ENCIT 2018

ABCM Associação Brasileira de Engenharia e Ciências Mecânicas

17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-0801
NUMERICAL SIMULATION OF THE BUBBLE PEN PHENOMENON

Marcelo de Oliveira e Silva
Fernanda Kelly de Jesus Gomes
Federal University of Pará, 01 Augusto Correa Street, Belém, PA, 66075-110, Brazil
mos@ufpa.br, fernanda.kelly@hotmail.com

Luis Ricardo Braga Pereira
Luany Karolynne David Braga
Fernanda Regina Freitas da Silva
João Wellington Amaral Perdigão de Souza
Kaynan Gabriel do Espirito Santo Leal
Federal University of Pará, 01 Augusto Correa Street, Belém, PA, 66075-110, Brazil
lrick_pa@yahoo.com.br, luany_karolynne@hotmail.com, fr_freitas@live.com, joaowapsouza@yahoo.com.br,
kaynan.gabrielleal@gmail.com

Abstract. *This paper presents preliminary numerical simulations of the bubble pen phenomenon. The numerical treatment is being developed with OpenFOAM free code software, based on the finite volume method for discretizing physical and temporal space. The Eulerian-Eulerian approach for incompressible multiphase flows already implemented in the software through the twoPhaseEulerFoam solver developed by Rusche (2003) is used to simulate dispersed flows, whose terms of change of momentum between the phases (drag, support and virtual mass) are included in their modeling. The phenomenon is reproduced numerically, adopting the same experimental characteristics of research carried out by the PPGEM. In order to validate the code, numerical profiles of void fraction and vertical velocity of the gas phase at three heights are compared with experimental data obtained in PPGEM. The experimental behavior of the vacuum fraction and gas velocity profiles were to some extent captured by the numerical treatment of the twoPhaseEulerFoam solver using some calibrations made in the model. However, a better understanding of the problem physics and the OpenFOAM software are necessary to obtain more satisfactory results.*

Keywords: *Bubble Pen, Multiphase Flow, OpenFOAM.*

1. INTRODUCTION

If a continuous flow of gas is released inside a liquid medium, the gas takes the bubble configuration (due to Kelvin-Helmholtz instabilities) and moves upwards due to the buoyancy caused by the density difference between the phases. The biphasic and cone-like region based on the free surface, vertex at the entrance orifice and height H. Due to the surface drag of the bubbles, the gas carries with it an appreciable amount of liquid, a phenomenon known as "entrainment". For reasons of continuity, the movement of the dispersed phase raises in the liquid an equally upward recirculation region (Barbosa, 1997). The phenomenon described and known as bubble plume can be observed in Fig.1.

The flow in bubble plume is considered to be axissimetric for punctiform gas sources and has been categorically divided into three distinct regions: the Zone of Flow Establishment (ZOFE), the Zone of Established Flow ZOEF) and the Zone of Surface Flow (ZOSF). These parts are determined by the balance of the forks that dominate each region of the pen. The following are theories that consider the flow in bubble feathers under a permanent regime, isothermal, totally turbulent, assuming the non-stratified medium and assuming the absence of cross currents (Barbosa, 1997).

In the Developing Stream Region, near the gas inlet orifice, the inertia and thrust forces have the same order of magnitude and the bubble plume still has jet characteristics (most of the researches performed on bubble plumes disregard analyzes of phenomena in this region). The driving force of the flow in the Developed Flow Region and the thrust, which dominates the inertial forces from a given region (ZOFE-ZOEF boundary) defined as the location where the bubbles reach their terminal velocity. In ZOSF, the forces due to surface tensions interfere in the flow and cause the deflection of the boom and the formation of waves in the surface (Barbosa, 1997).

The scientific community has been researching the characteristics of bubble plume flow dynamics and several studies that approach the theme can be found in the literature. Equally vast are the applications of the phenomenon in the industry to analyze processes of aeration, distillation, chemical reactions, or for the purpose of controlling possible leaks of gas

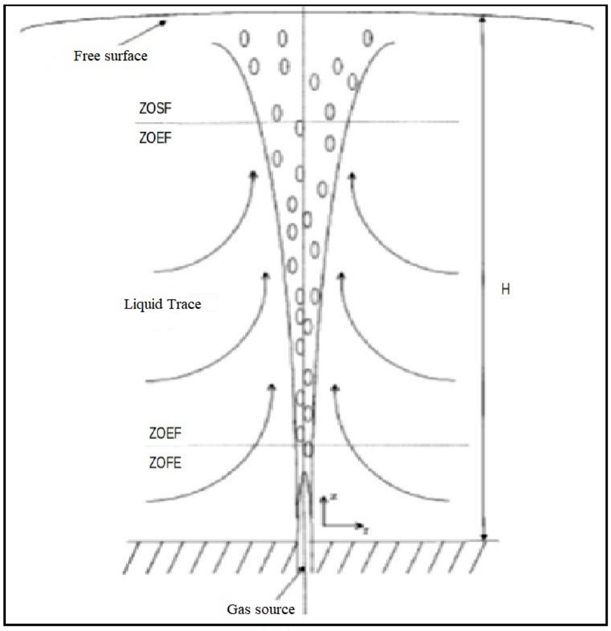

Figure 1. Illustration of a bubble pen.

or oil in perforations or operations of few submarines. In this last example, the formed bubble pen can generate waves when it reaches the surface and damage oil platforms and prevent the approach of rescue ships to control the leak. In addition, around the plume a marine current can be formed capable of homogenizing the temperature and breaking the marine ecological balance by mixing the surface waters with those of the bottom (Conti, 1983; Barbosa, 1997; Rusche, 2003; Rezende *et al.*, 2014; Paula *et al.*, 2015).

Although the construction of equipment capable of reproducing the phenomenon is simple in design and easy maintenance, the complexity of the flow patterns and the periodic fluctuation of the bubble pen, for example, still make the multiphasic description of the phenomena very difficult (Silva, 2008; Dionisio *et al.*, 2007). As an alternative, the numerical tool has become essential to investigate many critical aspects for which physical experimentation is not possible, in order to allow numerical treatment to be the most practical procedure and, in many cases, the only available for obtaining of data (Maliska, 2004; Fortuna, 2000).

2. METHODOLOGY

The study of any physical phenomenon should be based on previous theoretical analyzes. Therefore, it is essential to know the numerical solution steps of a CFD (Computational Fluid Dynamics) project. The numerical solution must begin with the physical modeling of the problem, where the physical quantities relevant to the system and the way in which the magnitudes affect the system itself are established. Physical principles guide the elaboration of the models from laws of conservation of mass, amount of movement and energy. As a result, the models are expressed by equations in the temporal and spatial domain relating the relevant quantities to each other (Maliska, 2004; Fortuna, 2000).

The next step is to properly treat the model computationally, carefully establishing the equations and the continuous region of validity of the model. The region continues and then divided into a number of points. At these points the solutions of the equations are obtained. The set of discrete points is known as mesh. The construction of the mesh and of vital importance to obtain a numerical solution representative of the phenomenon (Maliska, 2004; Fortuna, 2000).

Then the terms of the equations are written as a function of the values of the unknowns in adjacent discrete points and linear algebraic equations are obtained. Initial and boundary conditions, physical properties of the fluid and specific flow parameters are introduced in this step. The algebraic equations are solved and the solution of the problem can be analyzed. Thus, visualization techniques are applied and play a fundamental role in the comparison between numerical and experimental results, since the model can be adjusted until it adequately represents the physics of the problem (Maliska, 2004; Fortuna, 2000). In Figure 2 the numerical solution steps of any physical phenomenon are shown.

The numerical simulation of the bubble pen was performed in a cluster of 32 processing cores, which belongs to the Faculty of Mechanical Engineering of UFPA, where version 2.3.8 of OpenFOAM free code software is installed. The numerical study of the phenomena adopts the experimental characteristics of research carried out by PPGEM, such as: geometric information of the water tank and the air injector, air leakage in the liquid medium and height of the amount of liquid inside the tank.

The numerical methodology of OpenFOAM is based on the finite volume method for discretization of physical and

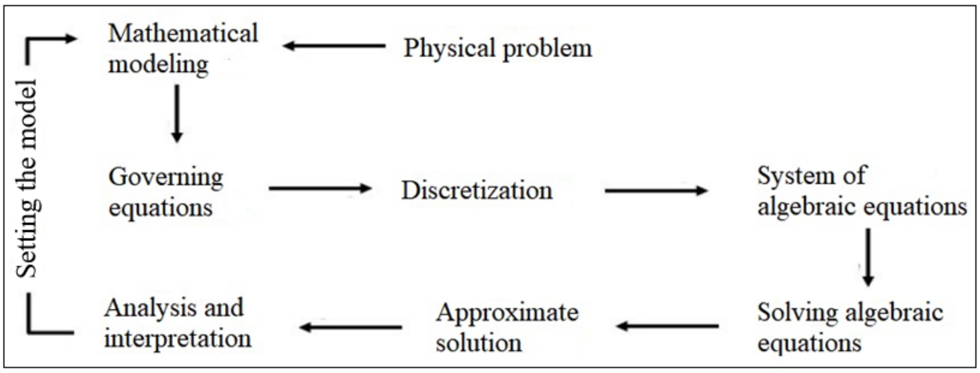

Figure 2. Synthesis of the steps of a numerical solution of any physical phenomenon.

temporal space (Rusche, 2003). Essentially, this technique consists of integrating the partial differential equations of continuous variables over a control volume and over a period of time to obtain a corresponding system of equations. The finite volume method is widely used in the CFD literature (Maliska, 2004; Fortuna, 2000).

3. RESULTS

The experimental characteristics of this study were obtained in a tank with dimensions of 0.5m long, 0.5m wide and 0.6m high. The level of non-tank water comprised 0.5m in height. A continuous stream of air in the liquid medium was inserted through a 1mm diameter movable injector of diameter by 30mm in height located at the bottom of the tank. Gas volumes corresponded to values of 1.8 and 3 l/minute. Experimental data of vacuum fraction and vertical velocity of the gas phase were measured at heights of 13cm, 18cm and 32.7cm above the bottom of the tank. A representation of the tank geometry, the water level and the heights of measurements are shown in Fig. 3.

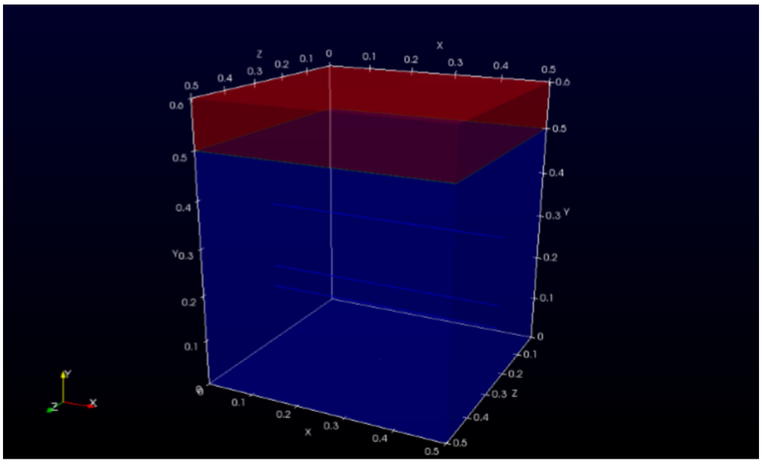

Figure 3. Representation of the water tank. The blue color determines the height of the water level. The blue lines inform the measurement locations of the vacuum fraction parameters and the vertical velocity of the gas phase. The injector is not represented in the figure.

The computational domain was discretized for the refinements presented in Tab. 1. The meshes are of the structured type, with refinement toward the injector inlet, as can be seen in Fig. 4 and Fig. 5. The results presented in this work were obtained with the mesh of 75x75x75 elements.

Table 1. Divisions of the computational domain.

Number of divisions	Cells	Points
25 x 25 x 30	15625	17576
49 x 49 x 50	120050	127500
75 x 75 x 75	421875	438976

The knowledge of the initial and boundary conditions are trivial to the reproduction of any physical phenomenon,

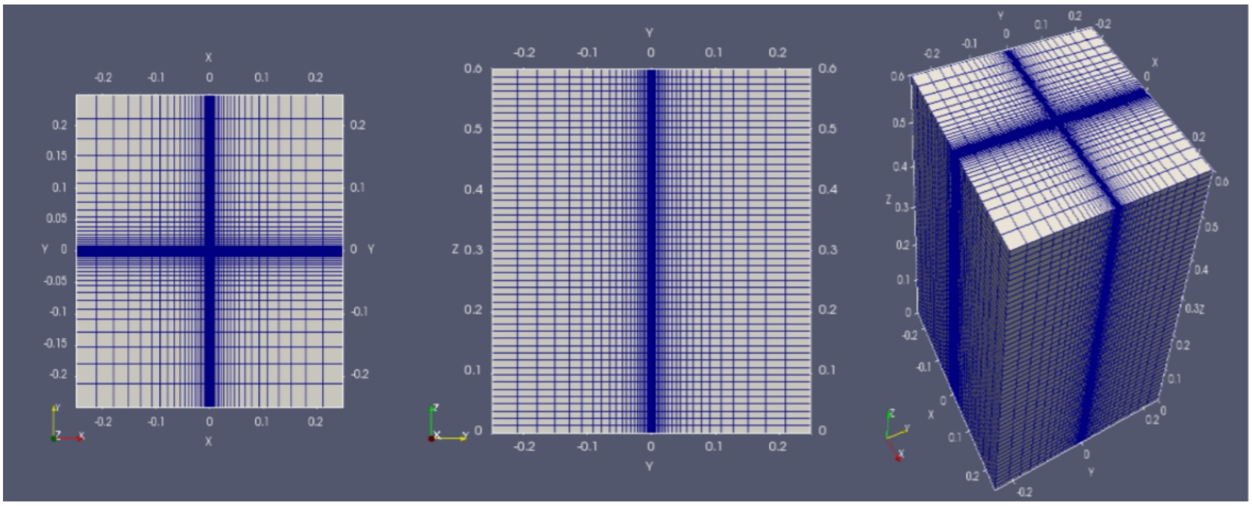

Figure 4. Top, side and panoramic views, respectively, mesh 49 x 49 x 50 divisions.

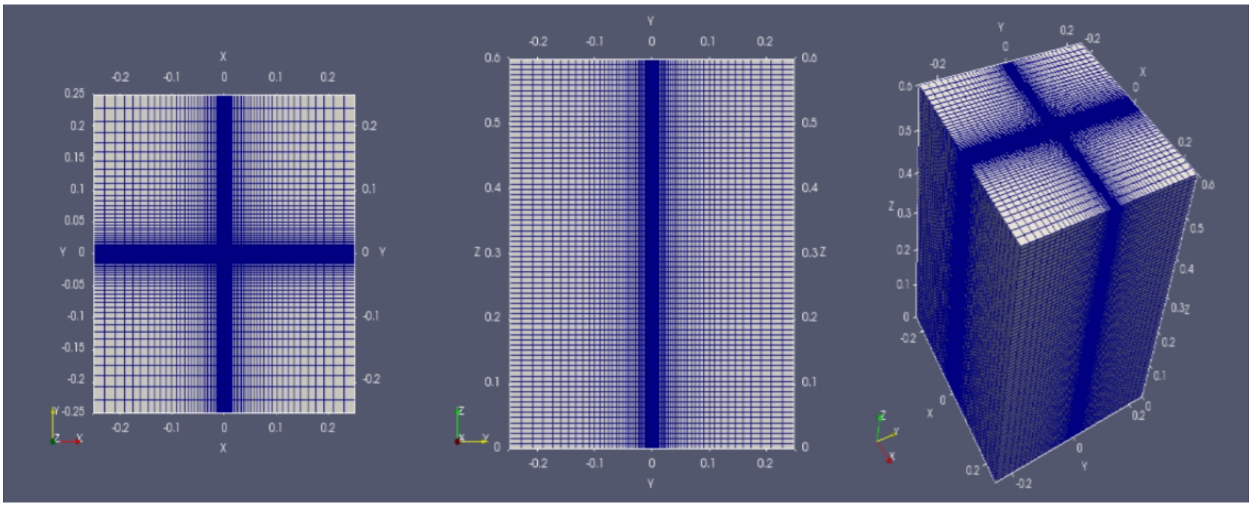

Figure 5. Top, side and panoramic views, respectively, mesh 75 x 75 x 75 divisions.

because characteristics inappropriate to the problem can compromise the development of the simulation or result in numerical simulations inconsistent with the physics of the problem (Fortuna, 2000).

The standard initial conditions of the solver were adopted for most variables, with the exception of the initial conditions of liquid and gas temperature defined as uniform internalField at 20°C and the turbulent variables defined with the initial value of $10^{-8}m^2s^{-3}$. The temperature values refer to the local temperature of the experimental measurements. According to Martin (2013), initial turbulent effects are neglected when inflow values for turbulent variables are determined.

The boundary conditions for domain boundary variables are described in OpenFOAM by the boundaryField statement. A procedure similar to the previous one was adopted, that is, the contour conditions already implemented in the solver were used for the great majority of the variables, except for gas leakage, void fraction and turbulent variables that were adjusted at the entrance of the domain. The gas leak at the entrance of the domain corresponds to the experimental values. 100% of air should be injected into the domain entry by the determination inlet 1 as done by Martin (2013), but only adjustments below 50% have obtained numerical solutions concordant with the experimental data.

The imposition of temperature for air and water in $20°C$ implied adjusting the kinematic viscosity values of water for $1.01e^{-6}m^2s^{-1}$ and air for $1.5e^{-5}m^2s^{-1}$ and also the density of water for $998kg/m^{-3}$. The remaining parameters remained unchanged.

The solution time interval of the equations for the domain volume elements was determined from the Courant number, (Co), an important dimensionless parameter that measures the stability of the numerical solution. A physical phenomenon can be well represented from the numerical point of view for values of $Co \leq 1$ (Fortuna, 2000). An $Co = 1$ was used, which resulted in the time interval of $2,8e^{-4}$ seconds. The data were recorded every $0,1$ second. A total time of 59 seconds of simulation was obtained.

The *TwoPhaseEulerFoam* adopts the following steps in the problem solving process: solving the continuity equation for each phase; updating coefficients for interfacial forces; construction of the numerical matrices for the momentum equation; obtaining the velocity fields, disregarding the pressure fields; correction of velocity fields for the new pressure fields; resolution of the equations for turbulence Rusche (2003).

Numerical and experimental profiles of vacuum fraction and vertical velocity of the gas phase obtained in the central region of the domain are compared with the experimental data in the three heights of measurements.

In Figure 6 the numerical vacuum fraction profiles are shown in comparison with the experimental ones. Qualitatively, we see that the curves of the experimental graphs show similar behaviors, even at different heights of medicos, where we see peaks in the central region above the injector decreasing of intensity with height. Similar behaviors are reproduced by the curves of the numerical graphs, where they exhibit the central vacuum fraction peaks decreasing in intensity as the distance from the base of the domain increases.

We see in Figure 6 that the numerical peak has values below the experimental one. The reverse happens in Fig. 6c. On the other hand, in figure 6b the numerical and experimental peaks show good agreement. However, we see that the vast majority of experimental data are overestimated by numerical profiles as they move away from the central medical region.

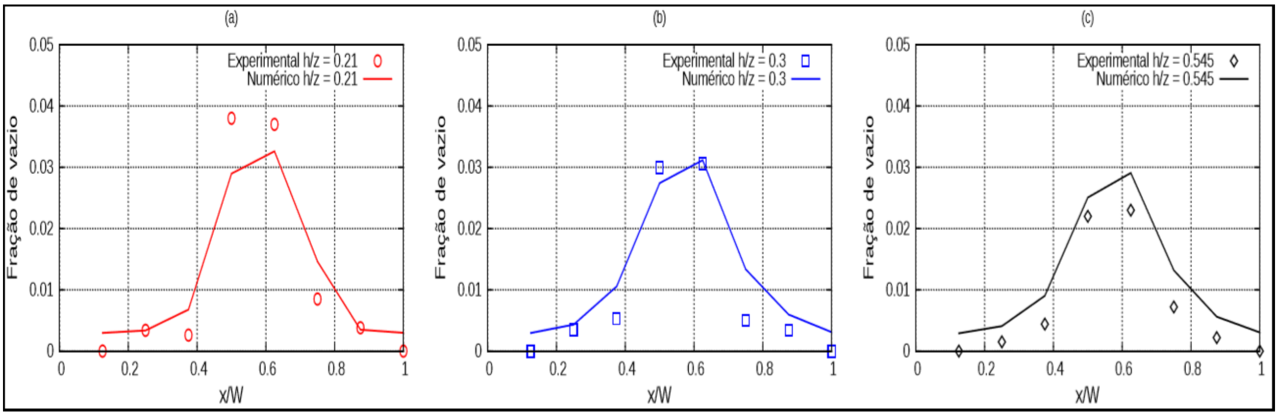

Figure 6. Experimental and numerical profiles of void fraction.

In Figure 7 the vertical velocity numerical profiles of the gas phase are shown in comparison with the experimental ones. Qualitatively, we observed that the experimental profiles show a decrease in the highest values of gas velocity with height. We also verified that this region of greater gas velocity is displaced a little to the right of the central region. The characteristic of the diminution of the profiles with the height are well represented by the numerical profiles, but these develop more protuberantes and centralized peaks.

In Figure 7a, we noticed the greatest disparity between the profiles, where most of the medications are underestimated by numerical treatment. In the figure 7b, we verified that the experimental profile becomes a little flatter, which can be identified in the numerical profile at the same height, but which is still underestimated by the numerical treatment. We have verified that the experimental data have an approximately parabolic shape in Fig. 7c, this characteristic is little presented by the numerical profile, but at this time of measurements the best concordances between experimental and numerical data take place. In the regions farthest from the central region overestimations of the numerical profiles prevail over the experimental data.

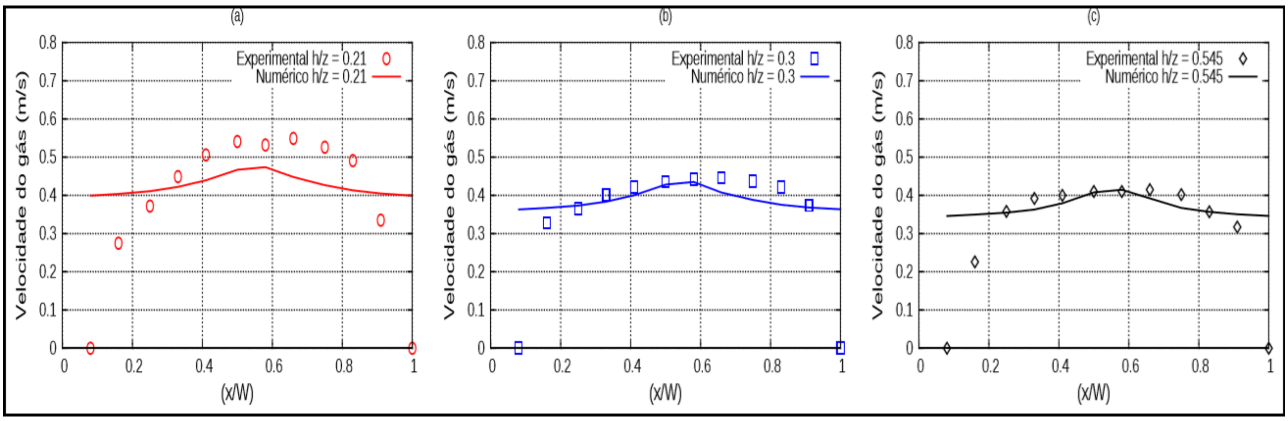

Figure 7. Experimental and numerical speed profiles.

4. CONCLUSION

In this work of numerical simulation of the bubble pen phenomenon, the preliminary results of the numerical profiles in comparison with the experimental ones showed that there is much to be ascertained about the physics of the problem and about the software OpenFOAM to better represent the average characteristics of this complex flow multiphasic. However, the experimental behavior of the void fraction and vertical velocity profiles of the gas phase were somehow captured by the numerical treatment of the twoPhaseEulerFoam solver by some calibrations made in the model. As a complement of the work, it is possible to numerically simulate the bubble pen near the borders of the domain to observe the coanda effect.

5. REFERENCES

Barbosa, J.R., 1997. *The method of the electro-resistive sensors used in a bubble pen*. Ph.D. thesis, Master's Dissertation, COPPE/UFRJ, Rio de Janeiro, 112p.

Conti, T.N., 1983. *Numerical simulation of adiabatic, two - dimensional, two - dimensional flow in a transient regime, applying the two - fluid model*. Ph.D. thesis, USP, São Paulo.

Dionisio, R.P. *et al.*, 2007. "Three-dimensional simulation of a column of bubbles - different geometric guides and modeling".

Fortuna, A.O., 2000. *Computational techniques for fluid dynamics: basic concepts and applications*. Edusp.

Maliska, C.R., 2004. "Computational fluid mechanics and heat transfer". *LTC, Rio de Janeiro, Brazil*.

Martin, S., 2013. *CFD study of gas-liquid flow from a subsea gas release*. Master's thesis, University of Stavanger, Norway.

Paula, D.M.L., Valls, E.M.L., Siqueira, A.M.O. and Batet, L., 2015. "Validation of the openfoam numerical code for gas-liquid biphasic flow models". *Blucher Chemical Engineering Proceedings*, Vol. 1, No. 2, pp. 6369–6376.

Rezende, R.V.P. *et al.*, 2014. "Simulation of the dynamics of the bubble column".

Rusche, H., 2003. *Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions*. Ph.D. thesis, Imperial College London (University of London).

Silva, L.F.L.R., 2008. "Development of methodologies for simulation of pollutants using free code". *PhD thesis in Sciences in Chemical Engineering., COOPE/ UFRJ, Rio de Janeiro*.

6. RESPONSIBILITY NOTICE