Numerical simulation of gas lift systems using 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. Oil production in reservoirs declines over time as pressure decreases, presenting a challenge in maintaining economically desired production rates. Artificial lift methods can be used to address this issue, considering the characteristics of the production system, such as reservoir and fluid properties, as well as facility constraints. Gas lift involves injecting gas into lower sections of the tubing through strategically placed valves along the pipeline, effectively reducing the density of the fluid mixture and facilitating its upward flow. This lifting method is applicable to offshore operations and is not limited by well depth, allowing for different types of operations, including continuous or intermittent lift. In the present work, a gas lift flow is numerically studied using the volume of fluid method and a fluid mixture of water-air. The method is based on defining the liquid volume fraction and employing a transport equation to capture the interface. The interFoam solver from OpenFOAM-10 is used to simulate a three-dimensional two-phase gas lift scenario. The setup consists of vertical water flow in a 44 mm pipe diameter, while air is injected through a 2 mm diameter orifice in the ortoghonal-flow direction. The total pressure drop is evaluated and compared with experimental data. The phase fraction and mean velocity profile are also obtained.

Keywords: Two-phase flow, Volume of Fluid, Gas-lift.

1 Introduction

In the oil recovery process, artificial lift methods are used to bypass situations when a reservoir eventually is unable to produce flow at the desired rate. Gas lift technology increases production rates by injecting gas into the lower sections of the pipe, enhancing the transportation of the mixture to the surface [1, 2]. While each artificial lift method has its own characteristics, the project design of such systems requires an understanding of the multiphase flow through both experimental and simulation studies.

Modeling approach for multiphase flow in the computational fluid dynamics of gas lift applications vary from the more frequently studied grid based methods, such as the Volume of Fluid or the Eulerian averaging methods [3–6], to the meshfree Smoothed Particle Hydrodynamics and the Lattice Boltzmann Method [7]. Defining the best modelling approach is challenging and its complexity is related to the characteristics of the flow regime and interface topology. The gas lift's flow regime can exhibit variations, transitioning between bubbly and slug patterns. Consequently, the comparative analysis of experimental data with distinct modeling approaches stands as an relavant development in gas lift design.

The present work focus on the the Volume of Fluid (VOF) using the OpenFOAM toolbox for gas lift applications. The VOF method can track the interface between two or more immiscible fluids [8]. It is based on a volume fraction variable in each cell: for a two-phase gas-liquid problem, this variable goes from 1, for when the cell is filled with the liquid, to 0, when the cell contains only the gas phase. The momentum equation and the continuity equation are solved as averages using the volume fraction as weight [9]. While OpenFOAM is a powerful, open source CFD framework, some aspects such as appropriate boundary conditions to reproduce experimental data are still a challenge.

OpenFOAM has been used by Tocci, Henkes and Bos [3] to study a water-air problem in a vertical riser, evaluating the total liquid holdup as the main property, but authors ultimately decided for a modified VOF implementation. Their simulated case is also one where air was injected by the same entrance as the water, which differs from gas-lift systems. A recent study by Rodrigues et. al. [10] simulating gas-lift systems via CFD focused on the effects of the various injection angles of the gas into the sytem, but using the software ANSYS CFX 13.0 .

Experimental data is available in other works, such as the one by Guerra et al. [11] that studied bubble diameter distribution in a gas-water system with different diameters and angles for injection sections. The pressure drop across the system and velocity profiles in different sections is provided, which allows this problem to be reproduced, with certain considerations, numerically via OpenFOAM. This present work presents problem definition and preliminar comparative results of the CFD simulation with the case presented by [11] on a cross-flow vertical gas-liquid pipe. This work aims to study the viability of using the Volume of Fluid (VOF) method for designing gas lift systems.

2 Numerical Model

The *interFoam* module is a solver for the Navier-Stokes equations for problems with two incompressible, isothermal and immiscible fluids and properties are constants in cells where only one fluid occurs, except at the interphase. The mathematical model uses the following governing equations:

2.1 Continuity Equation

The constant-density continuity equation is :

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

where 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, τ_{ij} and $\tau_{t_{ij}}$ are the laminar and turbulent tensors, respectively, g_i is the gravitacional acceleration, $f_{\sigma i}$ represents the surface tension and ρ is the density, such as:

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

If the cell has only fluid 1, α is 1 and its density is ρ_1, while in a cell with only fluid 2, the density is ρ_2 and α is 0. In a cell that contains both fluids, α can assume any value between 0 and 1, and that cell is said to contain the interphase between the fluids.

The surface tension is modelled as a continuum surface force (CSF). $f_{\sigma i}$ is calculated 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

The final governing equation is an adittional equation solved for the phase fraction α variable, used to determine the position of the interphase.

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

3 Metodology

3.1 Problem definition

The case study in this work is based on the multiphase results of Guerra, Loureiro and Freire [11] . The design is applicable for gas lift systems, as air is injected into liquid cross flow in a vertical pipe. The solver `interFoam` from OpenFOAM-10 was used for the CFD solution using VOF method. The simulation was simplified for a laminar bidimensional case with 2.252 m long and 44 mm wide and `blockMesh` utility was used for geometry and mesh construction. Air is injected at 1.45 m from the water inlet at the bottom of the main pipe by a 2 mm section, in a 0° angle with the main pipe. The water flow rate is 2 m^3/h while the air flow rate is 0.57 m^3/h. Fluid properties were constant and the density was $1000 kg/m^3$ for water and 1 kg/m^3 for air. The kinematic viscosity was $10^{-6} m^2/s$ for water and $1.4810^{-5} m^2/s$ for air. A surface tension of $0.07 N/m$ was used and gravity was $9.81\ m/s^2$ in the opposite direction of the inlet flow.

3.2 Boundary conditions

The computational domain of the 2D vertical pipe is defined by the following boundaries: a water inlet at the bottom plane, a gas inlet in the left wall and an outlet at the top. The boundary conditions set up for the case is based on fixing the velocity at the inlets and the pressure at the outlet. In OpenFOAM this can be done using different types of boundary conditions. The inlet velocities were prescribed using the `flowRateInletVelocity`, and the volumetric flow rate was applied with a laminar profile. The velocity at the outlet was calculated using the `pressureInletOutletVelocity` and no slip condtion was used at the walls. The pressure at the outlet was prescribed using the `prghTotalPressure` with a static pressure of $1.0110 \times 10^5 Pa$. Pressure at inlets and walls were calculated using `fixedFluxPressure`. Phase fraction (`alpha.water`) was prescribed at the bottom inlet as 1, and 0 at the gas inlet, using `fixedValue`. Phase fraction was calculated using `inletOutlet` at the outlet an `zeroGradient` at the walls.

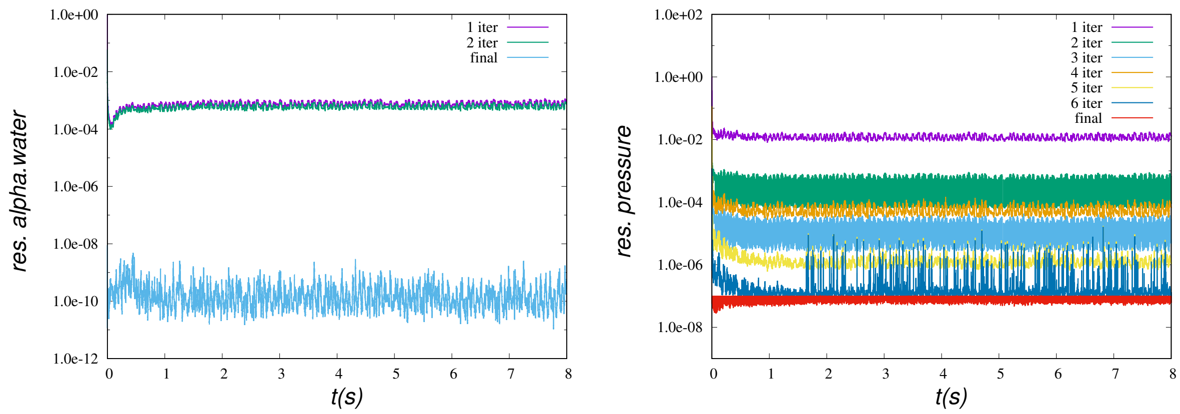
3.3 Numerical setup

The transient solution of the flow had the time step calculation with a maximum Courant criterion of 0.5. PIMPLE convergence was established using 2 outer iterations, no momentum predictior and 3 iteration cycles for the pressure equation. The pressure system was resolved using a Generalized Geometric-Algebraic MultiGrid solver and a tolerance of 10^{-7}. The phase fraction numerical solution was set up with semi-implicit MULES limiter with 5 iterations using a Gauss Seidel solver and a tolerance of 10^{-8}. The number of iterations of the phase fraction equation within a solution step was set up for 2.

Spatial discretization schemes are generally set up as second order and Euler method is used for time discretization. The advection scheme for the velocity is the second order upwind bounded `linearUpwind grad(U)`. The advection term of phase fraction equation was also discretized using the limited second order scheme `interfaceCompression vanLeer 1`.

4 Results

Numerical convergence results are analysed for a bidimensional 10^6 volumes mesh. The simulation was set up with two iterations in the PIMPLE convergence loop. PIMPLE was configured with one solution iteration for the phase fraction equation and three iterations for the pressure equation. At the end of each time step, the phase fraction equation was iterated twice and the pressure equation six times. Figure 1 shows the residues, at each time step, for the phase fraction and pressure variables. The legend indicates that 'i iter' are the initial residues of each iteration, that is, the residue before the algebraic system solution, while the 'final' is the final residue, after the algebraic system solution, at the end of the step of time. Figure 1(a) shows that the increment of iterations in the phase fraction solution does not result in better convergence for this variable. On the other hand, Figure 1(b) shows the relevance of iterations of the pressure solution to guarantee the convergence of this variable in the transient solution. At the end of each time step, convergence to the order of 10^{-10} and 10^{-7} was guaranteed for the phase fraction and pressure, respectively.

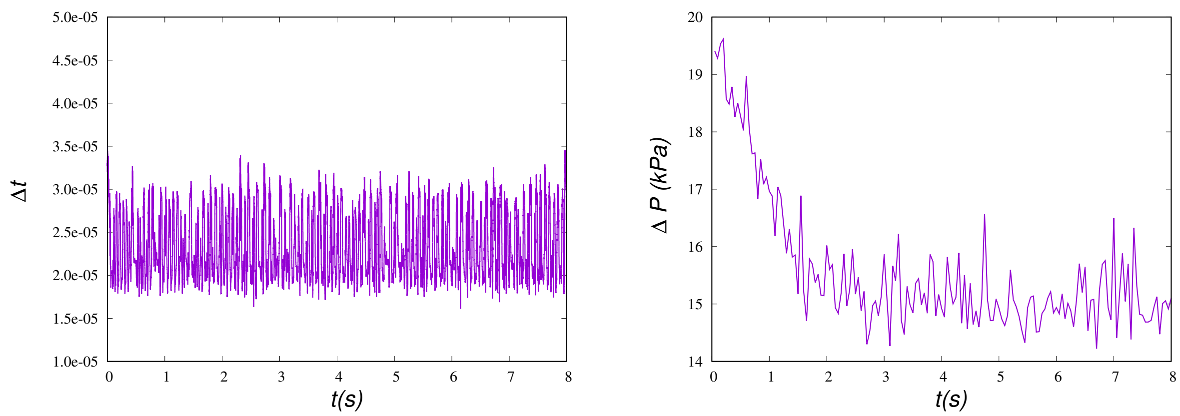


(a) Phase fraction (b) Pressure

Figure 1. Residuals, at each time step, using 120k mesh for fase fraction and pressure.

The transient solution time step was calculated based on a maximum Courant criterion of 0.5 . Figure 2(a) illustrates that the time step was approximately on the order of 10^{-5}. Note that the case studied in this work is simplified, that is, a 2D mesh with 10^{6} volumes. Nonetheless, even with this simplified scenario, the time step's order of magnitude for achieving convergence has been significantly reduced. This is a general drawback that must be observed when adopting a CFD simulation using VOF.

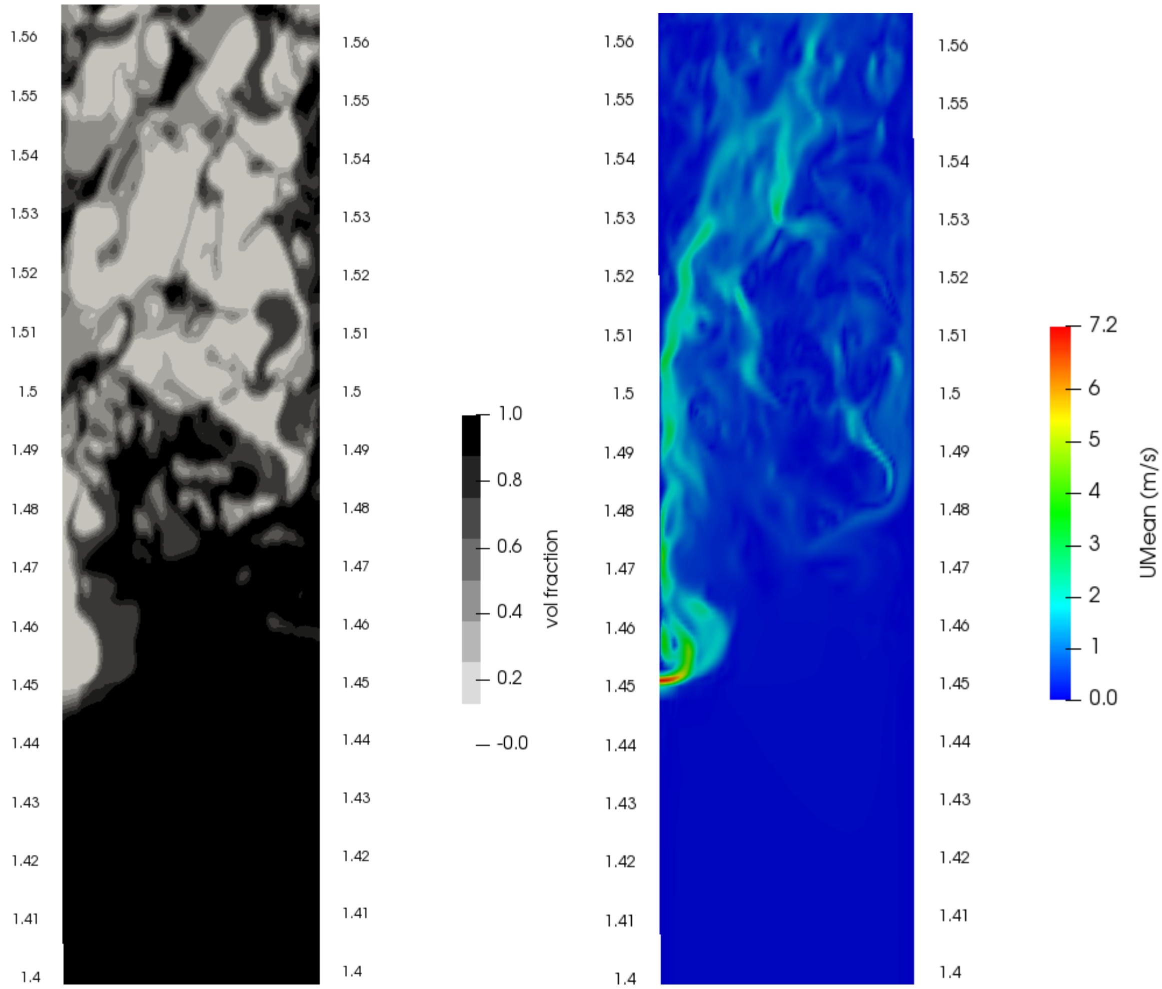
The analysis of the numerical convergence of the transient solution is completed by observing the value of the pressure drop in a 2-meter section of the pipe, over time, as shown in Figure 2(b). It can be noted that, from 2 seconds until the end of the simulation, the value of the pressure drop starts to fluctuate around an average value of $15.1\ kPa$, with a standard deviation of 0.5. The pressure drop estimated with the CFD simulation can be compared with the similar case from [11] and a 3.5% error was found. Therefore, even for a simplified 2D case,the VOF method was able to predict pressure drop for a gas lift system with a good accuracy.



(a) Time step. (b) Pressure drop.

Figure 2. Time step values and pressure drop over time.

The simulation was considered converged after 2 seconds. Field values between 2 and 8 seconds of simulation were used to calculate temporal averages. Figure 3 shows the value of the average phase fraction and magnitude of velocity in the pipeline section close to the gas injection.



(a) Average fase fraction (b) Average velocity magnitude.

Figure 3. Average values of fase fraction and velocity magnitude.

The phase fraction, as seen in Figure 3(a), shows the general behavior in the section above the gas injection point captured by the VOF method. Due to the low flow rates, the gas phase is quickly carried by the water, and the results observed here ressemble a mixed flow regime, while experiments show in a similar configuration the occurrence of bubbling flow, with bubbles being formed right above the gas injection [11]. With the current study, we cannot confirm if VOF is able to detect bubbling phenomena, such as formation and coalescence.

The mean velocity observed 3(b) shows that the after the injection, the air velocity rapidly decreases when the mixture with water starts. The gas phase first tends to travel closer to the wall where the injection occurs, with high velocities were obtained in first 5 cm of the pipe. After that region, the gas is found in a mixture with the water and the highest velocities start to be observed in areas in the middle of the pipe.

5 Conclusions

The Volume of Fluid method implementation in OpenFOAM was able to correctly capture the pressure drop in a pipe for a gas lift problem involving the injection of air into a system initially containing only water and the difference in values. When compared to a similar experimental case, the difference in the values was found to be 3.5%.

The phase fraction analysis shows that, for the simulated problem, the VOF method predicts a mixed flow regime, with air initially traveling close to the wall but rapidly mixing with water above the injection point. These show that the VOF method, in particular the implementation available in OpenFOAM, is able to capture phenomena reported in experiments on gas-lift systems

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] S. Guet and G. Ooms. Fluid mechanical aspects of the gas-lift technique. *Annual Review of Fluid Mechanics, 38*, vol. 38, 2006.
[3] F. Tocci, R. A. W. M. Henkes, and F. Bos. Cfd for multiphase flow in vertical rises. *18th International Conference on Multiphase Production Technology*, 2017.
[4] 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.
[5] Z. Rek, J. Gregorc, M. Bouaifi, and C. Daniel. Numerical simulation of gas jet in liquid crossflow with high mean jet to crossflow velocity ratio. *Chemical Engineering Science*, vol. 172, 2017.
[6] F. Qi, G. Yeoh, S. Cheung, J. Tu, E. Krepper, and D. Lucas. Classification of bubbles in vertical gas-liquid flow: Part 2-a model investigation. *International Journal of Multiphase Flow*, vol. 39, pp. 135–147, 2012.
[7] T. Douillet-Grellier, S. Leclaire, D. Vidal, F. Bertrand, and F. De Vuyst. Comparison of multiphase sph and lbm approaches for the simulation of intermittent flows. *Computational Mechanics*, pp. 1–, 2019.
[8] S. M. Damián. *An extended mixture model for the simultaneous treatment of short and long scale interfaces*. PhD thesis, Instituto de Desarrollo Tecnologico para la Industria Quimica, 2013.
[9] *Comparison of single drop impact simulations with experiments*. The West Virginia University, 2015.
[10] 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.
[11] 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.