

Boundary and Initial Conditions for 2D Modeling

Alex Sánchez, PhD

Senior Hydraulic Engineer

USACE, Institute for Water Resources, Hydrologic Engineering Center



US Army Corps
of Engineers®



1



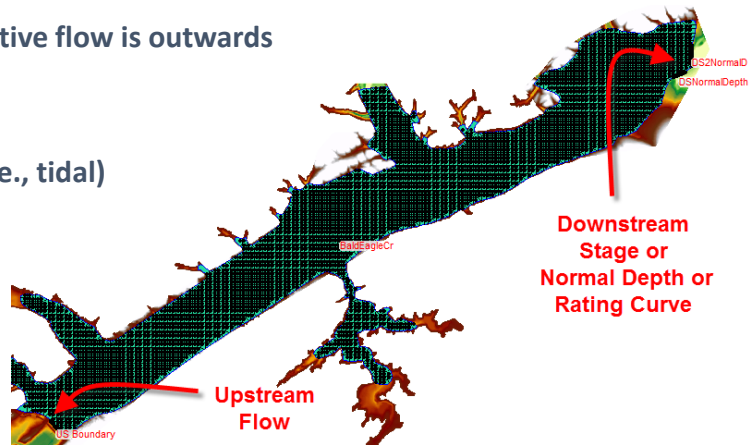
Overview

- Boundary Conditions (BC)
 - External
 - Internal
 - Global
- Initial Conditions (IC)
 - User-Specified
 - Ramp Up and Warm Up Periods
 - Restart File
 - Interpolate from Previous Results



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)

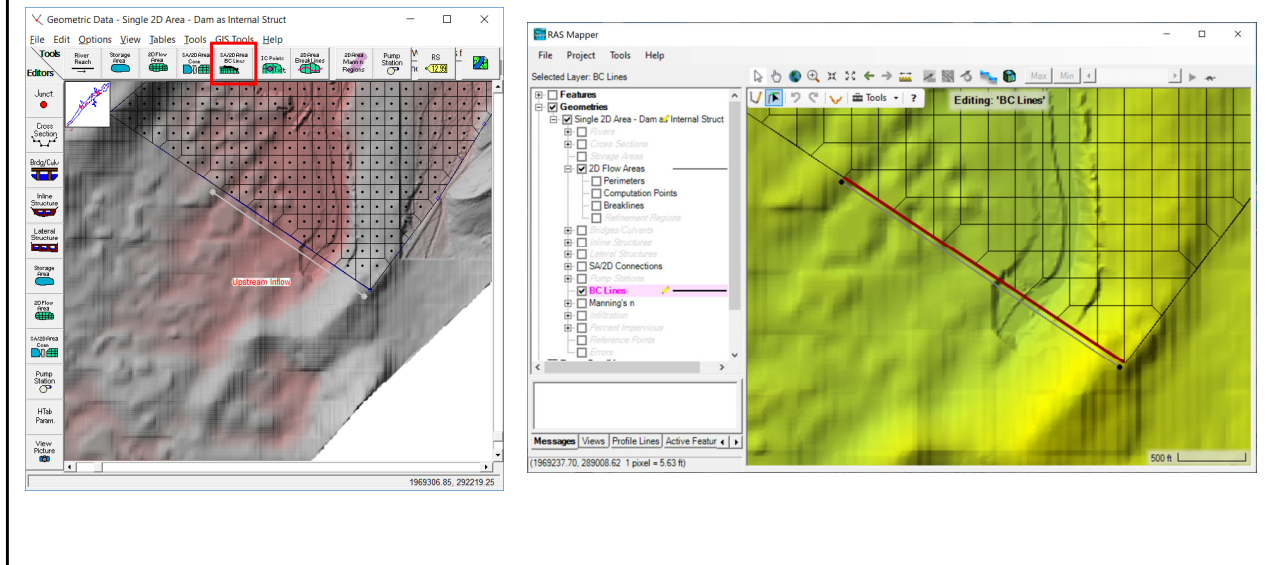


There are 4 boundary types:

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.



Creating External Boundary Condition Lines

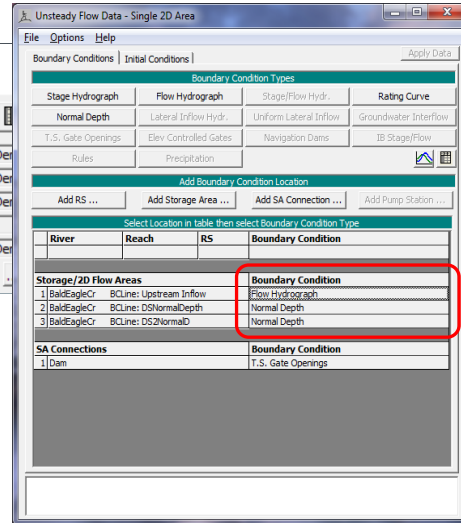
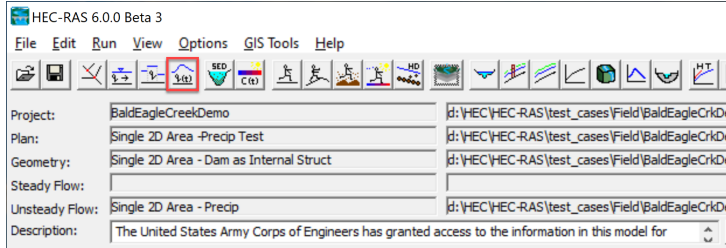


To create a 2D boundary:

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.



External Boundary Condition Data



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



Flow Hydrograph & EG Slope



Flow Hydrograph

SA: 2D Area BCLine: BC Upstream (Upstream name)

Read from DSS before simulation

File:
Path:

Enter Table

Select/Enter the Data's Starting Time Reference

Use Simulation Time: Date: 01JAN1999 Time: 1200
 Fixed Start Time: Date: Time:

Hydrograph Data			
	Date	Simulation Time (hours)	Flow (cfs)
1	01Jan1999 1200	00:00	400
2	01Jan1999 1201	00:01	416.67
3	01Jan1999 1202	00:02	833.33
4	01Jan1999 1203	00:03	1250

Time Step Adjustment Options ("Critical" boundary conditions)

Monitor this hydrograph for adjustments to computational time step

Max Change in Flow (without changing time step):

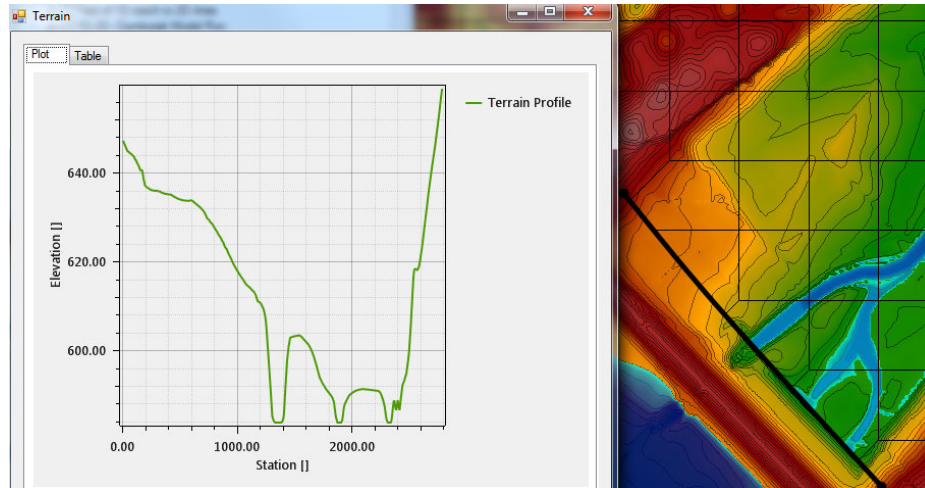
Min Flow: Multiplier: EG Slope for distributing flow along BC Line: 0.1 TW Check

- Time series can be specified from DSS file or entered as a table
- **EG Slope** required to compute normal depth
- Flow distributed along boundary based on conveyance
- **TW Check** option uses higher of TW and normal depth
- Face velocities computed for shallow water equations

- The flow hydrograph data can either be entered directly in the table or it can be read from DSS.
- The user is required to enter an energy slope that is associated with the flow boundary. This slope is used to compute normal depth based on the profile of the boundary faces and the flow for the given time step.
- If the optional TW Check is turned on, the water surface computed from normal depth will be compared to the current water surface in the 2D area next to the boundary. Unsteady will use the higher of these two water surfaces.
- Once the water surface is determined, the flow will be distributed along the boundary. If the user has selected to use the full momentum equation, the velocity associated with the boundary water surface will be transferred into the 2D area. So for example, **if the EG slope is steep (e.g. 0.1) and the TW Check is off, there can be a "jet" of water shooting into the 2D from the boundary.** If the TW Check is on, the velocity at the boundary will drop as the water depth inside of the 2D increases.



Flow Distribution along Boundary



- Flow is distributed to the appropriate cells based on EG Slope Conveyance or actual water surface

- If the 2D area starts dry, and the flow boundary crosses more than one cell (which it usually will), a methodology is needed to decide which cells get flow. Putting all of the flow into the cell with the lowest invert would be simple, but this can lead to poor results and instability at higher flows. Distributing the flow uniformly among all of the cells can lead to flow high on the “valley walls” which also gives poor results and/or instability.
- So instead, the user entered energy slope is used to compute normal depth and distribute the flow between the boundary cells. The flow is determined at the start of each time step. Using this flow, the user entered energy grade slope, and the conveyance properties of the boundary faces (station/elevation data and Manning’s n), a normal depth flow distribution is computed. This face conveyance is then used to distribute the flow among the boundary faces in the 2D area.
- Once there is a sufficient water surface inside of the 2D area along the flow boundary, Unsteady will use this water surface if it is higher and the TW Check is on.
- A steep energy slope without the TW Check can cause the water to shoot into the 2D area. A very shallow energy slope will always be higher than the tailwater and can cause flow to appear up on the sides of the valley that should actually be dry.
- **In general, it is recommended to use a steep energy slope and have the TW Check turned on.**



Stage Hydrograph



- DSS or Table
- **Use Initial Stage** option applies a horizontal water level from the boundary inwards
- All wet faces at the boundary assigned the same value
- Stage specified at boundary faces

Unsteady Flow Data - 2D Lower

File Options Help

Boundary Conditions | Initial Conditions | Apply Data

Boundary Condition Types

Stage Hydrograph | Flow Hydrograph | Stage/Flow Hydr. | Rating Curve

SA: BaldEagleCr BCLine: DS Stage

Read from DSS before simulation

Select DSS File: Path: Boundary Condition Location: Add SA Connection ... Add Pump Station ...

Enter Table

Select/Enter the Data's Starting Time Reference

Use Simulation Time: Date: 01JAN1999 Time: 1200

Fixed Start Time: Date: Time:

Data time interval: RS Boundary Condition

No. Ordinates Interpolate Missing Values Del Row Ins Row

No.	Date	Simulation Time (hours)	Stage (ft)
1	01Jan1999 1200	00:00	536.13
2	01Jan1999 1300	01:00	537.07
3	01Jan1999 1400	02:00	536.86
4	01Jan1999 1500	03:00	537.53
5	01Jan1999 1600	04:00	538.18
6	01Jan1999 1700	05:00	539.73
7	01Jan1999 1800	06:00	540.38
8	01Jan1999 1900	07:00	540.77

Use Initial Stage (recommended) Plot Data OK

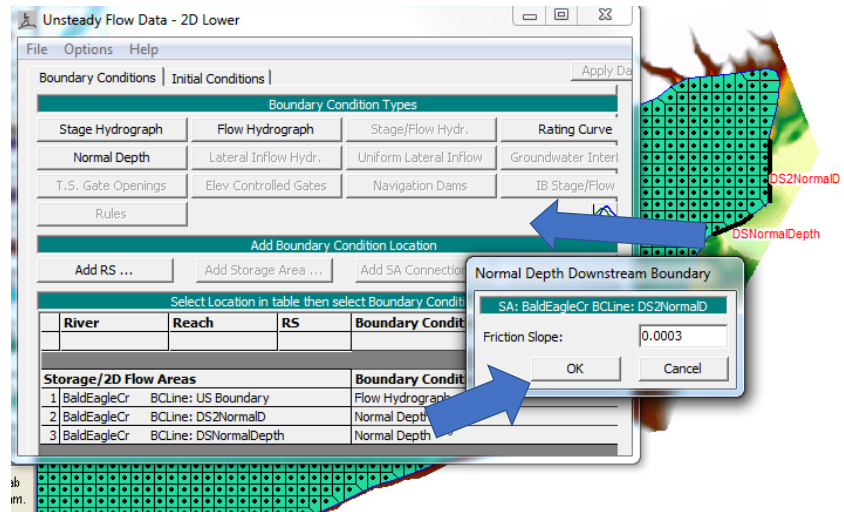
- The stage hydrograph has an option to Use Initial Stage (red circle). If this is selected, the initial stage from the stage hydrograph (536.13 in this example) will be projected into the 2D area at the start of the initial conditions for this 2D area. This option is generally recommended because it will allow the starting conditions in the 2D area to be established more quickly.
- However, if the user wants the 2D area to start dry, then this option should not be selected.
- There may be other situations where this option should NOT be used. For instance, if the stage boundary is at the “upstream” end of the 2D area, the projected water surface could inappropriately flood the whole area.
- All of the wet faces along the boundary are assigned the same water surface elevation.
- Internal cell values will usually be very close to boundary value but necessarily the same.



Normal Depth



- Only for outflow
- Friction slope constant for entire boundary
- Manning's Eq. used to compute flow per face
- Boundary stage can vary
- Should be reserved for situations where no other data is available
- Placed away from project area



In the case of the normal depth boundary condition, the user-specified friction slope is assumed constant for the whole boundary and is utilized to compute a flow for each face.

The flow is computed using Manning's equation as:

$$Q = A V$$

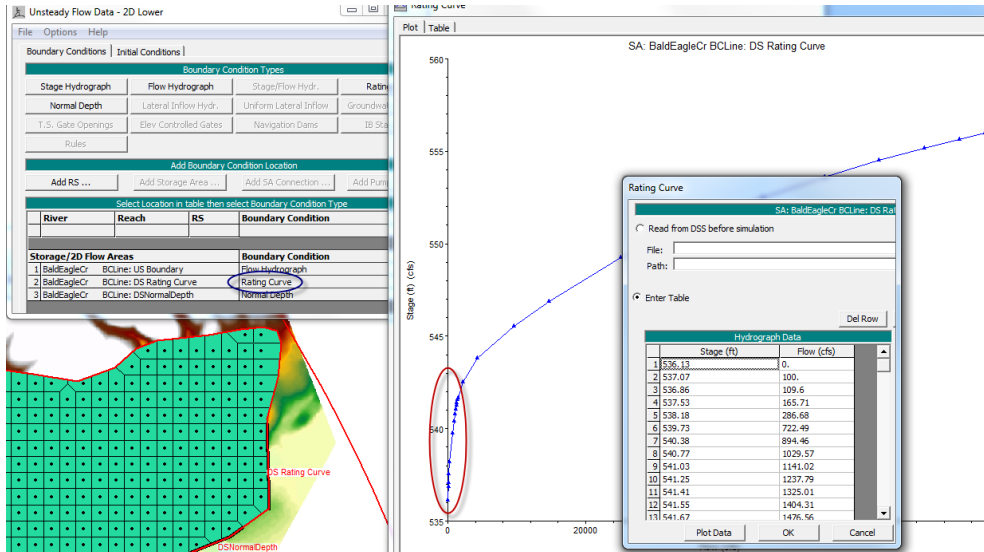
$$V = 1/n R^{2/3} \text{sqrt}(S_f)$$

where

- A : Face area [L²]
- V : Velocity [L/T]
- n : Manning's roughness coefficient [1/L^{1/3}]
- R : Hydraulic Radius [L]
- Sf : Friction slope [-]



Rating Curve



- To compute the flow at the Rating Curve boundary, first, an average (conveyance weighted) water surface at the boundary cells is computed. Next, a flow is then determined based on this water surface and the user entered rating curve. Finally, the flow (that is being removed from the 2D area) is distributed among the boundary cells based on the conveyance along the boundary.
- Rating curves tend to be very steep at the beginning (for low flows), as shown above. This can cause stability problems. A sharp transition in the curve (often caused by having too few of points) can also create stability problems.



Rating Curve Considerations

- Flow based on conveyance-averaged WSE
- Flow distribution based on conveyance
- Watch out for a steeply sloped curve and/or sharp transitions in the curve
- Watch out for “bad” low flow curve
- Zero flow point on Rating Curve does **NOT** have to be at invert (could be higher)
- Can have initialization problems (when RC is not consistent with cold-start conditions)

- Since 1D reaches (in RAS) are not allowed to go dry, some 1D rating curves do not have “real” data at the beginning of the curve. (Be careful using rating curves from the 1D model).
- A 2D Area that is starting dry may have problems with the initial, very low flows because of the shape of the rating curve. Either because the curve is inaccurate and/or it is too steep with sharp transitions.
- Therefore, it may be desirable to adjust the starting shape of the curve, this is especially true if the 2D area is starting dry but there is no need to get model results for the dry/extremely low flow condition. (For instance the model starts dry, but in reality the river channel would always have flow.)
- One option is to put the zero flow elevation a foot or two above the invert. This would allow a couple of feet of water to accumulate at the boundary before any water is allowed to flow out. This will improve the stability when a flow is finally computed.
- If initializing a model with a cold start and a high water level, the RC will immediately produce a large flow and “suck” too much water out of the domain, lowering the water levels close to the boundary. This produces an artificial draw down near the boundary.



External Boundary Locations

- Extend boundary condition locations away from study area of interest
 - **Normal Depth is an approximate boundary**
 - **Stage creates a horizontal WSE along the boundary cells (place normal to flow)**
 - **Rating curve does not take into for changes in flow for rising and falling-limbs of hydrograph**



Boundary Output



The screenshot displays the HEC-RAS 5.0.0 software interface. The main window is titled "Geometric Data - Single 2D Area - D...". The "Tools" menu is open, and the "Stage and Flow Hydrographs" option is circled in red. A context menu is visible over a boundary line in the 2D area, with the "Plot Stage and Flow Hydrograph ..." option selected. The "Stage and Flow Hydrographs" window is open, showing a plot of "Flow (cfs)" versus "Time" for the boundary line "DS2NormalD". The plot shows a flow rate that increases from approximately 25,000 cfs at 0200 to about 225,000 cfs by 0400, then levels off. The y-axis ranges from 0 to 250,000 cfs, and the x-axis ranges from 0200 to 1200. The plot title is "Plan: Single 2D BCLine: DSNormalDepth".

The stage and flow from a 2D boundary is part of the normal stage and flow output. This output can be selected by either clicking on the **Stage and Flow** button on the main RAS GUI or by clicking on the boundary line on the Geometric Editor.



Internal Boundary Conditions

- Flow Hydrographs
 - Must be completely inside of the 2D Flow Area
 - Can have positive and/or negative flows
 - No flow direction; Flow is only in Continuity Equation and not Momentum Equations
 - Flow is allocated to each cell based on length of line in cells
- Precipitation
 - Time-series applied to individual 2D areas
 - Precipitation rates constant for every cell
 - Specified as period cumulative depths

Precipitation creates an inflow to all of the cells in a 2D area.



Internal Flow Boundary Condition



The screenshot displays the HEC-RAS software interface. The main window shows a grid-based model of a river reach. A red rectangular box highlights a specific area within the grid, labeled "Internal Flow". The software interface includes a menu bar (File, Edit, Options, View, Tables, Tools, GIS Tools, Help), a toolbar with various modeling tools, and a left-hand panel with a tree view of the project components. The right-hand panel shows the "RAS Mapper" window, which displays a map of the model area with a red line indicating the "Upstream Inflow" and a red box indicating the "Internal Flow". The "RAS Mapper" window also includes a legend and a status bar at the bottom.



Internal Flow Boundary Condition



HEC-RAS 6.0.0 Beta 3

File Edit Run View Options GIS

Project: BaldEagleCreekDam
Plan: Single 2D Area - D...
Geometry: Single 2D Area - D...
Steady Flow:
Unsteady Flow: Single 2D Area - Precip
Description: The United States Army Co

Unsteady Flow Data - Single 2D Area - Precip

Boundary Conditions | Initial Conditions | Meteorological Data | Observed Data

Boundary Condition Types

- Stage Hydrograph
- Flow Hydrograph
- Stage/Flow Hydr
- Raking Curve
- Normal Depth
- Lateral Inflow Hydr
- Uniform Lateral Inflow
- groundwater Interflow
- T.S. Gate Openings
- Elev Controlled Gates
- Navigation Dams
- IB Stage/Flow
- Rules
- Precipitation

Add Boundary Condition Location

Add RS ... Add SA/2D Flow Area ... Add SA/2D Area Conn ... Add Pump Station ...

Select Location in table, then select Boundary Condition Type

River	Reach	RS	Boundary Condition
Storage/2D Flow Areas			
1	BaldEagleCr	BCLine: Internal Flow	Flow Hydrograph
2	BaldEagleCr	BCLine: Upstream Inflow	Flow Hydrograph
3	BaldEagleCr	BCLine: DSNormalDepth	Normal Depth
4	BaldEagleCr	BCLine: DS2NormalD	Normal Depth
5	BaldEagleCr		Precipitation
SA/2D Area Conns			
1	Dam		T.S. Gate Openings

Flow Hydrograph

SA: BaldEagleCr BCLine: Internal Flow

Read from DSS before simulation

File: Path:

Enter Table

Select the Data's Starting Time Reference

Use Simulation Time: Date: 09SEP2018 Time: 0000
 Fixed Start Time: Date: Time:

Data time interval: 1 Hour

No. Ordinates Interpolate Missing Values Del Row Ins Row

No. Ordinates	Date	Simulation Time (hours)	Flow (cfs)
1	08Sep2018 2400	00:00	
2	09Sep2018 0100	01:00	
3	09Sep2018 0200	02:00	
4	09Sep2018 0300	03:00	
5	09Sep2018 0400	04:00	
6	09Sep2018 0500	05:00	
7	09Sep2018 0600	06:00	
8	09Sep2018 0700	07:00	
9	09Sep2018 0800	08:00	
10	09Sep2018 0900	09:00	
11	09Sep2018 1000	10:00	
12	09Sep2018 1100	11:00	
13	09Sep2018 1200	12:00	
14	09Sep2018 1300	13:00	
15	09Sep2018 1400	14:00	

Time Step Adjustment Options ("Critical" boundary conditions)

Monitor this hydrograph for adjustments to computational time step

Max. Change in Flow (without changing time step):

Min Flow: Multiplier: EG Slope for distributing flow along BC Line: TW Check

Plot Data OK Cancel



Internal (2D Area) Precipitation BC



HEC-RAS 6.0.0 Beta 3

Unsteady Flow Data - Single 2D Area - Precip

Boundary Condition Types

Boundary Condition Types

Boundary Condition Types
Stage Hydrograph
Flow Hydrograph
Stage/Flow Hydro
Raking Curve
Normal Depth
Lateral Inflow Hydro
Uniform Lateral Inflow
Groundwater Interflow
T.S. Gate Openings
Elev. Controlled Gates
Navigation Dams
IB Stage/Flow
Routing
Precipitation

Add Boundary Condition Location

Add RS ... Add SA/ZD Flow Area ... Add SA/ZD Area Conn ... Add Pump Station ...

Select Location in table then select Boundary Condition Type

River	Reach	RS	Boundary Condition
Storage/2D Flow Areas			
1	BaldEagleCr		Precipitation
2	BaldEagleCr	BCLine: Upstream Inflow	Flow Hydrograph
3	BaldEagleCr	BCLine: DSNormalDepth	Normal Depth
4	BaldEagleCr	BCLine: DS2NormalD	Normal Depth
SA/ZD Area Conns			
1	Dam		T.S. Gate Openings

Precipitation Hydrograph

SA: BaldEagleCr

Read from DSS before simulation

File: Path:

Enter Table

Select/Enter the Data's Starting Time Reference

Use Simulation Time: Date: 09SEP2018 Time: 0000

Fixed Start Time: Date: Time:

Data time interval: 1 Hour

No. Ordinates Interpolate Missing Values Del Row Ins Row

Date	Simulation Time (hours)	Precipitation (in)
08Sep2018 2400	00:00	
09Sep2018 0100	01:00	
09Sep2018 0200	02:00	
09Sep2018 0300	03:00	
09Sep2018 0400	04:00	
09Sep2018 0500	05:00	
09Sep2018 0600	06:00	
09Sep2018 0700	07:00	
09Sep2018 0800	08:00	
09Sep2018 0900	09:00	
09Sep2018 1000	10:00	
09Sep2018 1100	11:00	
09Sep2018 1200	12:00	
09Sep2018 1300	13:00	
09Sep2018 1400	14:00	
09Sep2018 1500	15:00	
09Sep2018 1600	16:00	
09Sep2018 1700	17:00	
09Sep2018 1800	18:00	
09Sep2018 1900	19:00	

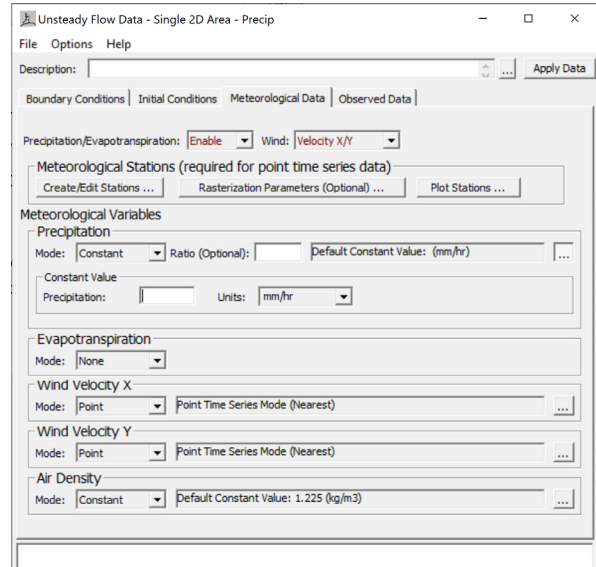
Plot Data OK Cancel



Global Boundary Conditions



- Applied to the entire project
- Specified in **Meteorologic Data** tab of **Unsteady Flow Data** editor
- Types of Boundaries:
 1. Precipitation
 2. Evapotranspiration Potential
 3. Wind
 4. Air Density
- Types of Data
 1. Point
 2. Constant
 3. Gridded



18

- Global Boundary Conditions are new to Version 6.0
- This is a really big topic that requires it's own lecture or several lectures.



2D Initial Conditions Overview

- **User Specified Initial Conditions**
 - Dry Initial Conditions
 - Single Horizontal WSE
 - Option to apply initial stage boundary values horizontally into domain
 - **2D Initial Condition Time** (Optional)
 - **Allows each 2D area to compute an initial profile**
 - **Somewhat similar to 1D initial backwater**
 - Optional entire model **Warm Up** period to settle simulation
- **Use Restart File**
 - Restart File created by previous run
- **Interpolate from Previous Results**
 - Interpolates wse, velocity, and flows for 1D and 2D
 - Geometries do not have to be the same

There are two options for starting a 2D RAS model:

Option 1: User-specified Initial Conditions

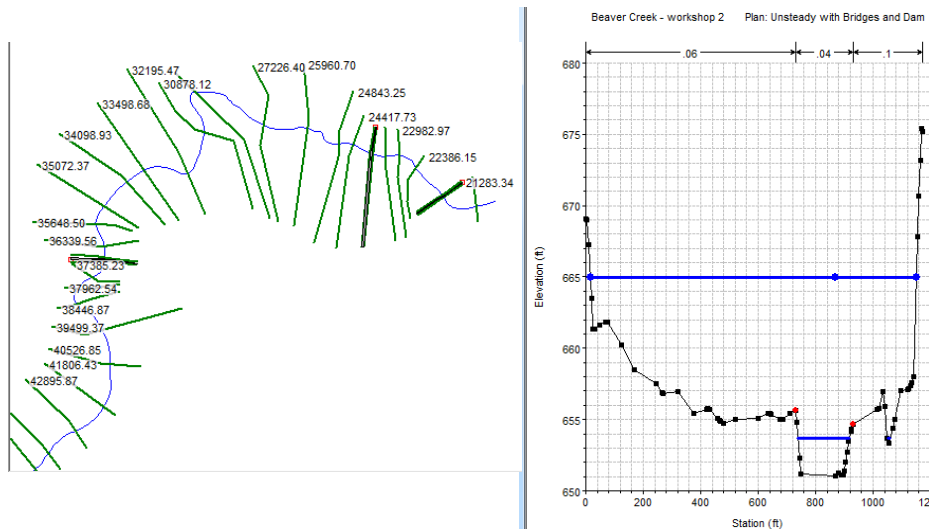
The model can use the default mode where the user specifies the initial conditions. This allows for an optional initial 2D water surface and an optional Initial Conditions period. There is also an optional warm up period that can be used to settle the simulation.

Option 2: Restart File

The model can use a Restart File. The restart file must have already been created by a previous run.



1D Finite-Difference Stays Wet



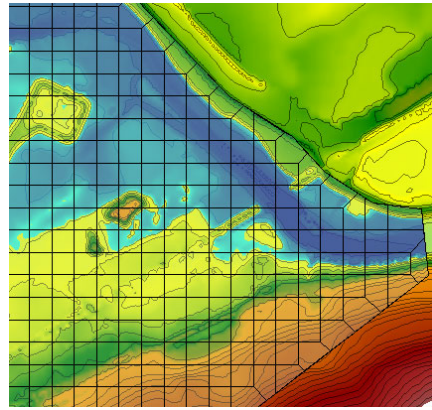
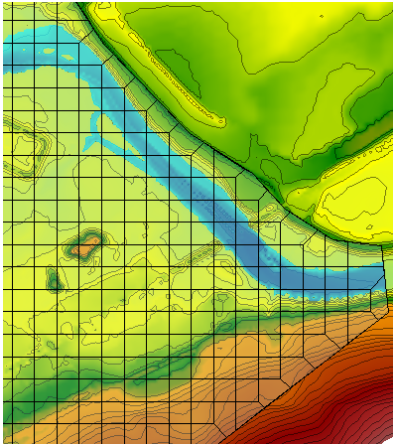
When the 1D RAS unsteady program computes the flow and stages for a given time step, it creates a system of equations that must be solved simultaneously by the unsteady solver. The current 1D solver does not allow for dry cross sections. All of the cross sections in the model must have water in them during the entire model run.

For an unsteady program that implicitly solves the full unsteady [Saint Venant] equation, allowing for wetting and drying has historically been a difficult task. For a 1D model, this was not a major limitation. An initial water surface profile can be computed using a steady-state model. For subcritical flow, this profile is determined from downstream to upstream using Manning's equation to compute the stage at each cross section one at a time.

This initial backwater solution can then be used as the starting condition for unsteady flow. As the flow varies over the model run, the stage in the cross sections will vary, but each cross section always has some water which means the unsteady solver never has to handle dry cross sections.



2D Wetting/Drying

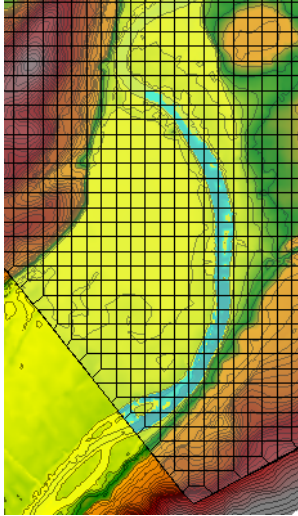




Filling 2D Channel



- Use Initial Conditions Time to fill channel



- Upstream flow will eventually fill channel
- May take a long time to fill, especially reservoirs
- Initial [horizontal] WSE can be used to speed things up
- A Restart File can save time

- Since the 2D does not have a steady-state solution, but it does have the ability to model wetting and drying, the easiest way to start up a 2D river channel is to start with a dry model and let it fill up by the using upstream flow.
- RAS refers to this part of the model solution as the Initial Conditions Time.
- For each 2D area, the user enters an Initial Condition time in hours. During this period flow will enter the 2D area from flow hydrographs and/or other external sources. If the Initial Condition time is long enough, the 2D profile should eventually settle down to a constant solution.
- It may take a long time to fill the model especially if there are any reservoirs.
- The user can enter a starting water surface to speed things up. This horizontal water surface is applied to all cells in the model, so care should be taken not to make it so high that it puts water in cells that should be dry. For instance, if there is a reservoir that goes to the downstream end of the model (and, for example, the dam is modeled as a storage area connector between two different 2D areas), the initial water surface could be entered as the pool elevation. However, if the 2D area extends below the dam, entering the pool elevation would inappropriately flood the area below the dam.
- For models that take a long time to run the Initial Conditions, a Restart File can save a lot of time.



Initial Conditions Ramp Up



- Ramp Up Period run before Warm Up Period
- Specified for each 2D area
- Ramp Up Fraction determines the period over which flow and stage are ramped up from the initial condition and then held constant

HEC-RAS Unsteady Computation Options and Tolerances

General | 2D Flow Options | 1D/2D Options | Advanced Time Step Control | 1D Mixed Flow Options

Use Coriolis Effects (not used with Diffusion Wave equation)

Parameter	(Default)	BaldEagleCr
1 Theta (0.6-1.0)	1	1
2 Theta Warmup (0.6-1.0)	1	1
3 Water Surface Tolerance [max=0.2](ft)	0.01	0.01
4 Volume Tolerance (ft)	0.01	0.01
5 Maximum Iterations	20	20
6 Equation Set	Diffusion Wave	Diffusion Wave
7 Initial Conditions Time (hrs)		
8 Initial Conditions Ramp Up Fraction (0-1)	0.5	0.5
9 Number of Time Slices (Integer Value)	1	1
10 Turbulence Model	None	None
11 Longitudinal Mixing Coefficient	0.3	0.3
12 Transverse Mixing Coefficient	0.1	0.1
13 Smagorinsky Coefficient	0.05	0.05
14 Boundary Condition Volume Check	<input type="checkbox"/>	<input type="checkbox"/>
15 Latitude for Coriolis (-90 to 90)		
16 Solver Cores	All Available	12 Cores
17 Matrix Solver	Pardiso (Direct)	Pardiso (Direct)
18 Convergence Tolerance		
19 Minimum Iterations	0	0
20 Maximum Iterations	0	0
21 Restart Iteration	10	10
22 Relaxation Factor	1.3	1.3
23 SOR Preconditioner Iterations	10	10

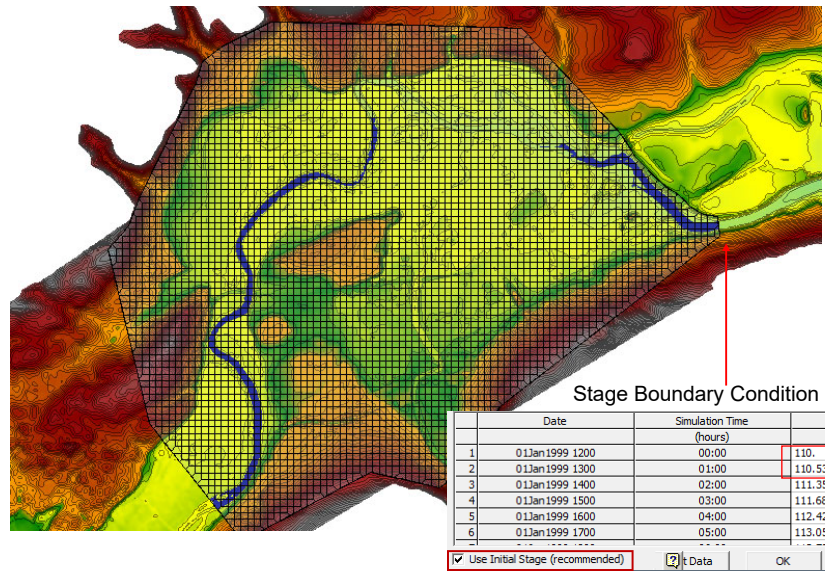
OK Cancel Defaults ...

23

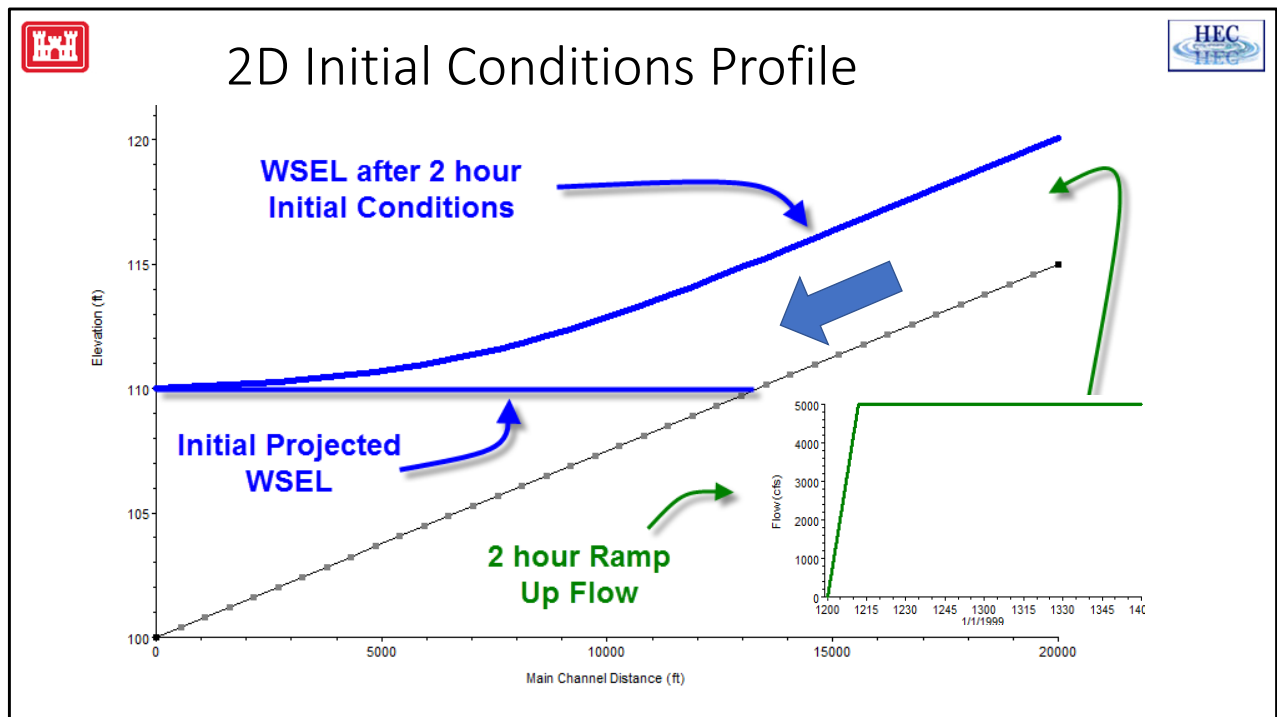
- Say, for instance, that a 2D area has an upstream flow boundary and a downstream stage boundary and the user has entered a 2-hr **Initial Conditions Ramp Up Time** with the **Boundary Fraction** at 0.5 (50%).
- Assume that the first flow on the flow boundary is 1000 cfs and the first stage of the downstream boundary has an elevation that corresponds to 10 ft of depth above the invert of the stage boundary (The invert is the lowest point along any part of the faces that make up the boundary).
- For the first hour of the initial conditions, the flow will increase linearly from 0 cfs up to 1000 cfs. The downstream stage boundary will transition from a depth of 0 ft up to a depth of 10 ft (and even though this is a “downstream” boundary, if the 2D area started out dry, then flow will initially come into the 2D area).
- For the second hour, the flow will held at 1000 cfs upstream and the depth at 10 ft downstream.



2D Initial Conditions Warm Up in Progress



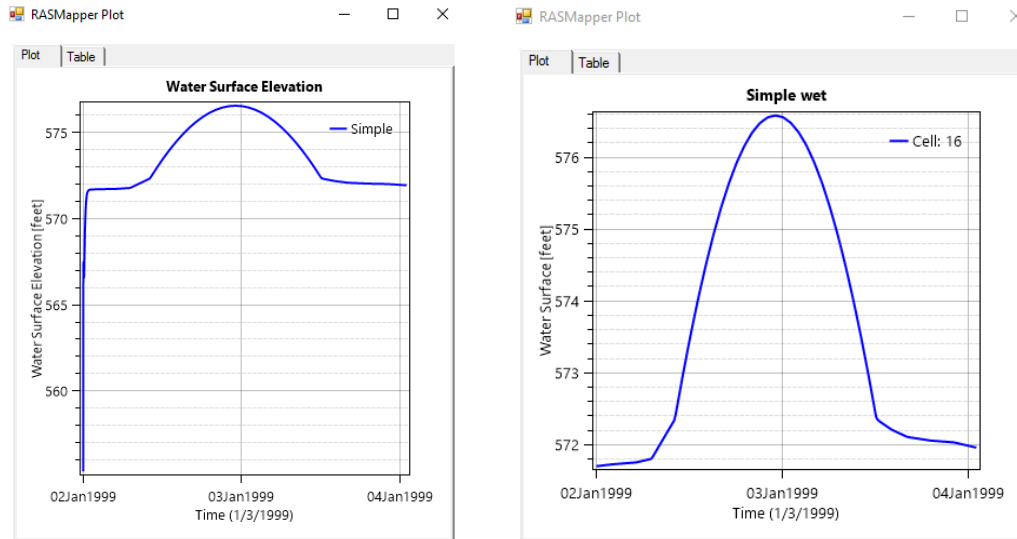
- For a model that has one or more 2D areas, but no 1D reaches, the Initial Conditions and Warm Up are similar. One difference is that only the Initial Conditions “ramps up” the flow. The second main difference is that the Warm Up period solves all of the 2D areas at once.
- The distinction between Initial Conditions and Warm Up is more important for models that incorporate both 1D reaches and 2D areas, as will be discussed in a separate lecture.
- In the slide above, the 2D area has an upstream flow hydrograph on the left hand side and a stage boundary downstream on the right hand side. The stage boundary has projected water a short distance up the channel. The Initial Conditions has been run long enough that the upstream flow has made it about halfway downstream. It should be noted that even after the upstream flow reaches the downstream water, additional Initial Conditions time (and/or Warm Up period time) will be required for the flow profile in the channel to settle down to equilibrium.



- The slide above shows a profile of a 2D channel that has a starting flow of 5000cfs, a starting stage boundary of 110 feet, a 2 hour Initial Conditions time and the default, 0.1 ramp up fraction.
- The lower part of the channel starts with water based on the starting stage being projected at an elevation of 110 feet. The flow starts at 0 cfs and goes up to 5000 cfs over the first 12 minutes and is held constant for the remaining 1 hour and 48 minutes. The higher blue line shows the profile after the initial 2 hours.



Dry vs Wet Start



The water surface time series shown above are from the same, small channel. The plot on the left is from a run that started with a dry channel. The plot on the right used an initial condition times to fill the channel before the main simulation started. The channel fills quickly enough that both runs give the same result during the peak flood hydrograph.

However, when plotting the results, the water surface for the dry run starts around 555' whereas the wet start is a little over 571'. The results from the dry start could cause confusion because those are not real water surfaces. Additionally, the dry start shows less detail during the flood hydrograph because the y axis has a greater range.



Initial Conditions Warm Up Period



- Run after Initial Conditions Period
- Length specified as number of time steps
- Time step is optional
- Hold all the BC's constant and allows the model to stabilize

HEC-RAS Unsteady Computation Options and Tolerances

General	2D Flow Options	1D/2D Options	Advanced Time Step Control	1D Mixed Flow Options	
1D Unsteady Flow Options				1D/2D Unsteady Flow Options	
Theta [implicit weighting factor] (0.6-1.0):	1.			Number of warm up time steps (0 - 100,000):	20
Theta for warm up [implicit weighting factor] (0.6-1.0):	1.			Time step during warm up period (hrs):	0
Water surface calculation tolerance [max=0.2](ft):	0.02			Minimum time step for time slicing (hrs):	0
Storage Area elevation tolerance [max=0.2](ft):	0.02			Maximum number of time slices:	20
Flow calculation tolerance [optional] (cfs):				Lateral Structure flow stability factor (1.0-3.0):	2.
Max error in water surface solution (Abort Tolerance)(ft):	100.			Inline Structure flow stability factor (1.0-3.0):	1.
Maximum number of iterations (0-40):	20			Weir flow submergence decay exponent (1.0-3.0):	1.
Maximum iterations without improvement (0-40):				Gate flow submergence decay exponent (1.0-3.0):	1.
				Gravity (ft/s ²):	32.174
Wind Forces				1D Numerical Solution	
Reference Frame:	Eulerian			<input checked="" type="radio"/> Finite Difference (classic HEC-RAS methodology)	
Drag Formulation:	Hsu (1988)			<input type="radio"/> Finite Difference Matrix Solver	
				<input checked="" type="radio"/> Skyline/Gaussian (Default: faster for dendritic systems)	
				<input type="radio"/> Pardiso (Optional: may be faster for large interconnected systems)	
				<input type="radio"/> Finite Volume (new approach)	
Geometry Preprocessor Options				Number of cores to use with Pardiso solver:	All Available
Family of Rating Curves for Internal Boundaries					
<input checked="" type="radio"/> Use existing internal boundary tables when possible.					
<input type="radio"/> Recompute at all internal boundaries					

OK Cancel Defaults ...

27

Warning: If you have 2D flow areas directly connected to 1D river reaches, you must use the 2D Initial Conditions Ramp Up option to get water all the way through the 2D flow area, such that the 1D/2D connection will be wet when the model starts up. If you do not do this, the 1D connection may go unstable right at the beginning of the simulation, because there is no water in the 2D area that it is connected to.



Using a Restart File

1. Write **Initial Condition File** out
2. Create a new **Unsteady Flow Plan** (“Save As...”)
3. Select **Use a Restart File** option in **Unsteady Flow Initial Conditions Tab**
4. Adjust starting date, and any hydrographs, if needed (if you start model later than previous run).

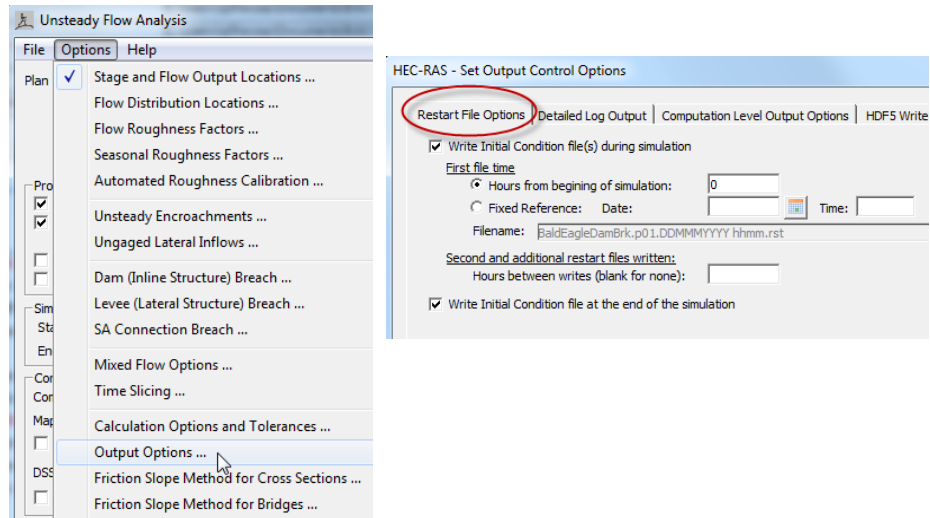
An initial condition file can be saved during a simulation. This restart file can then be used to start a new simulation (it replaces the Initial Condition phase and Warm Up period).

If the initial condition phase takes a long time to run (for instance to fill a very large 2D area), a restart file can be written out at time zero (zero hours from start of simulation) and this file can be used to more quickly start the simulations.

Note: Although it is possible to both create and use a restart file using the same unsteady flow and plan, this is not recommended. Instead, create a new flow and plan file using the “save as” option.



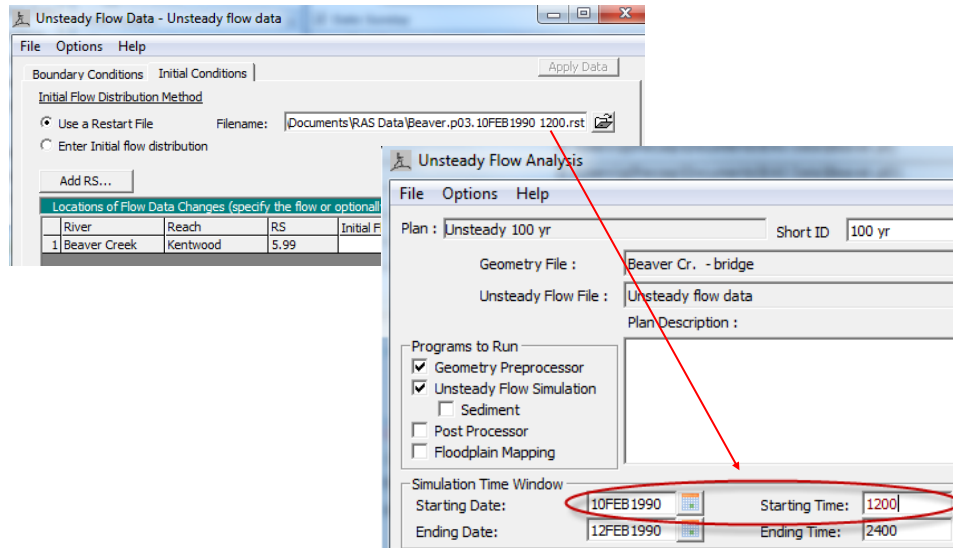
Write Initial Condition File



- The **Restart** file can also be used during model development. For instance, if a RAS model takes hours to run on a computer before it gets to a time period of user interest (the peak flow during the event, or the model goes unstable), a restart file can be written out just before the period of interest.
- The model can then be quickly and repeatedly run through the period of interest (perhaps with more detailed output).
- To create a Restart file, go to the output options editor, select the Restart File tab and enter the date and time and then run the model.
- The restart file starts the model in the same condition that it was saved. For instance, the state of all gate operations, navigation dams, levee breaches, etc. are saved.



Select Restart



- On the initial condition editor, browse and select the desired restart file. Note that the name of the restart file contains both the project and plan identification as well as the date and time stamp (the simulation time when the restart file was created).
- If the restart file is written out in the middle of a long run, it would usually be recommended to change the starting date and time on the compute window to match the date and time of the restart file.
- There is no requirement that the date and time match. For instance, if the restart file is being used in lieu of the initial backwater and warmup period, it may not matter.
- Note however...
- The user must maintain sufficient continuity between the restart file and the unsteady flow run at time zero. For instance, if a gate is only open 0.5 feet in the restart file, but the unsteady model (that is using the restart file) is using a time series gate opening and the given starting time specifies a gate opening of 10.0 feet, instability is likely to result.



Hydrograph Starting Time

The image shows two screenshots of the 'Flow Hydrograph' dialog box. The left screenshot shows the 'Use Simulation Time' option selected, with the Date set to 10FEB1990 and Time set to 1200. The right screenshot shows the 'Fixed Start Time' option selected, with the Date set to 01Jan1990 and Time set to 0000. Both screenshots show a table of hydrograph data with columns for Date, Simulation Time, and Flow.

No.	Date	Simulation Time (hours)	Flow (cfs)
1	10Feb 1990 1200	00:00	1075.53
2	10Feb 1990 1300	01:00	1301.64
3	10Feb 1990 1400	02:00	1676.94
4	10Feb 1990 1500	03:00	2199.12
5	10Feb 1990 1600	04:00	2864.95
6	10Feb 1990 1700	05:00	3670.34
7	10Feb 1990 1800	06:00	4610.32
8	10Feb 1990 1900	07:00	5679.08
9	10Feb 1990 2000	08:00	6870.05

No.	Date	Simulation Time (hours)	Flow (cfs)
1	31Dec1989 2400	00:00	1075.53
2	01Jan1990 0100	01:00	1301.64
3	01Jan1990 0200	02:00	1676.94
4	01Jan1990 0300	03:00	2199.12
5	01Jan1990 0400	04:00	2864.95
6	01Jan1990 0500	05:00	3670.34
7	01Jan1990 0600	06:00	4610.32
8	01Jan1990 0700	07:00	5679.08
9	01Jan1990 0800	08:00	6870.05

- If the hydrograph boundary conditions are being read from DSS, then changing the starting date (see previous slide) is all that is needed. However, if the boundary conditions have been entered in tables, then adjustments may be needed.
- In the example above, the model was originally run from 01Jan1990 000 and a restart file was written out at 10Feb1990 1200. For the plan that is using the restart file (see previous slide) the starting date has been changed in order to match the February starting date. On the flow hydrograph shown above, the first flow value is 1075.53 cfs and this is at 01 Jan 0000 (or 31Dec 2400). If the **Use Simulation Time** was selected when the hydrograph was first entered, then changing the starting date to 10Feb 1200 will cause this to be the first value used when using the restart file. But this is not the actual flow at 10Feb!
- The solution is to switch to using **Fixed Start Time** and to enter the original time and date of the original run that was used to create the restart file.



Restart File Notes

- Geometry must be the same
- Can switch 2D equation if desired
 - **Use to run DWE to create Restart for SWE**
- Can change the time step
- Can change the output interval
- Can (generally) change flow and plan data

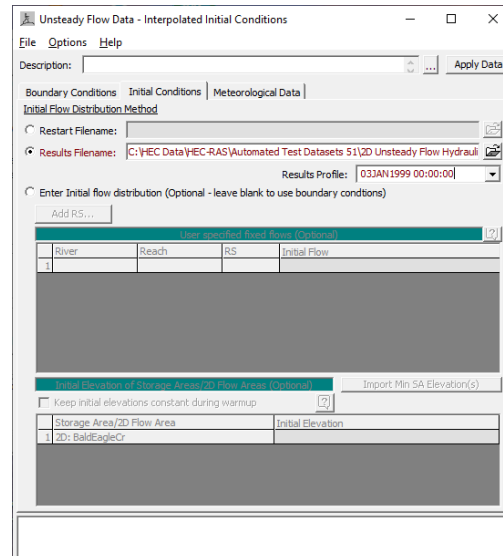
Note: To use the Restart file option, the geometry that is used when the restart file is created needs to be [pretty much] identical to the geometry that uses the restart file.

The simulation that creates the restart file does not need to use the same equation set as the simulation that uses the restart file. If it is desired to use the Shallow Water Equations but the 2D area is having problems during the initial phase/warm up, the initial phase could be modeled with diffusion wave and a restart file written out that is then run with full momentum. Although the diffusion wave does not use the velocities from the previous time step, a restart file written during a diffusion wave run will still write the velocities. So, for example, if a full momentum run uses a diffusion wave restart file, it will have the correct starting velocities.



Interpolate From Previous Results

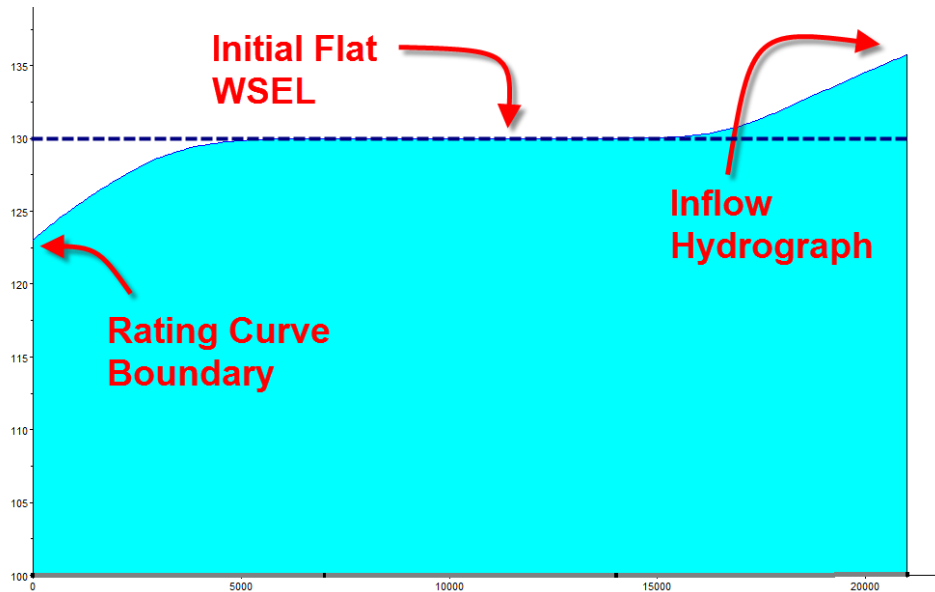
- New for Version 6.0
- User selects a previous plan results file (ProjectName.p##.hdf)
- Geometry does not have to be the same
- Interpolates water levels, velocities, and flows
- Works for 1D and 2D



33



Flat WSEL vs Momentum



- If an initial water surface has been entered, the user should be aware that the velocities at all of the cells [that have water] is zero. The above example has an upstream hydrograph, a downstream rating curve, and an initial water surface that is high enough to wet the 2D area the length of the channel.
- When solving for a given time step, the full momentum equation takes into account the velocities from the previous time step. When the velocities start at zero, it can take a long time for the full momentum solution to accelerate all of the water in the system. Instead of producing a smooth slope, the incoming flow creates a bulge at the upstream end. Similarly, at the downstream end, flow leaving the system can create a steep slope that only slowly propagates upstream. A long initial condition phase and/or warm up period may be needed.
- The diffusion wave solution does not take into account the velocities from the previous time step. Instead, the velocity that is computed each time step is based only on the differences in water surfaces. This allows the diffusion wave to “settle down” to an equilibrium condition much, more quickly.
- However, it should be noted, if a large system really did have an initial flat water surface, it really could take the system a long time to respond to changes at the edges and the full momentum would be appropriate.

Questions?



US Army Corps
of Engineers®

