Multiphase debris flow simulations with the discrete element method coupled with a lattice-boltzmann fluid

III International Conference on Particle-based Methods – Fundamentals and Applications
PARTICLES 2013
M. Bischoff, E. Oñate, D.R.J. Owen, E. Ramm & P. Wriggers (Eds)

MULTIPHASE DEBRIS FLOW SIMULATIONS WITH THE DISCRETE ELEMENT METHOD COUPLED WITH A LATTICE-BOLTZMANN FLUID

ALESSANDRO LEONARDI, FALK K. WITTEL, MILLER MENDOZA AND HANS J. HERRMANN

Institute for Building Materials, ETH Zurich
Computational Physics for Engineering Materials (COMPHYS)
Schafmattstr. 6, CH-8093 Zurich, Switzerland
E-mail: aleonardi@ethz.ch, web page: http://www.ifb.ethz.ch/comphys

Key words: Debris Flow, Particle-fluid, Hybrid, LBM, DEM

Abstract. Debris flows are dangerous natural hazards that occur mainly in mountainous terrains after heavy rainfall, responsible for casualties and damages reported yearly worldwide. Their heterogeneous composition, with a viscoplastic fluid and the presence of a relevant granular solid phase, leads to a non-trivial behaviour making them a challenging problem both for the physical description of the phenomena and for the design of effective protection measures. A numerical model is developed, fully coupling the two phases. A Discrete Element approach is used for the description of the solid phase, with a realistic particle size distribution, while the fluid phase is solved with a Lattice-Boltzmann Method. The effect of shape on the rolling mechanism is included with a simplified model, as well as the effects of non-Newtonian rheology and the presence of a free surface. The numerical results provide insight into complex segregation, transportation and sedimentation phenomena, essential for understanding and predicting the run-out mechanism of debris avalanches and their interaction with obstacles and retaining structures.

1 INTRODUCTION

Among all landslide phenomena, debris flows are well known for their large destructive effects, with extensive damage to properties and thousands of related casualties reported every year. These fast mass movements of mixed debris and water occur in mountainous terrain after heavy rainfall and can be imagined as liquefied landslides. The simultaneous presence of a fluid and a granular phase determines a non-Newtonian behavior and a counterintuitive rheology. They also show segregation phenomena, which make their physical properties non-uniform, and a non-trivial behavior when interacting with obstacles. To study all these features, numerical frameworks have been proposed in recent years. The flow is usually described either as a continuum [1] or as an ensemble of particles [2].

These models have been used to predict the behavior of dry granular flow, such as rock avalanches. However, a hybrid approach that extends their result to hybrid flows, taking into account their actual composition, is still missing. Recently, there has been growing interest in the development of full fluid-particle coupling models. In particular, the lattice-Boltzmann method (LBM) has been extensively used in this way. Feng et al. [3] developed a computational framework that employed the immersed boundary method for the LBM for the simulation of hybrid particle-fluid mixtures. Švec et al., following a similar approach [4], designed a framework able to simulated hybrid flows with a free surface and including a non-Newtonian constitutive relation for the fluid. Both models have been shown to be accurate and precise for both the description of the rheology properties of the hybrid mixture and the application to real cases. However, one of the drawbacks of the immersed boundary method is the increased computational cost compared to the standard bounce-back rule. This is usually balanced by the use of a coarser grid, since the lattice is not itself used to discretize particles. When particles are used to represent rocks, debris etc. in the model, an accurate description of the surface is not required and the standard bounce-back with a coarse grid can be used without loss of overall predictability. In this work, a 3D numerical framework is designed, focused on the application on the study of debris flow phenomena but also suitable for other fields, such as food processing or building materials. The standard bounce-back rule in the form used first by Ladd [5] and then developed by Aidun and Lu [6] is applied for the description of solid boundaries and for the particle-fluid interaction. This is done by solving only the exterior of the particles with the LBM, which can consistently reduce the computational load for dense particle packings. Moreover, the model is generalized for many non-Newtonian constitutive laws, to represent the rheology of muddy mixtures.

2 DEBRIS FLOWS

Debris flows are natural gravity-driven flows that contain more than one phase. This is due to the special triggering mechanisms that precede the events, usually some accumulation of water close to loose material or heavy rainfall. Much effort has been registered in the past to classify and describe their composition [7, 8]. They are constituted of a mixture of a solid granular material and an interstitial fluid. Their rheological characteristics depend heavily on the composition. Usually the granular phase is present with a complete size distribution, from particles of microscopical size to boulders that in some cases reach the size of meters. Viscosity and inertia can play relatively different roles depending on the size. As a first approximation, the *Bagnold number* can be used to identify this effect. It is defined as

$$\mathrm{Ba} = \frac{\rho_s d_s^2 \lambda_s^{1/2} \dot{\gamma}}{\mu_f}, \tag{1}$$

where ρ_s and d_s are density and diameter of the particles, λ_s is their linear concentration, $\dot{\gamma}$ is the shear rate and μ_f the dynamic viscosity of the flow. Under a threshold value of

the Bagnold number the rheological properties of the flow are dominated by viscosity. A continuum constitutive model can then be used with good approximation to model the mixture. The constitutive model to employ depends on many factors. However, non-Newtonian shear-thinning relationships are considered appropriate by researchers [9]. In particular, the Bingham plastic, a fluid constitutive model with an apparent yield stress σ_B, is widely used. The viscosity is then described by

$$\nu = \nu_{pl} + \frac{\sigma_y}{\dot{\gamma}}, \tag{2}$$

where $\dot{\gamma}$ is the second invariant of the shear rate tensor, and ν_{pl}, σ_y two parameters for the model, namely a plastic viscosity and a yield stress. At high Bagnold numbers, however, the rheology of the flow is dominated by collisional or frictional effects due to granularity. In this case, an explicit representation of particles is preferable.

For the solution of the solid phase a Discrete Element Method (DEM) is used. Only particles large enough to exhibit collisional-frictional behavior are solved explicitly. Smaller particles are incorporated into the rheology model of the fluid. In this way we are able to represent the effect of the full grain size distribution into the model. The fluid phase is solved with the use of a free-surface Lattice-Boltzmann Method (LBM). The two phases are accordingly fully coupled. The method can then be seen as a hybrid of two widely used techniques for the representation of gravitational geological flow, combining advantages of both. Section 2.1 describes the basic concept of the discrete element method used, while Section 2.2 focus on the lattice-Boltzmann solver for the fluid. Finally, Section 2.3 describes the tool used for the coupling of the two phases.

2.1 DISCRETE ELEMENT METHOD (DEM)

DEM is a well-established technique for the solution of particle systems. It has been successfully used in the framework of geohazards to model and predict the behavior of dry granular avalanches. Its main advantage is the possibility to explicitly represent every single constituent of the system, without the use of mean field approximations. Its limitation lies in the number of particle that can be simulated. This is due to the small time step that is necessary to resolve particle interactions.

In the model, particles are represented as rigid spheres. The tracking of 3 translational and 3 rotational degrees of freedom is therefore required in order to study the evolution of the system. Newton's equation of motion are sufficient to represent the dynamics of the system:

$$\ddot{\boldsymbol{x}}_i = \frac{1}{m_i}\boldsymbol{F}_i\left(\boldsymbol{x}_i, \dot{\boldsymbol{x}}_i, \boldsymbol{\phi}_i, \dot{\boldsymbol{\phi}}_i\right), \quad \ddot{\boldsymbol{\phi}}_i = \tfrac{1}{J_i}\boldsymbol{M}_i\left(\boldsymbol{x}_i, \dot{\boldsymbol{x}}_i, \boldsymbol{\phi}_i, \dot{\boldsymbol{\phi}}_i\right), \tag{3}$$

where $\boldsymbol{x}_i$ and $\boldsymbol{\phi}_i$ are position and orientation vectors. Forces and torques are computed from contacts laws. Among many different opportunities, we employ the Hertzian model for viscoelastic spheres. The contact law is symbolized in Figure 1(a). It can be represented as a nonlinear spring coupled with a dashpot for the normal direction. The

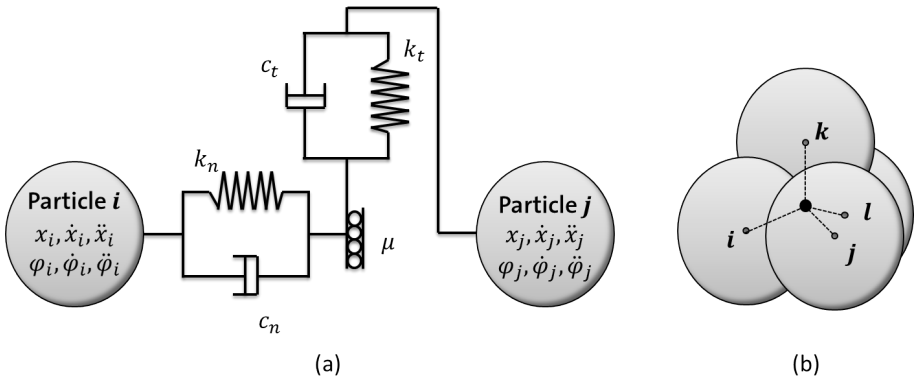


Figure 1: (a) Contact dynamics. Springs k, dashpots c, and Coulomb friction μ are used to represent the law. (b) A tetrahedral composite particle. Components are rigidly linked spheres.

tangential direction features also a friction coefficient. The system of equations that arises is then solved with a predictor-corrector scheme with a 5^{th} order accuracy.

DEM can reproduce efficiently systems composed of spherical particles. However, physical problems such as debris flows require a realistic representation of natural materials, such as stones or gravel. We chose to employ composite particles since they have to the same dynamics and contact resolution as spherical particles, but are composed of a number of rigidly connected spheres. The main advantage is that the rolling mechanism of natural stony material is properly recovered, without adding any further model for the dynamics of contacts. An composite particle example can be seen in Figure 1(b), where a tetrahedral cluster is represented.

2.2 LATTICE-BOLTZMANN METHOD (LBM)

LBM was developed over the last two decades to become an alternative to the simulation of partial differential equations. It is based on a regular time and space discretization of the Boltzmann equation for the kinematics of gases on a cubic lattice. Both lattice cell width and time step are set as unitary, so that all equations appearing below are dimensionless. The center of a cell is the location where physical variables are referenced, and is called node.

LBM differs from traditional computational fluid dynamics methods since the only dependent variable is the *probability density function*, $f_i(\boldsymbol{x}, t)$. It is defined as the probability of finding particles with a given velocity $\boldsymbol{c_i}$ at a certain range of locations at a given time. Due to the regular discretization in time and space, a limited set of microscopic velocities c_i is chosen according to the adopted lattice configuration. In this work, a lattice in 3 dimensions with 19 directions (the D3Q19) is used, since it is the most simple 3D lattice that ensures conservation of all symmetries required for incompressible fluid dynamics simulations. Macroscopic variables such as local density and velocity can then be defined from the two first moments of the distribution functions

$$\rho_f(\boldsymbol{x},t) = \sum_i f_i(\boldsymbol{x},t), \quad \boldsymbol{u}_f = \sum_i f_i(\boldsymbol{x},t)\boldsymbol{c}_i/\rho_f(\boldsymbol{x},t). \tag{4}$$

The evolution of the density functions is governed by the Boltzmann equation written in the lattice reference system as

$$f_i(\boldsymbol{x}+\boldsymbol{c}_i, t+1) = f_i(\boldsymbol{x},t) + f_i^F(\boldsymbol{x},t) + \Omega_i(\boldsymbol{x},t). \tag{5}$$

The term f^F is the contribution of external force fields, while Ω is the collision operator, representing the effects of collisions between particles in the fluid. This term is difficult to solve precisely, and in most cases an approximated form is used. The most common way to do this is substituting the analytical collision term with the so-called Bhatnagar-Gross-Krook operator [10], which is a linear approximation given by:

$$\Omega_i(\boldsymbol{x},t) = \frac{f_i^{eq}(\boldsymbol{u}_f,\rho_f) - f_i(\boldsymbol{x},t)}{\tau(\boldsymbol{x},t)}. \tag{6}$$

The value of the *relaxation time* τ is determined accordingly to the local viscosity of the fluid as:

$$\tau(\boldsymbol{x},t) = 1/2 + 3\nu(\boldsymbol{x},t). \tag{7}$$

The equilibrium functions f_i^{eq} are defined with a small-velocity expansion of the Maxwell-Boltzmann distribution from the local macroscopic velocity $\boldsymbol{u}_f$ and density ρ_f:

$$f_i^{eq}(\boldsymbol{u}_f,\rho_f) = \rho_f w_i \left(1 + 3\boldsymbol{c}_i\cdot\boldsymbol{u}_f + \frac{9}{2}\left(\boldsymbol{c}_i\cdot\boldsymbol{u}_f\right)^2 - \frac{3}{2}\boldsymbol{u}_f\cdot\boldsymbol{u}_f\right), \tag{8}$$

where w_i are the weights that arise from mass and momentum conservation. They are dependent on the lattice geometry and in our case are:

$$w_i = \begin{cases} 1/3 & \text{for } i = 1 \\ 1/18 & \text{for } i = 2,\ldots,7 \\ 1/36 & \text{for } i = 8,\ldots,19. \end{cases} \tag{9}$$

In this form the LBM can be used for discrete realizations of kinetic models for fluids.

To represent the solid phase, lattice nodes are divided into active and solid nodes, as can be seen in Figure 2(b). Streaming and collision are solved only for active nodes. Solid boundaries are located halfway between a solid and an active node and are solved with the use of the bounce-back rule, as developed first by Ladd [5] and then by Aidun and Lu [6]. Every time a density function is streaming towards a solid node, it gets reflected back in the opposite direction. A correction in the value of the bounced distribution function must be taken into account, due to the exchange of momentum between solid and fluid. This is done by imposing that

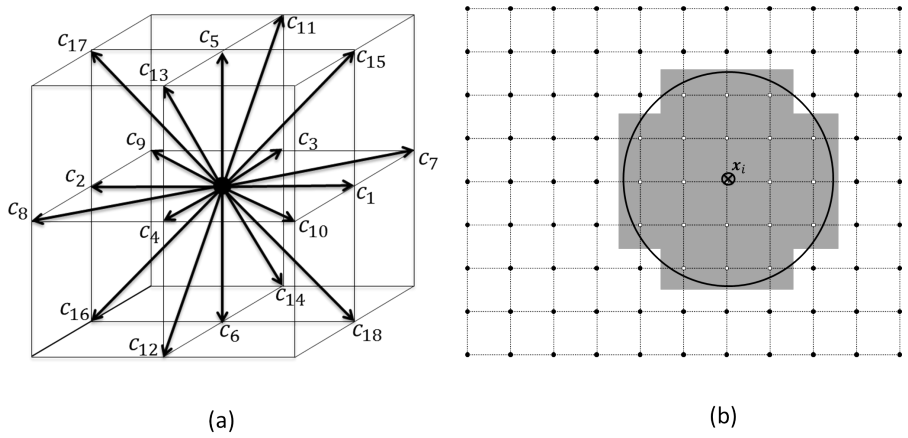


Figure 2: (a) Sketch of the D3Q19 lattice. Arrows represent the chosen set of velocities c_i. (b) Discretization of solid particles. The continuous line represent the ideal sphere, while the grey area represent the solid zone as in the fluid solver. Solid nodes are represented as empty squares, while fluid nodes as full black dots.

$$f_{i'}(\boldsymbol{x} + \boldsymbol{c}_{i'}, t+1) = f_i(\boldsymbol{x}, t) - 2w_i\rho\boldsymbol{u}_w \cdot \boldsymbol{c}_i, \tag{10}$$

where i' is the streaming direction opposite to i and u_w is the local velocity of the wall at the bounce-back location. The same momentum exchange is used to compute the hydrodynamic force on the particle.

A free surface is included in the model and the mass-tracking algorithm is used to represent it. In its general formulation, this technique can track the evolution of the surface between two immiscible fluids. However, when one of the two component has a negligible effect on the dynamics of the system, it can be neglected. This is the case of debris flows, where the gas phase has little to no influence. Moreover, for this class of flows the length scale is large enough to make superficial tension negligible, too. Under these conditions, the free surface can be seen as a special moving boundary condition with imposed pressure. Following Mendoza et al. [11], active nodes are further divided into fluid and interface nodes. Gas nodes represent points located outside the surface of the fluid. The lattice is organized so that an interface node is always located between a gas and a fluid node. An additional variable representing the mass is added to the system. For fluid nodes, it coincides with the density, and there is no modification to the scheme so far described. For interface nodes it represent the degree of filling. When the mass reaches the value of the local density ($m(\boldsymbol{x}, t) \geq \rho(\boldsymbol{x}, t)$), the interface node is converted to a fluid cell. When, on the other hand, the mass is less or equal to zero ($m(\boldsymbol{x}, t) \leq 0$), the interface node is converted to gas. The equation describing the evolution of the mass is

$$m(\boldsymbol{x}, t+1) = m(\boldsymbol{x}, t) + \sum_i \alpha_i \left[f_{i'}(\boldsymbol{x} + \boldsymbol{c}_i, t), -f_i(\boldsymbol{x}, t) \right], \tag{11}$$

where the parameter α_i is dependent on the neighbor node type in the i direction

$$\alpha_i = \begin{cases} \frac{1}{2}\left[m(\boldsymbol{x},t) + m(\boldsymbol{x}+\boldsymbol{c}_i,t)\right] & \text{if neighbor is interface} \\ 1 & \text{if neighbor is fluid} \\ 0 & \text{if neighbor is gas.} \end{cases} \tag{12}$$

This technique can be proved to conserve mass exactly. However, due to the discrete time step, when an interface node is turned into a fluid cell, some mass would be destroyed. To avoid this, the surplus is equally distributed among neighbor active nodes. The same is done with the mass deficit arising from interface nodes that are turned into gas.

The last ingredient of the framework is a non-Newtonian rheology constitutive law. This is a necessary step to correctly represent the physical behavior of the system, as seen in Section 2. To do this, a multiple relaxation time is adopted. This implies that the viscosity is not a global parameter, but is locally defined from the strain state of the node. Following the approach of Švec et al. [4], the shear rate tensor can be computed as

$$\gamma_{ab}(\boldsymbol{x},t) = \frac{3}{2\tau(\boldsymbol{x},t)} \sum_i \boldsymbol{c}_{i,a}\boldsymbol{c}_{i,b} f_i(\boldsymbol{x},t). \tag{13}$$

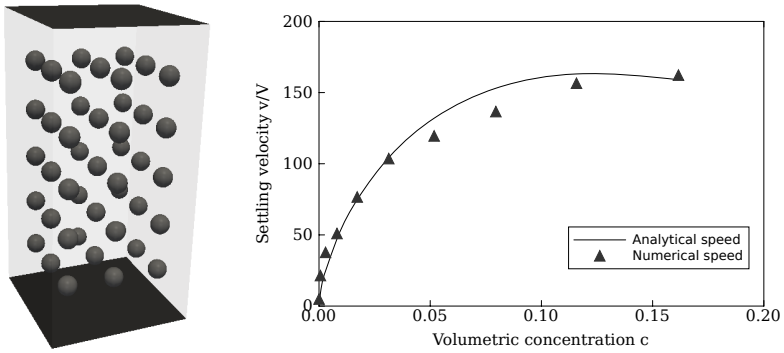
According to Equations 2 and 7, a Bingham fluid model for viscosity can then be implemented with

$$\tau(\boldsymbol{x},t+1) = \frac{1}{2} + 3\left(\nu_B + \frac{\sigma_B}{\dot{\gamma}(\boldsymbol{x},t)}\right), \tag{14}$$

where $\dot{\gamma}(\boldsymbol{x},t)$ is the local second invariant of the shear rate tensor. In order to ensure accuracy and stability, the relaxation time must fall into a range of allowed value. A condition must then be added to Equation 14, namely that $\tau_{min} \leq \tau(\boldsymbol{x},t+1) \leq \tau_{max}$.

2.3 FLUID-STRUCTURE INTERACTION

Fluid-particle coupling is achieved with consecutive iterations of the DEM solver and the LBM one. Since in general the time step of the DEM can be different (and usually smaller) than the LBM one, a number of DEM iterations can be performed between two LBM steps. Firstly, the fluid lattice is updated according to the current position of the solid particles. Each lattice node is assigned its properties and boundary conditions according to the properties of its neighbors. Now, collision and streaming steps are performed. The bounce-back rule is applied at all solid boundaries and at the free-surface interface. At the same time, it gives all information necessary to compute the hydrodynamic forces and moments on the particles. This is done by computing the momentum exchange that occurs at the bounce-back locations. In addition, the mass streaming is calculated for all interface nodes. The mass update allows the computation of the mass, and the consequent definition of the new free surface. The DEM solver is then called. Predicted values for position, velocity, acceleration, and other derivatives if necessary, are computed based on previous time step values. These values are then used



(a) (b)

Figure 3: (a) Simulation geometry. Particle subject to gravity are settling in a laminar fluid. (b) Analytical and numerical dimensionless settling velocity.

to calculate forces and moments generated by colliding particles. These forces are then summed with the hydrodynamic contributions coming from the fluid solver. If particles are close to contact, the LBM is not precise enough to reproduce effects due to lubrication. A correction between pairs of particles i and j closer than a threshold value d^{lub} is included as

$$F_{i,j}^{lub}(t) = -6\nu\rho_f \left(\boldsymbol{u}_i(t) - \boldsymbol{u}_j(t)\right) \cdot \boldsymbol{n}_{i,j} \frac{r_i^2 r_j^2}{(r_i + r_j)^2} \left(\frac{1}{d_{i,j}} - \frac{1}{d^{lub}}\right). \tag{15}$$

At this point the corrector can be called and the positions of particles are updated. This can then be used to update the lattice and start a new LBM step.

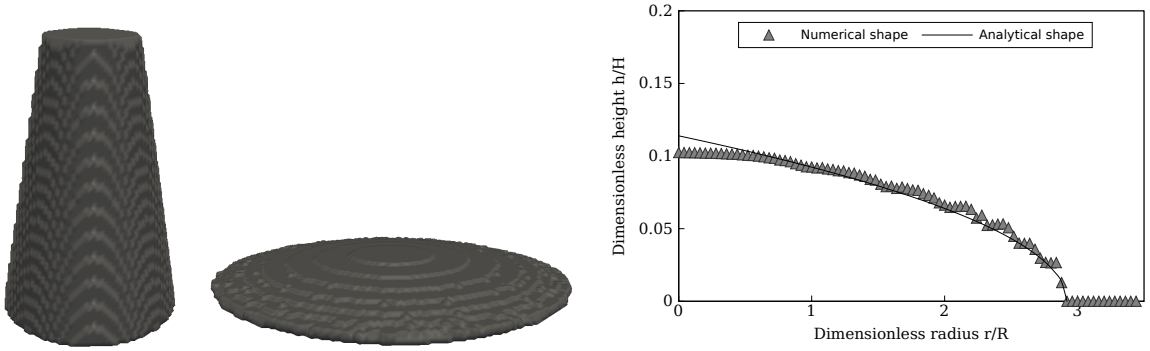
3 VALIDATION

Two series of tests were performed to study the applicability of the model to different scenarios. The first test is meant to validate the fluid-particle interaction. The settling of particles in a Newtonian fluid is investigated. The Stoke's law for hydrodynamic drag force in laminar viscous fluid is a well-known result of fluid mechanics. It leads to a relation for the settling velocity of one spherical particle in a fluid subjected only to a force field:

$$\boldsymbol{v}_{set} = \frac{2}{9} \frac{\rho_s - \rho_f}{\mu_f} \boldsymbol{f}_{ext} R^2, \tag{16}$$

where $\boldsymbol{v}_{set}$ is the stationary settling velocity, μ the dynamic viscosity of the fluid, $\boldsymbol{f}_{ext}$ denotes the external homogeneous force field, and R the radius of the particle.

Stoke's law for the settling velocity is valid only for single particle systems. Several law generalize its validity to systems with many particles. Richardson and Zaki [12] provided a relation which, despite its simplicity, is valid for a wide range of conditions. It can be



(a) (b) (c)

Figure 4: (a)(b) Cone shape before and after the slump. (c) Analytical and numerical shape comparison.

expressed as:

$$\boldsymbol{v}_{set,c} = \boldsymbol{v}_{set}\,(1-c)^{4.65}\,, \tag{17}$$

where c is the volumetric concentration of particles and $\boldsymbol{v}_{set,c}$ the settling velocity of the ensemble. Our test is set using a regular grid of particles immersed in a fluid and subject to gravity in the vertical direction. Figure 3 (a) shows the geometry of the test. The density of the particles is twice the density of the fluid. The simulations parameters are set to obtain a flow in the laminar regime. The simulation box has bounce-back boundary conditions at the upper and lower boundaries; the other four boundaries have periodic conditions. Particles are positioned in a regular grid. The volumetric concentration of particles is varied by changing the particle size:

$$c = l^3 / \left(\frac{4}{3}\pi R^3\right), \tag{18}$$

where l is the grid space of the particle distribution. Figure 3 (b) shows the comparison between the analytical prediction and the numerical results for the settling velocity of the ensemble. Values for velocity are scaled with the settling velocity of the smallest particle alone, V. The agreement is good and tends to prove that an approximated approach for the resolution of particles gives acceptable results for practical applications.

The second test is meant to validate both the mass-tracking algorithm for the representation of the free surface and the Bingham constitutive model. Following Švec et al. [4] we simulate the collapse of a conical sample of plastic fluid. This geometry is used also in practice to test the workability of concrete and is called *slump test*. The shape of the sample after deformation can be linked to the properties of the fluid and in particular to the yield stress. An analytical description of the flow is provided in [13]. At the limit when the radius-height ratio r/h of the sample is large, a pure shear flow can be a good approximation. At this limit, only the tangential stress is relevant. For this case, an

analytical expression exists for the final shape of the sample:

$$h\left(r\right) = \left(\frac{2\sigma_y \left(R - r\right)}{\rho_s F_{ext}}\right)^{1/2}. \tag{19}$$

The slump test was reproduced using the same parameters as in [4]. The numerical results are shown in Figure 4. Two snapshot of the simulation show the free surface before and after the deformation. In addition, the numerical shape of the surface is plotted together with the analytical one. Values are scaled using the initial height H and the initial basic radius R. The good agreement tends to prove the validity of the used method.

4 CONCLUSIONS

In this paper we described a method for the simulation of hybrid particle-fluid systems. The framework takes into account the presence of a free surface and can deal with different non-Newtonian models for the fluid. Due to the intrinsic locality of the lattice-Boltzmann method and its advantages over traditional CFD solvers, parallelization is naturally implemented. The biggest strength of the method lies in its simplicity and velocity. The scheme adopted for the fluid interaction with particles or other solid boundaries is the simple bounce-back scheme. This allows for a fast calculation of the hydrodynamic force exerted by the fluid on particles and works particularly well for dense systems, since only the exterior of the particles is discretized. The drawback of this method is that the same lattice is used for both, the solution of the fluid and the discretization of the particles. Said this, for systems where a correct representation of the shape of particles is not essential, such as geoflows, the method finds its most successful application. It can then be used without a loss of accuracy on coarse lattices. Other methods were developed in the last years for a better resolution of particles [3, 4], but we believe that a whole class of flows exist, where this is not necessary and can turn to be counterproductive. The mass-tracking algorithm has been used for the resolution of the free surface. This allows for a fast though accurate description of a variety of free-surface flows, where surface tension effects can be neglected. A whole series of non-Newtonian formulation for the fluid can be implemented. We included the Bingham plastic, a rheology model that can be used to describe natural flows. The method has been tested on the settling of particles. A monodisperse mixture under the effect of gravity has been simulated, showing a good agreement with the Richardson-Zaki law. In addition, the non-Newtonian formulation has been validated simulating a slump test, finding good agreement with analytical results and a correctly represented surface. The method has the capability to simulate real systems, like a debris flow. The method is quite successful and can be applied to a huge variety of physical systems. The most used models for the study of geoflows so far never explicitly resolved the interaction between solid and fluid phase without the adoption of severe simplistic assumptions. The main limitation of the method lies in the small range of turbulent regimes that can be simulated, namely only from low to moderate Reynolds

numbers. A successive improvement would be the addition of a turbulence model, to extend its applicability to a wider range of cases.

REFERENCES

[1] S. Savage and K. Hutter, "The motion of a finite mass of granular material down a rough incline," *Journal of Fluid Mechanics*, 1989.

[2] C. S. Campbell, P. W. Cleary, and M. Hopkins, "Large-scale landslide simulations: Global deformation, velocities and basal friction," *Journal of Geophysical Research*, vol. 100, no. B5, p. 8267, 1995.

[3] Y. T. Feng, K. Han, and D. R. J. Owen, "Coupled lattice Boltzmann method and discrete element modelling of particle transport in turbulent fluid flows : Computational issues," *International Journal for Numerical Methods in Engineering*, vol. 72, no. 1, pp. 1111–1134, 2007.

[4] O. Švec, J. Skoček, H. Stang, M. R. Geiker, and N. Roussel, "Free surface flow of a suspension of rigid particles in a non-Newtonian fluid: A lattice Boltzmann approach," *Journal of Non-Newtonian Fluid Mechanics*, vol. 179-180, pp. 32–42, July 2012.

[5] A. J. C. Ladd, "Numerical simulations of particulate suspensions via a discretized Boltzmann equation. Part 1. Theoretical foundation," *Journal of Fluid Mechanics*, vol. 271, pp. 285–309, 1994.

[6] C. K. Aidun and Y. Lu, "Lattice Boltzmann simulation of solid particles suspended in fluid," *Journal of Statistical Physics*, vol. 81, no. 1-2, pp. 49–61, 1995.

[7] O. Hungr, S. G. Evans, M. J. Bovis, and J. N. Hutchinson, "A Review of the Classification of Landslides of the Flow Type," *Environmental and Engineering Geoscience*, vol. 7, no. 3, pp. 221–238, 2001.

[8] T. Takahashi, *Debris Flow, Mechanics, Predistion and Countermeasures*. London: Taylor & Francis, 2007.

[9] P. Coussot, *Mudflow rheology and dynamics*. A. A. Balkema, Rotterdam, 1997.

[10] P. L. Bhatnagar, E. P. Gross, and M. Krook, "A Model for Collision Processes in Gases. I. Small Amplitude Processes in Charged and Neutral One-Component Systems," *Physical Review*, vol. 94, no. 3, pp. 511–525, 1954.

[11] M. Mendoza, F. K. Wittel, and H. J. Herrmann, "Simulation of Flow of Mixtures Through Anisotropic Porous Media using a Lattice Boltzmann Model," *The European Physical Journal E*, vol. 32, pp. 339–348, 2010.

[12] J. Richardson and W. Zaki, "The sedimentation of a suspension of uniform spheres under conditions of viscous flow," *Chemical Engineering Science*, vol. 3, pp. 65–73, Apr. 1954.

[13] N. Roussel and P. Coussot, "Fifty-cent rheometer for yield stress measurements: From slump to spreading flow," *Journal of Rheology*, vol. 49, no. 3, p. 705, 2005.