

TD 1

Numerical resolution of the Saint-Venant system

N. Aissiouene, E. Godlewski, A. Mangeney & J. Sainte-Marie

November 16, 2017

Objectives

The objectives of these short exercises is to guide the user for the use of the Freshkiss1d code. This numerical code solve a classical model in fluid mechanics, namely the Saint-Venant system. The details of the numerical scheme and its properties are not given or proved in this document, we focus on some well-known and typical test cases. Starting from these test cases, the user can modify the input file in order to simulate the desired situation.

With the proposed exercices, you will perform *numerical experiments* i.e. you will have the opportunity to evaluate the answer of a model in the situation you impose. This will allow you to understand the behavior of a rather complex model and to test the influence of several quantities

- the initial and boundary conditions,
- the bottom topography,
- the bottom friction laws, the fluid rheology
- the quality of the numerical approximation.

The Saint-Venant system

We consider a 1d chanel of length L with a bottom topography given by the function $z_b(x)$. The water depth is $H(x,t)$, the free surface is given by $\eta = H + z_b$ and the velocity (mean velocity over the water depth) is given by

$$\bar{u}(x,t) = \frac{1}{H} \int_{z_b}^{\eta} u(x,z,t) dz,$$

where u is the component of the fluid velocity along the horizontal axis Ox , see Fig. 1.

We consider the Saint-Venant system with a topography source term, a Navier friction coefficient and a Newtonian viscosity. The model writes

$$\frac{\partial H}{\partial t} + \frac{\partial(H\bar{u})}{\partial x} = 0, \tag{1}$$

$$\frac{\partial(H\bar{u})}{\partial t} + \frac{\partial(H\bar{u}^2)}{\partial x} + \frac{g}{2} \frac{\partial H^2}{\partial x} = -gH \frac{\partial z_b}{\partial x} + \frac{\partial}{\partial x} \left(4\mu H \frac{\partial \bar{u}}{\partial x} \right) - \kappa \bar{u}. \tag{2}$$

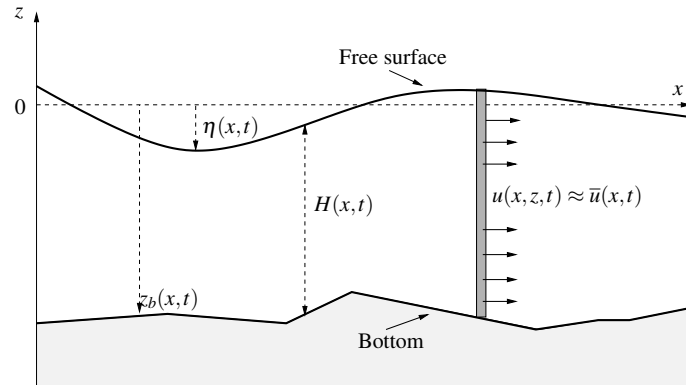


Figure 1: Notations.

Preliminaries

How to install the code The subroutines are written in `fortran`, you should have a `fortran` compiler available on your computer e.g. `gfortran`. For the visualization of the results, two free softwares are required `octave` and `paraview`. You can download them

- <https://www.gnu.org/software/octave/>
- <http://www.paraview.org>

In order to install the code, you have to

1. open a Terminal window and go (using the `cd` command) to the directory `Freshkiss1d` and then type `mkdir ../pipe`.
2. If your `fortran` compiler is not `gfortran`, you should modify the files `makefile` and `bin/makefile` by changing the line `F770=gfortran` by `F770='name_of_the_fortran_compiler'`.
3. In the window Terminal and in the directory `Freshkiss1d`, type `make`.
4. The compilation is in progress. If it ends with `Compilation successful`, you are ready to use the code.

First use In order to launch a simulation with the code, you should proceed as follows :

1. In the Terminal window indicating the directory `Freshkiss1d`, type `sh launch`.
2. Then the simulation of the test case defined by the input file `data.freshkiss1d` is carried out. For the visualization, you have to launch the software `paraview` and open (with `paraview`) the files sequence `resu2d_*.vtk`.
3. With `paraview`, you can visualize the time evolution of the solution (water depth H and velocity \bar{u}) for the considered geometry.

You can also draw the variations – at a given point over time or at a given instant for the whole domain – associated with the quantities H and \bar{u} . For that purpose, you have to :

1. in the directory `Freshkiss1d`, type `octave`.

2. A new window opens. In this new window, using the command `cd` go to the directory `Code_nh` and type `traceIPGP`.
3. Three figures will appear, the first one shows the variations of the quantities

$$x \mapsto H(x, t_1) + z_b(x),$$

and

$$x \mapsto z_b(x),$$

i.e. the bottom topography and the free surface at the instant $t_1 = 0.6$ second of the simulation. The second figure depicts the variations of

$$x \mapsto \bar{u}(x, t_1),$$

and the third figure gives the variations of

$$x \mapsto (H\bar{u})(x, t_1).$$

Another test case If you intend to modify the test case defined by the input file `data.freshkiss1d`, you have two solutions :

1. if the modifications you want only concern the parameter file `data.freshkiss1d`, it is enough to perform them, to save the file `data.freshkiss1d` and to type `sh launch`.
2. If the modifications also concern one or several `fortran` subroutines you should type `make` and then `sh launch`.

Important parameters and files The meaning of the variables appearing in the file `data.freshkiss1d` and the role of the `fortran` subroutines are explained in the documentation available in the directory `doc/`.

Other exercises

1 Some particular situations

A dam break We intend to simulate a sort of “dam break” problem, see Fig 2. This means we consider the following situation :

- at the boundaries $x = 0$ and $x = L$, a solid wall (or equivalently an inflow equals to zero) is imposed,
- we start from the following initial condition

$$u(x, t_0) = 0, \quad \forall x \in [0, L],$$

et

$$H(x, t_0) = \begin{cases} \eta_0 - z_b(x) & \text{si } x \leq x_1 \\ 0 & \text{si } x > x_1 \end{cases}$$

with $x_1 \in [0, L]$ and $\eta_0 > \max_{x \in [0, x_1]} z_b(x)$.

1. When $z_b(x) = cst = 0$, try to simulate the dam break. In fact, this is the default test case of the code. Therefore, you only have to type `sh launch` in the window `Terminal`.

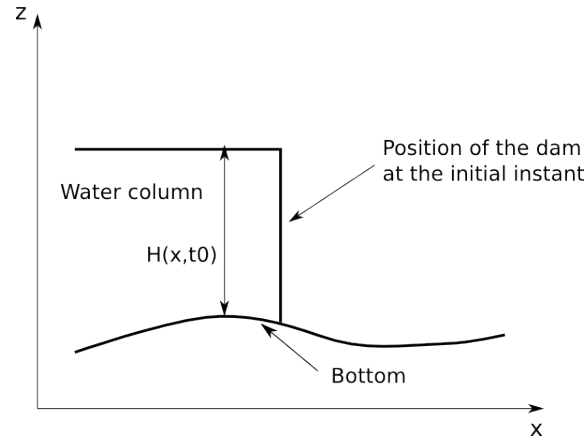


Figure 2: Dam break : situation at the initial instant.

2. Use paraview to visualize the results.
3. For this test case when $\kappa = \mu = 0$, an analytical solution is available, launch `octave` and type `traceIPGP` in the window Terminal. Three figures appear on your screen allowing to compare the analytical solution (water depth, velocity and flow) at the instant $t_1 = 0.6$ second.
4. Modify the space step by choosing different values for `ns` in the file `data.freshkiss1d` and compare the analytical solutions to the simulated ones. What is the influence of `ns` ?
5. Why the computation of the simulated solution is time consuming when `ns` is large ?

The analytical solution is given by:

$$H(x,t) = \begin{cases} h_1 & \text{if } x \leq x_A(t) \\ \frac{4}{9g} \left(\sqrt{gh_1} - \frac{x-x_1}{2t} \right)^2 & \text{if } x_A(t) \leq x \leq x_B(t) \\ 0 & \text{if } x \geq x_B(t) \end{cases}$$

and

$$\bar{u}(x,t) = \begin{cases} 0 & \text{if } x \leq x_A(t) \\ \frac{2}{3} \left(\frac{x-x_1}{t} + \sqrt{gh_1} \right) & \text{if } x_A(t) \leq x \leq x_B(t) \\ 0 & \text{if } x \geq x_B(t) \end{cases}$$

with $h_1 = H(0 \leq x < x_1, t_0)$ and

$$\begin{aligned} x_A(t) &= x_1 - t\sqrt{gh_1} \\ x_B(t) &= x_1 + 2t\sqrt{gh_1} \end{aligned}$$

6. Try to simulation a dam break with a non flat topography. This means to set to e.g. 1 the variable `ishape` in the file `data.freshkiss1d`.
7. What should be the effects of bottom friction over a dam break ?
8. Simulate a dam brak with a Navier friction coefficient (e.g. $\kappa = 1 \text{ s}^{-1}$) and visualize the obtained results.

Equilibrium at rest

Now, we are going to check if the numerical scheme preserves the lake at rest equilibrium.

1. How is defined/characterized the lake at rest equilibrium ?
2. Starting from a non trivial topography, check that the code preserves the static equilibrium. For that purpose, you will have to modify
 - o the initial conditions : set the variable `isurf_init` equals to 1 and have a look to the file `init.f`,
 - o the boundary conditions : set the variables `isort` and `ient` equals to 2 and have a look to the file `utils.f` to check if the imposed inflow at the boundaries are correct,
 - o trace : in order to trace the water depth and velocity profiles at the final instant with octave, you have to set the variable `choix` in the file `traceIPGP.m` to `choix = options{2}`.

Stationary solution

1. With a slight modification of the boundary conditions used in the previous questions, try to simulate a stationary smooth solution (not at rest).
2. With a slight modification of the boundary conditions used in the previous questions, try to simulate a stationary **non smooth** solution.

Analytical solution

We consider the Saint-Venant system (1)-(2).

1. Try to build an analytical continuous stationary solution for the system (1)-(2). For that purpose, you should define/determine:
 - o a bottom topography,
 - o boundary conditions (often a given inflow at the left entry and an imposed water depth at the right exit),
 - o the (space) variations of the water depth and the debit.
2. Verify that the numerical code well approximate the analytical solution you have found.
3. Verify that when the space step decreases, the simulated solution *converges* towards the exact one.
4. Try to build a discontinuous stationary analytical solution for the system (1)-(2). Verify that when the space step decreases, the simulated solution *converges* towards the exact one. (*Difficult question, to do after all the others*)

2 Tsunami (en 1d)

On va maintenant chercher à générer un tsunami. Pour cela, on considère que la topographie dépend du temps i.e. $z_b = z_b(x, t)$ et partant d'une topographie à l'instant initial $z_b(x, t_0)$ on va la modifier. La modification de la topographie est faite dans le fichier `zbt.f`. Pour que les variations de la topographie au cours du temps soient prises en compte, il faut que la variable `iseism` soit égale à 1 dans le fichier `data.freshkiss1d`.

1. Chercher à simuler un tsunami.
2. Tester plusieurs formes pour le déplacement du fond. Un choix simple est

$$z_b(x, t) = z_b(x, t_0) + d e^{-\left(\frac{x-x_{max}}{L}\right)^2} \left(1 - e^{-\frac{t-t_0}{\tau}}\right), \quad (3)$$

(voir dans le fichier `zbt.f`) où d , L et τ sont trois paramètres et x_{max} désigne la longueur du canal considéré.

3. Faire varier le déplacement du fond pour voir l'impact
 - o d'un mouvement lent vs. un mouvement rapide,
 - o d'un déplacement global i.e.

$$\int_0^T \int_0^{x_{max}} z_b(x, t) dx dt > 0,$$

vs. un déplacement *compensé* i.e.

$$\int_0^T \int_0^{x_{max}} z_b(x, t) dx dt = 0.$$

4. On se met maintenant dans la situation suivante. Un séisme a eu lieu quelque part dans un océan (ici un canal) et ce séisme a généré un tsunami. Grâce à 3 bouées, on a enregistré la forme de la vague du tsunami lors de son passage au niveau de chacune des bouées. Retrouver l'excitation $z_b(x, t)$ qui a généré ce tsunami ?

On suppose que l'excitation est de la forme

$$z_b(x, t) = z_b(x, t_0) + f_1(x) f_2(t),$$

et qu'elle ressemble donc à celle décrite par l'équation (3). Reste à identifier les paramètres d , L et τ . Attention, il y a une petite chose à ajouter à la formule (3).

La topographie de l'océan est obtenue en prenant `ishape` égal à 7. Choisir `ns` égal à 5119 et `xmax` égal à 1 000 000 mètres (soit 1 000 km). Cette topographie 1d est en fait une coupe de l'océan en large du Japon, cette coupe relie l'épicentre du séisme de mars 2011 avec la bouée DART 21418.

Vous pouvez choisir pour chaque bord, des conditions aux limites de type hauteur d'eau imposée i.e. `ient` et `isort` égaux à 3 et `h0e=1.0d0` ainsi que `h0s=1.0d0` dans le fichier `utils.f`. Dans le fichier `data.freshkiss1d`, la ligne définissant la variable `isurf_init` doit débiter par `1, 1.0, 0.0 # 'isurf_init'`. Cela revient à dire qu'à l'instant initial l'océan est au repos et que la cote de la surface libre est 1 mètre.

Vous pouvez également choisir `dt_save` égal à 10.0 et `tmax` égal à 500.0.

Pour mieux visualiser le domaine fluide (qui est très étendu 1 000 km pour 8 000 mètres de fond), vous pouvez "dilater" les figures `paraview` sur la verticale en mettant `scale_h` égal à 5.

L'abscisse des 3 bouées est :

$$x_1 = 390.7 \text{ km}, \quad x_2 = 586.05 \text{ km}, \quad x_3 = 781.4 \text{ km}.$$

Pour comparer les résultats de votre simulation aux mesures expérimentales enregistrées au niveau des 3 bouées, faire `choix = options{4}` dans `traceIPGP.m` et exécuter `traceIPGP` dans `octave`. Trois figures apparaissent représentant les variations de la surface libre mesurées (bouée DART) et simulées aux points x_1 , x_2 et x_3 .

3 Rhéologie complexe

On va maintenant chercher à simuler un écoulement de type shallow water avec une rhéologie (un peu) complexe. Pour cela on considère les équations de Saint-Venant pour un fluide ayant une rhéologie de type Coulomb. Les équations du modèle sont alors

$$\frac{\partial H}{\partial t} + \frac{\partial}{\partial x}(H\bar{u}) = 0, \quad (4)$$

$$\frac{\partial(H\bar{u})}{\partial t} + \frac{\partial(H\bar{u}^2)}{\partial x} + \frac{g}{2} \frac{\partial H^2}{\partial x} = -gH \frac{\partial z_b}{\partial x} + Hf, \quad (5)$$

la force de frottement $f = f(x,t)$ satisfaisant

$$\begin{cases} |f(x,t)| \leq g\mu, \\ \bar{u}(x,t) \neq 0 \Rightarrow f(x,t) = -g\mu \frac{\bar{u}(x,t)}{|\bar{u}(x,t)|}. \end{cases} \quad (6)$$

1. En définissant `ifrot` égal à 7 et en donnant une valeur numérique à `cfrot`, typiquement 0.5, simuler une rupture de barrage avec un fluide obéissant à une rhéologie de type Coulomb. Qu'observez-vous ?
2. A votre avis, où se situent les difficultés quant à l'approximation numérique de la loi de frottement (6).

Avant d'aborder les questions suivantes, supprimer le frottement dans le fluide i.e. mettre `ifrot` égal à 0.

4 Effets non-hydrostatiques

On compare maintenant le comportement du modèle de Saint-Venant avec son extension non-hydrostatique, voir le poly du cours pour en savoir plus sur ce modèle non-hydrostatique.

On se met dans les hypothèse suivantes :

- un canal de 30 mètres de long, `ns` valant 1000,
- `tmax` égal à 15 et `dt_save` égal à 0.05,
- la topographie correspond à `ishape` égal à 4,
- hauteur d'eau imposée i.e. `ient` et `isort` égales à 3,
- dans le fichier `utils.f`, on définit

$$h0e = 1.0d0 + 0.02d0 * d\sin(2.0d0 * pi/Ta * x),$$

il suffit de décommenter les lignes correspondantes.

Lorsque `inh` vaut 0 dans `data.freshkiss1d`, on résout les équations de Sait-Venant et lorsque `inh` vaut 1, on résout les équations de Sait-Venant non-hydrostatique.

1. Quelle différence observe-t-on entre les 2 modèles ? Vous pouvez visualiser les résultats avec `paraview` ou `octave` avec `choix = options{3}`.

5 Wind effects

To finish with this short introduction, we consider a channel with a wind blowing at the free surface. When the variable `iwind` is set to 1 and `cwind` has a non-negative value, the wind blows from the left to the right. of the domain.

1. Starting from the lake at rest situation with an inflow equals to zero at each boundary, what is the effect of the wind over the fluid domain ?
2. What happens `cwind` when increases ?
3. What is your conclusion concerning the validity of the Saint-Venant system in such a situation ?