

Initial Debris Flow Workshop: Part 2

Refining Your Model and Improving Stability

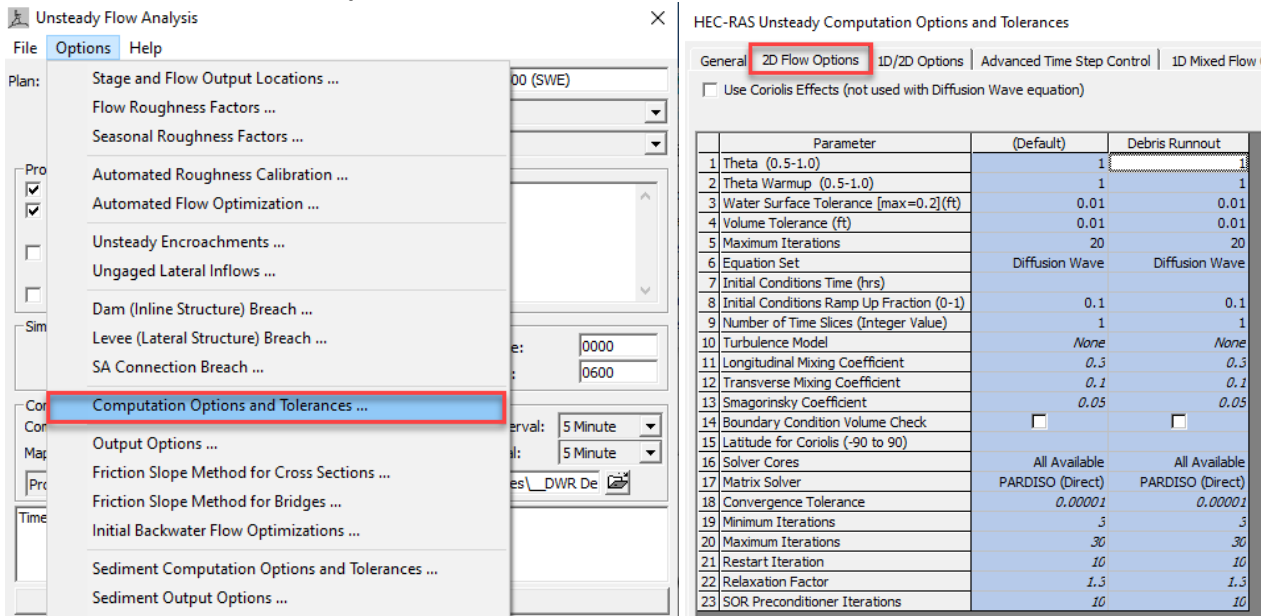
In the first part of this workshop we tried to get to a reasonable risk map and timeline as quickly as possible. In this workshop we are going to try to refine the model and improve the stability

1. Improve the Hydrodynamic Simulation

Open the Options Menu of one of your non-Newtonian Plan Files.

Go to **Options** → **Computation Options and Tolerances**.

Click on the **2D Flow Options** tab.

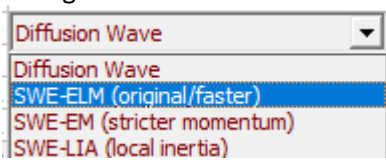


Which of these parameters might you want to change? _____

We usually recommend a warmup or initial conditions window for 2D HEC-RAS models, why wouldn't we necessarily use one for this model?

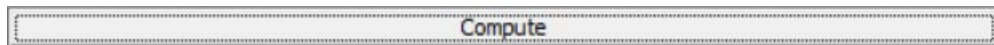
Our expedited models used the **Diffusion Wave** model.

Change that to the Shallow Water Flow Equation **SWE-ELM**



Save your plan AS and give it a new name (e.g. add a "(SWE)" to the name and short id).

From the Simulation (Plan) editor  press the **Compute** button.



How does the stability messages and the Max Errors compare to your Diffusion Wave simulation? _____

01JAN2100 00:06:30	Debris Runnout	Cell #	13441	5028.66	0.709	20	Max WSE Error
01JAN2100 00:06:35	Debris Runnout	Cell #	13441	5030.23	1.849	21	
01JAN2100 00:06:40	Debris Runnout	Cell #	13440	5025.90	1.592	21	
01JAN2100 00:06:45	Debris Runnout	Cell #	13441	5029.91	1.213	22	
01JAN2100 00:06:50	Debris Runnout	Cell #	13280	5038.30	0.453	20	
01JAN2100 00:06:55	Debris Runnout	Cell #	13441	5028.09	2.055	21	
01JAN2100 00:07:00	Debris Runnout	Cell #	13441	5028.76	1.892	23	
01JAN2100 00:07:05	Debris Runnout	Cell #	13268	4988.41	2.727	22	
01JAN2100 00:07:10	Debris Runnout	Cell #	13441	5028.26	1.553	21	
01JAN2100 00:07:15	Debris Runnout	Cell #	13266	4981.88	1.636	23	
01JAN2100 00:07:20	Debris Runnout	Cell #	13441	5028.18	2.717	20	
01JAN2100 00:07:25	Debris Runnout	Cell #	13441	5028.35	2.611	21	
01JAN2100 00:07:30	Debris Runnout	Cell #	13441	5027.61	3.937	22	
01JAN2100 00:07:35	Debris Runnout	Cell #	13441	5027.33	4.765	21	

These Water Surface Errors are in feet, do they seem acceptable? _____

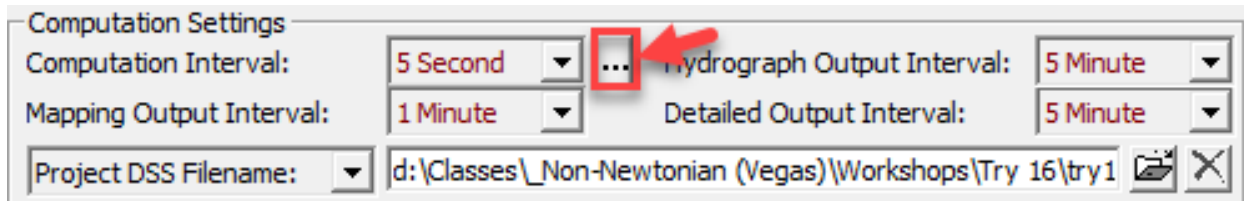
What could you do to improve them? _____

You selected the time step based on the Courant Condition, why isn't that time step appropriate for the whole simulation? _____

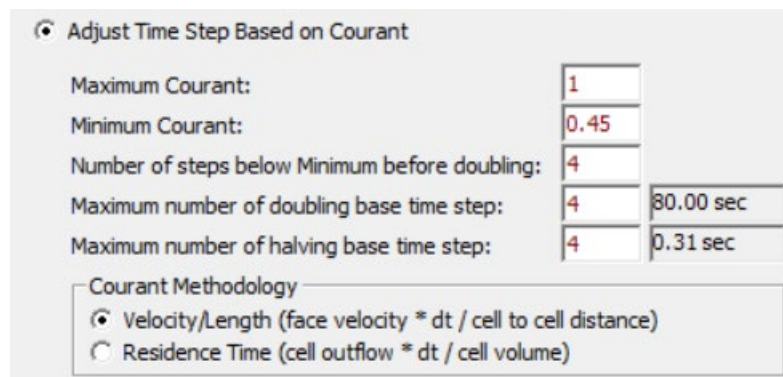
2. Improving the Model Time Step

Velocities change throughout the model, especially with Non-Newtonian models where flow can approach 0. Sometimes a variable time step that tries to optimize the Courant dynamically can improve stability.

To specify a variable time step, press the “...” button next to the **Computational Time Step**, or select **Options** → **Computation Options and Tolerances** and choose the **Advanced Time Step Control** tab.



Select the **Adjust Time Step Based on Courant** radio button.



Set the **Maximum Courant** to 1 for this exercise.

(Note: This can be as high as 2 or 4 for some models, but because we are not too concerned about run time, we set it for the strictest criteria.)

Then, the **Minimum Courant** should be a little less than half of the max (to avoid frequent switching). The final parameter (Max # of halving) is the most important of the remaining three, but you can set them all to 4.

How did changing the time step affect your iteration and error? _____

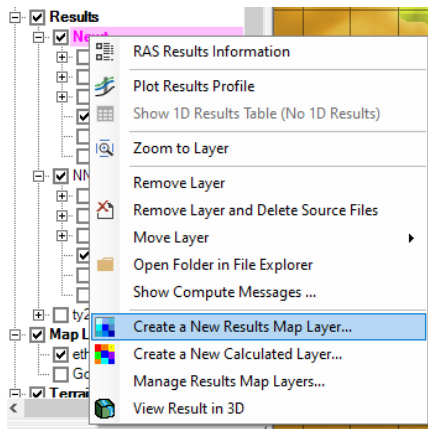
Why? _____

3. Evaluate Stability and Cell Connectivity Spatially

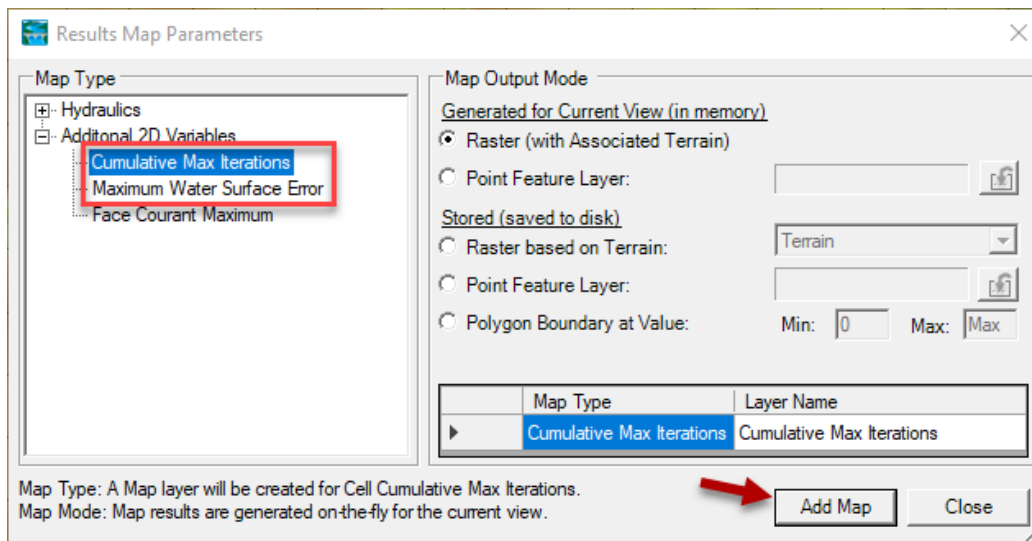
The Max Iteration/Water Surface Error in the runtime window can be helpful, but it only tells you the worst case. You could go search for that cell and fix it, but there might be tens, or hundreds, of other cells with similar behavior.

Therefore, it is extremely useful to view the iteration behavior and max error spatially.

Right Click on the Result and select **Create New Results Map Layer...**



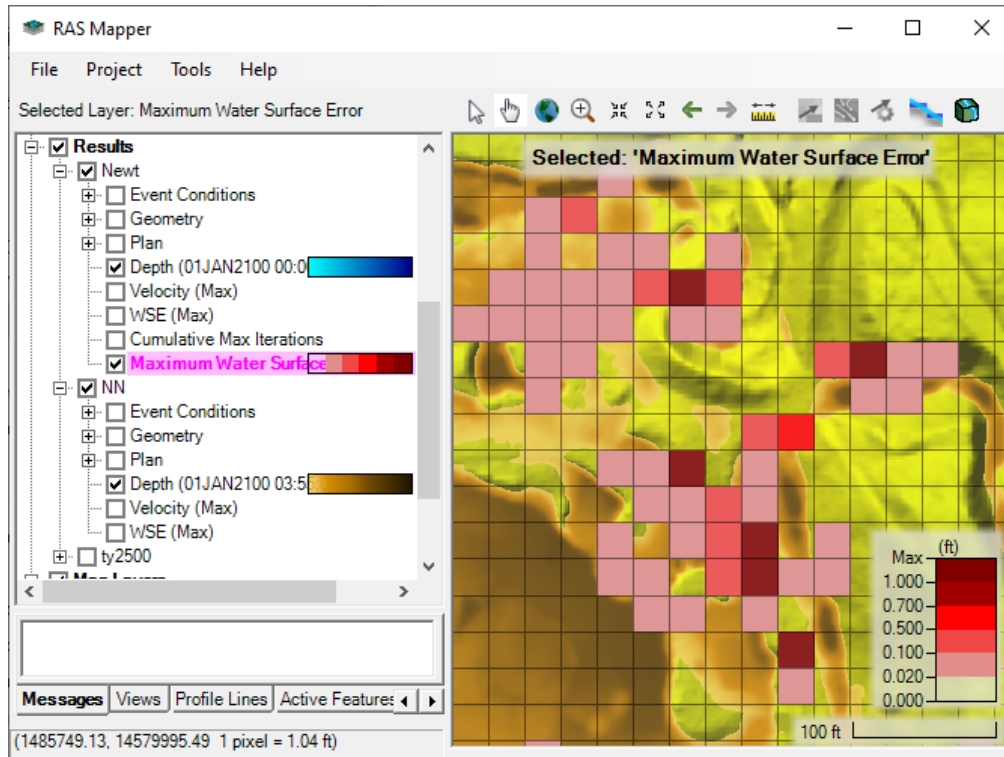
Expand **Additional 2D Variables** and add **Cumulative Max Iterations** and **Maximum Water Surface Error** by pressing **Add Map**.



Plot these error metrics and identify cells with substantial errors.

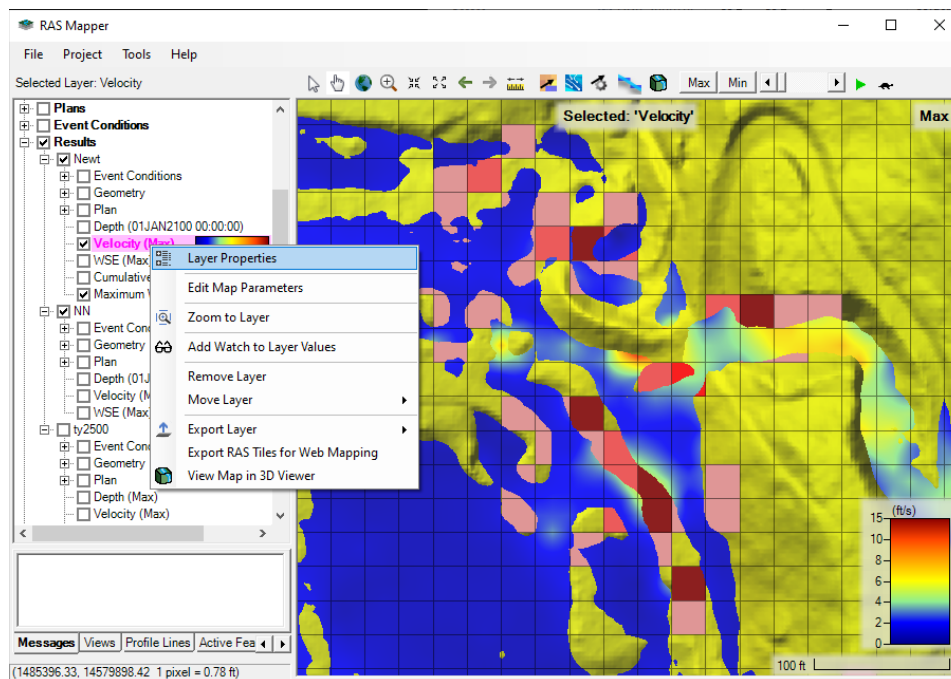
These will differ depending on how you drew your polygon.

But they often correspond to ridges or channels or “leaking meshes” that could be improved with break lines.

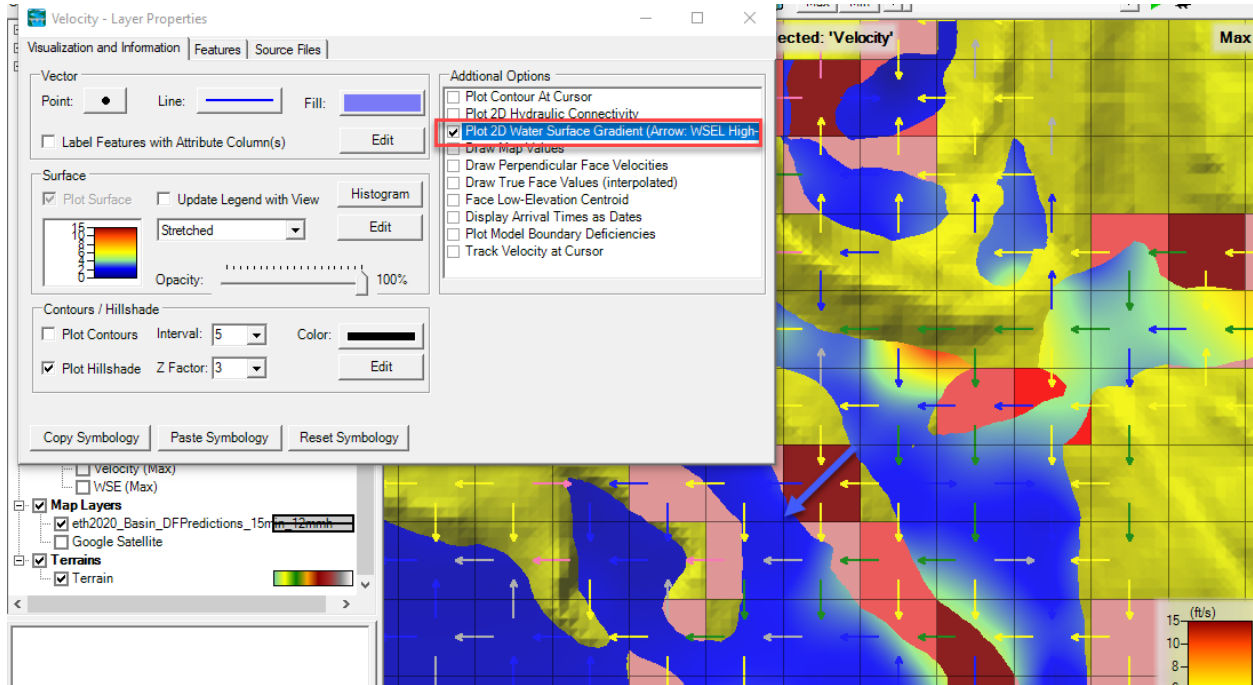


Plotting velocity can illustrate where flow is jumping ridges or high ground. (notice where the velocity gets high at the edge of the wetted area, which is also near high error cells.)

Evaluating continuity can also help you evaluate where your mesh needs improvement. Turn on Continuity Arrows by right clicking on Velocity and selecting **Layer Properties**.



Select the **Plot 2D Water Surface Gradient (Arrow WSEL...)** This illustrates that the cells with the most error tend to jump high ground ridges.



You are looking for cells that should not be connected hydraulically, but have arrows indicating water is flowing through them. In the image above, we drew a big blue arrow across a high iteration cell which should not be allowing flow, because it is a ridge. But because two corners of the cell are wet, the continuity arrows show that water is flowing through it. And because flow should not convey through this cell, the model is having trouble converging on a solution.

What could you do to fix this?

4. Improve your Mesh

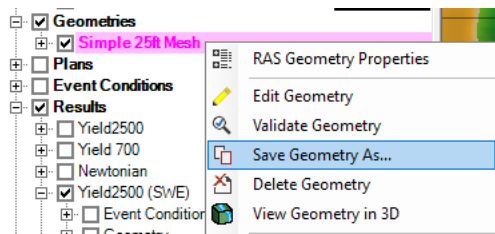
Finally, now that you understand the regions that your model is struggling, you can improve the mesh.

You have several tools to improve the result, including:

- Breaklines
- Refinement Regions
- n-Value Regions


But in this workshop, you will mostly use Breaklines, which are the most common and versatile tool.

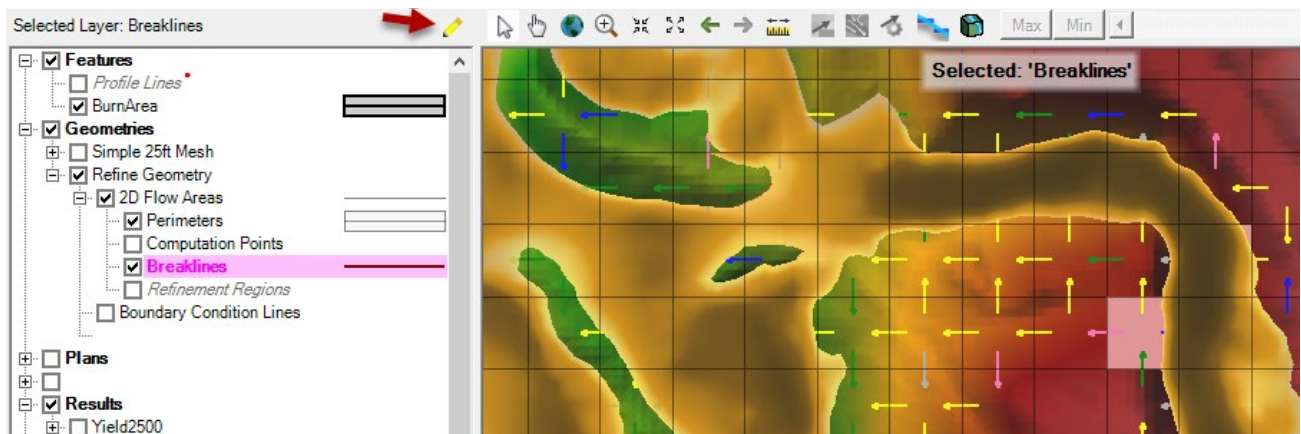
Return to the Geometry tree, Right Click on your Geometry and **Save Geometry As**.



Give the new Geometry a name like “Refine Geometry”

Expand 2D Flow Areas, and click the pencil to start editing.

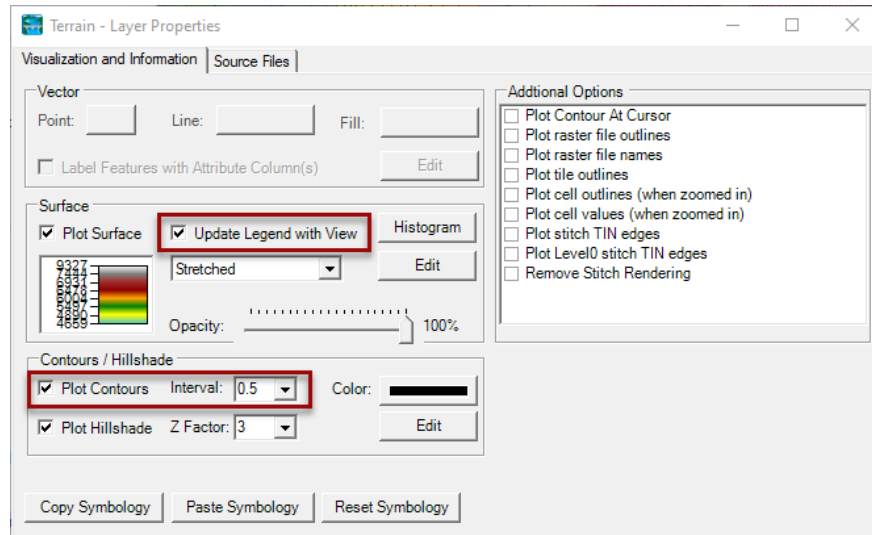
Choose the Draw  button and draw break lines along high ground.



Note: When you are trying to draw breaklines along high ground, there are a couple of visualization options that can help.

Right Click on the Terrain and select **Image Display Properties**. Select **Plot Contours** (experiment with a good interval) and **Update Legend with View** options.

Then right click on the velocity result, select **Layer Properties** and decrease the **Opacity** (e.g. decreasing opacity increases transparency) of the velocity result.



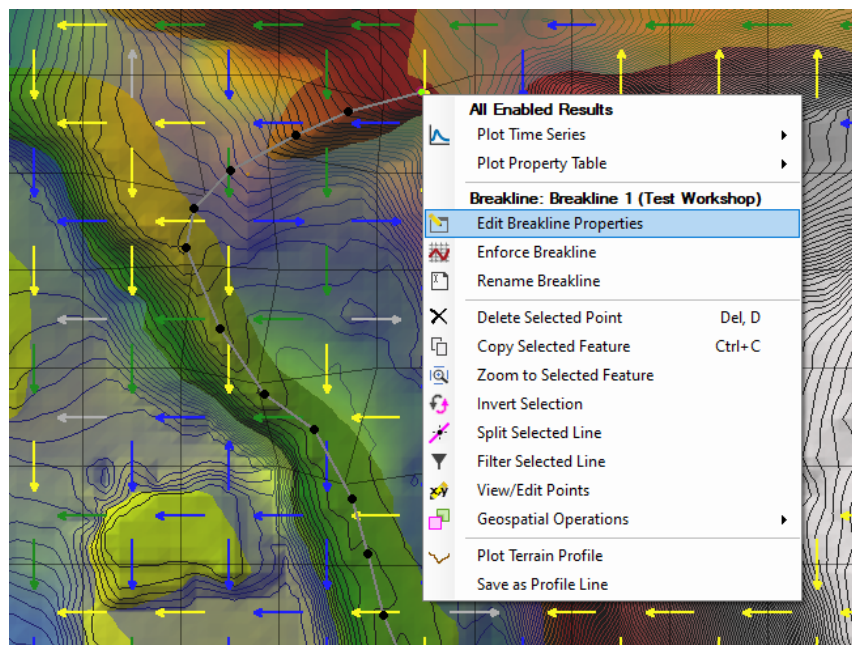
Draw a breakline along a ridge or high ground.

Double click to stop drawing when you are done.

You may notice that a few of the cells snap to the breakline but mostly the mesh does not change.

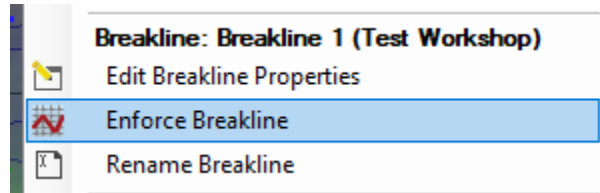
You need to edit the properties and enforce the breakline.

Right Click on the Break Line and select **Edit Breakline Properties**.



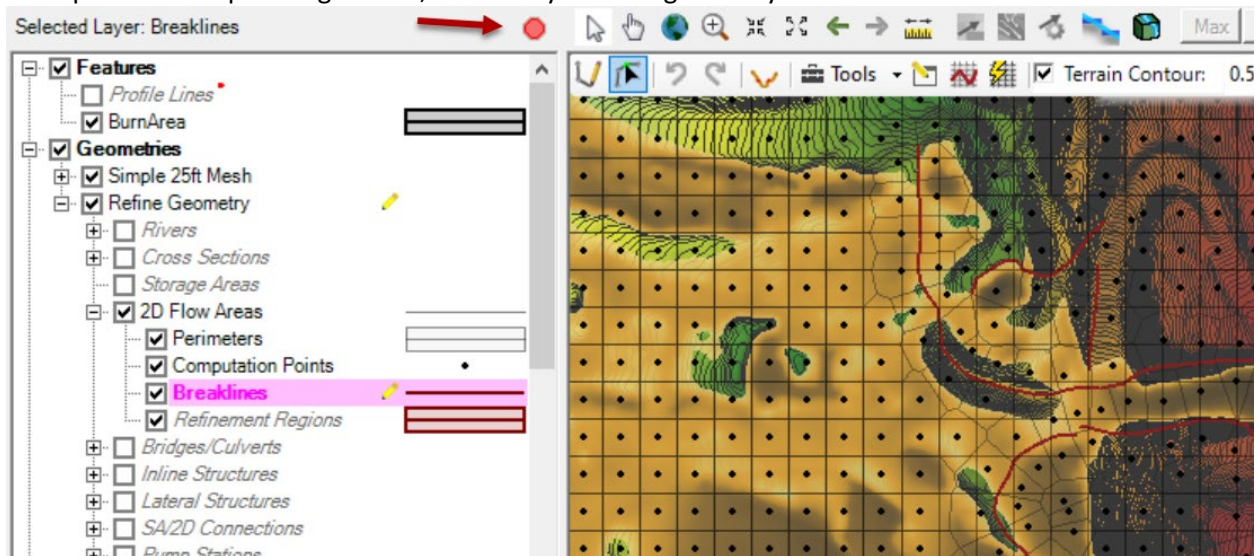
Set the near spacing to the overall spacing (25) and do not any repeats or far spacing.


Right click on the breakline again and choose **Enforce Breakline**.



Repeat this process for a couple ridges or high ground locations where you are having continuity and/or stability issues.

Then press the Stop Editing button, and save your new geometry.



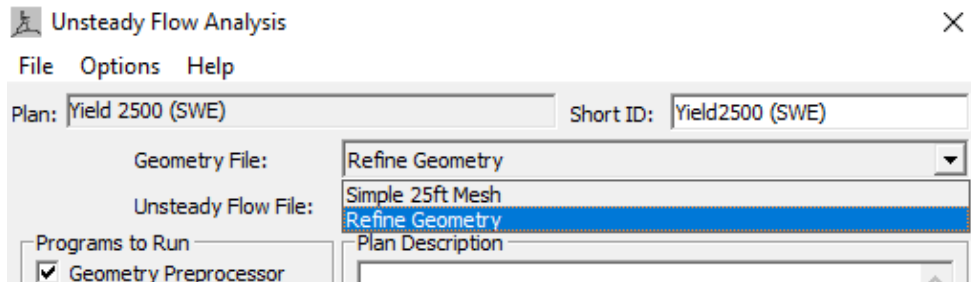
Return to the Plan Editor from the main HEC-RAS window by selecting **Run→Unsteady Flow** or the **Unsteady Plan Button**. 

Save the Plan AS (under the File menu) and give it a new name.

Then select the dropdown box next to **Geometry File**.

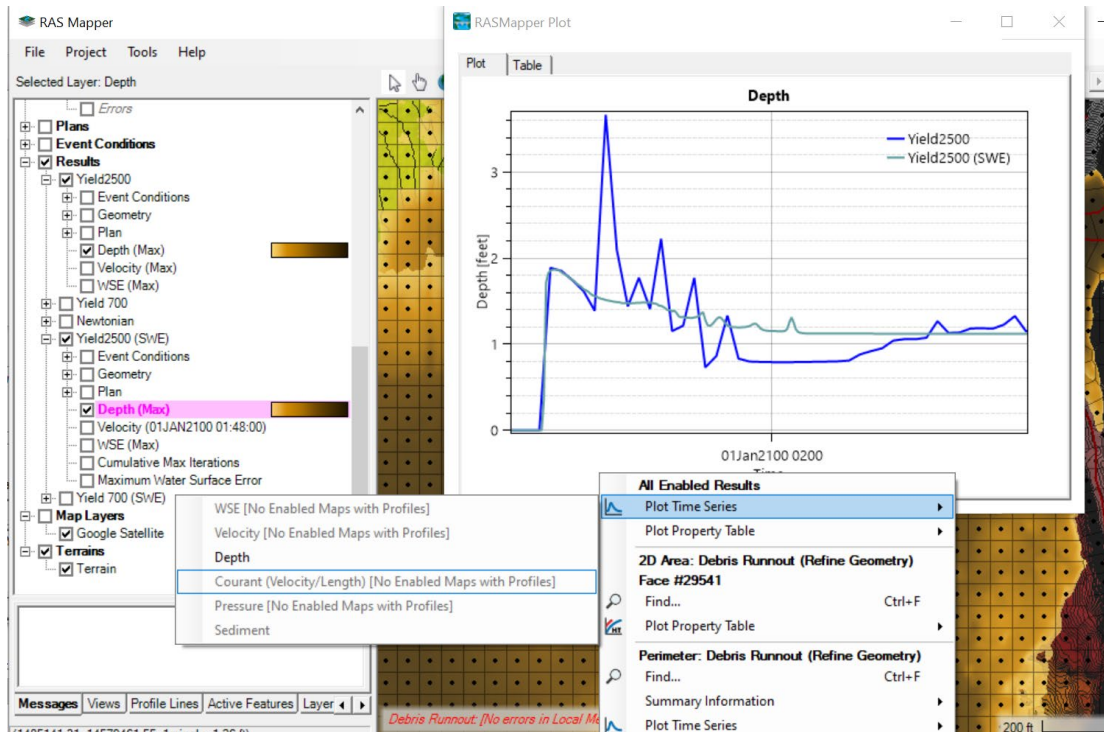
Choose the new, refined geometry you just made.

Compute and look at your new results.



Compare the time series results between the initial run and the refined simulation.

How did it change?



If you have extra time

Sometimes a quasi-1D model can be more stable, where a single cell spans a small, steep channel. Save your Geometry File As, and experiment with this approach.

