MESH UPDATING SCHEME FOR FLUID STRUCTURE INTERACTION APPROACH

Carlos Eduardo da Silva, kadu@lamce.coppe.ufrj.br
Daniel Afonso Barcarolo, barcarolo@peno.coppe.ufrj.br
Nestor Oscar Guevara Jr, guevara@lamce.coppe.ufrj.br
José Luis Drummond Alves, jalves@lamce.coppe.ufrj.br

Federal University of Rio de Janeiro, Laboratory of Computational Methods in Engineering, LAMCE/COPPE/UFRJ

Abstract. *The interaction of a structure with a flowing in which it is submersed rise to a rich variety of physical phenomena with applications in many fields of engineering, for example, the stability and response of floating system, the response of bridges and tall buildings to winds, and the vibration of turbine and compressor blades. In the present article, we emphasize recent developments in the treatment of moving mesh under ALE approach in finite elements problems with attention especial in the description of dynamic of body submersed subjected to hydrodynamics forces. The fluid is modeled on a stabilized finite element formulation for treat incompressible Navier-Stokes equations. This study presents the numerical results on the behavior of structures submersed in viscous fluid.*

Keywords*: finite element, moving mesh, fluid structure interaction, arbitrary lagrangian eulerian (ALE)*

1. INTRODUCTION

The crescent increase of capacity of the computers and the recent developments of the numerical methods, particularly in the finite element and finite volume, have been responsible by an important advance of the hydrodynamics problems encountered in many fields of engineering.

In practice, this hydrodynamic problem presents many challenges due to: complex geometry, effects turbulent, extremely conditions of the operating and environment, all these aspects are responsible by difficulty in the numerical treatment of the fluid structure interaction (FSI) problems.

For numerical simulation of such flows, detailed resolution of complex free surface topologies is important for understanding the physics and mechanisms of air entrainment and buble formation associate with wave breaking, which is one of the major sources of hydrodynamic instability, (Wilson *et. al* 2006).

Basically the developments presented in this work have a set of the four subproblems, which are: (a) evaluate the fields of velocity and pressure, (b) mapping of the free surface, (c) compute the motion rigid body and finally (d) update finite element mesh of the analysis domain. All these subproblems are coupled and solved in each time step of the transient analysis.

The sub problem (a) is normally called of incompressible fluid flow governed by Navier-Stokes (NS) equations. The firstly studies for modeling this equations via finite element approach were the developments presented in the works of the Hughes and Brooks (1979) and Brooks and Hughes (1982). Since then several researchers have been actively involved with this theme. A survey important about this topic can be encountered in the work of the Tezduyar (1992).

The mapping of free surface, subproblem (b) is the main computational challenges in the scenario of the FSI problems. For treated this problem was utilized a technique of interface-capture, in this case we use the Volume of Fluid Method (VOF), firstly proposed by Hirt and Nichols (1981). In this method the (NS) equation are solved over a non-moving mesh together with time-dependent advection equation governing the evolution of the interface function.

After or simultaneous to solve of the subproblems (1) and (2) the integration of the motion equations of the body rigid is necessarily. O motion of a floating body or immersed in fluid is a direct consequence of the flow induced forces acting on it while at the same time these forces are a function of the body movement itself. In general, forces exerted on a moving body in a viscous fluid exist of pressure forces and viscous drag forces. The real experimental or numerical characterization these drag forces is yet a challenge. Aspects about skin friction due to roughness of the surface in contact with the fluid can induce considerably the boundary layer around the body. Strategies for coupled this motion can be found in Fekken (2004) and Panahi 2006.

Finally, is commented the subproblem (d) which is responsible for accommodation of the internal nodes of mesh, this step of the analysis is computed from of the motion of the body rigid. In coupled solutions strategies for FSI problems the essential attribute required is maintain the quality of the mesh for all the successive time steps In order to solve this problem we employed a smooth potential with diffusion artificial. This method was present Masud *et. al* (2007), the main goal this strategies is not need a remesh method for each time step analysis.

The outline of this work is the following: in Section 2 we presented a formalism for the mesh moving strategy. In Section 3 and 4, are presented, respectively, the procedures for the treatment of coupled FSI problem and some test cases chosen for show the details of performance of the model proposed. In Section 5 we draw some conclusions.

2. MESH MOVING MODEL

The method employed for the mesh moving is based on the potential suavization with artificial diffusion (τ) no uniform on a discretized domain by finite element, proposed Masud *et. al.* (2007). The basic idea in this method consists in solving a Laplace equation for each direction of the moving mesh, and was proposed in the works by Lohner (1996) and Masud (1997). The formal statement of the boundary value problem for finding the field of the displacement (**u**) for each node of the mesh, is given by:

$$\nabla \cdot ([1+\tau]\nabla) u = 0 \tag{1}$$

$$u = g \ldots on \ldots \Gamma_m \tag{2}$$

$$u = 0 \ldots on \ldots \Gamma_f \tag{3}$$

The Equations (1)-(3) represent the governing equation, the imposed movement by external actions of the fluid on $\mathbf{\Gamma}_m$ and the fixed boundary conditions on $\mathbf{\Gamma}_f$, respectively. The definition of the artificial diffusion (τ) which is non-dimensional and appears in the equation (1) is defined only by the geometric proprieties of all finite elements. For a given element (*e*), its artificial diffusion is given by:

$$\tau^e = \frac{1 - V_{\min} / V_{\max}}{V^e / V_{\max}} \tag{4}$$

where V^e, V^{max} and V^{min}, are the volumes, respectively, of the current element (*e*), the largest and smallest element in a given mesh. This geometric parameter was designed to move the smaller element in the boundary layers together with the interface with the least amount of distortion so as to attain well-conditioned meshes for subsequent time steps.

For a brief analysis of the sensibility for the parameter (τ), curves are showed in the Fig 1 indicating the variation of (τ) versus the level of refinement for three cases: (i) a mesh with relation 1:20, i.e. V_{min} =0.5, V_{max} =10; (ii) mesh with relation 1:10, i.e. V_{min}=1.0, V_{max}=10 and (iii) mesh 1:2, i.e. V_{min} =5, V_{max} =10. Note that, for a homogeneous mesh the diffusion is null, in this case the Eq (1) is a typical Laplace equation.

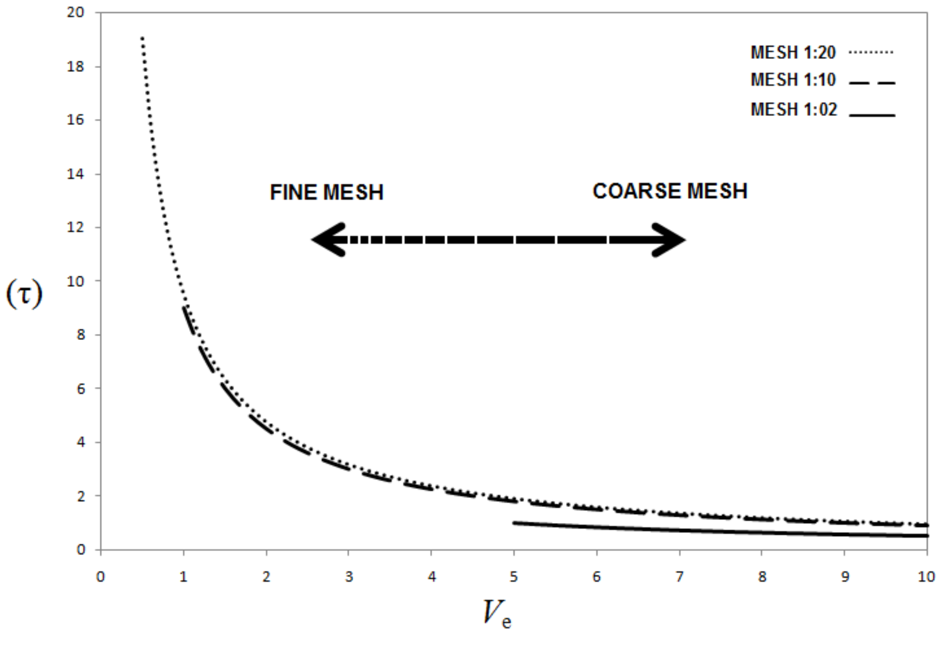


Figure 1. Curve of artificial diffusion for three typical mesh non-homogenous.

For a FSI problem, using the ALE approach, it is necessary the computation of the mesh velocity ($\boldsymbol{v}_{mesh}$) in each time step of the analysis, as follow:

$$v_{mesh}^{n+1} = \frac{u_{mesh}^{n+1} - u_{mesh}^{n}}{\Delta t} \quad (5)$$

3. FSI COUPLING

In this work the fluid flow will be considered by the motion of the two single-phase fluids (air-water), treated by incompressibility assumption and separated by an interface (free surface). The single-phase assumption requires that the effect of one fluid on the other is very small. This assumption is reasonably because the physical properties of water and of air are distinct by a ratio of the 1:1000.

All these assumptions should be valid in cases which the liquid-gas interface remains a free boundary. Although complex flow can be managed with the single-phase model, extremely cases of the small-scale as free surface turbulence, capillary waves, air entrainment and bubbles due to breaking waves were not resolved with satisfactory accuracy. Along of the interface the surface tension is negligible.

3.1. Incompressible fluid flow in the ALE approach

The governing equations for flow fluid are the incompressible Navier-Stokes equations written for two fluids in the ALE domain. These equations are:

$$\rho\left[\frac{\partial \mathbf{u}}{\partial \mathrm{t}} + \left(\mathbf{u} - \mathbf{u}_{\mathrm{mesh}}\right)\cdot\nabla\mathbf{u} - \mathbf{g}\right] - \nabla\cdot\boldsymbol{\sigma} = 0 \quad (6)$$

$$\nabla\cdot\mathbf{u} = 0 \quad (7)$$

where

$$\boldsymbol{\sigma} = -p\mathbf{I} + 2\mu\nabla^{s}v \quad (8)$$

Here $\mathbf{u}$, p, $\mathbf{g}$, ρ and μ are fluid velocity, pressure, gravitational force, density, and dynamic viscosity, respectively; and $\mathbf{u}_{mesh}$ is the mesh velocity given by Eq. (5).

To simulate the behavior of the free surface was designed an interface scalar function $\varphi(\mathbf{x},t)$ which assume only values between [0,1]. This function is the crucial ingredient of the volume-of-fluid (VOF) method, which was firstly proposed by Hirt and Nicholson 1981. This method will be briefly reviewed here, more details about the formulation of this method may be found in, Kleefsman (2005) and Elias and Coutinho (2007).

The scalar function φ is transported by time-dependent advection equation, as following:

$$\frac{\partial\varphi}{\partial \mathrm{t}} + \left(\mathbf{u} - \mathbf{u}_{\mathrm{mesh}}\right)\cdot\nabla\cdot\varphi = 0 \quad (9)$$

Initially, is assumed the value 1.0 in regions filled with water, null for air and for the free surface 0.5 is usually assumed. After evaluating $\varphi(\mathbf{x},t)$, the density and viscosity in the fluid flow solution are interpolated across the interface as follow:

$$\rho = \varphi(\rho_1) + (1-\varphi)(\rho_2)\ ,\ \upsilon = \varphi(\upsilon_1) + (1-\varphi)(\upsilon_2) \quad (10)$$

The finite formulation of Eq. (6), (7) and (9), will not be showed here, but a detailed description can be found in, Tezduyar and Liou 1990 and Franca and Frey (1992).

3.2. Rigid body dynamics

In this work the structure was considered as a rigid body. The motion consists in the superposition of the translation produced by resultant hydrodynamic forces and its weight and the rotation caused by moments around the body mass center. These hydrodynamic forces are evaluated following the solving of the fields ($\mathbf{u}$,p). The rigid body is moved attached to the mesh in each time step of the analysis.

The motion of a given rigid body is governed by the following equations:

$$m\mathbf{a} = \mathbf{F} + m\mathbf{g} \tag{11}$$

$$\mathrm{I}\dot{\omega} + \omega \times \mathrm{I} \cdot \omega = \mathbf{M} \tag{12}$$

where m and I, are the mass of the body and its inertia matrix; **a** and $\dot{\omega}$ are the its translation and angular accelerations of the body. Finally **F** and **M** are the external forces e torques, both caused by the fluid in contact with the body surface.

An explicit scheme of time integration is implemented for the Equations (11) and (12). For the translation equation of movement (6), a Crank-Nicholson operator is used for the velocity field in order to obtain the displacement field, as follows:

$$v_i^{n+1} = v_i^n + \frac{\Delta t}{2m} \cdot \left(3F_i^n - F_i^{n-1}\right) \tag{13}$$

$$u_i^{n+1} = u_i^n + \frac{\Delta t}{2}\left(v_i^{n+1} + v_i^n\right) \tag{14}$$

The same scheme of (13) and (14) is applied for the Euler Equation (12), but before two intermediary steps are presented. The first is an inversion of the matrix inertia

$$\dot{\omega} = \mathrm{I}^{-1}\left(\mathbf{M} - \omega \times \mathrm{I} \cdot \omega\right) \tag{15}$$

and the second step defines the matrix $\overline{\mathbf{M}}$

$$\overline{\mathbf{M}}^n = \left(\mathbf{M}^n - \omega^n \times \mathrm{I} \cdot \omega^n\right) \tag{16}$$

Finally, the rotation of the body is presented

$$\omega^{n+1} = \omega^n + \frac{\Delta t}{2}\mathrm{I}^{-1}\left(3\overline{\mathbf{M}}^n - \overline{\mathbf{M}}^{n-1}\right) \tag{17}$$

$$\theta^{n+1} = \theta^n + \frac{\Delta t}{2}\omega^{n+1} \tag{18}$$

3.3. Equations of the coupling

The internal forces of the elements surrounding the immersed body can be calculated by:

$$f_{\mathrm{int}}^e = \int_{\Omega_{(sub)}} \mathrm{B}^t \sigma d\Omega = \mathrm{B}^t \sigma V^e \tag{19}$$

where the tensor (σ) was defined by (8) and $\mathbf{B}^t$ and V^e being the discrete differential operator and the element volume. The integral (20) is evaluated on Ω_{sub}, which is the region defined by the finite elements with nodes attached to the body surface. Then the resulting force on the immersed body can be found by the sum of each element internal force contribution (equivalent nodal forces).

The resultant of the hydrodynamic force (11) on the body is given by:

$$\mathbf{F} = \sum_{n=1}^{nrb} \mathbf{f}_n \tag{20}$$

where $\mathbf{f}_n$ is the assemble of forces (20) for all nodes on surface of the rigid body designed by (nrb). After evaluating the force for each node of (nrb) the resulting moment is calculated by the following equation:

$$\mathrm{M} = \sum_{n=1}^{nrb} \mathrm{r}_n \times \mathrm{f}_n \tag{21}$$

with $\mathbf{r}_n$ being the position vector of each node (nrb) to the mass center.

4. TEST CASES

In order to realize some test the efficiency of the formulation presented in this work, are summarized four test cases in the following section. Firstly we present an analysis of the mesh performance quality for 2d problems, the following two test cases illustrate the severe distortions in the mesh, and the last test case show a reliability in real case of the hydrodynamic stability.

4.1. Analysis of the performance of mesh quality

Two cases were considered to verify the mesh quality performance: models with square and hemispherical bodies. The square body mesh has 1804 triangular elements and 962 nodes against 2571 triangles and 1138 nodes for the hemispherical body mesh. On the figures below the initial and the deformed mesh with the quality factor plotted can be seen for both cases. The quality factor, Branets (2005), for each element is given by:

$$q = \frac{4\sqrt{3}A}{l_1^2 + l_2^2 + l_3^2} \tag{22}$$

where l_1,l_2 e $l_3$represents each edge of an element and A its area. This factor varies on the interval [0,1] being 0 for an element with null area and 1 for an equilateral element.

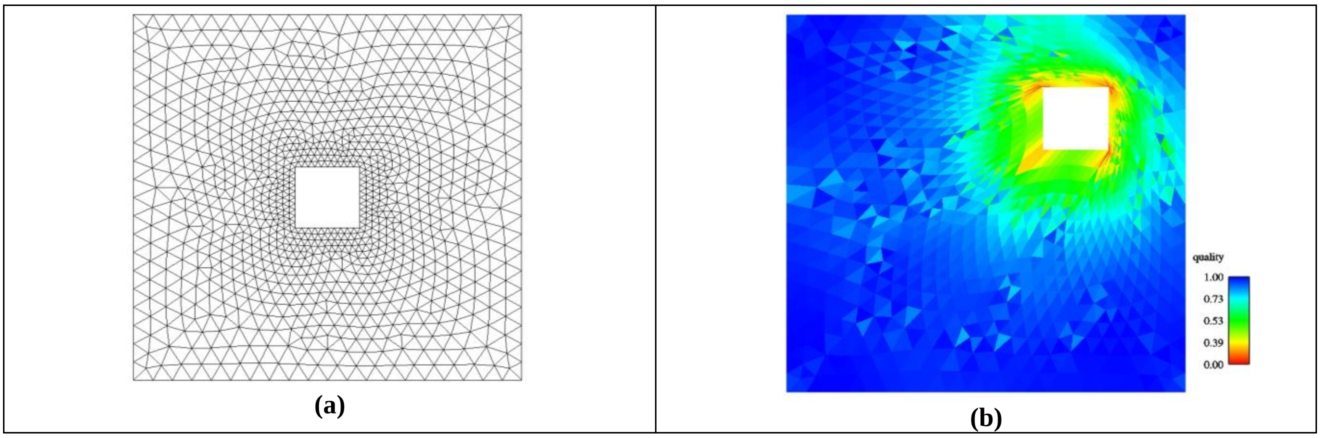


Figure 2. (a) Initial square body mesh and (b) deformed mesh with element quality factor plotted.

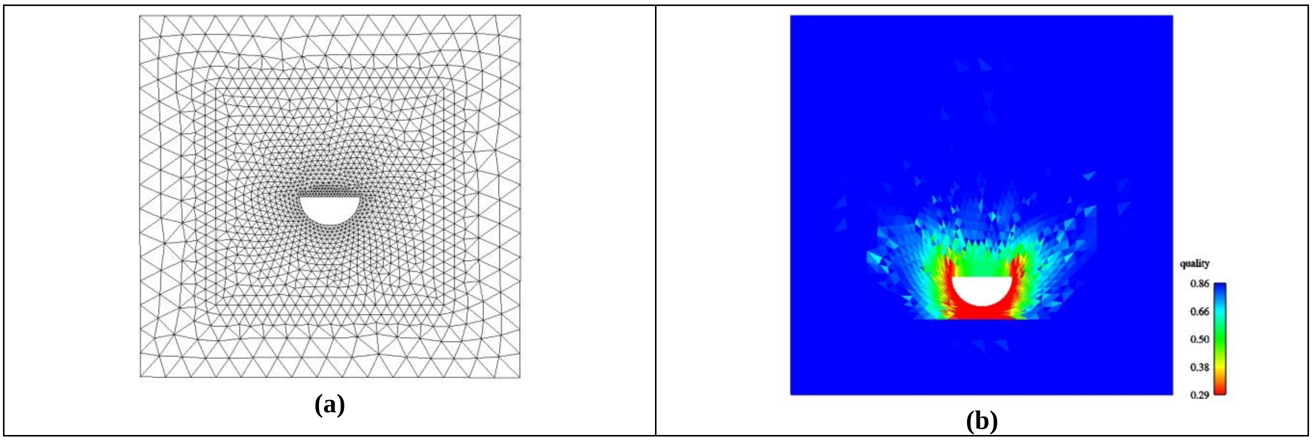


Figure 3. (a) Initial hemispherical body mesh and (b) deformed mesh with element quality factor plotted.

4.2 Suspension of a prism immersed in fluid

On this tested case a submersed prism with a density of 600 kg/m³ emerges towards the free surface. The dimension of the model can be seen on Figure 4, where L = 0,06m, together with its mesh that has 84546 tetrahedrons and 15723 nodes. Here, the objective is to verify the dynamics of a cubic body in suspension and the mesh behavior.

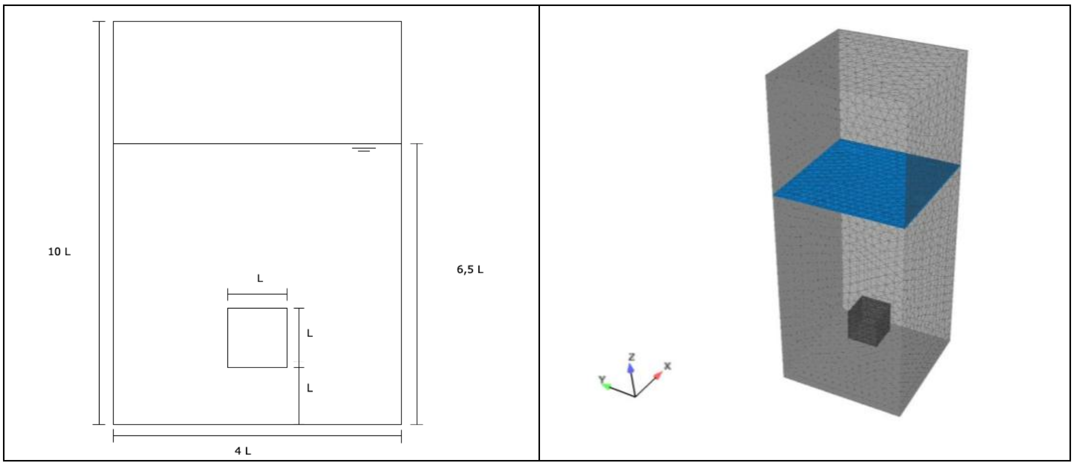


Figure 4. Prism model dimensions and mesh.

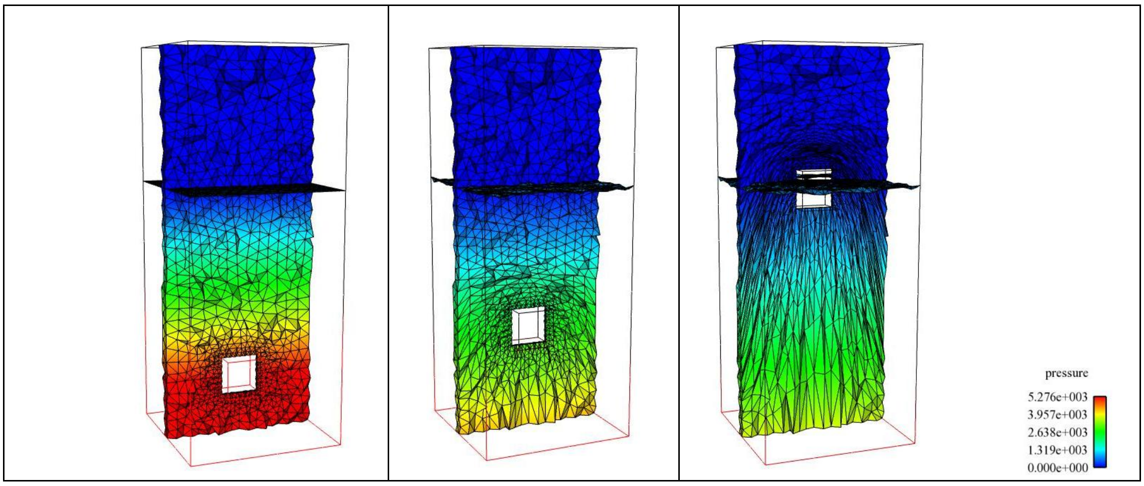


Figure 5. Mesh Behavior.

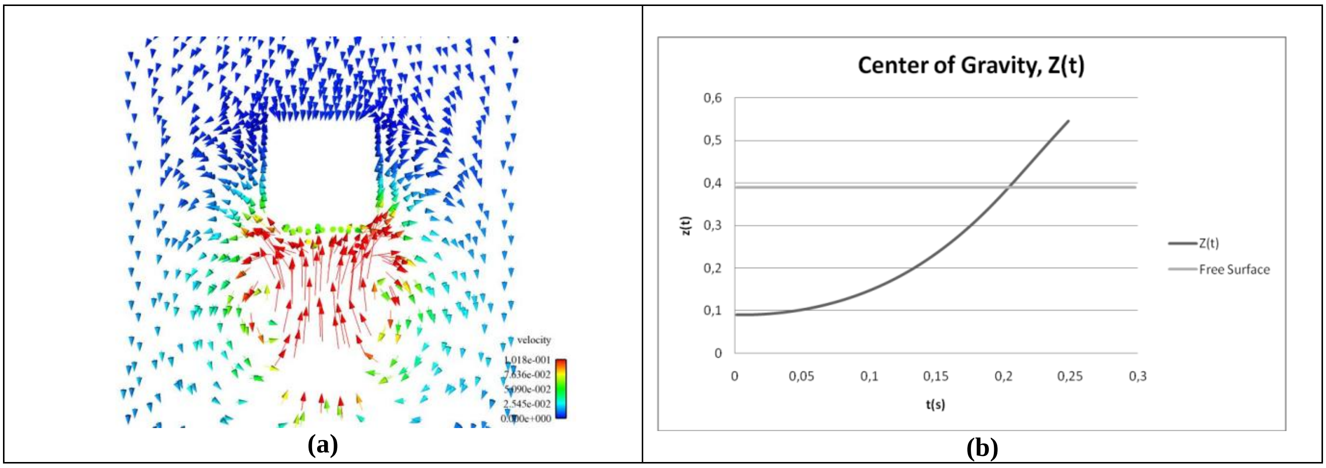


Figure 6. Velocity vectors of section of the model(t=0.1s) and (b) the Z center of gravity time history.

4.3 Suspension of a sphere immersed in fluid

On this tested case a submersed sphere with a density of 600 kg/m³ emerges towards the free surface. The dimension of the model can be seen on Figure 7, where L = 0,06m, together with its mesh that has 113272 tetrahedrons and 20777 nodes. Here, the objective is to verify the dynamics of a spherical body in suspension and the mesh behavior.

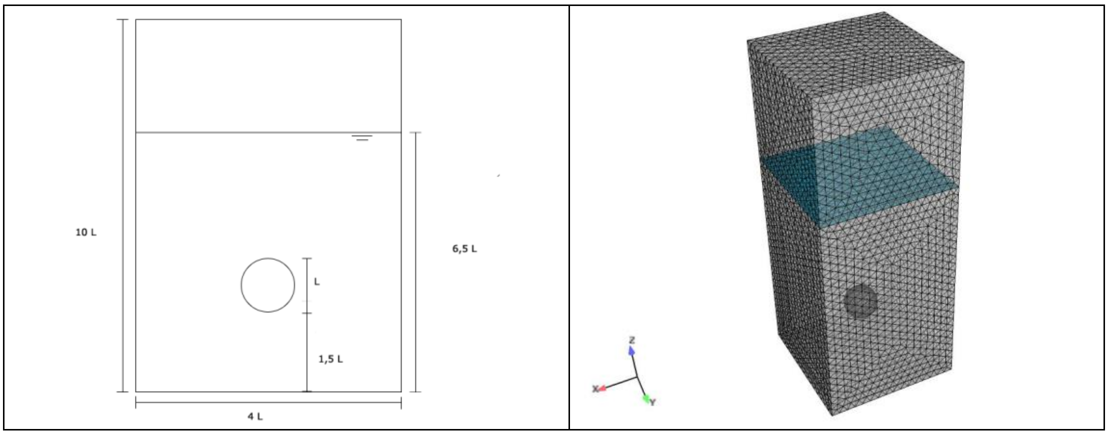


Figure 7. Sphere model dimensions and mesh.

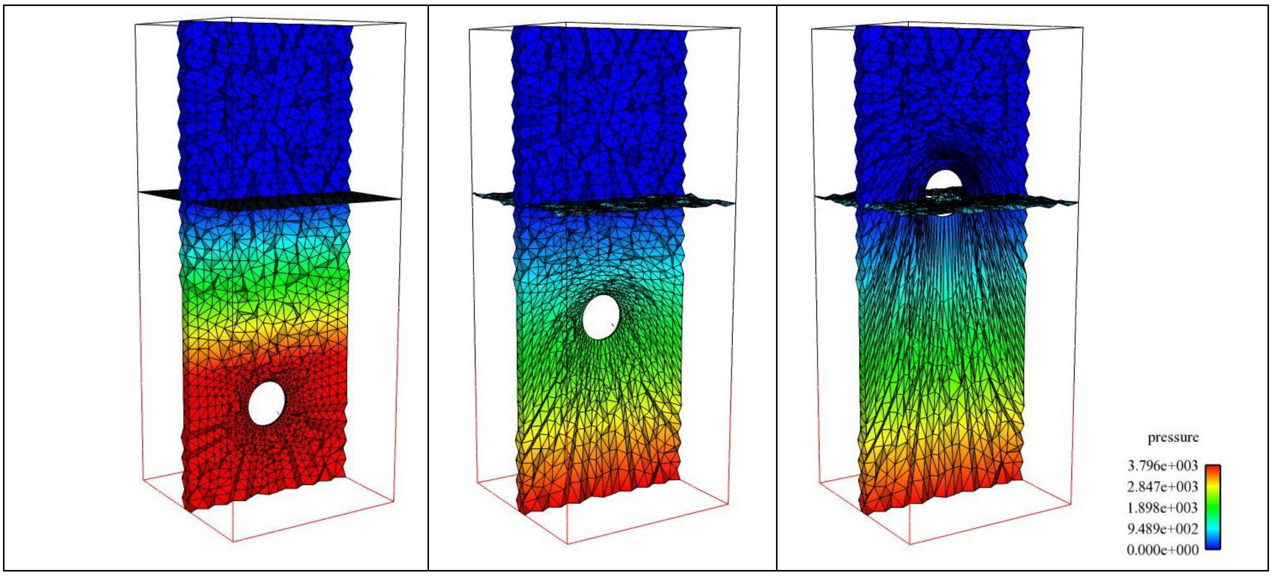


Figure 8. Mesh Behavior.

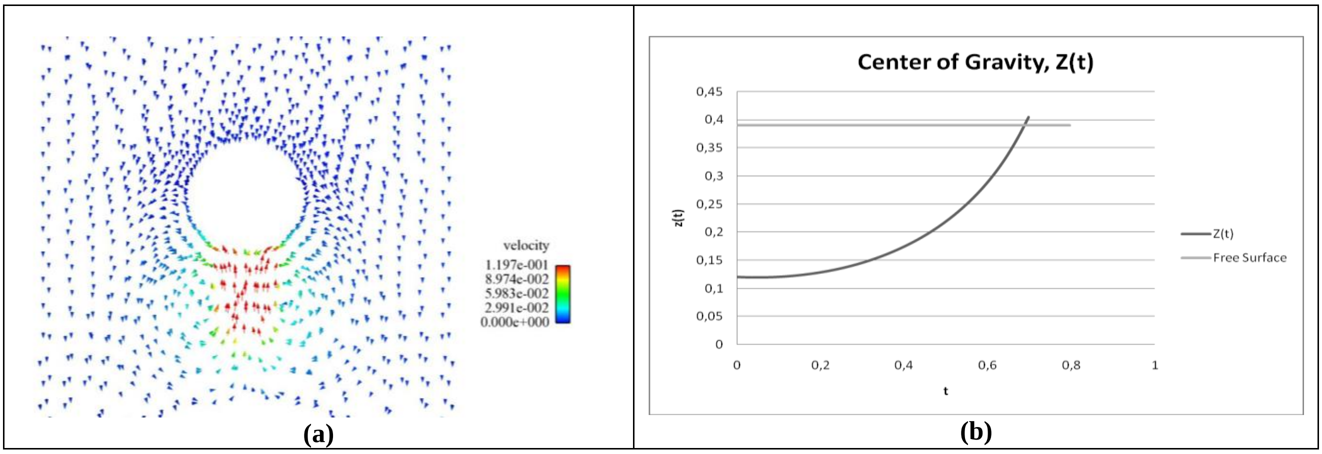
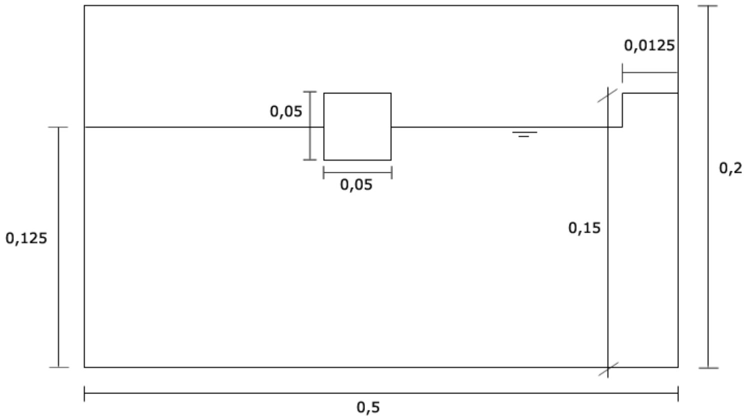


Figure 9. Velocity vectors of section of the model(t=0.3s) and (b) the Z center of gravity time history.

4.4 Analysis of the stability of a cylinder

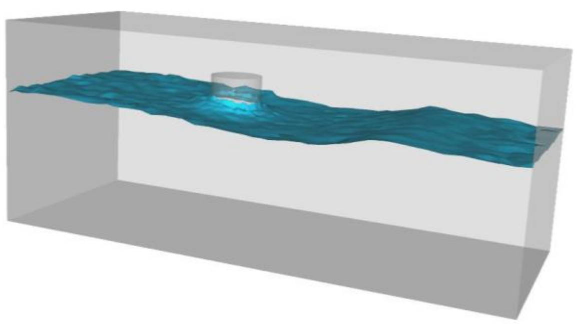
This model has as objective to study the stability of a floating cylinder by analyzing the buoyancy time history and the forces actuating on its surface. A small elevation on the free-surface was modeled in order to generate a wave so that the body is removed from its state of static equilibrium.



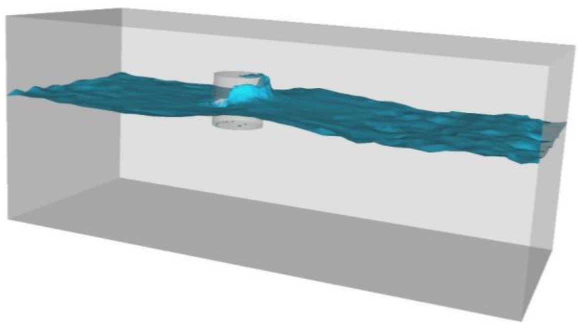
Model Data:

- Elements = 151085;
- Nodes = 27306;
- Density = 500kg/m³
- Radius = 0,025 m

Figure 10. Stability model dimensions and data.

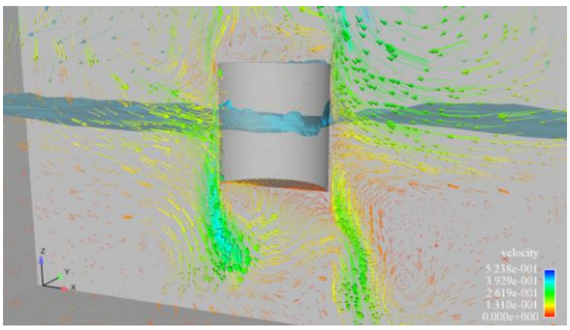


(a)

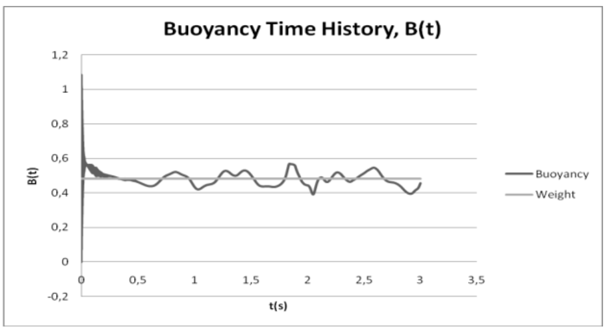


(b)

Figure 11. The free surface behavior: (a) t = 0.6s and (b) t = 0.86s .



(a)



(b)

Figure 12. (a) Velocity vectors of section of the model and (b) the buoyancy time history.

5. CONCLUSIONS

This work showed an unsteady single-phase method presented under an ALE approach for description of the Navier-Equations finite element equations. For the motion of the interface between the two fluids is used the VOF method.

There are still challenges on the mesh moving area, on the treatment of the free surface and finding robust methods remains a crucial objective to guarantee the physical reliability of the numerical codes. In this context some themes can be cited: consideration of the two-phase flow (air-water) and the developing of schemes to capture the superficial tension.

The formulation presented has shown to be a robust numerical tool to analyze large amplitude motions on problems involving fluid-structure interaction. The main goal of the numerical apparatus collection presented is the qualitative fidelity of the configurations, mainly the test case of the cylinder stability.

Finally, it is important to salient that the mesh moving method used on this article is not recommendable for problems with very high motions of the moving body, in some cases the aspect ratio of the elements can affect considerably the accuracy of the fluid flow solver.

6. ACKNOWLEDGEMENTS

This work has been supported by ANP (National Agency of Petroleum- Brazil), PETROBRAS Research Center and MCT/CNPq (The Brazilian Council for Scientific Research). The authors are grateful to Prof. Alvaro L.G. A. Coutinho and to Researcher Renato N. Elias, by the innumerous discussion about the theme of this work, and also by the offered support from EdgeCFD computational code developed at NACAD/COPPE/UFRJ, on which the formulations presented were implemented.

7. REFERENCES

Branets, L, Carey, G. F, A local cell quality metric and variational grid smoothing algorithm, Engineering and Computers, 21, pp 19-28, 2005

Brooks, A.N. and Hughes, T.J.R., “Streamline upwind/Petrov-Galerkin formulations for convection dominated flows with particular emphasis on the incompressible Navier-Stokes equations”, Computer Methods in Applied Mechanics and Engineering, 32 (1982) 199–259.

Chiandussi G., Bugeda G., Oñate E., A simple method for automatic update of finite element meshes, Commun. Numer. Methods Eng. 16 (2000) 1-19.

Donea, J. Arbitrary Lagrangian Eulerian methods. In Computational Methods for Transient Analysis, volume 1 of Computational Methods in Mechanics. North-Holland, Elsevier, 1983.

Donea, J. e Huerta, A., Finite Element Methods for Flow Problems, Wiley, 2003.

Elias, R.N and Coutinho, A.L.G.A, Stabilized edge-based finite element simulation of free-surface flows, Int. J. Numer. Meth. Fluids 2007; 54:965–993.

Fekken, G., Numerical simulation on free surface flow with moving rigid bodies, 2004. PhD Thesis, University of Groningen, Netherlands.

Formaggia, L., Miglio, E., Mola, A. and Parolini, N., Fluid–structure interaction problems in free surface flows: Application to boat dynamics, Int. J. Numer. Meth.Fluids 2008; 56:965–978

Hughes, T. J. R. & Brooks, A., 1979. A multi-dimensional upwind scheme with no cross wind diffusion. ASME, vol. 34.

Hirt, C. W. Hirt and Nichols, B. D., Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. J. Comp. Phys., 39:201–225, 1981.

Kleefsman, K.M.T., Fekken G., Veldman A.E.P., Iwanowski B. and Buchner B., A Volume-of-Fluid based simulation method for wave impact problems, Journal of Computational Physics 206 (2005) 363–393.

Kanchi, H; Masud, A., A 3D adaptive mesh moving scheme, International Journal for Numerical Methods in Fluids, 54(6-8): 923-944, 2007.

Masud, A., Bhanabhagvanwala, M., & Khurram, R. A., 2007. An adaptive mesh rezoning scheme for moving boundary flows and fluid-structure interaction. Computers and Fluids, vol. 36, pp. 77–91.

Panahi, R, Jahanbakhsh, E., Seif, M. S.. 2006. Development of a vof-fractinal step solver for oating body motion simulation. Applied ocean research, vol 28, pp 171-181.

Tezduyar, T.E., "Stabilized Finite Element Formulations for Incompressible Flow Computations", *Advances in Applied Mechanics*, **28** (1992) 1-44.

Wilson R., Carrica P. M. and Stern, F., 2006. Simulation of ship breaking blow waves and induced vortices and scars, International Journal for Numerical Methods in Fluids, in press.