Running Unsteady Simulations

The condition for having unsteady states require for you to have three different files additionally to the files you have created already using BlueKenue and FUDAA-PREPRO.

1. Boundary condition file with transient discharges (hydrographs) to be imposed in the upper boundary of the domain. This file has extension liq.
2. Boundary condition file specifying stage discharge curve –rating curves—for the lower boundary of the domain. This file has extension txt.

Once the geometry for a steady state as well as the boundary condition file has been generated, they should also be used for unsteady conditions, if including a hotstart file. Moreover, for an unsteady simulation, new requirements will be added as keywords in the steering file such as:

- STAGE-DISCHARGE CURVES: determine in which of the boundaries will the stage discharge information will be included if having multiple outputs on the domain.
- BOTTOM SMOOTHINGS: Smooths the bottom of the topography based on the bottom feature between 0 to 10, in increasing manner.
- TURBULENCE MODEL: Depends on the type of wave propagation and gravitational effects. Leave as 3, being K-Epsilon.
- TREATMENT OF NEGATIVE DEPTHS: Negative depths will propagate through the model in the beginning of the simulation because of the amount of water that is entering the domain. There are 3 options: 0 is no treatment, 1 is smoothing and 2 is flux control. Leave as flux control.
- TREATMENT OF THE LINEAR SYSTEM: Value based on the type of propagation that is done by the solver (see SOLVER) which can be 1: coupled or 2: wave equation.
- VARIABLE TIME STEP: important to manage instabilities. The variable time step will adjust so that a specified Courant number (CFL condition) can be achieved.
- DESIRED COURANT NUMBER: CFL condition between 0 and 1 (preferably 0.5) for the convergence of the partial differential equations imposed by the solver. It can only be included when the variable time step option is set as true.
- CONTINUITY CORRECTION: it applies corrections on the points in which the continuity equation has not been solved. Boolean.

- SUPG OPTION: It is an array option which is applied to 4 different variables. It is set for the solution of the upwind scheme (Finite Volume) applied on velocities u and v, water elevation H, tracers, K and Epsilon for turbulence model. The options are 0: no upwind scheme, 1: upwind scheme with classical SUPG method, 2: upwind scheme with modified SUPG method. Usually, you use 2 for velocities, tracers and K-Epsilon and 0 for water elevation because the modified SUPG is more accurate for Courant numbers that are less than 1.
- SOLVER: Solver used depending on the nature of the problem. It has 3 different keywords; SOLVER: hydrodynamic propagation, SOLVER FOR DIFFUSION OF TRACERS: diffusion for tracers (contaminants), SOLVER FOR K-EPSILON MODEL: solving turbulence model system. The keywords may have values between 1 and 9, which would correspond to different possibilities. For normal propagations, use 1: Conjugate gradient method.
- MASS-BALANCE: this Boolean should be set to true in unsteady simulations because there is a hydrograph imposed in the domain at each time step.
- MASS-LUMPING ON H: used for the solution of the linearized system when propagating the solutions. The matrix is diagonalized and simplified so that the computation of each of the steps is reduced considerably.
- PRESCRIBED ELEVATIONS: As a difference with steady state simulation, when using hotstart files there is no need to prescribe elevations because TELEMAC will take it directly from the PREVIOUS COMPUTATION FILE and the STAGE-DISCHARGE CURVES FILE. If not using hotstart, impose the elevation values separated by the colons (;).
- PRESCRIBED FLOWRATES: same as previously, if using hotstart file there is no need to prescribe any initial condition because TELEMAC will ready it from the PREVIOUS COMPUTATION FILE and the LIQUID BOUNDARIES FILE.

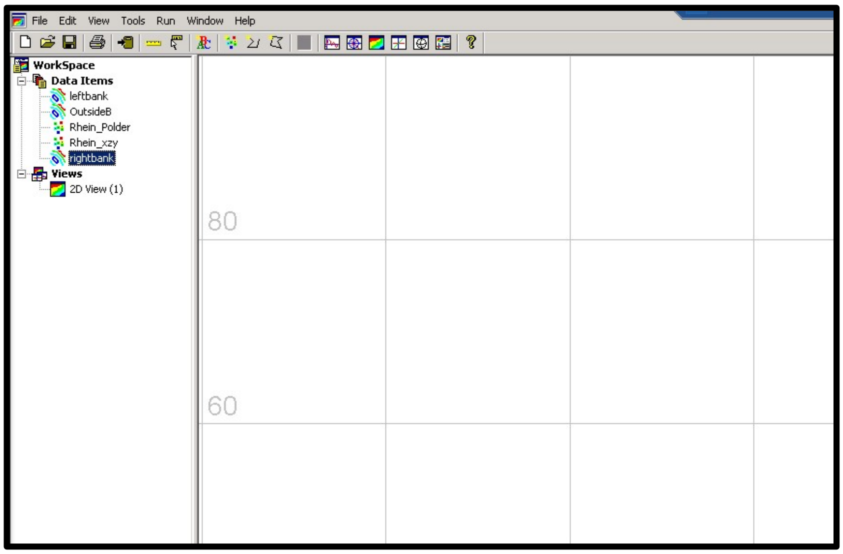
The once all the information has been made available to the steering file—including liquid boundaries, stage discharge curves—then the simulation is run just as with a steady state. Go to the directory where the files are located and use

Telemac2d.py nameofsteeringfile.cas

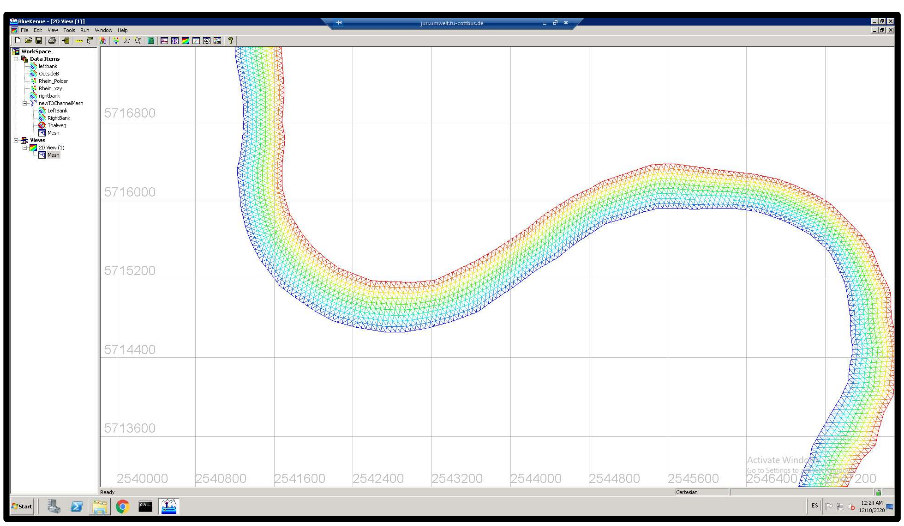
Dam break and control sections

When including dam break or dyke breaches, the geometry of the domain must be regenerated. This is because the geometry will be newly created based on hardlines or meshes that impose geometry features that change hydrodynamic properties. In the following exercise, the creation of a dyke will be done using both meshes and hardlines to be imposed within the domain.

Open BlueKenue, import the boundary t2s file, the right and left banks of the river, the coordinates for the polder and the river domain.

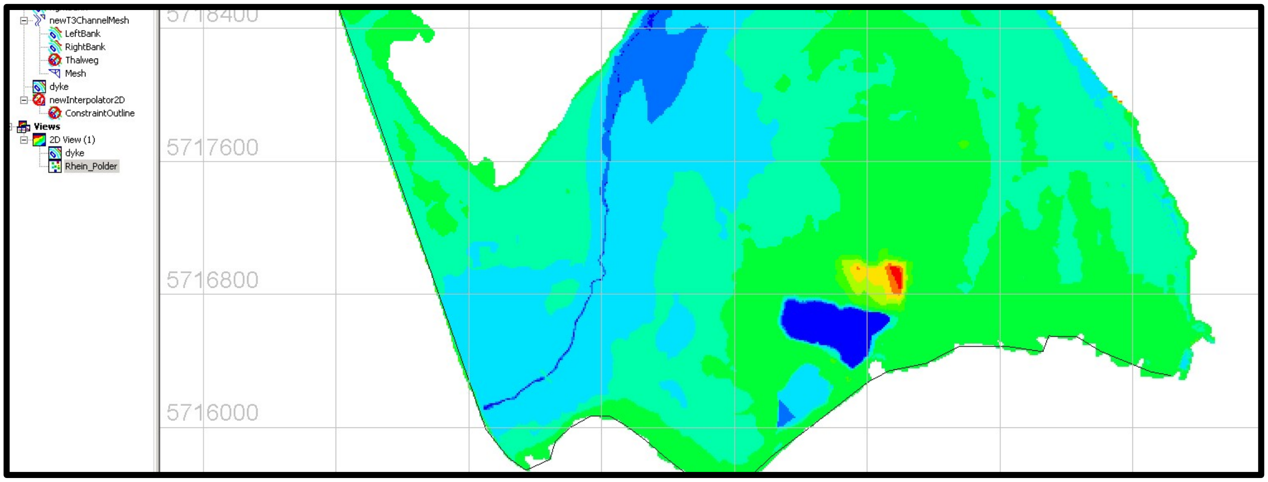


Follow the same steps for creating a mesh for the river, selecting T3 channel mesher with node count across the river of 10 and 50 of length.

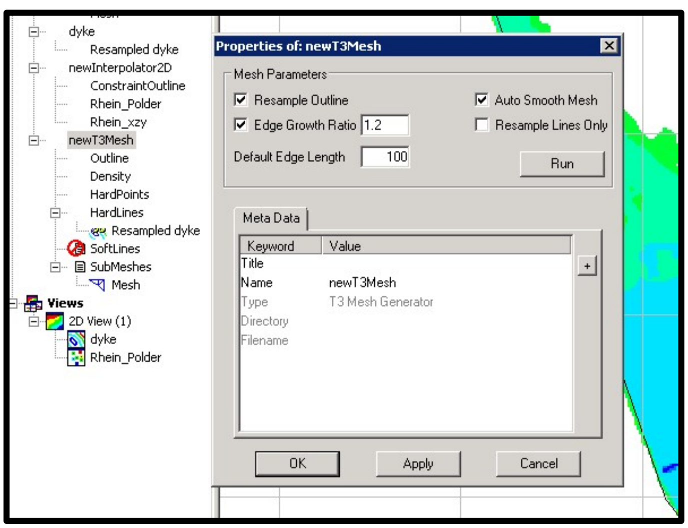


Create dyke using hardlines

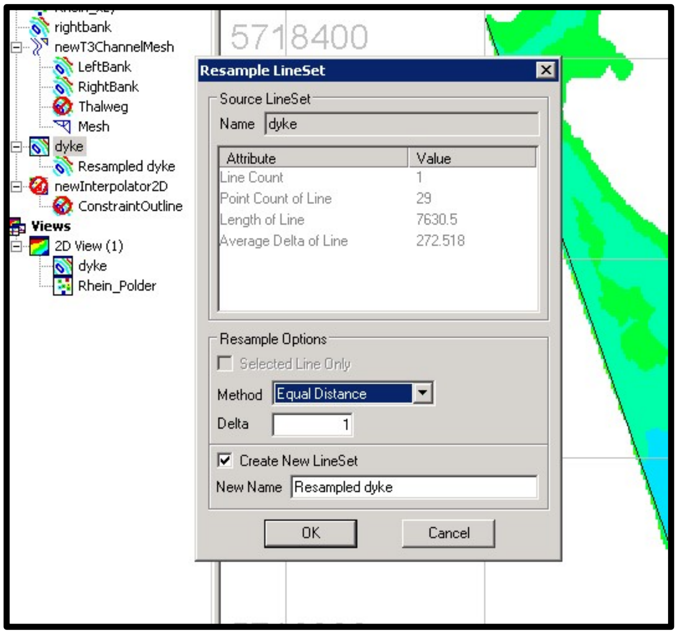
Create or import the hardlines of a dyke by using open lines.

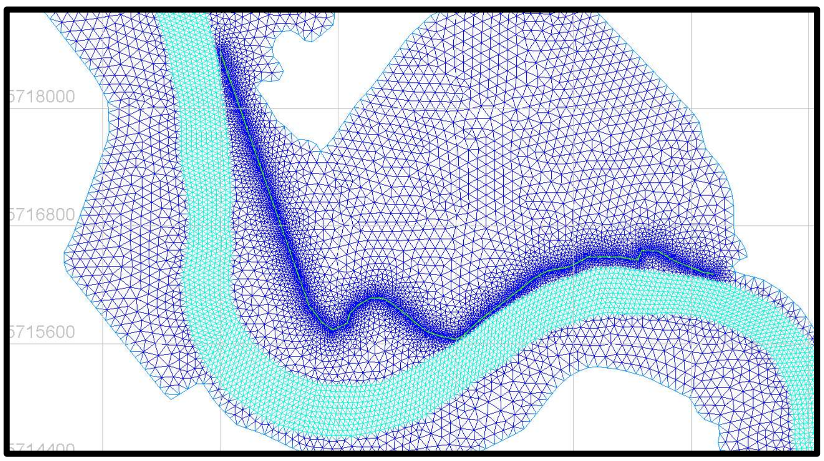


Create a new T3 mesher for the domain. Resample the dyke line so that it is spaced every 10 meters. Right click on the line -> resample -> select Equal distance and put 10 meters. Include it in the new T3 mesher as hardline, include the channel mesh as a submesh and finally, the boundary of the domain as the outline.

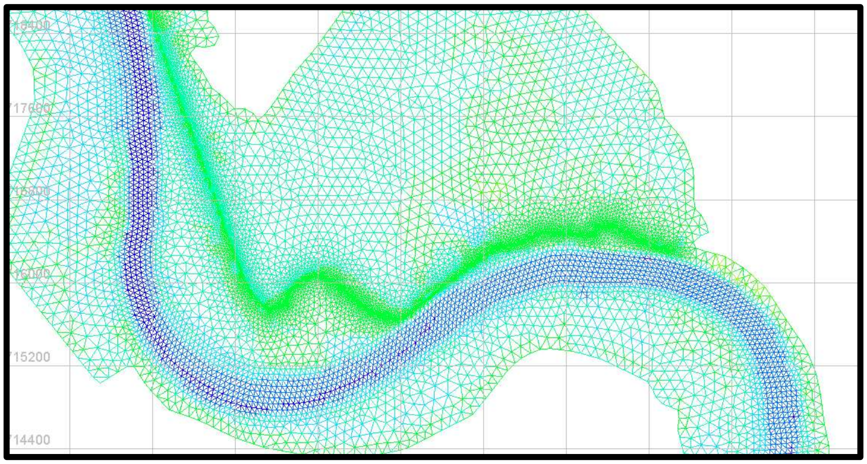


The lines are just a visual representation of the dyke right now and cannot be used for anything else but that. Considering this, we need to impose the bathymetric data within the line using the readily available coordinates.

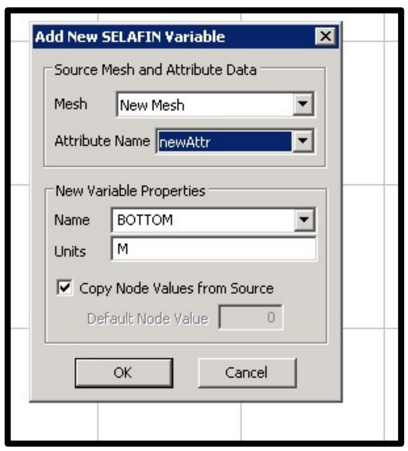




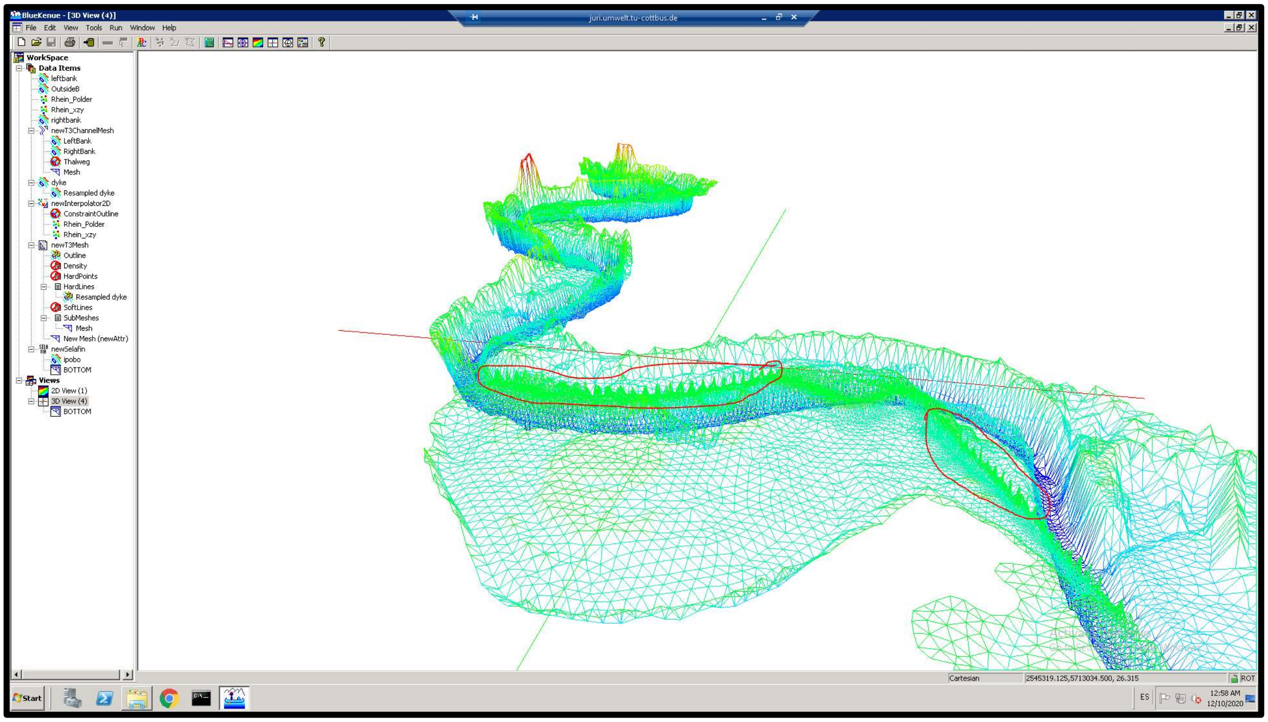
The mesh will be much finer around the edges of the dyke line, and coarser everywhere else. Create a 2D interpolator and include the points of the rhein and the polder and map these values upon the newly created mesh.



Create a new SELAFIN object, add a new variable called BOTTOM selecting the new mesh, with variable properties called BOTTOM and copy the node values from source.

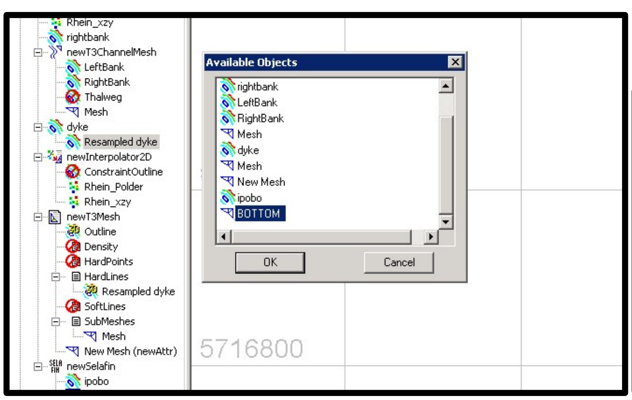


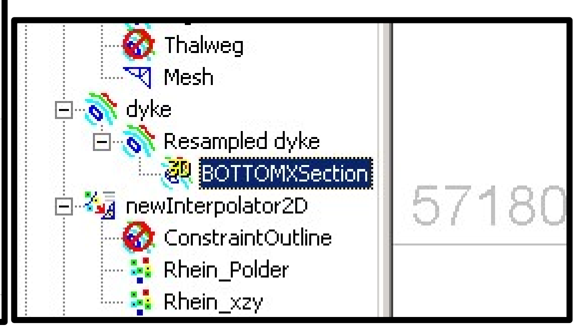
Map the BOTTOM mesh with the 2D interpolator. Drag this into a 3D viewer to see the results of the interpolation (quick way to know the bottom is actually correctly mapped!).



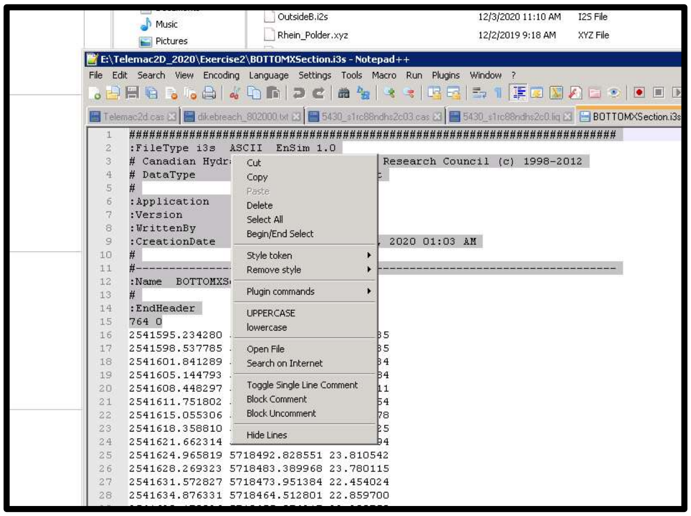
There is a slight issue with the hardline just drawn, because the resampled line interpolates the elevations based on the polder and domain points, it cannot take into account real elevation values. For instance, the elevation of the dyke at all points is around 22.8 meters. An easy fix to this, is to extract the dyke line and import it once again as t3s object.

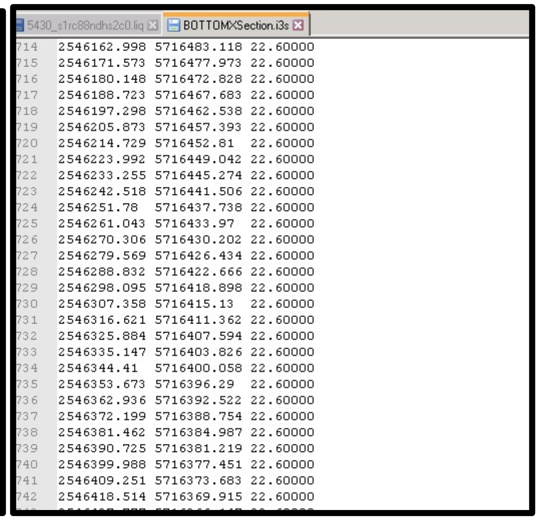
Select the resampled line and map the newly created bottom on it.



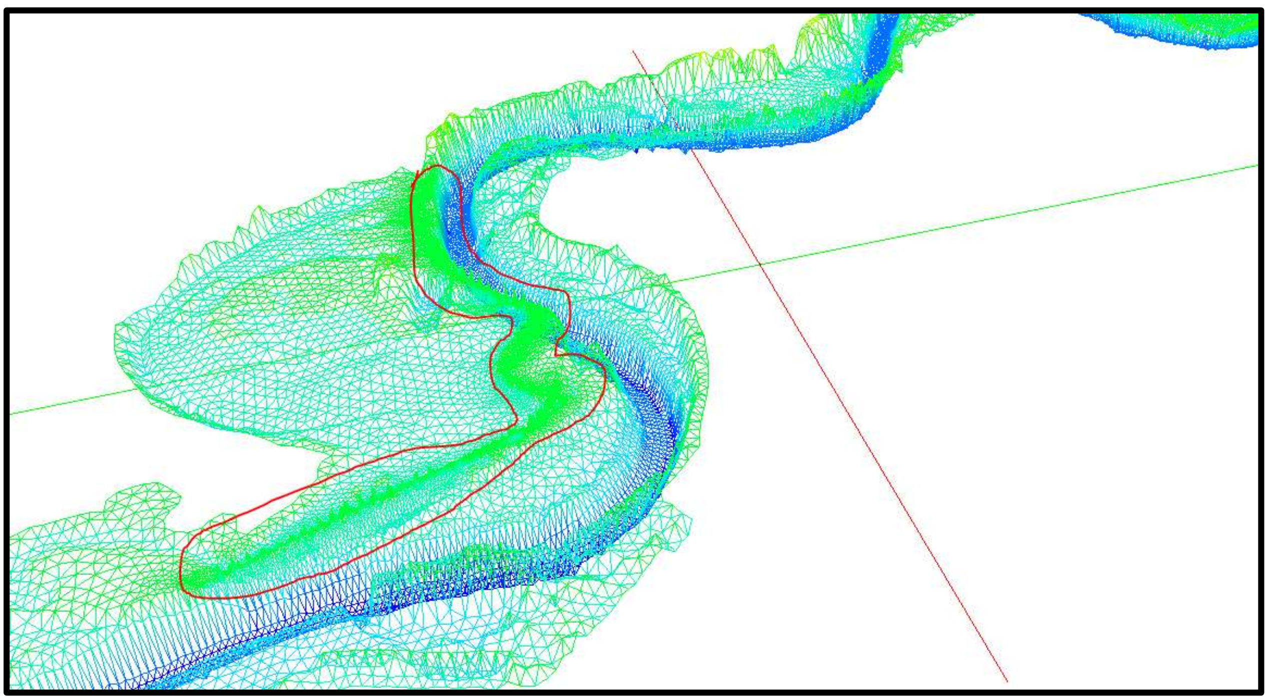


Save the newly created line in your local folder. Using the text editor, delete the information above the coordinates. Using excel or some other spreadsheet—even the same editor—change the elevations of each point and put them as 22.60.





Change the extension from i3s to xyz. Import this new file into the BlueKenue interface once again. Create a new mesh using the T3 mesher. Create a new 2D interpolator that includes the newly added points and repeat the same process.



Same as before, establish the boundary conditions using BOTTOM as the mesh, with flow in the upper boundary and free tracer and height in the lower boundary with free tracer too.

<u>Dyke breach</u>

The dyke breach is based on the newly created mesh and it is done by literally by specificying nodes and coordinates where its supposed to happen. It should be specified in a different file with new keywords once again.

```
# Number of breaches
1
# Bandwidth of the polyline defining the breach
80.0
# Breach definition
# Option for the breaching process
2
# Duration of the breaching process (0.0 = instant opening)
0.0
# option of lateral growth (1= bottom lowering, 2=dike opening by widening)
2
# Final bottom altitude of the breach (appr. Height of the land behind the dike at this location)
22.0
# Control level of breach (breach exist if water level exceed this value)
22.6
# Number of points of the polyline
5
# Description of the polyline
2544280.750 5715852.500
2544300.500 5715871.500
2544320.000 5715890.500
2544341.000 5715908.000
2544362.000 5715925.500
```

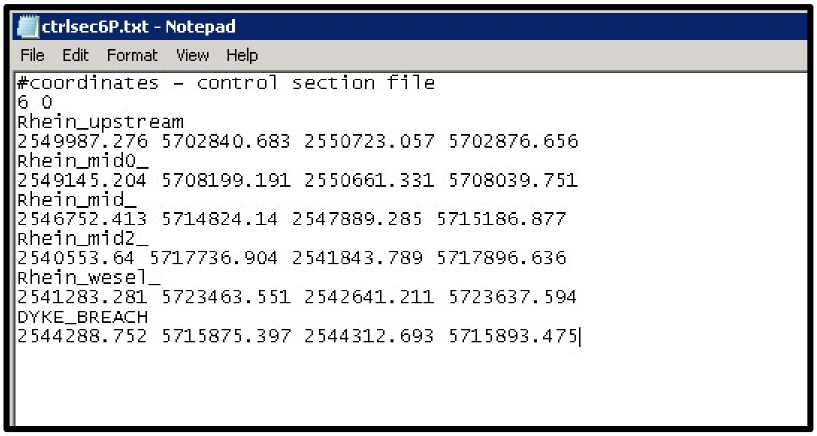
Depending on the number of points that you have on your resampled lines, you can select the width and coordinates of the location. Recall that the width comes as well based on the length of your resampling. If considering a dyke breach, in the steering file include the following keywords:

- BREACHES DATA FILE: breach data with coordinates. txt.
- BREACH: Information about breach happening. Boolean.

Sections

TELEMAC allows for control sections to be included either as points within the mesh or coordinates in the domain. The result information for each sections are the instantanous flow rates and cumulated positive and negative flow rates.

The section input file is similar to the dyke breach with specific formats depending if the info included is for a point or coordinates.




```
#coordinates - control section file
6 0
Rhein_upstream
2549987.276 5702840.683 2550723.057 5702876.656
Rhein_mid0_
2549145.204 5708199.191 2550661.331 5708039.751
Rhein_mid_
2546752.413 5714824.14 2547889.285 5715186.877
Rhein_mid2_
2540553.64 5717736.904 2541843.789 5717896.636
Rhein_wesel_
2541283.281 5723463.551 2542641.211 5723637.594
DYKE_BREACH
2544288.752 5715875.397 2544312.693 5715893.475
```

For specifying sections, include the following in the steering file:

- SECTIONS INPUT FILE: the file that contains info about the sections. Txt
- SECTIONS OUTPUT FILE: name of the location for the final results for each section per time step.

Once this is done, run telemac as usual.