Tenth International Conference on Computational Fluid Dynamics (ICCFD10), Barcelona,Spain, July 9-13, 2018

ICCFD10-2018-388

Simulation of floating platforms for marine energy generation

M. Moragues Ginard[1], N. C. Degirmenci[1,3], D. Castañón Quiroz[2], M. Leoni[1,3], J. Jansson[1,3], V. Nava[1,4], E. Krishnasamy[1,3] and J. Hoffman[3]
Corresponding author: mmoragues@bcamath.org

[1] BCAM Basque Center for Applied Mathematics, Bilbao, Spain.
[2] IMAG, Univ Montpellier, CNRS, Montpellier, France.
[3] KTH Computational Technology Laboratory, Stockholm, Sweden.
[4] Tecnalia R&I, Bilbao, Spain.

Abstract: The goal of this work is to study the dynamics of floating platforms that are designed for marine energy generation. This work is done in collaboration with Tecnalia R&I, a company settled in the Basque Country which designs this kind of platforms. To our purpose we present a method for the simulation of two-phase flow with the presence of floating bodies. We consider the variable density incompressible Navier-Stokes equations and discretize them by the finite element method with a variational multiscale stabilization. A level-set type method is adopted to model the interphase between the two fluids. The mixing or smearing in the interphase is prevented with a compression technique. Turbulence is implicitly modeled by the numerical stabilization. The floating device simulation is done by a rigid body motion scheme where a deforming mesh approach is used. The mesh deforms elastically following the movement of the body. Simulation of a decay test on a cube is performed and the results are presented in this paper. All the simulations are done with the open source finite elements parallel software FEniCS-HPC.

Keywords: Two-phase flow , finite elements, floating bodies, FEniCS-HPC.

1 Introduction

Free surface flows have been modeled by different methodologies, some of them being marker-and-cell, volumes of fluid, level-set, etc. A quite complete overview of the different methods can be found in [1]. In this work we consider the variable density incompressible Navier-Stokes equations and take a level-set type approach to model the interface between the two fluids. The space discratization is done by the finite element method (FE) with a variational multiscale stabilization (VMS) and a discontinuity capturing technique. In the level-set approach the mixing or smearing in the interphase is prevented with a compression technique [10, 2]. Turbulence is not modeled by an explicit Large eddy simulation (LES) scheme but implicitly modeled by the numerical stabilization as in the VMS-LES approaches. The floating device simulation is done by a rigid body motion scheme where a deforming mesh approach [3] is used. Time discretization is done by a Crank-Nicholson scheme, Newton's iterations are used for linearization, and velocity and pressure are solved separately by a projection method [4]. Implementation and simulations are done with the open source software framework FEniCS-HPC [5].

The present paper is organized as follows. In section 2 we present the method for the simulation of two-phase flows. Simulation results of a decay test on a cube are presented in section 3 being the final goal of this work the study of the dynamics of floating platforms related to marine energy generation in collaboration with Tecnalia R&I. Finally conclusions and future work are given in section 4.

2 Problem Statement

Two-phase flow is modeled in this work by means of the variable density incompressible Navier-Stokes equations that expressed in dimensionless form read

$$\rho\,\partial_t \mathbf{u} + \rho\,(\mathbf{u}\cdot\nabla)\mathbf{u} - \frac{1}{Re}\,\nabla\cdot(\mu\,\nabla\mathbf{u}) + \nabla p - \frac{1}{Fr^2}\,\rho\,\mathbf{e}_g = 0 \tag{1}$$

$$\nabla\cdot\mathbf{u} = 0 \tag{2}$$

$$\partial_t\rho + \mathbf{u}\cdot\nabla\rho = 0 \tag{3}$$

The equations (1)-(3) are defined in some space time domain, the unknowns being the velocity vector $\mathbf{u}$, the pressure p, and the density ρ. System (1)-(3) is composed of the equation for the conservation of the momentum, the continuity equation, and a transport equation for the density here used to track the interface as in the level-set type schemes. In the momentum equation μ is the dynamic viscosity, $Re = \frac{\rho_{\mathrm{ch}} L_{\mathrm{ch}} u_{\mathrm{ch}}}{\mu_{\mathrm{ch}}}$ is the Reynolds number, and $Fr = \frac{u_{\mathrm{ch}}}{\sqrt{g\,L_{\mathrm{ch}}}}$ is the Froude number, where u is the velocity norm, L is the length, and the subindex $(\cdot)_{\mathrm{ch}}$ has the meaning of characteristic or reference quantity that depends on each particular problem, $g = 9.81\,m/s^2$ is the gravity norm, and $\mathbf{e}_g$ is a unit vector in the direction of gravitation. The initial condition for the density is $\rho = \rho_1$ in the domain occupied by the first fluid and $\rho = \rho_2$ in the domain occupied by the second fluid, where $\rho_1 < \rho_2$.

System (1)-(3) is discretized in space by the finite element method. A variational multiscale stabilization (VMS) is applied to the finite element discrete form and turbulence is implicitly modeled by the stabilization diffusion-like terms. To prevent instabilities in the interface between the two fluids a discontinuity capturing technique is considered. And finally to avoid smearing in the interface, a compression technique is used. This leads to the following weak form of system (1)-(3):

$$\begin{aligned}&(\rho\,\partial_t\mathbf{u},\,\mathbf{v}) + (\rho\,(\mathbf{u}\cdot\nabla)\mathbf{u},\,\mathbf{v}) + \frac{1}{Re}\,(\mu\,\nabla\mathbf{u},\,\nabla\mathbf{v}) + (\nabla p,\,\mathbf{v}) - \frac{1}{Fr^2}\,(\rho\,\mathbf{e}_g,\,\mathbf{v})\\ &+ (\nabla\cdot\mathbf{u},\,q) + (\tau\,\mathbf{R}(\mathbf{u},p),\,\mathcal{L}^*(\mathbf{v},q)) + (\mu_{\mathrm{DC,m}}\,\nabla\mathbf{u},\nabla\mathbf{v}) = 0\end{aligned} \tag{4}$$

$$(\partial_t\rho,\,\eta) + (\mathbf{u}\cdot\nabla\rho,\,\eta) + (\tau_\rho\,\mathbf{u}\cdot\nabla\rho,\,\mathbf{u}\cdot\nabla\eta) + (\mu_{\mathrm{DC},\rho}\,\nabla\rho,\nabla\eta) = 0 \tag{5}$$

In (4)-(5), $\mathbf{v}$, q, and η are the finite element test functions for the momentum, continuity, and density equations, respectively. Test functions are here first order Lagrange polynomials. The seventh term in (4) and the third term in (5) are VMS stabilization terms [6, 7]. The last term in (4) and the fourth term in (5) are discontinuity capturing terms [8, 9]. A compression condition is also applied to the transport equation for the density [10, 2]. The strong residual of the momentum and continuity equations are

$$\mathbf{R}(\mathbf{u},p) = \begin{pmatrix} \mathbf{R}_{\mathrm{m}} \\ R_{\mathrm{c}} \end{pmatrix} = - \begin{pmatrix} \rho\,(\mathbf{u}\cdot\nabla)\mathbf{u} - \frac{1}{Re}\,\nabla\cdot(\mu\,\nabla\mathbf{u}) + \nabla p - \frac{1}{Fr^2}\,\rho\,\mathbf{e}_g \\ \nabla\cdot\mathbf{u} \end{pmatrix}\,, \tag{6}$$

and the corresponding space differential dual operator writes

$$\mathcal{L}^*(\mathbf{v},q) = \begin{pmatrix} -\rho\,(\mathbf{u}\cdot\nabla)\mathbf{v} - \frac{1}{Re}\,\nabla\cdot(\mu\,\nabla\mathbf{v}) - \nabla q \\ -\nabla\cdot\mathbf{v} \end{pmatrix}\,. \tag{7}$$

System (4)-(5) is discretized in time by a Crank-Nicholson scheme, linearization is done by Newton's iterations, and velocity and pressure are computed separately using a projection method [4].

3 Floating cube decay test

We simulate three-dimensional two-phase flow on a floating cube. The decay test consists of giving a downwards initial velocity to the cube and study the oscillation that occurs. We call x_1, x_2, and x_3 the three space coordinates. This is a preliminary test case were the force of the fluid on the cube is only taken into account in the vertical direction x_2, then the cube only moves in this direction. To this purpose, a simplification of the rigid body motion [11] is used here and briefly described in what follows. We compute the vertical force applied by the fluid on the cube boundary by $\mathbf{F} = \rho_c\,V_c\,g + \int_{\Gamma_c} p\,\mathbf{e}_2\,ds$, where ρ_c, V_c, and Γ_c are the density, the volume, and the boundary of the cube, respectively, and $\mathbf{e}_2$ is the unit vector in the x_2 direction. By the Newton's second law $\mathbf{F} = \rho_c\,V_c\,\partial_t\mathbf{w}$, we compute $\mathbf{w}$ which is the vertical velocity of the cube. The mesh moves by an ALE approach, the boundary of the cube is tracked with the mesh using the cube velocity $\mathbf{w}$ and the rest of the nodes of the mesh move following an elasticity equation [3].

The dimensions of the computational domain are $-4 \leq x_1 \leq 4$, $-4 \leq x_2 \leq 4$, and $-4 \leq x_3 \leq 4$. A mesh of $353,740$ grid points and $2,075,923$ tetrahedra is used for this simulation.The minimum edge lenght of the mesh being 0.0308. The mesh near the cube on a vertical cut and on the cube surface is represented in Figure 1. The floating cube is initially defined by $-0.5 \leq x_1 \leq 0.5$, $-0.5 \leq x_2 \leq 0.5$, and $-0.5 \leq x_3 \leq 0.5$, being the center of mass of the cube placed at the origin (0, 0, 0) when the simulation starts. The flow is initially at rest and the cube has an initial vertical velocity of (0, -2.5, 0). The non dimensional form of the Navier-Stokes equations (1)-(3) are used in this paper. Then the non dimensional densities of the two flows are $\rho_1 = 1e-3$ and $\rho_2 = 1$, and the non dimensional density of the cube is set to $\rho_c = 0.5$. The interphase between the two fluids is the plane $x_2 = 0.0$, then the initial density is ρ_2 when $x_2 \leq 0.0$ and ρ_1 otherwise. In this preliminary test case we prescribe a uniform non dimensional viscosity of $\mu = 1.0e-6$, in the future we should use different viscosities for the two fluids. The characteristic scales used in this test case for the nondimensionalization of the equations are: $\rho_{\mathrm{ch}} = 1000\,kg/m^3$, $\mu_{\mathrm{ch}} = 1000\,kg/sm$, $L_{\mathrm{ch}} = 1\,m$, $U_{\mathrm{ch}} = 1\,m/s$. Slip boundary conditions are set everywhere.

In Fig. 2 we plot the filled contours of the density on a x_1x_2-plane passing through the origin of the domain, for different times: $0\,s$, $0.28\,s$, $0.84\,s$, and $1.44\,s$. In Fig. 3 we plot the vertical displacement of the center of mass of the cube over time until a final time of $8\,s$. We observe the expected decay of the vertical movement of the cube when time advances.

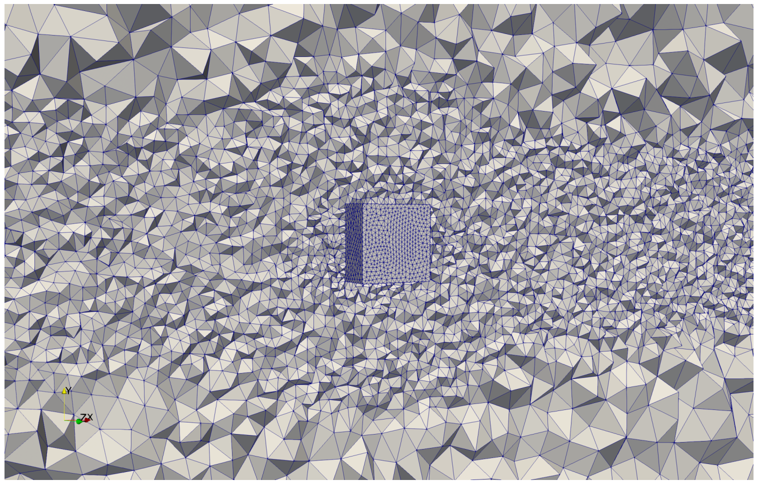

Figure 1: Mesh near the cube on a vertical cut and on the cube surface.

4 Conclusion and Future Work

A level-set type approach with compression has been introduced for the simulation of two-phase flows. A finite element method has been used for space discretization, stabilized by a VMS stabilization and a discontinuity capturing technique has been applied as well. Time integration has been done by a Crank-Nicholson scheme. Results of a decay test on a cube have been presented. As future work, the present method has to be further validated and applied to the simulation of floating platforms for marine energy generation.

References

[1] Marcela Cruchaga, Laura Battaglia, Mario Storti, and Jorge D'Elía. Numerical modeling and experimental validation of free surface flow problems. *Archives of Computational Methods in Engineering*, 23(1):139–169, 3 2016.

[2] Elin Olsson and Gunilla Kreiss. A conservative level set method for two phase flow. *Journal of Computational Physics*, 210(1):225 – 246, 2005.

[3] A.A. Johnson and T.E. Tezduyar. Mesh update strategies in parallel finite element computations of flow problems with moving boundaries and interfaces. *Computer Methods in Applied Mechanics and Engineering*, 119(1):73 – 94, 1994.

[4] G. Houzeaux, M. Vázquez, R. Aubry, and J. M. Cela. A massively parallel fractional step solver for incompressible flows. *Journal of Computational Physics*, 228:6316–6332, 2009.

[5] J. Hoffman, J. Jansson, and N. Jansson. FEniCS-HPC: Automated predictive high-performance finite element computing with applications in aerodynamics. In Dongarra J. Karczewski K. Kitowski J. Wiatr K. (eds) Parallel Processing Wyrzykowski R., Deelman E. and Applied Mathematics, editors, *Proceedings of the 11th International Conference on Parallel Processing and Applied Mathematics, PPAM 2015. Lecture Notes in Computer Science*, volume 9573, pages 356–365, Springer, Cham, 2016.

[6] G. Houzeaux and J. Principe. A variational subgrid scale model for transient incompressible flows. *International Journal of Computational Fluid Dynamics*, 22:135–152, 2008.

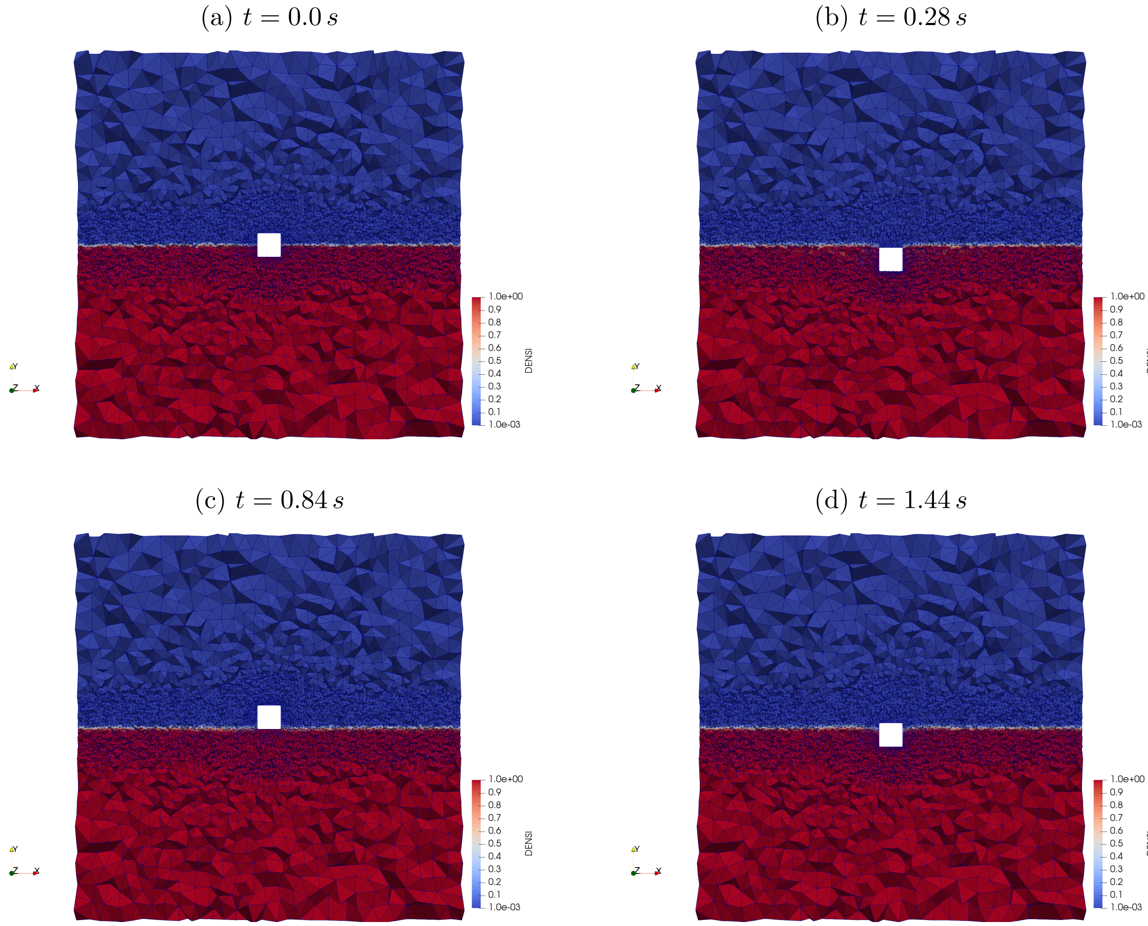

Figure 2: Filled contours of the density on a slice of the domain on the x_1x_2-plane, for different times, from $t = 0.0\,s$ to $t = 1.44\,s$.

[7] R. Codina. Stabilized finite element approximation of transient incompressible flows using orthogonal subscales. *Comput. Methods Appl. Mech. Engrg.*, 191:4295–4321, 2002.

[8] R. Codina. A discontinuity-capturing crosswind-dissipation for the finite element solution of the convection-diffusion equation. *Comput. Methods Appl. Mech. and Engrg.*, 110:325–342, 1993.

[9] C. Johnson, A. Szepessy, and P. Hansbo. On the convergence of shock-capturing streamline diffusion finite element methods for hyperbolic conservation laws. *Mathematics of Computation*, 54:107–129, January 1990.

[10] A. Harten. High resolution schemes for hyperbolic conservation laws. *Journal of Computational Physics*, 135(2):260–278, 1997.

[11] Roozbeh Panahi, Ebrahim Jahanbakhsh, and Mohammad S. Seif. Development of a VoF-fractional step solver for floating body motion simulation. *Applied Ocean Research*, 28(3):171 – 181, 2006.

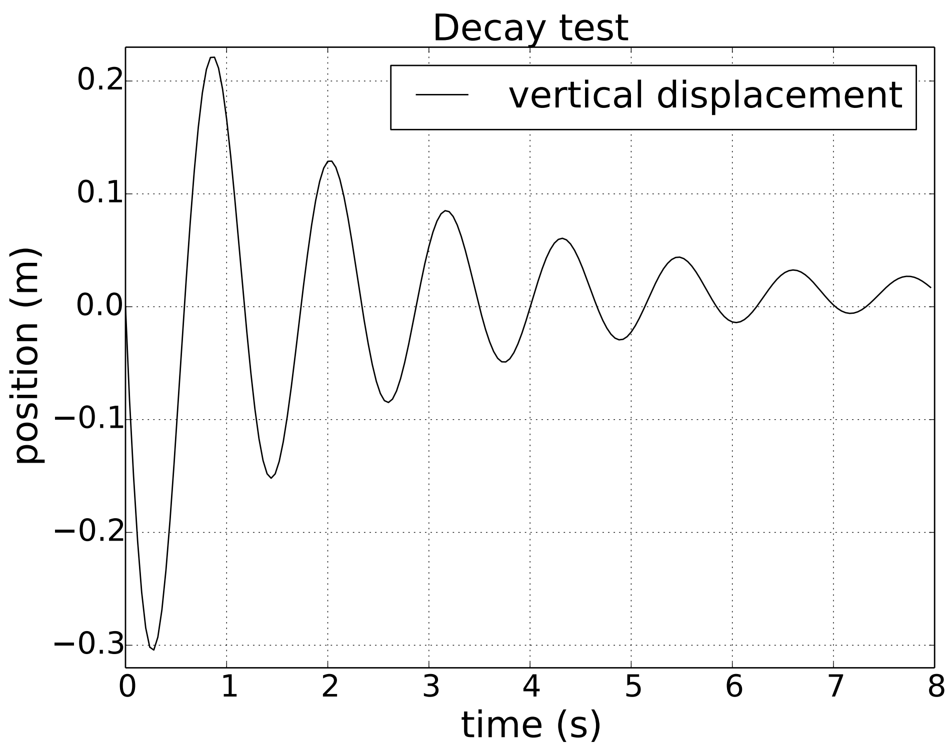

Figure 3: Vertical displacement of the floating cube.