Telemac2d

Validation Manual

Otto Mattic

Version 7.2
December 29, 2016

Contents

1. Cone

1.1 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.

1.2 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.

1.3 Results

The error norms, the maximum and the minimum value of the tracer function at the end of the simulation are reported in Table 1.1. The final profiles for every scheme are plotted in Figures 1.1 and 1.2. 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 1.1 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 1.1 and in Table 1.1). 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 1.1: 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	2.866e-03	1.479e-02	2.484e-01	0.0000	0.753
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	3.371e-03	1.757e-02	2.791e-01	0.0000	0.721
PSI PC2	4.698e-03	2.391e-02	3.573e-01	0.0000	0.643
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

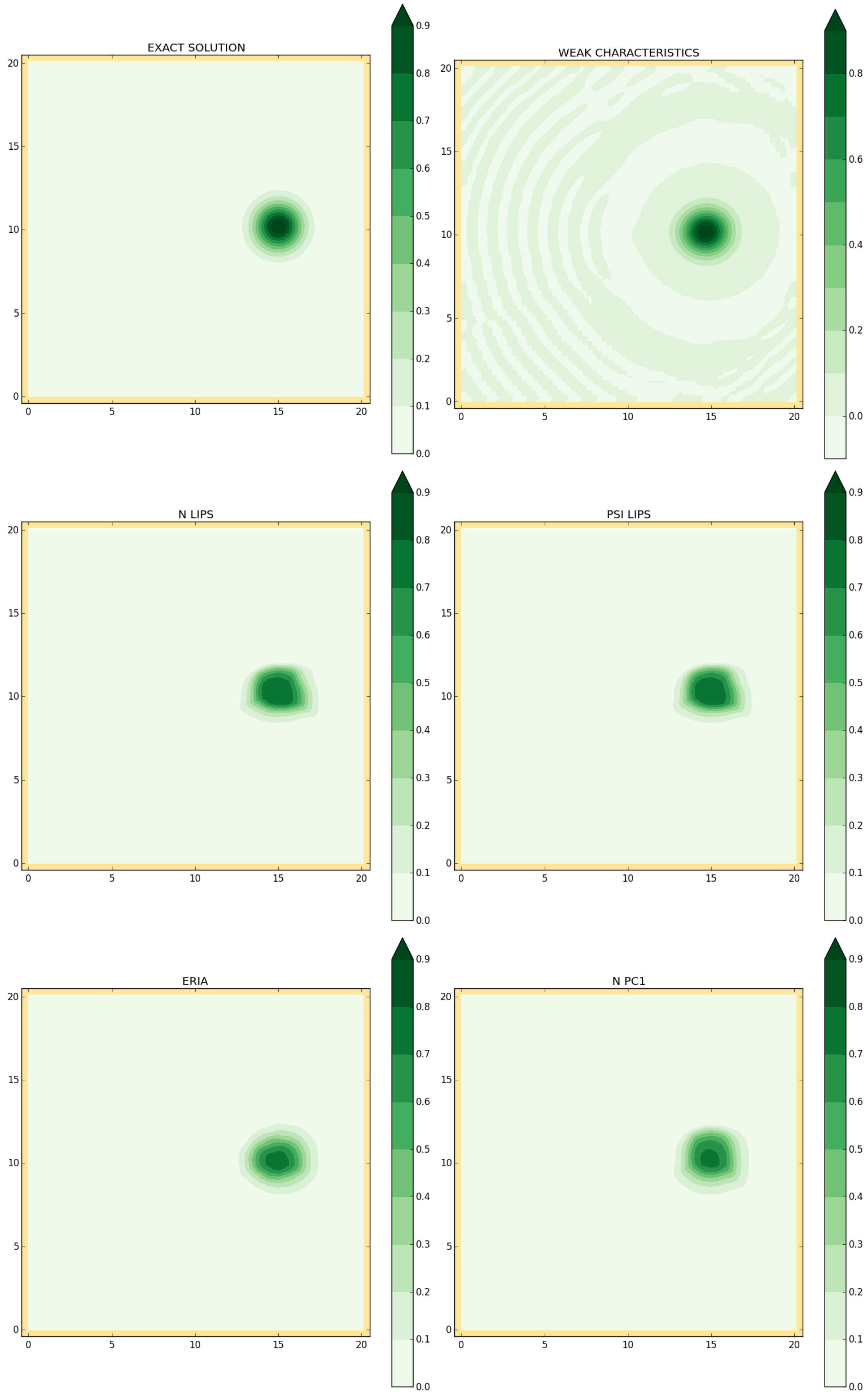


Figure 1.1: Cone test: contour lines of gaussian tracer functions after one period of rotation, for the advection schemes of TELEMAC-2D.

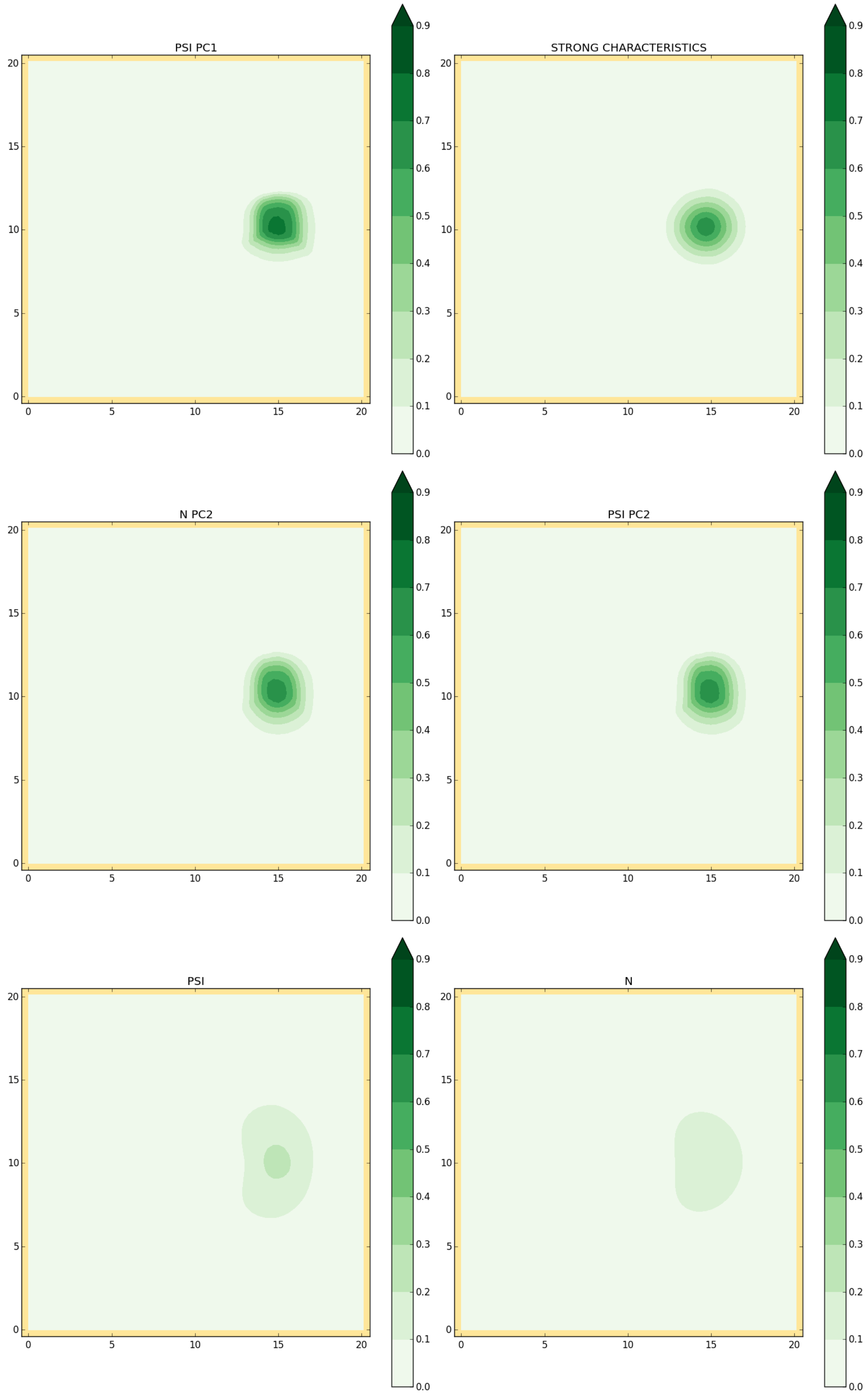


Figure 1.2: Cone test: contour lines of gaussian tracer functions after one period of rotation, for the advection schemes of TELEMAC-2D.

2. confluence

2.1 Purpose

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

2.2 Description of the problem

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

2.3 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.

2.4 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 2.1

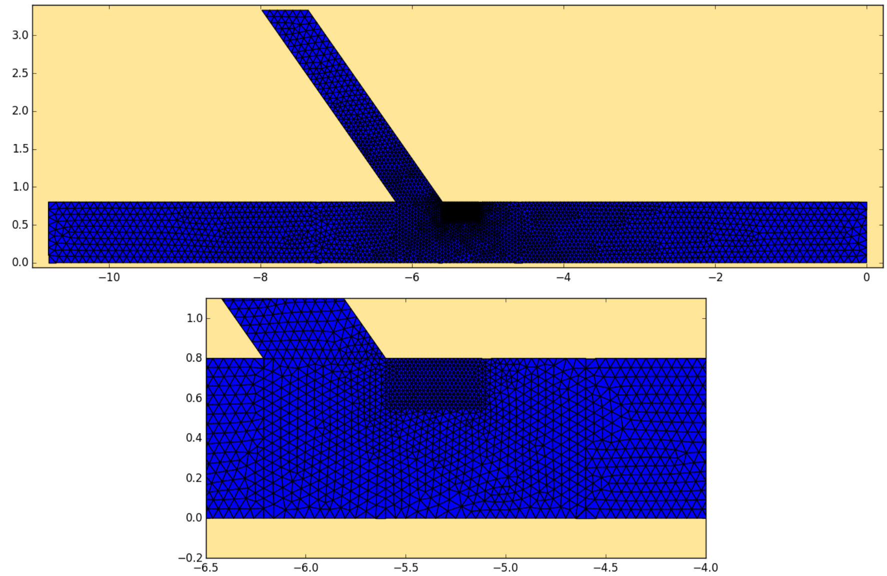


Figure 2.1: Mesh

2.5 Initial and Boundary Conditions

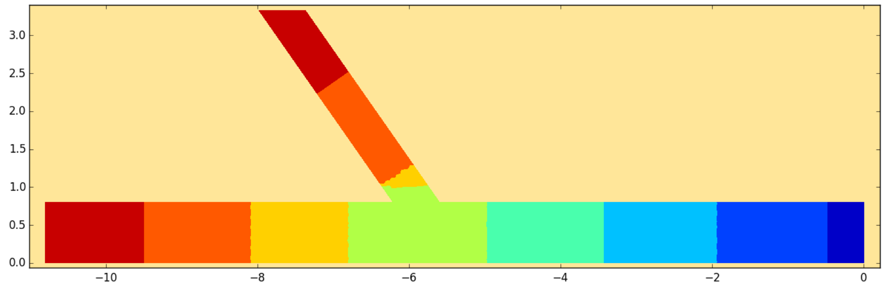
Boundaries:

- 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

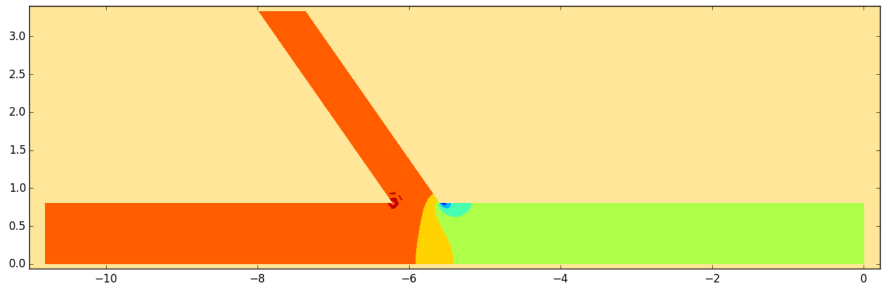
Bottom:

- Strickler formula with friction $coefficient = 62$

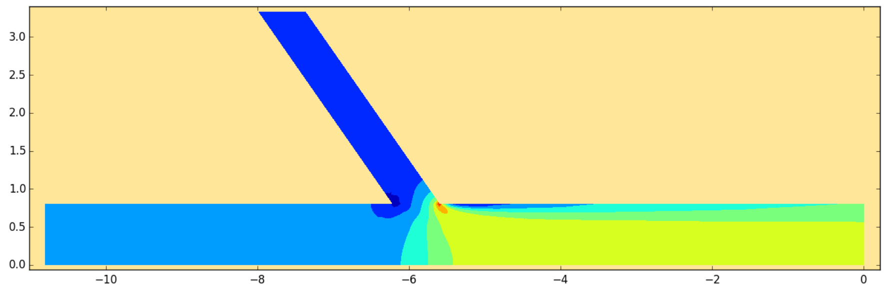
The mesh is shown on Figure 2.1 and the topography on Figure 2.2.



(a) bottom level



(b) free surface



(c) velocity

Figure 2.2: Results

Turbulence:

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

2.6 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

2.7 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 2.2).
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. [4]).

2.8 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”).

2.9 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.
```

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

3.1 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.

3.2 Description of the problem

3.2.1 Domain

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

3.2.2 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 3.1).

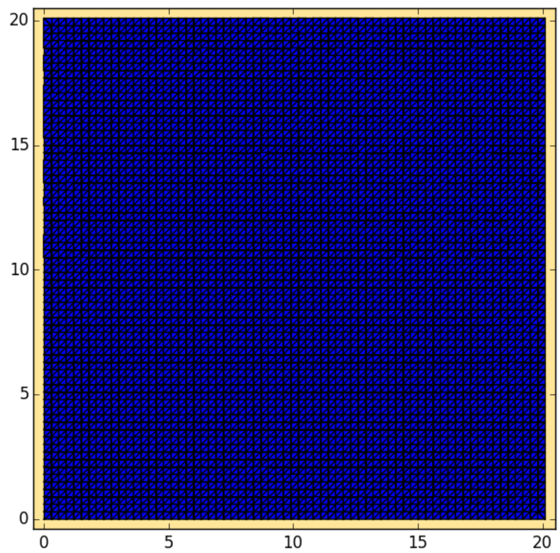


Figure 3.1: Gouttedo case: used mesh

3.2.3 Initial conditions

The fluid is initially at rest with a Gaussian free surface in the centre of a square domain (see Figure 3.2). Water depth is given by $H = 2.4 \exp \frac{-[(x-10)+(y-10)]}{4}$

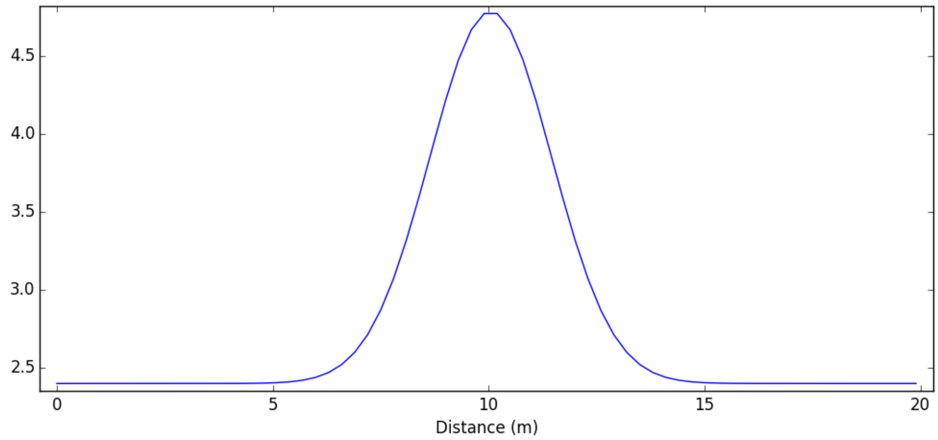


Figure 3.2: Gouttedo case: initial elevation

3.2.4 boundary conditions

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

3.2.5 Physical parameters

The physical parameters used for this case are the following:

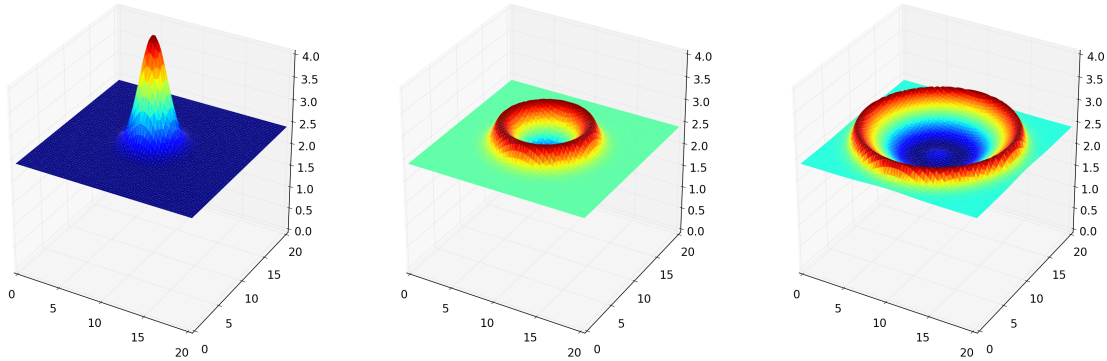
1. Friction: Strickler formula with K = 40
2. Turbulence: Constant viscosity equal to zero (or disactivation of diffusion step using the keyword *DIFFUSION OF VELOCITY = NO*)

3.2.6 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.

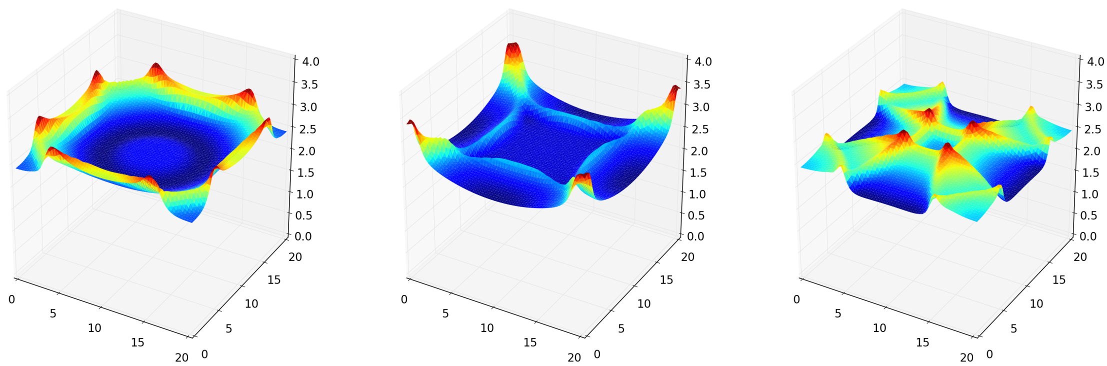
3.3 Results

The wave spreads circularly around the initial water surface peak elevation (Figures 3.3). 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 3.4).



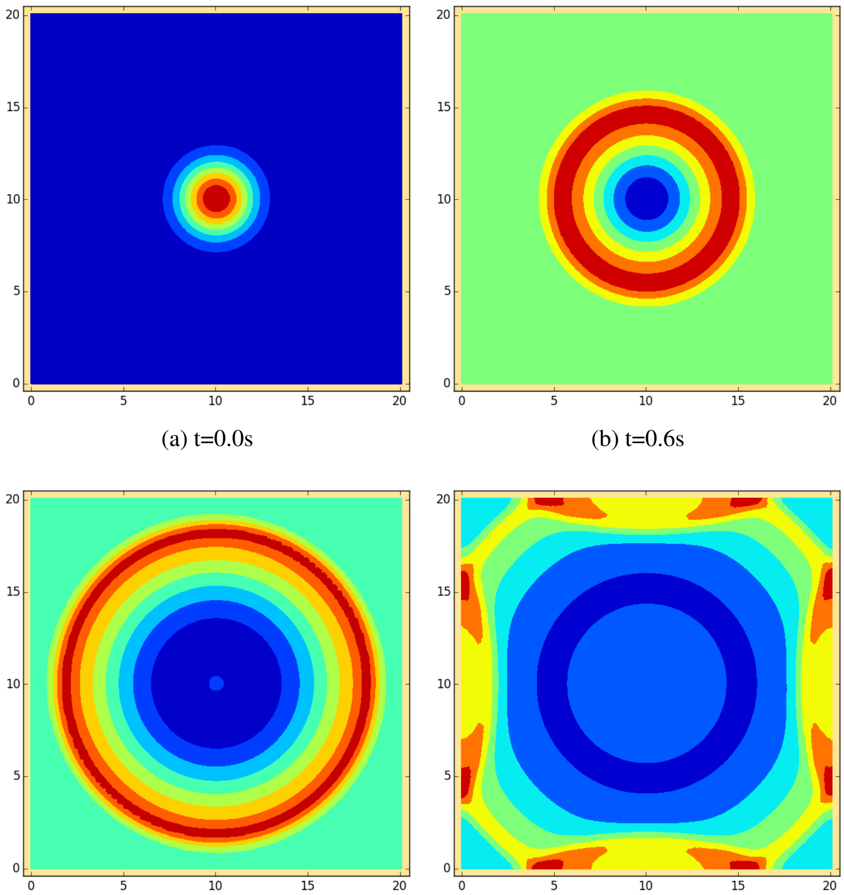
(a) t=0.0s (b) t=0.6s (c) t=1.2s

Figure 3.3: Gaussion water hill: Initial conditions and circular spreading



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

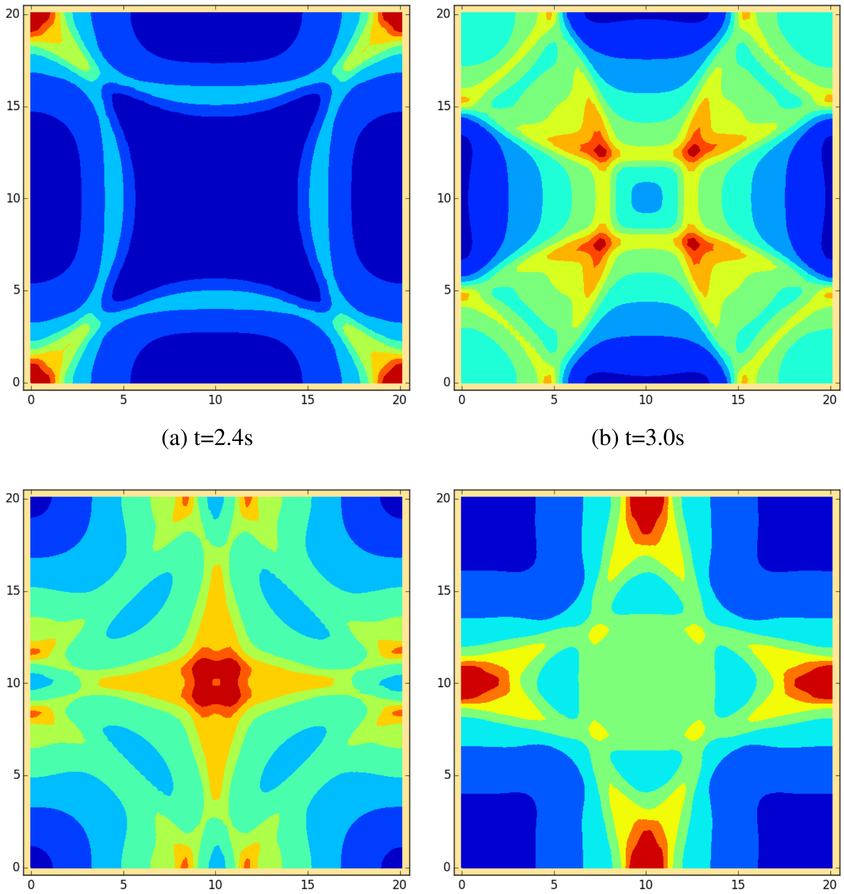
Figure 3.4: Gaussion water hill : reflexions



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

(c) t=1.2s (d) t=1.8s

Figure 3.5: 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 3.6: Evolution of the water depth from 2.4 sec. to 4.0 sec

3.4 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.

3.5 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.

3.6 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
```

4. malpasset

4.1 Purpose

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.

4.2 Description

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]. 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 "charac") has been kept. Nevertheless, the recommended advection scheme for velocities for such applications is now the NERD scheme (14). A simulation using a large mesh (named "large") is also performed.

4.2.1 Reference

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

4.2.2 Geometry and Mesh

Bathymetry

Real topography

Geometry

Size of the model domain $\approx$ 17 km $\times$ 9 km

Mesh

The mesh is refined in the river valley (downstream from the dam) and on the banks.
Regular mesh: 26,000 triangular elements / 13,541 nodes. Maximum size range is from 17 to 313 m
Large mesh: 104,000 triangular elements / 53,081 nodes. Maximum size range is from 8.5 to 156.5 m

4.2.3 Physical parameters

Constant viscosity equal to 1 m^2/s on horizontal directions
Coriolis: no
Wind: no

4.2.4 Initial and Boundary Conditions

Initial conditions

Full reservoir at initial time
No water in the downstream valley
No velocity

Boundary conditions

Channel banks: solid boundary without roughness (slip conditions)
Bottom: solid boundary with roughness. Strickler formula with friction coefficient = 30 $m^{1/3}/s$
Solid boundary everywhere

4.2.5 General parameters

Time step: 4 s for regular mesh cases (except the 1st Order Kinetic scheme with 1 s and the primitive equations with 0.5 s) and 1 s for large mesh case
Simulation duration: 4,000 s

4.2.6 Numerical parameters

Advection of velocities: NERD scheme (14) for tidal flats with the treatment of negative depths (regular + large meshes: "pos" and "large"), but also just for regular mesh: the new ERIA scheme ("ERIA", number 15), the historical method of characteristics ("charac"), the 1st Order Kinetic scheme ("cin") and the coupled primitive equations ("prim").

4.2.7 Comments

4.3 Results

Figure ??? illustrates the progression of the flood wave after the dam break (simulation using the treatment of negative depths that smoothes 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 when arriving to the sea. During the simulation, no negative water depths are observed.

4.4 Conclusion

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

5. Pildepon

5.1 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 5.1.

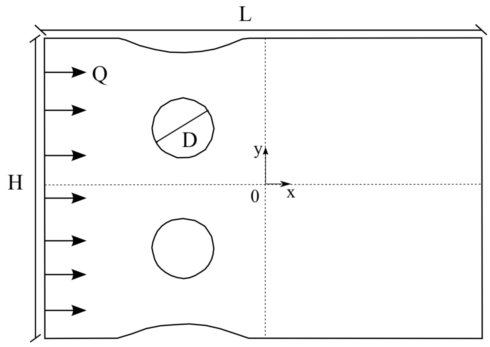


Figure 5.1: Geometry of the pildepon test case.

The section is trapezoidal (see the bottom in the Figure 5.2) 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 $k_s = 40\ \mathrm{m}^{1/3}\mathrm{s}^{-1}$.

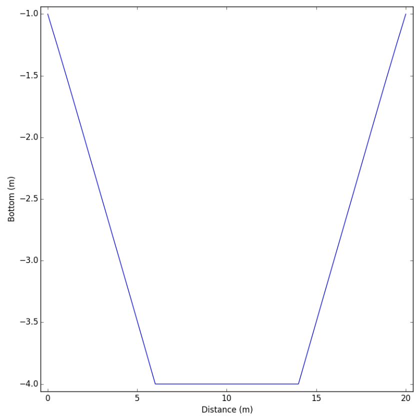


Figure 5.2: Topography at the inlet of the channel.

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:

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

5.2 Numerical parameters

The computational domain is made up by 4304 triangular elements and 2280 nodes and it is shown in Figure 5.3.

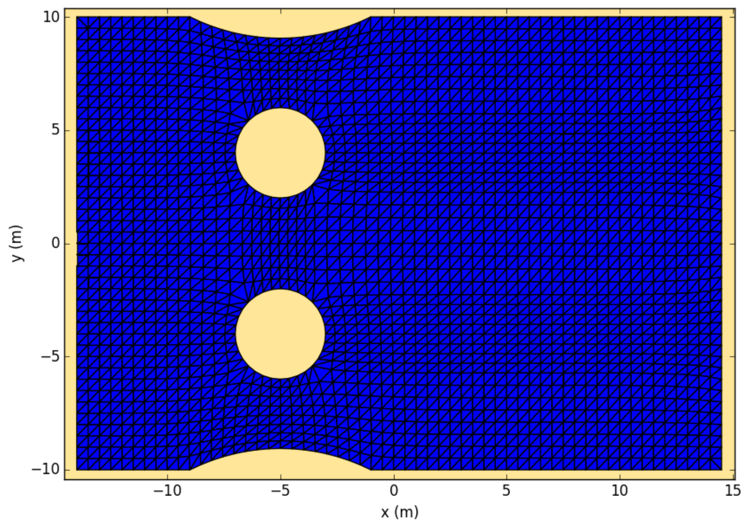


Figure 5.3: Mesh of the channel with two cylindrical piers.

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.

5.3 Results

Table 5.1 contains the Strouhal number obtained for the different numerical configurations.

Table 5.1: 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 5.2 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 5.2: 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
Weak Char.	1637.919	-0.2111681E+25	-0.2386819E+25	-0.2751374E+24	-0.1152737
N	1637.919	1787.794	149.8751	-0.8981260E-10	-0.5023654E-13
N PC1	1637.919	1780.652	142.7332	-0.1690933E-07	-0.9496142E-11
N PC2	1637.919	1780.529	142.6102	-0.1511921E-07	-0.8491415E-11
N LIPS	1637.919	1780.469	142.5495	0.6848631E-05	0.3846533E-08
PSI	1637.919	1780.675	142.7558	-0.1027729E-09	-0.5771570E-13
PSI PC1	1637.919	1780.043	142.1243	-0.1719627E-07	-0.9660591E-11
PSI PC2	1637.919	1780.004	142.0847	-0.1630201E-07	-0.9158414E-11
PSI LIPS	1637.919	1780.469	142.5495	0.6848631E-05	0.3846533E-08
NERD	1637.919	1788.786	150.8668	-0.2728484E-11	-0.1525327E-14

6. tide

6.1 Purpose

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

6.2 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

6.2.1 Reference

6.2.2 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

6.2.3 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

6.2.4 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.

6.2.5 General parameters

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

6.2.6 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 $h \geq 0$

6.2.7 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.

6.3 Results

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

6.4 Conclusion

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

Bibliography

[1] JOLY A., GOEURY C., and HERVOUET J.-M. Adding a particle transport module to telemac-2d with applications to algae blooms and oil spills. Technical Report H-P74-2013-02317-EN, EDF R&D-LNHE, 2013.

[2] AUTHOR. Title. *Journal de Mickey*, 666.

[3] PHAM C.-T., BOURBAN S., DURAND N., and TURNBULL M. Méthodologie pour la simulation de la marée avec la version 6.2 de telemac-2d et telemac-3d. Technical Report H-P74-2012-02534-FR, EDF R&D-LNHE, 2012.

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

[5] TSANIS I. Simulation of wind-induced water currents. *Journal of hydraulic Engineering*, 115(8):1113–1134, 1989.

[6] SMAGORINSKY J. General simulation experiments with the primitive equations. *Monthly Weather Review*, 91(3):99–164, March 1963.

[7] HERVOUET J.-M. *Méthodes itératives pour la solution des systèmes matriciels.* Rapport EDF HE43/93.049/A, 1996.

[8] HERVOUET J.-M. *Hydrodynamics of Free Surface Flows. Modelling with the finite element method.* Wiley, 2007.

[9] HERVOUET J.-M. Guide to programming in the telemac system version 6.0. Technical Report H-P74-2009-00801-EN, EDF R&D-LNHE, 2009.

[10] JANIN J.-M., HERVOUET J.-M., and MOULIN C. *A positive conservative scheme for scalar advection using the M.U.R.D technique in 3D free-surface flow problems.* XIth International Conference on Computional methods in water resources, 1996.

[11] GAUTHIER M. and QUETIN B. Modèles mathématiques de calcul des écoulements induits par le vent. In *17e congrès de l'AIRH*, Baden-Baden, August 1977.

[12] METCALF M. and REID J. *Fortran 90 explained.* Oxford Science Publications, 1990.