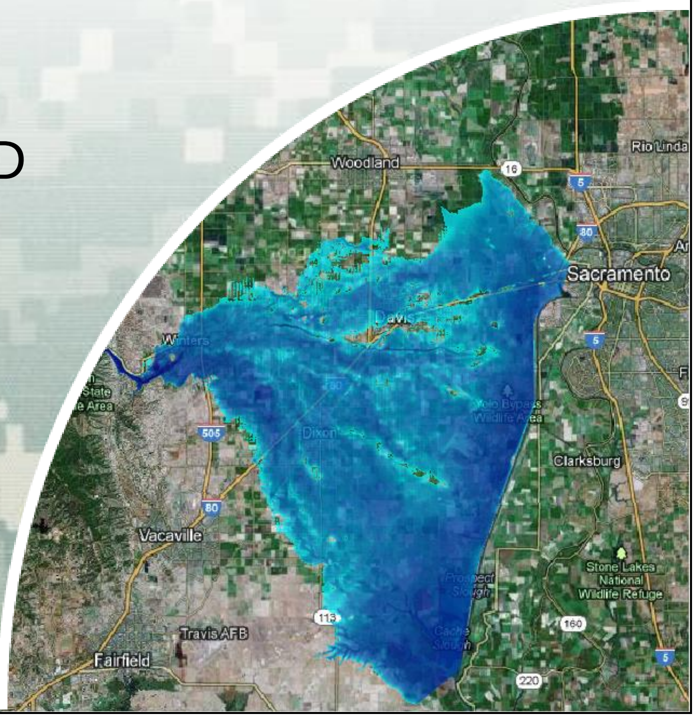
Dam Breaching Analysis with Combined 1D and 2D Elements

Stanford Gibson, PhD

Including slides by

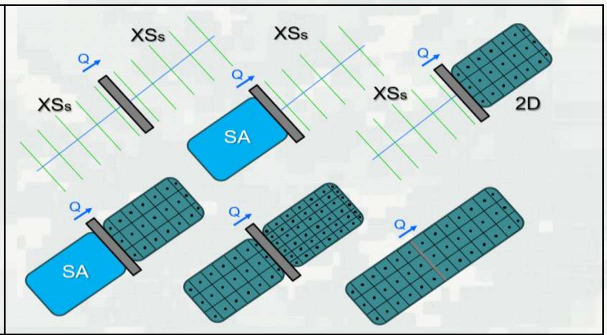
Gary Brunner, P.E.



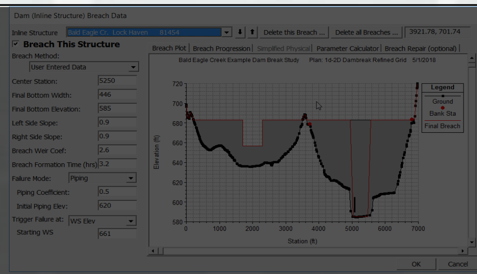
US Army Corps of Engineers
BUILDING STRONG®

1

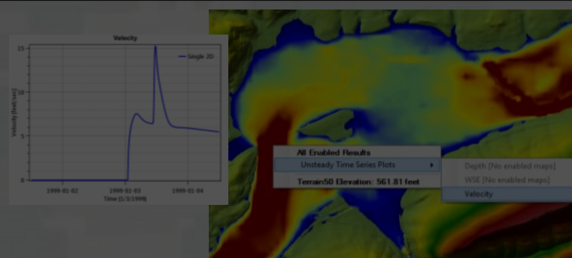
1. Six Dam Breach Model Configurations

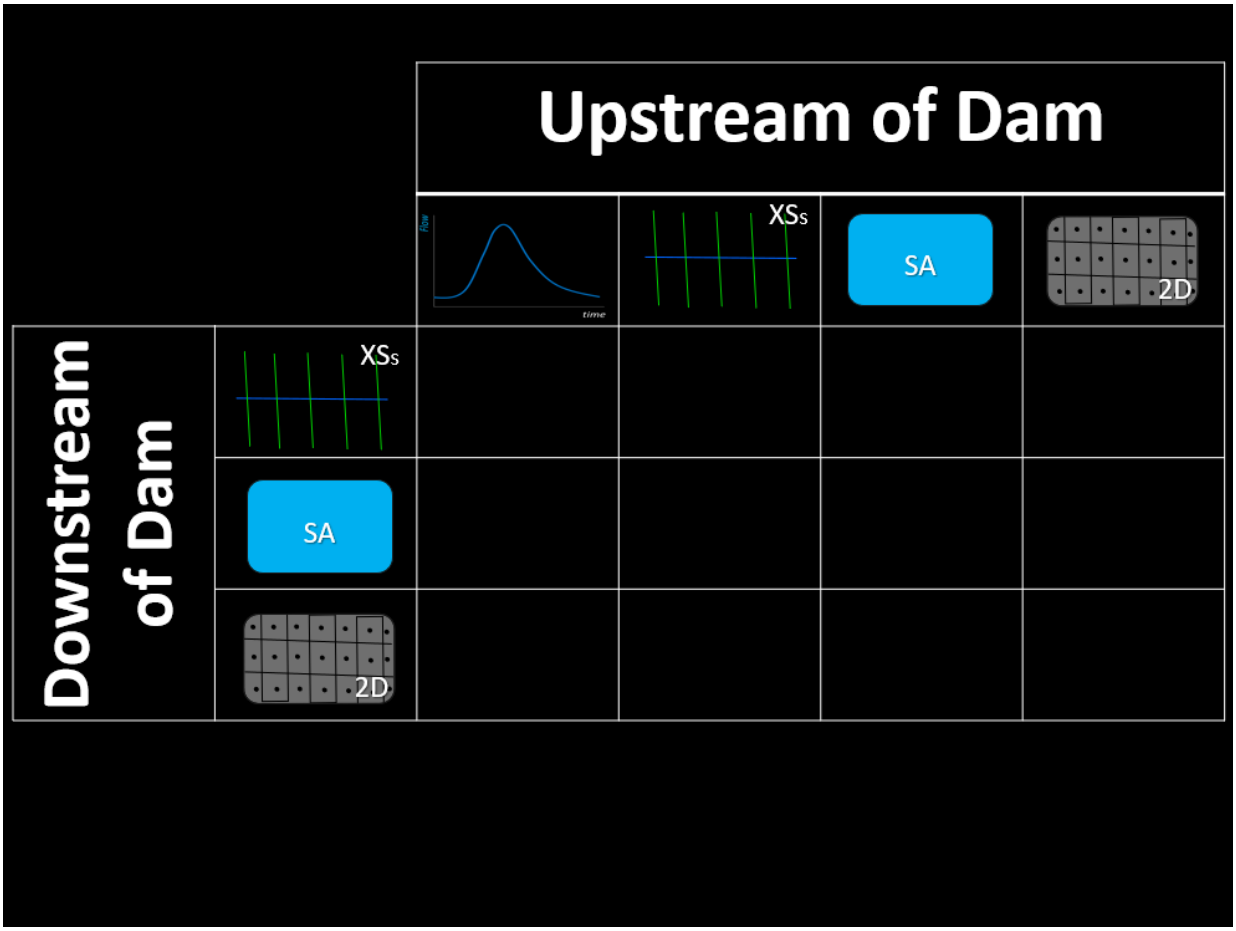


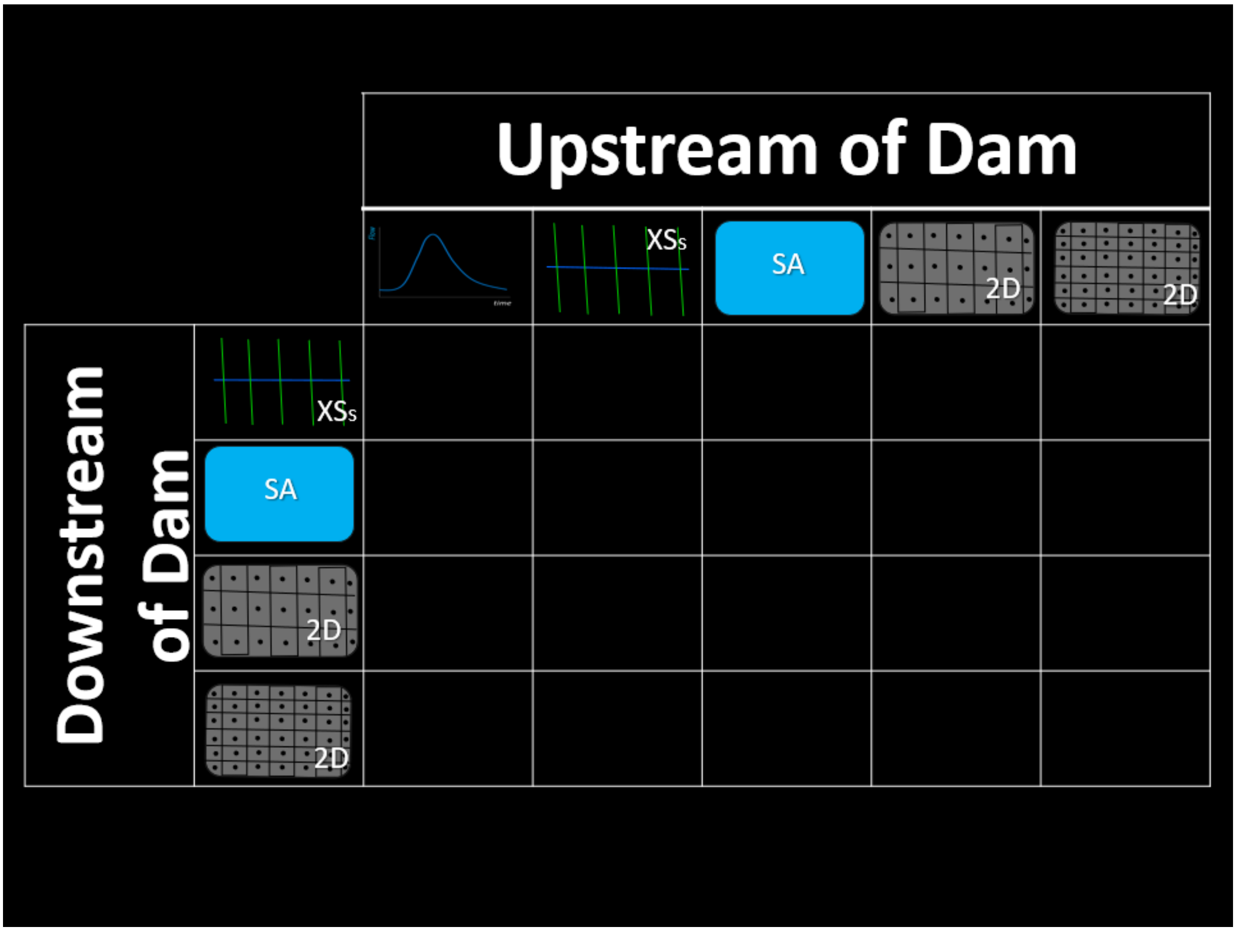
2. Breaching Options and Parameters



3. Breach Results and Visualization

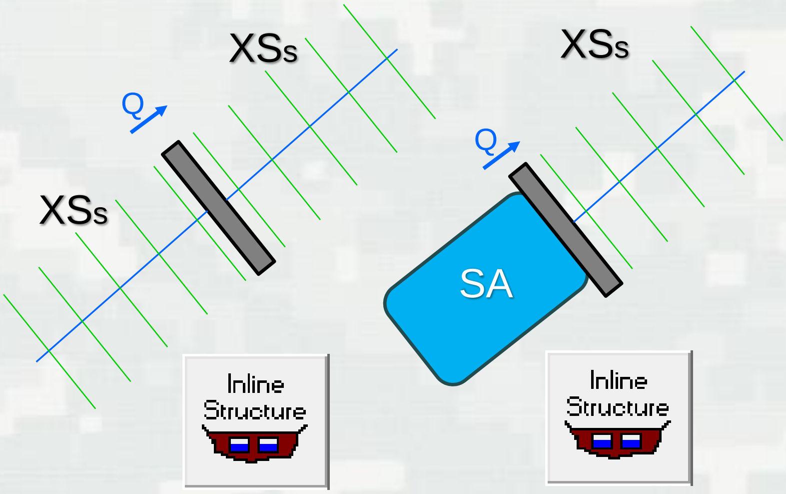



Upstream of Dam
Downstream of Dam
Flow
time
XSs
SA
2D
XSs
SA
2D



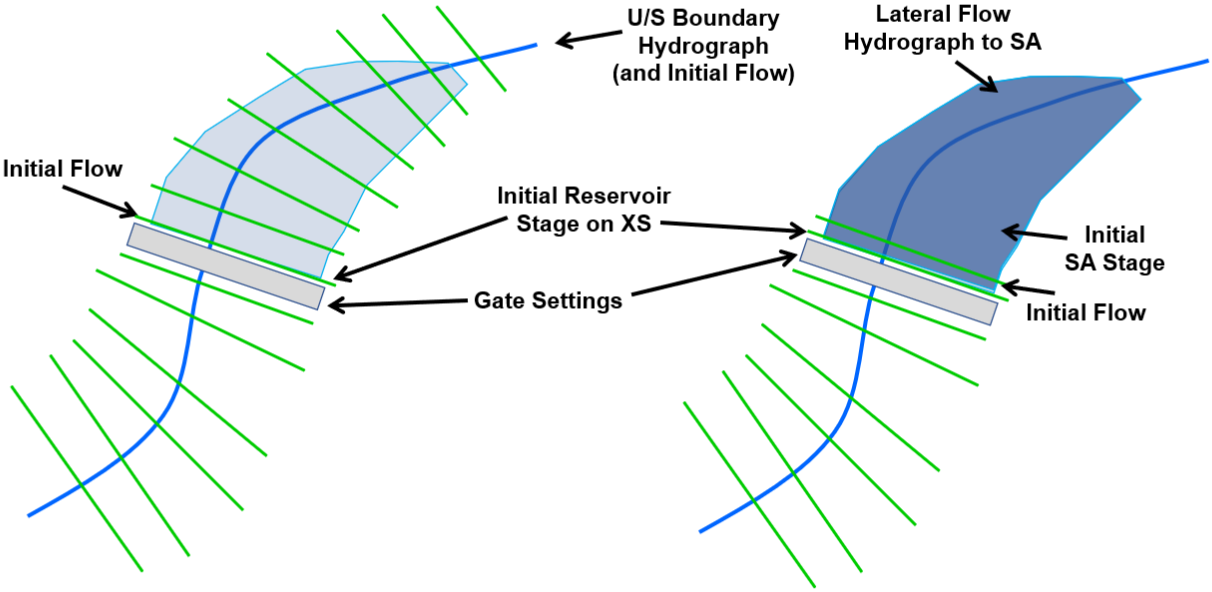
Upstream of Dam
XSs
SA
2D
2D
Downstream of Dam
XSs
SA
2D
2D

Breach Model Configuration Options



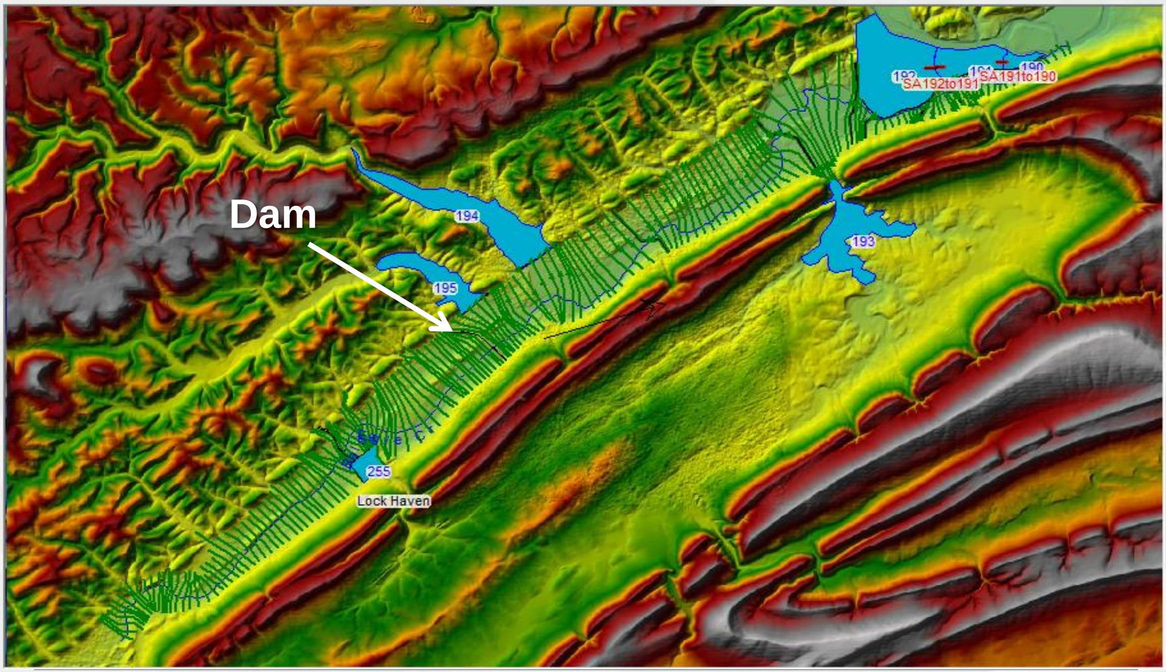
Pre-2D
Options

1D Reservoir Modeling Options



6

1D Dam Breaching Analysis
Cross Section for Pool and Downstream

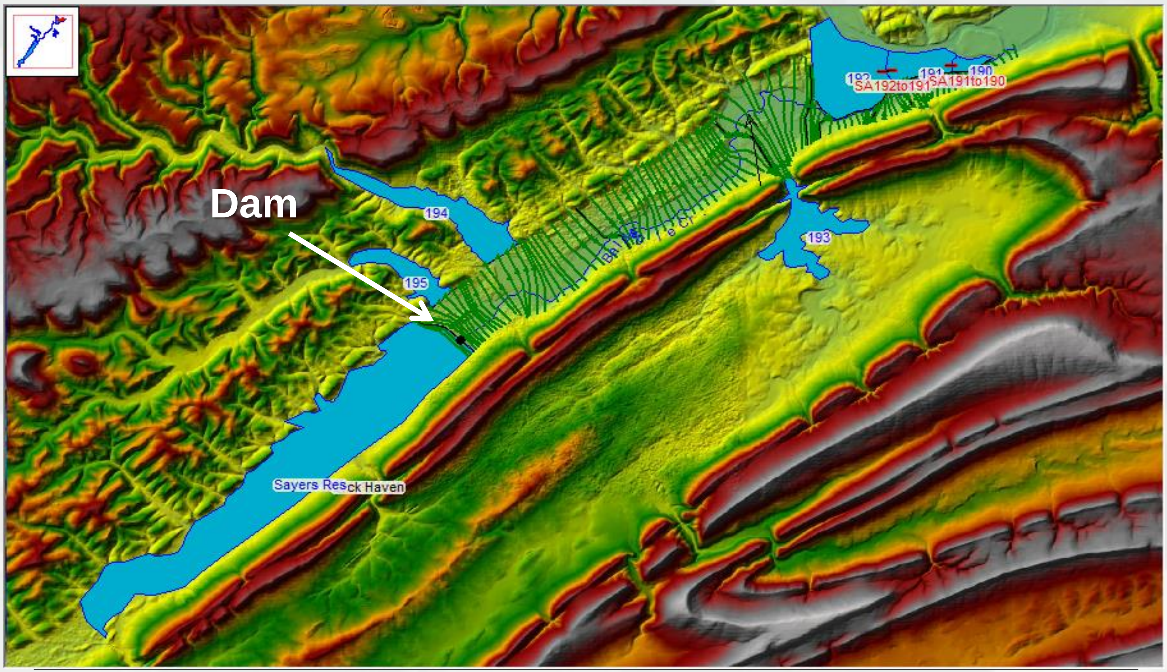


7

BUILDING STRONG®

This is a typical layout for modeling a Dam breach scenario with entirely 1D elements (River reaches and Storage areas). In this example the reservoir pool and the downstream river are modeled as a single reach with cross sections. There is a downstream levee system modeled with Lateral Structures, and the area behind the levee is modeled with interconnected storage areas. There are a few tributaries in which water will back up into during a dam break analysis, these have been modeled in a simple way with storage areas.

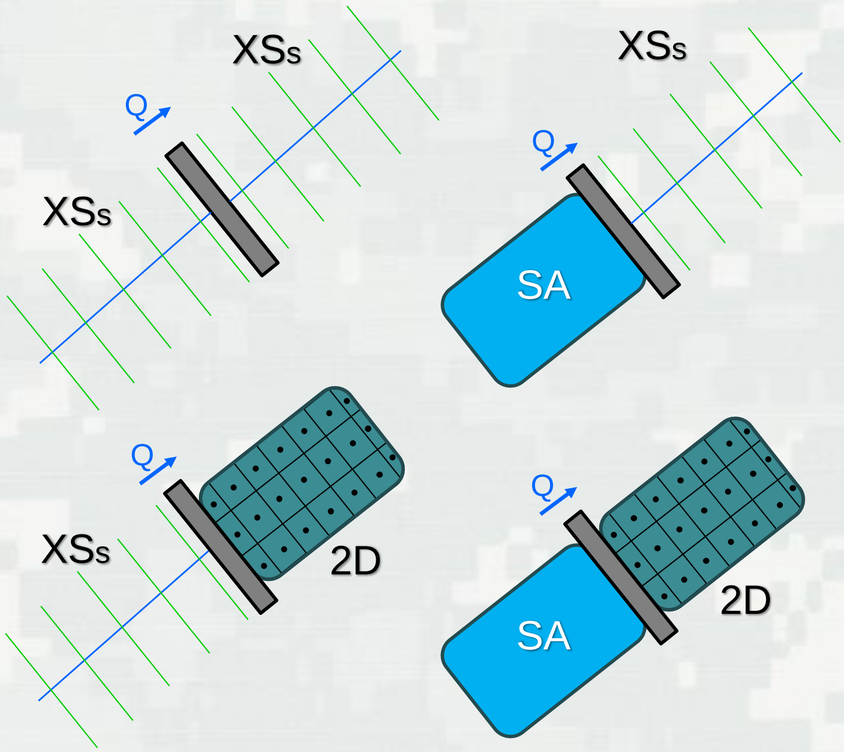
1D Dam Breaching Analysis
Storage Area Pool and XS Downstream



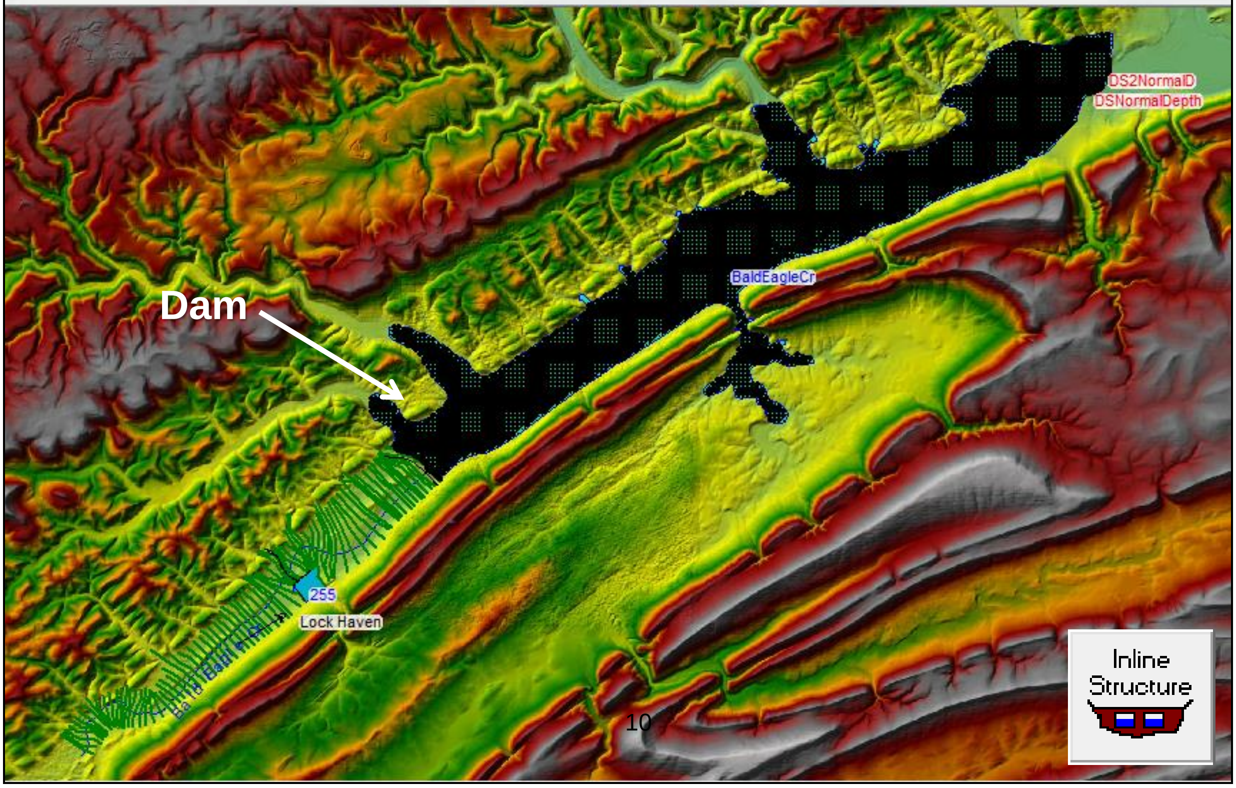
8 BUILDING STRONG®

This is another 1D only approach. The only difference in this model to the previous slide is that the reservoir pool is being modeled with a storage area (level pool routing).

Breach Model Configuration Options



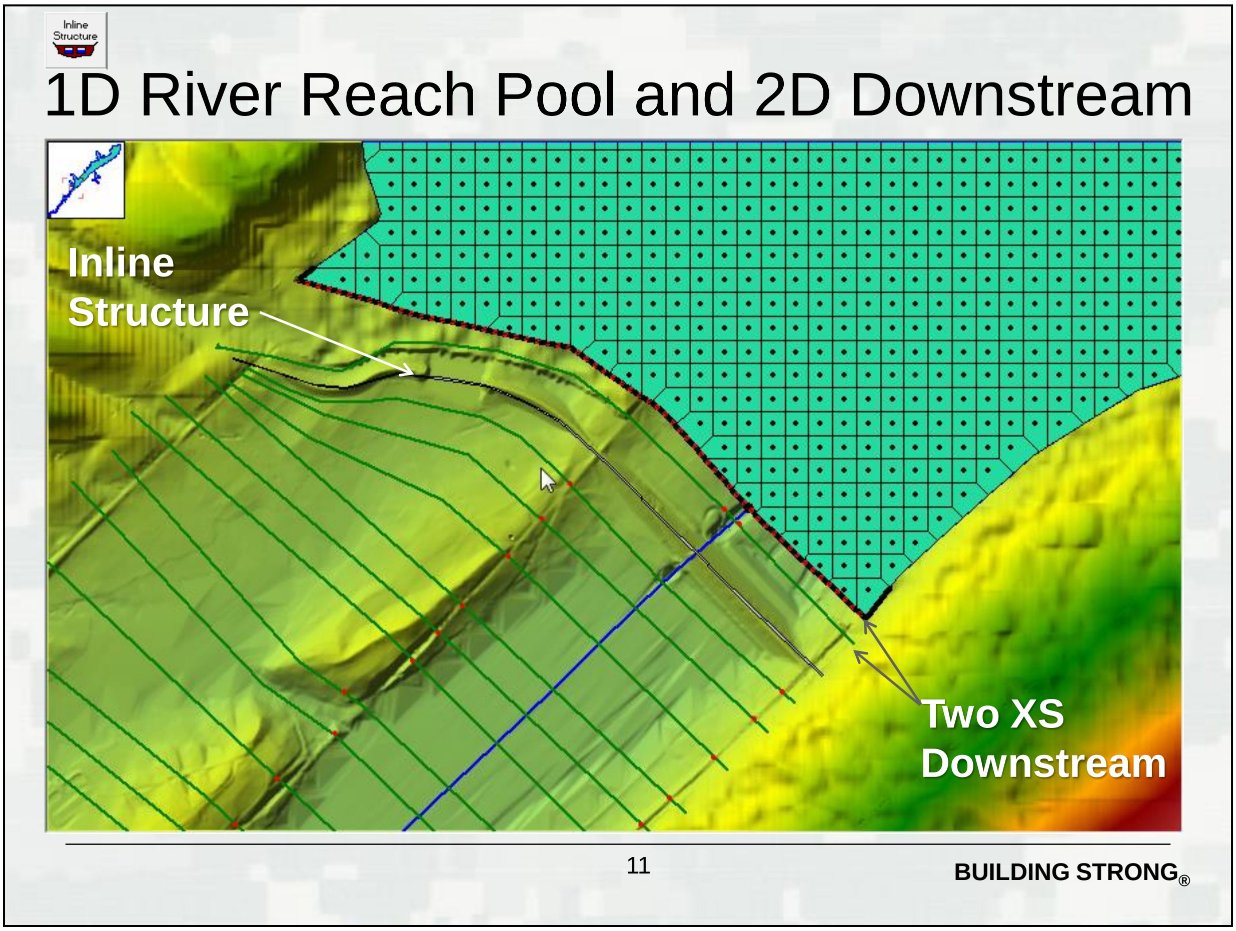
1D River Reach Pool and 2D Downstream



Users can connect a 1D River Reach directly to a 2D Flow Area. When this type of boundary condition is used, the last cross section of the 1D River Reach must be lined up with the upstream boundary of the 2D Flow Area (i.e., the last cross section of the 1D reach is directly linked to the boundary of the 2D area, so they need to be at the same exact location).

For this type of boundary condition, the 1D river reach passes flow each time step to the 2D Flow Area, while the stage in the cross section is based on the water surface elevation in the 2D cells that it is connected too. Flow is distributed to the 2D cells based on the conveyance distribution in the cross section, and the stationing of how the cells are linked to the cross section. The computed stage for the 1D cross section is based on computing a conveyance weighted stage from the connected boundary cells in the 2D Flow Area, and then forcing that stage on the 1D cross section each time step.

This type of boundary condition should only be placed in areas where the flow and stage are highly one-dimensional in nature. If the flow is not highly one-dimensional, you may need to turn on the option to allow the program to iterate back and forth between the 1D and the 2D computations during each time step, until the computed flow and stage at the boundary connection converges within a user specified tolerance. If the flow is highly one dimensional, 1D to 2D iterations are generally not necessary for this type of boundary condition.



To connect a 1D river reach to a 2D Flow Area, do the following:

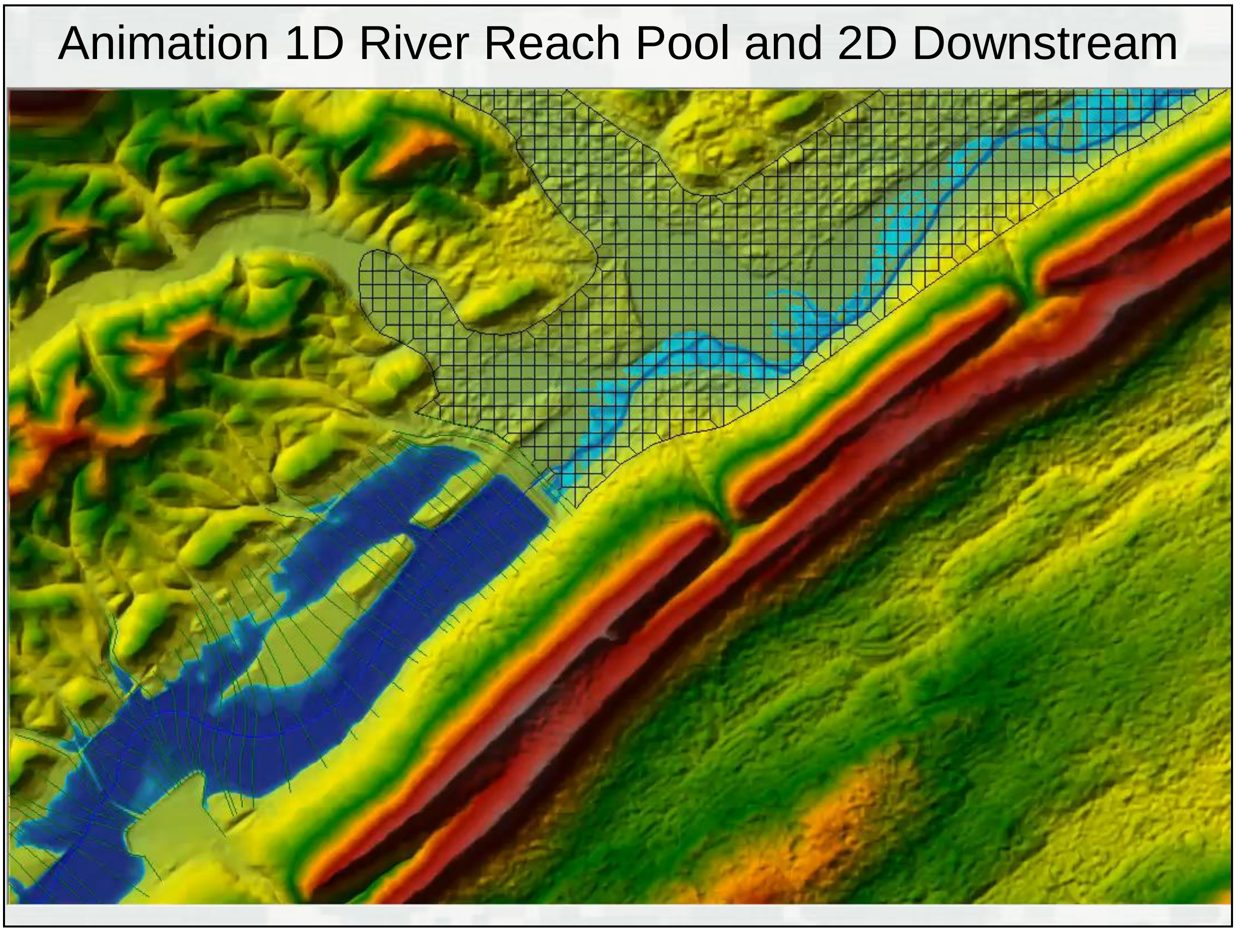
1. Draw the 2D area polygon such that the outer boundary at the upstream end is right on top of the last cross section of the 1D river reach.

2. Go to the **Edit** menu of the Geometric Editor, and turn on the Option to "Move Points/Objects".

3. Move the last point of the stream centerline inside of the 2D Area. The software will ask you if you want to connect the 1D River Reach to the 2D Flow Area. Select "Yes".

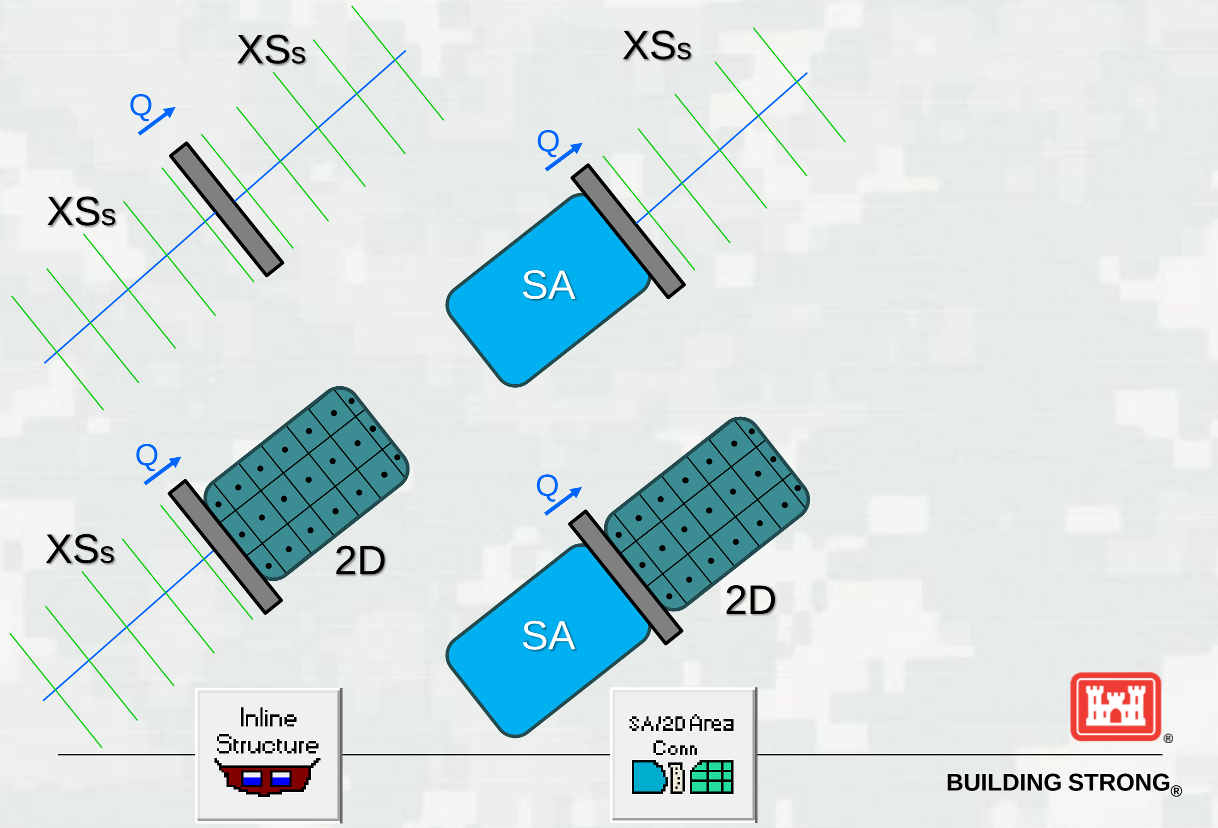
Once the 2D area and the 1D River Reach are connected, the software will draw a black line along the 2D Flow Area cells outer boundary to show you how it is connected. That is all that needs to be done for the connection.

Note: When a 1D River Reach is connected to a 2D area, the user will need to define the initial conditions for the 1D Reach and the 2D area. Initial conditions for the 2D Area can be: Dry; set to a single water elevation; set using a "Restart" file from a previous run; or the user can select to run a warm-up period at the beginning of the run, in which flow and stage boundaries connected to the 2D area will be applied slowly over time.

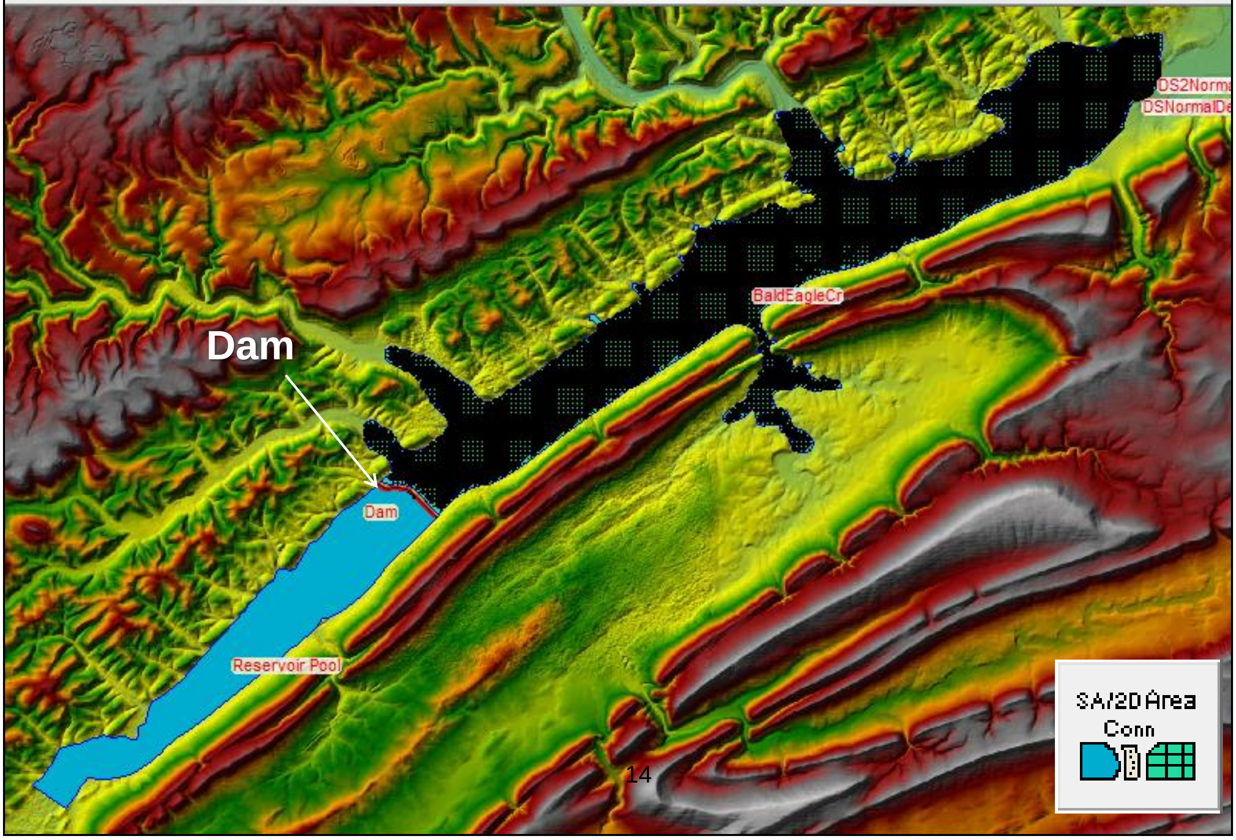


This is an animation of a Dam breaching analysis with a 1D River reach used for the reservoir pool, an Inline structure for the Dam, an then it is connected to a 2D Flow Area for modeling the area below the dam.

Breach Model Configuration Options

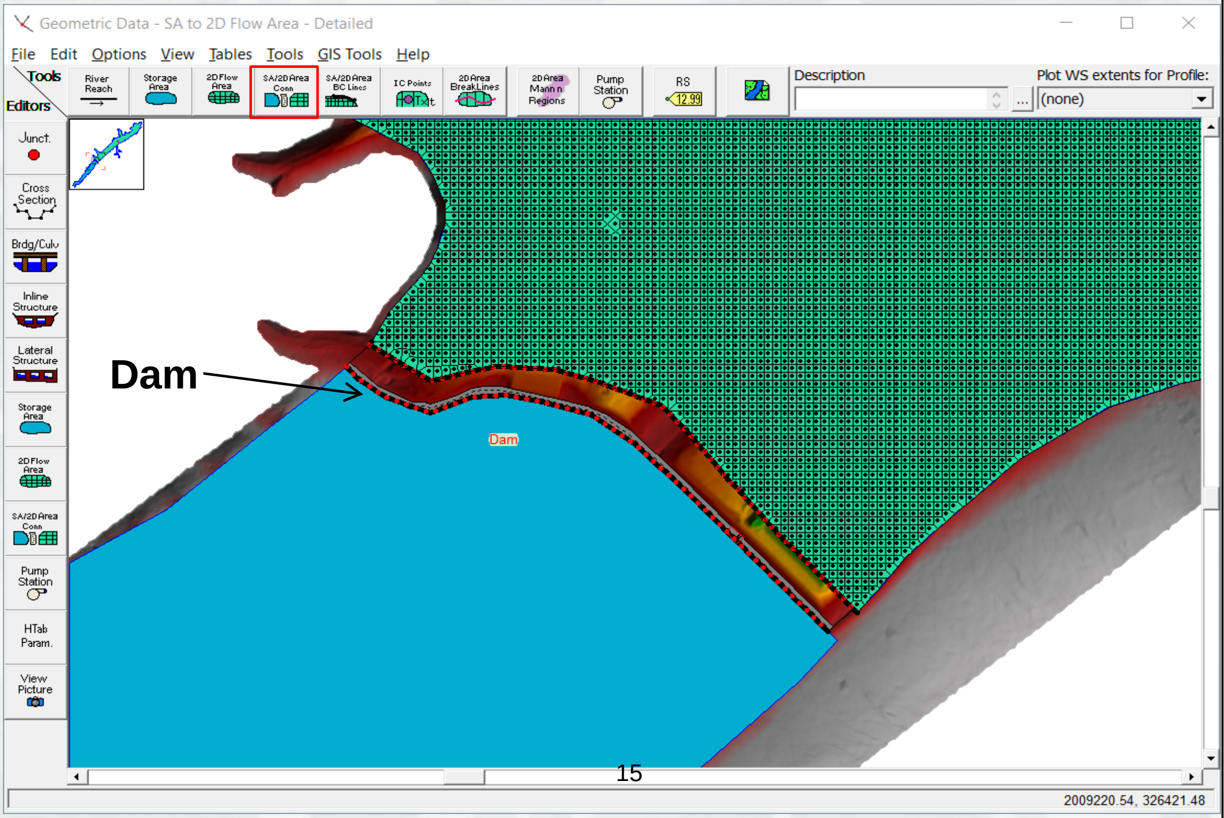


1D Storage Area Pool and 2D Downstream



A 2D Flow Area can be directly connected to Storage Area by using a hydraulic structure called a Storage Area/ 2D Flow Area Hydraulic Connector (**“SA/2D Area Conn”**)**.**

1D Storage Area Pool and 2D Downstream



In the example shown in the Figure above, the Storage Area is upstream of the 2D Flow Area, so the positive flow direction is from the storage area to the 2D Flow Area. Therefore, when defining the hydraulic structure that connects the two areas, the Storage Area will be considered the Headwater side, and the 2D Flow Area will be considered the Tailwater side. This can also be done the other way, in which the 2D Flow Area is on the upstream side (Headwater) and the Storage Area is on the downstream side (Tailwater). For the example shown, a Storage Area is being used to represent a reservoir pool. The hydraulic connection between the Storage Area and the 2D Flow Area is a dam (SA/2D Area Hydraulic Connection) in this example. The 2D Flow Area is being used to model the hydraulics of the flow downstream of the dam.

Steps to Connect a SA to a 2D Flow Area with a SA/2D Area Hydraulic Connection

- Draw the Storage Area and enter its data
- Draw the 2D Flow Area and create Mesh
- Using the **SA/2D Area Conn** drawing tool
  - draw the line that represents the hydraulic Structure from left to right looking downstream
- Select the **SA/2d Area Conn** data editor
  - Enter the “From” and “To” connections
  - Enter the top of dam and spillway profile
  - Enter any gate data, etc…

16 BUILDING STRONG®

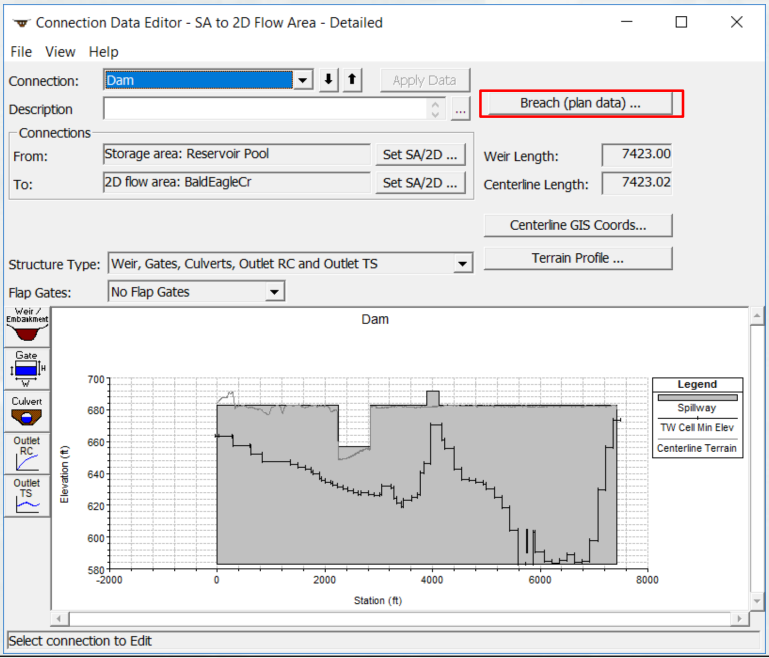
To hydraulically connect a Storage Area to a 2D Flow Area, do the following:

Draw the storage area polygon right up to the edge of the hydraulic structure. This can be as close to the hydraulic structure as you want for mapping purposes.

Draw the outer boundary of the 2D Flow Area right up to the other side of the hydraulic structure. This can also be very close to the hydraulic structure. However, keep in mind that the computed water surface elevations of the boundary cells of the 2D area will be used in the hydraulic calculations over/through the structure (don’t put very small cells down the face of a steep embankment). Generally, the water surface computed for the 2D cells should represent what you want for the water surface in the hydraulic calculations of flow over and through the hydraulic structure. That is, don’t put very small cells down the face of a steep embankment because the small boundary cells may end up with a transitional water surface that is between the “headwater” and the “tailwater” surfaces. If this happens, the accuracy of the hydraulic computations across the structure may be reduced. **Note: For any culverts and/or gates, the minimum elevation of the culvert/gate must not be below the minimum elevation of the cell it is connected too. This is another reason to use cells that are large enough to span at least to the bottom of the embankment.**

Select the Drawing tool at the top of the Geometric editor labeled “**SA/2D Area Conn**”. Then draw a line directly down the center of the hydraulic structure that will be used to connect the two flow areas. Draw this line from left to right looking downstream. This is how HEC-RAS will detect what is upstream (headwater) and what is downstream (tailwater). The interface will ask you for a label to define the hydraulic structure. See the red line shown in Figure 40.

Modeling the Dam with a SA/2D Area Hydraulic Connection



17

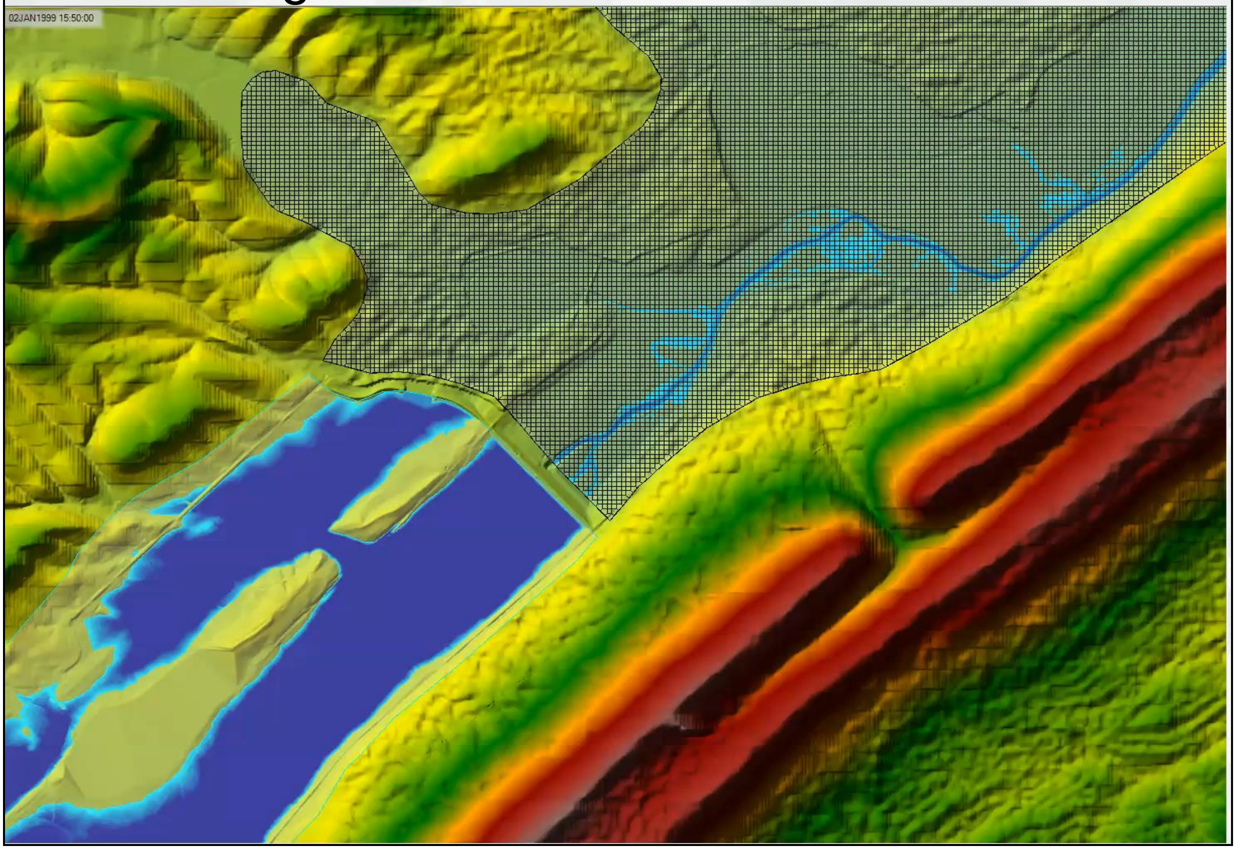
BUILDING STRONG®

Next, select the Storage Area/ 2D Flow Area Hydraulic Connection (**SA/2D Area Conn**) editor on the left panel of the Geometric data editor. This will bring up the editor shown in the Figure above.

On the **SA/2D Area Conn** editor set the "From" and "To" by selecting the buttons labeled "Set SA/2D Area". For this example, the Storage Area labeled "Reservoir Pool" is the "From" element, and the 2D Flow Area labeled "BaldEagleCr" is the "To" element.

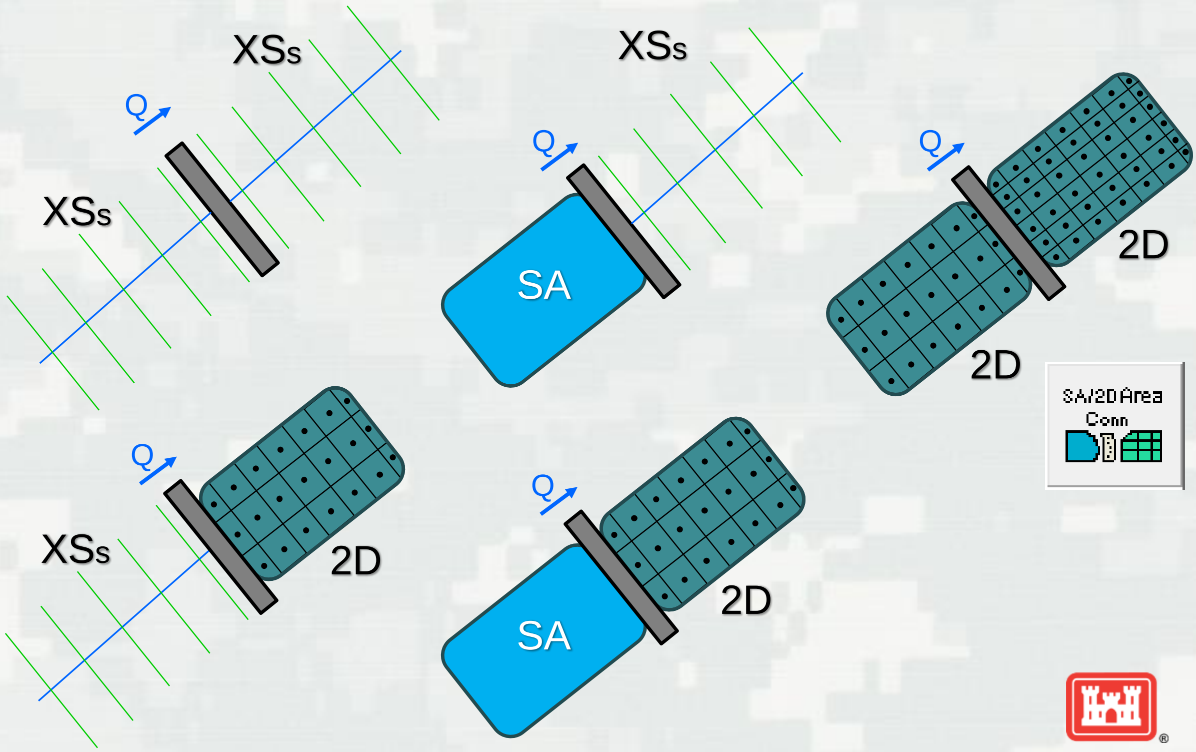
Enter all the hydraulic structure information for the connection. This will consist of a Weir/Embankment profile, and any additional hydraulic outlets, such as culverts, gates, etc… In the example shown in the Figure above, there is an embankment with an emergency spillway defined, and there are also low flow gates defined.

1D Storage Area Pool→2D Area Downstream



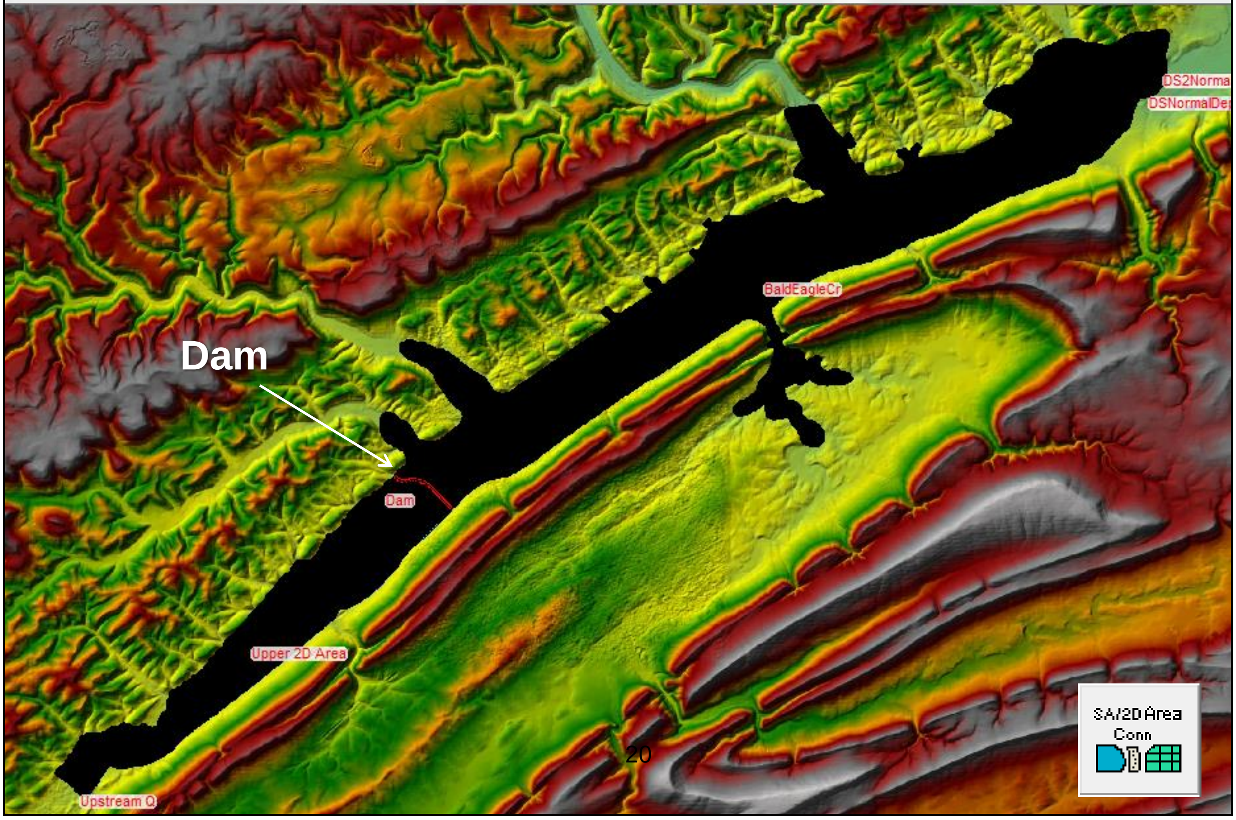
This is an animation of a Dam Breaching analysis done with a Storage Area for the reservoir pool, a SA/2D Area Hydraulic Connection for the Dam, and a 2D Flow Area for the downstream area below the dam. This is a very quick and easy way to model a dam. A user can put together a preliminary Dam Break model with this approach very easily. Rough answers could be developed very quickly with this type of approach. However, remember a detailed model would require refinement of the 2D Flow Area mesh, such that all of the levees, roads, and high ground barriers to flow were incorporated into the computational mesh. Also, any bridges, culvert weirs, and gated structures would also need to be added to have a detailed model.

Breach Model Configuration Options



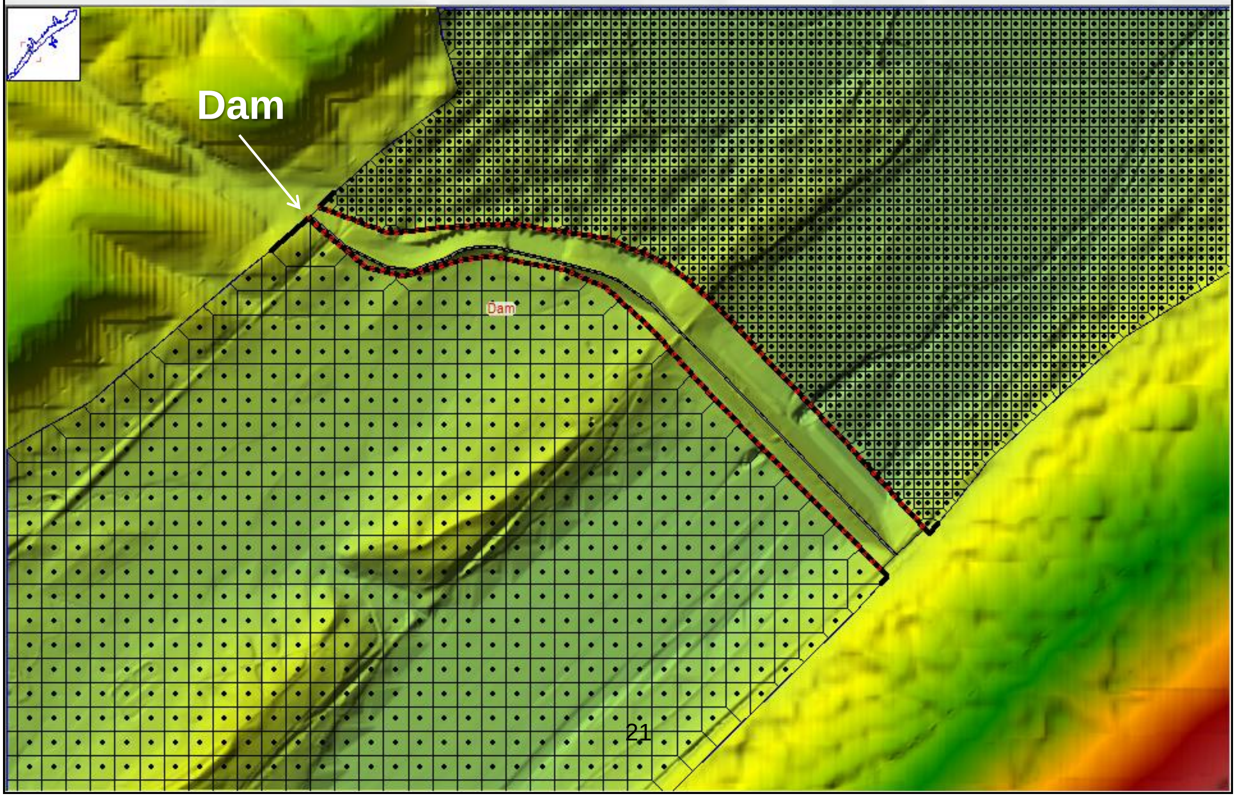
BUILDING STRONG®

2D Reservoir Pool and 2D Downstream



2D Flow Areas can be directly connected to other 2D Flow Areas by using a hydraulic structure called a Storage Area/ 2D Flow Area Hydraulic Connector (**“SA/2D Area Conn”).** In the example shown in the Figure above, there is a 2D Flow Area upstream of another 2D Flow Area, so the positive flow direction is from the upstream 2D Flow Area to the downstream 2D Flow Area. When defining the hydraulic structure that connects the two areas, the upstream 2D Flow Area will be considered the Headwater side, and the downstream 2D Flow Area will be considered the Tailwater side.

2D Reservoir Pool and 2D Downstream



To hydraulically connect one 2D Flow Area to another 2D Flow Area, do the following:

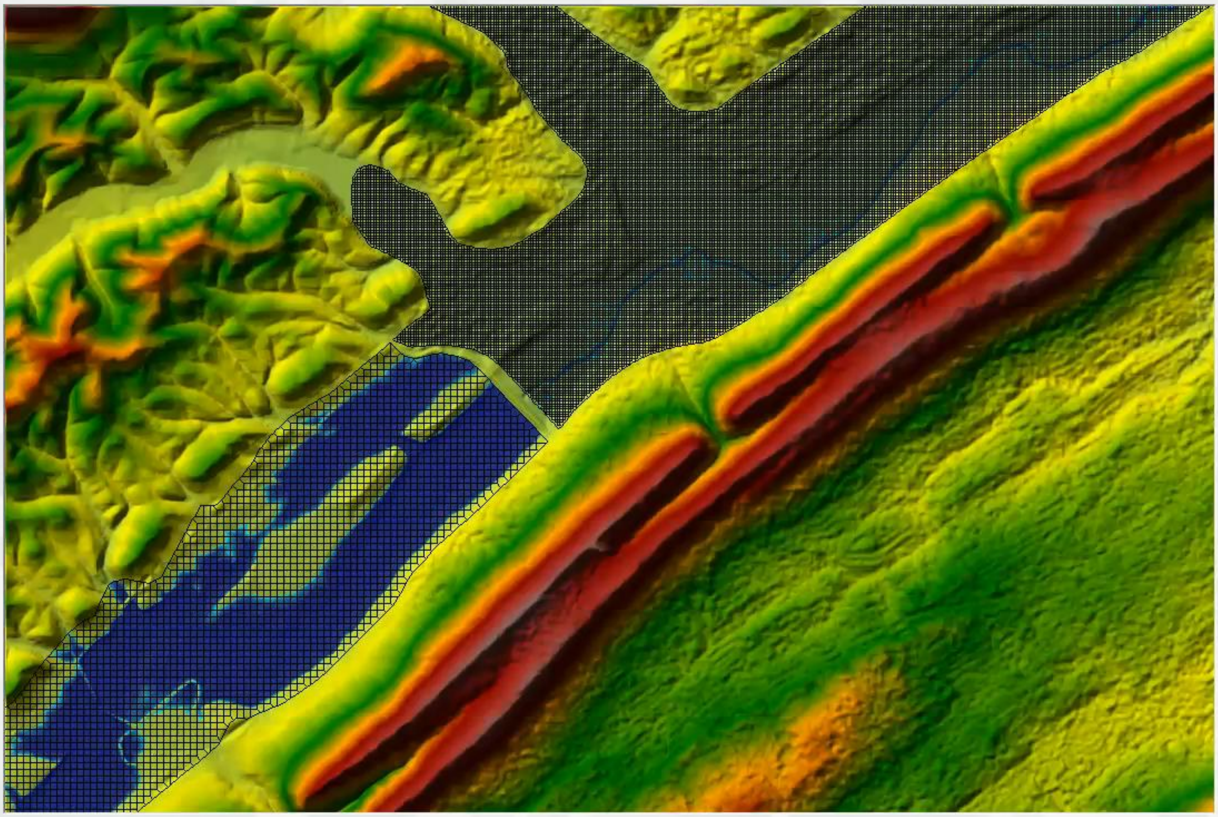
Draw the upstream 2D Flow Area polygon right up to the edge of the hydraulic structure. This should be relatively close to the hydraulic structure for mapping purposes.

Draw the outer boundary of the downstream 2D Flow Area right up to the other side of the hydraulic structure. This can also be very close to the hydraulic structure, however, keep in mind that the computed water surface elevations of the boundary cells of the 2D area will be used in the hydraulic calculations over/through the structure (i.e., don't put very small cells down the face of a steep embankment). Generally, the 2D cells computed water surfaces should represent what you want to be used for the water surface in the hydraulic calculations of flow over and through the hydraulic structure.

Select the Drawing tool at the top of the Geometric editor labeled "**SA/2D Area Conn**". Then draw a line directly down the center of the hydraulic structure that will be used to connect the two flow areas. The interface will ask you for a label to define the hydraulic structure. See the red line shown in the Figure above.

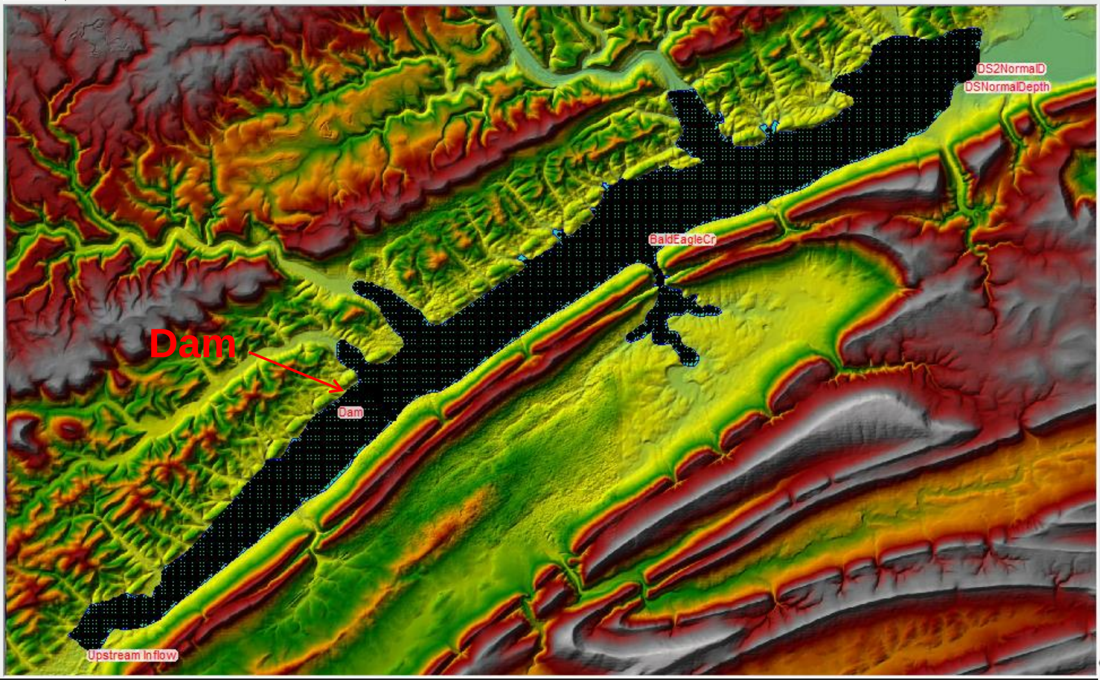
Next, select the Storage Area/ 2D Flow Area Hydraulic Connection (**SA/2D Area Conn**) editor on the left panel of the Geometric data editor. This will bring up the Hydraulic Structure editor, which will allow you to enter all of the data for the Dam, spillway, gates, etc…

Animation of 2D Reservoir Pool and 2D Downstream



This is an animation of a Dam Breaching analysis done with two 2D Flow Areas. The upstream 2D Flow Area represents the reservoir pool, and was done with a courser mesh. The downstream 2D Flow Area is used to model the area below the dam, this was dine with a finer mesh for more detail.

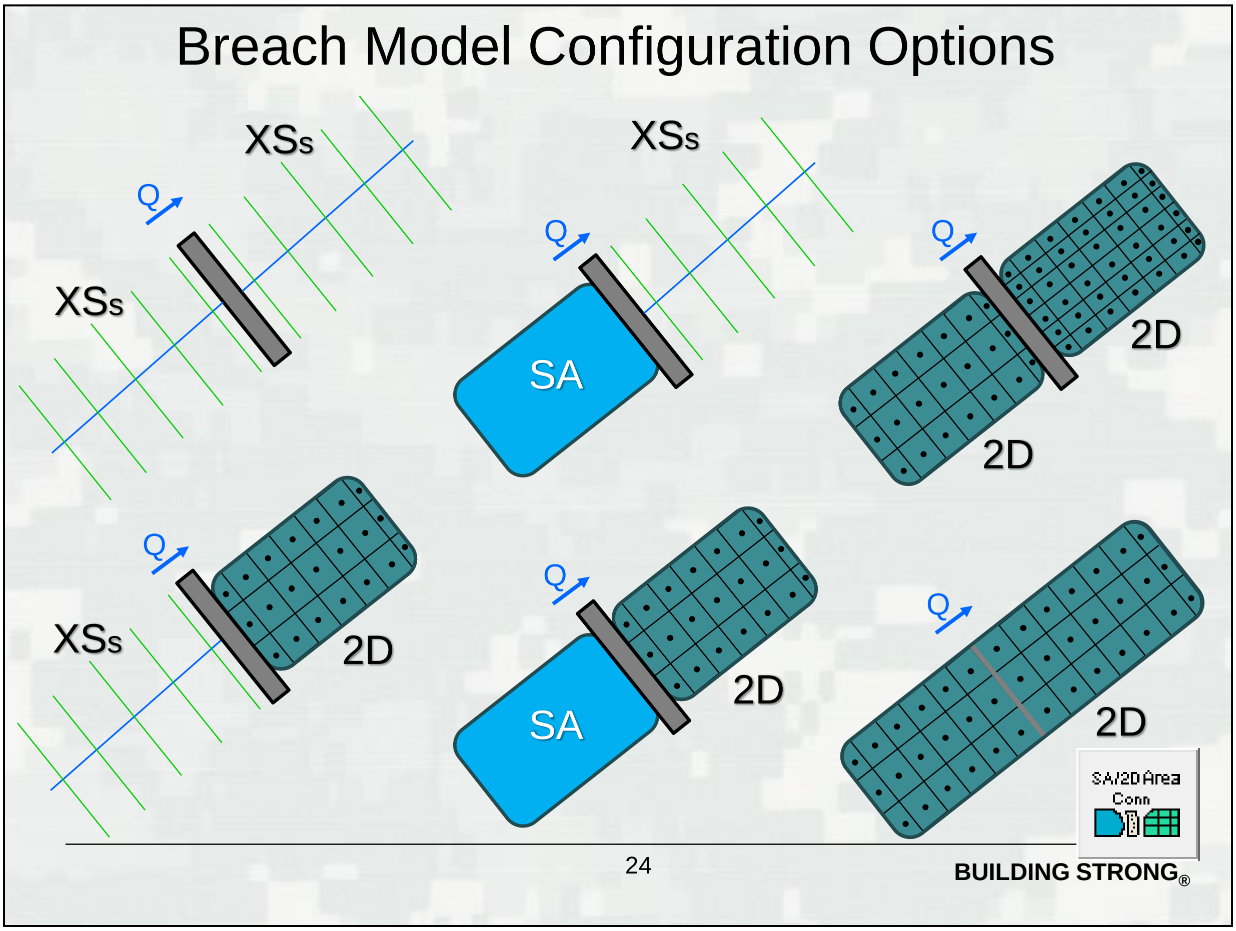
Single 2D Flow Area with Internal Hydraulic Structure for the Dam



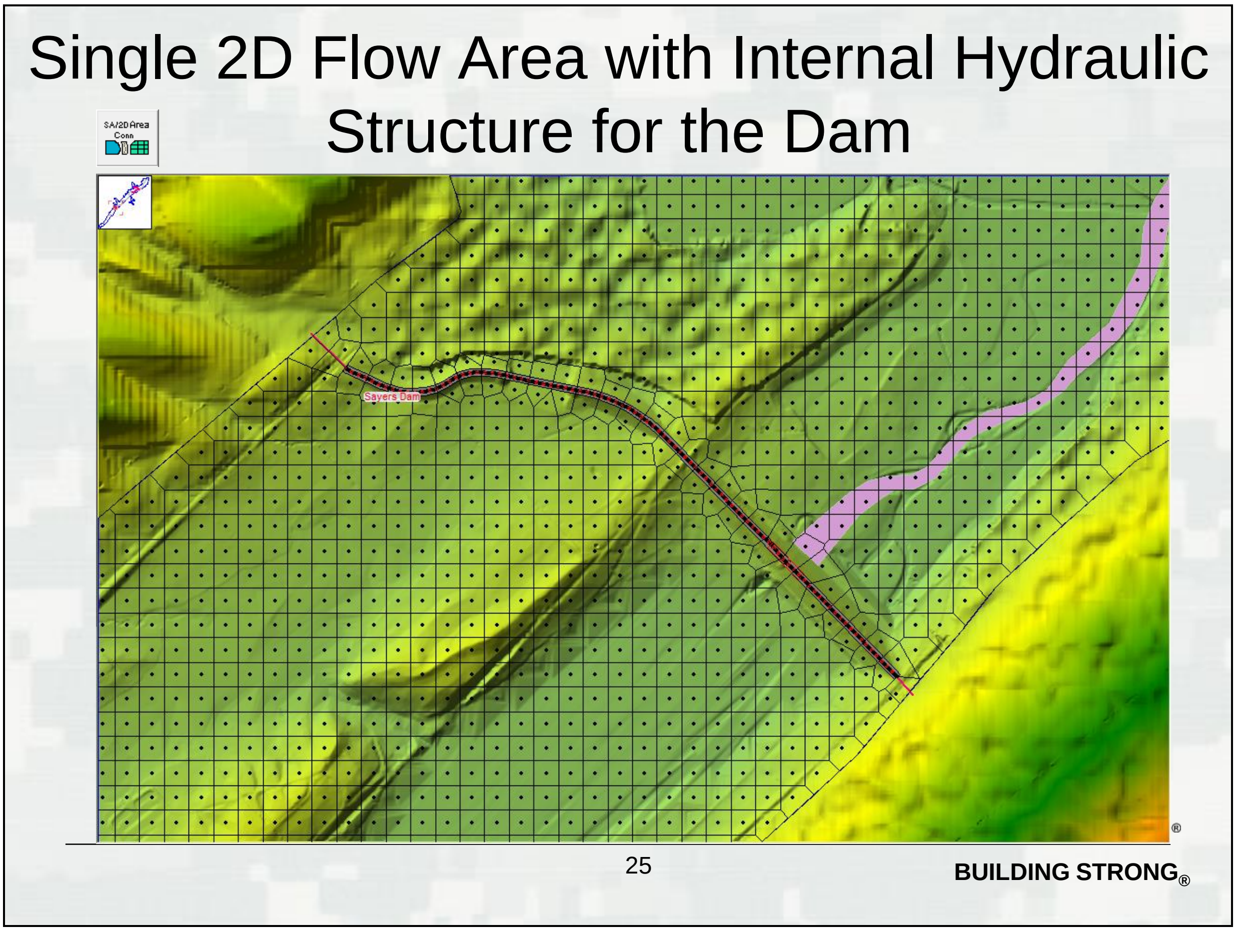
23

BUILDING STRONG®

This is an example of performing a Dam breaching analysis with a single 2D Flow Area for the entire model.



Breach Model Configuration Options
XSs
XSs
XSs
XSs
Q
Q
Q
Q
Q
Q
SA
SA
2D
2D
2D
2D
2D
2D
SA/2D Area
Conn
24
BUILDING STRONG®



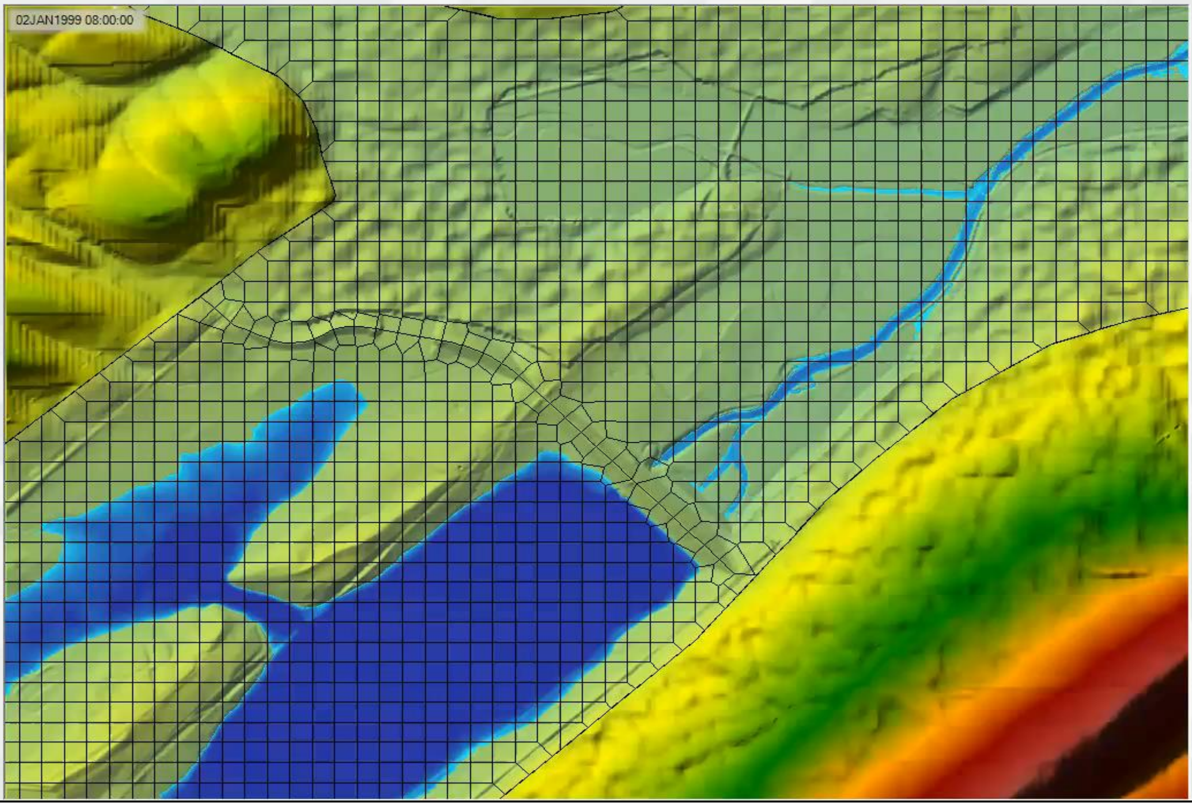
HEC-RAS has the ability to add hydraulic structures inside of 2D Flow Areas. This is accomplished by using the "**SA/2D Area Conn**" option to make a hydraulic structure in the middle of a single 2D Flow Area. The hydraulic structure must be laid out along the Faces of the 2D Cells (2D Cell Faces control flow movement). To add a hydraulic structure inside of a 2D Flow Area do the following:

First modify the 2D Flow Area mesh so that the faces of the cells go along the centerline of the top of the hydraulic structure. For example, as shown in the Figure above, a Dam is being modeled inside of a single 2D Flow Area. The 2D Flow Area mesh was modified (cell center points were added and moved) to have cells on both sides of the Dam, such that the faces between the Dam lined up on top of the Dam. This requires adding enough cells to get the correct detail, as well as placing cell centers at equal distances apart on each side of the structure. [Note: currently this is tedious with the HEC-RAS mesh editing tools. However, future versions of HEC-RAS will allow you to draw a break line on top of the levee (or hydraulic structure) and it will automatically modify the mesh to align the cell faces with the structure].

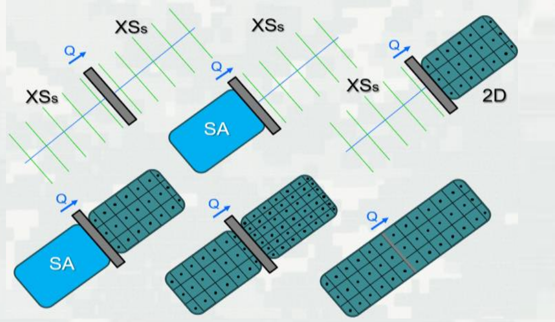
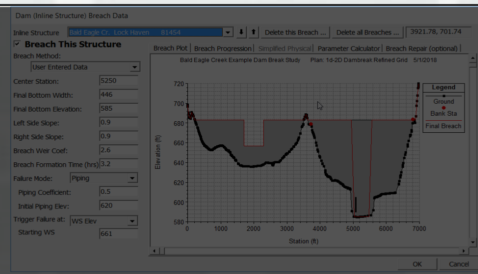
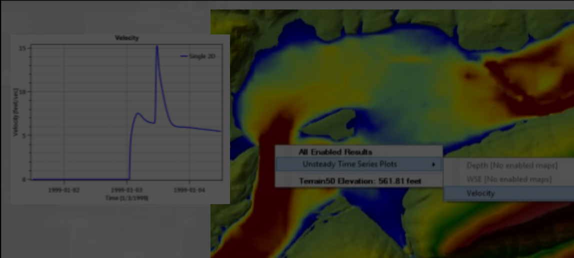
Next, select the Drawing tool at the top of the Geometric editor labeled "**SA/2D Area Conn**". Then draw a line directly down the center of the hydraulic structure along the cell faces that represent the structure (**NOTE: this line Must be drawn from left to right, while looking from what is considered to be upstream to downstream. This is how the program figures out what is considered to be the headwater side and the tailwater side.**). This line will be the hydraulic structure that will be used to connect the 2D Flow Areas cells on one side of it to the other side of it. The interface will ask you for a label to define the name of the hydraulic structure. See the black line in the Figure above.

Next, select the Storage Area/ 2D Flow Area Hydraulic Connection (**SA/2D Area Conn**) editor on the left panel of the Geometric data editor.

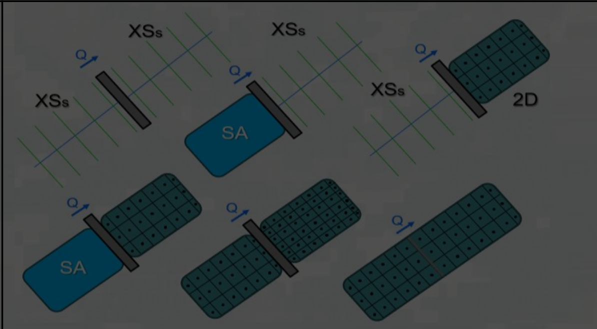
Animation of Single 2D Flow Area with Internal Hydraulic Structure for the Dam



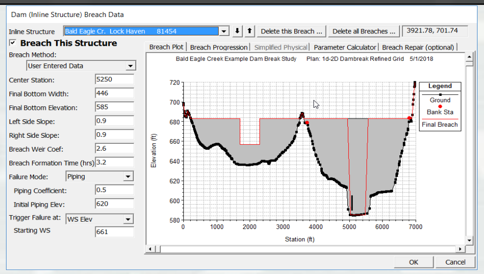
This is an animation of a Dam breaching analysis with a single 2D Flow Area for the entire model. The mesh was modified in the area of the Dam, then a SA/2D Area Hydraulic Connection was added to model the Dam. This structure allows for modeling flow going over the emergency spillway, through the low flow gates, and even through a hypothetical breaching of the Dam.

1. Six Dam Breach Model Configurations	
2. Breaching Options and Parameters	
3. Breach Results and Visualization	

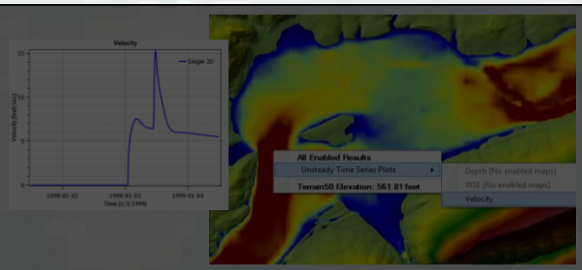
1. Six Dam Breach Model Configurations



2. Breaching Options and Parameters



3. Breach Results and Visualization

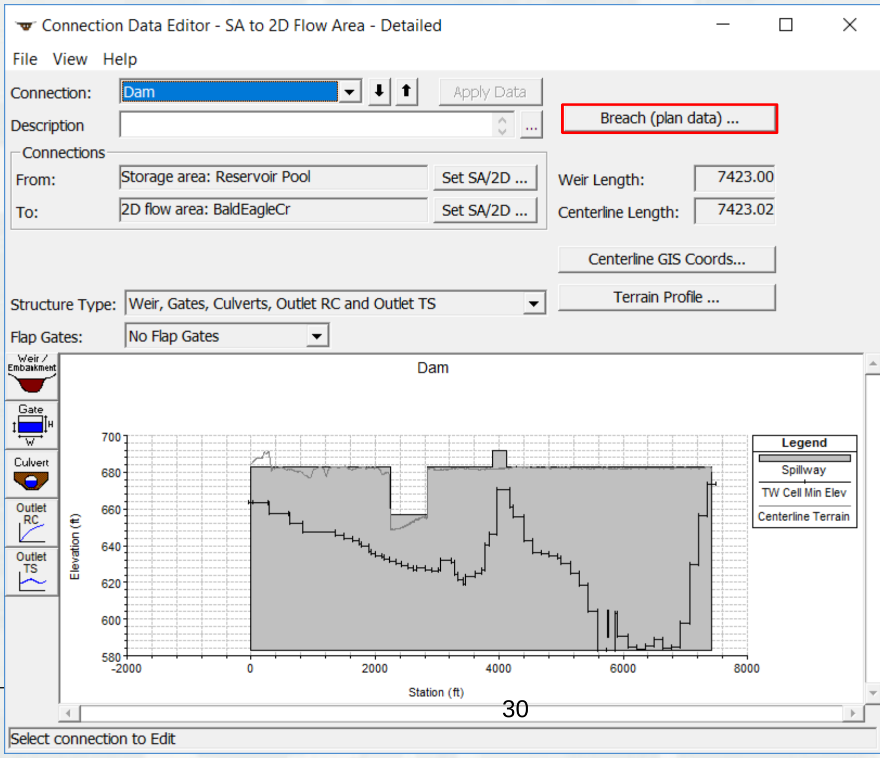


Dam Break Analysis in HEC-RAS

- Failure Modes
  - Overtopping and Piping
- Failure Initiation based on:
  - stage
  - simulation time
  - stage + duration, and immediate initiation stage.
- Breach progression
  - linear or nonlinear (user specified)
  - Simplified Physical Breaching Option
  - DL Breach

HEC-RAS can be used to model both overtopping as well as piping failure breaches for earthen dams. Additionally, the more instantaneous type of failures of concrete dams (generally occurring from earthquakes) can also be modeled. The resulting flood wave is routed downstream using the unsteady flow equations. Inundation mapping of the resulting flood can be done with the HEC RAS-Mapper.

Modeling the Dam with a SA/2D Area Hydraulic Connection



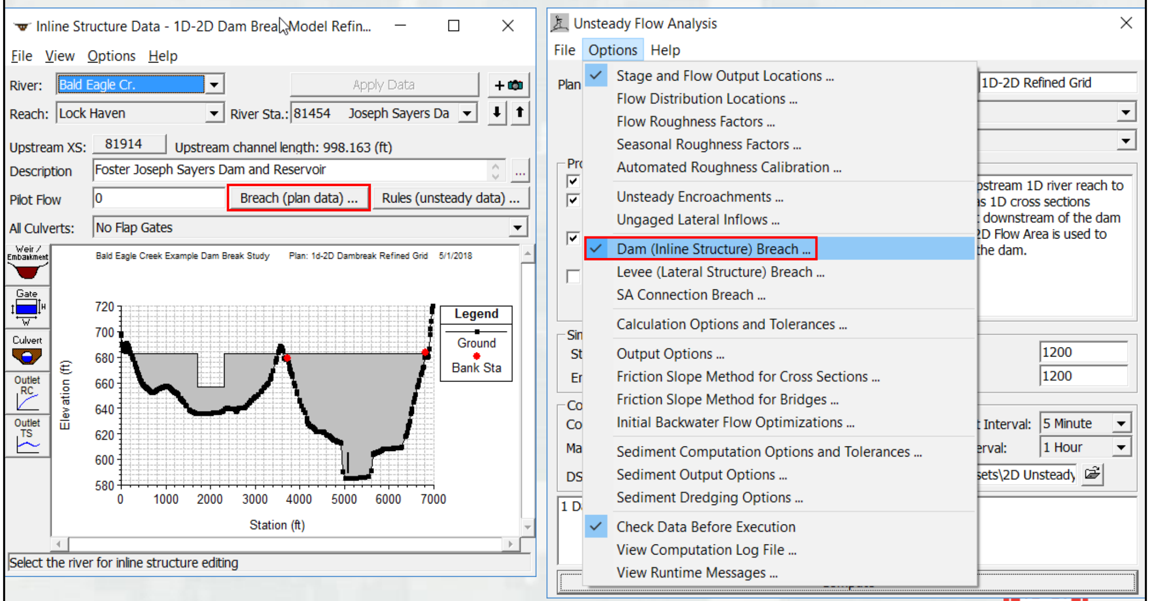
.DING STRONG®

Next, select the Storage Area/ 2D Flow Area Hydraulic Connection (**SA/2D Area Conn**) editor on the left panel of the Geometric data editor. This will bring up the editor shown in the Figure above.

On the **SA/2D Area Conn** editor set the “From” and “To” by selecting the buttons labeled “Set SA/2D Area”. For this example, the Storage Area labeled “Reservoir Pool” is the “From” element, and the 2D Flow Area labeled “BaldEagleCr” is the “To” element.

Enter all the hydraulic structure information for the connection. This will consist of a Weir/Embankment profile, and any additional hydraulic outlets, such as culverts, gates, etc… In the example shown in the Figure above, there is an embankment with an emergency spillway defined, and there are also low flow gates defined.

Inline Structure Dam Break Control



31

BUILDING STRONG®

A dam breach can be performed on an inline weir and a levee breach can be performed on a lateral weir. The breach editor can be accessed from either the unsteady flow analysis editor or by clicking the breach button on the inline or lateral weir editor. It is usually more convenient to just click on the button on the appropriate geometry editor, but the user should keep in mind that breach information is stored with the plan data not the geometry data. For instance, to compare a “breached” with a “not breached” run, the project should have two plans, but they can (and in many cases should) have the same geometry.

The dam breach data (next slide) is as follows:

Breach This Structure - This check box is used to decide if the program will perform the breach or not. This box was added to allow the user to turn certain breaches on or off, without loosing the user entered breach information.

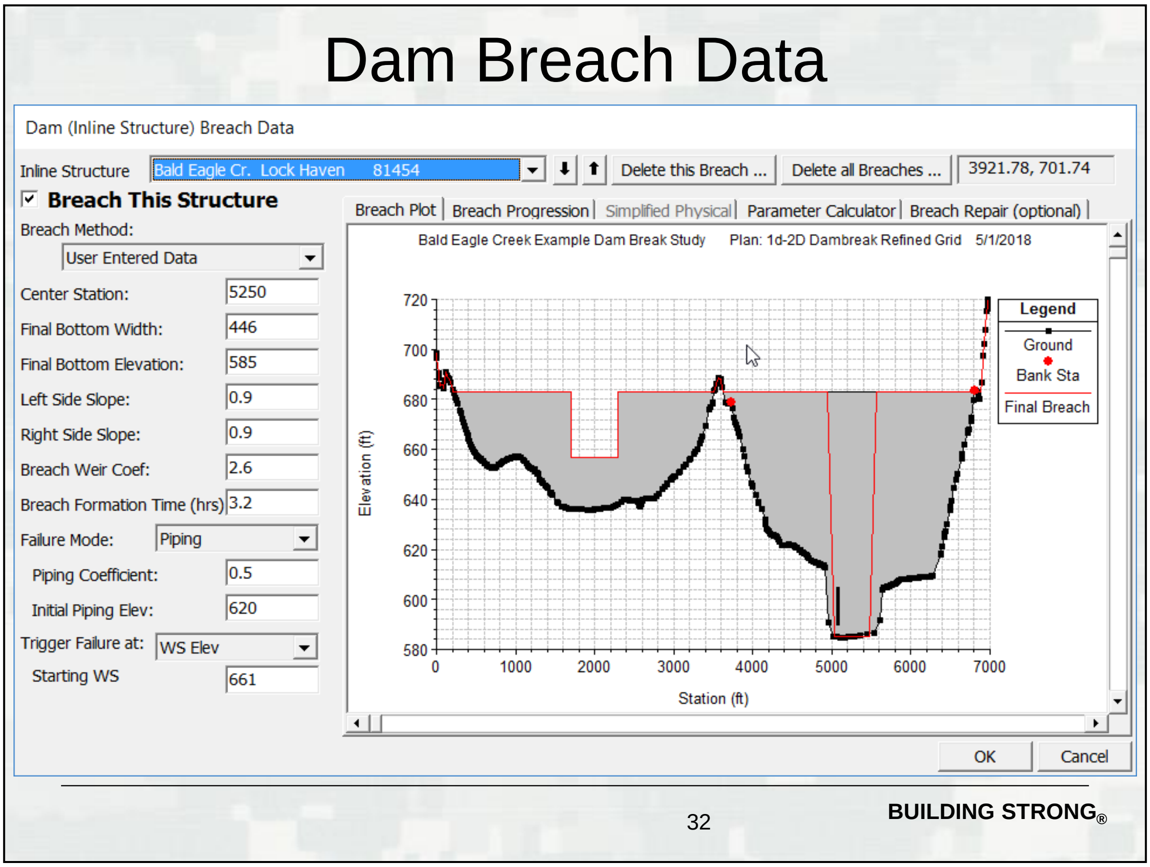
Center Station - The centerline stationing of the final breach.

Final Bottom Width - The bottom width of the breach at its maximum size.

Final Bottom Elevation - The elevation of the bottom of the breach after it has been fully developed.

Left Side Slope - This is the left side slope of the trapezoidal breach.

Right Side Slope - This is the right side slope of the trapezoidal breach.



Center Station - This field is the cross section stationing of the centerline of the breach.

Final Bottom Width - Bottom width of the breach when it has reached its maximum size.

Final Bottom Elevation - Bottom elevation of the breach at its maximum size.

Left Side Slope - Left side slope for the trapezoid that will represent the final breach shape. Side slopes are entered in values representing the horizontal to vertical ratio.

Right Side Slope - Right side slope for the trapezoid that will represent the final breach shape. Side slopes are entered in values representing the horizontal to vertical ratio.

Breach Weir Coefficient –Weir coefficient used for overtopping (weir) flow in the breach area.

Failure Mode - This option allows the user to choose between two different failure modes, an Overtopping failure and a Piping failure.

Piping Coefficient - If a piping failure mode is selected, the user must enter a piping coefficient.

Initial Piping Elev. - If a piping failure is selected, the user must enter an initial piping elevation.

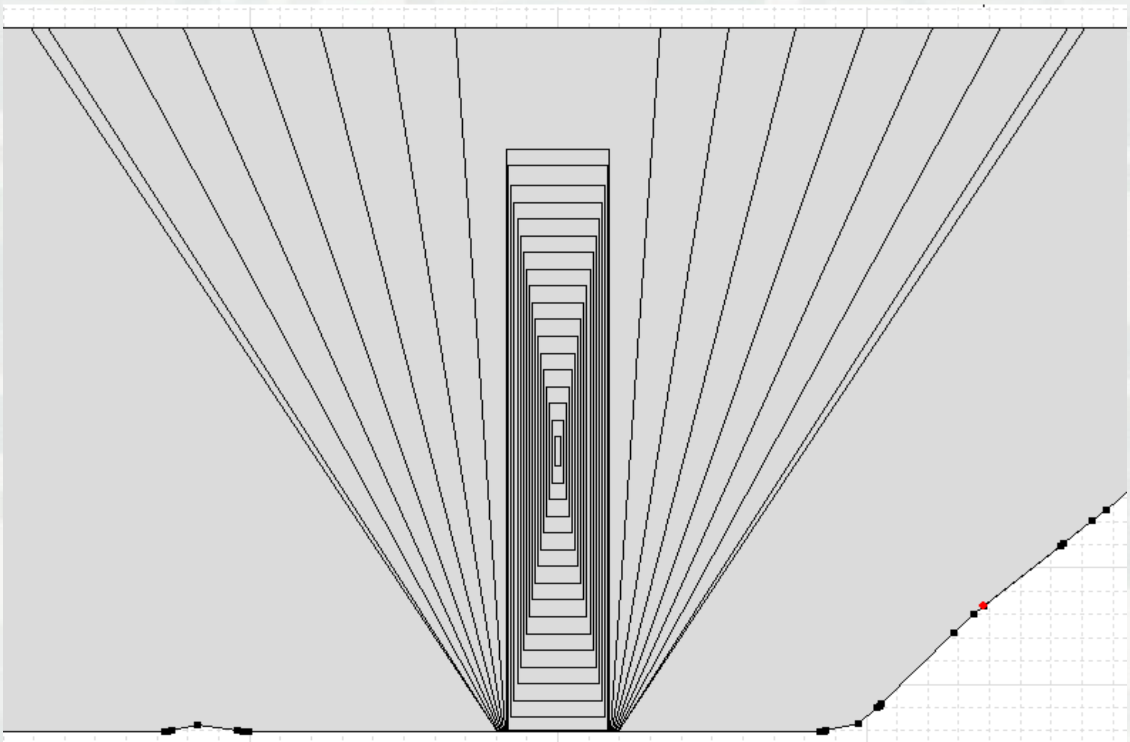
Trigger Failure At - This field is used to select one of two trigger methods for initiating the breach. The two trigger methods are a water surface elevation or a specific time and date.

Starting WS – If water surface elevation is the failure trigger mode, then this field represents the water surface elevation at which the breach should begin to occur.

Start Date – If starting date and time is the failure trigger mode, then this field must be entered. This field is used to enter the date at which the breach will begin to occur.

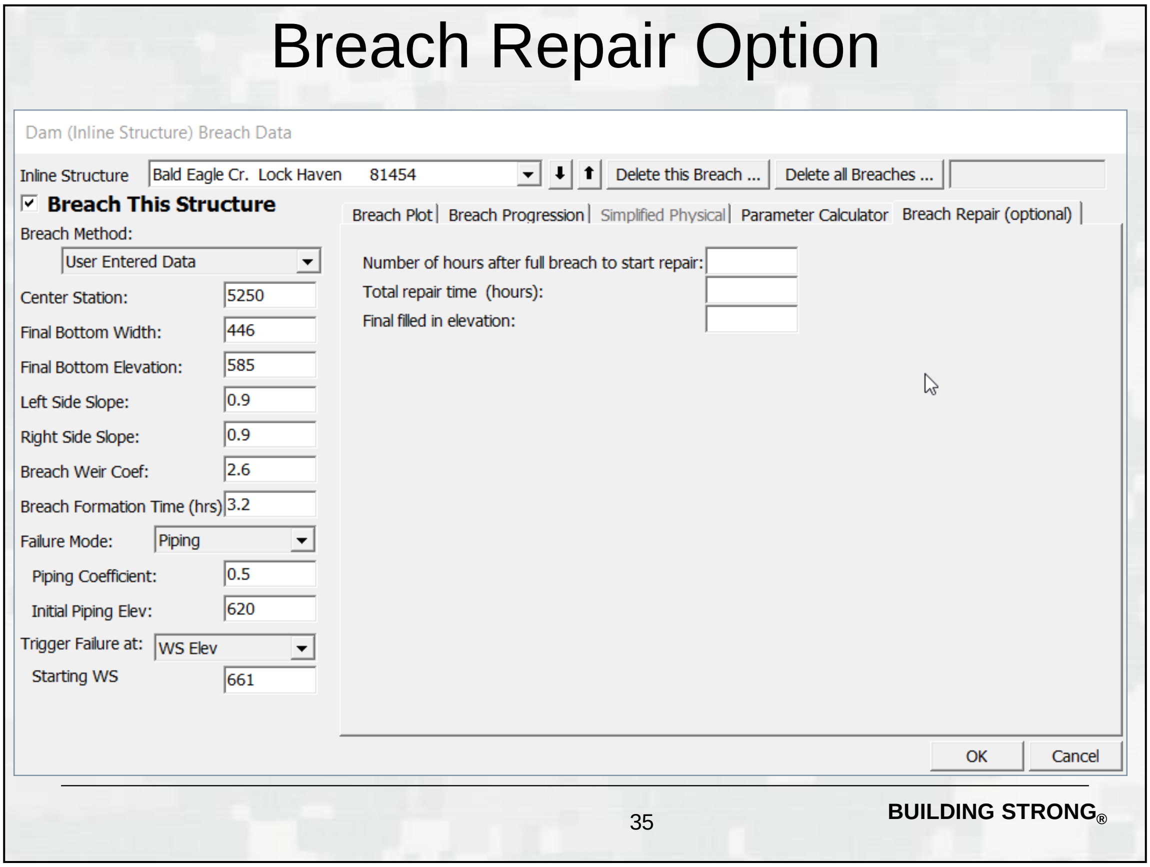
Start Time – If starting date and time is the failure trigger mode, then this field must be entered. This field is used to enter the time at which the breach will begin to occur.

Piping Failure Breach Growth Geometry



A piping breach starts out as a point—at the centerline station and elevation as specified by the user. Its growth is modeled as a rectangle.

When the top of the piping breach is greater than the water surface, it is assumed that the levee sloughs and the breach is modeled as an open breach. The side slopes are gradually increased until they reach the user specified values.



This option allows the user to have the breach fill back in during a simulation. This would most often be used for levee breaches, but could also be used for a dam breach if the user were running a long term simulation or if it was assumed that some effort would be put in place to fill a breach back in during a failure.

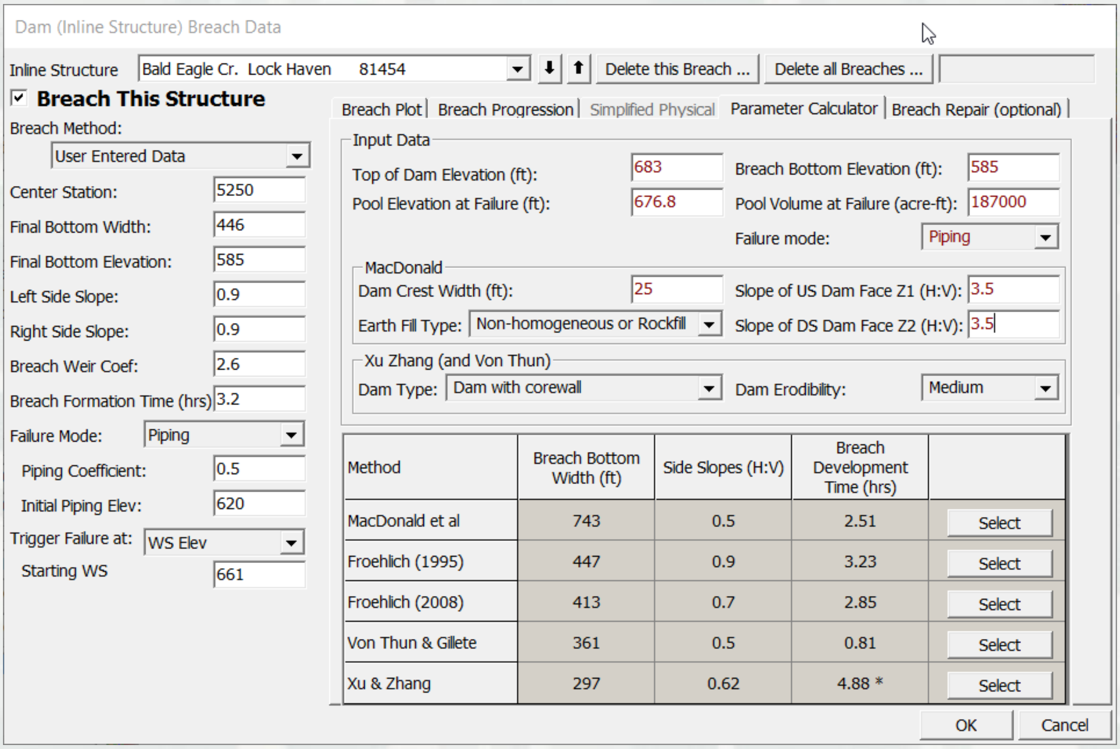
The Breach Repair Option requires the user to enter three pieces of information:

Number of hours after full breach to start repair: This field is used to enter the amount of time (in hours) it takes to start the repair process after the breach has occurred.

Total repair time (hours): This field is used to enter the total amount of time that it will take to perform the breach repair, in hours.

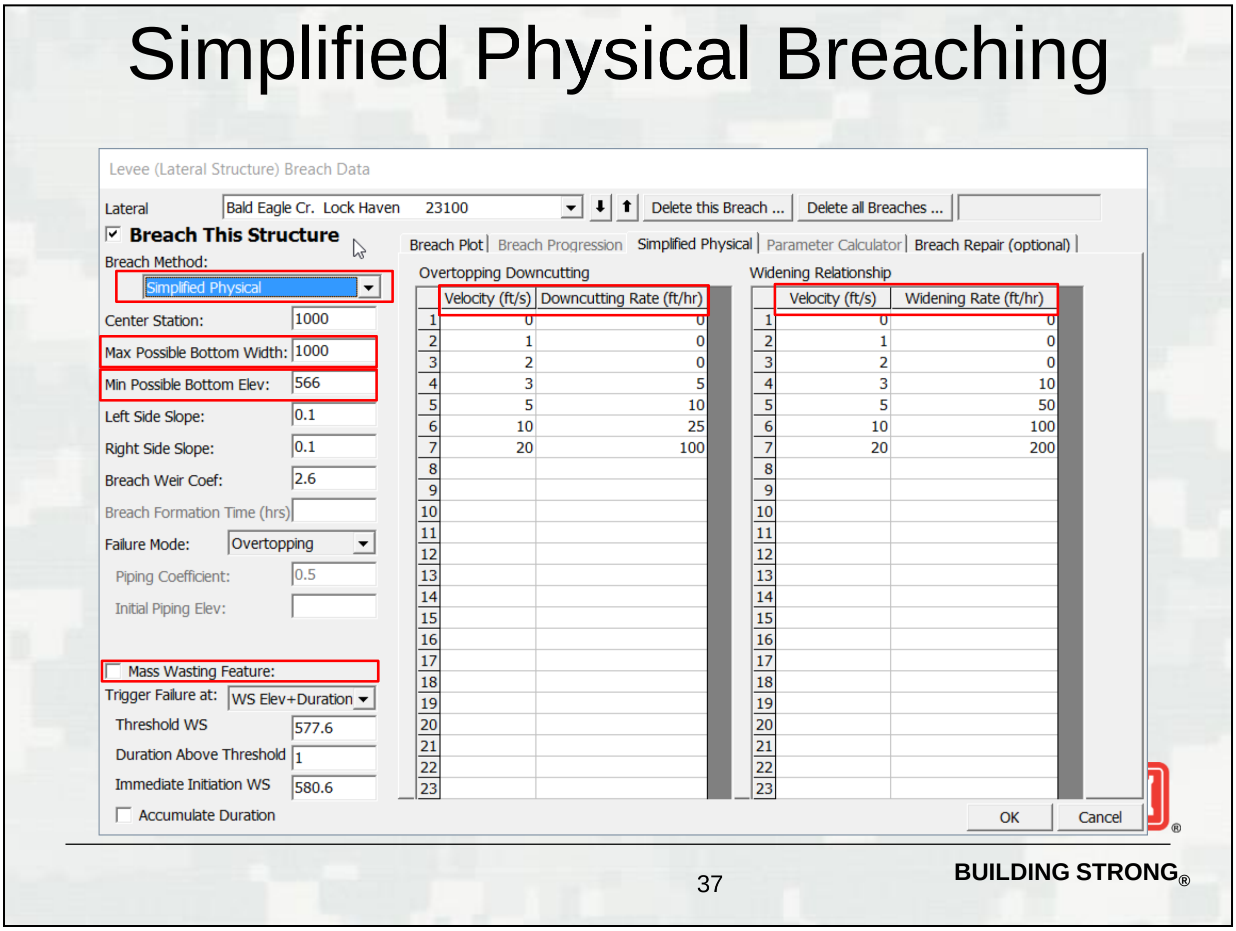
Final filled in elevation: This field is used to enter the top elevation of the final repaired breach.

Breach Regression Equation Calculator



To assist users in estimating the Breach dimensions and development time, HEC has added a “Parameter Calculator” to the Breach Data editor. To use this calculator select the **Parameter Calculator** Tab from the breach editor, and the editor will look like the screen above.

As shown in the Figure above, the Breach Parameter Calculator contains five regression equations (MacDonald\Langridge-Monopolis; Froehlich 1995; Froehlich 2008; Von Thun and Gillete; and Xu & Zhang 2009). The user is required to enter several parameters that describe the Dam and the volume of water behind the structure at the time of failure. These parameters include: Top of Dam elevation; Breach Bottom Width; Pool elevation at Failure; Pool Volume at Failure; Failure Mode; Dam Crest width, upstream and downstream embankment slopes; Earth Fill Type; Dam Type; and Dam Erodibility factor. Not all values are used for all regression equations as noted on the editor. Some of the variables are specific to the MacDonald equation, and the last two are specific to the Xu & Zhang equation. Once the values are entered the calculator computes Breach Bottom Width; Side Slopes; and Breach Development Times from each of the regression equations. The user can then select the answers from one of the equations to by pressing the **Select** button next to the equation results that they would like to use.



Once the User selects “Simplified Physical” breaching option, there are several fields in which labels change, some additional information required, and some previous information that is not required. The main changes between this method and the “User Entered Data” breach method, are the following:

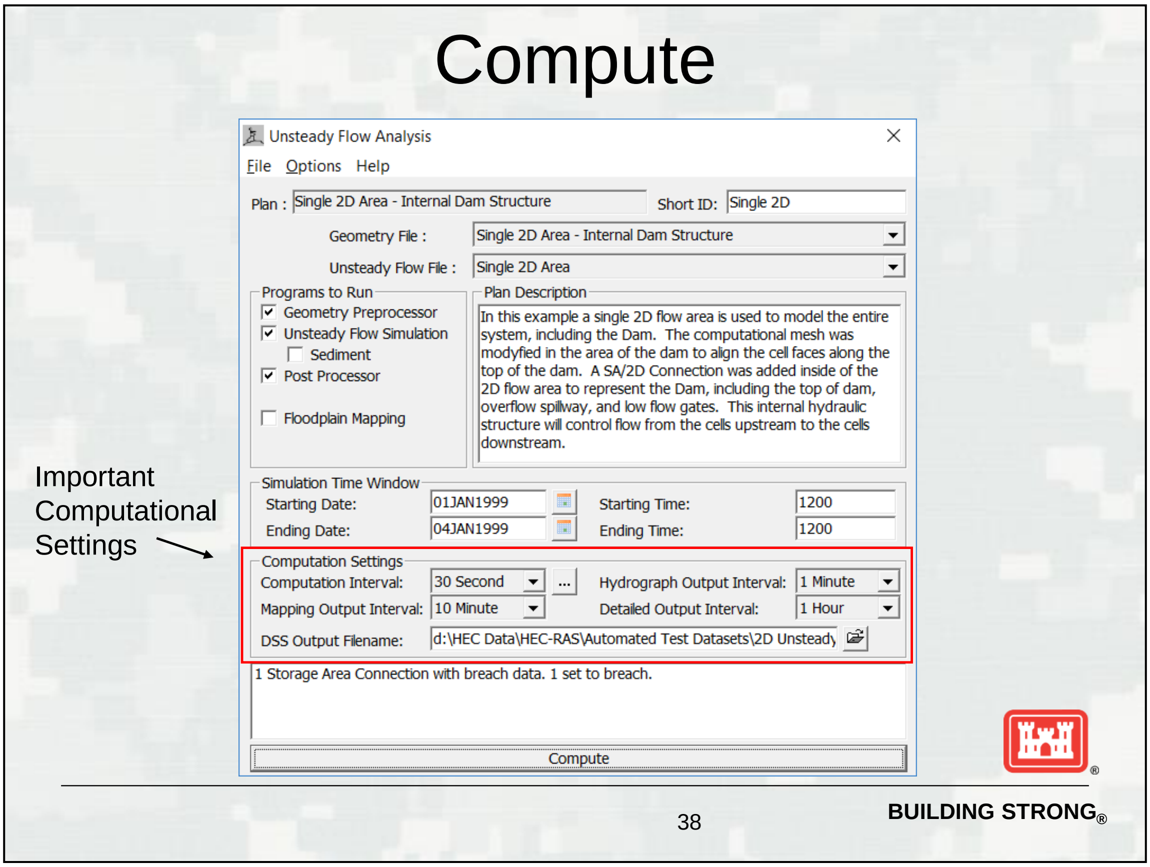
Max Possible Bottom Width – This field is now used to enter a maximum possible breach bottom width. This does not mean this will be the final breach bottom width, it is really being used to limit the breach bottom width growth to this amount. The actual bottom width will be dependent on the velocity verses erosion rate data entered, and the hydraulics of flow through the breach. This field is used to prevent breaches from growing larger than this user set upper limit during the run.

Min Possible Bottom Elev – This field is used to put a limit on how far down the breach can erode during the breaching process. This is not necessarily the final breach bottom elevation, it is a user entered limiter (I.e. the breach cannot go below this elevation). The final breach elevation will be dependent on the velocity verses erosion rate data entered, and the hydraulics of flow through the breach.

Starting Notch Width or Initial Piping Diameter – If the Overtopping failure mode is selected, the user will be asked to enter a starting notch width. The purpose of this is that the software will us this width at the top of the dam to compute a velocity, from the velocity it will get a down cutting erosion rate (based on user entered data), which will be used to start the erosion process. If a Piping Failure model is selected, the user must enter an initial piping diameter. Once the breach is triggered to start, this initial hole will show up immediately. A velocity will be computed through it, then the down cutting and widening process will begin based in user entered erosion rate data.

Mass Wasting Feature – This option allows the user to put a hole in the Dam or the Levee at the beginning of the breach, in a very short amount of time. This option would probably most often be used in a levee evaluation, in which a section of the levee may give way (Mass Wasting), then that initial hole would continue to erode and widen based on the erosion process. The required data for this option is a width for the mass wasting hole; duration in hours that this mass wasting occurs over (this would normally be a short amount of time); and the final bottom elevation of the initial mass wasting hole (it is assumed that the hole is open all the way to the top of the levee or Dam if this option is used).

Velocity vs. Downcutting and Widening Erosion Rates. When using the “Simplified Physical” breaching option, the user is required to enter velocity versus Downcutting erosion rates, as well as velocity versus erosion widening rates. To enter this data the user selects the “Simplified Physical” breach Tab.



When performing a Dam Break simulation the user must pay close attention to the Computational Setting that they apply.

Computational Interval: This is probably the most important parameter that will be entered by the user. The computational Interval will generally need to be very small when performing a dambreak analysis, on the order of 1 second to 1 minute. Consideration needs to be given to cross section spacing and 2D Cell size, as well as flood wave velocity, when choosing this parameter.

Hydrograph Output Interval: This interval controls the time interval at which the flow and stage hydrographs will be written out at. This should generally be small in order to capture the peak outflow coming out of the dam, as well as at various locations downstream. Generally around 5 to 15 minutes.

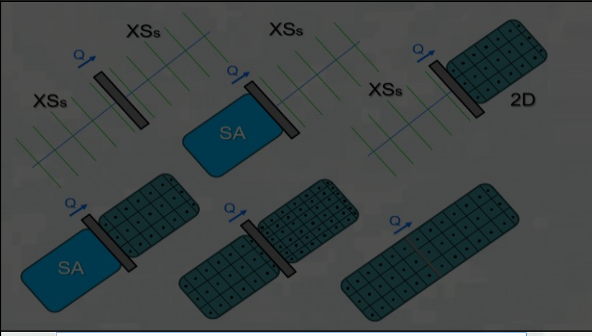
Detailed Output Interval: This is the interval at which detailed post processing of model output will be computed at, it will also control what snapshots in time can be sent to the GIS for inundation mapping. This is also the interval at which cross section and profile graphics can be viewed with animation of the results.

Mapping Output Interval: This is the interval that will be used by RAS Mapper for performing Dynamic Mapping of the flood inundation area.

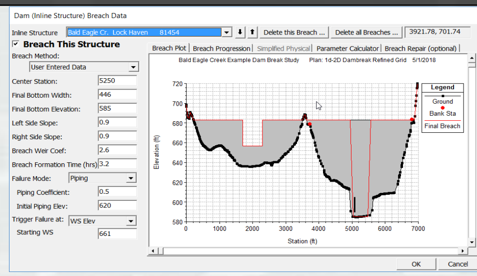
Computation Level Output: This option instructs the model to write out stage and flow at all locations for every computational time interval to a binary file. This should only be used as a tool for debugging model stability problems. **Turning this option on could generate a huge output file depending on the size of your model and the duration of the simulation.**

Mixed Flow Regime Analysis: This option allows the model to run in a mixed flow regime mode. This option should only be used in steep stream areas and/or areas where you have supercritical flow in the middle of the simulation.

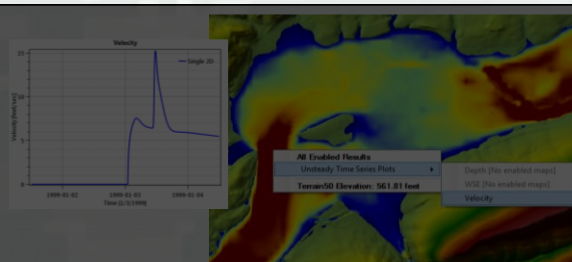
1. Six Dam Breach Model Configurations



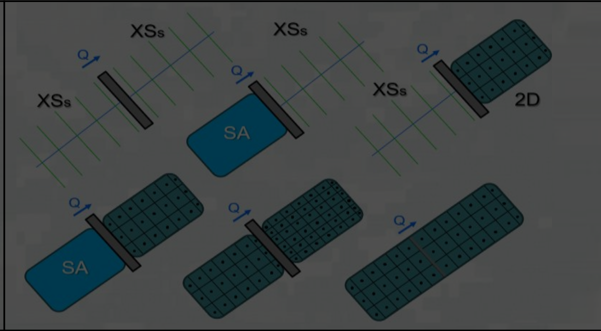
2. Breaching Options and Parameters



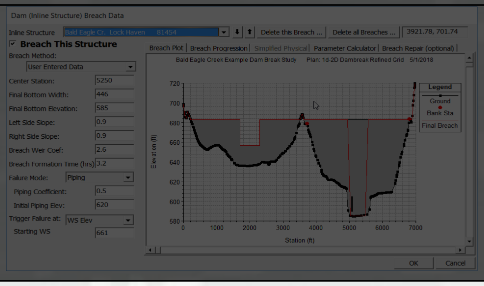
3. Breach Results and Visualization



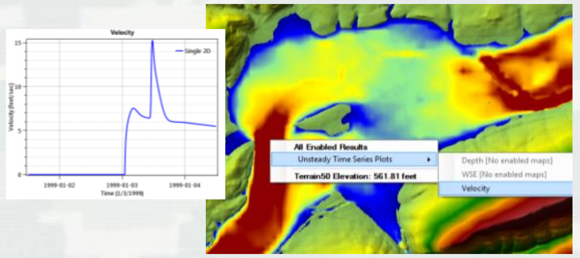
1. Six Dam Breach Model Configurations



2. Breaching Options and Parameters



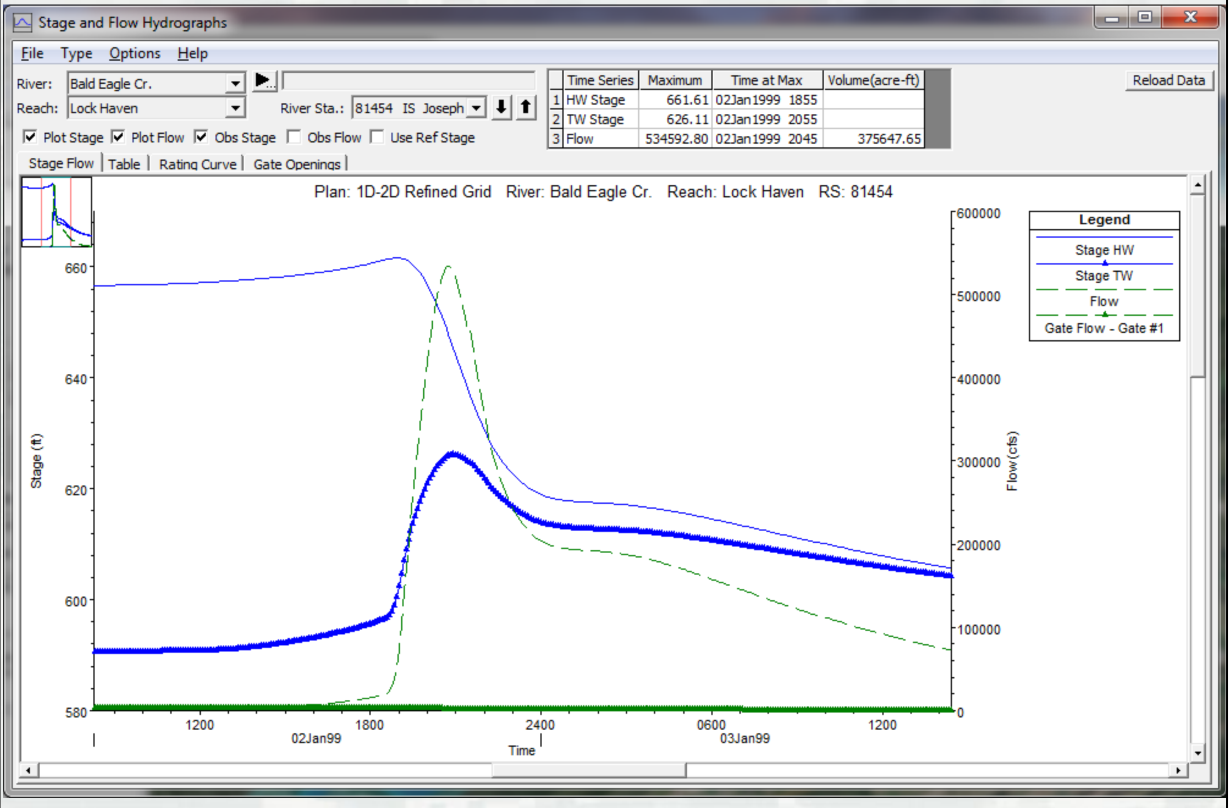
3. Breach Results and Visualization



Breach Results and Visualization

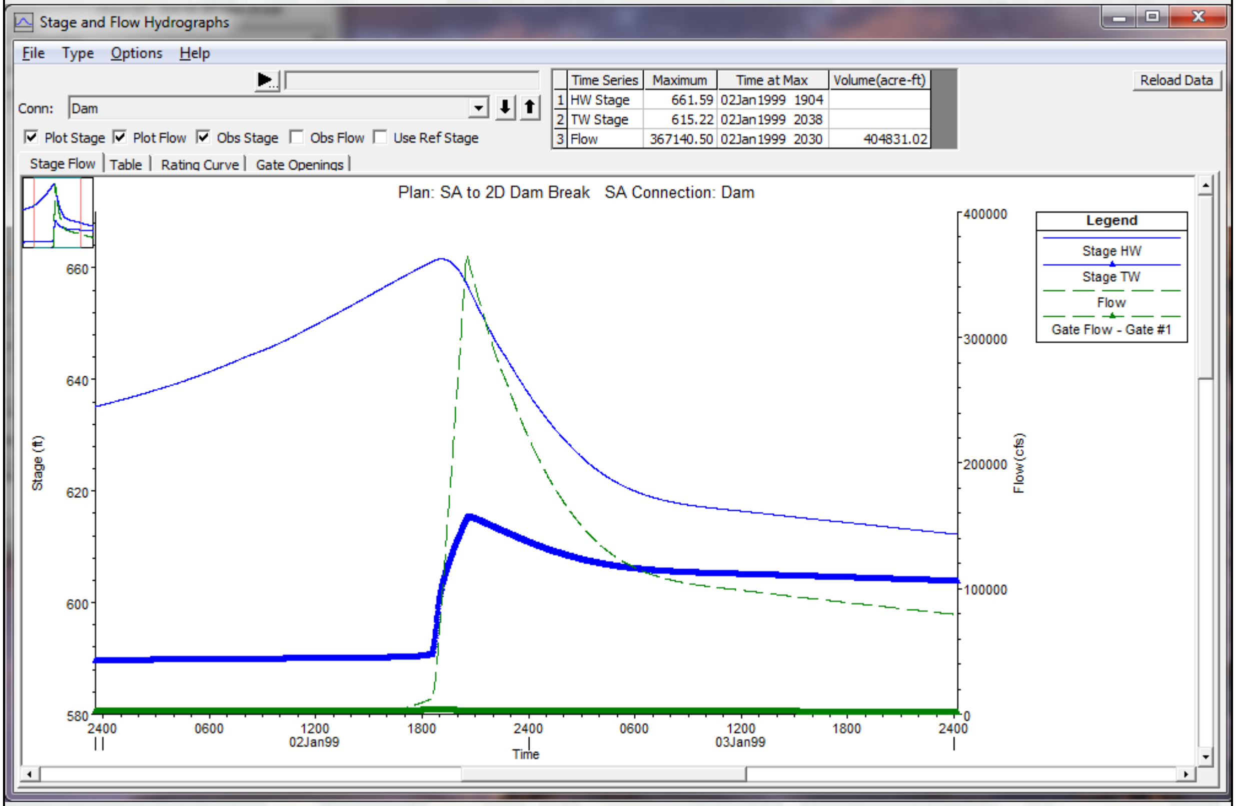
- Hydrographs at Inline Structures, SA/2D Hydraulic Connections, and Storage Areas
- Profile Plots for 1D Reaches
- Flow Hydrographs at 2D Area Boundaries
- Inundation Maps/Animations in RAS-Mapper
- Stage Hydrograph Plots in RAS-Mapper
- Velocity Time Series in RAS-Mapper

Inline Structure Hydrograph



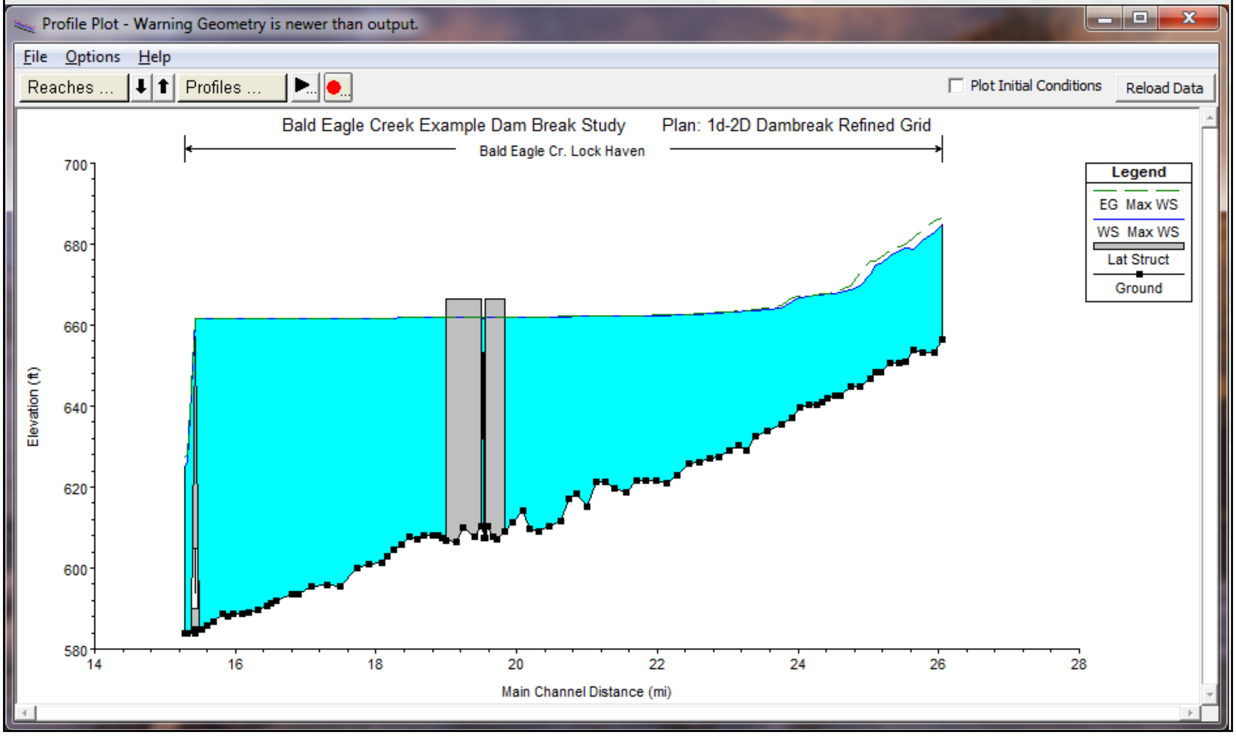
This is a hydrograph plot from an Inline Structure that was used to model the Dam in the scenario where a 1D reach was used to model the reseroir pool and the Dam, then connected to a 2D Flow Area downstream.

SA/2D Conn Hydrograph



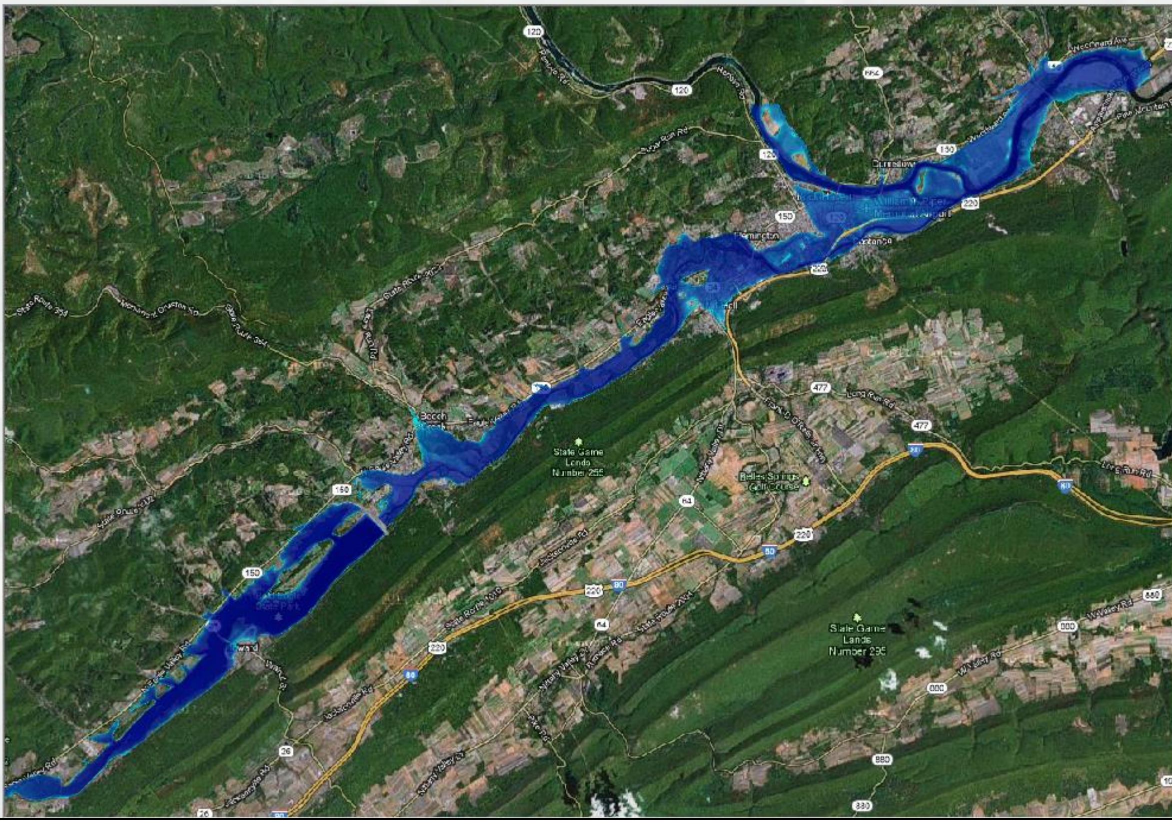
This is a hydrograph plot from a SA/2D Hydraulic Connection used to model a dam. Shown in blue are the Head water (reservoir pool) elevations and tailwater elevations (Downstream of the dam in the 2D Flow Area. The Tailwater location is based on the lowest outlet in the dam (Lowest flow movement point from the overflow weir, gates, culverts, etc…). So the Tailwater elevation is just showing the water surface elevation for a single cell. Keep in mind that headwater and tailwater are evaluated at every cell location along the dam seperately.

1D River Reach W.S Profile Plots



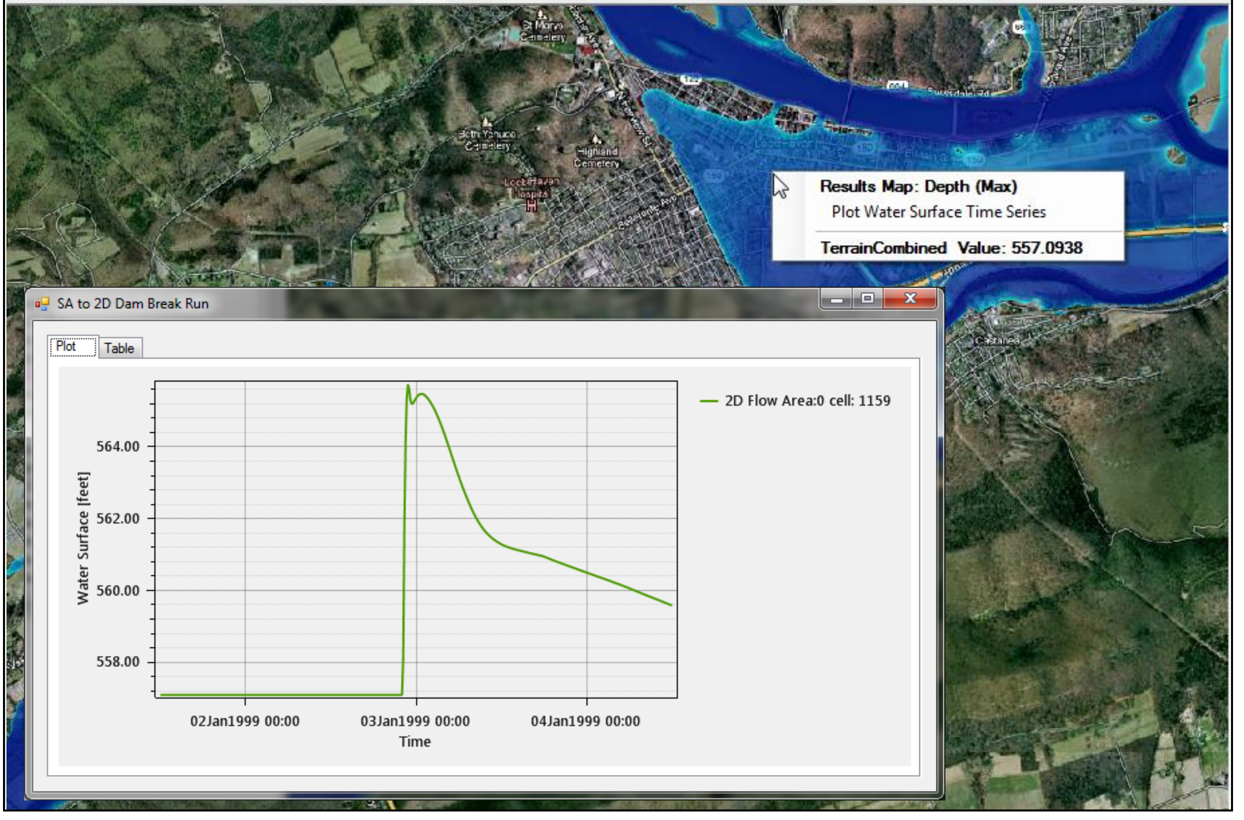
This is just a profile plot from a 1D river reach representing the upstream pool and dam, plus two cross sections downstream. This reach was then directly connected to a 2D Flow Area for the remainder of the downstream area.

Inundation Maps



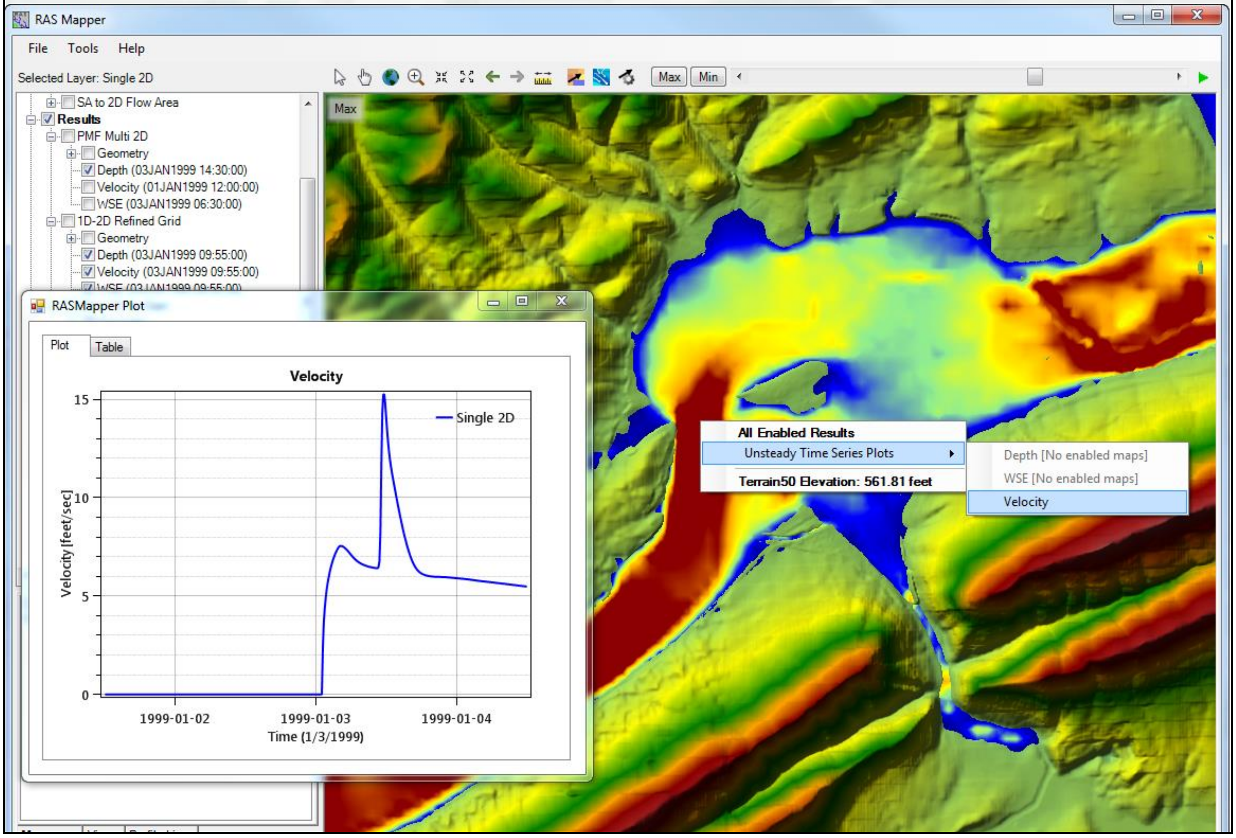
As you know by now, you can use RAS-Mapper to visualize inundation in both a Dynamic (Animations) and static (Stored maps) approach. You can also display water surface elevations, velocities, shear stress, arrival time grids, and flood duration grids.

Stage Hydrographs – RAS-Mapper



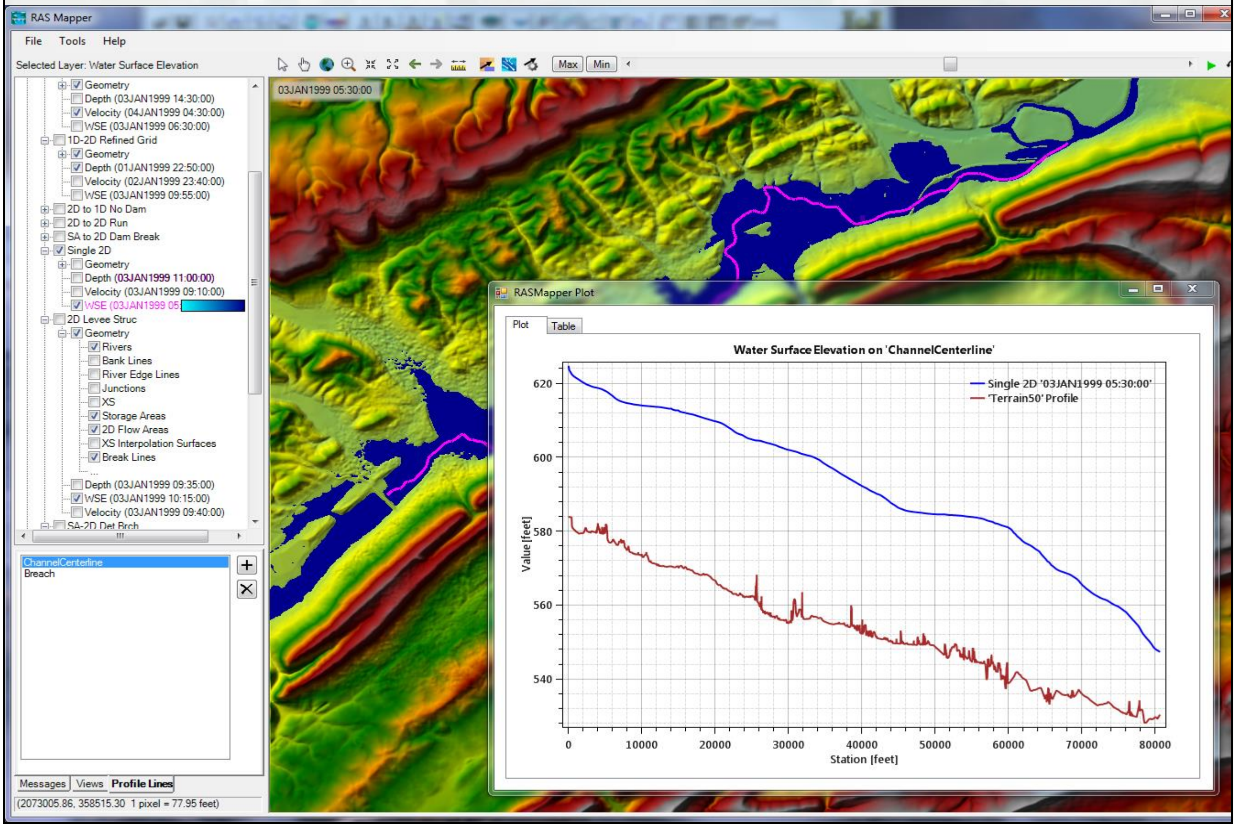
Also in RAS-Mapper, you can right click on an inundation map and request a Water Surface Time Series plot for that point location.

Velocity Time Series – RAS-Mapper



If you turn on the 2D Flow Area grid, you will also see options for plotting the Cell water surface time series; cell depth time series; 2D Face Point Velocity time series; 2D Face Velocity time series; 2D Face Shear time series; and the hydraulic property tables of the cells and faces.

Profile Lines – RAS-Mapper



You can also setup user defined profile lines. Then you can plot a profile of the data currently being displayed in RAS Mapper.