

# SPH NUMERICAL APPROACH IN MODELLING 2D MUDDY DEBRIS FLOW

L. MINATTI(\*) & A. PASCULLI(\*\*)

(\*)University of Florence, Department of Civil and Environmental Engineering, Italy

(\*\*)University G. d'Annunzio Chieti-Pescara, Department of Science, Italy

## ABSTRACT

2D muddy debris flow has been simulated according to a dam break like problem along a slope. The two sets of equations related to the fluid and solid phases, as considered by the debris flow mixture theory, have been simplified in only one set of equations, considering just one equivalent material. Then the Herschel-Bulkley fluid constitutive equations have been selected. The correct parameters of the HERSCHEL-BULKLEY model have been chosen in order to correctly simulate the behaviour of mudflows. The final mathematical model, has been solved numerically with the smoothed particle hydrodynamics (SPH) method. SPH is a particle mesh-free Lagrangian method, well suitable for computing highly transitory free surface flows of complex fluids in complex geometries. Finally a laboratory experimental test has been selected for comparison. Satisfactory results have been achieved. Nevertheless, further parametric analyses will be carried out and further considerations about both constitutive equations and numerical improvements will be employed and discussed in future papers.

**KEY WORDS:** SPH, 2D numerical modelling, muddy debris flow

## INTRODUCTION

Debris flows are characterized by a mixture of water and poorly sorted granular material flowing under the effect of gravity (PIERSON *et alii*, 1987; TAKAHASHI, 1991; COUSSOT *et alii*, 1996; IVERSON, 1997; CROSTA *et*

*alii*, 2001). The most widely used approaches to model debris flows are usually related either to “continuum” or to “granular” mechanics. A combination of them can sometimes be used, depending on the type of debris flow under study. The content of this paper belongs to the former kind of approach where the Navier-Stokes equations are considered as the master tool. In the “continuum” framework, a quite general mathematical model, that can be used to model a physical phenomenon like the one discussed in this paper, would consist of two sets of equations: a first set for the liquid phase and a second one for the solid phase. Interactions terms can then be added to the equations in order to model the forces exchanged by the two phases. A model of this kind can be derived from the mixture theory (ATKIN *et alii*, 1976): an implementation of such theory in the case of debris flows can be found in IVERSON (1997), where the possibility of using different rheological models for each phase is suggested.

The grain size distribution plays an important role in the physics of debris flows and recent developments in the study of water-solid mixtures have shown that the clay fraction can be of great importance too, as it influences the granular interactions. In particular, it is reported by LAIGLE *et alii* (1997) that debris flow mixtures with a high clay fraction, despite retaining their two-phase nature, behave like viscoplastic fluids and that a single phase model of such kind can reproduce their physics to a good degree of approximation. We then restrict our analysis to the specific case of

debris flows characterized by a high clay fraction (mudflows) and simplify the problem by employing a single equivalent phase model of a viscoplastic fluid.

## GOVERNING EQUATIONS

In light of the considerations reported in the introduction, we are therefore interested in the equations of motion of an incompressible non-Newtonian fluid. In order to develop a theoretical framework for a numerical model we resort to the basic principles governing the motion of a continuum, namely the mass conservation and the momentum equations. They can be written in Lagrangian form as:

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \underline{v} = 0 \quad (1)$$

$$\rho \frac{Dv}{Dt} = \rho \underline{f} + \nabla \cdot \underline{\sigma} \quad (2)$$

where:

$\underline{v}$  is the local velocity of the continuum

$\rho$  is the local density of the continuum

$\underline{f}$  is the body force per unit of mass exerted on the continuum;

$\underline{\sigma}$  is the local total stress tensor;

The total stress tensor is usually split into two parts: an isotropic and a deviatory one. The stress tensor decomposition is indicated as follows in the paper:

$$\underline{\sigma} = -p\underline{I} + \underline{\tau} \quad (3)$$

where:

$p$  is the isotropic pressure;

$\underline{I}$  is the unit tensor;

$\underline{\tau}$  is the deviatory part of the total stress tensor;

The isotropic part of the tensor depends on the pressure, while the deviatory part of the total stress tensor is usually expressed as a function – namely the constitutive law - of the strain rate tensor  $\underline{D}$ :

$$\underline{D}_{ij} = \frac{1}{2} \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) \quad (4)$$

The constitutive law used throughout the paper to simulate the behaviour of a mudflow is the Herschel-Bulkley law, which has also been recently used by LAIGLE *et alii* (2007):

$$\eta = K \sqrt{2Tr(\underline{D}^2)}^{(n-1)} + \frac{\tau_c}{\sqrt{2Tr(\underline{D}^2)}} \quad (5)$$

$$\underline{\tau} = 2\eta \underline{D} - \frac{2}{3} \nabla \cdot \underline{v} \underline{I} \quad \text{if } \sqrt{\frac{Tr(\underline{\tau}^2)}{2}} > \tau_c \quad (6)$$

$$\underline{D} = \underline{0} \quad \text{if } \sqrt{\frac{Tr(\underline{\tau}^2)}{2}} \leq \tau_c$$

where:

$\eta$  is the local dynamic viscosity of the fluid;

$\tau_c$  is the yield stress;

$K$  is called the liquid consistency;

$n$  is called the power law index;

$\underline{D}$  is the strain rate tensor of eqn. (4).

$\tau_c$ ,  $K$ , and  $n$  represent the constitutive law parameters: their values will be adjusted according to experiments (see Tab. 1) in order to correctly reproduce the behaviour of a mudflow. The physical meaning of the yield stress is immediate, representing the stress threshold below which the fluid starts to behave like a rigid body.

## NUMERICAL MODELLING

The physics and the complexity of debris flow phenomena require the selection of an appropriate numerical approach. The related bibliography is very extensive and, for the sake of completeness, we mention the fields of Computational Fluid Dynamics (CFD) and of Computational Granular Dynamics (CGD). CFD (CHUNG, 2006), among many others) and CGD (PÖSCHEL & SCHWAGER, 2005) are considered, respectively, as numerical tools related to continuum mechanics and granular mechanics. Molecular Dynamics is a further numerical tool used to study the interactions among particles, exploiting essentially Newton law related to both their translations and rotations. Nevertheless, according to the scope of the present paper, an appropriate numerical method, among the ones suitable for solving the Navier-Stokes equations, should be selected. In these regards, many approaches have been proposed: in particular, techniques capable of handling free surfaces and violent mass fluxes

|   |                        |
|---|------------------------|
| density ( $\rho_0$ )                              | 1410 kg/m <sup>3</sup> |
| yield stress ( $\tau_c$ )                         | 19 Pa                  |
| power law index ( $n$ )                           | 1/3                    |
| liquid consistency ( $K$ )                        | 3.5 Pa·s <sup>n</sup>  |
| Flume slope                                       | 21 %                   |
| Initial height of material (measured at the gate) | 14 cm                  |

Tab. 1 - Rheological characteristics and experimental parameters in LAIGLE *et alii* (1994) and LAIGLE (1997)

appear to be suitable for solving muddy debris flows problems.

Finite Elements (FEM), Finite Volumes (FVM), Finite Differences (FDM) are some of the most common used methods (CHUNG, 2006). For all the mentioned approaches, grid generation is an important step to be performed. And as far as grid generation is concerned, the possibilities are numerous: structured grids, unstructured grids and adaptive meshes.

Other important issues related to the numerical solutions of fluid flows problems modelled with the Navier-Stokes equations involve incompressibility and convective terms. In order to avoid the latter one, a Lagrangian approach is commonly selected instead of an Eulerian one.

Furthermore, particular care should be taken concerning the numerical issues related to large deformations. This is the main reason why a frequent (time consuming) update of the grid, aimed at lowering the excessive mesh distortion due to large deformations, is often necessary.

From the above discussion, it is clear that methods that avoid the numerical instability due to the convective terms, lower the grid generation time and that are capable of easily taking free surfaces into account, are very desirable. The “Meshless Finite Element Methods” include many of the above necessary discussed features (see IDELSOHN *et alii*, 2003; 2004; for an overview). The “Smoothed Particle Hydrodynamics” (SPH), briefly described in the following paragraphs and selected in our approach is among these. A similar tool, the “Particle Finite Element Method” (PFEM) (OÑATE *et alii*, 2004) is very promising as it includes all the advantages of the FEM without some of the difficulties of the SPH method (in particular, the imposition of boundary conditions), even if it requires some time, (which can still be reduced by using an extended Delaunay method), to update grid meshes.

Finally, it is worth to mention other approaches, reliable for granular debris flow, like the “Distinct Elements” and the “Alternate Lagrangian Eulerian” method (ALE) (CROSTA *et alii*, 2001).

## OVERVIEW OF THE SPH METHOD

The Smoothed Particle Hydrodynamic (SPH) method is a numerical technique that was initially developed during the 1970s to solve astrophysical problems (MONAGHAN, 2005). It is a fully meshless particle

method that is easy to code. Its meshless and Lagrangian nature make it very attractive for solving fluid flow problems where free surface boundary conditions and large strain rates are involved. The computational domain is filled with particles carrying flow field information (e.g. pressure, velocity, density) and capable of moving in space.

Particles are the computational frame used in the method to solve the flow describing PDEs, as a grid or a mesh to calculate spatial derivatives is not needed.

We shall refer to 2D cases throughout the rest of the paper, even though all the assumptions and results can be extended to a 3D case with little effort. The key idea on which the method is based is the well-known use of a convolution integral with a Dirac delta function to reproduce a generic function  $f(x)$ :

$$\hat{f}(x) = \int_D f(x') \cdot \delta(x - x') dx' \quad (7)$$

In the SPH method, the Dirac function is replaced by a “bellshaped” kernel function  $W$  (it ‘mimics’ the Dirac delta function), and the generic function  $f(x)$  is reproduced with a convolution integral, which in a discrete framework takes the form of a summation over particles:

$$\hat{f}(x_i) = \sum_{j=1}^n f(x_j) \cdot W(x_i - x_j) \cdot \Delta A_j \quad (8)$$

where:

$x_i$  and  $x_j$  represent the  $i$  and  $j$  particle positions in the given frame of reference;

$\Delta A_j$  represents the tributary area (or volume in a 3D case) associated with particle  $j$ ;

Summation is extended to all the particles located within the support domain of particle  $i$ .

The kernel function is chosen to be non negative, even and with a support domain  $\Omega_x$  (usually circular) whose radius is a multiple of a length  $h$ , named *smoothing length*. The kernel function is zero outside the support domain and the smoothing length serves as a scaling parameter for its arguments. It also has the property of converging to the Dirac function as the smoothing length approaches to zero.

It is possible to obtain the expression for the SPH approximation of a function gradient by writing the convolution integral of the function with the kernel and by using the Gauss-Green formula:

$$\nabla \hat{f}'(\underline{x}) = \int_{\Omega_i} \nabla f(\underline{x}') \cdot W(\underline{x} - \underline{x}') d\underline{x}' = \int_{\partial\Omega_i} f(\underline{x}') \cdot W(\underline{x} - \underline{x}') \cdot \underline{n} ds - \int_{\Omega_i} f(\underline{x}') \cdot \nabla W(\underline{x} - \underline{x}') d\underline{x}' \quad (9)$$

where:

the kernel is differentiated with respect to the  $\underline{x}'$  coordinate;

$\underline{n}$  represents the normal to the support domain boundaries (pointing outward).

The first term of the RHS of eqn. (9) is zero if the support domain isn't truncated by the computational domain boundaries, as the kernel is zero on the support domain boundaries. Another case when the term can be zero is when the support domain is truncated by the computational domain boundaries but there exists a boundary condition forcing the function  $f(\underline{x})$  to vanish on the boundaries (it may be the case when  $f(\underline{x})$  represents a velocity and a no-slip condition has to be enforced on the computational domain boundaries). If the first term of the RHS of eqn. (9) is zero, then the SPH approximation of  $f(\underline{x})$  gradient takes the following form in a discrete framework:

$$\nabla \hat{f}_h(\underline{x}_i) = - \sum_{j=1}^n f(\underline{x}_j) \cdot \nabla_j W(\underline{x}_i - \underline{x}_j) \cdot \Delta A_j \quad (10)$$

Eqn. (10) is often used, even when the first term of the RHS of eqn. (9) doesn't vanish. Nevertheless, it is possible to offset the errors induced by neglecting the term by introducing special treatments of the boundaries, as detailed later.

There are consistency conditions that the kernel has to satisfy in order to correctly reproduce continuous field functions up to a certain order of accuracy. They are related to the kernel moments and should be satisfied both in the continuous and in the discrete frameworks. More details can be found in LIU *et alii* (2003) & in LIU *et alii* (2006).

Numerical applications of SPH share these common features: The equations describing the continuum motion are written in Lagrangian form, by using Lagrangian instead of Eulerian time derivatives.

The spatial gradients involved in the PDEs are discretized over the particles, by using a SPH approximation.

Time integration of variables is performed particle-wise usually by using an explicit time stepping method.

Particles positions are evolved in time, according to the velocity of each particle.

## SPH DISCRETIZATION TECHNIQUE

### MAIN EQUATIONS

The equation briefly presented in the overview paragraph can be used to discretize the Herschel-Bulkley fluid flow governing equations in order to create an SPH algorithm.

There are many references where it is possible to find details on how fluid flow governing equations can be effectively discretized into SPH equations. Among the many of them, we indicate MONAGHAN (2005), LIU *et alii* (2003) & TAKEDA *et alii* (1994).

The SPH discretization of the mass conservation equation (1) used in the paper is as follows:

$$\frac{D\rho_i}{Dt} = \sum_{j=1}^n m_j \cdot \underline{v}_{ij} \cdot \nabla_i W_{ij} \quad (11)$$

where:

$m$  represents the particle mass;

$\underline{v}_{ij} = \underline{v}_i - \underline{v}_j$  represents the difference between the interacting particles velocity;

A widely used SPH discretization of the momentum equation (2) is as follows:

$$\frac{Dv_i}{Dt} = \underline{f}_i - \sum_{j=1}^n m_j \cdot \left( \frac{\rho_i}{\rho_j} + \frac{\rho_j}{\rho_i} \right) \cdot \nabla_i W_{ij} + \sum_{j=1}^n m_j \cdot \frac{8\eta_{ij}}{\rho_i \rho_j} \cdot \frac{\underline{v}_{ij} \cdot \underline{r}_{ij}}{\|\underline{r}_{ij}\|^2} \cdot \nabla_i W_{ij} \quad (12)$$

where:

$\underline{r}_{ij} = \underline{r}_i - \underline{r}_j$  represents the difference between the interacting particle position;

$\eta_{ij}$  is a symmetrised dynamic viscosity between interacting particles, such that  $\eta_{ij} = \eta_{ji}$ .

Usually, a symmetrised expression for the smoothing lengths of each pair of interacting particles in the summations is used when calculating the kernel gradient (e.g.  $h_{ij} = (h_i + h_j)/2$ ). If this is the case, it can be shown that particles exchange equal and opposite forces, making eqn. (12) capable of conserving the linear momentum of the particles system.

In the paper, the following expression has been used for the symmetrised viscosity:

$$\eta_{ij} = \frac{2\eta_i \eta_j}{\eta_i + \eta_j} \quad (13)$$

The term in the right hand side of eqn. (12) involving the symmetrised viscosity represents the deviatoric stress tensor divergence. Such an expression has been proposed by CLEARY (1998) & MONAGHAN (2005) and accounts for the presence of spatial gradients in viscosity too. It is possible to find a proof in Espanol et al. (2003) that it accounts also for a bulk viscosity coefficient, which is equal to 5/3 times the dynamic shear viscosity. Despite not having any bulk viscosity in the constitutive law that is being used in the paper, we still think it could be worth using the expression in eqn. (12) for the deviatoric tensor divergence for two main reasons:

It conserves the angular momentum in the particles system as discussed in MONAGHAN (1997), thus leading to more accurate results in the simulations;

It provides zero-th order consistency. This means that there are no viscous interactions between particles if the velocity field is constant in space;

The bulk viscosity induced by the expression should not significantly affect the simulation results, as the velocity field in the weakly compressible fluid approximation is nearly divergence-free;

A possible alternative to eqn. (12) is the one of directly calculating the total stress tensor from eqn. (10) and to use then the following SPH approximation for its divergence:

$$\frac{Dv_i}{Dt} = \underline{f}_i + \sum_{j=1}^n m_j \cdot \left( \frac{\sigma_i}{\rho_i^2} + \frac{\sigma_j}{\rho_j^2} \right) \cdot \nabla_i W_{ij} \quad (14)$$

This formulation is still able to preserve the linear momentum of the particles system and is able to exactly reproduce the Herschel- Bulkley rheology within the accuracy of the SPH approximation, without adding any kind of bulk viscosity to the flow. Nevertheless, it lacks some good properties of the formulation of eqn. (12) like the zero-th order consistency and the angular momentum conservation

#### ARTIFICIAL COMPRESSIBILITY

The simulation of an incompressible flow requires the solution of a Poisson equation for the pressure, which often leads to an increase of the computational time. Therefore, it is more practical to approximate the incompressible medium with a weakly compressible one, thus allowing the calculation of the pressure from the density with a stiff equation of state which introduces an artificial compressibility in the fluid.

It can be shown that density variation is proportional to the square of the Mach number (MONAGHAN (1994)). If an artificial equation of state where sound speed is larger than the bulk velocity of flow is used, then it is possible to keep density variations as low as desired. The artificial equation of state used in this paper is as follows:

$$p = c^2(\rho - \rho_0) \quad (15)$$

where:

$c$  is the artificial speed of sound;

$\rho_0$  is the reference density of the fluid at zero pressure;

There are other possible forms for the equation of state that can be used to simulate a weakly compressible flow [see MONAGHAN (1994)]. Nevertheless, we found that the choice of the equation of state doesn't significantly affect the results, as long as the artificial speed of sound is chosen to be large enough.

#### VISCOSITY REGULARIZATION

The Herschel-Bulkley constitutive law provides a viscosity diverging to infinite for strain rates approaching to zero. It is therefore impossible to numerically reproduce this behaviour in a straightforward manner. In the calculations, the expression of eqn. (5) for viscosity has been regularized and the infinite value that is obtained when the strain rate approaches to zero has been replaced with a finite value. The expression used to regularize viscosity has been proposed by PAPANASTASIOU (1987):

$$\eta = \left( K \sqrt{2Tr(\mathbf{D}^2)}^{(n-1)} + \frac{\tau_c}{\sqrt{2Tr(\mathbf{D}^2)}} \right) \cdot \left( 1 - \exp\left(-B\sqrt{2Tr(\mathbf{D}^2)}\right) \right) \quad (16)$$

The parameter  $B$  is related to the maximum viscosity that is returned by the regularization when the strain rate is zero. In this case the maximum viscosity value is given by:

$$\eta_{max} = B\tau_c \quad (17)$$

The value assigned to the parameter  $B$  has been chosen high enough to correctly simulate the physical behaviour of the fluid but small enough to avoid prohibitively small time steps.

#### BOUNDARY TREATMENT

Since the gradient formulation given by eqn. (10) is used even for fluid particles close to the boundaries,

special treatments are required in order to offset the errors introduced by neglecting the terms in the right hand side of eqn. (9).

There are a number of strategies to tackle boundary-related problems. RANDES *et alii* (1996) used ghost particles to treat a symmetrical surface boundary condition. Ghost particles have also been used in various manners for particle approximations near boundaries by TAKEDA *et alii* (1994), MORRIS *et alii* (1997) and FERRARI *et alii* (2009) by using point symmetry.

In this paper, boundaries have been treated by placing a layer of particles on them, whose spacing is three times narrower than the fluid particles one. Boundary particles prevent fluid particles from passing through the domain boundaries by exerting a normal force on them. They also interact with the fluid particles via SPH summations through their viscosity, thus enforcing the no-slip condition. The expression used for the boundary forces has been proposed by MONAGHAN *et alii* (2009):

$$\underline{F}_i = \sum_{j=1}^n K_b \frac{2m_i m_j}{m_i + m_j} \frac{W_{ij}}{\|\underline{r}_{ij}\|^2} \underline{r}_{ij} \quad (18)$$

where:

$K_b$  is a constant having the dimensions of a square velocity, used to correctly reproduce the bulk forces exerted by the boundaries on the fluid;

Summation is extended to all the boundary particles located within the support domain of fluid particle  $i$ .

It can be shown (MONAGHAN *et alii*, 2009), that the above summation gives negligible contribution along the direction parallel to the boundary, making boundary forces being exerted only along the normal direction. It is also a symmetrical formulation, thus conserving the linear momentum of the particles system.

### TIME INTEGRATION

Time integration has been performed by means of a symplectic Verlet scheme, as in KAJTAR *et alii* (2008). The time stepping scheme is explicit and conserves the linear momentum of the particles system. The time step  $\Delta t$  is controlled by a C.F.L. condition depending on the artificial speed of sound, the viscous interactions between particles and on the interactions with boundary particles, according to the following equation:

$$\Delta t = CFL \cdot \min \left( \frac{h_{ij}}{c_{ij}}, 0.1 \frac{\rho_{ij} h_{ij}^2}{\eta_{ij}}, \frac{\|\underline{r}_{ij}\| - dp / \beta}{\max(\|\underline{v}_i\|, K)} \right) \quad (19)$$

where:

The minimum time step value is sought over each couple of interacting particle;

$h_{ij} = (h_i + h_j) / 2$  is a symmetrised smoothing length between the pair of interacting particles;

$c_{ij} = (c_i + c_j) / 2$  is a symmetrised artificial speed of sound between the pair of interacting particles;

$\rho_{ij} = (\rho_i + \rho_j) / 2$  is a symmetrised density between the pair of interacting particles;

$dp$  is the initial fluid particles spacing;

$\beta$  is the ratio between the boundary and the fluid particles spacing;

The C.F.L. number, indicated as  $CFL$ , has been set equal to 0.5.

### MODEL TESTING

The ability of the SPH model to correctly reproduce a mudflow has been tested by simulating the experiments performed by LAIGLE *et alii* (1994) and (1997). Their experiments consisted in creating a mudflow dam break problem in a laboratory flume, by quickly opening a gate. The experimental setup they used is briefly shown in the figure below:

After the opening of the gate, the material stored behind it was released and the three ultrasonic gages, sketched in the picture, recorded the mudflow wave heights in time. The authors used waterclay mixtures prepared in laboratory with different concentrations in order to simulate mudflows. Herschel-Bulkley rheological parameters for the used mixtures were fitted to measures, carried out with a rheometer by the authors. They performed tests with four different kinds of mixtures, named A, B, C and D. The rheological

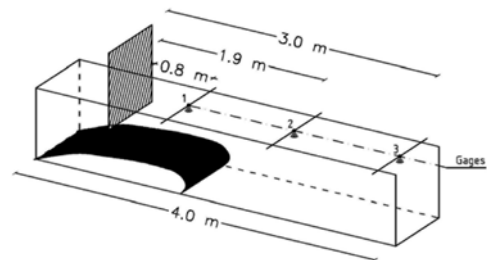


Fig. 1 - 3D sketch of the experimental setup in LAIGLE *et alii* (1994) and LAIGLE (1997)

characteristics of the A mixture and the experimental parameters are summarized in the following table:

The authors modelled the mudflow as a wide channel flow initially at rest: the resulting flow conditions were therefore completely determined by the height of the material stored behind the gate, the flume slope and the material characteristics. LAIGLE *et alii* (1994) and (1997) indicated two non dimensional scaling parameters controlling the resulting flow conditions. The values of the scaling parameters in the tests they performed can represent a wide range of real situations at smaller scales (COUSSOT, 1994). In particular, the material A, along with the experimental conditions of Tab. 1 represent a realistic natural material (with  $\rho_0 = 2200 \text{ kg/m}^3$ ,  $\tau_c = 900 \text{ Pa}$ ,  $K = 290 \text{ Pa}\cdot\text{s}^{1/3}$ ) in a 120 m long slope (LAIGLE *et alii*, 1997). A field determination of the Herschel-Bulkley rheological parameters of a natural debris flow can be found in COUSSOT *et alii* (1998), even though the values provided for the parameters in the paper are in a slightly different range than the ones indicated above. LAIGLE *et alii* (1994) and (1997) developed a 1D numerical model based on the shallow water equations and solved it with a Godunov conservative finite volumes scheme. We tested our SPH model by simulating the experiment of LAIGLE *et alii* (1997) using the rheological parameters of the Herschel-Bulkley model for the material A and the experimental conditions indicated in Tab. 1.

In our SPH simulations, particles were stored behind the gate and a layer of boundary particles was placed to simulate the closed gate at the initial time step. Particles were initially stored in a uniform lattice according to their spacing and were given a  $\rho_0 = 1410 \text{ kg/m}^3$  density. At first, around 3000 damped time steps, as in MONAGHAN (1994), were performed in order to build up a hydrostatic pressure distribution. After that, the gate opening was simulated by removing the layer of boundary particles placed on it, thus releasing the mudflow. The artificial Mach number was set to 0.2, the boundary force constant of eqn. (18) was set to  $K_b = gH_g$  (where  $g$  = gravity acceleration,  $H_g$  = height of material behind the gate) and the coefficient  $B$  of eqn. (17) for the viscosity regularization was set to 10 s.

Another important parameter is the initial particles spacing  $dp$ , which plays the same role of the grid spacing in finite differences schemes. A decrease in the spacing improves the accuracy and reduces the numeri-

cal dissipation but increases the computational time, as the number of particles required to fill the entire computational domain gets higher. As the physics of the flow is dominated by viscosity, the choice of the SPH discretization for the momentum equation plays an important role. We have tried both eqn. (12) and (14) with different particles spacing to reproduce the experiment of Tab.1. Accuracy is improved at higher resolutions for both equations but even though eqn. (12) seems to provide more accurate and less noisy results, it seems also to be more dissipative than eqn. (14). The influence of resolution and the issues regarding what would be the most appropriate choice for the momentum equation will be discussed further in a future paper

Here we present the results obtained by using eqn. (14) with a particles spacing of  $dp = 3 \text{ mm}$ . The following figure shows the initial particle disposition, right before the opening of the gate:

In the following figures the particle disposition is reported in two different instants of the simulation:

We then extrapolated the wave heights below the same locations of the ultrasonic gages used in the experiments of LAIGLE *et alii*. (1994) and (1997) from the numerical results of our simulation by measuring the thickness of the layer of particles passing below. Finally, a comparison of our numerical wave heights

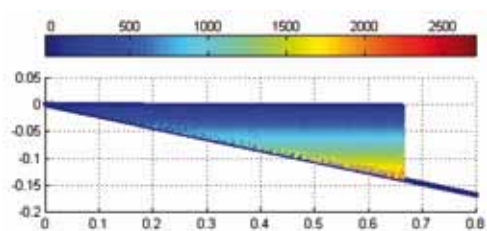


Fig. 2 - Initial particles disposition (lengths are in meters and particles are colour coded according to their pressure in Pa)

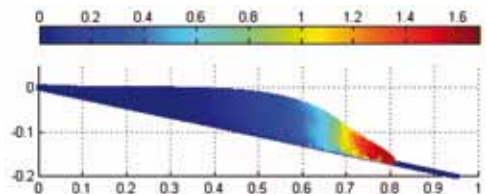


Fig. 3 - Particles disposition after 0.15 s from gate opening (lengths are in meters and particles are colour coded according to the module of their velocity in m/s)

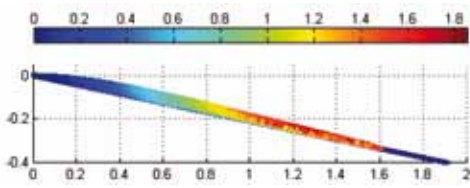


Fig. 4 - Particles disposition after 0.60 s from gate opening (lengths are in meters and particles are colour coded according to the module of their velocity in m/s)

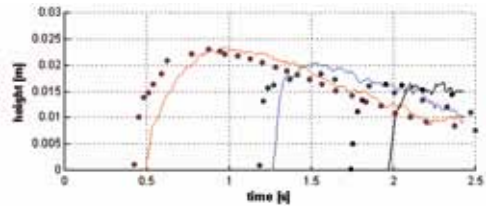


Fig. 5 - Time plot of the simulated wave heights at the locations of gage 1 (red line), gage 2 (blue line) and gage 3 (black line). Experimental measurements of gages are represented by dots coloured as the respective lines

with the experimental measurements of LAIGLE *et alii* (1994) and (1997) is reported in Fig. 5:

It can be observed that while the wave heights are well caught, the SPH model is still affected by some numerical diffusion, as the calculated wave's velocity is lower than in the experimental data. This can be improved either by further increasing the resolution or by switching to a less diffusive SPH formulation for the momentum equation.

## CONCLUDING REMARKS

A SPH model incorporating the Herschel-Bulkley non Newtonian rheology has been developed and has been tested to simulate some mudflow dam

break experiments made by LAIGLE *et alii* (1994) and (1997). The SPH model is fully two-dimensional and is capable of providing more information than the one-dimensional model. Nevertheless, a preliminary analysis shows that it is too diffusive, while still being able to produce a good agreement with experimental results. While the SPH model numerical diffusivity is reduced by increasing the resolution, other kinds of improvements haven't been fully tested yet, but they seem possible by using different SPH versions of the momentum equation. In further papers, parametric studies with comparison with actual debris flow phenomena will be discussed.

## REFERENCES

- ATKIN R.J. & CRAINE R.E. (1976) – *Continuum theories of mixtures: basic theory and historical development*. Q. J. Mech. Appl. **29**(2): 209-244.
- CHUNG T.J. (2006) – *Computational Fluid Dynamics*. Cambridge University Press, New York.
- CLEARY P.W. (1998) – *Modelling confined multi-material heat and mass flows using SPH*. Appl. Math. Modelling **22**: 981-993.
- COUSSOT P. (1994) – *Steady, laminar, flow of concentrated mud suspensions in open channel*. Journal of Hydraulic Research **32**(4): 535-559.
- COUSSOT P. & MEUNIER M. (1996) – *Recognition, classification and mechanical description of debris flows*. Earth-Sci. Rev. **40**: 209-227.
- COUSSOT P., LAIGLE D., ARATTANO M., DEGANUTTI A. & MARCHI L. (1998) – *Direct determination of rheological characteristics of debris flow*. Journal of Hydraulic Engineering **124**(8): 865-868.
- CROSTA G.B., CALVETTI F., IMPOSIMATO S., RODDEMAN D., FRATTINI P. & AGLIARDI F. (2001) – *Granular Flows and Numerical Modelling of Landslides*. Debrisfall Assessment in Mountain Catchments for local end-users Contract N. EVG1-CT-1999-00007.
- ESPANOL P. & REVENGA M. (2003) – *Smoothed dissipative particle dynamics*. Physical Review **67**: 026705.
- FERRARI A., DUMBSER M. & TORO E.F. (2009) - *A new 3D parallel scheme for free surface flows*. Computers & Fluids **38**: 1203-1217.
- IVERSON R.M. (1997) – *The physics of debris flows*. Reviews of Geophysics **35**(3): 245-296.
- KAJTAR J.B. & MONAGHAN J.J. (2008) – *SPH simulations of swimming linked bodies*. Journal of Computational Physics **227**: 8568-8587.
- IDELSOHN S.R., OÑATE E., CALVO N. & DEL PIN F. (2003) – *The meshless finite element method*. Int. J. Numer. Meth. Engng. **58**: 893-912.
- IDELSOHN S.R., OÑATE E. & DEL PIN F. (2004) – *The Particle Finite Element Method: a powerful tool to solve incompressible*



- flows with freesurfaces and breaking waves*. Int. J. Numer. Meth. Engng. **61**: 964-989.
- LAIGLE D. & COUSSOT P. (1994) – *Modélisation numérique des écoulements de laves torrentielles*. La Houille Blanche **3**: 50-56.
- LAIGLE D. & COUSSOT P. (1997) – Numerical modelling of mudflows. Journal of Hydraulic Engineering **123**(7): 617-623.
- LAIGLE D., LACHAMP P. & NAIM M. (2007) - *SPH-based numerical investigation of mudflow and other complex fluid flow interactions with structures*. Comput. Geosci. **11**: 297-306.
- LIU G.R. & LIU M.B. (2003) - *Smoothed Particle Hydrodynamics a meshfree particle method*. World Scientific Publishing.
- Liu M.B. & Liu G.R. (2006) - *Restoring particle consistency in smoothed particle hydrodynamics*. Applied Numerical Mathematics **56**: 19-36.
- MONAGHAN J.J. (1994) - *Simulating free surface flows with SPH*. Journal of Computational Physics **110**: 399-406.
- MONAGHAN J.J. (1997) – *SPH and Riemann solvers*. Journal of Computational Physics **136**: 298-307.
- MONAGHAN J.J. (2005) - *Smoothed particle hydrodynamics*. Reports on Progress in Physics **68**: 1703-1759.
- MONAGHAN J.J. & KAJTAR J.B. (2009) – *SPH particle boundary forces for arbitrary boundaries*. Computer Physics Communications **180**: 1811- 1820.
- MORRIS J.P., FOX P.J. & ZHU Y. (1997) - *Modeling low Reynolds number incompressible flows using SPH*. Journal of Computational Physics **136**: 214-226.
- OÑATE E., IDELSOHN S.R., DEL PINO F. & AUBRY R. (2004) – *The Particle Finite Element Method. An Overview*. International Journal. of Computational Methods **1**(2): 267-307.
- PAPANASTASIOU T.C. (1987) – *Flows of materials with yield*. Journal of Rheology **31**(5): 385.
- PIERSON T.C. & COSTA J.E. (1987) – *A rheologic classification of subaerial sediment-water flows*. Rev. Eng. Geol., Geol.Soc. Am., Boulder, Co., **7**: 1-12
- PÖSCHEL T. & SCHWAGER T. (2005) – *Computational Granular Dynamics*. Springer Berlin Heidelberg, New York
- RANDLES P.W. & LIBERSKY L.D. (1996) - *Smoothed particle hydrodynamics: some recent improvements and applications*. Computer Methods in Applied Mechanics and Engineering **138**: 375-408.
- TAKAHASHI T. (1991) – *Debris Flow*. IAHR Monograph, published for IAHR by A.A.Balkema, Rotterdam
- TAKEDA H., MIYAMA S.M. & SEKIYA M. (1994) – *Numerical simulation of viscous flow by smoothed particle hydrodynamics*. Progress of Theoretical Physics **92**: 939-960.