

Overview of Optional Capabilities

Alex Sanchez

USACE, Institute for Water Resources, Hydrologic Engineering Center



US Army Corps
of Engineers®



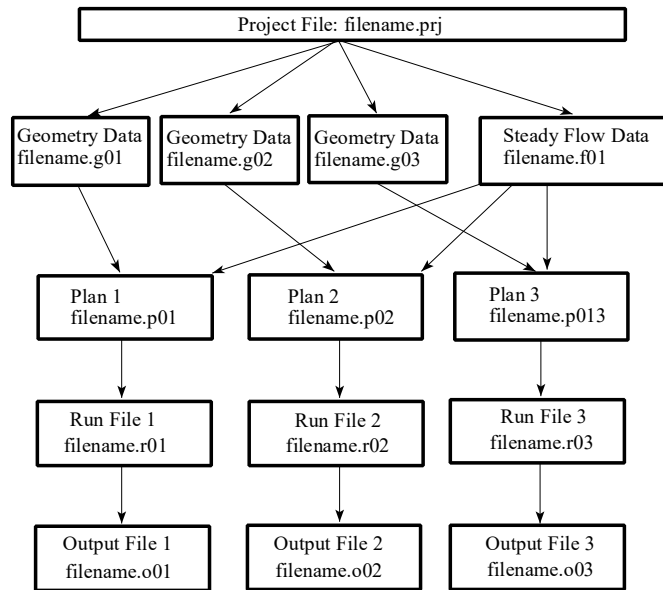


Optional Capabilities

- Multiple Plan Analysis
- Cross Section Interpolation
- Mixed Flow Regime Calculations
- Flow Distribution Calculations
- Inline Weirs and Gated Spillways



Multiple Plan Analysis



The HEC-RAS system has the ability to compute a series of water surface profiles for a number of different characterizations (**Plans**) of the river system. Modifications can be made to the geometry and/or flow data, and then saved in separate files. Plans are then formulated by selecting a particular geometry file and a particular flow file. The multiple plan option is useful when, for example, a comparison of existing conditions and future channel modifications are to be analyzed. Channel modifications can consist of any change in the geometric data, such as: the addition of a bridge or culvert; channel improvements; the addition of levees; changes in n values due to development or changes in vegetation; etc... The multiple plan option can also be used to perform a design of a specific geometric feature. For example, if you were sizing a bridge opening, a separate geometry file could be developed for a base condition (no bridge), and then separate geometry files could be developed for each possible bridge configuration. A plan would then consist of selecting a flow file and one of the geometry files. Computations are performed for each plan individually. Once the computations are performed for all the plans, the user can then view output in a graphical and tabular mode for any single plan or combination of plans.



Cross Section Interpolation May Be Needed When:

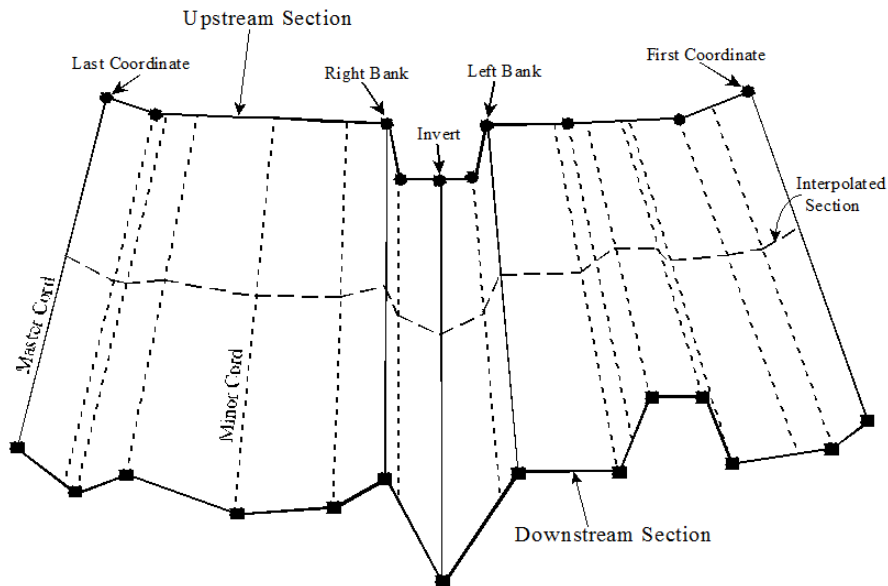
- The change in velocity head is too large
- To better model friction losses
- To prevent the program from defaulting to critical depth
- To generate a smoother plot of the water surface profile

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 change in 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. When cross sections are spaced too far apart, the program may end up defaulting to critical depth.

The HEC-RAS program has the ability to generate cross sections by interpolating the geometry between two user entered cross sections. The geometric interpolation routines in HEC-RAS are based on a string model, as shown on the next slide.



Cross Section Interpolation



The string model in HEC-RAS consists of cords that connect the coordinates of the upstream and downstream cross sections. The cords are classified as "Master Cords" and "Minor Cords". The master cords are defined explicitly as to the number and starting and ending location of each cord. The default number of master cords is five. The five default master cords are based on the following location criteria:

1. First coordinate of the cross section (May be equal to left bank).
2. Left bank of main channel (Required to be a master cord).
3. Minimum elevation point in the main channel.
4. Right bank of main channel (Required to be a master cord).
5. Last coordinate of the cross section (May be equal to right bank).

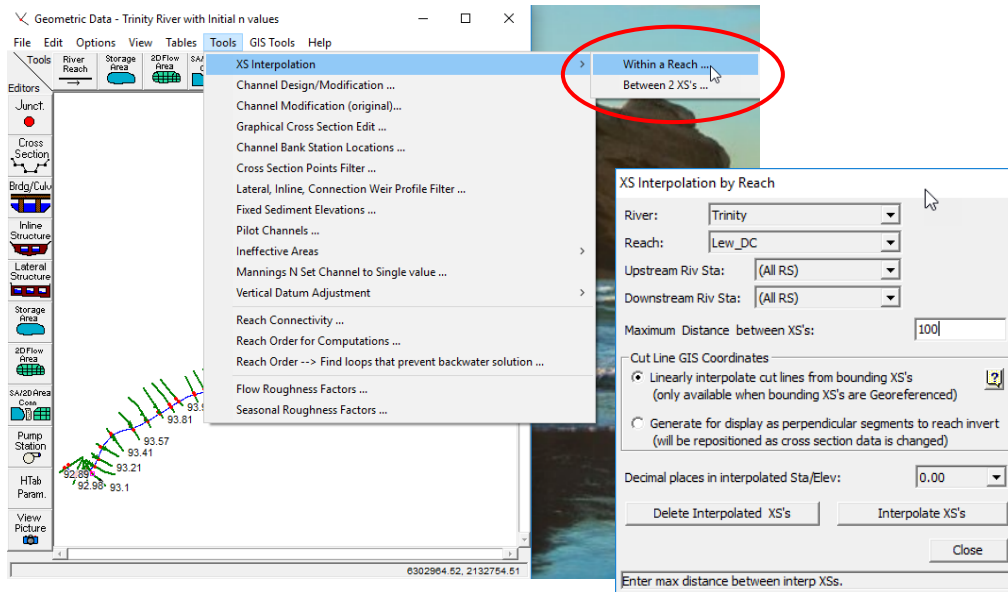
The interpolation routines are not restricted to a set number of master cords. At a minimum there must be two master cords, but there is no maximum. Additional master cords can be added by the user.

The 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 proportional distance that the known coordinate represents between master chords, and then applying the proportion to the distance between master cords of the opposite section. The number of minor cords will be equal to the sum of all the coordinates in the upstream and downstream sections minus the number of master cords.

Once all the minor cords are computed, the routines can then interpolate any number of sections between the two known cross sections. Interpolation is accomplished by linearly interpolating between the elevations at the ends of a cord. Interpolated points are generated at all of the minor and master cords. The elevation of a particular point is computed by distance weighting, which is based on how far the interpolated cross section is from the user known cross sections.



Interpolation within a Reach

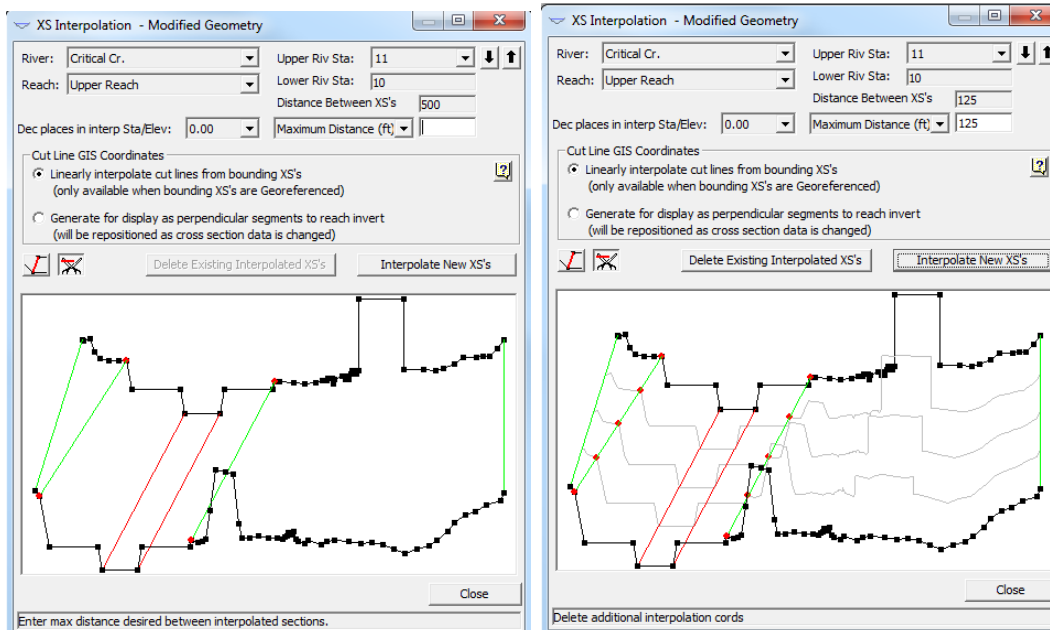


The first cross section interpolation option, **Within a Reach**, allows for automatic interpolation over a specified range of cross sections within a single reach. When this option is selected, a window will pop up as shown above. The user must first select the River and Reach that they would like to perform the interpolation in. Next the user must enter 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.

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 tic marks. 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.



Interpolation between 2 XS's

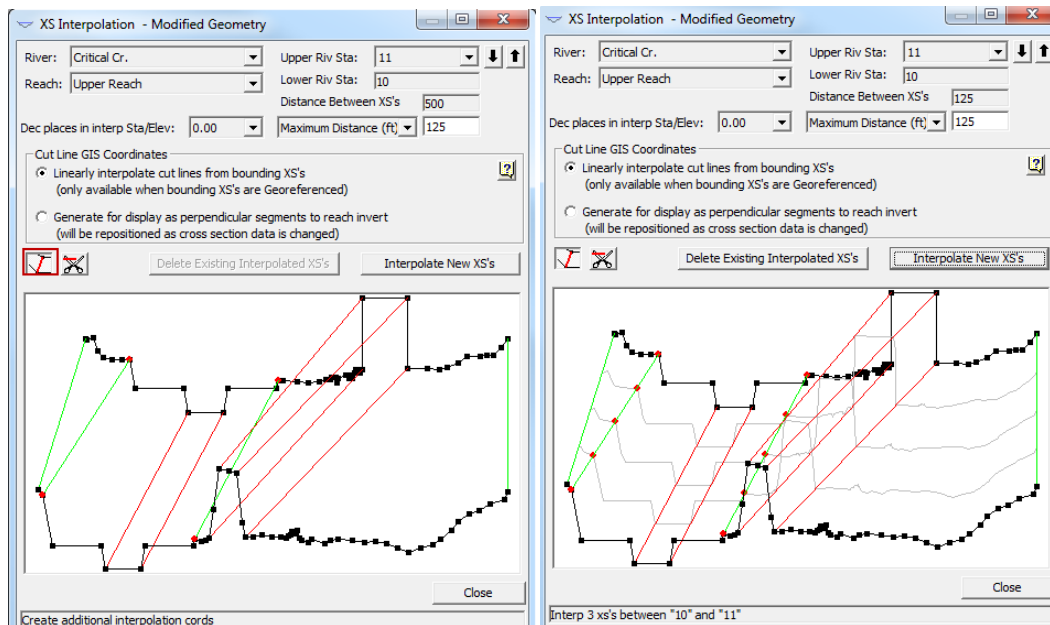


The same interpolation scheme is used in both of the automated interpolation options ("Within a Reach" and "Between 2 XS's"). The difference is that the **Between 2 XS's** option allows the user to define additional master cords. This can provide for a better interpolation, especially when the default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

As can be seen, 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.



Adding Additional Master Cords



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 top 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.

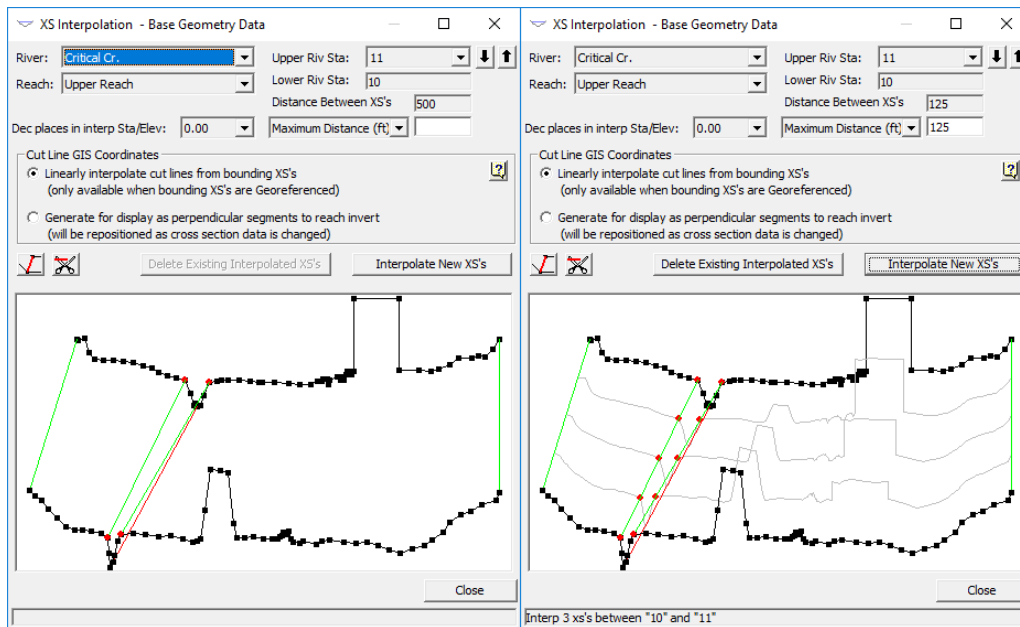
User defined master cords can also be deleted. To delete user defined master cords, press the **scissors** button to the right of the master cords button. When this button is pressed, simply move the mouse pointer over a user defined cord and click the left mouse button to delete the cord.

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.

In general, the best approach for cross section interpolation is to first interpolate sections using the **"Within a Reach"** method. Next, 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 sections can then be developed by adding additional master cords in order to improve the interpolation.



Interpolation between 2 XS's

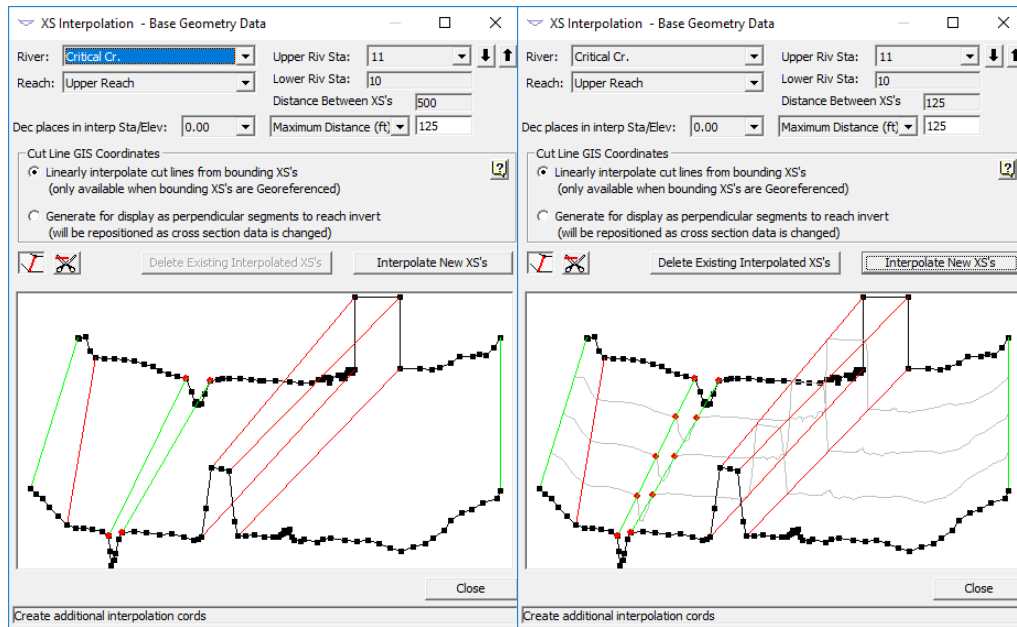


The same interpolation scheme is used in both of the automated interpolation options ("Within a Reach" and "Between 2 XS's"). The difference is that the **Between 2 XS's** option allows the user to define additional master cords. This can provide for a better interpolation, especially when the default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

As can be seen, 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.



Adding Additional Master Cords



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 top 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.

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.

In general, the best approach for cross section interpolation is to first interpolate sections using the "**Within a Reach**" method. Next, 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 sections can then be developed by adding additional master cords in order to improve the interpolation.

Cross-Section Interpolation

CAUTION: Automatic interpolation should not be used as a replacement for required cross section data!



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.



Mixed Flow Regime

- Subcritical Flow
- Supercritical Flow
- Mixed Flow

The HEC-RAS software has the ability to perform subcritical, supercritical, or mixed flow regime calculations.



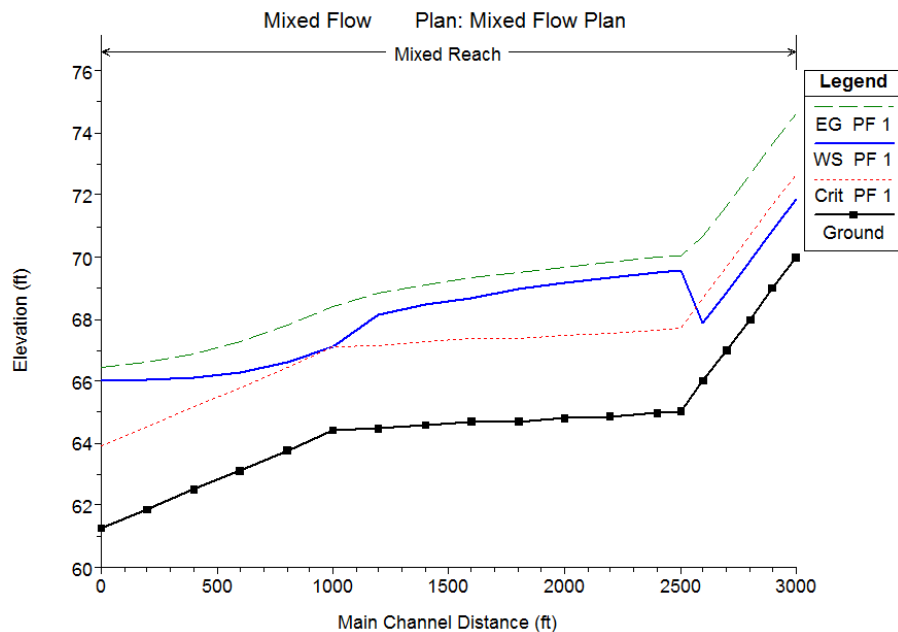
Specific Force Equation

$$\frac{Q_1^2 B_1}{g A_1} + A_1 \bar{Y}_1 = \frac{Q_2^2 B_2}{g A_2} + A_2 \bar{Y}_2$$

- Q = Discharge at each section
- B = Momentum coefficient (similar to alpha)
- A = Total flow area
- Y = Depth from the water surface to centroid of the area
- g = Gravitational acceleration



Mixed Flow



The mixed flow regime calculations in HEC-RAS are performed as follows:

First, a subcritical water surface profile is computed starting from a known downstream boundary condition.

Next, the program attempts to compute a supercritical profile. The program will start with the upstream end and compute a supercritical profile as long as the supercritical profile has a greater specific force than the previously computed subcritical answer at that location.

If, at a given location, the subcritical water surface has a greater specific force, then the program will keep the subcritical answer. In this case, the program will start searching downstream for a cross section that has a critical depth answer. From the critical depth cross section, the program will once again compute a supercritical profile as long as the supercritical answer has a greater specific force.

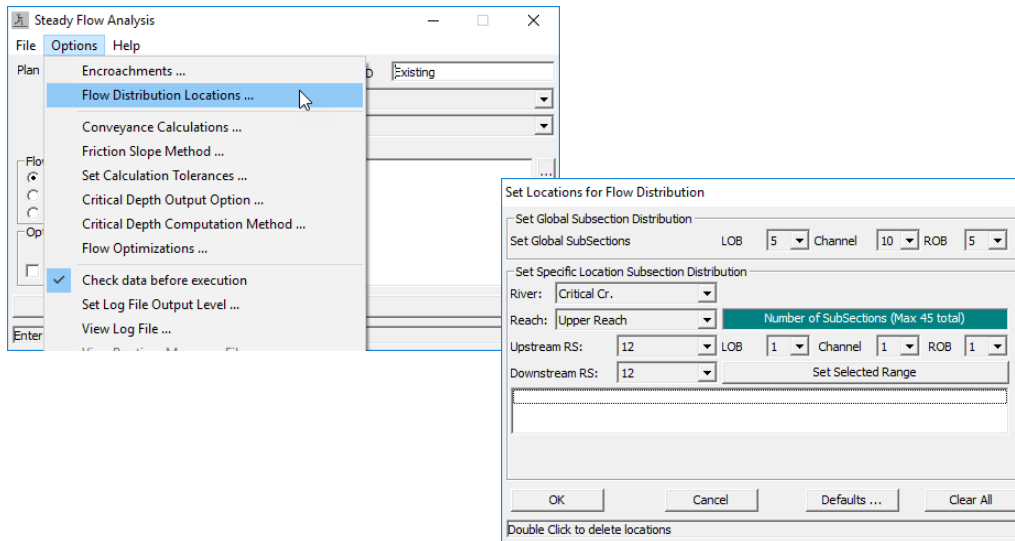


Flow Distribution Option

The flow distribution option allows the user to see the amount of flow, and the flow velocity, in different parts of the cross section.



Setting the Flow Distribution Locations

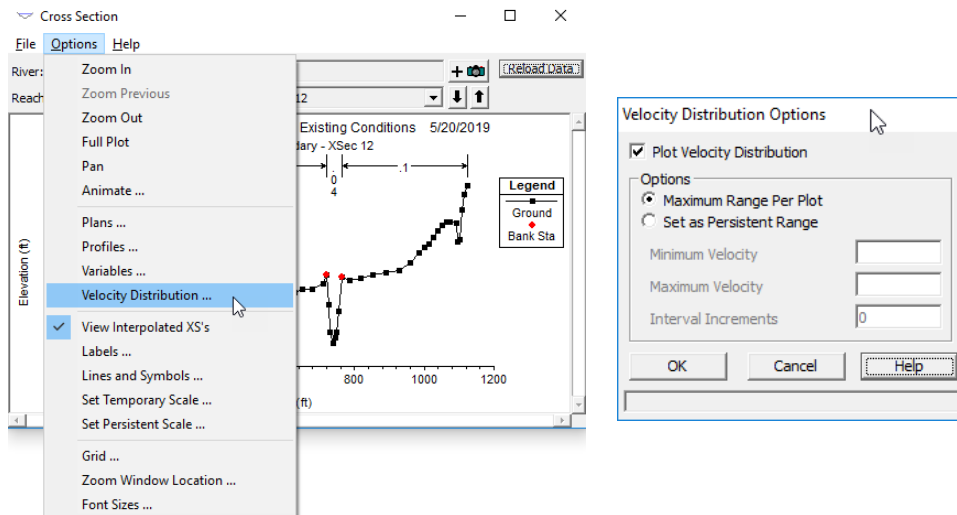


The flow distribution output can be obtained by first defining the locations that the user would like to have this type of output. As shown above, the user can either select specific locations or all locations in the model. Next, the number of slices for the flow distribution computations must be defined for the left overbank, main channel, and the right overbank. The user can define up to 45 total slices. Each flow element (left overbank, main channel, and right overbank) must have at least one slice. The user can change the number of slices used at each of the cross sections. The flow distribution output will be calculated for all profiles in the plan during the computations.

The final step is to perform the normal profile calculations. During the computations, at each cross section where flow distribution is requested, the program will calculate the percentage of flow, area, wetted perimeter, conveyance, and average velocity for each of the user defined slices.



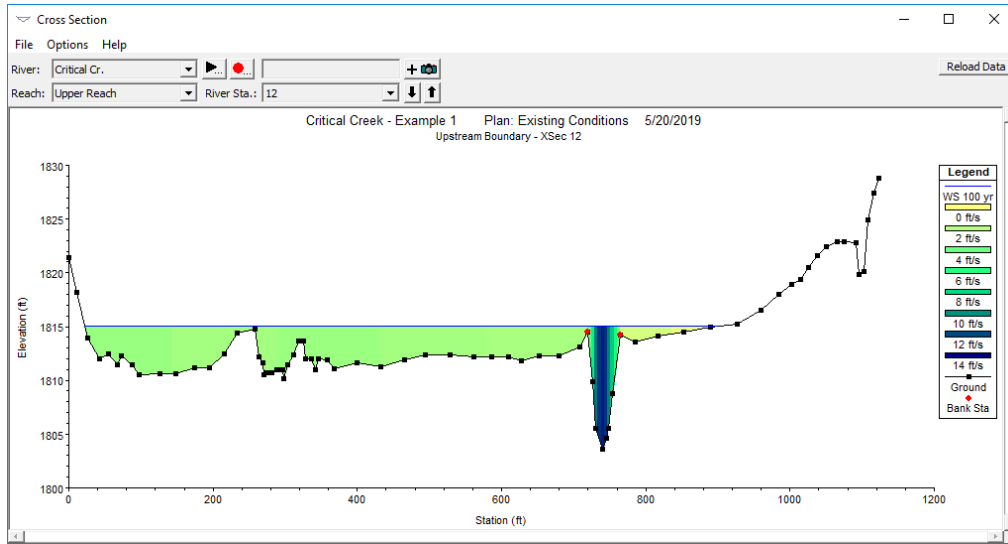
Plotting Velocity Distribution



The user has the option of plotting velocity distribution output from the cross section viewer. Velocity distributions can only be plotted at locations in which the user specified that flow distribution output be calculated during the computations. To view the velocity distribution plot, first bring up a cross section plot (select “Cross Sections” from the view menu of the main HEC-RAS window). Next, select the cross section in which you would like to see the velocity distribution output. Select **Velocity Distribution** from the **Options** menu of the cross section window. This will bring up a popup window that will allow you to set the minimum velocity, maximum; velocity, and velocity increment for plotting. In general, it is better to let the program use the maximum velocity range for plotting. Next, the user selects **Plot Velocity Distribution**. Then press the “OK” button and the velocity distribution plot will appear.



Velocity Distribution Plot





Flow Distribution Output



Flow Distribution Output

File Type Options Help

River: Critical Cr. Profile: 100 yr

Reach: Upper Reach RS: 12 Plan: Exist Cond

		Plan: Existing Con	Critical Cr.	Upper Reach	RS: 12	Profile: 100 yr					
	Pos	Left Sta (ft)	Right Sta (ft)	Flow (cfs)	Area (sq ft)	W.P. (ft)	Percent Conv	Hydr Depth(ft)	Velocity (ft/s)	Shear (lb/sq ft)	Power (lb/ft s)
1	LOB	0.00	144.00	1188.86	424.30	122.50	13.21	3.48	2.80	1.48	4.15
2	LOB	144.00	288.00	1047.96	420.84	145.02	11.64	2.92	2.49	1.24	3.09
3	LOB	288.00	432.00	1329.09	485.54	145.17	14.77	3.37	2.74	1.43	3.92
4	LOB	432.00	576.00	1061.19	422.84	144.01	11.79	2.94	2.51	1.26	3.15
5	LOB	576.00	720.00	901.16	383.44	144.11	10.01	2.66	2.35	1.14	2.68
6	Chan	720.00	724.50	40.64	10.27	5.67	0.45	2.28	3.96	0.77	3.07
7	Chan	724.50	729.00	190.14	26.30	5.89	2.11	5.84	7.23	1.91	13.82
8	Chan	729.00	733.50	447.72	41.90	5.22	4.97	9.31	10.68	3.44	36.71
9	Chan	733.50	738.00	600.64	47.52	4.60	6.67	10.56	12.64	4.42	55.89
10	Chan	738.00	742.50	669.24	50.61	4.58	7.44	11.25	13.22	4.73	62.55
11	Chan	742.50	747.00	603.69	47.85	4.64	6.71	10.63	12.62	4.41	55.62
12	Chan	747.00	751.50	418.81	39.86	5.09	4.65	8.86	10.51	3.35	35.21
13	Chan	751.50	756.00	245.25	28.89	5.08	2.73	6.42	8.49	2.43	20.66
14	Chan	756.00	760.50	120.56	18.77	5.01	1.34	4.17	6.42	1.60	10.29
15	Chan	760.50	765.00	34.29	8.83	5.01	0.38	1.96	3.89	0.75	2.93

Errors, Warnings and Notes

Warning: The velocity head has changed by more than 0.5 ft (0.15 m). This may indicate the need for additional cross sections.

Warning: The energy loss was greater than 1.0 ft (0.3 m) between the current and previous cross section. This may indicate the need for additional cross sections.

Select Profile

The Flow Distribution table type can be used to view the computed flow distribution output at any cross section where this type of output was requested. To bring up the flow distribution table, first select **Cross Section Table** from the **View** menu of the main HEC-RAS window. Once the cross section table is on the screen, the user can select **Flow Distribution** table from the **Type** menu of the cross section table.



CAUTION

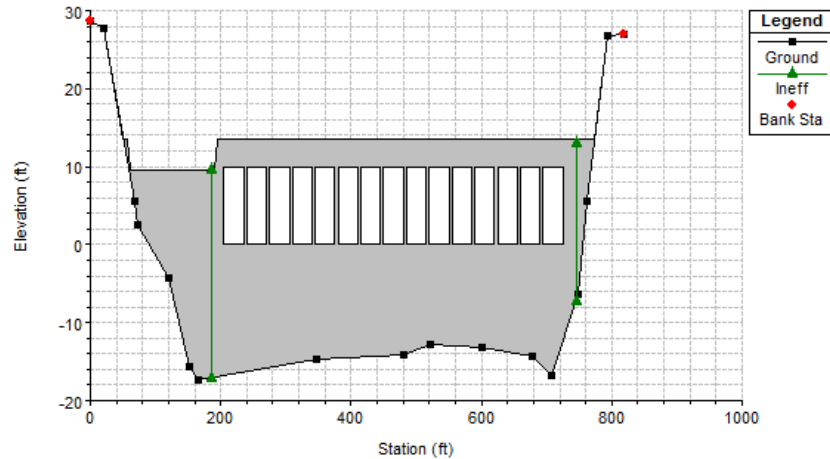
- Flow distribution varies vertically and horizontally (3-D problem)
- HEC-RAS Steady Flow is a 1-D model
- Flow distribution is based on area and wetted perimeter of each flow slice (using fewer slices is generally better)

In general, the results of the flow distribution computations should be used cautiously. Specifically, the velocities and discharges are based on a one-dimensional hydraulic model. A true velocity and flow distribution varies vertically as well as horizontally. To achieve such detail, the user would need to use a three-dimensional hydraulic model, or go out and measure the flow field. While the results for the flow distribution are better than the standard three subdivisions (left overbank, main channel, and right overbank) provided by the model, the values are still based on average estimates of the one-dimensional results. Also, the results obtained from the flow distribution option can vary with the number of slices used for the computations. In general, it is better to use as few slices as possible.



Inline Weirs and Gated Spillways

Inline structure - Example 12 Plan: Gated Spillway 11/9/2010

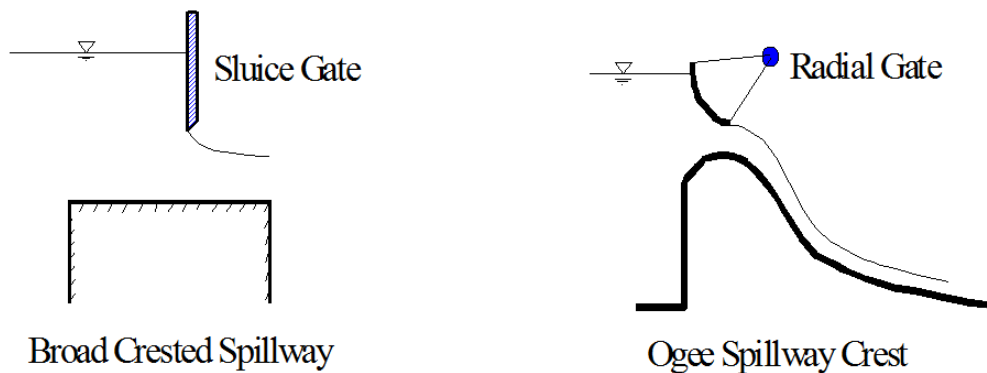


HEC-RAS allows the user to model inline gated spillways and overflow weirs. HEC-RAS has the ability to model radial gates (often called tainter gates) or vertical lift gates (sluice gates). The spillway crest of the gates can be modeled as either an ogee shape or a broad crested weir shape. In addition to the gate openings, the user can also define a separate uncontrolled overflow weir.

The overflow weir is entered as a series of station and elevation points across the stream, which allows for complicated weir shapes. The user must specify if the weir is broad crested or an ogee shape. The software has the ability to account for submergence due to the downstream tailwater. Additionally, if the weir has an ogee shaped crest, the program can calculate the appropriate weir coefficient for a given design head. The weir coefficient will automatically be decreased or increased when the actual head is lower or higher than the design head.



Sluice and Radial Gates

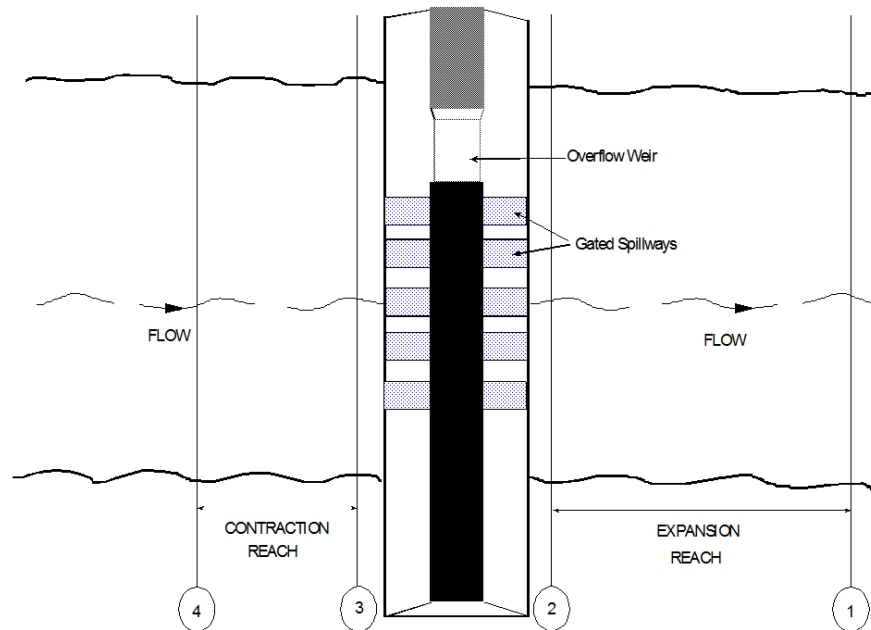


Gated Spillways within HEC-RAS can be modeled as radial gates (often called tainter gates) or vertical lift gates (sluice gates). The equations used to model the gate openings can handle both submerged and unsubmerged conditions at the inlet and outlet of the gates. If the gates are opened far enough, such that unsubmerged conditions exist at the upstream end, the program automatically switches to a weir flow equation to calculate the hydraulics of the flow. The spillway crest through the gate openings can be specified as either an ogee crest shape or a broad crested weir. The program has the ability to calculate both free flowing and submerged weir flow through the gate openings.

Up to 10 gate groups can be entered into the program at any one river crossing. Each gate group can have up to 25 identical gate openings. Identical gate openings must be the same gate type; size; elevation; and have identical gate coefficients. If anything about the gates is different, except their physical location across the stream, the gates must be entered as separate gate groups.



Cross Section Layout

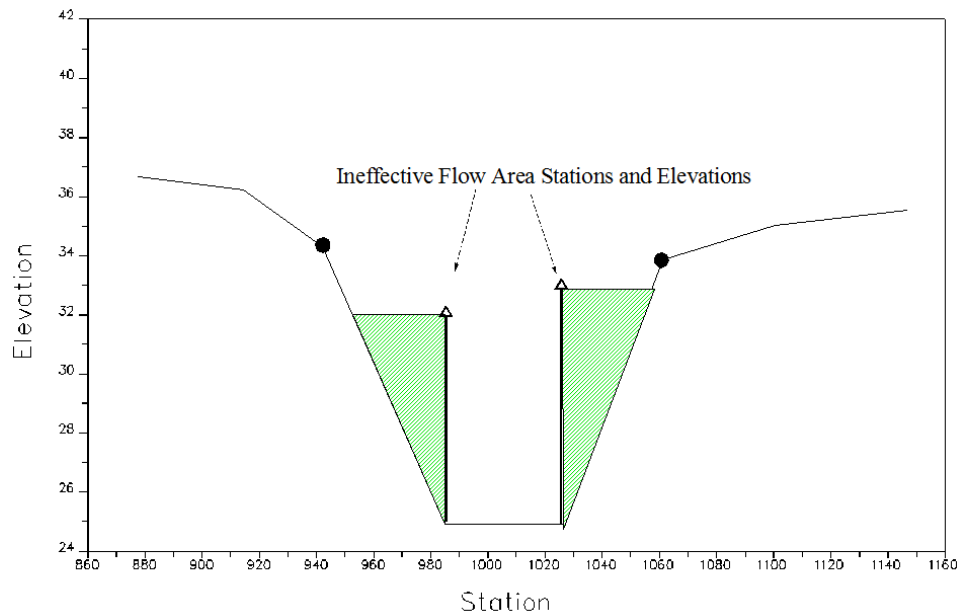


The inline weir and gated spillway routines in HEC-RAS require the same cross sections as the bridge and culvert routines. Four cross sections in the vicinity of the hydraulic structure are required for a complete model, two upstream and two downstream. In general, there should always be additional cross sections downstream from any structure (bridge, culvert, weir, etc...), such that the user entered downstream boundary condition does not affect the hydraulics of flow through the structure. In order to simplify the discussion of cross sections around the inline weir and gated spillway structure, only the four cross sections in the vicinity will be discussed. These four cross sections include: one cross section sufficiently downstream such that the flow is fully expanded; one at the downstream end of the structure (representing the tailwater location); one at the upstream end of the structure (representing the headwater location); and one cross section located far enough upstream at the point in which the flow begins to contract. Note, the cross sections that bound the structure represent the channel geometry outside of the embankment.

Cross Section 1. Cross section 1 for a weir and/or gated spillway should be located at a point where flow has fully expanded from its constricted top width caused by the constriction. The entire area of cross section 1 is usually considered to be effective in conveying flow.



Cross Section 2



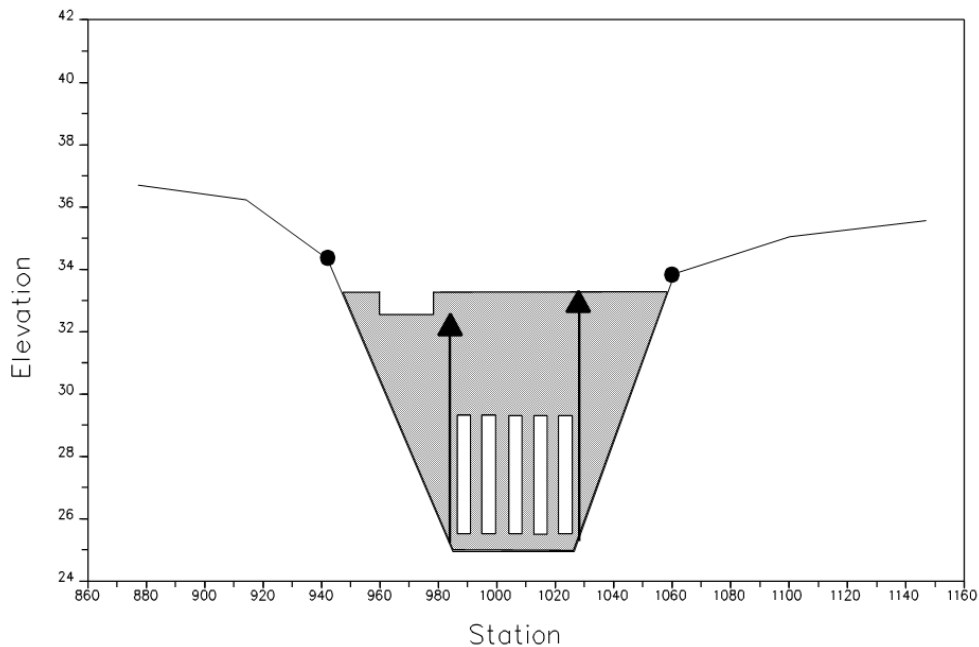
Cross Section 2. Cross section 2 is located a short distance downstream from the structure. The computed water surface at this cross section will represent the tailwater elevation of the weir and the gated spillways. This cross section should not include any of the structure or embankment, but represents the physical shape of the channel just downstream of the structure. The shape and location of this cross section is entered separately from the Inline Weir and Gated Spillway data (from the cross section editor).

The HEC-RAS ineffective area option is used to restrict the effective flow area of cross section 2 to the flow area around or near the edges of the gated spillways, until flow overtops the overflow weir and/or embankment. The ineffective flow areas are used to represent the correct amount of active flow area just downstream of the structure. Establishing the correct amount of effective flow area is very important in computing an accurate tailwater elevation at cross section 2. Because the flow will begin to expand as it exits the gated spillways, the active flow area at section 2 is generally wider than the width of the gate openings. The width of the active flow area will depend upon how far downstream cross section 2 is from the structure. In general, a reasonable assumption would be to assume a 1:1 expansion rate over this short distance.

Cross sections 1 and 2 are located so as to create a channel reach downstream of the structure in which the HEC-RAS program can accurately compute the friction losses and expansion losses that occur as the flow fully expands.



Cross Section 3



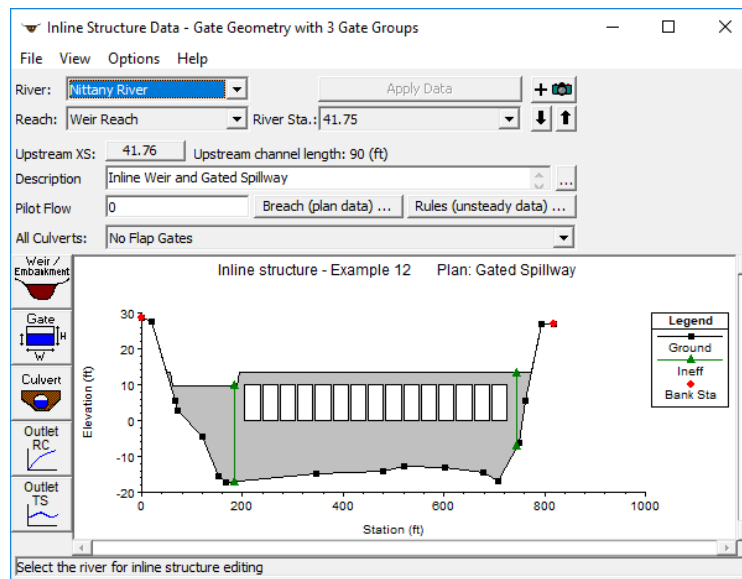
Cross Section 3. Cross section 3 of an inline weir and gated spillway model is located a short distance upstream of the embankment, and represents the physical configuration of the upstream channel. The water surface computed at this cross section represents the upstream headwater for the overflow weir and the gated spillways. The software uses a combination of the deck/road embankment data, cross section 3, and the gated spillway data, to describe the hydraulic structure and the roadway embankment. The inline weir and gated spillway data is located at a river station between cross section 2 and cross section 3.

The HEC-RAS ineffective area option is used to restrict the effective flow area of cross section 3 until the flow overtops the roadway. The ineffective flow area is used to represent the correct amount of active flow area just upstream of the structure. Because the flow is contracting rapidly as it enters the gate openings, the active flow area at section 3 is generally wider than the width of the gates. The width of the active flow area will depend upon how far upstream cross section 3 is placed from the structure. In general, a reasonable assumption would be to assume a 1:1 contraction rate over this short distance.

Cross Section 4. The final cross section in the inline weir and gated spillway model is located at a point where flow has not yet begun to contract from its unrestrained top width upstream of the structure. This distance is normally determined assuming a one to one contraction of flow. In other words, the average rate at which flow can contract to pass through the gate openings is assumed to be one foot laterally for every one foot traveled in the downstream direction. The entire area of cross section 4 is usually considered to be effective in conveying flow.



Inline Weir Editor



Inline weir and gated spillway data are entered in a similar manner as bridge and culvert data. To enter an inline weir and/or gated spillway press the **Inline Weir/Spillway** button from the Geometric Data window. Once this button is pressed, the Inline Weir and Gated Spillway Data editor will appear as shown above (except yours will be blank until you have entered some data).

To add an inline weir and/or gated spillway to a model, the user must do the following:

Select the river and reach that you would like to place this inline weir and/or spillway 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 Weir and/or Gated Spillway Data editor.

Go to the **Options** menu at the top of the window and select **Add an Inline Weir and/or Gated Spillway** 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 weir and/or gated spillway crossing.

Enter all of the data for the Inline Weir and/or Gated Spillway. This data will include a Weir/Embankment profile, and any gated spillways that you may be modeling. Gated spillways are optional.



Weir/Embankment



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 2. D.S Embankment SS 2.

Weir Data
Weir Crest Shape
 Broad Crested
 Ogee

Spillway Approach Height: 24.
Design Energy Head: 3. Cd ...

OK Cancel

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

The weir information is entered in a similar manner to the bridge and culvert data.

The station data is entered from left to right. Everything below this elevations will be filled in down to the ground.

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 weir/embankment, and are not effected by this elevation. If this field is left blank, the elevation that triggers weir flow is based on the lowest elevation of the station and elevation coordinates.



Gate Editor



The screenshot shows the 'Inline Gate Editor' dialog box with the following settings:

- Gate Group: Left Group
- Gate type (or methodology): Radial
- Gate Flow: Radial Gate Flow
 - Radial Discharge Coefficient: 0.8
 - Trunnion Exponent: 0.16
 - Opening Exponent: 0.72
 - Head Exponent: 0.62
 - Trunnion Height: 10
- Submerged Orifice Flow: Orifice Coefficient (typically 0.8): 0.8
- Head Reference: Sill (Invert)
- Weir Flow Over Gate Sill (gate out of water):
 - Weir Shape: Ogee
 - Weir Coefficient: 3.91
 - Spillway Approach Height: 1.4
 - Design Energy Head: 10
- Geometric Properties: Height: 10, Width: 30, Invert: 0
- Opening Centerline Stations: # Openings: 5
- Opening GIS Data: Length: (empty table)

Opening Name	Station	GIS Sta
1 Opening #1	220	
2 Opening #2	255	
3 Opening #3	290	
4 Opening #4	325	
5 Opening #5	360	
6		
7		

Gate Group - The Gate Group is automatically assigned to "Gate #1" the first time you open the editor. The user can enter up to 10 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, then only one gate group needs to be entered. If the user has gate openings that are different in shape, size, elevation, or have different coefficients, then additional Gate Groups must be added for each Gate type. 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.

Centerline Stations - 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".

Weir Data and Coefficient – When the gate is opened to an elevation higher than the upstream water surface, the flow is treated as weir flow and the program uses the Weir Data information. For sluice type flow, the program uses the Discharge Coefficient. When the flow is fully submerged, the Orifice Coefficient is used.



Setting the Gate Opening

The screenshot shows the 'Steady Flow Data - 7 Flow Profiles' window with the 'Options' menu open. The 'Gate Openings...' option is selected. A 'Spillway Gate Openings' dialog box is also open, showing the following table:

Gate Group	# Openings	Gate Ht (ft)	PF#1		PF#2		PF#3
			# Open	Open Ht	# Open	Open Ht	# Open
Left Group	5	10	0	0	2	3	3
Center Group	5	10	5	5	5	5	5
Right Group	5	10	0	0	2	3	3

The number of gates that are open, and the opening height, is set for each gate group for each profile.