COMEC
The 10th International Conference on
COMPUTATIONAL MECHANICS AND VIRTUAL ENGINEERING
Transilvania University of Brasov
FACULTY OF MECHANICAL ENGINEERING

25-27 October 2023

MODELING AND SIMULATION OF RIGID PARTICULATE FLOW IN AN INCOMPRESSIBLE NEWTONIAN FLUID

Salah ZOUAOUI[*1]

1. LMSE Laboratory, Mechanical Engineering Department Mouloud Mammeri University of Tizi-Ouzou P.O. Box 17 RP15000 Algeria, salah zouaoui@ummto.dz
[*]Corresponding author: salah.zouaoui@ummto.dz

Abstract: *We present a penalization method for the simulation of the interaction of an incompressible Newtonian fluid flow with rigid bodies. This method is based on a variational formulation throughout the fluid/solid domain, with constraints on the unknown and on the test functions. The rigid motion of the particle is enforced by penalizing the strain tensor on the rigid domain for canceling the deformation rate in the volume occupied by the particle. A simulations , implemented with FreeFem++, of a rigid particles in a incompressible Newtonian fluid flow are performed. The method is first validated by simulating the Sheared particle and then aplicated for simulating the sedimentation of one single particle and two identical particles in an incompressible fluid.*
Keywords: *Modelling, Simulation, Newtonian Fluid, fluid/particles interaction, DKT,*

1. INTRODUCTION

Transport phenomena and solid particles deposit in the context of hydraulic turbine engine systems is multidisciplinary. For modeling and simulation of such flows, there are several methods which use dynamic meshing. These methods follow the movement of the objects in a Lagrangian way [1, 2, 3]. However, the remeshing steps can be expensive and very difficult, especially in the 3D case. To overcome this problem the fictitious domain methods also called domain embedding methods are used. The idea is to extend a problem defined on a time-dependent, complex domain (the fluid domain)to a larger one (fixed) called the fictitious domain [4, 5]. Penalty methods are another class of fictitious domain strategies [6].

The penalty method is based on a reformulation of the stress tensor for canceling the deformation rate in the volume occupied by the particle. It consists on constraining the movement of the fluid to be a rigid body motion identical to that of a particle by locally increasing the viscosity of the fluid [7, 8, 9]. Recently this method has been extended to manage of stress rigid motion for a particle in a fluid to an approach of finite difference type and then by a finite elements method. [10, 11]. This method is implemented on a general Finite Element solver which we use to make numerical tests.

In this paper we present a method of simulating the movement of one or more convex rigid body in a Newtonian incompressible fluid. We used a penalty method which is based on a reformulation of the stress tensor which allows the canceling of the deformation rate in the volume occupied by the particle. The objective is to develop a code from FreeFem++ that simulates Stokes or Navier-Stokes flows (with low Reynolds number) in the presence of solid particles. To validate this method, we apply this method to simulate the case of a particle which is subjected to shear fields. Thereafter we applied this method for simulating the sedimentation of one single particle.

2. MATHEMATICAL FORMULATION OF THE FLUID-STRUCTURE PROBLEM

We consider a connected bounded and regular domain $\Omega \subset R^2$ and we denote by $(B_i)_{i=1,\dots,N}$ the rigid particles, strongly included in Ω. B denotes the whole rigid domain: $B = \cup_i B_i$ (see Figure 1).

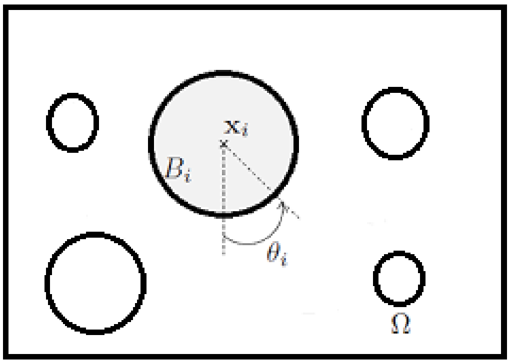


Figure 1: Particles B_i in a Newtonian fluid

The domain $\Omega\backslash\overline{B}$ is filled with Newtonian fluid governed by the Navier-Stokes equations. Here, μ is the viscosity of the fluid and ff the external forces exerted on it. Since we consider a Newtonian fluid, the stress tensor σ writes

$$\sigma = 2\mu D(u) - pId, \text{ where } D(u) = \frac{\nabla u + (\nabla U)^T}{2} \quad (1)$$

and p is the pressure. We will consider homogeneous Dirichlet conditions on $\partial\Omega$. On the other hand, viscosity imposes a no-slip condition on the boundary ∂B of the rigid domain.

At the initial time the particles with density ρ_i are distributed randomly over the fluid (without overlapping). The position of the center of the i^{th} particle is denoted by x_i, and by v_i and ω_i its translational and angular velocities. We denote by mi and Ji the mass and the kinematic momentum about its center of mass:

$$m_i = \int_{B_i} \rho_i \,, \qquad J_i = \int_{B_i} \rho_i \, \|x - x_i\|^2 \quad (2)$$

We have to find the velocity u = (u1; u2) and the pressure field p defined in $\Omega\backslash\bar{B}$, as well as the velocities of the particles $V := (v_i)_{i=1,..,N} \in R^{2N}$ and $\omega := (\omega_i)_{i=1,....,N} \in R^N$ such that:

$$\begin{cases} \rho_f\,(\frac{Du}{Dt} + u.\nabla u) - \mathrm{div}(\sigma) = \mathrm{f_f} \quad \text{in } \Omega\backslash\bar{B} \\ \nabla.u = 0 \quad \text{n } \Omega\backslash\bar{B} \\ u = 0 \quad \text{on } \partial\Omega \end{cases} \tag{3}$$

where ρ_f denotes the density of the fluid and $f_f = \rho_f\, g e_y$ is the external force exerted on the fluid (gravity forces). The viscosity imposes a no-slip boundary condition on ∂B:

$$u = v_i + \omega_i(x - x_i)^{\perp} \quad \text{on } \partial B_i,\ \forall i \in \{1, \dots, N\} \tag{4}$$

Finally, since the fluid exerts hydrodynamic forces on the particles, Newton's second law couples these equations:

$$\begin{cases} m_i\,\frac{dV_i}{dt} = \int_{B_i} f_i - \int_{\partial B_i} \sigma n \\ J_i\,\frac{d\omega_i}{dt} = \int_{B_i} (x - x_i)^{\perp}\,.f_i - \int_{\partial B_i} (x - x_i)^{\perp}\,.\sigma n \end{cases} \tag{5}$$

Here, f_i denotes the external non-hydrodynamical forces exerted on the sphere, such as gravity : $f_i = -\rho_i g e_y$.

Using a penalty method, we obtain The variational formulation obtained on the whole fluid/particle domain is given here after:

$$\begin{cases} \int_\Omega \tilde{\rho}(\,\frac{Du}{Dt} + \mathrm{u}.\nabla \mathrm{u}) - 2\mathrm{div}\big(\tilde{\mu}\mathrm{D}(\mathrm{u})\big) + \nabla p = \hat{\mathrm{f}} \\ \mathrm{div}(\mathrm{u}) = 0 \end{cases} \tag{14}$$

with

$$\tilde{\mu} := \rho_f 1_{\Omega\backslash\bar{B}} + \frac{1}{\varepsilon} 1_B \tag{15}$$

and

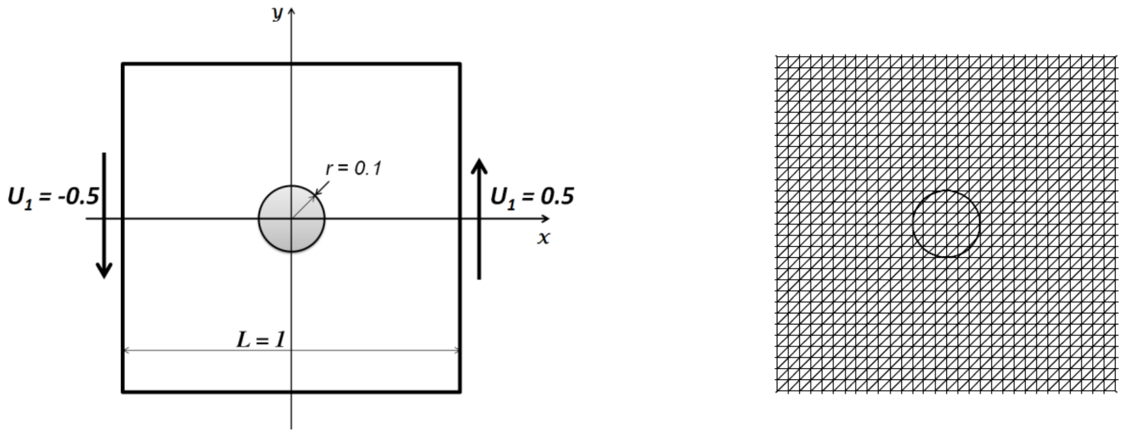
$$\hat{\mathrm{f}} := \tilde{\mathrm{f}} - \mathrm{f_f} = \sum_{i=1}^{N} (\mathrm{f}_i - \mathrm{f_f})\, 1_{B_i} \tag{16}$$

The time discretization is performed by using the method of characteristics [12].

3. Results

3.1. Sheared particle

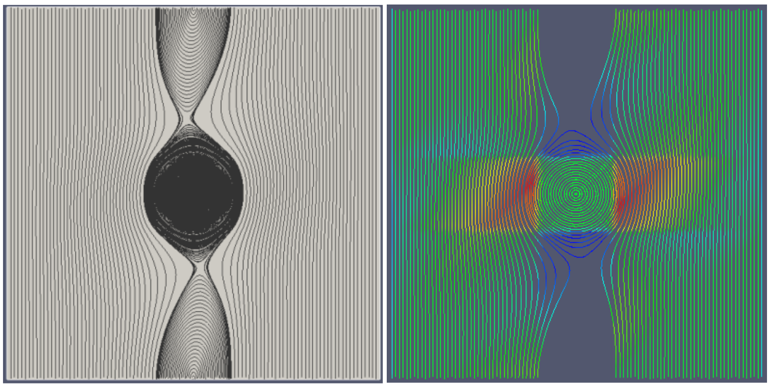
We consider the instantaneous problem of a particle in a Newtonnain fluid. The computational domain is a square 1cm wide and a particle of radius 0.1cm is located at its center . The right and left walls of the domain impose a shearing motion to the system, the viscosity of the fluid is equal to 1 . A cartesian mesh is used (see figure. 2).



a) **b)**

Figure 2: Sheared particle: a) Physical domain; b) Cartesian meshes

On table (1) below we present the angular velocity as a function of the particle radius. We show that the angular velocity of the particle converges to the theoretical value which is equal to 0.5 [13, 14]. The streamlines and the velocity field of the shear movement are respectively shown in figure 3. This figure shows the streamlines of the rotational motion.



a) **b)**

Figure 3: Sheared particle: a) Stream line; b)Velocity field

Table 1: Angular velocity of a sheared particle

Radius(r_p)	ω	**Radius(r_p)**	ω	**Radius(r_p)**	ω	**Radius(r_p)**	ω
0.6	0.1804	0.15	0.249977	0.00385	0.47697	0.0038049	0.499989
0.45	0.249947	0.1	0.251729	0.00383	0.487011	0.00380489	0.499995
0.25	0.249492	0.01	0.356297	0.00381	0.497318		
0.17	0.249895	0.0039	0.452976	0.003801	0.499937		

3.2. Spherical particle sedimentation

To validate our code, we considered the data used in references [15, 16]. In this test case, we performed the simulation of a rigid particle sedimentation, with density $\rho_s=1.5 g.cm^{-3}$, under the effect of the gravity force ($g=-980\ cm.s^{-2}$) in an incompressible viscous fluid of density $\rho_l=1.0 g.cm^{-3}$ and viscosity $\mu=0.01 g.(cm.s)^{-2}$. The computational domain is a *6cm* x *2cm* size rectangle. The particle of radius $r = 0.125\ cm$ is located at a height of *4 cm* in the middle of the rectangle. A comparison from a qualitative point of view, with the work of Bost [17], representing the iso-lines of vorticity created by the motion of the particle, shows very good agreement. Figure4 which shows the iso-lines of vorticity at different times, highlights the symmetry of the two vortices created at the particle-fluid interface. We can see very little vorticity inside the particle**.**

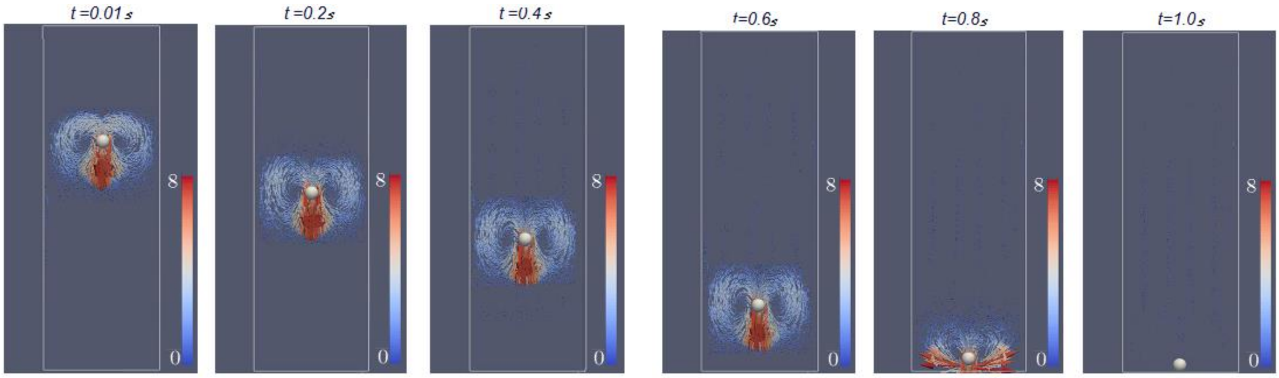


Figure 4: Iso-lines of vorticities at different time steps.

4. Conclusion

In this work, we have proposed a strategy for the numerical modeling of of a rigid particle motion in a Newtonian fluid. The rigid motion is imposed by penalizing the strain tensor. The time discretization is performed by using the method of characteristics.

The code was written in FreeFem++ version 3.26 and at each time step the generalized Navier-Stokes problem is solved by using standard finite elements. From the results, we notice that the stress of rigid motion is taken into account. These results are similar to those existing in the literature. For the Sheared particle, We show that the angular velocity of the particle converges to the theoretical value. The sedimentation of one circular particle in a rectangular box is used as the second test problem. The iso-lines of vorticity at different times, highlights the symmetry of the two vortices created at the particle-fluid interface. the sedimentation of two identical particles in an incompressible fluid show that our code can reproduce the physical behavior of non-stationary flows .

BIBLIOGRAFIE

[1] Hu, Howard H and Patankar, Neelesh A and Zhu, MY Direct numerical simulations of fluid–solid systems using the arbitrary Lagrangian–Eulerian technique, Journal of Computational Physics 169, 2, pp. 427-462 (2001).

[2] A. Lefebvre and B. Maury Apparent viscosity of a mixture of a Newtonian fluid and interacting particles, Comptes rendus mécanique 333, 12, pp. 923–933 (2005).

[3] B. Maury Direct simulations of 2D fluid-particle flows in biperiodic domains, Journal of computational physics 156, 2, pp. 325-351 (1999).R. Nicole, Title of paper with only first word capitalized, J. Name Stand. Abbrev, in press.

[4] R Glowinski, TW Pan, TI Hesla, DD Joseph, and J Periaux. A _ctitious domain approach to the directnumerical simulation of incompressible viscous flow past moving rigid

bodies: application to particulate floow. Journal of Computational Physics, 169(2):363_426, 2001.

[5] Neelesh A Patankar, Pushpendra Singh, Daniel D Joseph, Roland Glowinski, and T-W Pan. A new formulation of the distributed lagrange multiplier/fictitious domain method for particulate flows. International Journal of Multiphase Flow, 26(9):1509_1524, 2000.

[6] Philippe Angot, Charles-Henri Bruneau, and Pierre Fabrie. A penalization method to take into account obstacles in incompressible viscous flows. Numerische Mathematik, 81(4):497_520, 1999.

[7] J Caltagirone and S Vincent. Tensorial penalisation method for solving navier-stokes equations. Comptes Rendus de l'Academie des Sciences Series IIB Mechanics, 329(8):607_613, 2001.

[8] Stéphane Vincent, Jorge César Brändle De Motta, Arthur Sarthou, Jean-Luc Estivalezes, Olivier Simonin, and Eric Climent. A lagrangian vof tensorial penalty method for the dns of resolved particle-laden flows. Journal of Computational Physics, 256:582_614, 2014.

[9] Stéphane Vincent, Tseheno Nirina Randrianarivelo, Grégoire Pianet, and Jean-Paul Caltagirone. Local penalty methods for flows interacting with moving solids at high reynolds numbers. Computers & fluids, 36(5):902_913, 2007.

[10] Joao Janela, Aline Lefebvre, and Bertrand Maury. A penalty method for the simulation of fluid-rigid body interaction. In ESAIM: Proceedings, volume 14, pages 115_123. EDP Sciences, 2005

[11] Aline Lefebvre. Fluid-particle simulations with freefem++. In ESAIM: Proceedings, volume 18, pages 120_132. EDP Sciences, 2007.

[12] O. Pironneau, J.Liou and T.Tezduyar, Characteristic-galerkin and galerkin/least-squares space-time formulations for the advection-diffusion equation with time-dependent domains, Computer Methods in Applied Mechanics and Engineering 100, pp.117-141 (1994).

[13] Batchelor, GK and Green, J-T The hydrodynamic interaction of two small freely-moving spheres in a linear flow field, Journal of Fluid Mechanics,56, 02, pp.375–400 (1972).

[14] Salah ZOUAOUI. Modlisation et Simulation des Ecoulements Multiphasiques bases sur une Approche Multi-Echelles. -Application au Transport Solide-. PhD thesis, UniversitMouloud Mammeri de Tizi-Ouzou, 2016.

[15] Joao Janela, Aline Lefebvre, and Bertrand Maury. A penalty method for the simulation of fluid-rigidbody interaction. In ESAIM: Proceedings, volume 14, pages 115_123. EDP Sciences, 2005.

[16] Aline Lefebvre. Fluid-particle simulations with freefem++. In ESAIM: Proceedings, volume 18, pages 120_132. EDP Sciences, 2007.

[17] Claire Bost. Méthodes Level-Set et pénalisation pour le calcul d'interactions fluide-structure. PhD thesis, Universit Joseph-Fourier-Grenoble I, (2008).