**Detailed Bridge Hydraulics in HEC-RAS**
Solution

## Objective

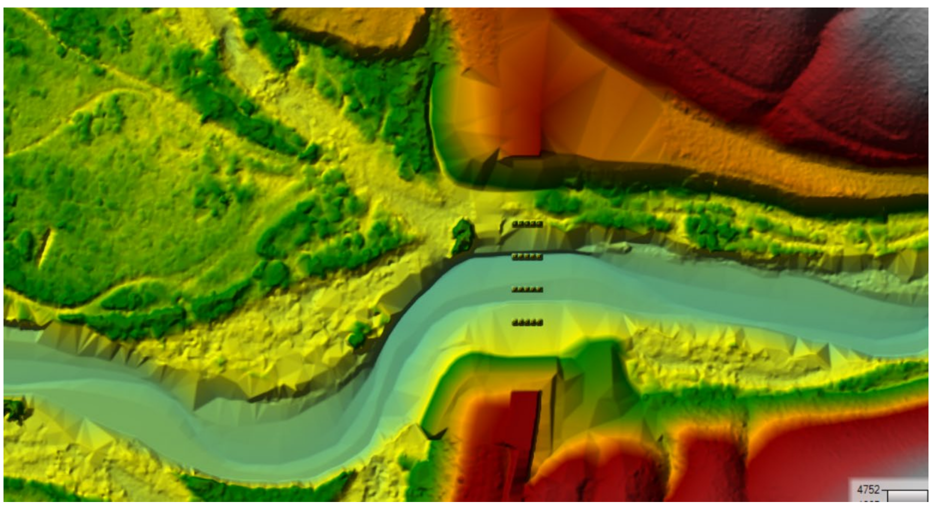
This workshop will help students how to use HEC-RAS to perform detailed modeling of a hydraulic structure (non-pressurized). This workshop specifically models a complex bridge.

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.

## 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.

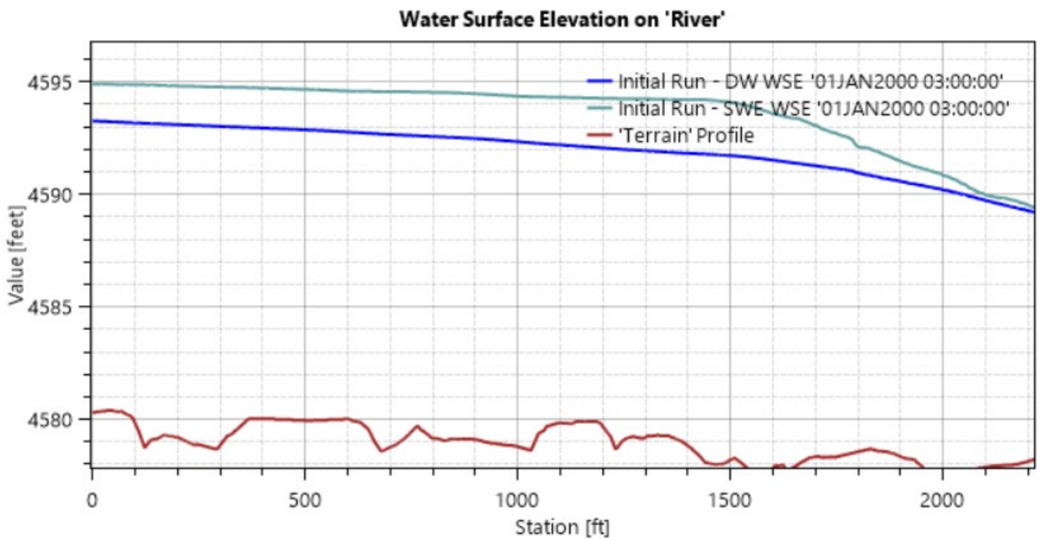
The flow event modeled does not overtop the bridge or hit the low chord.



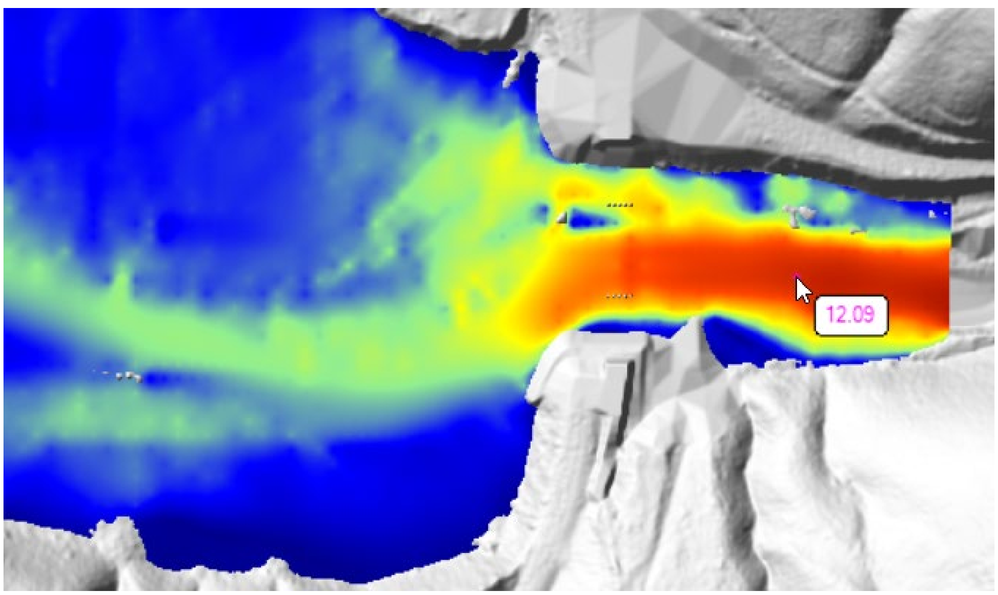
## Compare Results

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

The SWE solution results in a much higher water surface. This is reasonable as the SWE include local acceleration terms in the momentum equation and will result in losses due to contraction and expansion. This also indicates that the DW approximation should not be used to for this river system.




As we can see from the floodplain plot, the bridge is a narrowing of the floodplain creating significant contraction (and acceleration) of flow through the bridge. Velocities increase from 4 ft/s above the bridge to 12 ft/s below the bridge.




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

The water surface at the downstream of the model is **4589ft** and upstream of the bridge the water surface is **4593ft for DW** and **4595ft for SWE**. Assuming the SWE is correct, the bridge creates approximately **6ft** of water surface change.

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

No. In general, the 20ft cells will give a decent water surface through the bridge; however, the influence of the pier will not be modeled (they are simply removing volume from the cells but not influencing the cell faces). We will need to refine the bridge to do a better job of modeling the detailed velocity and water surface around the piers.

## Refine Bridge

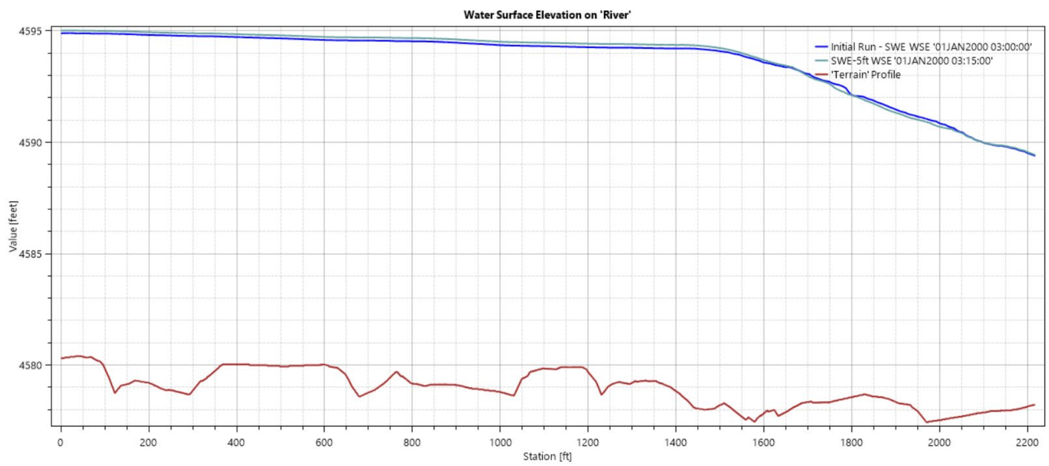
Now that we have some understanding of the model, let's start to improve the runs.

*<u>Question</u>: What is an appropriate time step?*

In an attempt to satisfy the courant condition (of 1), we would look at velocity (~10 ft/s) and cell size (5 ft). Therefore, we would need a cell size of ~0.5 ft/s.

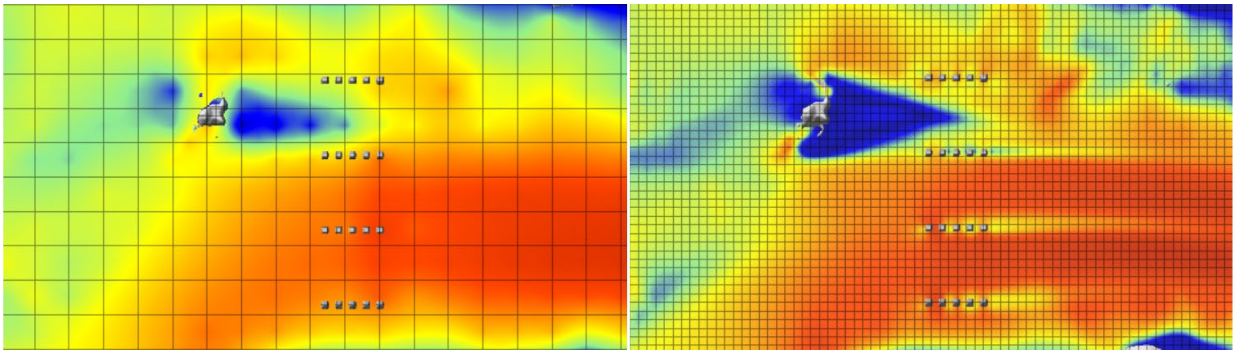
*<u>Question</u>: How does the Refined Plan compare with the Initial SWE results?*

The SWE runs with the 20ft mesh and refined mesh with 5ft in the bridge opening show very similar results for the water surface elevation profile. This supports our thought that we would get reasonable water surface elevation results with the larger cell size.




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

Looking at the velocity field, we see that we have "dramatically" different velocities. For the refined mesh you can see the acceleration of flow and the tail of low velocities downstream of the piers. This indicates that if you need detailed velocity information a refined mesh is necessary!



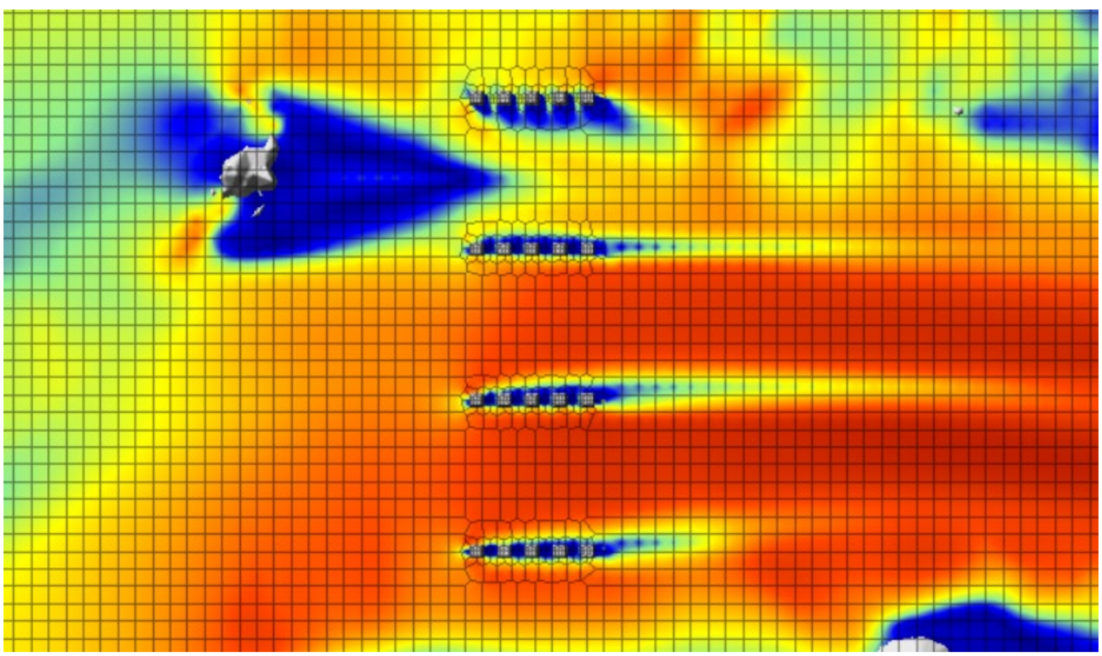
With the refined mesh, not only do you see the acceleration of flow, but you can see wave run-up and separation around piers, separation downstream of the piers, and eddy formations.

However, if you closely inspect the piers, you will see velocities that just don't quite make sense.  The piers are not adequately represented. A better solution will have smaller cells that include the piers.

## Refine Piers

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

The smaller 1ft cell size does a much more reasonable job of showing the fluctuations in velocities expected from the effect of piers on flow.  You can more clearly see the skewed flow pattern and reduced velocities in the wake of the piers (which we did not see with the 5ft cells).

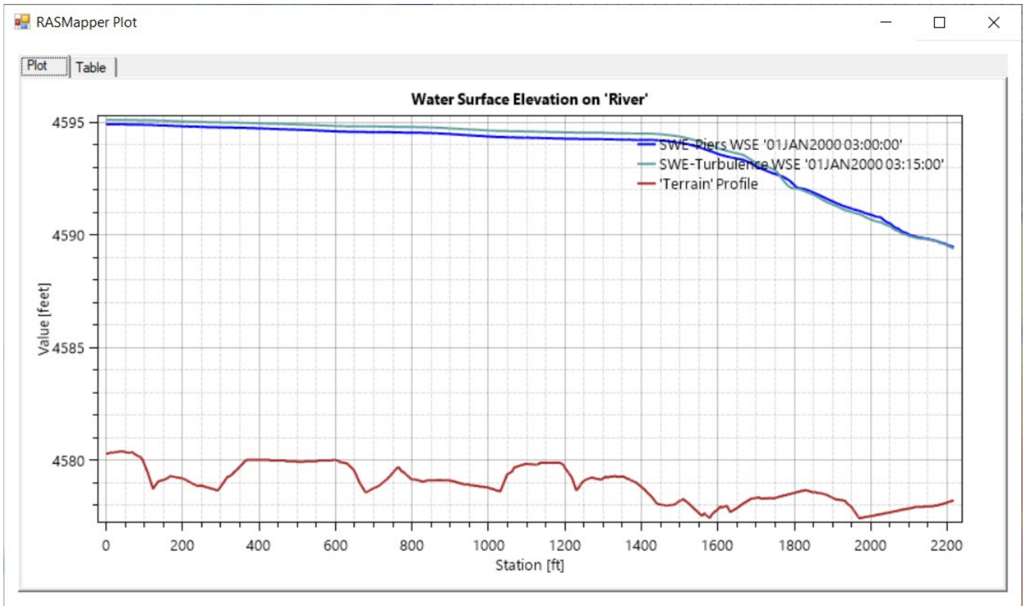


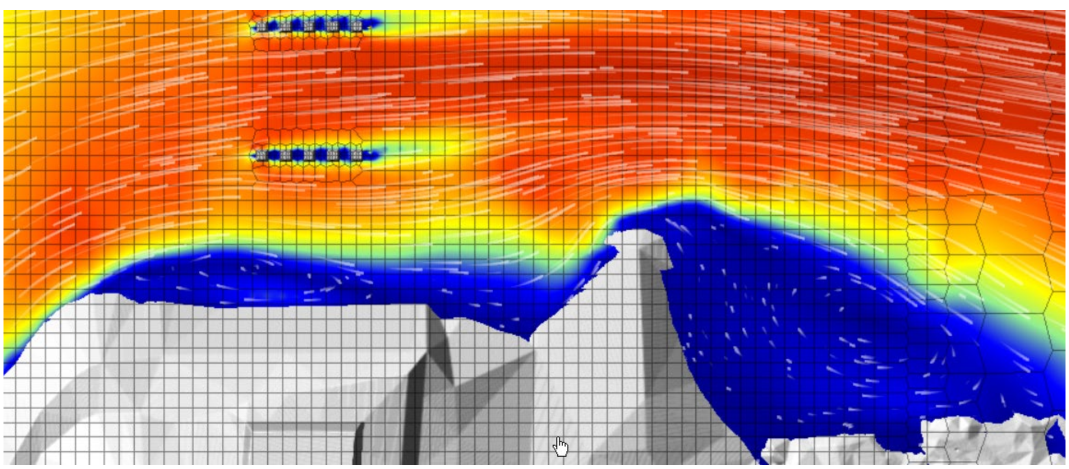
However, it looks like the areas of flow separation downstream of the piers are longer than they should be – flow should be accelerating much more quickly.

## Turbulence Modeling

*Question: How does the Refined model with Piers and Turbulence compare with other results? Which run is the better run?*

The water surface profile results with turbulence are very similar to the results without turbulence modeling, however slightly higher. This is due to the lower velocities on the downstream right abutment causing slightly more constriction.



The results for the turbelence modeling are definitely improved around the bridge piers. However, we don't really know how what turbulence modeling parameters we should be using. We would need to do a sensitivity analysis to see how impactful turbulence could be on the final solution. It would be valuable to observed data with which to calibrate.