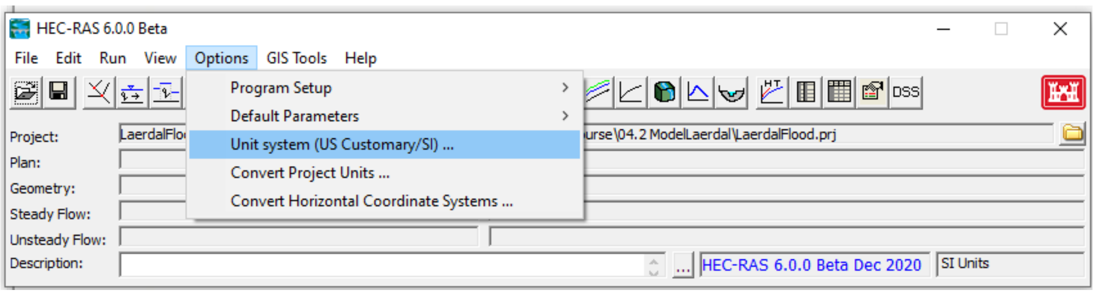
Exercise 5: HEC-RAS 2D

In this exercise we will set up a HEC-RAS 2D model in the municipality of Lærdalsøyri in Norway to simulate a flood with return period of 50 and 200 years.

Getting started

1. Install HEC-RAS 6.0 from this web page: https://www.hec.usace.army.mil/software/hec-ras/download.aspx
2. Choose HEC-RAS 6.0 Beta Update 1 Setup Package Documentation, and Example Data sets
3. Set Region format in the computer to English (United States)
4. Download the package Laerdal.zip
5. Open HEC-RAS, Options-> Unit system-> SI Units, to set Metric System in the program.



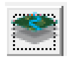
Now we are ready to start our new model, the steps to set up the model will be explained in the following points, I recommend to watch as extra support the following video:

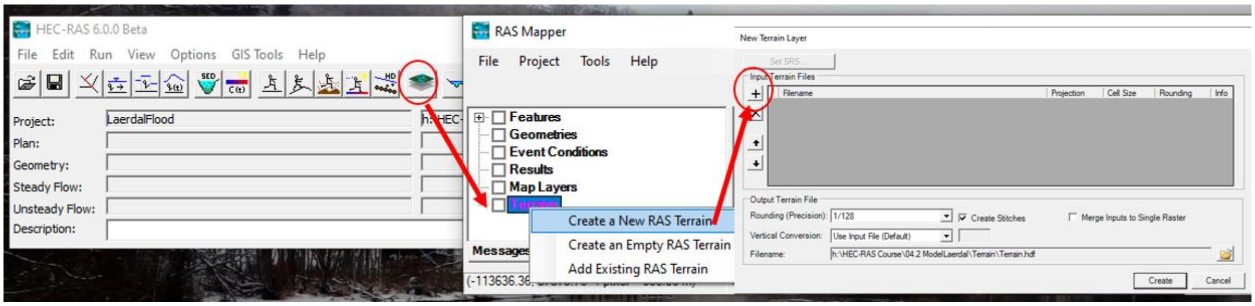
https://www.youtube.com/watch?v=rMOLYunwqU8&t=689s

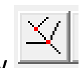
Tasks to report:

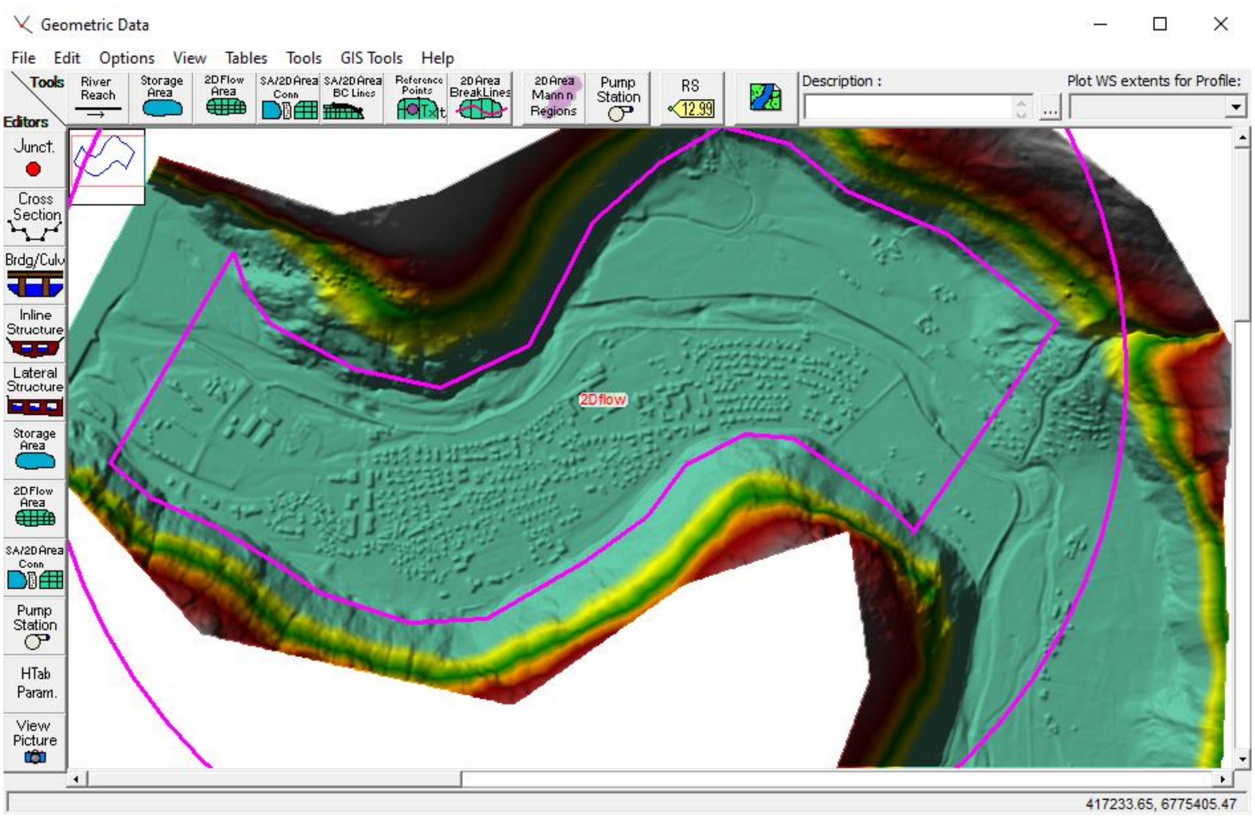
- Screenshot of depth and velocity maps for the 50 and 200 year flood (Part 1 and 2).
- Screenshot of 200-year flood after fixing the geometry as shown in Part 3.
- (Optional) Screenshot of 3D view with the new geometry fixed (Part 4).

Part 1. Simulate 200-year flood Q=920m³/s

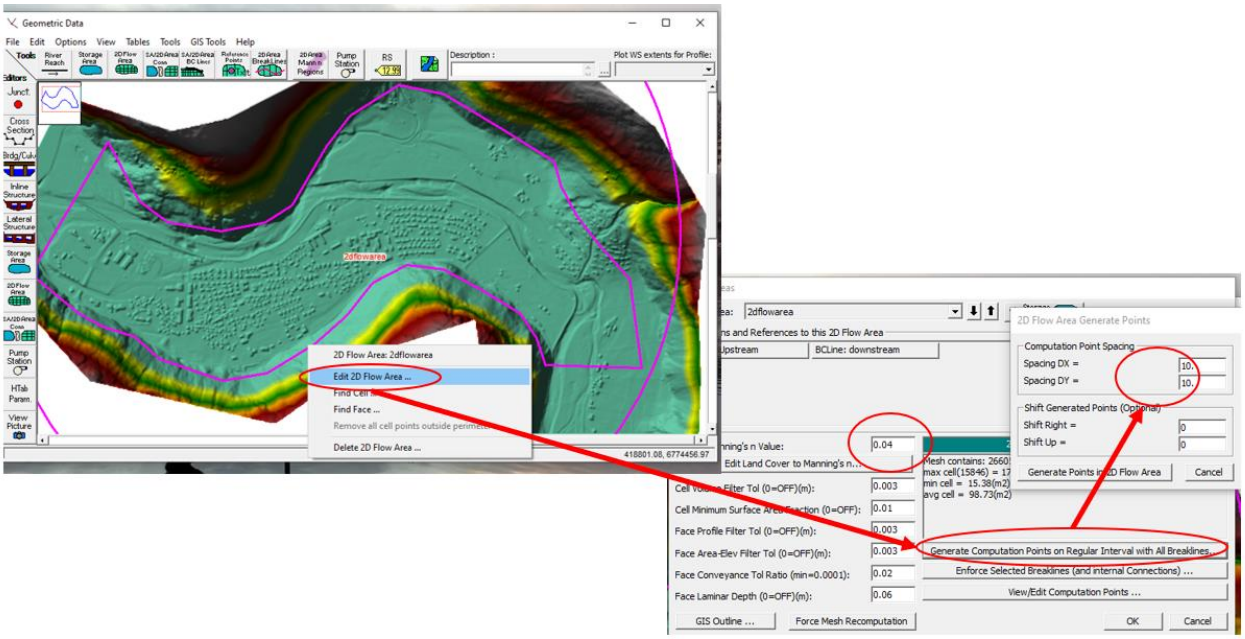
1. Name the project: go to File Save Project As-> (select a folder in your computer and give a title) -> Click OK
2. Go to Ras Mapper  ->Project-> Set Projection-> (look in Laerdal.zip for **25832.prj**)->Ok
3. In Ras Mapper, right click on Terrain-> Create a New RAS Terrain->click on + -> look for Laerdal.tif-> Create



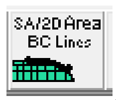
4. Close Ras Mapper and Open Geometry Data window . Click on 2D flow area. Now we can draw the boundaries of our geometry. Give it a name, for example 2Dflowarea.



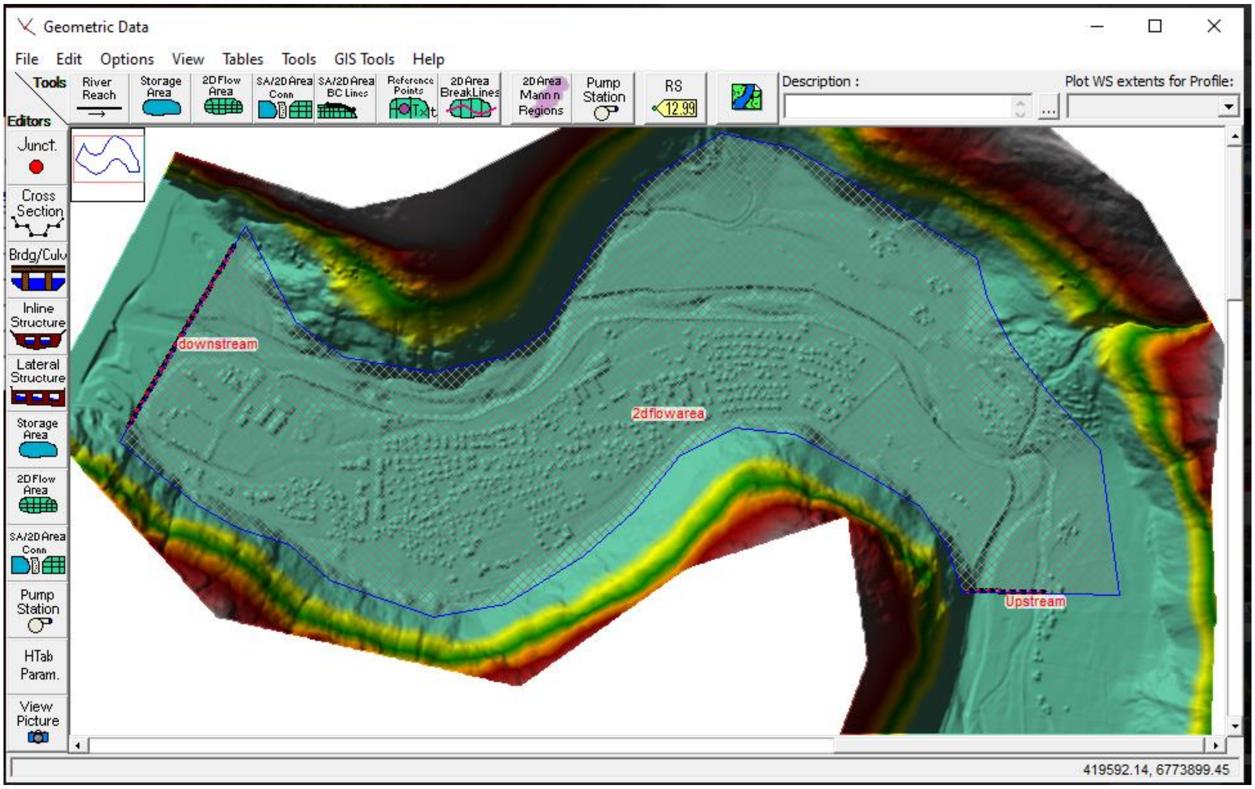
5. Click on the pink boundary -> Edit 2D Flow Area… -> Generate Computation Points on Regular… -> Insert Spacing **DX and DY 10m**.



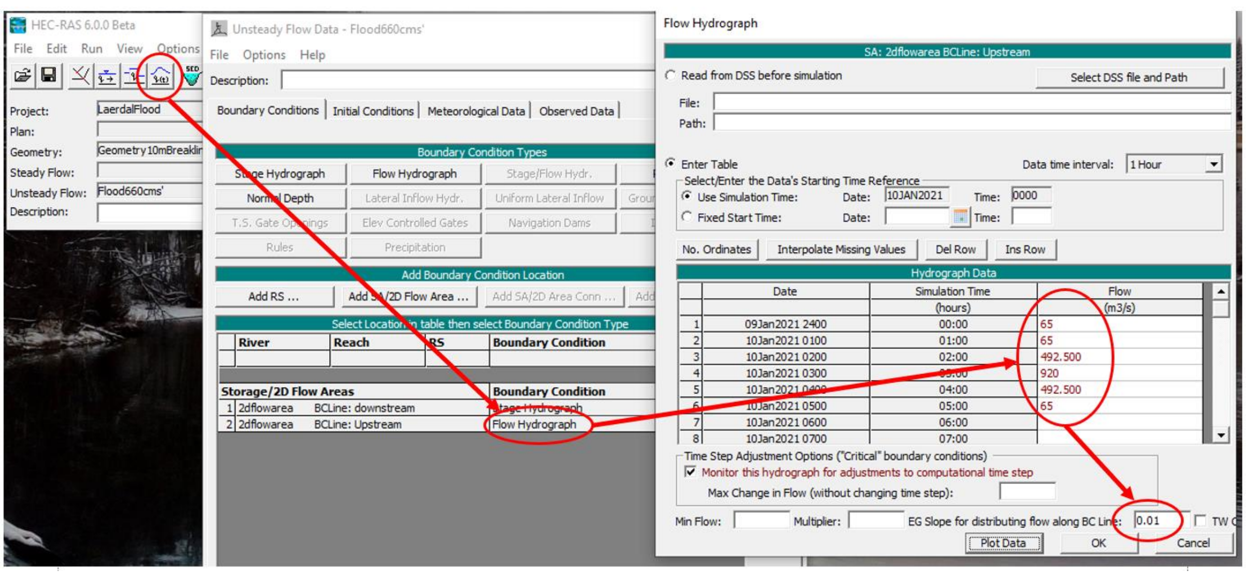
6. IMPORTANT: notice that in this window we can see **the Default Manning's n Value.** In our model we can insert a vectoral layer with roughness coefficient in RasMapper or we can leave the Default manning number. In this case we will leave a Default manning of n=0.04.

7. Click on SA/2D Area BC Lines 

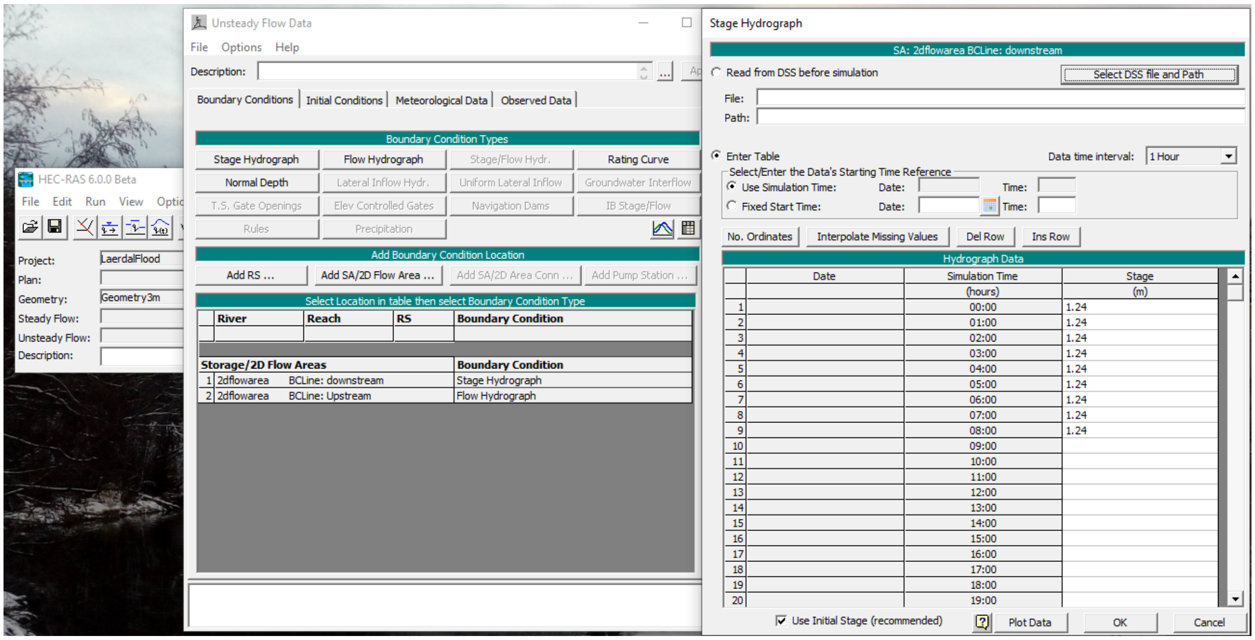
, we will **draw the upstream and downstream boundary conditions** in our model as shown below. Check the video to see how to draw this line: https://youtu.be/rMOLYunwqU8?t=468
Upstream on the right and downstream on the left. File-> Save Geometry Data As-> Geometry10m.



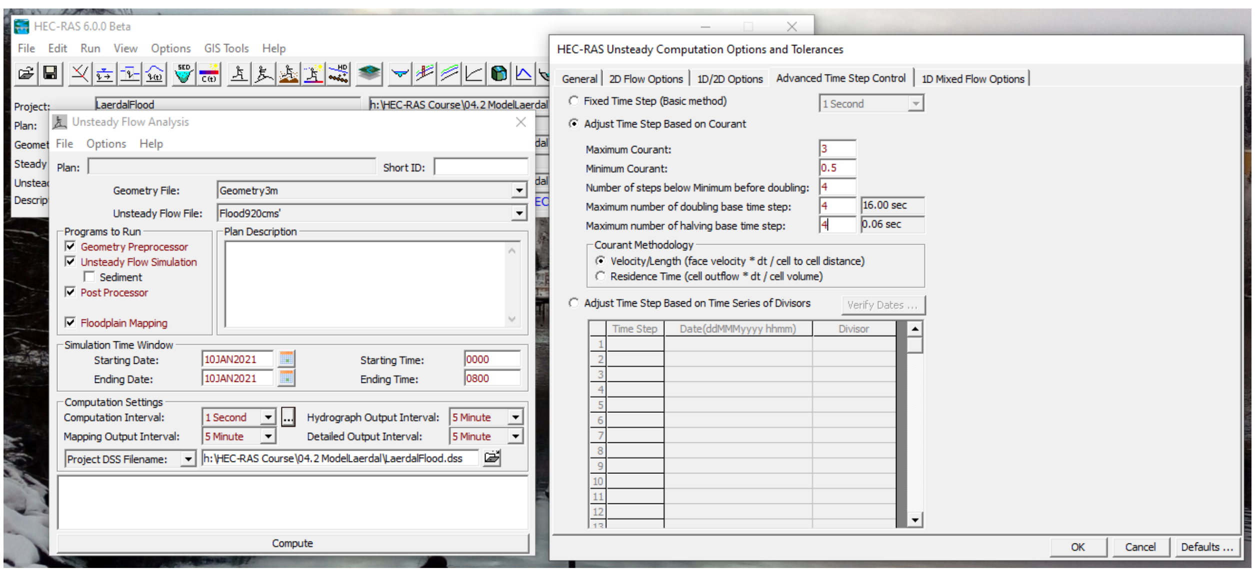
8. Close Geometry window and go to Edit Unsteady Flow Data 
9. Upstream boundary condition-> Flow Hydrograph



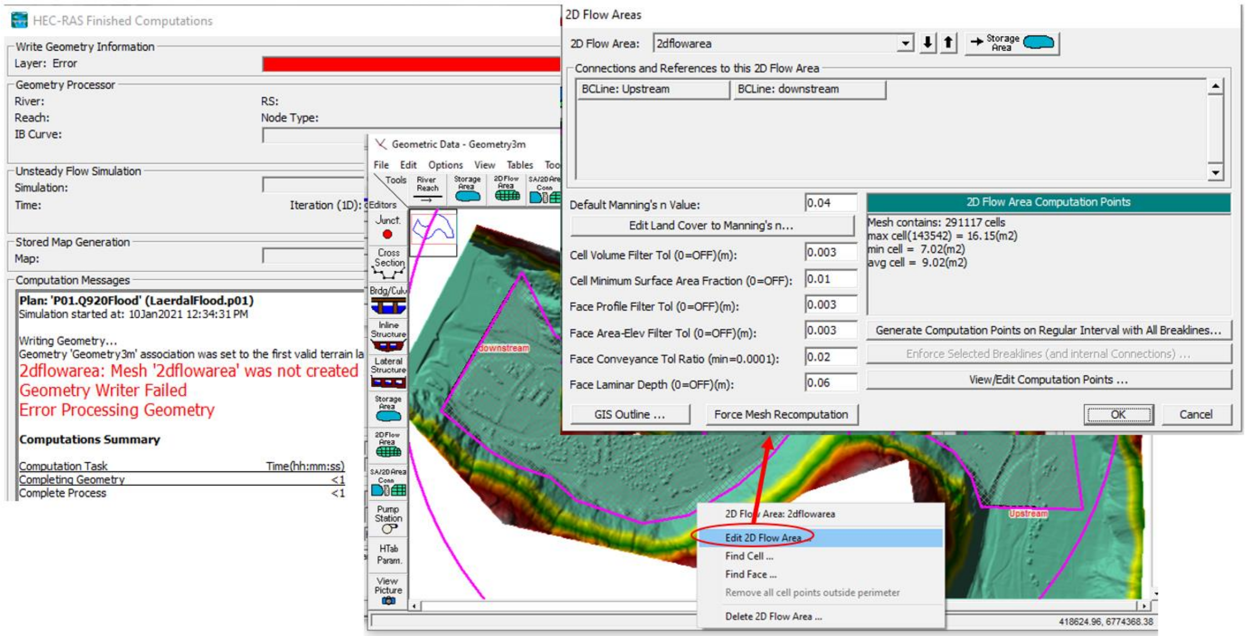
10. Downstream boundary condition-> Stage Hydrograph. We set the maximum average tide 1.24m



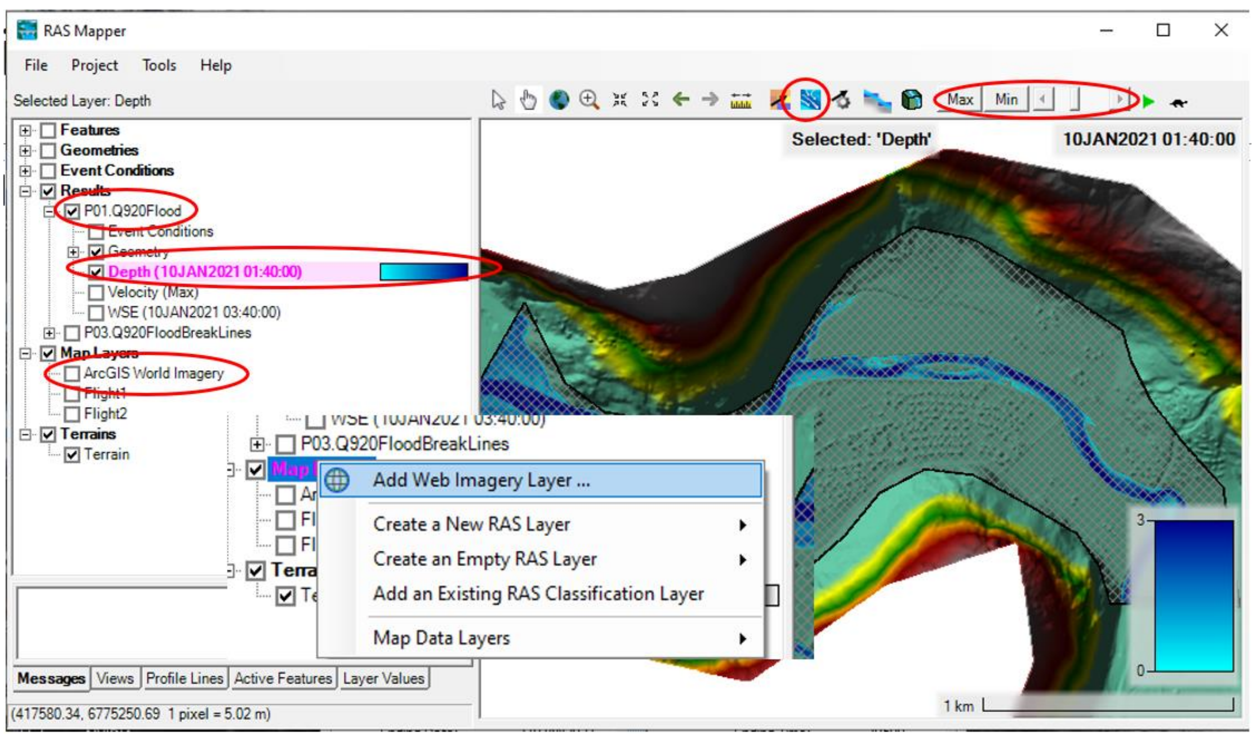
11. File-> Save Unsteady Flow Data-> Give the name Flood920cms
12. Close Unsteady Flow Data and open Unsteady Flow Analysis 



13. File-> Save Plan As-> P01.Q920Flood, press Ok, Shot plan Identifier P01.Q920Flood (same as name). Press Compute 😊
14. CHECK! Common error: “2Dflowarea was not created successfully”. Solution: go to Geometry Data window-> Edit 2D flow area-> Force mesh recomputation



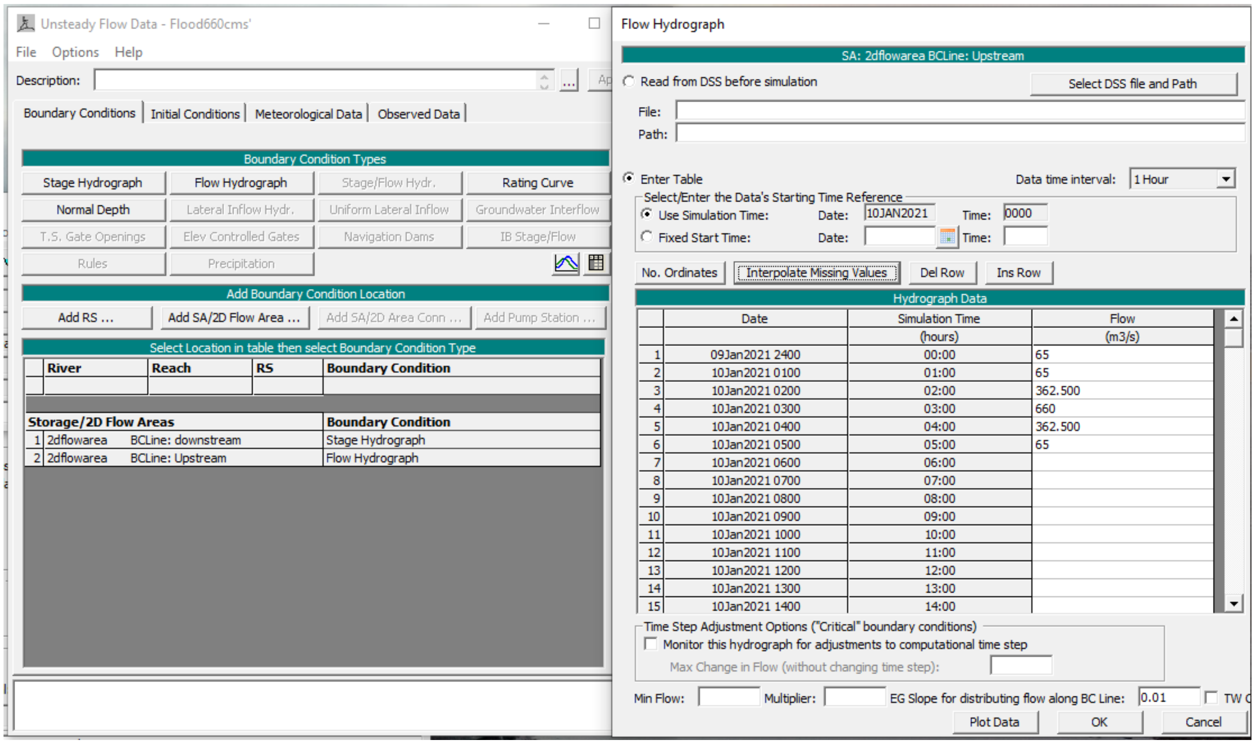
15. Visualizing results. Go to Ras Mapper  ->Results-> Select P01 and the map you want to visualize. Press Depth for example (it has to become pink) and the explore the time window in the up right. You can also add a background map. Right click on Map Layer-> Add Web Imagery Layer… Select ArcGIS World Imagery



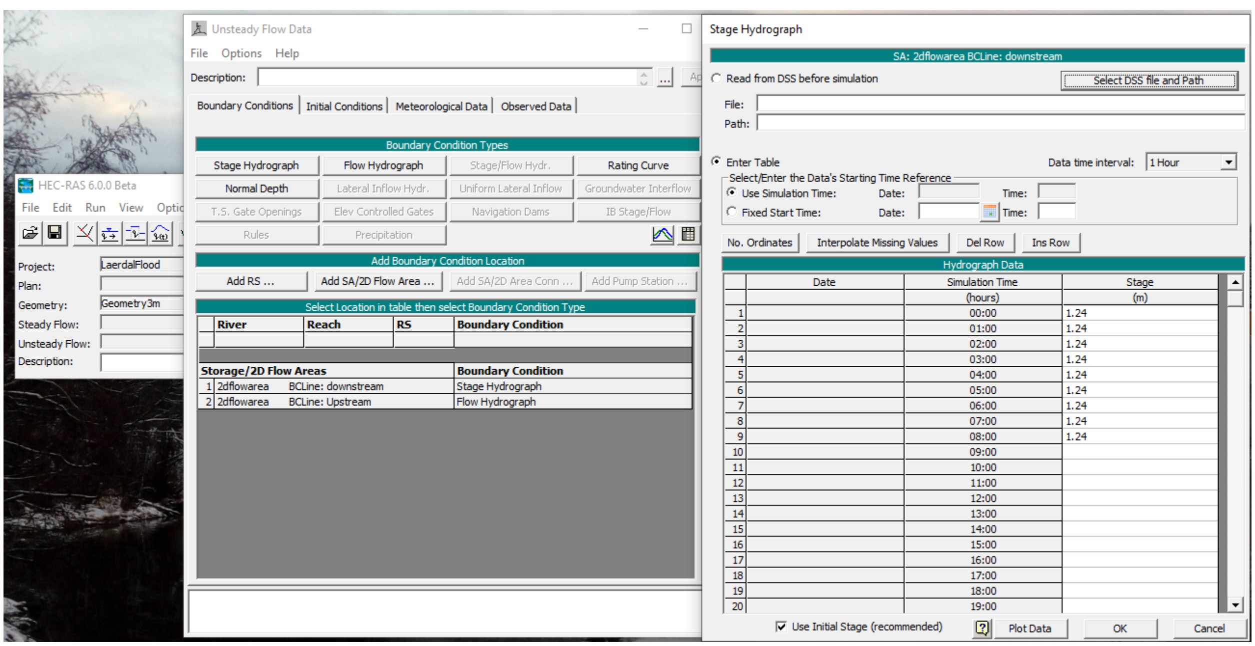
Part 2. Simulate 50-year flood Q=660 m^3/s

Repeat from step 8 in previous part.

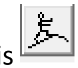
1. Upstream boundary condition-> Flow Hydrograph with Q max 660

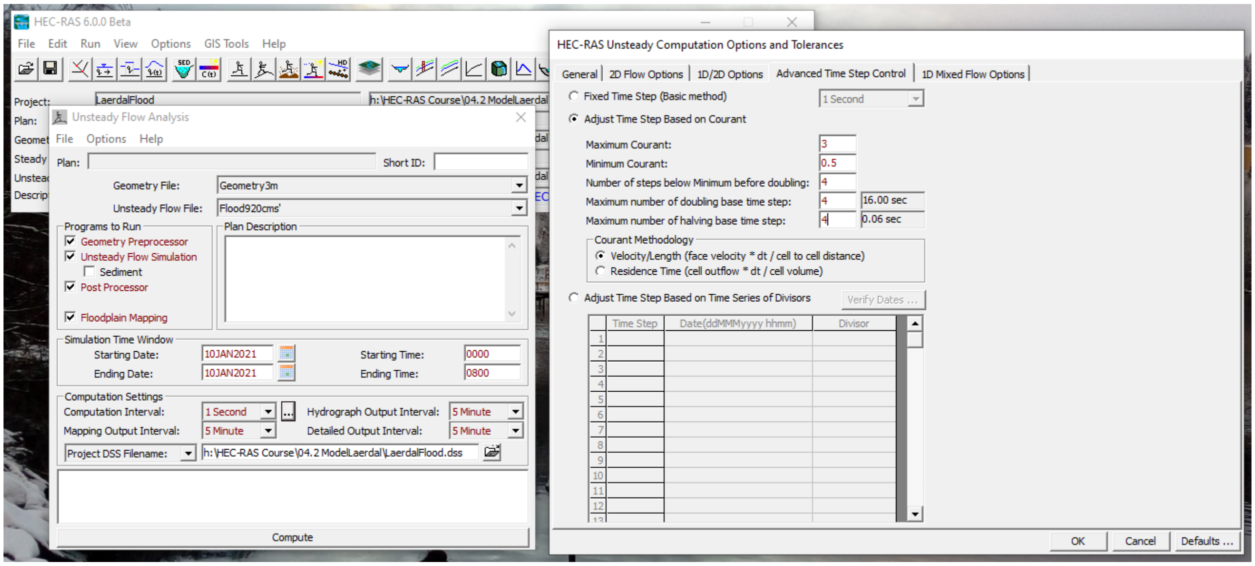


2. Downstream boundary condition-> Stage Hydrograph. We set the maximum average tide 1.24m



3. File-> Save Unsteady Flow Data-> Give the name Flood660cms

4. Close Unsteady Flow Data and open Unsteady Flow Analysis 

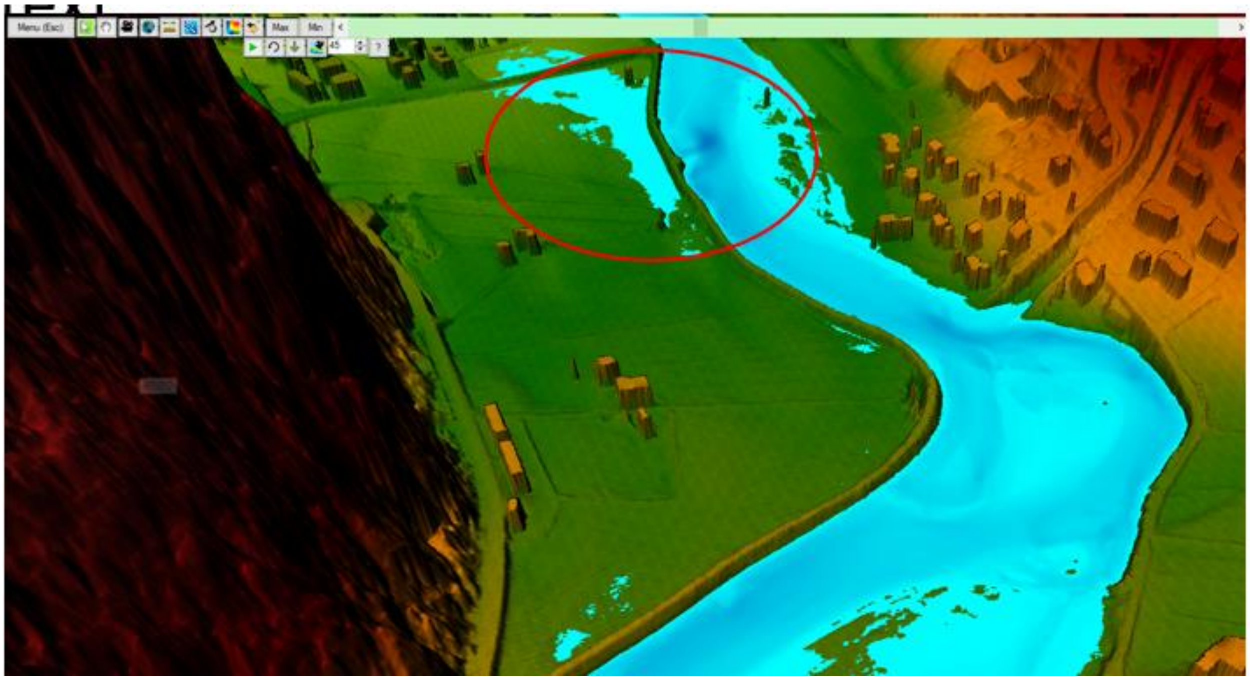


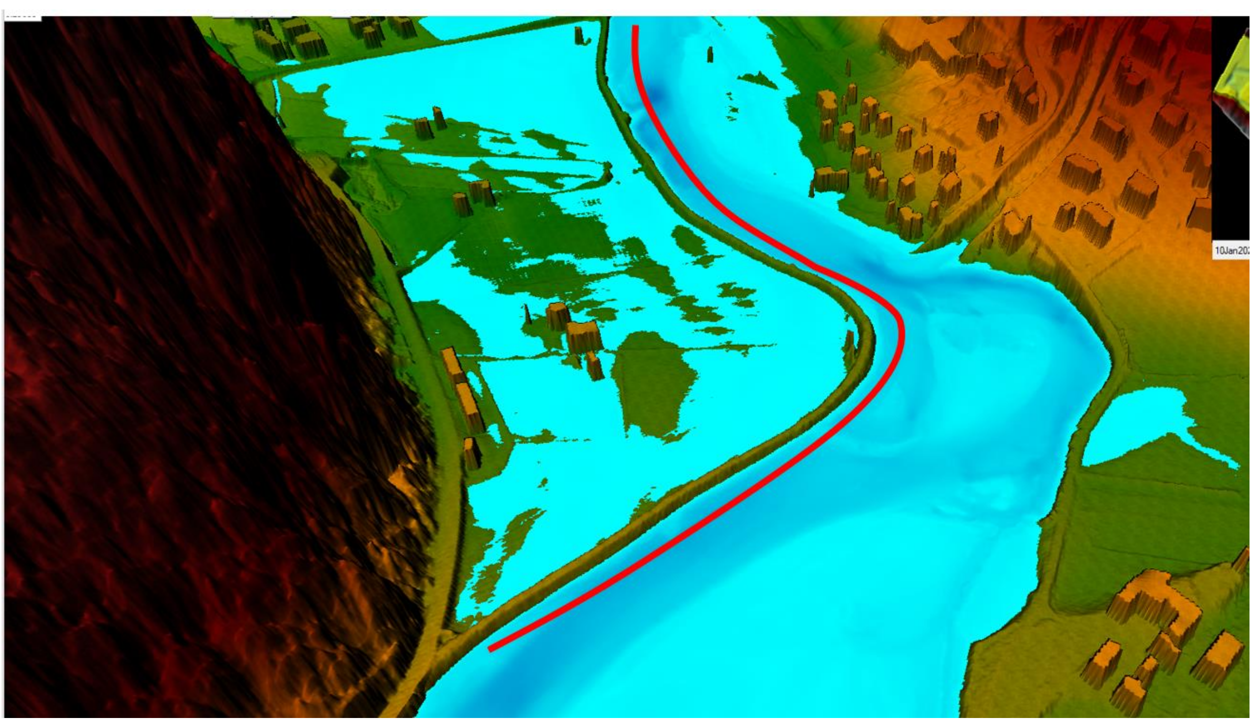
5. File-> Save Plan As-> P02.Q660Flood, press Ok, Shot plan Identifier P02.Q660Flood (same as name). Press Compute 😊

Part 3. Geometry check.

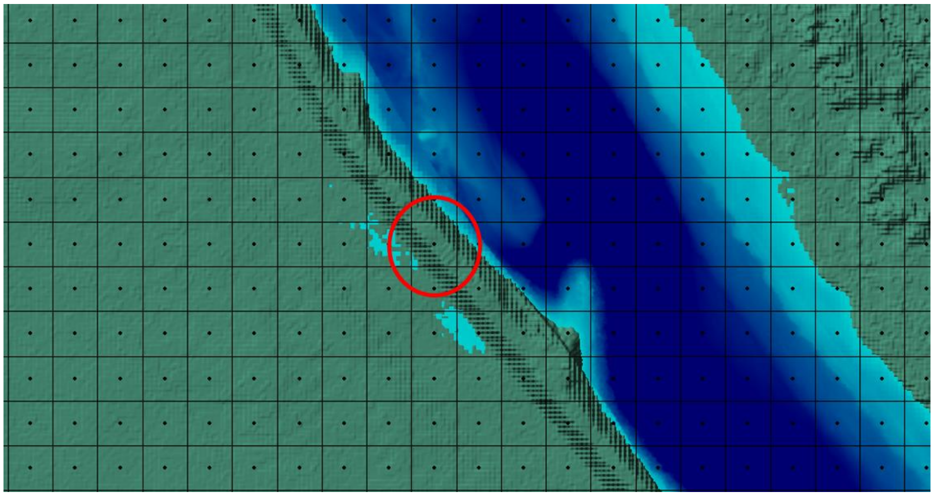
In this part you can learn how important is to check your model with and be critical about the results.

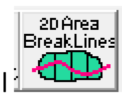
If you look in the upstream part of the model that we have just done the water starts flowing through a wall at the beginning (around time 01:50). Is that possible?!



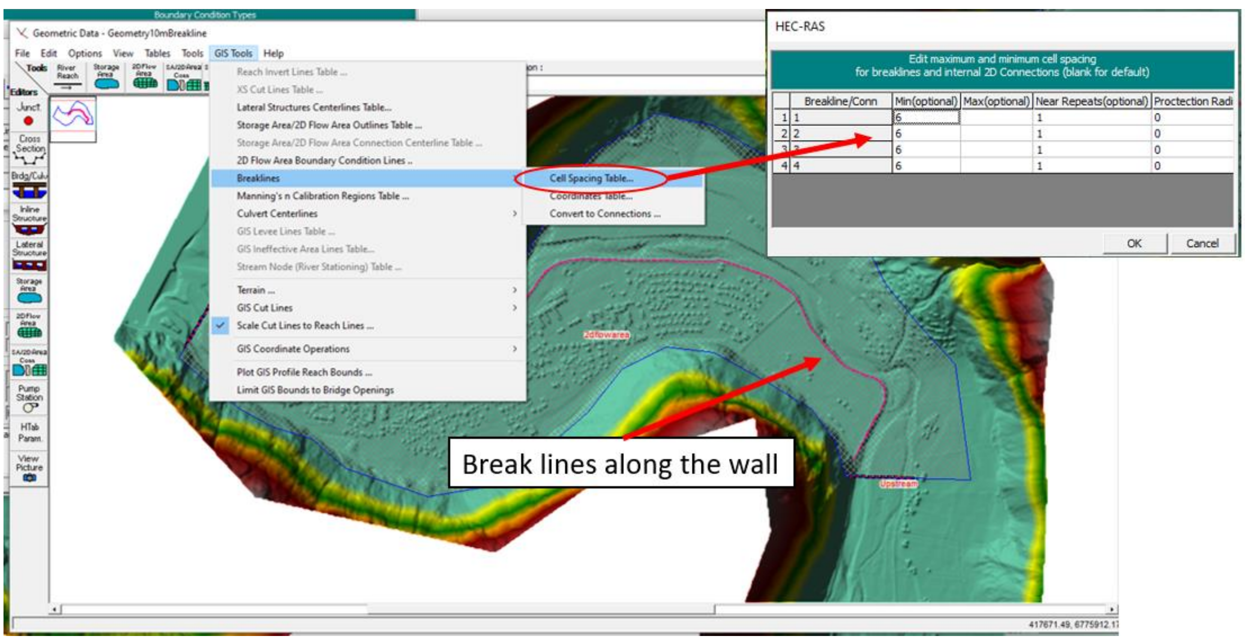


This happens when our cell size is so big that it does not represent the singularities in the geometry. This cell (see Figure below) is so big that it transferring the water from one side to the other one in the wall.



How to fix? Break lines! Go to the Geometry window and draw a Break line along the wall 

, (you can also draw instead of one several because it is easier to do it more precisely).



Part 4 (Extra). Explore 3D View 

This is one of the new features of HEC 6! Open 3D view . Go to Select a Flight Plan, Browse the Flight2.shp, set 45 degrees and press Ok.