Creating a 2D Model
Workshop Solution

Question: _What things should you consider when creating/defining the 2D Flow Area polygon?_

The polygon defining the 2D Mesh should capture the entire area of interest making sure that the polygon goes out to high ground. The high ground will ensure a "no flow" boundary (except at the inflow and outflow locations). However, the bigger your polygon the more cells you will have, the longer the computational run time, so sticking to the area of interest is important! If there are linear features in the land surface we want to make sure those are captured in the mesh (don't worry about it for this workshop, we'll do that in future workshops). Also, because of RAS quirks in mesh generation, making the polygon "smooth" will help with mesh generation, and reduce any problems in creating cells along the polygon boundary.

Also, since the water surface at the downstream boundary condition is unknown, in general we would want to go far enough downstream, such that when we apply a boundary condition like normal depth, that boundary condition does not have a significant effect on the water surfaces in our study area.

Question: _What is an appropriate time step to use for the simulation? How would you figure it out?_

Deciding on the proper time step for the 2D Simulation is similar to figuring out what time step to use for an unsteady-flow simulation in a 1D model. So, the question you should ask is "if my cross sections are spaced X feet apart (or in 2D my cell size is X feet), and my flood wave velocity is so fast, then to satisfy the Courant condition, my time step should be thisbig" **(time step <= spacing/velocity)**.

For a 500ft grid-cell spacing and an estimated flood wave velocity of 10 ft/s, then a time step of 100s is appropriate. Adopting a time step of 1min would certainly be appropriate, while a time step of 2min would probably work out just fine and will most likely reduce the simulation time. If using the Diffusion Wave form of the unsteady-flow equations, satisfying the Courant condition of C<= 1.0 is not nearly so necessary as when using the full shallow water equations.   In general, the Diffusion Wave equations are more stable.

For the 500 ft grid cell size, we selected a **1 minute time step** to use for the initial model.

Question: What is an appropriate Initial Conditions Time to use? What happens if you don't use an Initial Conditions Time? How else could you set the initial conditions?

The Initial Conditions time allows the 2D model to put water everywhere in the model where base flow should exist. This is not going to be just based on a travel time in the river, but rather how long it takes to fill areas that should be wet. After running an initial simulation, you will find when water has filled the low-flow portion of the river.

For the main Bald Eagle Creek portion of the river, the initial flow is very low. You would need to put in a reasonable "minimum flow" to fill up the channel. The initial flow from the inflow hydrograph is very, very low and, therefore, takes a very long time to fill the channel. Increasing the minimum flow to 400cfs, requires a warmup of about 20 hours to get the river stages to a reasonable elevation.

Additionally, the terrain is rough in some areas with artificial dams/bumps. Therefore, the terrain model would need to be corrected before an appropriate warm up was selected. Fixing the terrain model is not a reasonable step for this workshop, so it will be ignored; however, you should look at this closely in your terrain modeling.

One way to help speed up the simulation time is to make your first run and create a Restart file at the end of the warm up time period. This can be done from the **Options | Output Options** menu item on the Unsteady Flow Simulation window. Set a Restart file to be created at the beginning of the simulation. This creates a file "Simple2DModel.p01.DDMMMYYYY hhmm.rst" and can be used in the unsteady flow file for the initial conditions.
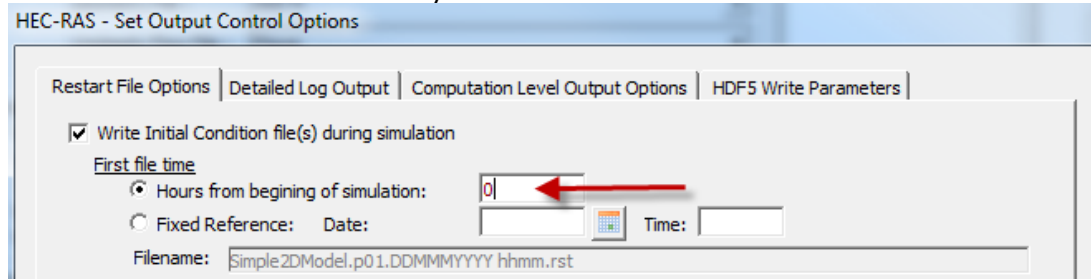
Next, go to the Unsteady Flow file, select the **Initial Conditions** tab and select the **Use a Restart File** option. You will then select the restart file to use.
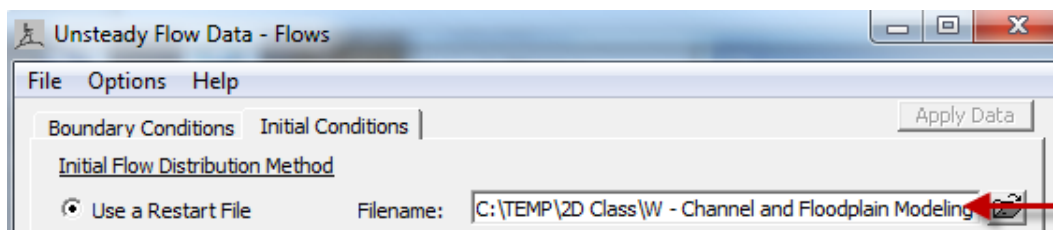
Question: What SHOULD we have done before simulating?

To create hydraulic tables for the 2D Computational Mesh requires a Terrain model; therefore, you should have <u>associated the Terrain with the Geometry</u>. In this case, since only one Terrain was specified, RAS figured out which Terrain to use. However, good practice is to NOT rely on RAS to make good decisions for you!
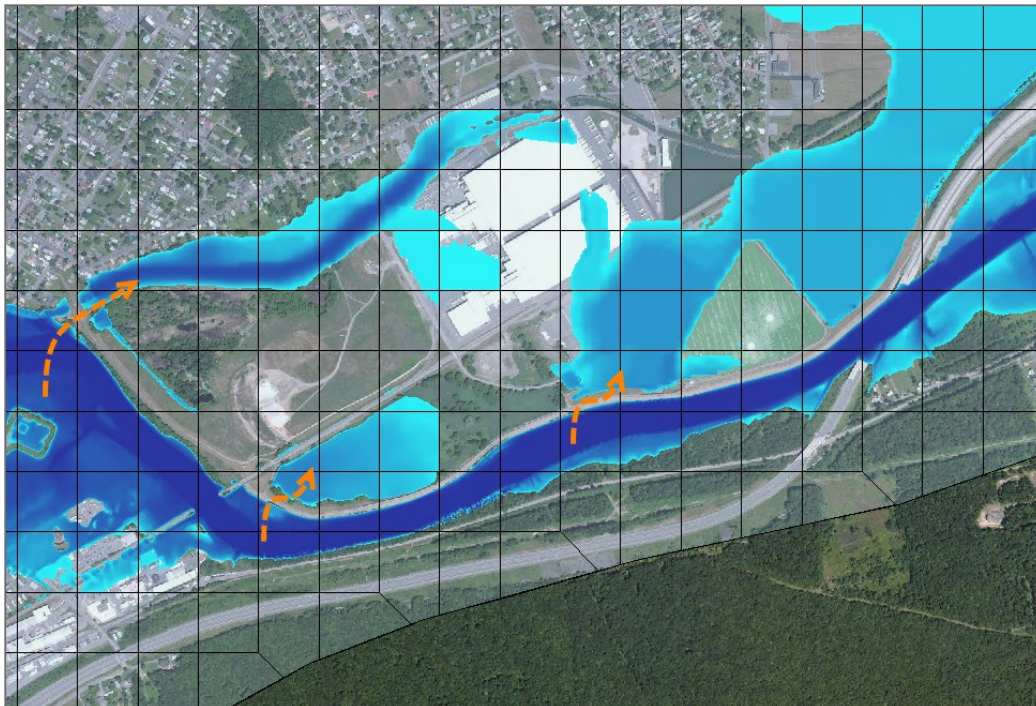
Question: What is the significance of the time step on the results? Evaluate by comparing the Stage or Depth hydrographs at several locations in the model from upstream to downstream, including the tributary areas.

> Several different time steps were used for the simulation (1min, 10min, and 30 seconds). The Water Surface Time Series option was used to plot the water surface at various grid cells, but specifically at the Walmart. It was noted that there is very little difference in the water surface for the 1min, 10min, and 30 seconds time step down near the town of Lock Haven (for the 10min time step <0.1ft difference is observed). However, the hydrographs below demonstrates the instability that will be produced by selecting an improperly large time step (the hydrograph on the left is up below the dam, the other is downstream near the town). See the jagged (unstable model) plot for the large time step compared with the appropriate time step, below.
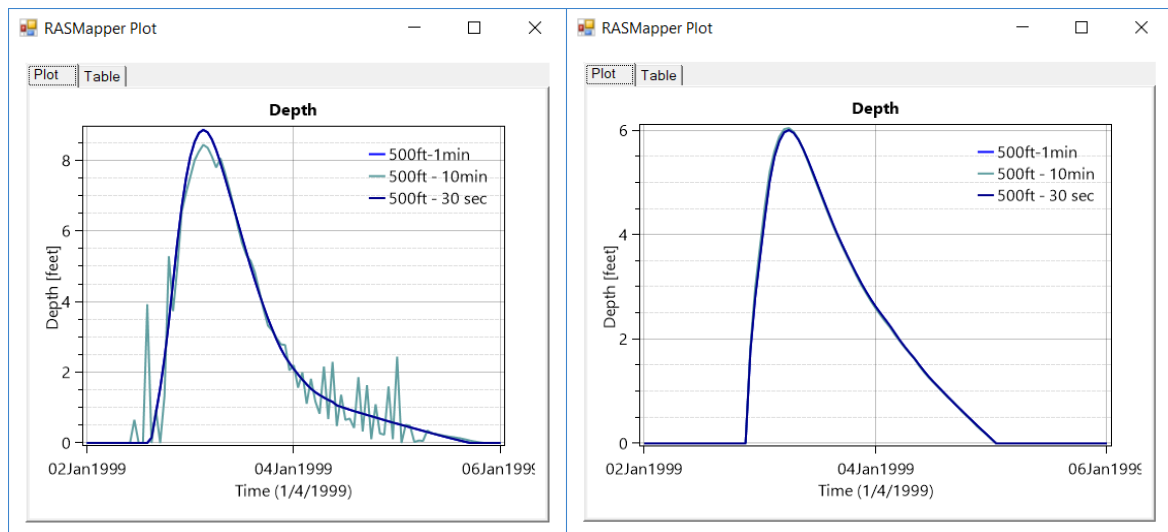
Question: How is the town of Lock Haven impacted by the flood event?

Lock Haven is significantly flooded in the simulation. It is noted that most of the water appears to be coming from the river on the western edge of town. However, even at the max inundation the top of levee appears to not be overtopped. This is due to the large grid cells used in modeling. The faces of the grid cells do not follow along the top of the levee, therefore the 2D mesh is not respective the elevation of the levee and preventing water from getting into Lock Haven. Regardless, the levee should be incorporated into the model to properly represent the land surface.

Question: What is the significance of the grid-cell size on model results?

The smaller grid-cell size appears to have made minor differences in total inundation area down near the confluence of Bald Eagle Creek and Fishing Creek. The smaller grid-cell size appears to be producing a slightly higher water surface (0.5 ft), and producing a larger inundation area.

Questions: Did the smaller cell size result in differences at the town of Lock Haven? What is geometrically significant about the Lock Haven area? How could we best model this area?

No. Lock Haven is still being inundated similarly to other runs. Lock Haven is protected by a levee. To properly model this, the levee must be incorporated into the model. This is done by aligning the faces of the computational mesh along the top of the levee so the that the sub-grid properties (cell face and corresponding elevation profile) capture the high ground and direct water to not cross cell boundaries until the water surface reaches an overtopping elevation. (You will learn how to do this later this week!)

Questions: *What is the effect of incorporating the levee on model results? What do you think of the results?*

Add the break line to the computational mesh improved the model results as water in confined to the channel for higher flows. However, because the terrain does not correctly represent the true elevation of the levee, there are still problems with water leaving the channel too soon.

This is clearly shown by looking at the terrain and zooming in. The top elevation of the levee shows a bit of a roller coaster ride. If we look at the profile plot for the cell faces in the area, we see a jagged up and down profile.

To properly model this levee you would need to correct the elevation data in the terrain model. Or you could use the 2D Area Connection option directly in HEC-RAS (to be discussed later this week).

WS – Creating a 2D Model/Ackerman