TELEMAC 2D TUTORIAL – CASE STUDY: RIVER RHINE

Authors: Alp Koyulhisarli – Carlos Erazo

Table of Contents

1. INTRODUCTION

In this tutorial, steady and unsteady flow scenarios for the river Rhine and Mehrum polder will be investigated using Telemac 2D. In the beginning, a steady-state simulation will be modelled to familiarize with the software. Afterwards, a dyke breach scenario will be created, and an unsteady simulation will be completed. To complete the simulation, BlueKenue, FudaaPrePro, and the command prompt of TELEMAC 2D are operated.

In this tutorial, a part of the river Rhine is analysed by taking Ruhrort as upstream and Wesel as downstream.

2. INSERTING THE DATA

For this case study, coordinates with the height of each point along the river have already been provided in the project folder.

From “File → Open”, the coordinate file (Rhein_xyz.xyz) can be found by selecting “All Files(*.*)” from the right bottom corner of the “Open” window.

After adding the data, drag it to the 2D view to visualize the coordinate file (Figure 1). To open map legend, right-click on the “Rhein_xyz” 2D view and select “Properties → Colour Scale → Show Legend”. The legend can be modified from that window.

Add the polder data (Rhein_Polder.xyz) by using the same procedure.

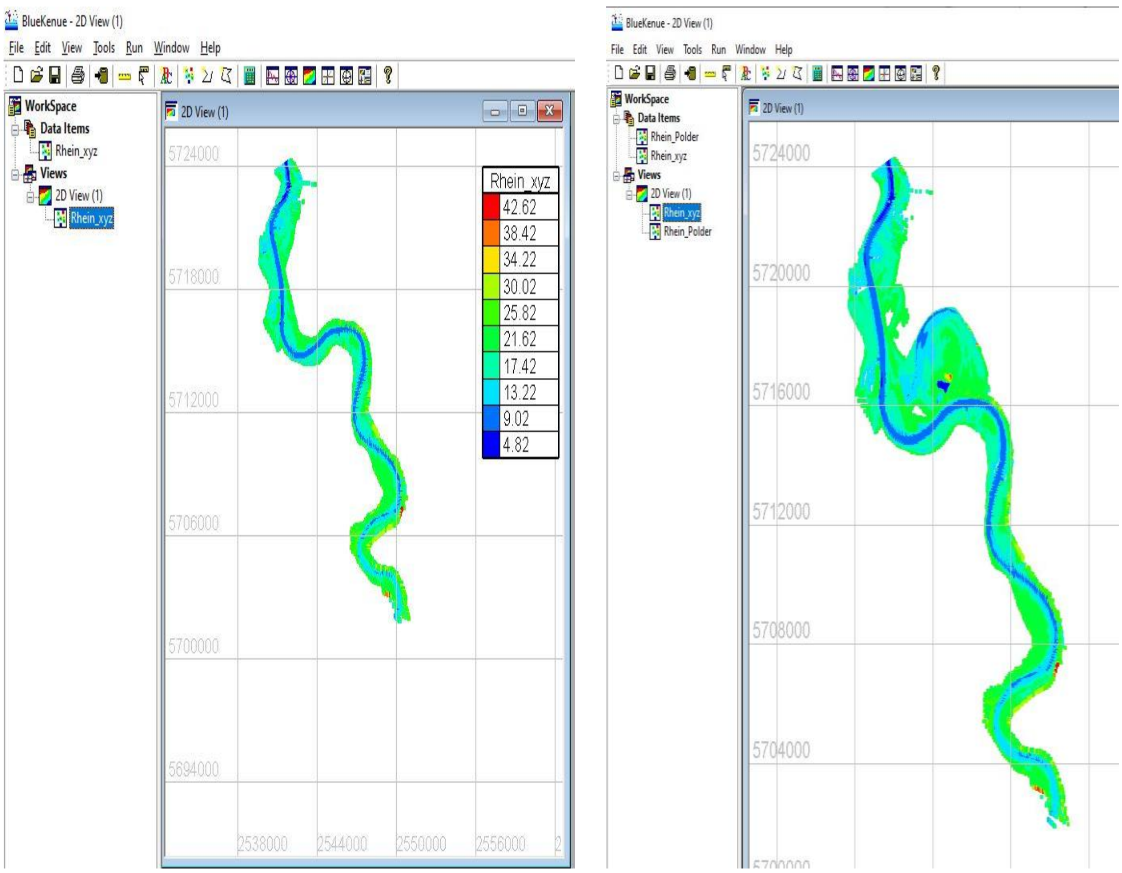


Figure 1. Rhein River and Mehrum Polder coordinate files

3. DRAWING RIVER BANKS, OUTLINE OF THE STUDY AREA, AND DYKE

To draw the channel banks, click “New Open Line” from the toolbar. While determining the banks, zoom in to the coordinate file and try to draw the line over the light blue dots as much as possible, as shown in Figure 2 (Keep in mind that the dark blue dots indicate the flow since they are the deepest points according to the legend. The lines should not intersect them). When there is no light blue point in the cross-section, select a point where dark blue points end and another colour starts. After finishing drawing a bank, hit the “Esc” button on your keyboard and name the drawing as which bank you draw and click “OK”. This process should be completed for each bank separately. After saving both bank drawings (clicking the floppy disk icon on the toolbar), drag them to the 2D view (Figure 3).

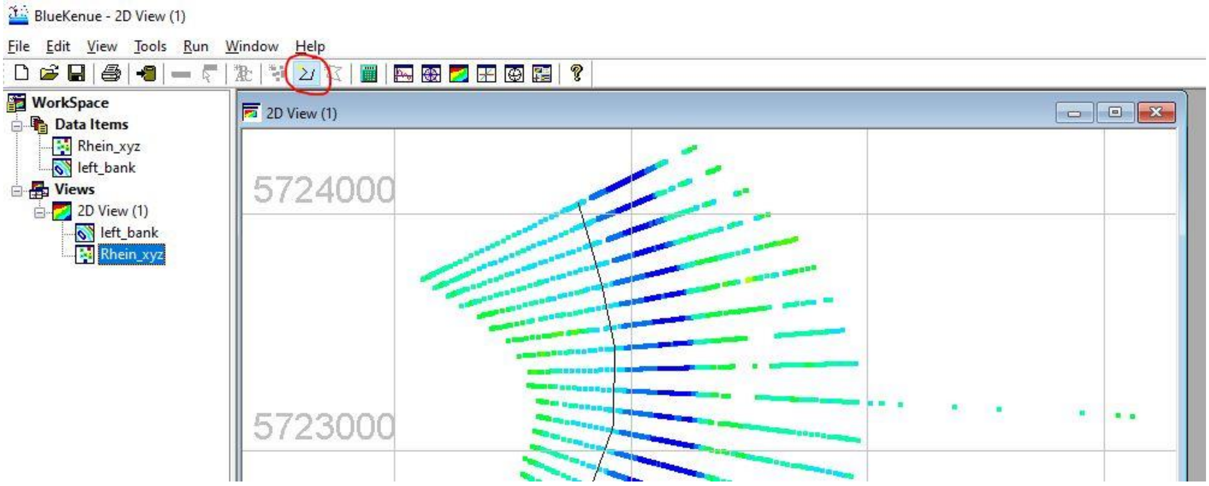


Figure 2. Drawing the left bank

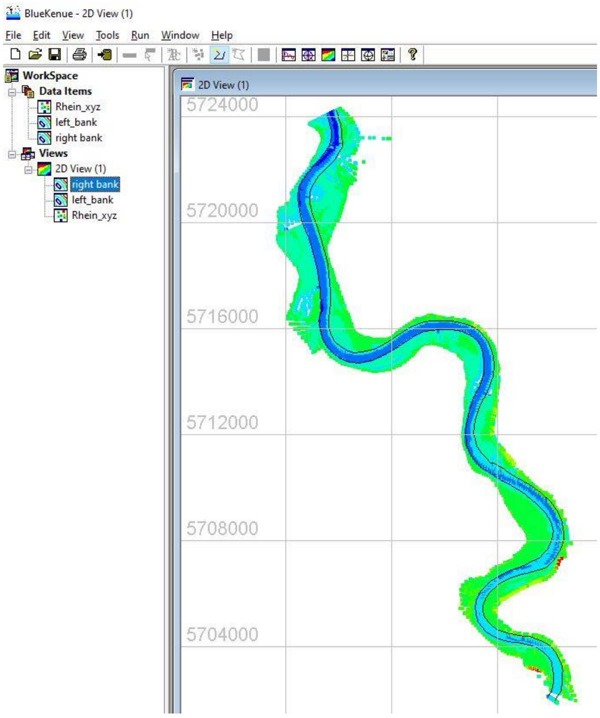


Figure 3. Left and right banks along the river

To define the outline, select “New Closed Line”. Try to include all the coordinate points while drawing. While trying to cover protruding dots, avoid drawing sharp corners (Figure 4). Instead, enclose them with rectangles as much as possible (Figure 5). Doing this will ease the creation of triangular mesh. Click “ESC” to save the outline file after surrounding the whole area and drag it to the 2D view (Figure 6). Save the outline drawing.

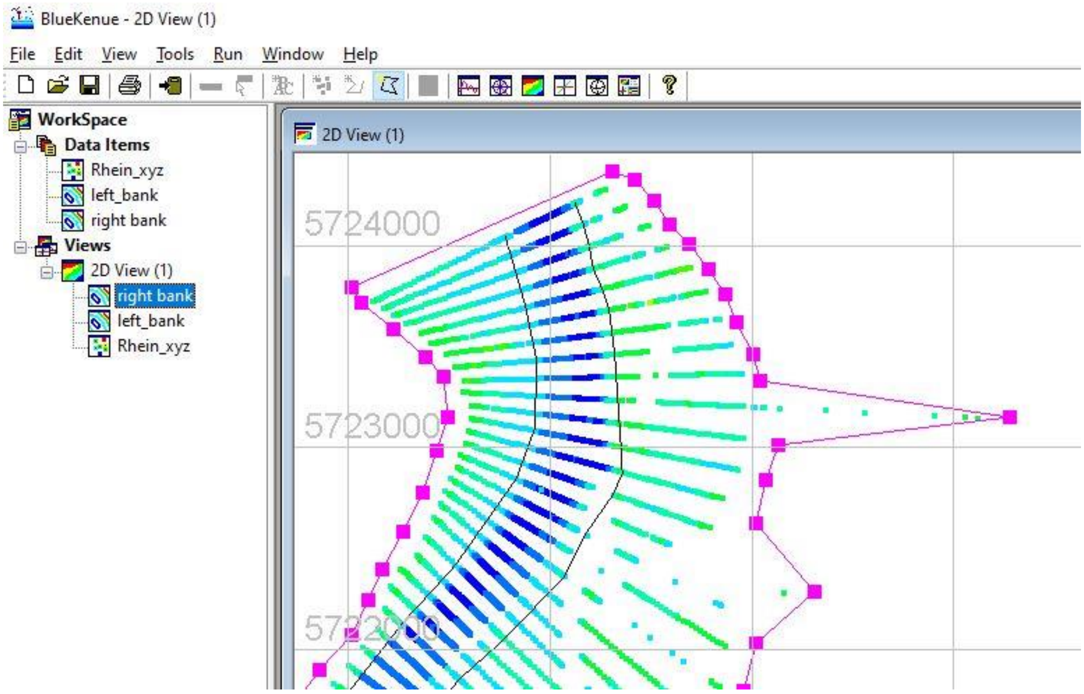


Figure 4. Spikes should be avoided

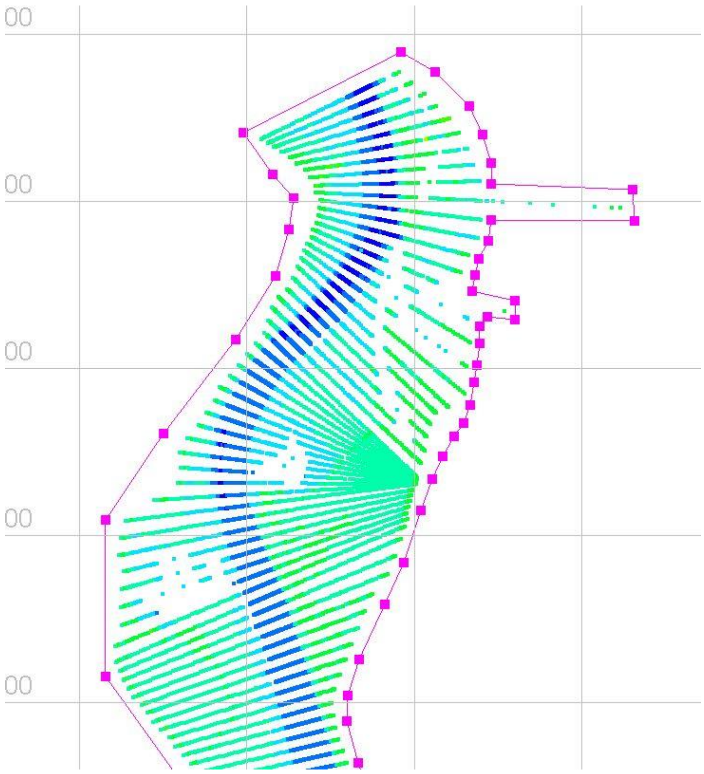


Figure 5. The better way to enclose protruding points

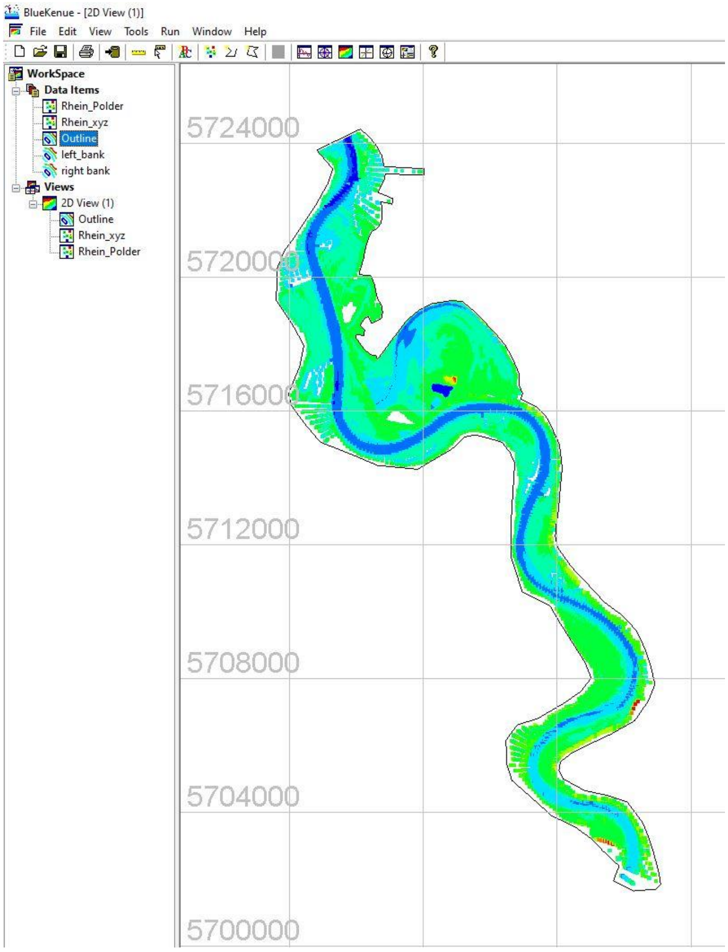


Figure 6. Outline of the study area

As the last step, a dyke will be implemented in the model. To do this, select “New Open Line” and draw the dyke between polder and river coordinates (Figure 7). After completing the drawing, save it as “dyke”.

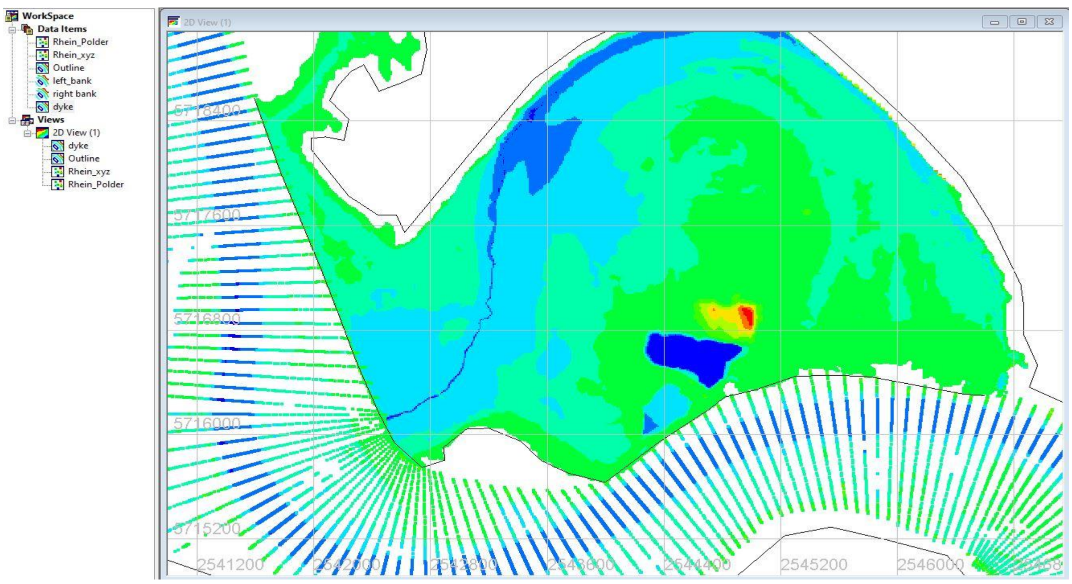


Figure 7. Drawing the dyke

The number of points along the dyke line can be seen by right-clicking on “dyke” and selecting “Show Attribute Table”. In the current line, there are 27 points which are placed irregularly (Figure 8). To have a better mesh, the dyke line should be resampled by right-clicking on “dyke” and selecting the “Resample” option (Figure 10). In the “Resample LineSet” window, select the method as “Equal Distance” and “delta” as 10. It means that there will be equally distanced points that have a 10 m distance between them (Figure 11). After completing the process, there will be a new item named “Resampled dyke”. Right-click on it and select “Show Attribute Table” to control if the resampling process worked properly (Figure 9).

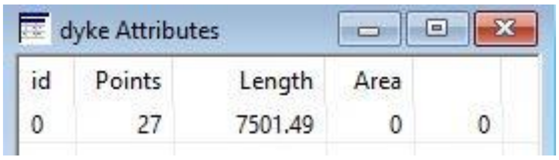


Figure 8. Attribute table of "dyke" item

Figure 9. Attribute table of "Resampled dyke" item

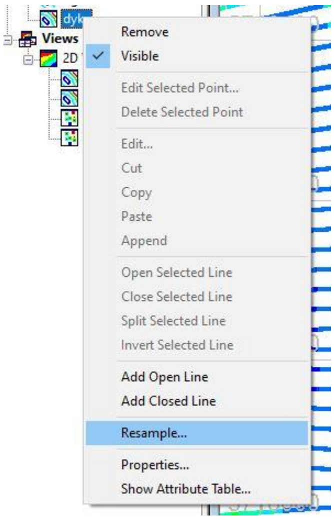


Figure 10. Selecting “Resample” option

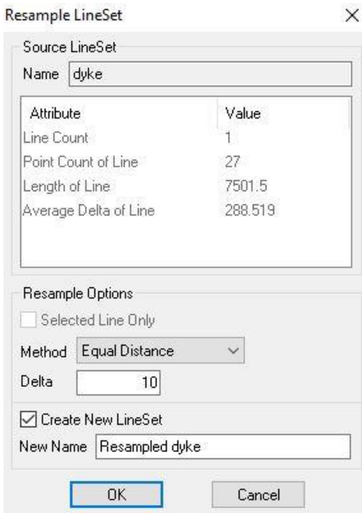


Figure 11. Adjusting the resampling properties

Suggestions for drawing

- Carefully draw the lines and do not misclick since undo option does not work for removing the mistakenly created points, and the drawing process should begin from scratch.
- Repeat sketching the banks and the outline several times to get used to the drawing process. Each repetition will take less time.
- Do not draw sloppy lines. They might decrease the outcome quality of the modelling, and even crash the simulation.
- While surrounding the most upstream and downstream parts, do not go very close to the points. Enclose them a bit wide to prevent mesh anomalies.
- Do not forget to save the drawings. Try using the same bank, outline and dyke drawings for analysing different scenarios in the same study area.

4. MESH GENERATION

TELEMAC-2D solves the depth-averaged Navier-Stokes equation utilizing both Finite Element (FE) and Finite-Volume (FV) formulations. All of these formulations require that a spatial representation of the domain be created using a computational mesh. BlueKenue has several tools for mesh generation and editing. Mesh types that BlueKenue can generate are unstructured and regular (via T3 Mesh Generator) triangular meshes.

In this tutorial, along with T3 Mesh Generator, T3 Channel Mesher is also used. To create it, click “File → New → T3 Channel Mesher”. From the properties window, adjust **Cross Channel Node Count** as 10 and **Along Channel Interval** as 50 (Figure 12) and click “OK”. Cross Channel Node Count decides the number of lines from the left bank to the right bank. Along Channel Interval adjusts the distance between the lines along the channel (Figure 13).

After closing the properties window, a new sub-menu is added to Work Space. For creating the channel mesh, drag the left and the right banks under “newT3ChannelMesh” accordingly. Then, right-click on “newT3ChannelMesh” and select “Properties”. Click “Run” and the channel mesh will be created in seconds (Figure 14). For the sake of upcoming steps, it is better to rename the channel mesh file. To do this, open the properties of “newT3ChannelMesh”, and select “Mesh → Meta Data”, and rename the file as “Channel Mesh”.

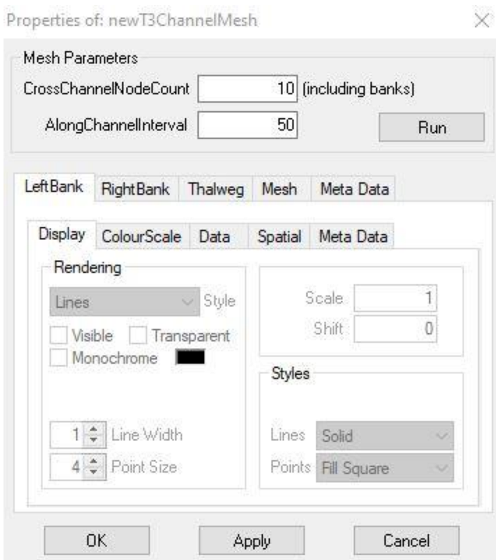


Figure 12. T3 Channel Mesher properties window

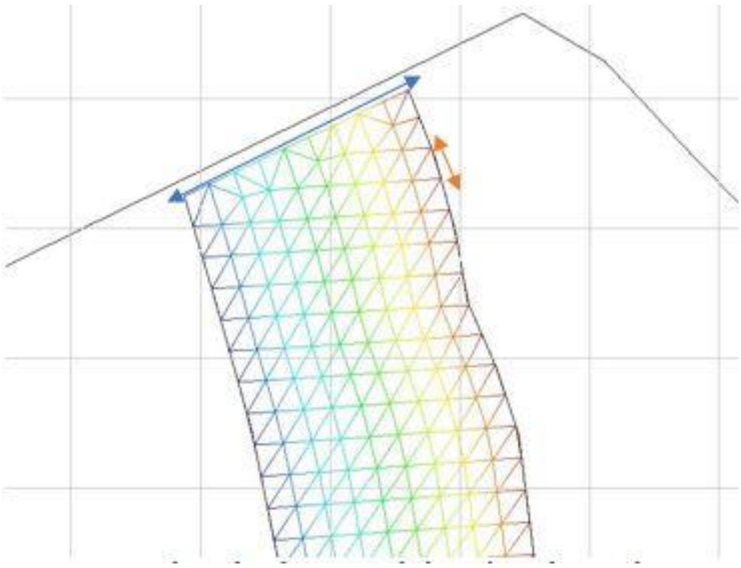


Figure 13. Thanks to “Cross Channel Node Count”, there are 10 vertical lines from the left bank to the right bank along the blue arrow. Changing the "Along Channel Interval" option will affect the distance between to points along the orange arrow. The distance is 50 m in this tutorial.

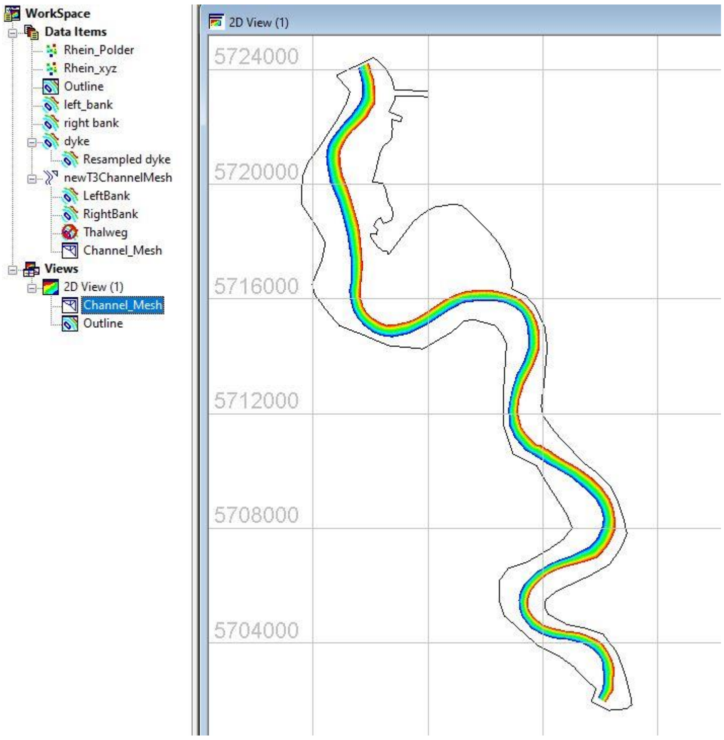


Figure 14. Channel Mesh

To produce mesh for the whole area, click "File → New → T3 Mesh Generator". From the properties window (Figure 15), adjust "Default Edge Length" to 100. "Edge Growth Ratio" can remain at 1.2. "Edge Growth Ratio" adjusts the growth rate between each segment, and "Default Edge Length" decides the length of an edge. Then click "OK" and close the window.

After closing it, another new sub menu is added to Work Space. For creating the mesh along the whole study area, drag the previously drawn outline file to "Outline", resampled dyke to "HardLines", and Channel Mesh file to "SubMeshes accordingly. Then, right click on "newT3Mesh" and select "Properties". Click "Run" and a new window will pop-up (Figure 16).

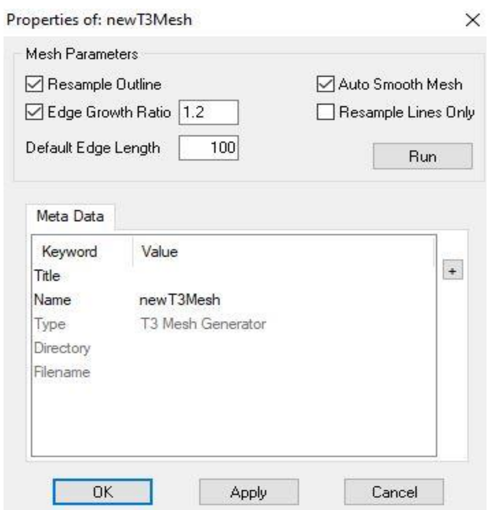


Figure 15. Property window of T3 Mesh Generator

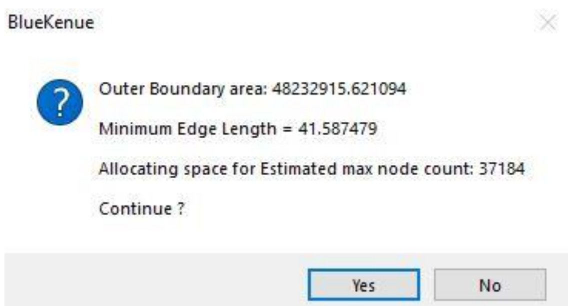


Figure 16. Confirmation window for running the T3 Mesh Generator

Keep in mind that the higher the estimated max node count, the longer the simulation time. Of course, it is almost impossible to get the same numbers with Figure 16, since all the left-right bank and outline drawings will be unique. After clicking "Yes", the mesh will be completed, and it will look as in Figure 17, after dragging the "New Mesh (NodeType) to the 2D View. Keep in mind that there should not be any abrupt triangles in the whole mesh as in Figure 18. If it happens, it is better to re-draw the outline (see Section 3 – Suggestions for drawing). For the next steps, New Mesh can be renamed as "Whole Mesh".

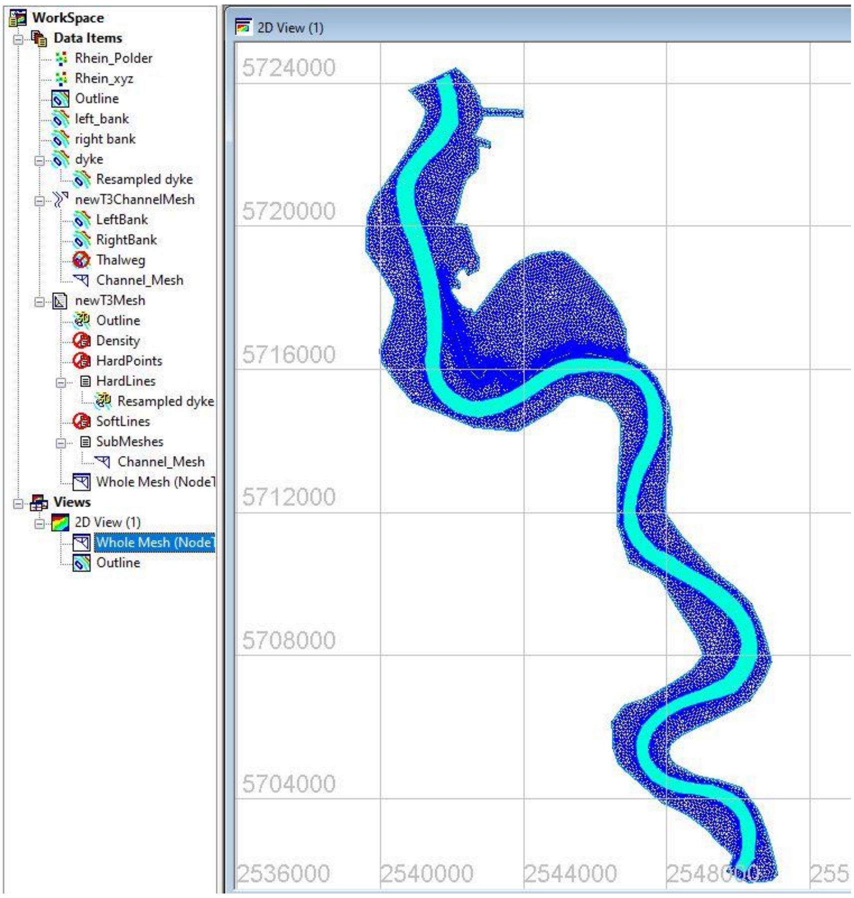


Figure 17. Generated mesh in the 2D View

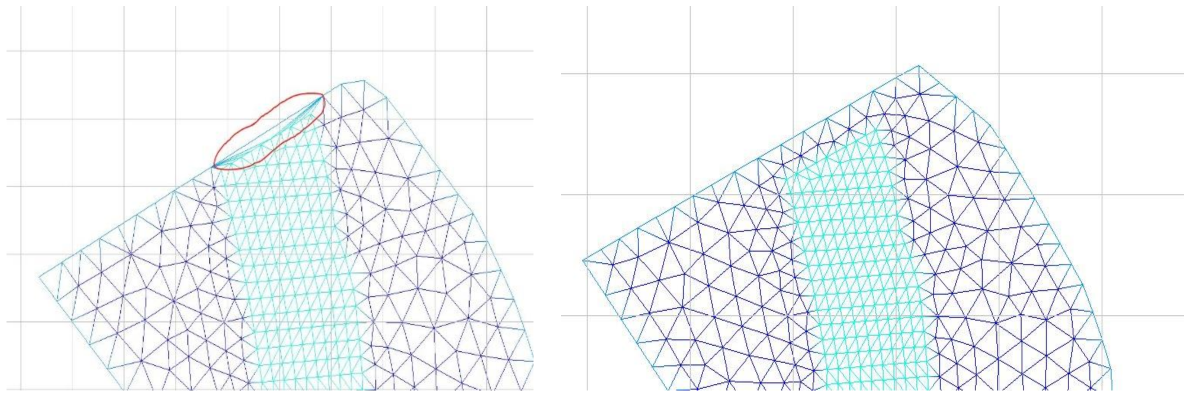


Figure 18. Left: Mesh anomality / Right: Properly created mesh

Inserting bathymetry data into the mesh

The bathymetry data has to be inserted into the previously created mesh via “File → New → 2D Interpolator”. A new submenu (newInterpolator2D) will be opened in Work Space. Drag the coordinate files (Rhein_xyz and Rhein_Polder) into this submenu. To insert the bathymetry, select Whole Mesh under the newT3Mesh submenu, then go to “Run→ Map Object” and choose “newInterpolator2D” to apply the bathymetry file into the mesh (Figure 19). In the Query window, leave the name as “newAttr” and click “OK”.

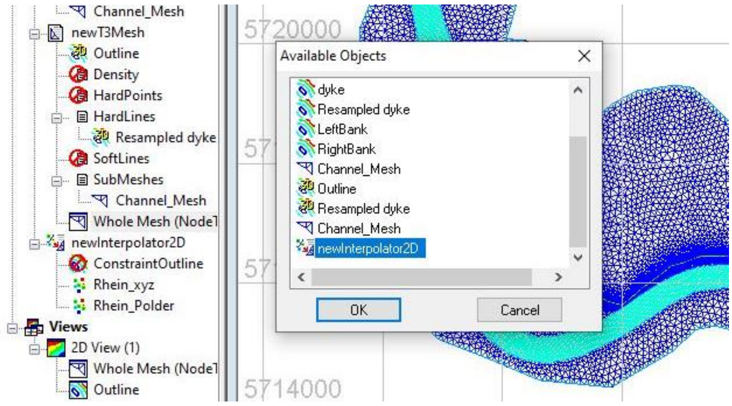


Figure 19. Mapping the “whole mesh” file with bathymetry data

Whole Mesh (NodeType) will be automatically converted to Whole Mesh (newAttr), and the mesh with bathymetry data will look as in Figure 20. To analyse this map better, a legend can be added as described in Section 2.

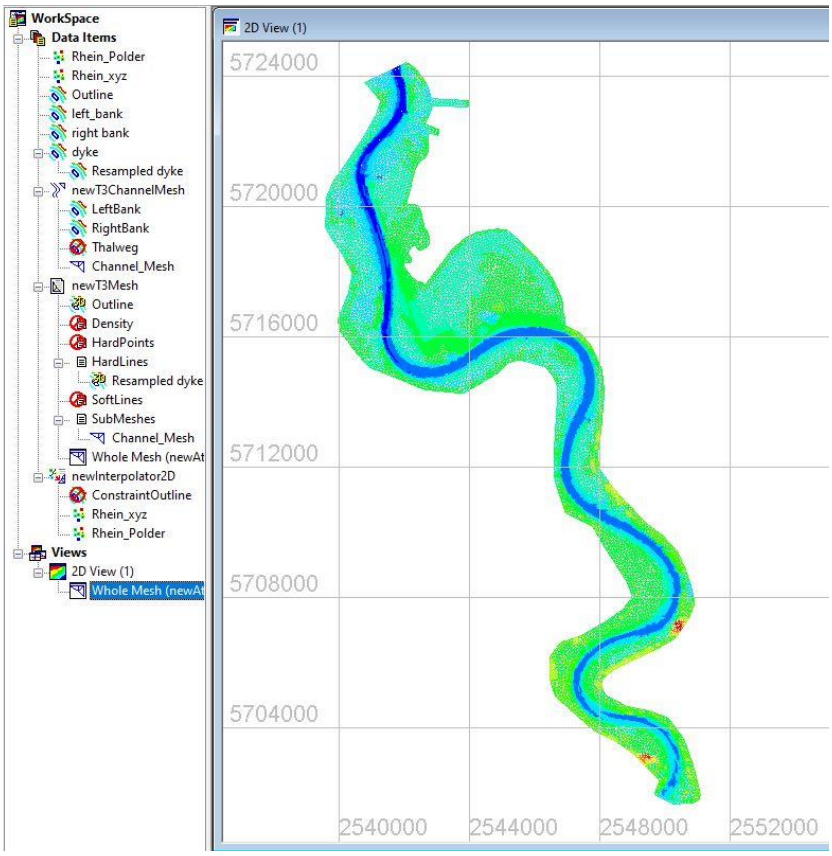


Figure 20. Mesh with the bathymetry data

Dyke height adjustment

After inserting the bathymetry data, the height of the dyke should be checked. To do this, select Resampled dyke and map it with Whole_Mesh (newAttr) item as described previously. As a result, a new item named "Whole_MeshXSection" will be added to the WorkSpace. Drag it into the 1D View (Figure 21).

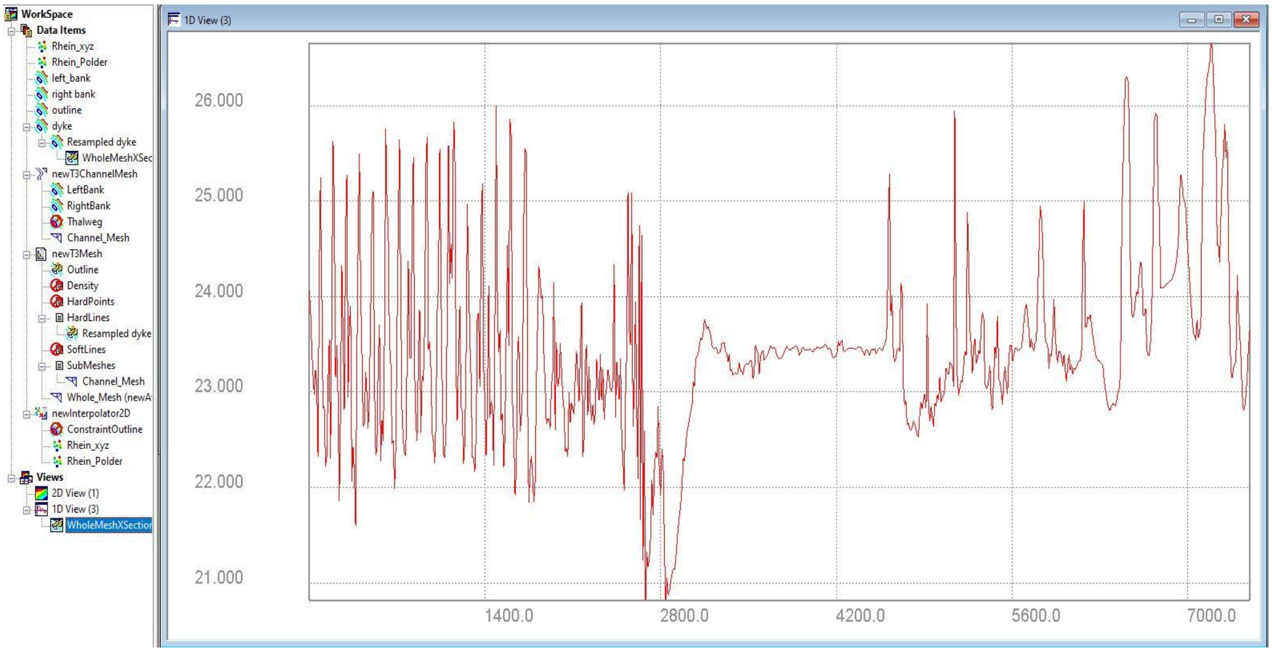
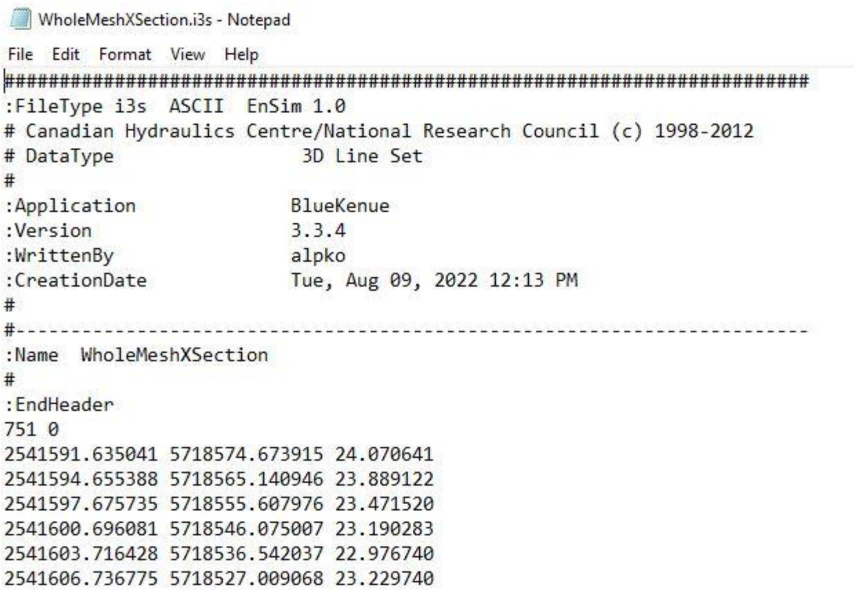


Figure 21. Dyke height after first mapping

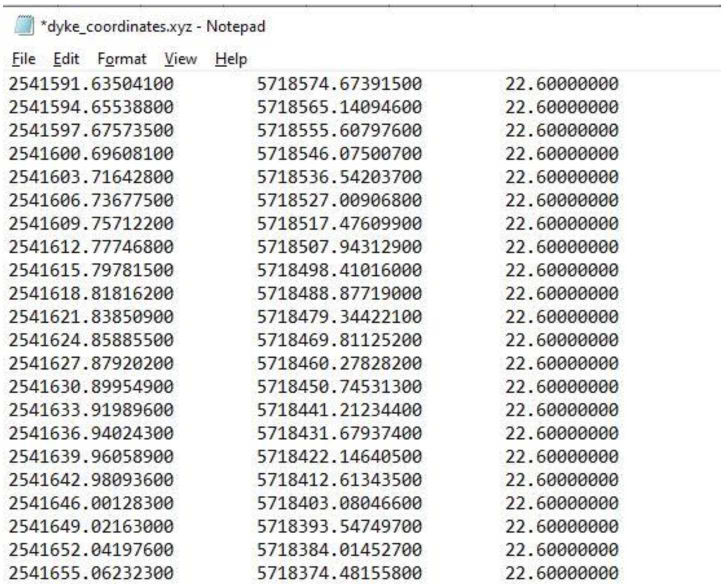
As it can be seen, the height differs at every point along the dyke. This situation is caused by not giving proper "z" values for the dyke in the 2D Interpolator. To solve this problem, save "WholeMeshXSection" as a ".i3s" file. Open this file with a text editor (Figure 22). Copy the coordinate data and paste them into an Excel sheet.



```
WholeMeshXSection.i3s - Notepad
File  Edit  Format  View  Help
#########################################################################
:FileType i3s  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2012
# DataType                 3D Line Set
#
:Application               BlueKenue
:Version                   3.3.4
:WrittenBy                 alpko
:CreationDate              Tue, Aug 09, 2022 12:13 PM
#
#------------------------------------------------------------------------
:Name  WholeMeshXSection
#
:EndHeader
751 0
2541591.635041 5718574.673915 24.070641
2541594.655388 5718565.140946 23.889122
2541597.675735 5718555.607976 23.471520
2541600.696081 5718546.075007 23.190283
2541603.716428 5718536.542037 22.976740
2541606.736775 5718527.009068 23.229740
```

Figure 22. Content of the ".i3s" file

If the data is only shown in one column after being pasted to Excel, go to "Data→ Text to Columns". From "Convert Text to Columns Wizard", select Fixed width, then click "Next" and "Finish", respectively. X, Y and Z data are now separated into three different columns. Since the dyke height is 22.6, change all the values under the third column to 22.6. Then, for each cell, increase the decimal to 8 (there will be <u>eight digits</u> in the decimal section). Finally, create a new text file and paste the values from excel into there (Figure 23). There is no need to give any column name here. Save the file as "dyke_coordinate.xyz".


*dyke_coordinates.xyz - Notepad

File Edit Format View Help

2541591.63504100	5718574.67391500	22.60000000
2541594.65538800	5718565.14094600	22.60000000
2541597.67573500	5718555.60797600	22.60000000
2541600.69608100	5718546.07500700	22.60000000
2541603.71642800	5718536.54203700	22.60000000
2541606.73677500	5718527.00906800	22.60000000
2541609.75712200	5718517.47609900	22.60000000
2541612.77746800	5718507.94312900	22.60000000
2541615.79781500	5718498.41016000	22.60000000
2541618.81816200	5718488.87719000	22.60000000
2541621.83850900	5718479.34422100	22.60000000
2541624.85885500	5718469.81125200	22.60000000
2541627.87920200	5718460.27828200	22.60000000
2541630.89954900	5718450.74531300	22.60000000
2541633.91989600	5718441.21234400	22.60000000
2541636.94024300	5718431.67937400	22.60000000
2541639.96058900	5718422.14640500	22.60000000
2541642.98093600	5718412.61343500	22.60000000
2541646.00128300	5718403.08046600	22.60000000
2541649.02163000	5718393.54749700	22.60000000
2541652.04197600	5718384.01452700	22.60000000
2541655.06232300	5718374.48155800	22.60000000

Figure 23. xyz file to give the dyke a proper height

Now, insert the dyke_coordinate.xyz file into BlueKenue. Create a new T3 Mesh with the same steps that are previously described, then name it Dyke_Whole_Mesh. Afterwards, create a 2D Interpolator and drag Rheix_xyz, Rhein_Polder and dyke_coordinate files under it. To prevent any mistakes, rename this 2D interpolator as Dyke_Interpolator. Map Dyke_Whole_Mesh with Dyke_Interpolator bathymetry data. Finally, map the Resampled dyke with Dyke_Whole_Mesh(newAttr) and drag it into 1D View (Figure 24).

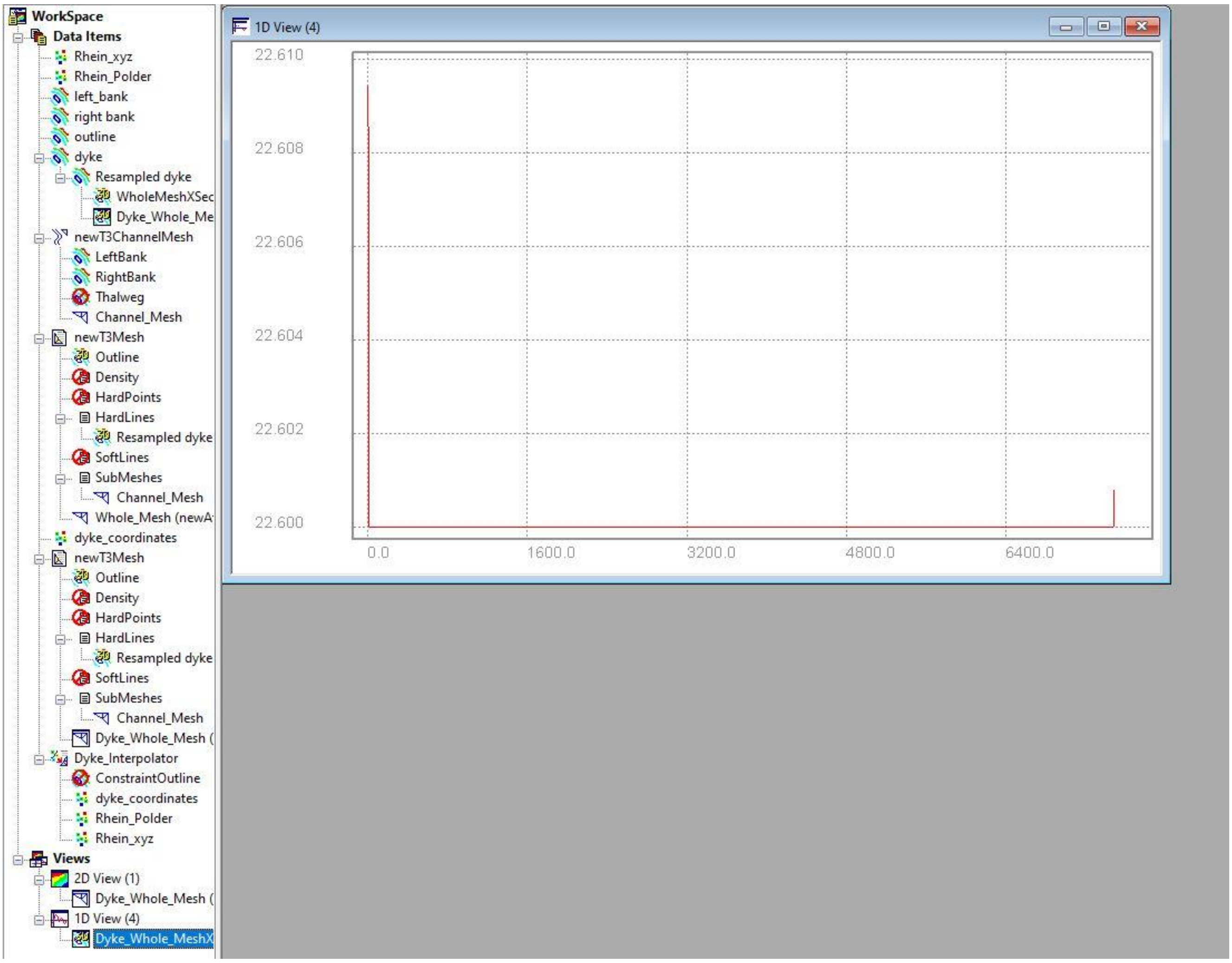


Figure 24. Corrected height of all points along the dyke

Ignore the abrupt values at both ends since the error is low in number but looks greater due to zoom on the graph. As can be seen in Figure 25, dyke has the same height on every point, thanks to the new 2D interpolator.

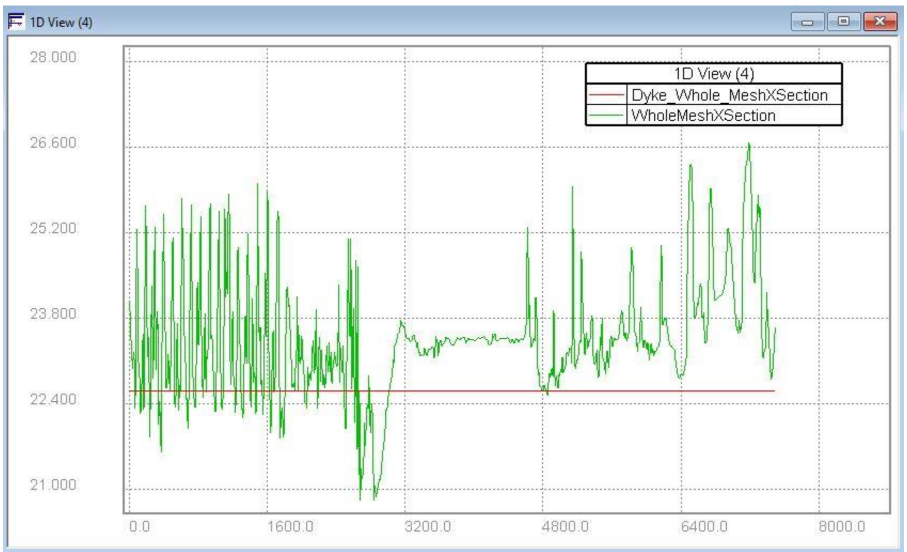


Figure 25. Comparison graph for dyke height for two meshes

5. CREATING THE SELAFIN FILE

A selafin and a boundary conditions files are compulsory to run the simulation. To create the selafin file, go to "File → New → Selafin Object". In WorkSpace, a new submenu named "newSelafin" will be opened. Right-click on "newSelafin" and select "new Variable". From the "Add New SELAFIN Variable" window, select the properties as it is shown in Figure 26 (To be able to change the Attribute Name as NewAttr, "Copy Node Values from Source" should be selected first). The "BOTTOM" variable helps to define the lowest points in the whole area. To check if the process is executed correctly, open 3D View from the Toolbar by clicking on "New 3D View" and dragging the BOTTOM file under the "3D View" submenu on the WorkSpace (Figure 27). If the 3D View cannot be obtained, please repeat the "Add New SELAFIN Variable" process carefully. Finally, save the "newSelafin" file with a ".slf" extension.

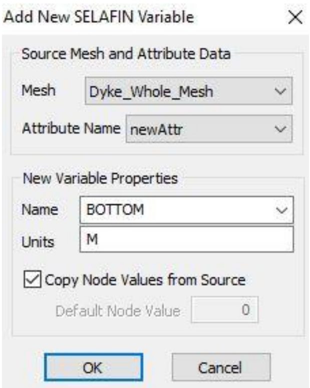


Figure 26. Adding "Bottom" as a new selafin variable

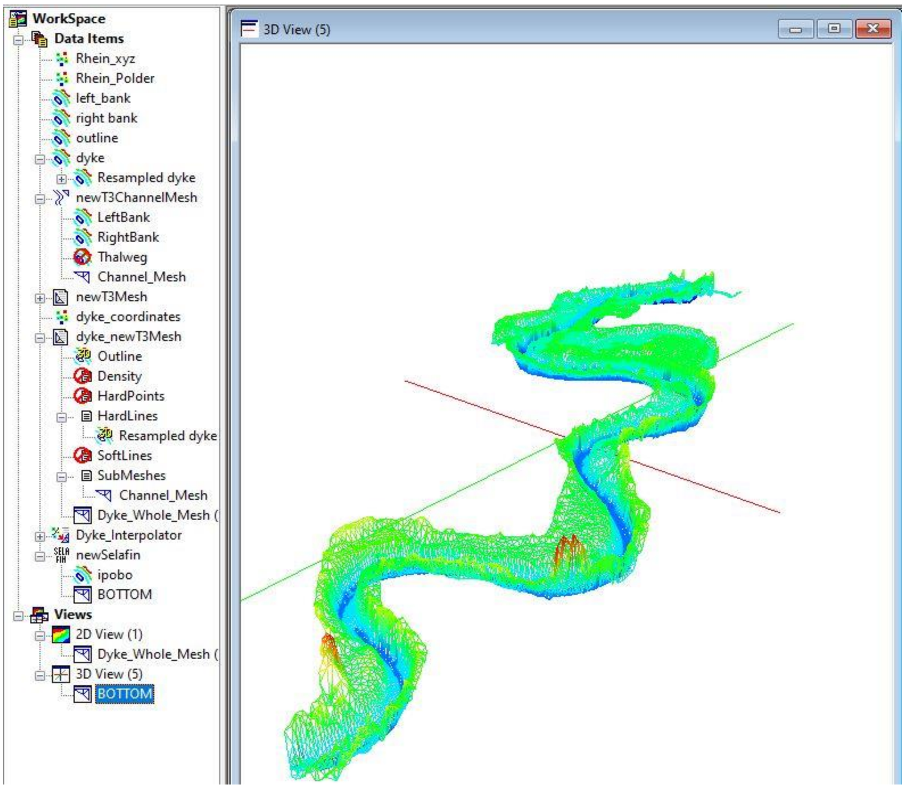


Figure 27. BOTTOM file in the 3D view

6. DEFINING THE BOUNDARY CONDITIONS

As a final step in BlueKenue, boundary conditions should be defined. First, a boundary conditions file should be created via "File → New → Boundary Conditions (Conlim)…". From the "Available t3s Objects" window, select "BOTTOM". A new submenu named "BOTTOM_BC" will be created in the WorkSpace. To visualize it, drag "BOTTOM_BC" into the 2D view (Figure 28).

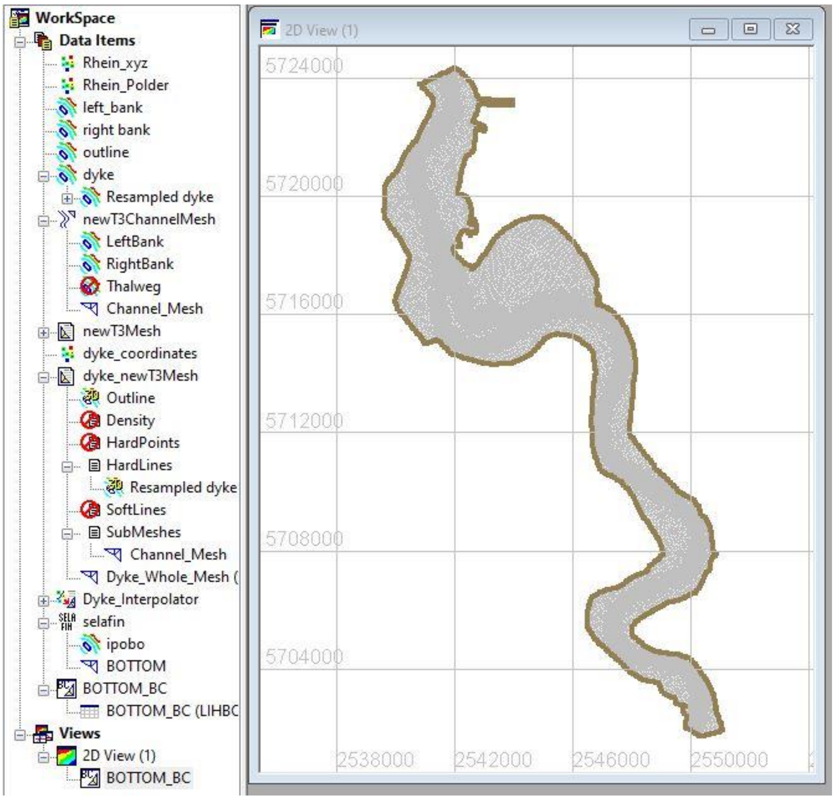


Figure 28. BOTTOM_BC file in 2D View

To determine the upstream boundary condition, zoom in on the upstream part. Select a node with a double click, as shown in Figure 29. For determining the boundary, press and hold the "Shift" key, and double click to the other node from the opposite side. The boundary line's colour will turn magenta (Figure 30). Right-click on the magenta-coloured line and click on "Add Boundary Segment". From the "CONLIM Boundary Segment Editor", select Boundary Code as "Open boundary with prescribed Q" and Tracer Code as "Open boundary with free Tracer". After clicking "OK", the colour of the upstream boundary segment will be blue (Figure 31). More details about these options can be found TELEMAC 2D User Manual.

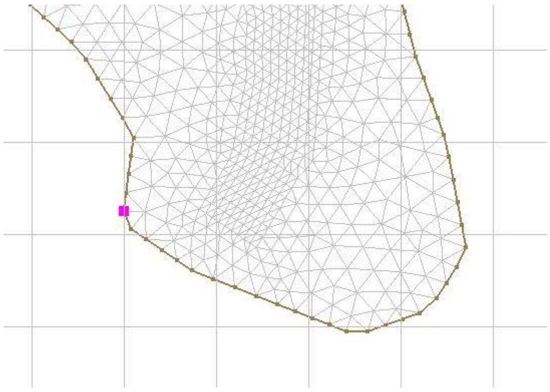


Figure 29. Selecting a node

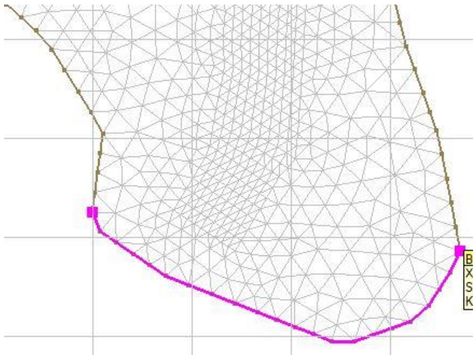


Figure 30. Determining the upstream boundary segment

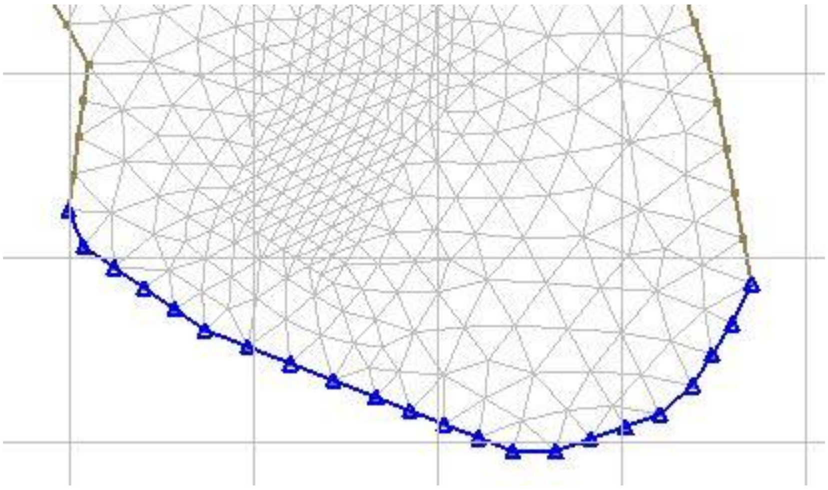


Figure 31. Upstream boundary segment with the prescribed Q condition

Follow the same steps for the downstream boundary and determine the boundary segment (Figure 32). Select Boundary Code as "Open boundary with prescribed H" and Tracer Code as "Open boundary with free Tracer". After clicking "OK", the colour of the downstream boundary segment will be green (Figure 33).

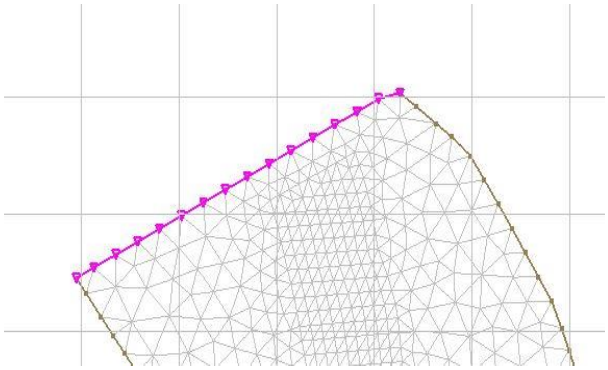


Figure 32. Determining the downstream boundary segment

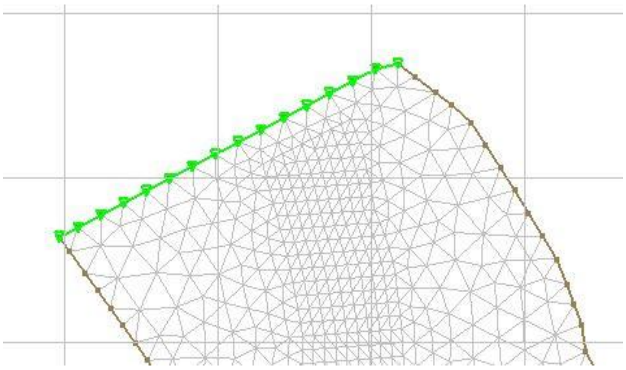


Figure 33. Downstream boundary segment with the prescribed H condition

Finally, from the "BOTTOM_BC" submenu, click on "BOTTOM_BC (LIHBOR) and save it with the ".cli" extension.

!!! <u>Do not close BlueKenue until the end of hotstart file section.</u>

7. CAS FILE

Creating the CAS file is the last step before running the simulation. CAS file specifies the correct parameters that are required to obtain meaningful results. The user must know the type of simulation, simulation time and necessary variables. FUDAA Pre-Pro, the software available from the TELEMAC-MASCARET ensemble, is used for creating the CAS file. Before using the FUDAA Pre-Pro, the output files from BlueKenue should be stored in the same folder.

From the search box inside of the taskbar, search Fudaa Pre-Pro and open it. A new supervisor screen will appear on the desktop (Figure 34).

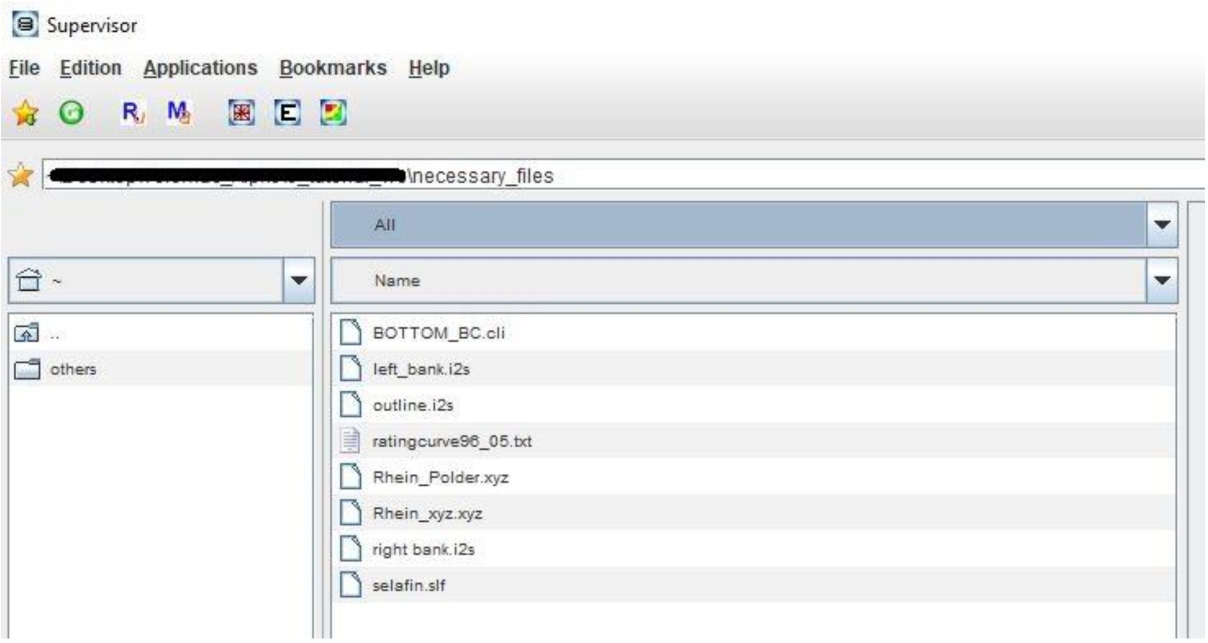


Figure 34. Fudaa Pre-Pro's supervisor window

Using the leftmost bars, navigate to the location folder in which the geometry (.slf) and boundary condition (.cli) files were placed. Afterwards, click "Applications" and select "Editor". From the Editor window, click "Create" (Figure 35).

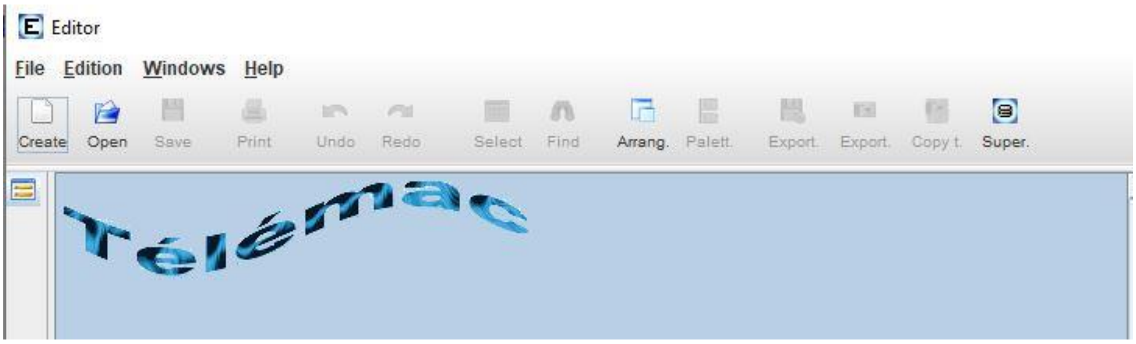


Figure 35. Editor window

In order to create the file, place the selafin and boundary conditions files respectively (Figure 36). For the steering file, open the file explorer by clicking the double dots and in write a relevant name for the steering file the "File Name" box, then click save (Figure 37). Then, click on the green check mark and ignore the "Cartesian grid is not supported" warning (Figure 38).

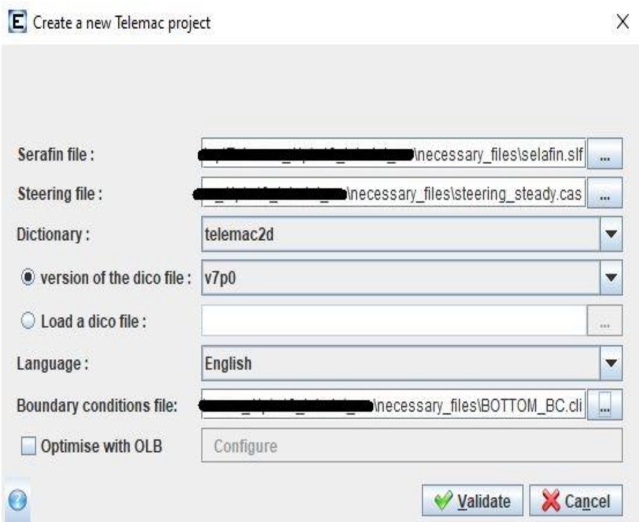


Figure 36. Inserting necessary files for creating the CAS file

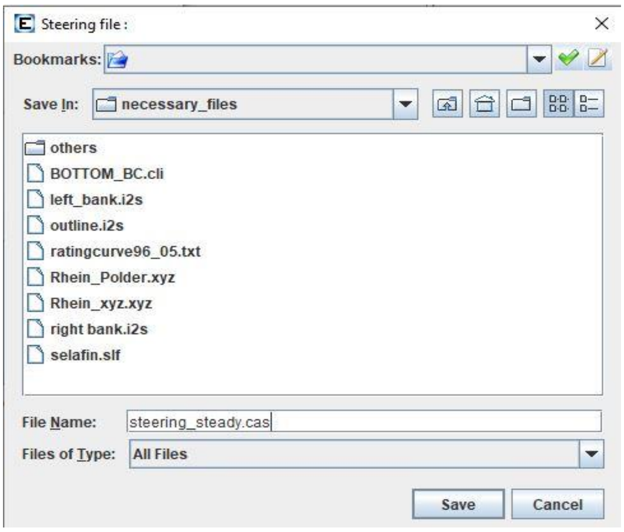


Figure 37. Filling the steering file box

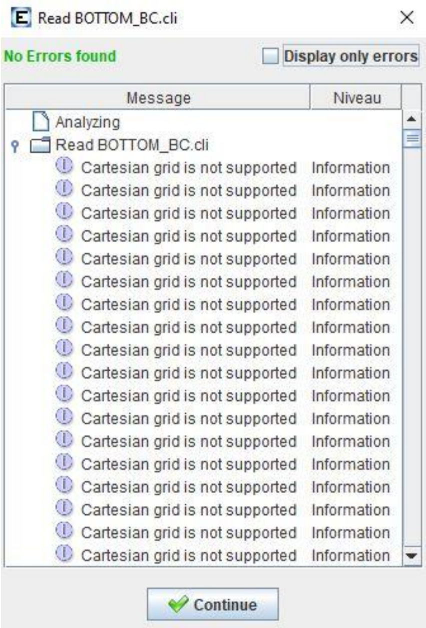


Figure 38. When "No Errors found" is written, click "Continue"

The geometry file with the liquid boundaries will be visible (Figure 39). To give the relevant information for the modelling, click on the "Home" icon, which represents the "General parameters", as marked in Figure 39.

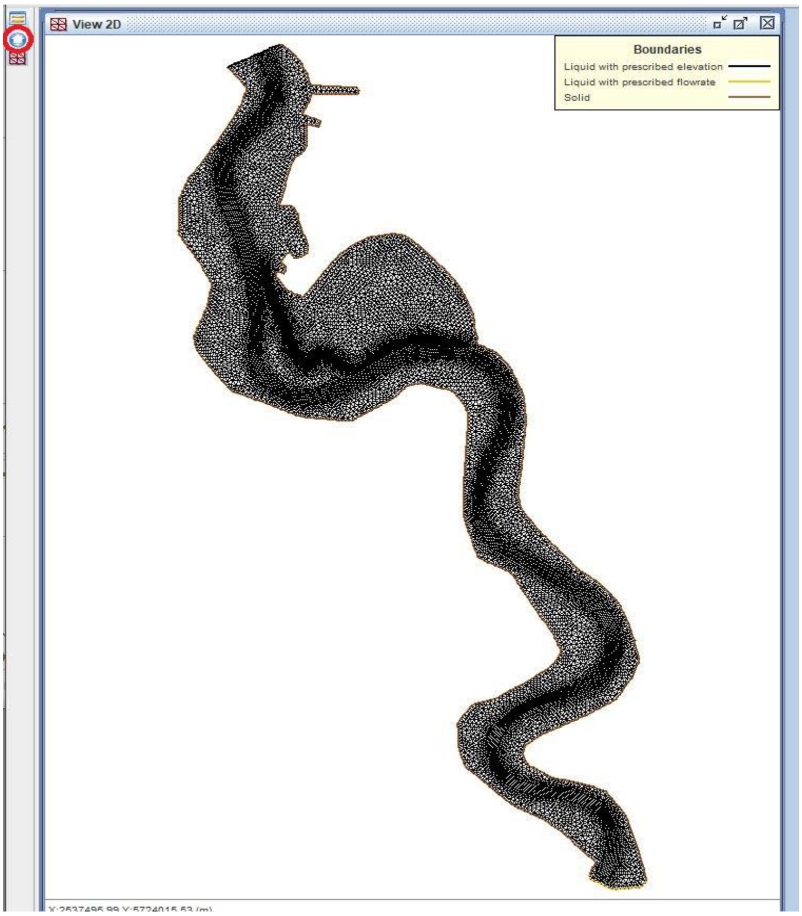


Figure 39. Visualization of the geometry file with the liquid boundaries

In the "General parameter" window, "Result File" should be manually filled as it was previously completed for the steering file (Figure 40).

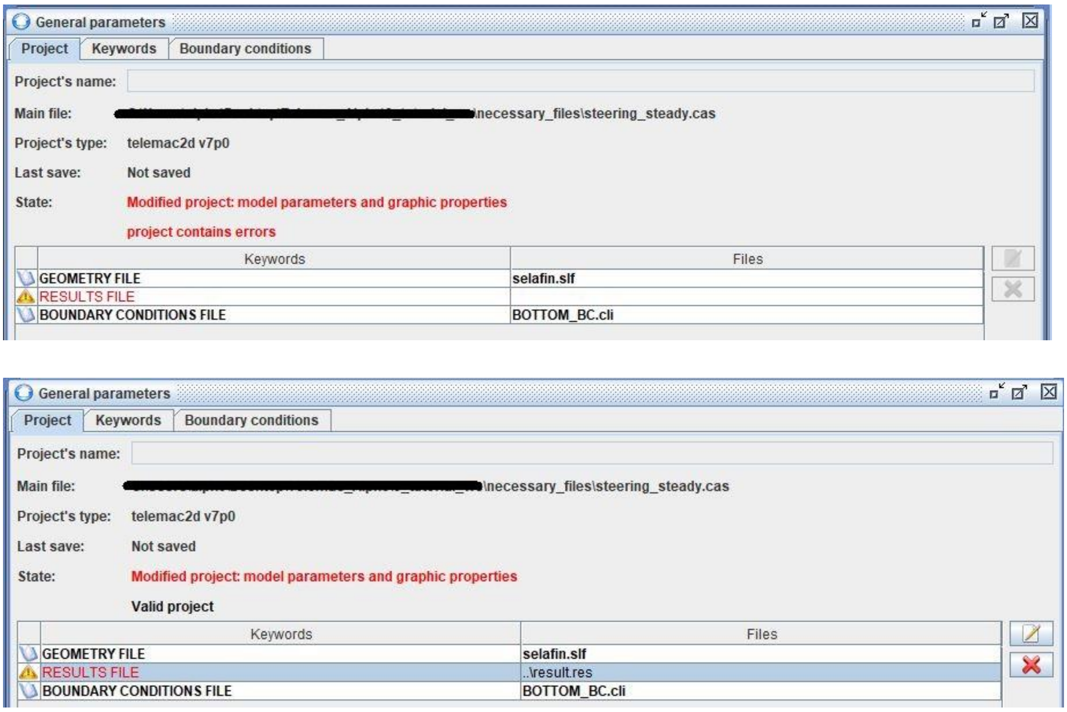


Figure 40. Filling the results file box and making the project valid

From the upper menu of the “General parameters” window, click on “Keywords”. On the left side of the window, the “Not modified” option is selected as default. When a parameter is modified, it can only be seen if the state is adjusted as “Modified”.

In the “Mode” submenu, “normal” is the default selection. However, it is better to select “Expert” from the “Mode” submenu to see all the parameters that can be altered.

If nothing is selected from the “Headings” submenu, all the parameters will be listed alphabetically. However, to find demanded parameters easily, it is better to use headings. Detailed information about the parameters can be found in the Telemac2D User Manual.

- <u>Boundary Conditions</u>

By clicking on Add button, insert two options. Adjust the first as 0 and the second as 1 (Figure 41).

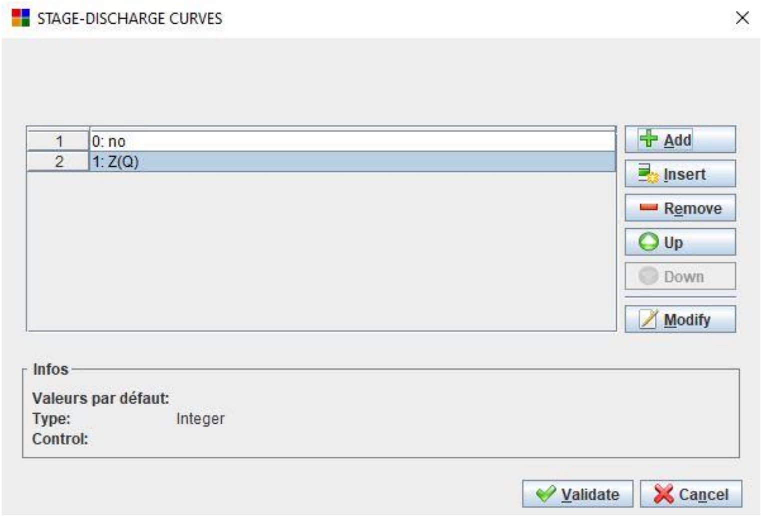


Figure 41. Stage-Discharge curves

- <u>Equations</u>

From “Equations”, adjust “Law of Bottom Friction” as 3 (Strickler), “Friction Coefficient” as 40 and “Turbulence Model” as 3 (Figure 42).

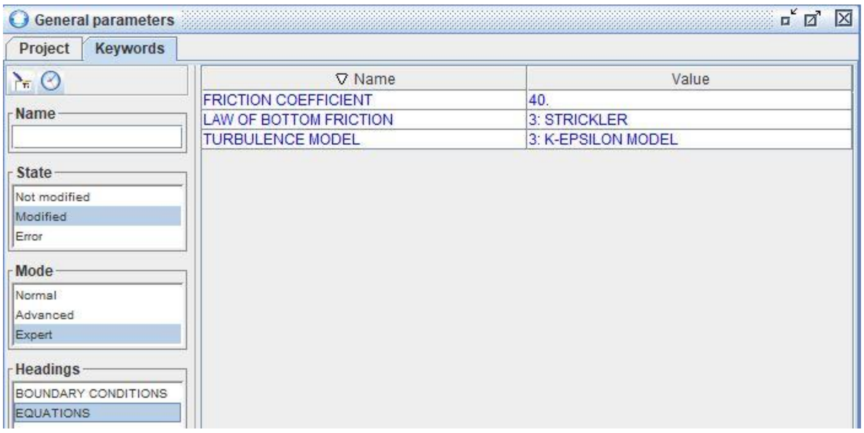


Figure 42. Modified parameters of the equations heading

- <u>Equations, Boundary Conditions</u>

In this heading, adjust "Option for Liquid Boundaries" as 1;1 (Figure 43). By default, this window will open blank. With the "+ Add" button, add two conditions for both upstream and downstream.

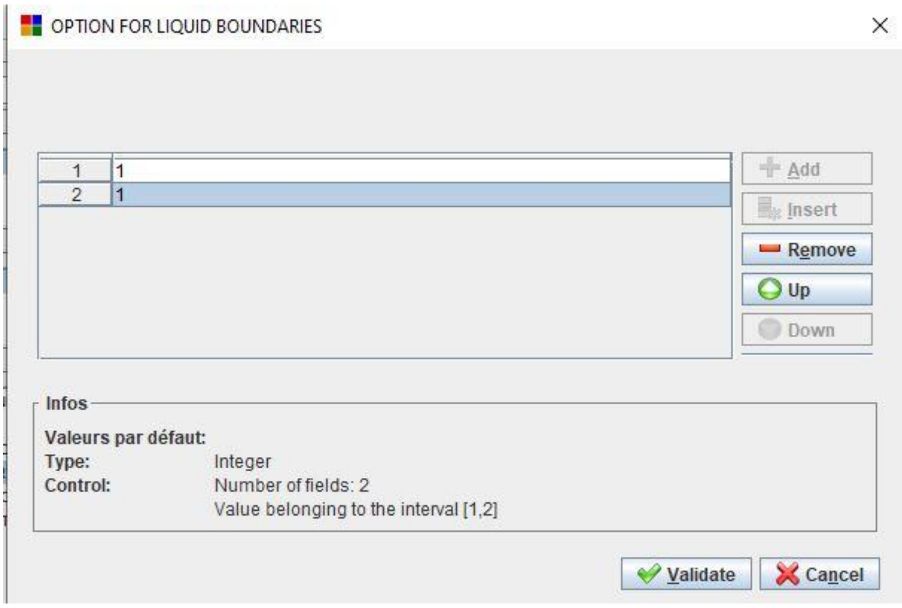


Figure 43. Adjustment of the liquid boundaries

By using the same method, adjust "Prescribed Elevations" as 0.0;0.0, "Prescribed Flowrates" as 7410.0;0.0, and "Velocity Profiles" as 5;1.

- <u>Equations, Initial Conditions</u>

Here, adjust "Initial Elevation" to 23 and select "Constant Elevation" from the "Initial Conditions" box (Figure 44).

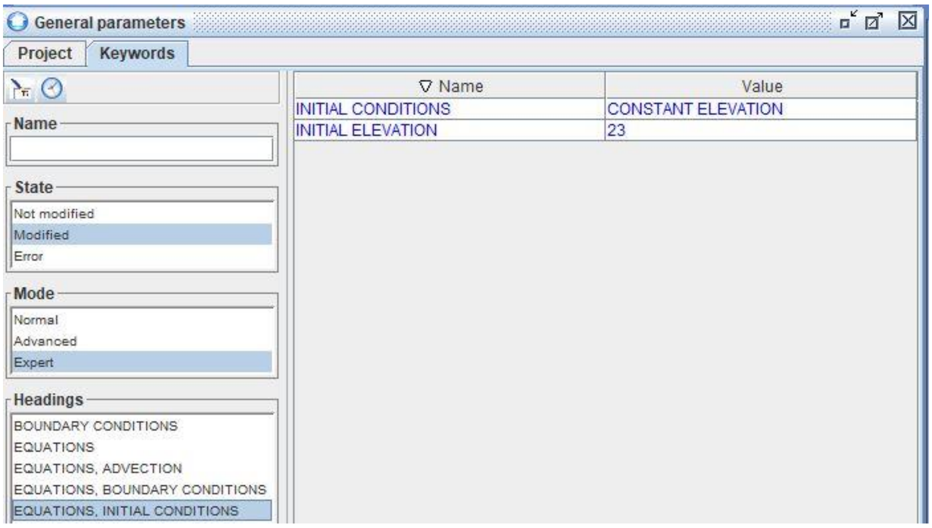


Figure 44. Initial conditions

- Input – Output, Files

Steering, geometry, boundary and results files have already been filled in the previous steps. Add the “ratingcurve96_05.txt” file as “Stage-Discharge Curves File”.

- Input – Output, Graphics and Listing

Select the “Variables for Graphic Printouts” as shown in Figure 45. These selected parameters can be visualized after a successful simulation. Adjust “Listing Printout Period” and “Graphic Printout Period” as 100 (unit of these values are second). To have a faster simulation with fewer details in the results file, the values of these two parameters can be increased. Please note that giving abnormally high values would crash the simulation. Finally, mark the Mass-Balance box (Figure 46).

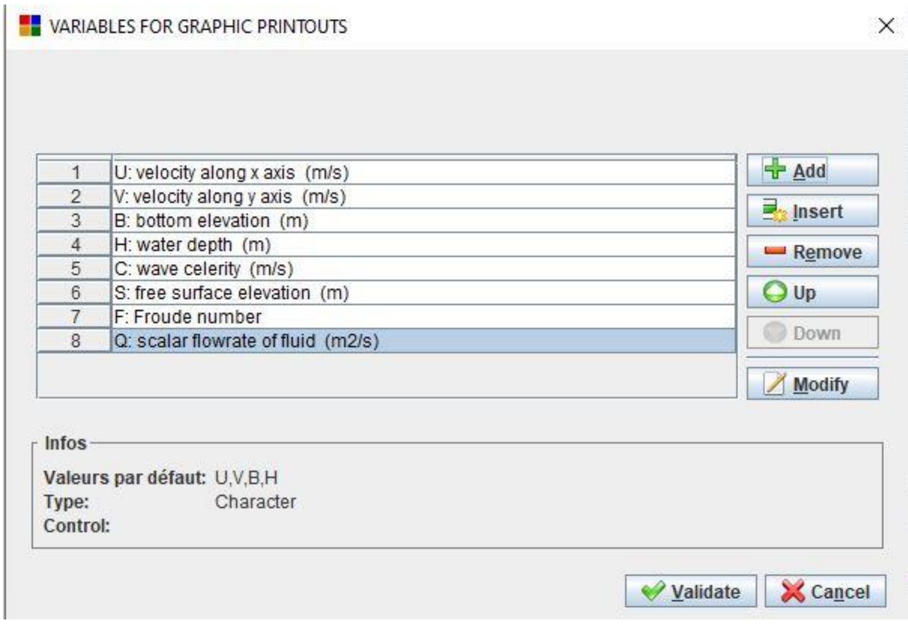


Figure 45. Selected variables for graphic printouts

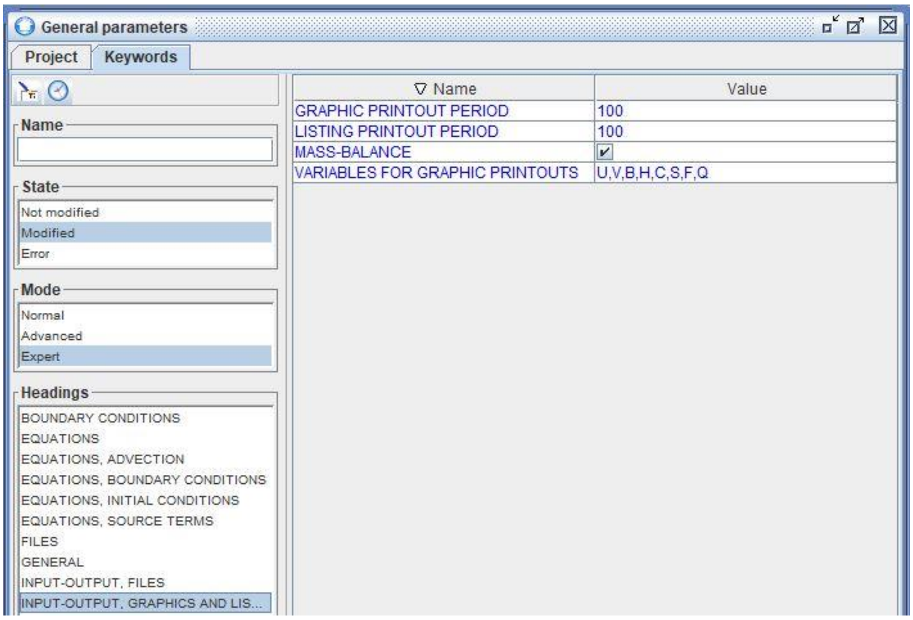


Figure 46. Modified parameters in Input-Output, Graphics and Listing heading

- Input – Output, Information

Give a definitive title to the CAS file from the “Title” box to remember its purpose of it in the future works.

- Numerical Parameters

Adjust “Time Step” as 10 seconds, “Duration” as 200000 seconds and “Number of Time Steps” as 1000. Since the event is steady, the duration of the simulation along with the number of time steps selected for it should be considered as arbitrary. Finally, mark the “Continuity Correction” box.

- Numerical Parameters, Solver

Open the “Numerical Parameters, Solver” heading. Adjust “Solver” as 2, “Maximum Number of Iterations for Solver” as 1000, and “Solver Accuracy as 1. E-3.

Lastly, save the CAS file by clicking the “Save” button from the top menu. An error message which is written in French can occur (Figure 47). Ignore this message and close that window. Check the destination folder of the CAS file to control if it is saved successfully.

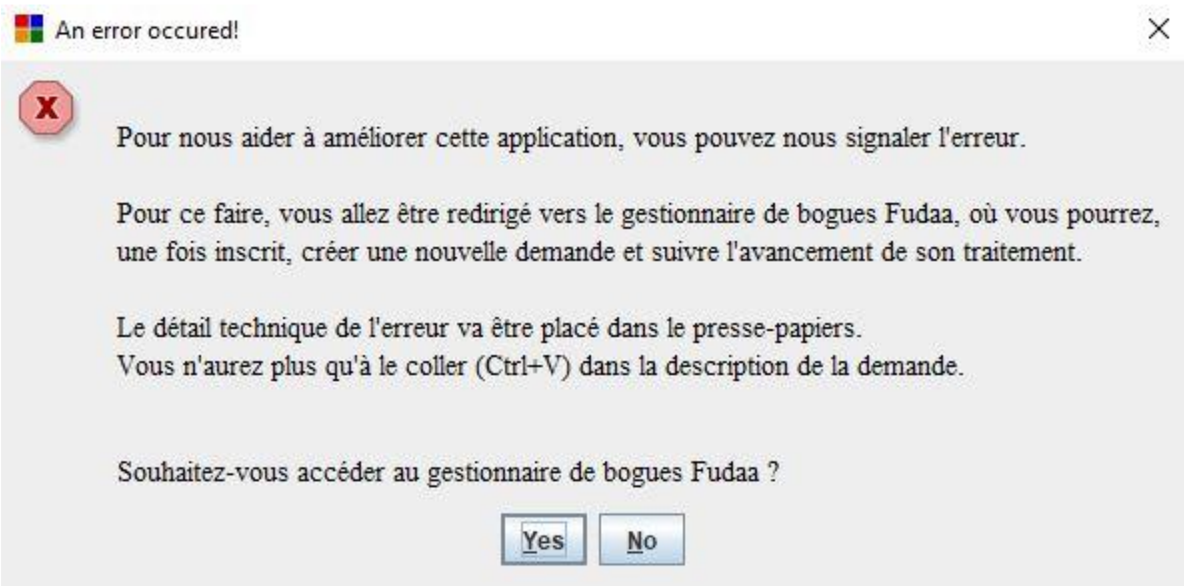


Figure 47. Error message that pops up while saving the CAS file

Please remember that the created CAS file can be used for simulations for the same study area with different boundary conditions or geometry files. Before starting the new simulation, open the CAS file with a text reader and change the boundary and geometry file names accordingly. Other parameters can also be changed directly via text editor. Always keep the CAS file in the same folder as other necessary documents that are stated in the CAS file (for example, the Q-H relation file).

8. RUNNING THE SIMULATION AND VISUALIZING THE RESULT FILE

After successfully creating the CAS file, it is time to run the simulation. To do this, open the TELEMAC Command Prompt (the name of this command prompt can differ according to the version). In the command prompt, write cd and paste the file destination where the CAS file has been saved (Figure 48). After the desired destination is given, type telemac2d.py and the full name of the CAS file (in this tutorial it is named steering_steady.cas) by leaving one space between them (Figure 48).

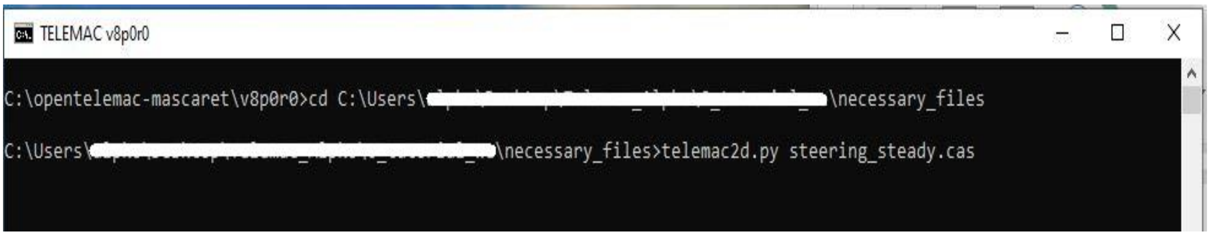


Figure 48. Running the simulation

At the beginning, "Exceeding Maximum Iterations" warning can be seen until the simulation reaches the steady state (Figure 49).

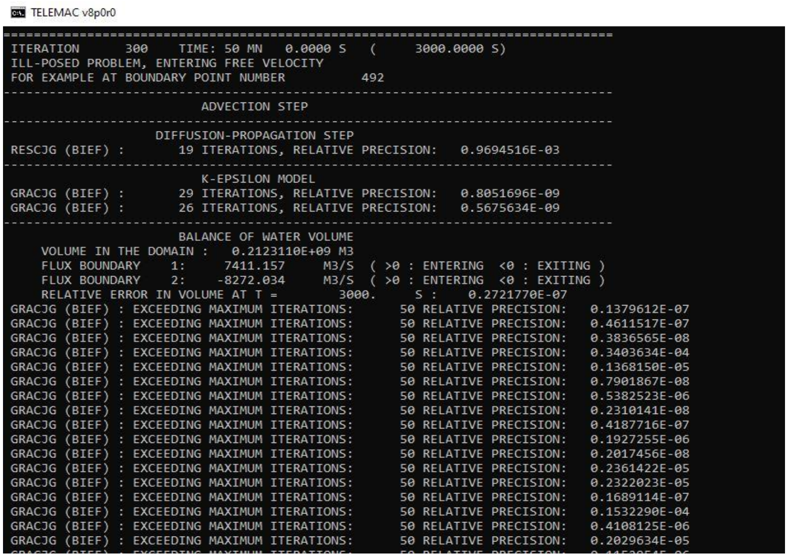


Figure 49. Exceeding Maximum Iteration warning

During the simulation, the values will be updated for each iteration (Figure 50).

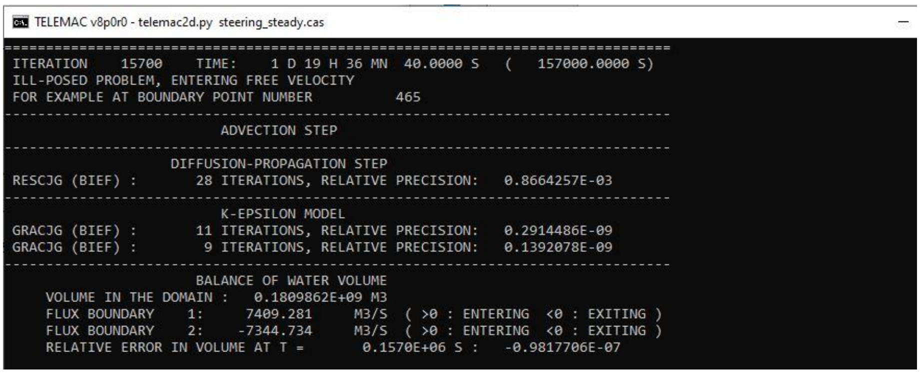
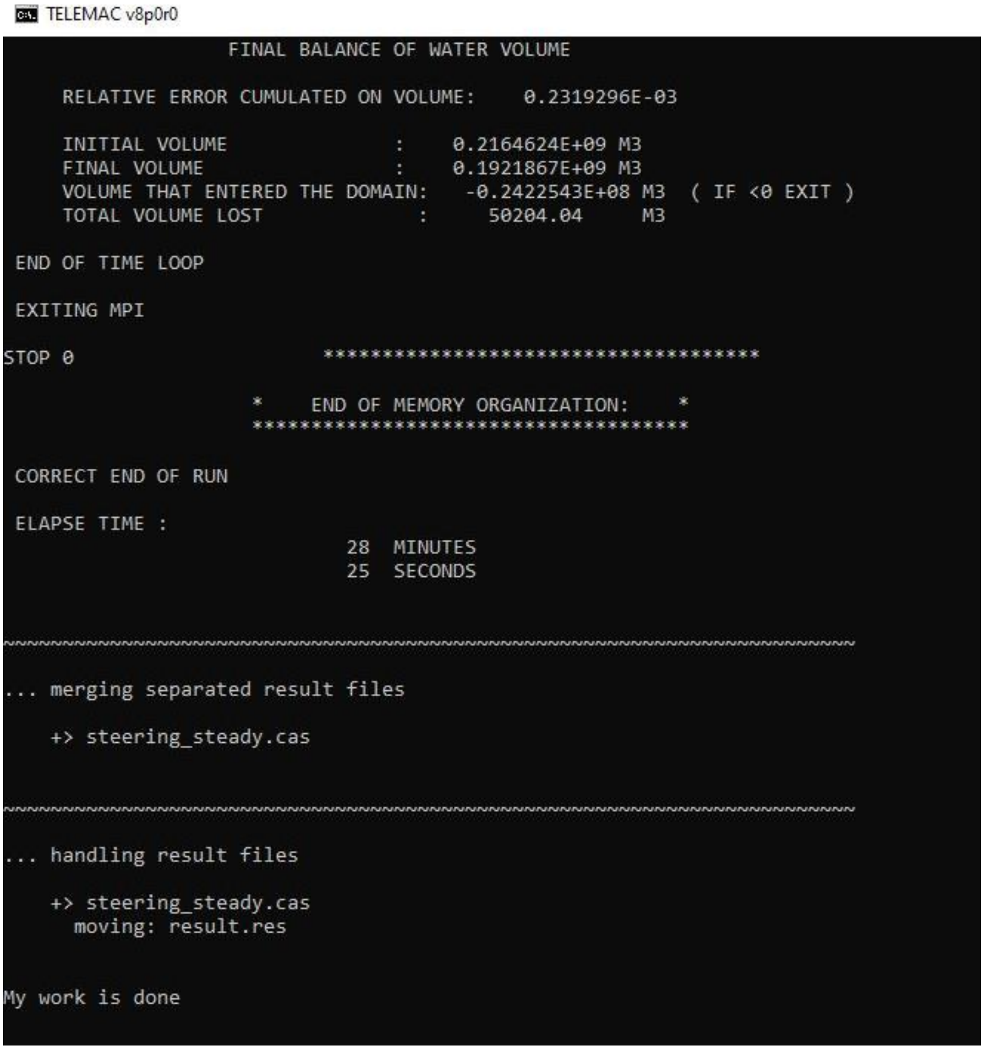


Figure 50. Values of an iteration

From the first line, number of iteration can be understood. Moreover, since the difference between the two flux boundaries is low, it can be understood that the model has reached a steady state. Of course, this gap can differ a lot in the unsteady simulation.

Moreover, the value of "Relative Error in Volume at T" can give ideas of the model. Since the value is very low, it can be understood that there is no water storage in the model and it works without any problem.

When the simulation ends without any problem, the "My work is done" message occurs in the command prompt (Figure 51). In the end screen, total volume lost, duration of simulation and name of the result file can be seen.



```
TELEMAC v8p0r0
                   FINAL BALANCE OF WATER VOLUME

     RELATIVE ERROR CUMULATED ON VOLUME:    0.2319296E-03

     INITIAL VOLUME              :    0.2164624E+09 M3
     FINAL VOLUME                :    0.1921867E+09 M3
     VOLUME THAT ENTERED THE DOMAIN:   -0.2422543E+08 M3  ( IF <0 EXIT )
     TOTAL VOLUME LOST             :     50204.04     M3

 END OF TIME LOOP

 EXITING MPI

STOP 0                    *************************************

                     *    END OF MEMORY ORGANIZATION:    *
                     *************************************

 CORRECT END OF RUN

 ELAPSE TIME :
                             28  MINUTES
                             25  SECONDS

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

... merging separated result files

    +> steering_steady.cas

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

... handling result files

    +> steering_steady.cas
      moving: result.res

My work is done
```

Figure 51. End screen of the simulation

Open the “result.res” file in BlueKenue software to visualize the results. Drag one of the parameters into the 2D View for visualization. In Figure 52, “Velocity UV” is animated by right-clicking on Velocity UV under the “2D View” and then selecting “Animate”. Afterwards, by clicking on the play button, the animation of the simulation can be started, and the behaviour of the Velocity UV parameter of each point can be seen (Figure 53).

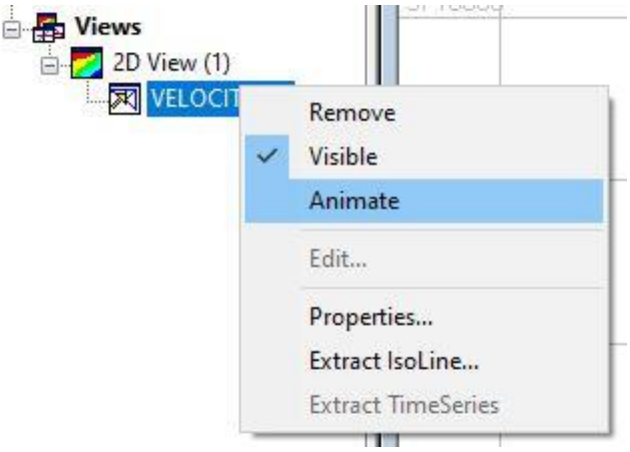


Figure 52. Animating the Velocity UV parameter

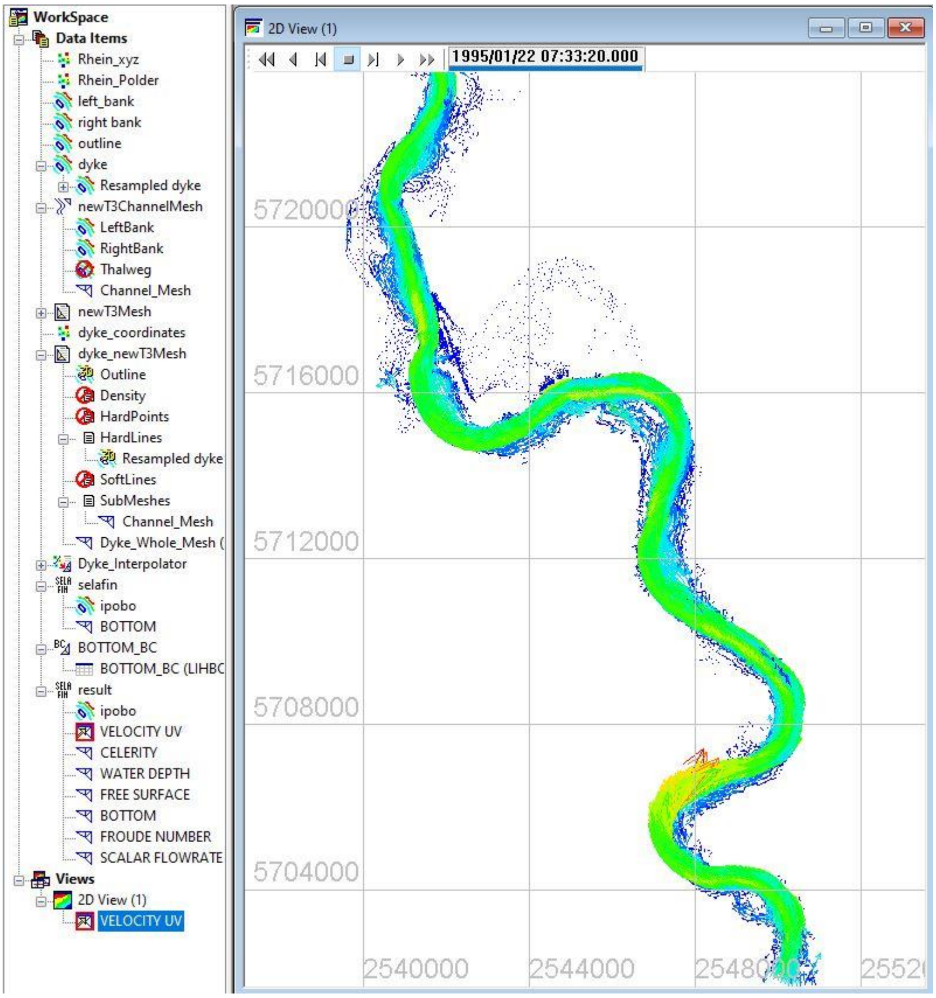


Figure 53. Velocity UV for each point at the end of simulation

A time series graph of any node can be plotted. For this purpose, click on any demanded node, then right-click and select “Extract Time Series” (Figure 54).

A new item will occur under the “Data Items” menu. Open the 1D View, then drag this new item under it. The time series plot of the node can be seen (Figure 55).

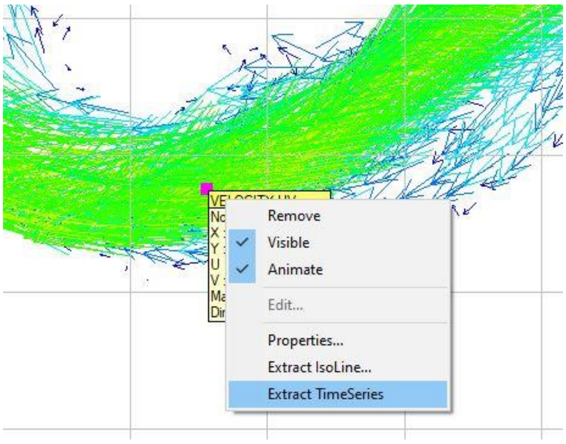


Figure 54. Creating the Time Series plot of a node

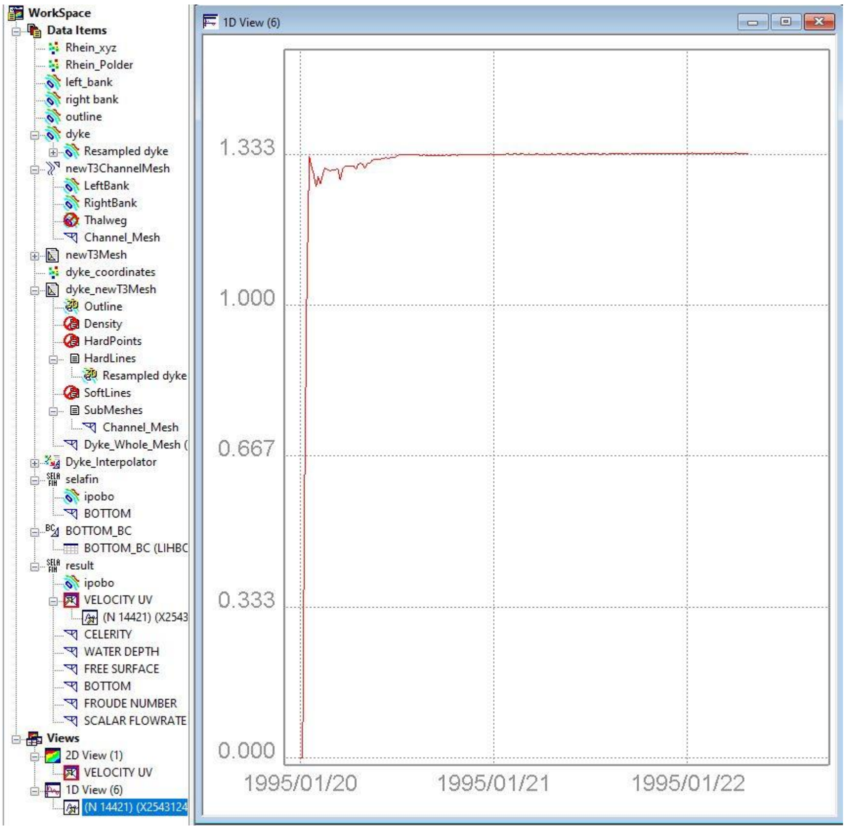


Figure 55. Time Series plot of a node

As can be seen in Figure 55, velocity increased at the beginning, and there was no critical change afterwards. This plot shows that the model has reached a steady state, and this result file can be used as a hot start file for the unsteady simulation.

Feel free to animate other parameters such as Water Depth and Free Surface and make other time series plots to understand their behaviour under steady-state conditions.

9. HOTSTART FILE

A hotstart file is used to reduce the simulation time required in an unsteady simulation. The idea is that a simulation that has already reached a steady state can be used to reduce the total simulation time. A good steady-state parametrization hotstart file should include the following variables:

- Velocity in the x-direction
- Velocity in the y-direction
- Water depth
- Free surface

Without closing BlueKenue after visualizing the results, create a new selafin object as described in Section 5. Change the name of this new selafin file to “Rhein-Hotstart”.

Drag the Velocity UV file into the 2D View and animate it as shown in Section 8. Let the animation end, because the end of it will be the last point in which the model has already achieved a steady state. After the animation ends, click on the simulation date from the animation window and see the number of the last step (it is also automatically written in the Calculator Screen of the selected variable). Select Velocity UV file and click “Tools → Calculator”. From the calculator screen, select the variable A as “Velocity UV” and select “U” (Figure 56). Write the same number to the Start box which is already written in the End box (in this simulation, it is 201 but it can differ for other simulations with different timestep). Just write “A” to the expression, give a proper name to the file in the name box and write the unit (for velocity parameter: M/S). Complete the same procedure for Velocity V and compare them visually by dragging files to the 2D view (Figure 57).

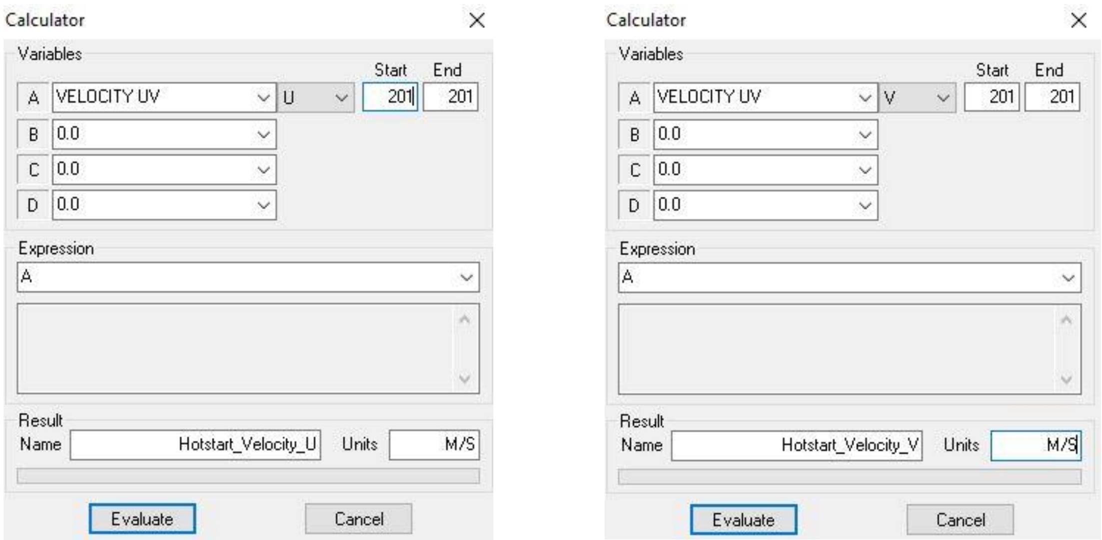


Figure 56. Calculator windows for Velocity U and Velocity V

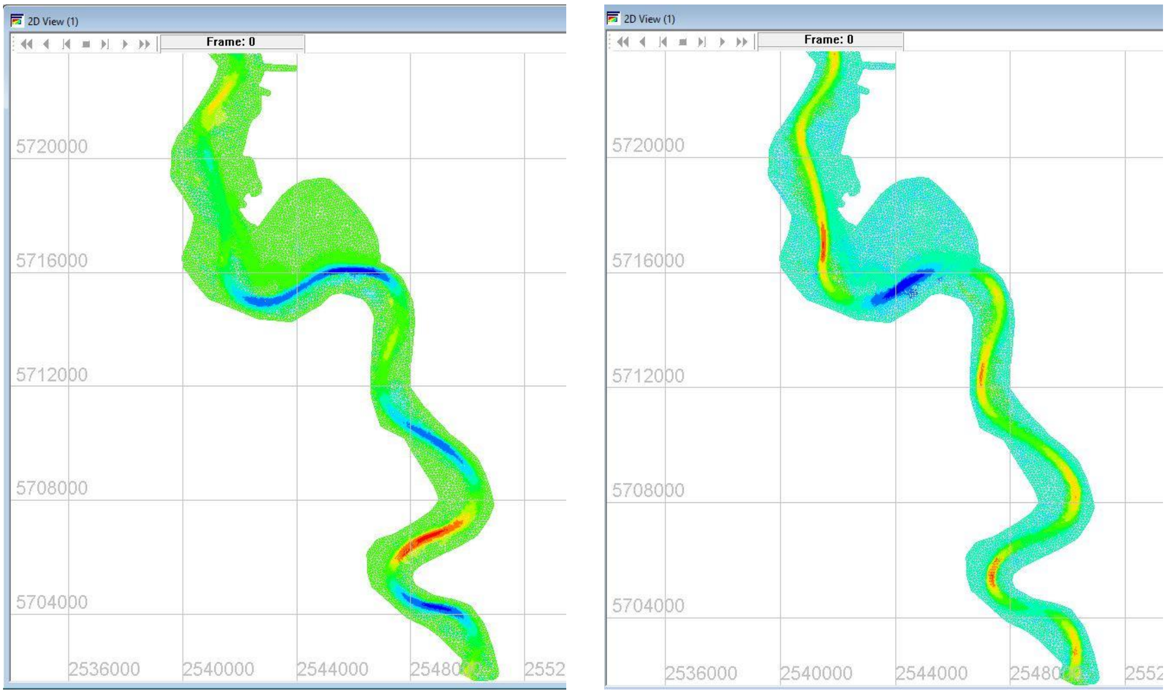


Figure 57. Velocity U (left) and Velocity V (right) files in 2D view

Now that the velocities are extracted, they need to be mapped accordingly in the new selafin object. To do this, right-click the "Rhein-Hotstart" and select "Add Variable". From the "Add New Selafin Variable" window, select Dyke_Whole_Mesh. To create a blank mesh, leave the attribute name as NodeType. Select "New Variable Properties" as Velocity U and M/S (Figure 58). After that, a new item (Velocity_U) will be created under the Rhein-Hotstart submenu. Drag Velocity_U into the 2D view and visualize the blank mesh which is completely red (Figure 60). By selecting it, go to "Tools → Map Object" and select "Hotstart_Velocity_U" (Figure 59). The velocity U data will be mapped into the blank mesh (Figure 61). Complete the same procedure for Velocity V.

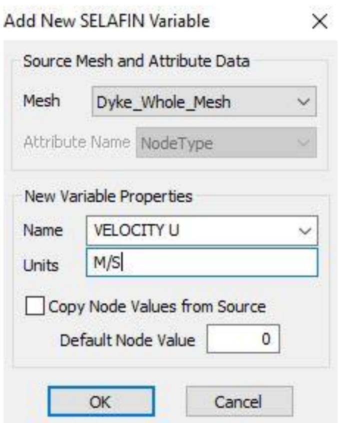


Figure 58. Creating blank map for Velocity U variable

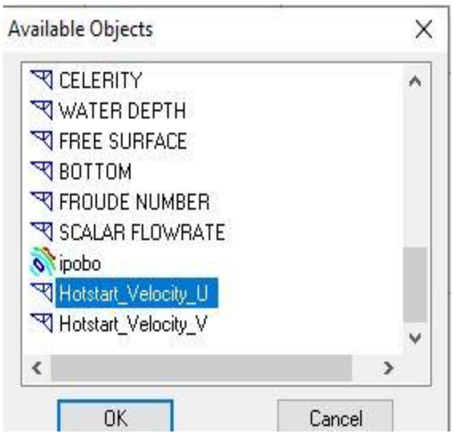


Figure 59. Mapping the blank mesh

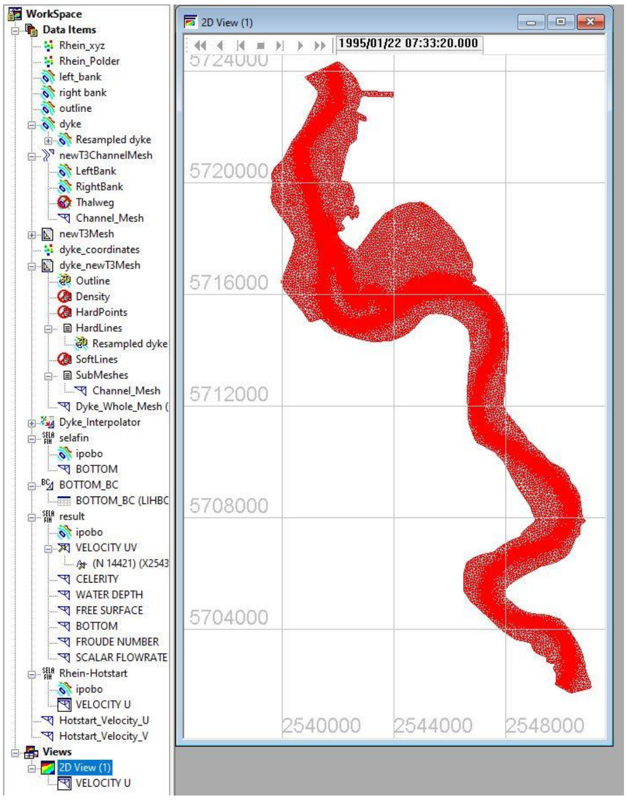


Figure 60. Blank mesh of Velocity U variable

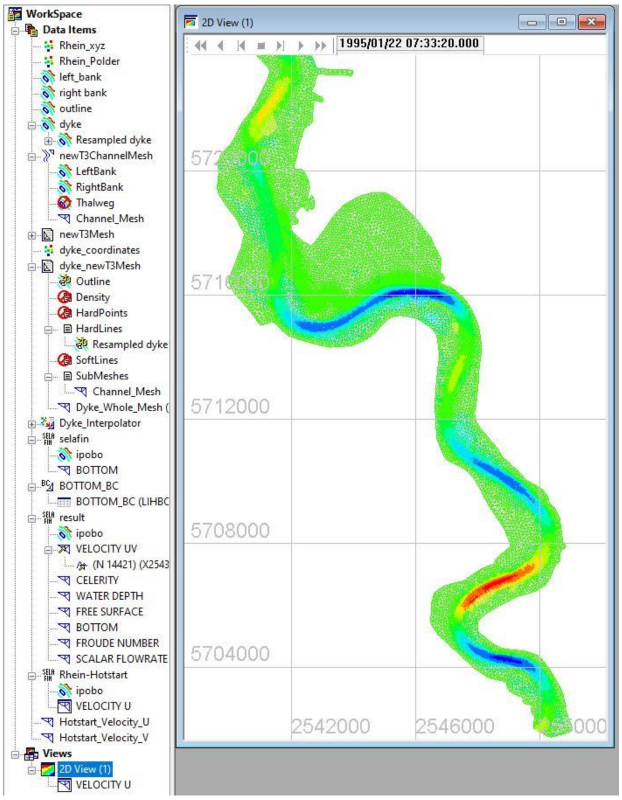


Figure 61. Mapped mesh of Velocity U variable

Complete the same steps for water depth and free surface values (Figure 62 & 63).

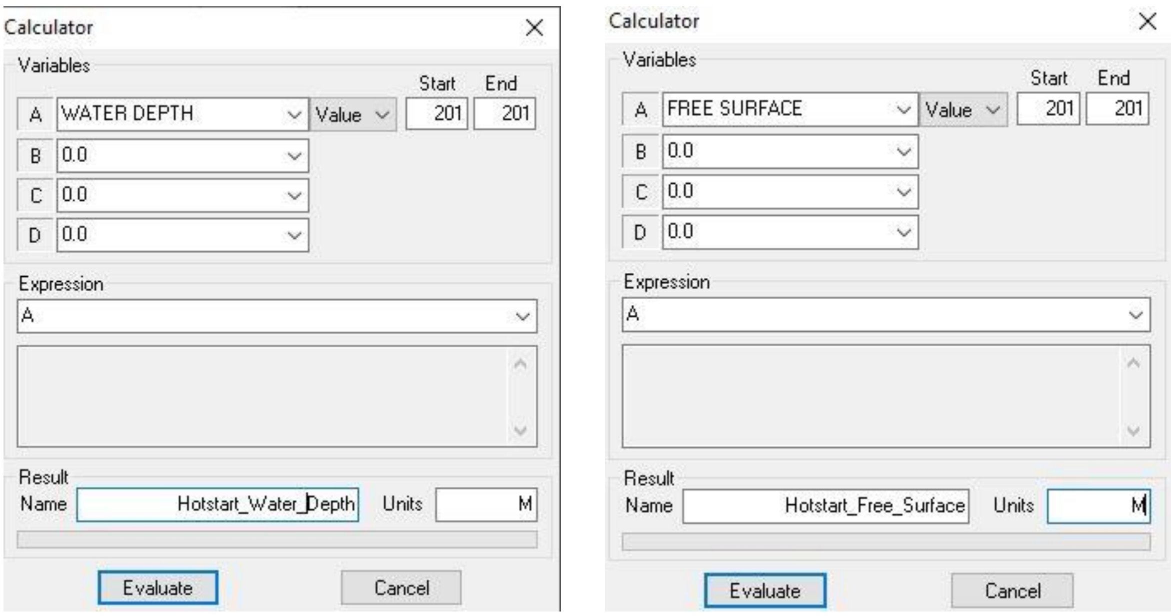


Figure 62. Calculating the last frame of Water Depth and Free Surface parameters

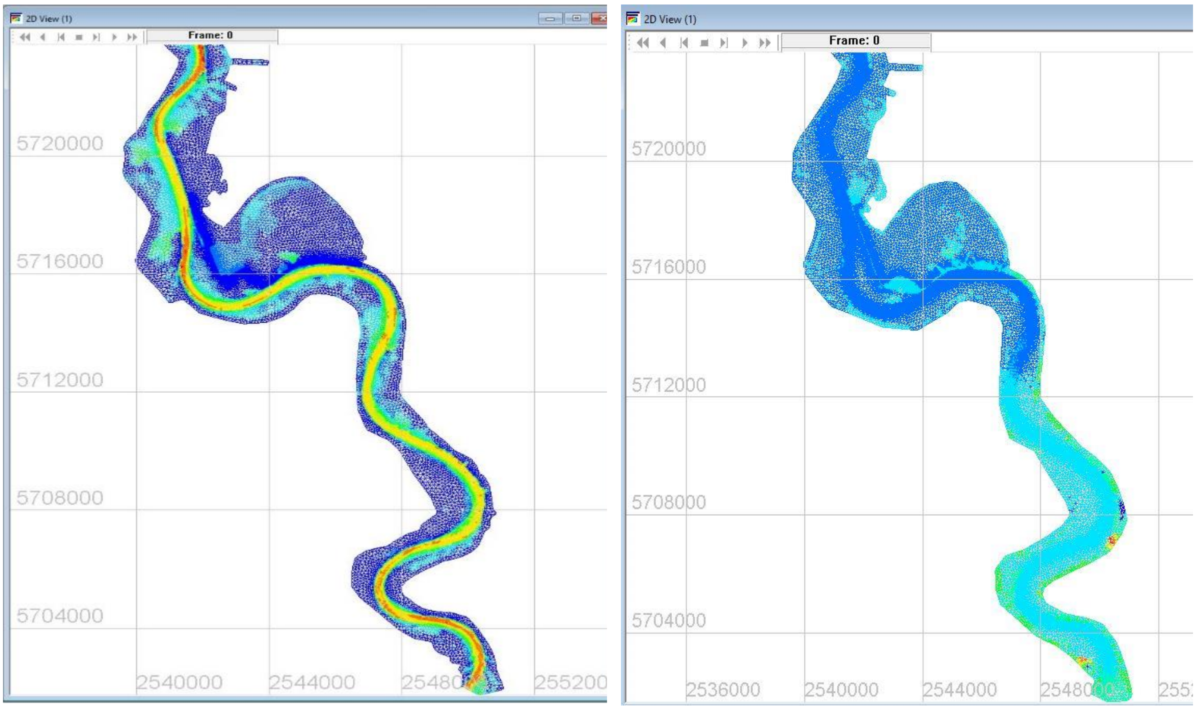
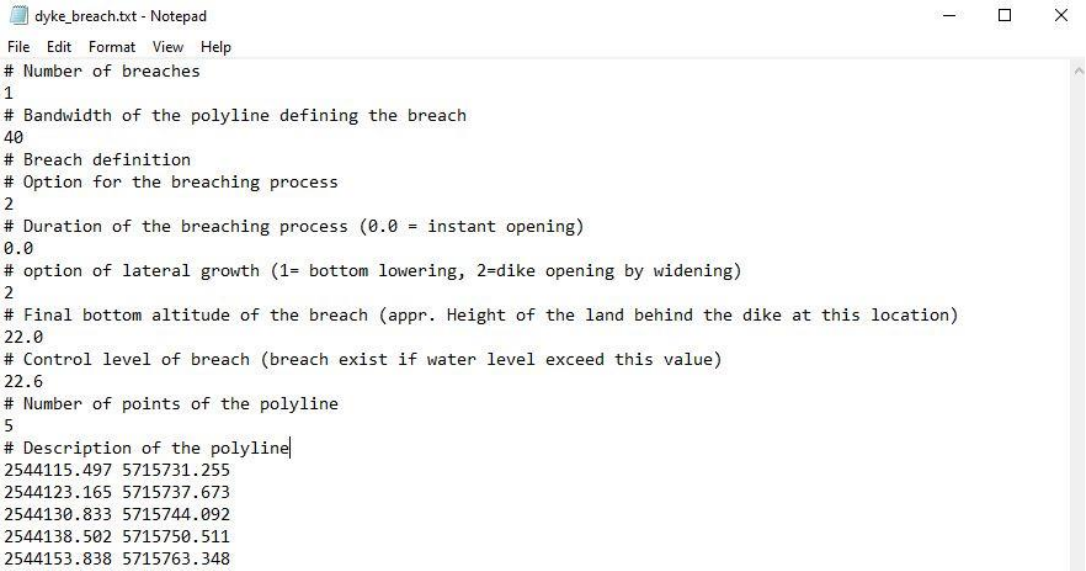


Figure 63. Water Depth (left) and Free Surface (right) hotstart data

After creating all four variables, save the Rhein-Hotstart file by clicking on Sloppy Disk Icon from the toolbar as “Rhein-Hotstart.slf”.

10. DYKE BREACH AND CONTROL SECTION FILES

To simulate a dyke breach scenario, a text file needs to be prepared. A sample file is given in the project folder named dyke_breach.txt (Figure 64).




```
# Number of breaches
1
# Bandwidth of the polyline defining the breach
40
# 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
2544115.497 5715731.255
2544123.165 5715737.673
2544130.833 5715744.092
2544138.502 5715750.511
2544153.838 5715763.348
```

Figure 64. Dyke breach text file

The necessary information should be given in this file.

- For this simulation, the number of the breach is given as 1. The width of the breach is adjusted under the bandwidth title. It is selected as 40 m. However, feel free to analyse the effect of different dyke breach widths on the water volume that enters to polder.
- To have an instant breach, the duration of the breaching process is given as 0 seconds. Yet, a specific duration can be given to have a delayed breach.
- Option of lateral growth is selected as 2, dyke opening by widening. The bottom lowering option is better for simulations of sedimentation and sediment erosion processes.
- Final bottom altitude and control level of breach parameters determine the height of the breach. According to the text, dyke breach will happen when the water level exceeds 22.6 m and dyke height on the breach area will drop to 22 m. Thus, dyke breach height is 0.6 m. Feel free to change this height for further analysis.
- In Dyke Height Adjustment, points along the dyke were resampled with a 10 m gap between each. Thus, nodes along the dyke have a 10 m difference. To have a 40 m dyke breach width, coordinates of five consecutive nodes should be given. The number of points of the polyline should be adjusted to five for this case, and coordinates should be written in the "Description of the polyline" part.
- Selection of the dyke breach place depends on the analysis. For this project, dyke breach place is selected a bit near the upstream part (Figure 65). Feel free to change the dyke breach place for further analysis.
- To select the dyke breach place, drag "resampled dyke" and "Dyke_Whole_Mesh" into the 2D view. Select the consecutive nodes that are under the resampled dyke line (Figure 65-66-67).

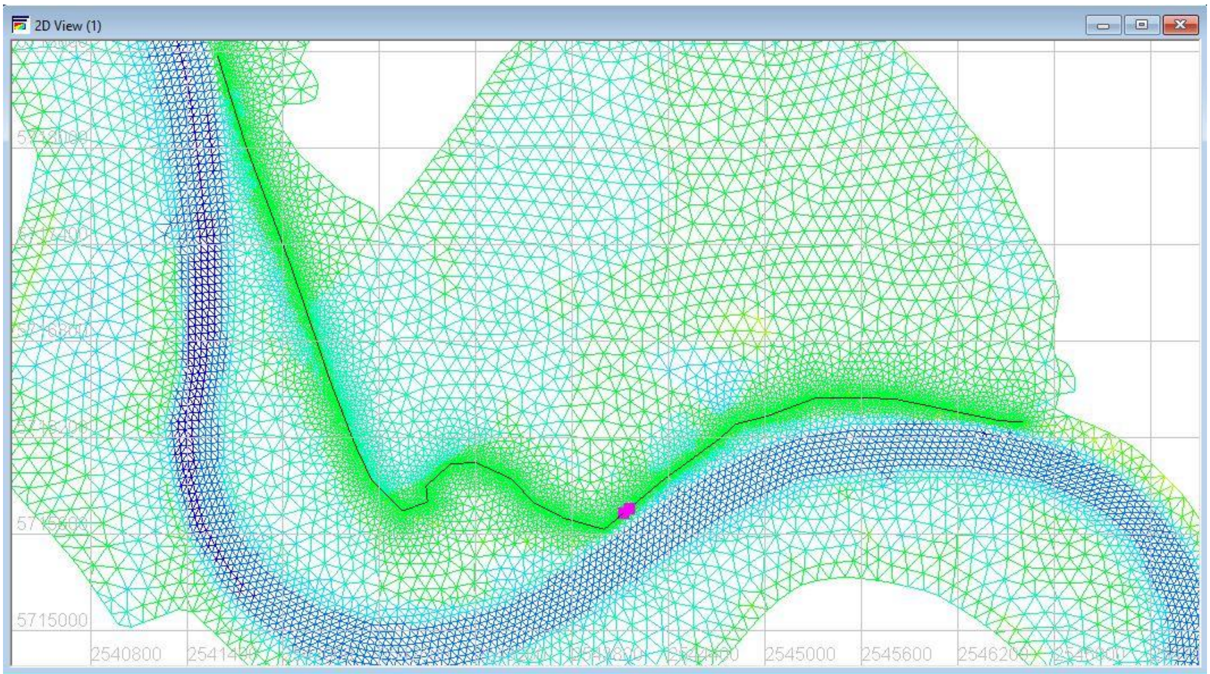


Figure 65. Selecting the dyke breach place

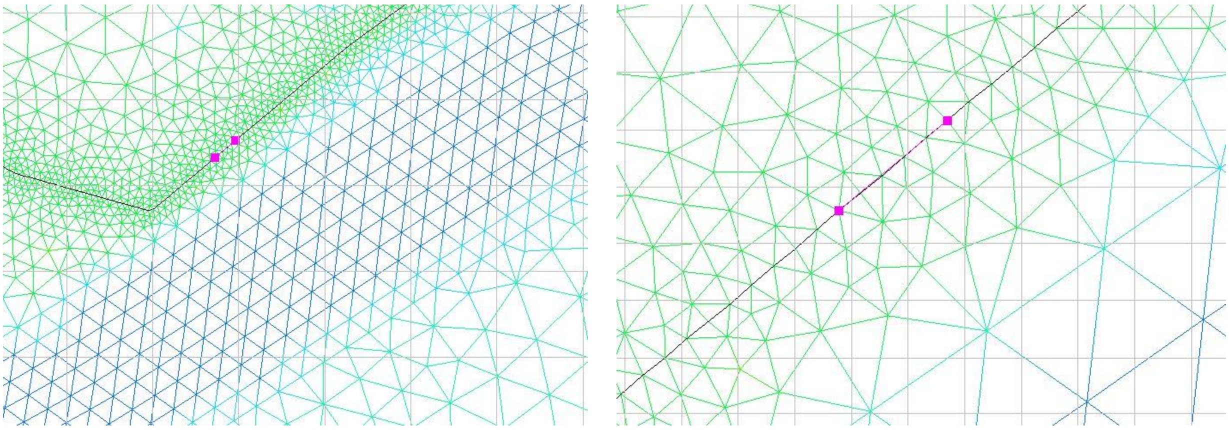


Figure 66. Zoomed view of dyke breach place

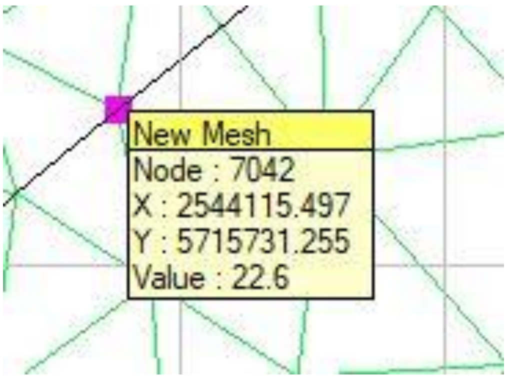
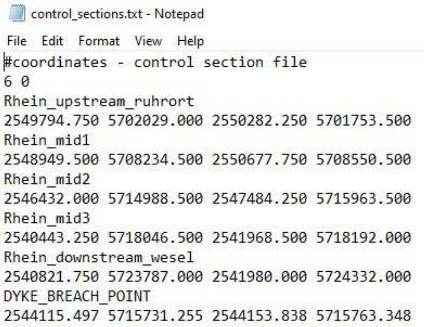


Figure 67. Select the node by clicking the junction point of the triangles

TELEMAC allows control sections to be included either as node number within the mesh or coordinates in the domain. The result information for each section is the instantaneous flow rates and cumulated positive and negative flow rates. The section input file (control_sections.txt in the project folder) is similar to the dyke breach with specific formats depending if the information is included for a point or coordinates (Figure 68).



```
control_sections.txt - Notepad
File  Edit  Format  View  Help
#coordinates - control section file
6 0
Rhein_upstream_ruhrort
2549794.750 5702029.000 2550282.250 5701753.500
Rhein_mid1
2548949.500 5708234.500 2550677.750 5708550.500
Rhein_mid2
2546432.000 5714988.500 2547484.250 5715963.500
Rhein_mid3
2540443.250 5718046.500 2541968.500 5718192.000
Rhein_downstream_wesel
2540821.750 5723787.000 2541980.000 5724332.000
DYKE_BREACH_POINT
2544115.497 5715731.255 2544153.838 5715763.348
```

Figure 68. Control section file

On the second line, it is written that six different control sections will be given into TELEMAC by using the coordinates of the nodes (0 represents this choice. If the number of the node will be used, adjust this number to 1). Start and End Point X-Y coordinates of the control section lines are given under the relevant control section name. For this work, control sections represent upstream, three different sections along the river, downstream and the breach place (Figure 69). Middle section lines are user dependent. To obtain their coordinates, draw an open line as in Figure 70, then write down the coordinates of the both ends, under the relevant title in the text file.

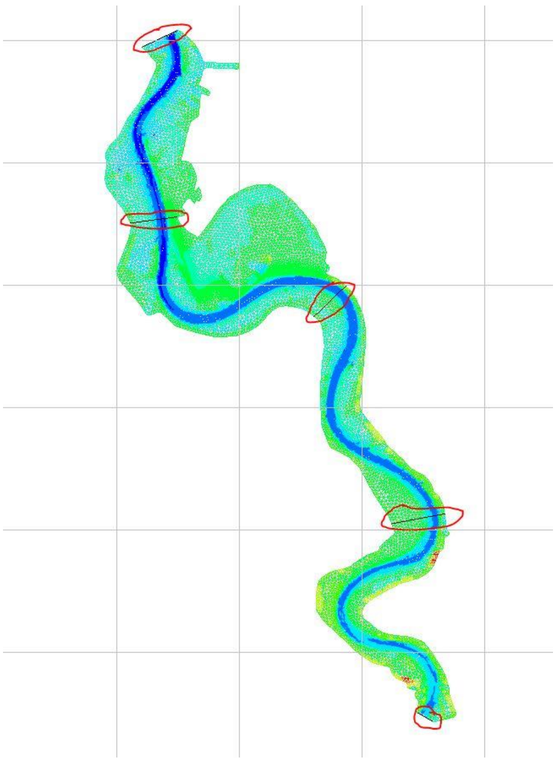


Figure 69. Control sections

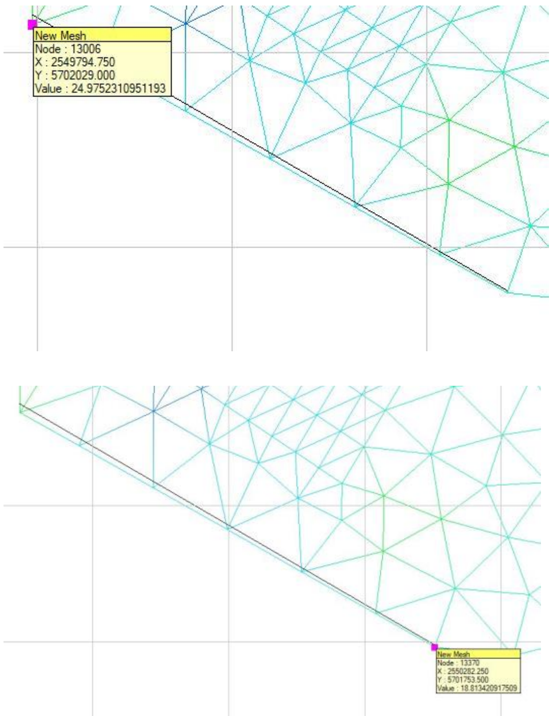


Figure 70. Determining the X-Y coordinates of both ends

11. ADJUSTING THE CAS FILE FOR UNSTEADY SIMULATION

For running an unsteady simulation, new requirements will be added as keywords in the CAS file via opening it in Fudaa Prepro. This time, a liquid boundary file will be used as the upstream boundary condition while the downstream boundary condition is still a Q-h file.

Equations

Adjust “Turbulence Model” as 3: K-EPSILON model.

Equations, Boundary Conditions

Adjust prescribed Flowrates as 0;0.

Equations, Initial Conditions

Delete the initial conditions because a liquid boundary conditions file will be inserted. If it cannot be done from Fudaa PrePro, delete this line from the CAS file by opening it with a text editor.

Input-Output, Files

Geometry, Stage-Discharge Curve and Boundary Conditions files will remain the same as in the steady simulation. However, rename the steering file as “steering_unsteady.cas” and the results file as “result_unsteady.res”. In addition, add the following files:

1) Breaches Data File: “dyke_breach.txt”

2) Sections Input File: “control_sections.txt”

3) Liquid Boundaries File: “5430_s1rc88ndhs2c0.liq”

4) Previous Computation File: “Rhein-Hotstart.slf”

5) Sections Output File: This file will be created by the user. Give a proper name, for example, “control_sections_output.txt”.

Input-Output, Graphics and Listing

Adjust Graphic Printout Period and Listing Printout Period as 720.

Input-Output, Information

Change the title to “Unsteady Simulation”. Check the “Computation Continued” box.

Numerical Parameters

Adjust “Original Date of Time” as 1995;01;20, since it will be the starting date of data in unsteady simulation.

Put a checkmark on the boxes for “Compatible Computation of Fluxes”, “Initial Time Set to Zero” and “Variable Time Step”.

Select “2: Wave Equation” for “Treatment of The Linear System” and 2 (Flux Control) for “Treatment of Negative Depths”.

Adjust “Desired Courant Number” to 0.7 and increase “Duration” to 2,000,000 seconds (around 23 days). Change the “Time Step” to 240, even if it is not relevant since the “Variable Time Step” is open.

Adjust SUPG Option as 2;0;2;2.

Adjust “Original Date of Time” as 1995;1;20.

<u>Numerical Parameters, K-Epsilon Model</u>

Adjust “Maximum Number of Iterations for K and Epsilon” as 100.

<u>Numerical Parameters, Velocity-Celerity-Hight</u>

Adjust “Mass-Lumping on H” as 1.

<u>Physical Parameters</u>

Mark the box of the “Breach” parameter.

12. RUNNING THE SIMULATION & RESULTS AND SECTION OUTPUT FILE

The process for running the simulation is similar to the steady simulation. The only different thing is the name of the CAS file. Open Telemac prompt window and enter the file location. Then, write “telemac2.py steering_unsteady.cas” (without “ ”).

This simulation will last relatively longer than the steady simulation. When the simulation has ended (Figure 71), open the results file in BlueKenue to visualize it. Different from the steady simulation, Q values at time t for entered control sections will be given for each iteration during the unsteady simulation.

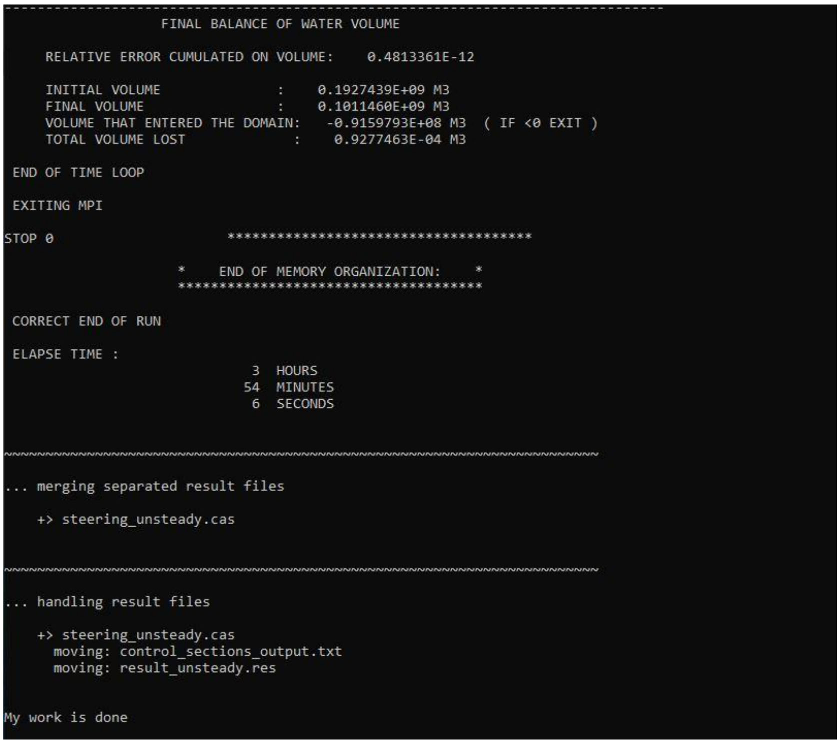


Figure 71. Final screen of the unsteady simulation

Open the results file in BlueKenue. Drag Velocity UV file into 2D View and animate it. Water entrance to the polder during the flood can be seen in Figure 72. The change of velocity in the meandering parts upstream can be seen in this Figure, too.

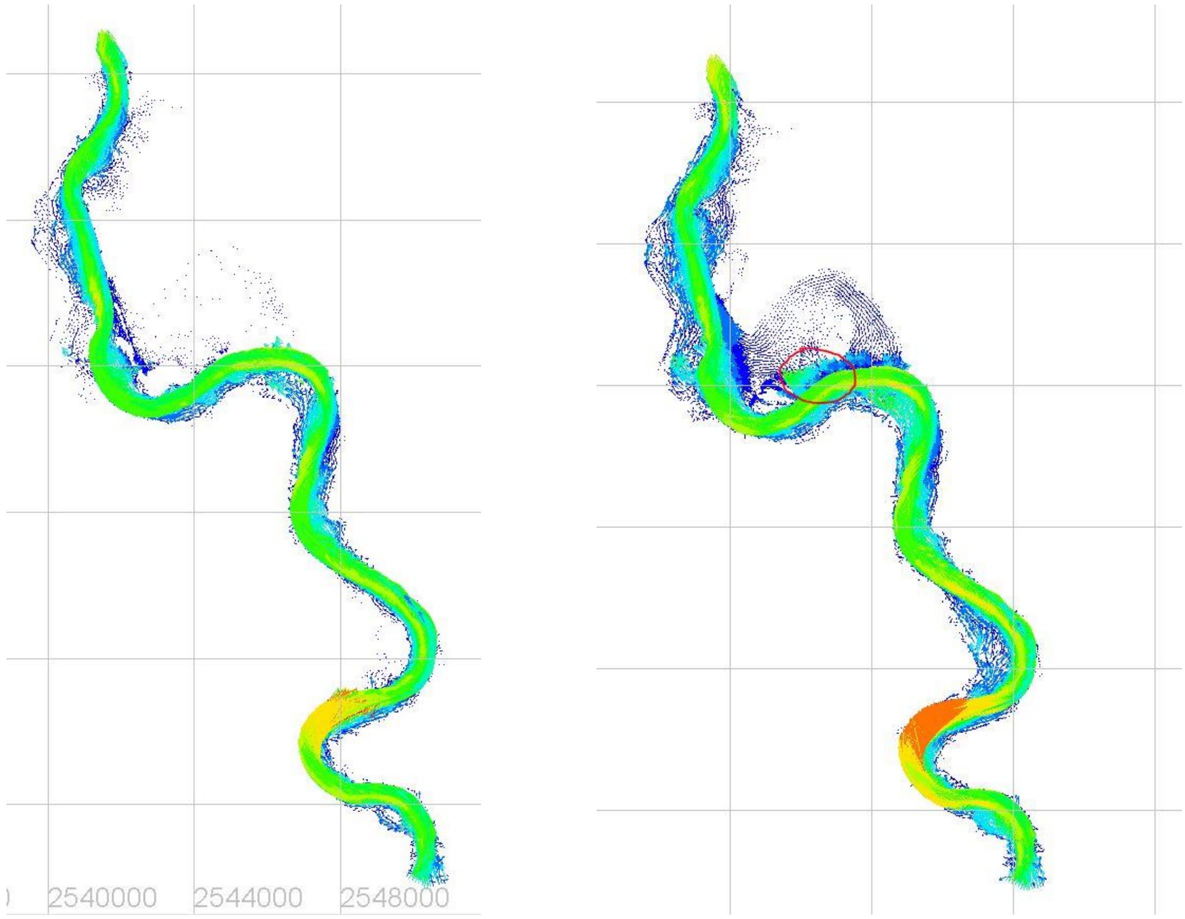
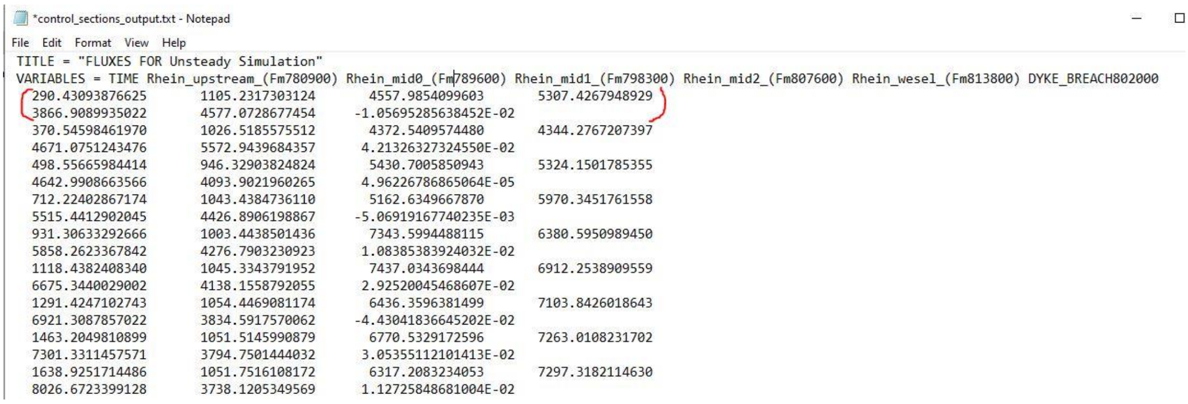


Figure 72. Visualizing the water entrance into the polder

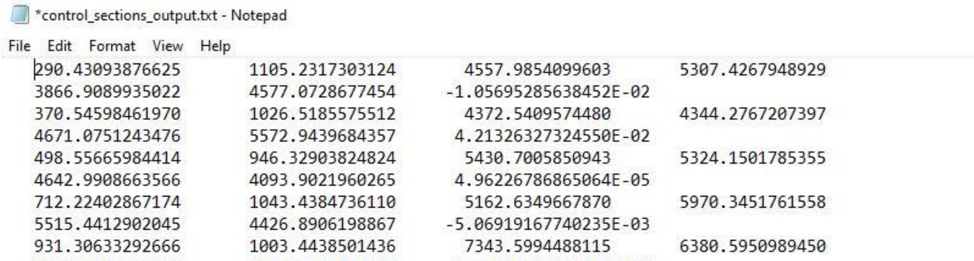
However, visualization does not provide the necessary information. The amount of water that enters the polder should be determined. For this purpose, cross_section_output.txt is used. TELEMAC created this document after the specific code given in the CAS file. Nonetheless, the interpretation of the numbers is hard due to the way the text file is created. The output file gives the time step and the discharge for each control sections at a particular time in the same line (Figure 73). To solve this issue, a python script is provided in the project folder for rearranging this data as a pandas data frame (Figure 75). Comparison graphs can be created from here or from Excel by carrying the data into it (Figure 76).

<u>Before running the code, please delete the first two lines from the text file (Figure 74).</u>



```
*control_sections_output.txt - Notepad
File  Edit  Format  View  Help
 TITLE = "FLUXES FOR Unsteady Simulation"
 VARIABLES = TIME Rhein_upstream_(Fm780900) Rhein_mid0_(Fm789600) Rhein_mid1_(Fm798300) Rhein_mid2_(Fm807600) Rhein_wesel_(Fm813800) DYKE_BREACH802000
  290.43093876625        1105.2317303124       4557.9854099603        5307.4267948929
  3866.9089935022        4577.0728677454      -1.05695285638452E-02
  370.54598461970        1026.5185575512       4372.5409574480        4344.2767207397
  4671.0751243476        5572.9439684357       4.21326327324550E-02
  498.55665984414        946.32903824824       5430.7005850943        5324.1501785355
  4642.9908663566        4093.9021960265       4.96226786865064E-05
  712.22402867174        1043.4384736110       5162.6349667870        5970.3451761558
  5515.4412902045        4426.8906198867      -5.06919167740235E-03
  931.30633292666        1003.4438501436       7343.5994488115        6380.5950989450
  5858.2623367842        4276.7903230923       1.08385383924032E-02
  1118.4382408340        1045.3343791952       7437.0343698444        6912.2538909559
  6675.3440029002        4138.1558792055       2.92520045468607E-02
  1291.4247102743        1054.4469081174       6436.3596381499        7103.8426018643
  6921.3087857022        3834.5917570062      -4.43041836645202E-02
  1463.2049810899        1051.5145990879       6770.5329172596        7263.0108231702
  7301.3311457571        3794.7501444032       3.05355112101413E-02
  1638.9251714486        1051.7516108172       6317.2083234053        7297.3182114630
  8026.6723399128        3738.1205349569       1.12725848681004E-02
```

Figure 73. Sections output text file



```
*control_sections_output.txt - Notepad
File  Edit  Format  View  Help
  290.43093876625     1105.2317303124      4557.9854099603     5307.4267948929
  3866.9089935022     4577.0728677454     -1.05695285638452E-02
  370.54598461970     1026.5185575512      4372.5409574480     4344.2767207397
  4671.0751243476     5572.9439684357      4.21326327324550E-02
  498.55665984414     946.32903824824      5430.7005850943     5324.1501785355
  4642.9908663566     4093.9021960265      4.96226786865064E-05
  712.22402867174     1043.4384736110      5162.6349667870     5970.3451761558
  5515.4412902045     4426.8906198867     -5.06919167740235E-03
  931.30633292666     1003.4438501436      7343.5994488115     6380.5950989450
```

Figure 74. The text file should look like this before running the python script

final - DataFrame

Index	TIME	stream (Fı	ı mid0 (Fm789	in mid (Fm7983	in mid2 (Fm807(	wesel (Fm81	E BREACH802
0	290.431	1105.23	4557.99	5307.43	3866.91	4577.07	-0.0105695
1	370.546	1026.52	4372.54	4344.28	4671.08	5572.94	0.0421326
2	498.557	946.329	5430.7	5324.15	4642.99	4093.9	4.96227e-05
3	712.224	1043.44	5162.63	5970.35	5515.44	4426.89	-0.00506919
4	931.306	1003.44	7343.6	6380.6	5858.26	4276.79	0.0108385
5	1118.44	1045.33	7437.03	6912.25	6675.34	4138.16	0.029252
6	1291.42	1054.45	6436.36	7103.84	6921.31	3834.59	-0.0443042
7	1463.2	1051.51	6770.53	7263.01	7301.33	3794.75	0.0305355
8	1638.93	1051.75	6317.21	7297.32	8026.67	3738.12	0.0112726
9	1812.89	1055.64	6147.63	7760.24	8232.12	3670.29	0.0133415
10	1992.02	1060.31	5974.09	8012.19	8398	3711.36	-0.00999995
11	2161.22	1055.13	5981.5	7919.24	8461	3736.12	0.017046
12	2336.01	1061.1	5598.3	7531.06	8249.21	3797.79	-0.000440728
13	2508.51	1066.93	5286.45	7402.4	8118.23	3815.08	0.00314719

Figure 75. Rearranged dataframe

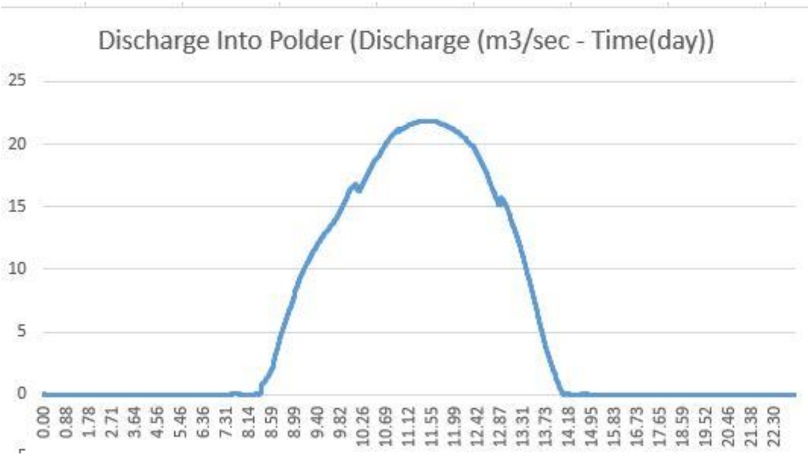


Figure 76. Discharge graph from Excel

13. FINAL REMARKS

- To familiarize yourself with the software, repeat the processes from the scratch a couple of times. Once the drawings are completed, they can be used for different simulations. However, a new mesh is needed for some changes, for example, resampling the dyke line because the number of nodes will change. Thus, a new hotstart file should be created, too.

- To decrease the simulation time, the dyke line can be interpolated with greater values. However, try to avoid implausible numbers.

- While adjusting the duration for the unsteady simulation, take the length of the liquid boundary condition file into consideration. If data in the liquid boundary file ends before the simulation, the program will crash when it cannot take any data. It means the program can run for hours and then crash due to a lack of new data. To not waste your time, adjust your duration so that the liquid boundary data covers the whole simulation.

- Do not forget to locate all necessary files into the same folder before running the simulation.

- In case any problem happens and the simulation does not work, you can look for the solution in the TELEMAC-MASCARET forum:
  http://www.opentelemac.org/index.php/assistance/forum5/16-telemac-2d