Common Model Stability Problems When Performing a Dam Break Analysis

Alex Sánchez, PhD

US Army Corps of Engineers®

1

Objectives

- For students to have a better understanding of what causes an unsteady flow model to go unstable.
- To become familiar with the available parameters and techniques within HEC-RAS that will allow you to develop a stable and accurate model.

2

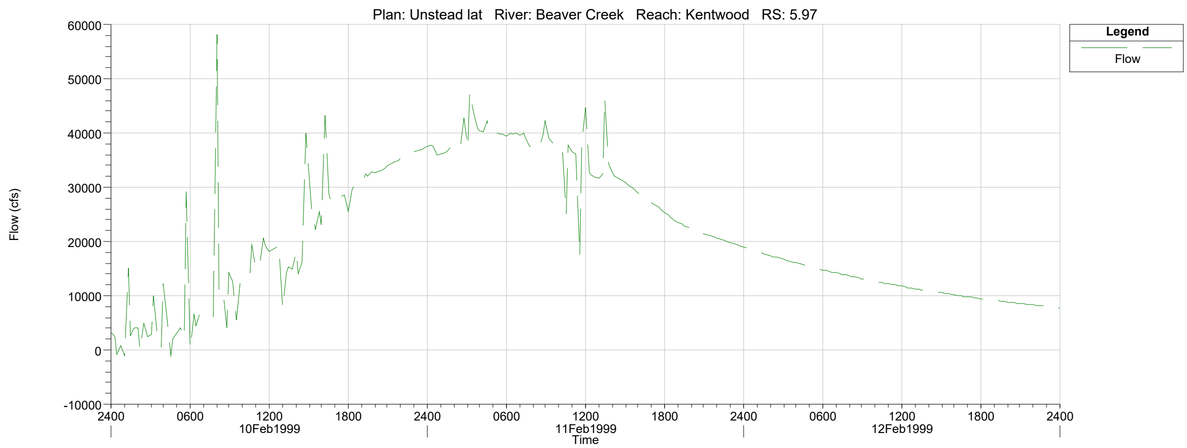
Overview

- Overview of Model Stability
- 1D Stability Issues
  - Cross section spacing
  - Computational time step selection
  - Theta weighting factor
  - Calculation tolerances and iterations
  - Lateral Structures/weirs
  - Manning's n values
  - Initial/Low flow conditions
  - Steep Streams/Mixed Flow regime
  - Drops in the bed profile
  - Bridge/Culverts
  - Cross section geometry and table properties
  - Breach characteristics
  - Junctions
  - Storage Areas
- 2D Model Stability Issues
  - Cell size and time step
  - Flood wave wetting front
  - Weird shaped/small cells
  - Channel Alignment/cell size
  - Partial cell wetting
  - Internal hydraulic structures

3

Model Stability

- An unstable numerical model is one for which certain types of numerical errors grow to the extent at which the solution begins to oscillate, or the errors become so large that the computations can not continue



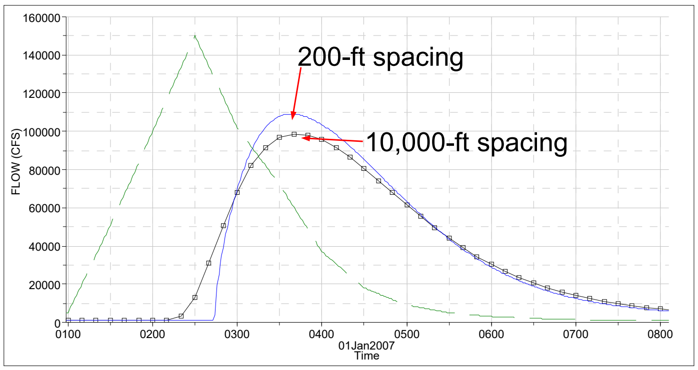
4

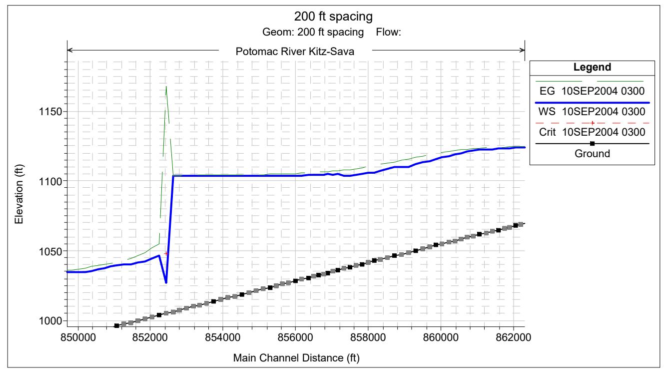
Developing a stable model is a common problem when working with an unsteady flow model of any size or complexity. Modeling a dam break flood wave is one of the most difficult unsteady flow problems to model.

Cross-Section Spacing

- Too large of a XS spacing can cause numerical damping of the flood wave (too low of a peak flow downstream), and/or model instability

- Too fine of XS spacing can cause wave steepening and model instability such as on the rising limb of the flood wave





5

Not enough cross sections: When cross sections are spaced far apart, and the changes in hydraulic properties are great, the solution can become unstable. In general, cross sections spaced too far apart will cause additional numerical diffusion, due to the derivatives with respect to distance being averaged over to long of a distance. Also, if the distance between cross sections is so great, such that the Courant number would be much greater than 1.0, then the model may also become unstable.

Cross Sections too Close. If the cross sections are too close together, then the derivatives with respect to distance may be overestimated, especially on the rising side of the flood wave. This can cause the leading edge of the flood wave to over steepen, to the point at which the model may become unstable.

XS Spacing Maximum and Minimum

Use Dr. Fread's and Samual's equations as a guide for maximum spacing.

Dr. Fread's Equation:

$$\Delta x \leq \frac{cT_r}{20}$$

Samuel's Equation:

$$\Delta x \leq \frac{0.15D}{S_0}$$

Where:

- Δx = Cross section spacing (feet)
- T_r = Time of rise of the main flood wave (seconds)
- c = Wave speed of the flood wave (ft/s)
- D = Average bank full depth of the channel (ft)
- S_0 = Average bed slope (ft/ft)

Minimum spacing for a dam break model should be in the range of 100 to 200 ft.

6

One of the first steps in stabilizing a dam breach model is to apply the correct cross section spacing. Fread's equation and Samuel's equations are good starting points. Samuels equation is a little easier to use since you only have to estimate the depth and slope. Frequently, bank full depth is used. For Fread's equation, although the time of rise of the hydrograph (Tr) is easy enough to determine, the wave speed (c) is a little more difficult to come by. Once a cross section spacing is decided upon, apply it to the entire reach using the HEC-RAS cross section interpolation routines. Make sure that the reach-wide method is applicable. At areas of extreme contraction and expansion, at grade breaks, or in abnormally steep reaches, further localized interpolation may be necessary.

Fread, D.L. (1988) (Revision 1991). "The NWS DAMBRK Model. Theoretical Background and User's Documentation." National Weather Service, Office of Hydrology, Silver Spring, Md.

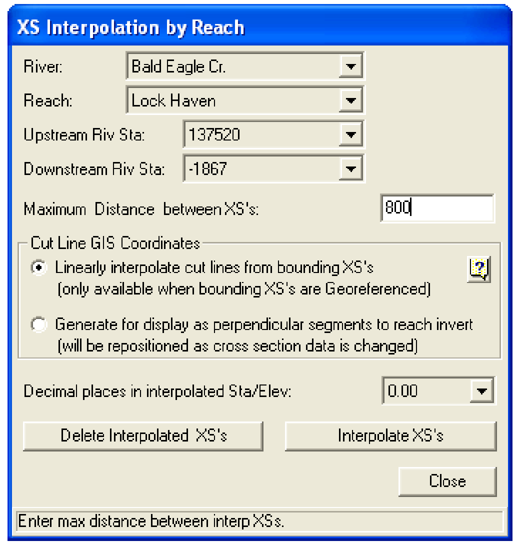
Fread, D.L., Lewis, J.M. (1993). "Selection of Dx and Dt Computational Steps for Four-Point Implicit Nonlinear Dynamic Routing Models" ASCE National Hydraulic Engineering Conference Proceedings, San Francisco, CA.

Samuels, P.G. (1989). "Backwater lengths in rivers", Proceedings -- Institution of Civil Engineers, Part 2, Research and Theory, 87, 571-582. Samuels equation should be

limited to being use for streams from 2 – 50 ft/mile in slope.

Cross-Section Interpolation

- Apply the XS interpolation to ensure a maximum spacing is not exceeded
- At problem areas you may need to use tighter spacing
  - Steep reaches
  - Transition zones
  - Grade breaks

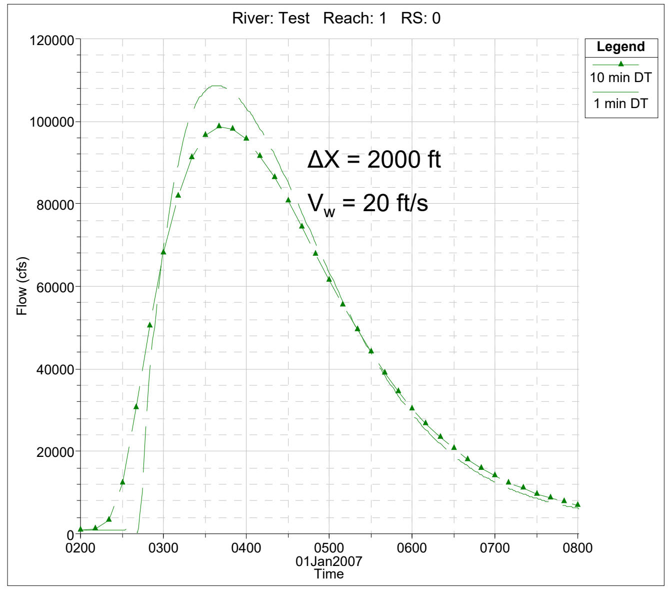


7

In general, it is always better to use real cross sections rather than interpolated. However, if acquiring more cross section data is not possible, then the cross section interpolation routines in HEC-RAS should be used to ensure that the cross section do not go over a maximum distance estimated from Samual's, or Fread's equation.

Computational Time Step

- Too large a time step will cause numerical diffusion (attenuation of the peak) and also model instability
- Too large of a time step can also lead to model instability
- Too small of a time step can lead to very long computation times



8

Too large of a time step: When the solution scheme solves the unsteady flow equations, derivatives are calculated with respect to distance and time. If the changes in hydraulic properties at a give cross section are changing rapidly with respect to time, the program may go unstable. The solution to this problem in general is to decrease the time step.

Too Small of a Time Step. If a time step is selected that is much smaller than what the Courant condition would dictate for a given flood wave, this can also cause model stability problems. In general to small of a time step will cause the leading edge of the flood wave to steepen, possible to the point of oscillating and going unstable.

Computational Time Step - continued

Stability and accuracy can be achieved by selecting a time step that satisfies the Courant Condition :

$$C_r = V_w \frac{\Delta t}{\Delta x} \leq 1 \qquad \Delta t \leq \frac{\Delta x}{V_w}$$

For most rivers, the flood wave velocity is calculated more accurately by:

$$V_w = \frac{dQ}{dA}$$

An approximate flood wave velocity can be calculated as:

$$V_w = \frac{3}{2}\overline{V}$$

9

Where: V_w = The flood wave speed, which is normally greater than the average velocity.

V = Average velocity of the flow

Δx = Distance between cross sections

Δt = computational time step

Q = flow rate

A = Flow area

User's should pay close attention to the Courant condition for selecting the computational interval.

Practical Time Step Selection

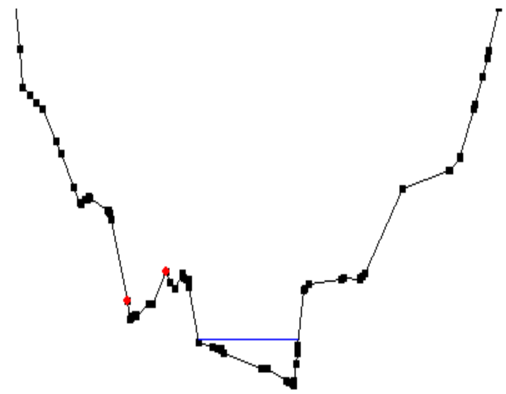
- For medium to large rivers the Courant condition may yield time steps that are to restrictive (i.e. a larger time step could be used and still maintain accuracy and stability)
- A practical time step is: $$\Delta t \leq \frac{T_r}{20}$$
- Remember that for Dambreak models, typical time steps are in the range of 1- 60 seconds due to the short time of rise and very fast flood wave velocities..

10

Where: T_r = Time of rise of the flood wave.

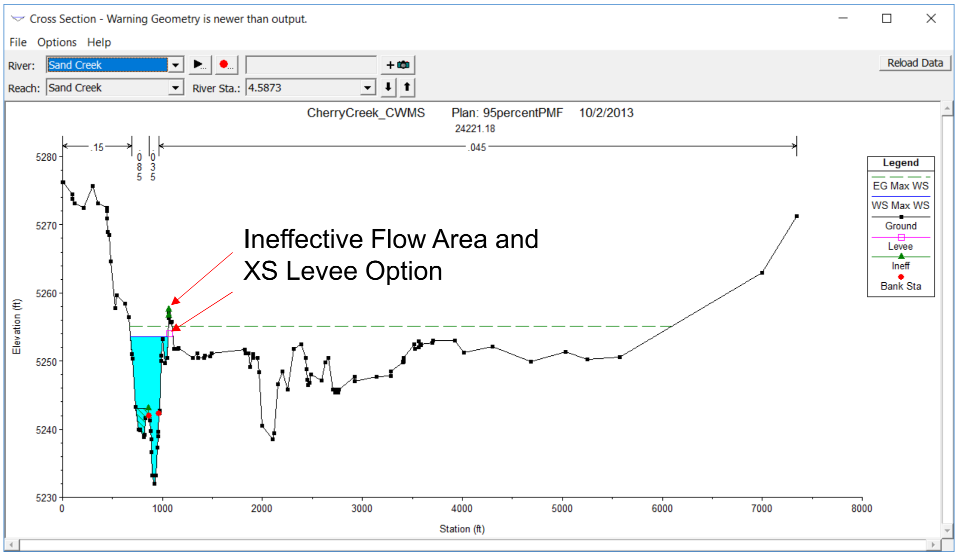
Bad Low Flow Channel Data

- Using the XS Plot can help spot isolated problems such as:
  - Missing or bad channel data.
  - Incorrect Bank Station locations
  - Bad Manning's n Values
  - Bad Station-elevation points
- Can help spot transition problems
  - Contraction/Expansion Areas
  - Ineffective Flow Areas
  - Levees



Once a location for an instability is determined on the profile plot, the cros-section plot can be used to investigate the cause of the instability. The cross-section plot will show isolated problems such as incorrectly placed bank stations, poor n-values, and bad station-elevation data. In addition, scrolling through its neighboring cross sections can give you an idea of transition problems like contractions and expansions that occur to abruptly, poorly defined ineffective flow areas, or incorrectly handled levees or natural high ground spots.

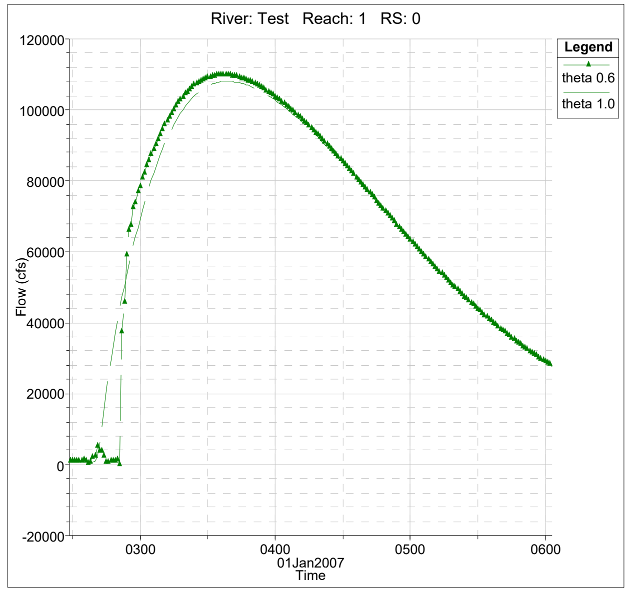
Ineffective Flow Areas and XS Levee Options



12

Theta – Implicit Weighting Factor

- Theta is a weighting applied to the finite difference approximations when solving the unsteady flow equations
- Theoretically Theta can vary from 0.5 to 1.0 (practical limit is from 0.6 to 1.0).
- Theta of 1.0 provides the most stability
- Theta of 0.6 provides the most accuracy, but can increase instabilities
- The default in HEC-RAS is 1.0
- Once you have your model developed, try to reduce theta towards 0.6, as long as the model stays stable



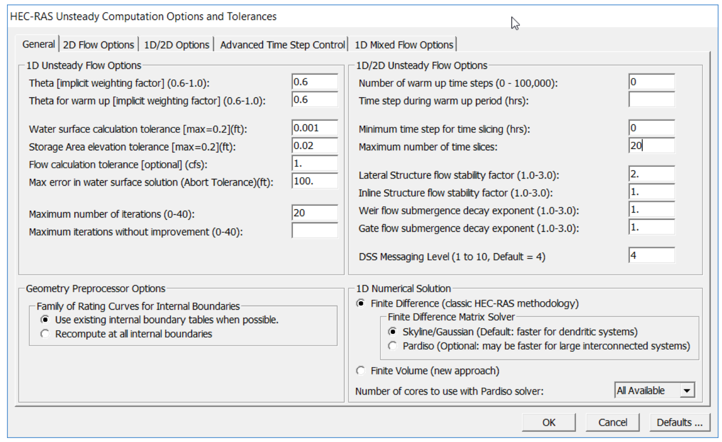
13

Larger values of theta increase numerical diffusion, but, by how much? Experience has shown that for short period waves that rapidly rise, theta of 1.0 can produce significant errors. However, these errors can be reduced by using smaller time steps.

When choosing theta, one must balance accuracy and computational robustness. Larger values of theta produce a solution that is more robust, less prone to blowing up. Smaller values of theta, while more accurate, tend to cause oscillations in the solution, which are amplified if there are large numbers of internal boundary conditions.

Calculation Options and Tolerances

- Theta
- Water surface/SA/flow tolerance
- Maximum Number of Iterations
- Warm up: duration and time step
- Lateral/Inline Structure Stability
- Weir/Gate flow submergence



14

Water Surface, Storage Area, and Flow Tolerances: Three solution tolerances can be set or changed by the user: Water surface calculation (0.02 default); Storage area elevation (0.05 default); and Flow calculation (Default is that it is not used). The default values should be good for most river systems. Only change them if you are sure!!!

Making the tolerances larger can reduce the stability of the solution. Making them smaller can cause the program to go to the maximum number of iterations every time.

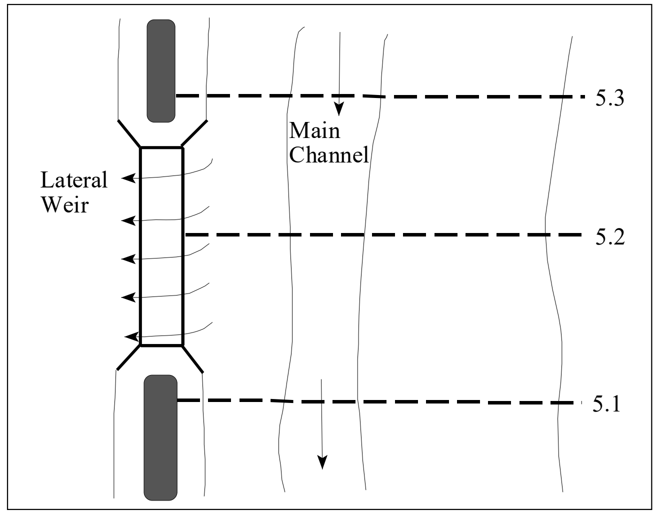
Maximum Number of Iterations: At each time step derivatives are estimated and the equations are solved. All of the computation nodes are then checked for numerical error. If the error is greater than the allowable tolerances, the program will iterate. The default number of iterations in HEC-RAS is set to 20. Iteration will generally improve the solution. This is especially true when your model has lateral weirs and storage areas.

Warm up time step and duration: The user can instruct the program to run a number of iterations at the beginning of the simulation in which all inflows are held constant. This is called the warm up period. The default is not to perform a warm up period, but the user can specify a number of time steps to use for the warm up period. The user can also specify a specific time step to use (default is to use the user selected computation interval). The warm up period does not advance the simulation in time, it is generally used to allow the unsteady flow equations to establish a stable flow and stage before proceeding with the computations.

Time Slicing: The user can control the maximum number of time slices and the minimum time step used during time slicing. There are two ways to invoke time slicing: rate of change of an inflow hydrograph or when a maximum number of iterations is reached.

Inline/Lateral Structure Stability Issues

- Long and flat lateral weirs can often be a source of model instability
  - Small change in stage in the river results in big change in flow going out the lateral structure
  - Flow is assumed to be constant over the time step
- Solutions:
  - Reduce the computation interval
  - Put a small slope on the lateral weir
  - Use lateral structure stability factors
- Opening and Closing Gates Quickly
  - Reduce the computational time step
  - Open and close gates slower



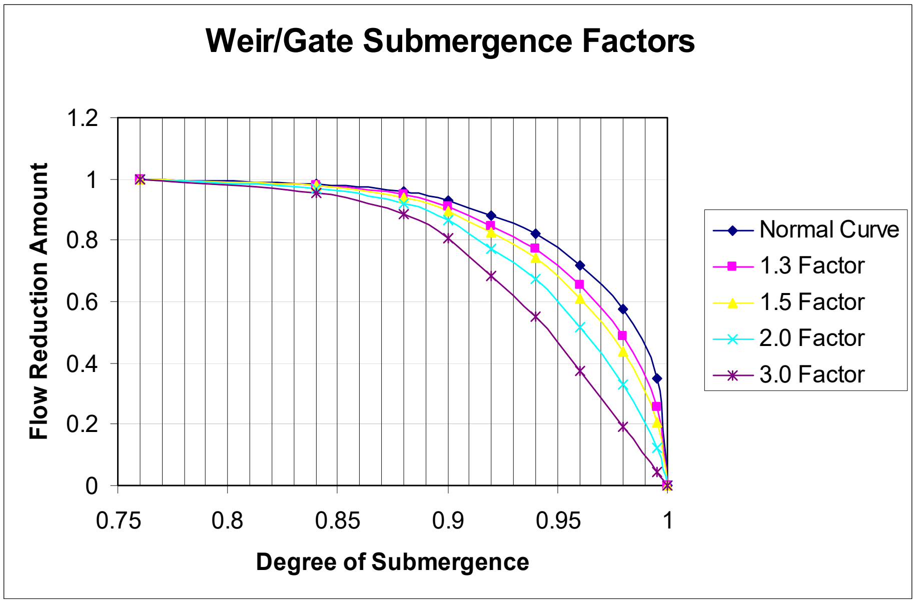
15

Inline and Lateral Structures can often be a source of instability in the solution. Especially lateral structures, which take flow away or bring it into the main river. During each time step, the flow over a weir/spillway is assumed to be constant. This can cause oscillations by sending too much flow during a time step. One solution is to reduce the time step. Another solution is to use Inline and Lateral Structure stability factors, which can smooth these oscillations by damping the computed flows. However, using these stability factors can reduce the accuracy of the computed values. The Inline and Lateral Structure stability factors can range from 1.0 to 3.0. The default value of 1.0 is essentially no damping of the computed flows. As you increase the factor you get greater dampening of the flows (which will provide for greater stability), but less accuracy.

Long and flat Lateral Weirs/Spillways: during the computations there will be a point at which for one time step no flow is going over the lateral weir, and then the very next time step there is. If the water surface is rising rapidly, and the weir is wide and flat, the first time the water surface goes above the weir could result in a very large flow being computed (i.e. it does not take a large depth above the weir to produce are large flow if it is very wide and flat). This can result in a great decrease in stage from the main river, which in turn causes the solution to oscillate and possible go unstable. This is also a common problem when having large flat weirs between storage areas. The solution to this problem is to use smaller computational time steps, and/or weir/spillway stability factors.

Opening gated spillways to quickly: When you have a gated structure in the system, and you open it quickly, if the flow coming out of that structure is a significant percentage of the flow in the receiving body of water, then the resulting stage, area and velocity will increase very quickly. This abrupt change in the hydraulic properties can lead to instabilities in the solution. To solve this problem you should use smaller computational time steps, or open the gate a littler slower, or both if necessary.

Weir/Gate Submergence Exponents

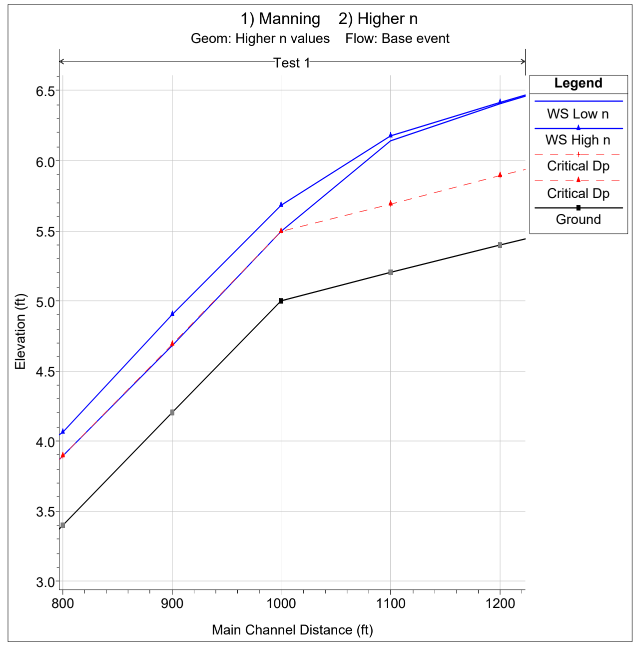


16

To reduce the oscillations, the user can increase the Weir/Gate Submergence decay exponent. This factor can vary from 1.0 to 3.0. A factor of 1.0 leaves the submergence criteria in its original form. Using a factor greater than one causes the program to use larger submergence factors earlier, and makes the submergence curve less steep at high degrees of submergence. A plot of the submergence curves for various factors is shown in the Figure above.

Manning’s n Values

- Manning’s n values that are too low can cause model instability
  - Lower depths
  - Higher velocities
  - Supercritical flow
  - Flow transitions
- Manning’s n values that are too high will locally increase stage and attenuate the hydrograph more as it moves downstream



17

Manning’s n values can also be a source of model instability. Manning’s n values that are too low, will cause shallower depths of water, higher velocities, and possibly even supercritical flows. This is especially critical in steep streams, where the velocities will already be high. User’s should check there estimated Manning’s n values closely in order to ensure reasonable values. It is very common to underestimate Manning’s n values in steep streams. Use Dr. Robert Jarret’s equation for steep streams to check your main channel Manning’s n values.

Over estimating Manning’s n values will cause higher stages and more hydrograph attenuation than may be realistic.

Low Flow Conditions

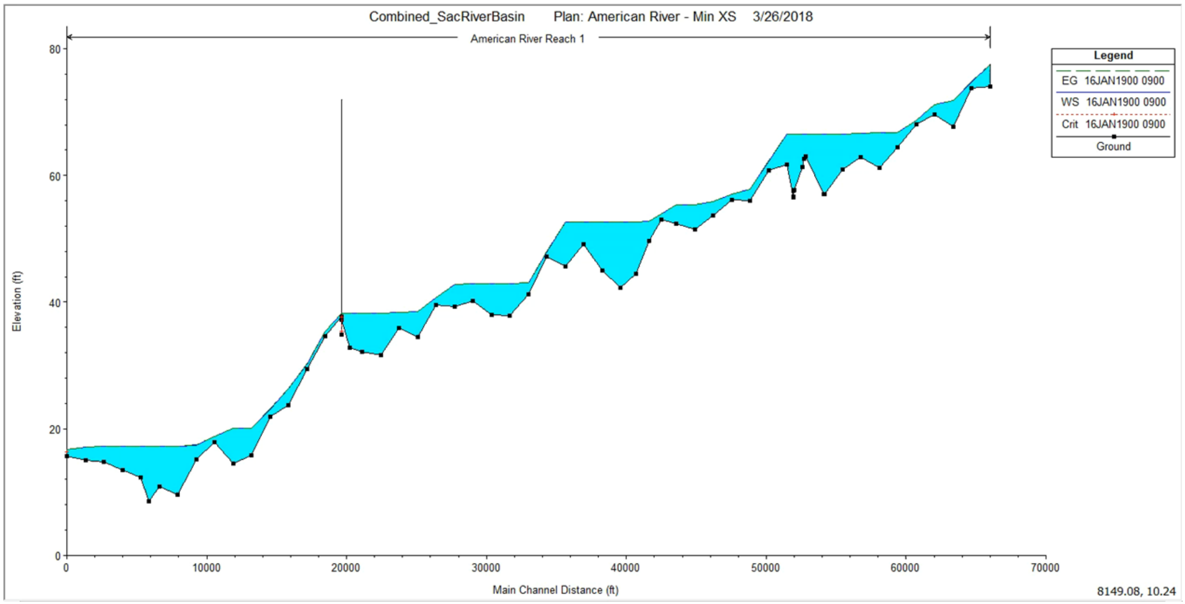
- Low flows are often a source of model instability
  - Pools and riffles (flow passes through critical depth)
  - Very shallow depths (When flood wave starts the change in depth/velocity is very large, therefore derivatives are large)
- Solutions
  - Increase Base Flow of Hydrograph (change hydrographs directly or use the Qmin option on the hydrograph editor)
  - Rule of thumb: Start with 1% of peak, don't exceed 10% of peak
  - Use a "Pilot Channel" to smooth our bed irregularities and provide some artificial depth.

18

If any portion of an inflow hydrograph is so low that it causes the stream to go through a pool and riffle sequence, it may be necessary to increase the base flow. The minimum flow value must be small enough that it is negligible when compared to the peak of the flood wave. A good rule of thumb is to start with a minimum flow equal to about 1 % of the peak flood (inflow hydrograph, or dam breach flood wave) and increase as necessary to 10%. If more than 10% is needed, then the problem is probably from something else.

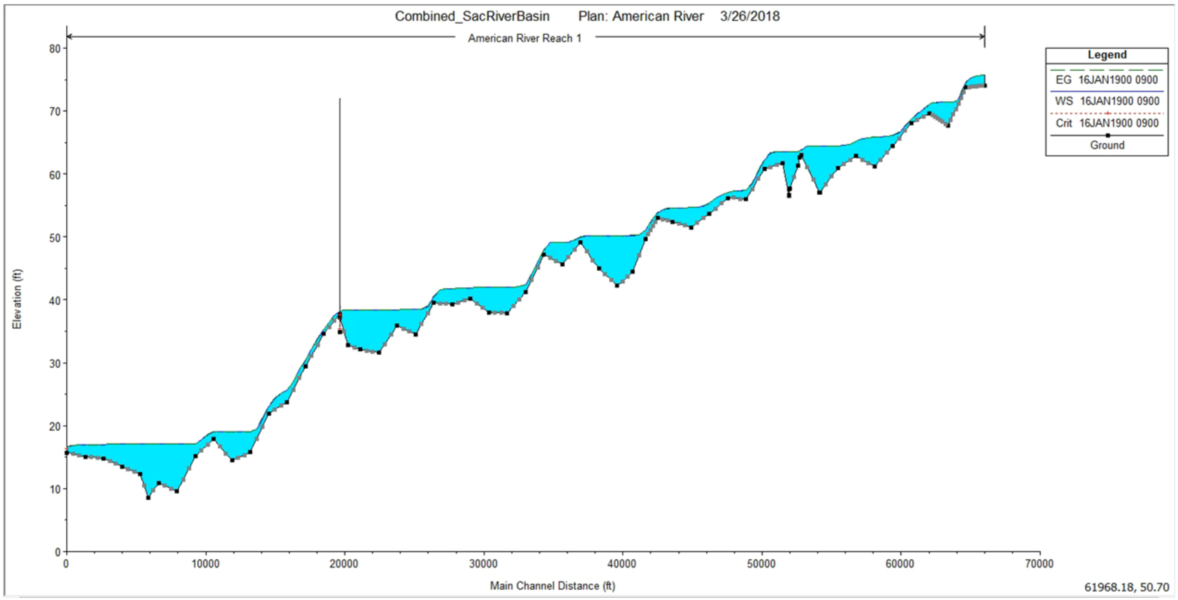
Very shallow depths of water: When starting a simulation it is very common to start the system at low flows. If you have some cross sections that are fairly wide, the depth will be very small. As flow begins to come into the river, the water surface will change quickly. The leading edge of the flood wave will have a very steep slope. Sometimes this steep slope will cause the solution to reduce the depth even further downstream of the rise in the water surface, possible even producing a negative depth. This is do to the fact that the steep slope gets projected to the next cross section downstream when trying to solve for its water surface. The best solution to this problem is to use what is called a pilot channel. A pilot channel is a small slot at the bottom of the cross section, which gives the cross section a greater depth, without adding much flow area. This allows the program to compute shallow depths on the leading edge of the flood wave without going unstable. Another solution to this problem is to use a larger base flow at the beginning of the simulation.

Profile Plot with Large XS Spacing



19

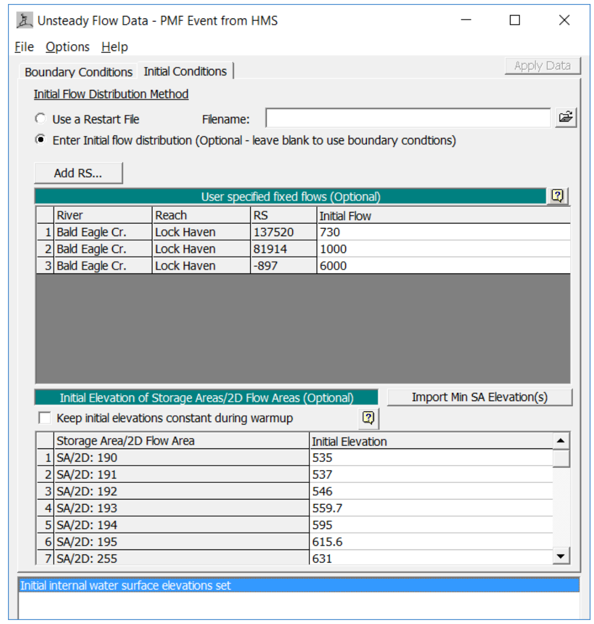
Profile Plot with Fine XS Spacing



20

Initial Conditions

- Initial condition flows need to be consistent with boundary condition flows at time zero
  - If dendritic system – can leave initial condition flows blank
- Initial Reservoir elevations need to be consistent with initial flows and gate settings used
- Initial Storage area elevations must be consistent with water elevations in connected streams
- Inconsistent, initial conditions can often cause the model to blow up right at the start of the simulation

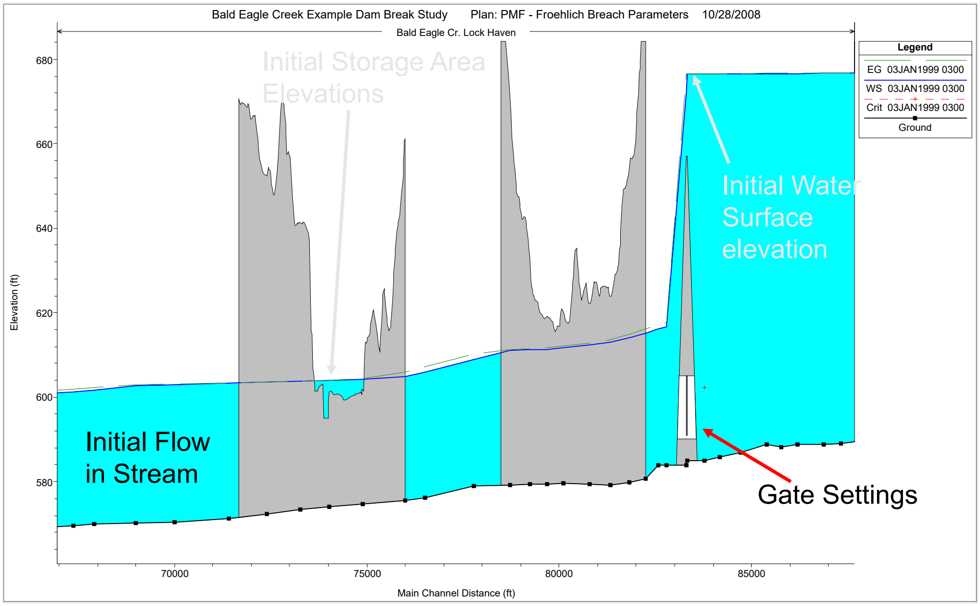


21

Make sure that the initial conditions flow is consistent with the first time step flow, or minimum flow value, which ever is greater. User's must also pay close attention to initial gate settings for the reservoir, and the initial stage of the pool in the reservoir. The initial condition flow values must be consistent with all inflow hydrographs, as well as the initial flows coming out of the reservoir.

Flows entered on the initial conditions tab are used for calculating stages in the river system based on steady flow backwater calculations. If these flows and stages are inconsistent with the initial flows in the hydrographs, and coming out of the reservoir, then the model may have computational stability problems at the very beginning of the unsteady flow computations.

Initial Conditions - Continued



22

The following items must be set correctly for the initial conditions:

1. Flows in all the reaches make sense and they add up correctly.
2. Water surface elevation at the Dam must be set by forcing the water surface elevation at the cross section just upstream of the Dam. Otherwise it will be computed based on the flow in the reach and gate settings.
3. Gate openings must be set to output the same flow as what you set for the initial condition flow.
4. All storage area elevations must be set to be consistent with the river stages for the reaches they are connected too.

Steep Streams and Mixed-Flow Regime

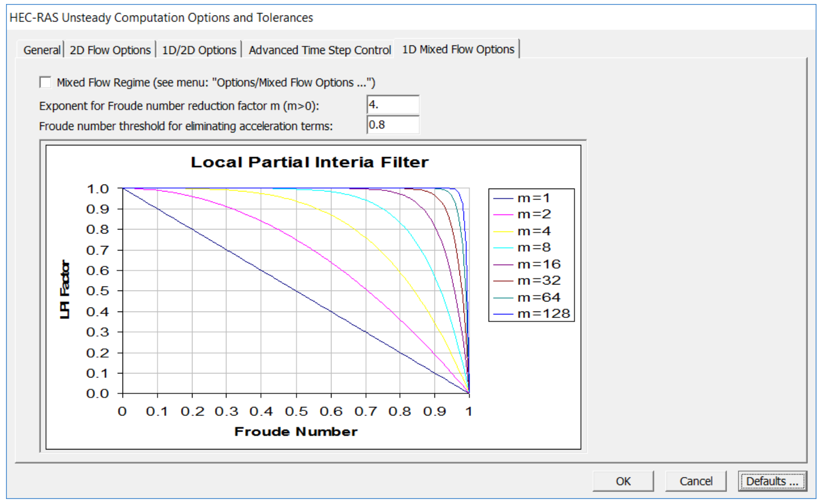
- Higher velocities and rapid changes in depth and velocity are more difficult to model and keep a stable solution
- As Froude number approaches 1.0 (critical depth), the inertial terms of the St. Venant equations and their derivatives tend to cause model instabilities
- Model goes to critical depth – RAS is limited to subcritcal flow for unsteady flow simulations, unless you turn on the mixed flow option
- Solutions:
  - Higher n values (n values often under-estimated in steep streams)
  - Increase base flow of hydrographs and initial condition flows
  - Try turning on the Mixed-Flow Regime option
  - Use Modified Puls Routing Option through steep reaches

23

Model goes to critical depth: The default solution methodology for unsteady flow routing within HEC-RAS is generally for subcritcal flow. However, the software does have an option to run in a mixed flow regime mode. However, this option should not be used unless you truly believe you have a mixed flow regime river system. If you are running the software in the default mode (subcritical only, no mixed flow), and if the program goes down to critical depth at a cross section, the changes in area, depth, and velocity are very high. This sharp increase in the water surface slope will often cause the program to overestimate the depth at the next cross section upstream, and possible underestimate the depth at the next cross section downstream (or even the one that went to critical depth the previous time step). One solution to this problem is to increase the Manning's n value in the area where the program is first going to critical depth. This will force the solution to a subcritical answer and allow it to continue with the run. If you feel that the true water surface should go to critical depth, or even to a supercritical flow regime, then the mixed flow regime option should be turned on. Another solution is to increase the base flow in the hydrographs, as well as the base flows used for computing the initial conditions. Increased base flow will often dampen out any water surfaces going towards or through critical depth due to low flows.

Mixed Flow Regime Option for Unsteady Flow

- HEC-RAS uses the Local Partial Inertia (LPI) method to model mixed flow regimes
- User can adjust 2 parameters for reducing the inertial terms via the LP factor



LP factor

Inertial Terms

$$\sigma\left[\frac{\partial Q}{\partial t}+\frac{\partial}{\partial t}\left(\frac{\beta Q^2}{A}\right)\right]+gA\left(\frac{\partial z_s}{\partial x}+S_f\right)=v_x q_l$$

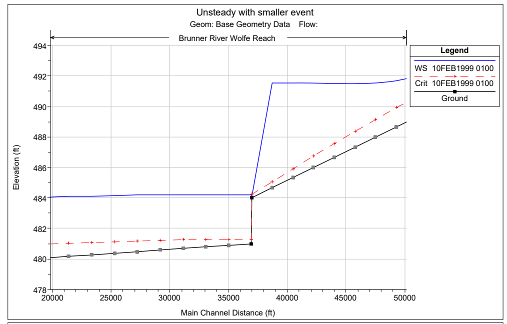
24

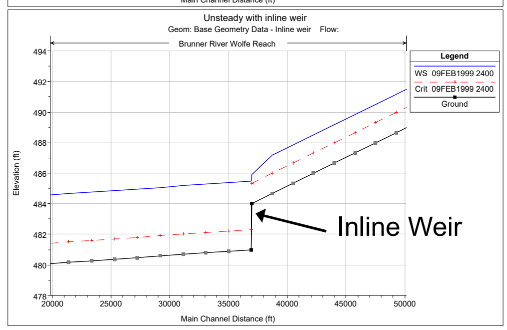
In order to solve the stability problem for a mixed flow regime system, Dr. Danny Fread (Fread, 1986) developed a methodology called the "Local Partial Inertia Technique." The LPI method has been adapted to HEC-RAS as an option for solving mixed flow regime problems when using the unsteady flow analysis portion of HEC-RAS. This methodology applies a reduction factor to the two inertia terms in the momentum equation as the Froude number goes towards 1.0.

The default values for the equation are FT = 1.0 (Froude Number Threshold) and m = 10 (exponent). When the Froude number is greater than the threshold value, the factor is set to zero. The user can change both the Froude number threshold and the exponent. As you increase the value of both the threshold and the exponent, you decrease stability but increase accuracy. As you decrease the value of the threshold and/or the exponent, you increase stability but decrease accuracy. To change either the threshold or the exponent, select **Mixed Flow Options** from the **Options** menu of the Unsteady Flow Analysis window.

Drops In the Bed Profile

- Significant drops in the elevation of the channel bed can cause flow to pass through critical depth and results in an unstable model solution
- Solutions:
  - Use an Inline Weir to model drop
  - Increase Base Flow
  - Rating curve for cross section at top of the drop
  - More cross sections and turn on mixed flow regime option





25

Significant drops in the bed profile can also be a source of model stability problems, especially at low flows. If the drop is very small, then usually an increase in baseflow will drowned out the drop, thus preventing the model from passing through critical depth. If the drop is significant, then it should be modeled with an inline structure using a weir. This will allow the model to use a weir equation for calculating the upstream water surface for a given flow, rather than using the unsteady flow equations. This produces a much more stable model, as the program does not have to model the flow passing through critical depth with the unsteady flow equations. HEC-RAS automatically handles submergence on the weir, so this is not a problem.

Bridges and Culverts

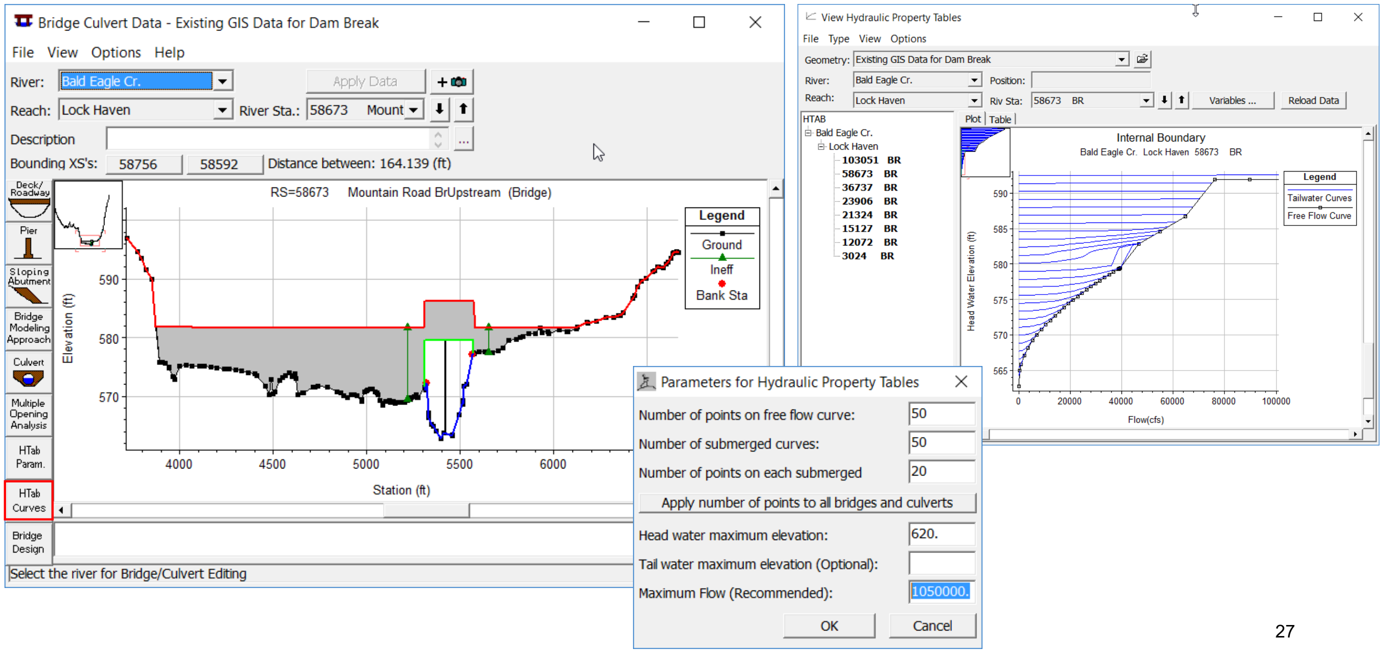
- It is very common to have rapid changes in depth and velocity at Bridges and Culverts, however this can be a source of model instability
- As flow transitions from low flow to pressure flow at the structure the water surface upstream will jump up very quickly.
- HEC-RAS pre-processes bridges/culverts into a family of curves (tailwater/headwater vs flow). Common Problems with the curves are:
  - Curve extents not high enough (change default extents)
  - Abrupt transitions in curves (adjust bridge parameters or use smaller ΔT)

26

Bridge/Culvert crossings can be a common source of model stability problems when performing a Dam Break analysis. Many bridges will be overtopped during such an event. Many of those bridges may in fact be washed out during such an event. Common problems at bridges/culverts are the extreme rapid rise in stages when flow hits the low chord of the bridge deck or the top of the culvert. Modelers need to check the computed curves closely and make sure they are reasonable. One solution to this problem is to use smaller time steps, such that the rate of rise in the water surface is smaller for a given time step. Modelers may also need to change hydraulic coefficients to get curves that have more reasonable transitions.

An additional problem is when the curves do not go high enough, and the program extrapolates from the last two points in the curve. This extrapolation can cause problems when it is not consistent with the cross-section geometry upstream and downstream of the structure.

Bridge/Culvert Family of Curves

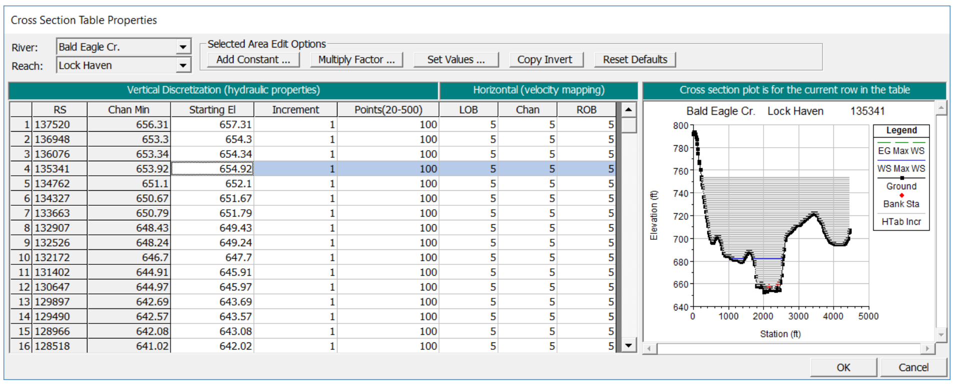


27

Bridge/Culvert Family of rating curves: The program creates a family of rating curves to define all the possible headwater, tailwater, and flow combinations that can occur at a particular structure. The user can control how many submerged curves get calculated (default = 50), how many points in each curve (default = 20), and the properties used to define the limits of the curves (maximum headwater, maximum tailwater, maximum flow, and maximum head difference). By default, the software will take the curves up to an elevation equal to the highest point in the cross section just upstream of the structure. This may lead to curves that are too spread out and go up to a flow rate that is way beyond anything realistic for that structure. These type of problems can be reduced by putting in specific table limits for maximum headwater, tailwater, flow, and head difference.

Cross Section Geometry and Table Properties

- Bad cross section properties, commonly caused by: levee options, ineffective flow areas, Manning's n values, etc…
- Cross section properties that do not go high enough, or are way too high (curves are spread too far apart)
- Not enough definition in the property tables



	Vertical Discretization (hydraulic properties)					Horizontal (velocity mapping)		
	RS	Chan Min	Starting El	Increment	Points(20-500)	LOB	Chan	ROB
1	137520	656.31	657.31	1	100	5	5	5
2	136948	653.3	654.3	1	100	5	5	5
3	136076	653.34	654.34	1	100	5	5	5
4	135341	653.92	654.92	1	100	5	5	5
5	134762	651.1	652.1	1	100	5	5	5
6	134327	650.67	651.67	1	100	5	5	5
7	133663	650.79	651.79	1	100	5	5	5
8	132907	648.43	649.43	1	100	5	5	5
9	132526	648.24	649.24	1	100	5	5	5
10	132172	646.7	647.7	1	100	5	5	5
11	131402	644.91	645.91	1	100	5	5	5
12	130647	644.97	645.97	1	100	5	5	5
13	129897	642.69	643.69	1	100	5	5	5
14	129490	642.57	643.57	1	100	5	5	5
15	128966	642.08	643.08	1	100	5	5	5
16	128518	641.02	642.02	1	100	5	5	5

28

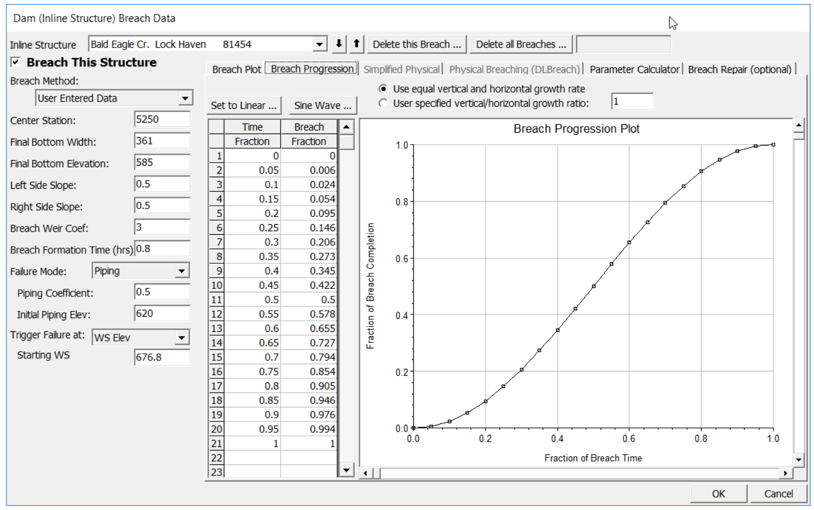
Bad cross section properties: All of the cross sections get converted to tables of hydraulic properties (elevation versus area, conveyance, and storage). If the curves that represent these hydraulic properties have abrupt changes with small changes in elevation, this can also lead to instability problems. This situation is commonly caused by: levees being overtopped with large areas behind them (since the model is one dimensional, it assumes that the water surface is the same all the way across the entire cross section); and ineffective flow areas with large amounts of storage areas that are turned on at one elevation, and then turn off at a slightly higher elevation (this makes the entire area now used as active conveyance area). There are many possible solutions to these problems, but the basic solution is to not allow the hydraulic properties of a cross section to change so abruptly. If you have a levee with are large amount of area behind it, model the area behind the levee separately from the cross section. This can be done with either a storage area or another routing reach, whichever is most hydraulically correct for the flow going over the levee or if the levee breaches. With large ineffective flow areas, the possible solutions are to model them as being permanently on, or to put very high Manning's n values in the ineffective zones.

Cross section property tables that do not go high enough: The program creates tables of elevation versus area, conveyance, and storage area for each of the cross sections. These tables are used during the unsteady flow solution to make the calculations much faster. By default, the program will create tables that extend up to the highest point in the cross section, however the user can override this and specify their own table properties (increment and number of points). If during the solution the water surface goes above the highest elevation in the table, the program simply extrapolates the hydraulic properties from the last two points in the table. This can lead to bad water surface elevations or even instabilities in the solution.

Not enough definition in cross section property tables: The counter problem to the previous paragraph is when the cross-section properties in a given table are spread too far apart, and do not adequately define the changes in the hydraulic properties. Because the program uses straight-line interpolation between the points, this can lead to inaccurate solutions or even instabilities. To reduce this problem, we have increased the allowable number of points in the tables to 100.

Breach Characteristics

- By default HEC-RAS uses a linear growth rate for forming the breach
- Rapid change in flow at the beginning of the breach can cause instability
- Try non-linear breach to smooth out change in flow (Sine Wave or User entered values)



	Time Fraction	Breach Fraction
1	0	0
2	0.05	0.006
3	0.1	0.024
4	0.15	0.054
5	0.2	0.095
6	0.25	0.146
7	0.3	0.206
8	0.35	0.273
9	0.4	0.345
10	0.45	0.422
11	0.5	0.5
12	0.55	0.578
13	0.6	0.655
14	0.65	0.727
15	0.7	0.794
16	0.75	0.854
17	0.8	0.905
18	0.85	0.946
19	0.9	0.976
20	0.95	0.994
21	1	1
22		
23		

29

If the user puts in a very large breach, over a very short period of time, and they use the linear breach growth rate, the model will have a very abrupt change in flow starting right at the beginning of the breaching process. This rapid change in flow at the leading edge of the flood wave may cause an instability at the beginning of the breaching process, just downstream of the dam. Some possible solutions to this are:

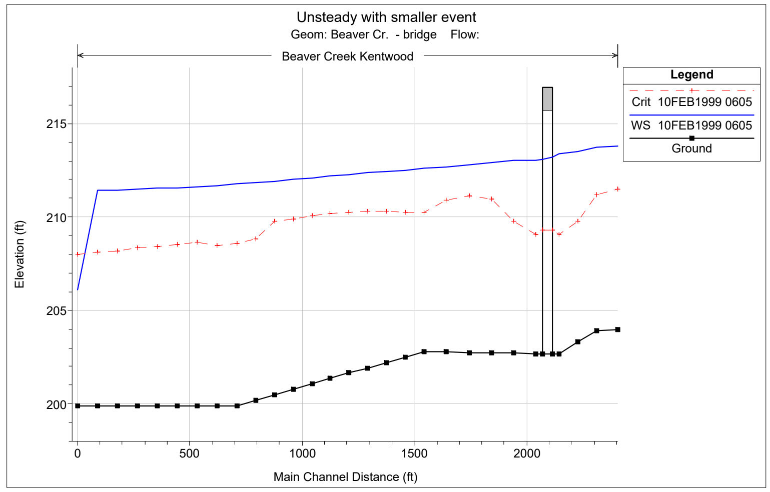
•Smaller time step

•Use the non-linear breach progression (sine wave or user entered)

•Increasing the overall breach time.

Bad Downstream Boundary Conditions

- Rating curve with bad points or not high enough
- Slope for normal depth to steep

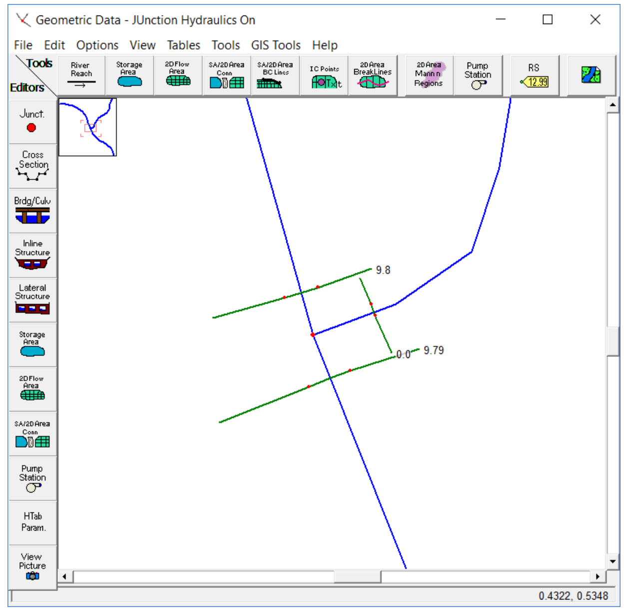


30

Bad downstream boundary condition: If the user entered downstream boundary condition causes abrupt jumps in the water surface, or water surface elevations that are too low (approaching or going below critical depth), this can cause oscillations in the solution that may lead to it going unstable and stopping. Examples of this are rating curves with not enough points or just simply to low of stages for a given flow; and normal depth boundaries where the user has entered to steep of a slope for the energy gradeline.

Stream Junctions

- Unsteady Flow Default at Junctions is to force the same water surface at all cross sections that bound a Junction
- Junction option to use Energy Balance to compute water surfaces across a junction



31

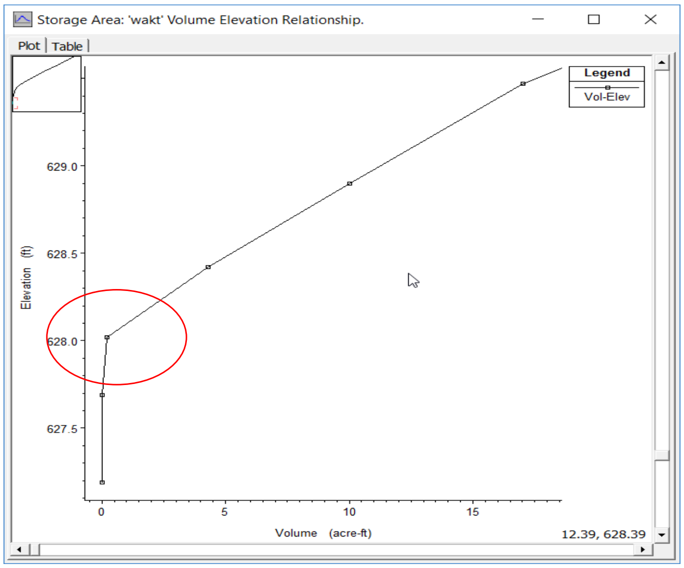
Junction Problems

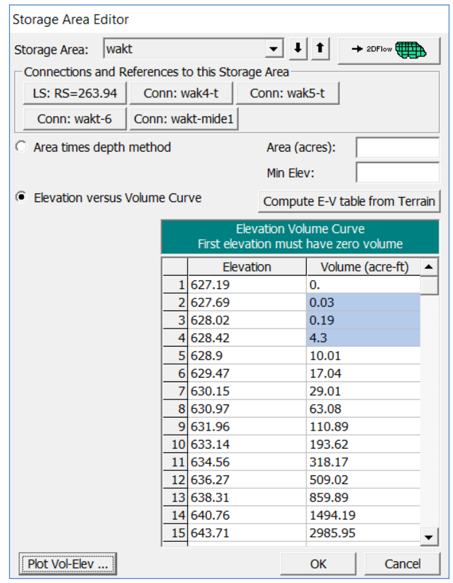
- Cross sections spaced too far apart for assumption of equal water surface
  - Solution: Place Cross section closer together around junctions
- Tributary cross section invert at much higher elevation that main stem cross sections. Assumption of equal water surface will cause tributary to have too low of water surface and go unstable!!!
  - Use Energy Balance Junction Option
  - Move cross sections closer to Junction
  - Adjust tributary cross section inverts at lower sections
  - Add Pilot channel up tributary sections

32

Storage Area Issues

- The main issue is very abrupt transitions at the low end of the curve

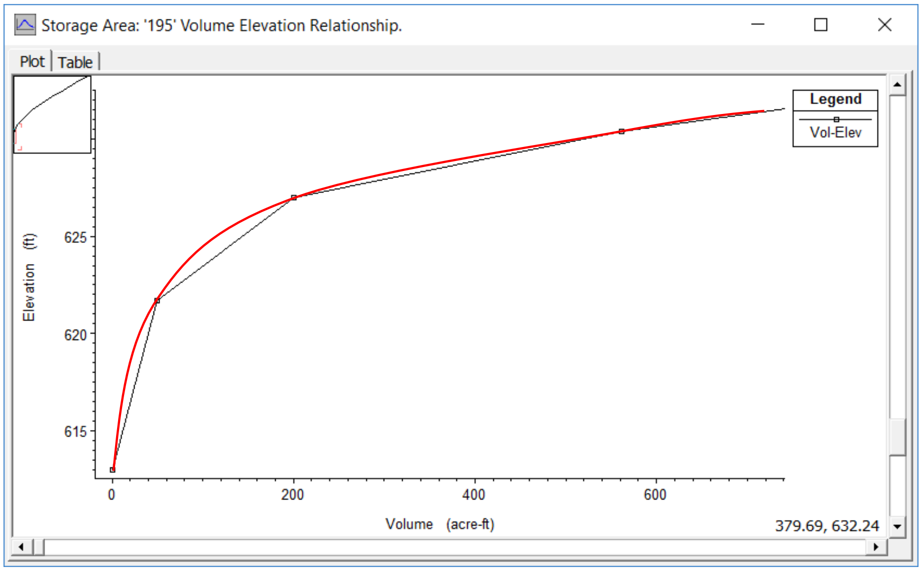




33

Storage Area Issues

- A second issue is to few points defining the curve at the low end



34

2D Flow Area Stability Issues

- Cell size and time step
- Flood wave wetting front
- Weird shaped/small cells
- Channel Alignment/cell size
- Partial cell wetting
- Internal hydraulic structures

35

Cell Size and Time Step

- Too large a time step for the cell size/velocity can cause model instability.
- Diffusion wave is more forgiving than Full eqns. But full St. Venant more accurate.
- Use Courant condition pick the best time step.
- The time step you use will also depend on how fast the hydrograph rises:
  - Fast rising = Lower time step/Courant number
  - Slow rising = Higher time step/Courant number

Courant Condition Guidelines

- Shallow Water Equations

$$\boldsymbol{C} = \frac{\boldsymbol{V\Delta T}}{\boldsymbol{\Delta X}} \leq \mathbf{1.0} \quad (with\ a\ \max\ C = 3.0)$$

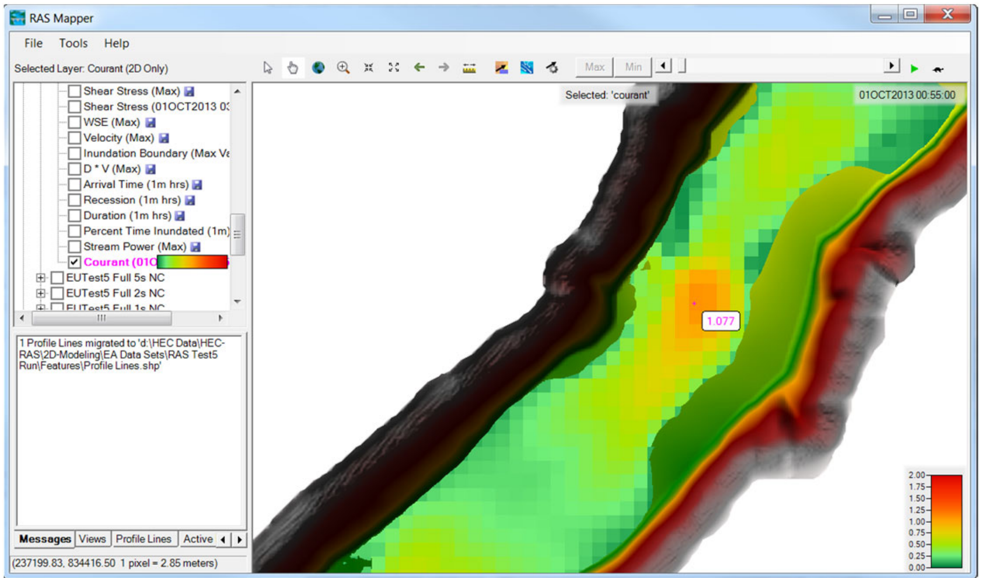
Or $\Delta T \leq \frac{\Delta X}{V}$ (With C = 1.0)

- Diffusion Wave Equations:

$$\boldsymbol{C} = \frac{\boldsymbol{V\Delta T}}{\boldsymbol{\Delta X}} \leq \mathbf{2.0} \quad (with\ a\ \max\ C = 5.0)$$

Or $\Delta T \leq \frac{2\Delta X}{V}$ (With C = 2.0)

RAS Mapper Courant Number Map



38

Floodwave Wetting Front

- 2D Models can often go unstable at the wetting front of the floodwave
  - Can cause model iterations
  - Can also cause bad max velocity plots
- Ways to improve this:
  - Poor Cell size – use polygon refinement tool
  - To large of an elevation change across a single cell – make cells smaller or larger
  - Breaklines for high ground barriers

39

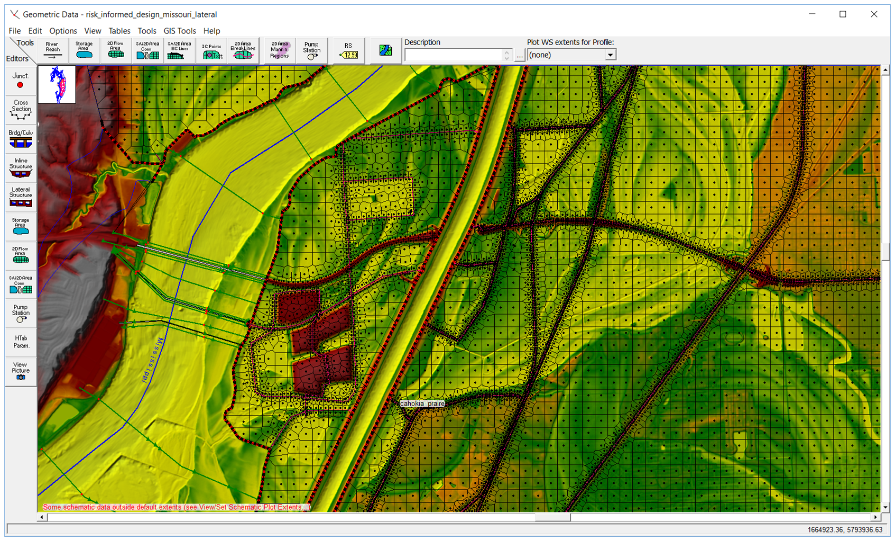
Weird Shaped Cells/Small Faces

- Cells need to transition in size slowly
  - No more than 50% change in size
- Cells with one face that is very small compared to other faces and cells – this may cause excessive model iterations.
- Minor adjustments in either moving the cell centers, deleting cells, or adding additional cells to smooth out the transitions, and remove small faces can get rid of model iteration issues.

40

Breaklines

- In general people do not use enough breaklines
- Use breaklines along high ground barriers to flow in order to align faces
  - This will improve accuracy
  - This will improve model stability



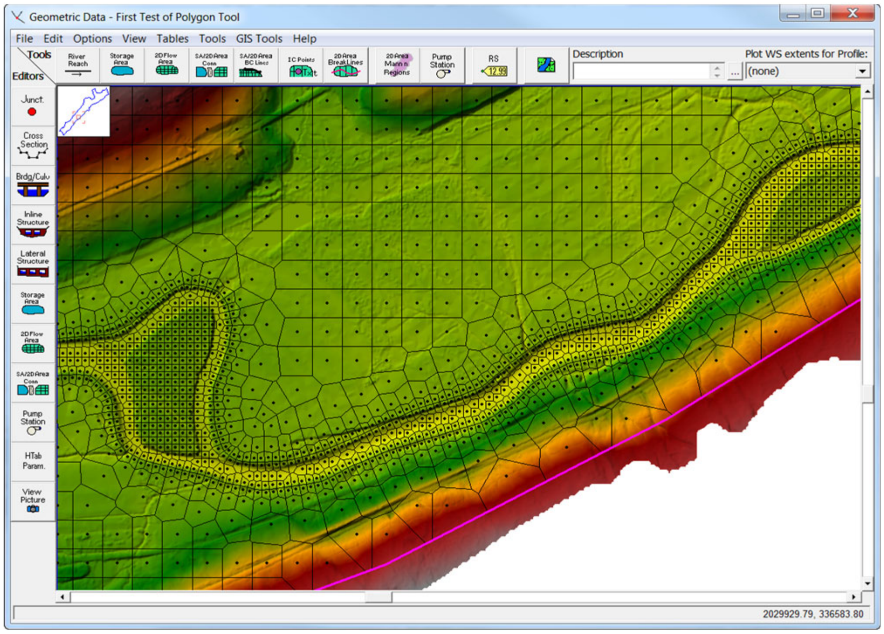
41

Channel Alignment and Cell Size

- Need to define the channel portion of the 2D mesh appropriately
- 2D Faces need to be aligned with high ground separating channel from floodplain
- Channel needs to have enough cells across the channel in order to get a good velocity profile. Recommend at least 7 to 10 cells across channel
- Fewer cells ok for water surface only
- Use Polygon Refinement tool to accomplish this

42

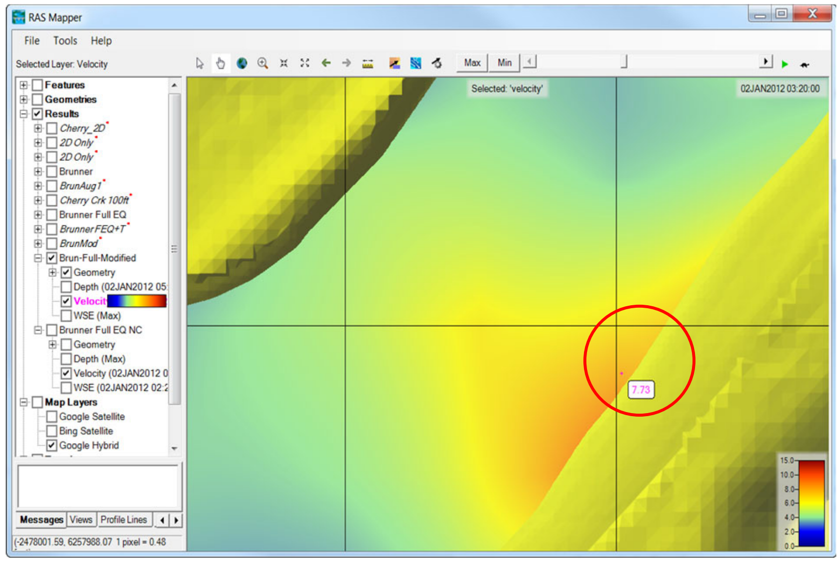
Example of Polygon Refinement Tool for Main Channel



43

Partial Cell Wetting Issue

- Excessive model iteration can occur when just a corner of a cell has flow and the velocity is high.
- This will be even more unstable when flow comes into a cell through a small portion of a face but can leave over a much larger portion of another face
- Adjust cell sizes, use breaklines and polygon refinement tool to fix



44

Internal Hydraulic Structures

- To small of cell sizes at invert of culvert or gate.
  - Small cells have less volume
  - Flow/volume for the culvert is computed over the time step as V = Q x T
- Highly submerged weirs with culverts and gates can have stability issues. “Weir and Gate Flow Submergence decay exponents”
- Flow over the embankment can be computed as weir flow or 2D Flow Equations
  - Use Weir options when the is a high embankment
  - Use 2D flow option for non weir flow situations

45