SPH-Based Numerical Study of the Impact of Mudflows on Obstacles

Dominique LAIGLE\(^1\) and Mathieu LABBE\(^1\)

\(^1\)Université Grenoble Alpes, Irstea, UR ETGR, 2 rue de la Papeterie-BP 76, F-38402 St-Martin-d’Hères, France. E-mail: dominique.laigle@irstea.fr

In this work, we study the impact of a transient free-surface flow of viscoplastic fluid on a rigid obstacle. This study is conducted numerically using the SPH (Smoothed Particle Hydrodynamics) method, and the Herschel-Bulkley rheological model. The SPH code and its specific adaptations to our needs are presented. The capacity of the code to meet the requirements of our objectives is validated on classic benchmarks. The virtual experiment setup is presented. The local characteristics of the flow near the obstacle, the length of the dead-zone of fluid at rest which forms upstream of the obstacle and the shape of the pressure signal applied to the obstacle are analyzed with reference to the inclination angle and Froude number of the incident flow. This analysis highlights the existence of two impact regimes referred to as the dead-zone and jet impact regimes respectively with a transition occurring for values of the Froude number about 1.3 to 1.5. These values are coherent with previous experimental studies.

Key words: debris flow, mudflow, impact force, SPH, protection structures

1. INTRODUCTION

The understanding and quantification of mudflow and debris flow – structure interactions in terms of modification of the incident flow and impact force applied to the obstacle are of paramount importance for the conception and design of structural countermeasures [Takahashi, 2014]. However, to date, only a limited number of studies have been carried out on that subject, with the consequence that the design of check-dams and other countermeasures against debris flows remains essentially empirical. Our study aims at determining local values of the mudflow velocity and pressure in the vicinity of a structure subjected to a mudflow impact, as well as the changes in these variables over time and space, for a given incident mudflow.

We use a numerical model based upon the SPH (smoothed particles hydrodynamics) numerical method. SPH is a particular method of treatment of fluid mechanics equations which is suitable for computing highly transitory free surface flows of complex fluids (viscoplastic, granular, etc.) in complex geometries. Thus, it is suitable for the treatment of debris-flow waves – structure interactions.

We analyze first the capacity of our model to meet the requirements of our study: simulate very unsteady flows impacting a structure and simulate viscoplastic fluid flows. In that aim, we carry out SPH simulations of a dam-break problem with water and compare the results to experimental data, including impact pressure records, and numerical results from the literature. We also simulate the velocity profile inside the free-surface steady flow of a viscoplastic fluid and compare the results to the theoretical profile.

In a second part, we present our numerical experiments setup derived from the laboratory experiments by Tiberghien et al. [2007]. In the presence of an obstacle, we simulate the local internal velocities immediately upstream of the obstacle at several times of the impact of a viscoplastic fluid wave. We also focus interest on the pressures developed on the structure during the impact which are computed thanks to numerical pressure sensors located on the upstream face of the obstacle.

2. THE SPH METHOD

The SPH method is a mesh-free numerical method initially introduced by Gingold and Monaghan [1977] and Lucy [1977] to study...
astrophysical problems. It has since found many applications in fluid dynamics [Monaghan, 1992]. Because of its mesh-free nature, it can handle large deformations of the simulated fluid. It also handles free surfaces naturally. These characteristics are crucial when modeling transient flows such as ours, as the fluid will spread on a long distance with regards to its thickness, and the free surface will deform significantly. This makes this method particularly well-suited for the simulation of the propagation of mudflows [Rodriguez-Paz and Bonet, 2004; Laigle et al., 2007].

SPH has been used in the simulation of the propagation of viscousplastic flows such as landslides and avalanches [Rodriguez-Paz and Bonet, 2004; Huang et al., 2011a], granular materials [Chambon et al., 2011] and their impact on a wall [Huang et al., 2011b]. It has been employed in studies of the interaction of water and mudflows with wall-like obstacles [Colagrossi and Landrini, 2003; Laigle et al., 2007; Sun et al., 2010].

2.1 Discretization and kernel function

SPH consists in discretizing the continuous medium into particles that represent small fluid elements, moving with the material velocity and carrying physical properties such as density, pressure and stresses. The value of a function $f$ and its spatial derivatives can be calculated at any point of the simulation domain by interpolating over the values of the neighboring particles:

$$ f(\mathbf{x}) = \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\mathbf{x}_j) W(\mathbf{x} - \mathbf{x}_j) $$

where $N$ is the number of SPH particles, and $m_j$, $\rho_j$, $\mathbf{x}_j$ are the mass, density and position of particle $j$. $W$ is the smoothing kernel function. Our code uses a cubic spline [Morris et al., 1997]:

$$ W(\mathbf{x}) = C \begin{cases} 1 - \frac{3s^2}{2} + \frac{3s^3}{4} & \text{if } s < 1, \\ \frac{1}{4}(2-s)^3 & \text{if } 1 \leq s < 2, \\ 0 & \text{if } s \geq 2, \end{cases} $$

with:

$$ s = |\mathbf{x}|/h \quad C = \frac{10}{7\pi^3 h^3} $$

The smoothing length $h$ is a parameter that determines the maximal distance at which a particle can interact with one of its neighbors. In our simulations, it is set at a constant value of $h = 1.2 \delta$, where $\delta$ is the initial spacing of the particles. This value is chosen so that a particle interacts with approximately 20 neighbors.

2.2 SPH implementation of continuity and momentum equations

In coherence with the weakly compressible SPH method we selected, our code solves the following continuity equation:

$$ \frac{d\rho}{dt} = -\rho \nabla \cdot \mathbf{u} $$

and the following momentum equation:

$$ \frac{D\mathbf{u}}{Dt} = \frac{1}{\rho} \nabla \cdot \mathbf{P} + \frac{1}{\rho} \nabla \cdot \mathbf{\tau} $$

where $\rho$ is the density of the fluid, $\mathbf{u}$ the local velocity and $D\mathbf{u}/Dt$ its material derivative, $\mathbf{P}$ is the total stress tensor, $\mathbf{\tau}$ is the deviatoric stress tensor, $P$ is the local pressure, and $g$ is the gravity. These Eqs. (4)-(5) are solved in two dimensions, in a vertical plane. While some SPH implementations enforce an incompressible fluid [Shao and Lo, 2003], we chose to use the more classical SPH method which is based on the assumption of a weakly compressible fluid.

Comparisons of the two methods have been made by Lee et al. [2008] and Hughes and Graham [2010]. A flux term, introduced by Ferrari et al. [2009], is added to the classical SPH mass-conservation equation to stabilize the pressure field, yielding the following equations:

$$ \frac{d\rho_i}{dt} = \sum_{j=1}^{N} m_j \left[ (\mathbf{u}_i - \mathbf{u}_j) \cdot \nabla \right] W_{ij} + e_i \cdot \nabla W_{ij} \left( \frac{\rho_i \rho_j}{\rho_i + \rho_j} \right) $$

$$ \frac{d\mathbf{u}_i}{dt} = \sum_{j=1}^{N} m_j \left[ \frac{\partial \mathbf{P}_{ij}}{\partial \rho_i} + \frac{\partial \mathbf{P}_{ij}}{\partial \rho_j} \right] \frac{\partial W_{ij}}{\partial \rho_i} + e_i $$

where $e_i$ is a unit vector pointing from particle $i$ to particle $j$. $W_{ij} = W(x_i-x_j)$, $e$ is the sound speed, and indices $\alpha$ and $\beta$ refer to Cartesian coordinates.

2.3 Computation of the pressure

SPH relies on an equation of state to compute the pressure $P$. Our code uses the following equation of state:

$$ P = c_s^2 \rho (\rho - \rho_0) $$

where $\rho_0$ is a reference density. This equation of state has been used previously on viscous flows with good results [Morris et al., 1997; Fang et al., 2009]. See also [Chambon et al., 2011] for a discussion of various equations of state used in previous studies. The value of $c$ must be chosen so that the artificial compressibility is negligible. Given that we do not expect our flows to reach velocities higher than 1.5 m/s (derived from the data of Tiberghien et al.
we determined that \( c = 150 \text{ m/s} \) is a good choice, as it ensures that the compressibility is smaller than 0.1\%.

2.4 Implementation of the Herschel-Bulkley rheological model

The rheology of mudflows and viscous debris flows can be described by the Herschel-Bulkley model [Coussot, 1994], in which the norm of the deviatoric stress tensor \( \tau \) is given by the following equation:

\[
|\tau| = \tau_c + K \dot{\gamma}^n \quad \text{if } \dot{\gamma} \neq 0,
\]

\[
|\tau| \leq \tau_c \quad \text{otherwise}
\]

where \( \tau_c \) is the yield stress, \( K \) is the consistency and \( n \) the flow index of the Herschel-Bulkley fluid, \( \dot{\gamma} \) is the shear rate. To simulate a Herschel-Bulkley fluid, we introduce an apparent dynamic viscosity \( \eta_{app} \) in our model:

\[
\eta_{app} = \frac{\tau_c}{\dot{\gamma}} + K \dot{\gamma}^{n-1}
\]

Let us consider what happens when the shear rate \( \dot{\gamma} \) goes to zero. Ideally, the apparent viscosity \( \eta_{app} \) would be infinite and the fluid would not move. However, we cannot deal with an infinite value of \( \eta_{app} \). Moreover, large values of the viscosity would lead to very small time steps, and thus very long simulation times. To solve these problems, we introduce a maximum viscosity value \( \eta_{max} \) meaning that for small values of \( \dot{\gamma} \), the fluid behaves like a highly viscous Newtonian fluid [Lachamp, 2003]. In such cases, we want the residual velocity of the fluid to be much smaller than the typical velocity of the flows we will simulate. We deal with free-surface flows of Herschel-Bulkley fluid for which the velocity profile is known in steady regime [Chen, 1988; Coussot, 1994]. When a maximum residual velocity \( u_{max} \) is fixed, the corresponding value \( \eta_{max} \) of the apparent viscosity can be computed as follows:

\[
\eta_{max} = \frac{\rho gh^2 \sin \theta}{2 u_{max}}
\]

where \( \theta \) is the slope angle, \( h \) is the flow depth, \( \rho \) is the fluid density and \( g \) is gravity.

The flows we want to simulate are derived from the experiments of Tiberghien et al. [2007]. They are characterized by flow depths of roughly \( h = 3 \text{ cm} \), an average slope angle about \( 7^\circ \) and velocities around 0.5 m/s. Therefore, a value of \( u_{max} < 0.0005 \text{ m/s} \) will keep the error below 1\%, which is more than enough. By plugging these numbers into Eq. (11), we obtain a value \( \eta_{max} = 1000 \text{ Pa.s} \). We will use this value in all the simulations of Herschel-Bulkley fluid flows presented in this work.

2.5 Boundary conditions

The SPH method was initially designed for simulating astrophysical problems without solid boundaries. Over the years, several techniques have been proposed [Liu et al., 2012], and the implementation of solid boundaries is still a matter of discussion. In our code, the boundaries are made of two rows of fixed SPH particles similar to those that constitute the fluid, but with specific properties. To prevent the penetration of fluid into the boundary, each boundary particle \( k \) interacting with a fluid particle \( i \) is given a virtual velocity \( u_{\text{virt}} \), opposite to \( u_i \), in the direction normal to the boundary. For no-slip boundary conditions, the velocity component parallel to the boundary \( u_{\text{parallel}} \) is fixed, and for free slipping, \( u_{\text{parallel}} = u_{\text{fluid}} \). In our case, to better approach natural conditions of a highly viscous fluid flowing over a rough bottom, we use no-slip conditions. Boundary particles are also given a virtual density to ensure that the hydrostatic pressure gradient is continuous across the boundary.

2.6 Time stepping

Once the acceleration of every particle is computed with Eq. (5), it is integrated over time to yield the velocity of the particles, while the velocity is used to compute the new positions of the particles, and Eq. (4) to compute the new density. Various methods exist for performing the timewise integration, both explicit [Monaghan, 1985; 1989; 1992] and implicit [Bonet and Kulasegaram, 2001]. In our code, the timewise integration is performed using a scheme equivalent to a leapfrog scheme [Monaghan, 2000]. Using initial values a predictor step is performed, and a corrector step yields the final values. The time step \( \Delta t \) has to be carefully chosen to ensure the stability of the integration scheme. The criterion on the time step takes into account the Courant-Friedrichs-Levy condition, the magnitude of particle accelerations, viscous diffusion, and the variation of particle density [Lachamp, 2003; Laigle et al., 2007].

2.7 Pressure field smoothing

Because SPH is prone to numerical noise in the pressure field (even with a diffusion term in the mass conservation Eq. (6)), that is mostly due to the propagation of sound waves in the medium, we have to implement ways to filter it. To help remove this noise, the code implements a periodical smoothing: every 20 time steps, the average density of all the
neighbors, i.e. all particles within a distance $2h_i$ (Eq. 3) of a particle $i$ (including particle $i$ itself), is computed and attributed to particle $i$ [Chambon et al., 2011]. In the case of our simulations, the typical periodicity of the smoothing is $10^{-3}$ s. We will see in section 3.4 that the shortest pressure peaks we simulate in the framework of this study have duration greater than $10^{-3}$ s, which is well above. Therefore the pressure smoothing should not impact too badly the capacity of the code to capture the pressure peaks.

2.8 Virtual pressure sensors

One of the goals of this work is to compute the pressure of the fluid in the vicinity of an obstacle. To this effect, we use virtual sensors, initially introduced by Lachamp [2003]. Sensors are constituted of rectangular areas of the simulation domain over which various properties of the fluid are recorded every N time steps. These properties include the average pressure, velocity, density and depth of the fluid. These average values are computed over all the particles located inside the boundaries of the sensor. This is the method we will use to compute the pressure on the wall, either at a given location (in which case the height of the sensor is equal to 1 cm, the height of the physical sensor used in flume experiments by Tiberghien et al. [2007]), or on the whole obstacle (in which case the vertical extent of the sensor matches the height of the wall). The width of the sensor along the x-axis is set equal to 4 $\delta$ in most simulations, so as to encompass enough particles to smooth out pressure instabilities near the wall.

2.9 Evaluation of the model capacities

2.9.1 The dam-break problem

The dam break test case is a classical benchmark used to verify that a code is able to reproduce the propagation of a transient flow of Newtonian fluid. The dam break experiment by Zhou et al. [1999] (using water) also includes impact pressure measurements on a rigid wall. These experimental data, previously compared to numerical results by Colagrossi and Landrini [2003] and Ferrari et al. [2009], are here compared to our own results.

In the experiments of Zhou et al. [1999], a rectangular water tank (Fig. 1) of dimensions $H = 60$ cm and $L = 2H$ is closed by a flap. At $t = 0$ s, the flap is lifted up instantly and the water flows on a dry deck towards a plate located at a distance $x_p = 5.367 H$ of the upstream wall of the tank. Depth probes located downstream, at $x_{H1} = 3.713 H$ and $x_{H5} = 5.542 H$, measure the height of the water. These sensors are capacitive wave gages which are sensitive to the wetted portion of the wire, and may thus indicate a water level lower than the maximum height reached by the water, in the presence of an entrapped air cavity [Colagrossi and Landrini, 2003]. A circular pressure transducer is present on the impact plate. It has a diameter of 90 mm (0.15 $H$) and its center is located 160 mm (0.267 $H$) above the deck. This experiment was reproduced using our code with a particle diameter $\delta = 10$ mm. To simulate water, the density of the fluid was set to $\rho = 1000$ kg/m$^3$. Dimensionless time is defined as $t^* = t(\rho H^3/\mu)$.

The computed height of the fluid at the location of the two depth probes is shown in Fig. 2 along with the experimental measurements and the results of Colagrossi and Landrini [2003] and Ferrari et al. [2009]. The results by Colagrossi and Landrini [2003] show the effective water thickness, obtained by removing the height of entrapped air cavities if present. Ferrari et al. [2009] did not perform this correction. We show both the maximum height water reaches (thick line), and the effective height (thin line). Our code performs as well as the previous SPH implementations, in fact giving near identical results until after the impact with the wall, when a wave flows back towards the left side of the deck (at $t^* \approx 6$). Some discrepancies are found with the experiments, some of which may be caused by the fact that our simulations are only 2D, while 3D effects can appear in breaking waves, as shown by Dalrymple and Rogers [2006].

The pressure computed on the pressure plate is shown in Fig. 3. In our code, the pressure transducer is modeled with a sensor box whose vertical extent is the same as the diameter of the one used in the experiments (90 mm), and its thickness is equal to the particle diameter $\delta = 10$ mm. Once again, our results are pretty similar to previous modeling. We do not recover the first impact (around $t^* \approx 3$) similarly to Colagrossi and Landrini [2003]. It should be noted that the pressure indicated by Ferrari et al. [2009] is measured below the actual center of the pressure transducer (120 mm instead of 160 mm). We do not have their results at the center, but can assume that their implementation would underestimate the pressure, with regards to the experiments. Indeed, if we place our virtual sensor lower, we get higher
velocities and thus obtain a better fit with the experimental data. After the initial peak, the pressure rises on our sensor and gets in close agreement with the experiments. As in previous simulations, the second peak \((t^* \approx 6)\) is late in comparison to the experiment, but we recover its amplitude better than Colagrossi and Landrini [2003]. Another peak \((t^* \approx 8)\) appears in both Colagrossi’s and our simulation, which does not show up in the experimental data. Overall, our data are in good agreement with the experiment, which means our code is capable of simulating a transient flow of Newtonian fluid and its interaction with a rigid structure. It provides a good prediction for the impact pressure and for the movement of the water.

2.9.2 Velocity field in a Herschel-Bulkley fluid steady flow

Another benchmark was performed to test the numerical implementation of the rheological model. This benchmark consists in reproducing a steady flow of a Herschel-Bulkley fluid on a slope, and comparing the computed velocity profile along the z-axis with the analytical solution for a free-surface Poiseuille flow with the same rheology [Chen, 1988].

The rheological characteristics of the fluid are \(\tau_c = 4.9\) Pa, \(K = 3.6\) Pa.s\(^n\), \(n = 0.42\) (see section 3.1). The particle diameter is \(\delta = 1.25\) mm. For a slope of 5° we computed an average fluid height \(h = 3.25\) cm, which is equal to 26.\(\delta\). The theoretical profile corresponding to this fluid height is shown in Fig. 4 along with the numerical results. Because of the finite size of the particles, there is an uncertainty on \(h\). To take this uncertainty into account, analytical solutions for \(h \pm \delta\) are traced in dashed lines. Even when the steady regime is reached, we observe a slight instability of the simulated flow leading to a slight variability on velocity \(u_x\). This variability is taken into account by computing three profiles at a one-second interval. The red dots correspond to the average value of the velocity over the three profiles, while the error bars show the minimum and maximum values.

The agreement is good, considering the small number of particles present in the sheared layer of the fluid (approximately 19). It must also be noted that one of the sources of the uncertainty on \(u_x\) is the sensitivity of the velocity at the free surface to the fluid depth. Indeed, in Fig. 4, we see that a variation of less than 4% in fluid height \(h\) leads to a variation of 20% in the velocity at the free surface. This sensitivity is inherent to shear-thinning fluids \((n < 1)\) and can lead to instabilities in the flow [Coussot, 1994].

3. NUMERICAL EXPERIMENTS

In this section we present the virtual flume and rheological parameter values of the fluid chosen in coherence with laboratory experiments presented in Tiberghien et al. [2007]. We present in a first part the characteristics of the flow, and notably the velocity...
field, upstream of an obstacle. We analyze in a second part the evolution of the pressure applied to the obstacle versus time.

3.1 Virtual experiments layout

Our simulation domain is derived from the experiments described in [Tiberghien et al., 2007]. The experimental setup was constituted of a 30 cm wide inclined flume. A 62 cm long tank was located upstream of the flume and stored some Carbopol, a transparent shear-thinning yield-stress fluid. This tank was closed by a gate which could be instantaneously partially removed to release the fluid. This experimental setup was designed to generate unsteady flows constituted of a steep front followed by a steady flow of constant depth \( h \). A rectangular obstacle of height \( H \) was located downstream of the flume. This height \( H \) was adjusted from one experiment to another so as to maintain a constant aspect ratio \( h/H \approx 0.86 \). This value was arbitrarily chosen but is the order of magnitude of the ratio encountered in the field when considering real debris flows and protection structures. In our simulations, we adopted a simplified representation of the experimental setup constituted of a 2 m long virtual flume (Fig. 5). The obstacle is located at the downstream extremity of the flume. At initial time of the simulation, all SPH particles are stored in the upstream tank. The length of this tank is 62 cm. It is limited downstream by a gate which is partially opened at initial time of the simulation. In practice, the height of the open gate is fixed at 2.5 cm. The fluid depth in this tank is maintained constant (value fixed at 5 cm) by introducing SPH particles during the time of the simulation. This numerical setup made it possible to generate unsteady flows constituted of a steep front followed by a steady flow of constant depth and velocity. Additionally, it was observed that after an initial acceleration phase, the front velocity remains almost constant. The inclination angle is modified by modifying the direction of gravity \( g \). The aspect ratio \( h/H \) is kept equal to 0.86.

Our objective is to study the evolution of the local flow features, in the vicinity of the obstacle, versus time for several values of the inclination angle. All other parameters of the virtual experiments (rheological parameters, height of the gate, and fluid level in the tank) are kept constant. This way, each inclination angle is related to a unique incident flow. In order to get a comparison criterion between numerical and flume experiments, and field flows the following results will be presented not only on the basis of the inclination angle. We’ll indicate also the value of the Froude number of the incident flow. We adopt here the non-classical definition proposed by Tiberghien et al. [2007] in the framework of their experiments:

\[
Fr = \frac{u_{front}}{\sqrt{gh \cos \theta}}
\]

(12)

where \( u_{front} \) is not the fluid velocity but an average velocity of the front between the gate and the obstacle. \( h \) is the depth of the steady flow following the front propagation and \( \theta \) is the inclination angle. Verification has shown that using the time-averaged front speed is roughly equivalent to using the depth-averaged steady flow speed far from the front. Indeed, the difference in the Froude number value is systematically lower than 8%. The Herschel-Bulkley model parameter values are identical to the laboratory experiments: \( \tau_0 = 4.9 \text{Pa}, K = 3.6 \text{Pa.s}^n, n = 0.42 \). The studied inclination angles in the simulations range from 3° to 16°, which corresponds to Froude number values from 0.52 to 2.88, thus of the same order of magnitude as field debris flows.

3.2 Velocity field in the vicinity of the obstacle

Figures 6 to 9 present series of snapshots of simulated flows immediately before and after the impact on the obstacle. They share a common color scale for velocities. In each of these figures, snapshots were recorded every 0.05 s of simulated time.

One can observe that for gentler slopes, between
Fig. 6 Velocity fields at different times of the simulation carried out with an inclination angle $\theta = 4^\circ$, $Fr = 0.71$

3° and 7°, illustrated by Fig. 6, the impact of the fluid on the obstacle is rather gentle. The fluid level goes up gradually along the upstream wall of the obstacle and finally overflows with permanent contact with the upper face of the obstacle. A surface wave looking like a hydraulic bore propagates upstream and a roughly triangular zone of fluid at rest forms upstream of the obstacle. This zone will later be referred to as the dead-zone. The fluid flows over this dead-zone which plays the role of a springboard. This observation is coherent with previous experimental results obtained by Tiberghien et al. [2007]; Armanini and Scotton [1993]; Zanuttigh and Lamberti [2006]. This kind of gentle impact will be later on referred to as the dead-zone impact regime.

One can observe that for steeper slopes, above 8°, illustrated by Fig. 9, the impact on the obstacle is much more violent. A vertical jet forms immediately when the fluid impacts the obstacle. A dead-zone forms upstream of the obstacle but it is much smaller than at gentle slopes. After the first impact a jet is still present over the obstacle and its angle with the horizontal direction diminishes with time and stabilizes when the steady regime is reached. The jet presents no contact with the upper face of the obstacle. This kind of violent impact will be later on referred to as the jet impact regime.

At intermediate inclination angles, between 7° and 8°, illustrated by Fig. 7 and 8, we observe a transition from the dead-zone regime to the jet regime. In Fig. 7, at 7°, the jet formed immediately after the impact partly falls down towards the upstream side of the obstacle while in Fig. 8, at 8°, the jet is sufficiently fast to fully flow over the obstacle.

In Fig. 6 to 9, we observe that, once the steady regime is established, the velocity isolines (stripes with same color in the figures) get more curved and narrow when the inclination increases. Almost parallel to the flume bottom on the last snapshot of Fig. 6, they form an angle of about 20° with the bottom on the last snapshot of Fig. 9. In parallel, the length of the dead-zone diminishes when the inclination increases. Consequently, the size of the dead-zone appears as an important factor of the deviation of the fluid in the vicinity of the obstacle.

3.3 Size of the dead-zone

In this section we aim at examining the evolution of the size of the dead-zone versus the inclination angle. We have indirectly measured the length of the dead-zone $L_{DZ}$ (Fig. 9) after it stabilizes to a constant value when the steady flow conditions are reached. This was done for all the simulations we carried out. In Fig. 10 we considered a dimensionless length given by $L_{DZ}/H$ where $H$ is the height of the obstacle. We observe that at gentle slopes the dead-zone can be very long, up to 25 times the height of the obstacle. When the inclination increases the length of the dead-zone strongly reduces and then almost
Fig. 8 Velocity fields at different times of the simulation carried out with an inclination angle \( \theta = 8^\circ, Fr = 1.53 \)

stabilizes for inclination angles above 8°. A transition is clearly visible in Fig. 10 at 8° which corresponds to a Froude number value \( Fr = 1.53 \)

Fig. 9 Velocity fields at different times of the simulation carried out with an inclination angle \( \theta = 12^\circ, Fr = 2.22 \)

3.4 Evolution of the pressure on the obstacle

In this section, we aim at examining the pressure applied by the fluid on the upstream wall of the obstacle. The pressures are recorded using a virtual pressure sensor as presented in section 2.8. The height of the sensor is identical to the obstacle height and its thickness is equal to 4 \( \delta \) (Fig. 5), \( \delta \) being the initial distance, at the beginning of the simulation, between SPH particles which can also be referred to as the diameter of the particles. This sensor thickness (4 \( \delta \)) has been chosen so that most of the time an average value of the pressure computed on about 100 particles is recorded by the sensor. This size aims at reducing the strong, very local pressure oscillations which may occur when the fluid impacts the obstacle. The main drawback of the technique is that it may underestimate the pressure of strong and short peaks. The pressure recorded by these sensors is recorded all over the simulation.

Figure 11 shows the evolution of the pressure versus time for a series of simulations carried out with an increasing virtual flume inclination. We note that a pressure peak is systematically present when the fluid impacts the obstacle as well as a pressure plateau when the steady regime is established. The respective values of these pressures strongly evolve with the flume inclination. On gentle slopes, in the dead-zone regime, a small peak with duration about 0.01 s is observed at the impact followed by a slow increase of the pressure lasting a few seconds. Between 3° and 6°, the first pressure peak, whose intensity is lower than the pressure of the plateau, grows up with the inclination. In parallel, the pressure of the plateau diminishes with the inclination. From 7° onwards, the peak intensity overcomes the pressure of the plateau. The local minimum in the curve connecting the peak to the plateau disappears at 8°. The duration of the peak increases with slope to reach values about 0.1 s.

These results evidence two different regimes. For the first one the maximum pressure is reached.
Section 3.2 evidenced, on the basis of local flow characteristics in the vicinity of the obstacle, a transition between what we named the dead-zone impact regime, where the dead-zone plays an important role in the impact, and the jet impact regime. We established that this transition occurred for values of the inclination angle between $\theta = 7^\circ$ ($Fr \approx 1.3$) and $\theta = 8^\circ$ ($Fr \approx 1.5$). In section 3.3 we mathematical principles behind the method and the equations it is designed to solve. We explained how we implemented the Herschel-Bulkley rheological model, detailed our choices of implementation and the various features specific to our code: boundary conditions, time-stepping algorithm, pressure field smoothing, and virtual sensors.

We analyzed the capacity of our model to meet the requirements of our study: simulate very unsteady flows impacting a structure and simulate viscoplastic fluid flows. In that aim, we carried out SPH simulations of a dam-break problem with water and compared the results to experimental data, and numerical results from the literature. We also simulated the velocity profile inside the free-surface steady flow of a viscoplastic fluid and compared the results to the theoretical profile. The quality of the results obtained has shown we could use the code with reasonable confidence for modeling a free-surface flow of viscoplastic fluid impacting a structure.

In a second part, we presented our numerical experiments setup derived from the laboratory experiments by Tiberghien et al. [2007]. In the presence of an obstacle, we simulated the local internal velocities immediately upstream of the obstacle at several times of the impact of a viscoplastic fluid wave. We also focused interest on the length of the dead-zone of fluid at rest which form immediately upstream of the obstacle on the pressures developed on the structure during the impact. The results were analyzed in view of the features of the incident flow characterized by its Froude number.

In section 3.2 we evidenced, on the basis of local flow characteristics in the vicinity of the obstacle, a transition between what we named the dead-zone impact regime, where the dead-zone plays an important role in the impact, and the jet impact regime. We established that this transition occurred for values of the inclination angle between $\theta = 7^\circ$ ($Fr \approx 1.3$) and $\theta = 8^\circ$ ($Fr \approx 1.5$). In section 3.3 we
evidenced a transition in the evolution of the length of the dead-zone versus inclination angle at $\theta = 8^\circ$ ($Fr \approx 1.5$). In section 3.4 we evidenced a transition in the features of the impact pressure signal applied to the obstacle between $6^\circ$ ($Fr \approx 1.1$) and $7^\circ$ ($Fr \approx 1.3$). These transitions occur for very similar values of the Froude number of the incident flow. This strongly reinforces the assumption of two different impact regimes where dominant physical processes change from one regime to another. Furthermore, these results are similar to those of Tiberghien et al. [2007] who experimentally evidenced a transition of impact regime for $Fr \approx 1.4$.

REFERENCES


Received: 20 July, 2015
Accepted: 28 March, 2016