VOF SIMULATION OF SEDIMENT TRANSPORT UNDER HIGH VELOCITY FLOW CASE OF A DAM-BREAK OVER MOBILE BED

Denis Morichon[1], Jonathan Desombre[2], Benjamin Simian[1241]

Abstract

Extreme flow conditions due to dike or dam failure can cause intense sediment transport. In some cases, volume of entrained material can reach the same order of magnitude as the volume of water initially released from the water flooding itself. Dam-break flows are usually characterized by violent free surface, rapid and transient multi-phases flow. Numerical simulation of this complex flow requires to consider flow-particle and particle-particle interactions. In this study we propose to apply a single fluid formulation vof Navier-Stokes model. In our approach, the bed is considered as water-sediment mixture which physical properties, density and viscosity, are assumed to be a function of the sediment fraction. Furthermore, the mixture is assumed to be a non-Newtonian fluid which can interact with water and air. The model is applied to the case of a dambreak over a mobile bed and compared with experimental data.

Key words: Sediment-water mixture, scouring, mobile-bed dambreak, non-Newtonian fluid, Navier-Stokes, vof

1. Introduction

Sudden and high velocity flow induced by dam or dike failure can lead to rapid and intense soil movements. The behavior of the fluid-like soil can be very complex and depends on the behavior of its constituents such as granular material, air and water.

The dam-break flow problem has been first study in the case of a fixed bottom. The problem is usually idealized according to the following configuration. A vertical barrier divides fluids of different depth until at time t=0 the barrier is instantaneously removed and fluid flows in the shallower region. An analytical description of the flow was obtained by Stoker (1957), who proposed a solution of the shallow water equations. The problem becomes more complex when dealing with a movable bed downstream of the structure. In this case, the sudden release of water can generate hydrodynamic conditions with highly concentrated sediments. The volume of entrained material can be considerable and can reach in certain cases the same order of magnitude as the volume of water initially released from the failed structure. The resulting downstream flow is so rapid and involved such intense rate of transport that granular components play an active role in the flow dynamic. Rapid bed changes induce a jump of the water surface whereas a scouring generally appeared at the dam location.

Numerical simulation of sediment transport generated by violent free surface flow induced by a dam-break over a movable bed implies to deal with large deformation, fragmentation of the free surface and multiple interfaces. One approach consists in solving the non-hydrostatic free surface flow and the sediment transport separately. First, the Navier-Stokes equations are solved, coupled with a volume of fluid method (VOF), to obtain the detailed vertical structure of the flow while capturing the air-water interface transformation. An advection-diffusion equation is then solved to obtain the sediment concentration. The bed changes are calculated by applying the sediment mass conservation between the bed and the suspension. In this single phase type model, sediments are assumed to be entrained by the flow without interacting with it. The effects of sediment transport and bed deformation on the movement of flow is thus not considered.

Recently, two-phase flow sediment transport models have been applied to simulate the influence of

[1] SIAME-IVS, University of Pau et des Pays de l'Adour, allée du Parc Montaury, Anglet 64600, France. denis.morichon@univ-pau.fr

[2] INGEROP, Bâtiment LE SARDE, 15 chemin de la Dhuy, Meylan 38240, France.

sediment particles on the carrier fluid. These models solve conservation of mass and momentum for both fluid and sediment phases (Gotoh andFresdoe, 2000). A multi-phase mesh free particle model was proposed by Shakibaeinia and Jin (2011) to simulate the mobile-bed dam break problem. Unlike traditional two-phase flow models, they treat the problem as a multi-density and multi-viscosity problem and solve a single set of equations. They showed that a non-Newtonian model is needed for reproducing the detailed features of flow and sediments.
In this work, we propose to apply an Eulerian mesh-based method to the mobile-bed dam break problem. This approach is expected to be less time consuming than the mesh-free particle method. We use a single fluid formulation to account for the complexity of multiphase flow in sediment transport while limiting the number of momentum equations to solve. We assume that the bed can be considered as a mixture of sediment and water. The physical properties of the mixture, its density and viscosity, are then assumed to be a function of the sediment fraction. The mixture is considered to be a non-Newtonian fluid that can interact with the ambient water and air flows.
In the following, we first present the modeling approach. We then apply the model to the simulation of bed scouring downstream of a rigid step. This test case is chosen for its simplicity as a two- dimensional sediment transport problem. We analyze the ability of the model to reproduce the different stages of scouring induced by a sudden uni-directional flow. The approach is then applied to the simulation of the dam-break problem over a mobile bed. The model is qualitatively compared with laboratory experimental data (Spinewine and Zech, 2007). Our study focuses on the near-field response to the dam-break waves.

2. **Numerical approach**

2.1. ***Governing equations***

Our numerical approach is based on the Thetis code developed by UMR TREFLE of the University of Bordeaux. The code is a VOF Navier-Stokes model that allows to simulate turbulent free surface flows. Lately, it was used to study the vertical structure of a transient and turbulent aerated flow (Desombre et al., 2013) induced by a dam break over a fixed bed. The code is based on a single fluid Navier-Stokes formulation in which air-water flow is considered following Kataoka (1986). In this formulation density and viscosity of the different phases are calculated as function of location.
A color function, or phase function, C(x,z,t) is used to locate the different phases stating C (x,z,t)=1 when a mesh is filled with water and C(x,z,t)=0 when it is filled with air. Intermediate values of C indicate the proportion of water in a mesh. In this work, the Navier Stokes equations are solved and coupled with a sub-grid scale turbulence model (LES). The following set of equations are thus solved assuming incompressibility of the fluids:

$$\nabla . u = 0 \tag{1}$$

$$\rho\left(\frac{\partial u}{\partial t}+(u.\nabla)u\right)=\rho g-\nabla(p)+\nabla.(\mu+\mu_t)\left(\nabla u+\nabla^T u\right) \tag{2}$$

where u(x,z,t) is the flow velocity, t the time, p(x,z,t) the pressure, g the gravity. The density ρ(x,z,t) and the viscosity μ(x,z,t) in a given cell are calculated according to:

$$\rho(x,z,t)=\rho_w C(x,z,t)+\rho_a(1-C(x,z,t)) \tag{3}$$

$$\mu(x,z,t)=\mu_w C(x,z,t)+\mu_a(1-C(x,z,t)) \tag{4}$$

where density and viscosity of water and air are respectively ρ_w = 1000 kg.m^{-3}, μ_w = 10^{-3} Pa.s and ρ_a = 1.18 kg.m^{-3}, μ_a = 1.85 10^{-5}Pa.s.

To deal with the air-water interface tracking and calculation, the Piecewise Linear Interface Calculation (PLIC) volume of fluid method is used. In the following simulations, the conservative nature of the PLIC method is ensured by automatically computing the time step using a Courant-Friedrichs-Levy number (CFL) set to 0.3. Furthermore, the PLIC method is coupled with a smoothing step in order to deal with complex interfaces.

2.2. *Movable-bed simulation*

In our approach, the sediment layer is considered as an additional phase trough a water-sediment mixture. This phase is treated as a non-Newtonian fluid. We use a Bingham plastic fluid model. This rheological model has been widely used to simulate granular flow, and more recently in a numerical study of the problem of dam-break over a mobile bed (Shakibaeinia and Jin, 2011).

The Bingham plastic model states that the bed behaves as a rigid solid for stress lower than a threshold value ♦$_s$, but flows as a Newtonian fluid for greater values. The sediment-water mixture density ❒$_m$(x,z,t) is calculated according to

$$\rho_m(x,z,t)=\rho_s C_s(x,z,t)+\rho_w(1-C_s(x,z,t)) \tag{5}$$

where C_s(x,z,t) refers to the fraction of sediment in a cell and ❒$_s$(x,z,t) the density of the sediment. The viscosity of the mixture ❍$_m$(x,z,t) is given by

$$\mu_m(x,z,t)=\begin{cases} \mu_s+\tau_s/\sqrt{\tfrac{1}{2}D:D} & if \ \tfrac{1}{2}\tau:\tau \geqslant \tau_s^2 \\ \infty & if \ \tfrac{1}{2}\tau:\tau < \tau_s^2 \end{cases} \tag{6}$$

where D is the shear rate and ❍$_s$ refers to the dynamic viscosity of the slurry which designs the mixture of a liquid (water in our case) and solid particles. This viscosity is described as relative to the viscosity of the liquid phase :

$$\mu_s=\mu_r.\mu_w \tag{7}$$

where μ_r is a dimensionless relative viscosity.

Critical conditions for incipient motion of sediments are usually expressed in a non-dimensional form with the critical Shields parameter :

$$\theta_c=\frac{\tau_s}{(\rho_s-\rho_w)g\,d} \tag{8}$$

where d is the sediment particle diameter. Cao et al. (2006) proposed an explicit formulation of the critical Shields parameter as a function of the fluid and sediment characteristics. The value of ♦$_s$ can then be derived from Equation 8.

The model is completed with an advection equation specifying that interface between water and water-sediment mixture moves with the flow velocity:

$$\frac{\partial C_s}{\partial t}+u\nabla.C_s=0 \tag{9}$$

2.3. *Scouring downstream of a rigid step*

In order to test the validity of our sediment transport model, we apply our approach to the case of scouring downstream of a rigid step (Figure 1). This test case consists in simulating bed evolution induced by a high velocity unidirectional flow. It has been chosen as it involves complex phenomenon such as two-phase turbulent flows while being relatively simple to simulate.

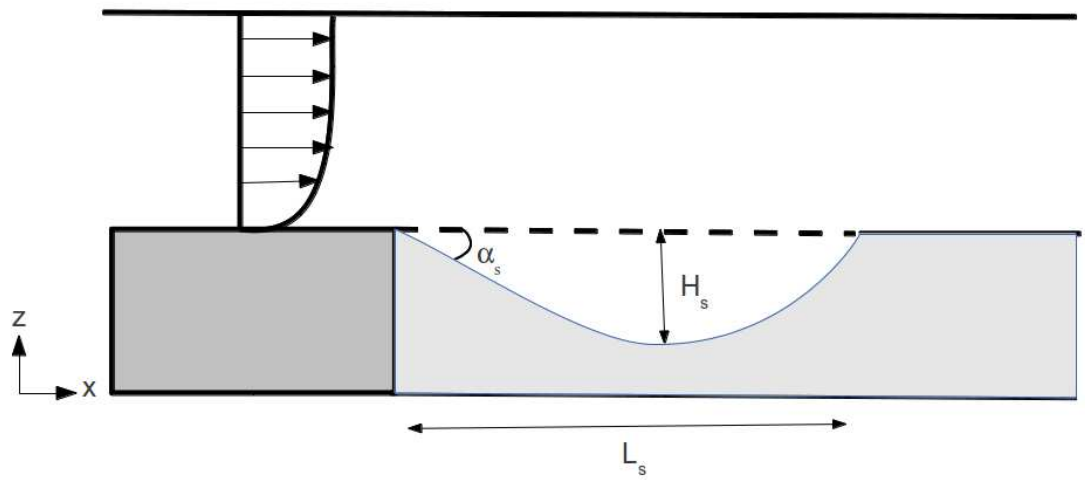


Figure 1. Scheme of scouring downstream of a rigid step

The scouring mechanism downstream of the step is a well documented problem (Breusers, 1967), and the geometric characteristics of the scour hole such as its length L_s, its depth H_s and the upstream scour slope α_s (Figure 1) have known behaviors (Breusers and Raudkivi, 1991; Hoffmans and Pilarczyk, 1995).
Our numerical set up consists in a canal of 0.6 m length and 0.1 m height filled with water. The bottom of the tank is initially covered with a horizontal saturated sediment layer of 5 cm thickness. The sediment concentration $C_s(x,z,t)$ in a cell is set to 0.6. The sediments grain diameter is taken equal to 0.25 mm and the sediment density $\rho_s(x,z,t)$ = 1500 kg.m^{-3}. The sediment characteristics were chosen to be representative of high mobility sediment materials. This choice is intended to limit the number of iterations needed to simulate significant bed changes downstream of the step while being representative of highly concentrated flow conditions. Two rigid rectangular steps of 0.1 cm length are placed on each side of the sediment layer.

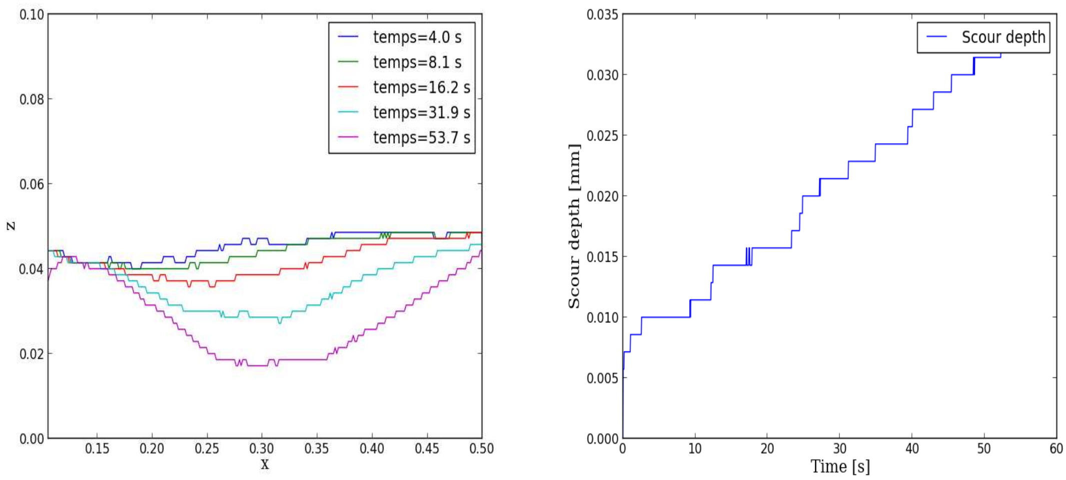
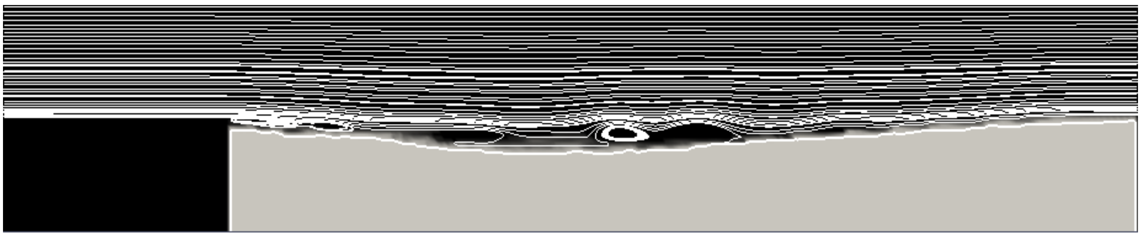


Figure 2.Time evolution of the bed (left panel) and maximum scour depth (right panel)

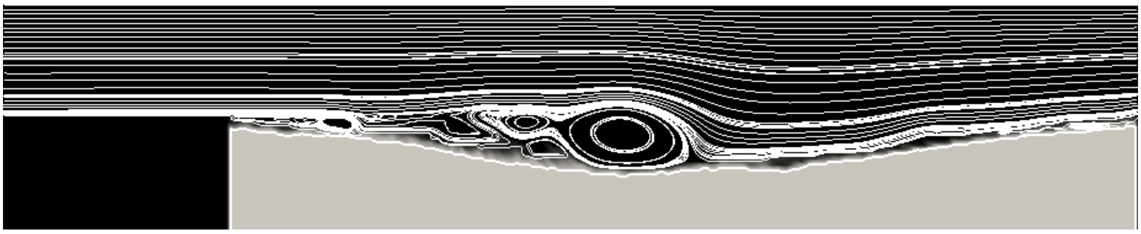
Those steps are simulated as a porous medium, for which porosity tends to zero, thus preventing the fluid to flow within it.

The computational domain is divided in a regular grid system with $\Delta x=\Delta z=1.5$mm. On the left and right boundaries a constant uniform velocity profile is set to 50 cm/s. This numerical set-up is intended to reproduce transient hydraulic conditions downstream of an apron following a sudden release of water, generated by a structure failure or a flood event, while simulating the development of a constant unidirectional flow. The top boundary is assumed to be a symmetry boundary condition and a no-slip boundary condition is applied to the bottom.

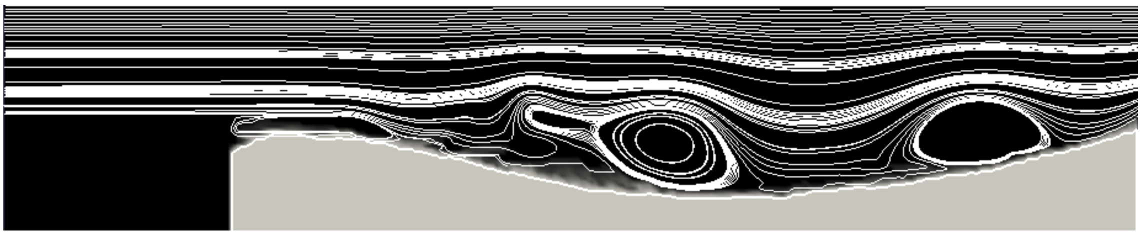
The spatial evolution of the interface between the clear water and the sediment layer is displayed on the left panel in Figure 2 at different times. The right panel displays the time evolution of the maximum scour depth. The model seems to simulate correctly the main stages of the scour development as described in Hoffmans and Pilarczyk (1995). At the early stage of the scour formation, erosion takes place all over the bed, but is more intense in the near field of the step. Then, the scour hole considerably increases in length and depth. Nevertheless, the hole shape, which can be expressed in term of ratio between L_s and H_s, remains relatively unchanged. The scour slope also increases with time and tends toward an equilibrium. The equilibrium angle of the upstream scour slope was estimated at 15°, which is consistent with the values indicated by Hoffmans and Pilarczyk (1995).



Time = 20 s



Time = 40 s



Time = 60 s

Figure 3. Bed form evolution at different time steps with streamlines (white solid line represents the interface between water and sediment layer).

In order to get a better insight of the flow above the sediment layer during the different development stages of the scour hole development, a sequence of snapshots with streamlines taken at three different times, is displayed in Figure 3. On the upper panel, which corresponds to the beginning of the development stage of the scour, the hole is underdeveloped. However, the entire bed experiences erosion. According to the shape of the streamlines, the flow in the clear water is unidirectional over the major part of the water column. Close to the bed, we observe a series of vortices. Their number and size change in time. Right downstream of the step, one vortex is present in each snap-shot. This vortex is representative of the well known reversal flow in the case of turbulent flow over a backward facing step. At the location of the maximum scour depth, an other vortex is visible in each snap-shot. Its height is quasi equal to the hole depth. In the last snap-shot, at time 60 s, we observe an additional detached vortex downstream of the maximum score depth location. Its size is similar to the size of the previous vortex. We can note that the time evolution of the number and spatial distribution of vortices corresponds to the time scale of the score hole development. The more developed is the score, the more vortices there are.

3. *Dam-break over an horizontal mobile bed*

In this section, the sediment transport model is applied to the case of a dam-break over a flat mobile bed. The simulations are compared with data obtained during the laboratory experiment performed by Spinewine and Zech (2007). The experiment was performed in the framework of the EC_funded IMPACT project and was intended to serve as a benchmark for sediment transport modeling.

In their experimental set-up, a wave tank is divided by a gate. The bottom of the tank is covered with fully saturated movable materials. The upstream side of the tank is then filled with water. The dam-break phenomenon is reproduced by uplifting the gate at very high speed. The rate of gate ascent was estimated at less than 50 ms. The experiment was conducted with PVC pellets. The grains are slightly cylindrical in shape with an equivalent spherical diameter of 3.5 mm. The flow was measured with two synchronized CCD cameras operating at frame rates of 200 images per second.

3.1. *Numerical set-up*

In our numerical set-up (Figure 4), the removal of the gate is not reproduced. Indeed, since the removal speed and the roughness of the gate are unknown, we assume that the gate is instantaneously removed. The sediment layer is simulated as a mixture of water and sediment with a sediment ratio of 0.6, which is equivalent to a fraction of void space in the saturated sediment layer of 40%. The relative density of the PVC pellets is $s = \rho_s/\rho_w = 1.54$.

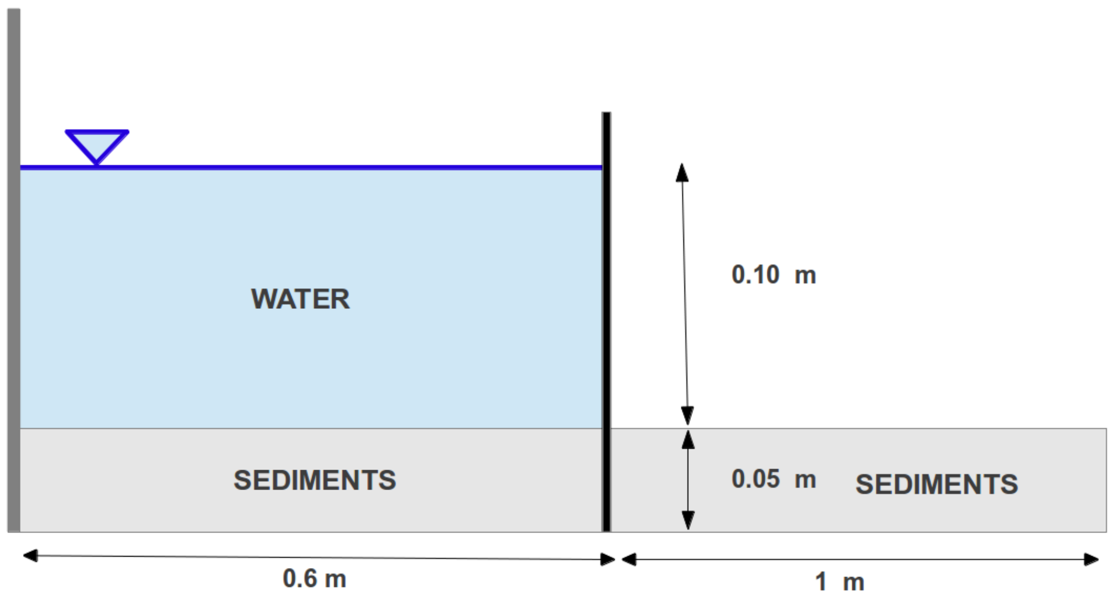


Figure 4. Numerical set-up of the dam-break problem over a flat mobile bed

The computational domain is 1.6 m length and 0.25 m height. The length of the reservoir is intended to simulate a constant water level upstream of the gate. We use a regular grid mesh with a spatial resolution $\Delta x = \Delta z = 2$ mm. A no-slip boundary condition was assumed on the bottom and right boundary. The left and top boundary conditions were assumed to be symmetry boundary condition.

3.2. *Results*

Figure 5 displays the comparisons between experimental data and model simulation shortly after gate removal. In panel a and panel c, the light grey corresponds to the sediment layer density and dark grey to the water density. Air density is displayed in black. The model reproduces the generation of a dam-break wave following the collapse of the water body (panel a). Then, the wave front propagates downstream and erodes intensely the sediment layer (panel c). In the simulation, a scouring zone appears near the initial position of the gate. Its size and shape is similar to the laboratory measurements. This scouring zone is still present during the wave propagation stage. This suggests that the sediment transport model verifies the conservation of mass of the sediment.

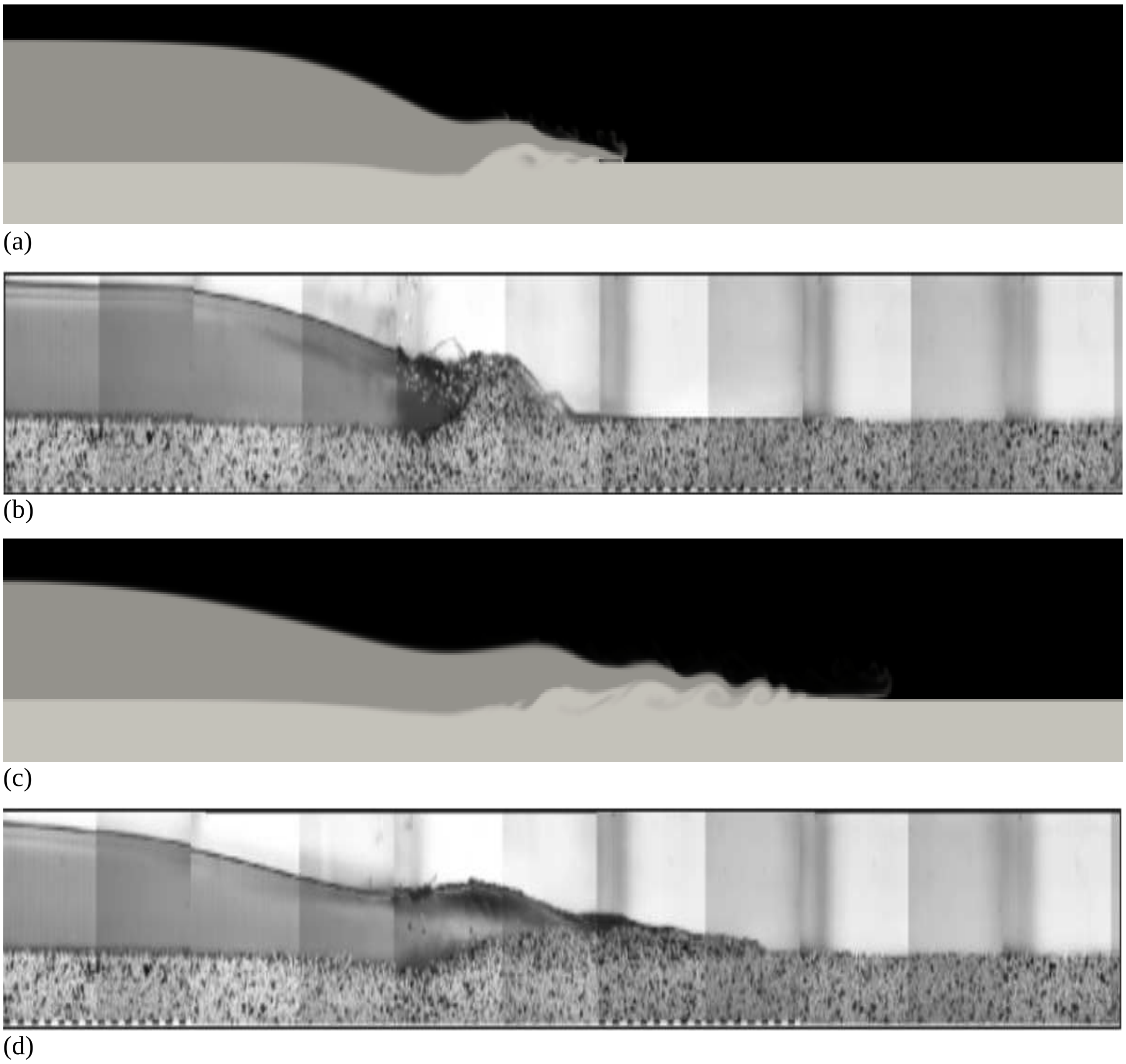
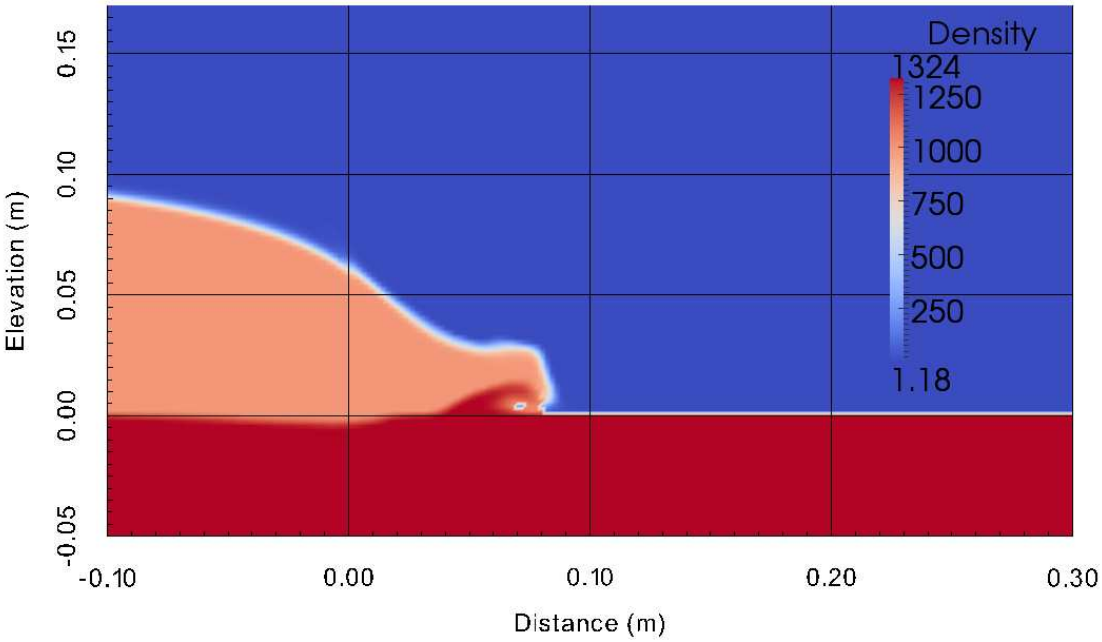


Figure 5. Comparisons between model simulations (panel a and panel c) and the corresponding experimental data (panel b and panel d) at t=0.25 s and t=0.5 s respectively

The model is able to reproduce the different flow regions as described by Capart (2000). Clear water constitutes the upper layer of the flow region. It propagates over the sediment layer which is divided into a

transport layer and a motionless layer.
At t= 0.25 s, the thickness of the measured sediment layer is higher than in the simulation. Discrepancies might be the result of the gate removal mechanism. Indeed, in their numerical study, Shakibaenia and Jin (2011) showed that the method of removing the gate has a significant effect on the generated wave form and its breaking process. At t= 0.5s, the evolution of the free surface of the flow suggests a strong coupling between the sediment layer dynamic and the wave dynamic. Ripples of the sediment layers coincide with the position of trough and crest of the dam break waves.



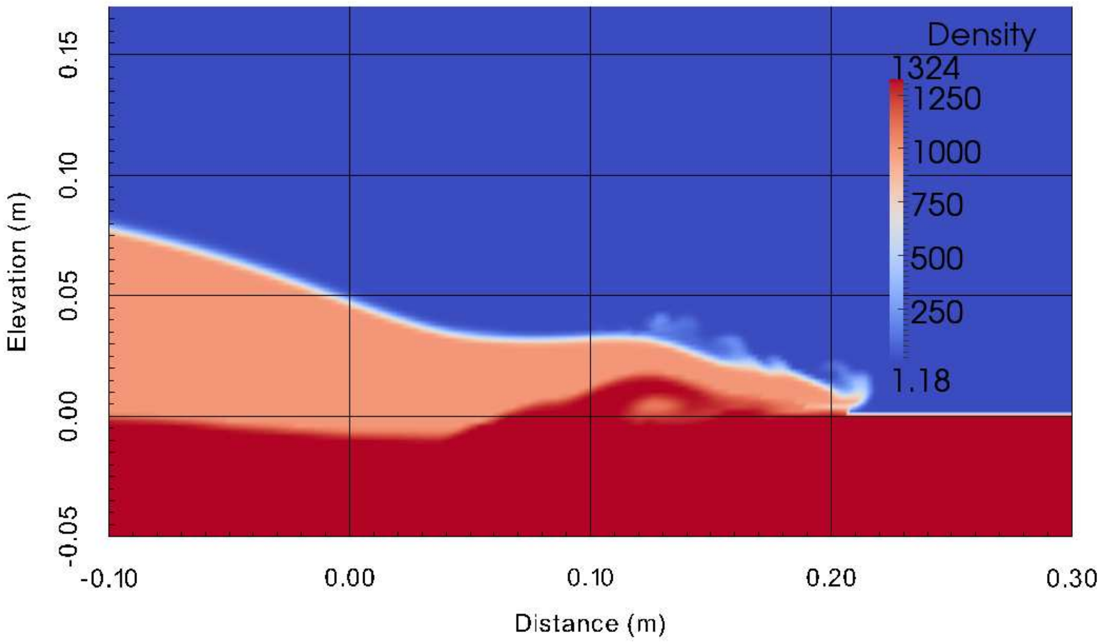


Figure 6. Computed density map at t = 0.1 s (top panel) and t = 0.25 s (bottom panel)

Figure 6 displays the density map shortly after the gate release. At t = 0.1 s (upper panel), the water jumps above a bulk of water-sediment mixture. At the location of the gate, a small scour has appeared. At t = 0.2 s its shape, in term of length and depth, has increased and the position of the maximum scour depth has moved downstream. At t = 0.1 s, the shear stress at the water front is greater than the failure stress of the sediment layer. As a consequence, a layer of the water-sediment mixture behaves as a Newtonian fluid, forming a breaking wave which then propagates over the motionless part of the bottom. At t = 0.2 s, water has been entrapped in a cavity following the reconnection of the mobile sediment layer interface.

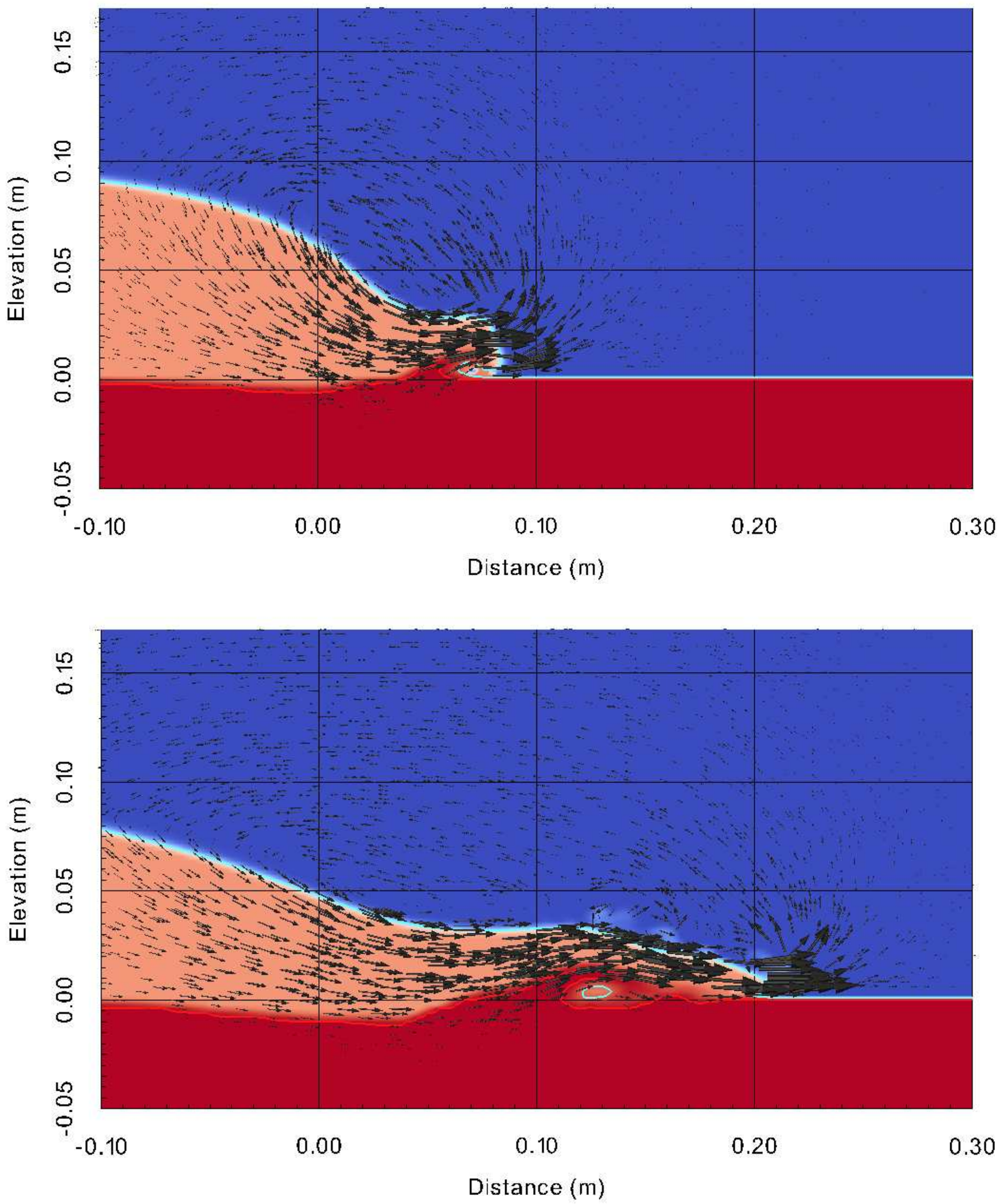


Figure 7. Computed velocity vectors at t = 0.1 s (top panel) and t = 0.25 s (bottom panel). The blue line represents the air-water interface and the red line the upper boundary of the water-sediment mixture.

In Figure 7, the velocity vectors field allows to distinguish the moving transport layer from the underlying motionless bed. We observe that the maximum velocity of the flow occurs near the water front. At t = 0.2 s (bottom panel), intense water flow is also observed above the bulk of sediment material in motion. In the rear of the sediment bulk, velocity decays.

4. ***Conclusions***

We proposed in this paper a sediment transport model to simulate very transient and multi-phases flows induced by dam-break over a flat mobile bed. The model was first applied to the case of scouring downstream of a rigid step. The main stages of the scour hole development are well reproduced. Analysis of the vertical structure of the flow above the sediment layer highlights the role of vortices on the sediment transport rate and subsequent shape of the scour.
The model is then applied to the dam-break problem. The model allows to reproduce the generation of the dam-break wave and its downstream propagation. Some discrepancies in the form of the sediment bulk shortly after gate removal suggests that the assumption of instant gate release is not valid. It would be interesting to gradually pullet it up and to study the influence on the wave shape. During the wave propagation, the model allows to reproduce the strong coupling between the free surface wave and the sediment layer in motion.
This preliminary work shows the ability of a single fluid formulation model, coupled with a VOF method, to reproduce the main characteristics of a transient and intense multi-phase flow problem. Further works, including sensitivity analysis of the model, are needed to improve the non-Newtonian treatment of the solid phase.

Acknowledgements

The authors wish to thank Thetis administrator, Stéphane Glockner (UMR TREFLE, Université de Bordeaux I) for his useful assistance. This work was partly funded by the Aquitaine Regional Council, France.

References

Breusers, H.N.C., 1967. Time scale of two-dimensional local scour. *Proceeding 12th IAHR Congress, Ft. Collins,* 3, 275-282

Breusers, H.N.C., and Raudkivi, A.J., 1991. *Scouring.* A.A. Balkema, Rotterdam

Capart, H., 2000. Dam-break induced geomorphic flows and the transition from solid to fluid-like behavior across evolving intefaces. *PhD thesis*, Université Catholique de Louvain, Belgium.

Cao, Z. Pender, G. and Meng J., 2006. Explicit formulation of the Shields diagram for incipient motion of sediment. *Journal of hydraulic engineering,* 1097-1099

Desombre, J., Morichon, D., and Mory, M., 2013. RANS v²-f simulation of a swash event: Detailed flow structure. *Coastal Engineering*,71, 1-12

Gotoh, H and Fredsoe, J., 2000. Lagrangian two-phase flow model of the settling behavior of fine sediment dumped into water. *Proceeding of the ICCE, Sydney, Australia,*3906-3919

Hoffmans, G.J.C.M., and Pilarczyk, K.W., 1995. Local scour downstream of hydraulic structures. *Journal Hydraulic Engineering,* 121 (4), 326-340;

Kataoka, I., 1986. Local instant formulation of two-phase flow. *International Journal of Multiphase Flow,* 12 (5), 745-758

Shakibaeinia, A., and Jin, Y. C., 2011. A mesh-free particle model for simulation of mobile-bed dam break. *Advances in Water Resources*,34, 794-807

Spinewine, B. and Zech, Y, 2007. Small-scale laboratory dam-break waves on movable-beds. *Journal of Hydraulic research ,Extra Issue,* 45, 73-86

Stoker, J.J., 1957. Water Waves: The Mathematical Theory with Applications. *Vol. a Interscience Publishers,* New York