CHAPTER 4

Overview of Optional Capabilities

HEC-RAS has numerous optional capabilities that allow the user to model unique situations. These capabilities include: multiple profile analysis; multiple plan analysis; optional friction loss equations; cross section interpolation; mixed flow regime calculations; modeling stream junctions; flow distribution calculations; and split flow optimization.

Contents

Multiple Profile Analysis

HEC-RAS can compute up to 500 profiles, for the same geometric data, within a single execution of the steady flow computations. The number of profiles to be computed is defined as part of the steady flow data. When more than one profile is requested, the user must ensure that flow data and boundary conditions are established for each profile. Once a multiple profile computation is made, the user can view output, in a graphical and tabular mode, for any single profile or combination of profiles.

For an unsteady flow analysis, the user can have detailed output computed for the maximum water surface profile, as well as profiles that represent specific instances in time during the unsteady flow simulation. The user can request detailed output for up to 500 specific time slices.

Warning, as the number of profiles (steady flow) or time slices (unsteady flow) is increased, the size of the output files will also increase.

Multiple Plan Analysis

The HEC-RAS system has the ability to compute 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.

Optional Friction Loss Equations

This option is limited to steady flow water surface profile calculations. The friction loss between adjacent cross sections is computed as the product of the representative rate of friction loss (friction slope) and the weighted-average reach length. The program allows the user to select from the following previously defined friction loss equations:

- Average Conveyance (Equation 2-13)
- Average Friction Slope (Equation 2-14)
- Geometric Mean Friction Slope (Equation 2-15)
- Harmonic Mean Friction Slope (Equation 2-16)

Any of the above friction loss equations will produce satisfactory estimates provided that reach lengths are not too long. The advantage sought in alternative friction loss formulations is to be able to maximize reach lengths without sacrificing profile accuracy.

Equation 2-13, the average conveyance equation, is the friction loss formulation that has been set as the default method within HEC-RAS. This equation is viewed as giving the best overall results for a range of profile types (M1, M2, etc). Research (Reed and Wolfkill, 1976) indicates that Equation 2-14 is the most suitable for M1 profiles. (Suitability as indicated by Reed and Wolfkill is the most accurate determination of a known profile with the least number of cross sections.) Equation 2-15 is the standard friction loss formulation used in the FHWA/USGS step-backwater program WSPRO (Sherman, 1990). Equation 2-16 has been shown by Reed and Wolfkill to be the most suitable for M2 profiles.

Another feature of this capability is to select the most appropriate of the preceding four equations on a cross section by cross section basis depending on flow conditions (e.g., M1, S1, etc.) within the reach. At present, however, the criteria for this automated method (shown in Table 4.1), does not select the best equation for friction loss analysis in reaches with significant lateral expansion, such as the reach below a contracted bridge opening.

The selection of friction loss equations is accomplished from the Options menu on the Steady Flow Analysis window.

Table 4.1
Criteria Utilized to Select Friction Equation

Profile Type	Is friction slope at current cross section greater than friction slope at preceding cross section?	Equation Used
Subcritical (M1, S1)	Yes	Average Friction Slope (2-14)
Subcritical (M2)	No	Harmonic Mean (2-16)
Supercritical (S2)	Yes	Average Friction Slope (2-14)
Supercritical (M3, S3)	No	Geometric Mean (2-15)

Cross Section Interpolation

Occasionally it is necessary to supplement surveyed cross section data by interpolating cross sections 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 in Figure 4.1

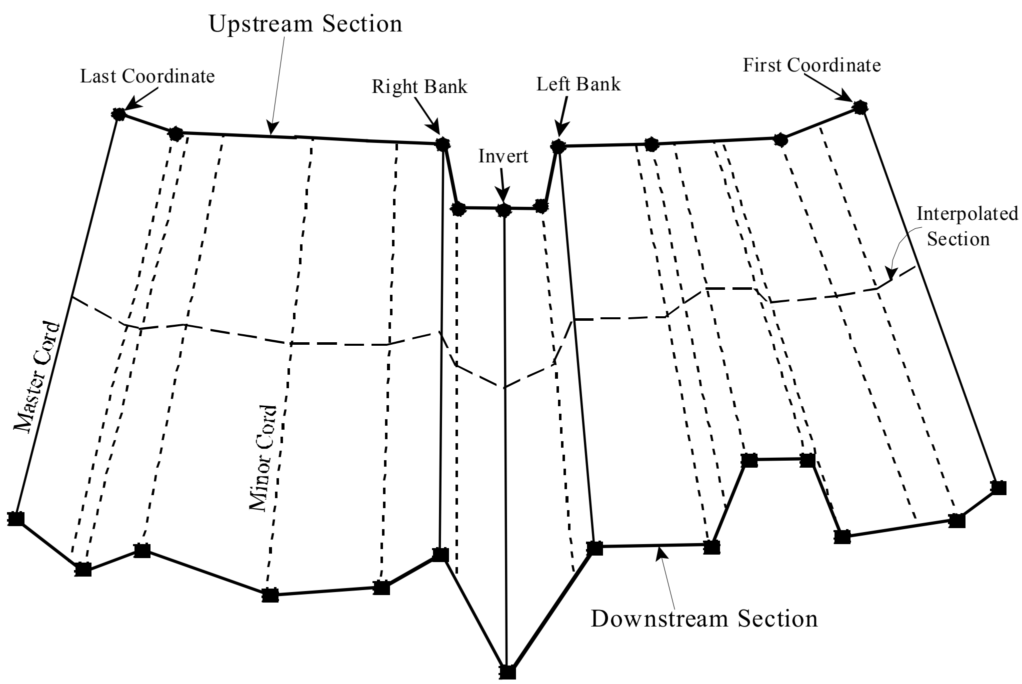


Figure 4.1 String Model for Geometric Cross Section Interpolation

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. This is explained in Chapter 6 of the HEC-RAS user's manual, under cross section interpolation.

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.

The interpolation routines will also interpolate roughness coefficients (Manning's n). Interpolated cross section roughness is based on a string model similar to the one used for geometry. Cords are used to connect the breaks in roughness coefficients of the upstream and downstream sections. The cords are also classified as master and minor cords. The default number of master cords is set to four, and are located based on the following criteria:

1. First coordinate of the cross section (may be equal to left bank).

2. Left bank of main channel.

3. Right bank of main channel.

4. Last coordinate of the cross section (may be equal to right bank).

When either of the two cross sections has more than three n values, additional minor cords are added at all other n value break points. Interpolation of roughness coefficients is then accomplished in the same manner as the geometry interpolation.

In addition to the Manning's n values, the following information is interpolated automatically for each generated cross section: downstream reach lengths; main channel bank stations; contraction and expansion coefficients; normal ineffective flow areas; levees; and normal blocked obstructions. Ineffective flow areas, levees, and blocked obstructions are only interpolated if both of the user-entered cross sections have these features turned on.

Cross section interpolation is accomplished from the user interface. To learn how to perform the interpolation, review the section on interpolating in Chapter 6 of the HEC-RAS user's manual.

Mixed Flow Regime Calculations

The HEC-RAS software has the ability to perform subcritical, supercritical, or mixed flow regime calculations. The Specific Force equation is used in HEC-RAS to determine which flow regime is controlling, as well as locating any hydraulic jumps. The equation for Specific Force is derived from the momentum equation (Equation 2-37). When applying the momentum equation to a very short reach of river, the external force of friction and the force due to the weight of water are very small, and can be ignored. The momentum equation then reduces to the following equation:

$$\frac{Q_1^2 \beta_1}{g A_1} + A_1 \overline{Y}_1 = \frac{Q_2^2 \beta_2}{g A_2} + A_2 \overline{Y}_2 \qquad (4\text{-}1)$$

Where: Q = Discharge at each section
β = Momentum coefficient (similar to alpha)
A = Total flow area
$\overline{Y}$ = Depth from the water surface to centroid of the area
g = Gravitational acceleration

The two sides of the equation are analogous, and may be expressed for any channel section as a general function:

$$SF = \frac{Q^2 \beta}{g A} + A\overline{Y} \tag{4-2}$$

The generalized function (equation 4-2) consists of two terms. The first term is the momentum of the flow passing through the channel cross section per unit time. This portion of the equation is considered the dynamic component. The second term represents the momentum of the static component, which is the force exerted by the hydrostatic pressure of the water. Both terms are essentially a force per unit weight of water. The sum of the two terms is called the Specific Force (Chow, 1959).

When the specific force equation is applied to natural channels, it is written in the following manner:

$$SF = \frac{Q^2 \beta}{g A_m} + A_t \overline{Y} \tag{4-3}$$

Where: A_m = Flow area in which there is motion
A_t = Total flow area, including ineffective flow areas

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

1. First, a subcritical water surface profile is computed starting from a known downstream boundary condition. During the subcritical calculations, all locations where the program defaults to critical depth are flagged for further analysis.

2. Next the program begins a supercritical profile calculation starting upstream. The program starts with a user specified upstream boundary condition. If the boundary condition is supercritical, the program checks to see if it has a greater specific force than the previously computed subcritical water surface at this location. If the supercritical boundary condition has a greater specific force, then it is assumed to control, and the program will begin calculating a supercritical profile from this section. If the subcritical answer has a greater specific force, then the program begins searching downstream to find a location where the program defaulted to critical depth in the subcritical run. When a critical depth is located, the program uses it as a boundary condition to begin a supercritical profile calculation.

3. The program calculates a supercritical profile in the downstream direction until it reaches a cross section that has both a valid subcritical and a supercritical answer. When this occurs, the program calculates the specific force of both computed water surface elevations. Whichever answer has the greater specific force is considered to be the correct solution. If the supercritical answer has a greater specific force, the program continues making supercritical calculations in the downstream direction and comparing the specific force of the two solutions. When the program reaches a cross section whose subcritical answer has a greater specific force than the supercritical answer, the program assumes that a hydraulic jump occurred between that section and the previous cross section.

4. The program then goes to the next downstream location that has a critical depth answer and continues the process.

An example mixed flow profile, from HEC-RAS, is shown in Figure 4.2. This example was adapted from problem 9-8, page 245, in Chow's "Open Channel Hydraulics" (Chow, 1959).

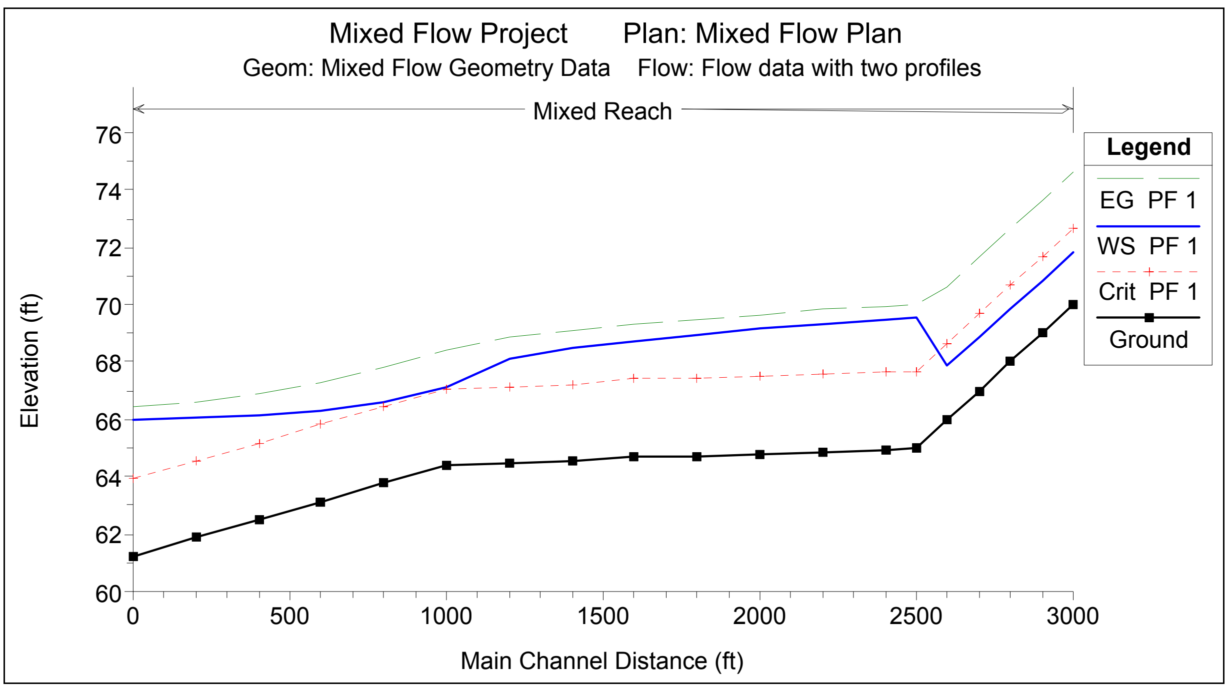


Figure 4.2 Example Mixed Flow Regime Profile from HEC-RAS

As shown in Figure 4.2, the flow regime transitions from supercritical to subcritical just before the first break in slope.

Modeling Stream Junctions

This option is only available for steady flow water surface profile calculations. Stream junctions can be modeled in two different ways within HEC-RAS. The default method is an energy based solution. This method solves for water surfaces across the junction by performing standard step backwater and forewater calculations through the junction. The method does not account for the angle of any of the tributary flows. Because most streams are highly subcritical flow, the influence of the tributary flow angle is often insignificant. If the angle of the tributary plays an important role in influencing the water surface around the junction, then the user should switch to the alternative method available in HEC-RAS, which is a momentum based method. The momentum based method is a one dimensional formulation of the momentum equation, but the angles of the tributaries are used to evaluate the forces associated with the tributary flows. There are six possible flow conditions that HEC-RAS can handle at a junction:

1. Subcritical flow - flow combining
2. Subcritical flow - flow split
3. Supercritical flow - flow combining
4. Supercritical flow- flow split
5. Mixed flow regime - flow combining
6. Mixed flow regime - flow split

The most common situations are the subcritical flow cases (1) and (2). The following is a discussion of how the energy method and the momentum based method are applied to these six flow cases.

Energy Based Junction Method

The energy-based method solves for water surfaces across the junction by performing standard step calculations with the one dimensional energy equation (Equation 2-1). Each of the six cases are discussed individually.

Case 1: *Subcritical Flow - Flow Combining.*

An example junction with flow combining is shown in Figure 4.3. In this case, subcritical flow calculations are performed up to the most upstream section of reach 3. From here, backwater calculations are performed separately across the junction for each of the two upstream reaches. The water surface at reach 1, station 4.0 is calculated by performing a balance of energy from station 3.0 to 4.0. Friction losses are based on the length from station 4.0 to 3.0 and the average friction slope between the two sections. Contraction or expansion losses are also evaluated across the junction. The water surface for the downstream end of reach 2 is calculated in the same manner. The energy equation from station 3.0 to 4.0 is written as follows:

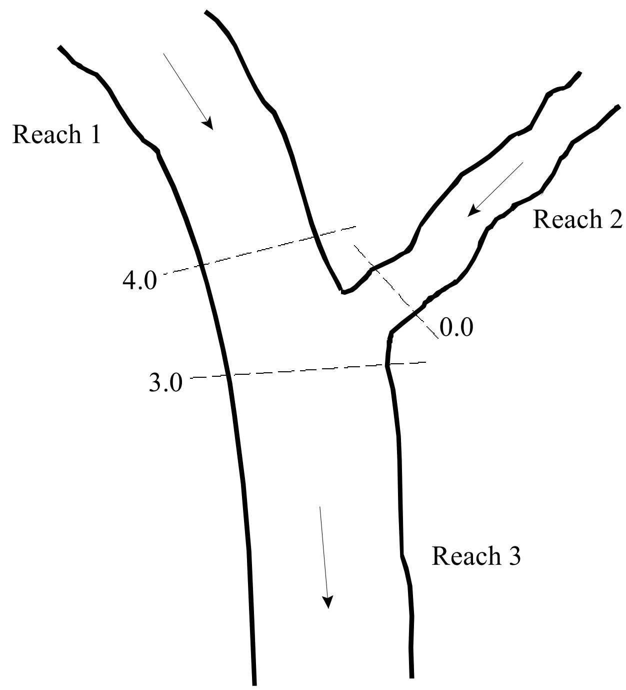


Figure 4.3 Example Junction with Flow Combining.

$$WS_4 + \frac{\alpha_4 V_4^2}{2g} = WS_3 + \frac{\alpha_3 V_3^2}{2g} + L_{4-3}\,\overline{S}_{f_{4-3}} + C\left|\frac{\alpha_4 V_4^2}{2g} - \frac{\alpha_3 V_3^2}{2g}\right| \qquad (4\text{-}4)$$

Case 2: *Subcritical Flow - Flow Split*

For this case, a subcritical water surface profile is calculated for both reaches 2 and 3, up to river stations 2.0 and 3.0 (see Figure 4.4). The program then calculates the specific force (momentum) at the two locations. The cross section with the greater specific force is used as the downstream boundary for calculating the water surface across the junction at river station 4.0. For example, if cross section 3.0 had a greater specific force than section 2.0, the program will compute a backwater profile from station 3.0 to station 4.0 in order to get the water surface at 4.0.

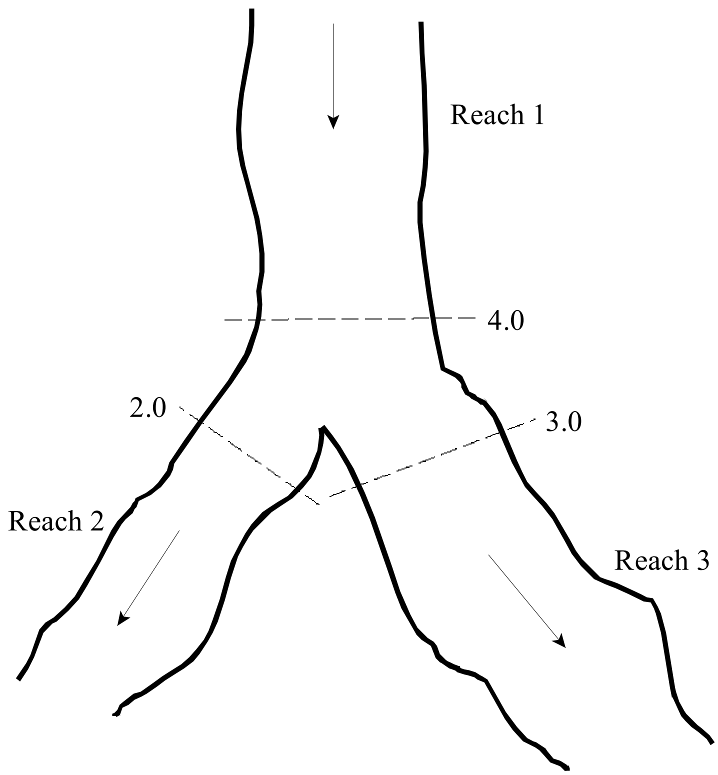


Figure 4.4 Example Flow Split at a Junction

Currently the HEC-RAS program assumes that the user has entered the correct flow for each of the three reaches. In general, the amount of flow going to reach 2 and reach 3 is unknown. In order to obtain the correct flow distribution at the flow split, the user must perform a trial and error process. This procedure involves the following:

1. Assume an initial flow split at the junction.

2. Run the program in order to get energies and water surfaces at all the locations around the junction.

3. Compare the energy at stations 2.0 and 3.0. If they differ by a significant magnitude, then the flow distribution is incorrect. Re-distribute the flow by putting more flow into the reach that had the lower energy.

4. Run the program again and compare the energies. If the energy at stations 2.0 and 3.0 still differ significantly, then re-distribute the flow again.

5. Keep doing this until the energies at stations 2.0 and 3.0 are within a reasonable tolerance.

Ideally it would be better to perform a backwater from station 2.0 to 4.0 and also from station 3.0 to 4.0, and then compare the two computed energies at the same location. Since the program only computes one energy at station 4.0, the user must compare the energies at the downstream cross sections. This procedure assumes that the cross sections around the junction are spaced closely together.

Case 3: *Supercritical Flow - Flow Combining*

In this case, a supercritical water surface profile is calculated for all of reach 1 and 2, down to stations 4.0 and 0.0 (see Figure 4.5). The program calculates the specific force at stations 4.0 and 0.0, and then takes the stream with the larger specific force as the controlling stream. A supercritical forewater calculation is made from the controlling upstream section down to station 3.0.

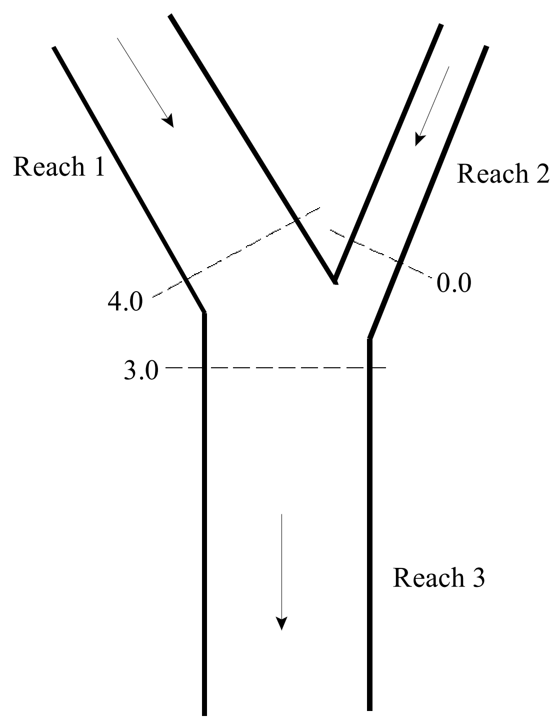


Figure 4.5 Example Supercritical Flow Combine

Case 4: *Supercritical Flow - Flow Split*

In this case a supercritical water surface profile is calculated down to station 4.0 of reach 1 (see Figure 4.6). The water surfaces at sections 3.0 and 2.0 are calculated by performing separate forewater calculations from station 4.0 to station 2.0, and then from station 4.0 to 3.0.

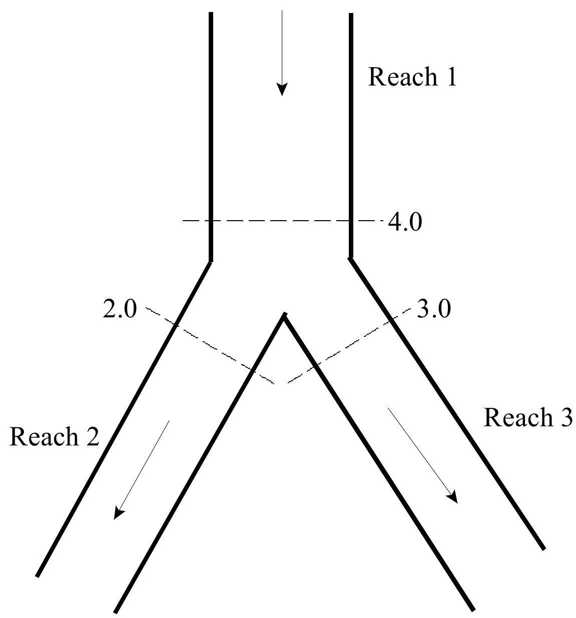


Figure 4.6 Example Supercritical Flow Split

Case 5: *Mixed Flow Regime - Flow Combining*

In the case of mixed flow, a subcritical profile calculation is made through the junction as described previously (see Figure 4.7). If the flow remains subcritical during the supercritical flow calculations, then the subcritical answers are assumed to be correct. If, however, the flow at either or both of the cross sections upstream of the junction is found to have supercritical flow controlling, then the junction must be re-calculated. When one or more of the upstream sections is supercritical, the program will calculate the specific force of all the upstream sections. If the supercritical sections have a greater specific force than the subcritical sections, then the program assumes that supercritical flow will control. The program then makes a forewater calculation from the upstream section with the greatest specific force (let's say section 4.0) to the downstream section (section 3.0).

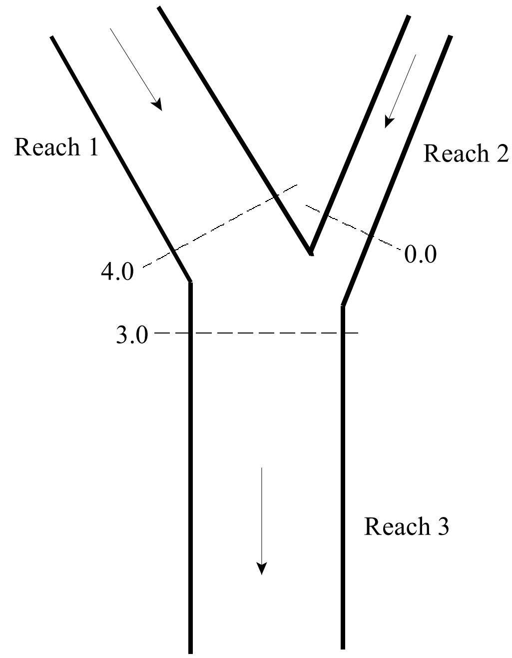


Figure 4.7 Example of Mixed Flow Regime at a Flow Combine

The program next computes the specific force of both the subcritical and supercritical answers at section 3.0. If the supercritical answer at section 3.0 has a lower specific force than the previously computed subcritical answer, then the program uses the subcritical answer and assumes that a hydraulic jump occurred at the junction. If the supercritical answer has a greater specific force, then the program continues downstream with forewater calculations until a hydraulic jump is encountered. Also, any upstream reach that is subcritical must be recomputed. For example, if reach two is subcritical, the water surface at section 0.0 was based on a backwater calculation from section 3.0 to 0.0. If section 3.0 is found to be supercritical, the water surface at section 0.0 is set to critical depth, and backwater calculations are performed again for reach 2. If there are any reaches above reach 2 that are affected by this change, then they are also recomputed.

Case 6: *Mixed Flow Regime - Split Flow*

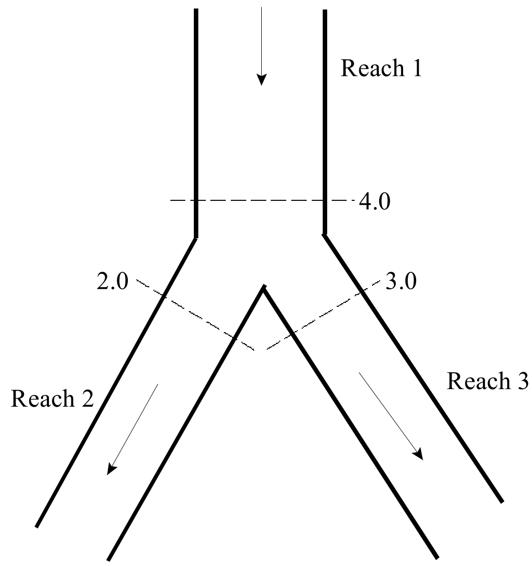


Figure 4.8 Example of Mixed Flow Regime at a Flow Split

In this case, a subcritical profile through the junction is computed as described previously. If during the supercritical flow pass it is found that section 4.0 (Figure 4.8) is actually supercritical, the program will perform forewater calculations across the junction. The program will make a forewater calculation from section 4.0 to 2.0 and then from 4.0 to 3.0. The program will then calculate the specific force of the subcritical and supercritical answers at sections 2.0 and 3.0. Which ever answer has the greater specific force is assumed to be correct for each location. Normal mixed flow regime calculations continue on downstream from the junction.

Momentum Based Junction Method

The user can choose a momentum-based method to solve the junction problem instead of the default energy based method. As described previously, there are six possible flow conditions at the junction. The momentum-based method uses the same logic as the energy based method for solving the junction problem. The only difference is that the momentum-based method solves for the water surfaces across the junction with the momentum equation.

Also, the momentum equation is formulated such that it can take into account the angles at which reaches are coming into or leaving the junction. To use the momentum based method, the user must supply the angle for any reach who's flow lines are not parallel to the main stem′s flow lines. An example of a flow combining junction is shown below in Figure 4.9. In this example, angles for both reaches 1 and 2 could be entered. Each angle is taken from a line that is perpendicular to cross-section 3.0 of reach 3.

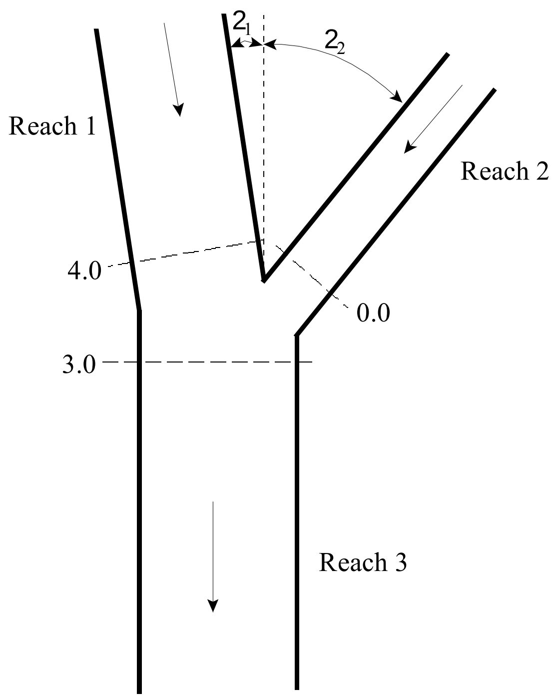


Figure 4.9 Example Geometry for Applying the Momentum Equation to a Flow Combining Junction

For subcritical flow, the water surface is computed up to section 3.0 of reach 3 by normal standard step backwater calculations. If the momentum equation is selected, the program solves for the water surfaces at sections 4.0 and 0.0 by performing a momentum balance across the junction. The momentum balance is written to only evaluate the forces in the X direction (the direction of flow based on cross section 3.0 of reach 3). For this example the equation is as follows:

$$SF_3 = SF_4 \cos\theta_1 - F_{f_{4-3}} + W_{x_{4-3}} + SF_0 \cos\theta_2 - F_{f_{0-3}} + W_{x_{0-3}} \tag{4-5}$$

Where: *SF* = Specific Force (as define in Equation 4.3)

The frictional and the weight forces are computed in two segments. For example, the friction and weight forces between sections 4.0 and 3.0 are based on the assumption that the centroid of the junction is half the distance between the two sections. The first portion of the forces are computed from section 4.0 to the centroid of the junction, utilizing the area at cross section 4.0. The second portion of the forces are computed from the centroid of the junction to section 3.0, using a flow weighted area at section 3.0. The equations to compute the friction and weight forces for this example are as follows:

<u>Forces due to friction</u>:

$$F_{f_{4-3}} = \overline{S}_{f_{4-3}} \frac{L_{4-3}}{2} A_4 \cos\theta_1 + \overline{S}_{f_{4-3}} \frac{L_{4-3}}{2} A_3 \frac{Q_4}{Q_3} \tag{4-6}$$

$$F_{f_{0-3}} = \overline{S}_{f_{0-3}} \frac{L_{0-3}}{2} A_0 \cos\theta_2 + \overline{S}_{f_{0-3}} \frac{L_{0-3}}{2} A_3 \frac{Q_0}{Q_3} \tag{4-7}$$

<u>Forces due to weight of water</u>:

$$W_{x_{4-3}} = S_{0_{4-3}} \frac{L_{4-3}}{2} A_4 \cos\theta_1 + S_{0_{4-3}} \frac{L_{4-3}}{2} A_3 \frac{Q_4}{Q_3} \tag{4-8}$$

$$W_{x_{0-3}} = S_{0_{0-3}} \frac{L_{0-3}}{2} A_0 \cos\theta_2 + S_{0_{0-3}} \frac{L_{0-3}}{2} A_3 \frac{Q_0}{Q_3} \tag{4-9}$$

To solve the momentum balance equation (Equation 4-5) for this example, the following assumptions are made:

1. The water surface elevations at section 4.0 and 0.0 are solved simultaneously, and are assumed to be equal to each other. This is a rough approximation, but it is necessary in order to solve Equation 4-5. Because of this assumption, the cross sections around the junction should be closely spaced in order to minimize the error associated with this assumption.

2. The area used at section 3.0 for friction and weight forces is distributed between the upper two reaches by using a flow weighting. This is necessary in order not to double account for the flow volume and frictional area.

When evaluating supercritical flow at this type of junction (Figure 4.9), the water surface elevations at sections 4.0 and 0.0 are computed from forewater calculations, and therefore the water surface elevations at section 3.0 can be solved directly from equation 4-5.

For mixed flow regime computations, the solution approach is the same as the energy based method, except the momentum equation is used to solve for the water surfaces across the junction.

An example of applying the momentum equation to a flow split is shown in Figure 4.10 below:

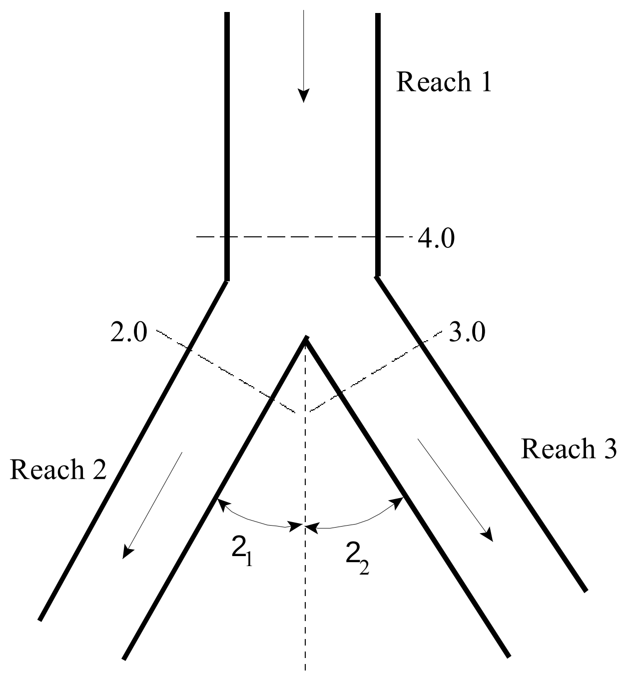


Figure 4.10 Example Geometry for Applying the Momentum Equation To a Flow Split Type of Junction

For the flow split shown in Figure 4.10, the momentum equation is written as follows:

$$SF_4 = SF_2 \cos\theta_1 + F_{f_{4-2}} - W_{x_{4-2}} + SF_3 \cos\theta_2 \; F_{f_{4-3}} - W_{x_{4-3}} \qquad (4\text{-}10)$$

For subcritical flow, the water surface elevation is known at sections 2.0 and 3.0, and the water surface elevation at section 4.0 can be found by solving Equation 4-10. For supercritical flow, the water surface is known at section 4.0 only, and, therefore, the water surface elevations at sections 3.0 and 2.0 must be solved simultaneously. In order to solve Equation 4-10 for supercritical flow, it is assumed that the water surface elevations at sections 2.0 and 3.0 are equal.

Mixed flow regime computations for a flow split are handled in the same manner as the energy based solution, except the momentum equation (Equation 4-10) is used to solve for the water surface elevations across the junction.

Flow Distribution Calculations

The general cross section output shows the distribution of flow in three subdivisions of the cross section: left overbank, main channel, and the right overbank. Additional output, showing the distribution of flow for multiple subdivisions of the left and right overbanks, as well as the main channel, can be requested by the user.

The flow distribution output can be obtained by first defining the locations that the user would like to have this type of output. 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 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 flow (discharge), area, wetted perimeter, percentage of conveyance, hydraulic depth, and average velocity for each of the user defined slices. For further details on how to request and view flow distribution output, see Chapters 7 and 8 of the HEC-RAS User's manual.

The computations for the flow distribution are performed after the program has calculated a water surface elevation and energy by the normal

methodology described in Chapter 2 of this manual. The flow distribution computations are performed as follows:

1. First, the water surface is computed in the normal manner of using the three flow subdivisions (left overbank, main channel, and right overbank), and balancing the energy equation.

2. Once a water surface elevation is computed, the program slices the cross section into the user defined flow distribution slices, and then computes an area, wetted perimeter, and hydraulic depth (area over top width) for each slice.

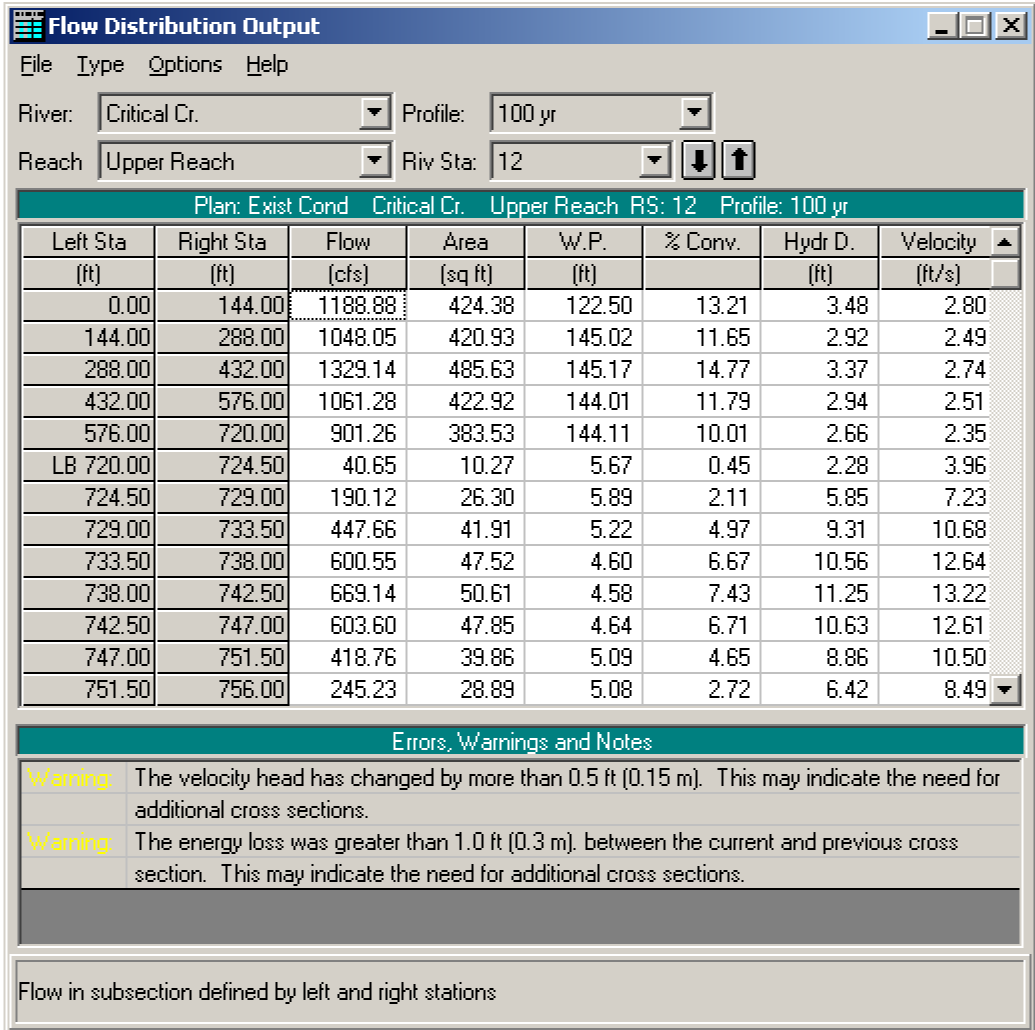
3. Using the originally computed energy slope (S_f), the cross section Manning's n values, the computed area and wetted perimeter for each slice, and Manning's equation, the program computes the conveyance and percentage of discharge for each of the slices.

4. The program sums up the computed conveyance for each of the slices. In general, the slice computed conveyance will not be the same as the originally computed conveyance (from the traditional methods for conveyance subdivision described in Chapter 2 of this manual). Normally, as a cross section is subdivided further and further, the computed conveyance, for a given water surface elevation, will increase.

5. In order to correct for the difference in computed conveyances, the program computes a ratio of the original total conveyance (from the normal calculations) divided by the total slice conveyance. This ratio is then applied to each of the slices, in order to achieve the same conveyance as was originally computed.

6. The final step is to compute an average velocity for each slice. The average velocity is computed by taking the discharge and dividing by the area for each of the user defined slices.

An example of the flow distribution output is shown in Figure 4.11.



Left Sta (ft)	Right Sta (ft)	Flow (cfs)	Area (sq ft)	W.P. (ft)	% Conv.	Hydr D. (ft)	Velocity (ft/s)
0.00	144.00	1188.88	424.38	122.50	13.21	3.48	2.80
144.00	288.00	1048.05	420.93	145.02	11.65	2.92	2.49
288.00	432.00	1329.14	485.63	145.17	14.77	3.37	2.74
432.00	576.00	1061.28	422.92	144.01	11.79	2.94	2.51
576.00	720.00	901.26	383.53	144.11	10.01	2.66	2.35
LB 720.00	724.50	40.65	10.27	5.67	0.45	2.28	3.96
724.50	729.00	190.12	26.30	5.89	2.11	5.85	7.23
729.00	733.50	447.66	41.91	5.22	4.97	9.31	10.68
733.50	738.00	600.55	47.52	4.60	6.67	10.56	12.64
738.00	742.50	669.14	50.61	4.58	7.43	11.25	13.22
742.50	747.00	603.60	47.85	4.64	6.71	10.63	12.61
747.00	751.50	418.76	39.86	5.09	4.65	8.86	10.50
751.50	756.00	245.23	28.89	5.08	2.72	6.42	8.49

Figure 4.11 Example Output for the Flow Distribution Option.

In general, the results of the flow distribution computations should be used cautiously. Specifically, the velocities and percentages of discharge are based on the results of 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 distribution in the field. While the results for the flow distribution, provided by HEC-RAS, 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.

Split Flow Optimization

This feature is for Steady Flow Analyses only. The HEC-RAS software has the capability to optimize flow splits at lateral weirs/spillways, hydraulic connections, storage areas, and stream junctions. This feature is available by selecting "Split Flow Optimizations" from the "Options" menu of the Steady Flow Analysis" window. When this option is selected, a window will appear as shown below.

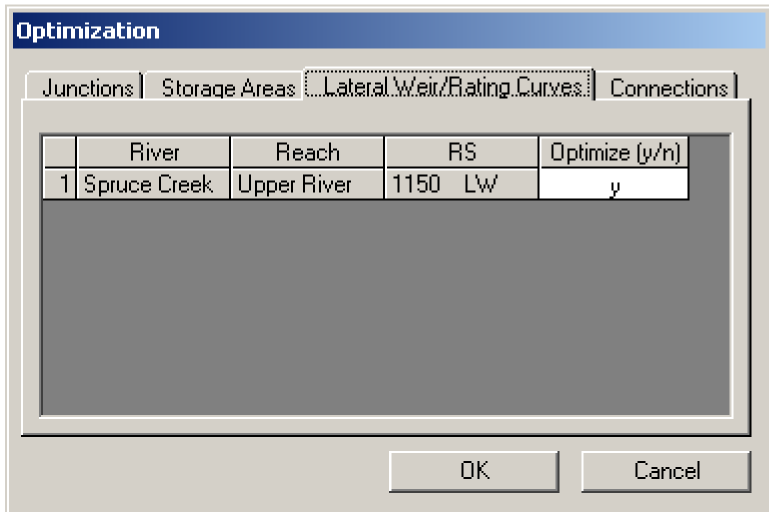


Figure 4.12 Split Flow Optimization Window

When the split flow optimization is turned on, the program will calculate a water surface profile with the first assumed flows. From the computed profile, new flows are calculated for the hydraulic structures and junctions and the profile is re-run. This process continues until the calculated and assumed flows match within a given tolerance. For more information on split flow optimization, please review Example 15 of the Applications Guide.