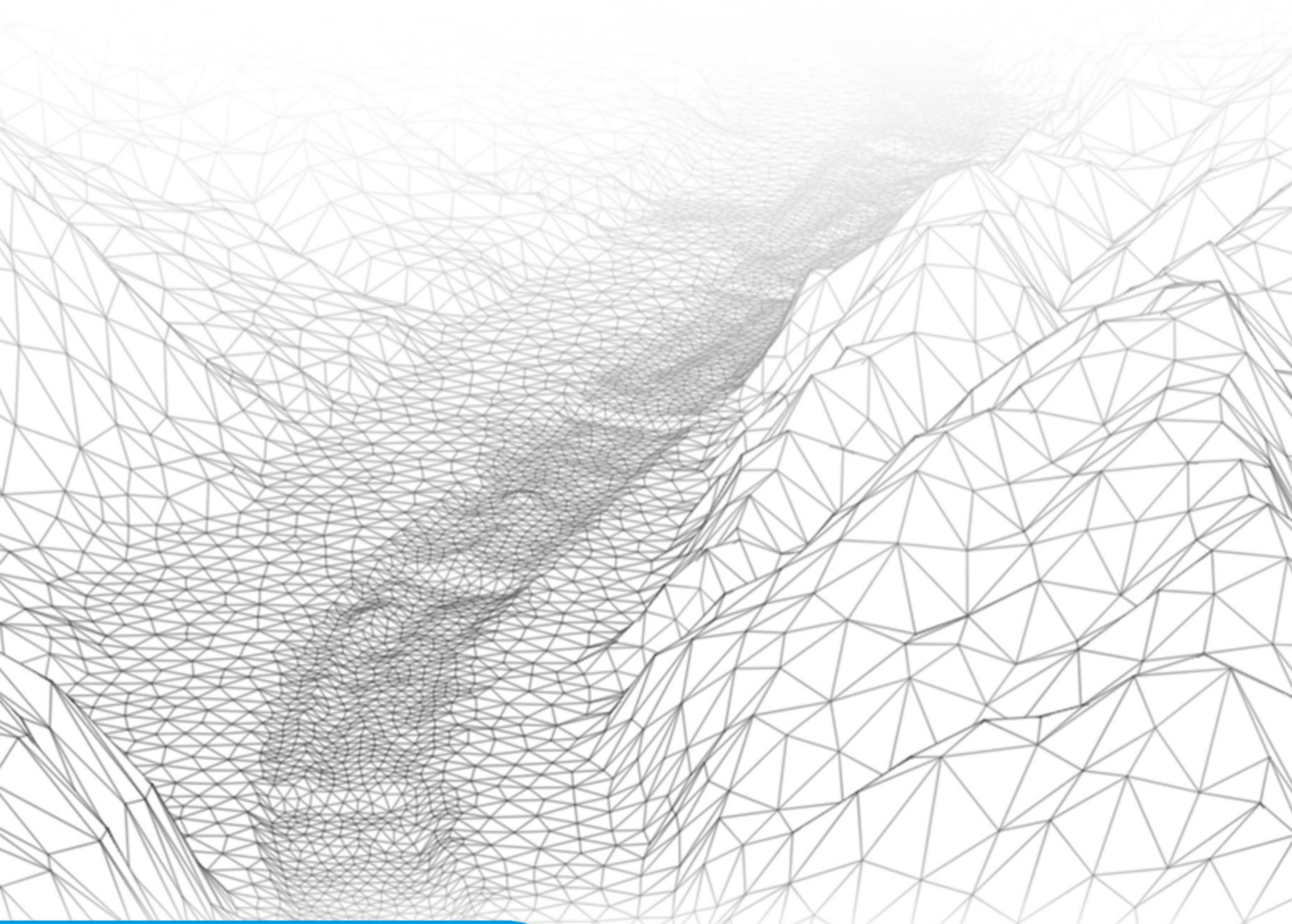


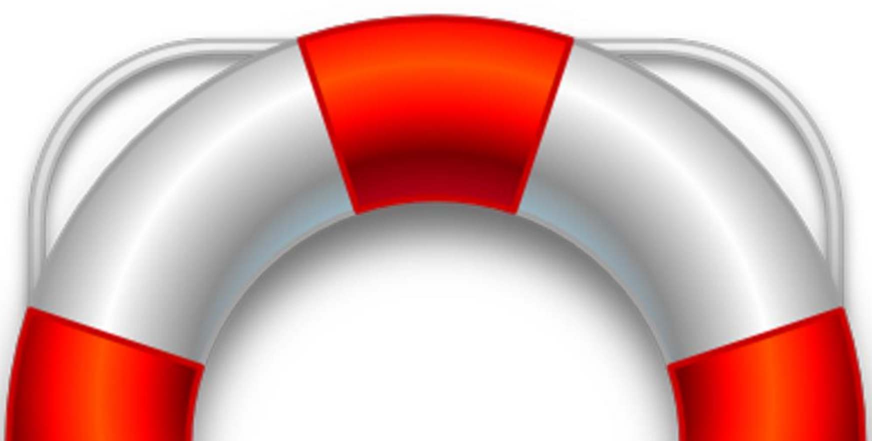
TELEMAC-2D

HYDROEUROPE

Erasmus+ · UNIVERSITAT POLITÈCNICA DE CATALUNYA · Vrije Universiteit Brussel · b-tu Brandenburgische Technische Universität Cottbus - Senftenberg · Newcastle University · POLITECHNIKA WARSZAWSKA · Université Nice Sophia Antipolis · POLYTECH NICE-SOPHIA

Unsteady flow simulation - Step by Step tutorial

By Marcin Kawka, Warsaw University of Technology



Tutorial 1: Basic Var river simulation

The goal of this tutorial is to get yourself familiar with basic TELEMAC-2D usage. You will learn how to create a simple mesh in BlueKenue, set up the simulation case scenario in Fudaa-Prepro and how to run the simulation. Finally results visualisation will be performed with BlueKenue and Fudaa Prepro. In the second part you will learn how to use spatially variable bed resistance coefficient.

The following data are required to be used during this exercises:

- *riverOutline.i2s* – an ASCII file with outline of the Var river flume (BlueKenue format)
- *DEM_Var_burned.asc* – an ASCII file with DEM of lower Var, interpolated on a regular grid with 5 meters resolution. This DEM has been manually modified, in order to remove bridges.
- riverFlume.shp – a vector file with polygons, containing different bed types in Var river flume.
- discharge.liq – text file with discharge data from November 1994 flood

All spatial layers are prepared using *RGF93 / Lambert-93* spatial coordinate system (EPSG code: 2154). We recommend that you keep this coordinate system for all calculations related to Var river.

Generating simple mesh

The first step before running your simulation is to generate the computational mesh. Open BlueKenue (BK) and from the top menu select *File→ Open,* change file format to *Line Sets (*.i2s,*.i3s)* and select *riverOutline.i2s* and click *Open*. In order to start new mesh generation, from the top menu of BlueKenue window select *File→ New→ T3Mesh Generator...* In the dialogue window enter 20 as the *Default Edge Length and* click *OK*. Drag the *riverOutline* layer in the *Data Items manager* (left pane of the BK window) and drop it on *2D View*, drag it once again and drop on *Outline* of *newT3Mesh.* Double click on the *newT3Mesh* to open its properties. In the properties dialogue window click *Run* to start mesh generation process. In a few seconds BK should complete the mesh generation process. After successful competition a new item should appear within the *newT3Mesh,* called New *Mesh (Node Type)*. Drag and drop it to *2D View.* Your results should resemble the ones from fig. 1.

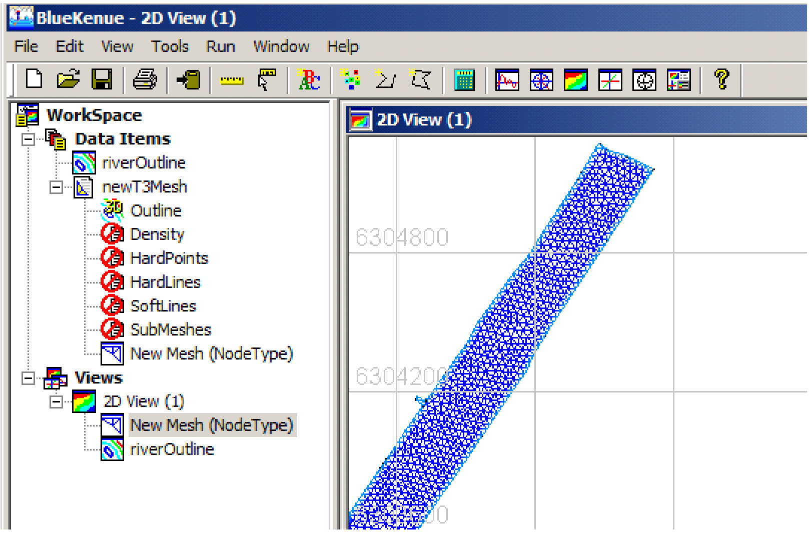


Figure 1 : Results of your first mesh generation in BlueKenue

As you generated your first mesh, river bottom elevation data have to be interpolated on mesh nodes. From the top menu of BK select *File→ Import→ ArcINFO (ASCII)* and select *DEM_Var_burned.asc* This file contains DEM of lower Var and will be used as a source of elevation data for our mesh.

Highlight the *New Mesh (Node Type)* layer and from the top menu select *Tools→ Map Object.* In the dialog window select the *DEM_Var_burned* to be mapped on the mesh and click OK. Your mesh should become multicolour, with colour levels representing river bed elevation.

You can adjust the colour scale in layer properties. Double click on layer name in the *Views* section of the left pane. In the properties dialog window, go to *ColorScale* tab. Change the interval to 8 meters and click *Apply*. In the *Meta Data* tab, set the keywords Name and Title to BOTTOM. This will inform TELEMAC, that values associated with mesh nodes are bottom elevations.

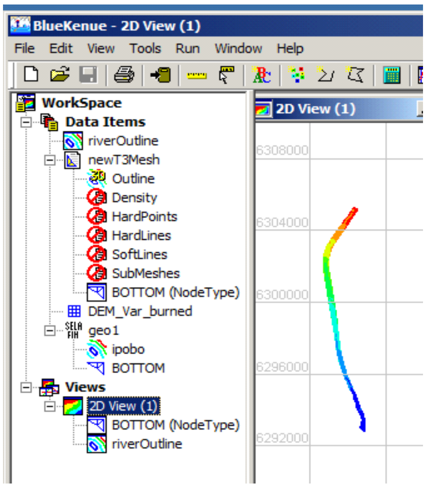


Figure 2: Mesh after DEM data interpolation.

To create a new file for mesh storage, select from the BK top menu *File→ New→ Selafin OBJECT…* In left pane of the window (Data Items manager), drag and drop your BOTTOM layer on the newSelafin file.Highlight the *newSelafin* file and select from the top menu *File → Save* and save it under the name *geo1.slf* .

In order to be recognized by Fudaa Prepro, the file name has to start with geo. That's the end of basic mesh generation, you can close Blue Kenue for a moment and proceed to simulation case file setup in Fudaa Prepro.

Setting up your first simulation

To start setting up scenario of your first simulation open *Fudaa Prepro*. The main window of Fudaa is called *Supervisor.* It acts similarly to a typical file and directory explorer (e.g. Windows Commander), however it recognizes the files associated with TELEMAC. You will see dedicated icons for case files, result files and meshes.

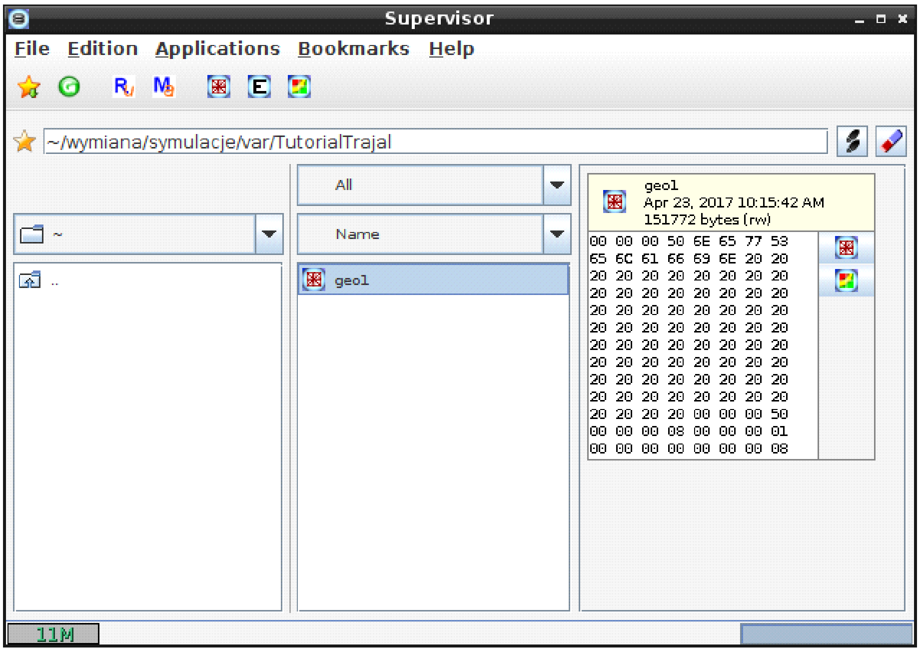


Figure 3: Fudaa Prepro Supervisor window, just after program launch

Navigate to the folder which contains your newly created mesh file (geo1.slf). Right click on the file name and choose *Rename* from the drop-down menu. Remove the extension *.slf* after the file name. After removing the file extension, your mesh should be recognized by Fudaa as a *geometry file* (fig. 3).

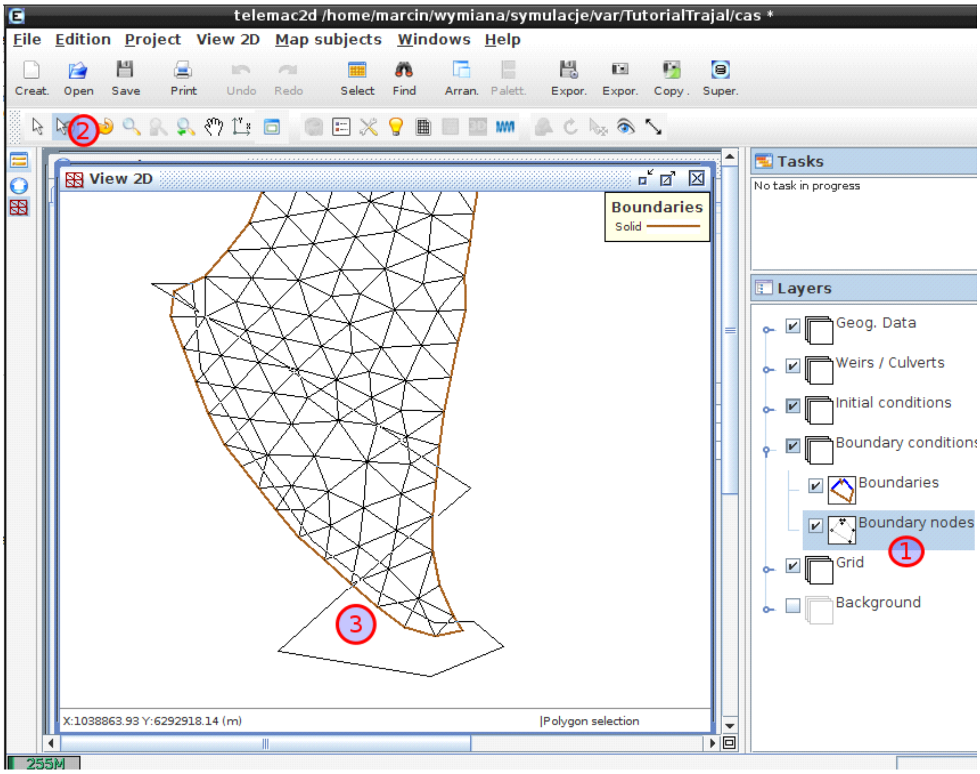


Figure 4 : View 2D window in Fudaa Prepro

Right click on the mesh file and select *Create a Telemac project* from the drop-down menu. In the dialogue window, make sure that *telemac2d* dictionary of the latest version is selected. The default *Steering file* name is *cas* and its' location is the current directory. You can modify the name; however it should start with cas, in order to be recognized by Fudaa. This tutorial assumes that you selected *English* as the language of the case file. Selecting *French* would stick you to French keywords for the rest of your work on this case file. Accept and close the dialogue window, by clicking *Validate.*

By default, Fudaa Prepro should open *View 2d* window now and ask you to initialize the boundary conditions. Using magnifier, zoom into mouth of river Var on your mesh. In the layers manager (right pane of the window – fig.4, pt. 1) select *Boundary nodes* layer. Use polygon selection tool (fig. 4 - pt. 2) to select boundary nodes at Var mouth. End your selection with double click. Selected nodes should remain highlighted. Right click on them and from the drop-down menu select *Insert liquid nodes…* Select liquid nodes with prescribed elevation at 0.0 meters (fig. 5). This will be the lower boundary of your model. Accept by clicking *Validate*. A new liquid boundary should appear on your mesh.

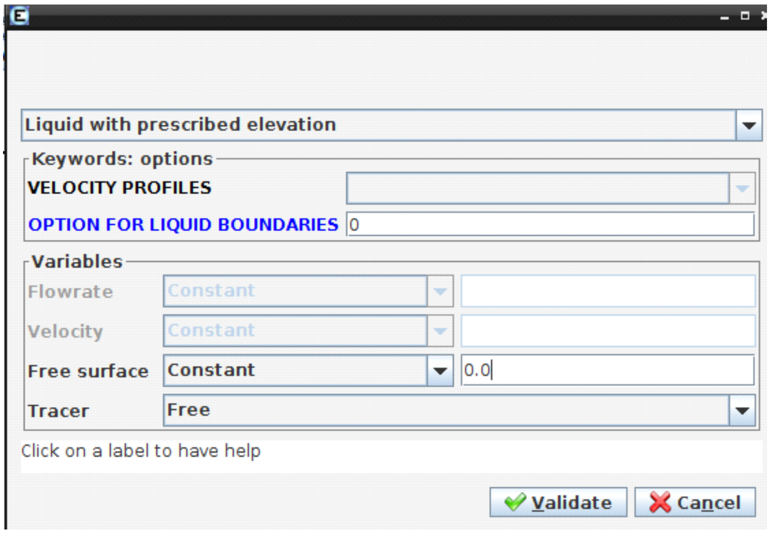


Figure 5: Liquid boundary options

Navigate to upper end of your mesh. An inflow boundary should be created here. Unfortunately, it's impossible to use inflow discharge hydrograph and dry domain initial condition at the same time (this is a common issue in many modelling systems). A popular way to overcome this issue is to create point sources on the boundary nodes, which are intended to be inflow boundary and "wet" the initially dry domain. In the Layers manager, highlight *Sources* (subgroup of *Weirs / Culverts*). Right click on the mesh and from the drop-down menu select *Add sources*. In the dialogue window enter 20 as water discharge of sources. Click on several nodes on upper boundary, to add new sources (fig.6).

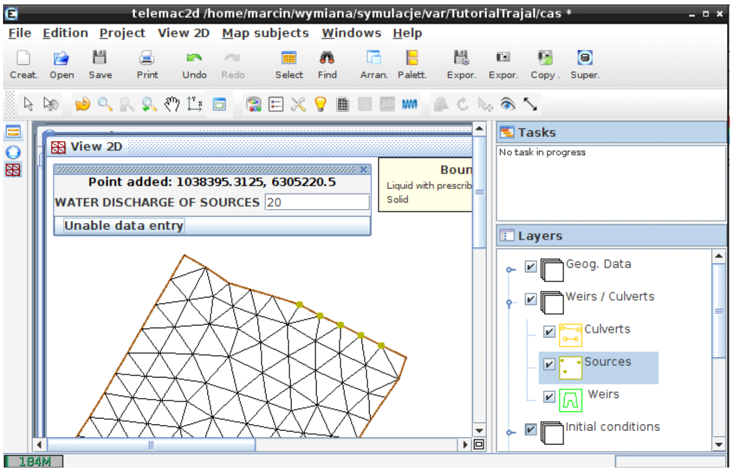


Figure 6: Adding new sources to mesh

After you finished adding new sources, close the *View 2D.* Open the *General Parameters* window (From the top menu of Fudaa window select *Project → General Parameters*).

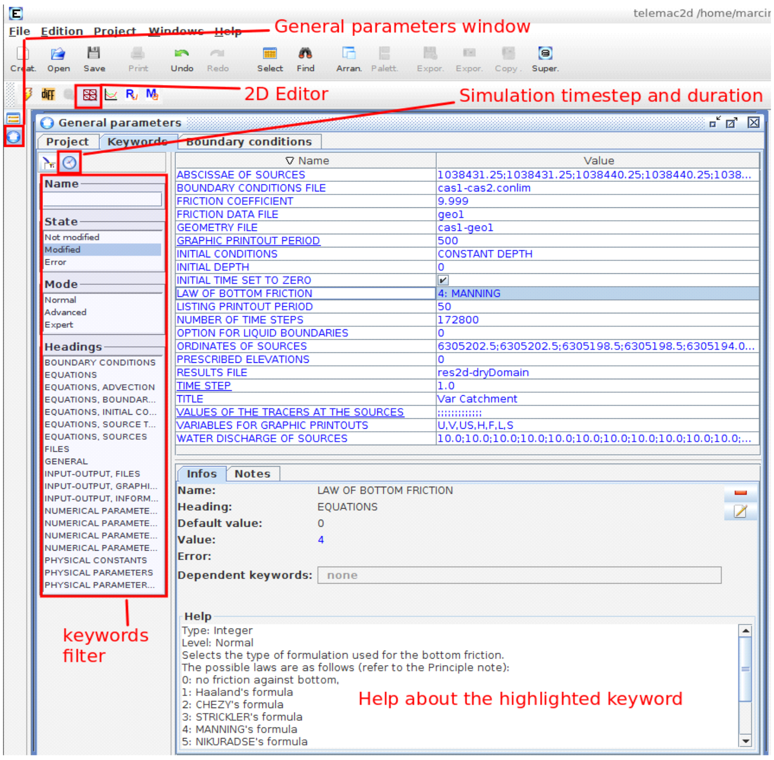


Figure 7: General parameters window

The general parameters are organized in a format keyword = value. In the left pane of the window you can intuitively filter listed keywords. The modified keywords are highlighted in blue, non-modified in black, keywords with errors (usually missing or improper values) are highlighted in red. A minimum set of parameters, which need to be set for your first simulation are given in the table below:

Name	**Value**	**Comment**
LAW OF BOTTOM FRICTION	4	Value indicates, that friction coefficient should be interpreted as Manning's *n*
FRICTION COEFFICIENT	0.035	Uniform value of Manning's n for the whole domain
RESULT FILE	res2d	Name of the file which will contain results of the simulation
LISTING PRINTOUT PERIOD	30	To limit the amount of the text output, your model will print listing information every 30 steps
GRAPHIC PRINTOUT PERIOD	600	To limit the size of graphic output file, your model will save results once per 600 steps
VARIABLES FOR GRAPHIC PRINTOUTS	B,U,V,H,S	Symbols of variables which will be saved in results file

After you set all the necessary parameters click the time step button (watch icon – fig.

7.) and set length of your simulation to 24h and time step to one second. Fudaa should automatically recalculate number of time steps to 86400.

Your first simulation scenario in TELEMAC2D is now ready to be run. Save your case file and exit Fudaa Prepro.

Running the first simulation

Before running, transfer your mesh and case files to the computer (or virtual machine), that you are planning tu run TELEMAC. Remember to copy not only the two files, but also any auxiliary files, which might have been produced by Fudaa. It's a good practice to create each new simulation in a new folder.

To run the simulation type into text terminal (or command prompt):

telemac2d.py case_file

It happens very often that first simulation fails to run on first attempt. In the last section of this document you can find explanations and solutions for the most common TELEMAC2D error messages.

Visualizing results of your simulation

You can use both Fudaa and BlueKenue to visualize your results. Visualization using Fudaa is more rapid-report-oriented (however it's still feasible to make animations), while BlueKenue is supposed to be more interactive. To start viewing your results in Fudaa Prepro, double click on the results file *res2d* in Fudaa Supervisor window. Figure 8 summarizes the most popular features which can be used for results visualisation. A very commonly used approach is to export the time step of maximum flood inundation to xyz (CSV) file. Such file can be further used to produce inundation maps or resilience analysis in GIS software (e.g. QGIS or ArcGIS).

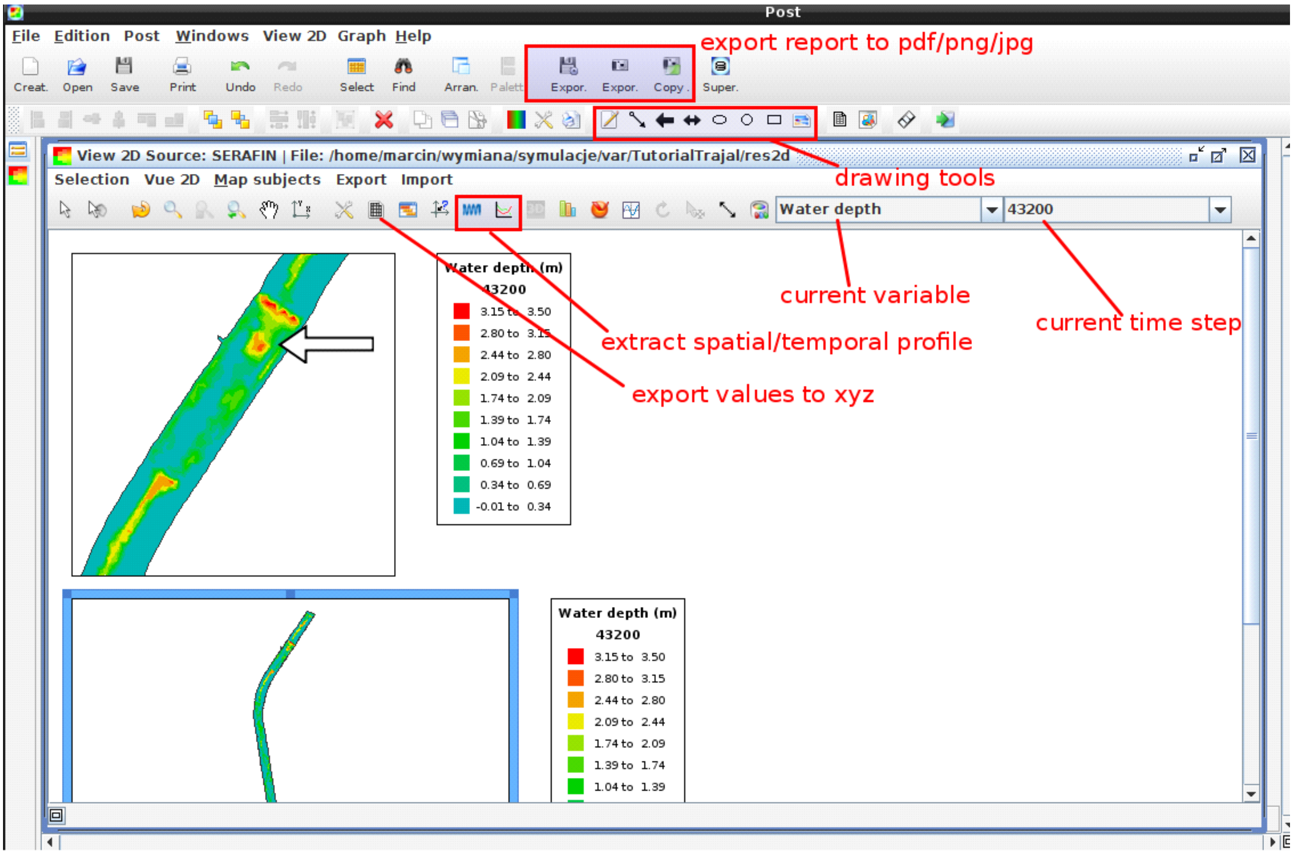


Figure 8: TELEMAC2D results visualization in Fudaa Prepro

To open the results in BlueKenue, select from the top menu *File→ Open*, switch file type filter to show all files and click on the res2d file. The idea of visualisation in BK is very similar to the concept of mesh generation. In the left pan of the window you drag and drop variables from data items to 2D or 3D views. You can change colour scale and transparency in properties of each variable (right click on the variable name). To enable animation, right click on the variable name and select *Animate*. Worth mentioning are also extraction tools (from the top menu *Tools→ Extract Surface/TimeSeries/*) and animation from a fly path (View→ *New Flight Path)*. Zoom in/out, rotation is available under mouse wheel, combined with Ctrl and Shift. Figure 9 summarizes visualisation interface of BlueKenue.

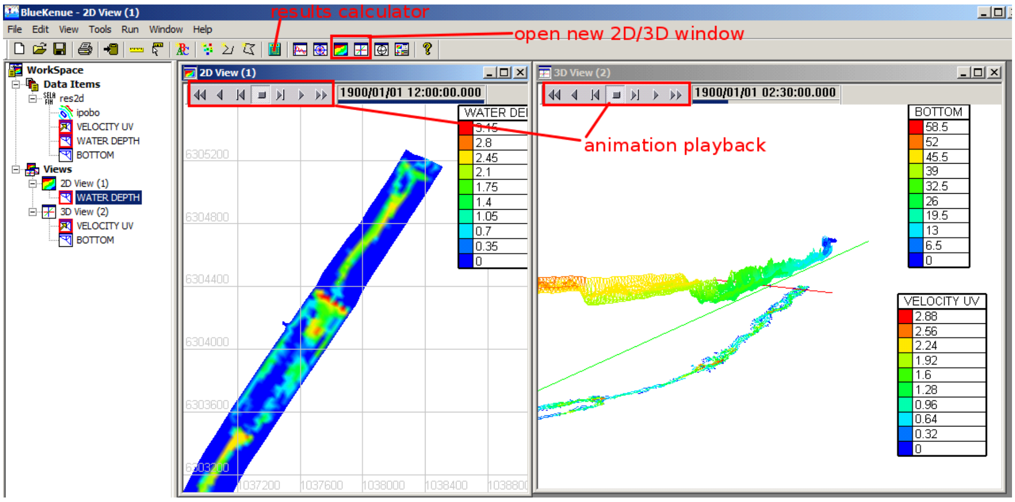


Figure 9 : TELEMAC2D results visualization in Blue Kenue

Using spatially variable bottom roughness

So far you assumed uniform roughness coefficient within the whole riverbed. In order to move your model closer to reality we will classify bed roughness into three classes:

- dense vegetation – Manning's coefficient equal to 0.05
- sparse vegetation – Manning's coefficient equal to 0.035
- pebbles or sandy material – Manning's coefficient equal to 0.02

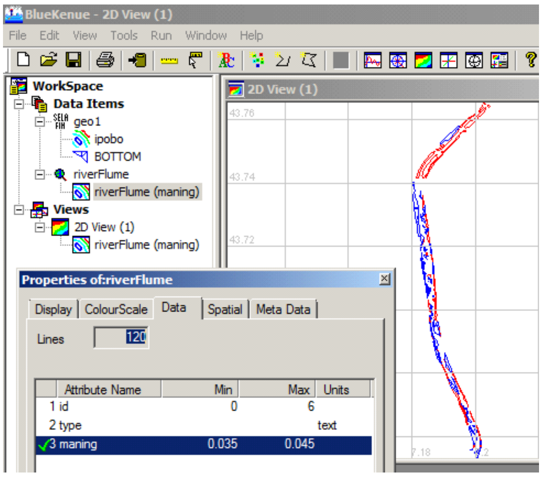


Figure 10: Loading shape file with non-uniform bed roughness in Blue Kenue

Open BlueKenue and start from loading your geometry file with the mesh which you have generated during previous steps of this tutorial. Load the vector file with areas of different roughness *riverFlume.shp* into BlueKenue (*File→Import → ArcView Shape file*). By default, the first column from *Attribute Table* is interpreted as a data field. Double click on the layer name to open its properties. Go to *Data* tab and tick the *maning* column as the one with roughness data (fig.10).

On the left pane of the window, right click on the file containing mesh (geo1) and select "*Add Variable*" from the drop-down menu. Select "BOTTOM FRICTION" as the name of the new variable and "s/m^(1/3)" as units, 0.02 as the default value. This should create a copy of BOTTOM layer, within the geo1 file with a uniform friction assigned.

To use the friction values from shape file, highlight the "BOTTOM FRICTION" layer and select from the top menu: *Tools→ Map Object* and select riverFlume from the available objects list. As a result – Manning friction coefficient values should be mapped on the grid. To check the results - drag the "BOTTOM FRICTION" layer on the 2D View (fig. 11).

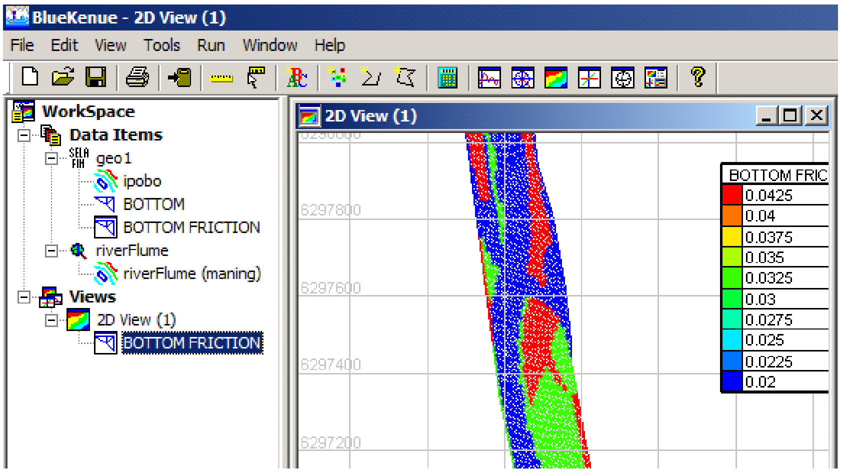


Figure 11: Bottom friction, mapped on the computational mesh

If you are satisfied with the friction coefficient map, highlight the geometry file and choose from the top menu *File → Save.*

In your case file you need to set the name of the file containing friction map, using the keyword "FRICTION DATA FILE". In areas, which are not covered by your spatial manning distribution it's a good practice to set an extremely large value of FRICTION COEFFICIENT (i.e. greater than 1), so that you will be sure that your friction map is really in use. The keyword GEOMETRY FILE should point to the same file as FRICTION DATA FILE.

Non-dry initial condition

So far, your previous simulation was using point sources with constant discharge, located on the upper end of the computational domain. To use discharge as upper boundary condition, non-dry initial condition is needed. Last step of your simulation with dry domain can be used as an initial condition in a simulation with discharge defined on the upper boundary. Open results of your simulation in Fudaa Prepro. Highlight the frame with results and from top subwindow menu select *Export → Export Data.*

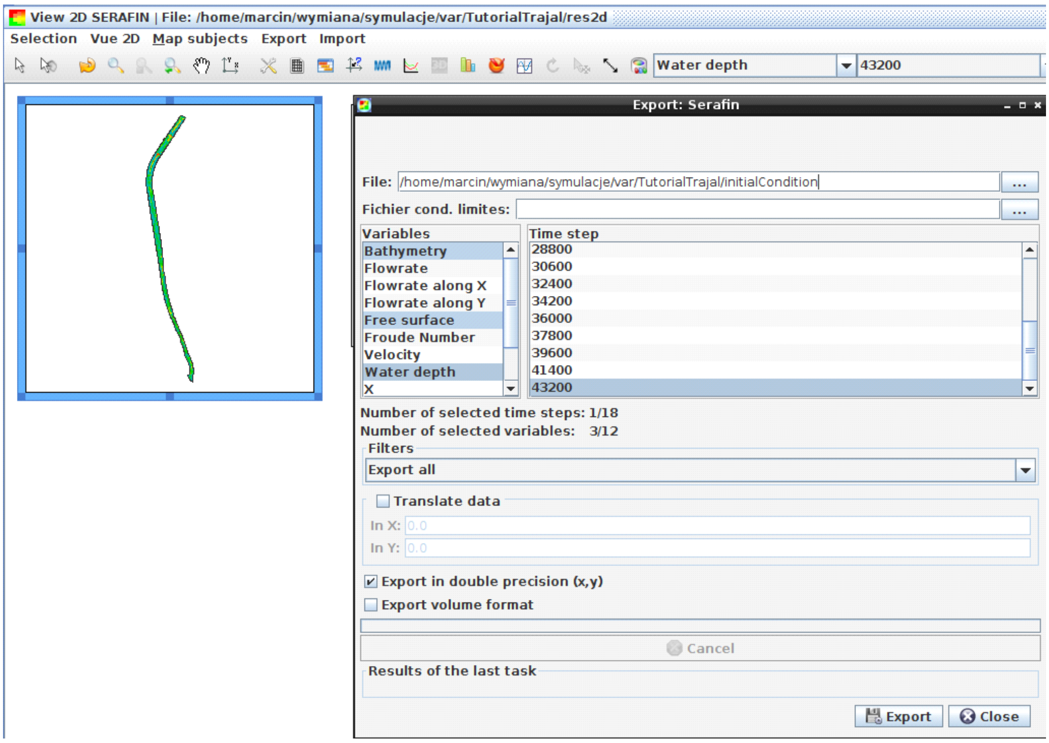


Figure 12: Creating initial condition, based on the results of previous simulations

As variables to be exported select Bathymetry, Free Surface, Water depth (hold Ctrl to select more than one variable). As time step, select the last time step, give a name for the file and click *Export.* Close Fuda Prepro post-processing window.

In Fudaa Prepro Supervisor window, right click on the mesh file (*geo1)* and create a new case scenario *cas2.* Follow the same steps as in your previous simulation, with one exception: on the upper boundary condition do not create sources. Instead, highlight boundary nodes (*Layers manager → Boundary conditions* in the right pane of the View 2D window), select nodes of the upper boundary. Right click on the highlighted nodes of upper boundary and from the drop-down menu select *Insert liquid nodes.* Choose type: *Liquid with prescribed flowrate* and insert constant flowrate 100m3/s.

In layers manager (right pane of the window) select *Initial conditions→ IC Nodes.* Right click and select *Enable Initial Conditions* from the drop-down menu. In the dialogue window choose *Initialization from a results file* and select the results file which you have just exported from your previous results. Click *Validate* to accept.

In general parameters window, make sure, that besides the parameters explained in the first part of this tutorial, also the following additional parameters are set:

Name	Value	Comment
COMPUTATION CONTINUED	true	Value indicates, that this is a continuation of previous simulation
INITIAL TIME SET TO ZERO	true	However, we would like our time coordinates to start again from the beginning
PRESCRIBED FLOWRATES	0;100	Flowrates assigned to each boundary 0 – at sea boundary will not be considered, 100 – discharge at upper boundary condition.
RESULT FILE	res2d_new	Name of the results file, in case your simulation is in the same folder as previous, make sure you do not overwrite its results

After setting all the parameters, save the case file. Transfer all the simulation files to computer with telemac2d and launch the simulation. After completing simulation, make sure you understand your results.

Non-steady boundary condition

```
# discharge at La Manda Bridge
# November 1994
T        Q(2)
s        m3/s
0        300
3600     320
7200     320
10800    300
```

Listing 1: Liquid boundary file, representing discharge time series at boundary 2

In order to use time series as flood hydrographs, you need to change your upper boundary condition. Open your case file in Fudaa Prepro and launch View 2D. Zoom in to upper boundary condition. In layers manager (right pane of the window), highlight *Boundary Conditions→ Boundaries* and right click on the upper boundary. Select *Edit the selected boundary* from the drop-down menu. Change flow rate type to variable in time and click *Validate.*

In general parameters window set the following keyword:

LIQUID BOUNDARY FILE = discharge.liq

Make sure that you assigned the discharge to proper boundary (it depends on the order the boundaries were created). The liquid boundary file is a self-explaining text file (listing 1.). Finally, adjust the simulation length to 48 hours, save the case file and run the simulation.

Troubleshooting

Here are some most common errors, which you may face during your first experiences with TELEMAC:

- ***file does not exist (cas.liq) for key LIQUID BOUNDARIES FILE*** *– due to bug in Fudaa, TELEMAC2D* is expecting liquid boundaries file, which is not necessary in very simple simulations with steady boundary conditions (like your first one). To fix this problem, edit the case file with a text editor and comment out or remove the line *LIQUID BOUNDARIES FILE*

- **I am looking for an FLOAT but found an inappropriate value set for keyword: VALUES OF THE TRACERS AT THE SOURCES** – due to bug Fudaa sometimes leaves empty keywords without values. This is unacceptable for TELEMAC2D. In such case fix the case file with text editor, either comment out the keyword with empty value or assign a value. For example:

  VALUES OF THE TRACERS AT THE SOURCES =;;;;

  should be replaced with:

  VALUES OF THE TRACERS AT THE SOURCES =0;0;0;0;0

  or commented out with a slash '/' at the beginning of the line

- **NO FRICTION LAW IS PRESCRIBED!** - bottom friction formula and friction coefficient have to be defined in case file

- **DEBIMP: PROBLEM ON BOUNDARY NUMBER 2**

  GIVE A VELOCITY PROFILE

  IN THE BOUNDARY CONDITIONS FILE

  OR CHECK THE WATER DEPTHS. - this message is usually a result of super critical flow at the boundary, thus water is entering dry domain. It usually indicates that your initial condition was not set up properly.