Detailed Bridge Hydraulics in HEC-RAS
Workshop

1 Objective

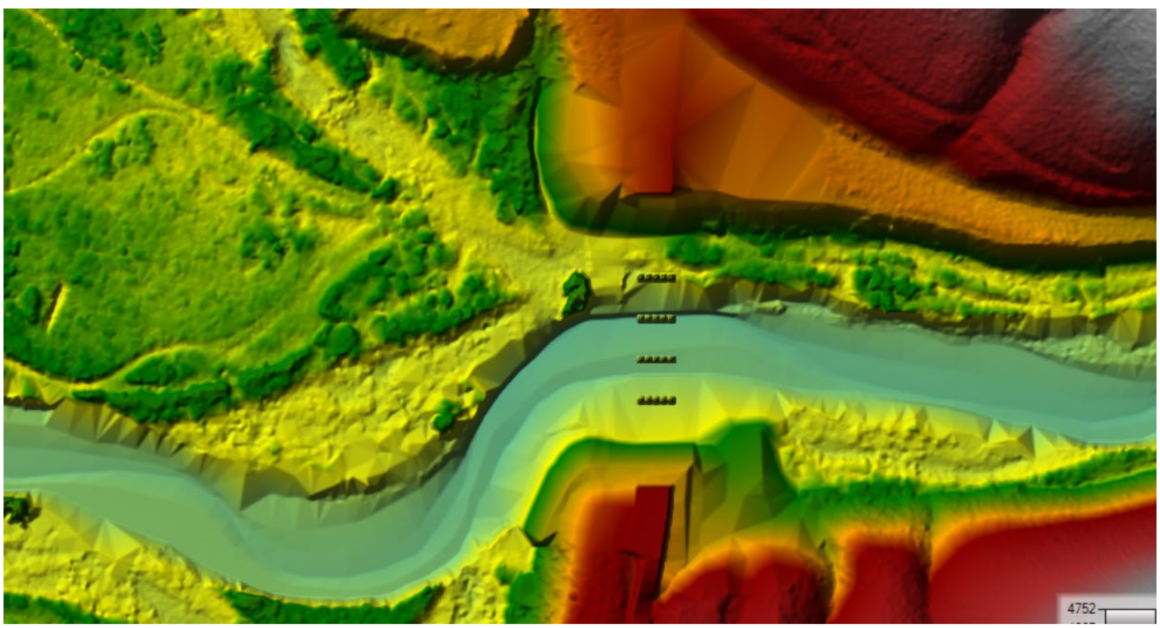
This workshop will help students how to use HEC-RAS to perform detailed modeling of a hydraulic structure (non-pressurized). This workshop specifically models a complex bridge.

The bridge will be modeled using both the Diffusion Wave approximation and full Shallow Water Equations and well as evaluate the use of Turbulence modeling.

2 Background

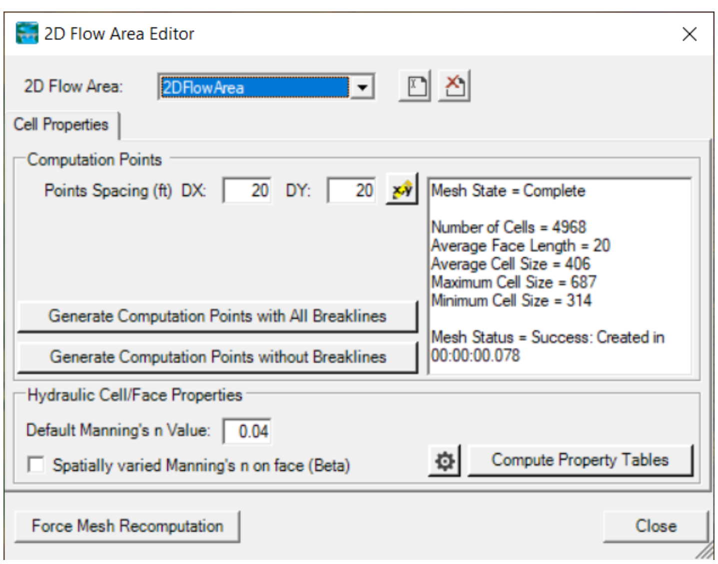
The figure below shows the terrain and bridge crossing (flow west to east). The floodplain is highly constricted at the bridge which includes four rows of piers, each row containing five piers. The square piers have 4ft sides.

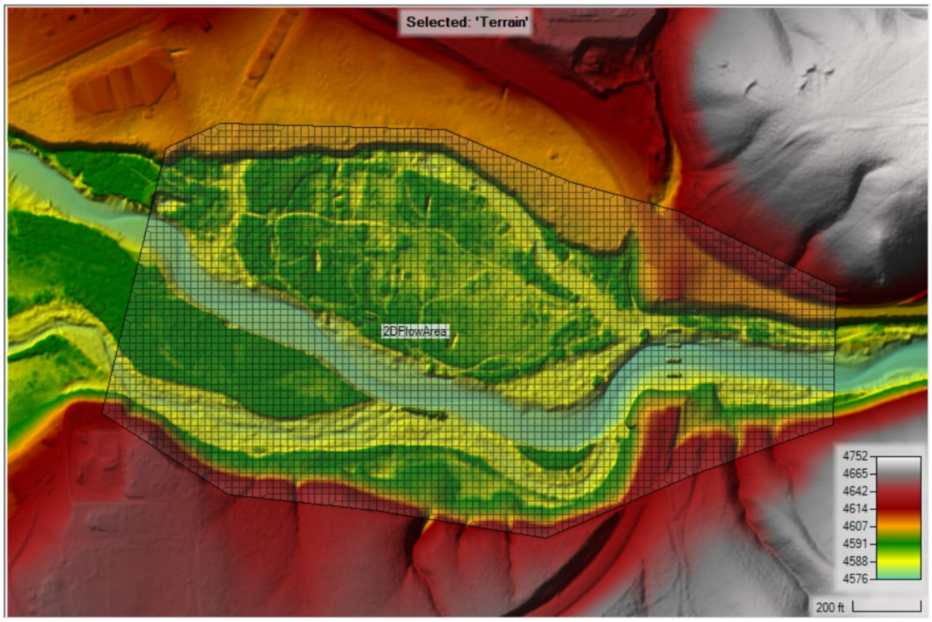
The flow event modeled does not overtop the bridge or hit the low chord.



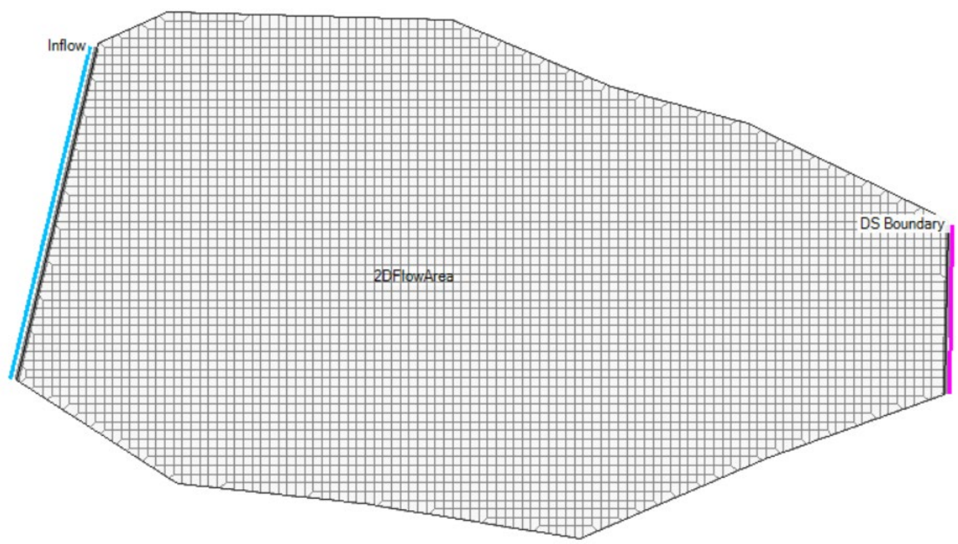
3 Create Initial Model Geometry

1. **Open HEC-RAS** and start a new project called **DetailedBridge** and save it in the workshop directory containing the "GIS_Data" and "Terrain" folders.
2. **Open RAS Mapper**
    a. Set the **Projection** (select *.prj file in "GIS_Data" folder)
    b. Create a **New Terrain** (select Terrain.hdf file "Terrain" folder)
    c. **Add** a **New Geometry**.
3. Create 2D Flow Area with a **20-ft** grid resolution for the entire study area. Try with a Manning's n value of **0.04**.





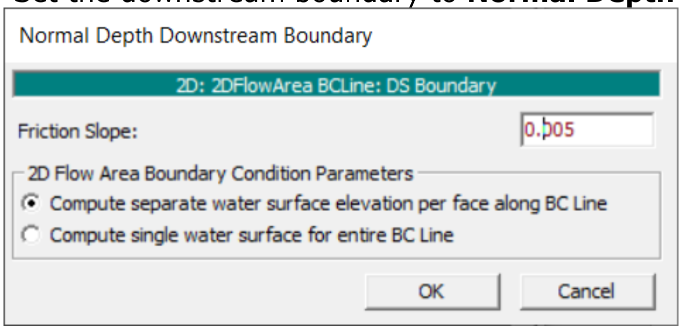
4. Create the upstream and downstream boundary condition lines
    a. **Upstream - Inflow**
    b. **Downstream - DS Boundary**



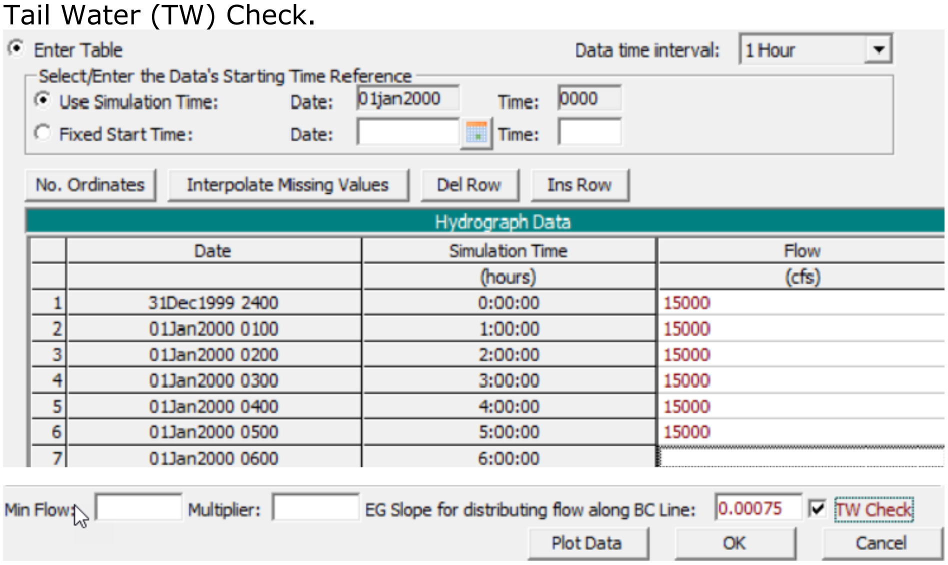
5. **Stop Editing** in RAS Mapper.
6. Close **RAS Mapper**
7. Open the **Geometric Data** editor.
    a. Open the initial Geometry
    b. Close the **Geometric Data** editor.

4 Enter Flow Data and Boundary Conditions

8. Open the **Unsteady Flow Data** editor.
9. Set the downstream boundary to **Normal Depth** with a slope of **0.005**



10. Set the inflow to be a **Flow Hydrograph**
Use a constant flow of **15,000 cfs**. Set the EG Slope to **0.00075**. Turn on the Tail Water (TW) Check.



11. **Save** the flow data

5 DW - Create a Plan and Simulate

12. Open the **Unsteady Flow Analysis** window
13. Set up the time window, time step, and mapping output interval.
14. Set the **Computation Options**
    a. Set the Equation Set to **Diffusion Wave**

	Parameter	(Default)	2DFlowArea
1	Theta (0.5-1.0)	1	1
2	Theta Warmup (0.5-1.0)	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)	0.1	0.1
9	Number of Time Slices (Integer Value)	1	1

15. **Save** the plan data
16. **Compute**

6 SWE - Create a Plan and Simulate

17. Open the **Unsteady Flow Analysis** window
18. **Save Plan As**, to create another run
19. Set the **Computation Options**
    a. Set the Equation Set to the **Shallow Water Equations**

	Parameter	(Default)	2DFlowArea
1	Theta (0.5-1.0)	1	1
2	Theta Warmup (0.5-1.0)	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	SWE-ELM (original/faster)
7	Initial Conditions Time (hrs)		
8	Initial Conditions Ramp Up Fraction (0-1)	0.1	0.1
9	Number of Time Slices (Integer Value)	1	1

20. Set the **time step** to something better for **SWE**.
21. **Save** the plan

    Compute

7 Compare DW and SWE Results

22. Create a **Profile Line** for the river centerline.
23. **Plot the WSE** and compare the results.

Question: Is there a difference between DW and SWE, why would there be a difference, and what does this tell us about the model?

Question: What is the change in water surface through the bridge constriction?

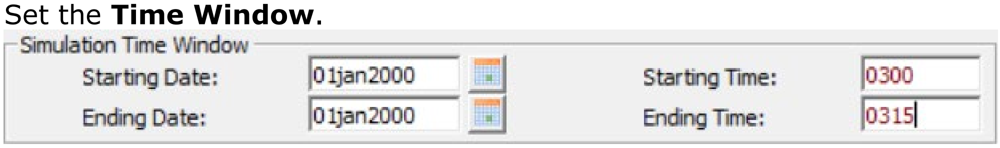
Question: Is the cell size adequate for modeling the terrain, bridge, piers, etc?

8 Refine the 2D Bridge Model

Now that we have some understanding of the model, let's start to improve the runs.

24. Create a copy of the initial geometry using the **Save Geometry As** option in RAS Mapper.
25. Use your knowledge of the contraction and expansion zone through the bridge to add a Refinement Region, using a smaller cell size (**5 ft**).
26. Create a new **Unsteady Flow Data** file with modified initial conditions.
    a. Click on **File | Unsteady Flow Data As** and name it "Flow with WSE"
    b. On the **Initial Conditions** tab, choose **Prior WS Filename**
    c. Select the results file (**.p02.hdf**) from the previous SWE run
    d. **Choose** the last profile

    e. **Save**
27. **Save** the SWE to a new Plan (**SWE-5ft**)
28. **Select** the new **Geometry**.
29. **Set** an appropriate **time step**.
30. Go back to the **Unsteady Flow Analysis** window.
31. **Re-save** the Plan.
32. Set the **Time Window**.



33. **Run the new plan.**

<u>Question</u>: How does the Refined Plan compare with the Initial SWE results?

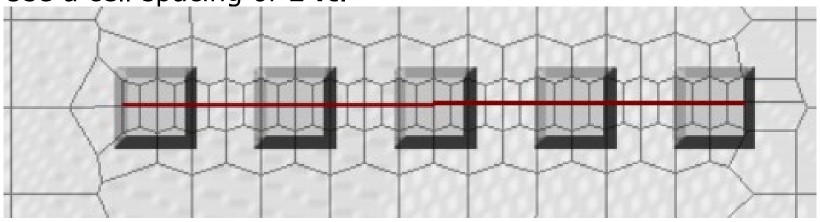
<u>Question</u>: How does the Refined Plan velocity field look?

9 Refine Piers

Continue to improve the model.

34. Create a copy of the refined geometry using the **Save Geometry As** option in RAS Mapper.

35. Refine the area around the Piers using **breaklines** to create smaller cells. Use a cell spacing of **1 ft.**



36. Save the SWE-5ft Plan to a new Plan (**SWE-Piers**)
37. **Select** the new **Geometry**.
38. **Set** an appropriate **time step**.
39. **Compute** results.

<u>Question</u>: How does the Refined Pier Plan compare with other results?

<u>Question</u>: How else can the mesh around the piers be refined besides breaklines?

10 Turbulence Modeling

Add Turbulence to the model and review.

40. Save the SWE-Piers Plan to a new Plan (**SWE-Turbulence**)
41. **Turn on Turbulence Modeling.**

	Parameter	(Default)	2DFlowArea
1	Theta (0.5-1.0)	1	1
2	Theta Warmup (0.5-1.0)	1	1
3	Water Surface Tolerance [max=0.2](ft)	0.02	0.02
4	Volume Tolerance (ft)	0.01	0.01
5	Maximum Iterations	20	20
6	Equation Set	Diffusion Wave	SWE-ELM (original/faster)
7	Initial Conditions Time (hrs)		
8	Initial Conditions Ramp Up Fraction (0-1)	0.1	0.1
9	Number of Time Slices (Integer Value)	1	1
10	Turbulence Model	None	Conservative
11	Longitudinal Mixing Coefficient	0.3	0.3
12	Transverse Mixing Coefficient	0.1	0.1
13	Smagorinsky Coefficient	0.05	0.05

42. **Compute** results.

<u>Question</u>: How does the Refined model with Piers and Turbulence compare with other results? Which run is the better run?