



# Adding boundary conditions on the bed for Telemac-3D simulations

Antoine Joly, Agnès Leroy

## ► To cite this version:

Antoine Joly, Agnès Leroy. Adding boundary conditions on the bed for Telemac-3D simulations. Telemac User Clup 2015, Oct 2015, Warrington, United Kingdom. hal-01702802

**HAL Id: hal-01702802**

**<https://hal.science/hal-01702802>**

Submitted on 7 Feb 2018

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

# Adding boundary conditions on the bed for Telemac-3D simulations

Antoine Joly & Agnès Leroy  
EDF R&D LNHE / LHSV  
6 quai Watier, 78401 Chatou, France  
Email: antoine.joly@edf.fr

**Abstract**—Telemac-3D has been developed to work on a two dimensional mesh that has been extruded along the z-axis. On-going work is done by the development team to move away from this structure. The work presented here, and available in version 7.1 of Telemac-3D, is the first step of this process as it will extend the available external boundary conditions to the bed. The impact of imposing a flow rate on the bed on the equations solved by Telemac-3D and on the hypotheses chosen will be developed. The use of these new boundary conditions in a Telemac-3D simulation will then be explained. These boundary conditions will then be illustrated through a simple test case through a comparison with source terms, which was the only available method in the previous versions of Telemac-3D to model inflow near the bed.

## I. INTRODUCTION

Historically, Telemac-3D has been developed to work on a two dimensional mesh that has been extruded along the z-axis. Work is ongoing within the Telemac-Mascaret development team to move away from this concept. The first step chosen was to work on the boundary conditions, with a focus on liquid boundary conditions on the bed. Aside from the structural modifications in the code, this has affected the hypothesis used to solve the Navier-Stokes equations.

Therefore, a brief description of the modified equations will be given, followed by an explanation of how to use this new functionality of Telemac-3D. To finish a simple validation case will be given illustrating the advantages of these new boundary conditions.

## II. REMINDERS OF THE EQUATIONS SOLVED IN TELEMAC-3D

The time-discretisation, from time  $t^n$  to  $t^{n+1} = t^n + \Delta t$ , of the Navier-Stokes equations is done following the sequence below:

$$\frac{\tilde{\mathbf{U}}^{n+1} - \mathbf{U}^n}{\Delta t} = -\mathbf{U}^n \cdot \mathbf{grad} \mathbf{U}^n + \mathbf{F}^{visc^{n+1}} + \tilde{\mathbf{F}}^{n+1} \quad (1a)$$

$$\frac{\eta^{n+1} - \eta^n}{\Delta t} = -\text{div}_{2D} \left( \int_b^\eta \tilde{\mathbf{U}}_{2D}^n dz \right) \quad (1b)$$

$$\text{div} \left( \frac{1}{\rho} \mathbf{grad} p_d^{n+1} \right) = \frac{1}{\Delta t} \text{div} \tilde{\mathbf{U}}^{n+1} \quad (1c)$$

$$\frac{\mathbf{U}^{n+1} - \tilde{\mathbf{U}}^{n+1}}{\Delta t} = -\frac{1}{\rho} \mathbf{grad} p_d^{n+1} \quad (1d)$$

Where:

- $\mathbf{U}$  is the three-dimensional velocity vector
- $\mathbf{U}_{2D}$  is the horizontal velocity vector
- $W$  is the vertical velocity vector
- $\eta$  is the free surface
- $\mathbf{F}^{visc}$  are the viscous forces, i.e.:

$$\mathbf{F}^{visc} = \frac{1}{\rho} \text{div} (\mu [\mathbf{grad} \mathbf{U} + (\mathbf{grad} \mathbf{U})^T]) \quad (2)$$

- $\tilde{\mathbf{F}}$  are external forcing terms, including buoyancy effects and hydrostatic pressure terms
- $p_d$  is the dynamic pressure and  $p_h$  is the hydrostatic pressure, i.e. total pressure  $p = p_d + p_h$

To solve for the fluid velocities, the momentum equation is usually separated into different stages (depending on the scheme used), which is the viscous forces are written under the symbol  $\mathbf{F}^{visc}$ .

## III. CONSIDERING FLUX ON THE BED

Considering flow on the bed will affect how the continuity equation can be used to solve for the free surface. Indeed to solve for equation 1b, one starts by integrating the continuity equation from the bed to the free-surface:

$$\int_b^\eta \text{div} \mathbf{U} dz = 0 \quad (3)$$

If we develop equation (3) to separate the horizontal and vertical directions, we get:

$$\begin{aligned} \int_b^\eta \text{div} \mathbf{U} dz &= \int_b^\eta \text{div}_{2D} \mathbf{U}_{2D} dz + \int_b^\eta \frac{\partial W}{\partial z} dz \\ &= \int_b^\eta \text{div}_{2D} \mathbf{U}_{2D} dz \\ &\quad + W(z = \eta) - W(z = b) \end{aligned} \quad (4)$$

Where  $z = \eta$  is the plane at the free surface, and  $z = b$  is the plane at the bed.

Leibniz's theorem then gives:

$$\int_b^\eta \text{div}_{2D} U_{2D} dz = \text{div}_{2D} \left( \int_b^\eta U_{2D} dz \right) - U_{2D}|_{z=\eta} \cdot \mathbf{grad}_{2D} \eta + U_{2D}|_{z=b} \cdot \mathbf{grad}_{2D} b \quad (5)$$

Using equation (5) to rewrite equation (4) we get:

$$\begin{aligned} & \text{div}_{2D} \left( \int_b^\eta U_{2D} dz \right) \\ & - U_{2D}|_{z=\eta} \cdot \mathbf{grad}_{2D} \eta \\ & + U_{2D}|_{z=b} \cdot \mathbf{grad}_{2D} b \\ & + W(z=\eta) - W(z=b) = 0 \end{aligned} \quad (6)$$

The kinematic condition on the free-surface states that [1]:

$$\frac{d}{dt}(z - \eta) = 0 \quad \text{on } \Gamma_\eta \quad (7)$$

Where  $\Gamma_\eta$  is the free surface boundary. This boundary condition can be rewritten as:

$$W(z=\eta) - \frac{\partial \eta}{\partial t} - U|_{\Gamma_\eta} \cdot \mathbf{grad} \eta = 0 \quad \text{on } \Gamma_\eta \quad (8)$$

Since the free surface ( $z = \eta$ ) is only dependent on  $x$  and  $y$ ,  $\mathbf{grad} \eta = \mathbf{grad}_{2D} \eta$ . Therefore, using (8), equation (6) can be rewritten as:

$$\frac{\partial \eta}{\partial t} + \text{div}_{2D} \left( \int_b^\eta U_{2D} dz \right) = F_b \quad (9)$$

Where the right-hand side of equation (9) is defined as the conditions on the bed boundary:

$$F_b = W(z=b) - U_{2D}|_{z=b} \cdot \mathbf{grad}_{2D} b \quad (10)$$

If the bed boundary is fixed, and because the bottom plane ( $z = b$ ) is only dependent on  $x$  and  $y$  then:

$$F_b = - U|_{z=b} \cdot \mathbf{n}_b \quad (11)$$

Where  $\mathbf{n}_b$  is the normal of the bed boundary pointing away from the domain.

#### IV. SOLVING FOR THE BED FLUX TERM IN FINITE ELEMENTS

In a finite element framework, let us define the domain as  $\Omega$ , its boundary as  $\Gamma$ , and  $\Psi$  is the basis function.

The term describing the bed fluxes  $F_b$  will also appear when the divergence of the velocity is necessary:

$$\begin{aligned} \int_\Omega \text{div}(U\Psi) d\Omega &= \int_\Omega \text{div}(U)\Psi d\Omega + \int_\Omega U \cdot \mathbf{grad}(\Psi) d\Omega \\ &= \int_\Omega U \cdot \mathbf{grad}(\Psi) d\Omega - \int_\Gamma U \cdot \mathbf{n} \Psi d\Gamma \end{aligned} \quad (12)$$

Where the last part of equation 12 found using Gauss' divergence theorem. Therefore, when using finite elements, the bed flux term  $F_b$  will be written as:

$$F_b = - \int_\Gamma U \cdot \mathbf{n} \Psi d\Gamma \quad (13)$$

In telemac-3D bed fluxes will therefore be added when solving free surface (equation 1b), also when solving the Poisson equation for the dynamic pressure (1d).

When solving for the momentum equation (equations 1a and 1c),  $F_b$  can be thought of as an additional source term. This become clear from the fact that:

$$\begin{aligned} U \cdot \mathbf{grad}(U) &= \text{div}(UU) - U \text{div}(U) \\ &= \text{div}(UU) \end{aligned} \quad (14)$$

Which can be solved in a finite element notation in the same way as equation 12:

$$\begin{aligned} \int_\Omega \text{div}(UU\Psi) d\Omega &= \int_\Omega \text{div}(UU)\Psi d\Omega + \int_\Omega UU \cdot \mathbf{grad}(\Psi) d\Omega \\ &= U \left[ \int_\Omega U \cdot \mathbf{grad}(\Psi) d\Omega + F_b \right] \end{aligned} \quad (15)$$

This last part will be solved differently according to the advection scheme chosen.

#### V. IMPOSING A FLUX ON THE BED IN TELEMAC-3D

To take into account the velocity on the bed, a choice has been made to create additional keywords which will allow the user to define a flow rate per boundary on the bed. This is done by adding the following keywords to the steering file:

```

/-----/
/          OPTION FOR BED FLUXES          /
/-----/
OPEN BOUNDARY CONDITIONS ON THE BED = YES
PRESCRIBED FLOWRATES ON THE BED = <Enter Flowrates>

```

The values following the keyword `PRESCRIBED FLOWRATES ON THE BED` follows the same structure as for other prescribed flowrates in a Telemac-Mascaret system code, it should be a list of numbers separated by a semi-colon. One number per liquid boundary on the bed must be given.

Furthermore, as the `BOUNDARY CONDITIONS FILE` only deals with horizontal boundaries the user has to define the liquid boundary on the bed by hand. This can be done by modifying the subroutine `LIMI3D` in the user fortran. For example to add a circular boundary of radius 50 m centred around coordinate (2000, 2000) m, the following modifications can be done:

```
...
!      BOUNDARY CONDITIONS ON VELOCITIES
!      *****
!
!      BOTTOM
!      =====
!
!      DEFAULT: IMPERMEABILITY AND LOG LAW
!
!      IF (BC_BOTTOM.EQ.1) THEN
!
!          DO IPOIN2 = 1, NPOIN2
!              LIUBOF%I(IPOIN2) = KLOG
!              LIVBOF%I(IPOIN2) = KLOG
!              LIWBOF%I(IPOIN2) = KLOG
!              USEFUL ? SHOULD NOT BE USED ANYWAY
!              UBORF%R(IPOIN2) = 0.D0
!              VBORF%R(IPOIN2) = 0.D0
!              WBORF%R(IPOIN2) = 0.D0
!              IF (SQRT((X(IPOIN2)-2000.D0)**2
!                  &      + (Y(IPOIN2)-2000.D0)**2)
!                  &      .LE.50.D0) THEN
!                  !5: IMPOSED FLOW RATE
!                  LIUBOF%I(IPOIN2) = 5
!                  LIVBOF%I(IPOIN2) = 5
!                  LIWBOF%I(IPOIN2) = 5
!                  NLIQBED%I(IPOIN2) = 1
!                  PRINT*, '===== '
!                  PRINT*, 'FOR POINT ', IPOIN2
!                  PRINT*, 'BEDFLO', BEDFLO(1)
!              ENDIF
!          ENDDO
!
!      ...
```

It should be noted that since `NLIQBED%I(IPOIN2) = 1` this is only applied to the first liquid boundary defined in the steering file.

This is all that needs to be defined by the user to deal with fluxes on the bed. The developments added to version 7.1 of Telemac-3D, will then calculate the velocity imposed on the bed from the imposed flowrate and the area of the liquid nodes. At the moment, only a constant velocity profile is available. The term  $F_b$  will then be calculated and added to the relevant steps.

## VI. VALIDATION OF THE IMPLEMENTATION THROUGH A SIMPLE TEST CASE

A new test case has been added to the Telemac-3D examples. It is known as `bottom_bc`. The configuration of this test case is simple, it is a square box of sides 4000 m. The depth is constant, and initially set to 500 m. A discharge  $Q$  of  $10\,000\text{ m}^3\text{s}^{-1}$  will be imposed inside a circle with diameter  $D$

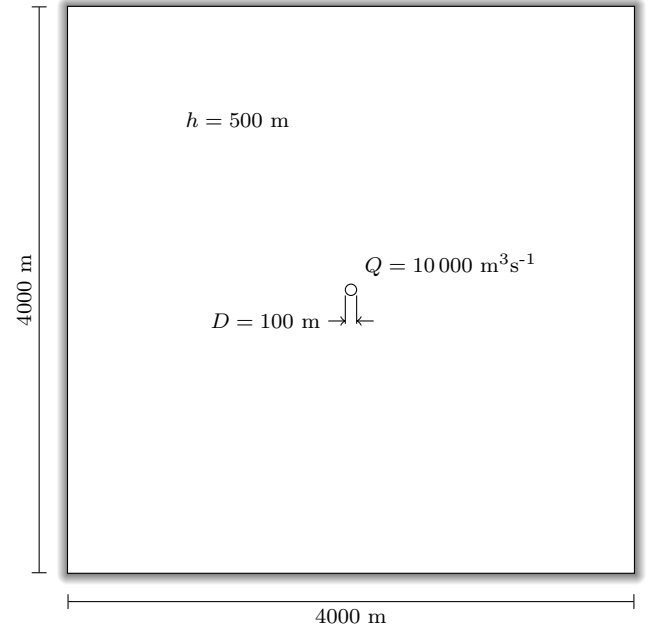


Fig. 1: Geometrical parameters of the test case.

of 100 meters placed at the centre of the box. The geometry of the test case is shown in figure 1.

This test case is solved in three different manners, two of which are given by steering file in `bottom_bc` and an additional method used to validate.

- 1) `t3d_bottom_inlet.cas`: This file launches a simulation where an inlet is imposed on the bed as a liquid boundary with an imposed flow rate.
- 2) `t3d_bottom_source.cas`: This file launches a simulation where the inlet is imposed as a source discharge.
- 3) `bottom_inlet_equiv_source`: This simulation is not present in the example folder. It is a reference solution where an inlet is imposed on the bed as a liquid boundary, but the fortran user file has been modified so that  $F_b$  is calculated to be equivalent to the source terms, and once it is calculated the velocity on the bed is fixed to zero, as source terms do not cancel the impermeability of the bed.

In essence, the simulation launched by file `bottom_inlet_equiv_source` should give the same results as `t3d_bottom_source.cas` and it is used to validate the fact that no bug has been introduced in the development of the liquid bed boundary conditions.

Furthermore, two different meshes will be used, a fine mesh and a coarse mesh. Since source terms are imposed on a node the coarse mesh is used to impose the inflow on a single node, and it will be used for `t3d_bottom_source.cas` and `bottom_inlet_equiv_source`. Since applying a flowrate can be imposed on several nodes on the bed, the finer mesh will be compared to the coarse mesh for `t3d_bottom_inlet.cas` simulation results. The coarseness of the mesh is also present for the distribution of the

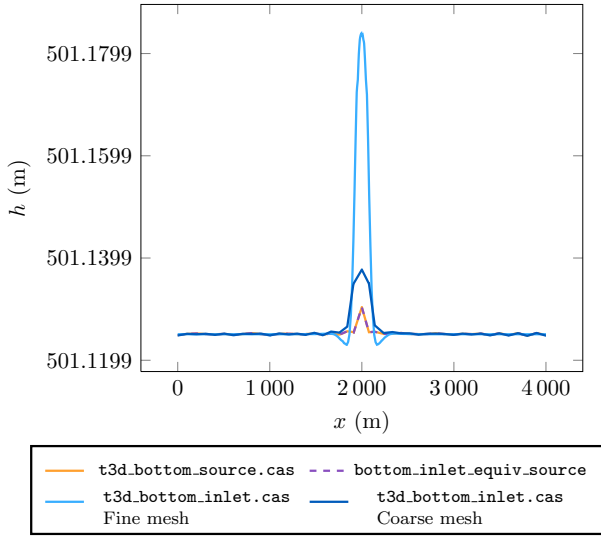


Fig. 2: Water depth profiles plotted along the line  $y = 2000$  at  $t = 1800$  s.

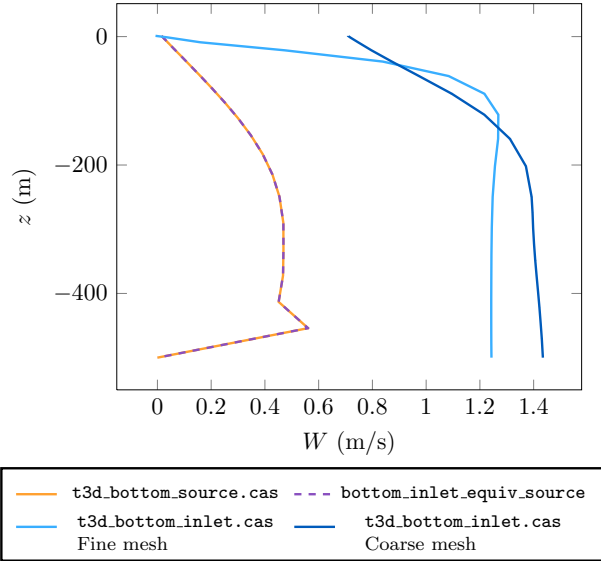


Fig. 3: Vertical velocity profiles plotted along the line  $x = 2000$  and  $y = 2000$  at  $t = 1800$  s.

planes in the simulation. The fine mesh has a smaller plane spacing near the bed and the free surface, whereas the coarse mesh has the same number of planes, but these are distributed evenly on the bottom half of the domain and the plane spacing decreases towards the free-surface.

Finally, simulation results of the imposed flowrate on the bed will be compared to a point discharge imposed through source terms because the use of  $F_b$  is close to those source terms, and great care was taken in previous versions of Telemac-3D to ensure that these source terms did not introduce any unphysical effects (see Telemac-3D release notes 5.7 [2]).

The simulation results will be shown in three figures. Firstly, profiles of the water depth will be plotted along the

line  $y = 2000$  at  $t = 1800$  s in figure 2. Secondly, profiles of the vertical velocity will be plotted along the line  $x = 2000$  and  $y = 2000$  at  $t = 1800$  s in figure 3. Finally, contour plots of this vertical velocity will be plotted along the plane  $y = 2000$  at  $t = 1800$  s in figure 4.

The first thing to observe when looking at figures 2 and 3 is that, as expected, the simulation results for `bottom_inlet_equiv_source` are equivalent to `t3d_bottom_source.cas`. This shows that one can expect to have results that are at least as good when imposing a flow rate on the bed than when using a source discharge. Furthermore, this is why the velocity contour plots for `bottom_inlet_equiv_source` are not shown in figure 4.

Looking in greater detail at the water depths presented in figure 2, it is shown that the correct volume of water has been introduced (i.e. a mean increase of depth of 1.125 m is expected). Furthermore, even though the differences may look big on the graph between the different simulation results, they are of the order a few centimetres (compared to a depth of 500 m). Furthermore, no method gives unphysical results (such as a dip in the free surface above the inlet). In conclusion, when looking at depths only all the methods appear to be equivalent and can introduce the correct volume.

However, when looking at figure 3 the differences between a source discharge and an imposed flow rate on the bed become apparent. When applying a source discharge the impermeability condition of the bed is maintained. This explains why the vertical velocity is 0 at the bed for those results. A quick calculation shows that a velocity of 1.27 m/s is expected towards the inlet ( $4Q/(\pi R^2)$ ). These values are close to what is calculated with the imposed flowrate on the bed. The coarse mesh has a value that is slightly greater, but this is because the velocity imposed on the bed is the flowrate divided by the area of the node (i.e. a third of the surface of the connected elements) which is not equal to  $\pi D^2/4$ . Aside from the differences of the bed velocity, the results from `t3d_bottom_inlet.cas` maintain this velocity almost throughout the bottom two thirds of the depth, whereas the result from the source discharge (`t3d_bottom_source.cas`) never reach more than half of this value (0.6 m/s). Therefore, for processes where the vertical velocity is important, an inlet should be modelled as a flowrate on the bed instead of through a discharge source.

Furthermore, refining the mesh in three dimensions will allow the vertical velocity calculated from source terms to be much more accurate than a coarse mesh, as the velocity on the free surface is close to 0, which was not the case for a coarse mesh. Figure 4 shows that the column of water with a high velocity is larger with a refined mesh, which also leads to a slightly higher water depth directly above the inlet, but more importantly there is a small drop of the water depth around the inlet and the water depth is a lot more stable away from the inlet (see figure 2).

## VII. CONCLUSION AND RECOMMENDATION FOR USE

In the beginning of this article, it was shown that imposing a velocity on the bed will only affect Telemac-3D by introducing a new term,  $F_b$ , to the several equations solved by the code. However, from a user point of view this is done by

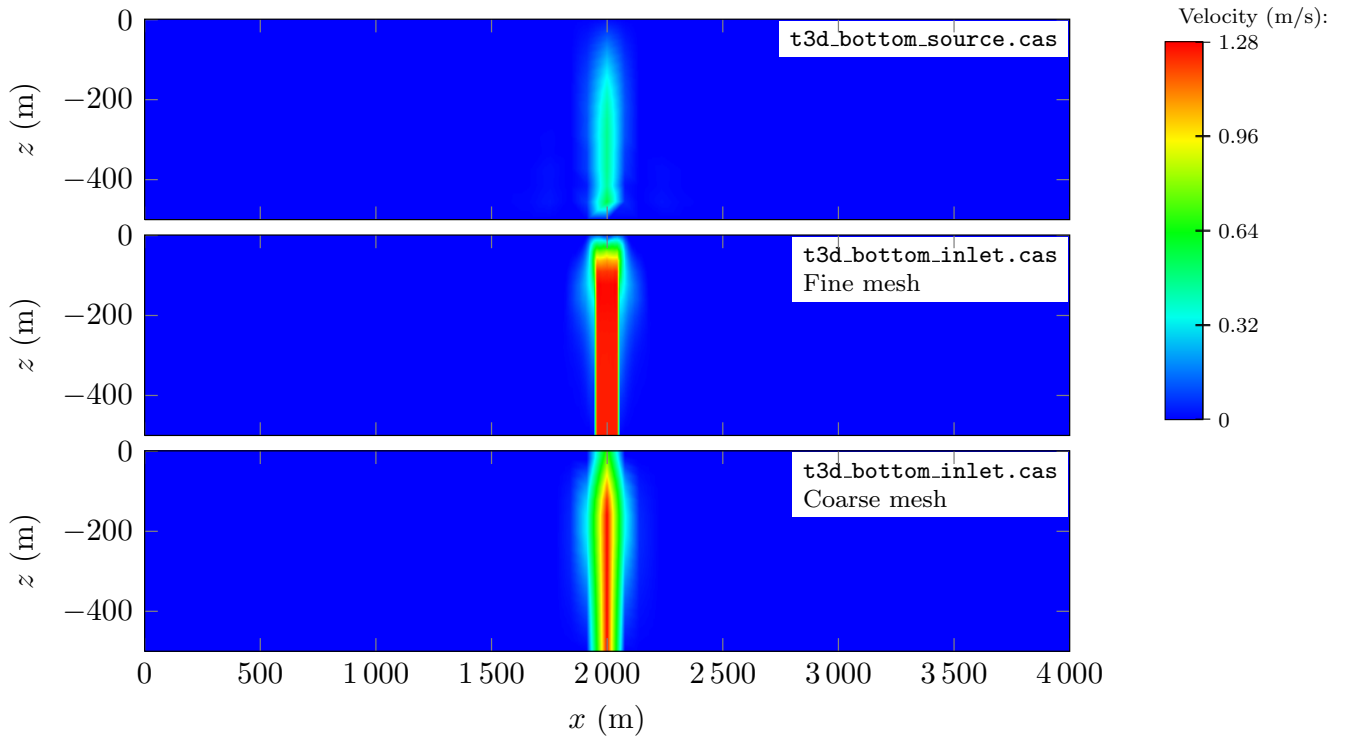


Fig. 4: Vertical velocity contours plotted along the plane  $y = 2000$  at  $t = 1800$ .

the use of keywords `OPEN BOUNDARY CONDITIONS ON THE BED` and `PRESCRIBED FLOWRATES ON THE BED` in the steering file and through the modification of `LIMI3D` in the user fortran.

The validation of these developpements through a simple test case have shown that introducing liquid boundaries on the bed will not affect greatly the free surface of the flow when compared to a source discharge, which was the only method available in previous versions to impose inflows or outflows near the bed. However, imposing a flow rate on the bed has a great impact on the vertical velocities modelled. In addition, these boundary conditions are a lot more flexible for the user than source terms close to the bed, as it is possible to spread the flowrate over an area covering several nodes. As such, it is recommended to properly impose liquid boundaries on the bed, rather than using source terms.

Nonetheless, these boundary conditions may still be improved. At the moment, it is not possible to specify the fluid density on these liquid boundary, and therefore it has not been tested on stratified flows. The same is true for turbulence.

Nonetheless, several tests have been done with these new boundary conditions. They have been tested in parrallel and in scalar implementations with minimal differences. These boundary conditions have been tested with all advection schemes. The mass losses are of the order of the machine precision if the `ACCURACY FOR PROPAGATION` is set to small enough value. It is also possible to add the `DYNAMIC PRESSURE IN WAVE EQUATION`, but this requires the keyword `VELOCITY PROJECTED ON BOTTOM` to be set to no.

In addition, when using these new liquid boundaries, these recommendations should be followed:

- Use the non-hydrostatic version of the code (it will work for hydrostatic flows, but a lot of the errors will be reported on the vertical velocities).
- Use a sigma transformation of the mesh, i.e. setting `MESH TRANSFORMATION` to 1 or 2.
- Refining along the bed and the free surface will help produce correct vertical velocity profiles.
- The time step needs to be chosen so that the CFL condition is also valid along the vertical.

Finally, it should be noted that when imposing a flowrate it will always be converted into a velocity along the normal of each node and that these conditions will work for outflows as well.

## REFERENCES

- [1] J.-M. Hervouet, *Hydrodynamics of free surface flows*. Chichester: Wiley, 2007.
- [2] J.-M. Hervouet and C.-T. Pham, "Telemac version 5.7, release notes. Telemac-2D and Telemac-3D," EDF R&D LNHE, Tech. Rep., 2007.