Telemac2d

Validation Manual

Otto Mattic

Version v7p3
March 16, 2018

Contents

1. bumpcri

1.1 Description of the problem

This test case presents a flow over a bump on the bed with super-critical condition. It allows to show that TELEMAC-2D is able to correctly reproduce the hydrodynamic impact of a changing bed slopes, vertical flow contractions and expansions. Furthermore, it allows to have a good representation of flows computed in steady and transient flow regimes.
The solution produced by TELEMAC-2D in a frictionless channel presenting an idealised bump on the bottom is compared with the analytical solution to this problem. Flow conditions are such that the bump generates a transition from sub-critical (fluvial) to super-critical (torrential) flow for this problem.

The geometry dimensions of the channel are 2 m wide and 20.5 m long. It is horizontal with a 4 m long bump in its middle (see Figure 1.1). The maximum elevation of the bump is 0.2 m (see Figure 1.2) with the bottom z_f describes by the following equation :

$$z_f = \begin{cases} -0.05(x-10)^2 \text{ m} & \text{if } 8 \text{ m} < x < 12 \text{ m} \\ -0.20 \text{ m} & \text{elsewhere} \end{cases}$$

Note that the horizontal viscosity turbulent is constant and equal to zero. However instead of prescribing a zero viscosity, no diffusion step could have been used (keyword DIFFUSION OF VELOCITY prescribed to NO).

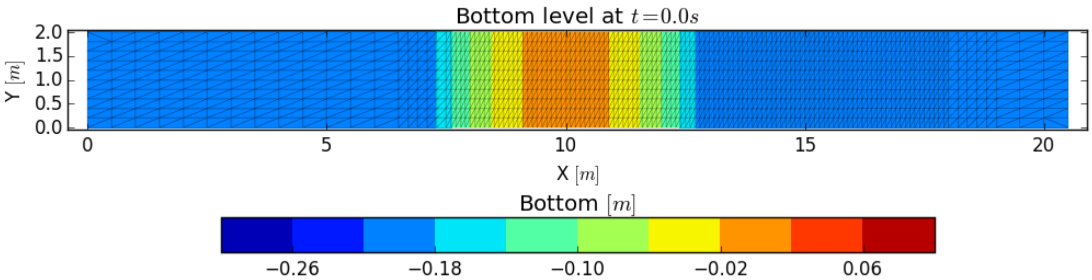


Figure 1.1: Mesh and topography of the channel.

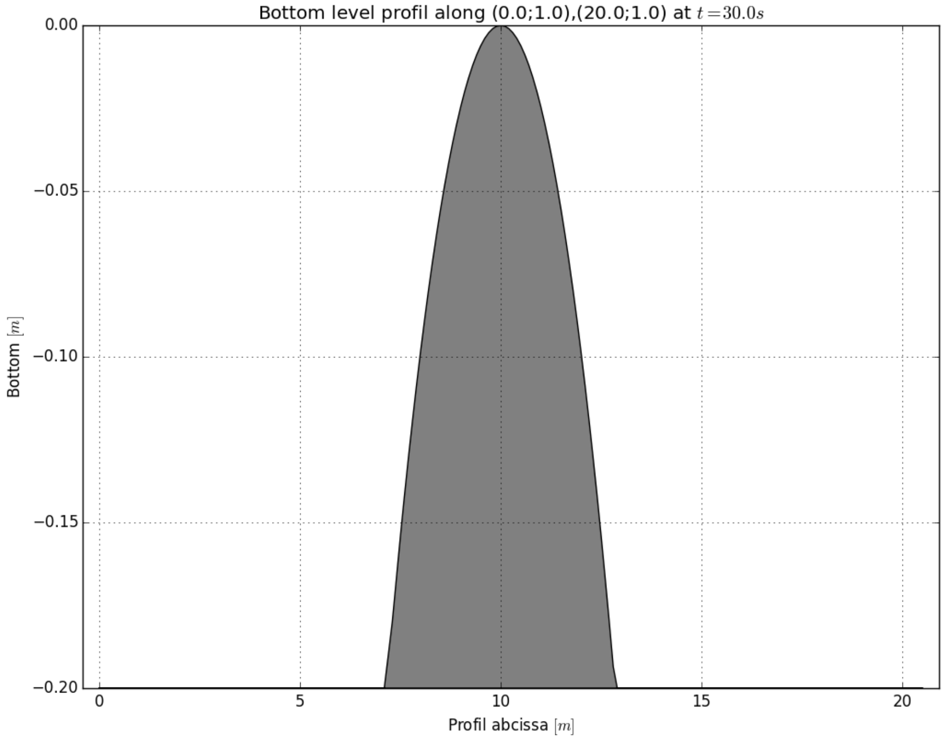


Figure 1.2: Profile of the bump.

1.2 Initial and boundary conditions

The initial conditions are a null velocity and an analytical solution (Equation (1.1)) for the water depth.

The boundary conditions are:

- At the channel entrance, the flow rate is $Q = 0.6\ \mathrm{m}^3\mathrm{s}^{-1}$ (note that $q = \dfrac{Q}{B}$, B being the channel width and the depth is $h = z_s - z_f = 0.5$ m, with the free surface elevation z_s).
- At the channel outlet, the free velocity and free water level are imposed on the liquid boundary.
- No friction is taken into account on the bottom and on the lateral walls.

1.3 Mesh and numerical parameters

The mesh is regular, with a higher resolution in the middle of the channel. It is made up with quadrangles split into two triangles. It is composed of 2,620 triangular elements (1,452 nodes) and the size of triangles ranges between 0.25 m and 0.5 m. The triangular elements types are linear triangles (P1) for velocities and for water depth.

The time step is 0.03 s for a period of 30 s. In fact, TELEMAC-2D is run forward in time until a steady state flow is obtained. The resolution accuracy for the velocity is taken at 10^{-5}.

Note that for numerical resolution, the conjugate gradient on a normal equation is used for solving the propagation step (option 1). Furthermore the treatment of linear system is done with a wave equation. To solve advection, the characteristics scheme is used for the velocities (scheme 1) and the conservative PSI scheme is used for the depth (scheme 5, mandatory). To finish, the implicitation coefficient for depth and velocities is equal at 1.

1.4 Results

From an analytical point of view, the critical depth is :

$$h_c = \left(\frac{q_c^2}{g}\right)^{1/3}$$

This value is obtained above the bump. No friction on the bottom thus allows to write the Bernoulli equation between the bump and any point A of abscissa x_A:

$$z_{fA} + h_A + \frac{q_A^2}{2gh_A^2} = z_{fc} + E_c$$

where the corresponding specific water depth energy E_c is:

$$E_c = h_c + \frac{q_c^2}{2gh_c^2} = \frac{3}{2}h_c$$

Therefore, water depth h_A is given analytically by:

$$h_A^3 + (z_A - z_c - E_c)h_A^2 + \frac{q_A^2}{2g} = 0 \tag{1.1}$$

At the foot of the bump, positive solutions are $h_{upstream} = 0.49535$ m and $h_{downstream} = 0.49535$ m. These solutions are respectively the upstream sub-critical and downstream super-critical water depths. Corresponding free surface elevations are $z_{s\ upstream} = 0.29535$ m and $z_{s\ downstream} = -0.094$ m.

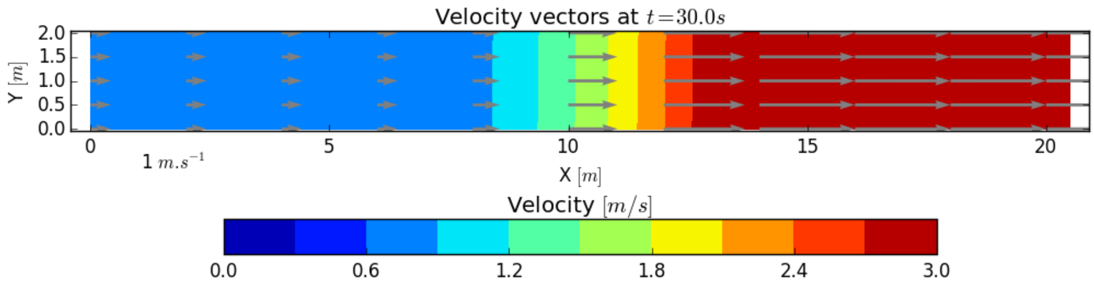


Figure 1.3: Velocity field for the steady state flow.

Numerical results are compared with the analytical solution, when the state flow is steady (Figure 2.3). Furthermore, the computed Froude number is also compared with the analytical solution $F_r = \frac{q}{h\sqrt{gh}}$. The solution produced by TELEMAC-2D is in close agreement with the analytical solution as shown on the Figures 1.4 and 1.5.

To conclude, this transition from a sub-critical to a super-critical flow regime over a bump in a channel without friction is adequately reproduced by TELEMAC-2D, provided the mesh resolution is fine enough. This transition occurs at critical depth as announced by channel flow hydraulics.

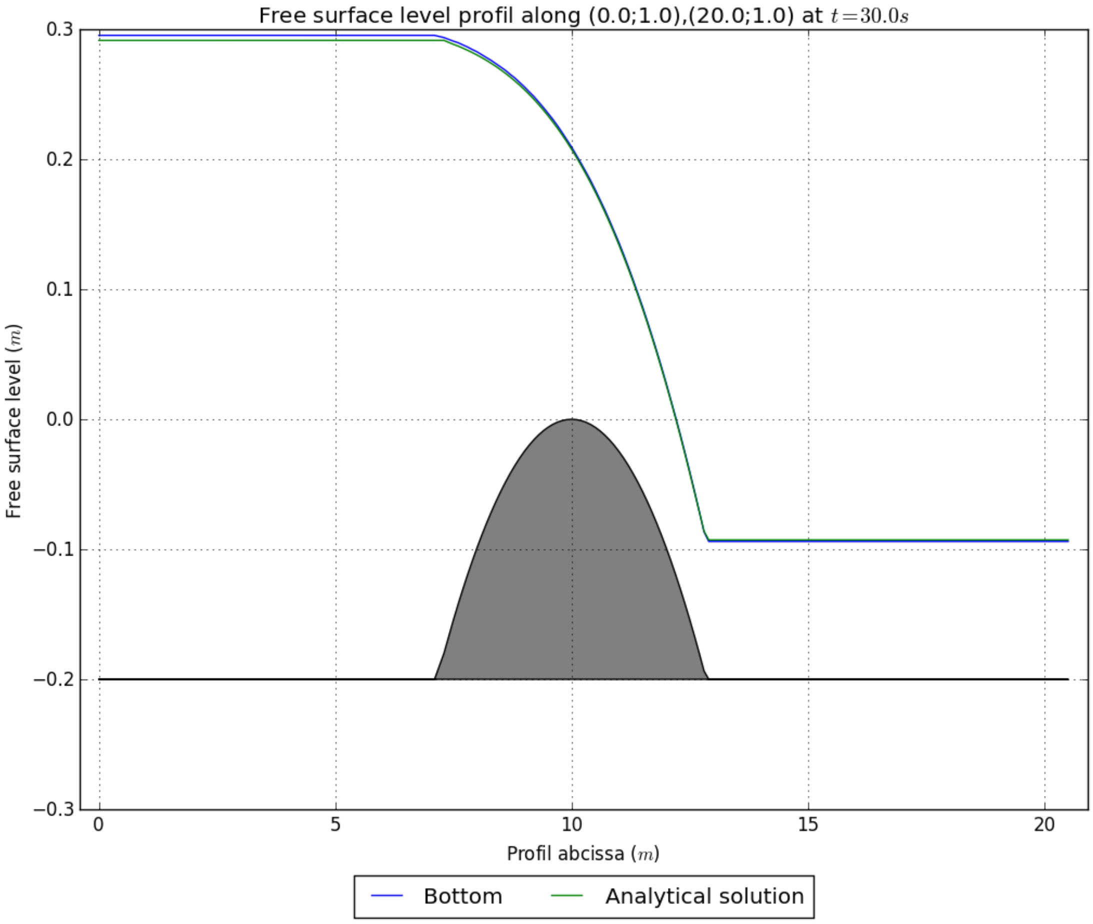


Figure 1.4: Comparison between analytical solution and TELEMAC-2D solution for the free surface elevation.

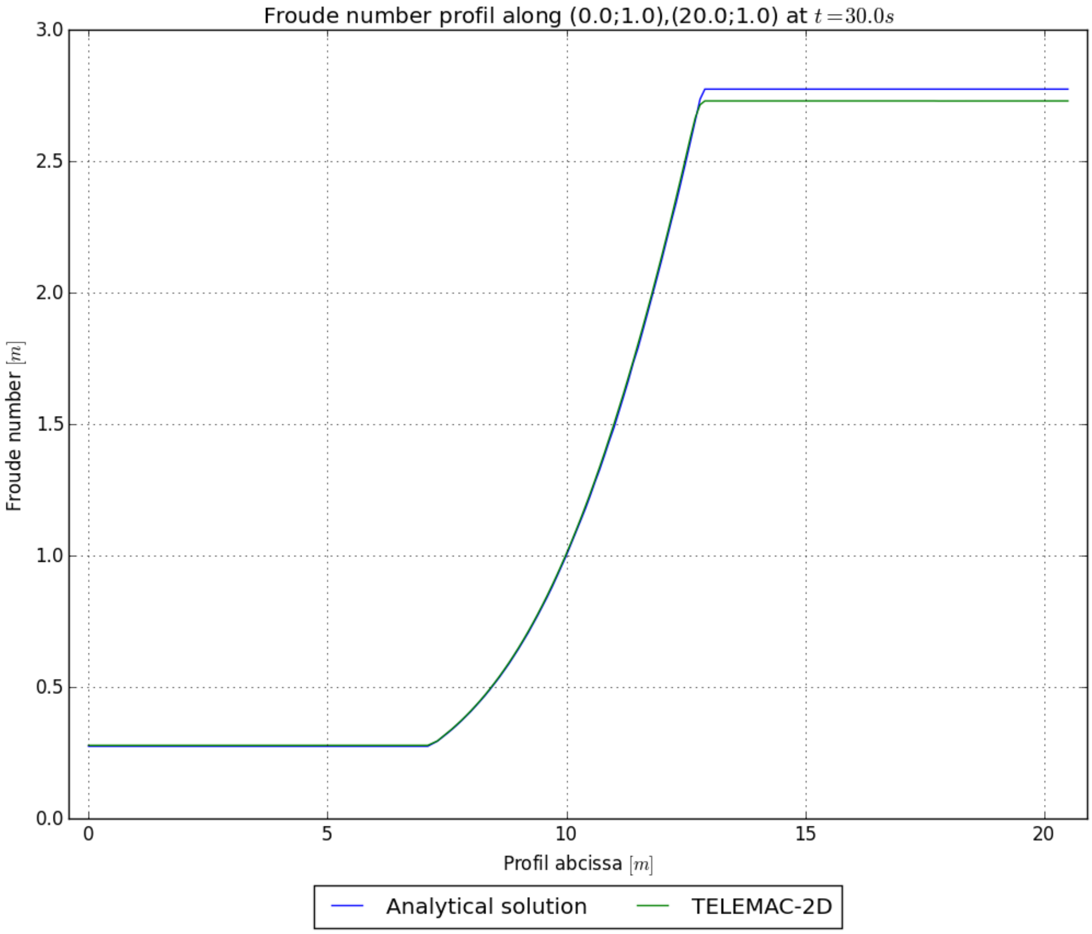


Figure 1.5: Comparison between analytical solution and TELEMAC-2D solution for the Froude number.

2. bumpflu

2.1 Description of the problem

This test case presents a flow over a bump on the bed with subcritical condition. It allows to show that TELEMAC-2D is able to correctly reproduce the hydrodynamic impact of a changing bed slopes, vertical flow contractions and expansions. Furthermore, It allows to have a good representation of flows computed in steady and transient flow regimes.
The solution produced by TELEMAC-2D in a frictionless channel presenting an idealised bump on the bottom is compared with the analytical solution to this problem. For this test case, the studied flow regime is sub-critical.

The geometry dimensions of the channel are 2 m wide and 20.5 m long. It is horizontal with a 4 m long bump in its middle (see Figure 2.1). The maximum elevation of the bump is 0.2 m (see Figure 2.2) with the bottom z_f describes by the following equation :

$$z_f = \begin{cases} -0.05(x-10)^2 & \text{if 8 m} < \text{x} < \text{12 m} \\ -0.20 & \text{elsewhere} \end{cases}$$

Note that the horizontal viscosity turbulent is constant and equal to zero. However instead of prescribing a zero viscosity, no diffusion step could have been used (keyword DIFFUSION OF VELOCITY prescribed to NO).

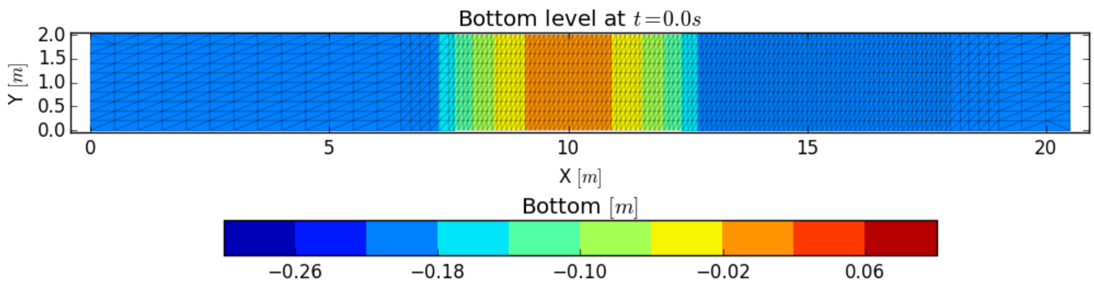


Figure 2.1: Mesh and topography of the channel.

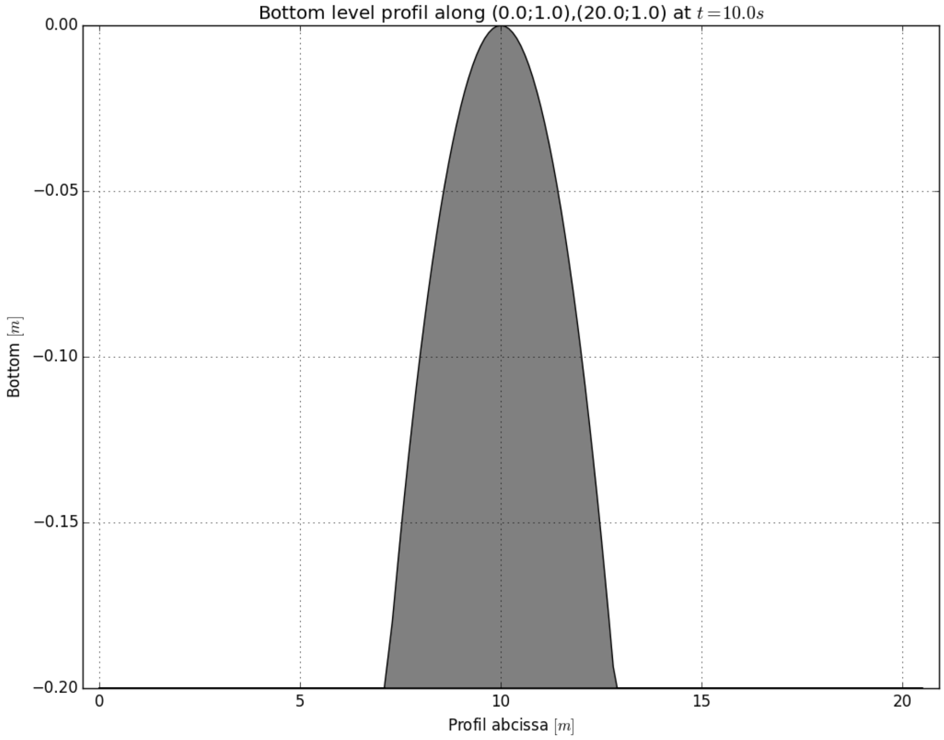


Figure 2.2: Profile of the bump.

2.2 Initial and boundary conditions

The initial conditions are a null velocity and an analytical solution (Equation (2.1)) for the water depth.

The boundary conditions are:

- At the channel entrance, the flow rate is $Q = 8.858893836$ m^{3}s^{-1} (note that $q = \frac{Q}{B}$, B being the channel width). The flow rate is taken so that the q value is equal at $\sqrt{gh}$ in entrance.
- At the channel outlet, the free surface elevation is $z_s = 1.8$ m. Therefore, the depth $h = z_s - z_f = 2$ m.
- No friction is taken into account on the bottom and on the lateral walls.

2.3 Mesh and numerical parameters

The mesh is regular, with a higher resolution in the middle of the channel. It is made up with quadrangles split into two triangles. It is composed of 2,620 triangular elements (1,452 nodes) and the size of triangles ranges between 0.25 m and 0.5 m. The triangular elements types are linear triangles (P1) for velocities and for water depth.

The time step is 0.01 s for a period of 10 s. In fact, TELEMAC-2D is run forward in time until a steady state flow is obtained. The resolution accuracy for the velocity is taken at 10^{-5}.

Note that for numerical resolution, the conjugate gradient on a normal equation is used for solving the propagation step (option 1). Furthermore the treatment of linear system is done with a wave equation. To solve advection, the conservative PSI scheme is used for the depth (scheme 5) and the conservative N scheme is used for the velocities (scheme 4). In addition, the treatment of linear system is done with a wave equation. To finish, the implicitation coefficient for depth and velocities is equal at 0.6.

2.4 Results

From an analytical point of view, no friction on the bottom allows to write the Bernoulli equation between the entrance of the channel E (where $q_E = 4.43$ m^{2}s^{-1} and $h_E = 2$ m) and point A of abscissa x_A:

$$z_{fA} + h_A + \frac{q_A^2}{2gh_A^2} = z_{fE} + h_E + \frac{q_E^2}{2gh_E^2}$$

Replacing by the values, the equation can be written as following:

$$z_{fA} + h_A + \frac{q_A^2}{2gh_A^2} = 2.25$$

Therefore, water depth h_A is given analytically by:

$$h_A^3 + (z_{fA} - 2.05)h_A^2 + \frac{q_A^2}{2g} = 0 \tag{2.1}$$

In this test case, the numerical results are compared with the analytical solution, when the state flow is steady (Figure 2.3). Furthermore, the computed Froude number is also compared with the analytical solution $F_r = \frac{q}{h\sqrt{gh}}$. The solution produced by TELEMAC-2D is in close agreement with the analytical solution as shown on the Figures 2.4 and 2.5.

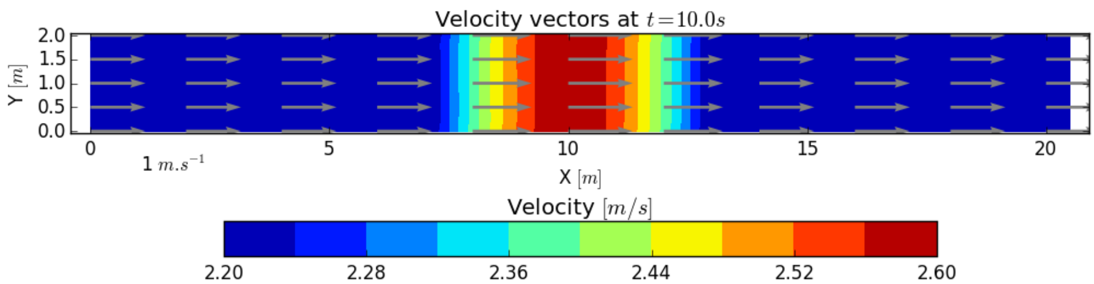


Figure 2.3: Velocity field for the steady state flow.

To conclude, this type of channel flow is driven by advection and pressure gradient terms. It is adequately reproduced by TELEMAC-2D in sub-critical flow regime.

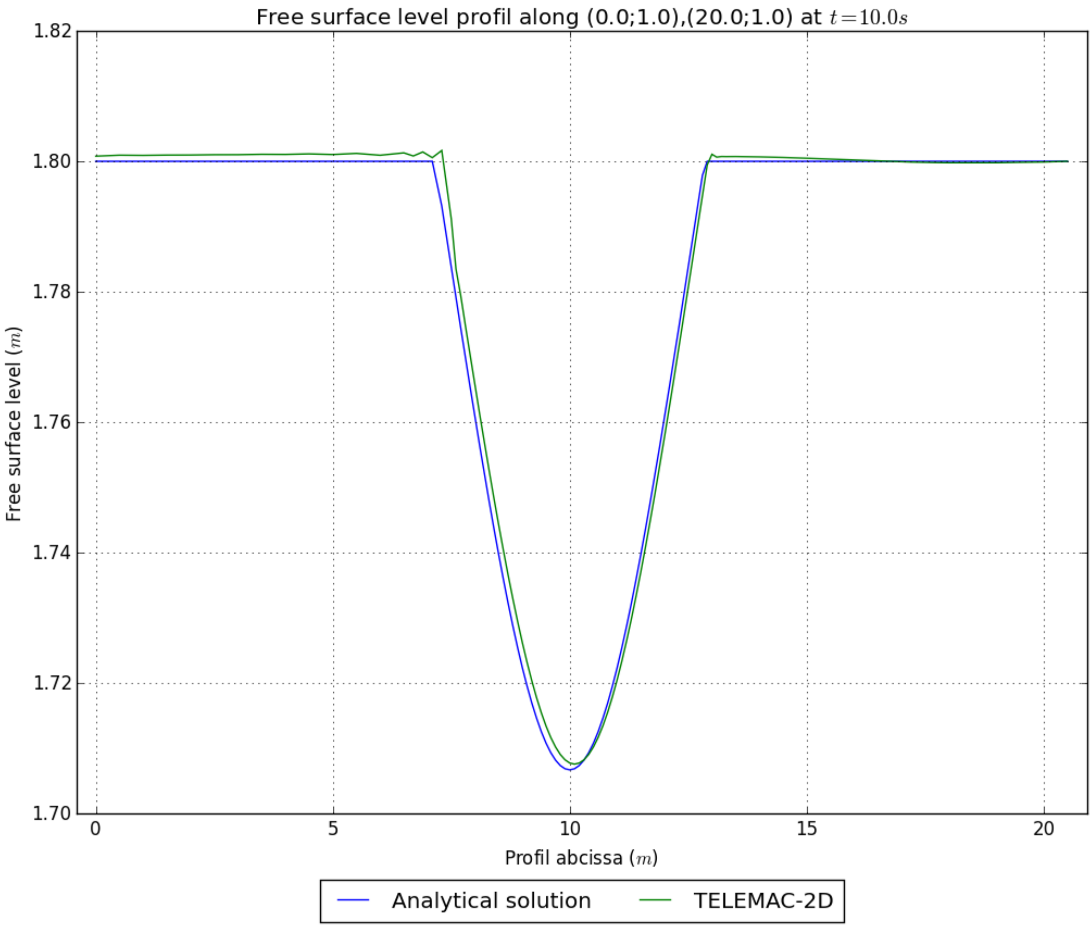


Figure 2.4: Comparison between analytical solution and TELEMAC-2D solution for the free surface elevation.

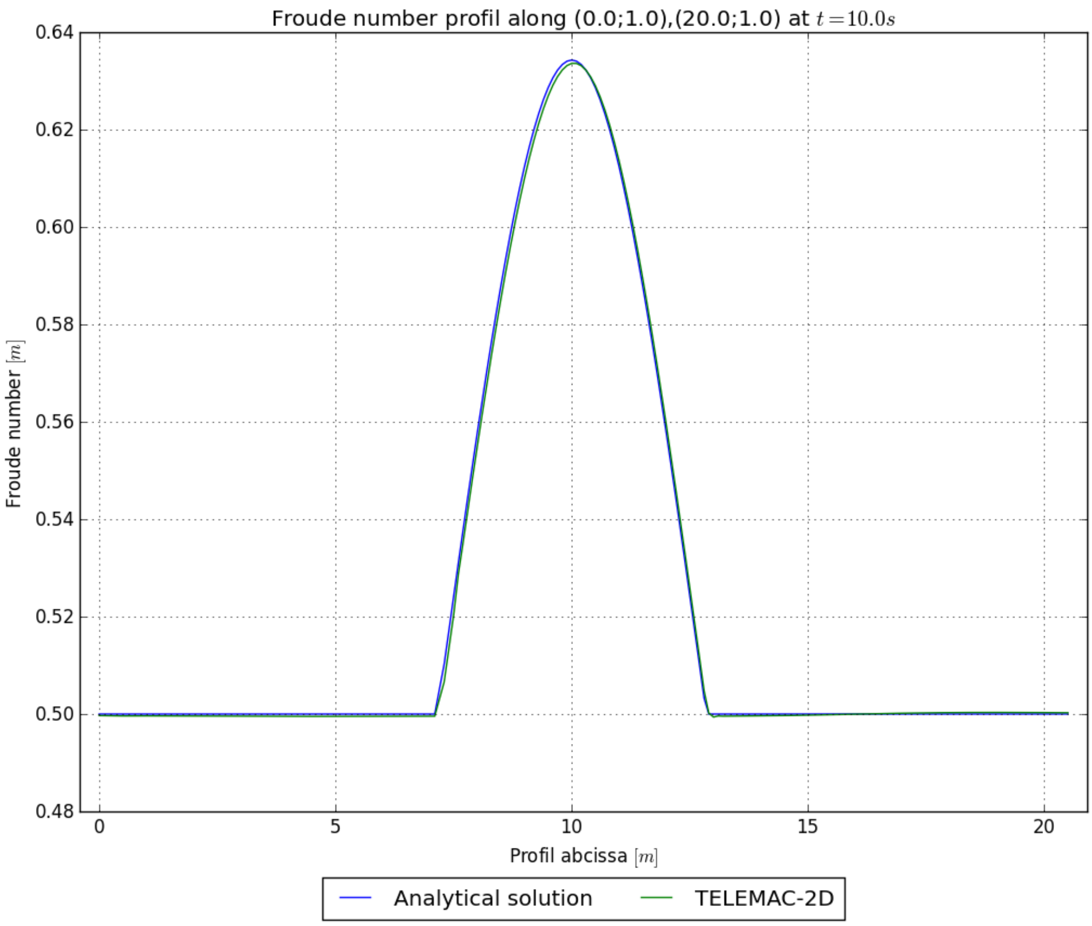


Figure 2.5: Comparison between analytical solution and TELEMAC-2D solution for the Froude number.

3. Cone

3.1 Description of the problem

This test shows the performance of the advection schemes of TELEMAC-2D for passive scalar transport in a time dependent case.
It shows the advection of a tracer (or any other passive scalar) in a square basin of dimensions $[d \times d] = [20.1 \times 20.1]$ m^2 with flat frictionless bottom and with open boundaries. The tracer is described by a gaussian function and is submitted to a rotating velocity field. In this case the scalar advection equation is solved using fixed hydrodynamic conditions. After one period we expect that the tracer function has the same position and the same values as the initial condition (i.e. maximum value equal to 1 at the center). The exact solution can indeed be computed using the theory of characteristics.
In order to evaluate the behaviour of the scheme, the error norms L^1, L^2, L^∞ and the maximum value of the gaussian function are measured after one rotation. The minimum value is computed as well, in order to check the respect of the maximum principle (or monotonicity).
The water depth is constant in time and in space, equal to 2 m. The velocity field is constant in time as well and is divergence free:

$$\boldsymbol{u} = \begin{cases} u(x,y) = -(y-d/2) \\ v(x,y) = (x-d/2) \end{cases}$$

The initial condition for the tracer is:

$$c^0(x,y) = e^{-\frac{[(x-15)^2+(y-10.2)^2]}{2}}$$

where c represents the concentration of the tracer.
The simulation time is one period of rotation equal to 2π, that is 6.28 s.

3.2 Numerical parameters

The computational domain for this test is made up by squares of side 0.3 m split into two triangles. The number of nodes is 4,624 and the number of elements is 8,978. The time step is chosen in order to do the whole period in 32 steps, so it is equal to 0.196349541 s.
For tracers advection, all the numerical schemes available in TELEMAC-2D are tested.
For weak characteristics the number of gauss points is set to 12. For distributive schemes, like predictor-corrector (PC) schemes (scheme 4 and 5 with options 2,3) and locally implicit schemes (LIPS: scheme 4 and 5 with options 4), the number of corrections is set to 5, which

is usually sufficient to converge to accurate results. For the locally implicit schemes (scheme 4 and 5 with option 4), the number of sub-steps is equal to 20.

3.3 Results

The error norms, the maximum and the minimum value of the tracer function at the end of the simulation are reported in Table 3.1. The final profiles for every scheme are plotted in Figures 3.1 and 3.2. Schemes can be listed from the most accurate to the least accurate: weak characteristics, N LIPS, PSI LIPS (these two are overlapped), ERIA, PSI PC1, N PC1, strong characteristics, PSI PC2, N PC2, PSI, N. The error computations of Table 3.1 are in agreement with the figures.
However, it is worth noticing that the weak characteristics give the highest maximum value but the scheme does not guarantee the maximum principle, indeed negative values are produced along the simulation (this is also visible on Figure 3.1 and in Table 3.1). On the contrary, all the distributive schemes guarantee the respect of the maximum principle: minimum and maximum extrema are never exceeded.
TELEMAC-2D is able to model passive scalar transport problems in shallow water flows. This test shows that to get higher accuracy and monotonicity in passive scalar transport cases the predictor-corrector distributive schemes (N PC, PSI PC or LIPS) are the most appropriate schemes, as well as the ERIA scheme.

Table 3.1: Cone test: error norms and extrema at the end of the simulation.

	$\|\|\varepsilon\|\|_{L^1}$	$\|\|\varepsilon\|\|_{L^2}$	$\|\|\varepsilon\|\|_{L^\infty}$	$\min(c)$	$\max(c)$
Strong characteristics	5.011e-03	2.719e-02	3.738e-01	0.0000	0.678
Weak characteristics	3.860e-03	1.992e-02	2.016e-01	-0.0001	0.996
ERIA	2.868e-03	1.487e-02	2.501e-01	0.0000	0.751
N	1.702e-02	6.631e-02	8.235e-01	0.0000	0.179
N LIPS	4.032e-03	1.910e-02	2.089e-01	0.0000	0.792
PSI	1.548e-02	6.237e-02	7.863e-01	0.0000	0.214
PSI PC1	3.412e-03	1.761e-02	2.752e-01	0.0000	0.725
PSI PC2	4.669e-03	2.387e-02	3.561e-01	0.0000	0.644
N PC1	3.424e-03	1.769e-02	2.759e-01	0.0000	0.724
N PC2	4.663e-03	2.386e-02	3.565e-01	0.0000	0.644
PSI LIPS	4.032e-03	1.910e-02	2.089e-01	0.0000	0.792

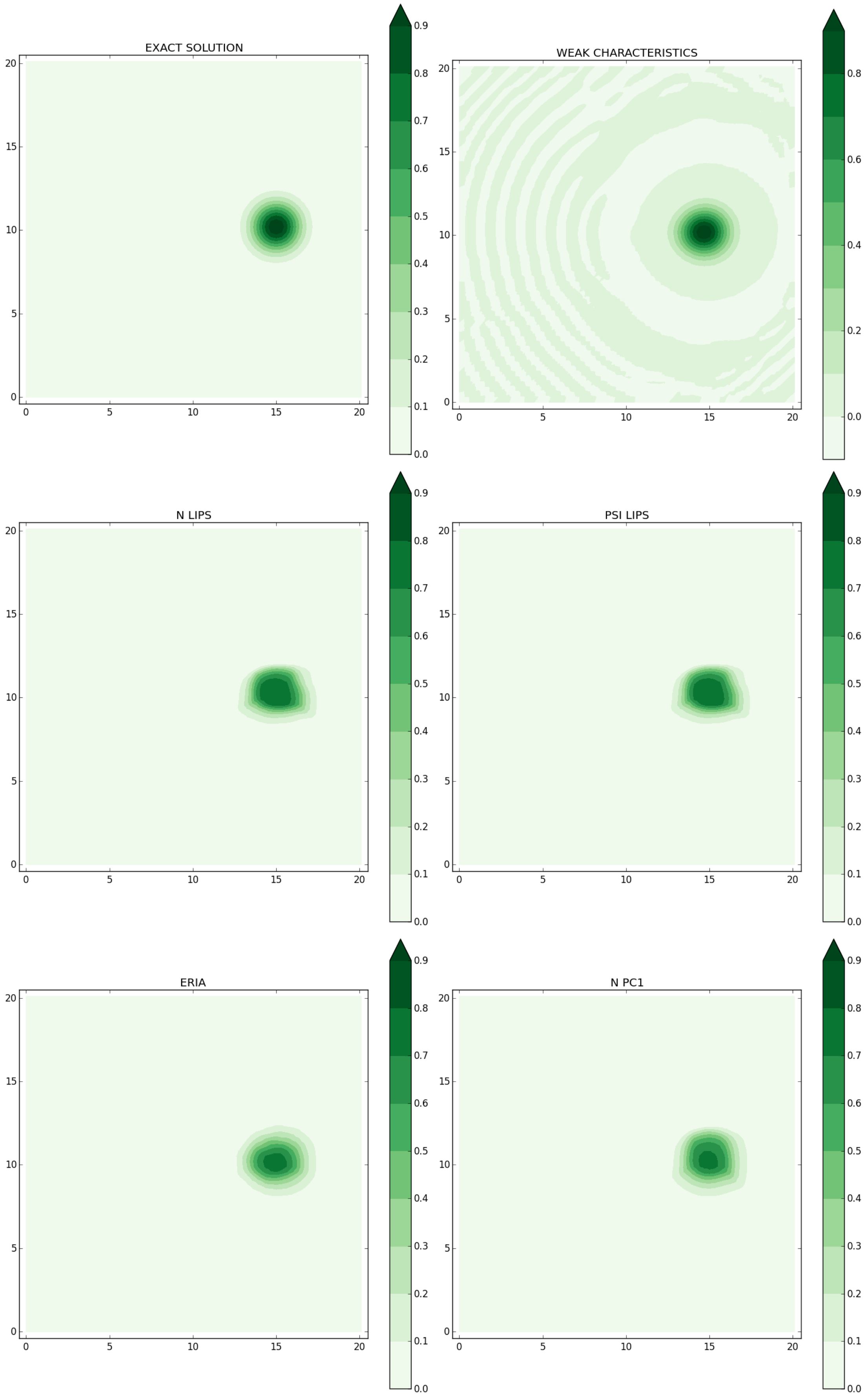


Figure 3.1: Cone test: contour lines of gaussian tracer functions after one period of rotation, for the advection schemes of TELEMAC-2D.

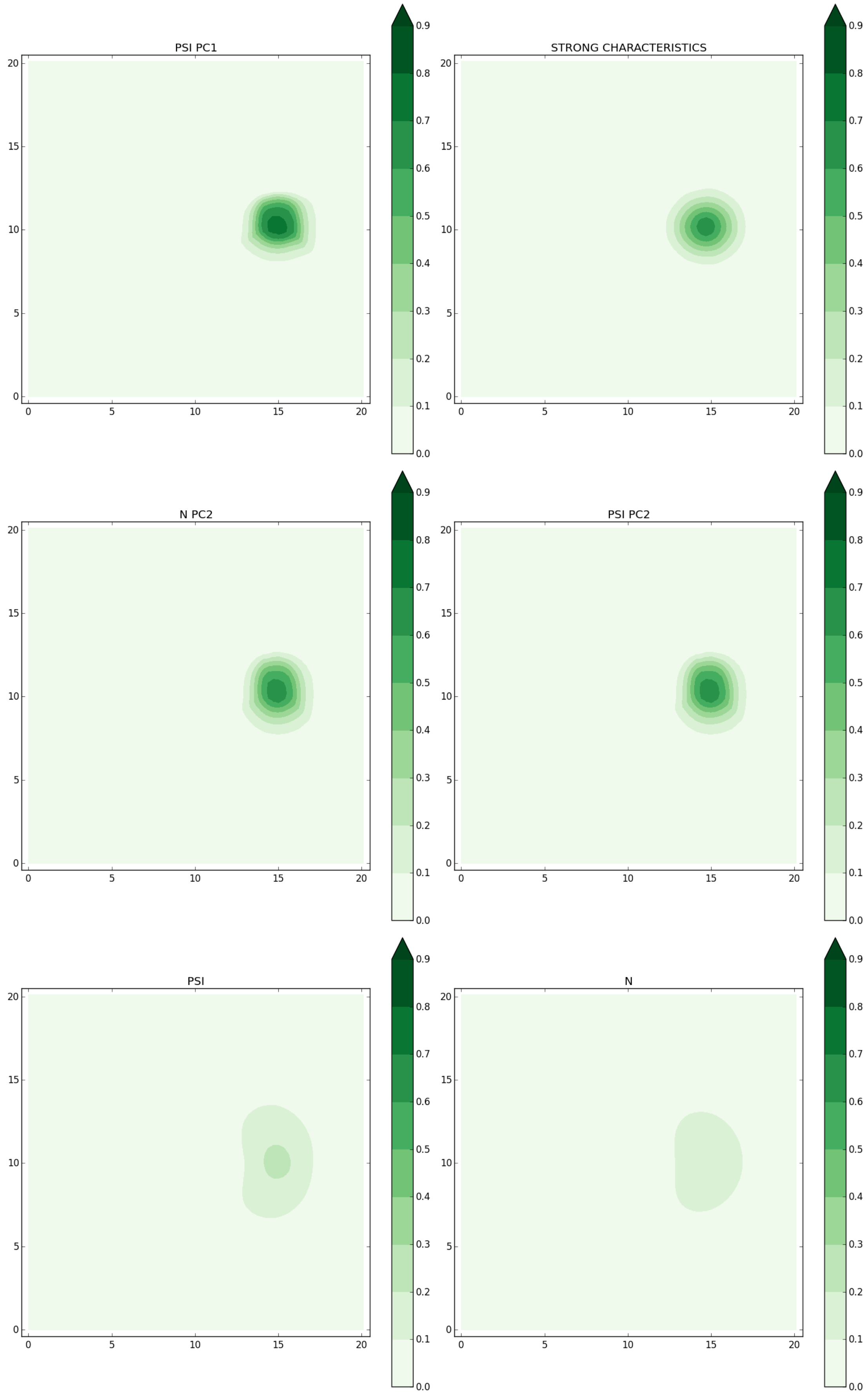


Figure 3.2: Cone test: contour lines of gaussian tracer functions after one period of rotation, for the advection schemes of TELEMAC-2D.

4. confluence

4.1 Purpose

To demonstrate that TELEMAC-2D can model the flow that occurs at a river confluence.

4.2 Description of the problem

The model represents the junction between two rectilinear laboratory channels with rectangular cross-sections and constant slope.

4.3 Physical parameters

The main channel is 0.8 m broad whereas its influent is 0.5 m broad. Both have a slope of 10^{-3} m/m. The two channels join with an angle of 55 °C.

4.4 Geometry and Mesh

Geometry:

- Size of the model :
  - main channel: 10.8 $m \times 0.8$ m
  - influent: 3.2 $m \times 0.5$ m
- Free surface at rest: 0.2852 m

Mesh:

- 6168 triangular elements
- 3303 nodes
- Maximum size range: from 0.03 to 0.1 m

The mesh is refined near the confluence as shown on Figure 4.1

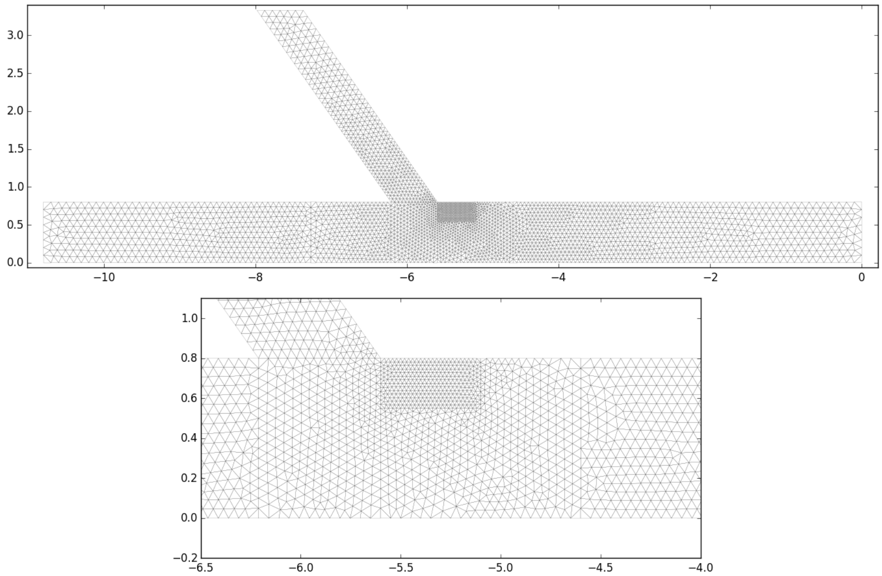


Figure 4.1: Mesh

4.5 Initial and Boundary Conditions

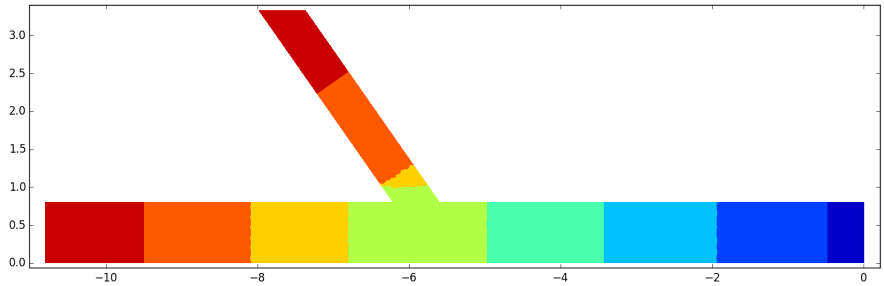
Boundaries:

- main channel:
  - channel entrance: $Q = 0.07\ m^3/s$
  - channel outlet: $H = 0.2852\ m$
- Influent:
  - channel entrance: $Q = 0.035\ m^3/s$
- Lateral boundaries: solid walls with slip condition in the channel

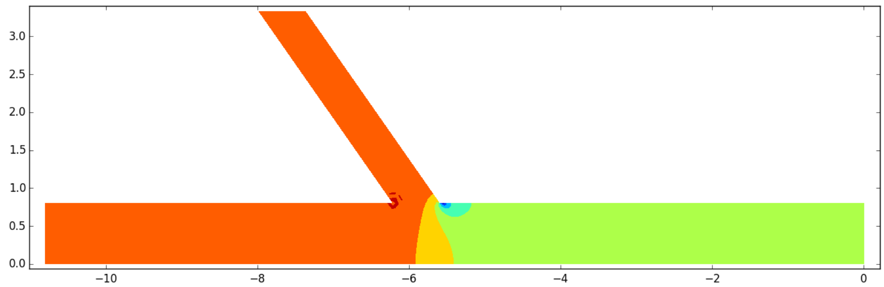
Bottom:

- Strickler formula with friction $coefficient = 62$

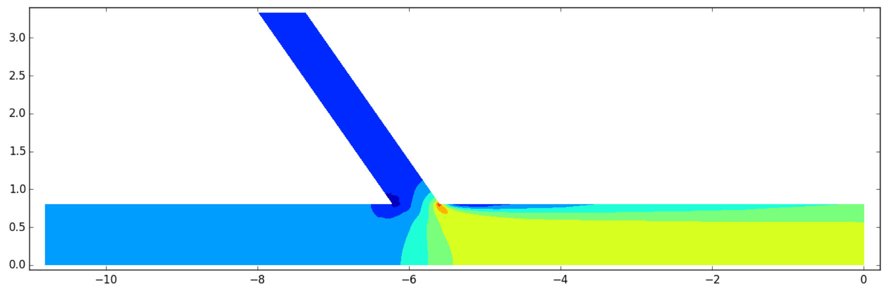
The mesh is shown on Figure 4.1 and the topography on Figure 4.2.



(a) bottom level



(b) free surface



(c) velocity

Figure 4.2: Results

Turbulence:

– Constant viscosity equal to 10^{-3} m^2/s

4.6 Numerical parameters

Algorithm:

– Type of advection:
  - Characteristics on velocities (scheme n°1)
  - Conservative + modified SUPG on depth (mandatory scheme)

– Type of element:
  - Linear triangle (P1) for h and for velocities

– Solver : Conjugate gradient

– Solver accuracy: 10^{-4}

– Implicitation for depth and for velocity: 1.0

Time data:

– Time step: 0.1 s
– Simulation duration: 100 s

4.7 Results

Initially the water level is horizontal. In the main channel and in the lateral channel, the free surface increases with time.
At the end of the calculation the water surface profile is constant in time downstream and upstream from the confluence which shows that the computation has converged.
The water depths in both channels (upstream and downstream) tend to be uniform. The water level upstream from the confluence is 0.30 m higher than the water level downstream.
Close to the confluence, the water surface is rapidly varying. The velocity field is regular in the whole domain (see Figure 4.2).
No back eddy is computed at the junction of the two rivers with the turbulence model used in this test case despite mesh refinement in this area; such back eddy has been observed on physical model experiments (see ref. [1]).

4.8 Conclusions

TELEMAC-2D reproduces appropriately free surface variations at a river confluence. However, in order to simulate in detail the flow pattern in such conditions, more sophisticated turbulence model should be used (see e.g; test case named "cavity").

4.9 Steering file

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v7p0
/ Validation test case 15
/----------------------------------------------------------------------
/----------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/----------------------------------------------------------------------

GEOMETRY FILE                      = geo_confluence.slf
FORTRAN FILE                       = t2d_confluence.f
BOUNDARY CONDITIONS FILE           = geo_confluence.cli
RESULTS FILE                       = r2d_confluence.slf
REFERENCE FILE                     = f2d_confluence.slf

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD            = 100
VARIABLES FOR GRAPHIC PRINTOUTS = 'U,V,H,S,B'
MASS-BALANCE                       = YES
GRAPHIC PRINTOUT PERIOD            = 1000

/----------------------------------------------------------------------
/ PARAMETERS
/----------------------------------------------------------------------

FRICTION COEFFICIENT    = 62.
```

```
LAW OF BOTTOM FRICTION = 3
TURBULENCE MODEL       = 1
VELOCITY DIFFUSIVITY   = 1.E-3

/----------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/----------------------------------------------------------------------

VELOCITY PROFILES     = 2;2;2
PRESCRIBED FLOWRATES  = 0.;0.035;0.070
PRESCRIBED ELEVATIONS = 0.2852;0.;0.

/----------------------------------------------------------------------
/ EQUATIONS, INITIAL CONDITIONS
/----------------------------------------------------------------------

INITIAL ELEVATION  = 0.2852
INITIAL CONDITIONS = 'CONSTANT ELEVATION'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, INFORMATION
/----------------------------------------------------------------------

VALIDATION =YES
TITLE      ='Validation test case 15'
/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

TIME STEP                         = 0.1
NUMBER OF TIME STEPS              = 1000
TREATMENT OF THE LINEAR SYSTEM    = 2
TYPE OF ADVECTION                 = 1;5
SUPG OPTION                       = 2;2
H CLIPPING                        = NO

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------
INFORMATION ABOUT SOLVER           = YES
SOLVER                             = 1
SOLVER OPTION                      = 3
MASS-LUMPING ON H                  = 1.
IMPLICITATION FOR DEPTH            = 1.
IMPLICITATION FOR VELOCITY         = 1.
```

5. convergence

5.1 Purpose

The purpose of this example is to test the framework for automatic convergence studies.

5.2 Description

It consists of a schematic case with an analytical solution. It only involves the diffusion of a tracer so that the convergence study measures the error done on the resolution of the time-discretised tracer equation:

$$\frac{T^{n+1} - T^n}{\Delta t} = K \nabla^2 T^{n+1} \tag{5.1}$$

where K is the coefficient of diffusion for the tracer and Δt is the time-step size.

5.2.1 Geometry and mesh

The case consists of a square basin of side $L = 200m$. The coarse mesh contains 40 elements, it will be refined 4 times. The mesh for each simulation and the mesh for the error calculation are shown in the Figure 5.1.

5.2.2 Initial and boundary conditions

The water height is constant and equal to $2m$ and the velocity is equal to zero. The tracer is initialised with:

$$T = \left(1 + \frac{2K}{\Delta t}\left(\frac{2\pi}{L}\right)^2\right) \sin\left(\frac{2\pi}{L}x\right) \sin\left(\frac{2\pi}{L}y\right) \tag{5.2}$$

K is taken equal to $1ms^{-2}$ and Δt equal to $1s$.

5.2.3 Analytical solution

After one time-step, the tracer should be equal to:

$$T = \sin\left(\frac{2\pi}{L}x\right) \sin\left(\frac{2\pi}{L}y\right) \tag{5.3}$$

5.2.4 Numerical parameters

The solver accuracy for the tracer diffusion is set to 10^{-10}. The advection and diffusion of velocities are deactivated in the steering file. The simulation is done for 1 iteration.

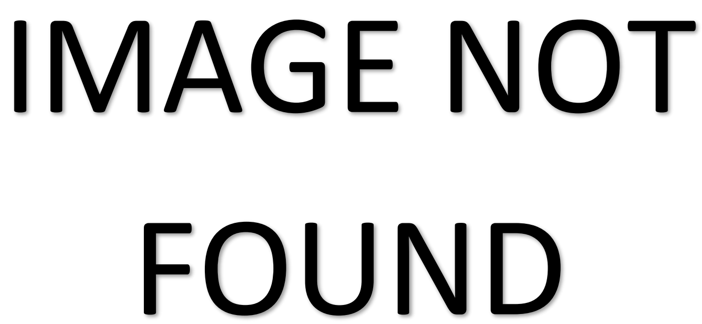
IMAGE NOT
FOUND

img/figure1.pdf is missing

Figure 5.1: Test on a schematic case: view of the meshes for each simulation – (a) initial mesh, (c), (d), (e) successive refinements and of the mesh for the error calculation (b).

5.3 Results

We compare the numerical solution for the tracer to the analytical solution after one time-step. Three refinement levels were asked for in the convergence study, yielding four TELEMAC-2D simulations.
The total time spent for the four successive runs is of $4s$ on one processor (only the diffusion matrix inversion for the tracer is performed, on one time-step). Figure 5.2 shows the results of the convergence study, regarding the L_1, L_2 and L_∞ errors. For the three of them, a first order convergence is obtained. For the simulation on the finest mesh, the error is slightly higher than expected, probably due to the worsened aspect ratio of the triangles after three refinements.



`img/figure2.pdf` is missing

Figure 5.2: Test on a schematic case: results of the convergence study with three refinement levels.

6. Gouttedo: Gaussian water surface centred in a square domain - Solid boundaries

6.1 Purpose

To demonstrate that the Telemac-2D solution is not polarised because it can simulate the circular spreading of a wave. Also to show that the no-flow condition is satisfied on solid boundaries and that the solution remains symmetric after reflection of the circular wave on the boundaries.

6.2 Description of the problem

6.2.1 Domain

The domain is square with a size of 20.1 m x 20.1 m with a flat bottom.

6.2.2 Mesh

The domain is meshed with 8978 triangular elements and 4624 nodes. Triangles are obtained by dividing rectagular elements on their diagonals. The mean size of obntained triangles is about 0.3 m (see figure 6.1).

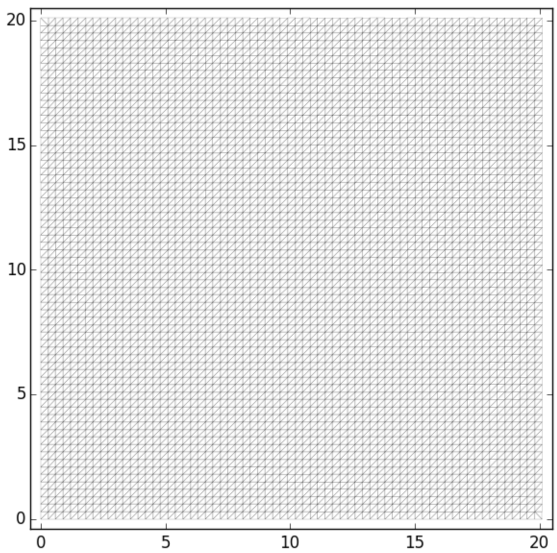


Figure 6.1: Gouttedo case: used mesh

6.2.3 Initial conditions

The fluid is initially at rest with a Gaussian free surface in the centre of a square domain (see Figure 6.2). Water depth is given by $H = 2.4 \exp\frac{-[(x-10)+(y-10)]}{4}$

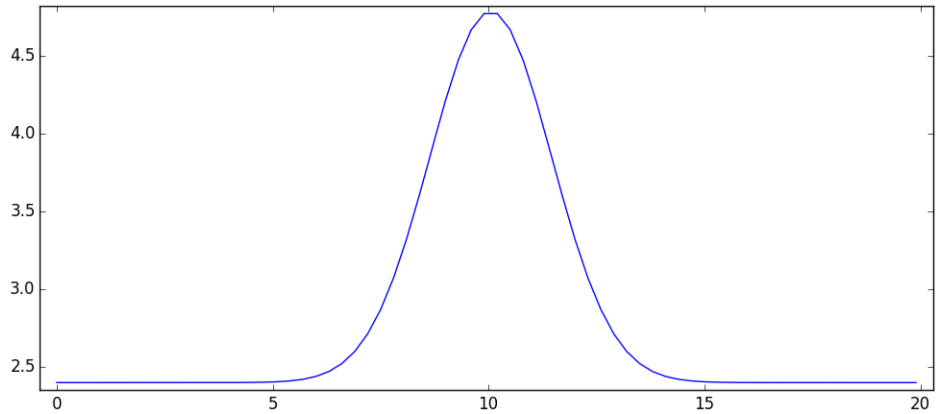


Figure 6.2: Gouttedo case: initial elevation

6.2.4 boundary conditions

Boundaries are considered as solid walls with perfect slip conditions (condition 2 2 2)

6.2.5 Physical parameters

The physical parameters used for this case are the following:

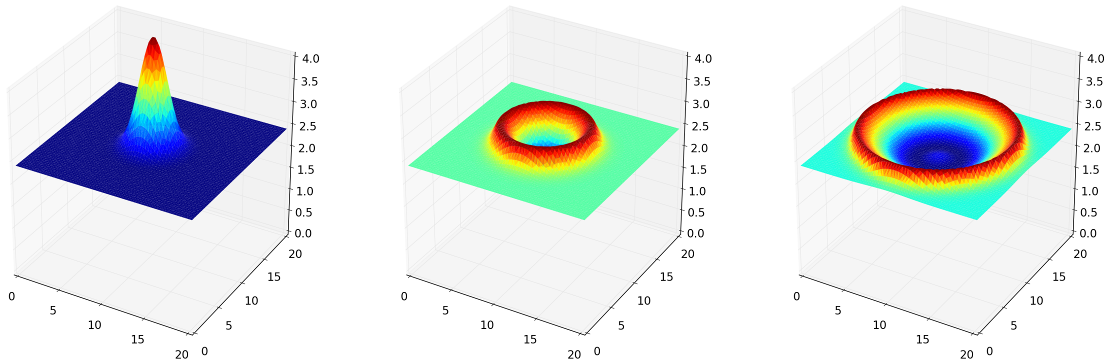
1. Friction: Strickler formula with K = 40
2. Turbulence: Constant viscosity equal to zero (or disactivation of diffusion step using the keyword *DIFFUSION OF VELOCITY = NO*)

6.2.6 Numerical parameters

1. Type of advection: centred semi-implicit scheme + SUPG upwinding on velocities (2=SUPG)
2. Type of advection: conservative + modified SUPG on depth (mandatory scheme)
3. Type of element: Linear triangle (P1) for velocities and Linear triangle (P1) for h
4. Solver : GMRES with an accuracy = 10^{-4}
5. Time step : 0.4 sec.
6. Simulation time : 4 sec.

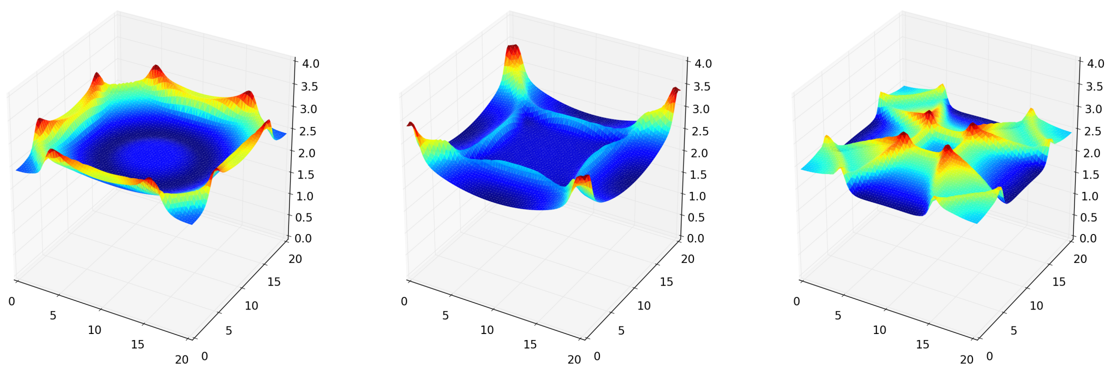
6.3 Results

The wave spreads circularly around the initial water surface peak elevation (Figures 6.3). When it reaches the boundaries, reflection occurs. Interaction between reflected waves issuing from the four walls can be observed after time 1.8 sec (Figures 6.4).



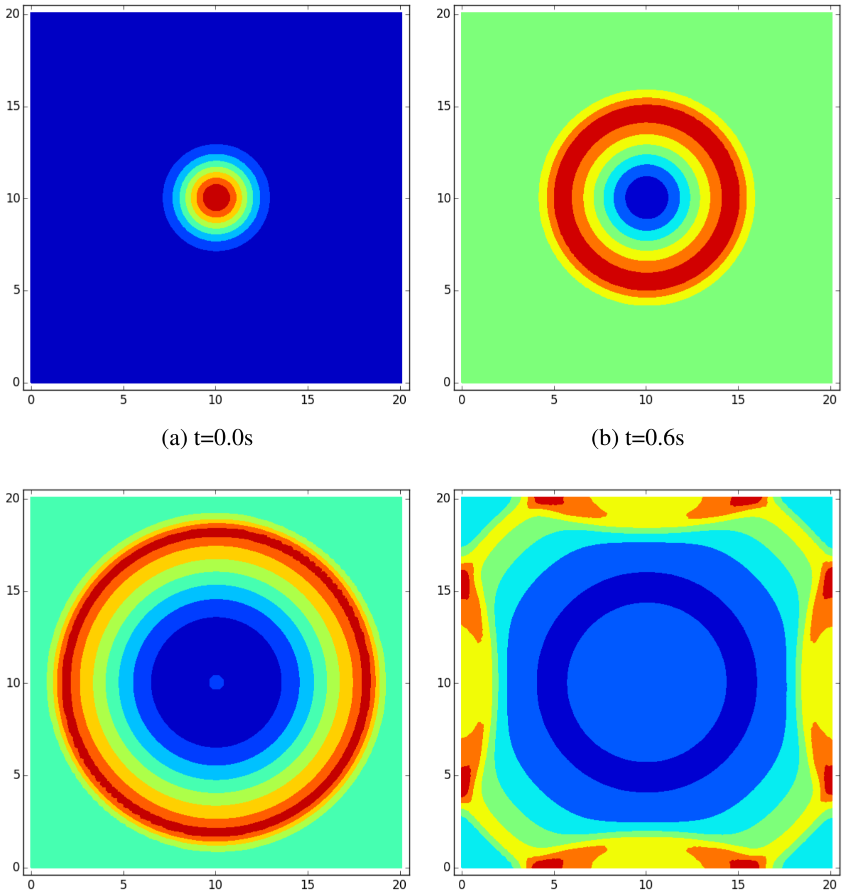
(a) t=0.0s (b) t=0.6s (c) t=1.2s

Figure 6.3: Gaussion water hill: Initial conditions and circular spreading



(a) t=1.8s (b) t=2.4s (c) t=3.0s

Figure 6.4: Gaussion water hill : reflexions



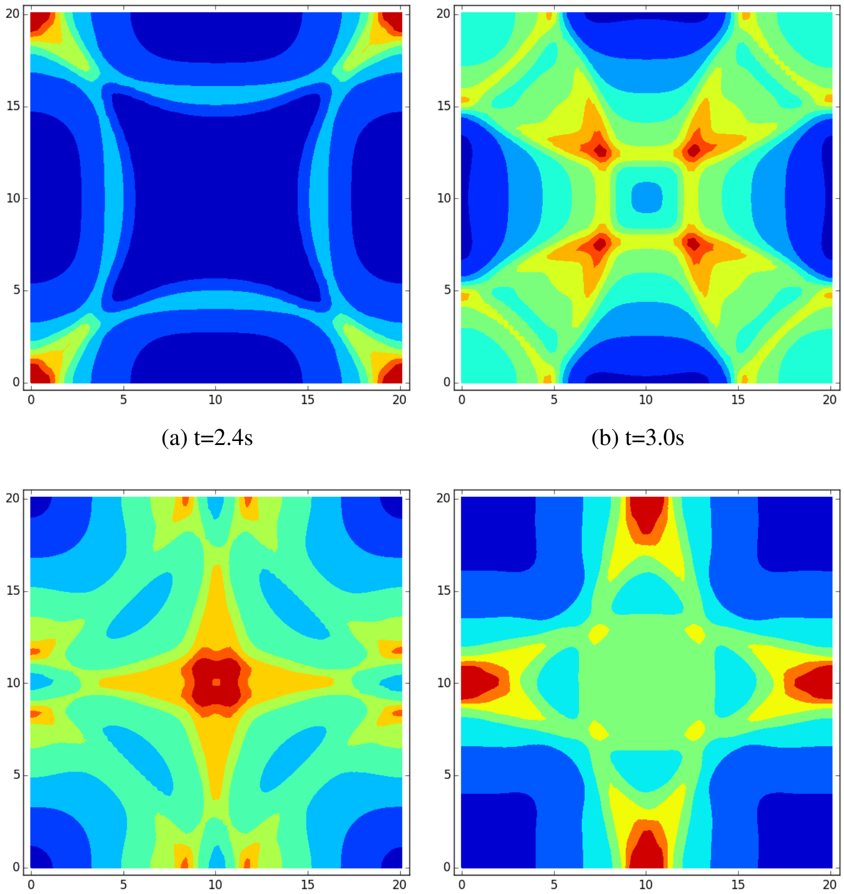
(a) t=0.0s

(b) t=0.6s

(c) t=1.2s

(d) t=1.8s

Figure 6.5: Evolution of the water depth from 0.0 sec. to 1.8 sec



(a) t=2.4s (b) t=3.0s

(c) t=3.6s (d) t=4.0s

Figure 6.6: Evolution of the water depth from 2.4 sec. to 4.0 sec

6.4 Reference

For this case, we do not have a reference to compare with. The aim is mainly to observe qualitatively the behavior of flow induced by a gaussian hill and then reflected on solid boundaries.

6.5 Conclusions

Even though the mesh is polarised (along the x and y directions and the main diagonal), the solution is not. Solid boundaries are treated properly: no biais occurs in the reflected wave. Water mass is conserved.

6.6 Steering file

```
/---------------------------------------------------------------------/
/   TELEMAC-2D        VALIDATION TEST CASE NUMBER 1                   /
/                   GAUSSIAN HILL WITH SOLID BOUNDARIES               /
/---------------------------------------------------------------------/
/----------------------------------------------
/ COMPUTER INFORMATIONS
/----------------------------------------------
/
GEOMETRY FILE                     = geo_gouttedo.slf
```

```
FORTRAN FILE                    = t2d_gouttedo.f
BOUNDARY CONDITIONS FILE        = geo_gouttedo.cli
RESULTS FILE                    = r2d_gouttedo_v1p0.slf
REFERENCE FILE                  = f2d_gouttedo.slf
/
/-----------------------------------------------
/ GENERAL INFORMATIONS - OUTPUTS
/-----------------------------------------------
/
TITLE                           = 'GAUSSIAN WALL'
VARIABLES FOR GRAPHIC PRINTOUTS = 'U,V,H,T'
GRAPHIC PRINTOUT PERIOD         = 5
LISTING PRINTOUT PERIOD         = 10
VALIDATION                      = YES
TIME STEP                       = 0.04
NUMBER OF TIME STEPS            = 100
MASS-BALANCE                    = YES
INFORMATION ABOUT SOLVER        = YES
/
/-----------------------------------------------
/ INITIAL CONDITIONS
/-----------------------------------------------
/
COMPUTATION CONTINUED           = NO
INITIAL CONDITIONS              = 'PARTICULAR'
/
/-----------------------------------------------
/ PHYSICAL PARAMETERS
/-----------------------------------------------
/
LAW OF BOTTOM FRICTION          = 3
FRICTION COEFFICIENT            = 40.
TURBULENCE MODEL                = 1
VELOCITY DIFFUSIVITY            = 0.
/
/-----------------------------------------------
/  NUMERICAL PARAMETERS
/-----------------------------------------------
/
TYPE OF ADVECTION               = 2;5
SOLVER                          = 7
SOLVER OPTION                   = 2;2
SOLVER ACCURACY                 = 1.E-4
DISCRETIZATIONS IN SPACE        = 11 ; 11
PRECONDITIONING                 = 2
INITIAL GUESS FOR H             = 1
IMPLICITATION FOR DEPTH         = 0.6
IMPLICITATION FOR VELOCITY      = 0.6
MATRIX-VECTOR PRODUCT           = 1
MATRIX STORAGE                  = 3
/
/-----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/-----------------------------------------------
/
/EQUATIONS                         = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME              = 6
/VARIABLE TIME-STEP                = YES
/DESIRED COURANT NUMBER            = 0.8
/DURATION                          = 4.
/-----------------------------------------------
```

```
&FIN
```

7. hydraulic jump

7.1 Description of the problem

This test case presents a flow over a bump on the bed with super-critical condition and hydraulic jump. It allows to show that TELEMAC-2D is able to correctly reproduce the hydrodynamic impact of a changing bed slopes, vertical flow contractions and expansions. Furthermore, It allows to have a good representation of flows computed in steady and transient flow regimes. Finally, with appropriate choice of the mesh resolution, this test case allows also to see that TELEMAC-2D produces solutions without wiggles (i.e. oscillations in space at a given time or oscillations in time at a given position).

The solution produced by TELEMAC-2D in a channel with friction presenting an idealised bump on the bottom is compared with the analytical solution to this problem. The studied flow regime is sub-critical upstream; a transition to super-critical flow conditions is located on the bump, and the downstream water level imposes the presence of an hydraulic jump in the downstream reach of the channel.

The geometry dimensions of the channel are 2 m wide and 20.5 m long. It is horizontal with a 4 m long bump in its middle (see Figure 7.1). The maximum elevation of the bump is 0.2 m (see Figure 7.2) with the bottom z_f is describe by the equation following :

$$z_f = \begin{cases} -0.05(x-10)^2 \text{ m} & \text{if } 8 \text{ m} < \text{x} < 12 \text{ m} \\ -0.20 \text{ m} & \text{elsewhere} \end{cases}$$

Note that the horizontal viscosity turbulent is constant and equal to zero. However instead of prescribing a zero viscosity, no diffusion step could have been used (keyword DIFFUSION OF VELOCITY prescribed to NO).

7.2 Initial and boundary conditions

The initial conditions are a null velocity and water level equal to 0.4 m along channel.

The boundary conditions are:

- At the channel entrance, the flow rate is $Q = 2 \text{ m}^3\text{s}^{-1}$.

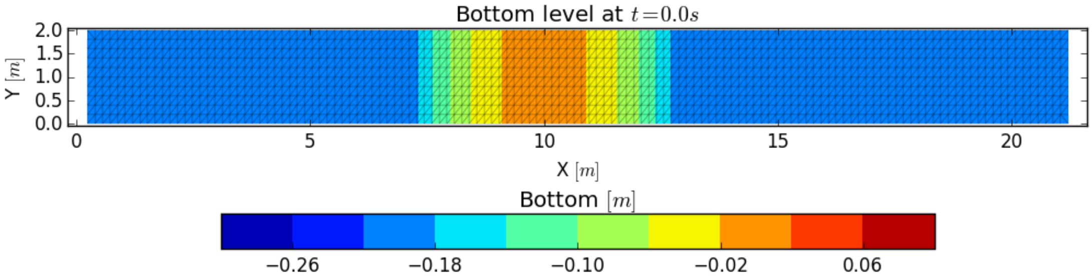


Figure 7.1: Mesh and topography of the channel.

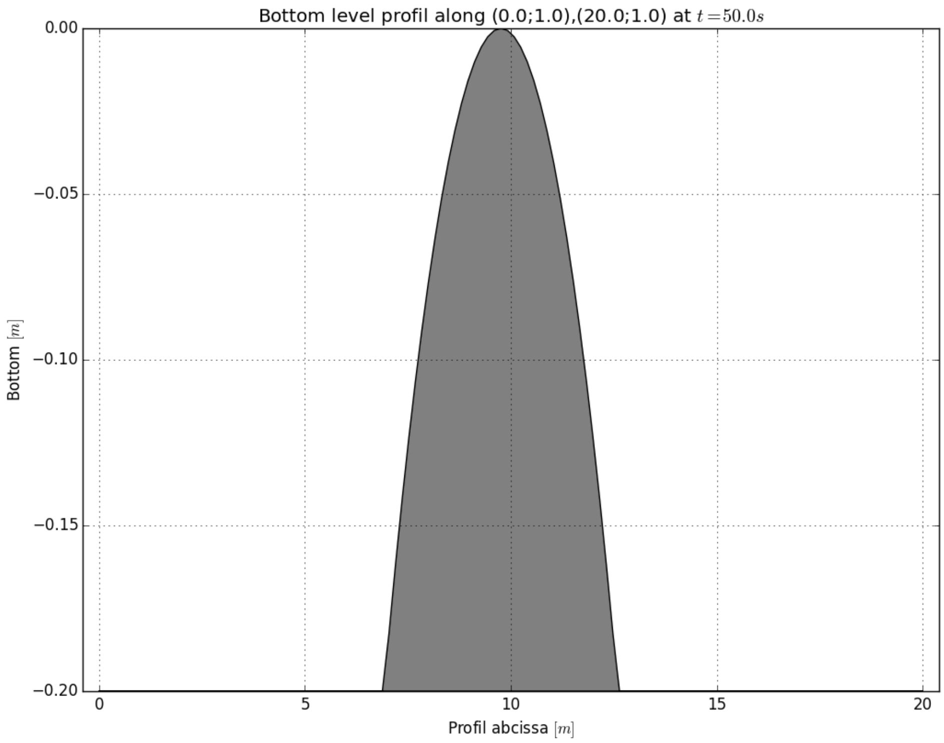


Figure 7.2: Profile of the bump.

- At the channel outlet, the free surface elevation is $z_s = 0.4$ m. Therefore, the depth $h = z_s - z_f = 0.6$ m.
- On the bottom Strickler formula with friction coefficient equal to 40 $m^{1/3}.s^{-1}$ is imposed.
- No friction on lateral walls.

7.3 Mesh and numerical parameters

The mesh is regular along channel. It is made up with quadrangles split into two triangles. It is composed of 2,620 triangular elements (1,452 nodes) and the size of triangles ranges between 0.25 m and 0.5 m. The triangular elements types are linear triangles (P1, 3 values per element, the corners) for water depth and quadratic triangle (6 values per element, the corners and the center of the edges) for velocities ("t2d_hydraulic_jump_v2p0.case" steering file).

The time step is 0.02 s for a period of 50 s. In fact, TELEMAC-2D is run forward in time until a steady state flow is obtained. The resolution accuracy for the velocity is taken at 10^{-5}.

Note that for numerical resolution, GMRES (Generalized Minimal Residual Method) is used for solving the propagation step (option 7). To solve advection, the characteristics scheme is used on velocities (scheme 1) and the conservative PSI scheme is used for the depth (scheme 5). In addition, the treatment of linear system is done with a wave equation. To finish, the implicitation coefficient for depth and velocities is equal at 0.6.

7.4 Results

As seen in the test case of the section 1, the sub-critical flow upstream the bump and the transition are correctly reproduced with TELEMAC-2D.
The prescribed downstream elevation defines a backwater line which takes into account roughness effects on the downstream reach. The super-critical flow issuing from the downstream foot of the bump also generates a water line. The hydraulic jump should be located in a position where the water depths of these two water lines are conjugate, i.e. they are related by the hydraulic jump equation:

$$\frac{h_1}{h_2} = \frac{1}{2}\left(\sqrt{1+8F_{r1}^2} - 1\right)$$

where index 1 relates to the position upstream the jump and index 2 relates to the position downstream the jump. Let us notice that it is necessary to have bottom friction to have a well-posed problem in this test case. In the contrary, the water lines issuing from the foot of the bump and from the downstream end respectively would be parallel to the bed, and the jump could not occur in the downstream reach.

The hydraulic jump computed by TELEMAC-2D occurs in a position where $h_1 = 0.18$ m , $h_2 = 0.41$ m and $F_{r1} = 1.64$ (see Figures 7.3 and 7.5). These values satisfy with more or less accuracy the hydraulic jump formula. In addition, the followings other results are well obtained :

- The output flow rate is equal to the input flow rate thank to flow acceleration after the bump (Figure 7.4).
- The Froude number is equal to 1 at the position of the bump (Figure 7.5).

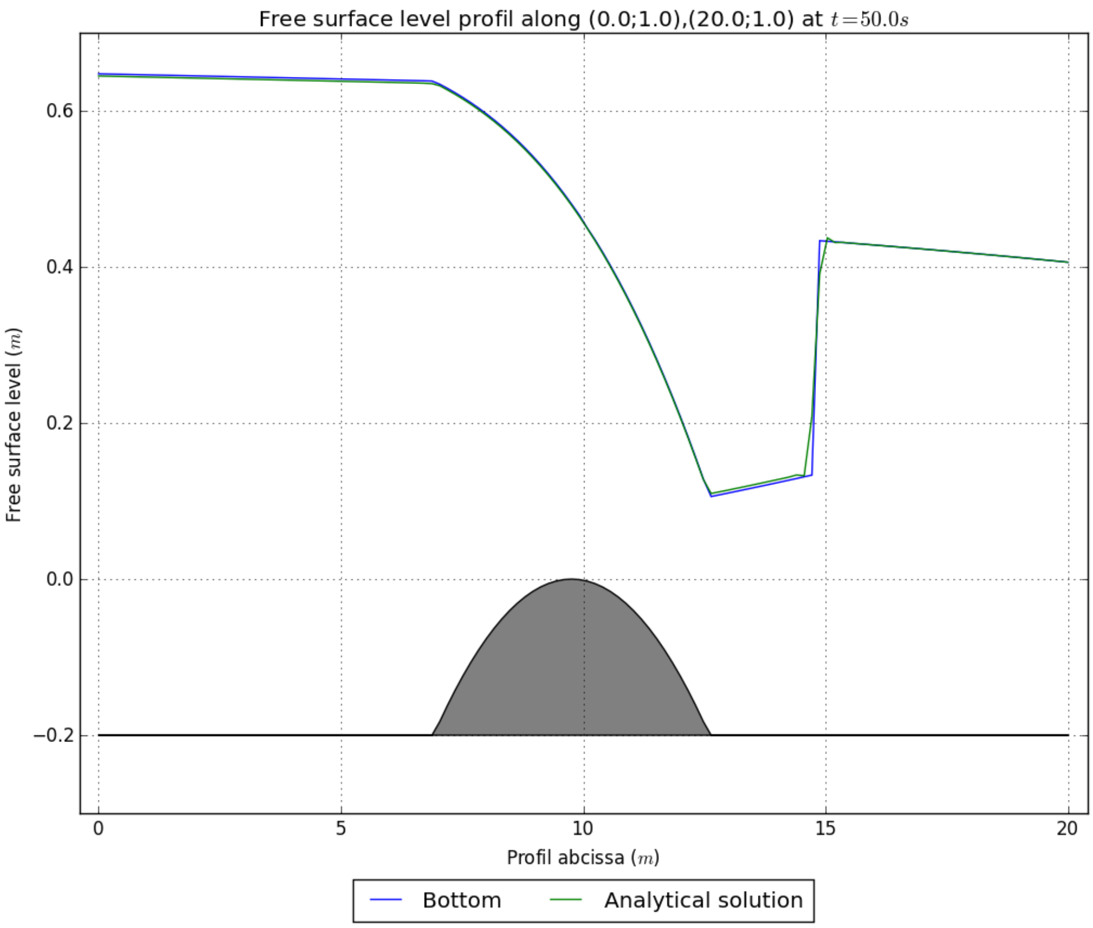


Figure 7.3: Comparison between analytical solution and TELEMAC-2D solution for the free surface elevation.

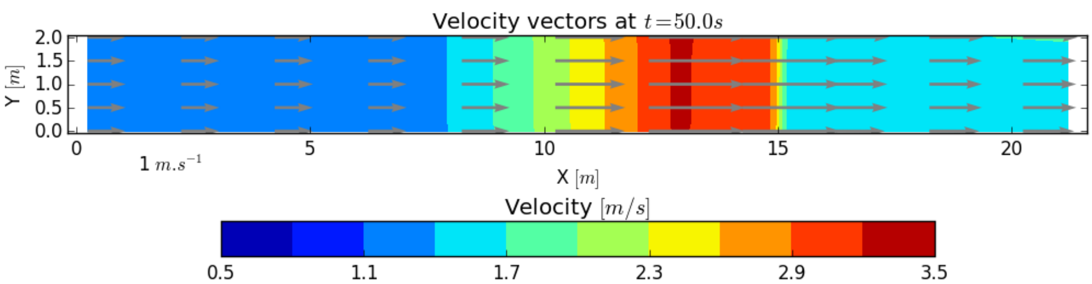


Figure 7.4: Velocity field for the steady state flow.

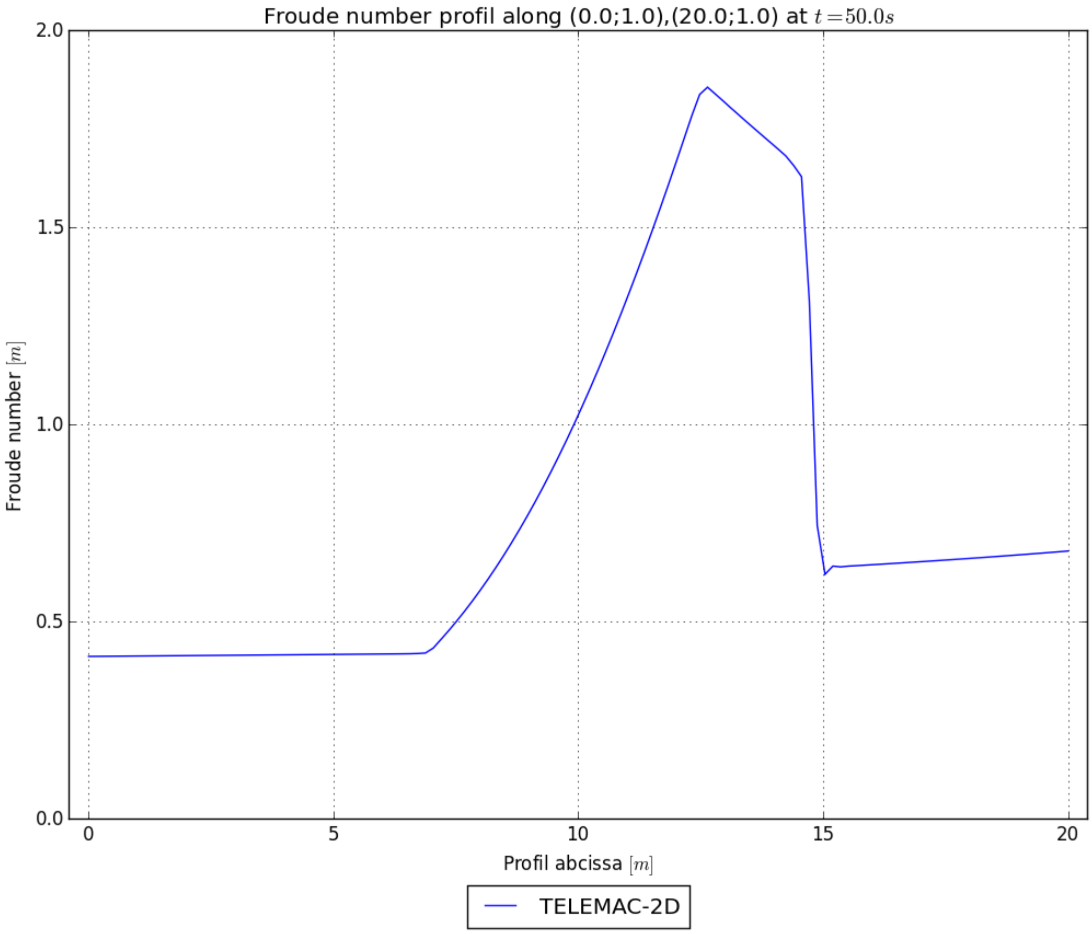


Figure 7.5: Froude number of TELEMAC-2D.

To conclude, the transition from a super-critical to a sub-critical flow regime through a hydraulic jump is properly reproduced by TELEMAC-2D, provided the mesh resolution is fine enough. The hydraulic jump formula is satisfied and the jump is located in the right position.

8. Malpasset

8.1 Description

This test illustrates that TELEMAC-2D is able to simulate a real dam break flow on an initially dry domain. It also shows the propagation of the wave front and the evolution in time of the water surface and velocities in the valley downstream.

This case is the simulation of the propagation of the wave following the break of the Malpasset dam (South-East of France). Such accident really occurred in December 1959. The model represents the reservoir upstream from the dam and the valley and flood plain downstream. The entire valley is approximately 18 km long and between 200 m (valley) and 7 km wide (flood plain). The complete study is described in details in [1].

Note that the simulation is performed using the treatment of negative depths introduced since TELEMAC-2D 7.0. The historical simulation using the method of characteristics (named "t2d_malpasset-small_charac.cas") has been kept. Nevertheless, the recommended advection scheme for velocities for such applications is now the NERD scheme (scheme 14) which is the presented cas here (named "t2d_malpasset-small_pos.cas"). A simulation using a finer mesh (named "t2d_malpasset-large") is also performed, but the results aren't presented.

The geometry dimensions of model are around 17 km long and 9 km wide. The real bathymetry is shown in the Figure 8.1.
Note also that the horizontal turbulent viscosity is constant and equal to 1 $m^2 \cdot s^{-1}$.

8.2 Initial and boundary conditions

At initial time, the velocity is null. The reservoir is full and the downstream valley is dry.

The boundary conditions are:

- Solid boundaries everywhere, e.i., no entrance and no outlet in the domain.
- On the bottom, Strickler formula with friction coefficient equal to 30 $m^{1/3}.s^{-1}$ is imposed.
- No friction on the solid boundaries (lateral walls).

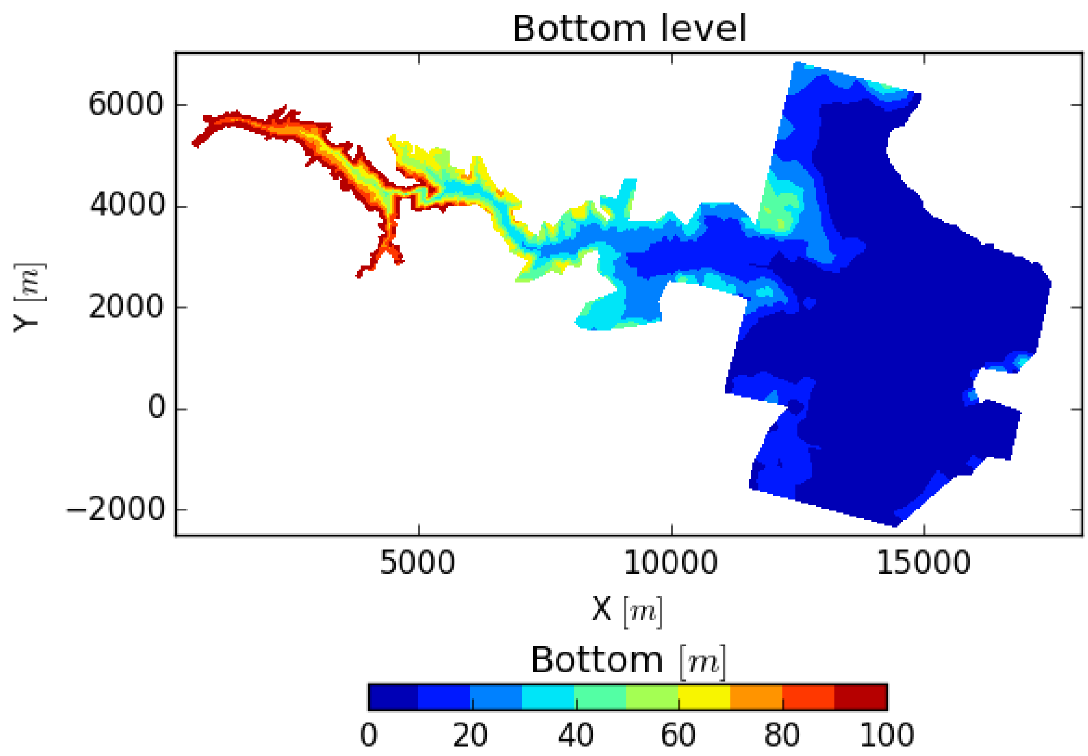


Figure 8.1: Bathymetry of Malpasset domain.

8.3 Mesh and numerical parameters

The mesh is regular (seen Figure 8.2). It's refined in the river valley (downstream from the dam) and on the banks. It is composed of 26,000 triangular elements (13,541 nodes) and the size of triangles ranges between 17 m and 313 m. The triangular elements types are linear triangles (P1, 3 values per element, the corners) for water depth and for velocities. The time step is 4 s

for a simulate period of 4,000 s. The resolution accuracy for the velocity is fixed at 10^{-4}.

Note that for numerical resolution, the conjugate gradient is used for solving the propagation step (option 1). To solve advection, the NERD scheme is used for the velocities (scheme 14) with the treatment of negative depths for tidal flats and the conservative PSI scheme is used for the depth (scheme 5). In addition, the treatment of linear system is done with a wave equation. To finish, the implicitation coefficient for depth and velocities is respectively equal at 1 and 0.55.

Note that the finer mesh is also regular and it is composed of 104,000 triangular elements (53,081 nodes) and the size of triangles ranges between 8.5 m and 156.5 m. The triangular elements types are linear triangles (P1, 3 values per element, the corners) for water depth and for velocities. The time step is 1 s due to mesh. The free surface gradient compatibility is equal at 0.9 and the implication for velocity is equal at 1. The other parameters are similar than those of previous test.

Other advection schemes for the velocities are also tested but the results are not presented here. These other schemes are:

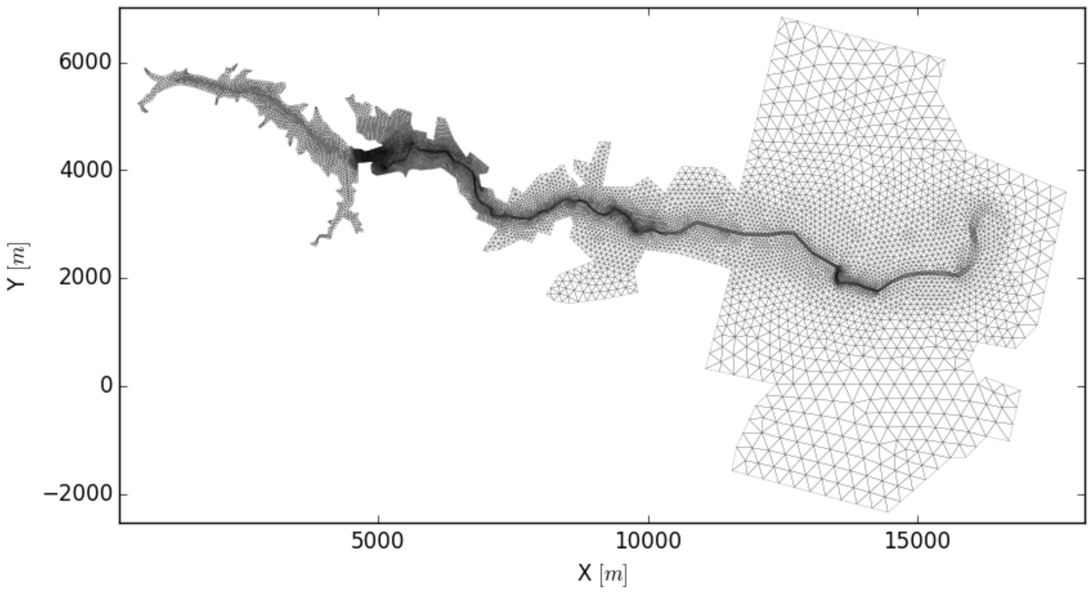


Figure 8.2: Mesh of Malpasset domain.

- the ERIA scheme ("t2d_malpasset-small_ERIA.cas", scheme 15)
- the historical method of characteristics ("t2d_malpasset-small_charac.cas", scheme 1)
- the 1st Order Kinetic scheme ("t2d_malpasset-small_cin.cas", finite volume method with the scheme 5)
- the coupled primitive equations ("t2d_malpasset-small_prim.cas", scheme 1 with method 1 for the treatment of linear system)

The time step is normally 4 s for these advection test cases except for the tests with the 1st Order Kinetic (1 s) and for the using of the primitive equations (0.5 s). Some other parameters may vary between these tests.

8.4 Results

Figure 8.3 illustrates the progression of the flood wave after the dam break (the simulation using the treatment of negative depths smooths the results on tidal flats). The propagation of the wave front is very fast. The water depths increase rapidly in the valley downstream from the dam location. The wave spreads in the plain before the sea. During the simulation, no negative water depths are observed.

Figure 8.4 represents the velocity patterns at successive times as given by the computation. Maximum velocities are close to 14 $\mathrm{m} \cdot \mathrm{s}^{-1}$ in sharp and narrow meanders of the river valley in which the dam break wave propagates. Water mass balance shows that the mass conservation is very good: the relative error cumulated on volume is $0.308 \cdot 10^{-14}$. A complete comparison between simulation results produced by TELEMAC-2D and in-situ data available collected immediately after the catastrophe has been reported by Hervouet in [1].

TELEMAC-2D is so capable to simulate the propagation of a dam break wave in a river valley initially dry.

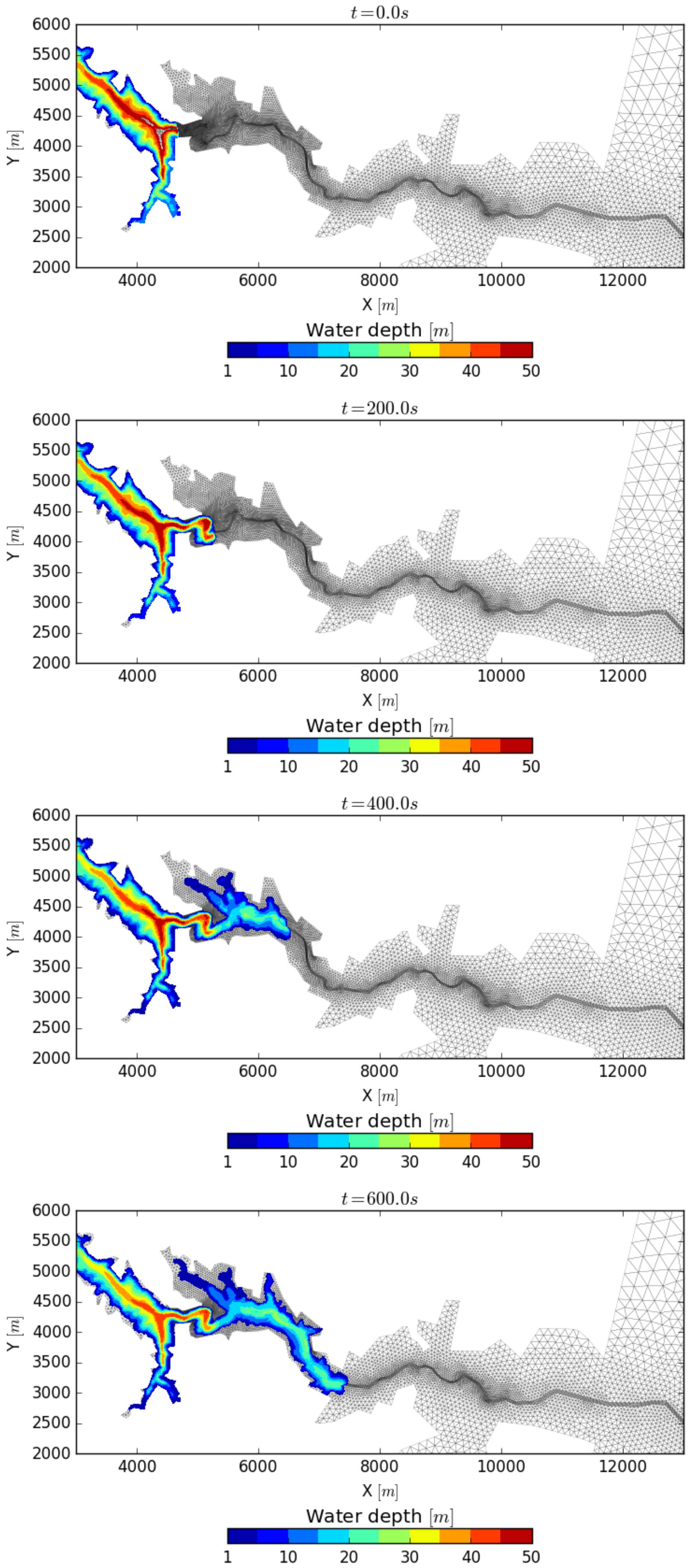


Figure 8.3: Evolution of the water depth in time.

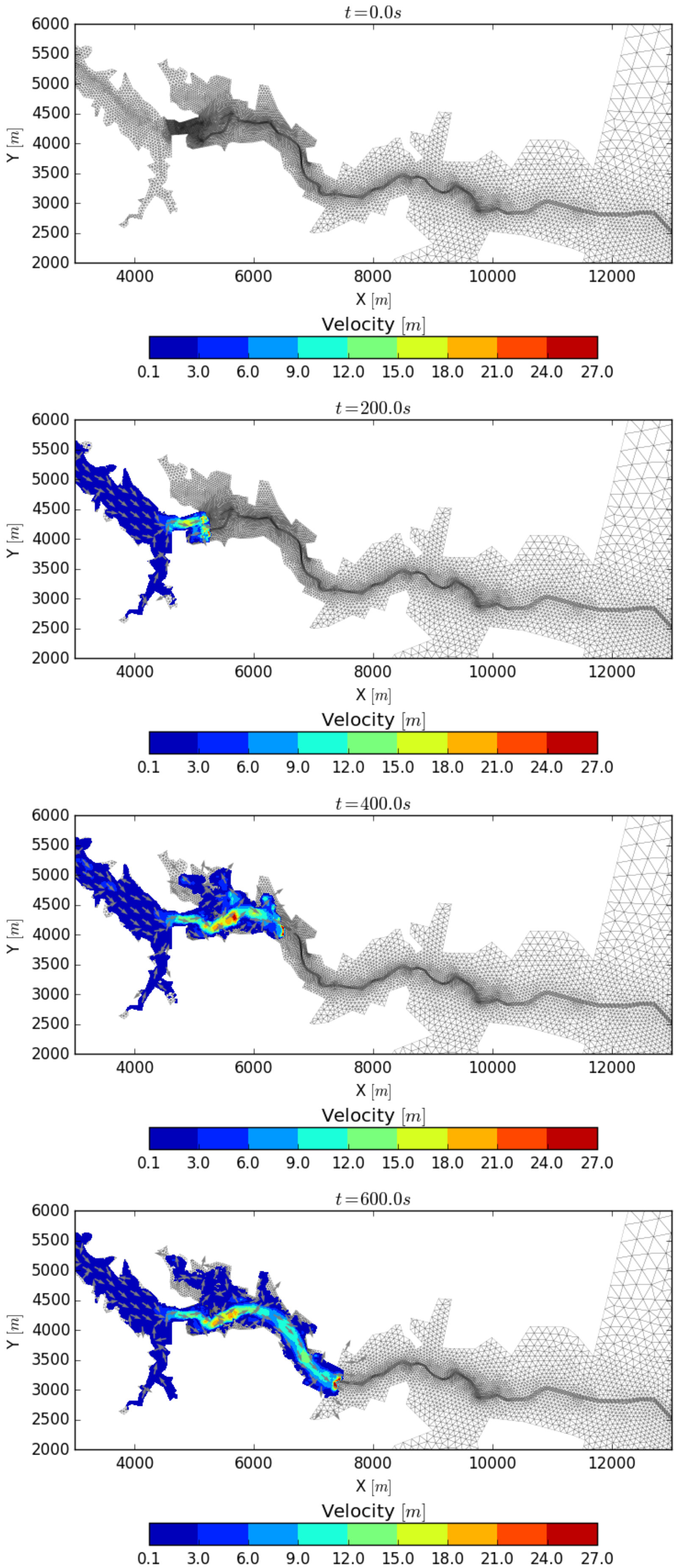


Figure 8.4: Evolution of the velocity field in time.

8.5 Reference

[1] Hydrodynamics of Free Surface Flows modelling with the finite element method. Jean-Michel Hervouet (Wiley, 2007) pp. 281-288.

9. Pildepon

9.1 Description of the problem

This test case shows that TELEMAC-2D is able to represent the impact of an obstacle on a channel flow : it simulates a laminar and very viscous flow in a channel with two cylindrical piers.
The channel is 28.5 m long and 20 m wide (L=28.5 m and H=20 m) with two bridge piers positioned at $P_1 = (-5,4)$, $P_2 = (-5,-4)$ and a diameter D of 4 m. The geometry is shown in the Figure 9.1.

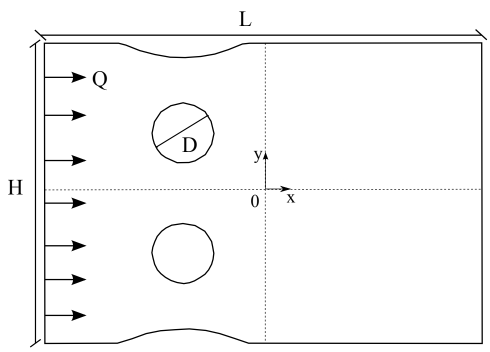


Figure 9.1: Geometry of the pildepon test case.

The section is trapezoidal (see the bottom in the Figure 9.2) and the minimum value of the bottom elevation is equal to -4 m in the main channel. The bottom friction is described by the Strickler law with a coefficient equal to $k_s = 40$ m$^{1/3}$s^{-1}.

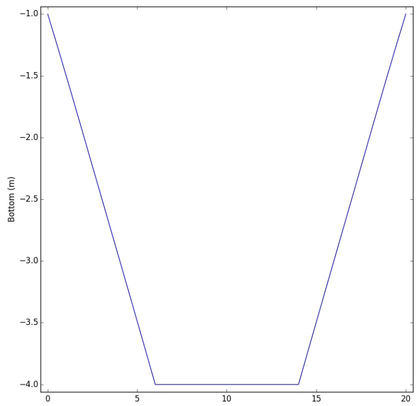


Figure 9.2: Topography at the inlet of the channel.

At the inlet of the channel, we gradually impose as upstream boundary condition a flow discharge Q = 62 m^3s^{-1}, while at the outlet a null free surface is imposed, which is also the initial condition. On the lateral walls and on the cylinder, slip boundary conditions are imposed. The fluid considered presents a kinematic viscosity $\nu = 0.021$ m^2s^{-1}. The average flow velocity in the upstream undisturbed field is about U = 0.95 m/s. Taking into account the diameter of the cylindrical pier, the Reynolds number is $Re = UD/\nu = 180$.
For this case, neither analytical nor experimental solutions are available, but the formation of von Karman vortex is expected behind the piers. The validation is performed computing the Strouhal number for the two piers, given by the following formula:

$$St = \frac{f_{lift}D}{U}$$

where f_{lift} is the lift frequency which usually corresponds to the vortex shedding frequency, D is the diameter of the cylinder and U is the average free-stream velocity. In order to compute the lift frequency, a FFT (Fast Fourier Transform) has been performed on the signal which describes the variation of the force with time. The force is computed as:

$$F = \int_0^l \int_0^h \rho\ gz\boldsymbol{n}\ dz\ ds$$

where ρ is the water density, g is the acceleration of gravity and $\boldsymbol{n}$ is the normal vector. The integral is performed on the cylinder with boundary l and along the vertical direction z.
To perform an appropriate analysis on several cycles, the simulation time is set to 1200 s.
Finally, in order to check the mass coservation of the advection schemes of TELEMAC-2D, a tracer is released at the inlet with the following boundary condition:

$$c(x = -13.5, y) = \begin{cases} 2\ g/l & \text{if} \quad H/2-9\ \leq \mathrm{y} \leq H/2-8 \\ 1\ g/l & \text{otherwise} \end{cases}$$

A free condition is imposed at the outlet. The error on the mass is computed as follows:

$$\varepsilon_M = M_{start} + M_{in} - M_{end}$$

$\mathrm{M_{start}} = \int_\Omega (hc)^n d\Omega$ is the mass at the beginning of the simulation, $\mathrm{M_{end}} = \int_\Omega (hc)^{n+1} d\Omega$ is the mass at the end of the simulation, $\mathrm{M_{in}} = \int_\Gamma hc\boldsymbol{un}d\Gamma$ is the mass introduced (and leaved) by the boundaries; where Ω is the computational domain and Γ is its boundary. The realtive error is computed as:

$$\varepsilon_{rel} = \frac{\varepsilon_M}{\max(|M_{start}|, |M_{in}|, |M_{end}|)}$$

9.2 Numerical parameters

The computational domain is made up by 4304 triangular elements and 2280 nodes and it is shown in Figure 9.3.

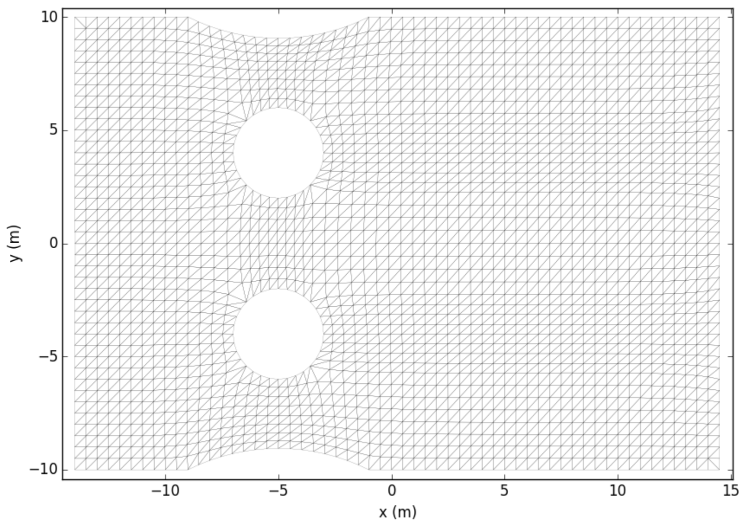


Figure 9.3: Mesh of the channel with two cylindrical piers.

Three different numerical configurations are tested:
CASE A

- Equations: Saint-Venant FE
- Type of element:
  - Linear P1 for velocities
  - Linear P1 for water depth
- Advection scheme for velocities: weak characteristics with 12 Gauss points
- Linear system: wave equation
- Time step: 0.8 s
- Solver: conjugate gradient with accuracy 10^{-5}

CASE B

- Equations: Saint-Venant FE
- Type of element:
  - Quadratic P2 for velocities
  - Linear P1 for water depth

- Advection scheme for velocities: strong characteristics
- Linear system: primitive equations
- Time step: 0.1 s
- Solver: GMRES with option 1 and accuracy 10^{-5}

CASE C

- Equations: Saint-Venant FV
- Finite volume scheme: kinetic order 2
- Time step: variable with CFL=0.9

For the tracer, all the advection schemes of TELEMAC-2D are tested in the configuration A. In the case of the predictor-corrector schemes, the number of corrections is set to 5; for the LIPS schemes, the number of sub-steps is equal to 10 and the accuracy for diffusion of tracers is set to 10^{-10}. It is important to note that the last parameter is used even if the keyword `DIFFUSION OF TRACERS` is set to NO. Indeed, when using the LIPS schemes, a linear system has to be solved and this parameter defines the accuracy of the solver.

9.3 Results

Table 9.1 contains the Strouhal number obtained for the different numerical configurations.

Table 9.1: Pildepon test case: Strouhal number for the upper and lower piers according to the different numerical configurations.

	Strouhal for upper pier	Strouhal for lower pier
CASE A	0.265	0.265
CASE B	0.347	0.347
CASE C	0.549	0.549

The results are reasonable for the various configurations.
Table 9.2 shows the mass balance at the end of the simulation according to the different advection schemes It can be noted that only the characteristics are not mass conservative.

Table 9.2: Pildepon test case: mass balance for the different advection schemes.

	M_{start}	M_{end}	M_{in}	ε_M	ε_{rel}
Strong Char.	1637.919	1774.378	946.0997	809.6411	0.4562958
N	1637.919	1787.794	149.8751	-0.8981260E-10	-0.5023654E-13
N PC1	1637.919	1780.652	142.7332	-0.1690887E-07	-0.9495886E-11
N PC2	1637.919	1780.529	142.6102	-0.1511967E-07	-0.8491671E-11
PSI	1637.919	1780.469	142.5495	0.6855931E-05	0.3850633E-08
PSI PC1	1637.919	1780.675	142.7558	-0.1027729E-09	-0.5771570E-13
PSI PC2	1637.919	1780.479	142.5600	-0.1458420E-07	-0.8191168E-11
PSI LIPS	1637.919	1780.409	142.4901	-0.1329022E-07	-0.7464699E-11
NERD	1637.919	1788.786	150.8668	-0.2728484E-11	-0.1525327E-14

10. tide

10.1 Purpose

This test demonstrates the availability of TELEMAC-2D to model the propagation of tide in a maritime domain by computing tidal boundary conditions.

10.2 Description

A coastal area located in the English Channel off the coast of Brittany (in France) close to the real location of the Paimpol-Bréhat tidal farm is modelled to simulate the tide and the tidal currents over this area. Time and space varying boundary conditions are prescribed over liquid boundaries.
Several databases of harmonic constants are interfaced with TELEMAC-2D:

- The JMJ database resulting from the LNH Atlantic coast TELEMAC model by Jean-Marc JANIN,
- The global TPXO database and its regional and local variants from the Oregon State University (OSU),
- The regional North-East Atlantic atlas (NEA) and the global atlas FES (e.g. FES2004 or FES2012...) coming from the works of Laboratoire d'Etudes en Géophysique et Océanographie Spatiales (LEGOS),
- The PREVIMER atlases.

In the tide test case, the JMJ database and the NEA prior atlas are used as examples. A TPXO-like example is also provided as an example but the user has to download the local solution available on the OSU website: http://volkov.oce.orst.edu/tides/region.html

10.2.1 Reference

10.2.2 Geometry and Mesh

Bathymetry

Real bathymetry of the area bought from the SHOM (French Navy Hydrographic and Oceanographic Service). ©Copyright 2007 SHOM. Produced with the permission of SHOM. Contract number 67/2007

Geometry

Almost a rectangle with the French coasts on one side 22 km $\times$ 24 km

Mesh

4,385 triangular elements
2,386 nodes

10.2.3 Physical parameters

Horizontal viscosity for velocity: 10^{-6} m^2/s
Coriolis: yes (constant coefficient over the domain = 1.10×10^{-4} rad/s)
No wind, no atmospheric pressure, no surge and nor waves

10.2.4 Initial and Boundary Conditions

Initial conditions

Constant elevation
No velocity

Boundary conditions

Elevation and horizontal velocity boundary conditions computed by TELEMAC-2D from an harmonic constants database (JMJ from LNH or NEA prior from LEGOS). If a tidal solution from OSU has been downloaded (e.g. TPXO, European Shelf), it can be used to compute elevation and horizontal velocity boundary conditions as well.

10.2.5 General parameters

Time step: 60 s
Simulation duration: 90,000 s = 25 h

10.2.6 Numerical parameters

Advection for velocities: Characteristics method
Thompson method with calculation of characteristics for open boundary conditions
Free Surface Gradient Compatibility = 0.5 (not 0.9) to prevent on wiggles
Tidal flats with correction of Free Surface by elements, treatments to have $h \geq 0$

10.2.7 Comments

If a tidal solution from OSU has been downloaded (e.g. TPXO, European Shelf), it can be used to compute initial conditions with the keyword `INITIAL CONDITIONS` set to TPXO SATELLITE ALTIMETRY. Thus, both initial water levels and horizontal components of velocity can be calculated and may vary in space.

10.3 Results

Tidal range, sea levels and tidal velocities are well reproduced compared to data coming from the SHOM or at sea measurements.

10.4 Conclusion

TELEMAC-2D is able to model tide in coastal areas.

11. weirs

11.1 Description of the problem

This test case presents a flow over three successive sills in a rectangular channel treated as singularities. It allows to show that TELEMAC-2D is able to treat a number of flow singularities as internal boundary conditions. Moreover, it allows also to check the tracers function for this type of problem in TELEMAC-2D.

The sills may be considered under the traditional approach made in open channel hydraulics through a relation between two superimposed open boundaries, short culverts may be treated as a couple of nodes with respective source and sink terms.
This sills description, which are represented in a channel as internal singularities, was introduced in TELEMAC-2D in order to avoid the multiplication of computational nodes and associated reduction in time step when a sill is represented thanks to variations in the bathymetry.
The weir law as traditionally used in channel hydraulics is prescribed through two boundary conditions: one upstream the weir and one downstream. It should be mentioned however that this option gives satisfactory results only if the flow is relatively perpendicular to the weir, which is the case in the present test.

In this test case, the sills are so represented as internal singularities, as just indicated. The geometry dimensions of rectangular channel are 848 m wide and 3,522 m long. The channel is flat bottom and it is decomposed of four part reaches 848 m long. The three upstream reaches are limited at their downstream end by 3 sills with crest heights 1.8 m (upstream sill), 1.6 m and 1.4 m (downstream sill).
Note that the turbulent viscosity is constant and equal to 1 $m^2 \cdot s^{-1}$.

11.2 Initial and boundary conditions

The initial conditions are a null velocity, a water depth of 1.35 m and a tracer value of 50.

The boundary conditions (Figure 11.1) are:

- At the channel entrance, the flow rate is $Q = 600\ m^3s^{-1}$ and the tracer value is 100.
- At the channel outlet, the water depth is $h = 1.35$ m.

- On bottom friction, the Strickler formula with friction coefficient equal to 30 $m^{1/3} \cdot s^{-1}$ is imposed.
- No friction is taken into account on lateral walls.

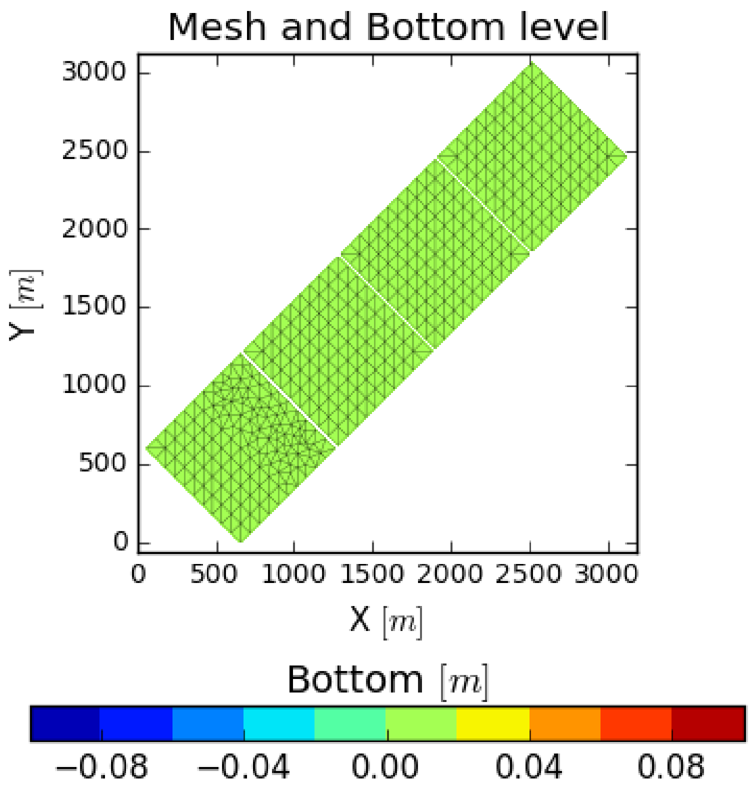


Figure 11.1: Mesh and topography of the domain.

11.3 Numerical parameters

The mesh is regular along the domain. It is generally made up with quadrangles split into two triangles. It is irregular in the upstream reach in order to test the sensitivity on this feature of the flow results above the sill (see Figure 11.1). It is composed of 870 triangular elements (519 nodes) and the size of triangles ranges between 53 m and 120 m. The triangular elements types are linear triangles (P1, 3 values per element, the corners) for water depth and quasi-bubble triangle (4 values per element, the corners and the element center) for velocities.

The time step is 150 s for a period of 6,000 s. The resolution accuracy for the velocity is taken at 10^{-10}.

Note that for numerical resolution, GMRES (Generalized Minimal Residual Method) is used for solving the propagation step (option 7). To solve advection, the characteristics scheme (scheme 1), and the conservative psi scheme (scheme 5) is used respectively for the velocities and for the depth. To finish, the implicitation coefficient for depth and velocities is equal at 0.55.
It should also be noted that a tracer is used. The solver for propagation is the conjugate gradient (option 1). For the advection resolution, it's used a conservative N-scheme(scheme 4) and the solver for diffusion of tracer is also the conjugate gradient (option 1).

11.4 Results

As it can be seen Figure 11.2, the velocity field remains regular in the different reaches of the channel.

The water level increases progressively as expected in the three upstream reaches during the simulated period (see Figure 11.3). The relations between the discharge (per unit of width) on the sill, the water levels upstream and downstream and the sill crest elevation must be respected. They are:

- Free overflow weir:
$$q = \mu\sqrt{2g}\,(z_{up} - z_{sill})^{3/2}$$
- drowned weir :
$$q = \frac{2}{3\sqrt{3}} C_d \sqrt{2g}\,(z_{down} - z_{sill})\sqrt{(z_{up} - z_{down})}$$

The transition from free overflow to drowned condition is defined by:

$$z_{down} \leq z_{sill} + \frac{2}{3}(z_{up} - z_{sill}) \tag{11.1}$$

Where z_{up}, z_{down} and z_{sill} are respectively water level upstream (m), water level downstream (m) and sill crest elevation (m). q is discharge per unit width ($\mathrm{m}^2 \cdot \mathrm{s}^{-1}$) and C_d is the discharge coefficient (usually between 0.4 and 0.5).
The TELEMAC-2D results respect well these relation (11.1) (Figure 11.3). Furthermore the tracer propagation is well carry out through internal singularities (Figure 11.4).

To conclude, TELEMAC-2D computes adequately weir flows as given by analytical hydraulic laws. This type of flow is represented as an internal singularity in the model.

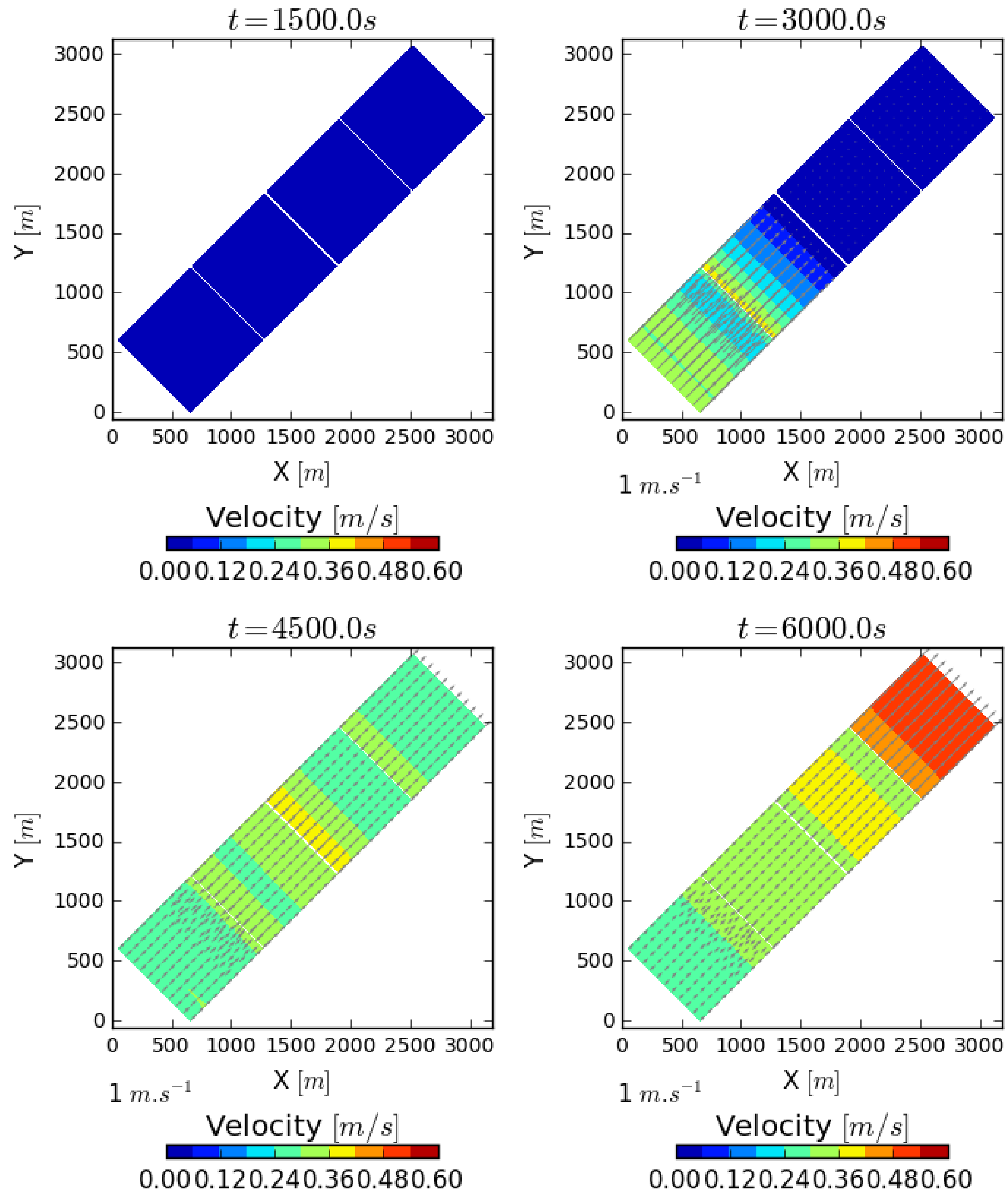


Figure 11.2: Evolution of velocity field in time.

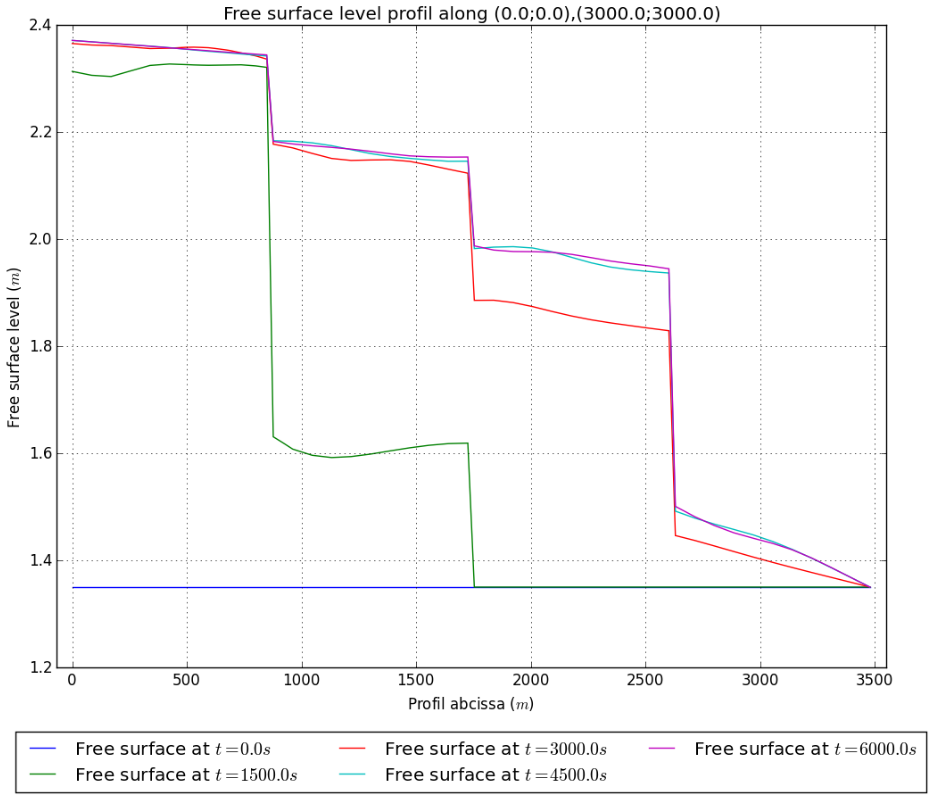


Figure 11.3: Evolution of the free surface elevation in time.

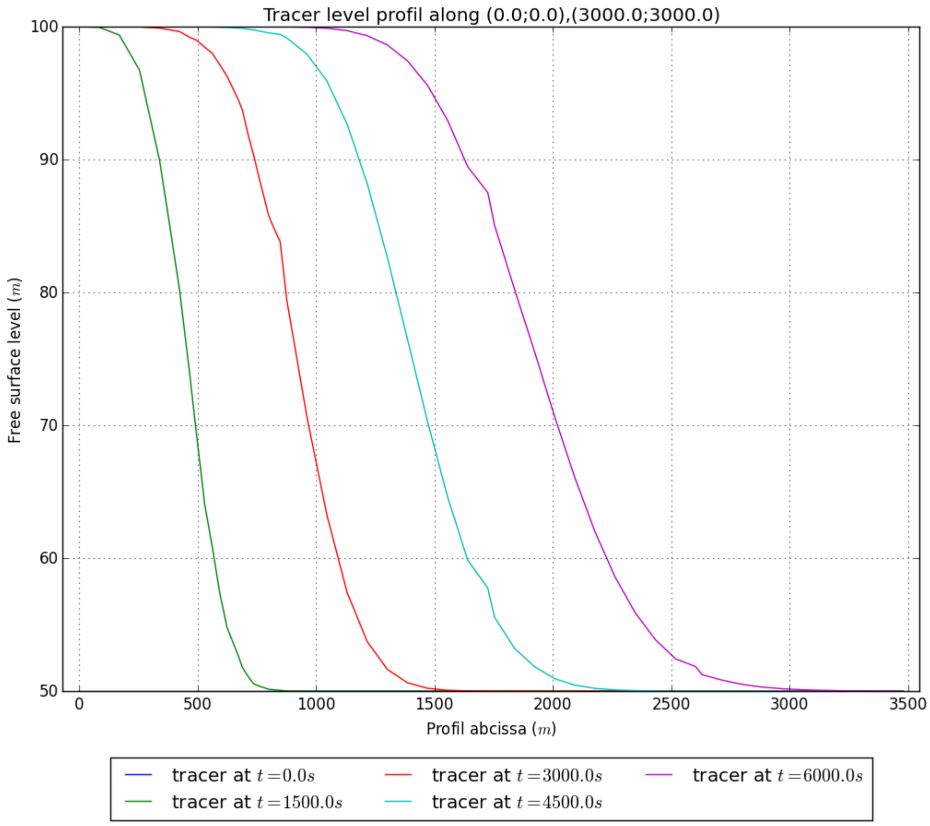


Figure 11.4: Evolution of the tracer in time.

12. wind

12.1 Description of the problem

This test case presents the hydrodynamics study resulting from wind set-up in a closed rectangular basin or channel. It allows to show that TELEMAC-2D is able to correctly simulate the effect of meteorological conditions such as surface layer motion generated by the wind blowing at the water surface provided a depth integration of this process is adequate. This test case allows also to demonstrate that TELEMAC-2D produces the expected one-dimensional solution even though the grid of triangles is irregular (various sizes and orientations of triangles).

This test case models the hydrodynamics behaviours due to wind blowing in a closed rectangular basin. The geometry dimensions of basin are 100 m wide and 500 m long. The basin has a flat bottom and the water depth is equal to 2 m depth. The wind blowing on whole basin produces a surface current in the direction of the wind and a bottom current in the opposite direction. The total discharge in each cross-section is null and the wind shear stress is balanced by the slope of the induced free surface. The solution produced by TELEMAC-2D is compared with the analytical solution to this problem.
Note that the turbulent viscosity is constant with velocity diffusivity equal to 0 $\mathrm{m}^2 \cdot \mathrm{s}^{-1}$.

12.2 Initial and boundary conditions

The initial water depth is 2 m with null velocity.

The boundary conditions (Figure 11.1) are:

- For the solid walls, a slip condition in the basin is used for the velocity.
- No bottom friction.
- The wind stress at the surface is imposed.

The wind conditions allowing to compute the wind stress are :

- The wind velocity is equal to 5 $\mathrm{m} \cdot \mathrm{s}^{-1}$ (West wind; the wind is coming from the West)

- The coefficient of wind influence is $a_{wind}\frac{\rho_{air}}{\rho_{water}} = 1.2615 \cdot 10^{-3}$ with ρ_{air}, ρ_{water} which are respectively the air density and the water density and with a_{wind} an addimentional coefficient.

Note that a pre-computation is carried out with the precedent conditions until 500 s ("ini_wind.slf"). For the pre-computation, the wind is applied progressively during the 4 first time step (40 s). The computation is after continued until 10 time steps. The results are so observed after 60 time step (600 s).

12.3 Mesh and numerical parameters

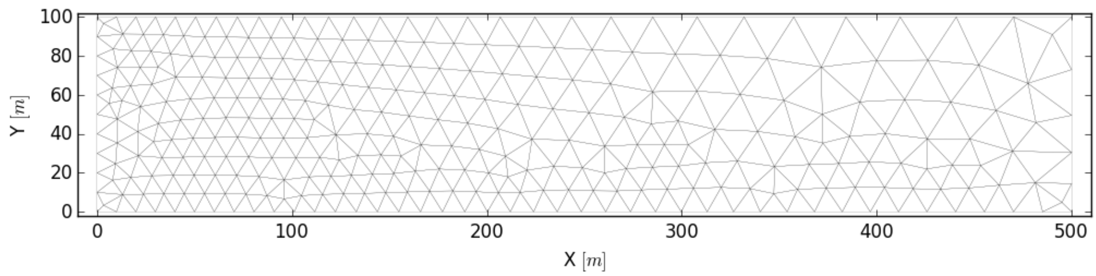


Figure 12.1: Mesh.

The mesh is irregular in the basin. The mesh is shown on Figure 12.1. It is composed of 551 triangular elements (319 nodes) and the size of triangles ranges between 14 m and 24 m. The triangular elements types are linear triangles (P1, 3 values per element) for water depth and for velocities.

The time step is 10 s for a period of 100 s which is an added at initial computation of 500 s. The simulation duration is then 600 s. The resolution accuracy for the velocity is taken at 10^{-3}.

Note that for numerical resolution, conjugate gradient on a normal equation is used for solving the propagation step (option 3). To solve advection, the characteristics scheme (scheme 1), and the conservative psi scheme (scheme 5) is used respectively for the velocities and for the depth. To finish, the implicitation coefficient for depth and velocities is equal at 0.5.

12.4 Results

When the wind blows on closed basin (Figure 12.2), a balance between the wind stress and the surface slope occurs. The analytical solution for this problem is given by the equation:

$$H(x) = \sqrt{H_0^2 + a_{wind}\frac{2\rho_{air}}{g\rho_{eau}}||\boldsymbol{Wind}||^2 x} \tag{12.1}$$

where $||\boldsymbol{Wind}||^2$ is norm of wind velocity vector. H_0 is depth water on the wind entrance side (west side in this case) and H_L is depth water at the end basin within L distance (east side in this case). In addition, H_0 is the solution of the following equation:

$$F(x) = \left(a_{wind}\frac{2\rho_{air}}{g\rho_{eau}}||\boldsymbol{Wind}||^2 L\right)^{3/2} - x^3 - a_{wind}\frac{3\rho_{air}}{g\rho_{eau}}||\boldsymbol{Wind}||^2 L \cdot x \cdot H_{initial} = 0 \tag{12.2}$$

With $H_{initial}$ is the initial water depth equal here to 2 m. In this test case, the analytic values of water depths in basin are $H_0 \approx 1.56431$ m and consequently $H_L \approx 2.37947$ m.

The obtained TELEMAC-2D solution reproduces well the behaviour of a balance between the wind stress and the surface slope occurs. When you consider the one-dimensional solution (independent of the y-axis, Figure 12.3) as shown on Figure 12.4, the water surface elevation difference between the two extremities of this 500 m long basin is $81.5116 \cdot 10^{-2}$ m whereas it should be $81.5164 \cdot 10^{-2}$ m according to the analytical solution, i.e. an error of 0.006%.

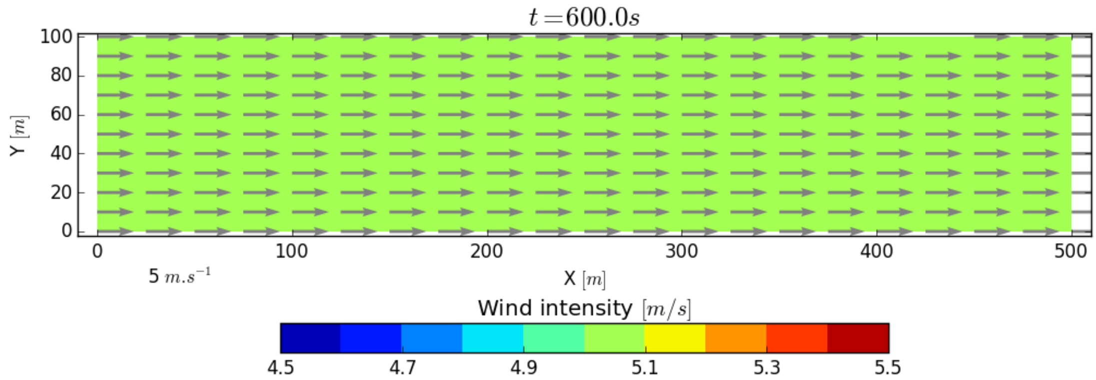


Figure 12.2: Wind velocity vector.

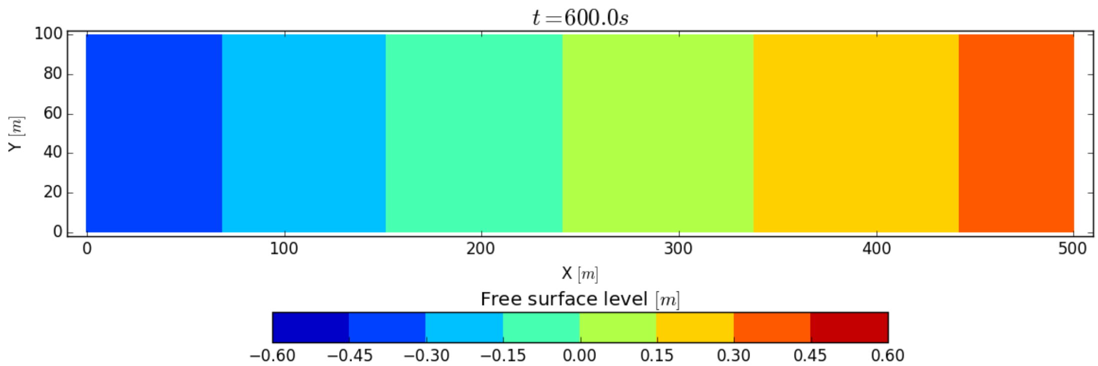


Figure 12.3: Free surface along basin.

TELEMAC-2D is able to compute wind generated flows on the basis of the empirical wind shear stress formulation (presented in the User's Manual). The solution computed by TELEMAC-2D in this one-dimensional test case is well independent of the computational grid characteristics although the grid meshes are very irregular.

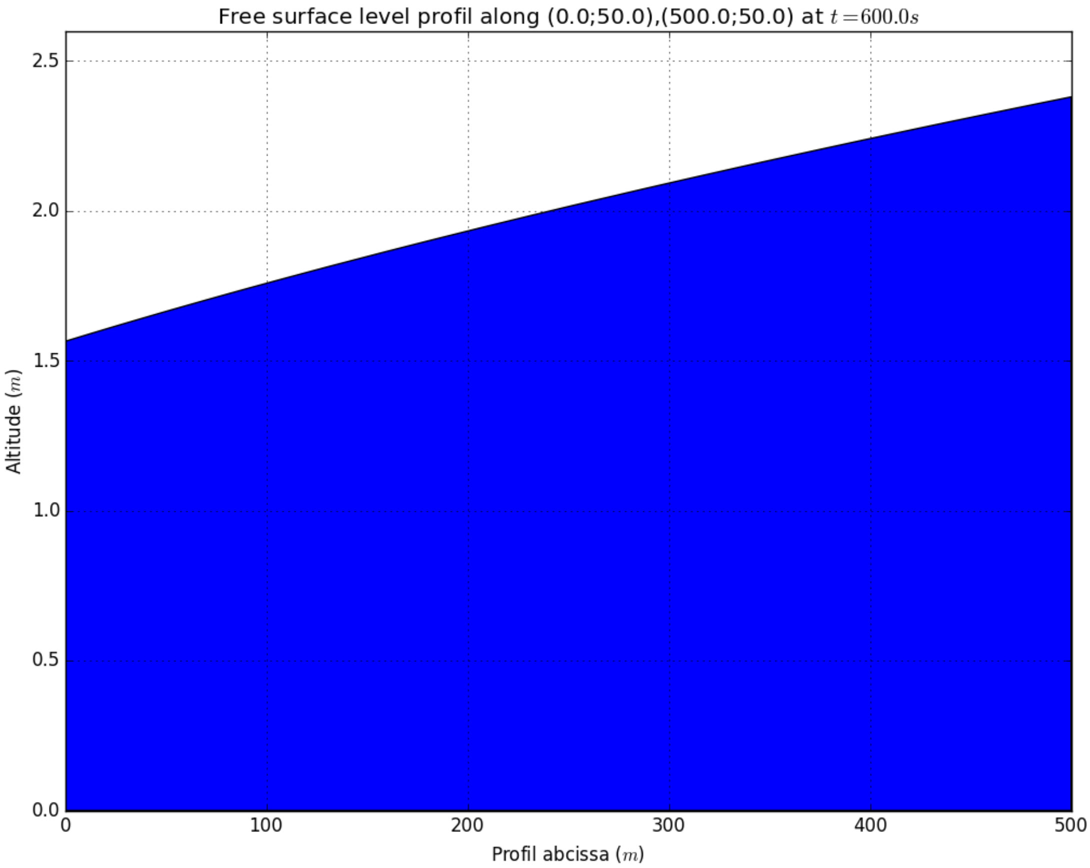


Figure 12.4: One-dimensional free surface.

[1] Sampath Kumar Gurram, Karam S. Karki, and Willi H. Hager. Subcritical junction flow. *Journal of Hydraulic Engineering*, 123(5):447–455, may 1997.