Unsteady Flow Modeling with HEC-RAS

Alex Sánchez, PhD

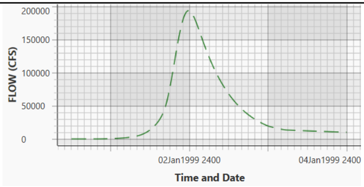
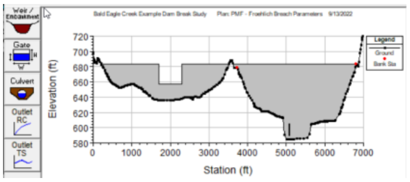
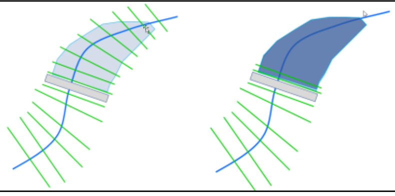
Stanford Gibson, PhD

Senior Hydraulic Engineer
USACE, Institute for Water Resources, Hydrologic Engineering Center

US Army Corps of Engineers®

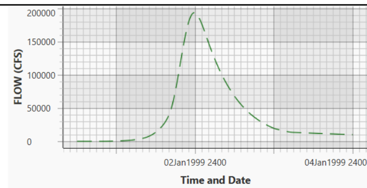
This lecture will discuss performing unsteady flow modeling with HEC-RAS for dam failure analysis.

Topics

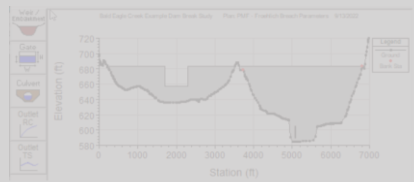
Boundary Conditions and Computation	
Inline Structures and Gates	
1D Reservoir Modeling Layout Options	

Topics

Boundary Conditions and Computation



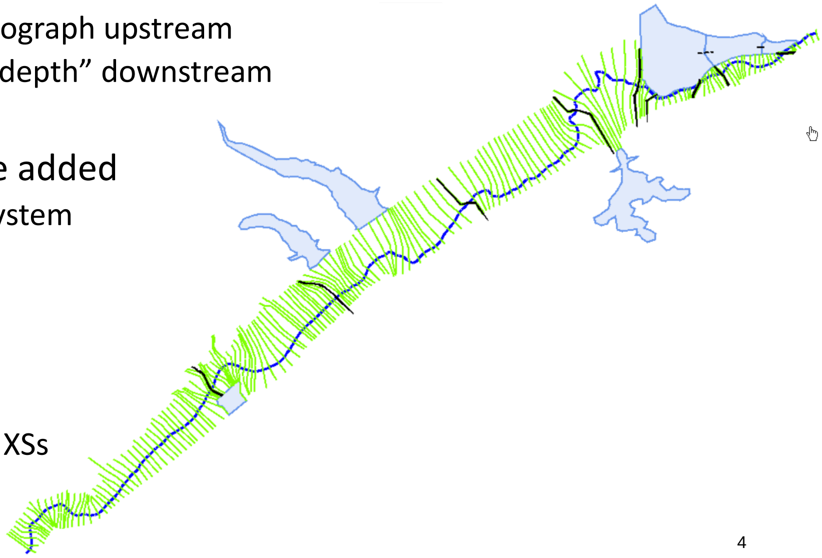
Inline Structures and Gates



1D Reservoir Modeling Layout Options

Unsteady Flow Data

- External Boundaries required
  - Upstream and Downstream ends of the river
  - Typically flow or stage hydrograph upstream
  - Typically rating or “normal depth” downstream
- Internal Boundaries can be added
  - Add flow within the river system
  - Define gate operation
- Initial Conditions
  - Steady Flow at XSs
  - Initialize SAs and Reservoir XSs

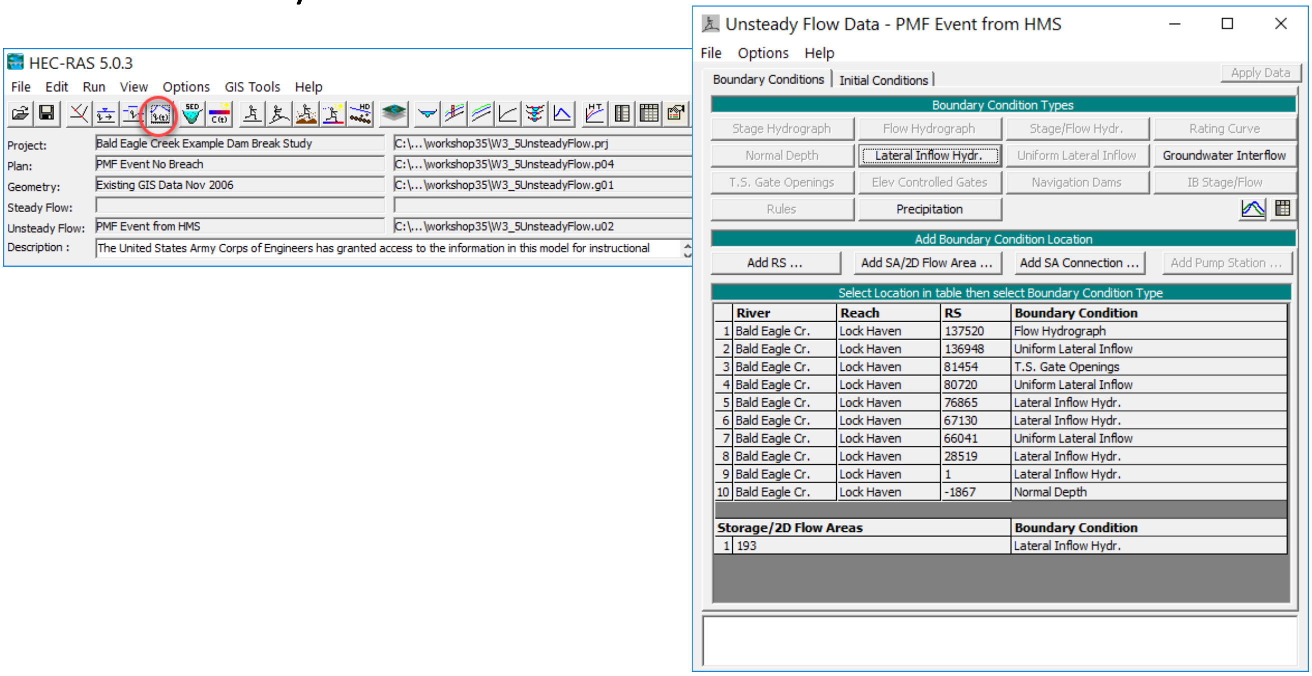


4

Boundary conditions are entered by first selecting the **Boundary Conditions** tab from the Unsteady Flow Data editor. River, Reach, and River Station locations of the external bounds of the system will automatically be entered into the table. Boundary conditions are entered by first selecting a cell in the table for a particular location, then selecting the boundary condition type that is desired at that location. Not all boundary condition types are available for use at all locations. The program will automatically gray-out the boundary condition types that are not relevant when the user highlights a particular location in the table. Users can also add locations for entering internal boundary conditions. To add an additional boundary condition location, select the desired River, Reach, and River Station, then press the **Add a Boundary Condition Location** button.

In addition to the boundary conditions, the user must establish the initial conditions of the system at the beginning of the unsteady flow simulation. Initial conditions consist of flow and stage information at each of the cross sections, as well as elevations for any storage areas defined in the system. Initial conditions are established from within the Unsteady Flow Data editor by selecting the Initial Conditions tab.

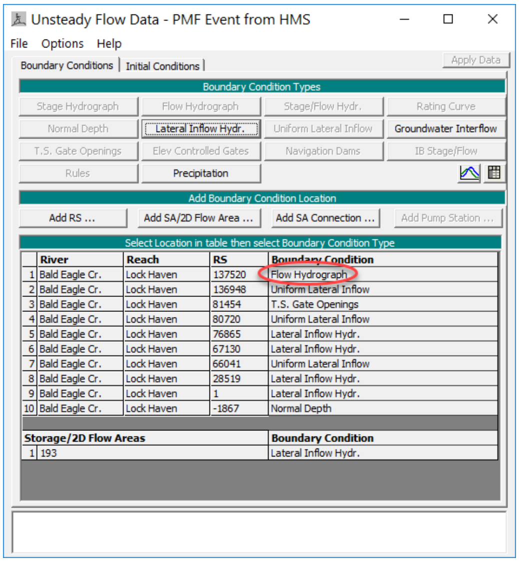
Unsteady Flow Data Editor

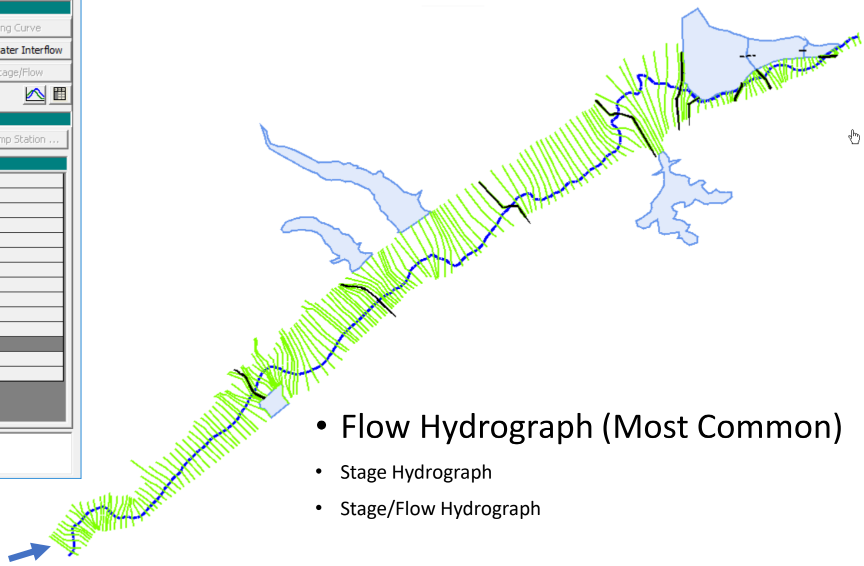


Press the Unsteady Flow button or choose Unsteady Flow under the Edit menu to get the unsteady flow editor.

Upstream and downstream boundary conditions are required – internal boundary conditions are optional.

Upstream Boundary Conditions





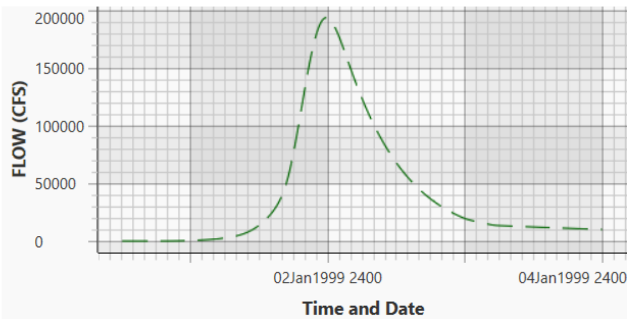
- Flow Hydrograph (Most Common)
  - Stage Hydrograph
  - Stage/Flow Hydrograph

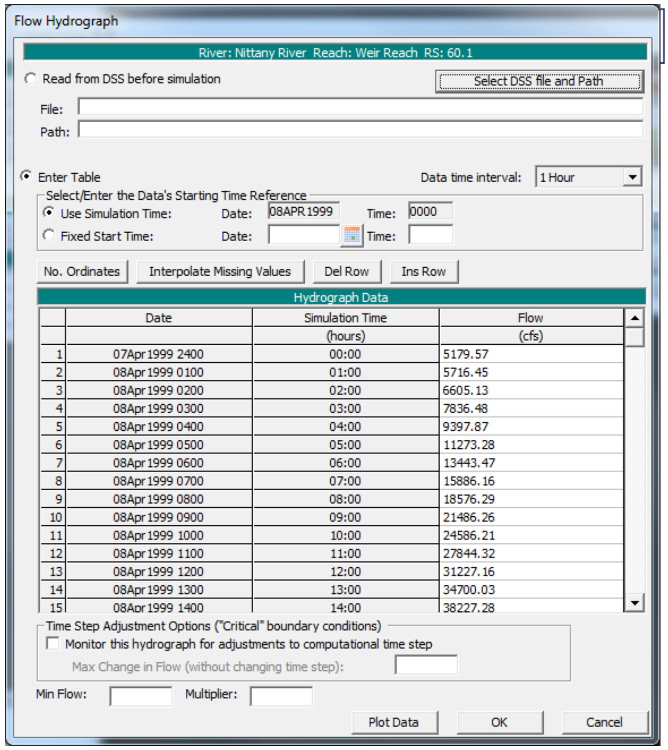
Most commonly used Upstream Boundary condition is the Flow Hydrograph boundary. Stage boundary allows the user to define a time series of stage values for the upstream boundary typically if you had observed stage data.

Stage/Flow Hydrograph is not a typical used – it is really only used in flood forecasting (only time) where you use stages for a known time window (hindcast) and then use flows to take over for the forecast period.

Flow Hydrograph

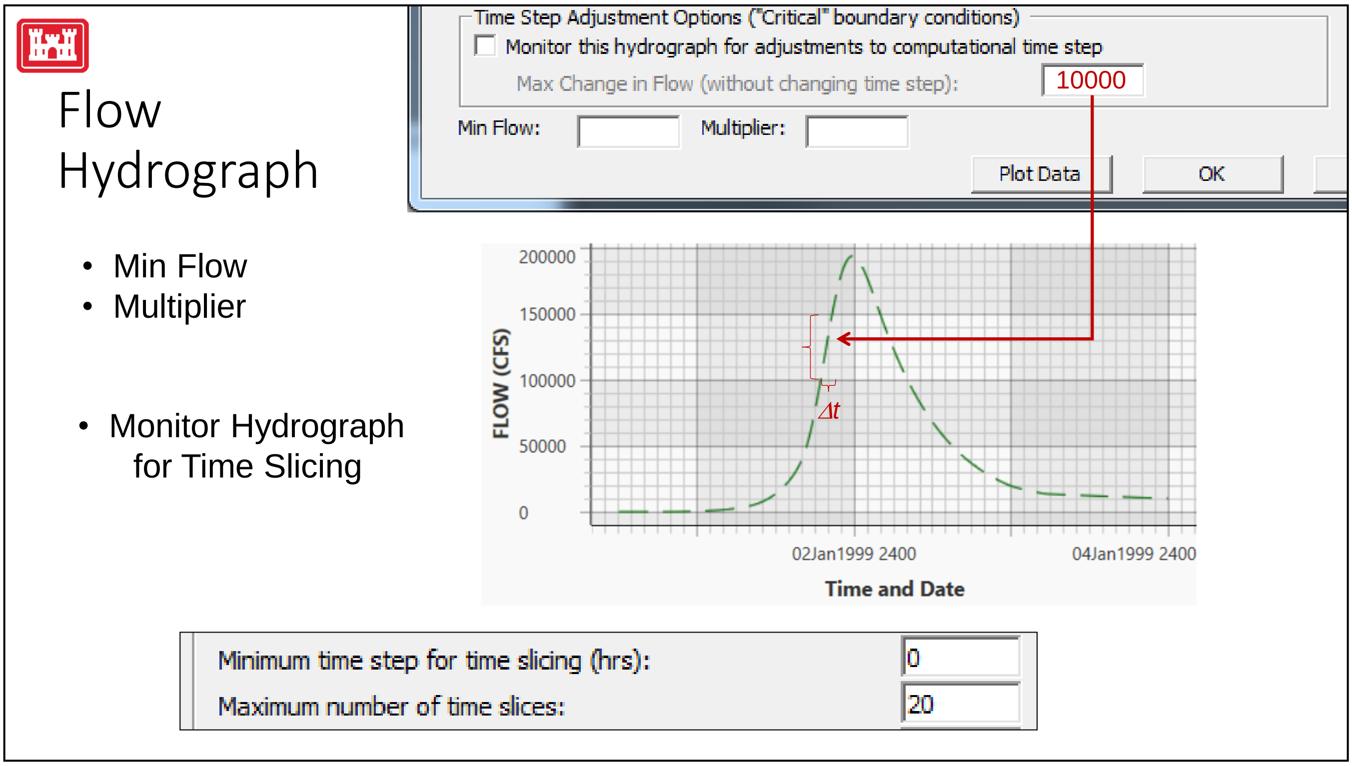
- Read from DSS
  - Select DSS file
  - Select Pathname
- Enter in Table
  - Select time interval
  - Select start date/time





When the flow hydrograph button is pressed, the window shown above will appear. As shown, the user can either read the data from a HEC-DSS (HEC Data Storage System) file, or they can enter the hydrograph ordinates into a table. If the user selects the option to read the data from DSS, they must press the "**Select DSS File and Path**" button. When this button is pressed a DSS file and pathname selection screen will appear. The user first selects the desired DSS file by using the browser button at the top. Once a DSS file is selected, a list of all of the DSS pathnames within that file will show up in the table.

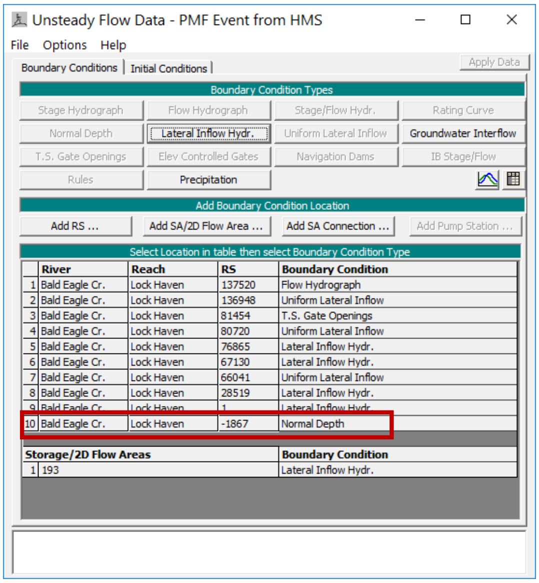
The standard Microsoft copy (Ctrl+C), cut (Ctrl+X) and paste (Ctrl+V) work in this editor if you want to bring data over from Excel. If you want to paste, however, you must first select the range of cells that you want to paste into (or a larger range of cells).

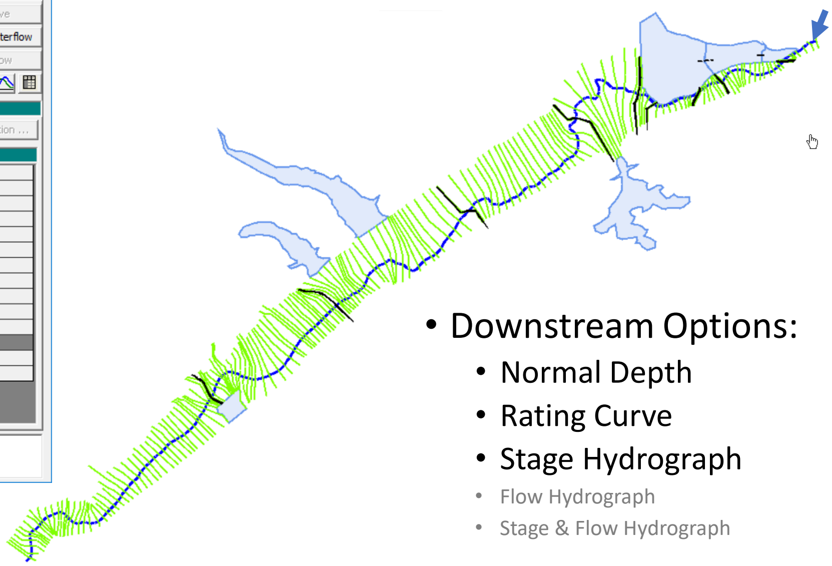


When the flow hydrograph button is pressed, the window shown above will appear. As shown, the user can either read the data from a HEC-DSS (HEC Data Storage System) file, or they can enter the hydrograph ordinates into a table. If the user selects the option to read the data from DSS, they must press the "**Select DSS File and Path**" button. When this button is pressed a DSS file and pathname selection screen will appear. The user first selects the desired DSS file by using the browser button at the top. Once a DSS file is selected, a list of all of the DSS pathnames within that file will show up in the table.

The standard Microsoft copy (Ctrl+C), cut (Ctrl+X) and paste (Ctrl+V) work in this editor if you want to bring data over from Excel. If you want to paste, however, you must first select the range of cells that you want to paste into (or a larger range of cells).

Downstream Boundary Conditions



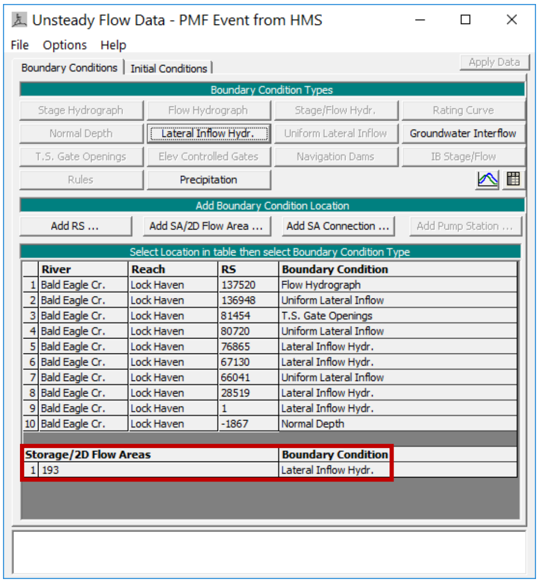


- Downstream Options:
  - Normal Depth
  - Rating Curve
  - Stage Hydrograph
  - Flow Hydrograph
  - Stage & Flow Hydrograph

Most commonly used Upstream Boundary condition is the Flow Hydrograph boundary. Stage boundary allows the user to define a time series of stage values for the upstream boundary typically if you had observed stage data.

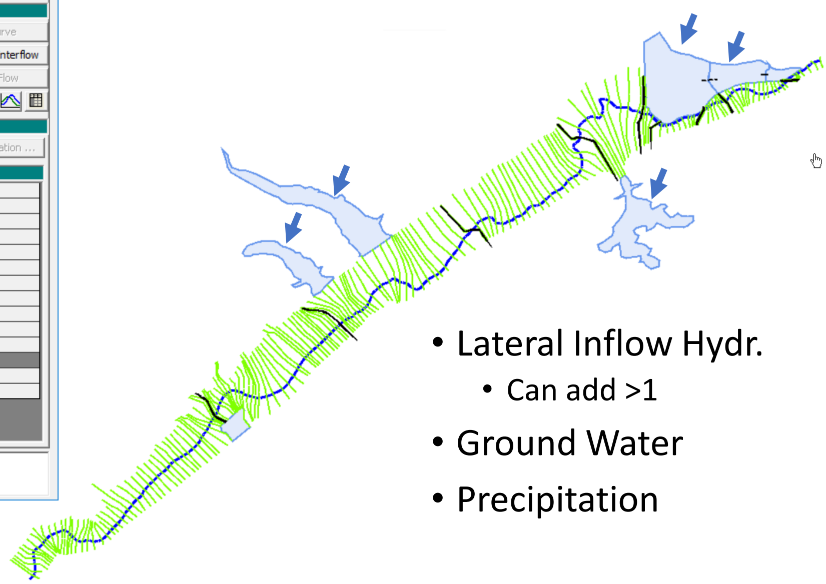
Stage/Flow Hydrograph is not a typical used – it is really only used in flood forecasting (only time) where you use stages for a known time window (hindcast) and then use flows to take over for the forecast period.

Storage Area Boundary Conditions



	River	Reach	RS	Boundary Condition
1	Bald Eagle Cr.	Lock Haven	137520	Flow Hydrograph
2	Bald Eagle Cr.	Lock Haven	136948	Uniform Lateral Inflow
3	Bald Eagle Cr.	Lock Haven	81454	T.S. Gate Openings
4	Bald Eagle Cr.	Lock Haven	80720	Uniform Lateral Inflow
5	Bald Eagle Cr.	Lock Haven	76865	Lateral Inflow Hydr.
6	Bald Eagle Cr.	Lock Haven	67130	Lateral Inflow Hydr.
7	Bald Eagle Cr.	Lock Haven	66041	Uniform Lateral Inflow
8	Bald Eagle Cr.	Lock Haven	28519	Lateral Inflow Hydr.
9	Bald Eagle Cr.	Lock Haven	1	Lateral Inflow Hydr.
10	Bald Eagle Cr.	Lock Haven	-1867	Normal Depth

	Storage/2D Flow Areas	Boundary Condition
1	193	Lateral Inflow Hydr.

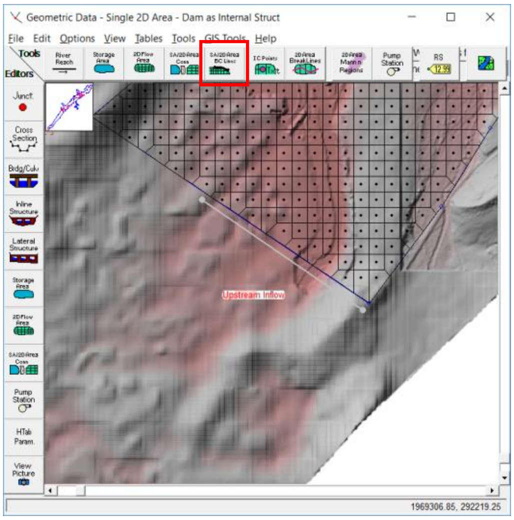


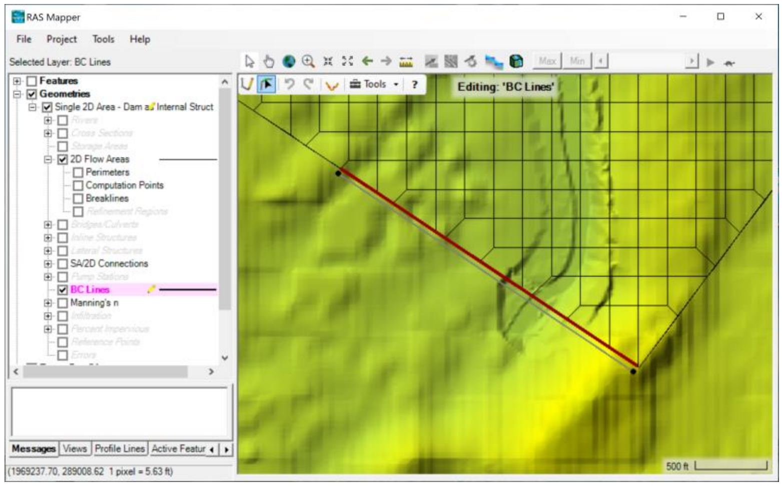
- Lateral Inflow Hydr.
  - Can add >1
- Ground Water
- Precipitation

Most commonly used Upstream Boundary condition is the Flow Hydrograph boundary. Stage boundary allows the user to define a time series of stage values for the upstream boundary typically if you had observed stage data.

Stage/Flow Hydrograph is not a typical used – it is really only used in flood forecasting (only time) where you use stages for a known time window (hindcast) and then use flows to take over for the forecast period.

2D External Boundary Conditions



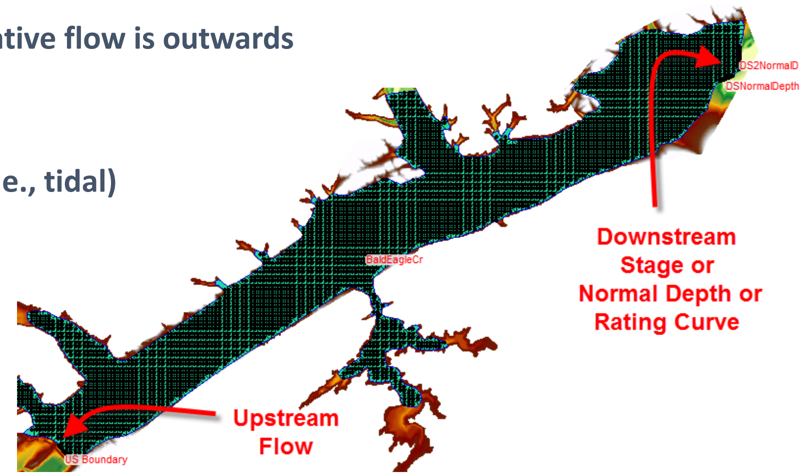


To create a 2D boundary:

1. If needed, zoom into the approximate location on the geometry editor.
2. Click on the **SA/2D Area BC** Lines button and then using the mouse/pen draw the location of the boundary on the outside of the 2D area in the appropriate location. Double click when finished and then enter an appropriate name when prompted by RAS.
3. The boundary condition will then appear in the Unsteady Flow Data editor. Click on the name of the boundary condition (to make that row active) and then select the desired boundary condition type (for instance Flow Hydrograph) by clicking on one of the boundary condition buttons.
4. Do not try to edit the boundary conditions from both the Geometric Editor and RAS Mapper. Please use RAS Mapper.

2D External Boundary Conditions

- Flow Hydrograph
  - **Usually for inflow (upstream/lateral)**
  - **Can also be used for outflow**
  - **Positive flow is inwards; Negative flow is outwards**
- Stage Hydrograph
  - **Usually for outflow**
  - **Can also be used for inflow (i.e., tidal)**
- Normal Depth (outflow only)
- Rating Curve (outflow only)

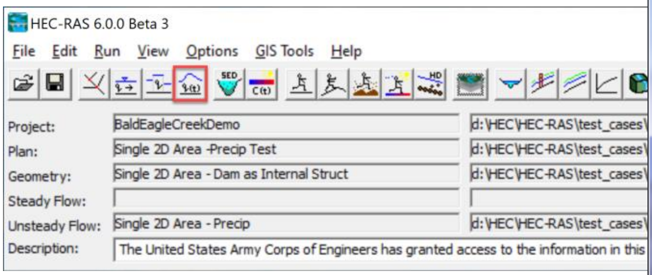


There are 4 boundary types:

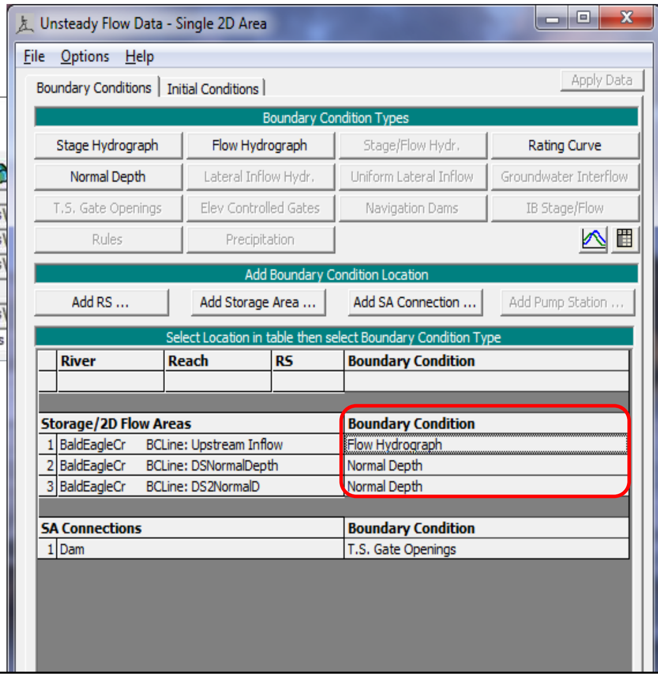
1. Flow hydrograph
2. Stage hydrograph
3. Normal depth
4. Rating curve.

- A flow boundary usually provides inflow to a 2D area, but it can also be used as an outflow by entering negative flows on the flow editor.
- A flow boundary can be entirely inside of a 2D area. Note: a flow boundary must be entirely external or entirely inside, it may not cross over the 2D bounding polygon.
- A stage boundary is typically used as an outflow/downstream boundary, but it can also be used for inflow (for instance as a tidal boundary where the flow changes direction).
- Normal depth and rating curve can only be used as outflow.

2D Boundary Conditions

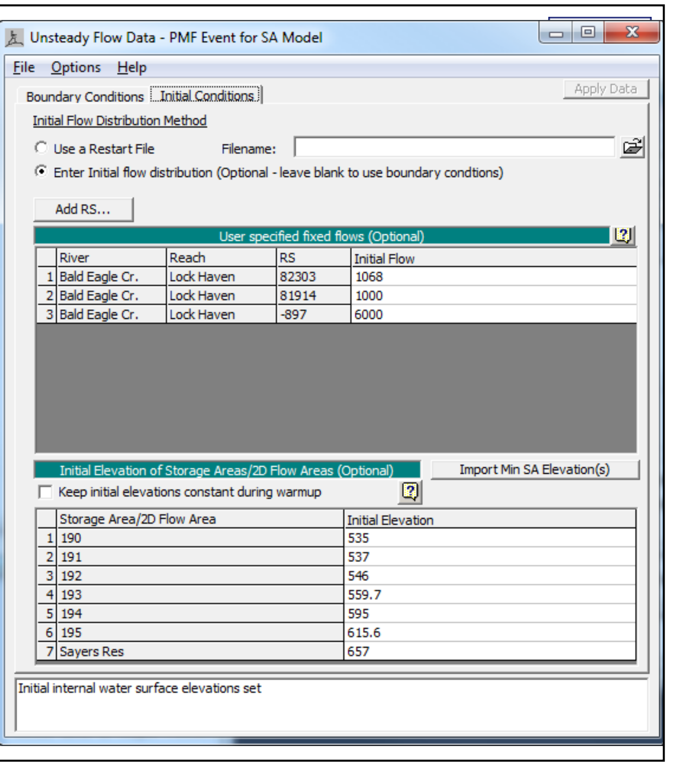


- Open Unsteady Flow Data editor
- Required BC's appear automatically



1D Initial Conditions

- Requires an initial flow for all reaches – can be left blank for dendritic systems
- Pool elevation for storage areas can be left blank
- Can change initial flow at any location
- Use system status from previous simulation (restart file)



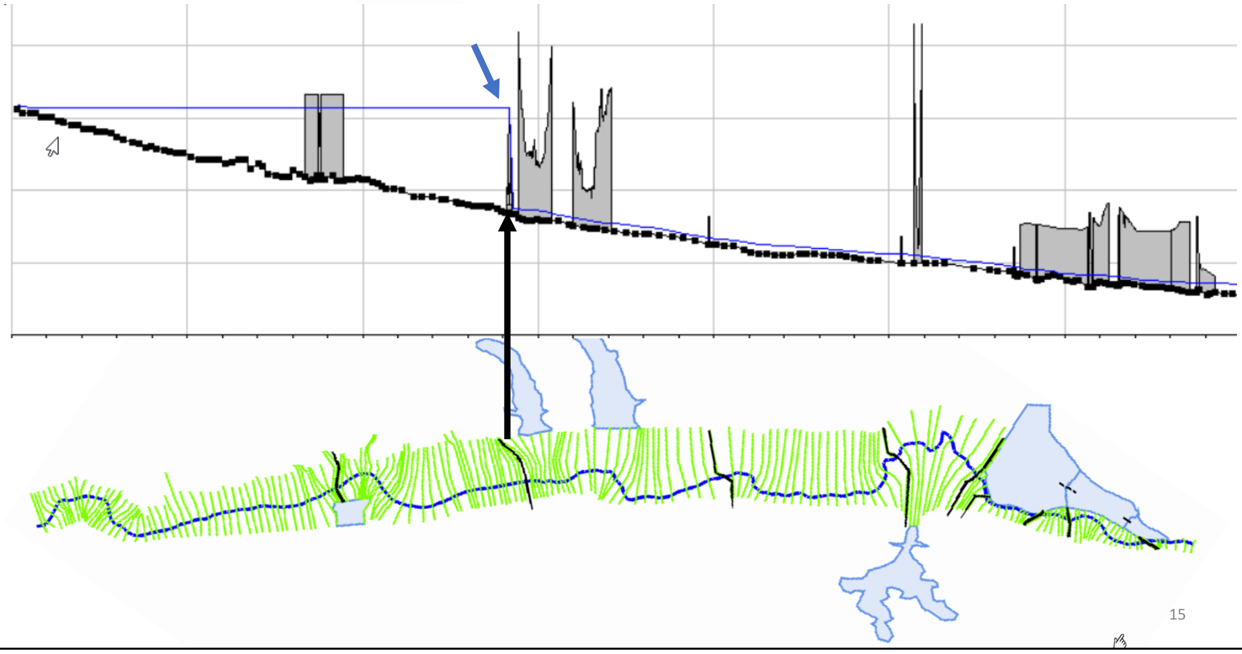
The differential equations used to compute the water surface elevation at the first time step require a water surface at time zero to make that calculation. HEC-RAS estimates an initial water surface by computing a steady flow water surface using 'Initial Flows' entered in the **Initial Conditions** tab of **Unsteady Flow Data Editor**. For standard, dendritic systems, if these initial flows are left blank, HEC-RAS will compute the steady flow profile based on the initial flow records in each of the flow hydrograph boundary conditions.

There are at least three reasons why you would want to input Initial Flows manually:

1) The transition between the steady flow profile computed for t=0 and the unsteady flow profile computed for the first times step is sometimes a difficult one. Since different equations are being used, there can be a discontinuity between these two profiles that leads to an instability. Specifying initial conditions allows you to 'tweak' them to increase the stability of this transition.
2) There are certain hydraulic configurations (e.g. looped networks) that RAS will not compute these for.
3) Models with regulation dams or routing dams will require specified Initial Flows because it will not get the gate flows right.

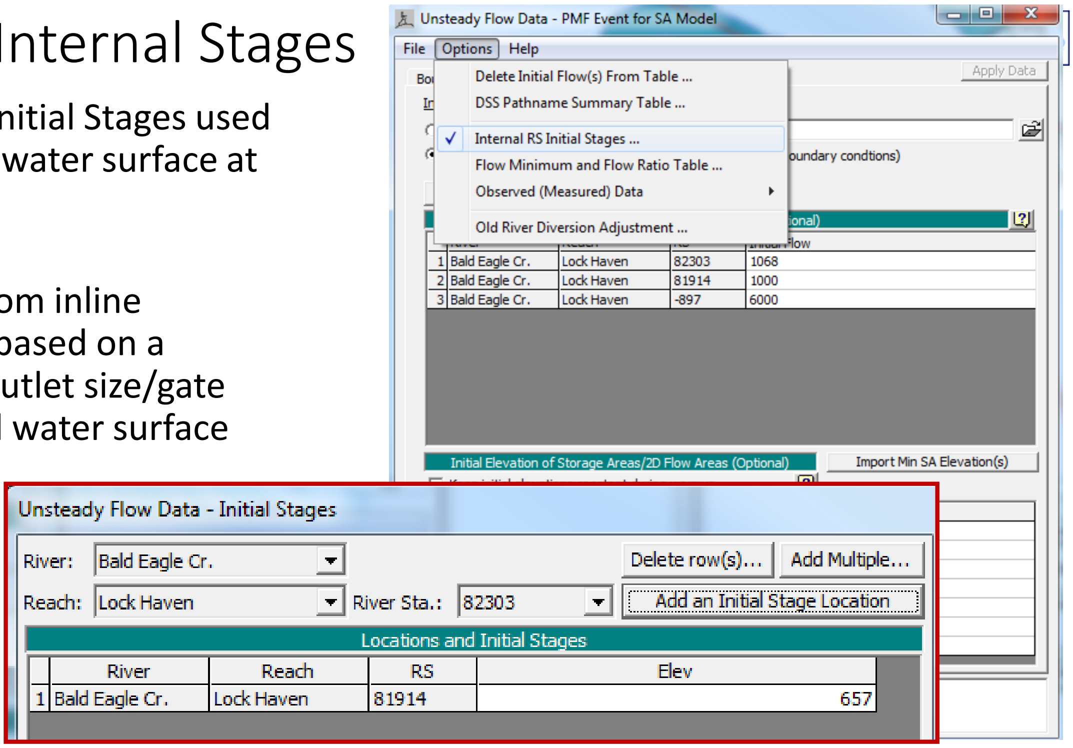
Secondly, you MUST specify the initial elevation of any storage areas in your system. If you are using a storage area as your reservoir (which we will talk about later) this is where you will specify your pool elevation.

Initial Reservoir Pool



Initial Internal Stages

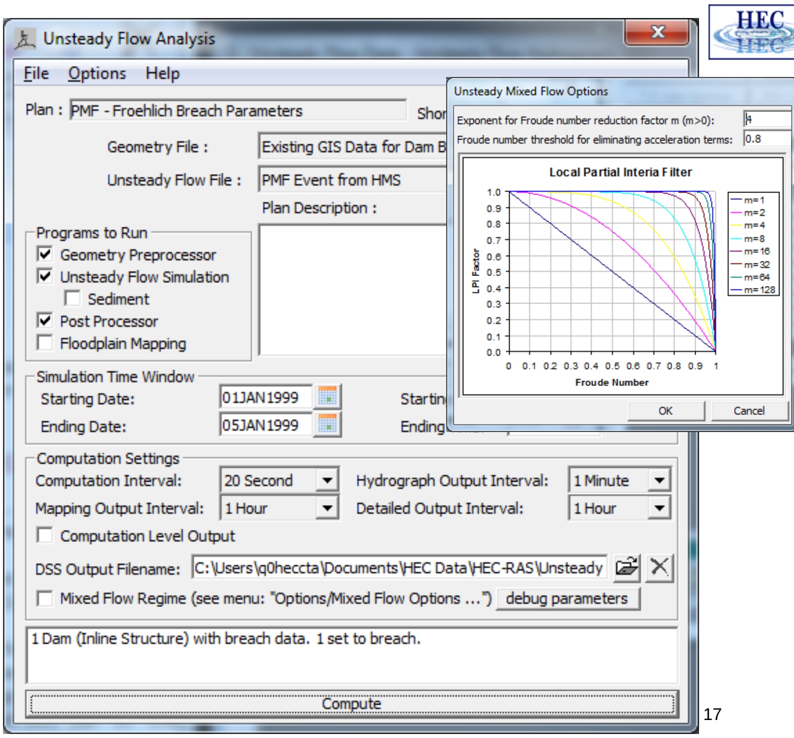
- Internal RS Initial Stages used to set initial water surface at a XS
- Stage U/S from inline structure is based on a balance of outlet size/gate opening and water surface



Internal RS Initial Stages. This option allows the user to specify starting water surface elevations for any internal cross section within the system. A common application of this would be to specify the starting pool elevation for the first cross section upstream of a dam (modeled with the inline weir/spillway option). The user specifies locations and water surface elevations, which are then used to establish the initial conditions of the system at the beginning of a run.

Computation

- Computation Time Step
- Hydrograph Output
- Detailed Output
- Mixed Flow Regime



17

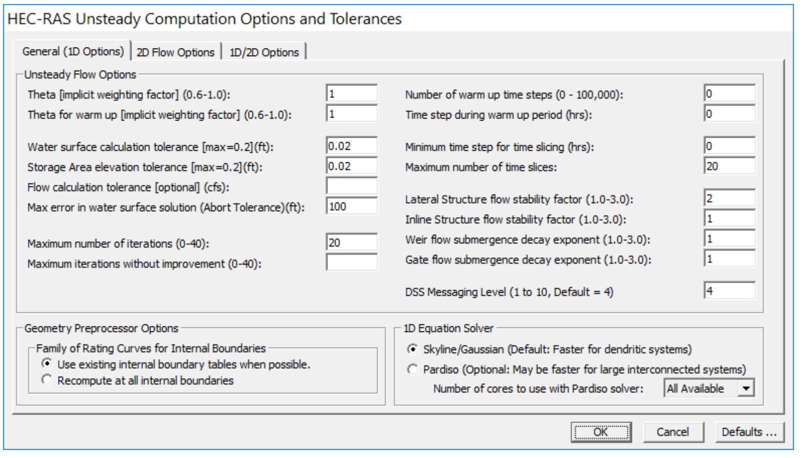
Once all of the geometry and unsteady flow data have been entered, the user can begin performing the unsteady flow calculations. To run the simulation, go to the HEC-RAS main window and select **Unsteady Flow Analysis** from the **Run** menu. The Unsteady Flow Analysis window will appear as depicted above.

Max WS is based on the Computation Interval

Mixed flow option should only be used for very rivers – it is not a panacea for troubled models!

Computation Options

- Theta
  1 = Most Stable
- Water Surface Tolerance
- Stability Factors
  1 = Most Accurate
  3 = Most Stable
  - Lateral Structure
  - Inline Structure
  - Weir Flow Submergence



18

Theta implicit weighting factor: This factor is used in the finite difference solution of the unsteady flow equations. The factor ranges between 0.6 and 1.0. A value of 0.6 will give the most accurate solution of the equations, but is more susceptible to instabilities. A value of 1.0 provides the most stability in the solution, but may not be as accurate for some data sets. The default value is set to 1.0. Once the user has the model up and running the way they want it, they should then experiment with changing theta towards a value of 0.6. If the model remains stable, then a value of 0.6 should be used. In many cases, you may not see an appreciable difference in the results when changing theta from 1.0 to 0.6. However, every simulation is different, so you must experiment with your model to find the most appropriate value.

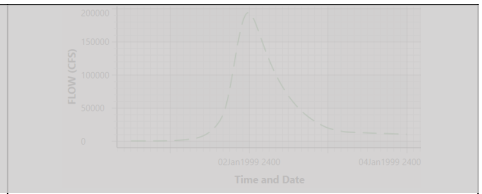
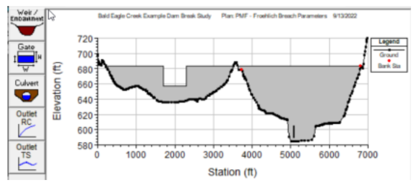
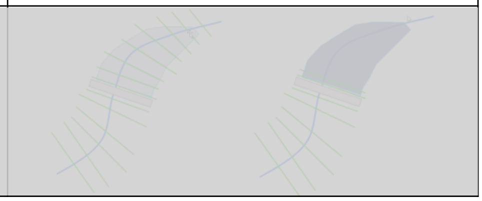
Water surface calculation tolerance: This tolerance is used to compare the difference between the computed and assumed water surface elevations at cross sections. If the difference is greater than the tolerance, the program continues to iterate for the current time step. When the difference is less than the tolerance, the program assumes that it has a valid numerical solution. The default value is set to 0.02 feet.

Storage area elevation tolerance: This tolerance is used to compare the difference between computed and assumed water surface elevations at storage areas. If the difference is greater than the tolerance, the program continues to iterate for the current time step. When the difference is less than the tolerance, the program can go on to the next time step. The default tolerance for storage areas is set to 0.05 feet.

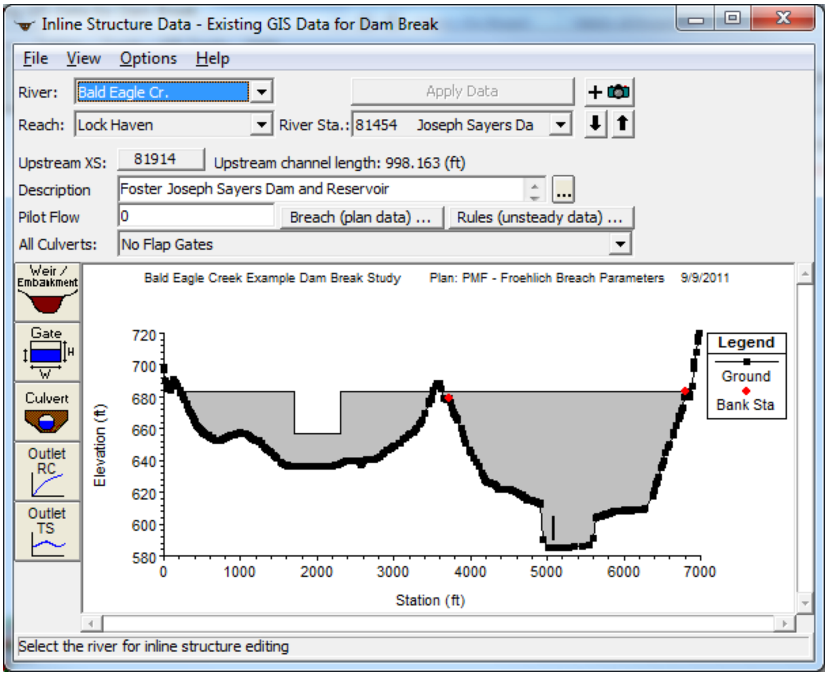
Flow calculation tolerance: This tolerance is used to compare against the numerical error in the computed flow versus the assumed flow for each iteration of the unsteady flow equations. The user enters a flow in cfs (or cms in metric data sets). The software monitors the flow error at all computational nodes. If the flow error is greater than the user entered tolerance, then the program will continue to iterate. By default, this option is not used, and is therefore only used if the user enters a value for the tolerance.

Weir flow stability factor: This factor is used to increase the stability of the numerical solution in and around a weir. This factor varies from 1.0 to 3.0. As the value is increased, the solution is more stable but less accurate. A value of 1.0 is the most accurate, but is susceptible to oscillations in the computed weir flow. The default value is 1.0. If you observe oscillations in the computed flow over the weir, you should first check to see if you are using a small enough computation interval. If the computation interval is sufficiently small, you should then try increasing this coefficient to see if it solves the problem.

Topics

Boundary Conditions and Computation	
Inline Structures and Gates	
1D Reservoir Modeling Layout Options	

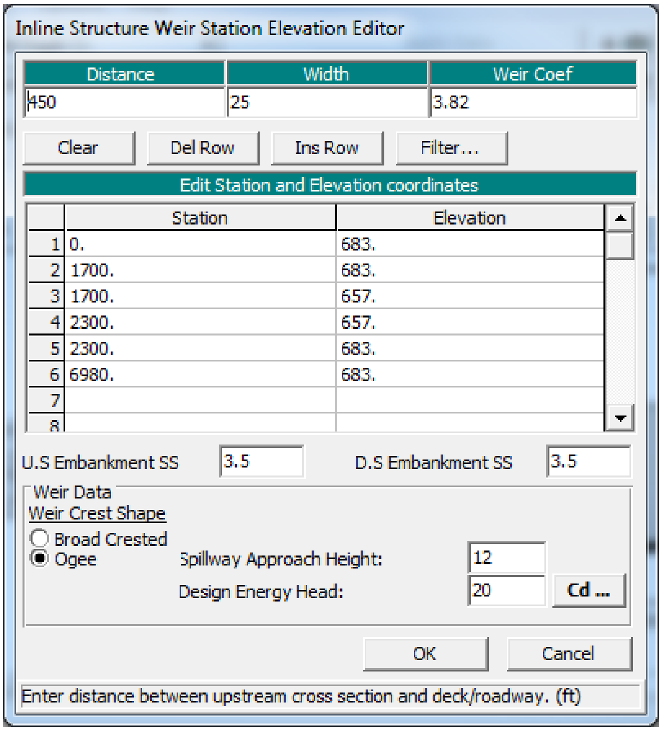
Entering Inline Structure Data



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

Weir and Embankment Profile

- Distance + Width < U/S XS Reach Length
- Weir include top of dam and spillway
- Weir Coef. used for both dam and spillway



Inline structure data are entered in a similar manner as bridge and culvert data. To enter an inline structure press the Inline Structure button from the Geometric Data window. Once this button is pressed, the Inline Structure Data editor will appear as shown above.

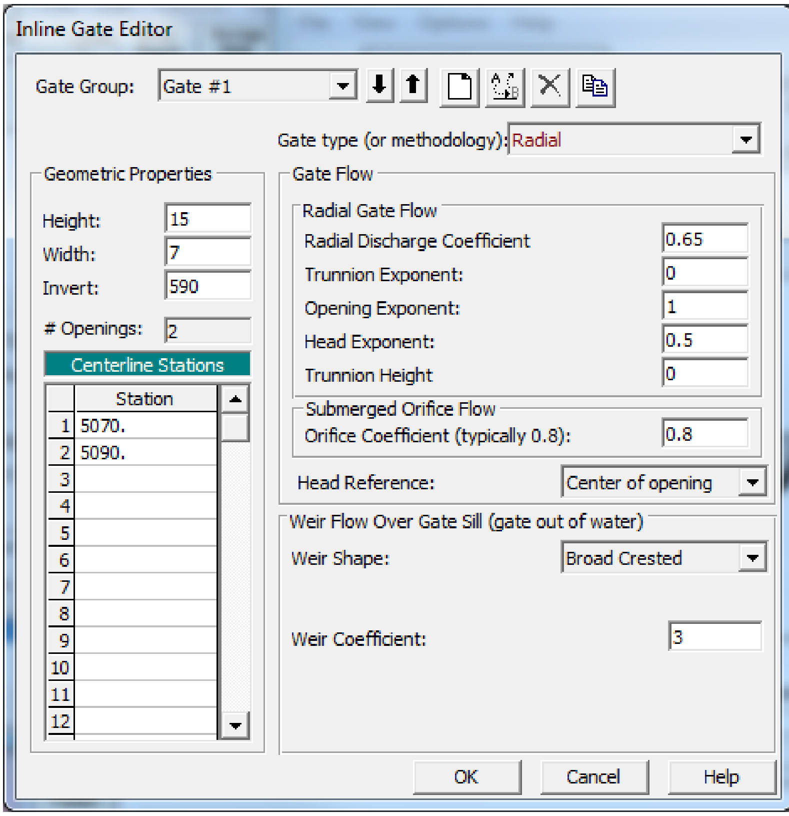
Distance - The distance field is used to enter the distance between the upstream side of the Weir/Embankment (the top of the embankment) and the cross section immediately upstream of the structure. This distance is entered in feet (or meters for metric).

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

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

Gates

- Sluice
- Radial
- Overflow
- User Defined Curves

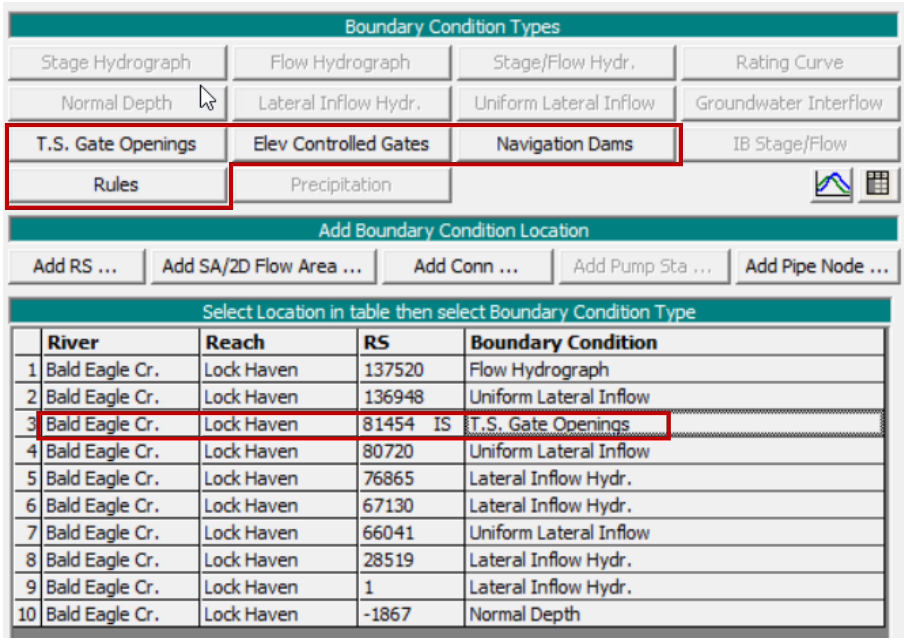


In addition to uncontrolled overflow weirs, the user can add gated spillways (this is optional). To add gated spillways to the structure, press the **Gate** button on the Inline Structure data editor. Once this button is pressed, the gated editor will appear as shown above.

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

Gate Settings

- The only mandatory Internal Boundary Condition
- Time Series
- Elev Controlled
  - Simple Rules
- Rules



Time Series of Gate Openings:

This option allows the user to enter a time series of gate openings for an inline gated spillway, lateral gated spillway, or a gated spillway

connecting two storage areas. The user has the option of reading the data from a DSS file or entering the data into a table from within the

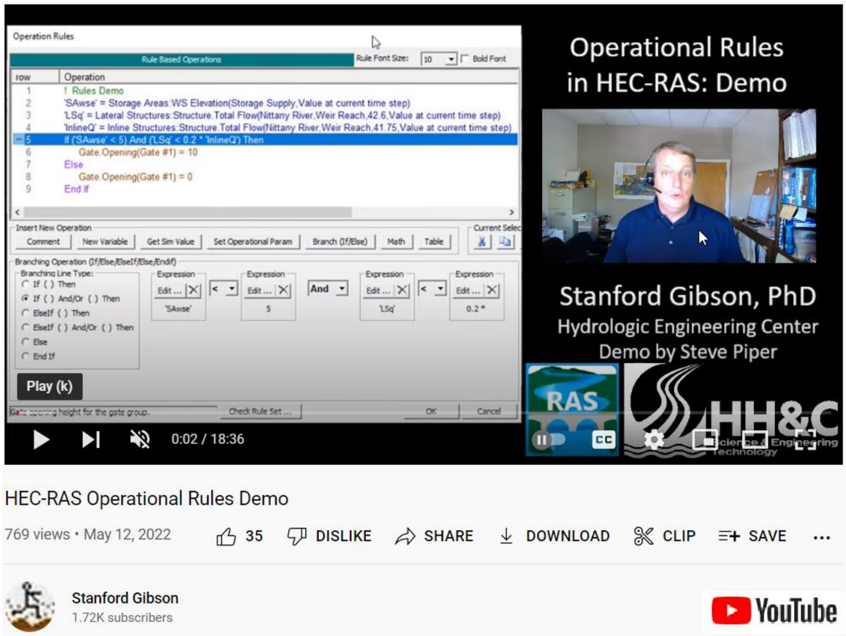
editor. Figure 8-5 shows an example of the Times Series of Gate Openings editor. As shown in Figure 8-5, the user first selects a gate

group, then either attaches a DSS pathname to that group or enters the data into the table. This is done for each of the gate groups

contained within the particular hydraulic structure.

Some Thoughts On Rules

- Underrated
- More Powerful Than You Think
- Easier Than You Think

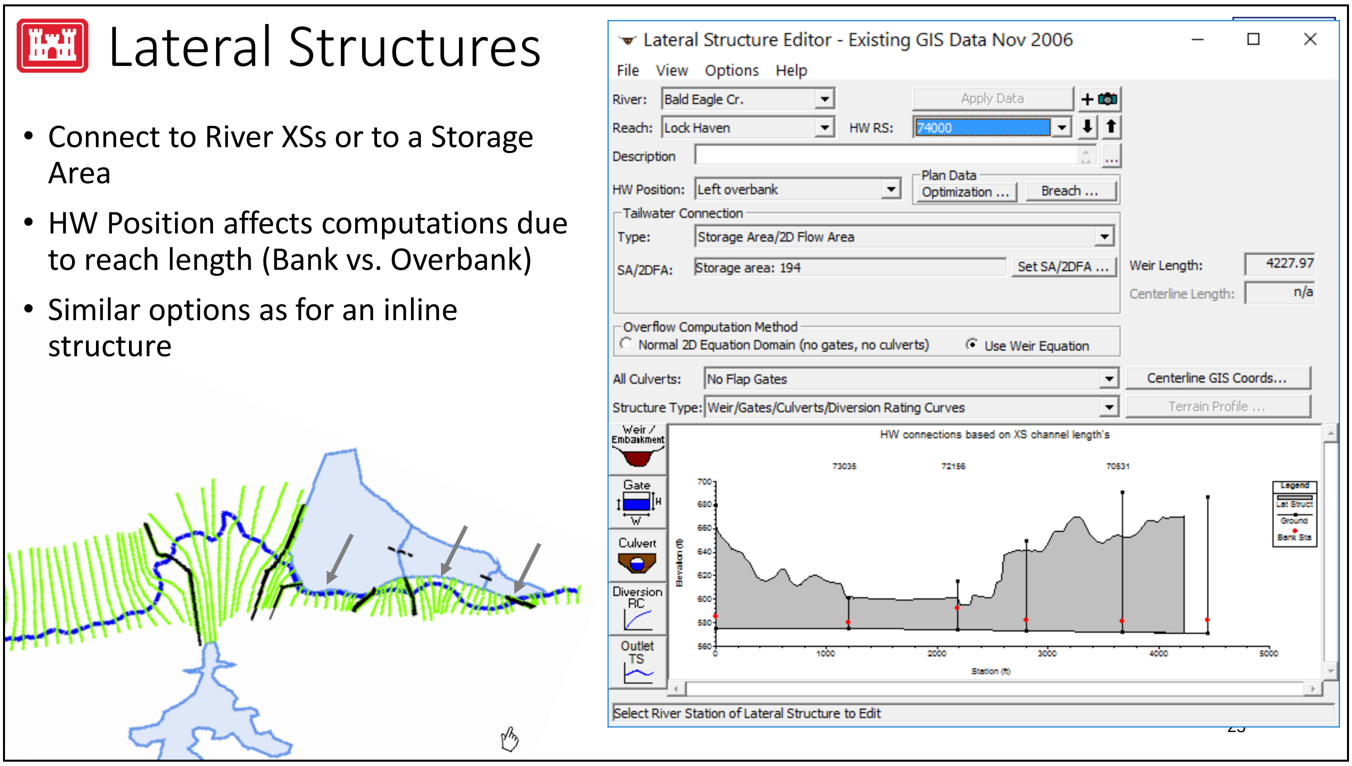


https://youtu.be/KMbV-cexP7w

Elevation Controlled Gate:

This option allows the user to control the opening and closing of gates based on the elevation of the water surface upstream of the structure; or based on the water surface at a user specified cross section or storage area (from any location in the model); or based on a

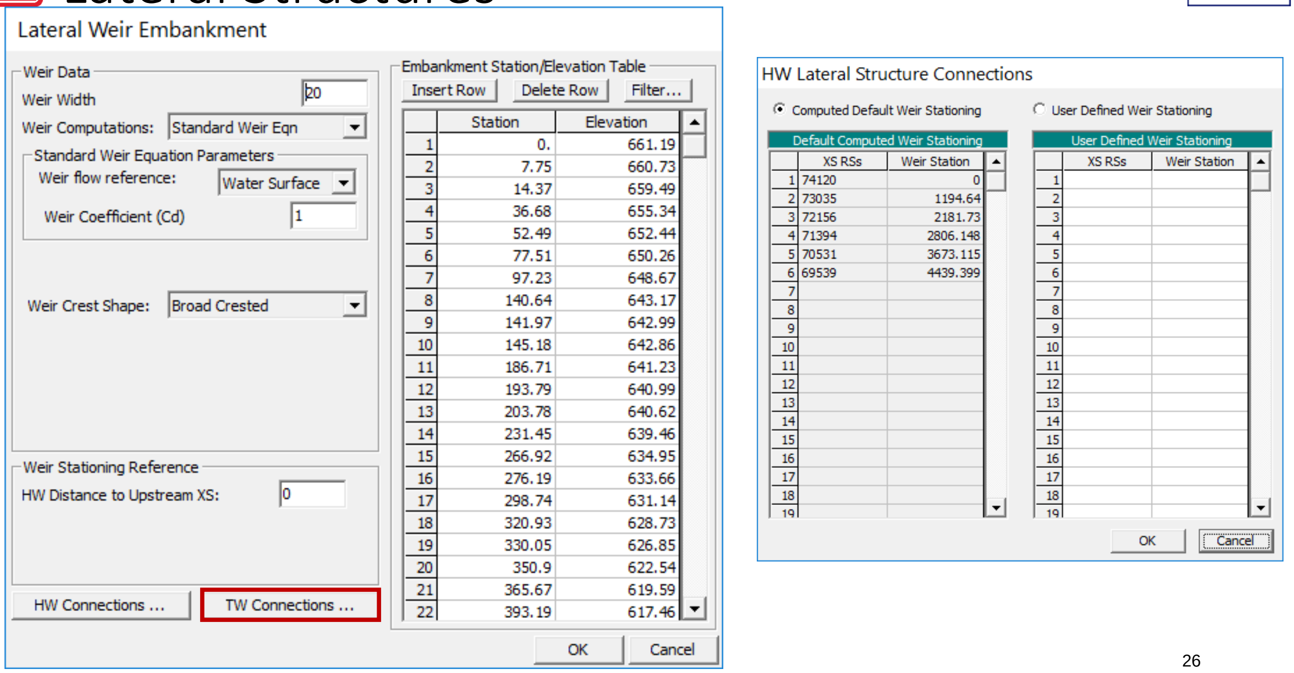
difference in water surface elevation from any two user defined reference locations. A gate begins to open when a user specified elevation is exceeded. The gate opens at a rate specified by the user. As the water surface goes down, the gate will begin to close at a user specified elevation. The closing of the gate is at a user specified rate (feet/min.). If the gate operating criteria is a stage difference, the user can specify a stage difference for when the gate should open and a stage difference for when the gate should begin to close. Stage differences can be positive, zero, or negative values. The user must also enter a maximum and minimum gate opening, as well as the initial gate opening.



The Lateral Structure in RAS is used to model high ground at the edge of cross sections (such as a levee) and has the same features as an inline structure. However, a lateral structure is connected either to cross sections on another river reach or to a storage area. The Headwater position indicates whether the structure is next to the river channel (Banks) or at the edge of the floodplain (Overbank). This is important, because RAS will use either the Channel Reach Lengths (Banks) or the Overbank Reach Lengths (Overbanks) for computing the water surface for flow computations over the lateral structure.

Lateral structures can also be breached, similar to the inline structure.

Lateral Structures

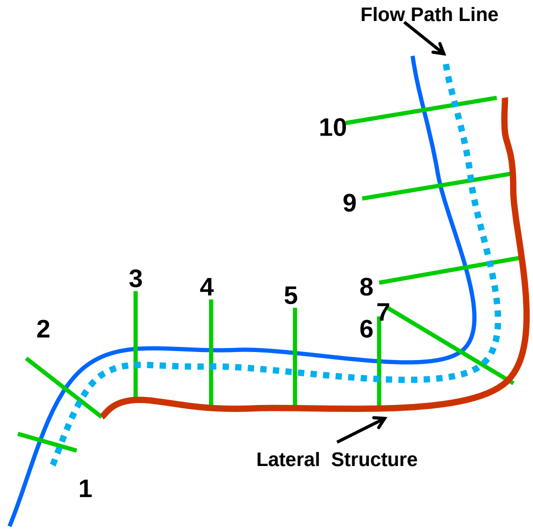


The weir embankment editor allows the user to specify whether to use the Water Surface or Energy Grade line for the weir flow computations. If the lateral structure is near the channel where velocities are high, the water surface should be used. If the lateral structure is over out at the end of a wide cross section on the outside of a river bend, the energy grade line should be used. The default weir coefficient is set to 2 to indicate most lateral structures may not be very efficient. If the lateral structure is used to connect a storage area for a tributary, a lower value of 0.5 or 1.0 is appropriate.

Lateral Structure

- User Specified Stationing

RAS XS	RAS Station	User Station
10	0	0
9	100	100
8	200	200
7	300	325
6	400	450
5	500	550
4	600	650
3	700	750
2	800	825
1	-	-

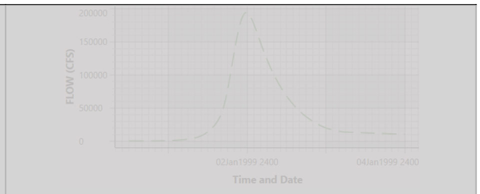
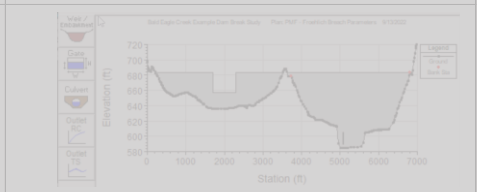
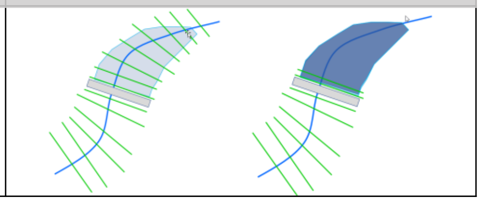


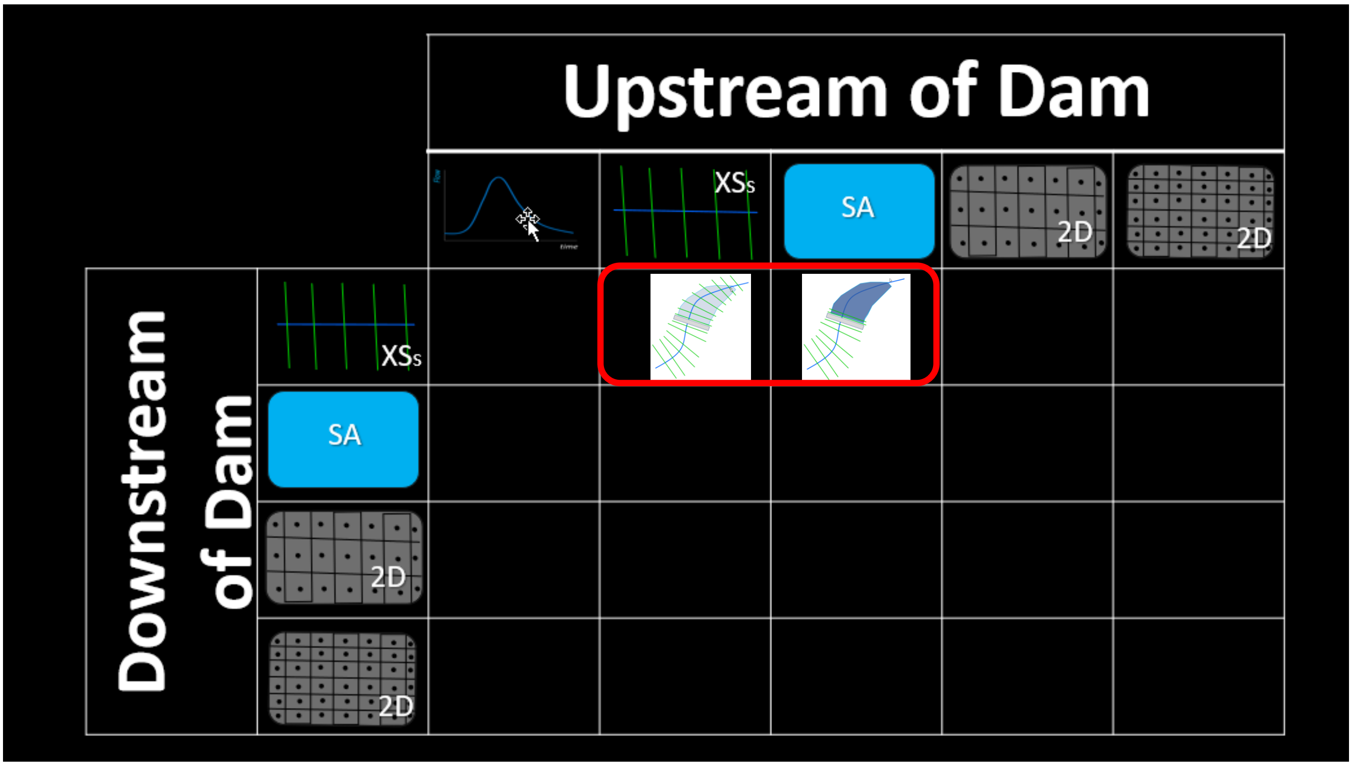
27

The default option in RAS is calculate the intersection of the lateral structure with the cross sections. This sometimes does not work correctly for the situation being modeled. In those cases, use the user specified stationing option. The user specified stationing option allows the user to specify the part of the lateral structure that intersects with the cross sections.

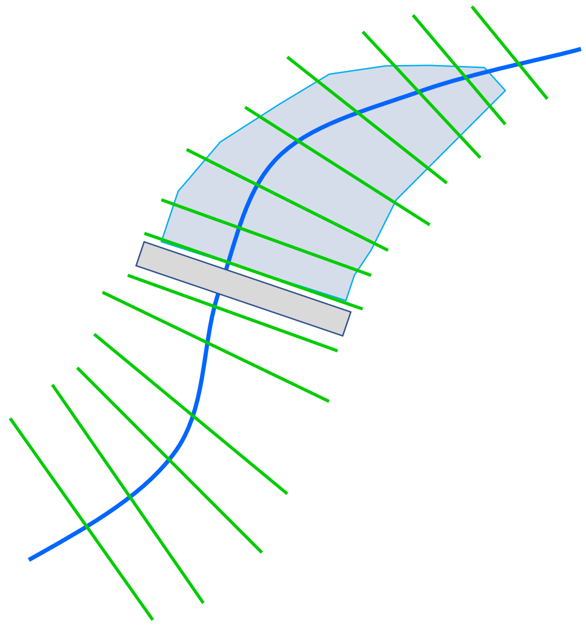
In the example above, each cross section has a left reach length of 100 units. The lateral structure on the left overbank is longer than 100 units on the outside of the bend from 8 to 7 and 7 to 6, and is shorter on the inside from 3 to 2.

Topics

Boundary Conditions and Computation	
Inline Structures and Gates	
1D Reservoir Modeling Layout Options	


Upstream of Dam
XSs
SA
2D
2D
Downstream of Dam
XSs
SA
2D
2D

1D Reservoir Modeling Option



- Model Reservoir with cross sections
- Cross sections must include channel information, especially around dam both u/s and d/s
- Allows for dynamic routing of water (sloped water surface)

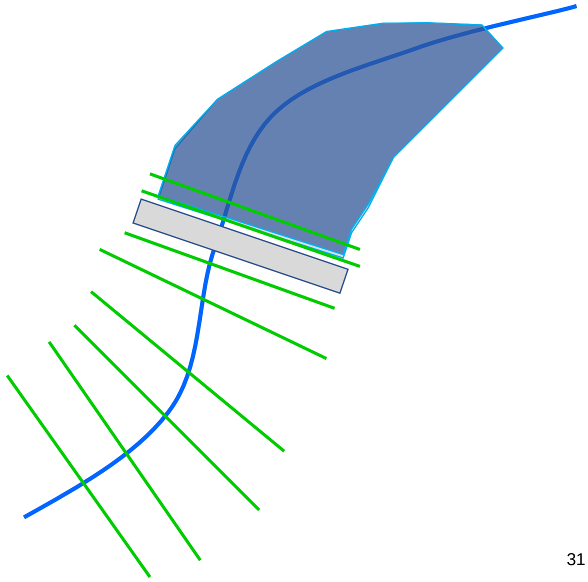
30

There are two options for modeling a reservoir in HEC-RAS. The most accurate way to model a reservoir is with an inline structure and cross sections up through the reservoir. The cross sections should include the full channel bathymetry.

Modeling the reservoir with cross sections allows for dynamic routing a of the flood hydrograph through the reservoir – allowing for a sloping water surface through the reservoir pool.

Reservoir Modeling Option

- Model Reservoir with a Storage Area
- Must have 2 cross sections U/S from inline structure
- Cross sections must include channel down to dam invert on both side of the inline structure
- Linear routing in storage area results in horizontal water surface

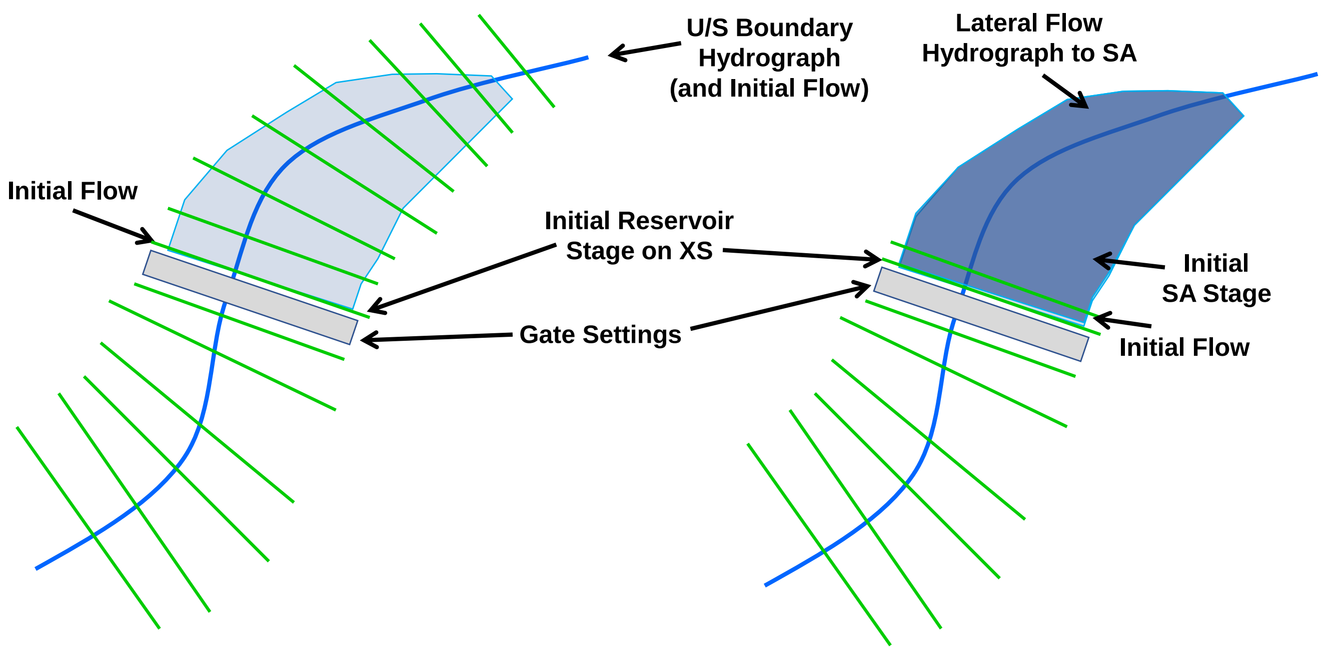


31

A second option in HEC-RAS for modeling a reservoir is to use a storage area. The storage area option requires a minimum of two cross sections upstream from the inline structure (one cross section as a boundary condition with the inline structure, the other cross section as the external boundary condition).

The cross section bounding the inline structure must go all the way down to the channel bottom of the reservoir (as low as the intended breach will go on the inline structure).

1D Reservoir Modeling Options



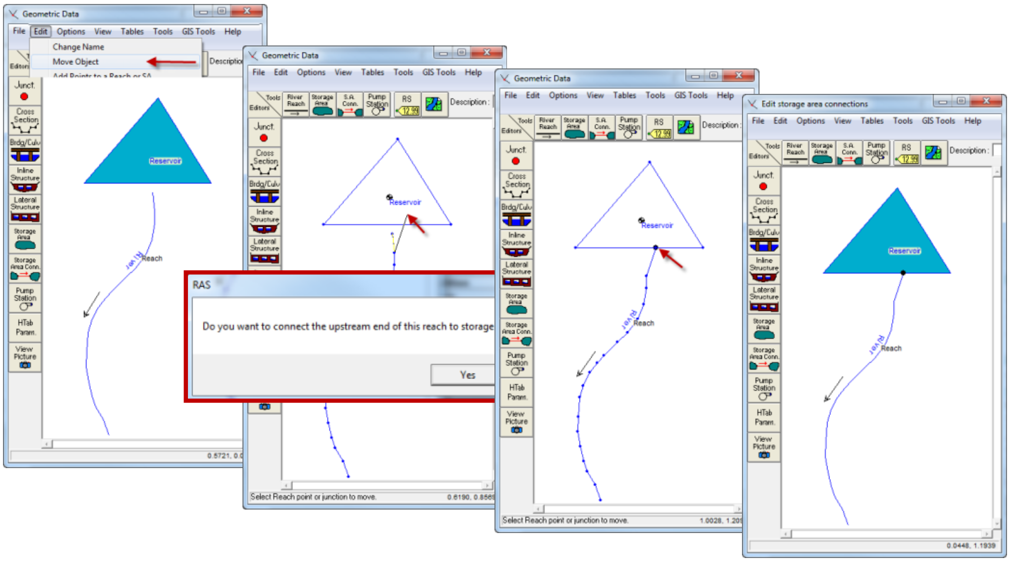
The different modeling approaches require similar boundary and initial conditions.

Both the XS Approach and the SA Approach require Gate Settings on the Inline Structure. It is also a good idea to set the Initial Stage on the XS just upstream from the Inline Structure to match what RAS will compute based on the Initial Gate opening.

For the SA Approach you also need to set the initial Reservoir Stage (which should match the Stage on the XS). Any flow into the reservoir is specified using the Lateral Inflow Hydrograph option. A storage area only allows one lateral inflow hydrograph, so all inflow must be combined.

For the XS Approach, you will need to specify the Inflow Hydrograph (or other) for the U/S Boundary Condition.

Connecting a Storage Area

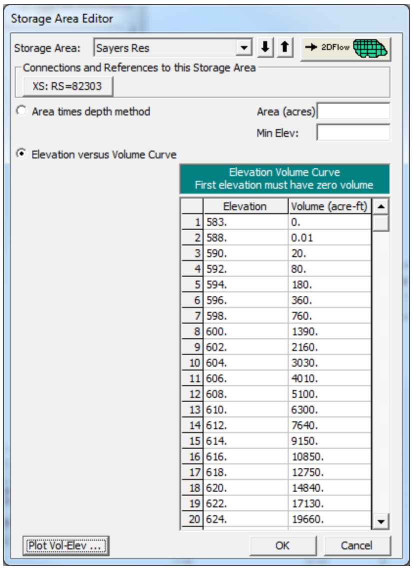


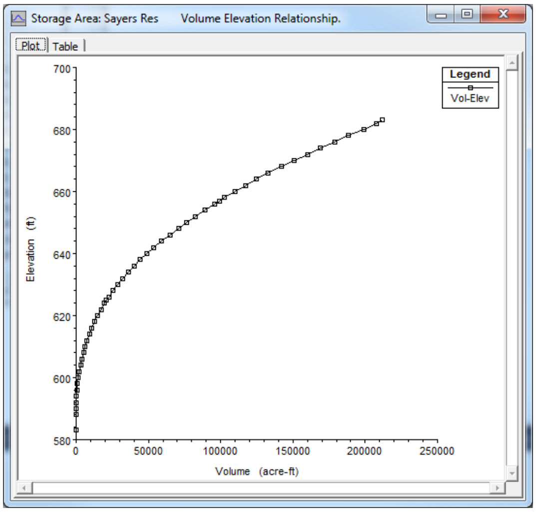
33

To utilize a storage area as an upstream boundary condition you must attach the upstream end of a reach to the storage area. To do this, Select **Move Object** from the **Edit** menu. This makes all of the nodes and labels availabe to move. Click on the upstream most stream centerline node and drag it into the outline of the storage area. You will be asked:

If you select **Yes** the storage area will become the upstream boundary condition for the reach.

Stage-Volume Curve





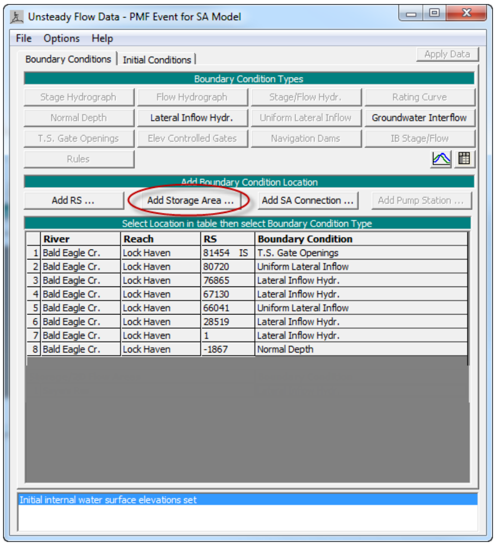
34

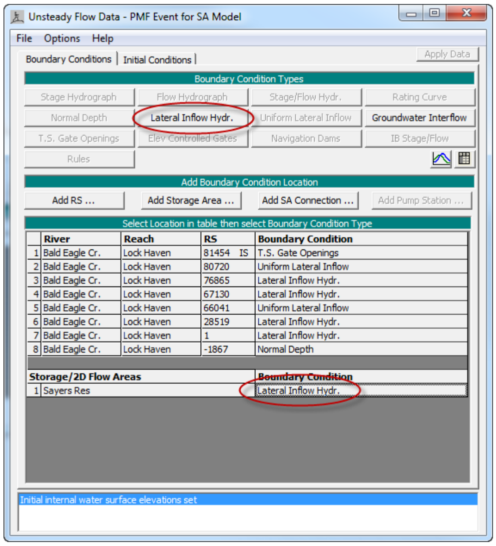
After the storage area is drawn and labeled, the user must enter data to describe the storage area. This is accomplished with the storage

area editor, which is one of the buttons on the left side of the geometric editor. Press the storage area editor button and the

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

Storage Area Inflow





35

We can now have multiple hydrographs connected to one storage area, you just need to add the storage area multiple times.