ESAIM: M2AN
Vol. 37, N° 4, 2003, pp. 699–708
DOI: 10.1051/m2an:2003052

ESAIM: Mathematical Modelling and Numerical Analysis

FLUID–PARTICLE SHEAR FLOWS

Bertrand Maury[1]

Abstract. Our purpose is to estimate numerically the influence of particles on the global viscosity of fluid–particle mixtures. Particles are supposed to rigid, and the surrounding fluid is newtonian. The motion of the mixture is computed directly, *i.e.* all the particle motions are computed explicitly. Apparent viscosity, based on the force exerted by the fluid on the sliding walls, is computed at each time step of the simulation. In order to perform long–time simulations and still control the solid fraction, we assume periodicity of the flow in the shear direction. Direct simulations are based on the so–called Arbitrary Lagrangian Eulerian approach, which we adapted to make it suitable to periodic domains. As a first step toward modelling of interacting red cells in the blood, we propose a simple model of circular particles submitted to an attractive force which tends to form aggregates.

Mathematics Subject Classification. 76T20, 65M60, 92-08.

1. Introduction

We consider here a mixture of (2D) newtonian fluid and rigid particles, submitted to constant shear between two parallel walls. Direct simulation is used to investigate the dependency of the apparent viscosity upon particle concentration. This problem still generates a considerable activity from the experimental point of view. Fluid–particle mixtures are typically located between two concentric cylinder. A shear flow is created by exerting a torque on one of the cylinders. Apparent viscosity is deduced from the shear rate and the torque (see for example [1]). Such experiments are still out of range of direct simulations, because the number of bodies (solid spheres in a polymer solution, red cells in the blood, ...) is huge in real situations. We therefore use a periodic model to focuse on a limited part of the flow, yet keeping the ability to control the solid volume fraction of the mixture. We present in this paper bidimensional simulations only (objects are actually infinite cylinders), which makes it difficult to compare with existing experimental data. For this reason, we shall restrict ourselves to some illustrations of what can be computed from direct simulations, and put off comparisons with experiments until 3D flows can be computed for similar configurations.

2. Continuous model

We consider a rectangular domain Ω filled with a mixture of a newtonian fluid and N particles. Viscosity and density of the fluid are denoted by μ and ρ, respectively. All particles are circular, with density ρ_p, radius r, mass m and moment of inertia J. We shall concentrate on a particular situation: the mixture domain Ω is a rectangle $2a \times L$ (see Fig. 1). The flow is periodic in the $y-$direction. Left and right walls are supposed to move vertically with velocities $-U_0$ and $+U_0$. Origin of the reference frame (x, y) is set on the centerline.

Keywords and phrases. Fluid–particle, ALE, Finite Element, Shear Flow.

[1] Laboratoire Jacques-Louis Lions, Université P. et M. Curie, 75252 Paris Cedex 05, France. e-mail: maury@ann.jussieu.fr

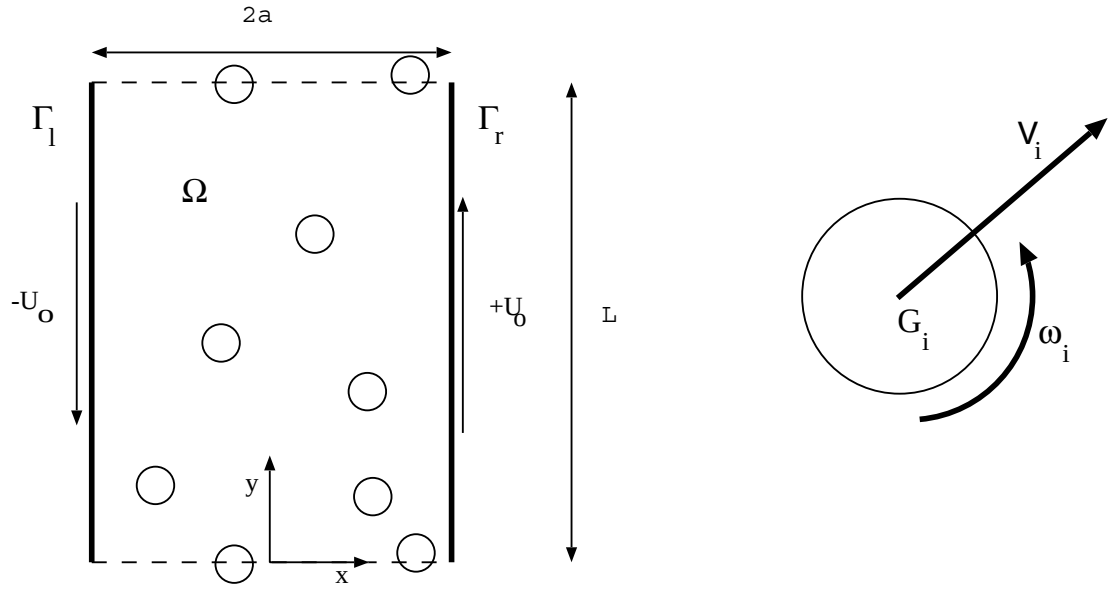

FIGURE 1. Notations.

The region occupied by the solid phase at time t shall be denoted by $B(t)$. The surrounding fluid is supposed to obey the incompressible Navier–Stokes equations in the moving fluid domain $\Omega \setminus B(t)$,

$$\begin{cases} \rho\left(\dfrac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\cdot\nabla\mathbf{u}\right) - \nabla\cdot\sigma = 0 \\ \nabla\cdot\mathbf{u} = 0, \end{cases} \tag{1}$$

while translational velocities $\mathbf{V}_i$ and angular velocities ω_i of the particles satisfy

$$m\,\frac{\mathrm{d}\mathbf{V}_i}{\mathrm{d}t} = -\int_{\Gamma_i(t)} \sigma\cdot\mathbf{n}\,, \tag{2}$$

$$J\,\frac{\mathrm{d}\omega_i}{\mathrm{d}t} = -\int_{\Gamma_i(t)} (\mathbf{x}-\mathbf{G}_i)\times\sigma\cdot\mathbf{n}, \tag{3}$$

where $\mathbf{G}_i$ is the center of particle i, and σ the stress tensor

$$\sigma = \mu(\nabla\mathbf{u} + \nabla^T\mathbf{u}) - p\mathrm{I_d}. \tag{4}$$

The no-slip condition on the i-th particle surface (denoted by $\Gamma_i(t)$) is

$$\mathbf{u} = \mathbf{V}_i + \omega_i\times(\mathbf{x}-\mathbf{G}_i) \qquad \forall\mathbf{x}\in\Gamma_i(t). \tag{5}$$

As a first step toward modelling interacting red cells in the blood, we add an attractive force between circular particles, which amounts to add

$$\mathbf{F}_i = -\sum_{j\neq i} \varphi(|\mathbf{G}_i\mathbf{G}_j|)\frac{\overrightarrow{\mathbf{G}_i\mathbf{G}_j}}{|\mathbf{G}_i\mathbf{G}_j|}, \tag{6}$$

to the left–hand side of equation (2), where φ represents the modulus of the interparticle force. Note that periodic images of particles have to be taken into account in the computation of interaction forces. In order to limit the computation, we actually did not consider distances over 20 times of a particle diameter.

3. Time and space discretizations

Several methods have been carried out to perform such simulations. The first approach consists in embedding the particle in a global domain which covers the physical domain occupied by the mixture (see for example Glowinski [2]). The second one, on which the present work is based, relies on a moving mesh which follows the fluid domain (see [3,4]). A detailed description of the algorithm which is used here is given in [5]. Equations are discretized within the Arbitrary Lagrangian Eulerian framework: a domain velocity is defined, which identifies to the particle velocities at their boundaries (lagrangian aspect), and which is arbitrary within the fluid domain (eulerian aspect). The nonlinear term in the momentum equation is approximated by a method of characteristics (see Pironneau *et al.* [6]), which can be adapted to the ALE framework, as indicated in [5]. The generalized Stokes problem which is obtained is dicretized in space within the Finite Element framework (see Fig. 2 for an example of mesh). Note that boundary integrals can be eliminated in the Galerkin formulation, so that no explicit calculation of hydrodynamical forces is needed. The two main difficulties are related to particle collisions, and mesh generation (each time it is necessary, a periodic mesh of the fluid domain is generated). We refer again to [5] for details.

4. Apparent viscosity

In case of a single newtonian fluid with no particle, the stationnary solution (to both Navier–Stokes and Stokes equations) is given by

$$\mathbf{u}(x,y) = U_0 \frac{x}{a} \mathbf{e}_y, \tag{7}$$

with constant pressure overall the domain.

First attempts to define the apparent viscosity were based on the rate of dissipated energy D. When there is no particles, it can be computed as

$$D_0 = \mu \int_\Omega \left(\frac{U_0}{a}\right)^2 = \mu\, U_0^2\, \frac{L}{a}. \tag{8}$$

Given now a mixture in the same geometry, on can define (and compute) at any time the dissipated energy

$$D(t) = \mu \int_{\Omega \setminus B(t)} |\nabla \mathbf{u}|^2 .$$

Identification of two formulae gives a value for an apparent viscosity from the dissipation point of view

$$\mu_{\text{diss}} = \frac{\mu a}{L U_0^2} \int_{\Omega \setminus B(t)} |\nabla \mathbf{u}|^2 .$$

This approach gives satisfactory results when the only active forces exerted on the fluid are those exerted by the walls. In the present case, particle may interact, and it leads to high dissipation within the fluid, and therefore to high values for μ_{diss}, which can no longer be considered a measure of the apparent (*i.e.* from the outside) viscosity.

We therefore switched to an expression based on the tangential forces exerted by the fluid on the lateral walls. Let us denote by $\mathbf{f}_l$ and $\mathbf{f}_r$ the forces along y-direction exerted on the left and right walls, respectively. As for the left side, one has

$$\mathbf{f}_l = -\int_{\Gamma_l} \mu \frac{\partial u_y}{\partial x} = -\int_{\Gamma_l} \mathbf{e}_y \cdot \sigma \cdot \mathbf{e}_x .$$

This force can be expressed in a weak form, by introducing a vector test function $\mathbf{w}$ which is equal to $\mathbf{e}_y$ (unit vector along y–direction) on Γ_l, and vanishes on the other boundaries (Γ_r and the boundaries of the particles).

Momentum equation (1) is multiplied by $\mathbf{w}$ and integrated over $\Omega \setminus B(t)$ (as $\mathbf{w}$ is supported within $\Omega \setminus B(t)$, it makes no difference to write integrals over the whole domain Ω):

$$\mathbf{f}_l = -\int_{\Gamma_l} \mathbf{e}_y \cdot \sigma \cdot \mathbf{e}_x = \int_{\Gamma_l} \mathbf{w} \cdot \sigma \cdot \mathbf{n} \tag{9}$$

$$= \int_\Omega \rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) \cdot \mathbf{w} + \mu \int_\Omega \nabla \mathbf{u} \cdot \nabla \mathbf{w} - \int_\Omega p \nabla \cdot \mathbf{w}. \tag{10}$$

At the discrete level, we used a test function directed along $\mathbf{e}_y$, and which is zero outside the first layer of triangles in contact with Γ_l. A similar expression is used to compute $\mathbf{f}_r$. We shall denote by F the sum of $|\mathbf{f}_l|$ and $|\mathbf{f}_r|$.

In the case with no particles (stationnary state described by (7)), this quantity can be computed explicitely, which gives the reference value

$$F_0 = 2 \int_{\Gamma_l} \mu \frac{U_0}{a} = 2\mu U_0 \frac{L}{a}.$$

Apparent viscosity is defined out of F and reference value F_0

$$\mu_{\text{app}} = \frac{a}{2LU_0^2} F.$$

We shall actually represent the adimensional apparent viscosity $\mu^\star_{\text{app}} = \dfrac{\mu_{\text{app}}}{\mu}$.

5. Numerical results

5.1. **Dependency upon solid fraction**

In the first set of simulation, the interaction is not taken into account. We considered different numbers N of particles in the fluid: $N = 0$, 200, 400, 600, which corresponds to solid volume fractions $c = 0$, 0.13, 0.25, and 0.38, respectively. The following numerical values are common to all simulations:

$$a = 1\,,\ L = 2\,,\ U_0 = 0.5\,,\ \rho_p = 1.1\,,\ \mu = 0.1\,,\ r = 0.02. \tag{11}$$

All simulations are started with a zero velocity field. Adimensional apparent viscosity is computed at each time step using a discretized version of formula (9), and the corresponding graphs are plotted in Figure 6. As expected, in the situation with no particles, $\mu^\star_{\text{app}}$ decreases to 1 as the velocity field converges to the analytic stationnary solution (7). When particles are present, $\mu^\star_{\text{app}}$ still decreases to a value greater than 1. The limit value increases with the solid volume fraction.

Remark 5.1. Those curves are not smooth (expect for the case $c = 0$, for which a single mesh is used for all time steps). Some bumps appear each time a new mesh is created. It is probably due to the fact that the projection of the velocity field from the old mesh onto the new one do not preserve H^1–norm, to which hydrodynamical forces are related.

Remark 5.2. The simulation for $c = 0.38$ stopped at iteration 140, because a failure occured during generation of the periodic mesh. Such a problem can be avoided by restarting the computation with different numerical parameters (parameters on which the mesh generator is based). We nevertheless kept the incomplete curve to illustrate the fact that those computations are not yet (at least for high solid volume fractions) fully robust.

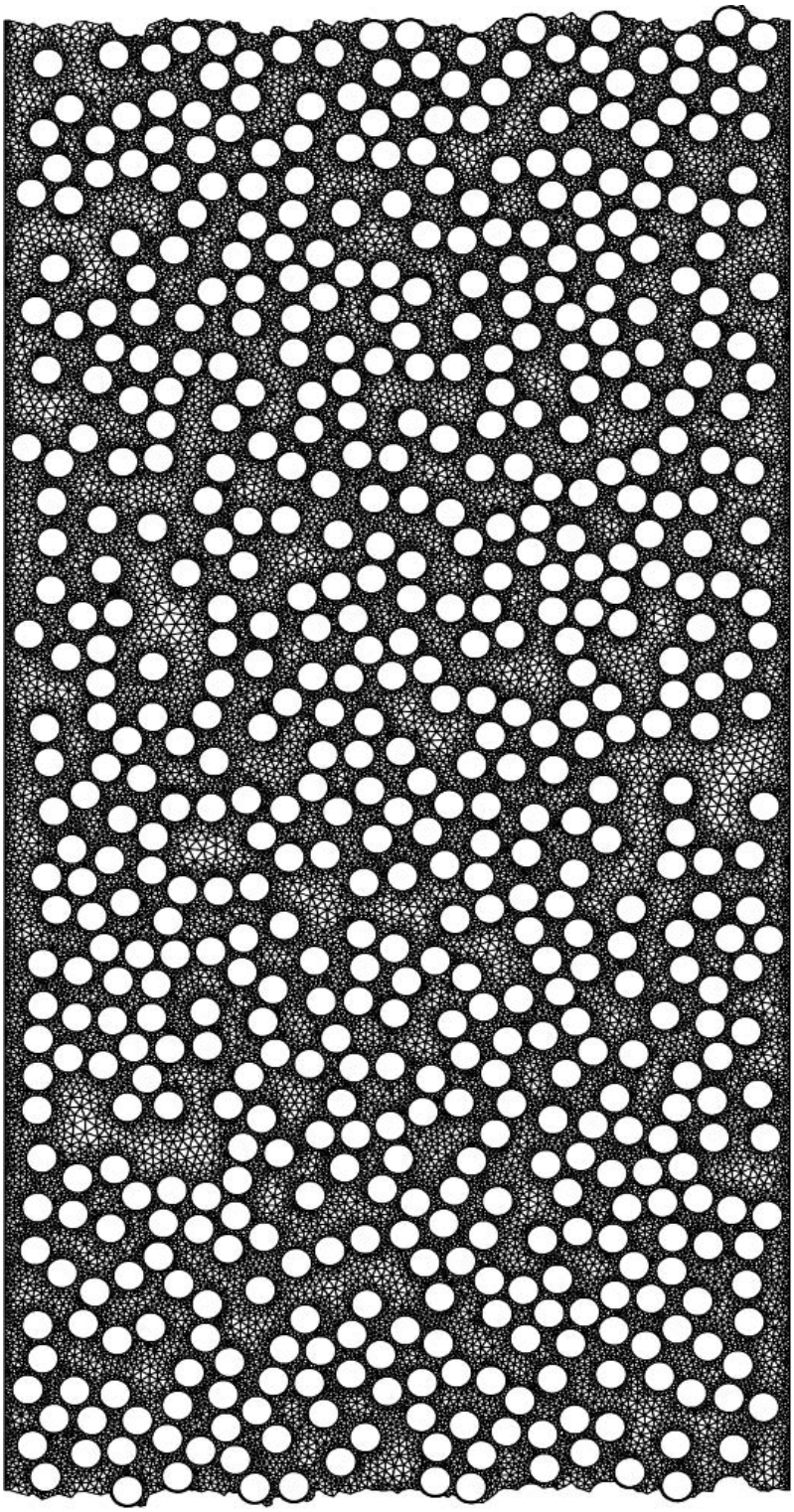

FIGURE 2. Mesh for $N = 600$, $c = 0.38$.

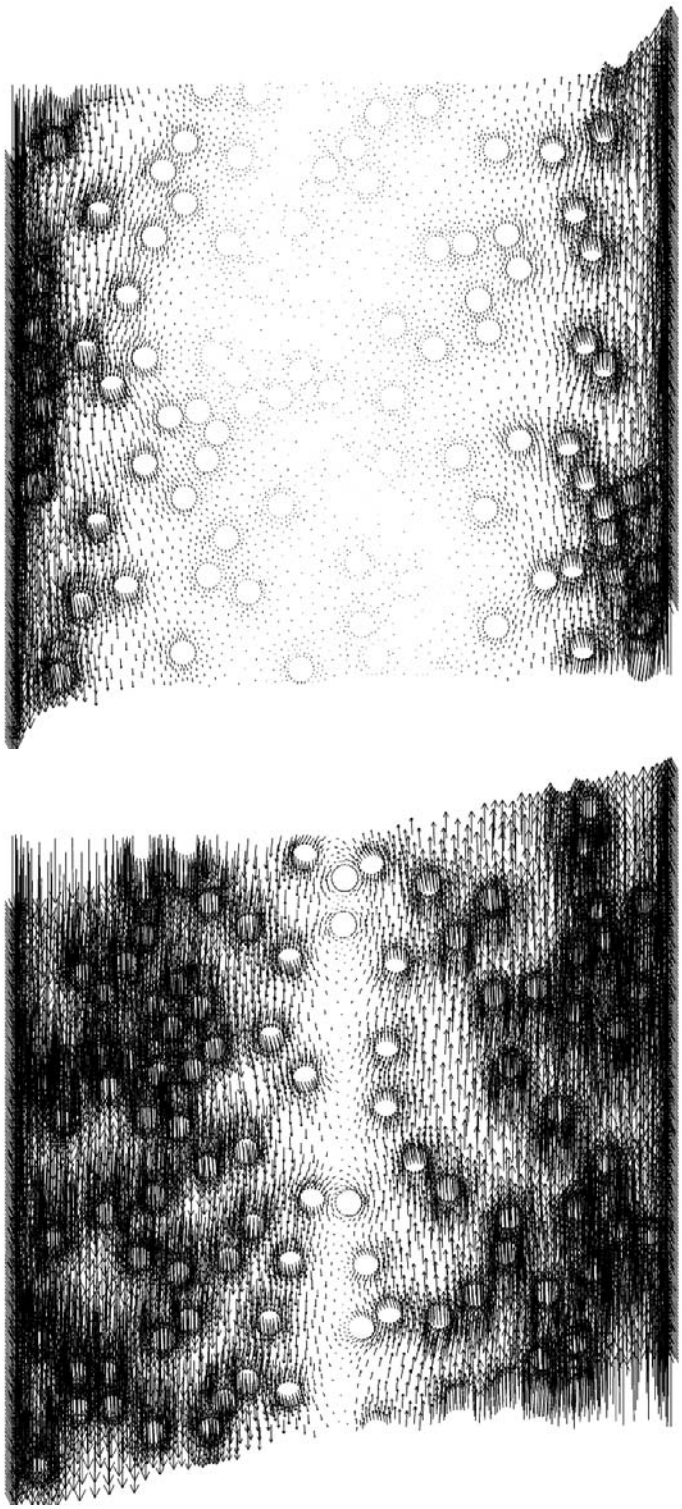

FIGURE 3. Velocity fields at times 0.01 and 1, case without interaction.

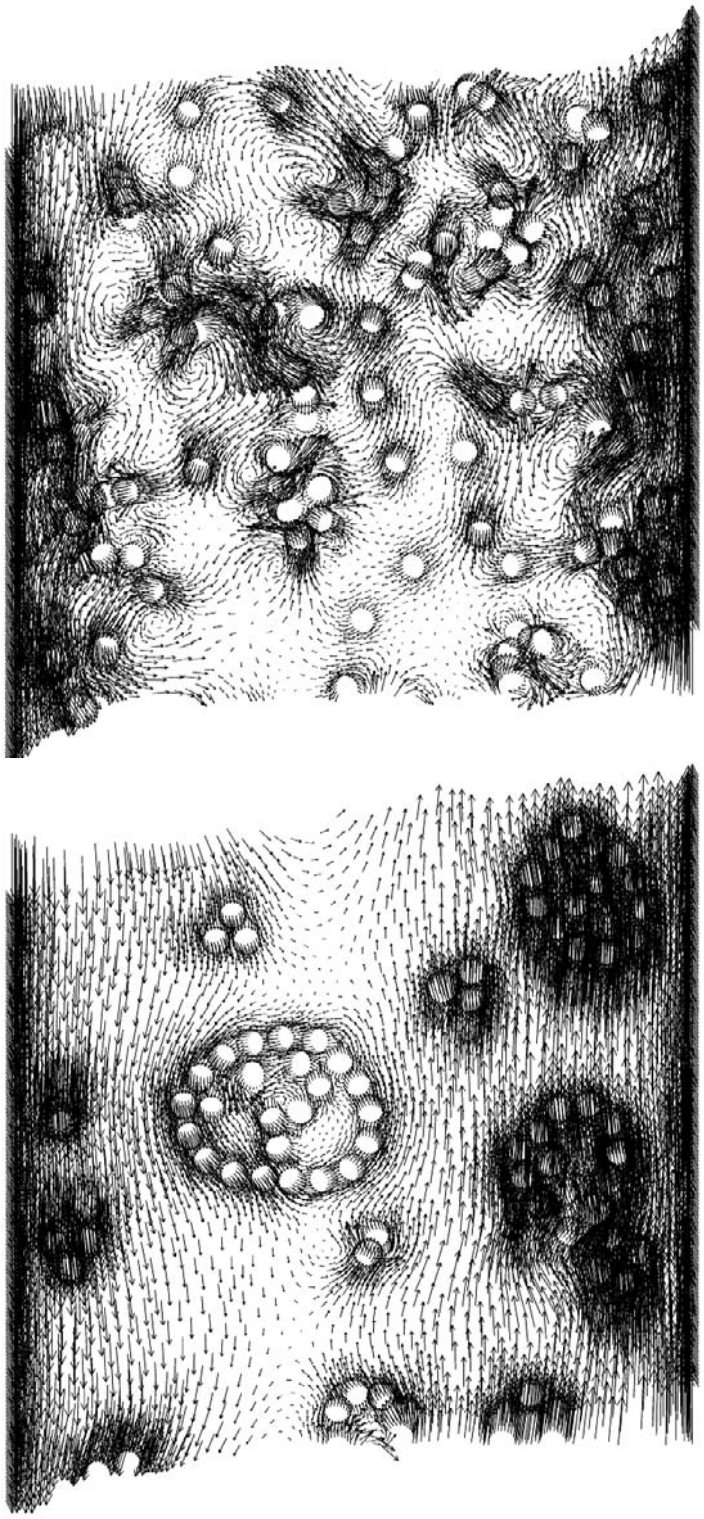

FIGURE 4. Velocity fields at times 0.01 and 1, case with interaction.

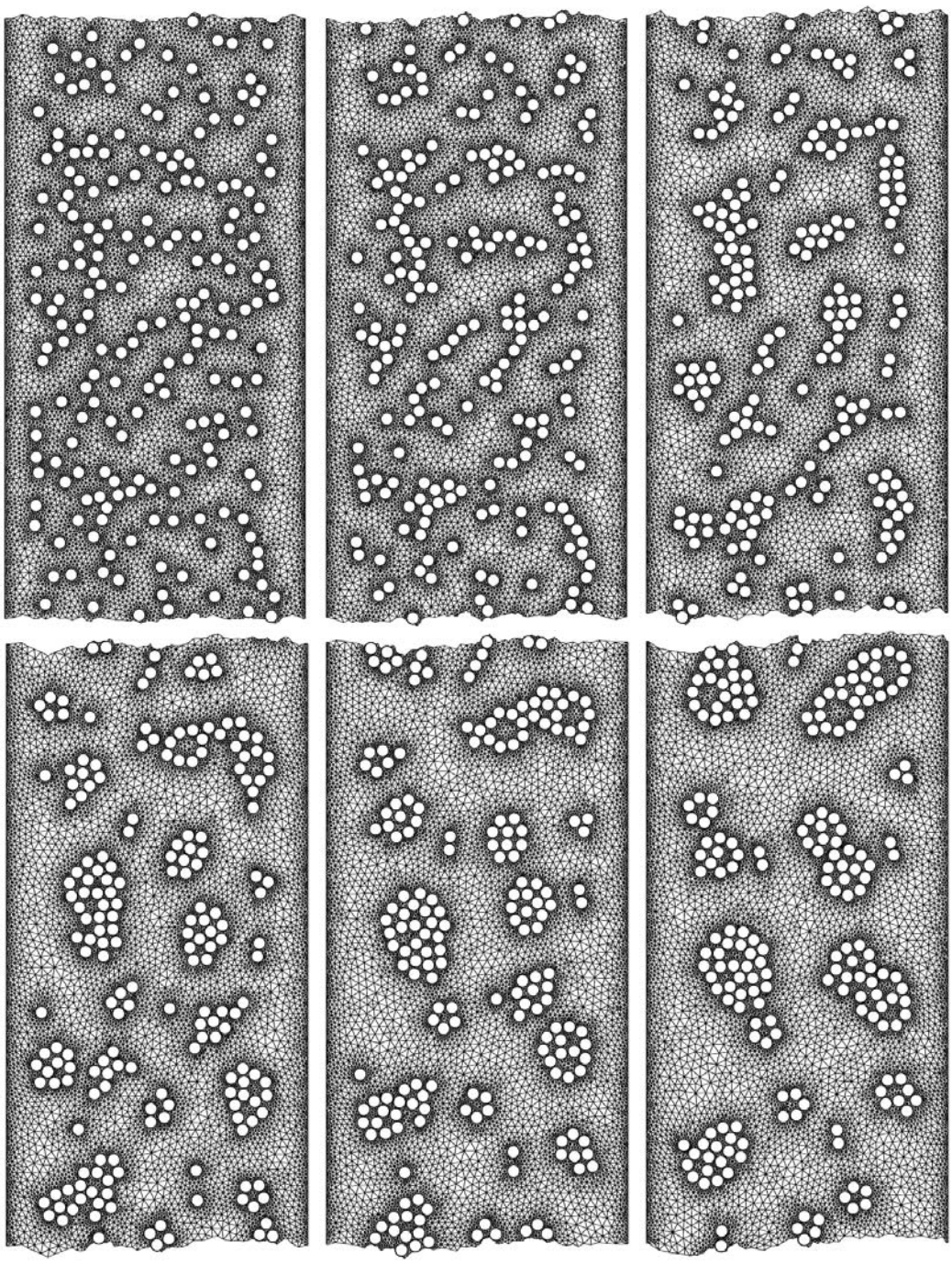

FIGURE 5. Meshes at time steps 0, 20, 50, 80, 110 and 140.

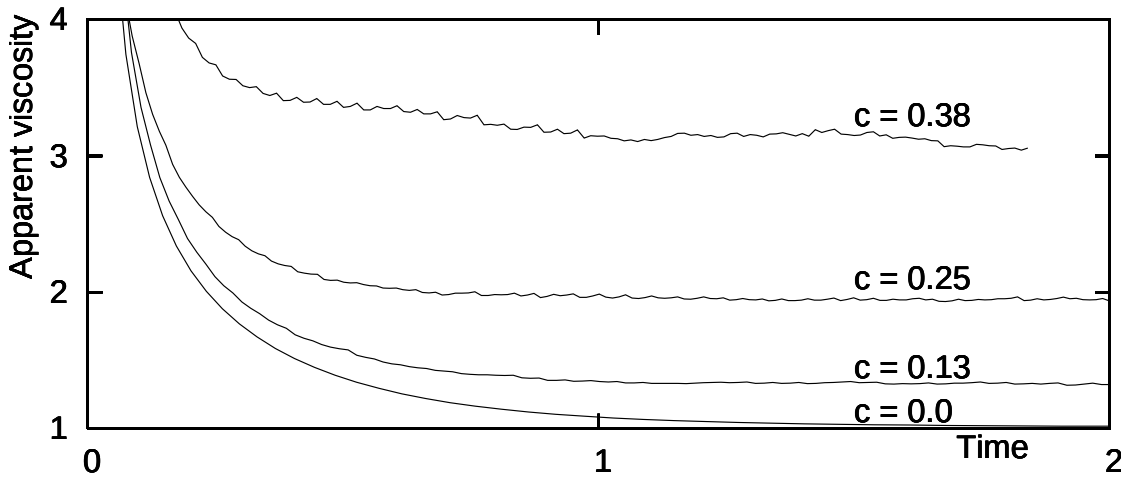

FIGURE 6. $\mu^{\star}_{\rm app}$ *versus* time for $c = 0$, 0.12, 0.25, 0.38.

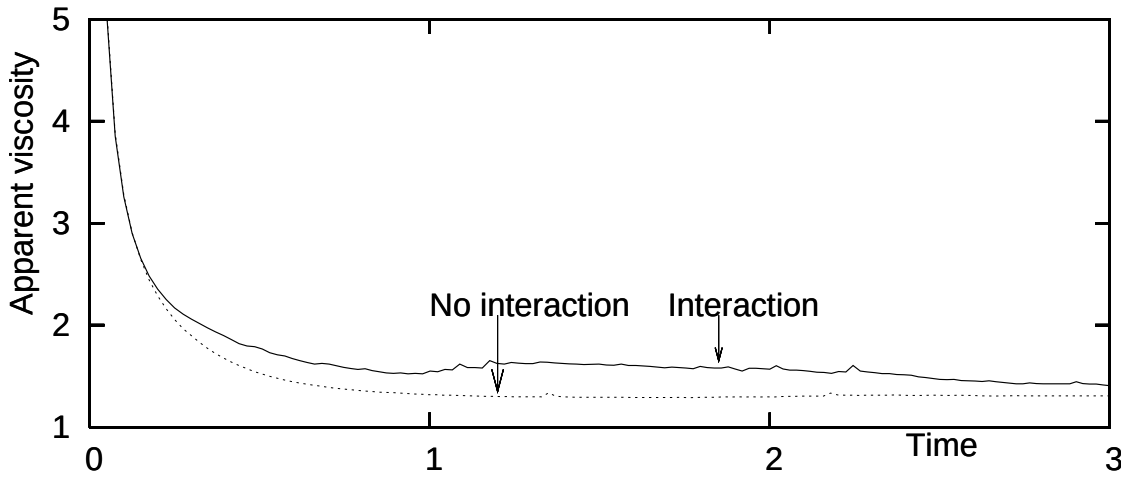

FIGURE 7. $\mu^{\star}_{\rm app}$ *versus* time for N = 200, with and without interaction.

5.2. **Interaction model**

In this second set of simulation interaction forces are added. We compare two simulations of the same population of particles ($N = 200$, $c = 0.12$), one with non–interacting particles, one with particles which interact according to force as expressed in equation (6). Following numerical values were used for the computation (r is the radius of a particle and ρ_p its density)

$$a = 1\,,\ L = 2\,,\ U_0 = 0.5\,,\ \rho_p = 1.1\,,\ \mu = 0.1\,,\ r = 0.02. \tag{12}$$

As for the interaction forces (see Eq. (6)), we chose

$$\varphi(\lambda) = \frac{0.05}{\lambda^3}\cdot \tag{13}$$

Figure 3 represents snapshots of the velocity field in the case without interaction force, near the beginning of the computation (top) and when the flow is quasi–steady (bottom). On the top we can see the lateral boundary layers, due to the fact that all velocities are initially set to zero. On the bottom, the velocity field is quite close to the analytic solution (7) (it is of course different, as this field is constrained to correspond to a rigid motion within particles). Figure 4 corresponds to the case where interaction is added. At first (top figure) the flow is clearly dominated by this interaction force. At time 1 (Fig. 4, bottom), interaction does no longer act in a predominating way, but the flow is highly modified by the presence of the clusters which have formed. Figure 5 represents meshes at different time steps.

Figure 7 plots $\mu^{\star}_{\text{app}}$ versus time in both cases (with and without interaction). Aggregates of particles which form during the simulation tend to increase slightly the apparent viscosity.

6. Conclusion

We presented a numerical framework to investigate the dependency of apparent viscosity of a fluid–particle mixture upon different parameters, including interaction between particles. Convincing comparisons with experiments will be made possible in the future only if some features of the method are improved:

- the computations should be tridimensional in order to reproduce shear flows of mixtures between parallel rotating disks or concentric rotating cylinders;
- realistic interaction models (*e.g.* interaction between red cells in the blood) must be used.

Both point are the subject of ongoing research.

References

[1] R. Folkersma, A.J.G. vanDiemen, J. Laven and H.N. Stein, Steady shear rheology of dilute polystyrene particle gels. *Rheol. Acta* **38** (1999) 257–267.

[2] R. Glowinski, T.-W. Pan, T.I. Hesla and D.D. Joseph, A Distributed Lagrange Multiplier/Fictitious Domain Method for Particulate Flows. *Int. J. Multiphas. Flow* **25** (1999) 755.

[3] H.H. Hu, Direct Simulation of Flows of Solid-Liquid Mixtures. *Int. J. Multiphas. Flow* **22** (1996) 335–352.

[4] A.A. Johnson and T.E. Tezduyar, Simulation of Multiple Spheres Falling in a Liquid-Filled Tube. *Comput. Methods Appl. M.* **134** (1996) 351–373.

[5] B. Maury, Direct Simulation of 2D Fluid–Particle Flows in Biperiodic Domains. *J. Comp. Phys.* **156** (1999) 325–351.

[6] O. Pironneau, J. Liou, T. Tezduyar, Characteristic–Galerkin and Galerkin Least Squares Space–Time Formulations for the Advection–Diffusion Equation with Time–Dependent Domains. *Comput. Meth. Appl. M.* **100** (1922) 117–141.