CHAPTER 5

Modeling Bridges

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 generally 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 generally contracting to get through the opening. This chapter discusses how bridges are modeled using HEC-RAS. Discussions include: general modeling guidelines; hydraulic computations through the bridge; selecting a bridge modeling approach; and unique bridge problems and suggested approaches.

Contents

General Modeling Guidelines

Considerations for modeling the geometry of a reach of river in the vicinity of a bridge are essentially the same for any of the available bridge modeling approaches within HEC-RAS. Modeling guidelines are provided in this section for locating cross sections; defining ineffective flow areas; and evaluating contraction and expansion losses around bridges.

Cross Section Locations

The bridge routines utilize four user-defined cross sections in the computations of energy losses due to the structure. During the hydraulic computations, the program automatically formulates two additional cross sections inside of the bridge structure. A plan view of the basic cross section layout is shown in Figure 5.1. The cross sections in Figure 5.1 are labeled as river stations 1, 2, 3, and 4 for the purpose of discussion within this chapter. Whenever the user is performing water surface profile computations through a bridge (or any other hydraulic structure), additional cross sections should always be included both downstream and upstream of the bridge. This will prevent any user-entered boundary conditions from affecting the hydraulic results through the bridge.

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 (the expansion reach length, L_e) should generally be determined by field investigation during high flows. 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.

Table 5.1 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 on Figure 5.1) is found by multiplying the expansion ratio by the average obstruction length (the average of the distances A to B and C to D from Figure 5.1). The average obstruction length is half of the total reduction in floodplain width caused by the two bridge approach embankments. In Table 5.1, b/B is the ratio of the bridge opening width to the total floodplain width, n_{ob} is the Manning n value for the overbank, n_c is the n value for the main channel, and S is the longitudinal slope. 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.

Table 5.1
Ranges of Expansion Ratios

		$n_{ob} / n_{c = 1}$	$n_{ob} / n_{c = 2}$	$n_{ob} / n_{c = 4}$
b/B = 0.10	S = 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	S = 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	S = 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

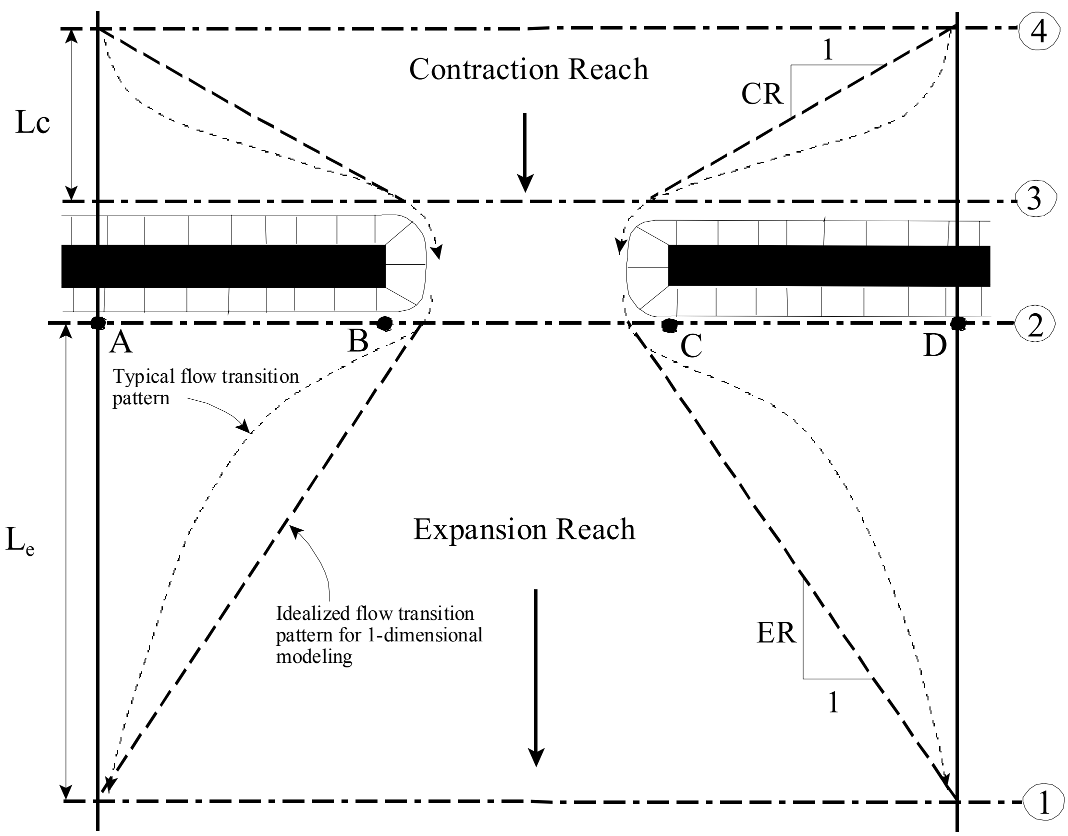


Figure 5.1 Cross Section Locations at a Bridge

A detailed study of flow contraction and expansion zones has been completed by the Hydrologic Engineering Center 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 this document.

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 thinks 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 ineffective flow option can be used to limit the effective flow area of the intermediate cross sections in the expansion reach.

Cross section 2 is located a short distance downstream from the bridge (i.e., commonly placed at the downstream toe of the road embankment). This cross section should represent the area just outside the bridge.
Cross section 3 should be located a short distance upstream from the bridge (commonly placed at the upstream toe of the road embankment). The distance between cross section 3 and the bridge 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 effective flow area just upstream of the bridge. Both cross sections 2 and 3 will have ineffective flow areas to either side of the bridge opening during low flow and pressure flow profiles. In order to model only the effective flow areas at these two sections, the modeler should use the ineffective flow area option at both of these cross sections.

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 used a criterion to locate the upstream cross section one times the average length of the side constriction caused by the structure abutments (the average of the distance from A to B and C to D on Figure 5.1). 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 this document.

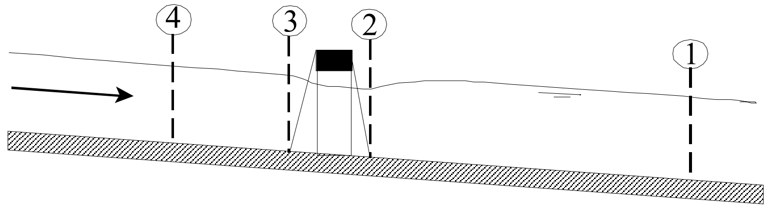
During the hydraulic computations, the program automatically formulates two additional cross sections inside of the bridge structure. The geometry inside of the bridge is a combination of the bounding cross sections (sections 2 and 3) and the bridge geometry. The bridge geometry consists of the bridge deck and roadway, sloping 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 of the bridge 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.

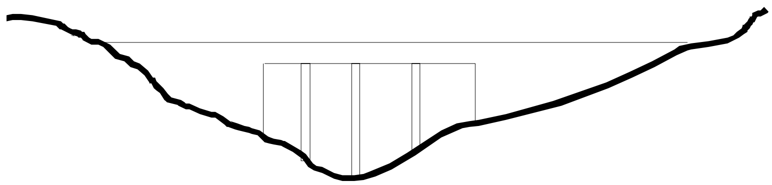
Defining Ineffective Flow Areas

A basic problem in defining the bridge data is the definition of ineffective flow areas near the bridge structure. Referring to Figure 5-1, the dashed lines represent the effective flow boundary for low flow and pressure flow conditions. Therefore, for cross sections 2 and 3, ineffective flow areas to either side of the bridge opening (along distance AB and CD) should not be included as part of the active flow area for low flow or pressure flow.

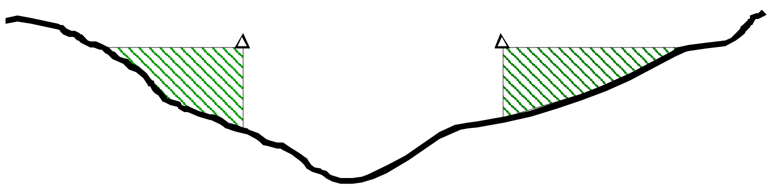
The bridge example shown in Figure 5.2 is a typical situation where the bridge spans the entire floodway and its abutments obstruct the natural floodplain. This is a similar situation as was shown in plan view in Figure 5.1. The cross section numbers and locations are the same as those discussed in the “Cross Section Locations” section of this chapter. The problem is to convert the natural ground profile at cross sections 2 and 3 from the cross section shown in part B to that shown in part C of Figure 5.2. The elimination of the ineffective overbank areas can be accomplished by redefining the geometry at cross sections 2 and 3 or by using the natural ground profile and requesting the program's ineffective area option to eliminate the use of the overbank area (as shown in part C of Figure 5.2). Also, for high flows (flows over topping the bridge deck), the area outside of the main bridge opening may no longer be ineffective, and will need to be included as active flow area. If the modeler chooses to redefine the cross section, a fixed boundary is used at the sides of the cross section to contain the flow, when in fact a solid boundary is not physically there. The use of the ineffective area option is more appropriate and it does not add wetted perimeter to the active flow boundary above the given ground profile.



A. Channel Profile and cross section locations



B. Bridge cross section on natural ground



C. Portion of cross sections 2 & 3 that is ineffective for low flow

Figure 5.2 Cross Sections Near Bridges

The ineffective area option is used at sections 2 and 3 to keep all the active flow in the area of the bridge opening until the elevations associated with the left and/or right ineffective flow areas are exceeded by the computed water surface elevation. The program allows the stations and controlling elevations of the left and right ineffective flow areas to be specified by the user. Also, the stations of the ineffective flow areas do not have to coincide with stations of the ground profile, the program will interpolate the ground station.

The ineffective flow areas should be set at stations that will adequately describe the active flow area at cross sections 2 and 3. In general, these stations should be placed outside the edges of the bridge opening to allow for the contraction and expansion of flow that occurs in the immediate vicinity of the bridge. On the upstream side of the bridge (section 3) the flow is contracting rapidly. A practical method for placing the stations of the ineffective flow areas is to assume a 1:1 contraction rate in the immediate vicinity of the bridge. In other words, if cross section 3 is 10 feet from the upstream bridge face, the ineffective flow areas should be placed 10 feet away from each side of the bridge opening. On the downstream side of the bridge (section 2), a similar assumption can be applied. The active flow area on the downstream side of the bridge may be less than, equal to, or greater than the width of the bridge opening. As flow converges into the bridge opening, depending on the abruptness of the abutments, the active flow area may constrict to be less than the bridge opening. As the flow passes through and out of the bridge it begins to expand. Because of this phenomenon, estimating the stationing of the ineffective flow areas at cross section 2 can be very difficult. In general, the user should make the active flow area equal to the width of the bridge opening or wider (to account for flow expanding), unless the bridge abutments are very abrupt (vertical wall abutments with no wing walls).

The elevations specified for ineffective flow should correspond to elevations where significant weir flow passes over the bridge. For the downstream cross section, the threshold water surface elevation for weir flow is not usually known on the initial run, so an estimate must be made. An elevation below the minimum top-of-road, such as an average between the low chord and minimum top-of-road, can be used as a first estimate.

Using the ineffective area option to define the ineffective flow areas allows the overbank areas to become effective as soon as the ineffective area elevations are exceeded. The assumption is that under weir flow conditions, the water can generally flow across the whole bridge length and the entire overbank in the vicinity of the bridge would be effectively carrying flow up to and over the bridge. If it is more reasonable to assume only part of the overbank is effective for carrying flow when the bridge is under weir flow, then the overbank n values can be increased to reduce the amount of conveyance in the overbank areas under weir flow conditions.

Cross section 3, just upstream from the bridge, is usually defined in the same manner as cross section 2. In many cases the cross sections are identical. The only difference generally is the stations and elevations to use for the ineffective area option. For the upstream cross section, the elevation should initially be set to the low point of the top-of-road. When this is done the user could possibly get a solution where the bridge hydraulics are computing weir flow, but the upstream water surface elevation comes out lower than the top of road. Both the weir flow and pressure flow equations are based on the energy grade line in the upstream cross section. Once an upstream energy is computed from the bridge hydraulics, the program tries to compute a water surface elevation in the upstream cross section that corresponds to that energy. Occasionally the program may get a water surface that is confined by the ineffective flow areas and lower than the minimum top of road. When this happens, the user should decrease the elevations of the upstream ineffective flow areas in order to get them to turn off. Once they turn off, the computed water surface elevation will be much closer to the computed energy gradeline (which is higher than the minimum high chord elevation).

Using the ineffective area option in the manner just described for the two cross sections on either side of the bridge provides for a constricted section when all of the flow is going under the bridge. When the water surface is higher than the control elevations used, the entire cross section is used. The program user should check the computed solutions on either side of the bridge section to ensure they are consistent with the type of flow. That is, for low flow or pressure flow solutions, the output should show the effective area restricted to the bridge opening. When the bridge output indicates weir flow, the solution should show that the entire cross section is effective. During overflow situations, the modeler should ensure that the overbank flow around the bridge is consistent with the weir flow.

Contraction and Expansion Losses

Losses due to contraction and expansion of flow between cross sections are determined during the standard step profile calculations. Manning's equation is used to calculate friction losses, and all other losses are described in terms of a 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, an expansion coefficient is used.

As shown in Figure 5.1, the flow contraction occurs between cross sections 4 and 3, while the flow expansion occurs between sections 2 and 1. The contraction and expansion coefficients are used to compute energy losses associated with changes in the shape of river cross-sections (or effective flow areas). The loss due to expansion of flow is usually larger than the contraction loss, and losses from short abrupt transitions are larger than losses from gradual transitions. Typical values for contraction and expansion coefficients under subcritical flow conditions are shown in Table 5.2 below:

Table 5.2
Subcritical Flow Contraction and Expansion Coefficients

	Contraction	**Expansion**
No transition loss computed	0.0	0.0
Gradual transitions	0.1	0.3
Typical Bridge sections	0.3	0.5
Abrupt transitions	0.6	0.8

The maximum value for the contraction and expansion coefficient is 1.0. As mentioned previously, a detailed study was completed by the Hydrologic Engineering Center entitled “Flow Transitions in Bridge Backwater Analysis” (HEC, 1995). A summary of this research, as well as recommendations for contraction and expansion coefficients, can be found in Appendix B.

In general, contraction and expansion coefficients for supercritical flow should be lower than subcritical flow. For typical bridges that are under class C flow conditions (totally supercritical flow), the contraction and expansion coefficients should be around 0.1 and 0.3 respectively. For abrupt bridge transitions under class C flow, values of 0.3 and 0.5 may be more appropriate.

Hydraulic Computations Through the Bridge

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 on the weir), pressure flow (orifice and sluice gate equations), pressure and weir flow, and highly submerged flows (the program will automatically switch to the energy equation when the flow over the road is highly submerged). This portion of the manual describes in detail how the program models each of these different flow types.

Low Flow Computations

Low flow exists when the flow going through the bridge opening is open channel flow (water surface below the highest point on the low chord of the bridge opening). 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. If the two sections are identical, the program selects the upstream bridge section as the controlling section. 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 controlling bridge section, 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 considered class C low flow.

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 weighted friction slope is based on one of the four available alternatives in the 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. 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 in the same way as friction and expansion losses between sections 1 and 2.

There are four methods available for computing losses through the bridge (sections 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 to be computed. This allows the modeler to compare the answers from several techniques all in a single execution of the program. If more than one method is selected, the user must choose either a single method as the final solution or direct the program to use the method that computes the greatest energy loss through the bridge as the final solution at section 3. Minimal results are available for all the methods computed, but detailed results are available for the method that is selected as the final answer. A detailed discussion of each method follows:

<u>Energy Equation (standard step method)</u>:

The energy-based method treats a bridge in the same manner as a natural river cross-section, except the area of the bridge below the water surface is subtracted from the total area, and the wetted perimeter is increased where the water is in contact with the bridge structure. As described previously, the program formulates two cross sections inside the bridge by combining the ground information of sections 2 and 3 with the bridge geometry. As shown in Figure 5.3, for the purposes of discussion, these cross sections will be referred to as sections BD (Bridge Downstream) and BU (Bridge Upstream).

The sequence of calculations starts with a standard step calculation from just downstream of the bridge (section 2) to just inside of the bridge (section BD) at the downstream end. The program then performs a standard step through the bridge (from section BD to section BU). The last calculation is to step out of the bridge (from section BU to section 3).

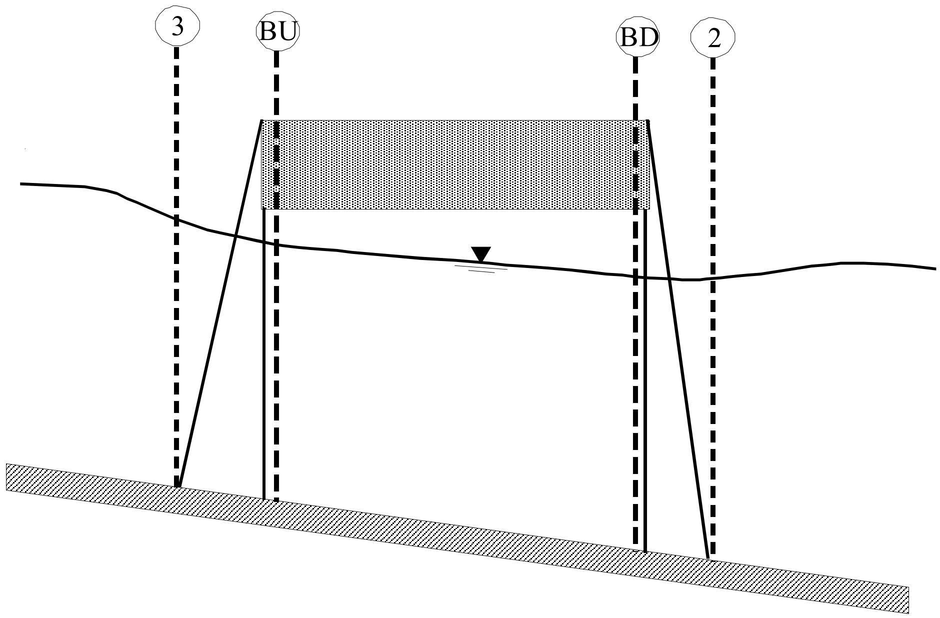


Figure 5.3 Cross Sections Near and Inside the Bridge

The energy-based method requires Manning's n values for friction losses and contraction and expansion coefficients for transition losses. The estimate of Manning's n values is well documented in many hydraulics text books, as well as several research studies. Basic guidance for estimating roughness coefficients is provided in Chapter 3 of this manual. Contraction and expansion coefficients are also provided in Chapter 3, as well as in earlier sections of this chapter. Detailed output is available for cross sections inside the bridge (sections BD and BU) as well as the user entered cross sections (sections 2 and 3).

<u>Momentum Balance Method</u>:

The momentum method is based on performing a momentum balance from cross section 2 to cross-section 3. The momentum balance is performed in three steps. The first step is to perform a momentum balance from cross section 2 to cross-section BD inside the bridge. The equation for this momentum balance is as follows:

$$A_{BD}\,\overline{Y}_{BD} + \frac{\beta_{BD}\,Q_{BD}^2}{g\,A_{BD}} = A_2\,\overline{Y}_2 + \frac{\beta_2\,Q_2^2}{g\,A_2} - A_{p_{BD}}\,\overline{Y}_{p_{BD}} + F_f - W_x \qquad (5\text{-}1)$$

where: A_2, A_{BD} = Active flow area at section 2 and BD, respectively

$A_{p_{BD}}$ = Obstructed area of the pier on downstream side

$\overline{Y}_2$, $\overline{Y}_{BD}$ = Vertical distance from water surface to center of gravity of flow area A_2 and A_{BD}, respectively

$\overline{Y}_{p_{BD}}$ = Vertical distance from water surface to center of gravity of wetted pier area on downstream side

β_2, β_{BD} = Velocity weighting coefficients for momentum equation

Q_2, Q_{BD} = Discharge

g = Gravitational acceleration

F_f = External force due to friction, per unit weight of water

W_x = Force due to weight of water in the direction of flow, per unit weight of water

The second step is a momentum balance from section BD to BU (see Figure 5.3). The equation for this step is as follows:

$$A_{BU}\,\overline{Y}_{BU} + \frac{\beta_{BU}\,Q_{BU}^2}{g\,A_{BU}} = A_{BD}\,\overline{Y}_{BD} + \frac{\beta_{BD}\,Q_{BD}^2}{g\,A_{BD}} + F_f - W_x \qquad (5\text{-}2)$$

The final step is a momentum balance from section BU to section 3 (see Figure 5.3). The equation for this step is as follows:

$$A_3\,\overline{Y}_3 + \frac{\beta_3\,Q_3^2}{g\,A_3} = A_{BU}\,\overline{Y}_{BU} + \frac{\beta_{BU}\,Q_{BU}^2}{g\,A_{BU}} + A_{p_{BU}}\,\overline{Y}_{p_{BU}} + \frac{1}{2}C_D\,\frac{A_{p_{BU}}\,Q_3^2}{g\,A_3^2} + F_f - W_x \qquad (5\text{-}3)$$

where: C_D = Drag coefficient for flow going around the piers. Guidance on selecting drag coefficients can be found under Table 5.3 below.

The momentum balance method requires the use of roughness coefficients for the estimation of the friction force and a drag coefficient for the force of drag on piers. As mentioned previously, roughness coefficients are described in Chapter 3 of this manual. Drag coefficients are used to estimate the force due to the water moving around the piers, the separation of the flow, and the resulting wake that occurs downstream. Drag coefficients for various cylindrical shapes have been derived from experimental data (Lindsey, 1938). The following table shows some typical drag coefficients that can be used for piers:

Table 5.3
Typical drag coefficients for various pier shapes

Pier Shape	**Drag Coefficient C_D**
Circular pier	1.20
Elongated piers with semi-circular ends	1.33
Elliptical piers with 2:1 length to width	0.60
Elliptical piers with 4:1 length to width	0.32
Elliptical piers with 8:1 length to width	0.29
Square nose piers	2.00
Triangular nose with 30 degree angle	1.00
Triangular nose with 60 degree angle	1.39
Triangular nose with 90 degree angle	1.60
Triangular nose with 120 degree angle	1.72

The momentum method provides detailed output for the cross sections inside the bridge (BU and BD) as well as outside the bridge (2 and 3). The user has the option of turning the friction and weight force components 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.

During the momentum calculations, if the water surface (at sections BD and BU) comes into contact with the maximum low chord of the bridge, the momentum balance is assumed to be invalid and the results are not used.

Yarnell Equation:

The Yarnell equation is an empirical equation that is used to predict the change in water surface from just downstream of the bridge (section 2 of Figure 5.3) to just upstream of the bridge (section 3). The equation is based on approximately 2600 lab experiments in which the researchers varied the shape of the piers, the width, the length, the angle, and the flow rate. The Yarnell equation is as follows (Yarnell, 1934):

$$H_{3-2} = 2K\left(K + 10\omega - 0.6\right)\left(\alpha + 15\alpha^4\right)\frac{V^2}{2g} \qquad (5\text{-}4)$$

Where: H_{3-2} = Drop in water surface elevation from section 3 to 2

K = Yarnell's pier shape coefficient

ω = Ratio of velocity head to depth at section 2

α = Obstructed area of the piers divided by the total unobstructed area at section 2

V_2 = Velocity downstream at section 2

The computed upstream water surface elevation (section 3) is simply the downstream water surface elevation plus H_{3-2}. With the upstream water surface known the program computes the corresponding velocity head and energy elevation for the upstream section (section 3). When the Yarnell method is used, hydraulic information is only provided at cross sections 2 and 3 (no information is provided for sections BU and BD).

The Yarnell equation is sensitive to the pier shape (K coefficient), the pier obstructed area, and the velocity of the water. The method is not sensitive to the shape of the bridge opening, the shape of the abutments, or the width of the bridge. Because of these limitations, the Yarnell method should only be used at bridges where the majority of the energy losses are associated with the piers. When Yarnell's equation is used for computing the change in water surface through the bridge, the user must supply the Yarnell pier shape coefficient, K. The following table gives values for Yarnell's pier coefficient, K, for various pier shapes:

Table 5.4
Yarnell's pier coefficient, K, for various pier shapes

Pier Shape	Yarnell K Coefficient
Semi-circular nose and tail	0.90
Twin-cylinder piers with connecting diaphragm	0.95
Twin-cylinder piers without diaphragm	1.05
90 degree triangular nose and tail	1.05
Square nose and tail	1.25
Ten pile trestle bent	2.50

FHWA WSPRO Method:

The low flow hydraulic computations of the Federal Highway Administration's (FHWA) WSPRO computer program, has been adapted as an option for low flow hydraulics in HEC-RAS. The WSPRO methodology had to be modified slightly in order to fit into the HEC-RAS concept of cross-section locations around and through a bridge.

The WSPRO method computes the water surface profile through a bridge by solving the energy equation. The method is an iterative solution performed from the exit cross section (1) to the approach cross-section (4). The energy balance is performed in steps from the exit section (1) to the cross section just downstream of the bridge (2); from just downstream of the bridge (2) to inside of the bridge at the downstream end (BD); from inside of the bridge at the downstream end (BD) to inside of the bridge at the upstream end (BU); From inside of the bridge at the upstream end (BU) to just upstream of the bridge (3); and from just upstream of the bridge (3) to the approach section (4). A general energy balance equation from the exit section to the approach section can be written as follows:

$$h_4 + \frac{\alpha_4 V_4^2}{2g} = h_1 + \frac{\alpha_1 V_1^2}{2g} h_{L_{4-1}} \tag{5-5}$$

where: h_1 = Water surface elevation at section 1

V_1 = Velocity at section 1

h_4 = Water surface elevation at section 4

V_4 = Velocity at section 4

h_L = Energy losses from section 4 to 1

The incremental energy losses from section 4 to 1 are calculated as follows:

From Section 1 to 2

Losses from section 1 to section 2 are based on friction losses and an expansion loss. Friction losses are calculated using the geometric mean friction slope times the flow weighted distance between sections 1 and 2. The following equation is used for friction losses from 1 to 2:

$$h_{f_{1-2}} = \frac{B\,Q^2}{K_2\,K_1} \tag{5-6}$$

Where B is the flow weighted distance between sections 1 and 2, and K_1 and K_2 are the total conveyance at sections 1 and 2 respectively. The expansion loss from section 2 to section 1 is computed by the following equation:

$$h_e = \frac{Q^2}{2\,g\,A_1^2}\left[2\beta_1 - \alpha_1 - 2\beta_2\left(\frac{A_1}{A_2}\right) + \alpha_2\left(\frac{A_1}{A_2}\right)^2\right] \tag{5-7}$$

Where α and β are energy and momentum correction factors for non-uniform flow. α_1 and β_1 are computed as follows:

$$\alpha_1 = \frac{\sum\left(K_i^3 / A_i^2\right)}{K_T^3 / A_T^2} \tag{5-8}$$

$$\beta_1 = \frac{\sum\left(K_i^2 / A_i\right)}{K_T^2 / A_T} \tag{5-9}$$

α_2 and β_2 are related to the bridge geometry and are defined as follows:

$$\alpha_1 = \frac{1}{C^2} \tag{5-10}$$

$$\beta_1 = \frac{1}{C} \tag{5-11}$$

where C is an empirical discharge coefficient for the bridge, which was originally developed as part of the Contracted Opening method by Kindswater, Carter, and Tracy (USGS, 1953), and subsequently modified by Matthai (USGS, 1968). The computation of the discharge coefficient, C, is explained in detail in appendix D of this manual.

From Section 2 to 3

Losses from section 2 to section 3 are based on friction losses only. The energy balance is performed in three steps: from section 2 to BD; BD to BU; and BU to 3. Friction losses are calculated using the geometric mean friction slope times the flow weighted distance between sections. The following equation is used for friction losses from BD to BU:

$$h_{f(BU\&BD)} = \frac{L_B Q^2}{K_{BU} K_{BD}} \tag{5-12}$$

Where K_{BU} and K_{BD} are the total conveyance at sections BU and BD respectively, and L_B is the length through the bridge. Similar equations are used for the friction losses from section 2 to BD and BU to 3.

From Section 3 to 4

Energy losses from section 3 to 4 are based on friction losses only. The equation for computing the friction loss is as follows:

$$h_{f(3\&4)} = \frac{L_{av} Q^2}{K_3 K_4} \tag{5-13}$$

Where L_{av} is the effective flow length in the approach reach, and K_3 and K_4 are the total conveyances at sections 3 and 4. The effective flow length is computed as the average length of 20 equal conveyance stream tubes (FHWA, 1986). The computation of the effective flow length by the stream tube method is explained in appendix D of this manual.

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 (section 3 of Figure 5.3) above critical depth and a downstream water surface (section 2) below critical depth. 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 to calculate an upstream water surface above critical depth and a downstream water surface below critical depth. If for some reason the momentum equation fails to converge on an answer during the class B flow computations, the program will automatically switch to an energy-based method for calculating the class B profile through the bridge.

Whenever class B flow is found to exist, the user should run the program in a mixed flow regime mode. If the user is running a mixed flow regime profile the program will proceed with backwater calculations upstream, and later with forewater calculations downstream from the bridge. Also, any hydraulic jumps that may occur upstream and downstream of the bridge can be located if they exist.

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 equation or the momentum equation to compute the water surface through the bridge for this class of flow.

High Flow Computations

The HEC-RAS program has the ability to compute high flows (flows that come into contact with the maximum low chord of the bridge deck) by either the Energy equation (standard step method) or by using separate hydraulic equations for pressure and/or weir flow. The two methodologies are explained below.

Energy Equation (standard step method). The energy-based method is applied to high flows in the same manner as it is applied to low flows. Computations are based on balancing the energy equation in three steps through the bridge. Energy losses are based on friction and contraction and expansion losses. Output from this method is available at the cross sections inside the bridge as well as outside.

As mentioned previously, friction losses are based on the use of Manning's equation. Guidance for selecting Manning’s n values is provided in Chapter 3 of this manual. Contraction and expansion losses are based on a coefficient times the change in velocity head. Guidance on the selection of contraction and expansion coefficients has also been provided in Chapter 3, as well as previous sections of this chapter.

The energy-based method performs all computations as though they are open channel flow. At the cross sections inside the bridge, the area obstructed by the bridge piers, abutments, and deck is subtracted from the flow area and additional wetted perimeter is added. Occasionally the resulting water surfaces inside the bridge (at sections BU and BD) can be computed at elevations that would be inside of the bridge deck. The water surfaces inside of the bridge reflect the hydraulic grade line elevations, not necessarily the actual water surface elevations. Additionally, the active flow area is limited to the open bridge area.

Pressure and Weir Flow Method. A second approach for the computation of high flows is to utilize separate hydraulic equations to compute the flow as pressure and/or weir flow. The two types of flow are presented below.

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 opening is flowing completely full. The HEC-RAS program will automatically select the appropriate equation, depending upon the flow situation. For the first case (see Figure 5.4), a sluice gate type of equation is used (FHWA, 1978):

$$Q = C_d \; A_{BU} \left[Y_3 - \frac{Z}{2} + \frac{\alpha_3 V_3^2}{2g} \right]^{1/2} \tag{5-14}$$

Where: Q = Total discharge through the bridge opening

C_d = Coefficient of discharge for pressure flow

A_{BU} = Net area of the bridge opening at section BU

Y_3 = Hydraulic depth at section 3

Z = Vertical distance from maximum bridge low chord to the mean river bed elevation at section BU

The discharge coefficient C_d, can vary depending upon the depth of water upstream. Values for C_d range from 0.27 to 0.5, with a typical value of 0.5 commonly used in practice. The user can enter a fixed value for this coefficient or the program will compute one based on the amount that the inlet is submerged. A diagram relating C_d to Y_3/Z is shown in Figure 5.5.

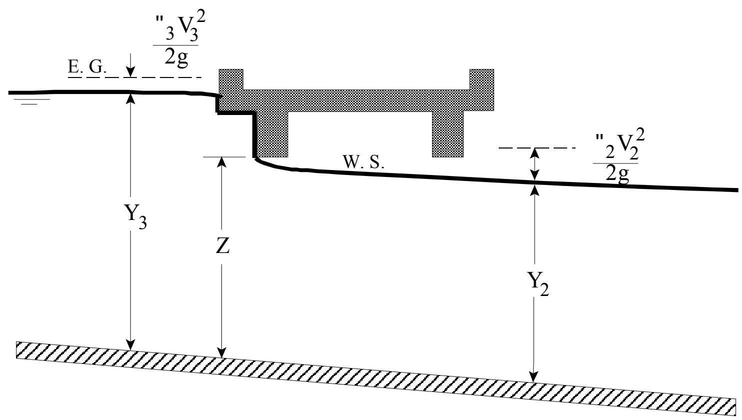


Figure 5.4 Example of a bridge under sluice gate type of pressure flow

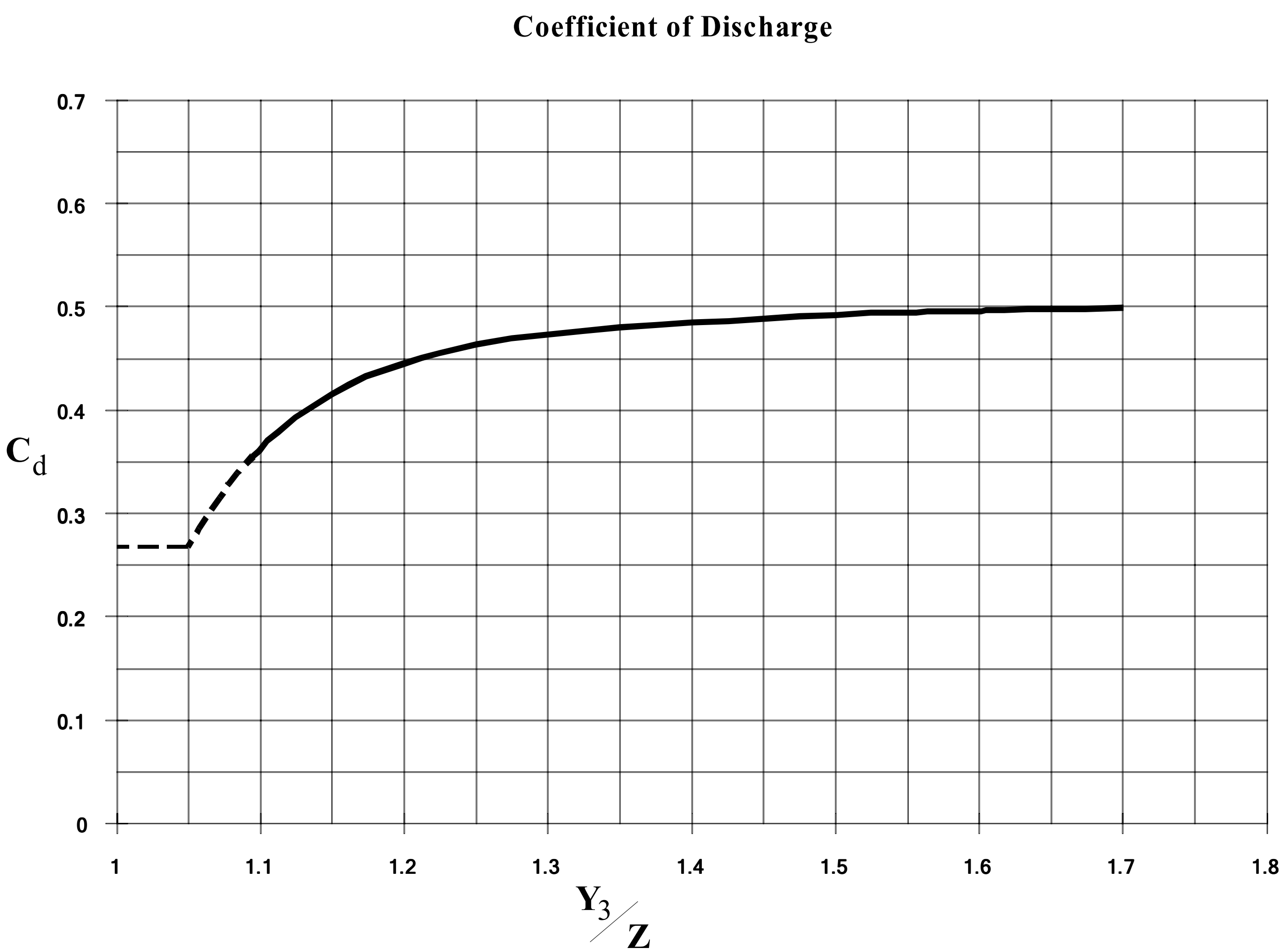


Figure 5.5 Coefficient of discharge for sluice gate type flow

As shown in Figure 5.5, the limiting value of Y_3/Z is 1.1. There is a transition zone somewhere between $Y_3/Z = 1.0$ and 1.1 where free surface flow changes to orifice flow. The type of flow in this range is unpredictable, and equation 5-14 is not applicable.

In the second case, when both the upstream and downstream side of the bridge are submerged, the standard full flowing orifice equation is used (see Figure 5.6). This equation is as follows:

$$Q = C\,A\sqrt{2gH} \tag{5-15}$$

Where: C = Coefficient of discharge for fully submerged pressure flow. Typical value of C is 0.8.

H = The difference between the energy gradient elevation upstream and the water surface elevation downstream.

A = Net area of the bridge opening.

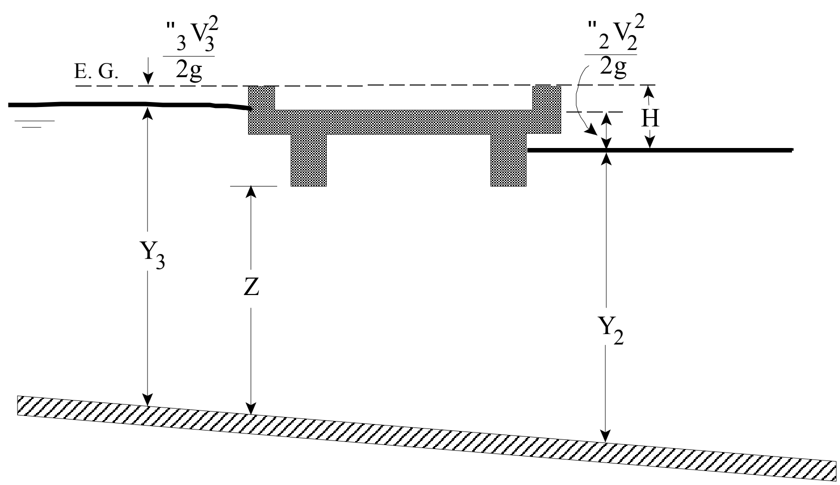


Figure 5.6 Example of a bridge under fully submerged pressure flow

Typical values for the discharge coefficient C range from 0.7 to 0.9, with a value of 0.8 commonly used for most bridges. The user must enter a value for C whenever the pressure flow method is selected. The discharge coefficient C can be related to the total loss coefficient, which comes from the form of the orifice equation that is used in the HEC-2 computer program (HEC, 1991):

$$C = A\sqrt{\frac{2gH}{K}} \tag{5-16}$$

Where: K = Total loss coefficient

The conversion from K to C is as follows:

$$C = \sqrt{\frac{1}{K}} \tag{5-17}$$

The program will begin checking for the possibility of pressure flow when the computed low flow energy grade line is above the maximum low chord elevation at the upstream side of the bridge. Once pressure flow is computed, the pressure flow answer is compared to the low flow answer, 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, is calculated using the standard weir equation (see Figure 5.7):

$$Q = CLH^{3/2} \tag{5-18}$$

where: Q = Total flow over the weir

C = Coefficient of discharge for weir flow

L = Effective length of the weir

H = Difference between energy upstream and road crest

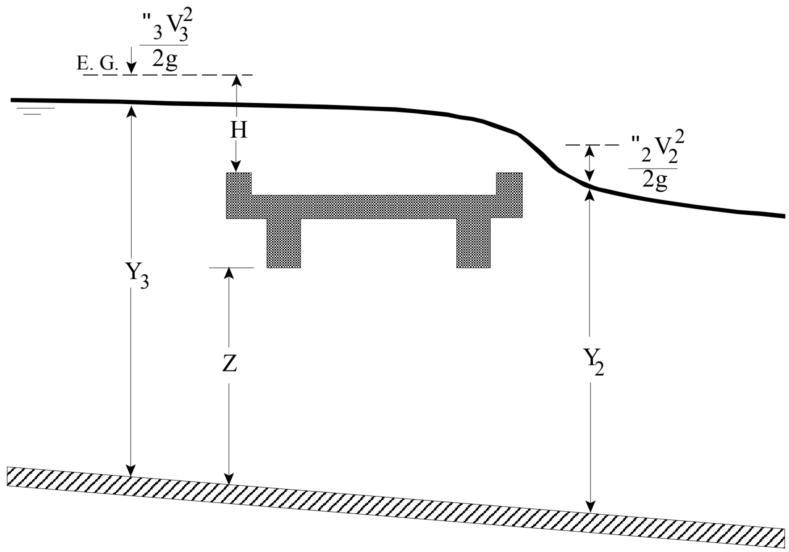


Figure 5.7 Example bridge with pressure and weir flow

The approach velocity is included by using the energy grade line elevation in lieu of the upstream water surface elevation for computing the head, H.

Under free flow conditions (discharge independent of tailwater) the coefficient of discharge C, ranges from 2.5 to 3.1 (1.38 - 1.71 metric) for broad-crested weirs depending primarily upon the gross head on the crest (C increases with head). Increased resistance to flow caused by obstructions such as trash on bridge railings, curbs, and other barriers would decrease the value of C.

Tables of weir coefficients, C, are given for broad-crested weirs in King's Handbook (King, 1963), with the value of C varying with measured head H and breadth of weir. For rectangular weirs with a breadth of 15 feet and a H of 1 foot or more, the given value is 2.63 (1.45 for metric). Trapezoidal shaped weirs generally have a larger coefficient with typical values ranging from 2.7 to 3.08 (1.49 to 1.70 for metric).

“Hydraulics of Bridge Waterways” (FHWA, 1978) provides a curve of C versus the head on the roadway. The roadway section is shown as a trapezoid and the coefficient rapidly changes from 2.9 for a very small H to 3.03 for H = 0.6 feet. From there, the curve levels off near a value of 3.05 (1.69 for metric).

With very little prototype data available, it seems the assumption of a rectangular weir for flow over the bridge deck (assuming the bridge can withstand the forces) and a coefficient of 2.6 (1.44 for metric) would be reasonable. If the weir flow is over the roadway approaches to the bridge, a value of 3.0 (1.66 for metric) would be consistent with available data. If weir flow occurs as a combination of bridge and roadway overflow, then an average coefficient (weighted by weir length) could be used.

For high tailwater elevations, the program will automatically reduce the amount of weir flow to account for submergence on the weir. Submergence is defined as the depth of water above the minimum weir elevation on the downstream side (section 2) divided by the height of the energy gradeline above the minimum weir elevation on the upstream side (section 3). The reduction of weir flow is accomplished by reducing the weir coefficient based on the amount of submergence. Submergence corrections are based on a trapezoidal weir shape or optionally an ogee spillway shape. The total weir flow is computed by subdividing the weir crest into segments, computing L, H, a submergence correction, and a Q for each section, then summing the incremental discharges. The submergence correction for a trapezoidal weir shape is from "Hydraulics of Bridge Waterways" (Bradley, 1978). Figure 5.8 shows the relationship between the percentage of submergence and the flow reduction factor.

When the weir becomes highly submerged the program will automatically switch to calculating the upstream water surface by the energy equation (standard step backwater) instead of using the pressure and weir flow equations. The criteria for when the program switches to energy based calculations is user controllable. A default maximum submergence is set to 0.95 (95 percent).

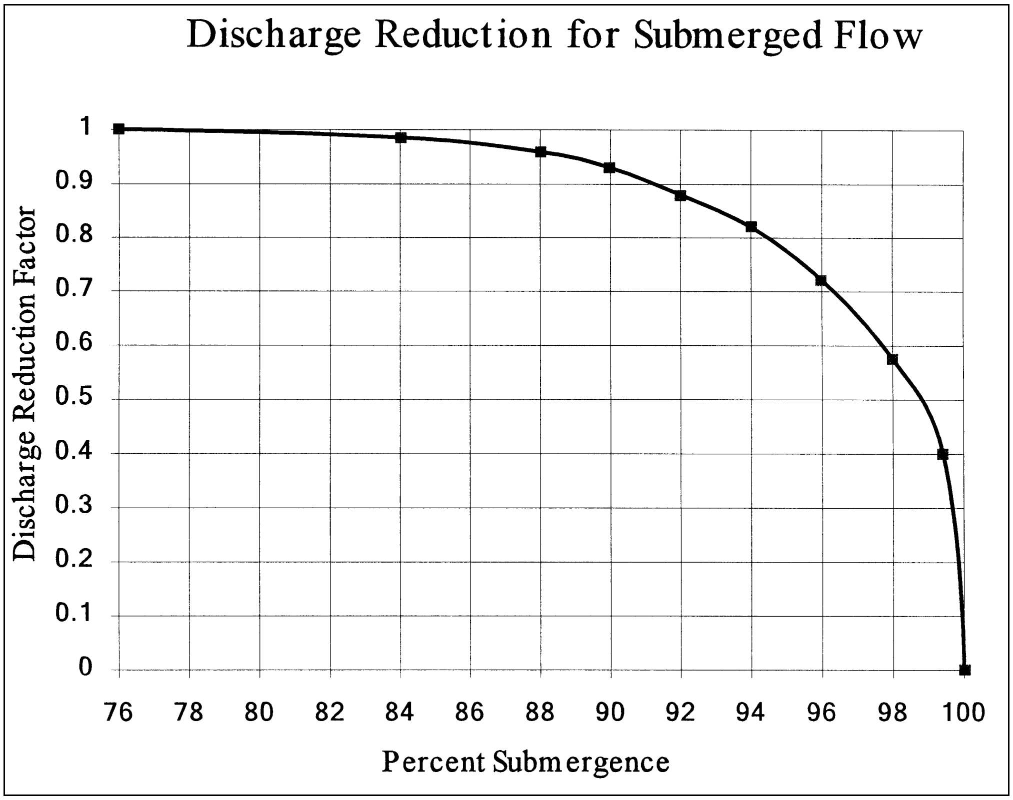


Figure 5.8 Factor for reducing weir flow for submergence

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. The program continues to iterate until both the low flow method (or pressure flow) and the weir flow method have the same energy (within a specified tolerance) upstream of the bridge (section 3). The combination of low flow and weir flow can only be computed with the energy and Yarnell low flow method.

Selecting a Bridge Modeling Approach

There are several choices available to the user when selecting methods for computing the water surface profile through a bridge. For low flow (water surface is below the maximum low chord of the bridge deck), the user can select any or all of the four available methods. For high flows, the user must choose between either the energy based method or the pressure and weir flow approach. The choice of methods should be considered carefully. The following discussion provides some basic guidelines on selecting the appropriate methods for various situations.

Low Flow Methods

For low flow conditions (water surface below the highest point on the low chord of the bridge opening), the Energy and Momentum methods are the most physically based, and in general are applicable to the widest range of bridges and flow situations. Both methods account for friction losses and changes in geometry through the bridge. The energy method accounts for additional losses due to flow transitions and turbulence through the use of contraction and expansion losses. The momentum method can account for additional losses due to pier drag. The FHWA WSPRO method was originally developed for bridge crossings that constrict wide flood plains with heavily vegetated overbank areas. The method is an energy-based solution with some empirical attributes (the expansion loss equation in the WSPRO method utilizes an empirical discharge coefficient). The Yarnell equation is an empirical formula. When applying the Yarnell equation, the user should ensure that the problem is within the range of data that the method was developed for. The following examples are some typical cases where the various low flow methods might be used:

1. In cases where the bridge piers are a small obstruction to the flow, and friction losses are the predominate consideration, the energy based method, the momentum method, and the WSPRO method should give the best answers.

2. In cases where pier losses and friction losses are both predominant, the momentum method should be the most applicable. But any of the methods can be used.

3. Whenever the flow passes through critical depth within the vicinity of the bridge, both the momentum and energy methods are capable of modeling this type of flow transition. The Yarnell and WSPRO methods are for subcritical flow only.

4. For supercritical flow, both the energy and the momentum method can be used. The momentum-based method may be better at locations that have a substantial amount of pier impact and drag losses. The Yarnell

equation and the WSPRO method are only applicable to subcritical flow situations.

5. For bridges in which the piers are the dominant contributor to energy losses and the change in water surface, either the momentum method or the Yarnell equation would be most applicable. However, the Yarnell equation is only applicable to Class A low flow.

6. For long culverts under low flow conditions, the energy based standard step method is the most suitable approach. Several sections can be taken through the culvert to model changes in grade or shape or to model a very long culvert. This approach also has the benefit of providing detailed answers at several locations within the culvert, which is not possible with the culvert routines in HEC-RAS. However, if the culvert flows full, or if it is controlled by inlet conditions, the culvert routines would be the best approach. For a detailed discussion of the culvert routines within HEC-RAS, see Chapter 6 of this manual.

High Flow Methods

For high flows (flows that come into contact with the maximum low chord of the bridge deck), the energy-based method is applicable to the widest range of problems. The following examples are some typical cases where the various high flow methods might be used.

1. When the bridge deck is a small obstruction to the flow, and the bridge opening is not acting like an pressurized orifice, the energy based method should be used.

2. When the bridge deck and road embankment are a large obstruction to the flow, and a backwater is created due to the constriction of the flow, the pressure and weir method should be used.

3. When the bridge and/or road embankment is overtopped, and the water going over top of the bridge is <u>not</u> highly submerged by the downstream tailwater, the pressure and weir method should be used. The pressure and weir method will automatically switch to the energy method if the bridge becomes 95 percent submerged. The user can change the percent submergence at which the program will switch from the pressure and weir method to the energy method. This is accomplished from the Deck/Roadway editor in the Bridge/Culvert Data editor.

4. When the bridge is highly submerged, and flow over the road is not acting like weir flow, the energy-based method should be used.

Unique Bridge Problems and Suggested Approaches

Many bridges are more complex than the simple examples presented in the previous sections. The following discussion is intended to show how HEC-RAS can be used to calculate profiles for more complex bridge crossings. The discussion here will be an extension of the previous discussions and will address only those aspects that have not been discussed previously.

Perched Bridges

A perched bridge is one for which the road approaching the bridge is at the floodplain ground level, and only in the immediate area of the bridge does the road rise above ground level to span the watercourse (Figure 5.9). A typical flood-flow situation with this type of bridge is low flow under the bridge and overbank flow around the bridge. Because the road approaching the bridge is usually not much higher than the surrounding ground, the assumption of weir flow is often not justified. A solution based on the energy method (standard step calculations) would be better than a solution based on weir flow with correction for submergence. Therefore, this type of bridge should generally be modeled using the energy-based method, especially when a large percentage of the total discharge is in the overbank areas.

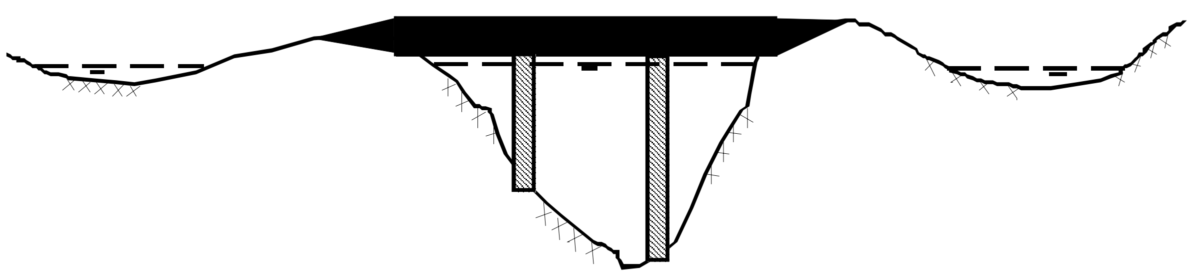


Figure 5.9 Perched Bridge Example

Low Water Bridges

A low water bridge (Figure 5.10) is designed to carry only low flows under the bridge. Flood flows are carried over the bridge and road. When modeling this bridge for flood flows, the anticipated solution is a combination of pressure and weir flow. However, with most of the flow over the top of the bridge, the correction for submergence may introduce considerable error. If the tailwater is going to be high, it may be better to use the energy-based method.

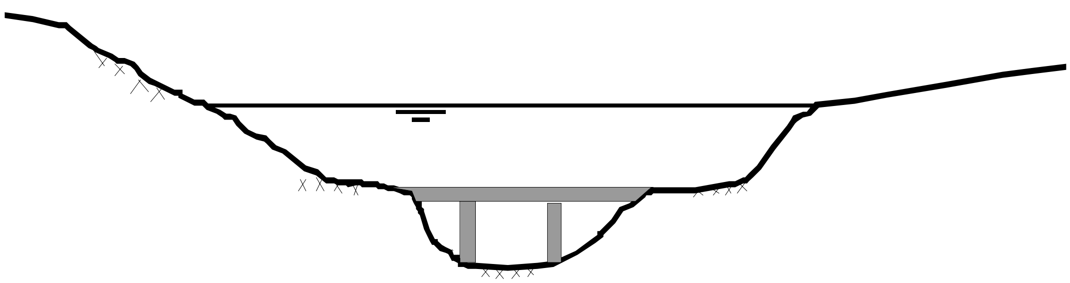


Figure 5.10 Low Water Bridge Example

Bridges on a Skew

Skewed bridge crossings (Figure 5.11) are generally handled by making adjustments to the bridge dimensions to define an equivalent cross section perpendicular to the flow lines. The bridge information, and cross sections that bound the bridge, can be adjusted from the bridge editor. An option called **Skew Bridge/Culvert** is available from the bridge/culvert editor.

In the publication "Hydraulics of Bridge Waterways" (Bradley, 1978) the effect of skew on low flow is discussed. In model testing, skewed crossings with angles up to 20 degrees showed no objectionable flow patterns. For increasing angles, flow efficiency decreased. A graph illustrating the impact of skew indicates that using the projected length is adequate for angles up to 30 degrees for small flow contractions.

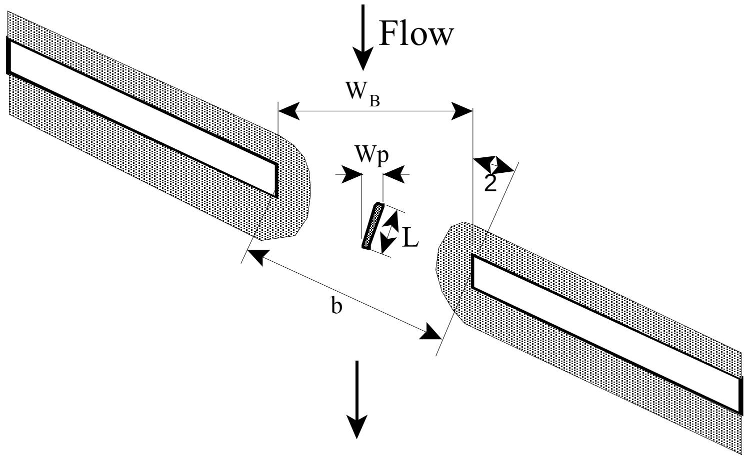


Figure 5.11 Example Bridge on a Skew

For the example shown in figure 5.11, the projected width of the bridge opening, perpendicular to the flow lines, will be computed with the following equation:

$$W_B = \cos\theta * b \tag{5-19}$$

Where: W_B = Projected width of the bridge opening, perpendicular to the flow lines

b = The length of the bridge opening as measured along the skewed road crossing

θ = The bridge skew angle in degrees

The pier information must also be adjusted to account for the skew of the bridge. HEC-RAS assumes the piers are continuous, as shown in Figure 5.11, thus the following equation will be applied to get the projected width of the piers, perpendicular to the flow lines:

$$W_p = \sin\theta * L + \cos\theta * w_p \tag{5-20}$$

Where: W_p = The projected width of the pier, perpendicular to the flow lines

L = The actual length of the pier

w_p = The actual width of the pier

Parallel Bridges

With the construction of divided highways, a common modeling problem involves parallel bridges (Figure 5.12). For new highways, these bridges are often identical structures. The hydraulic loss through the two structures has been shown to be between one and two times the loss for one bridge [Bradley, 1978]. The model results [Bradley, 1978] indicate the loss for two bridges ranging from 1.3 to 1.55 times the loss for one bridge crossing, over the range of bridge spacings tested. Presumably if the two bridges were far enough apart, the losses for the two bridges would equal twice the loss for one. If the parallel bridges are very close to each other, and the flow will not be able to expand between the bridges, the bridges can be modeled as a single bridge. If there is enough distance between the bridge, in which the flow has room to expand and contract, the bridges should be modeled as two separate bridges. If both bridges are modeled, care should be exercised in depicting the expansion and contraction of flow between the bridges. Expansion and contraction rates should be based on the same procedures as single bridges.

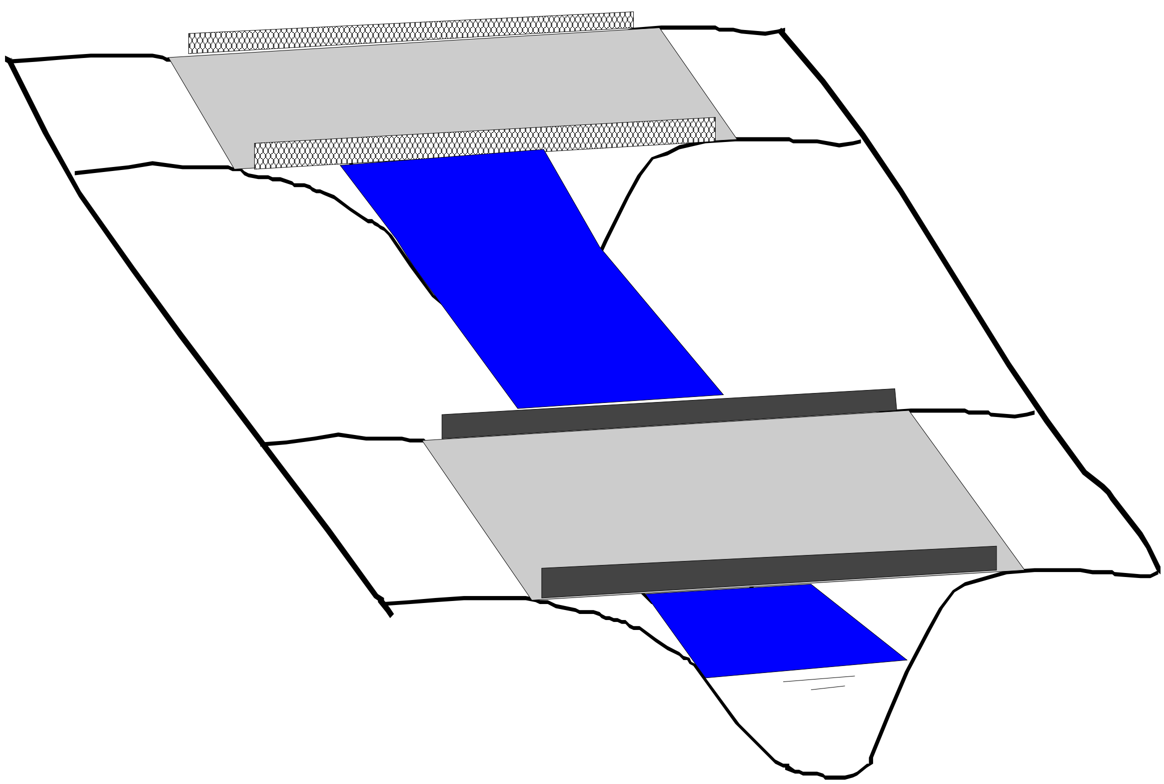


Figure 5.12 Parallel Bridge Example

Multiple Bridge Opening

Some bridges (Figure 5.13) have more than one opening for flood flow, especially over a very wide floodplain. Multiple culverts, bridges with side relief openings, and separate bridges over a divided channel are all examples of multiple opening problems. With more than one bridge opening, and possible different control elevations, the problem can be very complicated. HEC-RAS can handle multiple bridge and/or culvert openings. Detailed discussions on how to model multiple bridge and/or culvert openings is covered under Chapter 7 of the HEC-RAS Hydraulic Reference manual and Chapter 6 of the User's manual.

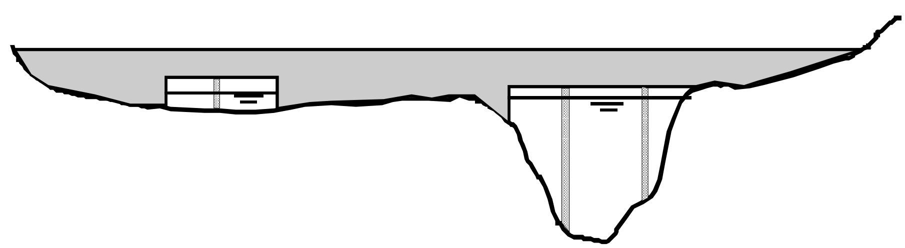


Figure 5.13 Example Multiple Bridge Opening

Modeling Floating Pier Debris

Trash, trees, and other debris may accumulate on the upstream side of a pier. During high flow events, this debris may block a significant portion of the bridge opening. In order to account for this effect, a pier debris option has been added to HEC-RAS.

The pier debris option blocks out a rectangular shaped area in front of the given pier. The user enters the height and the width of the given block. The program then adjusts the area and wetted perimeter of the bridge opening to account for the pier debris. The rectangular block is centered on the centerline of the upstream pier. The pier debris is assumed to float at the top of the water surface. That is, the top of the rectangular block is set at the same elevation as the water surface. For instance, assume a bridge opening that has a pier that is six feet wide with a centerline station of 100 feet, the elevation of water inside of the bridge is ten feet, and that the user wants to model pier debris that sticks out two feet past either side of the pier and is [vertically] four feet high. The user would enter a pier debris rectangle that is 10 feet wide (six feet for the pier plus two feet for the left side and two feet for the right side) and 4 feet high. The pier debris would block out the flow that is between stations 95 and 105 and between an elevation of six and ten feet (from an elevation of six feet to the top of the water surface).

The pier debris does not form until the given pier has flow. If the bottom of the pier is above the water surface, then there is no area or wetted perimeter adjustment for that pier. However, if the water surface is above the top of the pier, the debris is assumed to lodge underneath the bridge, where the top of the pier intersects the bottom of the bridge deck. It is assumed that the debris entirely blocks the flow and that the debris is physically part of the pier. (The Yarnell and momentum bridge methods require the area of the pier, and pier debris is included in these calculations.)

The program physically changes the geometry of the bridge in order to model the pier debris. This is done to ensure that there is no double accounting of area or wetted perimeter. For instance, pier debris that extends past the abutment, or into the ground, or that overlaps the pier debris of an adjacent pier is ignored.

Shown in Figure 5.14 is the pier editor with the pier debris option turned on. Note that there is a check box to turn the floating debris option on. Once this option is turned on, two additional fields will appear to enter the height and overall width of the pier debris. Additionally, there is a button that the user can use to set the entered height and width for the first pier as being the height and width of debris that will be used for all piers at this bridge location. Otherwise, the debris data can be defined separately for every pier.

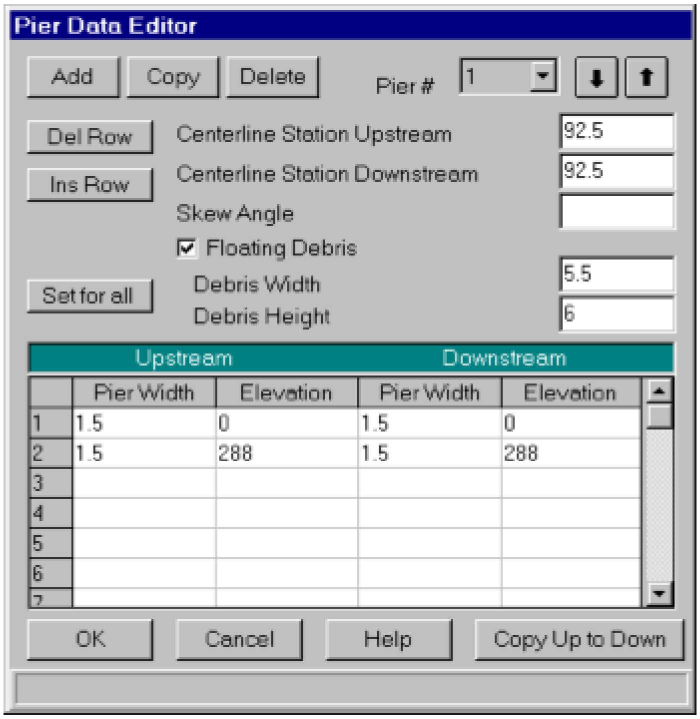


Figure 5.14 Pier Editor With Floating Debris Option

After the user has run the computational program with the pier debris option turned on, the pier debris will then be displayed on the cross section plots of the upstream side of the bridge (this is the cross sections with the labels "BR U," for inside of the bridge at the upstream end). An example cross-section plot with pier debris is shown in Figure 5.15.