

# Boundary and Initial Conditions for 2D Modeling

Chris Nygaard, PE

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
- Observed Data



## External Boundary Conditions

- **Flow Hydrograph**

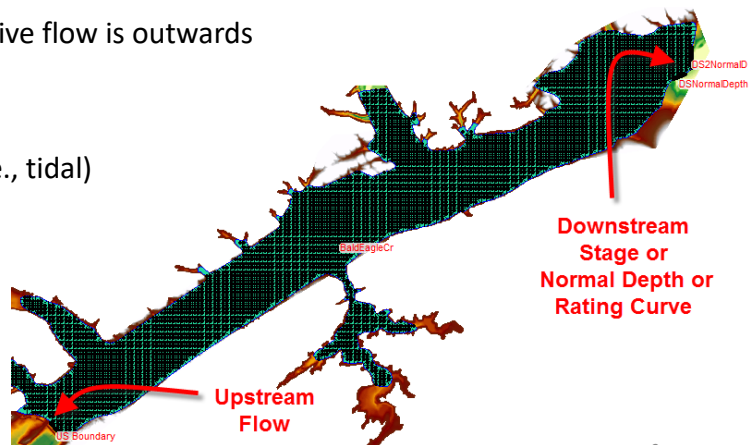
- Usually for inflow (upstream/lateral)
  - Can also be used for outflow
- Positive flow is inwards; Negative flow is outwards

- **Stage Hydrograph**

- Usually for outflow
- Can also be used for inflow (i.e., tidal)

- **Normal Depth (outflow only)**

- **Rating Curve (outflow only)**



3

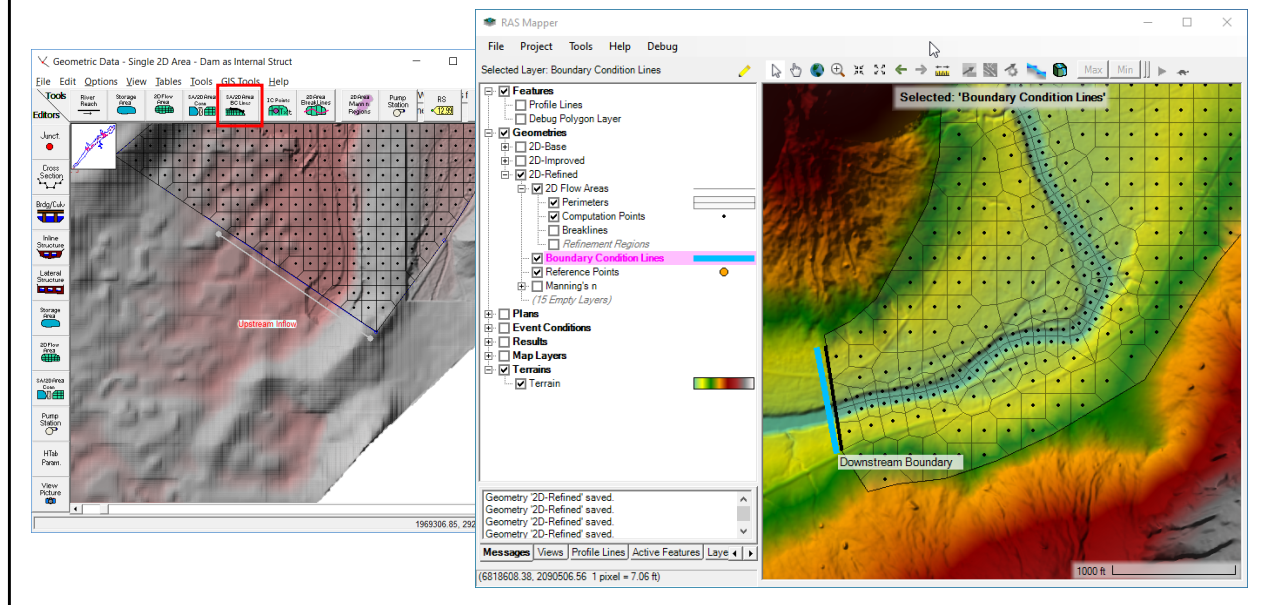
There are 4 external boundary types:

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

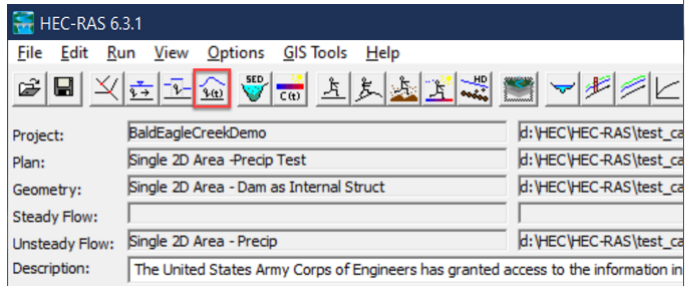


To create a 2D boundary:

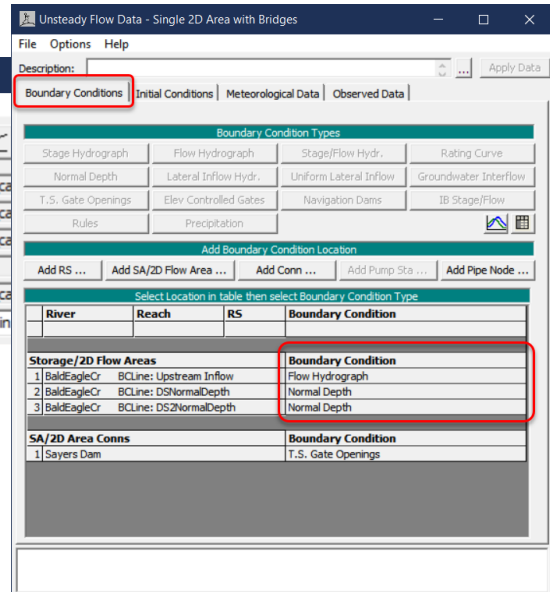
1. If needed, zoom into the approximate location on the geometry editor.
2. Click on the **SA/2D Area BC Lines** button and then using the mouse/pen draw the location of the boundary on the outside of the 2D area in the appropriate location. Double click when finished and then enter an appropriate name 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 for geometry





## Flow Hydrograph & EG Slope



Flow Hydrograph

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

☐ Read from DSS before simulation Select DSS file and Path

File:

Path:

☒ Enter Table Data time interval: 1 Minute

Select/Enter the Data's Starting Time Reference

☒ Use Simulation Time: Date: 01Jan1999 Time: 1200

☐ Fixed Start Time: Date:  Time:

No. Ordinates Interpolate Missing Values Del Row Ins Row

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

Plot Data OK Cancel

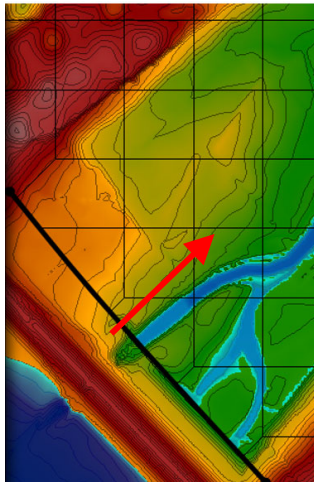
- Time series from DSS or table
- **EG Slope** required to compute normal depth
- Flow distributed to boundary cells boundary based on conveyance
- **TW Check** option uses higher of TW and normal depth
- Face velocities computed for shallow water equations

6

- 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.

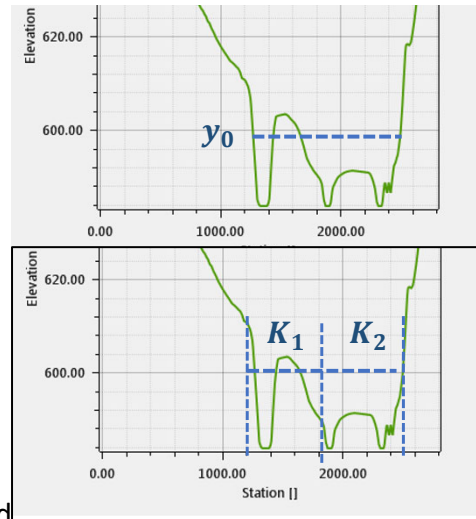


## Flow Distribution along Boundary



$$Q = \frac{1.49}{n} * AR^{2/3} S_f^{1/2}$$

$$K = \frac{1.49}{n} * AR^{2/3}$$



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

7

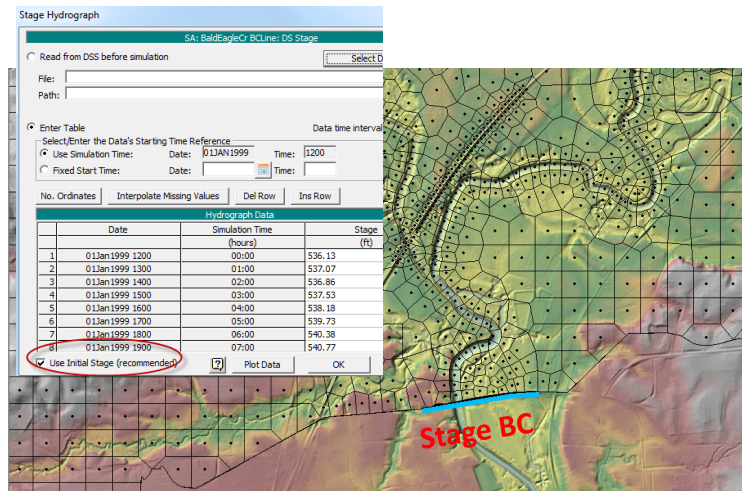
- If the 2D area starts dry, and the flow boundary crosses more than one cell (which it usually will), a methodology is needed to decide which cells get flow. Putting all of the flow into the cell with the lowest invert would be simple, but this can lead to poor results and instability at higher flows. 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



- Inflow or outflow
- All wet faces at the boundary assigned the same stage
- Flow computed per wet face
- **Use Initial Stage** option applies a horizontal water level from the boundary inwards



- 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 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 not necessarily the same.

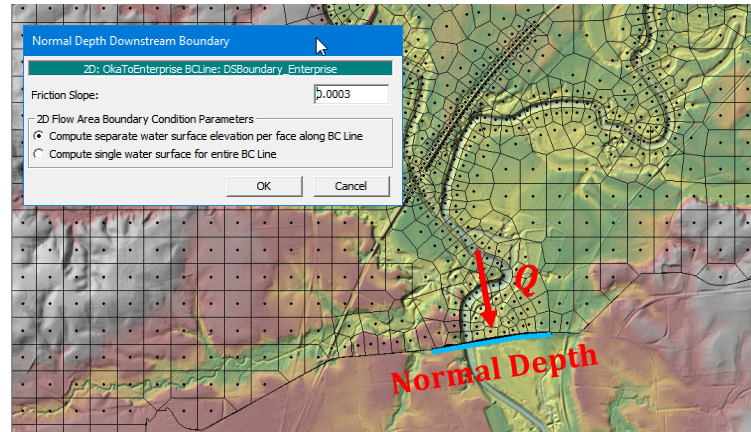




## Normal Depth



- Only for **outflow**
- $Q = \frac{1.49}{n} * AR^{2/3} S_f^{1/2}$
- Manning's Eq. used to compute **flow per face**
- Boundary stage can vary
- Should be reserved for situations where no other data is available



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 [L<sup>2</sup>]

V : Velocity [L/T]

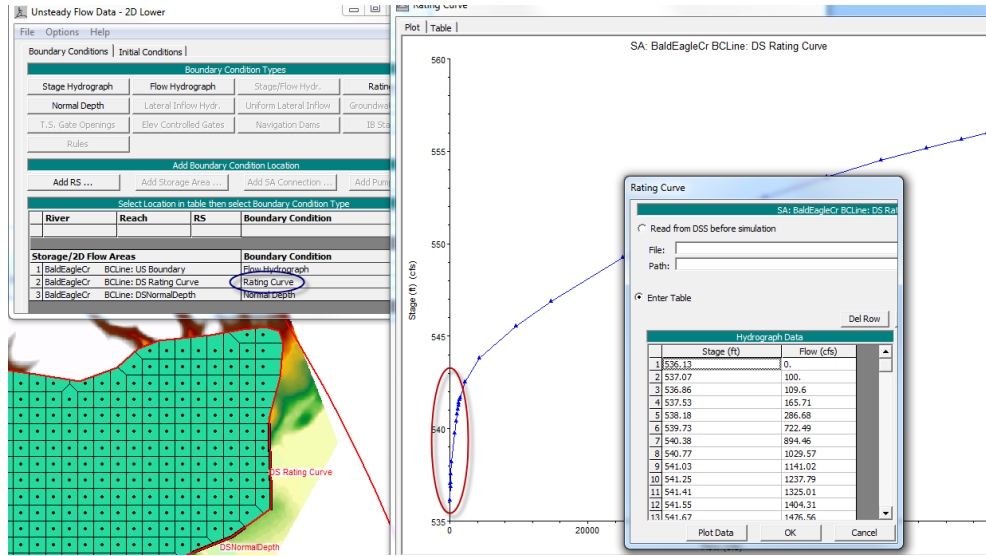
n : Manning's roughness coefficient [1/L<sup>1/3</sup>]

R : Hydraulic Radius [L]

S<sub>f</sub> : Friction slope [-]



## Rating Curve



10

- 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)

11

- 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.



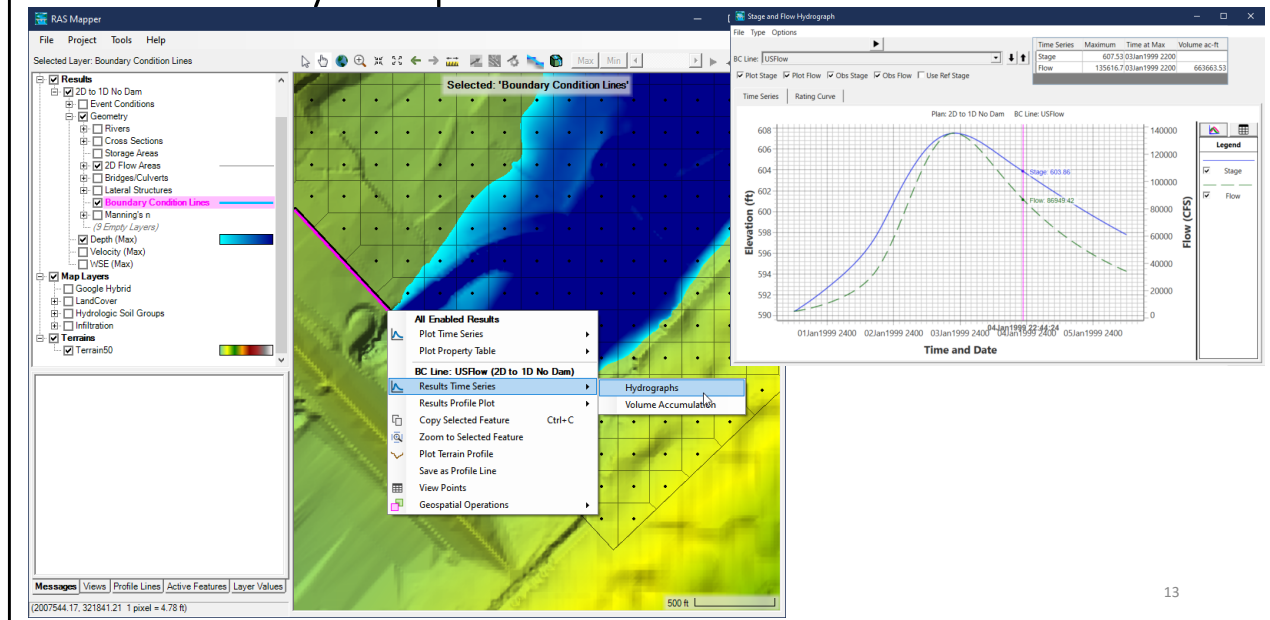
## External Boundary Locations

- 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 account for changes in flow for rising and falling-limbs of hydrograph

12



# Boundary Output



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 **inside** of the 2D Flow Area
- Can have positive and negative flows
- No flow direction; volume added only
- Flow is distributed to cells 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

14

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



# Internal Flow Boundary Condition Editing



**RAS Mapper**

File Project Tools Help

Selected Layer: Boundary Condition Lines

**Features**

- ☒ **Geometries**
  - ☐ Bald Eagle Multi 2D Areas
  - ☐ 1D-2D Dam Break Model Refined Grid
  - ☐ U.S. 2D - D.S. 1D No Dam
  - ☐ 2D to 2D Connection
  - ☐ SA to 2D Connection
  - ☐ Single 2D Area - Internal Dam Structure
  - ☒ SA to 2D Flow Area
    - ☐ Storage Areas
    - ☒ 2D Flow Areas
      - ☒ Perimeters
      - ☒ Computation Points
      - ☒ Breaklines
    - ☐ Refinement Regions
    - ☒ SA/2D Connections
    - ☐ Manning's n (12 Empty Layers)
    - ☐ SA to 2D Flow Area - Detailed

Geometry 'SA to 2D Flow Area' saved.  
Geometry 'SA to 2D Flow Area' saved.

**Messages** Views Profile Lines Active Features Layer Values

(2038212.74, 341839.77 1 pixel = 5.99 ft)

**Unsteady Flow Data - Single 2D Area - Precip**

File Options Help

Description: Apply Data

Boundary Conditions Initial Conditions Meteorological Data Observed Data

**Boundary Condition Types**

Stage Hydrograph	Flow Hydrograph	Stage/Flow Hydr.	Rating 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
<b>Storage/2D Flow Areas</b>			
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
<b>SA/2D Area Conns</b>			
1	Dam		T.S. Gate Openings



## Internal (2D Area) Precipitation BC



- Time-series applied to individual 2D areas
- Precipitation rates spatially uniform
- Specified as period cumulative depths

Precipitation Hydrograph

2D Levee Interior

☐ Read from DSS before simulation

File:

Path:

☒ Enter Table

Select/Enter the Data's Starting Time Reference

☒ Use Simulation Time: Date: 20May1984 Time: 0000

☐ Fixed Start Time: Date: Time:

No. Ordinates:  Interpolate Missing Values: ☐ Del Row:  Ins Row:

Date time interval: 15 Minute

No.	Date	Simulation Time (hours)	Precipitation (in)
1	20May 1984 2400	0:00:00	0.05
2	20May 1984 00:15	0:15:00	0.05
3	20May 1984 00:30	0:30:00	0.05
4	20May 1984 00:45	0:45:00	0.05
5	20May 1984 01:00	1:00:00	0.05
6	20May 1984 01:15	1:15:00	0.05
7	20May 1984 01:30	1:30:00	0.05
8	20May 1984 01:45	1:45:00	0.05
9	20May 1984 02:00	2:00:00	0.05
10	20May 1984 02:15	2:15:00	0.05
11	20May 1984 02:30	2:30:00	0.05

Plot Data OK Cancel

Unsteady Flow Data - Single 2D Area - Precip

File Options Help

Description:

Apply Data

Boundary Conditions Initial Conditions Meteorological Data Observed Data

Boundary Condition Types

Stage Hydrograph	Flow Hydrograph	Stage/Flow Hydro.	Rating Curve
Normal Depth	Lateral Inflow Hydro.	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		Precipitation
2	BaldEagleCr	BCLine: Upstream Inflow	Flow Hydrograph
3	BaldEagleCr	BCLine: DSHNormalDepth	Normal Depth
4	BaldEagleCr	BCLine: DSHNormalD	Normal Depth
SA/2D Area Conns			
1	Dam		T.S. Gate Openings





## Global Boundary Conditions

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

Unsteady Flow Data - Single 2D Area with Bridges

File Options Help

Description: [ ] [Apply Data]

Boundary Conditions | Initial Conditions | **Meteorological Data** | Observed Data

Precipitation/Evapotranspiration: [Enable] Wind: [Speed/Direction]

Meteorological Stations (required for point time series data)

Create/Edit Stations ... Rasterization Parameters (Optional) ... Plot Stations ...

Meteorological Variables (turn on precip and wind above)

Precipitation

Mode: [Gridded] Ratio (Optional): [ ] [DSS]

Gridded Data

Source: [DSS] Interpolation Method: [ ]

DSS Data

Filename: [C:\Users\qhecent\Desktop\RAS\HEC-RAS\_63\_Example\_Projects\Examp] [ ]

Path: [/SHG/MARFC/PRECIP/01SEP2018:0000/01SEP2018:0100/NEVRAD/] [ ]

Evapotranspiration

Mode: [None]

Wind Speed

Mode: [Point] Point Time Series Mode (Nearest) [ ]

Wind Direction

Mode: [Point] Point Time Series Mode (Nearest) [ ]

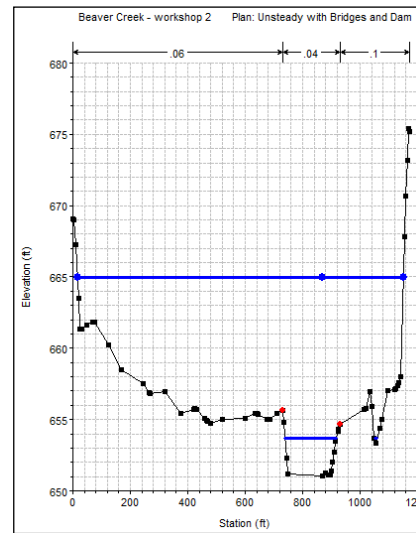
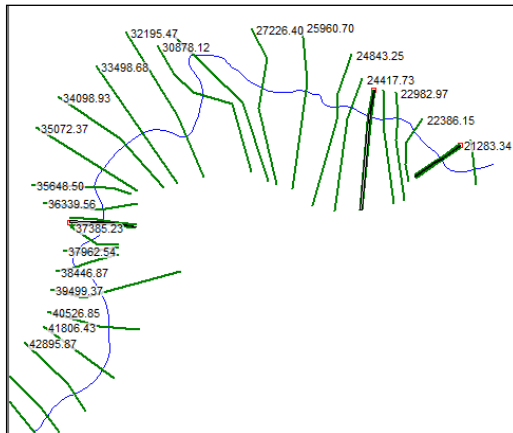
Air Density

Mode: [Constant] Default Constant Value: 1.225 (kg/m3) [ ]

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



## Initial Conditions in 1D



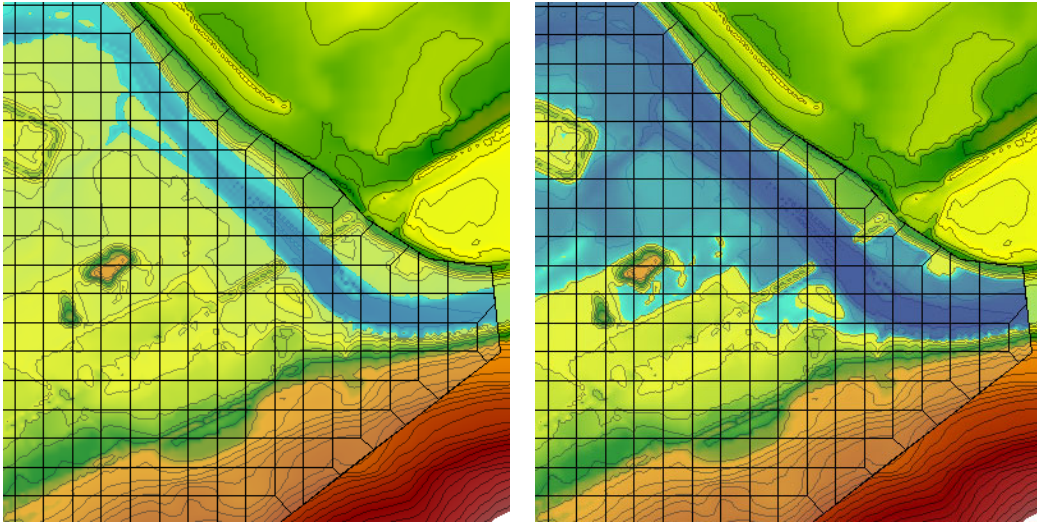
When the 1D RAS unsteady program computes the flow and stages for a given time step, it creates a system of equations that must be solved simultaneously by the unsteady solver. The current 1D solver does not allow for dry cross sections. All of the cross sections in the model must have water in them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



## Initial Conditions in 2D



19



## 2D Initial Conditions Overview

- **User Specified Initial Conditions**
  - Dry
  - Single horizontal WSE
  - Apply initial stage boundary values horizontally
  - Initial Conditions Points
    - Allows each 2D area to compute an initial profile
- **Ramp Up / Warm Up**
  - 2D Ramp Up (optional)
  - Warm Up (optional) 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

20

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.



## 2D Horizontal Initial Elev



- Single WSEL per 2D area
- Not so useful...by itself

Unsteady Flow Data - Single 2D Area with Bridges

File Options Help

Description: [ ] Apply Data

Boundary Conditions Initial Conditions Meteorological Data Observed Data

Initial Flow Distribution Method

☐ Restart Filename: [ ]

☐ Prior WS Filename: [ ] Profile: [ ]

☒ Enter Initial flow distribution (Optional - leave blank to use boundary conditions)

Add RS... [ ]

User specified fixed flows (Optional) [ ]

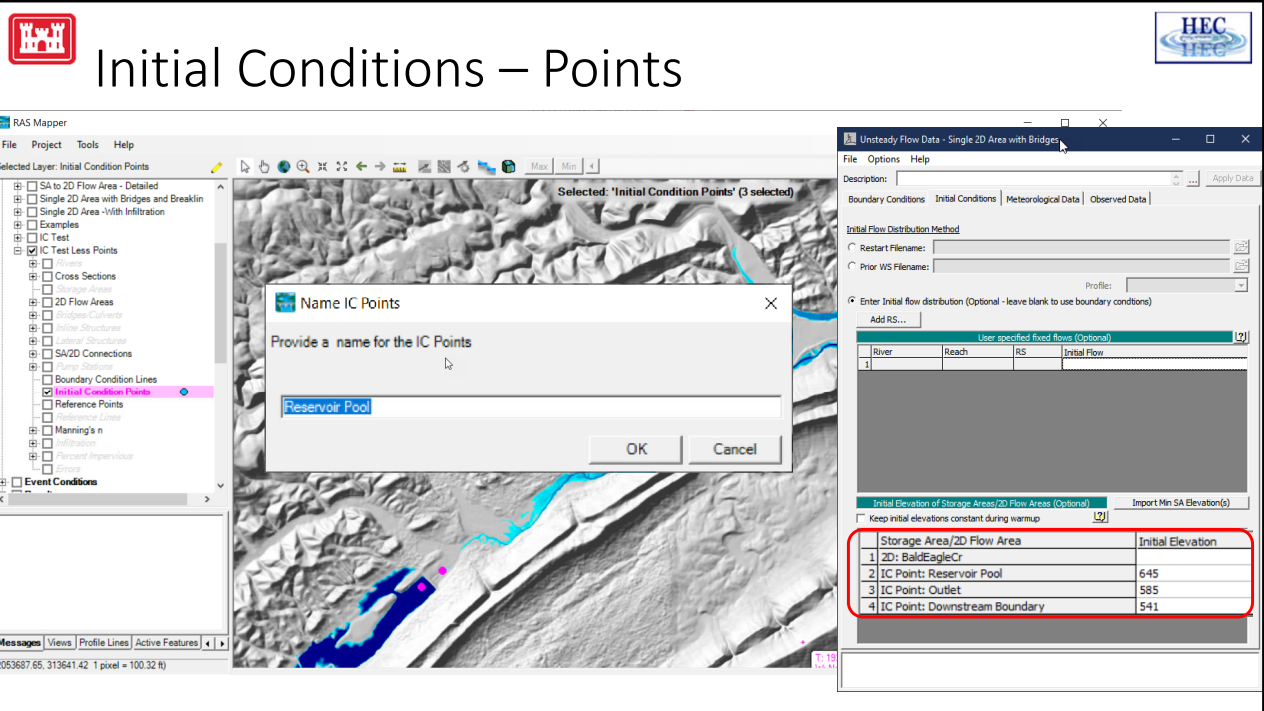
Row	Reach	RS	Initial Flow
1			

Initial Elevation of Storage Areas/2D Flow Areas (Optional) Import Min SA Elevation(s) [ ]

☒ Keep initial elevations constant during warmup [ ]

Storage Area/2D Flow Area	Initial Elevation
2D: Bald Eagle Cr	

21

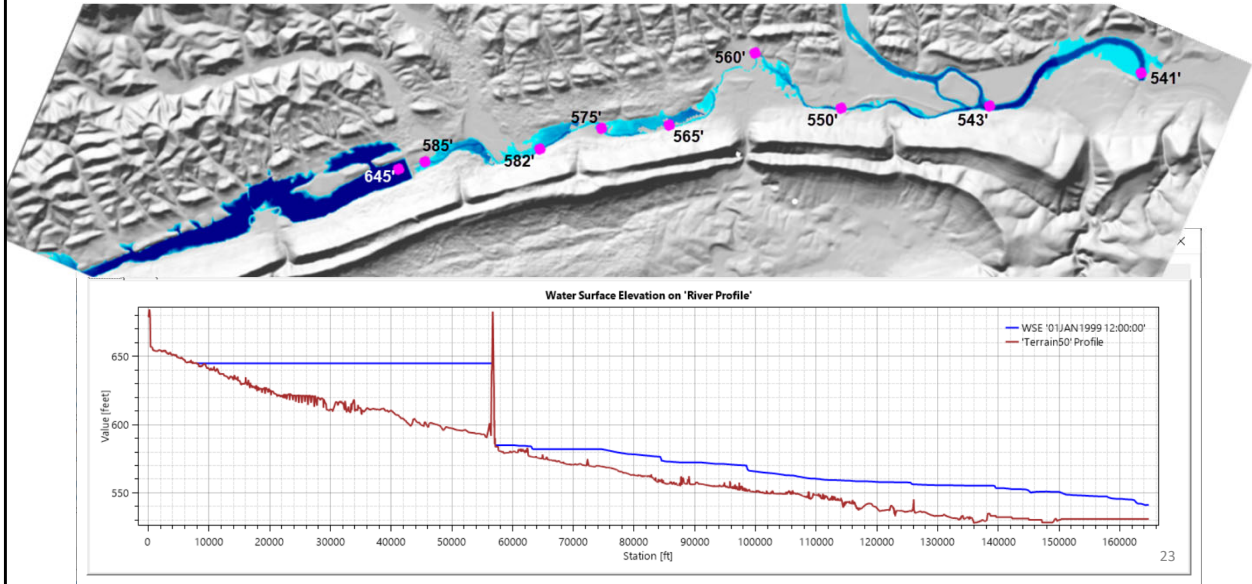


We also added a geospatial option for setting the initial conditions water surface for 2D Models. You can specify Initial Conditions Points and RAS will compute a water surface. This option essentially lets you set a known water surface along the river.

This is done by establishing IC Points in RAS Mapper. You then set the Initial Elevation through the unsteady-flow data editor.



## Initial Conditions Backwater Profile



Here is an example using many water surface elevation points along the river – for instance, if you have river gages. For any “exterior” points (most upstream or downstream), a horizontal “backwater” is computed. This is beneficial for locations behind a dam or that are in backwater. For all other points, a water surface is computed assuming normal depth is appropriate, using a unit-flow width approach.

If you want to improve the initial water surface, you can set the model run 100-500 warmup timesteps which holds the flows constant while running the unsteady flow engine.

The **Flow per Unit Width** Computation Procedure used in RAS Mapper to establish initial conditions has several assumptions built into the method. First, is that that **normal depth is achievable** given the prescribed geometry such that Manning's equation is appropriate. And assume the wide channel assumption that  $w \gg d$ .

Using the unit flow width equation, HEC-RAS iteratively solves for a flow that meets the water surface ( $d$ , depth) at the upstream initial conditions point using the specified downstream boundary water surface elevation (downstream initial conditions point). For all locations between an IC Point

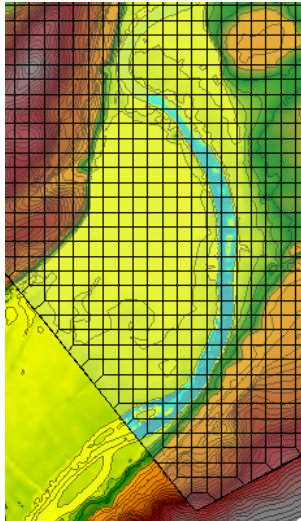
and the Downstream IC Point, we solve for a water surface (assuming normal depth) based on the computed flow per unit width. Wherever a higher water surface is computed throughout the domain, the higher water surface is used. This will result in reasonable starting water surface for the 2D domain. If the initial water surface is not satisfactory, the warm up time step option can be used to settle down the water surface. An illustration of the computational procedure is shown below for a simple river system comprised of a river (a-b) with a tributary (j-c).





## Ramp Up - Filling 2D Channel

- Use **Ramp Up Time** to fill channel



- Upstream flow will eventually fill channel
- May take a long time to fill, especially reservoirs
- Initial horizontal WSE or IC Points can help get to desired conditions faster.

24

- 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

- Specified for each 2D area
- Each 2D area Warms up individually
- **Initial Condition Time:**
  - How long to run ramp up
- **Ramp Up Fraction:**
  - Period over which flow & stage are ramped up to the initial condition
  - Flows & stages 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

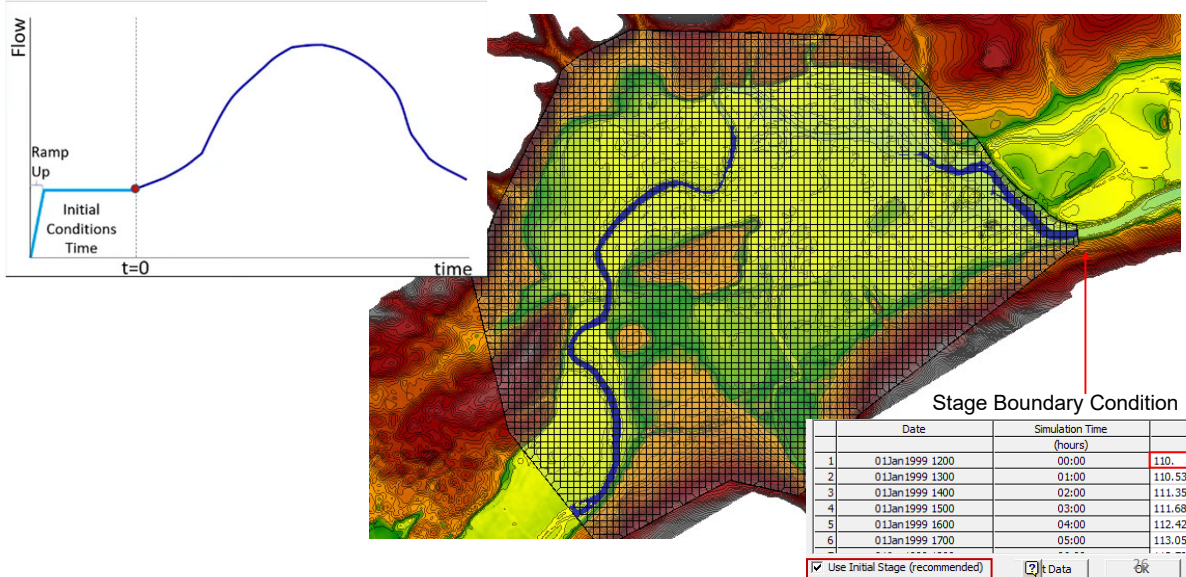
OK Cancel Defaults ...

25

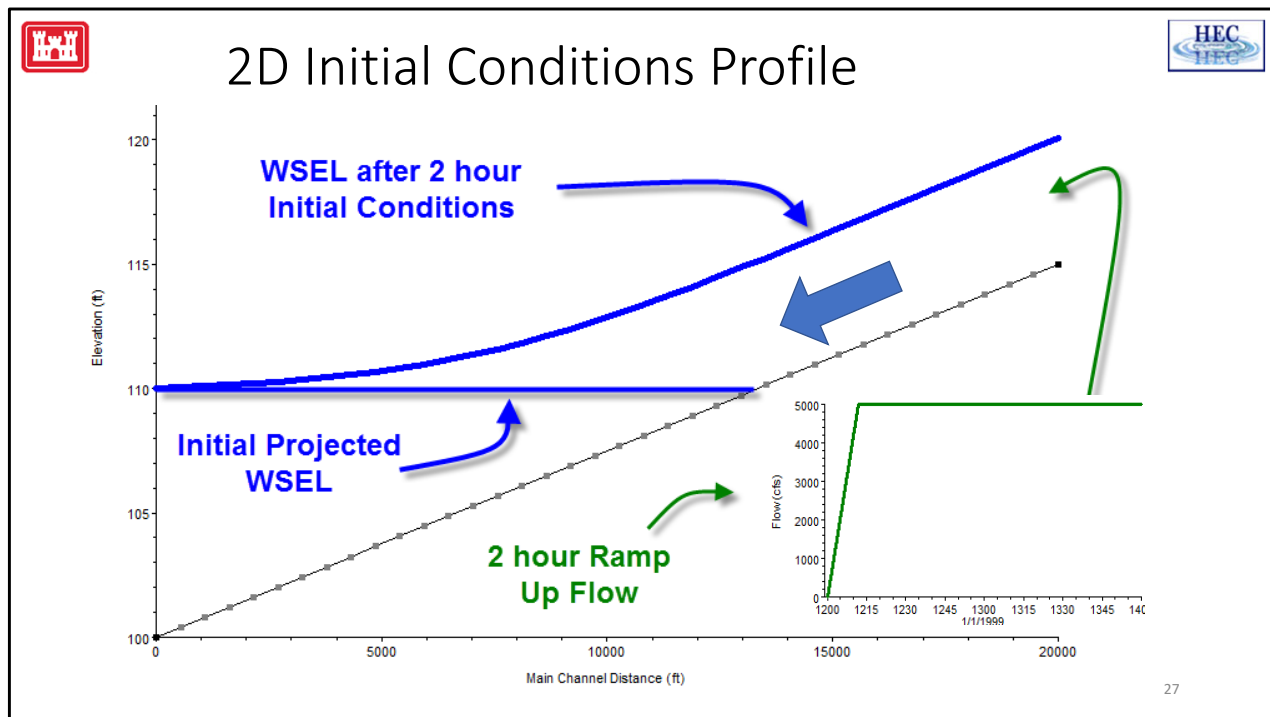
- Say, for instance, that a 2D area has an upstream flow boundary and a downstream stage boundary and the user has entered a 2-hr **Initial Conditions Ramp Up Time** with the **Boundary Fraction** at 0.5 (50%).
- Assume that the first flow on the flow boundary is 1000 cfs and the first stage of the downstream 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



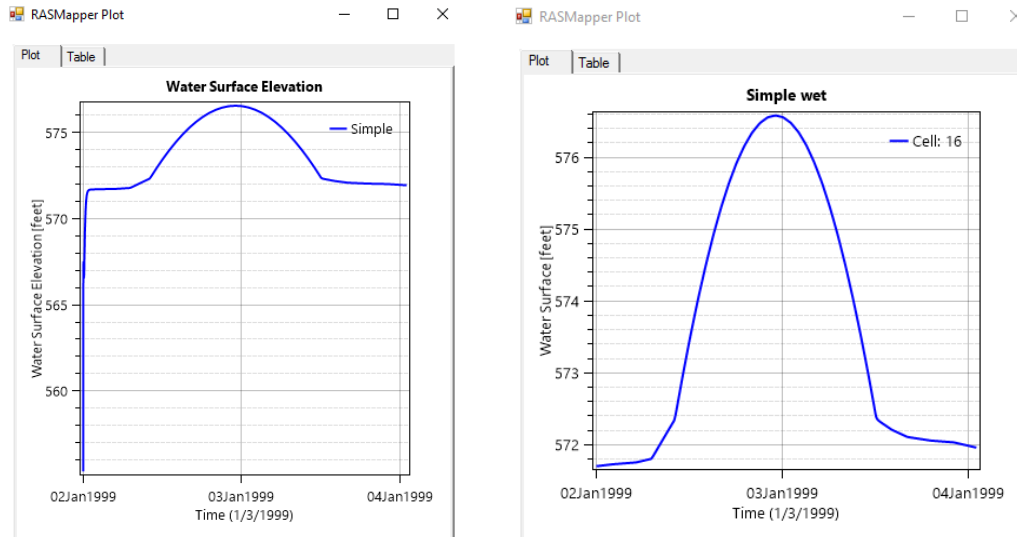
- For a model that has one or more 2D areas, but no 1D reaches, the Initial Conditions and Warm Up are similar. One difference is that only the Initial Conditions “ramps up” the flow. The second main difference is that the Warm Up period solves all of the 2D areas at once.
- The distinction between Initial 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



28

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.



## Warm Up Period



- Runs **whole model** domain
- Runs after Ramp Up Period
- Holds all the BC's constant and allows the model to stabilize
- Specify:
  - Number of timesteps
  - Timestep

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

Theta [Implicit weighting factor] (0.6-1.0): 1.0

Theta for warm up [Implicit weighting factor] (0.6-1.0): 1.0

Water surface calculation tolerance [max=0.2](ft): 0.02

Storage Area elevation tolerance [max=0.2](ft): 0.02

Flow calculation tolerance [optional] (cfs):

Max error in water surface solution (Abort Tolerance)(ft): 100.

Maximum number of iterations (0-40): 20

Maximum iterations without improvement (0-40):

Wind Forces

Reference Frame: Eulerian

Drag Formulation: Hsu (1988)

Geometry Preprocessor Options

Family of Rating Curves for Internal Boundaries

☒ Use existing internal boundary tables when possible.

☐ Recompute at all internal boundaries

1D/2D Unsteady Flow Options

Number of warm up time steps (0 - 100,000): 20

Time step during warm up period (hrs): 0

Minimum time step for time slicing (hrs): 0

Maximum number of time slices: 20

Lateral Structure flow stability factor (1.0-3.0): 2.

Inline Structure flow stability factor (1.0-3.0): 1.

Weir flow submergence decay exponent (1.0-3.0): 1.

Gate flow submergence decay exponent (1.0-3.0): 1.

Gravity (ft/s<sup>2</sup>): 32.174

1D Numerical Solution

☒ Finite Difference (classic HEC-RAS methodology)

Finite Difference Matrix Solver

☒ Skyline/Gaussian (Default: faster for dendritic systems)

☐ Pardiso (Optional: may be faster for large interconnected systems)

☐ Finite Volume (new approach)

Number of cores to use with Pardiso solver: All Available

OK Cancel Defaults ...

29

**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

### Avoid Ramp Up / Warm Up Compute Time!



1. Run initial conditions once
2. Write **Initial Condition File** out
3. Create a new **Unsteady Flow Plan**
4. Select **Use a Restart File**
5. Adjust starting date, and any hydrographs, if needed (if you start model later than previous run).

30

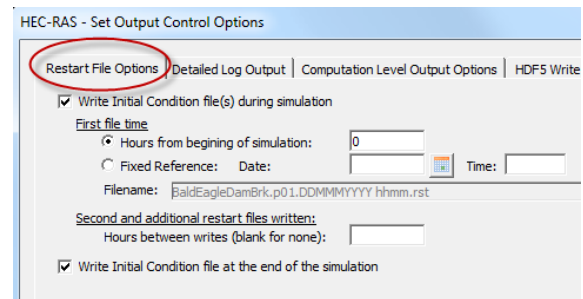
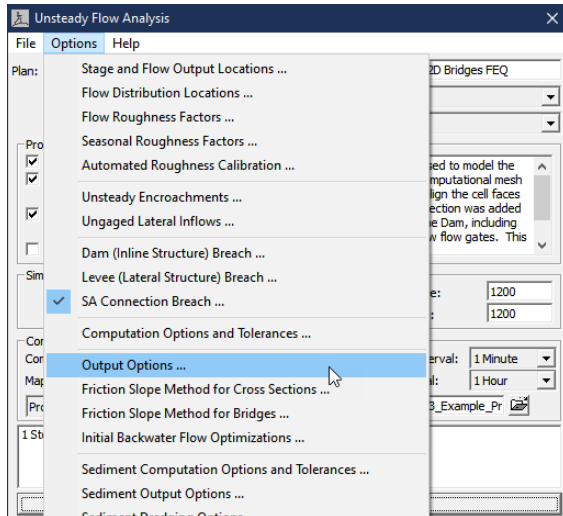
An initial condition file can be saved during a simulation. This restart file can then be used to start a new simulation (it replaces the Initial Condition phase and Warm Up period).

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



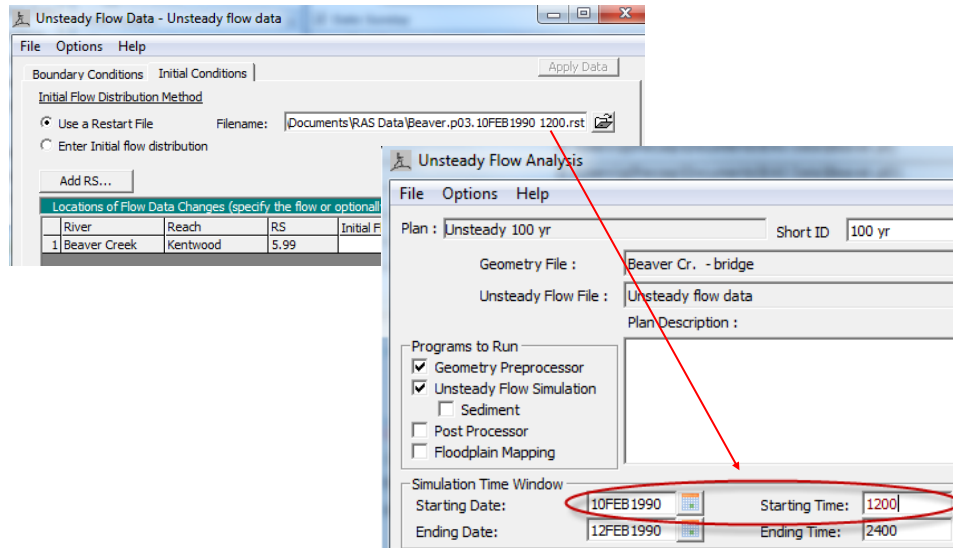
31

- 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



32

- 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



Flow Hydrograph

River: Bald Eagle Reach: Loc Hav RS: 138154.4

☐ Read from DSS before simulation

File:  Path:

☒ Enter Table

Select/Enter the Data's Starting Time Reference

☒ Use Simulation Time Date: 10FEB1990 Time: 1200

☐ Fixed Start Time Date:  Time:

No. Ordinates Interpolate Missing Values Del Row Ins Row

	Date	Simulation Time (hours)	Flow (cfs)
1	10Feb1990 1200	00:00	1075.53
2	10Feb1990 1300	01:00	1301.64
3	10Feb1990 1400	02:00	1676.94
4	10Feb1990 1500	03:00	2199.12
5	10Feb1990 1600	04:00	2864.95
6	10Feb1990 1700	05:00	3670.34
7	10Feb1990 1800	06:00	4610.32
8	10Feb1990 1900	07:00	5679.08
9	10Feb1990 2000	08:00	6870.05

Time Step Adjustment Options ("Critical" boundary conditions)

In Flow: ☐ this hydrograph for adjustments to computational time step

Max Change in Flow (without changing time step):

Min Flow:  Multiplier:

Plot Data OK Cancel

Flow Hydrograph

River: Bald Eagle Reach: Loc Hav RS: 138154.4

☐ Read from DSS before simulation

File:  Path:

☒ Enter Table

Select/Enter the Data's Starting Time Reference

☐ Use Simulation Time Date: 10FEB1990 Time: 1200

☒ Fixed Start Time Date: 01Jan1990 Time: 0000

No. Ordinates Interpolate Missing Values Del Row Ins Row

	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

Time Step Adjustment Options ("Critical" boundary conditions)

In Flow: ☐ this hydrograph for adjustments to computational time step

Max Change in Flow (without changing time step):

Min Flow:  Multiplier:

Plot Data OK Cancel

33

- 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 change flow and plan data

34

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.



## Initializing From Previous Results

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

**Unsteady Flow Data - Interpolated Initial Conditions**

File Options Help

Description: [ ] [?] Apply Data

Boundary Conditions Initial Conditions Meteorological Data

**Initial Flow Distribution Method**

☐ Restart Filename: [ ]

☒ Results Filename: [C:\HEC Data\HEC-RAS\Automated Test Datasets 51\2D Unsteady Flow Hydraulic] [?]

Results Profile: [03JAN1999 00:00:00] [v]

☐ Enter Initial flow distribution (Optional - leave blank to use boundary conditions)

+ Add R5... [?]

Use specified base flows (Optional) [?]			
River	Reach	R5	Initial Flow
1			

**Initial Elevation of Storage Area/2D Flow Area (Optional)** [?]

☐ Keep initial elevations constant during warmup

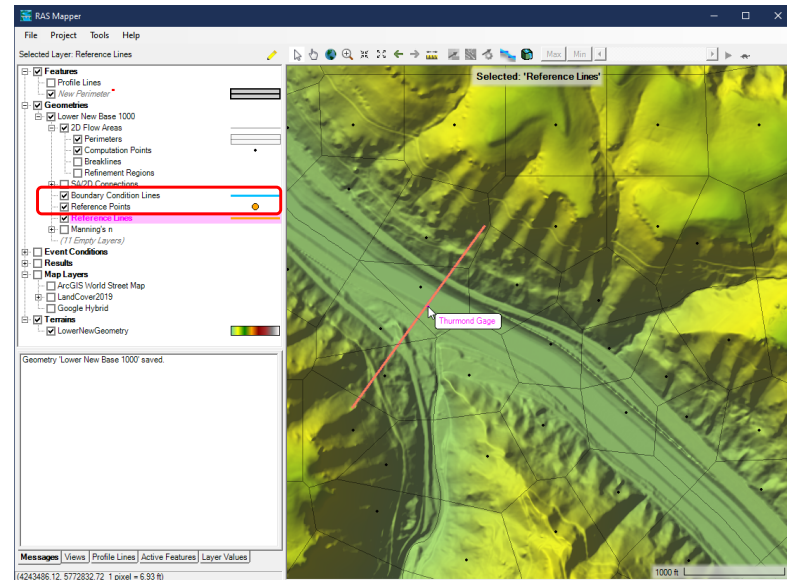
Import Min SA Elevation(s)

Storage Area/2D Flow Area		Initial Elevation
1	2D: BaldEagleCr	



## Observed Data Locations

- Reference Lines
  - Stage
  - Flow
  - Rating
  - HWM
- Reference Points
  - Stage
  - HWM





## Observed Data Entry

- Observed data *locations* associated with a geometry
- Observed data set in the Unsteady Flow File



Unsteady Flow Data - Jan1996

File Options Help

Description: June 2016 Calibration Storm Gridded Rainfall

Boundary Conditions Initial Conditions Meteorological Data **Observed Data** Apply Data

Set/Edit Observed Data RS Locations

No RS specified Select RS's ... Plot Locations ...

Observed Stages

Location	Summary	Edit
1 BC Line: Downstream Boundary	DSS: no data (feet)	...
2 Ref Line: Thurmond Gage	DSS: data range = 1034.80 to 1041.61 (feet)	...
3 Ref Line: Hinton Gage	DSS: data range = 1357.08 to 1359.48 (feet)	...

Observed Flows

Location	Summary	Edit
1 Ref Line: Thurmond Gage	DSS: Error reading dss pathname: \LOWER NEW\NE	...
2 Ref Line: Hinton Gage	DSS: data range = 3620.00 to 17100.00 (cfs)	...
3 Ref Line: Piney Creek Gage	DSS: Error reading dss pathname: \LOWER NEW\PI	...

Observed Rating Curves

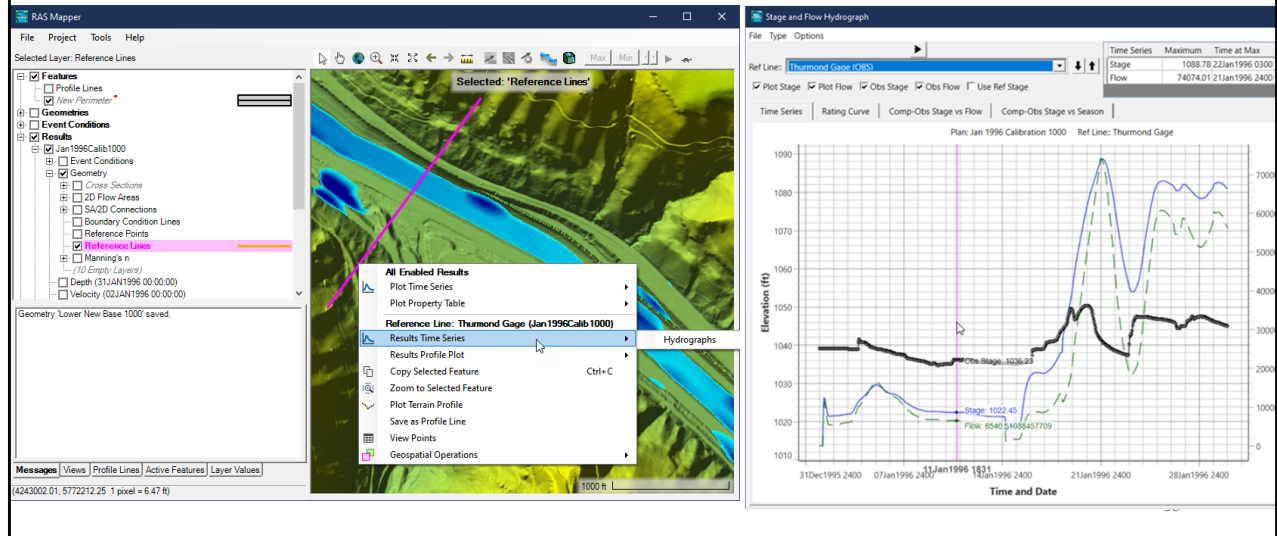
Location	Summary	Edit
----------	---------	------

High Water Marks Data

Location	Summary	Edit
----------	---------	------



# Accessing Observed Data Locations



# Questions?



US Army Corps  
of Engineers®



39