C H A P T E R 8

Performing an Unsteady Flow Analysis

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

Contents

Entering and Editing Unsteady Flow Data

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

Unsteady Flow Data

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

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

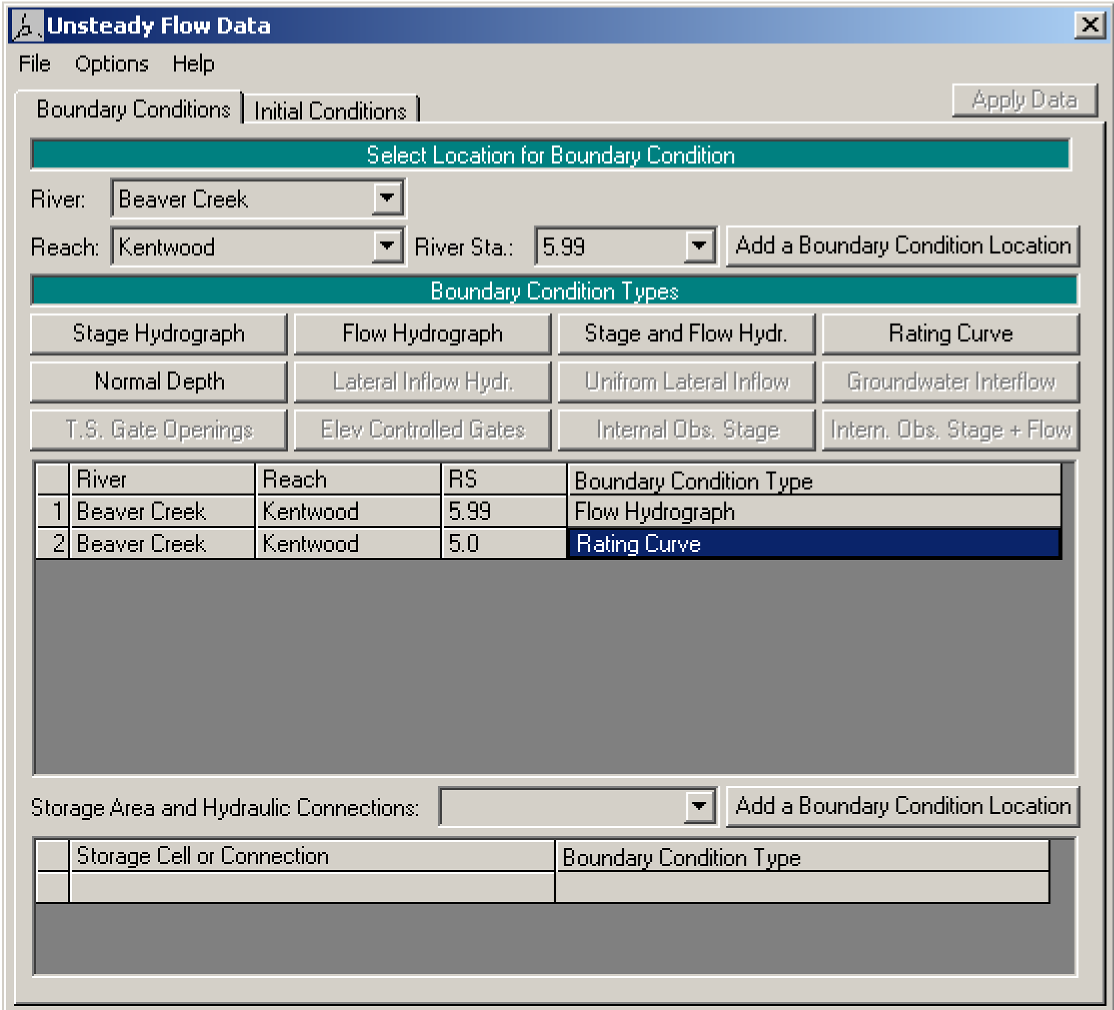

Figure 8.1 Unsteady Flow Data Editor

Boundary Conditions

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

Flow Hydrograph:

A flow hydrograph can be used as either an upstream boundary or downstream boundary condition, but is most commonly used as an upstream boundary condition. When the flow hydrograph button is pressed, the window shown in Figure 8.2 will appear. As shown, the user can either read the data from a HEC-DSS (HEC Data Storage System) file, or they can enter the hydrograph ordinates into a table. If the user selects the option to read the data from DSS, they must press the **Select DSS File and Path** button. When this button is pressed a DSS file and pathname selection screen will appear as shown in Figure 8.3. The user first selects the desired DSS file by using the browser button at the top. Once a DSS file is selected, a list of all of the DSS pathnames within that file will show up in the table. The user can use the pathname filters to reduce the number of pathnames shown in the table. The last step is to select the desired DSS Pathname and to close the window.

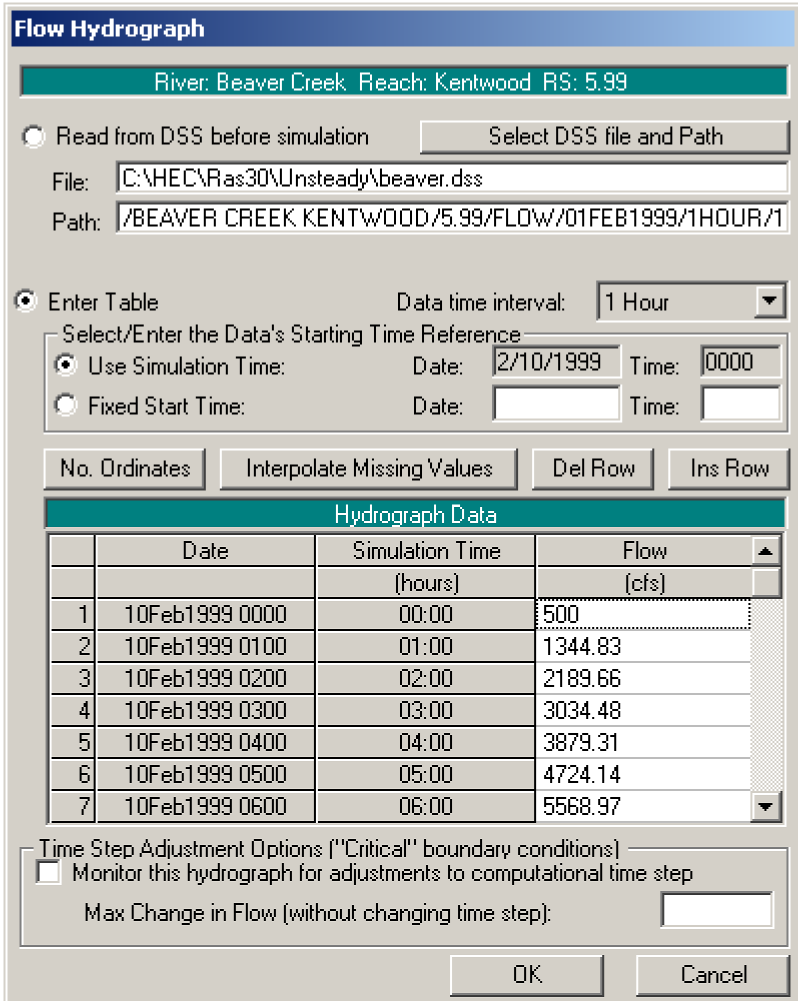

Figure 8.2 Example Flow Hydrograph Boundary Condition

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

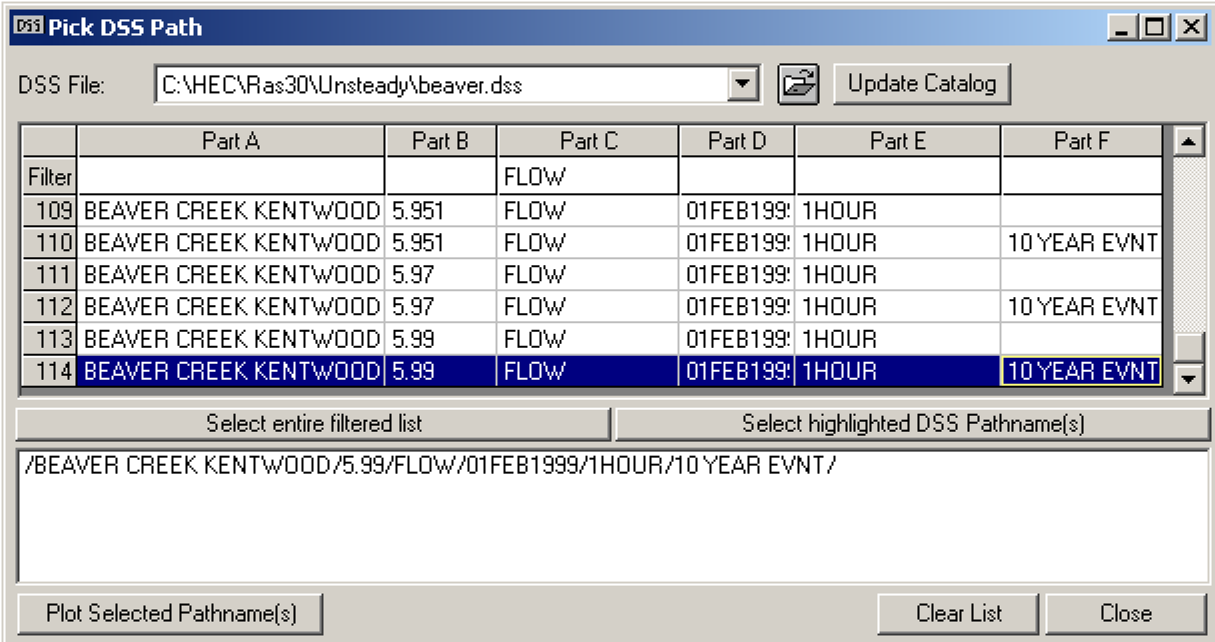

Figure 8.3 HEC-DSS File and Pathname Selection Screen

An additional option listed on 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.

Stage Hydrograph:

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

Stage and Flow Hydrograph:

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

Rating Curve:

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

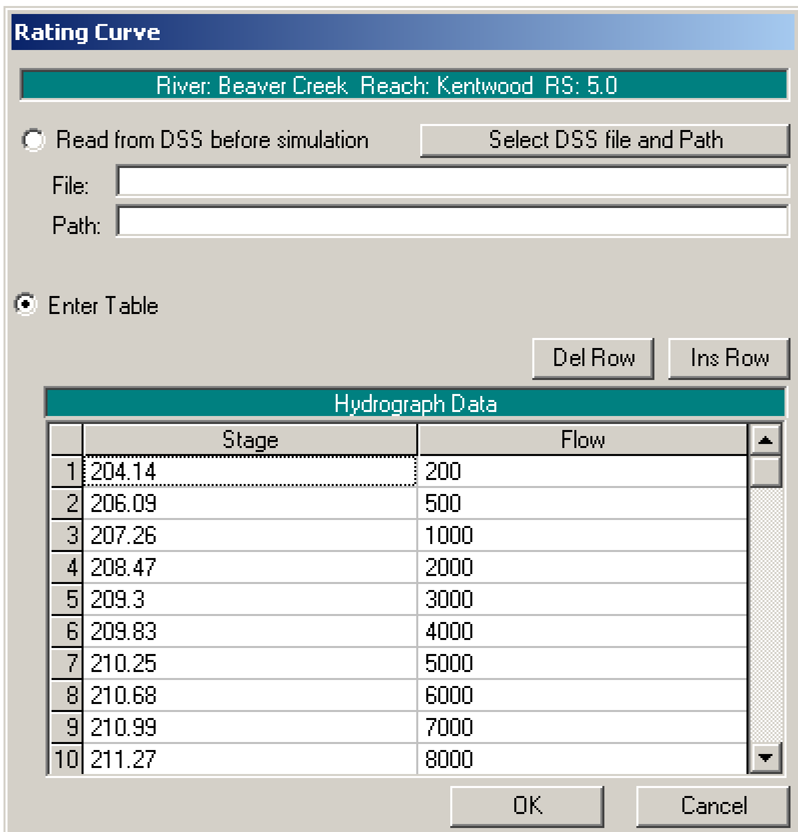

Figure 8.4 Example Rating Curve Boundary Condition Editor

Normal Depth:

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

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

Lateral Inflow Hydrograph:

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

Uniform Lateral Inflow Hydrograph:

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

Groundwater Interflow:

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

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 spillway connecting two storage areas. The user has the option of reading the data from a DSS file or entering the data into a table from within the editor. Figure 8.5 shows an example of the Times Series of Gate Openings editor.

As shown in Figure 8.5, the user first selects a gate group, then either attaches a DSS pathname to that group or enters the data into the table. This is done for each of the gate groups contained within the particular hydraulic structure.

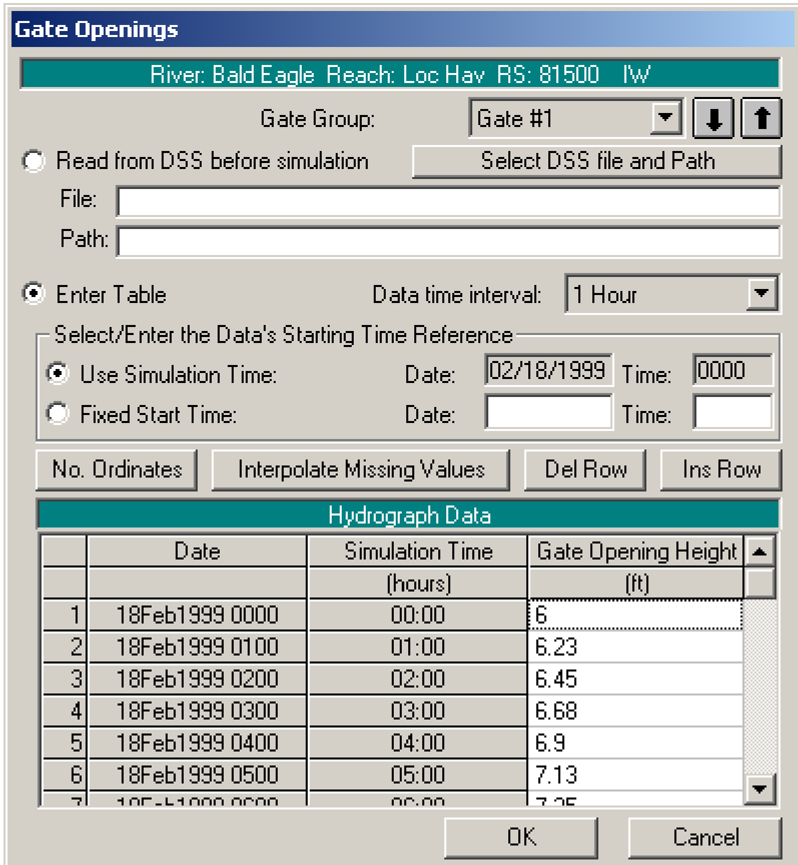

Figure 8.5 Example Time Series of Gate Openings Editor

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

Elevation Controlled Gate:

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

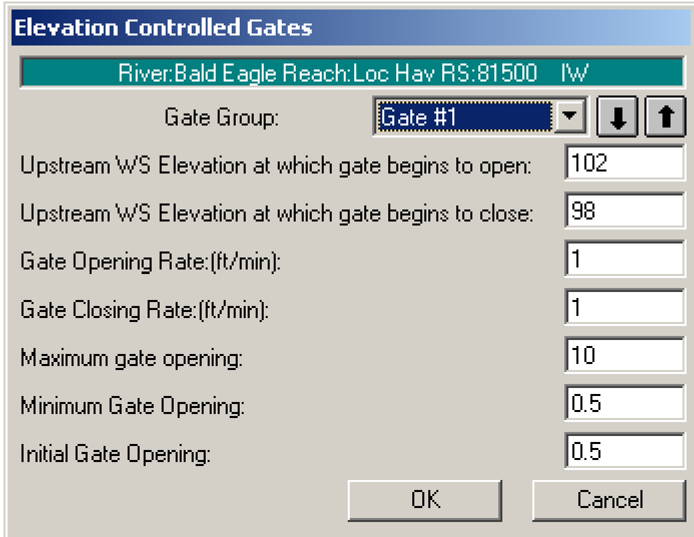

Figure 8.6 Elevation Controlled Gate Editor

Internal Observed Stage Hydrograph:

This option allows the user to enter an observed stage hydrograph to be used as an internal boundary condition just upstream of an inline structure with gated spillways. The observed stage hydrograph can either be entered into the editor directly or it can be read from HEC-DSS.

Internal Observed Stage and Flow Hydrograph:

This is a mixed boundary condition where a stage hydrograph is inserted as the observed boundary until the stage hydrograph runs out of data; afterward the flow hydrograph is used. The end of data in the stage hydrograph is identified by the HEC-DSS missing data code, -901.0. The stage and flow hydrographs can either be entered into a table or can be entered from HEC-DSS. The mixed boundary condition is primarily used for forecast models where the stage data is observed up to the forecast time and the flow hydrograph is the flow forecast. This option can only be used at the upstream cross section of an inline gated spillway structure.

Initial Conditions

In addition to the boundary conditions, the user must establish the initial conditions of the system at the beginning of the unsteady flow simulation. Initial conditions consist of flow and stage information at each of the cross sections, as well as elevations for any storage areas defined in the system. Initial conditions are established from within the Unsteady Flow Data editor by selecting the **Initial Conditions** tab. After the Initial Conditions tab is selected, the unsteady flow data editor will appear as shown in Figure 8.7.

As shown in Figure 8.7, the user has two options for establishing the initial conditions of the system. The first option is to enter flow data for each reach and have the program perform a steady flow backwater run to compute the corresponding stages at each cross section. This option also requires the user to enter a starting elevation for any storage areas that are part of the system. This is the most common method for establishing initial conditions. In the current version of HEC-RAS, the user is limited to entering only one flow per reach, for establishing the initial conditions (i.e. performing the steady flow backwater).

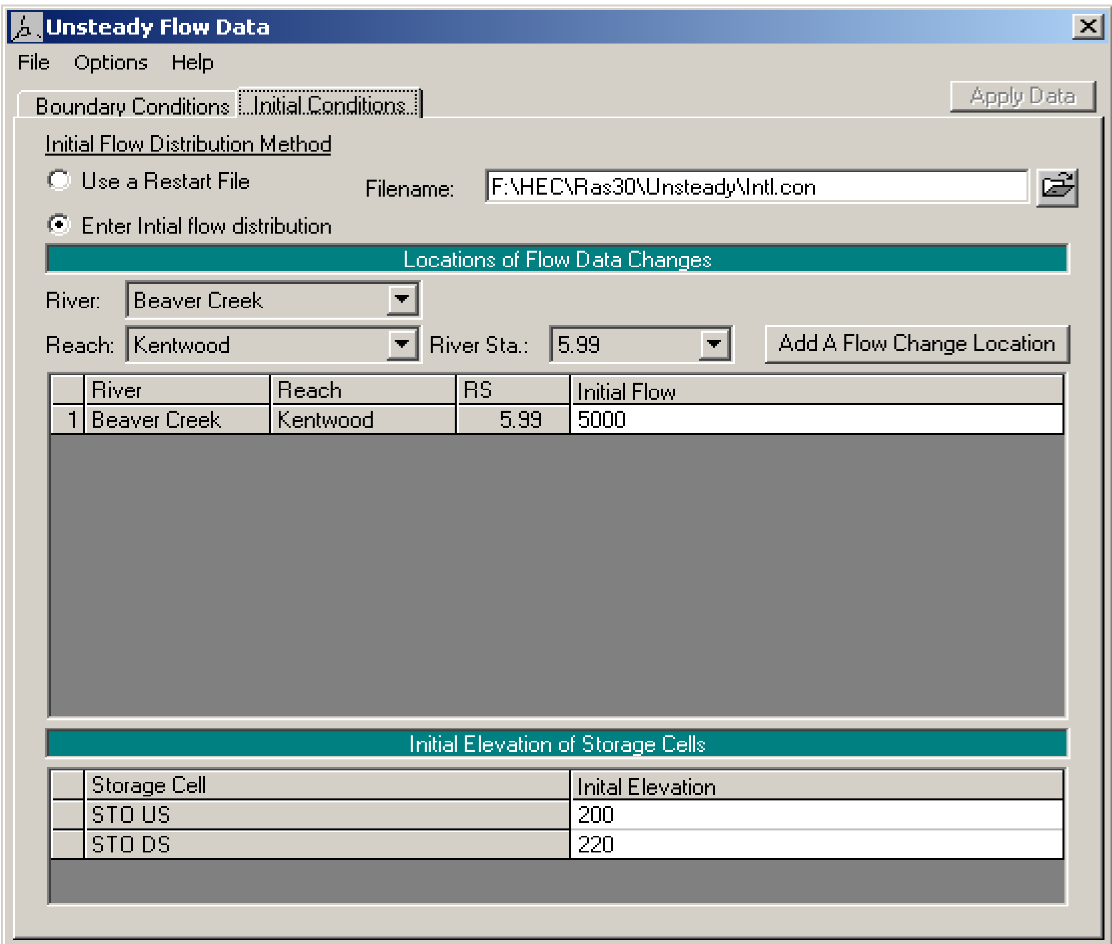

Figure 8.7 Initial Conditions Editor

A second method is to read in a file of stages and flows that was 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 for the constant flow, those conditions can be written out to a file, and then used as the starting conditions for the unsteady flow run.

Unsteady Flow Data Options

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

Undo Editing. This option allows the user to retrieve the data back to the form that it was in the last time the Apply Data button was pressed. Each time the Apply Data button is pressed, the Undo Editing feature is reset to the current information.

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.

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

To use this option, the user selects **Observed Data In DSS** from the **Options** menu of the Unsteady Flow Data editor. When this option is selected a window will appear as shown in Figure 8.8. As shown in the figure below, the user first selects a river, reach, and river station, and then presses the **Add selected location to table** button, in order to select a location to attach observed data to. This should be done for all the locations in which you have observed data. The next step is to open up the DSS file that contains the observed data. This is accomplished by pressing the open file button, which is next to the DSS filename field. When a DSS file is selected, a list of the available pathnames contained in that DSS file will show up in the lower table. To attach a DSS pathname to a particular river station, first select the river station row from the upper table. Then select the DSS pathname row from the lower table. Finally, press the button labeled **Select DSS Pathname**. Repeat this process for every location in which you wish to attach observed data. If you are going to have more than one data type (such as stage and flow) at a particular river station, you must have to entries in the upper table for that river station.

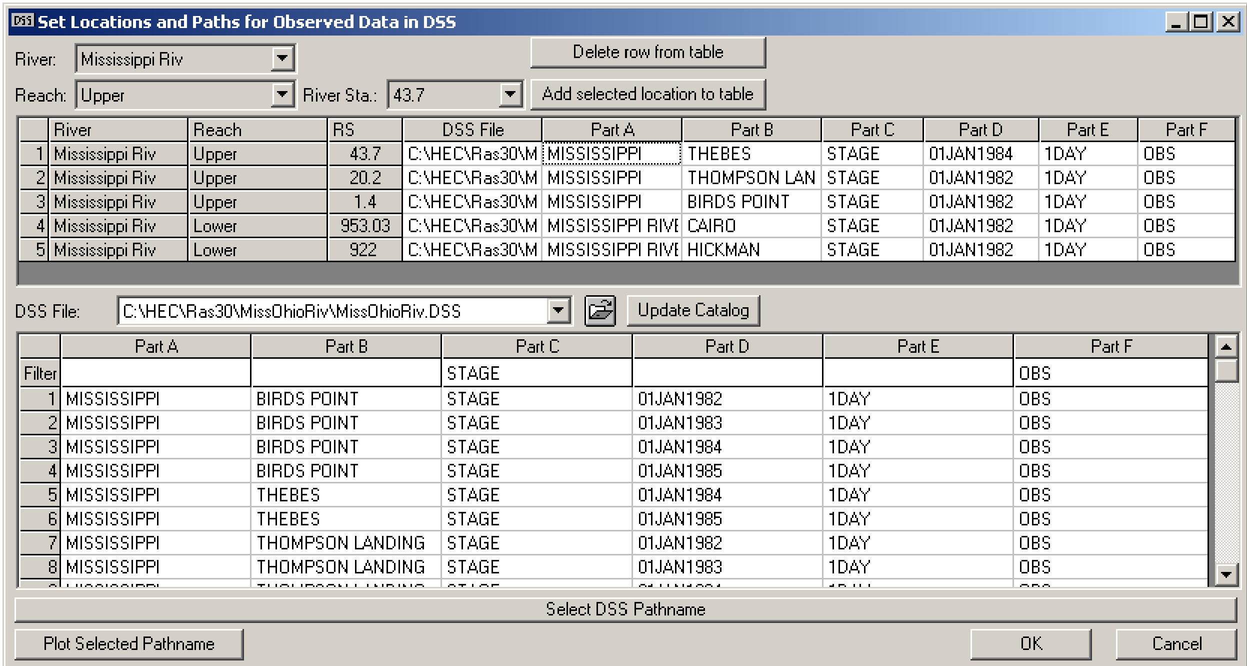

Figure 8.8 Editor for Establishing Locations of Observed Data

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 steady flow data editor. A pop-up window will appear prompting you to enter a title for the data.

Performing Unsteady Flow Calculations

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

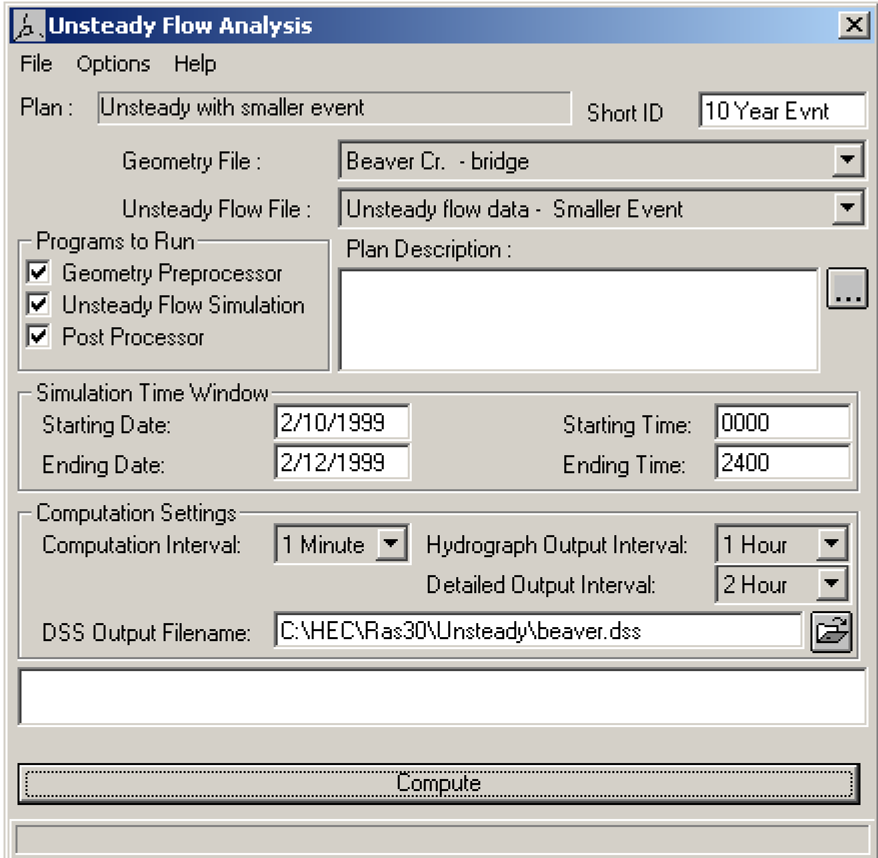

Figure 8.9 Unsteady Flow Analysis Window

Defining A Plan

The first step in performing a simulation is to put together a Plan. The Plan defines which geometry and unsteady flow data are to be used, as well as provide 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 (HTAB); the unsteady flow simulator (UNET); and an output post-processor.

Geometric Pre-Processor (HTAB)

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

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

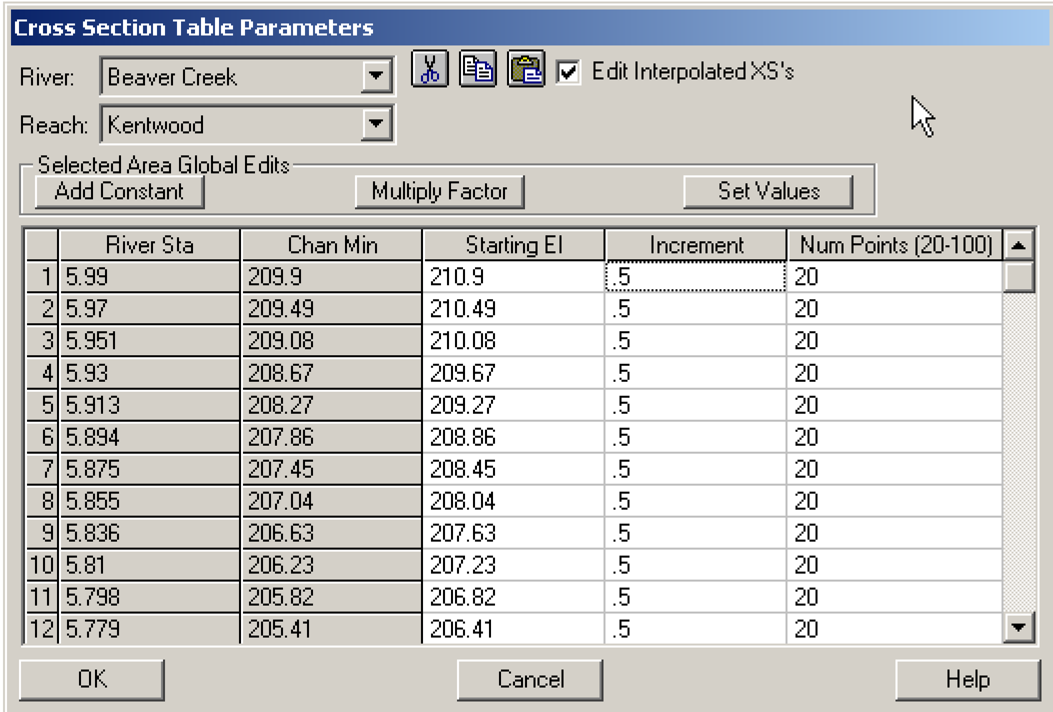

Figure 8.10 Hydraulic Table Parameters for Cross Sections

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

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

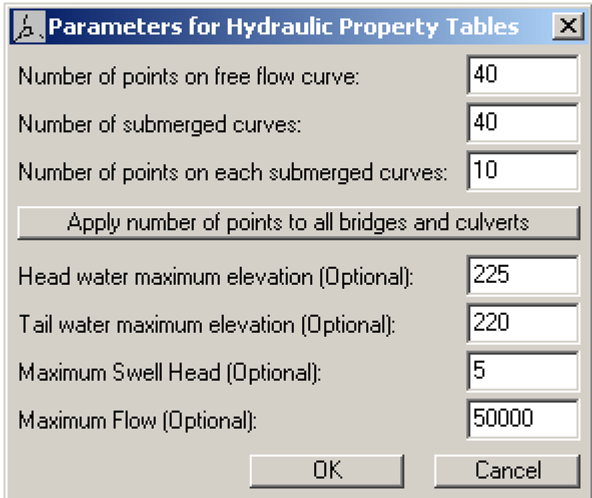

Figure 8.11 Hydraulic Properties Table for Bridges/Culverts

As shown in Figure 8.11, the user can set the number of points to be computed on the free-flow rating curve (maximum of 50 points); the number of submerged curves to be computed (maximum of 50); and the number of points on the submerged curves (maximum of 20). The default values for these parameters are 40, 40, and 10 respectively. Additionally, the user can refine the curves by putting limits on the extents of the curves. This can be accomplished by entering the head water maximum elevation; tail water maximum elevation; maximum swell head (difference between the head water and tailwater); and the maximum possible flow.

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

Unsteady Flow Simulation (UNET)

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

Post-Processor

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

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

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

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

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

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

Simulation Time Window

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

Computation Settings

The Computation Settings area of the Unsteady Flow Analysis window contains: the computational interval; hydrograph output interval; detailed output interval; and the name and path of the output DSS file.

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 floodwave. A general rule of thumb is to use a computation interval that is equal to or less than the time of rise of the floodwave divided by 24. In other words, if the flood wave goes from its base flow to its peak flow in 24 hours, then the computation interval should be equal to or less than 1 hour.

Additional considerations must be made for hydraulic structures, such as bridges, culverts, weirs, and gated spillways. Within bridges and culverts, when the flow transitions from unsubmerged to submerged flow, the water surface 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 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 as to reduce the amount of post-processing and storage required for detailed hydraulic output. One example for selecting this variable would be, if the time window of the simulation was set at 72 hours, then one might want to set the instantaneous profiles to an interval of every 6 hours. This would equate to 13 profiles being written out and having detailed hydraulic information computed for them.

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

Simulation Options

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

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

As shown in Figure 8.12, the user can select individual locations, groups of cross sections, or entire reaches. Setting these locations is important, in that, after a simulation is performed the user will only be able to view stage and flow hydrographs at the selected locations.

Flow Distribution Locations. This option allows the user to specify locations in which they would like the program to calculate flow distribution output. The flow distribution option allows the user to subdivide the left overbank, main channel, and right overbank, for the purpose of computing additional hydraulic information.
The user can specify to compute flow distribution information for all the cross sections (this is done by using the Global option) or at specific locations in the model. The number of slices for the flow distribution computations must be defined for the left overbank, main channel, and the right overbank. The user can define up to 45 total slices. Each flow element (left overbank, main channel, and right overbank) must have at least one slice. The flow distribution output will be calculated for all profiles in the plan during the computations.

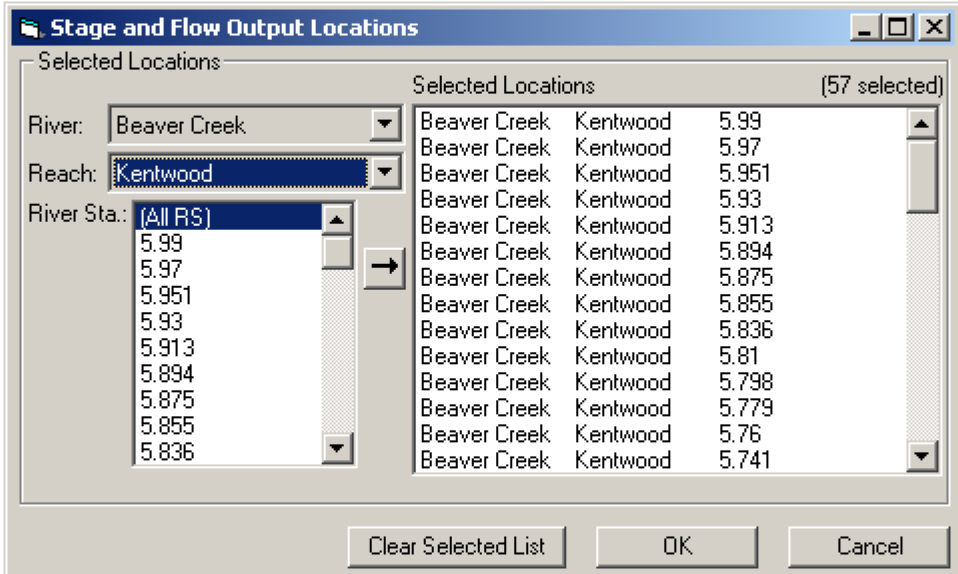

Figure 8.12 Stage and Flow Hydrograph Output Window

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

As shown in Figure 8.13, the user first selects a river, reach, and a range of cross sections to apply the factors to. Next a starting flow, flow increment, and a number of increments is entered. Finally, a roughness factor is entered into the table for each of the flows. The user can create several sets of these factors to cover a range of locations within the model. However, one set of factors cannot overlap with another set of factors. Hence, you can only apply one set of roughness change factors to any given cross section.

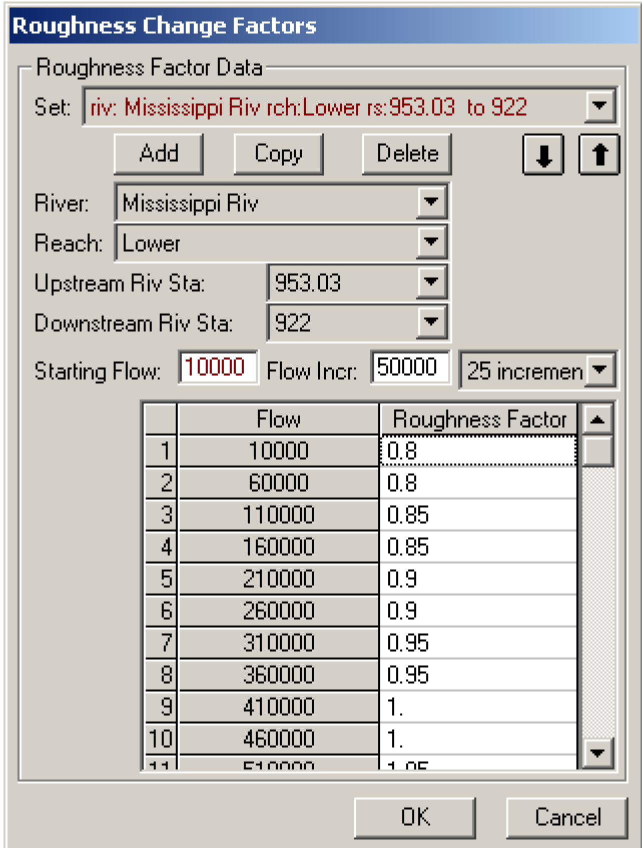

Figure 8.13 Flow versus Roughness Change Factors Editor

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

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

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

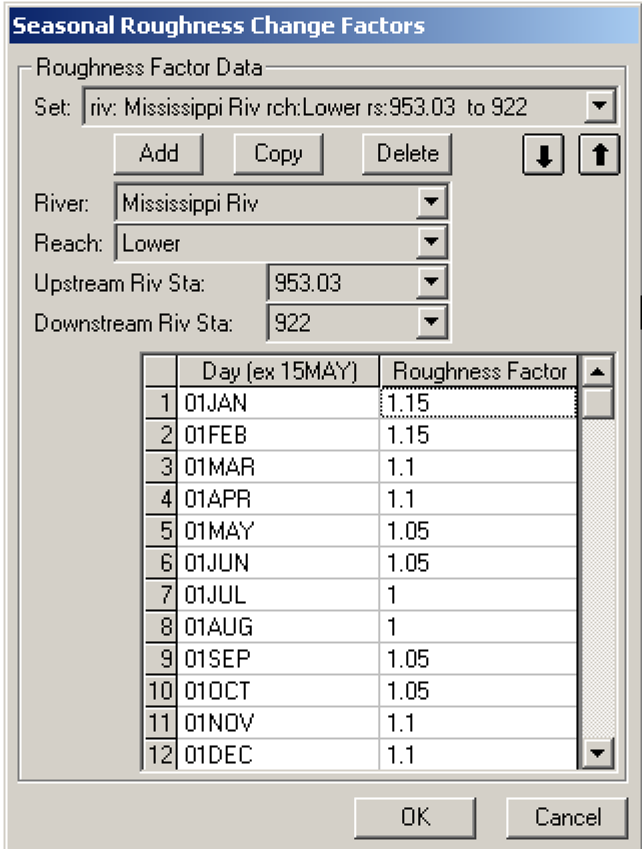

Figure 8.14 Seasonal Roughness Factors Editor

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

Theta implicit weighting factor: This factor is used in the finite difference solution of the unsteady flow equations. The factor ranges between 0.6 and 1.0. A value of 0.6 will give the most accurate solution of the equations, but is more susceptible to instabilities. A value of 1.0 provides the most stability in the solution, but may not be as accurate for some data sets. The default value is set to 1.0. Once the user has the model up and running the way they want it, they should then experiment with changing theta towards a value of 0.6. If the model remains stable, then a value of 0.6 should be used. In many cases, you may not see an appreciable difference in the results when changing theta from 1.0 to 0.6. However, every simulation is different, so you must experiment with your model to find the most appropriate value.

Water surface calculation tolerance: This tolerance is used to compare against 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 against 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 that the tolerance, the program can go on to the next time step. The default tolerance for storage areas is set to 0.1 feet.

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

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

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

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

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

Weir flow stability factor: This factor is used to increase the stability of the numerical solution in and around a weir. This factor varies from 1.0 to 3.0.

As the value is increased, the solution is more stable but less accurate. A value of 1.0 is the most accurate, but is susceptible to oscillations in the computed weir flow. The default value is 1.0. If you observe oscillations in the computed flow over the weir, you should first check to see if you are using a small enough computation interval. If the computation interval is sufficiently small, you should then try increasing this coefficient to see if it solves the problem.

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

Weir flow submergence decay exponent: This coefficient is used to stabilize the solution of flow over a weir for highly submerged weirs. This factor varies from 0.0 to 2.0. 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 zero has no effect. As you increase the coefficient, dampening of the oscillations will occur.

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 0.0 to 2.0. 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 zero has no effect. As you increase the coefficient, dampening of the oscillations will occur.

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

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

Write velocity profiles to DSS: When this option is turned on, the program will write velocity profiles to the HEC-DSS file. One velocity profile will be written for each corresponding water surface profile that is written.

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

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

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

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

Saving The Plan Information

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

Starting the Computations

Once all of the data have been entered, and a Plan has been defined, the unsteady flow computations can be performed by pressing the **Compute** button at the bottom of the unsteady flow simulation window. Once the compute button is pressed, a separate window will appear showing you the progress of the computations. The information that appears in the window is only there as an indicator of the programs progress during the computations. This window is not intended for viewing any output. When the computations have been completed, the user can close the computations window by clicking the upper right corner of the window. If the computations ended with a message stating "**Program Finished Normally**," the user can then begin to review the output. If the program does not finish normally, then the user should turn on the log file option and re-run the program. Then view the log file output to begin debugging the problem.