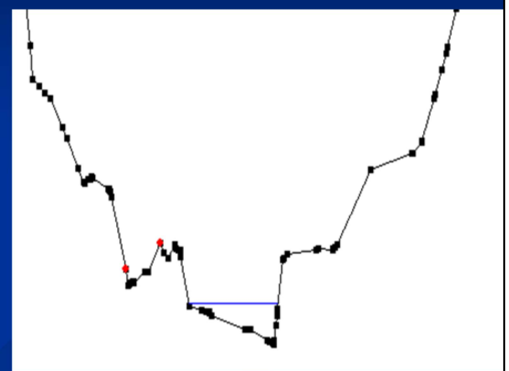


Visualization of computation level output can be accomplished with either **Spatial Plots** or **Time Series Plots**. From the Spatial Plots the user can view either a profile plot, a spatial plot of the schematic, or tabular output. The user can select from a limited list of variables that are available at the computation level output. These are water surface elevation (XS WSEl); Flow (XS Flow); computed maximum error in the water surface elevation (XS WSEL ERROR); computed maximum error in the flow (XS FLOW ERROR); and maximum depth of water in the channel (DEPTH). Each of the plots can be animated in time by using the video player buttons at the top right of the window. This type of output can often be very useful in debugging problems within an unsteady flow run. Especially plotting the water surface error and animating it in time.

The other type of plot available at the computation interval output level is the **Unsteady Flow Time Series Plot.** When this option is selected the user will get a plot as shown in the Figure above. Some of the same options and variables are available for the Time Series Plots as were available for the Spatial Plots.

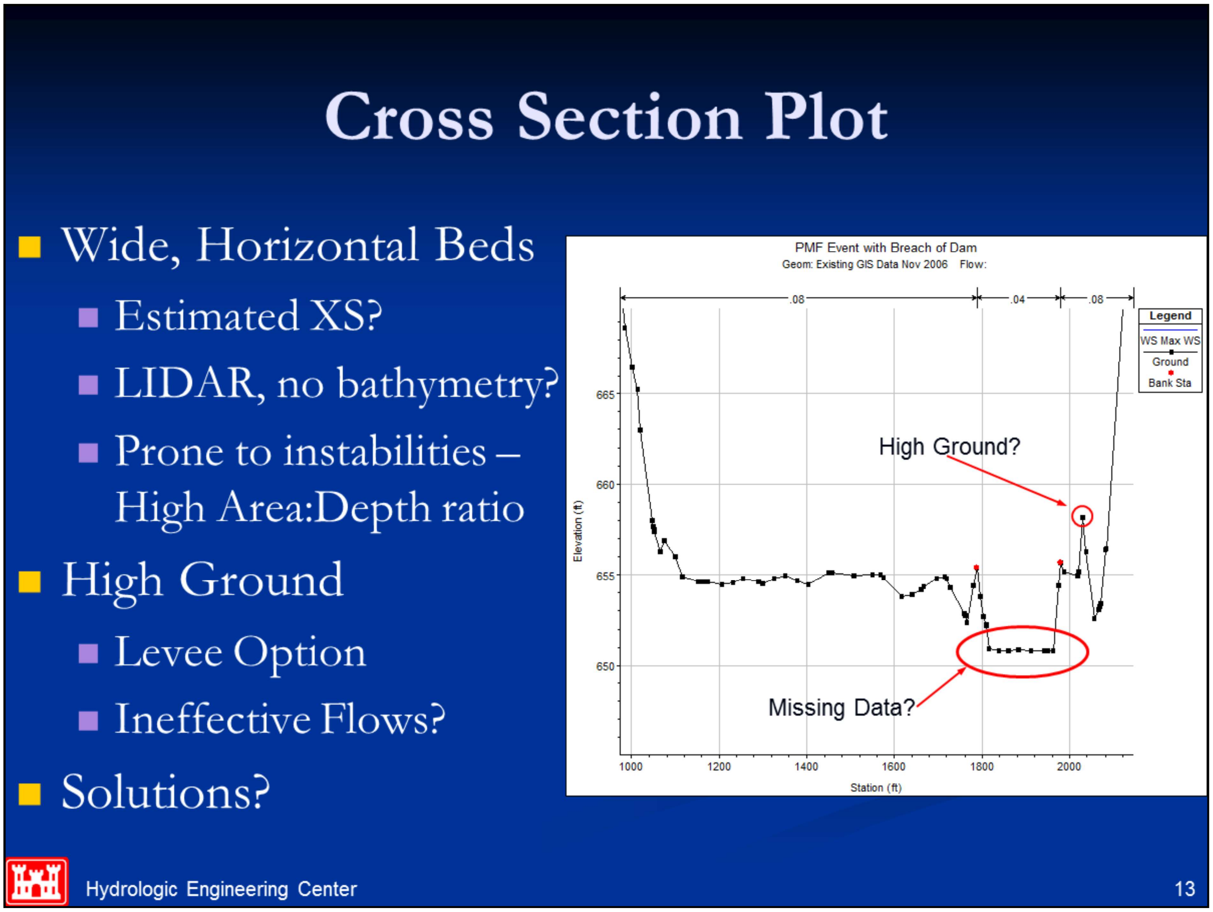
Utilizing the Cross Section Plot

- Can help spot isolated problems such as:
  - Incorrect Bank Station locations
  - Bad Manning's n Values
  - Bad Station-elevation points
- Can help spot transition problems
  - Contraction/Expansion Areas
  - Ineffective Flow Areas
  - Levees



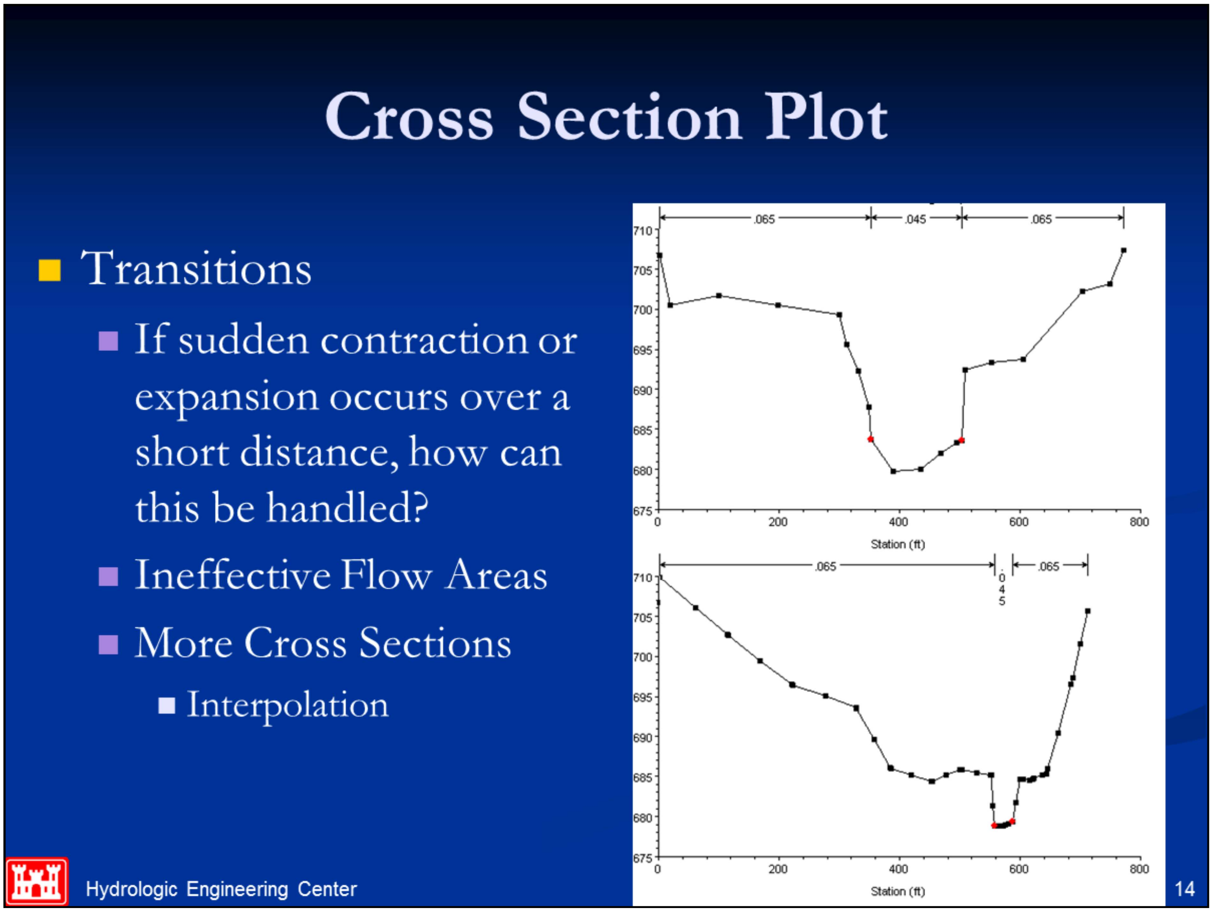
Hydrologic Engineering Center 12

Once a location for an instability is determined on the profile plot, the cross section plot can be used to investigate the cause of the instability. The cross section plot will show isolated problems such as incorrectly placed bank stations, poor n-values, and bad station-elevation data. In addition, scrolling through its neighboring cross sections can give you an idea of transition problems like contractions and expansions that occur to abruptly, poorly defined ineffective flow areas, or incorrectly handled levees or natural high ground spots.

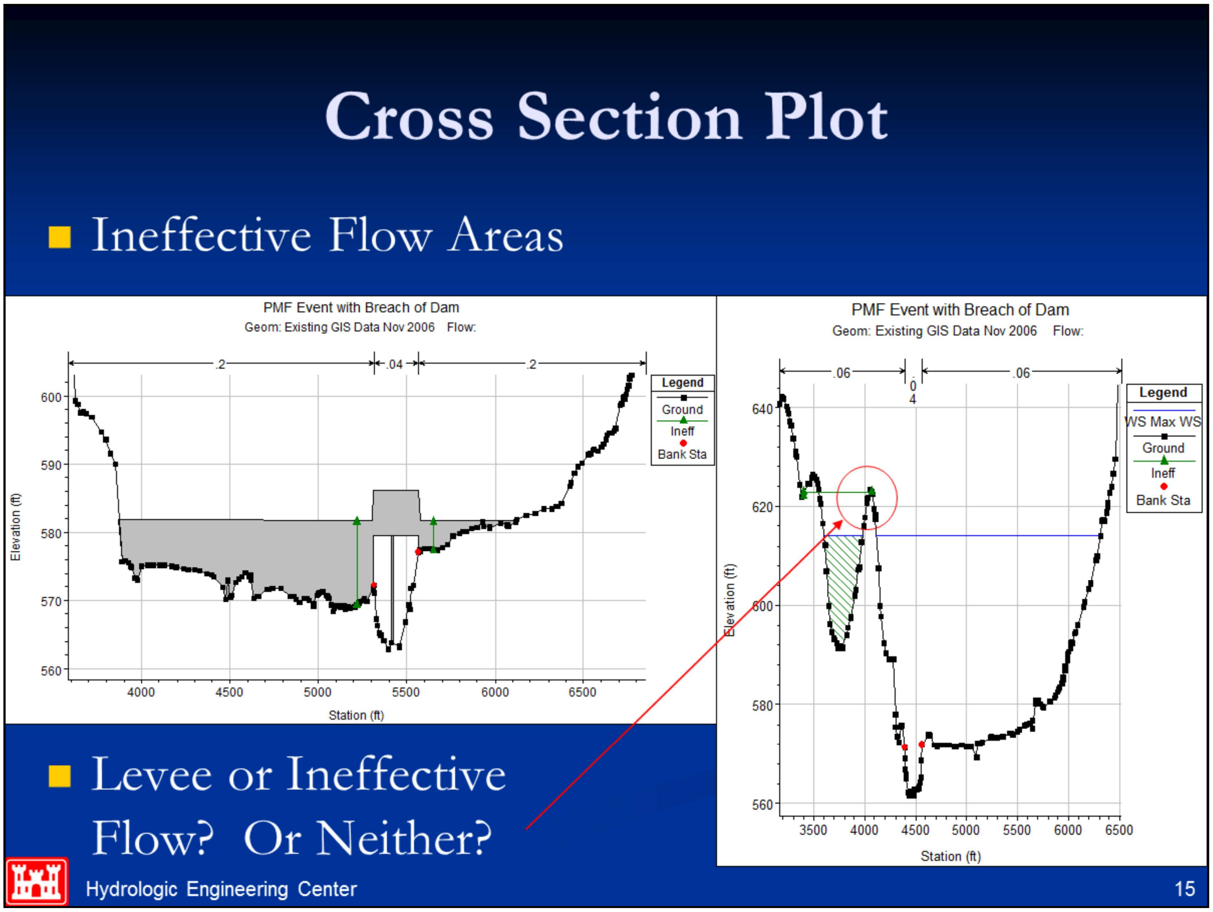


Another typical source of instabilities occurs when the main channel has a wide flat bed. This is usually found when cross sections are approximated or when terrain data is used to develop cross sections exclusive of real bathymetric data. Many times reaches are developed in GIS using LIDAR data or other aerial means. These survey methods don't penetrate water surfaces so the main channel is left with a flat horizontal bed equal to the water surface elevation. For shallow streams in dam breach analyses, this is normally okay, since the dam break flood wave is usually much greater than the depth of water. However, wide flat stream beds tend to cause instabilities because at lower flows, the area to depth ratio is very high. Again this presents the same problem of a small increase in depth amounting to a large relative increase.

Additionally, in the cross section plot, high ground that is not appropriately accounted for can be detected and fixed to remove sources of instabilities. High ground can be modeled as levees or with ineffective flows to remove the abrupt changes in storage and conveyance when the high ground is overtopped.



The example in this slide shows an abrupt transition from a wide main channel to a narrow main channel. If these cross sections are close enough, the flow man not be able to contract so suddenly and the approximate numerical methods may not be able to handle this situation. In this case, ineffective flow areas can be placed in the wide cross section to help smooth the transition from wide to narrow. If these cross sections are far enough apart, then perhaps additional interpolated cross sections are warranted.



Ineffective flow areas are required up and downstream of bridges and culverts to properly define the contraction and expansion zones. Unsteady flow models, and particularly dam breach models, need these zones to be adequately defined. When the bridge is overtopped, the ineffective flow areas will turn off. This sudden and large increase in conveyance can cause model instability. One solution is to use very high Manning's n values (.2 to 1.0) in the ineffective flow zones, so when they turn off the increase in conveyance is not so great. This is also more physically appropriate as the cross sections just upstream and downstream can not flow completely freely because of the bridge embankment.

When an isolated high ground area is causing an instability problem, the user must decide if this high ground is better modeled with the levee option or with ineffective flow.

Profile Summary Tables

- Sometimes visual clues are not available. Tabular output help.
  - lateral inflow/outflow
    - Tributaries
    - Interaction with storage areas
  - Lateral structure flow
  - Inline structure flow
  - Flow inconsistency
    - Main channel to overbanks
  - Other internal boundaries
    - Groundwater

Profile Output Table - Standard Table 1

File Options Std. Tables Locations Help

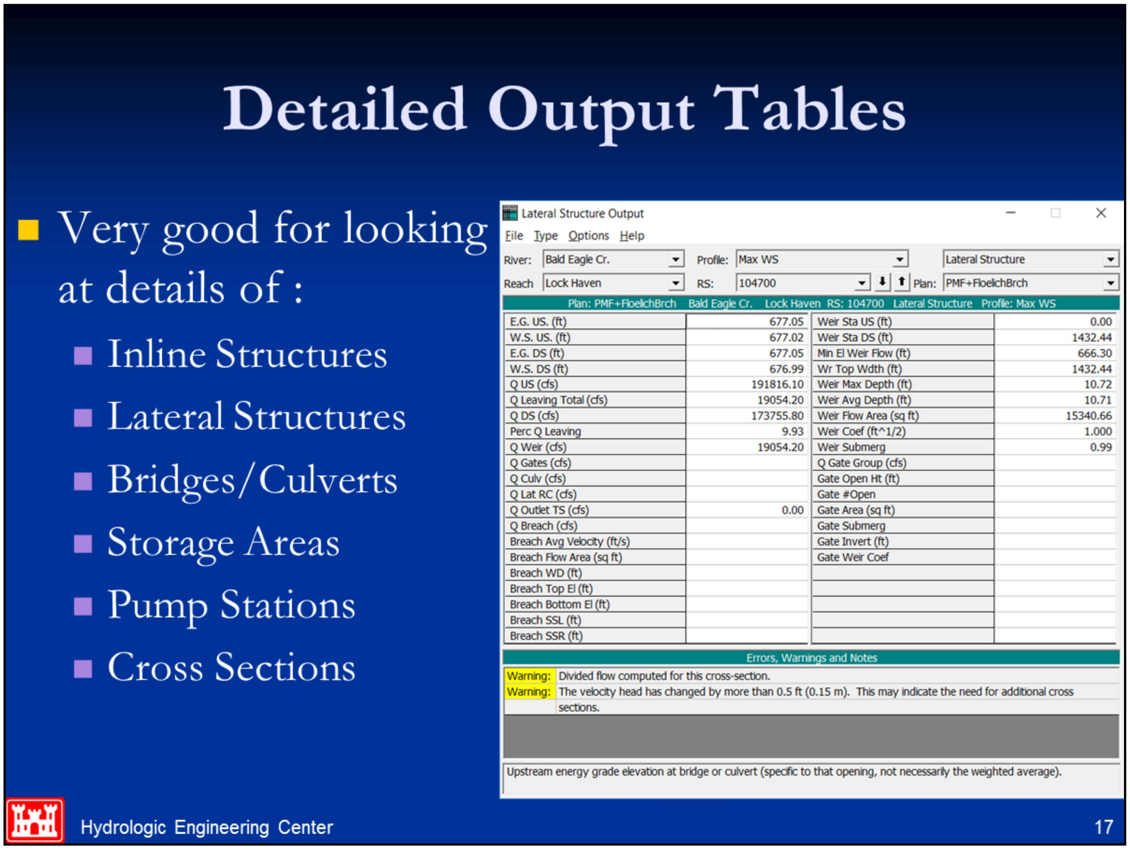
HEC-RAS Plan: PMF+FloelichBrch River: Bald Eagle Cr. Reload Data

Reach	River Sta	Profile	Q Total (cfs)	Min Ch El (ft)	W.S. Elev (ft)	Crit W.S. (ft)
Lock Haven	23595	03JAN1999 0300	239571.00	540.19	584.36	
Lock Haven	23191	03JAN1999 0300	239258.80	542.27	584.32	
Lock Haven	23100		Lat Struct			
Lock Haven	22438	03JAN1999 0300	222235.40	541.28	584.10	
Lock Haven	21398	03JAN1999 0300	199445.90	538.19	583.64	566.19
Lock Haven	21324		Bridge			
Lock Haven	21266	03JAN1999 0300	199445.90	537.51	583.61	
Lock Haven	21200		Lat Struct			
Lock Haven	20741	03JAN1999 0300	193066.80	538.52	582.32	
Lock Haven	20095	03JAN1999 0300	186219.50	540.37	578.88	
Lock Haven	19487	03JAN1999 0300	183837.20	541.07	575.51	
Lock Haven	18700	03JAN1999 0300	181599.10	540.12	576.90	
Lock Haven	17988	03JAN1999 0300	177019.20	537.81	577.52	
Lock Haven	17256	03JAN1999 0300	171642.60	536.91	576.79	
Lock Haven	16852	03JAN1999 0300	169344.60	537.25	576.04	
Lock Haven	16517	03JAN1999 0300	167809.50	537.53	575.71	
Lock Haven	15496	03JAN1999 0300	163777.30	532.71	576.09	554.14
Lock Haven	15127		Bridge			
Lock Haven	14914	03JAN1999 0300	163777.30	533.26	575.37	
Lock Haven	14800		Lat Struct			

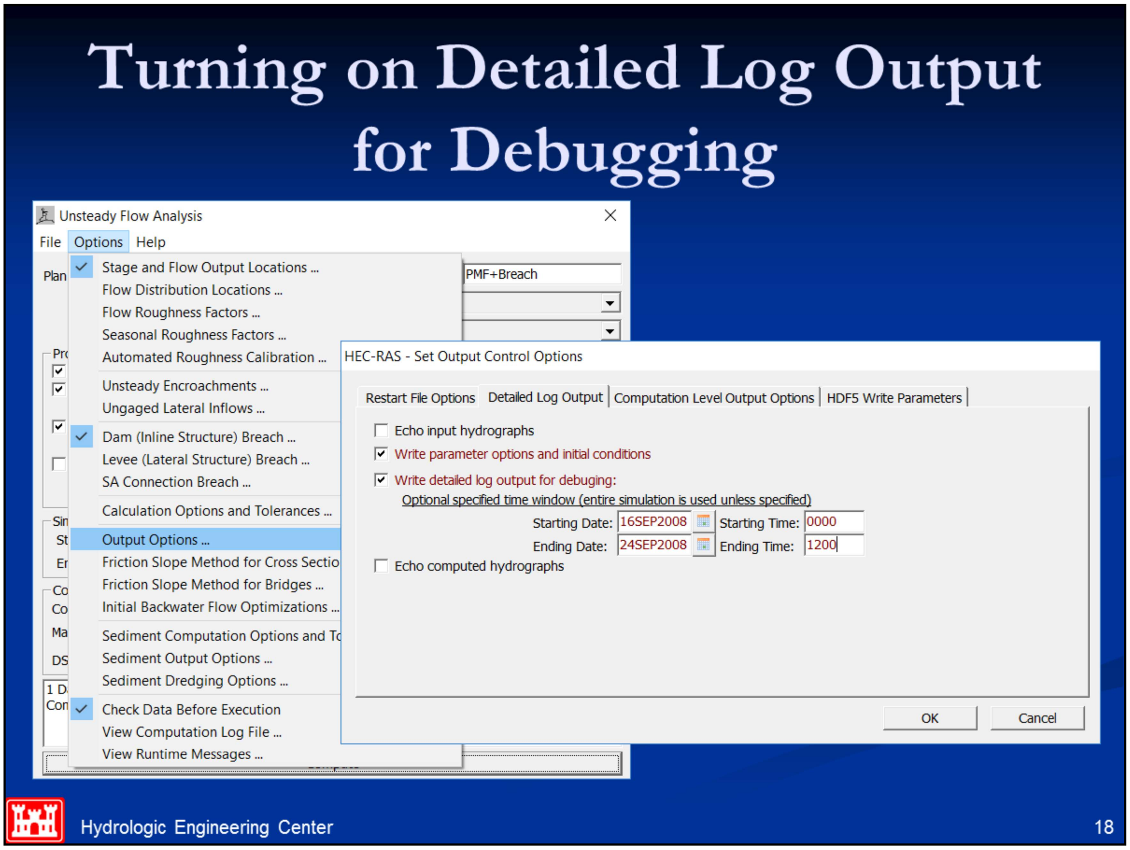
Total flow in cross section.

Hydrologic Engineering Center 16

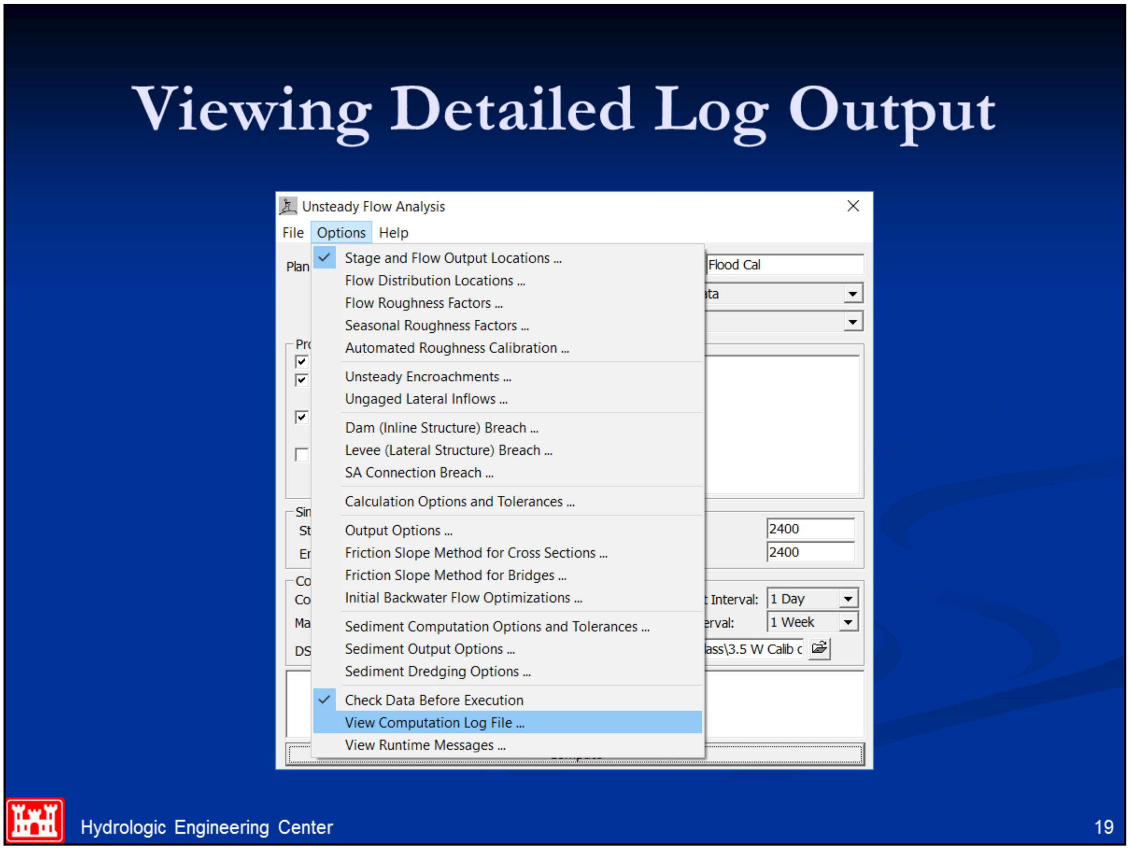
Often times the graphical options alone are not adequate to determine the source of instability. Another option is to go to the profile output table and analyze values of hydraulic parameters from one cross section to the next or from one profile to the next. Problems that don't always show up graphically are lateral inflows and outflows, groundwater interaction and the effects of lateral structures. It is imperative that the important hydraulic parameters (flow, depth, area, storage) change as gradually as possible. Flow consistency between the overbanks and the main channel is also important.



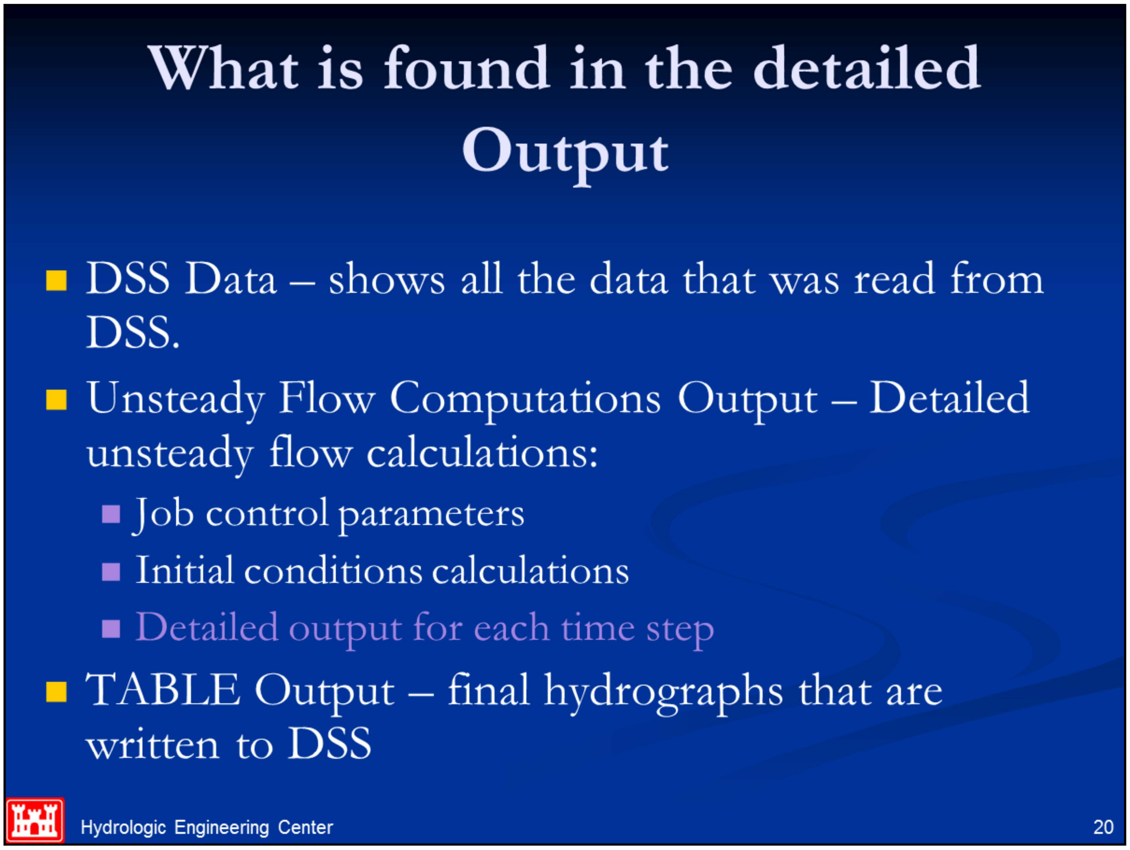
Detailed output tables are available for many types of nodes in HEC-RAS, including: cross sections, bridges, culverts, inline structures, lateral structures, storage areas, and pump stations. These detailed tables can be very helpful in seeing what is going on at that structure for a particular time step.



As shown in the figure above, the section at the bottom half of this editor is used for controlling the **detailed log output**. Three check boxes are listed. The first box can be used to turn on an echo of the hydrograph input to the model. This can be used to ensure that the model is receiving the correct flow data. The second check box can be used to turn on an echo of the computed hydrographs that will be written to the HEC-DSS. This is a good option for checking what was computed. However, if the user has selected to have hydrographs computed at many locations, this could end up taking a lot of file and disk space. The third check box is used to control the detailed output of results from the unsteady flow simulation. Selecting this options will cause the software to write detailed information on a time step by time step basis. This option is useful when the unsteady flow simulation is going unstable or completely blowing up (stopping). Checking this box turns on the detailed output for every time step. The user has the option to limit this output to a specific time window during the unsteady flow simulation. Limiting the log output is accomplished by entering a starting date and time and an ending date and time.



Viewing Detailed Log Output: After the user has turned on the detailed log output option, re-run the unsteady flow simulation. The user can then view the detailed log output by selecting **View Computational Log File** from the **Options** menu of the Unsteady flow simulation window. When this option is selected the detailed log output file will be loaded into the default text file viewer for your machine (normally the NotePad.exe program, unless you have changed this option within HEC-RAS).



The detailed log output file will contain the following output:

DSS Output: Shows all of the hydrograph data that will be used as input to the model, including data read from HEC-DSS.

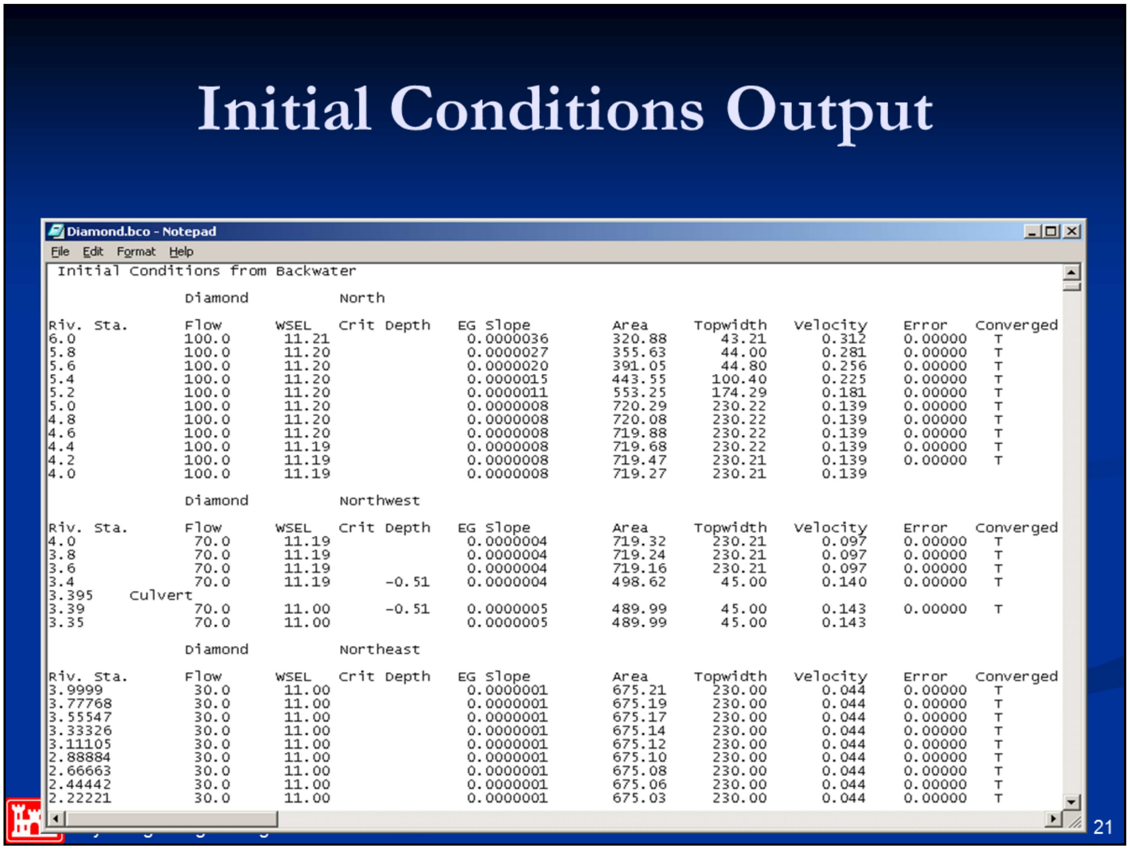
Unsteady Flow Computations Output: Detailed unsteady flow calculations including:

Job control parameters

Initial conditions calculations

Detailed output for each time step

Table Output: Final computed hydrographs that are written to HEC-DSS.



Initial Conditions Output

Diamond.bco - Notepad

Initial Conditions from Backwater

Diamond North

Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
6.0	100.0	11.21		0.0000036	320.88	43.21	0.312	0.00000	T
5.8	100.0	11.20		0.0000027	355.63	44.00	0.281	0.00000	T
5.6	100.0	11.20		0.0000020	391.05	44.80	0.256	0.00000	T
5.4	100.0	11.20		0.0000015	443.55	100.40	0.225	0.00000	T
5.2	100.0	11.20		0.0000011	553.25	174.29	0.181	0.00000	T
5.0	100.0	11.20		0.0000008	720.29	230.22	0.139	0.00000	T
4.8	100.0	11.20		0.0000008	720.08	230.22	0.139	0.00000	T
4.6	100.0	11.20		0.0000008	719.88	230.22	0.139	0.00000	T
4.4	100.0	11.19		0.0000008	719.68	230.22	0.139	0.00000	T
4.2	100.0	11.19		0.0000008	719.47	230.21	0.139	0.00000	T
4.0	100.0	11.19		0.0000008	719.27	230.21	0.139		

Diamond Northwest

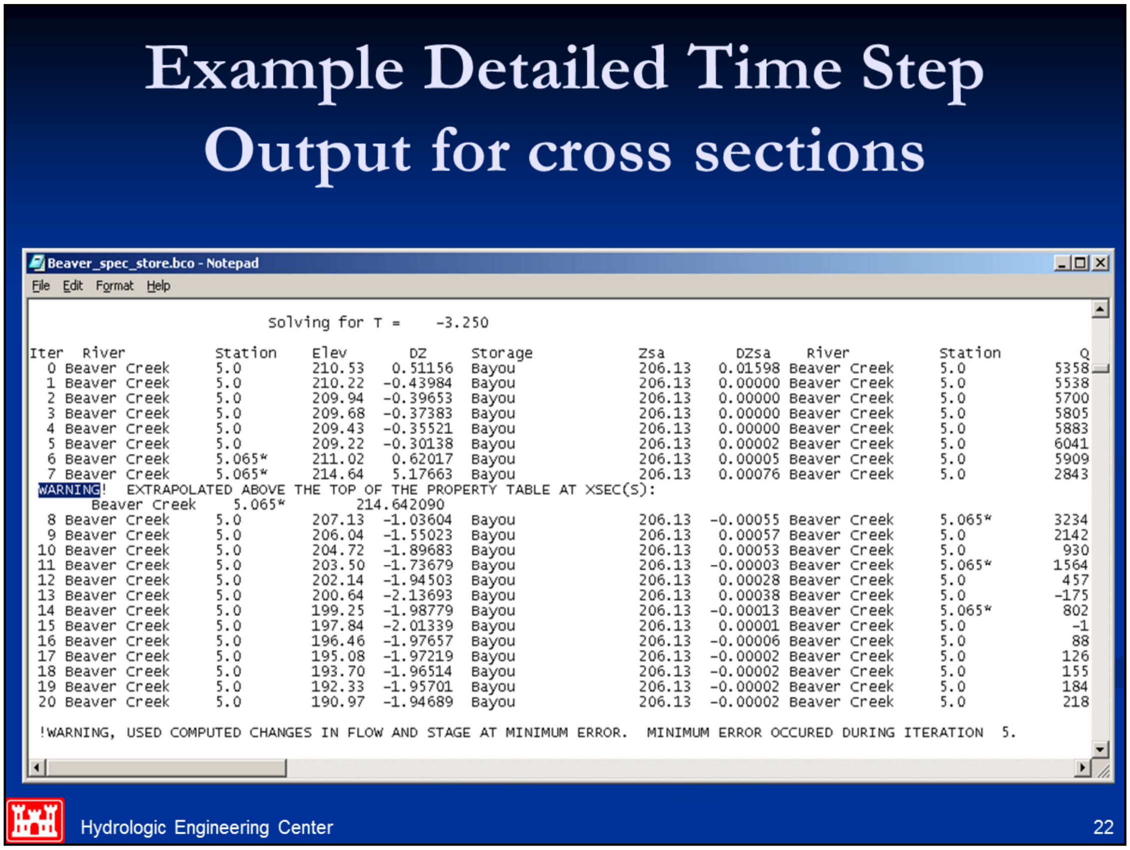
Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
4.0	70.0	11.19		0.0000004	719.32	230.21	0.097	0.00000	T
3.8	70.0	11.19		0.0000004	719.24	230.21	0.097	0.00000	T
3.6	70.0	11.19		0.0000004	719.16	230.21	0.097	0.00000	T
3.4	70.0	11.19	-0.51	0.0000004	498.62	45.00	0.140	0.00000	T
3.395 Culvert									
3.39	70.0	11.00	-0.51	0.0000005	489.99	45.00	0.143	0.00000	T
3.35	70.0	11.00		0.0000005	489.99	45.00	0.143		

Diamond Northeast

Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
3.9999	30.0	11.00		0.0000001	675.21	230.00	0.044	0.00000	T
3.77768	30.0	11.00		0.0000001	675.19	230.00	0.044	0.00000	T
3.55547	30.0	11.00		0.0000001	675.17	230.00	0.044	0.00000	T
3.33326	30.0	11.00		0.0000001	675.14	230.00	0.044	0.00000	T
3.11105	30.0	11.00		0.0000001	675.12	230.00	0.044	0.00000	T
2.88884	30.0	11.00		0.0000001	675.10	230.00	0.044	0.00000	T
2.66663	30.0	11.00		0.0000001	675.08	230.00	0.044	0.00000	T
2.44442	30.0	11.00		0.0000001	675.06	230.00	0.044	0.00000	T
2.22221	30.0	11.00		0.0000001	675.03	230.00	0.044	0.00000	T

21

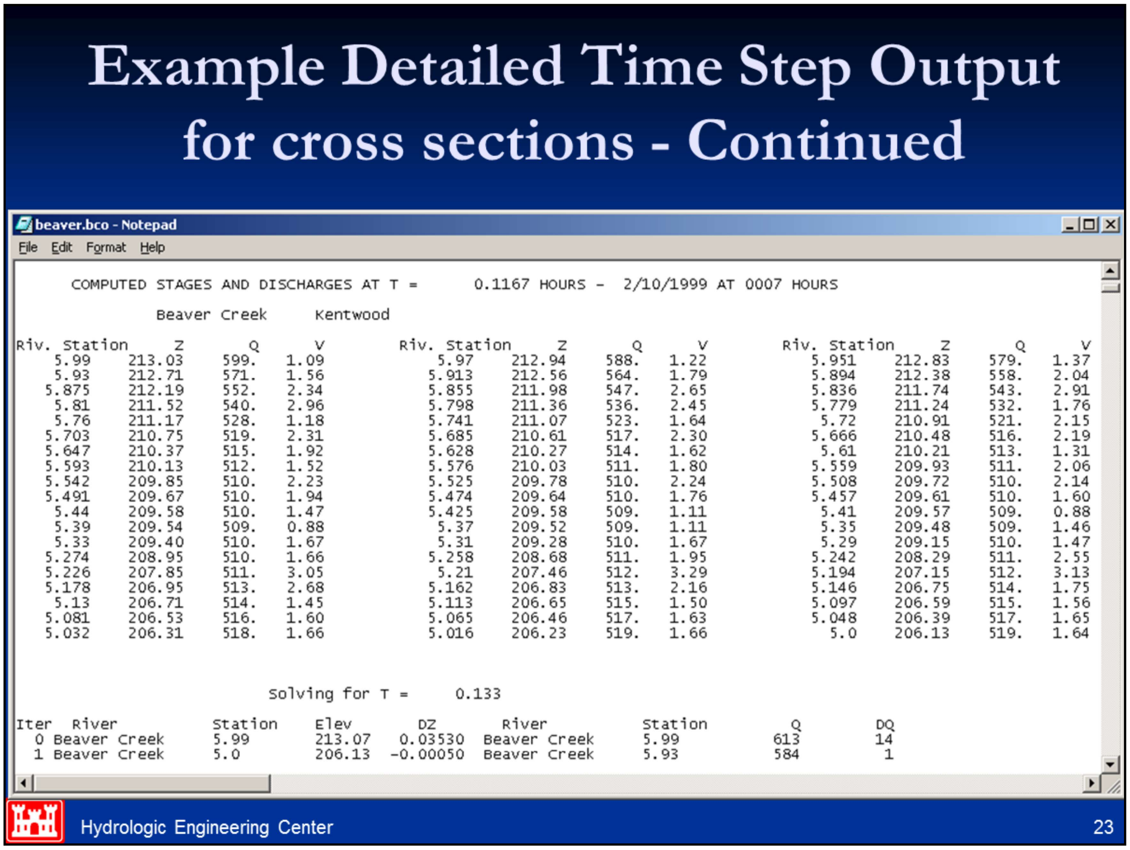
The program lists the computed initial conditions from a backwater calculation for each of the river/reaches. They are generally listed in an upstream to downstream order. However, they are computed from downstream to upstream under the assumption of subcritical flow.



One way to find and locate potential stability problems with the solution is to do a search in the file for the word "**WARNING**". The user then needs to look at the detailed output closely to try and detect both where and why the solution is going bad. The variables that are printed out during the iterations are the following:

Iter	= Iteration Number.
River	= River Name of the location with the largest error in stage.
Station	= River station with the larges error in the calculated stage.
ELEV	= Computed water surface elevation at that river station.
DZ	= The "Numerical Error" in the computed stage at that location.
Storage	= Name of the storage area that has the larges error for this iteration.
Zsa	= Computed elevation of the storage area.
Dzsa	= The "Numerical Error" in the computed storage area elevation.
River	= River Name of the location with the largest error in flow.
Station	= River station with the largest error in the calculation of flow.
Q	= Computed flow
DQ	= The "Numerical Error" in the computed flow at the listed river station

Note: If the program goes to the maximum number of iterations, it will choose the iteration that had the minimum amount of error, set that as the solution for the current time step, and then go to the next time step.



During the unsteady flow computations, the program will output detailed information for cross sections, bridges/culverts, inline weir/spillways, lateral weir/spillways, storage areas, and storage area connections. This information should be reviewed closely when the software is having stability problems. An example of the detailed output for cross sections is shown in the Figure above.