



Tutorial for Pamhydr2
January 2024

1D Modeling of the Hogneau River (Nord, France)

using Pamhydr2

INRAE Lyon-Grenoble Auvergne-Rhône-Alpes

RiverLy, river hydraulics

Authors :	Pierre-Antoine Rouby	pierre-antoine.rouby@inrae.fr
	Théophile Terraz	theophile.terraz@inrae.fr
	Lionel Pénard	lionel.penard@inrae.fr

Table des matières

1 Install Pamhyr2



Pamhyr2 can be downloaded from <https://gitlab.irstea.fr/theophile.terraz/pamhyr>.

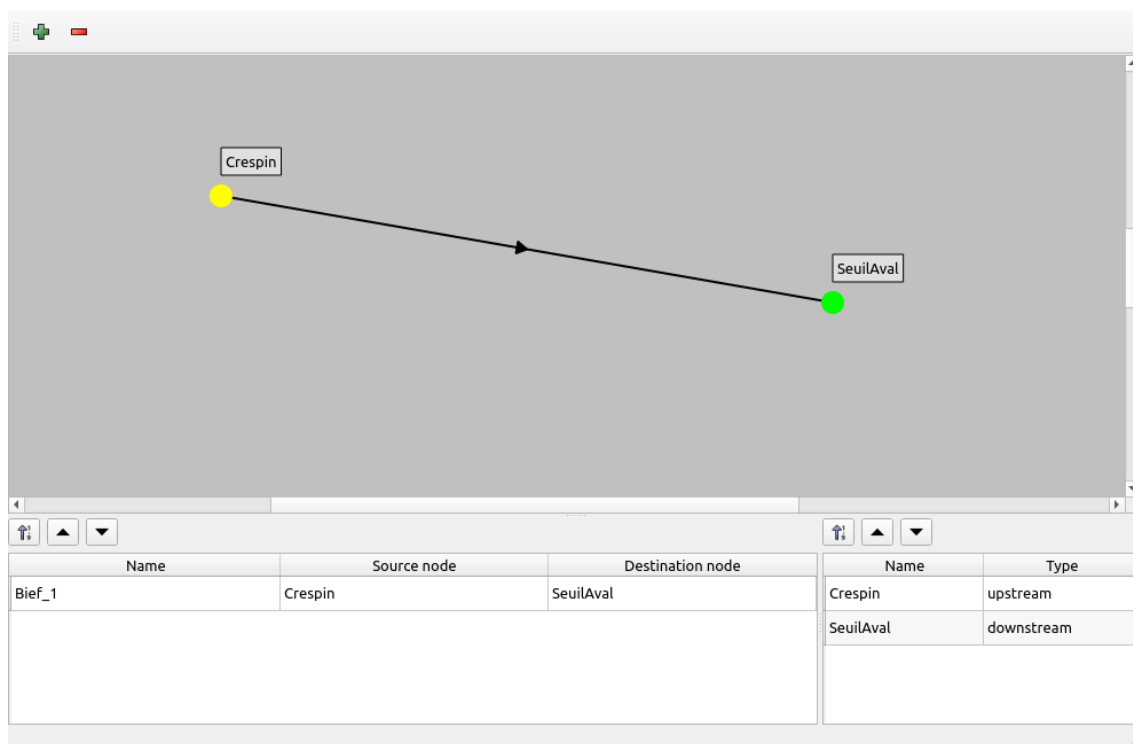


Use the GNU Linux or the Windows download button depending on your system. On windows, launch the installer. On Linux, unpack the archive and launch Pamhyr2.

2 Create your first study

On the main windows, click on [Files] => [New Study] to create a new study.

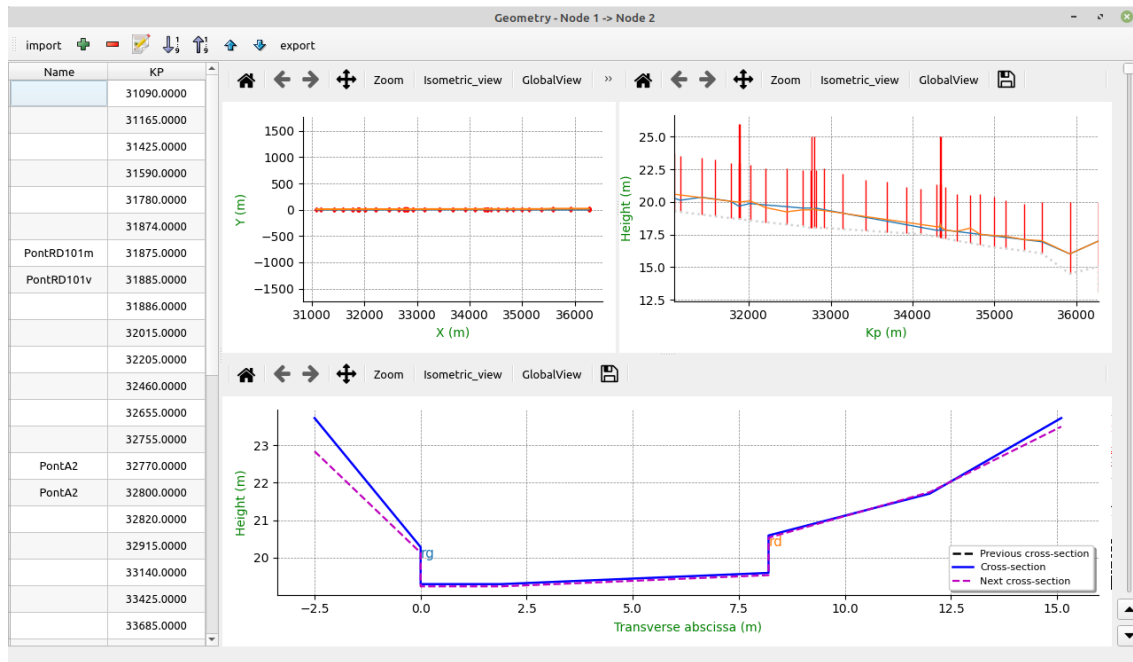
Click on [River Network] => [Edit River Network] to create the reaches of your river. In this window, you must define an oriented graph that represents the reaches of your river network : the edges are the reaches and the nodes are either upstream boundary conditions, downstream boundary conditions or junctions. Press the  to enter the *add* mode. Create two nodes by clicking in the grey zone of the window, and create a link by clicking again on each node. press  again to exit the *add* mode. You created your first reach, with an upstream node and a downstream node. In the lower part of the *Edit River Network* window you can rename the nodes and the reaches. As the reach you created is automatically selected, all the next steps will apply to this reach. The window should look like that :



Close the *Edit River Network* window.

3 Edit the river geometry

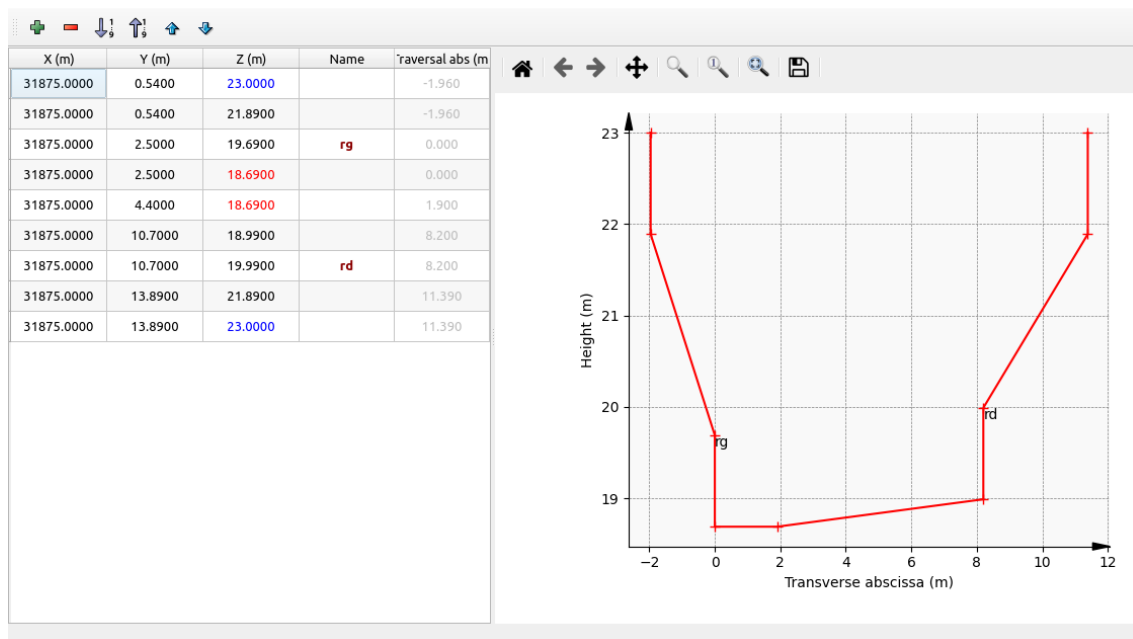
Click on [Geometry] => [Edit Geometry] to define the geometry of the selected bief. Click on the [Import] button and select the file Data/Bief_1.st. You should see :



On the left panel is a list of all the cross sections with their name and longitudinal abscissa. In the top left plot you can see the top view of the river, on the top right panel the longitudinal cross-section of the river and in the bottom plot you can see the selected cross-section (blue) along with the next one (dashed purple) and previous one (dashed black). You can move in the section list using the slider at the very right of the window.

You can edit the selected cross section by clicking on the  icon.

select the cross section named *PontrD101m* and open the edition window. You should see :

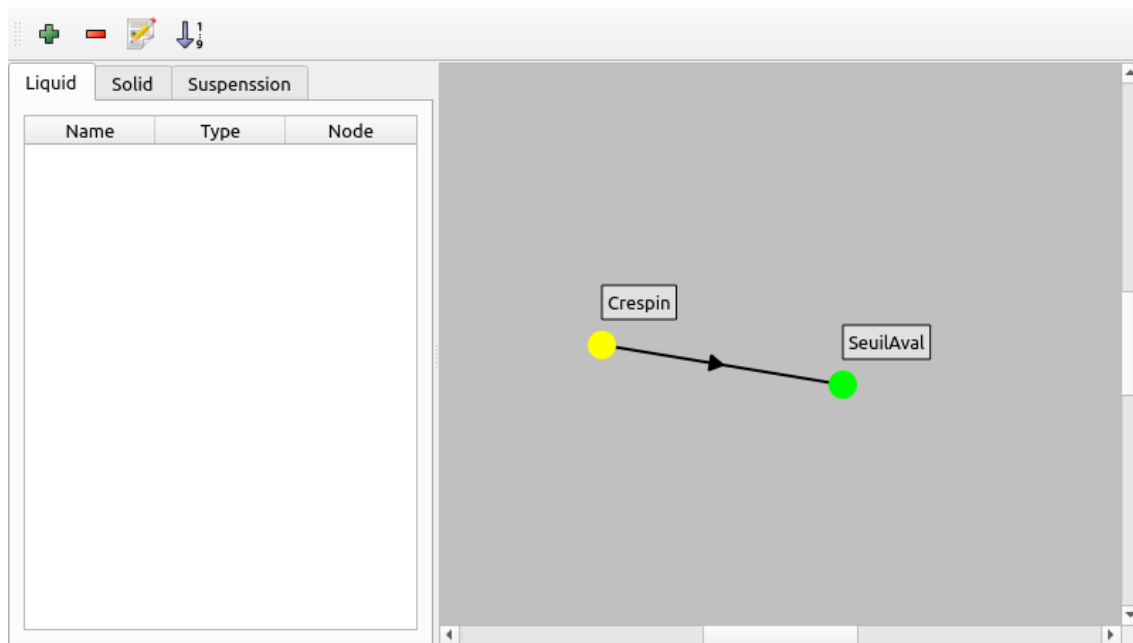



On the left panel is the list of all the points of the section, with their coordinates, their name and their transversal absisa. The Z coordinate of the highest point is written in blue and the lowest in red. Points can have a name. If a point with the same name exists in every section in a reach, it forms a longitudinal line. For example, here we have *rg* and *rd* which represent the left bank and the right bank of the main chanel.

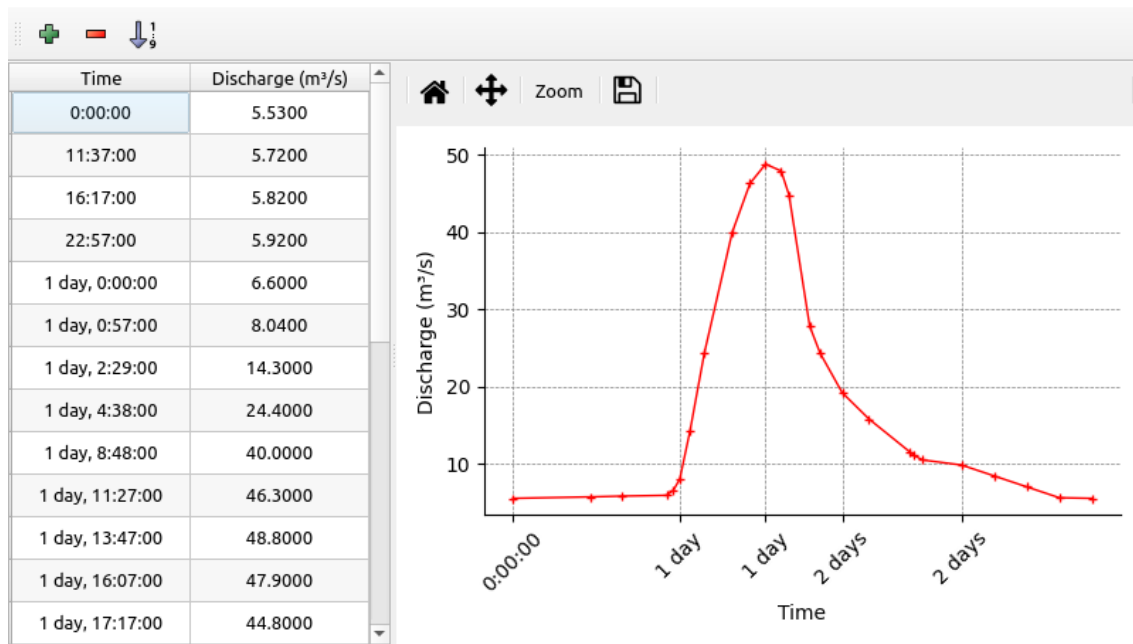
On the plot is a projection of the cross section. You can use [ctrl + click] to select a point in the plot and [shift + click] to select a water line and visualize usefull geometric data. You can close the cross section edition window and the geometry edition window.

4 Edit the boundary conditions

From the main window, click on [Hydraulics] => [Boundary conditions and punctual contributions]. You should see :



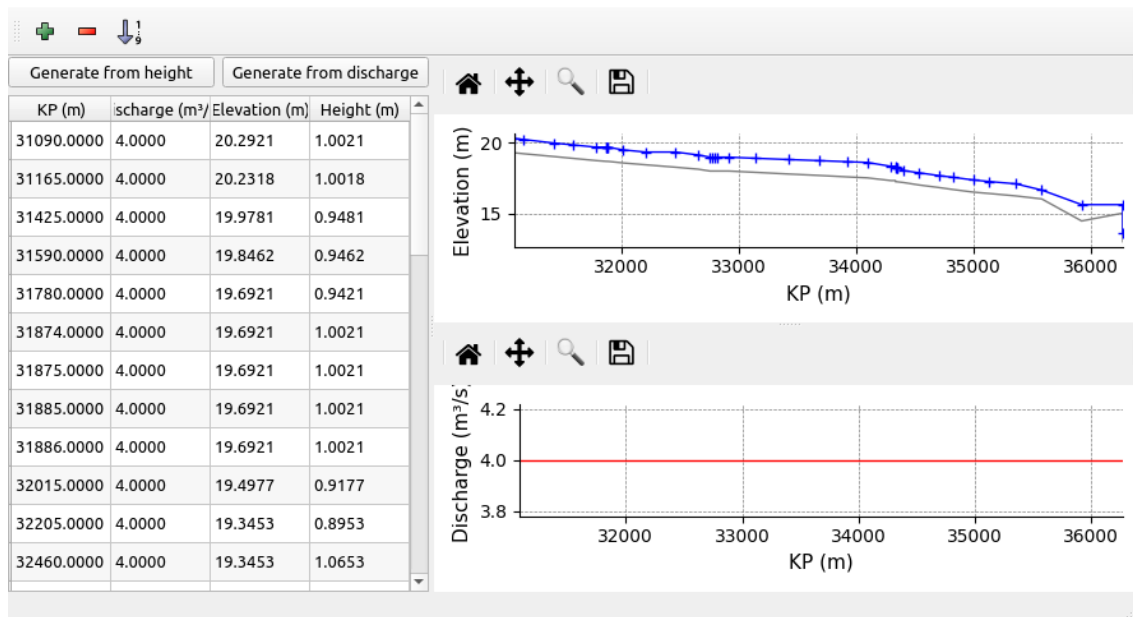
Use the *add* button on the top left of the window to add a liquid boundary condition. On the new line, click to select the whole line, double click to select the cell. Select the *Type* cell to give a name to the boundary condition. here, we will define the flow discharge mesured during the february 2002 flood. You can name this boundary condition "flood2002". Select the *Type* cell and use the combo box to put a Q(t) law. Select the *Node* cell and attribute this condition to the upstream node. Names of the nodes are recalled in network in the right panel. Now select the whole line and click on the edit button . You opened the *Edit Boundary Conditions* window. On a text editor, open the `Data/Fevrier_2002.txt` file. Copy the content of the file (for example with *ctrl+a ctrl+c*) and paste it in the left panel of the *Edit Boundary Conditions* window with *ctrl+v*. You can now see the flow discharge curve :



Close this window. Go back on the *Boundary Conditions* window. Add a new line, give it a name, give it the *textitZ(T)* type and associate it to the downstream node of the network. Open the *Edit Boundary Conditions* window (✎). Add two lines. In the first one, enter time : 0.00.00 and Z : 15.000. On the second one, time : 1.00.00 and Z : 15.000. It creates a constant downstream water elevation. For the computaion, Mage will extrapolate continuously the water elevation, that's why we only need to define one hour. You can close the the *Edit Boundary Conditions* and the *Boundary Conditions* window.


5 Create initial conditions

From the main window, click on [Hydraulics] => [Initial conditions]. If you don't know the initial conditions in water elevation and flow discharge of the river, you can use [Generate minimal height] or [Generate from discharge] buttons to let Pamhyr2 estimate an initial condition using the Manning-Strickler formula. Click on [Generate from discharge] and enter a discharge of $4m^3$ in the pop-up window to generate an initial water height condition based on the Manning-Strickler formula and a uniform discharge of $4m^3$. You should see :




Close the *Initial conditions* window.

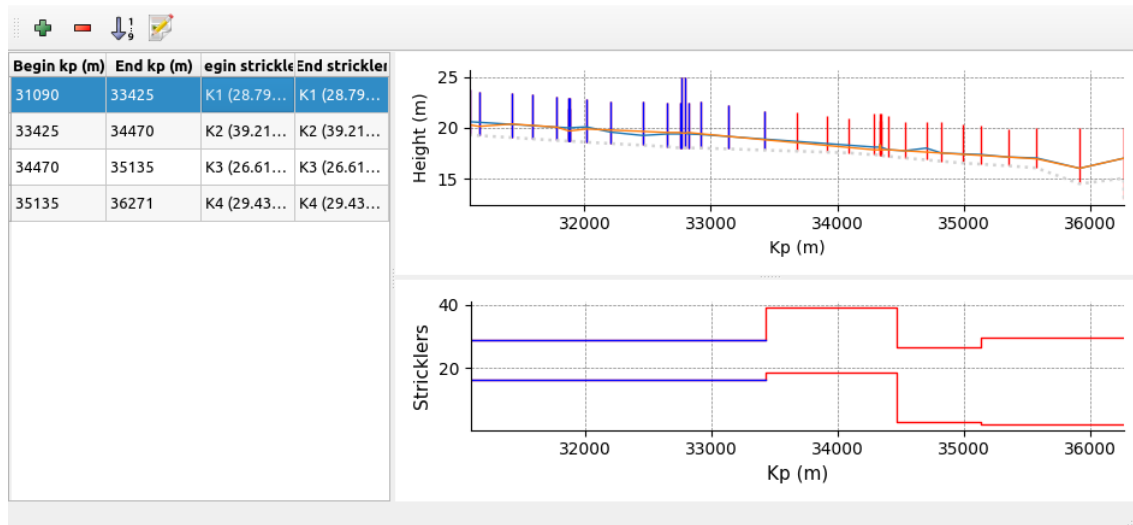
6 Edit friction coefficients

From the main window, click on [Hydraulics] => [Edit friction]. You first have to define sets of Strickler coefficients. Click on  to open the *Strickler* window. Here you can create couples of Strickler coefficients, the first one for the minor bed, the second one for the medium bed. Click on *add* four times to create four new couple. Give them the following values :

Study stricklers			
Name	Minor bed	Medium bed	Comment
K1	28.79	16.383	
K2	39.211	18.707	
K3	26.613	2.983	
K4	29.436	2.303	
Application stricklers			
Name	Minor bed	Medium bed	Comment


Close the *Strickler* window. On the *Edit friction* window, add four lines with the button  to create four friction zones. Each zone is defined by a *begin* and *end* KP and a *begin* and *end*

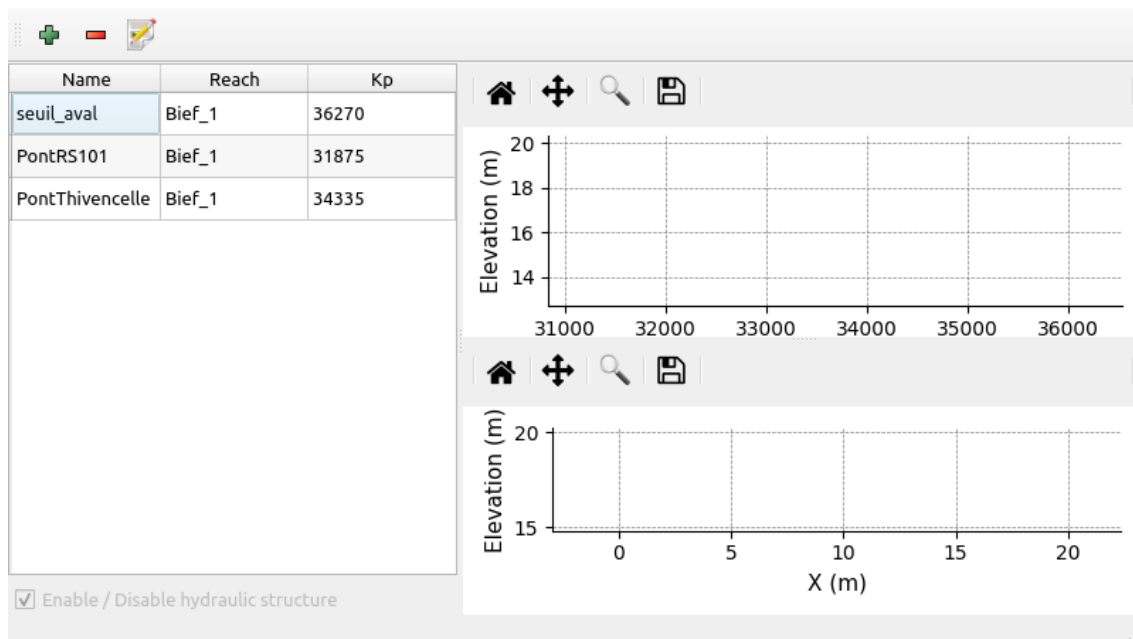
Strickler couple. The strickler coefficient couples inside a zone are interpolated from the *begin* and *end* couples. In our case, we will use constant coefficients per zone. Set the zones as follow :





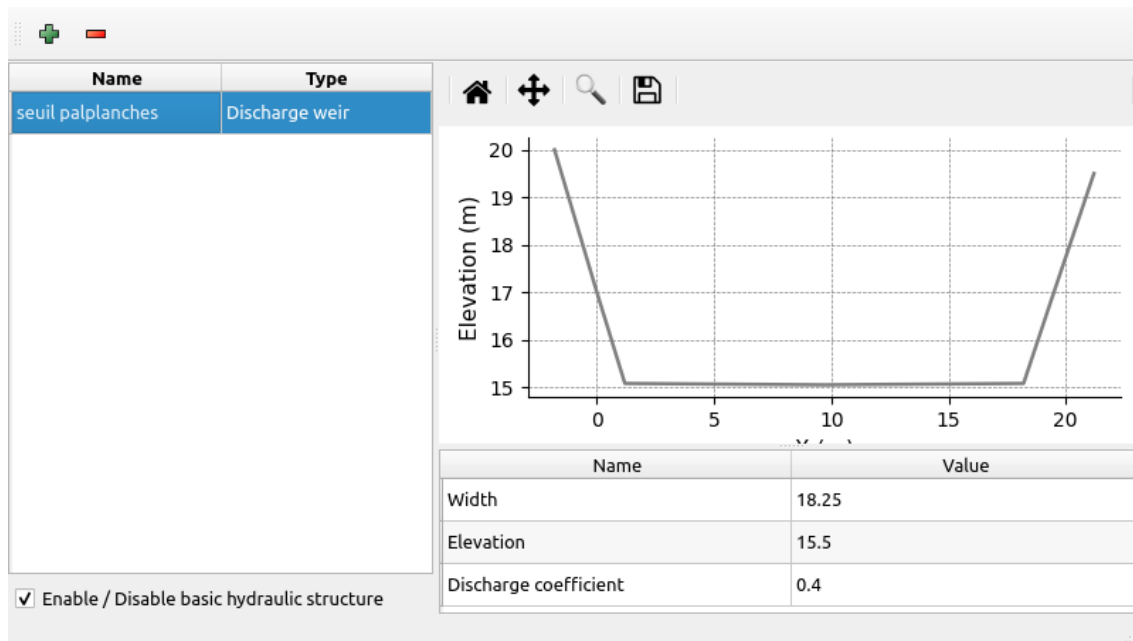
The selected zone is highlighted in blue. Close the *Edit friction* window.

7 Model hydraulic structures


Sometimes there can be cross-sections in which Shallow water equations can not be used to model the water flow. In that case, we have to define an other law to link the water elevation and the flow discharge. This is the case, for example, under bridges when the water elevation is too high, leading to a flow in charge. Pamhyr2 enables to define various hydraulic structures with laws that can be parametrized. In our case, a weir and two bridges have to be represented as hydraulic structures. From the main window, click on [Hydraulics] => [Hydraulic structures] to open the hydraulic structures window. Click tree times on the  button to create three hydraulic structures. Each structure can have a name and must have a reach and a KP. Set them as follow :

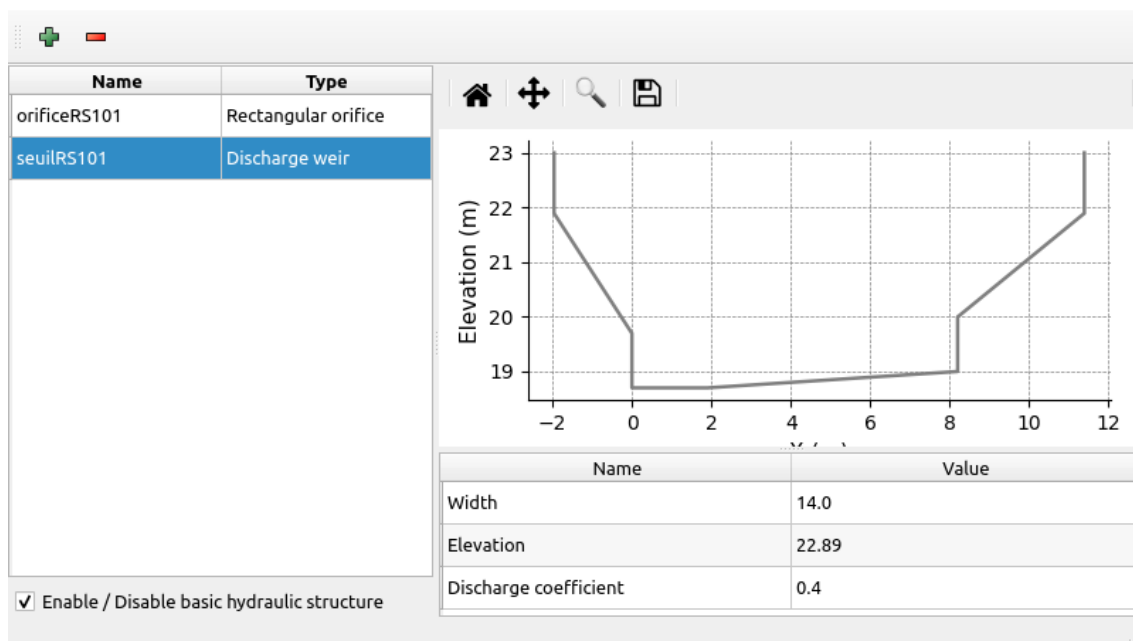


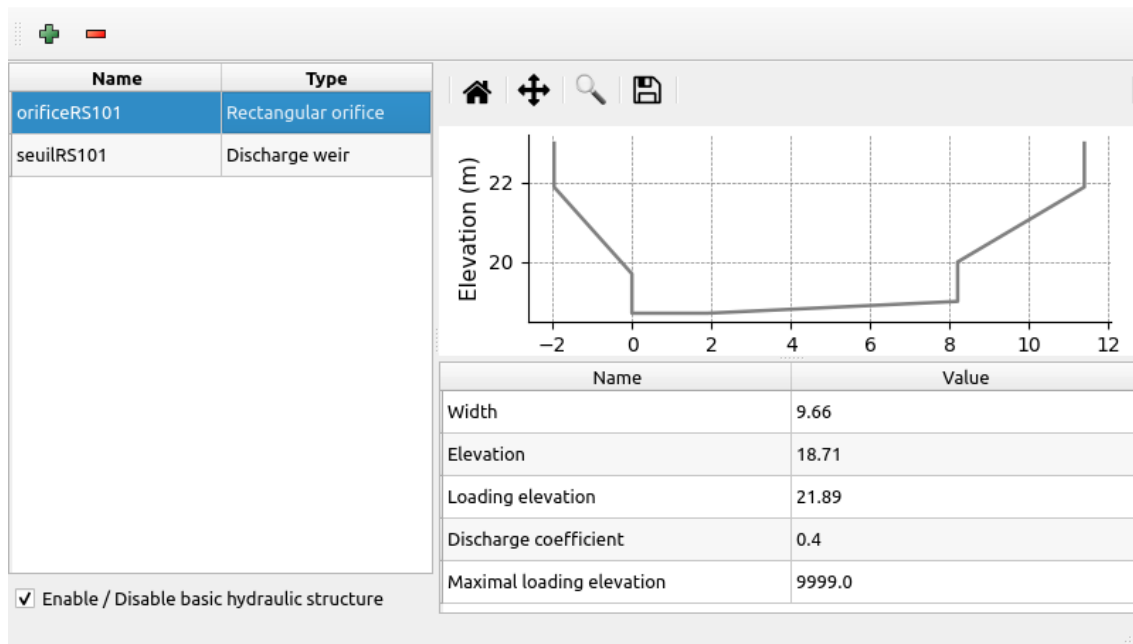
Select the downstream weir and click on  to edit the laws of this structure. Hydraulic structures are composed of basic hydraulic structures. You can combine the laws of several basic hydraulic structures to setup your structure. For this weir, we only need a weir basic hydraulic structure. Click on  to add a new basic hydraulic structure, give it the *weir* type and set it up as follow :



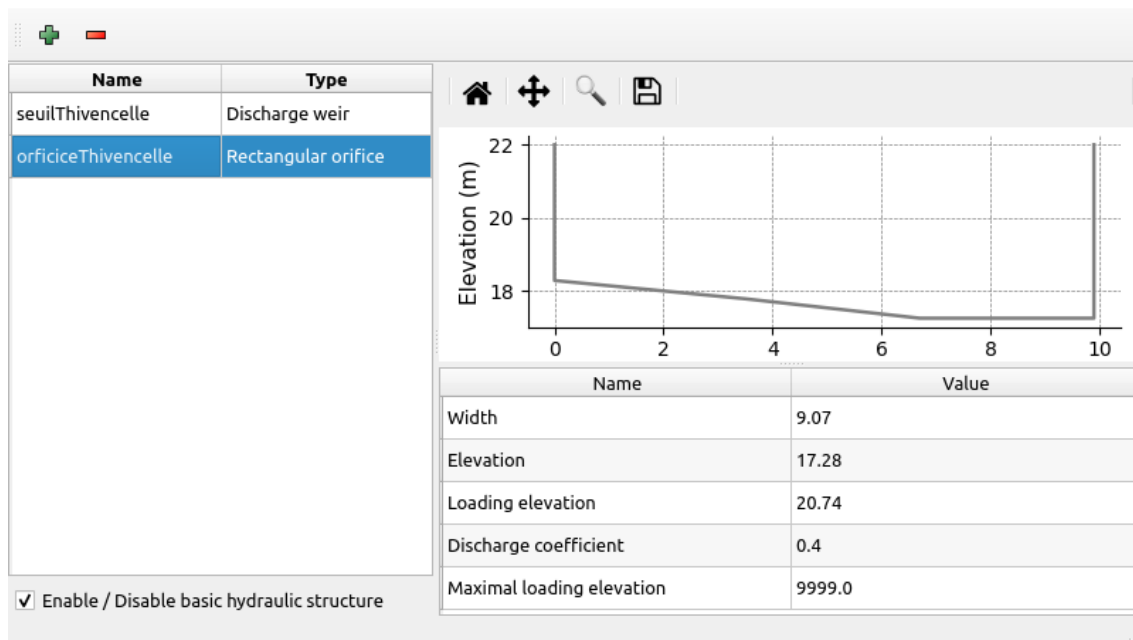
Go back to the *hydraulic structures* window.

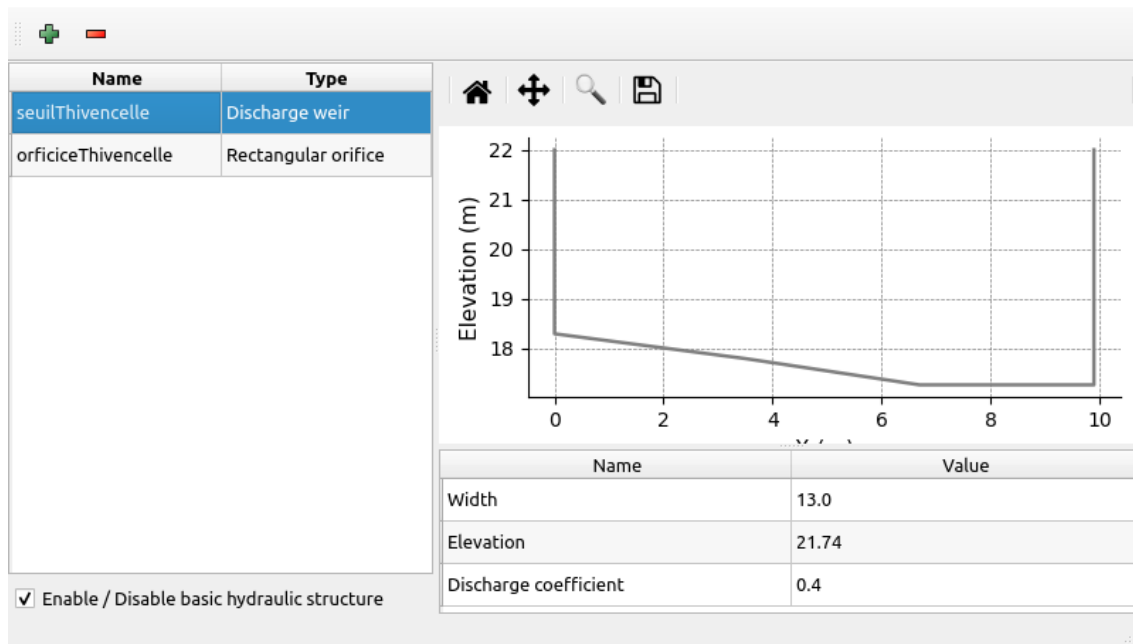
Select the RS101 bridge and click on  to edit the laws of this structure. A bridge can be modeled as a combination of an orifice for the flow under the bridge and a weir for the flow over the bridge. Create two basic hydraulic structures and set them as follow :





Go back to the *hydraulic structures* window and apply the same procedure for the Thivencelle bridge :





You can now close the *hydraulic structures* windows.

8 Solver parameters

From the main window, click on [Execute] => [Numerical parameters for solvers]. In the window *solver parameters* select the *Mag v8* tab. Set the solver parameters as follow :

Generic Mage v8	
Name	Value
Initial time (jj:hh:mm:ss)	000:00:00:00
Final time (jj:hh:mm:ss)	999:99:00:00
Timestep (second)	300.0
Command line arguments	
Minimum timestep (second)	0.01
Time step of writing on .TRA	3600
Time step of writing on .BIN	0
Implication parameter	0.65
Continuity discretization type (S/L)	S
QSJ discretization (A/B)	B
Stop criterion iterations (G/A/R)	R
Iteration type	-1
Smoothing coefficient	0
Maximum accepted number of CFL	10000
Minimum water height (meter)	0.1
Maximum number of iterations (< 100)	15
Timestep reduction factor	2
Reduction precision factor of Z	10
Reduction precision factor of Q	10
Reduction precision factor of residue	1
Number of iteration at maximum precision	4
Number of iteration before switch	99
Maximum accepted Froude number	2
Diffuence node height balance	-1
Compute reach volume balance (Y/N)	Y
Maximum reach volume balance	0.001
Minimum reach volume to check	1000.0
Use Mage internal initialization (Y/N)	Y


Close the *solver parameters* window.

9 Run the simulation

From the main window, click on [Execute] => [Run solver]. Select *Defaut-Mage - (Mage8)* and click on the *Run* button. It will open two windows : the *Check list* window and the *Solver log* window. The *Check list* window gives som hints about the validity of your model, and the *Solver log* window displays the outputs of the solver. From the *Solver log* window you can re-run the computation, and from the *textitSolver log* window you can click on the *Results* button to open the *Results* window.

10 Visualize the results

If you closed the *Solver log* window, you can click on [Results] => [Visualize last results] from the main window to open the *Results* window. The top lets panel let you select your reach, the

bottom left panel lets you select a cross-section in that reach. the three plots on the right show the reach and the cross-section the same way than in the *Geometry* window. You can use the bottom slider to visualize the water elevation at different timesteps. To visualize the flow discharge, switch to the *Hydrograph* tab. To create custom 2D plots, click on the  button on the top left of the window. Select the values you want on the X and Y axis and click on *OK*. You can now see a new tab with the custom 2D plot in the right panel of the *Results* window.