MASCARET v8.0

Application Guide

EDF - SA au capital de 924.433.331 euros - R.C.S. Paris B 552 081 317
CEREMA - Centre d'études et d'expertise sur les risques, l'environnement,
la mobilité et l'aménagement

Table of contents

Contents

1 Introduction

1.1 Modeling system

The one-dimensional software `MASCARET` 7.1 is designed to simulate flow propagation in open channels. The software supports steady and unsteady flow computations through three computational kernels :

- Steady flow: this kernel can handle subcritical, supercritical and mixed (i.e. hydraulic jumps) flow regimes in a single river reach or a dendritic system. Lateral inflows and structures, such as transverse/lateral weirs and spillways, may be considered;

- Unsteady subcritical flow: this component of the Mascaret modelling system is capable of simulating unsteady subcritical flow through a dendritic system or a full network of channels (i.e. river reaches split apart and then come back together, forming looped systems). Lateral inflows and structures may be considered;

- Unsteady trans-critical flow: this component is intended for computing mixed flow regime through a dendritic system or a full looped network of channels, including lateral inflows and structures. It was developed for simulating flood wave propagation (induced by dam-break waves for instance).

The objective of the `MASCARET` 7.1 system is to compute surface elevations, discharge and velocities at all locations of interest for either a given set of flow data (steady flow simulation), or by routing hydrographs through the system (i.e. unsteady flow simulation). The data needed to perform these computations are: geometric data (topography of the open channels and characteristics of the hydraulic structures), network description and boundary conditions (discharge data at the upstream end and additional conditions at the upstream/downstream ends depending on the flow regime).

1.2 Modeling Capabilities

The following table summarizes the modeling capabilities of `MASCARET` 7.1.

Capability	Steady kernel	Unsteady subcritical kernel	Unsteady trans-critical kernel
Trans-critical (i.e. hydraulic jump)	YES	NO	YES
Propagation on dry zones	NO	NO	YES
Unsteady flow	NO	YES	YES
Steady flow	YES	By convergence of an unsteady flow	By convergence of an unsteady flow
Non-hydrostatic terms	NO	NO	YES
Network (or looped) river system	NO	YES	YES (local 2D computation limited to 3 reaches per junction)
Dendritic river system	YES	YES	YES (local 2D computation limited to 3 reaches per junction)
Friction laws :			
- Manning-Strickler	YES	YES	YES
- Bazin	YES	YES	YES
- Colebrook	YES	YES	YES
Compound cross-sections :			
- Debord model	YES	YES	YES
- Bed-Bank model	YES	YES	NO
Storage areas	YES	YES	NO
Progressive overflow :			
- Main river	YES	YES	YES
- Storage areas	YES	YES	NO
Singularities:			
- Upstream weir f($Z_{upstream}$,q)	YES	YES	NO
- Unsubmerged weir	YES	YES	NO
- Weir defined by geometrical data	YES	YES	NO
- Weir defined by Q=f($Z_{upstream}$, coef)	YES	YES	YES
- Weir defined by h=f(t)	YES	YES	NO
- Weir defined by Q=f($Z_{upstream}$)	YES	YES	YES
Weirs taken into account in the geometry	YES	NO	YES
Singular head loss	YES	YES	YES
Lateral weir	YES	YES	NO
Lateral inflow	YES	YES	YES
Constraints on the time step	NO	NO	YES (CFL < 1), NO if implicit computation
Boundary conditions			
- Fixed level	YES	YES	YES
- Fixed flow	YES	YES	YES
- Rating curve	NO	YES	YES
- Fixed level and flow	NO	NO	YES
- Free outflow	NO	NO	YES

Table 1: Characteristics of the three computational kernels

The three kernels may be used for simulating solute transport processes, consisting of advection, diffusion and mass reduction/generation by physical, chemical and biological mechanisms. The flow and water quality equations are solved sequentially at each time step. The flow routing module of the MASCARET system is called first to provide the time-dependent hydraulic parameters throughout the model domain, and then these variables are passed to the water quality module for the solute transport simulation. However, simulation of water quality is only possible in a dendritic system formed by open channels without floodplains and lateral weirs (i.e. absence of diversion).

1.3 Install process

FUDAA-MASCARET software is the Graphical User Interface written in the Java language for MASCARET. It is distributed as a Java setup file (*.jar* file). Please, make sure that your computer has a version of the Java Runtime Environment (JRE 1.5 or 1.6) before installing FUDAA-MASCARET. Please do not use a version of JRE greater than 1.7. See the web site "http://www.java.com" if you need to install JRE on your computer.

To begin the installation, first uninstall all previous versions of the software and then click on the executable file "*Fudaa_Mascaret_Setup.jar*".

Even if FUDAA-MASCARET can be installed and used on various operating systems, the setup file only contains the Microsoft Windows 32 bits version of the computation code. In order to do calculations with the GUI on other systems, the user has to manually change the given executable in the working directory of software with the one compiled on his own system.

1.4 Working directory

If the permissions are given by the administrator, the user can install the software in the default location ("*C:\Program Files\Fudaa-Mascaret*" for example). This path is not restrictive and another location can be chosen.

Suppose the install directory is identified by "*~\Fudaa-Mascaret*", the working directory of MASCARET is "*~\Fudaa-Mascaret\serveurs\mascaret \mascaret_7_0*".

In this directory, you will find the binary file of computation code "*mascaret.exe*" plus some ASCII files describing the case study (geometry, numerical parameters, initial conditions, etc.). These data files are created by the GUI and read by the code each time the user clicks on the *Compute* button.

After a successful calculation, the results in binary or ASCII files written by the code in the working directory, are automatically read by the GUI to perform the post-processing phase.

1.5 Support

For any problem or question on `FUDAA-MASCARET`, please contact the authors by e-mail : *ot-consultancy@opentelemac.org*

2 Application - Steady Flow over a Weir

2.1 Purpose

The `MASCARET` system is applied to simulate a steady flow over a weir. The weir is represented as a cross-section. Changing the boundary conditions, various flow regimes can be tested: sub-critical flow, trans-critical flow with or without hydraulic jump. Herein, the two last configurations are presented.

Numerical runs are performed using the transcritical flow kernel. The steady state is reached by a converged analysis.

Input data are given in the directory permanent_weir.

2.2 Transcritical flow without hydraulic jump

Run the user interface `FUDAA-MASCARET`. Select **Hydraulics** and then **Computation Kernel**. Choose the trans-critical computation kernel and confirm by clicking the  icon. Remark : This simulation can be performed by the computation kernel dedicated to steady flow.

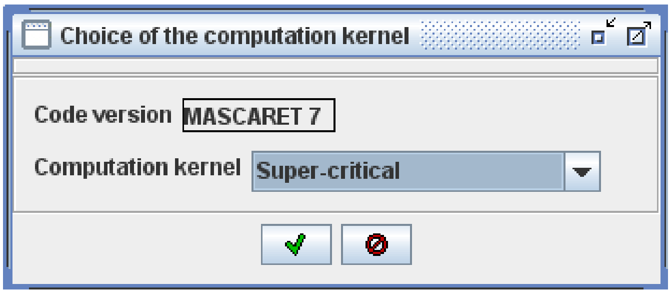


Figure 1: Selection of the computation kernel in `MASCARET`

2.3 Geometry

The channel is straight, 25 m long, 1 m width with rectangular cross-sectional shape. A parabolic weir profile, 0.2 m height, is located between $x = 9\ m$ and $x = 12\ m$. The longitudinal bed profile of the channel is described as :

$$\begin{cases} x < 9 \Rightarrow z_f = 0 \\ 9 \leq x \leq 12 \Rightarrow z_f = (x-9)(12-x)/11.25 \\ x > 12 \Rightarrow z_f = 0 \end{cases}$$

No overflow is considered.

2.3.1 Create the Hydraulic Network

From the main window of `FUDAA-MASCARET`, select **Hydraulics** and then **Edit the Hydraulic Network**. This will display the network window, which can be toggled on or off by clicking the Interactive Mode icon . If the interactive mode is toggled on, icons of the Hydraulic Network Editor are activated, enabling the creation of the hydraulic network.

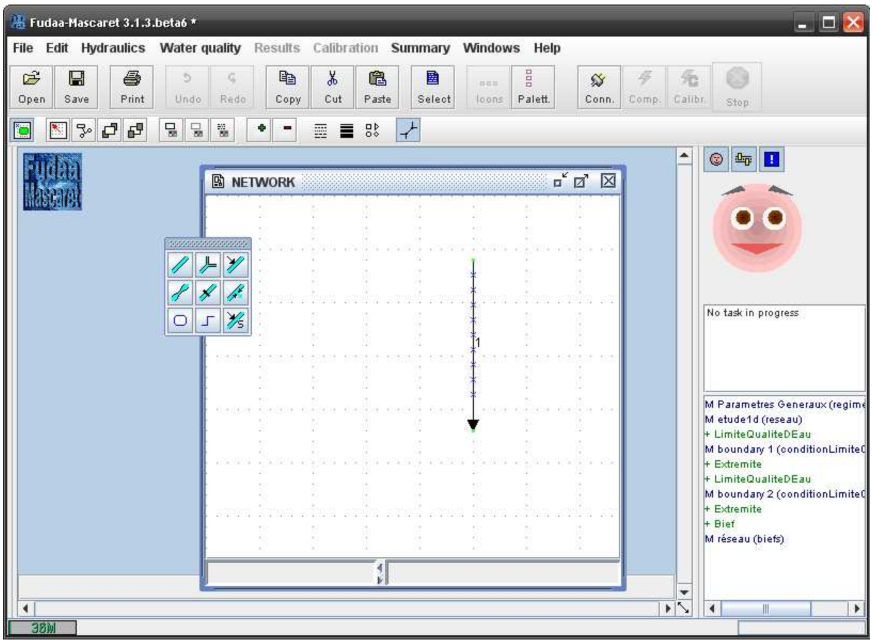


Figure 2: Hydraulic network : the window shows the river system schematic and network editor with its various components (reach, junction, lateral inflow...)

The channel is represented by a single reach. Click the icon . A window will display, showing different components of the hydraulic network. Click the icon - reach), a single reach is then drawn on the network window.

Because no additional components are needed in this case study, toggle off the interactive mode. The Network window shows the reach with ID 1 and open boundaries 1 and 2 (Figure 3)

Next, data describing the geometry of the channel (cross-sections) and weir should be entered. Move the cross in the network window and click on the reach. This activates the window that describes the channel geometry. You can either enter data nodes by nodes or import a geometry file (file *bottom.pro* from the example directory).

2.3.2 Manual Cross-Section Input

In the window describing the reach, you should enter all points defining the cross-sectional geometry. To create a cross-section, enter its name in the column “title” and abscissa in the second column. Then, click the “Create”

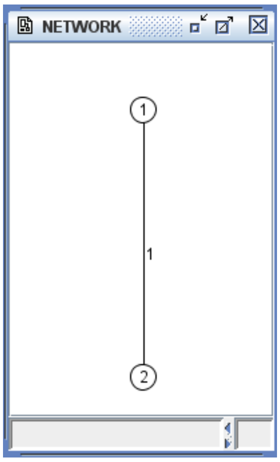


Figure 3: A single reach in the hydraulic network editor

button . A window will display asking for the shape of the cross-section. In the present test we need a simple geometry with vertical walls. Click on the  icon to create a user defined cross-section.

The following window "Edit Profile" allows entering the nodes defining the cross-section. Enter four nodes with bottom level 0 m, width 1m and wall height 100 m and validate by clicking on the  icon. Ignore the warning about vertical walls.

The next cross-section to create will be located just upstream from the weir at the abscissa 9 m. Create this cross-section by following the previous steps. Enter the bottom level 0.0 m. Then, create cross-sections describing the weir with space step of 0.1 m; the bottom level is given by $0.05(x - 9)(12 - x)$, in the zone between abscissas 9 m and 12 m. Finally, enter the bottom level 0 at the abscissa 25 m.

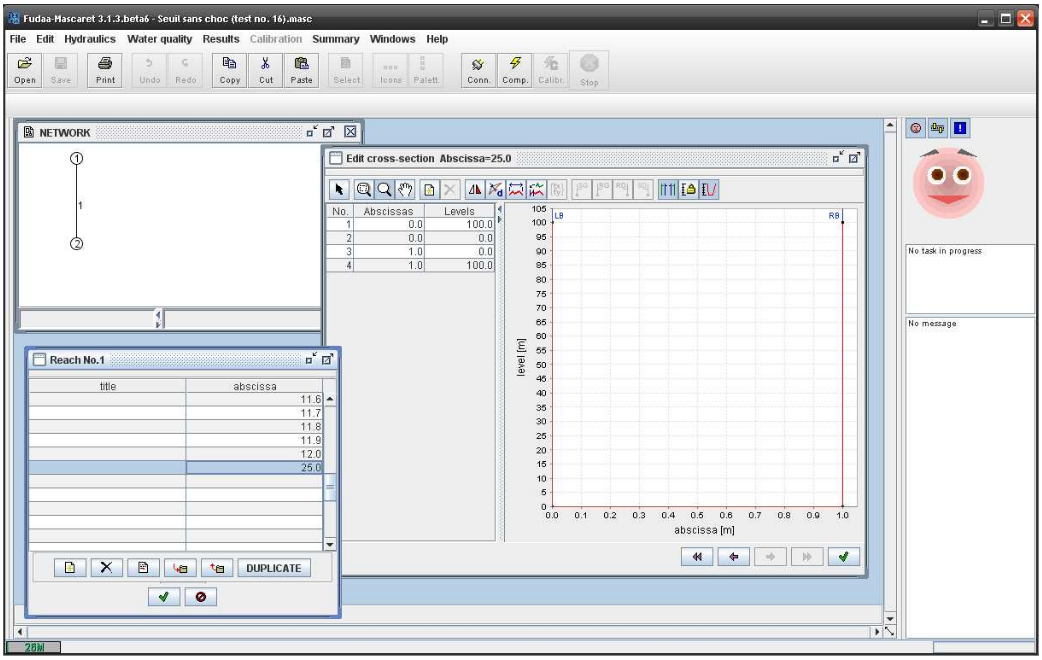


Figure 4: Editing the cross sections

2.3.3 Automatic Cross-Section Input

You can define the channel geometry by importing the file *geom_en.xls* (see the example directory). First, choose the Import icon from the window describing the reach and cross-sections. Then select the Excel file format and then *geom_en.xls* from the example directory.

To view a cross-section, click on the line in the table describing the reach and then on the Edit icon . A window will display, showing the selected cross-section. Using the arrows, you may remove to the next cross-section or come back to the previous one. Close the window and ignore the warning about vertical walls.

2.4 Boundary Conditions

Boundary conditions are set by clicking on the circle representing the open boundaries. The upstream boundary is the circle number 1. Click on this circle to display the boundary condition editor.

The upstream boundary condition is a constant inflow of 1.53 m^3/s. In the boundary editor, select "**boundary condition supplied by graph**". Then click on the "**EDIT GRAPH**" icon. The "**Graph Manager**" window appears. Click on the "**Create**" icon. Select a new "**Flow Hydrograph Q(t)**". The "**Hydraulic Graph**" window displays a spreadsheet and a graphical view of the data. Define a constant flow hydrograph by entering time 0 s and the large times of 10000 s. This last time must greater than the final simulation time. For both times, enter the same flow discharge of 1.53 m^3/s. The graph on the right part is then updated, showing the flow as a function of time.

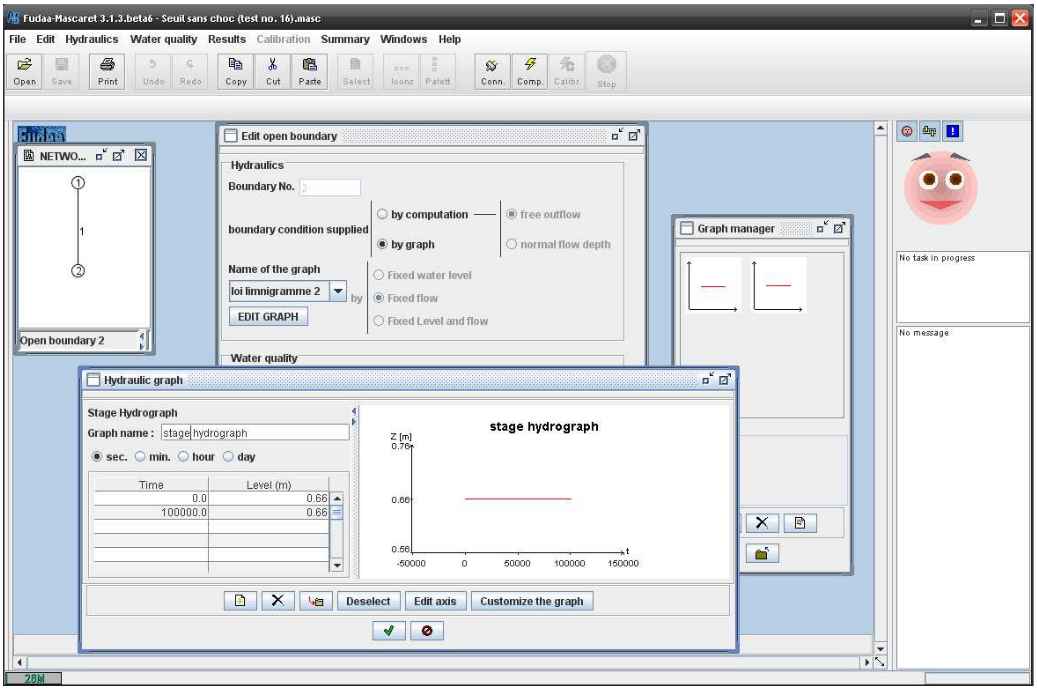


Figure 5: Flow hydrograph for upstream boundary condition

The downstream boundary condition is set as a constant water level of 0.66 m. In the network window, click on the circle 2. Select "**boundary condition defined by graph**" in the boundary editor, and then click on "**EDIT GRAPH**". Create a stage hydrograph by entering the water elevation (0.66 m) for two times : 0 s and 10000 s.

2.5 Mesh parameters

Two types of Mesh parameters should be set : the computational cross-sections and the transverse discretization of the cross sections.

Computational nodes may be added either by interpolating the user entered cross-sections, by using a grid map (fixed mesh size by reach), by entering them or by recovery from a previous computation. Here, we use a fixed mesh size for the whole reach. Select **"Hydraulics**" from the main menu, and then "**Mesh**". From the scroll-down list, select "computation cross sections from a grid map". Click on the icon **"Edit computation sections**", a spread-sheet appears. Then enter a constant grid size of 0.25 m for the reach $\#$ 1 (upstream abscissa 0 m, downstream abscissa 25 m). This space discretization value is user dependent. However, a much finer grid is needed around the location of the weir. Two choices are available : defining a finer mesh for this zone, or use the entered cross-sections. To do this, just click "**Yes**" on the check-box **"Computation cross-sections on the physical ones**". This will add all user entered cross sections as computation cross-sections.

Next, the menu point "**Vertical Discretization of the Cross Sections**" becomes active. It is now possible to define the transverse discretization by means of a spread-sheet. In the studied case, the discretization is constant for the reach number 1, from abscissa 0 to 25. The space step is equal to 0.05 m. Some slightly other values can be chosen and tested by the reader.

Close all opened windows by confirming your choices.

2.6 Initial conditions

Select "**Initial Conditions**" from the "**Hydraulics**" menu. In this example, the initial water level is constant. Choose "**No**" for the point "**Restart the computation**", and then "**Yes**" for "**Presence of an initial water level?**". Click on the "**Initial water level**" icon. A spread sheet will display, where the initial water level profile should be entered. Here, a level of 0.66 m and a discharge of 1.53 m^3/s are used for the whole reach (abscissa 0 m to 25 m). Validate by clicking on the  icon.

2.7 Temporal parameters

The "**Temporal Parameters**" entry is now active in the "**Hydraulics**". Click on it. Set the initial time of simulation at 0 s, the time step is 1 s. For the super-critical kernel, you can define a variable time step depending on the Courant number . Activate the check-box and set the Courant number to 0.8 for an optimal convergence of the kernel. This Courant number limitation is due to the explicit scheme (see the theoretical note). On the right side of the window, some criteria for stopping the calculation are provided. In the present case, calculation should stop once the steady state is reached. Proceed by a first trial and then adjust the final time or enter directly 300 s. Validate by clicking on the  icon.

2.8 General Parameters

The "**General Parameters**" that display in the "**Hydraulics**" menu allow defining the friction parameters, dam breaking parameters, flood marks and some advanced options. In the case of flow without hydraulic jump, no progressive overflow is calculated, friction is not conserved on vertical walls. The friction law is given by Strickler's formulation. To define the friction coefficient, click on the "**Dissipation Zones**" icon. A spread sheet will display, asking for the friction coefficient in the main channel and floodplain. Here, only one friction zone is defined for the whole channel : reach # 1, upstream abscissa 0 m, downstream abscissa 25 m, main channel coef. 10000 $m^{\frac{1}{3}}.s^{-1}$, Floodplain coef. 10000 $m^{\frac{1}{3}}s^{-1}$. Validate by clicking on .

2.9 Output Parameters

In the "**Edit results parameters**" menu, frequency on the output results is defined. First, enter a name for your case study. For example, "weir no jump".

The **"Recording"** frame defines the number of data points to be stored in the output files. The first part of the frame is the time frequency. Recording will start with the time step 1 (there is a minor bug, you can NOT write the initial state at time step 0). Then, the frequency of writing results file should be defined. This frequency is defined in number of time steps. Using a variable time step, this is somewhat tricky, for a nice resolution of your graphs chose 50 here.

Results will be stored at all computational cross-sections.

The listing file should be written within the computation progress. Activate this check box. You can store other data in the the listing file. To do this, just activate the check-box.

The post-processor to choose is Opthyca. The Opthyca format writes results in ASCII mode. The other Rubens format does it in binary mode. Limit the files size as shown in the Figure 6.

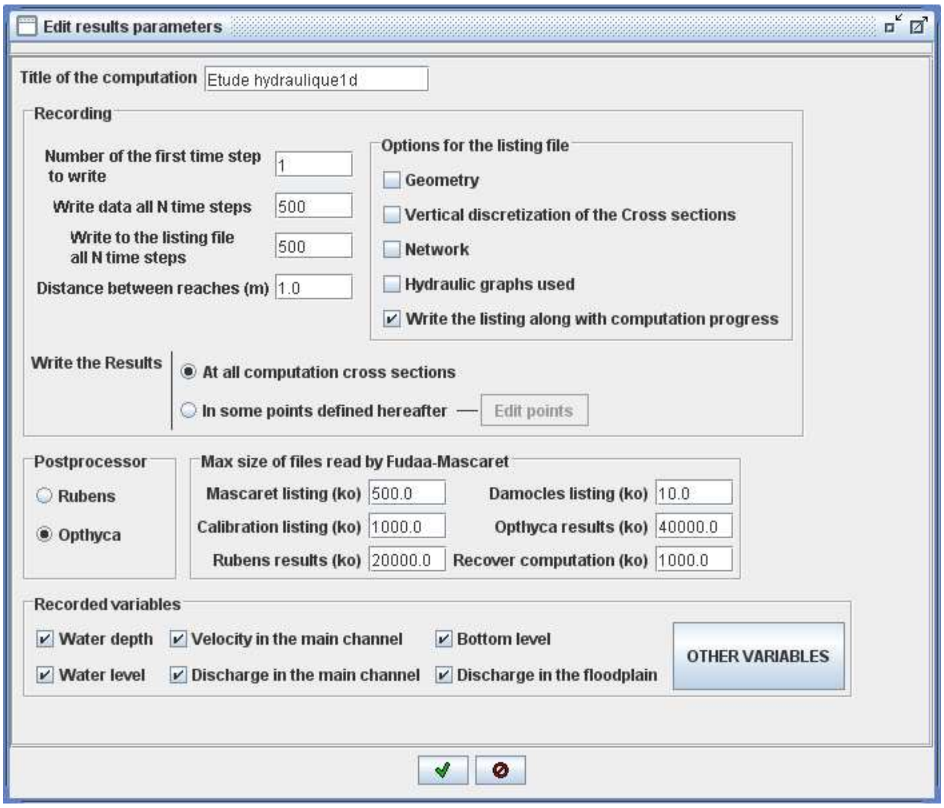


Figure 6: Output parameters for the application 1

The last point in the Results Parameter window concerns the Recorded Variables. Activate all variables down here, then click on the icon **"Other Variables"**. This opens a window with all variables that could be stored in the listing file. Activate the **"Froude Number"**.

Close all opened windows by validating the entries.

2.10 Results

Now, click on the computation icon 

of the main menu. The computation progress is shown in the status bar. Once the computation is finished, the **"General Results"** window is displayed. Check Warnings and Messages. Ignore all warnings and messages of Damocles.

The computation kernel should say "termine", and the last entry of the `MASCARET` listing should be "FIN" this means the computation was run without errors.

The aim of the simulation is a steady state, this means the flow discharge along the channel length should be constant and equal to the upstream discharge value. To check this, click on the menu entry **"Results"** and then on **"Graphs"**. A window will display to plot space and time profile variation of the selected variables. Check the total flow discharge as a function of time at the downstream cross-section of the model domain. Click on the **"Temporal Profile"** check-box. Choose the variable **"Total Discharge"** and last cross section (abscissa 25.0). Click on the  icon to plot the graph on the right side of the window. Results should be similar to the Figure 7. You can import the data to Excel.

Figure 7 demonstrates the convergence of the numerical run after about 50 s.

The next result to check is the final free surface profile. This profile will be compared to an analytical solution provided by [1]. To make comparison, the computed profile should be exported to Excel. Figure 8 shows a good agreement between the computed and analytical results. Also, a good agreement is found regarding the longitudinal profile of the flow discharge. Checking the space profile of the Froude Number shows the transition from the sub-critical to the super-critical regime around the weir location.

2.11 Flow with Hydraulic Jump

2.11.1 Parameters for the computation

Open the case study created for the weir without jump and save it under a different name (weir_jump.cas for example). The hydraulic network (a simple reach) appears on the NETWORK window. Click on the upstream boundary 1 to open the **Open Boundary Editor**. Click on the **NEW GRAPH** icon. Click on the existing flow hydrograph and edit the existing

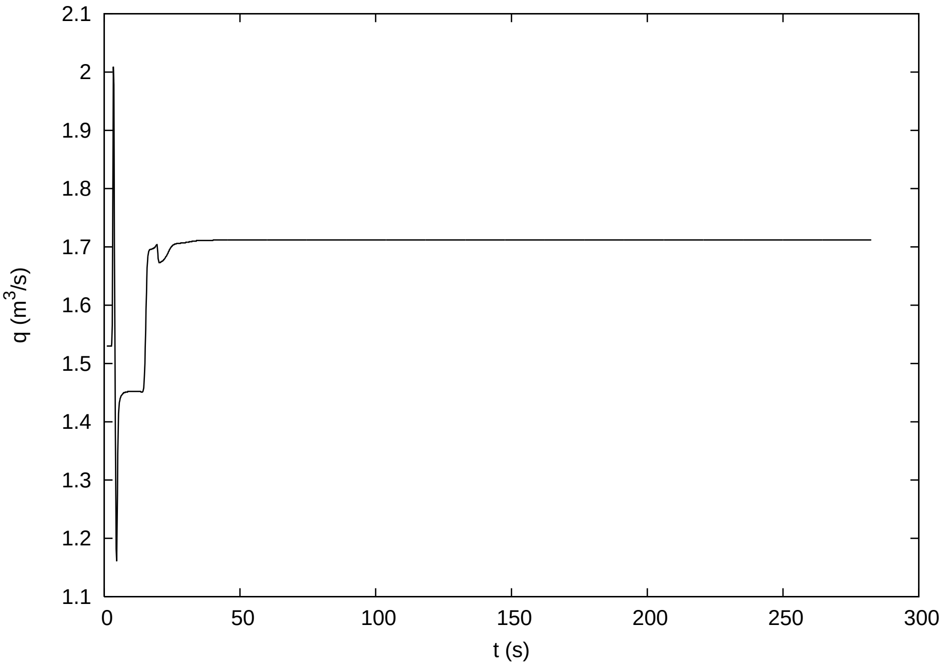


Figure 7: Convergence of the computation - the total flow hydrograph at the downstream end of the channel

graph, then change the constant inflow to 0.18 m^3/s. You can create a new flow hydrograph by clicking on the Create icon of the graph editor.

Set the downstream boundary condition as a constant stage hydrograph of 0.33 m.

Next, choose **Initial Conditions** from the main menu **Hydraulics**. Click on the **Initial Water Level** icon, and change the given values to 0.33 m and flow $= 0.18$ m^3/s. Confirm and click on **"Computation"**.

2.11.2 Results

After running the computation, check the general output (`MASCARET` listing and Warnings) for errors. If everything runs fine, you should read "FIN" at the end of the `MASCARET` Listing and "termine" at the end of the messages of the computation kernel.

Click on **"Results"** and select the **Graphs** entry. First, check the convergence of the computation by creating a temporal profile of the total discharge

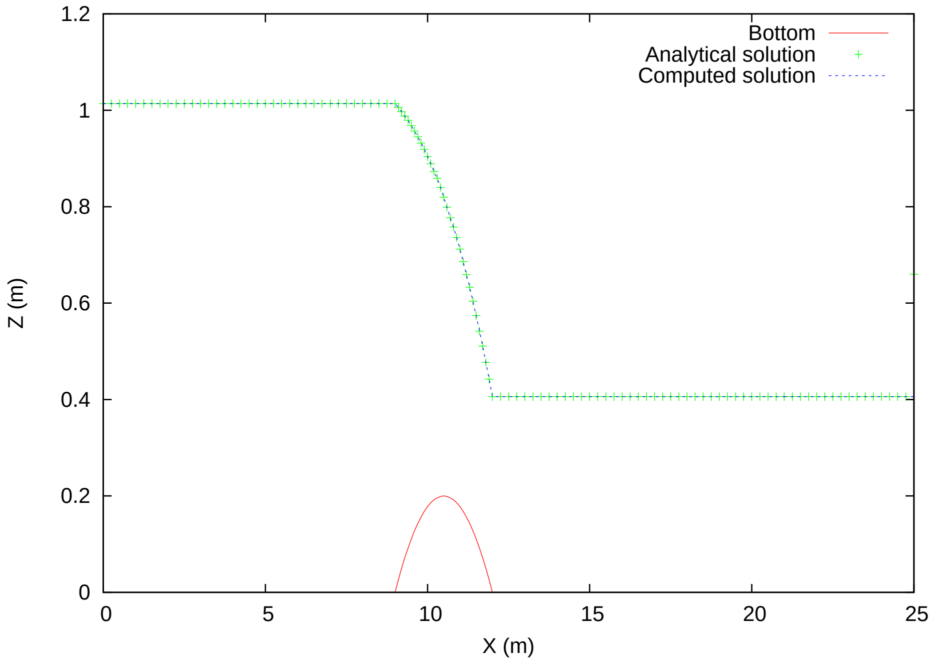


Figure 8: Water levels for the weir without jump: Comparison of analytical solution and computation

at the downstream end of the channel.

Figure 9 shows that the numerical run converges after about 150 seconds.

Note that this case study can be treated using the steady computation kernel of `MASCARET`. In this case, no initial water level is used, and only two time steps are necessary : the first one is used for the initialization, while the second one is used for computing the steady state. Save the present case study file under the name weir_steady.cas. Click on **Hydraulics**, and choose the menu entry **Computation kernel**. Select **"Steady"** from the scroll-down list, then confirm your choice. A window will display, imposing some changes due to the computation kernel (fixed time step). Now, choose the **Temporal Parameters** (in the **Hydraulics** menu). Set the time step to 10000 s and number of time steps to 2. Finally, select the menu **"Output Parameters"** from the results window and set the output frequency to 1.

Confirm all choices and re-run the test.

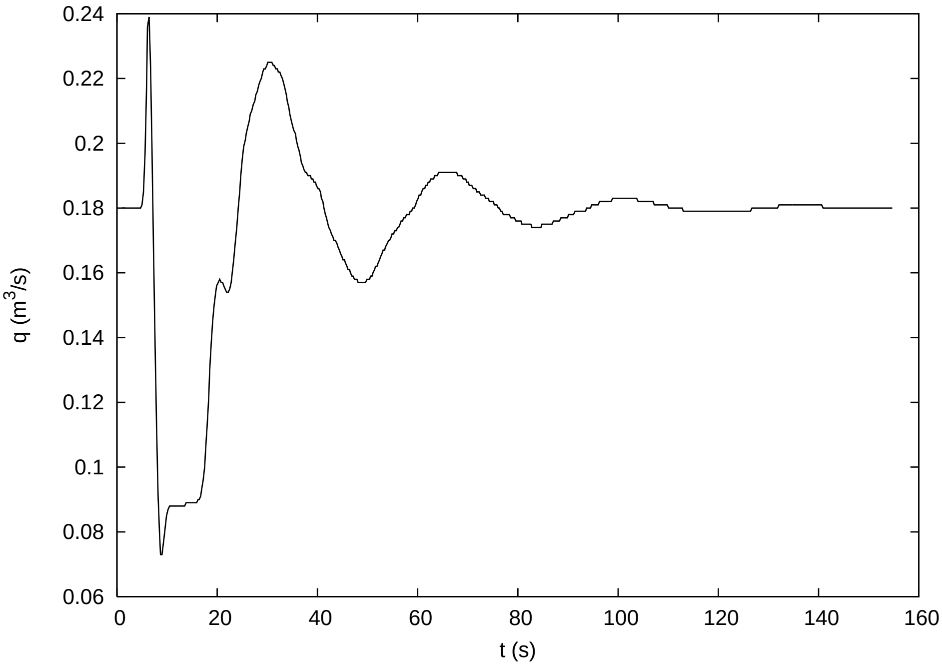


Figure 9: Convergence of the computation for the weir with jump - Flow discharge hydrograph at the downstream end of the channel

Figure 10 compares the water level profile computed by the steady computation kernel with results given by the trans-critical computation kernel and those of the analytical solution. A good agreement is obtained.

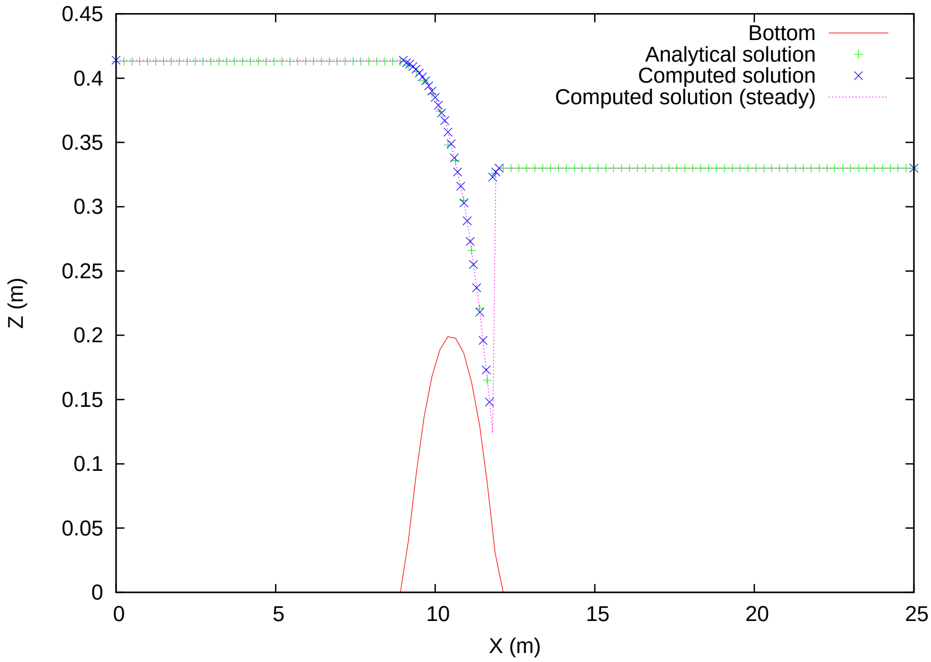


Figure 10: Water levels for the weir with hydraulic jump - comparison between the analytical solution and converged trans-critical and steady computations

3 Application - Junction

3.1 Purpose

This case study illustrates the use of Mascaret for simulating flood wave propagation in open channel with a junction. The flood wave may propagate downstream of the valley or upstream in the tributary. This flood wave division limits the height of the flood wave but increases the size of flooded area and may lead to the occurrence of a second wave in the main valley.

An accurate treatment of a junction is of high importance. `MASCARET` proposes a particular method to treat the junction, by coupling a 1D model and a 2D model. The 2-D zone is a simplified geometry and is created by `MASCARET` based on the model parameters [3].

The method is described in the report [3].

Here, we simulate flood wave propagation over a dry lake. The flood wave is coming from a lateral tributary. A progressive breaking of this dam is leading to a flood wave, propagating through the valley of the tributary and reaching the dam lake at the junction. The flood wave will propagate into both directions (upstream and downstream) of the dam lake.

The results of the flood wave propagation in the lake are compared to the simulations obtained with a two-dimensional depth averaged model (Computation with `TELEMAC2D`).

Data and files are stored in the example directory DIRECTORY.

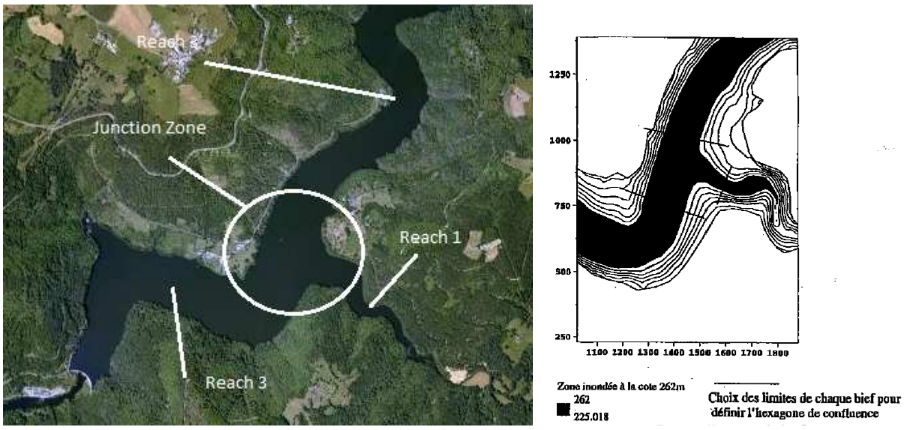


Figure 11: Left : The lake and the lateral tributary (reach 1). Right : result of the computation for a single reach and the obtained limits for the junction

3.2 Geometry

3.3 Defining the model domain

The lake is filled by a flood wave coming from a lateral tributary (reach number 1 in Figure 11) that has a slope of about 10%. The lake is divided in two parts : the downstream reach (reach number 2) has a length of 1000 m. The upstream part of the lake is the third reach (reach number 3) having a length of 12 km and a slope of 0.4%. The junction is the zone where these three reaches merge.

To define the size of the junction, a preliminary computation with a single reach was performed. The maximum water level was then used to define the geometry of the junction zone. Figure 11 shows the flooded zone for the single reach computation and limits of the junction chosen for each reach.

3.4 Creating the model in FUDAA-MASCARET

Open FUDAA-MASCARET. Choose the **Hydraulics** entry and then the **Computation Kernel**. Select the **super-critical kernel** from the scroll-down list and validate with the  icon.

3.5 Hydraulic Network

3.5.1 Preliminary Considerations

Figure 12 shows longitudinal bed elevation profiles of the three reaches, the junction as well as the direction of the wave propagation. This case study demonstrates several advanced features of MASCARET, such as 2-D modelling of a junction and flood propagation over a dry area.

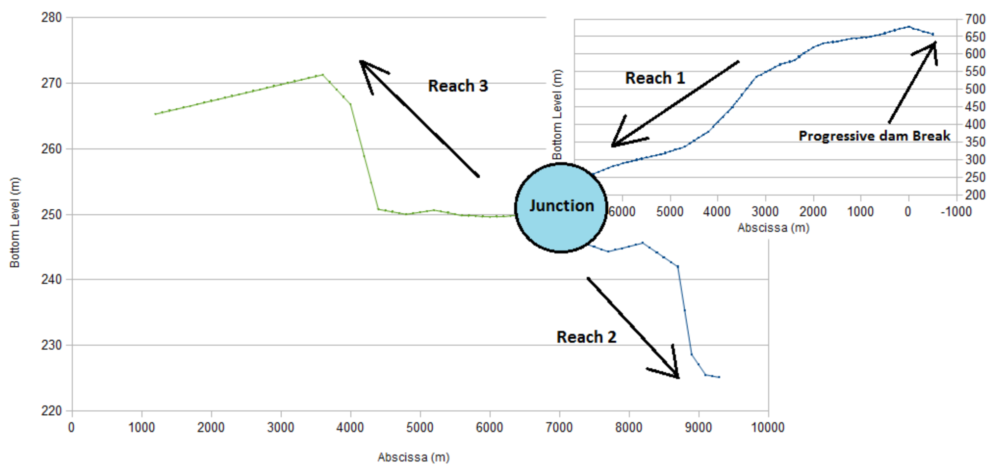


Figure 12: Bed elevation profiles of the three reaches and direction of the flood wave

Reach Abscissa:
In FUDAA-MASCARET, arrows indicate the direction of increasing abscissa, which is not necessary from upstream to downstream. Discharge is then

assumed positive in the direction of increasing abscissas (the direction of the arrow in `FUDAA-MASCARET`).

Dam breaking:

In this case, a progressive breaking of the dam located upstream of the reach 1 is considered. In `FUDAA-MASCARET`, when a progressive dam break is simulated, the dam is considered to be located at the upstream end of the reach number 1. A flow and stage hydrographs, modeling this progressive dam break, should be implemented as boundary conditions. If the option"dam-break wave computation" is highlighted, the propagation of the wave downstream is computed in a moving domain. Therefore, abscissa should increase in the direction of the wave (from upstream to downstream for the reaches of the main valley). All other reaches should be numbered from upstream to downstream and abscissa should increase from the junction to their open end (direction of the arrows in Figure 12).

In figure 12, reaches defining the main valley for the wave propagation are 1 and 2. Reach 2 is the downstream part of the lake and reach 3 is the tributary.

3.5.2 Creating the Hydraulic Network

Select **Hydraulics** in the main menu, and then **Edit the Hydraulic Network**. The network window will display in the interactive mode. The interactive mode may be toggled on or off by clicking on the **Interactive Mode** icon .

If the interactive mode is toggled on, icons of the Hydraulic Network Editor are activated, enabling the creation of the hydraulic network.

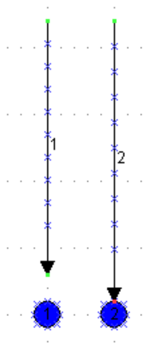


Figure 13: Joining a reach to a junction : Separated (left) and connected elements (right)

Now, click on the icon of the **Hydraulic Network Editor**. A window will display, showing the different elements of a hydraulic network described by `MASCARET`. The junction to be created is represented by three reaches : one inflow and two outflows.

First, create a reach by clicking on the icon of the editor. The created reach has the default number 1. The arrow shows the direction of the flow. This reach is the lateral tributary.

Next, create a junction by clicking on the icon  of the editor. Move the circle representing the junction next to the arrow of the reach number 1. Now, move the arrow of the reach number 1 to the limit of the circle representing the junction. The green dot at the end of the arrow becomes red, which means that the reach is connected to the junction.

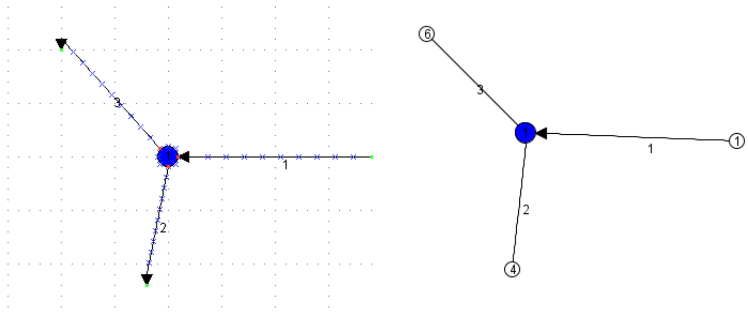
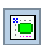


Figure 14: Final hydraulic network for the junction in interactive mode toggled on (left hand side) and off (right hand side)

NOTE : To connect the reach and the junction, drag and drop the end of the reach only (not the entire reach) within the junction.

Now, create reaches 2 and 3. Connect the upstream end of each reach to the junction (Figure 14).

Once the network is created, toggle OFF the interactive mode by clicking on the  icon.

3.6 Cross-Section Data

In this case study, cross-sections are quite complex. They are stored in the example directory and may be imported one by one for each reach. Click on the reach number 1. The **cross section editor** displays. Click on the Import icon and select the Excel file format and then the file *geom_r1.xls* from the example directory. Cross sections are now available in `FUDAA-MASCARET` and can be viewed by selecting a cross-section and clicking on the Edit icon . An example is shown in Figure 15. It is possible to switch to the next or previous cross-sections using icons and . Confirm the cross-section profile with the icon . Ignore all warnings about vertical walls and close the cross-section editor by clicking on .

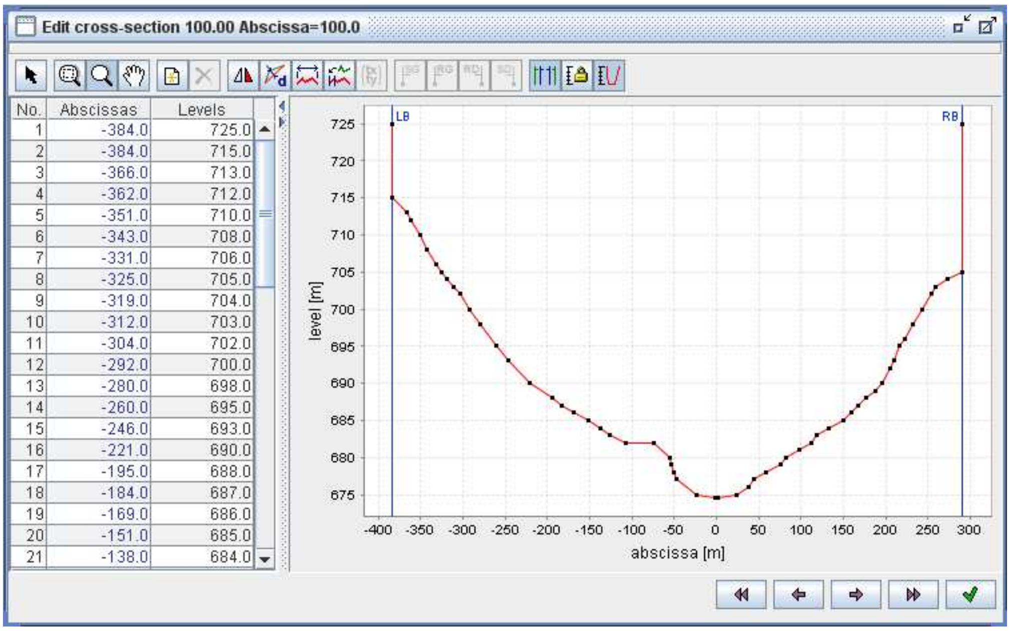


Figure 15: Cross-section profile editor

Create the two other reaches in the same manner as the previous one: click on the reach, then select Import from the cross-section editor. Select the file *geom_r2.xls* for the reach number 2 and *geom_r3.xls* for the reach number 3.

Now, all data for the geometry are entered, all three reaches are defined by abscissa and cross-sections.

3.7 Junction Data

The hexagon defining the limits of the local 2D computation was defined by a first computation using a single reach (not discussed here). The result is shown on the right hand side of the Figure 11. The cross-section data entered in the previous step took into account this step and cross-sections next to the junction are defined by this method.

From the hydraulic network, click on the junction, shown as a circle, The junction Editor will pop up. Two different methods are proposed : defining the junction area by coordinates of the external orthogonal of cross-sections at the limits of the reaches, or entering the start and end nodes of the cross-sections constituting the junction. Coordinates are given in a local system, particular to each reach and at the real scale, see section 7.1.

In the present case, the normal vectors of the cross-sections defining the junction are given by :

	X	Y	α
Extremity 2 (downstream end of reach 1)	0	0	-90
Extremity 4 (upstream end of reach 2)	-84.5	122.5	175.0
Extremity 6 (upstream end of reach 3)	80.5	175.0	5.5

Table 2: Enter these values into the table on the left hand side of the window

This data and cross sections at the extremities will define the zone of the local two dimensional model.

Confirm your choice by clicking on the icon .

3.8 Boundary Conditions

The hydraulic network created in the previous step has three open ends : The upstream end of the tributary where the flood-wave is coming from (open boundary number 1), the upstream end of the dam lake (open boundary number 6) and the downstream end of the lake (the dam, open boundary number 4).

At the open boundary number 1 (i.e. the upstream end of the tributary), a stage and flow hydrograph defining the progressive dam break is imposed. This hydrograph is given in the example directory, file junction_0.loi. Click on the circle number 1. The **open boundary editor** will pop up.

Choose "boundary condition supplied" by graph. Then, click on the **NEW GRAPH** button and create a new graph by clicking on the icon . Select the **“stage and flow hydrograph”** option. In the **hydraulic graph** window, click on the import icon and select the file junction_0.loi in the example directory. Choose the option **“Fixed flow”** in order to model the flow coming from the progressive dam break (the water level from the flow hydrograph will be used when the flow becomes super-critical). The stage and flow hydrograph is imported and drawn in the editor. Confirm all choices to close windows and come back to the network window.

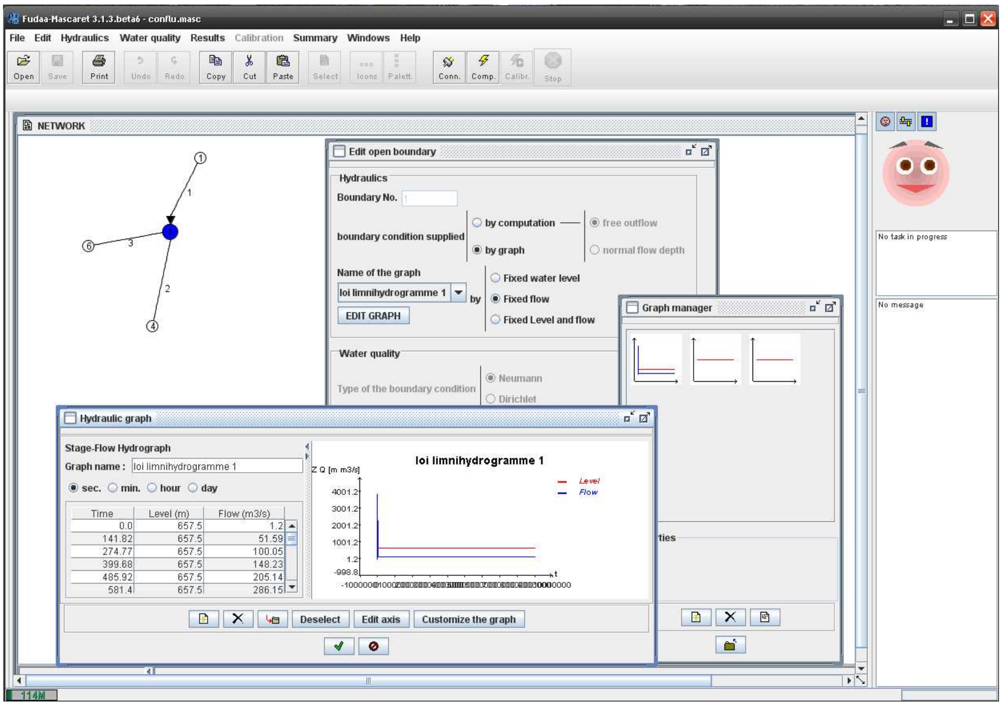


Figure 16: Defining the boundary conditions for the stage and flow hydrograph of the open end number 1

The open boundary number 4 is the downstream end of the reach number 2, corresponding to the location of the dam. The boundary condition is a fixed water level of 245 m. Indeed, the flow regime is super-critical at about 500 m upstream from the end of the reach. Click on the circle number 4. Select **“boundary condition supplied by graph”** and create a new stage hydrograph by clicking on the NEW GRAPH button and create a new graph. Select the **“stage hydrograph”** option. A spreadsheet window will display. Enter the water level of 245 m at times 0 s and 50000 s. Confirm all choices, close all windows and return to the network window.

The open boundary number 6 is the downstream end of the reach number 3, corresponding to the upstream end of the lake. At this boundary, a zero flow discharge is imposed. Define the flow hydrograph in the same manner as the open boundary 4. Enter a zero flow discharge at times 0 s and 50000 s. Confirm all choices and close all windows.

3.9 Mesh Parameters

All three reaches will be discretized using a regular mesh of 100 m. Select the **Mesh** from the **Hydraulics** menu. From the scroll-down list for the **Computation Method** select the **Grid Map** point. Then click on the button **Edit Computation Sections**. A Spread-sheet will display allowing the user to define the mesh size by parts. Here, each reach is assumed to be a single part, and each reach is discretized using a spatial step of 100 m for instance. Enter the reach number 1 in the first line. Clicking on the cells, Upstream Abscissa and Downstream Abscissa will automatically complete the data for the whole reach. Enter a grid size of 100 m in the last row. Proceed in the same manner for the reaches 2 and 3.

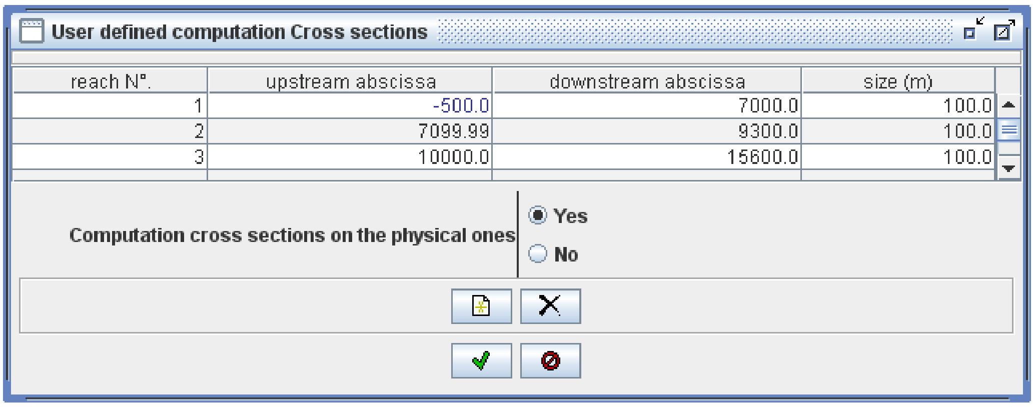


Figure 17: User defined cross-sections for the three reaches of the computation

Additionally, all physical cross-sections entered in the geometry part will be considered in the computation. To do this, choose **"Yes"** on the check-box just below the spread-sheet (Figure 17). Confirm by clicking on the icon .

From the hydraulics menu, select the **“Vertical Discretization of Cross-Sections”**. The space step is set at 0.5 m for all three reaches. This value can be modified in order to test the robustness of the solution. Proceed in the same manner as for the Cross-section discretization. Close all windows by confirming your choices.

3.10 Initial Conditions

Most part of the domain is considered to be initially dry. The reach number 1 (which represents the tributary) is wet only at the farthest upstream part.

From the main Fudaa-Mascaret window, select **Hydraulics** and then **“Initial Conditions”** to define the initial conditions. Here, the initial conditions are user-defined. Click **NO** on the question **“Restart the computation ?”** and choose **“Yes”** for **“Presence of an initial water level ?”**.

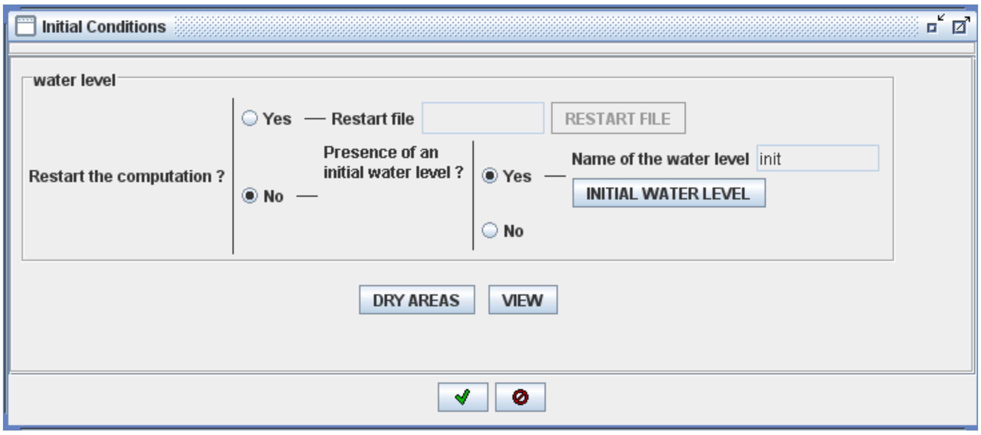


Figure 18: Initial conditions

Click on the Initial Water Level button and enter the following values into the table :

Reach No	Abscissa	Level (m)	Flow (m^3/s)
1	-500	677.1	0
1	0	677.1	0
1	102	674.6	0
1	6700	295.5	0
2	7100	245	0
2	8475	245	0
2	9300	245	0
3	10000	295.5	0
3	13200	271.3	0
3	16000	271.3	0

All other areas are considered dry. From the **Initial conditions window**, click on the button **"Dry Areas"**. Enter zones which are considered initially dry, as shown in the following table :

Reach No.	Upstream abscissa	Downstream abscissa
1	130.0	7000.0
2	7099.99	8770.27
3	10000.0	13240.0

It is worth noting that if some parts of the domain have an initial water level and then set as dry areas, they will be considered as initially dry by the code.

3.11 Temporal Parameters

From the **Hydraulics** menu, select the **Temporal Parameters**. Set the initial time to 0 and the time step to 1 s. The real value of the time step has not a great importance, because for the super-critical kernel a variable time-step will be used. Activate the Variable time step and set the Courant number to 0.8. The total time of simulation is 15000 s : Activate the Max

Time check box and enter the value 15000. Confirm your choices by clicking on the Validate button .

3.12 General Parameters

From the **Hydraulics** menu select **General Parameters**. Here, we define the general parameters for friction and dam breaking scenario.

The first point in the General Parameters Editor concerns the progressive overflow. In this case, no progressive overflow is considered, thus activation of the two check boxes is not required. There is no conservation of friction on vertical walls and no automatic head losses at junctions. Activate the corresponding check-boxes for both of them.

Here, the friction law is of type **"Strickler"**. Choose this option from the scroll-down list.

If a dam-breaking wave option is activated, it is necessary to choose **"Cross section in absolute abscissa - Yes"**, which means that the abscissa are not relative to a single reach. In this case, pay attention to the numbering of the reaches. The option **"Automatic head losses at junctions"** is NOT compatible with dam-breaking wave option.

The next point concerns the breaking parameters for the main dam. The main dam is the dam upstream of the first reach. In the present case, a progressive breaking of the main dam should be computed. So select Yes for **"Computation of a dam-break wave"** on the right side of the window and select a progressive breaking (Figure 19).

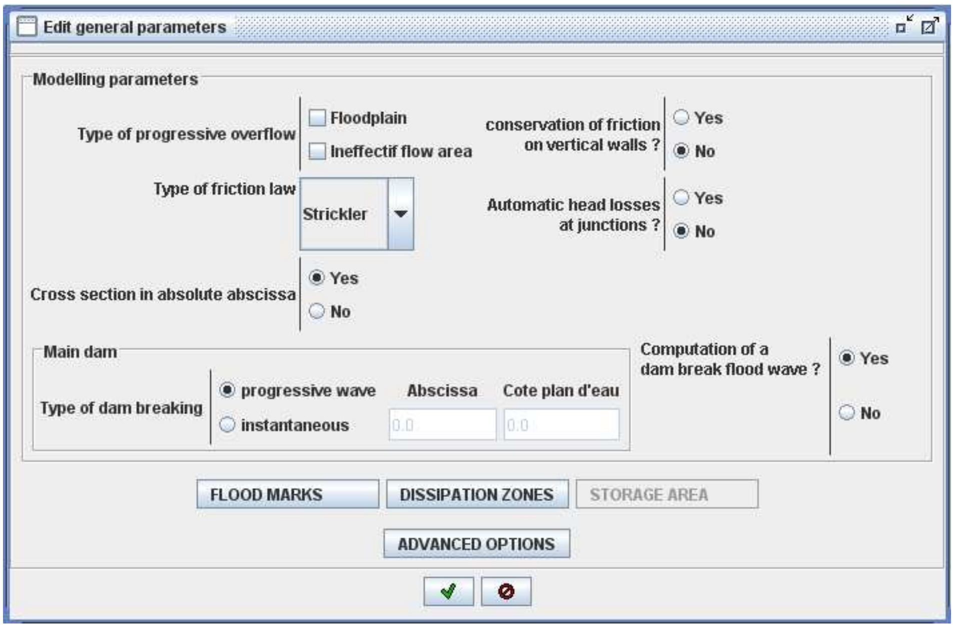


Figure 19: General Parameters and selected options for the case study

Now, the dissipation zones should be defined. Click the button **"Dissipation zones"** at the bottom of the window. A new window will pop-up. For all three reaches, the friction coefficient for the main channel is considered to be 40 $m^{1/3}.s^{-1}$, while a value of 15 $m^{1/3}.s^{-1}$ is set for the floodplain. For each reach, enter its number, and then click on the upstream-abscissa cell. Abscissa at the upstream and downstream ends will display automatically. Enter then the value 40 $m^{1/3}.s^{-1}$ for the main channel and 15 $m^{1/3}.s^{-1}$ for the flood-plain. Finally, confirm clicking on the icon.

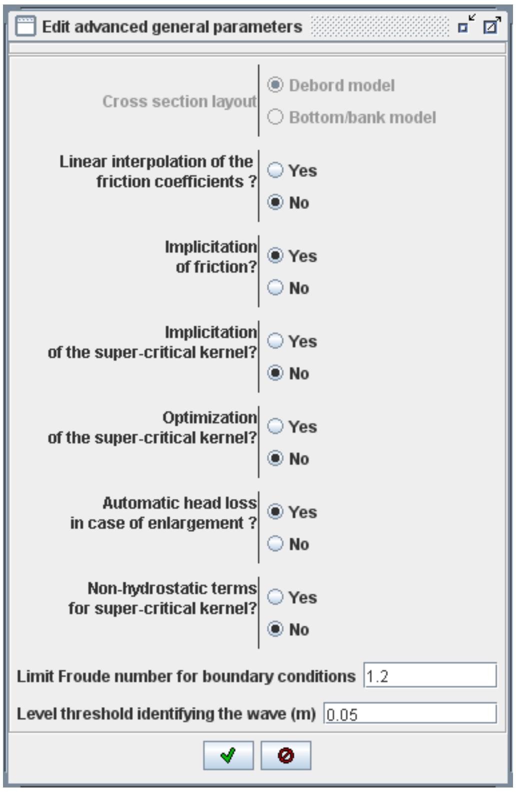


Figure 20: Advanced General Parameters

Go back to the **"General Parameter Editor"**, click the **Advanced Options**.

Activate the **Implicitation of friction** (for a more stable computation) and **Automatic Head Loss** in case of enlargement, but in case of rapid flow it is not recommended to choose this option. The Froude number for boundary conditions is set to 1.2 (what means that the continuity is used all along the computation to compute the boundary conditions) and the threshold level identifying the wave is set to 0.05. Implicitation of the super-critical kernel is not recommended. All choices are shown in Figure 20. Confirm by clicking on .

3.13 Output Parameters

From the **"Hydraulics"** menu select the **"Output Parameters"**. Data should be recorded all 10 time steps. Choose the same writing frequency for result and listing files. The first time step is the time step number 1 (it is not possible to save results at the time-step number 0, corresponding to the initial conditions). The listing file should be written along with computation progress, so activate this check-box.

The results should be written at all computation cross-sections, so activate this check-box.

Limit the file sizes as shown in the Figure 21.

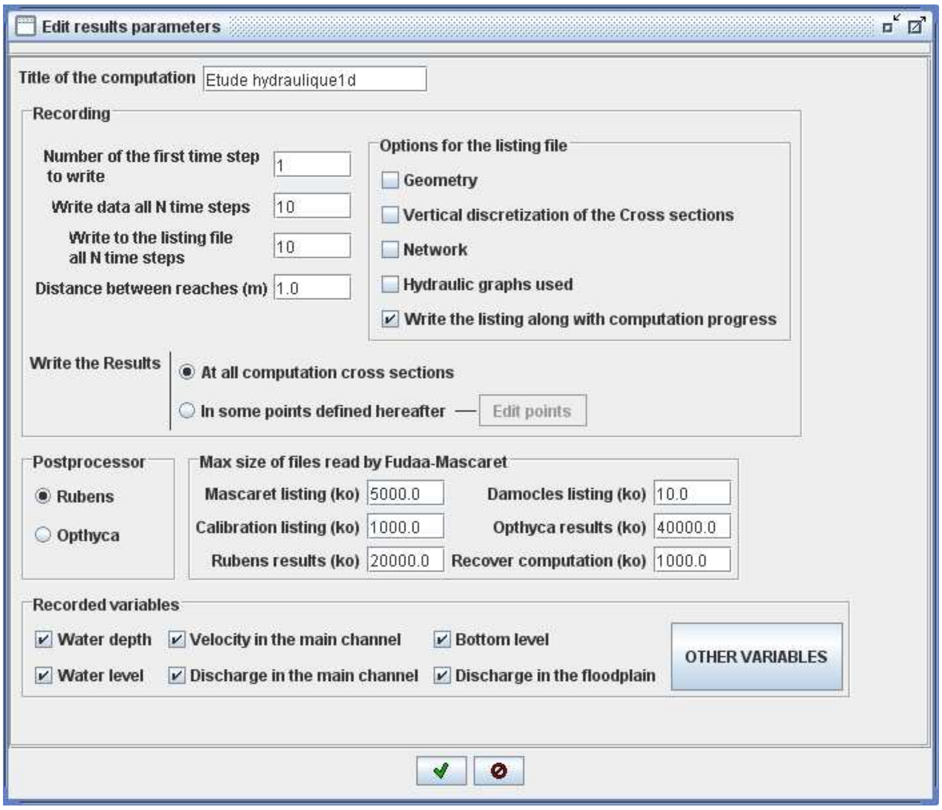


Figure 21: Output parameter options

3.14 Results

Finally, save the current case-study by selecting **"Save as ..."** from the **"File"** menu of the main window. Enter a file name like "junction" and save the present study.

Now, click on the Compute icon . The computation progress is shown on the bottom of the main window. Once the computation finished, the **General Results** window pops up. Check for warnings in the first tab. Ignore the warning about the size of the Damocles listing file.

The next tab concerns the messages of the computation kernel. Check here the computation time needed. On the last line, the computation kernel should say “termine”.

Ignore the Damocles listing tab.

The `MASCARET` listing should say at the end “FIN”.

If the computation has been successfully run, the Graphs point in the Results menu should be active. Click on it in order to check the results.

First, check the incoming discharge at the reach number 1, corresponding to the progressive dam break. The boundary condition was a stage and flow hydrograph, with the fixed flow option activated. This means that the boundary condition is the flow hydrograph if the flow regime is subcritical. If the flow regime becomes super-critical, both water level and discharge are used to define the boundary condition. Discharge and water level at the upstream end of the reach 1 are shown in Figure 22.

This flood-wave is propagating through the reach number 1 and joins the junction. Here, the flood-wave enters the reaches 2 and 3. Figure 23 shows the water level and the discharge for the three reaches at a distance of 1km from the junction. The discharge at these locations is compared to results from a full 2D simulation with `TELEMAC2D`, Figures 23, 24 and 25.

For reaches 2 and 3, results obtained with `MASCARET` are compared to a full 2D simulation with `TELEMAC2D`.

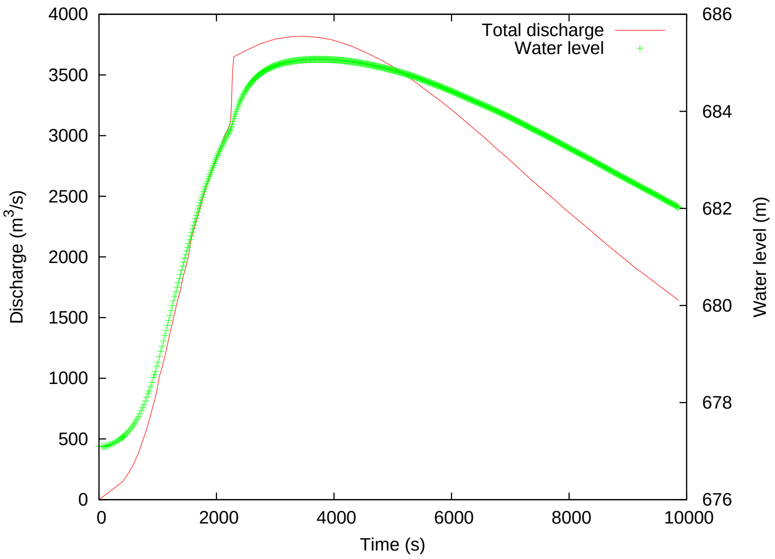


Figure 22: Discharge and water level at the upstream end of the reach 1

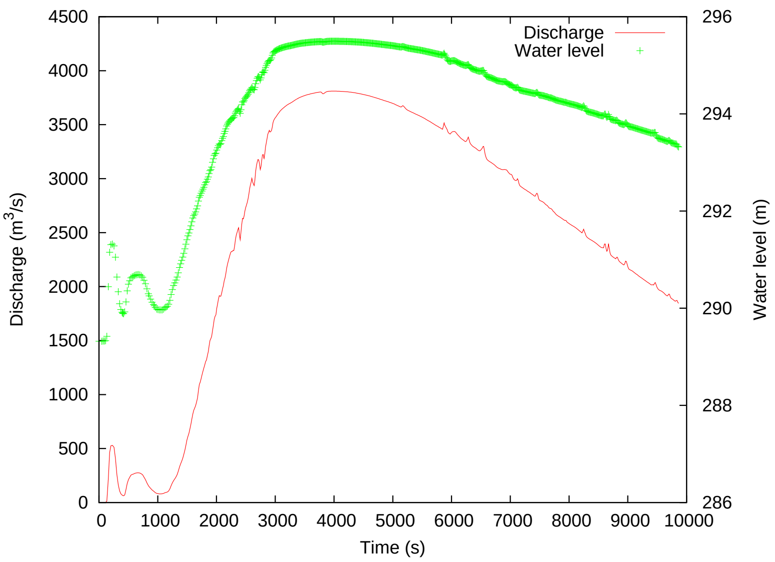


Figure 23: Reach 1- Discharge and water level at 1km upstream of the junction

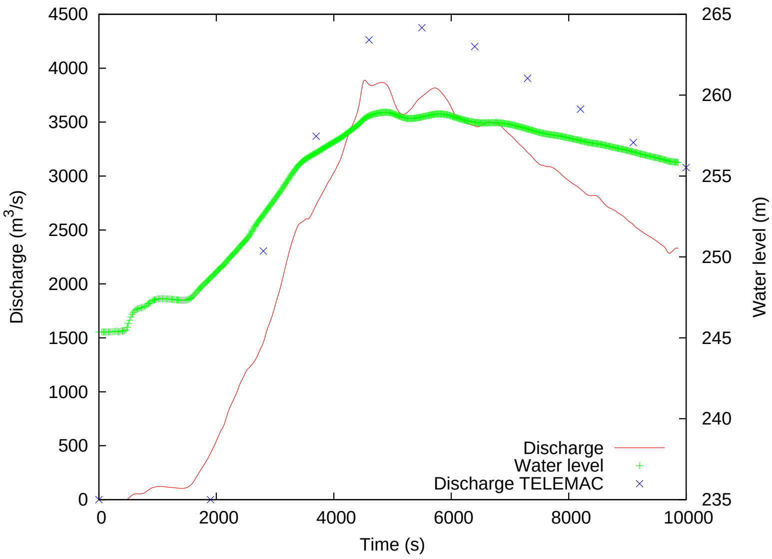


Figure 24: Reach 2- Discharge and water level at 1km downstream of the junction. `MASCARET` discharge is compared to numerical predictions obtained with `TELEMAC2D`

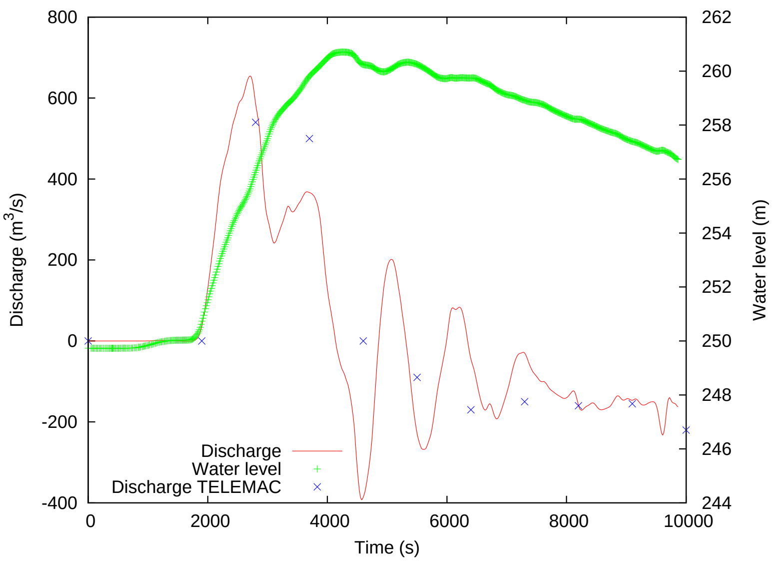


Figure 25: Reach 3- Discharge and water level at 1km downstream of the junction. `MASCARET` discharge is compared to numerical predictions obtained with `TELEMAC2D`

4 Application - Hydraulic Network with storage areas

4.1 Purpose

This case study shows an application invoking multiple computational features of the MASCARET code : hydraulic structures represented as singularities (inlined and lateral weirs, lateral inflow and local head-loss), the local two dimensional modeling of a junction and the computational kernel CASIER which allows to model storage areas linked to the reaches. Storage areas are represented within CASIER as variable volumes with a constant water level and a zero flow velocity. These storage areas are linked together and to the reaches by different exchange laws, function of the type of the link (culvert, channel, weir ...). This allows to model flooded areas which have no interaction with the flow in the main channel. It represents an extension of the 1D modeling approach, but does not replace a full 2D model, for example if active flow is present in these storage areas.

This application is not a real-case study but an example how to build up a more complex case study with multiple hydraulic structures.

4.2 Geometry

This case has a more complex geometry with several singularities corresponding to hydraulic structures, a junction, storage areas and connections between storage areas, called links. First, the graphical outline of the network will be created, section 4.3. Then, the parameters for the different structures will be entered, sections 4.4 to 4.7.

In order to create the present case study, open FUDAA-MASCARET or close the current project. The only active entry in the "Hydraulics" menu is then the "Computation Kernel" menu point. Click on it and select the **unsteady subcritical regime** from the scroll-down list. Click on the  button in order to valid your choice and to close the window.

4.3 Drawing the Hydraulic Network

Select the **"Hydraulic Network"** point from the **"Hydraulics"** menu. The **Network** window pushes up in the interactive mode, allowing to draw the hydraulic network with its different components. Open the **"hydraulic**

structures editor" by clicking on the symbol in the hydraulic network toolbar.

Start by creating a junction of three reaches as explained in the previous example 3, but arrange the reaches as shown in Figure 26 (reach 1 on the top, reach 2 down left and reach 3 down right).

Next, create two storage areas by clicking on the button of the hydraulic network components. Move one to the right hand side of the first reach, and the other one to the right hand side of the left downstream reach. Create a connection between the reach one and the first storage area : Click on the button of the **"hydraulic structures editor"**. Move the line representing the link in the space between the first reach and the first storage area. Move now the green dot of the right end to one of the blue crosses close to the center of the reach. The green dot becomes pink, which means that the link is connected. Now, move the green dot on the left end of the link to the border of the storage area (see left hand side of the figure). The green dot becomes pink, which means that the link is connected to the storage area (see center of the Figure 26).

Connect by a link in the same way the two storage areas and than the storage area 2 to the reach number 2. The result is shown in the right hand side of Figure 26

In the next step, the different hydraulic structures should be created. In this example, four types of structures are taken into account. The first one is a lateral weir, situated between the open end 1 of the reach 1 and the link of the reach 1 to the storage area 1. Beware : this lateral weir has no connection with the storage 1. In order to create this lateral weir, click on the **"lateral weir"** button in the **"hydraulic structures editor"** window. A small blue arrow showing from the left to the right with a green dot on the left shows up in the network editor. Move this arrow to one of the blue crosses of the first reach, situated between the open end 1 and the connection to the link. The green dot becomes pink, the lateral weir is connected to the reach.

The next structure on the reach 1 is a local head-loss, situated between the connection to the link and the junction of the three reaches. In the window showing the different hydraulic structures, click on the **"local head loss"** button . Attach the icon showing up on one of the blue crosses of

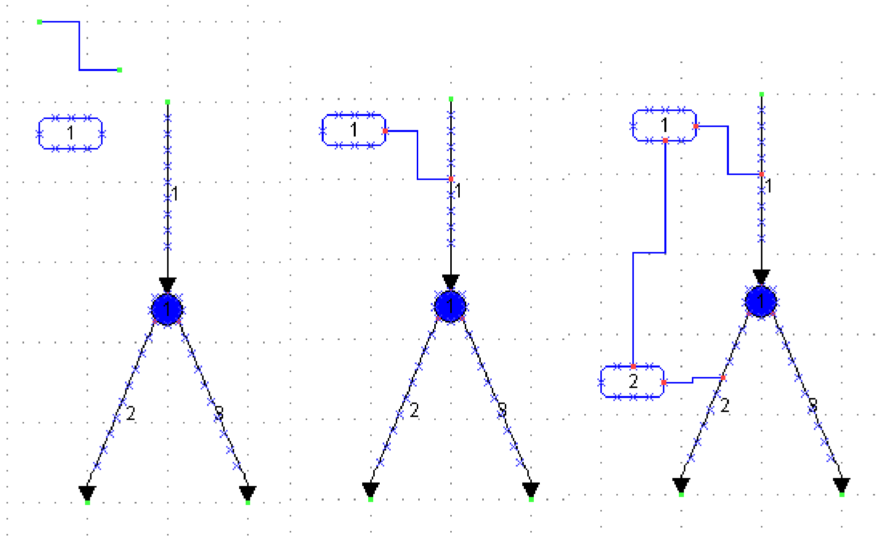


Figure 26: Connecting the storage areas to the reaches

the reach 1, situated between the connection to the link and the junction as shown in Figure 27. The green dot close to the icon should become pink, which indicates a correct connection to the reach.

Next, create an **“Dam/Weir”** object on the reach 2, situated between the junction and the connection to the storage area 2. Click on the symbol in the hydraulic structures editor. A window pops up, asking for the type of weir / spillway. Select the “Weir graph (crest level / discharge coeff.). Insert then the inlined weir on the reach 2 as shown in the Figure 27.

NOTE : here, a subcritical flow regime is considered, so a large choice of rating curve types is possible. In a super-critical flow regime, only the rating curve $Q = F(Z_{us})$ and the super-critical weir graph (crest, discharge) are available.

The last component is a **“Inflow”** , situated on the reach number 3. Toggle off the interactive mode of the network editor by clicking on .

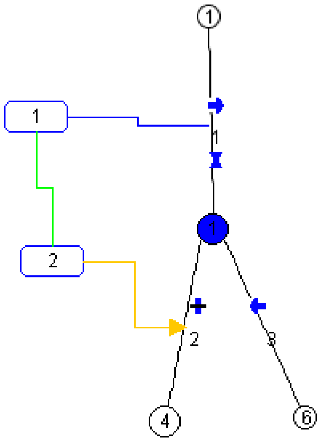


Figure 27: The final hydraulic network with the singularities and links between storage areas

4.4 Reach Cross Sections

In the present case, all three reaches are quite simple channels. In order to enter the cross-section data for the first reach, click on it. The **"cross section editor"** pushes up as a spread-sheet, where the abscissa should be entered. Enter the first abscissa named P1 with the value 0.

Select the first abscissa P1 and click then on the button **"Edit"** . As the cross-section contains no data yet, a window pushes up asking for the cross-section data. Here, a simple geometric form will be created, so answer **"Yes"**  in order to create the trapezoid. The following window allows for to define the shape of the cross-section. In the present case, only a main channel exists. Select a trapezoidal shape for the main channel and click on the E button. Enter a bottom width of 100 m, a height of 10 m and a wall slope of 1. Click on **"OK"**, and the cross-section shows up as in Figure 28. Validate this by clicking on  and ignore all warnings (no flood-plain in the present case study).

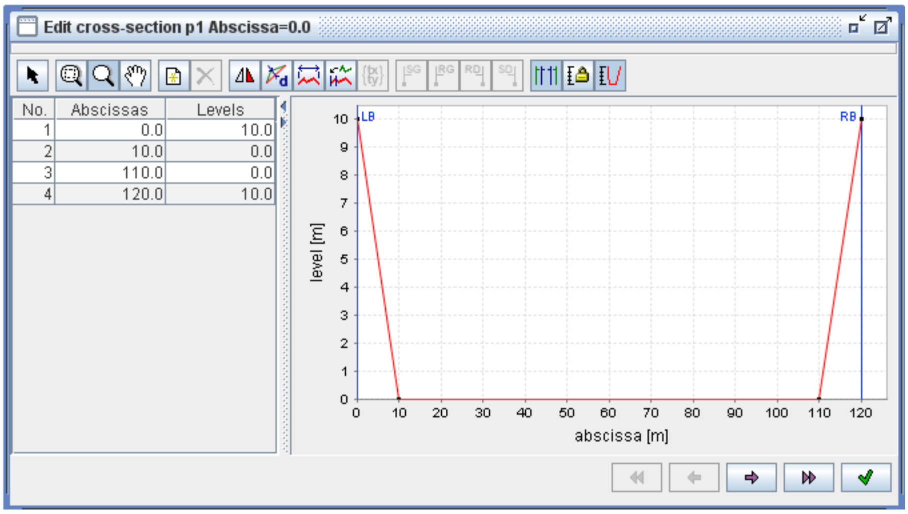
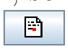


Figure 28: Cross section of the abscissa P1 for reach number 1

The second cross-section at the abscissa P2 is based on the first one, but is slightly modified. The best way is to duplicate the cross-section P1 and to modify it. Select the cross-section P1 and click on the button **"DUPLICATE"**. A new line is created, with a cross-section named P1bis. Rename it in P2 and change the abscissa to 1000. Then, select the line corresponding to the cross-section and edit it by clicking on .

The spread-sheet with the cross-section data comes up. The actual cross-section is a simple channel, which should be slightly modified. Add two new points by clicking on the **"NEW"** button .

The first new point (point number 5) has the abscissa 130 and the level 10, the second new point has the abscissa 131 and the level 11. The result should look like in Figure 29.

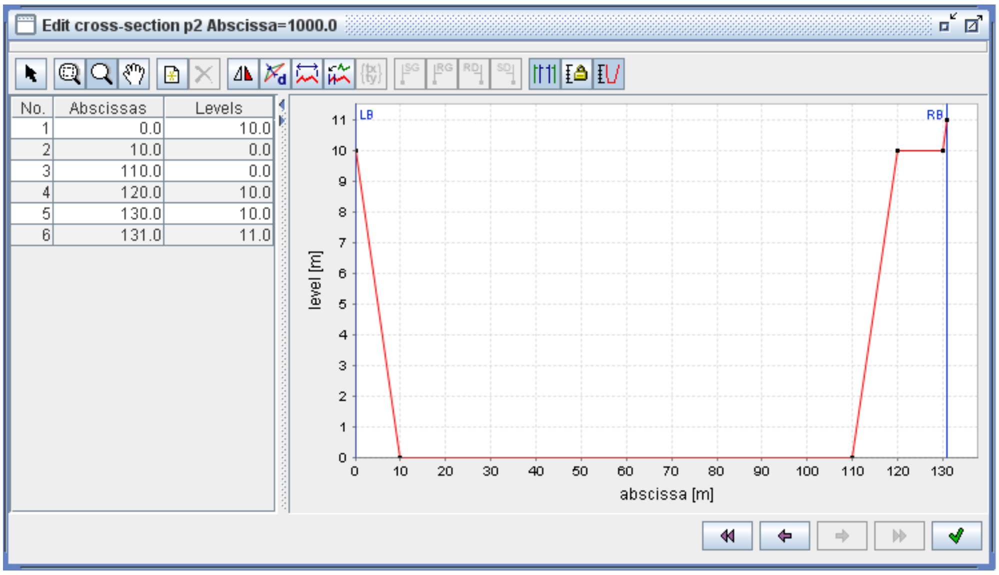


Figure 29: Cross section of the abscissa P2 of the reach number 1

The reach number 2 is defined by two identical cross-sections. Click on the reach and create the first cross-section by entering P3 for the title of the cross-section and an abscissa of 1001. Select this line and create a new profile, identical to the profile P1. Then, duplicate this profile, and change the name to P4 and the abscissa to 2000.

For the reach number 3, repeat these operation, this times with the cross-sections named P5 (abscissa 2001) and P6 (abscissa 3000).

Each reach is now defined by 2 cross-sections.

4.5 Storage Areas

Click on the storage area number 1 in the **"hydraulic network editor"**. A window pops up, allowing to enter the properties of the storage area. First, enter a name for the storage area, for example SA1. The initial water level is of 2.0 m. The storage area 1 has an external inflow, so select "Yes" for this option. This external inflow is defined as a flow hydrograph and may be due to rainfall or groundwater, for example. Click on the **"Edit Graph"** button, and in the **"graph manager"**, click on the **"new graph"** button . Select a new **"Flow Hydrograph Q(t)"**. The following window is a spread-sheet, defining the flow hydrograph. Enter a name for it, for example inflow_SA1.

Enter the following values into the spread-sheet :

Time (s)	Flow (m^3/s)
0	0
3600	2
8000	2

The resulting graph should look like in Figure 30. Validate this and return to the window Storage Area N. 1.

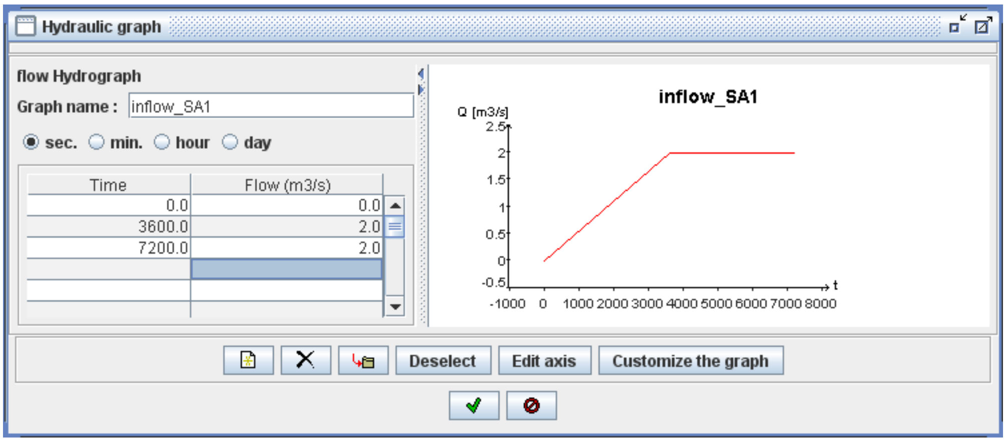


Figure 30: Inflow for the storage area 1

The last parameter for the storage area is about the representation of its geometry. The geometry is represented here by a **"Vertical Discretization"** of the storage area. Select this option in the scroll-down list and click then on the **"Edit Vertical Discretization"** button. A spread-sheet shows up, allowing to define the vertical discretization of the storage area. The bottom elevation of the storage area is 0 m. The step for the vertical discretization is 1 m. Enter these parameters on the top of the spread-sheet. The values for the level will then fill-in automatically. The surface of the storage area is 1000 m^2 for all sections. Enter this value in the column "Surface" for the first 8 points (up to a level of 7 m). The volume corresponding to the levels is then surface × level, enter the corresponding values. The final result should look like in Figure 31.

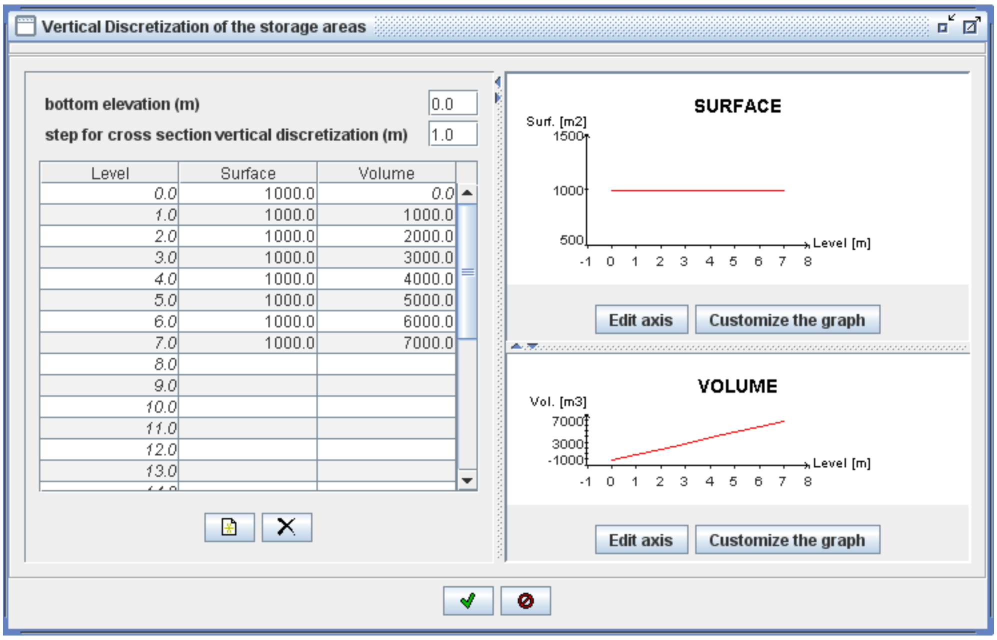


Figure 31: Vertical discretization of the storage area 1

Confirm all data for the storage area 1 and go back to the Network window.

Click on the storage area 2. In the **storage area editor**, enter a name for the second storage area, for example SA2. The initial water level is of 2 m. For this storage area, no external input has to be taken into account. The geometry is represented by a vertical discretization, as for the storage area 1. Select this option from the scroll-down list and click then on the **"Edit vertical cross-section"** button. In the following window, enter the parameters for the vertical discretization of the storage area. The bottom elevation is of 0 m. The step for the vertical discretization is 1 m. The surface of the storage area is 2000 m^2 up to level of 3 m. The corresponding volume is level X surface. Enter the corresponding values, the result should look like in Figure 32.

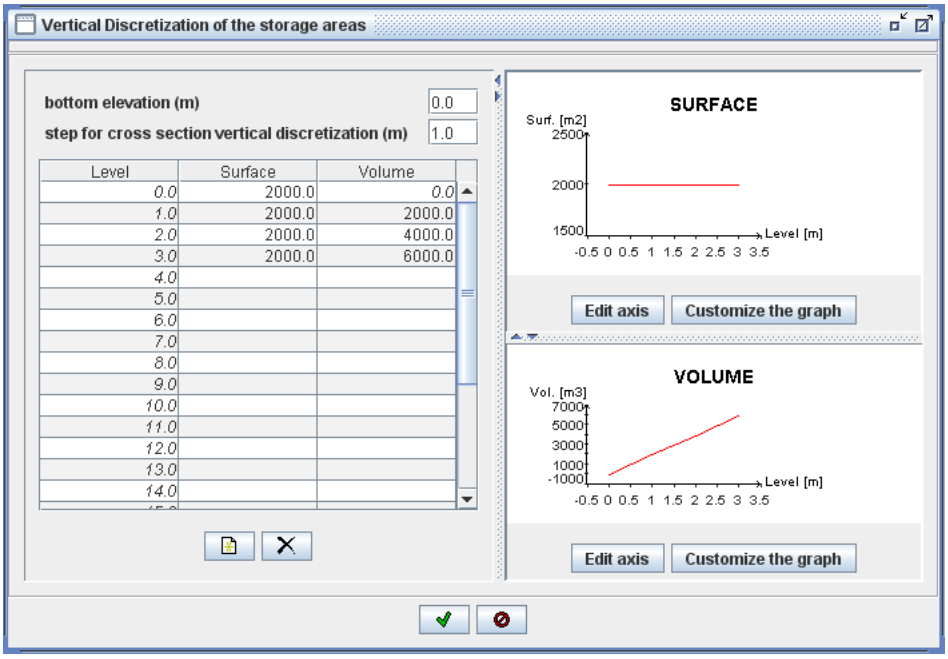


Figure 32: Vertical discretization of the storage area 2

Confirm all choices and return to the hydraulic network window.

4.6 Links to storage areas

The storage areas are connected to the other structures by links. Links exist between two storage areas or between a storage area and a reach. These links may represent different structures : channels, weirs, siphons or culverts.

Click on the link between the reach number 1 and the storage area 1. The first line in the **"Link Editor"** window concerns the abscissa of the reach, where the link is connected to the reach. Here, it is the abscissa 500 m of the reach number 1.

4.6.1 Link of type Weir

The type of this first link is a weir. Select this option. The mean level of the crest is of 1.9 m in the present case, and the width of the weir is of 1 m. The discharge coefficient is 0.4. The activation coefficient defines the limit between a submerged and an un-submerged flow over the weir. The type of the flow is described by the number:

$$R = \frac{Z_{ds} - Z_{crest}}{Z_{us} - Z_{crest}}$$

If R is less than the activation coefficient, the weir is considered as unsubmerged, if R is greater than the activation coefficient, the weir is considered as submerged. Here, the activation coefficient is 0.5.

Enter all values in the Weir part of the window and validate these entries by clicking on .

4.6.2 Link of type Channel

Next, click on the link between the storage area 1 and the storage area 2. This link is a channel. Select this option in the **"Link Editor"** for the link number 2. The mean bottom elevation of the channel is of 2 m, the length is 50 m, the width is 10 m and the Strickler coefficient is 40 $m^{1/3}.s^{-1}$. Enter these values into the channel section of the window and validate these entries by clicking on .

4.6.3 Link of type Culvert

The third link connects the storage area 2 to the reach number 2 by a culvert. Click on the link. First, enter the abscissa where the link joins the reach number 2 (here : 1800). Select the culvert option. The geometrical parameters for the culvert considered here are : Mean level = 2 m; Width = 0.5 m; Cross-section = 10 m^2. Note that in `MASCARET`, a rectangular culvert is considered, so the user should find the right similarity for the cross-section and the width. In general, the cross-section should be conserved and the width adjusted.

For the culvert, types of flow are considered. The flow of type "weir", which means that a free surface exists inside the culvert, and the flow of type "pipe flow" considers a completely filled culvert. In the present case, the Weir type Discharge coefficient is equal to 0.4 and the Pipe type discharge coefficient is equal to 0.4. The flow direction is "Storage area towards the reach". Validate these entries by clicking on .

4.7 Singularities

4.7.1 Lateral Weir

Click on the lateral weir symbol on the reach number 1, and the lateral weir editor pops up. Here, enter the abscissa, where the weir is placed (200.0) and a name for this weir ("lateral weir").

In the present case, the weir is defined by the crest level and the discharge coefficient, as shown in Figure 33. Enter the values as shown on this figure and close the window by confirming your choice .

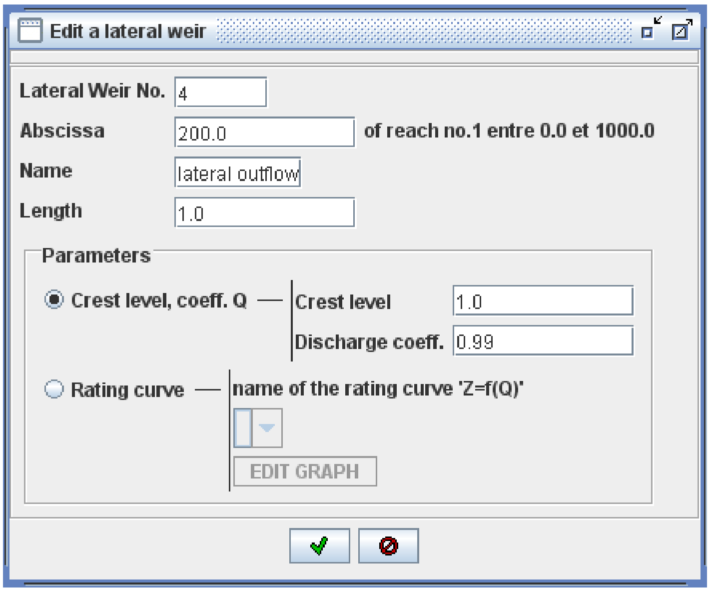


Figure 33: Parameters for the lateral weir

4.7.2 Local Head Loss

The local head loss is situated on the reach 1, between the link to the storage area 1 and the junction. Click on the symbol representing the local head loss on the reach. A window defining the parameters pops up. The number of the local head loss is 1, it is situated at the abscissa 600. Enter a name, for example "local head loss 1". The coefficient is 0.99. Valid these parameters by clicking on .

4.7.3 Inlined Weir

The inlined weir is situated on the reach number 2, between the junction and the link to the storage area 2. Click on the symbol in order to open the Inlined Weir editor. Enter a number for the weir ID, here it is 3. The second cell asks for the abscissa of the weir. The abscissa for the position of the current weir on the reach number 2 is 1500. Enter then a name for the weir, "Spillway / weir 3". The weir characteristics are the following ones : Crest level = 1, Discharge coefficient = 0.5, Thickness = thin. We can choose thin or thick (see more details in [1].

The dam breaking water level is considered to be 2.1 m.

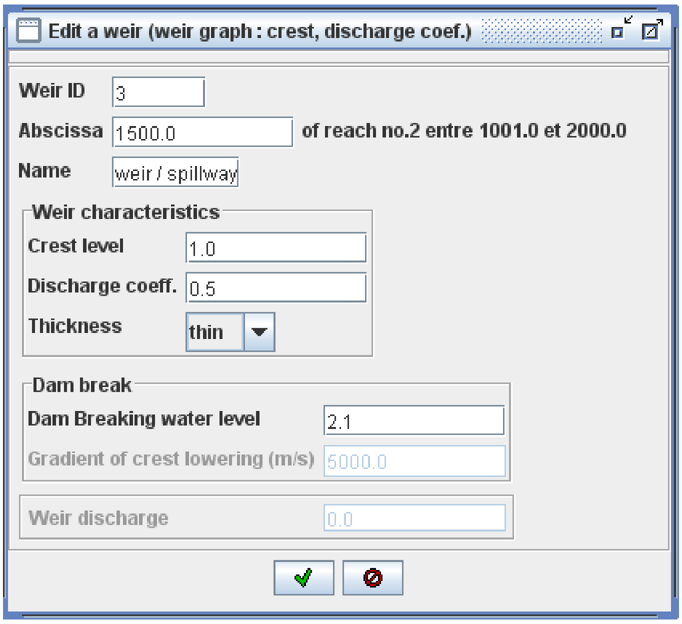


Figure 34: Parameters for an inlined weir / spillway

4.7.4 Lateral Inflow

On the reach number 3, a lateral inflow is considered. Click on the symbol in order to open the corresponding editor. Enter an ID for the lateral inflow, ID=2 here. The abscissa is 2500 m and the name is "lateral inflow 1". The type of lateral inflow is linear, so activate the corresponding option. The length is 5m.

It is necessary to supply a graph for the inflow. Click on the **"EDIT GRAPH"** button. The graph manager opens. Click then on the NEW button  in order to create a new graph. Choose the option "Flow hydrograph" in order to create a new flow hydrograph. Enter the following values into the spread-sheet :

Time (s)	Flow (m3/s)
0	0
3600	2
8000	2

Confirm all choices by clicking on the valid button  and return to the hydraulic network window.

4.8 Boundary Conditions

The model presents three open boundaries, the upstream end of the reach 1 and the downstream ends of the reaches 2 and 3. The upstream boundary condition is a flow hydrograph, the downstream boundary condition for the reaches 2 and 3 is a stage hydrograph.

In order to define the upstream boundary condition, click on the open end 1. The **"Open boundary editor"** pushes up. The boundary condition is supplied by graph, so choose this option in the window. Click on the **"EDIT GRAPH"** button in order to open the graph manager. Click here on **"Create"**  and select the option **"Flow Hydrograph Q(t)"**. A spread-sheet shows up, enter the name for the graph ("upstream", for example) and the following values for the flow hydrograph :

Time (s)	Flow (m3/s)
0	0
3600	20
8000	50

Confirm this by clicking on the **"Valid"** button , and go back to the **"Network Editor"** by closing / confirming all other windows.

Create now the boundary conditions for the downstream open end of the reach number 2, open end number 4. Click on it in order to open the **“Open Boundary Editor**”. Select “Boundary Condition supplied by Graph” and create a new graph by clicking on the **“EDIT GRAPH”** button and then on **“Create”** in the **“graph manager”** window. This times, a stage hydrograph is created. Select this option from window, a spread-sheet pops up allowing to enter a stage hydrograph. First, enter the name for the graph, for example “downstream”. In the present case, the stage hydrograph is a constant water level of 2 m. Enter this value for the time 0 and for a large time, 8000 s will be fine here.

The open boundary number 6, representing the downstream end of the reach number 3 has the same boundary condition than the open end number 4 of the reach number 2. Click on the open end number 6. In the upcoming **“Open Boundary Editor”** select “Boundary condition supplied by graph” and in the scroll-down list for the graphs select the graph “downstream” created just before. Confirm by clicking on the **“Valid”** button .

4.9 Mesh Parameters

The next active entry in the **“Hydraulics”** menu is the **“Mesh”** entry. In this menu point, the spatial discretization for the computation is defined. Here, the grid size is a user defined, constant length for the three reaches. `MASCARET` allows the user to enter a constant grid size for an entire reach or part of a reach.

Click on the entry **“Mesh”** of the **“Hydraulics”** point in the main menu. The mesh is here computed from a grid map. From the scroll-down list in the **“Mesh editor”** select **“Grid Map”** for the computation method for the grid. In order to enter the grid map, click on the button **“Edit Computation Sections”**. A spread-sheet pops up, allowing to enter the grid size for parts of the reaches.

Here, the grid size is constant and the same for all reaches, but the grid size should be entered reach by reach. Enter the number 1 in the first cell of the first line. The upstream and downstream abscissa for the reach will complete automatically. Enter then in the last cell for the first line a grid size of 10 m. Do the same for the reach number 2 and 3 (same grid size : 10 m). Confirm by clicking on the **“Valid”** button . Do the same for the **“Mesh”** window and return to the **“Network”** window.

Select then the **"Vertical Discretization of the Cross-Sections"** point from the **"Hydraulics"** menu. A spread-sheet shows up, allowing to enter the grid size for the vertical discretization for each reach. Proceed as described just above and enter for all three reaches a step of 0.5 m. Confirm by clicking on the **Valid** button  and return to the **Network** window.

4.10 Initial Conditions

Choose the point **"Initial Conditions"** from the **"Hydraulics"** menu. This menu point will ask for the initial water level and the associated flow. Answer **"Yes"** to the question **"Presence of an initial water level?"** and click then on the **"Initial Water Level"** button. A spread-sheet shows up, asking for the initial water levels and flows of the reaches. In the present case study, the initial condition for all three reaches is a constant level of 2 m and a constant zero flow. For each reach, create a line with the start abscissa and a second line with the end abscissa. Enter the corresponding values for level and flow. Confirm the values by clicking on the **Valid** button . Back to the **"Initial Conditions"** window, it is possible to check the initial water level and flow by clicking on the **"VIEW"** button. Figure 35 shows the spread-sheet and the graphical representation for the reach number 1.

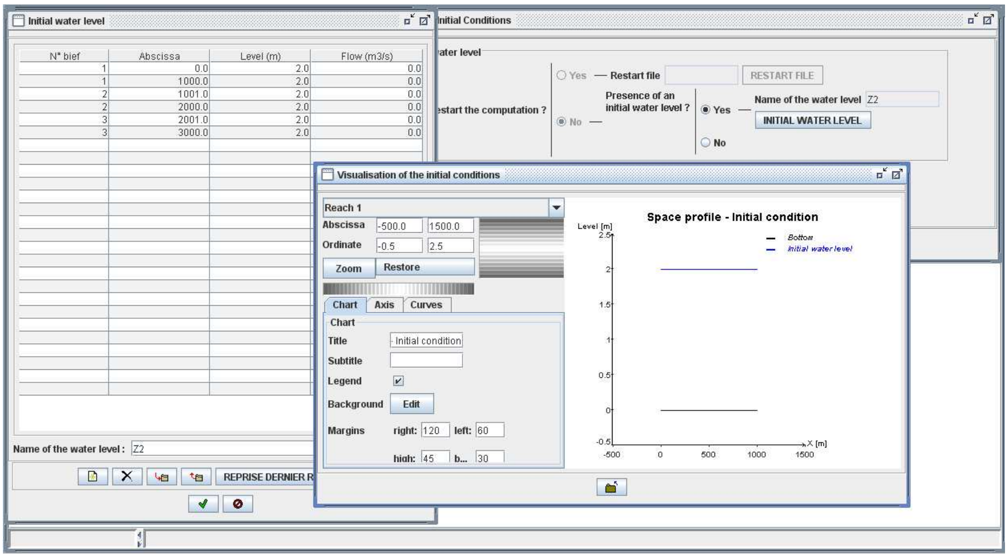


Figure 35: Initial conditions for the three reaches (spread-sheet) and the graphical representation for the reach number 1

4.11 Temporal Parameters

Once all spatial parameters given, the temporal parameters should be entered. Click on the **"Temporal Parameters"** point of the **"Hydraulics"** menu. The initial time is 0.0 s, the max time 8000 s. Set the time step to 30 s. The subcritical kernel has no variable time-step support, so this option is grayed out. Valid these entries by clicking on the **"Valid"** button .

4.12 General Parameters

Click on the **"General Parameters"** point of the **"Hydraulics"** menu. The first part is about the friction laws and the overflow behavior. In the present case study, no overflow is considered. Friction is not conserved on walls and no automatic head-losses at the junctions are considered. The type of the friction law is Strickler, select this from the scroll-down list. In order to enter the friction coefficients for the three reaches, click on the button **"Dissipation zones"** on the lower part of the window. A spread-sheet pops up, allowing to enter the values for the friction coefficients. Enter in the first cell the number of the reach, 1. The upstream and downstream abscissa will complete automatically. Friction is here constant for the whole reach, enter a coefficient of 40 for the main channel and the floodplain. Do the same for the reach number 2 and the reach number 3. Confirm your choice by clicking on the **"Valid"** button  (see top of the Figure 36).

In this case, we choose the option "Absolute abscissa" but it is not compulsory. The option "Absolute abscissa" is only compulsory when "dam break wave option" is highlighted.

The current case study models storage areas, so click on the button **"Storage Area"** in the lower part of the General Parameters window, in order to parametrize the storage area computation. Click on the check box on the lower part of the window **"Numerical Parameters of the Storage Areas"** in order to activate the model. The implicitation for the computation of the storage area and for the coupling is of 0.5, enter these values in the corresponding cells. The parameter "Max number of iterations for coupling" can be set to 1 in the present case. Confirm these parameters and return to the **"General Parameters"** window.

The "Advanced Options" are not changed in the present case study.

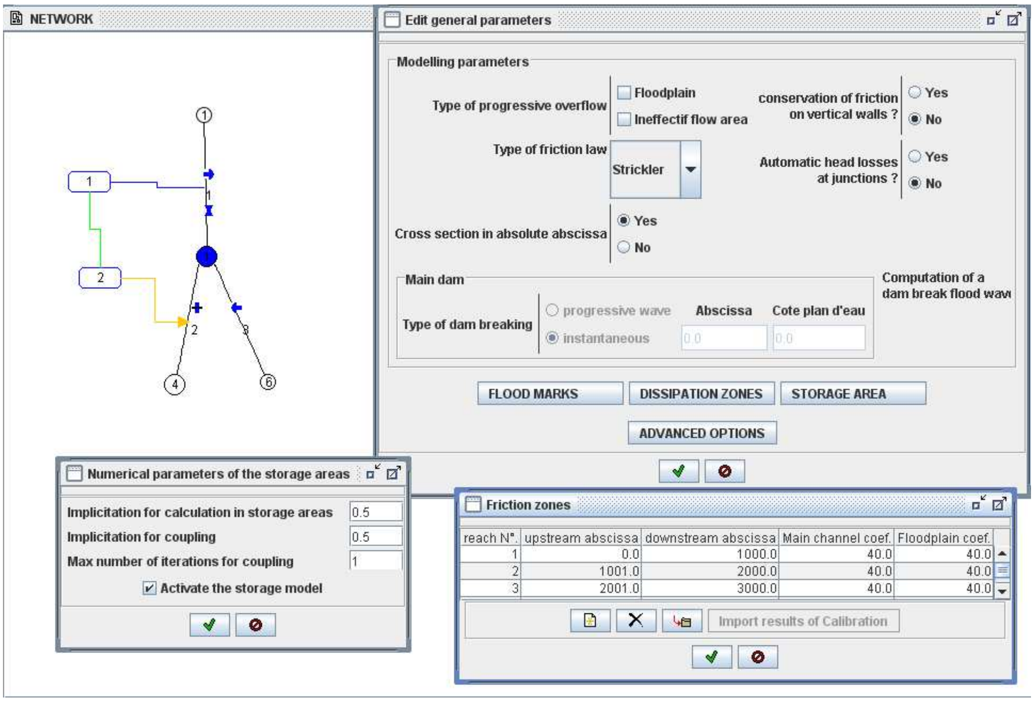


Figure 36: General Parameters window : the dissipation zones and the options for the Storage Areas

4.13 Output Parameters

The output parameters are quite standard in this case. In order to enter them, click on the “Output Parameters” point of the hydraulics menu. The window is divided in three parts : the first part defines the data points where results should be written (in space and in time), the second part defines the file format and limits the file sizes, and the third part defines the variables to write.

First, enter a title for the case study. The first time step to write is the time step 1 (due to a minor bug, it is not possible to enter here the time step 0). In the present case study, it is appropriated to write data all 10 time steps, as the time step is of 30 s, this means data are written all 300 s. The distance between reaches allows to connect the reaches at the junction. Spatial data should be written at all computation cross sections, as defined in the section “Mesh”.

Limit all file sizes as shown in the Figure 37. The output variables are the standard ones, MASCARET will add automatically all data corresponding to the storage areas, links and hydraulic singularities.

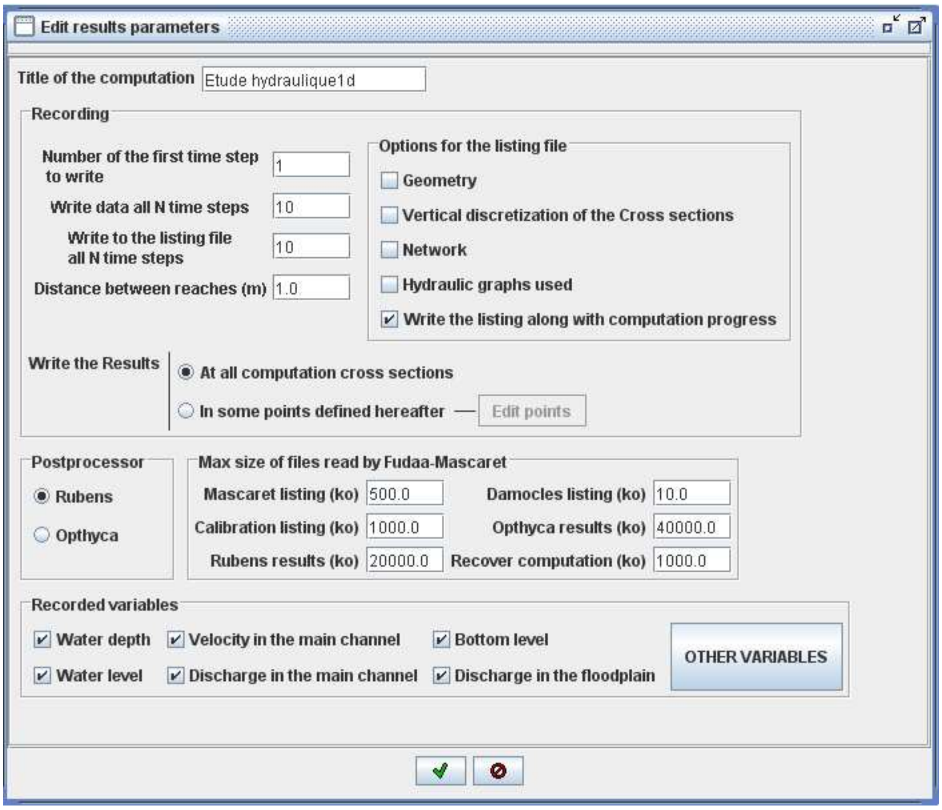


Figure 37: Output parameters for the application 3

4.14 Results

Once all parameters defined, save the current case-study by selecting **"Save as ..."** from the **"File"** menu of the main window. Enter a file name like "junction" and save the present study.

Now, click on the Compute button 

. The computation progress is shown in the lower part of the main window. Once the computation finished, the **General Results** window pushes up. Check for warnings in the first tab. Ignore the warning about the size of the Damocles listing file.

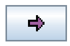
The next tab concerns the messages of the computation kernel. Check here the computation time needed. On the last line, the computation kernel should say “termine”.

Ignore the Damocles listing tab.

The `MASCARET` listing should say at the end “FIN”.

If everything was running fine, the Graphs point in the Results menu should be active. Click on it in order to check the results.

The upper left part of the window shows the list of reaches, storage areas and links. The middle left part allows to select a spatial or time profile for the data and to select the variables to plot. The lower left part will ask for the time step (in case of space profile) or for the abscissa (in case of time profile) to write. Note that it is NOT always consistent to select two reaches and to draw a space profile. This depends on the following choice : absolute or relative abscissa.

Select the reach 1 from the list of reaches, storage areas and links and draw a space profile by selecting “Profile”. Select the variables “Total discharge” and “spilled discharge” and the time step 300. (Note : multiple variables or time steps are selected by holding the key CTRL and clicking on the variables). Draw the graph by clicking on . It is possible to export the data to Excel for a more detailed analysis. Click on the button  in order to go to the next time step. Figure 38 shows the total discharge and the spilled discharge for the times 300 s and 1200 s. The spilled discharge corresponds to the discharge at the lateral weir, so at the position of the lateral weir, the total discharge is reduced by the spilled discharge. At the time 300 s, the flow is going from both ends of the reach towards the lateral weir, so the flow is negative behind the lateral weir. At the time 1200 s, the incoming flow reached the end of the reach, and the discharge is positive all over the reach. The lateral spillway is clearly visible by a difference in the total discharge in this point. At this time, the storage area 1 can be seen in the graph too : a slight increasing of the total discharge in the point where the link to the storage area is situated. Going forward in time, this link is more and more visible in the space profile of the discharge.

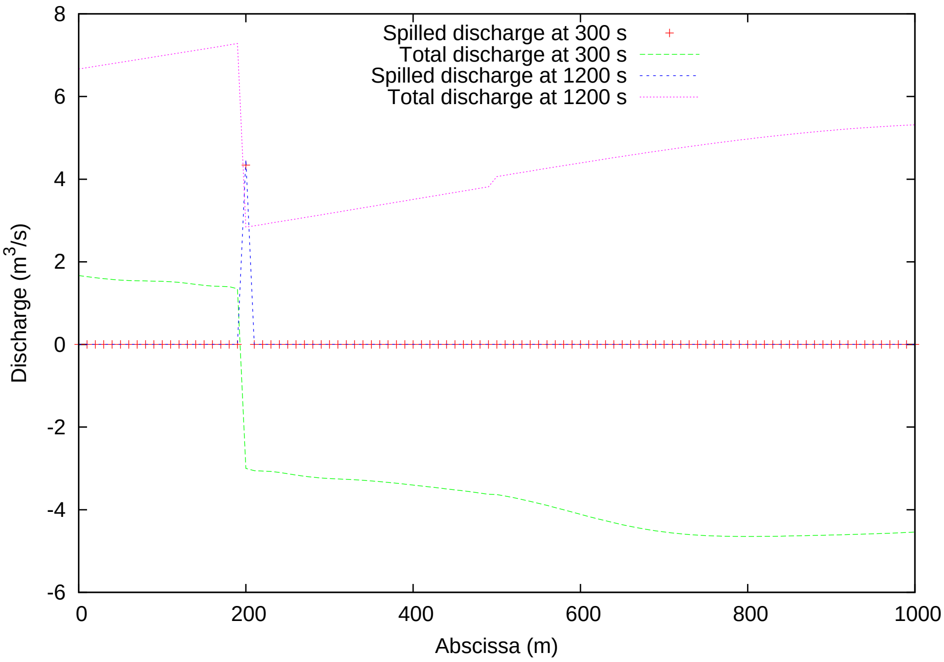


Figure 38: Space profile of the discharge for the reach 1, times 300 s and 1200 s

In order to plot the discharge exchanged by the link 1, select the link 1 and the Discharge in this link, as shown in Figure 39. The discharge is negative, means the flow is going from the storage area toward the reach and the discharge of the reach increases at the point where the link is connected. Analyzing the space profile of the discharge in the reach 2 and the discharge in the link 3, similar patterns can be found. The presence of the inlined spillway is visible too. Examining the flow pattern of the link 2 and the volumes of the storage areas, the conclusion is that the external inflow of the storage area 1 is the dominant variable. This external inflow causes the storage area 1 to supply as well the reach 1 as the storage area 2. So flow is going from the storage areas (1 and 2) towards the reaches (1 and 2).

For the reach number 3, the lateral inflow causes a jump in the discharge space profile at the point of the inflow.

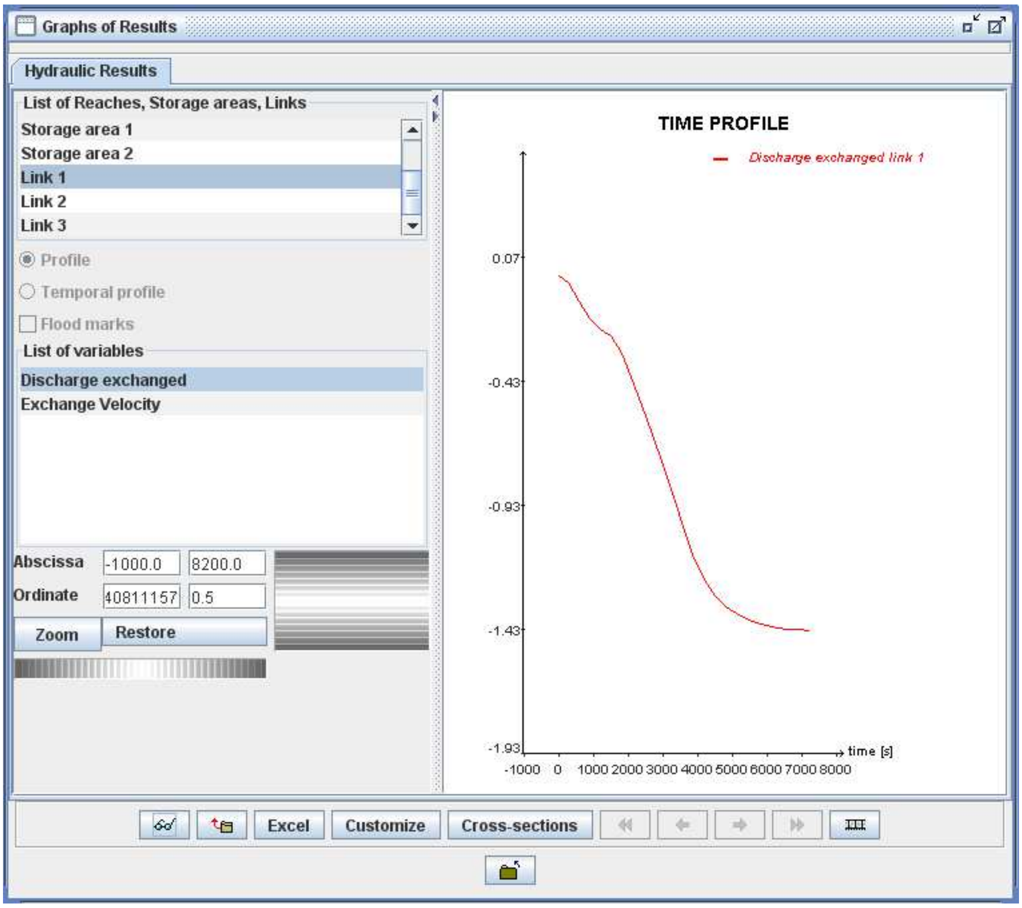


Figure 39: Discharge exchanged via the link 1 as a function of time

5 Application - Dam Break Wave

5.1 Purpose

This case study demonstrates the use of MASCARET for the computation of a dam break wave. Here, the real case of the dam Malpasset is considered.

Malpasset was a double-arched dam on the Reyran River, constructed from 1952 to 1954, in order to supply water and irrigation for the region. It is located in a narrow canyon of the river. The dam lake was filled with a volume of 55 Mm^3, when it collapsed in December 2, 1959, killing 423 people.

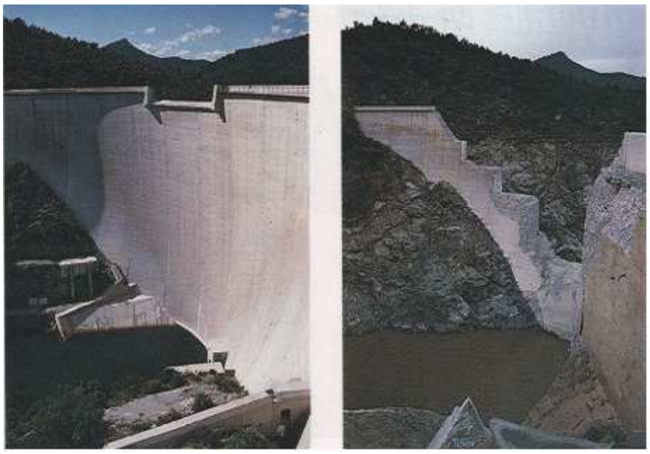


Figure 40: Dam of Malpasset : before and after the collapse

The three days before the dam break, heavy rain fall filled 4 m of the lake. The water level was only 28 cm from the edge. 5 hours before the breach, the water release valves were opened, too late to empty the dam in time.

The entire wall collapsed, large pieces of the dam were scattered throughout the area. The breach created a massive dam break wave, 40 m high and traveling with about 70 km/h. It reached Frejus in 20 min, still standing 3 m high.

This real case is considered as a reference case study for codes computing dam break waves.

A physical model was build up at LNH in 1964 and calibrated based on the observed events. This model contained 14 stations measuring the height of the dam break wave and its arrival time.

The present case study will model the dam breaking as an instantaneous collapse of an initial water level, simulating the filled dam lake. Thus, in the opposite of a progressive dam breaking, all the reservoir is integrated in the geometry.

All data files necessary to build up this case study can be found in the directory MALPASSET.

5.2 Geometry

5.3 General considerations

The dam break wave modified large parts of the valley, so the geometry is taken from maps dating from before the collapse. The user can get the cross-sections from the file cross-sections.xls of the example directory. The actual topography is shown on Figures 41 and 42.

The location of the cross-sections used for the computation are shown on Figure 43.

5.4 Mesh Parameters

The location of the dam is at abscissa 0. The upstream part (the reservoir) is considered for the whole length of the lake, 4820 m. The valley (from the upstream part of the lake down to the Mediterranean Sea) has a total length of about 20 km.

The case study does not take into account the singularities present in the real case : The dam was not completely destroyed; 1.5 km downstream : a bridge and parts of the highway were destroyed by the dam break wave; 9.5 km downstream : a railway.


la Gardiette
la Peyrière
Ambon
Meissonnière
Rossignole
le Pin de la Légue
le Capitou
de
l'Esterel
Quartier Lecocq
le Bonfin
Capitou
D 837
le Gardalen
Zone
Industrielle
du Capitou
Hôpital
Militaire
Quartier
Robert
St-Pons
Ste Brigitte
la Tour de Mare
la Muscadière
les Bosquets
la Combe de Roche
Malbousquet

Figure 41:

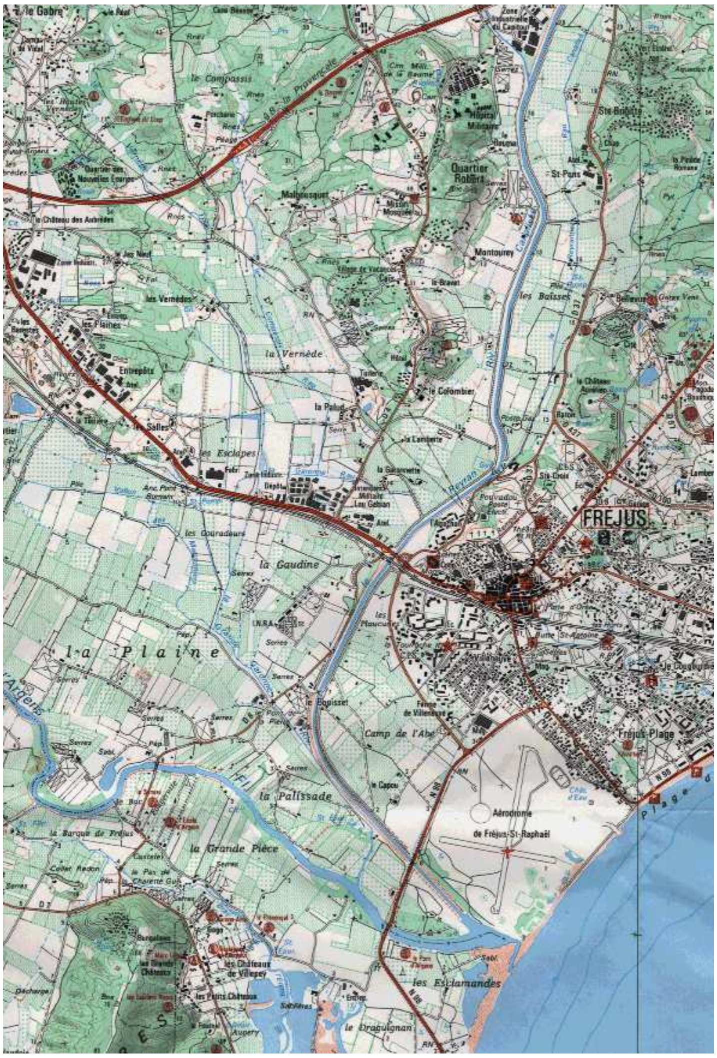

FREJUS
la Plaine
la Vernède
la Gaudine
la Palissade
la Grande Pièce
Camp de l'Abé
les Esclapes
Aérodrome
de Fréjus-St-Raphaël
les Esclamandes
Fréjus-Plage

Figure 42:

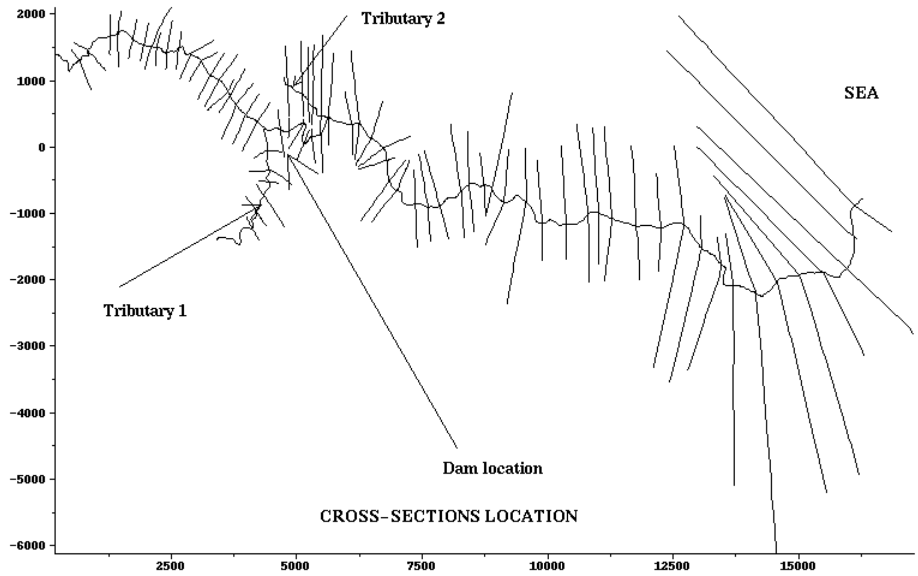


Figure 43: Cross-sections location for the case study Malpasset

5.5 Building up the model

Open FUDAA-MASCARET. Create a new case study by clicking on the **"Hydraulics"** point of the main menu and then on **"Computation Kernel"**. Select the Super-critical kernel from the scroll-down list and confirm by clicking on the valid button .

Now, go to the next entry of the **"Hydraulics"** menu : the **"Hydraulic Network"**. Create a single reach : click on the button  in order to open the **"hydraulic structures editor"**. Create a single reach by clicking on the symbol  in the hydraulic structures editor. Toggle off the interactive mode for the hydraulic network by clicking on . In the hydraulic network window, a simple reach with the open boundaries 1 and 2 is created.

5.6 Cross-sections

Cross-sections are quite complex for this real case study. Click on the reach, and the cross-section editor pops up. Import the cross-sections by clicking on the button . Choose the excel file format *.xls* and import then the file *malpasset-cs.xls*. You may view the cross-sections one by one by clicking on the edit button .

5.7 Boundary Conditions

The level of the dam lake is 100 m, the level of the sea (open boundary 2) is 0 m. The open boundary 1 is the upstream part of the dam lake. The initial level (before the dam break) is of 100 m, a flow of 50 m^3/s is considered coming from the upstream reach, filling the lake. For the dam break computation, the upstream boundary condition is then a stage and flow hydrograph with a fixed flow (the level changes, once the dam broken).

Click on the open boundary 1 in the hydraulic network window. The open boundary editor pops up. The boundary condition is supplied by graph. Click on the EDIT GRAPH button, in order to create a new Flow and Stage Hydrograph. Enter the values for the level (100 m) and for the flow (50 m^3/s) for two times : 0 s and 100000 s. Confirm the values and back to the open boundary editor, select the option “Fixed Flow”.

For the downstream open boundary, the boundary condition is supplied by computation (free outflow). Click on the open boundary 2 in the network window and select the corresponding options from the open boundary editor.

5.8 Mesh Parameters

5.9 Grid size

From the main menu, select the Hydraulics entry and then the Mesh point. In the present case study, the mesh is a constant grid of 25 m. For the computation of the grid size, select “Grid Map” from the scroll-down list. Click then on the button “Edit Computation Sections”. A spread-sheet pops up, allowing the user to enter a grid size for parts of a reach. Here, the grid size is the same for the whole reach. Enter the number of the reach, 1. The abscissa will auto-complete, enter a grid size of 25 m. Confirm by clicking on . You choose the option "Computational section on the physical ones". So all the cross-sections will be included in the mesh and consequently the distance between 2 sections could be different from the fixed space step.

5.10 Vertical discretization

The dam break wave considered here reached a height of 40 m. The vertical discretization used will be of 2 m, sufficient to analyze the results. Enter the value of 2 m for the reach number 1, once clicked on the point "Vertical discretization of the cross-sections" of the "Hydraulics" menu.

5.11 Initial Conditions

The initial condition is the filled dam lake. All other parts of the model are considered to be dry here, even if a small amount of water was present in the real case.

In order to enter the **initial conditions**, select the corresponding point in the **"Hydraulics"** menu. Answer **"Yes"** to the question **"Presence of an initial water level"**. Click on the button "Initial Water Level" in order to open the spread-sheet, allowing to enter the level values. The lake is filled, and the level is 100 m between the abscissa -4820 m and 05 m, for all other parts, the level is 0. Enter the values as shown in Figure 44. Validate these entries by clicking on the button .

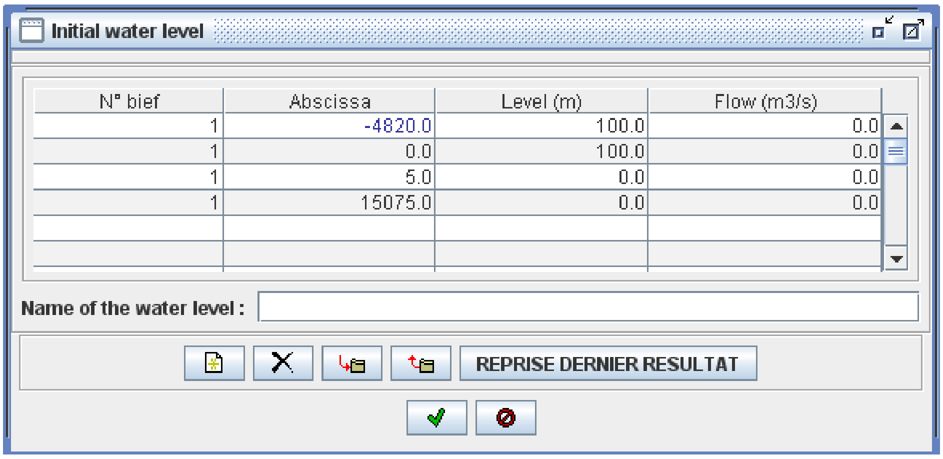


Figure 44: Initial water level for the Malpasset case study

Back to the initial conditions window, click on the button **"DRY AREAS"** in order to define the zone initially dry. For the reach number 1, from abscissa 5 m to 15075 m, the valley is considered dry. Enter these values in the spread-sheet and confirm by clicking on .

The initial water level and the dry zones may be viewed by clicking on the **VIEW** button of the initial conditions editor (see Figure 45)

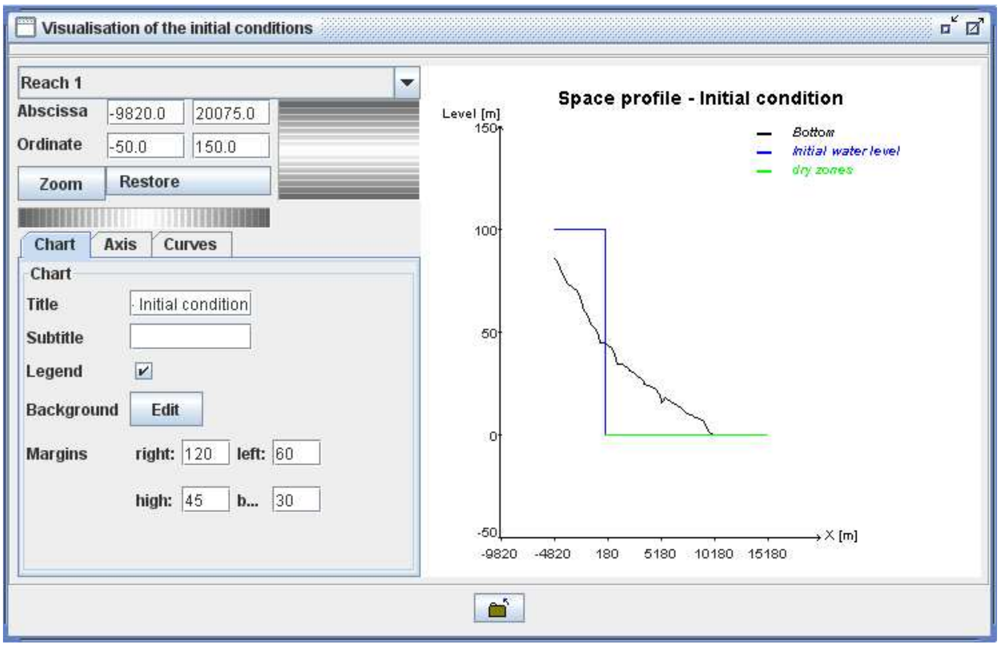


Figure 45: The initial conditions for the Malpasset case study

5.12 Temporal Parameters

Click on the point **"Temporal Parameters"** of the **"Hydraulics"** menu. In the temporal parameter editor, the initial time should be set to 0, the (initial) time step to 1 s. The max time is set to 20000 s.

A variable time step is taken (as a super-critical computation is considered here), with a Courant number of 2. The general case would be a Courant number of 0.8 with the explicit kernel, but as the implicit kernel will be used (see next section), a Courant number of 2 will be appropriated.

5.13 General Parameters

The next step concerns the general parameters of the computation. Click on the **"General Parameters"** point of the **"Hydraulics"** menu. The friction law used is Strickler and the friction on vertical walls is conserved. Here, a instantaneous breaking of the dam at a level of 100 m is considered. (see Figure 46).

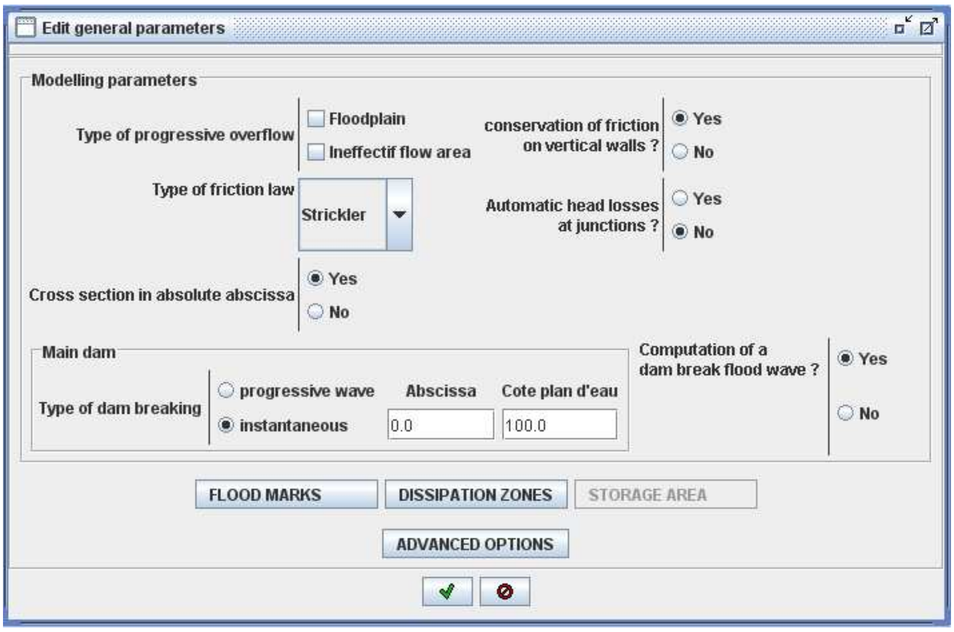


Figure 46: General Parameters for the Malpasset case study

Click on the button **"DISSIPATION ZONES"** located at the bottom of the window in order to enter the friction coefficients. A spread-sheet pops up and the following values should be entered :

Reach number	Upstream abscissa	Downstream abscissa	Main channel coefficient	Floodplain coefficient
1	-4820	0	35	35
1	0	199	35	30
1	199	498.1	15	15
1	498.1	851.3	5	10
1	851.3	1232.9	10	25
1	1232.9	15075	25	30

Note that the small values (high friction) are used in order to represent the 2D effects of the double bend of the river.

The "Advanced Options" button allows to change the options for the implicitation. Here, activate the implicitation for the friction and the super-critical kernel. For dambreak wave simulation, the option "implicitation for the friction" is compulsory. It avoids negative velocity of the dam-break wave. You can disable "implicitation of the supercritical kernel" but in this case the Courant number must be lower than 1. We recommand the value 0.8. Even you choose the option "implicitation of the supercritical kernel" , beware of the Courant number condition (for a dam-break simulation not greater than 2).

5.14 Output Parameters

Click on the **"Output Parameters"** point of the **"Hydraulics"** menu. Enter the values represented on the Figure 47.

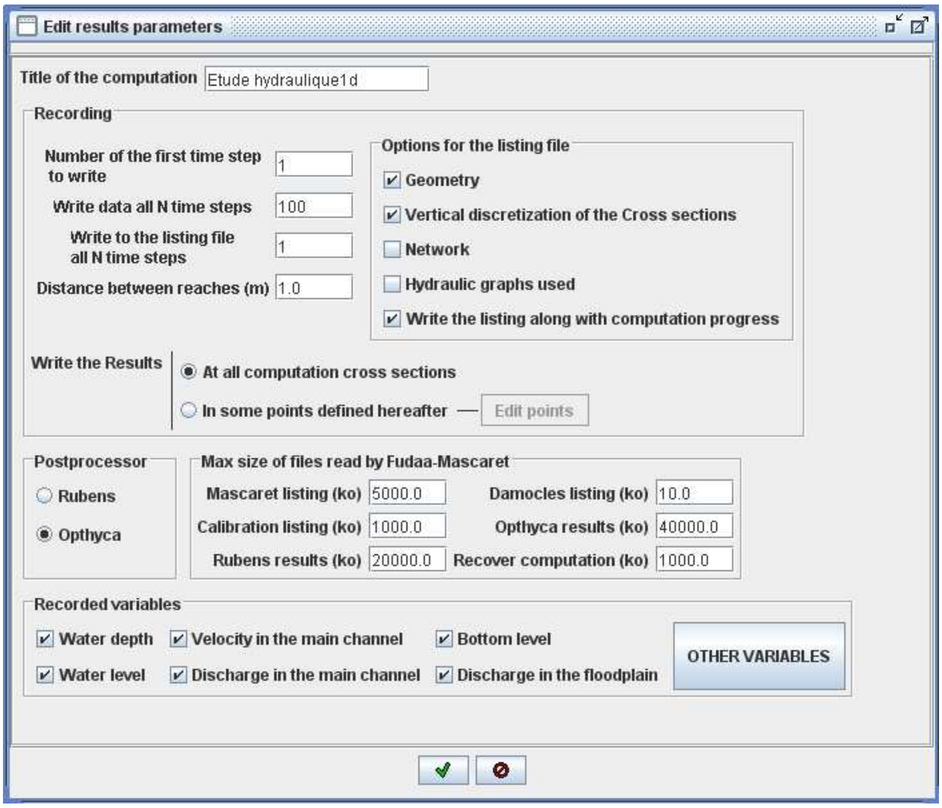


Figure 47: Output parameters

5.15 Results

Run the computation by clicking on the button 

on the tool bar. The current status of the computation is shown in the status bar. Once the computation finished, the listing window pops up. The computation kernel should say “termine” and the last message of the `MASCARET` listing should be “FIN”. Ignore all warnings of Damocles.

In order to draw the results, click on the point **“Graphs”** of the **“Results”** menu. Here, select the reach 1 (the only one). The user can then draw spatial and temporal profiles. In order to draw the arrival time of the dam break wave at the different points, select a spatial profile, the variable “Arrival time” and the last time step of the computation. In order to super-impose the measured data points, activate the “Flood Marks”. Click on in order to show the graph in the right hand part of the window. Results are compared to the data coming from the physical model build at LNH in 1964. These data are the arrival time of the dam break wave at different points and its height. Figures 48 and 49 show a quite good agreement between the data and the computation for the arrival time as well as for the maximum water level.

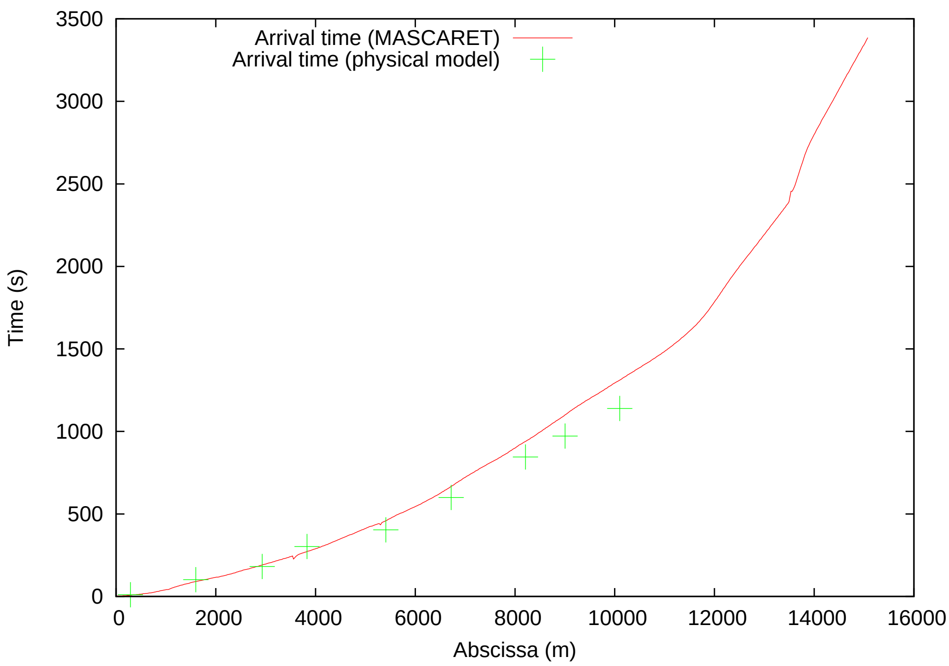


Figure 48: Arrival time of the dam break wave : comparison of the `MASCARET` simulation and the data of the physical model

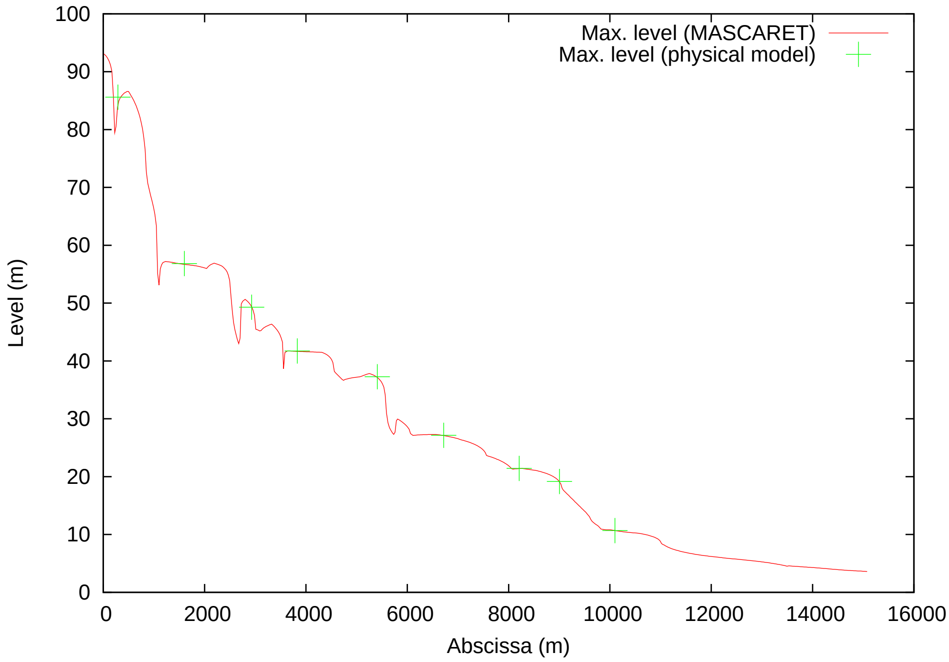


Figure 49: Maximum water level reached : comparison of `MASCARET` simulation and the data of the physical model

6 Application - Water quality analysis

6.1 Purpose

This test case illustrates the use of the water quality module for simulating the solute transport in an uniform channel: the convection and diffusion of a single tracer in a channel. The channel is 25 m long, 0.2 m wide, and has a slope of 0.247%. A steady flow rate of 0.0156 m^3/s is imposed at the upstream end of the channel.

The tracer is injected at the upstream end of the channel as a continuous function. Two empirical formulas for computing the dispersion coefficient are used and compared.

6.2 Hydraulic Parameters

Open `FUDAA-MASCARET`. Click on the "Hydraulics" point of the main menu. The only active entry is then "Computation Kernel". Click on it and select the Steady computation kernel.

6.3 Geometry

The next active entry in the "Hydraulics" menu is the "Hydraulic Network" point. Click on it, the network window will be pushed up in the interactive mode. The interactive mode may be toggled on or off by clicking on the Interactive Mode button If the interactive mode is toggled on, the buttons of the Hydraulic Network Editor are active. In order to draw the hydraulic network (a simple channel in this case), the interactive mode should be toggled ON.

The channel is represented by a simple reach. Click on the button of the Hydraulic Network Editor. A window pops up, showing the different elements of a hydraulic network described by `MASCARET`. Click on the first button (- reach), and the reach shows up in the Network window.

Because no other component is needed for this case study, toggle off the interactive mode. The Network window shows then the reach with the ID 1 and the open boundaries 1 and 2 as shown in Figure 50.

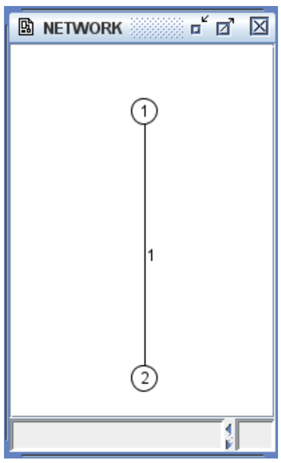
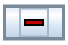


Figure 50: Single reach in hydraulic network editor

Now, the data describing the bottom of the channel, and so the weir, should be supplied. Move the cross in the network window to the center of the reach and click on it. The window describing the bottom level of the reach number 1 pops up. You may enter the data point by point or import directly a data file describing the channel (file *bottom.pro* from the example directory).

In the main window describing the reach, you should enter all points describing the cross-sections. In order to create a cross section, enter first a name in the column "title" and then the abscissa. Here, two cross-sections should be defined : the Upstream end at abscissa 0 m and the Downstream end at abscissa 25 m. Enter these data end select the first point, abscissa 0 m. Click on the "Create" button . A window will pop up, asking the shape of the cross-section at the abscissa 0. We need a simple channel with vertical walls here, click on the  button in order to create a user defined cross-section.

The following window "Edit Profile" allows to enter the points defining the cross-section. Enter the four points defining a channel with bottom level 10 m, width 0.2 m and wall level 10.4 m. Validate the entry by clicking on the  button. Ignore the warning about vertical walls.

The channel has a constant slope and the bottom level of the downstream end is of 9.93825 m. As described for the upstream cross-section, create now the cross-section for the downstream abscissa, but this times with a rectangular channel of width 0.2, bottom level 9.93825 m and wall level 10.4 m.

6.4 Boundary Conditions

The boundary conditions are defined by clicking with the mouse on the circle representing the open boundaries. The upstream open boundary is the circle number 1. Clicking on it will open the boundary condition editor.

The upstream boundary condition will be a constant inflow discharge of 0.0156 m^3/s: a constant flow hydrograph should be supplied. In the open boundary editor, select "**boundary condition supplied by graph**". Now, click on the **"EDIT GRAPH"** button, and the **"Graph Manager"** window shows up. Click on the "**Create**"  button. Select a new **"Flow Hydrograph Q(t)"**. The **"Hydraulic Graph"** window provides a spreadsheet and a graphical view of the data. Define a constant flow hydrograph by the means of two points : times 0 s and 50 s (at least over the duration of the simulation). For both time steps, enter a flow value of 0.0156 m^3/s. The graph in the right part is updated and shows the flow as a function of time as defined by the spreadsheet (see Figure 51).

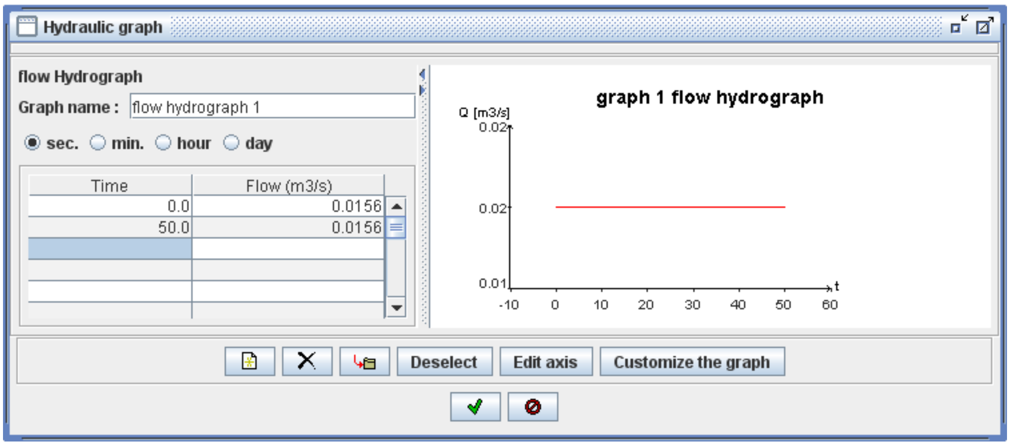


Figure 51: Flow hydrograph for the upstream boundary condition.

The boundary condition for the open end 2 is a constant stage hydrograph of 10.085 m. Proceed as described before, but select a **"Stage Hydrograph"** as boundary condition. Enter the values for two times, 0 s and 50 s.

6.5 Mesh Parameters

Two types of Mesh parameters should be supplied for `MASCARET` : the computational cross-sections and the vertical discretization of the cross sections.

The computational cross sections may be defined by the physical ones (defined above by the cross-sections describing the geometry), by a grid map (fixed mesh size by parts), one-by-one by the user or recovered from a previous computation. Here, we will use a fixed mesh size for the whole length of the reach. Select **"Hydraulics**" from the main menu, and then "**Mesh**". From the scroll-down list, select "computation cross sections from a grid map". Click on the button **"Edit computation sections**", a spread-sheet appears. Enter then for the reach N. 1 (upstream abscissa 0 m, downstream abscissa 25 m) a constant grid size of 0.1 m.

After this step, the menu point "**Vertical Discretization of the Cross Sections**" becomes active. It is now possible to define the vertical discretization by the means of a spread-sheet. In this case, the vertical discretization is constant for the reach number 1, from abscissa 0 to 25. A value of 0.1 m is chosen for the vertical step.

Close all open dialogue windows by confirming your choices.

6.6 Initial Conditions

As we consider the steady computation kernel, no initial conditions are supplied.

6.7 Temporal Parameters

Select the point **"Temporal Parameters"** from the **"Hydraulics"** menu. Here, the temporal parameters supplied are the initial time (0 s), the time step (0.01 s) and the maximal time for the computation (50 s). These parameters will be used by the water quality model too.

6.8 General Parameters

In this very simple example (for the hydraulics part), the general parameters are not changed. Click on the button *Dissipation Zones* to give the values of the friction coefficients.

6.9 Output Parameters

Select the point **"Output Parameters"** from the **"Hydraulics"** menu. First, enter a name for the case study, for example "Channel with water quality".

The upper part of the window requires input for the frequency of the output. A steady state for the hydraulics part is considered, but the frequency of writing into the files is used by the water quality model too. So, enter the following parameters : first time step to write is 1, write data all 10 time steps to the data and the listing file.

Keep the default values for the file size limits and the variables to write to the file.

6.10 Water Quality Parameters

6.11 Water quality model

Click on the **"Water Quality"** point of the main menu. Here, the parameters for the water quality model will be supplied. Select the first point from the menu list, **"Water Quality Model"** in order to open the window **"Choice of the water quality library"**. First, activate the water quality model by clicking on the check-box. Here, the transport of a simple tracer is considered, so select from the scroll-down list the water quality library "TRANSPORT_PUR". The number of constituents is 1 (see Figure 52).

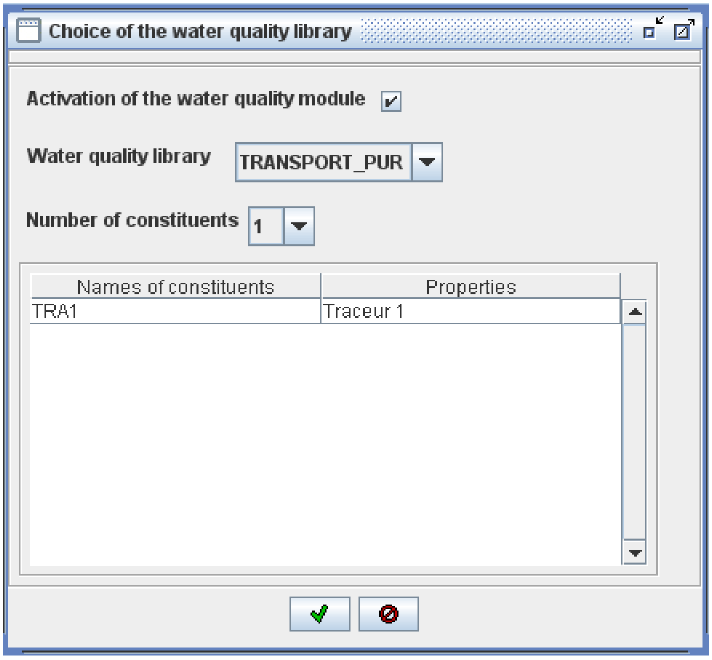
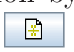


Figure 52: Choice of the water quality library for the channel test case

Confirm all choices by clicking on Confirm .

6.12 Boundary Conditions

Now, as the water quality model is activated, the boundary conditions should be defined for the water quality computations. Click on the open end number 1 in the Network window. The boundary conditions editor pops up. Now, the lower part of the window allows to define the boundary conditions for the water quality model. For the upstream open end (number 1) the type of the boundary condition is Dirichlet. Select this by clicking on the corresponding check-box.

Now, create a new graph for this boundary condition by clicking on the "Edit Graph" button. Create a new graph by selecting  in the graph manager. A spread-sheet pops up, allowing to define the boundary condition

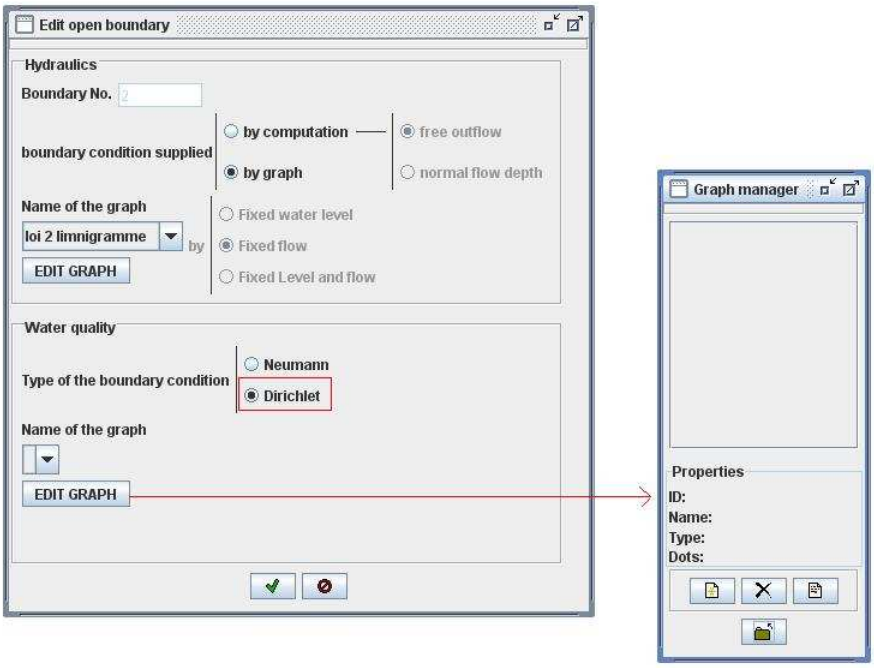


Figure 53: Upstream condition for water quality

as the concentration of the constituent 1 as a function of time. Enter the following values here :

6.13 Initial concentrations

In the present case study, the initial concentration is zero all over the domain. So it is not necessary to modify them.

6.14 General Parameters

The "General Parameters" point of the "Water quality" menu allows defining the computational parameters for the water quality module. The frequency of coupling is, in the present case, equal to 1, i.e. the time step is identical in both `MASCARET` and Tracer (no other option available in the current version).

The constituent should be transported by convection and diffusion. The convection method chosen here is a finite volume scheme of order 2, the slope limiter should be activated. For the diffusion method, a first run will

Time (s)	Concentration
0	0
2.16	17.8025
2.81	35.605
3.5	53.4075
5	71.21
6.28	53.4075
7.11	35.605
8.22	17.8025
25	0
50	0

be executed with the method of Fisher (1975), a second run with the more recent method of Kashefipur & Falconer (2002).

Figure 54 shows the general parameters window. Confirm all choices by clicking on .

6.15 Output Parameters

In order to finalize the parametrization of the water quality model, click on the point "Output parameters" of the "Water quality" menu. Choose a post-processor, for example Opthyca. The upper right part of the window allows the user to define the data to write to the file. As the initial concentration is zero all over the domain, it is not necessary to write this to the file. Choose to write the concentrations only. As a single constituent is used, a constituent balance is not necessary.

The frequency of writing to the files is the same as defined for the hydraulics output.

6.16 Results

Once all parameters entered, save the case study. Click then on the "Compute" button 

in order to launch the computation. The computation progress is shown in the status bar. Once the computation finished, the window showing the listings of the different modules pops up. Ignore all warnings of the Damocles part.

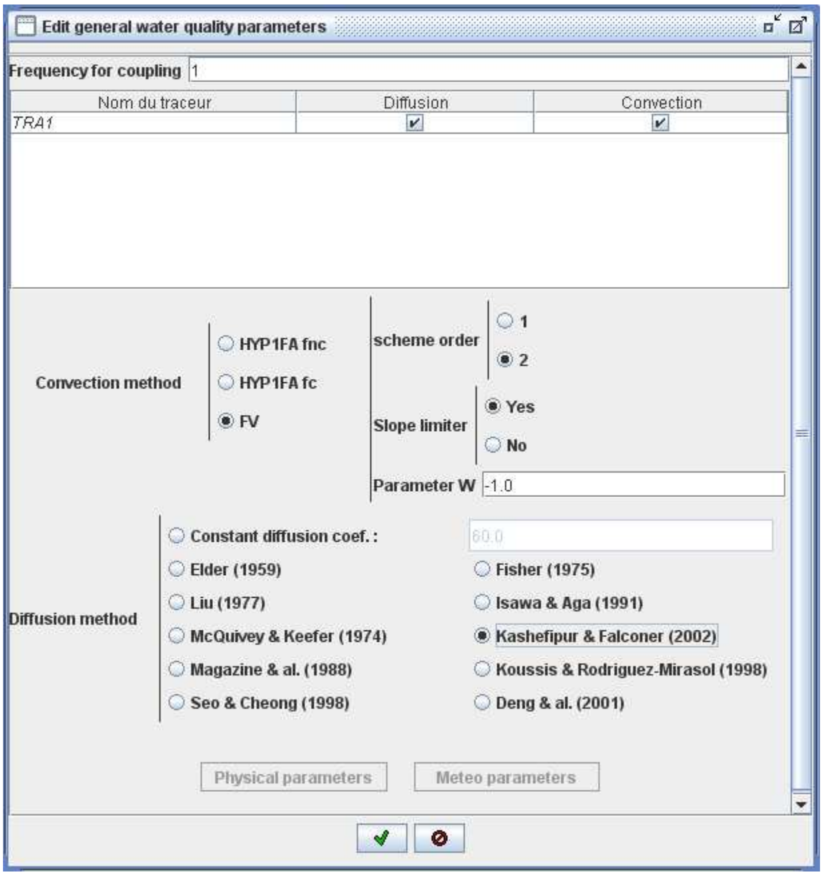
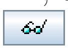
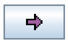


Figure 54: General Parameters window of the water quality module

If the computation finished, the point “Graphs” in the “Results” menu becomes active. Click on it in order to analyze the results of the computation.

The hydraulics results should just show the steady flow in the channel, in order to check it, choose the Hydraulic Results tab of the Graphs window. Select the reach 1, a temporal profile, the discharge variable and a arbitrary cross-section. Click on the button  in order to draw the graph. It should be a line with a constant discharge. It is possible to go to the next cross-section by simply clicking on .

Now, select the Water quality tab. The interesting data here are the time profiles of the concentration at different cross-sections. Select the Reach 1, the temporal profile option, the variable Concentration of constituent 1 and the first cross-section. Click on the button  in order to draw the graph. It should be a line with a constant discharge. It is possible to go to the next cross-section by simply clicking on  or to choose multiple cross-sections by holding the key CTRL while selecting the cross-sections with the mouse. Figure 55 shows the water quality results window for the present case with the cross-sections 0 m, 4 m, 8 m and 12 m.

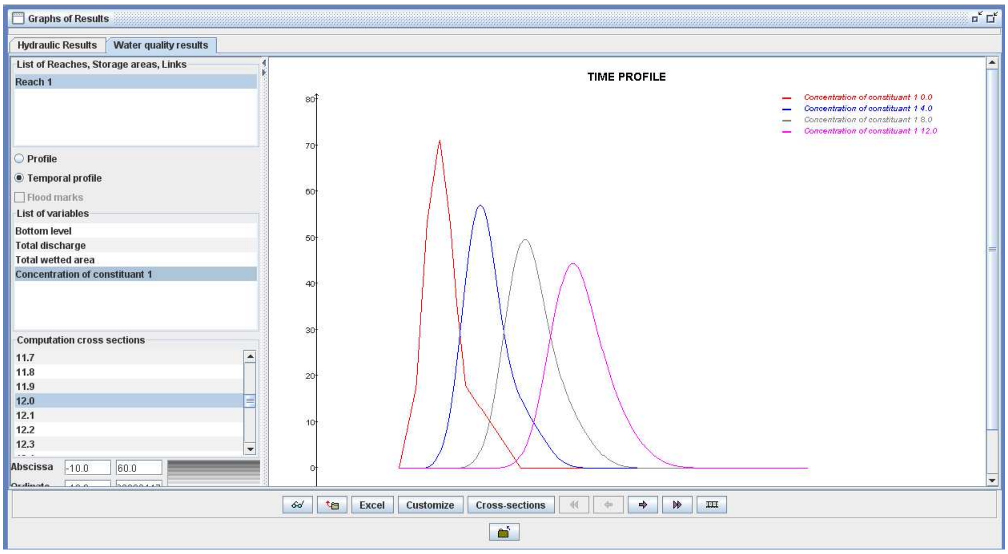


Figure 55: Results for the water quality module with time profiles at multiple cross-sections

The computation should be executed with two different diffusion methods (see the general parameters part of the water quality module). Figure 56 shows the space profiles of the concentration for both methods, at different cross-sections. The method of Fischer has a more diffusive action on the constituent, the cloud becomes larger and the maximum concentration is lower than for the method of Kashefipur & Falconer.

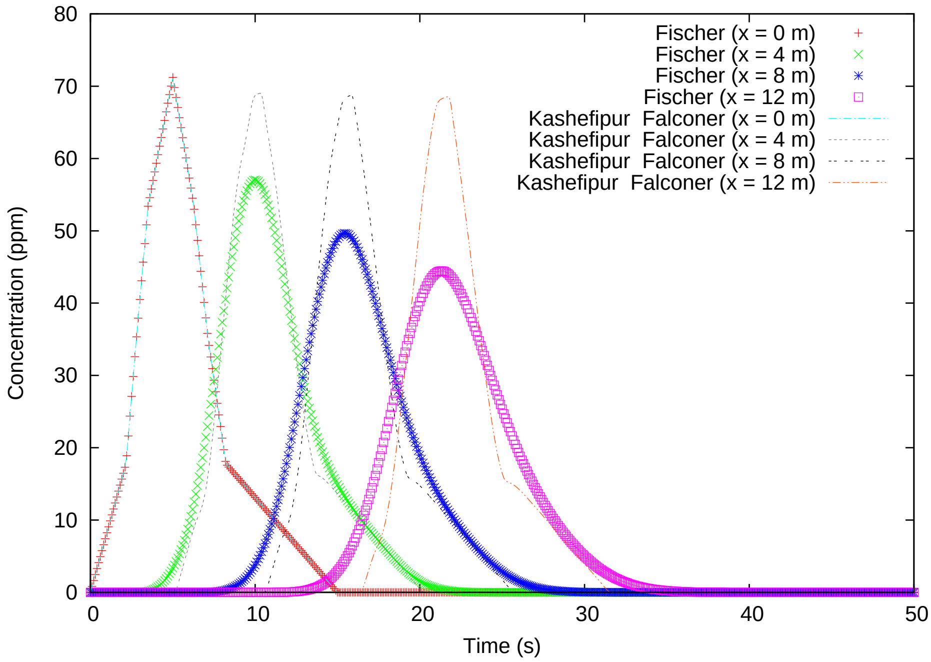


Figure 56: Comparison of the time profile for two different methods for the computation of diffusion : Fischer (1975) and Kashefipur & Falconer (2002)

References

[1] N. GOUTAL and F. MAUREL, *Note de principe de la version 4.1 du code* ***MASCARET***, EDF Report HE-43/96/075/B, 1996

[2] N. GOUTAL and F. MAUREL, *Proceedings of the* 2^{nd} *workshop on the dam-break wave simulation*, EDF Report HE-43/97/016/A, 1997

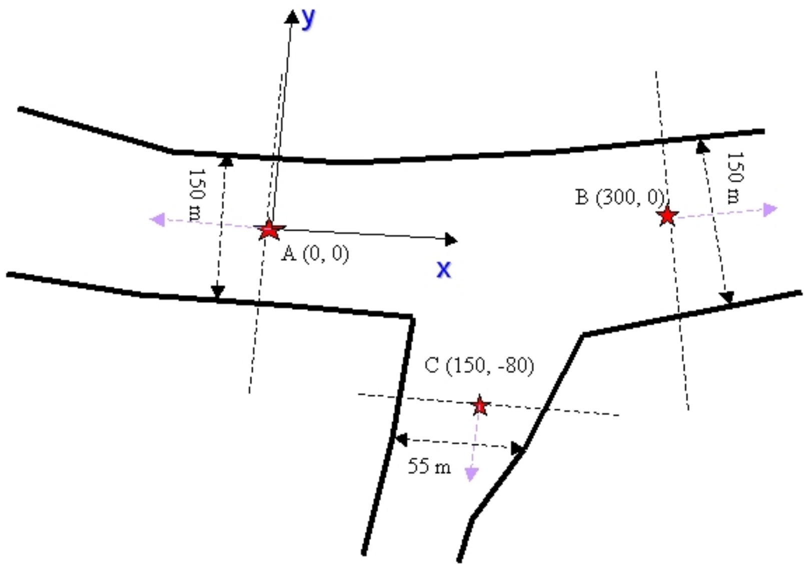
[3] F. MAUREL, *Traitement des confluents dans le logiciel* ***MASCARET*** *4.0 - Principes de la méthode et éléments de validation*, EDF Report HE-43/96/067/A,1996

[4] N. GOUTAL and F. ZAOUI,*Note de principe du code Mascaret 7.1*, EDF Report H-P73-2012-00188-FR

7 Appendix

7.1 Junction parameters for the super-critical flow

The points A, B and C are located in the center of the last cross-section of each river at the junction. The point A is considered the center of the cartesian coordinate system defined by the axes (Ax, Ay). The coordinates of the points B and C are then measured in this coordinate system. In order to define the angle α, the orthonormal, external vectors of all three points are taken into account. The angle α of the external orthogonality of each cross-section is measured in the cartesian coordinate system in the trigonometric sense.



7.2 Description of the file formats

7.2.1 Importing of Storage Areas files

The geometry of the storage areas can be described by three different ways :

- Points data describing the geometry
- Boundary points
- Rating curves

File of the points data describing the storage areas (*.geo).
CASIER A
1551.0 3742.0 150.0 F
1537.0 4063.0 150.0 F
1650.0 4417.0 150.0 F
2057.0 4870.0 150.0 F
2365.0 4842.0 150.0 F
2410.0 4382.0 150.0 F
2544.0 4024.0 150.0 F
2046.0 3917.0 150.0 F
1551.0 3742.0 150.0 F
1551.0 3742.0 150.0 F
1821.0 4405.0 141.2 I
1820.0 4404.0 141.2 I
1819.0 4097.0 142.2 I
2181.0 4728.0 142.5 I
2188.0 4472.0 142.5 I
2227.0 4284.0 142.5 I
2232.0 4013.0 142.5 I
2023.0 4201.0 142.8 I
2129.0 4101.0 143.3 I
CASIER B
... F for boundary point - I for inner point

File of boundary points of the storage area
XYZ(*.txt) CASIER A
1551.0 3742.0 150.0
1537.0 4063.0 150.0
1650.0 4417.0 150.0
2057.0 4870.0 150.0
2365.0 4842.0 150.0
2410.0 4382.0 150.0
2544.0 4024.0 150.0
2046.0 3917.0 150.0
1551.0 3742.0 150.0

File of rating curve : elevation - area (*.casier).
CASIER casier1
5.0 250000.0 0.0
6.0 250000.0 250000.0

7.0 250000.0 500000.0
8.0 250000.0 750000.0
9.0 250000.0 1000000.0
10.0 250000.0 1250000.0
11.0 250000.0 1500000.0
12.0 250000.0 1750000.0
13.0 250000.0 2000000.0
14.0 250000.0 2250000.0
15.0 250000.0 2500000.0

7.2.2 Importation of geometry file for a reach

It is the description of the file containing the cross-sections of all reaches. The cross-sections are given following an increasing order of the abscissa.The reaches are classified by increasing order.
File of the geometry of the reaches (*.pro).

PROFIL $Bief_1$ 1 0.0
0.0 48.3 T
0.0 43.49 T
31.7 43.44 T
49.92 43.59 T
51.99 42.98 T
92.12 42.8 T
122.43 42.56 T
151.88 42.6 T
183.9 42.39 T
211.4 41.8 T
241.93 40.13 T
272.92 40.77 T
300.7 38.3 T
332.43 38.96 T
691.98 42.09 T
724.07 40.97 T
842.03 41.89 T
853.43 40.35 T
874.09 40.57 T
904.92 39.96 T
934.64 39.15 T
969.62 39.09 T

983.56 39.19 B
985.77 39.19 B
988.27 38.0 B
991.07 35.9 B
996.82 33.9 B
1021.32 34.15 B
1025.57 35.35 B
1029.07 35.9 B
1035.57 38.0 B
1037.38 39.23 B
1040.47 39.34 B
1047.84 42.9 T
1093.56 44.11 T
1126.59 48.3 T
1126.59 48.3 T
PROFIL $Bief_1$ 2 60.17
0.0 48.28 T
...

T for topography (flood plain) - B for main channel
File of the cross-sections with a text format(*.txt).

profil 1
0
45
0 43.49
31.7 43.44
49.92 43.59
51.99 42.98
92.12 42.8
122.43 42.56
151.88 42.6
183.9 42.39
211.4 41.8
241.93 40.13
272.92 40.77
300.7 38.3
332.43 38.96
362.38 39.2
393.01 39.27

421.85 38.98
453.2 39.02
483.02 38.96
513.79 38.88
643.52 38.8
659.81 42.4
691.98 42.09
724.07 40.97
811.25 40.4
842.03 41.89
853.43 40.35
874.09 40.57
904.92 39.96
934.64 39.15
969.62 39.09
983.56 39.19
985.77 39.19
988.27 38
991.07 35.9
996.82 33.9
1009.07 33.75
1021.32 34.15
1025.57 35.35
1029.07 35.9
1035.57 38
1037.38 39.23
1040.47 39.34
1047.84 42.9
1093.56 44.11
1126.59 48.3
983.56 1040.47
36 13
profil 2
60.17
53
...

7.2.3 Importation of the graph file (*.loi).

File for a stage hydrograph.
hydrograph z

Temps (s) Cote
S
0.0 37.5
150000.0 37.5
File for a rating curve.

tarage znp

Cote Debit
37.0 300.0
38.5 450.0
39.1 550.0
39.5 600.0
File for a stage flow-hydrograph.

Stage-flow hydrograph

Temps (s) Cote Debit
S
0.0 418.0 0.0
100000.0 418.0 0.0

7.2.4 File restart simulation *.rep

RESULTATS CALCUL,DATE : 5 SEPTEMBRE 2005, 16 H 9
FICHIER RESULTAT MASCARET

IMAX = 31 NBBIEF= 1
I1,I2 = 1 31
X
0.0000000E+00 2.000000 4.000000 6.000000 8.000000
8.200000 8.400000 8.600000 8.800000 9.000000

9.200000 9.400000 9.600000 9.800000 10.00000
10.20000 10.40000 10.60000 10.80000 11.00000
11.20000 11.40000 11.60000 11.80000 12.00000
14.16667 16.33333 18.50000 20.66667 22.83333
25.00000
Z
0.4596793 0.4409126 0.4420789 0.4274720 0.4165675
0.4171953 0.4132192 0.4087039 0.4036271 0.3979653
0.3916991 0.3848078 0.3772775 0.3691527 0.3606673
0.3523874 0.3451217 0.3395015 0.3356378 0.3332120
0.3317785 0.3309636 0.3305122 0.3302676 0.3300386
0.3300028 0.3300002 0.3300000 0.3300000 0.3300000
0.3300000
QMIN
0.1800000 0.2519228 0.2515860 0.2073124 0.1618931
0.1460021 0.1361202 0.1257816 0.1149614 0.1036408
9.1798715E-02 7.9424441E-02 6.6562995E-02 5.3421162E-02 4.0529970E-02
2.8789369E-02 1.9147445E-02 1.2060605E-02 7.3135756E-03 4.3268478E-03
2.5150883E-03 1.4390950E-03 8.0939330E-04 4.4543188E-04 6.6328786E-05
4.9398795E-06 3.0439301E-07 1.5746586E-08 6.8334288E-10 2.4798563E-11
7.6527104E-13
QMAJ
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00
0.0000000E+00
VMIN
0.3995055 0.5829388 0.5806234 0.4947978 0.3965106
0.3928378 0.4070159 0.4184329 0.4255613 0.4264573
0.4187086 0.3995887 0.3665689 0.3184757 0.2573930
0.1902694 0.1275918 7.8132674E-02 4.4514839E-02 2.4097022E-02
1.2593262E-02 6.4130067E-03 3.1945726E-03 1.5550018E-03 2.0505085E-04
1.5272952E-05 9.4111942E-07 4.8685170E-08 2.1127542E-09 7.6671995E-11
2.3660590E-12
FIN
5.12703404996350
10.4000000000000
FIN

7.2.5 Importation of initial conditions file

File of the initial water-level(*.opt)
[variables]

"Cote du fond";"ZREF";"m";4
"Cote de l eau";"Z";"m";3
"Débit mineur";"QMIN";"m3/s";3
"Débit majeur";"QMAJ";"m3/s";3
"Coefficient de frottement mineur";"KMIN";"m1/3/s";0
"Coefficient de frottement majeur";"KMAJ";"m1/3/s";0
"Nombre de Froude";"FR";"";5
"Vitesse mineure";"VMIN";"m/s";4
"Hauteur d'eau maximale";"Y";"m";3
[resultats]
0.0;" 1";" 1"; 0.00; 5.0000; 0.000; 0.000; 0.000; 20.; 10.; 0.00000; 0.0000; 0.001
0.0;" 1";" 2"; 200.00; 4.8000; 0.000; 0.000; 0.000; 20.; 10.; 0.00000; 0.0000; 0.001
0.0;" 1";" 3"; 400.00; 4.6000; 0.000; 0.000; 0.000; 20.; 10.; 0.00000; 0.0000; 0.001
0.0;" 1";" 4"; 600.00; 4.4000; 0.000; 0.000; 0.000; 20.; 10.; 0.00000; 0.0000; 0.001
0.0;" 1";" 5"; 800.00; 4.2000; 0.000; 0.000; 0.000; 20.; 10.; 0.00000; 0.0000; 0.001
0.0;" 1";" 6"; 1000.00; 4.0000; 0.000; 0.000; 0.000; 20.; 10.; 0.00000; 0.0000; 0.001
10.0;" 1";" 1"; 0.00; 5.0000; 6.839; 30.000; 0.000; 20.; 10.; 0.19207; 0.8158; 1.839
10.0;" 1";" 2"; 200.00; 4.8000; 6.658; 30.000; 0.000; 20.; 10.; 0.18919; 0.8076; 1.858
10.0;" 1";" 3"; 400.00; 4.6000; 6.483; 30.000; 0.000; 20.; 10.; 0.18536; 0.7967; 1.883
10.0;" 1";" 4"; 600.00; 4.4000; 6.318; 30.000; 0.000; 20.; 10.; 0.18042; 0.7825; 1.918
10.0;" 1";" 5"; 800.00; 4.2000; 6.163; 30.000; 0.000; 20.; 10.; 0.17426; 0.7646; 1.963
10.0;" 1";" 6"; 1000.00; 4.0000; 6.020; 30.000; 0.000; 20.; 10.; 0.16688; 0.7428; 2.020

PK des sections de calcul. Numéro du bief Numéro de la section de calcul Numéro du bief. Pas de temps de calcul.

File of the initial water-level (*.lig)

RESULTATS CALCUL,DATE : 06/09/05 14:21

FICHIER RESULTAT MASCARET

IMAX = 20 NBBIEF= 1
I1,I2 = 1 20
X 0.00 60.17 1999.95 2648.58 4147.71
5728.58 5998.86 6256.28 7823.41 7888.82
7924.72 10067.27 10140.55 10234.29 11351.74
12194.60 12906.70 12937.12 12950.30 13038.97
Z
40.114 40.116 39.644 39.539 39.165
38.736 38.677 38.649 38.373 38.362
38.358 38.021 38.002 37.999 37.795
Q
350.000 350.000 350.000 350.000 350.000
350.000 350.000 350.000 350.000 350.000
350.000 350.000 350.000 350.000 350.000
350.000 350.000 350.000 350.000 350.000
FIN

File of the initial water-level(*.txt).

RESULTATS CALCUL,DATE : 06/09/05 14:21
FICHIER RESULTAT MASCARET

IMAX = 20 NBBIEF= 1
I1,I2 = 1 20
X 0.00 60.17 1999.95 2648.58 4147.71
5728.58 5998.86 6256.28 7823.41 7888.82
7924.72 10067.27 10140.55 10234.29 11351.74
12194.60 12906.70 12937.12 12950.30 13038.97
ZREF
33.7500 33.2000 33.7000 33.8500 33.7000
32.7000 33.1000 32.4000 32.8500 31.7000
30.2500 31.7500 32.3000 31.8500 32.3000

30.9000 32.3500 32.2000 32.0500 31.5500
Z
40.114 40.116 39.644 39.539 39.165
38.736 38.677 38.649 38.373 38.362
38.358 38.021 38.002 37.999 37.795
37.633 37.464 37.405 37.450 37.500
QMIN
334.235 329.216 346.594 337.527 347.537
349.564 350.000 349.236 309.464 350.000
349.653 347.193 346.967 346.978 350.000
349.825 349.624 350.000 349.886 349.971
QMAJ
15.765 20.784 3.406 12.473 2.463
0.436 0.000 0.764 40.536 0.000
0.347 2.807 3.033 3.022 0.000
0.175 0.376 0.000 0.114 0.029
KMIN
36. 36. 36. 36. 36.
36. 36. 36. 36. 36.
36. 36. 36. 36. 36.
36. 36. 36. 36. 36.
KMAJ
13. 13. 13. 13. 13.
13. 13. 13. 13. 13.
13. 13. 13. 13. 13.
13. 13. 13. 13. 13.
FR
0.26016 0.20304 0.40535 0.27730 0.26549
0.25162 0.21164 0.20149 0.25162 0.15485
0.17942 0.24732 0.26477 0.24390 0.19749
0.20361 0.25601 0.27064 0.22617 0.14289
VMIN
1.2777 1.1285 1.4628 1.2087 1.1954
1.3044 1.3221 1.1775 1.1925 1.0891
1.0924 1.3257 1.3573 1.2357 1.2706
1.3020 1.4378 1.7469 1.4346 0.8750
Y
6.364 6.916 5.944 5.689 5.465
6.036 5.577 6.249 5.523 6.662
8.108 6.271 5.702 6.149 5.495
6.733 5.114 5.205 5.400 5.950

FIN

File of the initial water-level (*.rub)
RESULTATS CALCUL,DATE : 06/09/05 14:21

FICHIER RESULTAT MASCARET

IMAX = 20 NBBIEF= 1
I1,I2 = 1 20
X
0.00 60.17 1999.95 2648.58 4147.71
5728.58 5998.86 6256.28 7823.41 7888.82
7924.72 10067.27 10140.55 10234.29 11351.74
12194.60 12906.70 12937.12 12950.30 13038.97
ZREF
33.7500 33.2000 33.7000 33.8500 33.7000
32.7000 33.1000 32.4000 32.8500 31.7000
30.2500 31.7500 32.3000 31.8500 32.3000
30.9000 32.3500 32.2000 32.0500 31.5500
Z
40.114 40.116 39.644 39.539 39.165
38.736 38.677 38.649 38.373 38.362
38.358 38.021 38.002 37.999 37.795
37.633 37.464 37.405 37.450 37.500
QMIN
334.235 329.216 346.594 337.527 347.537
349.564 350.000 349.236 309.464 350.000
349.653 347.193 346.967 346.978 350.000
349.825 349.624 350.000 349.886 349.971
QMAJ
15.765 20.784 3.406 12.473 2.463
0.436 0.000 0.764 40.536 0.000
0.347 2.807 3.033 3.022 0.000
0.175 0.376 0.000 0.114 0.029
KMIN
36. 36. 36. 36. 36.
36. 36. 36. 36. 36.
36. 36. 36. 36. 36.
36. 36. 36. 36. 36.
KMAJ

13. 13. 13. 13. 13.
13. 13. 13. 13. 13.
13. 13. 13. 13. 13.
13. 13. 13. 13. 13.
FR
0.26016 0.20304 0.40535 0.27730 0.26549
0.25162 0.21164 0.20149 0.25162 0.15485
0.17942 0.24732 0.26477 0.24390 0.19749
0.20361 0.25601 0.27064 0.22617 0.14289
VMIN
1.2777 1.1285 1.4628 1.2087 1.1954
1.3044 1.3221 1.1775 1.1925 1.0891
1.0924 1.3257 1.3573 1.2357 1.2706
1.3020 1.4378 1.7469 1.4346 0.8750
Y
6.364 6.916 5.944 5.689 5.465
6.036 5.577 6.249 5.523 6.662
8.108 6.271 5.702 6.149 5.495
6.733 5.114 5.205 5.400 5.950
FIN

7.2.6 Importation of the flood marks file

1 1000.0 40.09 L1
1 7000.0 38.64 L2
1 12000.0 37.70 L3