Instruction and reference material for performing modeling tasks in HEC-RAS.

1 Guides

Guides are intended to be short overviews of how to do something.

> **Terms of Use**
>
> Any use or reproduction of this material must be attributed to the US Army Corps of Engineers Hydrologic Engineering Center.

1.1 Creating an Combined 1D/2D Model

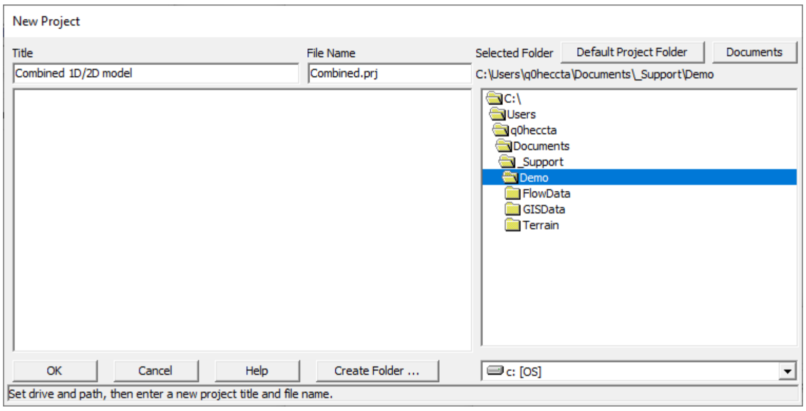
This document will describe the basic steps in creating an combined 1D/2D river hydraulics model in HEC-RAS. Steps for creating, refining, reviewing, and comparing to alternative plans will be discussed. This document will not provide the level of detail that you can get from the HEC-RAS User's Manual[1], HEC-RAS 2D User's Manual[2], or the HEC-RAS Mapper User's Manual[3] and should be considered a guiding document for **typical or general** steps, guidance, and problems you might encounter in building and refining an HEC-RAS model.

1.1.1 Create a New Project

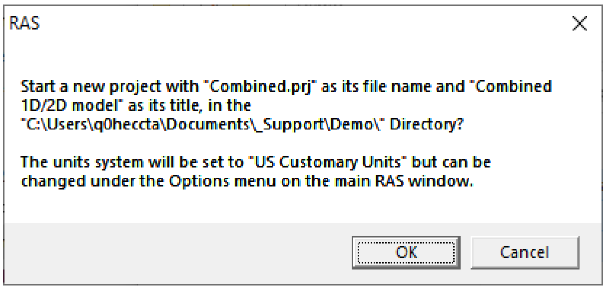
Before you can start building an HEC-RAS model, you need to create a new project. Choose the **File | New Project** menu item on the main RAS interface, as shown below.

1 https://www.hec.usace.army.mil/confluence/display/RASUM
2 https://www.hec.usace.army.mil/confluence/display/R2DUM
3 https://www.hec.usace.army.mil/confluence/display/RMUM/HEC-RAS+Mapper+User%27s+Manual

Provide a name (and filename for the project).



A window will informed you of the unit system to be used for the project.

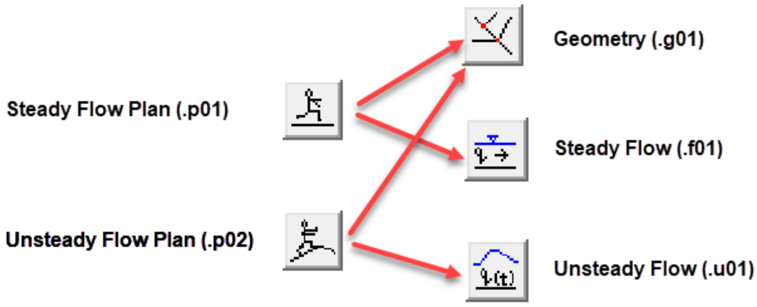


All project files will begin with the file name. In the example above, the model title is "Combined 1D/2D Model" and the project filename is "Combined.prj". When I create new data files (geometry, flow, plan, ...) the project name will be appended with with a suffix corresponding an enumeration for the type of file that is created.

So for this example, the following files will be created for the first of it's type.

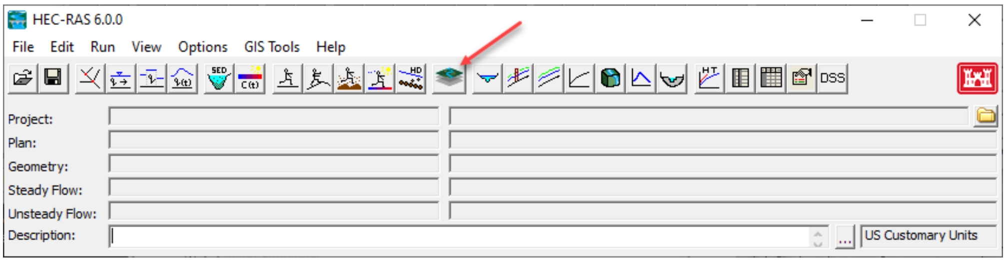
- Project File - Combined.prj
- Geometry File - Combined.g01
- Steady Flow File - Combined.f01
- Unsteady Flow File - Combined.u01
- Plan File - Combined.p01

A Plan is similar to an alternative and will be comprise of a geometry file and a flow file. A visual chart of the plan and file structure for HEC-RAS is shown below.



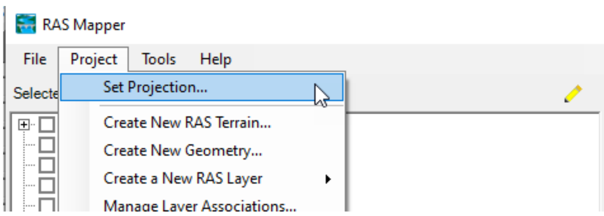
1.1.2 Base Project Data

RAS Mapper will be used to create the geometry for the HEC-RAS model. Click the **RAS Mapper** button from the main RAS interface.

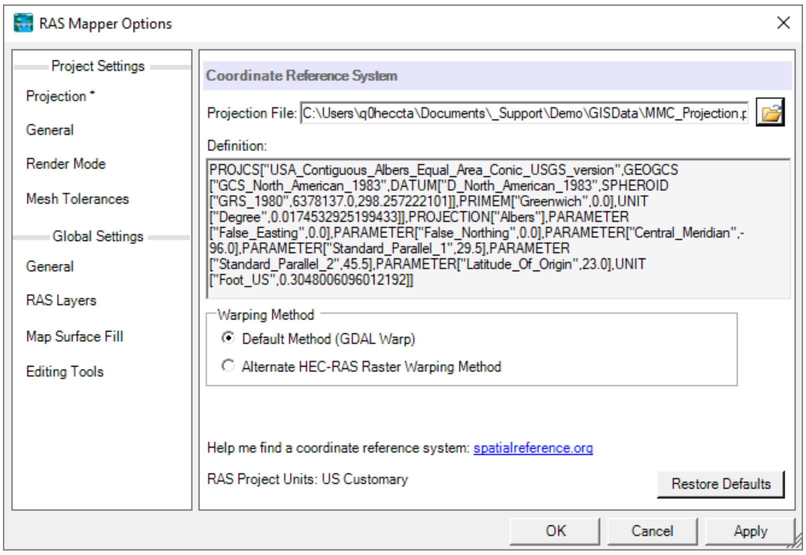


Projection

Set the coordinate system for the project. The projection will be used to reproject any data that is brought into RAS Mapper. This include terrain data and land cover data when you create new datasets or background data such as shapefiles and web imagery. To set the projection, choose the **Project | Set Projection** menu item.

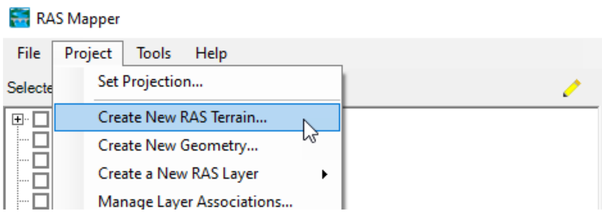


Select the **Browse** folder button to select an esri PRJ file. If GDAL doesn't recognize the projection, you will be provided a warning message.

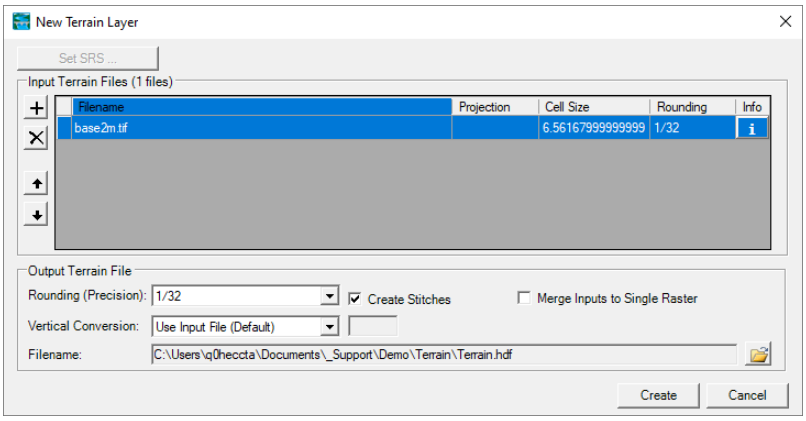


Terrain

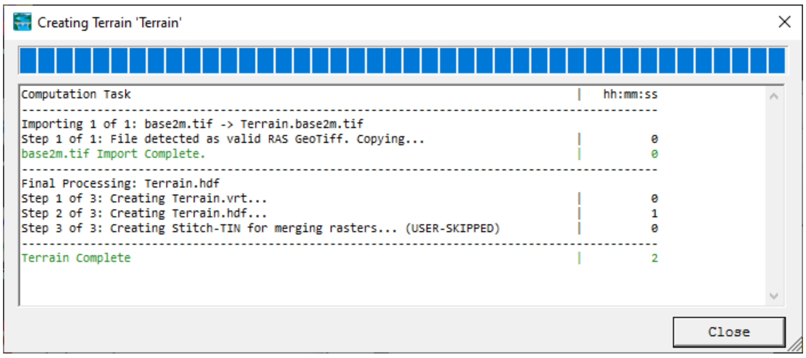
A good HEC-RAS model requires good terrain data. Especially when it comes to a 2D model as it is much more difficult to supplement data in a 2D model compared with a 1D cross sections. To begin using ground elevation data in HEC-RAS, select the **Project | Create New RAS Terrain** menu item in RAS Mapper.



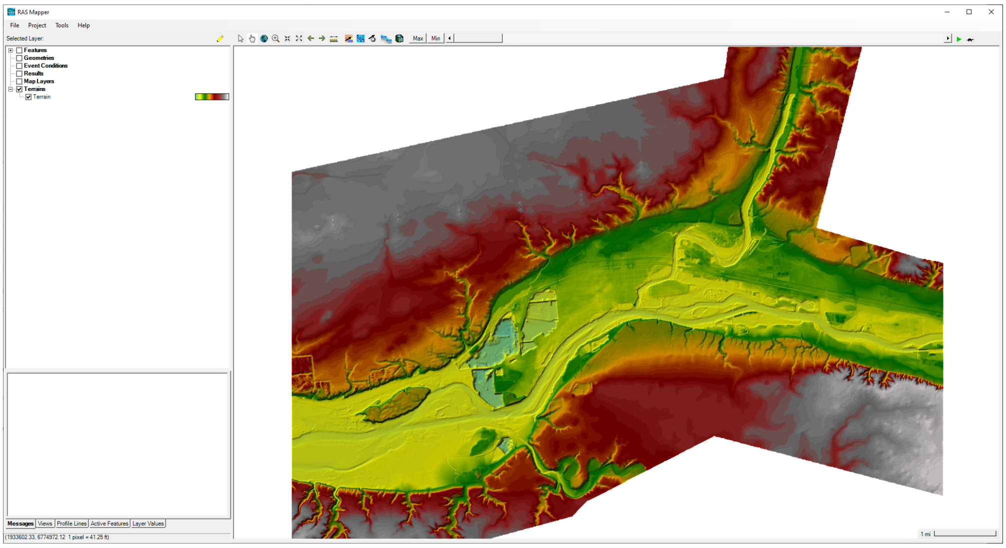
Select the terrain model(s) of interest. Specify the parameters for the new RAS Terrain (rounding, vertical conversion) and provide a unique filename.



Click the Create button. RAS Mapper will effectively import the terrain data (it creates a copy from the input dataset) based on the specified parameters. During the import process, the data will be rounded, converted, tiled (for zoom levels), and re-projected (if necessary) to the coordinate system specified.

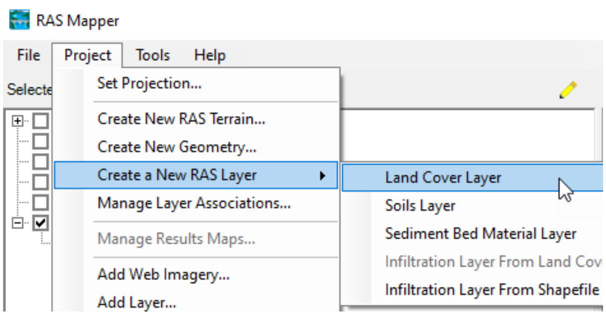


When complete, there will be new files created on disk: a Terrain.hdf file that contain information RAS Mapper uses to manage the terrain data, a Terrain.vrt file that can be used to visualize the terrain data in other programs, and a Terrain.Tif for the data that was imported (multiple .tif files will be created if the user specified multiple inputs). The Terrain.hdf file will then be loaded into RAS Mapper for visualization and use.

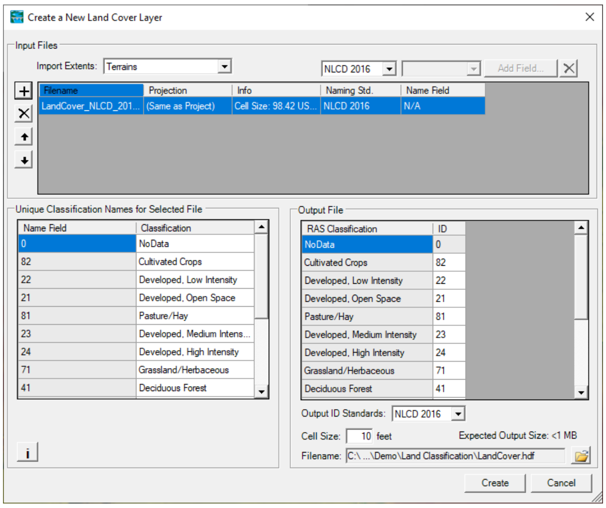


Land Cover Data

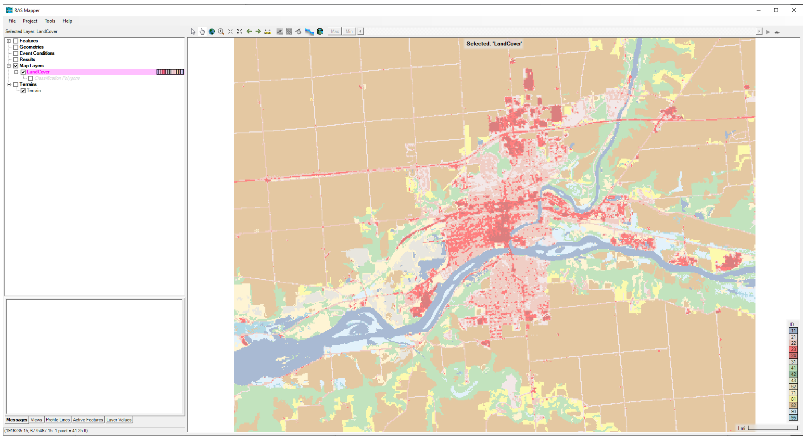
If you have a land cover datasets for use with Manning's n values, that might be the next thing you prepare for modeling. To import a land cover dataset, select **Project | Create a New RAS Layer | Land Cover Layer**.



Add the land cover layers of interest. At this time, you can re-classify the data if desired. Enter a cell size for new raster that is created and provide a unique filename. Click **Create** to create a byte raster (integer grid) of land cover data.

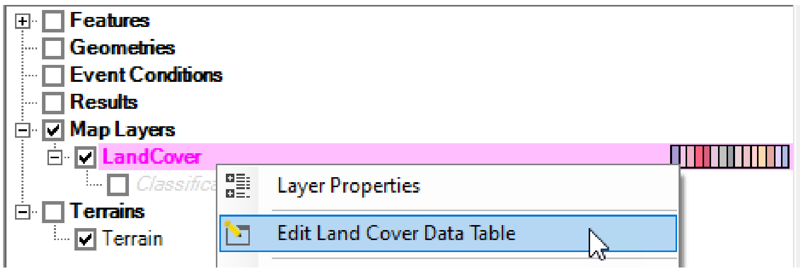


The data import will create a LandCover.hdf file for use in RAS Mapper and a LandCover.tif which holds the classification information. Once the data has been imported, it will be added to the map display in RAS Mapper.

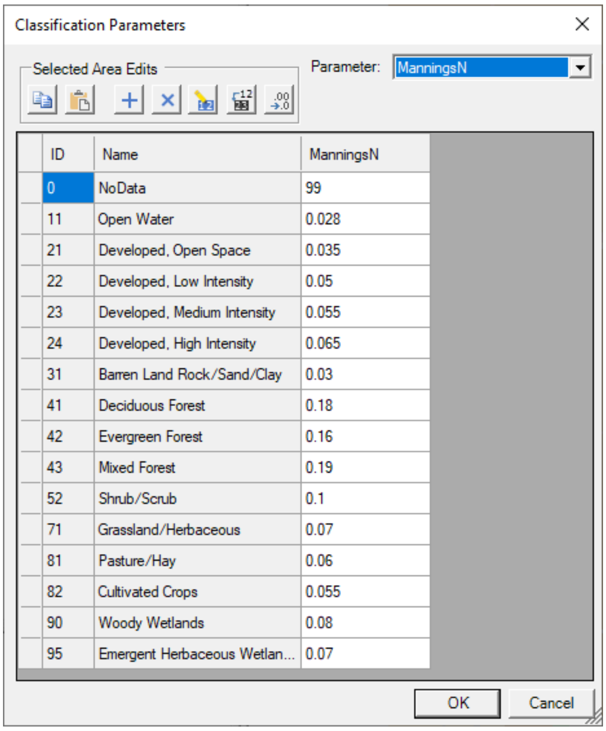


Set Manning's n Values

To assign Manning's n Values to the land cover dataset, right-click on the land cover dataset and choose the **Edit Land Cover Data Table**.



The Classification Parameters table will open and allow you to enter an n value for each classification type.

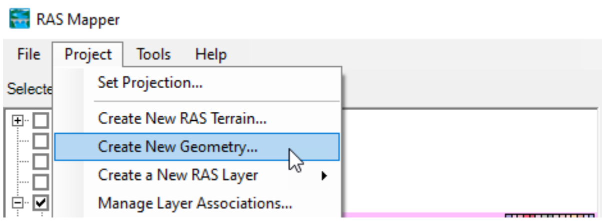


Classification Polygons

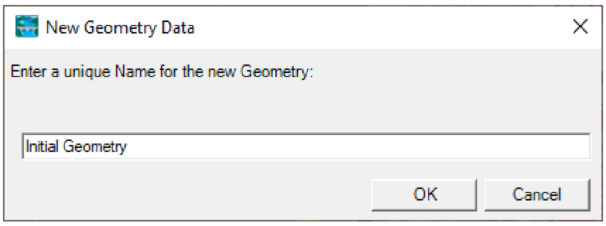
Land cover data can be refined using vector data on the land cover raster. The Classification Polygons are included as a child layer and can be used to reclassify the land cover data for areas where you wish to have more detailed data, such as the channel.

1.1.3 Create a New Geometry

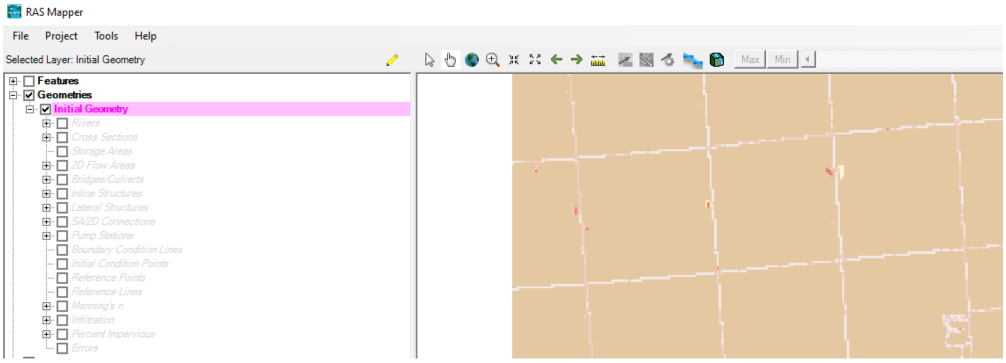
In RAS Mapper, select the **Project | Create New Geometry** menu item.



Provide a unique name for the geometry.



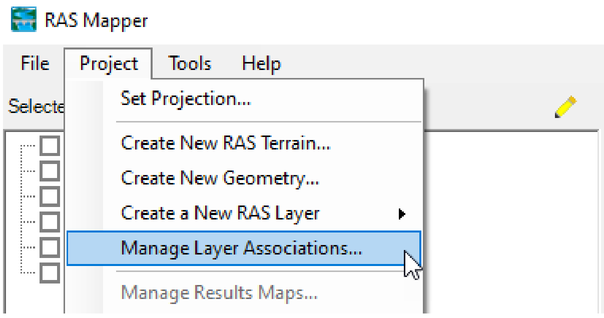
Press **OK** to create a new Geometry and add iit to RAS Mapper. Layers without data are shown in grey and as you create data the layer names will turn black and the layer's symbology will be shown.



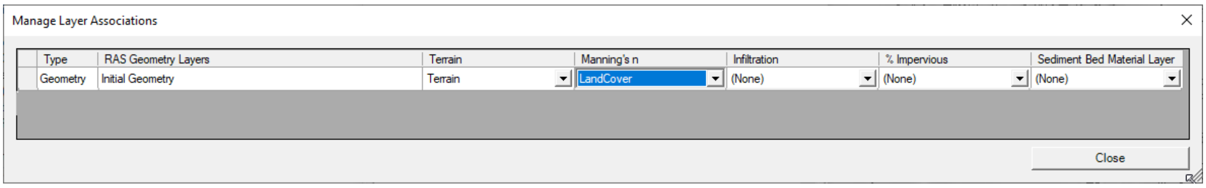
Once a Geometry has been added, it can then be place in editing mode to start creating model geometry.

Associate Base Data

Before you start creating model geometer, you need to associate the Terrain layer and the Land Cover layer with the Geometry. Select the **Project | Manage Associations** menu item.

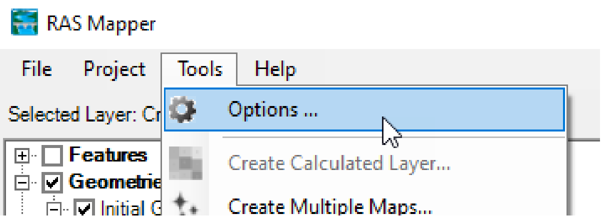


Associate the **Terrain** and the **Manning's n** dataset in the dialog that is provided. Press the **Close** button when finished.

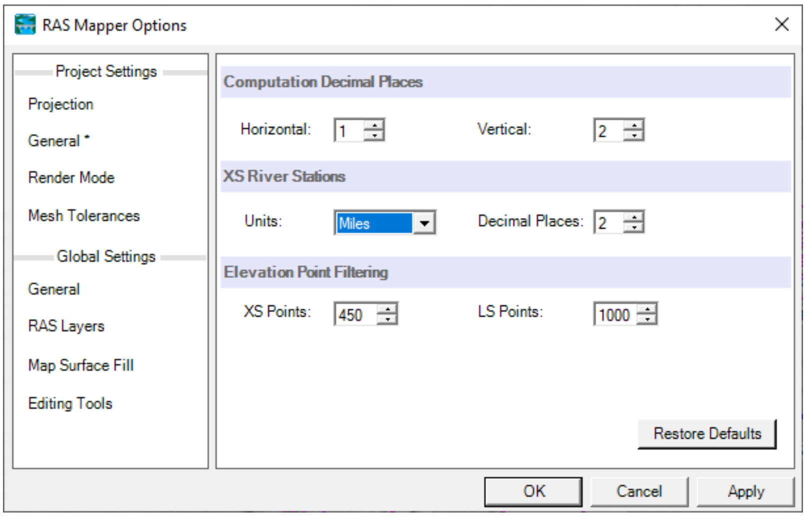


Options

There are several options available from within RAS Mapper to customize the data extraction and visualization. Access the Options from the **Tools | Options** menu item.



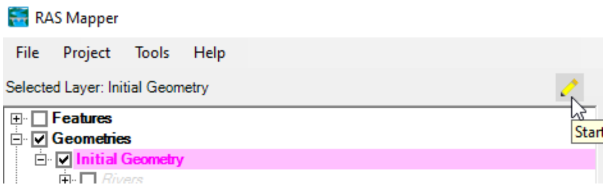
From the Option dialog you can set River Station unit system, elevation point filtering, RAS Layer symbology, Editing Tools, and many others.



1.1.4 Create a 1D Geometry

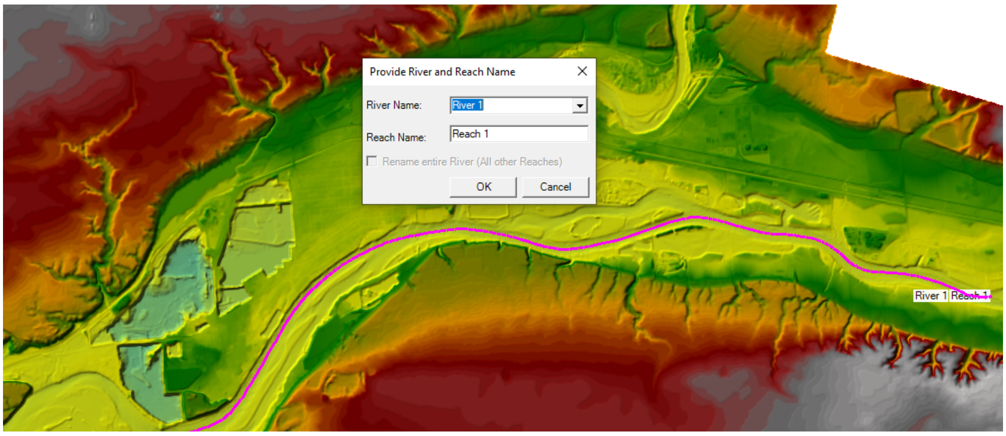
The first step in creating any steady or unsteady model is to create a steady, 1D model to get an understanding of the river system. Start creating the combined 1D/2D model by Editing the initial Geometry.

To start editing a geometry, select the Geometry layer of interest an click the **Start Editing** button (this button will be replaced with the **Stop Editing** button).

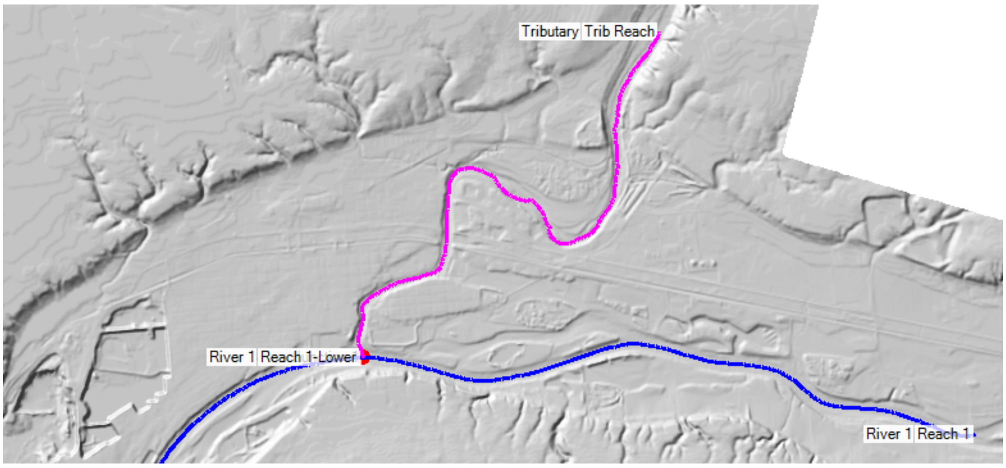


River

The first thing to create in a 1D model, is to digitize the river system. The River Layer is used to hold the river system. Select the Rivers Layer, and using the Add New Feature tool, create a river centerline. River centerlines are created from upstream to downstream through the main portion of the channel. To finish creating a river line, double-click the polyline. This will close the line and invoke the River and Reach Name dialog. Provide a unique river and reach name.

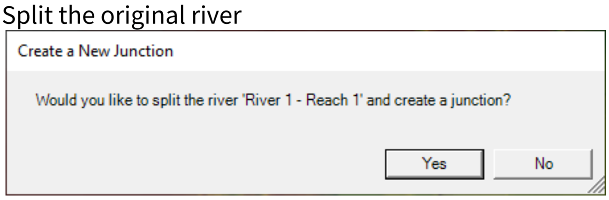


To add a another river, such a tributary, create another river reach.



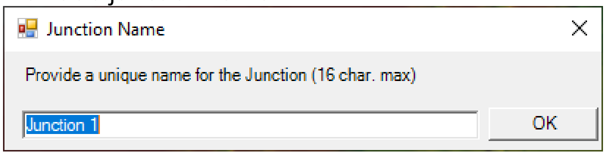
Double-click to end the tributary on the main river. If you are close to the original river (look for the red circle at the end of the river reach), you will be asked to create a junction. In RAS, a junction signifies a location to combine (or split) flow. The steps in creating a junction are listed below.

1. Split the original river



2. Rename the new river reach

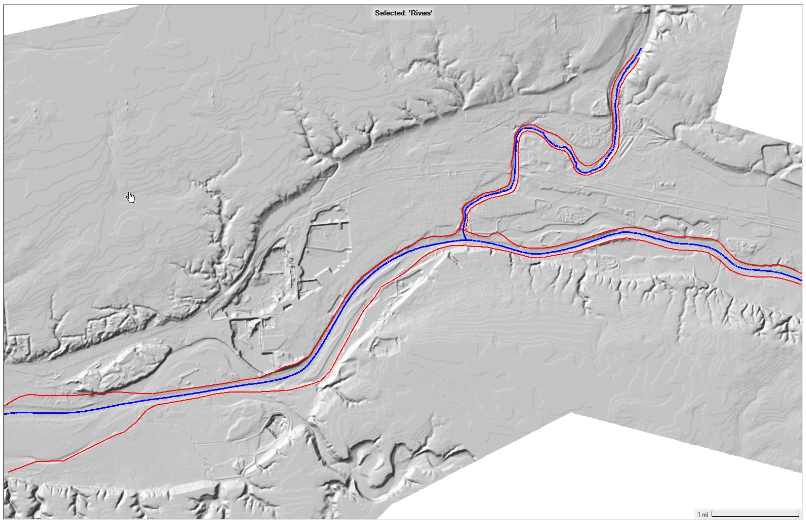
3. Provide a junction name



Junction information should be edited from the Geometric Schematic. Verify that the junctions lengths are appropriate and the desired modeling method is selected.

Bank Lines

Bank Lines are used to identify the main channel conveyance from the the overbanks. Create a bank line for the left and right overbanks for each river. The bank lines will also be used for creating an Interpolation Surface that is used for mapping results. Therefore, be careful to place the bank line as precisely as possible for where flow separation from the main channel and overbanks will occur. It is often helpful to use aerial imagery to assist in locating bank lines. Obviously, refinement of this layer will be necessary as you gain insight into the river system. Example bank lines are shown in the figure below.



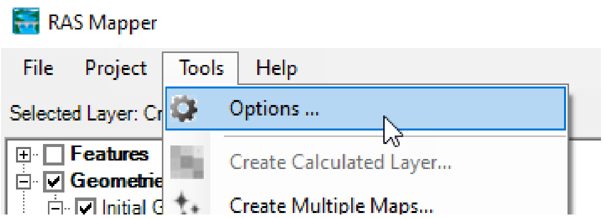
Flow Path Lines

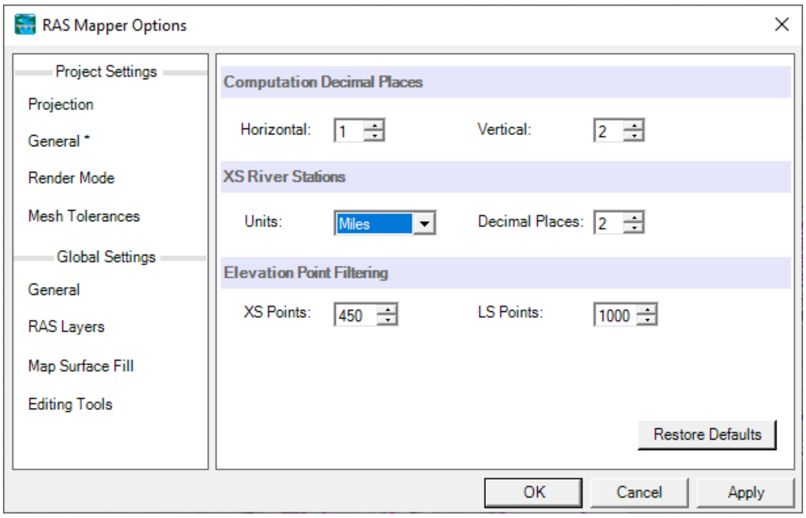
Flow Path Lines are used to compute the reach lengths for the left and right overbank. The River Lines will be used to compute the reach length in the main channel. The flow path lines should follow the center-of-mass of flow in the overbank.

Cross Sections

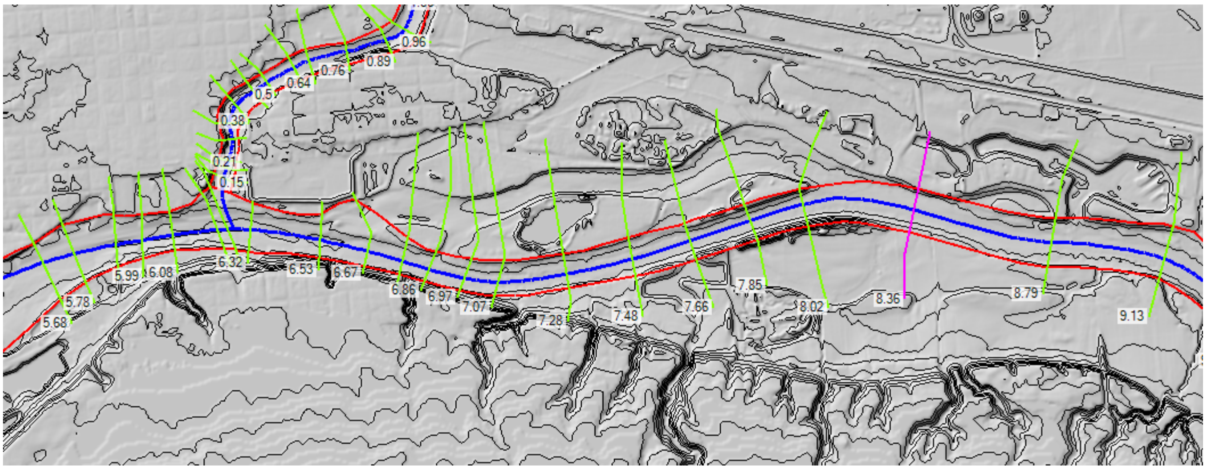
For the 1D portions of the model create cross sections covering the entire floodplain for the range of flows to be modeled. Cross sections should be located so that they capture controlling locations along the river as well as being created close enough together to smoothly capture the changes in terrain. A handy, back of the envelop, way to think about it is that no cross section should be no farther away that how wide it is. Cross sections should be constructed in the main channel and overbanks such that they are dog-legged perpendicular to flow. Having the bank lines and flow path lines available will assist in properly laying out cross sections.

Before laying out cross sections, be sure to set the River Station units for "numbering" the cross sections. The default units are feet/meters but for large river you should choose miles/kilometers. As you layout cross sections, RAS will compute the river station for the cross section and make the river station unique. So if you place the cross sections close together, RAS Mapper will keep incrementing the number of decimal places until the river station differs from its neighbor.

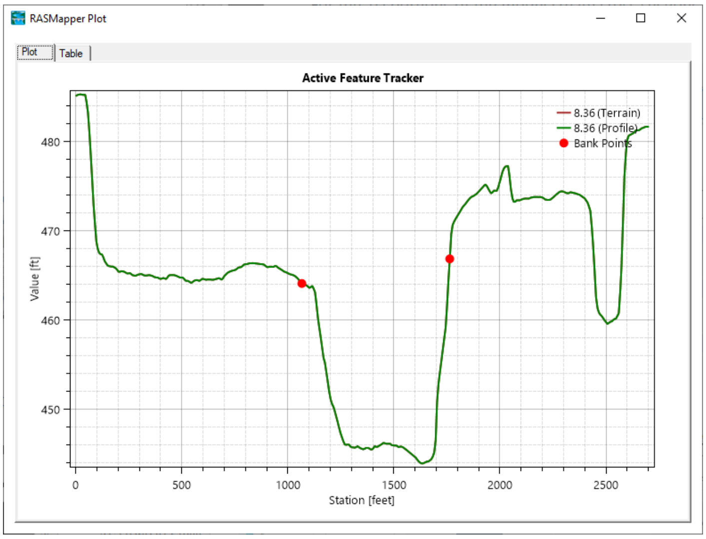




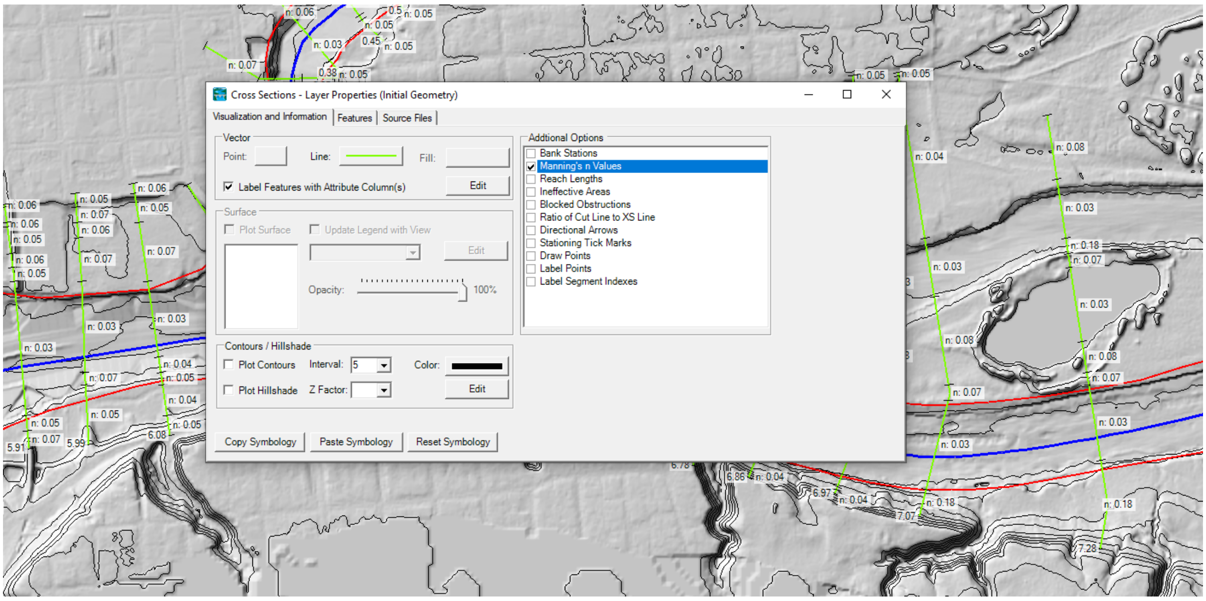
Another helpful option in RAS Mapper is to turn on the Contour Lines on the Terrain layer. The image below shows the location of cross sections on the Terrain with the river line, bank line, and contour lines.



If you want preview what the cross section will look like, use the the Cross Section Plot tool. The cross section will update each time you update a point on the cross section.



As you lay out and edit cross section locations, data will automatically be extracted. By default, River Stations will be plotted at the start of each cross section. Additional Plot Options available for the cross sections are accessed on the Layer Properties.



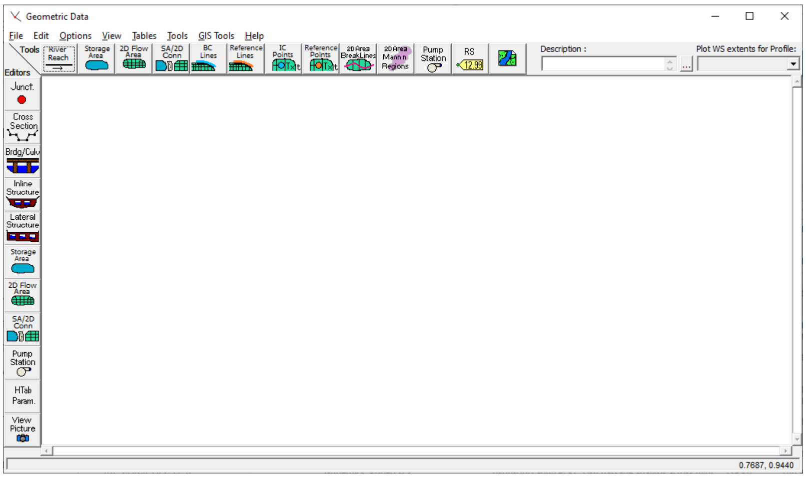
Additional Data

There are a multitude of addition 1D data you might want to create, from Ineffective Flow Areas, Blocked Obstructions, Bridges, Inline Structure, and Storage Areas. We will skip these data at this time.

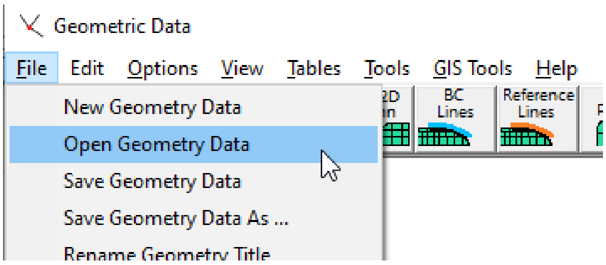
Geometric Data Editor

When finished creating data in RAS Mapper, **Stop Editing** and close RAS Mapper.

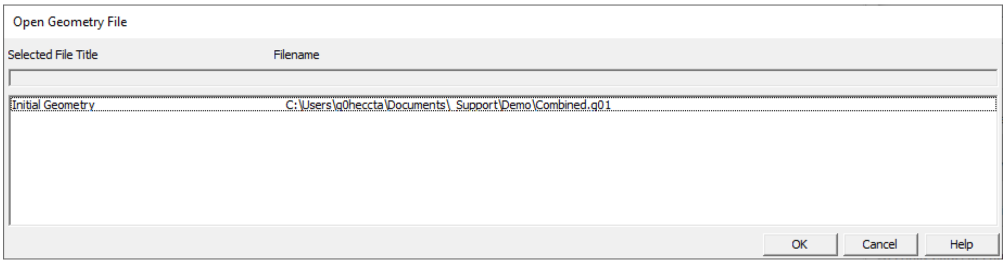
Open the **Geometric Data Editor** from the main HEC-RAS interface.



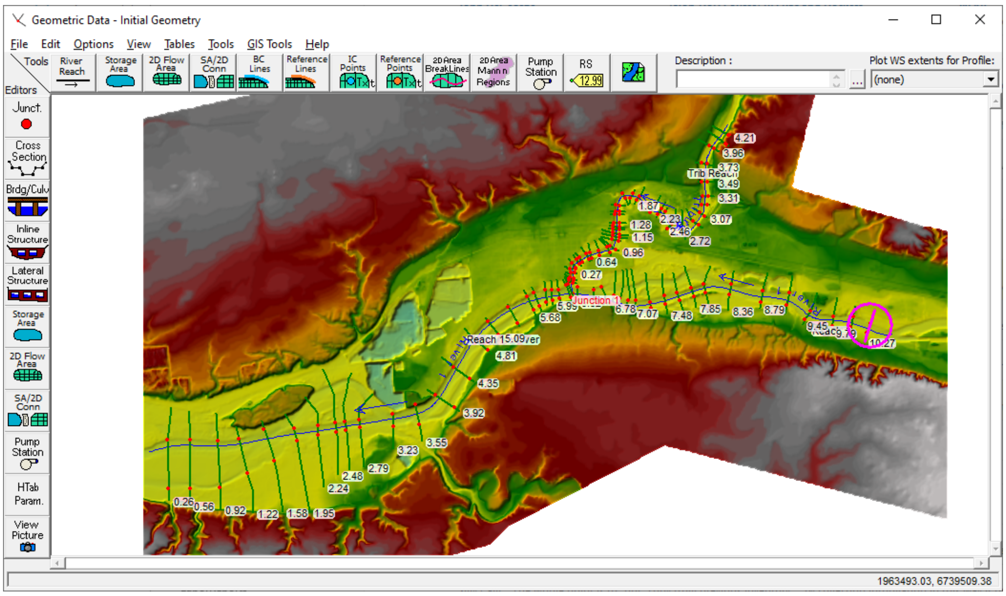
Load the Geometry by selecting **File | Open Geometry Data**.



Select the geometry you were working on in RAS Mapper.



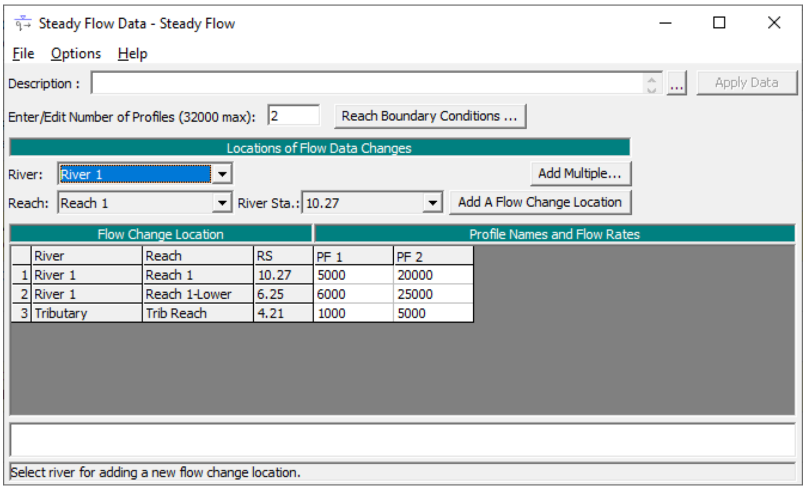
You may need to complete some of the geometry such as Manning's n values, filtering cross section points, adding ineffective flow areas, etc.



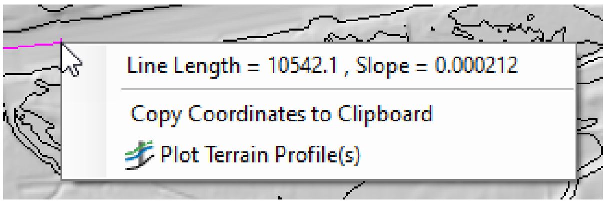
When you have finished making obvious edits, close the Geometric Data Editor.

1.1.5 Create 1D Steady Flow Data

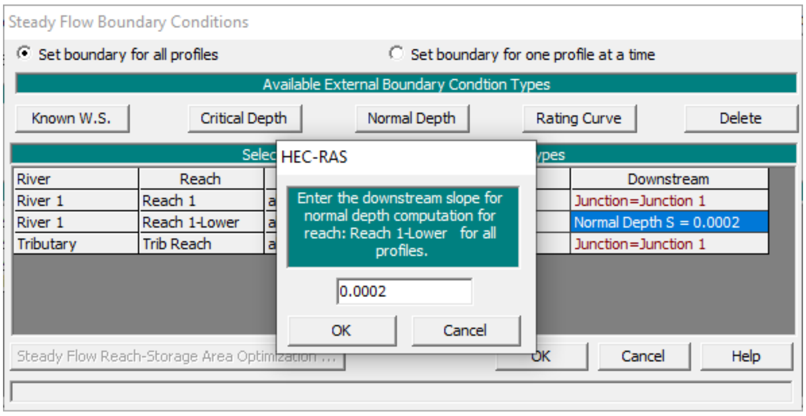
Open the Steady Flow Data Editor from the main HEC-RAS interface. Enter flow data for the range of flows you expect to model. Depending on the river system, entering a low flow like the 2-year flow can help identify the main channel and verify the location of bank stations. Entering a high flow will help you identify the entire floodplain and verify the cross section extents.



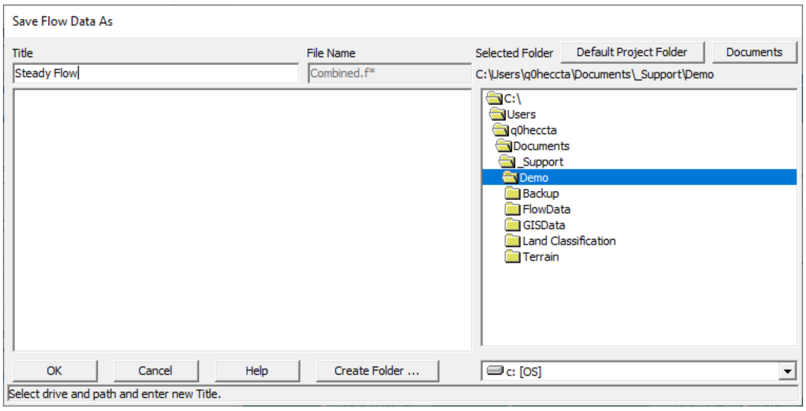
Next, enter a downstream boundary condition. Normal depth is a quick way to enter a boundary condition. Of course, you should have extended the downstream portion of the model downstream of the main area of interest and selected a highly 1-dimensional portion of the river. First, figure out the general slope of the river. This can be done in RAS Mapper, but selecting the measure **Measure Tool** and digitizing a portion of the river. After double-clicking to end the draw, the slope will be reported.



Enter the Normal Depth Slope but click on the cell for the downstream boundary of the river and clicking the **Normal Depth** button.

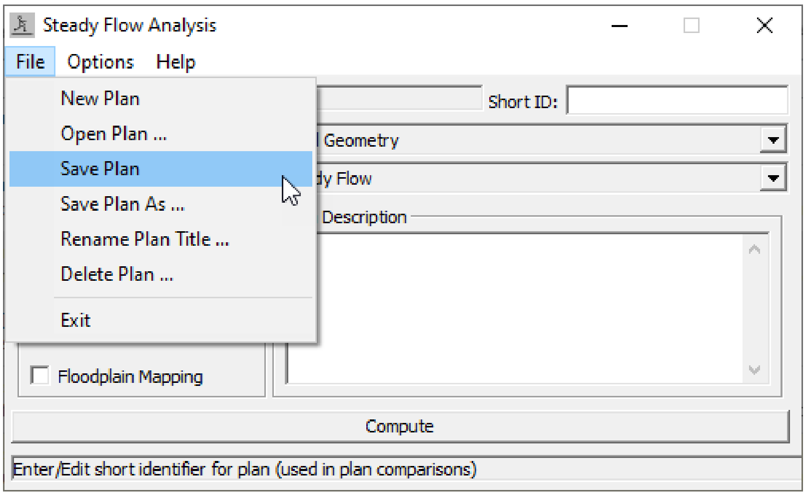


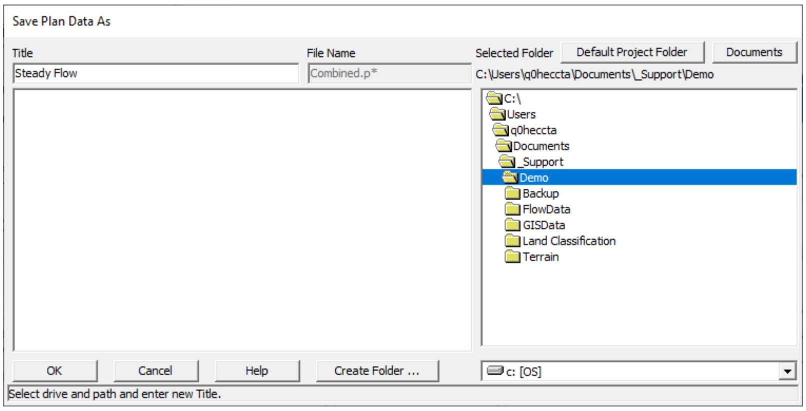
Save the flow data before moving to on to create a steady flow plan.



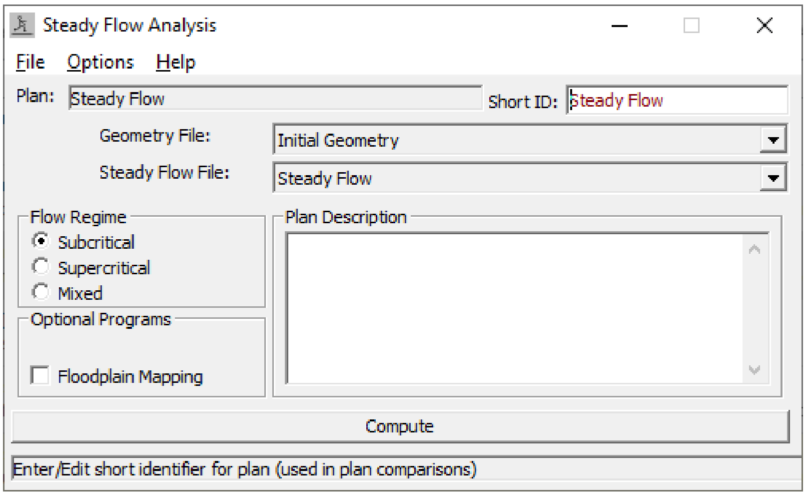
1.1.6 Create 1D Steady Flow Plan

In order to run the HEC-RAS model, create a Steady Flow plan using the initial geometry and steady flow data. From the Steady Flow Analysis window, select Save Plan and provide a plan name. You will also be prompted for a Short ID which is used for labeling output plots.

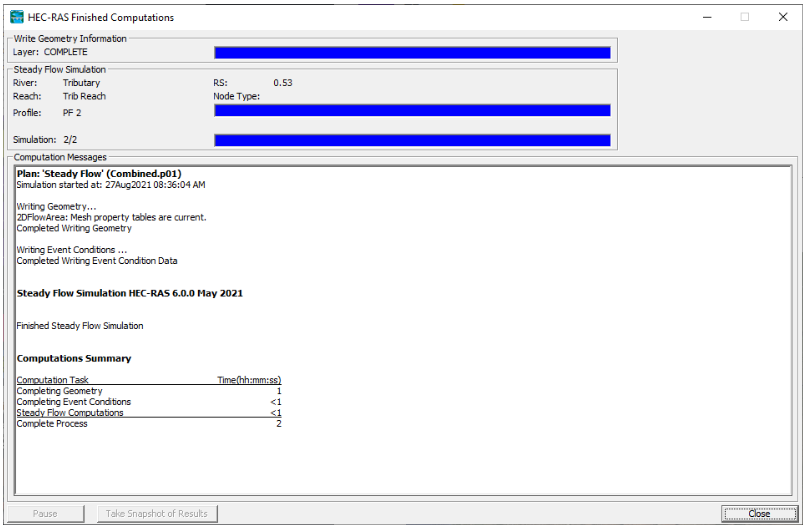




Once a plan is created, you are ready to compute hydraulic results.



Compute water surface profiles by clicking the Compute button. Provided data has been adequately entered, the interface will provide status messaging during the steady flow run.



After a completed run, you will be able to examine the model results and refine your model.

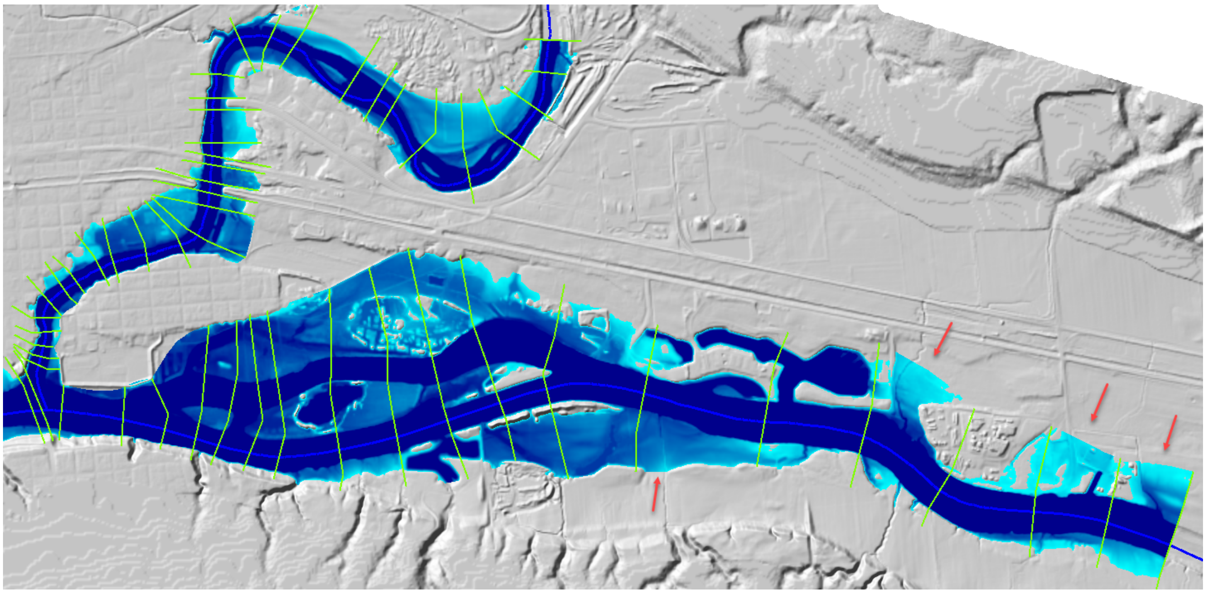
1.1.7 1D Model Evaluation and Refinement

Often when developing a model for a river system, the modeler comes the the river with little knowledge of how the system reacts to various flow conditions. While developing the river centerline, flow bath lines, and bank lines you can get a sense for how the river and floodplain will behave. However, the inundation depth, velocity, and boundary visualization provides a "first look" into the soul of the system. Developing the 1D river hydraulics model allows you to quickly gain an enormous amount of insight. Quickly, you will figure out just how little you actually know about the river...and how much you need to modify your initial geometry, before you develop the more complicated unsteady flow model.

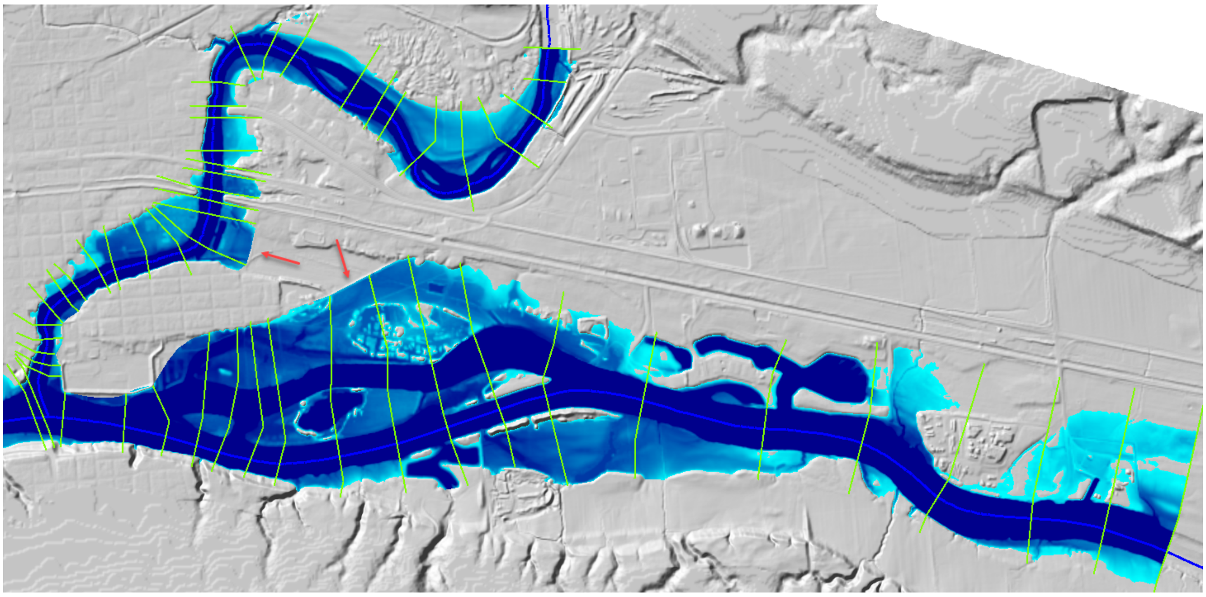
Cross Section Improvements

Running the steady flow model with the range of flows is allows you to quickly identify how to improve cross section layout. Using the simulation results, you will see locations where channel banks should be adjusted, locations to add cross sections (or remove), extend cross sections (or shorten), re-align to be perpendicular to flow, or to improve for inundation mapping.

In the below example, there are several cross sections (shown with red arrows) where the floodplain has been limited based on the cross section layout. There may be alternatives to cross section extension, however, for this example it is an appropriate improvement.

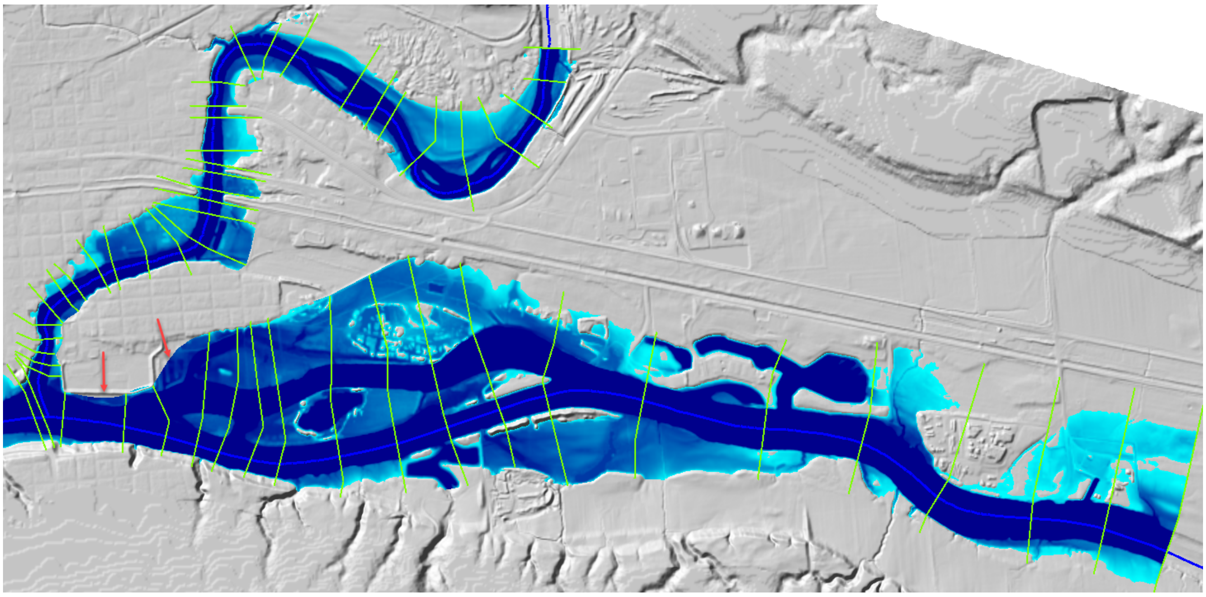


For this example, the river channels and floodplain are fairly well defined and reshaping the cross sections was not necessary to improve the model. However, there are still several locations which require improvement. As shown in the image below, there are additional locations (shown with red arrows) where the cross sections should be extended to capture the floodplain. We will improve these designated locations with a 2D Flow Area in a later step.

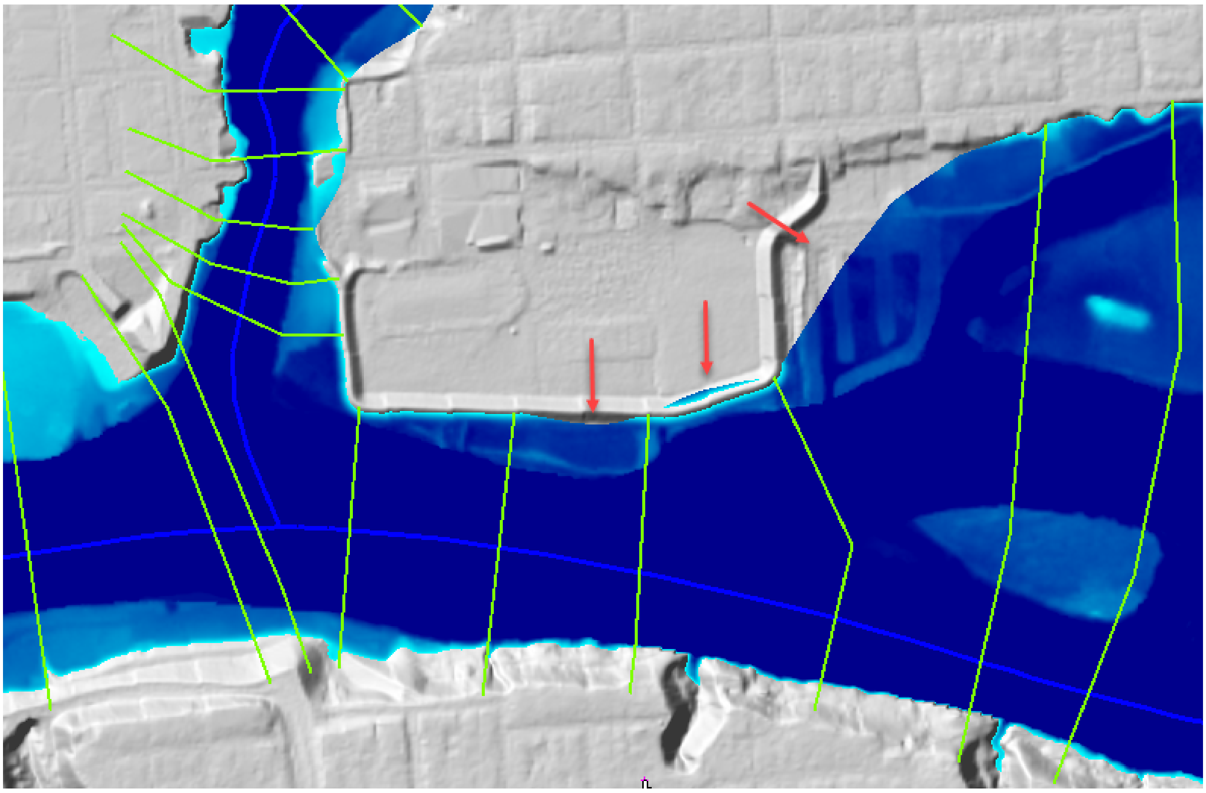


Inundation Mapping Improvements

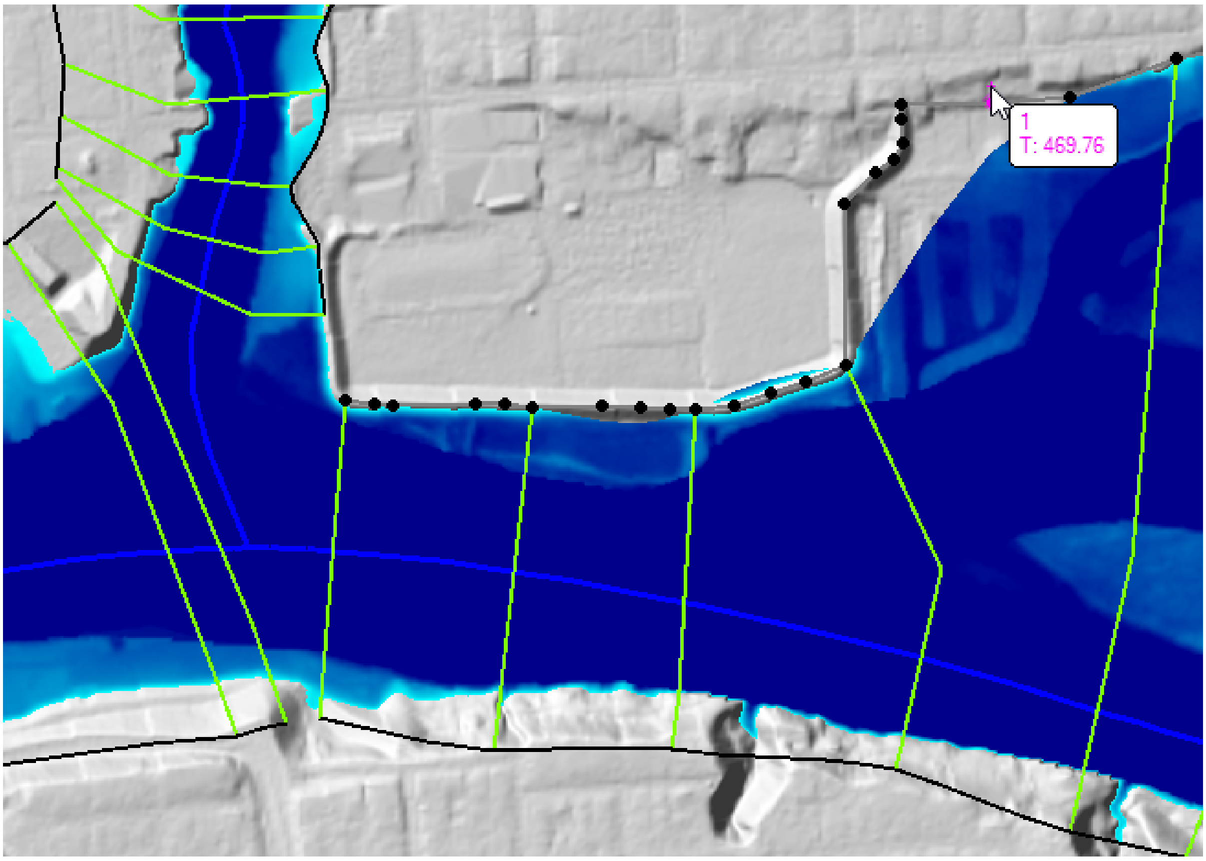
There are several other locations where the inundation mapping is not correct. This is especially visible around the levee at the confluence of the rivers. Why are there problems with the inundation? HEC-RAS connects the ends of the cross sections with what are called "edge lines" and the shape of them is created based on the shape of the river and bank lines. The most downstream part of the river we can improve by adding an additional cross section.

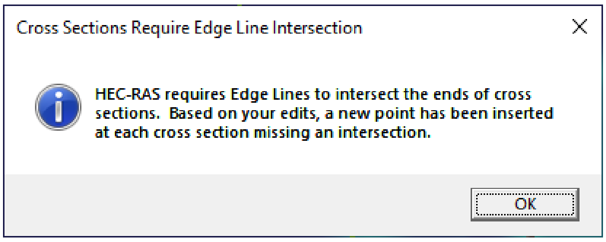


Improving the cross section can fix many mapping issues that you may run into. However, with enough modeling you will soon find that no amount of cross section manipulation can give you the mapping results you want the model to accurately reflect the hydraulic results. Take a look at the levee system and inundation mapping inconsistencies. Even with adding the new cross section, the river-side of the levee is still dry. There are still the problems on flooding on the interior side.

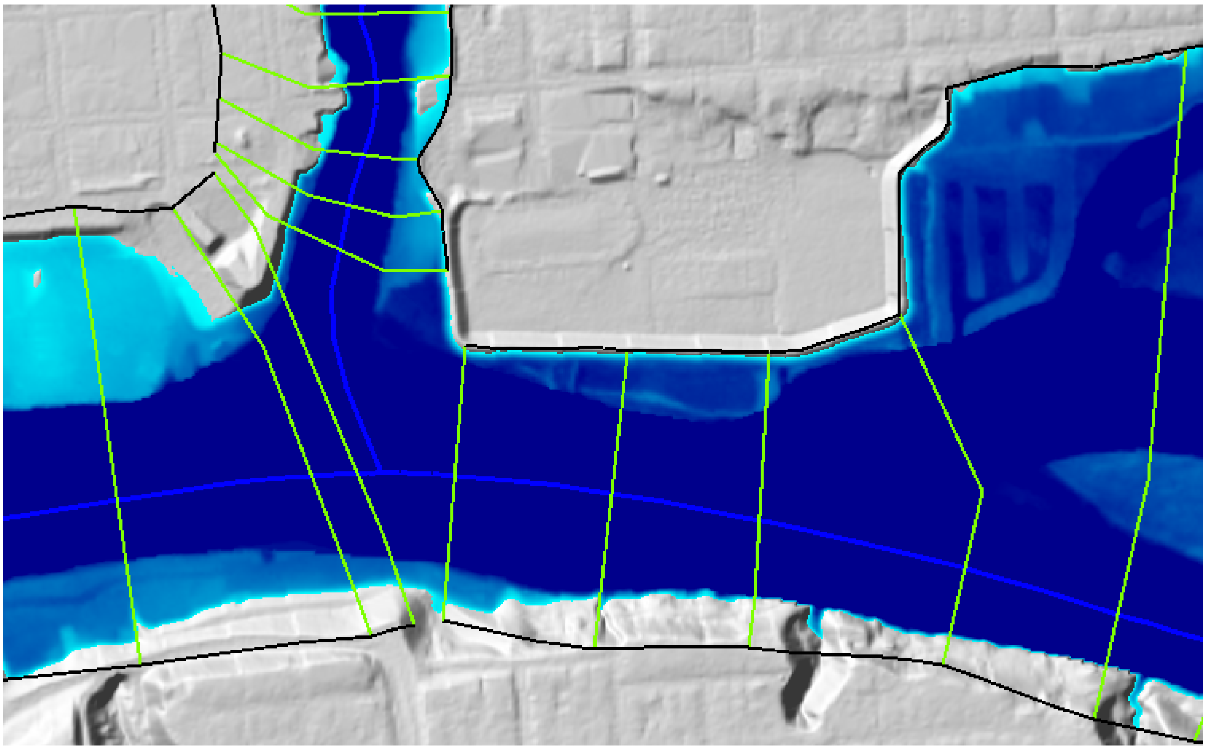


To improve the inundation mapping, you have the ability to edit the Edge Lines layer (grouped under the Cross Sections Layer). When you stop editing an edge line feature, RAS will make sure that the edge lines are connected with the end of the cross sections and provide a warning message that the edge lines are going to be modified. Further, the Edge Lines layer will now be saved in the results output during the simulation.





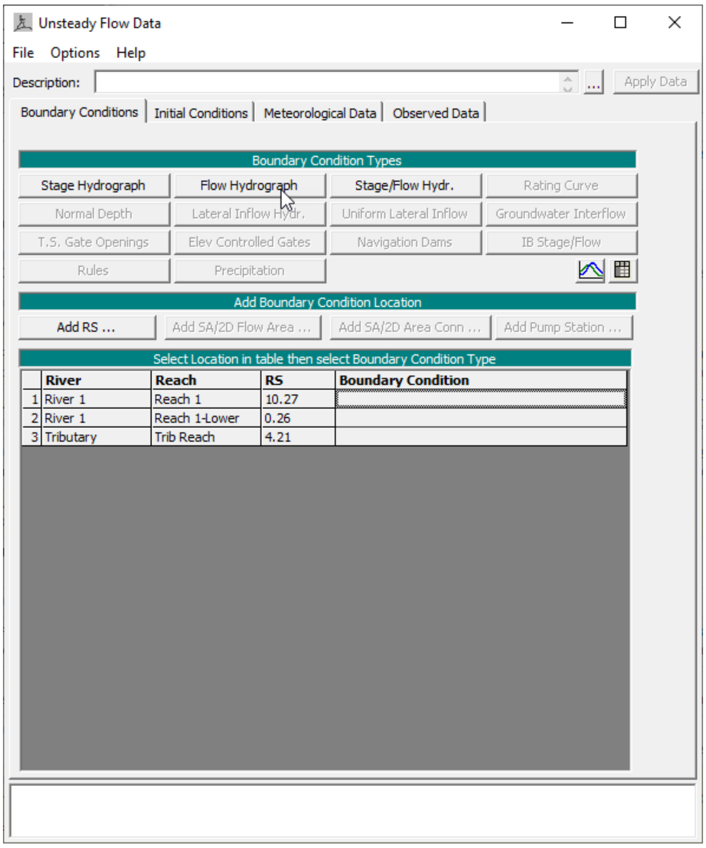
The resulting floodplain mapping is now hydraulically correct, as shown below.



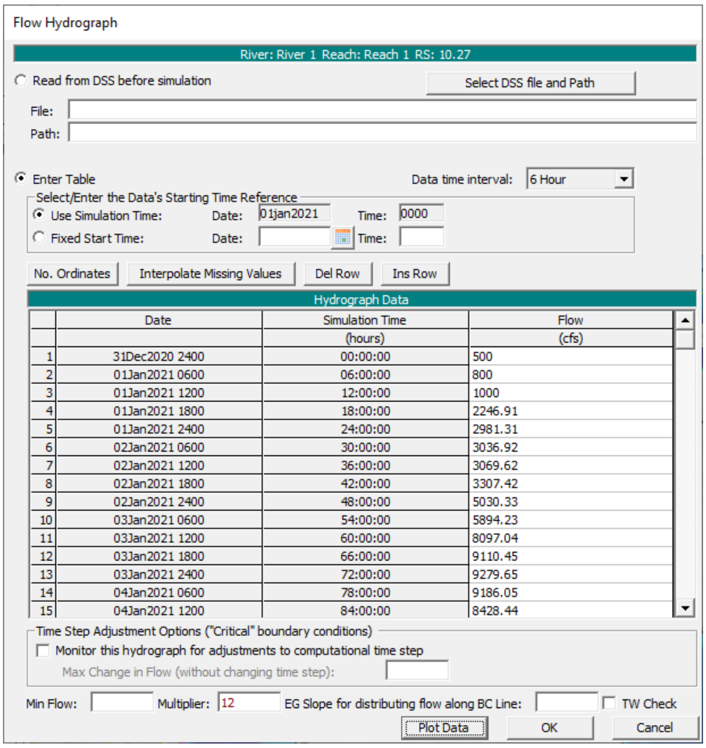
1.1.8 Create 1D Unsteady Flow Data

After creating, examining, and refining the 1D model it is time to move on to an unsteady flow model. Begin by opening the Unsteady Flow Data editor and providing unsteady flow data and boundary conditions.

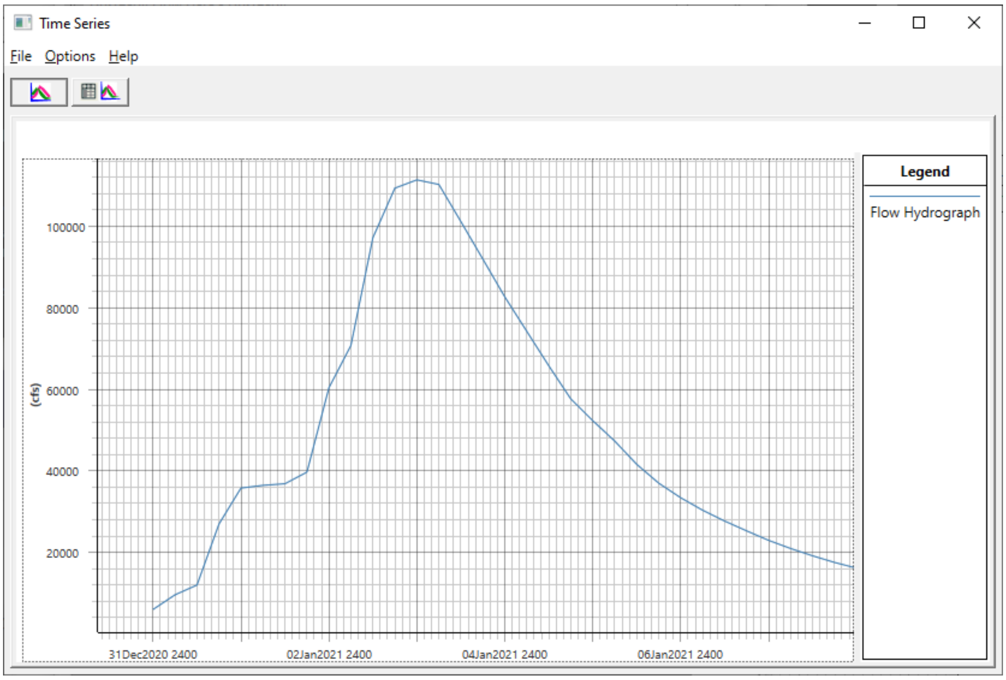
For this example we have a two river, three reach system. Therefore (at a minimum), we need to specify flow hydrographs at the upstream river stations for each river and specify the downstream boundary condition. We will use the same Normal Depth boundary condition as we used in the steady flow simulation. Use the Flow Hydrograph option for each river. To add a flow data, select the cell corresponding to the river reach and click the **Flow Hydrograph** button.



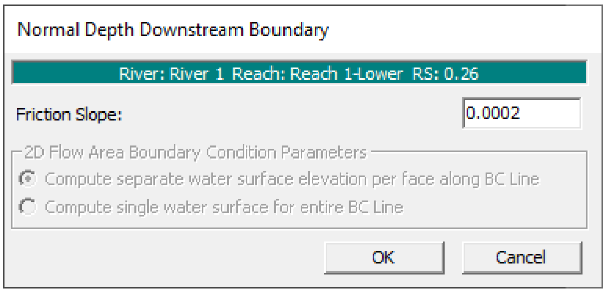
Enter the flow hydrograph using the provided table or use a connection to a DSS file. An example hydrograph data entry is shown below.



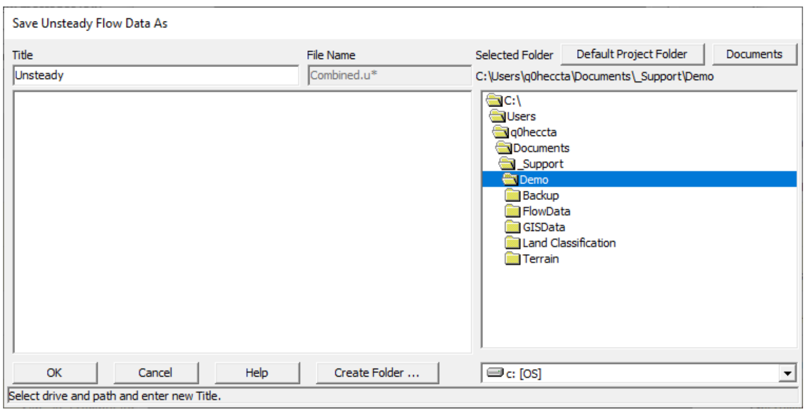
After entering information, always use the **Plot Data** button to verify the data entry! An example hydrograph plot is shown below.



Enter the downstream boundary by selecting the corresponding cell and clicking the **Normal Depth** button. Enter the normal depth slope as shown in the figure below and press **OK**.



Save the unsteady flow data using a useful name.

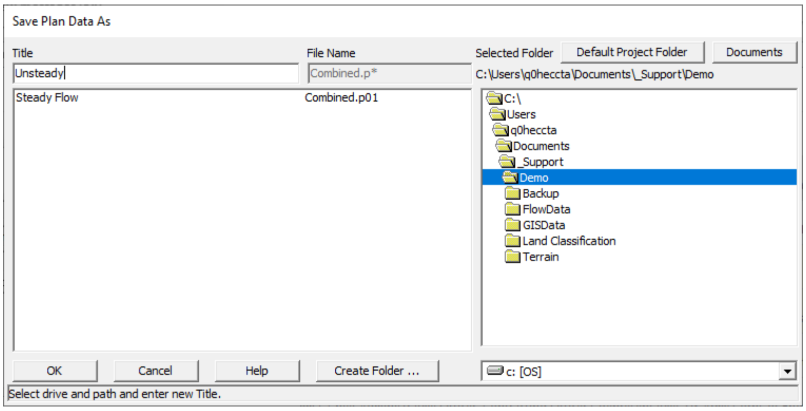


You are now ready to create an unsteady flow plan and simulate.

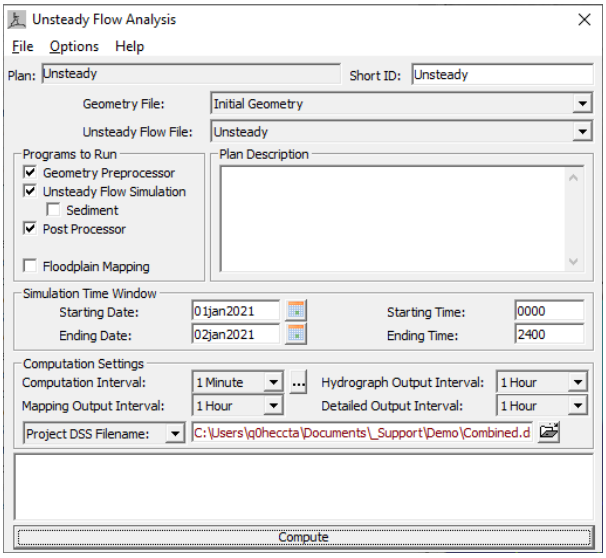
1.1.9 Create 1D Unsteady Flow Plan

Create the initial unsteady flow plan. Most likely there will be some data that you have need to complete, which the interface should report to you, or you model will go unstable. In either case, make sure to make thoughtful decisions for the Simulation Time Window, Computational Time Step, and other Computational Options and Tolerances.

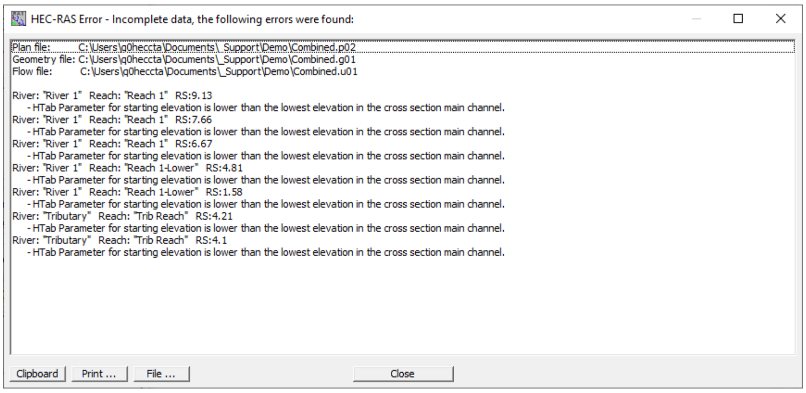
Save the plan with a descriptive name.



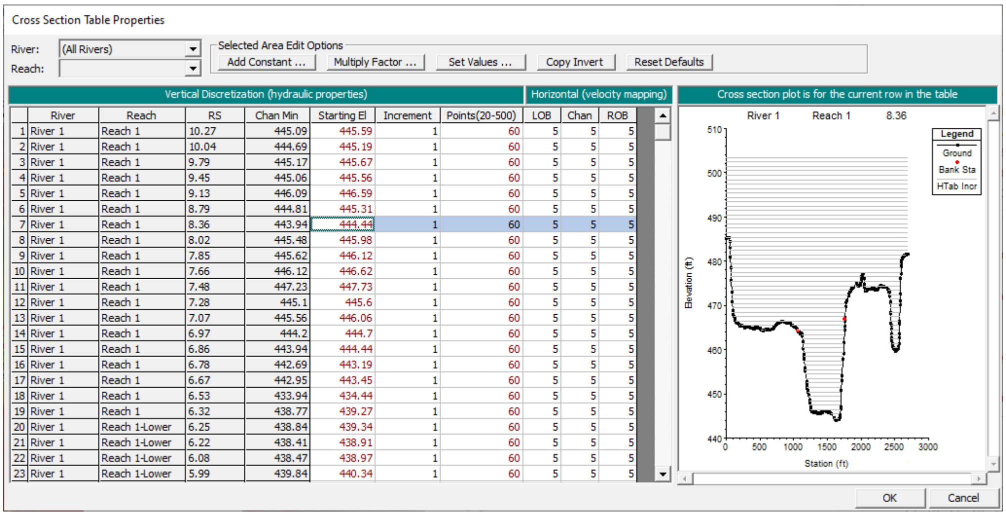
Setup the unsteady flow plan with a simulation time window that is short before you try to running a long simulation. Evaluate a time step that will satisfy the courant condition. Time step selection will be based on the cross section spacing and velocities.



A common error when moving from a steady flow to unsteady flow simulation is to forgot to set up the hydraulic table parameters for cross sections. If the parameters are not set, HEC-RAS will report the missing data when the simulation attempts to begin.



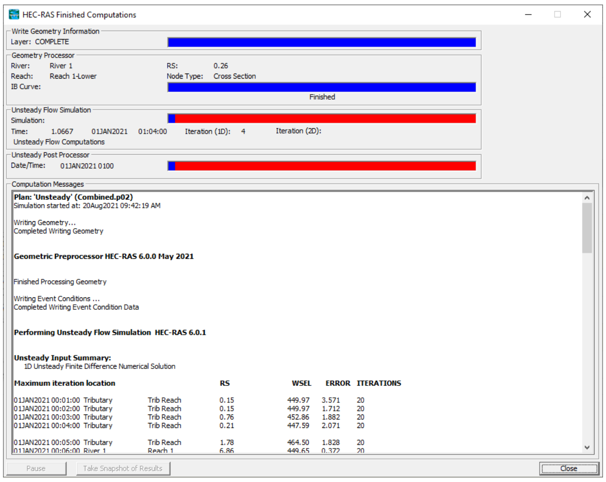
To update the cross section table properties, select the **Hydraulic Table Parameters** button from the Geometric Data Editor. Copy the invert to the Starting Elevation column and then add a value (the default used by RAS is 0.5ft). Set the number of points to cover the full range of water surface elevations expected to be generated from the range of flows. You can evaluate the stage range by scrolling through each cross section and looking at the vertical slicing in the plot on the right side of the window.



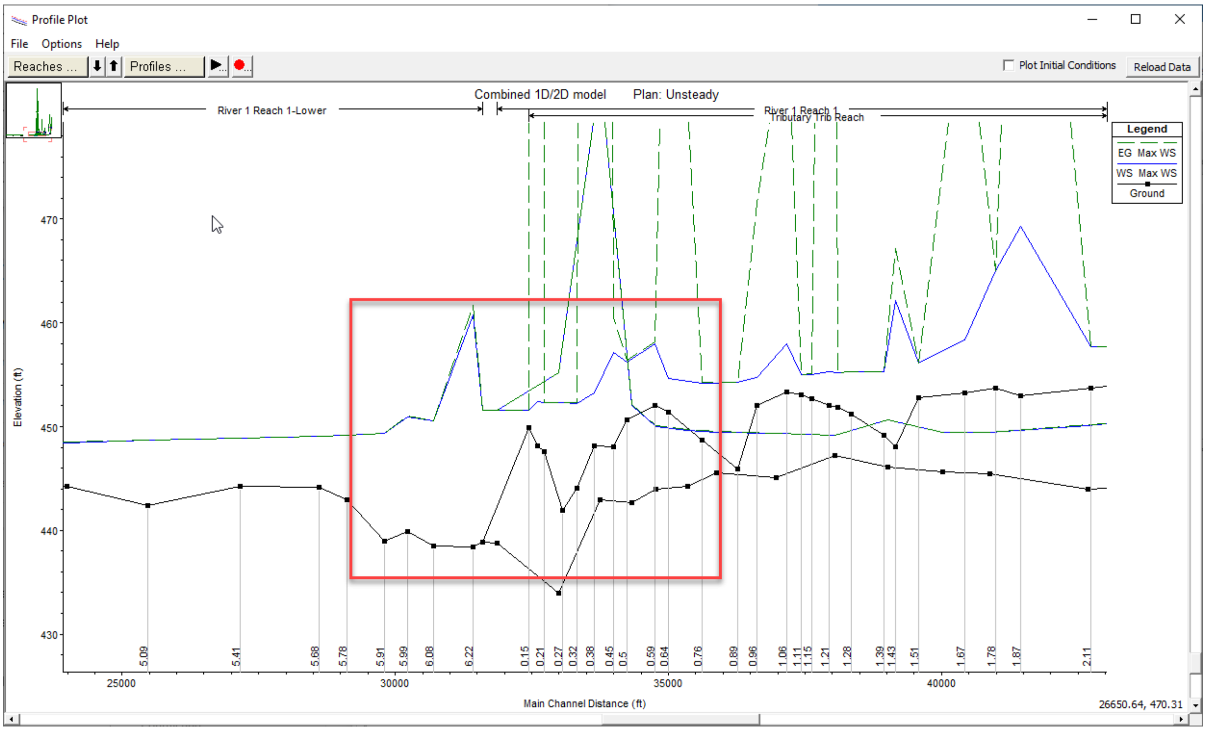
	Vertical Discretization (hydraulic properties)							Horizontal (velocity mapping)		
	River	Reach	RS	Chan Min	Starting El	Increment	Points(20-500)	LOB	Chan	ROB
1	River 1	Reach 1	10.27	445.09	445.59	1	60	5	5	5
2	River 1	Reach 1	10.04	444.69	445.19	1	60	5	5	5
3	River 1	Reach 1	9.79	445.17	445.67	1	60	5	5	5
4	River 1	Reach 1	9.45	445.06	445.56	1	60	5	5	5
5	River 1	Reach 1	9.13	446.09	446.59	1	60	5	5	5
6	River 1	Reach 1	8.79	444.81	445.31	1	60	5	5	5
7	River 1	Reach 1	8.36	443.94	444.44	1	60	5	5	5
8	River 1	Reach 1	8.02	445.48	445.98	1	60	5	5	5
9	River 1	Reach 1	7.85	445.62	446.12	1	60	5	5	5
10	River 1	Reach 1	7.66	446.12	446.62	1	60	5	5	5
11	River 1	Reach 1	7.48	447.23	447.73	1	60	5	5	5
12	River 1	Reach 1	7.28	445.1	445.6	1	60	5	5	5
13	River 1	Reach 1	7.07	445.56	446.06	1	60	5	5	5
14	River 1	Reach 1	6.97	444.2	444.7	1	60	5	5	5
15	River 1	Reach 1	6.86	443.94	444.44	1	60	5	5	5
16	River 1	Reach 1	6.78	442.69	443.19	1	60	5	5	5
17	River 1	Reach 1	6.67	442.95	443.45	1	60	5	5	5
18	River 1	Reach 1	6.53	433.94	434.44	1	60	5	5	5
19	River 1	Reach 1	6.32	438.77	439.27	1	60	5	5	5
20	River 1	Reach 1-Lower	6.25	438.84	439.34	1	60	5	5	5
21	River 1	Reach 1-Lower	6.22	438.41	438.91	1	60	5	5	5
22	River 1	Reach 1-Lower	6.08	438.47	438.97	1	60	5	5	5
23	River 1	Reach 1-Lower	5.99	439.84	440.34	1	60	5	5	5

Simulation

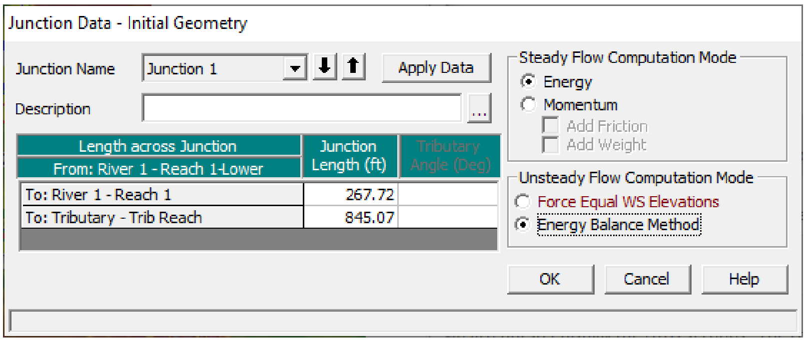
After pressing the Computation button, a status dialog (shown below) will provide progress and messaging during the simulation.



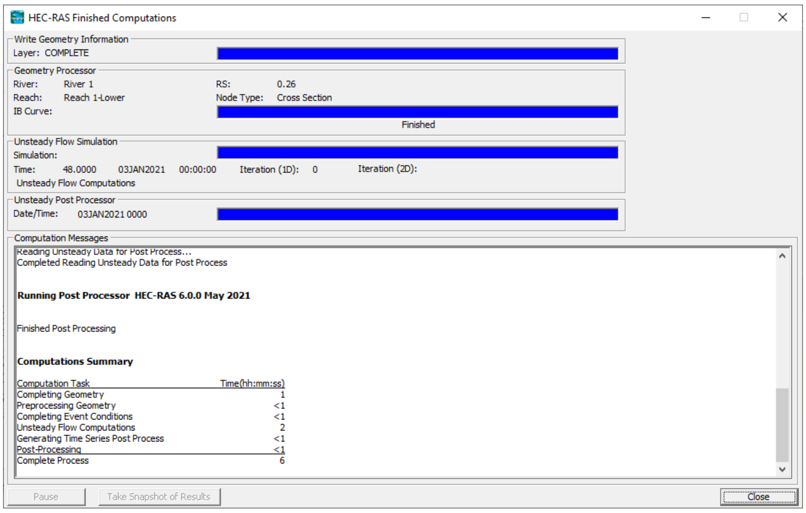
If the simulation goes unstable right away, it may likely be due to initial or boundary conditions. A typical example is shown here where the instability is occurring on the tributary reach. You can see this by looking at the computation output and observing that the simulation is going to maximum iterations at the most-downstream cross section for the tributary. This is due to the combination of relatively steep tributary reach coming into the main river which has computed a very low initial water surface (see image below).



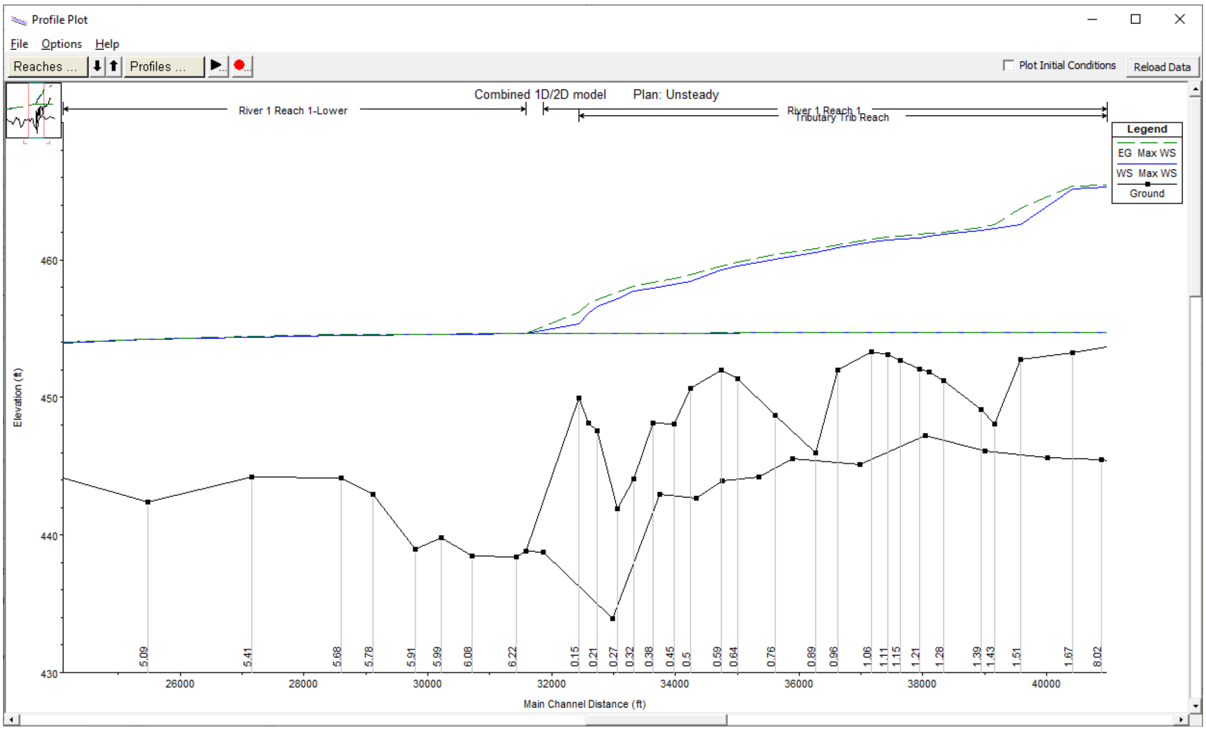
The unsteady equations are unable to solve at the junction due to poor geometric data. Bad geometric data, where the geometry is changing more rapidly than it should or is not consistent with the surrounding geometry, is often the source of model stability. For the case of junctions, HEC-RAS has an option to attempt to solve data inconsistencies. The default solution method at a junction is to force the same water surface at all cross sections. Using the method to compute the water surface elevations using an energy balance can help stabilize the model.



Simulation after turning on the "energy balance method" allowed the run to completion.



Simulation results with the water surface elevation computed across the junction look acceptable.

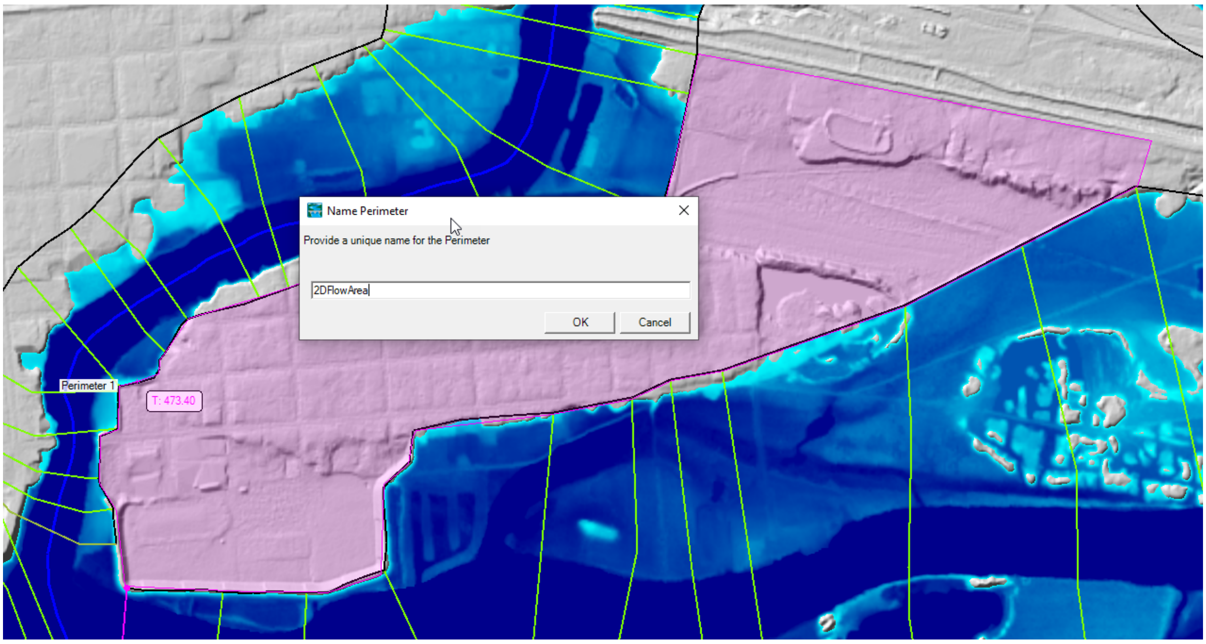


1.1.10 Create 2D Geometry

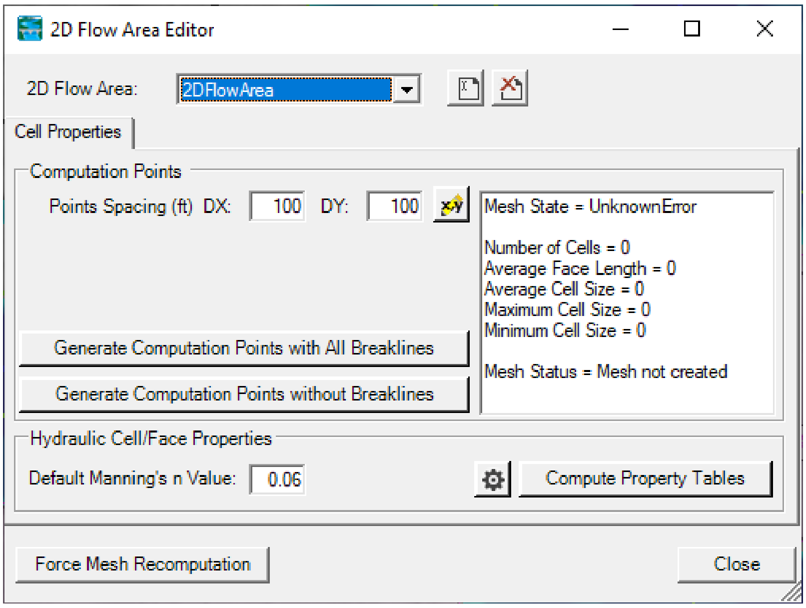
Once the 1D unsteady flow portion of the model is running stable, only should you consider adding 2D Flow Areas.

2D Flow Areas

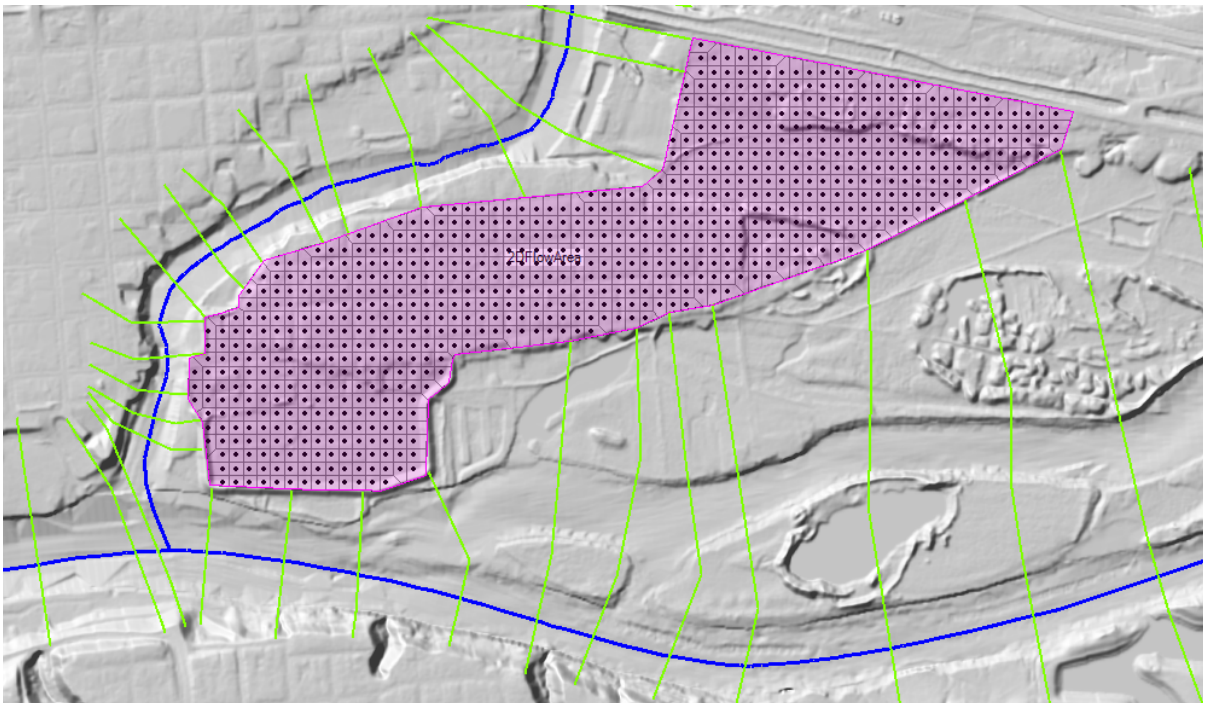
For this example, we are adding the a 2D mesh for the area between the river and tributary. The 2D flow area boundary should follow the cross section edge lines. When finished creating the 2D Flow Area, you will be asked to provide a name.



The 2D Flow Area editor will then be invoked. Enter a default cell size and default Manning's n value. For this example, the spatial n values will be used. **Generate Computation** points for the initial mesh. Consider starting with a large cell size and refine down to a smaller cell size as you become more comfortable with the model.



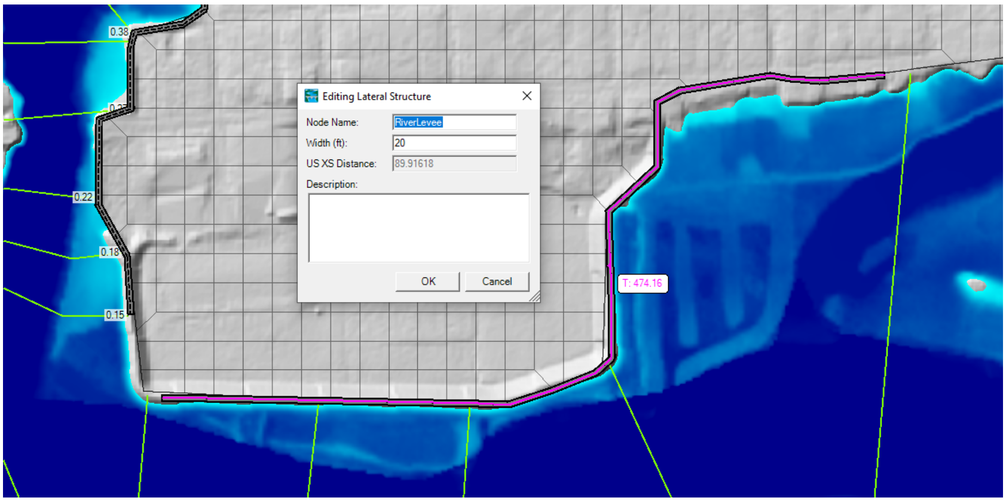
The initial 2D flow area with computation points is shown in the figure below.

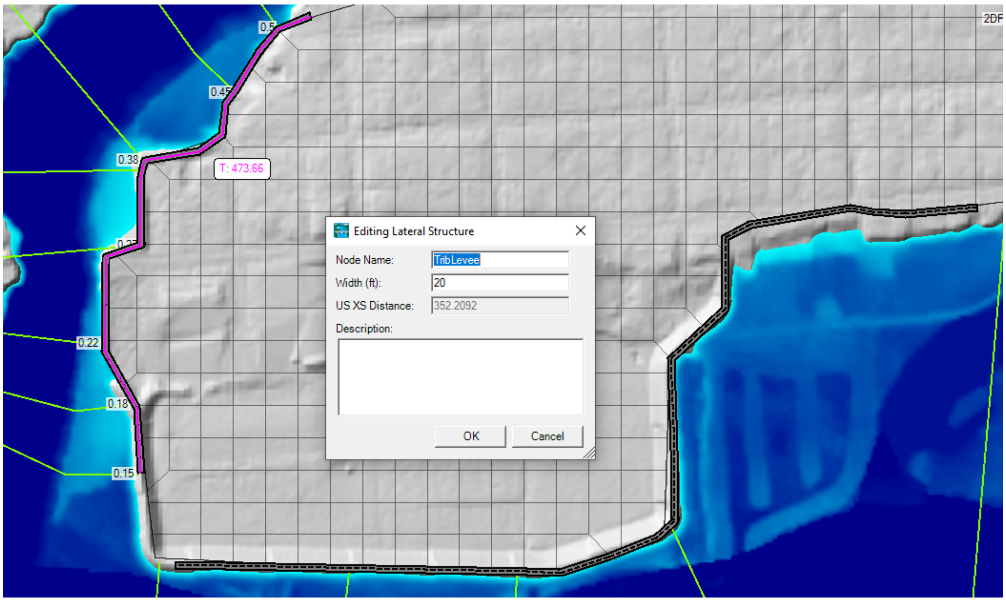


Lateral Structures

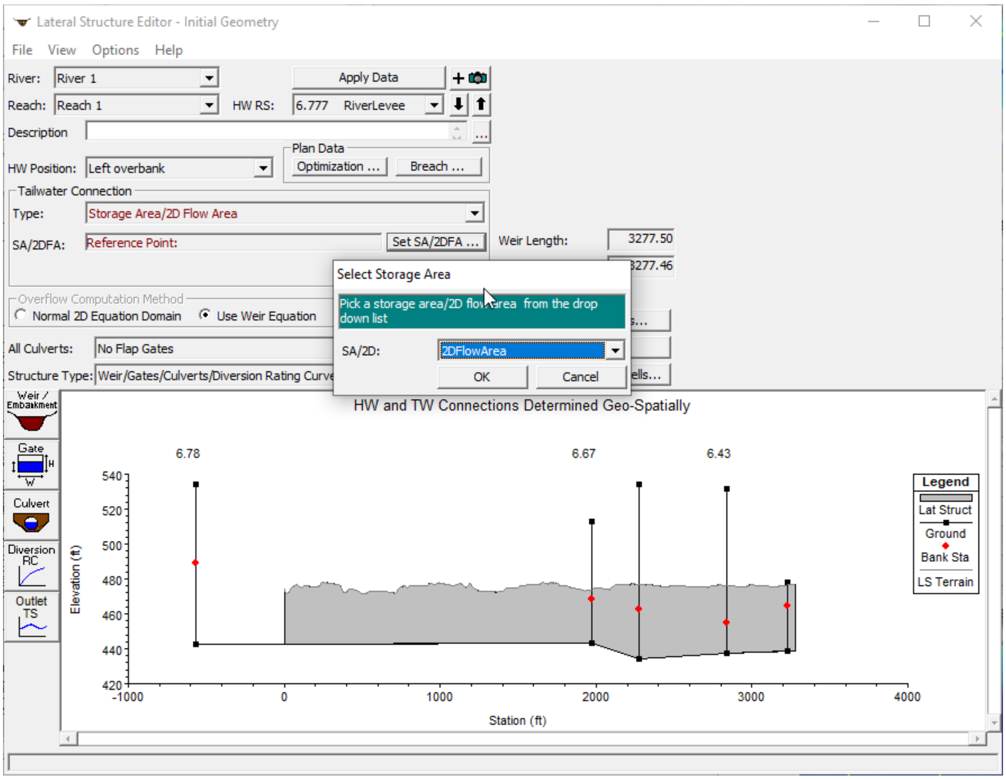
The 2D flow area is connected with the 1D portion of the model with lateral structures. Lateral structures should be placed along high ground at the ends of cross sections. Typically, we think of high ground as levees and roadways, but sometimes high ground can be more subtle. Another consideration with lateral structures is that if you need to know how much flow is passing over a certain location, you will need to limit the length of the lateral structure. If you are not concerned with flow accounting, you can create very long lateral structures.

Looking at the 1D unsteady flow model results will determine where to flow will move from the 1D to 2D model. Create the lateral structures in the downstream direction and RAS Mapper will pick up elevations for the weir crest from the Terrain model.

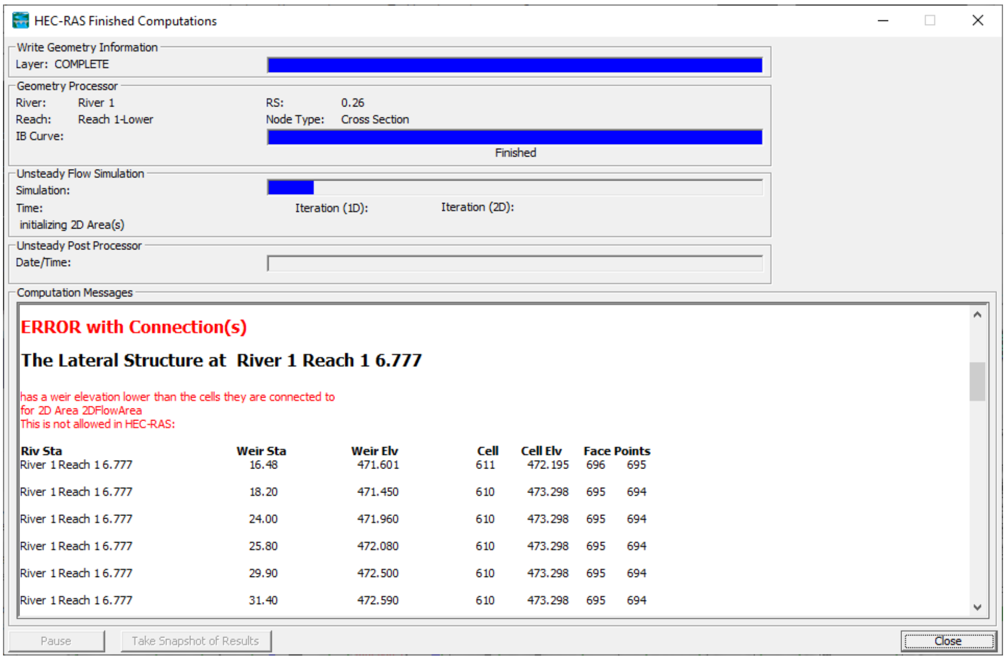




Lateral structure parameters will need to be entered from the Lateral Structure Editor. You will need to set the Tailwater Connection to the 2D Flow Area. You will also need to set the weir coefficient and verify the distance to the upstream cross section.

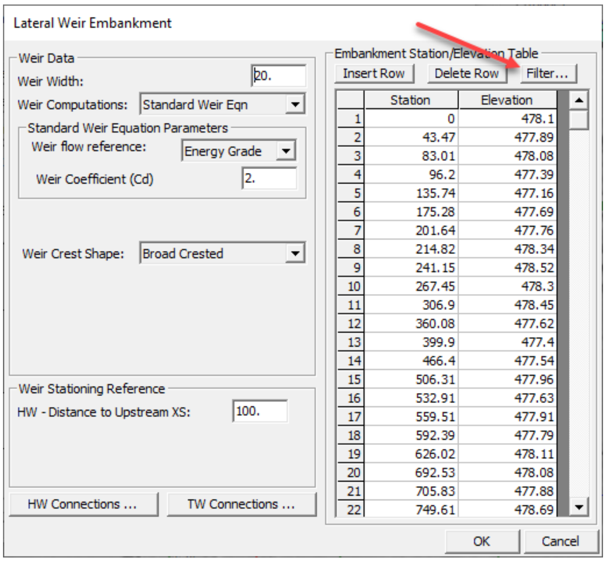


When you run the simulation, RAS will verify that the elevations of the lateral structure are higher than the minimum elevations of the connected 2D cells. You will receive a warning message if the lateral structure elevation is "below ground", as show below.

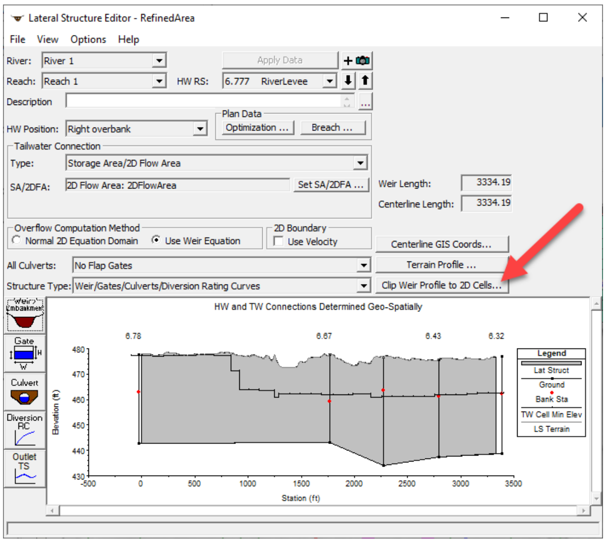


You can manually adjust the elevations or use the Clip Weir Profile to 2D Cells button on the Lateral Structure Editor. Before doing that, however, it is a good time to filter that lateral structure elevation points. The elevation profile from the terrain many hundreds (1000 is the default in RAS Mapper), but it is likely that the profile could be adequately defined by much fewer. Fewer points will mean feature computations with the unsteady flow engine because HEC-RAS computes the water surface between each set of points along the weir profile.

The weir profile is filtered using the **Filter** button on the Lateral Weir Embankment editor, as shown below.



After filtering the data, use the **Clip Weir Profile to 2D Cells** button on the Lateral Structure Editor (shown below) to ensure the weir profile is higher than the 2D cells.

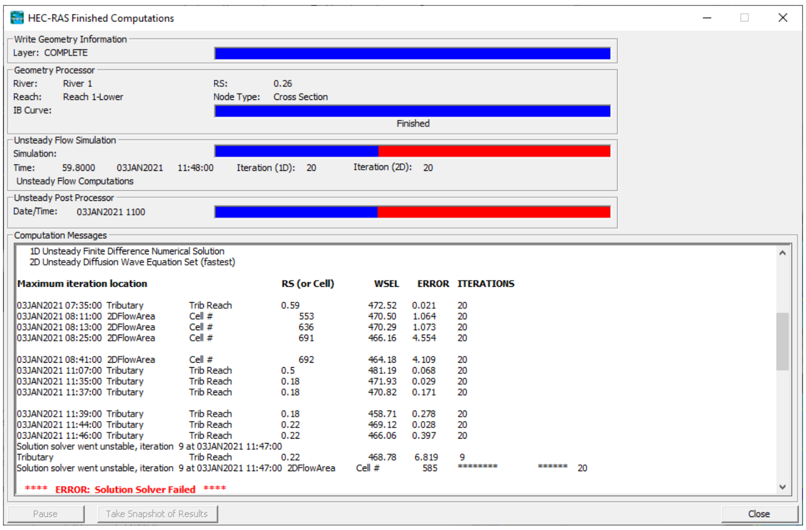


1.1.11 Combined Model Simulation

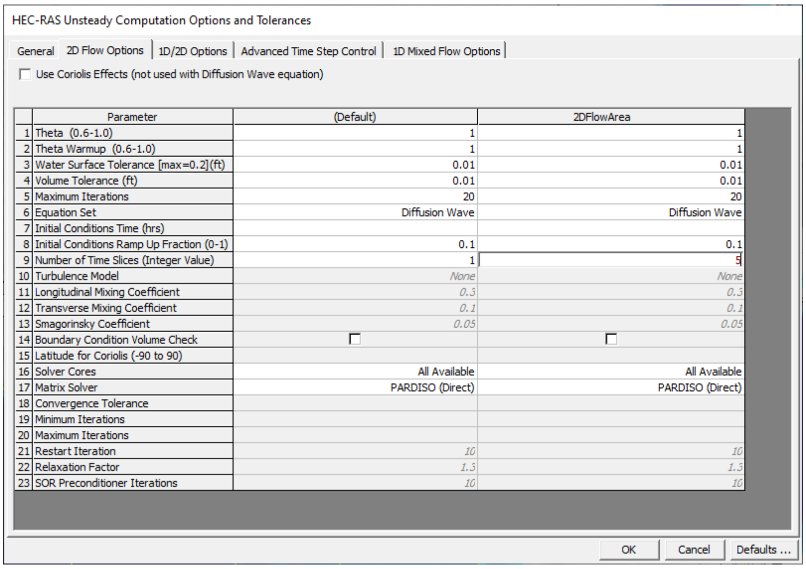
After setting up the 2D portion of the model. Then carefully connect the 2D flow area to the 1D geometry. Careful inspection of overflow areas will guide you in using lateral structures to move water between the 1D and 2D domain. However, you do not need to connect the model in every location. Consider adding into the model only one lateral connection at a time.

Time Step Considerations

It is not atypical for your first 1D/2D combined model simulation to go unstable. You can see the initial run begins to iterates to the maximum iterations in the 2D cells. For this example, the 2D cells are much smaller than the distance between cross sections; therefore, we should be using a smaller timestep in for the 2D domain.

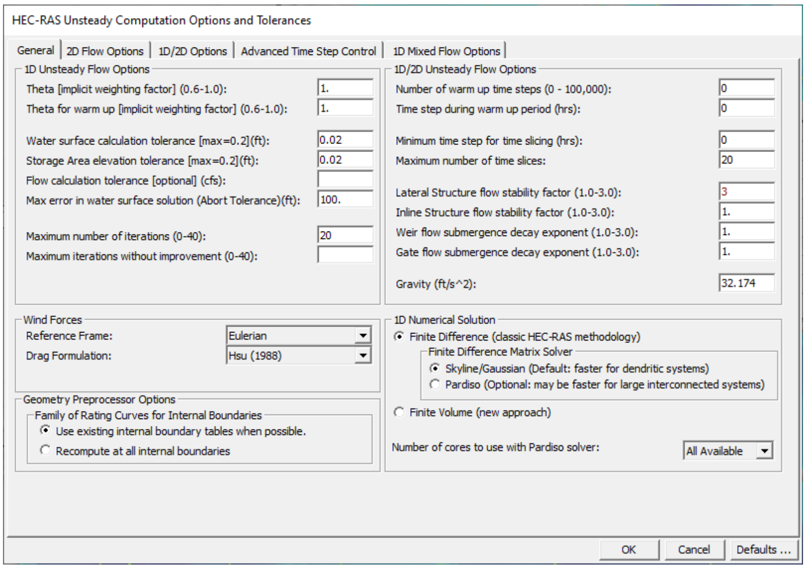


A smaller time step is achieved using Time Slices option, located in the 2D Flow Computation Option and Tolerances.

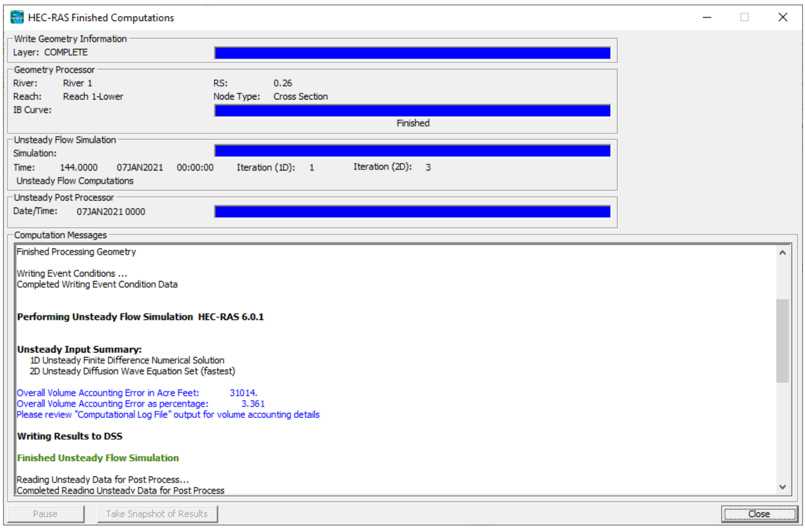


Lateral Structure Parameters

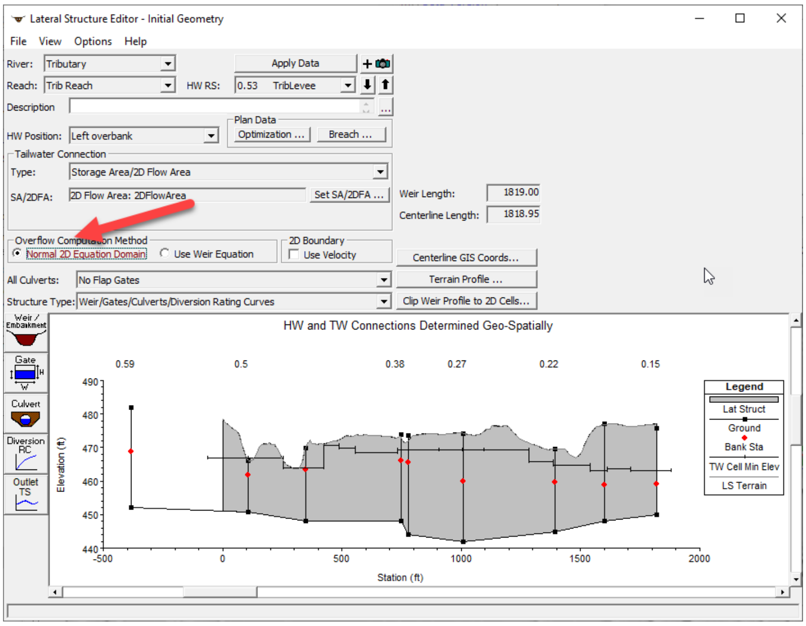
If you continue to have issues with stability (reaching max iterations at cross sections), consider the Weir Flow stability factor.



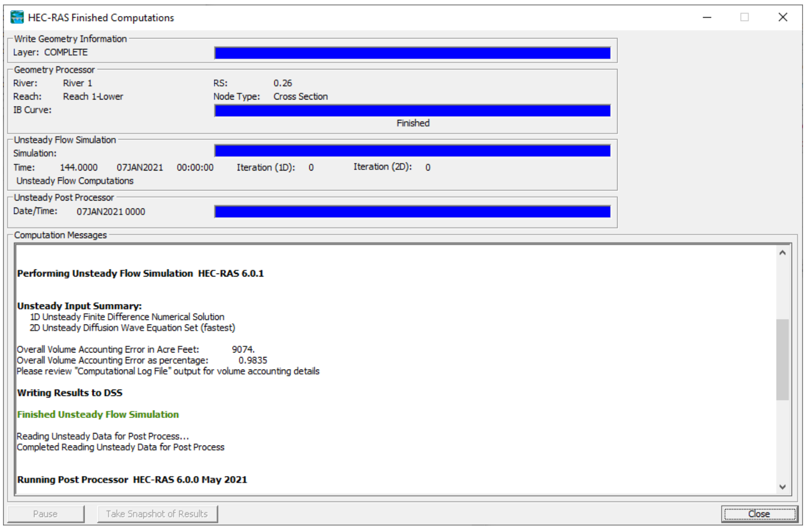
In our case, after the model became fairly stable, we still had a Volume Accounting that was in error.



Further, consider how the terrain and flow is interacting. Is the high ground along the lateral structure actually working like a levee? Lowering the weir coefficient can be used to dampen the effect of flow leaving the river system. After looking at the flow depths and terrain in detail around lateral structure on the tributary reach, we see flow really isn't controlled like a weir, it is more like overland flow. For the lateral structure in question, change the Overflow Computation Method to the Normal 2D Equation.



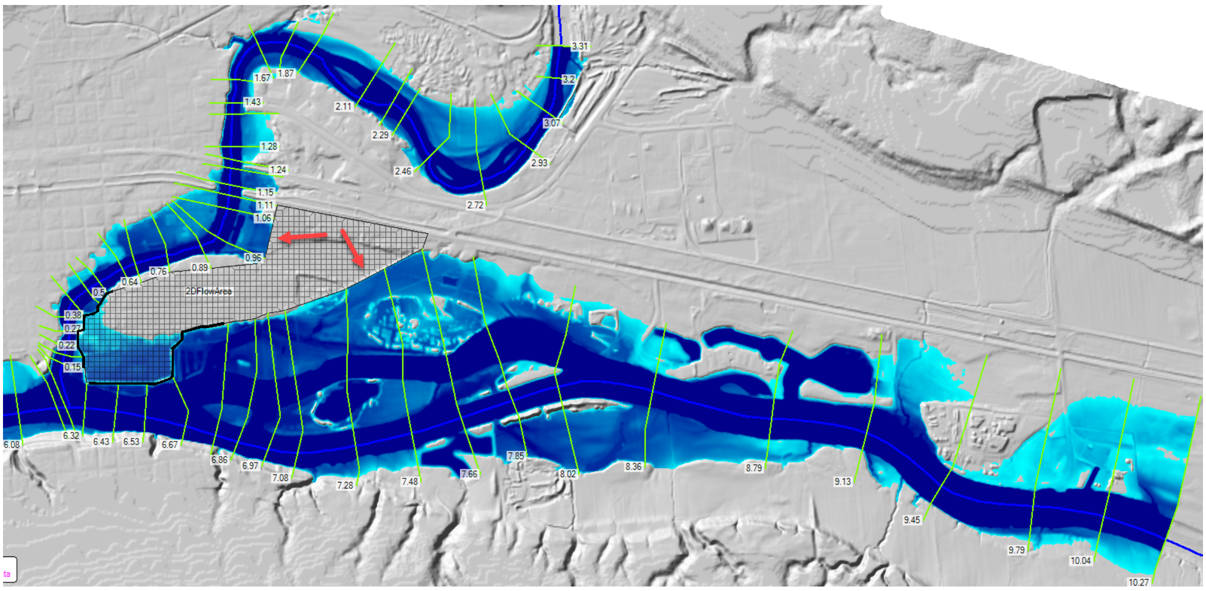
Changing the computational method results in fewer iterations, a model that runs faster, and a smaller volume accounting error.



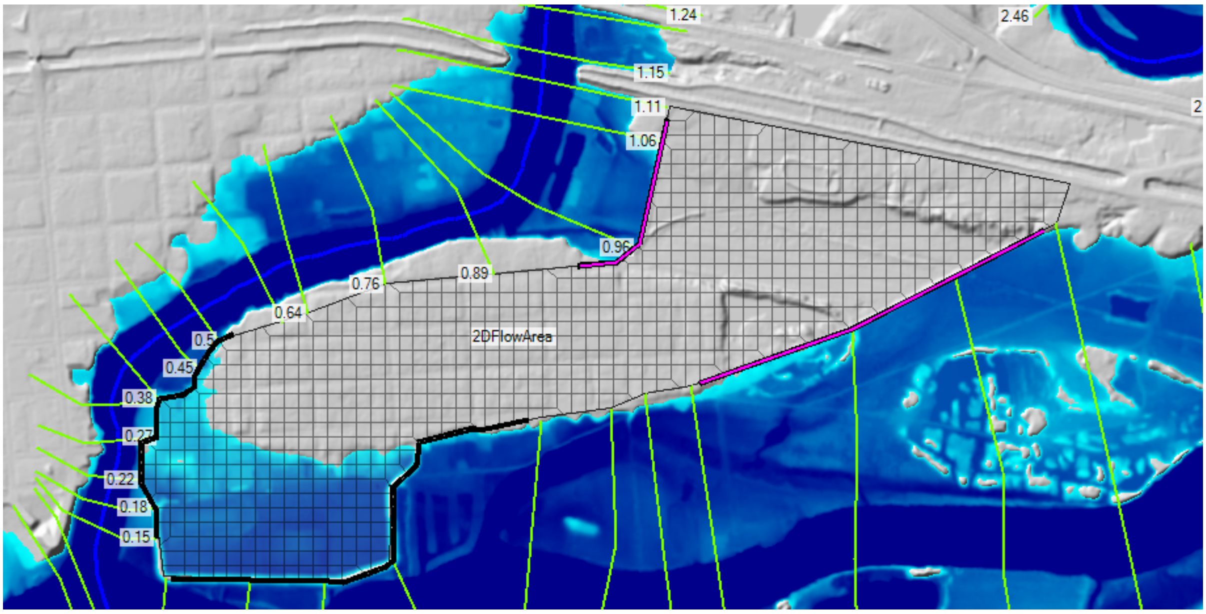
To further improve the volume accounting, we could decrease the timestep which would result in less flow being moved from the 1D domain to the 2D mesh during a single timestep. However, smaller timesteps will result in longer model run times.

1.1.12 Combined Model Evaluation and Refinement

After achieving model stability with the initial combined model, we can visualize results and get a handle on the system performance and resulting inundation. In the below image, we can see area where we should have included lateral structures to connect the floodplain to the river at the upstream end of the 2D flow area.



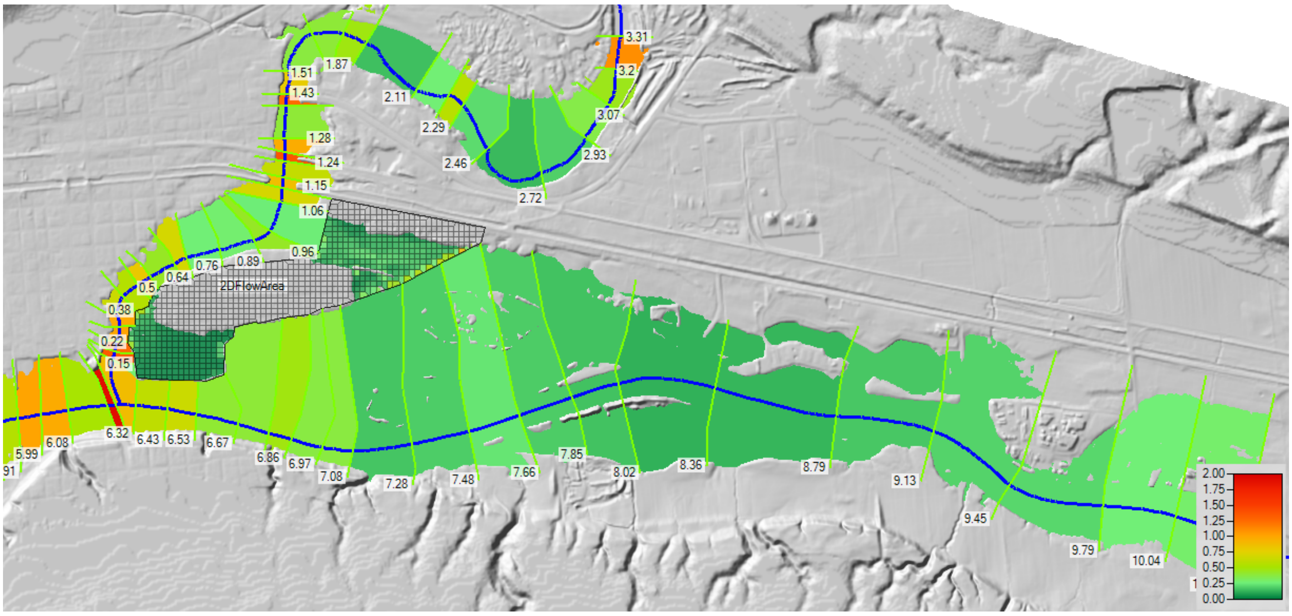
Continue to add lateral structures where appropriate to model the movement of water in the river system. In the image below, you can see we have added two more lateral structures to allow water to move into the 2D flow area. Be sure to complete all of data for each lateral structure. This includes selecting the headwater location, tailwater location, filtering the elevation profile, selecting a weir equation coefficient, selecting the modeling method, and verifying the headwater distance to the upstream cross section. You will also need to verify that the weir profile is higher than minimum elevation of the neighboring cells.



Evaluate the resultant floodplain for areas of improvement. This can be done using the various output plots in HEC-RAS and map output in RAS Mapper. Take a look at the inundation extent. Animate through the simulation profiles and look for discrepancies in how the water is moving through the system. Evaluating the courant number and velocities will provide insight to the current solution.

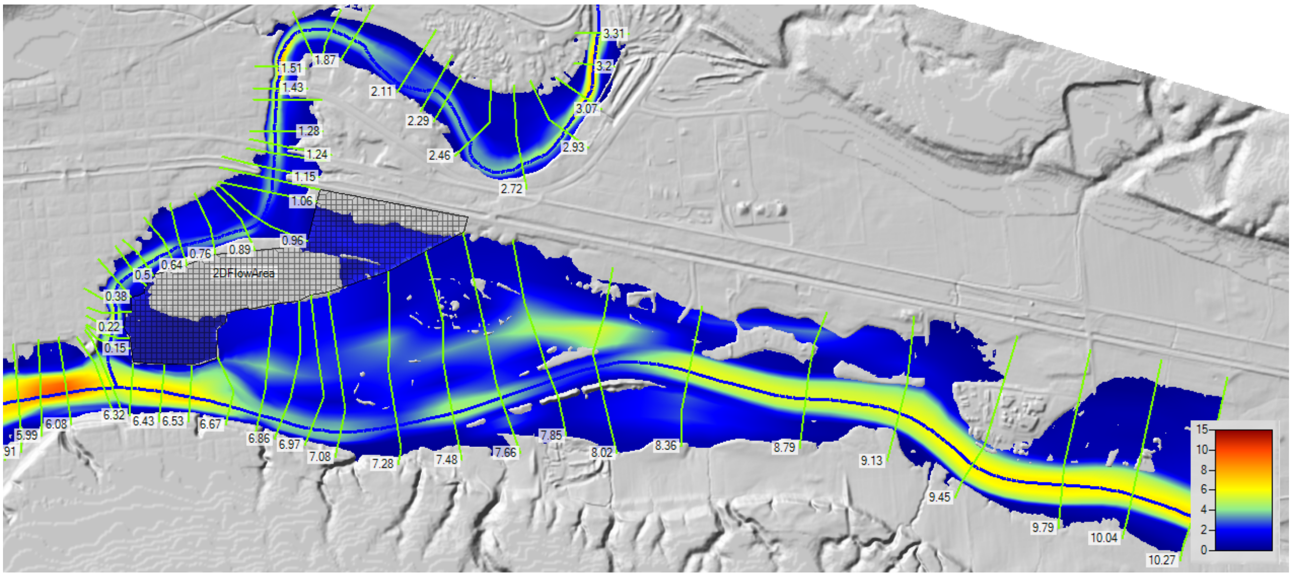
Courant Number

Plotting the courant number will help provide further understanding on the interaction of the selected timestep with the cross section spacing and the time slicing for the 2D flow area. Striving for a courant number near 1.0 is a noble effort, but can rarely be achieved for the entire model. If you identify particularly sensitive areas to the selected timestep, reduce and rerun.

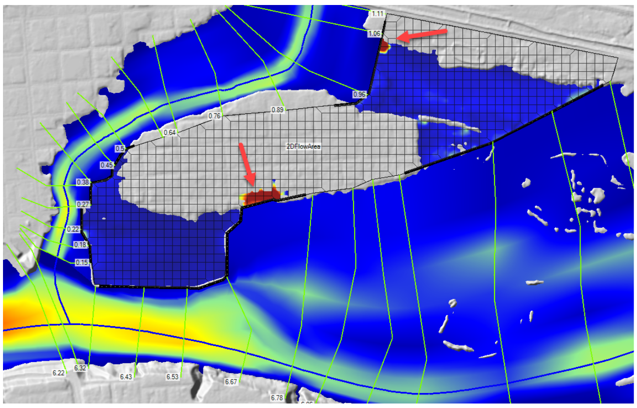


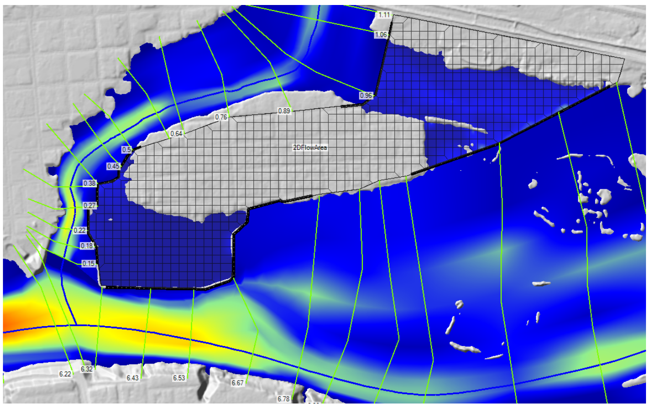
Velocities

The velocity map is ideal map for gaining insight to the river and floodplain. Create a velocity map and animate through the profiles. Pay special attention to high velocity locations or where velocities change rapidly.



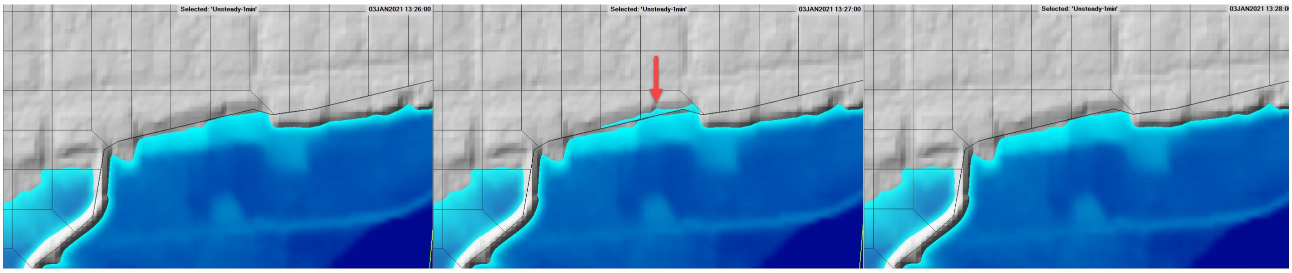
Not that if you plot the Max velocity, you will not get the a map that looks like any of the profiles that you animated through. This is because the maximum velocity is output separately from the data export at the "Mapping Output Interval" specified on the Unsteady Flow Data Editor. Investigate the max velocities to identify if it is the result of model instability or if the output interval did not catch the higher velocity. For the example shown below, the max velocity is reporting very high velocities near structures that are overtopped (that are not shown in the animation of the output). This is most likely due due to the diffusion wave solver struggling to solve for a stable solution just as the structure is overtopped.



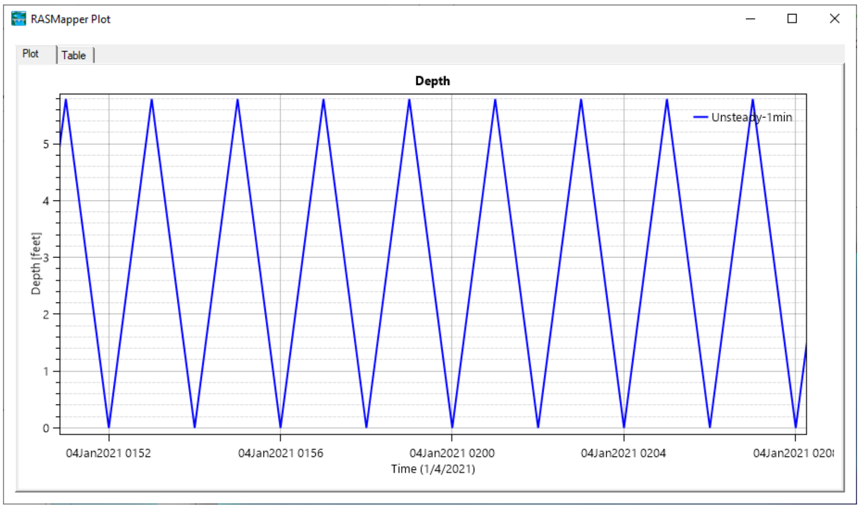


Make a run with the model, setting the mapping output equal to the time step. This will plot all of the computed results and allow you to see values that may have been skipped over. Zooming into the high velocity region, you can see the model is going unstable. Below is a depth plot for one of the cells showing how the depth is flipping between wet and dry. This indicates that the computed water surface elevation is very sensitive to the 1D/2D connection. This may be due to the weir coefficient or the location of the weir resulting in a poor elevation profile. Or maybe the timestep is just too big. Or a combination of factors.

In the below plot of the water surface time series, you can see the water surface oscillating - the 2D cell is wetting and drying every other timestep.

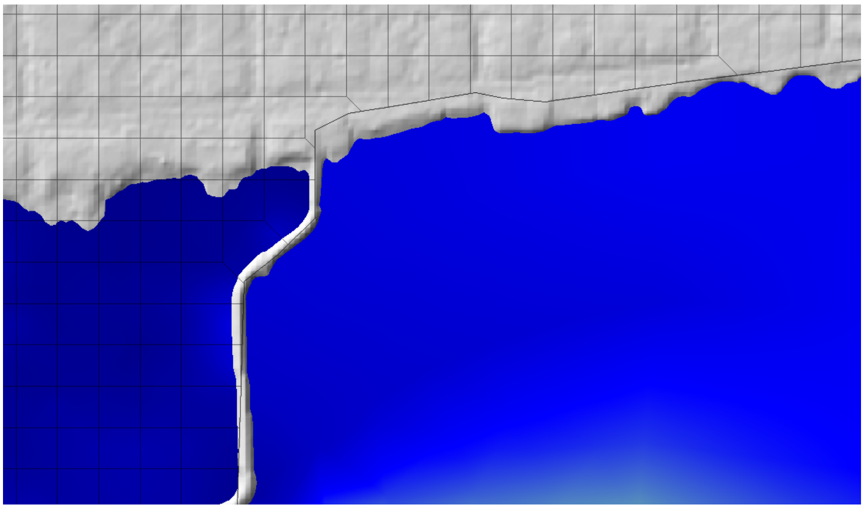


Plotting the time series of depths for the cell, you can see the large changes each timestep.



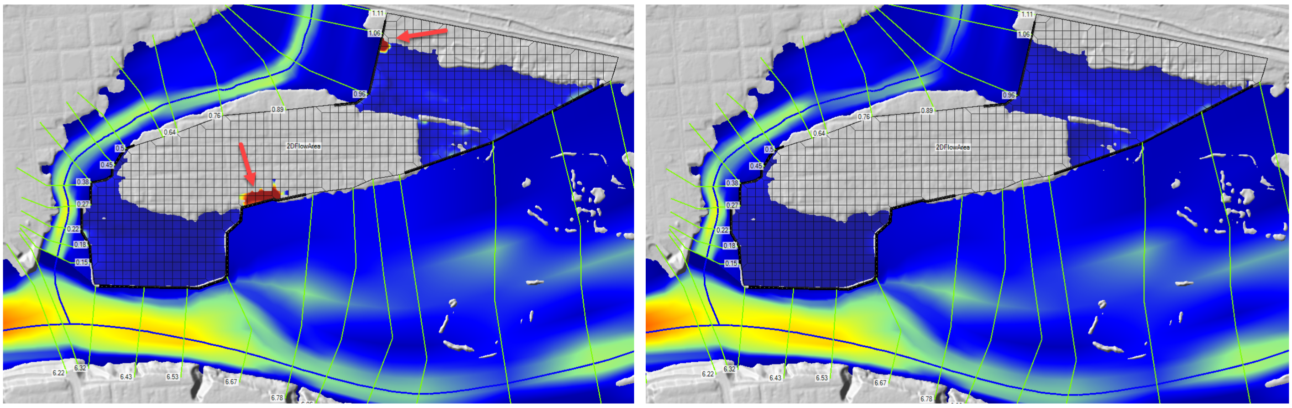
This demonstrates the importance of placing lateral structures on high ground and have the correct weir elevation profile. If we had survey information we could replace the weir profile. In order to fix this issue, we will need to move the lateral structure to high ground and adjust the edge of the 2D flow area.

Moving the lateral structure to high ground improved the solution and kept the water surface for moving out into the 2D area prematurely, iterating, and poor mapping. No longer due we have max velocity issue in this area, as shown in the max velocity mapping below.



Because lateral structures control flow into 2D flow areas and cell faces control how water moves within the 2D flow area, you must take precaution to ensure these controls are on high ground. Poor placement of cell faces and structures may not result in final water surface elevations that differ from the "perfect" solution, however, than can cause model instabilities that result in longer run times, volume accounting errors, local mapping issues, and other local anomalies. Take care to address each issue to improve model fidelity.

You may find yourself in the situation (shown below) where the Max velocity doesn't match any of the values contained in the output, despite setting the Mapping Output Interval equal to the Computation Time Step. This can happen if you have time slicing turned on. The max value could occur in the 2D cell during a time slice and not be reported at the output interval.



1.1.13 Model Sensitivity and Comparison

After reviewing initial model results and refining the geometry for river hydraulics model, you should spend more time trying to identify parameters in the model that are particularly sensitive. Hopefully, you have already been doing this initial investigation as you found model instabilities or data inconsistencies with previous simulations. There are many model data and parameters that may or may not impact the simulation results that are worth discussion. Saving an existing Plan to a new name and rerunning the model allows for an easy way to look at the affect of simulation parameters.

1.1.14 Parameters

One of the most important parameters that should be evaluated is the effect of the simulation time step on the model. Additional parameters that can and should be considered can be found on the Computation Options and Tolerances window. Some of the more important parameters are discussed below.

Time Step

You should always evaluate the affect of the computation time step on the hydraulic results. If reducing the time step resulted in significant changes to the water surface elevations or velocities, then the smaller time step should be used for future simulations. Various time steps should be evaluated and their effects on model stability, accuracy, and computational run time.

If you would like to have HEC-RAS figure out a time step to use base on the courant criteria, you can use the Advanced Time Step Control available on the Computation Options and Tolerances window.

Time Slices

The computation time step effects the base unsteady flow computation engine. For a combined 1D/2D model, if you require a smaller time step for the 2D Flow Areas, you must use the Time Slicing option. The optimal value can be identified through trial and error balancing model stability, accuracy, and computational run time.

Equations Set

2D model runs have the option to run either the using the Full Shallow Water Equations (SWE) or with the Diffusion Wave (DW) approximation of the momentum equation. The diffusion wave approximation is appropriate in where the dominant forces on flow are mainly gravitation and friction forces. Where local convective acceleration is important, the full shallow water equations will be more appropriate.

> ⓘ Once a model is up and running, you need to compare runs with the SWE to the DW. If you model shows significant difference, you should be using the SWE for all future model runs.

- **Diffusion Wave** - DW is more computationally stable and good for getting a 2D model up and running to get a rough estimate for inundation extents and depths. For complicated problems, DW is simply a first step before applying SWE. Some key points are listed below.
    - Very stable
    - Not good for rapid rising/falling hydrographs (temporal acceleration)
    - Not good for sharp contractions and expansions (accelerations)
    - Not good for sharp bends for modeling super-elevation
    - Not good for tidal boundaries (no wave propagation)
    - Can't model hydraulic jumps
- **Full Shallow Water Equation** - SWE is important for modeling rapid changes to flow due to acceleration whether that be do changes in hydrograph or geometry that results in rapidly varied flow. Some key points are listed below.
    - Less stable than DW requiring a smaller time step and longer computation times
    - Needed for rapid rising/falling hydrographs (temporal acceleration) like dam and levee breaches
    - Needed for sharp contractions and expansions (accelerations) like at hydraulic structures like a bridge opening
    - Needed for sharp bends for modeling super-elevation
    - Needed for tidal boundaries (no wave propagation)
    - Needed for modeling hydraulic jumps

Theta Implicit Weighting Factor

Theta is the implicit weighting factor used by HEC-RAS for solving the unsteady flow equations. The default value of 1.0 is the most stable and uses information on from the current time step weight the pressure gradient term in the momentum equation to solve the unsteady flow solution. Using a value smaller than 1.0 will result in the using the information from the previous time step with information from the current time step during the solution. HEC-RAS allows for a value of 0.6 to be used for the most accurate solution (at the cost of stability).

Stability Factors

Stability factors have been added in HEC-RAS to to keep flow from rapidly changing during the simulation. Experience has shown that dampening flow over lateral structures can greatly improve model stability. One the model is stable, these factors can be decrease back to a more accurate solution.

- **Lateral Structure Flow Stability Factor** - This factor ends up reducing the amount of flow change between time step at a structure. A value of 1.0 is the default and more accurate. A value of 3.0 will end up reducing flow and increase model stability.
- **Weir Flow Submergence Decay Exponent** - This factor is used to reduce the amount of flow over a weir that is submerged. Submergence occurs when the tailwater is high enough to slow down flow. Rather than waiting until the weir is highly submerged, weir flow can be reduced sooner resulting in smaller changes in flow and more model stability. The default value of 1.0 uses the default flow reduction. A value of 3.0 reduces under less submergence and is more stable.

Turbulence Modeling

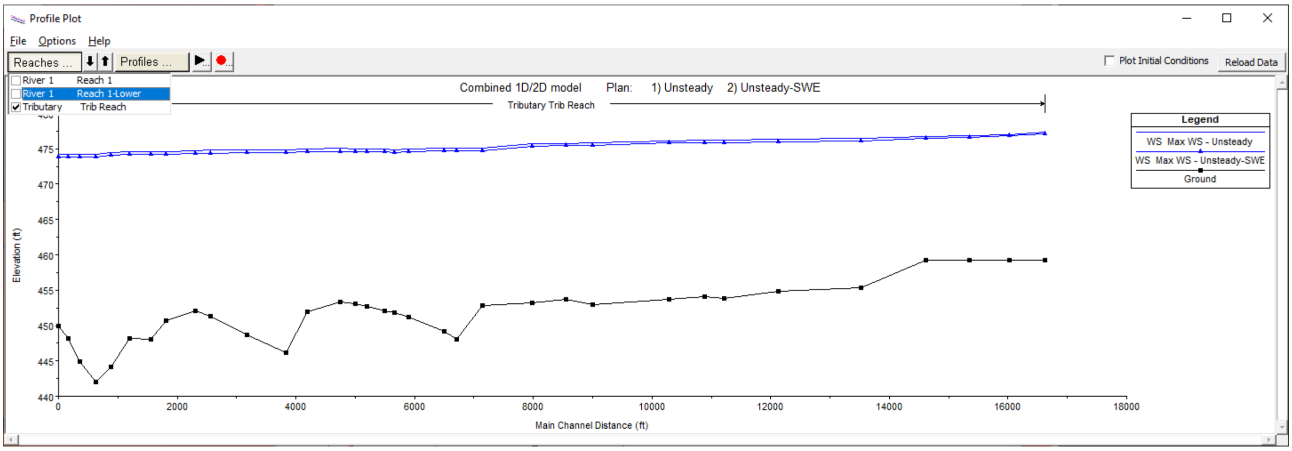
Turbulence modeling can be important in 2D unsteady flow modeling where slow water attempts to slow down faster water and fast water attempts to speed up slower water - the action that produces eddies. Turbulence modeling is only available when using the SWE because the DW approximation ignores all terms except the pressure gradient and bottom friction terms. Turbulence modeling requires the selection of a longitudinal mixing coefficient, transverse mixing coefficient, and Smagorinsky coefficient.

Model Comparison

To understand the model's sensitivity to each of the model parameters previously discussed, you will need to establish a base plan and then compare those results with various alternatives. There are many tools available for results comparison.

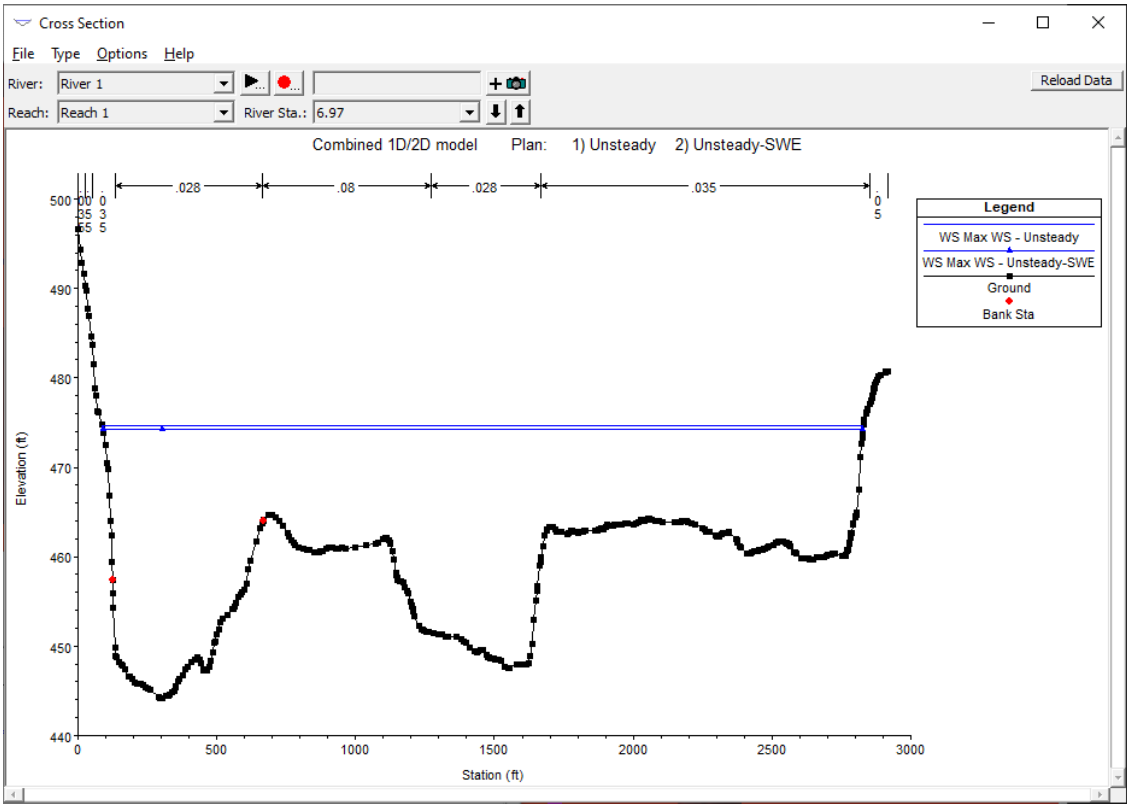
Profile Plot

An example profile plot from HEC-RAS is shown below for a single river reach with the max profile for two plans.



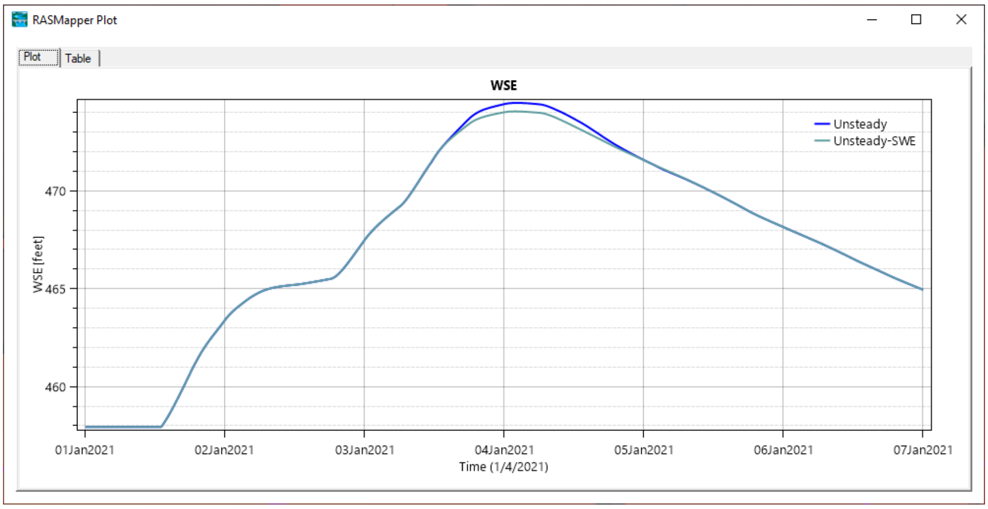
XS Plot

An example cross section plot from HEC-RAS is shown below with the max profile for two plans.



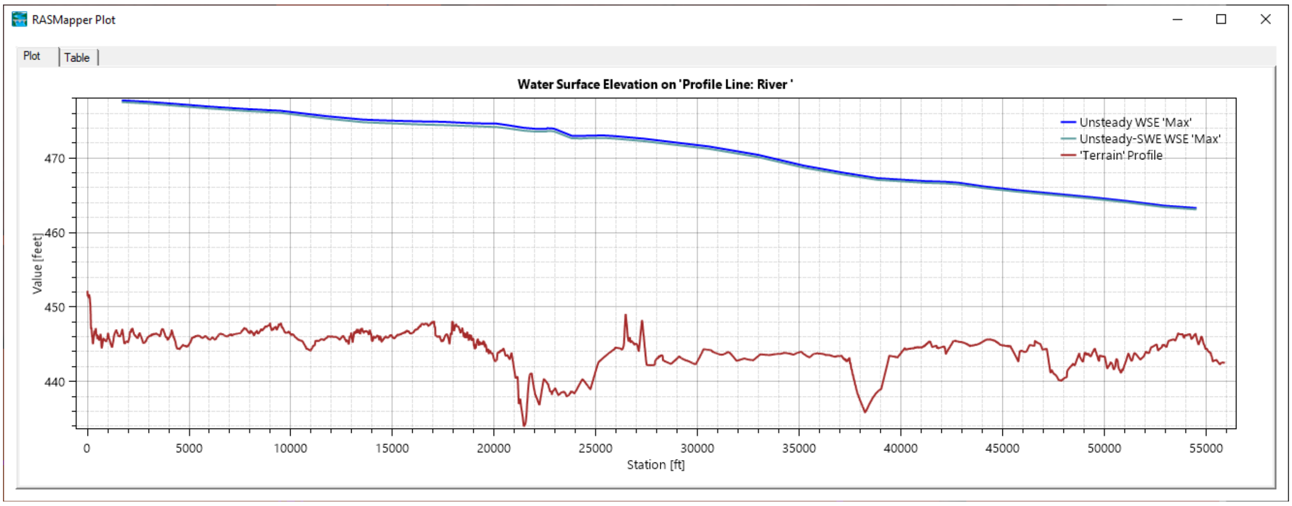
Hydrograph Plot

A hydrograph plot from RAS Mapper is shown with a water surface elevation time series plot for two simulations. Output is available from the map for whichever mapping results are visible.



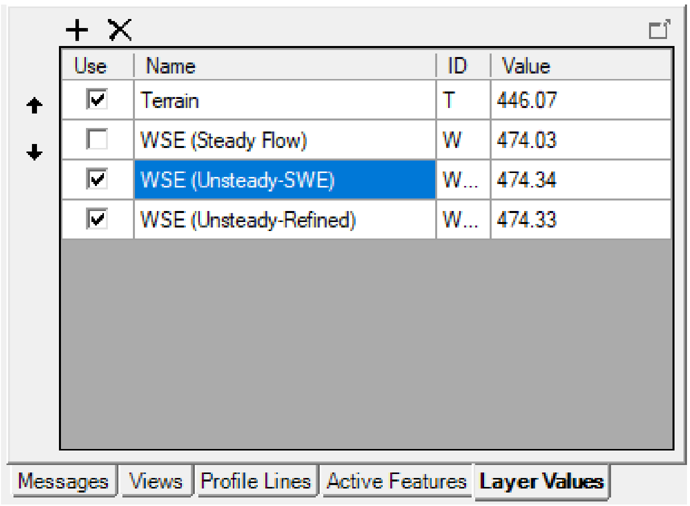
Profile Lines

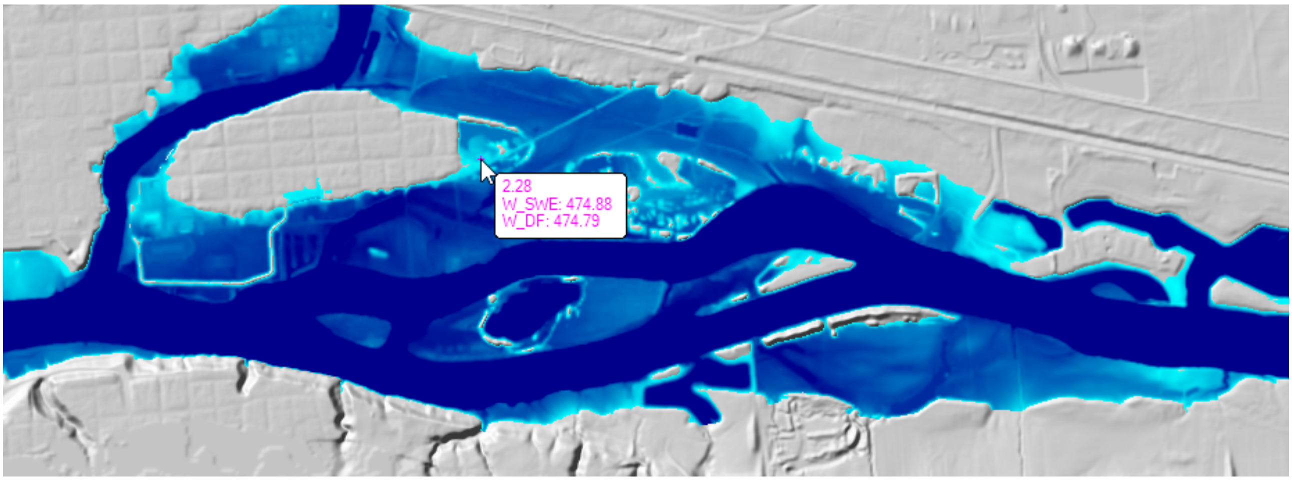
An example profile plot from HEC-RAS is shown below for a river reach with the max profile for two plans. Output is available from the map for whichever mapping results are visible.



Watch List

The Layer Values Watch List is created by right-clicking on any layer and choosing the Add Watch to Layer Values menu item. As you move the mouse across the map display, the cursor will report the map value next it's label (ID) as well as reporting the value in the Layer Values dialog. This is a valuable tool for a comparison of computed map results. Example output is shown below.



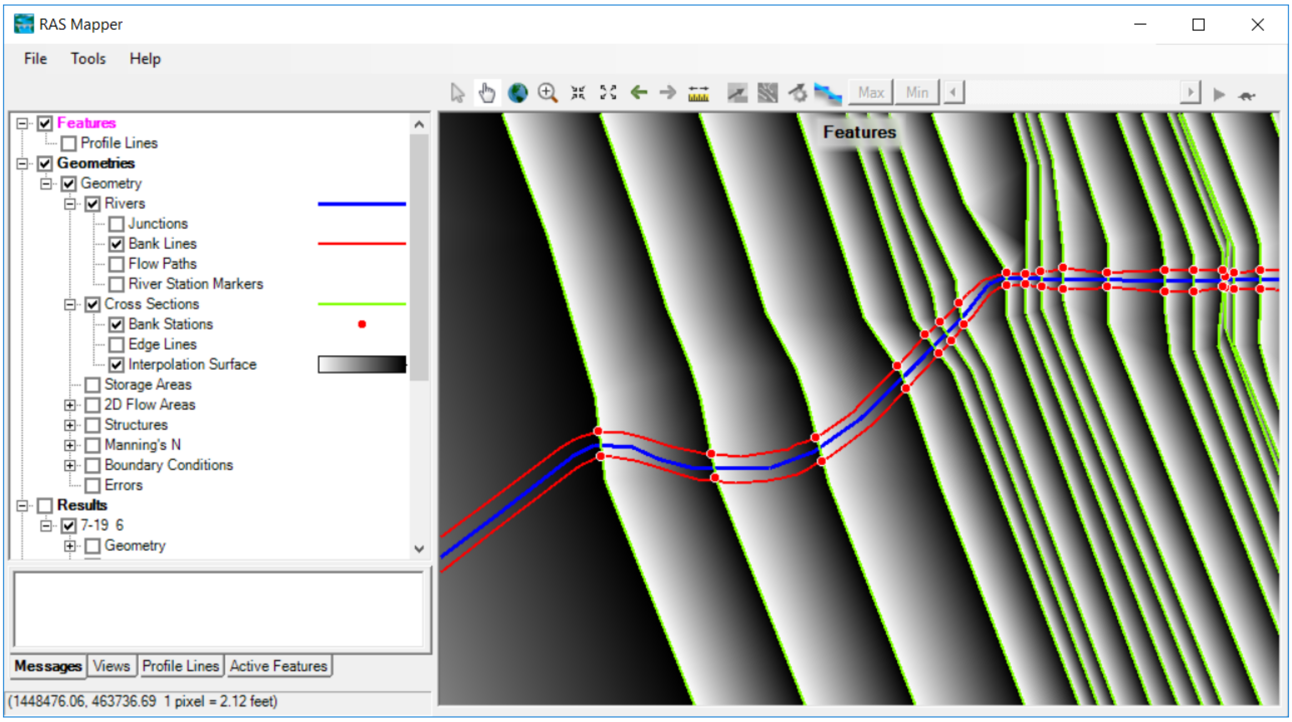


1.2 Export Channel Data for Terrain

RAS Mapper provides the capability to export channel (bathymetric) from an existing HEC-RAS Geometry to merge with other ground surface information in the Terrain Layer. This improves the visualization of computed water surface inundation extents and depths. Further, it allows the user to "cut" new cross sections at additional locations.

1.2.1 How does it work?

Prior to creating the channel surface, you should understand the process RAS uses to create the data. The channel surface gets the elevation data from the Cross Sections and will then be interpolated from cross section to cross section using the shape of the River Centerline, Bank Lines and Edge Lines. The Bank Lines will control the limit of the "channel" export, while the Edge Lines control the bounds of the interpolation. The results of the interpolation using the Cross Sections, River, Bank Lines, and Edge Lines can be visualized and inspected by the Interpolation Surface.



> The channel Bank Lines control how interpolation is performed. Make sure they do not cross each other or the River Centerline.

1.2.2 Step-by-step guide

To export the channel bathymetric data within RAS Mapper, perform the following steps listed below.

1. Begin Editing the Geometry of interest.
2. Right-click on the Bank Lines Layer and select **Compute Bank Lines from XS Bank Stations**.
    a. Inspect the Bank Lines
    b. Adjust the Bank Lines as needed to properly capture the channel
3. Right-click on the Interpolation Surface Layer and select **Compute Interpolation Surface**.
4. Right-click on the Geometry and select **Export Layer | Create Terrain GeoTiff from XS's (Channel Only)**.
5. Provide a layer name for the new GeoTiff.
6. Provide a rasterization cell size for the new GeoTiff.

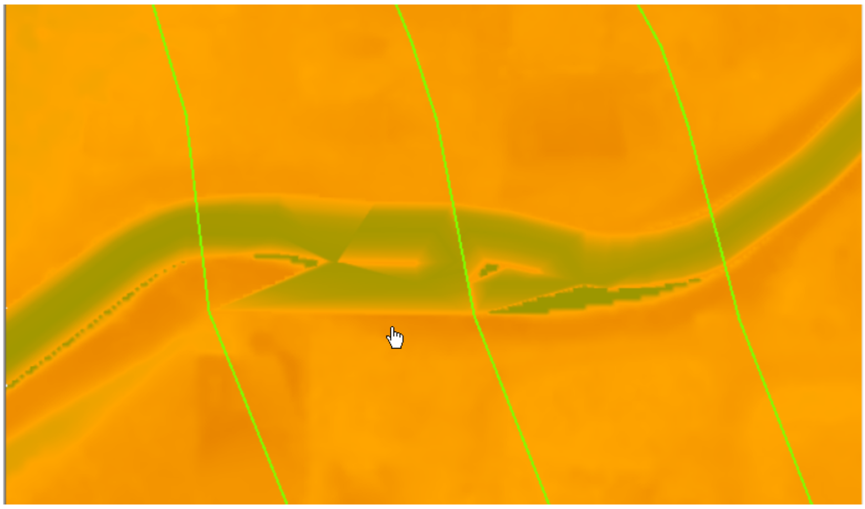
To merge the channel GeoTiff with an existing ground surface model, perform the following steps listed below.

1. Right-click on the Terrains group and select **Create a New RAS Terrain**.
2. **Add** the terrain models of interest.
3. **Prioritize** the terrain models, making sure the "channel" surface is on top.
4. Enter a unique name for the Terrain Layer.
5. Press the Create button.

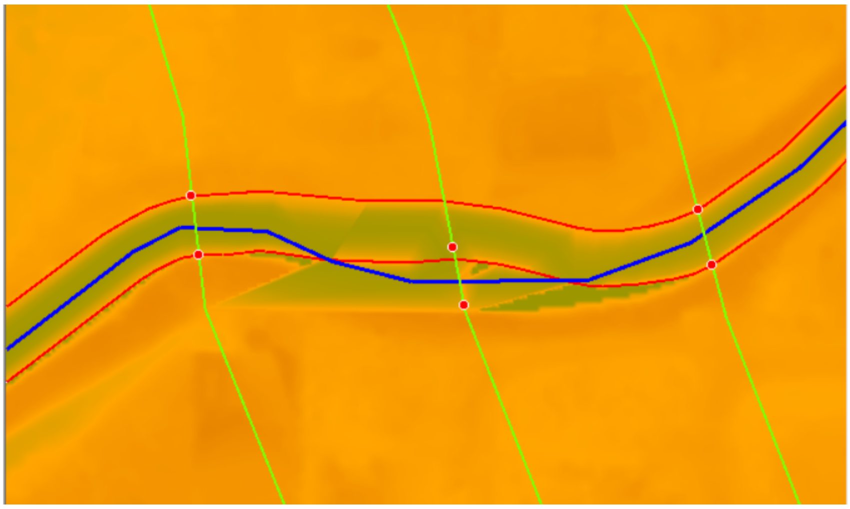
1.2.3 Common Problems

The results of the channel surface are dependent on the cross sections and bank line information. Because HEC-RAS allows users to store the geospatial "cut line" information separate from the station-elevation data, users might not get the surface they are expecting. The best way to identify problems that will occur with final channel surface is to inspect the Bank Lines Layers.

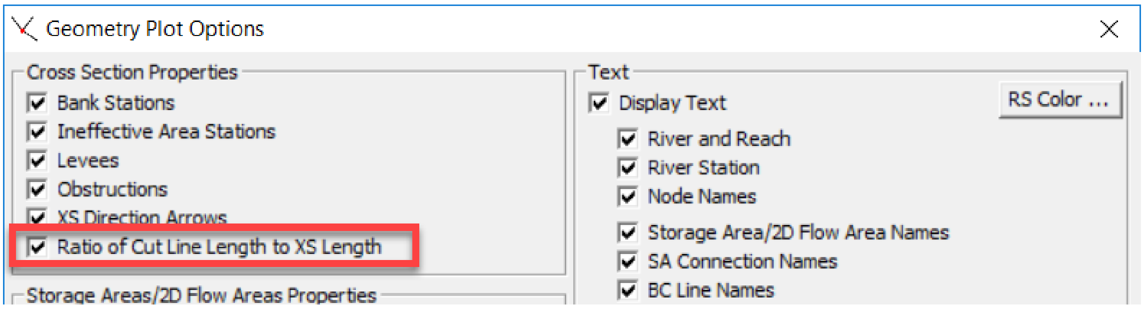
Below is an example demonstration problems with a channel surface that is created where the geospatial portion of the cross section differs from the station-elevation data. In the figure below, it is clear that the interpolation between the cross sections went poorly in **RAS Mapper**. Investigation of the interpolation surface showed no problems.



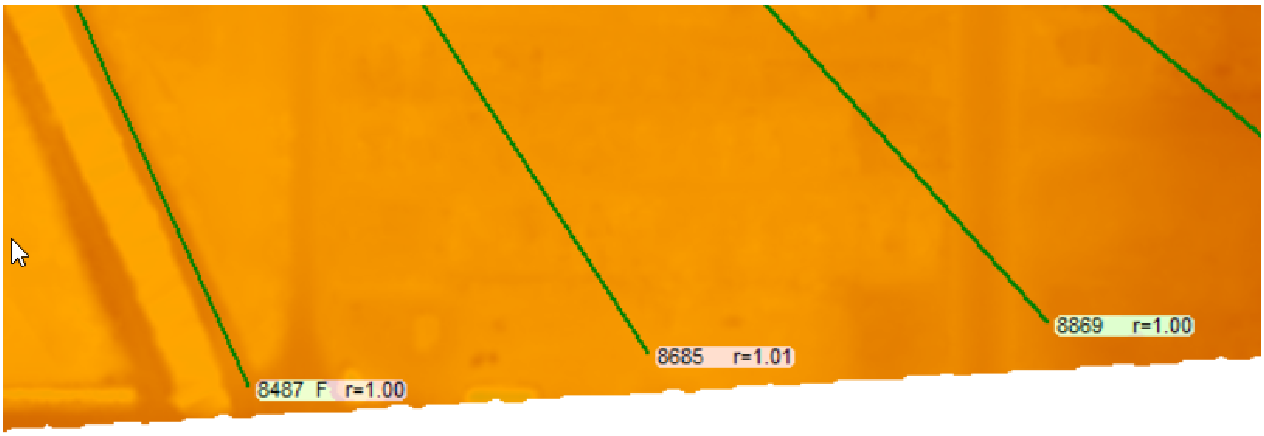
Right-click on the Bank Lines Layer and select **Compute Bank Lines from XS Bank Stations** (while Editing the geometry in RAS Mapper). Inspect the cross sections, river centerline, and bank lines together - there is an issue with the bank lines and bank stations. In RAS Mapper, the Bank Stations are created based on the length of the Station-Elevation data. The Bank Lines, however, were generated based on the geospatial length of the cross section. (We currently don't have a solution for this data problem in RAS Mapper, so we need to fix it over in the Geometric Schematic.)



Close RAS Mapper and open the **Geometric Schematic**. Go to the **View | View Options** dialog and turn on the **Ratio of Cut Line Length to XS Length** property.

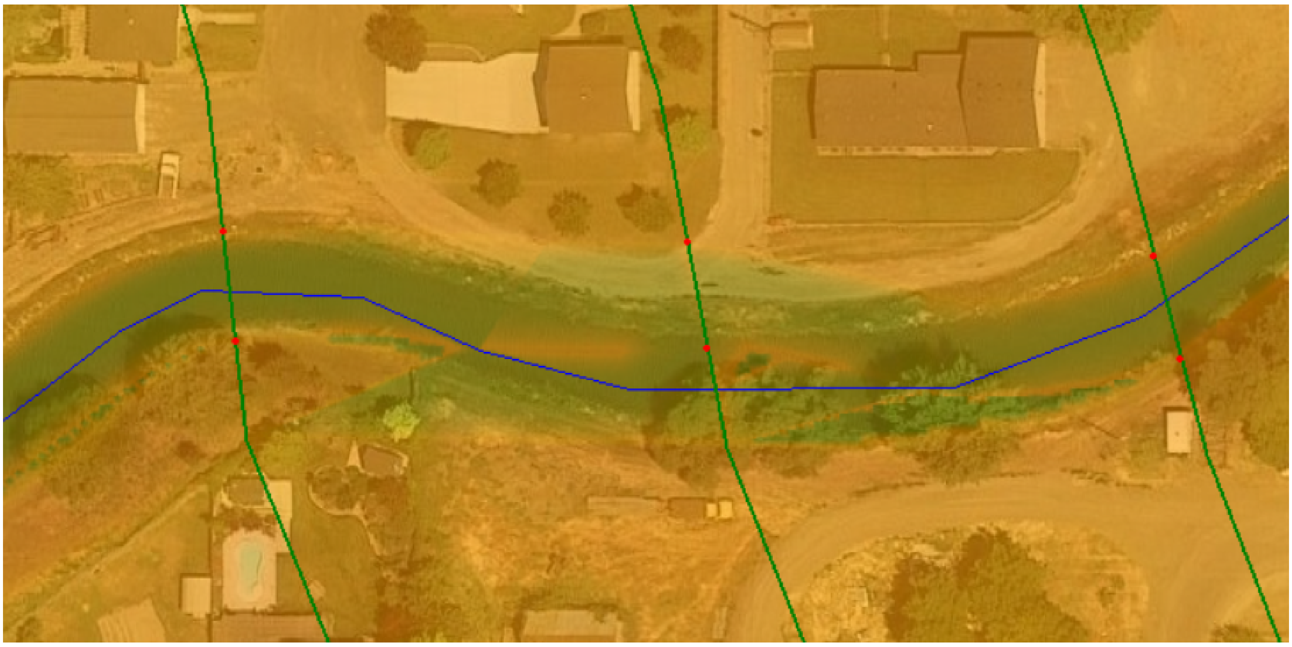


The computed ratio of the geospatial portion of the cross section and the station-elevation data will be displayed at the start of the cross section (left side) as shown in the figure below.

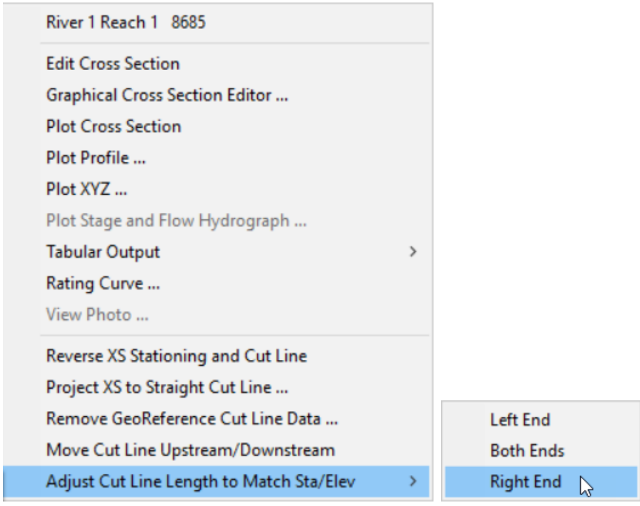


If the ratio (r) is greater than 1.0, the geospatial portion of the line is longer than the station-elevation data and the cut line should be shortened. If the ratio is less than 1.0 the geospatial portion of the line is shorter than the station-elevation data and should be lengthened.

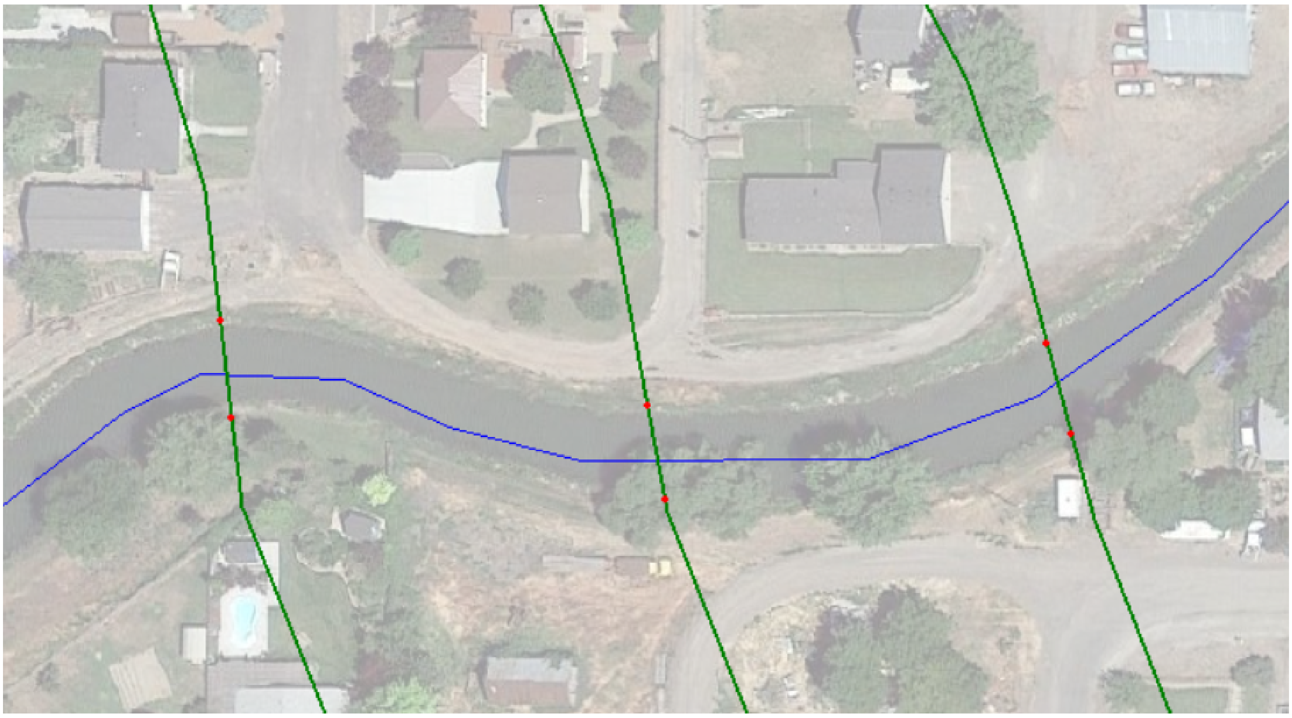
Because r=1.01, we have identified that the cut line length is too long, given the station elevation data. So how do we fix it? Based on the figure above (where the bank lines don't match the bank station information, we see that the bank lines are shifted too far "to the right" because the geospatial line length is too long. We can double check this information using background imagery. In the figure below, you can see the back station are indeed shifted off the channel into the right overbank. This means we need to shorten the geospatial line length, taking it away from the end.



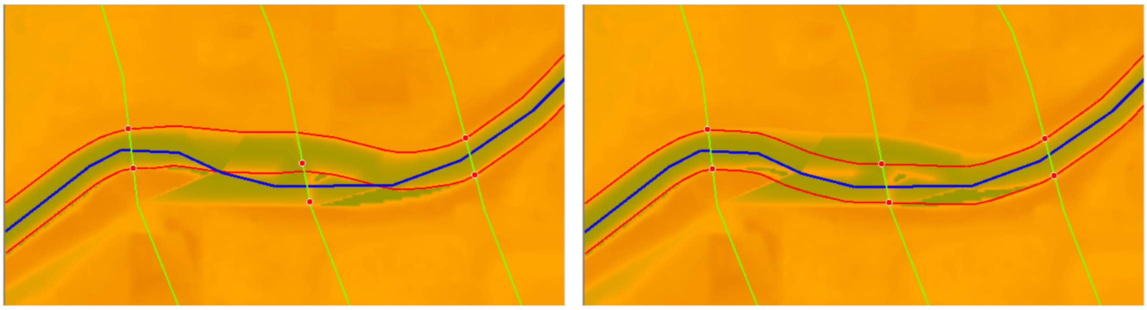
To shorten an cut line from the end, in the Geometric Schematic, left-click on the cross section and choose **Adjust Cut Line Length to Match Sta/Elev > Right End**.



(There is currently a replotting bug in HEC-RAS. So Save the geometry and then reopen it.) You will see the cut line changed and (likely) the bank stations will be in the correct location.

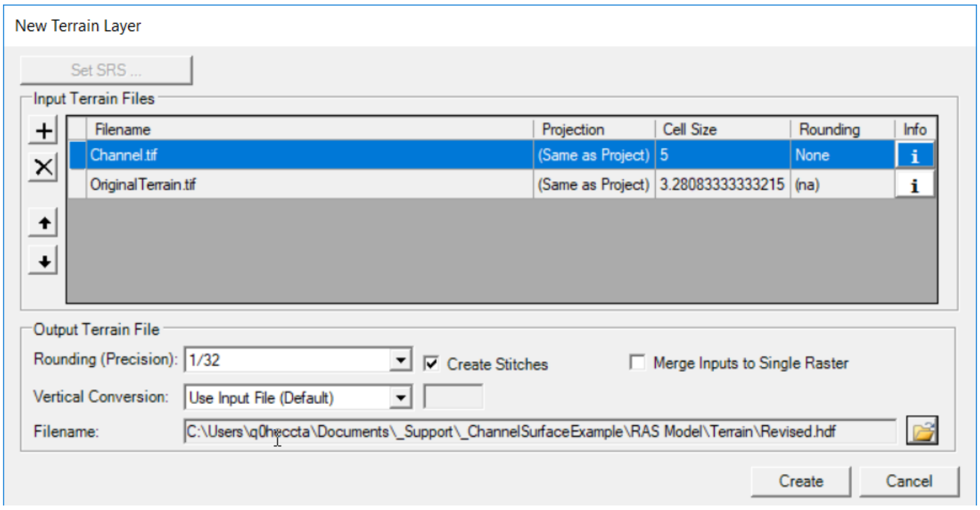


Now you can go back to RAS Mapper and perform the data export. Begin editing the geometry and **Compute Bank Lines from XS Bank Stations.** The original banks lines and new bank lines are shown in the figure below.

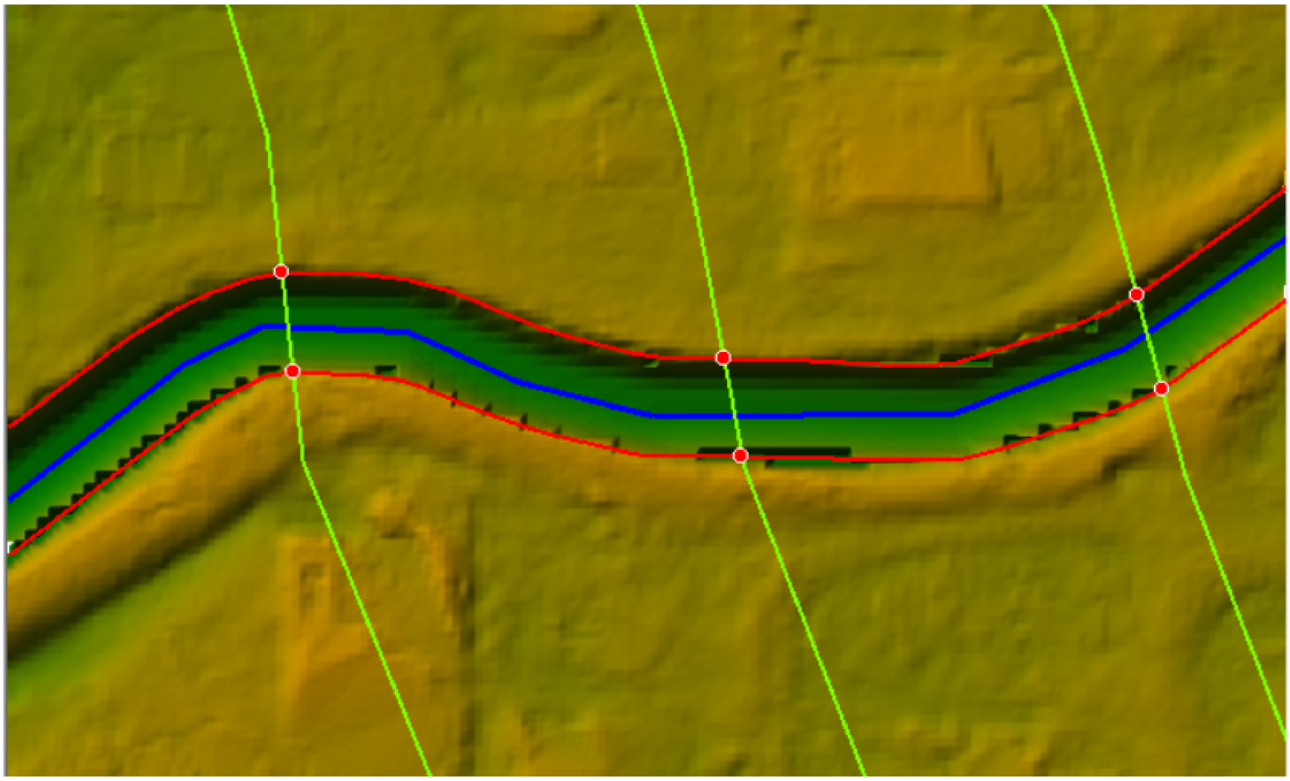


Inspect the bank lines, et. al and **Compute Interpolation Surface** and **Stop Editing**. Right-click on the Geometry and select **Export Layer | Create Terrain GeoTiff from XS's (Channel Only)**.

Next, merge the channel GeoTiff with an existing ground surface model by right-clicking on the Terrains group and select **Create a New RAS Terrain**. **Add** and **Prioritize** the terrain models.



Enjoy the fruits of your labor with a new terrain model that has channel information included.



1.3 Re-projecting Model Geometry

Georeferenced HEC-RAS models may need to be reprojected from one coordinate system to another. Typical use cases are a model that has been developed using a local coordinate system (e.g. State Plane) and needs to be converted to a National coordinate system (e.g. Albers projection). HEC-RAS provides tools to perform this coordinate system transformation with relative ease. These tools have been broadly used with 1D model data and have been adapted for 2D models but have not been used as extensively.

However, HEC-RAS does not support coordinate system transformation where a horizontal datum shift is required. Coordinate transformations in North America between spatial reference systems that use NAD83 and WGS84 or in Europe that use ETRS89 and WGS84 work because they share a very similar datum.

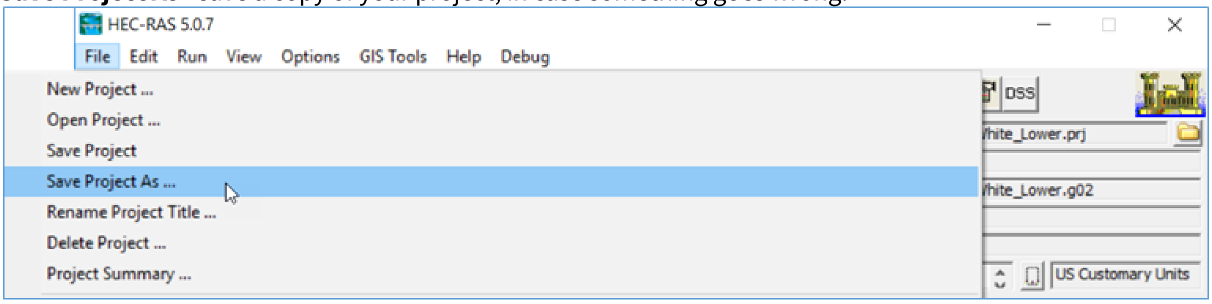
1.3.1 Step-by-step guide

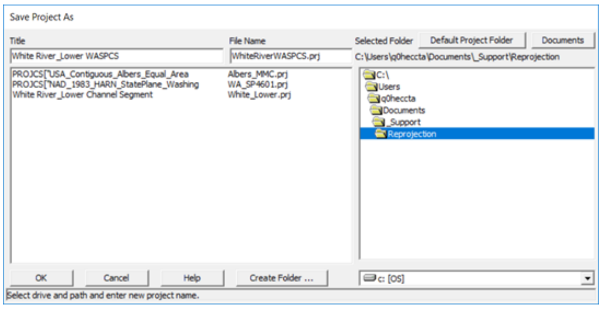
Prior to converting your HEC-RAS model, you will need to collect spatial reference system information for the current coordinate system an the coordinate system your are projecting the data into.

> ⓘ Currently, HEC-RAS only supports the esri projection file specification.

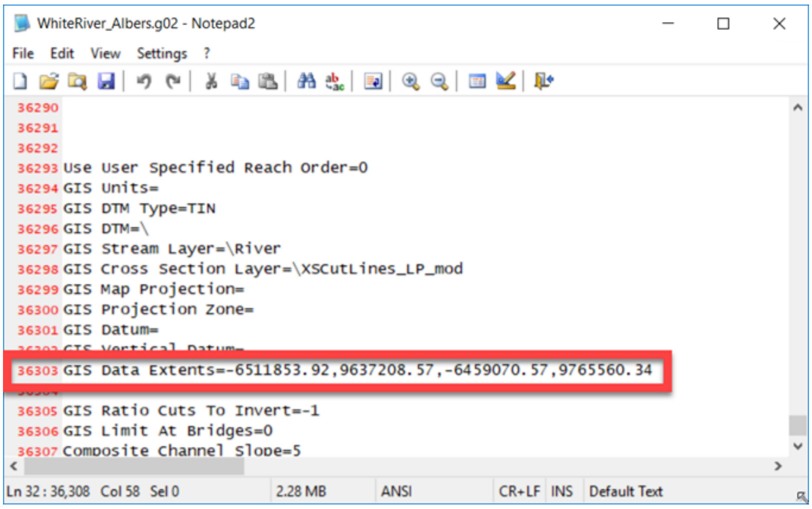
Geometry Conversion

1. **Save Project As** - save a copy of your project, in case something goes wrong.

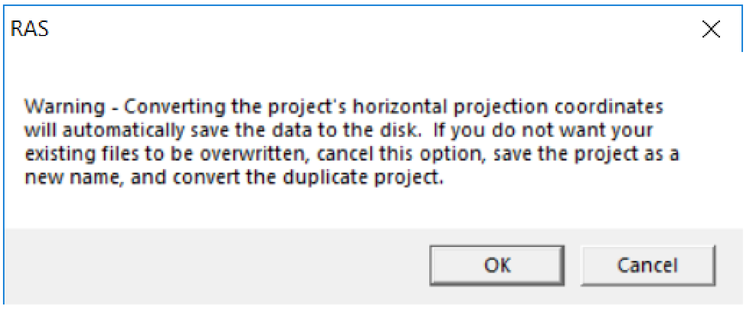




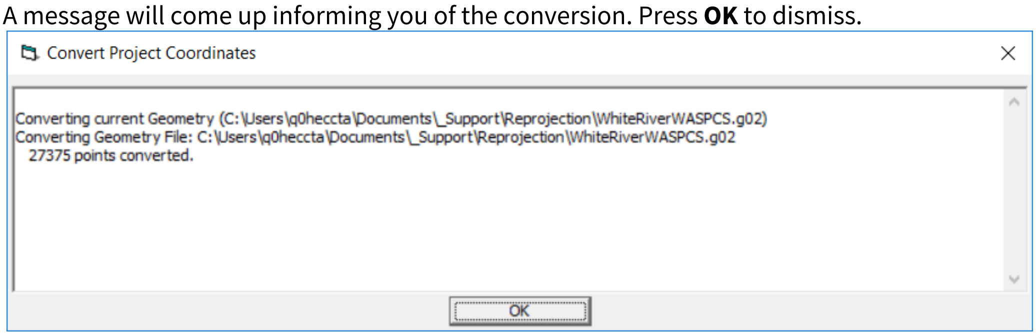
2. Check your geometry data file. There is a bug with versions 5.05 to 5.07 where the **GIS Data Extents** tag in the geometry file may have been improperly set. If the GIS Data Extents tag is empty, set it to "**GIS Data Extents=0,0,0,0**". If the data were generated with HEC-GeoRAS, the tag will be correct (see the example below).



3. **Convert the Geometry**
    a. Choose **Options | Convert Horizontal Coordinate System** from the main HEC-RAS window.
    b. Set the **FROM** coordinate system ("Current Project Spatial Reference System").
    c. Set the **TO** coordinate system ("Destination Project Spatial Reference System").
    d. Press **OK** to convert.
    e. A quick warning will come up - this is your last reminder to save your project as a copy of the original. Press **OK**.

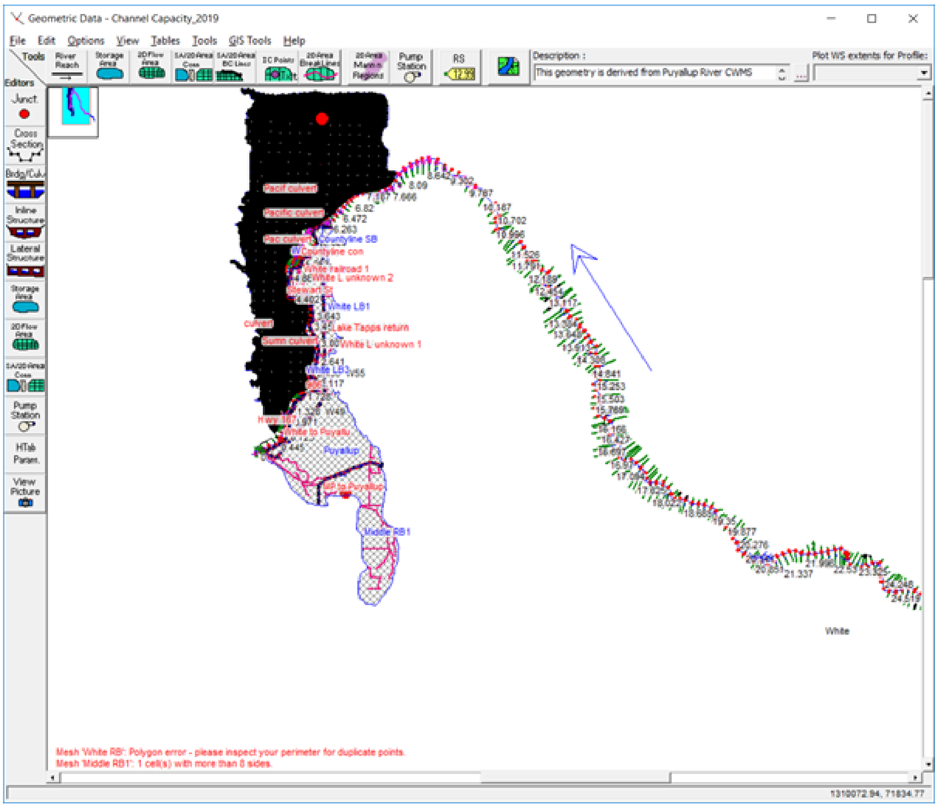


    f. A message will come up informing you of the conversion. Press **OK** to dismiss.



4. **Verify Geometry**
    a. The projection will *automatically* be set in RAS Mapper. Use background imagery to verify the data are in the correct location.
    b. Errors may have been created when the model was converted. This will be more likely to have happened with 2D Flow Areas. Errors in the geometry will be reported at the bottom of the Geometric

Data Editor.



Here are some things to look for that may need closer attention:

- Duplicate perimeter points
- Mesh cells with too many faces
- Breaklines **may** need to be enforced as points may not longer be exactly as desired
    - Previous hand edits of computation points will then be lost!
- Elevations extracted for cell faces may have “changed” with new terrain. This may result in elevations along Hydraulic Structures no longer being higher than minimum cell elevations. This may be true for Gates and Culverts as well.

Terrain Model Conversion

At this point you will have converted model geometry, but if you wish to visualize inundation results you will need a Terrain Layer in the same coordinate system. If you model is a 2D model, you will need the Terrain Layer for extracting hydraulic properties. The next step is to convert the terrain model.

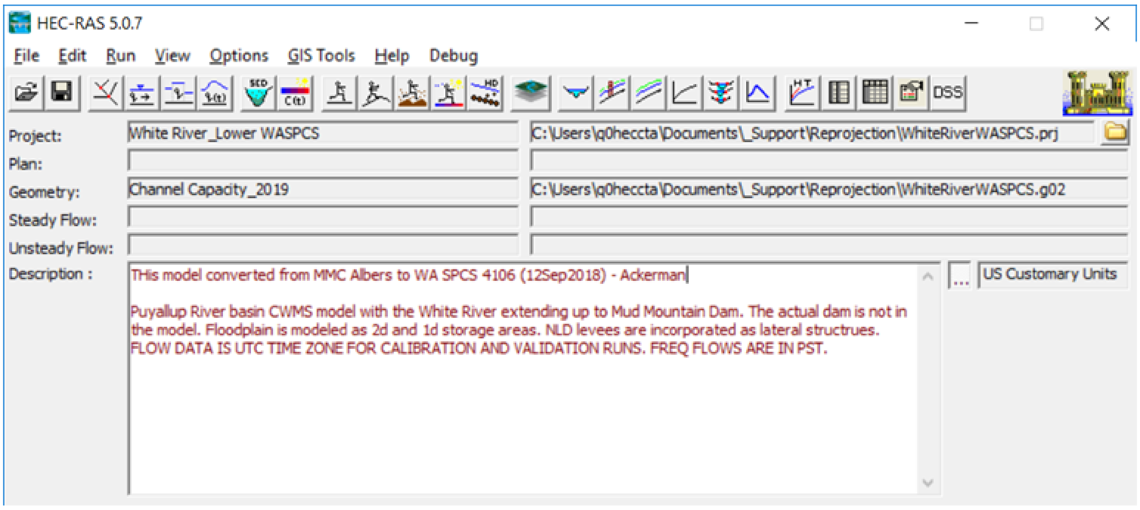
1. Open **RAS Mapper**.
2. Set the coordinate system with the new projection file by selecting the **Tools | Set Projection for Project** option.
3. **Create a New RAS Terrain** Layer
4. Verify that the data were converted on import.
5. Associate the RAS Terrain with the model geometry.

Manning's n Value Layer Conversion

If you have a geospatial Manning's n Value Layer, you will need to convert and associate that dataset as well.

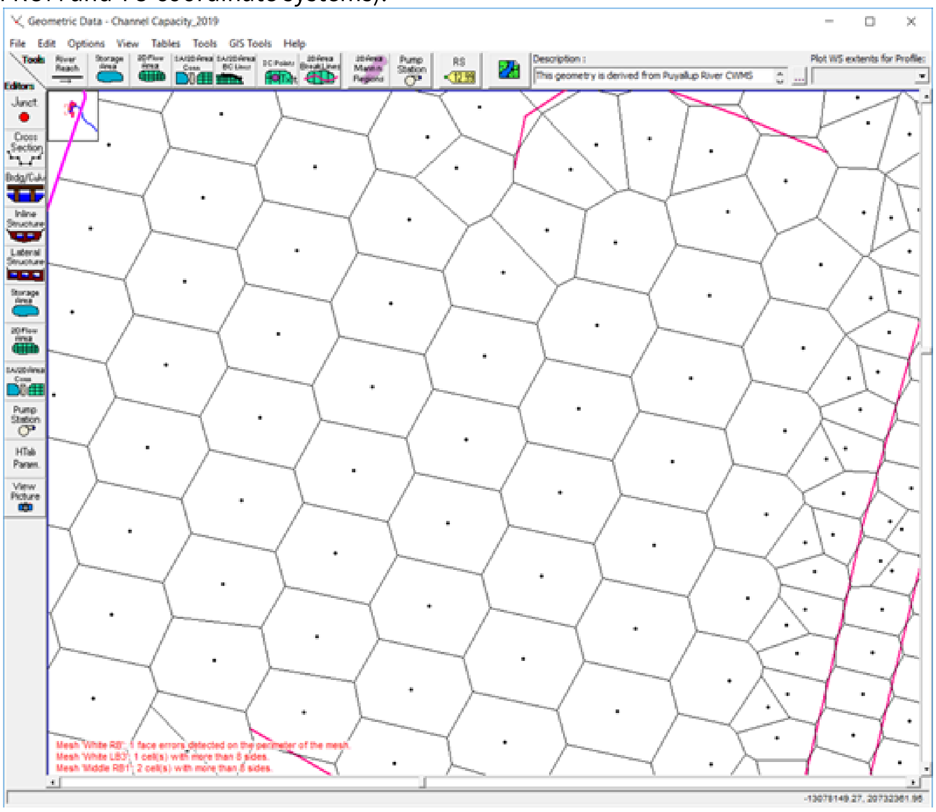
Document the Model

Document in the project file and the geometry files noting the coordinate system it is now in and any other information you think is pertinent



Problems?

If you see this cells with a lot of faces like the image below, you may have converted the data incorrectly. For instance, you may have inadvertently converted the data by using incorrect coordinate systems (e.g. switched the FROM and TO coordinate systems).

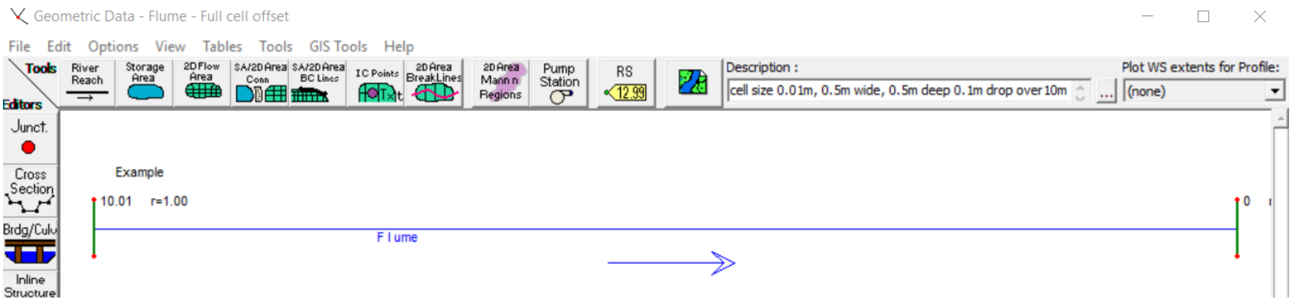


1.4 Creating a Terrain Dataset to Model a Flume Experiment

Typically, we use HEC-RAS to model real-world situations. However, sometimes we might find the need to model large-scale (small dataset) problems like a flume. The main problem in create a terrain model is that HEC-RAS uses a raster for storing data; however, this raster is going to be interpolated based on a cross section layout which will result in a terrain model that is not perfect. This document will discuss how we can use HEC-RAS to create a RAS Terrain which we then can used to create a 2D model and try to address some things to think about when creating it.

1.4.1 How Does it Work?

Before you get started you will need to know the dimensions of the dataset and arrive on the grid cell size that the resulting terrain model is going to be. All of the data will be created from cross sections and the variations in terrain will be based on the cell size selected. For this example, we have a flume that is 10m long, 0.5m wide, and 0.5m deep with a 0.1m drop over it's length. You will want enough cells to properly represent the terrain across the channel. For this case, we will use a cell size of 0.01m which will give us adequate defining in cross section (50 cells across).



You will use the flume dimensions and predetermined grid cell size to create a set of HEC-RAS cross sections. Once the HEC-RAS model geometry is constructed, you will export the geometry to a ground surface (TIF), and then use the ground surface to create a RAS Terrain. Creating the ground surface from cross sections ends up truncating the data around the edges by 1/2 a grid cell size, so you will need to consider that when making the data and make your model 1 grid cell longer and wider on all four sides of the model (2 cells longer, 2 cells wider).

Buffer the model domain by 1 grid cell length

Make your model 1 grid cell size longer at the upstream and downstream end and 1 grid cell wider on the left and right overbank.

1.4.2 Step-by-step guide

Perform the following step in HEC-RAS:

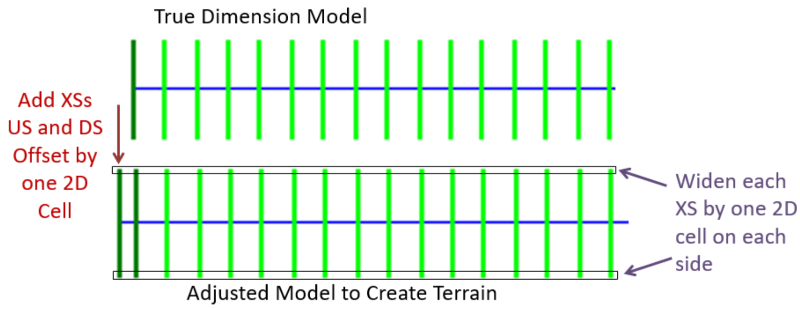
1. Create a new RAS project, set the Unit System, and start a new Geometry.
2. From the Geometric Data Editor, **draw a River Centerline**, providing a river and reach name.
3. Open the **GIS Tools | Reach Invert Lines Table** to enter the XY coordinates for the river line.
   Enter the geospatial coordinates for the river line. (Note: the river should be 1 cell farther upstream and downstream of the model limits - an easy way to do this is add a row at the start and add a row at the end.)

	Schematic X	Schematic Y
1	-0.01	0
2	0	0
3	10	0
4	10.01	0

For the example, above, the flume is 4.25m long. Additional, length (0.001m) was added based on 1 grid cell size upstream and downstream.

4. Create bounding cross sections at the top and bottom of the river reach. The "base" cross sections will be at RS 4.25 and RS 0. You will also need to add them to RS 4.251 and RS -0.001. Further, the cross sections must be wider (0.001m on both sides) than the true dimension.

RS 10.01/10		RS 0/-0.01	
Station	Elevation	Station	Elevation
-0.26	0.6	-0.26	0.5
-0.25	0.6	-0.25	0.5
-0.25	0.1	-0.25	0
0.25	0.1	0.25	0
0.25	0.6	0.25	0.5
0.26	0.6	0.26	0.5

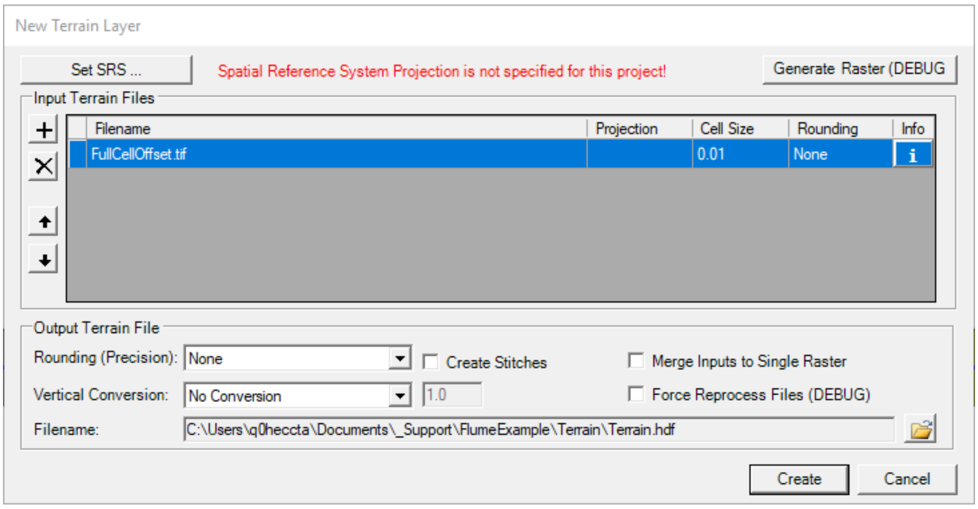


5. Reach lengths will vary for each cross section. For RS 4.25 the reach length is full length of the reach (4.25m). For RS 4.251 and RS 0 use the cell size (0.001m).
6. Set Bank Stations.
7. Open the **GIS Tools | CS Cut Lines Table** to enter the XY coordinates for each cross section. The cut lines must match the station-elevation data. (It's easiest to think of the main channel and then then extend the left and right sides by 1 grid cell length.)

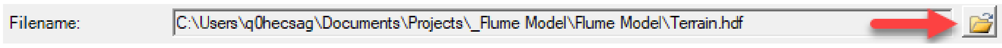
RS 10.01		RS 10		RS 0		RS -0.01	
X	Y	X	Y	X	Y	X	Y
-0.01	0.26	0	0.26	10	0.26	10.01	0.26
-0.01	0.25	0	0.25	10	0.25	10.01	0.25
-0.01	-0.25	0	-0.25	10	-0.25	10.01	-0.25
-0.01	-0.26	0	-0.26	10	-0.26	10.01	-0.26

8. **Save** the Geometry.
9. **Open RAS Mapper.**
10. Right-click on the Geometry and choose **Export Layer | Create Terrain GeoTiff from XS's (Overbanks and Channel).**
11. Provide a **filename** and **cell size** (0.01 for this example).
12. Right-click on the Terrains group and choose **Create a New RAS Terrain**.
13. Click **No** to disregard having a projection.

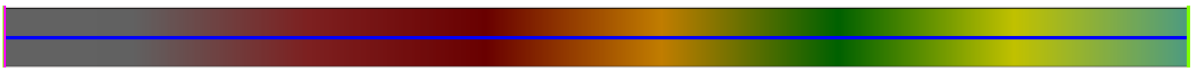
14. Add the ground surface raster, choose **None** for Rounding, **No Vertical Conversion**, and **uncheck** Create Stitches.



15. Provide a filename → click on the folder button to give the terrain a name other than the generic "Terrain.hdf"



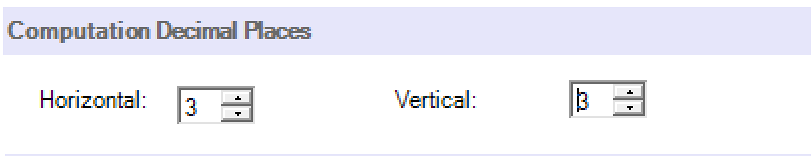
16. Click **Create**.
17. You can now inspect your Terrain model.



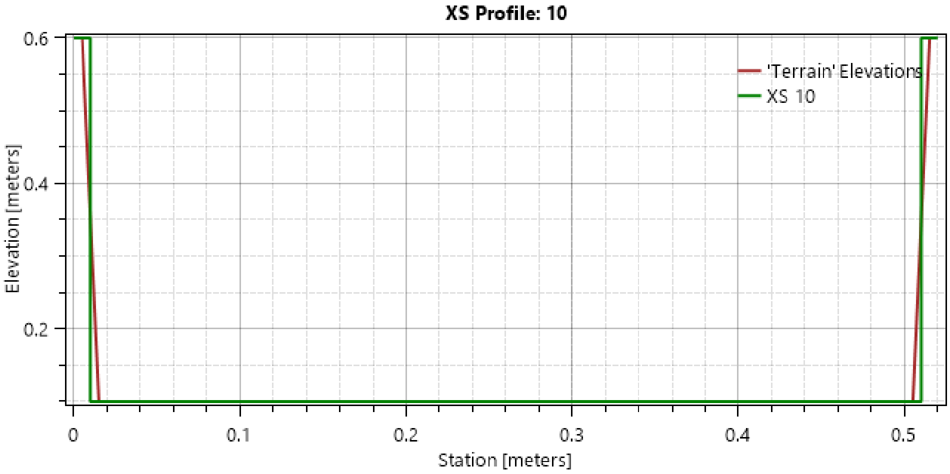
1.4.3 Some Things to Consider

You now have a terrain model with which you can visualize HEC-RAS results from a 1D model or you can use as the basis for creating a 2D model.

In order to properly plot the data (like in the figure below), you will need to set the precision for the horizontal and vertical data. Because we are using data to many decimal places, match the horizontal precision from the **Tools | Options** menu.



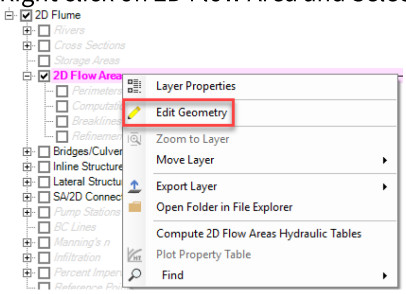
Take a look at your new RAS Terrain. Raster datasets have a single value per grid cell; however, RAS interpolates data from cell center to cell center so that a continuous profile may be extracted. This will result in a slope on the walls of the flume. You can experiment with elevations of the extended points in the station-elevation data to get the best representation of the channel for your purposes.



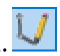
1.4.4 Creating a 2D Model

When creating a 2D model domain, the model should run from the upstream cross section (RS 10) to the downstream cross section (RS 0), ignoring the boundary cross sections. You can use the geospatial coordinates of the cross sections to create the mesh boundary.

1. In RAS Mapper, **Add a New Geometry**
2. Right click on 2D Flow Area and Select **Start Editing**



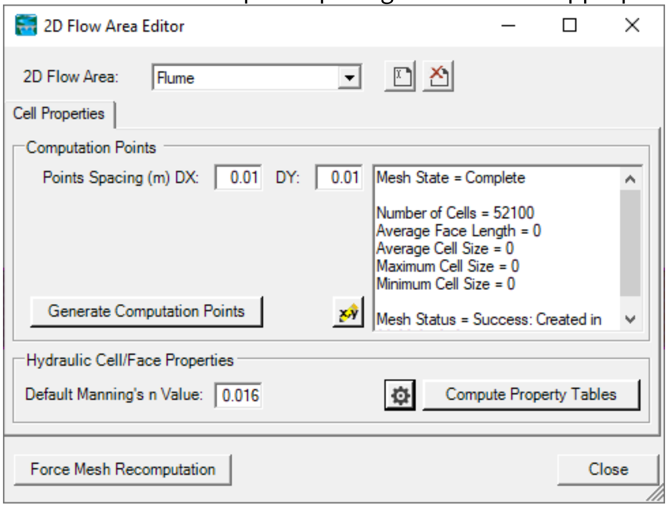
3. Add a perimeter: Click on **Perimeter** 

and then choose the draw tool.  (just 3 points is fine).
4. Select the perimeter, right-click on the perimeter and choose the **Geospatial Operations | View/Edit Points** menu item. (If you can't find this function, go to the geometry editor and select **GIS Tools → Storage Area/ 2D Area Outlines)**
5. Enter the true coordinates of the flume (without the extra cells added) points in consecutive order.

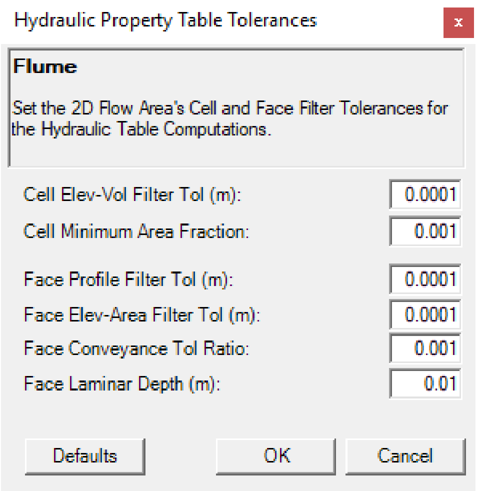
	X	Y
1	0	0.25
2	10	0.25
3	10	-0.25
4	0	-0.25

6. Edit the 2D Flow Area

7. Provide the cell mesh point spacing and enter the appropriate n value



8. Enter appropriate hydraulic table parameters

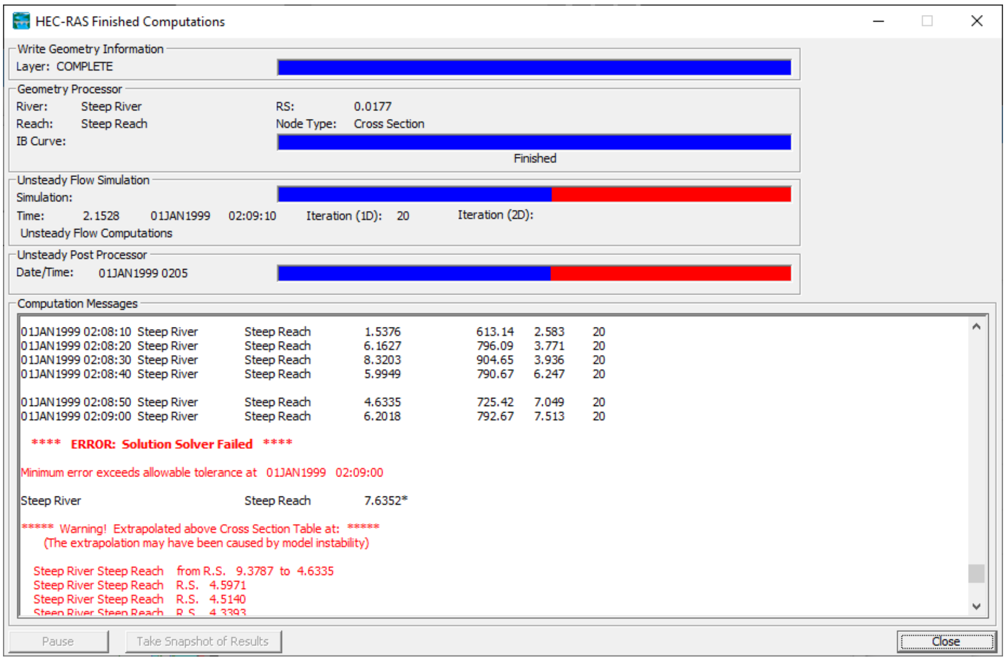


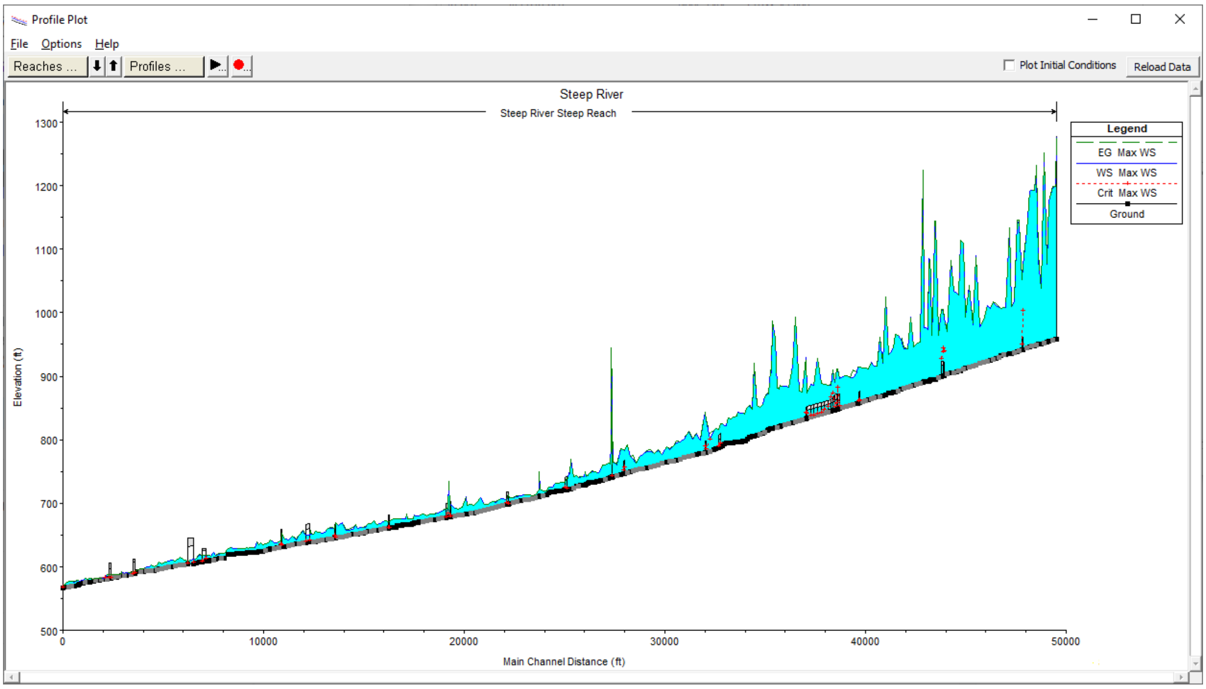
9. Create your upstream and downstream boundary conditions location in RAS Mapper.
10. Stop Editing.
11. Enter flow data for the upstream boundary.
12. Set up you Plan and Simulate.

1.5 Modeling Steep Reaches

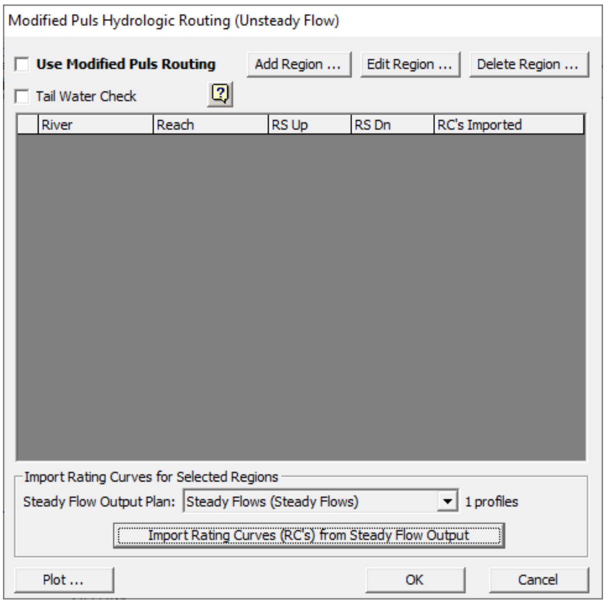
Frequently, we are developing an unsteady-flow river hydraulics model and the finite difference methodology goes unstable during the simulation. This can occur in extremely steep river reaches and when modeling dam breach events where the hydrograph is rapidly rising from a low base flow (usually in very steep terrain). The model may be going unstable for many other reasons; however, for the case where the reach is extremely steep (greater than 1% slope) and flows are low, the model is likely unable to solve the full shallow water flow equations because the water surface is going supercritical and flow depths are small. This is typically because, for the prescribed flow, the solution to the equations results in high velocities and shallow depths and the resulting derivatives for the change in water surface depth ends up being large compared with the depth from the previous time step.

There are many modeling capabilities in HEC-RAS than can be considered to improve model stability: increasing Manning's n values (see also Jarrett's equation), decreasing model time step, using the Advanced Time Step Control, adding Inline Structures for those extremely steep drops, turning on the Mixed Flow Regime, and/or incorporating Pilot Channels into the geometry. Each of the model capabilities should be considered; however, sometimes the model is just very steep and the finite difference solution scheme will just not stay stable. When you go to run your simulation, you end up with the dreaded "red" bar and, despite your best efforts, the model simulation continues to go unstable.

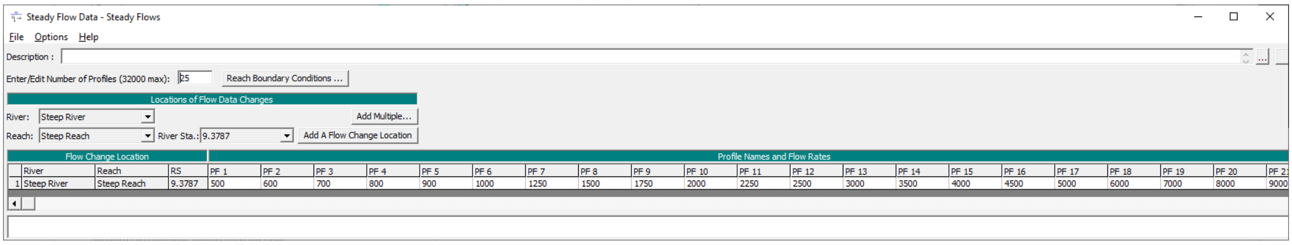




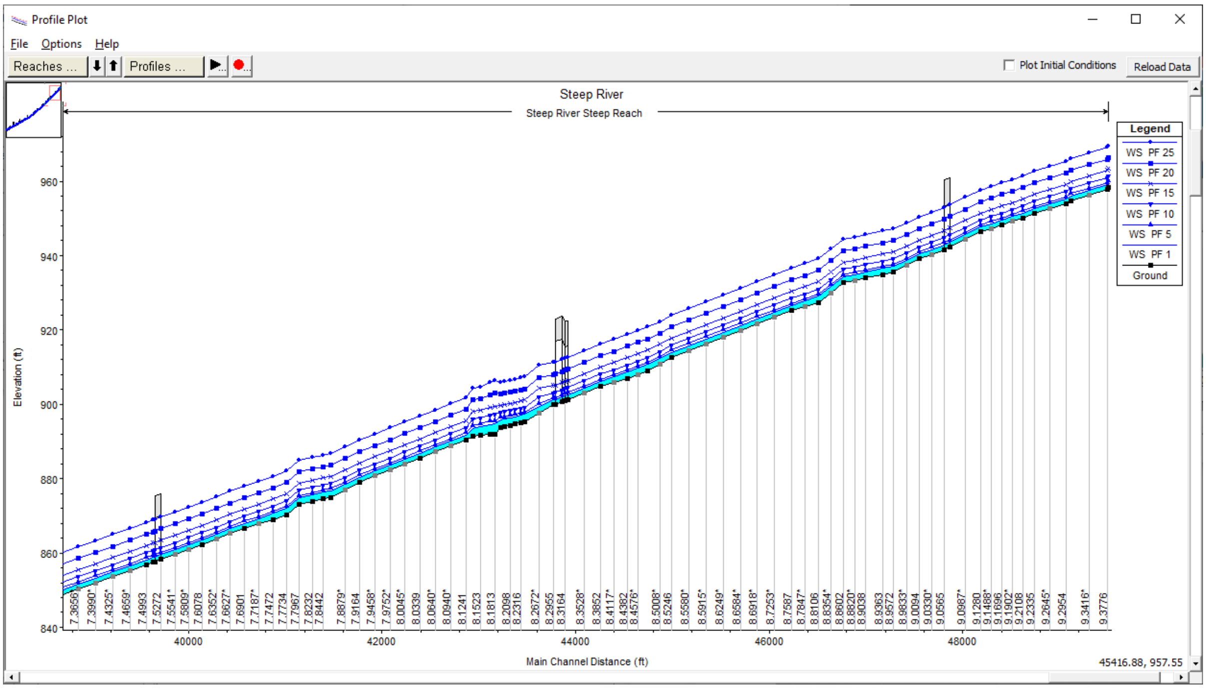
At some point, you should consider the Hydrologic Routing Method that is available in HEC-RAS from the **Geometric Data Editor | Options | Hydrologic Unsteady Routing** menu item. The Hydrologic Unsteady Routing option allow you to define portions of the model to be routed with a hydrologic routing technique instead of using the full unsteady-flow equations. The hydrologic routing method is based on the Modified Puls routing technique.



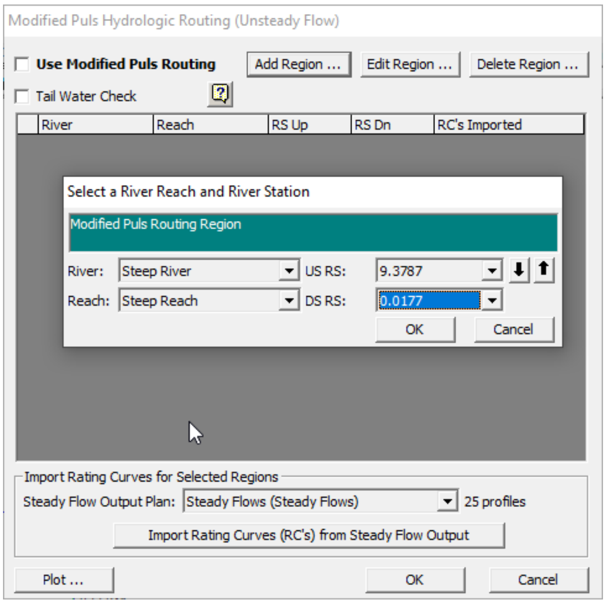
As you can see from the **Modified Puls Hydrologic Routing** window, you will need to specify a Region (routing reach) to utilize the routing and Import Rating Curves for that reach. This requires that you set up a steady flow plan that has multiple steady flow profiles representing the range of flows you will simulate (at least 20 is usually recommended).



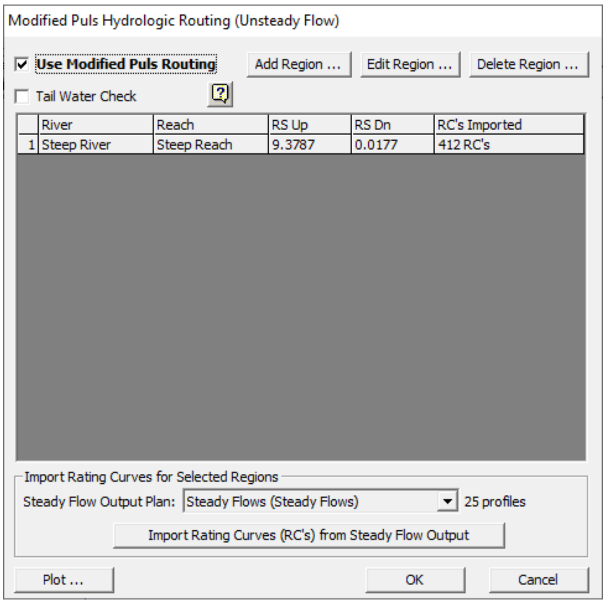
Run the Steady Flow Analysis to develop the water surface profiles for the multiple runs.



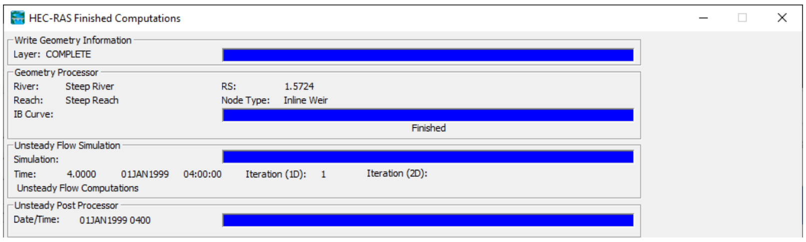
Next, set up the **Hydrologic Unsteady Routing** option. Click the Add Region button from the **Modified Puls Hydrologic Routing** window.



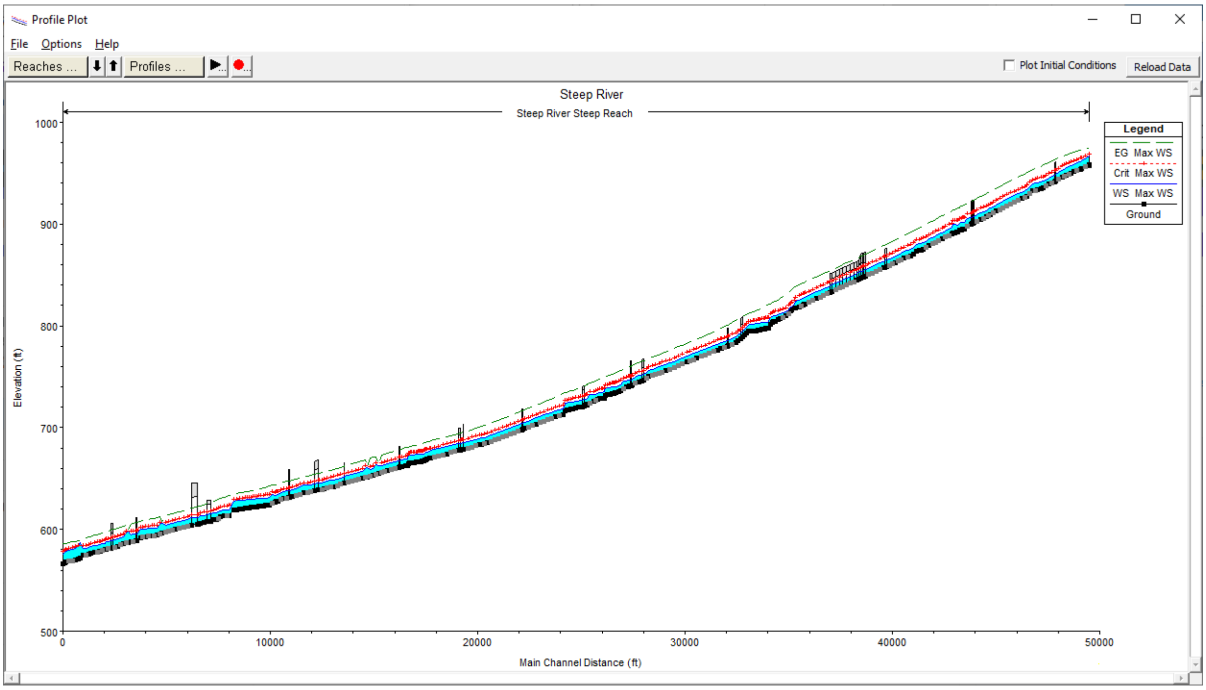
Then Import Rating Curves for the selected reach by clicking the **Import Rating Curves (RC's) from Steady Flow Output** button. You must have the same geometry as was used in the steady flow simulation. Once imported, the River Reach will show how many Rating Curves will be used - a rating curve for each cross section in the model. Make sure to turn on the hydrologic routing method by checking the **Use Modified Puls Routing**.



Run the unsteady-flow simulation. The areas where you used Modified Puls routing will solve, producing a stable solution.



The Profile Plot will should show a reasonable result with no steep spike due to model instability.



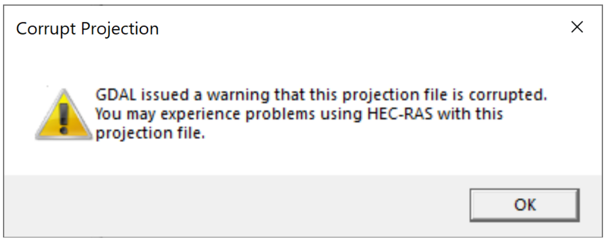
1.5.1 Limitations and Considerations

- Users often ask, whether Modified Puls routing (which is thought of as level-pool routing) can be used in the steep reaches. Yes, you can. The rating curves that are developed for use in unsteady-flow routing in HEC-RAS are built from sloped water surface profiles! Therefore, think of it as linear routing, not level-pool routing.
- To use the Modified Puls routing method, you must develop rating curves using the exact geometry used for the unsteady-flow simulation.
- Modified Puls routing reaches much be broken up at junctions.
- The Modified Puls routing can span across bridges and inline structures.
- However, the Modified Puls should be broken up at any inline structures that have gates. For a large inline structure/dam, you might want to start a Modified Puls region immediately downstream of the structure, but you wouldn't typically need a Modified Puls region immediately upstream of the structure.
- The Modified Puls can compute flow over lateral structures.
- For steep reaches, the Modified Puls can often provide good answers. One limiting factor may be whether the steady flow rating curves represent the unsteady flow. For steep reaches, the downstream tailwater effect does not propagate as far upstream as it would for shallower reaches. So the unsteady flow WSE often matches the steady flow rating curve. Nevertheless, if the Modified Puls is influenced by a tailwater that is not adequately captured by the steady flow rating curves, there will be a loss of hydraulic accuracy. For instance, if the downstream end of a Modified Puls region is influenced by a reservoir that has a varying water surface that is not strongly correlated with the flow. However, even some loss of hydraulic accuracy may be acceptable in order to prevent instability. This is especially true in the situations where the Modified Puls is only being used over a few problematic cross sections.
- The tailwater check option was intended to allow the Modified Puls region to incorporate the effects of when the tailwater was higher than the value from the rating curve. However as far as stability, this option is not as robust as using the Modified Puls without this option. It is recommended to develop the Modified Puls regions without the option. And then, if desired, selectively turn the option on checking that stability is maintained.

1.6 GDAL Projection File Warning

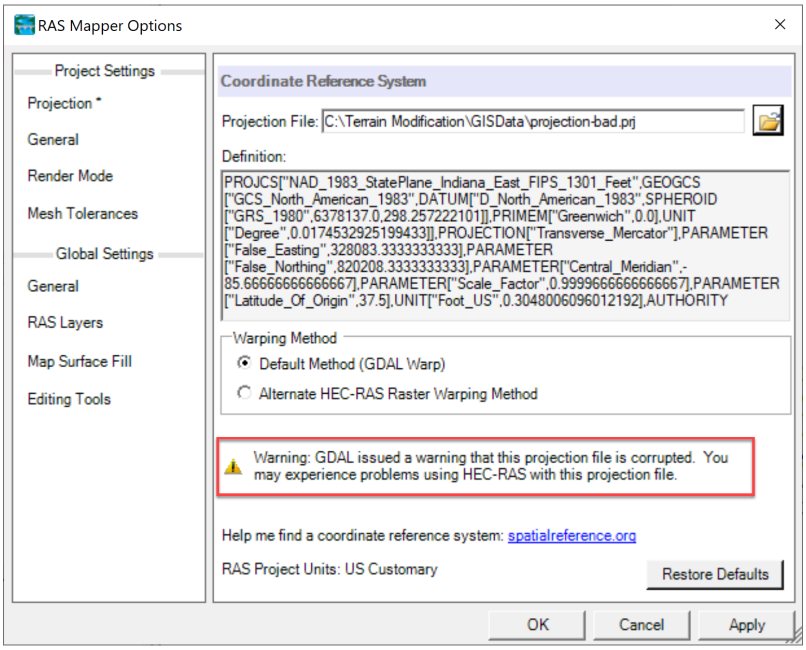
To learn more about projection files and how RAS Mapper uses spatial reference systems, go here[4].

Sometimes a spatial projection file (*.prj) will produce the following warning in RAS Mapper.



The warning message is also displayed in the Coordinate Reference System of the RAS Mapper Options window as shown below.

4 https://www.hec.usace.army.mil/confluence/rasdocs/rmum/latest/spatial-reference-system



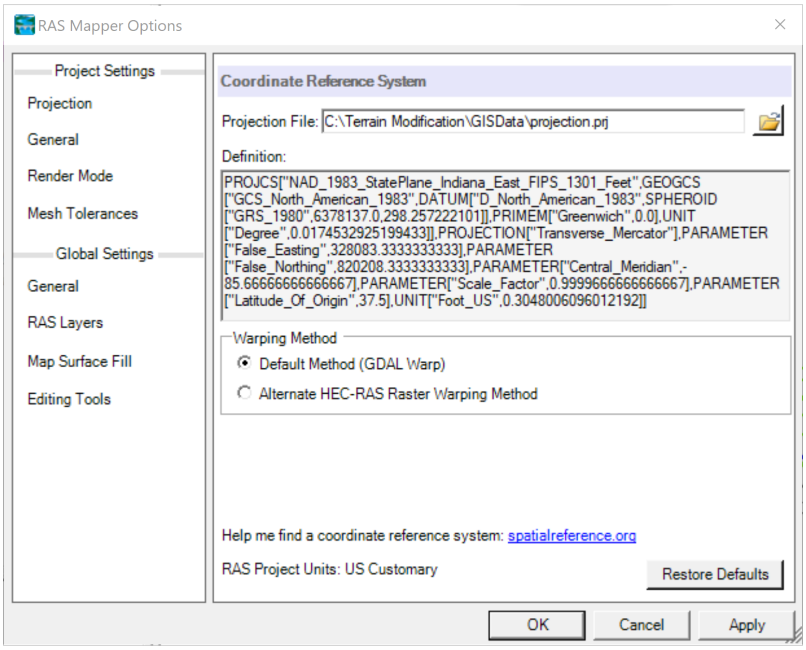
One common cause of this error is a field AUTHORITY["X",Y]. The field is usually at the end of the projection file as in the example below

```
UNIT["Foot_US",0.3048006096012192],AUTHORITY["EPSG",2965]]
```

To avoid the warning message, edit the projection file in a text editor and remove the AUTHORITY field as shown below

```
UNIT["Foot_US",0.3048006096012192]]
```

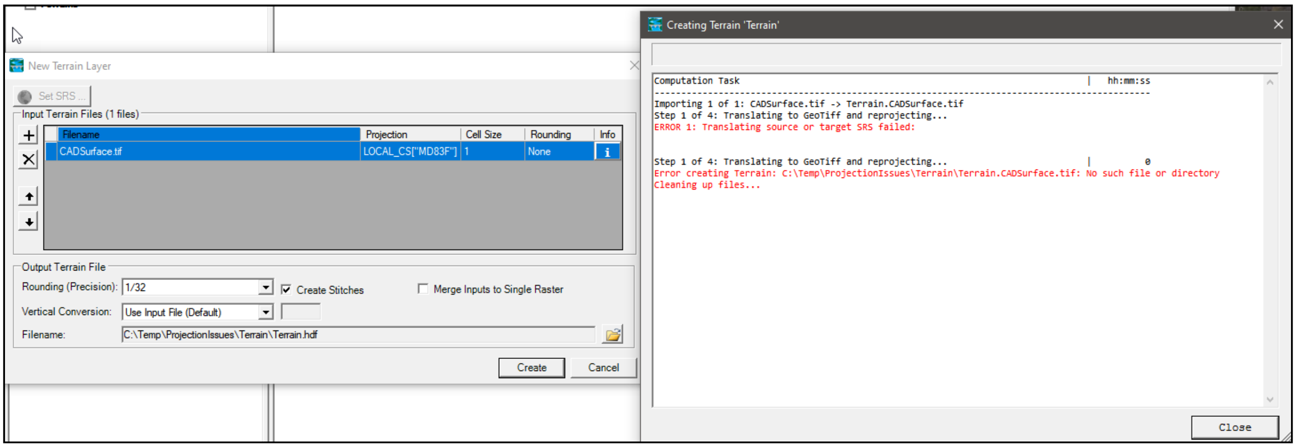
Then set the projection file again in RAS Mapper as the warning message should disappear (see figure below).



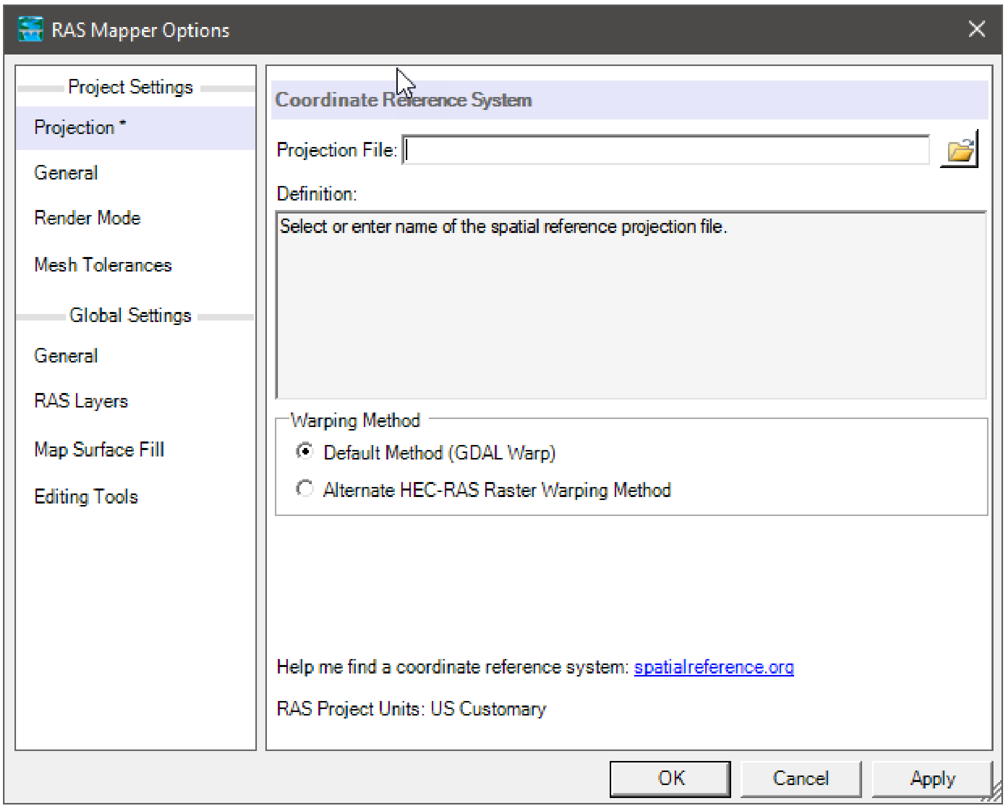
1.7 Skip SRS Translation For Terrain Imports

HEC-RAS Mapper can create terrains from a wide variety of raster data formats. Some of these formats such as GeoTIFF can include a spatial reference system (SRS) stored internally within the file. When this internal projection data is not stored properly, or is in a format that HEC-RAS geospatial libraries don't recognize, you will NOT be able to reproject the raster and create an HEC-RAS terrain. This guide discusses how to work around an unrecognized internal raster projection (or when NO projection is specified) by skipping the SRS translation step and importing the terrain dataset directly into a RAS Terrain.

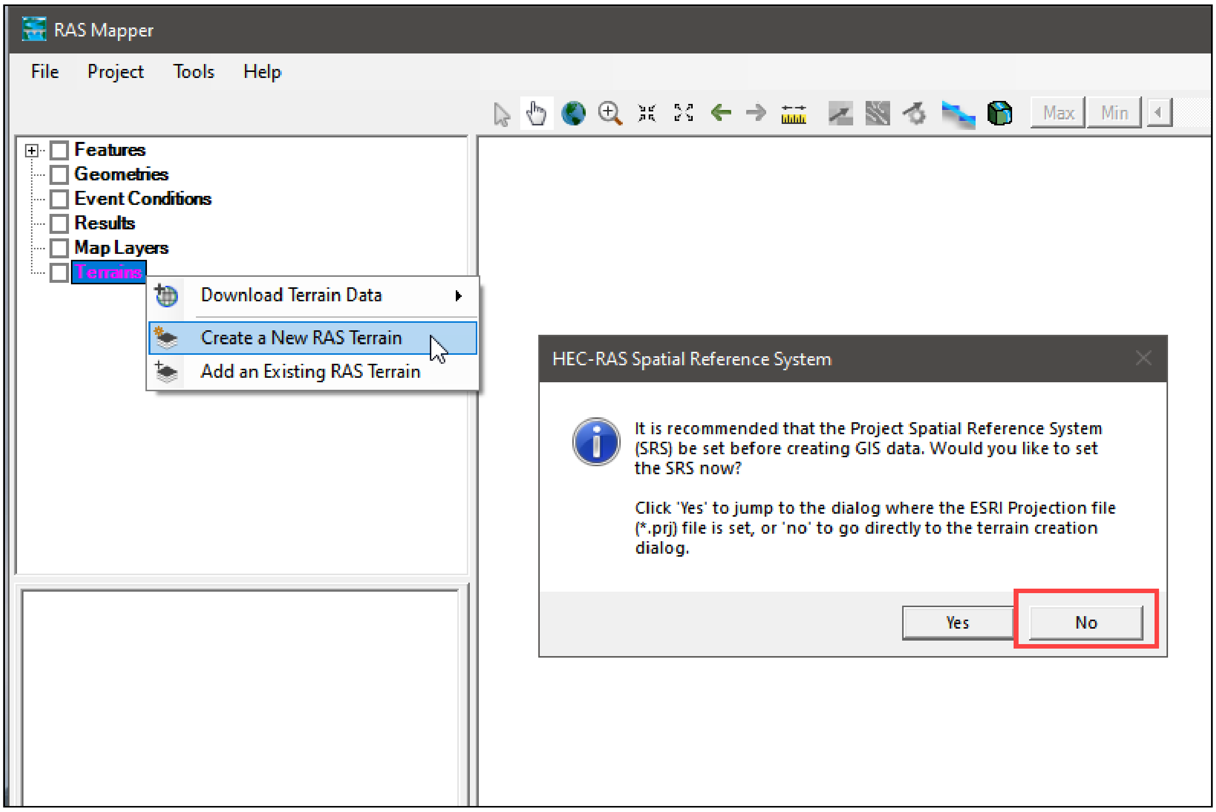
A raster surface was exported from a CAD application as a GeoTIFF with an internally stored SRS. When attempting to create a new terrain with this surface in HEC-RAS Mapper an error occurs indicating there was an issue translating the the SRS:



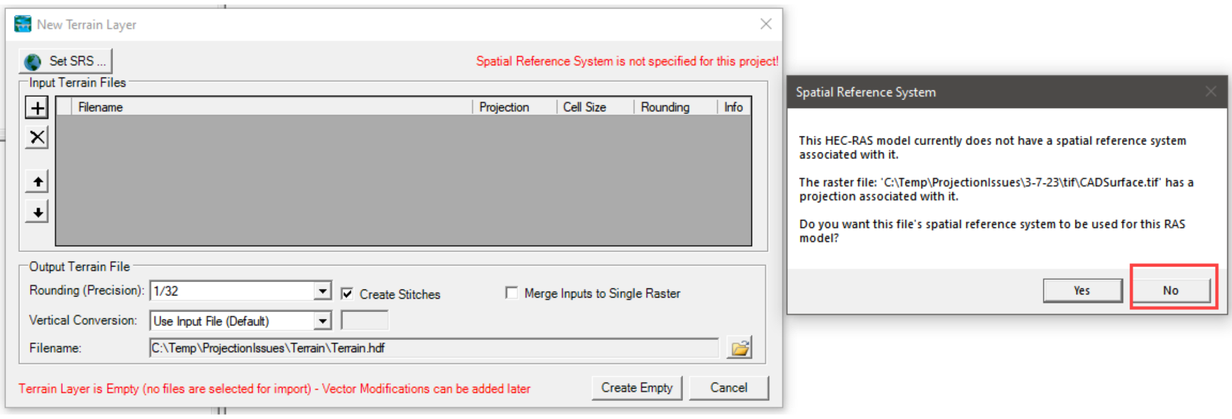
First, **clear the SRS** from the RAS Mapper Options and select **Apply**. This will ensure that HEC-RAS will not attempt to use the source raster internal SRS data to reproject it into the project SRS.



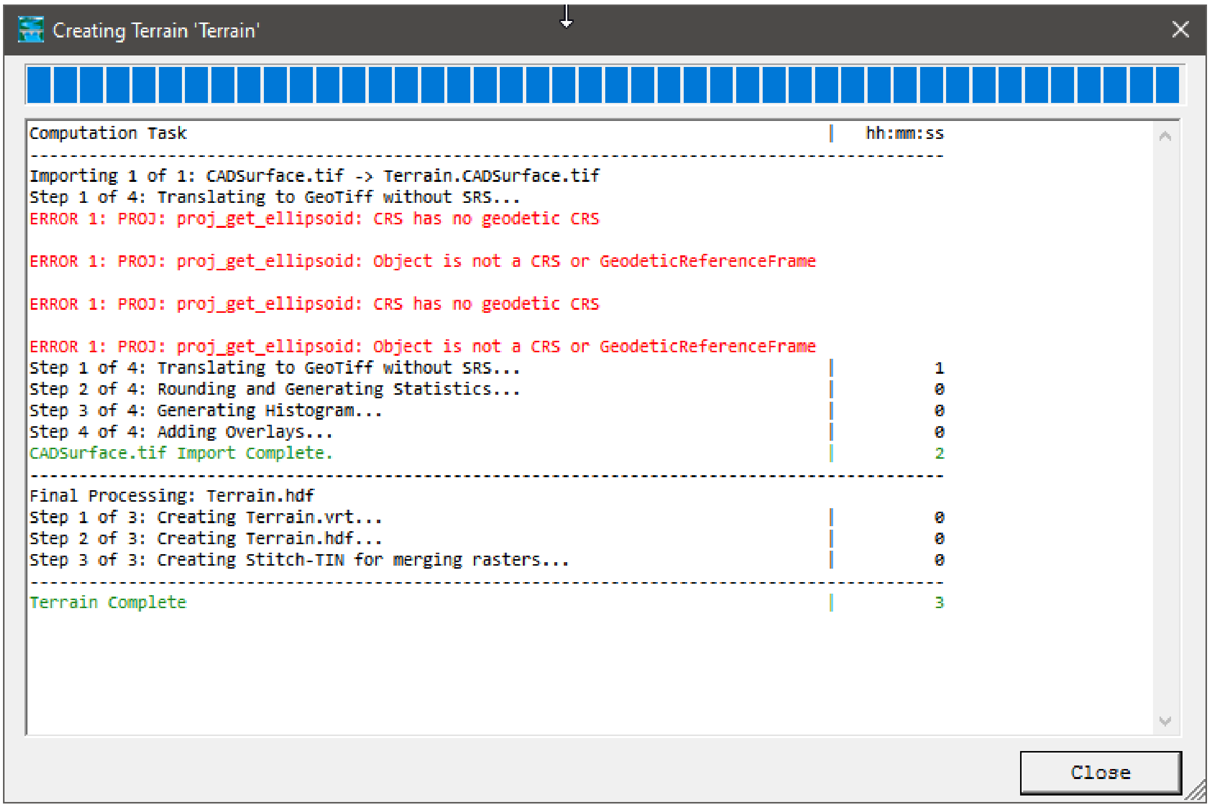
Next, create a new RAS terrain, and Respond "**No**" to the dialog requesting a to set Project SRS:



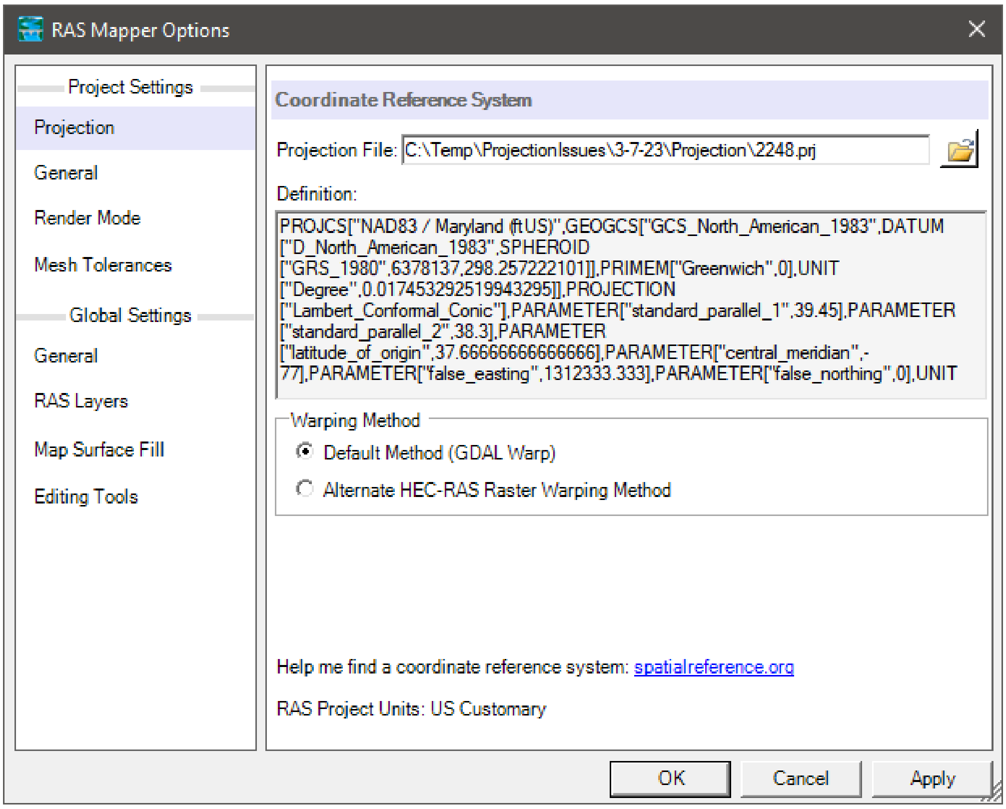
In the terrain importer select the source terrain data. Another message dialog will appear asking if you want to use the raster's SRS for the project. Select "**No**".



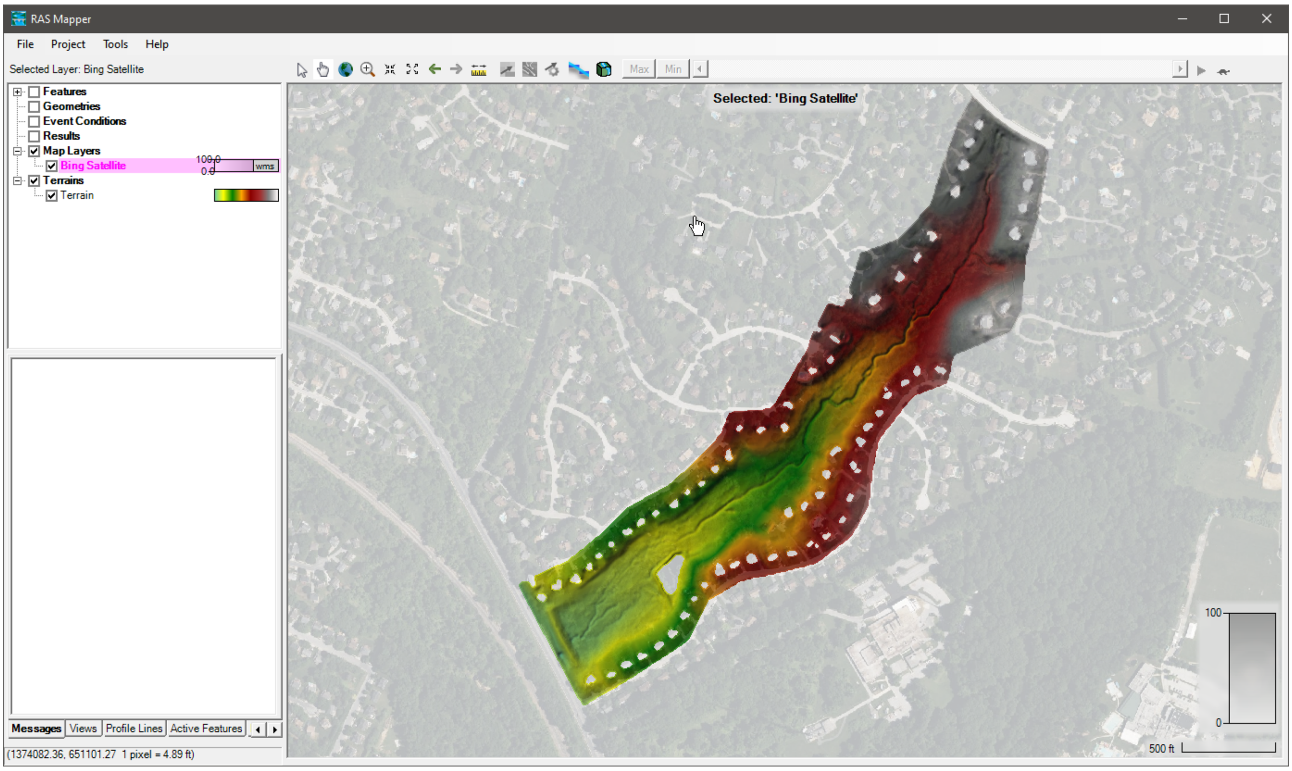
The terrain import process will complete, but errors are shown indicating that the SRS is missing and no reprojection was done: "*Step 1 of 4: Translating to GeoTIFF without SRS...*" Since the SRS was missing RAS Mapper imported the terrain data but skipped the reprojection step. That means the new HEC-RAS terrain will maintained the projection of the source raster data as is.



Next, set the Project SRS to the know projection of the terrain data.



Finally, verify the terrain is projected correctly using background imagery .



2 Tutorials

Tutorials are intended to walk users through a task using detailed instructions with example data.

2.1 Downloading Terrain Data

Ground surface elevation data is crucial to developing a good HEC-RAS model. Often, identifying terrain dataset for use in the study can be a difficult task. HEC-RAS is attempting to streamline the process of the getting the modeling process started by providing tools to assist in the acquisition of terrain data.

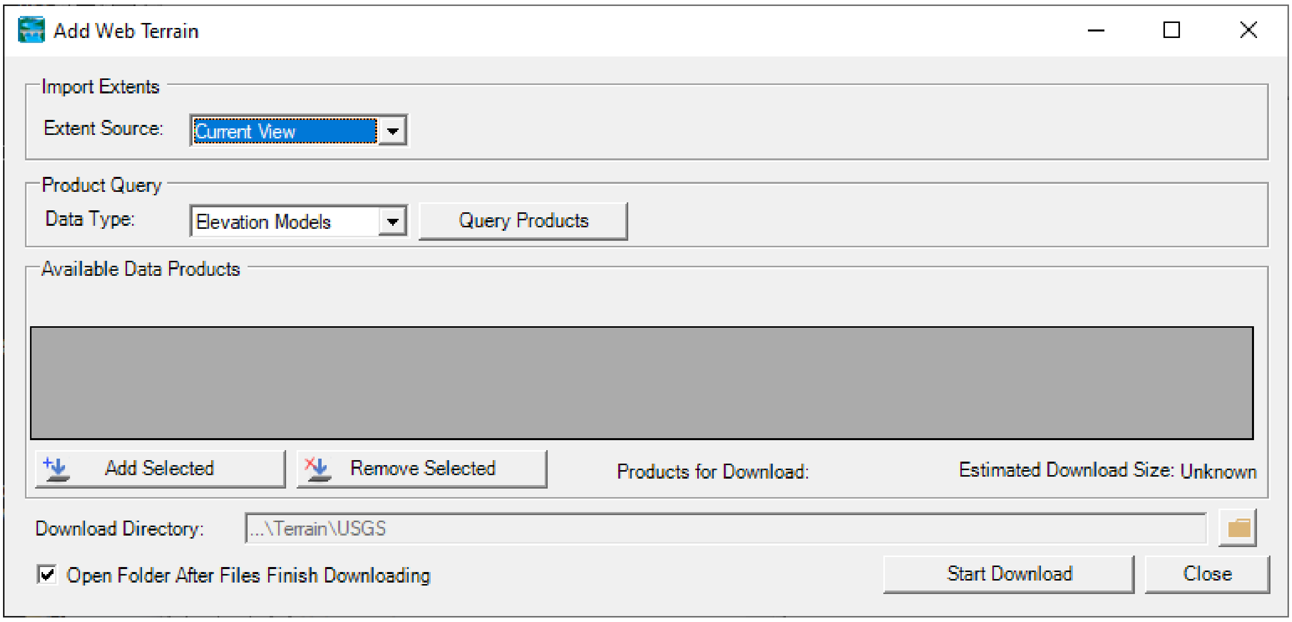
The USGS has made data for the United States available for download through the National Map Viewer[5]. In an effort to simplify the process of utilizing elevation data provided as part of the USGS 3DEP[6] (3D Elevation Program), HEC-RAS Mapper provides the capability to "automatically" download terrain data. While the USGS terrain data may be representative of the ground surface in the overbank areas, it most likely will not accurately represent the ground surface in the river channels. This will be evident due to flat features where water existed during data collection.

To utilize the download tool, you must first have a Projection defined and zoom to the area of interest.

2.1.1 Open RAS Mapper

To access the USGS Terrain download tool, select the **Project | Download Data | USGS Terrain** menu item. The figure shown below will be displayed.

5 https://apps.nationalmap.gov/viewer/
6 https://www.usgs.gov/3d-elevation-program



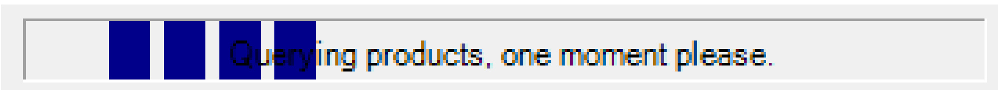
To get and use the data, you will follow four basic steps.

1. Query the USGS Product Database
2. Select the USGS Products of Interest
3. Download the USGS Datasets
4. Create a RAS Terrain.

Query Products

First, you will need to identify the **Import Extent** (the default option in the current view). The extent options include Current View, Geometry, Shapefile, and Manual Entry.

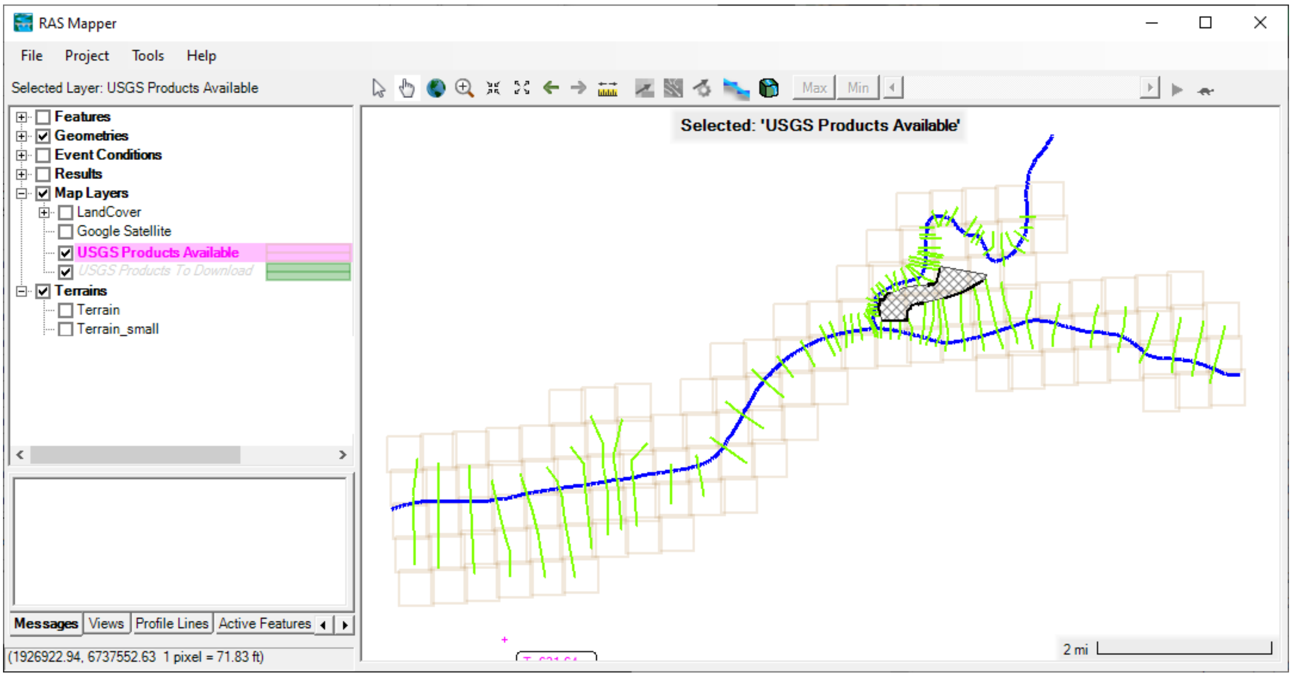
Next, press the **Query Products** button to ping the USGS server to see what data are available. At this time, you can ask for Elevation Model data or Topo Maps (in case you want those as background layers). When you query the USGS database, the interface will provide feedback that it is working, as shown below.



Once the list of available data has been received, the Available Data Products table will be populated as well as showing information about the dataset including Cell Size, Description, Date (Published), File Size, and Web Link to metadata. As shown in the figure below, there are likely to be several elevation data sources for your study area.

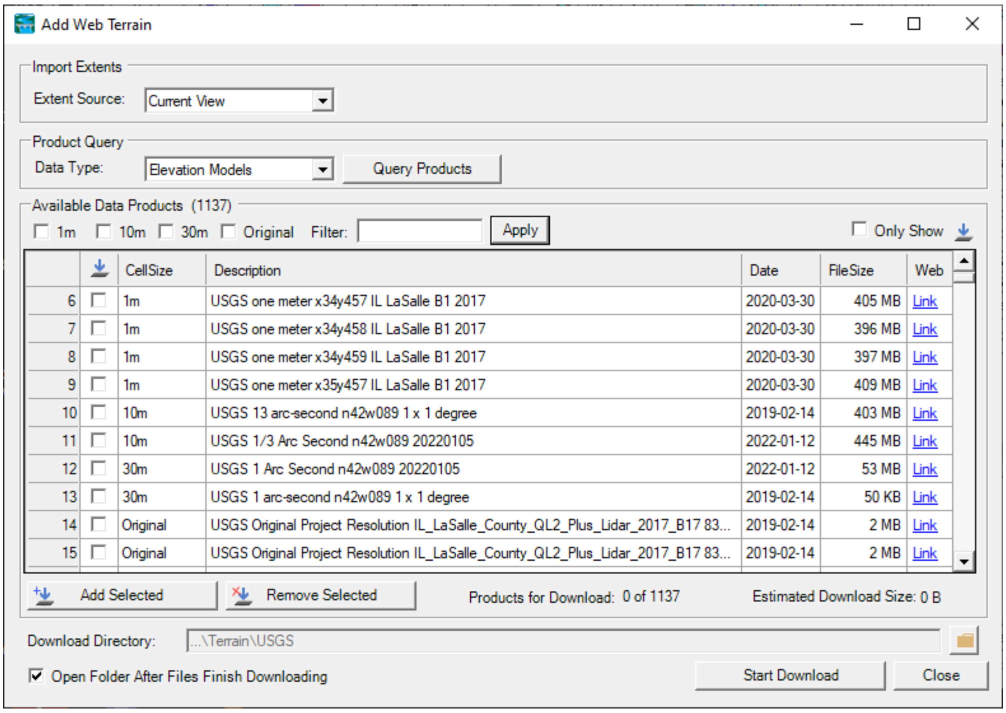
> ⓘ Note that the information about each data set will not necessarily be valid. The data download tool is passing on information scraped from the USGS database.

If you Query Projects using the Geometry for the source extent, the data download tool will automatically restrict the data products returned to just those that intersect the selected geometry. This can be a handy option if you already have an existing HEC-RAS model.

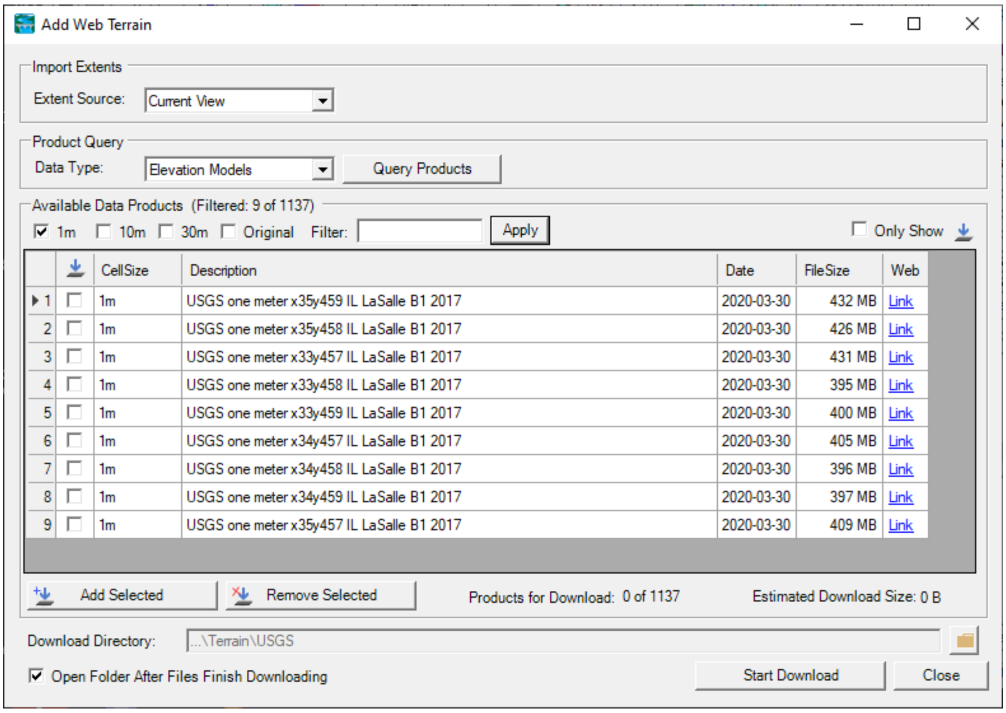


Select Products

To add a terrain dataset to the **Products for Download** list, select the product and click the **Add Selected** button or check the box. This will place a check mark in the "Data to Download" column.

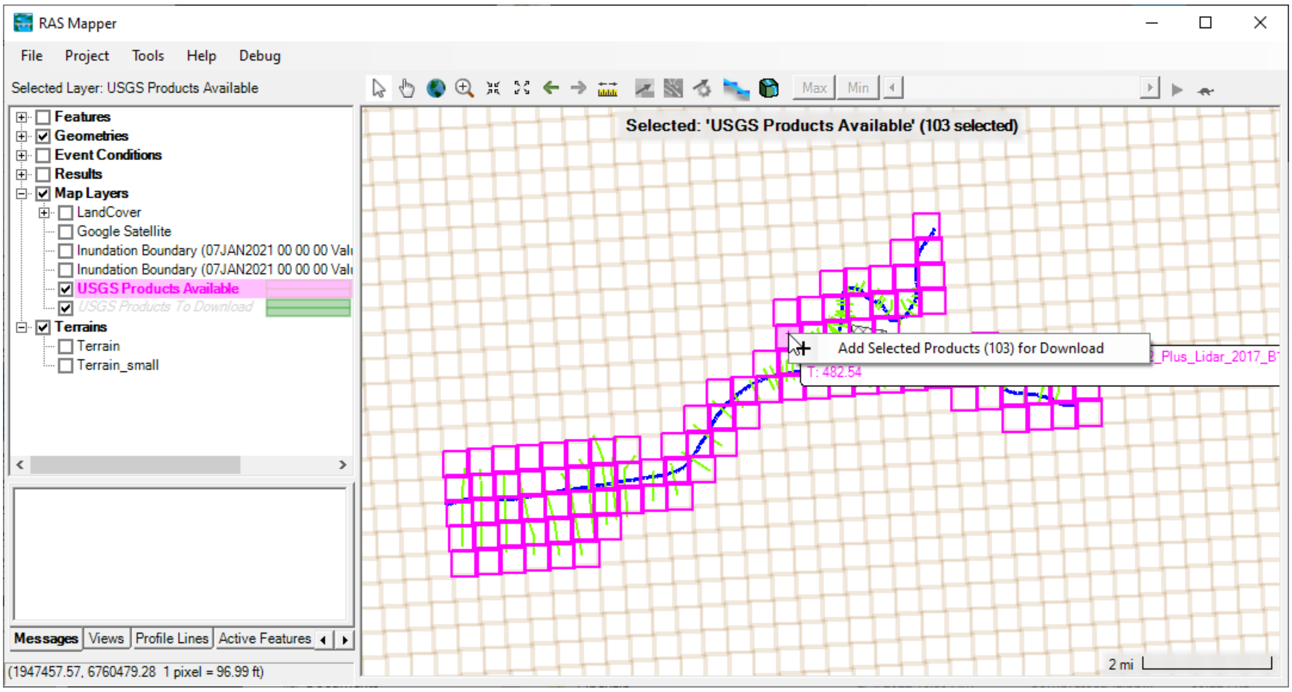


To assist you with identifying the data to download, the data download tool has a **Filter** mechanism. The filter capabilities have some built in filters based on grid resolution (1m, 10m, 30m, Original) or you can enter your own filter. For the example figure below, the data have been filtered for just the one meter data ("1m"). Note, this reduces the number files in the table to 9 (of 1137 ***Available Data Products***).

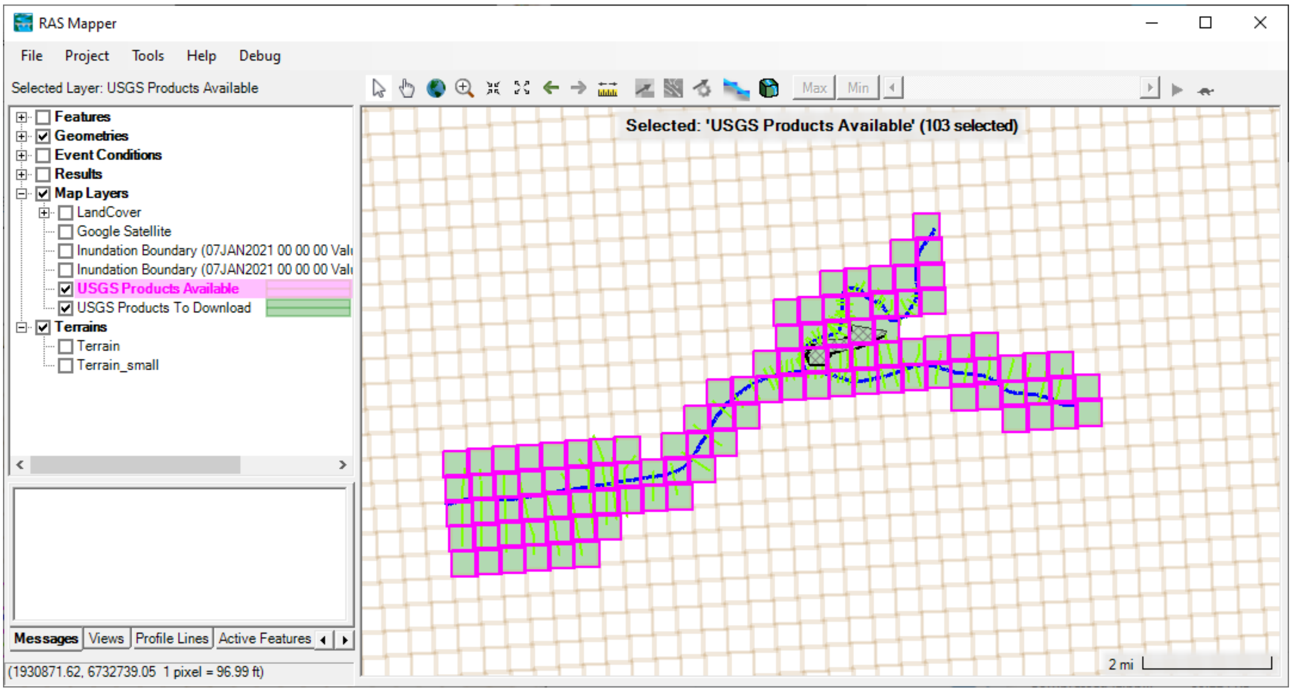


The data download tool also assists you in selecting data by providing an ***interactive*** layer. When the products are queried, 2 layers will be added to RAS Mapper: **USGS Products Available** and **USGS Products to Download**. The USGS Products to Download layer can be used to interactively select datasets. It will also show you what is selected in the Table on the download tool. Using the Selection Tool, in RAS Mapper, you can ***Add*** to the selection by using the **Ctrl** key or ***Remove*** from the selection using the **Ctrl+Shift** key. Once you have selected the terrain dataset, right-click (on the layer name or the features) and choose **Add Selected Products for Download**. **Remove Selected Products for Download** is likewise available.

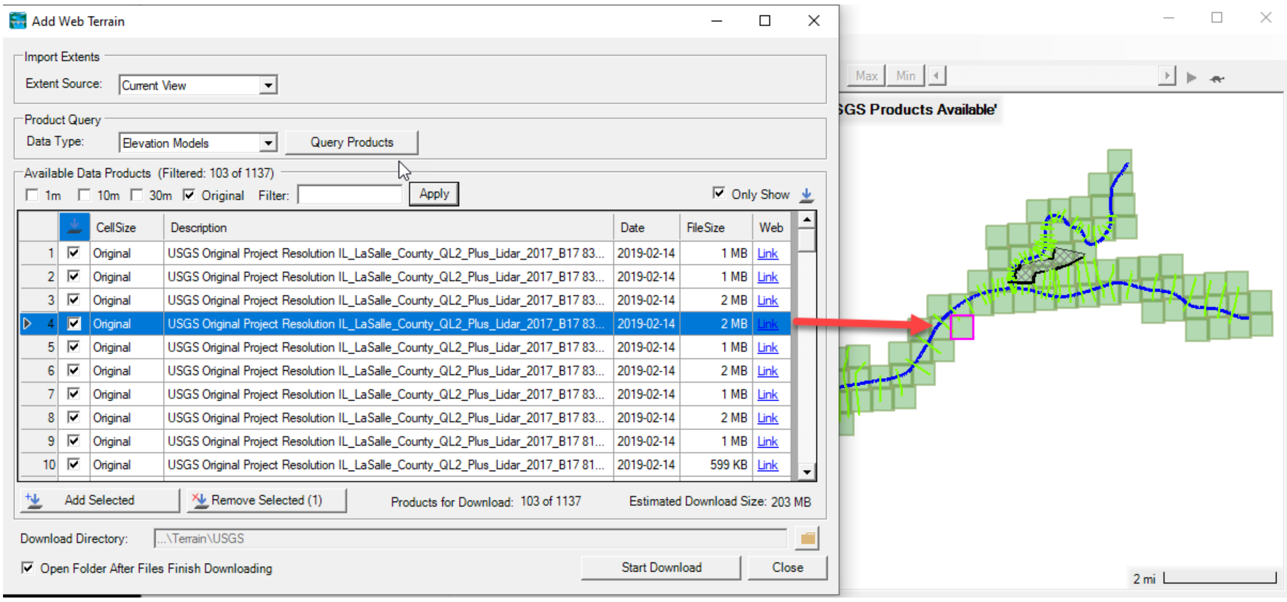
Ctrl + Click -> Adds to the current selection
Ctrl + Shift + Click -> Removes from the current selection



Once a product is selected for download, it will be add to the **USGS Products to Download** layer and the features will be painted using that layer's symbology (green is default).



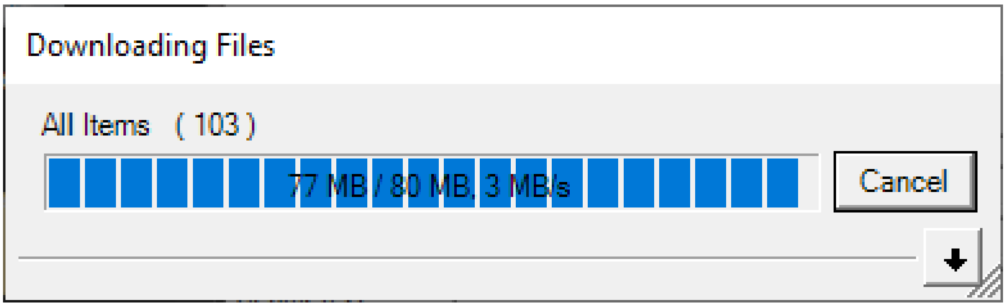
The selected products for download will be reflected in the product list with a check mark. You can verify each product with highlighting a row in the table and the selection in RAS Mapper will update (and vice versa).



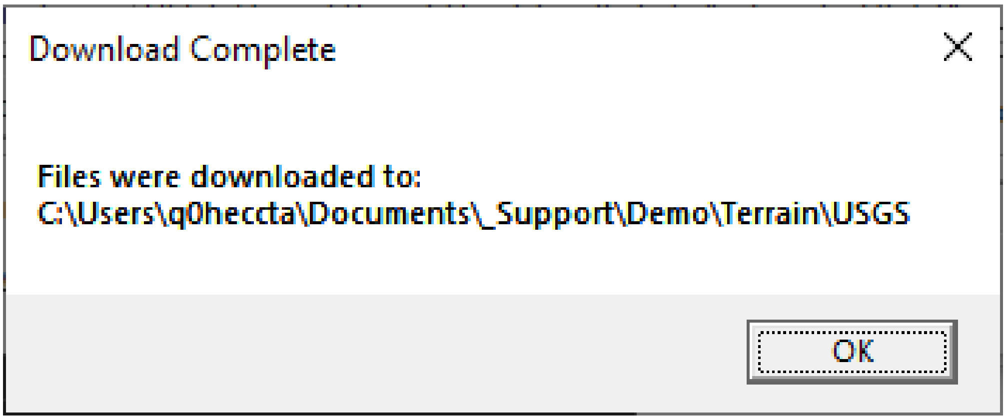
As products are added to the download list, the download tool will provide feedback. The ***Products for Download*** count and the ***Estimated Download Size*** will will be updated to inform you of the selection. (Note, the file size estimate comes from the file's metadata and is often incorrect.)

Download Products

By default, a ***USGS*** directory will be created in the ***Terrain*** folder. Clicking the **Start Download** button with begin the process of downloading the USGS data. The download process will happen asynchronously, so that you can continue using RAS Mapper. As the data download, a status window will appear. If you expand the window, you will get the status for each file.



Once the data have been downloaded, a window will be displayed informing the of the process. A file explorer window will also be open, if the the **Open Folder After Files Finish Downloading** check box is selected.



When you close the terrain download tool, the interactive USGS layers will be removed from RAS Mapper.

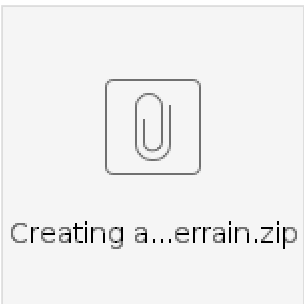
2.2 Creating a RAS Terrain

2.2.1 Objective

In this tutorial, you will learn how to create how to create a RAS Terrain from a raster representing ground surface elevations.

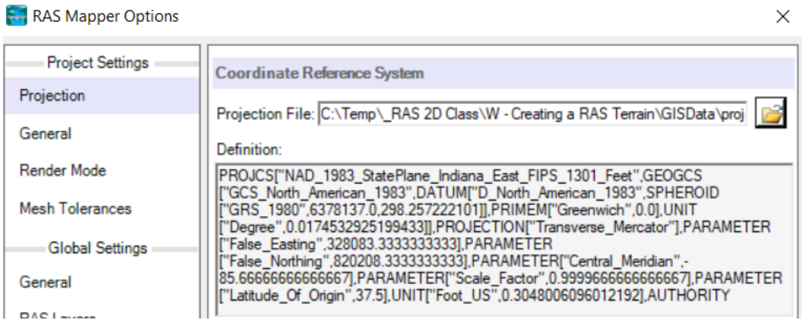
2.2.2 Background

You will be working with a section of the White River at Muncie, IN.

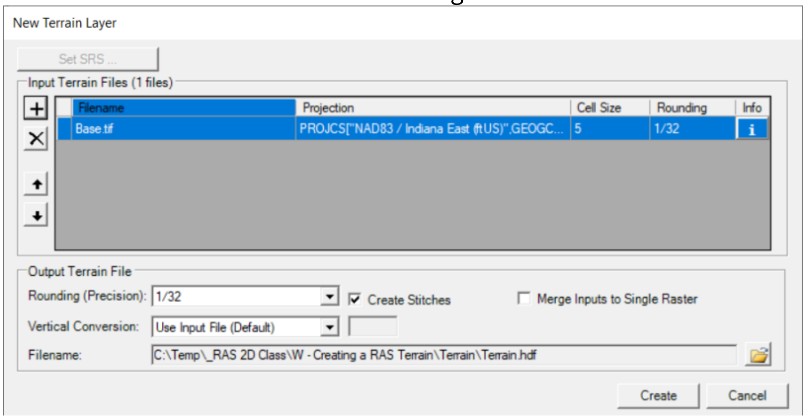


2.2.3 Creating a RAS Terrain

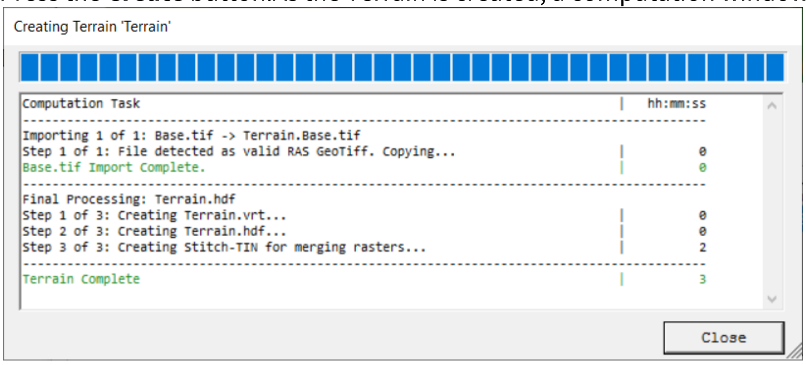
1. **Start HEC-RAS**.
2. Save the project using **File | New Project** and providing a project name.
3. Launch **RAS Mapper**  .
4. Select **Project | Set Projection for Project** and navigate to the "**projection.prj**" provided in the "GISData" folder. This sets the coordinate system for all the data you will view in RAS.



5. Press **OK** to accept the file.
6. Select the **Project | Create New RAS Terrain** menu item (or **right-click Terrains** and choose the **Create a New RAS Terrain** menu item) to import the terrain model.
7. Click the "**+**" button to add files and navigate to the "Terrain" folder. Select the "**base.tif**" file.

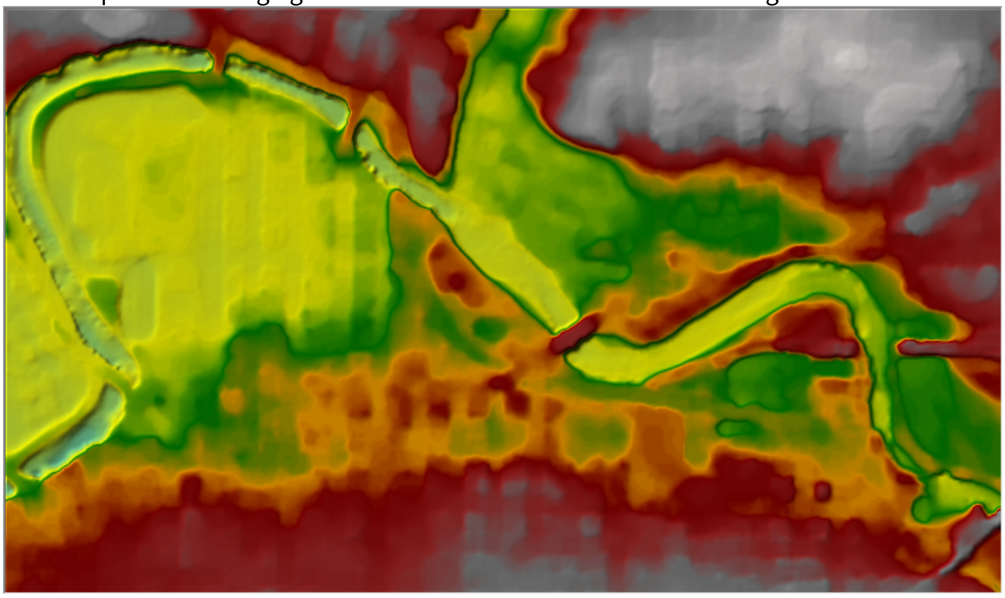


8. Press the **Create** button.As the Terrain is created, a computation window will inform you of progress.



9. **Double-click** on the **Terrain** Layer to access its Properties.
10. Click on the **Plot Hillshade** option.Play with the **Z Factor** to find a value you prefer.
11. Click on the **Plot Contours** option. Play with the **Interval**. Experiment with the option until the terrain looks good to you.

Note the presence of high ground in the middle of the channel at bridge locations.

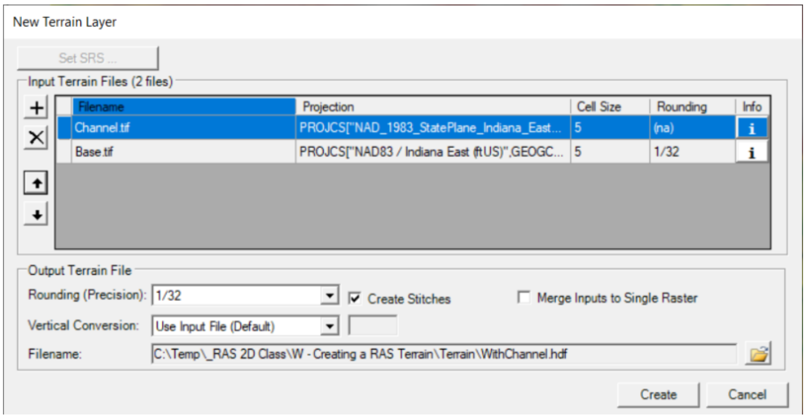


2.2.4 Creating a RAS Terrain from Multiple Datasets

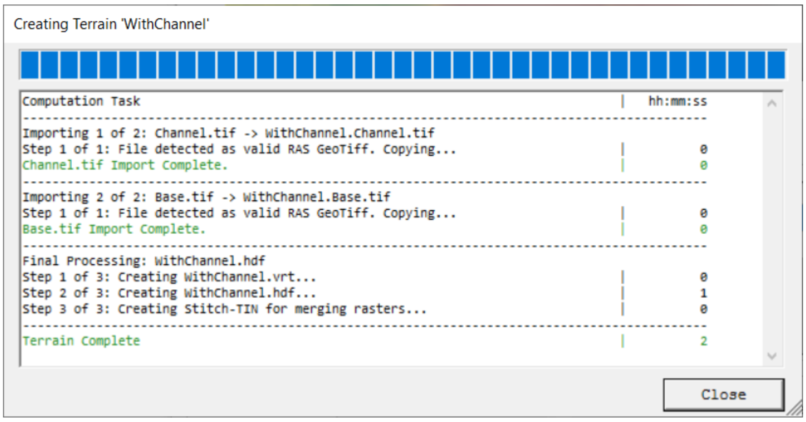
This task will take you through the process of creating a RAS Terrain from 2 different terrain models: channel data and overbank terrain, in order to capture the channel geometry where there is currently high ground at the bridge locations.

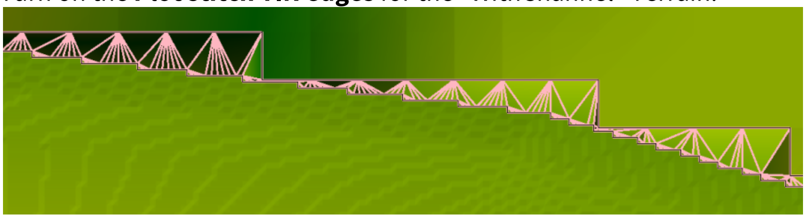
Create Grid of Channel Data

1. Open **RAS Mapper**
2. Select the **Project | Create New RAS Terrain** menu item
3. You will have to add to 2 different files. Make sure that the "priority" is set properly. The top layer should be the channel geometry, followed by the overbank. If you add them in the wrong order, you can reprioritize them by highlighting on of them, and clicking the up or down arrows on the left-hand side of the menu.
   a. Add the "**Channel.tif**"
   b. Add the "**Base.tif**"



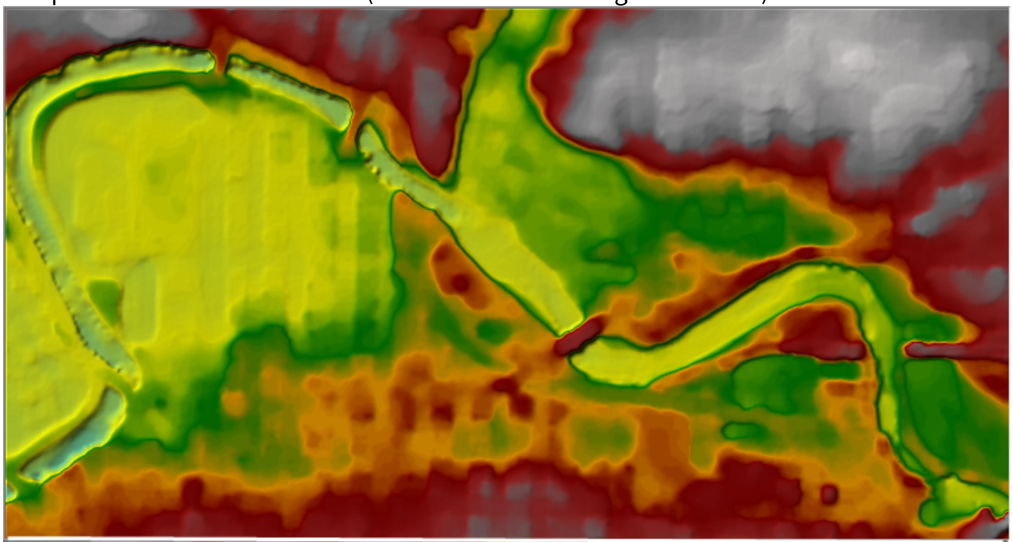
4. Change the output filename to "**WithChannel**" by clicking the open folder icon in the bottom right corner of the menu. Enter the new name, and click **save**.
5. Press the **Create** button.

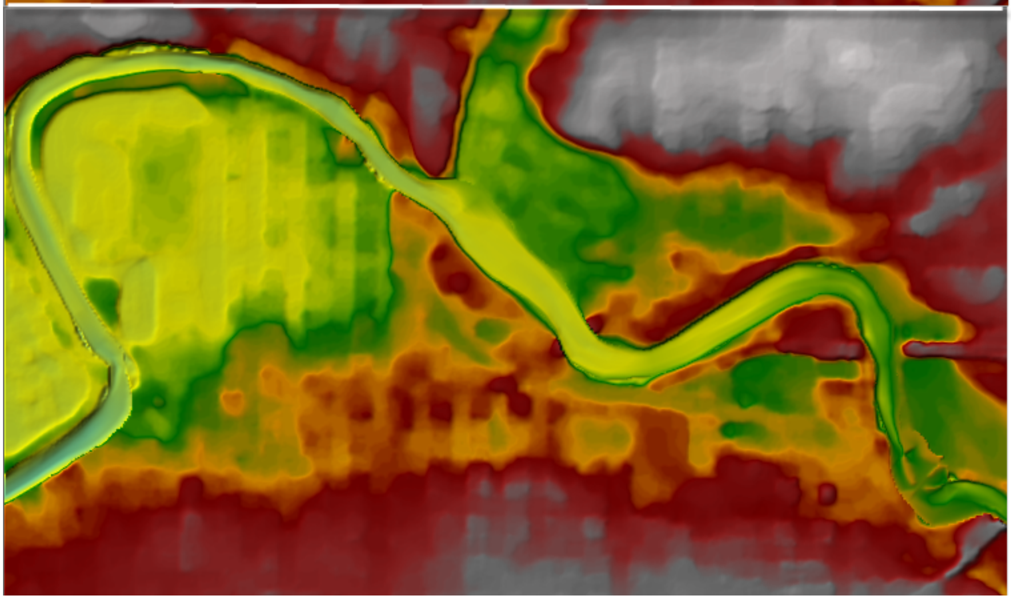
6. 

7. Change the **Layer Properties** for the "WithChannel" Terrain.
8. Turn on the **Plot stitch TIN edges** for the "WithChannel" Terrain.


9. Investigate the stitching – a TIN which is the interpolation between the in-channel data and the overbank data. Compare with the base Terrain model.

10. Compare the two terrain models (with and without bridge elevations).





2.3 Terrain Modification

2.3.1 Objective

In this tutorial, you will learn how to modify terrain models for use in HEC-RAS. You will learn how to create channel data to merge with overbank data from an existing RAS model and how to use the terrain modification tools.

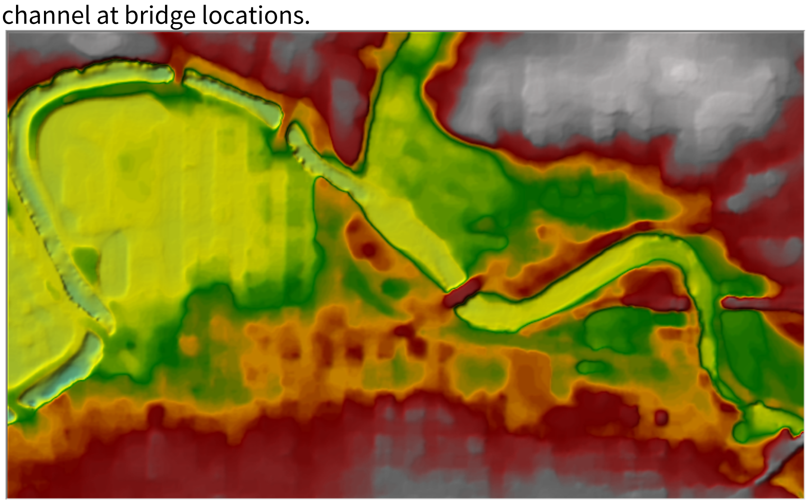
2.3.2 Background

You will be working with terrain data for a section of the White River at Muncie, IN. The data required for the tutorial is provided in the zip file below.

2.3.3 Merging XS Channel Data with Overbank Data

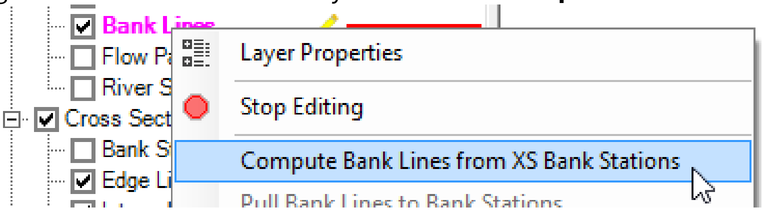
This task will take you through the process of merging channel cross section information with terrain data. This process simulates the situation where you have bathymetric data represented in cross sections to replace overwater elevations gathered using LiDAR.

Create Grid of Channel Data

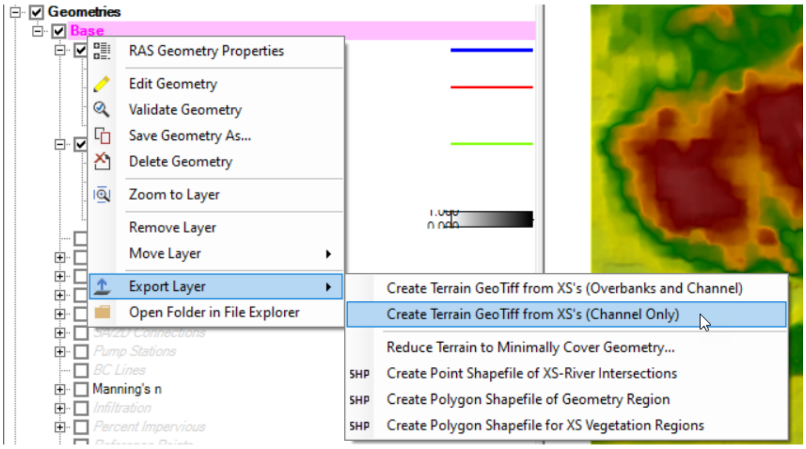
1. Start HEC-RAS and **open** the project titled "**Terrain Modification**".
2. Open **RAS Mapper**
3. **Create a New RAS Terrain** using the "**base.tif**". Note the presence of high ground in the middle of the channel at bridge locations.

4. **Turn on** the **Rivers**, **Bank Lines**, and **Cross Sections** Layers.These are the Layers that are used for creating the Interpolation Surface.


5. If the Bank Lines layer is missing. You can create bank lines from the cross section banks stations.
    a. **Start Editing**

   b. Right-click on the **Bank Lines** Layer and choose **Compute Bank Lines from XS Bank Stations**.



6. Note any problems with the bank lines. Improve the bank lines to do a bettter job representing the channel. For instance, the river line should not intersect the bank lines.
7. Right-click on the **Interpolation Surface** and select **Compute XS Interpolation Surface**. Turn in on. The Interpolation Surface is used in making the results maps and will be used to create the new channel raster data.
8. **Stop Editing**.
9. Right-click on the geometry and choose **Export Layer | Create GeoTIFF from XS's (channel only)**

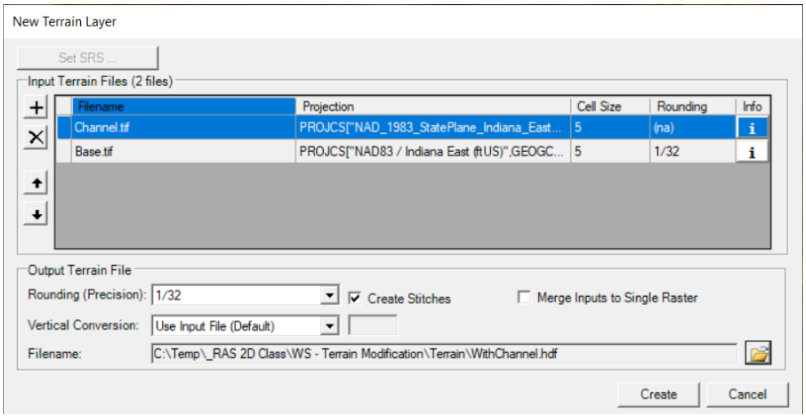


10. Provide a **filename** ("channel") and press **Save**.
11. Enter a raster cell size in the next dialog (this will depend on the size of your channel. **Enter 5** (ft).

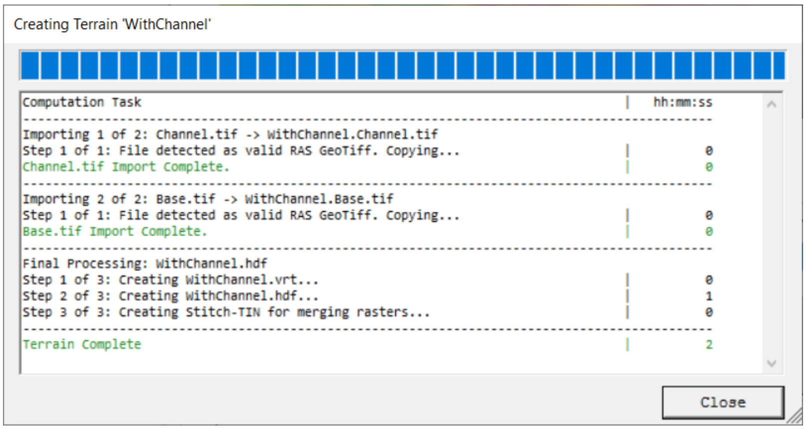
Merge Terrain Data

1. **Create a New RAS Terrain**.
2. You will have to browse to 2 different files. Make sure that the "priority" is set properly.
   a. Add the "**channel.tif**"

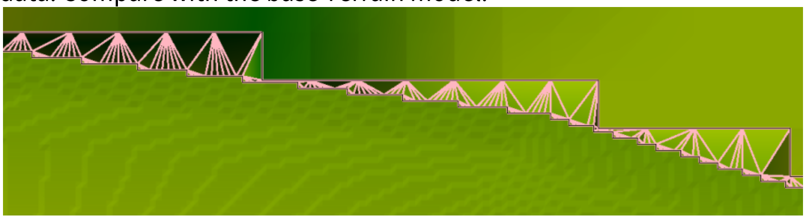
b. Add the "**base.tif**"



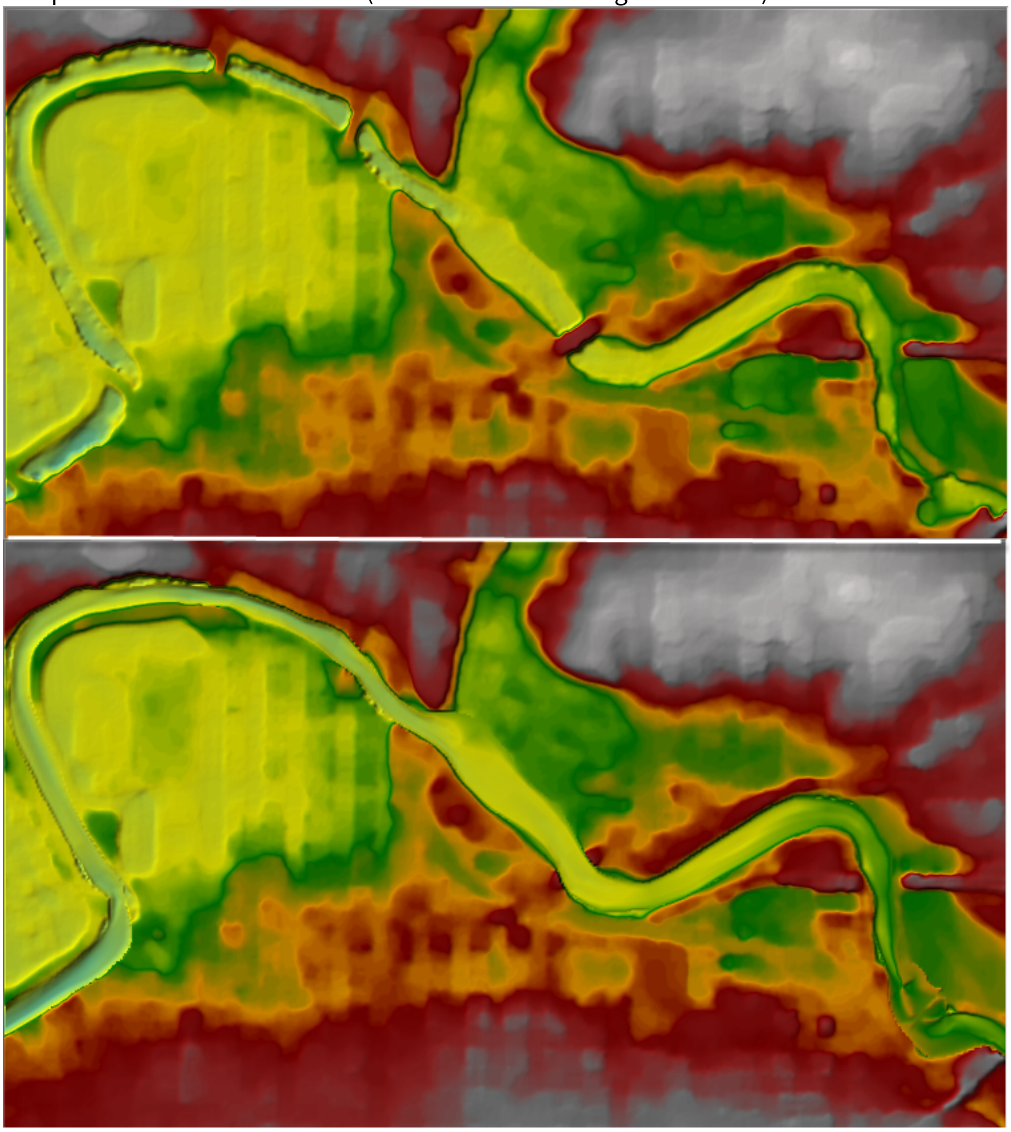
3. Change the output filename to "**WithChannel**".
4. Press the **Create** button.



5. Change the **Layer Properties** for the "WithChannel" Terrain.
6. Turn on the **Plot stitch TIN edges** for the "WithChannel" Terrain.
7. Investigate the stitching – a TIN which is the interpolation between the in-channel data and the overbank data. Compare with the base Terrain model.



8. Compare the two terrain models (with and without bridge elevations).



2.3.4 Channel Modification Tools

This task will take you through the process of cloning a RAS Terrain and using the Channel Modification tools to change the ground surface elevations for modeling.

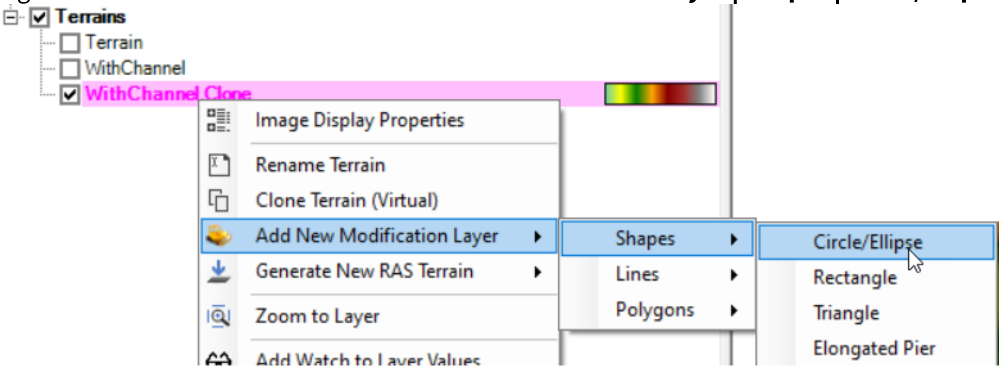
Clone the RAS Terrain

Cloning the RAS Terrain allows you to reuse datasets. You don't have "make a copy", rather we create a new terrain file for the modifications but point to the base terrain.

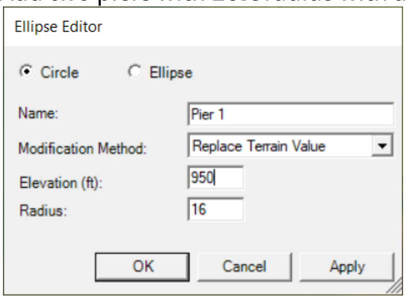
1. **Right-click** on the "**WithChannel**" Terrain and choose "**Clone Terrain**".
2. Provide a name (like "**Clone**") and press **OK**.
3. Turn on the cloned terrain.

Add Piers

1. Zoom into one of the bridge crossings (**View = Simple Pier**).
2. Right-click on the Clone and choose **Add New Modification Layer | Shapes | Circle/Ellipse**.

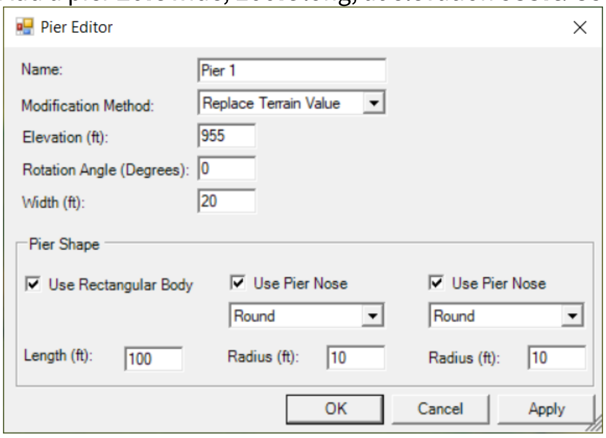


3. Add a name for the layer "**Simple Piers**".
4. Add two piers with **16ft** radius with a top elevation of **950ft**.



Add Elongated Pier

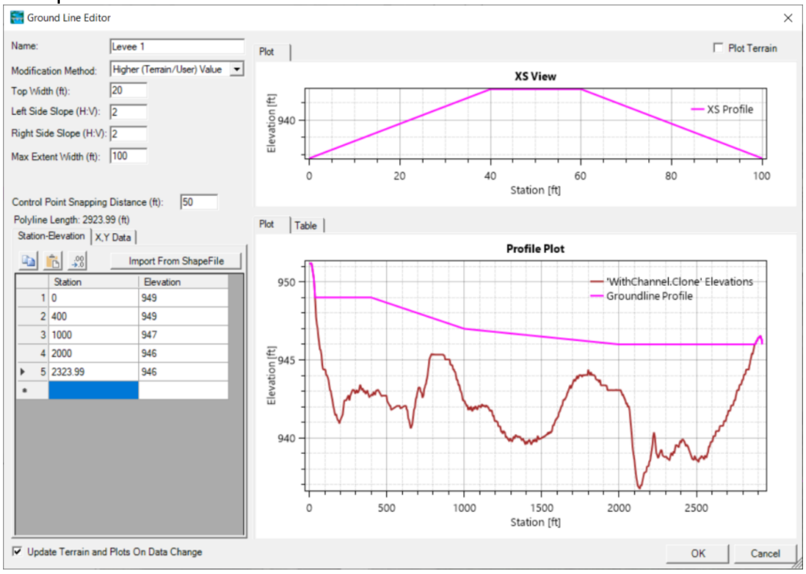
1. Zoom into one of the bridge crossings (**View = Elongated Pier**) .
2. Right-click on the Clone and choose **Add New Modification Layer | Shapes | Elongated Pier**.
3. Add a name for the layer "**Piers**".
4. Add a pier **20ft** wide, **100ft** long, at elevation **955ft**. Use round nose with a **10ft** radius.



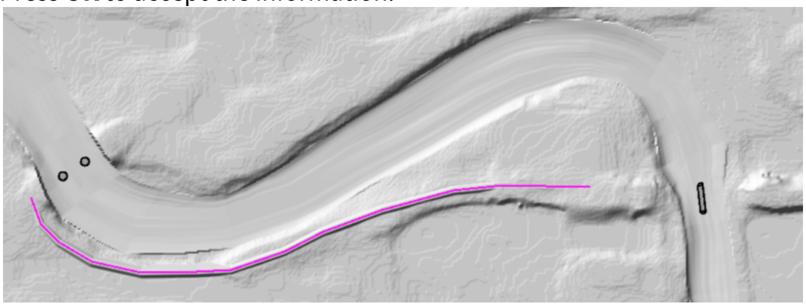
5. Use the edit tool to **rotate** the pier in line with flow.

Add a Levee

1. Zoom into one of the levee locations between the two bridges crossings (**View = Levee**).
2. Right-click on the Clone and choose **Add New Modification Layer | Lines | High Ground**.
3. Create a levee/floodwall alignment from the upper bridge to the lower bridge.
4. Set up the levee information as show below.



5. Press **OK** to accept the information.

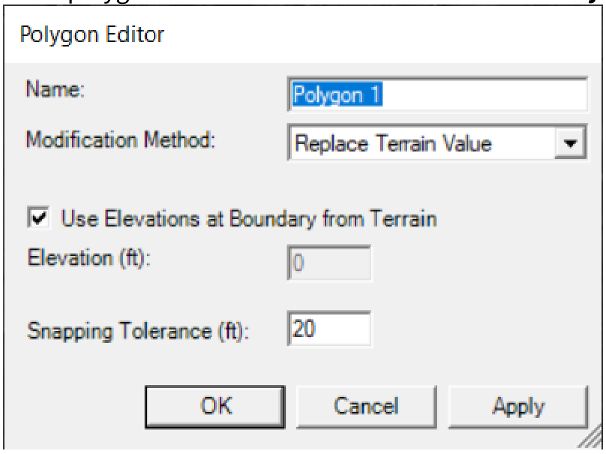


6. Evaluate how the terrain has changed.

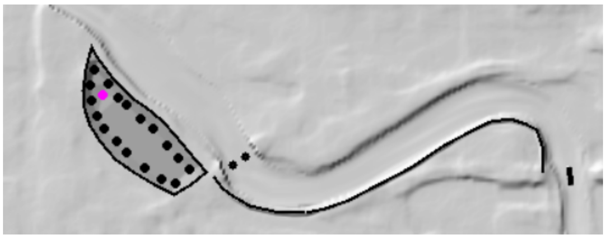
Add a Detention Pond

1. Zoom into one of the levee locations between the two bridges crossings (**View = Wetland**).
2. Right-click on the Clone and choose **Add New Modification Layer | Polygons | Multipoint**.
3. Provide a name like "Wetland"

4. Add a polygon – use the "**Use Elevations at Boundary from Terrain**" option.



5. Note what the terrain model looks like.
6. Now add Elevation Control Points to lower the terrain inside the polygon.
    a. Expand the Wetland modification to see the "Control Points" layer.
    b. Add control points and enter elevations around **930ft** with a low spot on the downstream end of **925ft**.

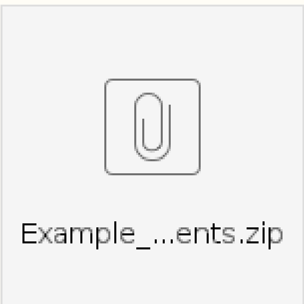


7. Evaluate the terrain model.

2.4 Floodway Encroachment Analysis - 1D Unsteady Flow

This tutorial demonstrates the approach to performing Floodway Encroachment Analysis for a simple 1D HEC-RAS example application.

> This is a new feature in HEC-RAS Version 6.4.

> **Project Files**
>
> 
>

2.4.1 Overview

HEC-RAS is the primary river hydraulics software used to perform Floodway Encroachment Analysis for FEMA. Floodway encroachment analysis is used to determine guidelines for allowable development withing the floodplain fringe. FEMA guidance dictates that the model analysis be performed using the 100-yr floodplain and limits development within the the floodplain based on the impact to the computed water surface profile. Historically, this capability in HEC-RAS was limited to Steady Flow analysis. Version 6.4 introduces the ability to perform this analysis with Unsteady Flow.

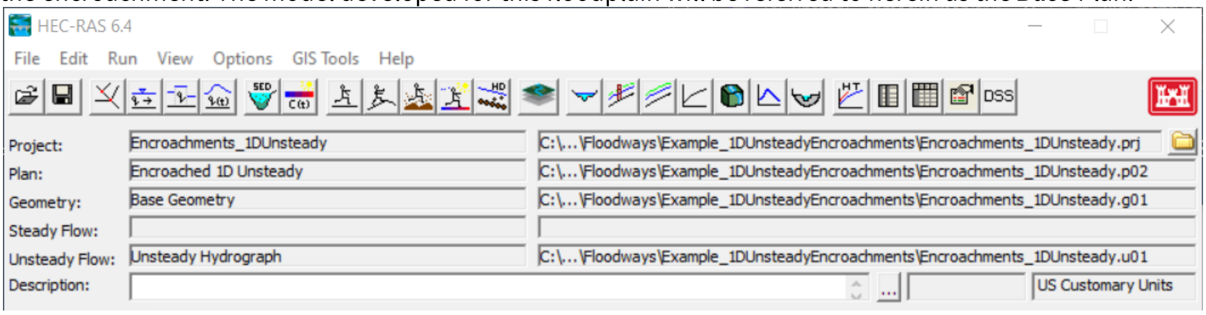
This example utilizes a simple 1D model developed for the Merced River in the Yosemite Valley. Terrain data used in this model was downloaded from the USGS NED and has been resampled. This model is not intended to be used to make floodplain management or engineering decisions, but simply to demonstrate the procedure for performing an encroachment analysis on a 1D model.

This example will take you through the procedure outlined below.

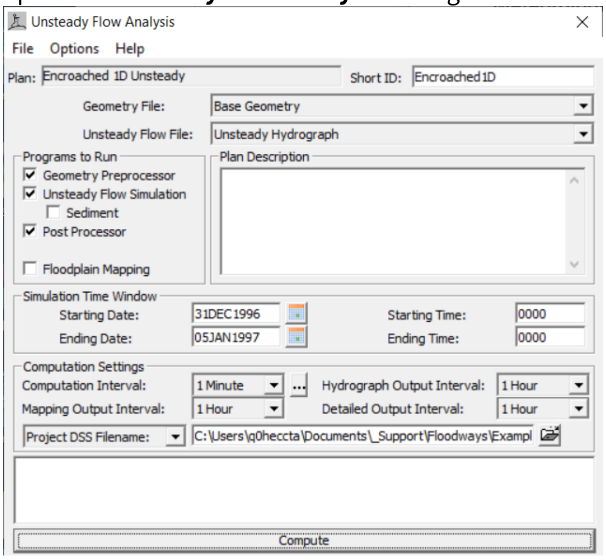
- Create a Calibrated model
- Save the Base plan to an Encroached plan
- Enter data into the Encroachment Table
- Perform the Encroached run
- Analyze results
- Refine the encroachment data
- Perform final model simulation using Method 1 or Encroachment Regions

2.4.2 Steps

1. **Open the HEC-RAS project "Encroachments_1Dunsteady"**
   Prior to beginning a Floodway Encroachment Analysis, the river hydraulics model should be calibrated to and validated with observed data. The downstream boundary condition for the model should utilize the Normal Depth boundary condition as it will allow for a changed stage as the flow hydrograph changes due to the encroachment. The model developed for this floodplain will be referred to herein as the ***Base*** Plan.



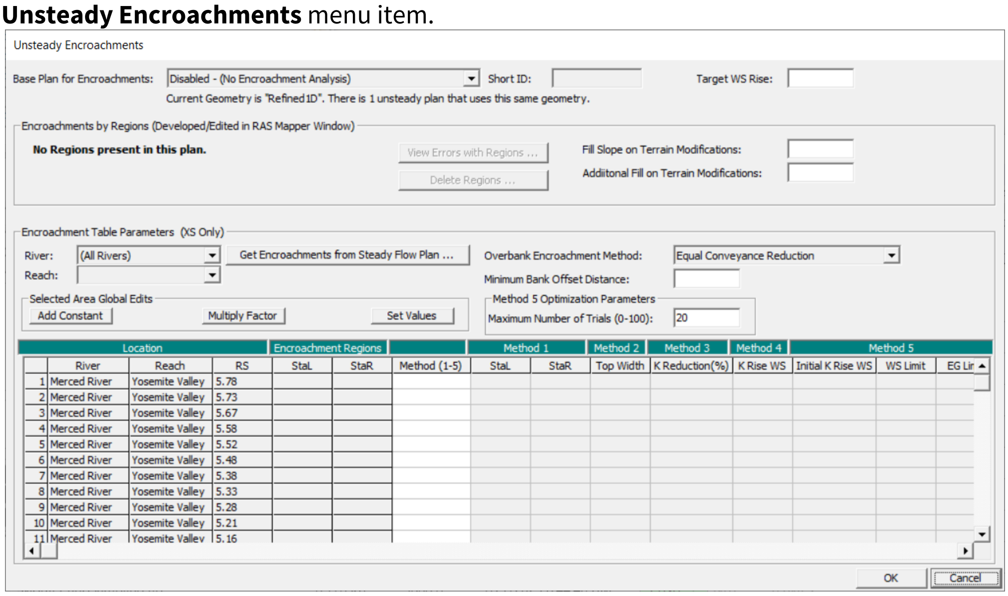
2. Open the **Unsteady Flow Analysis** Dialog



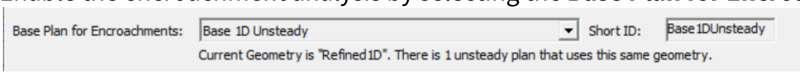
3. **Compute** the Base plan.
4. Copy the Base plan using the **File | Save Plan As** menu item on the **Unsteady Flow Analysis** dialog. Provide a new name for the encroached plan and hit okay, in the dialog. Provide a ShortID for the new plan.

> ⓘ The Encroached plan must utilize the identical geometry as the Base plan.

5. Access the Unsteady Flow Encroachment Table from the **Unsteady Flow Analysis** dialog **Options | Unsteady Encroachments** menu item.



6. Enable the encroachment analysis by selecting the **Base Plan for Encroachments**.



7. Set the **Target WS Rise** to **1.0**ft.

The Target WS Rise is the maximum allowable change to the computed water surface. The National standard is 1.0ft, but may be less than that in based on State, County, or Community regulations. For locations that follow a "no rise" that is equivalent to a Target WS Rise of 0ft and indicates that the computed water surface elevation should not be impacted by the proposed floodplain encroachment.

8. Set the **Fill Slope on Terrain Modifications** to **0.001**.
9. Set the **Minimum Bank Offset Distance** to **20**ft.

   Minimum Bank Offset Distance: 20

   The bank offset will provide a buffer to to the channel by not allowing encroachment near the river banks.
10. Set the Encroachment table to utilize **Method 5**
    a. **Initial K Rise WS** = **0.7** ft
    b. **WS Limit** = **1.0** ft
    c. **EG Limit** = **1.0** ft

	Location			Encroachment Regions			Method 1		Method 2	Method 3	Method 4	Method 5		
	River	Reach	RS	StaL	StaR	Method (1-5)	StaL	StaR	Top Width	K Reduction(%)	K Rise WS	Initial K Rise WS	WS Limit	EG Limit
1	Merced River	Yosemite Valley	5.78			5						0.7	1.0	1.0
2	Merced River	Yosemite Valley	5.73			5						0.7	1.0	1.0
3	Merced River	Yosemite Valley	5.67			5						0.7	1.0	1.0
4	Merced River	Yosemite Valley	5.58			5						0.7	1.0	1.0
5	Merced River	Yosemite Valley	5.52			5						0.7	1.0	1.0
6	Merced River	Yosemite Valley	5.48			5						0.7	1.0	1.0

11. Set the **Maximum Number of Trials** to **4**.

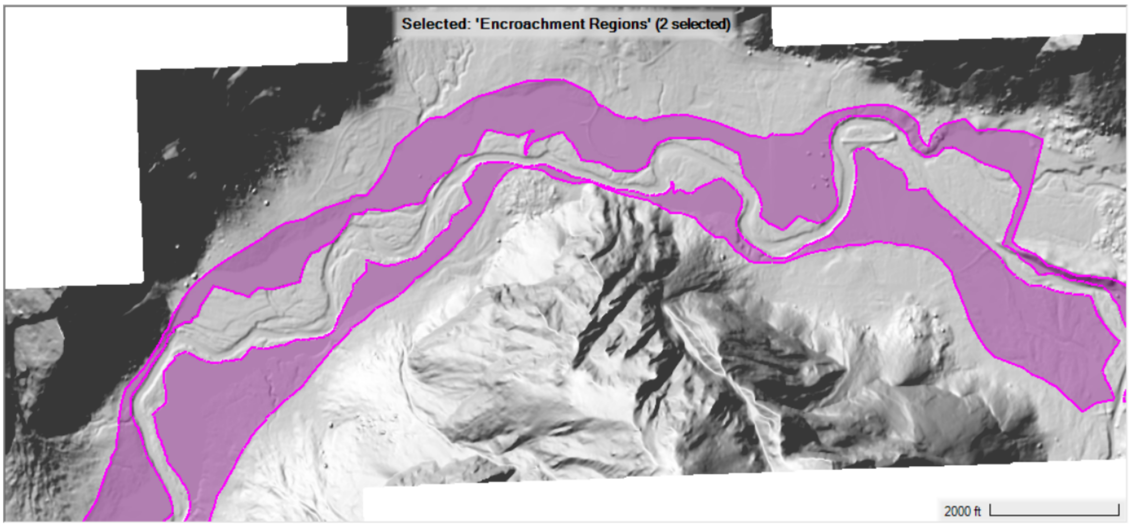
    Maximum Number of Trials (0-100): 4

12. Press **OK** to save the encroachment data.
13. **Compute** the Encroached plan.
    As the plan runs, you will be provided status on the the Trials.

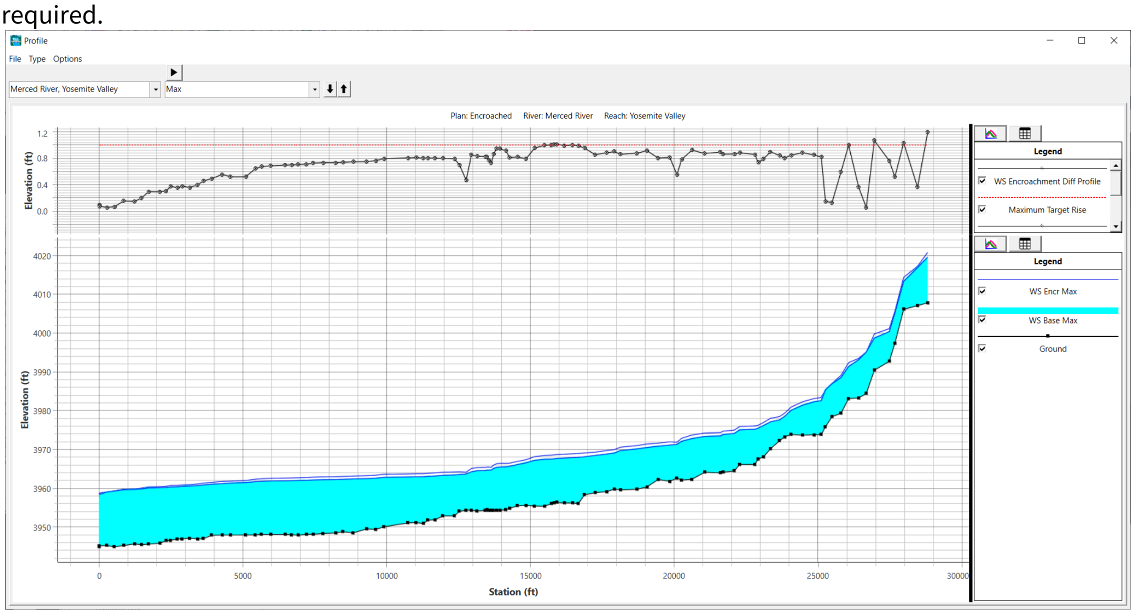
    Unsteady Input Summary:
    1D Unsteady Finite Difference Numerical Solution
    1D Unsteady Encroachments.

    Unsteady Encroachment Trial # 1
    Unsteady Encroachment Trial # 2
    Unsteady Encroachment Trial # 3
    Unsteady Encroachment Trial # 4
14. **Open RAS Mapper**
15. Utilize the capabilities in RAS Mapper to **evaluate the encroachment results** through profile Plots, Tables and geospatial Mapping.
    a. **Encroachment Regions**
       Encroachment Regions are automatically built for the plan Results and give you a visualization of the

Method 5 results.



b. **Profile Plot**
The Profile Plot is accessed by choosing the **Encroachment Regions** Layer **Plot Encroachment Results Profile** menu item. The Profile Plot shows a comparison of the Base and Encroached water surface profile as well as the difference between the two plots. Further, the Target WS Rise is shown along with the WS difference plot, indicating where areas along the river where refinement is required.



c. **Encroachment Table**
The Encroachment Table is accessed by choosing the **Encroachment Regions** Layer **Show Encroachment Table Results** menu item. There are many variables available from the Encroachment Results Table, including the Left and Right Encroachment Stations, Base and Encroached water surface elevations, and difference in water surface elevations. The table is

customizable to see the variables of interest.

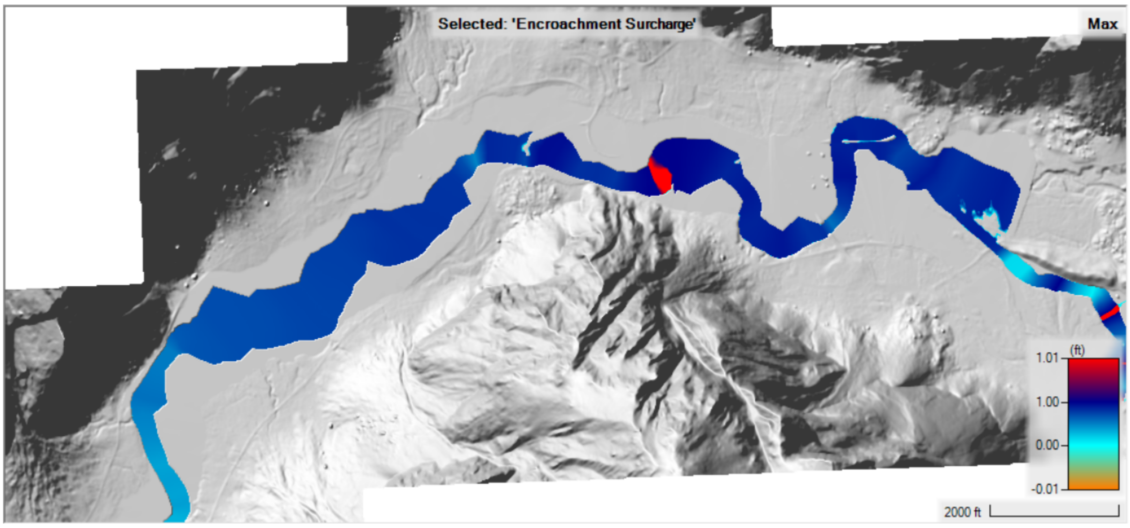
Encroachment Results Table

River: (All Rivers) Reach: (All Reaches) Select Columns

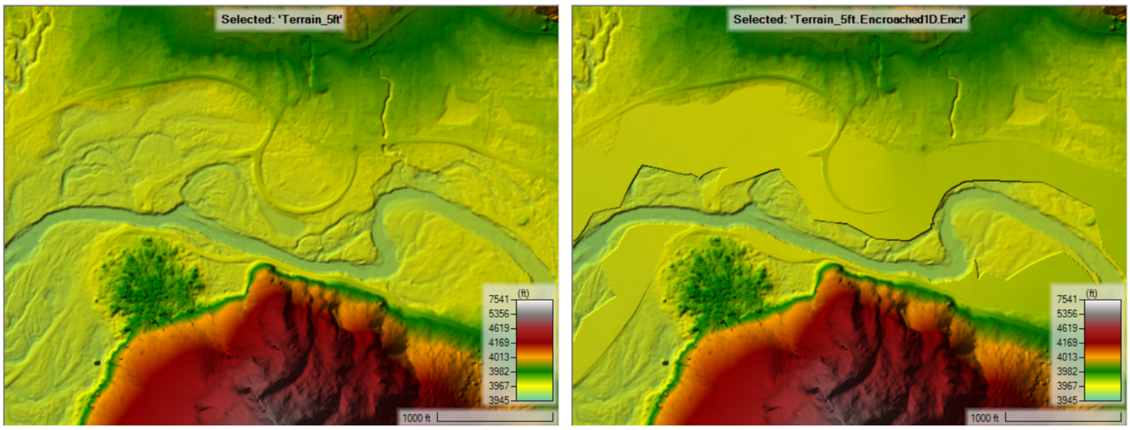
	River	Reach	River Sta	Left Station (ft)	Right Station (ft)	Base WSE (ft)	Encroached WSE (ft)	Delta WSE (Encr - Base) (ft)
1	Merced River	Yosemite Valley	5.78	527.5	669.8	4019.61	4020.86	1.25
2	Merced River	Yosemite Valley	5.73	742.1	903	4016.85	4017.31	0.46
3	Merced River	Yosemite Valley	5.67	808.5	1271.6	4013.32	4014.59	1.27
4	Merced River	Yosemite Valley	5.58	1220.089	1411.6	4005.13	4005.74	0.61
5	Merced River	Yosemite Valley	5.52	1267.6	1527.545	4000.37	4001.15	0.78
6	Merced River	Yosemite Valley	5.48	1359.3	1708.75	3998.7	3999.84	1.14
7	Merced River	Yosemite Valley	5.38	1516.841	1704.35	3995.1	3995.14	0.04

Close

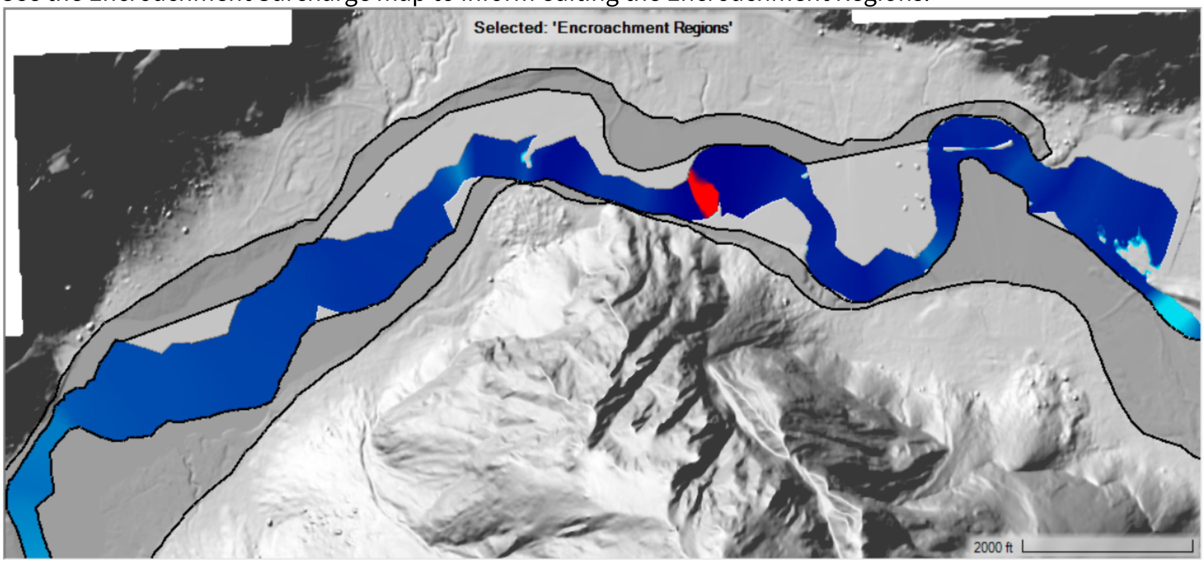
d. **Encroachment Surcharge**
The Encroachment Surcharge map is automatically added to the Encroached simulation results. The surface map allows you to quickly spatially identify locations where the water surface elevations changes more than the allowable (as specified by the Target WS Rise). Analysis of the Surcharge map while editing the Encroachment Regions will allow the user to improve the floodway encroachment boundaries.



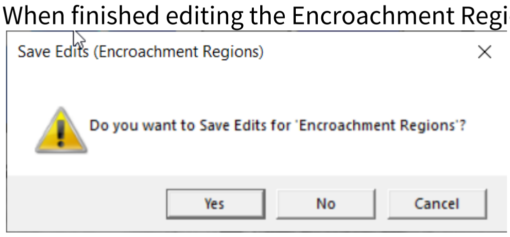
e. **Terrain**
A new Terrain (*Terrain_5ft.Encroached1D.Encr*) was created and associated with the Encroached plan during the simulation. This Terrain is Clone of the terrain used for the Base plan; however, a Terrain Modification was added based on the determination of the encroachment boundary and the Target WS Rise. This new Terrain shows you what would be filled in, should the floodway fringe be build out.



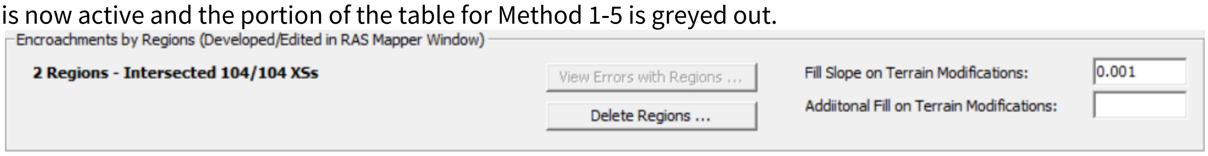
16. In RAS Mapper, expand the Plans group, Encroached plan, and show the Encroachment Regions
17. **Start Editing** the Encroachment Regions
18. Right-Click on the **Encroachment Regions** layer and choose the **Generate Floodway Encroachment Polygons | From 1D Encroachments** menu item and select the Encroached plan.
    This will create Encroachment Regions based on the simulation results (as shown in the Results/Plan layer).
19. Use the Encroachment Surcharge map to inform editing the Encroachment Regions.



20. When finished editing the Encroachment Regions, **Stop Editing**.

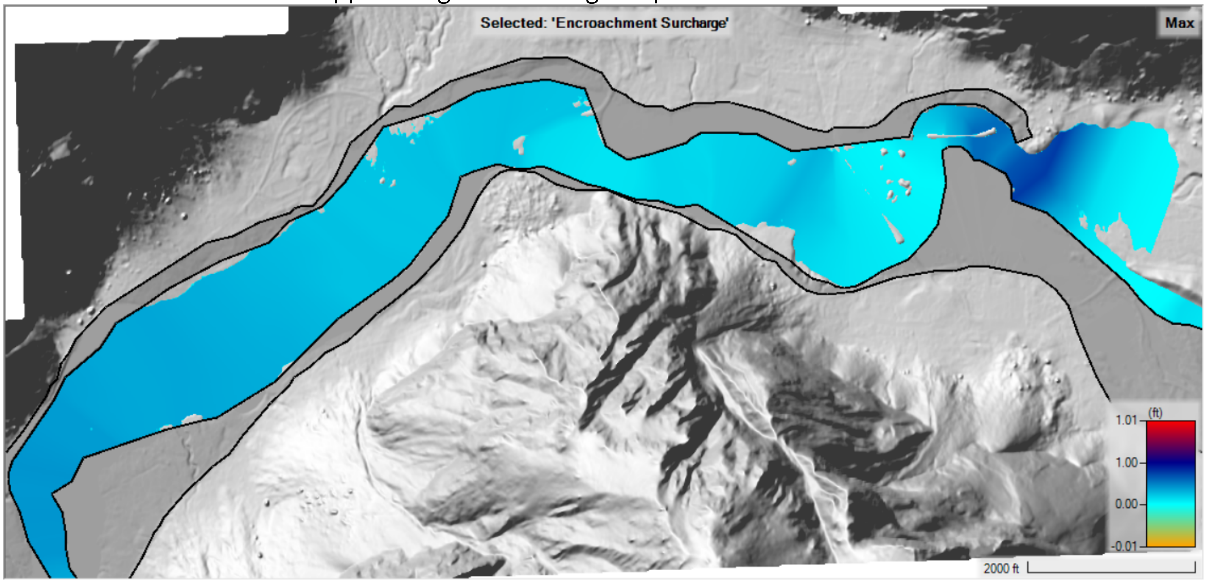


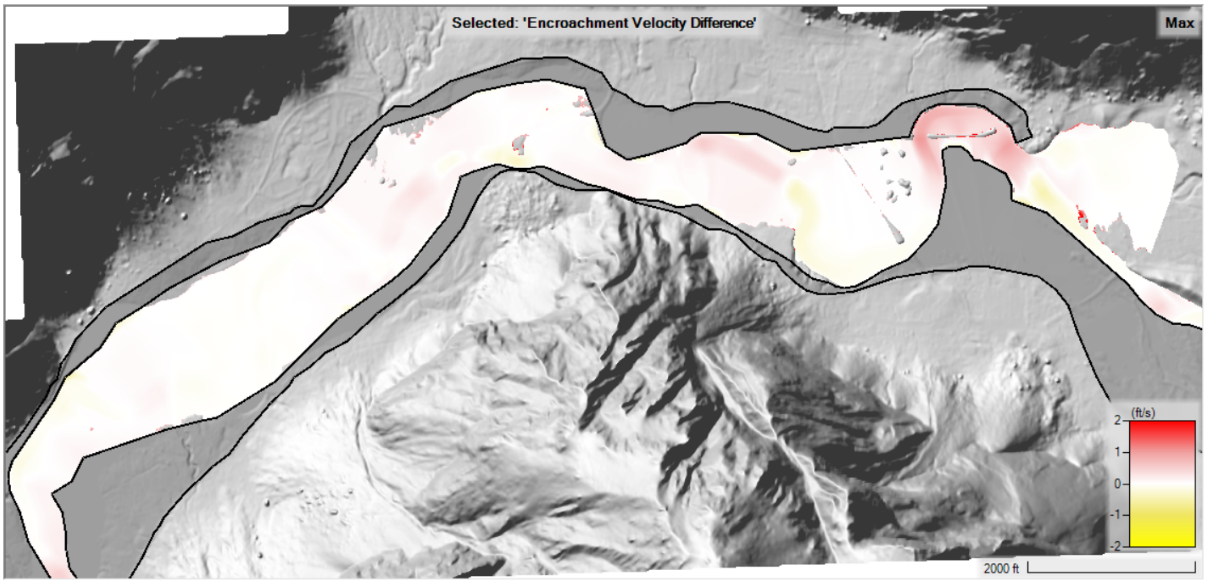
21. **Open** the **Unsteady Encroachment data editor**. Make note that the options for the Encroachment Regions is now active and the portion of the table for Method 1-5 is greyed out.



22. **Compute** the Encroachment plan.

23. The previous RAS Terrain will be replaced with a new RAS Terrain that reflects the Encroachment Regions
24. Evaluate the results in RAS Mapper using the Surcharge map.





25. Continue to refine the encroachment polygons in RAS Mapper as desired and re-computing the floodway analysis.

2.5 Flow Hydrograph Optimization

This tutorial demonstrates the approach to using the Flow Hydrograph Optimization capability for a simple HEC-RAS 2D model example application.

ⓘ This is a new feature in HEC-RAS Version 6.4.

Example_F...ation.zip

2.5.1 Overview

The flow hydrograph optimization capability in HEC-RAS is use to scale flow hydrograph boundary conditions to achieve a desired stage or flow at a specified location in the model. This feature can be used for both 1D and 2D unsteady flow modeling by specifying a Reference Location to be used for the optimization.

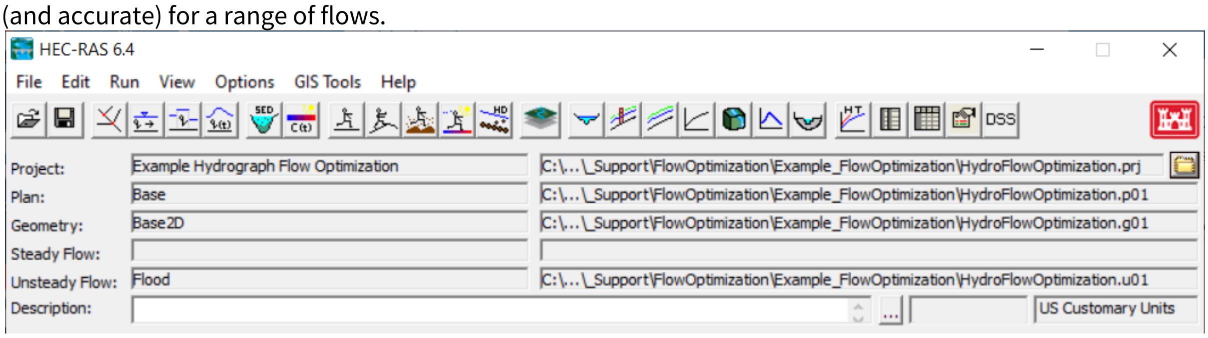
This example utilizes a simple 2D model developed for the Merced River in the Yosemite Valley. Terrain data used in this model was downloaded from the USGS NED and has been resampled. This model is not intended to be used to make floodplain management or engineering decisions, but simply to demonstrate the procedure for utilizing the flow hydrograph optimization capability in HEC-RAS. For this example, the goal will be to determine how much flow begins to flood a parking lot on the north side of the river near the Village Store.

This example will take you through the procedure outlined below.

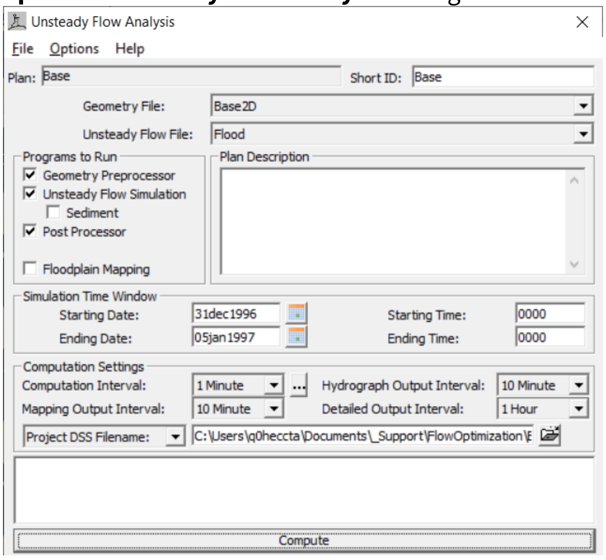
- Identify and create a Reference Location
- Enter Flow Hydrograph Optimization information
- Perform the Flow Hydrograph Optimization simulation
- Review Results

2.5.2 Steps

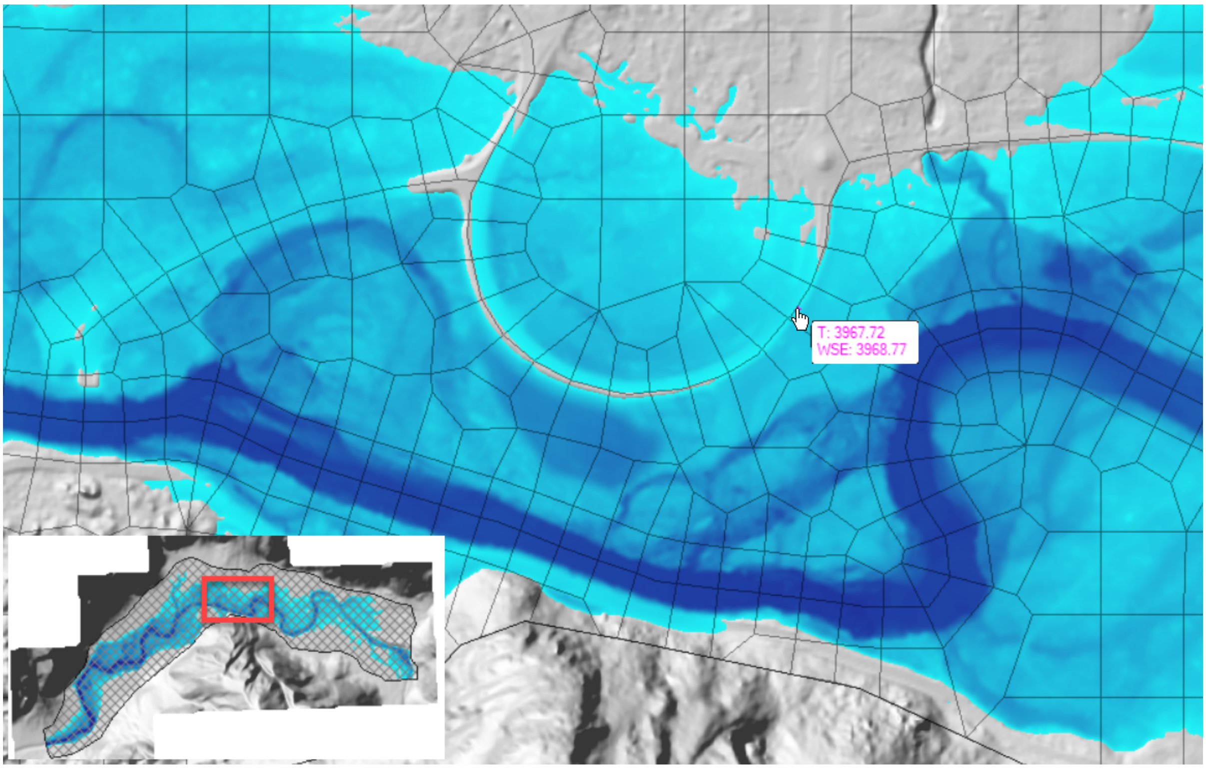
1. Open the HEC-RAS project "**Example Hydrograph Flow Optimization**"
   Prior to utilizing the flow hydrograph optimization capability, you should confirm that the model is stable (and accurate) for a range of flows.



2. **Open** the **Unsteady Flow Analysis** dialog.

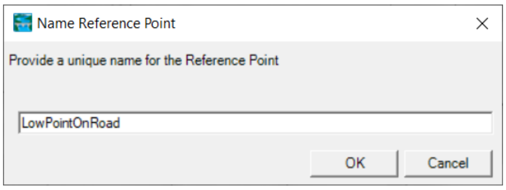


3. **Compute** the Base plan.
4. Open **RAS Mapper** and investigate the model to see what the computed water surface is and the Terrain elevation at the low point along the road that protects the parking lot.
   Once you have identified the location of the low point and recorded the elevation, you are ready to create a reference location.



5. Select the **Base2D** Geometry
6. **Start Editing**
7. Select the **Reference Point** layer
8. Add a the selected reference location using the **Add New Feature** tool.

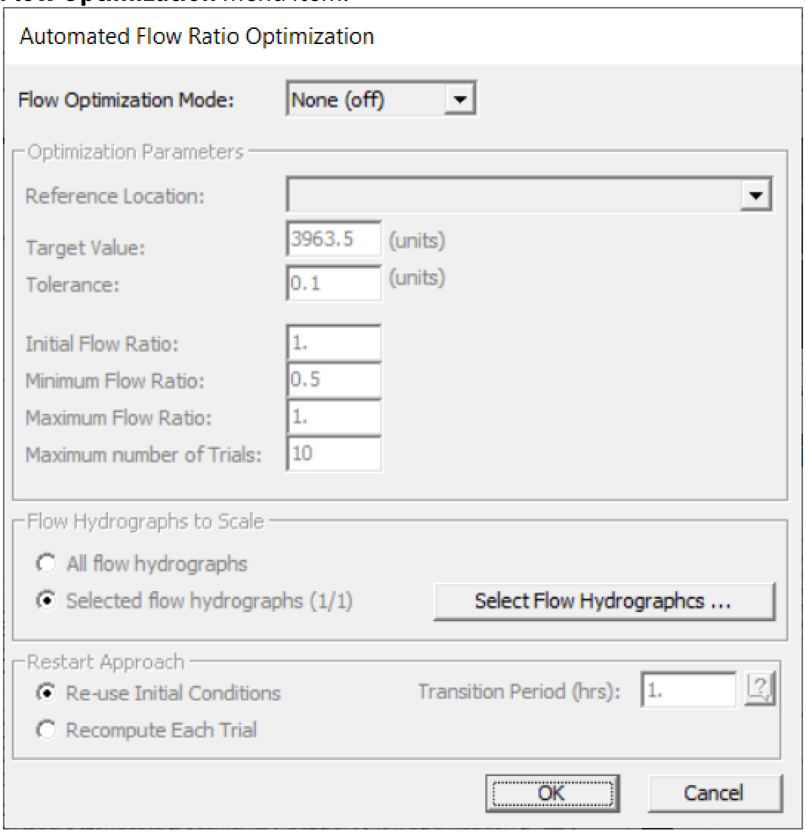
9. Provide a **Name** for the Reference Point



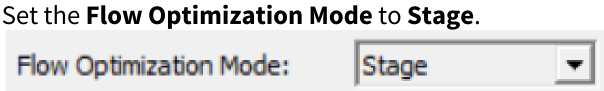
10. **Stop Editing**
11. Copy the Base plan using the **File | Save Plan As** menu item on the **Unsteady Flow Analysis** dialog. Provide a new name for the flow optimization plan and hit okay, in the dialog. Provide a ShortID for the new plan.

> ⓘ It is not required to create a new Plan; however, by creating an additional plan, you will be able to compare results.

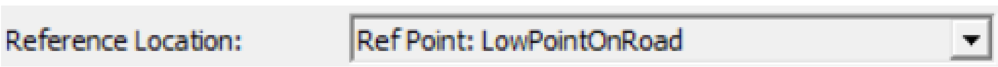
12. Access the flow optimization data entry from the **Unsteady Flow Analysis** dialog **Options | Automated Flow Optimization** menu item.



13. Set the **Flow Optimization Mode** to **Stage**.

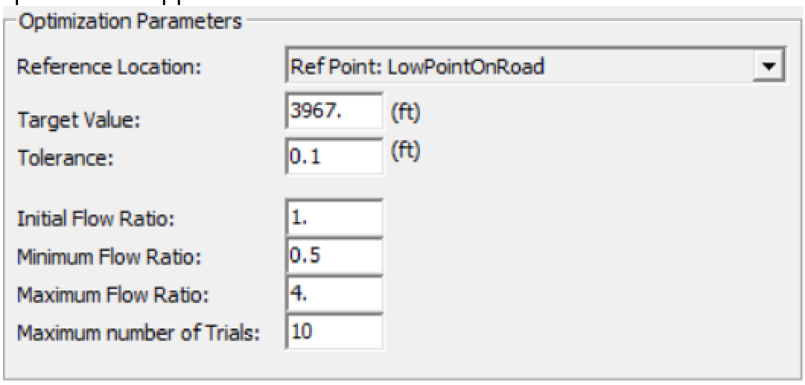


14. Set the **Reference Location** to the **Reference Point**.



15. Set the **Target Value** based the low elevation of the Terrain.
16. Select the remainder of the Optimization Parameters.
The remainder of the parameters can be set as desired, depending on how you want to limit the

optimization approach and number of simulations.



17. In this example, we only have one hydrograph to scale, so you can use the **All** option (or select the hydrograph).
18. Leave the Restart Approach to the default method.
19. Press **OK** to save the optimization information.
20. The **Unsteady Flow Analysis** dialog will now show a quick link that **Automatic Flow Ratio Optimization** has been enabled.

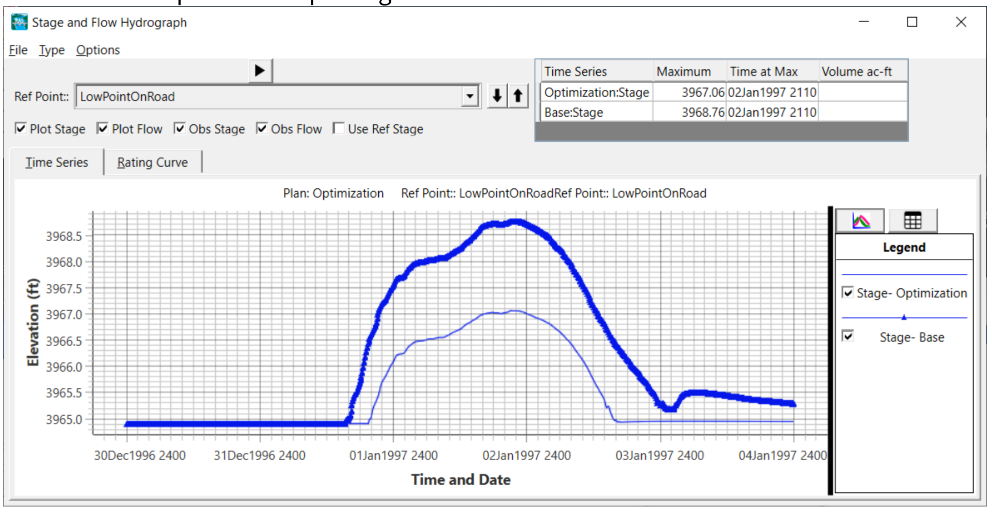
21. **Compute** the Optimization plan.
As the plan runs, you will be provided a status on the Trials.

22. Evaluate results using the **Hydrograph Plot**

> ⓘ To compare results at the Reference Point, you will need to re-run the Base plan. (The Reference Point is new to the Base run.)

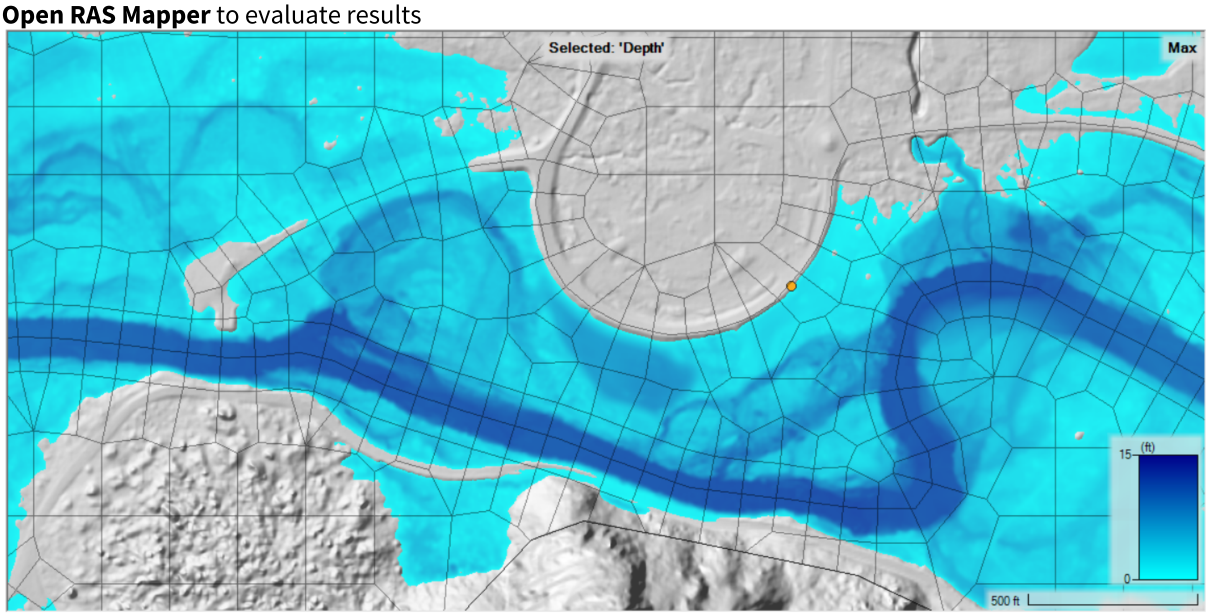
   a. From the **Hydrograph Plot**, choose the **Type | Reference Point**
   b. Select the **Options | Plans** menu item

c. Select the **Base** plan to compare against



Note from the output and the hydrograph plot, the Optimization did not quite get to the desired Stage (even though it was within the specified tolerance). Therefore, if you truly did not want the water surface higher than specified Target, you may need to lower the Target by the tolerance. This will be variable on how the flow optimization final solution approaches the desired Target.

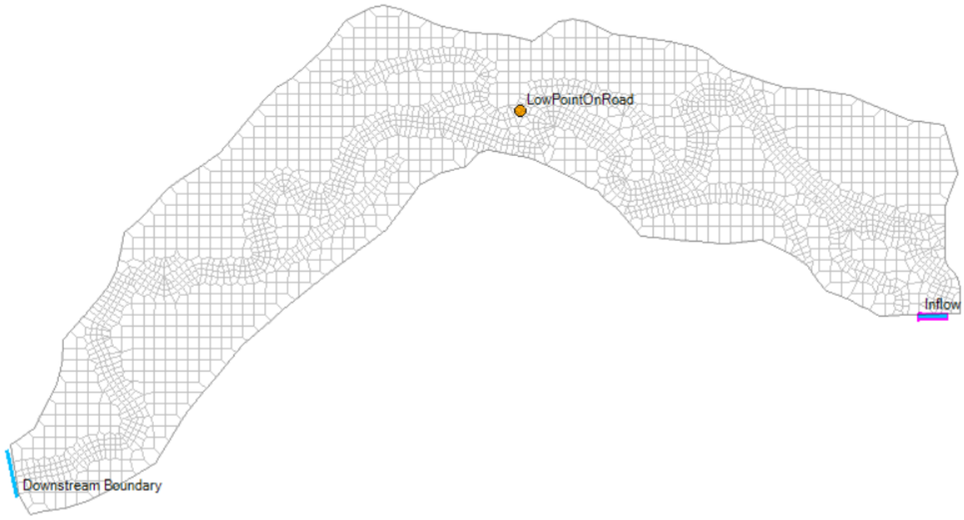
23. **Open RAS Mapper** to evaluate results



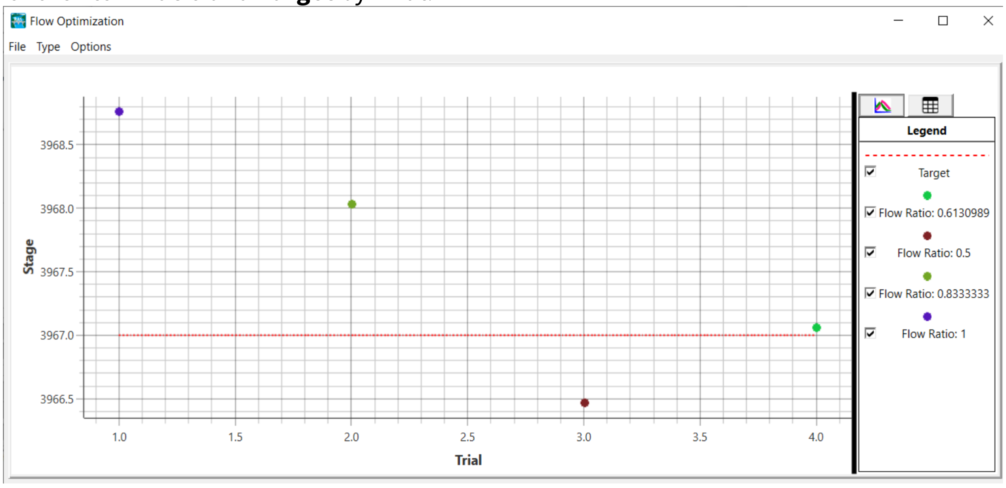
Note that the flow optimization was able to successfully scale the hydrograph to keep the desired location dry.

24. To access the Trial results, expand the Plan node to the **Flow Optimization** layer. The Flow Optimization layer can be used to identify the Reference Location and the hydrographs that were scaled. All boundary conditions are shown. Those that were used in the analysis ("Inflow" in the figure below) are shown in the

highlight color (magenta).



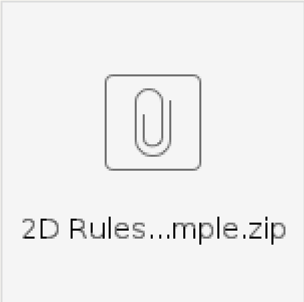
25. Right-click on the **Flow Optimization | View Flow Optimization Plot** to show you the optimization results for the **Flow Ratio** and **Target** by **Trial**.



2.6 2D Rules

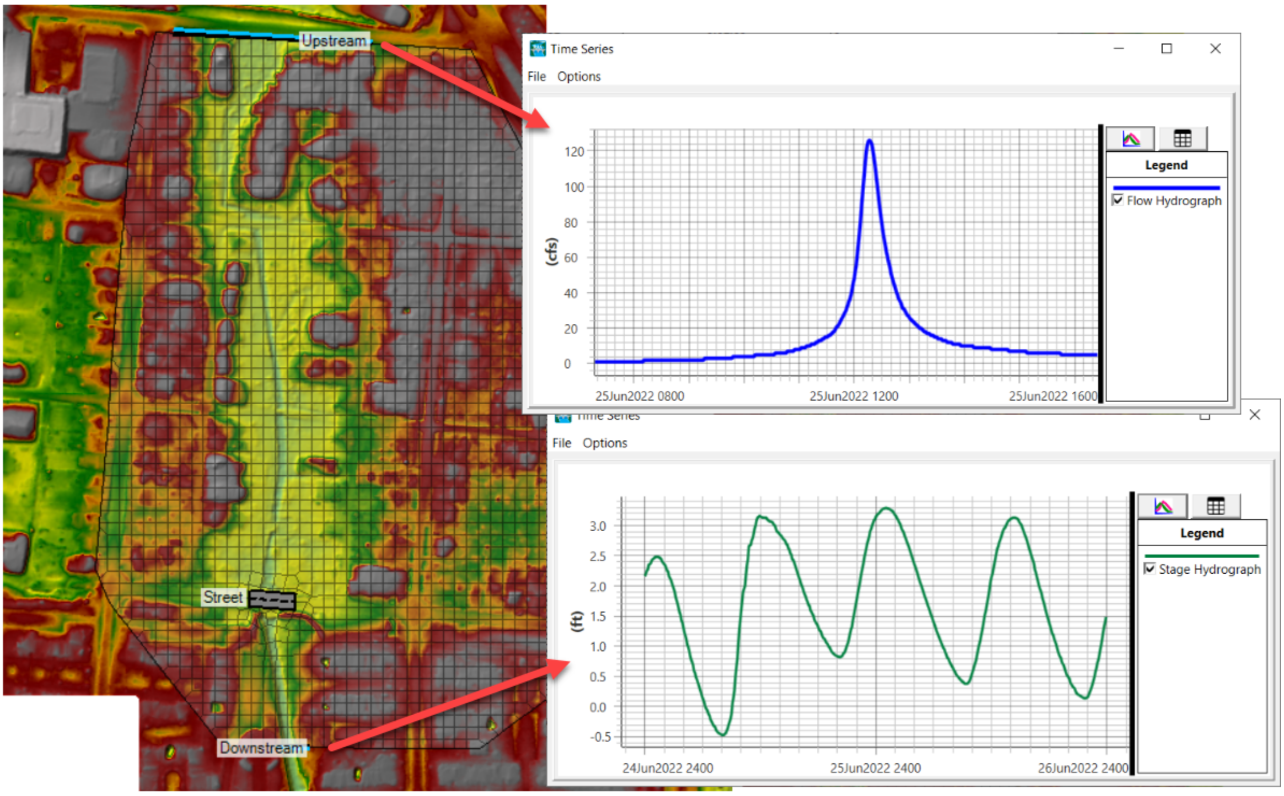
The Rules capability in HEC-RAS unsteady flow modeling is extremely powerful. This tutorial will walk the user through using the Rules capability within the 2D modeling framework.

Project Files

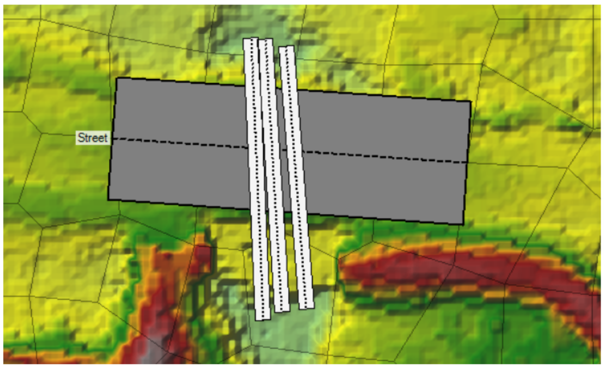


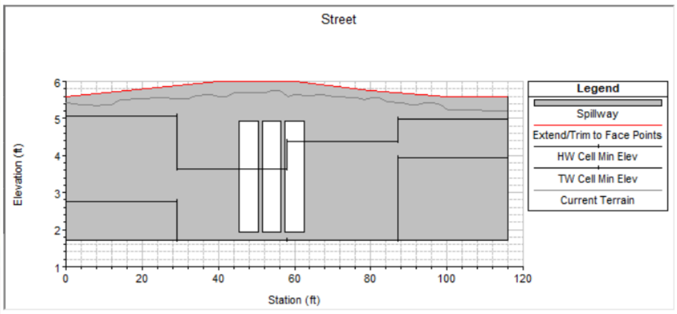
2.6.1 Overview

This example will demonstrate the use of the Rules capability to adjust gate settings for a 2D model that has a tidal boundary condition. The river used for the example has flow going from north to south - an inflow hydrograph is used for the ***Upstream*** boundary and a tidal stage boundary for the ***Downstream*** boundary condition. Gates are used to represent the culvert opening under the road crossing (***Street***) and is modeled using a 2D Connection. For the simple 2D model shown below, a 2D Flow Area has already been created with the road crossing.



The road crossing is modeled with a 2D Connection with the weir elevation set to the top of road elevation and three sluice gates (5' x 5') used to model the culverts.





The objective of the rule set described below is to hinder the intrusion of brackish water inland. Therefore, for the gates are closed if the downstream water surface elevation is too high and to be open the water surface upstream must be higher than the downstream water level. The (relatively simple) specific Gate Rules are defined below.

> **Gate Rules**
>
> - Gates OPEN when the Headwater elevation is greater than Tailwater by 0.7ft for more than 2min
> - Gates CLOSE when the Tailwater elevation is greater than 3.5ft for more than 5min.

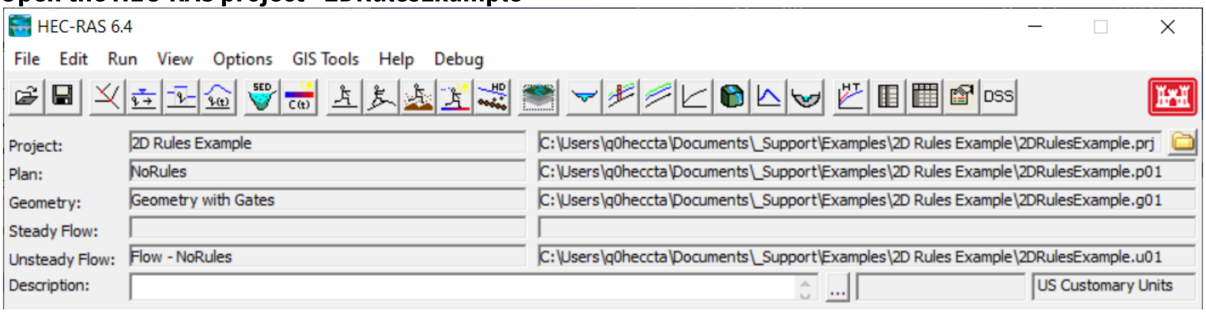
Reference Locations can be used in RAS to monitor Headwater and Tailwater water surface elevations.

This example will begin will a running model and take you through the process listed below.

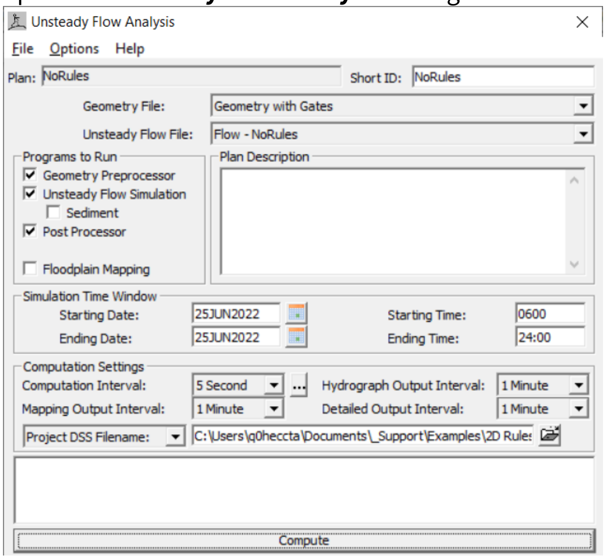
- Adding Reference Points
- Entering Rule Operations
- Visualizing Output
- Evaluating Results

2.6.2 Getting Started

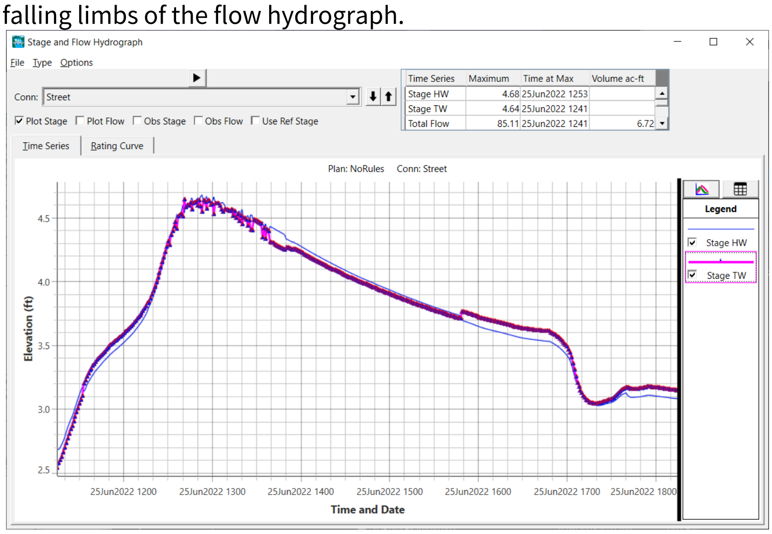
1. **Open the HEC-RAS project "2DRulesExample"**



2. Open the **Unsteady Flow Analysis** Dialog



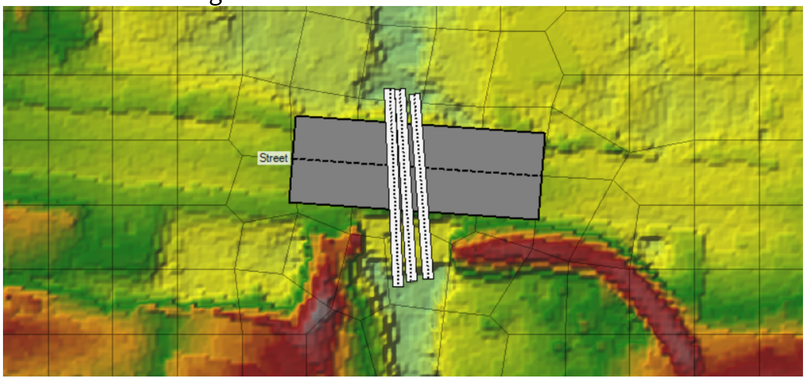
3. **Compute** the ***NoRules*** plan. Note that the Tailwater Stage is higher than the headwater on the rising and falling limbs of the flow hydrograph.



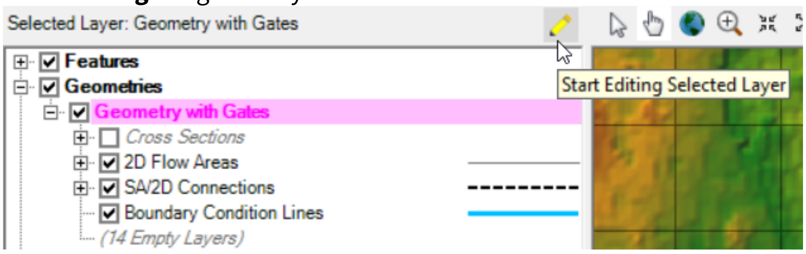
2.6.3 Add Reference Points

1. **Open RAS Mapper** - you need to add Reference Locations

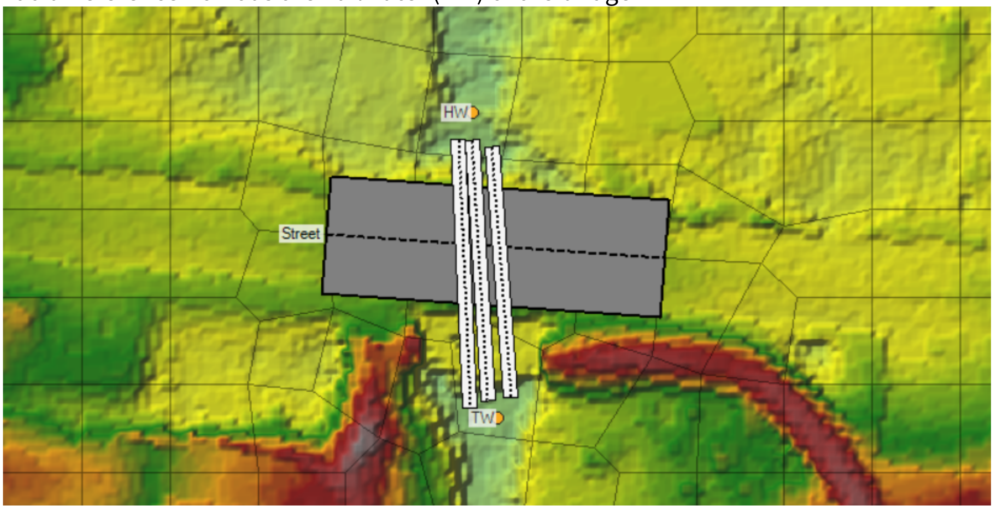
2. **Zoom** into the bridge



3. **Start Editing** the geometry



4. **Select** the **Reference Points** layer
    a. Add a Reference Point at the Headwater (**HW**) of the bridge
    b. Add a Reference Point as the Tailwater (**TW**) of the bridge

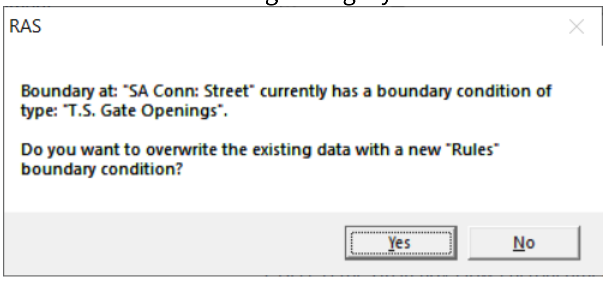


5. **Stop Editing** and Save Edits
6. **Close RAS Mapper**

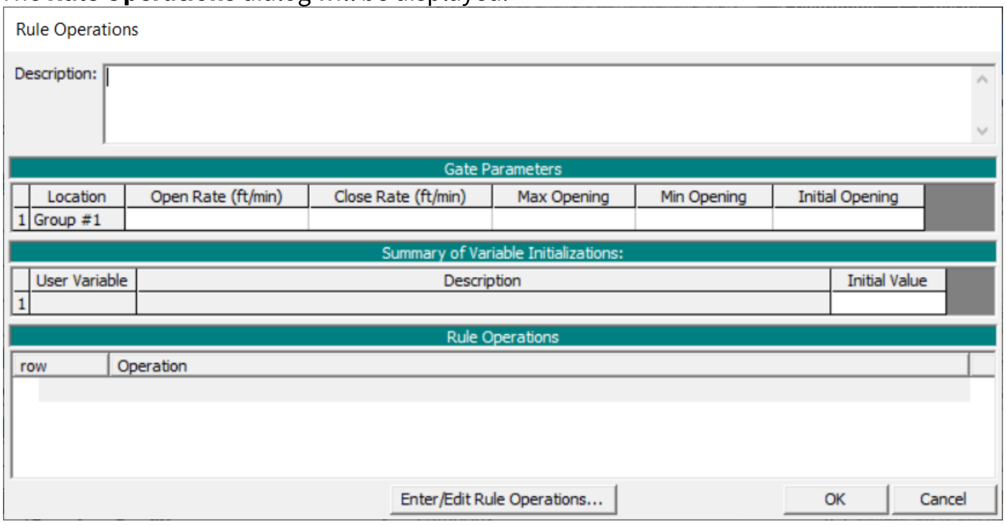
2.6.4 Rule Operations for Gates

1. Open the **Unsteady Flow Data** dialog
    a. Save the flow data using the **File | Save Unsteady Flow Data As...** menu item.
    b. Provide a new name for the flow file (Flow - Simple Rules)
2. Change the **Boundary Condition** for the Bridge (***Street***) by clicking on the **TS Gate Openings** and then clicking on the **Rules** button.

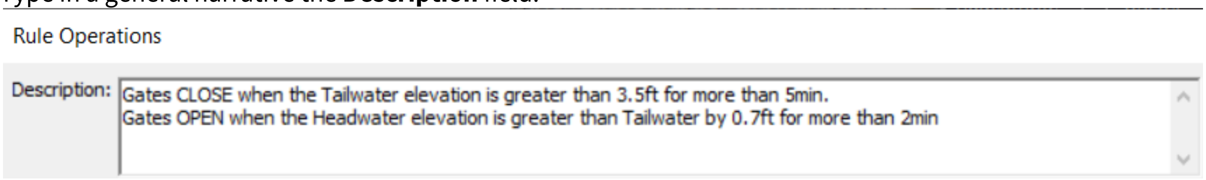
3. Choose Yes for the dialog asking if you would like to change the boundary condition.



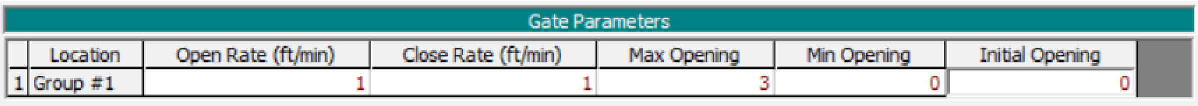
4. The **Rule Operations** dialog will be displayed.



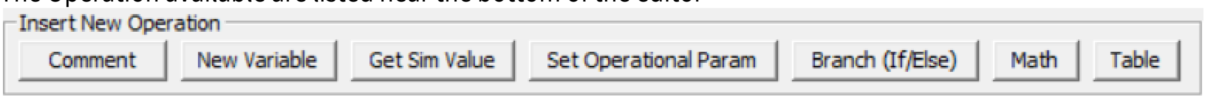
5. Type in a general narrative the **Description** field.



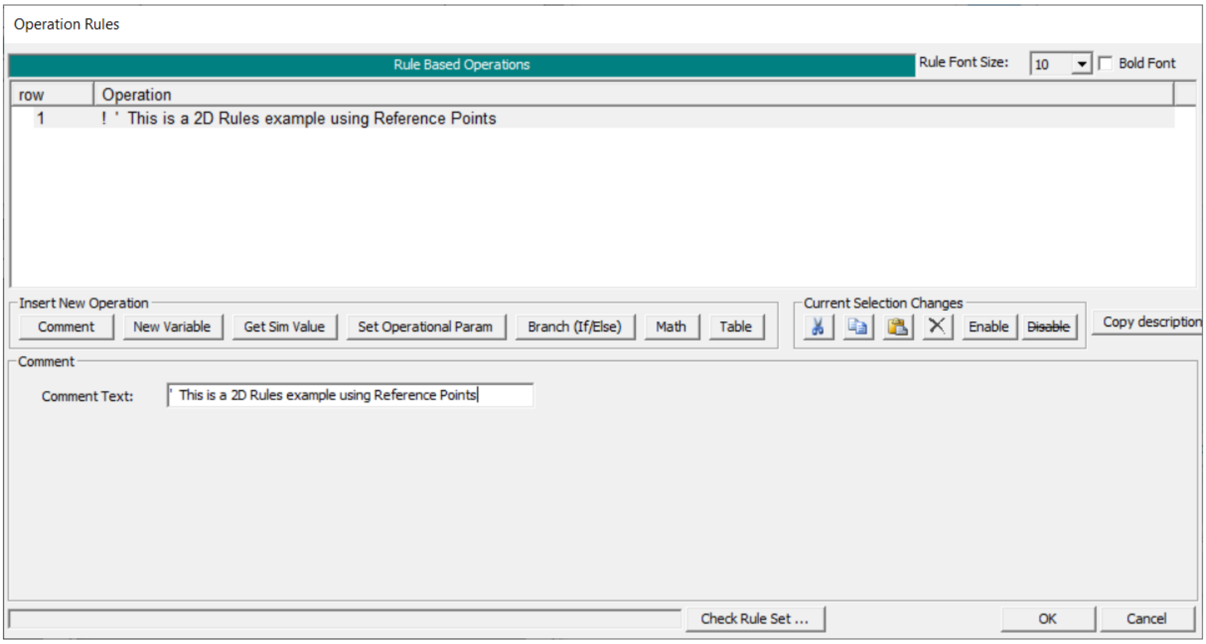
6. Provide base **Gate Parameter** information for how the gates will operate. These gates will Open and Close at a rate of 1ft/min and will open to a maximum height of 3ft. The simulation will begin with the gates closed.



7. Click on the **Enter/Edit Rule Operations** button to edit rules
   The Operation available are listed near the bottom of the editor



8. Press the **Comment** button to add a comment the rules set. Comments are helpful to remembering what you have done and to share information to other users.
   After pressing the comment button, a row will be added the Operations list. Type in your comment in the text box near the bottom of the Editor.

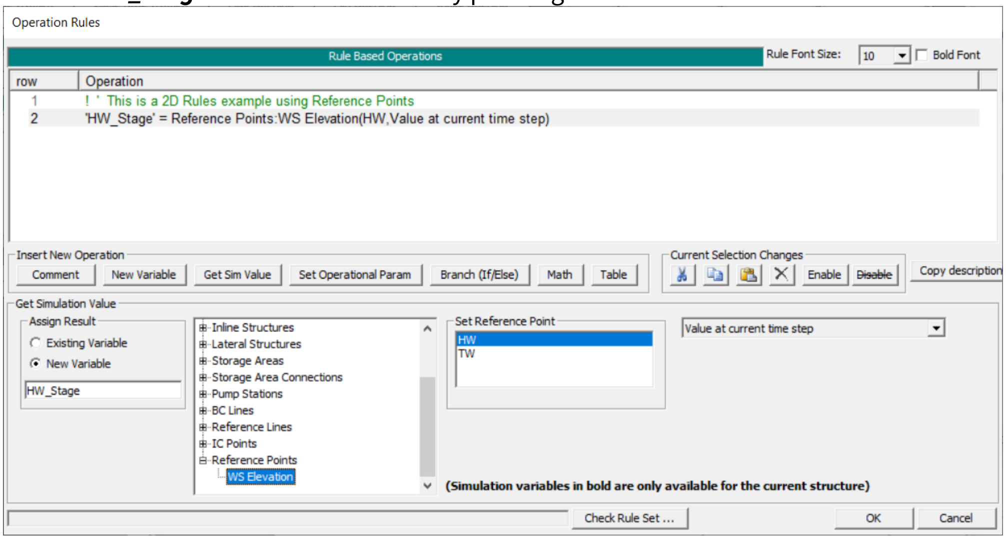


- New operations are added ABOVE the selected line.
- Click the line below to continue.
- Delete will delete the selected line.
- Copy/paste can be used to move/duplicate lines.

9. Set up variables to monitor the simulation values. A summary of variables is provided below.

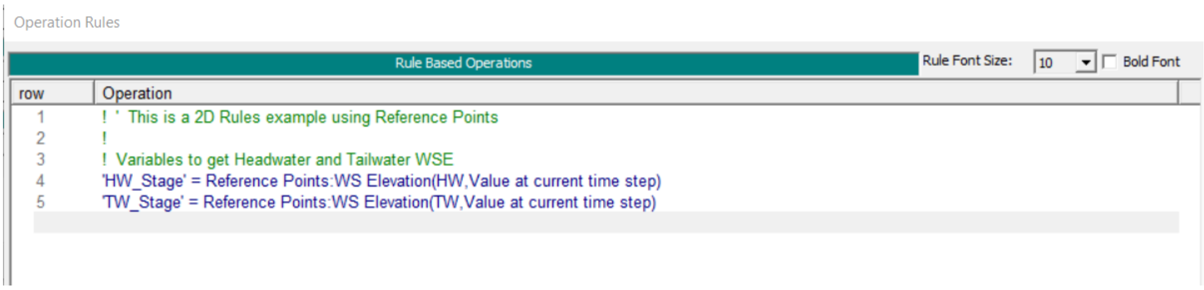
Variable Name	**Description**
HW_Stage	Stage of Headwater
TW_Stage	Stage of Tailwater
Timestep	Current Timestep
Time_TW_gt_3.5ft	Time that the Tailwater is greater than 3.5ft
Time_HWDiff_gt_0.7ft	Time that the Headwater - Tailwater is greater than 0.7ft
Done	Variable to monitor whether we have satisfied a rule (Done=1 is True)

10. Add the ***HW_Stage*** Simulation Variable by pressing the **Get Sim Value**.

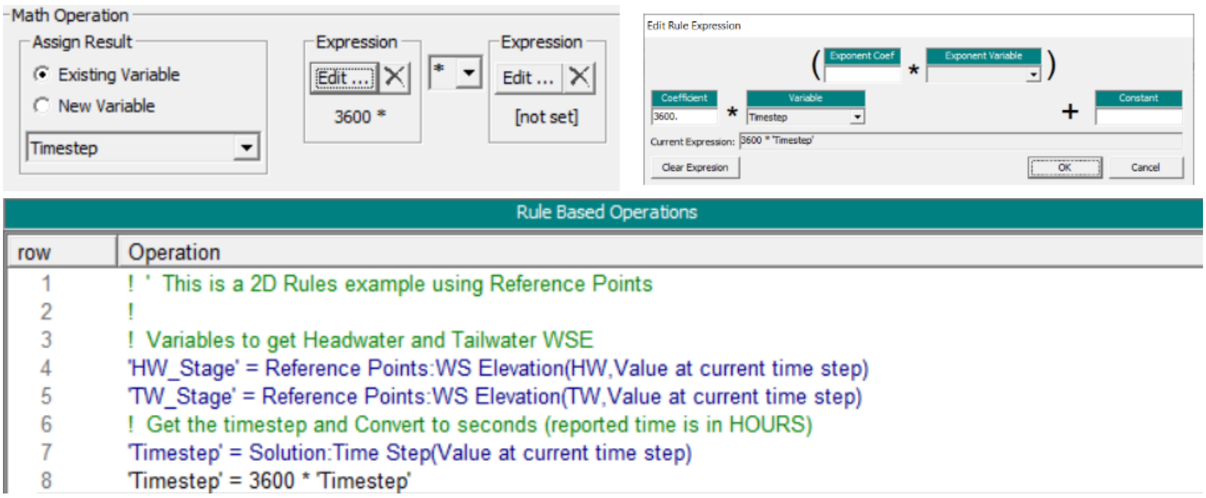


   a. Enter the **New Variable** name
   b. Select the **Reference Points | WS Elevation**
   c. Set variable to the **Reference Point** named ***HW***

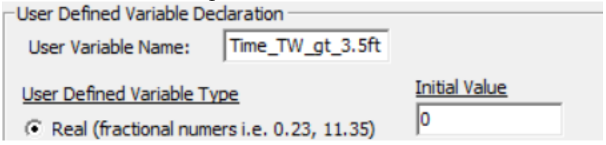
11. Click the **next row** in the list
12. Repeat for the ***TW_Stage***
13. Provide Comments!



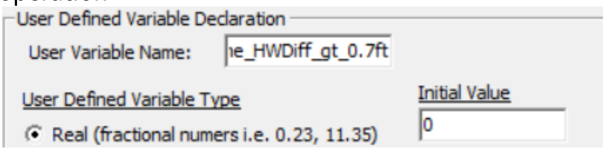
14. Add add the ***Timestep*** variable by pressing the **Get Sim Value**
   a. Set the variable to the **Solution | Timestep**
15. Convert the ***Timestep*** to seconds using the **Math** operation - HEC-RAS keeps track of simulation time in HOURS



16. Add the ***Time_TW_gt_3.5ft*** to monitor the Tailwater stage using the **New Variable** operation



17. Add the ***Time_HWDiff_gt_0.7ft*** to monitor the Headwater/Tailwater difference using the **New Variable** operation



18. Provide Comments for the variables
19. Create the ***Done*** Integer variable to keep track of the simulation - to know whether we have set a gate instruction and to stop evaluating the remainder of the rules for the current timestep - and
    a. Set it to **0** using the Math operation

```
! Variable to keep track if we are Done! (True=1)
Integer 'Done' (Initial Value = 0)
'Done' = 0
```

20. Create the **OPEN GATE** operation
    a. Create the first part of the rule to see if ***Done*** is True

```
! Gate OPENING: HW-TW>0.7ft
If ('Done' = 0) Then
End If
```

See operation steps ...

- Click on the **Branch (If/Else)** operation
- Select the **If () Then** option
  - Set the expression to ***Done = 0***
- Click the next row
- Click the **Branch (If/Else)** operation
  - Select the **End If** option

    b. Evaluate the Headwater/Tailwater difference and set the time counter accordingly

```
! Gate OPENING: HW-TW>0.7ft
If ('Done' = 0) Then
   If ('HW_Stage' > 'TW_Stage' + 0.7) Then
      'Time_HWDiff_gt_0.7ft' = 'Time_HWDiff_gt_0.7ft' + 'Timestep'
   Else
      ! Reset the time counter
      'Time_HWDiff_gt_0.7ft' = 0
   End If
End If
```

See operation steps ...

- Select the last End If row
- Click on the **Branch (If/Else)** operation
  - Set the expression to ***HW_Stage > TW_Stage + 0.7***
- Click the **Math** operation
  - Set the expression to ***Time_HWDiff_gt_0.7ft = Time_HWDiff_gt_0.7ft + Timestep***
- Click the next row
- Click the **Branch (If/Else)** operation
  - Select the **Else** option
- Click the next row
- Click the **Math** operation

    - Set the expression to ***Time_HWDiff_gt_0.7ft = 0***
  - Click the next row
  - **Click the Branch (If/Else) operation**
    - Select the **End If** option

c. Add the **Gate Opening** rule based on the Headwater Difference time counter previously evaluated

```
! Gate OPENING: HW-TW>0.7ft
If ('Done' = 0) Then
    If ('HW_Stage' > 'TW_Stage' + 0.7) Then
        'Time_HWDiff_gt_0.7ft' = 'Time_HWDiff_gt_0.7ft' + 'Timestep'
    Else
        ! Reset the time counter
        'Time_HWDiff_gt_0.7ft' = 0
    End If
    ! Start OPENING the Gates (120s = 2min)
    If ('Time_HWDiff_gt_0.7ft' > 120) Then
        Gate.Opening(Group #1) = 3
        ! Reset the Time counter for the HW evaluation
        'Time_TW_gt_3.5ft' = 0
        'Done' = 1
    End If
End If
```

See operation steps ...

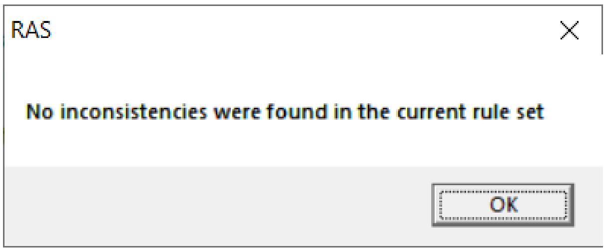
  - Select the last End If row
  - Click on the **Branch (If/Else)** operation
    - Set the expression to ***Time_HWDiff_gt_0.7ft > 120***
  - Click the **Set Operational Parameter** operation
    - Set the expression to ***Gate.Opening = 3***
  - Click the next row
  - Click the **Math** operation
    - Set the expression to ***Time_TW_gt_3.5ft = 0***
  - Click the next row
  - Click the **Math** operation
    - Set the expression to ***Done = 1***
  - Click the next row
  - **Click the Branch (If/Else) operation**
    - Select the **End If** option

21. Create the **CLOSE GATE** operation (repeat previous steps but evaluating the Tailwater condition)

```
! Gate CLOSURE: TW>3.5ft
If ('Done' = 0) Then
    If ('TW_Stage' >= 3.5) Then
        'Time_TW_gt_3.5ft' = 'Time_TW_gt_3.5ft' + 'Timestep'
    Else
        ! Reset the Time counter
        'Time_TW_gt_3.5ft' = 0
    End If
    ! Start CLOSING the Gates (300s = 5min)
    If ('Time_TW_gt_3.5ft' > 300) Then
        Gate.Opening(Group #1) = 0
        ! Reset otherTime counter for the TW Evaluation
        'Time_HWDiff_gt_0.7ft' = 0
        'Done' = 1
    End If
End If
```

22. Provide **Comments** in the Rule Operations where needed

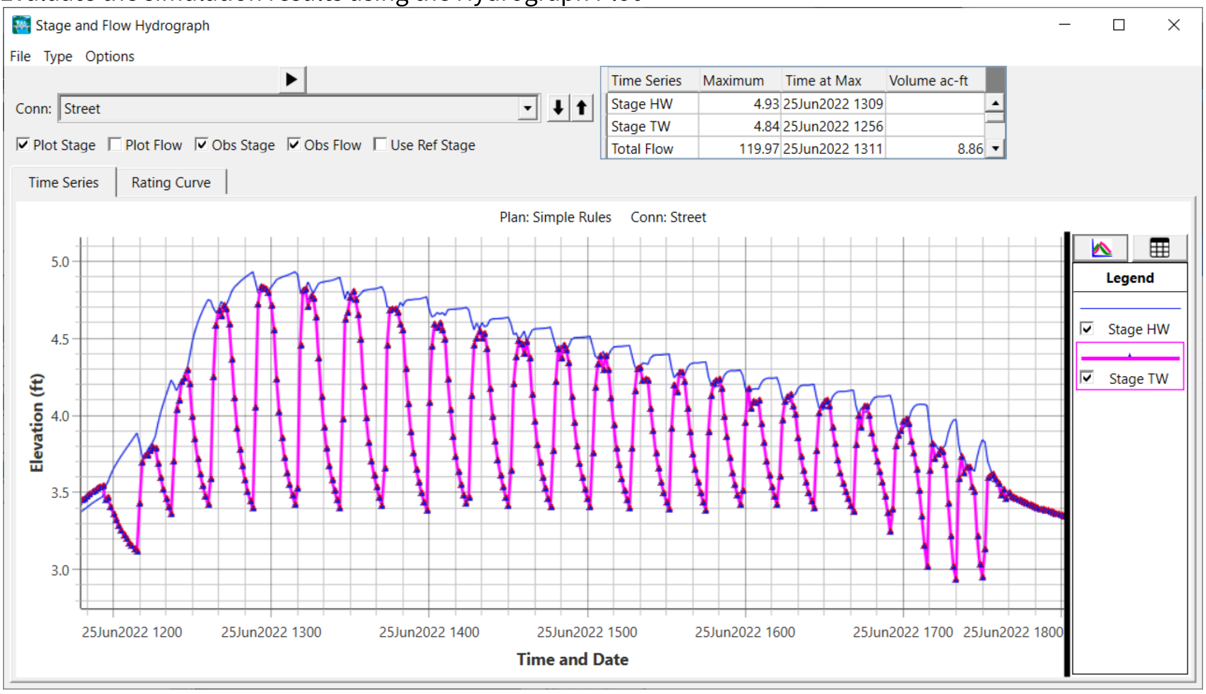
23. Press the **Check Rule Set** button to evaluate the Rules. Hopefully, you there are no syntax error.



24. Press **OK** on the **Operation Rules** editor
25. **Save** the Unsteady Flow Data

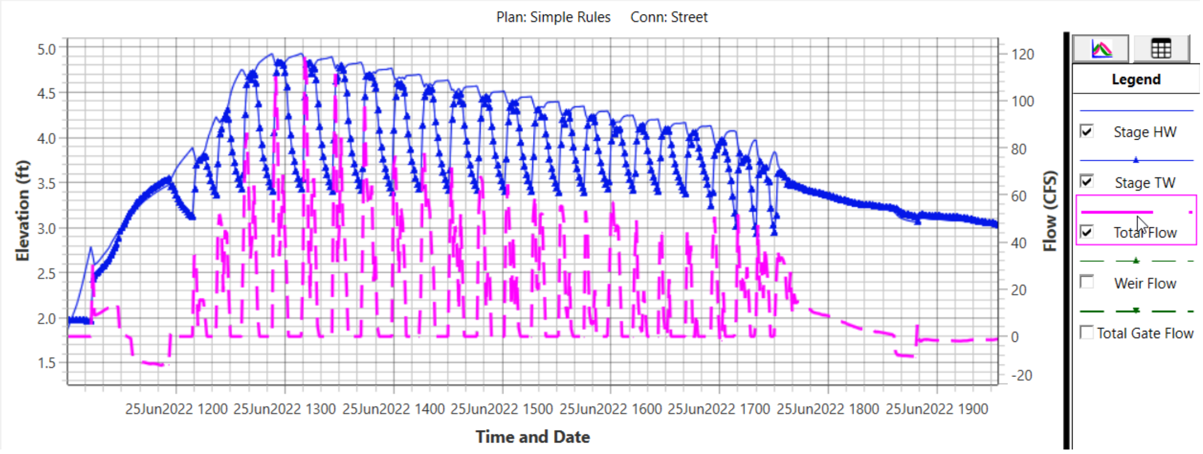
2.6.5 View Output

1. Go to the **Unsteady Flow Analysis** dialog
2. **Save** the Plan, providing a name (Simple Rules). Provide a ShortID for the new plan.
3. **Compute** the Simple Rules plan
4. Evaluate the simulation results using the Hydrograph Plot

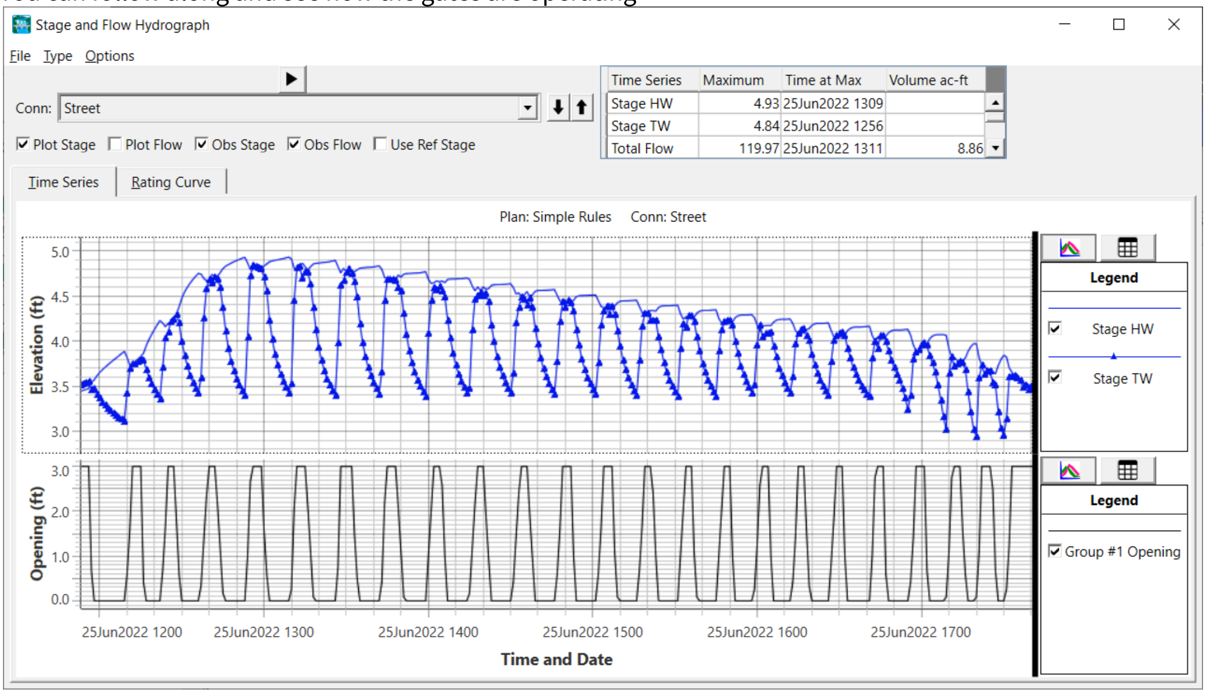


Note that the Tailwater no longer gets higher than the headwater

5. If you turn on Flow, you can see flow mostly "positive" in the downstream direction. Without the rules there was significant negative flows (flows going upstream based on the high tide).



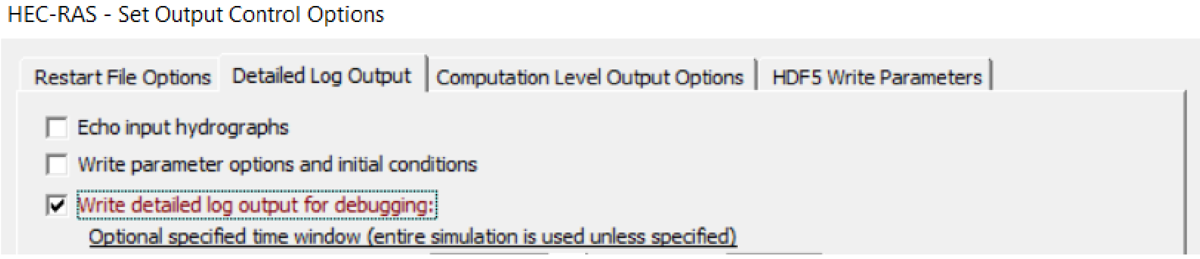
6. To see the Gate Operations, choose the **Options | Variables** menu item
7. Select the "Group #1 Opening" variable
8. You can follow along and see how the gates are operating



2.6.6 Evaluate Results

1. Go to the **Unsteady Flow Analysis** dialog
2. Choose the **Options | Output Options** menu item

3. Turn on the **Computation Level Output**

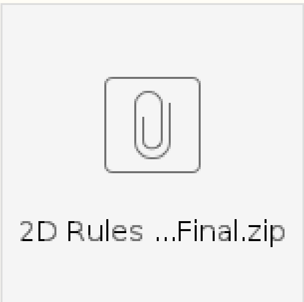


4. **Re-compute** the simulation
5. Choose the **Options | View Computation Log File** menu item
6. Evaluate the Rule Operations. You can find where the Rules were evaluated by searching the text file for "Rule Set"

```
                              Solving for Time Window =     6.88611     to       6.88750 Hours

    Rule Set for     Street

0004                             HW_Stage = 4.931848
0005                             TW_Stage = 3.389156
0007                             Timestep = 0.0013889
0008                             Timestep = 5.

0015                                 Done = 0.0
0018     True         0.0 =  0.0
0019     True    4.931848 >  4.089156
0020                 Time_HWDiff_gt_0.7ft = 125.

0026     True        125. >  120.
0027 SET            Gate.Opening Group #1 = 3.
0029                     Time_TW_gt_3.5ft = 0.0
0030                                 Done = 1.

0035     False         1. =  0.0
0051                          GateOpening = 0.0
```

Each line of the Rule Operation is enumerated in the left column. The result of each evaluation is shown in the next column, follow by the evaluation. For the timestep evaluated in the figure above, the Gate evaluation resulted in the decision to set the gate opening to 3ft. The Log File in concert with the Hydrograph Plots allow you visualize how HEC-RAS evaluated the operation during the simulation.

Final Project Files



2.7 1D Sediment Modeling Tutorial

Project Files (See Solution Files at the End)

Data Files (1D Se...ent Tutorial).zip

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

http://youtube.com/watch?v=d416442IC4c

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

http://youtube.com/watch?v=9YiL3Men9as

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

http://youtube.com/watch?v=X9xikwi0v-U

Solution Files (1D...ment Tutorial).zip

2.8 Modeling a 2D Half Pipe with Non-Newtonian Fluid

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

http://youtube.com/watch?v=cZDqpKYO7Ek

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

http://youtube.com/watch?v=kAZhWw-j0HI

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

http://youtube.com/watch?v=DvicPxAR9gg

ParsonsDebrisPipe (Solution).zip

2.9 Debris Flow Workshop

2.9.1 Full Workshop (Start with a Shape File)

Non-Newtonian D...S Download).pdf

Workshop Starting Files (US).zip

2.9.2 Abbreviated Workshop (Start with Terrain)

SedHyd Debris Fl...Abriviated).docx

SedHyd Debris Wo...g Files (US).zip

2.9.3 Solution

Debris Workshop Soln (US).zip

3 Reference Documents

Topic	Supporting Material
HEC-RAS Verification and Validation Document This document contains analytical and textbook datasets, laboratory datasets, and real world data with observed measurements. HEC-RAS computational results are compared to all of the datasets to verify the solutions of the equations, and to validate HEC-RAS's use for a wide range of computational problems. (Updated September 2020)	RD-52_HE...2020.pdf
Benchmarking of the HEC-RAS Two-Dimensional Hydraulic Modeling Capabilities This document summarizes how the HEC-RAS software performed in the two-dimensional modeling benchmark tests developed by the United Kingdom's (UK) Joint DEFRA (Department for Environment, Food, and Rural Affairs) Environment Agency. (Updated April 2020)	RD-51 HE...2020.pdf
Modeler Application Guidance or Steady vs. Unsteady and 1D vs. 2D vs. 3D Hydraulic Modeling All models, numerical or scale-physical, are simplified representations of the real world (prototype). Fortunately, there are numerous practical engineering problems for which simplified numerical models of the prototype are sufficient to provide usable descriptions of system behavior. The challenge for the modeler is to select an appropriate model to solve their particular engineering problem while recognizing that the model is not a perfect representation of the prototype. Selection of a model begins with developing an understanding of which aspects of the complex, real-world system are most important to the engineering problem being addressed. The purpose of this document is to provide entry to mid-level hydraulic engineer's with guidance on when to use Unsteady Flow modeling instead of Steady flow modeling; and how to select between one-dimensional (1D), two-dimensional (2D), or three-dimensional (3D) modeling for a given problem	TD-41.pdf

Topic	Supporting Material
Using HEC-RAS for Dam Break Studies This document provides information on how to use the HEC-RAS (River Analysis System) software when performing a dam break analysis. The document presents the unique hydraulic modeling aspects that are required, plus routing the inflow flood through a reservoir; estimating dam breach characteristics; and, downstream routing/modeling issues	TD-39.pdf
BSTEM Technical Reference and User's Manual The HEC-RAS software has included mobile bed capabilities since version 4.0. These capabilities compute vertical bed changes in response to dynamic sediment mass balance and bed processes. However, many riverine sediment problems involve lateral bank erosion that does not fit the current computational paradigm. The Bank and Stability Toe Erosion Model (BSTEM) developed by the United States Department of Agriculture Research Station is a physical based model that accounts for the dominant stream bank processes but requires an intermediate level of complexity and parameterization.	HEC-RAS_...nual.pdf
Comparison of One-Dimensional Bridge Hydraulic Routines from HEC-RAS, HEC-2, and WSPRO The hydraulics of flow through bridges is an important aspect of computing water surface profiles. The computation of accurate water surface profiles through bridges is necessary in flood damage reduction studies, channel design and analysis, and stream stability and scour evaluations. There are several one-dimensional water surface profile computer programs available for performing these types of computations. The purpose of this study was to evaluate the effectiveness of the new bridge hydraulics routines in HEC-RAS at sites with extensive observed data, and to compare HEC-RAS to HEC-2 and WSPRO, with respect to bridge modeling performance.	RD-41.pdf

Topic	Supporting Material
Flow Transitions in Bridge Backwater Analysis Bridges across floodplains may require special attention in one-dimensional hydraulic modeling if they cause severe contraction and expansion of the flow. The accurate prediction of the energy losses in the contraction reach upstream of the bridge and the expansion reach downstream of the bridge using one-dimensional models presents particular difficulty. Modeling these reaches requires the accurate evaluation of four parameters: the expansion reach length (Le), the contraction reach length (Lc), the expansion coefficient (Ce), and the contraction coefficient (Cc). This report presents research conducted by the author to investigate these four parameters through the use of field data, two-dimensional hydraulic modeling, and one-dimensional modeling.	RD-42.pdf
Making Your HEC-RAS Model Run Faster The purpose of this document is to provide guidance on model modifications that can be made to an HEC-RAS unsteady flow model (1D and/or 2D) in order to improve computational speed, while maintaining model accuracy.	Making HE...aster.pdf

4 Web Resources

Resource	Web Link
The National Map Viewer The National Map is a collaborative effort among the USGS and other Federal, State, and local partners to improve and deliver topographic information for the Nation.	USGS - National Map Viewer[7]
Spatial Reference A great website for getting a coordinate system for your project	spatialreference.org[8]
National Land Cover Database The National Land Cover Database (NLCD) provides nationwide data on land cover and land cover change at a 30m resolution with a 16-class legend based on a modified Anderson Level II classification system. NLCD 2019 represents the latest evolution of NLCD land cover products focused on providing innovative land cover and land cover change data for the Nation.	www.mrlc.gov/data[9]
USGS Land Cover USGS provides land cover data for the United States in raster or vector (shapefile) format.	https://water.usgs.gov/GIS/dsdl/ds240/index.html

7 https://apps.nationalmap.gov/viewer/
8 http://www.spatialreference.org
9 http://www.mrlc.gov/data

Resource	Web Link
Roughness Characteristics of Natural Channels Color photographs and data are provided for 50 stream channels for which Manning's roughness coefficients have been determined.	USGS Water Supply Paper 1849[10] wsp_1849.pdf
Determination of Roughness Coefficients for Streams in Colorado This report investigates Manning's roughness coefficients for steep streams.	USGS Report 85-4004[11] Jarrett (1985).pdf

10 https://pubs.usgs.gov/wsp/wsp_1849/html/pdf.html
11 https://https//pubs.usgs.gov/wri/1985/4004/report.pdf