Simulation of continuous gas-lift technique using OpenFOAM and the Volume of Fluid method

Naim J.S. Carvalho[1], Livia F. C. Jatobá[1], Grazione de Souza[1], Helio P.A. Souto[1]

[1]*Instituto Politécnico, Universidade do Estado do Rio de Janeiro, Rua Bonfim, 25, Vila Amélia, Nova Friburgo – RJ, CEP: 28.625-570, Brazil*
njscarvalho@iprj.uerj.br, liviajatoba@iprj.uerj.br, gsouza@iprj.uerj.br , helio@iprj.uerj.br

Abstract. To maintain optimal production rates in oil reservoirs as reservoir pressure declines over time due to oil extraction, artificial lift methods are employed. One such method is the gas lift technique, where compressed gas, typically a mixture of hydrocarbons with low molecular weight, is injected into the lower section of the pipeline through valves. This injection provides additional energy to propel oil to the surface, and the resulting gas-oil mixture has lower effective density, facilitating transport to the surface. This artificial lift method, when applied to restore productivity is not limited by the well depth and can be applied to offshore facilities and allows operation regimes in both continuous or intermittent lift. While computational fluid dynamics (CFD) simulations for gas lift problems commonly use commercial softwares, we propose CFD simulations using the free toolbox OpenFOAM-10 using the Volume of Fluid (VOF) method. This approach aims to simulate a gas lift scenario where a methane-like gas is injected horizontally into a pipe containing upward-flowing oil, replicating real-world oil industry conditions. The main goal is to investigate the gas lift process, analyzing how the gas propels oil and increases oil production compared to scenarios without gas lift. We focus on the continuous gas lift injection regime and compare VOF simulation results with those obtained from the Smoothed Particle Hydrodynamics (SPH) method for the same problem.

Keywords: Multiphase flow, gas-lift, Volume of Fluid, OpenFOAM.

1 Introduction

Fluid production from oil reservoirs declines over time as reservoir pressure decreases. To maintain desired production rates, artificial lift methods must be employed when natural driving mechanisms are insufficient. These techniques are all based on increasing reservoir energy via artificial means to enhance production pressure drawdown. One such method is gas lift, where compressed gas is injected into a lower section of the system where fluid is produced. It is expected that, as the gas enters the pipe, its expansion propels oil to the surface and aerates other fluids, resulting in a mixture with lower effective density that is easier to transport to the surface [1].

The use of computational fluid dynamics (CFD) tools to simulate multiphase flows is an option, with a broad range of methods and software available that can simulate various phenomena. Among the options in the literature, many problems are addressed using commercial software to simulate different gas lift scenarios, while few studies utilize free software.

Abdulkadir et al. [2] compared results obtained from experiments and CFD studies of slug flow in vertical risers using the commercial codes Star-CD and Star-CCM+. The authors simulated a domain comprising a 6 m long cylinder with a 0.067 m diameter, injecting air and silicone oil at the cylinder's bottom. They characterized observed flow regimes, identifying slug flow for the studied superficial velocities. The evaluation of phase distribution was conducted using the void fraction, with the authors reporting good agreement between CFD and experimental results. The Volume of Fluid (VOF) method was utilized to model slug flow in this work.

Sami and Turzo [3] studied intermittent gas lift technique using Ansys Fluent software, considering an 18 m long vertical tube with a 76 mm diameter. Their simulation involved a system where both gas and fluid were injected at the pipe's bottom, aiming to identify characteristics of the slug flow regime and capture velocity profiles for different injection pressures. They reported three major characteristics of slug flow: initial rapid slug creation and acceleration during gas injection, slug velocity reaching a constant value, and rapid acceleration of the slug near the production end of the tube. Consequently, the authors concluded that the VOF method used in their CFD tool showed good agreement with experimental results under similar scenarios.

Rodrigues et al. 4 simulated the effects of different injection angles on gas lift efficiency, also using Ansys CFX commercial CFD software. They simulated methane injection into upward flowing liquid water through a 3 m long tubing with a 127 mm diameter, with gas injected via a 12.7 mm orifice. Their main reported result was the average pressure drop comparison between gas injected countercurrent to the main flow versus injection in the same direction as the flow. The authors reported that CFD results predicted a reduction in local pressure drop at gas injection points, consistent with a theoretical model they developed. They concluded that for lower gas flow rates, injecting gas against the main flow direction could introduce a minor overall pressure drop.

In this work, we adopt the Volume of Fluid (VOF) method, implemented via OpenFOAM software, to simulate a simplified gas-oil problem that replicates a previous work of ours as reported in Carvalho et al. [5], which used the Smoothed Particle Hydrodynamics (SPH), a meshfree particle-based method, to implement a 2D gas-lift system. In the VOF method, the interface between two different phases is identified in a given cell of the domain using a void fraction (also known as phase fraction): this variable is solved as an additional equation in the mathematical model alongside the Navier-Stokes equations. It can take values of 1 when a cell is filled with liquid, 0 when it contains only the other phase, and any value in between indicates the presence of the interface within that cell. The OpenFOAM software has been previously employed for similar problems, as reported by Tocci et al. [6], who simulated an air-water problem in a vertical riser and evaluated the total liquid holdup in the system.

In a recent study [7], we explored the use of OpenFOAM software to simulate gas-lift systems, successfully reproducing certain results from experiments conducted by Guerra et al. [8], particularly regarding the pressure drop in the system. This current work extends upon those findings by simulating now a problem where methane gas is injected into a system which contains a heavy oil. We evaluate the numerical convergence of the solution, the total pressure drop in the system and the evolution of the phase distribution as the flow evolves.

2 Mathematical Model

OpenFOAM includes the *interFoam* module, used in this paper, which is a solver using the VOF method for the Navier-Stokes equations that can be used for problems with two immiscible, incompressible and isothermal fluids. In this method properties are constants in cells where only one fluid occurs, expect at the interphase. The simplified mathematical model, for one dimension, uses the following governing equations [9] :

2.1 Continuity Equation

The constant-density continuity equation is :

$$\frac{\partial u_j}{\partial x_j} = 0, \tag{1}$$

in which u is the velocity and x is the spatial coordinate.

2.2 Momentum Equation

$$\frac{\partial(\rho u_j)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j u_i) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}(\tau_{ij} + \tau_{t_{ij}}) + \rho g_i + f_{\sigma i}, \tag{2}$$

where p is the pressure, $\tau_{t_{ij}}$ and τ_{ij} are the turbulent and viscose stresses, g_i is the gravitacional acceleration, $f_{\sigma i}$ represents the surface tension and ρ is the density, written as:

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2. \tag{3}$$

Inside fluid 1, α (the phase fraction variable), is 1 and its density is ρ_1, while inside fluid 2, that has density ρ_2, it is 0. In a cell that contains both fluids, the phase fraction can assume any value between 0 and 1, and that cell is said to contain the interphase between the fluids [9].

The surface tension is modelled by the Continuum Surface Force (CSF) model, as proposed by Brackbill et al. [10]. In this $f_{\sigma i}$ is calculated as an extra source term, such as as:

$$f_{\sigma i} = \sigma \kappa \frac{\partial \alpha}{\partial x_i}, \tag{4}$$

where σ is the surface tension coefficient and κ represents a curvature, that can be approximated as:

$$\kappa = -\frac{\partial}{\partial x_i}\left(\frac{\partial \alpha / \partial x_i}{|\partial \alpha / \partial x_i|}\right). \tag{5}$$

2.3 Equation for the interphase

In order to determine where the interphase between the fluids is, the following adittional equation is solved for the α variable:

$$\frac{\partial \alpha}{\partial t} + \frac{\partial (\alpha u_j)}{\partial x_j} = 0. \tag{6}$$

3 Methodology and simulated problem

3.1 Problem definition

In the work reported in Carvalho et al. [5], we studied a problem in which a vertical pipe is initially filled with heay oil flowing upwards as methane gas is injected into the system. The main pipe diameter is 14 cm while the gas is injected via a 12 mm orifice at a 90° angle to the main pipe. Since we are only interested in the section right above the injection point, the domain can be safetly reduced to a 1 m long pipe, with the injection point located 0.5 m above the oil inlet at the bottom.

Fluid properties remain constant throughout the simulation, with oil density set to 728 kg/m^3 and methane density set to 0.717 kg/m^3. The kinematic viscosity is 9.2 x 10^{-4} m^2/s for the oil and 2.75 × 10^{-4} m^2/s for gas. Gravity acceleration is set to -9.81 m/s^2 vertically, and a surface tension of 0.045 N/m is applied.

The initial geometry was generated using the FreeCad software which is then exported as a stereolithography and treated with OpenFOAM's *cfMesh* utility to generate the mesh for the domain. Finally, we use OpenFoam version 10's *interFoam* module to solve the governing equations to obtain the numerical solution for the simulated flow, which considers a laminar, isothermal and incompressible problem.

3.2 Boundary and initial conditions

At t=0 s, the velocities for all fluids inside the domain were set to 0 m/s, with the main flow section initially filled with oil, and the system was set to standard atmospheric pressure (101.3 kPa). The sections for gas inlet and oil inlet have preescribed velocities equal as the ones used in Carvalho et al. [5], which lead to 0.1 m/s for the oil velocity and 0.48 m/s for the gas velocity. Solid boundary walls were given a no-slip boundary condition. The outlet velocity was calculated using the OpenFOAM's *pressureInletOutletVelocity* condition.

The pressure, at the outlet it is prescribed using the *prghTotalPressure* condition, with the static pressure equal to atmospheric pressure. Meanwhile, pressure at inlets and walls was calculated using OpenFOAM's *fixedFluxPressure* condition.

For the phase fraction, α, we use the *fixedValue* condition, prescribing it as 1 at the inlet at the bottom and 0 at the gas inlet. Walls received a *zeroGradient* condition for this phase, and finally, the outlet was calculated using the *inletOutlet* condition.

3.3 Numerical aspects

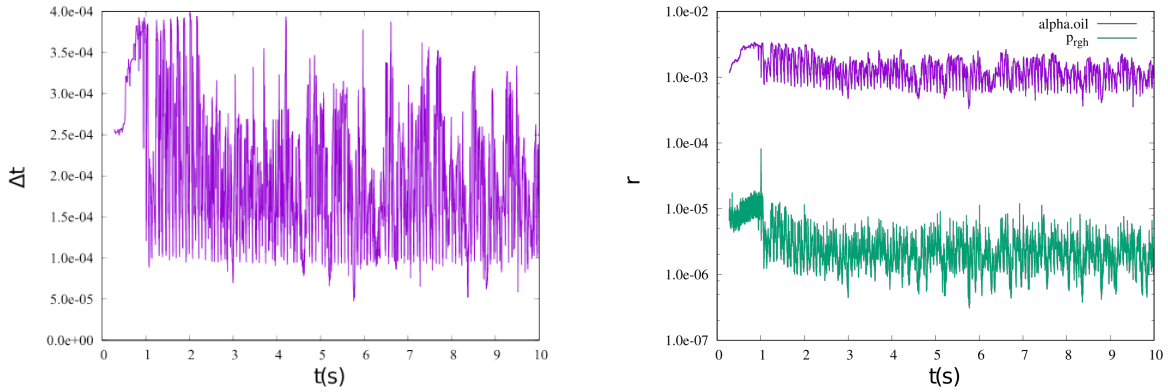
We define the maximum Courant for the time step calculation in the transient solution as 0.5. The solution for pressure and velocity were obtained via a PIMPLE algorithm, which combines the PISO (Pressure Implicit with Splitting of Operator) and SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) approaches. The PIMPE loop used a total of 6 iterations for both the phase fraction and the pressure per time step. The Generalized Geometric-Algebraic MultiGrid solver for the pressure was used, with a 10^{-7} tolerance. For the phase fraction variable the the solution used a Gauss-Seidel solver and 10^{-8} as tolerance. For the time discretization, we used the Euler method for the phase fraction and a second order scheme for the remaining equations, which was also the same order for the scheme used for the spatial dimensions.

4 Results

We simulate a total time of 10 s with 3 different meshes. First we use a mesh with 53,725 cells, then 186,340 cells and for the most refined mesh had 253,882 cells. We also use the results obtained with the less robust mesh as the initial conditions for the most refined ones.

Figures 1a and 1b shows the numerical convergence results for the 53k mesh. Similar results were found for the other meshes and omitted to avoid repetition. Figure 1a shows the evolution of the time step to fulfill the Courant criterion from the CFL condition. It is seen that, although as oscilation is observed in the initial second of the simulation, the time step for this mesh was consistently keept around 10^{-4} afterwards. We calculate its average value to be at 1.75 x 10^{-4} s.

Figure 1b shows the normalised residuals for both phase fraction and pressure. It can be noted that the residuals, for each time step, for the phase fraction do not fall below 10^{-3} and the ones for the pressure solution start at 10^{-5} and drop to 10^{-6} as the the simulation progresses. Even though we expected a more accurate result for the phase fraction, with the simulation paramaters the accurary was limited to 10^{-3} for the phase fraction. Given that the phase fraction has a lower absolute value when compared to the pressure variable, obtaining smaller residuals becomes harder, while the residuals for the pressure is close to the tolerance set, which indicates a good numerical convergence.



(a) Time step

(b) Residuals

Figure 1. Time step and residuals over time for 53k mesh.

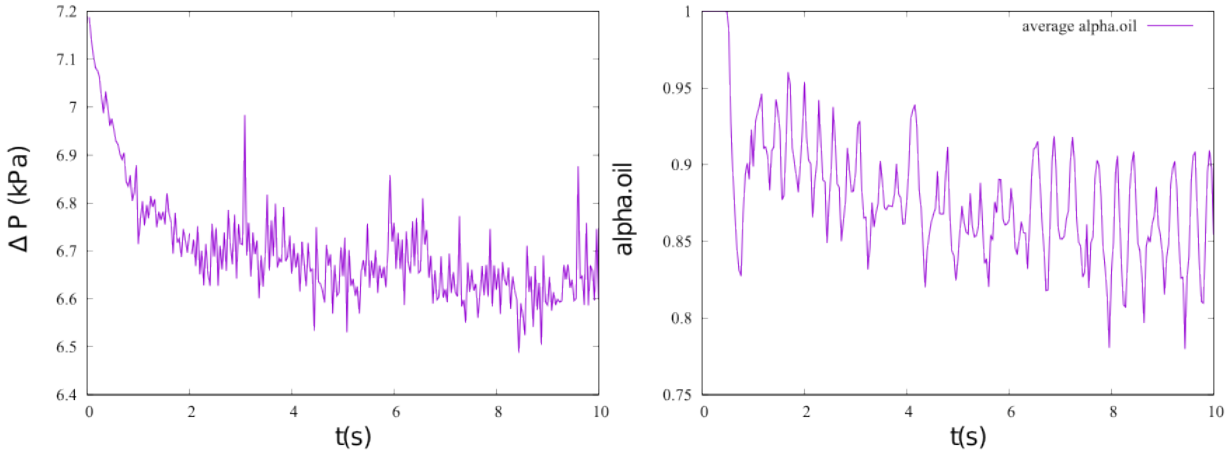
We also evaluate the pressure drop in the system by taking the difference in values for it in between two points: at the oil injection (y = 0 m) and at the top most section of the main pipe (y = 1 m). Figure 2a presents the total pressure drop, ΔP, for each time step.

As it is shown in the picture, the pressure difference quickly drops in the initial seconds of the simulation as the gas is injected but, from 5 s to 10 s, the pressure drop stabilizes. Therefore, we considered the transient simulation converged after 5 s and took average values in that time range.

Table 1 shows the average values for the 3 meshes: 53K, 186K and 253K, respectively. The value for the pressure drop for the 53k mesh is 6.65 kPa with a standard deviation of 0.07. For the 186k mesh, the corresponding values were 6.62 KPa and 0.05 and, for the 253K mesh, the total pressure drop obtained was 6.17 KPa but the highest deviation, 1.37, was also seen in this mesh. We evaluate the error among those solutions by comparing them to the most refined mesh, which leads to around 7% error in both cases, as also reported in Table 1.

We also analyse the flow development by plotting the the phase faction of the heaviest phase, the oil, over time. The phase fraction of the oil was a volume average value calculated in a section from y = 0.7 m to y = 0.8 m

(0.2 m above the gas injection). Figure 2b shows the general behavior of the oil phase by using the phase fraction variable in the less refined mesh.



(a) Pressure drop in the main pipe section.

(b) Phase fraction evolution above the gas injection.

Figure 2. Total pressure drop and phase fraction over time for 53k mesh.

Table 1 also presents the volume average value for the phase fraction of oil in the studied section, showing that the absolute error for this variable, which taken considering the most refined mesh as the best solution, is consistent among the meshes, which works a mesh independence check. The table also presents the average time step, which shows that even for a 2D simplified problem the time step has a low magnitude which could be even more limiting if we had considered a 3D simulation.

Table 1. Average results for different mesh refinements.

Mesh (cells)	Property	Value	Standard Deviation	Error
53K	$\overline{\Delta P}$	6.65 KPa	0.07	7.76 %
186K	$\overline{\Delta P}$	6.52 KPa	0.05	7.33 %
253K	$\overline{\Delta P}$	6.17 KPa	1.37	-
53K	$\overline{\Delta t}$	1.75 x 10^{-4}	-	-
186K	$\overline{\Delta t}$	1.36 x 10^{-4}	-	-
253K	$\overline{\Delta t}$	9.52 x 10^{-4}	-	-
53K	$\overline{\alpha_{oil}}$	0.860	-	1.485 x 10^{-3}
186K	$\overline{\alpha_{oil}}$	0.857	-	4.575 x 10^{-3}
253K	$\overline{\alpha_{oil}}$	0.852	-	-

We verified the incompressible flow hypothesis by calculating the continuity error over the domain, that is, the divergence of the velocity. At each time step, we obtained a maximum value of 10^{-9} and the cumulative over the whole simulation had magnitude of 10^{-3}. Next, we verified the velocity profile for the oil phase below the gas injection point, as shown in Figure 3. It can be noted that the entrance region was long enough for the development of the velocity profile, given that the expected parabolic behavior is seen immediately before the gas inlet.

Figure 4 shows a section of the geometry comparing side-by-side the visualizations obtained with the VOF method and the SPH method, as presented in Carvalho et al. [5] for the same time frame. In this figure, the flow regime can be inferred from the evolution of the oil phase fraction: in the initial second, it is at its maximum, as the pipe is initially filled with oil, but as time progresses, the gas injection causes this value to drop, representing the aeration of the mixture. The oscillatory behavior in the phase fraction indicates the formation of slugs in the pipe, as expected for this type of problem. These results closely resemble what was observed in Carvalho et al. [5] when the SPH method was used to simulate the same geometric configuration.

Qualitatively, the flow regime observed with both methods shows that bubble formation occurs above the gas injection section, with bubbles that don't fill the entire flow section and a higher concentration next to the wall

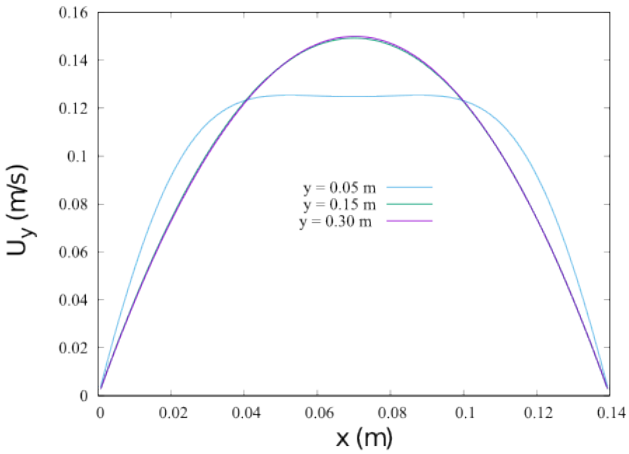
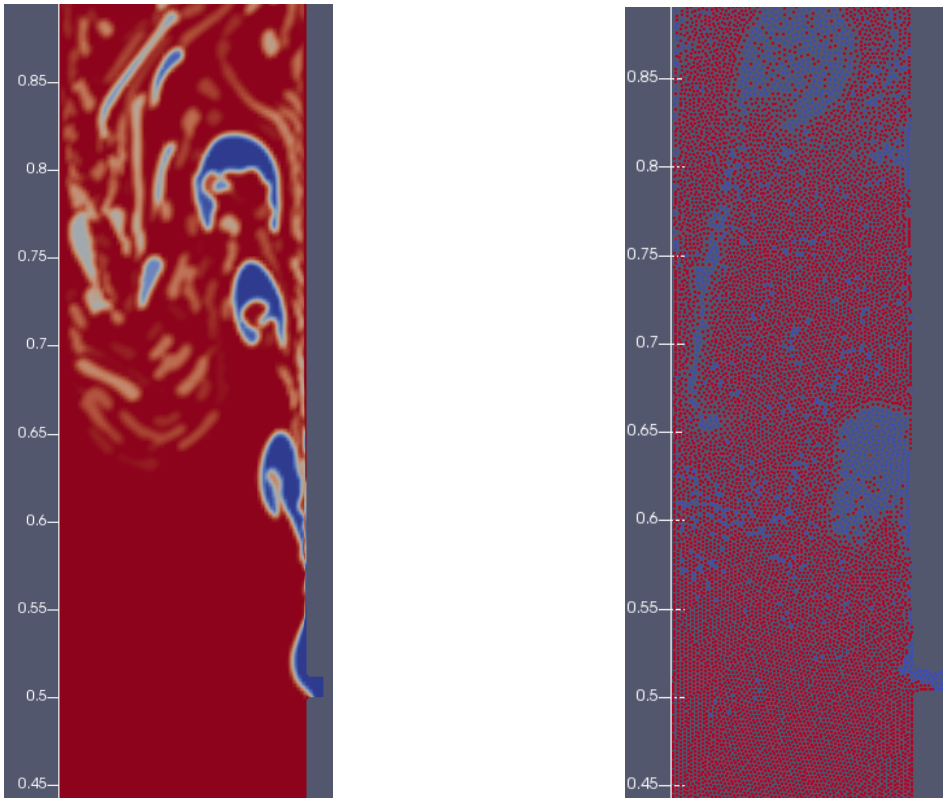


Figure 3. Velocity profile in the entry region (mesh 186K).

where the injection occurs.



(a) VOF (b) SPH

Figure 4. Snapshot of the main flow section at t = 5.0 s, with the VOF method and the SPH method.

5 Conclusions

The simulation of a gas-lift system has been successfully carried out using the Volume of Fluid (VOF) method via OpenFOAM. In this study, we studied a simplified 2D problem involving methane injection into a system, achieving consistent results across three different mesh refinements. Regarding computational cost, the simulation with the least refined mesh took approximately 5 hours, which is considerably less than the 22 hours required to simulate a similar problem using the Smoothed Particle Hydrodynamics (SPH) method. While comparing other

metrics between the two methods is challenging due to their different approaches to solving the same problem, the qualitative results from visualizing the two phases in the flow show good agreement. Some phenomena characteristic of this type of flow, such as bubble formation, were correctly captured by both methods, demonstrating the feasibility of their use in simulating such problems.

Acknowledgements. This study was financed in part by the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior – Brasil (CAPES) – Finance Code 001.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] G. Boyun, W. C. Lyons, and A. Ghalambor. *Petroleum Production Engineering - A Computer-Assisted Approach*. Elsevier Science & Technology Books, 2007.

[2] M. Abdulkadir, V. Hernandez-Perez, S. Lo, I. S. Lowndes, and B. J. Azzopardi. Comparison of experimental and computational fluid dynamics (cfd) studies of slug flow in a vertical riser. *Experimental Thermal and Fluid Science (EXP THERM FLUID SCI)*, vol. 68, pp. 468–483, 2015.

[3] N. A. Sami and Z. Turzo. Computational fluid dynamic (cfd) modelling of transient flow in the intermittent gas lift. *Journal of Petroleum Research*, vol. 5, pp. 144–193, 2020.

[4] H. T. Rodrigues, A. R. Almeida, D. C. Barrionuevi, and R. S. Fraga. Effect of the gas injection angle and configuration in the efficency of gas lift. *Journal of Petroleum Science and Engineering*, vol. 198, n. 108126, 2021.

[5] N. J. Carvalho, L. F. Jatoba, de G. Souza, and H. P. Souto. Gas-lift simulation using smoothed particle hydrodynamics. In *Proceedings of the XLIV Ibero-Latin-American Congress on Computational Methods in Engineering*. ABMEC, 2023a.

[6] F. Tocci, F. Bos, and R. Henkes. Cfd for multiphase flow in vertical risers, 2017.

[7] N. J. Carvalho, L. F. Jatoba, de G. Souza, and H. P. Souto. Numerical simulation of gas lift systems using the volume of fluid method. In *Proceedings of the XLIV Ibero-Latin-American Congress on Computational Methods in Engineering*. ABMEC, 2023b.

[8] L. A. O. Guerra, B. O. Temer, J. B. R. Loureiro, and A. P. Freire. Experimental study of gas-lift system with inclined gas jets. *Journal of Petroleum Science and Engineering*, vol. 216, n. 110749, 2022.

[9] C. Greenshields. Openfoam v10 user guide. ttps://doc.cfd.direct/openfoam/user-guide-v10. Accessed: 25 - July - 2023, 2022.

[10] J. Brackbill, D. Kothe, and C. Zemach. A continuum method for modeling surface tension. *Journal of Computational Physics*, vol. 100, n. 2, pp. 335–354, 1992.