C H A P T E R 8

Performing an Unsteady Flow Analysis

This chapter shows how to calculate unsteady flow water surface profiles. The chapter is divided into two parts. The first part explains how to enter unsteady flow data and boundary conditions. The second part describes how to develop a plan and perform the calculations.

Contents

■ Entering and Editing Unsteady Flow Data

■ Performing Unsteady Flow Calculations

■ Calibration of Unsteady Flow Models

■ Model Accuracy, Stability, and Sensitivity

Entering and Editing Unsteady Flow Data

Once all of the geometric data are entered, the modeler can then enter any unsteady flow data that are required. To bring up the unsteady flow data editor, select **Unsteady Flow Data** from the **Edit** menu on the HEC-RAS main window. The Unsteady flow data editor should appear as shown in Figure 8.1.

Unsteady Flow Data

The user is required to enter boundary conditions at all of the external boundaries of the system, as well as any desired internal locations, and set the initial flow and storage area conditions at the beginning of the simulation.

Boundary conditions are entered by first selecting the **Boundary Conditions** tab from the Unsteady Flow Data editor. River, Reach, and River Station locations of the external bounds of the system will automatically be entered into the table. Boundary conditions are entered by first selecting a cell in the table for a particular location, then selecting the boundary condition type that is desired at that location. Not all boundary condition types are available for use at all locations. The program will automatically gray-out the boundary condition types that are not relevant when the user highlights a particular location in the table. Users can also add locations for entering internal boundary conditions. To add an additional boundary condition location, select the desired River, Reach, and River Station, then press the **Add a Boundary Condition Location** button.

Figure 8.1 Unsteady Flow Data Editor

Boundary Conditions

There are several different types of boundary conditions available to the user. The following is a short discussion of each type:

Flow Hydrograph:

A flow hydrograph can be used as either an upstream boundary or downstream boundary condition, but is most commonly used as an upstream boundary condition. When the flow hydrograph button is pressed, the window shown in Figure 8.2 will appear. As shown, the user can either read the data from a HEC-DSS (HEC Data Storage System) file, or they can enter the hydrograph ordinates into a table. If the user selects the option to read the data from DSS, they must press the "**Select DSS File and Path**" button. When this button is pressed a DSS file and pathname selection screen will

appear as shown in Figure 8.3. The user first selects the desired DSS file by using the browser button at the top. Once a DSS file is selected, a list of all of the DSS pathnames within that file will show up in the table. The user can use the pathname filters to reduce the number of pathnames shown in the table. The last step is to select the desired DSS Pathname and to close the window.

Figure 8.2 Example Flow Hydrograph Boundary Condition

The user also has the option of entering a flow hydrograph directly into a table, as shown in Figure 8.2. The first step is to enter a "**Data time interval**." Currently the program only supports regular interval time series data. A list of allowable time intervals is shown in the drop down window of the data interval list box. To enter data into the table, the user is required to select either "**Use Simulation Time**" or "**Fixed Start Time**." If the user selects "Use Simulation Time", then the hydrograph that they enter will always start at the beginning of the simulation time window. The simulation starting date and time is shown next to this box, but is grayed out. If the user selects "Fixed Start Time" then the hydrograph is entered starting at a user specified time and date. Once a starting date and time is selected, the user can
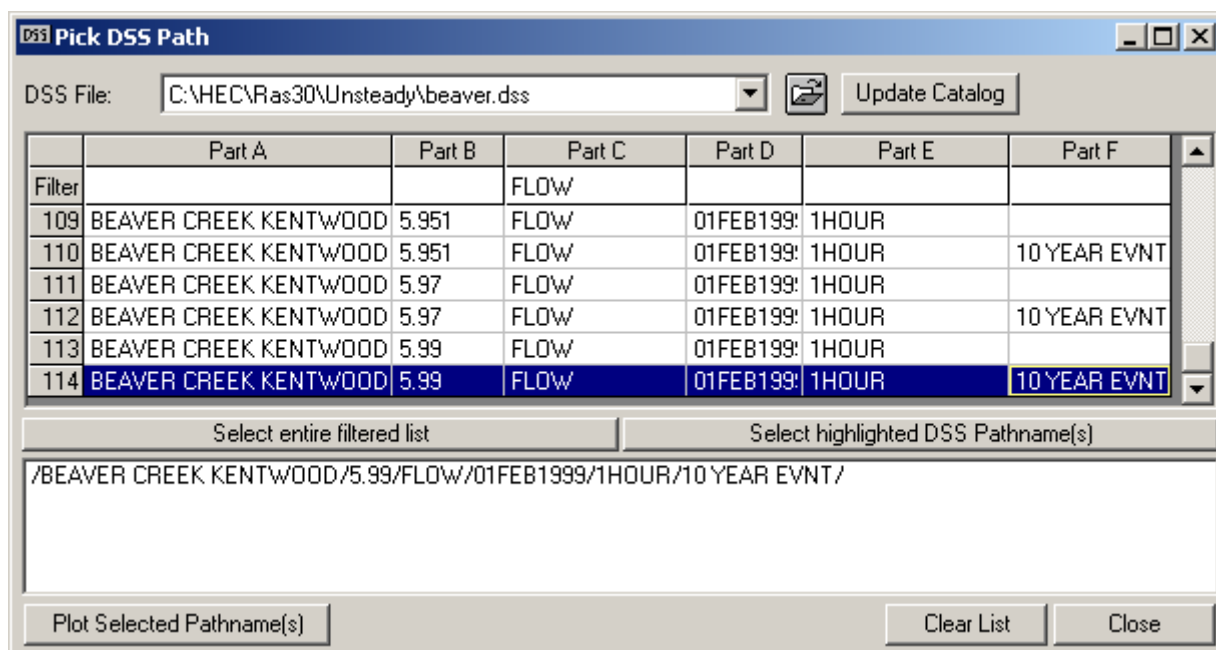
then begin entering the data.

Figure 8.3 HEC-DSS File and Pathname Selection Screen

An option listed at the bottom of the flow hydrograph boundary condition is to make this boundary a "**Critical Boundary Condition**." When you select this option, the program will monitor the inflow hydrograph to see if a change in flow rate from one time step to the next is exceeded. If the change in flow rate does exceed the user entered maximum, the program will automatically cut the time step in half until the change in flow rate does not exceed the user specified max. Large changes in flow can cause instabilities. The use of this feature can help to keep the solution of the program stable. This feature can be used at multiple hydrographs simultaneously. The software will evaluate all of the hydrographs, then calculate a time slice based on the hydrograph with the largest percentage increase over the user specified maximum flow change.

Two other options at the bottom of this editor are "**Min Flow**" and "**Multiplier**." Both of these options apply to user entered hydrographs or hydrographs read from HEC-DSS. The "Min Flow" option allows the user to specify a minimum flow to be used in the hydrograph. This option is very useful when too low of a flow is causing stability problems. Rather than edit the user entered hydrograph or the DSS file (depending upon where the hydrograph is coming from), the user can enter a single value, and all values below this magnitude will be changed to that value. The "Multiplier" option allows the user to multiply every ordinate of the hydrograph by a user specified factor. This factor will be applied to the user-entered hydrograph or a hydrograph read from HEC-DSS.

Stage Hydrograph:

A stage hydrograph can be used as either an upstream or downstream boundary condition. The editor for a stage hydrograph is similar to the flow hydrograph editor (Figure 8.2). The user has the choice of either attaching a HEC-DSS file and pathname or entering the data directly into a table.

Stage and Flow Hydrograph:

The stage and flow hydrograph option can be used together as either an upstream or downstream boundary condition. The upstream stage and flow hydrograph is a mixed boundary condition where the stage hydrograph is inserted as the upstream boundary until the stage hydrograph runs out of data; at this point the program automatically switches to using the flow hydrograph as the boundary condition. The end of the stage data is identified by the HEC-DSS missing data code of "-901.0". This type of boundary condition is primarily used for forecast models where the stage is observed data up to the time of forecast, and the flow data is a forecasted hydrograph.

Rating Curve:

The rating curve option can be used as a downstream boundary condition. The user can either read the rating curve from HEC-DSS or enter it by hand into the editor. Shown in Figure 8.4 is the editor with data entered into the table. The downstream rating curve is a single valued relationship, and does not reflect a loop in the rating, which may occur during an event. This assumption may cause errors in the vicinity of the rating curve. The errors become a problem for streams with mild gradients where the slope of the water surface is not steep enough to dampen the errors over a relatively short distance. When using a rating curve, make sure that the rating curve is a sufficient distance downstream of the study area, such that any errors introduced by the rating curve do not affect the study reach.

Figure 8.4 Example Rating Curve Boundary Condition Editor

Normal Depth:

The Normal Depth option can only be used as a downstream boundary condition for an open-ended reach. This option uses Manning's equation to estimate a stage for each computed flow. To use this method the user is required to enter a friction slope for the reach in the vicinity of the boundary condition. The slope of the water surface is often a good estimate of the friction slope.

As recommended with the rating curve option, when applying this type of boundary condition it should placed far enough downstream, such that any errors it produces will not affect the results at the study reach.

Lateral Inflow Hydrograph:

The Lateral Inflow Hydrograph is used as an internal boundary condition. This option allows the user to bring in flow at a specific point along the stream. The user attaches this boundary condition to the river station of the cross section just upstream of where the lateral inflow will come in. The actual change in flow will not show up until the next cross section downstream from this inflow hydrograph. The user can either read the hydrograph from DSS or enter it by hand.

Uniform Lateral Inflow Hydrograph:

The Uniform Lateral Inflow Hydrograph is used as an internal boundary condition. This option allows the user to bring in a flow hydrograph and distribute it uniformly along the river reach between two specified cross sections.   The hydrograph for this boundary condition type can be either read in from DSS, or entered by hand into a table.

Groundwater Interflow:

The Groundwater Interflow option allows the user to identify a reach of river that will exchange water with a groundwater reservoir. The stage of the groundwater reservoir is assumed to be independent of the interflow from the river, and must be entered manually or read from DSS. The groundwater interflow is similar to a uniform lateral inflow in that the user enters an upstream and a downstream river station, in which the flow passes back and forth. The computed flow is proportional to the head between the river and the groundwater reservoir. The computation of the interflow is based on Darcy's equation. The user is required to enter Darcy's groundwater loss coefficient (hydraulic conductivity), a time series of stages for the groundwater aquifer, and the distance between the river and the location of the user entered groundwater aquifer stages (this is used to obtain a gradient for Darcy's equation).

Time Series of Gate Openings:

This option allows the user to enter a time series of gate openings for an inline gated spillway, lateral gated spillway, or a gated spillway connecting two storage areas. The user has the option of reading the data from a DSS file or entering the data into a table from within the editor. Figure 8.5 shows an example of the Times Series of Gate Openings editor. As shown in Figure 8.5, the user first selects a gate group, then either attaches a DSS pathname to that group or enters the data into the table. This is done for each of the gate groups contained within the particular hydraulic structure.

Figure 8.5 Example Time Series of Gate Openings Editor

Warning: Opening and closing gates to quickly can cause instabilities in the solution of the unsteady flow equations. If instabilities occur near gated locations, the user should either reduce the computational time step and/or reduce the rate at which gates are opened or closed.

Elevation Controlled Gate:

This option allows the user to control the opening and closing of gates based on the elevation of the water surface upstream of the structure. A gate begins to open when a user specified elevation is exceeded. The gate opens at a rate specified by the user. As the water surface goes down, the gate will begin to close at a user specified elevation. The closing of the gate is at a user specified rate (feet/min.). The user must also enter a maximum and minimum gate opening, as well as the initial gate opening. Figure 8.6 shows and example of this editor.

Figure 8.6 Elevation Controlled Gate Editor

Navigation Dam:

This option allows the user to define an inline gated structure as a hinge pool operated navigation dam. The user specifies stage and flow monitoring locations, as well as a range of stages and flow factors. This data is used by the software to make decisions about gate operations in order to maintain water surface elevations at the monitor locations. A detailed discussion about Navigation Dams can be found in Chapter 16 of the user's manual.

Internal Observed Stage and Flow Hydrograph:

This option allows the user to enter an observed stage hydrograph or a stage and flow hydrograph, to be used as an internal boundary condition just upstream of an inline structure (inline weir/spillway, bridge, or culvert). If only an observed stage hydrograph is entered, the user must have values to cover the complete simulation range. If a stage and flow hydrpograph is entered, the stage hydrograph is used as the observed boundary until the stage hydrograph runs out of data; afterward the flow hydrograph is used. The end of data in the stage hydrograph is identified by the HEC-DSS missing data code, -901.0. The stage and flow hydrographs can either be entered into a table or from HEC-DSS. The mixed boundary condition is primarily used for forecast models where the stage data is observed up to the forecast time and the flow hydrograph is the flow forecast.

Initial Conditions

In addition to the boundary conditions, the user must establish the initial conditions of the system at the beginning of the unsteady flow simulation. Initial conditions consist of flow and stage information at each of the cross sections, as well as elevations for any storage areas defined in the system. Initial conditions are established from within the Unsteady Flow Data editor

by selecting the **Initial Conditions** tab. After the Initial Conditions tab is selected, the Unsteady Flow Data editor will appear as shown in Figure 8.7.

As shown in Figure 8.7, the user has two options for establishing the initial conditions of the system. The first option is to enter flow data for each reach and have the program perform a steady flow backwater run to compute the corresponding stages at each cross section. This option also requires the user to enter a starting elevation for any storage areas that are part of the system. This is the most common method for establishing initial conditions. Flow data can be changed at any cross section, but at a minimum the user must enter a flow at the upper end of each reach.

Figure 8.7 Initial Conditions Editor

A second method is to read in a file of stages and flows that were written from a previous run, which is called a "Restart File." This option is often used when running a long simulation time that must be divided into shorter periods. The output from the first period is used as the initial conditions for the next period, and so on. Additionally, this option may be used when the

software is having stability problems at the very beginning of a run. Occasionally the model may go unstable at the beginning of a simulation because of bad initial conditions. When this happens, one way to fix the problem is to run the model with all the inflow hydrographs set to a constant flow, and set the downstream boundaries to a high tailwater condition. Then run the model and decrease the tailwater down to a normal stage over time (use a stage hydrograph downstream boundary to do this). Once the tailwater is decreased to a reasonable value, those conditions can be written out to a file, and then used as the starting conditions for the unsteady flow run.

Unsteady Flow Data Options

Several options are available from the Unsteady Flow Data editor to assist users in entering and viewing the data. These features can be found under the **Options** menu at the top of the window. The following options are available:

Delete Boundary Condition. This option allows the user to delete a boundary condition from the table. To use this option, first select the row to be deleted with the mouse pointer. Then select **Delete Boundary Condition** from the options menu. The row will be deleted and all rows below it will move up one. Only user inserted boundary conditions can be deleted from the table. If the boundary condition is an open end of the system, the system will not allow that boundary to be deleted. There must always be some type of boundary condition at all the open ends of the system.

Internal RS Initial Stages. This option allows the user to specify starting water surface elevations for any internal cross section within the system. A common application of this would be to specify the starting pool elevation for the first cross section upstream of a dam (modeled with the inline weir/spillway option). The user specifies locations and water surface elevations, which are then used to establish the initial conditions of the system at the beginning of a run.

Observed Data In DSS. This option allows the user to attach observed data pathnames from a HEC-DSS file to specific river stations within the model. When an observed data pathname is attached to a specific river station location, the user can get a plot of the observed flow or stage hydrograph on the same plot as the computed flow and stage hydrographs. Additionally the observed data will show up on profile and cross section plots.

To use this option, the user selects **Observed Data In DSS** from the **Options** menu of the Unsteady Flow Data editor. When this option is selected a window will appear as shown in Figure 8.8. As shown in the figure below, the user first selects a river, reach, and river station. Then the user presses the **Add selected location to table** button in order to select a location to attach observed data. This should be done for all the locations in which you have observed data. The next step is to open up the DSS file that contains the observed data. This is accomplished by pressing the open file button, which is next to the DSS filename field. When a DSS file is selected, a list of the available pathnames contained in that DSS file will show up in the lower

table. To attach a DSS pathname to a particular river station, first select the river station row from the upper table. Then select the DSS pathname row from the lower table. Finally, press the button labeled **Select DSS Pathname**. Repeat this process for every location in which you wish to attach observed data. If you are going to have more than one data type (such as stage and flow) at a particular river station, you must have two entries in the upper table for that river station.

Figure 8.8 Editor for Establishing Locations of Observed Data

Minimum Flow and Flow Ratio Table. This option brings up a global editor that will show all the locations in which flow hydrographs have been attached as boundary conditions. The editor allows the user to enter a minimum flow or a flow factor for each flow hydrograph boundary condition. The minimum flow option will prevent any flow read from either HEC-DSS or a user entered hydrograph from going lower than the user specified minimum. Values that are lower than the minimum specified are automatically changed to the minimum value. The flow factor option allows the user to specify a factor to be multiplied by all ordinates of the flow hydrograph. This option is commonly used in planning type studies for performing sensitivity analysis (i.e. what if the flow were 20% higher?).

Saving The Unsteady Flow Data

The last step in developing the unsteady flow data is to save the information to a file. To save the data, select the **Save Unsteady Flow Data As** from the **File** menu on the Unsteady Flow Data editor. A pop-up window will appear prompting you to enter a title for the data.

Performing Unsteady Flow Calculations

Once all of the geometry and unsteady flow data have been entered, the user can begin performing the unsteady flow calculations. To run the simulation, go to the HEC-RAS main window and select **Unsteady Flow Analysis** from the **Run** menu. The Unsteady Flow Analysis window will appear as in Figure 8.9 (except yours may not have a Plan title and short ID).
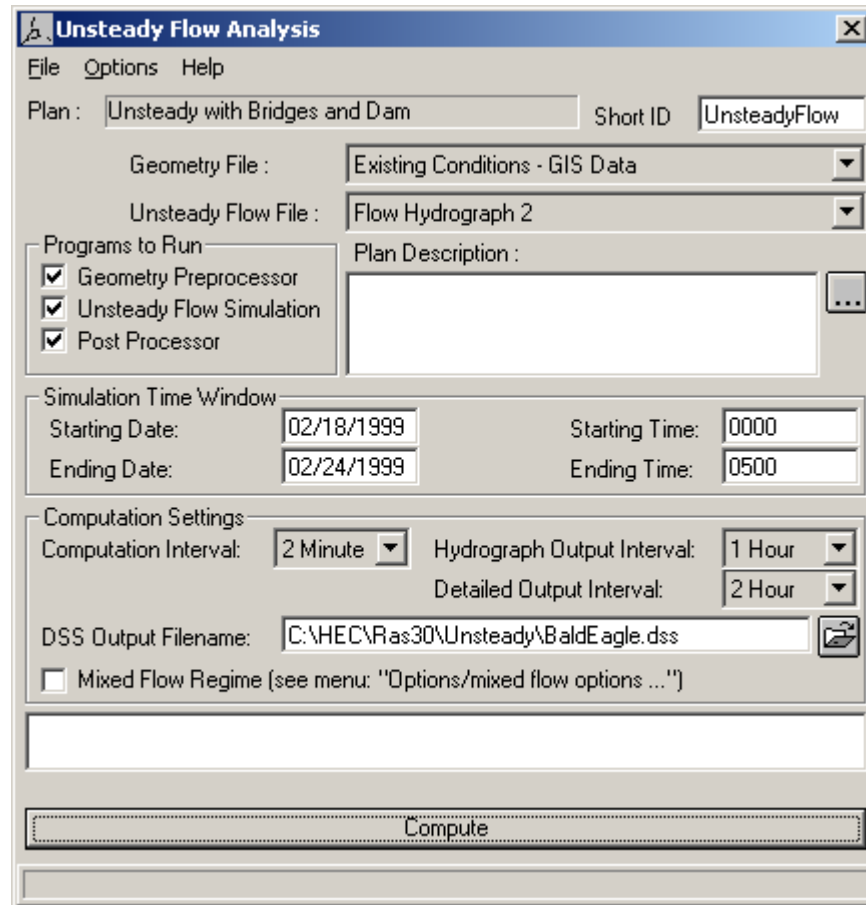
Figure 8.9 Unsteady Flow Analysis Window

Defining A Plan

The first step in performing a simulation is to put together a Plan. The Plan defines which geometry and unsteady flow data are to be used, as well as provides a description and short identifier for the run. Also included in the Plan information are the selected programs to be run; simulation time window; computation settings; and the simulation options.

Before a Plan is defined, the user should select which geometry and unsteady flow data will be used in the Plan. To select a geometry or unsteady flow file, press the down arrow button next to the desired data type. When this button is pressed, a list will appear displaying all of the available files of that type

that are currently available for the project. Select the geometry and unsteady flow file that you want to use for the current Plan.

To establish a Plan, select **Save Plan As** from the **File** menu on the Unsteady Flow Analysis window. When **Save Plan As** is selected, a window will appear prompting you to enter a title for the Plan. After you enter the title, press the **OK** button to close the window and accept the title. The user will also be prompted to enter a short identifier for the Plan. The short identifier is limited to 12 characters. It is very important to enter a short identifier that is descriptive of the Plan. When viewing multiple plan output from the graphics and tables, the Short ID will be used to identify each Plan.

Selecting Programs to Run

There are three components used in performing an unsteady flow analysis within HEC-RAS. These components are: a geometric data pre-processor; the unsteady flow simulator; and an output post-processor.

Geometric Pre-Processor

The pre-processor is used to process the geometric data into a series of hydraulic properties tables, rating curves, and family of rating curves. This is done in order to speed up the unsteady flow calculations. Instead of calculating hydraulic variables for each cross-section, during each iteration, the program interpolates the hydraulic variables from the tables. **The pre-processor must be executed at least once, but then only needs to be re-executed if something in the geometric data has changed.**

Cross sections are processed into tables of elevation versus hydraulic properties of areas, conveyances, and storage. Each table contains a minimum of 21 points (a zero point at the invert and 20 computed values), and can have up to a maximum of 100 points. The user is required to set an interval to be used for spacing the points in the cross section tables. The interval can be the same for all cross sections or it can vary from cross section to cross section. This interval is very important, in that it will define the limits of the table that is built for each cross section. On one hand, the interval must be large enough to encompass the full range of stages that may be incurred during the unsteady flow simulations. On the other hand, if the interval is to large, the tables will not have enough detail to accurately depict changes in area, conveyance, and storage with respect to elevation.

The interval for the cross section tables is defined as part of the geometric data. To set this interval, the user selects the **HTab Parameters** (**H**ydraulic **Tab**le Parameters) button from the Geometric Data editor. When this option is selected, a window will appear as shown in Figure 8.10.
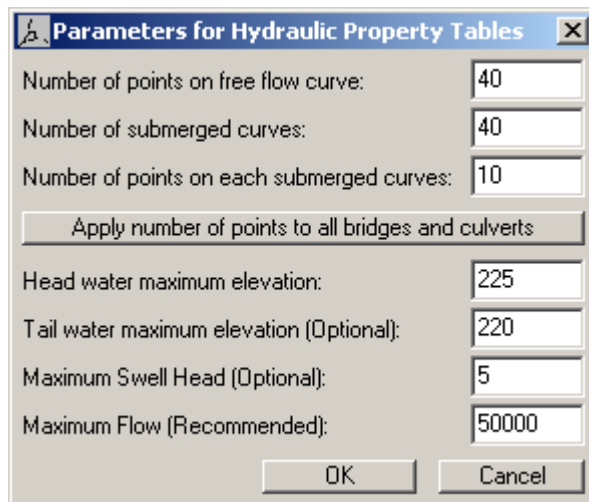
Figure 8.10 Hydraulic Table Parameters for Cross Sections

As shown in Figure 8.10, the table contains three columns in which the user can enter a Starting Elevation, Increment, and Number of Points. The first time the user opens this editor all of the columns are automatically filled. The starting elevation columns are automatically filled to an elevation one foot higher than the invert, however, the user can change these values to whatever they want. The second and third columns are used for the table increment and the number of points. These two variables will describe the extent to which the table encompasses the cross section data. A default value will be set for the increment and the number of points. Normally the increment will be set to one foot, and the number of points will be set to a value that will allow the table to extend to the top of the cross section. If this combination would end up with less than 20 points, then the number of points is set to 20 and the increment is reduced to get the table to the top of the cross section. The user can set these values individually for each cross section, or they can highlight a series of cross sections and use the **Set Values** button to enter the value for all of the highlighted sections. Other options are available to multiply highlighted fields by a factor or add a constant to all of them. Additionally, cut, copy, and paste buttons are available for manipulating the data.

Hydraulic structures, such as bridges and culverts, are converted into families of rating curves that describe the structure as a function of tailwater, flow, and headwater. The user can set several parameters that can be used in defining the curves. To set the parameters for the family of rating curves, the user can select the "**HTab Parameters"** button from the Bridge and Culvert editor or from the Hydraulic connection editor. When this button is pressed, the window in Figure 8.11 will appear:

Figure 8.11 Hydraulic Properties Table for Bridges/Culverts

As shown in Figure 8.11, the user can set the number of points to be computed on the free-flow rating curve (maximum of 50 points); the number of submerged curves to be computed (maximum of 50); and the number of points on the submerged curves (maximum of 20). The default values for these parameters are 40, 40, and 10 respectively. Additionally, the user can refine the curves by setting limits on the extent of the curves. This can be accomplished by entering the head water maximum elevation (reguired), tail water maximum elevation, maximum swell head (difference between the head water and tailwater), and the maximum possible flow (recommended). **In general, the curves will come out better if the user enters a headwater maximum elevation and a maximum flow rate.**

Structures that are gated, such as gated spillways, are not converted into curves because it would require a new family of curves for each possible gate setting. The hydraulics through gated structures is calculated on the fly during the unsteady flow calculations. No hydraulic table parameters are required for gated structures.

Unsteady Flow Simulation

The unsteady flow computations within HEC-RAS are performed by a modified version of the UNET (Unsteady NETwork model) program, developed by Dr. Robert Barkau (Barkau, 1992) and modified by HEC. The unsteady flow simulation is actually a three-step process. First a program called RDSS (Read DSS data) runs. This software reads data from a HEC-DSS file and then converts all of the boundary condition time series data into the user specified computation interval. Next, the UNET program runs. This software reads the hydraulic properties tables computed by the pre-processor, as well as the boundary conditions and flow data from the interface and the RDSS program. The program then performs the unsteady flow calculations. The final step is a program called TABLE. This software takes the results from the UNET unsteady flow run and writes them to a HEC-DSS file.

Post-Processor

The Post-Processor is used to compute detailed hydraulic information for a set of user specified time lines during the unsteady flow simulation period. In general, the unsteady flow computations only compute stage and flow at all of the computation nodes, as well as stage and flow hydrographs at user specified locations. **If the Post Processor is not run, then the user will only be able to view the stage and flow hydrographs and no other output from HEC-RAS**. By running the Post Processor, the user will have all of the available plots and tables for unsteady flow that HEC-RAS normally produces for steady flow.

By default, the Post-Processor will compute detailed output for a maximum stage water surface profile. This profile does not represent any specific instance in time, but rather represents a profile of the maximum stage that occurred at each cross section during the entire simulation. This profile is often useful for getting a quick view of the maximum extent of flooding during a specific event.

In addition to the maximum water surface profile, the user can request the software to write out a series of instantaneous profiles at a specific time interval. This is accomplished from the **Computation Settings** section of the **Unsteady Flow Analysis** window. The user turns on this option by selecting an interval from the box labeled **Detailed Output Interval.** The Post-Processor will then compute detailed output for each of the instantaneous profiles requested (Note: the Post-Processor is limited to 500 profiles). When the unsteady flow program runs, flow and stage water surface profiles are written to DSS for the entire system, starting with the beginning of the simulation and then at the user specified time interval for the entire simulation.

When the Post-Processor runs, the program reads from HEC-DSS the maximum water surface profile (stages and flows) and the instantaneous profiles. These computed stages and flow are sent to the HEC-RAS steady flow computation program SNET. Because the stages are already computed, the SNET program does not need to calculate a stage, but it does calculate all of the hydraulic variables that are normally computed. This consists of over two hundred hydraulic variables that are computed at each cross section for each flow and stage.

At hydraulic structures such as bridges and culverts, the unsteady flow program only reports the stage just upstream and downstream of the structure. During the Post-Processing of the results, the SNET program calculates the hydraulics of the structures by using the computed tailwater and flow, and then performing detailed hydraulic structure calculations. This is done so that the user can see detailed hydraulic information inside of the hydraulic structures for each of the profiles that are being post processed. However, this process can produce slightly different results for the upstream headwater elevation. Occasionally, you may notice a headwater elevation computed from the Post-Processor that is higher than the next upstream sections water surface. This difference is due to the fact that the unsteady flow simulation uses a pre-computed family of rating curves for the structure during the unsteady flow calculations. The program uses linear interpolation between the points of the rating curves to get the upstream headwater for a given flow and tailwater. The Post-Process performs the calculations through the structure and does not use rating curves (it solves the actual structure equations).

Once the Post-Processor is finished running, the user can view output from all of the HEC-RAS plots and tables. The maximum water surface profile and user specified instantaneous profiles can be viewed by selecting **Profiles** from the **Options** menu on each of the output windows (tables or plots). The overall maximum water surface profile will be labeled "**Max W.S.**", while the instantaneous profiles are labeled by the date and time. For example, a profile from January 5, 1999 at 1:00 p.m. would be labeled "**05Jan1999 1300**".

WARNING: Specifying a detailed output interval for post processing that is small can lead to long computational times and huge output files. Select this interval wisely, in that you only get detailed output when you really need it.

Simulation Time Window

The user is required to enter a time window that defines the start and end of the simulation period. The time window requires a starting date and time and an ending date and time. The date must have a four digit year and can be entered in either of the two following formats: **05Jan2000** or **01/05/2000**. The time field is entered in military style format (i.e. 1 p.m. is entered as 1300).

Computation Settings

The Computation Settings area of the Unsteady Flow Analysis window contains: the computational interval; hydrograph output interval; detailed output interval; the name and path of the output DSS file, and whether or not the program is run in a mixed flow regime mode.

The **computation interval** is used in the unsteady flow calculations. This is probably one of the most important parameters entered into the model. Choosing this value should be done with care and consideration as to how it will affect the simulation. The computation interval should be based on several factors. First, the interval should be small enough to accurately describe the rise and fall of the hydrographs being routed. A general rule of thumb is to use a computation interval that is equal to or less than the time of rise of the hydrograph divided by 24. In other words, if the flood wave goes from its base flow to its peak flow in 24 hours, then the computation interval should be equal to or less than 1 hour.

Additional considerations must be made for hydraulic structures, such as bridges, culverts, weirs, and gated spillways. Within bridges and culverts, when the flow transitions from unsubmerged to submerged flow, the water surface upstream of the structure can rise abruptly. This quick change in water surface elevation can cause the solution of the unsteady flow equations to go unstable. One solution to this problem is to use a very small time step, on the order of 1 to 5 minutes. This allows the module to handle the changes in stage in a more gradual manner. Additionally, when gates are opened or when flow just begins to go over a lateral weir, the change in stage and flow can be dramatic. Again, these types of quick changes in stage and flow can cause the solution of the unsteady flow equations to go unstable. The only solution to this problem is to shorten the computational time step to a very short interval. This may require the user to set the value as low as 1 to 5 minutes. The time step should be adjusted to find the largest value that will still solve the equations accurately. Additional variables that affect stability are the number of iterations and the Theta weighting factor. These two variables are discussed under the calculation tolerances section below.

The **Hydrograph Output Interval** is used to define at what interval the computed stage and flow hydrographs will be written to HEC-DSS. This interval should be selected to give an adequate number of points to define the shape of the computed hydrographs without losing information about the peak or volume of the hydrographs. This interval must be equal to or larger than the selected computation interval.

The **Detailed Output Interval** field allows the user to write out profiles of water surface elevation and flow at a user specified interval during the simulation. Profiles are not written for every computational time step because it would require too much space to store all of the information for most jobs. Also, when the Post-Processor is run, the program will compute detailed hydraulic information for each one of the instantaneous profiles that are written. This option is turned on by selecting an interval from the drop-down

box next to the detailed hydrograph output label. The selected interval must be equal to or greater than the computation interval. However, it is suggested that you make this interval fairly large, in order to reduce the amount of post-processing and storage required for a detailed hydraulic output. One example for selecting this variable would be, if the time window of the simulation was set at 72 hours, then one might want to set the instantaneous profiles to an interval of every 6 hours. This would equate to 13 profiles being written out and having detailed hydraulic information computed for them.

The field labeled **DSS Output Filename** is required before an execution can be made. The program will always write some results to a HEC-DSS file, so the user is required to select a path and filename to be used for this information.

Mixed Flow Regime. When this option is selected, the program will run in a mode such that it will allow subcritical flow, supercritical flow, hydraulic jumps, and draw downs (sub to supercritical transitions). **This option should only be selected if you actually have a mixed flow regime situation.** The methodology used for mixed flow regime analysis is called the **Local Partial Inertia (LPI)** solution technique (Fread, 1996). When this option is turned on, the program monitors the Froude number at all cross section locations for each time step. As the Froude number gets close to 1.0, the program will automatically reduce the magnitude of the inertial terms in the momentum equation. Reducing the inertial terms increases the models stability. When the Froude number is equal to or greater than 1.0, the inertial terms are completely zeroed out and the model is essentially reduced to a diffusion wave routing procedure. For Froude numbers close to 1.0, the program will use partial inertial effects, and when the Froude number is low, the complete inertial effects are used.

Note: more information about mixed flow regime calculations can be found in Chapter 16 of the HEC-RAS User's manual.

Simulation Options

From the **Options** menu of the Unsteady Flow Analysis window, the following options are available: stage and flow output locations; flow distribution locations; flow roughness factors; seasonal roughness factors; calculation options and tolerances; output options; checking data before execution, and viewing the computation log.

Stage and Flow Output Locations. This option allows the user to specify locations where they want to have hydrographs computed and available for display. By default, the program sets locations of the first and last cross section of every reach. To set the locations, the user selects **Stage and Flow Output Locations** from the **Options** menu of the Unsteady Flow Analysis window. When this option is selected a window will appear as shown in Figure 8.12.
As shown in Figure 8.12, the user can select individual locations, groups of

cross sections, or entire reaches. Setting these locations is important, in that, after a simulation is performed, the user will only be able to view stage and flow hydrographs at the selected locations.

Flow Distribution Locations. This option allows the user to specify locations in which they would like the program to calculate flow distribution output. The flow distribution option allows the user to subdivide the left overbank, main channel, and right overbank, for the purpose of computing additional hydraulic information.

The user can specify to compute flow distribution information for all the cross sections (this is done by using the Global option) or at specific locations in the model. The number of slices for the flow distribution computations must be defined for the left overbank, main channel, and the right overbank. The user can define up to 45 total slices. Each flow element (left overbank, main channel, and right overbank) must have at least one slice. The flow distribution output will be calculated for all profiles in the plan during the computations.
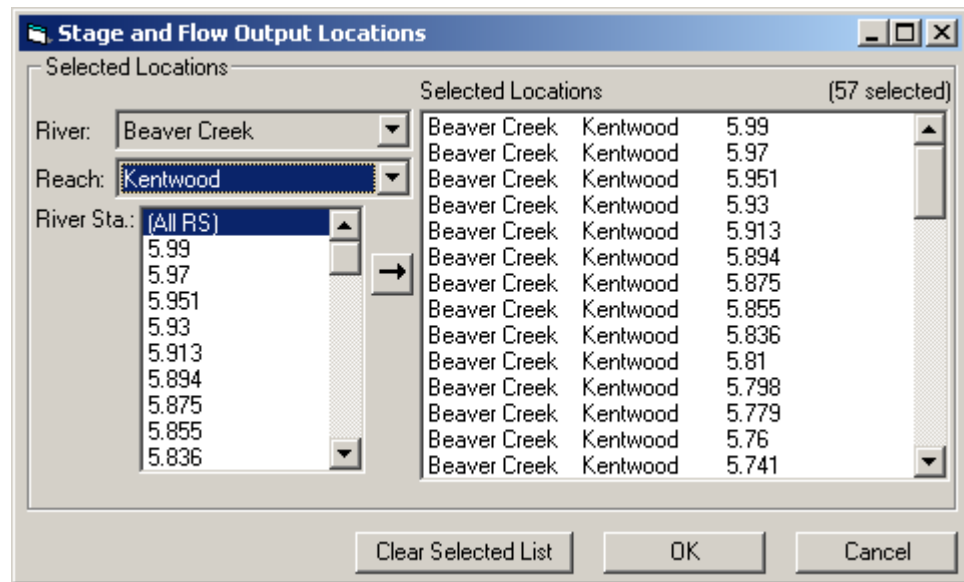
Figure 8.12 Stage and Flow Hydrograph Output Window

Flow Roughness Factors. This option allows the user to adjust roughness coefficients with changes in flow. This feature is very useful for calibrating an unsteady flow model for flows that range from low to high. Roughness generally decreases with increases flow and depth. This is especially true on larger river systems. This feature allows the user to adjust the roughness coefficients up or down in order to get a better match of observed data. To use this option, select **Flow Roughness Factors** from the **Options** menu of the Unsteady Flow Simulation manager. When this option is selected, a window will appear as shown in Figure 8.13.
As shown in Figure 8.13, the user first selects a river, reach, and a range of cross sections to apply the factors to. Next a starting flow, flow increment,

and a number of increments is entered. Finally, a roughness factor is entered into the table for each of the flows. The user can create several sets of these factors to cover a range of locations within the model. However, one set of factors cannot overlap with another set of factors. Hence, you can only apply one set of roughness change factors to any given cross section.

Figure 8.13 Flow versus Roughness Change Factors Editor

Seasonal Roughness Change Factors. This option allows the user to change roughness with time of year. This feature is most commonly used on larger river systems, in which temperature changes can cause changes in bed forms, which in turn causes changes in roughness. This factor can be applied in conjunction with the flow roughness change factors. When applying both, the seasonal roughness factor gets applied last.

To use this option, select **Seasonal Roughness Factors** from the **Options** menu of the Unsteady Flow Simulation manager. When this option is selected a window will appear as shown in Figure 8.14.

As shown in Figure 8.14, the user first selects a river, reach, and range of river station to apply the factors to. Next the user enters the day and month in the Day column, for each time that a new roughness factor will be entered. By default the program will automatically list the first of each month in this column. However, the user can change the day to whatever they would like. The final step is to then enter the roughness change factors.

Figure 8.14 Seasonal Roughness Factors Editor

Unsteady Flow Encroachments. This option allows the user to perform an encroachment analysis using the unsteady flow simulation option. Currently, encroachments are limited to method 1 within the unsteady flow analysis module. In general the user should first perform the encroachment analysis with the steady flow computations module, as documented in Chapter 10 of this manual. Once a good steady flow encroachment analysis is completed, the final encroachments can be imported into the unsteady flow plan for further analysis and refinement.   The user will need to have two unsteady flow plans, one without encroachments (representing the base flood) and one with encroachments (representing the encroached floodplain).

To add encroachments to an unsteady flow plan, the user selects **Unsteady Encroachments** from the **Options** menu of the Unsteady Flow Simulation editor. When this option is selected the following window will appear:

Figure 8.15 Unsteady Flow Encroachment Data Editor.

As shown in Figure 8.15, the user can enter a left station and a right station for the encroachments at each cross section. Additionally, the user has the option to import the encroachments calculated from a steady flow plan. This is accomplished by pressing the button labeled **Get Encroachments from Steady Flow Plan**, which is shown in the upper right part of the editor. When this button is pressed the user is asked to select a previously computed steady flow plan, and a specific profile from that plan. When the user presses the **OK** button, the program will go and get the final computed encroachments from that particular steady flow plan and profile.

Once all of the encroachments are entered, the user presses the **OK** button to have the interface accept the data. However, this information is not stored to the hard disk, the user must save the currently opened plan file for that to happen. The next step is to run the unsteady flow analysis with the encroachment data. The user should have two unsteady flow plans, one without encroachments and one with encroachments. Once both plans have been successfully executed, then comparisons between the plans can be made both graphically and in a tabular format.

Dam (Inline Structure) Breach. This option allows the user to perform a Dam Break analysis. The breach data is stored as "Plan" information. This is done so the user can try different breach locations, sizes, etc, without having to re-run the geometric pre-processor. Storing the data as Plan data is not that

important in a planning study, but it is very important when doing real time river forecasting. However, the user can get to the breach data in two different ways. First there is a button on the Inline Structure editor that is labeled **Breach** (Plan Data). Second, from the Unsteady Flow Simulation Manager, the user can select **Dam (Inline Weir) Breach** from the Options menu. When either option is selected, the following window will appear.
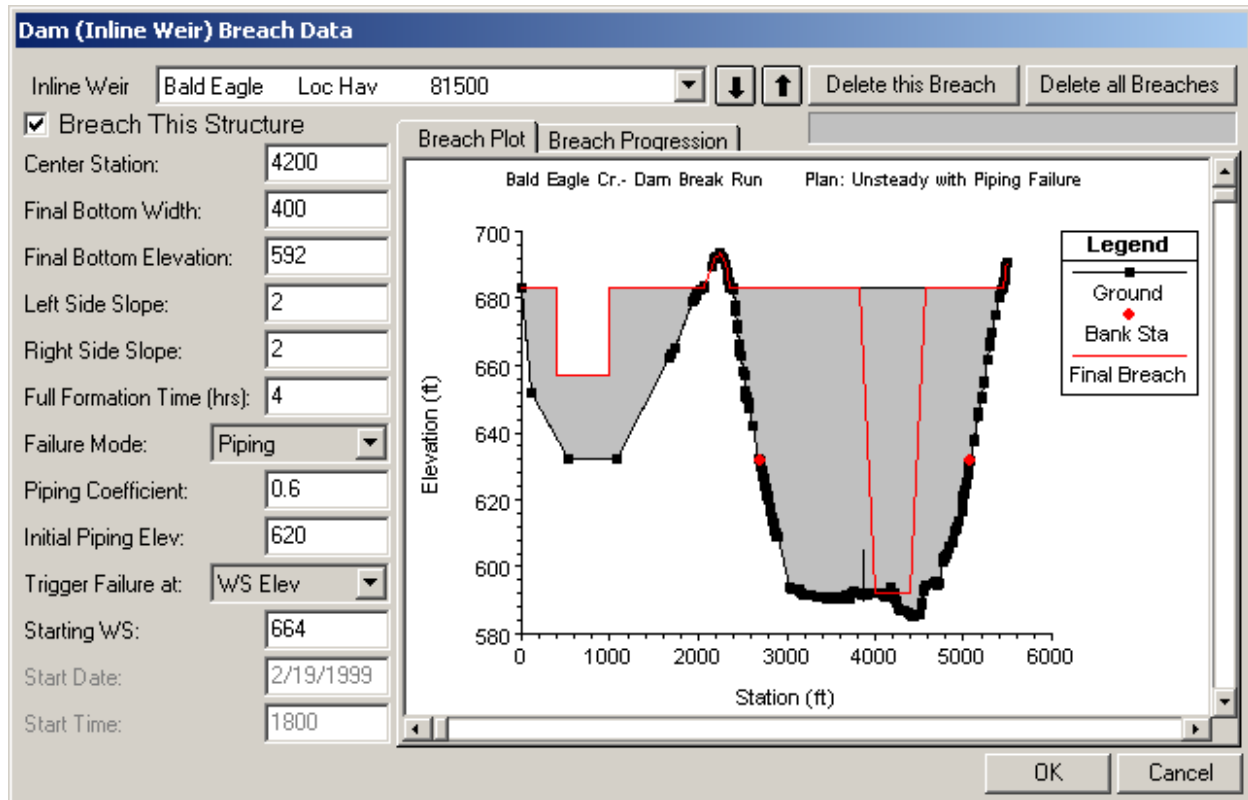
Figure 8.16 Dam Breach Editor.

As shown in Figure 8.16, the user selects a particular Inline Structure to perform the breach on. The following data must be entered for a breach:

Breach This Structure - This check box is used to decide if the program will perform the breach or not. In order for the breach to occur this box must be checked. This box was added to allow the user to turn certain breaches on or off, without loosing the user entered breach information.

Center Station - This field is used for entering the centerline stationing of the final breach.

Final Bottom Width - This field is used to enter the bottom width of the breach at its maximum size.

Final Bottom Elevation - This field is used to enter the elevation of the bottom of the breach after it has been fully developed.

Left Side Slope - This is the left side slope of the trapezoidal breach.

Right Side Slope - This is the right side slope of the trapezoidal breach.

Full Formation Time (hrs) - This field is used to enter the breach development time in hours. This time represents the duration from when the breach begins to have some significant erosion, to the full development of the breach.

Failure Mode - This option allows the user to choose between two different failure modes, an Overtopping failure and a Piping failure.

Piping Coefficient - If a piping failure mode is selected, the user must enter a piping coefficient. This coefficient is an Orifice coefficient, which is used while flow is coming out of the dam in a piping mode. Typical Orifice coefficients for a true designed orifice are around 0.8. However, for a piping breach, the coefficient should be lower to represent all of the additional energy losses occurring.

Initial Piping Elev. - If a piping failure mode is selected the user must enter an initial piping elevation. This elevation should be entered as the center of the piping flow while the breach develops.

Trigger Failure At - This field is used to select one of two trigger methods for initiating the breach. The two trigger methods are a water surface elevation or a specific time and date.

Starting WS - If the user selects water surface elevation for the failure trigger mode, then this field must be entered. This field represents the water surface elevation at which the breach should begin to occur.

Start Date - If the user selects a starting date and time as the failure trigger mode, then this field must be entered. This field is used to enter the date at which the breach will begin to occur.

Start Time - If the user selects a starting date and time as the failure trigger mode, then this field must be entered. This field is used to enter the time at which the breach will begin to occur.

In addition to all of the main breach information, the user also has the option to enter a user specified Breach Progression curve. By default the breach progression is assumed to be linear between the breach initiation and the full breach size (Full Formation Time). The user enters their breach progression curve by selecting the Breach Progression tab. When this tab is selected, the editor will now look like the following:
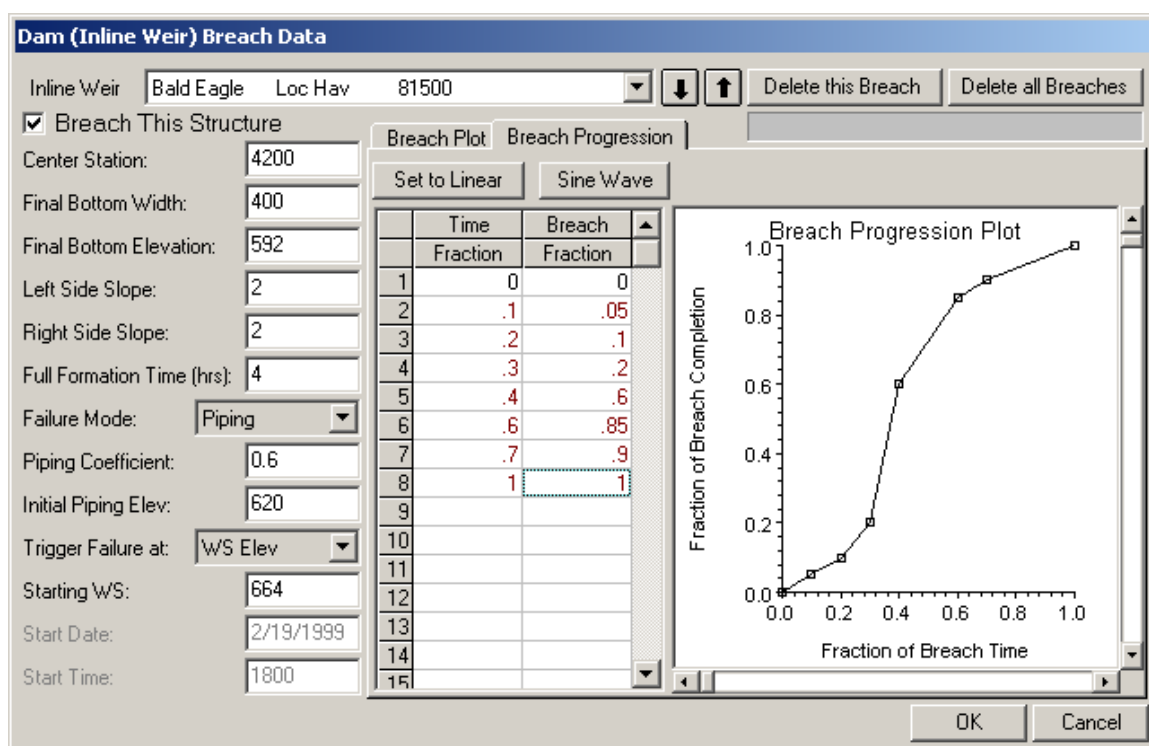
Figure 8.17 Dam Breach Editor with User Specified Breach Progression.

As shown in Figure 8.17, the user enters a Time Fraction (from zero to 1.0) and a Breach Fraction (from zero to 1.0). The user-entered data is plotted in the graphic next to the table. The breach progression curve is then used during the breach formation time to adjust the growth rate of the breach.

Note: More detailed information on performing a breach analysis can be found in the HEC-RAS User's Manual, chapter 16.

Levee (Lateral Structure) Breach. This option is very similar to the Dam Break option described previously. The only difference is that the breaching is performed on a levee. The options and data entered to describe the breach is the same as a Dam Break.

In order to use this option, the user must first define the levee as a lateral structure within HEC-RAS. The lateral weir profile is used to describe the top of the levee along the stream both at and between the cross sections. Second, a weir coefficient is entered for calculating the flow that may go overtop of the levee if the water surface gets high enough. Entering breach data for the levee can be accomplished from the lateral weir editor or from the **Levee (lateral structure) Breach** option from the Unsteady Flow Simulation window. The levee breaching data is stored as part of the unsteady flow plan file, just as it is for a dam break. When the levee breach option is selected, a breach editor will appear as shown in Figure 8.18.
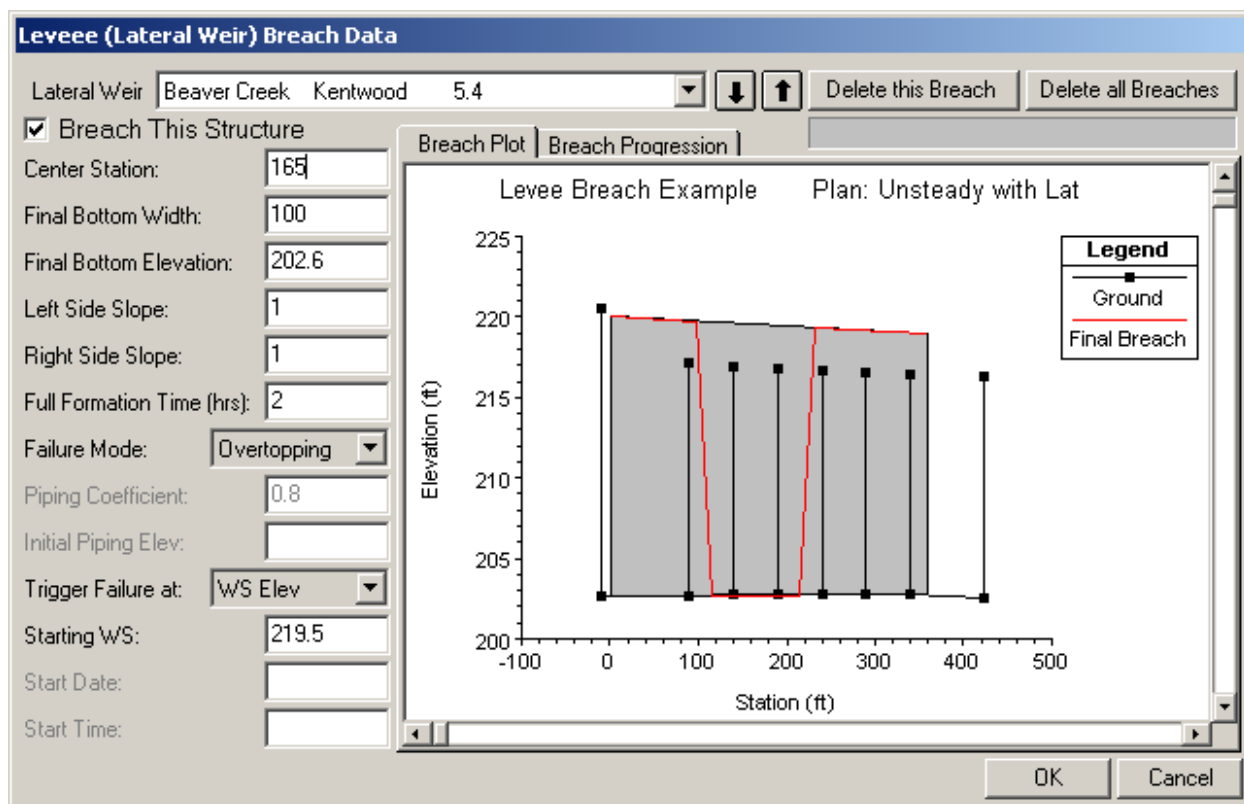
Figure 8.18 Levee Breaching Editor.

As shown in Figure 8.18, this editor contains the same information as the Dam Breach editor. For a description of the variables please review the section on Dam Breaching above. More detailed information about levee breaching can be found in Chapter 16 of this manual.

Mixed Flow Options. This option allows the user to change the parameters that control the computations of mixed flow regime within the unsteady flow simulation. This option was described previously in this chapter, under the section titled **Computational Settings**. Please review that section of this chapter for the details of how to use the mixed flow regime option and controlling the parameters.

Calculation Options and Tolerances. This option allows the user to set some computation options and to override the default settings for the calculation tolerances. These tolerances are used in the solution of the momentum equation. **Warning !!!** - Increasing the default calculation tolerances could result in computational errors in the water surface profile. The tolerances are as follows:

Theta implicit weighting factor: This factor is used in the finite difference solution of the unsteady flow equations. The factor ranges between 0.6 and 1.0. A value of 0.6 will give the most accurate solution of the equations, but is more susceptible to instabilities. A value of 1.0 provides the most stability in the solution, but may not be as accurate for some data sets. The default

value is set to 1.0. Once the user has the model up and running the way they want it, they should then experiment with changing theta towards a value of 0.6. If the model remains stable, then a value of 0.6 should be used. In many cases, you may not see an appreciable difference in the results when changing theta from 1.0 to 0.6. However, every simulation is different, so you must experiment with your model to find the most appropriate value.

Water surface calculation tolerance: This tolerance is used to compare the difference between the computed and assumed water surface elevations at cross sections. If the difference is greater than the tolerance, the program continues to iterate for the current time step. When the difference is less than the tolerance, the program assumes that it has a valid numerical solution. The default value is set to 0.02 feet.

Storage area elevation tolerance: This tolerance is used to compare the difference between computed and assumed water surface elevations at storage areas. If the difference is greater than the tolerance, the program continues to iterate for the current time step. When the difference is less than the tolerance, the program can go on to the next time step. The default tolerance for storage areas is set to 0.1 feet.

Maximum number of iterations: This variable defines the maximum number of iterations that the program will make when attempting to solve the unsteady flow equations using the specified tolerances. The default value is set to 20, and the allowable range is from 0 to 40.

Maximum number of warm-up time steps: Before the dynamic simulation, the program runs a series of time steps with constant inflows. This is called a warm-up period. This is done in order to smooth the profile before allowing the inflow hydrographs to progress. This helps to make a more stable solution at the beginning of the simulation. The default number of warm-up time steps is set to 20. This value ranges from 0 to 40.

Time step during warm-up period: During the warm-up period described in the previous paragraph, it is sometimes necessary to use a smaller time step than what will be used during the unsteady flow calculations. The initial conditions from the backwater analysis uses a flow distribution in the reaches which is often different than that computed by unsteady flow. This can cause some instabilities at the beginning of the simulation. The use of a smaller time step during the warm-up period helps to get through these instabilities. The default is to leave this field blank, which means to use a time step that is the same as for the unsteady flow simulation period.

Minimum time step for interpolation: The program has an option to interpolate between time steps when it finds a very steep rise in an inflow hydrograph, or a rapid change in stage at any cross section. This option allows the user to set a minimum time step to use during interpolation. This prevents the program from using to small of a time step during time slicing.

Maximum number of interpolated time steps: This option defines the

maximum number of interpolated time steps that the program can use during time slicing, as described in the previous paragraph.

Weir flow stability factor: This factor is used to increase the stability of the numerical solution in and around a weir. This factor varies from 1.0 to 3.0. As the value is increased, the solution is more stable but less accurate. A value of 1.0 is the most accurate, but is susceptible to oscillations in the computed weir flow. The default value is 1.0. If you observe oscillations in the computed flow over the weir, you should first check to see if you are using a small enough computation interval. If the computation interval is sufficiently small, you should then try increasing this coefficient to see if it solves the problem.

Spillway flow stability factor: This factor is used to increase the stability of the numerical solution in and around a gated spillway. This factor varies from 1.0 to 3.0.   As the value is increased, the solution is more stable but less accurate. A value of 1.0 is the most accurate, but is susceptible to oscillations in the computed spillway flow. The default value is 1.0. If you observe oscillations in the computed flow over the spillway, you should first check to see if you are using a small enough computation interval. If the computation interval is sufficiently small, you should then try increasing this coefficient to see if it solves the problem.

Weir flow submergence decay exponent: This coefficient is used to stabilize the solution of flow over a weir for highly submerged weirs. This factor varies from 1.0 to 3.0. As the headwater and tailwater stages become closer together, occasionally oscillations in the solution can occur. This exponent will prevent this from happening. The default value of one has no effect. As you increase the coefficient, dampening of the oscillations will occur. See the section called Model Accuracy, Stability, and Sensitivity later in this chapter for greater detail on this factor.

Spillway flow submergence decay exponent: This coefficient is used to stabilize the solution of flow over a gated spillway for highly submerged flows. This factor varies from 1.0 to 3.0. As the headwater and tailwater stages become closer together, occasionally oscillations in the solution can occur. This exponent will prevent this from happening. The default value of 1.0 has no effect. As you increase the coefficient, dampening of the oscillations will occur. See the section called Model Accuracy, Stability, and Sensitivity later in this chapter for greater detail on this factor.

Convert energy method bridges to cross-sections with lids: This option is used to convert bridges to normal cross sections, instead of being processed as a family of rating curves. If you have a bridge in which you are using the energy solution method for high and low flow solutions, there is no need to process this as a family of rating curves. Instead, you can have the program treat the two internal bridge cross sections as any other normal cross section. If you turn this option on, the program will create a separate table of elevation versus area and conveyance for each of the two bridge sections.

Output Options. This option allows the user to set some additional output flags. The following is a list of the available options:

Write Initial Conditions file: This option allows the user to write out a "Hot Start" file. A hot start file can be used to set the initial conditions of the system for a subsequent run. This is commonly done in real time forecasting, where you want to use the results at a specific time from a previous run to be the initial conditions of the next run. The user is require to enter a time in hours from the beginning of the current simulation, which represents the time at which the conditions of the system will be written to the "Hot Start" file.

Write Detailed Output for Debugging: This option allows the user to turn on detailed output that is written to a log file. The user has the option to set a specific time window in which the program will only output information within this time. This option is used when there is a problem with the unsteady flow solution, in that it may be oscillating or going completely unstable. When this occurs, the user should turn this option on and re-run the program. After the run has either finished or blown up, you can view the log file output by selecting **View Computation Log File** from the **Options** menu of the Unsteady Flow Simulation window. This log file will show what is happening on a time step by time step basis. It will also show which cross section locations the program is having trouble balancing the unsteady flow equations, as well as the magnitude of the errors.

Check Data Before Execution. This option provides for comprehensive input data checking. When this option is turned on, data checking will be performed when the user presses the compute button. If all of the data are complete, then the program allows the unsteady flow computations to proceed. If the data are not complete, or some other problem is detected, the program will not perform the unsteady flow analysis, and a list of all the problems in the data will be displayed on the screen. If this option is turned off, data checking is not performed before the unsteady flow execution. The default is that the data checking is turned on.

View Computation Log File. This option allows the user to view the contents of the unsteady flow computation log file. The interface uses the Windows Notepad program to accomplish this. The log file contains detailed information of what the unsteady flow computations are doing on a time step by time step basis. This file is very useful for debugging problems with your unsteady flow model.

Saving The Plan Information

To save the Plan information to the hard disk, select **Save Plan** from the **File** menu of the simulation window. Whenever any option is changed or modified on the Unsteady Flow Analysis window, the user should Save the Plan.

Starting the Computations

Once all of the data have been entered, and a Plan has been defined, the unsteady flow computations can be performed by pressing the **Compute** button at the bottom of the Unsteady Flow Simulation window. When the compute button is pressed, a separate window will appear showing you the progress of the computations (Figure 8.19). The information that appears in the window is there as an indicator of the programs progress during the computations, and to display any computational messages. When the computations have been completed, the user can close the computations window by clicking the upper right corner of the window, or the close button at the bottom. If the computations ended normally (i.e. all of the processes ran with no error messages), then the user can begin to look at the output. If the program does not finish normally, then the user should turn on the detailed log file output option and re-run the program. Then view the log file output to begin debugging the problem.
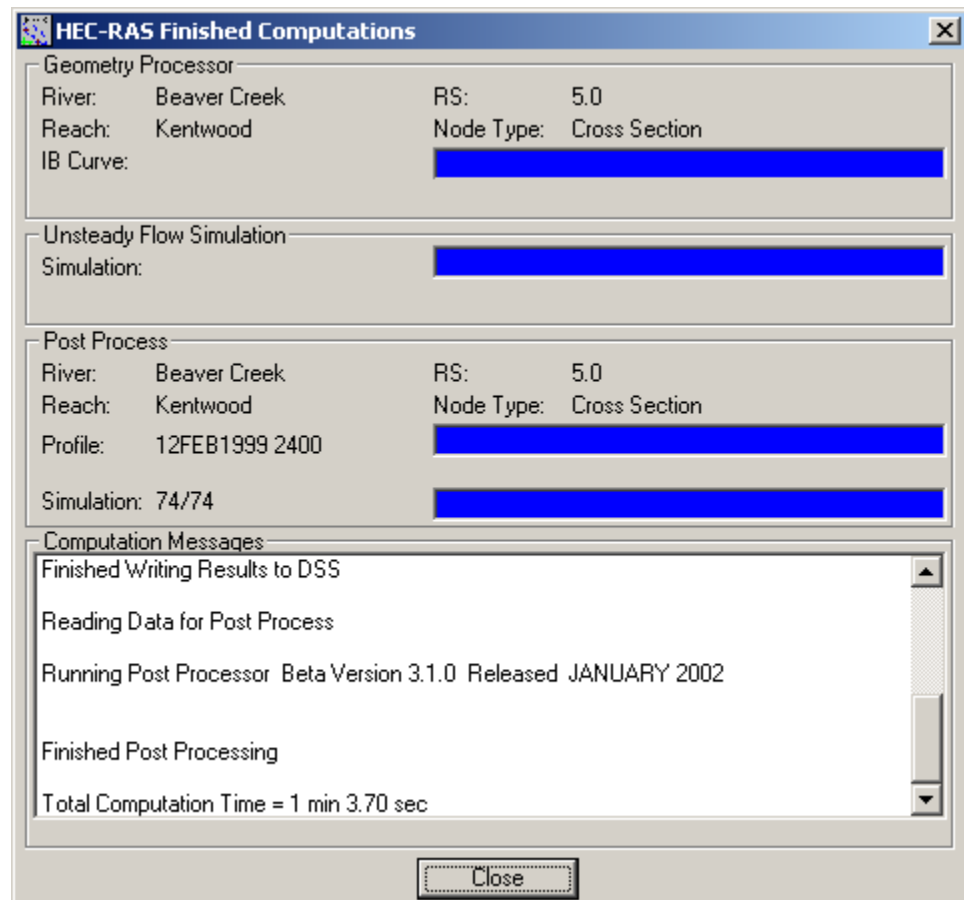
Figure 8.19 Unsteady Flow Computations Window

Calibration of Unsteady Flow Models

Calibration is the adjustment of a model's parameters so that it reproduces observed data to an acceptable accuracy. The following is a list of common problems and factors to consider when calibrating an unsteady flow model.

Observed Hydrologic Data

Stage Records. In general, measured stage data is our most accurate hydrologic data. Measured stage data is normally well within +/- 1.0 feet of accuracy. However, errors can be found in measured stage data. Some common problems are:

1.   The gages float gets stuck at a certain elevation during the rise or fall of the flood wave.

2.   The recorder may systematically accumulate error over time.

3.   The gage reader of a daily gage misses several days and guesses at the stage recordings.

4.   There is an error in the datum of the gage.

Flow Records. Flow records are generally computed from observed stages using single valued rating curves. These rating curves are a best fit of the measured data. The USGS classifies good flow measurements from Price current meters to be within ±5% of the true value. Some believe that this assumed error is optimistic. In any case, ±5%, on many river systems, translates into a stage error of ±1 foot. Acoustic velocity meters (AVM) provide a continuous record, but the current USGS technique calibrates these meters to reproduce measurements from Price current meters, so the AVM is as accurate as the current meter. Boat measurements are almost always suspect. In general it is very difficult to get accurate velocity measurements using a price current meter from a boat. Newer techniques using acoustic velocity meters with three beams mounted on boats are thought to be much better.

Published discharge records should also be scrutinized. Continuous discharge is computed from discharge measurements, usually taken at bi-weekly or monthly intervals and the continuous stage record. The measurements are compiled into a rating curve and the departures of subsequent measurements from the rating curve are used to define shifts. The shifts are temporary changes in the rating curve due to unsteady flow effects (looped rating curve) and short term geomorphic changes. The quality of the record depends on the frequency of discharge measurements and the skill of the hydrologist. The only way to depict the quality of the published flow data is to compare the measured flow values to the currently published rating curve. However, if the flow measurements are infrequent, one can only apply the flow record to the model and see how well the stage record is reproduced. Remember! Most

published flow records for large streams are in mean daily flow. The modeler must somehow assign time values to these records.

High Water Marks. High water marks are estimated from the upper limit of stains and debris deposits found on buildings, bridges, trees, and other structures. Wind and wave actions can cause the debris lines to be higher than the actual water surface. Capillary action can cause stains on buildings to migrate upward, depending on the material used for the building walls. High water marks in the overbank area are often higher than in the channel. The overbank water is moving slower and may be closer to the energy gradeline. High water marks on bridge piers are often equal to the energy gradeline, not the average water surface. This is due to the fact that the water will run up the front of the pier to an elevation close to the energy gradeline.

Shown in the Figure 8.20 below is a comparison between high water marks and the computed maximum water surface profile. Note the scatter in the high water marks, particularly around river station 230. Which mark is
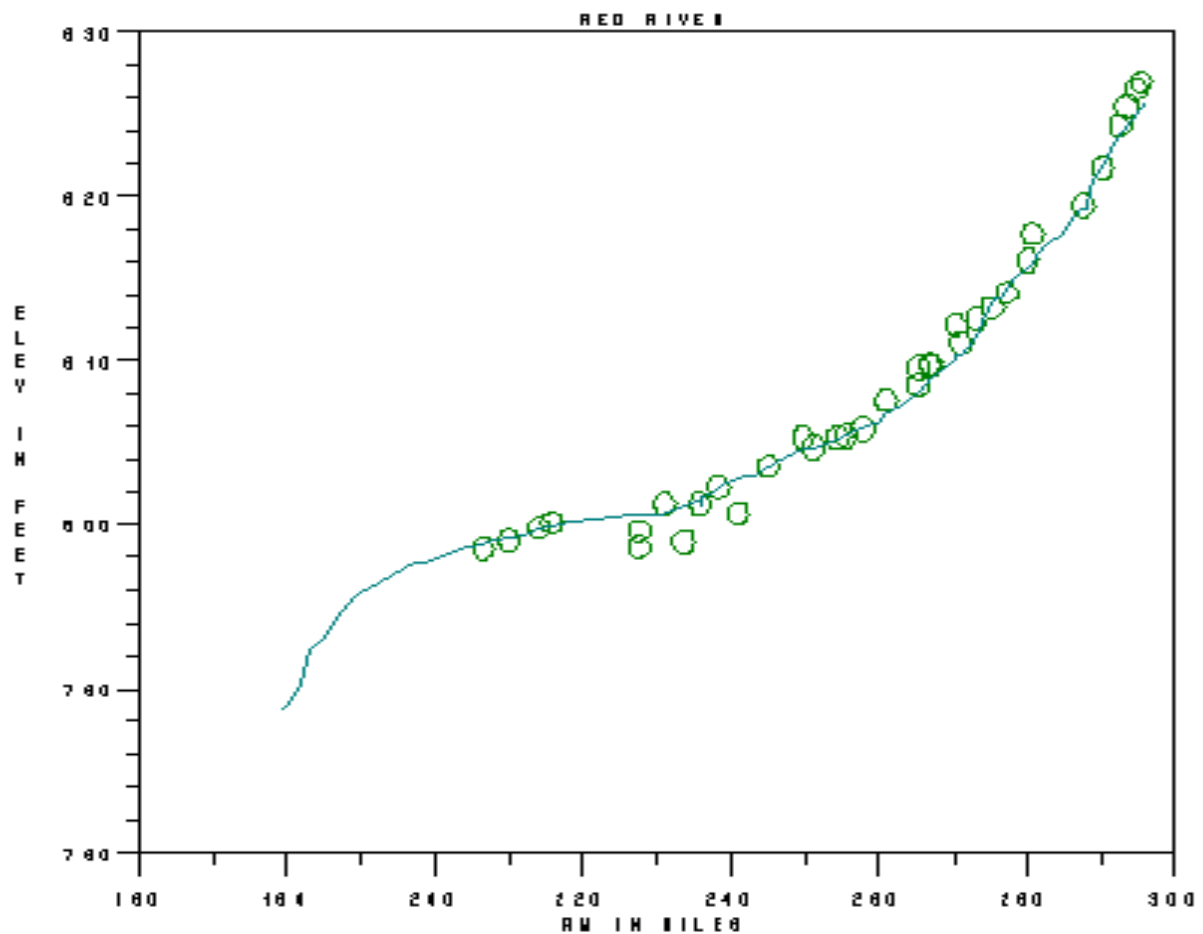
accurate?

Figure 8.20 Computed Water Surface Profile Versus Observed High Water Marks.

Ungaged Drainage Area. For an unsteady flow model to be accurate, it must have flow input from all of the contributing area. In many studies a significant portion of the area is ungaged. Discharge from ungaged areas can be estimated from either hydrologic models or by taking flow from a gaged watershed with similar hydrologic characteristics and multiplying it by a simple drainage area ratio.

An example of accounting for ungaged drainage area is shown below for the Red River of the North.

Stream	Station	River Mile	Gaged Drainage (Sq. Miles)
Red River	Grand Forks	296	30,100
Turtle River	Manvel	272.9	613
Forest River	Minto	242.5	740
Snake River	Alvarado	229.9	309
Middle River	Argyle	9.72	265
Park River	Grafton	221.9	695
	Total of Gaged Tributaries		2,622
Red River	Drayton	206.7	34,800
	Total Ungaged		2,078

Stream	River Mile	Ungaged Drainage (Sq. Miles)	Pattern Hydrograph	Drainage Area Ratio
Grand Marais Creek	288.6	298	Middle River	1.12
Tamarac River	218.5	320	Middle River	1.21
Remaining		1,460	Middle River	5.51

Figure 8.21 Example Drainage Area Accounting for Red River of the North.

As shown in Figure 8.21, ungaged areas can be accounted for by using a pattern hydrograph of a similar watershed (Middle River), then calculating a drainage area ratio of contributing areas (Ungaged area divided by pattern hydrograph area).

River and Floodplain Geometry

It is essential to have an adequate number of cross sections that accurately depict the channel and overbank geometry. This can be a great source of

error when trying to calibrate. Additionally, all hydraulic structures must be accurately depicted. Errors in bridge and culvert geometry can be significant sources of error in computed water surface profiles. Another important factor is correctly depicting the geometry at stream junctions (flow combining and splitting locations). This is especially important at flow splits, and areas in which flow reversals will occur (i.e. flow from a main stem backing up a tributary).

Also, a one-dimensional model assumes a constant water surface in each cross section. For some river systems, the water surface may vary substantially between the channel and the floodplain. If this is the case in your model, it may be necessary to separate the channel and the floodplain into there own reaches or model the overbank area as a series of storage areas.

Roughness Coefficients

Roughness coefficients are one of the main variables used in calibrating a hydraulic model. Generally, for a free flowing river, roughness decreases with increased stage and flow (Figure 8.22). However, if the banks of a river are rougher than the channel bottom (do to trees and brush), then the composite n value will increase with increased stage. Sediment and debris can also play an important role in changing the roughness. More sediment and debris in a river will require the modeler to use higher n values in order to match observed water surfaces.
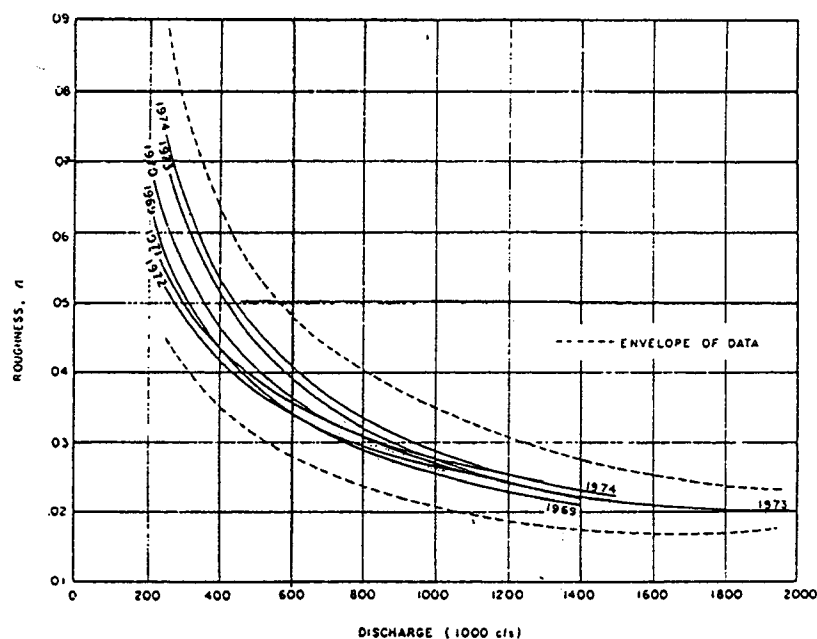
Figure 8.22 Roughness Versus Discharge for the Mississippi River at Arkansas City.

Looped Rating Curves. Excluding cataclysmic events such as meander cutoffs or a new channel, the river will pass any given flow within a range of stages. The shift in stage is a result of the following: shifts in channel bedforms; the dynamics of the hydrograph (how fast the flood wave rises and falls); backwater (backwater can significantly change the stage at a given cross section for a given flow); and finally, the slope of the river (flatter streams tend to have greater loops in the rating curve). Figure 8.23 below shows a looped rating for a single event. Generally, the lower stages are associated with the rising side of a flood wave, and the higher stages are associated with the falling side of the flood wave.
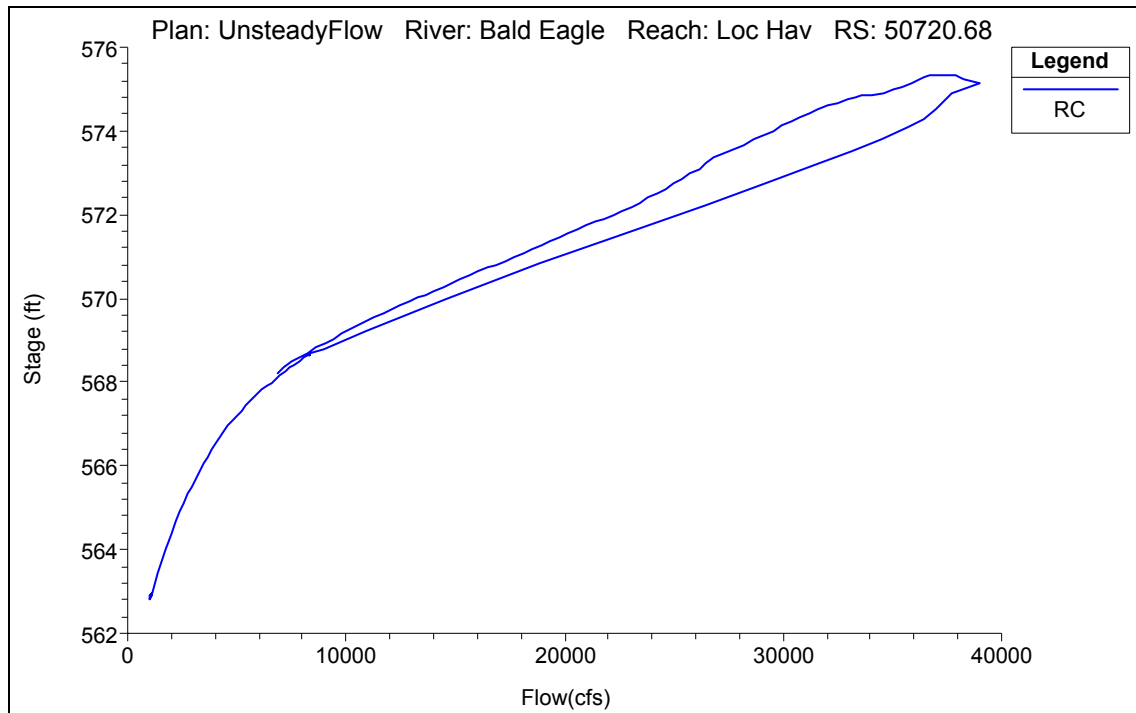
Figure 8.23 Looped Rating Curve Example

Alluvial Rivers. In an alluvial stream the channel boundary, as well as the meandering pattern of the stream, are continuously being re-worked by the flow of water. Alluvium is unconsolidated granular material, which is deposited by flowing water. An alluvial river is incised into these alluvial deposits. The flow characteristics of the stream are defined by the geometry and roughness of the cross-section below the water surface. The reworking of the cross section geometry and meander pattern is greatest during high flow, when the velocity, depth of water, and sediment transport capacity are the greatest. For some streams, which approach an equilibrium condition, the change in morphology (landforms) is small. For other streams, the change in morphology is much larger.   The change can be manifest as changes in roughness or a more dynamic change such as the cut-off of a meander loop, which shortens the stream and starts a process which completely redefines the bed.

A typical meandering river is shown in Figure 8.24 below. Pools are at the outside of bends, and a typical pool cross-section is very deep. On the inside of the bend is a point bar. Crossings are between the meander bends. A typical crossing cross-section is much shallower and more rectangular than a pool cross-section.
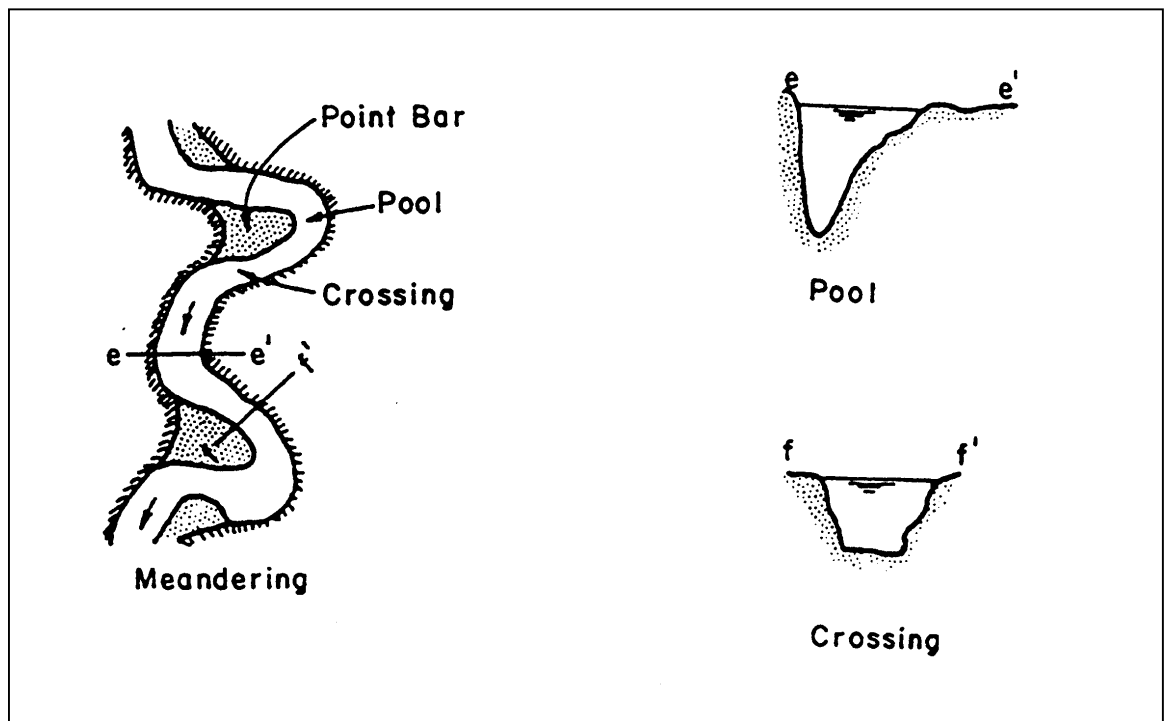
Figure 8.24 Morphology of a Meandering River

An invert profile for the Mississippi River is shown in the Figure 8.25. Note the pools and crossings. The water surface profile is controlled by the crossing cross-sections, particularly at low flow. The conveyance properties of pool cross-sections are only remotely related to the water surface. This poses a significant problem when calibrating a large river.
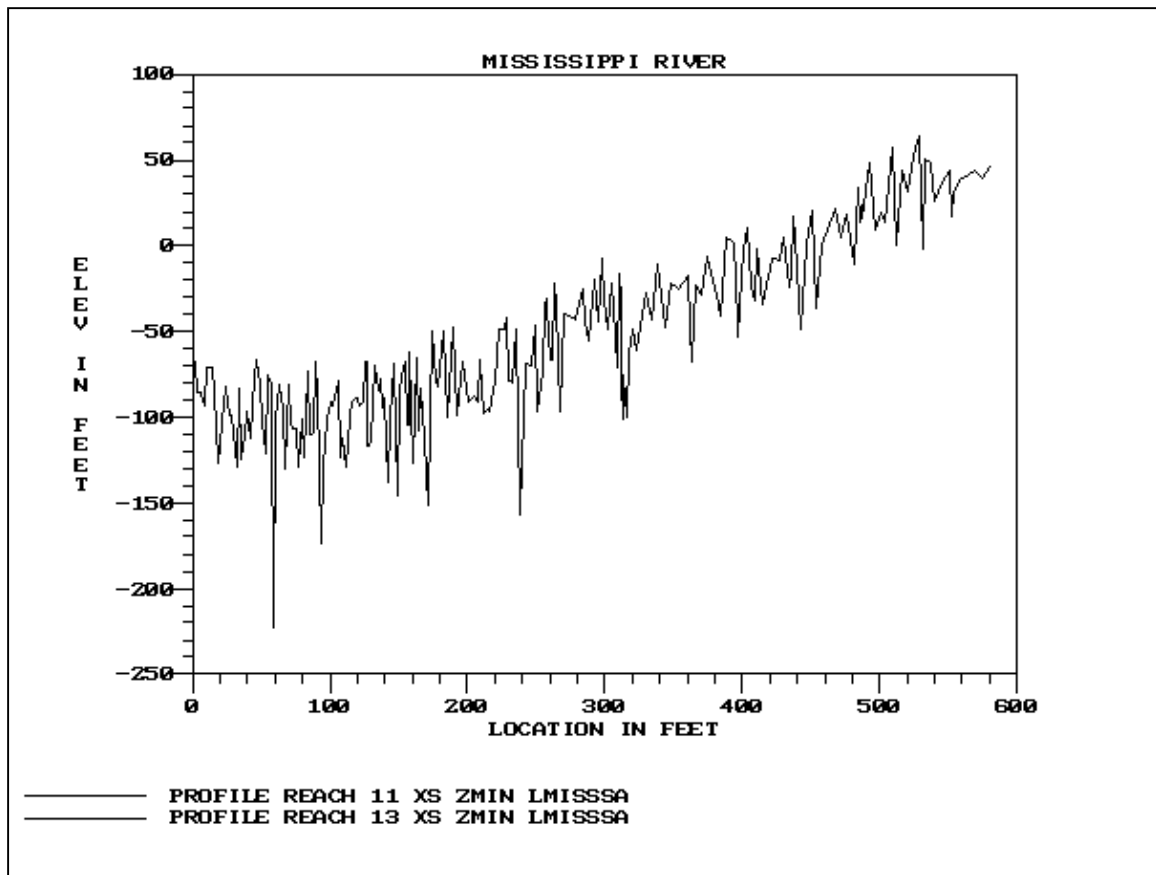
Figure 8.25 Invert Profile for Lower Mississippi River

As stage and flow increase you have an increase in stream power (stream power is a function of hydraulic radius, slope, and velocity). The bed forms in an alluvial stream tend to go through the following transitions:

1.   Plane bed without sediment movement.
2.   Ripples.
3.   Dunes.
4.   Plane bed with sediment movement.
5.   Anti-dunes.
6.   Chutes and pools.

Generally, anti-dunes and chutes and pools are associated with high velocity streams approaching supercritical flow. The bed form process is shown graphically in Figure 8.26.
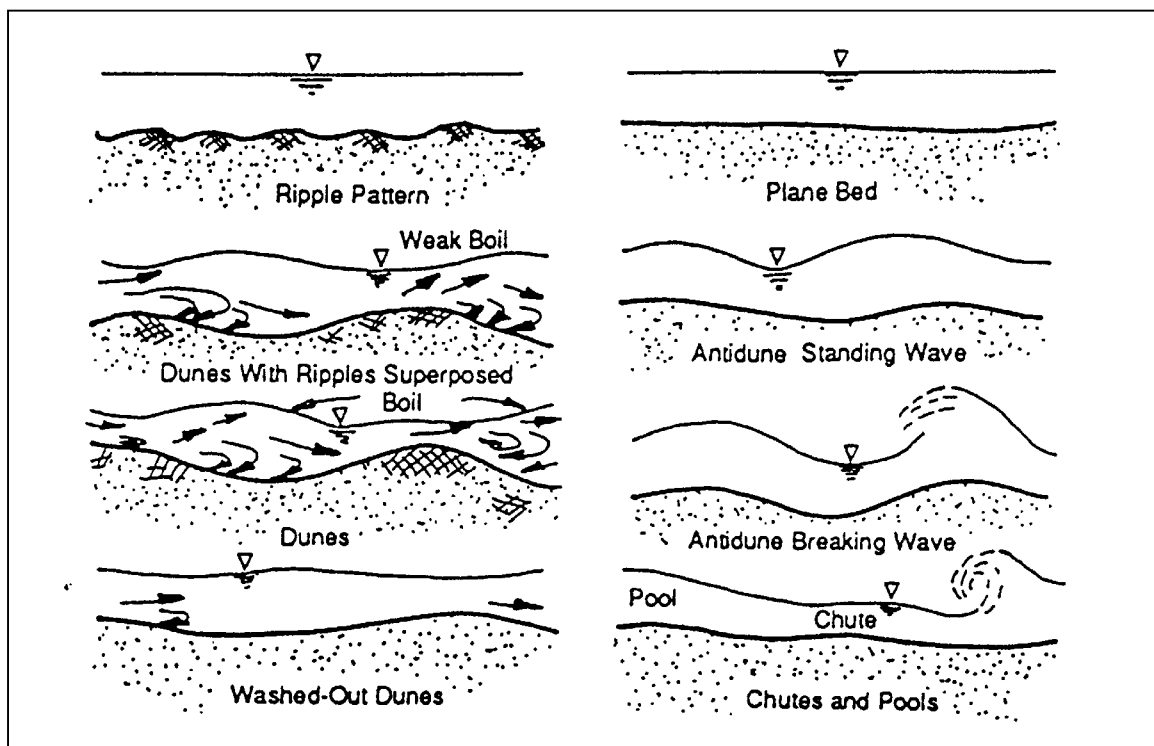
Figure 8.26 Transitions of Bed Forms in an Alluvial Stream

Typical Manning's roughness coefficients for the different bed forms presented above are shown in the following table:

Table 8-1 Roughness Variations for Alluvial Streams

Bed Forms	Range of Manning's n
Ripples	0.018 – 0.030
Dunes	0.020 – 0.035
Washed Out Dunes	0.014 – 0.025
Plane Bed	0.012 – 0.022
Standing Waves	0.014 – 0.025
Antidunes	0.015 – 0.031

Note: This table is from the book "Engineering Analysis of Fluvial Streams", by Simons, Li, and Associates.

Bed forms also change with water temperature. Because water is more viscous at lower temperatures, it becomes more erosive, reducing the height and the length of the dunes. At higher temperatures, when the water is less viscous, the dunes are higher and of greater length. Since the larger dunes are more resistant to flow, the same flow will pass at a higher stage in the summer than in the winter. Larger rivers such as the Mississippi River and the Missouri River show these trends. Figure 8.27 shows the seasonal shift for the Mississippi River at St. Louis.
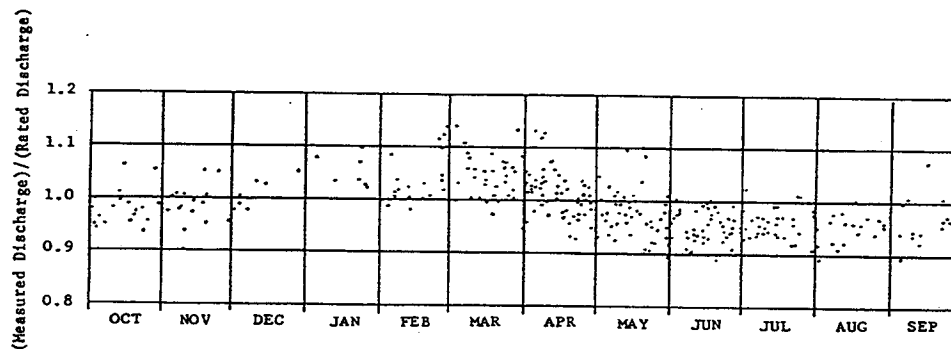
Figure 8.27 Changes in Roughness due to Temperature in The Mississippi River.

River and Floodplain Storage

Cross Sectional Storage. The active flow area of a cross section is the region in which there is appreciable velocity. This part of the cross section is conveying flow in the downstream direction. Storage is the portion of the cross section in which there is water, but it has little or no velocity. Storage can be modeled within a cross section by using the ineffective flow area option in HEC-RAS. The water surface elevation within the cross section storage is assumed to have the same elevation as the active flow portion of the cross section.

The storage within the floodplain is responsible for attenuating the flood hydrograph and, to some extent, delaying the flood wave.

Effects of Overbank Storage. Water is taken out of the rising side of the flood wave and returned on the falling side. An example of the effects of overbank storage is shown in Figure 8.28. In this example, the water goes out into storage during the rising side of the flood wave, as well as during the peak flow. After the peak flow passes, the water begins to come out of the storage in the overbank and increases the flow on the falling side of the floodwave.
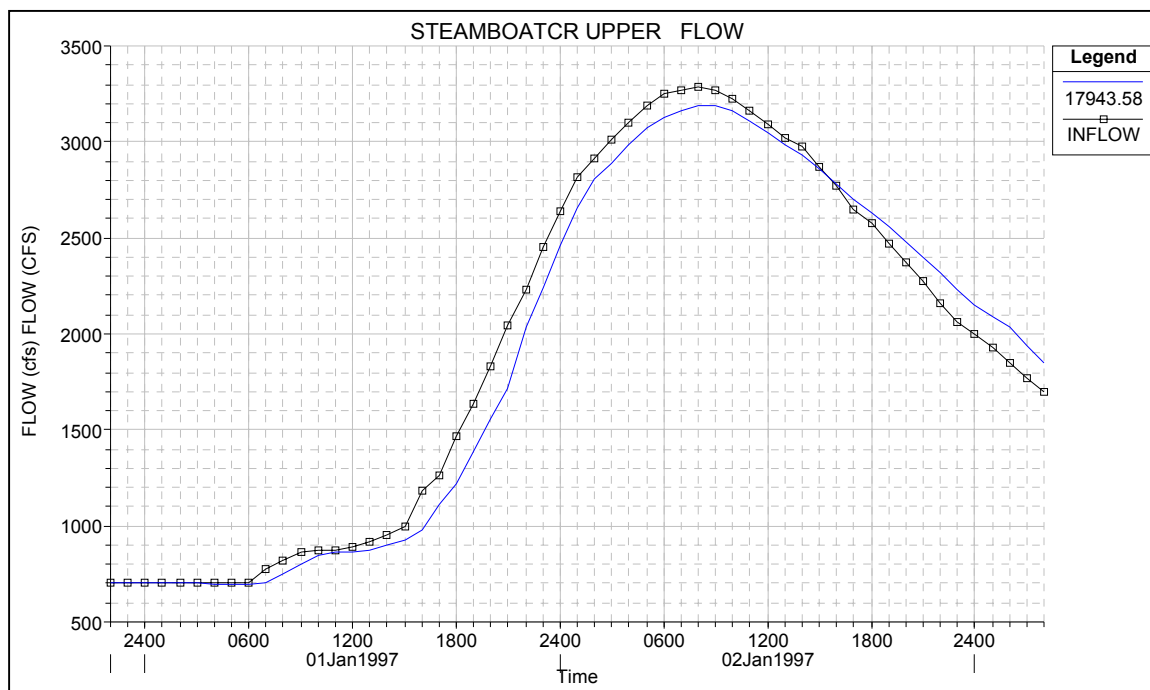
Figure 8.28 Example of The Effects of Overbank Storage

Off Line Storage. Off line storage is an area away from the main river in which water can go from the main river to the ponding area. The connection between the ponding area and the river may be a designed overflow, or it may just be a natural overflow area. The water in the ponding area is often at a different elevation that the main river, therefore, it must be modeled separately from the cross sections describing the main river and floodplain. Within HEC-RAS, ponding areas are modeled using what we call a storage area. Storage areas can be connected hydraulically to the river system by using a lateral weir/spillway option in HEC-RAS.

The effect that off line storage has on the hydrograph depends on the available volume and the elevation at which flow can get into and out of the storage area. Shown in Figure 8.29 is an example of an off-line storage area that is connected to the river through a lateral weir/spillway. The flow upstream and downstream of the offline storage area remains the same until the water surface elevation gets higher than the lateral weir. Water goes out into the lateral storage facility the whole time it is above the weir (i.e. the storage area elevation is always lower than the river elevation in this example). This continues until later in the event, when the river elevation is

below the lateral weir and flow can no longer leave the river. In this example, the flow in the storage area does not get back into the river system until much later in the event, and it is at a very slow rate (possible drained by culverts to a downstream location).
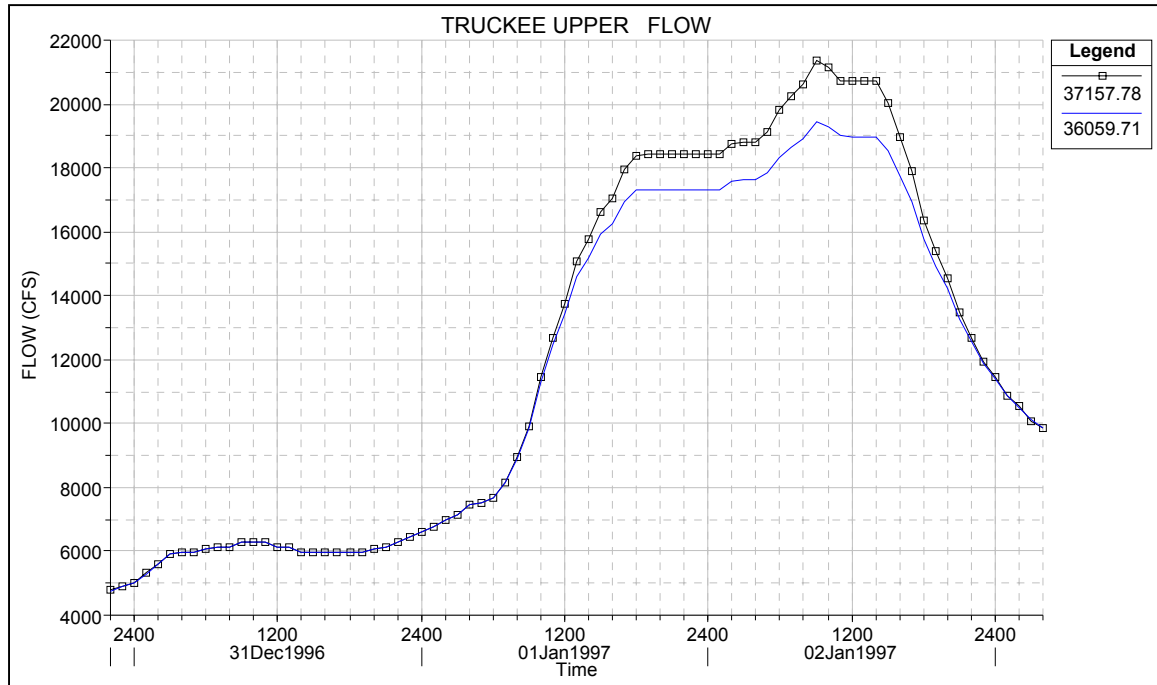
Figure 8.29 Example effects of off-line Storage

Hydraulic Structure Coefficients

Bridges and culverts tend to have a local effect on stage, and a minimum affect on the flow hydrograph (this depends on the amount of backwater they cause). The coefficients that are important in bridge modeling are: Manning's n values; contraction and expansion coefficients; pier loss coefficients, and pressure and weir flow coefficients for high flows. Culvert hydraulics are dependent upon the size of the culverts and shape of the entrance. Additional variables include Manning's n values and contraction and expansion coefficients.

The effects of Inline weirs/spillways can be substantial on both the stage and the flow attenuation of the hydrograph. The effects on the hydrograph will depend upon the available storage volume in the pool upstream of the structure, as well as how the structure is operated. Lateral weir/spillway structures can have a significant impact on the amount of water leaving the river system. Therefore gate and weir coefficients for these structures can be extremely critical to getting the right amount of flow leaving the system.

Steps To Follow in The Calibration Process

The following is a general list of steps to follow when calibrating an unsteady flow model:

1.   Run a range of discharges in the Steady-Flow mode, and calibrate n values to established rating curves at gages and known high water marks.

2.   Select specific events to run in unsteady flow mode. Ensure each event goes from low flow to high flow, and back to low flow.

3.   Adjust storage and lateral weirs to get good reproduction of flow hydrographs (Concentrate on timing, peak flow, volume, and shape).

4.   Adjust Manning's n values to reproduce stage hydrographs.

5.   Fine tune calibration for low to high stages by using "Discharge-Roughness Factors" where and when appropriate.

6.   Further refine calibration for long-term modeling (period of record analysis) with "Seasonal Roughness Factors" where and when appropriate.

7.   Verify the model calibration by running other flow events or long term periods that were not used in the calibration.

8.   If further adjustment is deemed necessary from verification runs, make adjustments and re-run all events (calibration and verification events).

General Trends When Adjusting Model Parameters

In order to understand which direction to adjust model parameters to get the desired results, the following is a discussion of general trends that occur when specific variables are adjusted. These trends assume that all other geometric data and variables will be held constant, except the specific variable being discussed.

Impacts of Increasing Manning's n. When Manning's n is increased the following impacts will occur:

1.   Stage will increase locally in the area where the Manning's n values were increased.

2.   Peak discharge will decrease (attenuate) as the flood wave moves downstream.

3.   The travel time will increase.

4.   The loop effect will be wider (i.e. the difference in stage for the same flow on the rising side of the flood wave as the falling side will be greater). An example of this is shown in Figure 8.30.
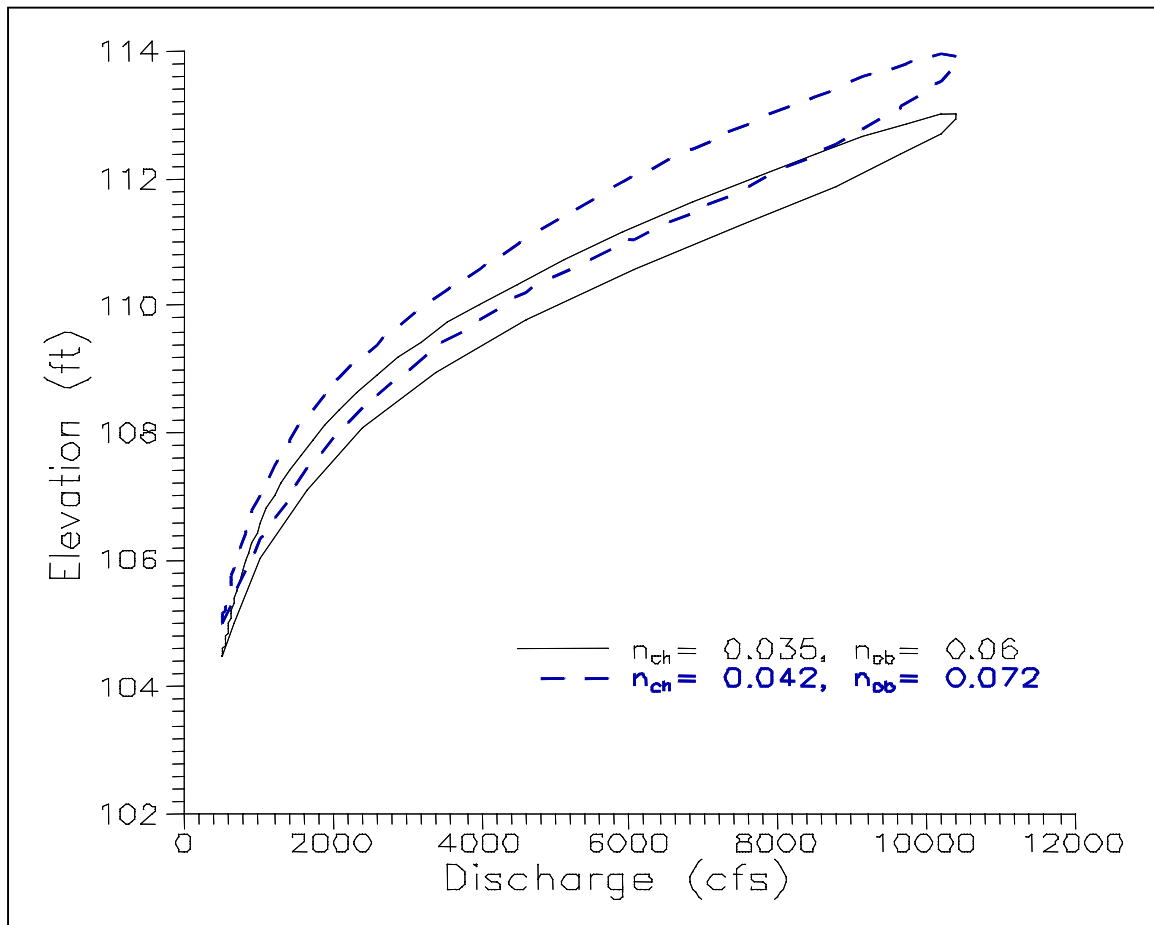
Figure 8.30 Example of Wider Loop for Higher Manning's n

Impacts of Increasing Storage. When storage within the floodplain is increased, the following impacts will occur:

1.   Peak discharge will decrease as the flood wave moves downstream.

2.   The travel time will increase.

3.   The tail of the hydrograph will be extended.

4.   The local stage (in the area of the increased storage) may increase or decrease. This depends upon if you are trading conveyance area for storage area, or just simply increasing the amount of storage area.

Calibration Suggestions and Warnings

The following is a list of suggestions and warnings to consider when calibrating an unsteady flow model:

1.   Calibrate mostly to stages. Flow data is derived from stage. Be wary of discharge derived from stage using single value rating curves.

2.   Do not force a calibration to fit with unrealistic Manning's n values or storage.

3.   If using a single-valued rating curve at the downstream boundary, move it far enough downstream so it doesn't effect the results in the study reach.

4.   Discrepancies may arise from a lack of quality cross-section data.

5.   The volume of off-channel storage areas is often underestimated, which results in a flood wave that travels to fast.

6.   Be careful with old HEC-2 and RAS studies done for steady flow only. The cross sections may not depict the storage areas. Defining storage is not a requirement for a steady flow model.

7.   Calibration should be based on floods that encompass a wide range of flows, low to high.

8.   For tidally influenced rivers and flows into reservoirs, the inertial terms in the momentum equation are very important. Adjusting Manning's n values may not help. Check cross sectional shape and storage.

9.   You must be aware of any unique events that occurred during the flood. Such as levee breaches and overtopping.

Model Accuracy, Stability, and Sensitivity

This section of the manual discusses model accuracy, stability, and sensitivity. In order to develop a good unsteady flow model of a river system, the user must understand how and why the solution of the unsteady flow equations becomes unstable. This knowledge will help you figure out why your particular model may be having stability problems. Additionally, it is important to understand the trade-offs between numerical accuracy (accurately solving the equations) and model stability. Finally, model sensitivity will be discussed in order to give you an understanding of what parameters affect the model and in what ways.

Model Accuracy

Model accuracy can be defined as the degree of closeness of the numerical solution to the true solution. Accuracy depends upon the following:

1.   Assumptions and limitations of the model (i.e. one dimensional model, subcritical flow only for unsteady flow).

2.   Accuracy of the geometric Data (cross sections, Manning's n values, bridges, culverts, etc…).

3.   Accuracy of the flow data and boundary conditions (inflow hydrographs, rating curves, etc…).

4.   Numerical Accuracy of the solution scheme (solution of the unsteady flow equations).

Numerical Accuracy. If we assume that the 1-dimensional unsteady flow equations are a true representation of flow moving through a river system, then only an analytical solution of these equations will yield an exact solution. Finite difference solutions are approximate. An exact solution of the equations is not feasible for complex river systems, so HEC-RAS uses an implicit finite difference scheme.

Model Stability

An unstable numerical model is one for which certain types of numerical errors grow to the extent at which the solution begins to oscillate, or the errors become so large that the computations can not continue. This is a common problem when working with an unsteady flow model of any size or complexity. The following factors will affect the stability and numerical accuracy of the model:

1.   Cross section spacing.

2.   Computation time step.

3.   Theta weighting factor for numerical solution.

4.   Solution iterations.

5.   Solution tolerances.

6.   Weir and spillway stability factors.

7.   Weir and spillway submergence factors.

Cross-Section Spacing. Cross sections should be placed at representative locations to describe the changes in geometry. Additional cross sections should be added at locations where changes occur in discharge, slope, velocity, and roughness. Cross sections must also be added at levees, bridges, culverts, and other structures.

Bed slope plays an important role in cross section spacing. Steeper slopes require more cross sections. Streams flowing at high velocities may require cross sections on the order of 100 feet or less. Larger uniform rivers with flat slopes may only require cross sections on the order of 1000 ft or more. However, most streams lie some where in between these two spacing distances.

The general question about cross section spacing is "How do you know if you have enough cross sections." The easiest way to tell is to add additional cross sections (this can be done through the HEC-RAS cross section interpolation option) and save the geometry as a new file. Then make a new plan and execute the model, compare the two plans (with and without interpolated cross sections). If there are no significant differences between the results (profiles and hydrographs), then the original model without the additional cross sections is ok. If there are some significant differences, then additional cross sections should be gathered in the area where the differences occur. If it is not possible to get surveyed cross sections, or even cross sections from a GIS, then use the HEC-RAS interpolated cross sections. However, at least check the reasonableness of the interpolated cross sections with a topographic map. Edit any cross sections that do not look reasonable.

Computational Time Step. Stability and accuracy can be achieved by selecting a time step that satisfies the Courant Condition:

$$C_r = V_w \frac{\Delta t}{\Delta x} \leq 1.0$$

Therefore:

$$\Delta t \leq \frac{\Delta x}{V_w}$$

Where: V_w = Flood wave speed, which is normally greater than the average velocity.

V = Average velocity of the flow.

Δx = Distance between cross sections.

Δt = Computational time step.

For most rivers the flood wave speed can be calculated as:

$$V_w = \frac{dQ}{dA}$$

However, an approximate way of calculating flood wave speed is to multiply the average velocity by a factor. Factors for various channel shapes are shown in the table below.

Table 8-2 Factors for Computing Wave Speed From Average Velocity

Channel Shape	Ratio Vw/V
Wide Rectangular	1.67
Wide Parabolic	1.44
Triangular	1.33
Natural Channel	1.5

Practical Time Step Selection. For medium to large rivers, the Courant condition may yield time steps that are too restrictive (i.e. a larger time step could be used and still maintain accuracy and stability). A practical time step is:

$$\Delta t \leq \frac{Tr}{20}$$

Where: T_r =   Time of rise of the hydrograph to be routed.

However, you may need to use a smaller time step when you have lateral weirs/spillways and hydraulic connections between storage areas and the river system. Also, if you are opening and closing gates quickly, or modeling a Dam or Levee breach, you will need to use very small time steps (less than a minute, maybe even as low as 5 seconds).

Theta Weighting Factor. Theta is a weighting applied to the finite difference approximations when solving the unsteady flow equations. Theoretically Theta can vary from 0.5 to 1.0. However, a practical limit is from 0.6 to 1.0
Theta of 1.0 provides the most stability. Theta of 0.6 provides the most accuracy. The default in HEC-RAS is 1.0. Once you have your model developed, reduce theta towards 0.6, as long as the model stays stable.

Larger values of theta increase numerical diffusion, but by how much? Experience has shown that for short period waves that rapidly rise, theta of 1.0 can produce significant errors. However, errors in the solution can be reduced by using smaller time steps.

When choosing theta, one must balance accuracy and computational robustness. Larger values of theta produce a solution that is more robust, less prone to blowing up. Smaller values of theta, while more accurate, tend to cause oscillations in the solution, which are amplified if there are large numbers of internal boundary conditions. Test the sensitivity of theta to your data set. If reducing theta does not change the solution, then the larger value should be used to insure greater stability.

Solution Iterations. At each time step derivatives are estimated and the equations are solved. All of the computation nodes are then checked for numerical error. If the error is greater than the allowable tolerances, the program will iterate. The default number of iterations in HEC-RAS is set to 20. Iteration will generally improve the solution. This is especially true when your model has lateral weirs and storage areas.

Solution Tolerances. Two solution tolerances can be set or changed by the user: water surface calculation (0.02 default) and Storage area elevation (0.10 default). The default values should be good for most river systems. Only change them if you are sure! Making the tolerances larger can reduce the stability of the solution. Making them smaller can cause the program to go to the maximum number of iterations every time.

Weir and Spillway Stability Factors. Weirs and Spillways can often be a source of instability in the solution. Especially lateral structures, which take flow away or bring it into the main river. During each time step, the flow over a weir/spillway is assumed to be constant. This can cause oscillations by sending too much flow during a time step. One solution is to reduce the time step. Another solution is to use Weir and spillway stability factors, which can smooth these oscillations by damping the computed flows. However, using these stability factors can reduce the accuracy of the computed values.

The weir and spillway stability factors can range from 1.0 to 3.0. The default value of 1.0 is essentially no damping of the computed flows. As you increase the factor you get greater dampening of the flows (which will provide for greater stability), but less accuracy.

Weir and Spillway Submergence Factors. When you have a weir or spillway connecting two storage areas, or a storage area and a reach, oscillations can occur when the weir or spillway becomes highly submerged. The program must always have flow going one way or the other when the water surface is above the weir/spillway. When a weir/spillway is highly submerged, the amount of flow can vary greatly with small changes in stage on one side or the other. This is due to the fact that the submergence curves, which are used to reduce the flow as it becomes more submerged, are very steep in the range of 95 to 100 percent submergence. The net effect of this is that you can get oscillations in the flow and stage hydrograph when you get to very high submergence levels. The program will calculate a flow in one direction at one time step. That flow may increase the stage on the receiving side of the weir, so the next time step it sends flow in the other direction. This type of oscillation is ok if it is small in magnitude. However, if the oscillations grow, they can cause the program to go unstable.

To reduce the oscillations, the user can increase the Weir/Spillway Submergence Factor. This factor can vary from 1.0 to 3.0. A factor of 1.0 leaves the submergence criteria in its original form. Using a factor greater than 1.0 causes the program to use larger submergence factors earlier, and makes the submergence curve less steep at high degrees of submergence. A plot of the submergence curves for various factors is shown in Figure 8.31.
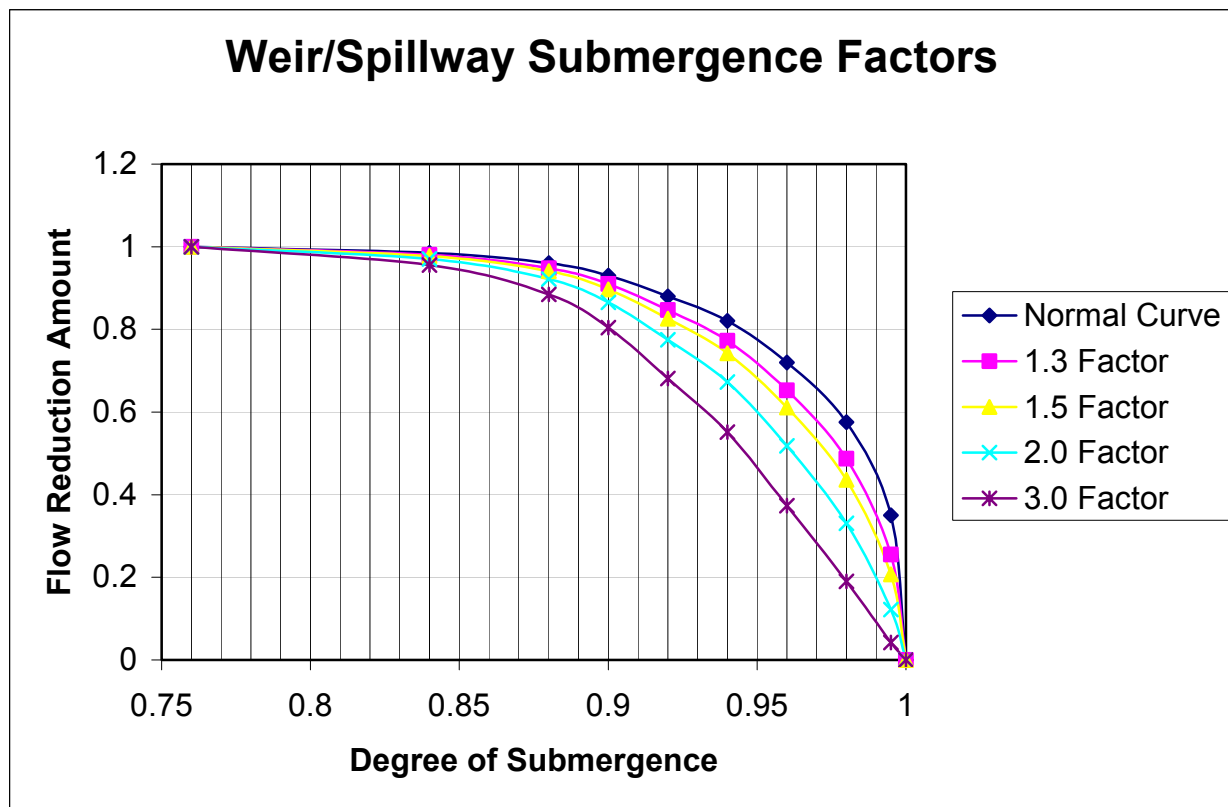
Figure 8.31 Weir/Spillway Submergence Factors

Common Stability Problems. The following is a list of some of the most common reasons for instabilities occurring in an unsteady flow model like HEC-RAS:

Too large of a time step: When the solution scheme solves the unsteady flow equations, derivatives are calculated with respect to distance and time. If the changes in hydraulic properties at a give cross section are changing rapidly with respect to time, the program will go unstable. The solution to this problem is to decrease the computational time step.

Not enough cross sections: When cross sections are spaced far apart, and the changes in hydraulic properties are great, the solution can become unstable. In situations where the flow area, depth, and/or velocity are changing rapidly, it is necessary to add additional cross sections to keep the solution stable.

Model goes to critical depth: The default solution methodology for unsteady flow routing within HEC-RAS is generally for subcritcal flow. However, the software does have an option to run in a mixed flow regime mode. This option should not be used unless you truly believe you have a mixed flow regime river system. If you are running the software in the default mode (subcritical only, no mixed flow), and if the program goes down to critical depth at a cross section, the changes in area, depth, and velocity are very high. This sharp increase in the water surface slope will often cause the

program to overestimate the depth at the next cross section upstream, and possible underestimate the depth at the next cross section downstream (or even the one that went to critical depth the previous time step). One solution to this problem is to increase the Manning's n value in the area where the program is first going to critical depth. This will force the solution to a subcritical answer and allow it to continue with the run. Obviously, this is creating a solution that may be higher than the true solution, but it may be necessary in order to allow the program to continue with the run. If you feel that the true water surface should go to critical depth, or even to a supercritical flow regime, then the mixed flow regime option should be turned on.

Bad downstream boundary condition: If the user entered downstream boundary condition causes abrupt jumps in the water surface, or water surface elevations that are too low (approaching or going below critical depth), this can cause oscillations in the solution that may lead to it going unstable and stopping. Examples of this are rating curves with not enough points or just simply to low of stages; and normal depth boundaries where the user has entered to steep of a slope.

Bad cross section properties: All of the cross sections get converted to tables of hydraulic properties (elevation versus area, conveyance, and storage). If the curves that represent these hydraulic properties have abrupt changes with small changes in elevation, this can also lead to instability problems. This situation is commonly caused by: levees being overtopped with large areas behind them (since the model is one dimensional, it assumes that the water surface is the same all the way across the entire cross section); and ineffective flow areas with large amounts of storage that are turned on at one elevation, and then turn off at a slightly higher elevation (this makes the entire area now used as active conveyance area). There are many possible solutions to these problems, but the basic solution is to not allow the hydraulic properties of a cross section to change so abruptly. If you have a levee with a large amount of area behind it, model the area behind the levee separately from the cross section. This can be done with either a storage area or another routing reach, whichever is most hydraulically correct for the flow going over the levee or if the levee breaches. With large ineffective flow areas, the possible solutions are to model them as being permanently on, or to put very high Manning's n values in the ineffective zones. Permanent ineffective flow areas allow water to convey over top of the ineffective area, so the change in conveyance and area is small. The use of high Manning's n values reduces the abruptness in the change in area and conveyance when the ineffective flow area gets turned off and starts conveying water.

Cross section property tables that do not go high enough: The program creates tables of elevation versus area, conveyance, and storage area for each of the cross sections. These tables are used during the unsteady flow solution to make the calculations much faster. By default, the program will create tables that extend up to the highest point in the cross section, however, the user can override this and specify their own table properties (increment and number of points). If during the solution the water surface goes above the

highest elevation in the table, the program simply extrapolates the hydraulic properties from the last two points in the table. This can lead to bad water surface elevations or even instabilities in the solution.

Not enough definition in cross section property tables: The counter problem to the previous paragraph is when the cross section properties in a given table are spread too far apart, and do not adequately define the changes in the hydraulic properties. Because the program uses straight-line interpolation between the points, this can lead to inaccurate solutions or even instabilities. To reduce this problem, we have increased the allowable number of points in the tables to 100. With this number of points, this problem should not happen.

Bad Bridge/Culvert Family of rating curves. The program creates a family of rating curves to define all the possible headwater, tailwater, and flow combinations that can occur at a particular structure. One free flow curve (headwater versus flow, with no influence from the tailwater) is calculated with fifty points to define it, then up to fifty submerged curves (headwater versus flow, staring at a particular tailwater) are calculated with up to 20 points to define each curve. The user can control how many submerged curves get calculated, how many points in each curve, and the properties used to define the limits of the curves (maximum headwater, maximum tailwater, maximum flow, and maximum head difference). By default, the software will take the curves up to an elevation equal to the highest point in the cross section just upstream of the structure. This may lead to curves that are too spread out and go up to a flow rate that is way beyond anything realistic for that structure. These type of problems can be reduced by entering specific table limits for maximum headwater, tailwater, flow, and head difference.

Wide and flat Lateral Weirs/Spillways: During the computations, there will be a point at which for one time step no flow is going over the lateral weir, and then the very next time step there is flow going over the weir. If the water surface is rising rapidly, and the weir is wide and flat, the first time the water surface goes above the weir could result in a very large flow being computed (i.e. it does not take a large depth above the weir to produce are large flow if it is very wide and flat). This can result in a great decrease in stage from the main river, which in turn causes the solution to oscillate and possibly go unstable. This is also a common problem when having large flat weirs between storage areas. The solution to this problem is to use smaller computational time steps. If this is not sufficient, there is also the option to use weir/spillway stability factors, as described previously in the chapter.

Opening gated spillways to quickly: When you have a gated structure in the system, and you open it quickly, if the flow coming out of that structure is a significant percentage of the flow in the receiving body of water, then the resulting stage, area and velocity will increase very quickly. This abrupt change in the hydraulic properties can lead to instabilities in the solution. To solve this problem you should use smaller computational time steps, or open the gate a littler slower, or both if necessary.

Very shallow depths of water: When starting a simulation it is very common to start the system at low flows. If you have some cross sections that are fairly wide, the depth will be very small. As flow begins to come into the river, the water surface will change quickly. The leading edge of the flood wave will have a very steep slope. Sometimes this steep slope will cause the solution to reduce the depth even further downstream of the rise in the water surface, possibly even producing a negative depth. This is due to the fact that the steep slope gets projected to the next cross section downstream when trying to solve for its water surface. The best solution to this problem is to use what is called a pilot channel. A pilot channel is a small slot at the bottom of the cross section, which gives the cross section a greater depth without adding much flow area. This allows the program to compute shallow depths on the leading edge of the flood wave without going unstable. Another solution to this problem is to use a larger base flow at the beginning of the simulation.

Detecting Stability Problems. One of the hardest things about using an unsteady flow model is to get the model to be stable, as well as accurate, for the range of events to be modeled. When you first start putting together an unsteady flow model, undoubtedly you will run in to some stability problems. The question is, how do you know you are having a stability problem? The following is a list of stability problem indicators:

1.   Program stops running during the simulation with a math error, or states that the matrix solution went unstable.

2.   Program goes to the maximum number of iterations for several time steps in a row (this is not always a stability problem).

3.   There are oscillations in the computed stage and flow hydrographs, or the water surface profiles.

If you detect a possible stability problem, in order to find the location of the problem you will need to turn on the detailed log output for debugging. Detailed log output is turned on by selecting **Options** and then **Output Options** from the Unsteady Flow Simulation manager. When this option is selected the following window will appear:
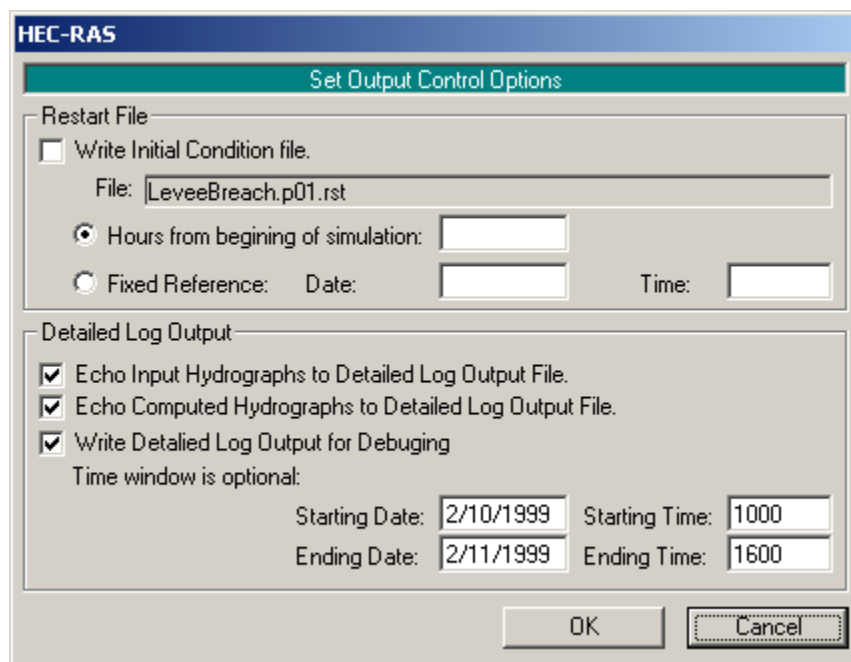
Figure 8.32 Detailed Log Output Control

As shown in Figure 8.32, 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:
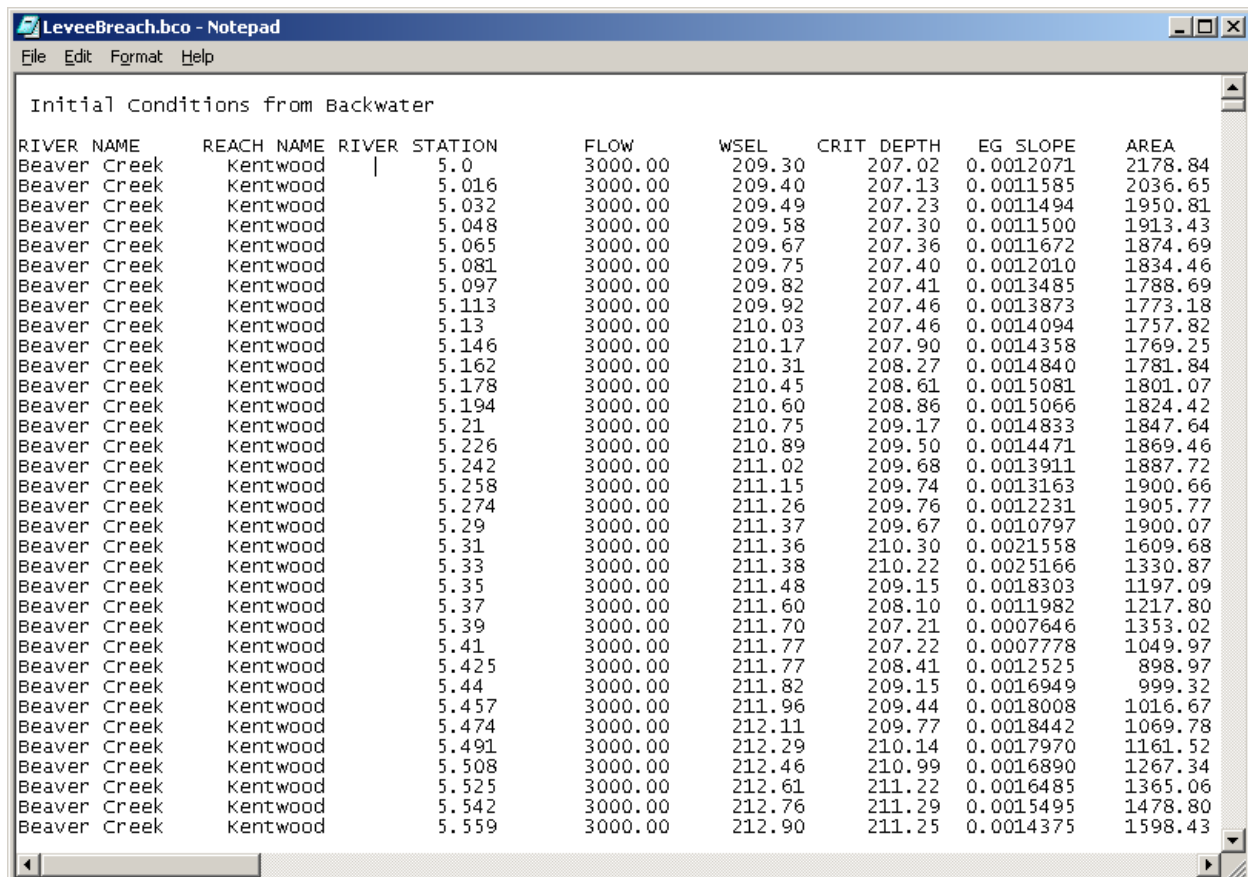
RDSS Output: Shows all of the hydrograph data that will be used as input to the model, including data read from HEC-DSS.

UNET 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.

The program lists the computed initial conditions from a backwater calculation for each of the river/reaches. They are listed in the order they were computed during the backwater analysis, which is downstream to upstream. An example of the initial conditions output is shown in Figure 8.33 below.

```
🗒 LeveeBreach.bco - Notepad                                                     _ □ ×
File  Edit  Format  Help

 Initial Conditions from Backwater

RIVER NAME      REACH NAME RIVER STATION      FLOW      WSEL    CRIT DEPTH   EG SLOPE      AREA
Beaver Creek      Kentwood    |    5.0      3000.00    209.30    207.02    0.0012071    2178.84
Beaver Creek      Kentwood         5.016    3000.00    209.40    207.13    0.0011585    2036.65
Beaver Creek      Kentwood         5.032    3000.00    209.49    207.23    0.0011494    1950.81
Beaver Creek      Kentwood         5.048    3000.00    209.58    207.30    0.0011500    1913.43
Beaver Creek      Kentwood         5.065    3000.00    209.67    207.36    0.0011672    1874.69
Beaver Creek      Kentwood         5.081    3000.00    209.75    207.40    0.0012010    1834.46
Beaver Creek      Kentwood         5.097    3000.00    209.82    207.41    0.0013485    1788.69
Beaver Creek      Kentwood         5.113    3000.00    209.92    207.46    0.0013873    1773.18
Beaver Creek      Kentwood         5.13     3000.00    210.03    207.46    0.0014094    1757.82
Beaver Creek      Kentwood         5.146    3000.00    210.17    207.90    0.0014358    1769.25
Beaver Creek      Kentwood         5.162    3000.00    210.31    208.27    0.0014840    1781.84
Beaver Creek      Kentwood         5.178    3000.00    210.45    208.61    0.0015081    1801.07
Beaver Creek      Kentwood         5.194    3000.00    210.60    208.86    0.0015066    1824.42
Beaver Creek      Kentwood         5.21     3000.00    210.75    209.17    0.0014833    1847.64
Beaver Creek      Kentwood         5.226    3000.00    210.89    209.50    0.0014471    1869.46
Beaver Creek      Kentwood         5.242    3000.00    211.02    209.68    0.0013911    1887.72
Beaver Creek      Kentwood         5.258    3000.00    211.15    209.74    0.0013163    1900.66
Beaver Creek      Kentwood         5.274    3000.00    211.26    209.76    0.0012231    1905.77
Beaver Creek      Kentwood         5.29     3000.00    211.37    209.67    0.0010797    1900.07
Beaver Creek      Kentwood         5.31     3000.00    211.36    210.30    0.0021558    1609.68
Beaver Creek      Kentwood         5.33     3000.00    211.38    210.22    0.0025166    1330.87
Beaver Creek      Kentwood         5.35     3000.00    211.48    209.15    0.0018303    1197.09
Beaver Creek      Kentwood         5.37     3000.00    211.60    208.10    0.0011982    1217.80
Beaver Creek      Kentwood         5.39     3000.00    211.70    207.21    0.0007646    1353.02
Beaver Creek      Kentwood         5.41     3000.00    211.77    207.22    0.0007778    1049.97
Beaver Creek      Kentwood         5.425    3000.00    211.77    208.41    0.0012525     898.97
Beaver Creek      Kentwood         5.44     3000.00    211.82    209.15    0.0016949     999.32
Beaver Creek      Kentwood         5.457    3000.00    211.96    209.44    0.0018008    1016.67
Beaver Creek      Kentwood         5.474    3000.00    212.11    209.77    0.0018442    1069.78
Beaver Creek      Kentwood         5.491    3000.00    212.29    210.14    0.0017970    1161.52
Beaver Creek      Kentwood         5.508    3000.00    212.46    210.99    0.0016890    1267.34
Beaver Creek      Kentwood         5.525    3000.00    212.61    211.22    0.0016485    1365.06
Beaver Creek      Kentwood         5.542    3000.00    212.76    211.29    0.0015495    1478.80
Beaver Creek      Kentwood         5.559    3000.00    212.90    211.25    0.0014375    1598.43
```

Figure 8.33 Example of Initial Conditions Output

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 Figure 8.34 below.

```
beaver.bco - Notepad                                                           _ □ ×
File  Edit  Format  Help

      COMPUTED STAGES AND DISCHARGES AT T =      0.1167 HOURS -  2/10/1999 AT 0007 HOURS

          Beaver Creek     Kentwood

Riv. Station     Z      Q      V      Riv. Station    Z      Q      V      Riv. Station     Z      Q      V
    5.99     213.03   599.    1.09        5.97    212.94   588.    1.22        5.951    212.83   579.    1.37
    5.93     212.71   571.    1.56        5.913   212.56   564.    1.79        5.894    212.38   558.    2.04
    5.875    212.19   552.    2.34        5.855   211.98   547.    2.65        5.836    211.74   543.    2.91
    5.81     211.52   540.    2.96        5.798   211.36   536.    2.45        5.779    211.24   532.    1.76
    5.76     211.17   528.    1.18        5.741   211.07   523.    1.64        5.72     210.91   521.    2.15
    5.703    210.75   519.    2.31        5.685   210.61   517.    2.30        5.666    210.48   516.    2.19
    5.647    210.37   515.    1.92        5.628   210.27   514.    1.62        5.61     210.21   513.    1.31
    5.593    210.13   512.    1.52        5.576   210.03   511.    1.80        5.559    209.93   511.    2.06
    5.542    209.85   510.    2.23        5.525   209.78   510.    2.24        5.508    209.72   510.    2.14
    5.491    209.67   510.    1.94        5.474   209.64   510.    1.76        5.457    209.61   510.    1.60
    5.44     209.58   510.    1.47        5.425   209.58   509.    1.11        5.41     209.57   509.    0.88
    5.39     209.54   509.    0.88        5.37    209.52   509.    1.11        5.35     209.48   509.    1.46
    5.33     209.40   510.    1.67        5.31    209.28   510.    1.67        5.29     209.15   510.    1.47
    5.274    208.95   510.    1.66        5.258   208.68   511.    1.95        5.242    208.29   511.    2.55
    5.226    207.85   511.    3.05        5.21    207.46   512.    3.29        5.194    207.15   512.    3.13
    5.178    206.95   513.    2.68        5.162   206.83   513.    2.16        5.146    206.75   514.    1.75
    5.13     206.71   514.    1.45        5.113   206.65   515.    1.50        5.097    206.59   515.    1.56
    5.081    206.53   516.    1.60        5.065   206.46   517.    1.63        5.048    206.39   517.    1.65
    5.032    206.31   518.    1.66        5.016   206.23   519.    1.66        5.0      206.13   519.    1.64



                  Solving for T =      0.133

Iter  River          Station    Elev     DZ      River         Station        Q      DQ
   0 Beaver Creek    5.99     213.07   0.03530  Beaver Creek    5.99        613      14
   1 Beaver Creek    5.0      206.13  -0.00050  Beaver Creek    5.93        584       1
```

Figure 8.34 Detailed Output at a Cross Section.

When the program has stability problems, it will generally try to solve them by iterating. An example of a stability problem is shown in Figure 8.35. In this example the program did not solve the equations to the specified tolerances, and therefore it was iterating to improve the solution. As shown in Figure 8.35, the program iterated to the maximum number of iterations. At the end of the iterations a warning message states **"WARNING USED COMPUTED CHANGES IN FLOW AND STAGE AT MINIMUM ERROR. MINIMUM ERROR OCCURRED AT ITERATION XX."** This message means that the program could not solve the unsteady flow equations to the required tolerance within the specified number of iterations (default number of iterations is 20). Therefore it used the iteration that had the least amount of error in the numerical solution.
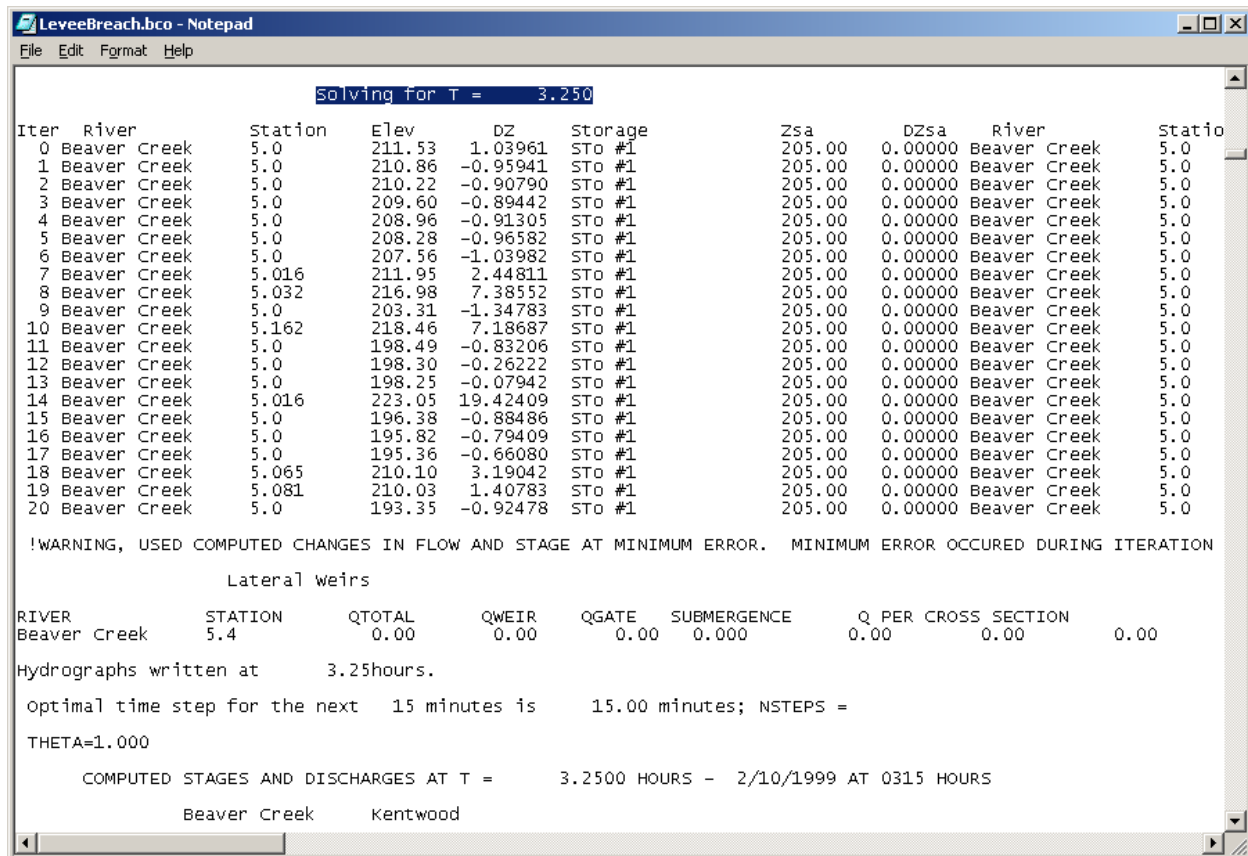
```
LeveeBreach.bco - Notepad

File  Edit  Format  Help

                  Solving for T =      3.250

Iter  River          Station    Elev      DZ      Storage           Zsa        DZsa     River         Statio
   0 Beaver Creek    5.0       211.53   1.03961  STo #1           205.00    0.00000 Beaver Creek     5.0
   1 Beaver Creek    5.0       210.86  -0.95941  STo #1           205.00    0.00000 Beaver Creek     5.0
   2 Beaver Creek    5.0       210.22  -0.90790  STo #1           205.00    0.00000 Beaver Creek     5.0
   3 Beaver Creek    5.0       209.60  -0.89442  STo #1           205.00    0.00000 Beaver Creek     5.0
   4 Beaver Creek    5.0       208.96  -0.91305  STo #1           205.00    0.00000 Beaver Creek     5.0
   5 Beaver Creek    5.0       208.28  -0.96582  STo #1           205.00    0.00000 Beaver Creek     5.0
   6 Beaver Creek    5.0       207.56  -1.03982  STo #1           205.00    0.00000 Beaver Creek     5.0
   7 Beaver Creek    5.016     211.95   2.44811  STo #1           205.00    0.00000 Beaver Creek     5.0
   8 Beaver Creek    5.032     216.98   7.38552  STo #1           205.00    0.00000 Beaver Creek     5.0
   9 Beaver Creek    5.0       203.31  -1.34783  STo #1           205.00    0.00000 Beaver Creek     5.0
  10 Beaver Creek    5.162     218.46   7.18687  STo #1           205.00    0.00000 Beaver Creek     5.0
  11 Beaver Creek    5.0       198.49  -0.83206  STo #1           205.00    0.00000 Beaver Creek     5.0
  12 Beaver Creek    5.0       198.30  -0.26222  STo #1           205.00    0.00000 Beaver Creek     5.0
  13 Beaver Creek    5.0       198.25  -0.07942  STo #1           205.00    0.00000 Beaver Creek     5.0
  14 Beaver Creek    5.016     223.05  19.42409  STo #1           205.00    0.00000 Beaver Creek     5.0
  15 Beaver Creek    5.0       196.38  -0.88486  STo #1           205.00    0.00000 Beaver Creek     5.0
  16 Beaver Creek    5.0       195.82  -0.79409  STo #1           205.00    0.00000 Beaver Creek     5.0
  17 Beaver Creek    5.0       195.36  -0.66080  STo #1           205.00    0.00000 Beaver Creek     5.0
  18 Beaver Creek    5.065     210.10   3.19042  STo #1           205.00    0.00000 Beaver Creek     5.0
  19 Beaver Creek    5.081     210.03   1.40783  STo #1           205.00    0.00000 Beaver Creek     5.0
  20 Beaver Creek    5.0       193.35  -0.92478  STo #1           205.00    0.00000 Beaver Creek     5.0

 !WARNING, USED COMPUTED CHANGES IN FLOW AND STAGE AT MINIMUM ERROR.  MINIMUM ERROR OCCURED DURING ITERATION

              Lateral Weirs

RIVER           STATION     QTOTAL      QWEIR    QGATE    SUBMERGENCE      Q PER CROSS SECTION
Beaver Creek    5.4          0.00        0.00     0.00    0.000           0.00        0.00        0.00

Hydrographs written at     3.25hours.

 Optimal time step for the next   15 minutes is     15.00 minutes; NSTEPS =

 THETA=1.000

      COMPUTED STAGES AND DISCHARGES AT T =      3.2500 HOURS  -  2/10/1999 AT 0315 HOURS

              Beaver Creek      Kentwood
```

Figure 8.35 Example Detailed Time step Output for Cross Sections.

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   = The name of the river in which the largest stage error is occurring.
Station = River station with the largest 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.
Zsa   = Computed elevation of the storage area
Dzsa   = The "Numerical Error" in the computed storage area elevation.
River   = The name of the river in which the largest flow error is occurring.
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

After the iterations output, the program will show the computed stages and flows for all of the cross sections in which the user has selected to have

hydrographs computed. This is also useful information for detecting model stability problems. It is not always obvious as to which cross section or modeling component is causing the problem. Sometimes the program may blow up at one cross section, but the real problem is caused by a cross section upstream or downstream from this location. Detecting, finding, and fixing stability problems will require lots of experience to become proficient at it. Good luck, and don't get to discouraged!!!

Model Sensitivity

Model sensitivity is an important part of understanding the accuracy and uncertainty of the model. There are two types of sensitivity analysis that should be performed, Numerical Sensitivity and Physical Parameter Sensitivity.

Numerical Sensitivity. Numerical Sensitivity is the process of adjusting parameters that affect the numerical solution in order to obtain the best solution to the equations, while still maintaining model stability. The following parameters are typically adjusted for this type of sensitivity analysis:

Computational Time Step - The user should try a smaller time step to see if the results change significantly. If the results do change significantly, then the original time step is probably too large to solve the problem accurately.

Theta Weighting Factor - The default value for this factor is 1.0, which provides the greatest amount of stability for the solution, but may reduce the accuracy. After the user has a working model, this factor should be reduced towards 0.6 to see if the results change.   If the results do change, then the new value should be used, as long as the model stays stable. Be aware that using a value of 0.6 gives the greatest accuracy in the solution of the equations, but it may open the solution up to stability problems.

 Weir/Spillway Stability Factors – If you are using these factors to maintain stability, try to reduce them to the lowest value you can and still maintain stability. The default value is 1.0, which is no stability damping.

Weir/Spillway Submergence Exponents – In general these parameters will not affect the answers significantly, they only provide greater stability when a spillway/weir is at a very high submergence. Try reducing them towards 1.0 (which is no factor) to see if the model will remain stable.

Physical Parameter Sensitivity. Physical Parameter Sensitivity is the process of adjusting hydraulic parameters and geometric properties in order to test the uncertainty of the models solutions. This type of sensitivity analysis is often done to gain an understanding of the possible range of solutions, given realistic changes in the model parameters. Another application of this type of sensitivity analysis is to quantify the uncertainty in the model results

for a range of statistical events (2, 5,10, 25, 50, 100 yr, etc…). The following data are often adjusted during this type of sensitivity analysis:

Manning's n Values – Manning's n values are estimated from physical data about the stream and floodplain. Sometimes Manning's n values are calibrated for a limited number of events. Either way, the values are not exact! The modeler should estimate a realistic range that the n values could be for their stream. For example, if you estimated an n value for a stream as 0.035, a realistic range for this might be 0.03 to 0.045. The modeler should run the lower Manning's n values and the higher Manning's n values to evaluate their sensitivity to the final model results.

Cross Section Spacing – Cross section spacing should always be tested to ensure that you have enough cross sections to accurately describe the water surface profiles. One way to test if you have enough cross sections is to use the HEC-RAS cross section interpolation routine, and interpolate enough cross sections to cut the average distance between cross sections in half. Re-run the model, if the results have not changed significantly, then your original model was probably fine. If the results do change significantly, then you should either get more surveyed cross sections or use the interpolated cross sections.   If you use the interpolated cross sections, then you should at least look at a topographic map to ensure that the interpolated cross sections are reasonable. If the interpolated cross sections are not reasonable in a specific area, then simply edit them directly to reflect what is reasonable based on the topographic map.

Cross Section Storage – Portions of cross sections are often defined with ineffective flow areas, which represents water that has no conveyance. The extent of the storage within a cross section is an estimate. What if the ineffective flow areas were larger or smaller? How would this effect the results? This is another area that should be tested to see the sensitivity to the final solution.

Lateral Weir/Spillway Coefficients – Lateral weir/spillway coefficents can have a great impact on the results of a simulation, because they take water away or bring water into the main stream system. These coefficients can vary greatly for a lateral structure, depending upon their angle to the main flow, the velocity of the main flow, and other factors. The sensitivity of these coefficients should also be evaluated.

Bridge/Culvert Parameters – In general, bridge and culvert parameters normally only effect the locally computed water surface elevations just upstream and downstream of the structure. The effect that a bridge or culvert structure will have on the water surface is much greater in flat streams (a small increase in water surface can back upstream for a long distance if the river is flat). However, the sensitivity of the water surface elevations around a bridge or culvert may be very important to localized flooding. The bridge and culvert hydraulic parameters should also be evaluated to test their sensitivity.