Numerical computation of a confined sediment-water mixture in uniform flow

L. Sarno, R. Martino[1] & M.N. Papa[2]
[1]*Department of Hydraulic and Environmental Engineering 'G.Ippolito', Naples University, Italy.*
[2]*Department of Civil Engineering, Salerno University, Italy.*

Abstract

The purpose of this paper is to simulate a laminar mud flow confined in a narrow rectangular open channel. The flow bed is an erodible layer made up of the same material involved in the flow; the equilibrium condition between the moving and non moving layer is assumed. The mud mixture under consideration is ruled by Herschel-Bulkley's shear thickening rheological law. It is supposed that the local volumetric concentration is linearly increasing with the depth and it is constantly equal to its maximum value where the local velocity is smaller than a threshold value. Relations among rheological parameters and concentration are obtained through laboratory rheometric tests. Turbulence effects and Coulombian stresses are ignored. To get the flow velocity, the momentum equation has to be integrated along the flow cross section. Unfortunately, it is very difficult to integrate this equation using H-B rheological law, since there are different stress functions and it is not possible to know a priori the sub-domains of them (plug, non plug and bed regions). In the present work a modified rheological law, continuous over the whole domain of integration is employed and the momentum equation is integrated numerically. This modified law is obtained by adding a constant correcting the denominator in the H-B stress functions: strictly speaking, there are not dead zones or plug any more. However it is noteworthy that, using a small constant, the model produces a good simulation of plug and dead zone: that is velocity gradient is very small there. The mathematical model has two parameters: maximum concentration and threshold velocity. These parameters are adjusted by back-analysis with measurements from laboratory flume experiments in uniform flow conditions.
Keywords: mud flow, Herschel-Bulkley rheological law, equilibrium, plug

1 Introduction

Mud flows are very dangerous phenomena for anthropic settlements and so, during these last years, they are widely studied in environmental engineering. Mud flow is characterized by motion of a two-phase mixture, consisting of water and high-concentrated fine-granulometry solid matter; therefore, its mechanical behavior is solid-like while the acting shear stresses are smaller than a fixed *yield stress*, and it is similar to a non-Newtonian fluid when the acting stresses are bigger. The ultimate goal of this research field is to obtain a resistance law that correlates the flow rate and the flow depth, taking into account both the natural mixtures and irregular-shape flow cross sections.

The present work reports a computational model to study the mud flow under some simplifying hypotheses. The numerical computation was performed in MATlab environment, by implementing a finite-difference method.

2 Constitutive law and definitions

There are two ways to approach the mechanical problem: by considering independently solid and liquid phases, or by using an equivalent fluid model with a rheological law, that, by accounting all the modes of resistance inside the mixture, relates the shear stress τ with the shear rate $\dot{\gamma}$. In this work the problem is tackled with the second approach: a rheological law, where parameters depend on the local volumetric concentration of solid matter, was used. Actually, the mixture capability to support shear stress depends on the relative distances among solid particles and, consequently, on the solid concentration.

The motion, under study, takes place in a rectangular-shaped open channel. Uniform and laminar conditions were assumed: that means the velocity is a scalar function defined over the flow cross section. The motion develops on erodible layer, consisting of the same solid matter which is in the mixture, and in equilibrium condition: there is a dynamic equilibrium between solid deposit and particles at motion inside the flow. Therefore, at equilibrium there is a zone, called *dead zone*, where solid matter is at rest. To tell the cross section sub-domain, where velocity is non-null, from the whole cross section, the first one will be called "active cross section" and the other one merely "cross section". The velocity function, in equilibrium condition, is marked out by null gradient at boundary, between active flow cross section and dead zone.

The following anticlockwise system of axes was assumed as reference frame: the x axis is parallel to the motion direction and so it is perpendicular to the cross section, y is perpendicular to the erodible bed and it lies on the cross section, z is parallel to the absolute bed plane and it lies on the cross section.

A scheme of the channel and the reference frame is reported in fig. 1.

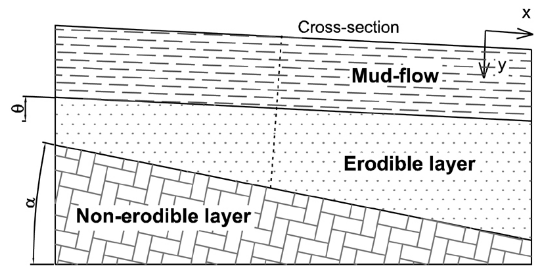

Figure 1: View of xy plane, z-axis is perpendicular to the sheet. It is shown that usually erodible layer slope is different to flume bed slope. The dashed line represents the cross section.

The mixture considered is ruled by Herschel-Bulkley's law, that has got the following one-dimensional form:

$$\tau = \tau_B + \mu \left(\frac{du}{dn} \right)^{\eta} \qquad \text{if } \tau > \tau_B \tag{1}$$

$$\frac{du}{dn} = 0 \qquad \text{if } \tau \leq \tau_B \tag{2}$$

where u is the velocity, τ the shear stress, τ_B the yield stress, μ the apparent viscosity. The second form is due to the fact that, when acting stress is smaller than yield stress, the behavior of mixture is solid-like.

In steady conditions, everywhere the acting shear stress is equal to the resistant one.

The general expression of H-B's law [1], valid for three-dimensional problems, is the following one:

$$\mathbf{T} - p\mathbf{I} = \frac{\tau_B \mathbf{D}}{\sqrt{-D_{II}}} + \frac{2^{\eta} \mu \mathbf{D}}{\sqrt{-D_{II}^{1-\eta}}} \qquad \text{if } \tau > \tau_B \tag{3}$$

$$\mathbf{D} = 0 \qquad \text{if } \tau \leq \tau_B \tag{4}$$

where $\mathbf{T}$ is the stress tensor, $\mathbf{I}$ the unit tensor, $\mathbf{D}$ the strain rate tensor, D_{II} the second invariant of the secular equation associated with tensor $\mathbf{D}$.

One can easily see there are two different rheological forms: second one postulates that, where acting stress is smaller than yield stress τ_B, there is no strain, that is the whole shear stress is supported by the solid matter. Direct consequence of that is the presence of a *plug* region, where the velocity vector is constant, set on the top of the flow, close to the free surface.

An example of a typical velocity distribution, where one can see *dead* and *plug* zones, is reported in the fig. 2, reproduced from [2].

Rheological parameters τ_B and μ strongly depend on the local concentration, but η is only dependent on the chemical-physical nature of the solid suspension.

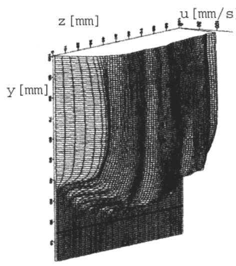

Figure 2: Velocity distribution, observed in laboratory flume experiments: *dead* and *plug* zones are evident. [2].

For solid matter used in laboratory flume experiments, coming from Sarno area (Italy), the following fitting forms are obtained, through rheometric tests:

$$\tau_B = 0.0589 \cdot e^{12.071 \cdot c},\ \mu = 0.0020 \cdot e^{9.382 \cdot c},\ \eta = 1.722 \tag{5}$$

where c is the local volumetric concentration. As one can see, this dependence is exponential so a good estimation of c is essential.

In this paper three kinds of concentration were used: volumetric local concentration c, mean concentration c_m over the cross section and flow concentration c_t. They are defined in the following expressions:

$$c(\mathbf{x}) := \lim_{V \to 0} \frac{V_s}{V_s + V_w},\ c_m := \frac{\displaystyle\int_\Omega c(\mathbf{x})\, dA}{\Omega},\ c_t := \frac{\displaystyle\int_\Omega c(\mathbf{x})\mathbf{u}(\mathbf{x}) \cdot \hat{\mathbf{n}}\, dA}{\displaystyle\int_\Omega \mathbf{u}(\mathbf{x}) \cdot \hat{\mathbf{n}}\, dA} \tag{6}$$

where V_s the volume taken up by the solid suspension, V_w the volume of water. Ω is the cross section domain, $\mathbf{u}(\mathbf{x})$ the velocity at $\mathbf{x}$ point, $\hat{\mathbf{n}}$ the unit vector normal to the cross section.

The local concentration is defined over the whole section, whereas the mean concentration and the flow concentration are features of the entire motion. The flow concentration means also the ratio between the solid flow rate and the total one.

By projecting on x-axis eqn (3), the following expressions are obtained:

$$\tau_{yx} = \frac{\tau_B + \mu \left[\left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial u}{\partial z} \right)^2 \right]^{\frac{\eta}{2}}}{\sqrt{\left[\left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial u}{\partial z} \right)^2 \right]}} \frac{\partial u}{\partial y} \tag{7}$$

$$\tau_{zx} = \frac{\tau_B + \mu \left[\left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial u}{\partial z} \right)^2 \right]^{\frac{\eta}{2}}}{\sqrt{\left[\left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial u}{\partial z} \right)^2 \right]}} \frac{\partial u}{\partial z}, \tag{8}$$

where τ_{zx} is the x-axis component of the shear stress vector, acting on the surface with normal z, and similarly τ_{yx} is the x-axis component of the one, acting on the surface with normal y.

3 Differential problem

Momentum equation, valid everywhere over the cross section domain, can be written as:

$$\rho \left(\mathbf{g} - \dot{\mathbf{u}} \right) = \nabla \cdot \mathbf{T} \tag{9}$$

where $\dot{\mathbf{u}}$ is the Lagrangian acceleration and $\mathbf{g}$ the gravity constant.

The x-axis component of eqn (9) can be written as follows:

$$\rho g \sin \theta + \frac{\partial \tau_{zx}}{\partial z} + \frac{\partial \tau_{yx}}{\partial y} = 0, \tag{10}$$

where ρ is the mean density ($\rho = c\, \rho_{solid} + (1-c)\ \rho_{water}$) and θ is the flow slope, usually different from that of non-erodible layer, which lies below.

If velocity boundary conditions and local concentration distribution were known, since functions τ_{zx} and τ_{yx} depend on u because of eqns (7 and 8), it would be possible to solve numerically the differential problem, associated to eqn (10), for the only function $u(y, z)$.

The domain of integration was a reference cross section, arbitrarily chosen in the whole flume. It is assumed that solution does not vary with total flow depth, if it is allowed to ignore Coulombian stresses.

3.1 Corrective term ϵ^2

The main difficulty to integrate eqn (10) is that it is a free-boundary problem: it is not possible to fix *a priori* the size of plug sub-domain and the value of velocity in the plug. Besides, eqn (10) is not defined in the plug. To overcome the problem, the functions (7 and 8) were replaced by the following ones, which are defined and continuous over the whole cross section:

$$\tau_{yx} = \frac{\tau_B + \mu \left[\left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial u}{\partial z} \right)^2 \right]^{\frac{\eta}{2}}}{\sqrt{\left[\epsilon^2 + \left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial u}{\partial z} \right)^2 \right]}} \frac{\partial u}{\partial y} \tag{11}$$

$$\tau_{zx} = \frac{\tau_B + \mu \left[\left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial u}{\partial z} \right)^2 \right]^{\frac{\eta}{2}}}{\sqrt{\left[\epsilon^2 + \left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial u}{\partial z} \right)^2 \right]}} \frac{\partial u}{\partial z}. \tag{12}$$

The critical effect, due to the ϵ^2, is the absence of yield stress, and so there are not dead or plug zones any more, strictly speaking [3]. The constant ϵ^2 should be as small as possible, for a Bingham fluid in [4] a value smaller than 10^{-16} is recommended. Really, it is obvious that the more ϵ^2 is small, the more expressions (11 and 12) come close to the original H-B's law and the more the u solution will have a zone, where the velocity is almost null with its gradient, and a zone with almost constant velocity. In this paper it is used an $\epsilon^2 = 10^{-3}$.

3.2 Concentration distribution

Unfortunately, it is not yet possible to obtain a reliable estimation of local concentration c through experimental measures, so it is required to formulate some hypotheses about it. Sure, c is increasing with the depth and there are some experimental results that confirm a nearly linear trend of c at solid boundary [5]. It is essential to make a good estimation of c, since rheological parameters depend on it.

In this work following hypotheses are assumed:

- concentration linearly increasing with the depth: $c = c_0 + k\,y$,
- existence of a maximum packing value of concentration c_{max}, independent of parameters of motion which vary from case to case (ex. slope, c_m),
- existence of a threshold value of velocity u_{thr}, under which concentration is equal to maximum packing value.

Values between $[0.66 - 0.69]$ for c_{max} and between $\left[10^{-4} - 10^{-3}\,m/s\right]$ for u_{thr} were tried.

3.3 Boundary conditions

The following boundary conditions are assumed:

- no-slip condition, that is null velocity everywhere at solid boundaries of channel,
- null shear stress at free surface.

Actually, this model could be tested also in slip condition at side solid border, which seems to be more realistic, but it's very difficult to obtain a reliable experimental estimation of velocity there.

4 Numerical implementation

A finite-difference discretization of differential problem was performed. First-order derivative of u in eqns (11 e 12) were replaced by their central approximations.

The integration domain was discretized in rectangular-shaped cells, of size $\Delta z \times \Delta y$. Continuous functions u, ρ become discrete functions, pertinent to the centre of gravity of cells: so they are implemented as matrices. Similarly functions τ_{yx} and τ_{zx} are implemented as matrices, with the following convention: $\tau_{zx}(i,j)$ is the shear stress acting over the right-hand side of the cell (i,j), and $\tau_{yx}(i,j)$ is the shear stress acting over the lower side of the cell (i,j). Stresses are regarded as positive when concordant with x-axis.

A scheme of conventions about τ stresses is reported in fig. 3.

So instead of momentum equation, a forces balance can be written for the generic cell (i,j):

$$[g\rho(i,j)\sin\theta] + [\tau_{zx}(i,j) - \tau_{zx}(i,j-1)]/\Delta z + [\tau_{yx}(i,j) - \tau_{yx}(i-1,j)]/\Delta y = 0. \quad (13)$$

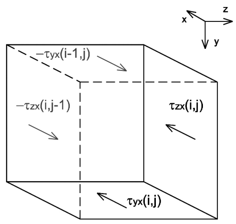

Figure 3: The convention assumed for τ stresses.

There are as many equations as are the cells, and so, as are unknown variables $u(i,j)$. It was used a 64×64 cells discretization: the differential problem was changed in a non-linear system of 4096 equations. Boundary conditions are implemented, by using ghost null values of u at solid boundary and by imposing $\tau_{yx} = 0$ at free surface.

To solve the non-linear system, it was used “Fsolve”, which is a *trust-region* algorithm, included in Optimization Toolbox [6]. Solution tolerance was set to 10^{-6}. To improve performances, it was exploited problem symmetry (this way the non-linear system will have only 2048 equations) and it was used a *pattern matrix*, which informs the computer about zeros, to obtain a faster computation of Jacobian matrix.

4.1 Loops to define c matrix and convergence of algorithm

Every time the algorithm solves the differential problem, a guessing distribution of c is assumed. The computation finishes when the solution is congruent with its distribution of c. It is possible to state two different congruence conditions:

- "threshold velocity condition", which is verified when the velocity solution is smaller than threshold velocity in cells where $c = c_{max}$ and only in them,
- "flow concentration condition", which is verified when c_f, calculated after solving the differential problem for u, is equal to c_f to simulate.

Really, the second condition is used to assure that it is simulated the particular motion observed in laboratory flume experiments and not any other.

Outside of the code which solves the differential problem, there are two nested *do-while* loops, responsible to verify the congruence conditions: the outer one is pertinent to the "flow concentration condition", the inner one to the "threshold velocity condition". The hypothesis that c_m is constant in every columns of the cross section is assumed. So, to fix a local concentration distribution there are $n+1$ freedom degrees, if n is the number of columns (in the case of this work 64): a degree is c_m and the n others ones are the packing positions in each column, that is the positions where c becomes equal to its maximum value c_{max}.

In fig. 4 it is reported the flowchart of algorithm.

After obtaining the u solution, it was calculated the flow rate Q, useful for the analysis of results.

Due to loops outside of differential problem, which can be regarded as *Turing-computable* functions but not analytical, it is very difficult to obtain a strict proof of convergence, by using spectral methods. An heuristic way was tackled: it was observed that even increasing the discretization level (up to 160×160 cells) the solution weakly changes. As well, the solution seems to be independent of first guessing concentration distribution [7].

5 Analysis of results and conclusions

Velocity and local concentration distributions, obtained by the simulation, are reported in fig. 5. As one can see, there is a discontinuity of the first type in c distribution: of course, it is a weak point of model, on which the future study will be focused.

Actually, even though it was used a modified rheological law with a rather big value of ϵ^2 (10^{-3}), one can easily see (fig. 5) a velocity distribution with well defined dead and plug zones: in other words, the integration method seems to be suitable for this kind of problems.

The expected boundary points of plug and dead zones, for the generic column j, is when the total shear stress is equal to the yield stress: $\sqrt{\tau_{yx}^2 + \tau_{zx}^2} = \tau_B(c)$.

It is very interesting, now, to compare boundary points of constant velocity zones, which can be observed in velocity distribution after simulation, with these expected boundary points, for each column of integration. The remarkable result

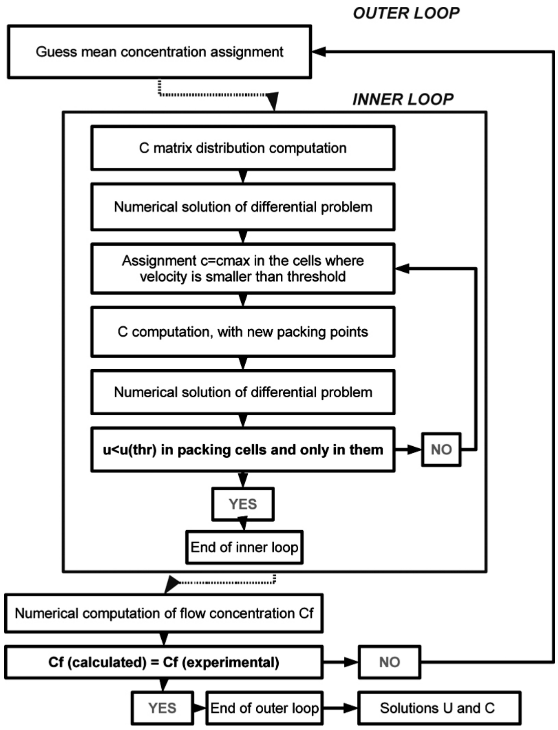

Figure 4: Algorithm flowchart.

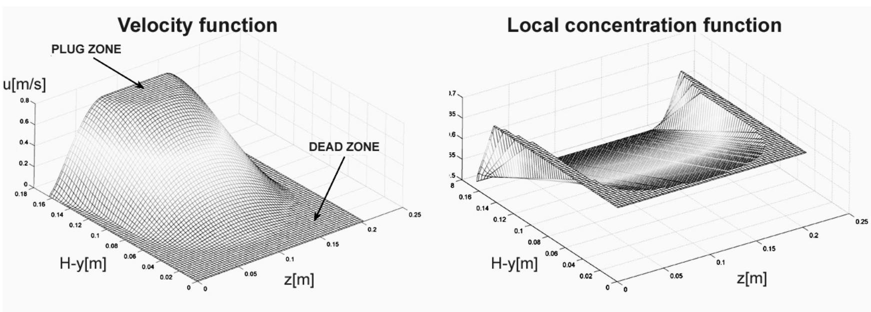

Figure 5: Velocity and local concentration distributions obtained by the simulation: constant velocity zones are highlighted.

of this work is that, everywhere in the integration domain, one can notice an encouraging correspondence between these points. Fig. 6 reports the comparison at middle column of the flume. That is a further confirmation that this modified-rheology numerical method seem to be working, even with high values of constant ϵ^2.

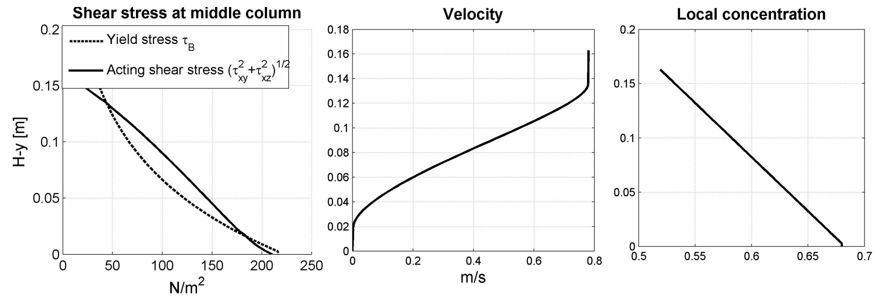

Figure 6: Comparison among total stress, yield stress, velocity distribution and local concentration.

The model needs two parameters, c_{max} and u_{thr}, not yet obtainable by direct measures. So they have to be fixed, by a back-analysis of some laboratory flume experiments.

The future work will be to compare the flow rates simulated with experimental measures, to improve the condition of threshold velocity, with the contribution of experimental results, to run the model in slip condition at solid boundary and, perhaps, to implement Coulombian stresses, which seem to be not negligible in the dead zones.

References

[1] Macosko, C.W., *Rheology - Principles, Measurements and Applications*. Wiley-VCH: New York, 1994.

[2] Dalrì, C., Fraccarollo, L., Larcher, M. & Armanini, A., Analisi sperimentale per la caratterizzazione del flusso di miscugli iperconcentrati di origine sintetica e naturale. *Proceedings of XXIX Convegno Nazionale di Idraulica e Costruzioni Idrauliche – IDRA*, Trento, 2004.

[3] Whipple, K.X., Open channel flow of bingham fluids: application in debris-flow research. *Journal of Geology*, **105**, pp. 243–263, 1997.

[4] Bercovier, M. & Engleman, M., A finite element method for incompressible non-newtonian flows. *Journal of Computational Physics*, **36**, pp. 313–326, 1980.

[5] Martino, R. & Papa, M.N., Effetto delle pareti nelle correnti detritiche: primi risultati. *Proceedings of XXIX Convegno Nazionale di Idraulica e Costruzioni Idrauliche – IDRA*, Trento, 2004.

[6] AA.VV., Optimization toolbox - *Trust-region* methods for nonlinear minimization. *Matlab 70 R14 Help*, 2004.

[7] Sarno, L., Simulazione di una colata di fango in regime visco-plastico e condizioni di moto uniforme. Degree thesis – Naples University, Naples, 2006. Supervisor: Martino R.