Gas-lift simulation using Smoothed Particle Hydrodynamics

Naim Jessé dos Santos Carvalho[1], Livia Flavia Carleti Jatobá[1], Grazione de Souza[1], Helio Pedro Amaral Souto[1]

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

Abstract. Oil reservoir production reduces over time as the pressure decreases, posing a challenge to maintaining economically desired production rates. We can use artificial lift methods to address this issue, considering the characteristics of the production system, such as reservoir and fluid properties or surface facility constraints. Here, we focus on the gas lift technique, which involves injecting compressed gas into lower sections of the tubing through valves installed along the pipeline. As the gas aerates the oil, it decreases the effective density of the fluid, making it easier to reach the surface. It is suitable for offshore installations and not limited by the well depth, allowing continuous or intermittent lift to restore productivity. However, fewer studies use particle-based methods such as the Smoothed Particle Hydrodynamics method (SPH). It is a Lagrangian mesh-free method, and we can use this method to simulate multiphase flows. Therefore, we chose it to reproduce a simplified gas lift test case that involves a two-dimensional two-phase flow within a vertical pipe that contains oil, and we inject high-pressure gas into the system through a horizontal valve. The main objective is to verify if the SPH method can reproduce the phenomena associated with gas lift operations.

Keywords: Gas lift, Multiphase flow, Smoothed Particle Hydrodynamics

1 Introduction

Although our main objective is the simulation of reservoirs, ranging from single-phase to multiphase flow considering slightly compressible fluids and rock, including well-reservoir coupling and advanced recovery techniques [1–3], we have recently turned our interest to the problem of transport of oil through pipelines from the reservoir to the surface installations.

As in advanced recovery techniques applied to stimulate oil production in reservoirs, there are ways to increase the flow of oil pumped from the deposits. Among them, we can mention the gas lift. In it, we inject gas into the pipelines containing the oil through valves positioned along them [4]. For example, we can use different strategies, continuous or pre-set time interval injections [5]. As the gas has a density lower than oil, their mixture with the oil should result in a fluid with a lower average value, thus increasing the quantity produced.

In addition to numerical methods that employ computational meshes, such as, for example, finite difference and finite volume methods, we have also been working with the mesh-free Lagranean Smoothed Particle Hydrodynamics method [6]. In addition to its well-known advantages, such as being easily parallelizable, there is a version aimed at simulating multiphase flows found in DualSPHysics [7], which is free software that can be redistributed or modified under the terms of the GNU Lesser General Public License.

Therefore, the main objective of this work is to verify the feasibility of using the multiphase version of DualSPHysics in the study of the gas lift technique. It is an initial study in which we are interested in simulating gas injection in a circuit containing a section of the transport pipeline and a reservoir for separating the gas-oil mixture. We emphasize that we will be focused on the qualitative aspects of the two-phase flow and not on the quantitative ones, which we will address in future works.

With regard to numerical simulations, due to the computational effort required, they were performed using the API (Application Programming Interface) CUDA (Compute Unified Device Architecture) developed by Nvidia for parallel computing using a graphics processing unit (GPU). In the case of our work, we used an Nvidia K80 GPU.

2 Governing equations

The continuity and Navier-Stokes equations govern the flow of compressible Newtonian fluids. Here, we present theses equations in their Lagrangian representations.

The continuity equation expresses the conservation of mass [6]

$$\frac{D\rho}{Dt} + \rho\nabla \cdot \vec{v} = 0, \tag{1}$$

where the equation is written in terms of the material derivative ($D\rho/Dt$), ρ is the density, t is the time, and $\vec{v}$ is the velocity vector.

On the other hand, the conservation of momentum is represented by the Navier-Stokes equation [6]

$$\rho\frac{D\vec{v}}{Dt} = -\nabla P + \mu\nabla^2\vec{v} - \frac{2}{3}\nabla\left(\mu\nabla \cdot \vec{v}\right) + \vec{f_e}, \tag{2}$$

where P represents the thermodynamic pressure, μ the viscosity of the fluid, and $\vec{f_e}$ the external force vector. However, when the flow is turbulent, the momentum equation is modified using the Moving Particle Semi-implicit model [8].

As we consider the flow of compressible fluids, an equation of state is used to determine the pressure [9]

$$P(\rho) = \frac{c_s^2\rho_0}{\gamma}\left[\left(\frac{\rho}{\rho_0}\right)^\gamma - 1\right] + P_f - a\rho^2, \tag{3}$$

where γ is an isentropic expansion factor (relationship between specific heats at constant pressure and volume), ρ_0 is the initial density of the fluid, c_s is the speed of sound, P_f is the background pressure (calculated by the code during the numerical simulation of the problem) and the last term is responsible for introducing the cohesive forces that act between the particles of a phase. The coefficient a is determined based on the properties of the different phases and the characteristic length of the problem, L_c,

$$a = 1,5g\left(\frac{\rho_\alpha}{\rho_\beta}\right)L_c, \tag{4}$$

where g is the magnitude of the acceleration due to gravity, and ρ_α and ρ_β are the initial densities of the two phases present in the flow. The characteristic length is an empirical constant that depends on the dimensions of the domain and the initial distance between the particles.

3 Discretized equations

Using the integral and particle approximations and taking $v_{ij} = v_i - v_j$, the discretized forms of the governing Equations (1) and (2) can be obtained in the formalism of the SPH method. Initially, we present the discretized form of the continuity equation [6]

$$\frac{D\rho_i}{Dt} = \sum_{j=1}^{N} m_j v_{ij} \cdot \nabla W_{ij}^h. \tag{5}$$

In this representation, m_j represents the mass of the particle j, $W_{ij}^h = W(r_{ij}, h)$, W is the kernel function, $r_{ij} = |x_i - x_j|$, h is the smoothing length, and the gradient is determined with respect to the particle *i* such that:

$$\nabla_i W_{ij}^h = \frac{x_{ij}}{r_{ij}}\frac{\partial W_{ij}^h}{\partial r_{ij}}, \tag{6}$$

where x_{ij} represent the distance between two particles *i* and j ($x_{ij} = x_i - x_j$). Here, we performed all simulations using a quintic spline type kernel function [10], one of the two available in DualSPHysics and standard when solving example cases.

Due to the presence of a gaseous phase and the discontinuities in the density at the interface separating the phases, the traditional formulation used in the SPH method is no longer applicable in this context. Therefore, an extra term is added to the momentum equation (the fourth term on the right side of the equal sign) to account for the cohesive forces of the less dense phase [11]

$$\frac{D\vec{v}_i}{Dt} = -\sum m_j \left(\frac{P_i + P_j}{\rho_i \rho_j}\right) \nabla_i W_{ij} + \sum_j m_j \left[\frac{4\mu r_{ij} \cdot \nabla_i W_{ij}}{(\rho_i + \rho_j)\left(r_{ij}^2 + \eta^2\right)}\right] + \sum_j m_j \left(\frac{\tau_{ij}^j}{\rho_j^2} + \frac{\tau_{ij}^i}{\rho_i^2}\right) \nabla_i W_{ij} \underbrace{- 2a\rho_\alpha^2 \sum_j \frac{m_j}{\rho_j} \nabla_i W_{ij}}_{\text{for the gas phase}} + \vec{F}_s + \vec{g}, \tag{7}$$

where viscous stress effects were decomposed into two terms: a contribution due to laminar viscous stress and another to SPS viscous stress (*Sub-Particle Scale*). The concept was proposed by Gotoh, Shibahara and Sakai [8] to introduce the effects of turbulence in the MPS model (Moving Particle Semi-implicit model), and τ_{ij} represents the components of the sub-particle stress tensor

$$\frac{\tau_{ij}}{\rho} = \nu_t \left(2S_{ij} - \frac{2}{3} k \delta_{ij}\right) - \frac{2}{3} C_I \Delta l^2 \delta_{ij} |S_{ij}|^2, \tag{8}$$

where $\nu_t = (C_S \Delta l)^2 \sqrt{2S_{ij} S_{ij}}$ is the turbulent eddy viscosity, C_S the Smagorinsky constant, Δl represents the initial distance between the particles, k the SPS turbulence kinetic energy, C_I a constant parameter, δ_{ij} is the Kronecker delta, S_{ij} the components of the SPS strain tensor, and η is a small number introduced to keep the denominator different from zero and is generally taken to be equal to $0.1h$. Besides, we have the surface force per unit area $\vec{F}_s = \kappa \sigma_\alpha \vec{n}$, where σ_α is the surface tension of the α phase, $\vec{n} = \nabla C / |\nabla C|$ is the unit normal to the separation surface κ, and C is the color function

$$C_i^\alpha = \begin{cases} 1 & \text{se } i \in \alpha, \\ 0 & \text{se } i \notin \alpha. \end{cases} \tag{9}$$

The surface tension force, introduced into the momentum equation (7) as an external force [7], can be computed in the form

$$\vec{F}_s = \sum_j m_j \frac{\Pi_i^{\alpha\beta} + \Pi_j^{\alpha\beta}}{\rho_i \rho_j} \frac{\partial W}{\partial r_{ij}}, \tag{10}$$

where [12]

$$\Pi_i^{\alpha\beta} = \sigma^{\alpha\beta} \frac{1}{\left|\nabla C_i^{\alpha\beta}\right|} \left(\frac{1}{d} \left|\nabla C_i^{\alpha\beta}\right|^2 \delta^{\alpha\beta} - \nabla C_i^{\alpha\beta} \otimes \nabla C_i^{\alpha\beta}\right), \tag{11}$$

d is the spatial dimension (1, 2 or 3), $\delta^{\alpha\beta}$ is the Kronecker delta, and

$$\nabla C_i^{\alpha\beta} = \sum_j \frac{m_j}{\rho_j} \left(C_j^\beta - C_i^\beta\right) \frac{\partial W}{\partial r_{ij}}. \tag{12}$$

In the case of two-phase liquid-gas flow, it is necessary to consider the effect of forces at the interface when the two immiscible fluids are in contact. Therefore, we use the surface tension to describe the forces acting at the interface.

4 Resolution of the system of ordinary differential equations

We used Verlet's numerical integration algorithm to solve the system of ordinary differential equations

$$\frac{D\vec{v}_i}{Dt} = \vec{F}_i, \qquad \frac{D\rho_i}{Dt} = R_i, \qquad \frac{D\vec{r}_i}{Dt} = \vec{v}_i, \tag{13}$$

where the first represents the momentum equation, the second the continuity equation, and we use the third to calculate the displacement of fluid particles. Generally used in molecular dynamics [13] due to its low computational cost, when we apply the Verlet scheme in the resolution of quasi-compressible flows, we get

$$\vec{v}_i^{\,n+1} = \vec{v}_i^{\,n-1} + 2\Delta t \vec{F}_i^{\,n}, \quad \vec{r}_i^{\,n+1} = \vec{r}_i^{\,n} + \Delta t \vec{v}_i^{\,n} + \frac{1}{2}\Delta t^2 \vec{F}_i^{\,n}, \quad \rho_i^{n+1} = \rho_i^{n-1} + 2\Delta t R_i^n. \tag{14}$$

Due to the nature of the method, we decouple the calculation of the density and velocity values in $n+1$ time since they do not consider their respective values determined at the instant of time n (using only the information available in $n-1$). This fact can lead to divergence of the numerical method [7]. Thus, we require an intermediate step for every N_s step:

$$\vec{v}_i^{n+1} = \vec{v}_i^n + \Delta t \vec{F}_i^n, \quad \vec{r}_i^{n+1} = r_i^n + \Delta t \vec{v}_i^n + \frac{1}{2}\Delta t^2 \vec{F}_i^n, \quad \rho_i^{n+1} = \rho_i^n + 2\Delta t R_i^n \tag{15}$$

where the superscript $n+1$ indicates the next instant of time $t^{n+1} = t^n + \Delta t$. As we use an explicit method, we must impose restrictions on the value of the time increment to guarantee its stability and convergence. In this work, we determine the Δt according to the proposal by Monaghan [14].

We chose this integration algorithm because it is a low computational cost scheme, second order in space, which does not require multiple steps within an iteration.

5 Numerical results

As already announced in the introductory part of this work, these are the first simulations we are running for the gas lift problem. Therefore, initially, we focus our attention to the definition of the domain, containing a pipeline for the transport of oil, gas injection, and a region destined to phase separation. The idealized test problem does not necessarily correspond to a real case. Nevertheless, we tried to keep the relationships between the pipe diameters and the oil and gas injection flow rates close to the values known in the literature. We reinforce that we are only considering the transportation of oil through pipelines. Furthermore, the separation between the injected gas and the oil takes place in the separation tank.

We intend to verify whether the SPH method can reproduce the phenomena that occur in multiphase flow using the gas lift technique since the literature lacks information on this type of problem with this method.

In Table 1, we present the parameters used in all simulations, and we can find the specific properties of oil and gas in Table 2. The surface tension value was 0.045 N/m. The total number of particles representing the oil was 37,397, while for the gas, we used 12,162. The maximum simulation time was 10 seconds.

Table 1. Parameters for the simulations

Properties	Unit	Value	Properties	Unit	Value
C_I	-	0.066	L_c	m	0.14
C_S	-	0.12	N_s	-	40
g	m/s^2	9.81	Δl	m	0.002

Table 2. Oil and gas properties

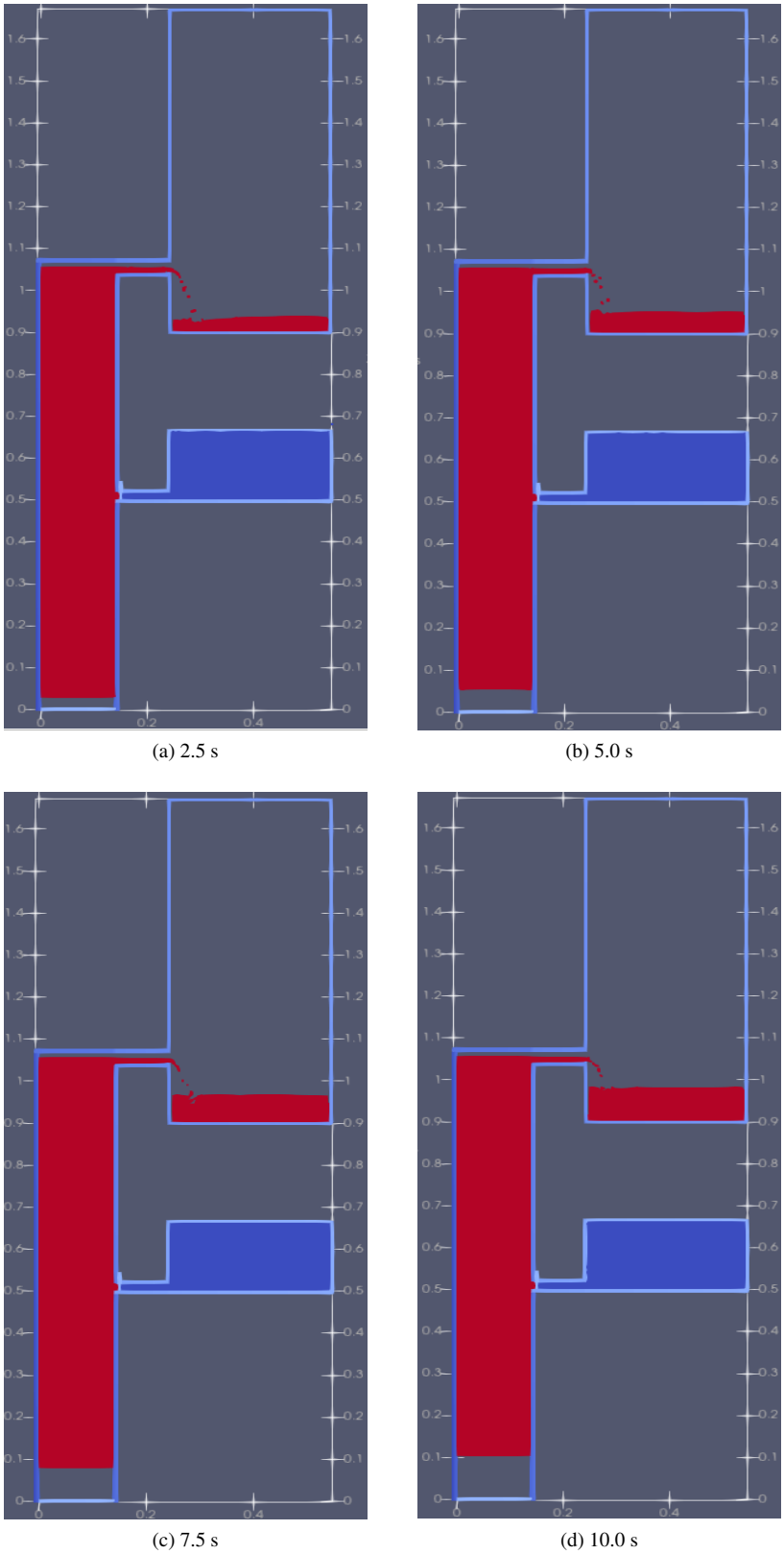
Properties	Unit	Oil	Gas
c_s	m/s	100	343.28
γ	-	7	1.32
μ	N.s/m^2	0.0092	0.000275
ρ_0	kg/m^3	728	0.717

We simulated two test cases: one without gas injection, and a second where we inject gas continuously until the final time. In the present work, both the oil and the gas are set in motion through the displacement of a piston whose speed is specified.

The first simulation dealt solely with oil flow without any gas injection. Our intention is to use this case as a reference to attest to the influence of gas injection. We can see the displacement of the oil particles in Fig. 1 for four pre-fixed time instants and for a piston velocity equal to 0.01 m/s.

On the other hand, in the second case, gas is continuously injected with a piston velocity equal to 0.025 m/s during the entire period of time in which the oil flow occurs. We can also see the flow of oil (red particles) and gas (blue particles) in Fig. 2 for the same instants of time considered previously. We draw attention to the fact that the gas expands inside the separation tank.

From the results obtained, we can verify that the gas lift provided a gas production approximately four times greater than that without injection. We calculated the volume of gas in the separation tank using the known values of the mass and density of the oil particles inside it.

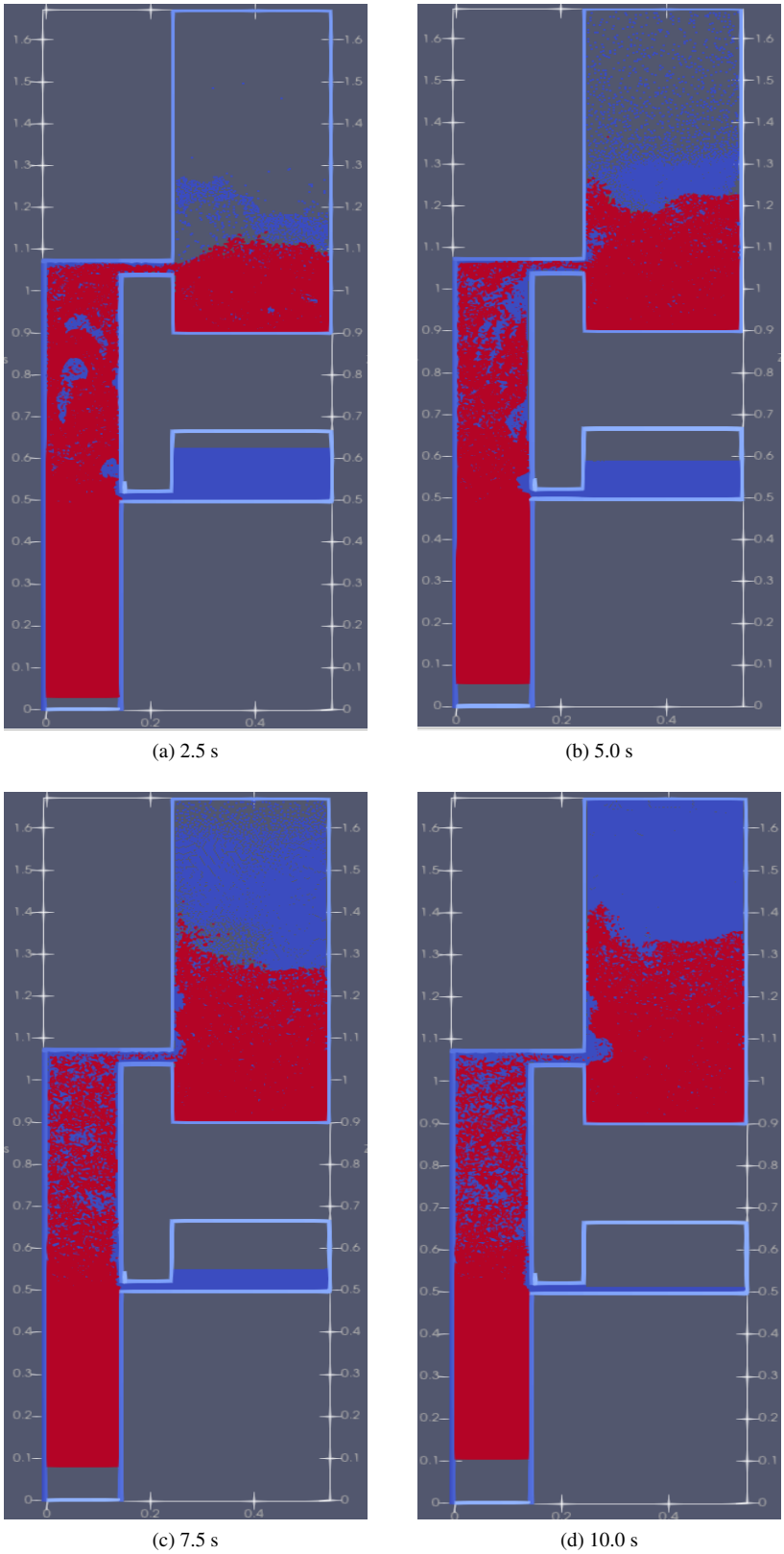


(a) 2.5 s (b) 5.0 s

(c) 7.5 s (d) 10.0 s

Figure 1. Flow of the oil phase without gas injection

We performed all simulations on a K80 graphics card with 2,496 Nvidia CUDA cores and 12 GB of GDDR5 memory. The average simulation time was approximately 13 hours for the first case, and 27 hours for the second.



(a) 2.5 s (b) 5.0 s

(c) 7.5 s (d) 10.0 s

Figure 2. Flow of the oil phase with a continuous injection of gas

6 Conclusions

We tested numerous configurations until we reached the results presented here. It was clear to the authors that we still have to progress to simulate some practical cases. However, as we could observe, the gas injection provided, as expected, a higher volume of gas in the separation tank. We could verify this by comparing the values obtained with and without gas injection. Shortly, we intend to implement inlet and outlet boundary conditions to simulate problems close to real ones without using pistons.

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] R. C. D. do Rosário, G. de Souza, and H. P. Amaral Souto. A comparative study of some well-reservoir coupling models in the numerical simulation of oil reservoirs. *International Journal of Advanced Engineering Research and Science*, vol. 7, n. 9, pp. 126–148. DOI:10.22161/ijaers.79.16, 2020.
[2] J. D. dos Santos Heringer, G. de Souza, H. P. Amaral Souto, and A. P. Pires. On the numerical simulation of non-isothermal heavy oil flow using horizontal wells and horizontal heaters. *Brazilian Journal of Chemical Engineering*. DOI:10.1007/s43153-023-00313-2, 2023.
[3] J. G. Souza Debossam, G. de Souza, H. P. Amaral Souto, and A. P. Pires. Numerical simulation of single-phase two-component non-darcy flow in naturally fractured reservoirs for enhanced gas recovery and carbon dioxide storage. *Brazilian Journal of Chemical Engineering*. DOI:10.1007/s43153-023-00318-x, 2023.
[4] de L. Sousa Santos, K. M. F. de Souza, M. R. Bandeira, V. R. Ruiz Ahón, F. C. Peixoto, and D. M. Prata. Dynamic optimization of a continuous gas lift process using a mesh refining sequential method. *Journal of Petroleum Science and Engineering*, vol. 165, pp. 161–170. DOI:10.1016/j.petrol.2018.02.019, 2018.
[5] N. A. Sami and Z. Turzo. Computational fluid dynamic (CFD) modelling of transient flow in the intermittent gas lift. *Petroleum Research*, vol. 5, n. 2, pp. 144–153. DOI:10.1016/j.ptlrs.2020.03.001, 2020.
[6] G. Liu and M. Liu. *Smoothed Particle Hydrodynamics: A Meshfree Particle Method*. World Scientific Publishing Company, 2003.
[7] A. J. C. Crespo. *DualSPHysics: User wiki*, 2019.
[8] H. Gotoh, T. Shibahara, and T. Sakai. Sub-particle-scale turbulence model for the MPS method. *Computational Fluid Dynamics Journal*, vol. 9, pp. 339–347, 2001.
[9] A. Mokos. *Muti-phase Modelling of Violent Hydrodynamics using Smoothed Particle Hydrodynamics (SPH) on Graphics Processing Units (GPU)*. PhD thesis, School of Mechanical, Aerospace and Civil Engineering, Faculty of Engineering and Physical Sciences, University of Manchester, Manchester, 2013.
[10] H. Wendland. Piecewise polynomial, positive definite and compactly supported radial functions of minimal degree. *Adv Comput Math*, , n. 4, pp. 389–396, 1995.
[11] O. Shoham. *Mechanistic Modeling of Gas-liquid Two-phase Flow in Pipes*. Society of Petroleum Engineers, 2013.
[12] B. Lafaurie, R. Scardovelli, C. Nardone, S. Zaleski, and G. Zanetti. Modelling merging and fragmentation in multiphase flows with SURFER. *Journal of Computational Physics*, vol. 113, n. 1, pp. 134–147, 1994.
[13] L. Verlet. Computer "Experiments" on Classical Fluids - Thermodynamical Properties of Lennard-Jones Molecules. *Physical Review*, vol. 159, pp. 98–103, 1967.
[14] J. J. Monaghan. Simulating Free Surface Flows with SPH. *Journal of Computational Physics*, vol. 110, pp. 399–406, 1994.