Isabela State University

TRAINING MANUAL

Flow and Sediment Transport Modelling in River Basins using TELEMAC 2D and 3D Numerical Codes

ifwarm@isu.edu.ph

IFWaRM
Integrated Flood and Water Resources Management

Training Manual on Flow and Sediment Transport Modelling in River Basins using TELEMAC 2D and 3D Numerical Codes

Training Manual

Isabela State University

Water Resources Development Center (WReDC)

Integrated Flood and Water Resources Management in ASEAN Basins for Integrated Development (IFWaRM)

Cagayan State University

Kyoto University-Disaster Prevention Research Institute (DPRI)

Asia Pacific Network (APN)

Department of Science and Technology

National Irrigation Administration

Contents

List of Figures

Rationale

Climate change threatens the world with devastating floods and droughts, with Japan, Vietnam, and the Philippines among the worst-affected countries.

Human interventions such as deforestation, dam construction, and irrigation system development have increased the adverse effects of climate change. Furthermore, flood and sediment management in the river basin scale has also become a crucial issue for the Cagayan River Basin in the Philippines.

To help resolve these issues, reliable numerical tools are needed. Not only these numerical models may be used to characterize flood inundation maps, but they can also be used to create mitigation measures and assist in successful land use and emergency planning. In the same light, significant efforts have been made to use remotely sensed data (e.g., Classified Aerial Imagery, Light Detection and Ranging (LiDAR) Terrain Measurements) capable of providing high-resolution Digital Model Terrain (DTM) in 2D or even in 3D space, thereby guiding mesh generation and model parameterization.

TELEMAC's application range is very wide and comes with a range of modules for two-dimensional (2D) and three-dimensional (3D) modeling of hydro-morphodynamic processes of various water bodies from mountain rivers to coastal deltas under the influence of tides. Multiple sediment transport phenomena can also be modeled, as well as steady and unsteady flow conditions.

Consequently, this training workshop provides an overview of the capabilities, limitations, and challenges related to hydraulic modelling of flow and sediment transport in rivers. As a result, Isabela State University in the Philippines hosted a workshop training course in conjunction with Kyoto University. The details of the training course and other relevant information are given hereafter.

Objectives

After completing this training, the participants are expected to:

1. Generate a computational mesh from a topographic data set using free pre-processing software Blue Kenue (www.nrc-cnrc.gc.ca/eng/solutions/advisory/blue_kenue_index.html);
2. Create input files to execute TELEMAC2D and TELEMAC3D;
3. Assess the impact of imposed flow on near- and far-field river hydrodynamics; and
4. Analyze and visualize TELEMAC output files using third-party charting and data analysis software.

TELEMAC - MASCARET

TELEMAC-MASCARET or simply TELEMAC, is an integrated suite of solvers for use in the field of free-surface flow, waves, water quality, and sediment transport (Galland, Goutal, & Hervouet, 1991). Having been used in the context of many studies throughout the world, TELEMAC has become a major standard in its field. It was made "open source" in 2010 to be freely available to the whole community of consultants, universities, and researchers while being actively supported and continuously developed by a consortium of industrials, consultants, and research organizations. Anyone can, therefore, take advantage of TELEMAC and assess its performance. For more information about TELEMAC-MASCARET see www.openTELEMAC.org.

Furthermore, this training course covers a broad range of modelling techniques for fluvial processes using the TELEMAC suite of computer models. Lectures will provide both theoretical background and practical aspects of modelling flow and sediment transport processes. Introductory topics, which can be grasped by the participants without a background in numerical methods, will be followed by advanced topics such as mesh generation/optimization, selection of numerical schemes, and flow and sediment interactions.

Additionally, the capabilities and benefits of the TELEMAC modelling system will be shown by a hands-on application using data sets acquired from the Magat rivers

Main Applications

TELEMAC can be used in the following but not limited to:

(Adapted from the www.openTELEMAC.org website)

- Flood Studies
- Estuary Management and Development
- River Restoration Studies
- Dam Bursts
- Hydro Sedimentary Studies
- Water Quality
- Dumping of dredged materials
- Impact of Hydraulic Structures
- Study of Marine Climates
- Harbor Installation Studies

Modules

(Adapted from the www.openTELEMAC.org website)

1. GAIA (SISYPHE) - Sediment Transport and Bed Evolution

   It is a state-of-the-art sediment transport and bed evolution module of the TELEMAC-MASCARET modelling system which can be used to model complex morphodynamics processes in diverse environments such as coastal, rivers, lakes, and estuaries for different flow rates, sediment size classes, and sediment transport modes.
2. NESTOR- Modelling Dredging Operations in the River Bed

   Using documented data from dredging operations to model the ensuing changes in the bottom level is possible with NESTOR and the morphodynamic module SISYPHE. The period, the volume of silt removed during that period, and the size and location of the dredging site can be defined. This information can then be utilized to calculate the site's bottom level.
3. MASCARET- One-Dimensional Free Surface Flow Modelling

   MASCARET includes one-dimensional free surface flow modelling engines. Different modules can describe varied phenomena over large areas and for various geometries using the Saint-Venant equations: meshed or branched networks, subcritical or supercritical flows, and steady or unsteady flows.
4. TELEMAC-2Ds

   TELEMAC-2D is a two-dimensional horizontal space simulation program that simulates free-surface flows. At each point of the mesh, the program calculates the depth of water and the two velocity components.
5. TELEMAC- 3D

   TELEMAC-3D is a three-dimensional (3D) model based on TELEMAC-2D's horizontally unstructured mesh. The wave formulation for the updating of the free surface is used for efficiency. The model mesh is developed as a series of model planes between the bed and surface planes. Furthermore, the usage of a sigma grid is possible due to the freedom in the placement of these planes (each plane at a given proportion of the spacing between bed and surface) or several other strategies for intermediate plane location. Include certain planes that are at a fixed distance below the water surface or above the bed, for example. In the presence of a near surface thermocline or halocline, this is advantageous in so far to avoid the mixing of water between the near surface planes where the greatest density gradients are located.
6. TOMAWAC

   Wave propagation in coastal locations is modelled using TOMAWAC. It solves a simplified equation for the spectro-angular density of wave activity using a finite-element approach. This is done for steady-state conditions (i.e., with a fixed depth of water throughout the simulation).

7. ARTEMIS

   ARTEMIS is a scientific software dedicated to the simulation of wave propagation towards the coast or into harbors over a geographical domain of about a few square kilometers. Long waves or resonance simulations may require a bigger domain. The frequency dependence and directional spreading of the wave energy are considered by ARTEMIS.

i. Pre-requisites

Individuals having a basic understanding of hydrological/ hydraulic modelling, rainfall-runoff analysis, river catchment, open-channel flow, and sediment transport mechanics would benefit greatly from this training session. It can serve as an introduction to numerical modelling of flow and sediment transport in rivers.

ii. Training Guide

This training manual is based on the user manuals provided by the TELEMAC developers at http://wiki.opentelemac.org and the Hydro-Informatics website, https://hydro-informatics.com/index.html by Sebastian Schwindt © Copyright 2022.

TELEMAC's application range is very wide and comes with a range of modules for two-dimensional (2D) and three-dimensional (3D) modeling of hydro-morphodynamic processes of various water bodies from mountain rivers to coastal deltas under the influence of tides. Multiple sediment transport phenomena can also be modeled and connected with constant or variable flow conditions.

On this note, this manual will only feature the following tutorials:

- Generating a geometry file in Selafin *.slf* format using BlueKenue
- Creating a boundary condition *.cli* file using BlueKenue
- Creating and understanding a Steering *.cas* file.
- Setup a purely hydrodynamic, steady Telemac2d simulation as a first scenario
- Couple hydrodynamics (i.e., Telemac2d) with morphodynamics (i.e., Sediment transport) as the second and third scenario simulations.

iii. Software Resources Options

Option I – Automatic Installer

This provides a complete setup of the latest version of the software, including all its dependencies and prerequisites: MED, MUMPS, GOTM, AED2, METIS, MS-MPI, GFortran and Python, as well as all Python packages required to run TELEMAC examples and notebooks.

http://www.opentelemac.org/index.php/component/jdownloads/summary/23-installation-files/3431-telemac-v8p4-setup?Itemid=54

Option II - Manual Installation of TELEMAC-MASCARET and Prerequisites

For developers and Linux users, the other option is to clone the latest source code from the Git Repository and build the modules under your own operating system and Fortran compiler. This option provides more flexibility for power users and is also recommended for those who want to stay permanently up to date with the latest corrections and improvements to the TELEMAC codebase.

https://gitlab.pam-retd.fr/otm/telemac-mascaret/-/releases

iv. General Computer Requirements

- Intel Core i5-8th Gen or Ryzen 5 3000 Series processors and above
- RAM: 8 GB Minimum; 32 GB and above is much preferred
- Disk Space: at 2.22 GB of free disk is required
- CPU Cores: 4 cores minimum; 8 cores and above are preferred
- Processor speed: 2.3 GHz and above

1. The TELEMAC File Structure

1.1. Mandatory Files

1.1.1. The Geometry File

The geometry file in *.slf (selafin or SERAFIN) or *.med (MED file) format contains binary data about the mesh with its nodes. It can be created using Blue Kenue or through SALOME-HDRO.

1.1.2. The Boundary Conditions File

Information regarding inflow and outflow nodes is contained in the boundary file in *.cli format (coordinates and IDs). Any text editor can open and modify the *.cli file, but this is not suggested to avoid inconsistencies. For generating and/or editing *.cli files, Fudaa PrePro or Blue Kenue are recommended.

1.1.3. The Steering File

The steering file is the main simulation file that contains information on required files (such as selafin geometry and boundary), optional files, and simulation parameters. A basic text editor or complex software such as Fudaa PrePro or Blue Kenue can be used to create or change the steering file.

1.2. Optional Files

1.2.1. Stage-discharge (or WSE-Q) File (txt - ASCII)

It defines a stage-discharge file to use a stage (water surface elevation WSE) - discharge relationship for boundary conditions. Such files typically apply to the downstream boundary of a model at control sections (e.g., a free overflow weir).

Also, stage-discharge files can be used by defining the following keyword in the steering file: "STAGE-DISCHARGE CURVES FILE: YES".

1.2.2. Friction Data File (tbl/txt - ASCII)

This optional file allows for the definition of bottom friction, as well as the corresponding function coefficients and roughness law.

1.2.3. The Results/Restart file (SLF or MED)

A restart file stems from a previous TELEMAC simulation but is not needed to exist before any simulations. It can, however, speed up simulations.

Definitions adapted from the Hydro-informatics website https://hydro-informatics.com/index.html.

2. Installation Procedures

Note: Enable .NET Framework 3.5 (Includes .NET 2.0 and 3.0)

MPICH2, one of the prime components of TELEMAC requires the older version .NET Framework 2, which must be enabled through the Windows Features. This will prevent some future errors regarding using the MPICH.

Please see Appendix I for instructions.

2.1. TELEMAC Installation

Notes:

- If the software is already installed, please proceed to the next section.
- Account for at least 2 hours (depending on internet speed) for the installation.
- Internet connection is required.

1. Open your browser and head to www.opentelemac.org and Click "**Log in**".

Figure 1 The open TELEMAC-MASCARET webpage

2. To be able to download a copy of the installer, you must create an account first and login. Create your account by clicking "**REGISTER**".

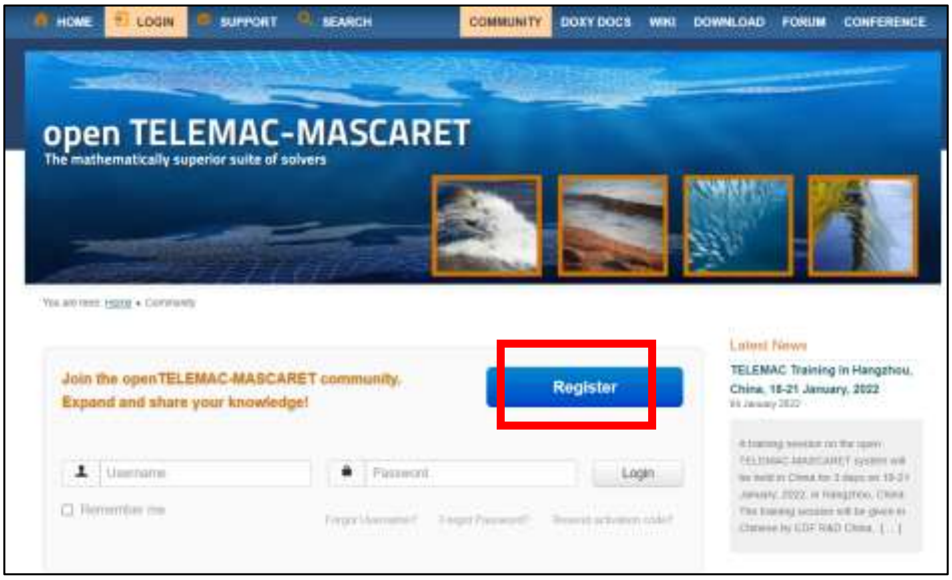
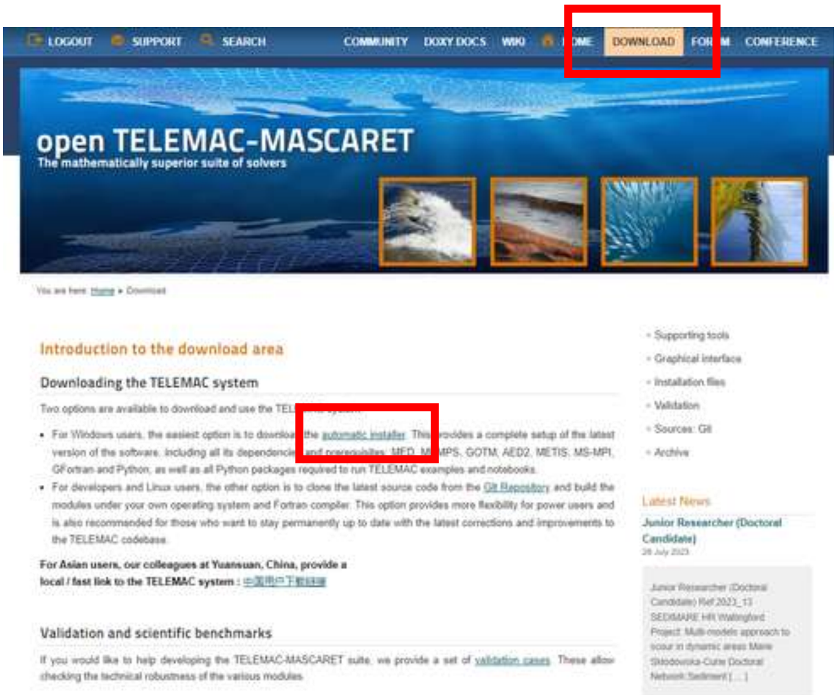


Figure 2 Registration Section Used in Creating and Logging in to Account

3. Once logged in, head to the "**DOWNLOAD**" section and find the "**automatic installer**" option.

Figure 3 The Automatic Installer Link



4. Scroll down until the "**TELEMAC V8P4 Setup**" section and click the link.

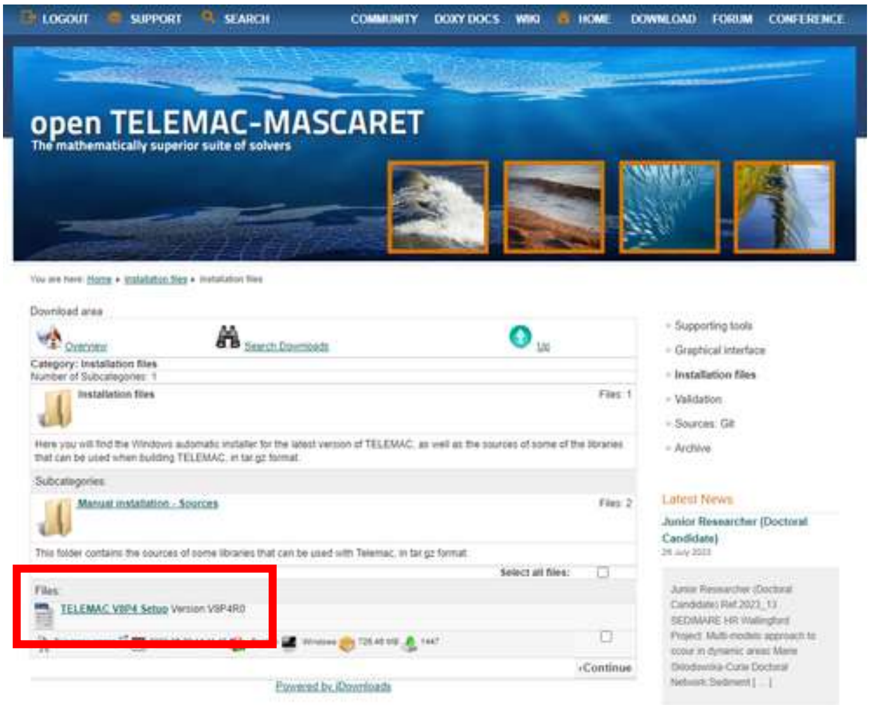


Figure 4 The Automatic Installation: Opentelemac-Windows Link

5. Read provided License Agreement, and, if you accept, tick the "**I Accept the above-listed conditions**" option and then click "**Download**".

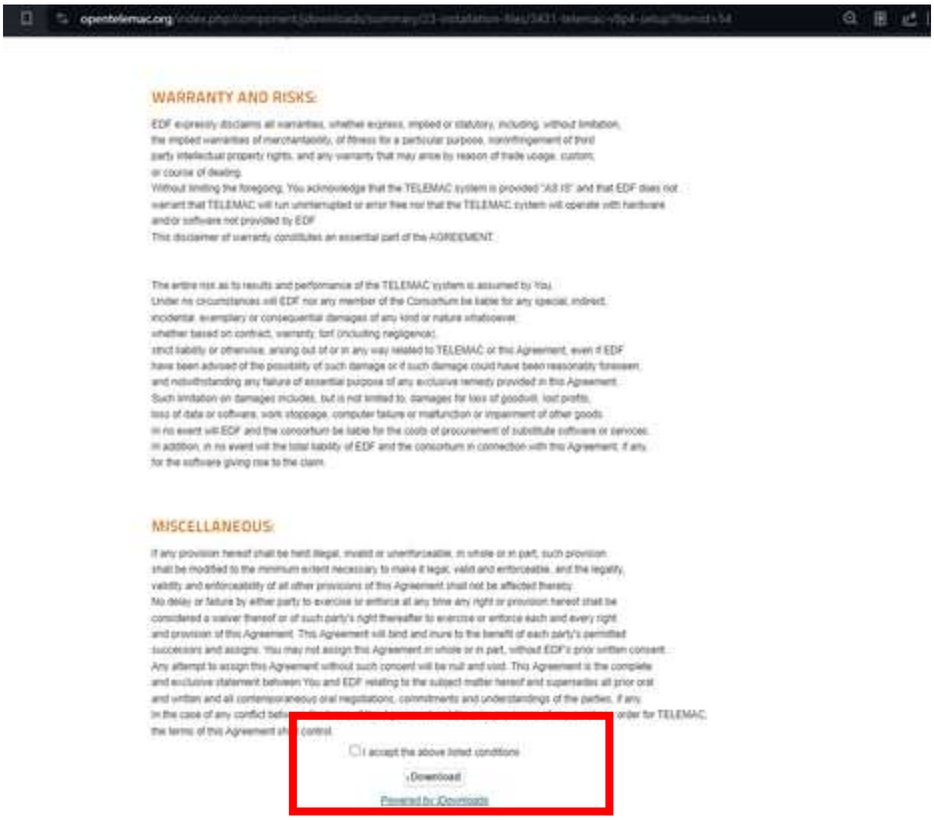


Figure 5 The TELEMAC-MASCARET License Agreement

6. Save the file to your computer and wait for it to finish. The file is about 726 MB.

7. Alternatively, if the current download system is not working, you can download the V8P4 installer from this direct link:

http://opentelemac.org/downloads/Installation%20files/telemac_v8p4r0_setup.exe

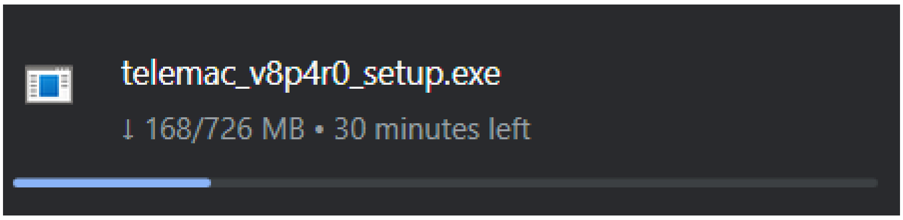


Figure 6 Approximate download size of the installer

8. Once finished, open the installer and click "**YES**" to proceed.

9. Once opened, click "**NEXT**" in the setup wizard to continue.

Figure 7 The Open TELEMAC-MASCARET Setup Wizard

10. Select desired installation location where setup will install TELEMAC V8P4. For this example, the installation directory will be set at C:\TELEMAC\V8P4. The reason for doing so is to prevent having a white space within the workig directory (The default directory is at C:\Program Files). Click "**Browse**" if a different location is desired. Click "**Next**" to continue.

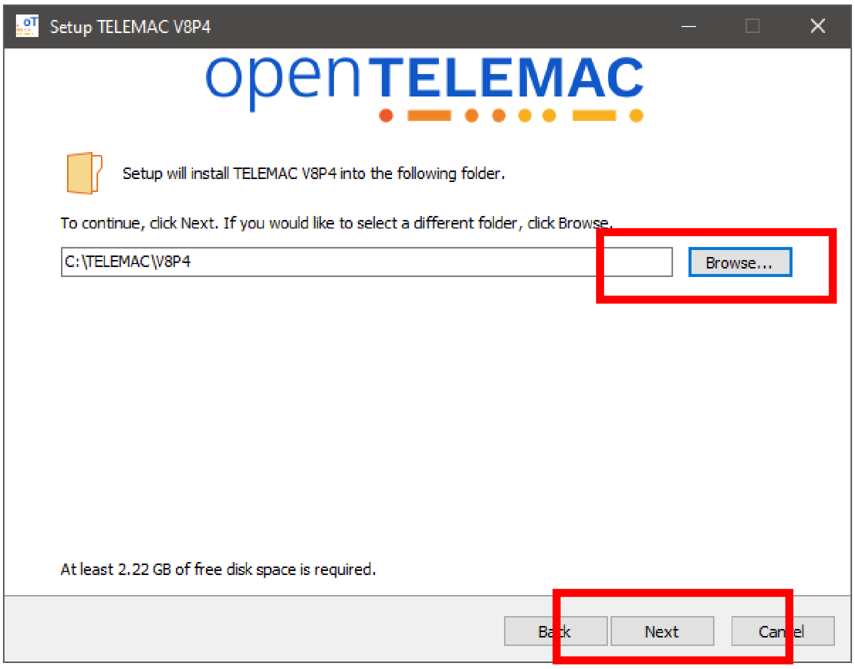
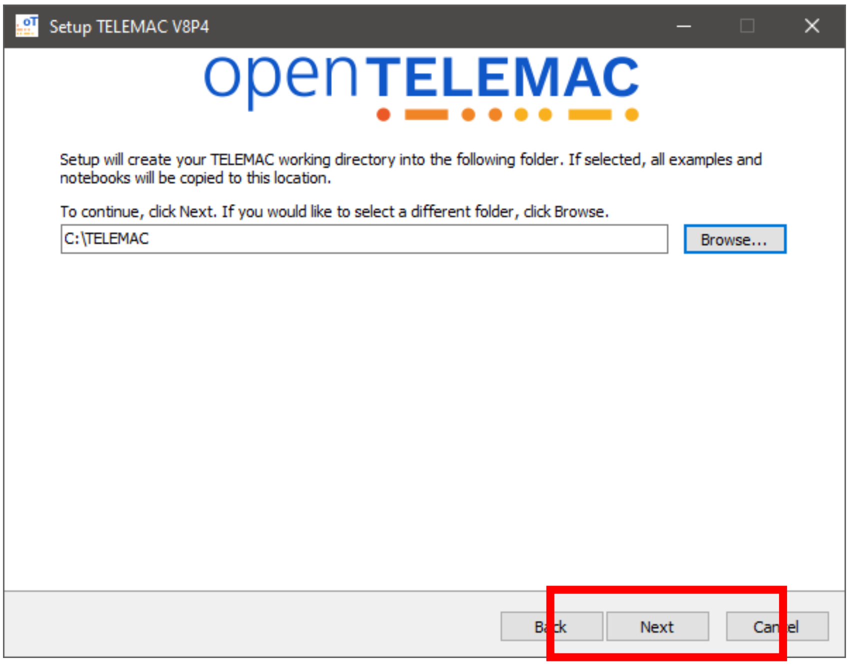
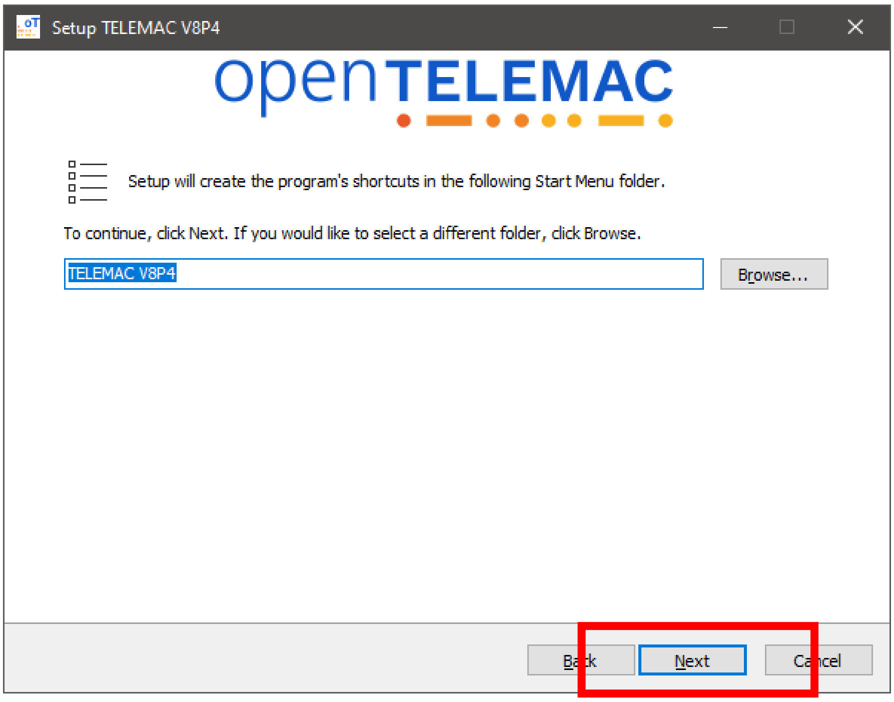


Figure 8. Selected directory for installation

11. For the "Working Directory" folder, navigate to C:\. Setup will copy all examples and notebooks to this location. Click "**Next**" to continue.



12. Leave the default location for program shortcuts. Click "**Next**" to continue.



13. You can choose additional tasks. Click “**Next**” to continue.

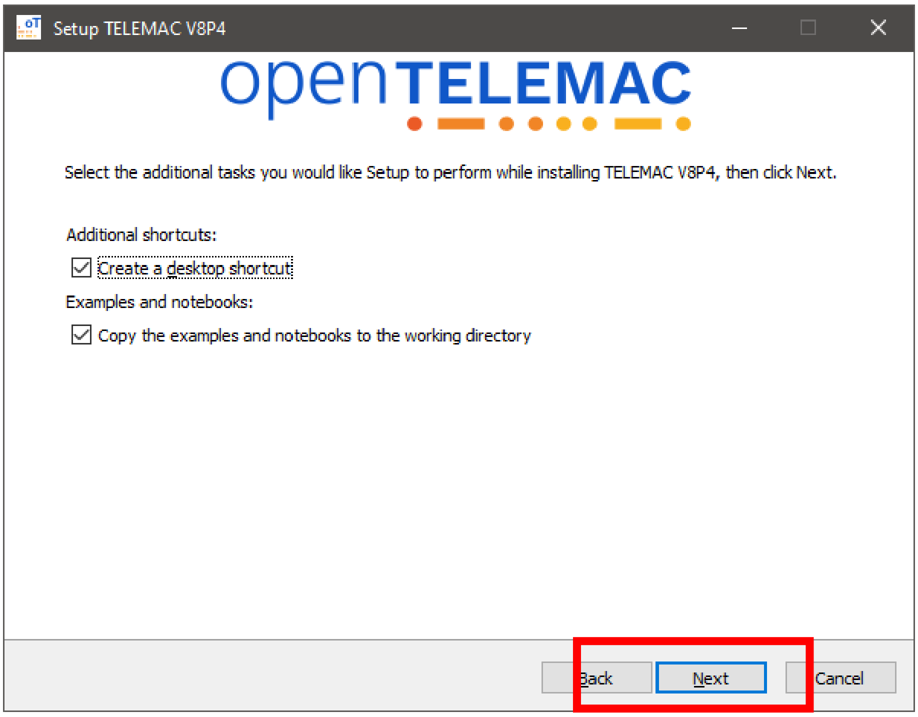
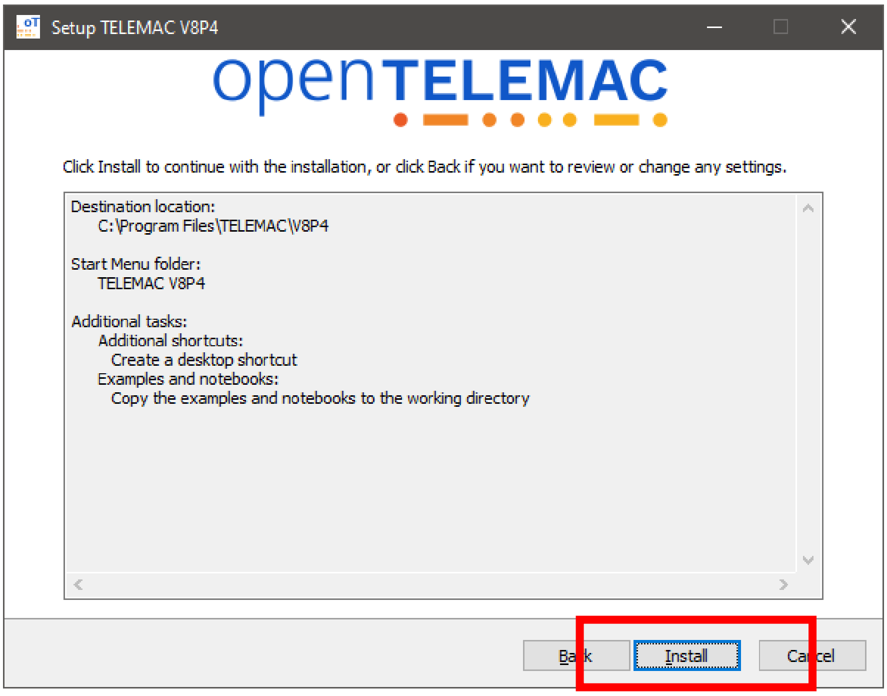
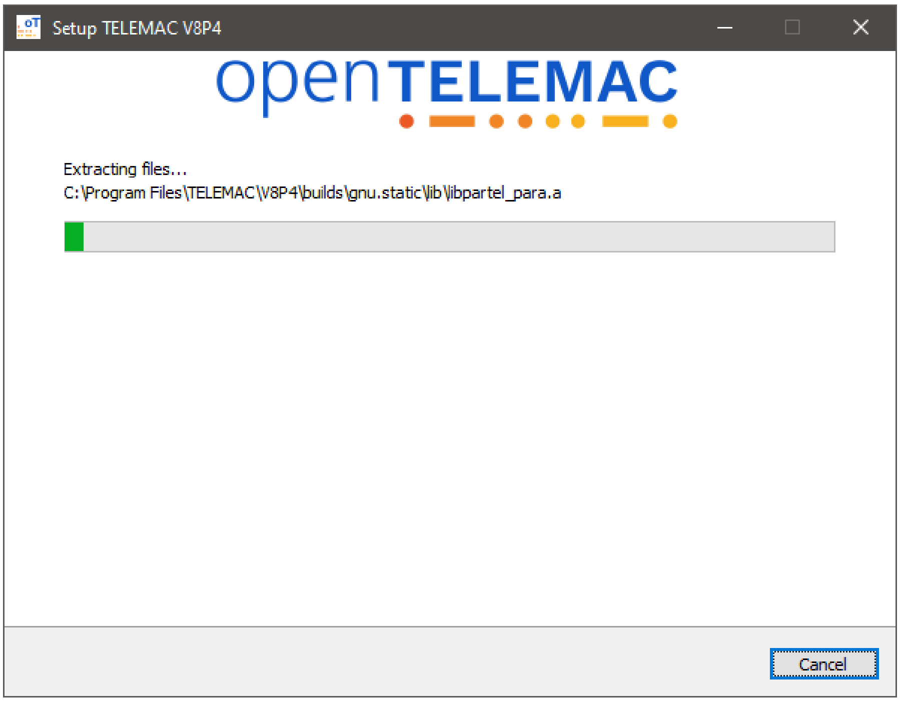


Figure 9 The License Agreement before proceeding to the installation

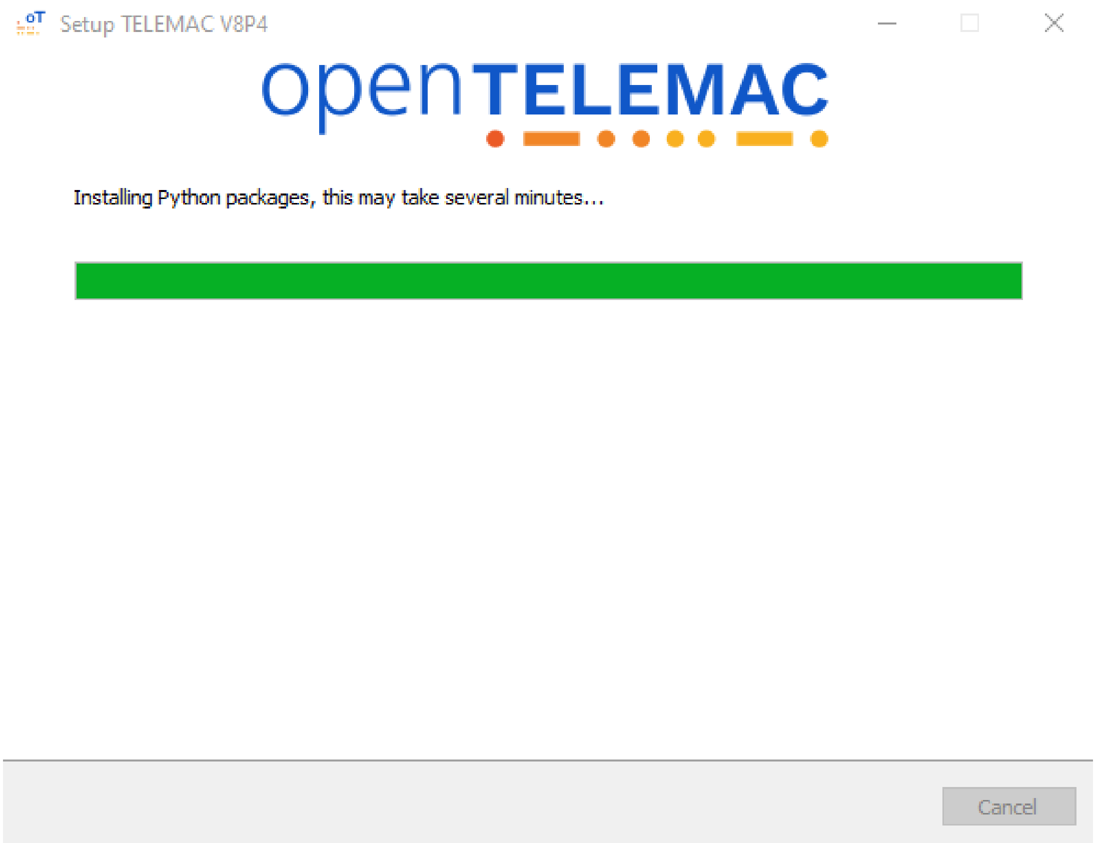
14. Click “**Install**” to proceed with the installation.



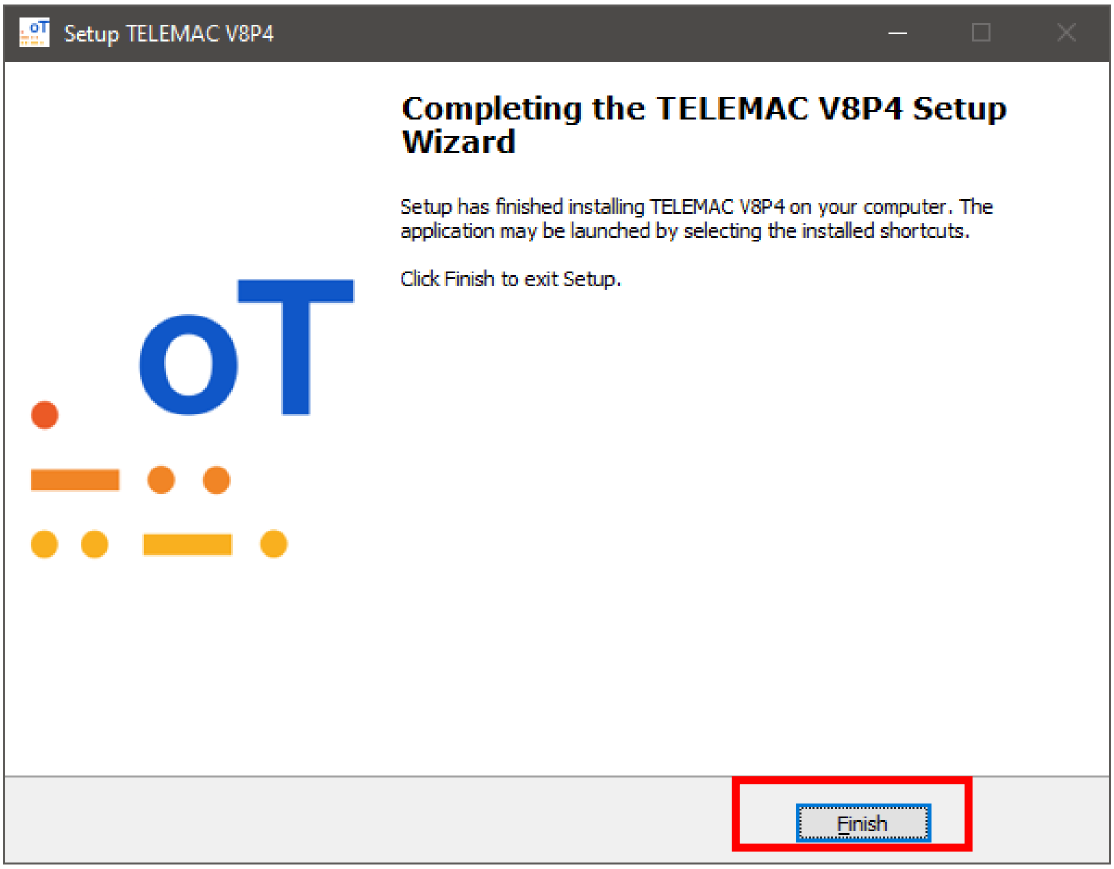
15. The setup will start extracting files with the set settings.



16. Wait for the setup to finish.



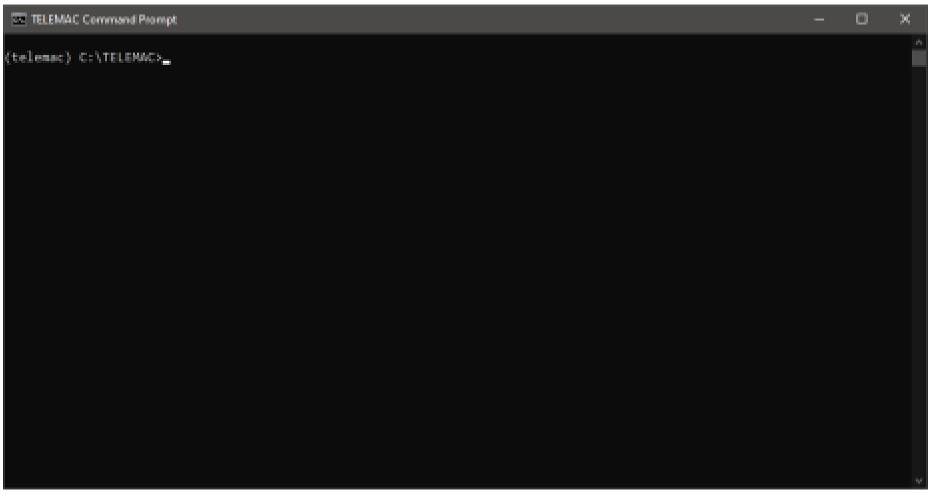
17. Click “**Finish**” to exit setup.



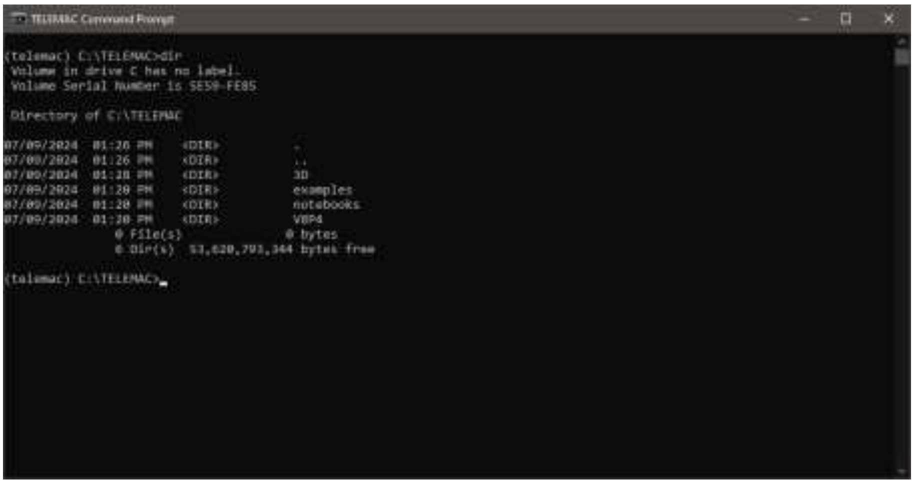
18. The next step would be to test whether the installation is successful or not. Proceed to the next section to run a Sample test.

2.1.1. Example Run Test

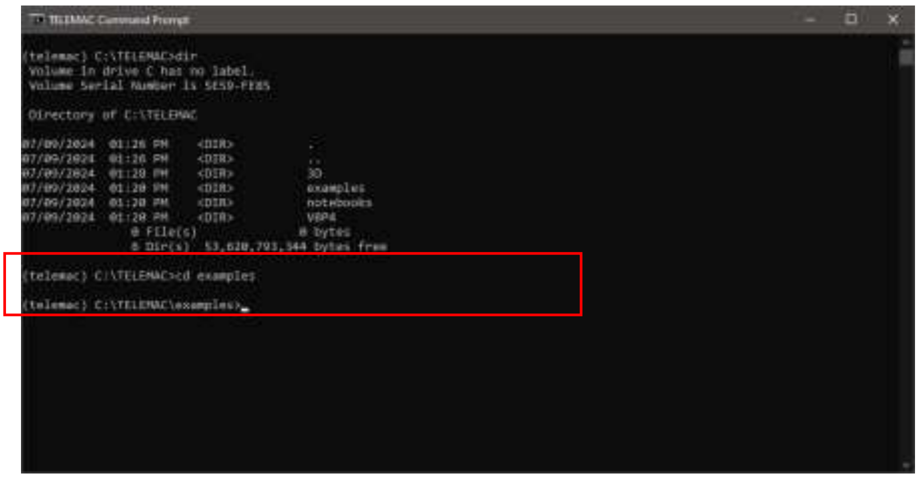
1. Open the generated shortcut called "**TELEMAC Command Prompt**" located in the desktop.



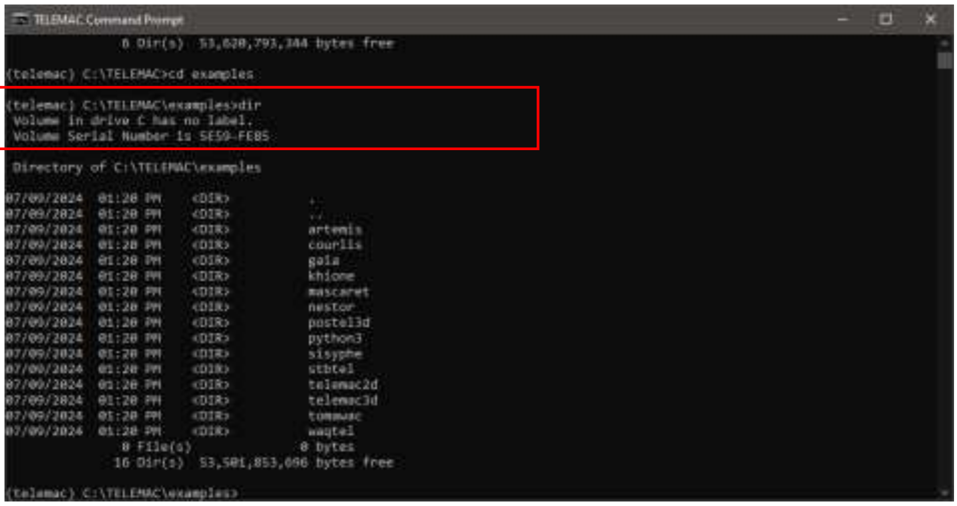
2. Type "**dir**" and press <**Enter**> to show the contents of current directory.



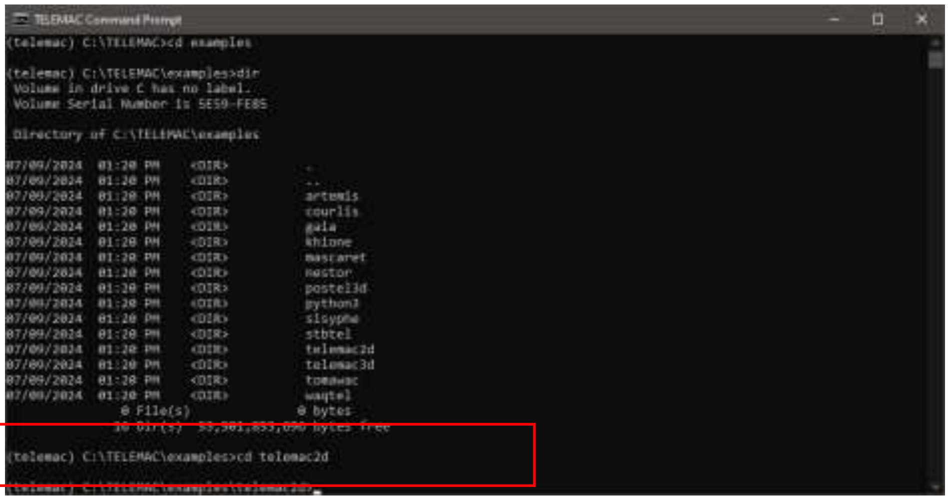
3. Change directory into the examples folder by typing "**cd examples**" and pressing <**Enter**>.



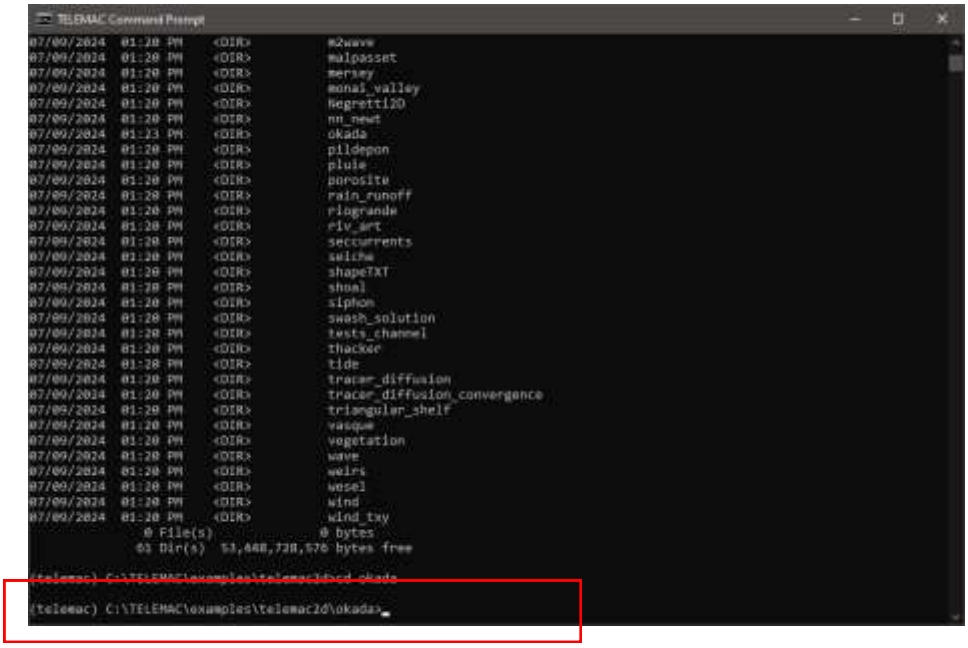
4. You are now inside the “**examples**” folder. Type “**dir**” and press <**Enter**> again to show the contents.



5. For this test run, type “**cd telemac2d**”to select telemac2d examples.



6. Type “**dir**” and press <**Enter**> again to show contents of current directory.
7. For this test run we’ll be running a sample from the “**okada**” folder. So go ahead and type “**cd okada**” and press <**Enter**>.



8. Now that we are in the directory where simulation files (okada) are located, we can start the simulation properly.

9. Type **dir** to display the contents of the said folder.

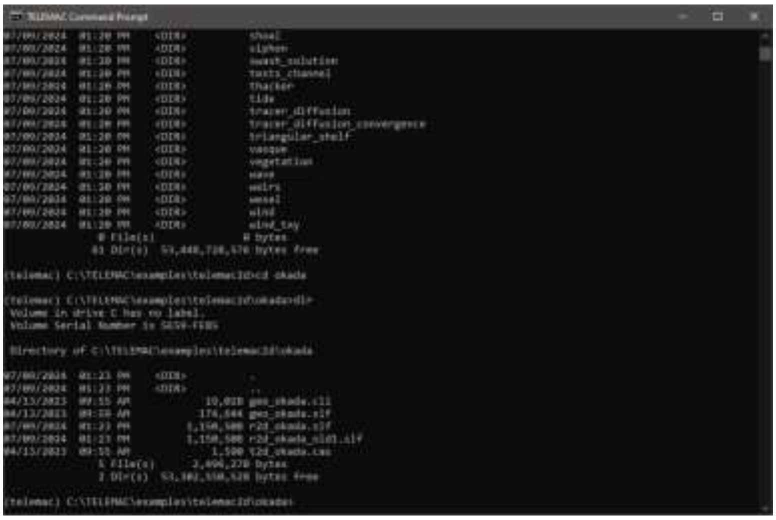


Figure 10 Typing dir to Display Contents

10. Note that all the mandatory files (i.e., the *.cli, *.slf, and *.cas files) are available in this folder.

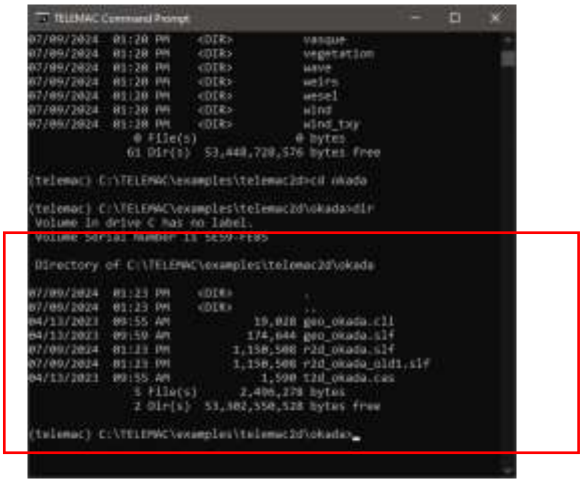


*Figure 11 The Contents of the Okada Folder after Running the **dir** Command*

11. To start the simulation, type "**python C:\TELEMAC\V8P4\scripts\python3\telemac2d.py t2d_okada.cas t2d_okada.cas**" and press <**Enter**>.

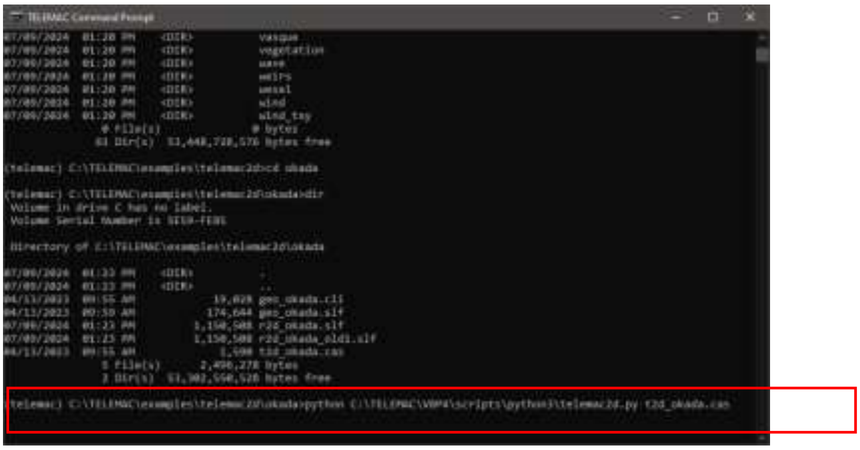


Figure 12 The Command Used to Start the Simulation

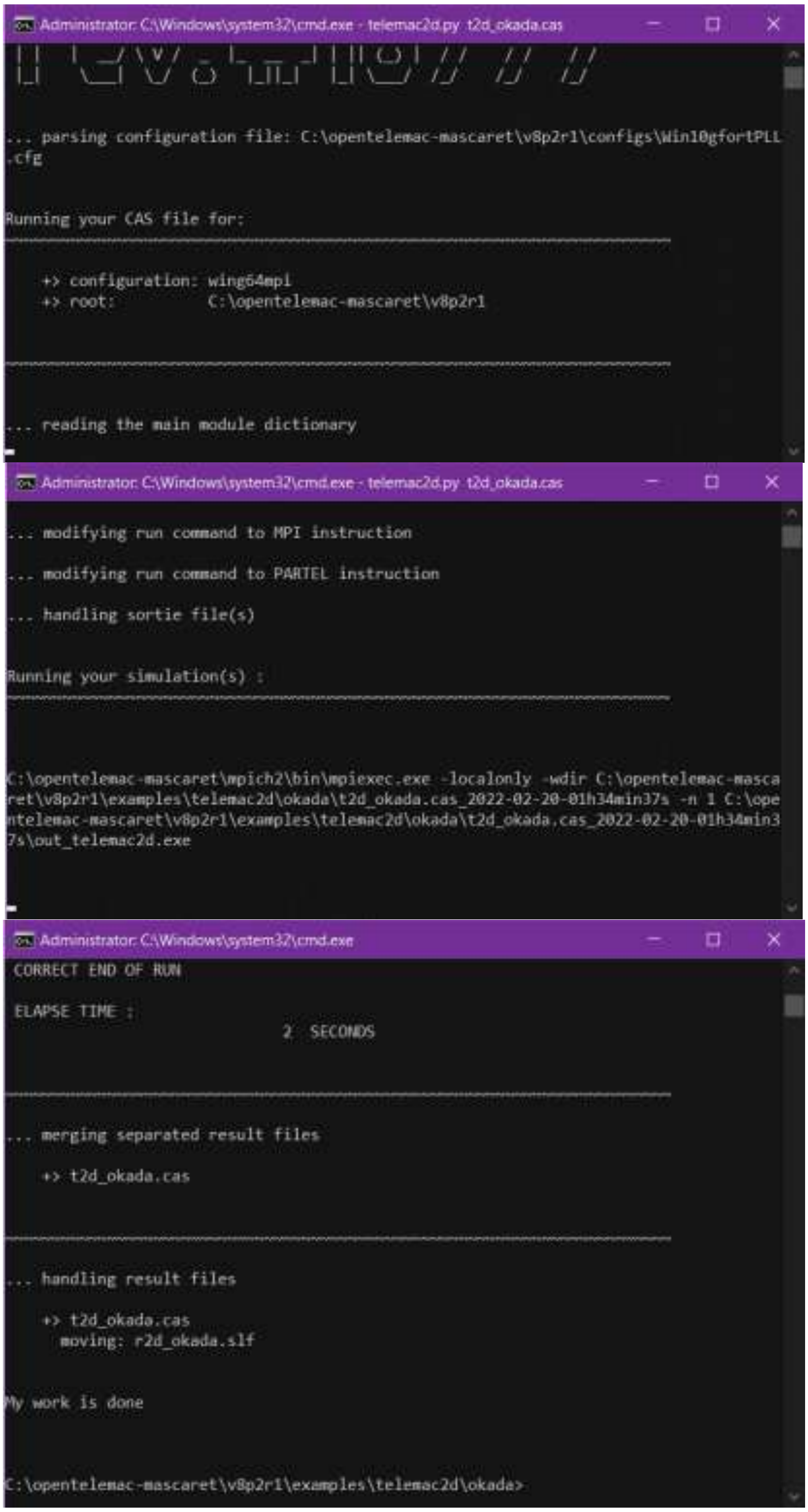


Figure 13 Correct end of run

12. The "**My work is done**" message signifies that our installation worked and that TELAMAC has been successfully installed and properly running.

2.2. Installing Blue Kenue

1. Open your web browser and navigate to https://nrc.canada.ca/en/research-development/products-services/software-applications/blue-kenuetm-software-tool-hydraulic-modellers. Click the "**Download Blue Kenue**" button.

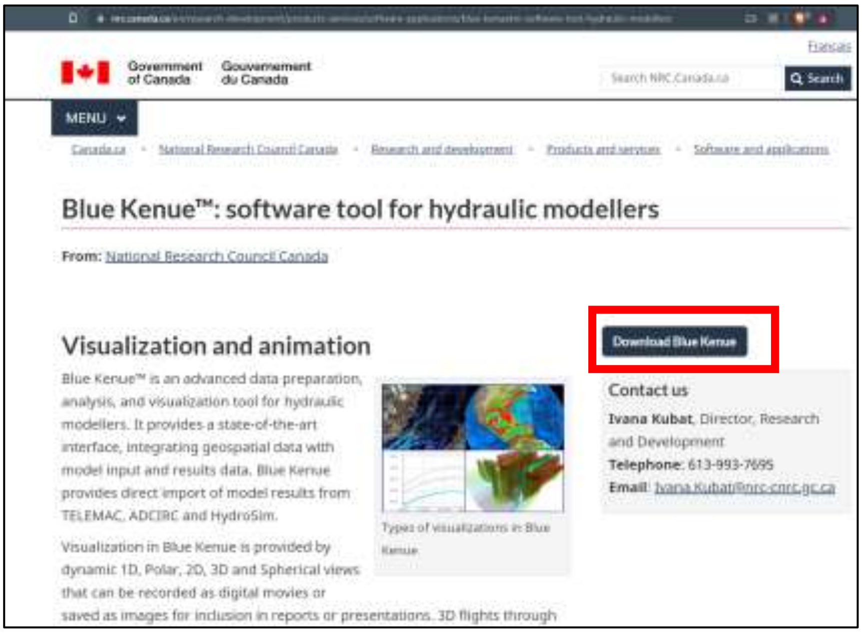


Figure 14. The National Research Council Webpage

2. Fill up the request form then click "**Submit**" once finished.

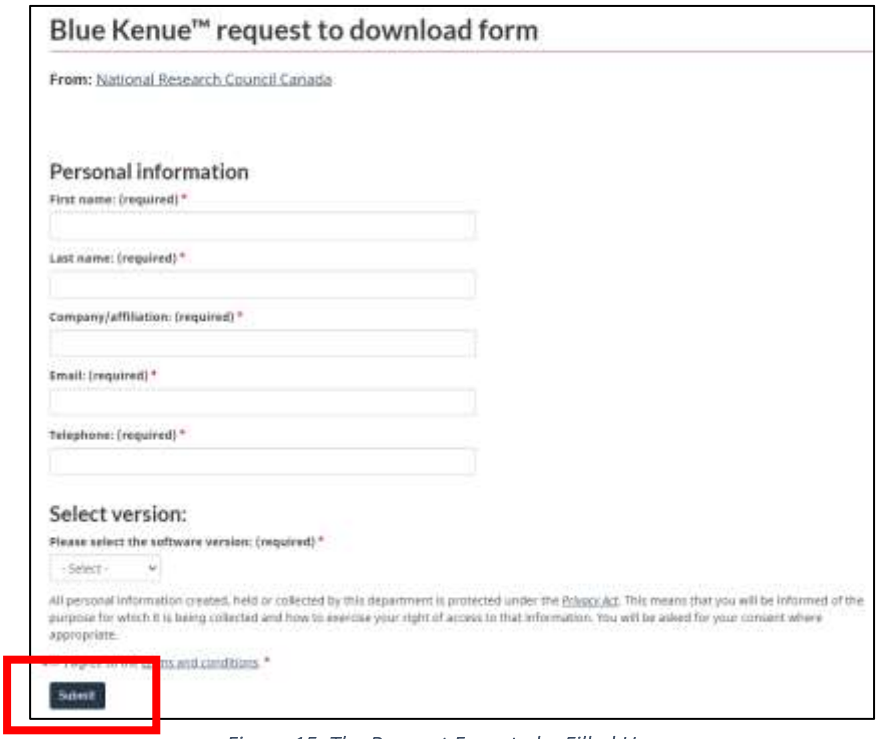


Figure 15. The Request Form to be Filled Up

3. Download the application, wait for it to finish, and then open the file to start installing. Then click “**Next**”>>” **Next**”>>” **Next**” to proceed.

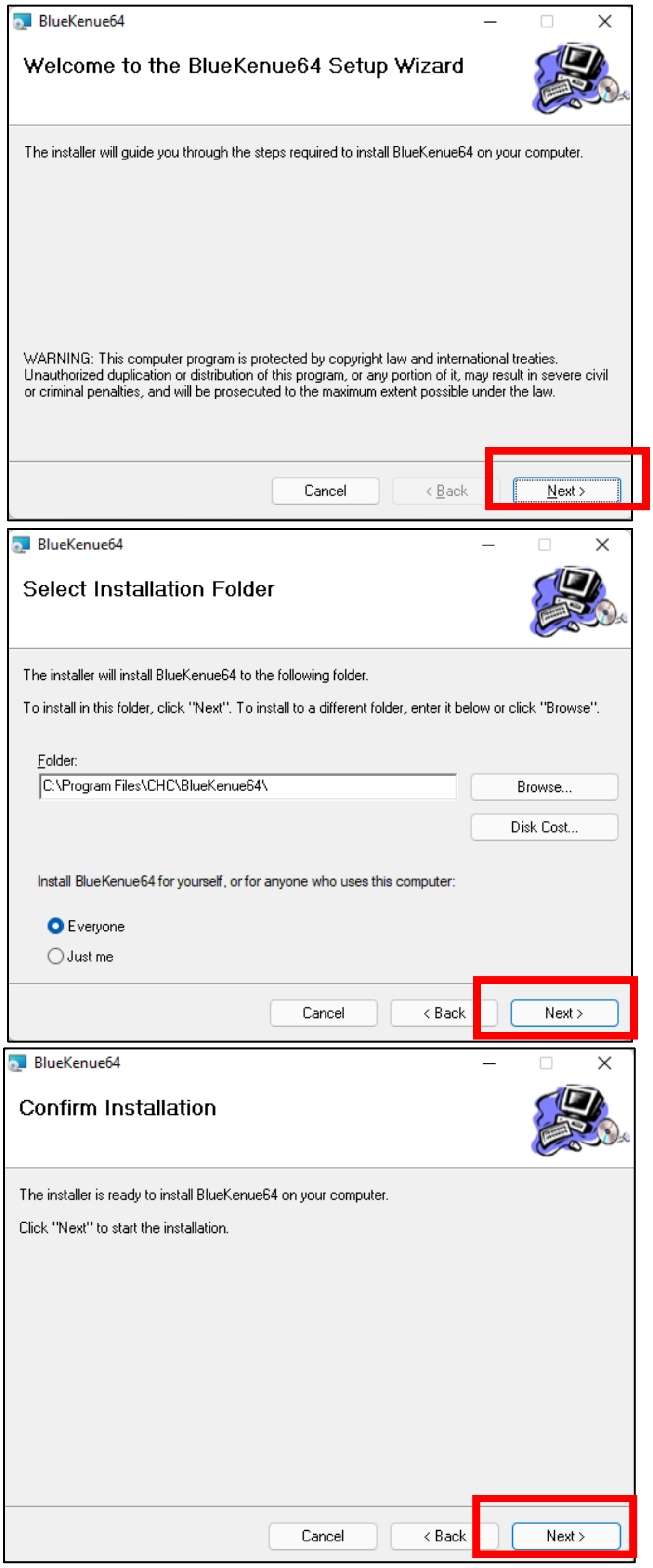


Figure 16. The Confirmation of the Installation

4. Wait for a few seconds and once finished, you can click close to finish the installation.

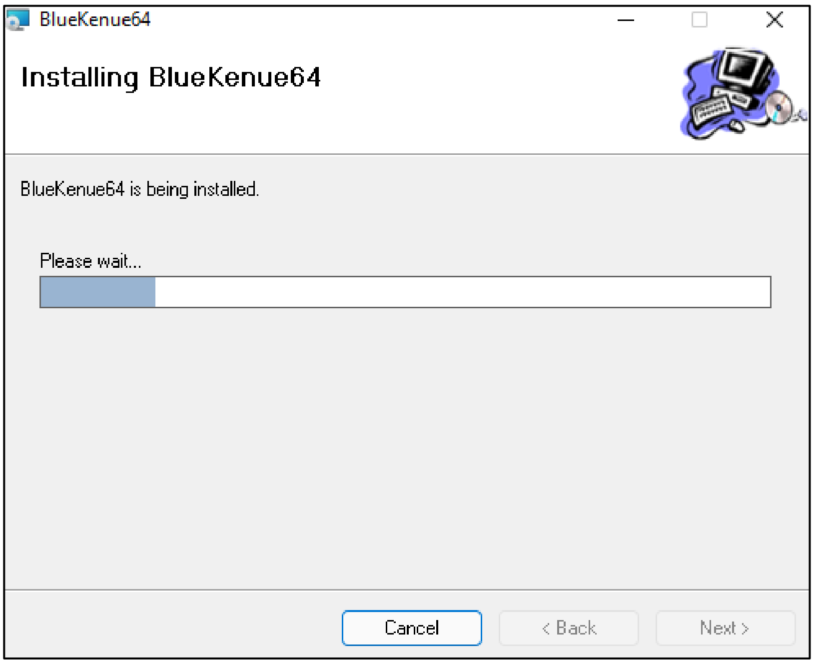


Figure 17. The Duration of the Installation

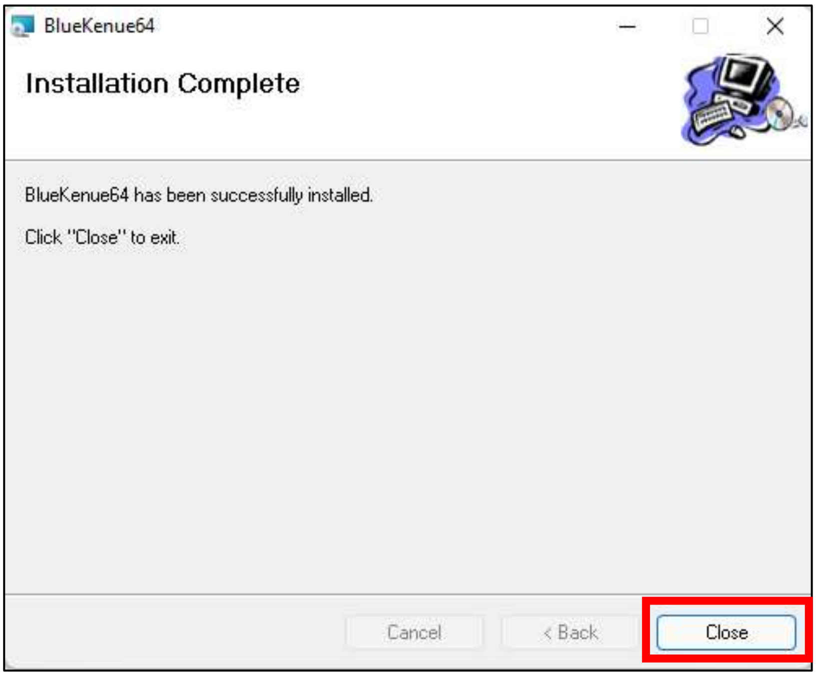


Figure 18. Finishing the Installation Process

3. Pre-processing utilizing Blue Kenue

Pre-processing encompasses the abstraction of the river landscape into a computational mesh (grid) with boundary conditions and interpolation of DEM data into the generated computational mesh. Various software programs are available for this purpose, which include but are not limited to the following:

1. QGIS and the BASEmesh plugin
2. The National Research Council Canada's Blue Kenue GUI software (primarily for Windows but can be used in Linux systems using Wine).
3. SALOME-HYDRO for generating computational meshes in the MED files format

However, this manual will only focus on utilizing Blue Kenue since it is advantageous for computation speed and model stability.

3.1. Generating Selafin File using Blue Kenue (Geometry File)

Blue Kenue™ is an advanced data preparation, analysis, and visualization tool for hydraulic modelers. It provides a state-of-the-art interface, thereby integrating geospatial data with model input and results from data. It also provides direct import of model results from TELEMAC and other software.

3.1.1. The Blue Kenue Interface

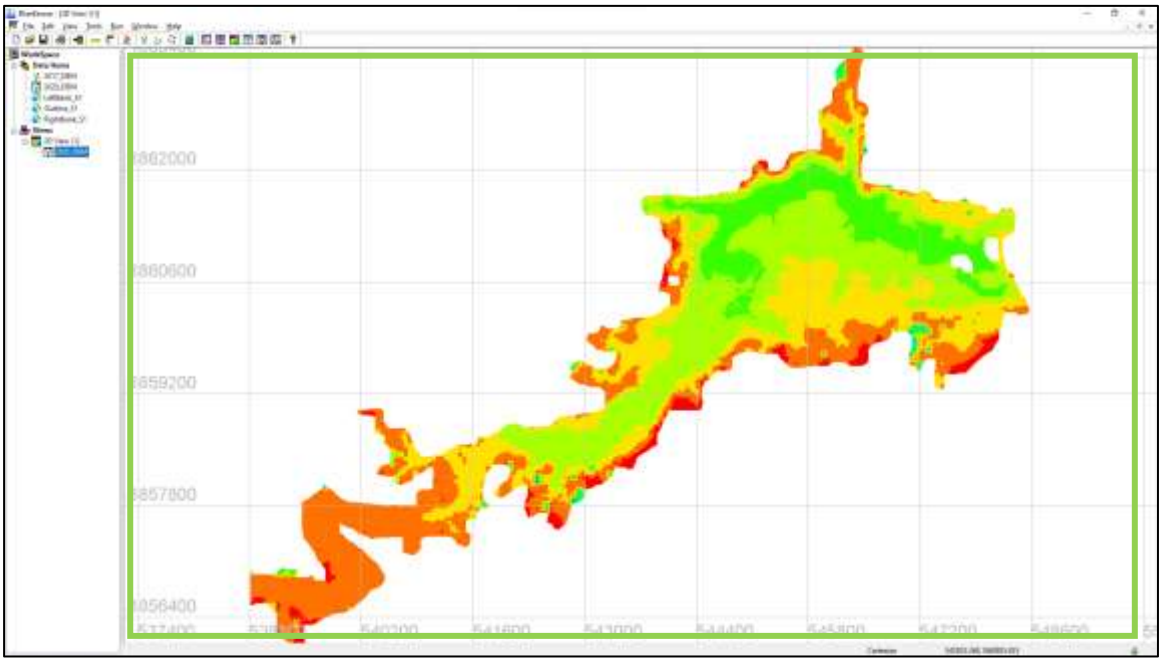


Figure 19. The Blue Kenue User Interface

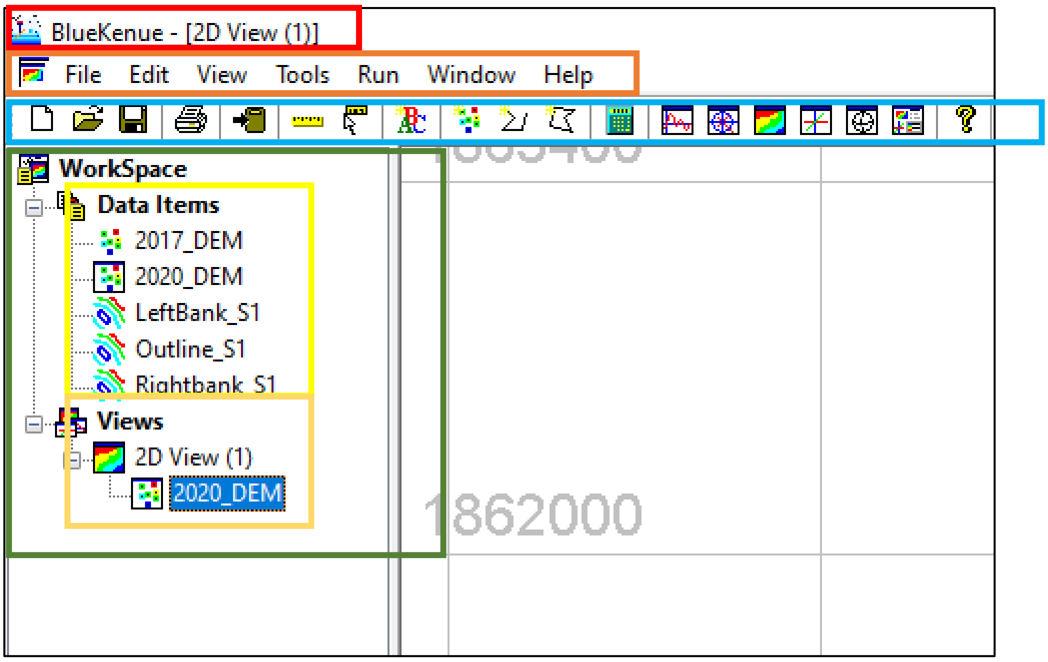


Figure 20. Parts of the Blue Kenue User Interface

1. Title Bar
2. Menu Bar
3. Tool Bar
4. Workspace
5. Data Items
6. Views
7. Canvass

3.1.2. Opening/Importing Data in Blue Kenue

1. Download and extract the TELEMAC Training Folder from https://drive.google.com/file/d/1FwfrIDsYLjTGenbGoXajNGO23k5hOrvQ/view?usp=sharing and extract the contents into your desired directory. In this manual, we are going to extract the contents into the **desktop folder**. The contents of the extracted TELEMAC Training folder are shown on the next page:

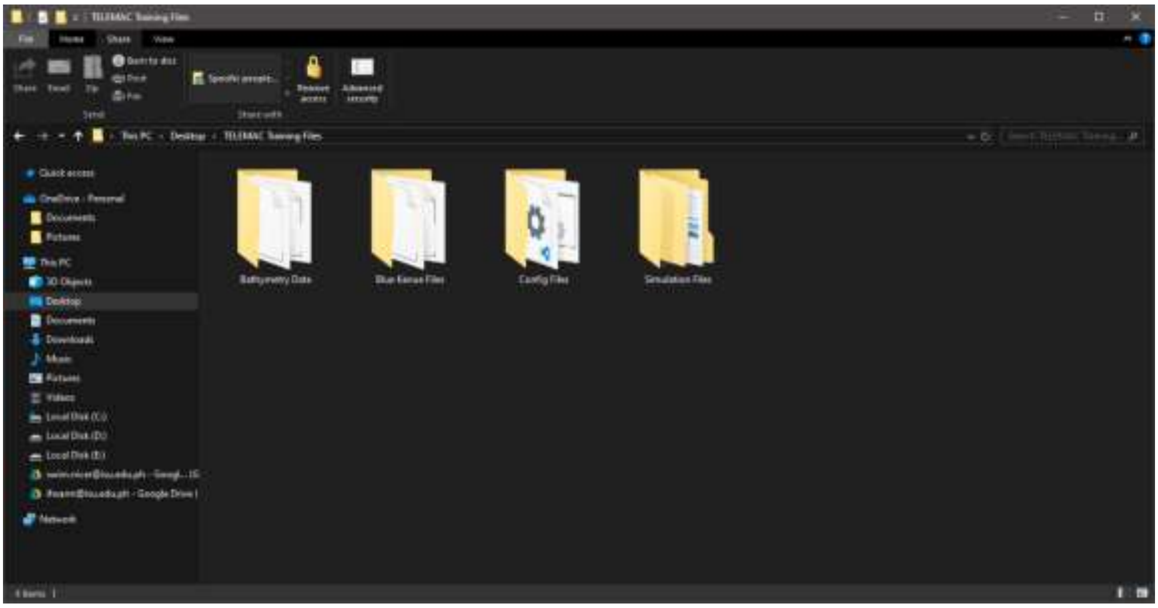


Figure 21. The Extracted Contents of the TELEMAC Training Folder

2. Start the Blue Kenue application and open the (.xyz) geometry file from the extracted **TELEMAC Training Folder** >> **Bathymetry_Data** >> **2017_DEM.xyz**. To see your XYZ dataset, change the file type drop-down menu from Selafin to All Files as shown in Figure 84.

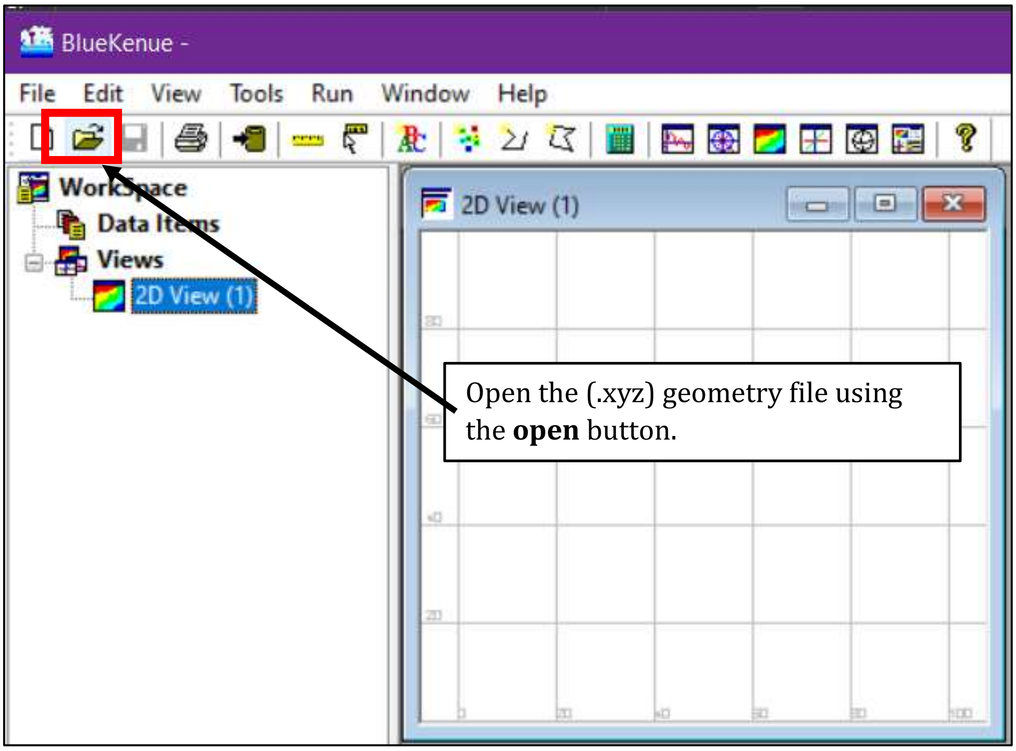


Figure 22. Importing the Bathymetry Data Using the Open Button

3. From the "**Open Files**" window, click the "**Telemac Selafin File(*.slf)**" and change it to "**All Files**" to see your XYZ data set.

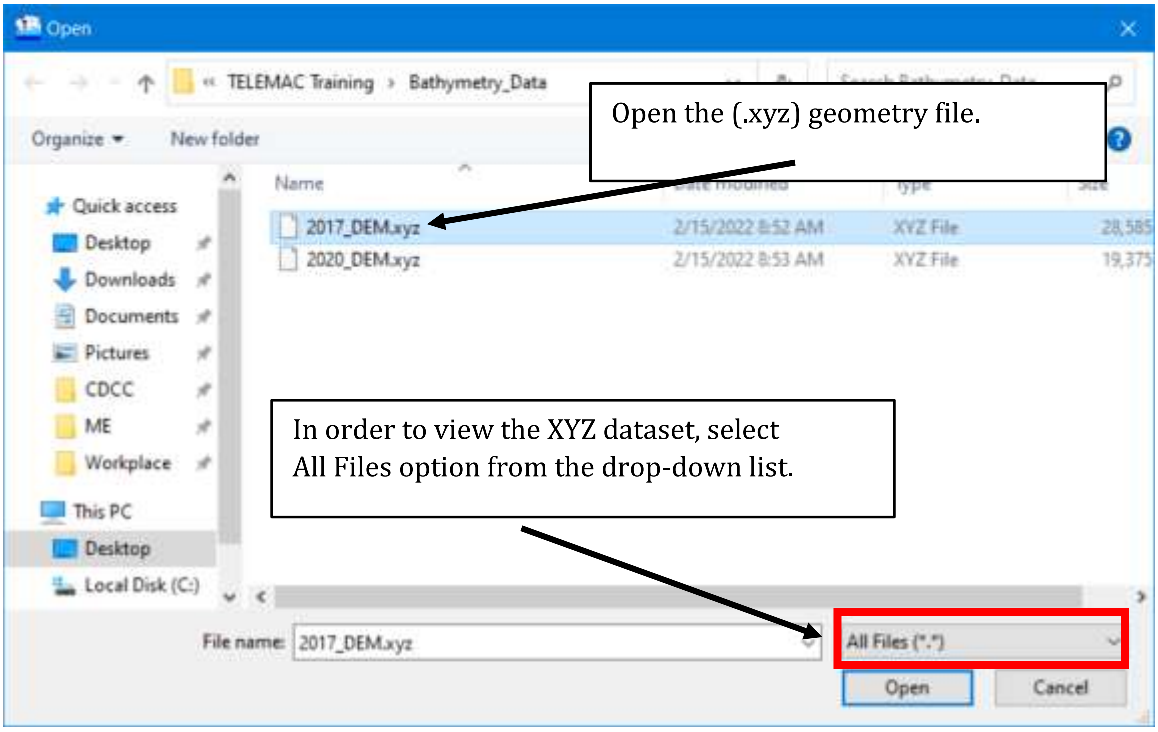
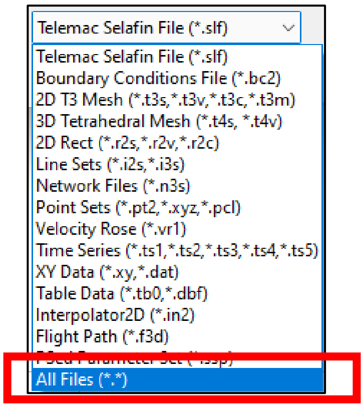


Figure 23. Toggling All Files (*.*) in Opening/Importing Bathymetry Data



3.1.3. Viewing Data in Blue Kenue

1. Notice that the "**DEM_2017**" is now displayed under the Data Items. Drag it into the "**2D View**" to display the item in the canvass.

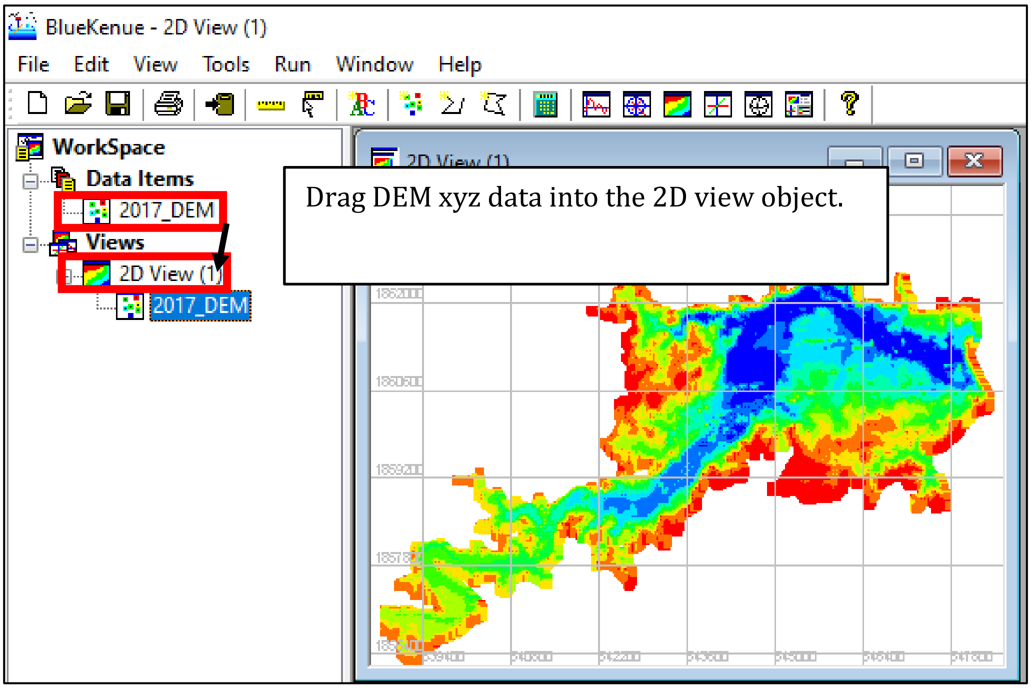


Figure 24. Viewing Items in Blue Kenue through a Views Object

*Note: In Blue Kenue, data items are not automatically displayed in the canvas. You must drag the item into the "**View**" section. Additionally, other view options like "**1D View**", "**3D View** ", "**Polar Views**", "**Spherical Views**", and "**Report View**" can also be used from the tool bar. Try to hover your mouse over each option. The example shown in Figure 87 features the 3D view option.*

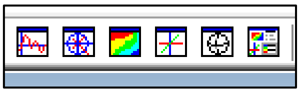


Figure 25. Other View Options Available in Blue Kenue

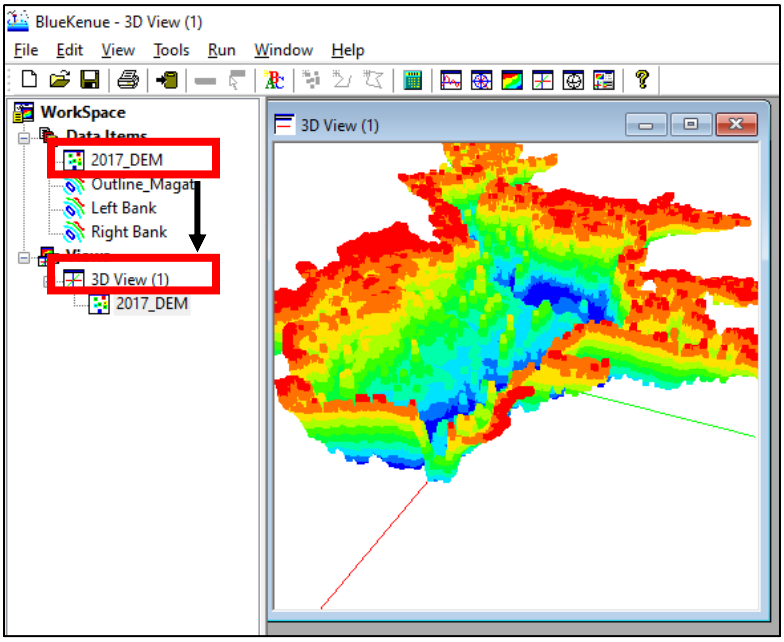


Figure 26. An Example of a 3D Isometric View

3.1.3. Mesh Generation

Step 1: Model Boundary (Closed Line)

1. In Blue Kenue, the "**New Closed Line**" tool is used to delineate the outline or boundary of the area to be studied.

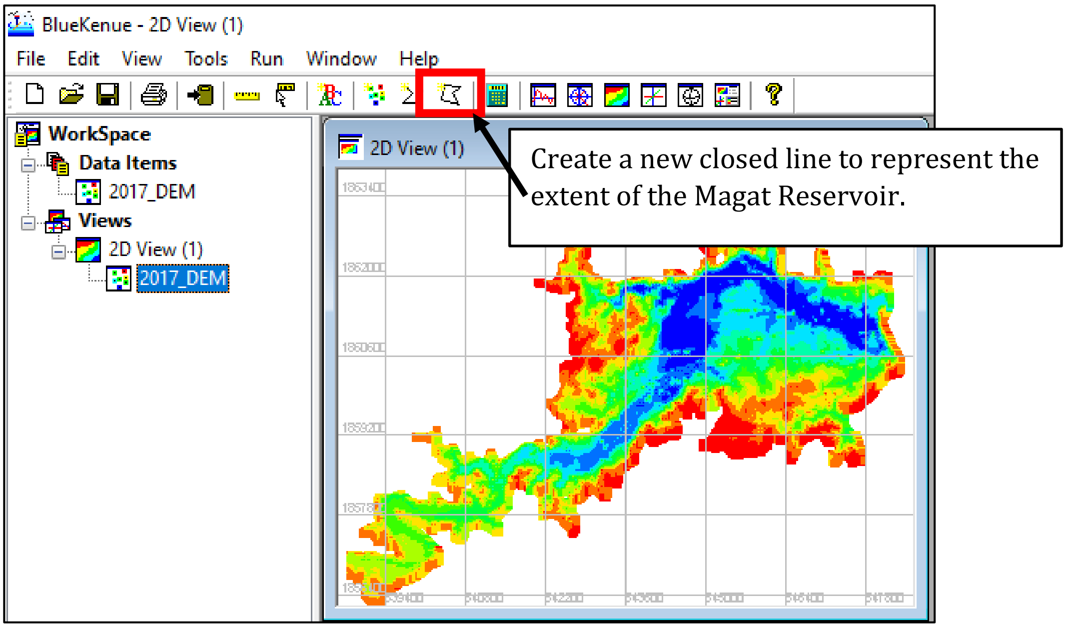


Figure 27. Step 1: Creating New Closed Line to Represent the Reservoir Boundaries

2. Click the "**New Closed Line**" icon. Notice that your mouse pointer icon will change into something like this: . Try to follow the extent of the DEM file and delineate the outline boundaries. You can zoom in and out with the mouse wheel button, and do a pan with the left click then drag.

3. Start by clicking the edge of the DEM and work your way around it by patiently left-clicking along the edge lines. Notice the magenta-colored connecting lines being generated.

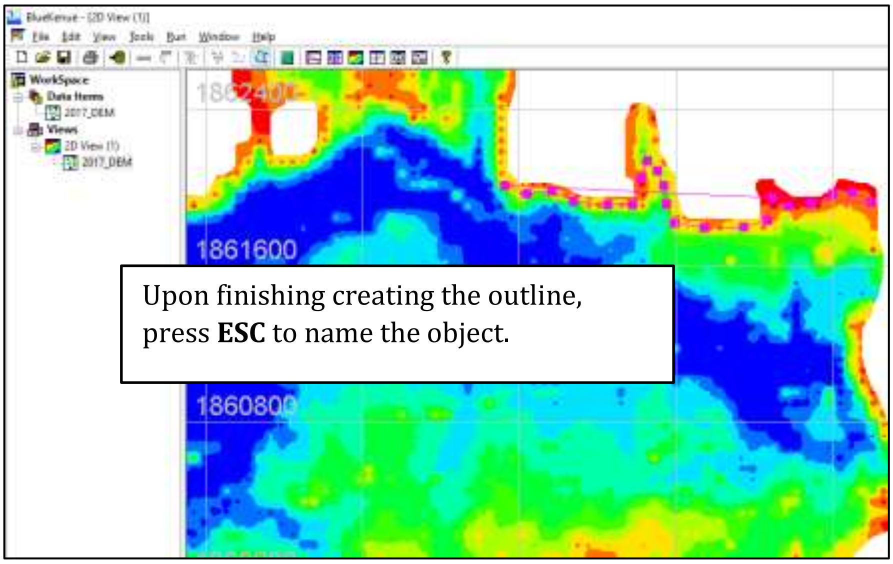


Figure 28. Naming the Newly Created Outline

4. Once finished, press the "**ESC**" button on your keyboard. A query toast will pop up. Enter the desired outline name (i.e., Outline Magat). The units should be in meters (m) and leave the rest at their default values.

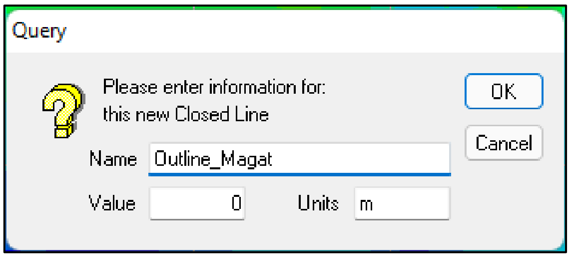


Figure 29. Query upon Pressing ESC

5. Again, drag the newly created item into the 2D view. See figure 85 for reference.

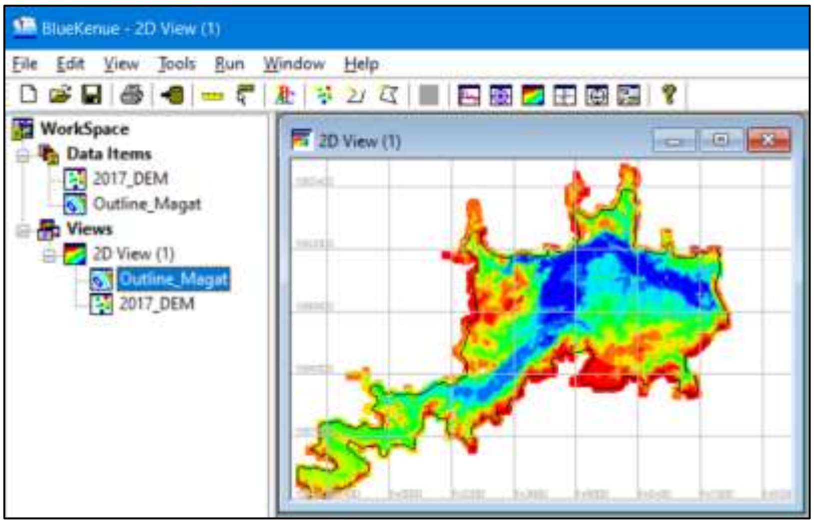


Figure 30. The Current Workspace with the Outline in Dragged into the View Object

Note: Please see Appendix VI: Importing an existing shapefile as an outline.

Step 2: The T3 Mesh Generator

To create a spatial representation of the domain to be studied (in this case, the Magat River Main Reservoir), a computational mesh must be created first.

Now, in generating a mesh, a closed line (outline) is enough and can be generated using the steps shown [above](#).

Following the above steps, you should now have a workspace in Blue Kenue with the following files open:

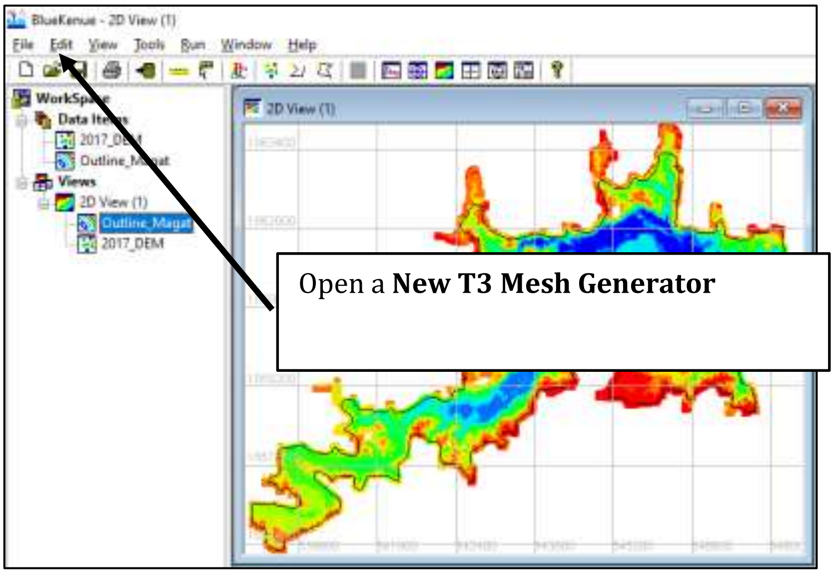


Figure 31. Current Workspace

1. From the Menu bar, click "**File**" > "**New**" > "**T3 Mesh Generator**".

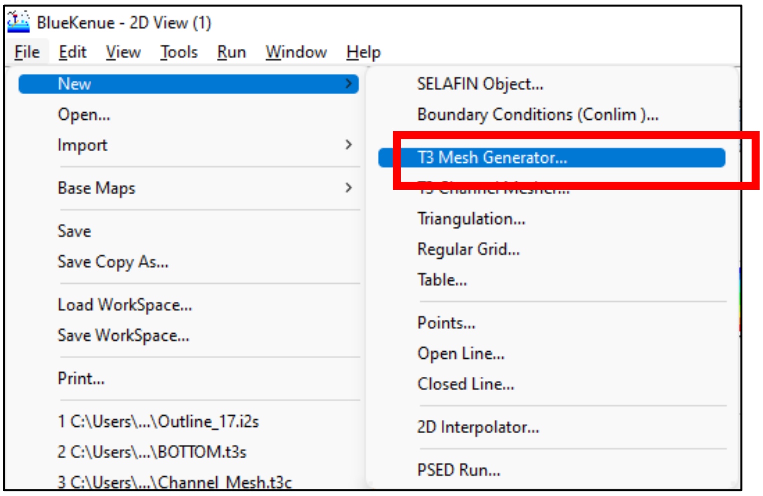


Figure 32. Opening a New T3 Mesh Generator

Note: The T3 Mesh Generator is used to produce two-dimensional scalar triangular meshes that can be used in models such as TELEMAC and similar programs

2. Upon clicking, a properties window for the "**newT3Mesh**" will pop up:

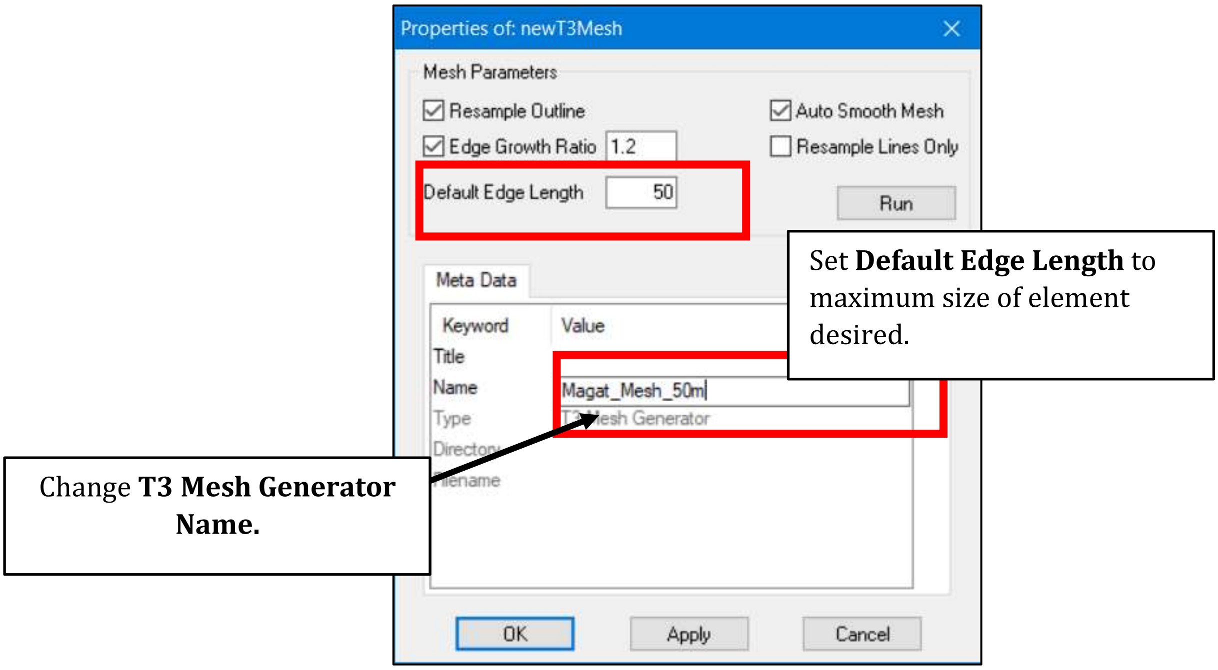


Figure 33. The New T3 Mesh Properties Window

3. Change "**Default Edge Length**" to "**50".** Leave the rest settings at their default settings. If desired, you can change the "**Name**" of newT3Mesh by simply clicking on the "**newT3Mesh**" and typing the desired new name (i.e., "**Magat_Mesh_50m**". Don't run yet, just click "**OK**".

   Note:

   - *Resample Outline: If this box is checked, the Mesher will resample the supplied outline, considering the values provided by the various density objects and the default edge length. If this box is unchecked, the triangular mesh will only employ the points that make up the polygon used to form the contour as edge nodes. This box is checked by default.*
   - *Auto Smooth Mesh: If this box is checked, the mesh will be adjusted by the Laplacian mesh smoothing algorithm after the mesh has been generated. This box is checked by default.*
   - *Edge Growth Ratio: This box, if checked, defines the maximum ratio between the lengths of edges at a given node. This box is checked by default and has a default value of 1.2.*
   - *Resample Lines Only: If this box is selected, the outline and soft lines will be resampled according to the density objects, but no mesh will be generated.*
   - *Default Edge Length: This value defines the default distance between nodes using the same units as the closed polygon that defines the perimeter of the mesh. The default value is 1, whereas the lower edge length is equal to a higher density.*

4. Drag “**Outline_Magat**” into the “**Outline**” slot.

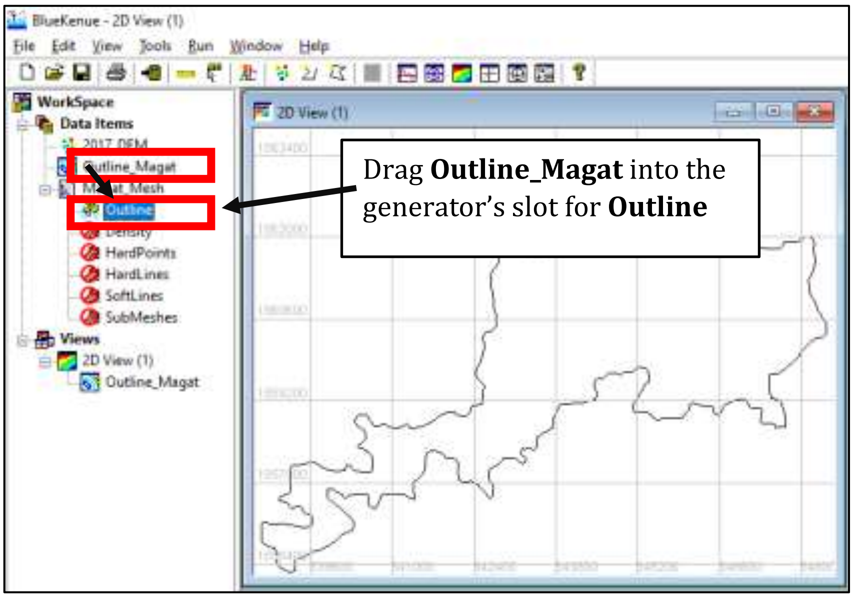


Figure 34. Dragging the Generated Outline into the Generator

Note:

- *For now, we can leave the other slots untouched.*
- *Hard points specify the locations of certain nodes as well as modify the local density of nodes.*
- *Any combination of points, polylines, closed polygons, triangular meshes, or rectangular grids is used by the density object to define areas of constant or varying density.*
- *Closed hard lines are used to produce islands with fixed outline geometry. These are also used in generated break lines with fixed geometry.*
- *Closed soft lines are used to produce islands with geometry determined from the density object. On the other hand, open soft lines produce break lines where the node spacing is determined by the density object.*

5. Once the “**Outline**” slot is filled, go to the “**Magat_Mesh_50m**” properties again by either double-clicking it or right-click >> P**roperties**.

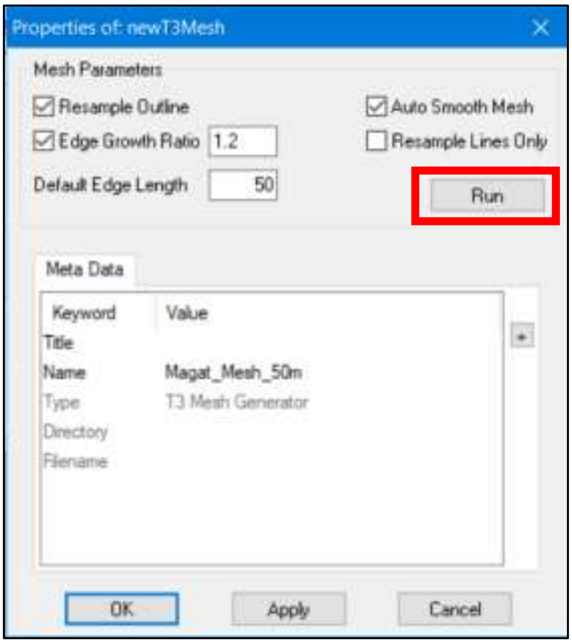


Figure 35. Running the Mesh Generator

6. We can now click "**Run**". A dialog containing Outer Boundary Area and other parameters will appear. Simply click "**Yes**" to continue.

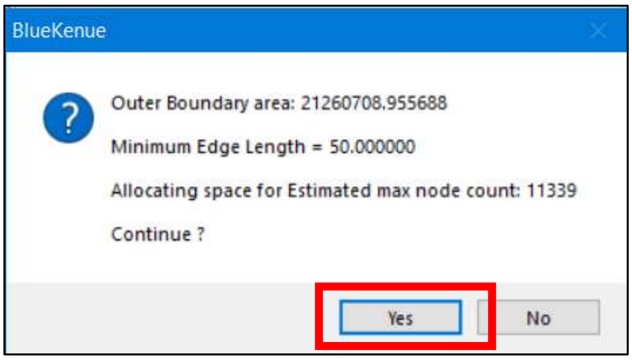


Figure 36. Running the Generator

7. In this example, the program can generate mesh within about a few seconds.

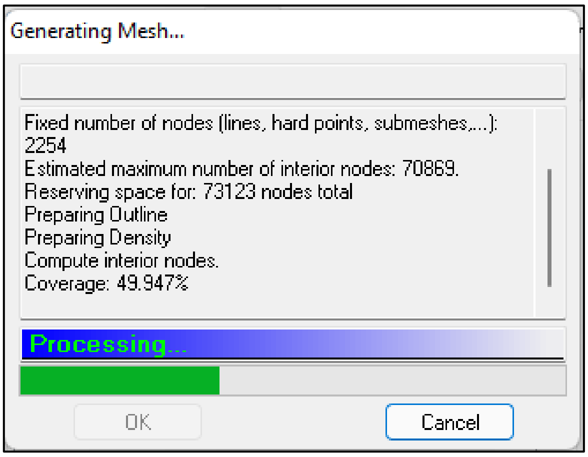


Figure 37. The Mesh Generator

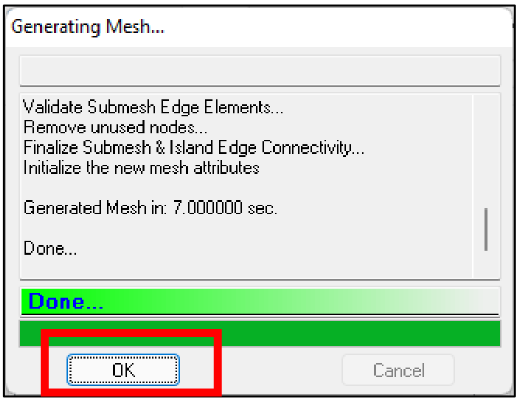


Figure 38. The Mesh Generator after a Successful Run

8. Click "**Ok**">> "**Ok".**
9. Once finished, notice that a child object called "**New Mesh (NodeType)**" has been added under the "**Magat_Mesh_50m**" item. Then, rename it into "**Magat_Full_Mesh**".

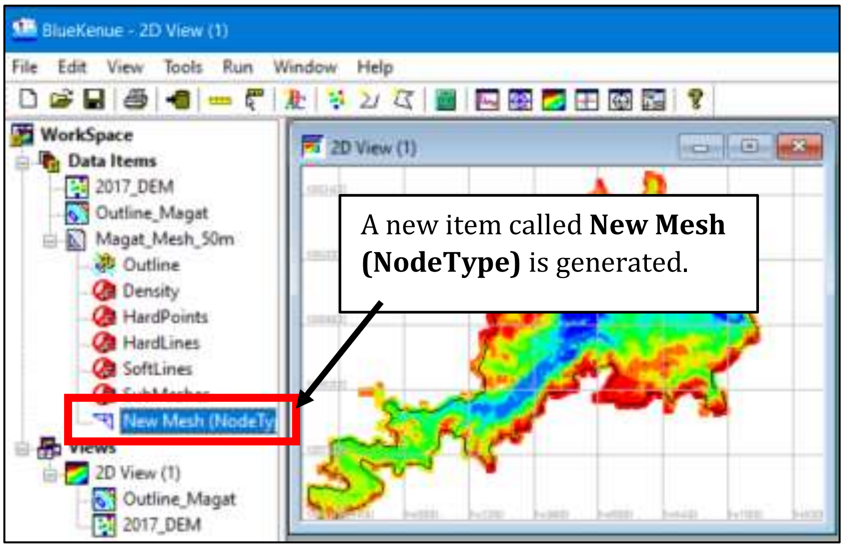


Figure 39. A Newly Generated Mesh

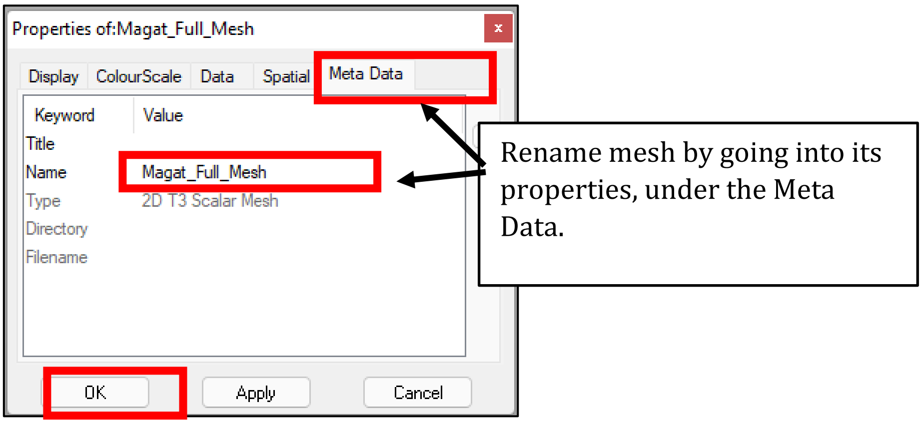


Figure 40. Renaming the Mesh

10. To see whether the process has been successful, drag the newly generated and renamed "**Magat_Full_Mesh**" into the 2D View.

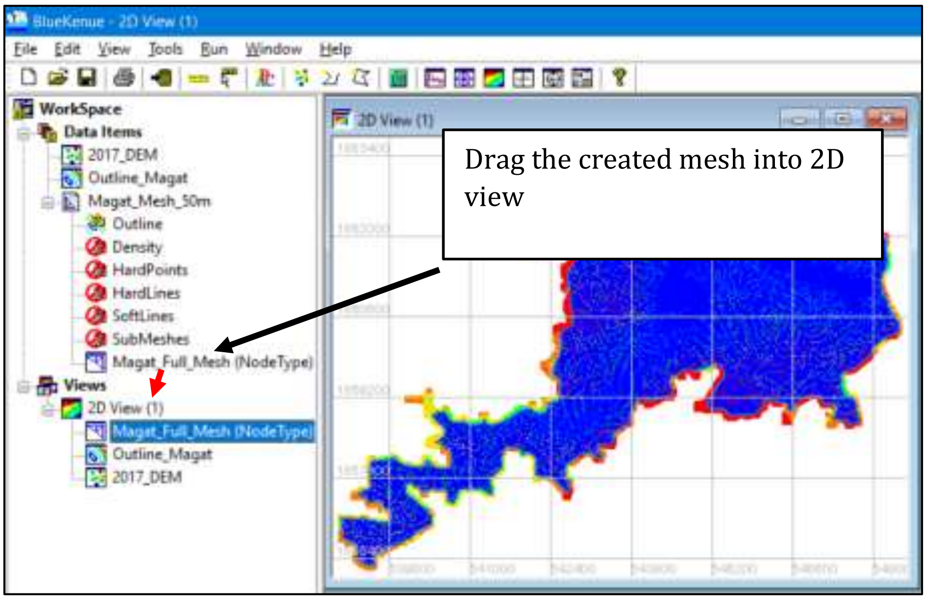


Figure 41. The Newly Created Mesh in 2D View

11. To deactivate an object in the 2D view, simply right-click the object of interest and click "**Visible**".

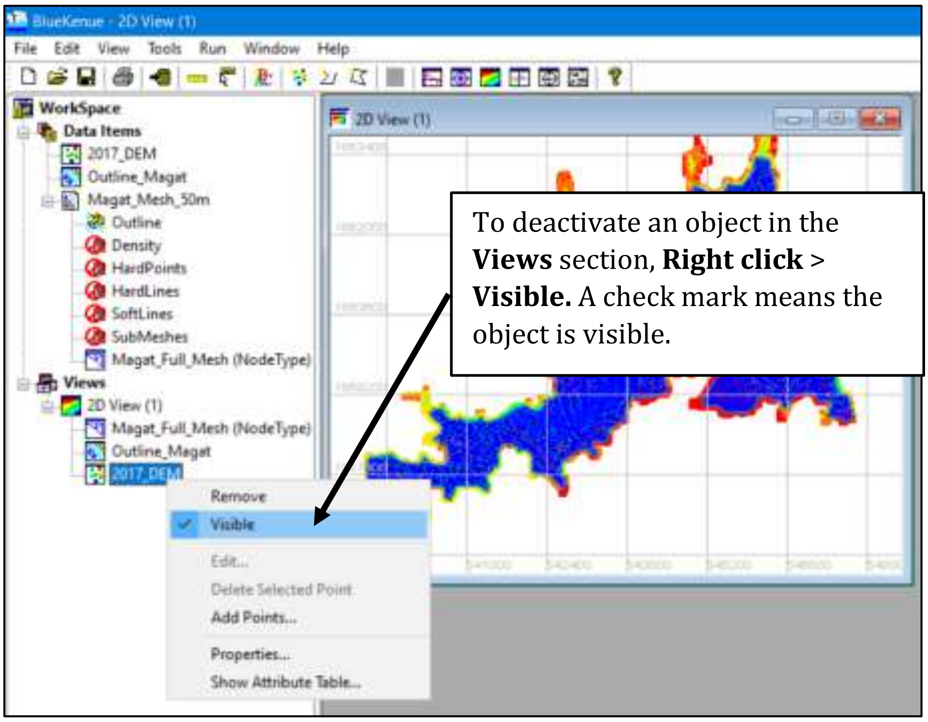


Figure 42. Process of Deactivating an Object in the View Section

3.1.4. Interpolating the DEM into the Created Mesh

1. Continuing from the last section, create a new Selafin object by clicking the "**New Selafin Object**" 🗋 icon, which will create a "**newSelafin**" object under the Data Items Menu.

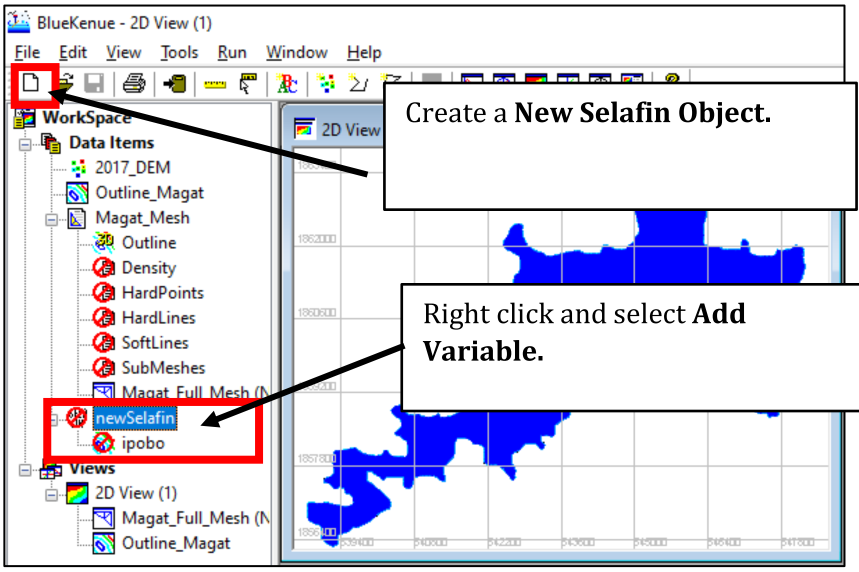


Figure 43. Creating a New Selafin Object

2. Right click the said Selafin object and select "**Add Variable**".

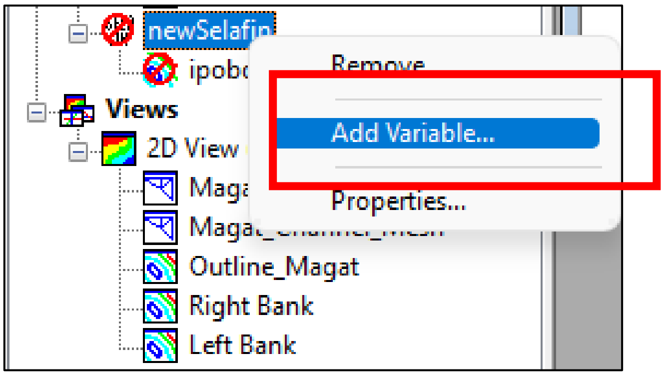


Figure 44. Adding a New Variable to the Selafin Object

3. After which, an "**Add New SELAFIN Variable**" window opens.

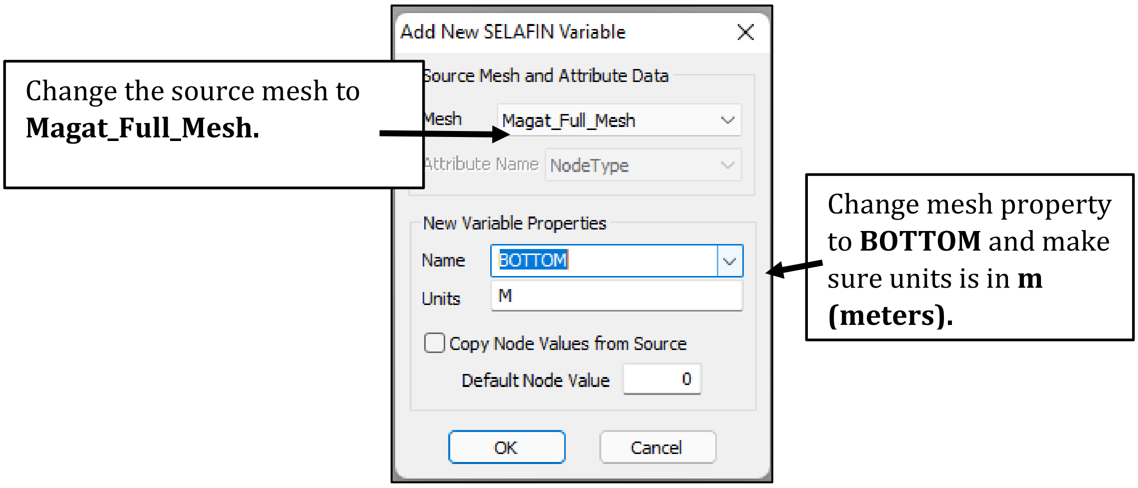


Figure 45. The Add New SELAFIN Variable Window

4. Leave the rest at their default values, then click "**OK**".

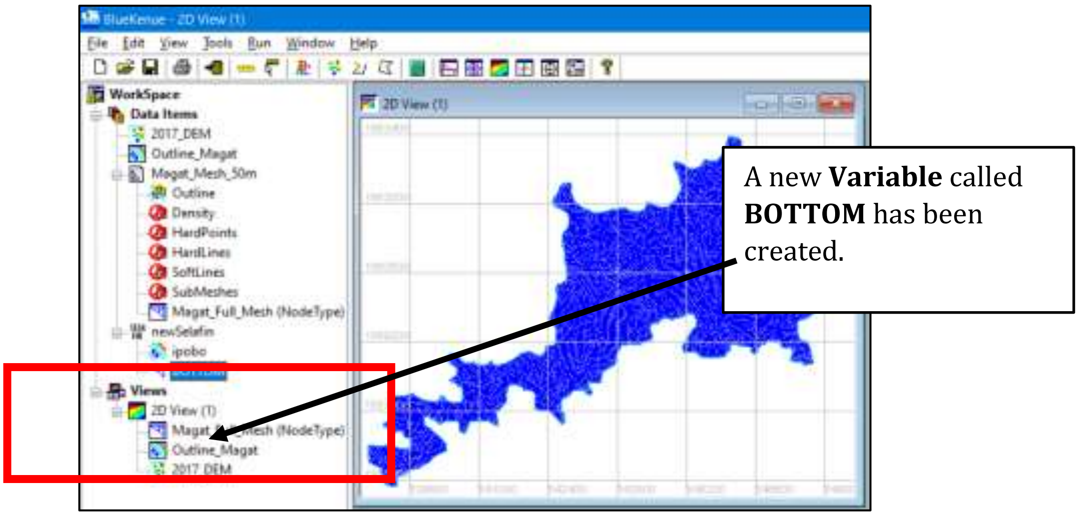


*Figure 46 The new **Variable** has been added*

5. Next, from the Menu Bar, click "**File**" >> "**New**" >> "**2D Interpolator**".

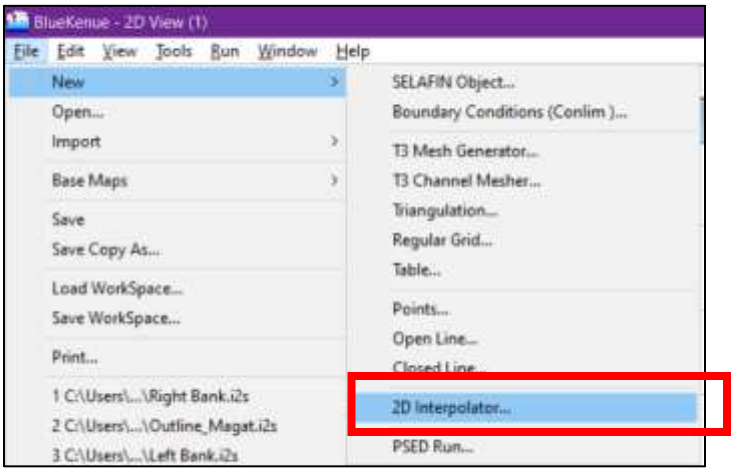


Figure 47. Adding a 2D Interpolator

6. Notice that a “**newInterpolator2D**” is now available in the Data Items menu.
7. Drag the “**2017_DEM**” into the newly created 2D interpolator.

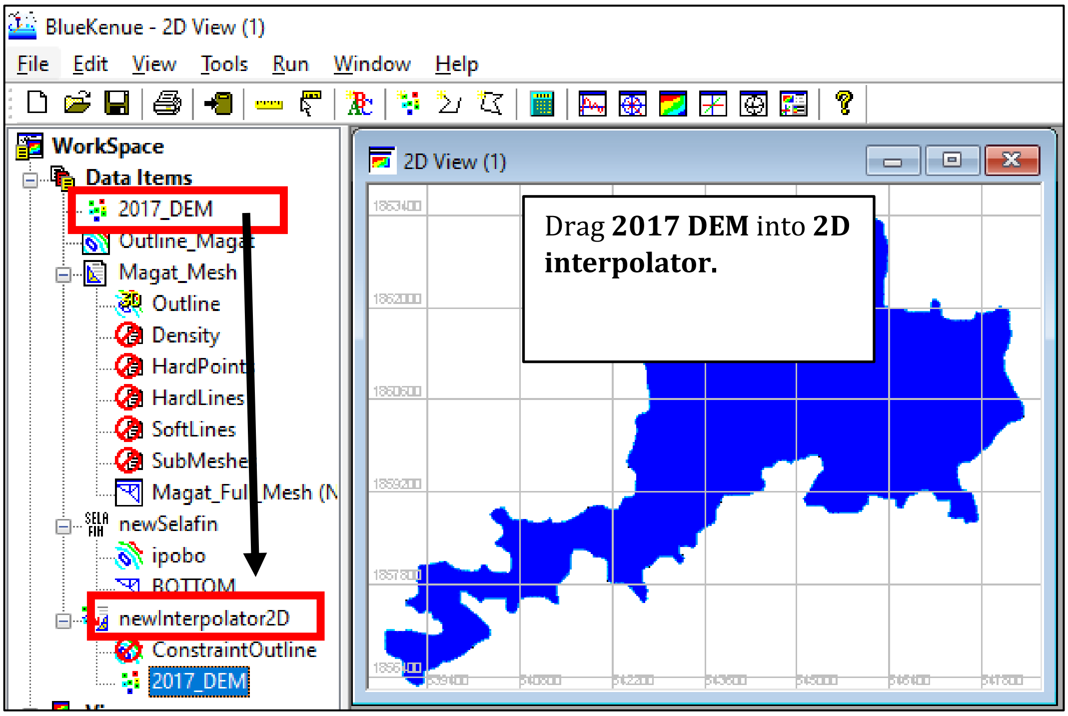


Figure 48. Dragging the 2017_DEM Item into the Newly Added 2D Interpolator Tool

8. Next, click the “**BOTTOM**” item under the **newSelafin** file to highlight it.

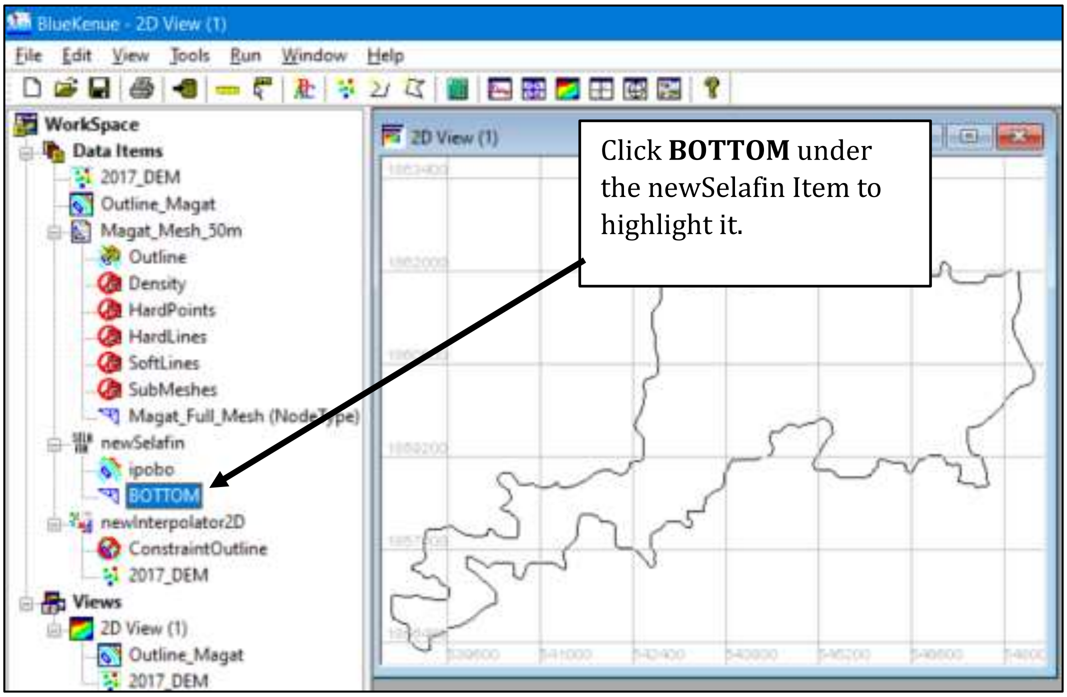


Figure 49. Selecting the Bottom Item

9. Then from the Menu Bar, click “**Tools**” >> “**Map Object**”.

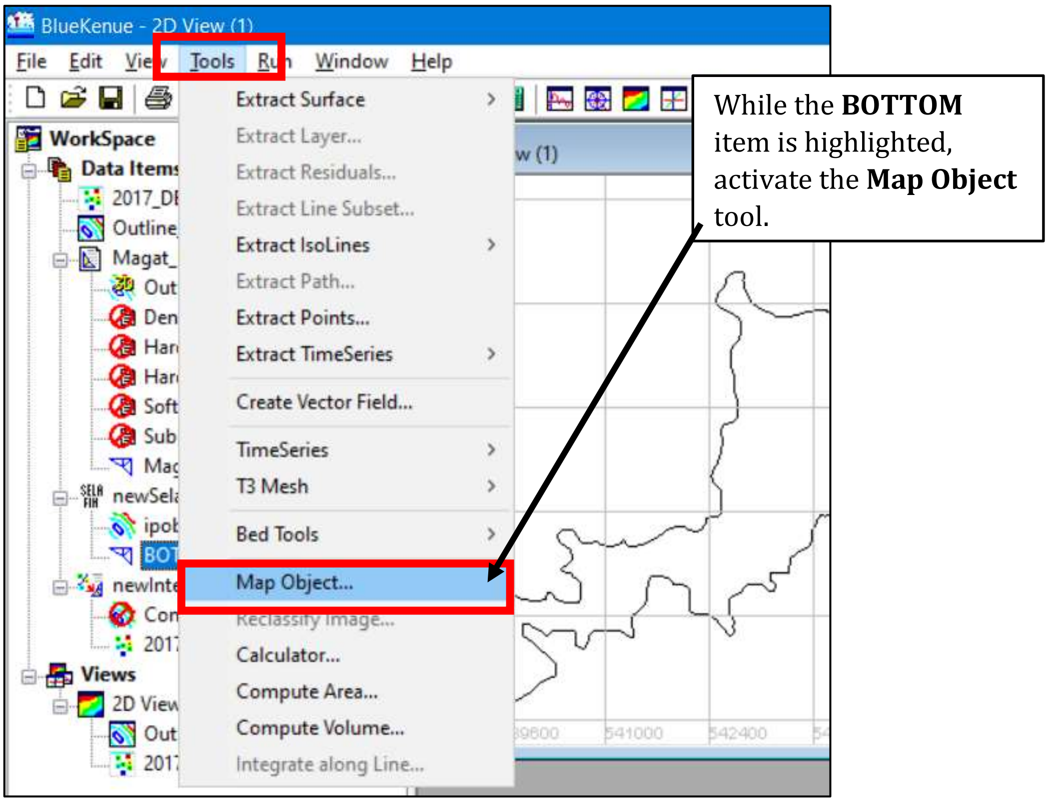


Figure 50. The Map Object Tool

10. A pop-up window appears showing the **Available Objects** from which you select the “**newInterpolator2D**” that we have just created. Then click “**OK**”.

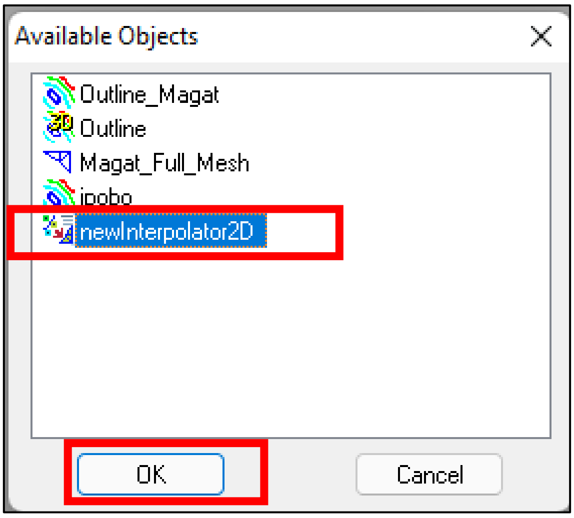


Figure 51. Choosing the NewInterpolator2D

11. A processing window will be shown, then once done, click the "**OK**" button.

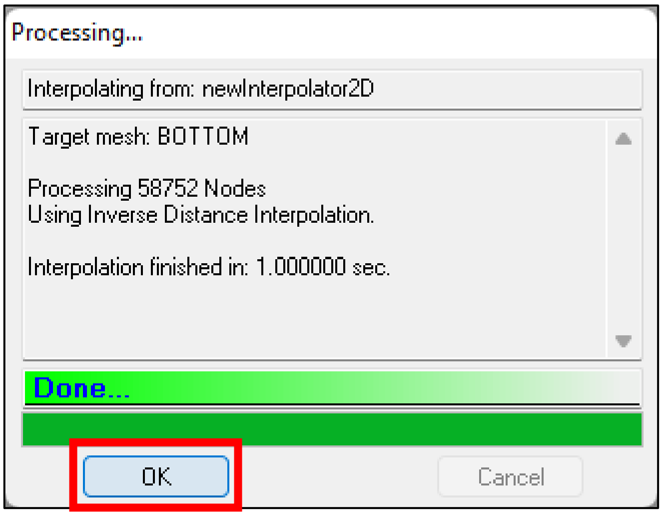


Figure 52. A Successful Interpolation Finished in 1.000 Seconds

12. Now, to check whether the process is successful, drag the "**BOTTOM**" item into the 2D View.

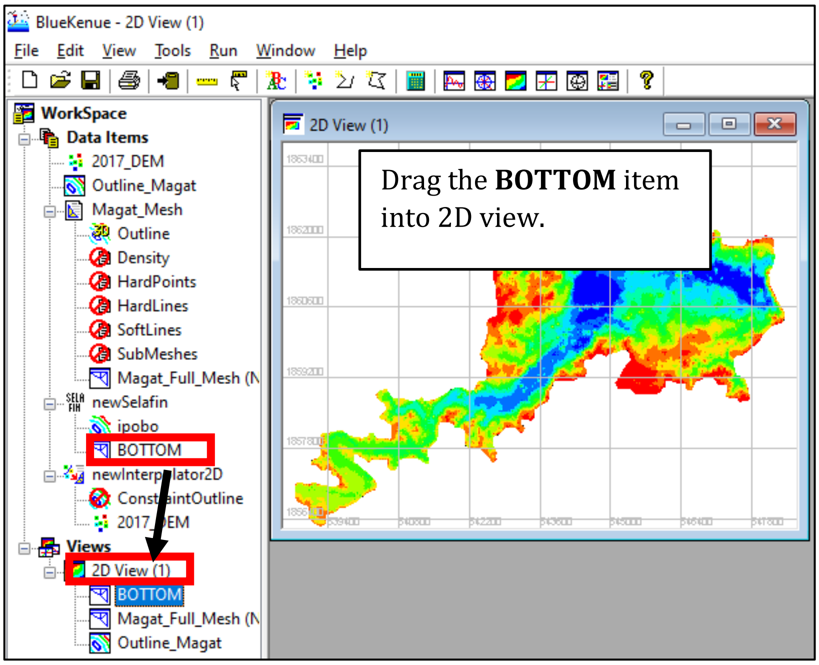


Figure 53. Bottom Item in 2D View

13. If you try to zoom in, you can see that the mesh now possesses the same color as that of the DEM file used.

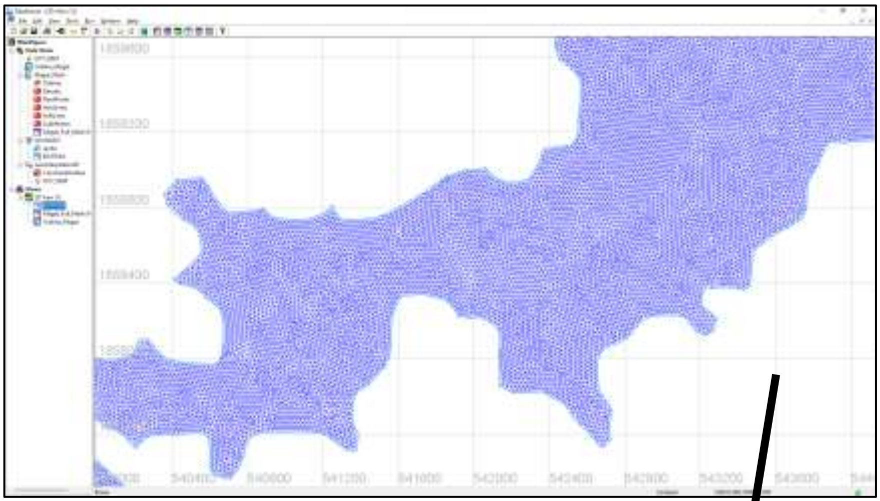


Figure 54. The Mesh before the Interpolation

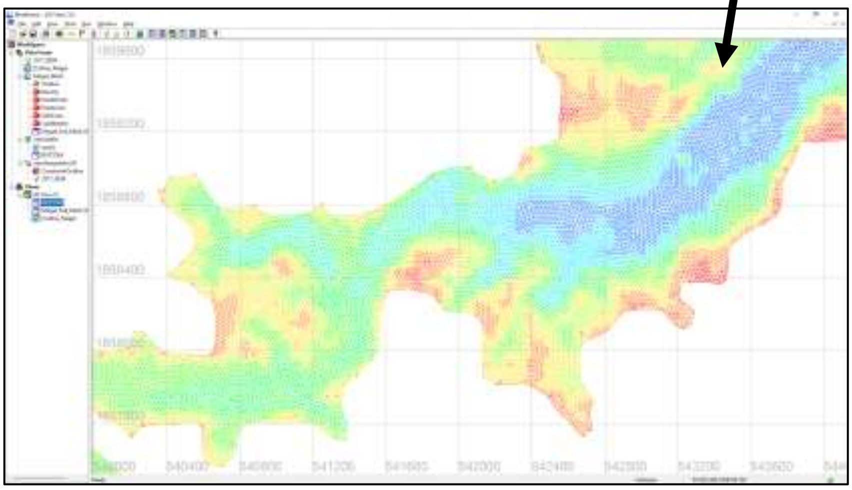


Figure 55. The Mesh after the Interpolation

14. The final step would be to save the Selafin file which is one of the mandatory files needed to run a TELEMAC simulation.

15. Click the "**newSelafin**" item to highlight it, then click the **Save** icon in the file.

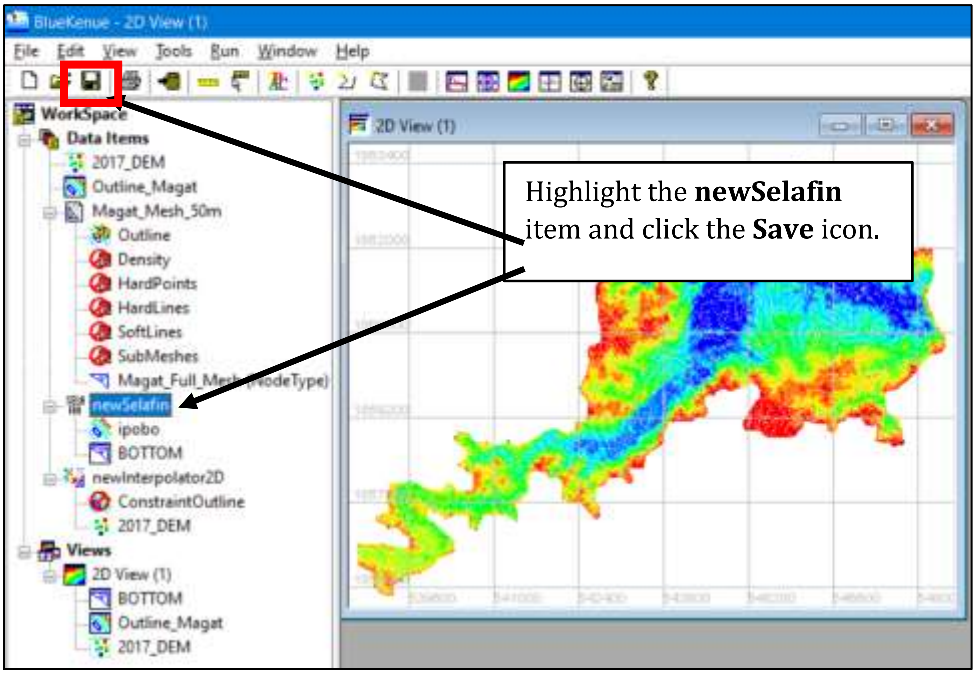
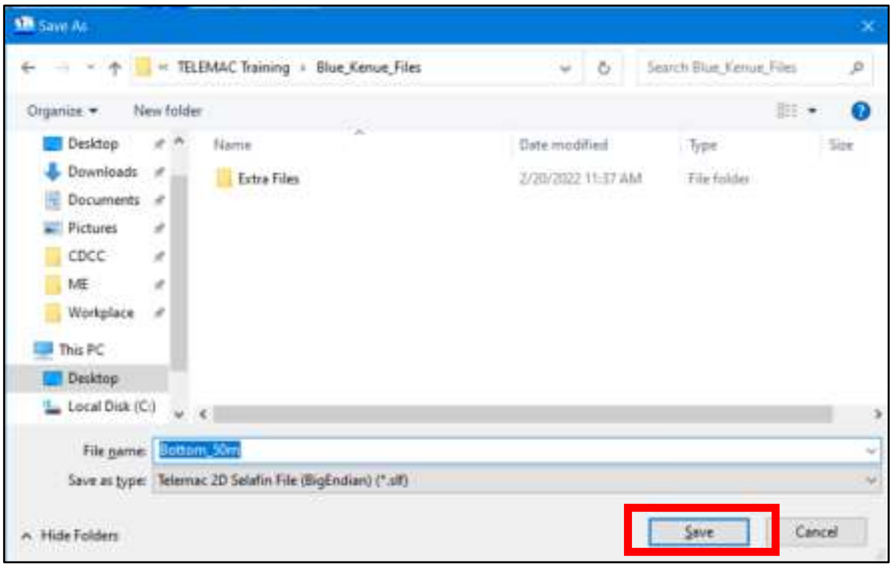


Figure 56. Highlighting Selafin Item and Saving it into File

16. Enter an appropriate name (i.e., Bottom_50m.slf), then click "**Save**".



17. Finally, save your workspace by following the steps described on the next page:

3.1.5. Saving Files and the Workspace

1. We can save the whole workspace into a file which can be useful when transferring to another computer or when picking up from an unfinished session.

2. From the menu bar, click "**File**" >> "**Save Workspace**".

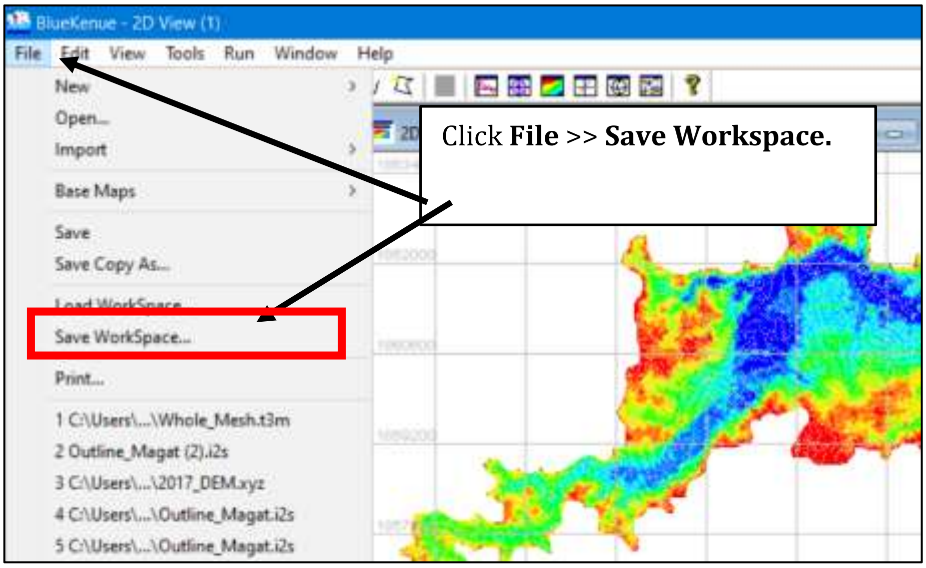


Figure 57 Saving an entire Workspace in Blue Kenue

3. Enter a suitable file name for the Workspace (i.e., Magat_Workspace) then click "**Save**".

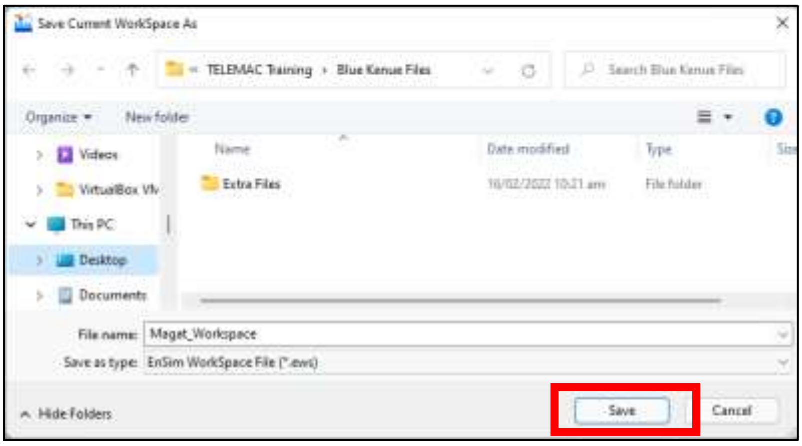


Figure 58. Entering the File Name and Saving Your Workspace

4. A toast with a message that the unsaved objects must be saved first before the Workspace can be saved will pop up. Click "**Yes**" to continue.

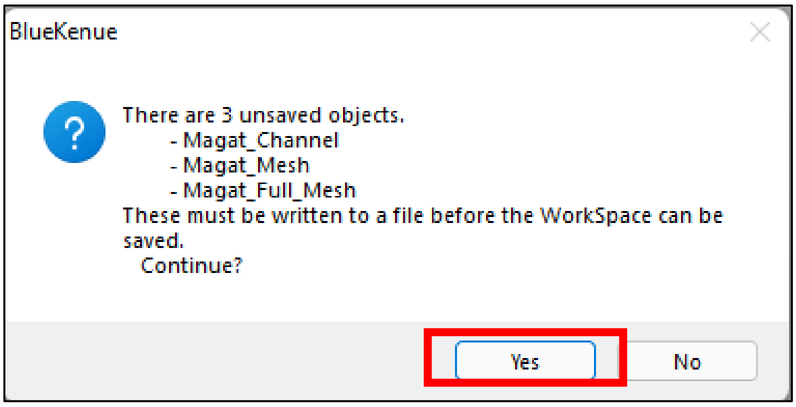


Figure 59. Saving All Unsaved Items

5. Enter appropriate names for each file then click "**Save**".

Note: The Workspace File contains all the data object settings, including color scales, legend options, scaling, rendering options, line width, point size, etc. for all currently open objects. Aside from saving data object settings, it also saves the settings for all the views and the object view relationships.

3.2. Generating Boundary Conditions File Using Blue Kenue

Boundary conditions in the TELEMAC modeling system are specified through a table of variables known as the CONLIM table. Each record in this table corresponds to a boundary (or edge) node in a model's finite element mesh.

Information regarding inflow and outflow nodes is contained in the boundary file in *.cli format (coordinated and IDs).

1. Continuing from the previous section, we have a workspace containing all the generated meshes and Selafin objects.

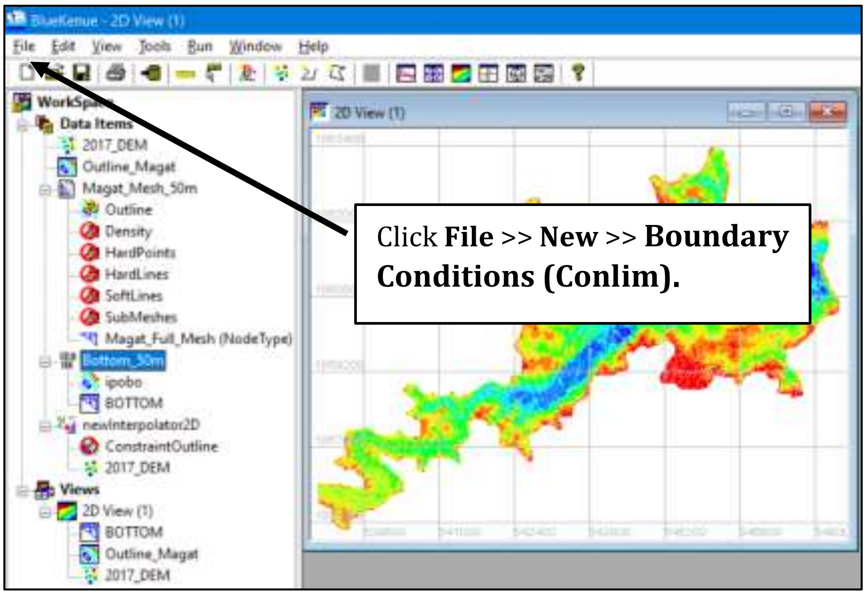


Figure 60. Current Workspace

2. From the menu bar, again, click "**File**" >> "**New**" >> "**Boundary Conditions (Conlim).**"

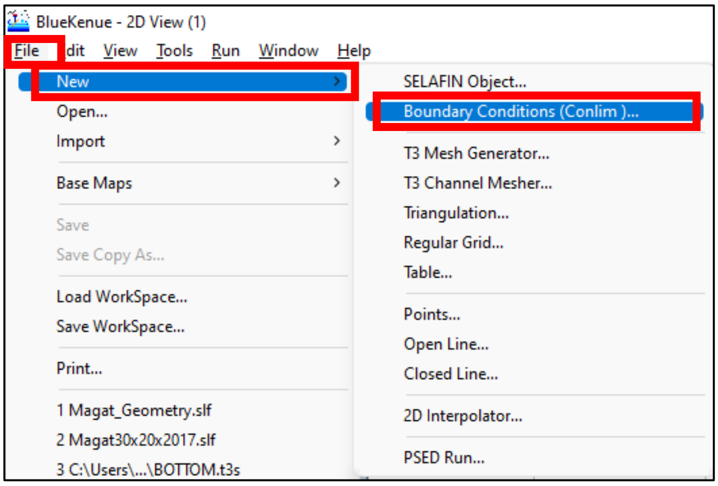


Figure 61. Steps in Creating a New Boundary Conditions Item

3. A window showing the available t3s Objects will show. Select “**Bottom**”, then click “**OK**”.

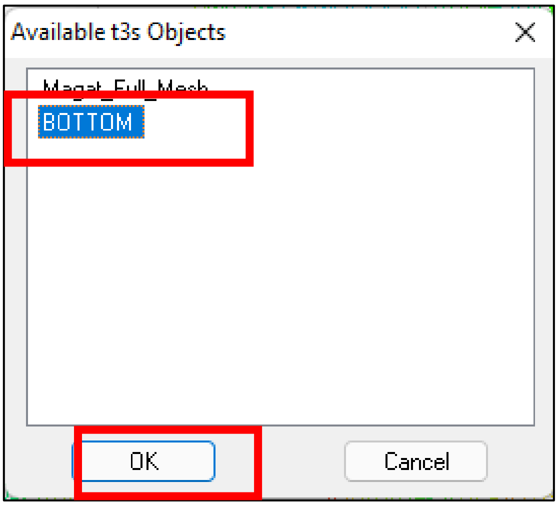


Figure 62. Available t3s Objects

4. Notice that a “**BOTTOM_BC**” item has been added under the Data Item menu.

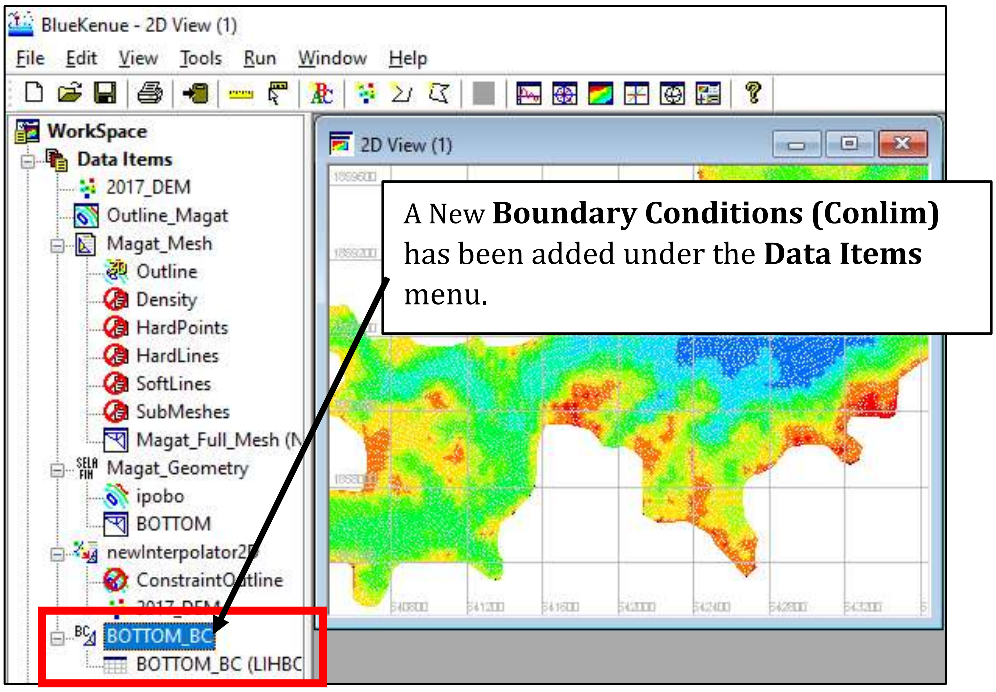


Figure 63. The Newly Created Boundary Conditions Item

5. To start working on the boundary conditions file, drag the "**BOTTOM_BC**" item into the **2D View.**

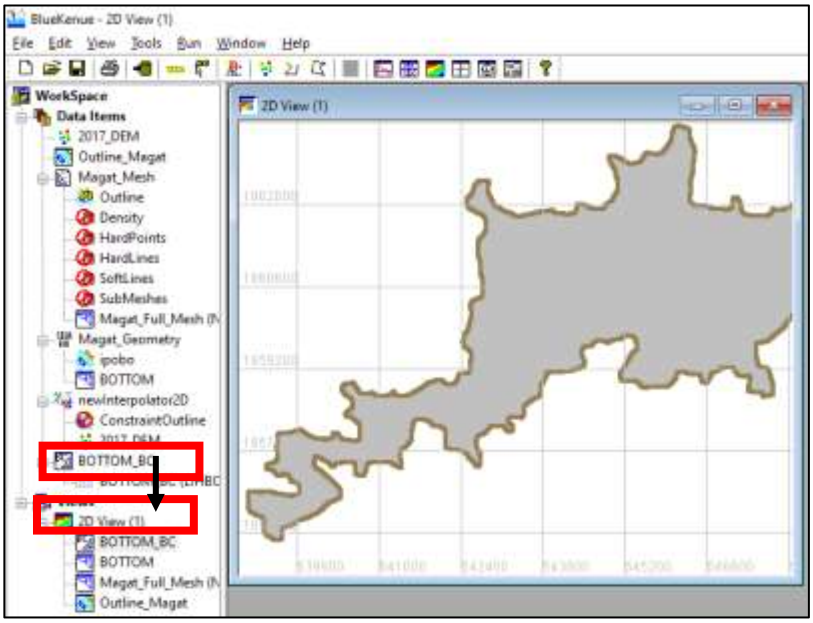


Figure 64. Dragging BOTTOM_BC into 2D View

6. Notice that the color of the mesh in the 2D view now turned gray and the boundary nodes are colored brown. By default, this node color representation means a **Closed Boundary (wall)**.

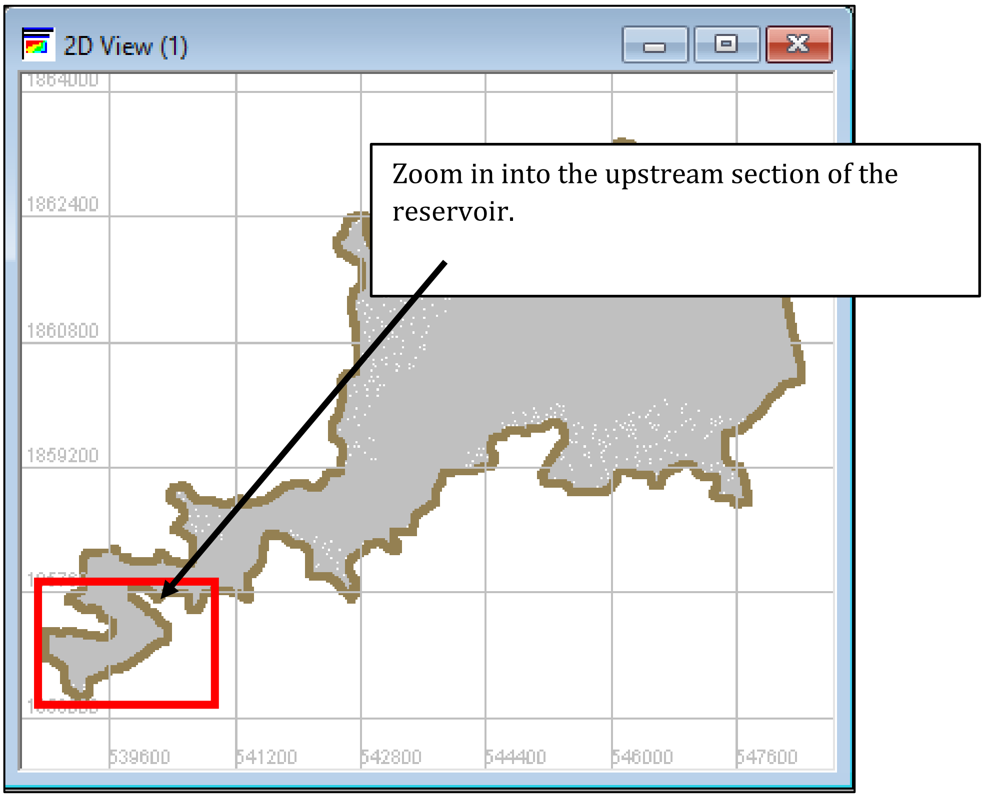


Figure 65. The 2D View after Dragging BOTTOM_BC

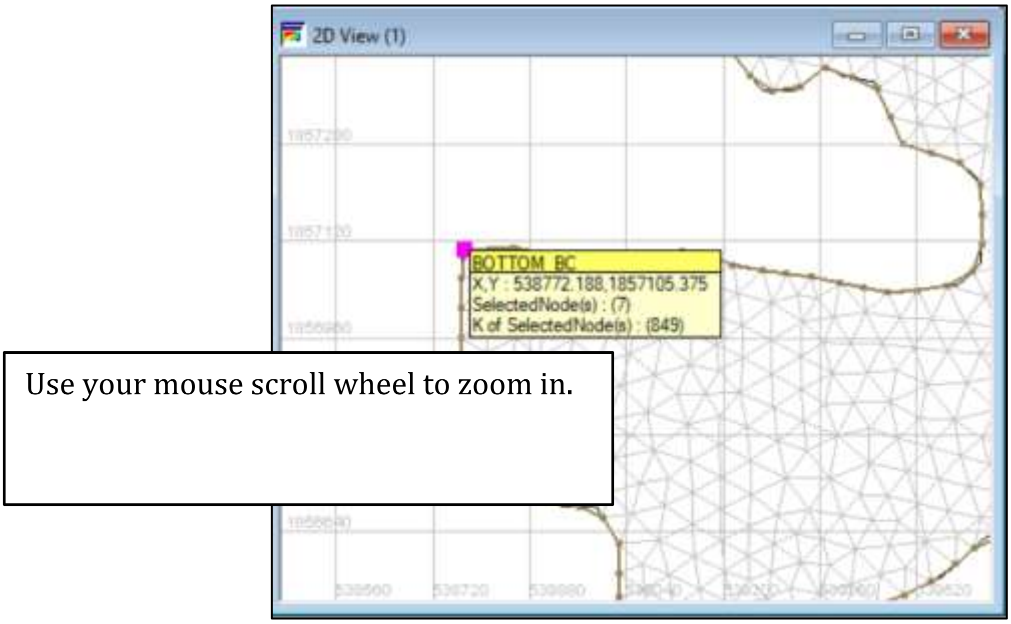


Figure 66. Zooming in on the Upstream Section of the Reservoir

7. It would help if you maximize your 2D View window.

8. Once zoomed in, point the cursor into a node at the edge and double-click. This will create a magenta-colored dot. Now hold the "**SHIFT**" key on your keyboard and double-click on the other side of the mesh. This will create a magenta-colored line with nodes.

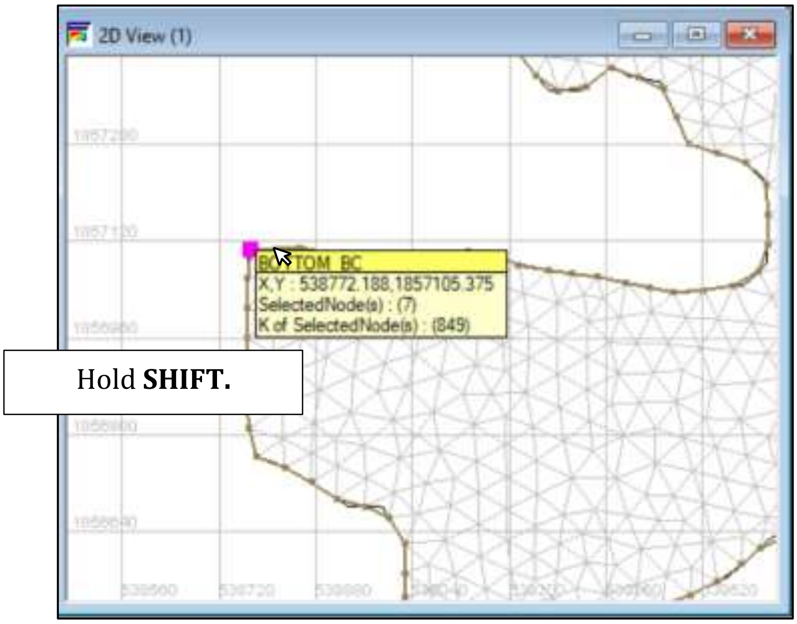


Figure 67. Double-clicking a Node to Start

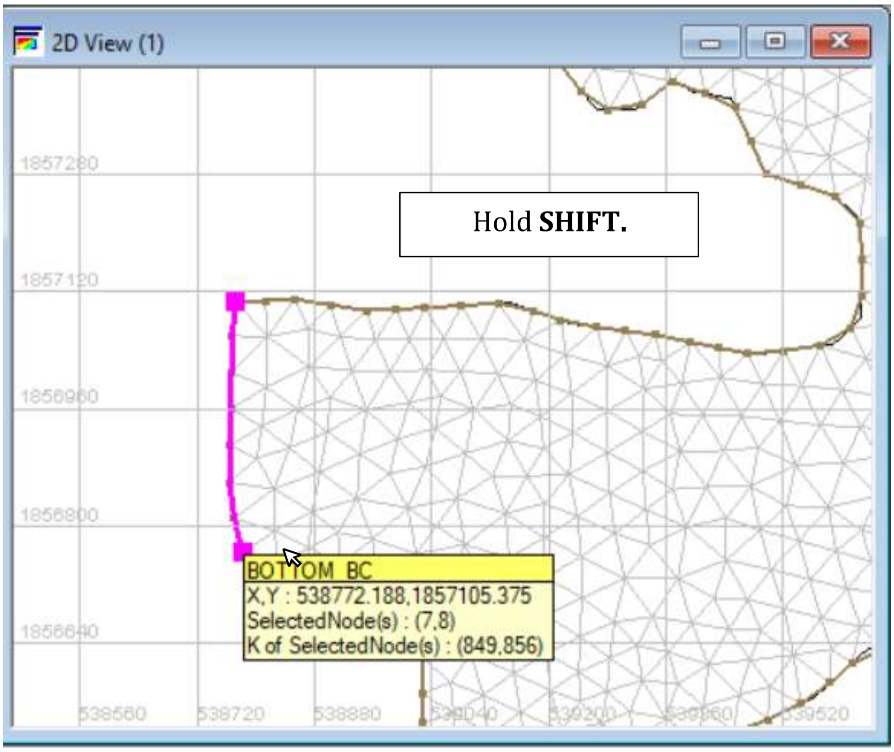


Figure 68. Double-clicking the Other End of the Upstream Section

9. Now, right-click the magenta-colored line and select "**Add Boundary Segment**".

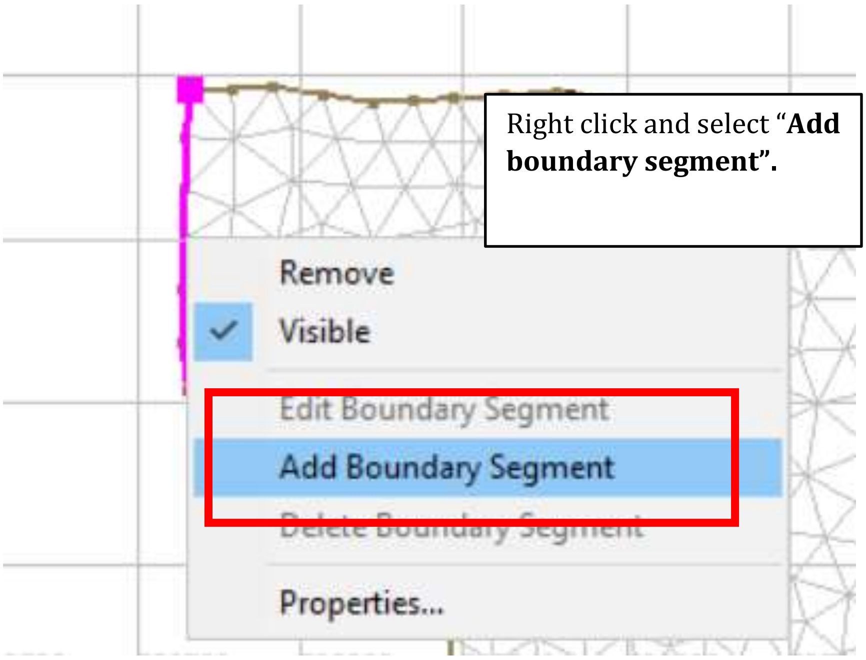


Figure 69. Selecting Add Boundary Segment

10. A “**Conlim Boundary Segment Editor**” window will pop up.

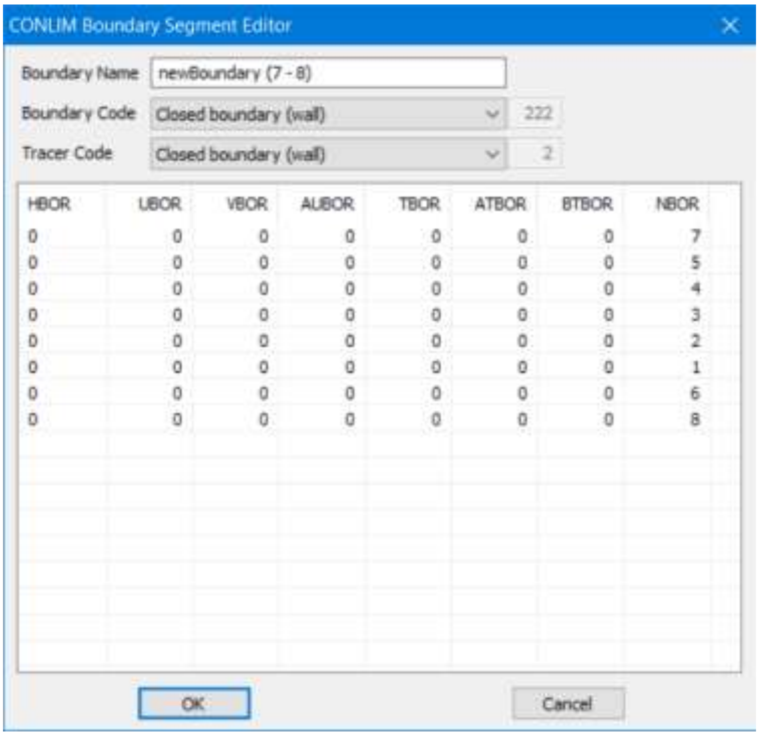


Figure 70. The Conlim Boundary Segment Editor

11. In the “**Boundary Name**” section, type “**Inflow**”, while in the “**Boundary Code**”, choose “**Open boundary with prescribed Q**”. Leave the rest unmodified, and click “**OK**”.

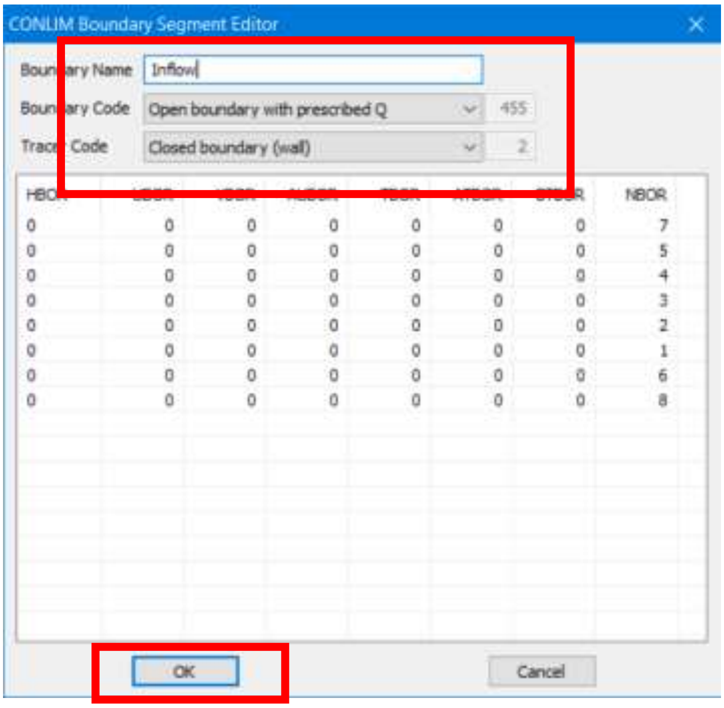


Figure 71. Boundary Properties for the Upstream Boundary Section

Note:

- *Q stands for volumetric flowrate, H is for water depth, and UV is for velocity.*
- *The magenta-colored nodes have been changed into blue nodes, which represents an open boundary with prescribed Q.*

Boundary Type	Code	Colour	Point Style
Closed boundary (wall)	222	brown	filled square
Open boundary with prescribed Q	455	blue	triangle
Open boundary with prescribed H	544	green	inverted triangle
Open boundary with prescribed Q and H	555	cyan	diamond
Open boundary with prescribed UV	466	red	half filled square
Open boundary with prescribed UV and H	566	orange	square
Open boundary with incident waves	111	yellow	circle
Custom	???	black	star

12. Do the same process for the outflow/downstream boundary. A GIS-derived image is attached herein as a reference.

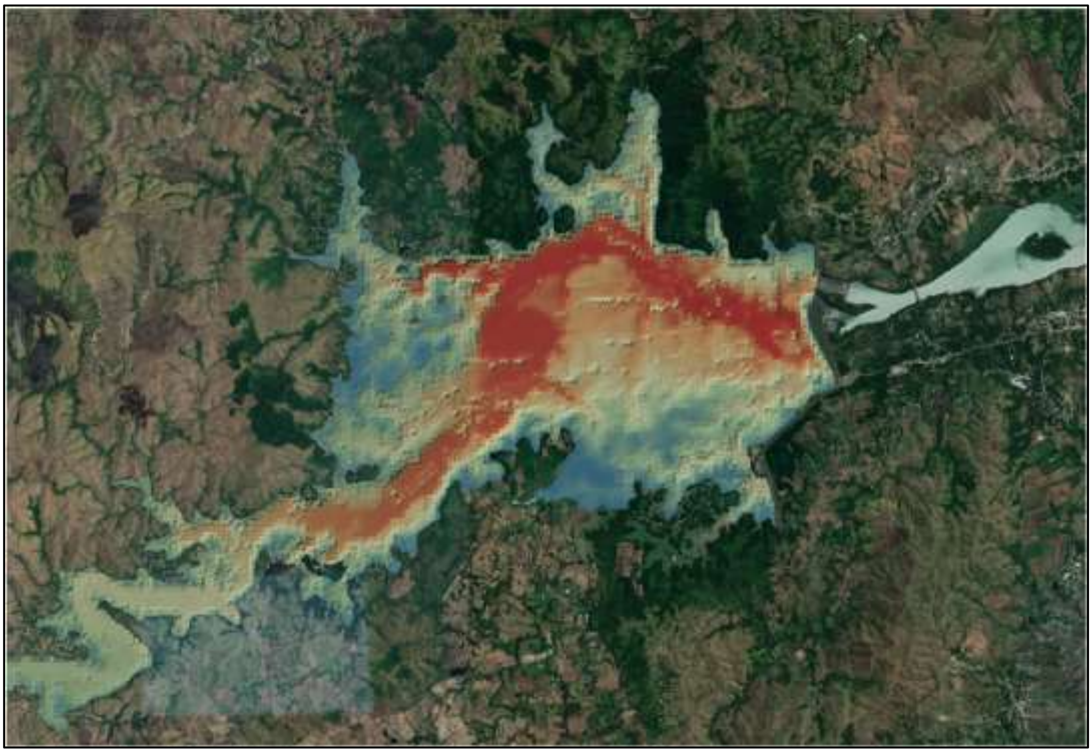


Figure 72. Base Map and the Digital Elevation File for Magat Reservoir

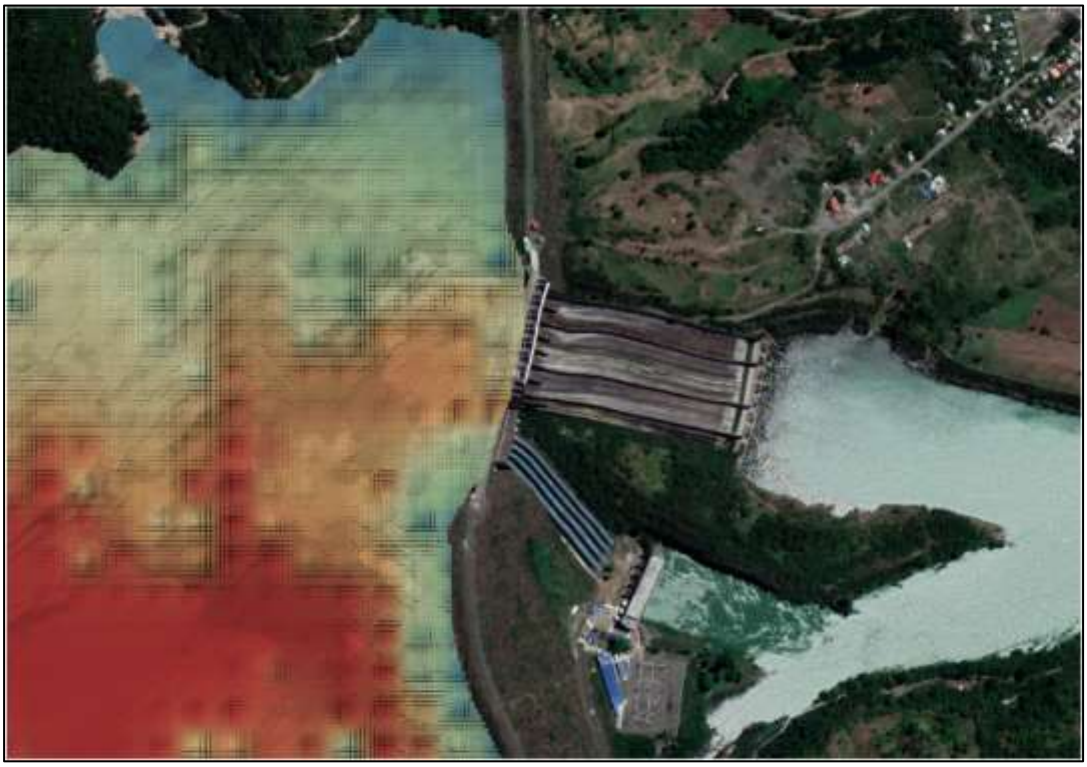


Figure 73. Zooming in on the Discharge Section

13. Back to Blue Kenue, zoom in on the discharge boundary.
14. Execute the same process, double-click to start, hold the **SHIFT** key, then double-click again approximately where the penstock pipes end.

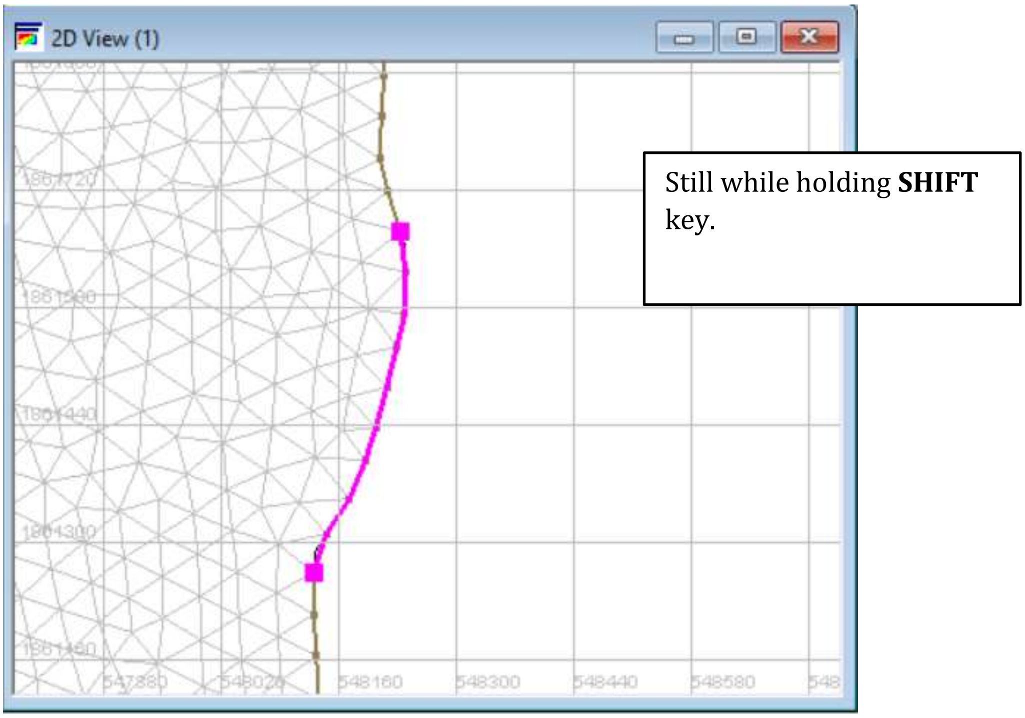


Figure 74. Double-clicking up to the End of the Downstream Section

15. Right-click and select “**Add Boundary Segment.**”

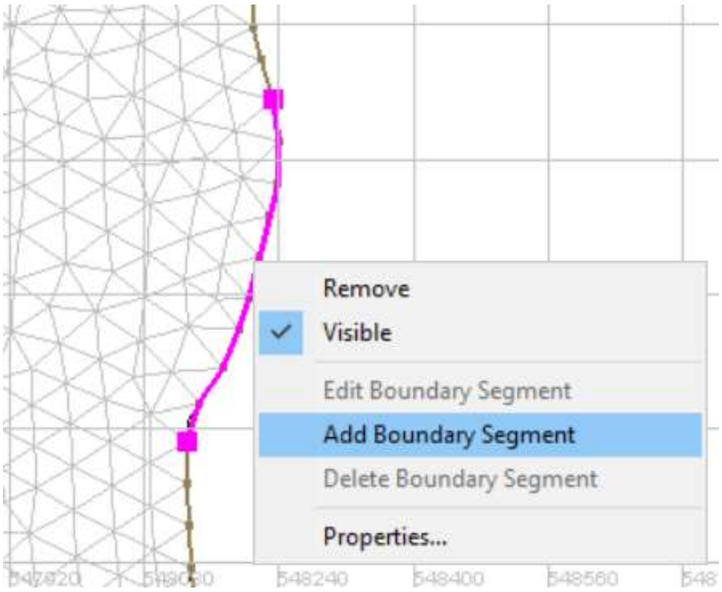


Figure 75. Adding Boundary Segment for the Downstream Boundary

16. In the "**Boundary Name**" section, type "**Outflow**", while in the "**Boundary Code**", choose "**Open boundary with prescribed H**". Leave the rest unmodified, then click "**OK**".

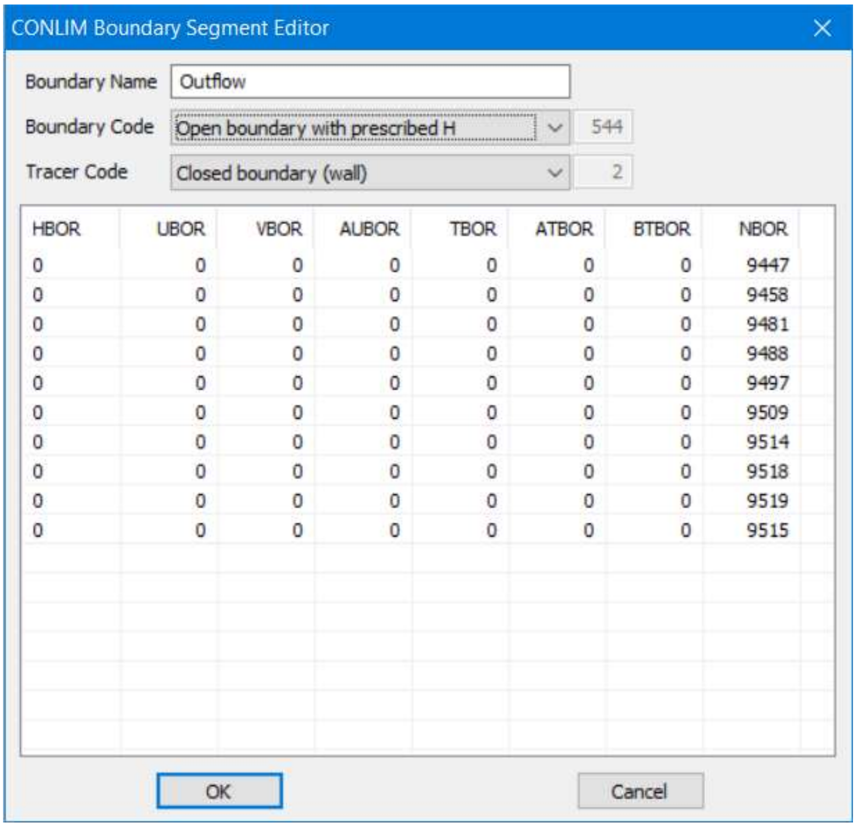


Figure 76. Boundary Properties for the Downstream Boundary Section

17. Again, the magenta-colored nodes now turn to green, a color representation for **Open boundaries with prescribed H nodes**.

18. Rename the newly generated Conlim file into "**Boundary_Conditions**".

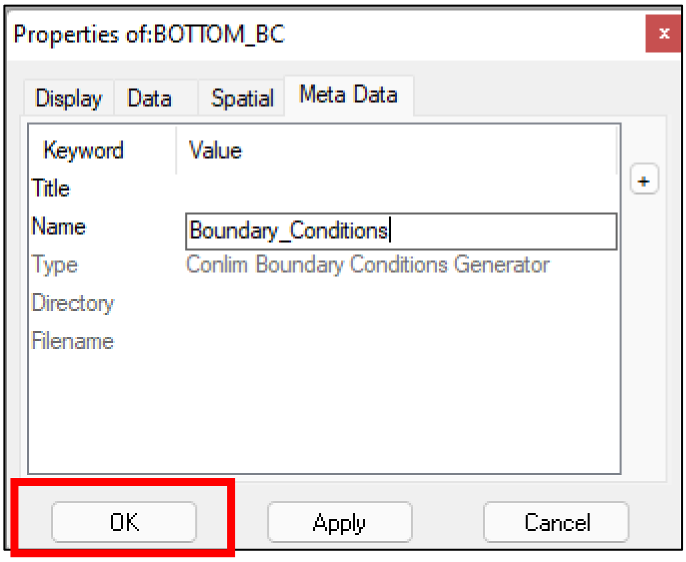


Figure 77. Rename the Items

19. Finally, highlight the child object under it named "**Boundary_Conditions (LIHBOR)** and click the "**Save**" icon to save it into a file.

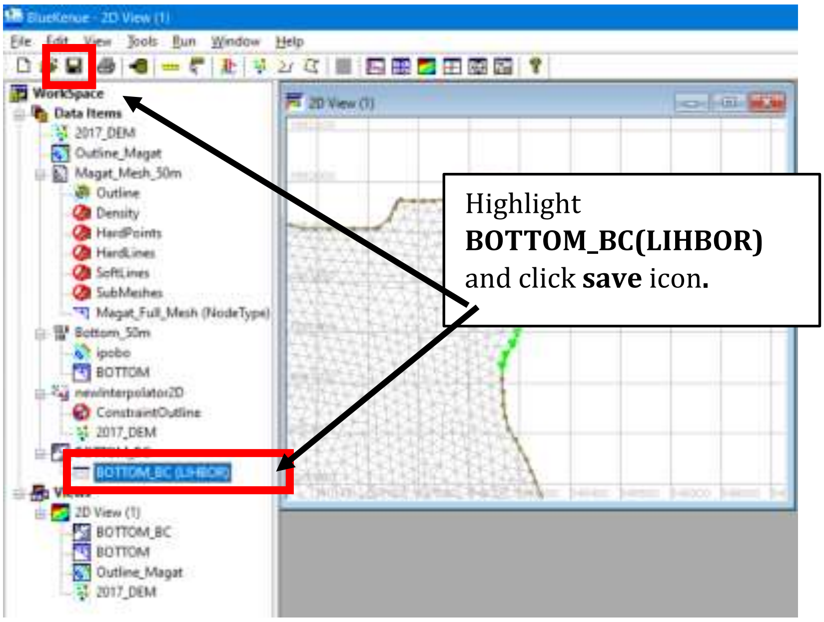


Figure 78. Saving the Generated Conlim Item

20. Enter the appropriate name (i.e., "**Boundary_Conditions**") and then save.

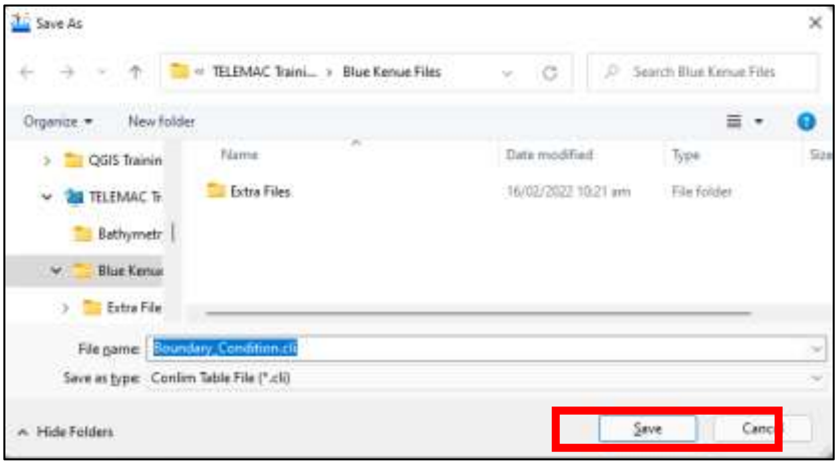


Figure 79. Saving the Boundary Conditions into a File

Note:

Various Types of Boundary Conditions and their descriptions.

Position	Name	Description
1	LIHBOR	Code for Depth
2	LIUBOR	Code for U (velocity or flow)
3	LIVBOR	Code for V (velocity or flow)
4	HBOR	Prescribed value for Depth
5	UBOR	Prescribed value for U (velocity or flow)
6	VBOR	Prescribed value for V (velocity or flow)
7	AUBOR	Friction coefficient
8	LITBOR	Code for Tracer
9	TBOR	Prescribed Tracer value
10	ATBOR	Tracer coefficient A
11	BTBOR	Tracer coefficient B
12	NBOR	Global node number in mesh
13	K	Sequential boundary node number

Figure 80. Various Types of Boundary Conditions

21. Finally, save your workspace again by following the steps described above.

3.3. Understanding and Creating a Steering File (CAS File)

The steering file is the file ending in the *.cas file extension. It is the main simulation file with information about references to the two mandatory files (i.e., the SELAFIN *.slf geometry and the *.cli boundary files) and optional files, as well as definitions of simulation parameters. It can be created or edited with a basic text editor or advanced GUI software such as Fudaa PrePro or BlueKenue.

In this manual, we will try to show a sample steering file ***before*** and ***after*** corrections and adjustments.

3.3.1. Sample Steering File for a **Purely Hydrodynamic Scenario** in Magat Reservoir (Initial/Uncorrected)

```
/---------------------------------------------------------------------
/                FILE DEFINITION
/---------------------------------------------------------------------
BOUNDARY CONDITIONS FILE                                    : 'BOTTOM_BC.cli'
GEOMETRY FILE                                               : 'Bottom_50m.slf'
RESULTS FILE                                                : 'Magat_V2.slf'
/---------------------------------------------------------------------
/                 GENERAL OPTIONS
/---------------------------------------------------------------------
TITLE                                                       : 'Magat'
/
VARIABLES FOR GRAPHIC PRINTOUTS                             : 'U,V,F,H,B,S,Q,W,L'
TIME STEP                                                   : 60 /it should be determined by Courant Number
NUMBER OF TIME STEPS                                        : 600
GRAPHIC PRINTOUT PERIOD                                     : 1
LISTING PRINTOUT PERIOD                                     : 10
TIDAL FLATS                                                 : YES
OPTION FOR THE TREATMENT OF TIDAL FLATS                     : 1
/---------------------------------------------------------------------
/               BOUNDARY CONDITIONS
/---------------------------------------------------------------------
PRESCRIBED ELEVATIONS                                       :188.28; 0.0 /downstream boundary
PRESCRIBED FLOWRATES                                        :0.0; 183.78 /upstream boundary
/---------------------------------------------------------------------
/                INITIAL CONDITIONS
/---------------------------------------------------------------------
INITIAL CONDITIONS                                          :'CONSTANT ELEVATION'
INITIAL ELEVATION                                           : 188.28 /initial water level
/---------------------------------------------------------------------
/                NUMERICAL OPTIONS (not necessary to modify for beginner)
/---------------------------------------------------------------------
MASS-BALANCE                                                : YES
SOLVER                                                      : 1
SOLVER OPTION                                               : 2
SOLVER ACCURACY                                             : 1.E-4
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER                     : 1000
PRECONDITIONING                                             : 2
TYPE OF ADVECTION                                           : 14;14
SUPG OPTION                                                 : 0;0
DISCRETIZATIONS IN SPACE                                    : 11; 11
MATRIX STORAGE                                              : 3
IMPLICITATION FOR DEPTH                                     : 1.0
IMPLICITATION FOR VELOCITY                                  : 1.0
VELOCITY PROFILES                                           : 1;1
MASS-LUMPING ON H                                           : 1.0
MASS-LUMPING ON VELOCITY                                    : 1.0
MASS-LUMPING ON TRACERS                                     : 1.0
TREATMENT OF THE LINEAR SYSTEM                              : 2
FREE SURFACE GRADIENT COMPATIBILITY                         : 0.5
CONTINUITY CORRECTION                                       : YES
TREATMENT OF NEGATIVE DEPTHS                                : 2
OPTION FOR LIQUID BOUNDARIES                                : 1;1
```

```
OPTION FOR THE DIFFUSION OF VELOCITIES                    : 1
/---------------------------------------------------------------------
/                  TURBULENCE MODEL
/---------------------------------------------------------------------
TURBULENCE MODEL                                   : 3 / 1:constant; 3:K-EPSILON; 4:SMAGORINSKI; 5:MIXING LENGTH
VELOCITY DIFFUSIVITY                                      : 1.E-6
TURBULENCE REGIME FOR SOLID BOUNDARIES                     : 2
/-------------------------------------------------------/
/   FRICTION PARAMETERS                          /
/-------------------------------------------------------/
LAW OF BOTTOM FRICTION                                     : 4 /2:CHEZY; 4:MANNING; 5:NIKURADSE
FRICTION COEFFICIENT                                       : 0.023
/---------------------------------------------------------------------
```

Note:

Please see Appendix III to see visualized results files

These files will be provided for the actual training session. See the ***Simulation_Files*** *folder.*

3.3.2. Sample Steering File for a Purely Hydrodynamic Scenario in Magat Reservoir (Adjusted/Corrected)

```
/---------------------------------------------------------------------
/                 FILE DEFINITION
/---------------------------------------------------------------------
BOUNDARY CONDITIONS FILE                                   : 'BOTTOM_BC.cli'
GEOMETRY FILE                                              : 'Bottom_50m.slf'
RESULTS FILE                                               : 'Magat_V2.slf'
/---------------------------------------------------------------------
/                  GENERAL OPTIONS
/---------------------------------------------------------------------
TITLE : 'Magat'
/
VARIABLES FOR GRAPHIC PRINTOUTS                              : 'U,V,F,H,B,S,Q,W,L'
TIME STEP                                           : 25 /it should be determined by Courant Number
NUMBER OF TIME STEPS                                        : 3456
GRAPHIC PRINTOUT PERIOD                                     : 10
LISTING PRINTOUT PERIOD                                     : 10
TIDAL FLATS                                                                              :
YES
OPTION FOR THE TREATMENT OF TIDAL FLATS                    : 1
/---------------------------------------------------------------------
/                BOUNDARY CONDITIONS
/---------------------------------------------------------------------
PRESCRIBED ELEVATIONS                                      :191.65; 0.0 /downstream boundary
PRESCRIBED FLOWRATES                                       :0.0; 4847.74 /upstream boundary
/---------------------------------------------------------------------
/                 INITIAL CONDITIONS
/---------------------------------------------------------------------
INITIAL CONDITIONS                                         :'CONSTANT ELEVATION'
INITIAL ELEVATION                                          : 191.65 /initial water level
/---------------------------------------------------------------------
/                 NUMERICAL OPTIONS (not necessary to modify for beginner)
/---------------------------------------------------------------------
/
MASS-BALANCE                                               : YES
SOLVER                                                     : 1
SOLVER OPTION                                              : 2
SOLVER ACCURACY                                            : 1.E-4
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER                    : 1000
PRECONDITIONING                                            : 2
TYPE OF ADVECTION                                          : 14;14
SUPG OPTION                                                : 0;0
DISCRETIZATIONS IN SPACE                                   : 11; 11
MATRIX STORAGE                                             : 3
IMPLICITATION FOR DEPTH                                    : 1.0
IMPLICITATION FOR VELOCITY                                 : 1.0
VELOCITY PROFILES                                          : 1;1
MASS-LUMPING ON H                                          : 1.0
MASS-LUMPING ON VELOCITY                                   : 1.0
MASS-LUMPING ON TRACERS                                    : 1.0
TREATMENT OF THE LINEAR SYSTEM                             : 2
FREE SURFACE GRADIENT COMPATIBILITY                        : 0.5
CONTINUITY CORRECTION                                      : YES
TREATMENT OF NEGATIVE DEPTHS                               : 2
OPTION FOR LIQUID BOUNDARIES                               : 1;1
OPTION FOR THE DIFFUSION OF VELOCITIES                     : 1
/---------------------------------------------------------------------
/                 TURBULENCE MODEL
/---------------------------------------------------------------------
TURBULENCE MODEL                   : 3 / 1:constant; 3:K-EPSILON; 4:SMAGORINSKI; 5:MIXING LENGTH
VELOCITY DIFFUSIVITY               : 1.E-6
TURBULENCE REGIME FOR SOLID BOUNDARIES     : 2
/-----------------------------------------------------/
/  FRICTION PARAMETERS                                /
/-----------------------------------------------------/
LAW OF BOTTOM FRICTION                : 4 /2:CHEZY; 4:MANNING; 5:NIKURADSE
FRICTION COEFFICIENT               : 0.023
```

3.3.3. Sample Steering File for Modelling a Sediment Transport Scenario using TELEMAC coupled with the GAIA Module (Initial/Uncorrected)

The steering files used in coupling hydrodynamics (TELEMAC 2D/3D) and morphodynamics (Gaia) are fairly the same as the sample described above. However, a couple of new keywords for the sediment transport must be added to the steering file:

```
/---------------------------------------------------------------------
/                FILE DEFINITION
/---------------------------------------------------------------------
BOUNDARY CONDITIONS FILE                          : 'BOTTOM_BC.cli'
GEOMETRY FILE                                     : 'Bottom_50m.slf'
RESULTS FILE                                      : 'Magat_V2.slf'
/---------------------------------------------------------------------
/                 GENERAL OPTIONS
/---------------------------------------------------------------------
TITLE                                             : 'Magat'
/
VARIABLES FOR GRAPHIC PRINTOUTS                   : 'U,V,F,H,B,S,Q,W,L'
TIME STEP                                         : 60 /it should be determined by Courant Number
NUMBER OF TIME STEPS                              : 600
GRAPHIC PRINTOUT PERIOD                           : 1
LISTING PRINTOUT PERIOD                           : 10
TIDAL FLATS                                       : YES
OPTION FOR THE TREATMENT OF TIDAL FLATS           : 1
/---------------------------------------------------------------------
/                            COUPLING
/---------------------------------------------------------------------
COUPLING WITH                           : 'GAIA'
GAIA STEERING FILE                                : 'Gaia_Magat_2D.cas' /steering file name of gaia
/---------------------------------------------------------------------
/               BOUNDARY CONDITIONS
/---------------------------------------------------------------------
PRESCRIBED ELEVATIONS                             :188.28; 0.0 /downstream boundary (water level)
PRESCRIBED FLOWRATES                              :0.0; 183.78 /upstream boundary (inflow discharge)
PRESCRIBED TRACERS VALUES                         :0.0; 1.0 /upstream boundary (inflow sediment)
/---------------------------------------------------------------------
/                INITIAL CONDITIONS
/---------------------------------------------------------------------
INITIAL CONDITIONS                                :'CONSTANT ELEVATION'
INITIAL ELEVATION                                 : 188.28 /initial water level
/---------------------------------------------------------------------
/                 NUMERICAL OPTIONS (not necessary to modify for beginner)
/---------------------------------------------------------------------
MASS-BALANCE                                      : YES
SOLVER                                            : 1
SOLVER OPTION                                     : 2
SOLVER ACCURACY                                   : 1.E-4
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER           : 1000
PRECONDITIONING                                    : 2
TYPE OF ADVECTION                                 : 14;14
SUPG OPTION                                       : 0;0
DISCRETIZATIONS IN SPACE                           : 11; 11
MATRIX STORAGE                                    : 3
IMPLICITATION FOR DEPTH                           : 1.0
IMPLICITATION FOR VELOCITY                        : 1.0
VELOCITY PROFILES                                 : 1;1
MASS-LUMPING ON H                                 : 1.0
MASS-LUMPING ON VELOCITY                          : 1.0
MASS-LUMPING ON TRACERS                           : 1.0
TREATMENT OF THE LINEAR SYSTEM                    : 2
FREE SURFACE GRADIENT COMPATIBILITY               : 0.5
CONTINUITY CORRECTION                             : YES
TREATMENT OF NEGATIVE DEPTHS                      : 2
OPTION FOR LIQUID BOUNDARIES                      : 1;1
OPTION FOR THE DIFFUSION OF VELOCITIES            : 1
/---------------------------------------------------------------------
/                 TURBULENCE MODEL
/---------------------------------------------------------------------
TURBULENCE MODEL                                  : 3 / 1:constant; 3:K-EPSILON; 4:SMAGORINSKI; 5:MIXING LENGTH
VELOCITY DIFFUSIVITY                              : 1.E-6
TURBULENCE REGIME FOR SOLID BOUNDARIES            : 2
/-----------------------------------------------------/
/  FRICTION PARAMETERS                                /
/-----------------------------------------------------/
LAW OF BOTTOM FRICTION                             : 4 /2:CHEZY; 4:MANNING; 5:NIKURADSE
FRICTION COEFFICIENT                              : 0.023
/---------------------------------------------------------------------
```

Added keywords are highlighted

As stated, an additional steering file called “**Gaia_Magat_2D.cas**” must be also present and is described herein as:

```
/ TITLE: Gaia_Magat_2D.cas
/---------------------------------------------------------------------/
GEOMETRY FILE                                          = Bottom_50m.slf
BOUNDARY CONDITIONS FILE                               = BOTTOM_BC.cli
RESULTS FILE                                           = gai_Magat.slf
/---------------------------------------------------------------------/
/ OUTPUTS                                     /
/---------------------------------------------------------------------/
 VARIABLES FOR GRAPHIC PRINTOUTS                       : 'U,V,H,S,R,B,E,TOB,KS' /evolution du fond
 MASS-BALANCE                                          : YES
/---------------------------------------------------------------------/
/  PHYSICAL OPTIONS
/---------------------------------------------------------------------/
 CLASSES TYPE OF SEDIMENT                              : CO
 CLASSES SEDIMENT DENSITY                               : 2650.0
 CLASSES SEDIMENT DIAMETERS                            = 0.000008
 CLASSES SETTLING VELOCITIES                            : -9
 NUMBER OF LAYERS FOR INITIAL STRATIFICATION            : 1
 LAYERS INITIAL THICKNESS                              : 0.D0
 LAYERS MASS TRANSFER                                  : 0.D0
 LAYERS PARTHENIADES CONSTANT                          : 5.D-3
 LAYERS MUD CONCENTRATION                               : 900.D0
 LAYERS CRITICAL EROSION SHEAR STRESS OF THE MUD       : 0.05
 CLASSES CRITICAL SHEAR STRESS FOR MUD DEPOSITION       : 1000
 SKIN FRICTION CORRECTION                              : 0
/---------------------------------------------------------------------/
/ NUMERICAL OPTIONS
/---------------------------------------------------------------------/
 ADVECTION-DIFFUSION SCHEME WITH SETTLING VELOCITY      : 1 / SET_DIF scheme
 SOLVER FOR DIFFUSION OF SUSPENSION                    = 1
```

3.3.4. Sample Steering File for Modelling a Sediment Transport Scenario by Coupling TELEMAC with the GAIA Module (Adjusted/Corrected)

Adjusted parameters are shown in bold letters

```
/-----------------------------------------------------------------------
/                    FILE DEFINITION
/-----------------------------------------------------------------------
BOUNDARY CONDITIONS FILE                          : 'BOTTOM_BC.cli'
GEOMETRY FILE                                     : 'Bottom_50m.slf'
RESULTS FILE                                      : 'Magat_V2.slf'
/-----------------------------------------------------------------------
/                    GENERAL OPTIONS
/-----------------------------------------------------------------------
TITLE                                             : 'Magat'
/
VARIABLES FOR GRAPHIC PRINTOUTS                   : 'U,V,F,H,B,S,Q,W,L,T*'
TIME STEP                                         : 25 /it should be determined by Courant Number
NUMBER OF TIME STEPS                              : 3456
GRAPHIC PRINTOUT PERIOD                           : 10
LISTING PRINTOUT PERIOD                           : 10
TIDAL FLATS                                       : YES
OPTION FOR THE TREATMENT OF TIDAL FLATS           : 1
/-----------------------------------------------------------------------
                         COUPLING
/-----------------------------------------------------------------------
COUPLING WITH                                     : 'GAIA'
GAIA STEERING FILE                                : 'Gaia_Magat_2D.cas' /steering file name of gaia
/-----------------------------------------------------------------------
/                    BOUNDARY CONDITIONS
/-----------------------------------------------------------------------
PRESCRIBED ELEVATIONS                             :191.65; 0.0 /downstream boundary (water level)
PRESCRIBED FLOWRATES                              :0.0; 4847.74 /upstream boundary (inflow discharge)
PRESCRIBED TRACERS VALUES                         :0.0; 19.07 /upstream boundary (inflow sediment)
/-----------------------------------------------------------------------
/                    INITIAL CONDITIONS
/-----------------------------------------------------------------------
INITIAL CONDITIONS                                :'CONSTANT ELEVATION'
INITIAL ELEVATION                                 : 191.65 /initial water level
/
/-----------------------------------------------------------------------
/                    NUMERICAL OPTIONS (not necessary to modify for beginner)
/-----------------------------------------------------------------------
MASS-BALANCE                                      : YES
SOLVER                                            : 1
SOLVER OPTION                                     : 2
SOLVER ACCURACY                                   : 1.E-4
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER           : 1000
PRECONDITIONING                                   : 2
TYPE OF ADVECTION                                 : 14;14
SUPG OPTION                                       : 0;0
DISCRETIZATIONS IN SPACE                          : 11; 11
MATRIX STORAGE                                    : 3
IMPLICITATION FOR DEPTH                           : 1.0
IMPLICITATION FOR VELOCITY                        : 1.0
VELOCITY PROFILES                                 : 1;1
MASS-LUMPING ON H                                 : 1.0
MASS-LUMPING ON VELOCITY                          : 1.0
MASS-LUMPING ON TRACERS                           : 1.0
TREATMENT OF THE LINEAR SYSTEM                    : 2
FREE SURFACE GRADIENT COMPATIBILITY               : 0.5
CONTINUITY CORRECTION                             : YES
TREATMENT OF NEGATIVE DEPTHS                      : 2
OPTION FOR LIQUID BOUNDARIES                      : 1;1
OPTION FOR THE DIFFUSION OF VELOCITIES            : 1
/-----------------------------------------------------------------------
/                    TURBULENCE MODEL
/-----------------------------------------------------------------------
/
TURBULENCE MODEL                                  : 3 / 1:constant; 3:K-EPSILON; 4:SMAGORINSKI; 5:MIXING LENGTH
VELOCITY DIFFUSIVITY                              : 1.E-6
TURBULENCE REGIME FOR SOLID BOUNDARIES            : 2
/
/-------------------------------------------------------/
/  FRICTION PARAMETERS                                  /
/-------------------------------------------------------/
LAW OF BOTTOM FRICTION                            : 4 /2:CHEZY; 4:MANNING; 5:NIKURADSE
FRICTION COEFFICIENT                              : 0.023
/-----------------------------------------------------------------------
```

3.3.5. Sample Steering File for Modelling a Sediment Transport Scenario using TELEMAC coupled with GAIA Module and imposing a Hydrograph File (Initial/Uncorrected)

```
/----------------------------------------------------------------------
/                FILE DEFINITION
/----------------------------------------------------------------------
BOUNDARY CONDITIONS FILE                              : 'BOTTOM_BC.cli'
GEOMETRY FILE                                         : 'Bottom_50m.slf'
RESULTS FILE                                          : 'Magat_V2.slf'
LIQUID BOUNDARIES FILE                                = Magat_Boundary.txt /hydrograph B.C. file
/----------------------------------------------------------------------
/                 GENERAL OPTIONS
/----------------------------------------------------------------------
TITLE                                                 : 'Magat'
/
VARIABLES FOR GRAPHIC PRINTOUTS                       : 'U,V,F,H,B,S,Q,W,L'
TIME STEP                                             : 60 /it should be determined by Courant Number
NUMBER OF TIME STEPS                                  : 600
GRAPHIC PRINTOUT PERIOD                               : 1
LISTING PRINTOUT PERIOD                               : 10
TIDAL FLATS                                           : YES
OPTION FOR THE TREATMENT OF TIDAL FLATS               : 1
/----------------------------------------------------------------------
/                   COUPLING
/----------------------------------------------------------------------
COUPLING WITH                                         : 'GAIA'
GAIA STEERING FILE                                    : 'Gaia_Magat_2D.cas' /steering file name of gaia
/----------------------------------------------------------------------
/               BOUNDARY CONDITIONS
/----------------------------------------------------------------------
PRESCRIBED ELEVATIONS                                 :188.28; 0.0 /downstream boundary (water level)
PRESCRIBED FLOWRATES                                  :0.0; 183.78 /upstream boundary (inflow discharge)
PRESCRIBED TRACERS VALUES                             :0.0; 1.0 /upstream boundary (inflow sediment)
/----------------------------------------------------------------------
/                INITIAL CONDITIONS
/----------------------------------------------------------------------
INITIAL CONDITIONS                                    :'CONSTANT ELEVATION'
INITIAL ELEVATION                                     : 188.28 /initial water level
/
/----------------------------------------------------------------------
/                NUMERICAL OPTIONS (not necessary to modify for beginner)
/----------------------------------------------------------------------
MASS-BALANCE                                          : YES
SOLVER                                                : 1
SOLVER OPTION                                         : 2
SOLVER ACCURACY                                       : 1.E-4
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER               : 1000
PRECONDITIONING                                       : 2
TYPE OF ADVECTION                                     : 14;14
SUPG OPTION                                           : 0;0
DISCRETIZATIONS IN SPACE                              : 11; 11
MATRIX STORAGE                                        : 3
IMPLICITATION FOR DEPTH                               : 1.0
IMPLICITATION FOR VELOCITY                            : 1.0
VELOCITY PROFILES                                     : 1;1
MASS-LUMPING ON H                                     : 1.0
MASS-LUMPING ON VELOCITY                              : 1.0
MASS-LUMPING ON TRACERS                               : 1.0
TREATMENT OF THE LINEAR SYSTEM                        : 2
FREE SURFACE GRADIENT COMPATIBILITY                   : 0.5
CONTINUITY CORRECTION                                 : YES
TREATMENT OF NEGATIVE DEPTHS                          : 2
OPTION FOR LIQUID BOUNDARIES                          : 1;1
OPTION FOR THE DIFFUSION OF VELOCITIES                : 1
/----------------------------------------------------------------------
/                TURBULENCE MODEL
/----------------------------------------------------------------------
TURBULENCE MODEL                                      : 3 / 1:constant; 3:K-EPSILON; 4:SMAGORINSKI; 5:MIXING LENGTH
VELOCITY DIFFUSIVITY                                  : 1.E-6
TURBULENCE REGIME FOR SOLID BOUNDARIES                : 2
/-------------------------------------------------------/
/   FRICTION PARAMETERS                                /
/-------------------------------------------------------/
LAW OF BOTTOM FRICTION                                : 4 /2:CHEZY; 4:MANNING; 5:NIKURADSE
FRICTION COEFFICIENT                                  : 0.023
```

3.3.6. Sample Steering File for Modelling a Sediment Transport Scenario using TELEMAC coupled with GAIA Module and imposing a Boundary File (Corrected/Adjusted)

```
/---------------------------------------------------------------------
/                FILE DEFINITION
/---------------------------------------------------------------------
BOUNDARY CONDITIONS FILE                          : 'BOTTOM_BC.cli'
GEOMETRY FILE                                     : 'Bottom_50m.slf'
RESULTS FILE                                      : 'Magat_V2.slf'
LIQUID BOUNDARIES FILE                            = Magat_Boundary.txt /hydrograph B.C. file
/---------------------------------------------------------------------
/                 GENERAL OPTIONS
/---------------------------------------------------------------------
TITLE                                             : 'Magat'
VARIABLES FOR GRAPHIC PRINTOUTS                   : 'U,V,F,H,B,S,Q,W,L,T*'
TIME STEP                                         : 25 /it should be determined by Courant Number
NUMBER OF TIME STEPS                              : 20592
GRAPHIC PRINTOUT PERIOD                           : 10
LISTING PRINTOUT PERIOD                           : 10
TIDAL FLATS                                       : YES
OPTION FOR THE TREATMENT OF TIDAL FLATS           : 1
/---------------------------------------------------------------------
/                                  COUPLING
/---------------------------------------------------------------------
COUPLING WITH                                     : 'GAIA'
GAIA STEERING FILE                                : 'Gaia_Magat_2D.cas' /steering file name of gaia
/---------------------------------------------------------------------
/               BOUNDARY CONDITIONS
/---------------------------------------------------------------------
PRESCRIBED ELEVATIONS                             :189.02; 0.0 /downstream boundary (water level)
PRESCRIBED FLOWRATES                              :0.0; 301 /upstream boundary (inflow discharge)
PRESCRIBED TRACERS VALUES                         :0.0; 1.26 /upstream boundary (inflow sediment)
/---------------------------------------------------------------------
/                INITIAL CONDITIONS
/---------------------------------------------------------------------
INITIAL CONDITIONS                                :'CONSTANT ELEVATION'
INITIAL ELEVATION                                 : 189.02 /initial water level
/---------------------------------------------------------------------
/                NUMERICAL OPTIONS (not necessary to modify for beginner)
/---------------------------------------------------------------------
MASS-BALANCE                                      : YES
SOLVER                                            : 1
SOLVER OPTION                                     : 2
SOLVER ACCURACY                                   : 1.E-4
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER           : 1000
PRECONDITIONING                                   : 2
TYPE OF ADVECTION                                 : 14;14
SUPG OPTION                                       : 0;0
DISCRETIZATIONS IN SPACE                          : 11; 11
MATRIX STORAGE                                    : 3
IMPLICITATION FOR DEPTH                           : 1.0
IMPLICITATION FOR VELOCITY                        : 1.0
VELOCITY PROFILES                                 : 1;1
MASS-LUMPING ON H                                 : 1.0
MASS-LUMPING ON VELOCITY                          : 1.0
MASS-LUMPING ON TRACERS                           : 1.0
TREATMENT OF THE LINEAR SYSTEM                    : 2
FREE SURFACE GRADIENT COMPATIBILITY               : 0.5
CONTINUITY CORRECTION                             : YES
TREATMENT OF NEGATIVE DEPTHS                      : 2
OPTION FOR LIQUID BOUNDARIES                      : 1;1
OPTION FOR THE DIFFUSION OF VELOCITIES            : 1
/---------------------------------------------------------------------
/                TURBULENCE MODEL
/---------------------------------------------------------------------
TURBULENCE MODEL                                  : 3 / 1:constant; 3:K-EPSILON; 4:SMAGORINSKI; 5:MIXING LENGTH
VELOCITY DIFFUSIVITY                              : 1.E-6
TURBULENCE REGIME FOR SOLID BOUNDARIES            : 2
/----------------------------------------------------/
/  FRICTION PARAMETERS                               /
/----------------------------------------------------/
LAW OF BOTTOM FRICTION                            : 4 /2:CHEZY; 4:MANNING; 5:NIKURADSE
FRICTION COEFFICIENT                              : 0.023
/---------------------------------------------------------------------
```

The Hydrograph (Boundary Condition File)

Note: Displaying the first 60 hours only

Open boundary file of Magat_upstream_2020
Simulation duration is from 11/9/2020 00:00 to 11/14/2020
#

T	SL(1)	Q(2)	TR(1,1)	TR(2,1)
s	m	m3/s	kg/m3	kg/m3
0	189.02	301	0	1.26209784
3600	189.04	408	0	1.700375418
7200	189.07	453	0	1.88397049
10800	189.11	535	0	2.2176069
14400	189.16	675	0	2.784938155
18000	189.34	1751	0	7.087911415
21600	189.53	1823	0	7.373416954
25200	189.74	2009	0	8.10995036
28800	189.95	2108	0	8.501411778
32400	190.17	2184	0	8.80167725
36000	190.35	1896	0	7.662657678
39600	190.51	1784	0	7.218796697
43200	190.65	1517	0	6.158340916
46800	190.8	1629	0	6.603596377
50400	190.92	1418	0	5.764220215
54000	191.03	1538	0	6.241874992
57600	191.1	1477	0	5.999164123
61200	191.12	1251	0	5.098119621
64800	191.16	1417	0	5.760236452
68400	191.18	1291	0	5.257818321
72000	191.2	1218	0	4.966291423
75600	191.2	1038	0	4.245914694
79200	191.2	1048	0	4.285997515
82800	191.2	1067	0	4.362133864
86400	191.2	1094	0	4.470281102
90000	191.2	1111	0	4.538346341
93600	191.18	915	0	3.75223932
97200	191.16	939	0	3.848665248
100800	191.13	868	0	3.563257342
104400	191.09	728	0	2.999070049
108000	191.06	870	0	3.571303221
111600	191.02	751	0	3.091896846
115200	190.98	727	0	2.995032784
118800	190.95	747	0	3.075757189
122400	190.93	814	0	3.345875484
126000	190.89	701	0	2.890024441
129600	190.85	710	0	2.926382154
133200	190.79	588	0	2.43269486
136800	190.74	624	0	2.578568992
140400	190.7	716	0	2.950615502
144000	190.66	745	0	3.067686713
147600	190.6	557	0	2.30693814
151200	190.56	732	0	3.015218002
154800	190.51	672	0	2.772807663
158400	190.45	571	0	2.363748379
162000	190.41	749	0	3.083827233
165600	190.38	800	0	3.289470873
169200	190.34	785	0	3.229015439
172800	190.3	742.71	0	3.058445485
176400	190.27	832.55	0	3.420581768
180000	190.24	861.78	0	3.538232282
183600	190.2	738.72	0	3.042342596
187200	190.17	849.3	0	3.488010307
190800	190.14	878.53	0	3.605614756
194400	190.1	813.73	0	3.344787865
198000	190.07	742.06	0	3.055822326
201600	190.04	833.29	0	3.423561269
205200	190	735.73	0	3.030274378
208800	189.95	704.48	0	2.904083857
212400	189.89	618.61	0	2.556739349
216000	189.86	742.64	0	3.058162993

3.3.7. Creating a Steering File Using Notepad (or any basic text editing software available)

1. Copy the text files described in Section 3.3.2. [*Sample Steering File for a Purely Hydrodynamic Scenario in Magat Reservoir (Adjusted)*]
2. In the Windows start menu, type “**Notepad**”.

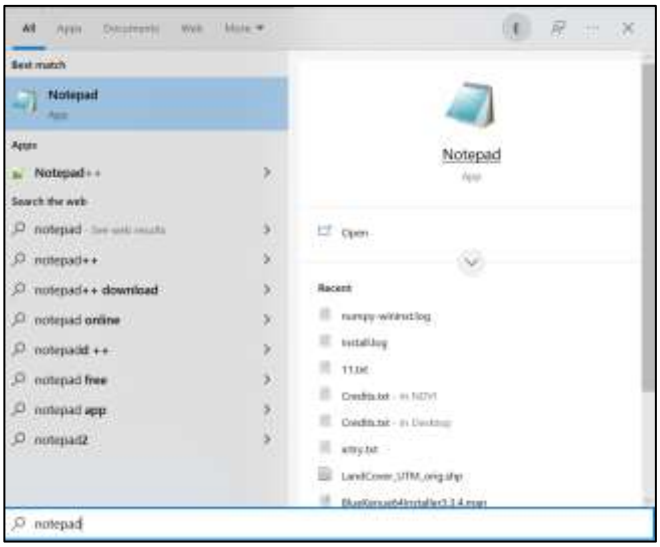
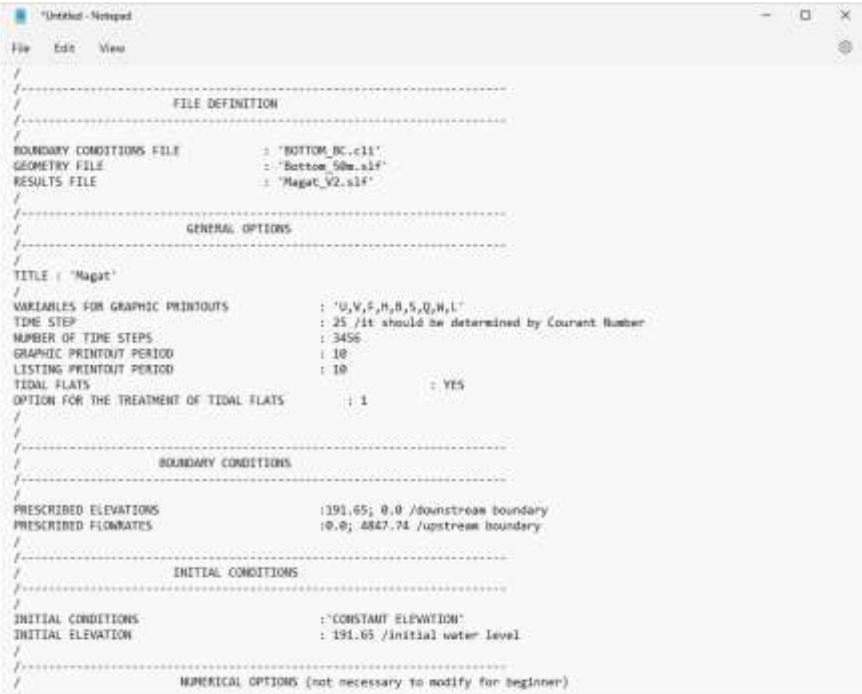


Figure 81. Using Notepad in the Windows Files and Programs Search Option

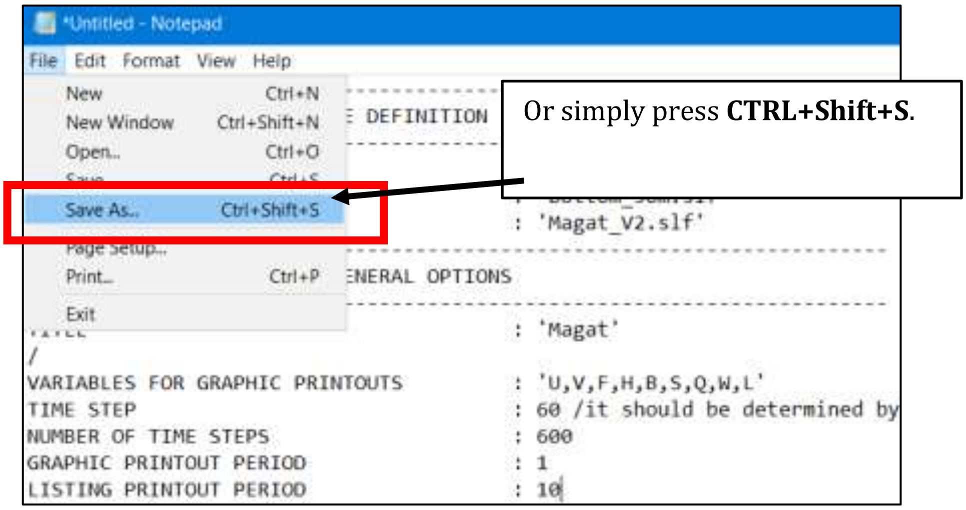
3. Paste (or manually type if preferred) the copied text files.



```
/
/---------------------------------------------------------------------
/                      FILE DEFINITION
/---------------------------------------------------------------------
/
BOUNDARY CONDITIONS FILE            : 'BOTTOM_BC.cli'
GEOMETRY FILE                       : 'Bottom_50m.slf'
RESULTS FILE                        : 'Magat_V2.slf'
/
/---------------------------------------------------------------------
/                       GENERAL OPTIONS
/---------------------------------------------------------------------
/
TITLE : 'Magat'
/
VARIABLES FOR GRAPHIC PRINTOUTS              : 'U,V,F,H,B,S,Q,W,L'
TIME STEP                                    : 25 /it should be determined by Courant Number
NUMBER OF TIME STEPS                         : 3456
GRAPHIC PRINTOUT PERIOD                      : 10
LISTING PRINTOUT PERIOD                      : 10
TIDAL FLATS                                           : YES
OPTION FOR THE TREATMENT OF TIDAL FLATS         : 1
/
/
/---------------------------------------------------------------------
/                    BOUNDARY CONDITIONS
/---------------------------------------------------------------------
/
PRESCRIBED ELEVATIONS                        :191.65; 0.0 /downstream boundary
PRESCRIBED FLOWRATES                         :0.0; 4847.74 /upstream boundary
/
/---------------------------------------------------------------------
/                     INITIAL CONDITIONS
/---------------------------------------------------------------------
/
INITIAL CONDITIONS                           :'CONSTANT ELEVATION'
INITIAL ELEVATION                            : 191.65 /initial water level
/
/---------------------------------------------------------------------
/                      NUMERICAL OPTIONS (not necessary to modify for beginner)
```

Figure 82. A Sample Steering File for a Purely Hydrodynamic Scenario

4. Click **File** >> **Save as:**



5. Enter the appropriate file name (i.e., Magat_2D.cas) and do not forget to add the **.cas** extension, and then click "**Save**".

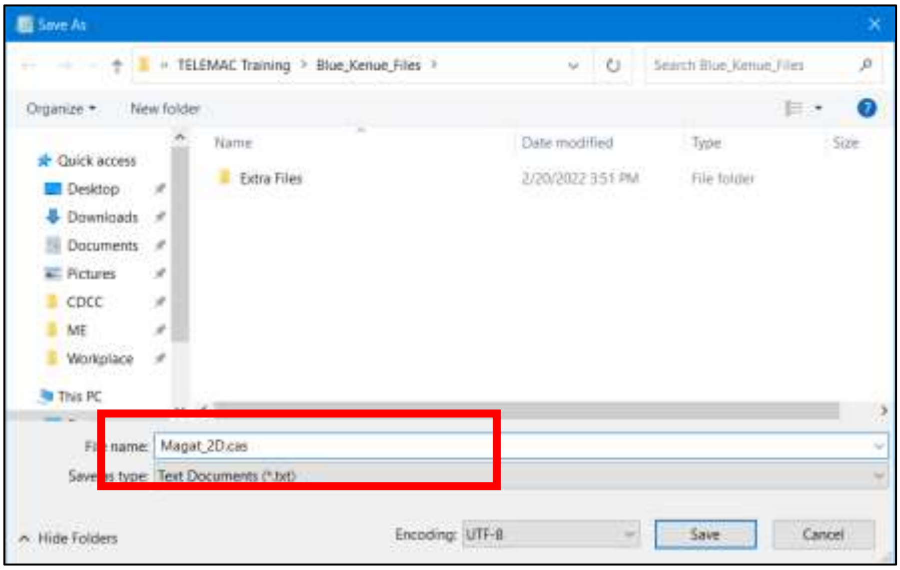


Figure 83. Adding .cas File Extension

4. Running the Model Setup

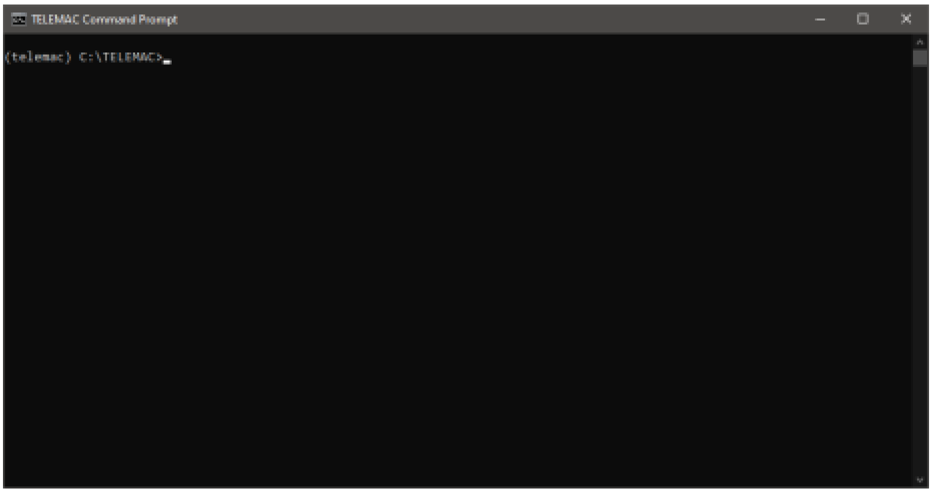
With the steering file (*.cas) generated from the last section, all the necessary files for the TELEMAC 2D simulation are now available. In summary, we have also generated the geometry file (*.slf) which can be found on page 34, and the boundary conditions (*.cli) files on page 57.

Make sure that all these files are now located in a single simulation folder. For the trainees' convenience, these files have been generated beforehand and are housed in the **Simulation_Files** folder.

4.1. Running a Hydrodynamic Simulation on Windows Environment

Step 1: Open TELEAMC Command Prompt.

a. Open the generated shortcut called "**TELEMAC Command Prompt**" located in the desktop.



Step 2: Change the directory in the **Simulation_Files** folder

a. Given that the files from the drive folder have been downloaded and extracted into your **Desktop** folder, type the following in the next cmd line:

C:\Users*username*\Desktop\TELEMAC Training Files\Simulation Files\Magat_Initial\Sce1_hydroFiles\Magat_Initial\Sce1_hydro

*Note: Insert your computer's user name at the *username*, and make sure to remove the asterisk signs afterward. Do not forget to press **Enter**.*

Step 3: Start the simulation.

a. Continuing from the previous step, type **dir** in the next line.

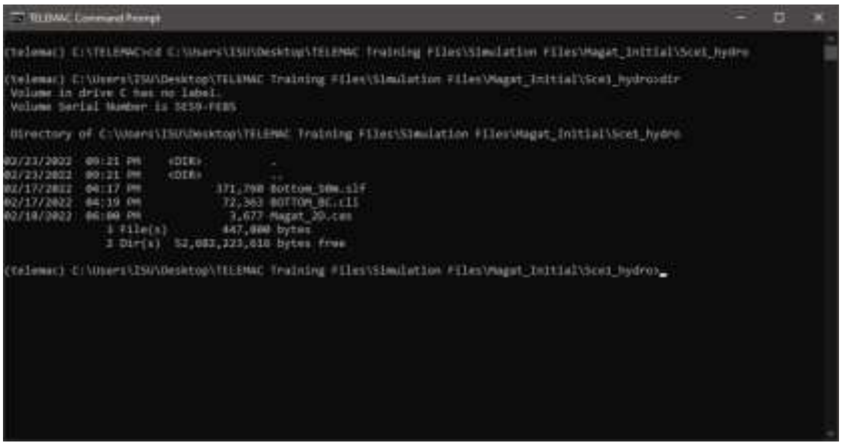


Figure 84. Typing dir in the Next Line

Figure 85. The Contents of the Current Directory

b. Now, type "**python C:\TELEMAC\V8P4\scripts\python3\telemac2d.py Magat_2D.cas**" and press <**Enter**>.

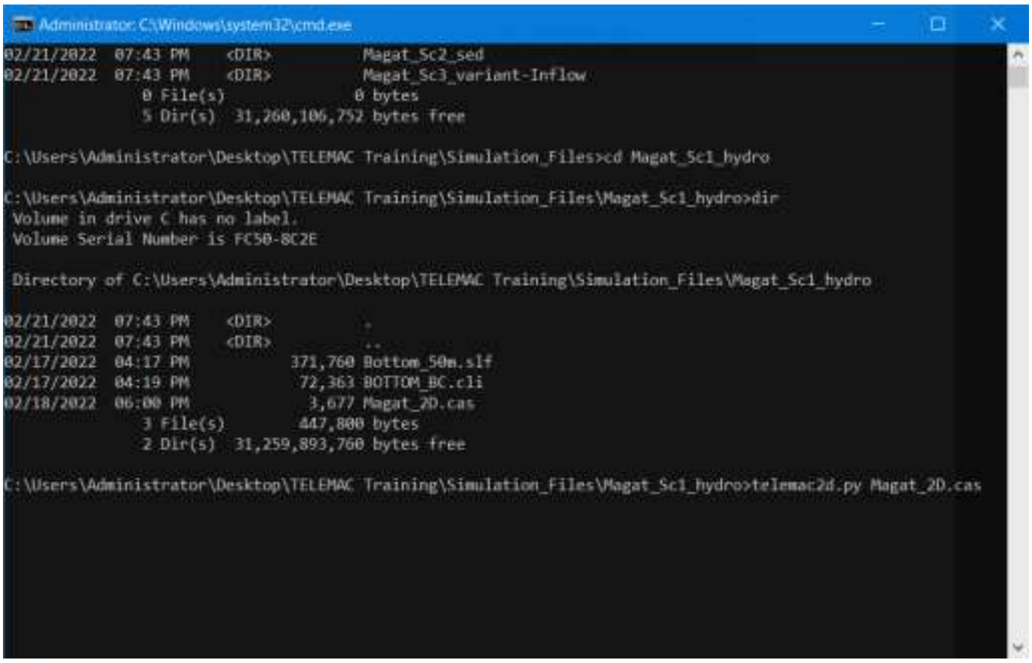


Figure 86. Starting the Simulation through Line of Code

c. Wait for the simulation to finish.

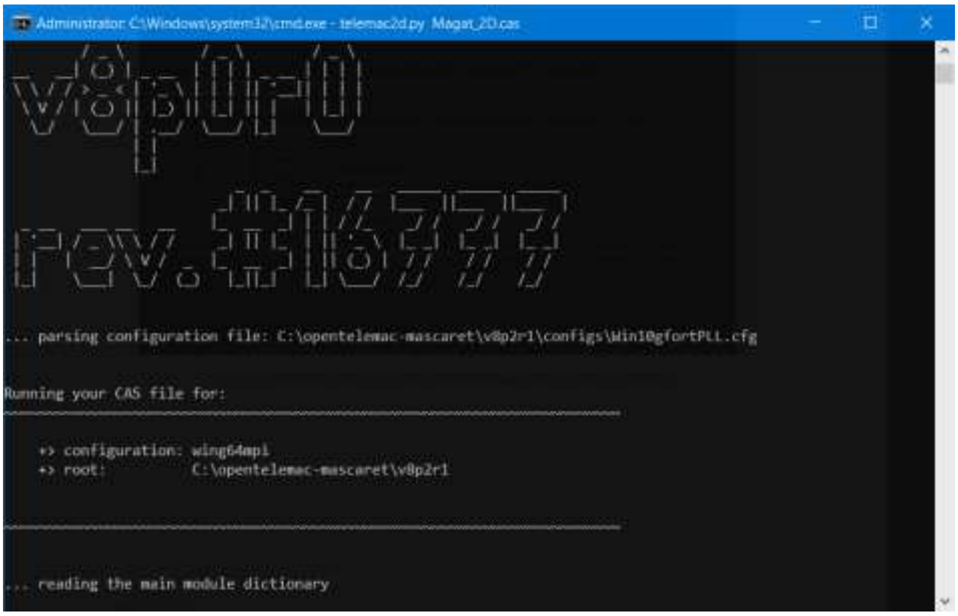


Figure 87. A Snapshot at the Start of the Simulation

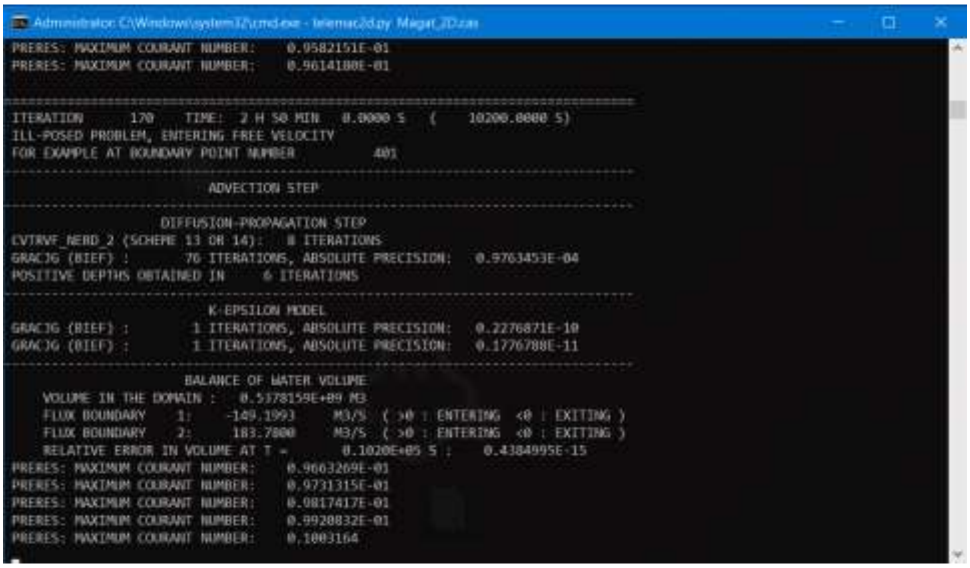


Figure 88. A Snapshot during the Simulation

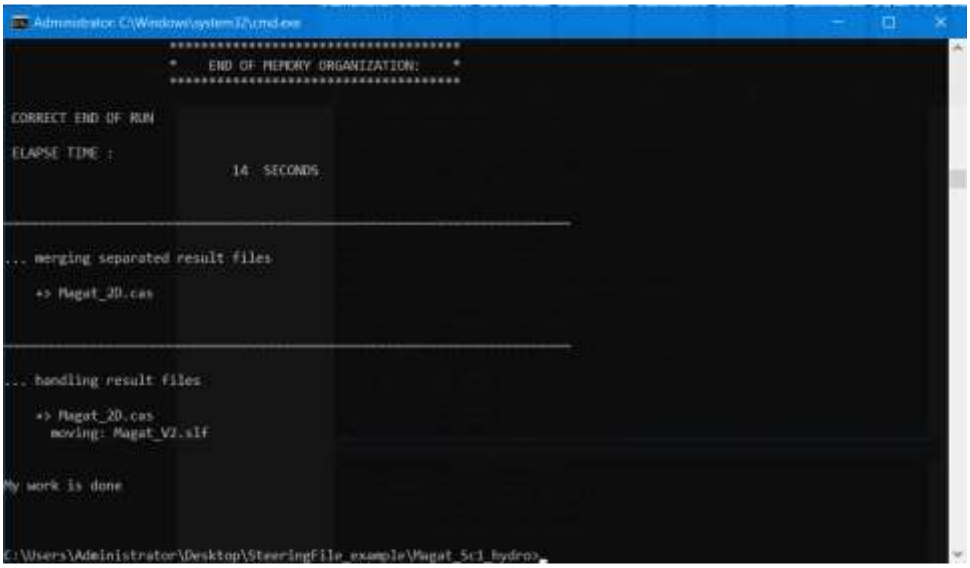


Figure 89. A Successful End of Run

Note:

The steps shown above portray the steps in running the uncorrected steering file. To run the corrected steering file for the scenarios, simply change the directory where the files are stored and follow the same steps shown above.

5. Post-Processing

5.1. Post-Processing Using Blue-Kenue

1. Continuing from a successful run, a new *.slf file called “**Magat_V2.slf**” should have been created inside our simulations file folder.

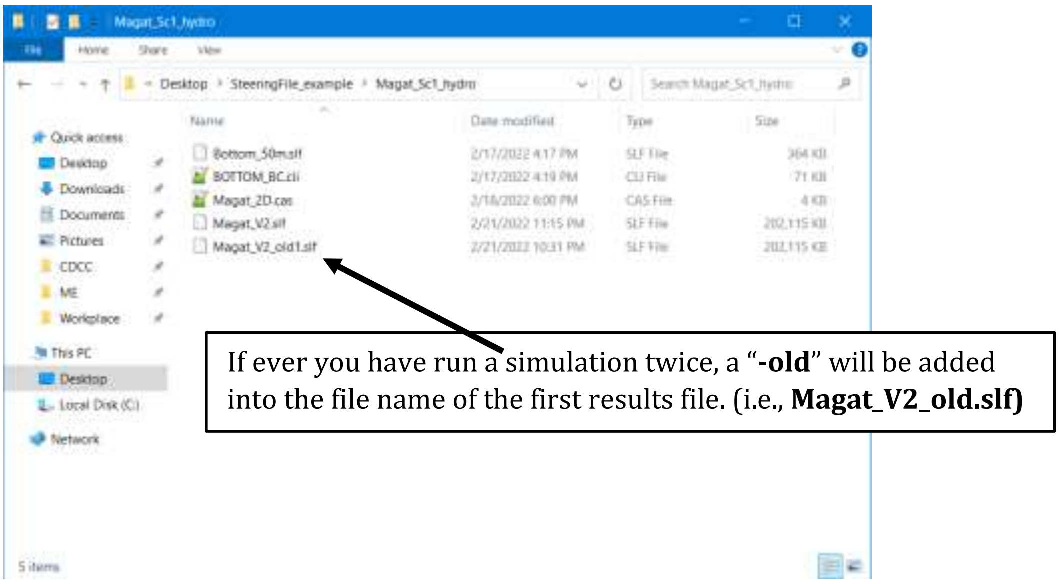


Figure 90. The New File Called Magat_V2.slf

2. Open the Blue Kenue application.

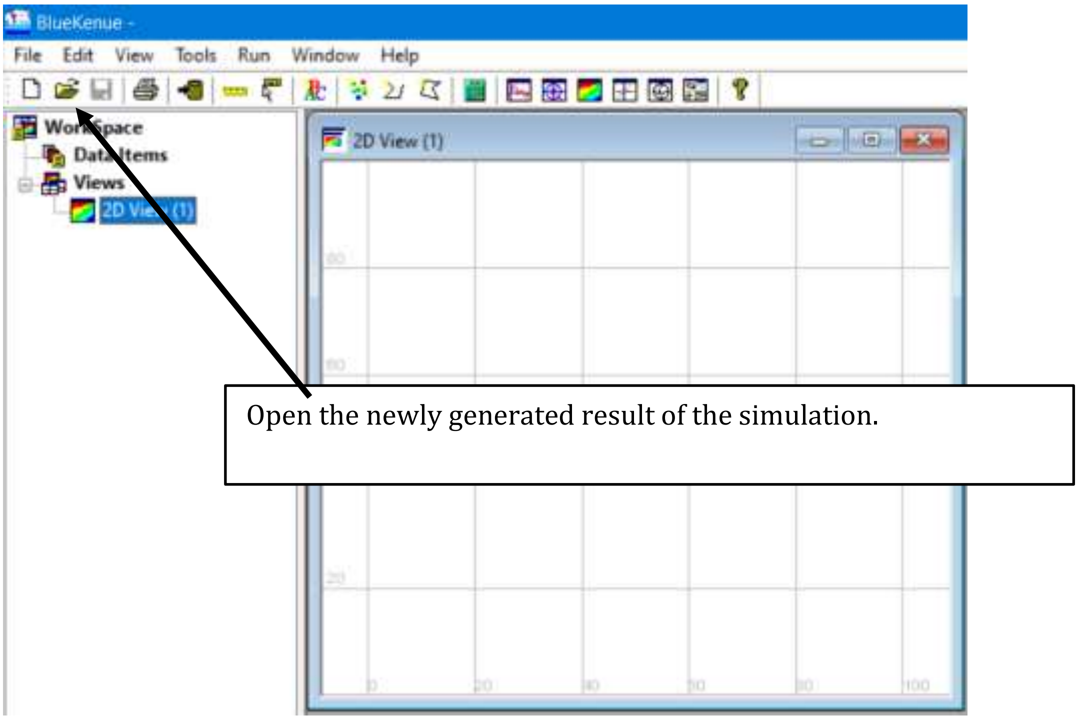


Figure 91. The Blue Kenue User Interface

3. Open the Magat_V2.slf file.

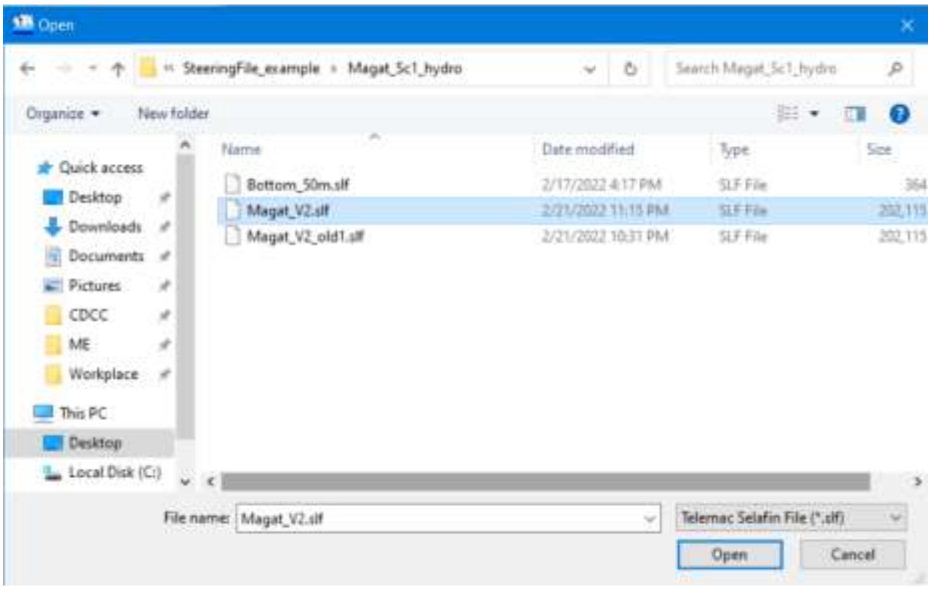


Figure 92. Opening Magat_V2.slf

4. Blue Kenue will read the Selafin file and after a few seconds, display a **Done** message window.

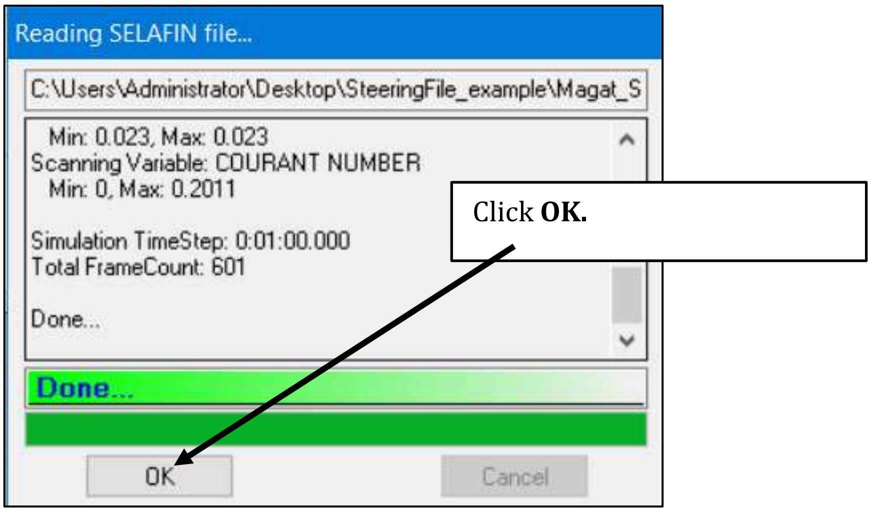


Figure 93. A successful Reading and Opening of the Results File

5. Results are now loaded and can be seen under the **Data Items** menu.

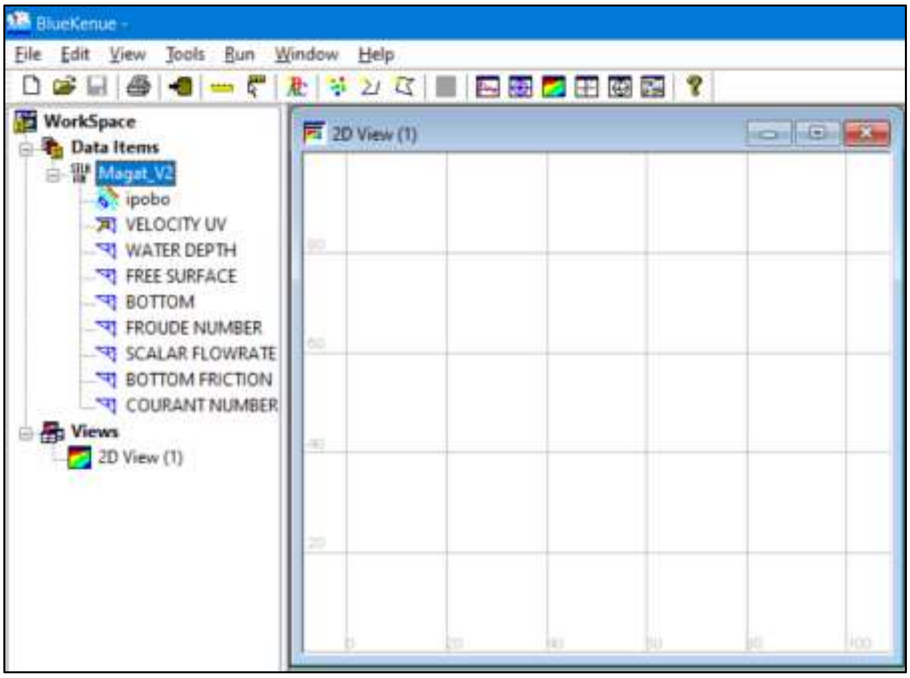


Figure 94 Results file content successfully loaded

5.1.1. Visualizing Velocity in Blue Kenue

1. Drag the **VELOCITY UV** item into the **2D View.**

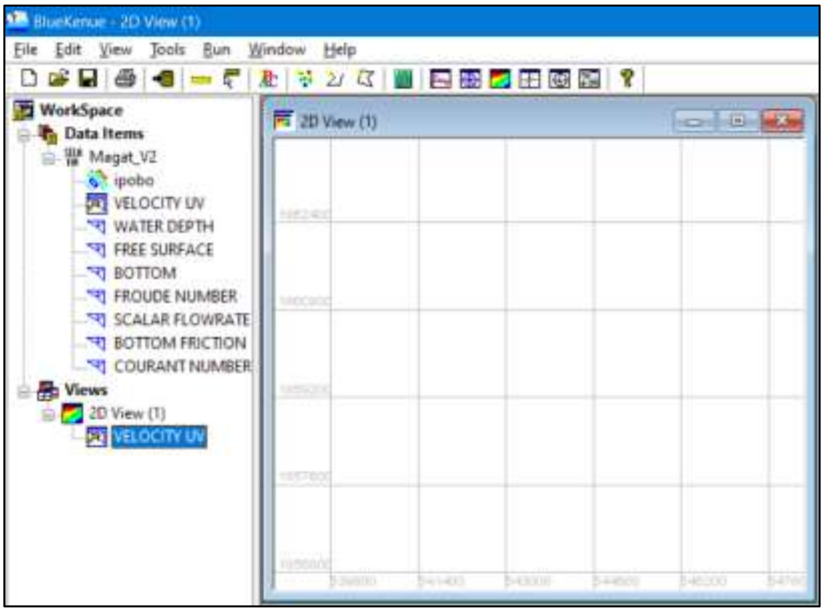


Figure 95. Velocity UV under 2D View

2. Notice that the 2D view window is still empty even after dragging the **VELOCITY UV item** into it.
3. To display the result, right-click the **VELOCITY UV** (the one under 2D View) and select **Animate.**

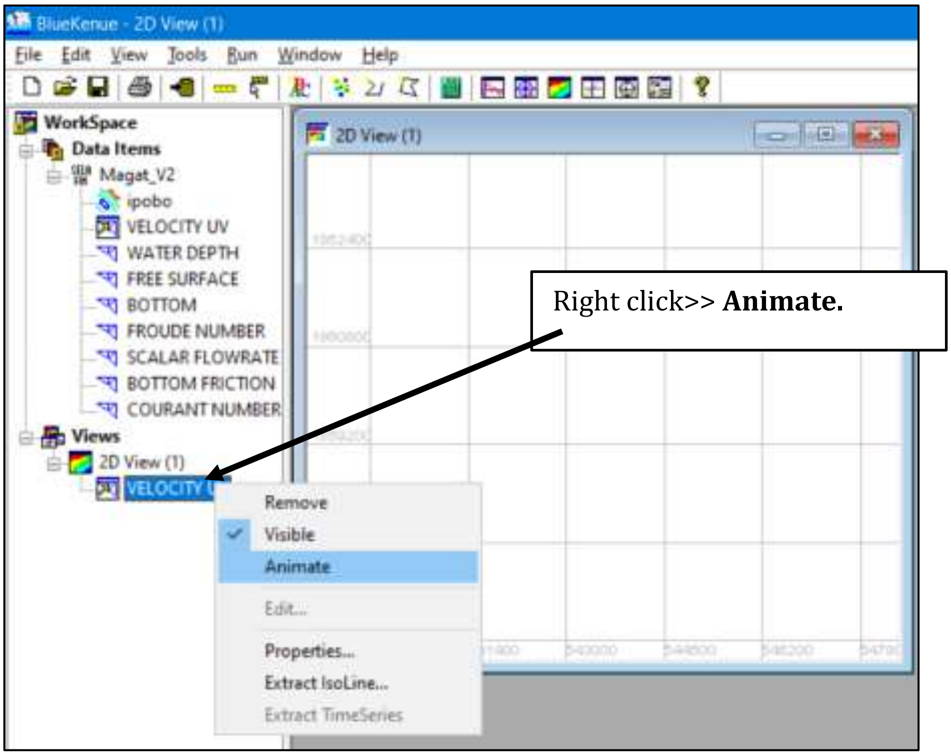


Figure 96. Right Clicking on the VELOCITY UV to Display the Result

4. Notice new **Playback options** are now added to the 2D View. Hit the "**play**" icon.

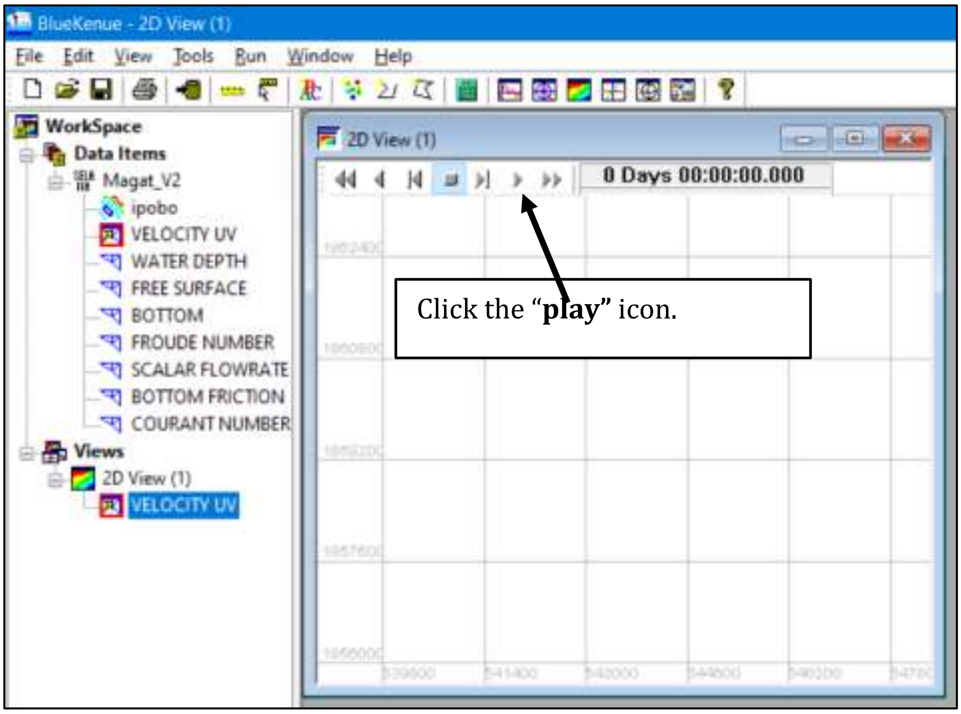


Figure 97. The 2D View with the Animate Playback Option

5. An animation showing the simulated velocity will be shown:

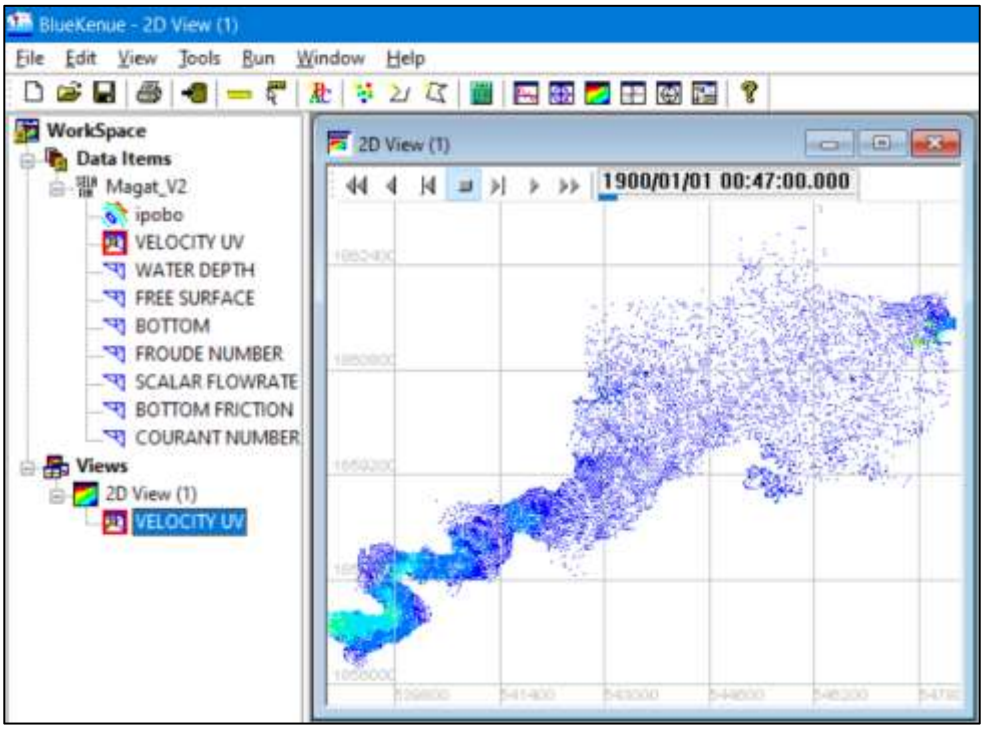


Figure 98. The Animated VELOCITY UV

6. You can modify the properties of the item to show another rendering style, show legend, etc.

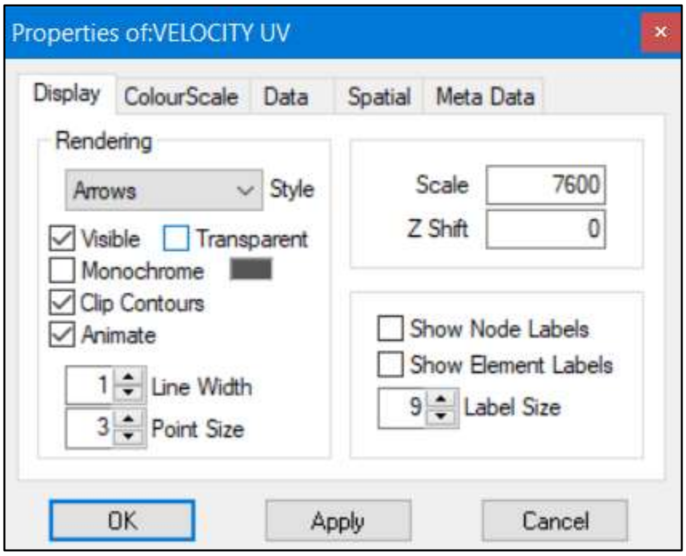


Figure 99. The Display Option Properties

7. Feel free to explore and tinker with the settings to suit your preference.
8. To show the legend, select the "**ColourScale**" tab and activate the "**Legends**" tick box.

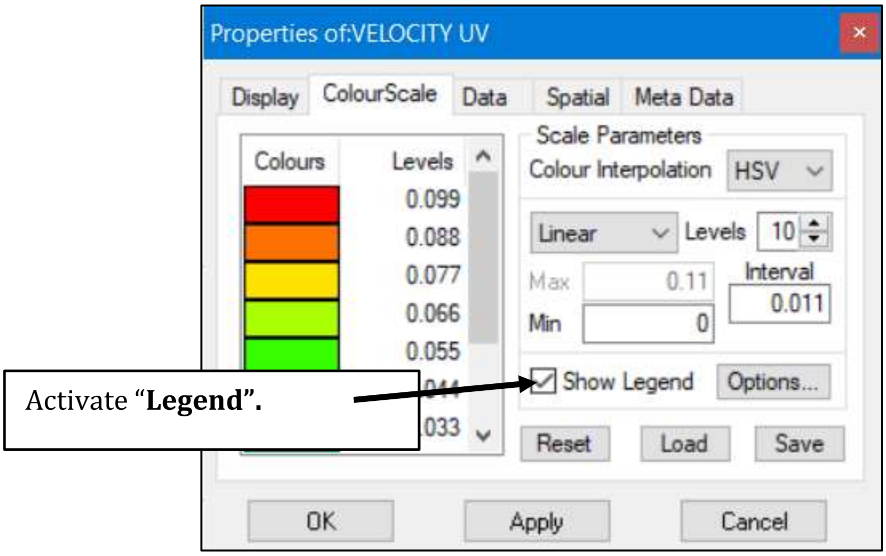


Figure 100. The Activation of the Legend Box

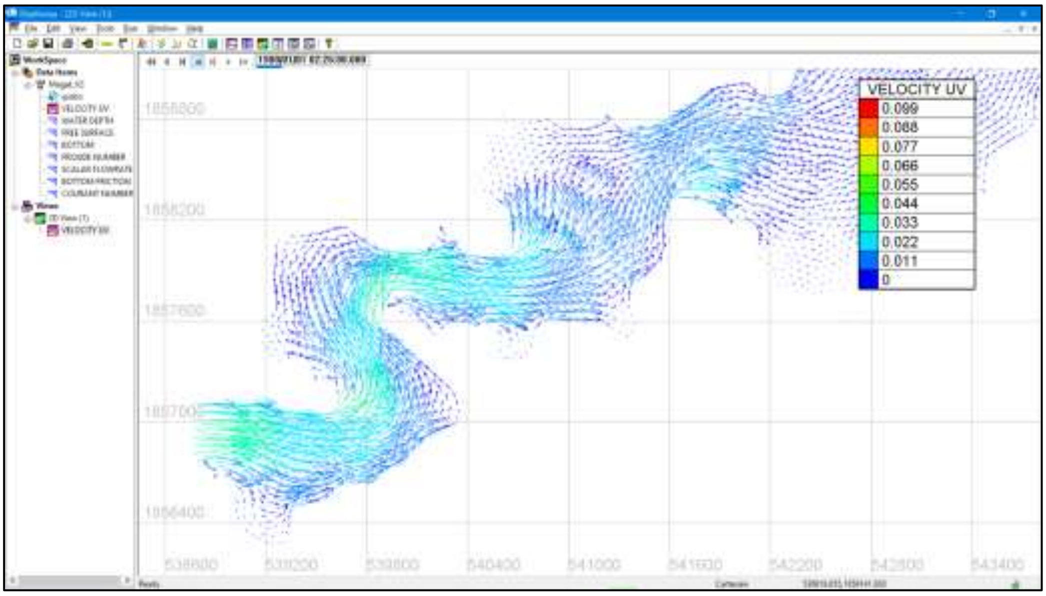


Figure 101. A Simple Visualization of the VELOCITY Profile Using Blue Kenue

Appendix I: Enabling .NET Framework 3.5

1. Type and search for "**Windows Features**" on your windows start menu.

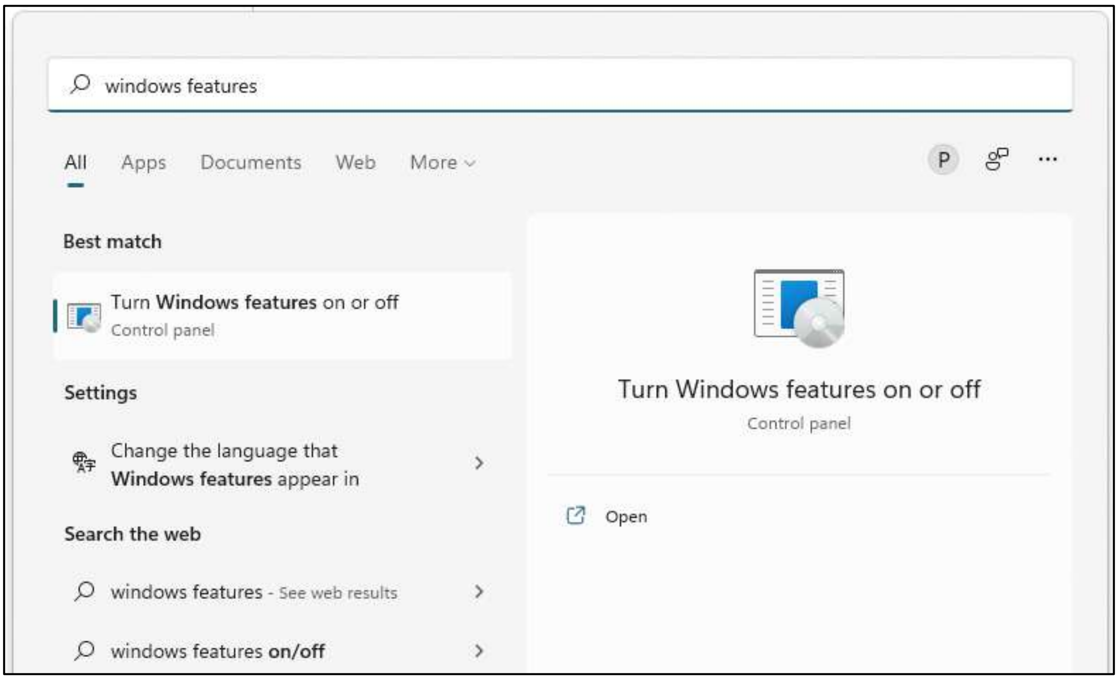


Figure 102. Searching for Windows Features

2. Select "**Turn Windows features on or off**".

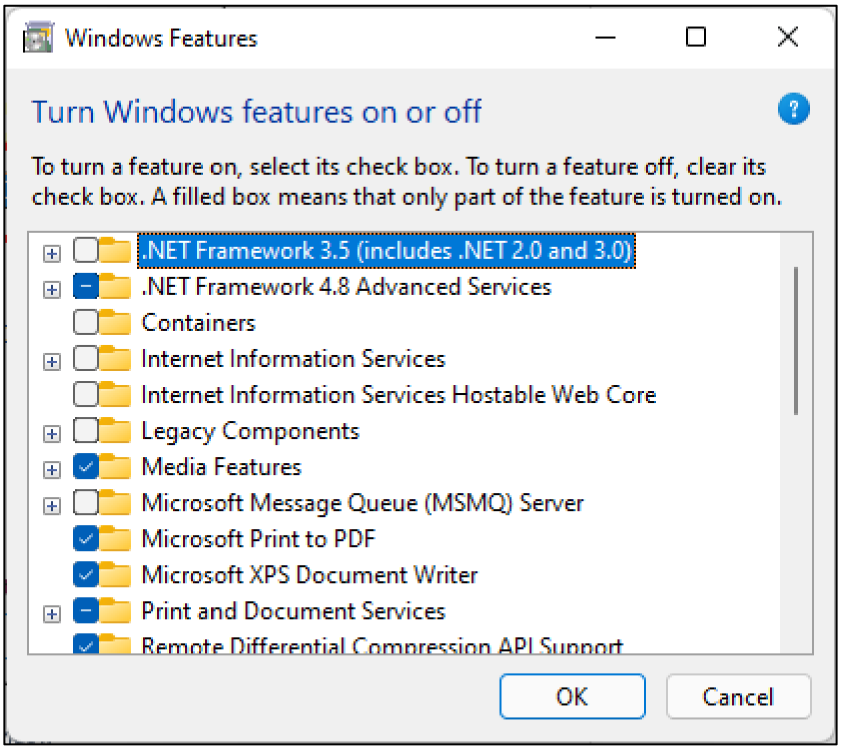


Figure 103. The Windows Features window

3. Make sure to select and enable the "**.NET Framework 3.5 (includes .NET 2.0 and 3.0)**".

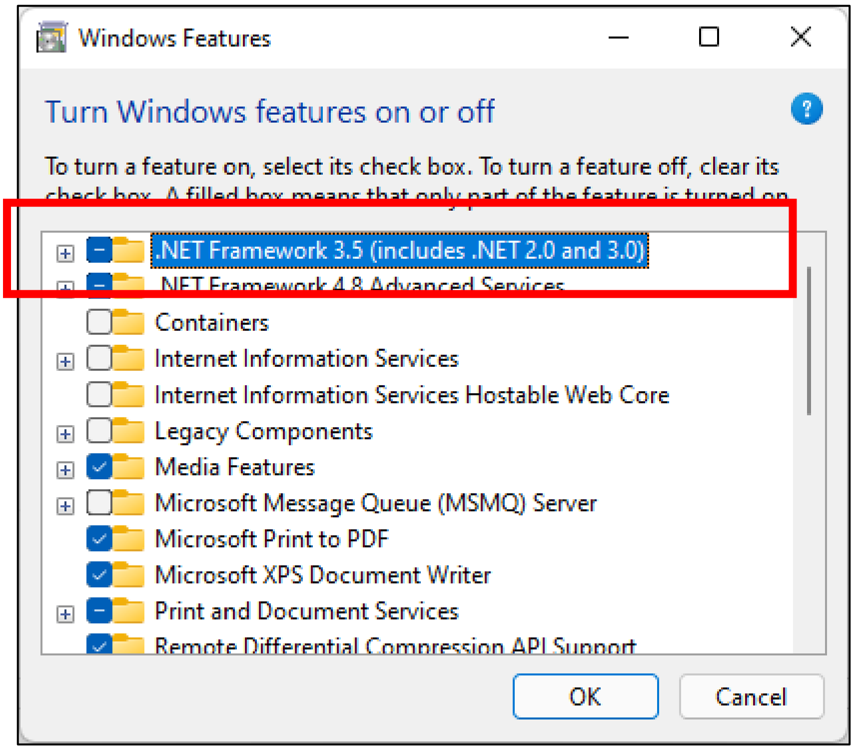


Figure 104. Enabling the .NET Framework 3.5

4. Click **"OK"** to finish. On some devices, your computer/s might have to search and download some components. This will be done automatically so the user/s don't have to worry about it.

Appendix II: Adding items into the System Variables Path

1. Once python and the libraries are installed, the next step is to make sure that the **path** to python3.10 and your newly created folder is recognized by your system. To do so, go to Windows start menu or simply press the windows start button and type "**variables**" and press "**Enter**".

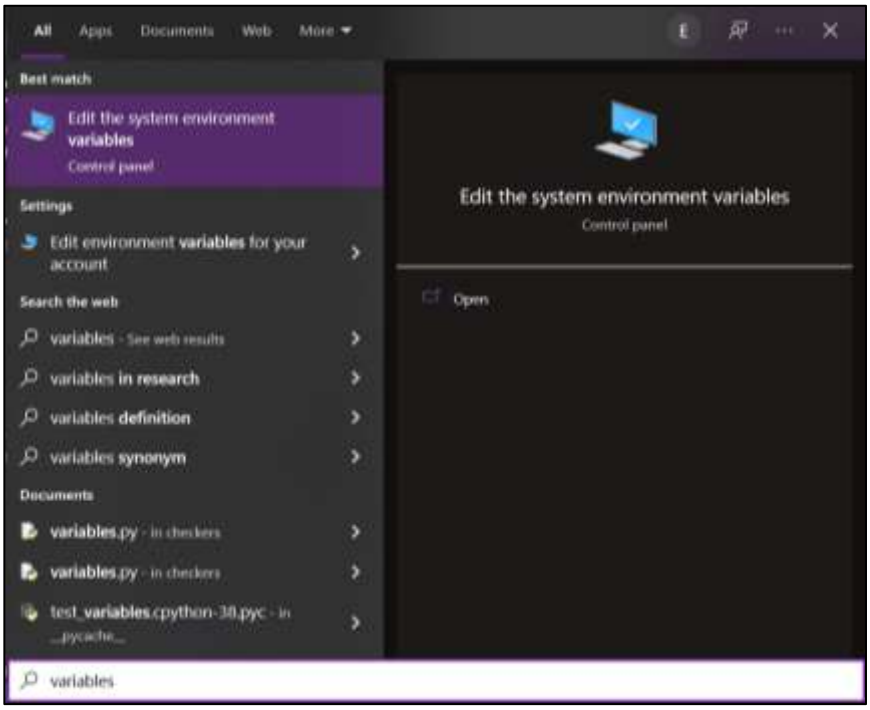


Figure 105 Editing the System Environment Variables

2. After pressing Enter, a system properties window will open then click on "**Environment variables**".

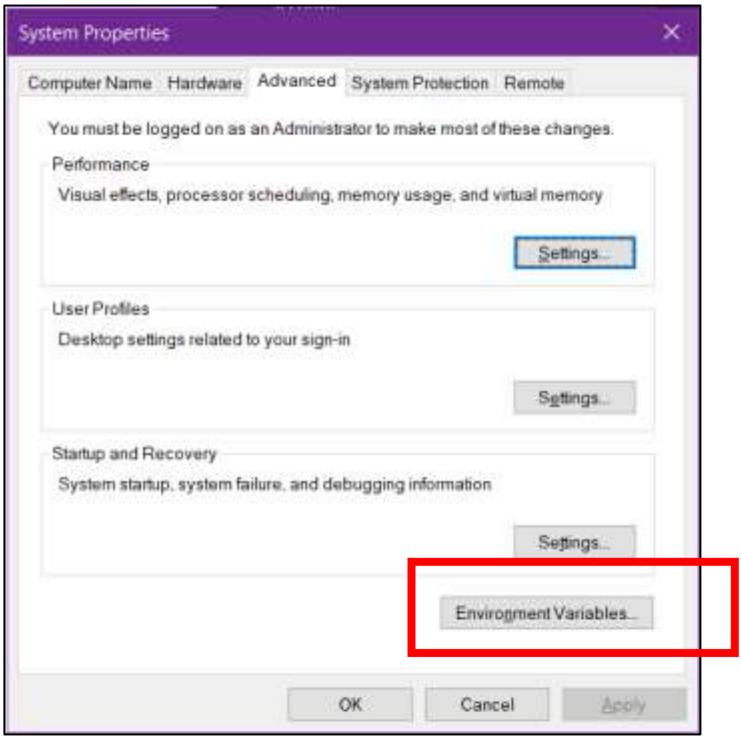


Figure 106 The System Properties Window

3. After that, another window for **Environment Variables** will pop up.

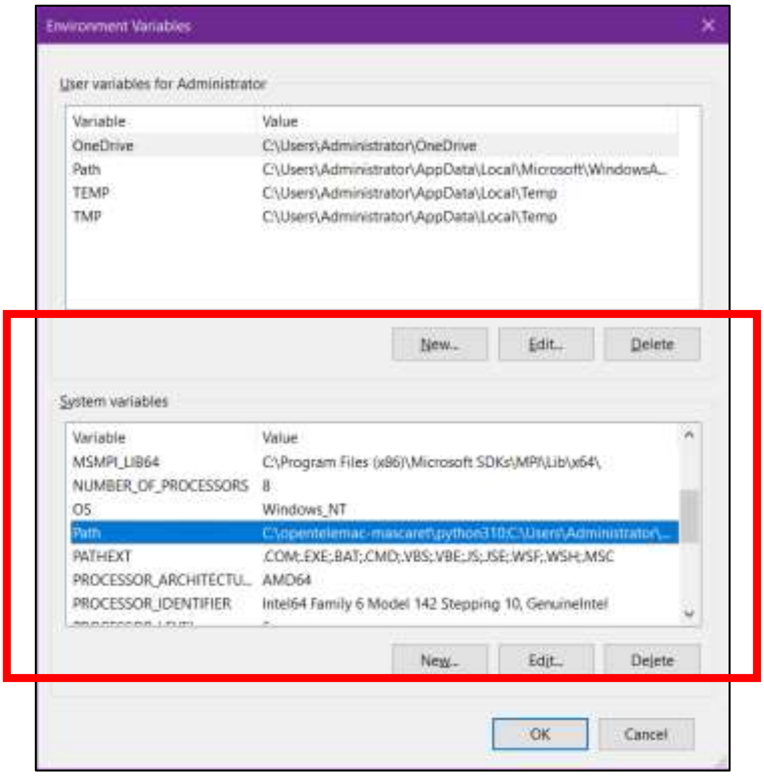


Figure 107 The Environment Variables Window

4. Under **System Variables**, double click on “**Path**”. Doing so will open another window called “**Edit Environment Variable**” and in this window, make sure that the following directory paths are added:

```
C:\Users\*your_username*\AppData\Local\Programs\Python\Python310\
C:\opentelemac-mascaret\mingw64\bin
C:\opentelemac-mascaret\mpich2\bin
C:\opentelemac-mascaret\mpich2\lib
C:\opentelemac-mascaret\python27
```

Note: Insert your computer’s user name at the <your_username>, and make sure to remove the asterisk signs afterwards.

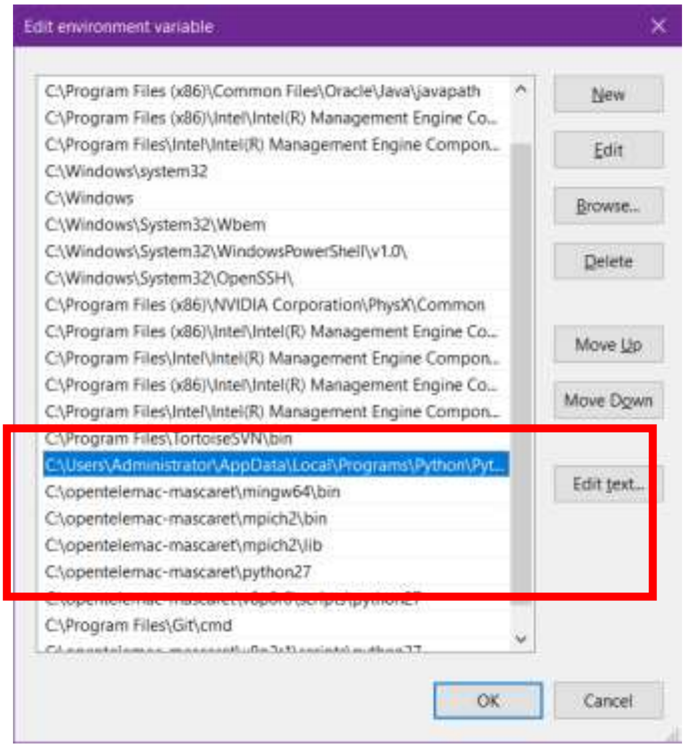


Figure 108 The Edit Environment Variable Window

5. If they are not yet present, add them individually by clicking “**New**” typing each of them individually, and clicking “**OK**” once finished.

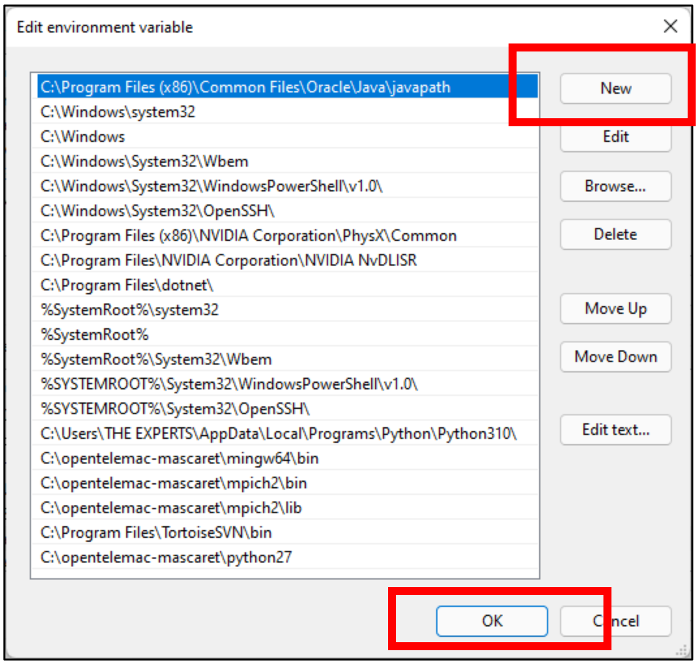


Figure 109 Adding a New Path

6. Click “**OK**” for each of the windows previously opened.

Appendix III: List of Possible Errors

1. This happens when TELEMAC automatic installer fails.

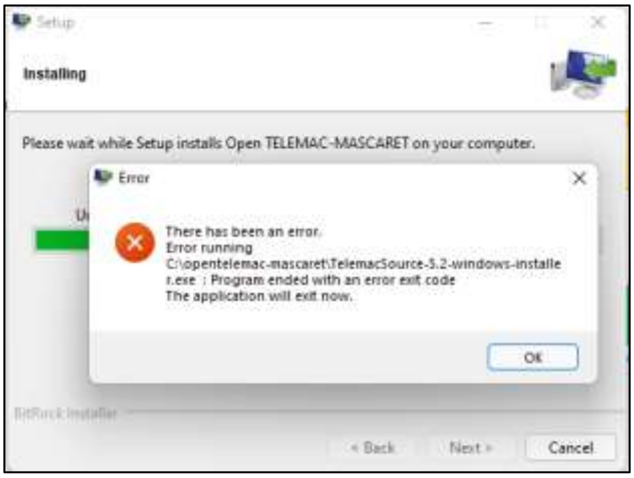


Figure 110. TELEMAC Automatic Installation Failure

This error can be caused by an unstable internet connection. To solve this, please see the Automatic Installer Failure section.

2. This error occurs when pip3 is not recognized as an internal or external command, operable program, or batch file.

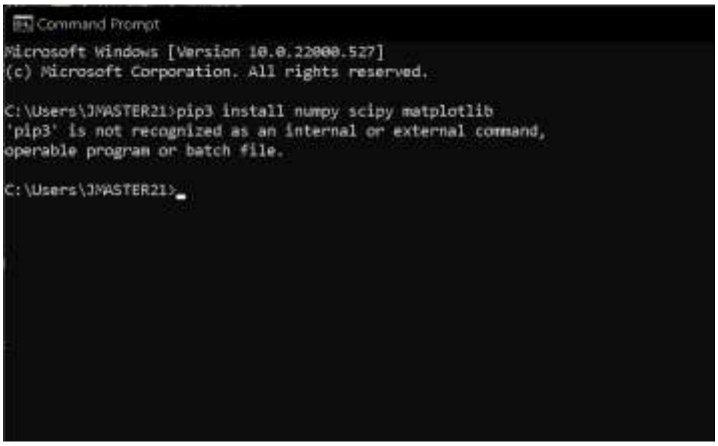


Figure 111. pip3 Error

This error can be possibly encountered after trying to manually install the numpy, scipy, and matplotlib libraries. Please see page 30 for reference.

Probable cause:

- Python 3.10 (or the newly installed python) is not added to the path.

Solution:

- Follow properly the steps in [Installing Python 3](). or
- Manually add it to the path. See [Adding items into the System Variables Path]().

3. ImportError: No module named configparser.

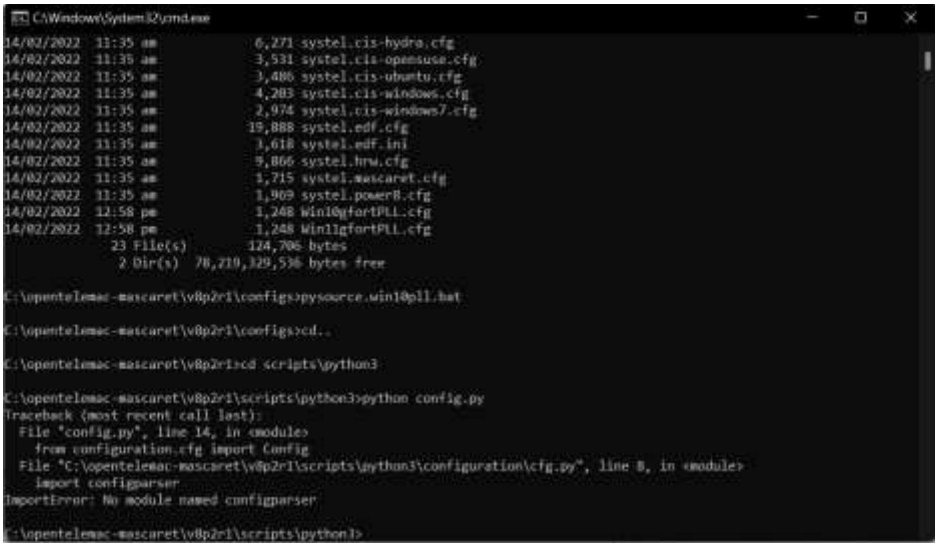


Figure 112. The Configparser Error

This same error might be encountered after trying to load the **config.py** file. This can be solved by making sure that both the config and batch files (**pysource.win10pll.bat** and **Win10gfortPLL.cfg**) are extracted in the correct directory. Please see Figure 43 on page 35.

Additionally, make sure also that the steps in [Adding items into the System Variables Path]() in the system variables are correctly followed.

4. ImportError: No module named request.

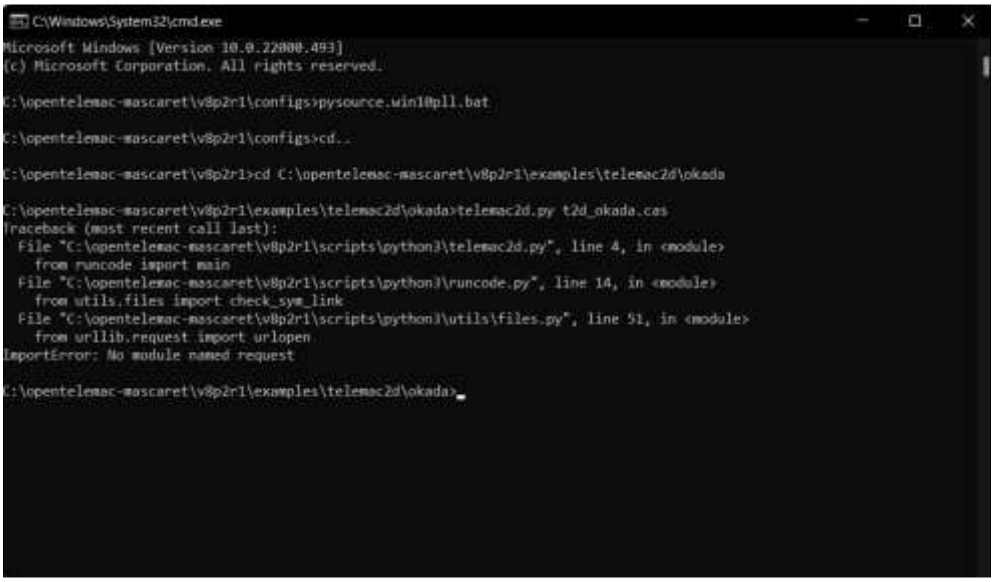
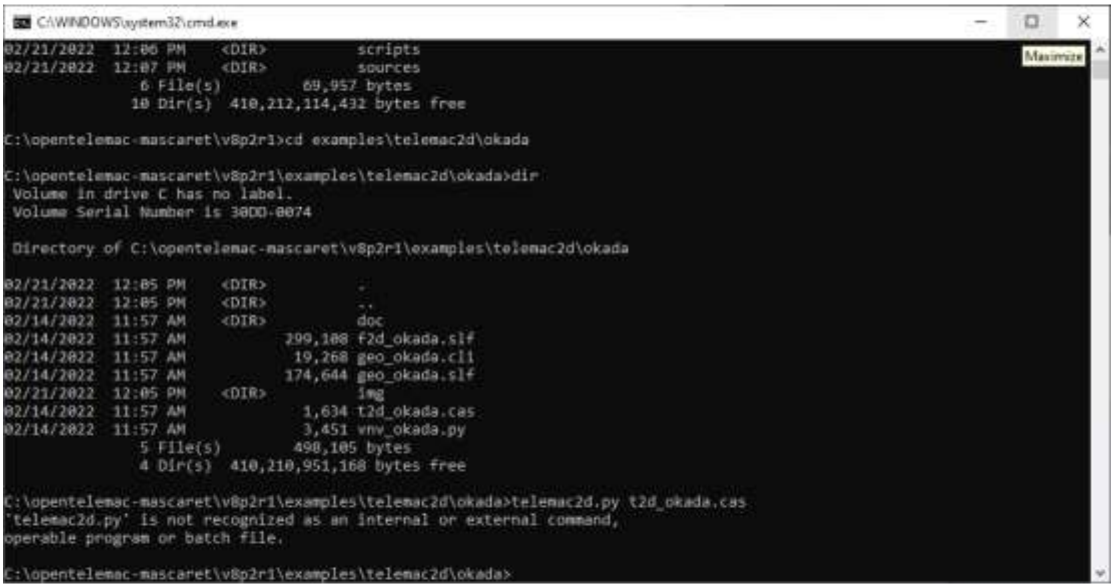


Figure 113. No Module Named Request Error

This error can be resolved also by making sure that the steps in [Adding items into the System Variables Path]() in the system variables are correctly followed.

5. This occurs when telemac2d.py is not recognized as an internal or external command operable program or batch file.

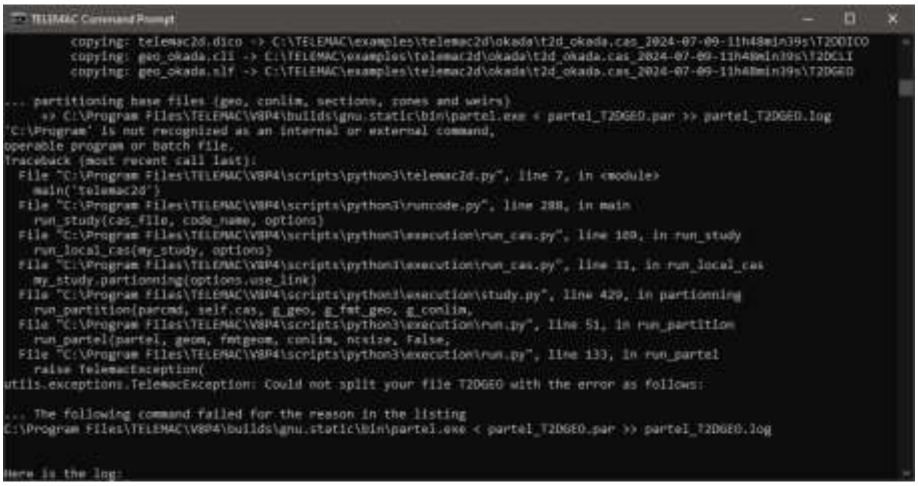


This error can be resolved also by making sure that the steps in [Adding items into the System Variables Path] in the system variables are correctly followed.

Note:

If, for some reason, the above steps do not solve the problem, try to uninstall Python and reinstall it again by following the steps described in the [Installing Python 3] section.

6. Issue with installing at default location. Contains white space at "C:\Program Files" and may cause "Unrecognized command" especially at parallel computations.



Appendix IV: General Summary of Steps in Running a Model

For Windows Environment

1. Load cmd environment.
2. Change the directory where simulation files are located.
3. Start the simulation by typing telemac2d.py *filename*.cas.

Note:

Please see the Running the Model Setup section for more information.

Appendix V: Visualized Results Files

Scenario 1 – Initial/ Uncorrected Steering File

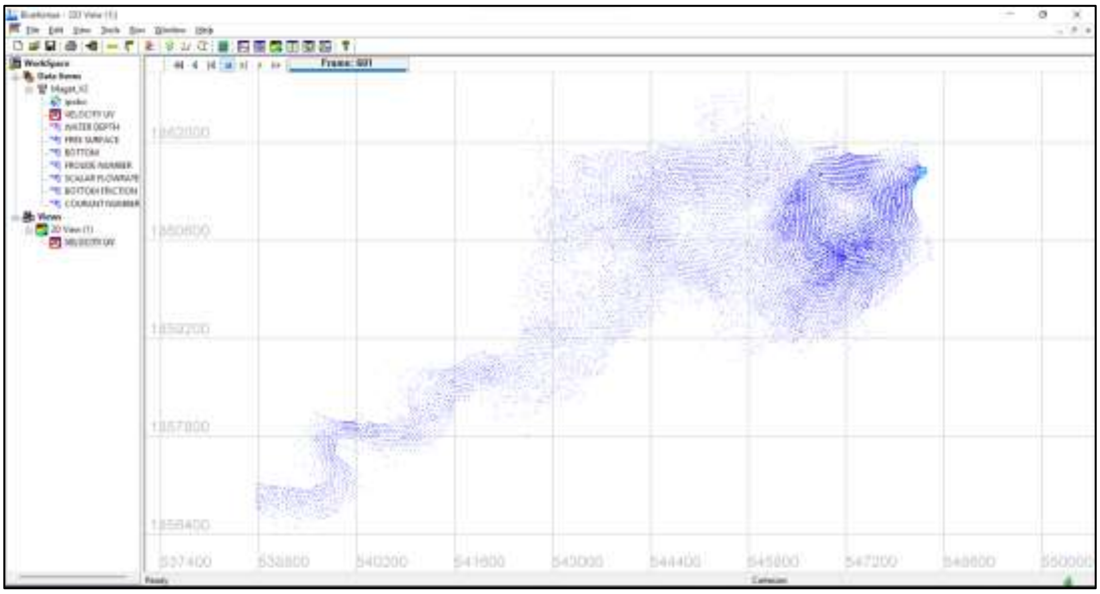


Figure 114. Scenario 1 Velocity Profile Visualized in BlueKenue

Scenario 1– Adjusted Steering File

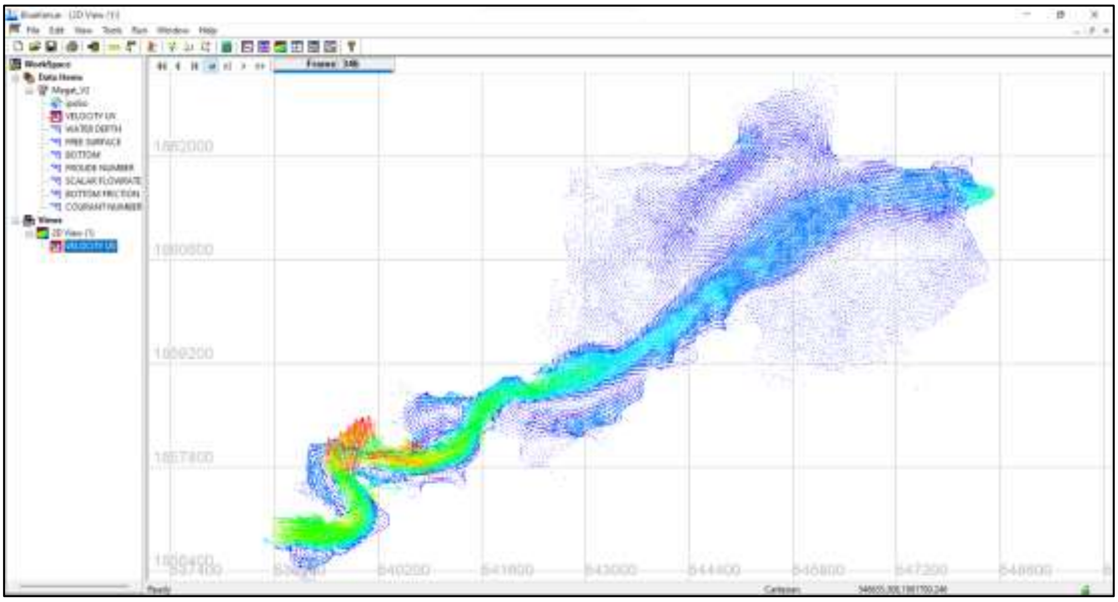


Figure 115 Scenario 1 Velocity Profile Visualized in BlueKenue

Scenario 2 – Initial/ Uncorrected Steering File

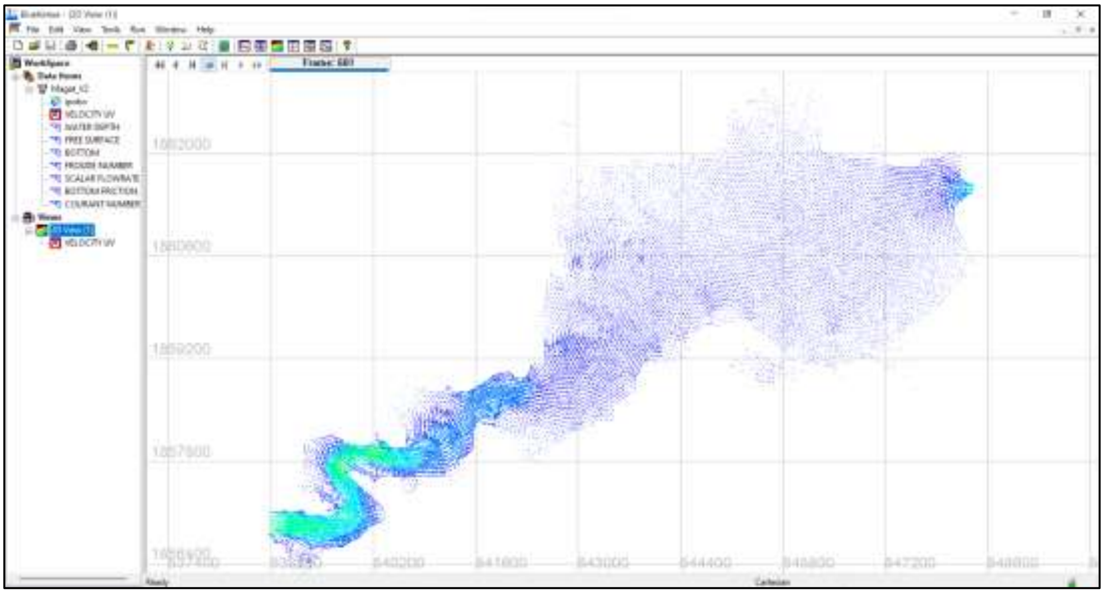


Figure 116. Scenario 2 Velocity Profile Visualized in BlueKenue

Scenario 2 – Adjusted Steering File

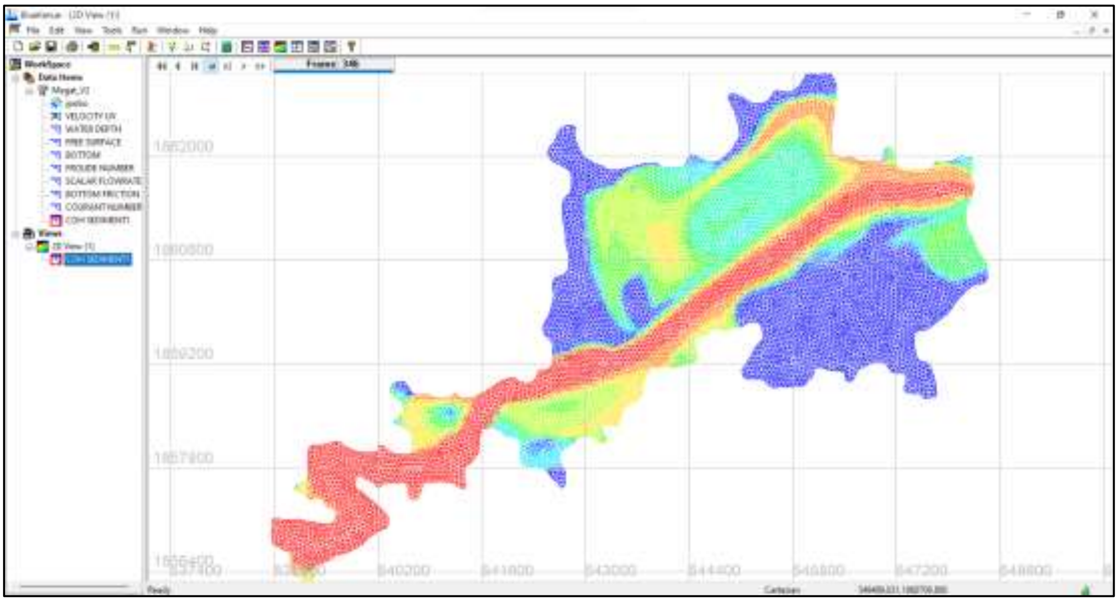


Figure 117. Scenario 2 Sediment Transport Profile Visualized in BlueKenue

Scenario 3 – Initial/ Uncorrected Steering File

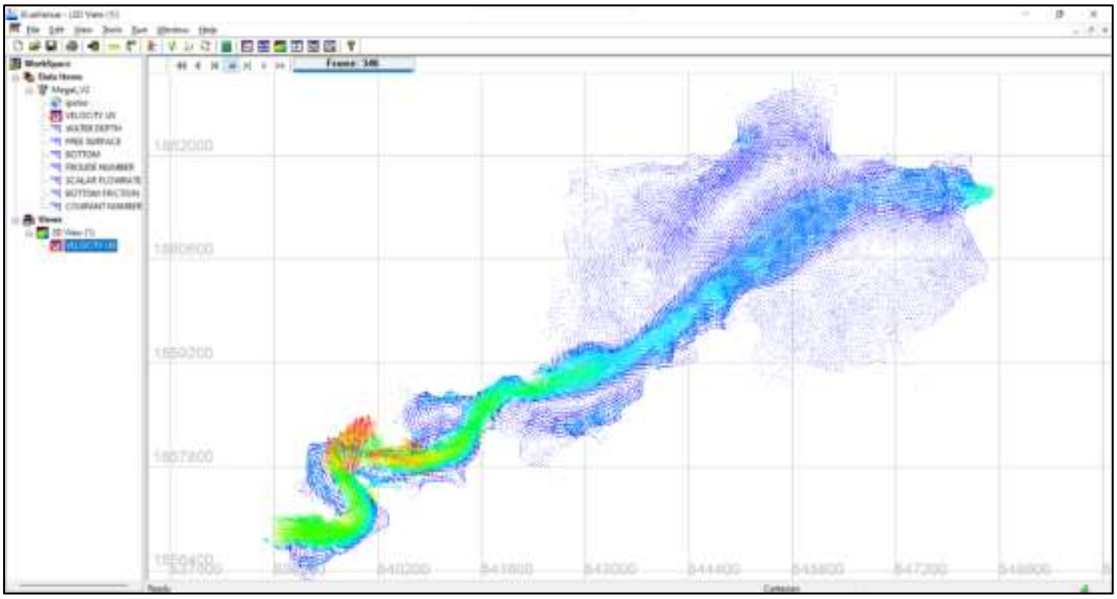


Figure 118. Scenario 3 Velocity Profile Visualized in BlueKenue

Scenario 3 – Adjusted Steering File

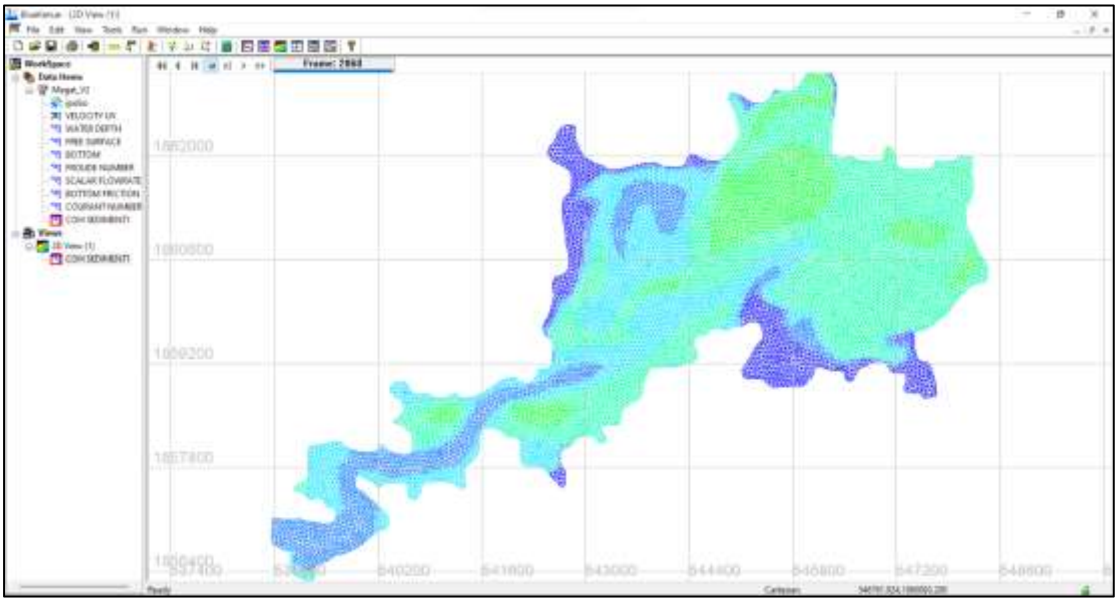


Figure 119. Scenario 3 Sediment Transport Profile Visualized in BlueKenue

Appendix VI: Downloading the latest release, TELEMAC v8p5

1. Go to http://www.opentelemac.org/ and log in using your account.

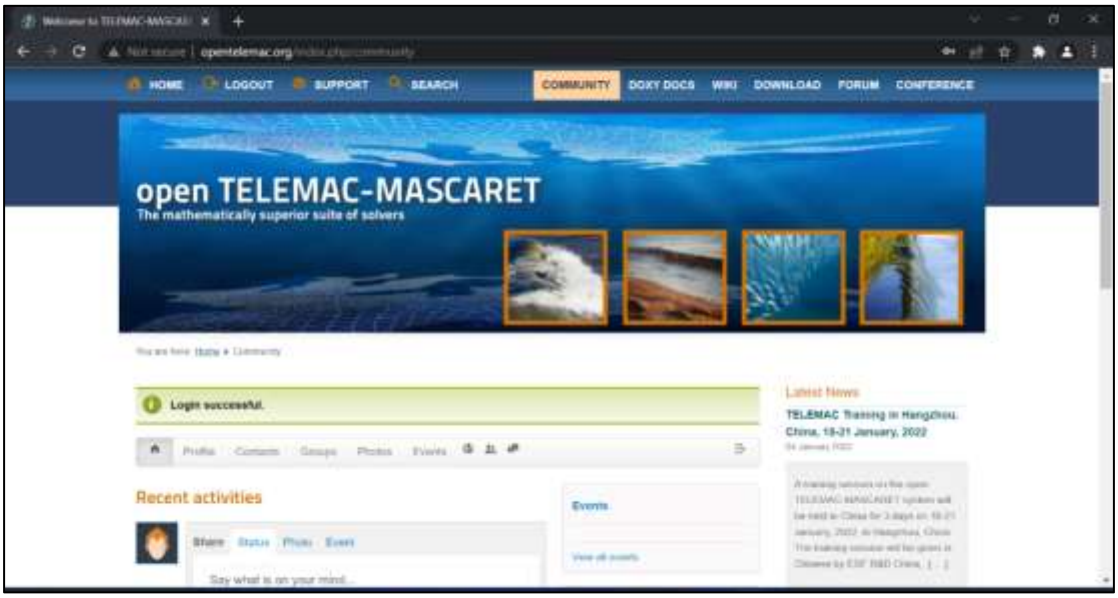


Figure 120. Logging in to your Telemac account

2. On the search bar, type "**v8p4**" or you can check the latest version in the **Latest News** section located on the right side of your screen.

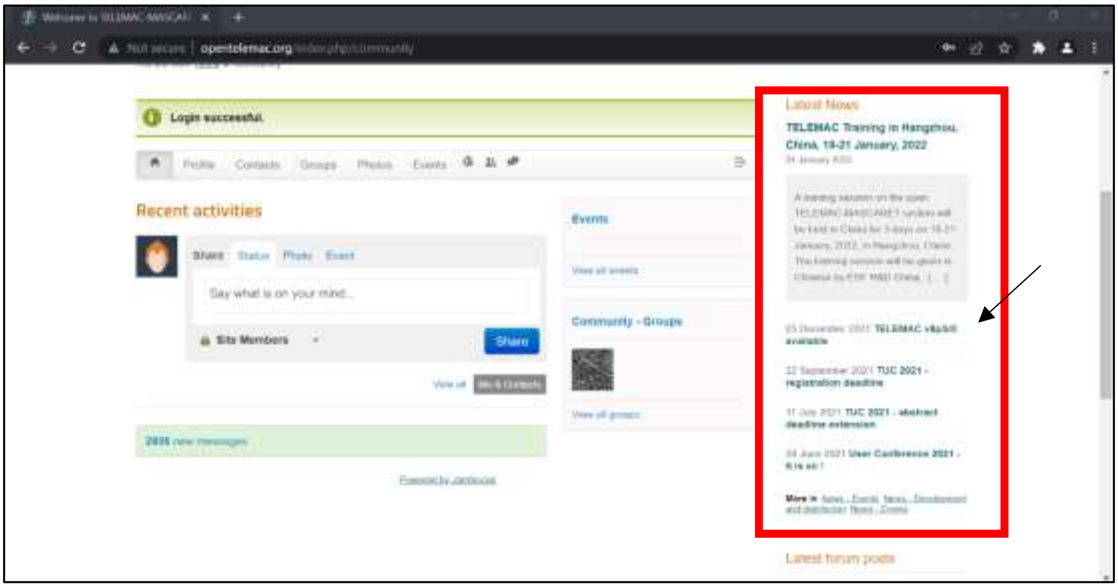


Figure 121. The Latest News section of the Telemac website

3. Click “**v8p4**” to navigate to the downloading page.

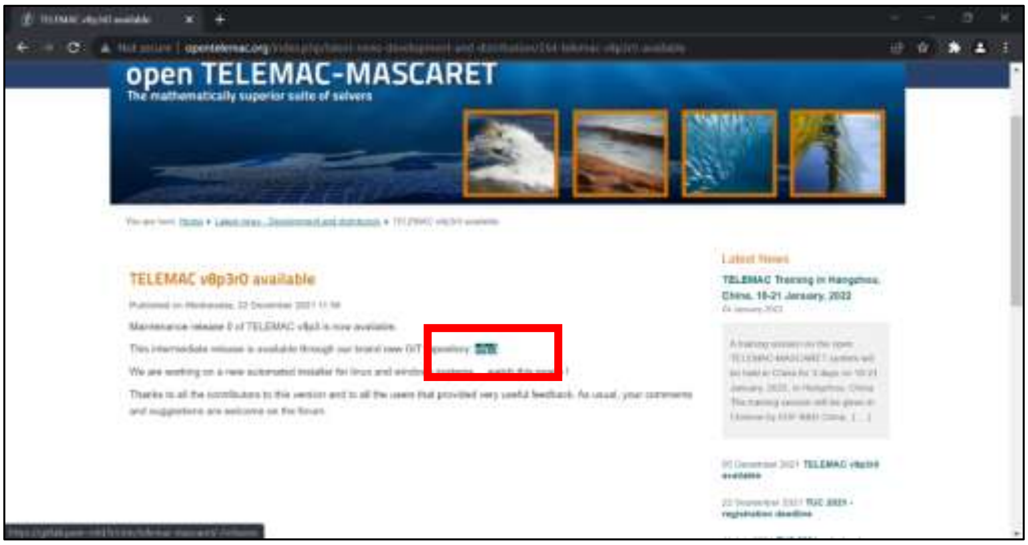


Figure 122. A short release statement regarding the availability of v8p3 version

4. The latest release is hosted through Git and will provide a faster checkout process compared to the traditional SVN. Click on “**Source Code”** to start downloading. Note: The latest release is V8P5R0.

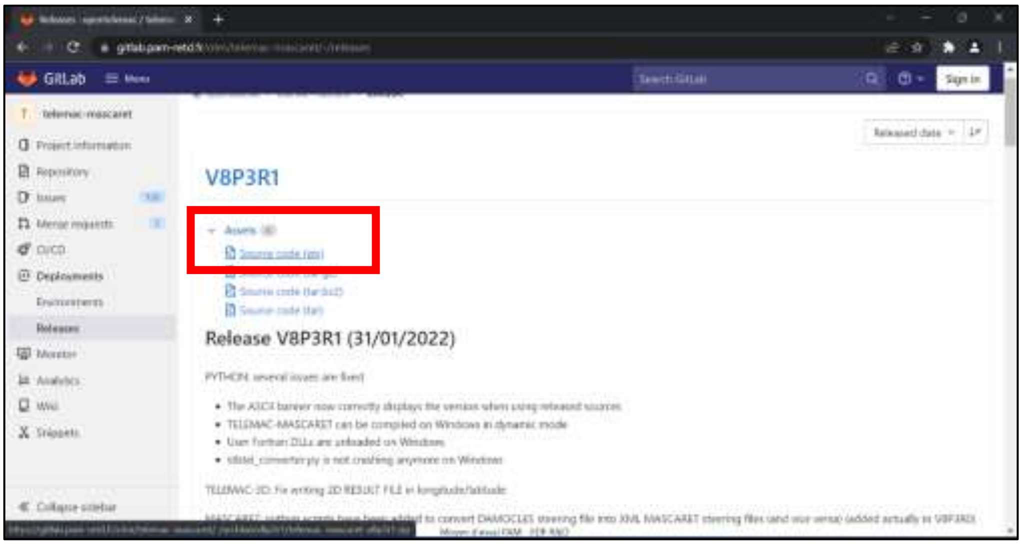


Figure 123. The source code download section

5. Download and save the compressed file into your TELEMAC installation folder (**C:\opentelemac-mascaret**).

6. Extract the contents of the downloaded compressed file.

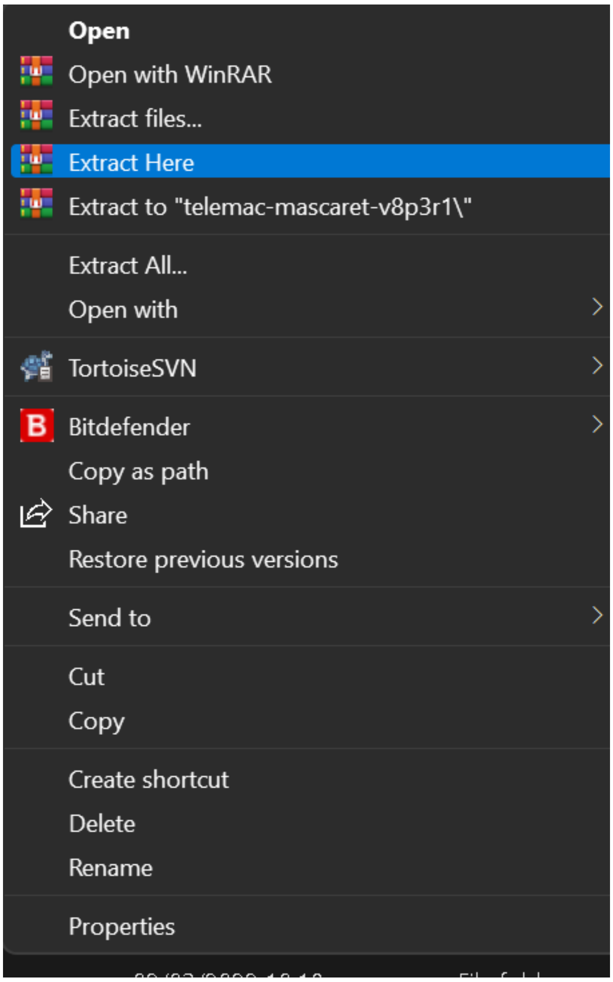


Figure 124. Extracting the contents of the downloaded compressed file

Appendix VI: Importing an existing shapefile as an outline.

If, for instance, you have already generated an outline for your study area using any GIS application, you can use and import it into BlueKenue to save some time in drawing your outline.

1. Click on File >> Import >> ArcView Shape File.

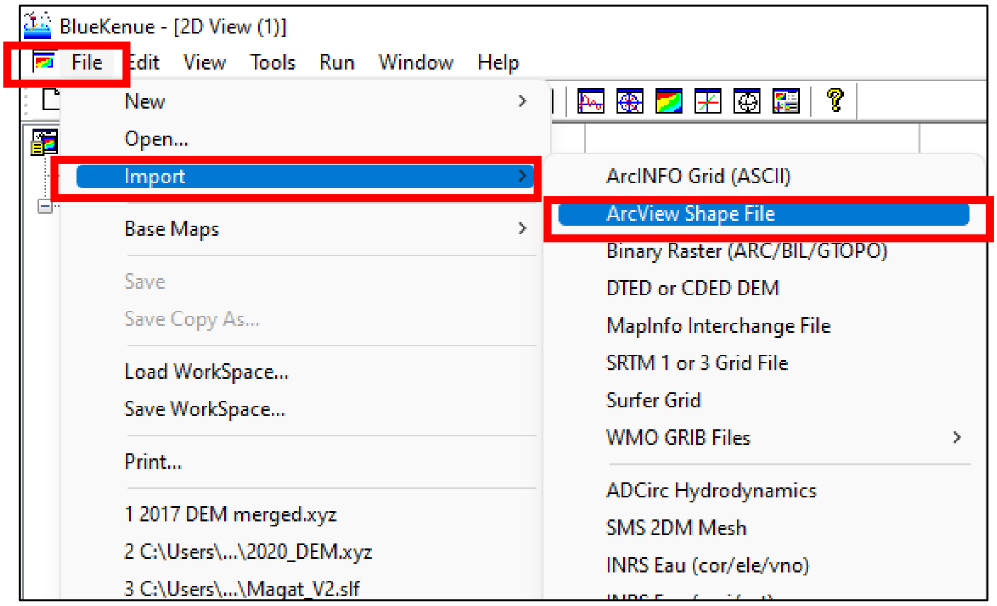
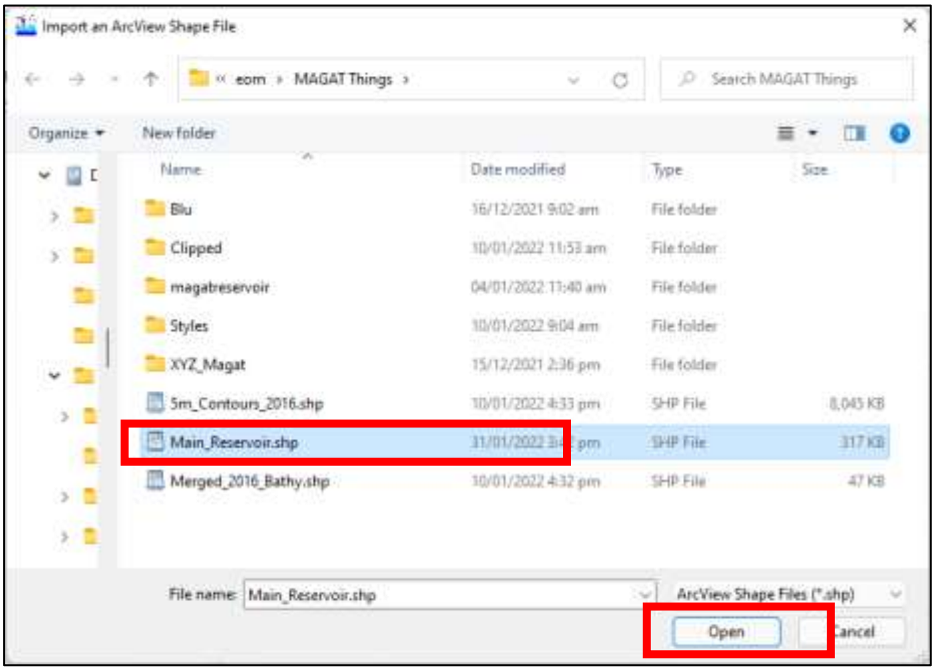
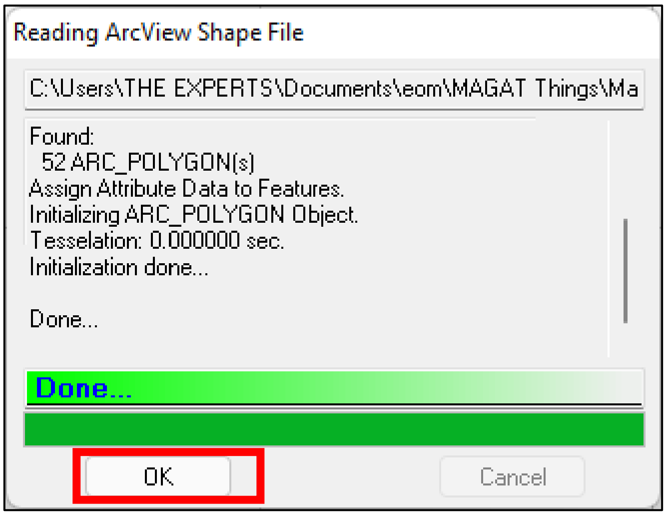


Figure 125. Importing a shapefile

2. Navigate into where your .shp is located and select and open it.



3. BlueKenue will try to read and load your file. If no errors were encountered, a “**DONE**” message shall appear. Click OK to close the message.



4. You can now drag and drop the loaded shp file into your 2D view to visualize it.

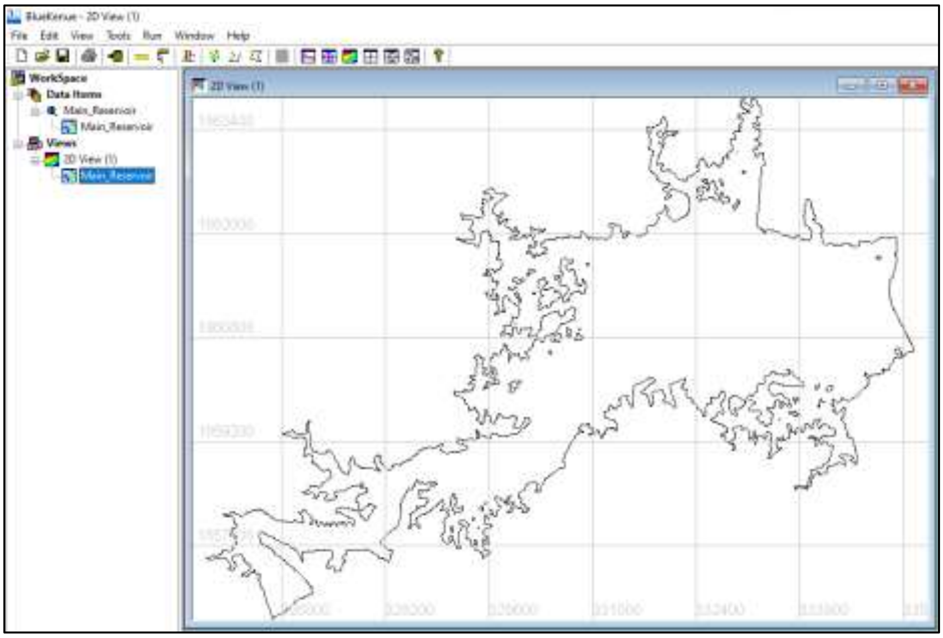


Figure 126. The Magat Reservoir outline loaded as a shapefile

References

Canadian Hydraulics Center (CHC) - National Research Council (NRC). (2011, September). Blue Kenue Reference Manual.

ELECTRICITE DE FRANCE (EDF). (2022, February). *Modules*. Retrieved from open TELEMAC-MASCARET: http://www.opentelemac.org/index.php/modules-list

Galland, J.-C., Goutal, N., & Hervouet, J.-M. (1991). TELEMAC: A new numerical model for solving shallow water equations. *Advances in Water Resources, 14*(3), 138-148.

Leon, A., & Gifford-Miears, C. (2013). Tutorial on the use of TELEMAC-2D Hydrodynamics model and Pre-/Post-processing with BlueKenue for flood-inundation mapping in Unsteady Flow Conditions. Oregon State University.

Schwindt, S. (2022). *TELEMAC.* Retrieved February 2022, from Hydro-Informatics: https://hydro-informatics.com/numerics/telemac.html