

6 ENTERING AND EDITING GEOMETRIC DATA

Geometric data consist of establishing the connectivity of the river system (River System Schematic), entering cross-section data, defining all the necessary junction information, adding hydraulic structure data (bridges, culverts, dams, levees, weirs, etc...), pump stations, storage areas, and two-dimensional flow areas. The geometric data is entered by selecting Geometric Data from the Edit menu on the HEC-RAS main window. Once this option is selected, the Geometric Data window will appear as shown in Figure 5-1. The drawing area will be blank on your screen, until you have either drawn in your own river system schematic or imported data from a GIS.

IMPORTANT NOTE: The current version of HEC-RAS has the ability to lay out the geometry of a model (i.e. stream centerline, cross sections, storage areas, 2D flow areas, and hydraulic structures) in either the Geometric Data editor or HEC-RAS Mapper. HEC is moving towards having a completely geospatial layout system for developing the geometry using HEC-RAS Mapper. The tools in the Geometric data editor are the original tools for developing a geometric model. All of the new tools for geospatially laying out the model geometry are in HEC-RAS Mapper. If you want to use HEC-RAS Mapper to layout the geometric data model, please read the **HEC-RAS Mapper User's manual**. Once you have a model laid out within HEC-RAS Mapper, you can extract terrain data for cross sections, storage areas, 2D flow areas, and hydraulic structures. However, to complete the parameterization of the model (i,e, cross section coefficients, structure modeling approaches and coefficients, etc...) you must come back into the Geometric data editor to complete all the required Geometric data for the model.

This chapter describes how to enter and edit all of the necessary geometric data for a river system using the Geometric Data editor.

The following topics will be discussed in this section:

- Developing the River System Schematic
- Cross Section Data
- Stream Junctions
- Bridges and Culverts
- Multiple Bridge and/or Culvert Openings
- Inline Structures (Weirs and Gated Spillways)
- Lateral Structures (Weirs, Gates, Culverts, and Rating Curves)
- Storage Areas
- Two-Dimensional Flow Areas
- Storage Area/2D Flow Area Hydraulic Connections
- Pump Stations
- Cross Section Interpolation
- River Ice
- Viewing and Editing Data Through Tables
- Importing Geometric Data
- Geometric Data Tools
- Georeferencing an HEC-RAS Model
- Attaching and Viewing Pictures
- Saving the Geometric Data

Developing the River System Schematic



The modeler develops the geometric data by either first drawing in the river system schematic on the Geometric Data window (Figure 5-1) or by developing a schematic in HEC-RAS Mapper (See the separate HEC-RAS Mapper User's manual). The River System Schematic is a diagram of how the stream system is connected together. The river system is drawn on a reach-by-reach basis, by pressing the **River Reach** button and then drawing in a reach from upstream to downstream (in the positive flow direction). Each reach is identified with a **River Name** and a **Reach Name**. The River Name should be the actual name of the stream, while the reach name is an additional qualifier for each hydraulic reach within that river. A river can be comprised of one or more reaches. Reaches start or end at locations where two or more streams join together or split apart. Reaches also start or end at the open ends of the river system being modeled.

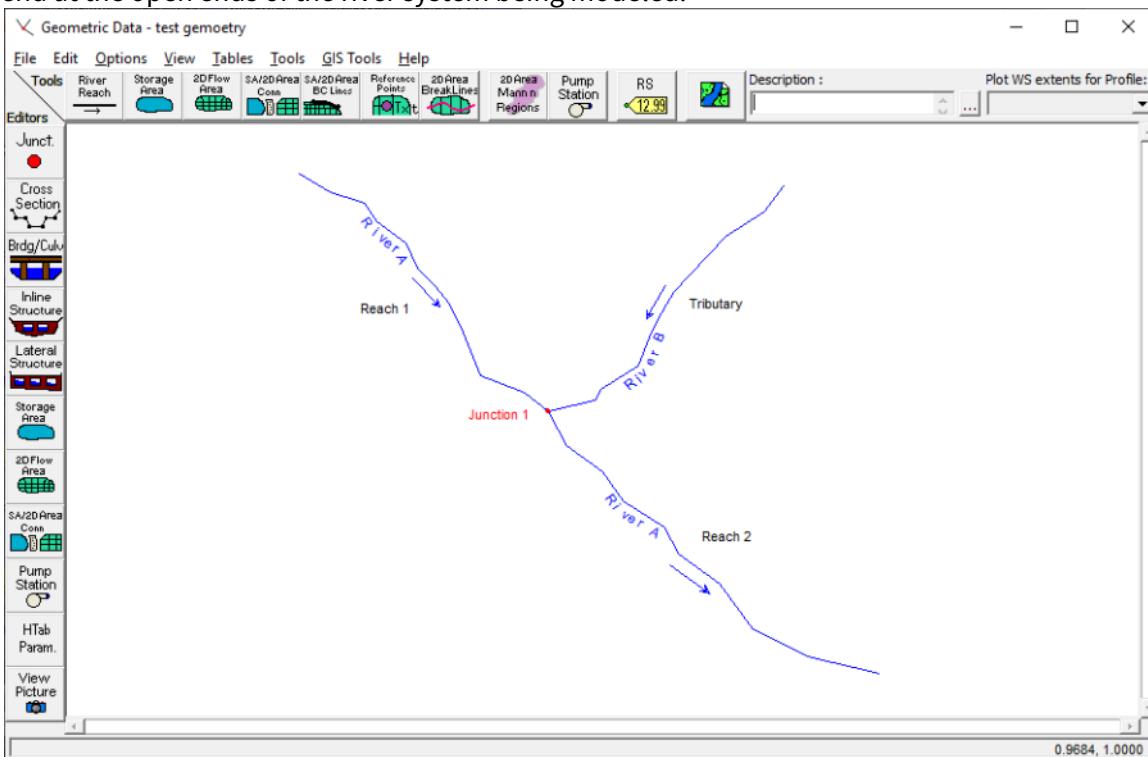


Figure 5 1 Geometric Data Editor Window

Reaches are drawn as multi-segmented lines. Each reach must have at least two points, defining the start and end of the reach. However, it is more typical to draw a reach with several points that would follow along the main channel invert of the stream (this can be accomplished by first loading a background map into the Geometric Data editor). To draw a reach, first press the **River Reach** button at the top of the Geometric Data editor, on the tools button bar. Move the mouse pointer to the location on the drawing area that you would like to have the reach begin (upstream end of the reach). Click the left mouse button once to define the first point of the reach. Move the mouse and continue to click the left mouse button to add additional points to the reach. To end a reach, move the mouse pointer to the location in which you would like the last point of the reach to be located, then double click the left mouse button. After the reach is drawn, the user is prompted to enter the **River Name** and the **Reach Name** to identify the reach. The river and reach identifiers are limited to

sixteen characters in length. If a particular River Name has already been entered for a previously defined reach of the same river, the user should simply select that river name from the list of available rivers in the river name text box. As reaches are connected together, **Junctions** are automatically formed by the interface. The modeler is also prompted to enter a unique identifier for each junction. Junctions are locations where two or more streams join together or split apart. Junction identifiers are also limited to sixteen characters. An example of a simple stream system schematic is shown in Figure 5-1.

In addition to river reaches, the user can draw **Storage Areas, 2D Flow Areas, Storage Area/2D Flow Area Connections, Storage Area/2D Flow Area Boundary Condition Lines, and Pump Stations**. A storage area is used to define an area in which water can flow into and out of. The water surface in a storage area is assumed to be a level pool. Storage areas can be connected to river reaches as well as other storage areas. The user can connect a storage area directly to the end of a reach or it can be connected to a reach by using the lateral structure option. To connect a storage area to the end of a river reach, simply draw or move the end point of a reach inside of the storage area. Storage areas can be connected to other storage areas hydraulically by using the **SA/2D Area Conn** option. Storage Area/2D Flow Area connections consist of culverts, gated spillways and a weir. The user can set up a Storage Area/2D Flow Area Connection as just a weir, a weir and culverts, or a weir and gated spillways.

To draw a **Storage Area**, select the storage area button at the top of the geometric editor window. Storage areas are drawn as polygons. Move the mouse pointer to the location in which you would like to start drawing the storage area. Press the left mouse button one time to start adding points to define the storage area. Continue using single left mouse clicks to define the points of the storage area. To end the storage area, use a double left mouse click. The storage area will automatically be closed into a polygon. Once you have finished drawing the storage area, a window will appear asking you to enter a name for the storage area. To enter and edit the data for a storage area, use the storage area editor button on the left panel of the geometric data window.

To draw a **2D Flow Area**, select the 2D Flow Area button at the top of the geometric editor window. 2D Flow areas are drawn as polygons, just like storage areas. Move the mouse pointer to the location in which you would like to start drawing the storage area. Press the left mouse button one time to start adding points to define the storage area. Continue using single left mouse clicks to define the points of the storage area. To end the 2D Flow Area, use a double left mouse click. The 2D Flow Area will automatically be closed into a polygon. Once you have finished drawing the 2D Flow Area, a window will appear asking you to enter a name. To enter and edit the data for a 2D Flow Area, use the 2D Flow Area editor button on the left panel of the geometric data window.

To enter a Storage Area/2D Flow Area Connection, select the **SA/2D Area Conn** button at the top of the Geometric data editor (this is a hydraulic structure that can be used to connect two storage areas, a storage area to a 2D Flow Area, or two 2D Flow Areas). Move the mouse pointer to the location of left end of the hydraulic structure. SA/2D Area Connections should be drawn from left to right looking in the positive flow direction. Click the left mouse pointer one time to start the drawing of the SA/2D Area Connection. You can continue to use single mouse clicks to add as many points as you want into the line that represents the SA/2D Area Connection. When you want to end the line, double click the mouse pointer. A window will pop up asking you to enter a name for the SA/2D Area Connection. The direction in which you draw the structure is important for establishing the positive flow direction for the flow. If you want the program to output positive flow when the flow is going from one area to another area, then you must draw from left to right looking in the positive flow direction. If flow happens to go in the other direction during the calculations, that flow will be output

as negative numbers. To enter and edit the data for a SA/2D Flow Area connection use the **SA/2D Area Conn** data editor on the left panel of the geometric data window.

To add boundary conditions directly to a 2D Flow Area or a Storage Area, select the **SA/2D Area BC Lines** button at the top of the Geometric data editor. Move the mouse point to the area of the outer boundary of the 2D Flow Area (or Storage Area) in which you would like to start the boundary condition line. Then click the left mouse pointer one time to start the drawing of the boundary condition line. You can continue to use single mouse clicks, along the outer boundary of the 2D Flow Area, to add as many points as you want into the line that represents the boundary condition. When you want to end the boundary condition line, place the mouse pointer over the location of where you want the line to end, and then double click the mouse pointer. A window will pop up asking you to enter a name for the boundary condition line. These boundary condition lines will show up in the Unsteady Flow Data editor, and will require you to select a boundary condition type (Flow Hydrograph, Rating Curve, or Stage Hydrograph) and enter the necessary data.

Pump stations can be connected between two storage areas, between a storage area and a river reach, or between two river reaches. To add a pump station to the schematic, click the **Pump Station** button at the top, under the tools button bar, and then left click on the schematic at the location where you want to place the pump station. To connect the pump station, either left click over top of the pump station and select edit, or just click on the pump station editor from the edit tool bar. Connecting pumps is accomplished by picking from and to locations from the pump data editor.

Adding Tributaries into an Existing Reach

If you would like to add a tributary or bifurcation into the middle of an existing reach, this can be accomplished by simply drawing the new reach, and connecting it graphically to the existing reach at the location where you would like the new junction to be formed. This is accomplished by ending the new river reach (tributary) right on top of the location of the main river, where you want the new Junction to be formed. Once the new reach is connected into the middle of an existing reach, you will first be prompted to enter a River and Reach identifier for the new reach. After entering the river and reach identifiers, you will be asked if you want to "Split" the existing reach into two reaches. If you answer "yes", you will be prompted to enter a new Reach identifier for the lower portion of the existing reach (the original reach name is kept for the upper portion of the reach) and a Junction name for the newly formed stream junction.

Editing the Schematic

There are several options available for editing the river system schematic. These options include: changing labels; moving Points/objects (such as labels, junctions, points in a reach or cross section, and 2D area cell centers); adding points (to a reach, cross section, storage area, 2D area boundary, and 2D Flow Area cell centers); deleting points; editing the schematic lines and symbols (line and point types and colors); changing the reach and river station text color; deleting entire reaches, junctions, storage areas/2D Flow Areas, storage area connections, pumps, nodes (cross sections, bridges/culverts, inline structures, lateral structures), and SA/2D Area boundary condition lines. Editing functions for the schematic are found under the **Edit** menu of the geometric data window. When a specific editing function is selected, the interaction of the user with the schematic is restricted to performing that type of operation. When the user is finished performing that editing function they should turn off that editing function by selecting it again from the **Edit** menu. When

none of the editing functions are turned on, the schematic goes back to the default mode of interaction. The default interaction mode for the schematic is described in the "Interacting with the Schematic" section of this document. A description of each editing function follows:

- **Change Name:** This option allows the user to change the identifiers of any reach or junction. To change an identifier, you must be in the Change Name edit mode. This is accomplished by selecting the **Change Name** option from the **Edit** menu. Once you are in the Change Name edit mode, you then select the particular label that you would like to change by clicking the left mouse button over that label. When a label is selected, a pop up window will appear allowing you to enter a new label. The user can continue to change names by simply selecting the next label to be changed. The **Change Name** option can only be turned off by re-selecting it from the edit menu or by selecting any other edit option.
 - **Move Points/Objects:** This option allows you to move any label, junction, points in the stream centerline of a reach, points defining the layout of cross sections, storage areas, 2D Flow Area boundaries, and 2D Flow Area cell centers. This is accomplished by first selecting **Move Object** from the **Edit** menu, then selecting the particular object that you would like to move. To select an object and then move it, simply place the mouse pointer over the object, then press the left mouse button down. Move the object to the desired location and then release the left mouse button. The **Move Object** option will remain in effect until the user either turns it off (which is accomplished by re-selecting it) or selects any other edit option.
 - **Add Points:** This option allows the user to add additional points to the line that defines a reach, cross section, storage area, 2D flow area boundaries, and 2D flow Area cell centers. This allows the user to make the schematic be drawn spatially correct as it would be on a map, as well as modify the 2D Flow Area mesh computational cells . To add additional points, first select **Add Points** from the **Edit** menu. Move the mouse pointer to the location in which you would like to add an additional point on the reach line, then click the left mouse button. After you have finished adding points to a reach, you can move them around by selecting the **Move Object** option from the **Edit** menu. To turn the "Add Points" mode off, simply re-select it from the Edit menu, or select any other edit function.
 - **Remove Points:** This option allows the user to remove points from a reach line, cross section line, storage area, 2D flow area boundary, or 2D Flow Area cell centers. To use this option, first select **Remove Points** from the **Edit** menu. Move the mouse pointer over the point that you would like to delete and then click the left mouse button. This option can only be turned off by either re-selecting the option from the Edit menu or by selecting another edit function.
 - **Lines and Symbols:** This option allows the user to change the line and symbol types, colors, and widths for the information on the stream system schematic. When this option is selected a window will appear that shows each line type being used in the schematic. The user can select a particular line type, then change the properties of that line.
 - **Reach and RS Text Color:** This option allows the user to change the color of the text for reaches and river stations plotted on the schematic. The default color is black.
- The **Edit** menu contains an option labeled **Delete**. The **Delete** menus has several submenus in order to delete the following objects.
- **Delete Reach:** This option is used to delete a reach. This is accomplished by selecting the **Delete Reach** option from the **Edit** menu. A list box containing all the available reaches will appear allowing you to select those reaches that you would like to delete. **Warning - Be careful when you delete reaches. When you delete a reach, all of its associated data will be deleted also.**
 - **Delete Junction:** This option is used to delete a junction. This is accomplished by selecting the **Delete Junction** option from the **Edit** menu. A list box containing all the available junctions will appear allowing you to select those junctions that you would like to delete.
 - **Delete Storage Areas/2D Flow Areas:** This option is used to delete a storage area or a 2D Flow Area. This is accomplished by selecting **Delete Storage Areas/2D Flow Areas** from the **Edit** menu. A selection box will appear allowing you to pick the storage areas or 2D Flow Areas that you would like to delete.
 - **Delete SA/2D Area Connections:** This option is used to delete a storage area or a 2D Flow Area hydraulic connection. This is accomplished by selecting the **Delete SA/2D Area Connections** option from the **Edit** menu. A list box containing all the available storage area and 2D Flow Area connections will appear allowing you to select the ones that you would like to delete.

- **Delete Pump Station:** This option allows the user to select one or more pump stations to be deleted from the schematic. This is accomplished by selecting **Delete Pump Station** from the **Edit** menu. A list box containing all the available pump stations will appear allowing you to select the ones that you would like to delete.
- **Delete Nodes (XS, Bridges, Culverts, etc...):** This option allows the user to delete multiple locations at one time. For example, you can delete multiple cross sections at one time with this option. When this option is selected, a window will appear allowing you to select all of the nodes (cross sections, bridges, culverts, Inline structures, lateral structures, etc.) that you would like to delete.
- **Delete SA/2D Flow Area Boundary Condition lines:** This option is used to delete a storage area or a 2D Flow Area Boundary Condition lines (BC Lines). This is accomplished by selecting the **Delete SA/2D Area Boundary Condition lines** option from the **Edit** menu. A list box containing all the available storage area and 2D Flow Area boundary condition lines will appear allowing you to select the ones that you would like to delete.
- **Delete 2D Flow Area Breaklines:** This option allows the user to delete previously drawn 2D Flow Area Breaklines.
- **Delete 2D Flow Manning's Regions:** This option allows the user to delete previously drawn 2D Flow Area Manning's n value regions.

Interacting with the Schematic

In addition to modifying the river schematic, there are options available from the **View** menu to zoom in, zoom previous, zoom out, full plot, pan, set schematic plot extents, and to display or not, many of the river system schematic labels. Additionally, the user has the ability to use the mouse to interact with the schematic. This is accomplished by moving the mouse pointer over an object (river reach line, junction, bridge, culvert, etc...) on the schematic and pressing down the left mouse button. Once the left mouse button is pressed down, a popup menu will appear with options that are specific to that type of object. For example, when the left mouse button is pressed down over a cross section, a menu will appear allowing the user to select options to: edit the cross section, plot the cross section, plot the profile for the reach that the cross section is in, display tabular output for the cross section, and plot the computed rating curve for that cross section.

Another way of interacting with the schematic is to press the right mouse button while the mouse pointer is located anywhere over the schematic drawing area. This will bring up a popup menu that is exactly the same as the View menu at the top of the drawing. This option is providing for convenience in getting to the View menu options.

Cool Graphics Window Tools: Most of the HEC-RAS graphical windows have some cool short cut tools. These tools include the following options:

- **Measuring Tool:** On any of the graphical windows (Geometric Schematic, Profile Plot, Cross Section Plot, etc...) if you hold down the **Cntrl Key**, you will get a measuring tool. The measuring tool can be used to draw multi-point lines (polyline) on the graphic window. To draw a polyline, hold down the **Cntrl Key** and then use single clicks of the left mouse button to start and add points to the line. Double click the left mouse button to end the line. Once you end the line, a window will pop up giving you information about that line, such as: the length of the line; the area if the first and last point were connected to form a polygon; the X-axis length; the Y-axis length; and the slope of the line (dx/dy , from the first point to the last point). Additionally, the X and Y coordinates of the line get sent to the Windows Clipboard, so you can paste those coordinates into a table of some other application. This is a very handy feature for digitizing the coordinates of a cross section, measuring a length, or estimating a slope (i.e. on the profile plot).
- **Pan Tool:** When you are zoomed in on the graphic within a window, if you hold down the **Shift Key**, the mouse pointer will change to a hand icon, and you can pan the graphic window. Releasing the **Shift Key** will change the mouse point back to the original icon.

- **Mouse Wheel Feature:** Now on any of the HEC-RAS graphical windows, the mouse wheel can be used to **Zoom In** and **Zoom Out**. Additionally the graphic will be re-centered based on where the mouse pointer is when using the mouse wheel to zoom in and out.

The following is a list of options available from the **View** menu:

- **Zoom In:** This option allows the user to zoom in on a piece of the schematic. This is accomplished by selecting **Zoom In** from the **View** menu, then specifying the area to zoom in on with the mouse. Defining the zoom area is accomplished by placing the mouse pointer in the upper left corner of the desired area. Then press down on the left mouse button and drag the mouse to define a box containing the desired zoom area. Finally, release the left mouse button and the viewing area will display the zoomed in schematic. Also displayed will be a small box in the upper right corner of the viewing area. This box will contain a picture of the entire schematic, with a rectangle defining the area that is zoomed in. In addition to showing you where you are at on the schematic, this zoom box allows you to move around the schematic without zooming out and then back in. To move the zoomed viewing area, simply hold down the left mouse button over the rectangle in the zoom box and move it around the schematic. The zoom box can also be resized. Resizing the zoom box is just like resizing a window.
- **Zoom Previous:** When this option is selected the program will go back to the previously defined viewing window of the schematic. For example, if the user zooms in on the display window of the geometric data editor, then selects the **Zoom Previous** option, the schematic drawing area will be put back to the previous display area before the last zoom in. The Zoom Previous option will remember up to the last 10 drawing rectangles displayed in the schematic window, so the user can select this option several times in a row to get back to a previous view.
- **Zoom Out:** This option zooms out to an area that is twice the size of the currently zoomed in window. Zooming out is accomplished by selecting **Zoom Out** from the **View** menu on the geometric data window.
- **Full Plot:** This option re-draws the plot to its full original size. The Full Plot option is accomplished by selecting **Full Plot** from the **View** menu on the geometric data window.
- **Pan:** This option allows the user to move around when in a zoomed in mode. The pan option is accomplished by selecting **Pan** from the **View** menu of the geometric data window. When this option is selected, the mouse pointer will turn into a hand. Press the left mouse button and hold it down, then move the mouse. This will allow the user to move the zoomed in graphic. To turn the pan mode off, re-select the pan option from the view menu. A short cut way to use the pan option is to simply hold down the **Shift Key** while the mouse is over the schematic. This will change the pointer to a hand graphic. Hold down the left mouse button and move the graphic. To stop panning, and change the pointer back to normal, release the Shift key.
- **Set Schematic Plot Extents:** This option allows the user to set the extents of the viewing area for the river system schematic. The user can enter a specific coordinate system, or utilize the default data system. The default data plot extents are from 0 to 1 for both the X and Y axis.
- **Find:** This option allows the user to have the interface locate a specific feature on the schematic. This is especially useful when very large and complex river systems are being modeled.

The following **View** menu options can be found on a new popup window by selecting **View**, then **View Options**. When **View Options** is selected, a window labeled **Geometry Plot Options** will appear that will allow you to toggle various objects on and off, such as: cross section properties; storage area/2D flow area properties; and Text labels.

Cross Section Properties:

- **Bank Stations:** This option allows the user to display the main channel bank stations on the cross section lines of the schematic. This is accomplished by checking **Display Bank Stations** from the **Geometry Plot Options** window.
- **Ineffective Areas:** This option allows the user to display the location of ineffective flow areas on top of the cross section lines of the schematic. This is accomplished by selecting **Display Ineffective Areas** from the **Geometry Plot Options** window.

- **Levees:** This option allows the user to display the location of levees on the cross section lines of the schematic. This is accomplished by selecting **Display Levees** from the **Geometry Plot Options** window.
- **Obstructions:** This option allows the user to display the location of blocked obstructions on the cross sections lines of the schematic. This is accomplished by selecting **Display Obstructions** from the **Geometry Plot Options** window.
- **XS Direction Arrows:** This option allows the user to display arrows along the cross sections in the direction in which they were extracted. This option is useful when you have coordinates defined for the cross section, such that the software can detect the direction that the cross section was extracted. Cross-sections are supposed to be entered from left to right while looking downstream. If a cross section has not been entered in this manner, it should be reversed. HEC-RAS has an option to reverse the cross section stationing. This option can be found under the **Tools** menu bar of the geometric data editor. To display the cross section direction arrows, select **Display XS Direction Arrows** from the **Geometry Plot Options** window.
- **Display Ratio of Cut Line Length to XS Length:** This option will display a ratio next to each cross section. The ratio represents the length of the cross section cut line (based on the GIS coordinates) divided by the length of the cross section (based on station/elevation points). If this number is greater than 1.0 then the GIS cross section cut line is longer than the station/elevation points of the cross section. If this number is less than 1.0, then the cross section cut line is shorter than the length of the cross section station/elevation points. When the value is exactly 1.0, then the cross section cut line and the station elevations points are consistent with each other.

Storage Area/2D Flow Area Properties:

- **Fill in Storage Areas/2D Flow Areas:** This option allows the user to turn on and off the fill in color for the storage areas and the 2D Flow Areas. Turning this off is very useful when a background picture is loaded.
- **2D Flow Area Cell Points:** This option turns on the black points that represent the 2D Cell centers.
- **2D Flow Area Cell Point Numbers:** This option turns on the numbering scheme for the 2D Flow Area cells.
- **2D Flow Areas Boundary Face Point Numbers:** This option allows you to display the Face Point numbers of a 2D flow area on the schematic. To use this option select **2D Flow Area Face Point Numbering** from the **Geometry Plot Options** window.
- **Display Break Lines Used in 2D Flow Areas:** This option turns the display of the 2D Flow Area Breaklines on and off.
- **Display Land Cover Calibration Regions:** This option turns the Land Cover regions on and off.

Text Labels:

- **Disable Text:** HEC-RAS has several options for display text labels on the river system schematic. This option will turn all of the text labels off or on simultaneously. This option can be turned on or off by selecting **Disable Text** from the **Geometry Plot Options** window.
- **River and Reach:** This option allows the user to display text labels for the River and Reaches. This is accomplished by selecting **Display River and Reach Text** from the **Geometry Plot Options** window.
- **River Stationing:** This option allows you to display river station identifiers on the schematic. This is accomplished by selecting **Display River Stationing Text** from the **Geometry Plot Options** window.
- **Node Names:** This option can be used to display User entered Node Names that may have been added to cross sections or hydraulic structures. Node Names are longer text labels that can be added to any node to further describe that location. User's can add and change node names from the **Tables** menu option, then select **Names**, then **Node Names**.
- **Storage Area/2D Flow Area Names:** This option allows you to display the text labels for storage areas and 2D Flow Areas. To use this option select **Display Storage Area/2D Flow Area Text** from the **Geometry Plot Options** window.
- **SA/2D Area Connection Names:** This option allows you to display the text labels for storage area connections. To use this option check the **SA/2D Area Connection Names** from the **Geometry Plot Options** window.
- **BC Line Names:** This option allows the user to turn the text labels for 2D Flow Area boundary conditions on and off.
- **Breakline Names:** This option allows the user to turn the text labels for 2D Flow Area breaklines on and off.

- **Land Cover Region Names:** This option allows the user to turn the text labels for Land Cover Regions on and off.
- **Pump Station Names:** This option allows the user to turn the text labels for pump stations on and off.
- **Junction Names:** This option allows you to display the text labels for Junctions. To use this option select **Junction Names** from the **Geometry Plot Options** window.
- **Results:** This option allows the user to display water surface or flow rate results, as numeric values, directly on the schematic. This option works in conjunction with the "**Plot WS extents for Profile**" option, which is available in the upper right hand corner of the Geometric Data editor window. If a user first turns on a specific profile to plot for the **Plot WS extents for Profile** option, then the computed water surface extents will be plotted in blue on top of each cross section. Additionally, if a user then checks either of the **Results** option from the **View** menu, they can choose to have the interface also plot numeric values for the cross section stage and flow rate, right next to the text label of all the cross sections.

Highlight:

- **Highlight Active Node:** This option will put a red circle around the active node (cross section, bridge, culvert, etc...) on the river system schematic. This option can be very handy when working with large complex data sets. The active node changes every time the user selects a new node from an editor or graphical plot.
- **Adjust Current Extents to Ensure Active Node is Visible:** This option will move the viewing area of the stream system schematic to ensure that the active node is in the view. When fully zoomed out, this option has no effect. When zoomed in, the viewing area will move to show the active node. To turn this option on select **Adjust Current Extents to Ensure Active Node is Visible** from the **View** menu.

Background Map Layers



Another option available to users is the ability to display background images and terrain data behind the river system schematic. To display terrain data and other map layers in the Geometric data editor, the user must use HEC-RAS Mapper to do the following:

1. Establish a Horizontal Coordinate Projection to use for your model, from within HEC-RAS Mapper. This is normally done by selecting an existing projection file from an ESRI shapefile or other mapping layer.
2. Develop a terrain model in HEC-RAS Mapper. The terrain model is a requirement for 2D modeling, as it is used to establish the geometric and hydraulic properties of the 2D cells and cell faces. A terrain model is also need in order to perform any inundation mapping in HEC-RAS Mapper.
3. Build a Land classification data set within HEC-RAS Mapper in order to establish Manning's n values within the 2D Flow Areas (Optional). Additionally HEC-RAS has option for user defined polygons that can be used to override the Land Classification data or as calibration zones.
4. Add any additional mapping layers that may be needed for visualization, such as aerial photography, levee locations, road networks, etc...

Once you have accomplished the necessary steps outlined above, then you can display the terrain data and available map layers developed in RAS Mapper within the Geometric data editor. This is accomplished by selecting the **Background Map Layer/Pictures** button at the top of the Geometry editor. Pressing this button will bring up a window showing you the available Map Layers and terrain that can be turned on or off by checking that layer.

Cross Section Data

After the river system schematic is completed, the next step for the modeler is to enter the cross section data. Cross section data represent the geometric boundary of the stream. An accurate representation of the stream channel and the overbank area (floodplain) are absolutely necessary to create an accurate hydraulics model. Cross sections are located at relatively short intervals along the stream to characterize the flow carrying capacity of the stream channel and its adjacent floodplain. Cross sections are required at representative locations throughout the stream and at locations where changes occur in discharge, slope, shape, roughness, at locations where levees begin and end, at hydraulic structures (bridges, culverts, inline weirs/spillways, and lateral weirs/spillways), and closely spaced around stream junctions. Cross sections can be entered by hand in the Geometric editor, or they can be extracted from terrain data from RAS Mapper.

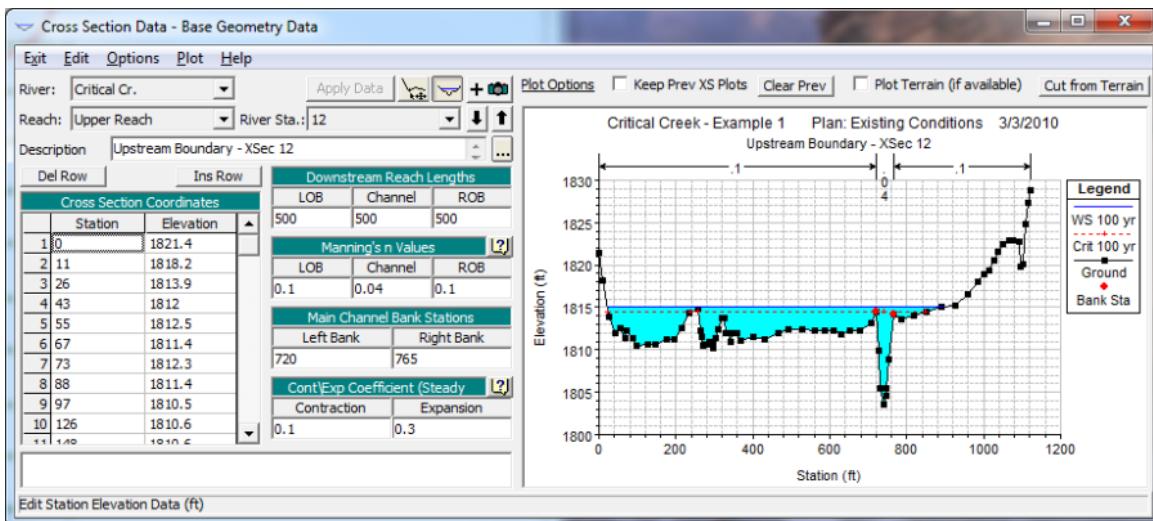
Entering Data



To enter cross section data directly in the Geometric Data editor, the user presses the **Cross Section** button on the Geometric Data window. Once the cross section button is pressed, the Cross Section Data Editor will appear as shown in the figure below (Except yours will be blank until you have added some data). To add a cross section to the model, the user must do the following:

1. From the Cross Section Editor, select the river and the reach that you would like to place the cross section in. This is accomplished by pressing the down arrow on the River and Reach boxes, and then selecting the river and reach of choice.
2. Go to the **Options** menu and select **Add a new Cross Section** from the list. An input box will appear prompting you to enter a river station identifier for the new cross section.
3. Enter all of the required data for the new cross section. Required data is the data that is openly displayed in the cross section editor window.
4. Enter any desired optional information (i.e., ineffective flow areas, levees, blocked obstructions, etc.). Optional cross section information is found under the **Options** menu.
5. Press the **Apply Data** button in order for the interface to accept the data. The apply data button does not save the data to the hard disk, it is used as a mechanism for telling the interface to use the information that was just entered. If you want the data to be saved to the hard disk you must do that from the **File** menu on the geometric data window.

The required information for a cross section consists of: the river, reach and river station identifiers; a description; X & Y coordinates (station and elevation points); downstream reach lengths; Manning's roughness coefficients; main channel bank stations; and contraction and expansion coefficients. All of the required information is displayed openly on the Cross Section Data editor (Figure 5-2). A description of this information follows:



Cross Section Data Editor

Cross section data entered in the manner described above is not geo-referenced (Does not have any horizontal coordinates describing its spatial location on the earth). If you want to have a model that is georeferenced, currently you must develop your model schematic and object layout using HEC-GeoRAS and ArcGIS. Future versions of HEC-RAS will allow you to do this directly inside of HEC-RAS using RAS Mapper.

- ⓘ If you want to do any inundation mapping of the model results, or if you want to use 2D Flow Areas, your model must be georeferenced, and you must bring terrain data into HEC-RAS from within HEC-RAS Mapper.

River, Reach, and River Station. The River and Reach boxes allow the user to select a specific hydraulic reach from the available reaches in the schematic diagram. The river and reach labels define which reach the cross section will be located in. The River Station tag defines where the cross section will be located within the specified reach. The river station tag does not have to be the actual river station of the cross section, but it must be a numeric value. Cross sections are ordered in the reach from highest river station upstream to lowest river station downstream. The up and down arrow buttons next to the river station box can be used to sequentially move through the river stations.

Description. The description box is used to describe the cross section location in more detail than just the river, reach, and river station. This box has a limit of 512 characters. The first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description can be used as a label for cross section plots and tables.

Cross Section Coordinates. This table is used to enter the station and elevation information of the cross section. Station and elevation information is entered in feet (meters for metric). Cross section stationing must be in increasing order. However, two or more stations can have the same value to represent vertical walls.

- ⓘ The cross section stationing (x-coordinates) must be entered from left to right looking in the downstream direction.

Downstream Reach Lengths. The downstream cross section reach lengths describe the distance between the current cross section and the next cross section downstream. Cross section reach lengths are defined for the left overbank, main channel, and the right overbank. Cross section reach lengths are entered in feet (meters for metric).

Manning's n Values. At a minimum, the user must specify Manning's n values for the left overbank, main channel, and the right overbank. Alternative roughness options are available from the **Options** menu, in which you can have horizontally varying Manning's n values (up to 20 across each cross section) and vertically varying Manning's n values.

Main Channel Bank Stations. The main channel bank stations are used to define what portion of the cross section is considered the main channel and what is considered left and right overbank area. The bank stations must correspond to stations entered on the cross section X & Y coordinates table. If the user enters a value that does not correspond to the station points of the cross section, the interface will ask the user if they would like the value to be automatically interpolated and added to the cross section data.

Contraction & Expansion Coefficients (Steady Flow). Contraction and expansion coefficients are used to evaluate the amount of energy loss that occurs because of a flow contraction or expansion. The coefficients are multiplied by the change in velocity head from the current cross section to the next downstream cross section. In other words, the values entered at a particular cross section are used to compute losses that occur between that cross section and the next downstream cross section. Values entered at this location are used for steady flow hydraulic computations only. Values for Unsteady Flow modeling can be entered from the Tables menu of the Geometric data editor.

Once all of the required data for the cross section are entered, make sure you press the **Apply data** button to ensure that the interface accepts the data that was just entered.



Plotting the Cross Section. A display of the cross section can be seen directly from the cross section editor. Simply press the cross section plot button at the top of the editor to either display or un-display the cross section graphic.



Attaching and Viewing Pictures at Cross Sections. To attach or view a previously attached picture to a cross section, press the picture button at the upper right hand side of the cross section editor. When this button is pressed, the picture viewer will pop up. You can attach a photo to a location, delete a picture, and view a currently attached photo.

Graphical Cross Section Editing. To jump to the Graphical Cross Section editor, press the button just to the right of the Apply Data button. This will bring up the current cross section in the Graphical Cross Section editor, and allow you to graphically edit it. To learn more about Graphical Cross Section editing, view the section called "Graphical Cross Section Editing" under the Geometric Data Tools section of this manual.

Editing Data

The bulk of the cross section data is the station and elevation information. There are several features available under the **Edit** menu to assist the user in modifying this information. These features include the following:

Undo Editing. This editing feature applies to all of the information on the cross section data editor. Once data has been entered and the **Apply Data** button has been pressed, the **Undo Editing** feature is activated. If any changes are made from this point, the user can get the original information back by selecting the **Undo Edit** option from the **Edit** menu. Once the Apply Data button is pressed, the new information is considered good and the **Undo Edit** feature is reset to the new data.

Cut, Copy, and Paste. Cut, Copy, and Paste features are available for the station and elevation information on the cross section editor. These features allow the user to pass cross section station and elevation data to and from the Windows Clipboard. To use this feature, first highlight a cell or multiple cells on the station and elevation table. Cells are highlighted by pressing down on the left mouse button and moving it over the cells that you would like to be highlighted. Next select either the **Cut** or **Copy** feature from the **Edit** menu. If **Cut** is selected, the information is placed in the Windows Clipboard and then it is deleted from the table. If **Copy** is selected, the information is placed in the Windows Clipboard, but it also remains in the table. Once the information is in the Windows Clipboard it can be pasted into the station and elevation table of any cross section. To paste data into another cross section, first go to the cross section in which you would like the data to be placed. Highlight the area of the table in which you want the data to be placed. Then select the **Paste** option from the **Edit** menu. The cut, copy, and paste features can also be used to pass station and elevation information between HEC-RAS and other programs.

Delete. This option allows the user to delete a single cell or multiple cells in the station/elevation table. Once the cells are deleted, everything below those cells is automatically moved up. To use this option, first highlight the cells that you would like to delete, then select the **Delete** option from the **Edit** menu. If you would like to clear cells, without moving the data below those cells, simply highlight the cells and press the delete key.

Insert. This option allows the user to insert one or several rows in the middle of existing data in the station/elevation table. To use this option, first highlight the area in the table that you would like to be inserted. Then select **Insert** from the **Edit** menu. The rows will be inserted and all of the data will be moved down the appropriate number of rows. The user can also insert a single row by placing the cursor in the row just below where you would like the new row to be inserted. Then select **Insert** from the **Edit** menu. The row will be inserted and all of the data below the current row will be moved down one row.

Options

Information that is not required, but is optional, is available from the **Options** menu at the top of the cross section data editor window (Figure 5-2). Options consist of the following:

Add a new Cross Section. This option initiates the process of adding a cross section to the data set. The user is prompted to enter a river station tag for the new cross section. The river station tag locates the cross section within the selected reach. Once the river station is entered, the cross section data editor is cleared (except for some default values that get set) and the user can begin entering the data for the cross section. Whenever a new cross section is added to the data set, default values will appear for the contraction and expansion coefficients (0.1 and 0.3 respectively). Also, if the new cross section is not the first or most upstream cross section of the reach, the program will set default Manning's n values equal to the n values of the cross section just upstream of the new cross section. If the user does not want these default values, they can simply change them to whatever values they would like.

Copy Current Cross Section. This option allows the user to make a copy of the cross section that is currently displayed in the editor. When this option is selected, the user is prompted to select a river and reach for the new section, and then enter the river station. Once the information is entered, the new cross section is displayed in the editor. At this point it is up to the user to change the description and any other information about the cross section. This option is normally used to make interpolated cross sections between two surveyed cross sections. Once the section is copied, the user can adjust the elevations and stationing of the cross section to adequately depict the geometry between the two surveyed sections.

Rename River Station. This option allows the user to change the River Station of the currently displayed cross section.

Delete Cross Section. This option will delete the currently displayed cross section. The user is prompted with a message stating specifically which cross section is going to be deleted, and requesting the user to press the **OK** button or the **Cancel** button. Once the OK button is pressed, the user will be prompted with a question of whether or not they would like the cross section reach lengths to be automatically adjusted to account for the removal of the cross section. If the user answers **YES** then the reach lengths of the current cross section, that is being deleted, will be added to the reach lengths of the next upstream cross section. If the user answers **NO**, then the cross section will be deleted without adjusting any reach lengths.

Adjust Elevations. This option allows the user to adjust all of the elevations of the currently displayed cross section. Positive or negative elevation changes can be entered. Once the value is entered, the interface automatically adjusts all the elevations in the table.

Adjust Stations. This option allows the user to adjust the stationing of the currently displayed cross section. Two options are available. The first option (**Multiply by a Factor**) allows the user to separately expand and/or contract the left overbank, main channel, and the right overbank. When this option is selected, the user is prompted to enter a multiplier for each of the three flow elements (left overbank, main channel, and right overbank). If the multiplier is less than one, the flow element is contracted. If the multiplier is greater than one, the flow element is expanded. Once the information is entered, and the user hits the **OK** button, the interface automatically performs the contraction and/or expansions. The cross section should be reviewed to ensure that the desired adjustments were performed. The second option (**Add a Constant**) allows the user to add or subtract a constant value from all the stations in the cross section. This would allow the entire cross section to be shifted to the right or the left.

Adjust n or k Values. This option allows the user to either increase or decrease all the n or k values of the current cross section. The user is prompted for a single value. This value is then used as the multiplier for all of the n or k values of the current cross section.

Skew Cross Section. This option allows the user to adjust the stationing of a cross section based on a user entered skew angle. Cross-sections are supposed to be taken perpendicular to the flow lines. This may not always be the case, such as at bridges. In order for the program to use the correct flow area, the cross section stationing must be adjusted by taking the cosine of the skew angle times the stationing. When this option is selected, a window will appear allowing the user to enter a skew angle. Once the angle is entered, the software will automatically adjust the cross section stationing. The user can get back to the original stationing by putting a zero skew into the field.

Ineffective Flow Areas. This option allows the user to define areas of the cross section that will contain water that is not actively being conveyed (ineffective flow). Ineffective flow areas are often used to describe portions of a cross section in which water will pond, but the velocity of that water, in the downstream direction, is close to or equal to zero. This water is included in the storage calculations and other wetted cross section parameters, but it is not included as part of the active flow area. An example of an ineffective flow area is shown in Figure 6.3. The cross-hatched area on the left of the plot represents the ineffective flow area.

Two alternatives are available for setting ineffective flow areas. The first option allows the user to define a left station and elevation and a right station and elevation (**normal ineffective areas**).

When this option is used, and if the water surface is below the established ineffective elevations, the areas to the left of the left station and to the right of the right station are considered ineffective.

Once the water surface goes above either of the established elevations, then that specific area is no longer considered ineffective. In other words, the program now assumes that the area will be conveying water in the downstream direction, such that it now uses that area in the conveyance calculations of the active flow area. However, the user has the option to set the ineffective flow areas to permanent, which will prevent them from turning off. When this option is used, water is allowed to go over top of the ineffective flow area.

The second option allows for the establishment of **blocked ineffective flow areas**. Blocked ineffective flow areas require the user to enter an elevation, a left station, and a right station for each ineffective block. Up to ten blocked ineffective flow areas can be entered at each cross section. Once the water surface goes above the elevation of the blocked ineffective flow area, the blocked area is no longer considered ineffective. However, the user has the option to set the blocked ineffective flow areas to permanent, which will prevent them from turning off. When this option is used, water is allowed to go over top of the ineffective flow area.

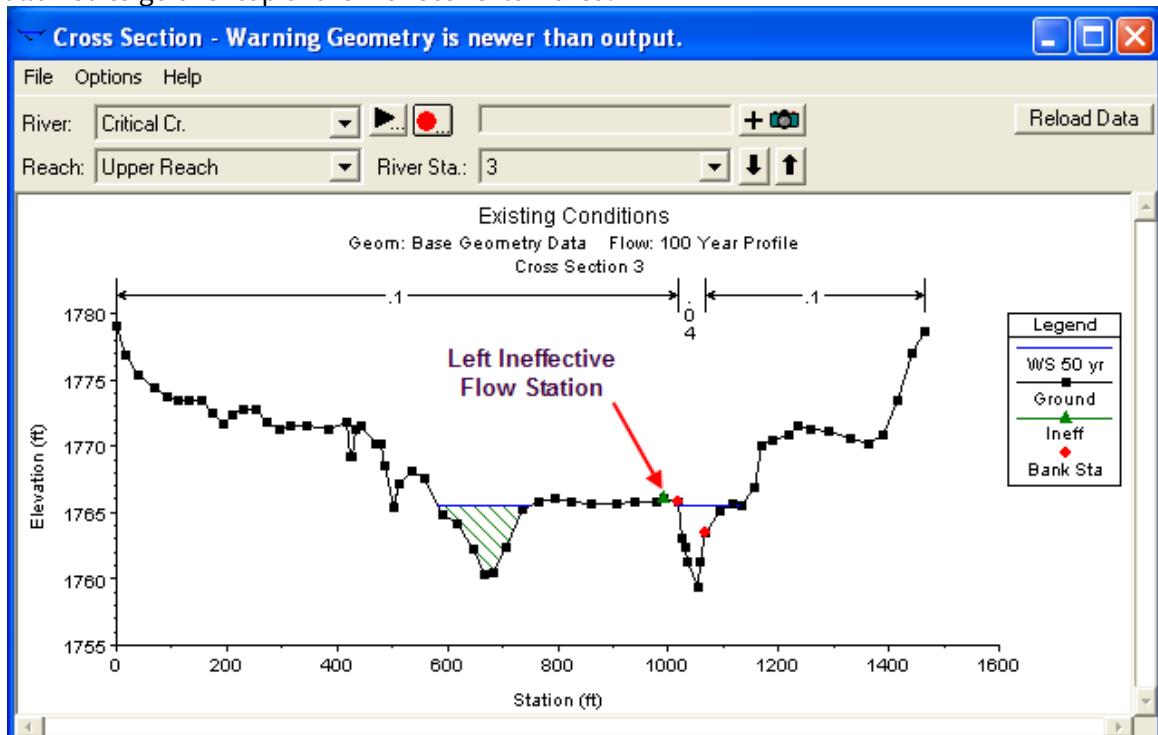
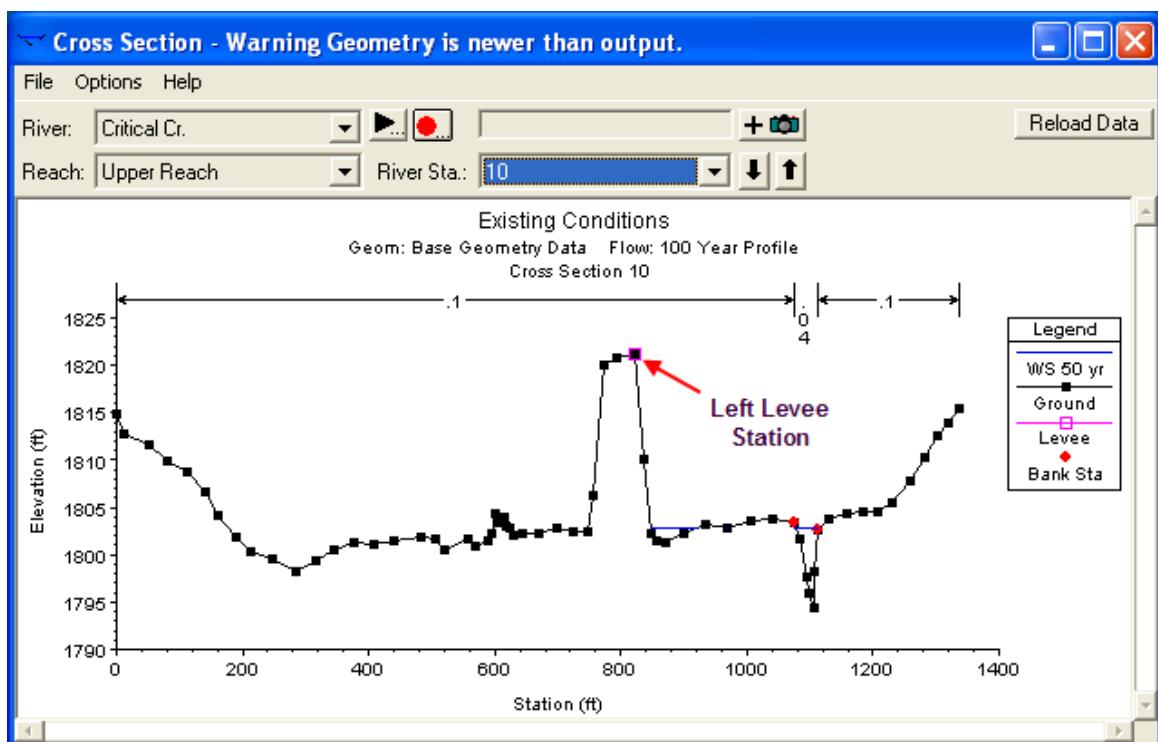


Figure 5.3 Cross section with ineffective flow areas

Levees. This option allows the user to establish a left and/or right stationing and elevation on any cross section, in which the water surface will be confined to inside of these stationing values, until the water surface goes above their trigger elevation (elevation at which this option turns off). When levees are established, no water can go to the left of the left levee station or to the right of the right levee station until either of the levee elevations is exceeded. Levee stations must be defined explicitly, or the program assumes that water can go anywhere within the cross section. An example of a cross section with a levee on the left side is shown in the figure below. In this example the levee station and elevation is associated with an existing point on the cross section.

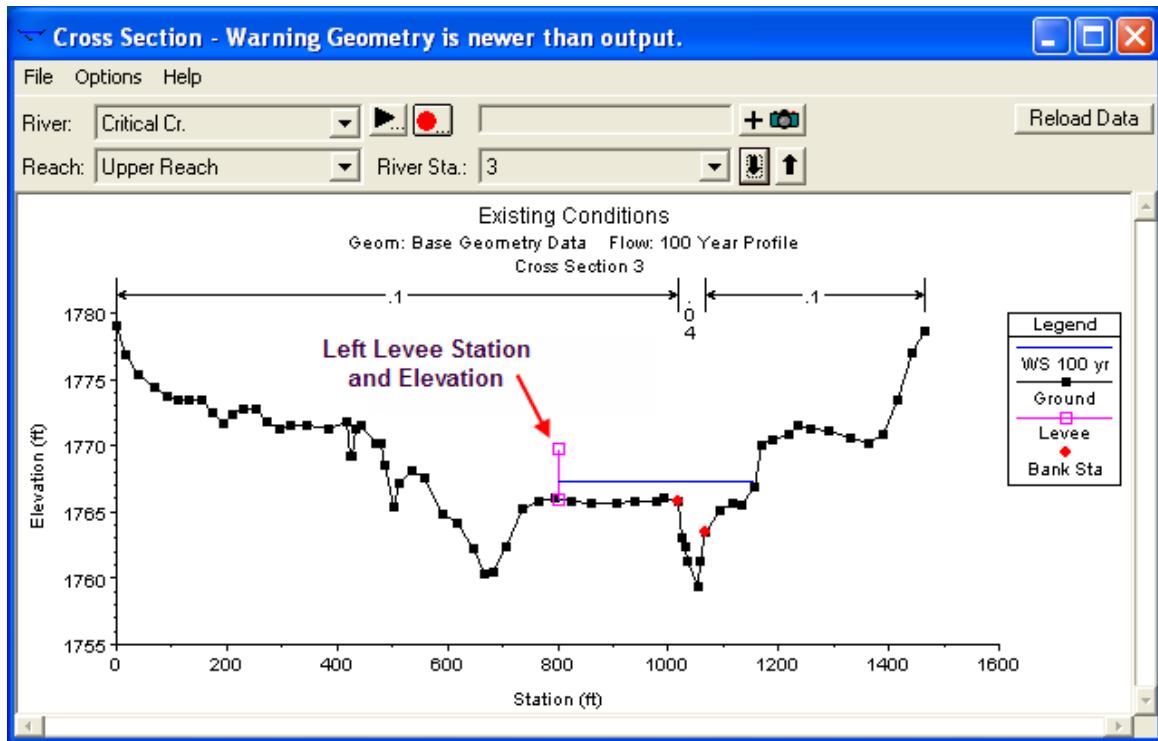
- ① The Levee option is a cross section option to prevent water from getting to certain portions of the cross section, until the water level rises above user specified elevations. However, the water surface is assumed to be horizontal across the entire wetted portion of the cross section. If you have a constructed levee, and you want to model over topping or breaching of that levee, you should end your cross section at the top of that levee. Model the actual levee with a lateral structure, and model the area behind the levee with a 2D Flow Area, Storage Area, or another river reach. This approach is much more accurate for modeling constructed levees, as it allows for different water surface elevations to be computed inside the leveed area, and a much more detailed analysis of levee overtopping and breaching.



Example of the Levee Option

The user may want to add levees into a data set in order to see what effect a levee will have on the water surface. A simple way to do this is to set a levee station and elevation that is above the existing ground. If a levee elevation is placed above the existing geometry of the cross section, then a vertical

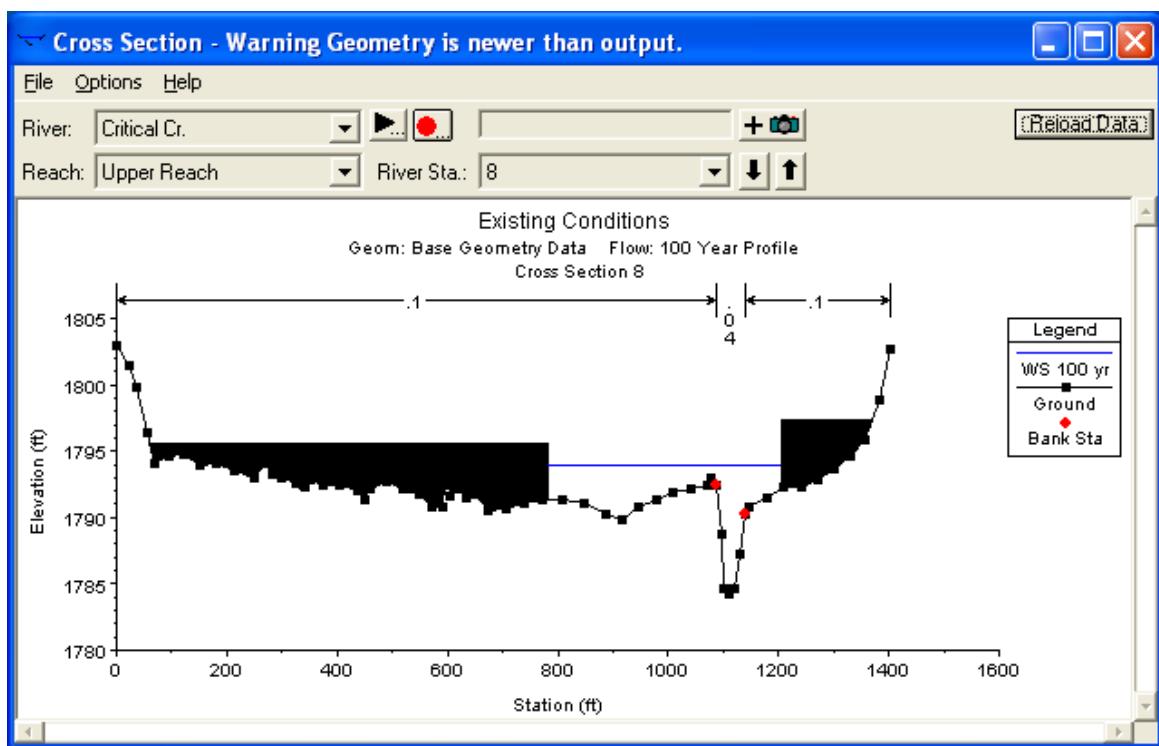
wall is placed at that station up to the established levee height. Additional wetted perimeter is included when water comes into contact with the levee wall. An example of this is shown below.



Example Levee Added to a Cross Section

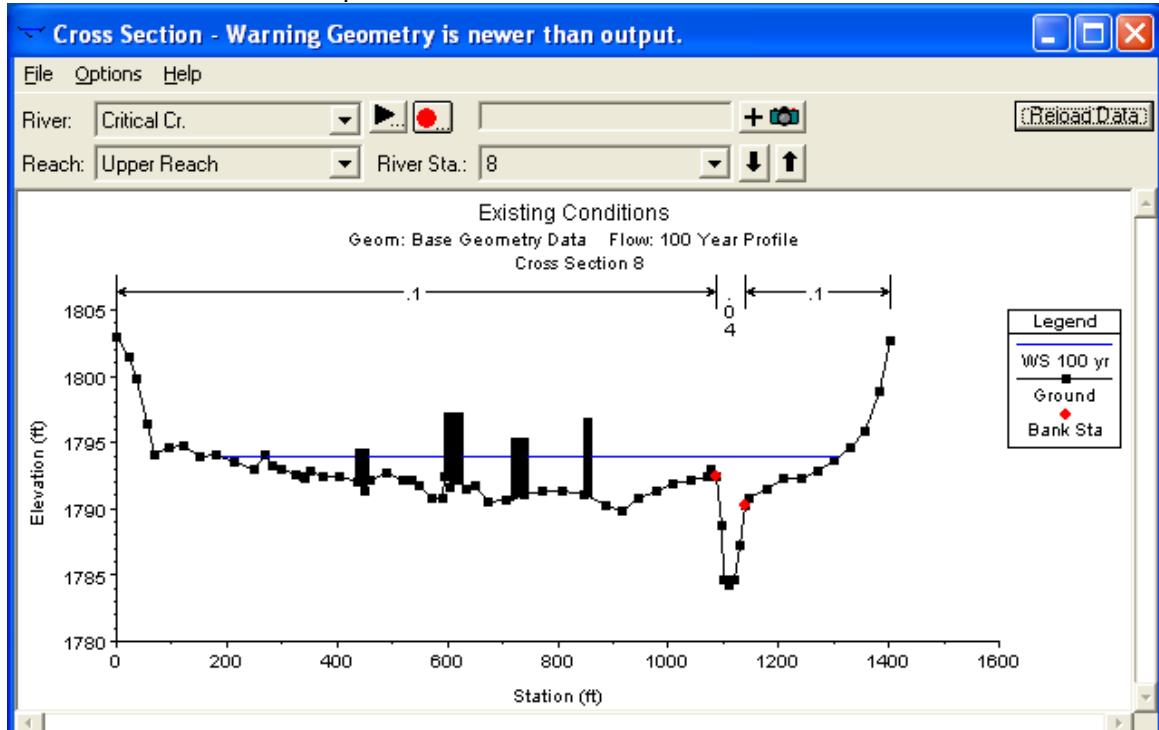
Obstructions. This option allows the user to define areas of the cross section that will be permanently blocked out. Obstructions decrease flow area and add wetted perimeter when the water comes in contact with the obstruction. An obstruction does not prevent water from going outside of the obstruction.

Two alternatives are available for entering obstructions. The first option allows the user to define a left station and elevation and a right station and elevation (**normal obstructions**). When this option is used, the area to the left of the left station and to the right of the right station will be completely blocked out. An example of this type of obstruction is shown in the figure below.



Example of Normal Obstructions

The second option, for obstructions, allows the user to enter up to 20 individual blocks (**blocked obstructions**). With this option the user enters a left station, a right station, and an elevation for each of the blocks. An example of a cross section with blocked obstructions is shown below.



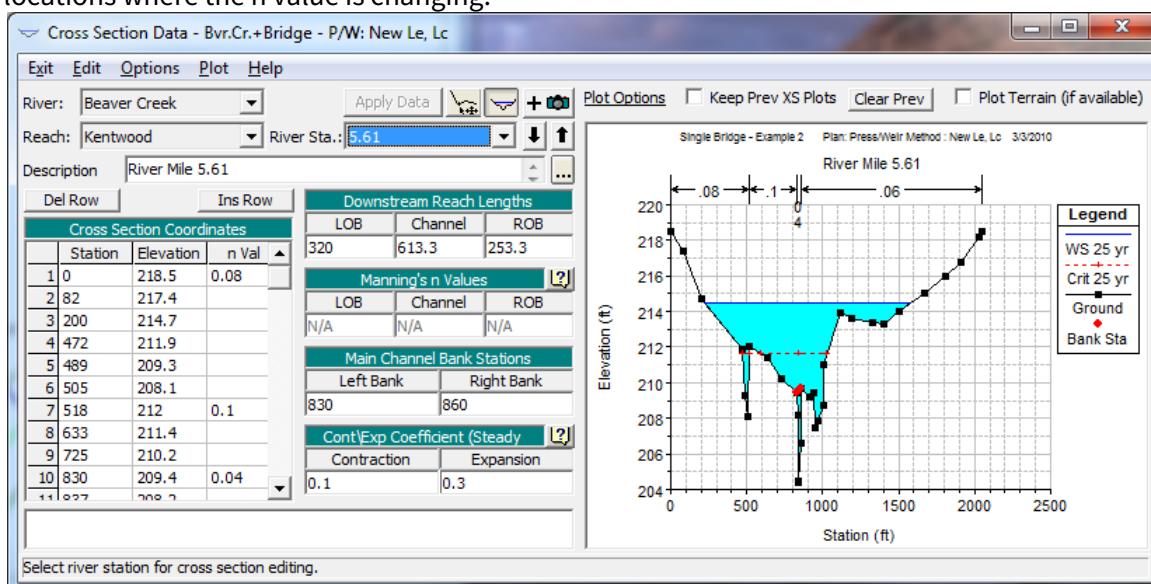
Example of a Cross Section with Blocked Obstruction

Add a Lid to XS. This option allows the user to add a lid (similar to a bridge deck/roadway) to any cross section. This is commonly used when trying to model a long tunnel. The ground geometry can be used to describe the bottom half of the tunnel, while the lid can describe the top half. A lid can be added to any number of cross sections in a row. The program treats cross sections with lids just like any other cross section. The energy equation is used to balance a water surface, with the assumption of open channel flow. The only difference is that the program will subtract out area and add wetted perimeter when the water surface comes into contact with the lid. For unsteady flow models, there is a check box to select the **Preissmann Slot** option when solving the unsteady flow equations for cross sections with lids. This option allows the unsteady flow equations to solve for a pressure flow water surface using the open channel flow equations.

Add Ice Cover. This option allows the user to enter ice cover for the currently opened cross section. For a detailed discussion of ice cover, and ice modeling, please review the section called **Modeling Ice Cover** later in this chapter.

Add a Rating Curve. This option allows the user to add a rating curve to a cross section as an alternative to the program computing the water surface. The user is required to enter flow versus elevation information for the rating curve. When the program is executed in a steady flow mode, the program will interpolate a water surface elevation from the rating curve for the given flow of a particular profile.

Horizontal Variation in n Values. This option allows the user to enter more than three Manning's n values for the current cross section. When this option is selected, an additional column for n values is added to the cross section coordinates table as shown in the figure below. A Manning's n value must be placed in the first row of the table. This n value is good for all cross section stations until a new n value shows up in the table. The user does not have to enter an n value for every station, only at the locations where the n value is changing.



Cross Section with Horizontal Variation of n Values Selected

Horizontal Variation in k Values. This option allows the user to enter k values (roughness heights) instead of n values. The k values are entered in the same manner as the horizontal variation of n

values. To learn more about k values and how they are used in the program, see Chapter 3 of the Hydraulic Reference manual.

Vertical Variation in n Values. This option allows the user to enter Manning's n values that vary both horizontally as well as vertically. The user can vary the n value either by elevation or by flow. When this option is selected a window will appear as shown the figure below. The user enters the stationing for horizontal changes in n values across the top in row 0 (these stations are entered in the same manner as the horizontal variation of Manning's n value option). The elevations in which changes occur are entered in the first column. Then the actual Manning's n values are entered in rows 1-20 (columns 2-21). The program will interpolate Manning's n values whenever the actual water surface is between the entered elevations. If the water surface is below the first elevation entered, then the values from that elevation will be used. Likewise, if the water surface is above the last elevation entered, the program will use the n values from the last elevation specified. No extrapolation is done on either side of the user entered values.

Vertical Variation in Manning's n Values

Vertical n based on Water Surface Vertical n based on Flow

Row 0: Starting Stations		Rows 1-20: Mannings n Values				
	Elev\Sta	Station1	Station2	Station3	Station4	Station5
0	0	512	830	860		
1	204	.08	0.1	0.04	0.06	
2	210	.08	0.1	0.038	0.06	
3	214	.07	0.09	0.035	0.06	
4	220	.065	0.08	0.032	0.05	
5						

Enter Manning's n data for station: 860 at elevation: 220

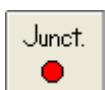
Vertical Variation of Manning's n Values Window

Plotting

Once all the data have been entered for a cross section, you should plot the cross section to inspect it for possible data errors. To plot the current cross section from the cross section editor, you can either select **Plot Cross Section** from the **Plot menu** (this will bring up a separate plot window), or you can use the cross section plot button at the top of the cross section editor (this attaches a plot window to the cross section editor).

Stream Junctions

Entering Junction Data



Stream junctions are defined as locations where two or more streams come together or split apart. Junction data consist of a description; reach lengths across the junction; tributary angles; and

modeling approach. To enter junction data the user presses the **Junction** button on the Geometric Data window (Figure 5-1). Once the junction button is pressed, the junction editor will appear as shown in Figure 5-10.

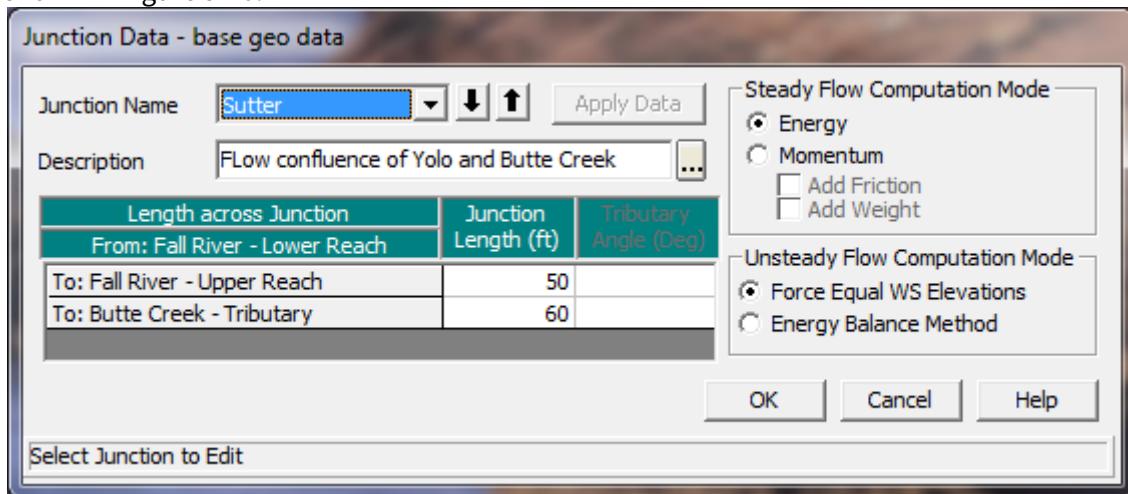


Figure 5 10 Junction Data Editor

The junction editor will come up with one of the junctions loaded. Fill out the description and reach lengths for the junction. Reach lengths across the junction are entered here instead of the cross section data editor. This allows for the lengths across very complicated confluences (i.e., flow splits) to be accommodated. In the cross section data, the reach lengths for the downstream cross section of each reach upstream of the junction will be overridden by the lengths in the junction editor.

- ① When laying out cross sections around a junction (Upstream and downstream on the main stem river and tributaries connected to the junction), place the cross sections as close to the junction as possible. This is especially important for unsteady flow modeling, as the default computational option is that the model assumes the same water surface elevation at all cross sections bounding the junction. If this is a bad assumption, turn on the option labeled "**Energy Balance Method**" under the **Unsteady Flow Computational Method**. Cross sections laid out very far from the junction can lead to model stability issues if the elevation of the channel bottom for the cross sections that bound the junction are very different (Have very different invert elevations).

Selection a Modeling Approach

For **steady flow hydraulics** in HEC-RAS, a junction can be modeled by either the energy equation or the momentum equation. The energy equation does not take into account the angle of a tributary coming in or leaving, while the momentum equation does. In most cases the amount of energy loss due to the angle of the tributary flow is not significant, and using the energy equation to model the junction is more than adequate. However, there are situations where the angle of the tributary can cause significant energy losses. In these situations it would be more appropriate to use the momentum approach. When the momentum approach is selected, an additional column is added to the table next to the junction lengths. This column is used to enter an angle for any river reach that is coming into or exiting the main river. For the reaches that are considered to be the main river, the angle should be left blank or set to zero. Also, the user has the option to turn friction and weight

forces on or off during the momentum calculations. The default is to have the weight force turned off.

For **unsteady flow hydraulics** there are two options for modeling the hydraulics at a junction. The default option makes some simplifying assumptions for the hydraulics at a junction. If the junction is a normal flow combining junction, then all cross sections that bound the junction are given the same water surface each time step, based on the computed water surface at the downstream side of the junction. If the junction is a flow split the water surfaces at the junction are based on the computed water surface at the upstream side of the junction. This simplifying assumption requires user's to place cross sections fairly close together around a junction, depending on the slope of the stream. If cross sections are too far apart, model stability problems can arise from the force water surfaces at all cross sections that bound the junction.

A new junction hydraulics option called the **Energy Balance Method** has been added for unsteady flow modeling. When this option is turned on, an energy balance is performed across the junction in order to compute the water surfaces, rather than forcing them to all be the same. This is a very useful option for medium to steep streams, or where junction reach lengths are fairly lengthy.

If there is more than one junction in the river schematic, the other junctions can be selected from the Junction Name box at the upper left corner of the window. Enter all the data for each junction in the river system then close the window by pressing the **OK** button in the lower left corner of the window. When the junction data editor is closed the data are automatically applied.

Bridges and Culverts



Once all of the necessary cross-section data have been entered, the modeler can then add any bridges or culverts that are required. HEC-RAS computes energy losses caused by structures such as bridges and culverts in three parts. One part consists of losses that occur in the reach immediately downstream from the structure where an expansion of flow takes place. The second part is the losses at the structure itself, which can be modeled with several different methods. The third part consists of losses that occur in the reach immediately upstream of the structure where the flow is contracting to get through the opening.

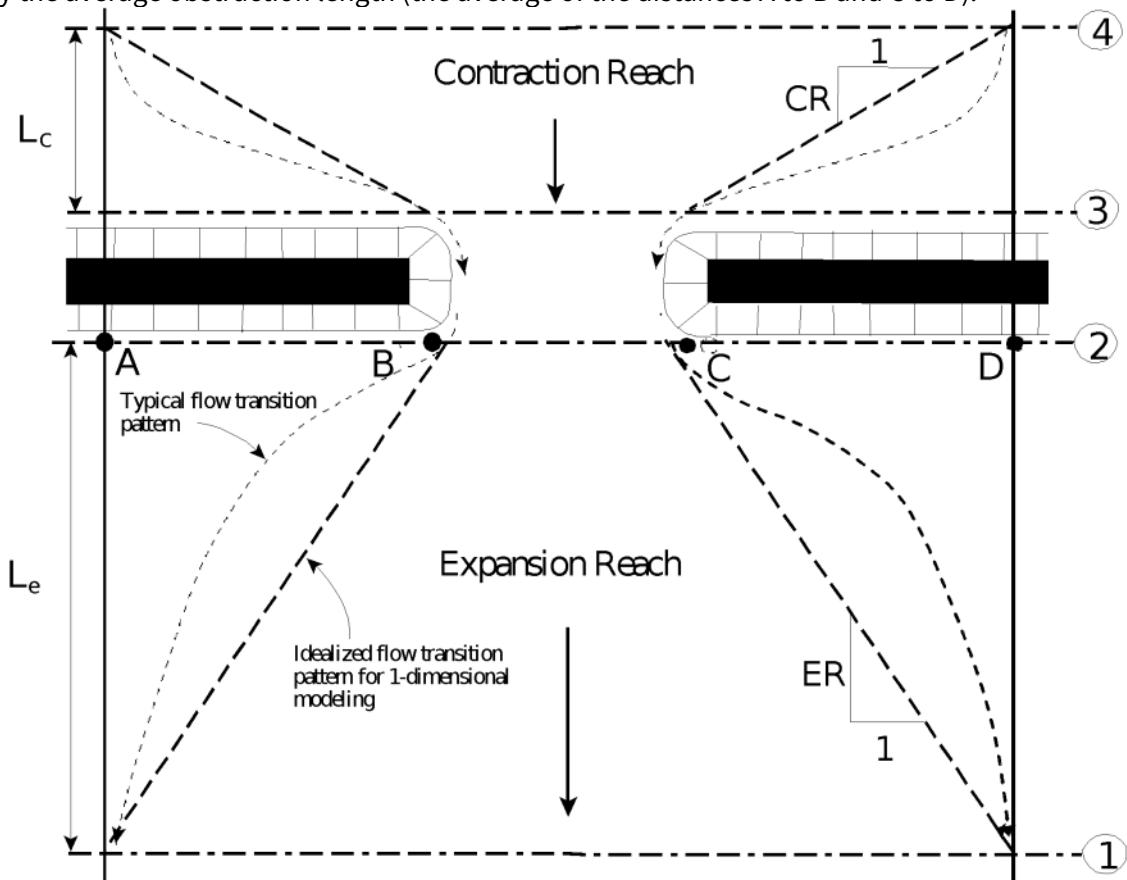
The bridge routines in HEC-RAS allow the modeler to analyze a bridge with several different methods without changing the bridge geometry. The bridge routines have the ability to model low flow (Class A, B, and C), low flow and weir flow (with adjustments for submergence), pressure flow (orifice and sluice gate equations), pressure and weir flow, and high flows with the energy equation only. The model allows for multiple bridge and/or culvert openings at a single location.

The culvert hydraulics in HEC-RAS are based on the Federal Highway Administrations (FHWA) standard equations from the publication Hydraulic Design of Highway Culverts (FHWA, 1985), for inlet control situations, and a detailed energy balance into, through, and out of the culvert, for outlet control computations. The culvert routines include the ability to model circular, box, elliptical, arch, pipe arch, low profile arch, high profile arch, semi circular culverts, and ConSpan culvert shapes. The HEC-RAS program has the ability to model multiple culverts at a single location. The culverts can have different shapes, sizes, elevations, and loss coefficients. The user can also specify the number

of identical barrels for each culvert type. Culverts can also be buried into the ground and have different roughness coefficients for the bottom, versus the top and sides.

Cross Section Locations

The bridge and culvert routines utilize four user defined cross sections in the computations of energy losses due to the structure. A plan view of the basic cross section layout is shown in the figure below. **Cross section 1** is located sufficiently downstream from the structure so that the flow is not affected by the structure (i.e., the flow has fully expanded). This distance should generally be determined by field investigation during high flows. However, generally field investigation during high flows is not possible. The expansion distance will vary depending upon the degree of constriction, the shape of the constriction, the magnitude of the flow, and the velocity of the flow. If no detailed information is available, a rough estimate of a 2:1 expansion ratio can be used for a first cut estimate of the expansion reach length. Additionally, the table below offers ranges of expansion ratios, which can be used for different degrees of constriction, different slopes, and different ratios of the overbank roughness to main channel roughness. Once an expansion ratio is selected, the distance to the downstream end of the expansion reach (the distance L_e) is found by multiplying the expansion ratio by the average obstruction length (the average of the distances A to B and C to D).



Cross Section Locations at a Bridge or Culvert

The average obstruction length is half of the total reduction in floodplain width caused by the two bridge approach embankments. In table below, b/B is the ratio of the bridge opening width to the total floodplain width, n_{ob} is the average Manning n value for the overbanks, n_c is the n value for the

main channel, and *Slope* is the average longitudinal bed slope through the bridge reach. The values in the interior of the table are the ranges of the expansion ratio. For each range, the higher value is typically associated with a higher discharge.

Ranges of Expansion Ratios

Opening Ratio	Slope	nob / nc = 1	nob / nc = 2	nob / nc = 4
b/B = 0.10	1 ft/mile	1.4 – 3.6	1.3 – 3.0	1.2 – 2.1
	5 ft/mile	1.0 – 2.5	0.8 – 2.0	0.8 – 2.0
	10 ft/mile	1.0 – 2.2	0.8 – 2.0	0.8 – 2.0
b/B = 0.25	1 ft/mile	1.6 – 3.0	1.4 – 2.5	1.2 – 2.0
	5 ft/mile	1.5 – 2.5	1.3 – 2.0	1.3 – 2.0
	10 ft/mile	1.5 – 2.0	1.3 – 2.0	1.3 – 2.0
b/B = 0.50	1 ft/mile	1.4 – 2.6	1.3 – 1.9	1.2 – 1.4
	5 ft/mile	1.3 – 2.1	1.2 – 1.6	1.0 – 1.4
	10 ft/mile	1.3 – 2.0	1.2 – 1.5	1.0 – 1.4

A detailed study of flow contraction and expansions at bridges was undertaken by the Hydrologic Engineering Center. The results of this study have been published as a research document entitled "Flow Transitions in Bridge Backwater Analysis" (RD-42 HEC, 1995). The purpose of this study was to provide better guidance to hydraulic engineers performing water surface profile computations through bridges. Specifically the study focused on determining the expansion reach length, L_e ; the contraction reach length, L_c ; the expansion energy loss coefficient, C_e ; and the contraction energy loss coefficient, C_c . A summary of this research, and the final recommendations, can be found in Appendix B of the HEC-RAS Hydraulic Reference manual.

The user should not allow the distance between cross section 1 and 2 to become so great that friction losses will not be adequately modeled. If the modeler feels that the expansion reach will require a long distance, then intermediate cross sections should be placed within the expansion reach in order to adequately model friction losses. The user will need to estimate ineffective flow areas for these intermediate cross sections.

Cross section 2 is located a short distance downstream from the bridge or culvert. This cross section should represent the natural ground (main channel and floodplain) just downstream of the bridge or culvert. This section is normally located near the toe of the downstream road embankment. This cross section should **Not** be placed immediately downstream of the face of the bridge deck or the culvert opening (for example some people wrongly place this cross section 1.0 foot downstream of the bridge deck or culvert opening). Even if the bridge has no embankment, this cross section should be placed far enough from the downstream face of the bridge to allow enough distance for some flow expansion due to piers, or pressurized flow coming out of the bridge. If a culvert is being modeled, the culvert routines automatically account for an exit loss. Therefore, cross section 2 should be located far enough downstream from the culvert to capture the immediate expansion of flow in which the exit losses occur over. This distance will vary with the size of the bridge opening or culvert.

Cross section 3 should be located a short distance upstream from the bridge or culvert. This distance should only reflect the length required for the abrupt acceleration and contraction of the flow that occurs in the immediate area of the opening. Cross section 3 represents the natural ground

of the channel and overbank area just upstream of the road embankment. This section is normally located near the toe of the upstream road embankment. This cross section should **Not** be placed immediately upstream of the bridge deck or culvert opening (for example some people wrongly place this cross section 1.0 foot upstream of the bridge deck or culvert opening). The bridge and culvert routines used between cross sections 2 and 3 account for the contraction losses that occur just upstream of the structure (entrance losses for the culvert routines). Therefore, this cross section should be placed just upstream of the area where the abrupt contraction of flow occurs to get into the bridge opening or culvert. This distance will vary with the size of the bridge opening or culvert. Both cross sections 2 and 3 will have ineffective flow areas to either side of the bridge or culvert opening during low flow and pressure flow. In order to model only the effective flow areas at these two sections, the modeler should use the ineffective flow area option. This option is selected from the cross section data editor. For a detailed discussion of how to set the ineffective flow area stations and elevations, see Chapter 5 of the Hydraulic Reference manual.

Cross section 4 is an upstream cross section where the flow lines are approximately parallel and the cross section is fully effective. In general, flow contractions occur over a shorter distance than flow expansions. The distance between cross section 3 and 4 (the contraction reach length, L_c) should generally be determined by field investigation during high flows. Traditionally, the Corps of Engineers recommends locating the upstream cross section a distance equal to one times the average length of the side constriction caused by the structure abutments (i.e. 1:1 contraction ratio). The 1:1 contraction ratio is a reasonable first estimate for placing cross section 4 if no other detailed information or field data is available to further refine that estimate. The contraction distance will vary depending upon the degree of constriction, the shape of the constriction, the magnitude of the flow, and the velocity of the flow. As mentioned previously, the detailed study "Flow Transitions in Bridge Backwater Analysis" (RD-42, HEC, 1995) was performed to provide better guidance to hydraulic engineers performing water surface profile computations through bridges. A summary of this research, and the final recommendations, can be found in Appendix B of the HEC-RAS Hydraulic Reference manual.

When the user adds a bridge at a particular river station, the program automatically formulates two additional cross sections inside of the bridge structure. The geometry inside of the bridge is a combination of the bounding cross sections (2 and 3) and the bridge geometry. The bridge geometry consists of the bridge deck, abutments if necessary, and any piers that may exist. The user can specify different bridge geometry for the upstream and downstream sides of the structure if necessary. Cross section 2 and the structure information on the downstream side are used as the geometry just inside the structure at the downstream end. Cross section 3 and the upstream structure information are used as the bridge geometry just inside the structure at the upstream end. The user has the option to edit these internal bridge cross sections, in order to make adjustments to the geometry.

For a more detailed discussion on laying out cross sections around bridges and culverts, the user is referred to chapters 5 and 6 of the Hydraulic Reference Manual.

Contraction and Expansion Losses

Losses due to the contraction and expansion of flow between cross sections are determined during the standard step profile calculations. Contraction and Expansion losses are described in terms of coefficient times the absolute value of the change in velocity head between adjacent cross sections. When the velocity head increases in the downstream direction a contraction coefficient is used; and when the velocity head decreases in the downstream direction, an expansion coefficient is used. For

a detailed discussion on selecting contraction and expansion coefficients at bridges, the user is referred to chapter 5 of the HEC-RAS Hydraulic Reference Manual.

Bridge Hydraulic Computations

Low Flow Computations

For low flow computations the program first uses the momentum equation to identify the class of flow. This is accomplished by first calculating the momentum at critical depth inside the bridge at the upstream and downstream ends. The end with the higher momentum (therefore most constricted section) will be the controlling section in the bridge. The momentum at critical depth in the controlling section is then compared to the momentum of the flow downstream of the bridge when performing a subcritical profile (upstream of the bridge for a supercritical profile). If the momentum downstream is greater than the critical depth momentum inside the bridge, the class of flow is considered to be completely subcritical (i.e., class A low flow). If the momentum downstream is less than the momentum at critical depth in the bridge, then it is assumed that the constriction will cause the flow to pass through critical depth and a hydraulic jump will occur at some distance downstream (i.e., class B low flow). If the profile is completely supercritical through the bridge then this is class C low flow. Depending on the class of flow the program will do the following:

Class A low flow. Class A low flow exists when the water surface through the bridge is completely subcritical (i.e., above critical depth). Energy losses through the expansion (sections 2 to 1) are calculated as friction losses and expansion losses. Friction losses are based on a weighted friction slope times a weighted reach length between sections 1 and 2. The average friction slope is based on one of the four available alternatives in HEC-RAS, with the average-conveyance method being the default. This option is user selectable. The average length used in the calculation is based on a discharge-weighted reach length.

There are four methods for computing losses through the bridge (from 2 to 3):

- Energy equation (standard step method)
- Momentum balance
- Yarnell equation
- FHWA WSPRO method

The user can select any or all of these methods in the computations. If more than one method is selected, the user must choose either a single method as the final solution or tell the program to use the method that computes the greatest energy loss through the bridge as the answer at section 3. This allows the modeler to compare the answers from several techniques all in a single execution of the program. Minimal results are available for all the methods computed, but detailed results are available for the method that is selected as the final answer.

Energy losses through the contraction (sections 3 to 4) are calculated as friction losses and contraction losses. Friction and contraction losses between sections 3 and 4 are calculated the same as friction and expansion losses between sections 1 and 2.

Class B low flow. Class B low flow can exist for either subcritical or supercritical profiles. For either profile, class B flow occurs when the profile passes through critical depth in the bridge constriction. For a **subcritical profile**, the momentum equation is used to compute an upstream water surface above critical depth and a downstream water surface below critical depth, using a momentum balance through the bridge. For a **supercritical profile**, the bridge is acting as a control and is

causing the upstream water surface elevation to be above critical depth. Momentum is used again to calculate an upstream water surface above critical depth and a downstream water surface below critical depth. The program will proceed with forewater calculations downstream from the bridge. *Class C low flow.* Class C low flow exists when the water surface through the bridge is completely supercritical. The program can use either the energy or the momentum equation to compute the water surface through the bridge.

Pressure Flow Computations

Pressure flow occurs when the flow comes into contact with the low chord of the bridge. Once the flow comes into contact with the upstream side of the bridge, a backwater occurs and orifice flow is established. The program will handle two cases of orifice flow: the first is when only the upstream side of the bridge is in contact with the water; and the second is when the bridge constriction is flowing completely full. For the first case, a sluice gate type of equation is used, as described in "Hydraulics of Bridge Waterways" (FHWA, 1978). In the second case, the standard full flowing orifice equation is used. The program will begin checking for the possibility of pressure flow when the energy grade line goes above the maximum low chord elevation. Once pressure flow is computed, the pressure flow answer is compared to the low flow answer and the higher of the two is used. The user has the option to tell the program to use the water surface, instead of energy, to trigger the pressure flow calculation.

Weir Flow Computations

Flow over the bridge and the roadway approaching the bridge will be calculated using the standard weir equation. For high tailwater elevations the program will automatically reduce the amount of weir flow to account for submergence on the weir. This is accomplished by reducing the weir coefficient based on the amount of submergence. When the weir becomes highly submerged, the program will automatically switch to calculating losses based on the energy equation (standard step backwater). The criterion for when the program switches to energy based calculations is user controllable.

Combination Flow

Sometimes combinations of low flow or pressure flow occur with weir flow. In these cases an iterative procedure is used to determine the amount of each type of flow.

Entering and Editing Bridge Data



To enter bridge data the user presses the **Bridge/Culvert** button on the geometric data window (Figure 5-1). Once the bridge/culvert button is pressed, the Bridge/Culvert Data Editor will appear as shown in Figure 5-12 (your bridge/culvert editor will come up with a blank window until you have entered the bridge data). To add a bridge to the model, do the following:

1. Select the river and reach that you would like to place the bridge in. Selecting a reach is accomplished by pressing the down arrow on the river and reach box, then selecting the river and reach of choice.

2. Go to the **Options** menu and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new bridge.
3. Enter all of the required data for the new bridge. This includes:
 - Bridge Deck
 - Sloping Abutments (optional)
 - Piers (optional)
 - Bridge modeling approach information
1. Enter any desired optional information. Optional bridge information is found under the Options menu at the top of the window.
2. Press the **Apply Data** button for the interface to accept the data.

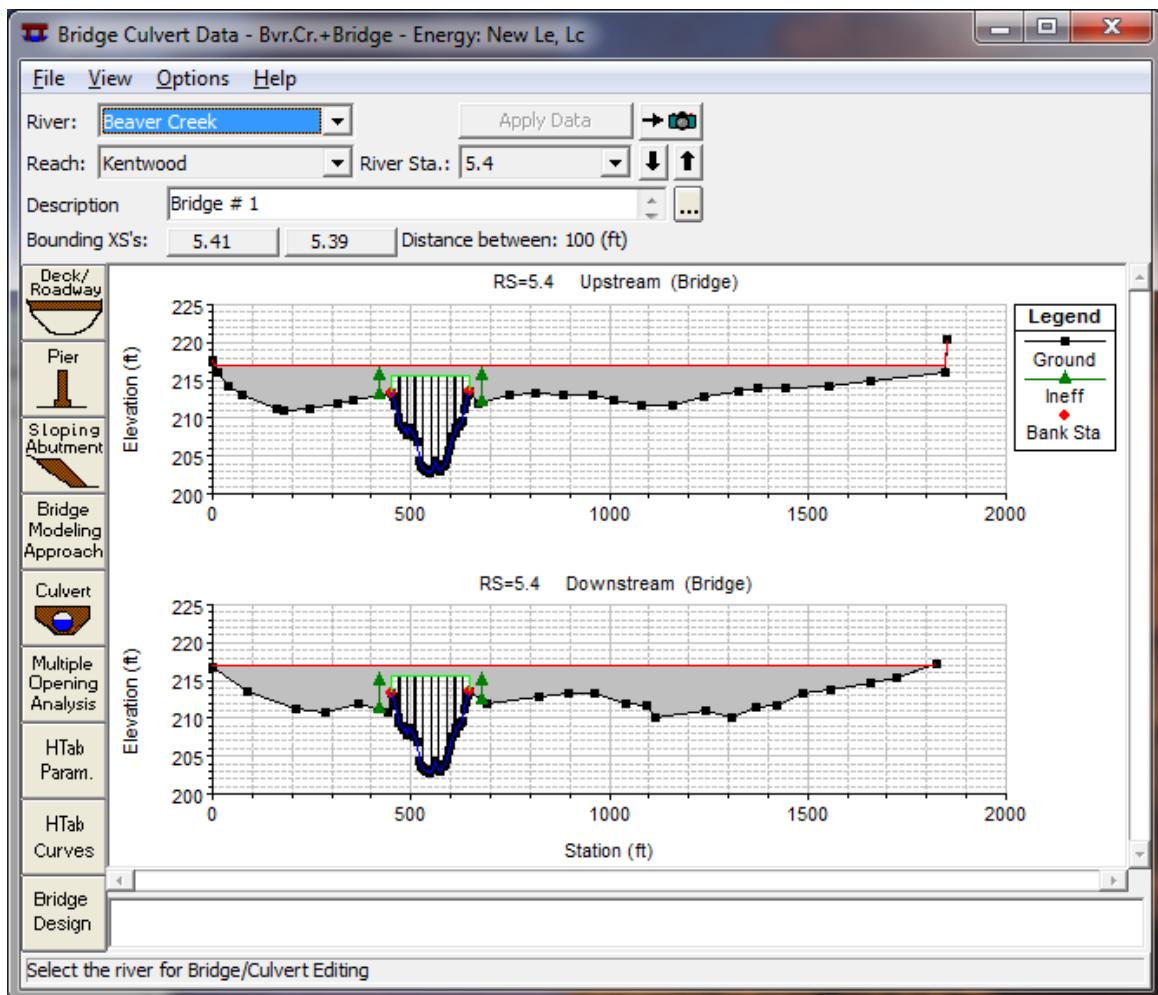


Figure 5 12 Bridge/Culvert Data Editor

The required information for a bridge consists of: the river, reach, and river station identifiers; a short description of the bridge; the bridge deck; bridge abutments (if they exist); bridge piers (if the bridge has piers); and specifying the bridge modeling approach. A description of this information follows:

River, Reach and River Station. The River and Reach boxes allow the user to select a river and reach from the available reaches that are defined in the schematic diagram. The reach label defines which reach the bridge will be located in. The River Station tag defines where the bridge will be located

within the specified reach. The river station tag does not have to be the actual river station of the bridge, but it must be a numeric value. The river station tag for the bridge should be numerically between the two cross sections that bound the bridge. Once the user selects Add a Bridge and/or Culvert from the options menu, an input box will appear prompting you to enter a river station tag for the new bridge. After the river station tag is entered, the two cross sections that bound the bridge will be displayed on the editor.

Description. The description box is used to describe the bridge location in more detail than just the reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for bridge plots and tables.

Bridge Deck/Roadway. The bridge deck editor is used to describe the area that will be blocked out due to the bridge deck, road embankment and vertical abutments. To enter bridge deck information the user presses the **Deck** button on the Bridge/Culvert Data Editor. Once the deck button is pressed, the Deck Editor will appear as in Figure 5-13 (except yours will be blank). The information entered in the deck editor consists of the following:

Upstream		Downstream				
	Station	high chord	low chord	Station	high chord	low chord
1	0.	216.93	200.	0.	216.93	200.
2	450.	216.93	200.	450.	216.93	200.
3	450.	216.93	215.7	450.	216.93	215.7
4	647.	216.93	215.7	647.	216.93	215.7
5	647.	216.93	200.	647.	216.93	200.
6	2000.	216.93	200.	2000.	216.93	200.
7						
8						

U.S Embankment SS D.S Embankment SS
 Weir Data
 Max Submergence: Min Weir Flow El:
Weir Crest Shape
 Broad Crested
 Ogee

Enter distance between upstream cross section and deck/roadway. (ft)

Figure 5 13 Bridge Deck/Roadway Data Editor

Distance - The distance field is used to enter the distance between the upstream side of the bridge deck and the cross section immediately upstream of the bridge (see Figure 5 14, "Upstream Distance"). This distance is entered in feet (or meters for metric).

Width - The width field is used to enter the width of the bridge deck along the stream (Figure 5 14, "Bridge Width"). The distance between the bridge deck and the downstream bounding cross section will equal the main channel reach length minus the sum of the bridge "width" and the "distance" between the bridge and the upstream section. The width of the bridge deck should be entered in feet (meters for metric).

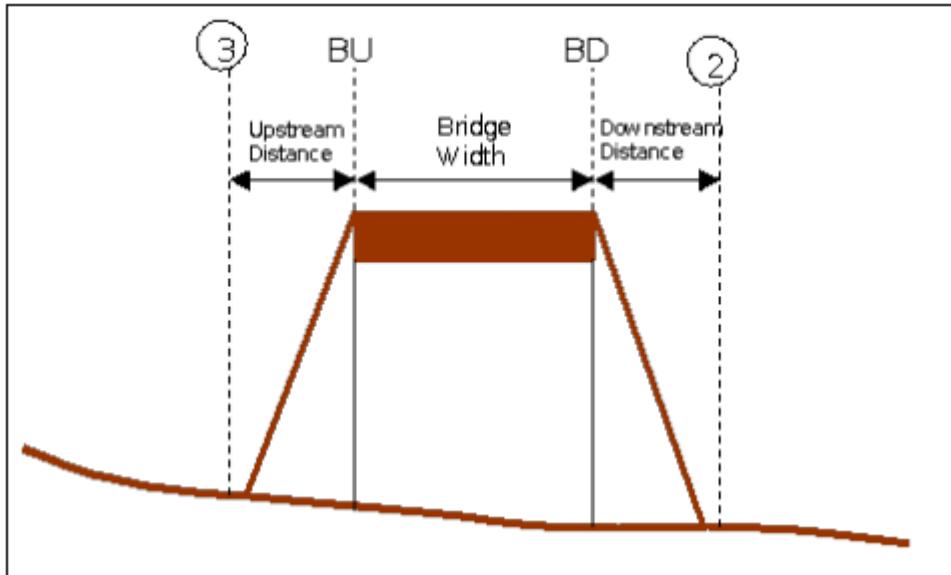


Figure 5 14 Bridge Profile with Upstream Distance, Bridge Width, and Downstream Distance

Weir Coefficient - Coefficient that will be used for weir flow over the bridge deck in the standard weir equation.

Upstream Stationing, High Chord, and Low Chord - This table is used to define the geometry of the bridge deck on the upstream side of the bridge. The information is entered from left to right in cross section stationing. The deck is the area between the high and low chord elevation information. The stationing of the deck does not have to equal the stations in the bounding cross section, but it must be based on the same origin. The **Del Row** and **Ins Row** buttons allow the user to delete and insert rows.

Downstream Stationing, High Chord, and Low Chord - This portion of the table is used to define the geometry of the bridge deck on the downstream side of the bridge. If the geometry of the downstream side is the same as the upstream side, then the user only needs to press the **Copy US to DS** button. When this button is pressed, all of the upstream bridge deck information is copied to the downstream side. If the bridge deck information on the downstream side is different than the upstream side, then the user must enter the information into the table.

U.S. Embankment SS - This field is used to enter the slope of the road embankment on the upstream side of the bridge. The slope should be entered as the horizontal to vertical distance ratio of the embankment. This variable is generally not used in the computations, but is used for display purposes in the profile plot. However, if the user has selected the FHWA WSPRO Bridge method for low flow, this field will be used in the computation of the bridge discharge coefficient.

D.S. Embankment SS - This field is used to enter the slope of the road embankment on the downstream side of the bridge. The slope should be entered as the horizontal to vertical distance ratio of the embankment. This variable is generally not used in the computations, but is used for

display purposes in the profile plot. However, if the user has selected the FHWA WSPRO Bridge method for low flow, this field will be used in the computation of the bridge discharge coefficient.

Max Submergence - The maximum allowable submergence ratio that can occur during weir flow calculations over the bridge deck. If this ratio is exceeded, the program automatically switches to energy based calculations rather than pressure and weir flow. The default value is 0.95 (95 percent submerged).

Submergence Criteria - When submergence occurs there are two choices available to figure out how much the weir coefficient should be reduced due to the submergence. The first method is based on work that was done on a trapezoidal shaped broad crested weir (FHWA, 1978). The second criterion was developed for an Ogee spillway shape (COE, 1965). The user should pick the criterion that best matches their problem.

Min Weir Flow El - This field is used to set the minimum elevation for which weir flow will begin to be evaluated. Once the computed upstream energy becomes higher than this elevation, the program begins to calculate weir flow. However, the weir flow calculations are still based on the actual geometry of the deck/roadway, and are not affected by this elevation. If this field is left blank, the elevation that triggers weir flow is based on the lowest high chord elevation on the upstream side of the bridge deck. Also, weir flow is based on the elevation of the energy grade line and not the water surface.

Once all of the bridge deck information is entered, the user should press the **OK** button at the bottom of the window. Pressing the **OK** button tells the interface to accept the data and close the window. Once the deck editor closes, the graphic of the bridge deck will appear on the Bridge/Culvert Data window. An example of this is shown in Figure 5-15. **Note! The data are not saved to the hard disk at this point.** Geometric data can only be saved to the hard disk from the **File** menu of the Geometric Data window.

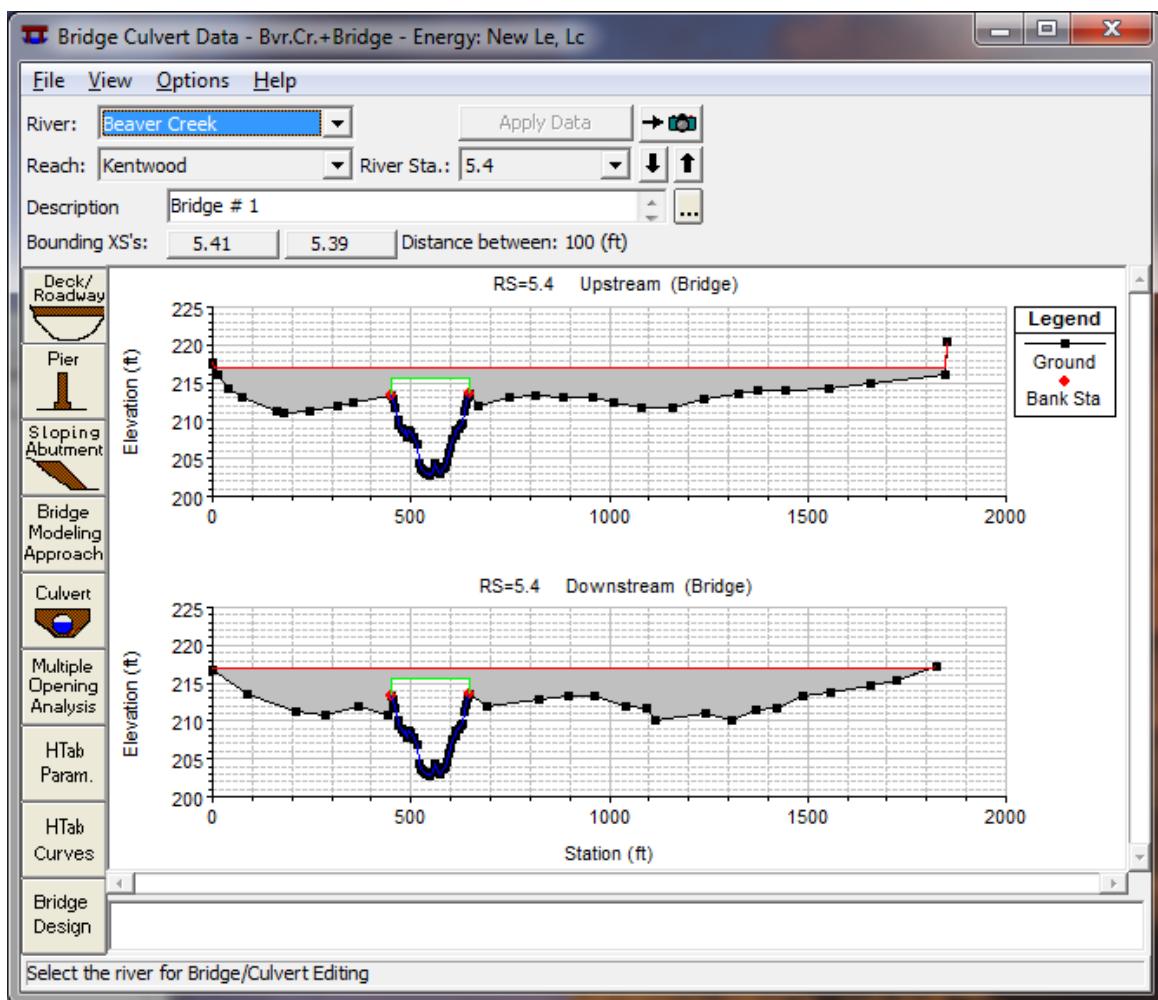


Figure 5 15 Example Bridge Deck Plotted on Bounding Cross Sections

Sloping Bridge Abutments. The sloping bridge abutments editor is used to supplement the bridge deck information. Whenever bridge abutments are protruding towards the main channel (sloping inward abutments), it will be necessary to block out additional area that cannot be accounted for in the bridge deck/roadway editor. If the bridge has vertical wall abutments, then it is not necessary to use this editor. Vertical wall abutments can be included as part of the bridge deck/roadway data. To add sloping abutments, the user presses the

Sloping Abutment button on the Bridge/Culvert Data editor. Once this button is pressed the Abutment data editor will appear as in Figure 5-16.

Sloping abutments are entered in a similar manner to the bridge deck/roadway. When the editor is open, it has already established an abutment # of 1. Generally a left and right abutment is entered for each bridge opening. Sloping abutment data are entered from left to right, looking in the downstream direction. In general it is usually only necessary to enter two points to describe each abutment.

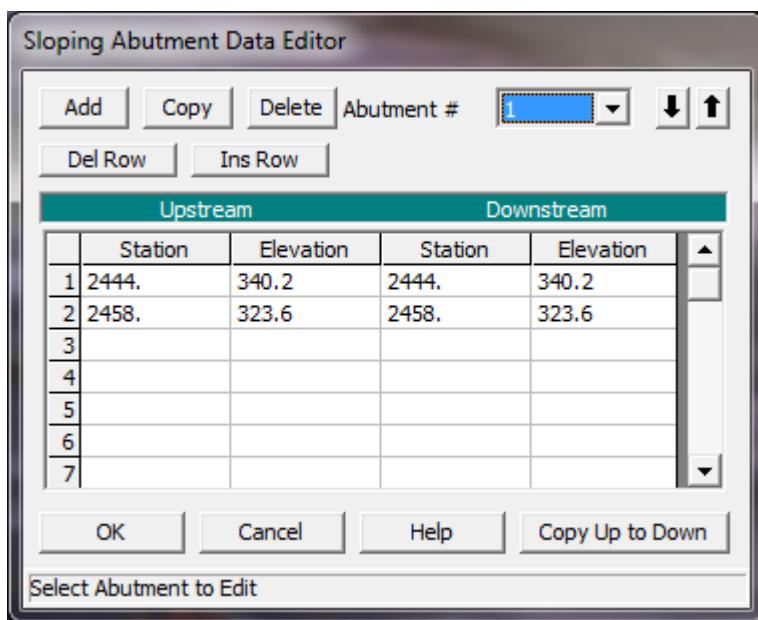


Figure 5 16 Abutment Data Editor

The data for each abutment consist of a skew angle (this is optional) and the station and elevation information. The station and elevation information represents the high chord information of the abutment. The low chord information of the abutment is assumed to be below the ground, and it is therefore not necessary to enter it. The geometric information for each abutment can vary from upstream to downstream. If this information is the same, then the user only needs to enter the upstream geometry and then press the **Copy Up to Down** button.

To add additional sloping abutments, the user can either press the **ADD** or the **Copy** button. To delete an abutment, press the **Delete** button. Once all of the abutment data are entered, the user should press the **OK** button. When the **OK** button is pressed, the abutment information is accepted and the editor is closed. The abutments are then added to the bridge graphic on the Bridge/Culvert Data editor. An example of a sloping bridge abutment is shown in Figure 5-17. This graphic is zoomed in on the left abutment of the bridge.

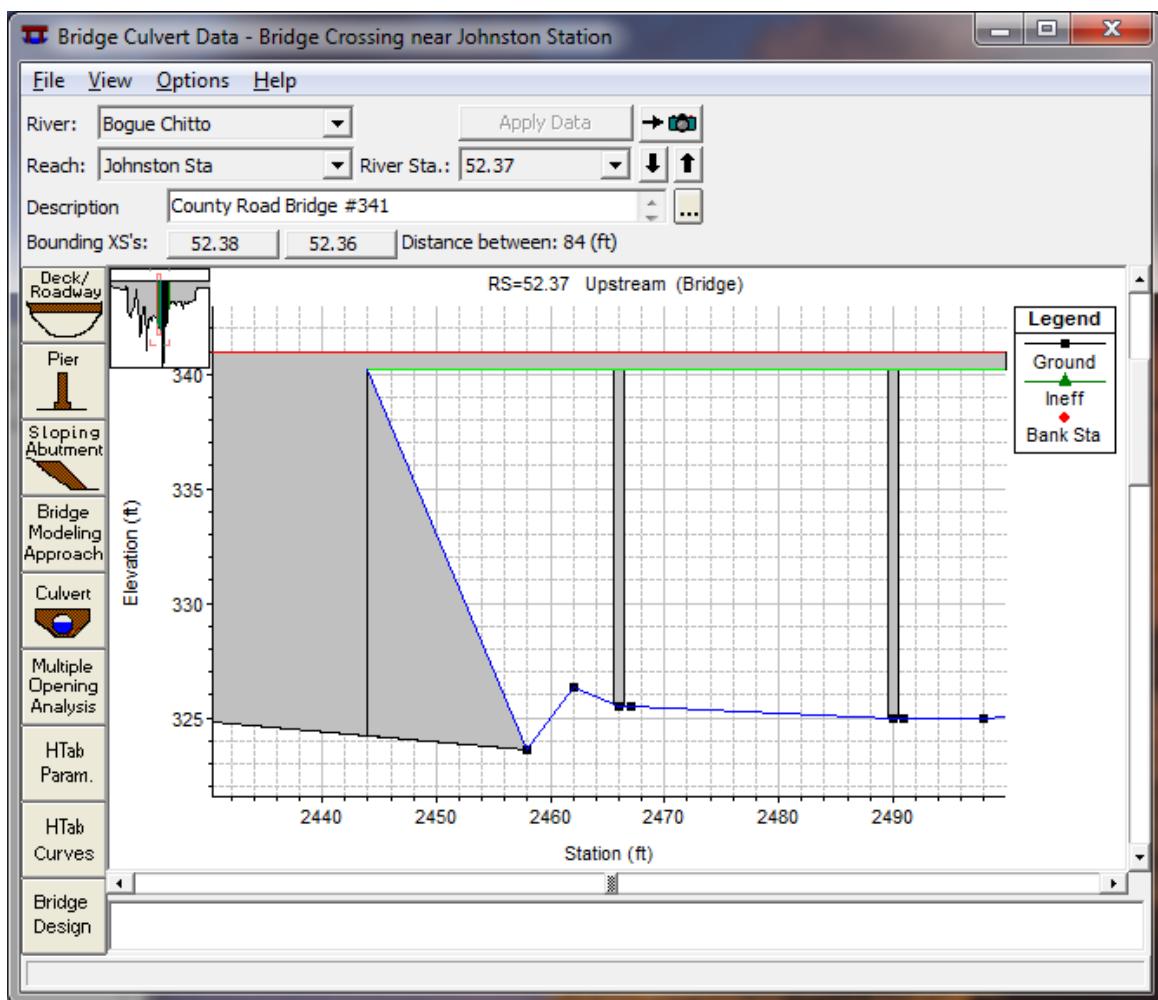


Figure 5 17 Example of a Sloping Abutment

Bridge Piers. The bridge pier editor is used to describe any piers that exist in the bridge opening.

Note! All piers must be entered through the Pier Editor, they should not be included as part of the ground or bridge deck. Several of the low flow bridge computations require that the piers be defined separately in order to determine that amount of area under the water surface that is blocked by the piers. If the piers are included with the ground or the bridge deck, several of the methods will not compute the correct amount of energy loss for the piers.

To enter pier information, the user presses the **Pier** button on the Bridge/Culvert Data editor. Once the pier button is pressed, the pier data editor will appear as in Figure 5-18 (Except yours will not have any data in it yet).

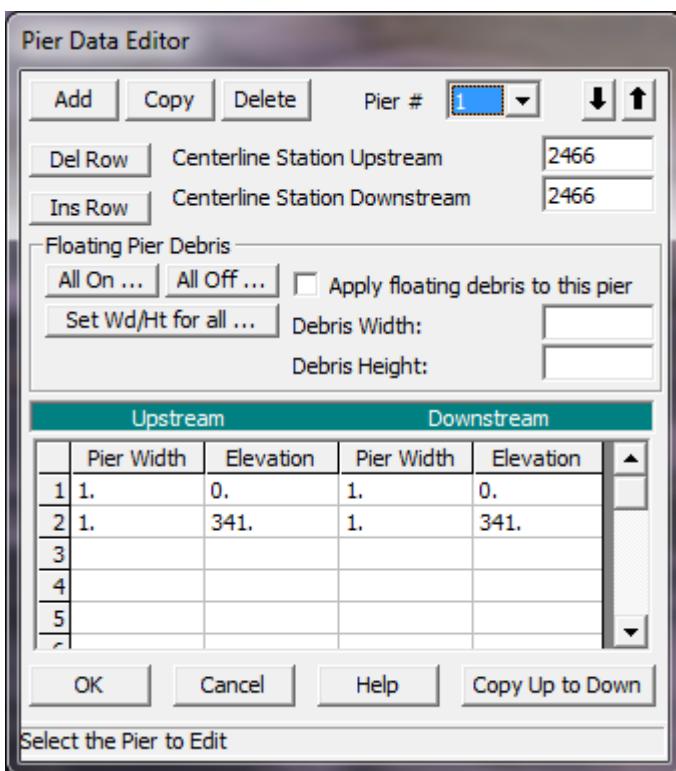


Figure 5 18 Pier Data Editor

When the pier data editor appears it will have already defined the first pier as pier # 1. The user is required to enter a centerline station for both the upstream and downstream side of the pier. The pier geometry is entered as pier widths and elevations. The elevations must start at the lowest value and go to the highest value. Generally the elevations should start below the ground level. Any pier area below the ground will be clipped off automatically. Pier widths that change at a single elevation are handled by entering two different widths at the same elevation. The order of the widths in the table is very important. Keep in mind that the pier is defined from the ground up to the deck. If the pier geometry on the downstream side is the same as the upstream side, simply press the **Copy Up to Down** button after the upstream side data are entered.

The user also has the option of defining floating pier debris. If the **Floating Debris** option is selected, the user will need to enter a width and a height for the debris. The user can set a different height and width of debris for each pier, or there is a button that will allow the user to enter a single height and width that will be used for all of the piers (**Set Wd/Ht for all...**). Additionally there are buttons to turn pier debris on or off for all of the piers of the bridge (**All On...** and **All Off...**).

Additional piers can be added by pressing either the **Add** or the **Copy** button. If the piers are the same shape, it is easier to use the copy button and simply change the centerline stations of the new pier. To delete a pier, simply press the **Delete** button and the currently displayed pier will be deleted. Once all of the pier data are entered, press the **OK** button. When the OK button is pressed, the data will be accepted and the pier editor will be closed. The graphic of the bridge will then be updated to include the piers. An example bridge with piers is shown in Figure 5-19. This graphic is only the upstream side of the bridge with a zoomed in view.

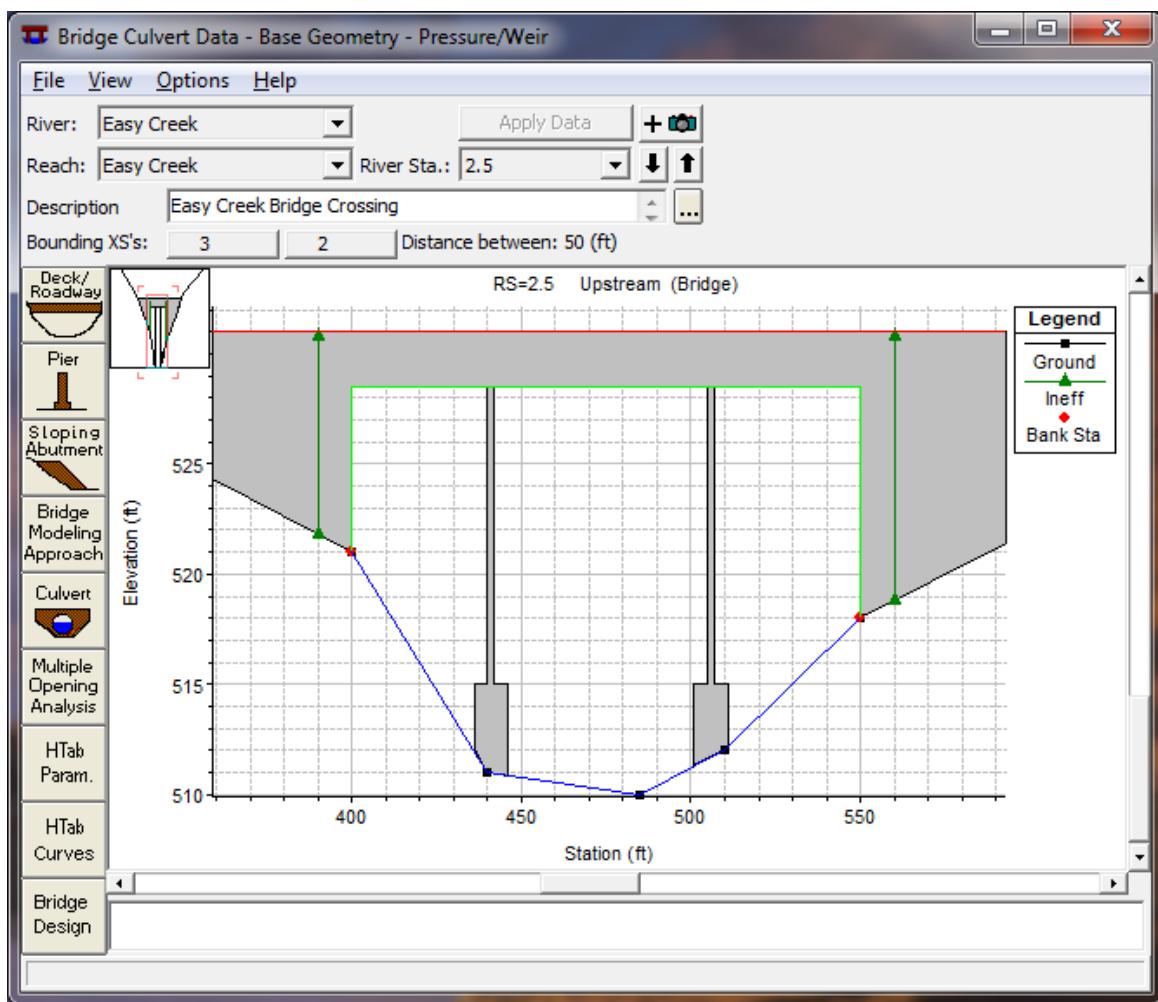


Figure 5 19 Bridge with Piers, zoomed in view

Bridge Modeling Approach. The Bridge Modeling Approach editor is used to define how the bridge will be modeled and to enter any coefficients that are necessary. To bring up the Bridge Modeling Approach editor press the **Bridge Modeling Approach** button on the Bridge/Culvert Data editor. Once this button is pressed, the editor will appear as shown in Figure 5-20 (Except yours will only have the default methods selected).

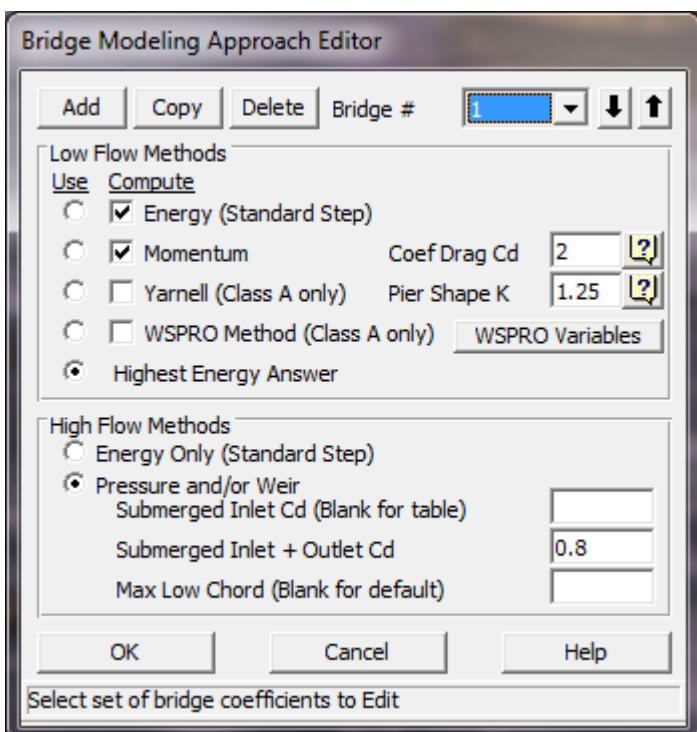


Figure 5 20 Bridge Modeling Approach Editor

When the Bridge Modeling Approach editor comes up it will be ready to enter data for the first bridge opening (coefficient set # 1). If there is more than one bridge opening at the current location, the user can either use a single set of modeling approaches and coefficients, or establish a different set for each bridge opening.

Establishing a bridge modeling approach consists of defining which methods the program will use for low flow computations and high flow (flow at or above the maximum low chord) computations. The user can instruct the program to use any or all of the low flow methods during the computations by clicking the buttons under the **Compute** column. If either the Momentum or Yarnell method are selected, the user must enter a value for the pier loss coefficient that corresponds to that method. If the WSPRO method is selected, the user must press the "WSPRO Variables" button and enter additional information that is required for the method. Once the **WSPRO Variables** button is pressed, a data editor as shown in Figure 5-21 will appear.

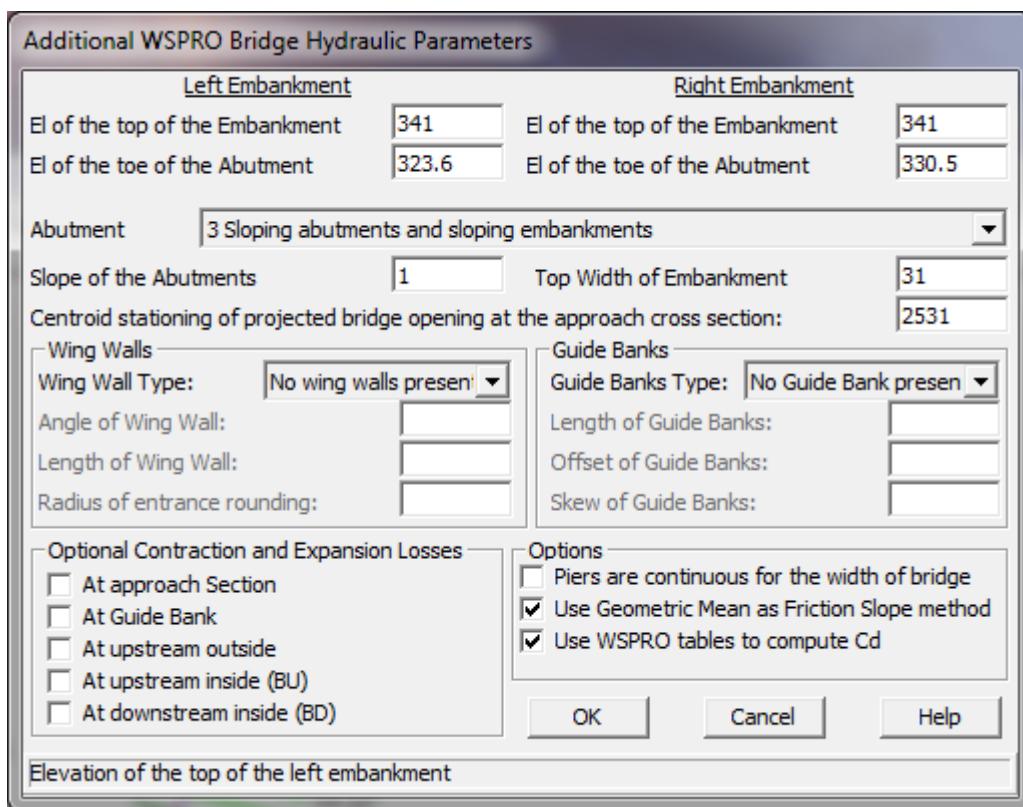


Figure 5 21 WSPRO Data Editor

As shown in Figure 5-21, there are several variables that must be entered as well as some options that are available to the user. All of the required variables shown on the WSPRO data editor are used in the computation of the discharge coefficient, C, which is used in the WSPRO expansion loss equation. A detailed discussion of how the discharge coefficient is computed can be found in appendix D of the HEC-RAS Hydraulic Reference manual. The following is a description of each of the variables on the WSPRO Data Editor:

Elevation of the top of the embankment - These fields are used for entering the elevation of the top of the embankment (top of road) at the edges of the bridge opening. An elevation must be entered for both the left and right side of the bridge opening.

Elevation of the toe of the abutment - These fields are used for entering the elevation of the abutment toe (elevation at the station in which the abutment toe intersects with the natural ground inside the bridge opening) on both the left and right side of the bridge opening.

Abutment Type - This field is used for selecting the type of abutments. There are four abutment types available from this selection box.

Slope of the abutments - This field is used for entering the slope of the abutments. This slope is taken as the horizontal distance divided by the vertical distance. If the abutments are vertical walls, then this field should be left blank or set to zero. If the left and right abutments do not have the same slope, take an average of the two and enter that into this field.

Top Width of Embankment - This field is used for entering the width of the top of the road embankment, in the area of the bridge opening. If the topwidth of the embankment varies from one end of the bridge opening to the other, use an average of the two widths.

Centroid stationing of the projected bridge opening at the approach cross section - For the WSPRO bridge method, it is necessary to calculate the water surface topwidth inside of the bridge opening, and then project that width onto the approach cross section. The program calculates the conveyance within this projected width at the approach cross section. This conveyance is used in calculating a channel contraction ratio, which is an integral part in the calculation of the discharge coefficient. If this field is left blank, the program will automatically center the computed topwidth, such that the center of the topwidth will be at the center of conveyance at the approach cross-section. The user can override this by entering their own centroid stationing value for the approach cross section.

Wing Walls - This field is used for selecting the type of wing walls. There are three choices available in the selection box: No wing walls present; Angular wing walls; and Rounded wing walls. If the user selects "Angular wing walls", then the fields labeled "Angle of Wing Wall" and "Length of Wing Wall" become active and must be filled out. If the user selects "Rounded wing walls", then the fields "Length of wing walls" and "Radius of entrance rounding" become active and must be filled out. If the user selects "No wing walls present" then no other information on wing walls is necessary. For more information on wing walls see appendix D of the HEC-RAS Hydraulic Reference manual.

Guide Banks Type - This field is used for selecting the type of guide banks if any exist. There are three choices available from the selection box: No guide bank present; Straight; and Elliptical. If the user selects "Straight" then the fields labeled "Length of guide banks", "Offset of Guide Banks", and "Skew of Guide Banks" become active and must be filled out. If the user selects "Elliptical" then only the fields "Length of Guide Banks" and "Offset of Guide Banks" become active. If the user selects "No Guide Bank present" then no other information about guide banks is necessary. For more information on Guide Banks see appendix D of the HEC-RAS Hydraulic Reference manual.

Optional Contraction and Expansion Losses - This box allows the user to turn on contraction and expansion losses at locations that are traditionally not in the WSPRO methodology. The basic WSPRO bridge method only computes expansion losses in the expansion reach (between the exit cross section and the section just downstream of the bridge). This option allows the user to turn on contraction and expansion losses individually at the following locations: downstream inside of the bridge; upstream inside of the bridge; upstream outside of the bridge; at the end of a guide bank (if guide banks exist); and at the approach cross section. The default for the WSPRO method is that contraction and expansion losses will not be calculated at these locations. Users should not turn these options on unless they feel that the standard WSPRO bridge approach is not producing enough energy loss through the bridge.

Three other options that the user has control over are: specifying that the piers are continuous the whole way through the bridge or not; using the Geometric Mean friction slope averaging technique through the bridge computations (from exit to approach section); and using the WSPRO tables to compute the C_d coefficient, rather than the theoretical equation. The default for the WSPRO methodology is to assume that the piers are continuous through the bridge, to use the Geometric Mean friction slope method, and compute C_d with the theoretical equation.

After all of the variables have been entered, the user must press the **OK** button for the WSPRO variables to be accepted. For more information about the computation of the discharge coefficient, and these data variable, see appendix D of the HEC-RAS Hydraulic Reference manual.

Once the user has selected which low flow bridge methods will be computed, they must also specify which of those methods will be used as the final answer to continue the computations on upstream

with. Only one of the methods can be selected as the answer to "**Use**" in order to continue the computations upstream. An alternative to selecting a single method to use is to instruct the program to use the answer with the highest computed upstream energy elevation. This is accomplished by pressing the button under the "**Use**" column that corresponds to the **Highest Energy Answer** text field.

For a **High Flow Method**, the modeler can choose between Energy based calculations or pressure and weir flow calculations. If pressure and weir flow is the selected high flow method, the user must enter coefficients for the pressure flow equations. The first coefficient applies to the equation that is used when only the upstream side (inlet) of the bridge is submerged. If this coefficient is left blank, the program selects a coefficient based on the amount of submergence. If the user enters a coefficient, then that value is used for all degrees of submergence. The second coefficient applies to the equation that is used when both the upstream and downstream end of the bridge is sub-merged. Generally this coefficient is around 0.8. For more information on pressure flow coefficients see *Hydraulics of Bridge Waterways* (FHWA, 1978).

Max Low Chord - This field is used to set the maximum elevation of the deck low chord, and therefore the elevation at which pressure flow begins to be calculated. If this field is left blank, then the elevation that triggers pressure flow calculations is based on the highest low chord elevation on the upstream side of the bridge deck. If the user enters a value in this field, then the value set will be used to trigger when pressure flow calculations begin. Pressure flow is triggered when the energy elevation exceeds the maximum low chord. When pressure flow is calculated, the answer is compared to the low flow answer and the higher of the two is selected. Alternatively, the user can tell the program to use the water surface instead of the energy elevation to trigger pressure flow calculations. This option can be found under the **Bridge and Culvert Options** section of this manual.

Once all of the bridge modeling approach information is entered, the user should press the **OK** button. When the OK button is pressed the information will be accepted and the editor will close.

Remember! The data are not saved to disk at this point, it is only accepted as being valid. To save the geometric data, use the **File** menu from the Geometric Data Editor window.

Bridge Design Editor

The bridge design editor allows the user to enter or modify bridge data quickly and conveniently. With this editor the user can enter the deck/roadway data, sloping abutments, and pier information. To put together a bridge with this editor, the user would do the following:

1. From the Geometric Data window, open the Bridge/Culvert data editor. Select the River and Reach in which you would like to place the bridge.
2. Go to the **Options** menu and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new bridge.
3. Open the Bridge Design editor by pressing the **Bridge Design** button on the lower left side of the Bridge/Culvert Data editor.
4. Enter the required data for the bridge deck/roadway, sloping abutments (optional), and piers (optional).

When the **Bridge Design** button is pressed, a window will appear as shown in Figure 5-22. The user only has to enter a minimal amount of information to build or edit the bridge. To create the bridge deck/roadway, the user must enter a high cord elevation (top of road) and a low cord elevation (maximum elevation inside of the bridge opening).

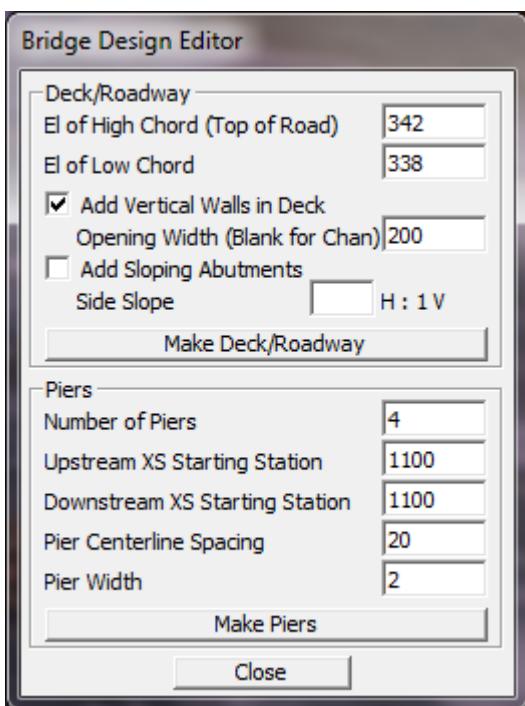


Figure 5 22 Bridge Design Editor

The user has the option to limit the width of the bridge opening by selecting the **Add Vertical Walls in Deck** option. When this option is selected, the bridge opening will be limited to either the main channel bank stations (this is the default) or a user specified width (this is optional). Everything left and right of the bridge opening will be completely filled in all the way to the ground elevations. If the user enters a bridge opening width, the opening will be centered between the main channel bank stations.

The user also has the option to enter sloping abutments. Sloping abutments should only be entered after selecting to limit the width of the bridge opening with the vertical walls option. To enter sloping abutments, the user only has to enter a slope in units of horizontal to vertical. The program will automatically build a left and right sloping abutment that starts in the upper left and right corners of the bridge opening.

Once all of the bridge deck/roadway information is entered, the user can have the program build the deck/roadway by pressing the **Make Deck/Roadway** button.

The last option available in the Bridge Design editor is to enter pier information. The user enters the number of piers, the upstream and downstream stationing of the left most pier, the spacing between the centerline of the piers, and the width of the piers. The user then presses the **Make Piers** button to have the interface build the piers.

After all of the bridge data are entered, the user presses the Close button to get out of the editor. The bridge data can be changed at any time by either going back into the Bridge Design editor and entering new values, or by going to the more detailed editors for the bridge deck/roadway, sloping abutments, and piers.

Culvert Hydraulic Computations

The culvert hydraulic computations in HEC-RAS are similar to the bridge hydraulic computations, except the Federal Highway Administration's (FHWA) standard equations for culvert hydraulics under inlet control are used to compute the losses through the structure (when a culvert is under inlet control conditions). Outlet control hydraulics are handled by balancing the energy equation from downstream to upstream. The HEC-RAS culvert routines are also capable of handling all 6 of the flow classifications outlined in the USGS publication "Measurement of Peak Discharge at Culverts by Indirect Methods" (USGS, 1976).

Because of the similarities between culverts and other types of bridges, the cross section layout, the use of ineffective areas, the selection of contraction and expansion coefficients, and many other aspects of bridge analysis apply to culverts as well.

The culvert routines in HEC-RAS have the ability to model nine different types of culvert shapes. These shapes include box (rectangular), circular, elliptical, arch, pipe arch, semi circular, low profile arch, high profile arch, and Con Span culverts.

The analysis of flow in culverts is complicated. It is common to use the concepts of "Inlet" control and "Outlet" control to simplify the analysis. **Inlet control** flow occurs when the flow carrying capacity of the culvert entrance is less than the flow capacity of the culvert barrel. Which means the culvert entrance is controlling the resulting headwater elevation for a given flow passing through the culvert. **Outlet control** flow occurs when the culvert carrying capacity is limited by downstream conditions or by the flow capacity of the culvert barrel. The HEC-RAS culvert routines compute the headwater required to produce a given flow rate through the culvert for inlet control conditions and for outlet control conditions. In general, the higher headwater "controls," and an upstream water surface is computed to correspond to that energy elevation.

Inlet Control Computations. For inlet control, the required headwater is computed by assuming that the culvert inlet acts as an orifice or a weir. Therefore, the inlet control capacity depends primarily on the geometry of the culvert entrance. Extensive laboratory tests by the National Bureau of Standards, and the Bureau of Public Roads (now, FHWA), and other entities resulted in a series of equations which describe the inlet control headwater under various conditions. These equations are used by HEC-RAS in computing the headwater associated with inlet control.

Outlet Control Computations. For outlet control flow, the required headwater must be computed considering several conditions within the culvert and the downstream tailwater. For culverts flowing full, the total energy loss through the culvert is computed as the sum of friction losses, entrance losses, and exit losses. Friction losses are based on Manning's equation. Entrance losses are computed as a coefficient times the velocity head in the culvert at the upstream end. Exit losses are computed as a coefficient times the change in velocity head from just inside the culvert (at the downstream end) to outside the culvert.

When the culvert is not flowing full, the direct step backwater procedure is used to calculate the profile through the culvert up to the culvert inlet. An entrance loss is then computed and added to the energy inside the culvert (at the upstream end) to obtain the upstream energy (headwater). For more information on the hydraulics of culverts, the reader is referred to Chapter 6 of the HEC-RAS Hydraulics Reference manual.

Entering and Editing Culvert Data

Culvert data are entered in the same manner as bridge data. To enter culvert data the user presses the **Bridge/Culvert** button on the Geometric Data window (Figure 5-1). Once this button is pressed, the Bridge/Culvert Data Editor will appear (Figure 5-12). To add a culvert group to the model the user must then do the following:

1. Select the river and reach that you would like to place the culvert in. This selection is accomplished by pressing the down arrow on the river and reach boxes and then selecting the river and reach of choice.
2. Go to the **Options** menu of the Bridge/Culvert editor and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new culvert group. After entering the river station, press the OK button and the cross sections that bound the new culvert group will appear in the editor.
3. Enter all of the required data for the culvert group. This includes the road embankment information and the culvert specific data. The roadway information is entered in the same manner as a bridge (using the deck/roadway editor). To enter culvert specific data, press the Culvert button on the Bridge/Culvert Data editor.
4. Once all of the culvert data are entered, press the OK button in order for the interface to accept the information.

River, Reach and River Station. The River and Reach boxes allow the user to select a river and reach from the available reaches that were put together in the schematic diagram. The reach label defines which reach the culvert will be located in. The River Station tag defines where the culvert will be located within the specified reach. The River Station tag does not have to be the actual river station of the culvert, but it must be a numeric value. The River Station tag for the culvert should be numerically between the two cross sections that bound the culvert. Once the user selects **Add a Bridge and/or Culvert** from the options menu, an input box will appear prompting you to enter a River Station tag for the new culvert. After the River Station tag is entered, the two cross sections that bound the culvert will be displayed on the editor.

Description. The description box is used to describe the culvert location in more detail than just the river, reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for culvert plots and tables.

Culvert Road Embankment. The culvert road embankment is virtually the same as the bridge deck/roadway information. The road embankment is used to describe the area blocking the stream and the roadway profile. The only difference in the information for culverts is that the low chord elevations should be left blank or set to elevations below the ground data. This will cause the road embankment to completely fill the channel up to the roadway elevations (high chord data). Therefore, the only opening below the roadway will be whatever culvert openings are entered.

To enter the culvert roadway information, press the **Deck/Roadway** button on the Bridge/Culvert Data Editor window. For an explanation of the deck information, please review the section entitled **Bridge Deck/Roadway** found earlier in this chapter.

Culvert Data. To enter culvert specific information, press the **Culvert** button on the Bridge/Culvert Data Editor window. When this button is pressed, the Culvert Data Editor will appear as shown in Figure 5-23 (Except yours will be blank). The information entered in the Culvert Data Editor consists of the following:

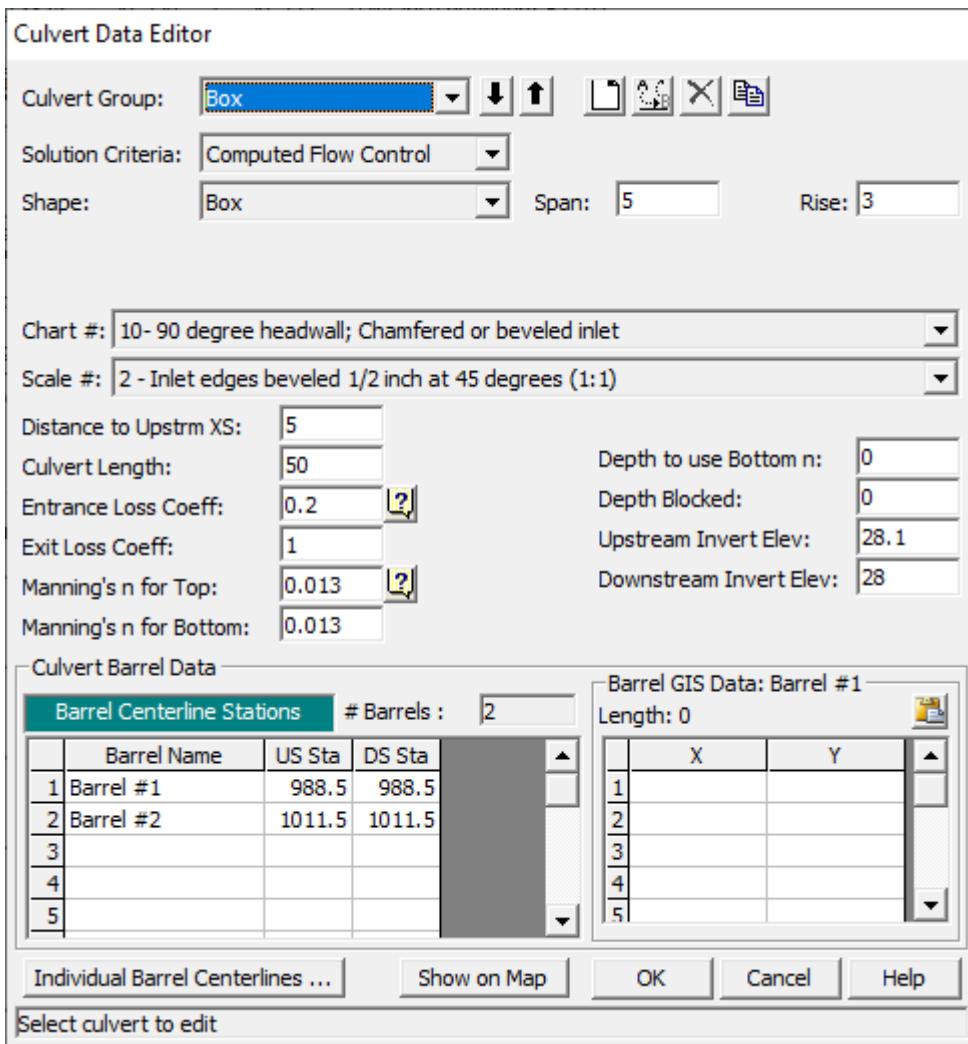


Figure 5 23 Culvert Data Editor

Culvert Group - The culvert group is automatically assigned to "Culvert #1" the first time you open the editor. The user can enter up to ten culvert types if they are working on a multiple culvert/opening problem. If all of the culvert barrels are exactly the same, then only one culvert group should be entered. The number of barrels is an input parameter in the culvert data. If the user has culverts that are different in shape, size, elevation, or loss coefficients, then additional culvert groups must be added for each culvert type. To add an additional culvert group you can either use the **Add** or **Copy** buttons. The Add button increments the culvert group and clears the culvert editor. The Copy button increments the culvert group and makes a copy of the original culvert data. Once a copy is made of a culvert, the user can change any of the existing culvert information. Culverts can be deleted by pressing the **Delete** button.

Solution Criteria - This option allows the user to select between taking the higher of the inlet control and outlet control answers (Computed Flow Control), or specifically selecting the Inlet control or Outlet control answer. The default is to let the program compute both and take the higher of the two. In general this should be left this way. The only time a user should specifically select Inlet control or Outlet control, is when they feel the program is in error by selecting the higher of the two answers.

Rename - This button allows the user to put in their own identifier for each of the culvert types. By default the culvert types will be labeled "Culvert #1," "Culvert #2," and so on. The user can enter up to twelve characters for each culvert type.

Shape - The shape selection box allows the user to select from one of the nine available shapes. This selection is accomplished by pressing the down arrow on the side of the box, then selecting one of the nine available shapes.

Span - The span field is used to define the maximum width inside of the culvert. The span is left blank for circular culverts.

Rise - The rise field describes the maximum height inside of the culvert.

Chart # - This field is used to select the Federal Highway Administration Chart number that corresponds to the type and shape of culvert being modeled. Once the user has selected a culvert shape, the corresponding FHWA chart numbers will show up in the chart # selection box. More information on FHWA chart numbers can be found in the Hydraulics Reference manual.

Scale# - This field is used to select the Federal Highway Administration Scale number that corresponds to the type of culvert entrance. Once the user has selected a culvert shape and chart #, the corresponding FHWA scale numbers will show up in the scale selection box. More information on FHWA scale numbers can be found in the Hydraulics Reference manual.

Distance to Upstream XS - This field is used to locate the culvert in space, relative to the two cross sections that bound the culvert crossing. The user should enter the distance between the upstream cross section and the upstream end of the culvert barrel.

Culvert Length - The culvert length field describes the length of the culvert along the centerline of the barrel.

Entrance Loss Coefficient - The coefficient entered in this field will be multiplied by the velocity head inside of the culvert at the upstream end. This value represents the amount of energy loss that occurs as flow transitions from the upstream cross section to inside the culvert barrel. This coefficient is used in the outlet control computations, and will not affect inlet control computations, as they are performed with the Federal Highway Inlet Control equations directly.

Exit Loss Coefficient - The coefficient entered in this field will be multiplied by the change in velocity head from inside the culvert to outside the culvert at the downstream end. This value represents the energy loss that occurs as water exits the culvert. This coefficient is used in the outlet control computations.

Manning's n for Top - The n-value fields are used for entering the Manning's n values of the culvert barrel. This version of HEC-RAS allows the user to enter a separate n value for the top (which includes top and sides) of the culvert, as well as for the bottom. If the culvert has the same roughness for the top and bottom, the user can enter the value for the top. The Manning's n value for the bottom will automatically be copied from the top field.

Manning's n for Bottom - This field is used to enter a Manning's n value for the bottom of the culvert. This n value will be used up to a user specified depth inside of the culvert. When the water surface gets higher than that depth, a composite Manning's n value is computed based on the bottom and top n values and their corresponding wetted perimeters.

Depth to use Bottom n – This field is used to specify the depth that the "Bottom n value" is applied inside of the culvert. The surface of the culvert below this depth is given the n value for the bottom of the culvert, while the surface of the culvert above this depth is given the n value for the top of the culvert.

Depth Blocked – This field is used to block off a portion of the bottom of the culvert. When a value is entered into this field, the culvert is completely blocked up to the depth specified. This blocked out area persists the whole way through the culvert.

Upstream Invert Elevation - This field is used to describe the elevation of the culvert invert at the upstream end.

Downstream Invert Elevation - This field is used to describe the elevation of the culvert invert at the downstream end.

Barrels - This field is used to **display** the number of identical barrels. The number of identical barrels is limited to 25. To enter more than one identical barrel, the user must provide different centerline stationing information for each barrel. As the centerline stationing information is added, the number of identical barrels will automatically change to reflect the number of centerline stations. The user does not enter anything into this field, it is just used to display the number of identical barrels.

Barrel Centerline Stations - This table is used to enter the stationing of each culvert barrel. Centerline stations must be provided for both the upstream and downstream side of each culvert barrel.

Barrel GIS Data – This table is used to enter X, Y coordinates for a line representing each barrel. X, Y coordinates are only used when culverts are connected to 2D Flow Areas. When a Culvert is put into a Bridge/Culvert crossing of a 1D River Reach, then the X, Y coordinates are not needed or used. This table is only relevant to Lateral Structures and SA/2D Area Connections in which culverts will be connected to 2D Flow Area cells. It is not needed for pure 1D bridge/Culvert crossings.

Once all of the culvert information is entered, the user should press the **OK** button at the bottom of the window. Pressing the **OK** button tells the interface to accept the data and close the window.

Once the culvert editor is closed, the graphic of the culvert will appear on the Bridge/Culvert Data editor window. An example culvert with two culvert types and two identical barrels for each culvert type is shown in Figure 5-24. **Note! The data are not saved to the hard disk at this point.**

Geometric data can only be saved from the **File** menu on the Geometric Data window.

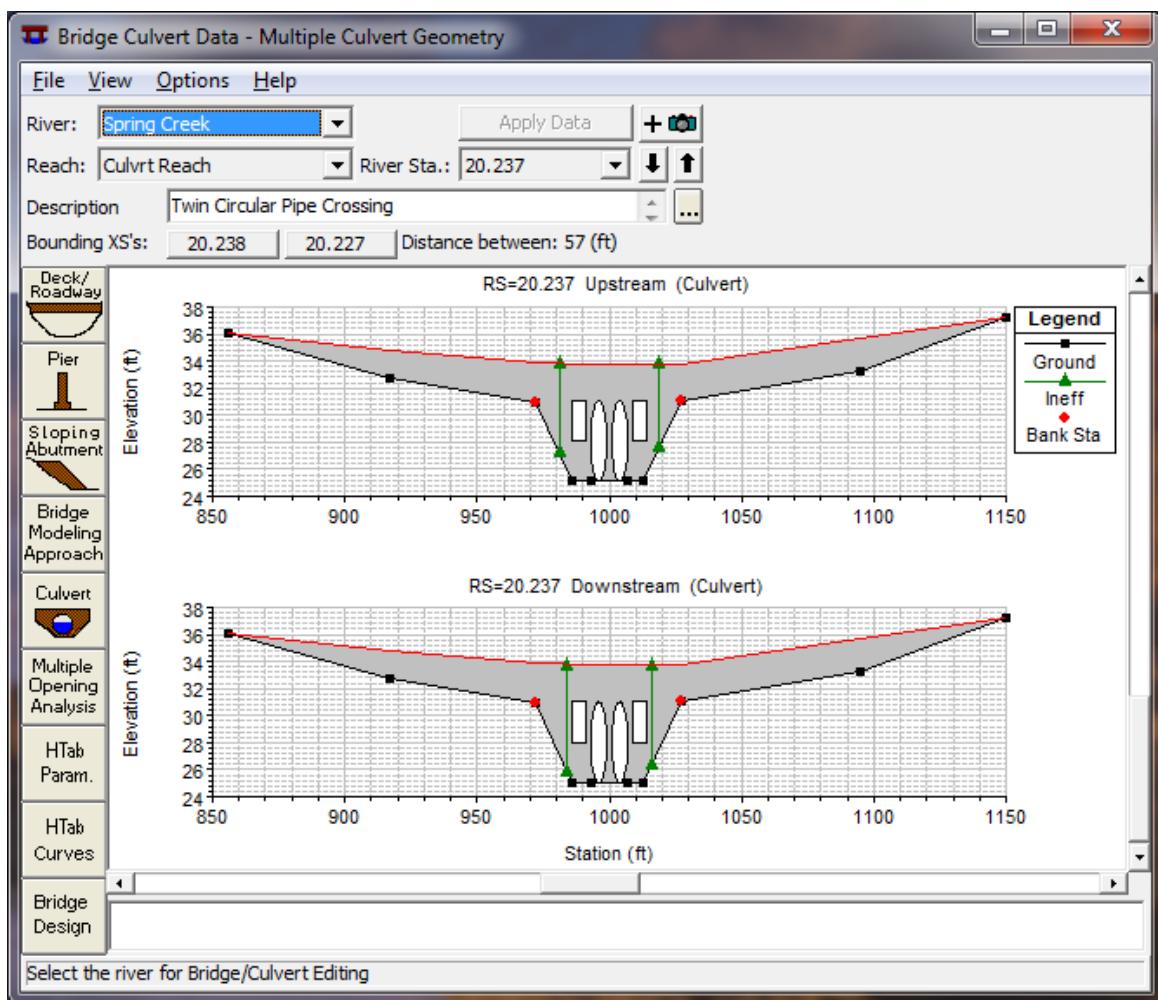


Figure 5 24 Bridge/Culvert Data Editor with example culvert

Bridge and Culvert Options

Some additional options that are available, but not required, are found under the **Options** menu from the Bridge/Culvert Data Editor. These include the following:

Add a Bridge and/or Culvert. This option initiates the process of adding a bridge or culvert to the data set. The user is prompted to enter a river station tag for the new bridge or culvert. The river station tag locates the bridge or culvert within the selected reach. Once the river station is entered, the Bridge/Culvert Data editor is cleared and the user can begin entering the data for that new bridge or culvert.

Copy Bridge and/or Culvert. This option allows the user to make a copy of the bridge and/or culvert crossing and place it in another reach and/or river station within the current project.

Rename River Station. This option allows the user to change the river station of the currently opened Bridge and/or Culvert crossing.

Delete Bridge and/or Culvert. This option will delete the currently displayed bridge or culvert. The user is prompted with a message stating specifically which bridge or culvert is going to be deleted, and requesting them to press the **OK** button or the **Cancel** button.

Internal Bridge Cross-Sections. This option allows the user to edit the two cross sections inside of a bridge. These two cross sections are a copy of the cross sections just upstream and downstream of the bridge. If the ground elevations inside of the bridge are different than just outside of the bridge, then the internal bridge cross sections should be modified to reflect the changing elevations. This option allows the user to change the station and elevation data, roughness coefficients, and main channel bank stations for each of the two internal bridge cross sections.

Momentum Equation. This option allows the user to change the components of the momentum equation. The momentum equation is one of the optional low flow methods in the bridge routines. The user has the option of turning the friction and weight force components on or off. The default is to include the friction force but not the weight component. The computation of the weight force is dependent upon computing a mean bed slope through the bridge. Estimating a mean bed slope can be very difficult with irregular cross section data. A bad estimate of the bed slope can lead to large errors in the momentum solution. The user can turn this force on if they feel that the bed slope through the bridge is well behaved for their application (smooth, with no deep scour hole).

Momentum Class B Defaults. If the program computes that the flow must pass through critical depth inside the bridge (Class B flow), critical depth will automatically be located inside the bridge at the most constricted cross section. If both cross sections are identical, the program will locate critical depth at the upstream inside cross section. This option allows the user to control where the program sets critical depth for class B flow. If the user feels that it would be better to set critical depth inside the bridge at the downstream end, then this can be selected.

Pressure Flow Criteria. This option allows the user to select either the energy grade line or the water surface, to be used as the criterion for when the program begins checking for the possibility of pressure flow. By default the program uses the energy grade line. This does not change how pressure flow is calculated, only when the program will begin checking for pressure flow.

Ice Option. This option allows the user to select how ice will be handled inside of the bridge during ice computations. This option is only pertinent if the user is performing a profile computation with the effects of ice included. When this option is selected, a window will appear asking the user to select one of three available options. These options include: no ice inside of the bridge; a constant amount of ice through the bridge; dynamic ice effects are to be computed through the bridge.

Skew Bridge/Culvert. This option allows the user to make adjustments to bridge/culvert data that is skewed (i.e. not perpendicular to the flow lines going through the bridge/culvert. When this option is selected, a window will appear allowing the user to enter a skew angle for the deck/roadway, as well as the piers. The stationing of the deck/roadway is reduced, by multiplying it by the cosine of the user entered skew angle. Additionally, the user has the option to adjust the upstream and downstream cross sections bounding the bridge by the same skew angle. A separate skew angle is entered for bridge piers. The piers are assumed to go the whole way through the bridge as a single continuous pier. For more details on modeling bridges that are skewed to the flow, see the section called "Bridges on a Skew" in chapter 5 of the Hydraulic Reference Manual.

Bridge and Culvert View Features

Several options are available for viewing the bridge/culvert geometric data. These options include: Zoom In; Zoom Out; Display Upstream XS; Display Downstream XS; Display Both; Highlight Weir, Opening Lid and Ground; Highlight Piers; and Grid. These options are available from the **View** menu on the bridge/culvert data editor.

Zoom In. This option allows the user to zoom in on a piece of the bridge or culvert. This is accomplished by selecting **Zoom In** from the **View** menu, then specifying the area to zoom in on with the mouse. Defining the zoom area is accomplished by placing the mouse pointer in the upper left corner of the desired area. Then press down on the left mouse button and drag the mouse to define a box containing the desired zoom area. Finally, release the left mouse button and the viewing area will display the zoomed in area of the bridge or culvert.

Zoom Out. This option displays the bridge or culvert back into its original size before you zoomed in. Zooming out is accomplished by selecting **Zoom Out** from the **View** menu bar on the bridge/culvert data editor.

Full Plot. When this option is selected, the graphic is automatically redrawn back to its full extent, showing the entire bridge/culvert.

Pan. When this option is selected, the user can move the zoomed in portion of the graphic. This is accomplished by first selecting the Pan option, then pressing and holding down the left mouse button while over the graphic. Next, move the graphic in the desired direction, and then release the left mouse button. The graphic will be redrawn with a new portion of the graphic shown in the zoomed in area.

Display Upstream XS. When this option is selected, only the upstream side of the bridge or culvert will be displayed.

Display Downstream XS. When this option is selected, only the downstream side of the bridge or culvert will be displayed.

Display Both. When this option is selected, both the downstream and upstream sides of the bridge will be displayed in the viewing area.

Highlight Weir, Opening Lid and Ground. When this option is selected, various portions of the bridge/culvert graphic will be highlighted. The program will highlight in red the combination of the deck/roadway high cord and any ground to the left and right of this data. The red color shows what the program will use for weir flow if the Pressure and Weir option is selected for high flows. The program will also highlight any bridge openings. Within the bridge opening, the ground information will be highlighted in blue and the lid of the opening (deck/roadway low cord data) will be highlighted in green. If any of these three colors show up in an area where they should not be, then there must be a geometric mistake in the data. This option is very useful for detecting any data entry errors that may otherwise go unnoticed.

Highlight Piers. When this option is turned on the interface will highlight what it thinks is the extent of the pier information. This option allows the user to see exactly what the program thinks piers are, and to see how the pier information has been clipped. Piers are clipped below the ground and above the low chord of the bridge.

Grid. This option allows the user to have a grid overlaid on top of the bridge or culvert graphic.

Multiple Bridge/Culvert Openings

HEC-RAS has the ability to model multiple bridge and/or culvert openings at any individual river crossing. Types of openings can consist of bridges, culvert groups (a group of culverts is considered to be a single opening), and conveyance areas (an area where water will flow as open channel flow, other than a bridge or culvert opening). Up to seven openings can be modeled at a given location, and any combination of bridges and culvert groups can be used. Conveyance type openings can range from zero to a maximum of two, and the conveyance areas must be located on the far left and far right of the river crossing.

An example multiple opening is shown in Figure 5-25. As shown in this example, there are three types of openings: a conveyance area (left side, labeled as opening #1), a bridge (labeled as opening #2), and a culvert group (labeled as opening #3). During low flow conditions, flow will be limited to the bridge opening. As flow increases, the culverts will begin to take some of the flow away from the bridge opening. The conveyance area was defined as ineffective flow (no conveyance) until the water surface goes above the top of the bridge. This was accomplished by setting blocked ineffective flow areas. In this example, three blocked ineffective flow areas were established: one to the left of the bridge (which encompasses the whole conveyance area), one between the bridge and the culvert group, and one to the right of the culvert group.

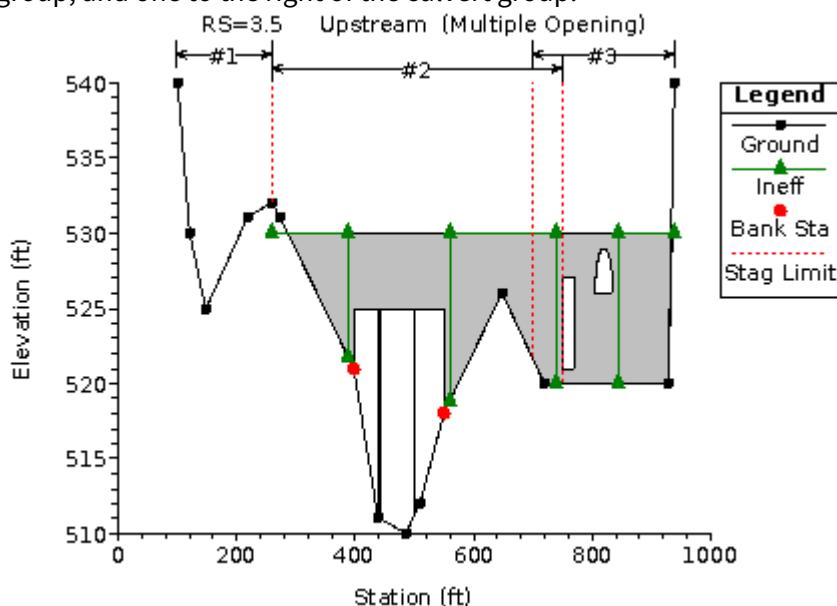


Figure 5 25 Example Multiple Opening River Crossing

Entering Multiple Opening Data

Multiple opening data are entered in the same manner as any other bridge or culvert crossing. In general, the user should perform the following steps to enter multiple opening data:

1. Press the Bridge/Culvert button on the Geometric Data window.
2. Select the river and reach in which you would like to place the multiple opening river crossing. This is accomplished from the River and Reach boxes near the top of the window.
3. Select **Add a Bridge and/or Culvert** from the **Options** menu of the bridge and culvert editor. Enter the river station at which you want to place the multiple opening crossing. Once you have done this, the two cross sections that bound this river station will appear in the window. These two cross sections, along with the

bridge and culvert information, will be used to formulate the two cross sections inside the multiple opening river crossing.

4. Enter the deck and road embankment data by using the Deck/Roadway editor.
5. Enter any piers or sloping abutments that are required.
6. Select the **Bridge Modeling Approach** button and enter a set of coefficients and modeling approaches for each bridge opening.
7. Enter Culvert data for any culvert openings.
8. Select the **Multiple Opening Analysis** button on the bridge and culvert editor. Enter the types of openings and their station limits. Start at the left most station of the crossing and work your way to the right end. This is explained in greater detail under the section entitled "Defining the Openings".

Deck/Road Embankment Data. There can only be one deck and road embankment entered for any bridge and/or culvert crossing. The deck editor is used to describe the area that will be blocked out due to the bridge deck and road embankment. As shown by the gray shaded area in Figure 5-25, the deck and roadway data are used to block out area around the bridge as well as around the culverts. In the area of the bridge, high and low chord information is entered in order to define the top of road as well as the bridge opening. In the area of the culverts, the high chord information is entered to define the rest of the top of the road embankment. However, the low chord information can be left blank, or set to elevations below the ground, because the culvert data define the culvert openings.

Piers and abutments. All piers are entered from the pier editor, which was described previously under bridge data. The number of bridge openings has no impact on how pier data are entered. Piers are treated as separate information. Once the user establishes that there is more than one bridge opening, the program is smart enough to figure out which piers go with which opening. If any sloping abutment data are required for a bridge opening, it can be entered as described previously under the bridge data section.

Bridge Modeling Approach. A bridge modeling approach and coefficient set must be established for at least one bridge opening. If there is more than one bridge opening, and the user has only established a single coefficient set and bridge modeling approach, those data will be used for all of the bridge openings. The user can establish a different set of coefficients and modeling approaches for each bridge opening.

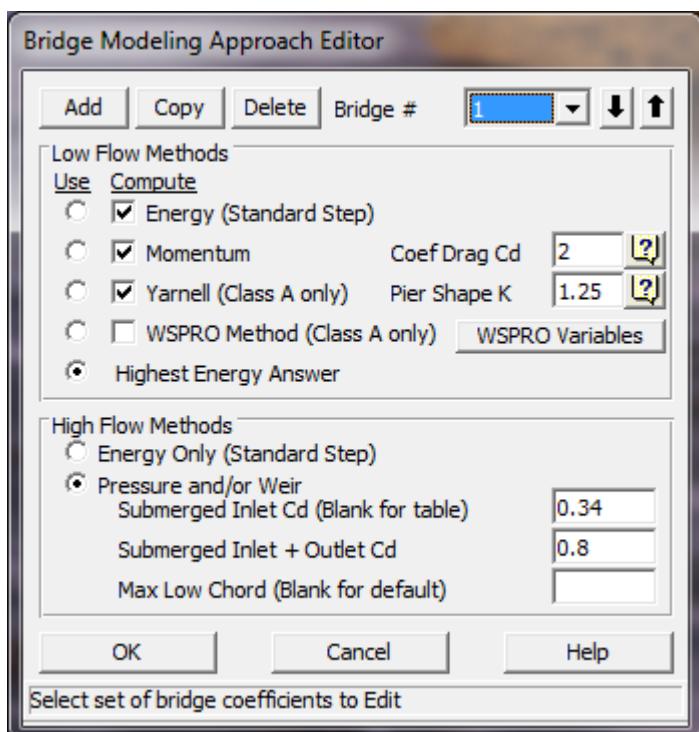


Figure 5 26 Bridge Modeling Approach Editor

As shown in Figure 5-26, the user must enter information under the Bridge Modeling Approach editor for at least one bridge Opening. Bridge openings are referred to as Bridge # 1, Bridge # 2, etc., up to the number of bridge openings. Bridge # 1 represents the left most bridge opening while looking in the downstream direction. Bridge # 2 represents the next bridge opening to the right of Bridge # 1, and so on. The user can enter additional coefficient sets and modeling approaches by selecting either the **Add** or **Copy** button. If either of these buttons is selected, the Bridge # will automatically be incremented by one. The user can then enter or change any of the information on the editor for the second bridge opening. Any bridge opening that does not have a corresponding coefficient set and modeling approach, will automatically default to what is set for Bridge # 1.

Culvert Data. Culvert information is added in the same manner as described in the previous section called "Entering and Editing Culvert Data." Culverts will automatically be grouped based on their stationing.

Defining the Openings

Once all of the bridge and/or culvert data are entered for a multiple opening river crossing, the last step is to define the number and type of openings that are being modeled. This is accomplished by pressing the Multiple Opening Analysis button on the Bridge/Culvert Data editor. Once this button is pressed, an editor will appear as shown in Figure 5-27 (except yours will be blank the first time you bring it up).

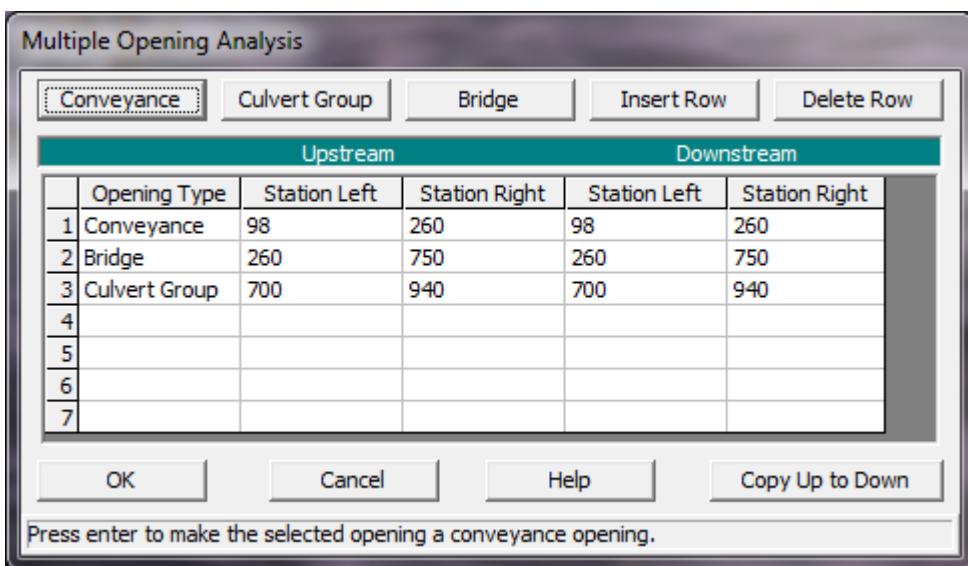


Figure 5 27 Multiple Opening Analysis window

The user selects from the three available opening types: Conveyance; Culvert Group; and Bridge. Openings must be established in order from left to right, while looking in the downstream direction. In addition to establishing the number and types of openings, the user must also enter a Station Left and a Station Right for each opening. These stations are used to establish limits for each opening as well as stagnation points. Stagnation points are the locations at which flow separates (on the upstream side) from one opening to the next adjacent opening. Stagnation points can either be set to fixed locations or they can be allowed to migrate within limits.

As shown in Figure 5-27 (numerical representation) and Figure 5-25 (graphical representation), there are three openings established in this example. The first opening is defined as a conveyance area, and it ranges from station 98 (the left most station of the section) to station 260. That means that any water in this area will be treated as normal open channel flow, and the water surface will be calculated by performing standard step calculations with the energy equation. The second opening is the bridge opening. This opening has a left station of 260 and a right station of 740. This bridge will be modeled by using the cross section data, bridge deck, and pier information that lie within these two stations (260 and 740). The bridge coefficients and modeling approach for this opening will be based on the data entered for bridge opening #1, since it is the first bridge opening. The third opening is a culvert group. This opening has a left station of 650 and a right station of 940. Any culverts that lie within these stations will be considered as being in the same culvert group.

Notice that the right station of the bridge opening overlaps with the left station of the culvert group. This is done on purpose. By overlapping these stations, the user is allowing the program to calculate the location of the stagnation point between these two openings. This allows the stagnation point to vary from one profile to the next. In the current version of the HEC-RAS software, stagnation points are allowed to migrate between any bridge and culvert group openings. However, stagnation points must be set to a fixed location for any conveyance opening type. A more detailed explanation of stagnation points, and how the program uses them, can be found in the HEC-RAS Hydraulics Reference manual, under the section on Multiple Openings (Chapter 7).

Once the user has entered all of the information into the Multiple Opening Analysis window, simply press the **OK** button to accept the data.

Multiple Opening Calculations

Multiple opening calculations are computationally intensive. An iterative solution approach is used, by which the amount of flow through each opening is adjusted until the computed upstream energies of each opening are balanced within a predefined tolerance. The general approach of the solution scheme is as follows:

1. The program makes a first guess at the upstream water surface by setting it to the computed energy of the cross section just downstream of the bridge.
2. The program sets an initial flow distribution. This is accomplished by first calculating the amount of active flow area in each opening, based on the water surface from step one. The program then apportions the flow by using an area weighting (i.e., if an opening has 40 percent of the active flow area, then it will receive 40 percent of the flow).
3. Once a flow distribution is established, the program then calculates the water surface and energy profiles for each opening, using the estimated flow.
4. Once the program has computed the upstream energy for each opening, a comparison is made between the energies to see if a balance has been achieved (i.e., all energies are within the predefined tolerance). If the energies are not within the set tolerance, the program re-distributes the flow based on the computed energies.
5. The program continues this process until either the computed energies are within the tolerance or the number of iterations reaches a pre-defined maximum. The energy balance tolerance is set as 3 times the user entered water surface calculation tolerance (The default is 0.03 feet or 0.009 meters). The maximum number of iterations for multiple opening analysis is set to 1.5 times the user entered maximum number of iterations from the normal water surface calculations (the default is 30 for multiple openings).

A more detailed discussion of how the program performs the multiple opening analyses can be found in Chapter 7 of the HEC-RAS Hydraulic Reference manual.

Inline Structures (Dams, Weirs and Gated Spillways)

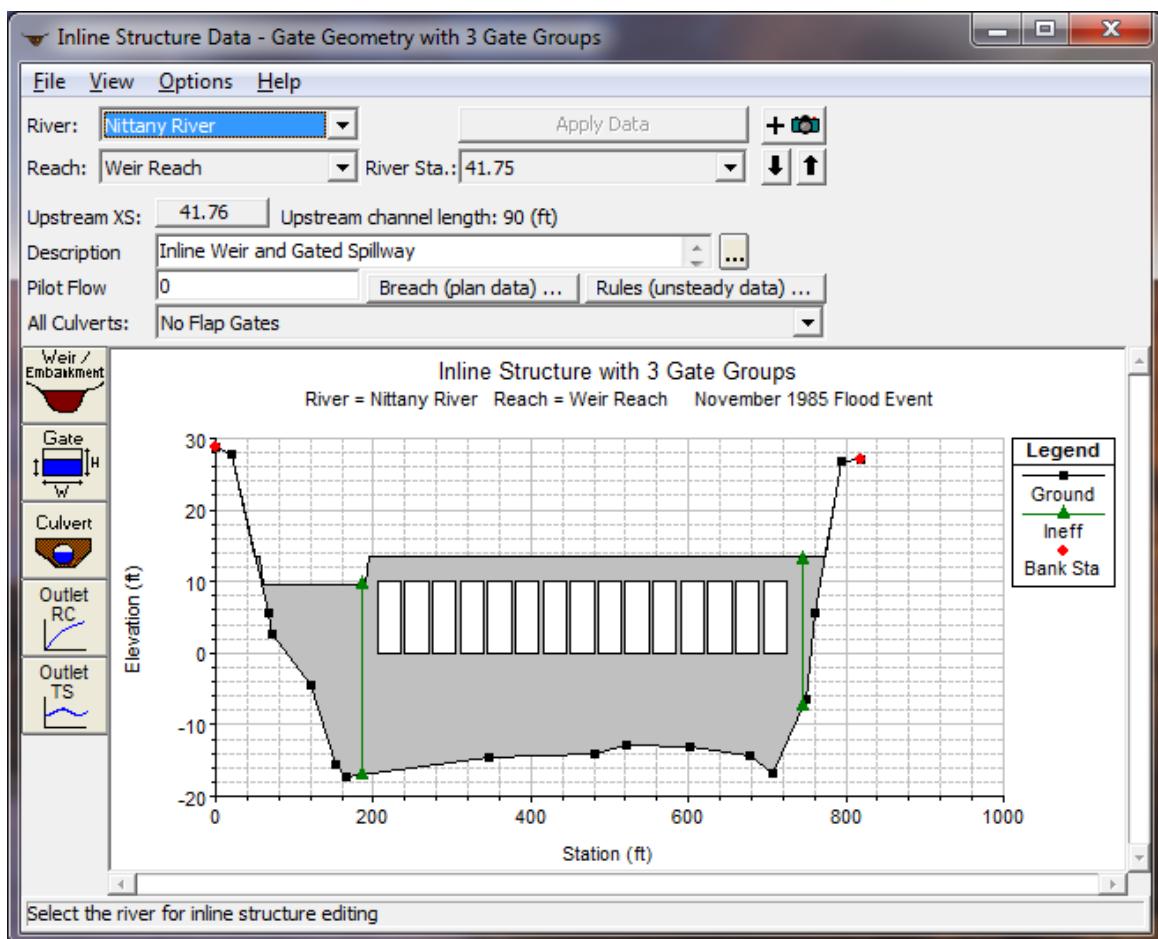


HEC-RAS has the ability to model inline dams, weirs, and gated structures with radial gates (often called tainter gates), vertical lift gates (sluice gates), overflow gates (open to the air or with a closed top), gates modeled with user defined curves, culverts, culverts with flap gates, user defined outlet rating curves, and user specified outlet time series. The spillway crest of the gates can be modeled as an ogee shape, broad crested weir, or a sharp crested weir shape.

This section of the User's manual will describe how to enter the data for inline structures. For information on general modeling guidelines and the hydraulic computations of Inline Structures, please see Chapter 8 of the HEC-RAS Hydraulic Reference manual. To find out how to view specific results for an inline structure, see Chapter 9 of this User's manual.

Entering and Editing Inline Structure Data

Inline structure data are entered in a similar manner as bridge and culvert data. To enter an inline structure, press the Inline Structure button from the Geometric Data window. Once this button is pressed, the Inline Structure Data editor will appear as shown in the figure below (except yours will be blank until you have entered some data).



Inline Structure Data Editor

To add an inline structure to a model, the user must do the following:

1. Select the river and reach that you would like to place this inline structure into. This is accomplished by first selecting a River, then selecting a specific reach within that river. The River and Reach selection buttons are at the top of the Inline Structure Data editor.
2. Go to the **Options** menu at the top of the window and select **Add an Inline Structure** from the list. An input box will appear asking you to enter a river station identifier for locating this structure within the reach. After entering the river station, press the **OK** button and a copy of the cross section just upstream of this river station will appear on the screen. This cross section is used in formulating the inline structure crossing.
3. Enter all of the data for the Inline structure. This data will include a Weir/Embankment profile, any gated spillways that you may be modeling, culverts, and/or outlet rating curves. All of the outlet types are optional, except the embankment profile of the inline structure. If the user does not enter any gated spillways, culverts, etc..., then the program assumes that there is only an inline weir.
4. Once all of the Inline Structure data are entered, press the **Apply Data** button in order for the interface to accept the data. The editor can then be closed by selecting **Exit** from the **File** menu at the top of the window.

River, Reach, and River Station. The River and Reach boxes allow the user to select a river and reach from the available reaches that were put together in the schematic diagram. The river and reach labels define which river and reach the inline structure will be located in. The River Station tag defines where the structure will be located within the specified reach. The River Station tag does not have to be the actual river station of the structure, but it must be a numeric value. The River Station tag for the inline structure should be numerically between the two cross sections that bound the structure. Once the user selects **Add an Inline Structure** from the options menu, an input box will

appear prompting you to enter a River Station tag for the new structure. After the River Station tag is entered, the cross section just upstream of the Inline Structure will be displayed on the editor.

Description. The description box is used to describe the Inline Structure location in more detail than just the river, reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for the Inline Structure plots and tables.

Pilot Flow. This option allows the user to put in a flow rate that will be used as a minimum flow release from the structure. If you have an inline structure in HEC-RAS, no cross section in the model can go dry during the simulation. While you can have a zero flow at the structure, the upstream and downstream cross sections must always have water in them. The pilot flow option is a simple way to ensure that there is always some minor flow going through the structure.

Breach (Plan Data). This button allows the user to define information for evaluating the breaching of this inline structure. The data is actually stored in the currently opened plan file. The editor can also be brought up from the plan editor. This option is only for unsteady flow modeling. To learn more about this option, see Chapter 8 "Performing an Unsteady Flow Analysis."

Rules (unsteady Data). This button brings up the Unsteady Flow Data and Boundary Conditions editor and allows the user to define a set of rules for controlling the gate openings. For more details on the Rules editor, please see Chapter 8, Performing an Unsteady Flow Analysis", in this manual.

All Culverts: This drop down selection box allows user to add flap gates to any culverts entered on the inline structure. The default is for "No Flap Gates", which means flow can go in both directions through the culverts. The other options include "Flaps prevent Negative Flow", which means flow can only go in the positive flow direction through the culverts (Downstream), and "Flaps prevent Positive Flow", which means flow can only go in the negative direction through the culverts (upstream).

Weir/Embankment Editor

The Embankment and Weir data are entered together, and are used to describe the embankment blocking the stream as well as any uncontrolled overflow weirs. To enter the weir and embankment data, press the **Weir/Embankment** button and the editor will appear (see figure below). The Weir/Embankment Data editor is similar to the Deck/Roadway editor for bridges and culverts. The data on the Weir/Embankment editor is the following:

Distance - The distance field is used to enter the distance between the upstream side of the Weir/Embankment (the top of the embankment) and the cross section immediately upstream of the structure. This distance is entered in feet (or meters for metric).

Inline Structure Weir Station Elevation Editor

Distance	Width	Weir Coef
20	50	3.95

Clear **Del Row** **Ins Row** **Filter...**

Edit Station and Elevation coordinates

	Station	Elevation
1	0.	13.5
2	57.	13.5
3	61.	9.5
4	190.	9.5
5	194.	13.5
6	1000.	13.5
7		
8		

U.S Embankment SS D.S Embankment SS

Weir Data

Weir Crest Shape

Broad Crested
 Ogee

Spillway Approach Height:
Design Energy Head: **Cd ...**

OK **Cancel**

Enter distance between upstream cross section and deck/roadway. (ft)

Weir and Embankment Data Editor

Width - The width field is used to enter the width of the top of the embankment along the stream. The distance between the top of the downstream side of the embankment and the downstream bounding cross section will equal the main channel reach length of the upstream cross section minus the sum of the weir/embankment "width" and the "distance" between the embankment and the upstream section. The width of the embankment should be entered in feet (meters for metric).

Weir Coefficient - Coefficient that will be used for weir flow over the embankment in the standard weir equation.

Station and Elevation Coordinates - This table is used to define the geometry of the Weir and the Embankment. The information is entered from left to right in cross section stationing. The user enters stations and elevations of the top of the embankment and weir. The stationing does not have to equal the stations in the bounding cross section, but it must be based on the same origin. Everything below these elevations will be filled in down to the ground. The **Del Row** and **Ins Row** buttons allow the user to delete and insert rows.

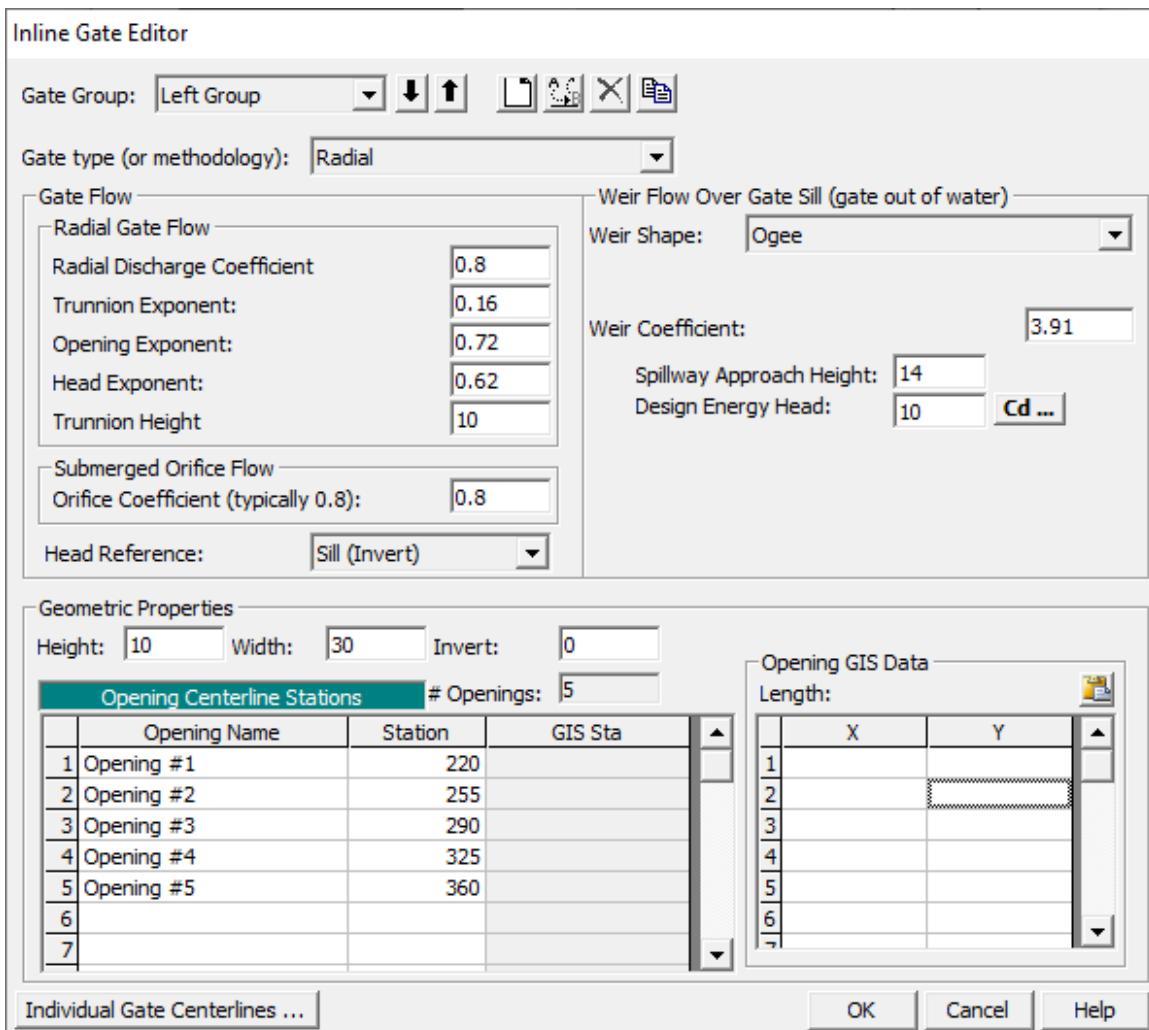
U.S. Embankment SS - This field is used to enter the slope of the road embankment on the upstream side of the structure. The slope should be entered as the horizontal to vertical distance ratio of the embankment.

D.S. Embankment SS - This field is used to enter the slope of the road embankment on the downstream side of the structure. The slope should be entered as the horizontal to vertical distance ratio of the embankment.

Weir Crest Shape - When submergence occurs over the weir there are two choices available to figure out how much the weir coefficient should be reduced due to the submergence. These two criteria are based on the shape of the weir. The first method is based on work that was done on a trapezoidal shaped broad crested weir (FHWA, 1978). The second criterion was developed for an Ogee spillway shape (COE, 1965). The user should pick the criterion that best matches their problem. If the user selects the Ogee Spillway shape, then some additional information is required. For an Ogee shaped weir the user must enter the "Spillway Approach Height" and the "Design Energy Head". The spillway approach height is equal to the elevation of the spillway crest minus the mean elevation of the ground just upstream of the spillway. The design energy head is equal to the energy grade line elevation (at the design discharge) minus the elevation of the spillway crest. In addition to these two parameters, the user has the option to have the program calculate the weir coefficient at the design discharge. This is accomplished by pressing the **C_d** button. Once this button is pressed, the program will compute a weir coefficient for the Ogee spillway based on the design head. During the weir calculations, this coefficient will fluctuate based on the actual head going over the spillway. The curves used for calculating the Ogee spillway coefficient at design head, and discharges other than design head, were taken from the Bureau of Reclamation publication "Design of Small Dams", Figures 249 and 250 on page 378 (Bureau of Reclamation, 1977).

Gated Spillway Editor

In addition to uncontrolled overflow weirs, the user can add gated spillways (this is optional). To add gated spillways to the structure, press the **Gate** button on the Inline Structure data editor. Once this button is pressed, the gated editor will appear as shown in the figure below (Except yours will be blank until you have entered some data).



Gated Spillway Editor

The Gated Spillway editor is similar to the Culvert editor in concept. The user enters the physical description of the gates, as well as the required coefficients, in the Gated Spillway editor. The functionality of the gates is defined as part of the Unsteady Flow Data editor or the Steady Flow data editor (on a per profile basis). The following is a list of the data contained on this editor:

Gate Group - The Gate Group is automatically assigned to "Gate #1" the first time you open the editor. The user can enter up to 20 different Gate Groups at each particular river crossing, and each gate group can have up to 25 identical gate openings. If all of the gate openings are exactly the same, and any opened gates will be operated in the same manner, then only one gate group needs to be entered. However, if the user has gate openings that are different in shape, size, elevation, have different coefficients, or they will be operated differently, then additional Gate Groups must be added for each Gate type. To add an additional gate group you can either use the **Add** or **Copy** buttons. The Add button increments the Gate # and clears the gate editor. The Copy button increments the Gate # and makes a copy of the original Gate group data. Once a copy is made of the gate data, the user can change any of the existing gate information. Gate groups can be deleted by pressing the **Delete** button. Also, if the gates are identical, but the user wants to be able to open the gates to different elevations, then the user must have a separate gate group for each set of gates that

will be opened to different elevations. In steady flow computations, the user can specify the number of gates in a group to be opened, but in unsteady flow computations all of the gates in a group are opened in exactly the same way (this is a limitation of our unsteady flow implementation of gates currently)

Geometric Properties of the Gates

Height - This field is used to enter the maximum possible height that the gate can be opened in feet (meters for metric).

Width - This field is used for entering the width of the gate in feet (meters).

Invert - This field is used for entering the elevation of the gate invert (sill elevation of the spillway inside of the gate) in feet (meters for metric). For overflow gates this is the lowest elevation that the gate will open to.

Opening Centerline Stations - This table is used for entering the centerline stationing of the gate openings. The user should enter a different centerline stationing for each gate opening that is part of the current gate group. All gate openings within the same gate group are exactly identical in every way, except their centerline stationing. As a user adds new centerline stationing values, the number of identical gates in the group is automatically incremented and displayed in the field labeled "# Openings".

Opening GIS Data - This table is used to enter X, Y coordinates for a line representing each gate. X, Y coordinates are only used when gates are connected to 2D Flow Areas. When a Gate is put into a Inline Structure of a 1D River Reach, then the X, Y coordinates are not needed or used. This table is only relevant to Lateral Structures and SA/2D Area Connections in which culverts will be connected to 2D Flow Area cells. It is not needed for pure 1D Inline Structures.

Gate Flow Coefficients

Gate Type (or methodology) - This field is used for selecting the type of gate. Five options are available for gate types: sluice (vertical lift gate), radial (tainter gate), Overflow (closed top), Overflow (open to the air), and User Defined Curves. Once a gate type is selected, the right hand side of the gate editor will change to show the required information for that gate type. Not all of the information is required for each gate type.

Discharge Coefficient - This field is used for entering the coefficient of discharge for the gate opening. This coefficient ranges from 0.6 to 0.8 for Radial gates and 0.5 to 0.7 for sluice gates. This coefficient is not required for overflow gates that are open to the air.

Trunnion Exponent - This field is used to enter the trunnion height exponent, which is used in the radial gate equation. The default value for this field is 0.0.

Opening Exponent - This field is used to enter the gate opening exponent, which is used in the radial gate equation. A default value of 1.0 is automatically set for this field.

Head Exponent - This field is used to enter the upstream energy head exponent, which is used in the radial gate equation. A default value of 0.5 is automatically set for this field.

Trunnion Height - This field is used for entering the height from the spillway crest to the trunnion pivot point. This data is only used for radial gates. See Chapter 8 of the Hydraulic Reference manual for more details on this variable.

Orifice Coefficient - This field is used to enter an orifice coefficient, which will be used for the gate opening when the gate becomes more than 80 percent submerged. Between 67 percent and 80 percent submerged, the program uses a transition between the fully submerged orifice equation and the free flow gate equations. When the flow is less than 67 percent submerged, the program uses the free flow gate equations. This coefficient is not required for overflow gates that are open to the air.

Head Reference – This field is used to select the reference point for which the upstream energy head will be computed from. The default is the gate sill (invert), which is normally used when the flow through the gate goes out into a channel. If the gate causes the flow to jet out freely into the atmosphere, then the head reference should be selected as the centerline elevation of the gate opening. If the gate crest is an ogee spillway crest, then the center of the gate opening should be used. Ogee spillway crests are normally designed to follow the shape of water jetting freely into the atmosphere.

Coefficients for Weir Flow over the Gate Sill

If a gate is opened to the point at which the top of the gate is no longer touching the water (or if an open air overflow gate is being used), then the flow through the gate is modeled as weir flow. The program will automatically transition from gate flow to weir flow when the upstream head is between 1.0 to 1.1 times the height of the gate opening. The following parameters are required to model weir flow through the gate opening.

Weir Shape - This parameter allows the user to select between a Broad Crested shape weir, an Ogee shaped weir, or a sharp crested weir. Depending on which shape is selected, the program will use a different submergence criteria during the calculations. In addition to the submergence criteria, if the user selects the Ogee shape, the program will bring up additional data entry fields that must be entered by the user. For the ogee weir shape, the additional fields are the Spillway Approach Height and the Design Energy Head, which are explained below. Once these fields are entered, the user should press the button labeled Cd. When this button is pressed, the program will compute a weir coefficient for the Ogee spillway based on the design head. During the weir calculations, this coefficient will fluctuate based on the actual head going over the gated spillway. The curves used for calculating the Ogee spillway coefficient at design head, and discharges other than design head, work taken from the Bureau of Reclamation publication "Design of Small Dams", Figures 249 and 250 on page 378 (Bureau of Reclamation, 1977).

Weir Method – This field is only available when the Sharp Crested Weir shape is selected. If a sharp crested weir shape is selected, then the user has three choices for defining the weir coefficient: User Entered Coefficient; Compute with Rehbock equation; and Compute with Kindsvater-Carter Equation. If the "User Entered Coefficient" option is selected, then the user simply enters a coefficient that will be used for weir flow through the gate, for all head ranges. If the "Rehbock equation" is selected, the user is asked to enter a spillway approach height (explained below), and the weir coefficient is then computed with the Rehbock equation. If the "Kindsvater-Carter equation" is selected, then the user must enter a spillway approach height, and also select which form of the Kindsvater-Carter equation will be used. The form of the Kindsvater-Carter equation is based on selecting one of eleven equations that are based on varying L/b. Where L is the width of the gate opening, and b is the top width of the approaching water upstream of the gate. If more than one gate

is defined at a particular opening, you must figure out an average approach width for flow going to each gate.

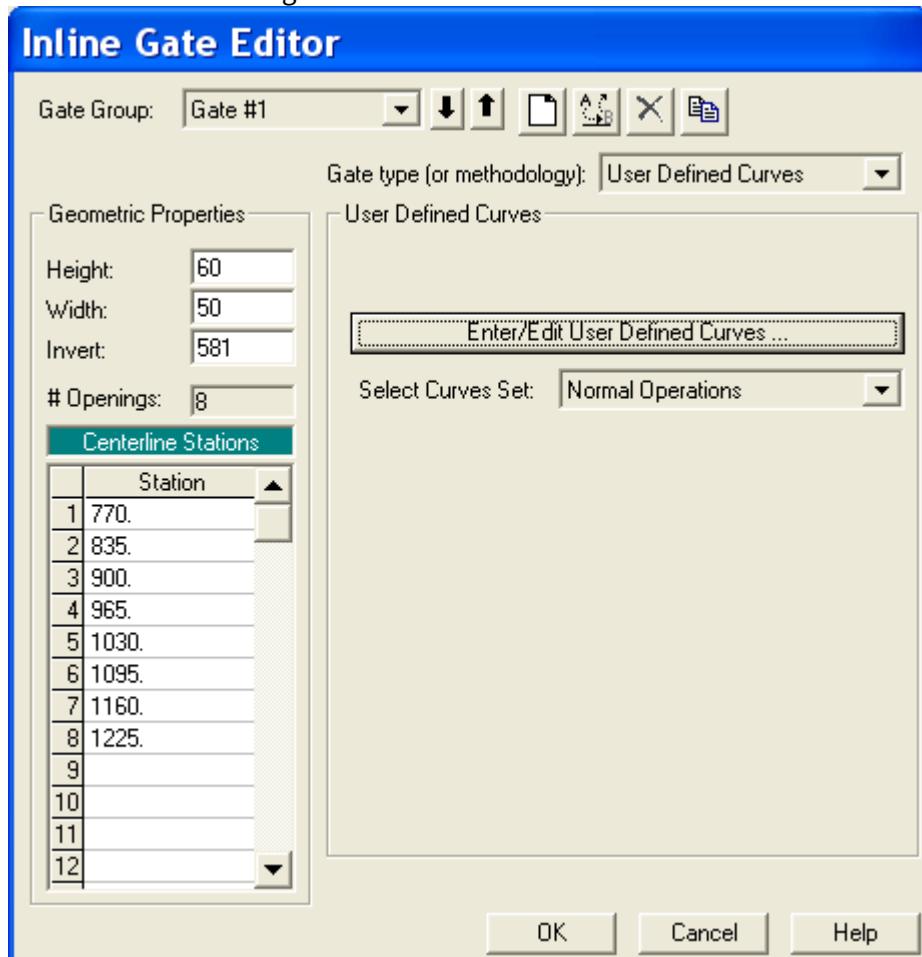
Weir Coefficient - This field is used for entering a weir coefficient that will be used for the gate opening. This coefficient will only be used when the gate is opened to an elevation higher than the upstream water surface elevation. When this occurs, the flow through the gate is calculated as weir flow. If the Kindsvater-Carter equation is selected, then this field is used to select which form of the equation will be used to compute the coefficient.

Spillway Approach Height - The spillway approach height is equal to the elevation of the spillway crest minus the mean elevation of the ground just upstream of the spillway.

Design Energy Head - The design energy head is equal to the energy grade line elevation (at the design discharge) minus the elevation of the spillway.

User Defined Gate Curves

When the user selects "User Defined Curves" for the gate type, then the editor will change to look like the one shown in the figure below.



Gate Editor with User Defined Curves Selected.

As shown in the figure above, the user must select the button that says "Enter/Edit User Defined Curves". This will bring up another editor that allows the user to enter the curves into a table. The user is also required to give a name for each set of curves. More than one curve set can be entered, and the user can then select a different curve set for each gate group (if desired). Each curve set represents the head versus flow relationships for one single gate opening. When there are 2 or more identical gates in a group, each gate in the group gets the same curve set applied to it. When the "Enter/Edit User Defined Curves" button is pressed, a new editor will appear as shown in figure below.

- ① The user defined gate curves represent a set of curves, one curve for each gate opening, in which the curve is based on headwater only control, and no downstream tailwater influence. To include the influence of tailwater, the user would have to enter a full family of rating curves (Headwater, tailwater and flow) for every possible gate opening). This is beyond the scope of this option. The normal mode of computing gate flow from the gate flow equations takes into account the influence of tailwater, but this user entered curve method does not.

User Defined Gate Performance Curves									
Set:		Normal Operations							
Enter head water in first row, gate openings in first column, resulting flows in rest of table									
		HW	HW	HW	HW	HW	HW	HW	HW
1	Gate Open Ht\Hw	581	585	590	595	600	605	610	
2	0	0	0	0	0	0	0	0	
3	1	0	1450	1500	1750	1850	1900	1950	
4	3	0	4000	4750	5000	5250	5350	5500	
5	5	0	6700	7900	8400	8600	8800	9000	
6	7	0	9200	10900	11600	12000	12300	12500	
7	9	0	11700	13800	14750	15300	15600	16000	
8	11	0	14000	16700	17800	18500	18850	19400	
9	13	0	16400	19450	20800	21600	22200	22600	
10	15	0	18500	22200	23900	24800	25400	26000	
11	17	0	20500	24800	26750	27900	28550	29300	
12	19	0	22600	27500	29700	30950	31700	32500	
13	21	0	24500	30100	32500	34000	34800	35700	
14	23	0	26000	32650	35500	37000	38000	39000	
15	25	0	28200	35200	38250	40000	41100	42200	
16	27	0	28200	37700	41100	43000	44200	45450	
17	29	0	28200	40000	43900	46050	47350	48600	
18	31	0	28200	42250	46750	49100	50600	52000	
19	33	0	28200	45500	49500	52100	53700	55300	
20	35	0	28200	45500	53000	55250	57000	58550	
21	37	0	28200	45500	55500	58500	60200	62000	
22	38	0	28200	45500	55500	60000	61750	63600	
23	39	0	28200	45500	55500	61750	63500	65300	
24	40	0	28200	45500	55500	63500	65500	70000	

Editor for Entering/Editing Gate Curves.

As shown in figure above, the user enters gate openings in the first column. Headwater elevations are entered in the first row. The remaining fields are the corresponding flow for a given gate opening and upstream headwater elevation. To enter a new curve set, the user must first select the "New

"User Curve" button at the top of the editor. When this button is selected the user will be prompted to enter a name for the curve set. Other buttons at the top of the editor are for renaming the curve set, deleting the curve set, and copying the curve set to a new name. Once the curve set, or sets, are entered, the user simply selects a curve set for each gate group desired. The user also has the option to use curve sets for some gate groups and have the program calculate the flow from equations for others.

Once all of the data for the gates has been entered, the user needs to press the **OK** button for the data to be accepted. If the user does not want to use the new data, and would like to go back to the original data they had before entering the Gate Editor, press the **Cancel** button. If the user presses the **OK** button, this does not mean that the data is saved to the hard disk, it is only stored in memory and accepted as being good data. This data is part of the geometry data, and is stored in the geometric data file. The data can be stored to the hard disk by selecting one of the save options from the File menu of the Geometric Data window.

Culverts

User can also enter culverts at inline structures. Culverts can be entered as groups of identical culvert barrels (up to 25 barrels per culvert group), or the user can have up to 20 different culvert groups in which the culverts can be all kinds of shapes, sizes, elevations, roughness, etc... Culverts can compute flow in both directions, or user's have the option to have flap gates to prevent either negative flow (Flow upstream), or positive flow (Flow downstream) through the culverts. For details on the specific data to model a culvert, please review the section on culverts earlier in this chapter.

Outlet Rating Curves

If a user has an outlet type that does not exist in HEC-RAS, or cannot be modeled accurately with the available weirs, gates, and culverts, then an outlet rating curve can be used to model that specific outlet. The Outlet Rating curve can be based on upstream water surface elevations versus outlet flow, or it can be based on upstream total flow versus the flow through the outlet. This method does not take into account downstream tailwater influences on the flow rate.

Outlet Time Series

This option allows the user to specify a time series of flows to be used as an additional outlet through the inline structure. When this option is selected, the user enters a name to identify the outlet. For example, let's say you call the time series "Hydropower", which may represent flows going through a hydropower station at the structure. Then, the user can attach a flow hydrograph to the inline structure in the Unsteady Flow Data editor. This flow is then assigned to the user specified Outlet time Series specified on the inline structure. Only one, Outlet time series can be defined per inline structure, and only one hydrograph can be entered/attached to the inline structure in the unsteady flow data editor. If you have more than one time series you would like to use, combine them into one flow hydrograph outside of HEC-RAS, and then use that as the time series data.

Lateral Structures (Weirs, Gated Spillways, Culverts, and Diversion Rating Curves)



At any lateral structure HEC-RAS has the ability to model lateral weirs, gated spillways, culverts, diversion rating curves, and an outlet time series. The user can set up a single lateral weir, a weir and separate set of gates, a weir and group of culverts, or any combination of weir, gates, culverts, rating curves, and a time series outlet. The gated spillways can have either radial gates (often called tainter gates), vertical lift gates (sluice gates), overflow gates (open to the air or with a covered top), or user defined gate curves. The spillway crest of the gates can be modeled as either an ogee shape, broad crested weir, or sharp crested weir shapes. The culverts can be any of the available shapes from the standard HEC-RAS culvert capability. The diversion rating curve can be used alone, or in conjunction with the other hydraulic outlet types. The rating curve can be used to represent an entire structure or a particular outlet that could not be modeled with HEC-RAS. Lateral structures can be connected to storage areas, 2D Flow Areas, or another river reach.

The lateral structure option can also be used to model a levee. In general, the user should end their cross sections at the inside top of the levee, and then use the lateral structure option to represent the top of the levee along the stream. The area behind the levee could be represented with either a 2D Flow Area, storage area (or combination of interconnected storage areas), or another river reach. Water that goes above the levee will be modeled as weir flow. The user also has the option to evaluate levee breaching.

HEC-RAS now has the option to have georeferenced lateral structures. Under the menu item labeled **GIS Tools**, there is now a table option called **Lateral Structure Centerlines Table**. User can use the **Measure Tool** to draw a line that would represent the lateral structure geospatial X and Y coordinates, then paste those coordinates into the Lateral Structure Centerline Table (This is optional). If a user inserts geospatial coordinates for a lateral structure, not only will it be drawn geospatially correct, but HEC-RAS will figure out how elements (1D cross sections and 2D Face Points) are connected to the lateral structure based on its spatial location.

- ⓘ If you put in a Geospatial centerline for a lateral structure, the length of the lateral structure weir/embankment stationing must be within 0.5% of the length of the centerline put in (i.e. they need to be consistent with each other in terms of length).

This section of the User's manual will describe how to enter the data for lateral weirs, gated spillways, culverts, lateral rating curves, and outlet time series. For information on general modeling guidelines and the hydraulic computations of lateral weirs, gated spillways, and culverts, please see Chapter 8 of the HEC-RAS Hydraulic Reference manual. To find out how to view specific results for a lateral structure, see Chapter 9 of this User's manual.

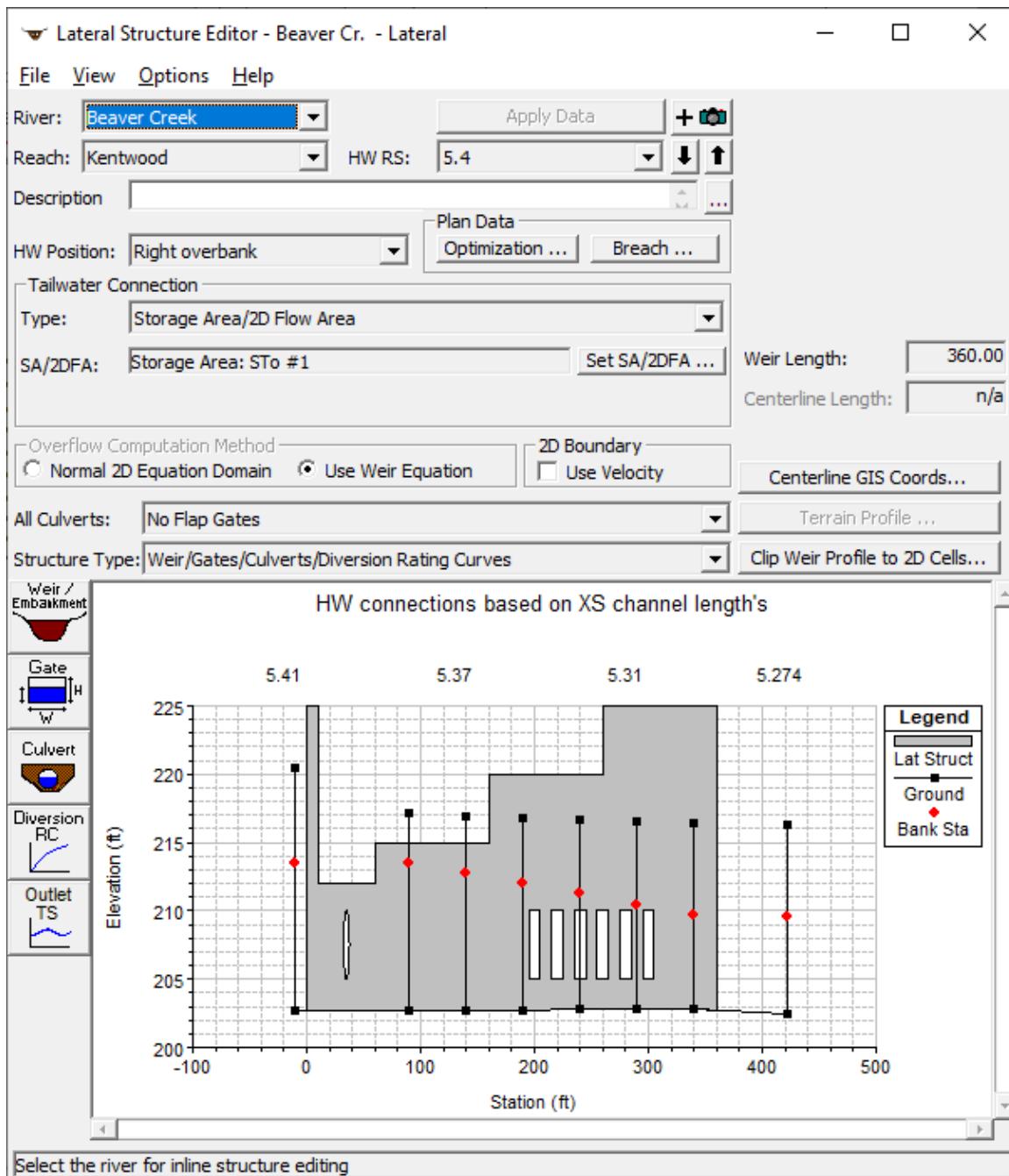
Entering and Editing Lateral Structure Data

Lateral weir, gated spillway, and culvert data are entered in a similar manner as bridge and culvert data. To enter a lateral structure, press the **Lateral Structure** button from the Geometric Data

window. Once this button is pressed, the Lateral Structure Data editor will appear as shown in the figure below (except yours will be blank until you have entered some data).

To add a lateral structure to a model, the user must do the following:

1. Select the river and reach that you would like to place this lateral structure into. This is accomplished by first selecting a River, then selecting a specific reach within that river. The River and Reach selection buttons are at the top of the Lateral Structure Data editor.
2. Go to the **Options** menu at the top of the window and select **Add a Lateral Structure** from the list. An input box will appear asking you to enter a river station identifier for locating this structure within the reach. The river station you enter will represent the location of the upstream end of the lateral structure. The river station must be unique, and should be numerically between the river station values of the upstream cross section and the next section downstream. After entering the river station, press the **OK** button and a profile plot of the channel invert and cross sections in the vicinity of the lateral weir/spillway will be displayed.



Lateral Weir, Gated Spillway, and Culvert Editor

- Enter all of the data for the Lateral Weir, Gated Spillways, Culverts, Diversion Rating Curves, and Outlet Time Series. All of the outlet types are optional, and can be mixed and matched to form a single lateral structure. If the user does not enter any gated spillways, culverts, rating curves, or outlet time series, then the program assumes that there is only a lateral weir. If the user wants to enter only gated spillways, culverts, or a rating curve, and no lateral weir, they must still enter a weir embankment.
- Once all of the data are entered, press the **Apply Data** button in order for the interface to accept the data. The editor can then be closed by selecting **Exit** from the **File** menu at the top of the window.

3. If any gated spillways were entered, the user must go to the Steady or Unsteady Flow Data Editor to control the gate settings for each individual event. If an outlet time series was entered, then the user must attach a flow hydrograph to the lateral structure in the Unsteady Flow Data editor.

The user can have up to two lateral structures defined between any two given cross sections. However, the lateral structure must be placed on opposite sides of the channel (i.e. one on the left and one on the right), and the river stations of each lateral structure must be different (though still contained within the two cross section river station values). Also, any lateral structure can be longer than the distance between cross sections. The user can have a lateral structure that extends downstream, encompassing up to 100 cross sections. If you have a lateral structure that is longer than that, you must break it up into separate lateral structures.

River, Reach, and River Station. The River and Reach boxes allow the user to select a river and reach from the available reaches that were defined in the schematic diagram. The river and reach labels define which river and reach the lateral structure will be located in. The Head Water River Station (HW RS) tag defines where the structure will be located within the specified reach. The River Station tag does not have to be the actual river station of the structure, but it must be a numeric value. The River Station tag for the lateral structure should be numerically between the two cross sections that bound the upstream end of the structure. Once the user selects **Add a Lateral Structure** from the options menu, an input box will appear prompting you to enter a River Station tag for the new structure. After the River Station tag is entered, a profile plot of the reach thalweg will be displayed for the bounding cross sections in the graphic window. The river and reach in which the lateral structure is defined is considered to be the headwater side of the structure. Whatever the user connects the lateral structure to is considered to be the tailwater side of the structure.

Description. The description box is used to describe the Lateral Structure location in more detail than just the river, reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for the Lateral Structure plots and tables.

HW Position. The headwater position box is used to define where the lateral structure is located spatially within the reach that it is defined. The user can select one of the following: Left overbank; Next to left bank station; Next to right bank station; and Right overbank. When the user selects "Left overbank", the weir is assumed to be located at the left end (beginning cross section station) of the cross section data, looking in the downstream direction. When the user selects "Next to left bank station", the weir is assumed to be located on the left edge of the main channel. When the user selects "Next to right bank station", the weir is assumed to be located on the right edge of the main channel. When the user selects "Right overbank", the weir is assumed to be located at the right end of the cross section data.

Tailwater Connection. This area of the editor is used to define what the lateral structure is connected to (i.e. where the water leaving from the main river will be going). A lateral structure can be connected to a storage area; 2D Flow Area; cross sections in another river reach; or nothing at all (defined as leaving the system). To set the tailwater connection, first select the connection type from the area labeled: **Type**. Then, depending on the type of tailwater connection, other information may be required. If the tailwater connection type is **Out of the System**, then no other information is required. If the tailwater connection type is a **Storage Area/2D Flow Area**, then the user is required to select a storage area or a 2D Flow Area from a drop down list of the currently defined storage areas. If the tailwater connection type is **Cross Sections of a River/Reach**, then the user is required to select the river, reach, and range of cross sections that the lateral structure is connected to. The

tailwater connection can be to a single cross section (all the flow goes to one point), or it can be set to a range of cross sections (the flow will be distributed over the range of cross sections). In addition to the cross section(s) the user must define if the connection is on the right over bank; next to the right bank station; next to the left bank station, or on the left overbank. Water can also flow in the reverse direction through a lateral structure if the connected to location has a higher water surface than the from location. Reverse flow gets labeled as negative flows for a lateral structure.

Structure Type. This field is used to select the type of routing that will be used for this structure. There are two options, **Weir/Gates/Culverts/ Diversion Rating Curves** (the default) and **Linear Routing**. The default option is where the program calculated the flow across the structure by performing detailed hydraulic calculations for the weir, gated spillways, culverts, and any rating curve. The second option, Linear Routing, is a simplified method in which the user just puts in a linear routing coefficient. This coefficient can vary between 0.0 and 1.0, with 1.0 representing sending the maximum flow over the structure and 0.0 representing no flow. The linear routing method is a simple storage accounting method. This method can be very useful when the user has many lateral structures connected to storage areas, and a detailed flow calculation over each structure is not necessary. Also, the linear routing method is computationally faster and more stable. Typical values for the linear routing coefficient are from 0.05 to 0.2. However, this coefficient needs to be calibrated.

Culvert Flap Gates. The drop down box right above the Structure Type pertains to having flap gates on culverts. This option only affects the flow through the culverts, not the weir or the gated structures. The options are no flap gates (default), Flaps prevent negative flow, and flaps prevent positive flow. No flap gates means that flow is allowed to go in both directions through the culvert. The "Flaps prevent negative flow" option means that flow can only go in the positive flow direction through the culverts. Positive flow is assumed to be taking flow away from the river for a lateral structure. Therefore, the "Flaps prevent negative flow" option would allow water to go away from the main river through the culverts, but not back into the river. The final option, "flaps prevent positive flow", would only allow water to come into the main river through the culverts, but not away from the main river.

Overflow Computation Method. This option is used to control what equations are used to compute the flow across a Lateral Structure. By default the "Weir" equation is used. However, if the Lateral Structure is connected to a 2D Flow Area, then there is the option to use the 2D Flow equations to figure out how much flow is going across the Lateral Structure, and in which direction. When the 2D Flow Equations overflow method is selected, an average water surface elevation is computed one the 1D river side, in front of each 2D cell that is connected with the Lateral Structure. This water surface is then applied as a stage boundary condition for each individual cell of the 2D area. So, if water is above the weir profile, and the elevation is higher in the 1D river than the 2D cell, then water will go from the 1D river to the 2D cell. If the 2D cell is higher than the 1D river, then water will go from the 2D cell to the 1D river. If the water is below the elevation of the weir, then the flow is zero for that cell.

2D Boundary. This option is used only when a Lateral Structure is connected to a 2D Flow Area. In general for this type of flow connection, flow is computed across the structure and passed into and out of the 2D cells, and no velocity is considered at the face. If this option is turned on, in addition to flow, a velocity is computed and applied to the face of the 2D cells. By default this is off, which produces a more stable solution, but less velocity accuracy. If turned on, it will produce more accurate flow velocities into the 2D area, but may be less stable.

Optimization. This option is for steady flow modeling, or the initial conditions of an Unsteady flow model. When modeling in a steady flow mode (or unsteady flow initial conditions), the user can have the software figure out how much flow will leave through the lateral structure, and how much will continue on downstream. This calculation requires an iterative solution. Pressing the Optimization button brings up an editor that allows the user to turn the optimization option on. When optimization is not turned on, the program will assume all of the water is still going downstream, though it will calculate what could have gone out the lateral weir based on the computed water surface. When optimization is turned on, the software calculates the flow out of the lateral structure, reduces the flow in the main river, and then recalculates the profile in the main river. This operation continues until there is a balance between the calculated and assumed flows for the main river.

Breach. This button allows the user to define information for evaluating the breaching of this lateral structure. The data is actually stored in the currently opened plan file. The editor can also be brought up from the plan editor. This option is only for unsteady flow modeling. To learn more about this option, see Chapter 16 "Advanced Features for Unsteady Flow Routing."

Weir/Embankment Editor

The Embankment and Weir data are entered together, and are used to describe the embankment in which the outlets will be placed, as well as any uncontrolled weirs. To enter the weir and embankment data, press the **Weir/Embankment** button and the editor will appear as shown in the

figure below.

Lateral Weir Embankment

Weir Data	<input type="text" value="10."/>
Weir Width	<input type="text" value="10."/>
Weir Computations:	Standard Weir Eqn
Standard Weir Equation Parameters	
Weir flow reference:	Water Surface
Weir Coefficient (Cd)	<input type="text" value="2."/>
Weir Crest Shape:	Broad Crested
Weir Stationing Reference	
HW - Distance to Upstream XS:	<input type="text" value="10."/>
HW Connections ... TW Connections ...	
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Embankment Station/Elevation Table

Station	Elevation
1	225
2	225
3	212
4	212
5	215
6	215
7	220
8	220
9	225
10	225
11	
12	
13	
14	
15	
16	
17	
18	
19	
20	
21	

Lateral Weir/Embankment Editor

The Lateral Weir/Embankment Data editor is similar to the Deck/Roadway editor for bridges and culverts. The data on the Weir/Embankment editor is the following:

Weir Width - The width field is used to enter the width of the top of the embankment. This value will only be used for graphical plotting, and does not have any effect on the computations. The width of the embankment should be entered in feet (meters for metric).

Weir Computations – This field allows the user to select either the standard weir equation or Hager's lateral weir equation. When the standard weir equation is selected, the user will also need to enter a weir flow reference head, and a weir coefficient. If Hager's lateral weir equation is selected, the user must also enter: default weir coefficient; weir average height; an average bed slope, and a weir angle in degrees if it is anything other than parallel to the stream.

Weir flow reference - This value is used to select whether weir flow is computed by using the energy gradeline or the water surface from the cross sections. The default is to use the energy gradeline.

Weir Coefficient - Coefficient that will be used for weir flow over the embankment in the standard weir equation.

Weir Crest Shape - When submergence occurs over the weir/embankment there are four choices available to figure out how much the weir coefficient should be reduced due to the submergence. These four criteria are based on the shape of the weir. The first method is based on work that was done on a trapezoidal shaped broad crested weir (FHWA, 1978). The second criterion was developed for an Ogee spillway shape (COE, 1965). The third is for a sharp crested weir. The last selection is when there really is no weir, and flow is just traveling overland. This is called a "Zero Height" weir. The user should pick the criterion that best matches their problem. If the user selects the Ogee Spillway shape, then some additional information is required. For an Ogee shaped weir the user must enter the "Spillway Approach Height" and the "Design Energy Head". The spillway approach height is equal to the elevation of the spillway crest minus the mean elevation of the ground just upstream of the spillway. The design energy head is equal to the energy grade line elevation (at the design discharge) minus the elevation of the spillway crest. In addition to these two parameters, the user has the option to have the program calculate the weir coefficient at the design discharge. This is accomplished by pressing the **C_d** button. Once this button is pressed, the program will compute a weir coefficient for the Ogee spillway based on the design head. During the weir calculations, this coefficient will fluctuate based on the actual head going over the spillway. The curves used for calculating the Ogee spillway coefficient at design head, and discharges other than design head, were taken from the Bureau of Reclamation publication "Design of Small Dams", Figures 249 and 250 on page 378 (Bureau of Reclamation, 1977).

HW Distance to Upstream XS - This field is used to enter the distance between the upstream end of the Weir/Embankment (based on where the user will start to enter the embankment data) and the cross section immediately upstream of the structure. This distance is entered in feet (or meters for metric).

TW Flow Goes – If the lateral structure is set to send flow to another river reach, this field will be active. This field is used to select where the flow will go to in the tailwater (TW) reach. Flow can be set to go into a single point, or it can be set to go over a range of cross sections.

TW Distance to Upstream Cross Section – If the lateral structure is set to send flow to another river reach, this field will be active. This field is used to enter a distance between the connected tailwater cross section, and the actual location in which the weir begins to connect. Default value is zero, meaning the weir starts at the selected tailwater cross section and continues downstream from there. Only positive values can be entered. Any number greater than zero means that the weir connection starts downstream of the connected cross section, by the user entered distance.

Weir Station and Elevation - This table is used to define the geometry of the Weir and the Embankment. The information is entered from upstream to downstream in stationing. The user enters stations and elevations of the top of the embankment and weir. The stationing is relative, so it can be started at any number (i.e. 0, 100, etc...). The user enters stations and elevations from the upstream end to the downstream end of the lateral structure. Everything below these elevations will be filled in to the ground. By default, the lateral structure will be lined up with the river/reach by comparing the stationing entered with the reach lengths of the river/reach. If the lateral structure is connected to the right overbank of the reach, then the right overbank reach lengths are used. If the lateral structure is connected to the right or left bank station of the main channel, then the main channel reach lengths will be used. The **Filter** button allows the user to filter the station and elevation points in order to reduce the total number of points. This feature is often used when a lateral weir is used to represent a natural overflow area, and the data has come from a GIS.

Headwater Connections – This table will by default show the weir stationing that intersects with the cross sections in the river/reach that the structure is defined in. The software automatically aligns the weir with the cross sections in the reach based on the weir stationing and the reach lengths in the cross sections (either left overbank, main channel, or right overbank reach lengths). However, if the user does not like how the defined weir intersects with the cross sections in the reach, they can define their own intersection points by entering the desired weir stationing to intersect with each of the cross sections in the reach. Water surface elevations for the lateral structure will then be interpolated based on the user entered stationing.

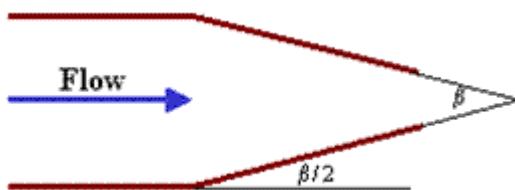
Tailwater Connections – This table can be used to line up the weir to either a 2D Flow Area or cross sections in another reach in a user controlled manner. By default the weir is automatically lined up to any 2D Flow Area or cross sections of another river reach that it is connected too. However, this editor will allow users to hand change the connection locations to the 2D Flow Area Face points, or cross section river station locations of the river reach. The user can select to use "User specified intersections", which will allow them to describe where the stationing of the weir hits the tailwater cross sections or 2D Flow Area Face points. If this option is used the user must completely define where the entire length of the weir intersects the various tailwater cross sections or 2D Flow Area. If Hager's Lateral weir equation is selected from the "Weir Computations" field, then the following additional fields will appear:

Default Weir Coefficient (C_d) – This weir coefficient will be used for the first iteration of trying the Hager lateral weir equation. The equation is iterative, and requires hydraulic results in order to make a weir coefficient calculation. The default weir coefficient is only used for the first guess at the hydraulic computations.

Weir Average Height – This field is used for entering the average height of the weir above the ground.

Average Bed Slope (Optional) – This field is used for entering the average slope of the stream bed in the reach of river that contains the lateral weir. If the user does not enter this field, the HEC-RAS program will compute the slope by estimating an average bed elevation for each cross section, then computing the slope of the average bed elevation. Average bed elevation of an irregular cross section is obtained by subtracting hydraulic depth from the water surface elevation.

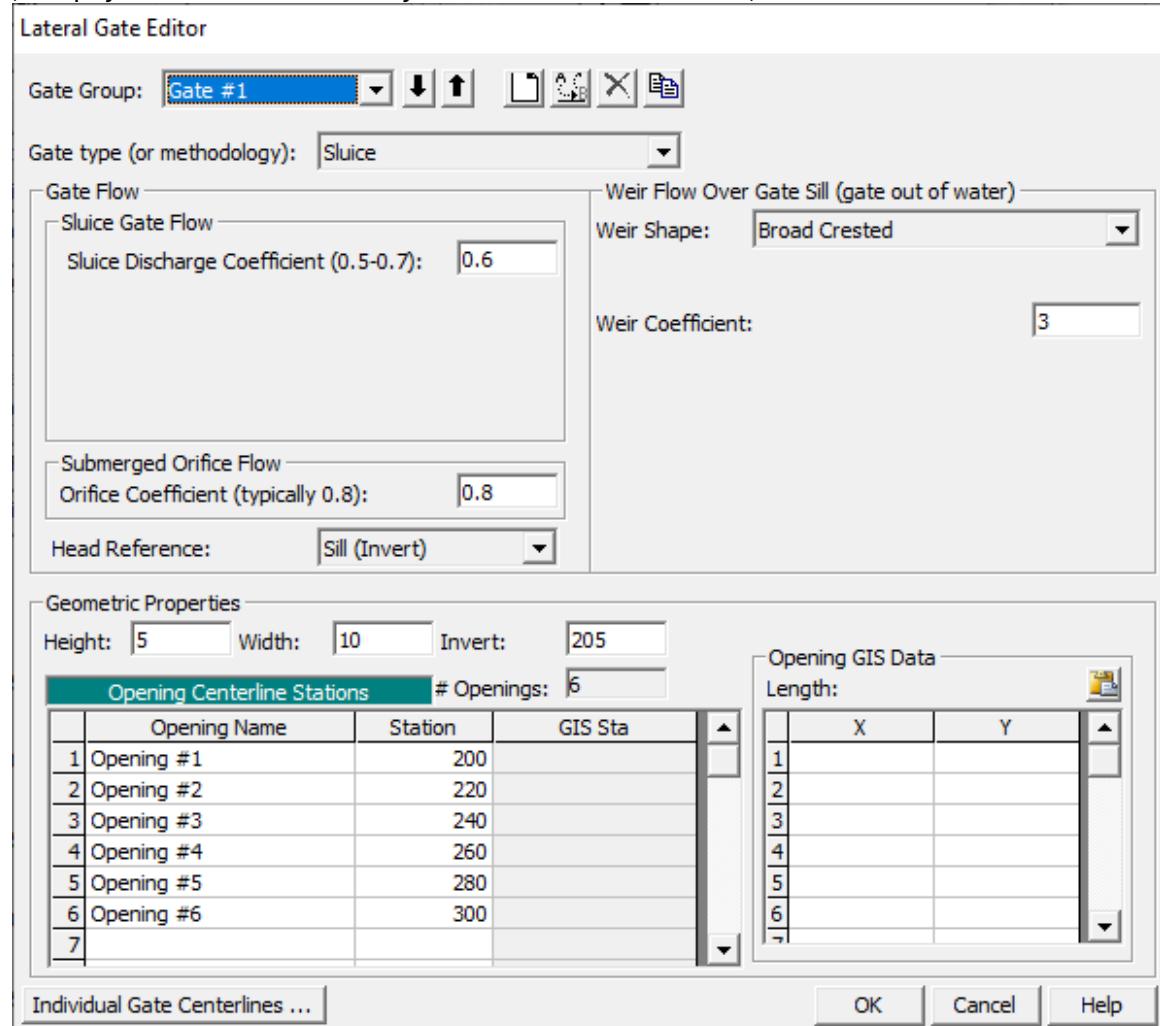
Weir Angle in Degrees (Optional) – This field can be used to enter an angle for the weir. If the weir is parallel to the stream, the angle is assumed to be zero. If the weir is angled inwards towards the center of the river, an angle (β) is required. This is used for channels that have a contraction, and weir flow is allowed to go over the contracted section. A diagram showing the angle (β) is shown below:



Gated Spillway Editor

In addition to uncontrolled overflow weirs, the user can add gated spillways (this is optional). To add gated spillways to the structure, press the Gate button on the Lateral Weir and Gated Spillway data

editor. Once this button is pressed, the lateral gated editor will appear as shown in the figure below (except yours will be blank until you have entered some data).

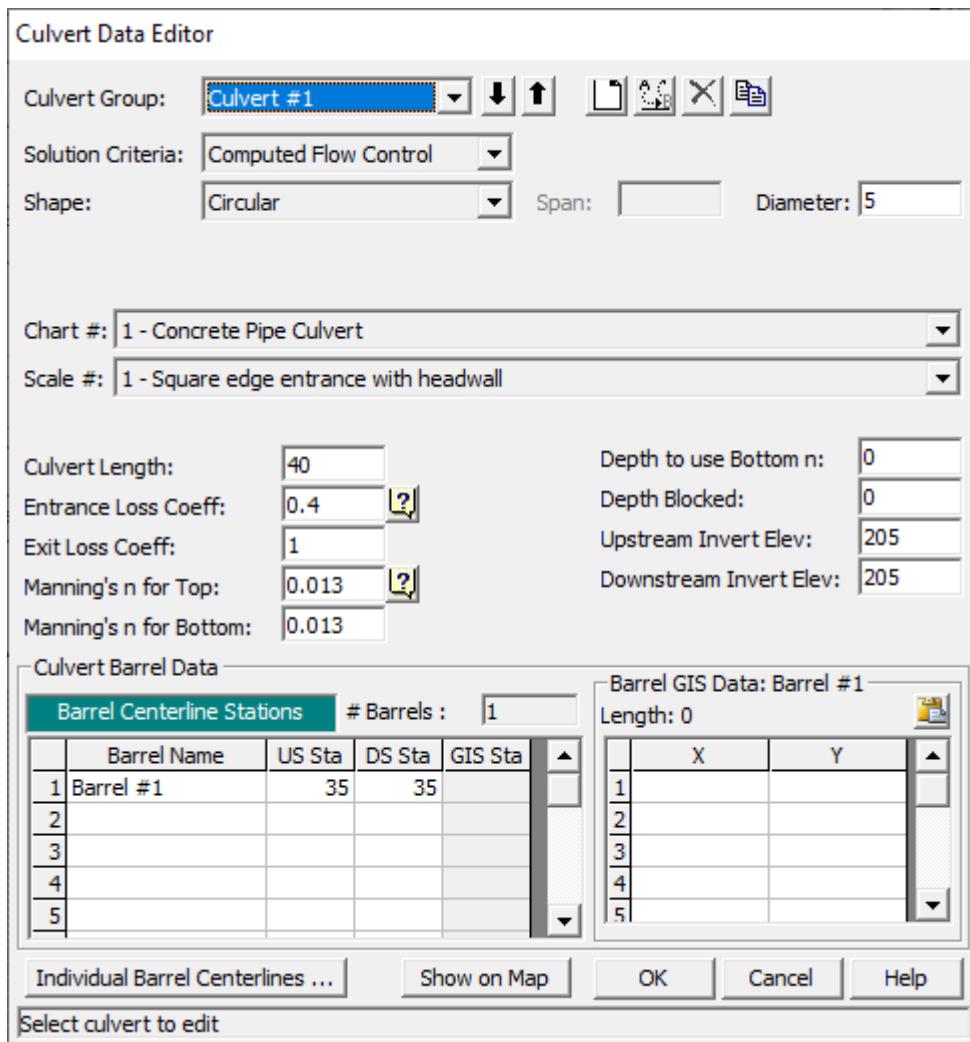


Lateral Gated Spillway Editor

The Gated Spillway editor is similar to the Culvert editor in concept. The user enters the physical description of the gates, as well as the required coefficients, in the Gated Spillway editor. The functionality of the gates is defined as part of the Unsteady flow Data, or the Steady Flow data (on a per profile basis). The data for modeling a gate in a lateral structure is the same as the data for modeling a gate in an inline structure. Please refer to the previous section on inline structures to get a detailed explanation of the data for the gate editor.

Culvert Editor

In addition to the lateral weir and gates, the user can also enter lateral culverts. To add culverts to the structure, press the **Culvert** button on the lower left side of the editor. When this option is selected the following window will appear.



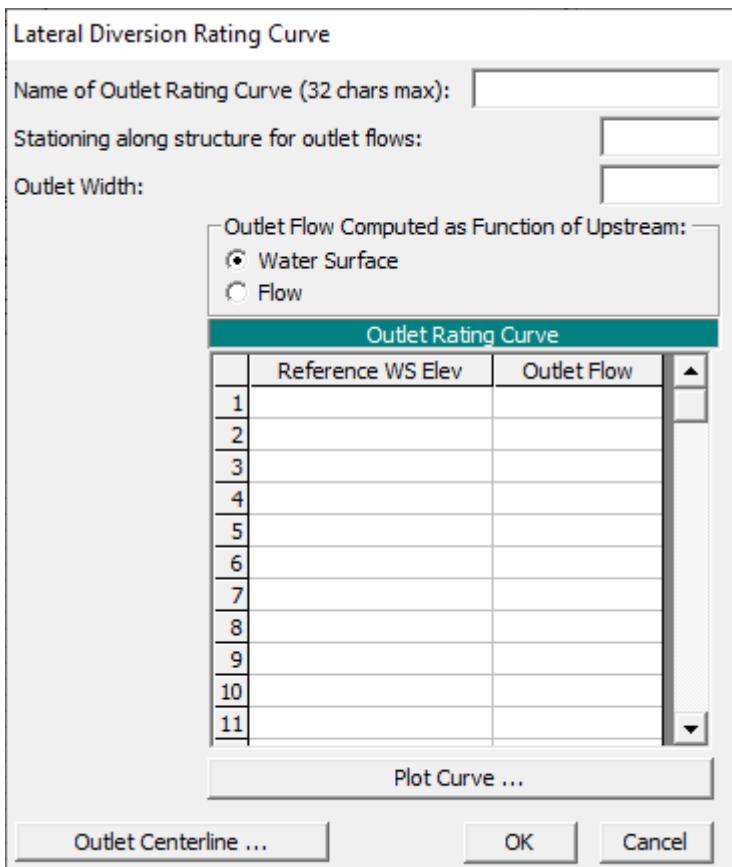
Culvert Editor for Lateral Culverts

The required culvert data is the same as for an inline culvert. To see an explanation of each field on the editor, review the information on culverts found earlier in this chapter. The only difference is that the centerline stationing of each culvert is based on the stationing entered in the Weir/Embankment editor.

Diversion Rating Curve Editor

Diversion rating curves are used to remove flow from a main river. The diversion, rating curve can be used in conjunction with a lateral weir, gated structures, and culverts, or it can be used alone.

To add a diversion, rating curve to the system, press the "**Diversion Rating Curve**" button on the left hand side of the Lateral Structure editor. When this button is pressed, the following editor will appear:



Lateral Rating Curve Editor

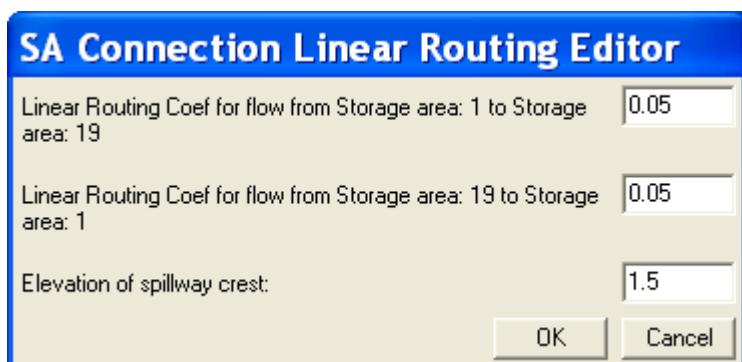
The user first selects the type of rating to be used: channel water surface versus diverted flow or channel flow versus diverted flow. Next, the distance between the location of the diversion and the cross section just upstream of the structure must be entered in order to locate the diversion. Finally, the user enters the actual rating curve. The curve is entered as the amount of flow leaving, versus the elevation of the water in the main river or flow in the main river. **NOTE:** This rating curve does not take into account any influence of the tailwater elevation in order to reduce the flow.

Outlet Time Series

This option allows the user to specify a name for an Outlet time series. Then a Flow Hydrograph can be specified for the Lateral Structure in the Unsteady Flow Data editor. The flow time series will be labeled in the output based on the user entered name for the Outlet Time series.

Linear Routing Option

The user can choose to use a linear routing option instead of entering structure information and having the program compute the flow from the structures. The linear routing option is selected by going to the Structure Type pull down and selecting "**Linear Routing**" from the list. This option uses a coefficient times the difference in available storage between the too and from connection. When this option is selected, a linear routing button will appear on the left side of the window. Selecting the linear routing button will bring up the following window:



Simple Spillway Data Editor

The equation used for the linear routing computations is the following:
 $Q = K \text{ (Available Storage)}/\text{hour}$

Where:

Q = Flow per hour

K = Linear Routing Coefficient (0.0 to 1.0)

Available Storage = ΔZ (Surface Area)

Surface Area = surface area of receiving storage area.

ΔZ is the difference between the headwater and tailwater water surface elevation on each side of the lateral structure. If both water surfaces are below the spillway crest elevation, then the flow is zero. If one water surface is above the spillway elevation and the other is not, then ΔZ is compute as the water surface above the spillway crest minus the spillway crest elevation.

The flow is computed in cfs per hour. If the user selected time step is not 1 hour, then the flow for the time step is compute by multiplying the flow by the ratio of the user entered time step divided by 1 hour.

As shown in the Linear Routing editor, the user must enter a **linear routing coefficient** for both the positive and negative flow directions. Additionally, the minimum **elevation of the spillway crest** must be entered. If both water surfaces go below the spillway crest, no flow is passed over the structure. Also, the user must enter the **HW Distance to Upstream XS**, which allows the program to figure out where the location of the lateral connection with respect to the upstream cross section. This location will be used for interpolating water surface elevations on the river side of the connection.

Storage Areas



Storage Areas are lake like regions in which water can be diverted into or from. Storage areas can be located at the beginning of a reach (as an upstream boundary to a reach), at the end of a reach (as a downstream boundary to a reach), or they can be located laterally to a reach. Storage areas can be connected to a river reach by using a lateral structure connection. Storage areas can be connected to another storage area by using a storage area connection. Storage area connections can consist of a weir and gated spillways; a weir and culverts; just a weir; or a linear routing option. An example river system with storage areas is shown in Figure 5.39.

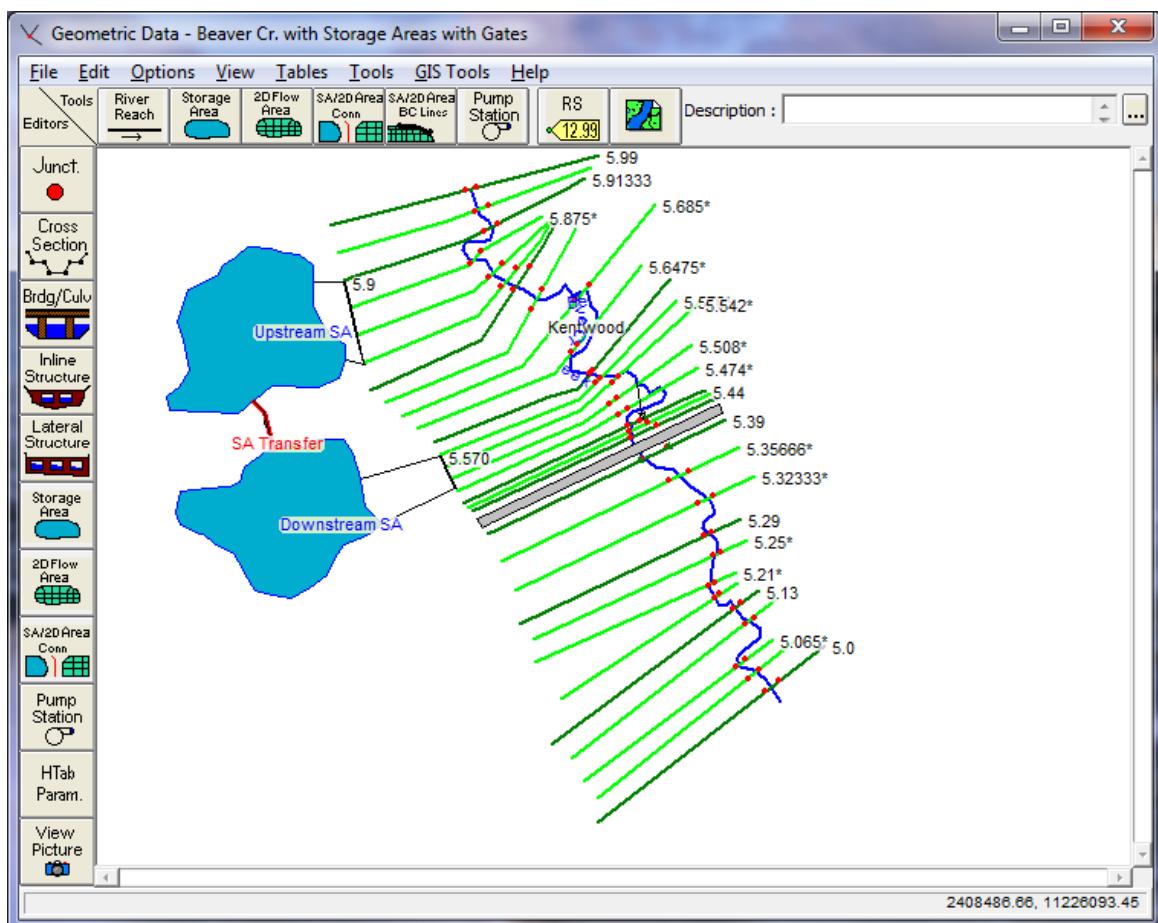


Figure 5 39 Example Schematic with Storage Areas

To add a storage area to the river system, first use the storage area drawing tool at the top of the geometric editor. Once the storage area, drawing tool is selected, the user single clicks the left mouse button to begin drawing the storage area. Additional points are added by moving the mouse and single clicking. The storage area will be represented as a polygon. To finish drawing the storage area, double click the left mouse button. The first and last point will then be connected, and the storage area will be filled in with a blue color. The user will then be prompted to enter a name for the storage area.

After the storage area is drawn and labeled, the user must enter data to describe the storage area. This is accomplished with the storage area editor, which is one of the buttons on the left side of the geometric editor. Press the storage area editor button and the following editor will appear:

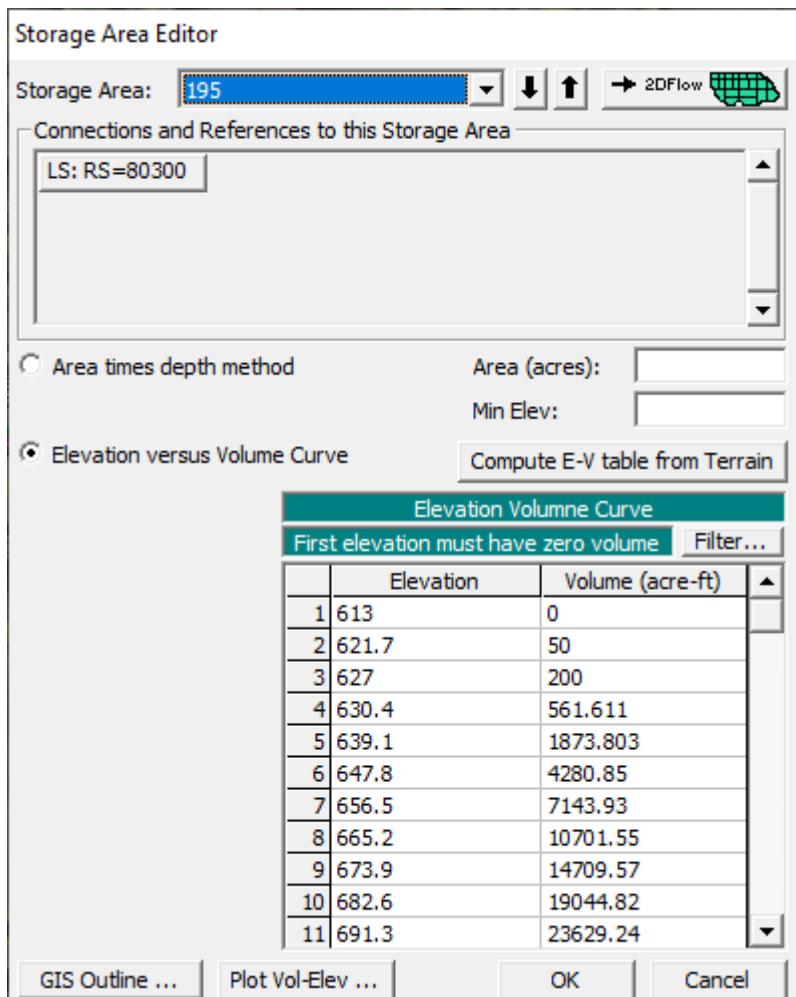


Figure 5 40 Storage Area Editor

As shown in the figure above, the user has two options for entering information about the volume of the storage area. The first option is a simple area times a depth option. The user enters the area of the storage, and a minimum elevation. The storage area is assumed to have the same area at all elevations, therefore the volume is simply the depth times the area. The second option is to enter an elevation versus volume relationship for the storage area. This option provides more detail and accuracy, and is the recommended method whenever possible. Also shown in the storage area editor are any connections or references to that particular storage area. Valid connections to a storage area are: lateral structures, storage area connections, and a cross section connected directly to a storage area.

Storage areas can be converted to two-dimensional flow areas by pressing the button labeled "-> 2DFlow". When this option is selected, the interface will now think that this area is a 2D Flow Area, and the user will then need to define the 2D computational mesh. See 2D Flow Areas below for more information on how to do this.

2D Flow Areas



Two Dimensional Flow Areas (2D Flow Areas) are regions of a model in which the flow through that region will be computed with the HEC-RAS two dimensional flow computation algorithms. 2D Flow Areas are defined by laying out a polygon that represents the outer boundary of the 2D Flow Area. Then the user must define the computational mesh.

2D Flow Area can be located at the beginning of a reach (as an upstream boundary to a reach), at the end of a reach (as a downstream boundary to a reach), or they can be located laterally to a reach. 2D Flow Areas can be connected to a river reach by using a lateral structure connection. 2D Flow Areas can be connected to another 2D Flow area or a Storage Area by using a SA/2D Area Connection (this is described later in this chapter).

The HEC-RAS 2D modeling capability uses a Finite-Volume solution scheme. This algorithm was developed to allow for the use of a structured or unstructured computational mesh. This means that the computational mesh can be a mixture of 3-sided, 4-sided, 5-sided, etc... computational cells (up to 8 sided cells). However, the user will most likely select a nominal grid resolution to use (e.g. 200 X 200 ft cells), and the automated tools within HEC-RAS will build the computational mesh.

Note: for a more detailed description of how to use the HEC-RAS two-dimensional modeling capabilities, please see the separate User's manual that comes with HEC-RAS called "**2D Modeling User's Manual**".

A 2D Flow Area, and computational mesh, is developed in the HEC-RAS Geometry editor by doing the following:

Draw a Polygon Boundary for the 2D Flow Area

The user must add a 2D flow area polygon to represent the boundary of the 2D area using the 2D flow area drawing tool in the Geometric Data editor (just as the user would create a Storage Area). The best way to do this in HEC-RAS is to first bring in terrain data and aerial imagery into HEC-RAS Mapper. Once you have terrain data and various Map Layers in RAS Mapper, they can be displayed as background images in the HEC-RAS Geometry editor. Additionally, the user may want to bring in a shapefile that represents the protected area, if they are working with a leveed system. The background images will assist the user in figuring out where to draw the 2D flow area boundaries in order to capture the tops of levees, floodwalls, and any high ground that will act as a barrier to flow.

Use the background mapping button on the HEC-RAS Geometry editor to turn on the terrain and other Map Layers, in order to visualize where the boundary of the 2D Flow Area should be drawn.

- ① The boundary between a 1D river reach and a 2D Flow Area should be high ground that separates the two.

For levees and roadways this is obviously the centerline of the levee and the roadway. However, when using a lateral structure to connect a main river to the floodplain (when there is no actual levee), try to find the high ground that separates the main river from the floodplain. Use this high ground as a guide for drawing the 2D boundary, as well as defining the Lateral Structure Station Elevation data.

To create the 2D Flow Area, use the 2D Flow Area tool (the button on the Geometric Editor Tools Bar labeled 2D Flow Area, highlighted in red on Figure 12). Begin by left-clicking to drop a point along the 2D Flow Area polygon boundary. Then continue to use the left mouse button to drop points in the 2D Flow Area boundary. As you run out of screen real-estate, right-click to re-center the screen. Double-click the left mouse button to finish creating the polygon. Once you have finished drawing the 2D area polygon by double clicking, the interface will ask you for a Name to identify the 2D Flow Area. Shown in Figure 5-41 is an example 2D Flow Area polygon for an area that is protected by a levee. The name given to the 2D Flow Area in this example is: "2D Interior Area".

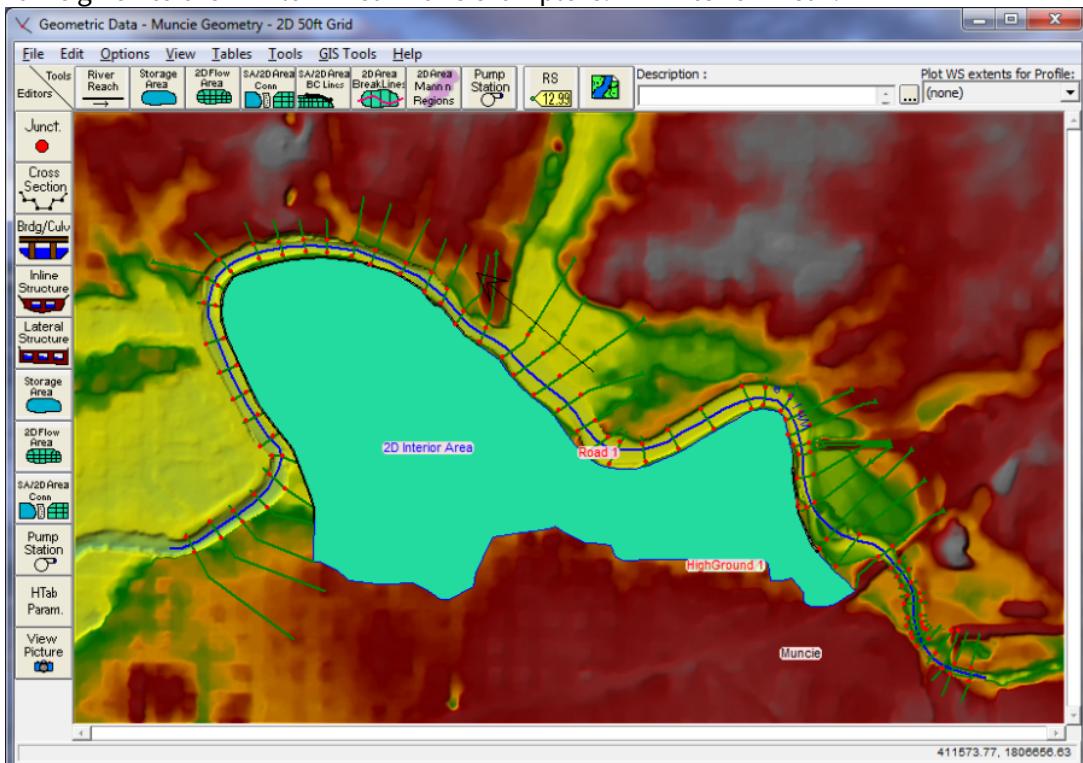


Figure 5 41. Example 2D Flow Area polygon.

Adding Break Lines inside of the 2D Flow Area

Before the computational mesh is created the user may want to add break lines to enforce the mesh generation tools to align the computational cell faces along the break lines. Break lines can also be added after the main computational mesh is formed, and the mesh can be regenerated just around that break line. In general, break lines should be added to any location that is a barrier to flow, or controls flow/direction.

Break lines can be imported from Shapefiles (GIS Tools/Breaklines Import from Shapefile); drawn by hand; or detailed coordinates for an existing breakline can be pasted into the break line coordinates table (GIS Tools/Breaklines Coordinates Table). To add break lines by hand into a 2D flow area, select the 2D Area Break Line tool (highlighted in Red in Figure 3-2), then left click on the geometry window to start a break line and to add additional points. Double click to end a break line. While drawing a breakline, you can right click to re-center the screen in order to have more area for drawing the breakline. Once a break line is drawn the software will ask you to enter a name for the break line. Add break lines along levees, roads, and any high ground that you want to align the mesh

faces along. Break lines can also be placed along the main channel banks in order to keep flow in the channel until it gets high enough to overtop any high ground berm along the main channel.

Creating the 2D Flow Area Computational Mesh

Select the 2D Flow Area editor button on the left panel of the Geometric Data editor (Under the Editors set of buttons on the left) to bring up the 2D Flow Area editor window:

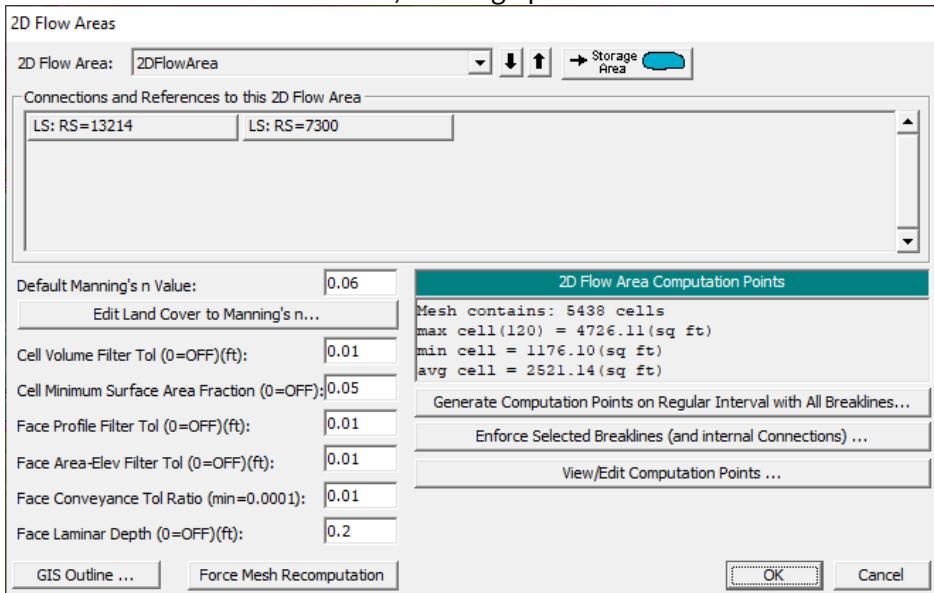


Figure 5 42. 2D Flow Area Mesh Generation Editor

The **2D Flow Area** editor allows the user to select a nominal grid size for the initial generation of the 2D flow area computational mesh. To use this editor, first select the button labeled **Generate Computational points on regular Interval** This will open a popup window that will allow the user to enter a nominal cell size. The editor requires the user to enter a **Computational Point Spacing** in terms of DX and DY (see Figure 3-5). This defines the spacing between the computational grid-cell centers. For example, if the user enters DX = 50, and DY = 50, they will get a computational mesh that has grids that are 50 x 50 everywhere, except around break lines and the outer boundary. Cells will get created around the 2D flow area boundary that are close to the area of the nominal grid-cell size you selected, but they will be irregular in shape.

Since the user can enter break lines, the mesh generation tools will automatically try to "snap" the cell faces to the breaklines. The cells formed around break lines may not always have cell faces that are aligned perfectly with the break lines. An additional option available is **Enforce Selected Breaklines**. The **Enforce Selected Breaklines** option will create cells that are aligned with the breaklines, which helps ensures that flow cannot go across that cells face until the water surface is higher than the terrain along that break line. When using the Enforce Selected Breaklines option, the software will create cells spaced along the breakline at the nominal cell size entered by the user. However, the user can enter a different cell spacing to be used for each breakline. This is accomplished by selecting **GIS Tools/Breaklines Cell Spacing Table**, and then entering a user defined cell spacing for each breakline.

The popup editor has an option to enter where the user would like the cell centers to start, in terms of an upper left X and an upper left Y coordinate. These Starting Point Offset fields are not required.

By default it will use the upper left corner of the polygon boundary that represents the 2D flow area. Use of the **Shift Generated Points** option allows the user to shift the origin of the grid cell centers, and therefore the location of the cell centers.

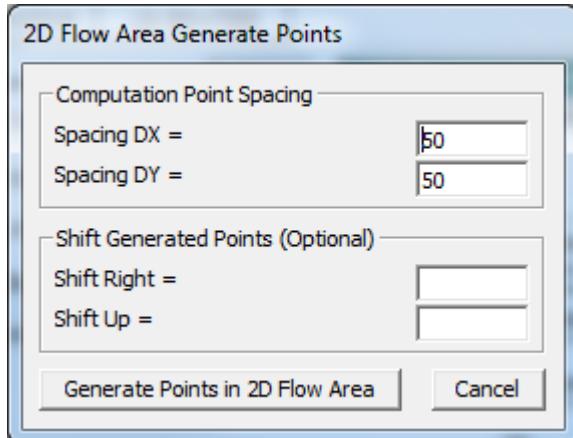


Figure 5 43. 2D Flow Area Computational Point Spacing Editor

After the Computational Point Spacing (DX and DY) has been entered, press the Generate Points in 2D flow area button. Pressing this button will cause the software to compute a series of X and Y coordinates for the cell centers. The user can view these points by pressing the View/Edit Computational Point's button, which brings the points up in a table. The user can cut and paste these into a spreadsheet, or edit them directly if desired (It is not envisioned that anyone will edit the points in this table or Excel, but the option is available).

- ① Warning: If there is an existing computational mesh and the "Generate Points in 2D Flow Area" option is used, all of the existing mesh points will be replaced with the newly generated points. Any hand editing that was done by the user will be lost.

There are five additional fields on the 2D Flow Areas editor (Figure 5-42) that are used during the 2D pre-processing. These fields are:

Default Manning's n Value: This field is used to enter a default Manning's n values that will be used for the Cell Faces in the 2D Flow Area. User's have the option of adding a spatially varying landuse classification versus Manning's n value table (and a corresponding Land Classification layer in RAS-Mapper), which can be used to override the base Manning's n values where polygons and roughness are defined. Even if a Land Use Classification versus Manning's n value table is defined, for any areas of the 2D Flow Area not covered by that layer, the base/default Manning's n value will be used for that portion of the 2D Flow Area.

Cell Volume Filter Tol: This tolerance is used to reduce the number of points in the 2D cell elevation volume curves that get developed in the 2D Pre-processor. Fewer points in the curve will speed up the computations, but reduce the accuracy of the elevation volume relationship. The default tolerance for filtering these points is 0.01 ft.

Face Profile Filter Tol: This filter tolerance is used to reduce the number of points that get extracted from the detailed terrain for each face of a 2D cell. The default is 0.01 ft.

Face Area-Elev Filter Tol: This filter tolerance is used to reduce the number of points in the cell face hydraulic property tables. Fewer points in the curves will speed up the computations, but reduce the accuracy of the face hydraulic property relationships. The default is 0.01 ft.

Face Conveyance Tol Ratio: This tolerance is used to figure out if more or less points are required at the lower end of the face property tables. It first computes conveyance at all of the elevations in the face property tables. It then computes the conveyance at an elevation half way between the points and compares this value to that obtained by using linear interpolation (based on the original points). If the computed value produces a conveyance that is within 2% (0.02) of the linear interpolation value, then no further points are needed between those two values. If linear interpolation would produce a value of conveyance that is more than 2% from the computed value at that elevation, then a new point is added to that table. This reduces the error in computing hydraulic properties, and therefore conveyance due to linear interpolation of the curves. A higher tolerance will result in fewer points in the hydraulic property tables of the cell faces, but less hydraulic accuracy for the flow movement across the faces. The default value is 0.02, which represents a 2% change.

Face Laminar Depth: This field is used to define the depth of water at which turbulent flow would transition to laminar flow for sheet flow flowing over a plane. The default is 0.2 feet.

Once a nominal grid size has been selected and a base Manning's n-value has been entered, the user should press the OK button to accept the data and close the editor. When the OK button is selected the software automatically creates the computational mesh and displays it in the Geometric Data Editor graphics window (See Figure 5-44).

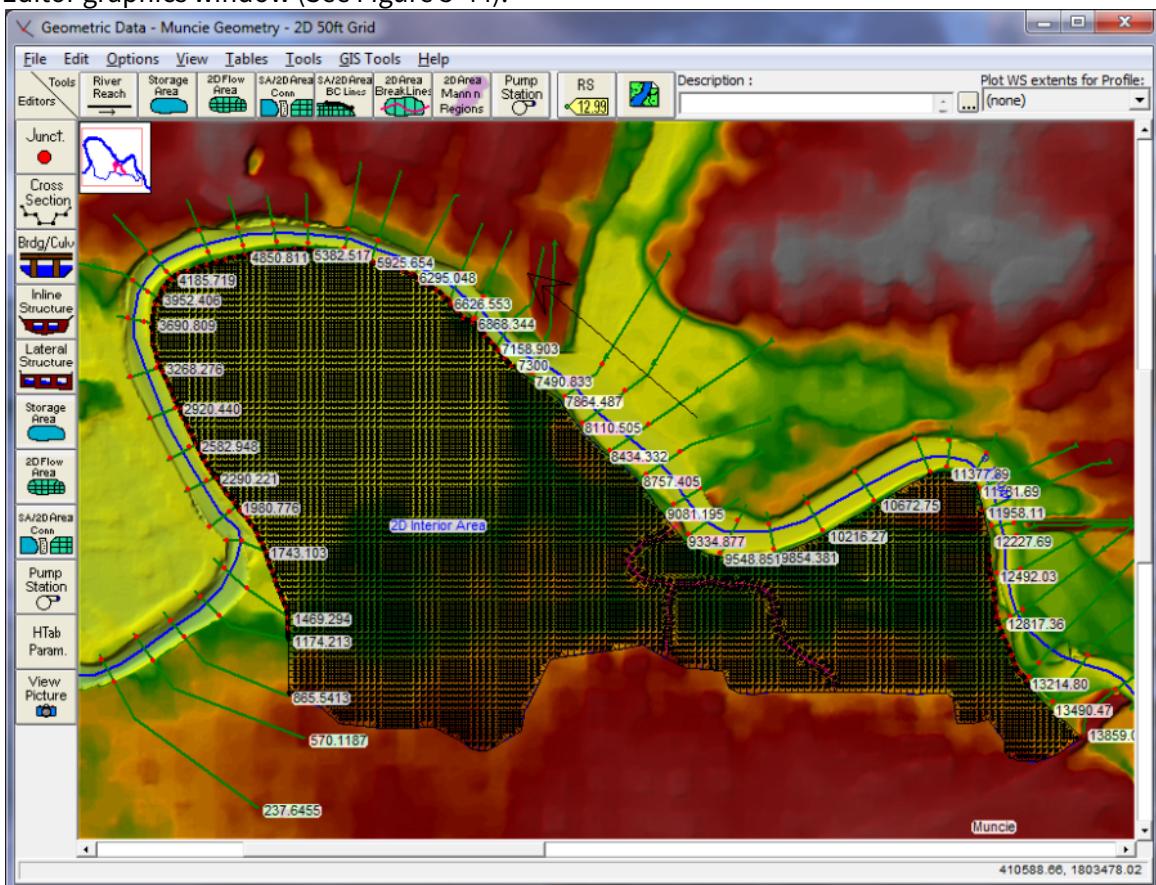


Figure 5 44. Example 2D computational mesh for an interior of a levee protected area.

As mentioned previously, cells around the 2D Flow Area boundary will be irregular in shape, in order to conform to the user entered polygon. The mesh generation tools utilize the irregular boundary, as well as try to ensure that no cell is smaller in area than the nominal cell size. The cells around the boundary will be equal to or larger than the nominal cell size; therefore, if a boundary cell is going to be smaller than the nominal cell size it gets combined with a neighbor cell. Additionally, breaklines can be placed inside of a 2D Flow Area in order to align the mesh to a geometric feature (levee, road, etc...) Shown in Figure 5-45, is a zoomed in view of a mesh with break lines on top of levees.

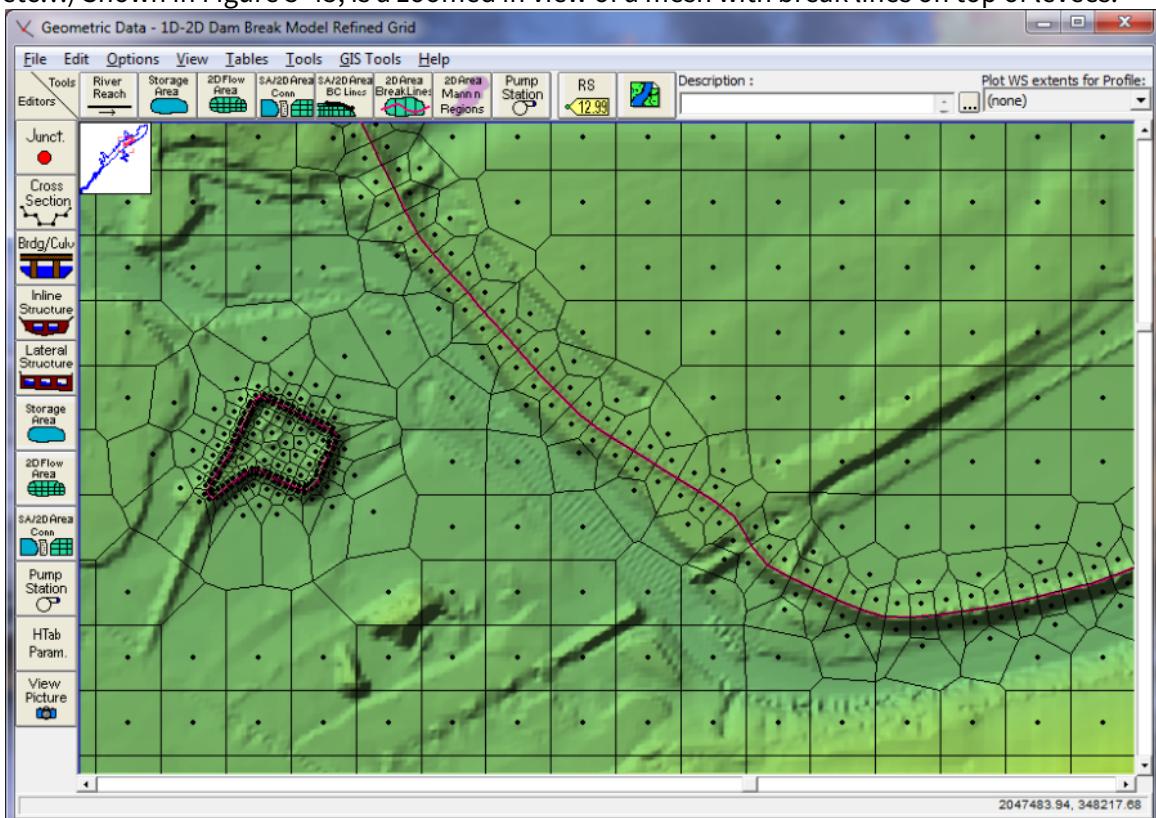


Figure 5 45. Zoomed in view of the 2D Flow Area computational mesh.

The HEC-RAS terminology for describing the computational mesh for 2D modeling begins with the 2D Flow Area. The 2D Flow Area defines the boundary for which 2D computations will occur. A computational mesh (or computational grid) is created within the 2D Flow Area.

Each cell within the computational mesh has the following three properties:

Cell Center: The computational center of the cell. This is where the water surface is computed for the cell.

Cell Faces: These are the cell boundary faces. Faces are generally straight lines, except along the outer boundary of the 2D Flow Area, in which a cell face can be a multi-point line.

Cell Face Points: The Cell Face Points (FP) are the ends of the cell faces. Later on in this document the Face Point (FP) numbers for the outer boundary of the 2D Flow Area will be used to hook the 2D Flow Area to a Lateral Structure.

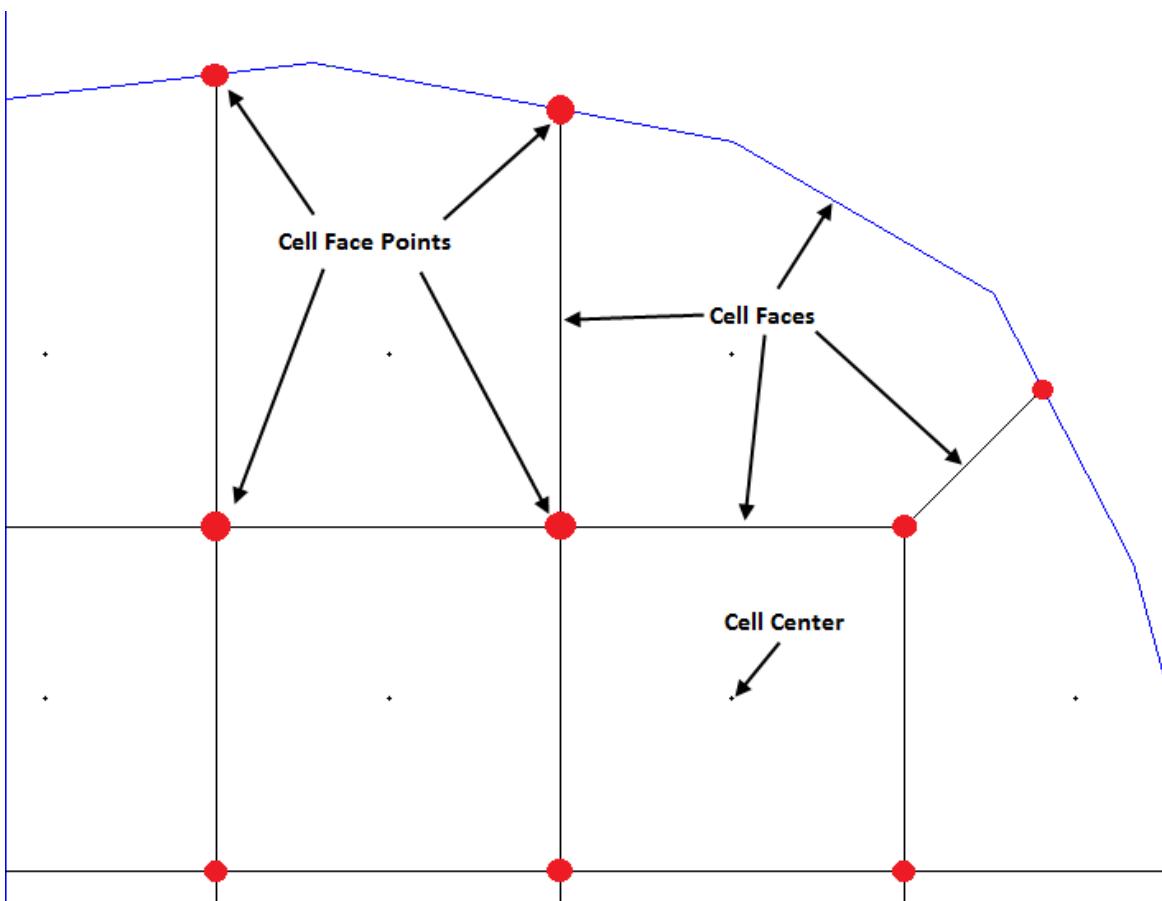


Figure 5 46. Description of HEC-RAS 2D modeling computational mesh terminology.

Edit/Modify the Computational Mesh

The computational mesh will control the movement of water through the 2D flow area. Specifically, one water surface elevation is calculated for each grid cell center at each time step. The computational cell faces control the flow movement from cell to cell. Within HEC-RAS, the underlying terrain and the computational mesh are preprocessed in order to develop detailed elevation–volume relationships for each cell, and also detailed hydraulic property curves for each cell face (elevation vs. wetted perimeter, area, and roughness). By creating hydraulic parameter tables from the underlying terrain, the net effect is that the details of the underlying terrain are still taken into account in the water storage and conveyance, regardless of the computational cell size. However, there are still limits to what cell size should be used, and important considerations for where smaller detailed cells are needed versus larger coarser cells.

In general, the cell size should be based on the slope of the water surface in a given area, as well as barriers to flow within the terrain. Where the water surface slope is flat and not changing rapidly, larger grid cell sizes are appropriate. Steeper slopes, and localized areas where the water surface elevation and slope change more rapidly will require smaller grid cells to capture those changes. Since flow movement is controlled by the computational cell faces, smaller cells may be required to define significant changes to geometry and rapid changes in flow dynamics.

The computational mesh can be edited/modified with the following tools: break lines; moving points; adding points, and removing points.

Break Lines

The user can add new break lines at any time. HEC-RAS allows the user to enter a new break line on top of an existing mesh and then regenerate the mesh around that break line, without changing the computational points of the mesh in other areas. The user can draw a new break line, then left click on the break line and select the option Enforce Break line in 2D Flow Area. Once this option is selected, new cells will be generated around the break line with cell faces that are aligned along the break line. Any existing cell centers that were already in the mesh in the area of the break line are removed first (within a buffer zone around the break line, based on the cell size used around the break line).

Additionally the user can control the size/spacing of cells along the break line. To control the cell spacing along a break line, right click on the break line and select the option Edit Break Line Cell Spacing. A window will appear allowing the user to enter a minimum and maximum cell spacing to be used when forming cells along that break line. The minimum cell spacing is used directly along the break line. The software will then increase the cell size around the break line, in order to provide a gradual cell size transition from the break line to the nominal cell size being used for the mesh. The user can enter a Maximum cell size if desired. If no maximum cell size is entered, the software automatically transitions the cells from the minimum cell size around the break line, to the default mesh cell size. To enforce the new cell spacing, the user must select the Enforce Break line in 2D Flow Area option, after entering the break line cell spacing. Break line cell spacing's are saved, such that if the mesh is regenerated, the user defined break line cell spacing will automatically be used. User can also bring up a table that will show all of the break lines and any user entered break line cell spacing values. To open this table, select GIS Tools, then Break Lines Cell Spacing Table. Once the table is open, users can add or change break line cell spacing values from the table. Then if the user regenerates the whole mesh, or just the area around a specific break lines, the new break line cell spacing will be used.

When creating a mesh around a break line, it may be desirable or even necessary to use smaller cells than the nominal cell size used in other areas of the mesh. However, transitions from a larger cell size immediately to a smaller cell size, may not produce the most accurate computational model. So it is better to transition cell sizes gradually. The HEC-RAS mesh generation tools allow the user to enter a minimum and a maximum cell spacing to use around break lines. The mesh generation tools will automatically transition from the smaller cell size right at the break line to the larger cell size away from the break line.

Hand Based Mesh Editing Tools

The hand editing mesh manipulation tools are available under the Edit menu of the HEC-RAS Geometric Data editor. If the user selects Edit then Move Points/Object, the user can select and move any cell center or points in the bounding polygon. If a cell center is moved, all of the neighboring cells will automatically change due to this movement. If the user selects Edit then Add Points, then wherever the user left-clicks within the 2D flow area, a new cell center is added, and the neighboring cells are changed (once the mesh is updated). The software creates a local mesh (Just the area visible on the screen, plus a buffer zone), such that while you are editing, just the local mesh will get updated. The entire mesh only updates once the user has turned off the editing feature, which saves computational time in creating the new mesh. If the user selects Edit then Remove Points, then any

click near a cell center will remove that cell's point, and all the neighboring cells will become larger to account for the removed cell.

The user may want to add points and move points in areas where more detail is needed. The user may also want to remove points in areas where less detail is needed. Because cells and cell faces are preprocessed into detailed hydraulic property tables, they represent the full details of the underlying terrain. In general, the user should be able to use larger grid cell sizes than what would be possible with a model that does not preprocess the cells and the cell faces using the underlying terrain. Many 2D models simply use a single flat elevation for the entire cell, and a single flat elevation for each cell face. These types of 2D models generally require very small computational cell sizes in order to model the details of the terrain.

HEC-RAS makes the computational mesh by following the Delaunay Triangulation technique and then constructing a Voronoi diagram (see Figure 5-47 below, taken from the Wikimedia Commons, a freely licensed media file repository):

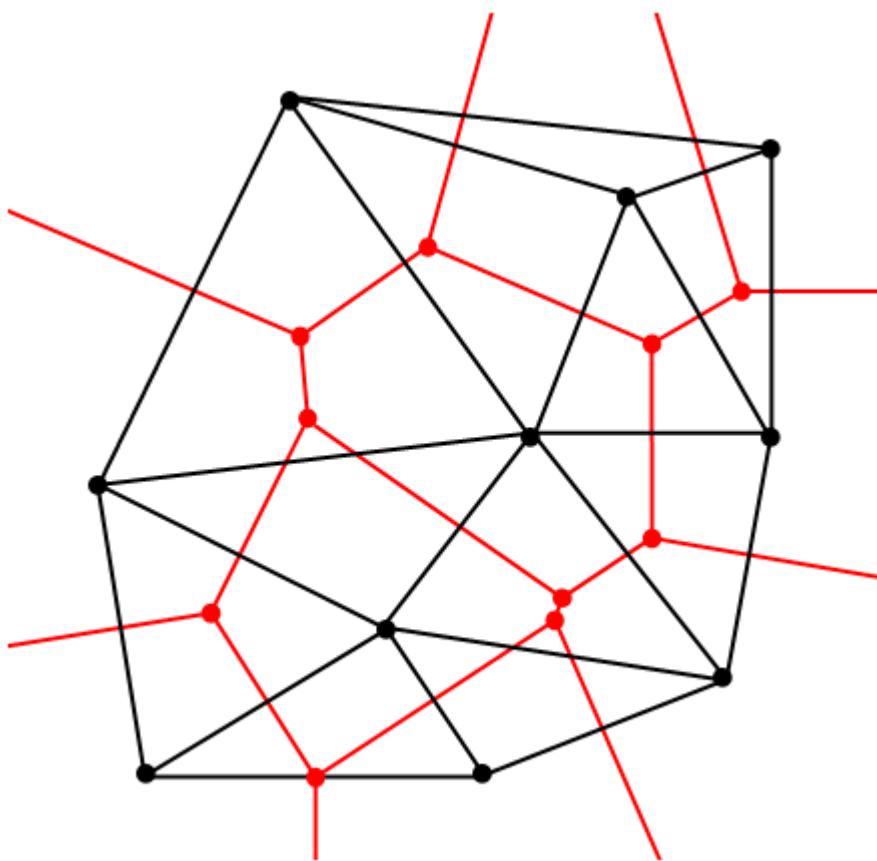


Figure 5-47. Delaunay - Voronoi diagram example.

The triangles (black) shown in Figure 18 are made by using the Delaunay Triangulation technique (http://en.wikipedia.org/wiki/Delaunay_triangulation). The cells (red) are then made by bisecting all of the triangle edges (Black edges), and then connecting the intersection of the red lines (Voronoi Diagram). This is analogous to the Thiessen Polygon method for attributing basin area to a specific rain gage.

You may want to add points and move points in areas where you need more detail. You may also want to remove points in areas where you know you need less detail. Because cells and cell faces are pre-processed into detailed hydraulic property tables, they represent the full details of the

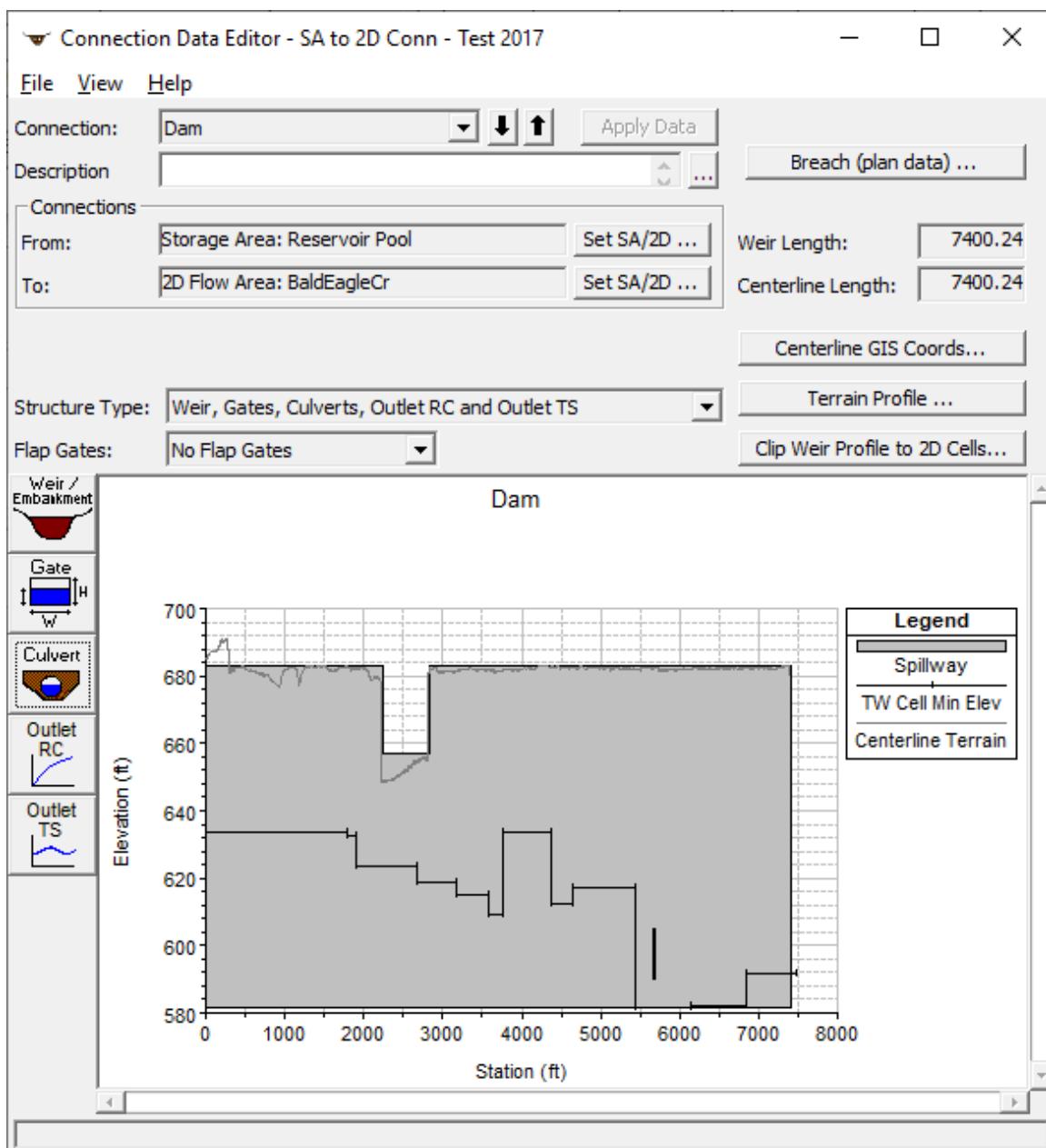
underlying terrain. In general, you should be able to get away with larger grid cell sizes than what you would be able to with a model that does not do this pre-processing of the cells and the cell faces using the underlying terrain. Many 2D models simply use a single flat elevation for the entire cell, and a single flat elevation for each cell face. These types of 2D models generally require very small computational cell sizes in order to model the details of the terrain.

Storage Area and 2D Flow Area Connections



Storage Area/2D Flow Area connections (SA/2D Area Conn) are used to link two storage areas together with a hydraulic structure, or two 2D Flow Areas, or a storage area to a 2D Flow Area. The SA/2D Area Conn tool can also be used to place a hydraulic structure in the middle of a 2D Flow Area in order to control how flow travels from one series of cells to another series of cells. The SA/2D Area Conn has three Structure Type options: 1) Weir, Gates, Culverts, Outlet RC and Outlet TS; 2) Linear Routing option (The Linear Routing option is for storage areas only, not 2D Flow Areas); or 3) Bridge (internal to a 2D Flow Area). To establish a hydraulic connection between two storage areas, 2D flow areas; or inside of a 2D Flow Area, press the "**SA/2D Area Conn**" button at the top of the geometric data window. Once the storage area connection drawing tool is invoked, the user simple presses the left mouse button one time to start drawing the centerline of the hydraulic structure. Continue left clicking to digitize the centerline of the hydraulic structure, then double click to end. This structure should be drawn from left to right while looking in the positive flow direction (i.e. downstream). If this structures is drawn between two storage areas, a storage area and a 2D flow area, or between two 2D flow areas, the user will need to define the **From** and **To** locations within the SA/2D Area Connection editor. If the structure is drawn completely inside of a single 2D Flow Area, then the To and From connections are automatically set to the 2D area.

Once a SA/2D Area Connection is drawn, the user must enter information describing the hydraulics of the connection. This is accomplished by pressing the **SA/2D Area Conn** editor button on the left hand side of the geometric data editor. When this button is pressed, the following window will appear:

*SA/2D Area Connection Editor*

As shown in the figure above, this example is for a hydraulic structure connecting a storage area to a 2D Flow Area. The user should first enter a description for the SA/2D area connection. Next the **From** and **To** connections must be set correctly. After that the user should select the **Structure Type** from the drop down box. As mentioned previously, the user has the choice of three different structure types: 1) Weir, Gates, Culverts, Outlet RC and Outlet TS; 2) Linear Routing option (The Linear Routing option is for storage areas only, not 2D Flow Areas); or 3) Bridge (internal to a 2D Flow Area). Once a structure type is selected, the window will place editor buttons specific to that type of structure onto the left side of the editor. In this example, the default structure type (Weir, Gates, Culverts, etc.) is shown on the window.

Note: The user also has the option to perform a breaching analysis of any SA/2D Flow Area connection. The breach data is stored in the currently opened **Plan**, however, the user can get to the breach editor by pressing the button labeled "Breach (plan data)" on this editor. Breach information for a SA/2D Flow Area is the same as for a Dam or a levee breaching analysis.

Weir Embankment Editor

The user must enter data for the **Weir/Embankment** editor at a minimum. The other editors (gates, culverts, rating curves, and time series outlet) are optional. In this example data was entered for weir/embankment and Culverts. When the weir/embankment editor is selected, the window shown below will appear.

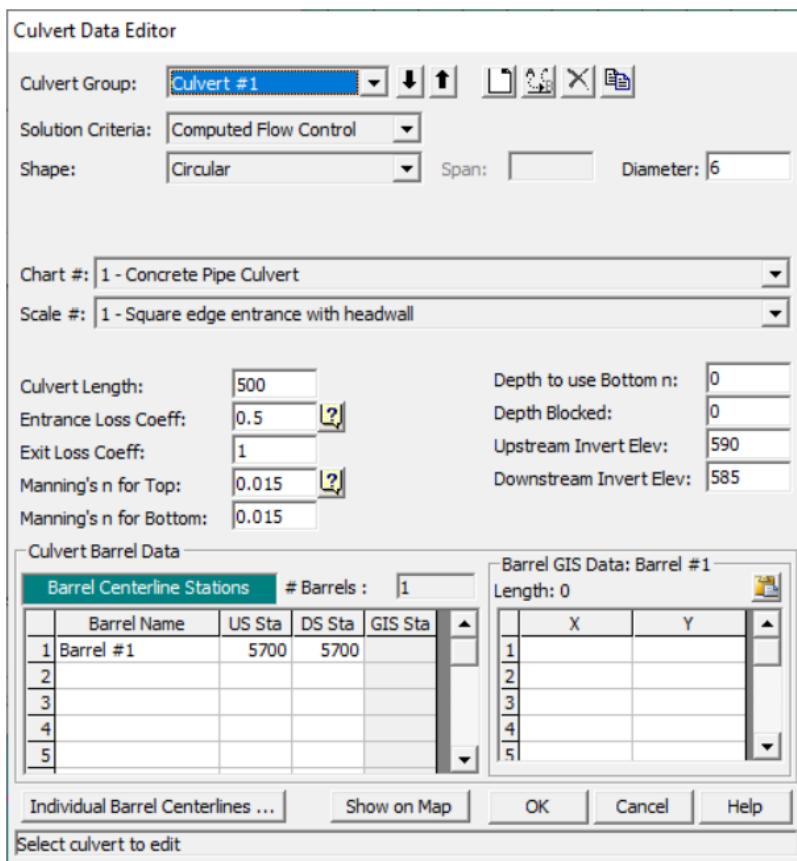
Station	Elevation
1	575.07
2	575.45
3	574.83
4	573.81
5	574.98
6	574.67
7	573.77
8	575.03
9	574.63
10	575.59
11	575.68
12	574.89
13	575.42
14	574.98
15	575.14
16	574.07
17	574.8
18	573.2
19	574.75
20	574.22
21	575.07
22	575.07

Weir/Embankment Editor for Storage Area Connections

To complete the data for the weir/embankment, the user enters a **Weir Width** (used only for drawing the schematic); a **Weir Coefficient** (used in the weir flow calculations); a **Weir Crest Shape** (used to assist in the calculation of the weir coefficient, as well as defining submergence criteria); and the **Station/Elevation Points** that describe the top of the weir/embankment profile. The weir/embankment can have up to 500 points to describe the profile. The program will use all of the information entered by the user for calculating weir flow between the two storage areas. After all of the data is entered, simply press the **OK** button to have the data accepted by the program.

Culvert Data Editor

When the culvert editor button is selected, the following window will appear.

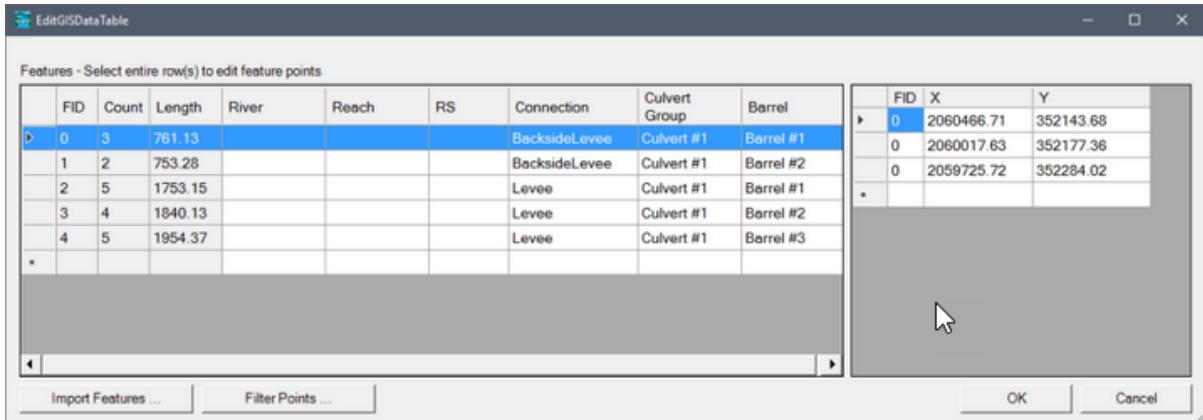


The culvert editor shown above has exactly the same information as the normal culvert editor used at a 1D river crossing. For detailed information about this editor, please review the Bridge/Culverts section found earlier in this document.

If the user is connecting Culverts to 2D Flow Areas, it may be advantageous to define coordinates for each of the culvert barrels. This will allow you to connect the ends of a barrel to a specific cell. In the **Culvert Data Editor** there is a button on the lower left portion of the editor labeled "**Individual Barrel Centerlines**" (**Error! Reference source not found.**). This new button opens the **Edit GIS Data Table** editor (**Error! Reference source not found.**), for entering the X and Y coordinates for the centerline of each culvert barrel added to the model, which allow users to view the barrels spatially. **[Note:** Separate centerlines must be added even for identical barrels within the same culvert group, and the barrels may also be connected to different cells.]

All Culvert centerlines (as well as gates, rating curves, and flow time series outlets), must be drawn from upstream to downstream. Keep in mind that is how the original centerline of the SA/2D Area Connection is drawn which defines upstream and downstream. Therefore, when users draw the centerline for the SA/2D Area Connection, it is drawn from left to right looking in the downstream direction. Based on that convention, when the centerlines for the hydraulic outlets (culverts, gates, rating curves, etc.) are drawn, yet again the centerlines must be drawn from the upstream side of the structure to the downstream side of the structure. For the example provided in **Error! Reference source not found.**, the structure being used to model the levee was drawn from the south end of the levee to the north end of the levee. Therefore, the culverts were drawn from the right hand side of the structure (head water) to the left hand side of the structure (tailwater).

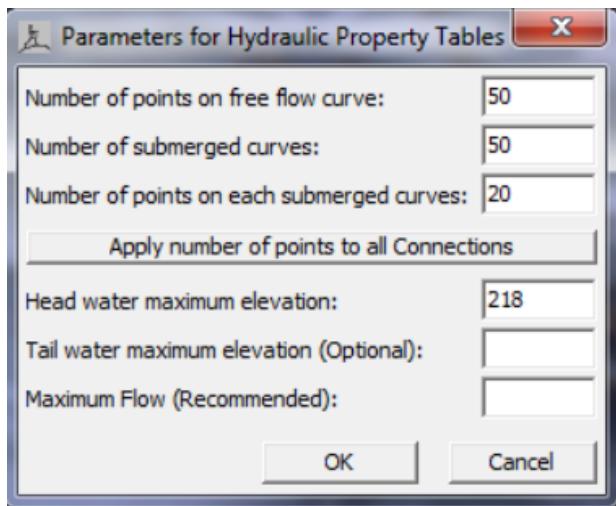
If the user presses the button labeled "**Individual Barrel Centerlines**" from the **Culvert Data Editor**, the **EditGISDataTable** centerline coordinate editor opens.



As shown in the figure below, The Feature Centerline Table contains the centerline X and Y coordinate data for all of the structures in the model. When the **Edit GIS Data Table** editor is opened, it will highlight the culvert that was open and selected in the **Culvert Data Editor**. To use this editor, from the Features table (located on the left of the editor), select a single culvert barrel (highlight it), and then past in the X and Y coordinates for the barrel into the data table (on the right hand side of the editor). **Hint:** The easiest way to define the culvert barrel X and Y centerline coordinates is to go back to the **Geometric Data** editor, hold down the **Ctrl** key, and digitize the culvert barrel centerline from the headwater side of the structure to the tailwater side of the structure. This digitized line can be copied to the clipboard from the **Measure Line** editor that pops up once digitizing the line is complete. Once all of the barrel coordinates have been entered, close all of the SA/2D Area Connection windows, and the digitized culvert(s) will appear in the **Geometric Data** editor window. Additionally, there is an option to import the culvert X and Y centerline coordinates at the bottom of the editor.

Note: In HEC-RAS, the process for adding centerline X and Y coordinates for individual hydraulic outlets (gates, rating curves and flow time series, etc.) is exactly the same as described above for culverts.

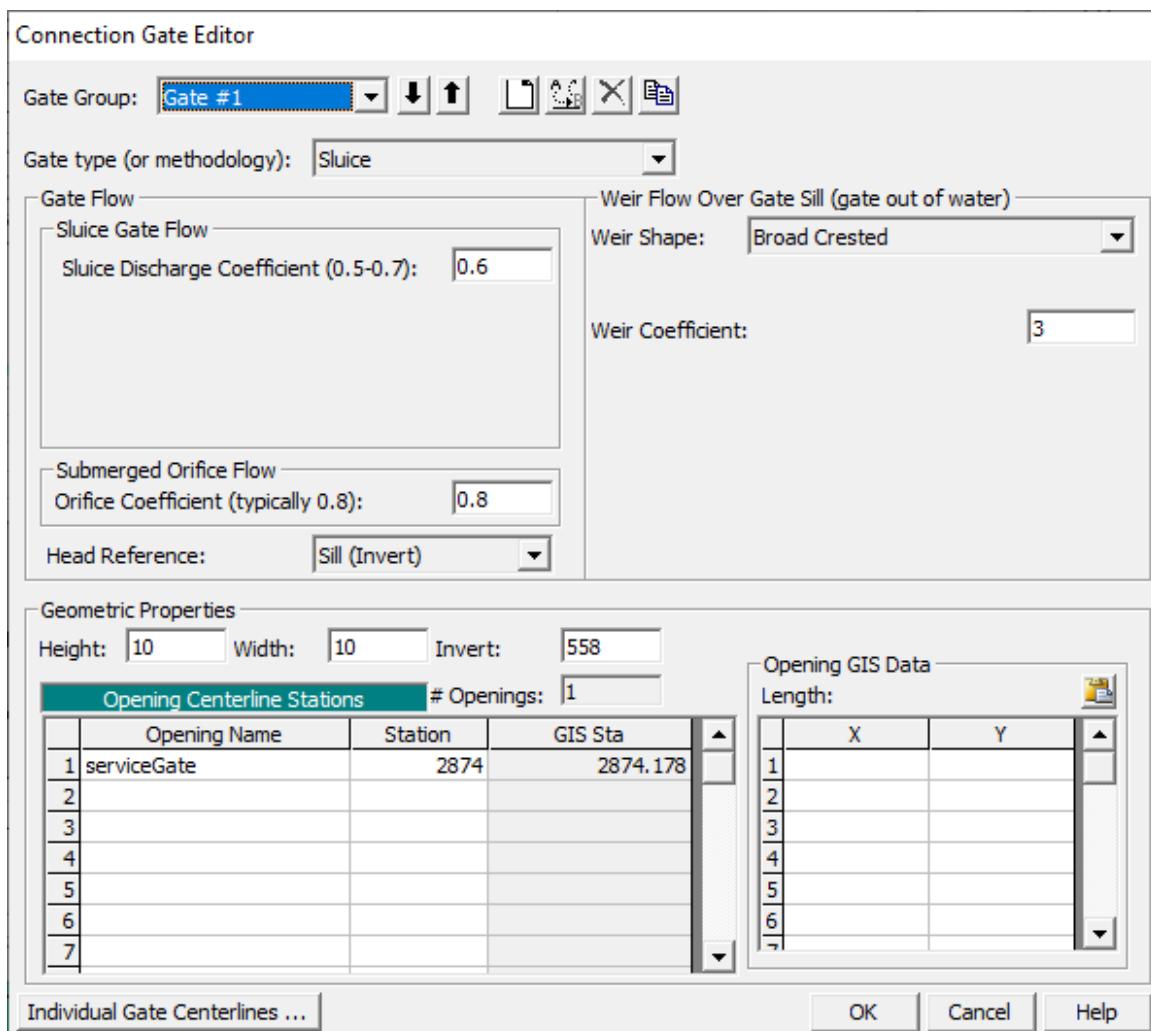
If the user is connecting a storage area to another storage area, an additional button will appear on the editor labeled "**Htab Param**". This editor is used to define the limits of the family of rating curves that will be developed for the storage area to storage area connection. When this button is pressed, the following editor will appear:



As shown in figure above, the user must enter a number of points for the free flow curve (default is 50, max 80); number of submerged curves (default is 50, max 60); number of points on the submerged curves (default is 20, max 50); and head water maximum elevation. Additionally, the user can enter a tailwater maximum elevation and a maximum flow rate. All of these parameters are used to define the limits of the family of rating curves that get created when the geometric pre-processor runs.

Gate Data Editor

If the user has decided to use the **Gates** option, pressing this button will bring up the window shown in the figure below. This editor is the same gate editor that is used for inline and lateral gated spillways. For information about this editor, please review the sections on inline gated spillways found earlier in this chapter.

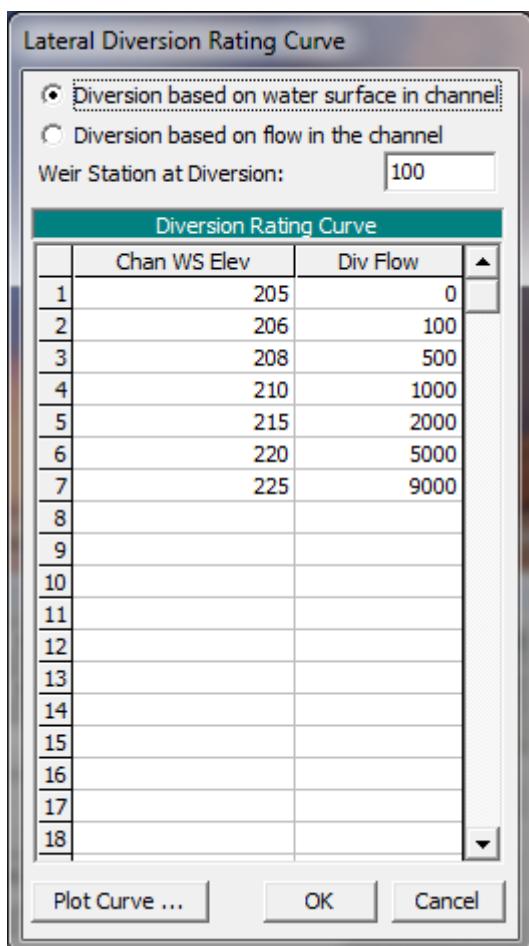


Gated Spillway Editor for Storage Area Connections

Diversion Rating Curve

Diversion rating curves are used to remove flow from a main river. The diversion, rating curve can be used in conjunction with a lateral weir, gated structures, and culverts, or it can be used alone.

To add a diversion, rating curve to the system, press the "**Diversion Rating Curve**" button on the left hand side of the Lateral Structure editor, shown below.



Lateral Rating Curve Editor

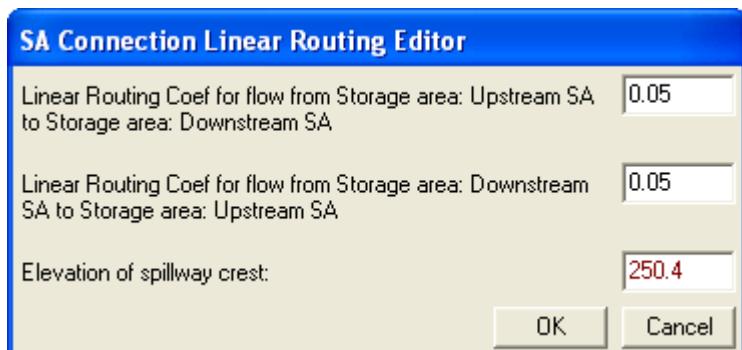
The user first selects the type of rating to be used: channel water surface versus diverted flow or channel flow versus diverted flow. Next, the distance between the location of the diversion and the cross section just upstream of the structure must be entered in order to locate the diversion. Finally, the user enters the actual rating curve. The curve is entered as the amount of flow leaving, versus the elevation of the water in the main river or flow in the main river. **NOTE:** This rating curve does not take into account any influence of the tailwater elevation in order to reduce the flow.

Outlet Time Series

This option allows the user to specify a name for an Outlet time series. Then a Flow Hydrograph can be specified for the Lateral Structure in the Unsteady Flow Data editor. The flow time series will be labeled in the output based on the user entered name for the Outlet Time series.

Linear Routing Option

The final option for connecting a Storage Area to another Storage Area is to model the connection as a **Linear Routing Method**. This option uses a coefficient times the difference in available storage between the two storage areas, divided by the time step.



Simple Spillway Data Editor

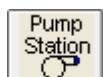
As shown in the Linear Routing editor, the user must enter a linear routing coefficient for both the positive and negative flow directions. Additionally, the minimum elevation of the spillway crest must be entered. If both water surfaces go below the spillway crest, no flow is passed between the storage areas. The Linear routing option can only be used to connect two storage areas. It cannot be used when one of the connections is a 2D Flow Area.

2D Flow Area and Storage Area External Boundary Conditions



User can define boundary condition location lines spatially along 2D Flow Areas and storage areas. This is accomplished using the **SA/2D Area BC Lines** drawing tool on the Geometric Data editor Tools button bar. To use this option, select the **SA/2D Area Conn** tool button at the top of the geometric data editor, then draw a line along the outer boundary of the 2D flow Area (or storage area) where you want the boundary condition to be located. Once the line is drawn the interface will ask you to enter a unique name for the boundary condition line. After you have drawn all the boundary condition lines you want along 2D flow Areas and Storage areas, save the geometry data. Then you can go into the Unsteady Flow Data editor and define the boundary condition types/data for each of these boundary condition lines.

Pump Stations



A pump station can be used to pump water between two storage areas, a storage area and a river reach, between two river reaches, a river reach and a 2D Flow Area, a Storage Area and a 2D Flow Area, between two 2D Flow Areas, or from one cell to another cell within the same 2D Flow Area. Each pump station can have up to ten different pump groups, and each pump group can have up to twenty identical pumps. To add a pump station to the system, select the Pump Station drawing tool at the top of the geometric data editor. When this button is pressed, move the mouse to the location that represents where the pump station will be located, and click the left mouse button. An editor will pop up asking you to enter a name for the Pump Station. This will establish a pump station location and icon.

Once a pump station is added to the system, the user must edit the pump station and fill in the required data. To bring up the pump station editor, select the pump station editor button on the left hand side of the geometric data editor, or move the mouse over the pump station icon on the schematic, press down on the left mouse button, then select **Edit Pump Station**. When the Pump Station editor is selected, the following window will appear:

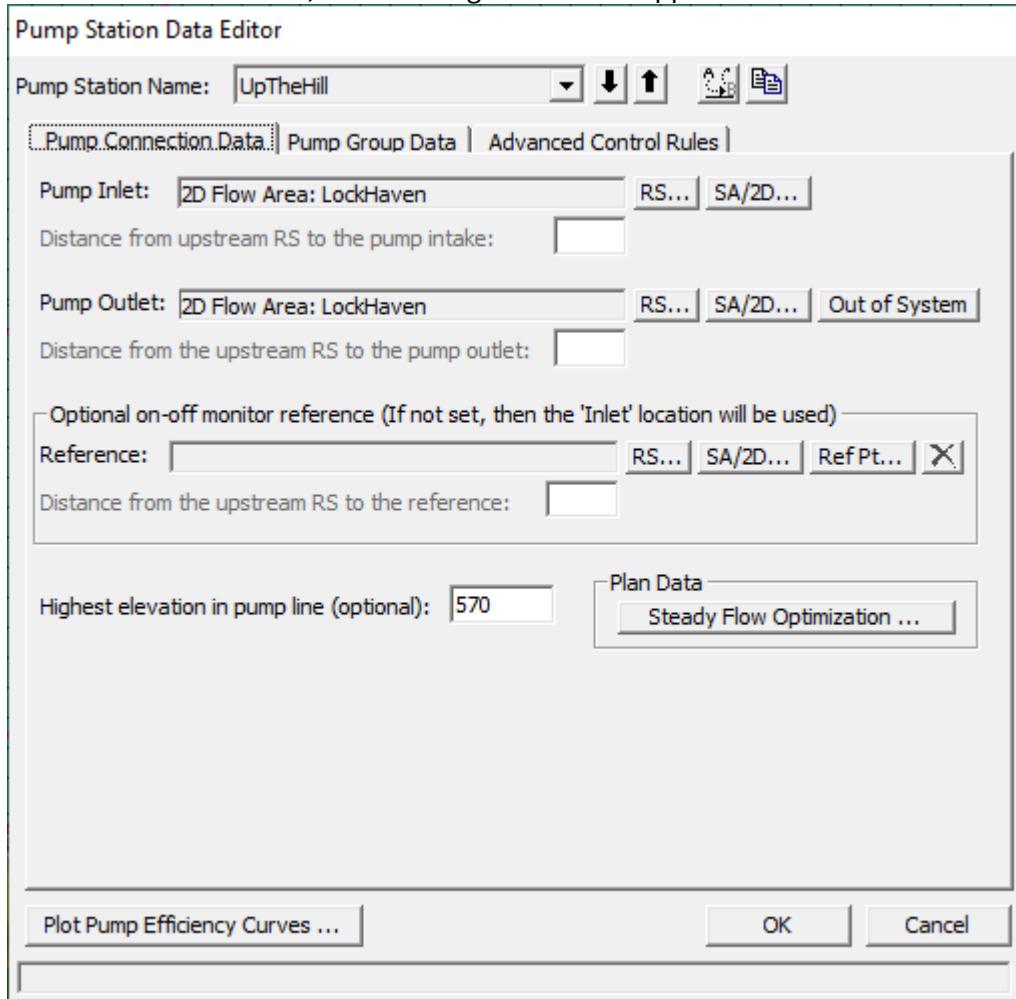


Figure 5 56 Pump Station Editor with Pump Connection Data

As shown in Figure 5 56, there are three tabs on the pump station editor, the first is for the pump connection data, the second is for the pump group data, and the third is for applying advanced rule controls over the pump station. The Pump Connection Data contains the following data:

Rename Pump Station: This option allows the user to rename the pump station to something other than the default.

Pump Inlet: This is the location of where the pump station is pumping from. This can be either a storage area, 2D Flow Area, or a river station from a river reach. The **Set RS** button allows the user to connect a pump from a river station of a reach, the **Set SA/2D** button allows the pump to be connected from a Storage Area or a 2D Flow Area.

Pump Outlet: This is the location of where the pump station is pumping to. This can be either a storage area or 2D Flow Area (use Set SA/2D button) or a river station from a river reach (use Set RS button).

Optional On-Off Monitor Reference: By default the program uses the "Pump Inlet" location to determine when the pump should turn on or off. However, the user has the option to set a different location to be used as the monitor point for determining whether the pump should be turned on or off. This optional monitor location can be a storage area, 2D Flow Area (if you are using a 2D Flow Area, you must add a reference point into the geometry data within that 2D flow Area); or a river station within a river reach.

Highest elevation in pump line: This option allows the user to enter an elevation to be used as the highest elevation in the pump line. One example of where this may be useful is if a pump station was being used to pump water over top of a levee. In this situation, the too and from water surface elevations does not completely quantify the required head to pump the water over the levee. So it is necessary to enter the elevation of the highest point in the pump line (top of the levee) in order to accurately compute the flow going through the pump.

Steady Flow Optimization: This option is for steady flow modeling only. If water is being pumped from or to a river reach, the amount of flow going into or out of the reach should be accounted for when computing the water surface profiles. However, the water surface profiles will affect the computation of the amount of flow through the pumps. Therefore, to calculate this accurately, the pump flow and water surface profiles must be calculated iteratively until a balance is found between the river flows and the pump flows. This optimization feature is not done automatically by the steady flow program, however, the user can have the program do this by selecting **Steady flow optimization**. When this option is selected, a window will appear allowing the user to turn the pump flow optimization on.

In addition to the pump connection data the user must fill out the pump group data. Select the **Pump Group Data** tab and the editor will look like the following (Figure 5-57):

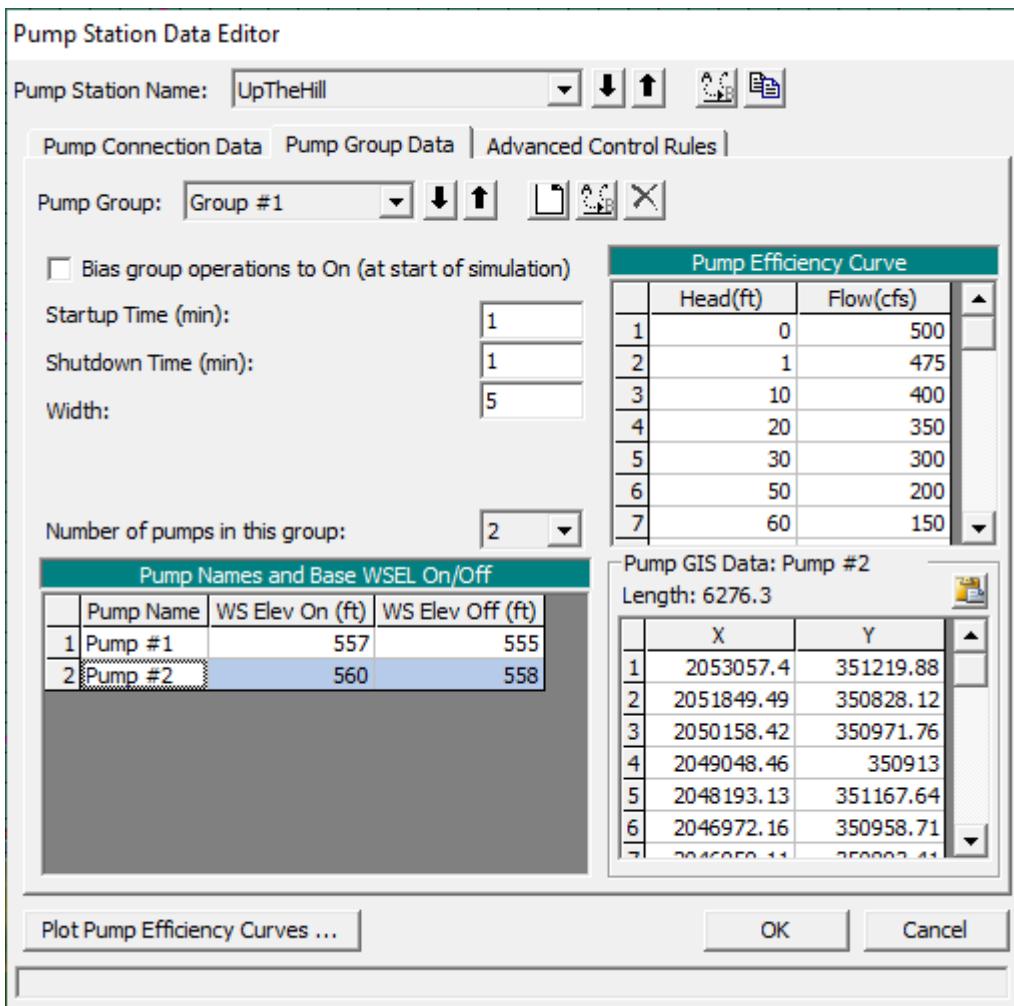


Figure 5 57 Pump Station Editor with Pump Group Data

As shown in Figure 5-57, the pump group data consists of the following:

Pump Group Name: By default the first pump group is called "Group #1", and the second would be "Pump Group #2", etc. The user has the option to rename any pump group to whatever they would like. This is done by pressing the "Rename Group" button.

Add Group: This button is used to add another pump group. If you have pumps that have different flow capacities and use different pump efficiency curves, they must be entered as a separate pump group.

Rename Group: This allows you to rename a pump group.

Delete Group: This button is used to delete the current pump group.

Bias group operations to on (Steady Flow Only): This option is only relevant for a steady flow run. When this option is selected, and a particular water surface profile is between the on and off elevation for a pump, the program will assume the pump is turned on. If this option is not checked, then the program will assume the pump is off when the water surface is between the on and off elevations.

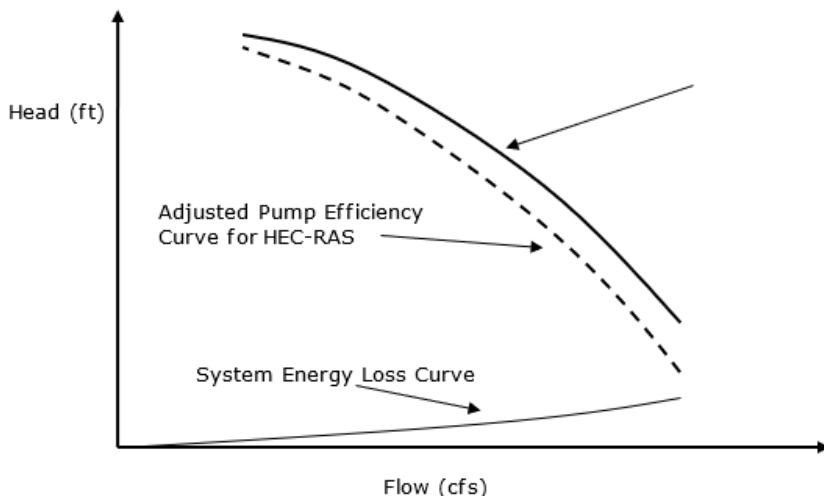
Startup (min): This option is used for unsteady flow only. When a pump is triggered to turn on, the default operation is that the pump turns on instantly and starts pumping to full capacity the very next time step. This option allows the user to enter a start up time in which the pumps will transition from zero flow to full capacity over the user entered time step in minutes. This option is very useful to prevent the unsteady flow computations from going unstable when to large of a flow change is experienced from a pump turning on.

Shutdown (min): This option is used for unsteady flow only. When a pump is triggered to turn off, the default operation is that the pump turns off instantly and stops pumping the very next time step. This option allows the user to enter a shut down time in which the pumps will transition from full capacity to zero flow over the user entered time step in minutes. This option is very useful to prevent the unsteady flow computations from going unstable when to large of a flow change is experienced from a pump turning off so abruptly.

Pump Width: This field is only used for drawing the width of the pump line onto the geometric data window.

Number of Pumps in Group: This field is used to enter the number of identical pumps in the current pump group. Identical pumps must use the same pump efficiency curve but can have different on and off trigger elevations.

Pump Efficiency Curve: This table is used to enter the pump efficiency curve, which is a table of static heads versus flow rates. The head represents the total head in the system, which is normally the difference in the water surface elevations between the from and the to location. **Note: The entered flow is the pump rate capacity at that particular head. In HEC-RAS, the flows entered for a given head difference, must already account for all energy losses in the pump line (friction, bends, junctions, etc...).** Do not enter the rated pump curve from the manufacturer, that curve does not account for losses in the pump line. An example of how to compute a pump efficiency curve is shown in Figure 5-58 below. As shown in Figure 5-58, the user must compute all energy losses in the system, between the two static pools. The energy losses in the line are subtracted from the manufacturer pump efficiency curve to get the curve for use in HEC-RAS. The pump efficiency curve can be plotted for visual inspection by pressing the **Plot Pump Curves** button at the bottom of the window.



Pump Efficiency Curve for HEC-RAS

Pump Operations: This table is used to define the trigger elevations for when the pumps will turn on and off. The monitor location for triggering a pump on or off is by default the from location, unless otherwise specified in the Optional On-Off Reference field. In general, the pump on elevation must be higher than the pump off elevation. Trigger elevations must be specified for all of the pumps. If the user puts the pump off elevation higher than the pump on elevation, then the pump turns on when the water surface goes below the on elevation, and the pump remains on until the water surface gets higher than the pump off elevation. This would be for example, pumping water up to a storage tank. When the pump off elevation is lower than the pump on elevation (typical way of using it), the pump turns on when it goes above the on elevation, and the pump turns off when it goes below the off elevation. This is the typical use of the pumps for interior ponding areas.

The bottom half of the window shows a table with all the individual pumps in the group. The table contains the following:

Pump Name: This field contains the name of each of the individual pumps. Pumps are automatically named "Pump #1", then "Pump #2", etc. The user can double click in the Pump Name field and change these names to be more specific to the location.

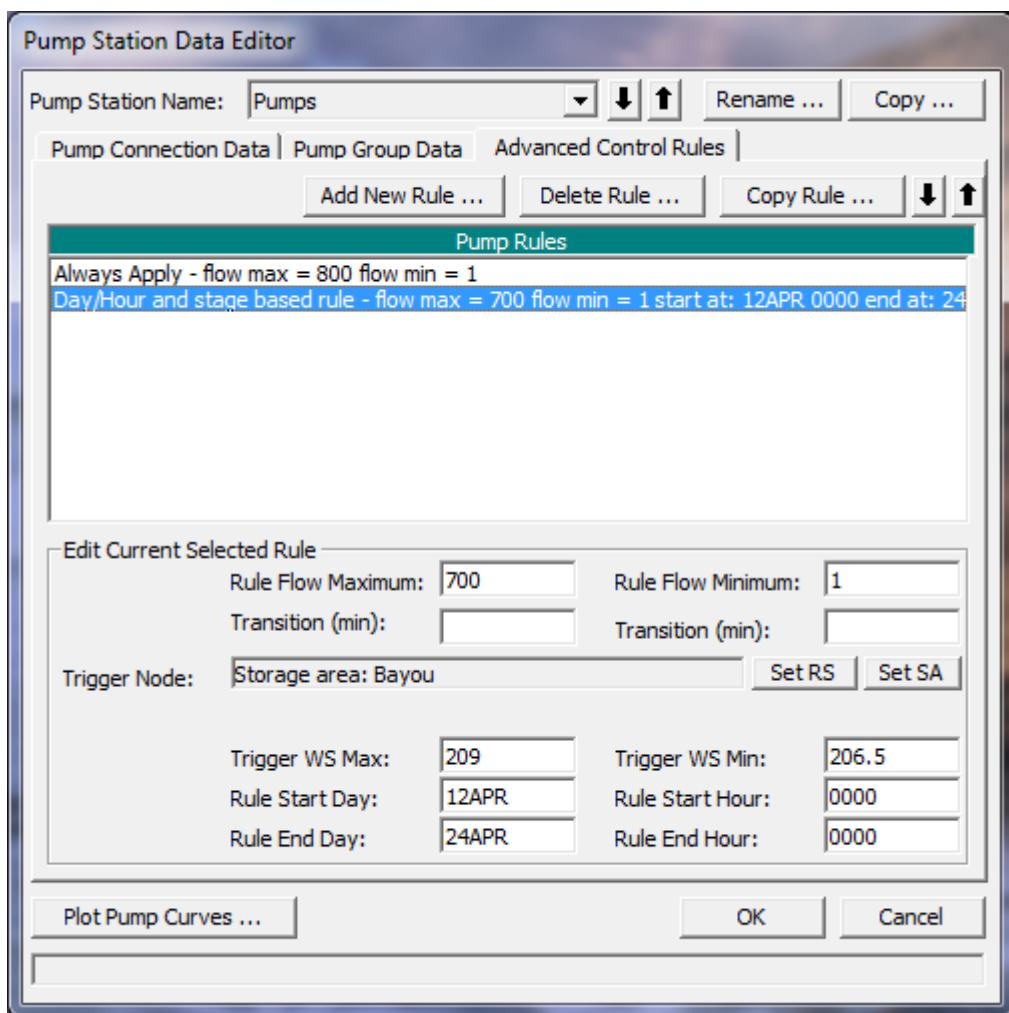
WS Elev On: This is the elevation at which the pump will be turned on. This is based on the elevation of the water surface at the "Inlet" location connected to the pump.

WS Elev Off: This is the elevation at which the pump will be turned off. This is based on the elevation of the water surface at the "Inlet" location connected to the pump.

Pump GIS Data: Another table directly to the right of the individual Pump Name table, is a table containing the GIS coordinates of the individual pumps. If the user clicks on a pump row in the Pump Names table (ex, if Pump #1 was selected), then the X, Y coordinates in the

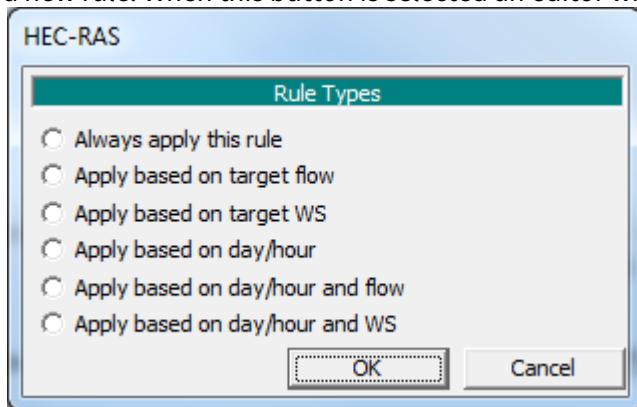
Pump Coordinates table are for Pump #1. If another Pump is selected, then the X, Y coordinates will be for that pump. Entering X, Y coordinates for a pump is only needed when connecting pumps to 2D Flow Areas. This is required in order to figure out which cell(s) the pump is connected to in the 2D Flow Area. User must draw a spatial line that goes from the Pump Inlet to the Pump Outlet. This line will be used to show the Pump connection spatially in the Geometric Data editor, as well as establish which cell the pump is connected to. X, Y coordinates are not required if you are not connecting pumps to a 2D Flow Area. But if even one end of the pump is connected to a 2D Flow Area, the X, Y coordinates are required.

The final tab, labeled **Advanced Control Rules**, is an optional tab used to specify rules that will override the physical pump data. When this tab is selected the editor will appear as follows:



5 59. Pump Editor with Advanced Control Rules Tab Selected.

As shown in the Figure 5-59 the Advanced Control Rules tab has three buttons at the top of the editor, **Add New Rule**; **Delete Rule**, and **Copy Rule**. The Delete Rule button will delete the currently selected rule from the list of pump rules shown in the text box labeled **Pump Rules**. The Copy Rule button makes a copy of the currently opened rule. The Add New Rule button allows the user to enter a new rule. When this button is selected an editor will appear as shown below:



5 60. Rule Types Editor.

As shown in Figure 5-60, there are six types of rules that can be applied to a pump station. Each of the six rule types allow the user to specify a minimum and maximum flow for the entire pump station. This minimum and maximum flow will narrow the range of possible flows that have been computed for the pump station based on the physical pump data. The rule types only differ in how and when the minimum and maximum flow range gets applied.

The first rule type, **Always apply this rule**, is applied at for all time steps in the solution. The second rule type, **Apply Based on Target Flow**, is applied only when a target minimum and/or maximum flow is exceeded (flow is greater than specified maximum or less than specified minimum) at a user specified flow monitoring location. The flow monitoring location can be a cross section within a river reach, or a storage area. The third rule type, **Apply based on target WS**, is applied only when a target minimum and/or maximum water surface elevation is exceeded (stage is greater than specified maximum or less than specified minimum) at a user specified stage monitoring location. The fourth rule type, **Apply based on Day/Hour**, is only applied only during a user specified time window. The user enters a starting day and time, and an ending day and time. The specified maximum and minimum flows are then applied to the pump station only during the user specified time window. The fifth rule type, **Apply based on day/hour and flow**, is a combination of a user specified time window, and a maximum and/or minimum flow target at a user specified flow monitoring location. The last rule type, **Apply based on day/hour and WS**, is a combination of a user specified time window, and a maximum and/or minimum stage target at a user specified stage monitoring location.

The user can also apply a transition time in minutes to the maximum and minimum flow for each of the rules. Therefore if a rule will change the flow from the currently computed value to a user entered maximum, the transition time is used to allow for the flow change to occur over a user specified time. This same concept is used for the minimum flow rate also.

The user can specify as many rules as they want for each pump station. The rules will be applied to the pump station in the order that they have been entered (which is also the order in which they appear in the editor). The user can move a rule up or down in the list by highlighting a rule, then using the up and down arrow buttons to move the rule.

After all of the pump data are entered, press the **OK** button to have the data excepted by the program. This does not save the data to the hard disk, it only allows it to be used in the current execution of the program. To save the data permanently, you must save the geometry data from the File menu of the Geometric Data Editor.

Cross Section Interpolation

Occasionally it is necessary to supplement surveyed cross section data by interpolating cross sections in between two surveyed sections. Interpolated cross sections are often required when the change in velocity head is too large to accurately determine the energy gradient. An adequate depiction of the change in energy gradient is necessary to accurately model friction losses as well as contraction and expansion losses.

Cross section interpolation can be accomplished in three ways from within the HEC-RAS interface. The first method is to simply copy one of the bounding cross sections and then adjust the station and elevation data. The cross section editor allows the user to raise or lower elevations and to shrink or expand various portions of any cross section. The second and third options allow for automatic interpolation of cross section data. From the Geometric Data editor, automatic interpolation options

are found under the Tools menu bar as shown in Figure 5-61.

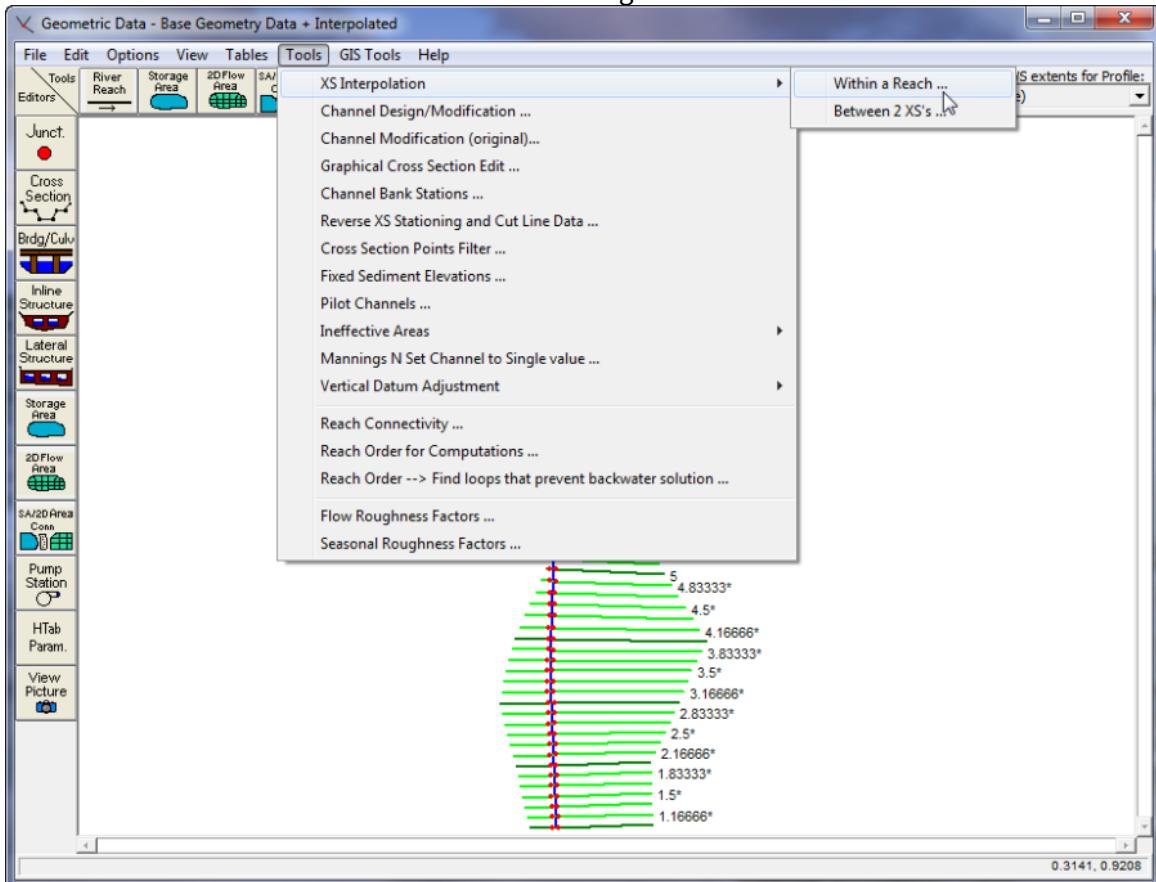


Figure 5 61 Automatic Cross Section Interpolation Options

The first cross section interpolation option, **Within a Reach**, allows for automatic interpolation over a specified range of cross sections within a single reach. When this option is selected, a window will pop up as shown in Figure 5-62. The user must first select the River and Reach that they would like to perform the interpolation in. Next the user must select a starting River Station and an ending River Station for which interpolation will be performed. The user must also provide the maximum allowable distance between cross sections. If the main channel distance between two sections is greater than the user defined maximum allowable, then the program will interpolate cross sections between these two sections. The program will interpolate as many cross sections as necessary in order to get the distance between cross sections below the maximum allowable. Additionally the user can specify the number of decimal places used for the stationing and elevations of the interpolated cross sections.

Cut Line GIS Coordinates. When cross sections are interpolated, their location on the river system schematic is also interpolated. HEC-RAS has two options for interpolating the coordinates of the cross section cut lines: **Linear Interpolation cut lines from the bounding XS's** and **Generate for display as perpendicular segments to reach invert**. The default method is linear interpolation from the bounding cross sections. This method simply draws straight lines between the two cross sections and interpolates the cross section coordinates based on main channel distance. The second method (perpendicular segments to the reach invert line) scales the cross sections along the river reach invert line. A perpendicular segment across the river reach is drawn for the main channel. However, the overbanks are based on average slopes of the invert line upstream and downstream

from the point of intersection.

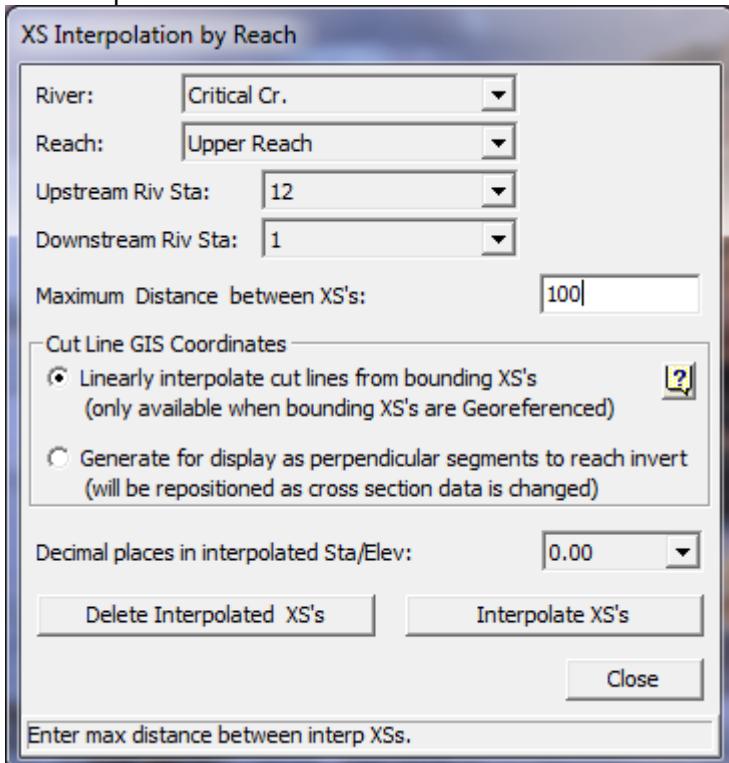


Figure 5 62 Automatic Cross Section Interpolating Within a Reach

Once the user has selected the cross section range and entered the maximum allowable distance, cross section interpolation is performed by pressing the **Interpolate XS's** button. When the program has finished interpolating the cross sections, the user can close the window by pressing the **Close** button. Once this window is closed, the interpolated cross sections will show up on the river schematic as light green lines. The lighter color is used to distinguish interpolated cross sections from user-entered data. Interpolated cross sections can be plotted and edited like any other cross section. The only difference between interpolated sections and user-defined sections is that interpolated sections will have an asterisk ★ attached to the end of their river station identifier. This asterisk will show up on all input and output forms, enabling the user to easily recognize which cross sections are interpolated and which are user defined.

The second type of automatic cross section interpolation, **Between 2 XS's**, allows the user to have much greater control over how the interpolation is performed. When this option is selected, a Cross Section Interpolation window will appear as shown in Figure 5-63.

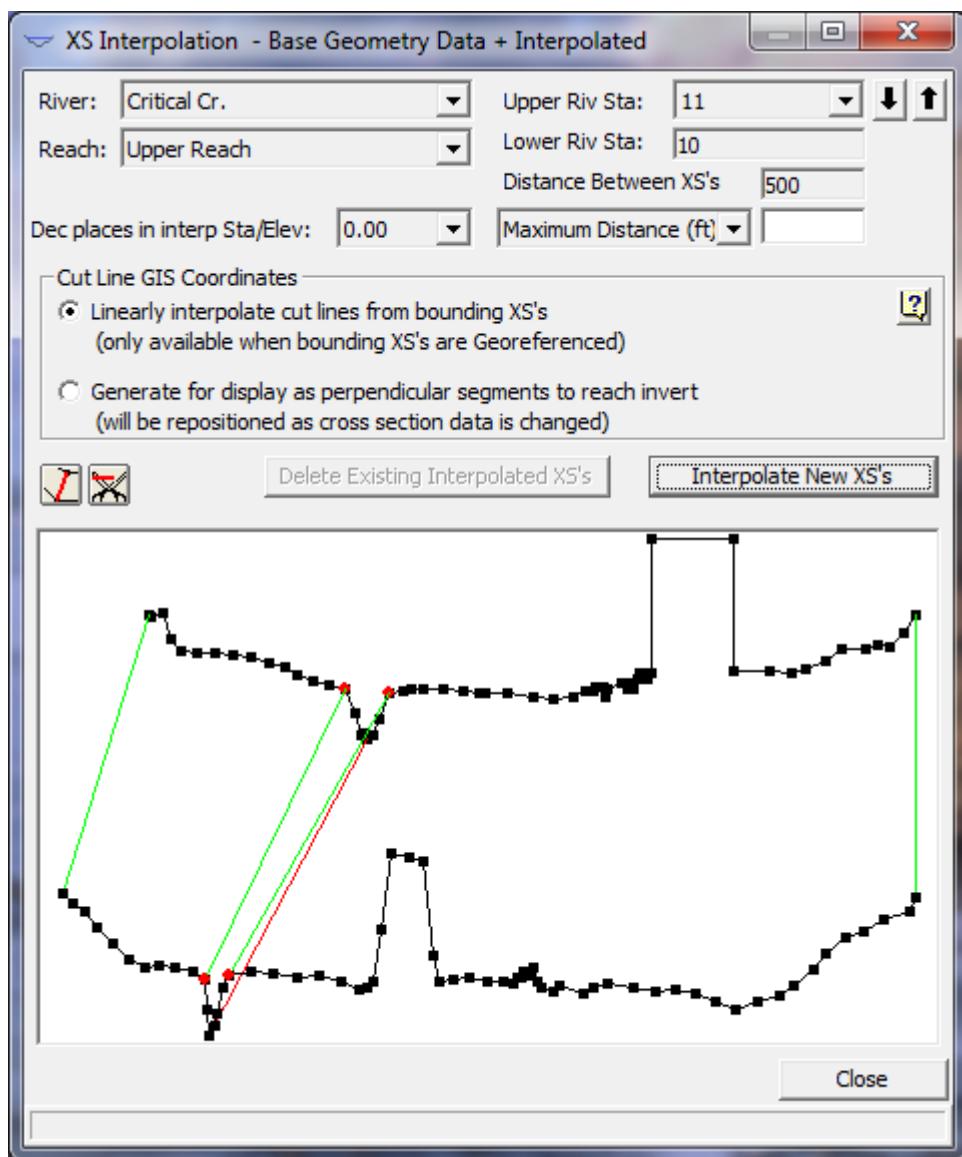


Figure 5 63 Detailed Cross Section Interpolation Window

This cross section interpolation window displays only two cross sections at a time. The user can get to any two cross sections from the River, Reach and River Station boxes at the top of the window. Interpolated cross section geometry is based on a string model as graphically depicted in Figure 561. The string model consists of chords that connect the coordinates of the upstream and downstream cross sections. The cords are classified as master and minor cords. As shown in Figure 5-63, five master cords are automatically attached between the two cross sections. These master cords are attached at the ends of the cross sections, the main channel bank stations, and the main channel inverts. Minor cords are generated automatically by the interpolation routines. A minor cord is generated by taking an existing coordinate in either the upstream or downstream section and establishing a corresponding coordinate at the opposite cross section by either matching an existing coordinate or interpolating one. The station value at the opposite cross section is determined by computing the decimal percent that the known coordinate represents of the distance between master cords and then applying that percentage to the opposite cross section master cords. The number of minor cords will be equal to the sum of all the coordinates of the upstream and

downstream sections minus the number of master cords. Interpolation at any point in between the two sections is then based on linear interpolation of the elevations at the ends of the master and minor cords. Interpolated cross sections will have station and elevation points equal to the number of major and minor cords.

This interpolation scheme is used in both of the automated interpolation options ("Within a Reach" and "Between 2 XS's"). The difference is that the Between 2 XS's option allows the user to define additional master cords. This can provide for a better interpolation, especially when the default of five major cords produces an inadequate interpolation. An example of an inadequate interpolation when using the default cords is shown below.

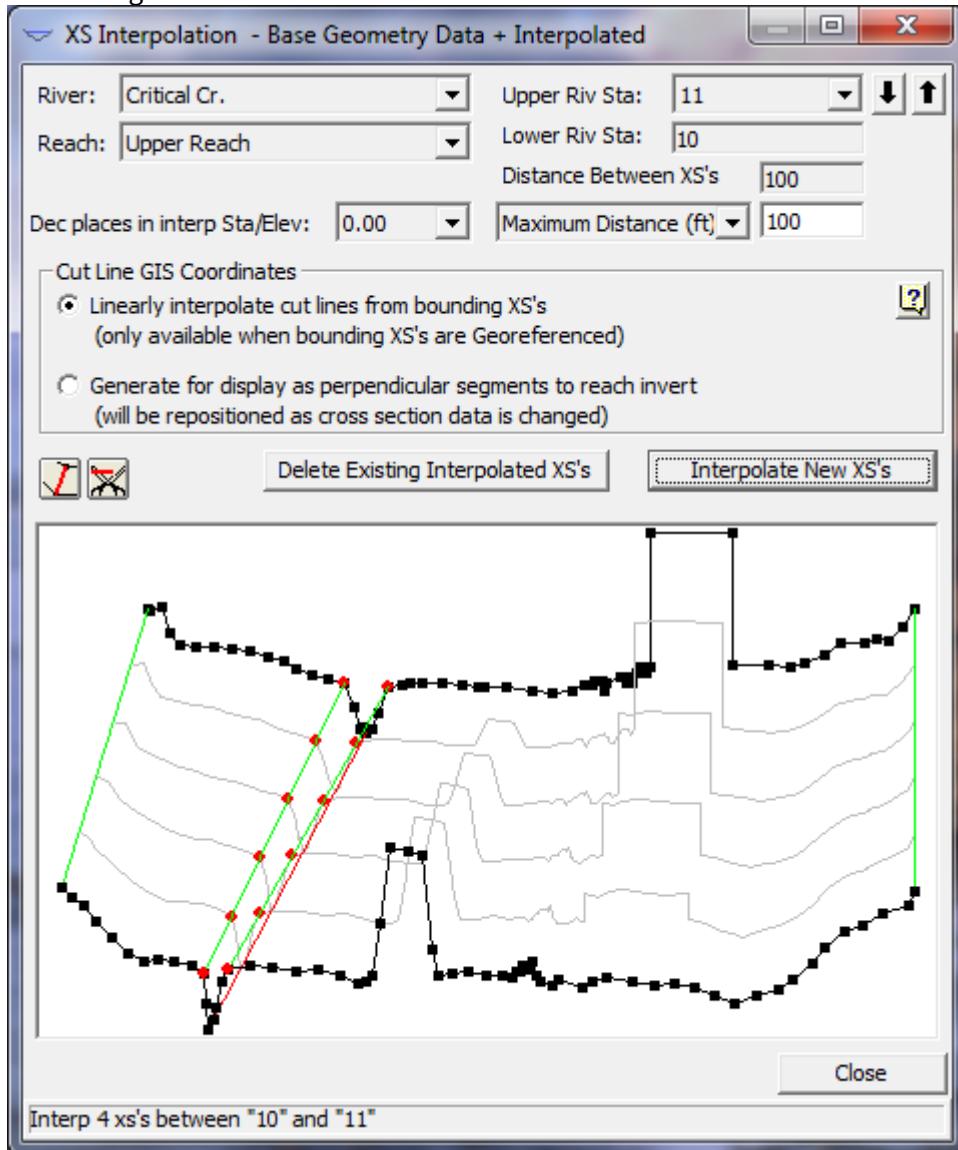


Figure 5 64 Cross Section Interpolation Based on Default Master Cords

As shown in Figure 5-64, the interpolation was adequate for the main channel and the left overbank area. The interpolation in the right overbank area failed to connect two geometric features that could be representing a levee or some other type of high ground. If it is known that these two areas of high ground should be connected, then the interpolation between these two sections should be

deleted, and additional master cords can be added to connect the two features. To delete the interpolated sections, press the **Del Interp** button.

Master cords are added by pressing the **Master Cord** button that is located to the right of the Maximum Distance field above the graphic. Once this button is pressed, any number of master cords can be drawn in. Master cords are drawn by placing the mouse pointer over the desired location (on the upper cross section), then while holding the left mouse button down, drag the mouse pointer to the desired location of the lower cross section. When the left mouse button is released, a cord is automatically attached to the closest point near the pointer. An example of how to connect master cords is shown in Figure 5-65.

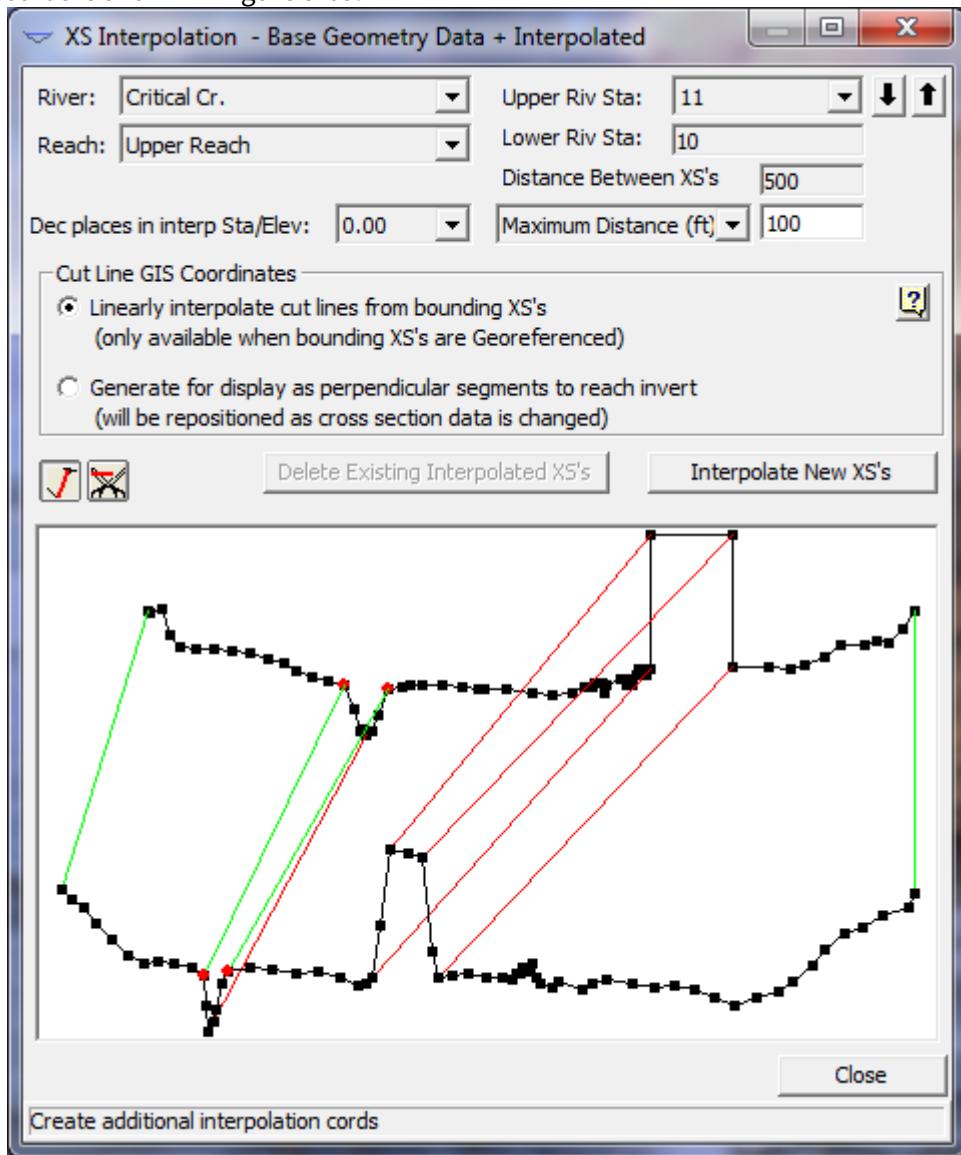


Figure 5 65 Adding Additional Master Cords for Interpolation

User defined master cords can also be deleted. To delete user defined master cords, press the **scissors** button to the right of the master cords button. When this button is pressed, simply move the mouse pointer over a user defined cord and click the left mouse button to delete the cord.

Once you have drawn in all the master cords that you feel are required, and entered the maximum distance desired between sections, press the **Interpolate** button. When the interpolation has finished, the interpolated cross sections will automatically be drawn onto the graphic for visual inspection. An example of this is shown in Figure 5-66.

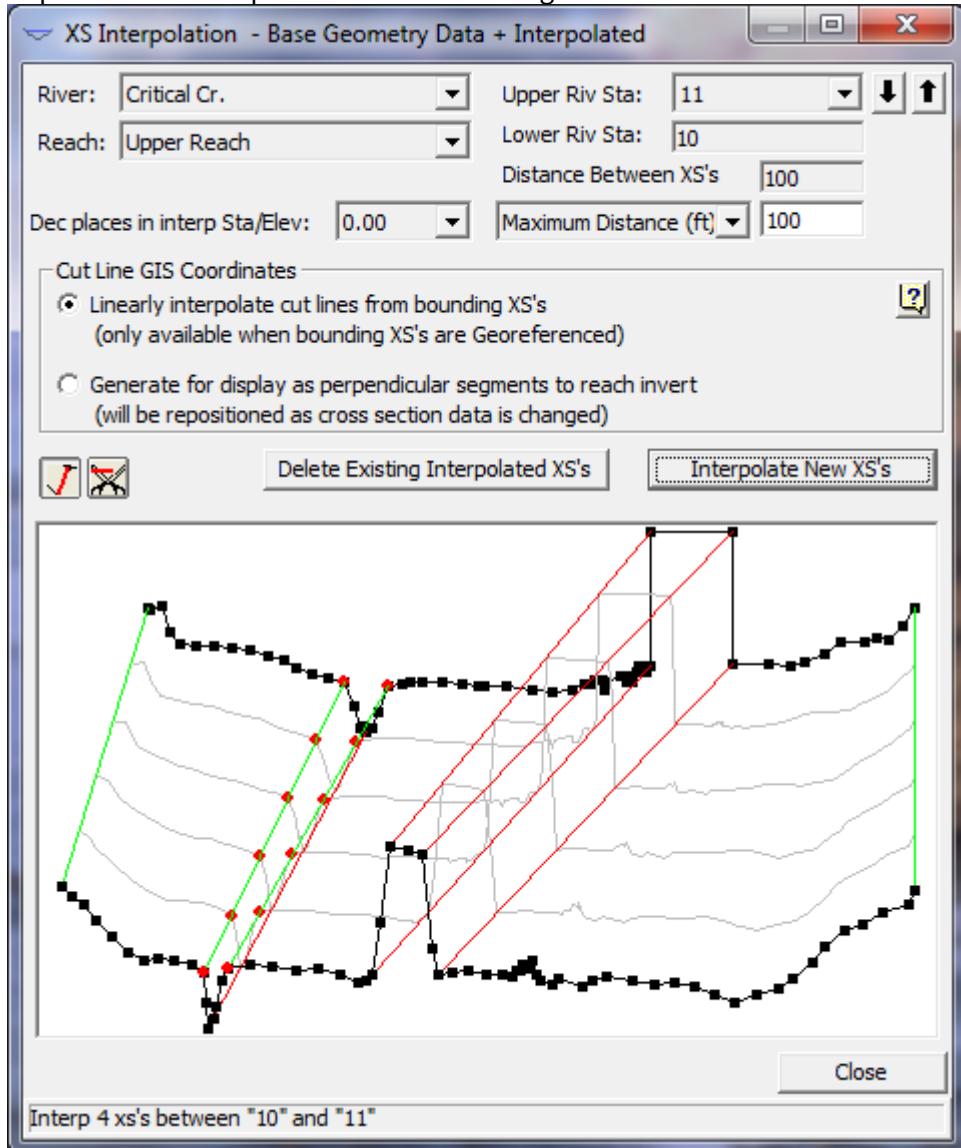


Figure 5 66 Final Interpolation with Additional Master Cords

As shown in Figure 5-66, the interpolation with the addition of user defined master cords is very reasonable.

In general, the best approach for cross section interpolation is to first interpolate sections using the **"Within a Reach"** method. This provides for fast interpolation at all locations within a reach. The "Within a Reach" method uses the five default master cords, and is usually very reasonable for most cross sections. Once this is accomplished, all of the interpolated sections should be viewed to ensure that a reasonable interpolation was accomplished in between each of the cross sections. This can be done from the **"Between 2 XS's"** window. Whenever the user finds interpolated cross sections that are not adequate, they should be deleted. A new set of interpolated cross sections can be developed

by adding additional master cords. This will improve the interpolation.

An additional option available in the "Between Two XS's" interpolation method is the ability to specify a constant distance for interpolation and to specify a specific location to interpolate a cross section. The **Constant Distance** option allows the user to put in a distance. This distance will be used to interpolate cross sections starting from the upstream cross section and moving downstream. Once the user entered distance can no longer be met between the two cross sections, then interpolation stops. The second option, **Set Location (ft)**, allows for the interpolation of a single cross section at a specified distance from the upstream cross section.

CAUTION: Automatic geometric cross section interpolation should not be used as a replacement for required cross section data. If water surface profile information is required at a specific location, surveyed cross section data should be provided at that location. It is very easy to use the automatic cross section interpolation to generate cross sections. But if these cross sections are not an adequate depiction of the actual geometry, you may be introducing error into the calculation of the water surface profile. Whenever possible, use topographic maps to assist you in evaluating whether or not the interpolated cross sections are adequate. Also, once the cross sections are interpolated, they can be modified just like any other cross section.

If the geometry between two surveyed cross sections does not change linearly, then the interpolated cross sections will not adequately depict what is in the field. When this occurs, the modeler should either get additional surveyed cross sections, or adjust the interpolated sections to better depict the information from the topographic map.

River Ice

The current version of HEC-RAS allows the user to model ice-covered channels. This section of the user's manual will describe how to enter the data describing the ice cover and the ice cover properties. If the ice cover geometry is known, that is, if the ice cover thickness and roughness are known throughout the reaches of interest, the user can supply these data and describe the ice cover directly. If the ice cover results from a wide-river type jam, HEC-RAS will estimate the jam thickness in reaches where the ice jam occurs. In this case, the user can supply the material properties of the jam or use the default values. To find out how to view specific results for a channel with an ice cover, see Chapter 9 of this User's manual.

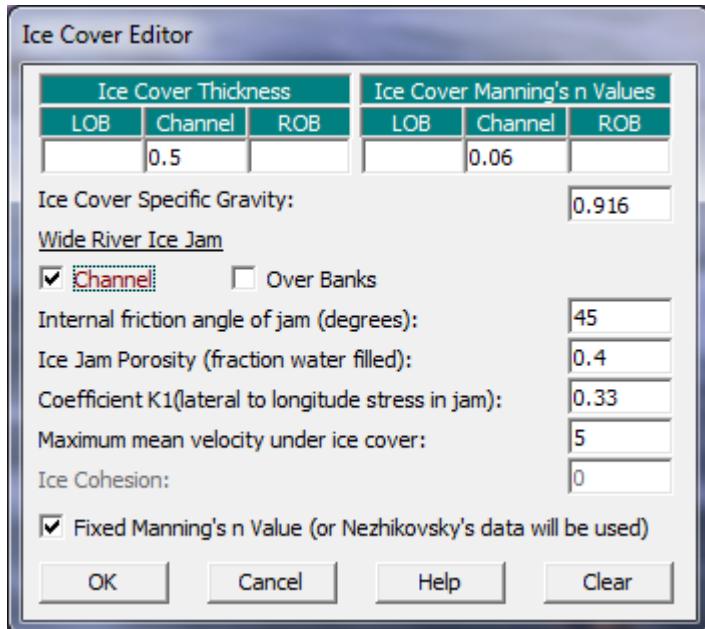
Entering and Editing Ice Data

River ice data can be entered in two ways: by using the **Add Ice Cover** option under the **Options** Menu found at the top of the Cross Section Data Editor, or by using **Tables** Menu found at the top of the Geometric Data window. Both ways of entering data will be described below. It is important to remember that at least two cross sections are required to define the ice cover. A cross section should be placed at the upstream and downstream ends of each ice-covered reach.

Entering Ice Data at a Cross Section

To enter river ice data the user presses the Cross Section button on the Geometric Data window. Once the cross section button is pressed the Cross Section Data Editor will appear. See the CROSS SECTION DATA section of the User's Manual, for information on selecting the appropriate river, reach, and cross section in the Cross Section Data Editor. Once a cross section with an ice cover has been selected, choose the "Add ice cover..." option under the Options Menu found at the top of the

Cross Section Data Editor. This will open the Ice Cover Editor (see figure below). All ice data for this cross section can be entered with this editor.



Ice Cover Editor

Ice Cover Thickness. The ice cover thickness in the left overbank (LOB), main channel (Channel), and right overbank (ROB), are entered here. If there is no ice in any of these areas, a thickness of zero should be entered.

Ice Cover Manning's n Values. The Manning's n value of the ice cover in the left overbank (LOB), main channel (Channel), and right overbank (ROB), are entered here. If any part of a cross section has a non-zero ice thickness, a Manning's n value must be supplied.

Ice Cover Specific Gravity. The default value is 0.916. The user can supply an alternative value here.

Wide River Ice Jam. The boxes under this option are checked if this section is to be treated as a wide river ice jam. In this case, HEC-RAS will estimate the jam thickness using the complete ice jam force balance as described in the Hydraulic Reference Manual. The user can confine the jam to the main channel or allow the jam to be in the channel and overbank areas by checking the proper boxes. If the ice cover is confined to the channel, the overbanks can have a known ice thickness (including an ice thickness of zero) assigned to them in the Ice Cover Thickness option. If the Wide River Ice Jam option is selected, an ice cover thickness must be supplied for the main channel using the **Ice Cover Thickness Option** or through the Ice Tables (see below). This ice cover thickness will be used as the initial estimate of the ice jam thickness and will also serve as the minimum thickness allowed for the ice jam at that section. If the jam is allowed in the overbank areas, the channel and overbanks hydraulic properties will be combined to calculate a single jam thickness for the channel and overbanks. **NOTE:** A wide river jam cannot be selected for an entire river channel. A cross section with fixed ice cover geometry must be included at the upstream end and the downstream end of the wide river ice jam to serve as the boundary conditions for the jam. There is no limit to the number of separate wide river jams that can exist in a river network. However, every ice jam must have a cross section with fixed ice geometry at its upstream and downstream limit. Ice jams can extend through

any number of junctions. However, the jam will only be extended between reaches that have identical reach names.

Internal Friction Angle of the Jam (degrees). This describes the "strength" of the ice jam as a granular material. The default value is 45_degrees.

Ice Jam Porosity (fraction water filled). This describes the fraction of the ice jam that is filled with liquid water. The default value is 0.4.

Coefficient K1 (longitudinal to lateral stress in jam). This describes the ratio of the lateral stress and the longitudinal stress in the jam. It is the efficiency of the jam in transferring longitudinal stress into lateral stress against the channel banks. The default value is 0.33

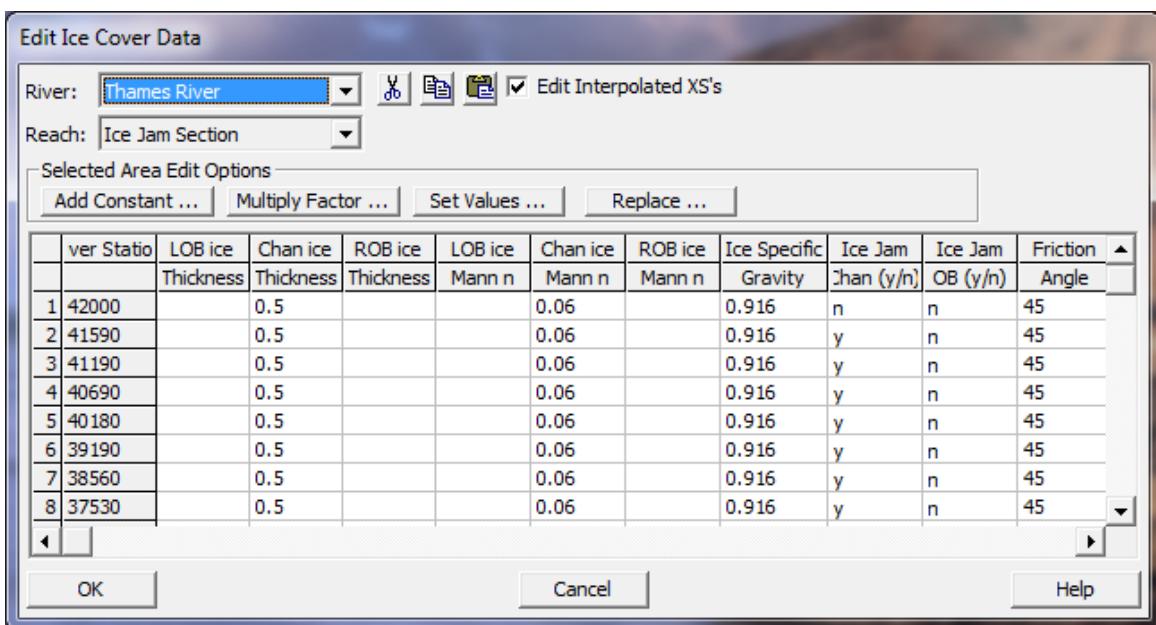
Maximum mean velocity under ice cover. This option limits the maximum mean velocity under a wide river ice jam. The default value is 5 fps. If the maximum mean velocity is greater than this, the ice cover will be thinned until the maximum velocity is attained, or the minimum ice thickness supplied by the user is reached. In any case, the jam thickness will not be allowed to be thinner than the user supplied thickness. This option prevents the jam from thickening to such an extent that the entire cross sectional area of the channel would become blocked.

Ice Cohesion. At present, the ice jam cohesion is set to the default value of zero. This cannot be changed by the user. A value of zero is appropriate for breakup ice jams.

Fixed Manning's n Value (or Nezhikovsky's data will be used). The Manning's n value of the ice jam can be specified by the user or estimated using the empirical relationships developed from Nezhikovsky's data (1964). The empirical relationships estimate the Manning's n value on the basis of the jam thickness and the total water depth. The default is the user supplied Manning's n value. Once all the ice data have been entered and edited, click the **OK** button. At the bottom of the Cross Section Data Editor, in the space entitled "List of special notes for cross section," the words "Ice cover" will now appear. The user can now click on the words "Ice cover" to return to the ice cover editor for that cross section.

Entering Ice Data Through a Table

Ice cover information can also be entered using the Tables Menu found at the top of the Geometric Data Window. To enter data the user selects the **Ice Cover** Option under the **Tables** Menu. All the information that can be entered under the Ice Cover Editor can also be entered using the Ice Cover table. It is often very convenient to enter and view data for more than one cross section at a time (figure below).



Entering Ice Information Using a Table

The user has the option of entering the ice thickness in the left overbank (LOB ice Thickness), the main channel (Chan ice Thickness), and the right overbank (ROB ice Thickness); the Manning's n value of the left overbank ice cover (LOB ice Mann n), the main channel ice cover (Chan ice Mann n), and the right overbank ice cover (ROB ice Mann n); and the specific gravity of the ice cover (Ice gravity). The user can also choose if the ice cover in the main channel is the result of a wide river ice jam (Ice Jam Chan. **Note: only y or n can be entered here**), and choose if the overbanks are also included in the wide river ice jam (Ice Jam OB. **Note: only y or n can be entered here**). The user can further select the internal friction angle of the ice jam (Friction Angle); the porosity of the ice jam (Porosity); the longitudinal to lateral stress ratio of the ice jam (Stress K1 ratio); the maximum allowable under ice flow velocity (Max Velocity); and if the Manning's n value of the ice jam is fixed, that is selected by the user, or if the Manning's n value will be determined by HEC-RAS (**Note: only y or n can be entered here**).

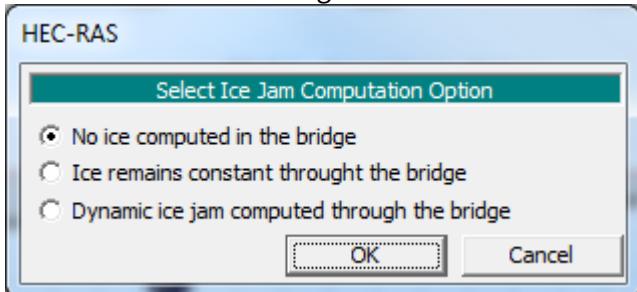
As in all instances where a Table is used to enter data, in each column the user has the option of entering one or more values, adding a constant to one or more of the values, multiplying a group of values by a factor, or changing a group of values to a specific value. Additionally, cut, copy, and paste buttons are provided to pass data to and from the Windows Clipboard.

Entering Ice Data at Bridges

The influence of ice on the hydraulics of bridges is a relatively unstudied area. Little is known about the ways in which a wide river ice jam interacts with the various components of a bridge. The important components of a bridge that may interact with an ice jam include the piers, low chord, approaches, and deck. Previous investigations of ice jams in rivers with bridges have largely ignored their presence, arguing that observed ice jams did not contact the low steel significantly. Removing the bridge information for an ice jam study still remains an option. However to allow a user to efficiently use HEC-RAS with ice *and* with bridges, three separate options are provided. These options allow the user to selectively decide at each bridge whether or not the ice cover can interact

with the structure. When modeling ice at bridges, users should carefully evaluate the results for consistency and accuracy.

Ice information at bridges is entered using the Bridge/Culvert editor found under the Geometry editor. Use the options menu in the Bridge/Culvert editor to select the ice option. This will open a window as shown in the figure below.



Entering Ice Information at Bridges

No ice computed in the bridge. In this case no ice calculations will be performed at the bridge itself and the ice thickness at the bridge will be assumed to be zero.

Ice remains constant through the bridge. In this case, the ice thickness at the cross section immediately upstream of the bridge will be used. If the ice thickness is calculated as a wide river jam, this thickness will be used.

Dynamic ice jam computed through the bridge. In this case, the wide river ice jam calculations will be performed at the bridge cross section. The user must check for inconsistent results, especially if any part of the ice jam is above the low chord of the bridge.

Setting Tolerances for Ice Jam Calculations

The user can override the default settings for the ice jam calculation tolerances which are used in the solution of the ice jam force balance equation. The tolerances are set as multiples of the *water surface calculation tolerance* used in the solution of the energy equation, described in the Simulation Options section of Chapter 7. The user can change the values of these tolerances by changing the *water surface calculation tolerance*. The tolerances are as follows:

Ice thickness calculation tolerance. This tolerance is compared with the difference between the computed and assumed ice thickness at a cross section. It is set to ten times the *water surface calculation tolerance*. Its default value is 0.1 ft.

Global ice thickness calculation tolerance. This tolerance is compared with the difference between the computed ice thickness at each cross section between successive solutions of the ice jam force balance equation and the energy equation. It is set to ten times the *water surface calculation tolerance*. Its default value is 0.1 ft.

Global water level calculation tolerance. This tolerance is compared with the difference between the computed water surface elevations at each cross section between successive solutions of the ice jam force balance equation and the energy equation. It is set to six times the *water surface calculation tolerance*. Its default value is 0.06 ft.

Maximum number of ice jam iterations. This variable defines the maximum number of times for successive solutions of the ice jam force balance equation and the energy equation. It is set to 2.5 times the *maximum number of iterations*. Its default value is 50.

Viewing and Editing Data through Tables

Once cross-section and hydraulic structure data are entered, the user can view and edit certain types of data in a tabular format. The following options are available from the **Tables** menu option on the **Geometric Data** editor:

Manning's n or k values

It is often desirable to view and edit the Manning's n values or roughness heights (k values) for several cross sections all at the same time. From the **Geometric Data** editor, the user can select **Manning's n or k values** from the **Tables** menu item. Once this option is selected, a window will appear as shown in Figure 5-70.

As shown in Figure 5-70, the user has the options of selecting either n or k values to be used as the roughness coefficient, add a constant to one or more of the n or k values, multiply a group of n or k values by a factor, or change a group of n or k values to a specific value. Additionally, cut, copy, and paste buttons are provided to pass data to and from the Windows Clipboard. The user can optionally select to view all regions of the cross sections, the left overbank only, the main channel only, the right overbank only, or both overbanks. The main channel roughness coefficients are shown shaded in green in order to make it easier to distinguish between overbank and channel roughness coefficients in the table.

To add a constant to a group of n or k values, the user must first highlight the values that they would like to change. Highlighting is accomplished by placing the mouse in the upper left cell of the desired cells to highlight, then press the left mouse button and drag the cursor to the lower left corner of the desired cells to highlight. When the left mouse button is released, the cells that are selected will be highlighted (except the first cell). Once the user has highlighted the desired cells to be modified, press the **Add Constant** button. This will bring up a pop up window, which will allow the user to enter a constant value that will be added to all cells that are highlighted.

Edit Manning's n or k Values

River Station	Frctn (n/K)	n #1	n #2	n #3	n #4	n #5	n #6
1 5.99	n	0.1	0.14	0.04	0.14		
2 5.875*	n	0.1	0.12	0.04	0.14		
3 5.76	n	0.1	0.04	0.14			
4 5.685*	n	0.09	0.1	0.04	0.1		
5 5.61	n	0.08	0.1	0.04	0.06		
6 5.525*	n	0.07	0.09	0.1	0.04	0.06	
7 5.44	n	0.06	0.1	0.04	0.06		
8 5.41	n	0.15	0.25	0.04	0.15		
9 5.4	Bridge						
10 5.39	n	0.15	0.2	0.04	0.2	0.15	0.2
11 5.29	n	0.04	0.06	0.04	0.06		
12 5.21*	n	0.07	0.08	0.04	0.08	0.06	
13 5.13	n	0.1	0.04	0.1	0.06		
14 5.065*	n	0.1	0.04	0.1	0.08		
15 5.0	n	0.1	0.04	0.1			

Figure 5 70. Manning's n Data View and Editing Table

To multiply a group of n or k values by a factor, the user first highlights the desired cells. Once the cells are highlighted, pressing the **Multiply by a Factor** button will bring up a pop up window. This window allows the user to enter a value that will be multiplied by each of the highlighted cells.

To set a group of n or k values to the same number, the user must first highlight the values that they would like to change. Once the cells are highlighted, pressing the **Set Values** button will bring up a pop up window. This window will allow the user to enter a specific n or k value, which will replace all of the highlighted values.

The last option (**Replace**) is to find and replace a specific number with a new number. When this option is selected the user is asked to put in a number to search for and also a number to replace it with. This option only searches and replaces data in highlighted fields.

An additional option is the **Reduce Channel to a Single n-Value**. This option will find the Manning's n or k value at the midpoint between the main channel bank station, then change the entire channel to a single roughness coefficient equal to that midpoint value.

The user can also go directly into the table and change any individual values.

Reach Lengths

The user has the ability to view and edit cross section reach lengths in a tabular format. This is accomplished by selecting **Reach Lengths** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear as shown in Figure 5-71. The user has the same editing features as described previously for the n values table. See the discussion under Manning's n or k values, in the previous section, for details on how to edit the data.

Edit Downstream Reach Lengths

River: Beaver Creek Reach: Kentwood

Edit Interpolated XS's

Selected Area Edit Options

Add Constant ... Multiply Factor ... Set Values ... Replace ...

	River Station	LOB	Channel	ROB
1	5.99	440	600	400
2	5.875*	440	600	400
3	5.76	225	400	275
4	5.685*	225	400	275
5	5.61	240	460	190
6	5.525*	240	460	190
7	5.44	270	170	500
8	5.41	100	100	100
9	5.4	Bridge		
10	5.39	320	500	580
11	5.29	410	416.5	410
12	5.21*	410	416.5	410
13	5.13	310	355	340
14	5.065*	310	355	340
15	5.0	0	0	0

OK Cancel Help

Figure 5 71. Reach Lengths View and Editing Table

Contraction and Expansion Coefficients (Steady Flow)

The user has the ability to view and edit contraction and expansion coefficients for Steady flow hydraulics in a tabular format. This is accomplished by selecting **Contraction/Expansion Coefficients (Steady Flow)** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear as shown in Figure 5-72. The user has the same editing features as described previously for the n values table. See the discussion under Manning's n values, in the previous section, for details on how to edit the data.

Edit Contraction/Expansion Coefficients (Steady Flow)

River: Beaver Creek Reach: Kentwood Edit Interpolated XS's

Selected Area Edit Options: Add Constant ... | Multiply Factor ... | Set Values ... | Replace ...

	River Station	Contraction	Expansion
1	5.99	0.1	0.3
2	5.875*	0.1	0.3
3	5.76	0.1	0.3
4	5.685*	0.1	0.3
5	5.61	0.1	0.3
6	5.525*	0.1	0.3
7	5.44	0.3	0.5
8	5.41	0.3	0.5
9	5.4	Bridge	
10	5.39	0.3	0.5
11	5.29	0.1	0.3
12	5.21*	0.1	0.3
13	5.13	0.1	0.3
14	5.065*	0.1	0.3
15	5.0	0.1	0.3

OK | Cancel | Help

Figure 5 72. Contraction and Expansion Coefficients (Steady Flow) Table

Contraction and Expansion Coefficients (Unsteady Flow)

In general, contraction and expansion losses are not used in unsteady flow, and therefore the default coefficients are 0.0. Forces due to contractions and expansion are handled in the momentum equation through pressure force differences. However, because HEC-RAS is a one-dimensional unsteady flow model, the one-dimensional momentum equation does not always capture all of the forces acting on the flow field at a sharp contraction and/or expansion zone. In order to better approximate the forces acting on the water, and the resulting water surface elevation, at a contraction and/or expansion, the user can enter empirical contraction and expansion coefficients for unsteady flow modeling. These coefficients will be multiplied by a change in velocity head, just like in steady flow modeling, but the resulting energy loss gets converted to an equivalent force for placement into the momentum equation.

The user has the ability to view and edit contraction and expansion coefficients for Unsteady Flow hydraulics in a tabular format. This is accomplished by selecting **Contraction/Expansion Coefficients (Unsteady Flow)** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear similar to the one shown in Figure 5-73. The user has the same editing features as described previously for the n values table. See the discussion under Manning's n values, in the previous section, for details on how to edit the data.

Edit Contraction/Expansion Coefficients (Unsteady Flow)

River: Bald Eagle Edit Interpolated XS's

Reach: Loc Hav

Selected Area Edit Options

Add Constant ... Multiply Factor ... Set Values ... Replace ...

	River Station	Contraction	Expansion
106	57250.60	0.1	0.3
107	56775.92	0.1	0.3
108	56243.79	0.1	0.3
109	55700.55	0.1	0.3
110	54696.51	0.1	0.3
111	53922.53	0.1	0.3
112	52879.19	0.1	0.3
113	51784.74	0.1	0.3
114	50720.68	0.1	0.3
115	49715.77	0.1	0.3
116	48965.94	0.1	0.3
117	48110.09	0.1	0.3
118	47453.14	0.1	0.3
119	46722.59	0.1	0.3
120	46310.48	0.1	0.3

OK Cancel Help

Figure 5 73. Contraction and Expansion Coefficients (Unsteady Flow) Table

Minor Losses

Minor losses due to bends, junctions, etc... can be added to both the steady flow and the unsteady flow solution. Minor losses are computed by the user entering a K loss coefficient at a specific cross section. The K loss coefficient can vary from 0.0 to 1.0. This loss coefficient gets multiplied by the velocity head at that specific cross section in order to compute the minor energy loss. This energy loss gets added to the energy equation for steady flow computations. For unsteady flow computations, the energy loss is converted to an equivalent force and inserted into the momentum equation. In both cases the energy loss is assumed to act as a force in the upstream direction to slow the flow down.

To use the minor loss option, select **Minor Losses** from the **Tables** menu at the top of the Geometric editor. When this option is selected, a table will appear allowing the user to enter a K loss coefficient at any desired cross section.

Bank Stations

This table allows the user to set or move the main channel bank stations. If the main channel bank stations have not been set, and the user brings up this table, the bank stations will be set to the ends of the cross section. If the bank stations are already set the user can adjust the bank stations by adding a constant, multiplying them by a factor, or setting them to a specific value. If the new bank stations do not exist in the cross section, the program will automatically interpolate them.

Levees

This table allows the user to easily enter and edit levee stations and elevations. The editor allows the user to quickly move levee stations and/or elevations. This can be very useful when trying to decide on a location for levee setbacks and/or the levee elevations.

Ice Cover

This option allows the user to enter ice cover data in a tabular form. A detailed discussion of ice cover information was presented earlier in this chapter.

Names

This option allows the user to change the name of any of the objects that make up the model schematic of the river system. When this option is selected, a submenu will appear that lists all of the objects of the schematic in which the user may want to change the current name of. This list includes: River and Reach Names; River Stations; Node Names; Node Descriptions; Junctions; Storage Areas; Storage Area Connections; and Pump Stations.

River and Reach Names

This option will bring up a table of all the River and Reach names in the model. The user can change any name by simply going to the appropriate cell and changing the current label.

River Stationing

This option allows the user to view and edit the cross section river stationing in a tabular form. This is accomplished by selecting **Names**, then **River Stations** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear as shown in Figure 5-74. This table allows the user to change the river stationing of individual cross sections, add a constant value to the river stationing of selected cross sections (those cross sections highlighted by the user), multiply the selected cross sections river stationing by a factor, or to renumber the cross section river stationing based on the main channel reach lengths

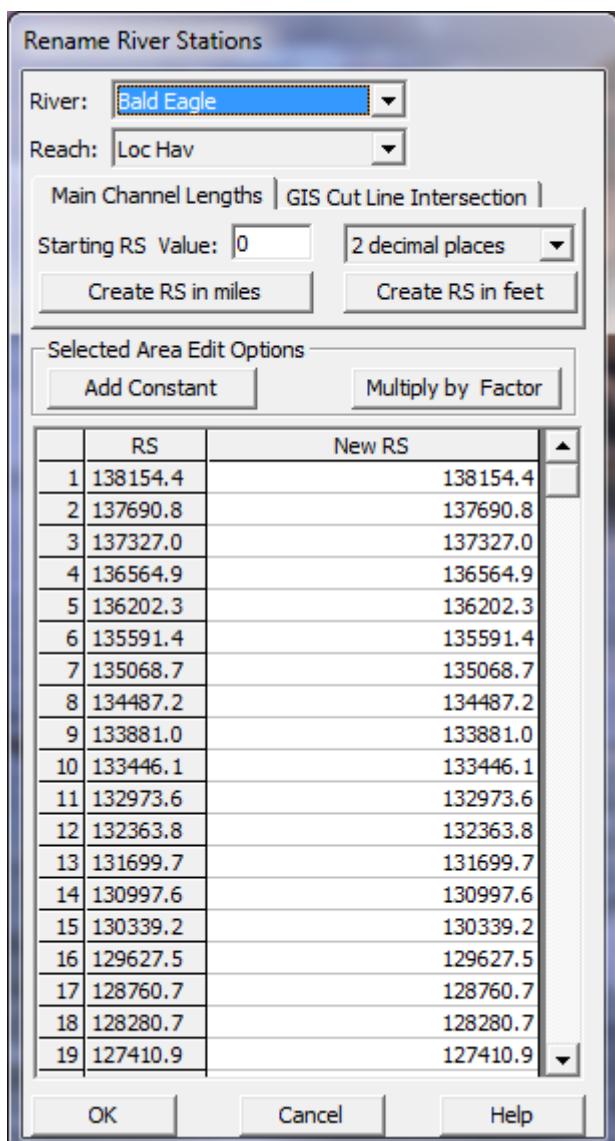


Figure 5 74. Cross Section River Stationing View and Editing Table

Node Names

This option allows the user to add an additional name to a node (a node is a cross section, bridge, culvert, inline structure, etc...). The name can be up to 16 characters long. The user can request that the name be displayed on a profile plot or on a cross-section plot. To use this feature, select **Names**, then **Node Names** from the **Tables** menu. When this option is selected a window will appear as shown in Figure 5-75. Enter any text name that you want at a desired location within the model.

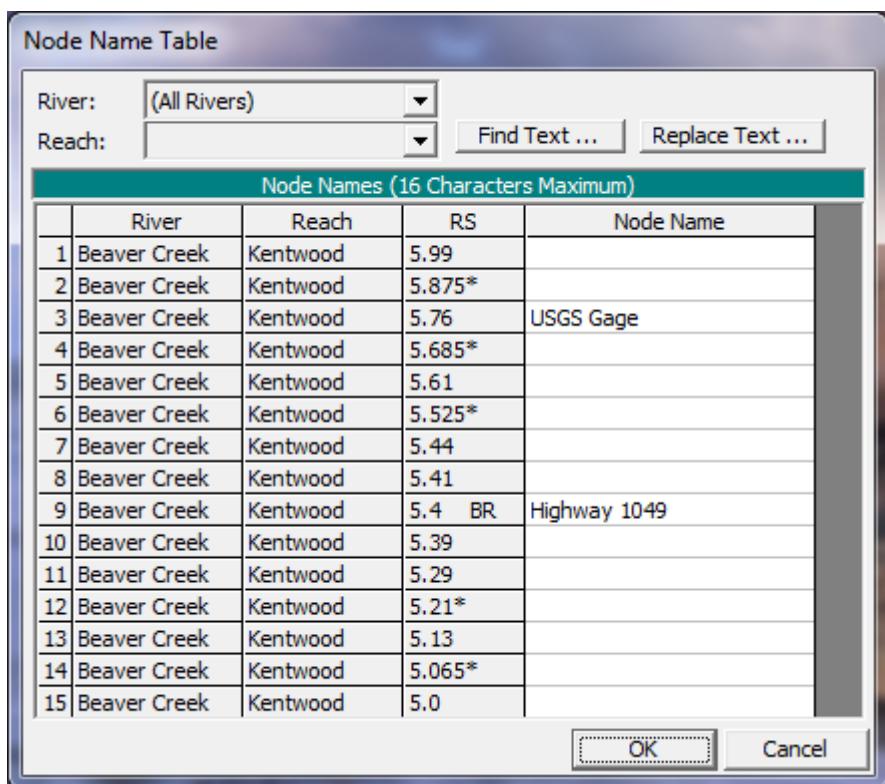


Figure 5 75. Node Name Table Editor

Node Descriptions

This table allows the user to enter a description for any node (cross section, bridge, culvert, inline structure, lateral structure, and pump stations). The description can be up to six rows of text. The table allows the user to display any number of the rows at one time. The user can request that the description be displayed on a profile plot or on a cross-section plot. To use this feature, select **Names**, then **Node Descriptions** from the **Tables** menu. Enter any text description that you want at a desired location within the model.

Junctions

This option allows the user to change the name of any junction that is currently in the model.

Storage Areas

This option allows the user to change the name of any storage area that is currently in the model.

Storage Area Connections

This option allows the user to change the name of any storage area connection that is currently in the model.

Pump Stations

This option allows the user to change the name of any pump station that is currently in the model.

Picture File Associations

This table allows the user to see and edit the directories that pictures are attached to for the project. For example, if all of the pictures for a project were in a directory separate from the project, and these pictures were then moved to another directory, this table would allow you to easily change the associated directory for the pictures.

Ineffective Flow Area Elevations

This table allows the user to see the trigger elevations for all of the ineffective flow areas in the model. The user can change any of the elevations directly from this table.

Bridge Width Table

This option allows the user to view and/or modify bridge width and distance data. Previous versions of HEC-RAS (versions 2.21 and earlier) allowed the user to enter a zero length between the cross sections inside of a bridge and the cross sections just outside of the bridge. This creates an unrealistic water surface profile in the vicinity of the bridge. Current versions require the user to maintain some distance between the outside cross sections and the bridge structure. This table was added to make the process of modifying old data sets less painful. When this option is selected, a window will appear as shown in Figure 5-76. As shown in Figure 5-76, the user is given the length between the cross sections that bound the bridge (**Dist Avail**), the distance between the upstream cross section and the bridge (**Upstream Dist**), the bridge width (**Bridge Width**), and the distance between the downstream cross section and the bridge (**Downstream Dist**). The user must ensure that the upstream and downstream distances are greater than zero. This will require entering an upstream distance, and then changing the bridge width to allow for a positive downstream distance.

Bridge Width and Upstream Distance Table

River: Bald Eagle

Reach: Loc Hav

Selected Area Edit Options

Add Constant ... Multiply Factor ... Set Values ... Replace ...

	River Station	Dist Avail	Upstream Dist	Bridge Width	Downstream Dist
1	103245	247.45	100	42	105.45
2	75960	85.15	30	30	25.15
3	58780	127.7	40	20	67.7
4	36713	106.12	35	30	41.12
5	23828	88.41	20	50	18.41
6	21241	83.41	30	20	33.41
7	15100	593.53	240	210	143.53
8	11985	169.43	50	50	69.43
9	2920	316.24	50	160	106.24
10	2436	129.19	15	100	14.19

OK Cancel Help

Figure 5 76. Bridge Width and Distance Table

Weir and Gate Coefficient Table

This table allows the user to see and edit all of the weir and gate coefficients for all of the inline and lateral structures within the model.

Summary of the weir/gated structures						
	Location	Overflow Weir Coef	Sluice Gate Coef	Radial Gate Coef	Gate Orifice Coef	Gate Weir Coef
1	Clear_Fork Clear_Fork 60.4 IS	3.3	0.6	0.6	0.8	3
2	Clear_Fork Clear_Fork 57.825 IS	2.6				
3	Clear_Fork Clear_Fork 56.7 LS	0.1				
4	Clear_Fork Clear_Fork 55.39 IS	2.6				
5	Clear_Fork Clear_Fork 54.61 IS	2.6				
6	Clear_Fork Clear_Fork 53.6 IS	2.6				
7	Clear_Fork Clear_Fork 52.79 IS	2.6				
8	Clear_Fork Clear_Fork 52.02 IS	2.6				
9	Clear_Fork Clear_Fork 50.39 IS	2.6				
10	Clear_Fork Clear_Fork 50.06 IS	2.6				
11	Clear_Fork Clear_Fork 49.80 IS	2.6				
12	Clear_Fork Clear_Fork 49.54 IS	2.6				
13	Clear_Fork Clear_Fork 49.27 IS	2.6				
14	Clear_Fork Clear_Fork 48.15 LS	0.1				
15	Clear_Fork Clear_Fork 47.2991IS	2.6				
16	Clear_Fork Clear_Fork 47.2 LS	2				
17	Clear_Fork Clear_Fork 46.3716 IS	2.6				
18	Clear_Fork Clear_Fork 45.2 LS	2				
19	Clear_Fork Clear_Fork 44.7 LS	2				
20	Clear_Fork Clear_Fork 43.0692 IS	2.6				
21	Clear_Fork Clear_Fork 40.9276 IS	2.6				
22	Clear_Fork Clear_Fork 36 LS	0.1				
23	Clear_Fork Clear_Fork 28.9 LS	0.1				
24	Clear_Fork Clear_Fork 10.9 LS	0.1				
25	Clear_Fork Clear_Fork 6.33 LS	0.5				
26	Clear_Fork Clear_Fork 5.43 LS	0.5				

HTAB Internal Boundaries Table

This table allows the user to see and edit all of the Hydraulic Tables properties that have been specified for internal boundaries, such as bridges, culverts, inline structures, lateral structures, and storage area connections. These hydraulic table properties are used in defining the limits that the pre-processor uses for building the family of curves for each internal boundary structure. This option is only used in an unsteady flow simulation.

HTab Parameters for Internal Boundaries							
Selected Area Edit Options							
Summary of the parameters for internal boundary rating curves							
	Location	#Pt FF Curve	# RC	#Pt on RC	HW Max	TW Max(Opt.)	Max Flow(Rec.)
1	SA Conn:SAC460L	50	50	20	500		
2	SA Conn:SAC461L	50	50	20	500		
3	SA Conn:SAC461R	50	50	20	500		
4	SA Conn:SAC463L	50	50	20	500		
5	SA Conn:SAC463R	50	50	20	500		
6	SA Conn:SAC464L	50	50	20	500		
7	SA Conn:SAC465R	50	50	20	500		
8	SA Conn:WF_Loop_Con1	50	50	20			
9	SA Conn:WF_Loop_Con2	50	50	20			

Linear Routing Coefficients

This option allows the user to view and edit any linear routing coefficients that have been entered for either lateral structures or storage area connections.

Linear Routing Coefficients					
Selected Area Edit Options					
Summary of the Linear Routing Coefficients					
	Location		Positive Flow Coef	Negative Flow Coef	Elevation
1	Clear_Fork	Clear_Fork	56.7 LS	0.05	0.05 345

Preissmann Slots on Lidded XS's

This table allows the user to turn the Preissmann slot option on or off for any or all of the cross sections that have lids added to them. An example of this table is shown below in Figure 5-77.

Preissmann's Slot Table				
River:	(All Rivers)	Reach:		
Check for a Preissmann's Slot to be added to XS's with a lid				
	River	Reach	RS	Add Preissmann Slot
1	17th Street	3	36020	<input checked="" type="checkbox"/>
2	17th Street	3	35819.8*	<input checked="" type="checkbox"/>
3	17th Street	3	35619.6*	<input checked="" type="checkbox"/>
4	17th Street	3	35419.4*	<input checked="" type="checkbox"/>
5	17th Street	3	35219.2*	<input checked="" type="checkbox"/>
6	17th Street	3	35019.*	<input checked="" type="checkbox"/>
7	17th Street	3	34818.8*	<input checked="" type="checkbox"/>
8	17th Street	3	34618.6*	<input checked="" type="checkbox"/>
9	17th Street	3	34418.4*	<input checked="" type="checkbox"/>
10	17th Street	3	34218.2*	<input checked="" type="checkbox"/>

Figure 5 77. Table Editor for Preissmann Slot Option.

As shown in the Figure above, every cross section that has a lid will show up in the table. If the user wants the Preissmann slot option to be used during unsteady flow calculations, then that cross section must be checked in the column labeled **Add Preissmann Slot**. An easy way to turn this option on or off for all of the cross sections is to click on the column heading of the check box column (which will highlight all of the locations in the table). Then, pressing the space bar will toggle the check mark on and off for all of the highlighted locations.

The **Preissmann Slot** option will instruct the computational code to treat this cross section and lid as a pressurized pipe. This option is only for unsteady flow computations. If the option is turned on, the conveyance curves for the cross section will be truncated at the maximum low chord elevation of the lid. Preissmann slot theory will be used for modeling the flow once it becomes pressurized. For more on modeling pressurized pipes in HEC-RAS, please review the section on modeling pressurized pipes in Chapter 16 of this manual, as well as the HEC-RAS Hydraulic Reference manual.

Manning's n by Land Classification

This table allows the user to enter Manning's n values for various Land Classification types. This option requires the user to have added a Land Classification Map layer into the HEC-RAS project inside of RAS Mapper. If a land Classification Layer has been developed within HEC-RAS Mapper, that layer can then be selected (associated with a specific geometry file). Once a Land Classification layer is selected, the types of land classifications contained within that layer will show up in the table (See Figure below). The user can then enter a Manning's n values to associate with each Land Classification type (name). Currently this option for entering Land Classifications, and associating Manning's n values with them, is only used for defining Manning's n values for 2D Flow Areas. See the Figure below for this example:

Land Cover to Manning's n (2D Flow Areas Only)				
Set Manning's n to Override Default Land Cover Values				
Selected Area Edit Options				
	Add Constant ...	Multiply Factor ...	Set Values ...	Replace ...
	Land Cover Layer	Default Mann n	Base Mann n (blank for default)	Flat Area
1 nodata			0.06	0.054
2 building		10	100	100
3 medium density residential		0.08	0.08	0.072
4 open space		0.04	0.04	0.036
5 park		0.06	0.06	0.054
6 trees		0.12	0.12	0.108
7 urban		0.1	0.1	0.09

Associated Layer: d:\...\Example Data\2D Unsteady Flow Hydraulics\Muncie\LandCover\LandCoverUserShapefile.tif

OK Cancel

Figure 5 78. Manning's n by Land Class Table.

As shown in the Figure above, the editor displays the Manning's n values that may be contained within the Land Cover file (**Default Mann n**). The user can override these default Manning's n values for this specific geometry file by entering their own defaults into the column labeled **Base Mann n (Blank for default)**. If this column is left blank, the software will use the Manning's n values from the Land Cover Layer. If the user enters values in this column, then these values will be used for those Land Cover layer types.

Additionally, the user has the option to draw polygons on top of the geometry, in which they can redefine the Manning's n values within that polygon. This option could be used for calibrating Manning's n values within that region (Inside of the polygon), or it could be used for defining main channel Manning's n values based on user defined regions (polygons). To use this option, draw polygons onto the geometric editor, using the drawing tool labeled **2D Area Mann n Regions**. After drawing the polygon the user is required to give it a unique name. Once the polygons are drawn they will show up in the Manning's n by Land Cover Table. Right now these Manning's n values only work for 2D Flow Areas.

Importing Geometric Data

HEC-RAS has the ability to import geometric data in several different formats. These formats include: a GIS format (developed at HEC); the USACE Standard Surveyor format; HEC-2 data format; HEC-RAS data format; UNET geometric data format; and the MIKE11 cross section data format. Data can be imported into an existing HEC-RAS geometry file or for a completely new geometry file. Multiple data files can be imported into the same geometric data file on a reach-by-reach basis.

Supported File Formats

GIS Format

A file format for interfacing HEC-RAS with GIS/CADD systems has been developed at HEC. A detailed description of the file format is contained in Appendix B of this manual. Chapter 14 of this manual provides detailed discussions on how to import GIS/CADD data into HEC-RAS, as well as how to export computed water surface profiles back to GIS/CADD systems.

USACE Survey Data Format

The U.S. Army Corps of Engineers (USACE) has developed a standard file format for survey data. This format is documented in Chapter 6 of Engineering Manual (EM) 1110-1-1005. The USACE survey format encompasses a wide range of data types. The current version of HEC-RAS has the capability to read this file format, but only cross section data are extracted from the file. At this time all other data are ignored.

HEC-2 Data Format

The HEC-2 program was the predecessor to the HEC-RAS software package. The HEC-2 program was used for many years to compute steady flow water surface profiles. Consequently, thousands of data sets exist in the HEC-2 data format. HEC-RAS has two ways of importing HEC-2 data. The first way is accomplished through the use of the **Import HEC-2 Data** option from the **File** menu on the main HEC-RAS window. When this method is used, it is assumed that the user has started a new project; and therefore all of the HEC-2 data is imported (geometric data, flow data, and plan information). A second way of importing HEC-2 data is provided from the geometric data editor. This way of importing HEC-2 data allows the user to bring the data into existing HEC-RAS geometric data files. This method also allows the user to import multiple HEC-2 data files into the same HEC-RAS geometric data file. However, when importing HEC-2 data from the geometric data window, only the geometric data contained in the HEC-2 files will be imported. All of the other data (flow data and plan information) will be ignored.

HEC-RAS Format

This option allows the user to combine several HEC-RAS geometry files into a single geometry file. For example, if several pieces of a river system were developed as separate HEC-RAS models, this option could be used to put them together into one model.

UNET Geometric Data Format

This option allows the user to import a UNET geometric data file (CSECT geometry file). UNET is an unsteady flow program developed by Dr. Robert Barkau. The Corps, as well as many other agencies, has used this software for many years. UNET models are often very complex, consisting of many river reaches that can be connected in numerous ways. The HEC-RAS UNET importer does not have enough information to draw the schematic in the proper manner. The river reaches and storage areas will be connected correctly, but the user will need to edit the schematic to make it look like the actual river system.

MIKE11 Cross-Section Data

This option allows the user to import cross section data from the MIKE11 program. MIKE11 is a one-dimensional river hydraulics model developed by the Danish Hydraulic Institute. Users must first export the MIKE11 data to a raw text file. This is an available option from MIKE11. Once the data is in the text file format, it can be imported into HEC-RAS.

CSV (Comma Separate Value) Format

This option allows for the import of comma separated value data. It is only intended to import cross section geometry and does not resolve the river network. River system connectivity must be completed by forming junctions after all of the cross section data has been imported.

The data must be in the format of "River Station", "X", "Y", "Z" or "River Station", "Station", "Elevation", as shown in Figure 5-83. Once a file has been selected, you must choose the file format and select the Column Headers that correspond to the HEC-RAS geometry convention. By default, HEC-RAS looks for keywords (such as "River", "Reach", "X", "Y", "Z", "Elevation", etc...) to automatically populate the selected data list item. The River and Reach data is optional – if it is not specified, all cross sections will be assigned to the same river reach.

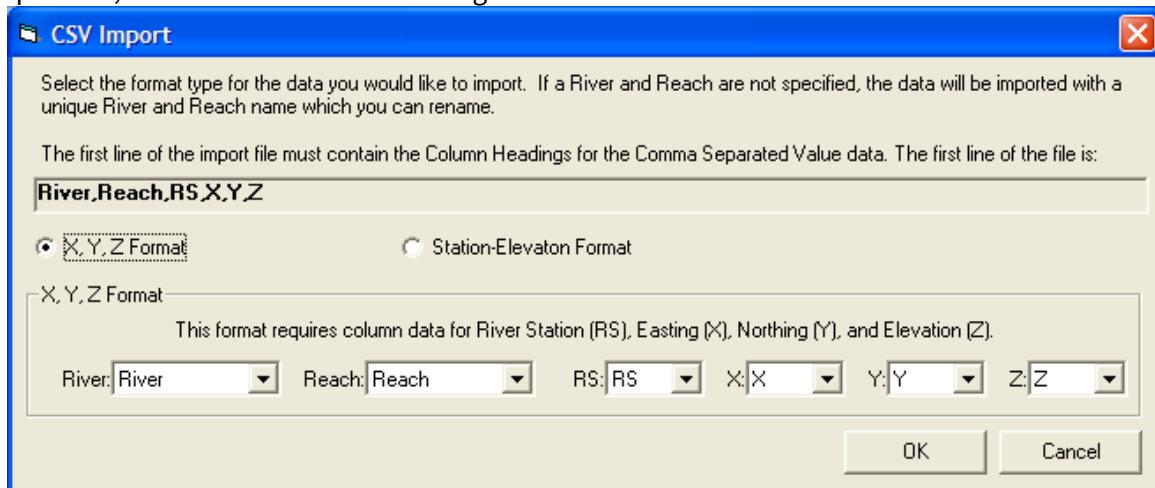


Figure 5 83. CSV data import window.

After identifying the Column Headers, using the dropdown lists, for the data in the CSV file, the standard RAS Geometric Data Import window (Figure 5-82) will appear to guide you through the import process of identifying exactly what data you would like to import. After the cross sections are imported, most likely you will need to adjust the river centerlines and establish connectivity with junctions.

One option for completing the River Network is to create a georeferenced stream centerline (or a stick figure diagram of the system). Then import the RAS cross sections onto the stream network. A detailed discussion on creating a georeferenced stream centerline is provided in a later Section of this document: Example of Georeferencing an HEC-RAS Model.

Geometric Data Importer

To import data into a HEC-RAS geometric data file, the user selects the **File | Import Geometric Data** menu option on the Geometric Data window. Once this option is selected, the user then selects one of the available formats from the list. Once this choice is made, a window will appear allowing the user to select a file containing the data to import. After the user has selected a file, an import window will appear to guide you through the import process.

The Import Options window will guide you through the process of importing all or part of the import file. The initial tab of the Import Options dialog is the Intro tab, shown in Figure 5-79. HEC-RAS will read the import file and look for a "UNITS" tag. Based on the value associated with the tag, you will be offered the option to import the data in the current unit system or to convert the data from one unit system to another. If no unit system is found in the file the import dialog will default to your current RAS project units.

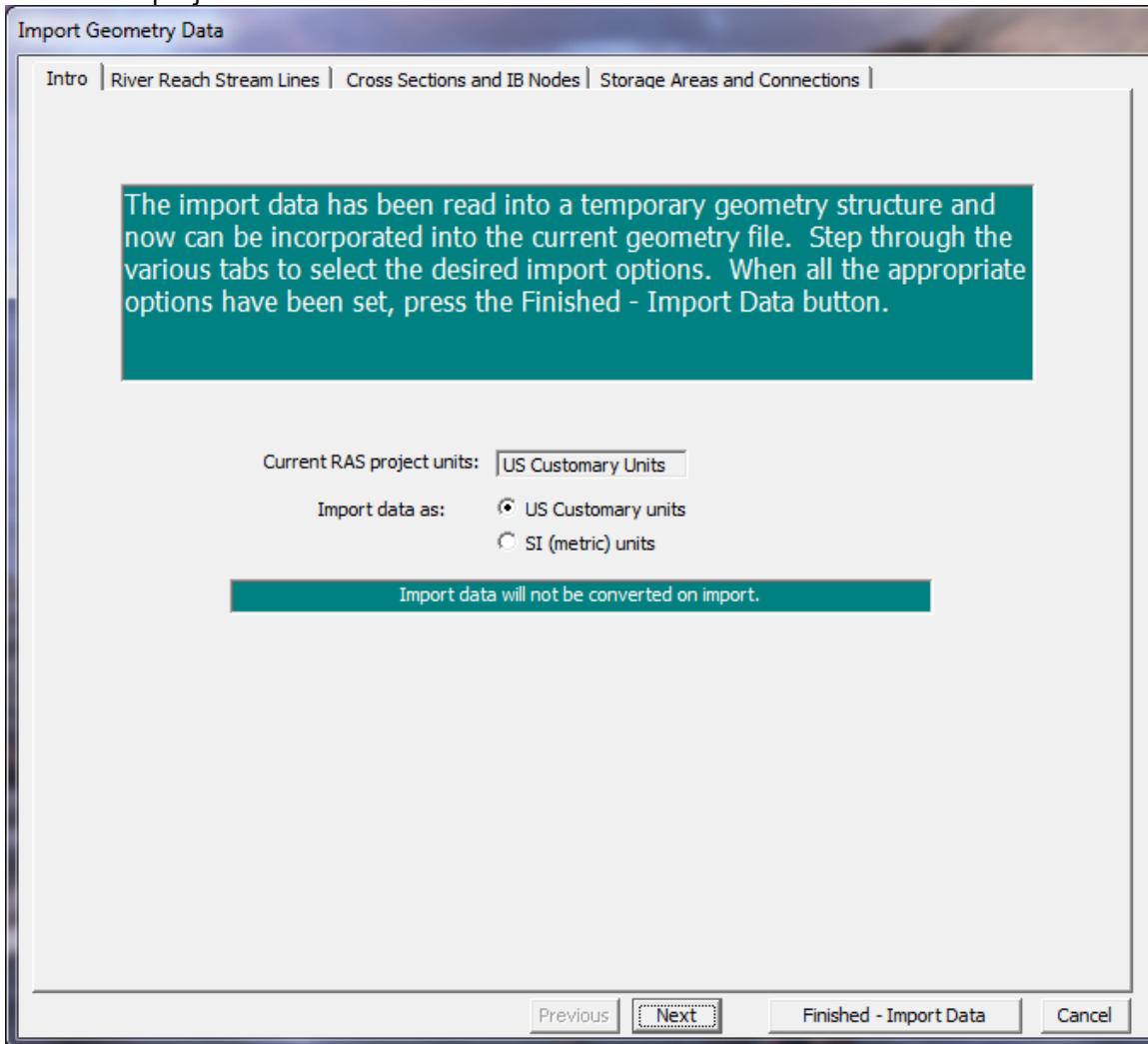


Figure 5 79. Unit system conversion is an import option in HEC-RAS

River Reach Stream Lines

The next tab on the import options window is the River Reach Stream Lines (Figure 5-80). This set of options allows you to specify which river reaches to import, how to import the data, and what to

name the river and reach. Import options for the river and reaches are summarized in Table 5-2.

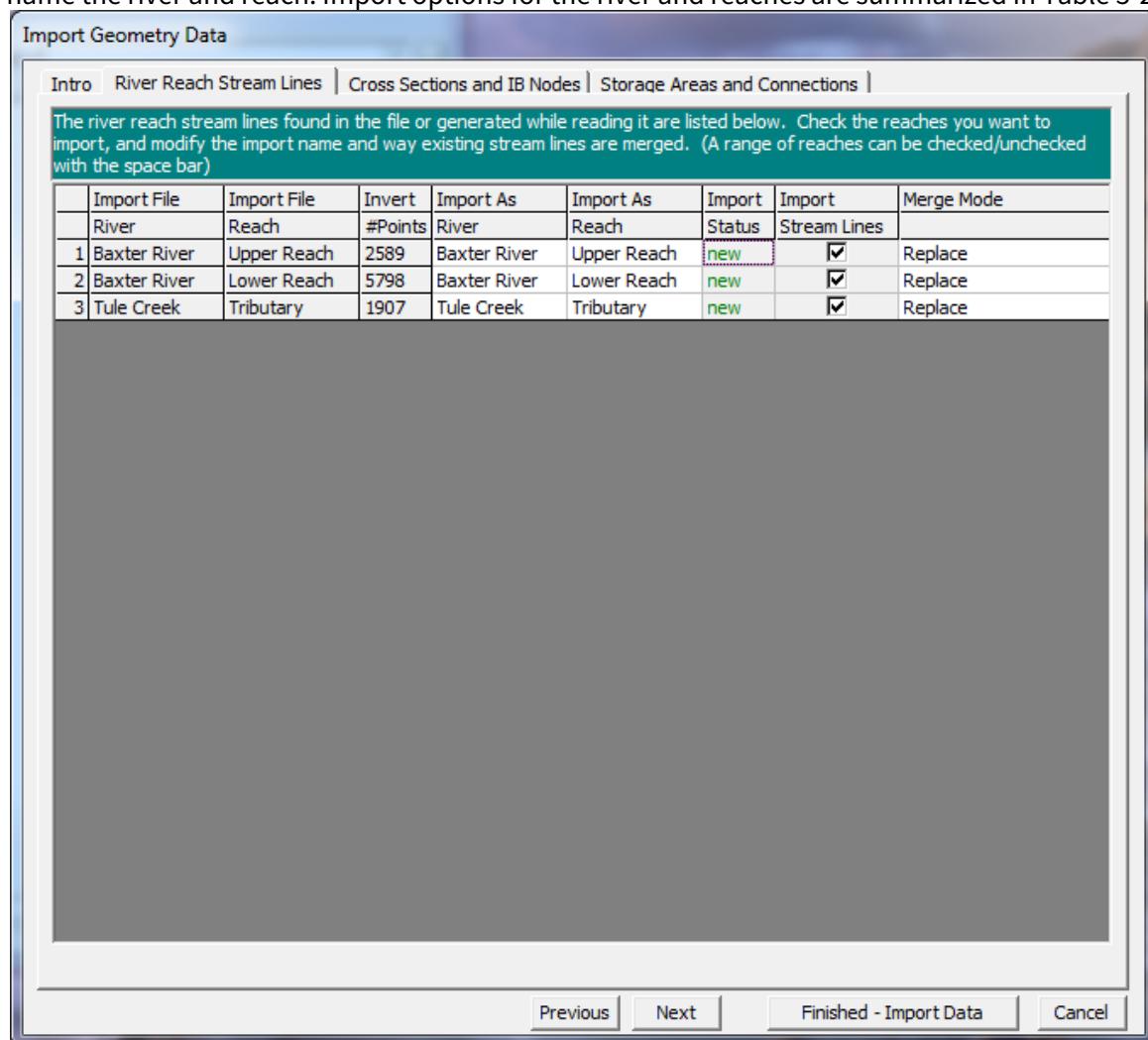


Figure 5 80. River and reach import options.

Table 5-2. Summary of River Reach Import option fields.

Column	Description
Import As River	The name of the River once it is imported to RAS.
Import As Reach	The name of the Reach once it is imported to RAS.
Import Status	Identifies whether the river reach exists in the RAS geometry file or is new.
Import Stream Lines	Checkbox to choose what river reaches to import. Use the spacebar to toggle the checkbox. All rows can be selected by clicking on the column header.
Merge Mode	The river reach can replace existing data, append upstream, or append downstream.

Cross Section and IB Nodes

The next tab on the Import Options window allows you to import cross sections and internal boundaries (bridges and inline structures). The Cross Sections and IB Nodes screen options are shown in Figure 5-81.

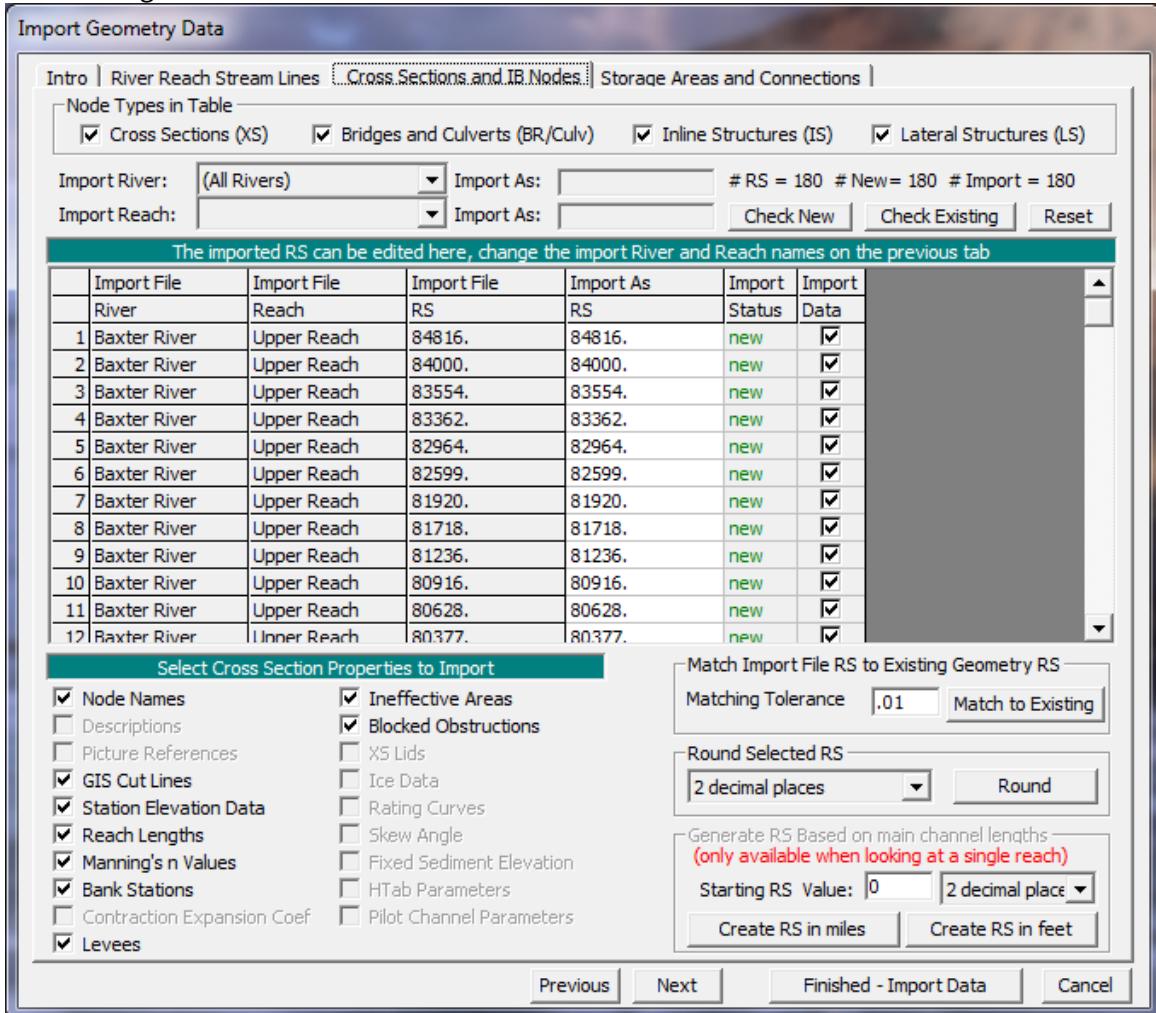


Figure 5 81. Cross section and internal boundary import options.

There are several options for importing cross-sectional data. You must first specify the Import River and Import Reach upon which the cross sections reside. The import dialog will inform you what river and reach name the data will import to (Import As) in the HEC-RAS geometry. (The Import As information was specified on the River Reach Stream Lines tab). You then specify the cross sections to import and the specific cross section properties to import.

Only those cross-sectional properties available from the import file will be available for import. Properties selected will be imported for each cross section specified during the import process. The properties import option will allow you to update individual pieces of data (such as bank station data) without modifying the other data already specified in HEC-RAS.

The cross sections that will be imported and how they will be imported are specified in the import table. Import table options are summarized in the table below.

Table 5-3. Summary of Cross Section and IB Nodes Import option fields.

Column	Description
Import File River	The name of the River in the import file. Refer to the associated <i>Import As</i> field to see the name of the river that the cross section will be imported into.
Import File Reach	The name of the Reach in the import File. Refer to the associated <i>Import As</i> field to see the name of the reach that the cross section will be imported into.
Import File RS	The name of the River Station in the import file.
Import As RS	The name of the River Station the cross section will be imported into. This data may be user-specified and changed using the provided tools. The "Reset" button will replace the river station data with the data in the import file.
Import Status	The Import Status will be "New" or "Exists". New will add the cross section to the data. Exists will update (replace) the existing data with the properties specified.
Import Data	Checkbox to choose what river stations to import. Use the spacebar to toggle the checkbox. All rows can be selected by clicking on the column header. You can also use the buttons provided to select all of the New cross sections (Check New) or those that Exist (Check Existing).

There are also several tools provided to change the river station name. River station identifiers are the link between the GeoRAS generated data and the HEC-RAS data. Cross-sectional river stations must be numbers in HEC-RAS. HEC-RAS will use the river stations (along with River names) for determining the order of cross sections for performing water surface profile calculations. River station numbers must increase in the upstream direction. Import options for river stations allow you to match river stations to the existing geometry, round the river station value for import, and create river stationing.

Match River Stations to Existing Geometry

The *Match Import File RS to Existing Geometry RS* option allows you to specify a numeric tolerance to search for duplicate cross sections in existing geometry files. This tool is useful when you are re-importing cross section data where you may have modified the stream centerline or cross section layout. The newly computed river stations may differ from the original stationing due to small spatial changes made in the GIS. This tool is also convenient if you are updating cross sections that have river stations that were rounded during the initial import of the data.

Round Selected River Stations

GeoRAS may export the river stationing to more decimal places than are necessary. You can round the river stations to the precision appropriate for your study.

Create River Stations

By default, GeoRAS will compute river stations in the unit system of the digital terrain model and will use a zero station at the most downstream end of each river reach. If you wish to change the river stationing you can do so in the GIS, or you can do so during the import process. It is recommended

that you document the method used if you change the river stations. Documenting the method used to compute new river stations will be important if you need to re-import cross-sectional data – the river station identifier is the link between the GeoRAS generated data and the HEC-RAS data.

Storage Areas and Connections

The Storage Areas and Connections tab, shown in Figure 5-82, allows you to specify storage areas and storage area connections to import and what name to import them with.

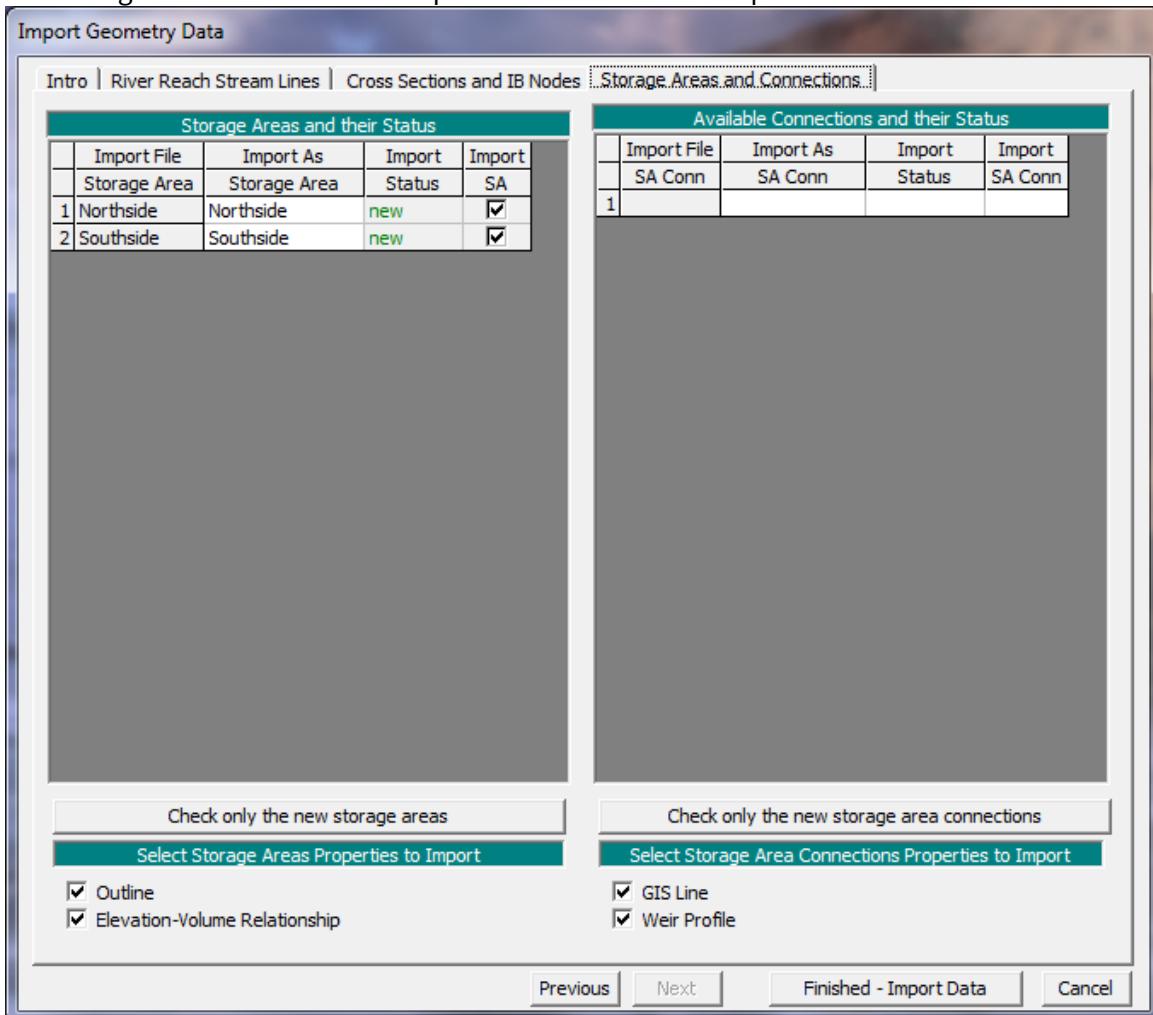


Figure 5 82. Storage Areas and Connections import options.

Geometric Data Tools

Several tools are available from the Geometric Data editor to assist you in the development and editing of data. These tools consist of: cross section interpolation; Channel design/modification; channel modification; graphical cross section editor; channel bank stations; reverse stationing data; set ineffective flow areas to permanent mode; cross section points filter; fixed sediment elevation; pilot channels; and GIS cut line check. The cross section interpolation tool has been described

previously in this chapter. Channel modification is described separately in Chapter 12 of this manual. The following is a short description of each of the tools.

Graphical Cross Section Editor

A graphical cross section editor is available from the **Tools** menu of the Geometric Data Editor window. When this option is selected, a window will appear as shown in Figure 5-84.

The user has the option to move objects (objects are ground points, main channel bank stations, Manning's n value station locations, ineffective flow areas, levees, and blocked obstructions), delete objects, or add new objects. To move an object, the user first selects **Move Objects** from the **Options** menu. Then move the mouse pointer over the object that you want to move, press down the left mouse button, and then move the object. When you are finished moving the object, simply release the left mouse button and the object will be moved. To delete an object, first select **Delete Objects** from the **Options** menu. Next, move the mouse pointer over the object that you would like to delete and click the left mouse button. Whatever object is closest to the mouse pointer will be deleted. To add an object to the cross section, first select the type of object you want to add from the available list under the **Options** menu. Once you have selected an object type to add, move the mouse pointer to the location where you would like to add it and click the left mouse button. If the object that you are adding requires more than one point, such as blocked ineffective flow areas and blocked obstructions then continue to move the mouse pointer and click the left mouse button to add the additional points.

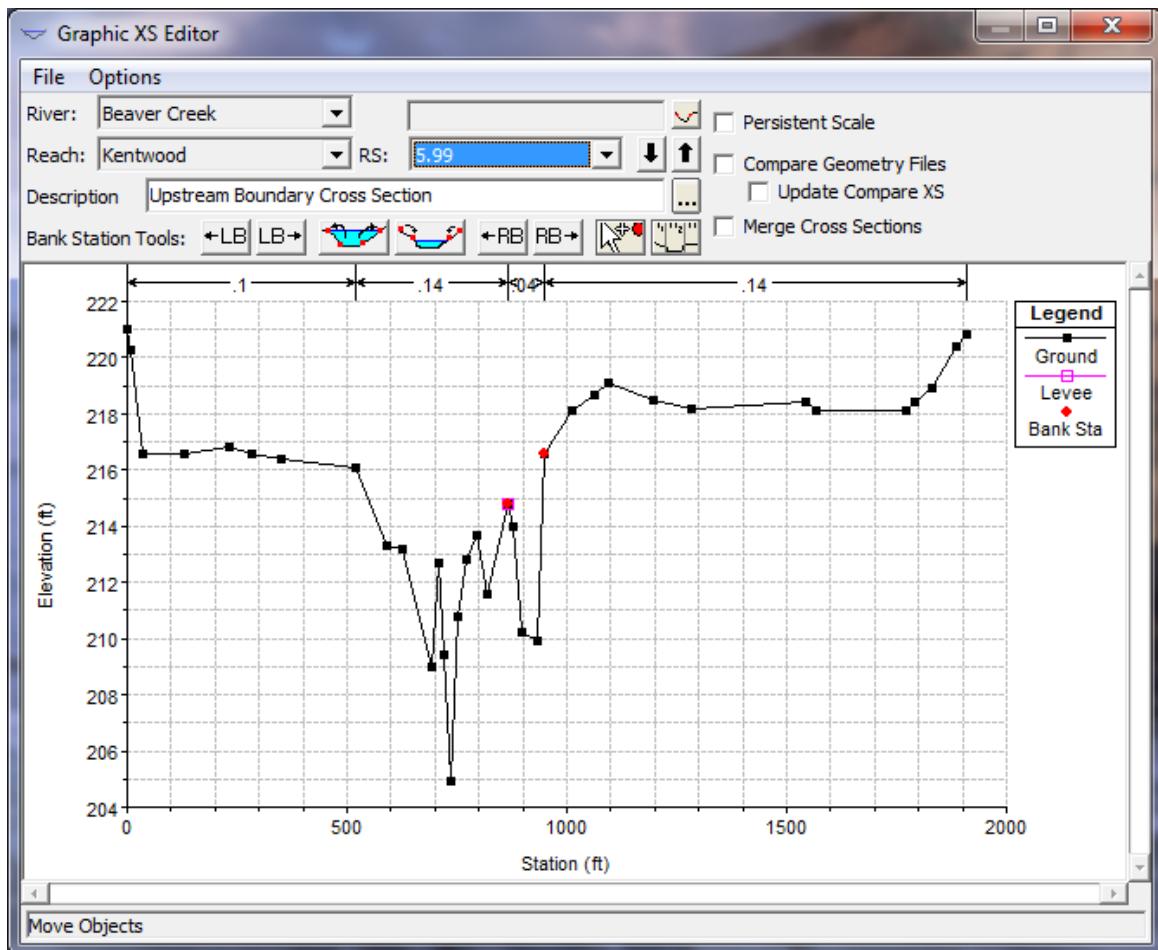


Figure 5.84. Graphical Cross Section Editor

Other available options from the Graphical Cross Section editor are the ability to zoom in and zoom out, full plot, pan, overlay a grid onto the cross section plot, and to undo all of the graphical editing. When the **Undo Edits** option is selected, the cross section is automatically returned to its original state before this particular editing session began. However, once this editor is closed, or if the user selects a different cross section from the editor, it is assumed that the user is happy with the changes that were made and they are saved in memory. The data is not saved to the hard disk, so it is still possible to get the original data back if needed.

Bank Station Tools

Several tools have been added to make it convenient to change the location of the main channel bank stations. These tools are in the form of buttons across the top of the graphic window displaying the currently opened cross section.

Tool	Description
LB	Allows the user to move the left bank of the main channel one point at a time to the left. Simply click this button to have the left main channel bank station move one point to the left.

Tool	Description
 LB→	Allows the user to move the left bank of the main channel one point at a time to the right. Simply click this button to have the left main channel bank station move one point to the right.
	Allows the user to move the main channel bank stations to cross section points that are just above the currently displayed water surface profile. The user can turn on any of the previously computed water surface profiles to be displayed on the plot. Once a water surface profile is displayed, and the user presses this button, the main channel bank stations will be move to the first two points that are above the water line on the plot. This option is very useful if you would like to set the main channel bank stations to a consistent stationing based on a particular flow event. For example, if a water surface profile is computed for the 2-year flow event, the user could then move all of the main channel bank stations just outside of this flow profile.
	This tool is very similar to the previously described tool. It does the same thing, only moving the main channel bank stations inward to just above the water line (assuming the current main channel bank stations are above the water line).
 ←LB	Allows the user to move the right bank of the main channel one point at a time to the left. Simply click this button to have the right main channel bank station move one point to the left.
 RB→	Allows the user to move the right bank of the main channel one point at a time to the right. Simply click this button to have the right main channel bank station move one point to the right.
	Allows the user to set the left or right bank of the main channel to a stationing of one of the existing points in the cross section. Once this button is pressed, the user can set the stationing of the left and right bank of the main channel by simply moving the mouse to the desired location and clicking the left mouse button. The program assumes the left bank when the mouse is left of the lowest point in the view area, and it assumes the right bank when the mouse pointer is right of the lowest point in the view area.
	Allows the user to move the station locations of existing Manning's n values. When this option is selected, vertical lines will appear at all of the current Manning's n value (or K-value) break points. The user can move the mouse pointer over the n value break locations at the top of the graphic, press and hold the left mouse button down, and then move the n value break location to the new desired location.

Persistent Scale

This option allows the user to set a persistent scale, in both the vertical and horizontal, to be used when plotting any cross section. When this option is checked, additional data fields will be displayed to the right of the option. The user has the option to set a left and right stationing, or a maximum width for the X-axis. Likewise, a top and bottom elevation, or a height can be set for the Y-axis. By setting a persistent scale, as the user moves from one cross section to the next, it is much easier to visualize how cross sections are changing from one to the next.

Compare Geometry Files

This option allows the user to compare cross sections from two different geometry files (one being the currently opened geometry file). When this option is selected, additional data fields and buttons will show up to the right of this option. The user first selects the second geometry file to use for comparing to the currently opened geometry file. Next the user selects the specific river, reach, and

river stationing to plot against the currently displayed cross section. The second geometry, which will be displayed in pink on the graphic, is only for visualization, as it cannot be modified by the graphical editor.

Merge Cross Sections

This option allows the user to merge data from one cross section into another. This option works in conjunction with the Compare Geometry Files option described above. To use this option, the user must first turn on the compare geometry files option and select the desired geometry file and cross section to compare to the current geometry file and cross section. Next the user turns on the **Merge Cross Sections** option. When this option is selected some additional data fields will show up next to the Compare Geometry Files data fields. Additionally two red vertical lines will show up on the plot, defaulted to the main channel bank stations of the existing cross section. An example of what the Graphical Cross Section Editor will look like when the two options are turned on is shown in the figure below.

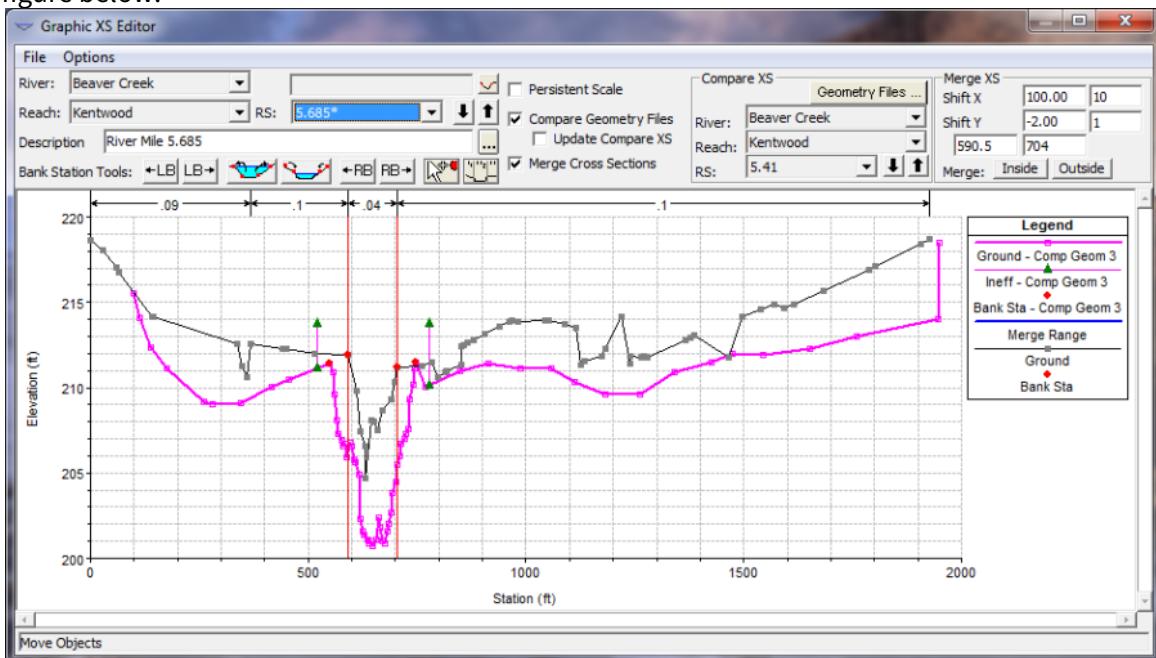


Figure 5.85. Graphical Cross Section Editor with Merge Cross Section Option.

As shown in the Figure above, the two red lines indicate the zone for merging data. The merge data zone can be everything inside of the two lines, or everything outside the two lines. The merge data zone can be changed by either graphically grabbing one of the vertical lines or moving it, or by entering a value in the appropriate fields under the Merge XS data area at the upper right hand corner of the window. Before the data is merged, the user may want to shift the comparison cross section (shown in purple) left or right, and/or up and down, to align the appropriate portion of the cross section with the appropriate portion of the current cross section (shown in black). Shifting the comparison cross section can be accomplished by entering a Shift X and/or Shift Y value into the appropriate fields in the upper right corner of the window. Additionally the cross section can be shifted by holding down the Shift key and then pressing the arrow keys. The amount of shift for each press of an arrow key can be controlled by entering a value for the X and Y shift amount in the fields in the upper right corner of the window. Once the merge zone has been set, and the comparison cross section has been shifted appropriately, the user can press either the button labeled **Inside** or

Outside. If the button labeled **Inside** is pressed, then the data for the current cross section, inside of the merge zone (between the two red lines), is replaced with the data from the comparison cross section. If the button labeled **Outside** is pressed, then the data outside of the two red lines is replaced (i.e. the data from the current cross section, outside of the red lines, is replaced with the data from the comparison cross section).

Channel Bank Stations

This tool allows the user to select a water surface profile from a previous run, and then have the program move the main channel bank stations to the station/elevation points closest to the edge of the water surface.

Reverse Stationing Data

Cross section data should be entered into HEC-RAS from left to right when looking downstream. This is the assumed direction for all of the cross sections and other structure data. If you have data that has not been entered from left to right while looking downstream, this editor will allow you to reverse the data to the assumed direction. To bring up this editor, select **Reverse Station Data** from the **Tools** menu of the geometric data editor. When this option is selected a window will appear as shown in Figure 5-86.

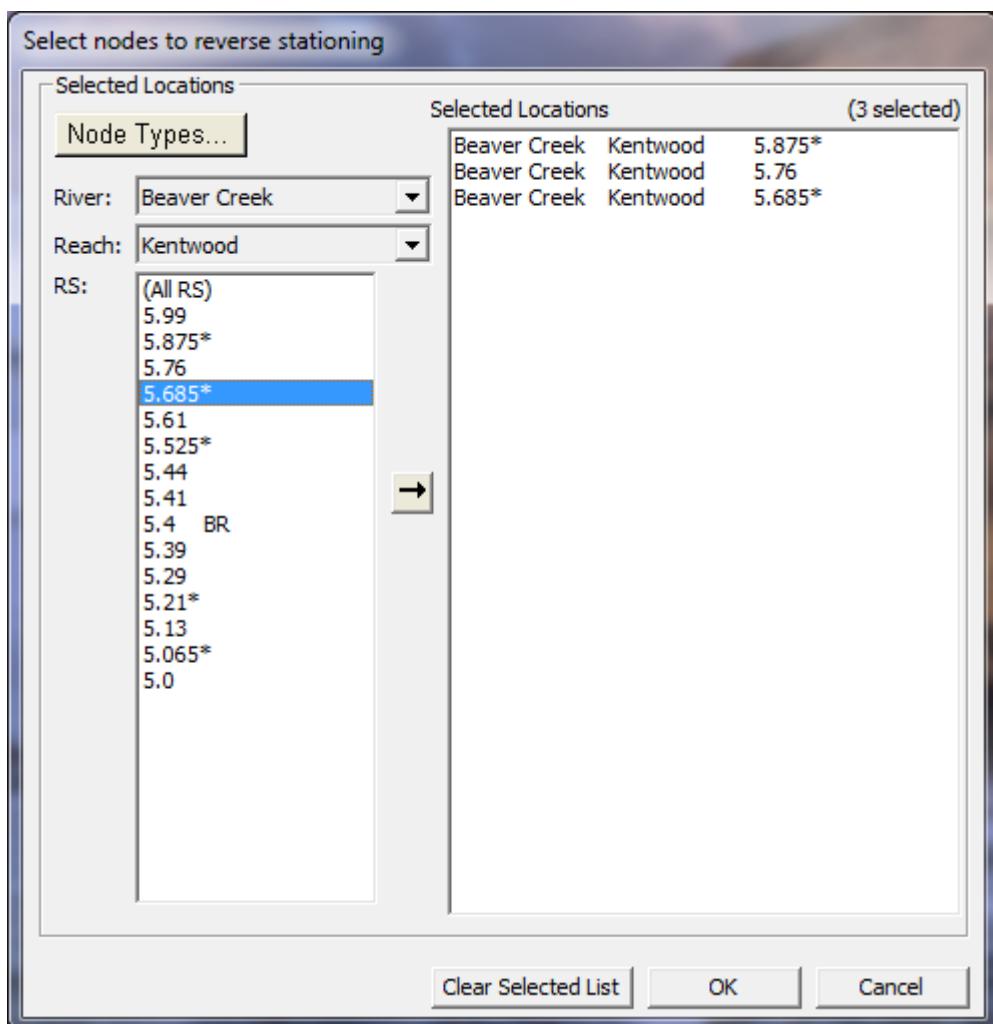


Figure 5 86. Reverse Cross Section Stationing Editor

As shown in Figure 5-86, you first select the river and reach which contains the data to be reversed. Then select the particular river stations of the data that is not in the correct format (left to right looking downstream). Add those locations to the box on the right side of the editor, by pressing the arrow button in the middle of the editor. Continue to do this until you have all of the cross section that you want to reverse the stationing for. Finally, press the **OK** button and the data will be reversed.

Cross Section Points Filter

This tool allows a user to filter out unnecessary points in cross sections. With the use of GIS data, cross sections can contain many more points than actually necessary to describe the terrain. HEC-RAS has a limit of 500 points in any cross section. Because of this limit, it is occasionally necessary to filter out points that are not needed. To bring up this editor, select **Cross Section Points Filter** from the **Tools** menu of the geometric data editor. When this option is selected a window will appear as shown in Figure 5-87.

As shown in Figure 5-87, the editor allows the user to filter points on a cross section by cross section basis, or for a range of cross sections at one time (Multiple Locations option tab). To filter a single cross section, the user selects the river, reach, and river station they want to work on. Then press the button labeled **Filter Points on Selected XS** to filter the points.

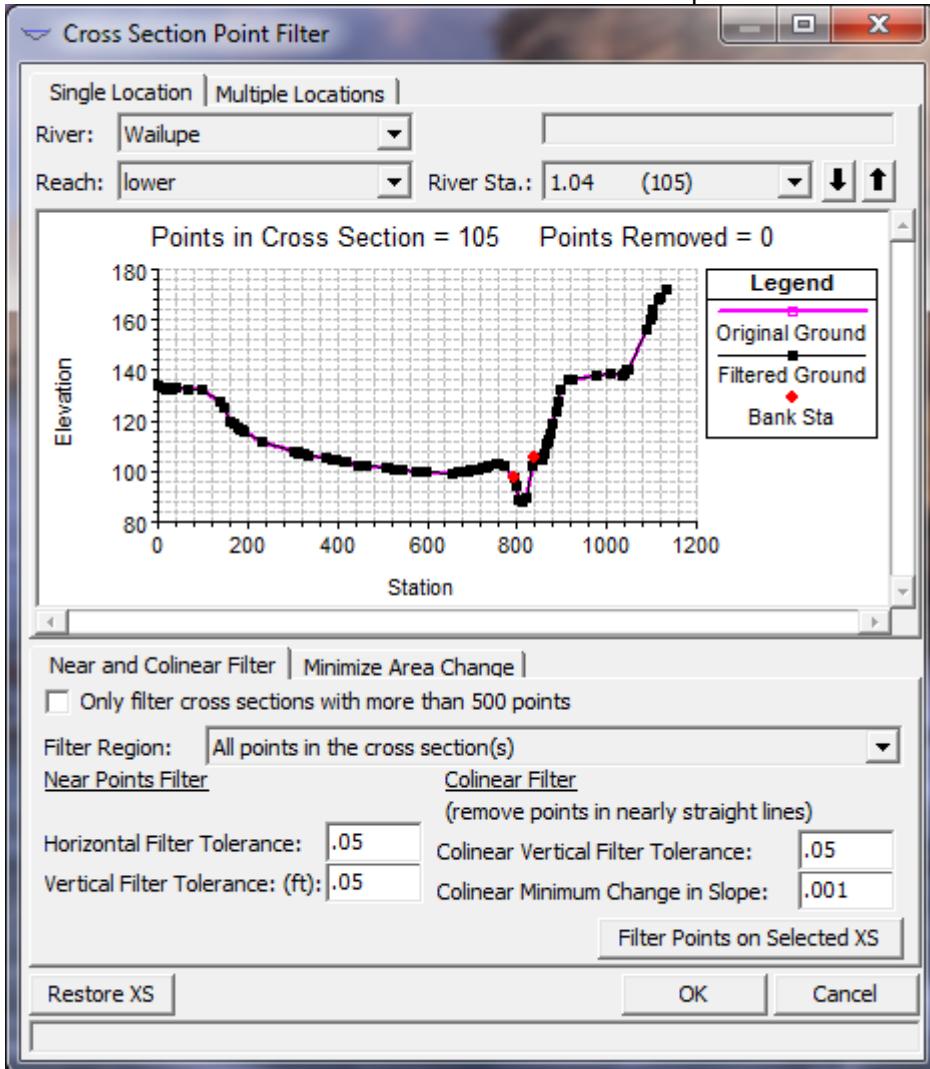


Figure 5 87 Cross Section Points Filter Editor

The cross section points filter performs two different types of filtering on each cross section. The first type is called a **Near and Colinear Points Filter**, this method simply searches for points that are close together. If two points are found to be within the horizontal and vertical distance tolerance, then the second point is removed. The second pass of this filter searches for points that are in a straight line, or nearly in a straight line. This filter searches to find three consecutive points that may be in a straight line. If a line is connected between points one and three, and point two is less than a predefined tolerance from that line (vertical filter tolerance based on a distance perpendicular to the line), then the second point is a candidate to be removed. A second check is done to ensure the slope of the line that connects point one and two together, is not changing significantly when point one and three are connected (minimum change in slope tolerance). Options are available to only filter cross sections that have more than 500 points, as well as to restore a cross section back to the original points before filtering occurred.

The second type of filter is called **Minimize Area Change**. To use this filter press the Minimize Area Change tab below the graphic window. When this tab is selected the user will be asked to enter the number of points that they would like the new cross section to be trimmed down to. After a number is entered, the user presses the **Filter Points on Selected XS** button to perform the filtering. This filtering method will drop out one point at a time until the cross section is down to the user desired number of points. The decision process for dropping a point is to find the point in the cross section that will cause the area of the cross section to change the least.

Additionally, this editor allows the user to select multiple cross sections and perform the filter operation on all of them at once. This is done by first selecting the **Multiple Locations** tab. Then select the cross sections that you would like to filter. Set the filter tolerances to any desired values, and then press the **Filter Points on Selected XS** button.

Fixed Sediment Elevations

This option allows the user to fill in portions of cross sections with sediment. The sediment is assumed to be at a constant elevation in any particular cross section. To use this option select Fixed Sediment Elevations from the Tools menu of the geometric data editor. When this option is selected, a window will appear as shown in Figure 5-88.

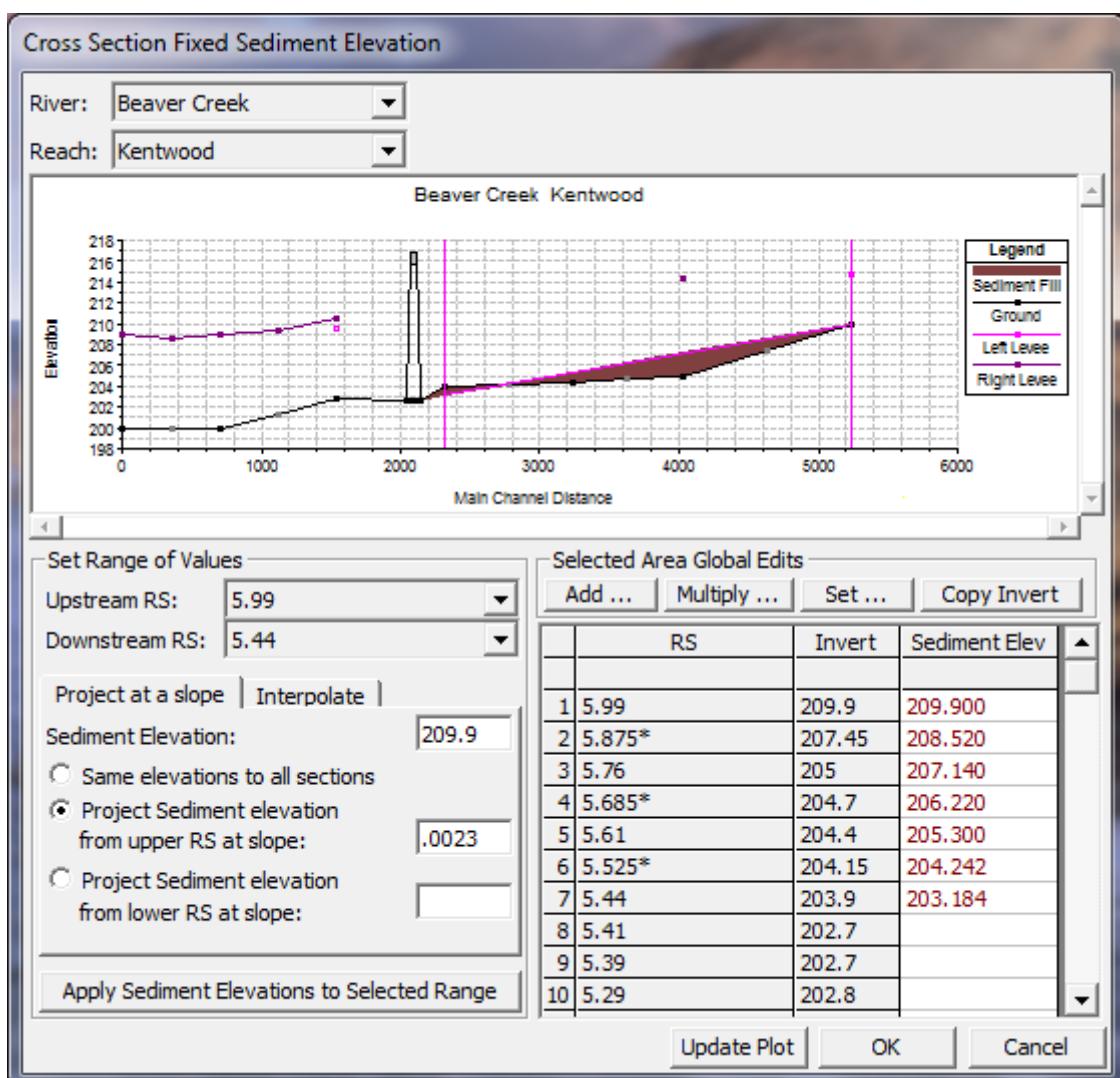


Figure 5-88. Fixed Sediment Elevation Editor

As shown in Figure 5-88, the user selects a particular river and reach to work on, then a range of cross sections to apply the sediment fill to. There are three options for having a sediment fill over a range of cross sections. The first option is to enter a sediment elevation at an upstream or downstream cross section then project the sediment fill on a slope over the range of selected cross sections. The second option is to set the upstream and downstream elevations, then allow the program to use linear interpolation for the cross sections in between. The final option is to set the sediment elevation individually on a cross-section by cross-section basis.

The lower left hand portion of the editor is used to set the sediment values over a range of sections. The table on the lower right hand side of the editor shows the actual values that are applied to each cross section. The user can change any value in the table directly, or they can highlight a section of values and use the four buttons above the table to modify the values. These four buttons allow for adding a constant; multiplying the values by a factor; setting all of them to a specific value; or setting them to the cross section invert.

Pilot Channels

Pilot channels are an option that was added for unsteady flow modeling. Occasionally, when modeling low flows (such as at the beginning or end of a storm event), the program will go unstable. This instability can occur for many reasons. The following is a list of some of the main causes for instabilities at low flows:

1. At low flows the depths are very small. As the flood wave begins to come into the reach, the depths change dramatically percentage wise. Unsteady flow models use derivatives that are based on the change in depth with respect to time and distance. If the depth changes significantly during any time step, the derivatives can become very large, and oscillations will occur. These oscillations can grow to the point where the solution becomes unstable.
2. Also during low flows, it is much more likely that your river may be flowing in a pool and riffle sequence. At the riffles, the flow may be passing through critical depth and going supercritical. By default, the unsteady flow solver in HEC-RAS cannot handle flows going down to or passing through critical depth (unless the mixed flow option is turned on). This again causes instabilities in the solution, and may eventually cause the solution to go unstable.

Pilot channels are one of the available options to help prevent the model from going unstable. A pilot channel cuts a rectangular notch into the bottom of the cross section. Generally this notch is not very wide (often 1 ft is used), but it provides depth to the cross section at low flows (typically make it 5 to 10 feet deep). Additionally, the use of a pilot channel can smooth-out irregularities in the channel bottom. This also helps the stability of the model solution. The pilot channel area and conveyance are barrowed from the lower portion of the main channel, such that the total area and conveyance properties of the cross section relate to the original cross section at higher flows. In other words, when the depth of flow gets higher, the area and conveyance of the pilot channel are ignored. To use the pilot channel option, select **Pilot Channel** from the **Tools** menu of the geometric data editor. When the pilot channel option is selected a window will appear as shown in Figure 5-89.

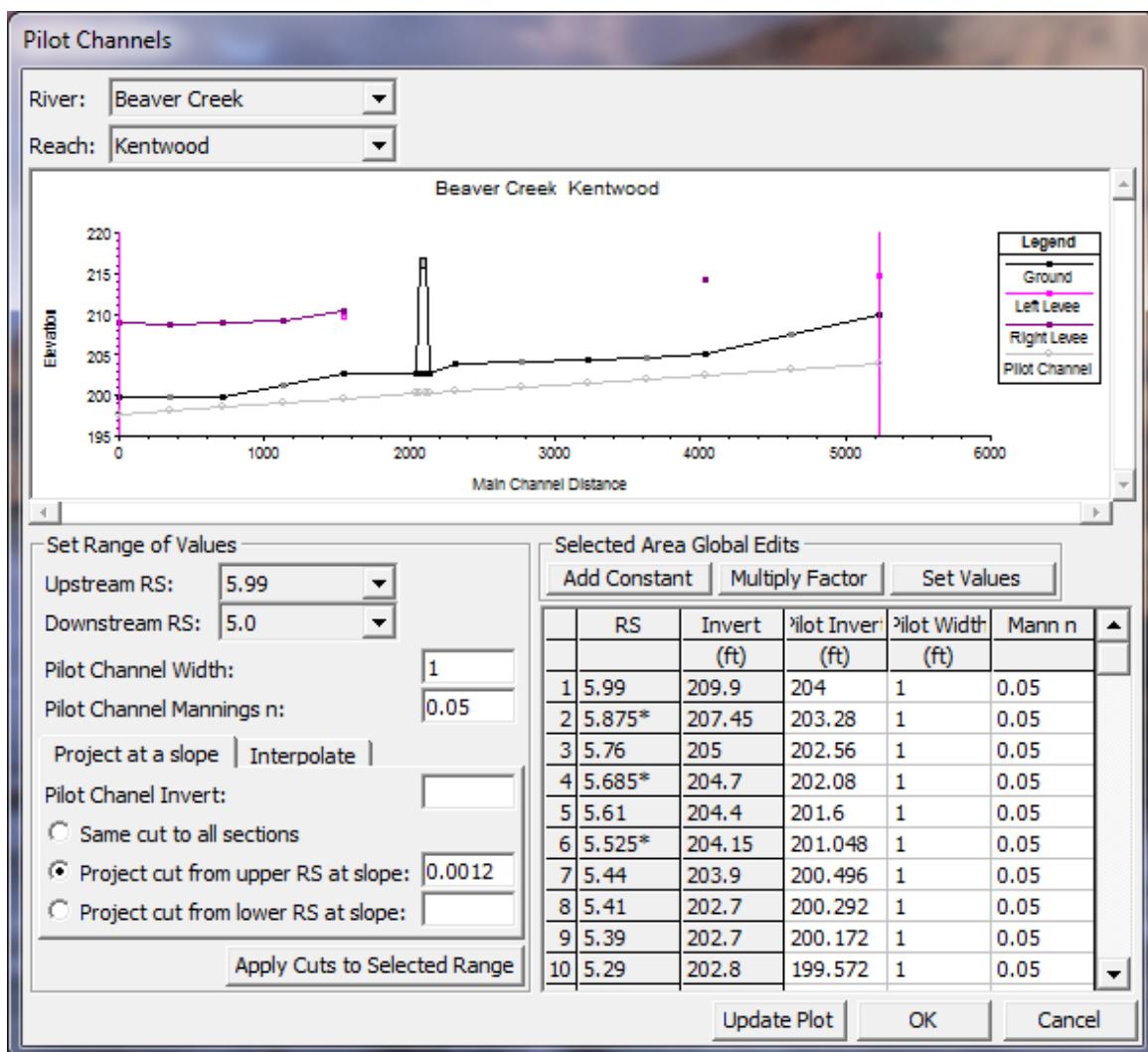


Figure 5.89. Pilot Channel Editor

As shown in Figure 5.89 the user selects a river, reach, and range of river stations to apply the pilot channel too. On the lower left hand side of the form are some utilities to enter the pilot channel information. The user enters the pilot channel width (typically the width should be narrow), and the Manning's n value (should be equal to or higher than the main channel n value). The user can either enter an elevation for the invert of the pilot channel and project it on a slope over the range of cross sections, or they can enter an upstream and a downstream invert elevation and have the program use linear interpolation for the cross sections in between. A list of the final pilot channel values for each of the cross sections is shown in the table on the lower right hand side of the editor. The user can modify the table directly and change any value on a cross section-by-cross section basis. The profile plot on the editor will display the invert elevation of the pilot so you can compare it to the actual channel invert. Once you have finished adding the pilot channel information, press the OK button, and then save the geometric data.

Ineffective Areas - Set to Permanent Mode

The default method for ineffective flows is that the area defined as ineffective will contain water but have no conveyance (the velocity is assumed to be zero). This remains true until the water surface reaches a trigger elevation (an elevation set by the user, as to when the ineffective flow area should become effective again). Once the water surface is higher than the trigger elevation, the entire ineffective flow area becomes effective. Water is assumed to be able to move freely in that area based on the roughness, wetted perimeter and area of each subsection.

Occasionally you may have a need to have these ineffective flow areas remain ineffective permanently. The ineffective flow areas can be set to the permanent mode individually from the cross section editor, or through a table from the geometric data editor. To bring up the table, select **Set Ineffective Areas to Permanent Mode** from the **Tools** menu of the geometric data editor. When this option is selected a window will appear as shown in Figure 5-90.

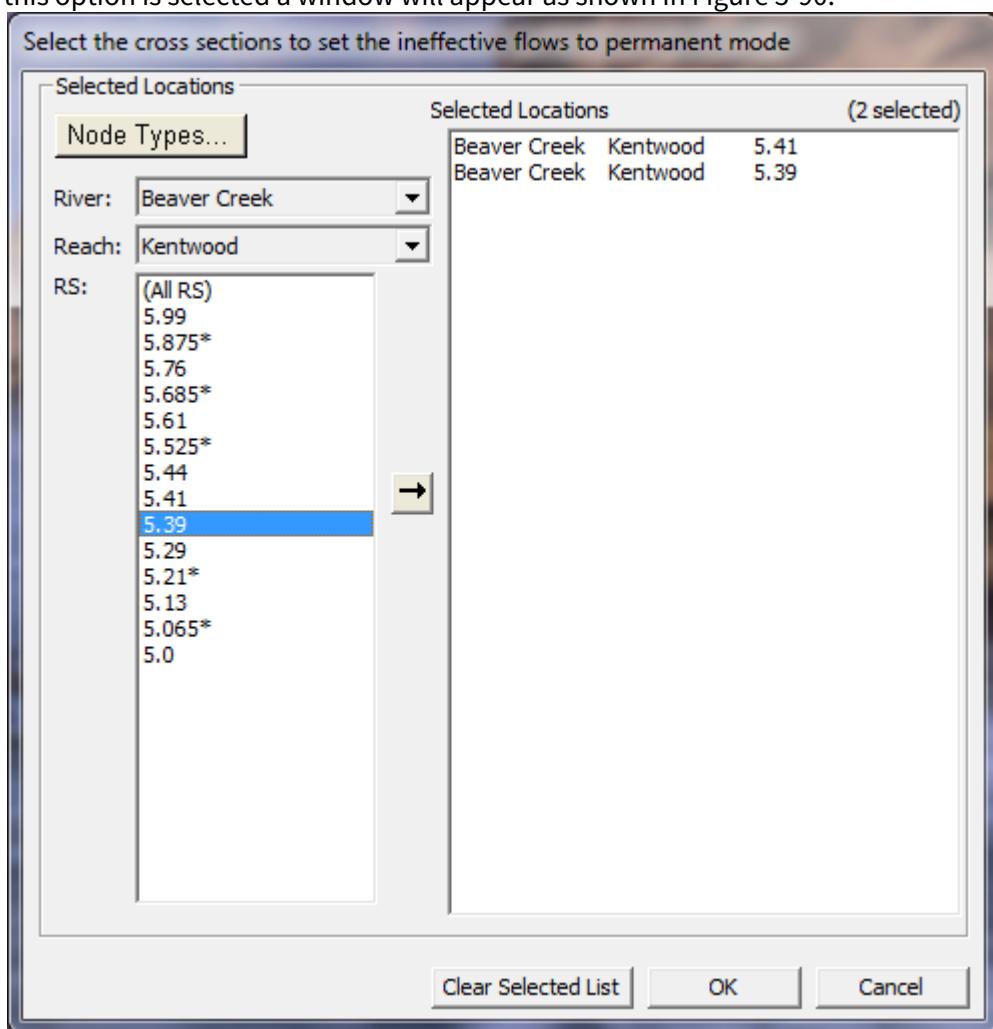


Figure 5 90. Editor to Set Ineffective Flow Areas to Permanent

The editor for this option allows the user to select the river, reach, and river stations, of the cross sections in which you want to set the ineffective flow areas to the permanent mode. Add those locations to the box on the right side of the editor, by pressing the arrow button in the middle of the

editor. Continue to do this until you have all of the cross section that you want. Finally, press the **OK** button and the data will be reversed.

Ineffective Areas – Fix Overlapping

This tool will search through all of the cross sections and check to see if the user has entered blocked ineffective areas that overlap each other. The computational program does not like to have overlapping ineffective flow areas so this must be rectified before the computations can begin. This tool will find all the cross sections that have overlapping ineffective flow areas and it will adjust the stationing of the block with the lower elevation so it does not overlap the adjacent ineffective area.

Ineffective Areas – Convert Multiple Blocks to Single “Normal” Ineffective

This tool will convert cross sections with multiple blocked ineffective flow areas to a cross section with the "Normal" ineffective flow areas (just one left and one right ineffective flow area in the overbanks).

Manning's n Set Channel to Single Value

This option allows the user to select a set of cross sections in which the Manning's n value will be set to a single value. This option will change any cross section that has more than one n value inside of the channel to a single value. The n value at the low point in the main channel is what is used for the entire channel.

Vertical Datum Adjustment

These tools allows the user to adjust the elevation data of the entire model or selected nodes (cross sections, bridges/culverts, structures, etc...), storage areas, storage area connections, or pump stations contained in the project. When this option is selected a submenu will come up that allows the user to select to adjust the **Entire Geometry** model with a single value; define a **Table** of locations and adjustment values; or select various types of nodes (cross sections, bridges, culverts, etc...) in the project. Once an option is selected a new window will appear allowing the user to enter a factor(s) to multiply or add to all of the elevations of the selected nodes.

Reach Connectivity

This tool allows the user to view how reaches are connected together. When working with a very large model with lots of reaches, it can often be difficult to see how the reaches are connected, or if there is a problem with a connection. This tool displays the connections in a text table.

Reach Order for Computations

When this option is selected a window will pop up showing the order of the reaches for computations (see Figure 5-91). The reaches are listed from upstream to downstream order. For complex models the reach order is very important. The HEC-RAS software will automatically compute the order for computations. However, the user can change the order if they do not like the order that the program came up with. **Warning:** changing the computational order for reaches can cause the computational programs not to work correctly. Before changing the computational order,

be sure you fully understand what the steady flow and unsteady flow computational modules require for computational orders.

Cut, Paste and Move to Complete List		Move:
	Default Order	User Specified Order
1	Audubon 1	17th Street 2
2	Claiborne 2	17th Street 3
3	ClaiTo17th 1	Wash-Orleans 1
4	17th Street 3	ClaiTo17th 1
5	Claiborne 1	Audubon 1
6	Napoleon 2	Washington 1
7	Napoleon 1	Washington 2
8	Wash-Orleans 1	Napoleon 1
9	Washington 2	Napoleon 2
10	Washington 1	Claiborne 1
11	17th Street 2	Claiborne 2

Figure 5.91. HEC-RAS Reach Computational Order Table

Reach Order – Find Loops

When this option is selected, the program will search through the model schematic and find any loops in the system that will cause the backwater computations to fail. This can occur if the user puts in a looped system that would continue to loop water.

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 **Tools** menu of the Geometric Data editor. When this option is selected, a window will appear as shown in Figure 5-92.

As shown in Figure 5-92, 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.

Note: 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.

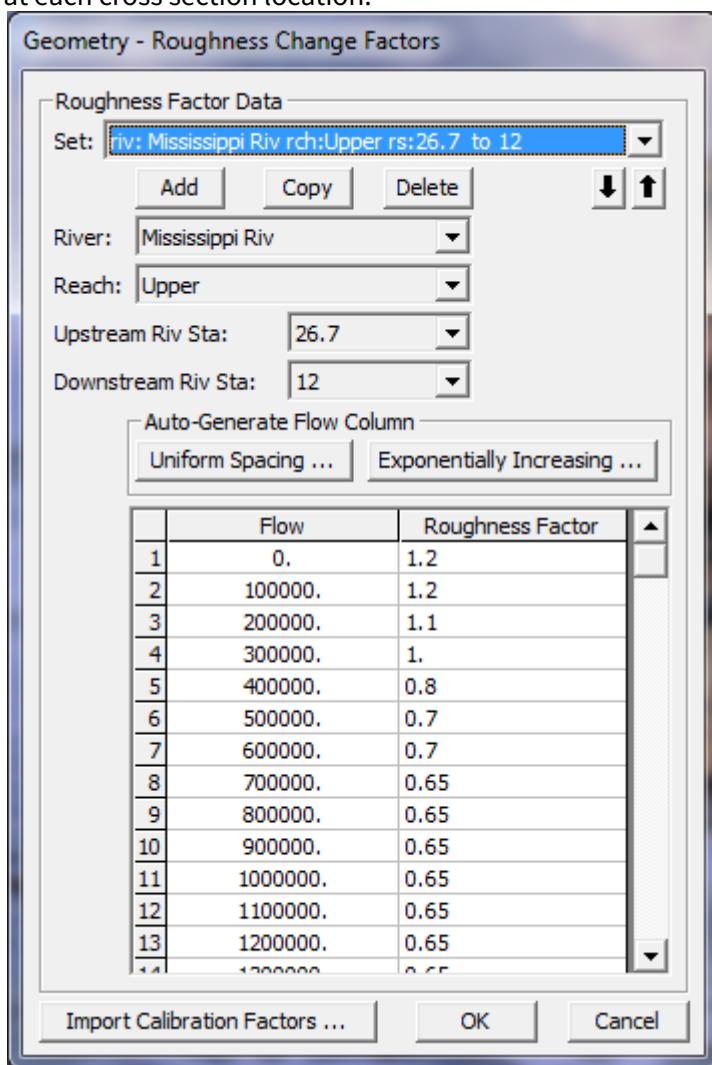


Figure 5 92. Flow versus Roughness Change Factors Editor

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 **Tools** menu of the Geometric Data editor. When this option is selected a window will appear as shown in Figure 5-93.

As shown in Figure 5-93, 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.

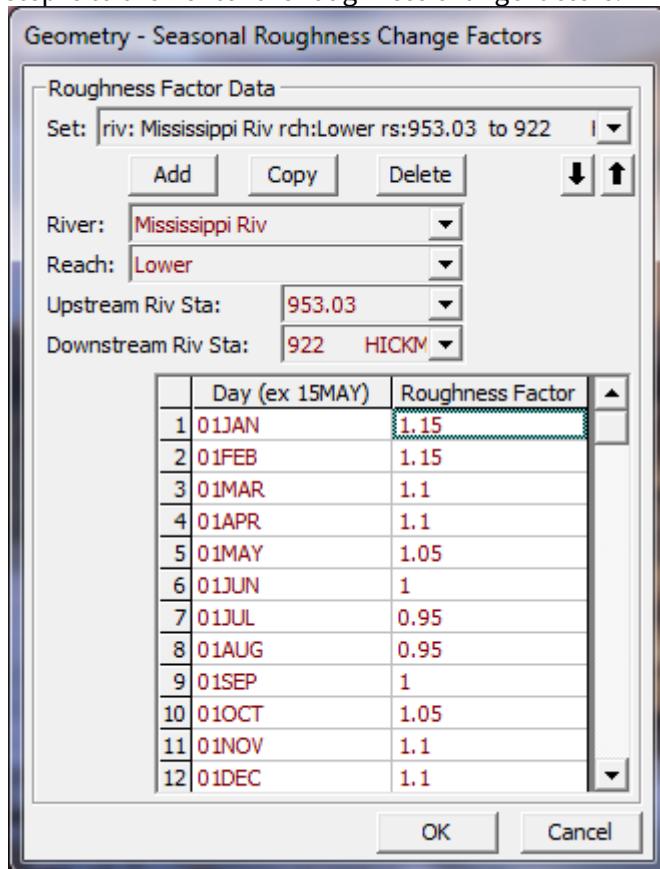


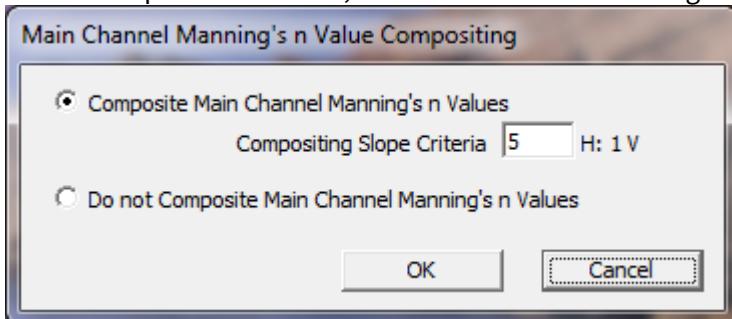
Figure 5 93. Seasonal Roughness Factors Editor

Geometric Data Options

A few new options have been added to the HEC-RAS Geometric Data editor. To select one of the options go to the **Options** menu at the top of the Geometric Data Editor, and select the desired option. The following is a list of the currently available options:

Main Channel Manning's n Value Compositing

When this option is selected, the window shown in the figure below will be invoked.



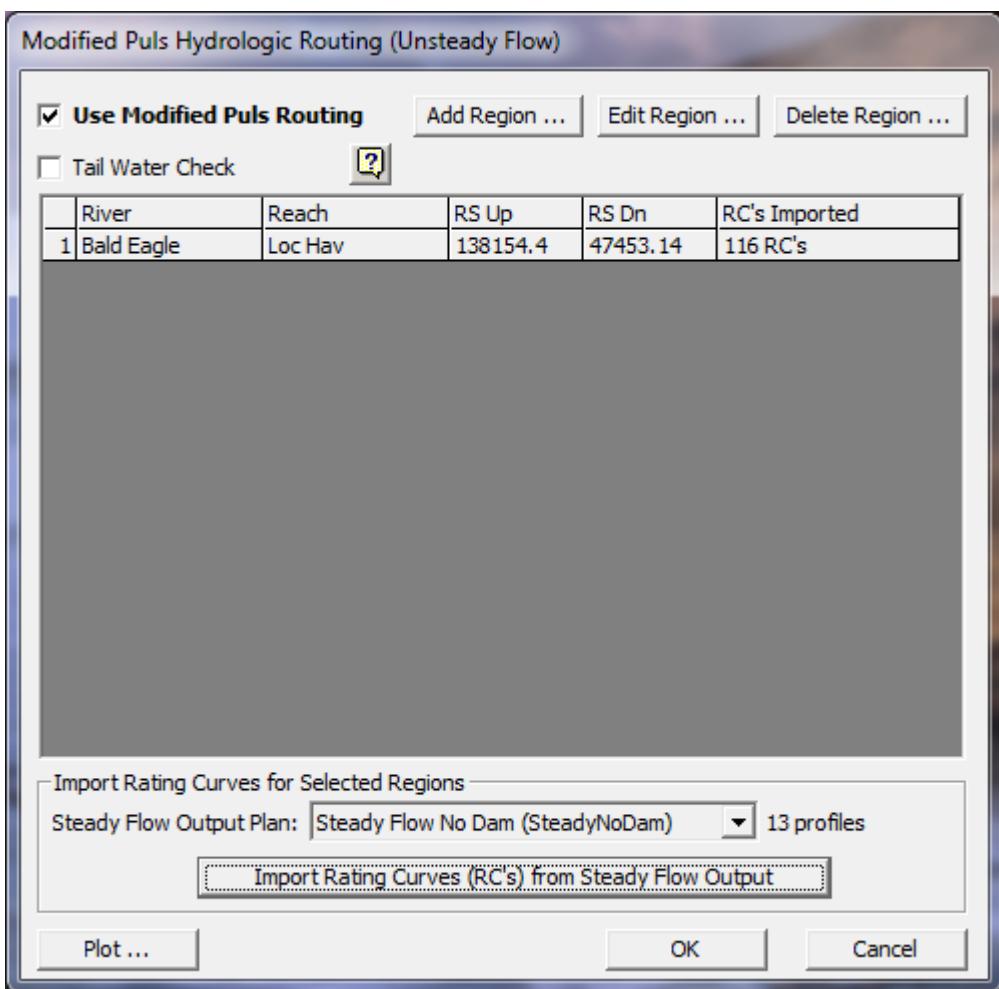
Main Channel Manning's n Value Compositing Editor

This editor allows the user to control how HEC-RAS will composite Manning's n values for the main channel only. The default option is the top option. This option will composite all of the main channel manning's n values into a single n value, as long as the side slopes of the main channel are greater than 5H:1V. The user has the option to change this slope criterion. The second option is to tell HEC-RAS to not composite Manning's n values for any of the cross sections in the model. For more details on Manning's n value compositing for the main channel, see the Hydraulic reference manual.

Hydrologic Unsteady Routing

This option allows the user to define portions of a model to be routed with a hydrologic routing technique instead of using the full unsteady flow equations. The software will simultaneously solve the unsteady flow equations and the hydrologic routing reaches each time step. This option is very useful when encountering portions of the model that are very steep and full unsteady flow routing is either unstable or not possible at all. Currently the only hydrologic routing method available is Modified Puls routing. This option only works as part of an unsteady flow model, and is ignored when using a geometry file in steady flow mode.

To use the hydrologic routing option, go to the **Options** menu at the top of the Geometric Editor and select the option called **Hydrologic Unsteady Routing**. When this option is selected the following window will appear.



Hydrologic Unsteady Flow Routing Editor

To use the Modified Puls routing option, the user must first create a steady flow plan with the exact same geometry file. The purpose of the steady flow plan is to compute a range of water surface profiles from very low to the highest expected flow rate. The results from the steady flow run are used within the hydrologic routing reaches in order to provide the necessary discharge-volume relationships required by Modified Puls routing. As shown in the figure above, a Steady Flow Output Plan must be selected at the bottom of the editor, which will be used for importing the computed rating curves. Based on the computed rating curves from the steady flow run, and the known distances between cross sections, the program can compute a volume for any flow rate on the fly, in order to solve the Modified Puls equations.

Users can establish hydrologic routing reaches almost anywhere in the model. A hydrologic routing reach must be at least two cross sections long. A hydrologic routing reach can be an upstream piece, downstream piece, or an intermediate piece of any existing HEC-RAS unsteady flow river reach. A hydrologic routing reach can also encompass an entire HEC-RAS river reach. Hydrologic routing reaches can contain bridges/culverts and lateral structures, but it cannot contain an inline structure. If you have an inline structure within a reach that you want to perform hydrologic routing, you must stop the hydrologic routing reach at least two cross sections upstream of the structure, and you can start a new routing reach downstream of the structure. The hydraulics of bridges and culverts will be

incorporated into the routing through the resulting steady flow water surface profiles. Flow over lateral structures are computed each time step as they would normally be for unsteady flow routing.

To establish a piece of a model as a hydrologic routing reach, select the **Add Region** button. When this button is pressed, a window will appear in which you can select a river, reach, and a range of cross sections (upstream and downstream end of the reach) to establish as a hydrologic routing reach. Multiple hydrologic routing reaches can be set within the same model. There is also a button to edit the limits of an existing reach, and a button to delete reaches.

Two check boxes exist at the top of the window. The first check box is labeled **Use Modified Puls Routing**. If this box is not checked, the modified puls routing option will be ignored, and HEC-RAS will perform full unsteady flow routing at all of the cross sections. When this box is checked, any hydrologic routing reaches listed in the table will be modeled with the Modified Puls routing method. The second check box is labeled **Tailwater Check**. When this option is turned on, the downstream interface of any hydrologic routing reach will be monitored to see if the next cross section downstream has a higher computed water surface than the last section of the hydrologic routing reach. If the downstream water surface (tailwater) is higher than the last cross sections water surface of the hydrologic routing reach, then that water surface is forced into the downstream portion of the hydrologic routing reach. This allows for downstream conditions to influence the water surface, volume, and flow rate in the hydrologic routing reach.

The last step required to use the hydrologic routing reach is to import computed rating curves from a previously run steady flow model that used the exact same geometry file. This is accomplished by selecting the steady flow output plan and then pressing the **Import Rating Curves (RC's) from Steady Flow Output** button. There is also an option to plot the rating curves at the bottom of the editor, in order to visually inspect them. In general, users should set up a steady flow model with many water surface profiles (at least 20) in order to get good definition in the flow versus elevation points of the rating curves. Flows should range from lower than expected to route, to higher than expected to route within the unsteady flow model.

Georeferencing an HEC-RAS Model

Georeferencing is the process by which real-world coordinates are assigned to an HEC-RAS model to reference it to other data. An HEC-RAS model that shares a common coordinate system with GIS layers allows the developer of the model to visualize limits of the defined study area using background data.

The use of aerial photographs, for example, can be extremely beneficial for identifying the location of a landform such as levee or road or structures such as bridges and weirs while showing the user its representation in the hydraulic model. Therefore, background data is quite useful for model development. HEC-RAS results may also be quickly visualized using automated GIS procedures using tools such as HEC-RAS Mapper and HEC-GeoRAS. Additionally, floodplain maps may also be developed if the HEC-RAS model is georeferenced.

HEC-RAS allows the user to have a stream system schematic, and hydraulic elements, that is drawn spatially correct. This requires that the stream centerlines, cross sections, storage areas, and 2D Flow Areas have GIS coordinate information in order to draw them in a georeferenced framework. Generally, GIS coordinate information can be developed by laying out the model geospatially using HEC-RAS Mapper. However, HEC-RAS also has editors and tools for allowing the user to enter and manipulate GIS coordinate information directly in the HEC-RAS Geometric Data editor. The following is a list of the GIS tools available from the **GIS Tools** menu at the top of the Geometric Data editor.

GIS Tools in HEC-RAS

GIS tools in HEC-RAS are provided on the Geometric Data editor on the GIS Tools menu shown in Figure 5-96. The GIS Tools provide capabilities for editing and modifying x and y coordinates associated with the river network, cross sections, and other features in HEC-RAS. These GIS coordinate data can be edited directly through the different table options or computed based on the data available. The GIS Tools also provide visual displays of the data that can be exported to the GIS for processing.

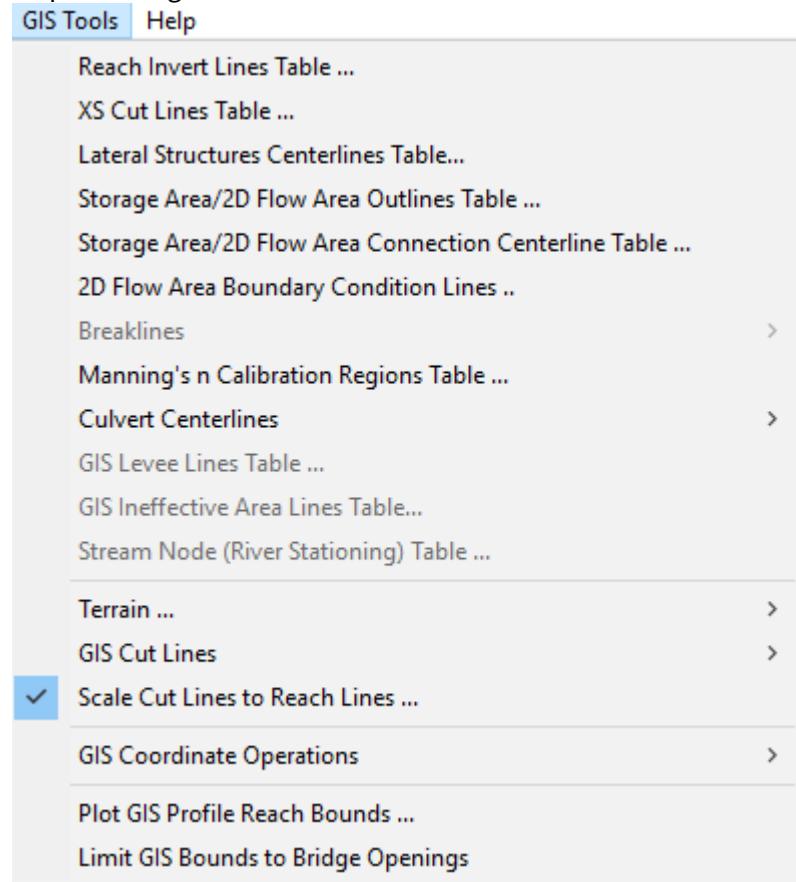


Figure 5 96. GIS Tools Menu Options

Tables

The x and y position that establish the location of the river reaches, cross-sectional cut lines, storage area outlines, storage area connections, levee lines, and ineffective areas are available through Tables from the GIS Tools menu. These tables identify the object and provide simple spreadsheet operations including cut, copy, and paste. This allows you easy access to geo-reference an individual object, such as a cross section cut line. The following is a list of the available tables from the GIS Tools menu:

Reach Invert Lines Table: This option allows the user to numerically edit the coordinates of the river reach schematic lines. When the river system schematic is hand drawn on the screen, the coordinates of the river reach lines are put into a simple coordinate system that ranges from 0.0 to 1.0 in both the X and Y direction. However, the user has the option of taking real world coordinates

(such as UTM or State Plane coordinates) off of a map and entering them into this table. If the user decides to use real world coordinates, real world coordinates must be added for all of the reaches of the schematic. If this is not done, the schematic will still be displayed in the simple 0.0 to 1.0 coordinate system (the hand drawn coordinates). Once real world coordinates have been entered for all of the river reaches, then the schematic will be drawn in that coordinate system. To enter/edit the reach schematic lines, select the **Reach Invert Lines Table** option from the **GIS Tools** menu. Once this option is selected, a window will appear allowing the user to enter/edit the coordinates of any of the reaches defined in the schematic (Figure 5-97).

	Schematic X	Schematic Y
1	2244841.0819507	5190870.1608279
2	2244841.0819507	5191869.7486222
3	2244669.7240432	5193526.2083962
4	2244669.7240432	5194697.1540988
5	2244783.9626482	5195925.2191036
6	2245212.3574174	5197210.4034112
7	2245697.8714891	5198324.2298119
8	2246383.3031197	5198981.1017923
9	2247011.6154475	5199666.533421
10	2247297.2119606	5200294.8457491

Figure 5 97. GIS coordinates for River/Reach lines

XS Cut Lines Table: This option allows the user to numerically edit the coordinates of the cross section schematic lines. When the river system is hand drawn on the screen, the default coordinate system is a simple 0.0 to 1.0 range for both the X and Y direction. However, the user can add spatially referenced map layers into the HEC-RAS Geometry editor, to be used as background map layers. If the user adds spatially referenced background map layers, then the coordinate system used will be based on the background maps. As cross sections are entered, they are automatically scaled based on the coordinates of the river reach line and the main channel distance between cross sections. Each cross section is drawn as a straight line perpendicular to the river reach schematic line. The user has the option of entering the real world coordinates (UTM or State Plane) of the cross section schematic lines. Each cross section schematic line must have at least two points, a start and an end, but additional points can be added if the cross section was taken as a multi segmented line. In order for the cross section schematic lines to be plotted in the real world coordinate system, the user must enter real world coordinates for all of the cross sections in the reach. To enter/edit the cross section schematic lines, select the **XS Cut Lines Table** option from the **GIS Tools** menu. Once this option is selected, a window will appear allowing the user to enter/edit the coordinates of any of the cross section schematic lines. User's can also use the **Measuring Tool** feature built into HEC-RAS to digitize the appropriate locations of a cross section. The measure tool is accessed by holding down the **Cntrl Key** and using the mouse to lay out a line. Once the line is ended, statistics about the line

are shown on the screen and the X and Y coordinates of that line are sent to the Windows clipboard. The user can then paste them into the Cross Section Cut Line table if desired.

Lateral Structure Centerline Tables: This table allows user to enter/edit the X and Y coordinates that define the centerline of a Lateral Structure. Users can cut/past coordinates into the table. Additionally X, Y coordinates can be imported from Shapefiles (**Import Lines**). Also, the number of points in the table can be filtered down to a more manageable number of points using the **Filter** option.

Storage Area/2D Flow Area Outlines Table: This option allows the user to enter/edit the X and Y coordinates that define the outline of a storage area or a 2D Flow Area. If you are using GIS data, these coordinates may be in the UTM or State Plane coordinate system. If you are not using GIS data, the coordinate system for the storage areas will just be in a 0 to 1.0 system. The coordinates of a storage area can also be edited graphically by using the **Move Object** option from the edit menu. The button called "**Filter Multiple Lines**" allows the user to select multiple storage areas and/or 2D Flow Areas, for filtering the number of points in the polygon that describes the storage area or 2D Flow Area. Once the user selects the storage areas and 2D Flow Areas to be filtered, they will be prompted for a point filter tolerance (default is 0.1). This tolerance is a collinear point filtering tolerance. The software puts a line between points 1 and 3, then if point 2 is a distance less than the tolerance away from that line, then it assumes that point two is not needed (i.e. the three points are basically on a straight line), so point 2 is dropped from the polygon perimeter.

Storage Area/2D Flow Area Connection Centerline Table: This option is used to define the coordinates of the hydraulic structure that is being used to connect two storage areas, or a storage area to a 2D Flow Area, or two 2D Flow Areas. The connection should be drawn from left to right looking in what the user considers to be the positive flow direction. This line will be drawn on the river system schematic, and will represent the hydraulic structure being used to connect two areas.

2D Flow Area Boundary Condition Lines: This option will show the coordinates of any 2D Flow Area external Boundary Condition lines.

Breaklines: This option brings up a table that shows the X and Y coordinates for each of the breaklines. Users can edit the X and Y coordinates directly, they can cut/paste values, and they can import breaklines from Shapefiles.

GIS Levee Lines Table: This option allows the user to edit/enter the coordinates for a user specified levee. Levees can be defined in the GIS system and imported into HEC-RAS. The coordinates will consist of X, Y, and Z (elevation).

Manning's n Calibration Regions Table: This option allows the user to enter/edit the X and Y coordinates that define the outline of the Manning's n Override/Calibration regions polygons.

Culvert Centerlines: There are two options under this menu option. The first is to **Import Barrel Centerline** from a Shapefile, and the second is to enter/edit the X and Y culvert barrel centerline coordinates.

GIS Ineffective Flow Area Table: This option allows the user to edit/enter the coordinates for a user specified ineffective flow areas. Ineffective flow areas can be defined in the RAS Mapper as a polygon. The coordinates will consist of X, Y, and Z (trigger elevation for when it will be turned off).

GIS Cut Lines

The GIS Cut Lines menu item provides useful tools for quickly computing geospatial information for cross sections. The tools allow to georeference cross sections and to adjust the geospatial length of the cross section to match the width specified by the cross-section's station-elevation data.

Accept Displayed Locations (as Georeferenced). This tool allows the user set the GIS cut line x and y positions to the coordinates displayed in the Geometric data editor. You can do this for several cross sections at a time using this menu option, or you can do this one cross section at a time by using the left mouse button to click on a specific cross section.

Remove Georeferenced Cut Line Data: This option allows user's to select cross sections and then request that all the georeferenced coordinates for those cross sections be removed, and the cross section go back to being laid out as straight lines along the stream centerline. The location of the cross sections will be based on their river stationing and reach lengths between the cross sections.

Adjust Cut Line Lengths to Match XS Lengths. You can adjust the length of the cut line to match the width defined by the cross section's station-elevation data. This tool will invoke the table (Figure 5-98) showing the river station, cut line length, cross section length, and ratio of cut line length to cross section length. You then specify how you would like to adjust the cut line: whether RAS should adjust the left side, right side, or both sides the line equally to make the cut line length equal to the cross section length.

Once you have decided which side to extend or trim, you then choose to adjust the cut line lengths or cross section lengths using the provided buttons.

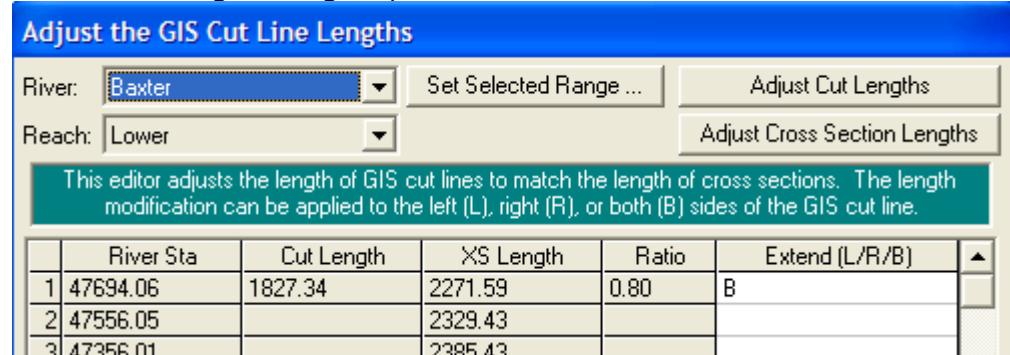


Figure 5.98. Dialog for adjusting cross section and cut line lengths

Extend Cut Lines and Sta/Elev. The menu option to extend the cut line and station elevation data allows you to extend both the cut line and cross section in the left bank or right bank. This is used to extend the cross section to improve floodplain mapping and should only be used once the cross section has reached high ground. The dialog use for extending a cross section is shown in Figure 5-99.

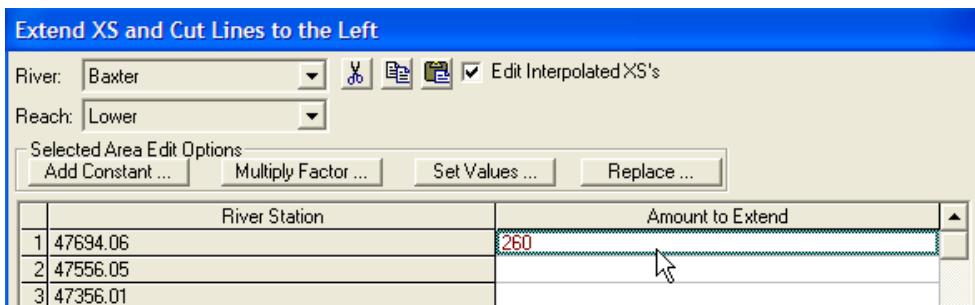


Figure 5 99. Dialog for extending a cut line.

Project XS to Straight Cut Line. This is an option to straighten a cross section based on its first and last point in the XS Cut Line. This option adjusts the Cut Line and the internal station/elevation data to match the new length of the line.

Reverse Cut Lines. This option will reverse the order of the horizontal coordinates of the cross section cut line. Cross sections must be laid out from left to right looking downstream. You can ask HEC-RAS to show you the direction a cross section line was laid out. If the cross section cut line is not laid out from left to right (but the station/elevation data are), you can reverse the cross section cut line with this option. If you need to reverse the order of the cross section cut line and the station/elevation data, there is a separate tool to do that under the **Tools** menu of the Geometric Data editor.

Scale Cut Lines to Reach Lines

As discussed previously, the cross sections may look georeferenced, but they actually may not be. Cross sections that do not have geospatial information are drawn perpendicular to the stream centerline and the spacing is based on the downstream reach lengths.

By default, the Geometric Schematic scales the display of the cross sections based on the river network. This is handy when the data is not georeferenced (when the river schematic is very short when compared with real world distance), but we want to turn this option off when we georeference the cross sections. Select the **GIS Tools | Scale Cut Lines to Reach Lines** menu item to turn it on or off, the default is on.

GIS Coordinate Operations

The GIS Coordinate Operations allow you to switch the x an y coordinates; add and multiply the coordinates by a factor; and to filter points in the GIS lines for a few of the objects in HEC-RAS: reaches, cross sections, and storage areas/2D Flow Areas. These tools are useful if you accidentally imported the northing and easting data incorrectly.

Other tools allow you to add and multiple the coordinate values. This is convenient for performing simple coordinate transformations or for removing a false northing or easting. Coordinate manipulation options are shown in Figure 5-100.

Reaches - Swap X and Y Coordinates ...
Reaches - Add and Multiply Coordinates ...
Reaches - Filter Coordinates ...
Cut Lines - Swap X and Y Coordinates ...
Cut Lines - Add and Multiply Coordinates ...
Cut Lines - Filter Coordinates ...
Storage Areas - Swap X and Y Coordinates ...
Storage Areas - Add and Multiply Coordinates ...
Storage Areas - Filter Coordinates ...

Figure 5 100. Coordinate manipulation menu options.

Plot GIS Profile Reach Bounds

If you intend to perform a floodplain delineation based on the computed HEC-RAS results, you will need to verify what RAS thinks are the limits of the model. You can plot this information using the Plot GIS Profile Reach Bounds menu option for each water surface profile. After selecting this option you will need to choose the profile(s) to plot and river reach(es). The bounds will be plotted in magenta (by default) as illustrated in Figure 5-101.

The GIS Profile Reach Bounds is also referred to as the Bounding Polygon. This data will be used in the GIS to keep the floodplain delineation to occur only over the limits of the hydraulic model.

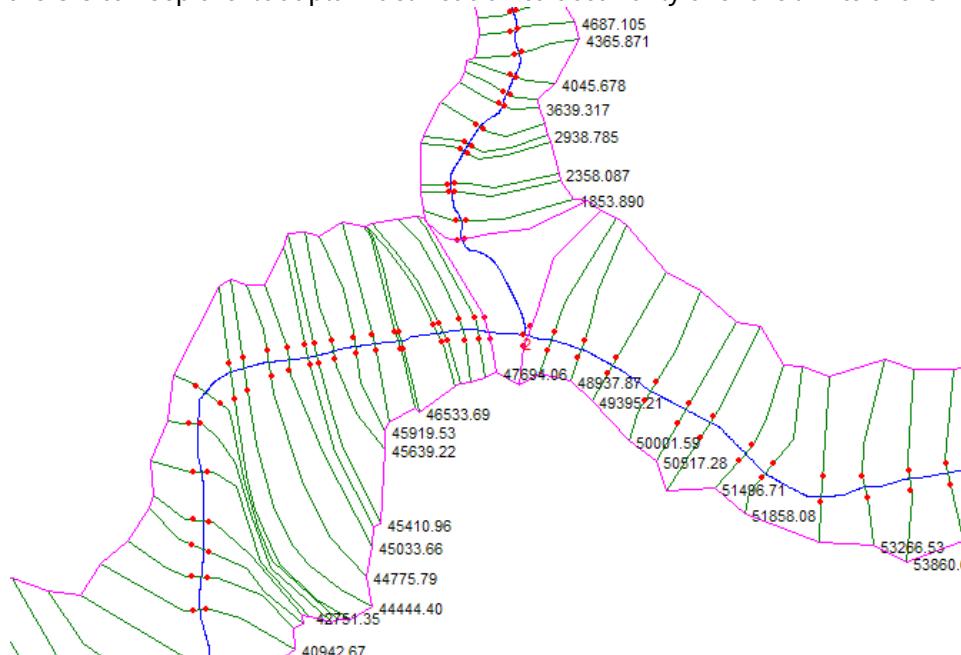
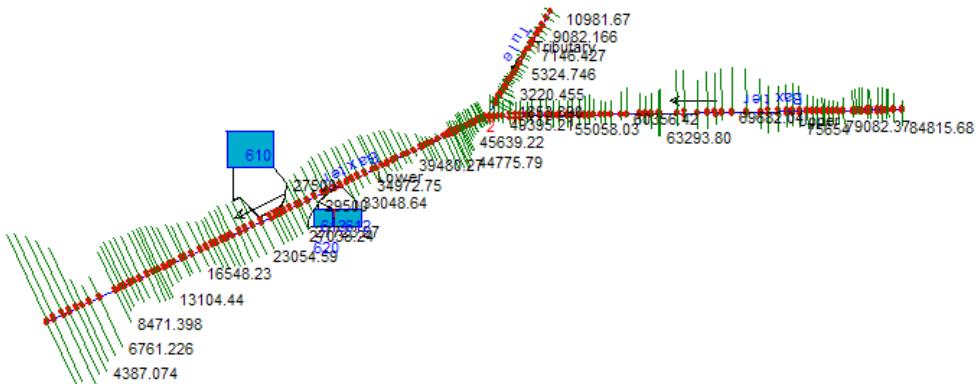


Figure 5 101. GIS bounding polygon information limits floodplain delineation to the area modeled in HEC-RAS

Limit GIS Bounds to Bridge Openings. This option will limit the GIS bounding polygon to the inside of a Bridge opening at all bridge locations.

Example of Georeferencing an HEC-RAS Model

In this example the Baxter River dataset will be used demonstrate how to georeference an HEC-RAS model. Specifically, the River Network, Cross Sections, Storage Areas, and Storage Area connections, shown in the figure below, will be georeferenced using methods in HEC-RAS and using the GIS.



A non-georeferenced model schematic.

To successfully georeference a RAS model, you decide on a coordinate system and have at least one background dataset that has been projected to that coordinate system. A digital raster graphic (DRG) of a topographic quad sheet, an aerial photograph, or a digital terrain model (DTM) may be available for reference.

Once the coordinate system has been defined and a background dataset acquired, you should establish the stream centerline first. Georeferencing the river network will assist you in spatially locating the cross sections.

The river network may be created in either the GIS using HEC-GeoRAS or created directly in HEC-RAS; however, you will not have access to production level digitizing tools in HEC-RAS so this option is limited to very simply river networks.

Creating the River Network using HEC-GeoRAS

Open ArcMap and load the 3D Analyst and Spatial Analyst Extensions. Turn on the HEC-GeoRAS toolbar and Save the ArcMap document.

Set the Coordinate System for the data frame (map) and load the background data (image, DTM, etc). The GeoRAS tools for establishing the Stream Centerline topology require that you load a terrain model.

The Stream Centerline is used to establish the river reach network and is shown in Figure 5-103. The river network must be digitized in the direction of flow with reach end points coincident at junctions.

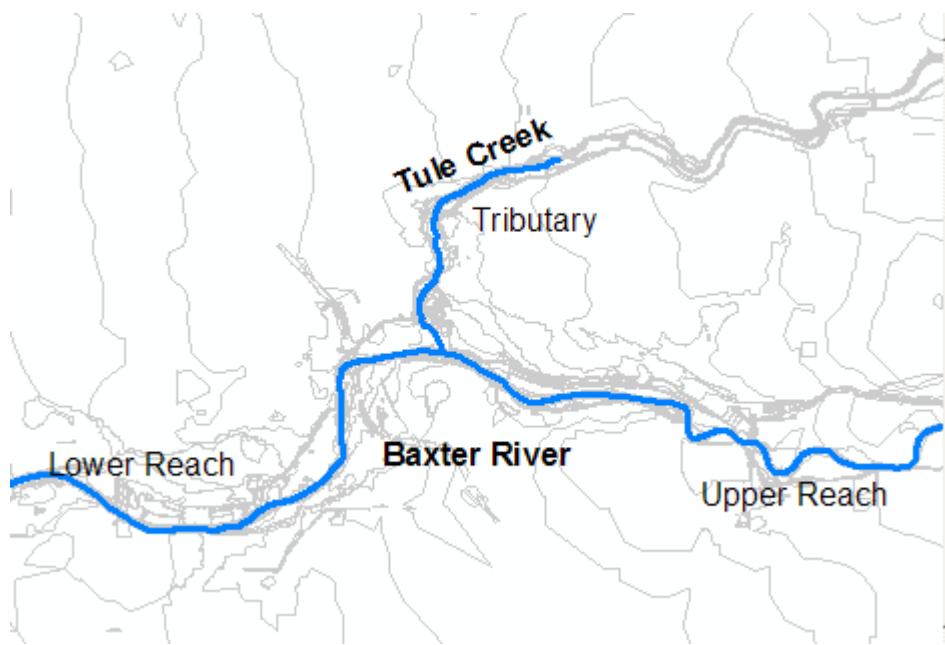


Figure 5 103. River network for the Baxter River example dataset.

Digitize the stream centerlines for the river reaches shown. From the GeoRAS toolbar, select the **RAS Geometry | Create Layer | Stream Centerline** menu item.

Start editing the feature class by selecting the **Editor | Start Editing** menu item. The stream centerline must be created in the direction of flow, so start at the top end of the river and zoom in so that the channel is easily identified.

Select "**Create New Feature**" for the Task and "**River**" for the Target feature class. Select the **Sketch** tool and begin digitizing the line in the downstream direction. (Left-click drops a vertex.) Continue digitizing the line until you reach the junction. If you need to pan, simply select the **Pan** tool, pan through the map, and re-select the Sketch tool to continue digitizing. To finish the reach line at the junction, double-click to drop the endpoint.

Digitize each river reach, individually. There are three river reaches in total, with one junction at the confluence of Tule Creek. You will create the junction after creating all the reaches.

Creating a Junction. To create a junction, the endpoints of each reach must be coincident. While in Edit mode, select "**Modify Features**" for the Task. Next set the snapping tolerance, by selecting the **Editor | Options** menu item. On the **General** tab, set the **Snapping Tolerance** to "**10**" **map units**.

Next select the **Editor | Snapping** menu item. Click on snapping to the **End** points for the River layer, as shown Figure 5-104.

Layer	Vertex	Edge	End
XSCutLines	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
River	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>

Figure 5 104. Select the option for snapping at end points

Next, select the **Edit** tool and grab the endpoint of a river reach line by pressing and holding the left mouse button. Move it towards another reach endpoint. When the point is within the snapping tolerance, a sketch of the endpoint will appear and snap to the endpoint. Release the mouse button and the endpoint will snap. The progression of steps to snap endpoints is illustrated in Figure 5-105.

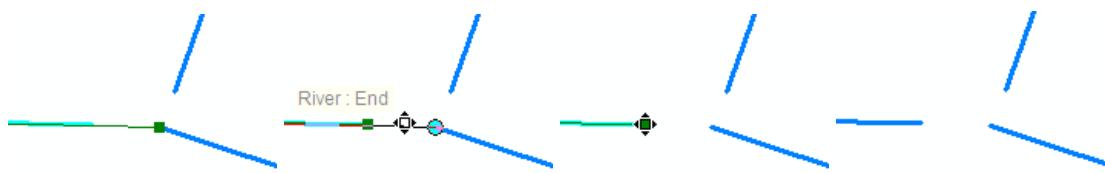


Figure 5 105. Progression for creating a junction using snapping.

Repeat the snapping process for the other reach. Verify that the reach network has been created in the downstream direction by changing the line symbol to include and arrow at the end of the line. In a later step, you will use the GeoRAS tools to double-check the connectivity.

River and Reach Names. Each river must have a unique river name, and each reach within a river must have a unique reach name. Use the (River Reach ID) tool to give each river reach a name. Click on the **River Reach ID** tool to make it active. Use the cursor to select each river reach. The River and Reach Name dialog (shown in Figure 5-106) will appear allowing you to enter the river and reach name. For this example, the **Baxter River** has an **Upper Reach** and a **Lower Reach** and **Tule Creek** is a **Tributary**.



Figure 5 106. River and Reach name assignment dialog.

After labeling each River reach, look at the attributes for the River layer and verify that the River and Reach information was provided for each reach. To open the attribute table, right-click on the River layer and select the **Open Attribute Table** menu item.

Network Connectivity. To verify the river reach connectivity, select the **RAS Geometry | Stream Centerline Attributes | Topology** menu item. The fields *FromNode* and *ToNode* will be populated with integer data. Verify that the endpoints at the junction all share a common node number. The complete attribute table for the River layer is shown in Figure 5-107.

Attributes of RiverNetwork							
River	Reach	FromNode	ToNode	ArcLength	FromSta	ToSta	
Baxter River	Upper Reach	1	2	41221.3	48157.1	89378.4	
Baxter River	Lower Reach	2	3	48157.1	0	48157.1	
Tule Creek	Tributary	4	2	12551.5	0	12551.5	

Figure 5 107. Completed Stream Centerline attribute table.

Lastly, run the **RAS Geometry | Stream Centerline Attributes | Lengths/Stations** menu item. This computes the length of each reach for determining the cross-section river stationing. The *FromSta* and *ToSta* fields will be populated with the *FromSta* being the downstream endpoint of the reach. The *FromSta* and *ToSta* data are "backwards" from the *FromNode* and *ToNode* because the actual river stationing is calculated from downstream to upstream!

Export the data by selecting the **RAS Geometry | Extract GIS Data** menu item. The dialog shown in Figure 5-108 will be invoked allowing you to choose the destination directory and filename.



Figure 5 108. Filename and location for GIS export.

After pressing **OK**, GeoRAS will export the GIS data to an XML file and then convert the XML file to the SDF format. Two files will be created: "GIS2RAS.xml" and "GIS2RAS.RASImport.sdf". This process will take several seconds. The dialog shown in Figure 5-109 will appear when the process has successfully created the files. You now have a file you can import into your RAS model.



Figure 5 109. Successful GIS data export dialog

Creating the River Network using HEC-RAS

If you don't have access to GIS tools or your river network is very simple, creating the river network in HEC-RAS is an option. To get started, open the RAS project that needs to be georeferenced. Open the Geometric Data editor and create a New Geometry File. Add the background data by

clicking on the **Add Background Data** button. If the image does not come in correctly, select the **View | Set Schematic Plot Extents** menu item and press the **Set to Computed Extents** button in the Geometry Extents window shown in Figure 5-110. This will zoom out the bounds of the image.

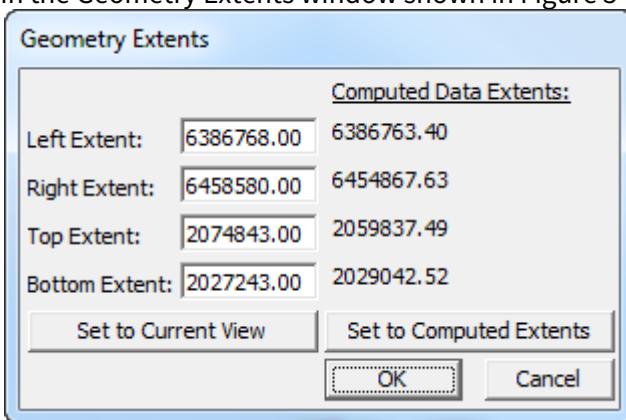


Figure 5 110. Dialog for setting the display extents in HEC-RAS.

Press the **OK** button to use the new extent coordinates and to dismiss the window.

Select the **River Reach** creation tool and digitize the **Baxter River**. You will want to be zoomed in to a reasonable scale. Digitize the centerline of the river in the downstream direction. You can pan by right clicking. This will pan the display window so that the mouse is centered.

After double clicking to end the river the dialog shown in Figure 5-111 will be invoked to provide you a place to enter the River and Reach name.



Figure 5 111. River and reach name data entry dialog.

Create the **Tule Creek Tributary**. Make sure to end the line on the Baxter River at the confluence. You will then be asked if you would like to split the Baxter River. Select **Yes** and provide a reach name for the **Lower Reach** of the Baxter River. Next, provide a junction name.

Save the geometry. You will now have an HEC-RAS geometry that has a georeferenced River Network like that in Figure 5-112. This can be imported into an existing RAS geometry.

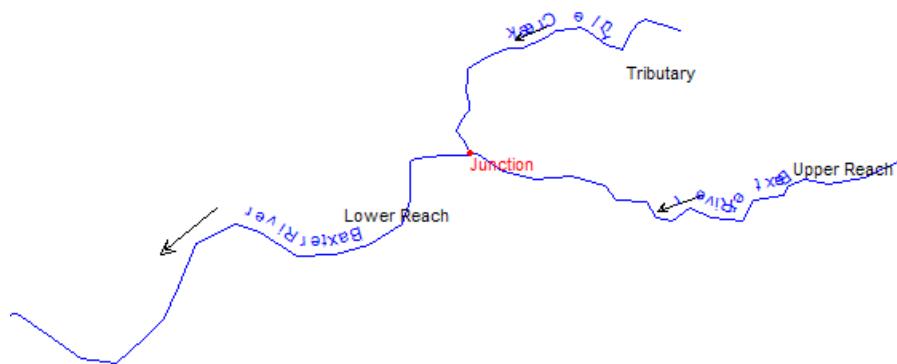


Figure 5 112. A georeferenced river network in HEC-RAS.

Importing the Stream Centerline

Once you have an import file created using GeoRAS or a geometry file created in RAS with a georeferenced stream centerline, you can import the stream centerline from either the GeoRAS export file or the RAS geometry file. The RAS Geometric Data importer works the same for either file.

Open HEC-RAS and load the non-georeferenced geometry shown in Figure 5-113. Save the geometry to a new geometry so that if anything goes wrong you won't destroy your existing model data.

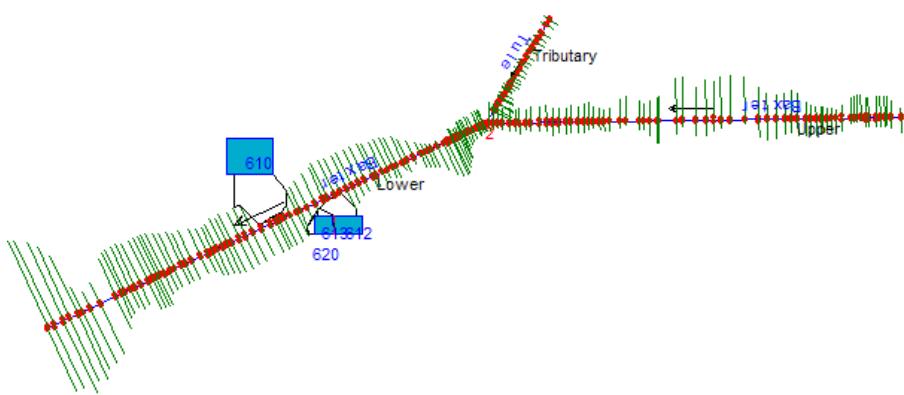


Figure 5 113. A non-georeferenced appears as a stick diagram in HEC-RAS.

Select the **File | Import Geometry Data | GIS Import** menu item. Select the **RASImport.sdf** file to import created using GeoRAS. (If you created the centerline in a RAS geometry file, select **File | Import Geometry Data | HEC-RAS Format** instead and choose the geometry file.)

HEC-RAS may display a warning or error message because it is expecting cross section data and the file only has the river network information. Continue through the error message.

Click on the **River Reach Stream Lines tab**. Note that the importer, as shown in Figure 5-114, thinks that the river reaches are all "new".

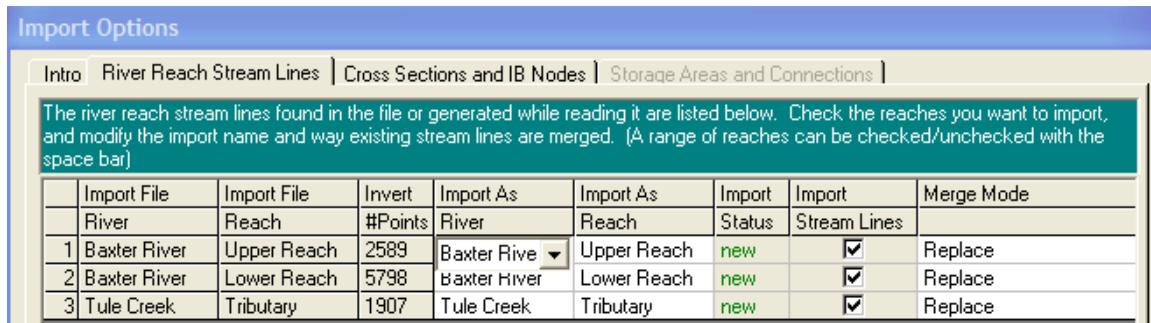


Figure 5 114. The HEC-RAS geometry importer looks to match the names in the import file with names in the RAS geometry file.

HEC-RAS thinks that the river reaches are new because the names in the import file do not match those that are in the existing RAS model. Select the appropriate names to import the river and reaches to by using the list boxes in the **Import As** columns. When completed, the **Import Status** will change to "**exists**", as shown in Figure 5-115.

Import Options								
Intro River Reach Stream Lines Cross Sections and IB Nodes Storage Areas and Connections								
The river reach stream lines found in the file or generated while reading it are listed below. Check the reaches you want to import, and modify the import name and way existing stream lines are merged. (A range of reaches can be checked/unchecked with the space bar)								
	Import File	Import File	Invert	Import As	Import As	Import	Import	Merge Mode
1	River	Reach	#Points	River	Reach	Status	Stream Lines	
1	Baxter River	Upper Reach	2589	Baxter	Upper	exists	<input checked="" type="checkbox"/>	Replace
2	Baxter River	Lower Reach	5798	Baxter	Lower	exists	<input checked="" type="checkbox"/>	Replace
3	Tule Creek	Tributary	1907	Tule	Tributary	exists	<input checked="" type="checkbox"/>	Replace

Figure 5 115. When river and reach names in the import file and geometry file match, the import status is "exists".

You are now ready to import the stream centerlines. Press the Finished-Import Data button. The stream centerline will import, replacing the existing river network. The HEC-RAS model will no longer look like a stick diagram, but will look georeferenced with a river network like that in Figure 5-116.

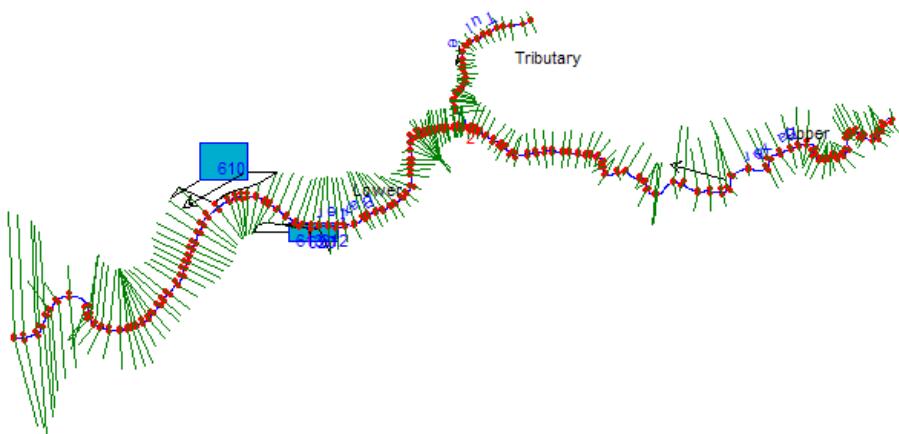


Figure 5 116. An HEC-RAS model with a georeferenced river network.

The cross sections look like they could be georeferenced, but they are not. They are actually drawn perpendicular to stream centerline and are spaced based on the channel downstream reach lengths. You can verify that the cross sections are not georeferenced by looking at the Cut Lines table. Select the **GIS Tools | XS Cut Lines Table** menu option. In the river station list box there is a label of "NO DATA!!" after each river station (see Figure 5-117).

Edit Cross Section lines for plan view on schematic plot														
River:	<input type="text" value="Baxter"/>	<input type="button" value="..."/>	<input type="button" value="New"/>	<input type="button" value="Open"/>	<input type="button" value="Save"/>	<input type="button" value="Delete"/>	<input type="button" value="Filter Line ..."/>							
Reach:	<input type="text" value="Upper"/>	<input type="button" value="..."/>	RS:	<input type="text" value="84815.68 (NO DATA !!)"/>	<input type="button" value="Down"/>	<input type="button" value="Up"/>								
Selected Area Edit Options														
<input type="button" value="Add Constant ..."/> <input type="button" value="Multiply Factor ..."/> <input type="button" value="Set Values ..."/> <input type="button" value="Replace ..."/>														
<table border="1"> <thead> <tr> <th>Schematic X</th> <th>Schematic Y</th> </tr> </thead> <tbody> <tr> <td>1</td> <td></td> </tr> <tr> <td>2</td> <td></td> </tr> </tbody> </table>							Schematic X	Schematic Y	1		2			
Schematic X	Schematic Y													
1														
2														

Figure 5 117. Cross sections without cut line information display "NO DATA!!" in the river station list.

Note that the storage areas are also not georeferenced. We will complete the remainder of the georeferencing within the HEC-RAS Geometric Data editor.

Storage Areas and Connections

Load the background image using the **Add Background Data** button and turn it on. Let's georeference the storage areas first.

Zoom into the area around the storage area on the north side of the river. Use the line creation tool available in RAS to create the outline of where the storage area should be. The line creation tool becomes active in any display in RAS when you press the Ctrl key.

Press the **Ctrl** key, while depressed, **left-mouse click** around the area that the storage area represents. When the polygon is completed, **release the Ctrl** key. The dialog shown in Figure 5-118 will be displayed summarizing some of the line information and the x and y coordinates will be copied to the clipboard. Press the **OK** button to continue.

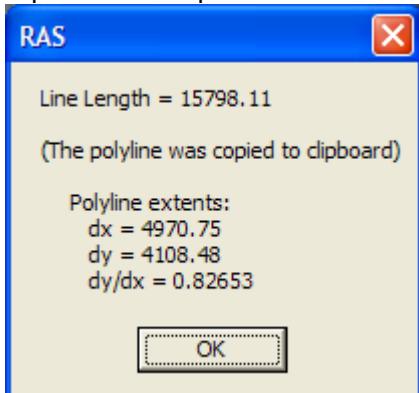


Figure 5 118. Summary dialog from the line creation tool.

Now you need to paste the spatial information into the storage area table. Select the **GIS Tools | Storage Area Outlines Table**. The table shown in Figure 5-119 will open with the previous outline information.

The screenshot shows the "Storage Area/2D Flow Area Outlines" dialog box. At the top, there is a dropdown menu labeled "SA: Northside" and a "Filter Line ..." button. Below this is a toolbar with icons for copy, paste, and other operations. A section titled "Selected Area Edit Options" contains buttons for "Add Constant ...", "Multiply Factor ...", "Set Values ...", and "Replace ...". The main area is a table with three columns: "Schematic X", "Schematic Y", and "Schematic Z". The table has 6 rows of data. The data is as follows:

	Schematic X	Schematic Y	Schematic Z
1	6402893.517147	2045925.8146822	
2	6400936.6848734	2047592.1129065	
3	6400030.9022489	2047096.4959987	
4	6400295.800941	2048386.8089826	
5	6401449.3920193	2048890.9710095	
6	6402688.4342886	2049318.2269644	

At the bottom of the dialog are buttons for "OK", "Cancel", and "Help".

Figure 5 119. Storage area outline information.

Select the entire table by clicking in the upper left hand corner of the table, as shown in Figure 5-120, and paste the information using the **Ctrl+V** paste command. The coordinates of the line you create will replace the previous data. Press **OK** to accept the changes.

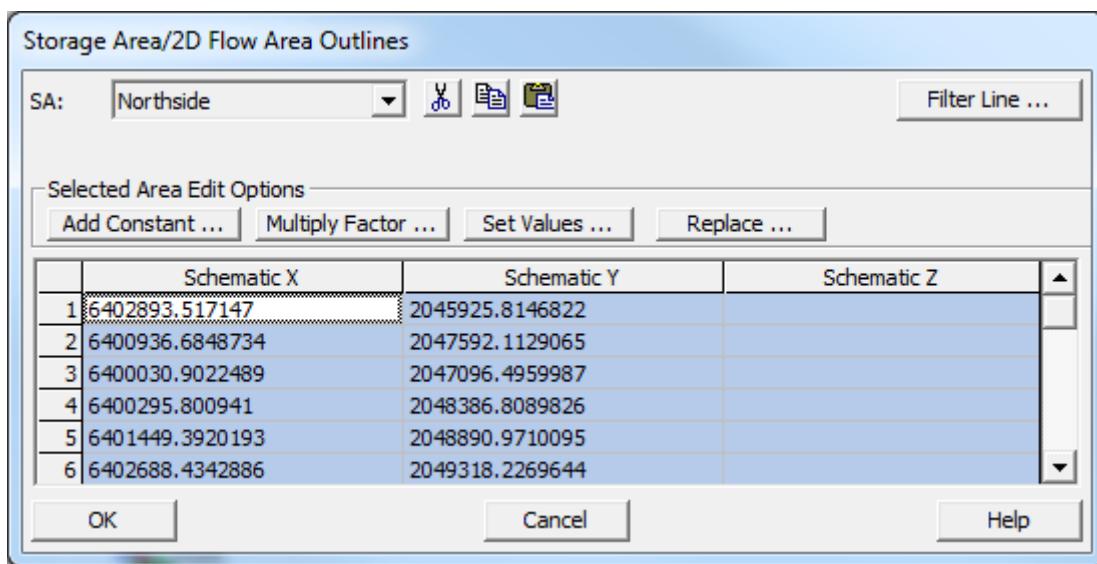


Figure 5 120. Storage area outline information replaced with georeferenced data.

Georeference each storage area as you did for the first. The storage areas should finally look like those in Figure 5-121.

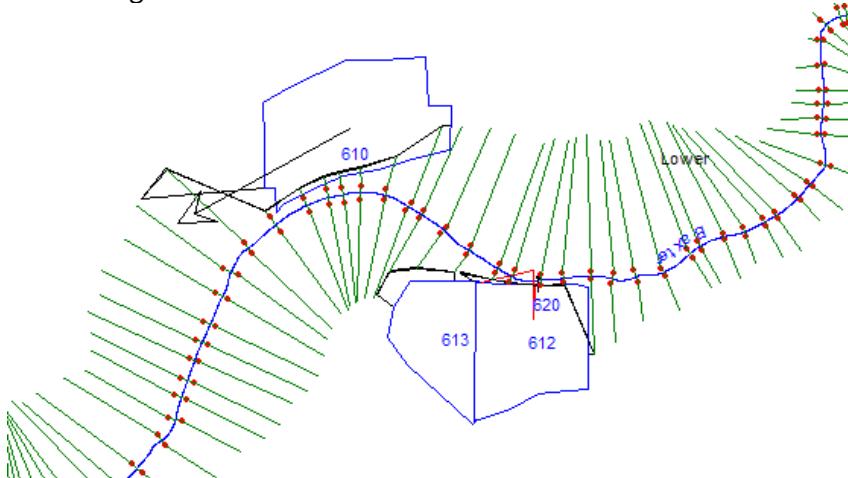


Figure 5 121. Geometric schematic with georeferenced storage areas.

Note that the storage area connection on the south side of the river is not georeferenced. You can georeference it the same as with the storage areas. Use the line creation tool to draw a line between the storage areas. Paste the results in the **Storage Area Connections Lines Table** accessible from the **GIS Tools** menu and the storage area connection will be drawn between the storage areas (see Figure 5-122).

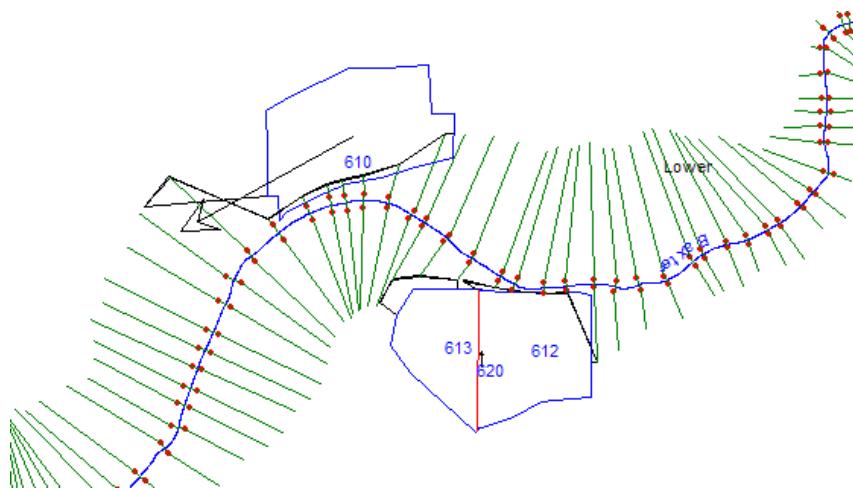


Figure 5-122. Geometric schematic with a georeferenced storage area connection.

Cross Sections

As discussed previously, the cross sections may look georeferenced, but they are not. Cross sections that do not have geospatial information are drawn perpendicular to the stream centerline and the spacing is based on the downstream reach lengths.

By default, the Geometric Schematic scales the display of the cross sections based on the river network. This is handy when the data is not georeferenced (when the river schematic is very short when compared with real world distance), but we want to turn this option off when we georeference the cross sections. Select the **GIS Tools| Scale Cut Lines to Reach Lines** menu item, as shown in Figure 5-123, to **remove** the check.

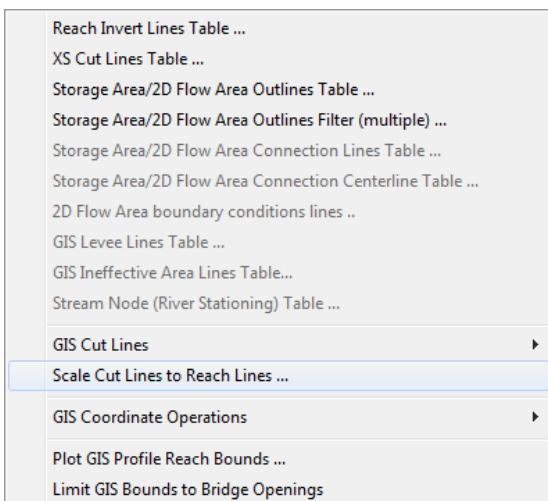


Figure 5-123. The Scale Cut Lines to Reach Lines menu item should be unchecked when using georeferenced data.

Cross sections should be georeferenced first at locations where you have a pretty good idea where they should go. The hydraulic structures in your model are the place to start. If you do not have bridges in your model, you should start by locating where the most downstream and most upstream

cross sections are located. This will allow HEC-RAS to establish where the cross sections should be along the river using the channel downstream reach lengths.

For this example, we are going to start with the most upstream bridge on the Upper Reach of the Baxter River. As shown in Figure 5-124, the bridge sections are located approximately 3000 ft upstream of the bridge and will need to be moved to the correct location.



Figure 5-124. The bridge sections are upstream of the real world location.

Use the mouse to **left click** on the downstream bridge cross section and select the **Move Cut Line Upstream/Downstream** option. The selected cross section and river centerline will be highlighted. Next, use the crosshairs and **left click** on the stream centerline just downstream of the bridge. The cross section will be repositioned (see Figure 5-125) and RAS will georeference it based on the stream centerline, cross section width, and bank stations.

The model schematic will then update to reflect the changes to all the cross sections based on the position of the georeferenced cross sections. A message will appear at the bottom of the schematic explaining to the user that all the cross sections do not have GIS information. The georeferenced cross sections are green, while those without geospatial data will be brown.

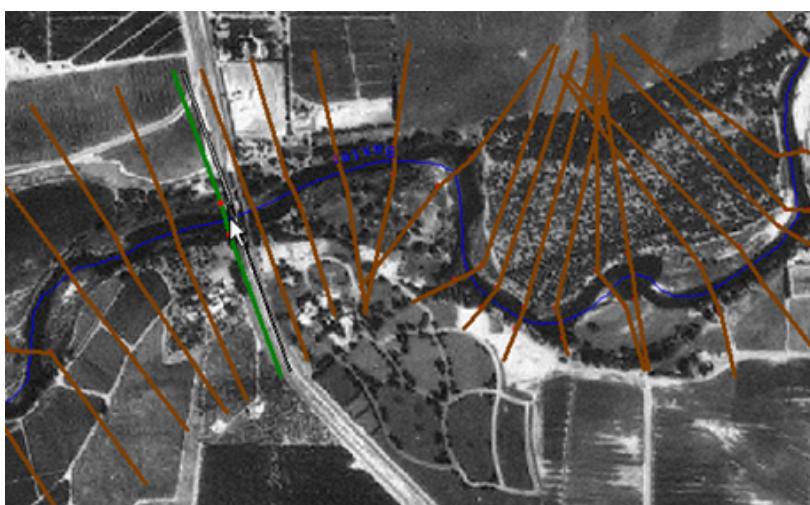


Figure 5-125. The downstream bridge location has been georeferenced.

The downstream bridge cross section is not exactly where it should be, so use the edit tools to move the endpoints of the cross section. Select the **Edit | Move Object** menu item. This will place vertices on each cut line, as shown in Figure 5-126, and allow you to move the points using the mouse. Move the points on the downstream bridge cross section.

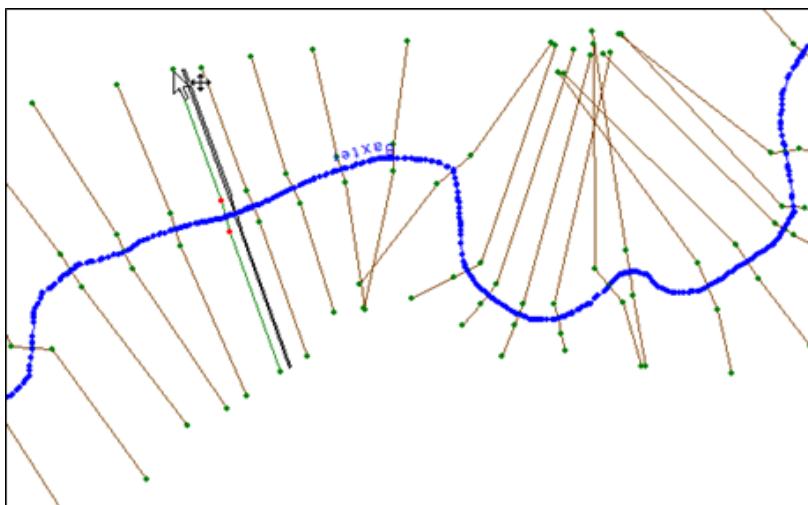


Figure 5-126. Vertices are displayed on the cut lines while in Move Object mode.

Do the same for the cross section on the upstream side of the bridge (see Figure 5-127), as well. Note that as soon as you move a point on a cross section that is not georeferenced it becomes georeferenced!

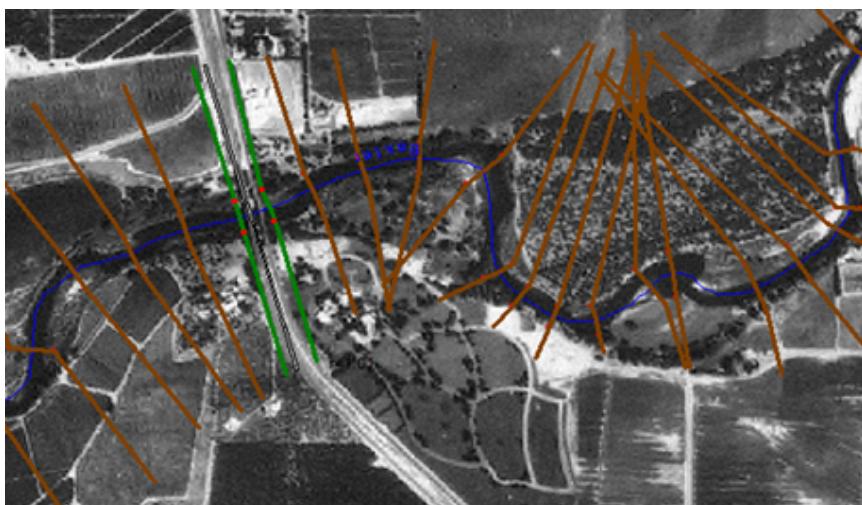


Figure 5-127. The upstream and downstream bridge cross sections have been georeferenced.

HEC-RAS will use the main channel reach lengths for determining where to lay out the cross sections, however, the stream centerline you created for georeferencing the model may not be identical to the centerline used when the river hydraulics model was first created. So you need to continue to position cross sections with known locations.

Zoom into the RAS schematic to the bridge just upstream from the airport runway. Note that the bridge river station position is about right on (see Figure 5-1286). It was placed there based on the downstream reach length. But to properly model this bridge, the cut line must follow the bridge out to high ground. Therefore, the cut line had to bend to follow the road. In this case, just moving the cut line points would be more difficult than using the line creation tool. Use the line creation tool to draw a line where the downstream bounding bridge cross section should be located.

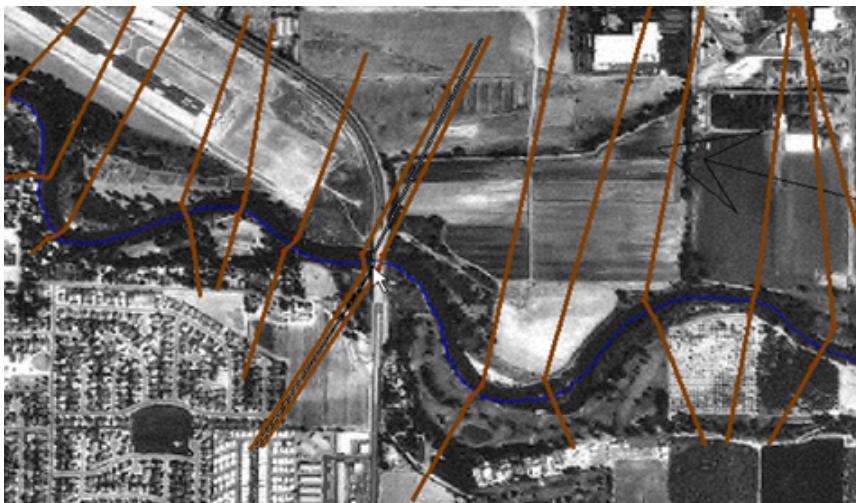


Figure 5-128. Bridges not perpendicular to the river require moving the bounding sections.

Once you have copied the cross section to the clipboard, paste it into the cut line table shown in Figure 5-129, as accessed through the **GIS Tools | XS Cut Lines Table**.

Edit Cross Section lines for plan view on schematic plot	
River:	Baxter River
Reach:	Upper Reach
Selected Area Edit Options	
Add Constant ...	Multiply Factor ...
Set Values ...	Replace ...
Schematic X	Schematic Y
1 6434658.9310454	2045482.6597695
2 6434634.8563554	2046570.0345306
3 6434646.8937004	2047573.1477819
4 6434651.5171713	2048162.9193713
5 6434612.8921591	2048484.6236115
6 6434566.0549328	2048644.421313

Figure 5-129. Cross section cut lines table data.

Press **OK** to accept your edits and the cross sections will look like those shown in Figure 5-130.

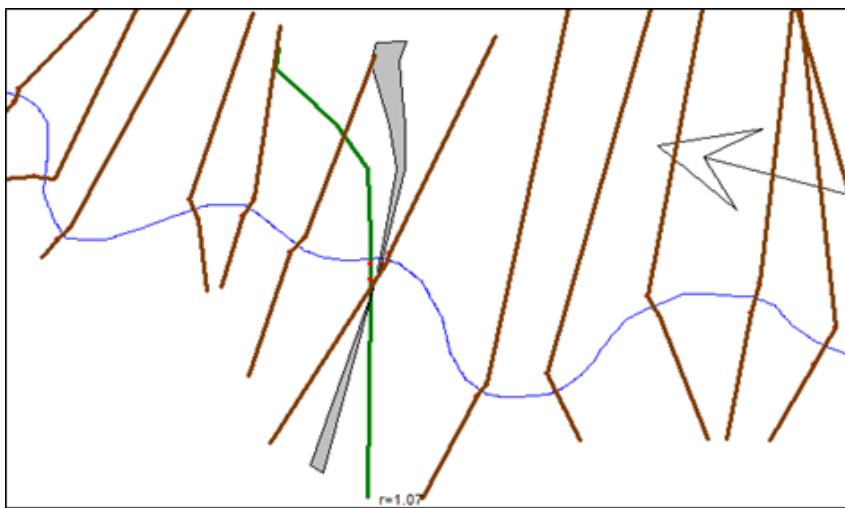


Figure 5-130. The ratio of the cut line to cross section width is displayed next to the cross section ($r=1.07$).

The downstream bridge cross section is now georeferenced, but there are two key pieces of information displayed that show you a mistake (1) the bank stations are not on either side of the stream centerline and (2) the ratio of the cut line to cross section width is 1.07 [$r=1.07$]. You need to adjust the cut line by shortening the left side. Use the Edit Object mode to move the left endpoint in until the banks are positioned correctly and the ratio is 1. The display of the cut line to cross section ratio is turned on/off through the **View | Display Ratio of Cut Line Length to XS Length** menu item.

What happens if the bank positions move to the correct position but the ratio remains greater than 1? Move the right bank in. The **GIS Tools | GIS Cut Lines | Adjust Cut Line Lengths to Match Sta/Elev** option can also be used to adjust the cut line lengths.

Obviously, the next step is to georeferenced the cross section just upstream of the bridge. Again, use the line creation tool and paste the cut line information into the cut line table. Adjust the cut line as necessary to look like Figure 5-131.

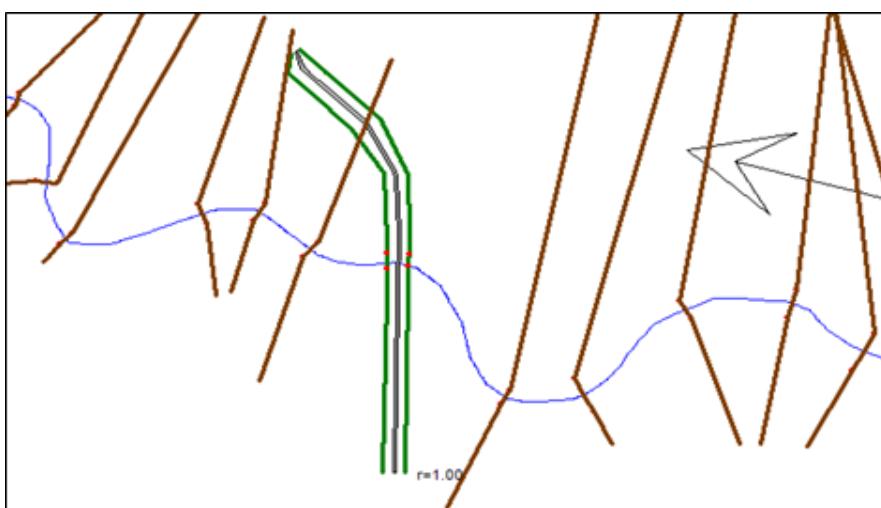


Figure 5-131. The downstream and upstream cut lines have been georeferenced.

It looks like there are some problems with how cross sections intersect near the bridge. We will need to remedy that problem of cross section intersecting. Not only hydraulically incorrect but attempts to perform floodplain delineation will not be successful. Prior to fixing overlapping cross sections, position all cross sections for which you know their location.

Reposition all cross sections around bridges. Next, look at any cross sections that HEC-RAS may have placed in the correct location. If any non-georeferenced cross section looks good, left click on that cross section and select the **Accept Displayed Locations (as Georeferenced)** menu item. This will store the cross section line to the XS Cut Lines table. If you want to accept the cut line information for more than one cross section, select the **GIS Tools | GIS Cut Lines | Accept Displayed Locations (as Georeferenced)** menu item and select the cross sections in the selection dialog. The color of all of the cross sections will turn green. The final, georeferenced geometry will look similar to that shown in Figure 5-132. Now you can re-run the model and export the results to the GIS for floodplain mapping.

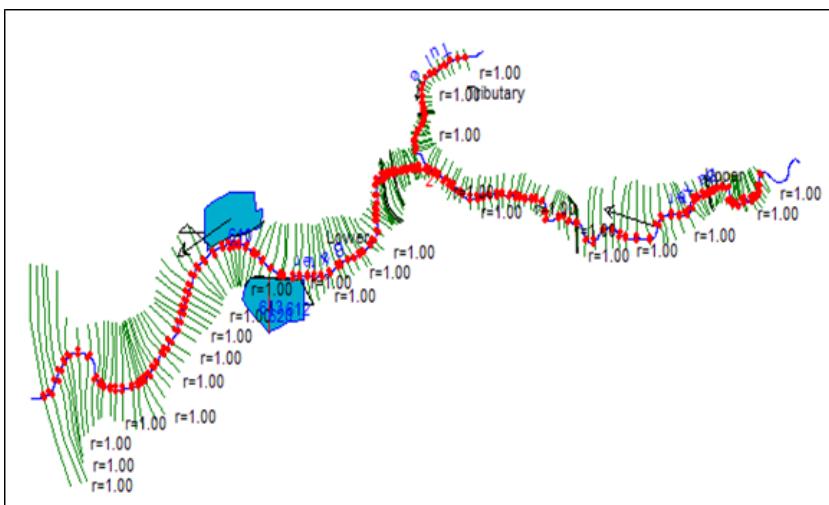


Figure 5-132. A completely georeferenced HEC-RAS model.

Attaching and Viewing Pictures

The user can attach a picture to any cross section or hydraulic structure (bridge, culvert, etc.). Once pictures are attached, they can be viewed from a picture viewer within the HEC-RAS geometric data editor. The picture viewer supports the following graphics formats: bit map (**.bmp**); icon (**.ico**); windows metafile (**.wmf**); GIF (**.gif**); and JPEG (***.jpg**).

Pictures are attached to cross sections or hydraulic structures from within the picture viewer. To bring up the picture viewer, go to the geometric data editor and click on the **View Picture** button with the left mouse button. An editor will appear as shown in Figure 5-133. To attach a picture to a particular river station, first select the River, Reach, and River Station in which you would like to attach the picture. Next select the **Add Picture** button, and a file selection box will appear allowing you to select a graphics file to attach to the selected location. If the picture file is not in the same location as your data files, you can select the drive and path of the picture from within the file selection box. Once a graphic file is located and selected, press the **Open** button to attach it to the selected location. The picture should automatically show up inside of the picture viewer. An example picture is shown in Figure 5-134. Additional pictures can be added by selecting a different location,

then select the **Add Picture** button to attach the picture. Only one picture can be attached to a model object.

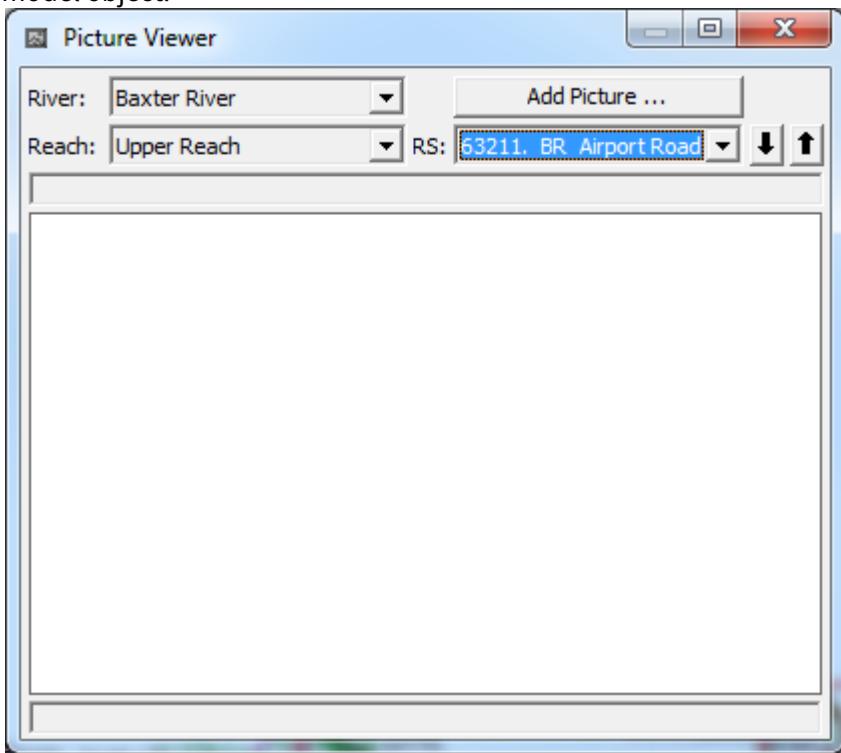


Figure 5 133. HEC-RAS Picture Viewer

Once pictures are attached to the viewer, the user can move to different pictures by using the up and down arrow buttons, or selecting a specific river stationing that has a picture attached to it. Options are available to zoom in, zoom full, and Pan by right clicking on the picture. Additionally, when a picture is loaded, the **Add Picture** button changes to **Remove Picture** in order to delete it. The user can resize the picture viewer to whatever size they want. However, if you are viewing a bitmap picture, and you make the window larger than the actual picture resolution, the photo will begin to distort.

Once pictures are attached to the geometry file, a small red square will be displayed on the river system schematic at each location where a picture exists. When the user clicks the left mouse button over a cross section, a pop up menu will appear. If that particular cross section has a picture attached to it, one of the menu options will be to view the picture. Selecting the **View Picture** option from the pop up menu will bring up the picture viewer and automatically load that particular picture.

The pictures are stored as part of the geometry data (not the actual picture, but its location on the hard disk). In general, it is a good idea to keep the picture files in the same directory as your project data files. This will make it easier to keep track of all the files associated with a particular project.

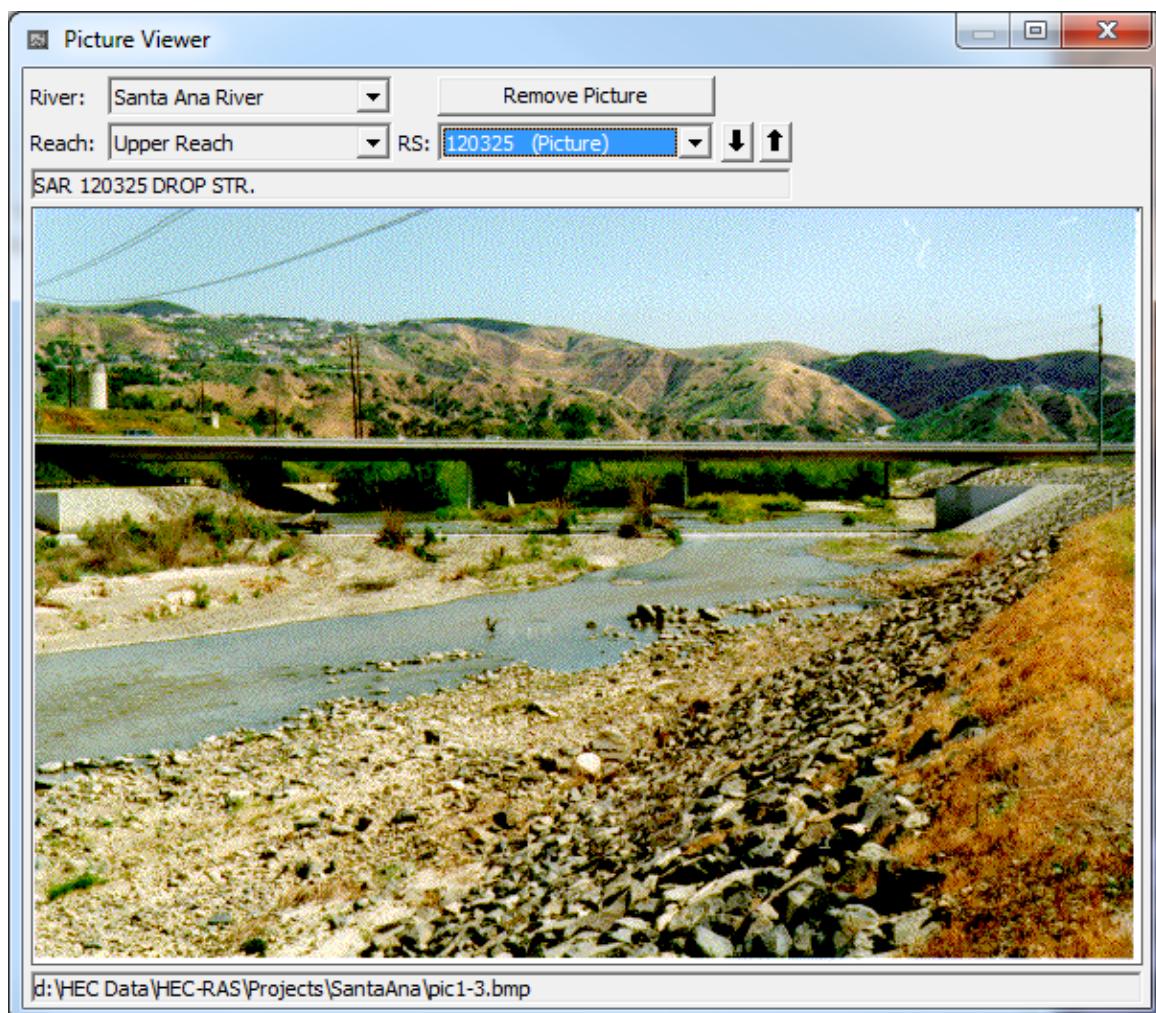


Figure 5 134. Picture Viewer with Example Bit Map Photo

Saving the Geometric Data

To save the geometric data, use the **Save Geometry Data As** option from the **File** menu of the Geometric Data window. When this option is selected, the user is prompted to enter a title for the geometric data. Once you have entered the title, press the **OK** button and the data will be saved to the hard disk. If the geometric data have been saved before (and therefore a title has already been entered), then it is only necessary to select the Save Geometry Data option. When this option is selected, the geometry data are saved with the previously defined title.

In general, it is a good idea to periodically save your data as you are entering them. This will prevent the loss of large amounts of information in the event of a power failure, or if a program error occurs in the HEC-RAS user interface.