

# CANALFLOWMODEL

## Reference manual

(CLOUD computing 1D hydraulic flow simulation)

Author: Pilar García-Navarro  
Mario Morales-Hernández

# Contents

1. Introduction	2
2. Step I: Preprocess	3
2.1. Geometry	3
2.2. Initial Conditions	4
2.3. External Boundary Conditions	5
2.4. Internal Boundary Conditions	5
2.4.1. Lateral Gates	5
2.4.2. Cross Gates	6
2.5. Numerical Parameters	7
3. Step II: Simulation	7
4. Step III: Postprocess	8

## 1. Introduction

Canalflowmodel is a software designed at the University of Zaragoza by the Computational Hydraulics Group (<http://www.ghc.unizar.es>) to numerically solve unsteady flows in open channels, based on cloud computing without the need for local installation. The application is executed by going to the address <http://canalflowmodel.net> that opens the screen of Figure 1. It offers the creation of new cases or the possibility to open an existing case through the ticket system.

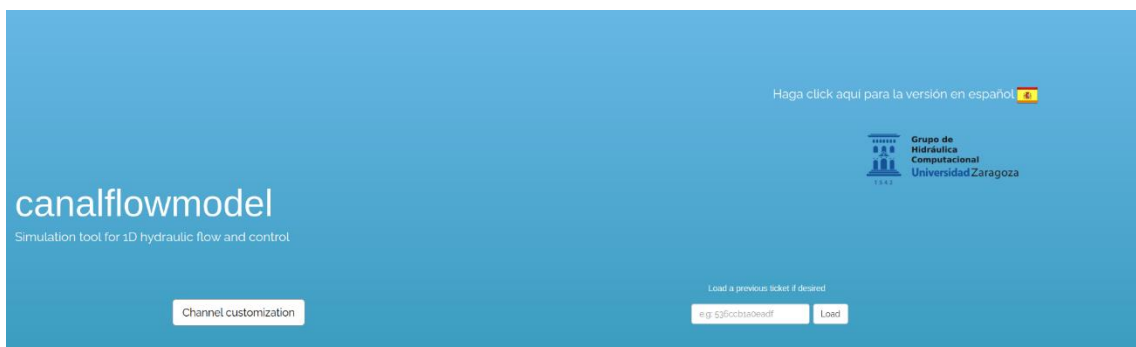


Figure 1: Access screen to the application

We describe below the sequence to follow to create a new case. If you start from a ticket with a preloaded case, you have to follow these same steps but with less effort. When starting the execution of the canalflowmodel program (cannel customization button), a window like the one illustrated in Figure 2 appears on the top bar of the screen.

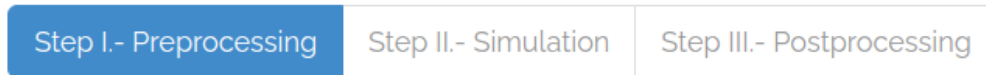


Figure 2: Steps to perform a simulation

## 2. Step I: Preprocess

The preprocess defines the geometry of the case, the initial conditions of the simulation, the boundary conditions that define the type of input and output in the region to simulate and the rest of the parameters required by the numerical method.

### 2.1. Geometry

**NOTE:** It is important to define at the beginning the number of cross sections in the channel and the number of points that define each cross section. This is done by pressing the *Properties* button.

This section contains the elements that will configure the geometric part of the case. The geometry consists of two parts: the first is the definition of the section to be simulated and it will be given mainly by its length and the number of involved sections. This is done in the panel shown in Figure 3. On the other hand, these sections will complete the geometry. This operation is done in the panel shown in Figure 4.

In addition to the geometry, it is necessary to specify values associated with the friction forces generated by the material from which the channel is built or the roughness of the bottom of the channel. In this model, this roughness is provided by the Manning's friction coefficient.

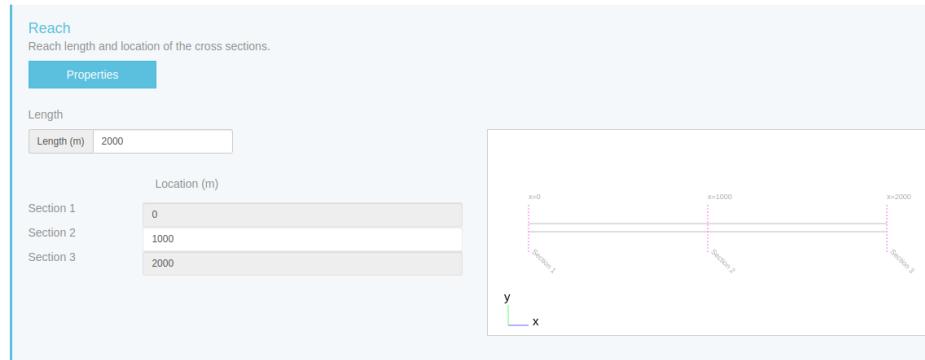


Figure 3: Definition of the location of the cross sections.

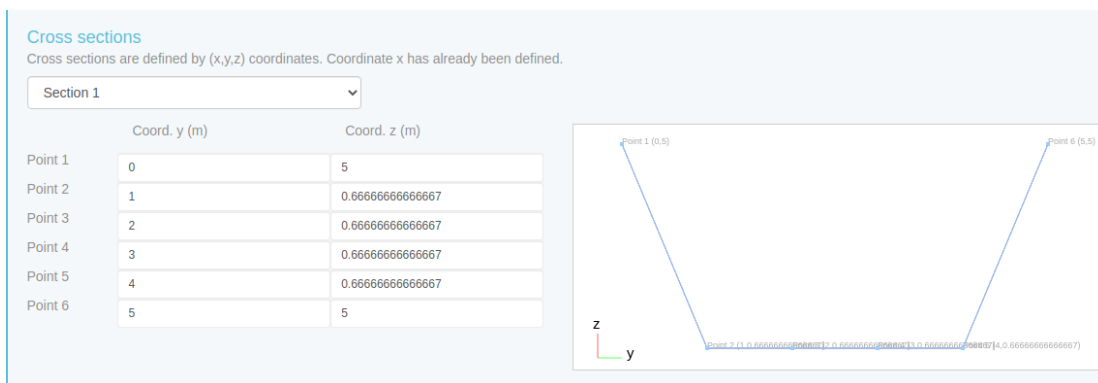


Figure 4: Coordinates of the points defining a cross section

## 2.2. Initial Conditions

This section generates the conditions at the initial time of the simulation. The available options are as follows:

- Dry channel: zero flows and depths at the beginning
- $h + z = \text{const}$ : A constant water surface condition with zero velocity is generated.
- $Q = \text{const}$ : The corresponding state is generated with a steady flow.
- $h = \text{const}$ : All cells will have the same depth and zero velocity.
- Dam break (dambreak): Discontinuity of water surface level with zero velocity.

## 2.3. External Boundary Conditions

The external boundary conditions are defined at the inlet and outlet of the one-dimensional domain. The inlet boundary condition specifies the behavior at the upstream section, being the direction of the incoming flow in the domain while the outlet boundary condition details the downstream section.

The options available at the entrance are the following:

- $h + z = h+z(t)$ : Surface level as a function of time.
- $Q = Q(t)$ : Discharge as a function of time
- $h = h(t)$ : Depth as a function of time.

The options available at the outlet are the following:

- Free flow: No conditions are imposed
- $h + z = h+z(t)$ : Surface level as a function of time.
- $Q = Q(h + z)$ : Discharge as a function of the surface level
- Constant Froude number
- Weir: defines a weir at the outlet with some user defined parameters
- Uniform flow depending on the slope

## 2.4. Internal Boundary Conditions

Canalflomodel includes the possibility of defining the presence of lateral and cross gates. The latter can work in automated form according to a goal. To begin with, the total number of control elements is entered in a window like the one shown in Figure 5.

### 2.4.1. Lateral Gates

Each lateral gate is characterized by its position, its width, its discharge coefficient (less than or equal to 1) and the time history of openings as shown in Figure 6.

**Gates configuration**  
 Defines regulation elements

Number of gates

Figure 5: Definition of the number of gates

Parameters Gate 0

Position (m)  Width (m)  K   Lateral  Controlled

Number of points

Time (s)	Opening/Target (m)
0	10
720	10
1440	10
2160	10
2880	10

Figure 6: Parameters to define a lateral gate

### 2.4.2. Cross Gates

Each cross gate is characterized by its position, its discharge coefficient (less than or equal to 1) and the time history of openings or target values of the upstream draft, as shown in Figure 6. To enable target-based regulation, check the *Automatic* box and configure the PID controller as shown in Figure 7.

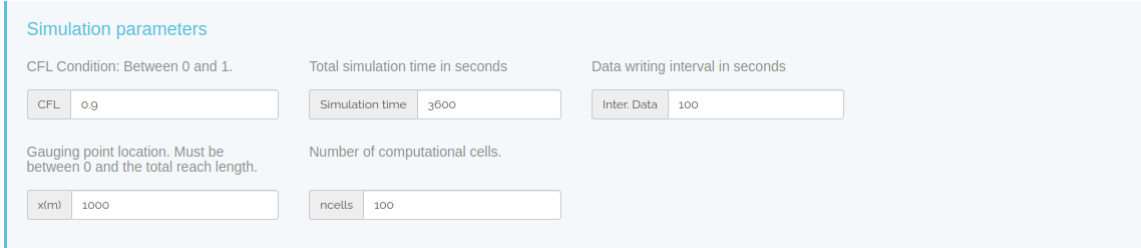
**Regulation**  
 Configuration of the parameters used to regulate the gates using PID control.

Integration period <input type="text" value="30"/> TI	Derivation period <input type="text" value="10"/> TD	Sampling period <input type="text" value="20"/> TS
Maximum gate displacement speed (I.S.) <input type="text" value="0.01"/> v Max	Maximum gate displacement (I.S.) <input type="text" value="2"/> z Max	Dead band range for gate actuation (I.S.) <input type="text" value="0.01"/> B.M.
Present time coefficient. NOTE: value between 0 and 1 <input type="text" value="1"/> k (t)	Previous time coefficient (t-1). NOTE: value between 0 and 1 <input type="text" value="0.8"/> k (t-1)	(t-2) time coefficient . NOTE: value between 0 and 1 <input type="text" value="0.6"/> k (t-2)

Figure 7: PID regulation parameters for the cross gates.

## 2.5. Numerical Parameters

The CFL number defines the time step size  $\Delta t$ . By default, the program uses the value 0.9 which optimizes the calculation speed without loss of precision. In addition, you have to choose the start and end time of the simulation and the time frequency of the data dump. This is shown in Figure 8.



The screenshot shows a 'Simulation parameters' form with the following fields and values:

Parameter	Value
CFL	0.9
Simulation time	3600
Inter. Data	100
x(m)	1000
ncells	100

Figure 8: Numerical parameters required by the simulation

## 3. Step II: Simulation

To start Step II, press the green *Simulate* button. The application will show a summary panel of the simulation configuration and will assign a code (ticket) to this case. Afterwards, a screen like the one in Figure 9 will appear where the application represents discharge in the inlet and outlet sections over the simulation time as well as the time step that is calculated from dynamic form and the conservation error of mass.

Then you access Step III.

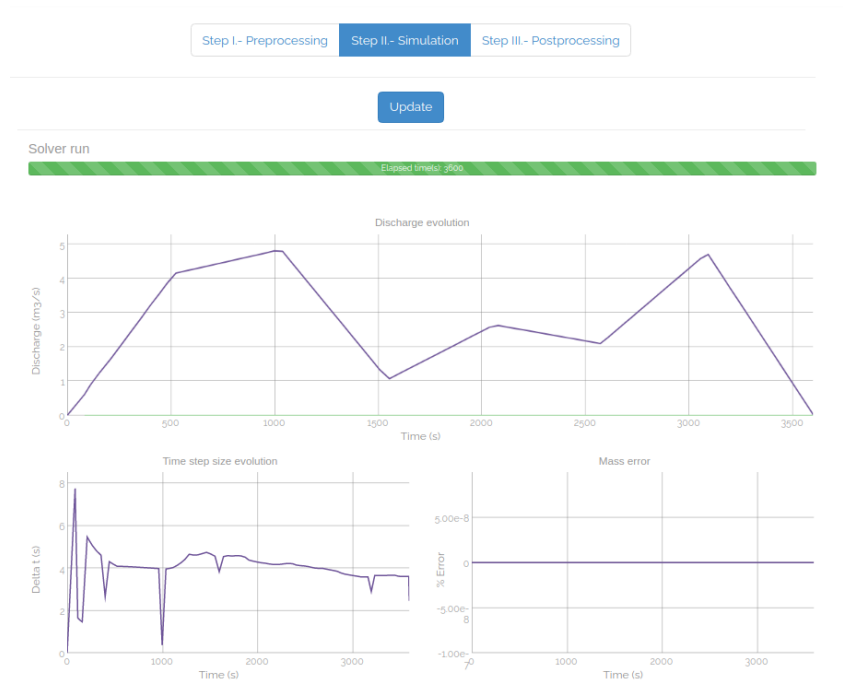


Figure 9: Time evolution during the calculation

## 4. Step III: Postprocess

The results show:

- Longitudinal surface level profiles along the canal at different instants of time.
- Longitudinal flow profiles along the canal at different times.
- Temporal evolution of opening and flow at the lateral gates.
- Temporal evolution of opening, calculated depth and target depth (if automatic) in the cross gates

An example is shown in Figure 10. Finally, it is important to note that the results can be downloaded to external files using the button "*Download results*" located at the top left.





Figure 10: Screenshot of simulation results