

8 PERFORMING A 1D UNSTEADY FLOW ANALYSIS

This chapter shows how to calculate unsteady flow water surface profiles using the 1D Unsteady Flow solver in HEC-RAS. If the user is interested in 2D modeling, or combined 1D/2D, please review the **2D Modeling User's Manual**. The chapter is divided into four 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. The third part focuses on calibrating an unsteady flow model, and the last part talks about 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 7-1.

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 either the **Add RS** button or the **Add Storage Area** button. The **Add RS** button allows users to enter additional river station locations for boundary conditions. The **Add Storage Area** button allows user to add storage area locations for insertion of a boundary condition.

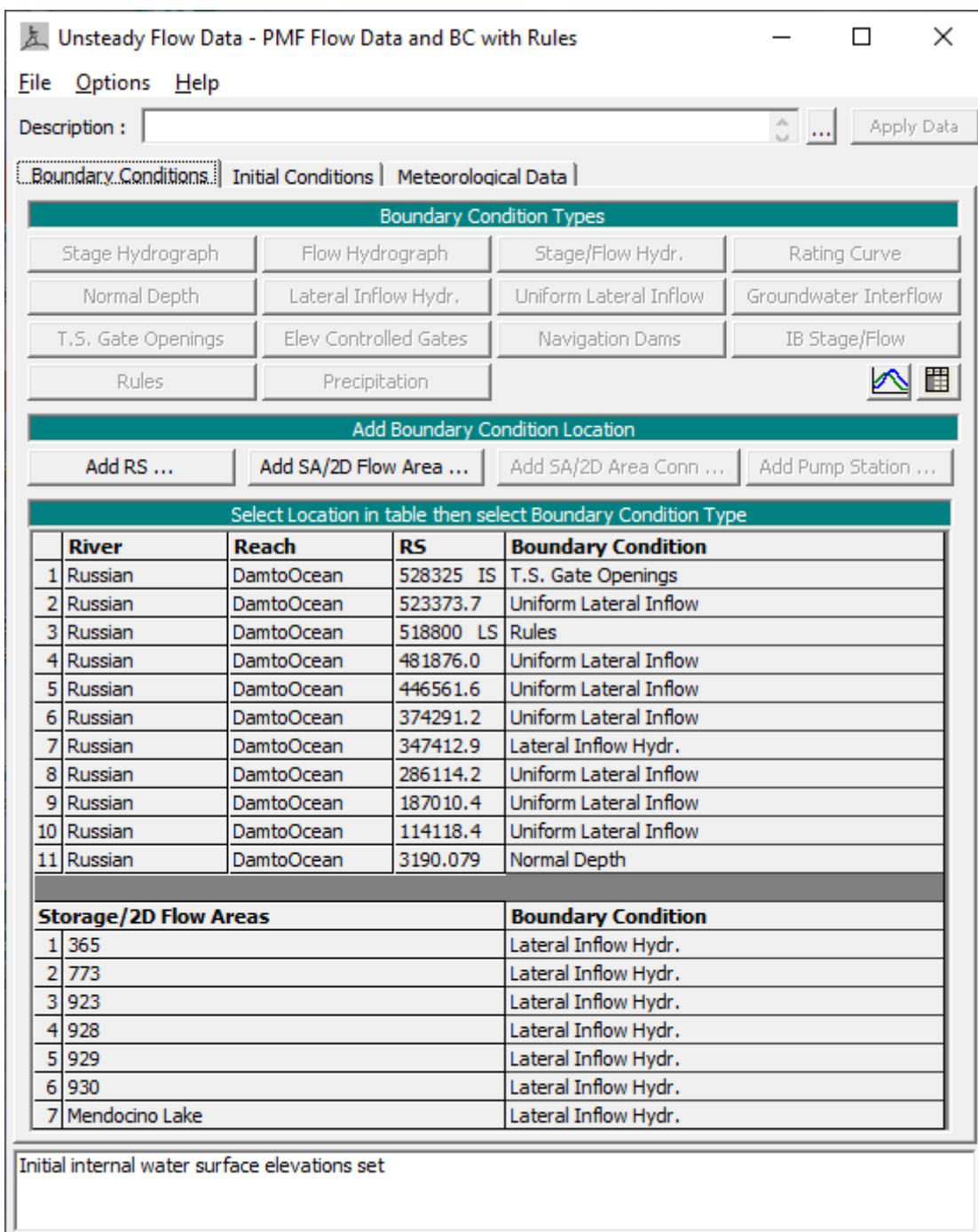


Figure 7.1. Unsteady Flow Data Editor

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 it is most commonly used as an upstream boundary condition. When the flow hydrograph button is pressed, the window shown in Figure 7-2 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 7-3. 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.

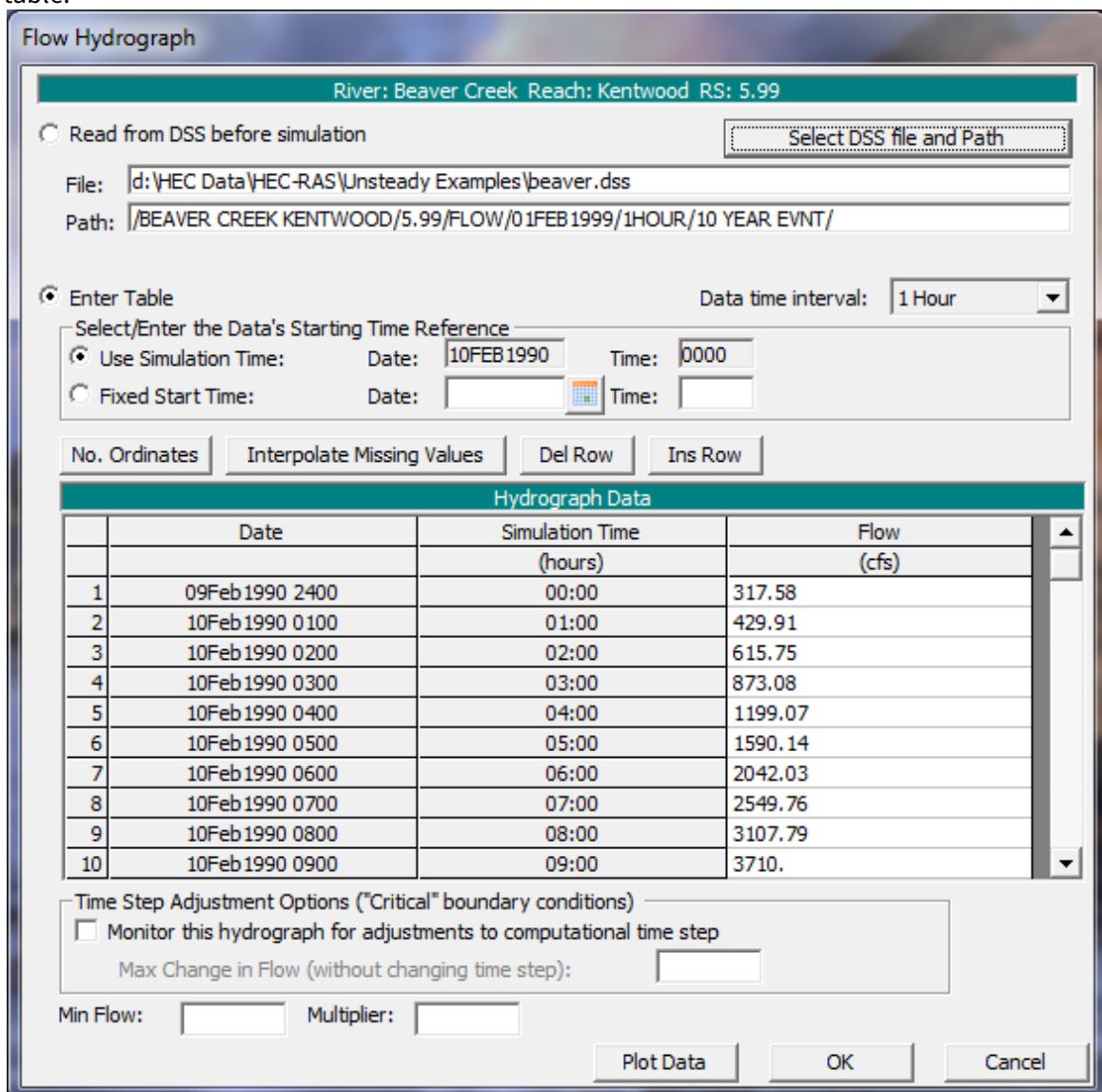


Figure 7.2. Example Flow Hydrograph Boundary Condition

The user also has the option of entering a flow hydrograph directly into a table, as shown in Figure 7-2. 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.

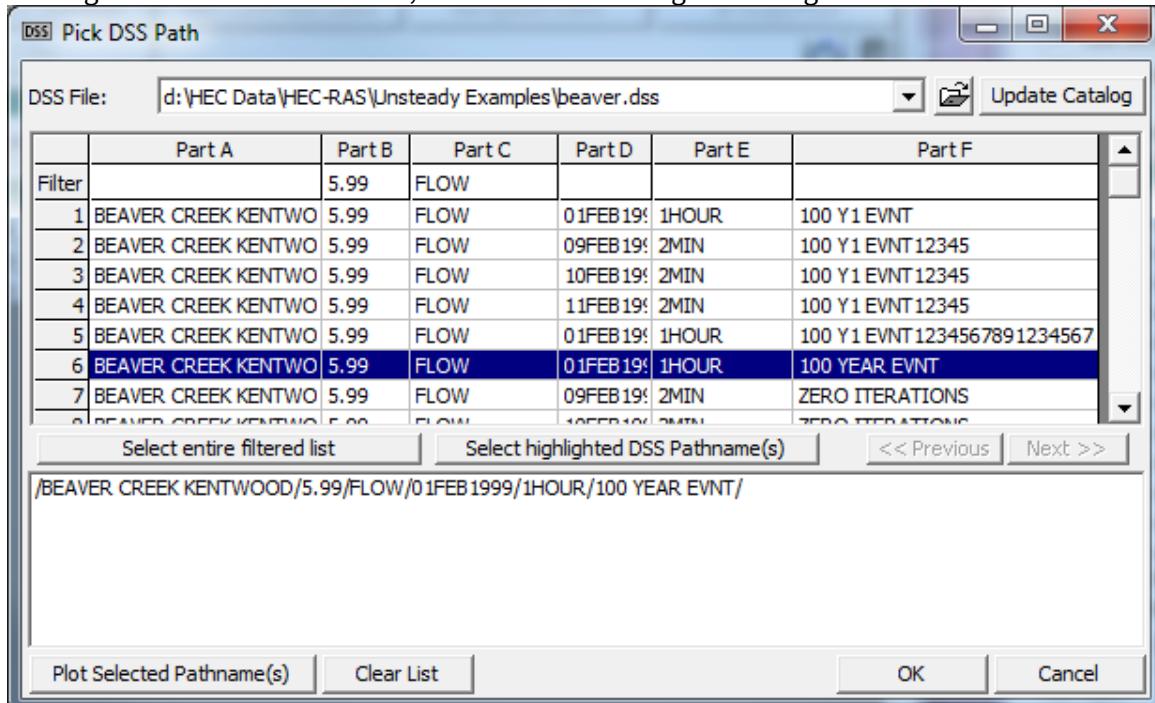


Figure 7.3. HEC-DSS File and Pathname Selection Screen

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 for 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 7-2). 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 7-4 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.

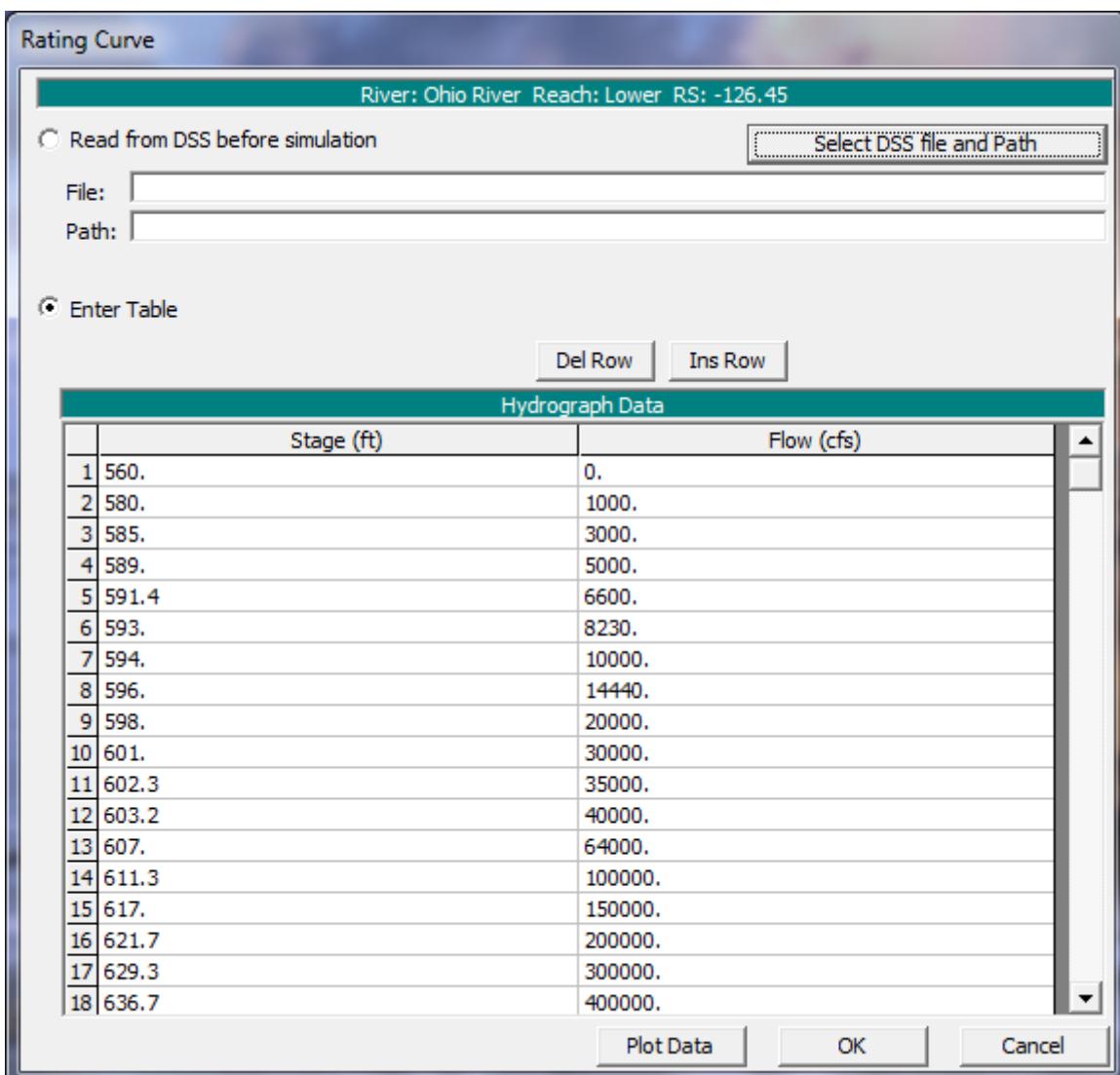


Figure 7.4. Example Rating Curve Boundary Condition Editor

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 (slope of the energy grade line) 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, however this is hard to obtain ahead of time. The average bed slope in the vicinity of the boundary condition location is often used as an estimate for 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 user specified cross section locations. 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 boundary condition can be applied to a river reach or a storage area. Groundwater can flow into, or out of, a reach or storage area, depending on the water surface head. 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 groundwater interflow option can also be linked directly to a storage area, for modeling groundwater exchange with ponding areas. The computed flow is proportional to the head between the river (or storage area) and the groundwater reservoir.

The computation of the interflow is based on Darcy's equation. The user is required to enter the coefficient of permeability (hydraulic conductivity, K, in feet/day), 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). If the groundwater time-series represents the stage immediately at the cross section (for instance if the user has detailed results from a groundwater model), then the user should enter the "K of the streambed sediment" and the "Thickness of the streambed sediments." If the groundwater time-series is the stage some distance from the cross section (such as a monitoring well), then the user should enter an "Average K of the groundwater basin" and the "Distance between the river and the location of the groundwater stage."

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 7-5 shows an example of the Times Series of Gate Openings editor. As shown in Figure 7-5, 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.

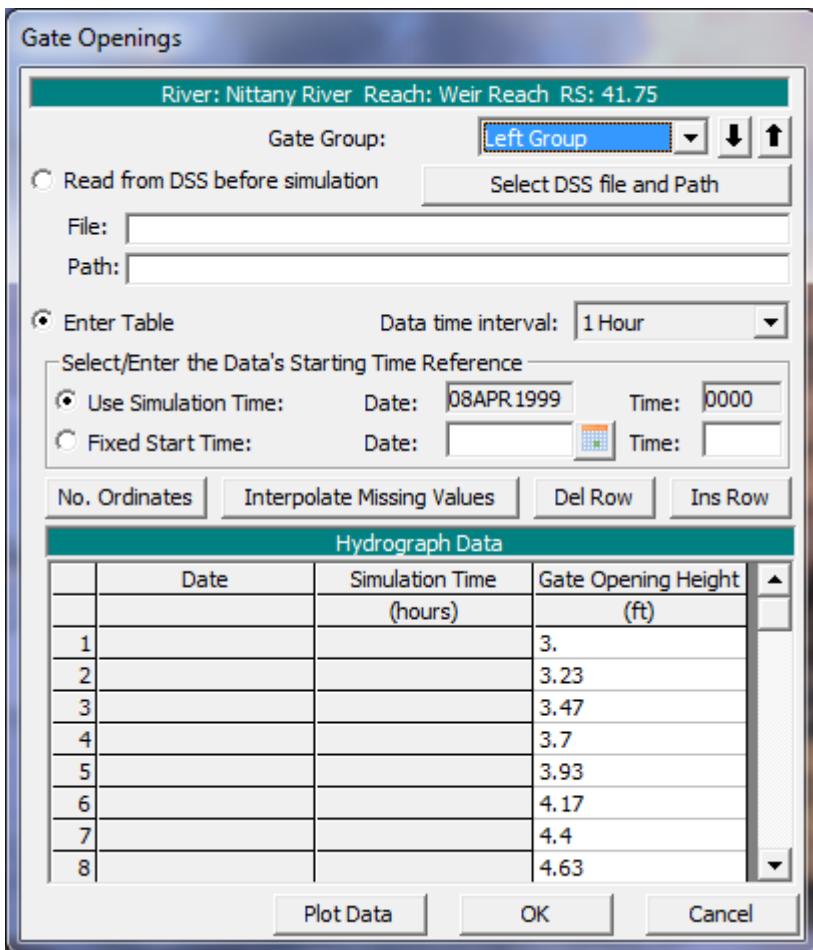


Figure 7.5. Example Time Series of Gate Openings Editor

Warning: Opening and closing gates too 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 (**Based on upstream WS**); or based on the water surface at a user specified cross section or storage area (from any location in the model) (**Based on specified reference**); or based on a difference in water surface elevation from any two user defined reference locations (**Based on difference in stage**). 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.). If the gate operating criteria is a stage difference, the user can specify a stage difference for when the gate should open and a stage difference for when the gate should begin to close. Stage differences can be positive, zero, or negative values. The user must also enter a maximum and minimum gate opening, as well as the initial gate opening. Figure 7-6 shows an example of this editor.

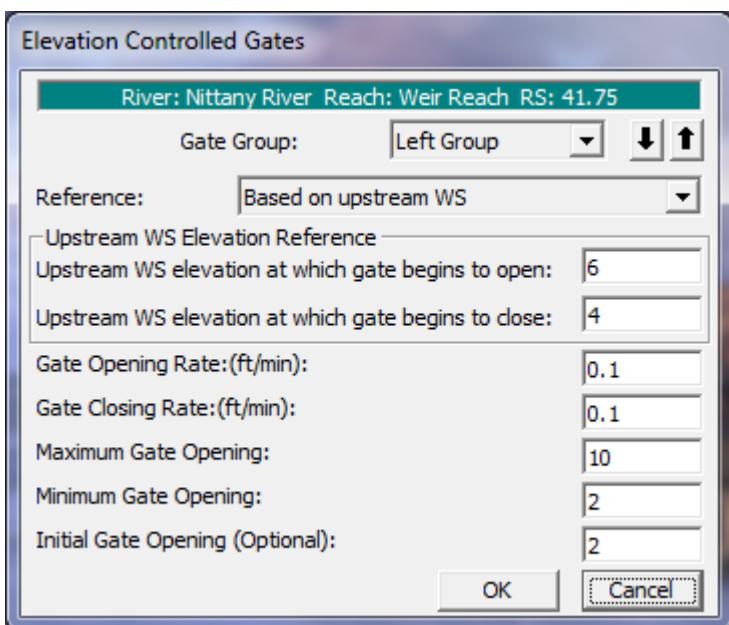


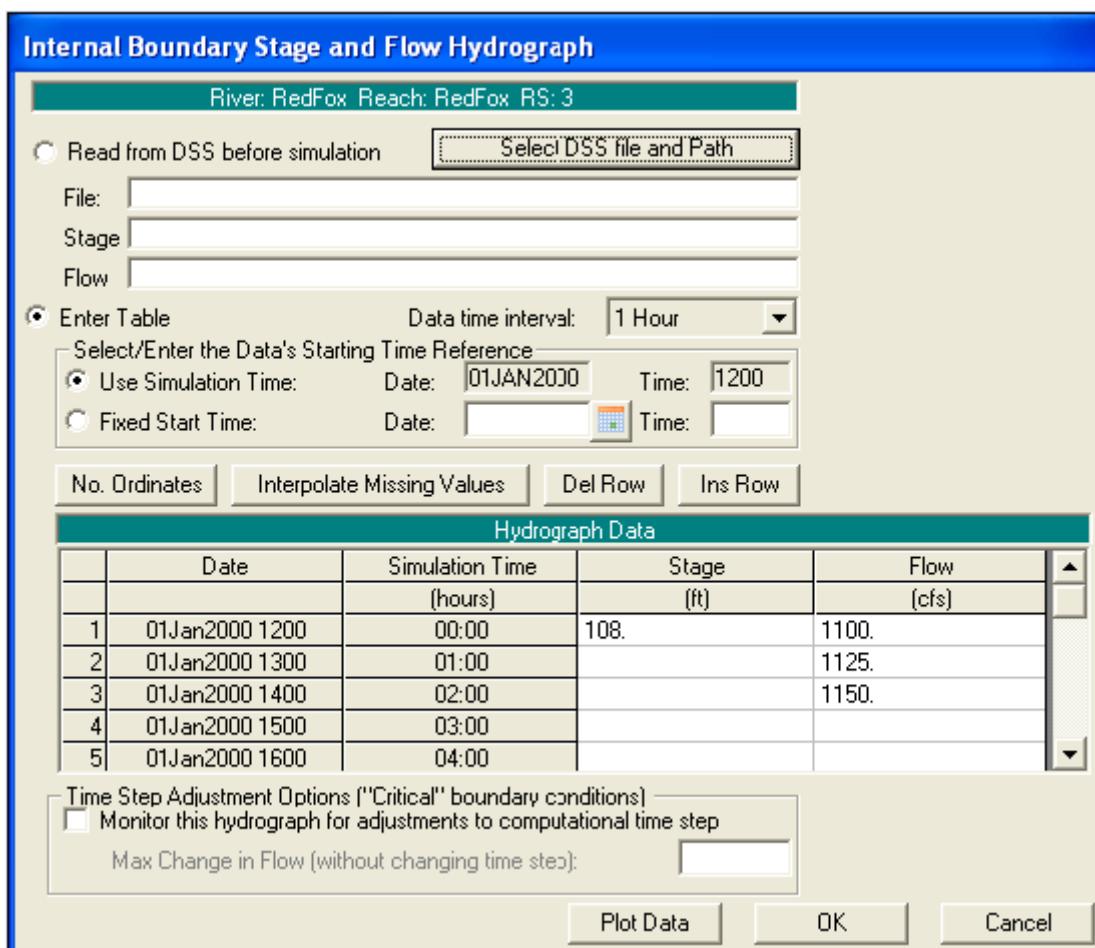
Figure 7 6. Elevation Controlled Gate Editor

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 can be found in the [Navigation Dams](#) section.

Internal Boundary Stage and/or Flow Hydrograph

This option allows the user to enter a known stage hydrograph and/or a flow hydrograph, to be used as an internal boundary condition. The boundary condition can be used at a cross section immediately upstream of an inline structure in order to force a known stage and/or flow for part or all the simulation. It can also be used at an "open" cross section (one not associated with a hydraulic structure). For example, in order to force the water surface to match the water surface from known gage data. If the user enters only a stage hydrograph, then the program will force the stage at this cross section and it will solve for the appropriate flow (in order to balance the unsteady continuity and momentum equations). Similarly, if the user enters only a flow hydrograph, then the program will force the flow at this cross section and it will solve for the appropriate stage. The user may also enter both stage and flow data. In this case, the program will force the stage (and solve for flow) as long as there is stage data. Once the stage data runs out, the program will start using the flow data (force the flow and solve for stage). This can often be useful when performing a forecast. Regardless of whether a stage and/or flow hydrograph is entered, if all of the time series data runs out before the end of the simulation, then the program will treat the cross section as a regular cross section and will solve for both flow and stage in the normal manner.



The Internal Boundary (IB) Stage and Flow Hydrograph editor is shown in Figure 7-7. In the example above, the program will force the stage to be 108 feet up until the start of the simulation time (that is, during the initial backwater and warm up period) and then it will start using the flow data. The flow for the first \[non-warm up\] time step will be 1100 cfs transitioning to 1125 cfs over the next hour. After two hours, the program will no longer force either stage or flow but, instead, will solve for both in a normal manner. Note: For an inline structure that has known discharges for all or part of the simulation, entering a single stage at the start of the simulation is a quick and easy way to enter an initial stage. (The alternative way to enter an initial stage is to go to the *Options* menu on the Unsteady Flow Data editor and select *Internal RS Stages...{*}) For the simulation in Figure 7-7, if the user had entered another stage, for example 107.5 feet at 01:00 hours, then the program would transition down to a stage of 107.5 feet during the first hour. At the start of the second hour, the flow would immediately be forced to a value of 1125 cfs. These sudden transitions may cause stability problems if the values are too far out of balance. For instance, if the program computed a flow value of 4400 cfs in order to force a water surface of 107.5 at the end of the first hour, then the system will receive a "shock" as the flow is forced to 1125 during the next time step. If a known hydrograph is entered for an inline structure (i.e. for the cross section immediately upstream of the inline structure) that has gates, then a boundary condition for the gate operations must still be entered (time series, elevation control, etc.). When the hydrograph information runs out, the program will use the gate operations from the boundary conditions and will solve for the flow and stage at the inline structure in a normal manner (computing weir and/or gate flow). As with the change from a stage hydrograph

to a flow hydrograph, the change from known (stage or flow) hydrograph to normal inline operations can cause a "shock" if the new computed value is too far out of balance from the previous one. If the stage and/or flow hydrograph for a reservoir is entered for the entire simulation, then the physical characteristics of the inline structure will not affect the solution. In this situation, the inline structure does not even have to be entered into the geometry—the IB hydrograph could just be attached to an "open" cross section and the headwater and tailwater stages and flows would be the same. However, neither does it matter if the inline structure is included, and it may be convenient for display and/or output. If the user elects to use a DSS stage/flow hydrograph, then the "end of data" should be entered as a -901 in the DSS record, which is the missing data code for HEC-DSS. If the data is entered in the table, as shown in the example, then the end of data is a blank line. Do not enter -901 in the table (unless this happens to be a real value).

"Rules" Editor

Currently HEC-RAS has the ability for the user to develop their own set of rules for controlling Inline structures, Lateral Structures, and Storage area Connections. The details of how the "Rules" editor can be used can be found in the [User Defined Rules for Hydraulic Structures](#) section.

Precipitation

This option allows the user to enter a precipitation hyetograph to either a Storage Area or a 2D Flow Area. This option allows the user to enter incremental precipitation vs time. The precipitation is used for the entire storage area or 2D flow area, with no spatial variability. If you want to use precipitation that is variable both spatially and in time, you can do that from the **Meteorological Data** Tab. Please see the section on Meteorological Data later in this chapter.

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 7-8.

As shown in Figure 7-8, 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. To add additional river station locations to the table use the **Add RS** button. If the river system is dendritic (no loops anywhere in the system), the user can leave all of the flow data fields blank. When this is done the software gets flow data from the first value of all of the boundary condition hydrographs (Upstream and lateral inflows). Flows are set from upstream to downstream by adding flows together at junctions as appropriate.

The user can also enter an initial water surface elevation for all storage areas in the model. A storage area can start out dry by entering the minimum elevation of the storage area. The button called

Import Min SA Elevations is an option to assist in getting the minimum elevation for any storage area. A new option introduced into HEC-RAS since version 5.0, is that user's can leave storage area elevations blank. If a storage area's initial conditions is left blank then generally it will start out dry. However, if that storage area is connected to a lateral structure, and the water surface in the river is higher than the lateral structures minimum weir elevation, then the storage area will be set to an average elevation of the water surface going over the weir. This option is very useful for setting the initial conditions in storage areas that are hydraulically connected right at the beginning of the simulation. Additionally, if a storage area is an upstream boundary of a river reach, and the initial conditions are left blank, then the storage area will be set to the same water surface elevation at the upstream cross section of that reach.

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.

Another option available is to set the initial flow and stage from a profile from a previous run. This option can be selected from the **File** menu, and it is called **Set Initial Conditions (flow and stage) from previous output profile**. When this option is selected, a window will appear allowing the user to select a plan and profile from an existing run in the project. Once the **OK** button is selected, then all of the flows and stages from that plan/profile will be set for the initial conditions of the current unsteady flow file. This option can be very handy when the initial conditions are causing some oscillation or stability problems in the run.

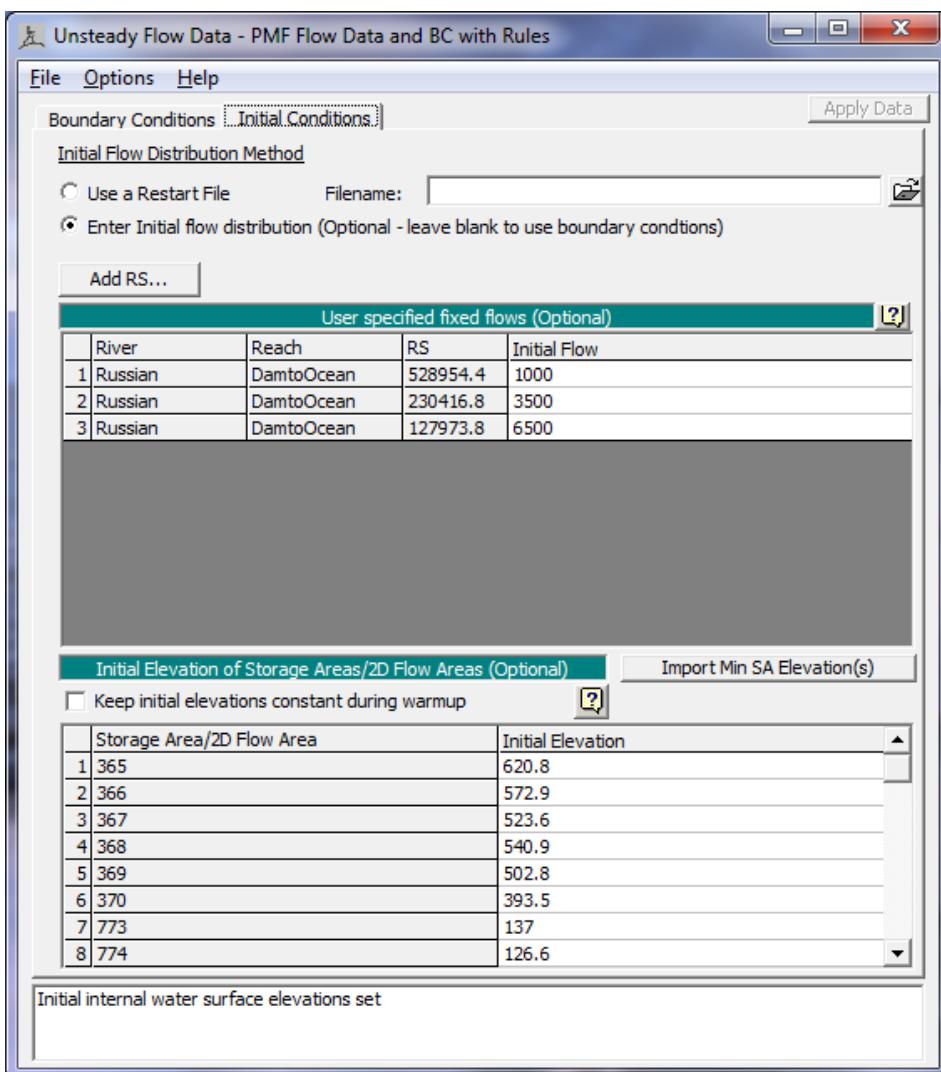


Figure 7.8. Initial Conditions Editor.

Meteorological Data

HEC-RAS now has the capability to have spatially varying precipitation, wind, and infiltration. Spatial precipitation and wind data are added into the Unsteady Flow Boundary Conditions editor. While infiltration information is defined as a spatial layer in HEC-RAS Mapper.

Spatial precipitation and wind data can be entered into HEC-RAS as either gridded data or point gage data. The spatial precipitation/wind data is added into HEC-RAS through the Unsteady Flow Boundary Conditions editor. When that editor is open, you will find a tab on the main window called "Meteorological Data".

NOTE: To learn how to enter and use spatial precipitation and/or wind data with HEC-RAS, please review the section called "**Global Boundary Conditions**" in the **2D Modeling User's Manual**.

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 structure 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.

Flow Minimum 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?).

Observed Data

Observed data are entered into the Unsteady Flow Data editor from the **Observed** Data tab. This option allows the user to enter data in the form of **observed time series data, high water marks**, or an **observed rating curves** at a gage. When an observed time series 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. If high water marks are entered, they will show up on the cross section and profile plots when the Max Water Surface profile is being plotted. If an observed Rating curve from a gage is entered it will appear on the rating curve plot for comparison to the computed stage versus flow values.

Observed Stage and Flow Time Series Data

To use the observed time series option, the user selects the **Observed Data** tab from the Unsteady Flow Data editor. When this tab is selected the window will appear as shown in Figure 7-9. As shown in the figure below, the user first selects the button labeled "**Select RS's**". This will bring up a new window that will allow the user to choose cross section locations to add into the observed data table. In addition to cross section locations, the observed data tables will include any 2D Flow Area Boundary Condition lines (**BC Lines**), as well as any **Reference Points** added to 2D flow areas or storage areas. Observed flow and stage time series can be added to the BC Lines, while only observed stage time series can be added to reference points.

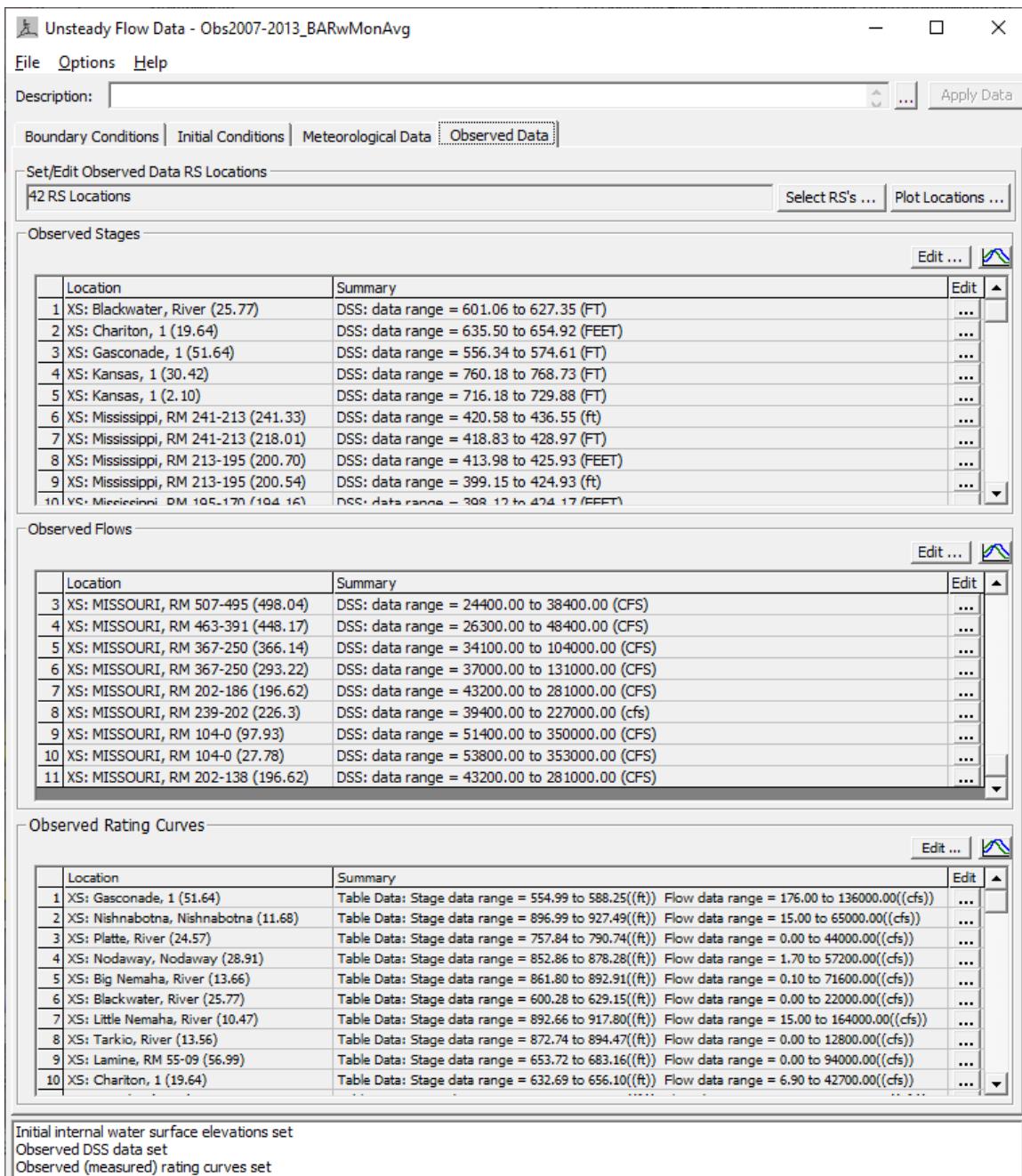


Figure 7.9. Observed Data tab for entering observed data

Once the user has entered locations for the observed data, the next step is to press the "Edit" button above the data type you want to enter (i.e. Observed Stages; Observed Flows; or Observed Rating Curves). If the user presses the **Edit** button above the Observed Stage or Flow table, a window will appear as shown in Figure 7-10.

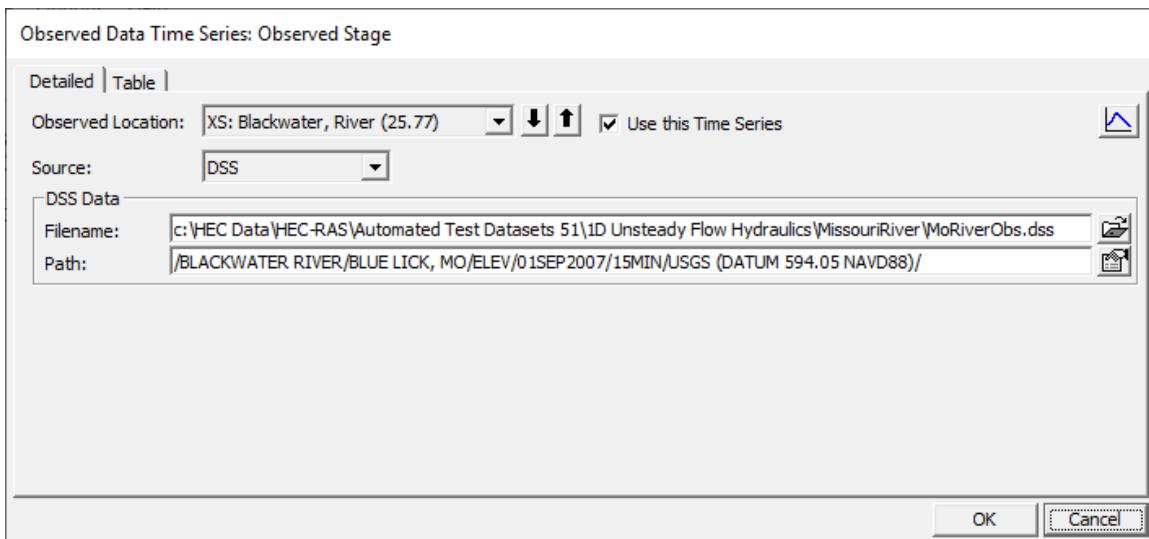


Figure 7 10. Observed Stage Time Series Editor

As shown in Figure 7-10, the window has two tabs. A "**Detailed**" tab and a "**Table**" tab. The detailed tab allows the user to view and edit data for one location at a time. The **Table** Tab allows the user to see the information for all the observed data locations. Whether you view the data in the detailed or table view, the user must first select a **Source** for the data. The "Source" options are: **DSS; Table; Constant**. Selecting **DSS** means the user has the observed data stored in an HEC-DSS file and wants to read the data from that DSS file. Selecting **Table** means the user wants to enter the data into a table directly in this editor. Selecting **Constant** means the user only has a "High Water Mark" for this location and wants to enter that High Water Mark as a "Constant" value. High Water Marks will be discussed in more detail below.

There is also an option called "**Use this Time Series**". This option must be turned on for all observed data that you want to be used for a run. If this option is not turned on, then it will not be used during the run and the user will not be able to plot the observed data against the computed results in any of the plots.

If the **DSS** source is selected, then the user must select the DSS file to use from the Filename chooser, as shown in Figure 7-10. Then the user must select the specific DSS pathname that contains the observed data. The DSS filename and location, as well as the DSS pathname can be cut and pasted into these fields.

If the **Table** source is selected, then the window will add a table for entering the data. An example Table is shown in Figure 7-11 below.

Observed Data Time Series: Observed Stage

Detailed | Table |

Observed Location: XS: Chariton, 1 (19.64) Use this Time Series

Source: Table

Table Data

| | Date/Time (ddmmmyyyy hh:mm:ss) | Simulation Time (Hours) | Value (ft) |
|----|--------------------------------|-------------------------|------------|
| 1 | 02Sep2008 00:00:00 | 0 | 637.3 |
| 2 | 02Sep2008 01:00:00 | 1 | 637.29 |
| 3 | 02Sep2008 02:00:00 | 2 | 637.29 |
| 4 | 02Sep2008 03:00:00 | 3 | 637.28 |
| 5 | 02Sep2008 04:00:00 | 4 | 637.3 |
| 6 | 02Sep2008 05:00:00 | 5 | 637.27 |
| 7 | 02Sep2008 06:00:00 | 6 | 637.3 |
| 8 | 02Sep2008 07:00:00 | 7 | 637.28 |
| 9 | 02Sep2008 08:00:00 | 8 | 637.28 |
| 10 | 02Sep2008 09:00:00 | 9 | 637.27 |
| 11 | 02Sep2008 10:00:00 | 10 | 637.28 |
| 12 | 02Sep2008 11:00:00 | 11 | 637.28 |
| 13 | 02Sep2008 12:00:00 | 12 | 637.28 |
| 14 | 02Sep2008 13:00:00 | 13 | 637.29 |
| 15 | 02Sep2008 14:00:00 | 14 | 637.29 |
| 16 | 02Sep2008 15:00:00 | 15 | 637.27 |
| 17 | 02Sep2008 16:00:00 | 16 | 637.28 |

Data Type: INST-VAL
Units: ft

OK | Cancel

Figure 7.11. Table for entering Observed Time Series Data.

As shown in Figure 7-11, the table has a drop down at the top. This drop down allows the user to select how they want to enter the data. The options are:

Create Regular Interval Time Series: This option allows the user to enter the observed data at a regular interval, such as 1 hour, etc... The user can select to have the Observed data start at the beginning of the simulation time (Which is based on the time window for the run) or they can select to enter a fixed start time for the data.

Enter Date/Time in All Rows: This option allows the user to enter the data as irregular data, meaning the dates and time are not on a regular interval. For this method, the user must enter the data/time for all the observed values.

Enter Simulation Time in All Rows: This option allows the user to enter the simulation time. Additionally, the user must select the **Data Type**; **Time Interval**; and the **Units** of the data. The Data Type can be: INST-VAL; PER-AVE; or PER-CUM. "INST-VAL" stands for instantaneous values, which is the most common for this type of data. "PER-AVE" stands for Period Average data, which means that the data is an average over the time period. "PER-CUM" stands for Period Cumulative, which means the data are cumulative values over the time period, and they are stored with a date and time at the end of the time period. PER-CUM does not make sense for observed time series data, but it is used for other data types, such as precipitation data.

If the **Constant** value is selected for the "Source" of the data, the window will change and only require the user to enter a single value and the units of that value.

High Water Marks

To enter Observed High Water Marks into HEC-RAS, select the **Edit** within the **Observed Stages** area, then select the **Constant** values as the data source. The editor will look like what is shown in Figure 7-12

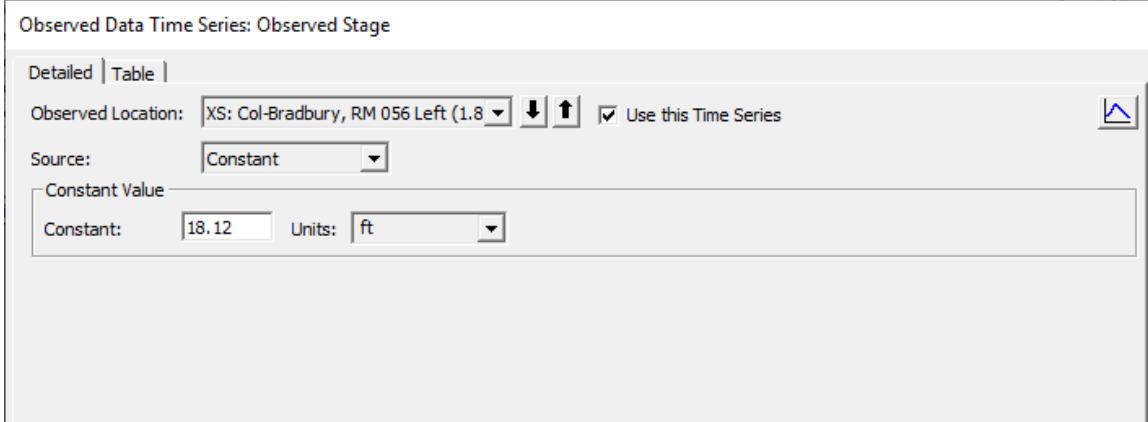


Figure 7-12. Observed High Water Marks entered as Constant values.

If the high water mark is not exactly at one of the user entered cross section locations, then the user should pick the cross section just upstream of the observed location and then enter the distance from that cross section to the observed high water mark under the column labeled **Dn Dist.**

Observed Rating Curves

To use the Observed Rating Curve option, the user presses the **Edit** button within the **Observed Rating Curves** portion of the window (Near the bottom). When this button is selected a window will appear as shown below in Figure 7-13.

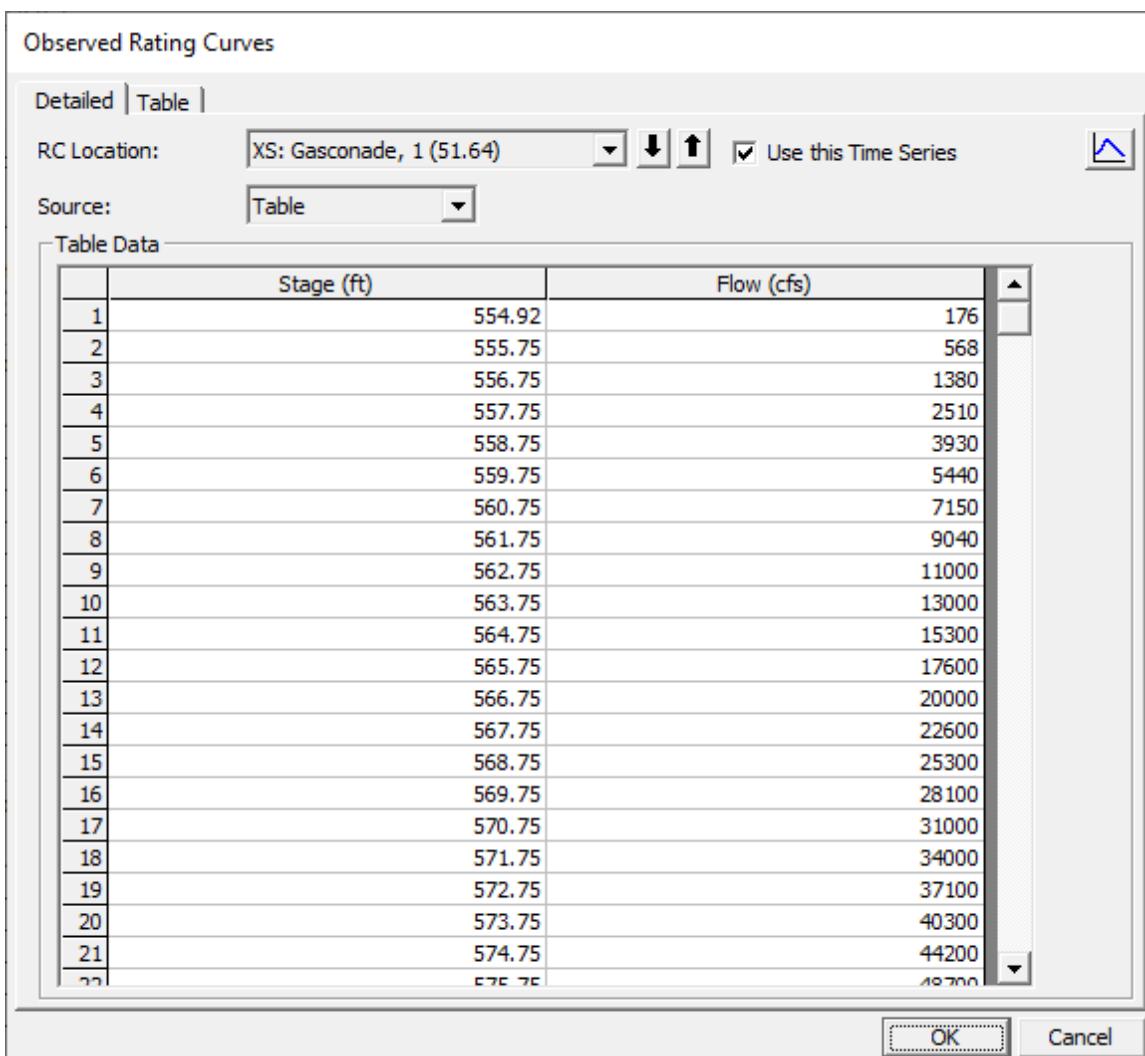


Figure 7.13. Observed Rating Curve Editor.

As shown in Figure 7-13, the user can either enter the data in a Table, or it can be read in from DSS.

Water Temperature (for Unsteady Sediment).

This option allows the user to put in a time series of water temperature data to be used in conjunction with the Unsteady Flow Sediment Transport capabilities. Water Temperatures are used for Fall velocities in computing sediment deposition.

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 7-14 (except yours may not have a Plan title and short ID).

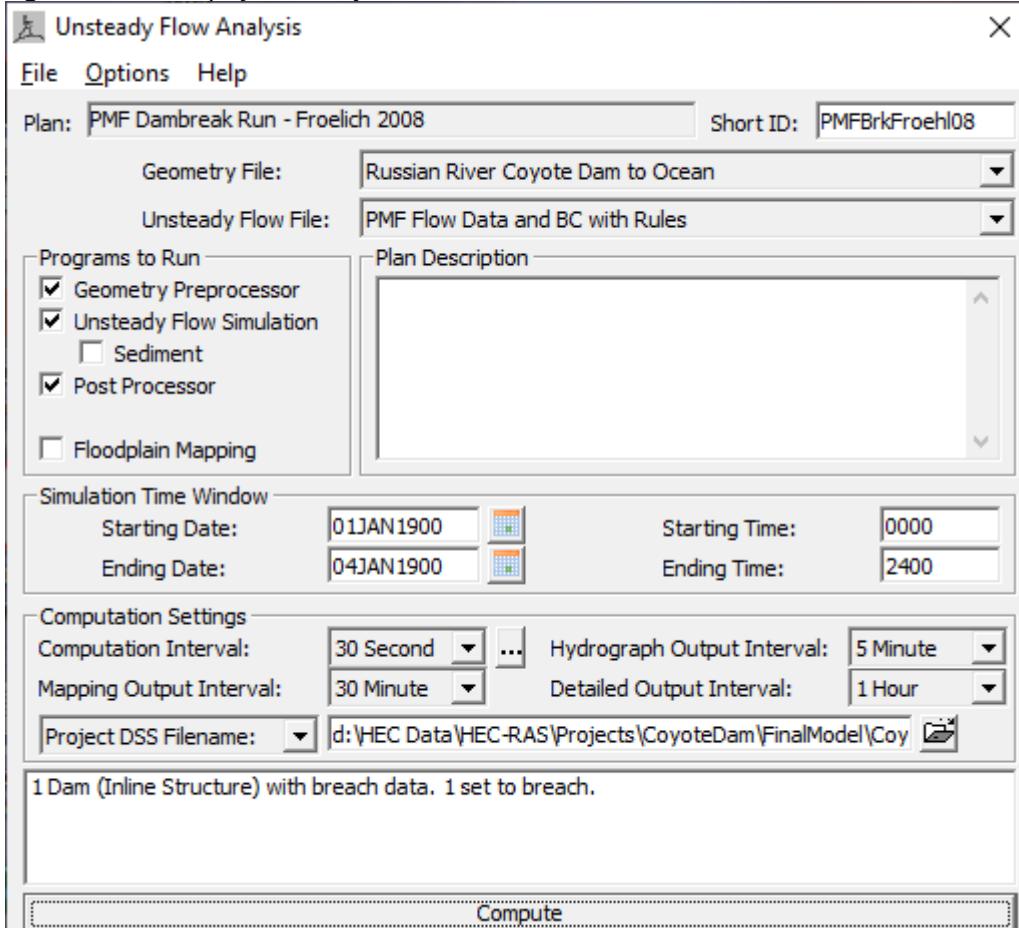


Figure 7 14. Unsteady Flow Analysis Window

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. Additionally, there is an option to automate the process of computing a Static flood inundation map (Depth Grid), or other static map layers (i.e. Arrival time grids, flood duration grids, etc...) after the unsteady flow simulation has finished. Computing static (Grids to disk) inundation maps is accomplished by checking the **Floodplain Mapping** option under the Programs to Run area. However, this option will only work if you have previously set up the computation of Static Map Layers for this particular Plan in HEC-RAS Mapper. Please see the Chapter on HEC-RAS Mapper in this document, for details on how to do this.

Geometric Preprocessor

The Geometric Preprocessor 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 preprocessor must be executed at least once, but then only needs to be re-executed if something in the geometric data has changed.**

Cross Section Property Tables

Cross sections are processed into tables of elevation versus hydraulic properties of areas, conveyance, and storage. The preprocessor creates hydraulic property tables for the main channel, and the floodplain. The floodplain hydraulic property tables are a combination of the left overbank and right overbank properties summed together in a single set of curves called the "Floodplain". The Unsteady flow engine uses a simple arithmetic average of the left and right overbank reach lengths in order to compute a floodplain reach length (i.e. the floodplain reach length is computed as the left overbank reach length plus the right overbank reach length, then divide by 2). The floodplain reach length is used to calculate storage volume, friction losses, etc... and all other hydraulic properties used in the continuity and momentum equation for the floodplain area. This is different than what is done in the steady flow computational program, in which it computes left overbank and right overbank hydraulic properties separately.

Each hydraulic property 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 too 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 (Hydraulic Table Parameters)** button from the **Geometric Data editor**. When this option is selected, a window will appear as shown in Figure 7-15.

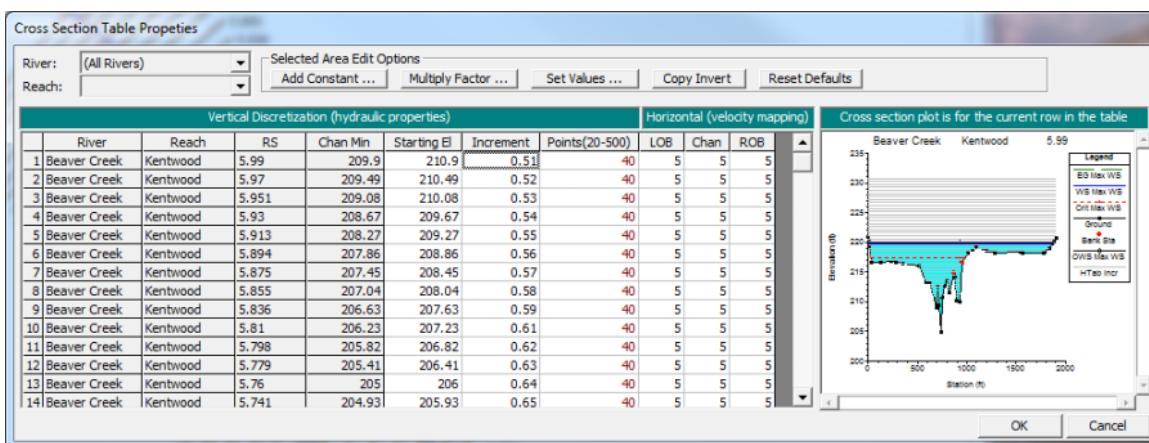


Figure 7.15. Hydraulic Table Parameters for Cross Sections

As shown in Figure 7-15, 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 the starting elevation 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 are available for manipulating the data (using standard windows function keys). **Warning: The hydraulic tables for cross sections must be high enough to capture all possible water surface elevations. It is up to the user to ensure that the combination of the increment and number of points produces a table that will extend to a high enough elevation. If the computed water surface goes above the table, properties are extrapolated by extending the last to points linearly. This extrapolation can often cause the model to go unstable.**

Three additional fields are also in the table, under the heading of "**Horizontal Velocity Mapping**". These fields are used for computing more detailed velocity estimates within a cross section, than the default of a single average value in the left overbank, main channel, and the right overbank. These fields can be used to enter a number of slices to further discretize the computation of average velocities within a cross section. These vertically and horizontally averaged velocities are used for mapping velocity within the 1D cross sectional flow fields.

Once the Geometric preprocessor is run, the user can view the cross section hydraulic property tables for the channel and floodplain from the HEC-RAS interface.

Hydraulic Structure Property Tables

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 **Storage Area Connection editor**. When this button is pressed, the window in Figure 7-16 will appear:

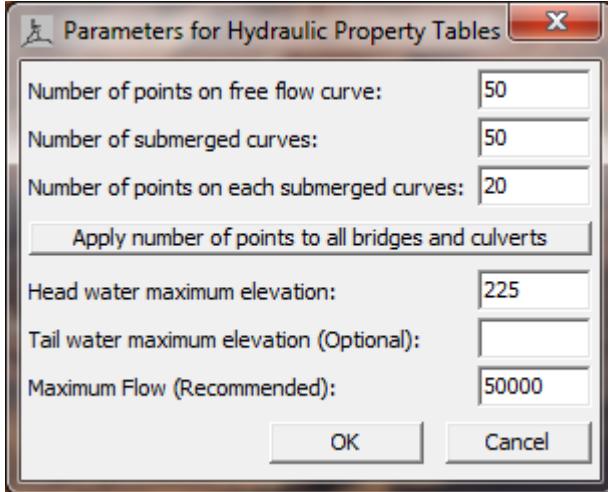


Figure 7-16. Hydraulic Properties Table for Bridges/Culverts

As shown in Figure 7-16, the user can set the number of points to be computed on the free-flow rating curve (maximum of 100 points); the number of submerged curves to be computed (maximum of 60); and the number of points on the submerged curves (maximum of 50). The default values for these parameters are 50, 50, and 20 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 (required), tail water maximum elevation (optional), 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.**

Viewing the Preprocessor Hydraulic Properties Tables



Once Hydraulic Table parameters have been entered for the cross sections and the structures, and the preprocessor has been run, the user can view the computed curves by either selecting **Hydraulic Property Plots** from the **View** menu of the main HEC-RAS window, or by pressing the **HT** button on the main HEC-RAS window. When this option is selected the following window will appear.

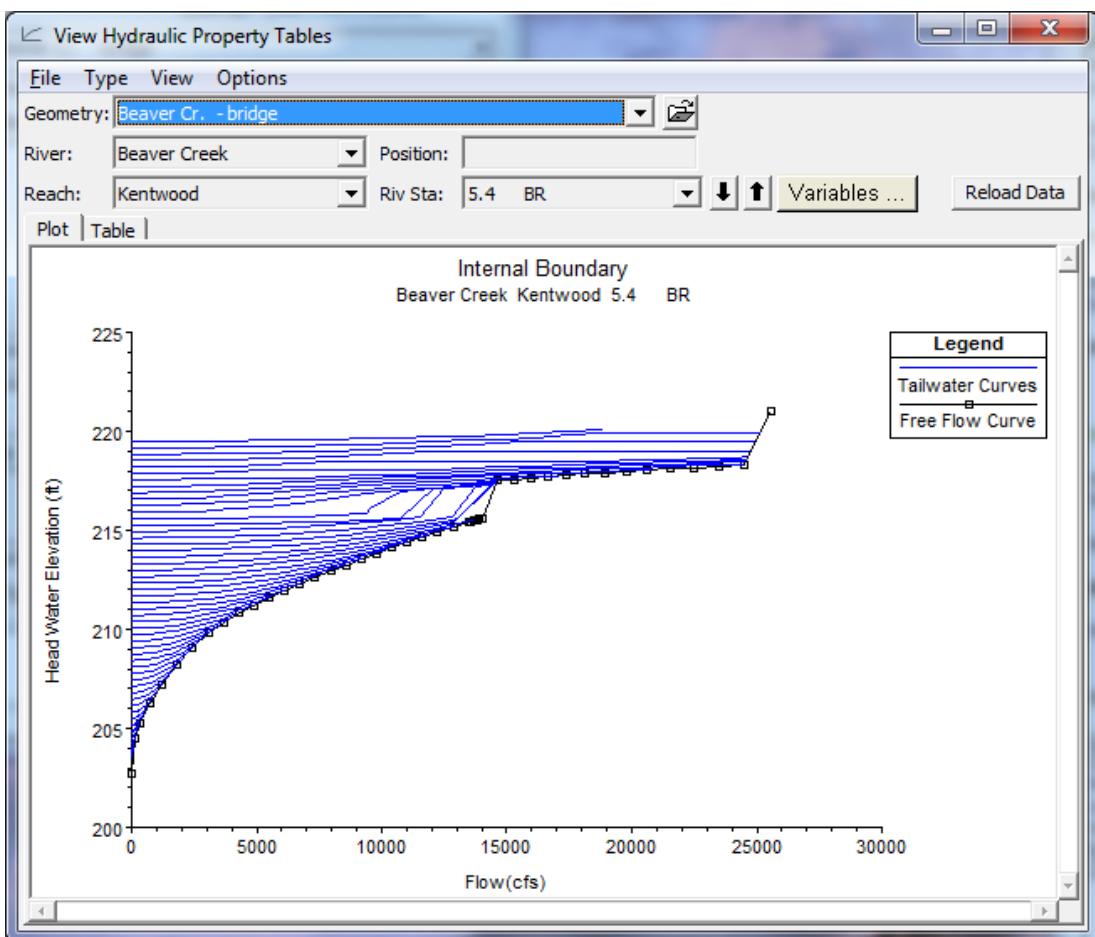


Figure 7.17. Plot of Hydraulic Property Tables Computed by the Preprocessor.

As shown in Figure 7-17, the user can plot properties for cross sections, internal boundaries (Bridges and culverts), or storage area connections by selecting the type of plot from the **Type** menu option. Additionally the information can be viewed in tabular form by selecting the **Table** tab on the plot. **The user should view all of the computed curves for their model closely to ensure that they are hydraulically appropriate.**

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. Additionally, if a storage area connection is set up as a weir only, it can either be preprocessed into curves, or the user has the option of having the program compute the hydraulics on the fly for each time step.

Unsteady Flow Simulation

The unsteady flow computational program in HEC-RAS uses the same hydraulic calculations (cross section properties, bridge and culvert hydraulics, weirs, gated structures, etc...) that HEC developed for steady flow; however, the solution of the unsteady flow equations (Continuity and momentum equation) are solved using a unique skyline matrix solver developed by Dr. Robert Barkau for his UNET (Unsteady NETwork model) program. The unsteady flow simulation is actually a three-step process. First data is read from HEC-DSS, if necessary, and then converted into the user specified

computation interval. Next, the RasUnsteady.exe 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. The program then performs the unsteady flow calculations. The final step is a program called RasDSSWriter.exe. This software takes the results from the RasUnsteady.exe run and writes the stage and flow hydrographs to an 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 inundation mapping (HEC-RAS Mapper), 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. If you only want to get this maximum envelope profile, then simple select "**Max. Prof.**" from the Detailed Output Interval drop down list.

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, as well as the maximum stage envelope profile. 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 and the instantaneous profiles (stages and flows). These computed stages and flow are sent to the HEC-RAS steady flow computation program RasSteady.exe. Because the stages and flows are already computed, the RasSteady.exe program does not need to calculate a stage, but it does calculate all of the hydraulic variables that are normally computed for steady flow. This consists of over three hundred hydraulic variables that are computed at each cross section for each flow and stage.

WARNING: The purpose of the post processor is to compute hydraulic output variables that are not computed by the Unsteady Flow computational engine. However, the Post Processor is actually just the Steady Flow computational engine. There are a few differences between assumptions made in the steady flow and the unsteady flow computations engines that will affect some of the post processor output. For example, the unsteady flow engine combines the left and right overbanks areas into a single "Floodplain" flow area, while the steady flow engine computes hydraulics for the left and right overbank separately. Therefore, some of the post processor output is computed like the steady flow program performs computations, even though the unsteady flow program makes some different assumptions. Specifically, the computation of left and right overbank cumulative volumes is based on separate reach lengths in the steady flow program, and the post processor. However, the unsteady flow program combines the left and right overbank hydraulic properties in the equations, and uses a simple average of the two overbank lengths to compute volumes between the cross sections in the

floodplain area. Also, the bridge computations are performed with a set of pre computed curves in the unsteady flow engine, while the steady flow engine computes the hydraulics through the bridge each time. These differences can lead to some small differences in some of the post processor output, as compared to how the Unsteady flow engine actually performs hydraulic calculations.

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 Steady.exe 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 water surface elevation computed from the Post-Processor that is higher at the structure 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).

Unsteady Flow Computation Settings

The Computation Settings area of the Unsteady Flow Analysis window contains: the computational interval; hydrograph output interval; detailed output interval; Mapping Output Interval; a computation level output flag; 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 20. In other words, if the flood wave goes

from its base flow to its peak flow in 10 hours, then the computation interval should be equal to or less than 0.5 hours (30 Minutes). This way of estimating the time step tends to give an upper boundary as to what the value should be.

A second way of computing the appropriate time step is by applying a numerical accuracy criteria called the Courant condition. The Courant condition criteria looks at cross section spacing and flood wave velocity. The basic premise is that the computational interval should be equal to or less than the time it takes water to travel from one cross section to the next. A detailed description of the Courant condition can be found under the Model Accuracy, Stability, and Sensitivity section of this chapter. Use of a time step based on the Courant condition will give the best numerical solution, but it may cause the model to take a lot longer to run.

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.

Mapping Output Interval. This field is used to enter the interval at which the user will be able to visualize mapping output within HEC-RAS Mapper. A limited set of output is written to a separate

HDF5 output file that corresponds to the Plan file. For example, if you are running Plan 1, the Plan filename will be something like "filename.p01". An output file will be written out by the unsteady flow computational program, with the name "filename.p01.hdf". This file will contain results from the unsteady flow simulation, written out at the interval the user defines for the **Mapping Output Interval**. The user will then be able to visualize spatial mapping output in HEC-RAS Mapper based on this time interval.

Computation Level Output. This option will instruct the program to write out a limited set of variables for each cross section at each computational time step to a separate output file. The variables that are written to this file are: water surface elevation; flow; maximum depth of water in the main channel; numerical error in the calculated water surface elevation; numerical error in the calculated flow; average velocity in the channel; and average velocity of the entire section. After a simulation is complete the user can plot and tabulate each of these variables spatially, or as a time series at an individual cross section. Because this output is at the computational time step level, it can be very useful in debugging model stability problems. The user can get to plots and tables of this information by selecting the **Unsteady Flow Spatial Plot** or the **Unsteady Flow Time Series Plot** options from the **View** menu of the main HEC-RAS window. Detailed descriptions of plotting and tabulating this output can be found under the Viewing Results Chapter of this manual.

Warning: Turning the **Computation Level Output** option on can create very large output files and it will also potentially slow down the computations. This option writes several output variables to a file for all locations in the model and for every computational time step. If you have a large data set and/or are running it for a long simulation time period, this file will be very large.

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. Hydrographs written to the DSS file are based on the user selected hydrograph output locations, as well as some default locations that HEC-RAS will always output a hydrograph.

Unsteady Flow 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; unsteady encroachments; dam breach; levee breach; ungaged lateral inflows; mixed flow options; time slicing; calculation options and tolerances; output options; friction slope methods for cross sections and bridges; initial backwater flow optimizations; run time computational options; checking data before execution, viewing the computation log, and viewing the computational log file.

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 7-18.

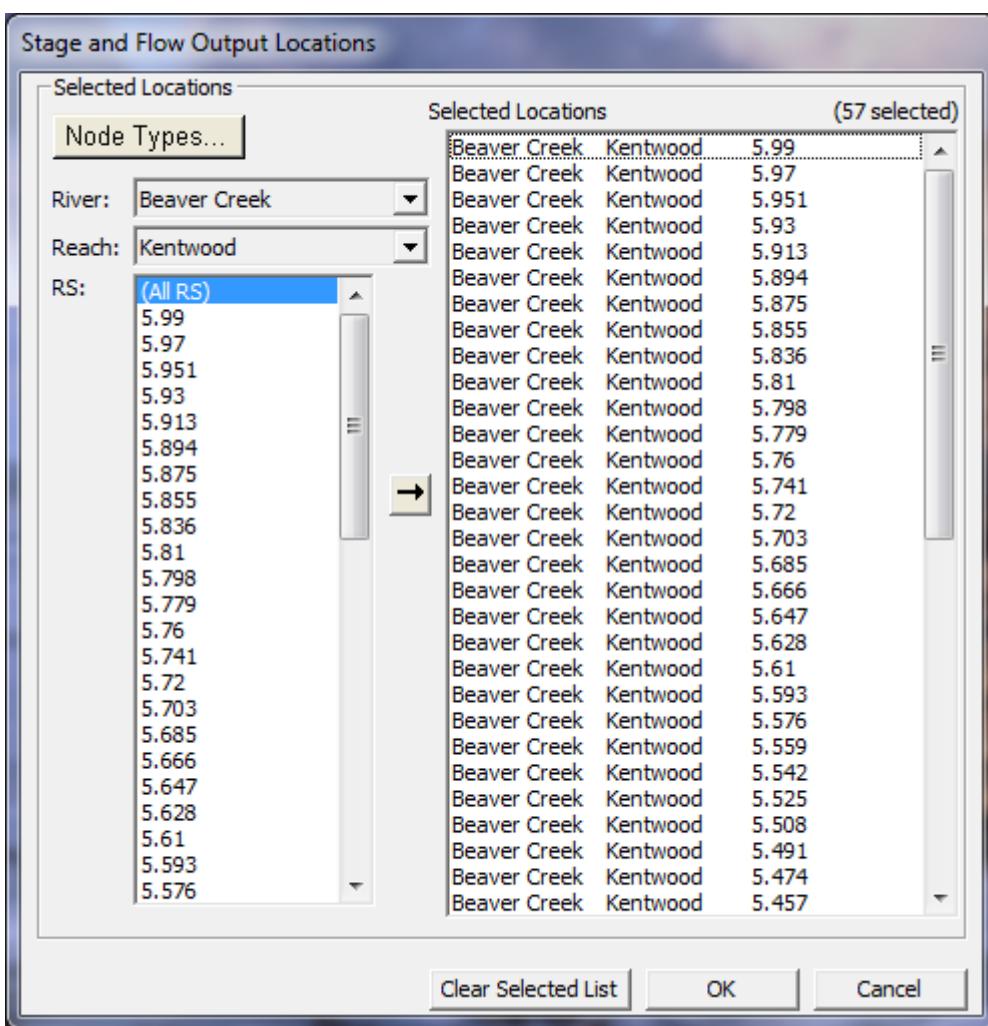


Figure 7 18. Stage and Flow Hydrograph Output Window.

As shown in Figure 7-18, 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.

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 7-19.

As shown in Figure 7-19, 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. Between the user entered flows, the model will use linear interpolation to obtain a roughness factor. If a flow is greater than the last user entered value, then that value is held constant. 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.

- ⓘ Flow Roughness factors is available as an option in both the Geometry Editor (saved with the geometry data) and the Plan Editor (saved in the plan file). If you put these factors in the geometry editor, they will be applied to every event that uses that geometry file. Normally you would do this when flow roughness factors are needed for a wide range of events, and you can come up with a set of factors that works well from low to high events. If you want to use flow roughness factors for real time calibration, or you believe they are much more event specific, then you would place them in the Plan File. When placed in the Plan file, they only get applied to that event being run with that plan.

Additionally, if you place these factors in both the geometry and the plan file, then you will be factoring the roughness twice. Factors placed in the geometry file get used during the Geometry Pre-Processing, and they will affect the hydraulic tables that get computed. Factors placed in the Plan file are used in real time during the unsteady flow computations, and they get applied each time step at each cross section location.

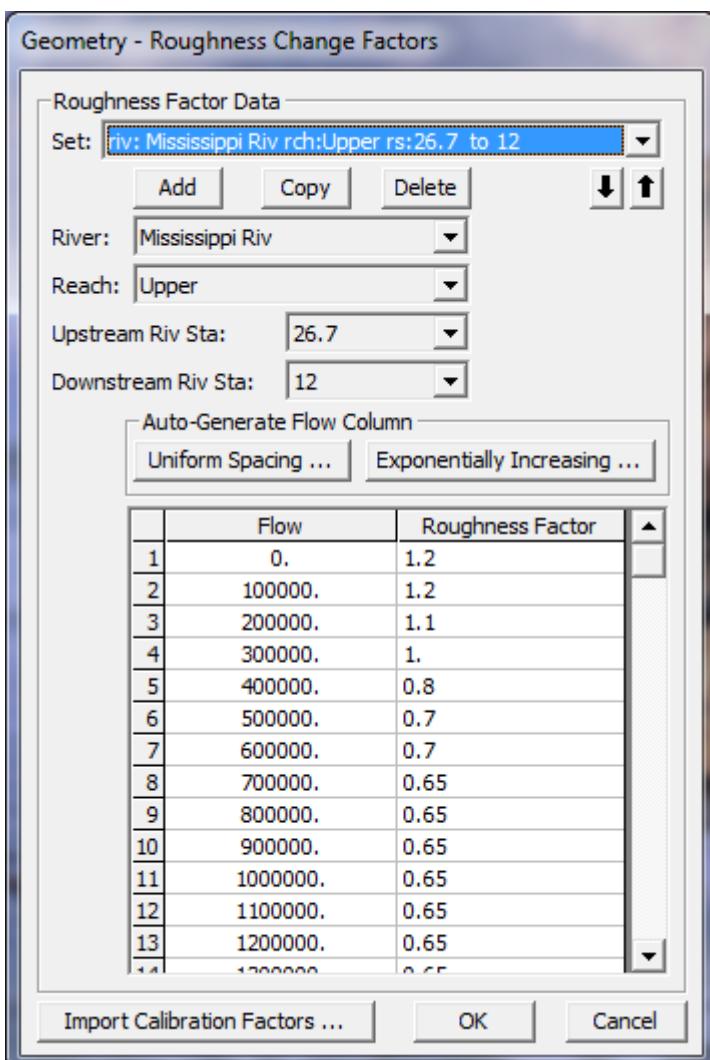


Figure 7.19. Flow versus Roughness Change Factors Editor

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 7-18.

As shown in Figure 7-20, 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. During the simulation, roughness factors are linearly interpolated between the user entered values.

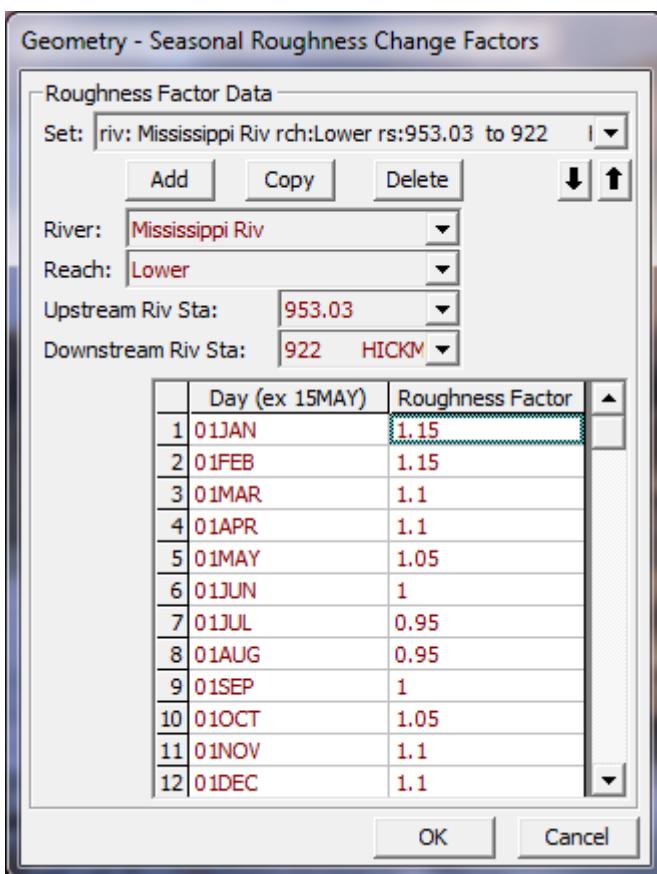


Figure 7.20. Seasonal Roughness Factors Editor

Automated Roughness Calibration. This option allows the user to perform an automated manning's n value calibration for an unsteady flow model. The results of this option are a set of flow versus roughness relationships, that can be applied in order to obtain a model that is calibrated from low to high flows.

To use this option, the user should first adjust the base Manning's n values (Channel and overbank values) in order to get a reasonable starting point for the automated calibration technique. The base set of Manning's n values should be within what is considered to be reasonable values for the type of river and overbank land use they are being applied to.

After a base set of Manning's n values are established, the user must break up the model into reaches for the purpose of applying and calibrating flow versus roughness factors. Once the flow versus roughness factors are set up, then the automated calibration of those reaches can be performed.

To use this option, select **Automated Roughness Calibration** from the **Options** menu of the Unsteady Flow Simulation manager. When this option is selected a window will appear as shown in Figure 7-21.

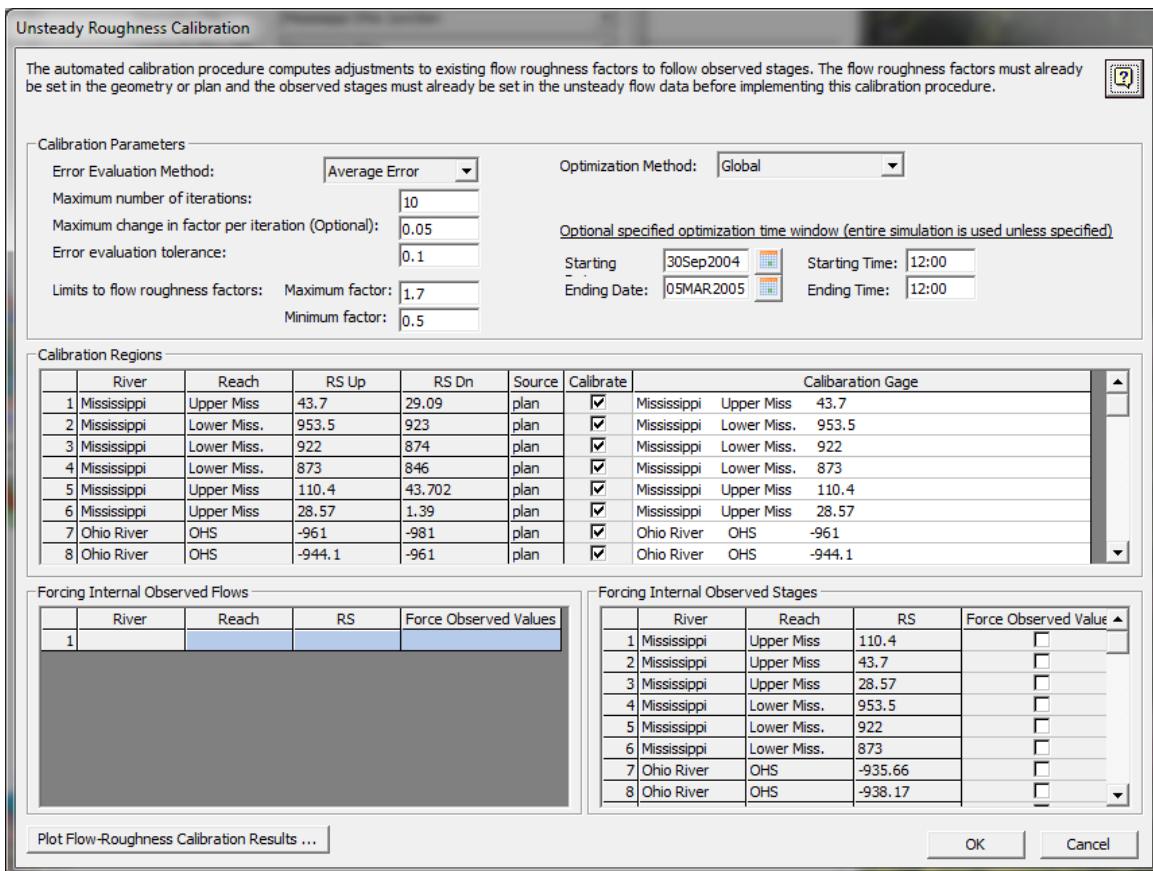


Figure 7.21. Automated Manning's n Value Editor

For a detailed discussion of the Automated Manning's n value editor and how to use this feature, please see Chapter 16 Advanced Features for Unsteady Flow Routing.

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:

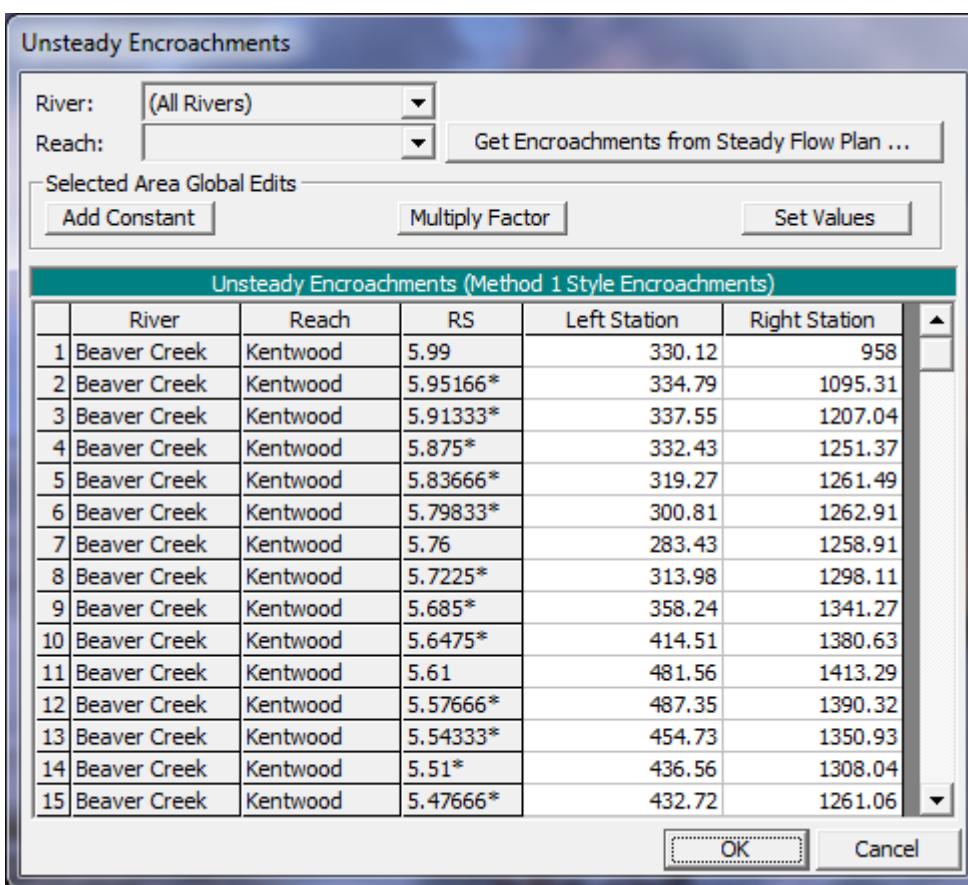


Figure 7.22. Unsteady Flow Encroachment Data Editor

As shown in Figure 7-22, 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.

Ungaged Lateral Inflows. This option can be used to automatically figure out the contribution of runoff from an ungaged area, given a gaged location with observed stage and flow. The software will compute the magnitude of the ungaged area hydrograph, based on routing the upstream flow hydrograph and subtracting it from the observed downstream flow hydrograph to get the ungaged inflow contribution. This is an iterative process, in which the program figures out a first estimate of the ungaged inflow, then reroutes the upstream and ungaged inflow again, until the routed

hydrograph matches the downstream observed hydrograph within a tolerance. More details of this simulation option can be found in Chapter 16 of this manual.

Computational 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 unsteady flow equations. There are separate tabs for: General; 2D Flow Options; 1D/2D Options; Advanced Time Step control; and 1D Mixed Flow Options.

- ◆ Increasing the default calculation tolerances could result in computational errors in the water surface profile.

When the "Calculation Options and Tolerances: option is selected, a window will appear as shown in Figure 7-23.

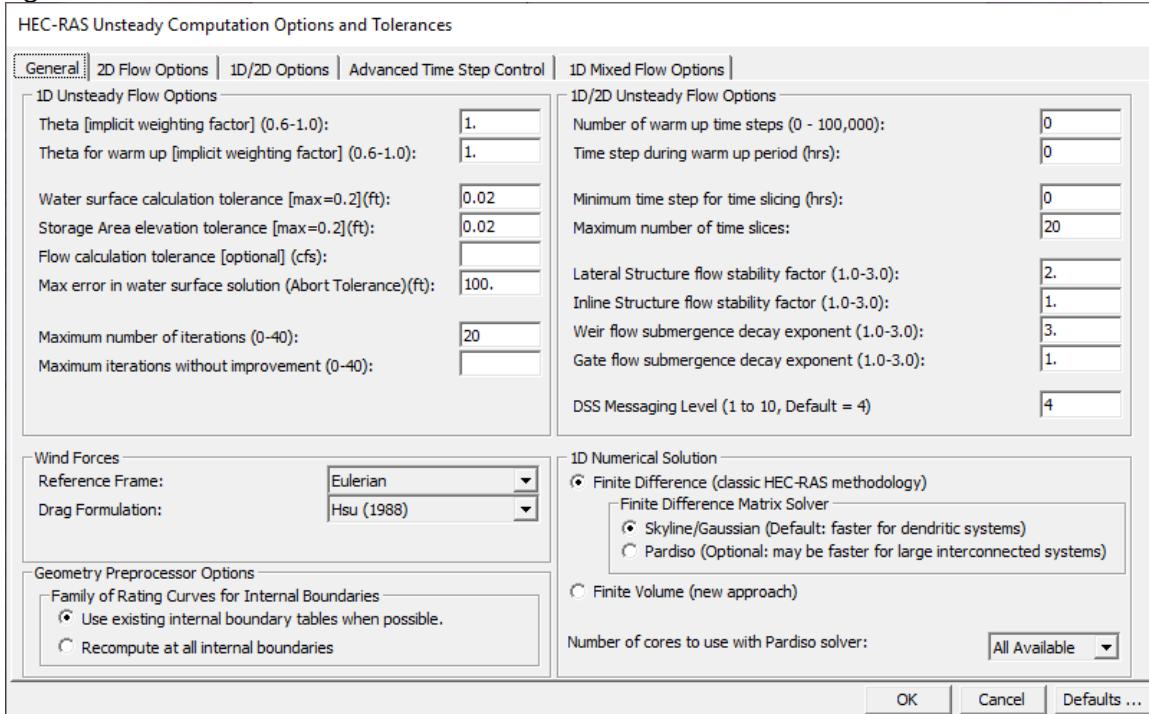


Figure 7-23 Computational Options and Tolerances with 1D Options tab shown.

The calculation options and tolerances are as follows:

General (1D Options) (see Figure 7-23):

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.

Theta for warm up: The unsteady flow solution scheme has an option to run what we call a "warm up period" (explained below). The user has the option to set a different value for theta during the warm up period versus the simulation period.

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 0.05 feet.

Flow calculation tolerance: This tolerance is used to compare against the numerical error in the computed flow versus the assumed flow for each iteration of the unsteady flow equations. The user enters a flow in cfs (or cms in metric data sets). The software monitors the flow error at all computational nodes. If the flow error is greater than the user entered tolerance, then the program will continue to iterate. By default, this option is not used, and is therefore only used if the user enters a value for the tolerance.

Maximum error in water surface solution: This option allows the user to set a maximum water surface error that will cause the program to stop running if it is exceeded. The default value is 100 ft. If during the computations, a numerical errors grows larger than this tolerance at any node, the program will stop the simulation at that point and issue a message saying that the maximum water surface error tolerance has been exceeded.

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 Iterations without improvement (0-40): This option allows the user to set a maximum number of iterations in which the solver can iterate without improving the answer. This option is not used by default. If turned on, it can increase the speed of the computations, but may cause larger errors or instabilities to occur if not used correctly. A good starting point for this option would be 5. What this means is, that if the program iterates five times and does not improve any of the solution errors, at all nodes in the system, then it will stop iterating and use the previous iteration that had the best answers up to that point. The premise here is that if the iteration scheme is not able to improve the solution for 5 consecutive iterations, more than likely it is not going to improve, and there is no point in wasting computational time by iterating all the way to the maximum number of iterations.

Number of warm-up time steps: Before the user entered simulation period begins, the program can run 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 0. This value ranges from 0 to 200.

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 the time step that has been set for the unsteady flow simulation period.

Minimum time step for time slicing: The program has an option to interpolate between time steps when it finds a very steep rise in an inflow hydrograph (see flow hydrograph boundary conditions earlier in this chapter). This option allows the user to set a minimum time step to use when the program starts reducing time steps during a steep rise or fall in flow at a flow boundary condition. This prevents the program from using too small of a time step during time slicing.

Maximum number of time slices: This option defines the maximum number of interpolated time steps that the program can use during time slicing, as described in the previous paragraph.

Lateral Structure flow stability factor: This factor is used to increase the stability of the numerical solution in and around a lateral structure. 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 lateral flow. The default value is 1.0. If you observe oscillations in the computed flow over a lateral structure, 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.

Inline Structure flow stability factor: This factor is used to increase the stability of the numerical solution in and around an Inline Structure. 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 flow. The default value is 1.0. If you observe oscillations in the computed flow over the inline structure, 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.

Gated 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.

DSS Messaging Level: This option will control the amount and detail of messages that get written to the Log File when reading and writing data to HEC-DSS. A value of 1 is minimal information and a value of 10 turns on the maximum amount of information. The default for this variable is 4.

Wind Forces: This option allows you to pick a frame of reference for computing wind forces, as well as which Drag (friction) Formulation equation to use. The default Reference Frame is **Eulerian**, which means the speed of the wind is based on a fixed point and does not account for the direction or speed of the water. If the user selects **Lagrangian**, then the software will take into account the speed and direction of the water to obtain the net speed of the air over the water. The Drag Formulation allows the user to select from one of four Drag coefficient equations, or to enter a constant drag coefficient. The default is the Hsu 1988 equation. Please see the Hydraulic Reference manual for more information on these equations.

Geometric Preprocessor Options: This section of the 1D computational options allows the user to control whether or not the program will read in previously computed curves for internal boundary conditions, like bridges and culverts, etc.. Or the user can force the geometric pre-processor to recompute all internal boundary conditions.

1D Numerical Solution: By default the 1D equation solver uses a **Finite Difference scheme** with a Skyline/Gaussian reduction matrix solution technique to solve the 1D Unsteady flow equations. There is an option to use the "Pardiso" matrix equation solver. We have found that even though the Pardiso solver can make use of multiple processor cores, the Skyline matrix solver is generally faster for dendritic river systems. However, for large, complex, looped systems, the Pardiso solver may produce faster computational results. The user should try the Pardiso solver if they have a larger complex system of interconnected reaches. If you turn on the Pardiso solver, there is an option to control the number of processor cores used for solving the matrix. By default, "All Available" will be used. However, the speed of the solution is sensitive to the number of cores used, and it is not always faster with using the maximum number of cores available. So user's should test the number of cores to get the maximum computational speed benefit.

A new **1D Finite Volume** solution scheme has been added to HEC-RAS as an option. This numerical scheme is similar to what we do in our 2D Finite Volume solver.

The current 1D Finite Difference solution scheme has the following deficiencies:

1. Cannot handle starting or going dry in a XS
2. Low flow model stability issues with irregular XS data.
3. Extremely rapid rising hydrographs can be difficult to get stable
4. Mixed flow regime (i.e. flow transitions) approach is approximate
5. Stream junctions do not transfer momentum

The new 1D Finite Volume approach has the following positive attributes:

1. Can start with channels completely dry, or they can go dry during a simulation (wetting/drying).
2. Very stable for low flow modeling
3. Can handle extremely rapid rising hydrographs without going unstable.
4. Handles subcritical to supercritical flow, and hydraulic jumps better.
5. Junction analysis is performed as a single 2D cell when connecting 1D reaches (continuity and momentum is conserved through the junction).

However, there are some draw backs to the new 1D Finite Volume solution scheme. These deficiencies are:

1. You cannot use lidded cross sections with the new 1D Finite Volume solution scheme. Which also means that it cannot handle pressurized flow, even with the Priestman slot option turned

on. If you have lidded cross sections, as soon as the water hits the high point of the lids low chord it will go unstable.

2.The 1D Finite Volume solution scheme is sensitive to the volume of water between any two cross sections. The equations are written from a "volume" perspective. If two cross sections are very close together, then there is very little volume between those two cross sections. The 1D Finite Volume scheme will require smaller time steps in order to handle the change in volume over the time step for this type of situation. The 1D Finite Difference scheme handles this better, because the equations are written in terms of change in water surface and velocity, which may be a small change for cross sections that are closer together. Therefore, for the 1D Finite Volume scheme to work well with larger time steps, users may need to remove cross sections that are very close together.

3.The 1D Finite Volume scheme is computationally slower, than the 1D Finite Difference solution scheme. This is due to the fact that the 1D Finite difference scheme combined the properties of the left and right overbank together (area, wetted perimeter, average length between cross sections, etc.) and solved the equations for a main channel and a single floodplain. The new 1D Finite Volume solution scheme keeps the left and right overbank properties completely separate. Thus the equations are written for a separate left overbank, main channel, and right overbank, and then solved. This is more computational work for every time step, but it is computationally more accurate.

2D Flow Options

When selecting the Tab labeled "2D Flow Options" on the Computational Options and Tolerances window, the editor will look like what is shown in Figure 7-24.

| HEC-RAS Unsteady Computation Options and Tolerances | | | | | |
|--|---|--------------------------|--------------------------|--------------------------|--------------------------|
| General 2D Flow Options 1D/2D Options Advanced Time Step Control 1D Mixed Flow Options | | | | | |
| <input type="checkbox"/> Use Coriolis Effects (only when using the momentum equation) | | | | | |
| 1 | Parameter | (Default) | 193 | 194 | LockHaven |
| 1 | Theta (0.6-1.0): | 1 | 1 | 1 | 1 |
| 2 | Theta Warmup (0.6-1.0): | 1 | 1 | 1 | 1 |
| 3 | Water Surface Tolerance [max=0.2](ft) | 0.01 | 0.01 | 0.01 | 0.01 |
| 4 | Volume Tolerance (ft) | 0.01 | 0.01 | 0.01 | 0.01 |
| 5 | Maximum Iterations | 20 | 20 | 20 | 20 |
| 6 | Equation Set | Diffusion Wave | Diffusion Wave | Diffusion Wave | Diffusion Wave |
| 7 | Initial Conditions Time (hrs) | | | | |
| 8 | Initial Conditions Ramp Up Fraction (0-1) | 0.5 | 0.5 | 0.5 | 0.5 |
| 9 | Number of Time Slices (Integer Value) | 1 | 1 | 1 | 1 |
| 10 | Longitudinal Mixing Coefficient | | 0 | 0 | 0 |
| 11 | Transverse Mixing Coefficient | | 0 | 0 | 0 |
| 12 | Smagorinsky Coefficient | | | | |
| 13 | Boundary Condition Volume Check | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| 14 | Latitude for Coriolis (-90 to 90) | | | | |
| 15 | Solver Cores | 8 Cores | 8 Cores | 8 Cores | 8 Cores |
| 16 | Matrix Solver | Pardiso (Direct) | Pardiso (Direct) | Pardiso (Direct) | Pardiso (Direct) |
| 17 | Convergence Tolerance | | | | |
| 18 | Minimum Iterations | | | | |
| 19 | Maximum Iterations | | | | |

Figure 7-24 Computation Options and Tolerances with 2D Flow Area Options shown

As shown in Figure 7-24, there are several computational options and tolerances that can be set for the 2D module. These Options are discussed below.

Use Coriolis Effects: Only used in the Full Momentum Equation

This option allows the user to turn on the effects of the Earth's rotation on the solution (Coriolis Effect). When this option is turned on, the user must enter the latitude of the center of the 2D Flow Area in degrees (this is the field labeled "**Latitude for Coriolis**" in the table). A latitude with a value greater than zero is considered in the northern hemisphere, and a value less than zero is considered in the southern hemisphere.

Theta (0.6 – 1.0): 1.0 (default)

This is the implicit weighting factor that is used to weight spatial derivatives between the current solution time line and the previously solved solution time line. Theta of 1.0 (Default), uses only the currently solved time line for the spatial derivatives. This provides for the most stable solution, but possibly at the loss of some accuracy. Theta of 0.6, provides for the most accurate solution of the equations, but tends to be less stable. In general it has been found that in application of most real world flood runoff types of events, Theta of 1.0, will give about the same answers as Theta of 0.6. However, this should be tested for each model due to site specific geometry and flood propagation, in which it may make a difference in the results.

Theta Warm-up (0.6 – 1.0): 1.0 (default)

This is the value of Theta (see description above) that is used during the model warmup and ramp up periods. This value of Theta is only used if the user has turned on the unsteady flow warm-up option, or the Boundary Condition Ramp up Option for 2D areas.

2D water surface calculation tolerance (ft): 0.01 (default)

This is the 2D water surface solution tolerance for the iteration scheme. If the solution of the equations gives a numerical answer that has less numerical error than the set tolerance, then the solver is done with that time step. If the error is greater than the set tolerance, then the program will iterate to get a better answer. The program will only iterate up to the maximum number of iterations set by the user. The default is set to 0.01 ft based on experience in using the model for a range of applications.

Maximum Number of iterations (0 – 40): 20 (Default)

This is the maximum number of iterations that the solver will use while attempting to solve the equations (in order to get an answer that has a numerical error less than the user specified tolerance at all locations in the 2D computational mesh domain). The default is set to 20. However, the user can change it from 0 to 40. It is not recommended to change this unless you are sure that changing the value will either improve the chances that the model will converge (I.e. increasing the value), or speed up the computations without causing any significant errors.

Equation Set: Diffusion Wave (Default); SWE-ELM (original/faster); and SWE-EM (stricter momentum).

The HEC-RAS two-dimensional computational module has the option of running the following equation sets: **2D Diffusion Wave** equations; **Shallow Water Equations** (SWE-ELM) with a **Eulerian-Langrangian** approach to solving for advection; or a new Shallow Water Equation solver (SWE-EM), that uses an **Eulerian** approach for advection. The new SWE equation solution method is more momentum conservative, but may require smaller time steps and produce longer run times. The default is the 2D Diffusion Wave equation set. In general, most flood applications will work fine with the 2D Diffusion Wave equations. The Diffusion Wave equation set will run faster and is inherently more stable. However, there are definitely applications where the 2D SWE should be used for greater accuracy. The good news is that it easy to try it both ways and compare the answers. It is simply a matter of selecting the equation set you want, and then running it. Create a second Plan file, use the

other equation set, run it, and compare it to the first Plan for your application.

The new SWE solver (SWE-EM) is an explicit solution scheme that is based on a more conservative form of the momentum equation. This solver requires time steps to be selected to ensure the Courant number will be less than 1.0, in general (not always). This solver produces less numerical diffusion than the original SWE solver. However, in general, this new solver is only needed when user's are interested in taking a very close look at changes in water surfaces and velocities at and around hydraulic structures, piers/abutments, and tight contractions and expansions. The original SWE solver is more than adequate for most problems requiring the full momentum equation based solution scheme.

Initial Conditions Ramp up Time (hrs): Default is Blank (not used)

This option can be used to "Ramp up" the water surface from a dry condition to a wet condition within a 2D area (or from a flat water surface if an initial water surface elevation was entered). When external boundary conditions, such as flow and stage hydrographs (or 1D reaches), are connected to a 2D area, the first value of the connected flow or stage may be too high (i.e. a very large flow or a stage much higher than the cell elevation it is attached to). If the model were to start this way, such a high discontinuity could cause a model instability. This option allows the user to specify a time (in hours) to run the computations for the 2D Flow Area, while slowly transitioning the flow boundaries from zero to their initial value, and the stage boundaries from a dry elevation up to their initial wet elevation. The user specifies the total "Initial Conditions Ramp up Time" in this field (10 hours, for example). The user must also specify a fraction of this time for Ramping up the boundary conditions. A value of 0.5 means that 50% of the Initial Conditions time will be used to Ramp Up the boundary conditions to their initial values, the remaining time will be used to hold the boundary conditions constant, but allow the flow to propagate through the 2D Flow Area, thus giving it enough time to stabilize to a good initial condition throughout the entire 2D Flow Area. The Ramp up time for the boundary conditions is entered in the next row, which is labeled "**Boundary Condition Ramp up Fraction**".

Boundary Condition Ramp up Fraction (0 to 1.0): 0.5 (50%) Default value

This field goes along with the previous field "Initial Conditions Ramp up Time". This field is used to enter the fraction of the Initial Conditions Ramp up Time that will be used to ramp up the 2D Flow Area boundary Conditions from zero or dry, to their initial flow or stage. User's can enter a value between 0.0 and 1.0, representing the decimal fraction of the Initial Conditions Ramp up Time.

Number of Time Slices (Integer Value): 1 (Default)

This option allows the user to set a computational time step for a 2D area that is a fraction of the overall Unsteady flow computation interval. For example, if the user has set the Unsteady Flow overall computation interval to 10 minutes, then setting a value of 5 in this field (for a specific 2D area) means that the computation interval for that 2D area will be 1/5 of the overall computation interval, which for this example would be 2 minutes (e.g. $10/5 = 2$). Different values can be set for each 2D Flow Area. The default is 1, which means that 2D Flow Area is using the same computational time step as the overall unsteady flow solution (computation Interval is entered by the user on the unsteady flow analysis window).

Eddy Viscosity Transverse Mixing Coefficient: Default is Blank (not used)

The modeler has the option to include the effects of turbulence in the two dimensional flow field. Turbulence is the transfer of momentum due to the chaotic motion of the fluid particles as water contracts and expands as it moves over the surface and around objects. Turbulence within HEC-RAS is modeled as a gradient diffusion process. In this approach, the diffusion is cast as an Eddy Viscosity coefficient. To turn turbulence modeling on in HEC-RAS, enter a value for the Eddy Viscosity

Transverse or Longitudinal Mixing Coefficient for that specific 2D Flow Area. These coefficients require calibration in order to get at an appropriate value for a given situation. The default in HEC-RAS is zero for this coefficient, meaning it is not used. Additional diffusion using the Eddy Viscosity formulation can be obtained by entering a value greater than zero in this field. HEC-RAS also has an option to add an additional amount of turbulence using the Smagorinsky formulation.

Note: For the details on how to use the HEC-RAS Turbulence modeling, please review the section in the HEC-RAS 2D User's Manual. For information of the equations used for turbulence modeling, please review the section in Chapter 2 of the Hydraulic Reference manual.

Boundary Condition Volume Check: Default is not checked

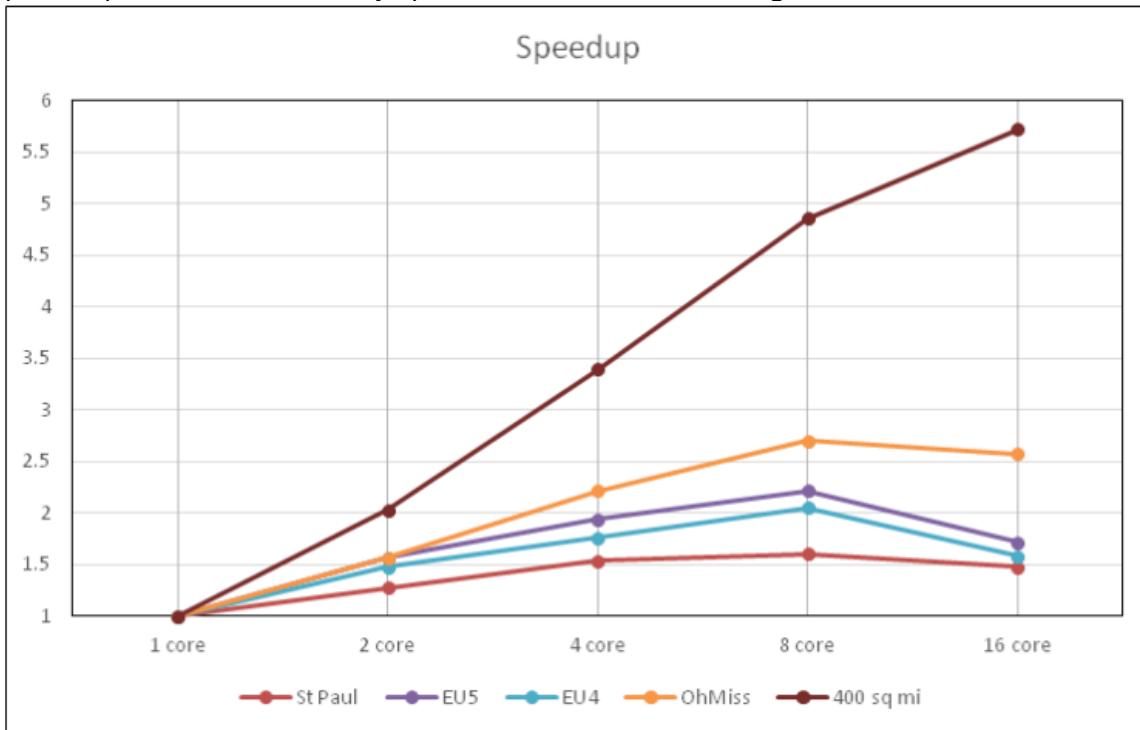
This option is used to keep track of potential volume errors that may arise when 1D connections are made to 2D Flow Areas (i.e. Lateral structure connected from a 1D River to a 2D Flow Area). If this is turned on, the program will evaluate the volume transfer across the hydraulic connection to ensure that the volume passing over the computational time step is not more water than is truly available in the 2D Flow Area. Where this comes into play is when water is leaving a 2D flow area and going into a 1D River Reach over a lateral structure. Flow use computed by a weir equation and is based on the water surface elevations on both sides of the weir. However, there may not be enough volume left in the 2D flow area to satisfy that flow rate over the computational time step. If this option is turned on the program will make a check for this, and if the 2D Flow Area can not send that amount of water out, the software will redo the time step with a lower flow transfer rate. Turning this option on can improve the accuracy of these types of computations, but it will also slow down the computations.

Number of Cores to use in computations: All Available (Default)

The HEC-RAS two-dimensional computational module was developed from the ground up with parallel processing in mind. The HEC-RAS 2D computations will use as many CPU cores as there are available on your machine (which is the default mode for running). However, HEC-RAS provides the option to set the number of cores to use for the 2D computations. In general, it is recommended to use the default of "All Available". However, you may want to experiment with this for a specific data set to see if it will either speed up or slow down computations based on a specific number of cores. The ideal number of cores for a given problem is size and shape dependent (shape of the 2D Flow Area). As you use more cores, the problem is split into smaller pieces, but there is overhead in the communications between the pieces. So, it is not necessarily true that a given problem will always run faster with more cores. Smaller data sets (2D areas with fewer cells) may actually run faster with fewer cores. Large data sets (2D Areas with lots of cells) will almost always run faster with more cores, so use all that is available.

Shown below are the results of testing a few data sets by running them with different numbers of Cores. Each model was run several times with the number of cores set to: 1, 2, 4, 8, and 16. As you can see four of the data sets had speed improvements up to 8 cores, but actually ran slower with 16 cores. These are smaller data sets ranging from 10,000 to 80,000 cells. However, one data sets had

speed improvements all the way up to 16 cores. This was the largest data set, with 250,000 cells.



Matrix Solver. By default HEC-RAS uses a matrix solver for the 2D equations called "Pardiso". This is a direct solver. For HEC-RAS 6.0 and newer some additional "Iterative" solvers have been added. In general, the Pardiso solver produces a more stable solution. The new Iterative solvers "may" produce a faster solution (Not always), but run the risk of increasing model instability. In general user's should get their model up and running with the Pardiso matrix solver, then they can try the iterative solvers to see if they get any performance increases with their data set.

1D/2D Options

When selecting the Tab labeled "1D/2D Options" on the Computational Options and Tolerances window, the editor will look like what is shown in Figure 7-25.

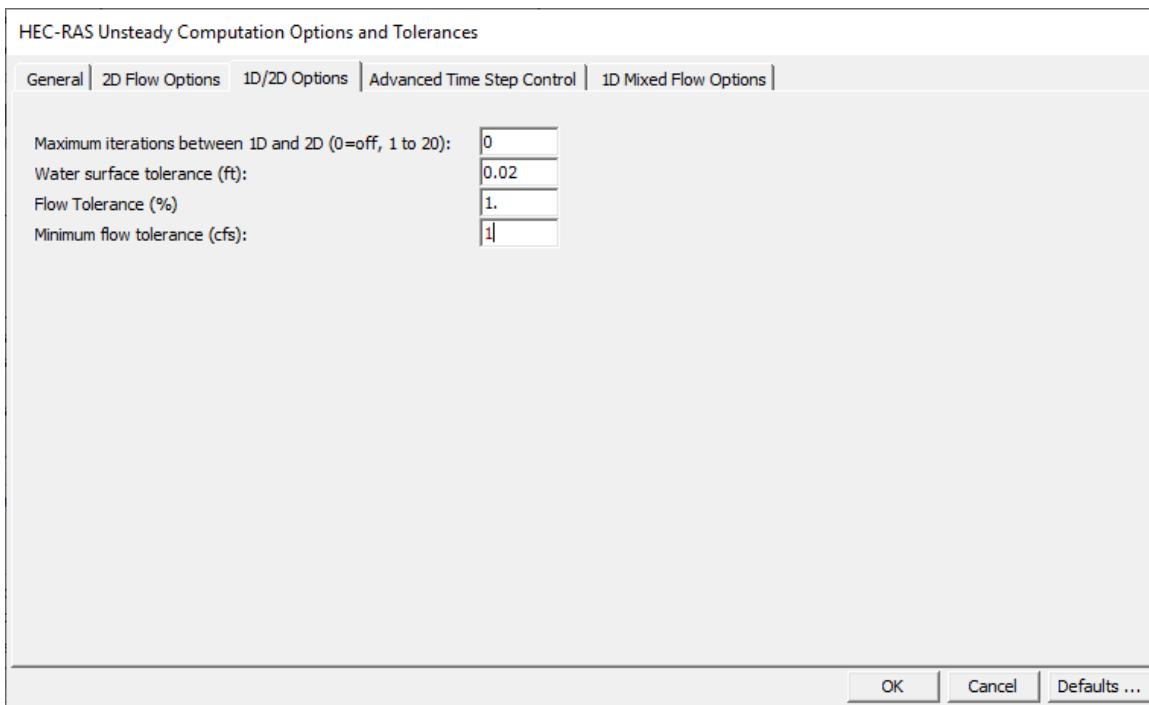


Figure 7.25 Calculation Options and Tolerances with 1D/2D Options tab shown

Maximum iterations between 1D and 2D. Default is zero (meaning this is not turned on)

There are also some options for Controlling 1D/2D Iterations, which can be used to improve the computations of flow passing from a 1D element (reach or storage area) to a 2D Flow Area. By default this option is turned off, and most 1D to 2D connections will not need iterations. However, when the 1D/2D hydraulic conditions become highly submerged, or there are flow reversals, or tidally influence stages/flows, then iterating between the 1D solution and 2D solution may be necessary to get an accurate and stable answer. To turn on the 1D/2D iterations option, select the "**1D/2D Options**" tab. Then you can set the **Maximum iterations between 1D and 2D**, as well as tolerances for controlling the convergence criteria. Iteration can be set from 0 to 20, with zero meaning that it does not do any extra iterations (this is the default). In general, only use this option if you are having a stability problem at a 1D/2D hydraulic connection. Set the number of 1D/2D Iterations to as low as possible in order to get a stable answer between a 1D and 2D connection that is having stability problems. The Number of 1D/2D Iterations will cause the entire solution to be done multiple times for each time step in order to get the desired convergence. **This could dramatically lengthen run times.** If you turn this option on, it is suggested that you start with a low value, like 3 or so. If the stability problem still exists with that number of iterations, then increase it from there.

The convergence criteria for 1D/2D iterations consists of a **Water Surface Tolerance, Flow Tolerance (%)**, and a **Minimum Flow Tolerance**. The water surface tolerance is currently only used when an upstream 1D reach is connected to a downstream 2D Flow Area. In this situation, the 1D region is computed, then the 2D region. The assumed water surface elevation at the boundary is re-evaluated. If the water surface has changed more than the **Water Surface Tolerance**, then the program will iterate. When the water surface elevation at the boundary has change less than the tolerance, the solution stops iterating and moves on to the next time step.

The **Flow Tolerance (%)** gets used for the following 1D/2D connections: Lateral Structure; SA/2D Hydraulic Connection (SA to 2D, or 2D to 2D); and 2D Flow Area to 1D Reach connection (Currently in

the 5.0 Beta version, this only works when an upstream 2D flow Area is connected to a downstream 1D river reach). The default value for the Flow Tolerance (%) is 0.1 %. If 1D/2D iterations are turned on, then the flow between these types of 1D/2D connections gets recomputed after each trial to see if it has changed more than the user defined Flow Tolerance (%). If it has changed more than the flow tolerance, then the program iterates. A companion tolerance to the Flow Tolerance, is the **Minimum Flow Tolerance (cfs)**. The purpose of this tolerance is to prevent the program from iterating when the flow passed between a 1D and 2D element is very small, and not significant to the solution. For example, you may have a connection from a 1D reach to a 2D Flow Area via a Lateral Structure, in which the flow under certain conditions is very low, so the actual change in the flow from one iteration to the next could be very small (put the percent error is very high). Such a small flow may have no significance to the solution, so iterating the entire solution to improve this small flow between the 1D and 2D elements makes no sense, and may be just wasting computational time. In general it is a good idea to set a minimum flow when turning on 1D/2D iterations. The default value is 1 cfs, however, this is most likely model specific.

Advanced Time Step Control Tab

Some new variable time step capabilities have been added to the unsteady flow engine for both 1-dimensional (1D) and 2D unsteady flow modeling. Two new options are available. One is a variable time step based on monitoring Courant numbers (or residence time within a cell), while the other method allows users to define a table of dates and time step divisors. The variable time step option can be used to improve model stability, as well as reduce computational time (not all models will be faster with the use of the variable time step). The information contained in this section is supplemental to Chapter 8 of the HEC-RAS 5.0 User's Manual.

These new Variable Time Step options are available from the Unsteady Flow Analysis window and also by going to the Computational Options and Tolerances window. There is a new button right next to the Computation Interval directly on the Unsteady Flow Analysis window, which will take users to this new feature.

When the Variable Time Step control editor ellipse button is clicked, the modified HEC-RAS Unsteady Computation Options and Tolerances window opens to the new Advanced Time Step Control tab (Figure 7-26). Alternatively, users can open the window and navigate to the new tab, from the Options | Computation Options and Tolerances menu option.

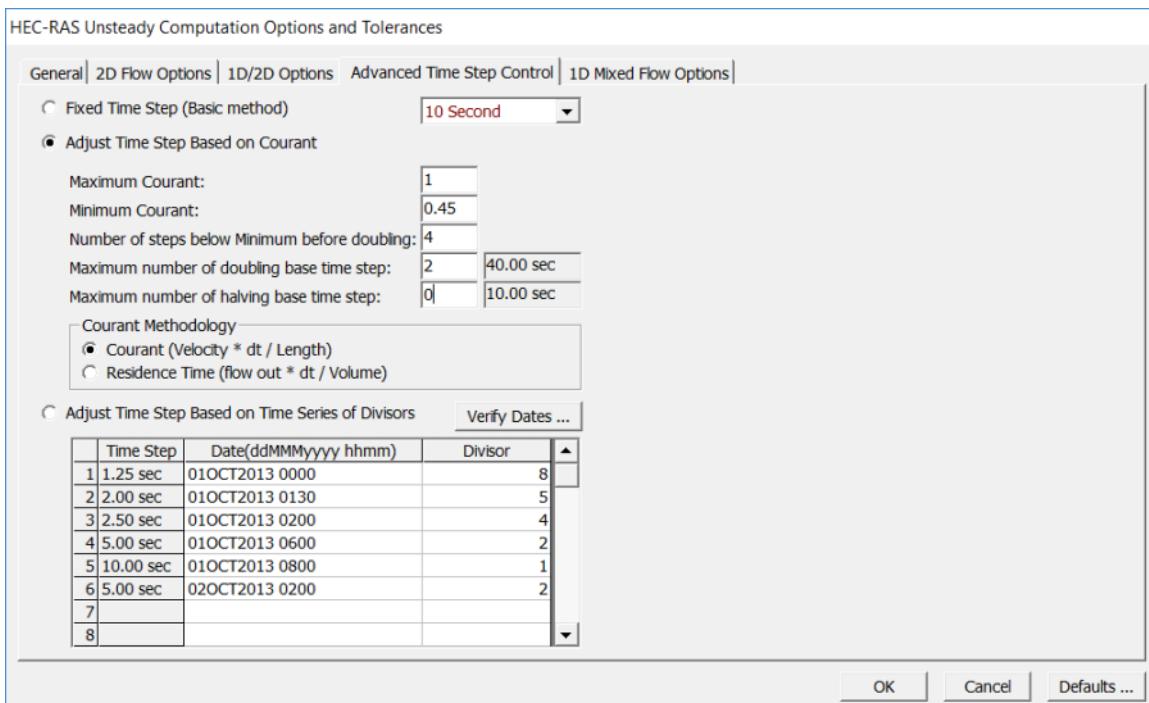


Figure 7 26. Variable Time Step Editor within the Computational Options and Tolerances.

As shown in Figure 7-26, the new Advanced Time Step Control tab now has three different methods for selecting and controlling the computational time step: (1) Fixed Time Step (default); (2) Adjust Time Step Based on Courant which is a variable time step based on the Courant number; and (3) Adjust Time Step Based on Time Series of Divisors which is a variable time step based on a user entered table of dates, times, and time step divisor's. The two new variable time step options (Courant number and Time Step Divisor) are discussed in the following sections.

Courant Number Method

The first new variable time step option is to use the Courant number method, from the new Advanced Time Step Control tab. In the example shown in Figure 7-26, the Courant number is being used for the variable time step. To use this method, select the Adjust Time Step Based on Courant option and provide the following information:

- **Maximum Courant:** This is the maximum Courant number allowed at any 2D cell or 1D cross section. If the maximum Courant number is exceeded, then the time step is cut in half for the very next time interval. Because HEC-RAS uses an implicit solution scheme, Courant numbers can be greater than one, and still maintain a stable and accurate solution. In general, if the flood wave is rising and falling slowly (depth and velocity are changing slowly), the model can handle extremely high Courant numbers. For these types of cases, users may be able to enter a Maximum Courant number as high as 5.0 or more. However, if the flood wave is very rapidly changing (depth and velocity are changing very quickly over time), then the Maximum Courant number will need to be set closer to 1.0. The example shown in Figure 2 6 is for a Dam break type of flood wave, in which depth and velocity are changing extremely rapidly. Because of the rapid changes in depth and velocity, the Maximum Courant number was set to 1.0.
- **Minimum Courant:** This is the minimum Courant number threshold for 2D cells and 1D cross sections. If the Courant number at "all" locations goes below the minimum, then the time step will be doubled. However, the time step will only be doubled if the current time step has been used for enough time steps in a row to satisfy the user entered criteria called "Number of steps below

"Minimum before doubling" (see below for an explanation of this field). The "Minimum Courant" value should always be less than half of the "Maximum Courant" value. If the Minimum Courant value is equal to or larger than half the Maximum Courant value, the HEC-RAS Version 5.0.4 software will just flip back and forth between halving and doubling the time steps. In the example shown in Figure 2-6, since the Maximum Courant number was set to 1.0, the minimum was set to 0.45 (less than half of the maximum), which allowed the model to stay stable, but also run faster.

- *Number of steps below Minimum before doubling:* This field is used to enter the integer number of time steps in which the Courant number must be below the user specified minimum before the time step can be increased. This can prevent the model from increasing the time step too quickly and/or from flipping back and forth between time steps. Typical values for this field may be in the range of 5 to 10.
- *Maximum number of doubling base time step:* This field is used to enter the maximum number of times the base time step can be doubled. For example, if the base computation interval is 10 seconds, and the user wants to allow it to go up to 40 seconds, then the value for this field would be 2 (i.e., the time step can be doubled twice: 10s to 20s to 40s). The value displayed in the box to the right of the user entered value is what the entered maximum time step will end up being.

NOTE: The HEC-RAS software requires that all time steps end up exactly hitting the Mapping Output Interval. This requirement is because output for HEC-RAS Mapper must be written to the output file for all cross sections, storage areas, and 2D cells at the mapping interval. Because of this fact, if users enter a "Maximum number of doubling base time step" that results in a computation interval that does not exactly land on the mapping output interval, then the unsteady flow computational program will compute its own time steps that work with the parameters entered in the Adjust Time Step Based on Courant section. Furthermore, the base time step will be changed to something close to what was entered, but when doubling it, all values will still line up with the mapping output interval. When the model runs it will list what time step it is currently using in the message window of the computational output window.

- *Maximum Number of halving base time step:* This field is used to enter the maximum number of time that the base computation interval can be cut in half. For example, if the base computation interval is 10 seconds, and the user wants to allow it to go down to 2.5 seconds, then the maximum number of halving value would need to be set to 2 (i.e., the time step can be cut in half twice: 10s to 5s to 2.5s). The value displayed in the box to the right of the user entered value is what the entered maximum time step will end up being.

For the Courant number method, the default approach for computing the Courant number is to take the velocity times the time step divided by the length (between 1D cross sections, or between two 2D cells). For 2D, the velocity is taken from each face and the length is the distance between the two cell centers across that face. For 1D, the velocity is taken as the average velocity from the main channel at the cross section, and the length is the distance between that cross section and the next cross section downstream.

An optional approach to using a traditional Courant number method is to use Residence Time. With this method, the HEC-RAS software is computing how much flow is leaving a 2D cell over the time step, divided by the volume in the cell. The Residence Time method is only applied to 2D cells. When this method is turned on, it is used for the 2D cells, but 1D cross sections still use the traditional Courant number approach.

User Defined Dates/Time vs Time Step Divisor

Another option available from the Advanced Time Step Control tab, is to set the variable time step control based on a user defined table of dates and times versus a time step divisor (Figure 7-26).

As shown in Figure 7-26 the user can select the option called "Adjust Time Step Based on Time Series of Divisors", from the Advanced Time Step Control tab. When this option is selected the user must

enter a table of Dates and times verses time step Divisors. The first date/time in the table must be equal to the starting date/time of the simulation period. To use this method, enter a base time step equal to the maximum time step desired during the run. Then in the table, under the Divisor column, enter the integer number to divide that time step by for the current date/time in the table. Once a time step is set for a date/time, the Unsteady Flow Analysis compute will use that time step until the user sets a new one.

In the example shown in Figure 7-26, the base computational interval was set at 1 minute. Based on the table of dates/times and Divisors entered, the actual time steps that will be used are displayed in the first column labelled "Time Step".

The Time Step Divisor method for controlling the time step requires much more knowledge by the user about the events being modelled, the system being routed through, as well as knowledge of velocities, cross section spacing, and 2D cell sizes. However, if done correctly, this method can be a very powerful tool for decreasing model run times and improving accuracy.

1D Mixed Flow Options.

When this option is selected, the program will run in a mode such that it will allow the 1D Finite Difference solution scheme to handle 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 can increase 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 14 of the HEC-RAS User's manual.

Output Options. This option allows the user to set some additional output flags. When this option is selected a window will appear as shown in Figure 7-27. The following is a list of the available options:

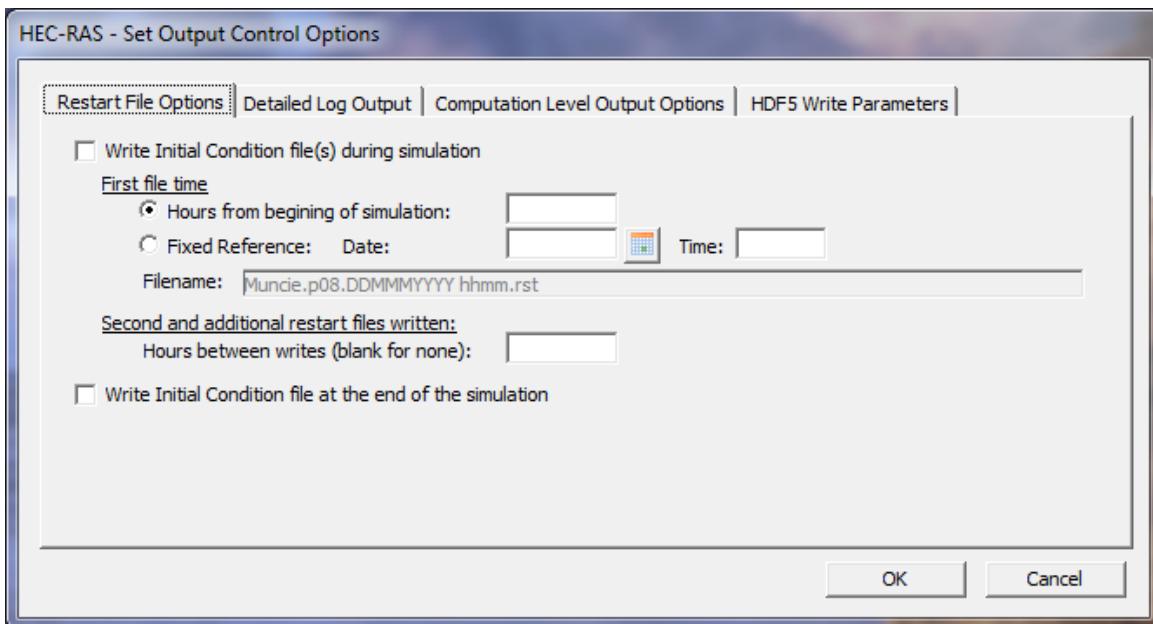


Figure 7.27. Unsteady Flow Output Control Options Window

Restart File Options: This tab allows the user to write out an Initial conditions file or files, sometimes called 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 can either enter a time in hours from the beginning of the current simulation, or they can enter a specific Date and Time. This time represents the time at which the conditions of the system will be written to the "Hot Start" file. The program writes flow and stage at all of the computational nodes, as well as the stage in all of the storage areas to the file. An additional option is available to write multiple restart files out from a single run. This is accomplished by first specifying how and when you want the first file to be written, then entering the number of hours between subsequent writes of the file. The last option of this section allows the user to ensure that the very last time step also gets written out as a separate restart file. Filenames for restart files are labeled as follows:

Projectname.p##.DDMMYYYY hhmm.rst
p##=plan number
DD=Day
MM=Month
YY=Year
hh=hour
mm=minute
rst=file extension

Detailed Log Output: This tab allows the user to turn on detailed output that is written to a log file. The user can have the Input Hydrographs written to the log file; the final computed hydrographs; and they also can have the software write detailed information for each iteration of the unsteady flow equations (**Write Detailed Log Output for Debugging**). The detailed output at the iteration level can be written for the entire simulation period, or 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. There is an additional option to turn on this detailed log output only when a certain number of iterations has been met or exceeded (**Automatic Detailed Log Output**).

Computation Level Output: This tab allows the user to write out a limited list of variables at the computational time step level. This is a very useful tool for assisting in debugging an unsteady flow model. It is often very helpful to seem a few output variables, like water surface and flow, at the detailed computational time step in order to see when and where a model is going unstable. By default, only the water surface and the flow rate will be written out when the computational output option is turned on from the Unsteady Flow Computational window. This option allows the user to select additional variables to be written to that file. The following variables are available to be written tot the computational level output file:

WSEL=Water surface elevation (Default)

Flow=Flow rate (Default)

WS Error=Numerical error in water surface comp.

Flow Error=Numerical error in computed flow

Depth =Depth of the water from channel invert

Invert=Elevation of main channel invert

Vel Channel=Average velocity in main channel

Vel Total=Average velocity of entire cross section

Courant Chan=Courant Number for main channel only

Courant Total=Courant Number for entire cross section

Diff Eqn Parts =Separate components of the unsteady flow equations (Momentum and Continuity equations)

Friction Slope Method for Cross Sections. By default the program uses the Average Friction Slope method for determining friction forces for the momentum equation during an unsteady flow run. This option allows the user to select one of the other five available methods in HEC-RAS. To learn more about the friction slope averaging techniques in HEC-RAS, see chapter 2 of the hydraulic reference manual.

Friction Slope Method for Bridges. By default the program uses the Average Conveyance friction slope averaging technique for computing frictional forces through bridges. This has been found to give the best results at bridge locations. This option allows the user to select one of the other five available methods.

Initial Backwater Flow Optimizations. If your model has a flow split, lateral structure, or pump stations, it may be necessary to optimize the flow splits during the initial backwater computations in order to get a reasonable initial condition for the unsteady flow computations. This option allows the user to turn on flow optimizations at the various locations where flow may be leaving the system and it is a function of the water surface elevation (which would require optimization to get the right values).

Sediment Computational Options and Tolerances. As of HEC-RAS version 5.0 and newer, there is now the ability to perform Unsteady Flow Sediment Transport (deposition and erosion) computational capabilities. This Option, and the next two (called: Sediment Output Options and Sediment Dredging Options), are used to control the sediment transport computations within the

Unsteady Flow computational program. Please refer to Chapter 17 for the details of performing sediment transport computations within HEC-RAS.

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. The user can turn this option off if they feel the software is erroneously stopping the computations from running. If this does happen the user should report this as a bug to the HEC-RAS development team.

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.

View Runtime Messages. This option allows the user to bring up a viewer that will show the user all of the messages that were written to the computational window for the last time the model was computed.

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 7-28). The information that appears in the window is an indicator of the programs progress during the computations, and a list of any computational messages that come up during the run. 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 view the computational messages to see what problems may have occurred, then begin debugging the problem.

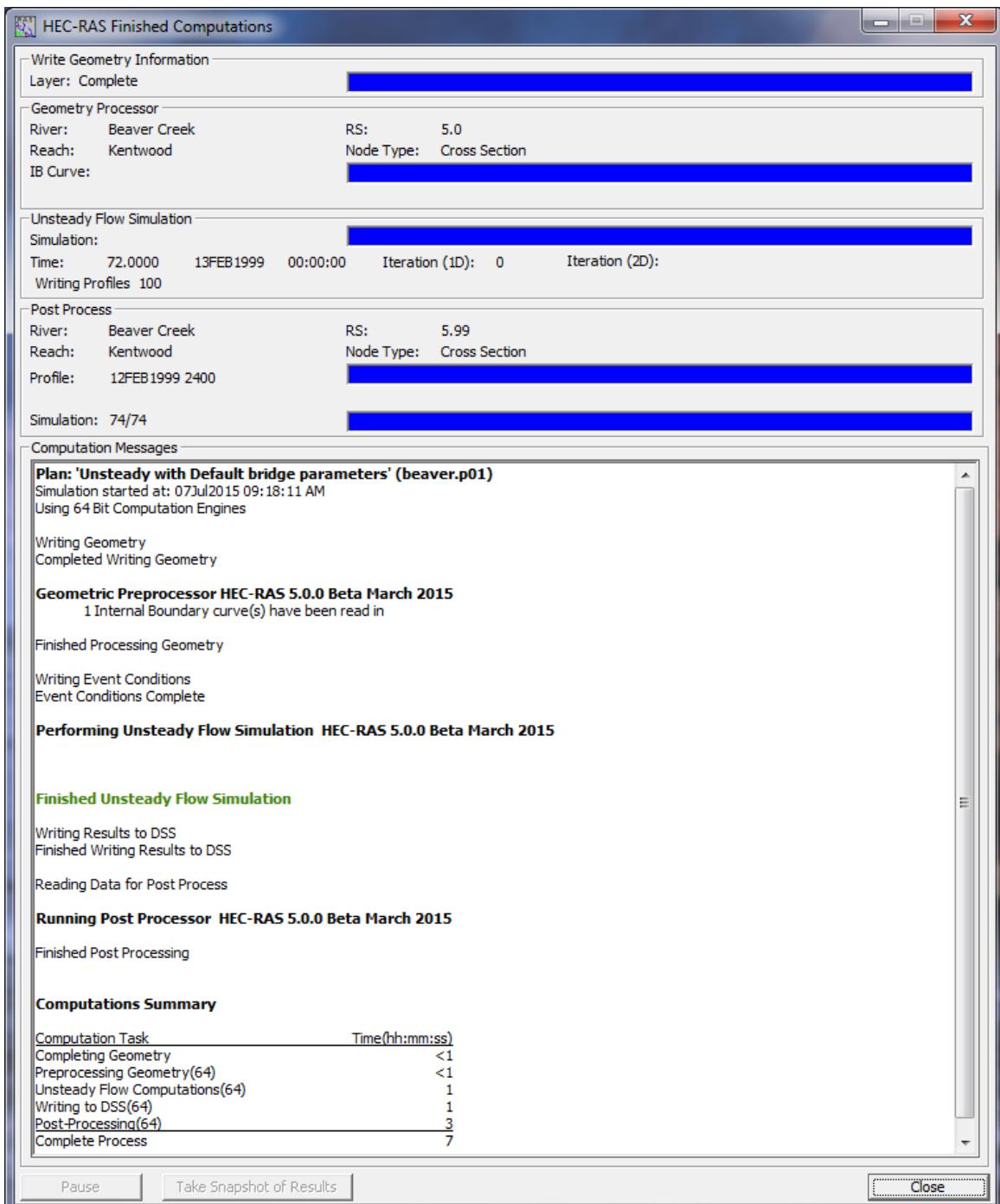


Figure 7.28. Unsteady Flow Computations Window

Calibration of Unsteady Flow Models

Calibration is the adjustment of a model's parameters, such as roughness and hydraulic structure coefficients, 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 (cooperative stream gage program).
4. There is an error in the datum of the gage.
5. Subsidence over time causes errors in the stage measurement.

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 $\pm 5\%$ of the true value. Some believe that this assumed error is optimistic. In any case, $\pm 5\%$ on many river systems, translates into a stage error of ± 1.0 feet. 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 7-29 below is a comparison between high water marks and the computed maximum water surface profile. Note the scatter in the high water marks, which mark is accurate?

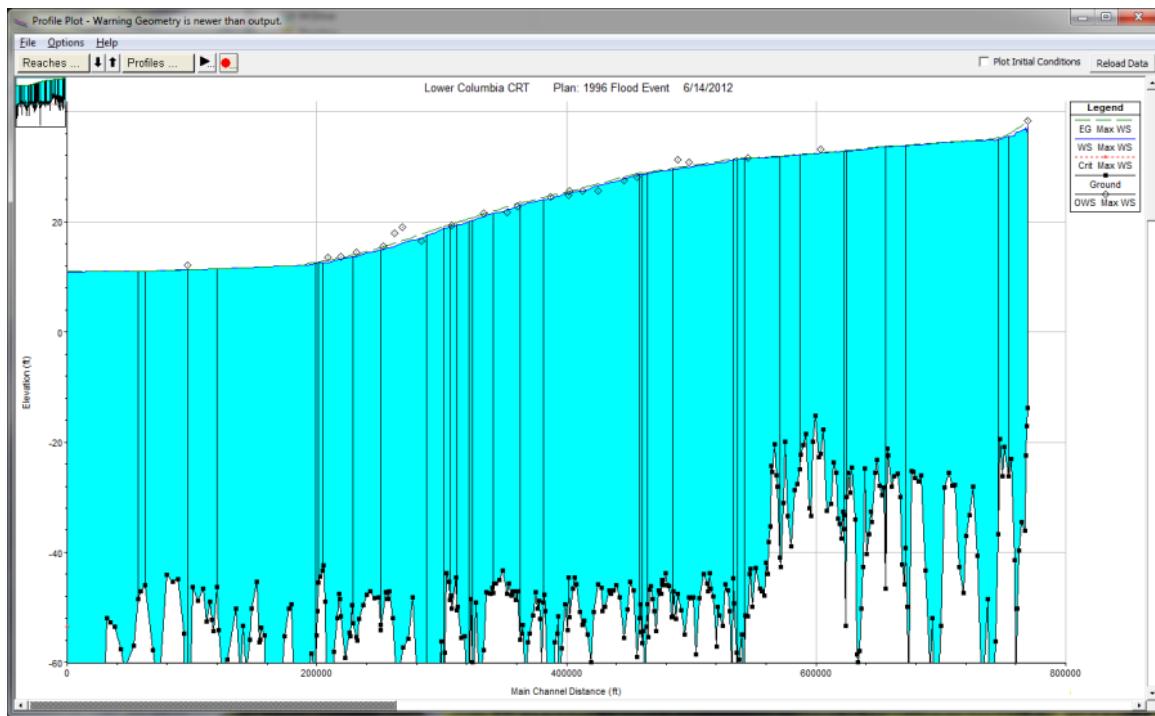


Figure 7.29. Computed Water Surface Profile Versus Observed High Water Marks

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 7.30. Example Drainage Area Accounting for Red River of the North

As shown in Figure 7-30, ungaged areas can be accounted for by using a pattern hydrograph of a hydrologically 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 across 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 their 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 7-31). However, if the banks of a river are rougher than the channel bottom (due 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.

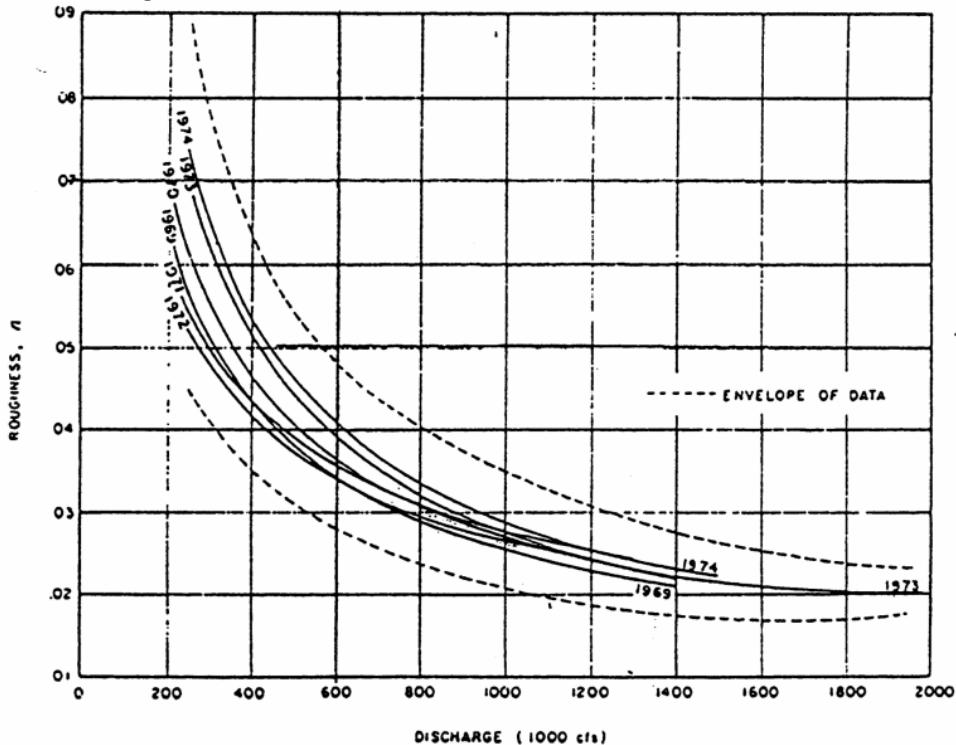


Figure 7.31. Roughness Versus Discharge for the Mississippi River at Arkansas City

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 geometry or bed forms; 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 7-32 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.

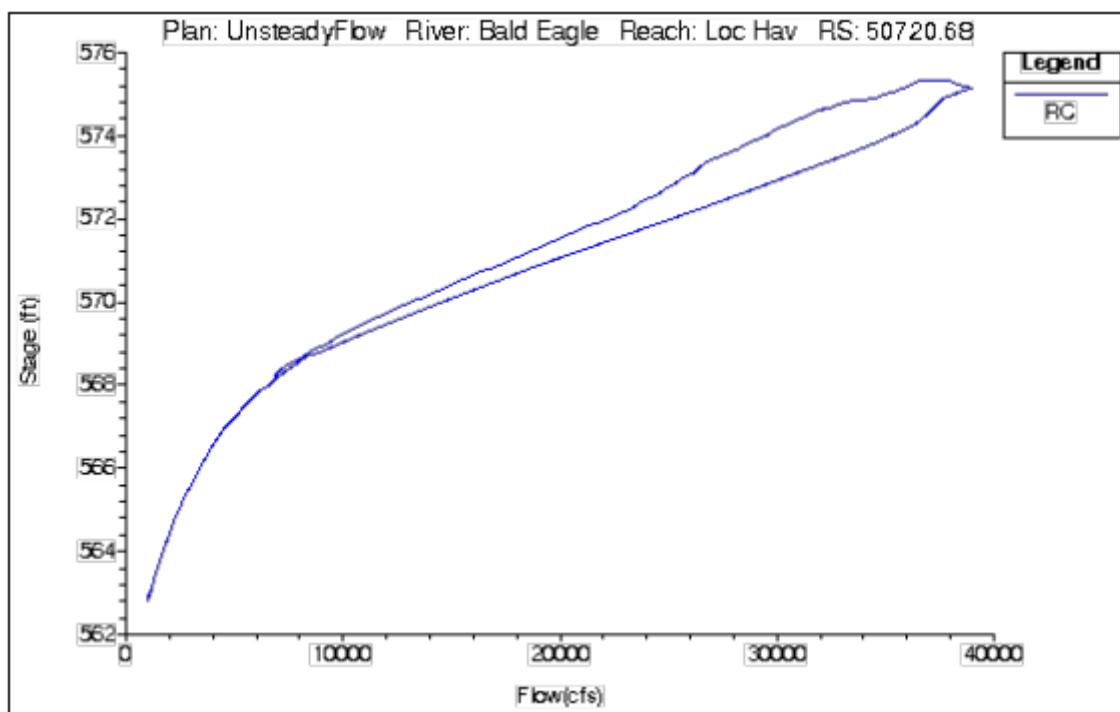


Figure 7.32. Looped Rating Curve Example

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 7-33 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.

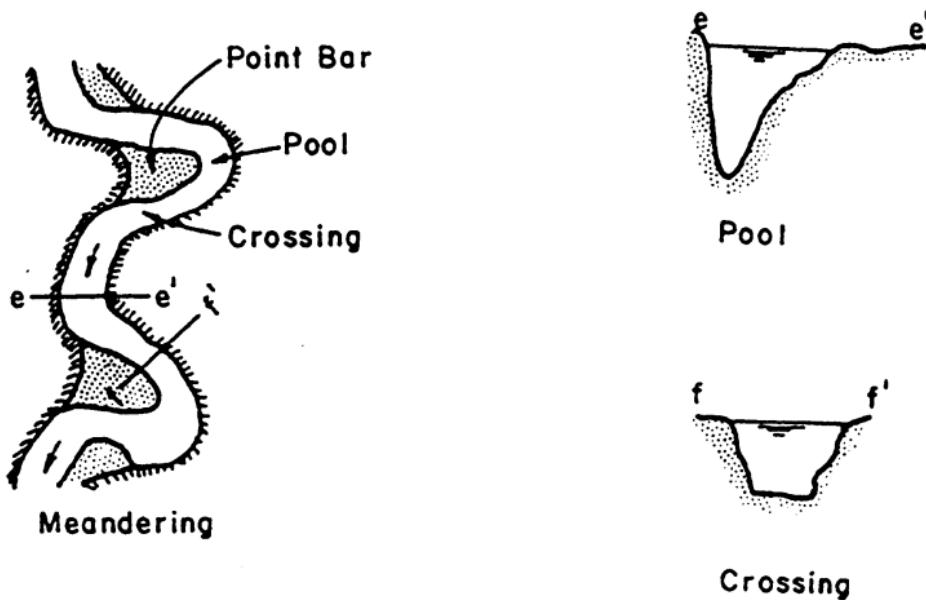


Figure 7.33. Morphology of a Meandering River

An invert profile for the Mississippi River is shown in the Figure 7-34. Note the pools and crossings. The water surface profile is controlled by the crossing cross-sections (high points in the invert), 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.

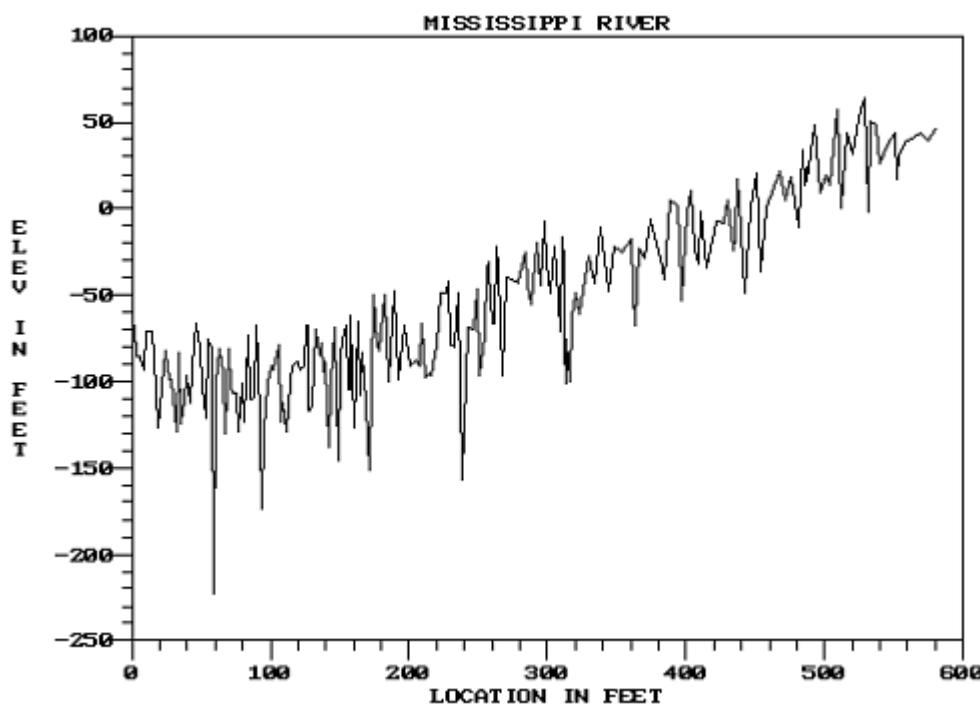


Figure 7.34. Invert Profile for Lower Mississippi River

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 7-35.

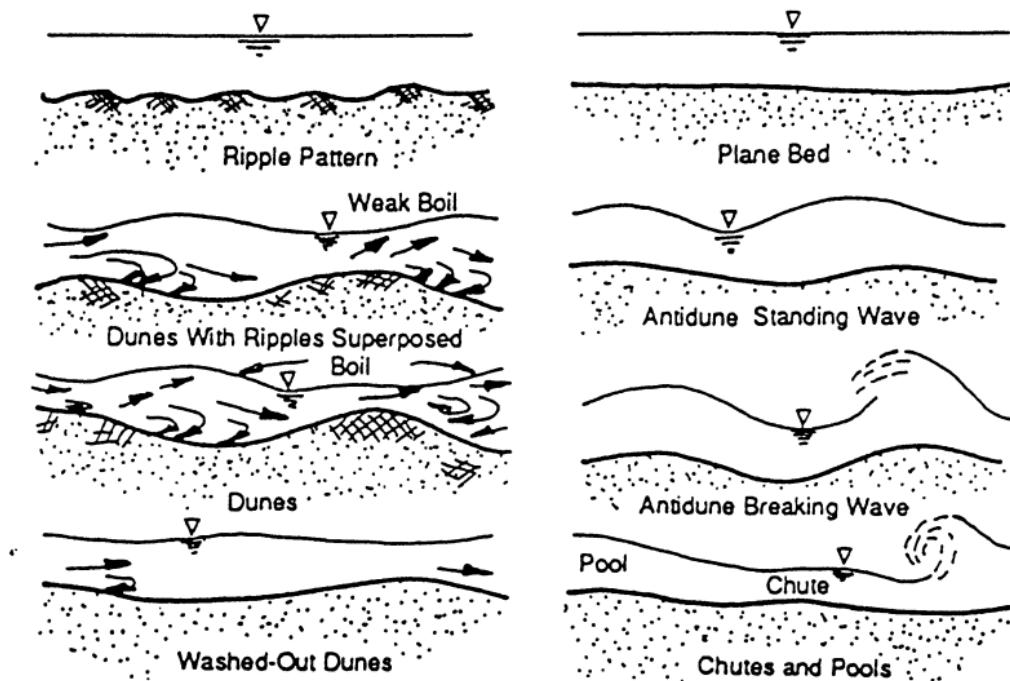


Figure 7-35. Transitions of Bed Forms in an Alluvial Stream

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 7-36 shows the seasonal shift for the Mississippi River at St. Louis.

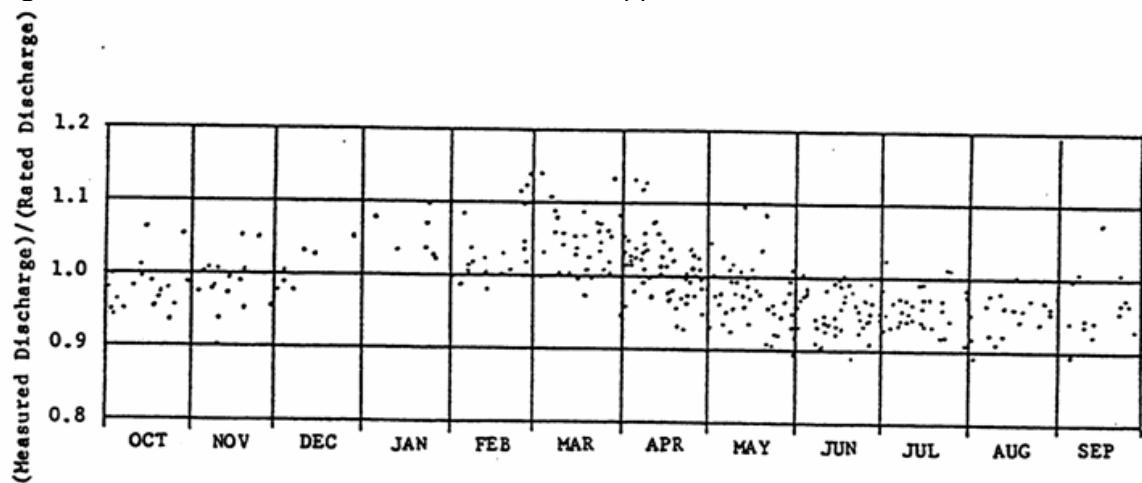


Figure 7-36. Changes in Roughness due to Temperature in the Mississippi River

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 7-37. 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.

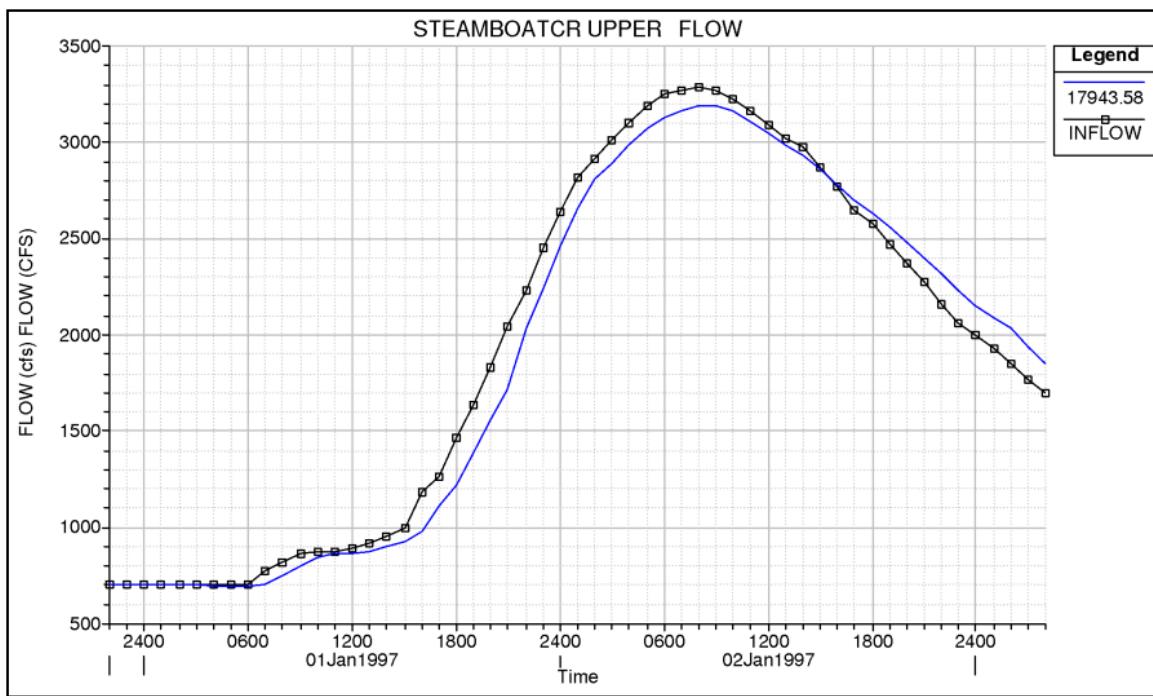


Figure 7.37. Example of the Effects of Overbank Storage

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 than 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 7-38 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 (possibly drained by culverts to a downstream location).

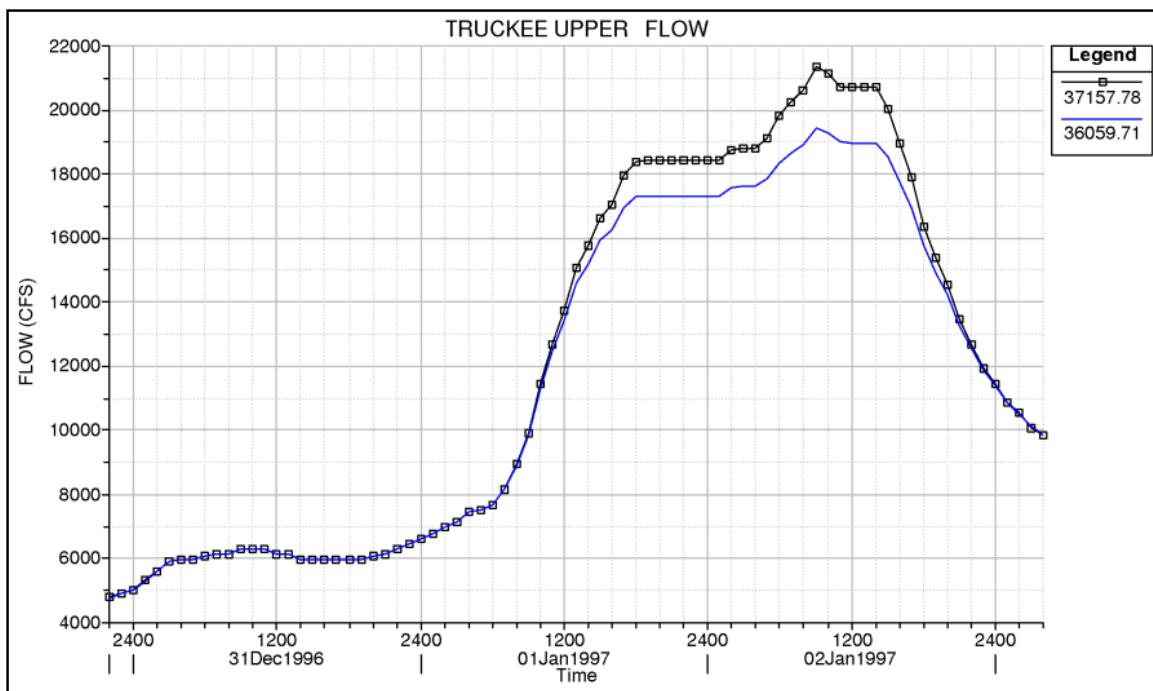


Figure 7.38. Example Effects of Off-line Storage

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 and the steepness of the stream). However, in flat streams, increases in a water surface at a structure can cause a backwater upstream for a substantial distance (depends on amount of stage increase and slope of the stream). 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.

Tools Available in HEC-RAS to aid in Model Calibration

The following is a list of tools that the user should be aware of and use frequently during any calibration of an HEC-RAS model:

1. Manning's n Value Tables (Tables for other parameters).
2. Flow versus Roughness Factors options.
3. Graphical Plots with Observed data options.
 - Profile Plots
 - Cross section Plot

- Hydrograph (Stage and Flow) Plots
- 4. Tabular Output Tables.

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 (if possible), 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 encompasses the full range of flows from low to high and back to low flow.
3. Adjust cross section storage (ineffective areas) 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 7-39.

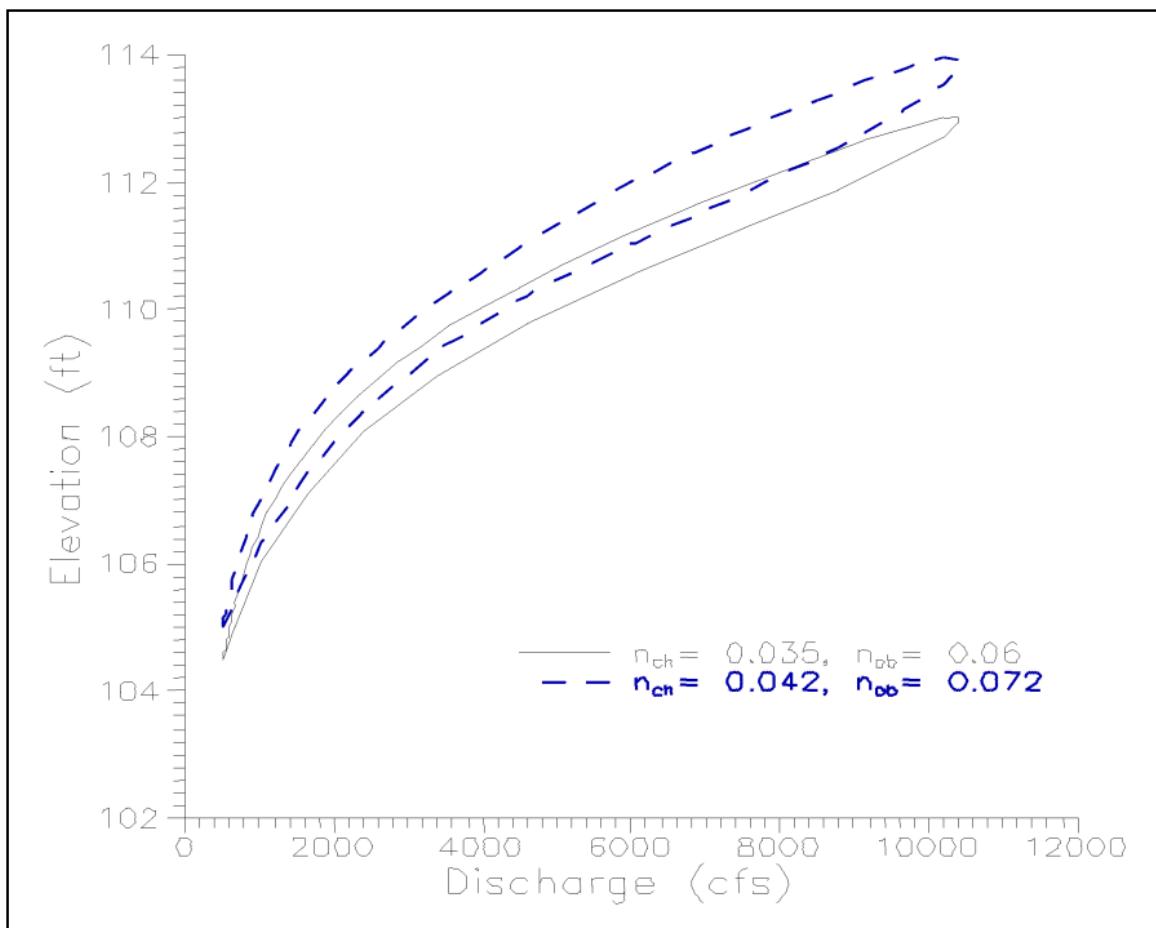


Figure 7.39. Example of Wider Loop for Higher Manning's n

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. You may be able to get a single event to calibrate well with parameters that are outside of the range that would be considered normal for that stream, but the model may not work well on a range of events. Stay within a realistic range for model parameters. If the model is still not calibrating well, then there must be other reasons why.
3. If using a single-valued rating curve at the downstream boundary, move it far enough downstream so it doesn't affect the results in the study reach.

4. Discrepancies may arise from a lack of quality cross-section data. If you are using cross sections cut from a 10 meter DEM, then you should not expect to be able to get a good model calibration with such poor terrain data.
5. The volume of off-channel storage areas is often underestimated, which results in a flood wave that travels to fast and will generally have to high of a peak downstream. Try to closely evaluate all of the areas that water can go and include them in the model.
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 to get a correctly computed water surface elevation.
7. Calibration should be based on floods that encompass a wide range of flows, low to high. Be careful, to low of a flow can cause an unsteady flow model to go unstable. This is general caused by flow passing through critical depth between pools and riffles.
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. Also, setting Theta towards a value of 0.6 will often help with the numerical accuracy in tidal situations.
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, single water surface across each cross section, etc...).
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 cannot continue. This is a common problem when working with an unsteady flow model of any size or complexity. Figure 7-40 is an example of a model that ran all the way through, but produced an

unstable solution.

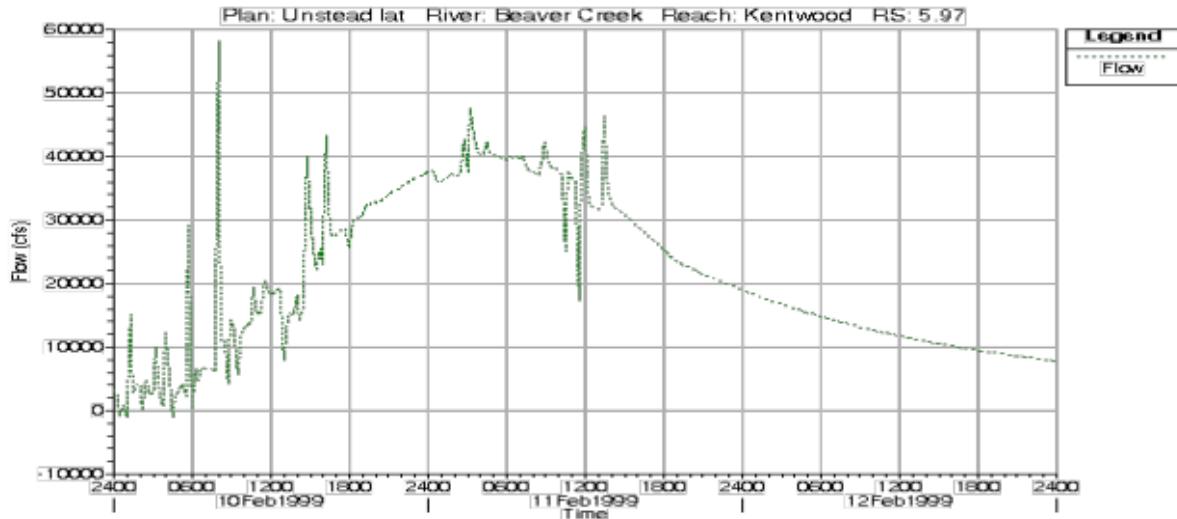


Figure 7.40. Hydrograph from an unstable solution.

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. Calculation Options and Tolerances.
5. Lateral Structures/Weirs
6. Steep streams/mixed flow regime
7. Downstream Boundary Conditions
8. Cross section geometry and table properties
9. Bridges and Culvert crossings
10. Initial/low flow conditions
11. Drops in bed profile.
12. Manning's n values
13. Missing or bad main channel data

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 5000 ft or more. However, most streams lie some where in between these two spacing distances.

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 general, cross sections spaced too far apart will cause additional numerical diffusion, due to the derivatives with respect to distance being averaged over to long of a distance. Also, if the distance between cross sections is so great, such that the Courant number would be much smaller than 1.0, then the model may also become unstable. An example of varying cross section spacing is shown in Figure 7-41. Figure 7-41 shows an inflow

hydrograph (dashed green line) and two outflow hydrographs (solid blue and black line with squares). As shown in the figure, as cross section spacing is increased, the hydrograph will show some numerical attenuation/diffusion.

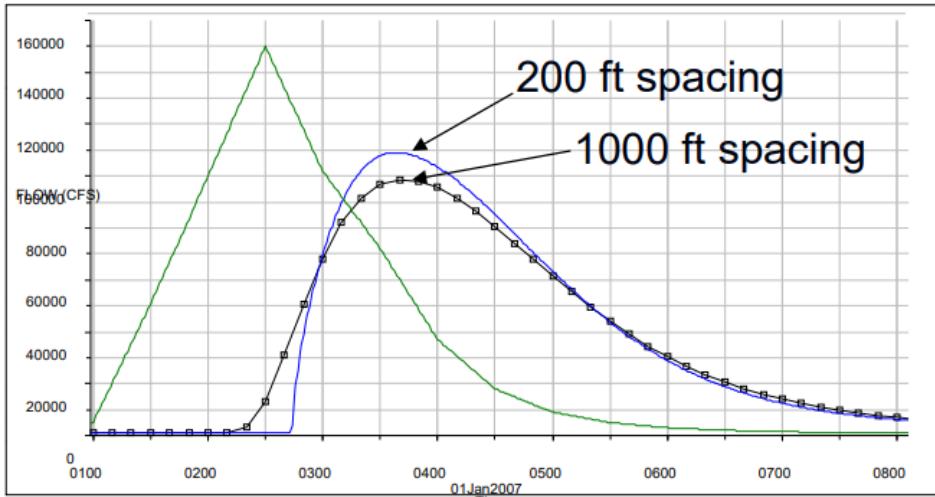


Figure 7-41. The affects of cross section spacing on the hydrograph.

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.

Cross Sections too Close. If the cross sections are too close together, then the derivatives with respect to distance may be overestimated (computed as steeper slopes than they should be), especially on the rising side of the flood wave. This can cause the leading edge of the flood wave to over steepen, to the point at which the model may become unstable. Figure 7-42 is an example where cross sections were placed very close together, and a very dynamic hydrograph was run through the river reach. The leading edge of the flood wave over steepened, and caused the model to produce an unstable result, which appears as a wall of water building just upstream of the flow going through critical depth. The solution to this problem is to remove some cross sections, which will allow the model to do a better job at computing the special derivatives.

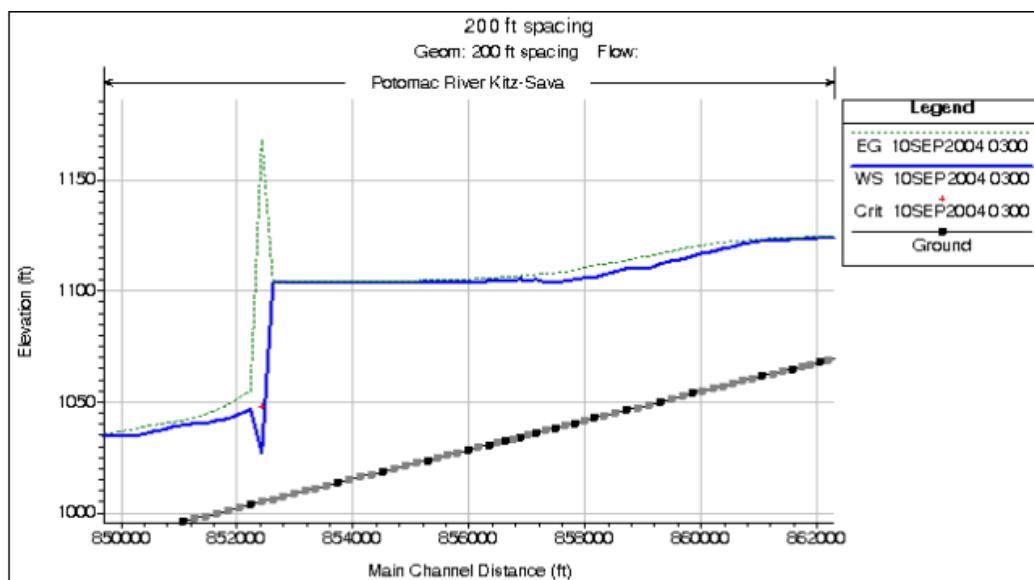


Figure 7.42.

Stability problem from cross sections spaced to close together.

One of the first steps in stabilizing an unsteady flow model is to apply the correct cross section spacing. Dr. Danny Fread equation and P.G. Samuel's have developed equations for predicting maximum cross section spacing. These two equations are good starting points for estimating cross section spacing. Dr. Fread's equation is as follows:

$$\Delta x \leq \frac{c T_r}{20}$$

Where: X=Cross section spacing (ft)

T_r =Time of rise of the main flood wave (seconds)

C=Wave speed of the flood wave (ft/s)

Samuel's equation is as follows:

$$\Delta x \leq \frac{0.15 D}{S_0}$$

Where 😊=Average bank full depth of the main channel (ft)

S_0 =Average bed slope (ft/ft)

Samuels equation is a little easier to use since you only have to estimate the average bank full depth and slope. For Fread's equation, although the time of rise of the hydrograph (T_r) is easy enough to determine, the wave speed (c) is a little more difficult to come by. At areas of extreme contraction and expansion, at grade breaks, or in abnormally steep reaches, inserting more cross sections may be necessary.

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

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

Therefore:

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

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

C_r = Courant Number. A value = 1.0 is optimal.

Δx =Distance between cross sections.

Δt =Computational time step.

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

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

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 7-2 Factors for Computing Wave Speed from Average Velocity

| Channel Shape | Ratio V_w/V |
|------------------|---------------|
| Wide Rectangular | 1.67 |
| Wide Parabolic | 1.44 |
| Triangular | 1.33 |
| Natural Channel | 1.5 |

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 may go unstable. The solution to this problem in general is to decrease the time step. An example of a hydrograph routed with two different time steps (1 minute and 10 minutes) is shown in Figure 7-43 below. As shown in the Figure, the hydrograph routed with a 10 minute time step has a much lower peak flow, and the leading edge of the floodwave is not as steep. This is due to the fact that the time based derivatives in the solution are averaging the changes in the floodwave over too long of a time step, thus numerically dampening the floodwave.

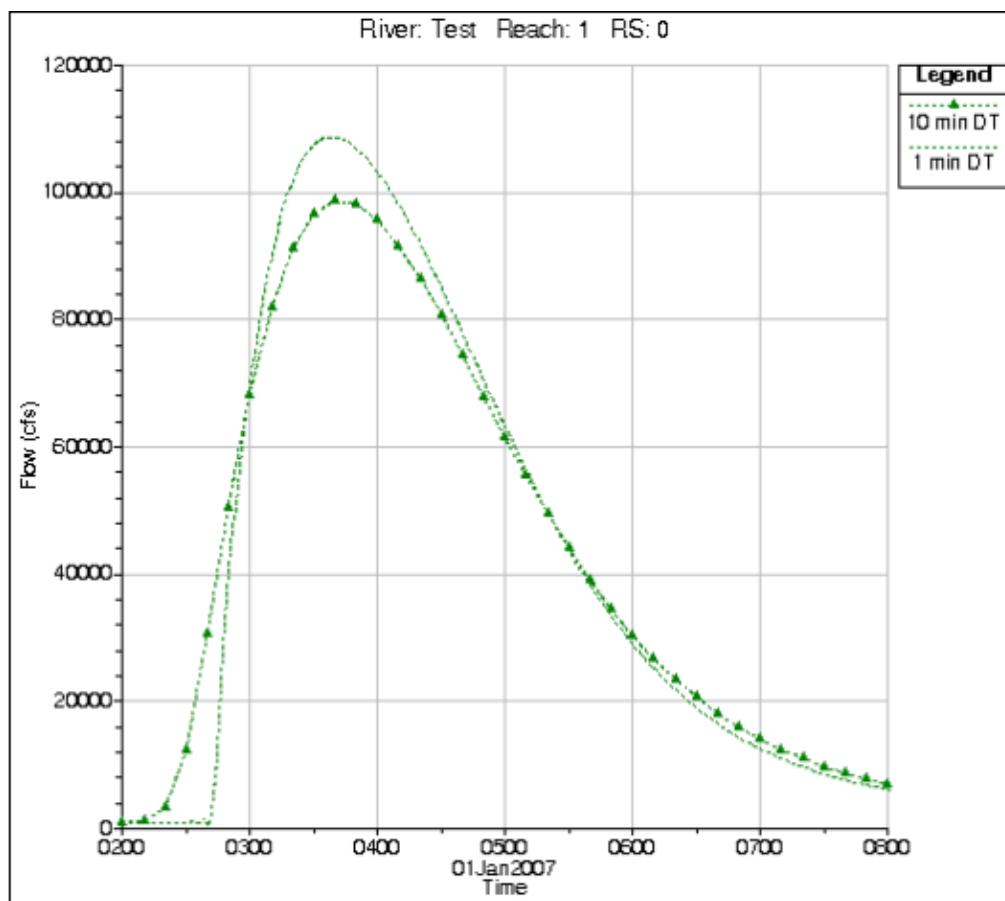


Figure 7.43.

Hydrograph routed with two different time steps (1 and 10 minutes).

Too Small of a Time Step. If a time step is selected that is much smaller than what the Courant condition would dictate for a given flood wave, this can also cause model stability problems. In general too small of a time step will cause the leading edge of the flood wave to steepen, possibly to the point of oscillating and going unstable.

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: Tr=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, but less numerical accuracy.

Theta of 0.6 provides the most accuracy, but less numerical stability. 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.

For rivers with tidal boundaries, in which the rising tide will propagate upstream, the user should always try to use a theta value as close to 0.6 as possible. Tidal waves are very dynamic. In order for the solution to be able to accurately model a tidal surge, theta must be close to 0.6.

Calculation Options and Tolerances. Within the HEC-RAS software there are several calculation options and tolerances that can affect the stability and accuracy of the solution. Some of the more important calculation options and tolerances are:

Calculation Tolerances: Three solution tolerances can be set or changed by the user: Water surface calculation (0.02 default); Storage area elevation (0.02 default); and Flow calculation (Default is that it is not used). The default values should be good for most river systems. Only change them if you are sure!!!

Making the tolerances larger will actually make the mode less stable. Making the tolerances smaller will make the model more stable, but may cause the program to iterate more, thus increasing the run time.

Maximum Number of 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.

Warm up time step and duration: The user can instruct the program to run a number of iterations at the beginning of the simulation in which all inflows are held constant. This is called the warm up period. The default is not to perform a warm up period, but the user can specify a number of time steps to use for the warm up period. The user can also specify a specific time step to use (default is to use the user selected computation interval). The warm up period does not advance the simulation in time, it is generally used to allow the unsteady flow equations to establish a stable flow and stage before proceeding with the computations.

Time Slicing: The user can control the maximum number of time slices and the minimum time step used during time slicing. There are two ways to invoke time slicing: rate of change of an inflow hydrograph or when a maximum number of iterations is reached.

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.

Inline and Lateral Structure Stability Issues. Inline and Lateral Structures 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 Inline and Lateral Structure stability factors, which can smooth these oscillations by damping the initial estimate of the computed flows.

The Inline and Lateral Structure 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 initial guess of the flows (which will provide for greater stability).

Long 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. 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 a 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, and/or weir/spillway stability factors.

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 little slower, or both if necessary.

Weir and Gated Spillway Submergence Factors. When you have a weir or gated spillway connecting two storage areas, or a storage area and a reach, oscillations can occur when the weir or gated 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 7-44.

The net result of using a weir flow submergence factor greater than 1.0, is the flow rate will be reduced more at lower degrees of submergence, but the computations will be much more stable at high degrees of submergence.

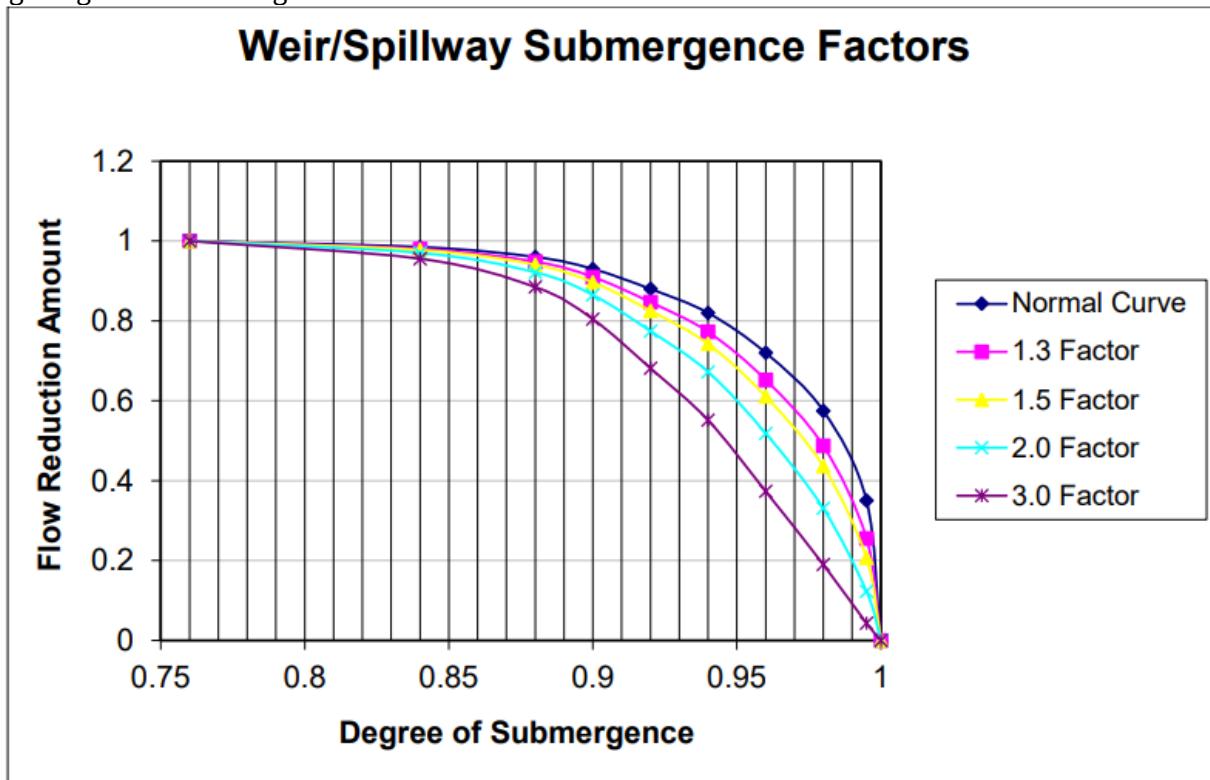


Figure 7-44. Weir/Spillway Submergence Factors.

Steep Streams and Mixed Flow Regime: Higher velocities and rapid changes in depth and velocity are more difficult to model and keep a stable solution. As the Froude number approaches 1.0 (critical depth), the inertial terms of the St. Venant equations and their associated derivatives tend to cause model instabilities (For the 1D Finite Difference solver). The default solution methodology for unsteady flow routing within HEC-RAS is generally for subcritical flow. The software does have an option to run in a mixed flow regime mode. However, 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 possibly 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. 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. Another solution is to increase the base flow in the hydrographs, as well as the base flows used for computing the initial conditions. Increased base flow will often dampen out any water surfaces going towards or through critical depth due to low flows.

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 too low of stages; and normal depth boundary conditions where the user has entered too steep of a slope. Shown in Figure 7-45 is an example in which a Normal depth boundary condition was used with too steep of an energy slope entered by the user. The net affect was that for any given flow, the water surface elevation was computed much lower than it should have been, as shown in the figure. The water surface just upstream of the boundary condition becomes very steep, and potentially can lead to an unstable solution.

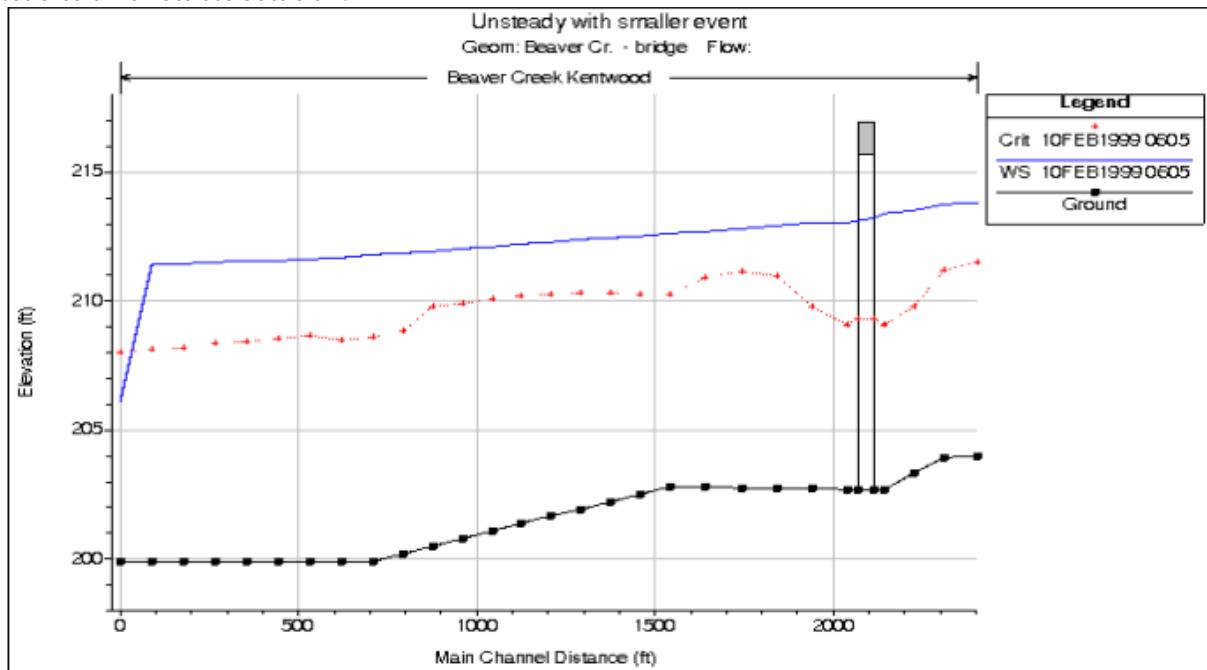


Figure 7-45. Example of a bad downstream boundary condition.

Cross section Geometry and Table 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.

Bridge and Culvert crossings. Bridge/Culvert crossings can be a common source of model stability problems when performing an unsteady flow analysis. Bridges may be overtapped during an event, or even washed out. Common problems at bridges/culverts are the extreme rapid rise in stages when flow hits the low chord of the bridge deck or the top of the culvert. Modelers need to check the computed curves closely and make sure they are reasonable. One solution to this problem is to use smaller time steps, such that the rate of rise in the water surface is smaller for a given time step. Modelers may also need to change hydraulic coefficients to get curves that have more reasonable transitions.

An additional problem is when the curves do not go high enough, and the program extrapolates from the last two points in the curve. This extrapolation can cause problems when it is not consistent with the cross section geometry upstream and downstream of the structure.

For bridge and culvert crossing 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, starting 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. An example set of curves are shown in Figure 7-46.

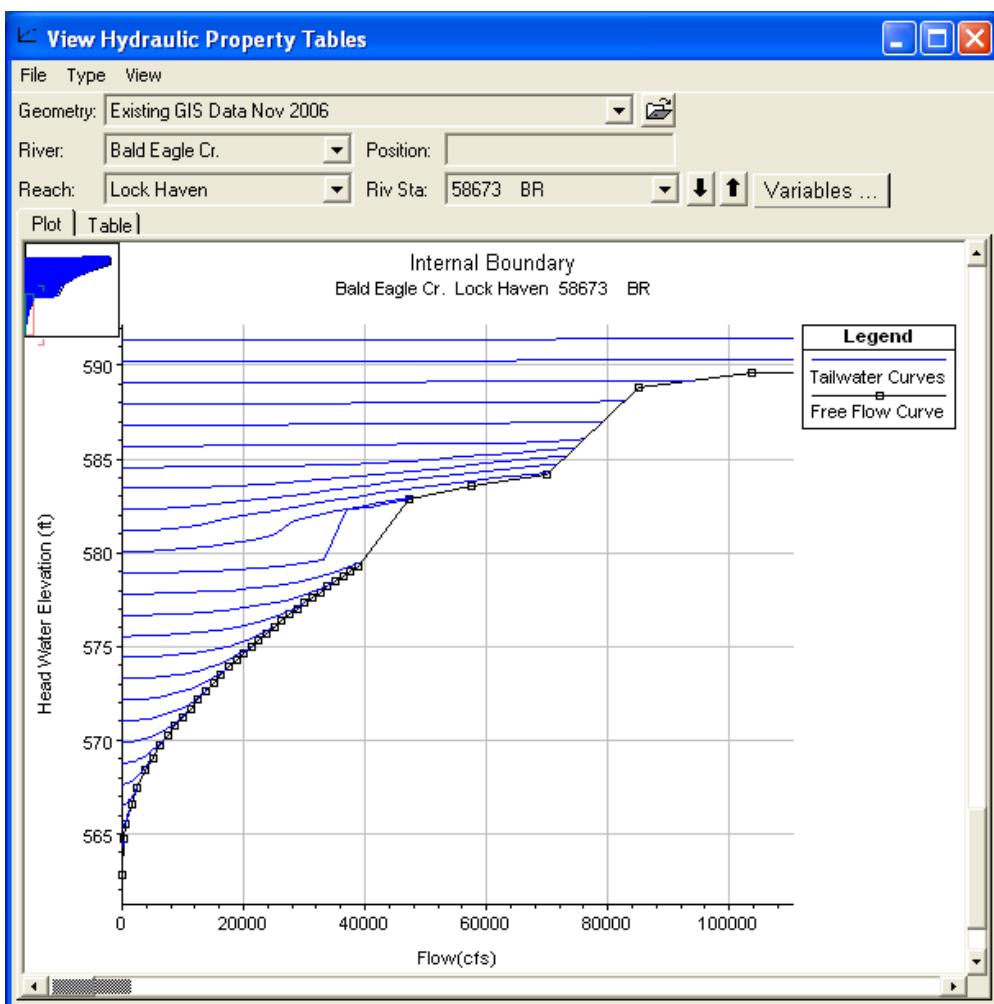


Figure 7 46.

Example Family of Curves for a Bridge crossing.

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

Initial Conditions and Low Flow: When starting a simulation it is very common to start the system at low flows. Make sure that the initial conditions flow is consistent with the first time step flow from the unsteady flow boundary conditions. User's must also pay close attention to initial gate settings and flows coming out of a reservoir, as well as the initial stage of the pool in the reservoir. The initial condition flow values must be consistent with all inflow hydrographs, as well as the initial flows coming out of the reservoir.

Flows entered on the initial conditions tab of the Unsteady Flow Data editor are used for calculating stages in the river system based on steady flow backwater calculations. If these flows and stages are

inconsistent with the initial flows in the hydrographs, and coming out of the reservoir, then the model may have computational stability problems at the very beginning of the unsteady flow computations.

If any portion of an inflow hydrograph is so low that it causes the stream to go through a pool and riffle sequence, it may be necessary to increase the base flow. The minimum flow value must be small enough that it is negligible when compared to the peak of the flood wave. A good rule of thumb is to start with a minimum flow equal to about 1 % of the peak flood (inflow hydrograph, or dam breach flood wave) and increase as necessary to 10%. If more than 10% is needed, then the problem is probably from something else.

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.

Drops in the Bed Profile. Significant drops in the bed profile can also be a source of model stability problems, especially at low flows. If the drop is very small, then usually an increase in baseflow will drown out the drop, thus preventing the model from passing through critical depth. If the drop is significant, then it should be modeled with an inline structure using a weir. This will allow the model to use a weir equation for calculating the upstream water surface for a given flow, rather than using the unsteady flow equations. This produces a much more stable model, as the program does not have to model the flow passing through critical depth with the unsteady flow equations. HEC-RAS automatically handles submergence on the weir, so this is not a problem. An example of a profile drop that causes a model stability problem is shown in Figure 7-47.

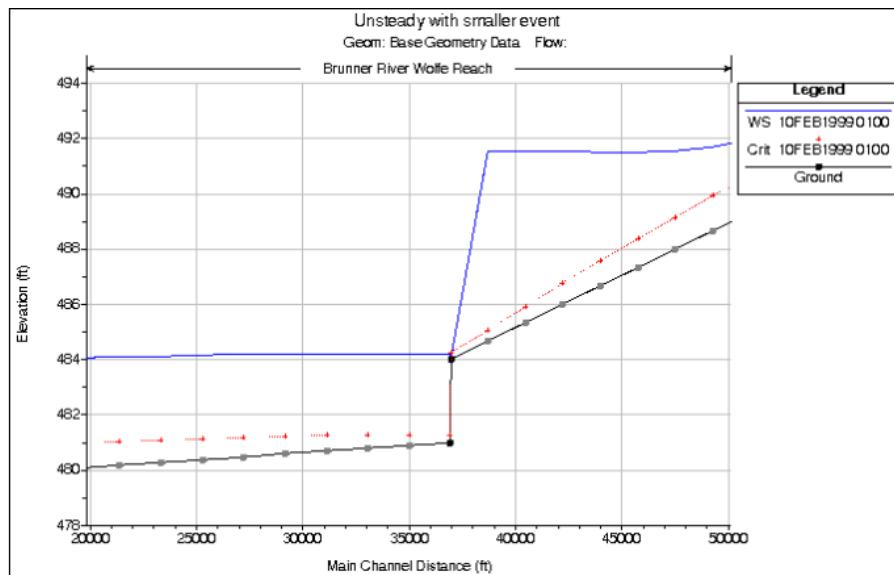


Figure 7.47. Stability

Problem caused by drop in bed profile.

When an Inline Structure (weir) is added to the above data set, the model is able to obtain a stable and accurate solution of the profile (Figure 7-48).

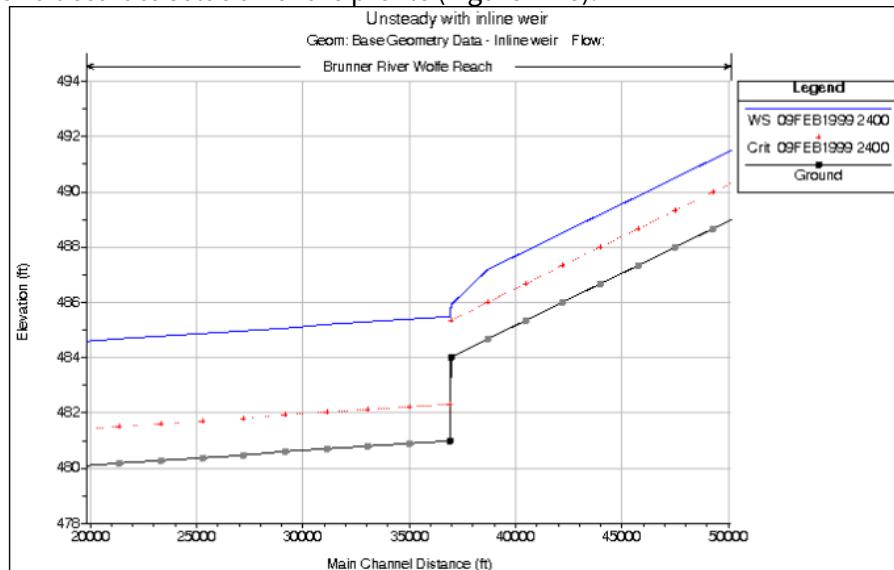


Figure 7.48. Stable solution

using Inline Structure to represent profile drop.

Some additional solutions to the problem of significant drops in the channel invert are: increase the base flow to a high enough value to drown out the drop in the bed profile; put a rating curve into the cross section at the top of the drop (this will prevent the unsteady flow equations being solved through the drop, the rating curve will be used instead); and add more cross section, if the drop is gradual, and run the program in mixed flow regime mode.

Manning's n Values. Manning's n values can also be a source of model instability. Manning's n

values that are too low, will cause shallower depths of water, higher velocities, and possibly even supercritical flows. This is especially critical in steep streams, where the velocities will already be high. User's should check there estimated Manning's n values closely in order to ensure reasonable values. It is very common to underestimate Manning's n values in steep streams. Use Dr. Robert Jarrets equation for steep streams to check your main channel Manning's n values. An example model stability problem due to too low on Manning's n values being used in steep reaches is shown in Figure 7-49.

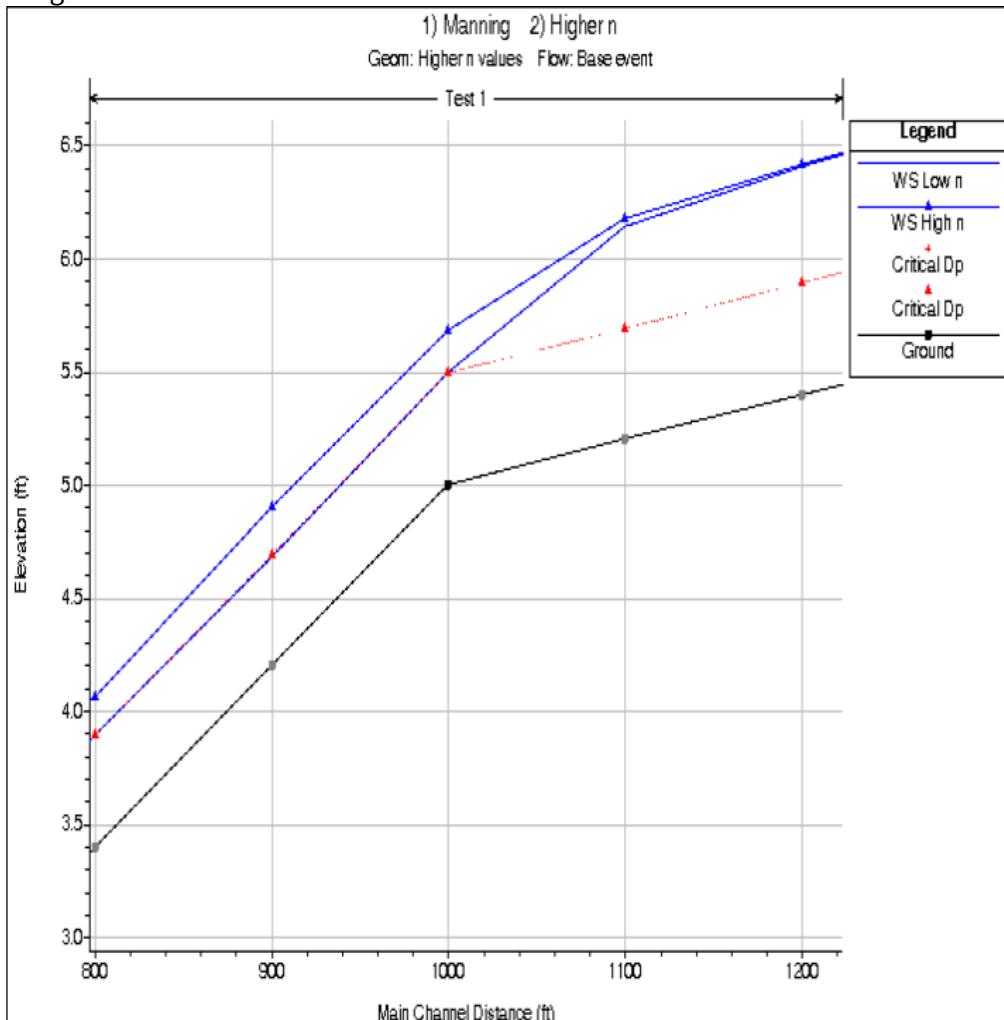


Figure 7-49.

Model stability problem due to low Manning's n values.

Over estimating Manning's n values will cause higher stages and more hydrograph attenuation than may be realistic.

Missing or Bad Channel Data. Another typical source of instabilities occurs when the main channel has a wide flat bed. This is usually found when cross sections are approximated or when terrain data is used to develop cross sections exclusive of real bathymetric data. Many times reaches are developed in GIS using LIDAR data or other aerial means. These survey methods don't penetrate water surfaces so the main channel is left with a flat horizontal bed equal to the water surface

elevation (Figure 7-50). For dam breach analyses, shallow streams are normally okay, since the dam break flood wave is usually much greater than the depth of water in the channel. However, wide flat stream beds tend to cause instabilities because at lower flows, the area to depth ratio is very high. When this occurs, a small increase in depth is seen as a large relative increase in depth.

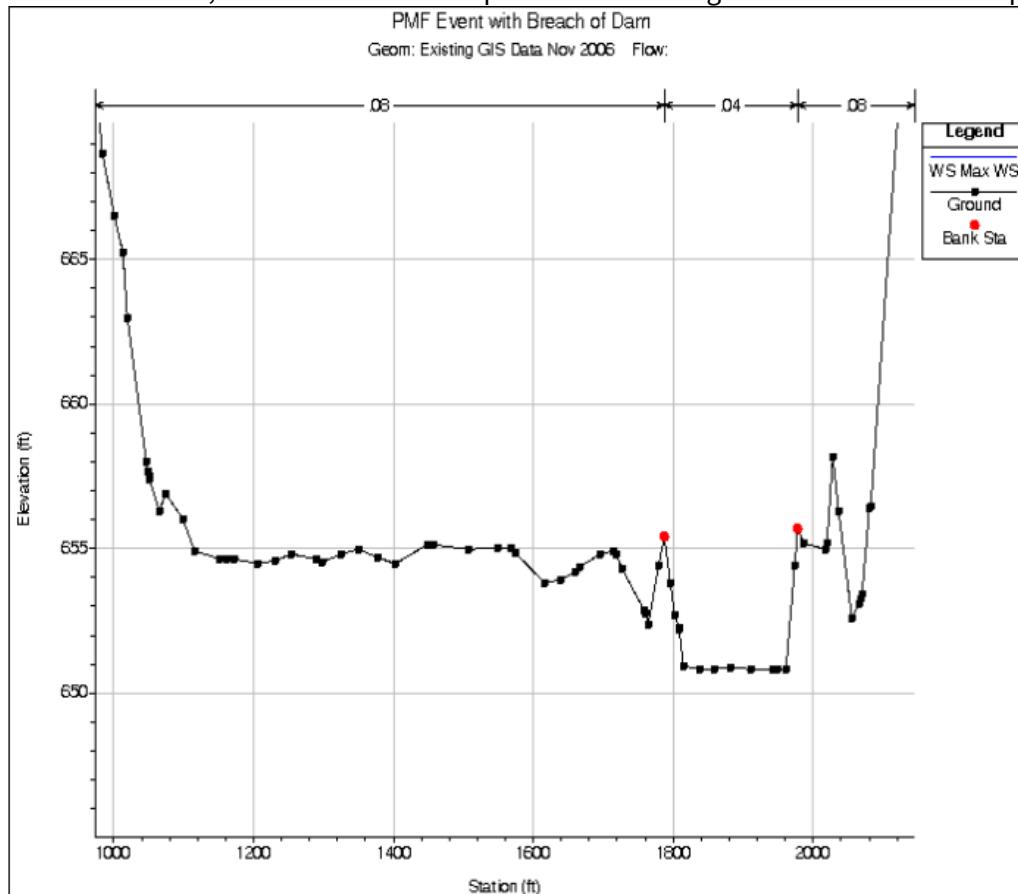


Figure 7-50.

Example Cross Section missing most of the main channel.

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

Stream Junction Issues. The unsteady flow equation solver, by default, makes a simplifying assumption at stream junctions. The unsteady solver forces the same exact water surface at all cross sections that bound a junction (Upstream and downstream of the junction). This simplifying assumption is fine for flat to moderately sloping streams. However, once you get to medium to steep sloping streams, this is normally a bad assumption, and can even cause model instability issues.

In general, cross sections placed around stream junctions should be placed as close to the junction as possible, such that the assumption of equal water surface elevation is not so bad. Sometimes this is not possible. For example, in steep streams, the first cross section of a tributary coming into a main stem may have a higher channel invert than the main stem river. If you are starting the model at a low flow, the program computes the water surface in the main river below the junction, then forces that water surface on the cross sections upstream of the junction, both in the main river and

the tributary. This can often end up with way too low of a water surface elevation in the tributary, for the given flow rate, which very quickly causes the model to go unstable in the tributary reach near the junction.

The solution to this problem, is to first ensure you have the cross sections bounding the junction, as close to the junction as possible. Second, compare at the main channel elevations of all the cross sections that bound the junction. If one cross section is much higher than the others (say the tributary one), then there will be a problem trying to run this model at low flow. Either extract a new cross section closer to the junction (thus having a lower main channel), or adjust the main channel data of that cross section.

An additional option that has been added to HEC-RAS to assist in this problem, is the option to use an "Energy Balance Method" to compute the water surface elevations across the junction during the unsteady flow computations. This option will allow for sloping water surface elevations across the junction and can help alleviate many model stability issues at junctions in medium to steep sloping streams.

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 coefficients 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.

Finding and Fixing Model Stability Problems

Detecting Model 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 with large numerical errors.
3. There are unrealistic oscillations in the computed stage and flow hydrographs, or the water surface profiles.
4. The computed error in the water surface elevation is very large.

What do you do when this happens?

- Note the simulation time and location from the computation window when the program either blew up or first started to go to the maximum number of iterations with large water surface errors.
- Use the HEC-RAS Profile and Cross Section Plots as well as the Tabular Output to find the problem location and issue.
- If you cannot find the problem using the normal HEC-RAS output - Turn on the "Computation Level Output" option and re-run the program.
- View the time series and profile output associated with the Computation Level Output option. Locate the simulation output at the simulation time when the solution first started to go bad.
- Find the river station locations that did not meet the solution tolerances. Then check the data in this general area.

The Computational Window is the first place to look for problems. When the maximum number of iterations is reached, and solution error is greater than the predefined tolerance, the time step, river, reach, river station, water surface elevation and the amount of error is reported. When the error increases too much, the solution will stop and say "Matrix Solution Failed". Often the first river station to show up on the window can give clues to the source of instabilities.

An example of the Computation Window with an unstable model solution is shown in Figure 7-51.

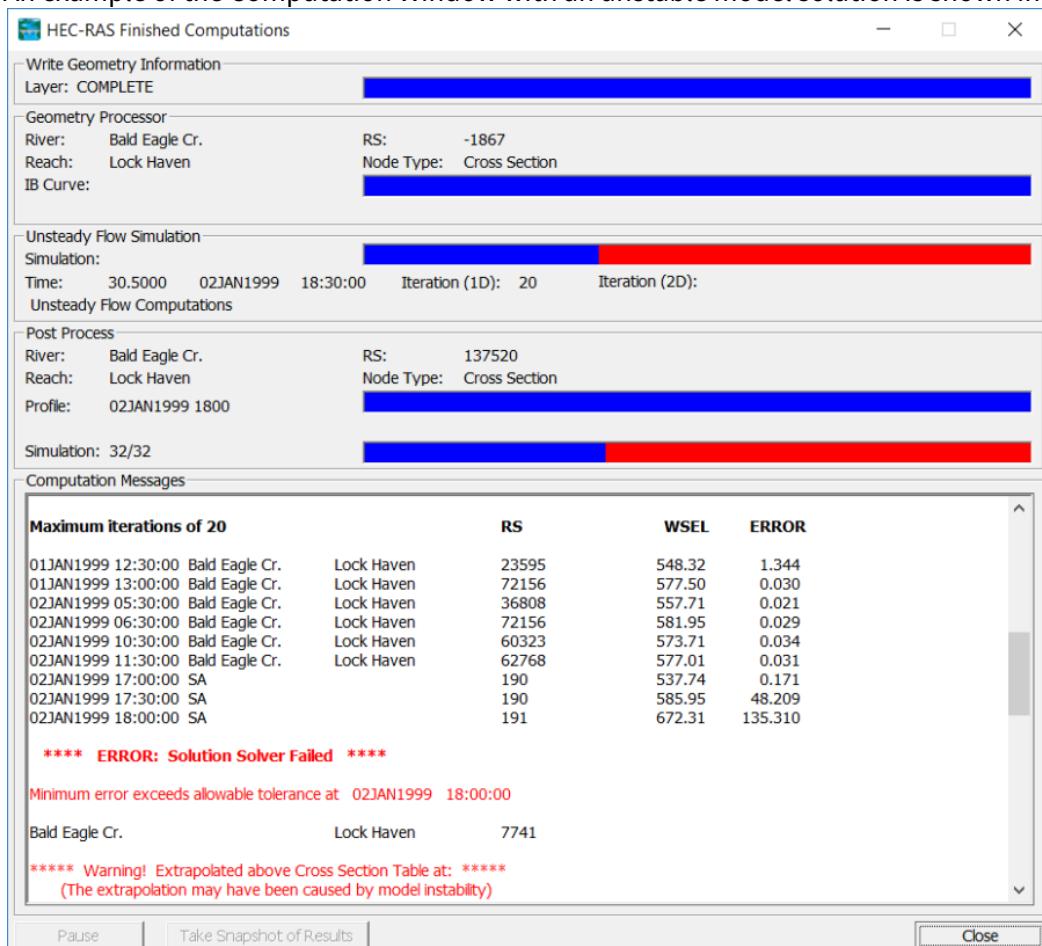
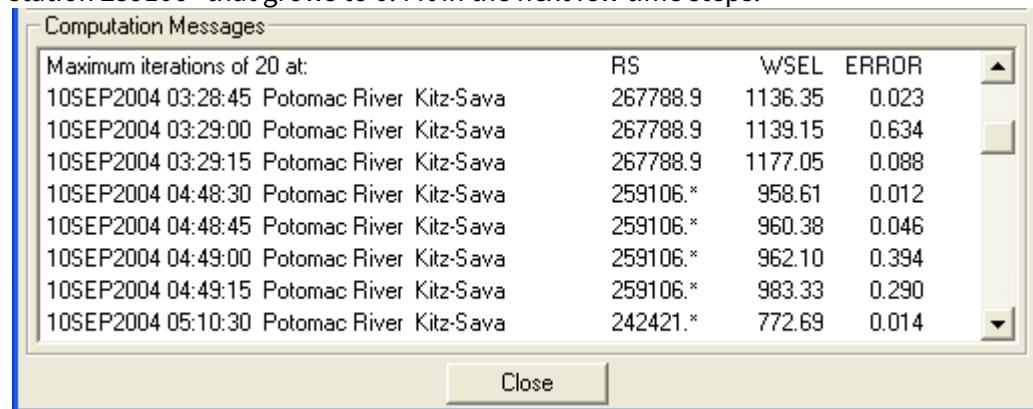


Figure 7 51. Example Unsteady Flow Computation window with unstable solution.

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

Sometimes the first error to occur is at the beginning of the simulation and is just a result of the model settling out after the transition from initial conditions to the first time step. Particularly if the error only occurs once for that given river station. It is better to focus on reoccurring errors or compounding errors first. The example shown in Figure 7-52 shows a relatively small error at river station 259106* that grows to 0.4 ft in the next few time steps.



The screenshot shows a window titled "Computation Messages". The table lists errors occurring at various river stations (RS) and times. The columns are labeled RS, WSEL, and ERROR. The RS column includes entries like "259106.*" which indicates an unstable node. The ERROR column shows values increasing over time, such as from 0.012 to 0.394.

| RS | WSEL | ERROR |
|--|----------|---------|
| 10SEP2004 03:28:45 Potomac River Kitz-Sava | 267788.9 | 1136.35 |
| 10SEP2004 03:29:00 Potomac River Kitz-Sava | 267788.9 | 1139.15 |
| 10SEP2004 03:29:15 Potomac River Kitz-Sava | 267788.9 | 1177.05 |
| 10SEP2004 04:48:30 Potomac River Kitz-Sava | 259106.* | 958.61 |
| 10SEP2004 04:48:45 Potomac River Kitz-Sava | 259106.* | 960.38 |
| 10SEP2004 04:49:00 Potomac River Kitz-Sava | 259106.* | 962.10 |
| 10SEP2004 04:49:15 Potomac River Kitz-Sava | 259106.* | 983.33 |
| 10SEP2004 05:10:30 Potomac River Kitz-Sava | 242421.* | 772.69 |

Figure 7 52. Example of growing computational errors.

Utilizing the Profile Plot.

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

Computational Level Output for Debugging

An additional feature that has been added to HEC-RAS to help user's find model stability problems, is the "Computational Level Output" option. When performing an unsteady flow analysis the user can optionally turn on the ability to view output at the computation interval level. This is accomplished by checking the box labeled **Computation Level Output** on the Unsteady Flow Analysis window (In the Computations Settings area on the window). When this option is selected an additional binary file containing output at the computation interval is written out. Users can control what output is written to this file from the options called **Output Options**. There is a tab labeled "Computation Level Output Options" on this window that allows you to control what gets written to this file, and

there is also a time window option for controlling the time period for writing this data. After the simulation the user can view computation level output by selecting either **Unsteady Flow Spatial Plot** or **Unsteady Flow Time Series Plot** from the **View** menu of the main HEC-RAS window.

Visualization of computation level output can be accomplished with either **Spatial Plots** or **Time**

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

Especially plotting the water surface error and animating it in time. An example of the computational level output spatial plot is shown in Figure 7-53.

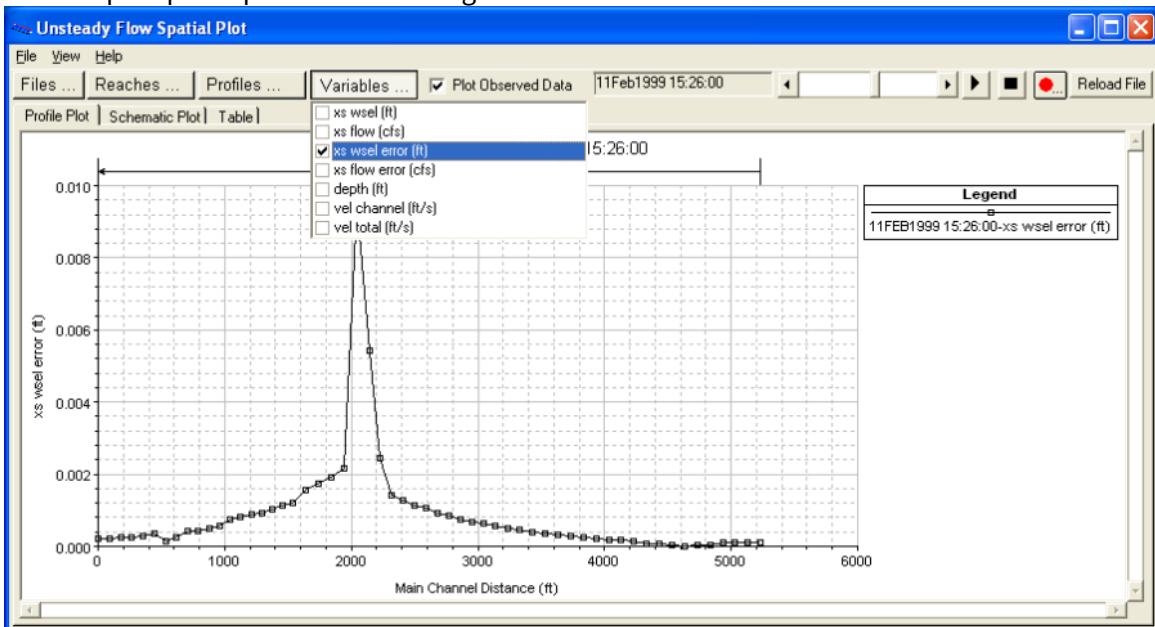


Figure 7-53. Example Spatial Plot from Computational Level Output.

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

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

Detailed Log Output for Debugging

If you detect a possible stability problem, and you are unable to find the location using the graphical output discussed above, another option for finding the location of the problem is 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:

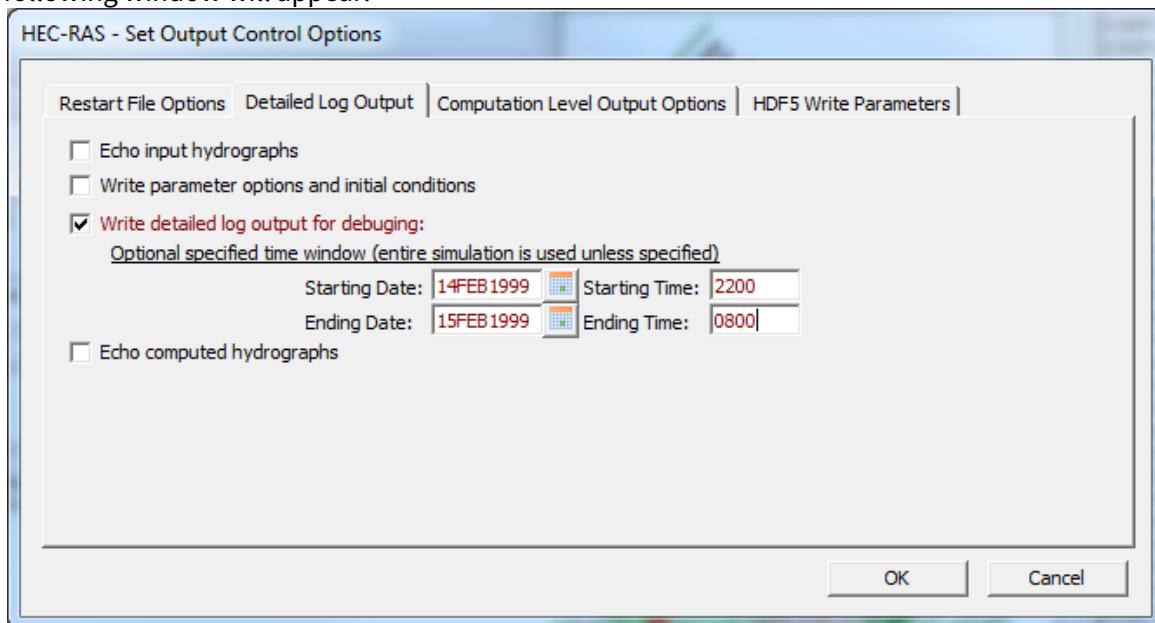


Figure 7 54. Detailed Log Output Control

As shown in Figure 7-54, the tab labeled "Detailed Log Output" allows the user to control this option. 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, labeled "Write detailed log output for debugging", is used to control the detailed output of results from the unsteady flow simulation. Selecting this option 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. Additionally the user can request that detailed log output only be written when the program reaches a certain number of iterations.

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

The detailed log output file will contain the following output:

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

Unsteady Flow Computations Output: Detailed unsteady flow calculations including:

- Job control parameters
- Initial conditions calculations
- Detailed output for each time step

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

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 7-55 below.

beaver.bco - Notepad

File Edit Format View Help

Initial Conditions from Backwater Solution
(Displayed upstream to downstream)

| Beaver Creek | | Kentwood | | | | | |
|--------------|--------|----------|------------|-----------|--------|----------|----------|
| Riv. Sta. | Flow | WSEL | Crit Depth | EG Slope | Area | Topwidth | velocity |
| 5.99 | 500.00 | 212.94 | 209.36 | 0.0010149 | 523.27 | 232.82 | 0.956 |
| 5.97 | 500.00 | 212.86 | 210.12 | 0.0008803 | 462.49 | 218.25 | 1.081 |
| 5.951 | 500.00 | 212.76 | 210.28 | 0.0010507 | 405.21 | 204.34 | 1.234 |
| 5.93 | 500.00 | 212.64 | 210.51 | 0.0012506 | 352.26 | 186.90 | 1.419 |
| 5.913 | 500.00 | 212.50 | 210.60 | 0.0014622 | 304.26 | 166.14 | 1.643 |
| 5.894 | 500.00 | 212.33 | 210.51 | 0.0016595 | 263.13 | 150.78 | 1.900 |
| 5.875 | 500.00 | 212.15 | 210.21 | 0.0018348 | 227.94 | 134.41 | 2.194 |
| 5.855 | 500.00 | 211.95 | 209.85 | 0.0019770 | 200.95 | 116.86 | 2.488 |
| 5.836 | 500.00 | 211.74 | 209.49 | 0.0020606 | 185.99 | 96.42 | 2.688 |
| 5.81 | 500.00 | 211.55 | 209.14 | 0.0017232 | 183.89 | 90.51 | 2.719 |
| 5.798 | 500.00 | 211.40 | 208.77 | 0.0014500 | 216.29 | 149.27 | 2.312 |
| 5.779 | 500.00 | 211.29 | 208.40 | 0.0010977 | 307.30 | 258.66 | 1.627 |
| 5.76 | 500.00 | 211.22 | 208.04 | 0.0007327 | 459.43 | 329.33 | 1.088 |
| 5.741 | 500.00 | 211.11 | 208.35 | 0.0012069 | 327.98 | 288.06 | 1.524 |
| 5.72 | 500.00 | 210.97 | 208.62 | 0.0015937 | 245.94 | 197.64 | 2.033 |
| 5.703 | 500.00 | 210.82 | 209.02 | 0.0013225 | 231.52 | 112.44 | 2.160 |
| 5.685 | 500.00 | 210.70 | 208.84 | 0.0011421 | 231.12 | 108.38 | 2.163 |
| 5.666 | 500.00 | 210.58 | 208.61 | 0.0010798 | 244.18 | 155.00 | 2.048 |
| 5.647 | 500.00 | 210.47 | 208.30 | 0.0010939 | 278.95 | 235.54 | 1.792 |
| 5.628 | 500.00 | 210.38 | 208.03 | 0.0009833 | 336.36 | 276.64 | 1.487 |

Figure 7-55. 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 7-56 below.

beaver.bco - Notepad

File Edit Format View Help

COMPUTED STAGES AND DISCHARGES AT T = 0.1667 HOURS - 2/10/1999 AT TIME 00:10:00

| Beaver Creek | | | | Kentwood | | | |
|--------------|--------|--------|------|--------------|--------|--------|------|
| Riv. Station | Z | Q | V | Riv. Station | Z | Q | V |
| 5.99 | 212.14 | 336.30 | 0.93 | 5.798 | 210.61 | 328.06 | 2.29 |
| 5.97 | 212.06 | 334.72 | 1.08 | 5.779 | 210.54 | 327.78 | 1.84 |
| 5.951 | 211.95 | 333.31 | 1.26 | 5.76 | 210.49 | 327.35 | 1.28 |
| 5.93 | 211.81 | 332.12 | 1.50 | 5.741 | 210.40 | 326.99 | 1.82 |
| 5.913 | 211.64 | 331.06 | 1.78 | 5.72 | 210.28 | 326.84 | 2.13 |
| 5.894 | 211.44 | 330.13 | 2.15 | 5.703 | 210.13 | 326.77 | 2.03 |
| 5.875 | 211.22 | 329.48 | 2.52 | 5.685 | 210.00 | 326.77 | 1.96 |
| 5.855 | 211.02 | 328.97 | 2.69 | 5.666 | 209.89 | 326.84 | 1.95 |
| 5.836 | 210.85 | 328.57 | 2.66 | 5.647 | 209.79 | 326.99 | 1.85 |
| 5.81 | 210.71 | 328.30 | 2.51 | 5.628 | 209.69 | 327.24 | 1.63 |
| 5.44 | 209.07 | 331.79 | 1.17 | 5.274 | 208.53 | 339.53 | 1.78 |
| 5.425 | 209.07 | 332.48 | 0.84 | 5.258 | 208.22 | 340.42 | 2.18 |
| 5.41 | 209.07 | 333.39 | 0.66 | 5.242 | 207.81 | 341.41 | 2.77 |
| 5.39 | 209.07 | 333.39 | 0.66 | 5.226 | 207.29 | 342.20 | 3.70 |
| 5.37 | 209.06 | 334.40 | 0.84 | 5.21 | 206.79 | 342.71 | 3.89 |
| 5.35 | 209.03 | 335.26 | 1.12 | 5.194 | 206.33 | 343.29 | 3.94 |
| 5.33 | 208.97 | 336.35 | 1.54 | 5.178 | 205.98 | 343.98 | 3.56 |
| 5.31 | 208.86 | 337.52 | 1.63 | 5.162 | 205.75 | 344.83 | 3.04 |
| 5.29 | 208.74 | 338.58 | 1.47 | 5.146 | 205.63 | 345.90 | 2.47 |

Figure 7-56. 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 7-55. 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 7-57, 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.

Stability2.bco - Notepad

| Iter | River | Station | Elev | DZ | Storage | Zsa | Dzsa | River | Station | Q | DQ |
|------|--------------|---------|--------|----------|---------|--------|----------|--------------|---------|-------|------|
| 0 | Beaver Creek | 5.0 | 211.72 | 0.77162 | STO #1 | 205.82 | 0.06177 | Beaver Creek | 5.0 | 11835 | 8277 |
| 1 | Beaver Creek | 5.0 | 211.33 | -0.55761 | STO #1 | 205.82 | 0.00012 | Beaver Creek | 5.0 | 12548 | 1019 |
| 2 | Beaver Creek | 5.0 | 211.03 | -0.43827 | STO #1 | 205.82 | -0.00002 | Beaver Creek | 5.0 | 13160 | 874 |
| 3 | Beaver Creek | 5.0 | 210.78 | -0.35800 | STO #1 | 205.82 | -0.00007 | Beaver Creek | 5.0 | 13611 | 644 |
| 4 | Beaver Creek | 5.0 | 210.57 | -0.29612 | STO #1 | 205.82 | -0.00006 | Beaver Creek | 5.0 | 13953 | 488 |
| 5 | Beaver Creek | 5.0 | 210.39 | -0.25120 | STO #1 | 205.82 | -0.00004 | Beaver Creek | 5.0 | 14205 | 360 |
| 6 | Beaver Creek | 5.0 | 210.24 | -0.21750 | STO #1 | 205.82 | -0.00003 | Beaver Creek | 5.0 | 14393 | 269 |
| 7 | Beaver Creek | 5.0 | 210.11 | -0.18898 | STO #1 | 205.82 | -0.00002 | Beaver Creek | 5.0 | 14545 | 216 |
| 8 | Beaver Creek | 5.0 | 209.99 | -0.16978 | STO #1 | 205.82 | -0.00002 | Beaver Creek | 5.0 | 14654 | 156 |
| 9 | Beaver Creek | 5.0 | 209.88 | -0.14935 | STO #1 | 205.82 | -0.00002 | Beaver Creek | 5.0 | 14755 | 144 |
| 10 | Beaver Creek | 5.0 | 209.79 | -0.13614 | STO #1 | 205.82 | -0.00001 | Beaver Creek | 5.0 | 14829 | 106 |
| 11 | Beaver Creek | 5.0 | 209.70 | -0.12276 | STO #1 | 205.82 | -0.00001 | Beaver Creek | 5.0 | 14896 | 96 |
| 12 | Beaver Creek | 5.0 | 209.62 | -0.11216 | STO #1 | 205.82 | -0.00001 | Beaver Creek | 5.0 | 14952 | 79 |
| 13 | Beaver Creek | 5.0 | 209.54 | -0.11670 | STO #1 | 205.82 | -0.00001 | Beaver Creek | 5.016 | 13156 | 54 |
| 14 | Beaver Creek | 5.0 | 209.47 | -0.10709 | STO #1 | 205.82 | -0.00001 | Beaver Creek | 5.0 | 14986 | 46 |
| 15 | Beaver Creek | 5.0 | 209.40 | -0.09978 | STO #1 | 205.82 | 0.00000 | Beaver Creek | 5.0 | 15020 | 49 |
| 16 | Beaver Creek | 5.0 | 209.33 | -0.09408 | STO #1 | 205.82 | 0.00000 | Beaver Creek | 5.0 | 15049 | 42 |
| 17 | Beaver Creek | 5.0 | 209.27 | -0.08969 | STO #1 | 205.82 | 0.00000 | Beaver Creek | 5.0 | 15072 | 33 |
| 18 | Beaver Creek | 5.0 | 209.21 | -0.08645 | STO #1 | 205.82 | 0.00000 | Beaver Creek | 5.0 | 15090 | 25 |
| 19 | Beaver Creek | 5.0 | 209.15 | -0.08429 | STO #1 | 205.82 | 0.00000 | Beaver Creek | 5.0 | 15103 | 18 |
| 20 | Beaver Creek | 5.0 | 209.09 | -0.08318 | STO #1 | 205.82 | 0.00000 | Beaver Creek | 5.0 | 15110 | 11 |

!WARNING, USED COMPUTED CHANGES IN FLOW AND STAGE AT MINIMUM ERROR. MINIMUM ERROR OCCURED DURING ITERATION 20.

Figure 7-57. 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 discouraged!!!

Performing a Dam or Levee Breach Analysis with HEC-RAS

HEC-RAS has the ability to perform Dam or Levee breaching analyses. User can perform a breach analysis on multiple Dams and/or levees simultaneously within the same unsteady flow run (There is no limit to the number of breach locations). Breach data can be entered for any and all Dams (Inline Structures) and Levees (Lateral Structures), however there is an option to turn specific locations on or off for any specific analysis.

The breach data entered, and the erosion process used in HEC-RAS for a Dam or a Levee is almost identical. Dams are entered as Inline Structures in HEC-RAS, and Levees are entered as Lateral Structures. Both types of structures have a breaching option, and the breaching editor used is almost identical. There are some minor differences in the data entered. Most of the differences are in how it is applied hydraulically in the model.

Currently the user has two Breaching Methodologies to choose from, either "User Entered Data" or "Simplified Physical". The **User Entered Data** method requires the user to enter all of the breach information (i.e. breach size, breach development time, breach progression, etc...). The **Simplified Physical** breaching method allows the user to enter velocity versus breach down-cutting and breach widening relationships, which are then used dynamically to figure out the breach progression versus the actual velocity being computed through the breach, on a time step by time step basis.

Note: The documentation in this chapter for dam and levee breaching is just an overview of how to use the User Interface to enter the data. For a more detailed discussion of Dam and Levee Breach, please go to the section on Dam and Levee Breaching in Chapter 14 of this Manual (Advanced Features for Unsteady Flow Analysis)

Dam (Inline Structure) Breach.

This option allows the user to perform a Dam Breaching 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. 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 7-58).

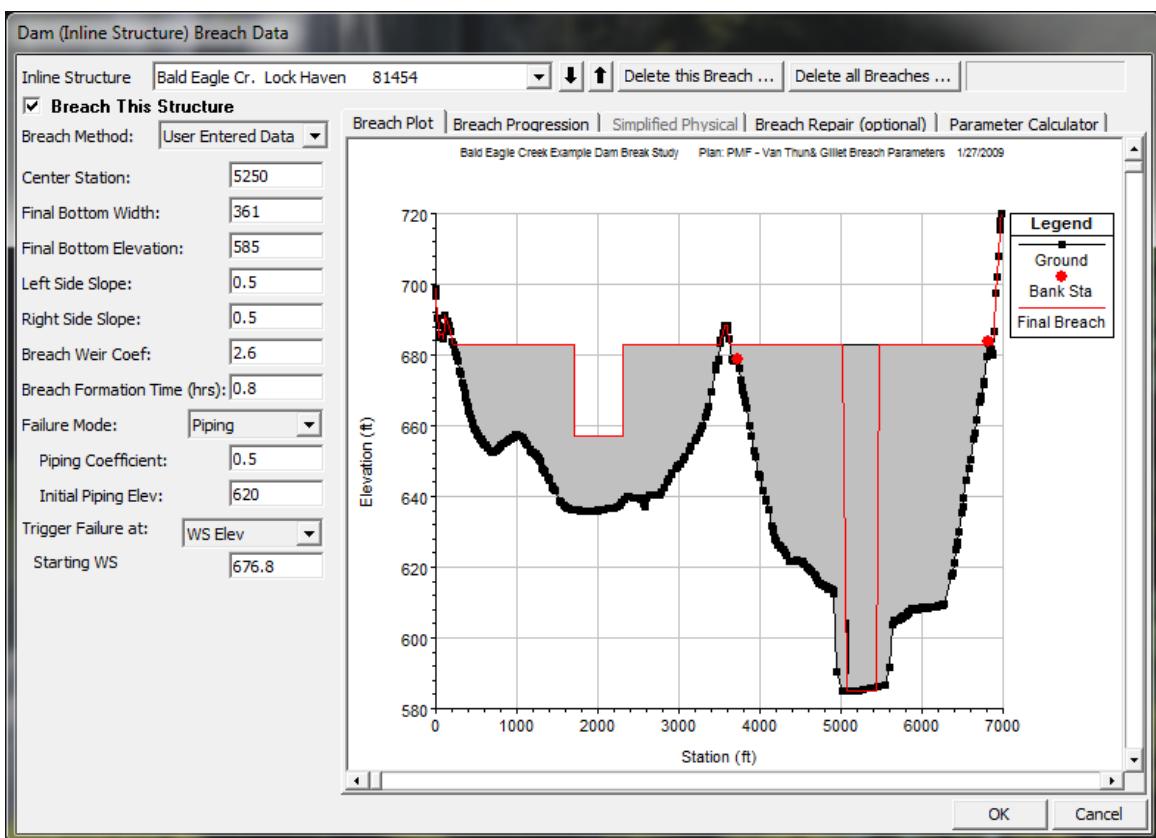


Figure 7 58. Dam Breach Editor

As shown in Figure 7-58, the user selects a particular Inline Structure to perform the breach on. At the top left of the editor is an option labeled: *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 losing the user entered breach information.

Next the decision needs to be made as to whether the user is going to compute the breach dimensions and development time and enter it as "User Entered Data", or if they want to use the "Simplified Physical" breaching option (his option requires velocity versus erosion rate information).

User Enter Breach Data

If the "**User Entered Data**" Breaching Method is selected, then the following data must be entered for the breaching analysis:

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.

Breach Weir Coef – This field is used for entering a weir coefficient for the breach area. For an overtopping failure, or when the top of a piping failure collapses, the program uses a weir equation to calculate the flow through the breach. Suggested range of values are 2.0 to 3.2, with 2.6 as a default value for most earth dams.

Breach 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 three trigger methods for initiating the breach. The three trigger methods are: a water surface elevation, a water surface elevation plus a duration of time that the water is above that elevation, and a specific time and date.

WS Elev - If the user selects water surface elevation for the failure trigger mode, then an additional field labeled **WS Start** must be entered. This field represents the water surface elevation at which the breach should begin to occur.

WS Elev+Duration – If the user selects **WS Elev+Duration** for the trigger mechanism of the failure, then they have three additional fields of data to enter. The first variable is **Threshold WS**. This variable is the water surface elevation at which the program starts to monitor the flow for duration above this water surface. The second variable is **Duration above Threshold**. This variable is used to specify the duration that the water surface must be above the threshold water surface elevation before the failure will initiate. The final variable, **Immediate initiation WS**, is used to instruct the program to begin the breach if the water surface in the structure reaches this elevation or higher, regardless of the duration requirement. This last field is optional.

Set Time - If the user selects the **Set Time** option, then a starting date and time to initiate the breach must be entered.

Breach Progression - 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 look like the following (Figure 7-59):

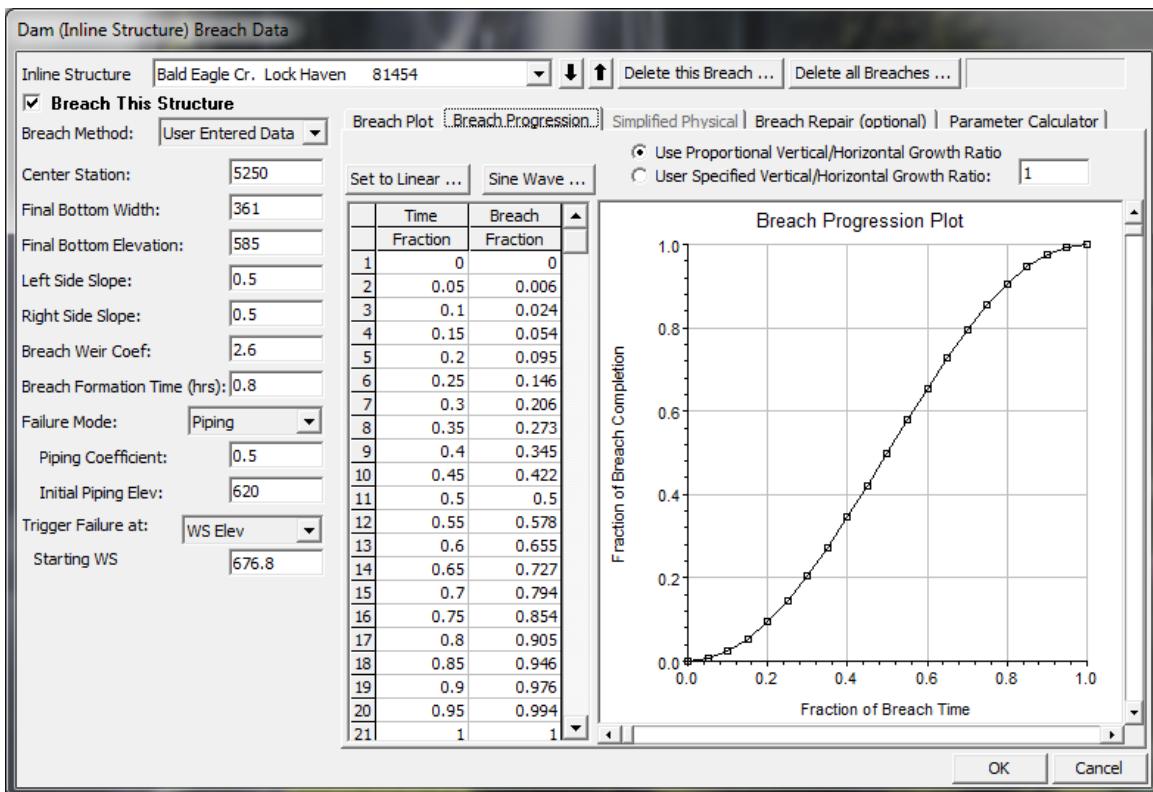


Figure 7.59. Dam Breach Editor with User Specified Breach Progression Tab Selected

As shown in Figure 7-59, 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: Previous to version 4.2, the horizontal and vertical growth rate of the breach was base on reaching the maximum breach depth and width at the user entered "Breach Formation Time". This means if a user put in breach dimensions of 400 ft wide and 100 ft deep, over a period of 2 hours, the horizontal growth rate was 200ft/hr and the vertical growth rate was 50 ft/hr. While this was generally ok for Dam breaches, it was not ok for levee breaching, in that levee breaches are much wider than they are tall. As of version 4.2, RAS computes the breach growth rate based on the breach "Final Bottom Width" and the user entered "Breach Formation Time". Then this same breach growth rate is used for the vertical down cutting of the breach. So in the previous example of a 400 ft breach bottom width and a 2 hour breach development time, the growth rate is 200 ft/hr, which is used for both the down cutting and widening rates. User's can change the vertical Breach Growth Rate by entering a value other than 1.0 under the option labeled "User Specified Vertical/Horizontal Growth Ratio" and the Breach Progression Tab. If a user enters a value of 0.5, that means you want the vertical growth rate to be half of what the Horizontal growth rate gets computed to be.

WARNING: The breach growth rate change described in the previous paragraph will generally results in RAS version 4.2 and newer yielding a higher peak flow through the breach, than versions 4.1 and older. If the user wants the same results as version 4.1 and older, you must compute a vertical/horizontal growth rate that will results in the breach reaching its maximum width and depth at the end of the breach formation time. For example (assuming an overtopping breach), if you specified a 400 ft breach bottom width and a 2 hour breach formation time, this is a horizontal growth rate of 200 ft/ hour. However, if you Dam is only 100 ft high, then to reproduce the version 4.1 or older results, the user would need to enter a "User Specified Vertical/Horizontal Growth Ratio" of 0.25. This

would cause the program to grow the breach vertically down to the 100 ft depth in exactly 2 hours. Piping breaches are more complicated, in that they have an initial elevation for the hole, and the vertical growth is both up and down.

The fourth Tab on the Breach editor is labeled **Breach Repair (Optional)**. This option allows the user to have the breach fill back in during the unsteady flow simulation. This could represent attempts to fill a breach during an event, or it could represent a repair of the breach after the event. Depending on the length of time being simulated, this may be a necessary option to represent what happened over the longer time frame. If this option is selected the user is required to enter three pieces of information: the number of hours after the full breach to start the repair; total repair time; and the final filled in elevation of the repair work. In general, this option was added for levee breaching analysis, and is not normally used during a Dam Breaching analysis.

The last Tab on the Breach Editor is Labeled **Parameter Calculator**. This option allows the user to enter some physical data about a Dam, and then using regression equations, it will compute potential Dam Breach bottom Widths, side slopes, and breach development times. Currently there are 5 different regression equations that have been programmed into this calculator, they are: MacDonald et al (1984); Froehlich (1995); Froehlich (2008); Von Thun & Gillete (1990); and Xu and Zhang (2009).

Simplified Physical Breaching

If the User selects to use the "**Simplified Physical**" Breach Method from the drop down at the top of the editor, the Breaching Editor will change to look the following way:

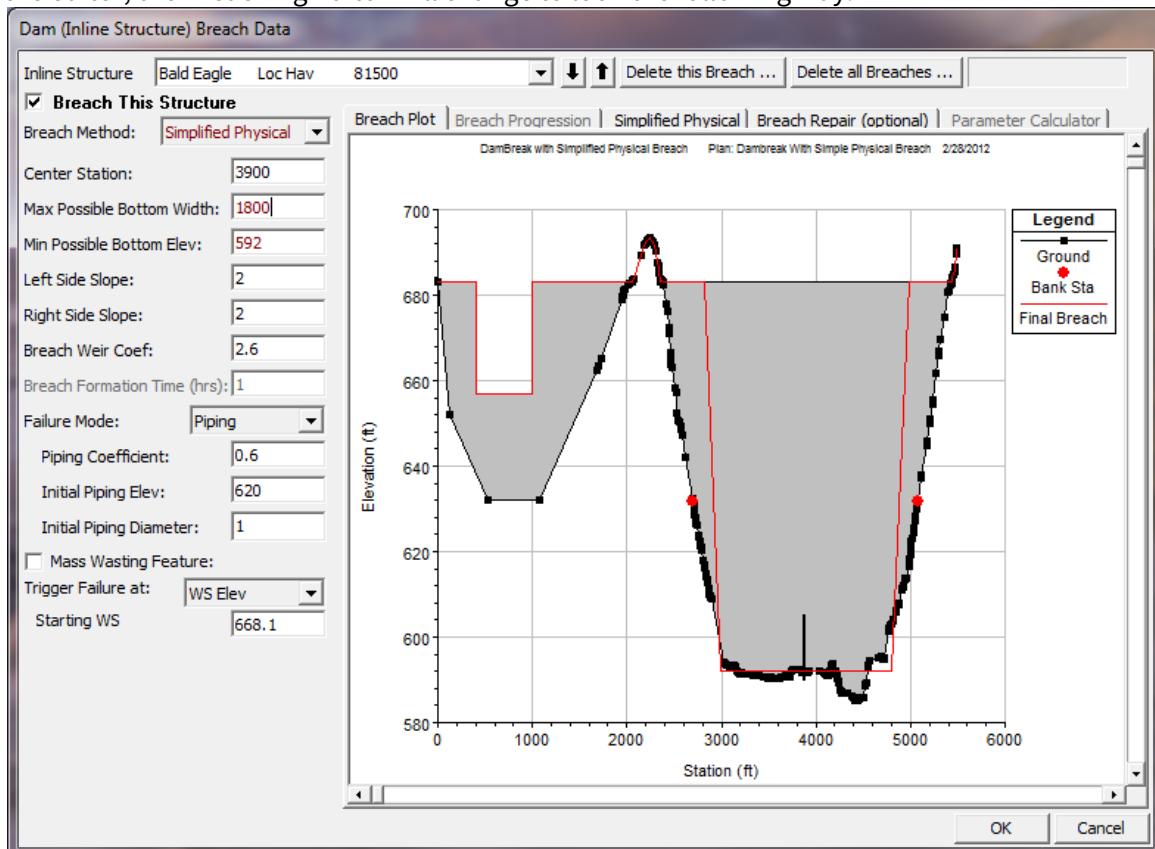


Figure 7 60. Simplified Physical Breaching Option for HEC-RAS

Once the User selects "Simplified Physical" breaching option, there are several fields in which labels change, some additional information required, and some previous information that is not required. The main changes between this method and the "User Entered Data" breach method are the following:

Max Possible Bottom Width – This field is now used to enter a maximum possible breach bottom width. This does not mean this will be the final breach bottom width, it is really being used to limit the breach bottom width growth to this amount. The actual bottom width will be dependent on the velocity versus erosion rate data entered, and the hydraulics of flow through the breach. This field is used to prevent breaches from growing larger than this user set upper limit during the run.

Min Possible Bottom Elev – This field is used to put a limit on how far down the breach can erode during the breaching process. This is not necessarily the final breach bottom elevation, it is a user entered limiter (i.e. the breach cannot go below this elevation). The final breach elevation will be dependent on the velocity versus erosion rate data entered, and the hydraulics of flow through the breach.

Starting Notch Width or Initial Piping Diameter – If the Overtopping failure mode is selected, the user will be asked to enter a starting notch width. The purpose of this is that the software will use this width at the top of the dam to compute a velocity, from the velocity it will get a down cutting erosion rate (based on user entered data), which will be used to start the erosion process. If a Piping Failure model is selected, the user must enter an initial piping diameter. Once the breach is triggered to start, this initial hole will show up immediately. A velocity will be computed through it, then the down cutting and widening process will begin based in user entered erosion rate data.

Mass Wasting Feature – This option allows the user to put a hole in the Dam or the Levee at the beginning of the breach, in a very short amount of time. This option would probably most often be used in a levee evaluation, in which a section of the levee may give way (Mass Wasting), then that initial hole would continue to erode and widen based on the erosion process. The require data for this option is a width for the mass wasting hole; duration in hours that this mass wasting occurs over (this would normally be a short amount of time); and the final bottom elevation of the initial mass wasting hole (it is assumed that the hole is open all the way to the top of the levee or Dam if this option is used).

When using the "Simplified Physical" breaching option, the user is required to enter velocity versus Downcutting erosion rates, as well as velocity versus erosion widening rates. To enter this data the user selects the "Simplified Physical" breach Tab. When this Tab is selected the editor will look like the following:

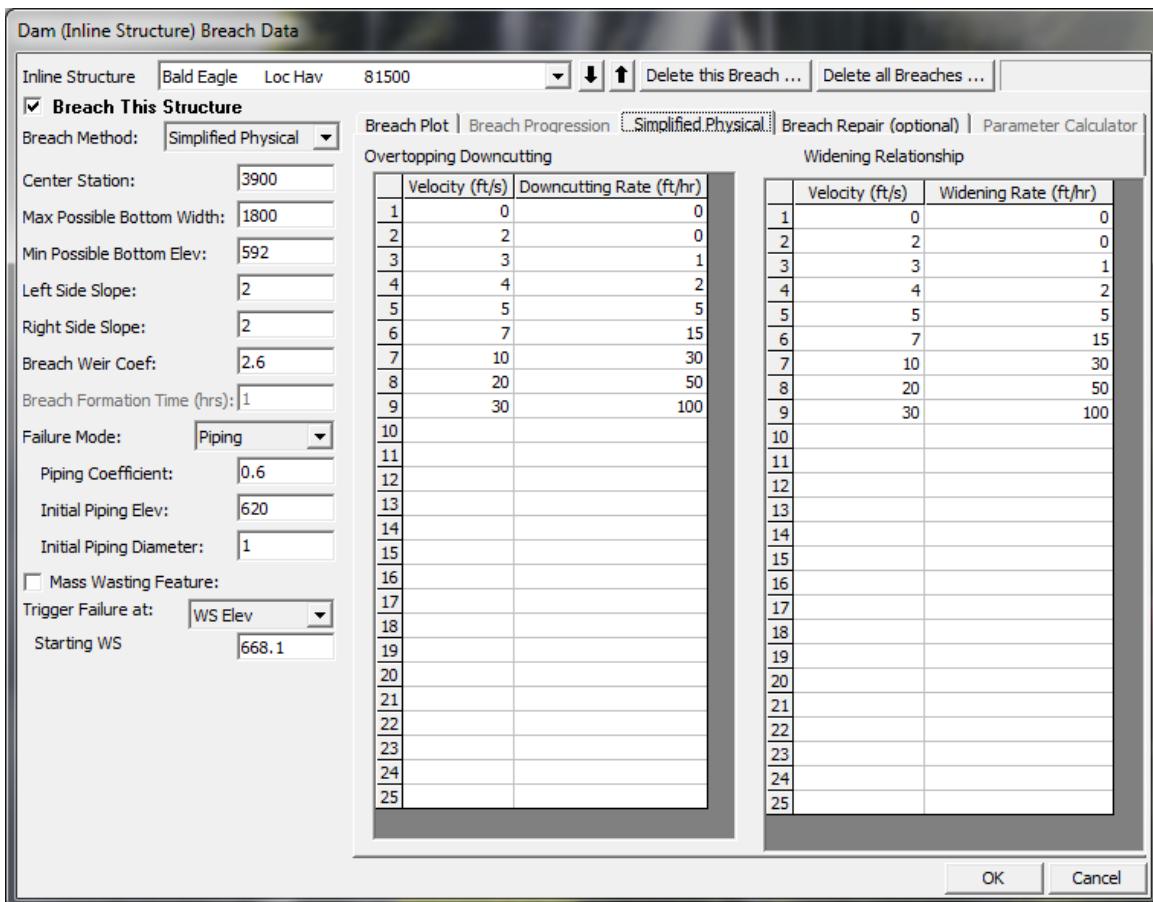


Figure 7 61. HEC-RAS Simplified Physical Breach Option.

As shown in Figure 7-61 above, the user is required to enter Velocity versus Down-cutting erosion rates and velocity versus erosion widening rates. This data is often very difficult to come by. User's will need to consult with Geotechnical engineers to come up with reasonable estimates of this data for your specific Levee or Dam. Another way to estimate this information is to try to derive it by simulating a historic Levee or Dam breach, and adjusting the velocity versus erosion rate data until the model simulates the correct breach width and time. This is obviously an iterative process, and may require the user to perform this at multiple locations to see if there is a consistent set of erosion rates that will provide a reasonable model for simulating Levee breaches (or Dams) in your geographical area.

We realize that this data is not readily available for any specific levee or dam. The hope is that over time we will be able to develop guidelines for these erosion rates based on analyzing historical levee and dam breaches.

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 over top 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 7-62.

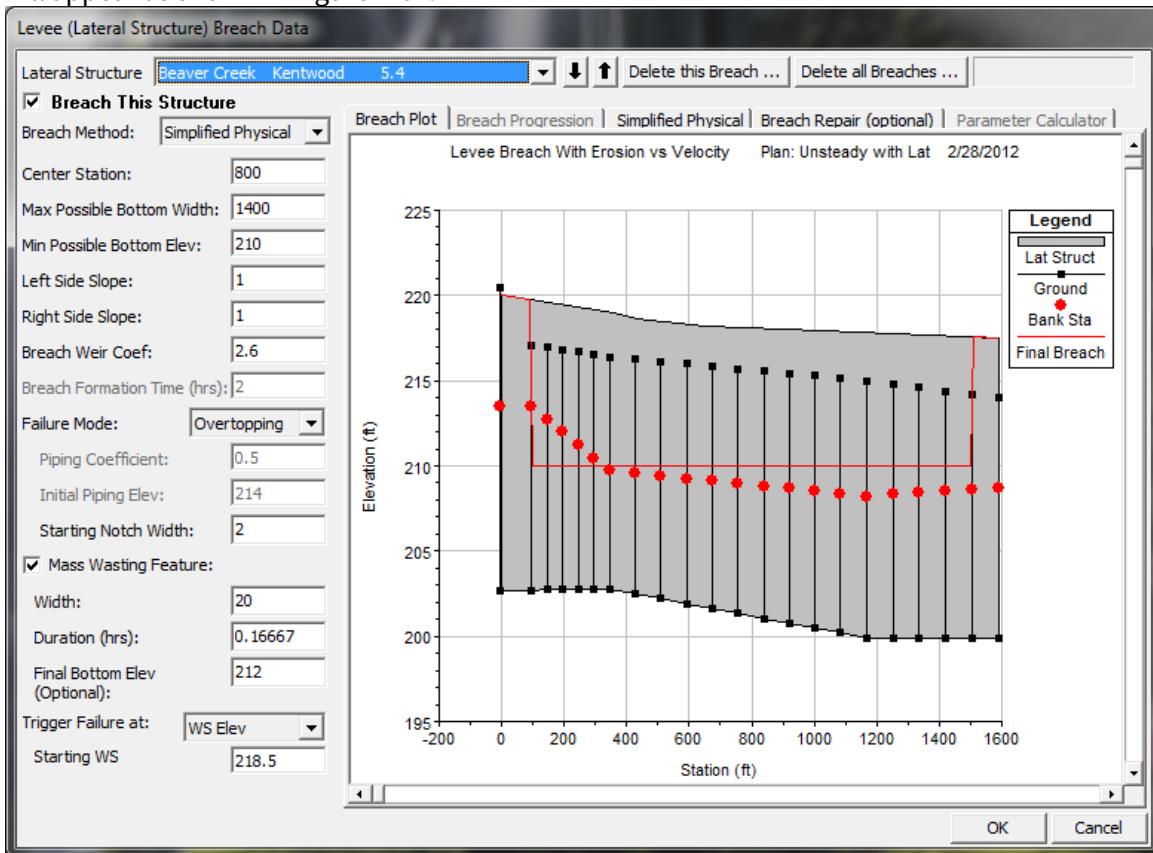


Figure 7-62. Levee Breaching Editor

As shown in Figure 7-62, 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 14 of this manual.

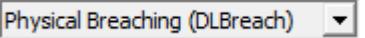
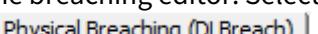
DL Breach

DL Breach is a physical breaching algorithm developed by Dr. Weiming Wu ([Wu, 2016](#), [Wu, 2013](#)). Dr. Wu collaborated with HEC to include this method in HEC-RAS to provide a breaching algorithm that estimates breach development based on physical processes and parameters. DLBreach computes breach development for overtopping and piping failures through cohesive, cohesionless, or composite structures. The HEC-RAS version of DLBreach, combines the breach development algorithms from Wu (2013) with HEC-RAS hydraulics.

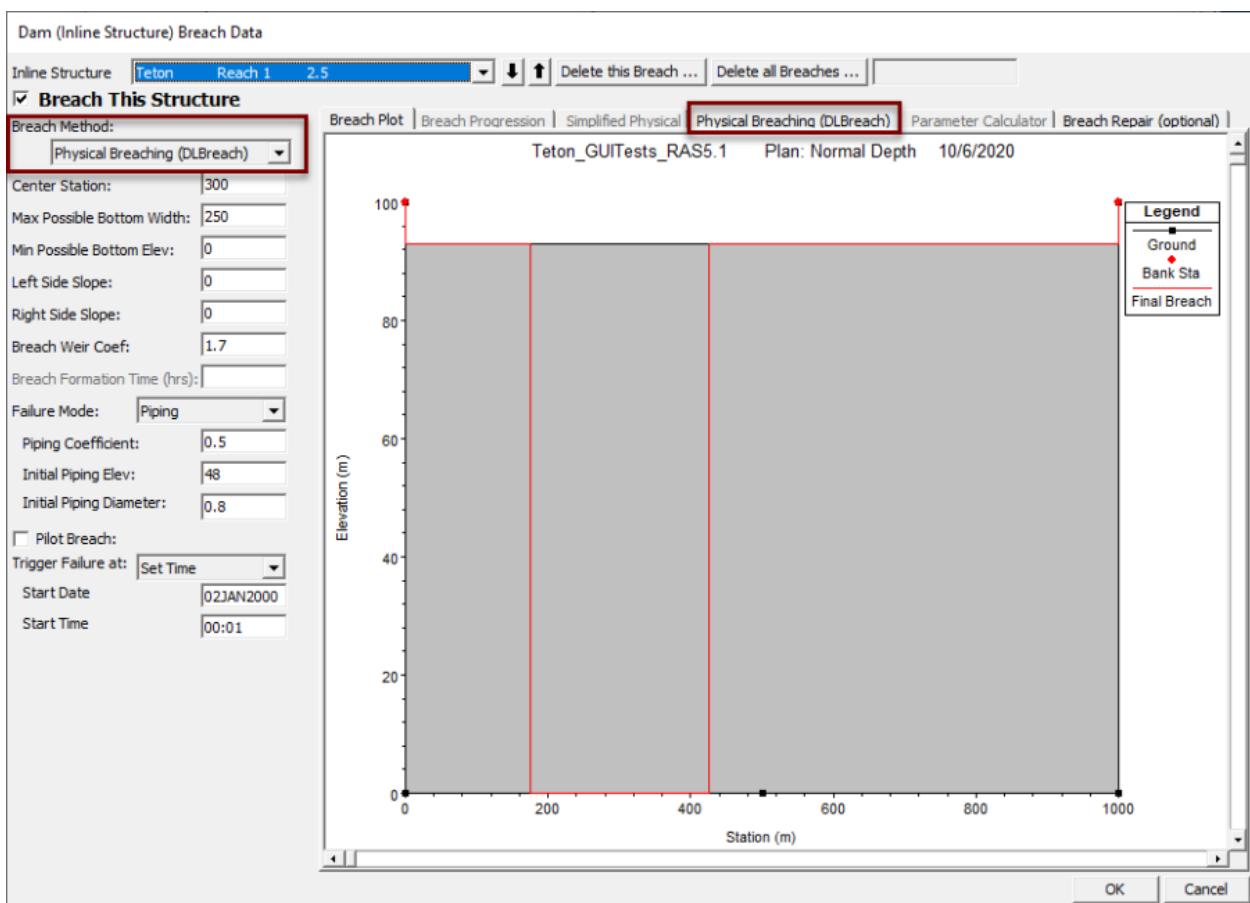
- [DL Breach Acknowledgements and References](#)
- [Transport Parameters](#)
 - [Overtopping Erosion Model](#)
- [DLBreach Data and Parameterization in HEC-RAS](#)

- Standard Breach Data (Left Pane)
- Embankment Parameters

DLBreach Data and Parameterization in HEC-RAS

To use DL Breach, select **Physical Breaching (DLBreach)**  from the **Breach Method** drop down in the top left corner of the breaching editor. Selecting DLBreach will make the **Physical Breaching (DLBreach)** data tab  (See Figure Below). DLBreach requires most of the same data and parameters that the other breach methods require. Enter these data in the left panel of the breach editor. However, some of these values have different definitions or purposes in DLBreach. Define the following parameters and data in the breach editor for DLBreach:

- Standard Breach Data (Left Pane)
- Embankment Parameters



Standard Breach Data (Left Pane)

Center Station

The lateral, cross section station of the breach centerline. The breach will start at this lateral station (e.g. 50 m or the midpoint of the dam in Figure 1).

Max Possible Bottom Width.

The maximum *bottom* width of the final breach.

Note:

The top width will generally be larger, based on the breach side slopes.

Min Possible Bottom Elev.

The model will erode through the dam until it reaches this elevation. This is the vertical limit of the breach. HEC-RAS will not erode to a negative elevation (e.g. for an arbitrary laboratory datum) with this method. Adjust the model datum so the **Min Possible Bottom Elev:** is positive if this value would otherwise be negative.

Note:

The current version of HEC-RAS will not erode below elevation Zero.

Left Side Slope

Define the left side slope of the final breach opening. Side slopes in the breaching editor are a horizontal to vertical (H:V) ratio or the base divided by the height. (Remember, displays in HEC-RAS are vertically distorted, so these slopes may appear steeper in HEC-RAS editors.) For example, a side slope of 2 represents 2 horizontal units for every vertical unit or a side slope that is twice as wide as it is high. Note: the H/V convention in HEC-RAS is opposite the convention in the stand alone version of DLBreach.

Right Side Slope

Define the right side slope of the final breach opening (Horizontal : Vertical – see note in previous entry).

Note: Unlike the other breach algorithms in HEC-RAS DLBreach assumes the right and left side slopes are the same. Therefore, if a DLBreach model defines different breach side slopes, HEC-RAS will average the side slopes and use the average for both the right and left side slopes in DLBreach.

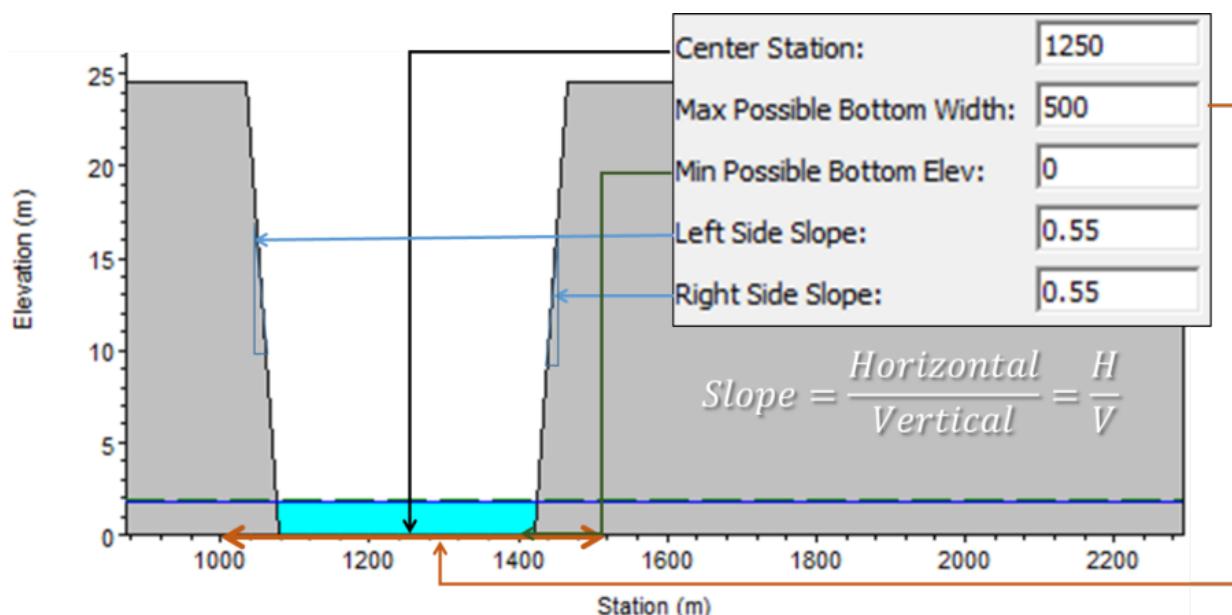


Figure 2: Diagram of the breach geometry variables in the left pane. In this example, the breach reached the minimum possible elevation but the widening stopped before it reached the maximum width.

Breach Weir Coeff

Enter the weir coefficient that HEC-RAS will use to compute flow through the breach. HEC-RAS uses the broad crested weir equation to compute flow through the breach for overtopping breaches (or after the piping breach collapses). This is the broad crested weir coefficient for that breach.

Failure Mode

Select the failure mode in this drop-down menu. DLBreach includes the same two failure modes as the other breaching methods in HEC-RAS, Piping or Overtopping. The interface will update based on this choice, activating different data fields appropriate for each method.

Piping Parameters and Data:

A piping failure starts with an enclosed flow channel that forms through the embankment. HEC-RAS simulates this flow channel with uniform diameter pipe and uses the orifice equation to compute flow in the pipe. DLBreach uses cohesive or non-cohesive transport equations to compute pipe sidewall erosion and increases the diameter of the pipe as the flow erodes material. When the pipe grows large enough that it can no longer hold a competent "ceiling" in the breach material, it collapses, and transitions into an overtopping breach phase.

Erosion rate tends to be directly and non-linearly related to flow, making the pipe expand faster at higher pipe flow rates. Therefore, the total breach time can be very sensitive to the initial breach diameter and the pipe flow parameters.

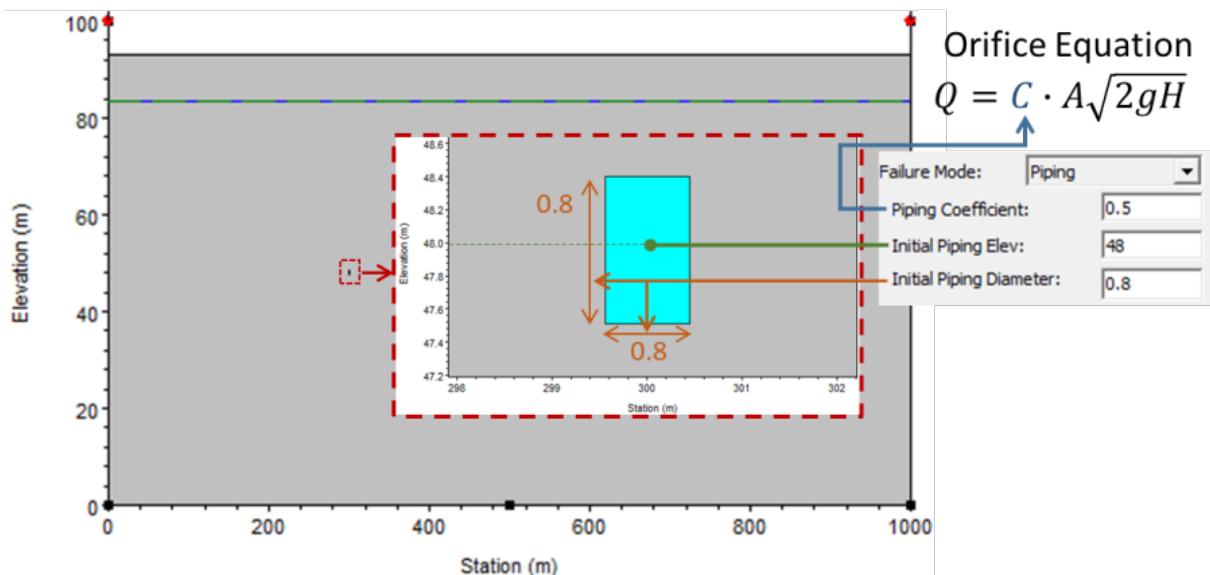


Figure 3: Piping geometry parameters and coefficient.

Piping Coefficient

HEC-RAS computes flow through the pipe – during the piping phase of a breach – with the orifice equation. The orifice equation has a coefficient that regulates the flow efficiency through the pipe. A Piping Coefficient of 1 would be completely efficient. The HEC-RAS default for a piping breach is 0.5. Good data on the conduit efficiency of piping failures are not available. But it is likely that these natural conduits are less efficient than the straight, smooth pipes used to simulate them. So reducing the flow efficiency with this coefficient is appropriate. This coefficient will regulate flow through the piping failure and will affect the breach time. It is also one of the most significant difference between DLBreach in HEC-RAS and the stand alone DLBreach equation.

Initial Piping Elev

Define the center elevation of the piping conduit. HEC-RAS and DLBreach assume that the piping conduit is horizontal (only has one elevation) and rectangular. As the conduit expands, it grows symmetrically from this initial elevation. The **Initial Piping Elevation** will be the center of the piping breach until the pipe ceiling collapses and it transitions to an overtopping breach.

Initial Piping Diameter

Define the initial "diameter" for the piping conduit. Piping conduits in HEC-RAS are square. Therefore, the "diameter" is actually the width and the height of the piping conduit. Piping failures progress slowly at small diameters (with smaller flows and lower erosion rates), making the breach time sensitive to this parameter.

Overtopping Parameters and Data:

Pilot Breach (aka Mass Wasting Feature) (Overtopping Failure Only). The other breaching functions in HEC-RAS have a "Mass Wasting" feature, which simulates a pressure build up behind the embankment that "blows out" a chunk of the embankment quickly. DLBreach does not use the mass wasting model. However, overtopping breaches in DLBreach require a "Pilot Breach," a starting notch that guides the overtopping erosion. HEC-RAS uses the Mass Wasting interface to define the pilot breach for DLBreach.

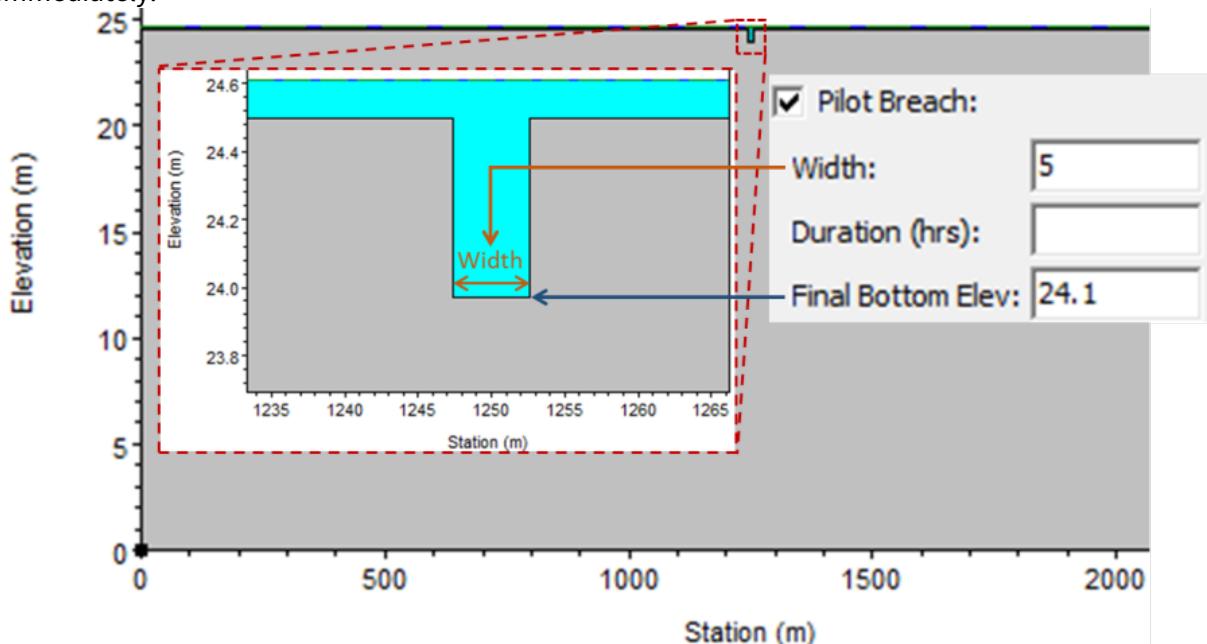
The **Pilot Breach** option requires a **Width** and **Bottom Elevation**.

Note:

The bottom elevation convention is different than the Pilot Channel Depth convention in DLBreach.

Duration

The **Duration** is optional. If the **Duration** is zero or blank, the full **Pilot Breach** will form immediately.



Trigger Failure At

HEC-RAS has three overtopping breach-initiation options. Because DLBreach starts simulations once the breach begins, HEC-RAS users can select any of these three options to initiate a physical breach simulation just like they would for the other breach methods. The three methods allow users to initiate the breach:

- at a specific time (**Set Time**),
- when the water reaches a specific a water surface elevation (**WS Elev**), or
- after the water surface has exceeded a specified water surface elevation for a specified period of time (**WS Elev + Duration**) – with the option to add a second elevation that triggers immediate failure (regardless of duration).

Starting WS

If the water surface trigger elevation (**WS Elev**) is selected, enter the trigger a water surface elevation into this field. HEC-RAS monitors the water surface elevation at the breach **Center Station** the user enters. Dam breaches have horizontal water surface elevations, so the water surface elevation is the same across the embankment. But levees have sloped water surfaces, making the water surface site-specific.

WS Elev+Duration

The **WS Elev+Duration trigger** mechanism monitors the time that the water on the embankment exceeds a threshold and triggers the failure when the water exceeds that elevation for a specified

| | |
|---|-------------------------|
| Trigger Failure at: | WS Elev+Duration |
| Threshold WS | 375 |
| Duration Above Threshold | 24 |
| Immediate Initiation WS | 381 |
| <input checked="" type="checkbox"/> Accumulate Duration | |

time. This option has four additional fields or options:

Threshold WS

HEC-RAS will monitor the time that the water surface exceeds this water surface elevation at the breach **Center Station**. If that time exceeds the **Duration Above Threshold** (next field), HEC-RAS will trigger the breach and start the DLBreach simulation.

Duration Above Threshold:

This is a time in hours. If the water surface elevation at the breach **Center Station** exceeds the **Threshold WS** for the **Duration Above Threshold**, DLBreach will initiate the breach.

Immediate Initiation WS:

The **WS Elev+Duration** option supports a failure mechanism where a moderate water surface saturates and loads an embankment long enough to cause a failure. However, the same embankment might fail immediately if the water surface gets high enough. The Immediate Initiation WS initiates a breach, without waiting for the **Duration Above Threshold**, if the water surface at the breach **Center Station**, reaches this (higher) water surface.

three additional fields of data to enter. The first variables **Threshold WS**. This variable is the water.

Set Time

If the user selects the **Set Time** option, then a starting date and time to initiate the breach must be

entered.

Dam (Inline Structure) Breach Data

Inline Structure | Teton | Reach 1 | 2.5 | | | |

Breach This Structure

Breach Method: Physical Breaching (DLBreach)

Center Station: 300
Max Possible Bottom Width: 250
Min Possible Bottom Elev: 0
Left Side Slope: 0
Right Side Slope: 0
Breach Weir Coef: 1.7
Breach Formation Time (hrs):
Failure Mode: Piping
Piping Coefficient: 0.5
Initial Piping Elev: 48
Initial Piping Diameter: 0.8
 Pilot Breach:
Trigger Failure at: Set Time
Start Date: 02JAN2000
Start Time: 00:01

Breach Plot | Breach Progression | Simplified Physical | Physical Breaching (DLBreach) | Parameter Calculator | Breach Repair (optional) | Model a cover layer

Embankment Geometry:

| | |
|-------------------|--------------|
| Embankment Width: | 10.5 (m) |
| Slope (H:B) | Roughness: |
| US Slope: | 0.3333 0.016 |
| Flat Top: | 0.016 |
| DS Slope: | 0.4 0.016 |

Soil Parameters:

| | |
|--------------------|----------------|
| Soil Type: | Cohesive |
| Sediment Diameter: | 0.00003 (mm) |
| Porosity: | 0.3 (0.0-1.0) |
| Specific Gravity: | 2.65 |
| Clay Content: | 0.3 (0.0-1.0) |
| Cohesion: | 25000. (Pa) |
| Friction Angle: | 0.65 (degrees) |

Erosion Model (Overtopping Only):

| | |
|------------------------|---------------------------|
| Surface Erosion | |
| Critical Shear Stress: | 0.15 (Pa) |
| Erodibility (kd): | 8. (cm ³ /N-s) |
| Adaptation λ: | |

Clay Cover and Core Parameters:

| | | |
|------------------------|--------------|------------------------|
| Parameters | Cover | Core |
| Core Height: | | (m) |
| Core Crest Width: | | (m) |
| Core Center Location: | | (m) |
| Core US Slope: | | |
| Core DS Slope: | | |
| Core Manning n: | | |
| Sediment Diameter: | | (mm) |
| Porosity: | | (0.0-1.0) |
| Specific Gravity: | 2.65 | |
| Clay Content: | | (0.0-1.0) |
| Cohesion: | | (Pa) |
| Friction Angle: | | (degrees) |
| Soil Type: | Cohesionless | |
| Critical Shear Stress: | | (Pa) |
| Erodibility (kd): | | (cm ³ /N-s) |
| Top Thickness: | | (m) |
| US Slope Thickness: | | (m) |
| DS Slope Thickness: | | (m) |

Breach Direction: One Way

Embankment Parameters

Enter data and parameters specific to DLBreach the remaining embankment parameters in the **Physical Breaching (DLBreach)** tab.

Embankment Geometry

The first few entries in the **Physical Breaching (DLBreach)** tab continue to define the embankment geometry, with parameters the other methods do not need.

Embankment Width

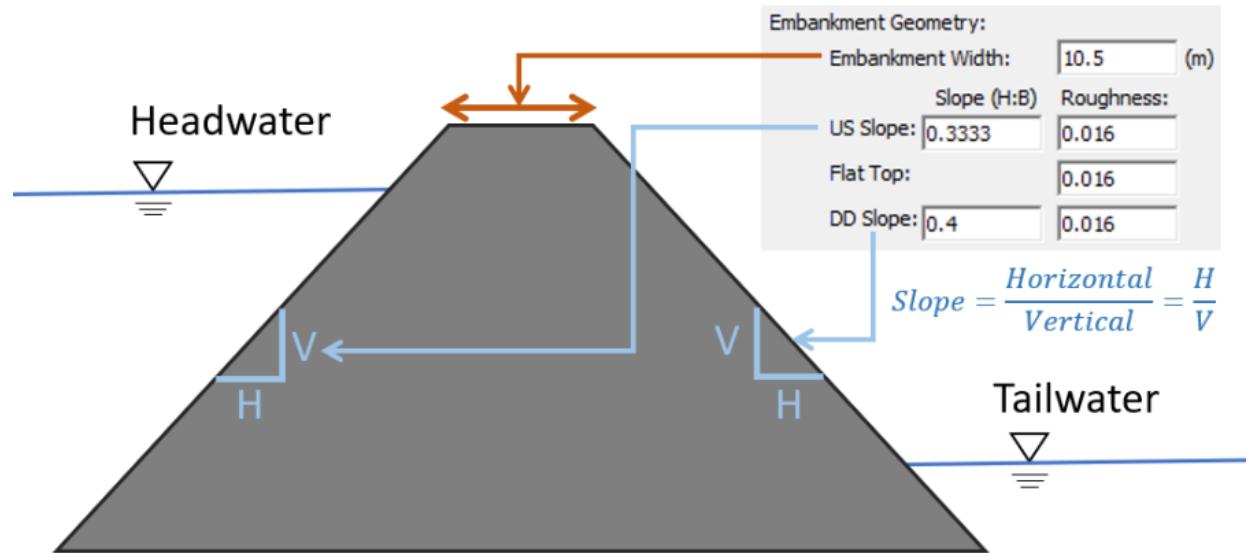
The embankment width is the thickness of the (flat) top of the embankment between the sloping sections. In an **Inline Structure** (dam) this is the a longitudinal distance (along the stream) and in a **Lateral Structure** (e.g. levee) this distance is lateral to the main flow direction

US Slope

This field is used to enter the slope of the embankment on the upstream side. The slope is entered in values representing the horizontal to vertical ratio. For example, a value of 2 represents 2 units moved horizontally for every 1 unit moved vertically. Note the H/V convention in HEC-RAS is opposite of the slope convention in the stand alone DLBreach

DS Slope

This field is used to enter the slope of the embankment on the downstream side. The slope is entered in values representing the horizontal to vertical ratio. For example, a value of 2 represents 2 units moved horizontally for every 1 unit moved vertically.



Breach Roughness

This field is used to define the Manning's Roughness Coefficient n for the breach. The breach is divided into three segments: upstream slope, flat top, and downstream slope. The user can have the same or different Manning's n values for the three segments.

Embankment Soil

| | |
|-------------------------|----------------|
| Soil Parameters: | |
| Soil Type: | Cohesive |
| Sediment Diameter: | 0.00003 (mm) |
| Porosity: | 0.3 (0.0-1.0) |
| Specific Gravity: | 2.65 |
| Clay Content: | 0.3 (0.0-1.0) |
| Cohesion: | 25000. (Pa) |
| Friction Angle: | 0.65 (degrees) |

| | |
|-------------------------|----------------|
| Soil Parameters: | |
| Soil Type: | Cohesionless |
| Sediment Diameter: | 0.0002 (mm) |
| Porosity: | 0.35 (0.0-1.0) |
| Specific Gravity: | 2.65 |
| Clay Content: | 0 (0.0-1.0) |
| Cohesion: | 0 (Pa) |
| Friction Angle: | 0.37 (degrees) |

Soil Type

Specify the soil type of the embankment (cohesive or cohesionless).

Cohesive Soils:

Have enough (<0.063 mm) silt and clay (0.004 mm) content that the particles "stick" together. The surface area-to-mass ratio for these particles is so large that the electro-chemical forces between particles is larger than the frictional or gravitational forces.

If the embankment is cohesionless, DLBreach will use a non-equilibrium transport approach to erode .

Both cohesionless and cohesive soils, the following parameters must be entered: **Sediment Diameter, Porosity, Specific Gravity, Clay Content, Cohesion, and Friction Angle.**

All Material Types

Sediment Diameter (mm)

Enter the median sediment diameter of the core (d_{50}).

Porosity (decimal fraction):

Define the porosity of the embankment material. The porosity is the fraction of voids and should be entered as a **decimal** (*not percent*). So if the soil porosity is 30%, enter **0.3**.

Specific Gravity (unitless):

Enter the specific gravity of the embankment material. The default is 2.65, which is appropriate for almost all embankment materials.

Clay Content (decimal fraction):

Enter the clay content of the embankment **if the Soil Type is Cohesive**. Like porosity, enter the value as a **decimal** between 0.0 and 1.0 (where 0.0 is 0% and 1.0 is 100%). Do Not enter this parameter as a percent.

Cohesion (Stress Units – lbf/ft² or Pa):

Enter the cohesion of the embankment material if the **Soil Type is Cohesive**.

Friction Angle (degree)

This field is used to enter the internal friction angle of the core material. The soil friction angle can be estimated according to soil types. The [average friction angles](#) are usually between:

| Soil Type | Range |
|-----------|---------|
| Gravel | 34°-40° |
| Sand | 32°-38° |
| Silt | 24°-33° |
| Clay | ~22° |

Wu (2016) provides the following guidance on these parameters:

"The actual friction angle may vary by ±3°–6° from the average value, because it is also affected by other properties of the soil, such as moisture, clay fraction, chemical composition, and compaction. Once the friction angle is estimated, the soil cohesion can thus be determined by using the known final breach slope."

"Furthermore, the soil cohesion and friction angle are important factors for the stability of headcut, pipe top block, and clay core, and in turn influence the breach peak discharge, breach width and failure time. Therefore, the estimated soil cohesion and friction angle can be validated indirectly by comparing these calculated breach properties against measured data."

Transport Parameters

The last three variables, the adaptation λ , critical shear stress, and erodibility are transport parameters. Both the overtopping and piping use these transport parameters. Piping uses cohesionless transport (and λ) or excess shear for cohesives (which uses critical shear stress and erodibility). Overtopping (including the post-collapse phase of piping) also uses these parameters. Surface erosion uses λ for cohesionless and critical shear/erodibility for cohesive, and the headcut algorithms use critical shear and erodibility to determine the vertical erosion rates.

λ is part of the non-equilibrium transport used for the cohesionless erosion model. Therefore DL Breach only requires λ if it applies the transport model to a cohesionless embankment. Critical shear stress and erodibility – on the other hand – are cohesive parameters of the excess shear equation. They are only required for cohesive embankments (or cohesive core materials, which are defined separately, under the **Core** column).

Cohesionless Transport Parameters

Adaptation λ

Cohesive Transport Parameters

Critical Shear Stress (τ_c)

Erodibility (k_d)

Adaptation λ : (unitless)

Non-equilibrium sediment transport models use an *adaptation length* to compute the rate of erosion. A non-equilibrium model does not just erode the computed capacity but takes time to entrain the full capacity. It uses an adaptation length (L_s) to control this temporal lag, such that:

$$\frac{\partial(AC_t)}{\partial t} + \frac{\partial(QC_t)}{\partial x} = -\frac{Q}{L_s} (C_t - C_{t^*})$$

L_s Adaptation Length

Where A is the cross section flow area in the breach channel, Ct is the transported sediment concentration, C_{t^*} is the transport capacity (or, strictly, the equilibrium concentration under the flow conditions for the sediment parameters) and Q is the flow through the breach.

Instead of requiring users to define an adaptation length (which can be an abstract parameter that can be difficult to estimate) DLBreach makes L_s a function of the breach channel width. So users define a linear multiplier (λ) such that:

$$\text{Adaptation Length} = \lambda * \text{Breach Channel Width}$$

Wu (2016) recommends:

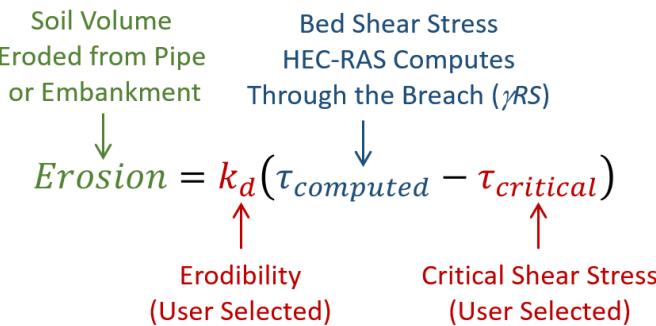
$$\lambda = \begin{cases} 6.0, & \text{for field cases} \\ 3.0, & \text{for lab cases} \end{cases}$$

HEC-RAS defaults to 6.0 based on this recommendation.

| | |
|------------------------|---|
| Critical Shear Stress: | <input type="text"/> (Pa) |
| Erodibility (kd): | <input type="text"/> (cm ³ /N-s) |
| Adaptation λ: | <input type="text"/> 6. |

Critical Shear Stress (Stress Units – lbf/ft² or Pa):

If the **Soil Type** is Cohesive, DLBreach uses the excess shear stress to erode the pipe or embankment. The excess shear stress requires two user parameters, Critical Shear Stress and erodibility, to compute erosion from the pipe or embankment:



These parameters are difficult to measure and highly variable. However, Wu et al. (2013) constrained the critical shear stress for breach calculations. He calibrated 30 different breaches by setting critical shear stress to 0.15 Pa (US Customary) and adjusting Erodibility (see next), and recommends that approach to these parameters.

| | |
|------------------------|--|
| Critical Shear Stress: | <input type="text"/> 0.15 (Pa) |
| Erodibility (kd): | <input type="text"/> 10.3 (cm ³ /N-s) |
| Adaptation λ: | <input type="text"/> |

Erodibility (kd)

If the **Soil Type** is Cohesive, then the user must use this field to enter an erodibility coefficient for core material. Wu et al (2013) calibrated 30 breaches by setting critical shear stress and adjusting the erodibility. The calibrations (some of which were also sensitive to initial pipe diameter and other, interrelated model variables) required erodibilities that varied between 2.5 and 30 cm³/N-s (US Customary).

✓ Note:

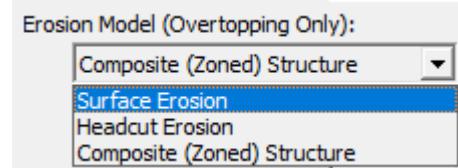
Erodibility Unit Diversity in HEC-RAS Erodibility has a variety of units, and it has different units in different places in HEC-RAS. The cohesive editor of the sediment transport module has units of mass/area/time (e.g. kg/m³/hr or lb/ft³/hr). BSTEM has units of Volume/Force-time (m³/N-s or ft³/lbf-s).

Overtopping Erosion Model

DLBreach can compute dam and levee breaches with a wide range of process and failure modes. Figure 4 organizes the different failure processes that DLBreach simulates into a flow chart, where the nodes represent user decisions. Figure 5 replicates this process flow chart with the actual

interface options that direct these decisions in the model.

The user decisions that direct how DLBreach computes the embankment failure include the **Failure Mode** (Piping or Overtopping), the **Soil Type (Cohesive or Cohesionless)** and, finally, the **Erosion Model**.



The DLBreach **Erosion Model determines which process and equations (surface, composite or headcut) DLBreach will use to erode the embankment.**

These are [overtopping algorithms](#), and are not used in the While DLBreach is in a piping mode. However, DLBreach will switch to these routines after the piping conduit collapses in a piping breach, so they are required for either failure mode. must be selected and the data corresponding to the selected erosion model must be entered for the breaching analysis. For all overtopping Erosion Models, users are required to enter Embankment Sediment Parameters and users have the option to Model a cover layer.

DLBreach includes three overtopping Erosion Models:

- Surface Erosion
- Headcut Erosion
- Composite (Zoned) Structure

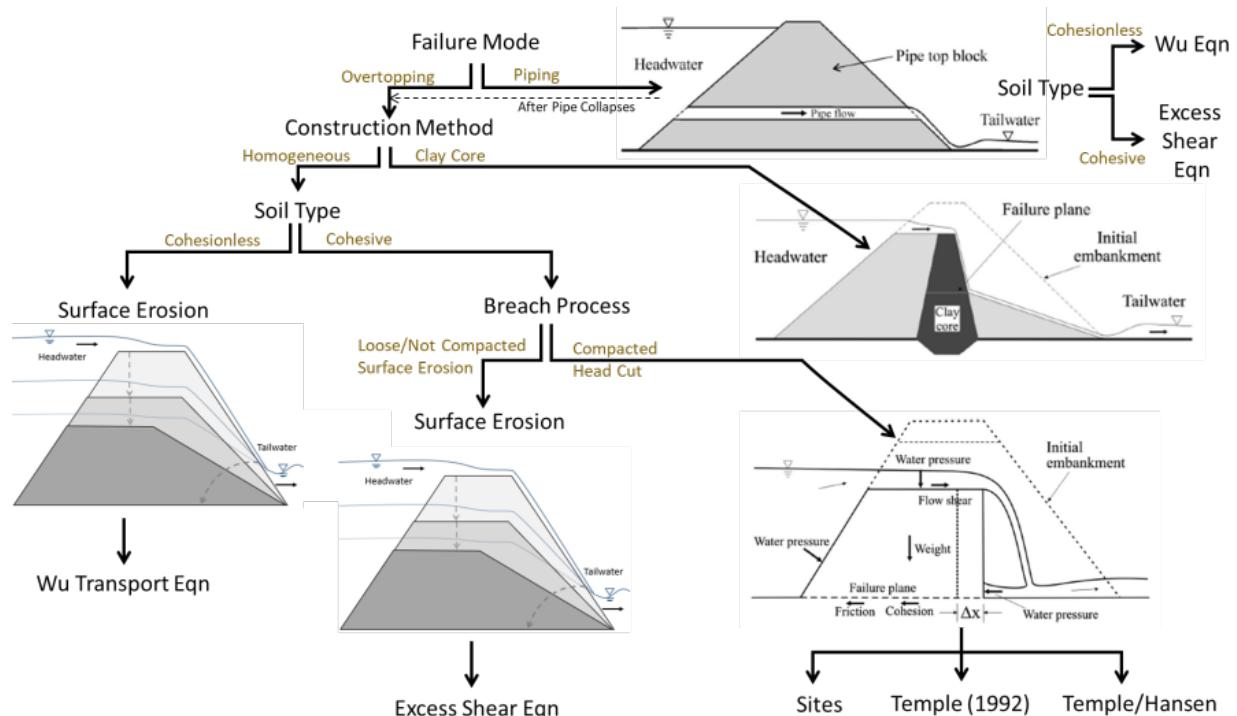


Figure: Flow chart of the different failure models included in DLBreach. The interface options used to select these are included in a similar flow chart in the next Figure).

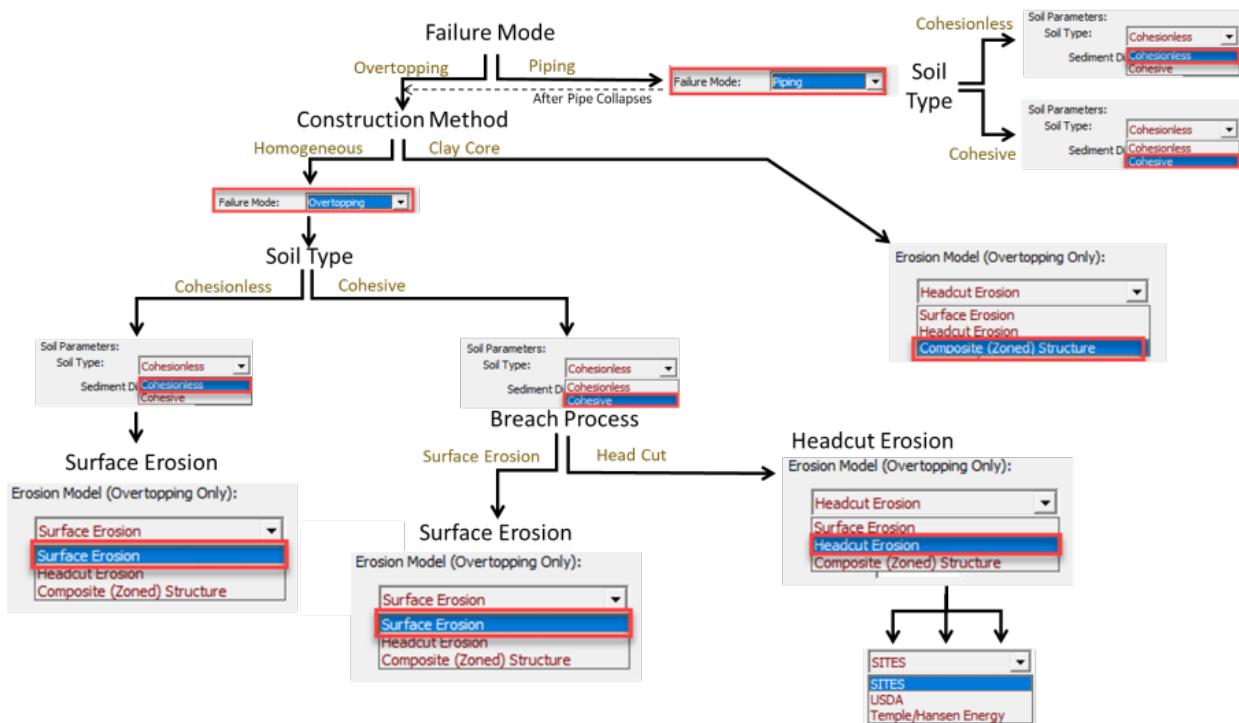


Figure: Flow Chart of HEC-RAS options to activate the different failure modes, processes, and algorithms from the previous flow chart.

Surface Erosion Model

Surface Erosion is appropriate for non-cohesive embankments or loose cohesive embankments without compaction. Therefore, HEC-RAS allows **Surface Erosion** with cohesive or cohesionless methods, but compacted, cohesive embankments should use the **Headcut Erosion** method (see next section). This method erodes sediment from the downstream slope (in a one-way breach), rotating the downstream slope around the downstream toe to account for the sediment eroded with the transport (cohesionless) or excess shear (cohesive) equation. At the same time, DLBreach lowers the flat-top weir representing the breach.

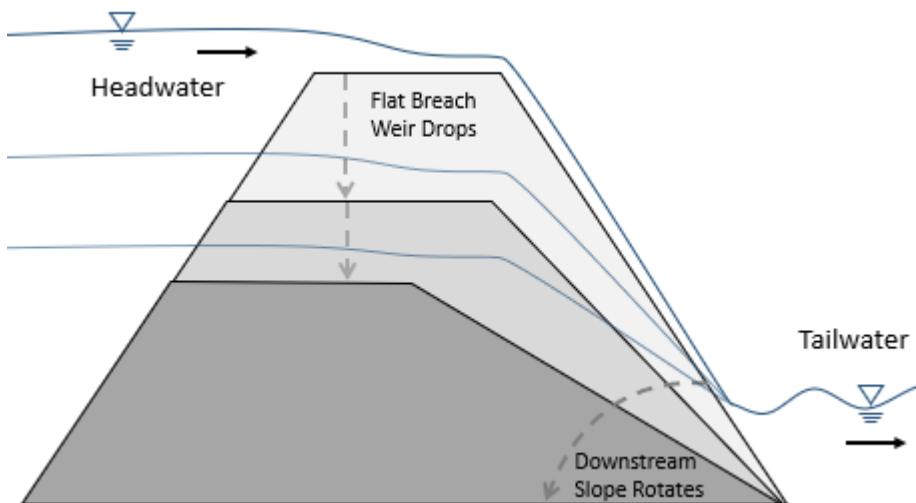


Figure: Conceptual diagram of three sequential stages of a one-way Surface Erosion breach in DL Breach.

If the Surface Erosion model is selected, the model will not require additional data.

Headcut Erosion Model

Most breach experiments and field observations of actual breaches included a headcut phase. As water flows over the breach, it erodes the top of the embankment. But the flow over the breach also forms a plunge pool on the downstream end of the embankment (the Figure below), which attacks the downstream toe of the embankment, pushing the headcut upstream. DLBreach computes the headcut by idealizing the downstream face of the breach with a vertical face, and computing the rate that this vertical, downstream, embankment face translates upstream. By eroding the berm in two directions (vertically and upstream) DLBreach estimates the impact of these two interacting processes (observed in actual breaches), which adds a non-linear effect accelerating the breach. When the headcut advances to the upstream slope, the embankment crest vanishes, and the breach flow increases significantly until the upstream water surface falls.

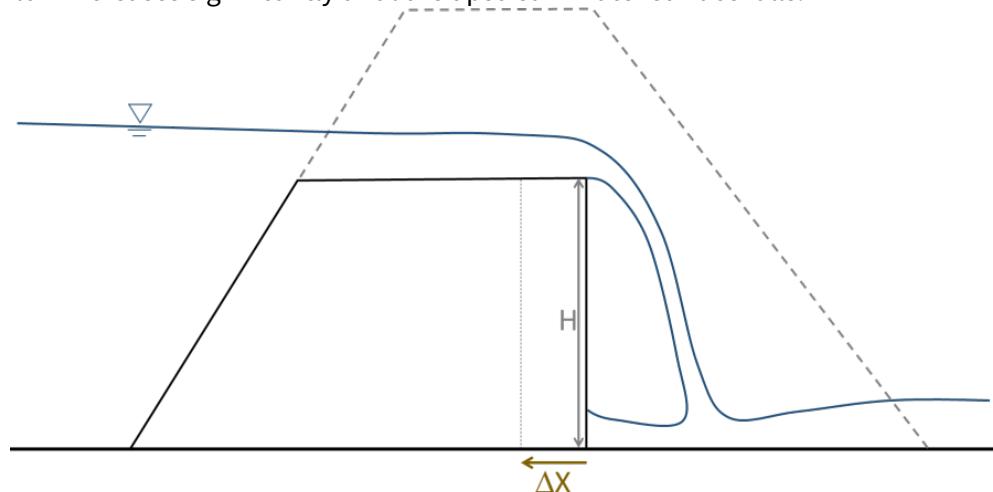


Figure: Idealized headcut geometry in DLBreach. The headcut equations compute the rate of upstream erosion, which pushes the vertical downstream face of the embankment upstream by a computed ΔX .

If modelers selects **Headcut Erosion**, the Headcut Method options will appear in the interface below the transport parameters.

| | |
|--|-----------------------------|
| Erosion Model (Overtopping Only): | |
| Headcut Erosion | |
| Critical Shear Stress: | 0.15 (Pa) |
| Erodibility (kd): | 10.3 (cm ³ /N·s) |
| Adaptation %: | |
| Headcut Method: | Temple (1992) |
| CT | 0.0049 |

DLBreach includes three methods to compute the headcut migration. Select the method from the dropdown box next to the **Headcut Method**. HEC-RAS will populate one or two data fields for the parameters that these methods require. The three headcut methods available in DLBreach are:

- SITES (USDA-NECS, 1997)

- Temple (1992)
- Temple/Hansen Energy (Temple et al. 2005)

These three equations have similar forms. They all compute the rate of the headcut migration (X/t) as a function of the product of the square or cube root of the unit flow (q) through the breach and the head drop (H) over the breach, so that:

$$\frac{\Delta X}{\Delta t} \propto C^m \sqrt{q} \sqrt[n]{H}$$

Where m and n are 2 or 3 and C is a linear, user-defined coefficient.

Temple (1992)

The 1992 Temple equation computes head cut migration the square root of the breach height and the cube root of the unit flow. Define CT to use Temple (1992) to compute headcut migration.

| | |
|-----------------|---------------|
| Headcut Method: | Temple (1992) |
| CT | 0.0049 |

$$\Delta X = C_T q^{\frac{1}{3}} H^{\frac{1}{2}} \Delta t$$

Temple et al. (2005)

The subsequent 2005 version of this equation is similar, except it raises both unit flow and breach height to the cube root. Therefore, the empirical coefficient has different units and behavior. Define C2 to compute headcut migration with this method.

| | |
|-----------------|----------------------|
| Headcut Method: | Temple/Hansen Energy |
| C2 | |

$$\Delta X = C_2 (qH)^{\frac{1}{3}} \Delta t$$

SITES (USDA-NECS, 1997)

The SITES method has the same cube-root form as Temple et al (2005), but adds a threshold, computing headcut migration as the difference between the cube root of the qH product and a user defined threshold. This method requires this threshold A_o and, a linear coefficient (C_1) similar to the other equations (with the same units as C_2 , but different behavior because of the threshold).

| | |
|-----------------|-------|
| Headcut Method: | SITES |
| C1 | |
| Ao | |

$$\Delta X = C_1 \left[(qH)^{\frac{1}{3}} - A_o \right] \Delta t$$

Composite (Zoned) Structure Model

Some embankments include resistant, internal cores that affect breaching processes. These cores can include internal clay, steel or concrete cores or a concrete floodwall on the crest. DLBreach includes algorithms to account for these composite structures. Breaching composite structures is a third overtopping mechanism, mutually exclusive with **Surface Erosion** and **Headcut** methods.

To simulate this process, select **Composite (Zoned) Structure** from the **Erosion Model** drop down.

Erosion Model (Overtopping Only): **Composite (Zoned) Structure**

This option activates the **Core** Data on in the right column of the editor.

Breach Plot | Breach Progression | Simplified Physical | **Physical Breaching (DLBreach)** | Parameter Calculator | Breach Repair (optional) |

Embankment Geometry:

| | |
|-------------------|------------|
| Embankment Width: | 4. (m) |
| Slope (H:B) | Roughness: |
| US Slope: 0.3333 | 0.025 |
| Flat Top: | 0.025 |
| DD Slope: 0.4 | 0.016 |

Soil Parameters:

| | |
|--------------------|----------------|
| Soil Type: | Cohesionless |
| Sediment Diameter: | 0.0008 (mm) |
| Porosity: | 0.43 (0.0-1.0) |
| Specific Gravity: | 2.65 |
| Clay Content: | 0 (0.0-1.0) |
| Cohesion: | 0 (Pa) |
| Friction Angle: | 0.55 (degrees) |

Erosion Model (Overtopping Only):

| | |
|-----------------------------|------------------------|
| Composite (Zoned) Structure | |
| Critical Shear Stress: | (Pa) |
| Erodibility (kd): | (cm ³ /N-s) |
| Adaptation λ: | 6. |

Breach Direction: One Way

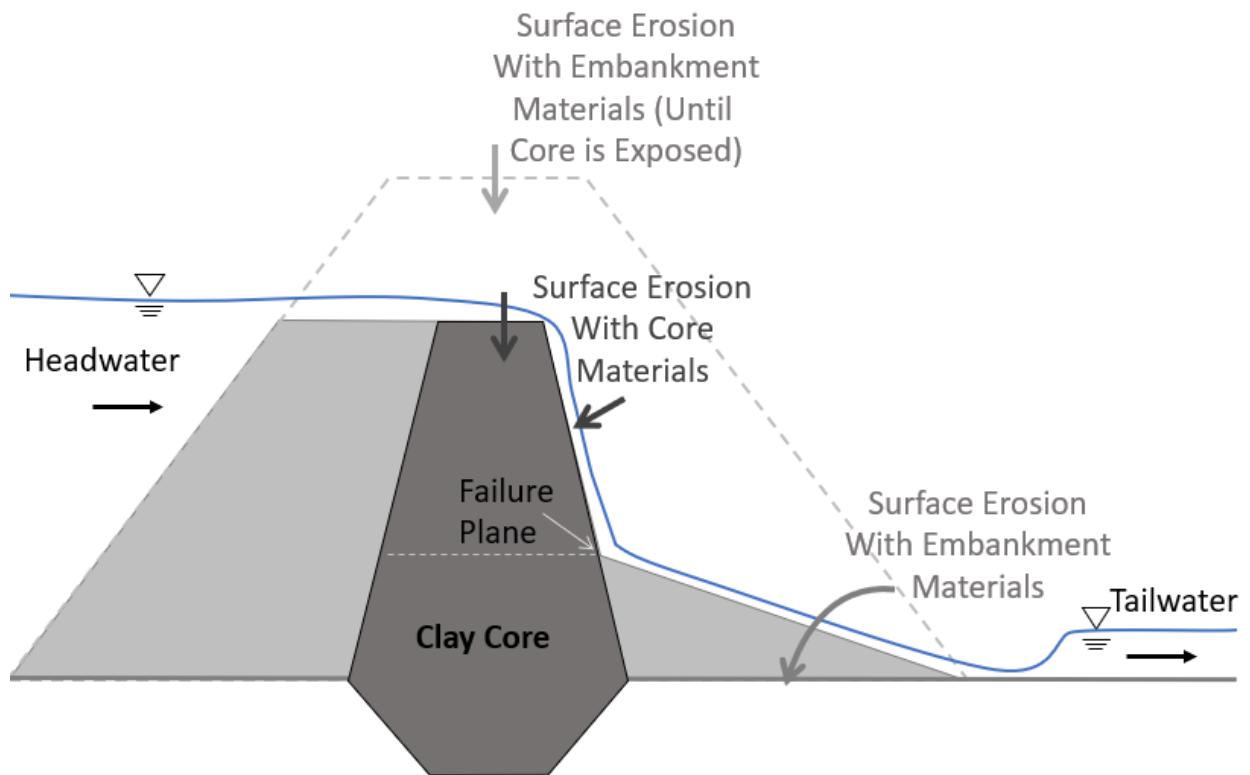
Model a cover layer

Clay Cover and Core Parameters:

| Parameters | Cover | Core |
|------------------------|----------------------------|----------|
| Core Height: | | 5.1 (m) |
| Core Crest Width: | | 0.8 (m) |
| Core Center Location: | | -1.6 (m) |
| Core US Slope: | | 5.88 |
| Core DS Slope: | | 5.88 |
| Core Manning n: | | 0.016 |
| Sediment Diameter: | 0.00003 (mm) | |
| Porosity: | 0.38 (0.0-1.0) | |
| Specific Gravity: | 2.65 | |
| Clay Content: | 0.29 (0.0-1.0) | |
| Cohesion: | 21000. (Pa) | |
| Friction Angle: | 0.325 (degrees) | |
| Soil Type: | Cohesive | |
| Critical Shear Stress: | 0.15 (Pa) | |
| Erodibility (kd): | 3.9 (cm ³ /N-s) | |
| Top Thickness: | | (m) |
| US Slope Thickness: | | (m) |
| DS Slope Thickness: | | (m) |

Figure: Core geometry and soil parameters associated with the Composite Structure Algorithms.

In a one-way breach, the model will use the surface erosion algorithms (described above) to erode the top and downstream face of the embankment, until it encounters the core (see Figure below). DLBreach then computes erosion along the top of the core and the (steeper) downstream slope of the core using the core soil parameters highlighted in Figure 8. The example in Figure 8 uses the cohesionless surface transport equations to erode the top of the embankment and the downstream face until the clay core is exposed. When the clay core is exposed, the example in Figure 8 uses the excess shear equation to erode the cohesive material in the core. Core materials are usually less erodible than the embankment materials, but the downstream slope of the core is also, often, steeper, exposing them to higher shear stresses.



Schematic of the DLBreach approach to one-way breaching for embankments with a clay core.

At some point, however, the core can erode enough, that it becomes unstable, and fails. DLBreach evaluates the core stability each time step by balancing the forces acting on a failure plane. The failure plane is the horizontal line from the place the downstream embankment slope intersects with the core material (see figure above).

If the driving forces on the core exceed the resisting forces, DLBreach immediately removes all of the core material above the failure plane and the upstream embankment soil above the failure plane to simulate the core failure. After a core failure, DLBreach continues to apply the surface erosion equations (cohesionless transport and/or excess shear) to erode the breach.

Core Geometry

DLBreach requires data that define the core geometry. These are in the top section of the core data column and are illustrated in the Figure below. The core geometry fields are described below:

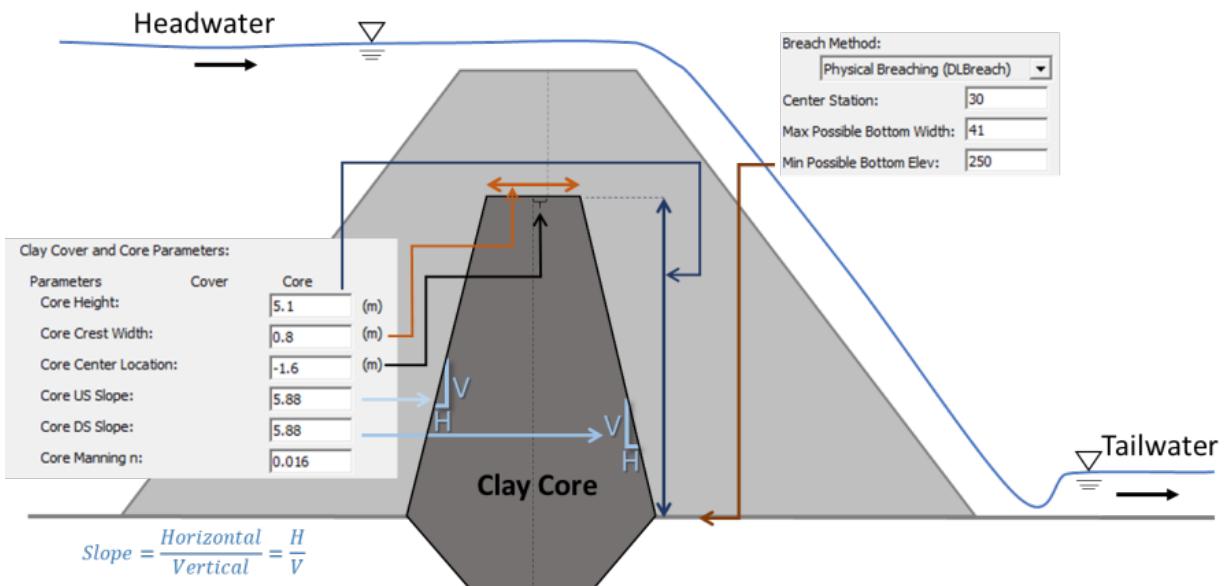


Figure: The core geometry (and some related embankment geometry) data in HEC-RAS.

Core Height: The vertical **distance between the Min Possible Bottom Elev** in the main breach embankment editor (left panel) and the top of the core.

Core Width: The width of the flat top of the core, in the direction of flow.

Core Center Location: This is the centerline station of the core relative to the **Center Station** of the embankment. The stationing is based on the inline structure stationing.

Core US Slope: The upstream slope of the core (horizontal displacement divided by vertical displacement).

Core DS Slope: The downstream slope of the core (horizontal/vertical).

Core Manning n: The manning n value used to compute hydraulic parameters for surface erosion equations on the exposed core.

Core Soil Parameters

The core soil parameters are the same as the embankment soil parameters. See that section above for these parameter descriptions.

Cover

Some dams and levee include cover material that is more-or-less resistant to flow than the embankment material. DLBreach allows a cover layer with any of the erosion models. To turn on the cover layer, check the **Model a cover layer** **Model a cover layer** option to activate the cover fields. The Cover Soil Parameters are the same as the core and embankment soil parameters. See these parameter descriptions above.

Breach Plot | Breach Progression | Simplified Physical | **Physical Breaching (DLBreach)** | Parameter Calculator | Breach Repair (optional) |

Embankment Geometry:

| | | |
|-------------------|------------|-----|
| Embankment Width: | 4. | (m) |
| Slope (H:B) | Roughness: | |
| US Slope: 0.3333 | 0.025 | |
| Flat Top: | 0.025 | |
| DD Slope: 0.4 | 0.016 | |

Soil Parameters:

| | | |
|--------------------|--------------|-----------|
| Soil Type: | Cohesionless | |
| Sediment Diameter: | 0.0008 | (mm) |
| Porosity: | 0.43 | (0.0-1.0) |
| Specific Gravity: | 2.65 | |
| Clay Content: | 0 | (0.0-1.0) |
| Cohesion: | 0 | (Pa) |
| Friction Angle: | 0.55 | (degrees) |

Erosion Model (Overtopping Only):

| | | |
|-----------------------------|----|------------------------|
| Composite (Zoned) Structure | | |
| Critical Shear Stress: | | (Pa) |
| Erodibility (kd): | | (cm ³ /N-s) |
| Adaptation λ: | 3. | |

Breach Direction: One Way

Clay Cover and Core Parameters:

| Parameters | Cover | Core |
|------------------------|----------|----------------------------|
| Core Height: | | 5.1 (m) |
| Core Crest Width: | | 0.8 (m) |
| Core Center Location: | | -1.6 (m) |
| Core US Slope: | | 5.88 |
| Core DS Slope: | | 5.88 |
| Core Manning n: | | 0.016 |
| Sediment Diameter: | 0.00003 | 0.00003 (mm) |
| Porosity: | 0.38 | 0.38 (0.0-1.0) |
| Specific Gravity: | 2.65 | 2.65 |
| Clay Content: | 0.29 | 0.29 (0.0-1.0) |
| Cohesion: | 21000. | 21000. (Pa) |
| Friction Angle: | 0.325 | 0.325 (degrees) |
| Soil Type: | Cohesive | |
| Critical Shear Stress: | 0.15 | 0.15 (Pa) |
| Erodibility (kd): | 3.9 | 3.9 (cm ³ /N-s) |
| Top Thickness: | 0 | 0 (m) |
| US Slope Thickness: | 0 | 0 (m) |
| DS Slope Thickness: | 0.3 | 0.3 (m) |

Figure: Cover material parameters and local thickness.

The three data fields unique to the cover layer are all related to the cover thicknesses. DLBreach allows users to define separate cover thicknesses for the top of the embankment, the upstream slope and the downstream slope. These fields control those initial cover thickness as illustrated in the Figure below.

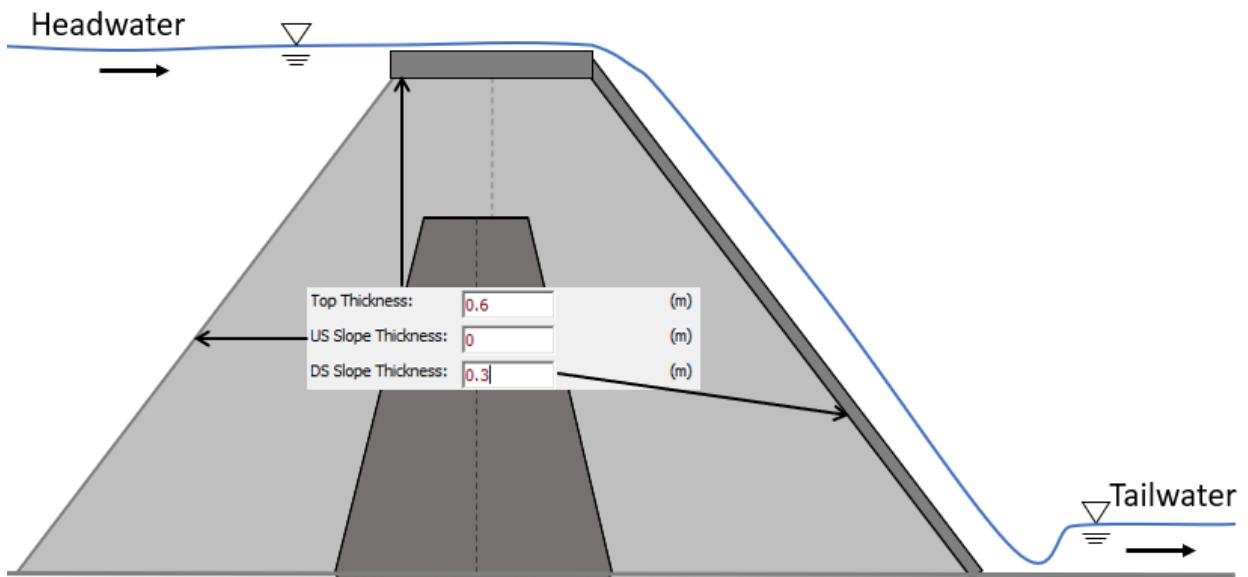


Figure: Cover thicknesses in DL Breach and the HEC-RAS interface.

DL Breach Acknowledgements and References

Acknowledgments

DLBreach was developed by Dr. Weiming Wu (Clarkson University).

References

Wu, W. (2016) *Introduction to DLBreach – A Simplified Physically-Based Dam/Levee Breach Model*, Technical Report, 120 p.

Wu, W. (2013) Simplified Physically Based Model of Earthen Embankment Breaching, ASCE Journal of Hydraulic Engineering, 139(8) .

Conversions:

$$1 \frac{cm^3}{N - s} \frac{1 m^3}{1,000,000 cm^3} \frac{1 ft^3}{0.0283168 m^3} \frac{1 N}{0.2248 lbf} = 1.57E - 4 \frac{ft^3}{lbf - s}$$

$$1 \frac{ft^3}{lbf - s} \frac{1 m^3}{35.317 ft^3} \frac{1,000,000 cm^3}{1 m^3} \frac{1 lbf}{4.44828 N} = 6,367.376 \frac{cm^3}{N - s}$$

$$1 \frac{cm^3}{N - s} \frac{1 m^3}{1,000,000 cm^3} \frac{1 ft^3}{0.0283168 m^3} \frac{1 N}{0.2248 lbf} = 1.57E - 4 \frac{ft^3}{lbf - s}$$

$$1 \frac{ft^3}{lbf - s} \frac{1 m^3}{35.317 ft^3} \frac{1,000,000 cm^3}{1 m^3} \frac{1 lbf}{4.44828 N} = 6,367.376 \frac{cm^3}{N - s}$$

Using HEC-RAS to Compute Ungaged Lateral Inflows

This section of the manual illustrates how to use the ungaged lateral inflow option in HEC-RAS. The ungaged inflow can be used either to recreate historical, lateral inflows, or it can be used in forecast mode to determine the inflows for future downstream routing. This section discusses the required input data and viewing the output. A more technical discussion of the numerical modeling can be found in the hydraulic reference manual.

Observed Stage and Flow Data

After an unsteady RAS project has been calibrated, the first step in developing an ungaged inflow model is to enter the internal observed stage and flow data at the known gage locations. From the Unsteady Flow editor, select the river station that the gage is at, press the **"Add a Boundary Condition Location"** button and then click on **"IB Stage/Flow"** (Figure 7-63).

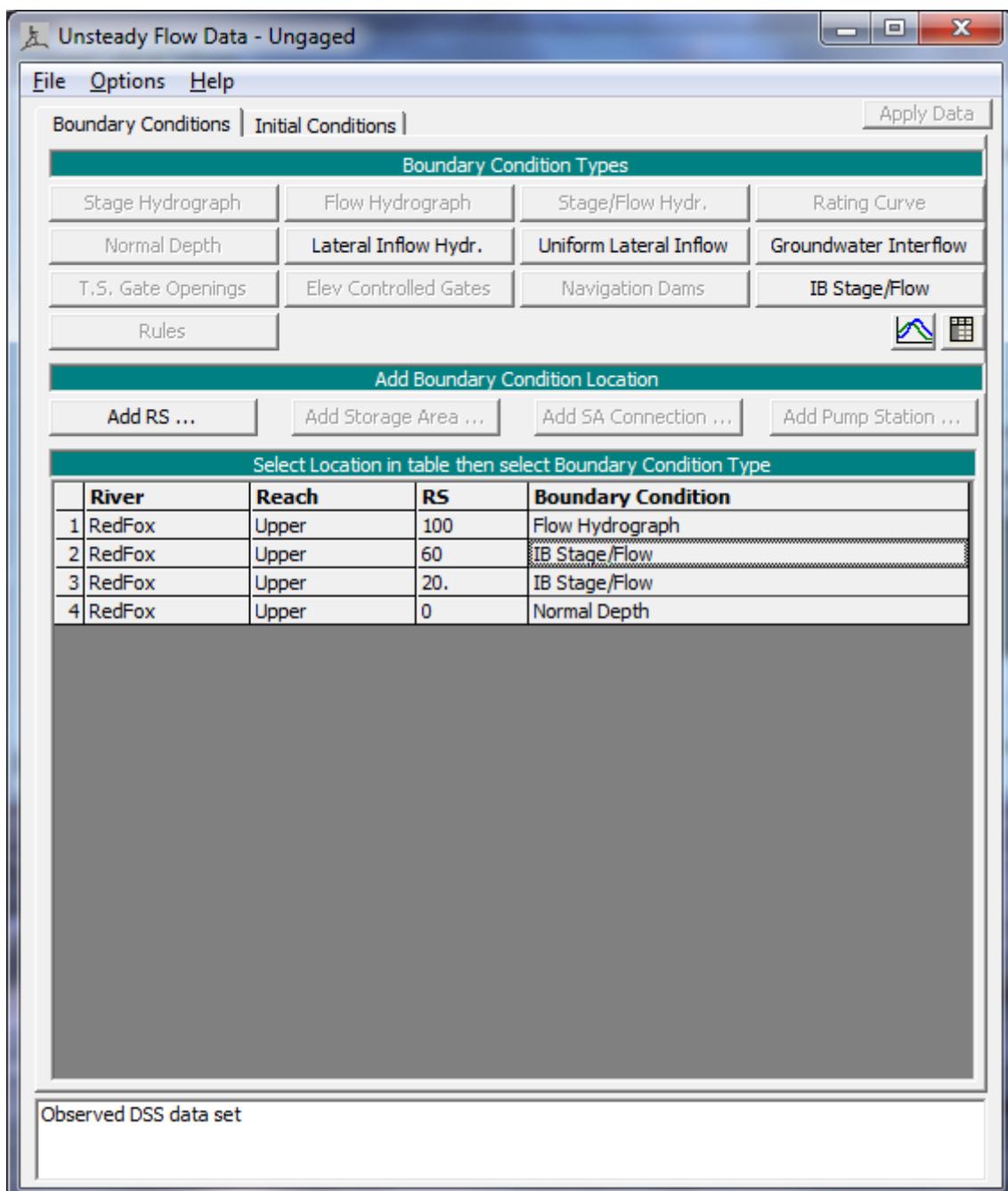


Figure 7 63. Unsteady Flow Boundary Conditions Editor.

This will bring up the Observed Stage and Flow Hydrograph editor:

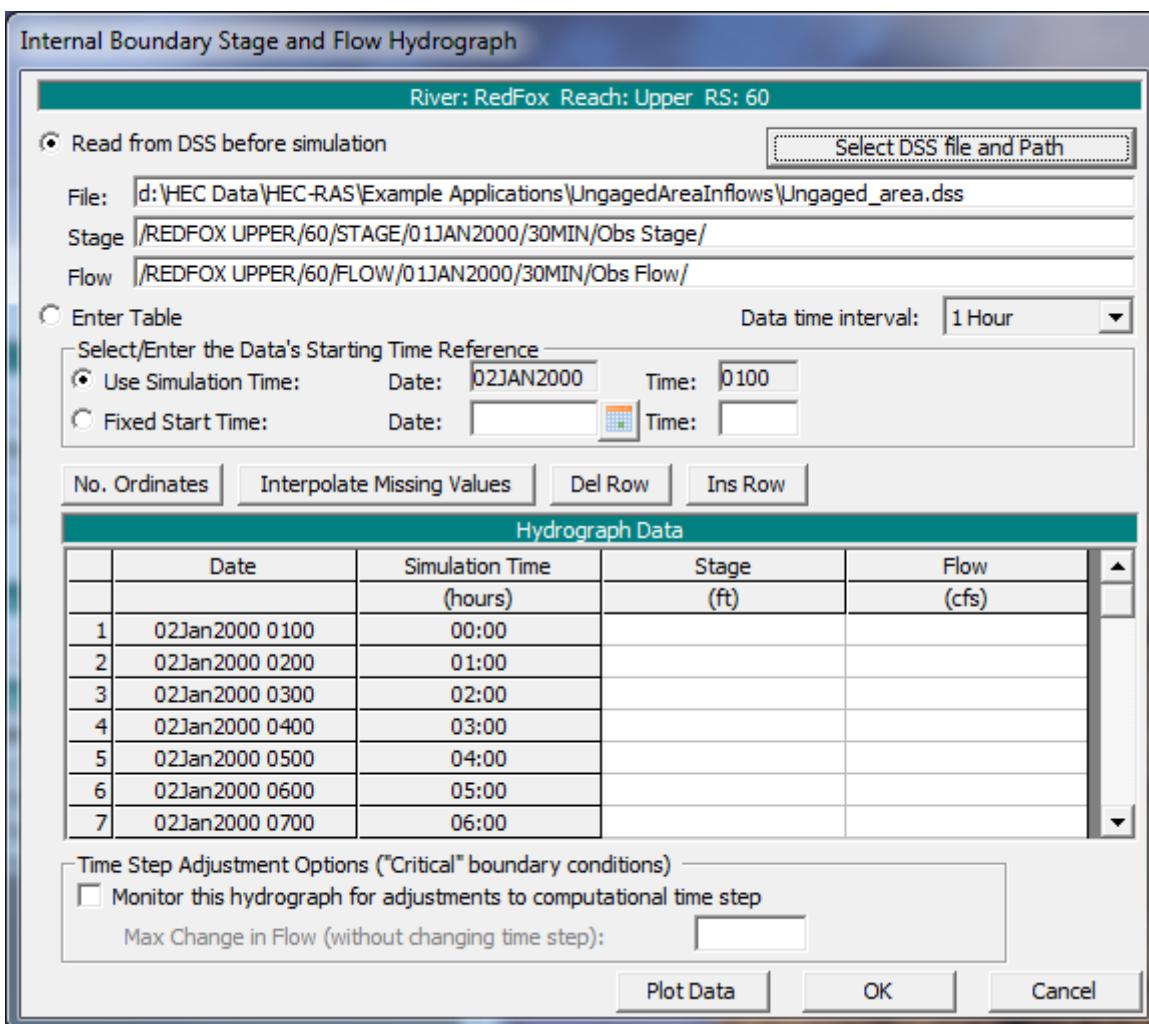


Figure 7 64. Internal Stage and Flow Boundary Condition Editor.

The gage data may either be linked to a DSS file or it can be entered in the table. For use in forecast mode, (optimization based on stage), stage is the only data that is required. For use in historical flow mode, (optimization based on flow), both stage and flow data is required (see hydraulic reference for more info).

Lateral Inflow Computations

Next, the required data for the ungaged lateral inflow should be entered. From the Unsteady Flow Analysis editor, choose Options and click on ungaged lateral inflow. (After ungaged inflow data has been entered, there will be a text message at the bottom of the unsteady run editor indicating that this plan has ungaged inflow. The ungaged lateral inflow editor can be accessed directly by clicking on this message.)

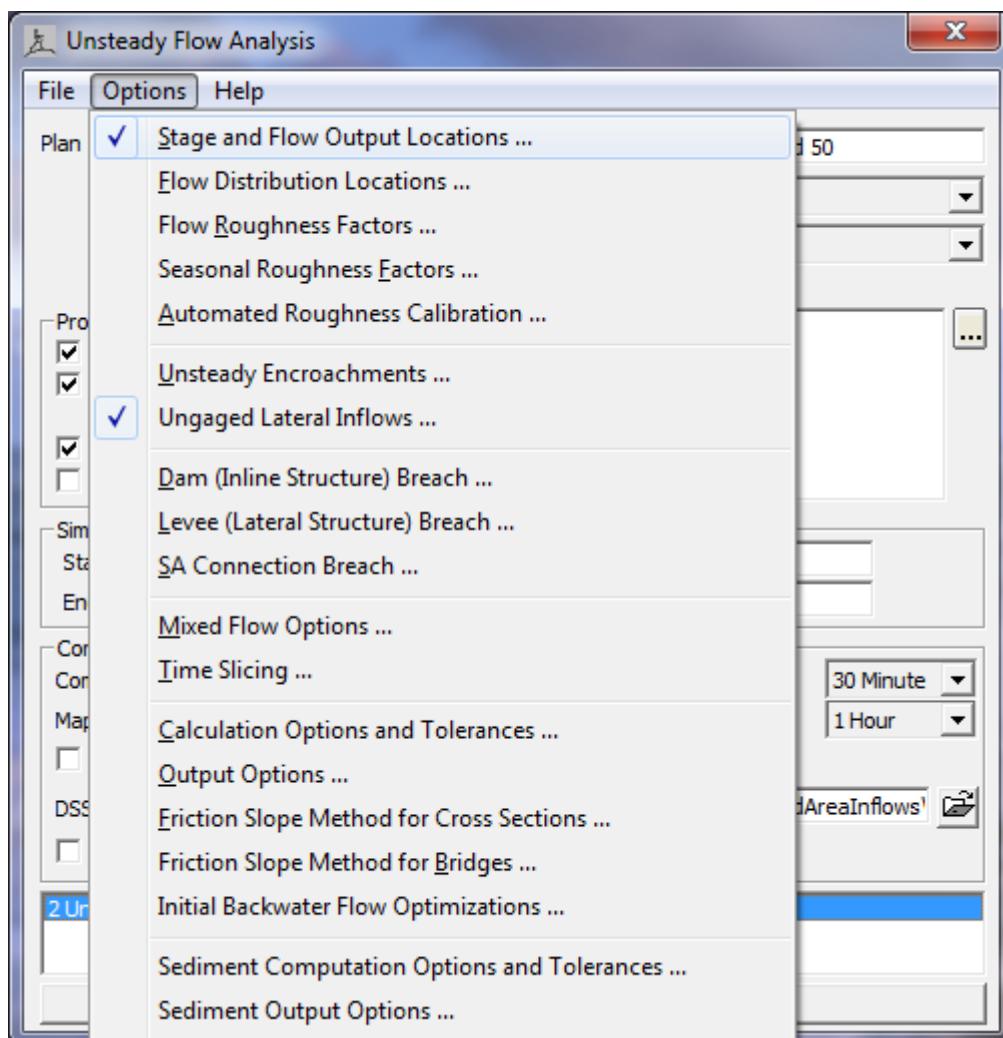


Figure 7 65. Selecting the Ungaged Lateral Inflow Option.

Now the data for the ungaged lateral inflow can be entered (Figure 7-66).

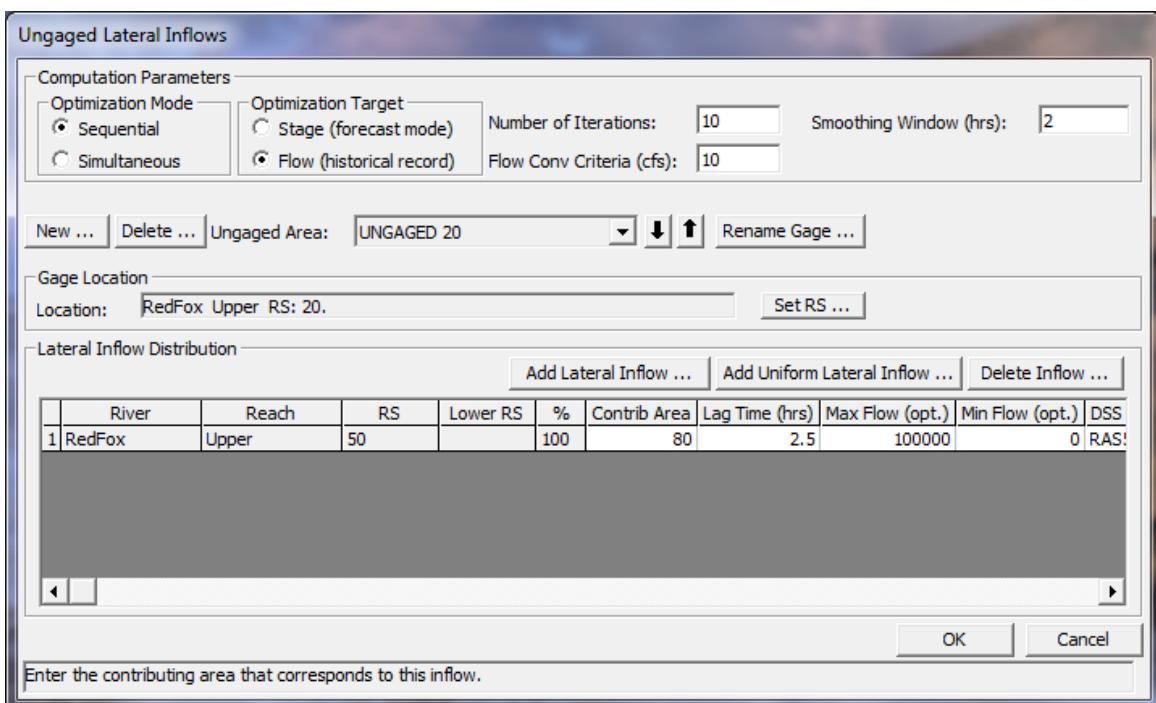


Figure 7.66. Ungaged Lateral Inflow Editor.

By default, the Optimization Mode and Optimization Target are set to Sequential and Stage, respectively (see hydraulic reference manual). The "Number of Iterations" is the maximum number of times that HEC-RAS will iterate (estimate ungaged inflow, lag and route the lateral inflow, run the unsteady model, and check for convergence) when computing the ungaged inflow. The "Flow Conv Criteria" is the flow tolerance (in cfs or m³/s). RAS will iterate until the flow tolerance is met (based on average, least-squared difference—see hydraulic reference) or the maximum number of iterations has been met. If the user enters a value for the "Smoothing Window," then RAS will apply a running-average when computing the ungaged inflow (hydraulic reference...). The previous data (the information at the top of the editor) is only entered once and it applies to all of the known gage/routing reaches.

To enter information for a particular gage, the first step is to click on <New> and enter the name of the gage (or other descriptive name for the inflow into this routing reach). Next, clicking on <Set RS> allows the user to enter the location of this gage. (Important: this river station should match a river station that was entered on the unsteady flow editor for Internal Observed Stage and Flow!)

The unknown lateral inflows are added by clicking on <Add Lateral Inflow> (for a stream or point source) or by clicking on <Add Uniform Lateral Inflow> (for a disperse inflow). The "Contrib. Area" column is used to enter the contributing area for each of the unknown lateral inflows for the current reach. These areas are summed and then a percentage is assigned to each lateral inflow. This is only applicable if a particular known gage/routing reach has more than one unknown inflow. After RAS has determined the total unknown inflow for a given routing reach, the program will proportion the flow between the various lateral inflows based on the ratio of Contributing Areas. In the above example, the contributing area has been entered as a percentage. The uniform inflow starting at river station 70 will get 20% of the flow and the lateral inflow at RS 50 will get the remaining 80%. The contributing areas could also have been entered as a decimal fraction (0.2 and 0.8). Alternately, the user could enter the actual size of the contributing watersheds (e.g., 100 and 400 sq. mi.). Since the division is proportionately based, the units of contributing area do not need to be specified.

Regardless of how the area is entered, RAS will compute and display the percent flow for each area. The "Lag Time" is used by the program when routing the unknown inflow. The amount of unknown inflow is computed at the gage location. However, this unknown inflow enters the river system upstream (that is, at an earlier point in time). The lag time is the amount that the program will shift the lateral hydrograph back in time when it routes the inflow and reruns the program. It should be the approximate travel time from the lateral inflow location to the gage station.

For each unknown inflow, the user can specify an optional maximum and minimum flow. The program will limit the lateral inflow to these values. This may be needed to prevent the program from computing hydraulically unreasonable flows.

Finally, an Optional DSS B part can be specified. After the inflows have been computed, RAS will write the lateral hydrograph out to DSS. The B part of the pathname will either be that specified by the user, or the default name. The default name is, "UNGAGED INFLOW #," where # is the river station for a particular lateral inflow (either 70 or 50 in the above example).

After all the lateral inflow information has been entered, additional routing reaches can be added (only applicable if the given RAS project has more than one known gage) by clicking the <New> button and repeating the process. Different routing reaches can be selected from the drop down menu next to the Ungaged Area Name. Clicking <Delete> will remove the actively displayed routing reach.

Computations and Output

After all the information has been entered, the ungaged inflows can be computed by running RAS normally (clicking the <Compute> button on the main run editor). The results can be viewed by using the DSS viewer inside of RAS.

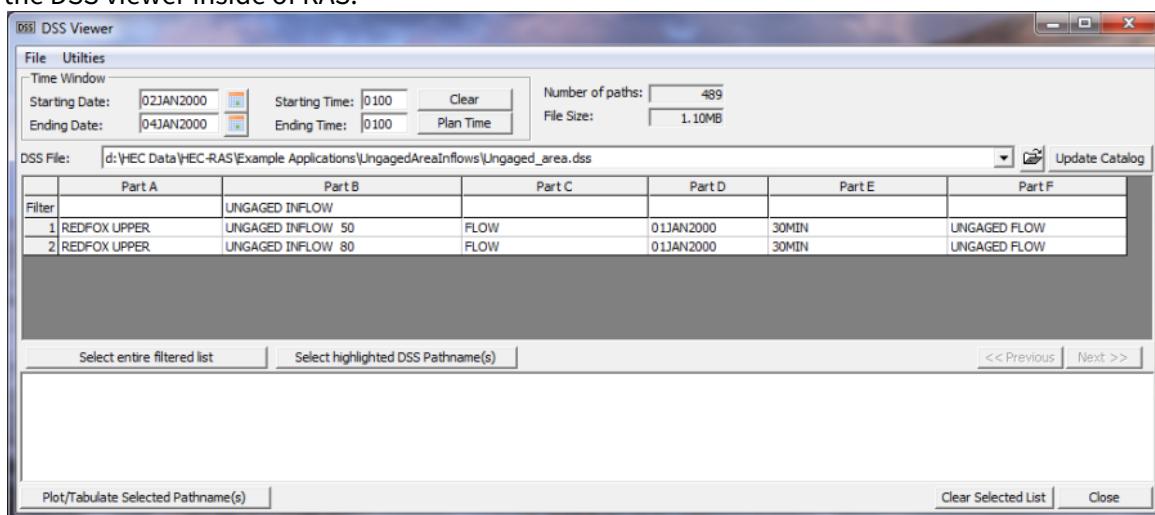


Figure 7 67. HEC-DSS Viewer inside of HEC-RAS

Once a record is selected to view, press the **Plot/Tabulate Selected Pathname(s)** button and the plot will appear as shown in Figure 7-68.

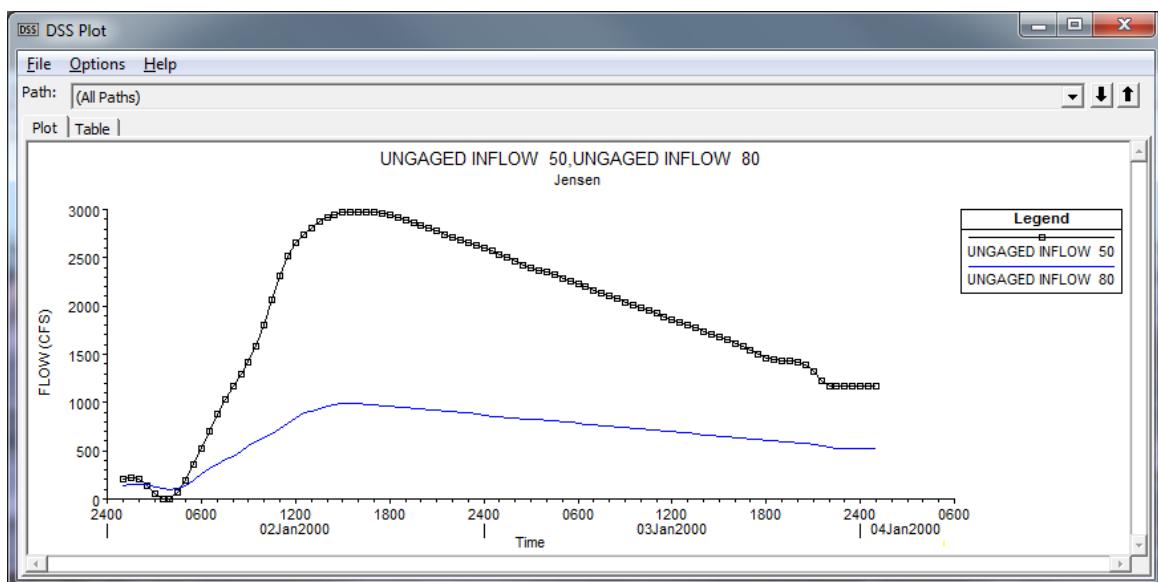


Figure 7 68. Plot of Computed Ungaged Inflows.