XLVI CILAMCE
VITÓRIA - ES - BRAZIL - 2025
XLVI IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGENEERING

Numerical Analysis of the Hydrodynamic Behavior of Monoethylene Glycol (MEG) Injection in Two-Phase Annular Flow

Deisiane S. de Oliveira[1], Nathália R. de Melo[2], Ellen R. M. de Lima[3], Rebeca N. da Silva[4], João A. F. Neto[3], Anderson V. do Nascimento[2], Eduardo S. Neves[1], Jonathan da C. Teixeira[5], Giselle M. L. L. da Silva[6], Tiago C. Freitas[6], Jussara de M. Silva[6], Rosane A. Fontes[6], Monica T. da Silva[6], Alessandro R. E. Antunes[4], Paulo R. M. Lyra[3], Darlan K. E. de Carvalho[3]

[1]*Postgraduate Program in Mechanical Engineering (PPGEM), Federal University of Pernambuco - UFPE*
Arquitetura avenue, Cidade Universitária, Recife, 50740-550, Pernambuco, Brazil
deisiane.dso@ufpe.br, eduardo.neves@ufpe.br
[2]*Postgraduate Program in Civil Engineering (PPGEC), Federal University of Pernambuco - UFPE*
Arquitetura avenue, Cidade Universitária, Recife, 50740-550, Pernambuco, Brazil
nathalia.roseane@ufpe.br, anderson.viananascimento@ufpe.br
[3]*Mechanical Engineering Department (DEMEC), Federal University of Pernambuco - UFPE*
Arquitetura avenue, Cidade Universitária, Recife, 50740-550, Pernambuco, Brazil
ellen.mlima@ufpe.br, joao.aferreira@ufpe.br, paulo.lyra@ufpe.br, darlan.ecarvalho@ufpe.br
[4]*Technology Center, Agreste Academic Center (CAA), Federal University of Pernambuco - UFPE*
Av. Marielle Franco- Km 59, Caruaru, 55014-900, Pernambuco, Brazil
rebeca.nicolle@ufpe.br, alessandro.antunes@ufpe.br
[5]*Technology Center (CTEC), Federal University of Alagoas - UFAL*
Av. Lourival Melo Mota, Tabuleiro do Martins, Maceió, 57072-900, Alagoas, Brazil
jonathan.teixeira@ctec.ufal.br
[6]*Petróleo Brasileiro S.A., Cenpes*
Av. Horácio de Macedo, 950, Ilha do Fundão, Rio de Janeiro, 21941-915, Rio de Janeiro, Brazil
gisellelopes@petrobras.com.br, ticfreitas@petrobras.com.br, jussarams@petrobras.com.br, rafontes@petrobras.com.br, monicats@petrobras.com.br

Abstract. The injection of inhibitors carried in monoethylene glycol (MEG) is a widely used strategy in the oil industry to prevent inorganic scaling in transport systems. This work presents a three-dimensional numerical analysis of the behavior of MEG behavior in two-phase flows, composed of oil and an aqueous phase (formation water + MEG), in oil production pipelines. The methodology evaluated different flow rates, combining one-dimensional simulations in ALFAsim to obtain operational profiles and three-dimensional simulations in ANSYS Fluent for local analysis of MEG dispersion. The 3D numerical model considered a 20-meter section of pipeline characterized by an annular flow pattern. The results showed that MEG disperses primarily in the aqueous phase, with an initially non-uniform distribution that tends toward homogenization along the flow. The flow rates did not significantly affect the order of magnitude of the mass fractions. The methodology proved effective in predicting MEG behavior, contributing to the optimization of injection and scale mitigation strategies, with direct impact on flow assurance in offshore systems.

Keywords: Scaling inhibitors, Monoethylene Glycol (MEG) injection, Oil pipelines, Two-phase flow, 3D Numerical Modeling

1 Introduction

According to Crowe et al. [1] and Silva [2], multiphase flows are characterized by the simultaneous presence of two or more phases (solid, liquid, or gas), with moving and deformable interfaces between them, and with distinct physical properties. A biphasic flow can occur, for example, between immiscible liquids such as oil and water, or between a liquid phase. Conversely, multicomponent flows involve the presence of more than one chemical species dissolved or mixed within the same phase, possibly without a clear interface between components. These flows exhibit a variety of flow patterns, which depend on factors such as flow rates, pipeline geometry,

physical properties of the phases, and operational conditions, according to Shoham [3] and Pacheco et al. [4].

According to Santos [5] and Melo et al. [6], in the oil industry, multiphase and multicomponent flow typically occurs in production lines that transport mixtures of oil, formation water, gas, and impurities. These systems are subject to flow assurance problems involving complex solid deposition phenomena, such as paraffins, asphaltenes, hydrates, and inorganic scale, which compromise the integrity of pipelines and production equipment. A common practice adopted to mitigate these problems is the injection of chemical scaling inhibitors, often carried in monoethylene glycol (MEG - $C_2H_6O_2$), with the aim of keeping the crystals dissolved in the aqueous phase and preventing their deposition, according to Jordan et al. [7] and de Melo [8].

In this study, a numerical analysis was carried out using computational tools based on Computational Fluid Dynamics (CFD) models to investigate the transport of monoethylene glycol in an oil pipeline with biphasic and multicomponent flow. The methodology aimed to predict the behavior of MEG in a biphasic flow composed of formation water and oil, evaluating its effectiveness under different injection rates.

2 Methodology

This research combines coupled 1D and 3D simulations to obtain a detailed understanding of the transport and dispersion of monoethylene glycol (MEG) in a biphasic flow composed of oil and an aqueous phase, consisting of formation water and MEG. The cases analyzed consist of a production line, including a well and riser. The chemical injection mandrel (CIM) is located at a height of 277.7 m from the perforated zone. The well is perfectly vertical and has a length of 3,539 m, while the riser has a vertical projection of 2,015 m and a horizontal projection of 5,000 m. In total, the line has a length of 9,956.8 m. The internal diameter of the pipes is 4.7" (≈ 0.12 m).

2.1 1D simulation

The one-dimensional simulation step was performed using commercial software ALFAsim, version 2024.3.1. ALFAsim employs the multiphase model (Multi-Field Model), which describes flow patterns in which one phase is dispersed in another, such as liquid droplets in a gas phase or gas bubbles in a liquid phase. The formulation supports bimodal or trimodal simulation, assigning two subfields to each phase — one continuous and one dispersed, according to Bonizzi and Issa [9] e Bonizzi et al. [10]. The resulting mass balance equation for each field k is:

$$\frac{\partial(\alpha_k \rho_k)}{\partial t} + \frac{\partial(\alpha_k \rho_k u_k)}{\partial x} = \sum_{s}^{N_{\text{sources}}} \Gamma_{k,s}. \tag{1}$$

where α_k is the volume fraction, ρ_k is the density, u_k is the velocity, and $\Gamma_{k,s}$ is the source term of field k.

The momentum balance equation for each field k is given by:

$$\begin{aligned}\frac{\partial(\alpha_k \rho_k u_k)}{\partial t} + \frac{\partial(C_k \alpha_k \rho_k u_k u_k)}{\partial x} = &-\alpha_k \frac{\partial p_k}{\partial x} + \alpha_k \rho_k g \sin\theta \\ &+ (p_{ki} - p_k)\frac{\partial \alpha_k}{\partial x} + \frac{\tau_{wk} S_{wk}}{A} + \frac{\tau_i S_i}{A} \quad \pm F_k^{vm} + F_k^{drag} + \sum_{s}^{N_{\text{sources}}} \Gamma_{k,s}\, \bar{u}_{k,s}.\end{aligned} \tag{2}$$

where θ is the pipe inclination, A is the cross-section area, τ_{wk} is the wall shear stress, τ_i is the interfacial shear stress, S_{wk} is the wall wet perimeter, S_i is the interface wet perimeter, C_k^u is the distribution parameter of phase k (set to 1 by default), $\bar{u}_{k,s}$ is the reference velocity that accounts for the momentum transported by the mass source $\Gamma_{k,s}$, F_k^{vm} represents volumetric forces (such as Reynolds stress), and F_k^{drag} is the drag force.

In this work, the (global) mixture energy equation was used for the total energy balance: The total (global) mixture energy equation is:

$$\frac{\partial}{\partial t}\sum_k \left[\alpha_k \rho_k \left(h_k + \frac{u_k^2}{2}\right)\right] + \frac{\partial}{\partial x}\sum_k \left[\alpha_k \rho_k u_k \left(h_k + \frac{u_k^2}{2}\right)\right] = \frac{\partial p}{\partial t} + \sum_k \alpha_k \rho_k u_k g \sin\theta + \dot{Q}_{\text{source}} + \dot{Q}_{\text{wall}}. \tag{3}$$

with h_k is the total enthalpy of phase k, $\dot{Q}_{\text{source}}$ is the source term, and $\dot{Q}_{\text{wall}}$ is the external heat flow term.

The simulation was performed considering the entire production pipeline (9,956.8 m), assuming a uniform pipe roughness of 4.57 × 10 m. A production scenario was considered with an oil flow rate of 1,146.6 sm³/d, a water flow rate of 770.33 sm³/d, and a gas–oil ratio at the bubble point of 243. Under standard conditions, the oil density was taken as 891.67 kg/m³ and the water density as 1001 kg/m³. The simulation was carried out in steady state, as the objective was solely to provide boundary and initial condition data for a subsequent three-dimensional simulation of MEG flow.

2.2 Description of the 3D scenario

Three-dimensional simulations were conducted to characterize the multiphase and multicomponent flow resulting from the injection of monoethylene glycol (MEG) (Fig. 1), used as a solvent for inorganic scale inhibitors, in a two-phase annular flow pattern. The oil phase flows in the central region, while the aqueous phase (formation water + MEG) occupies the region near the wall. The simulated domain corresponds to a 20 m section of a steel pipe with an internal diameter of 0.12 m.

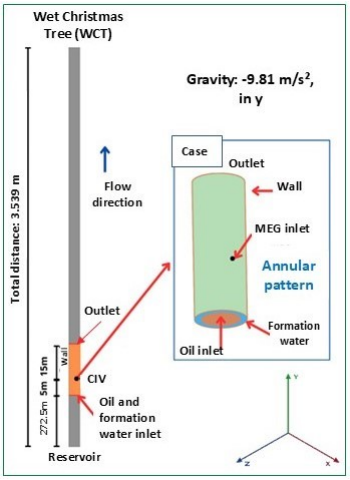


Figure 1. Schematic profile, not to scale, of the section of the pipeline analyzed.

The Figure 1 shows the schematic profile of the three-dimensional section of the pipeline, with turbulent flow in an annular pattern (BSW = 36%) and MEG injection through a chemical injection valve (CIV), represented by a circular inlet with a diameter of $2.02x10^{-2}$ m, located at y = 5 m. The valve body was disregarded. MEG injection was simulated for 3.056 h.

The system contains three components: oil (oil phase), MEG with formation water (aqueous phase), with surface tension σ_{aq} = 0.067 N/m, accordingly to Silva et al. [11]. The Reynolds numbers were obtained according to Cazarez-Candia and Piedra-González [12], with $Re_{\text{w}} \approx 163,638$ and $Re_{\text{o}} \approx 22,367$. In this study, three flow conditions were considered: one nominal (constant, provided by the manufacturer) and two real, measured in the field for valves 1 and 2. The respective injection velocities were applied as boundary conditions at the MEG inlet (Fig. 2).

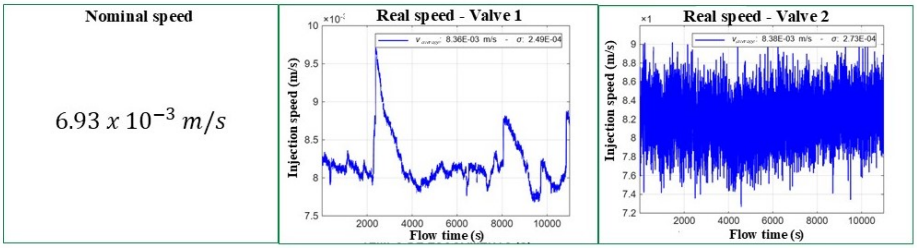


Figure 2. Nominal injection speed of MEG and speed profiles of valve 1 and valve 2.

The boundary conditions adopted in the simulation were defined as follows:

- **Oil inlet**: velocity of 2.445 m/s, temperature of 352.341 K, secondary phase volume fraction equal to 1, MEG mass fraction equal to 0, turbulence intensity of 5% and turbulent viscosity rate of 10;
- **Formation water inlet**: velocity of 2.445 m/s temperature of 352.341 K, secondary phase volume fraction equal to 0, MEG mass fraction equal to 0, turbulence intensity of 5% and turbulent viscosity rate of 10;
- **MEG inlet**: nominal velocity and transient profiles according to Fig. 2, from valves 1 and 2; temperature of 353.15 K, secondary phase volume fraction equal to 0, MEG mass fraction equal to 1, turbulence intensity of 5% and turbulent viscosity rate of 10;
- **Wall**: no-slip condition, with stationary walls, temperature of 352.341 K and zero diffusive flow for the chemical species;
- **Outlet**: gauge pressure of 38.93 MPa, temperature of 352.341 K, secondary phase volume fraction equal to 0, and MEG backflow mass fraction equal to 0.

The reference pressure used was 38.93 MPa, at coordinates x = 0.06 m, y = 20 m, and z = 0 m. Gravitational acceleration g = 9.81 m/s^2 was considered, acting in the y direction. The initial conditions included: gauge pressure equal to 38.93 MPa, velocities equal to 0 m/s on the x, y, and z axes, temperature of 352.341 K, mass fraction of monoethylene glycol equal to 10^{-6}, and volume fraction of the secondary phase equal to 10^{-6} at the outlet of the duct.

2.3 Mathematical models used in 3D simulation

The 3D simulations were carried out using Ansys Fluent® 2024 R1, considering a multiphase, multicomponent, incompressible, and non-isothermal flow. To capture the interface between immiscible phases, the Volume of Fluid (VOF) model was adopted. The distribution of monoethylene glycol (MEG) was modeled using the species transport model, without chemical reactions, and the effects of turbulence were addressed using the standard k–ε model with enhanced wall treatment.

The Volume of Fluid (VOF) multiphase model is an Eulerian approach developed by Hirt and Nichols [13], to solve free-surface problems, which correspond to the interfaces between two fluids or between fluids and solids, considering such interfaces deformable under internal or external forces. According to ANSYS [14], in this model, the phases are treated as interpenetrating continua; surface tension effects at the interface between each pair of phases can be computed, and, letting α_{q} be the volume fraction of the qth phase, with $0 \leq \alpha_{\mathrm{q}} \leq 1$, the scalar conservation equation for the volume fraction of the qth phase is given by:

$$\frac{1}{\rho}\left[\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) = S_{\alpha_q} + \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp})\right]. \tag{4}$$

where S_{α_q} is the mass source term, $\dot{m}_{pq}$ is the mass transfer from phase p to phase q, and $\dot{m}_{qp}$ is the mass transfer from phase q to phase p. The variables $\vec{v}_q$ and ρ_q represent the velocity and density of phase q, respectively.

In VOF, only one momentum equation and one energy equation are solved for all fluids. Thus, for linear momentum balance, we have the vector equation:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot \left[\mu\left(\nabla\vec{v} + \nabla\vec{v}^T\right) - \frac{2}{3}\nabla \cdot \vec{v}I\right] + \rho\vec{g} + \vec{F}. \tag{5}$$

where ρ is the density, $\vec{v}$ is the velocity, p is the pressure, μ is the viscosity, T is the temperature, $\vec{g}$ is the acceleration due to gravity and $\vec{F}$ s the volumetric force.

The (scalar) energy equation is given by:

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot \left(k_{\text{eff}}\nabla T - \sum_{q}\sum_{j} h_{j,q}\vec{j}_{j,q} + (\bar{\tau}_{\text{eff}}\vec{v})\right) + S_h. \tag{6}$$

where S_h is the heat source term, $\vec{J}_{j,q}$ represents the diffusive mass flow of species j in phase q, and $\bar{\tau}_{\text{ef}}$ is the effective viscous stress tensor of the fluid. The variable E corresponds to the total specific energy, determined by the weighted average of the masses.

According to ANSYS [14], the species transport model considers the multicomponent transport of chemical species through convection, molecular diffusion, mass sources, and chemical reactions. In this context, the mass conservation equation governs the transport of the mass fraction of the $N-1$ species i in the q_{th} phase, expressed by:

$$\frac{\partial}{\partial t}\left(\rho_q \alpha_q Y_{i_q}\right) + \nabla \cdot \left(\rho_q \alpha_q \vec{v}_q Y_{i_q}\right) = -\nabla \cdot \alpha_q \vec{J}_{i_q} + \alpha_q R_{i_q} + \alpha_q S_{i_q} + \sum_{p=1}^{n} \dot{m}_{p^i q^j} + \mathcal{R}. \tag{7}$$

where $Y_{i,q}$ represents the mass fraction of species i in phase q, while $R_{i,q}$ corresponds to the source term associated with the chemical reactions involving species i in phase q. The term $\vec{J}_{i,q}$ refers to the diffusive mass flow of species i in phase q, resulting from the concentration gradients present in the medium. $S_{i,q}$ represents an additional source term, for external effects applied directly to species i in phase q.The equation can also include the term $\mathcal{R}$, known as the residual term, wish is used in iterative numerical methods to control convergence and correct mass balances during the solution process.

Among the turbulence models available, this study opted to use the standard κ-ε model developed by Launder and Spalding [15]. Within the standard κ-ε model, we used the improved wall treatment, which combines the two-layer model developed by Wolfshtein [16] and improved wall functions by Jongen [17].

2.4 Numerical strategies used in the 3D simulation

The geometry of the 20 m long section of the pipeline was discretized in a computational domain with an unstructured mesh composed of 3,037,062 finite volumes. In the numerical simulations, a pressure-based coupled solver was used, was used, whose algorithm performs simultaneous coupling between the pressure and velocity fields, according to ANSYS [14]. To calculate the gradients of the variables within the computational domain, the Least Squares Cell-Based method was used, according to ANSYS [14]. The spatial discretization of convective terms in the transport equations was performed using the Second Order Upwind (SOU) scheme, by Hirsch and Nichols [18]. For pressure interpolation, the PRESTO (Pressure Staggering Option) scheme by Barth and Reddy [19] was adopted, while the transient terms were discretized using the Bounded Second Order Implicit scheme, according to ANSYS [14]. As a convergence criterion, a maximum number of 20 iterations per time step and an absolute residual error of less than 10^{-3} were considered for the equations of mass conservation, linear momentum balance, turbulent kinetic energy and turbulent kinetic energy dissipation rate, volumetric fraction of the secondary phase, and mass fraction of MEG.

3 Results and discussions

This section presents the results of the numerical simulations performed for the scenario described in subsection 2.2, in which the cases were defined based on different monoethylene glycol (MEG) injection flow rates in the pipeline. The results correspond to the instant t = 3.056 h. The Figure 3 shows the contour maps of the volumetric fraction of the primary phase for the nominal flow rate at different pipeline elevations, as well as the average volumetric fraction profiles of the aqueous phase for the three cases analyzed. From Figure 3, it can be seen that the aqueous phase followed a typical and well-established annular behavior in the flow, with the oil phase occupying most of the cross section. The same behavior was obtained in the simulations performed with the actual injection velocity of valves 1 and 2.

The figure 4 shows the color maps of MEG mass fraction for the nominal velocity at different pipeline elevations and the mass fraction profiles of monoethylene glycol (MEG) for the three cases analyzed. It was observed that, in all injection flows analyzed, MEG diffused along with the aqueous phase, as predicted by the modeling, but in a non-uniform manner, with a higher concentration on the side of the pipeline corresponding to the injection region by the chemical injection valve (CIV). With upward flow, the distribution tends to become uniform. The maximum, average, and minimum mass fractions of MEG remain within the same order of magnitude across the cases, although some local variations are observed. The lowest values occur in regions with a low volume fraction of the aqueous phase or where MEG dispersion is limited by the presence of the oil core. As dispersion becomes more uniform along the pipeline, with a decrease in the maximum mass fraction values and an increase in the minimum values are observed.

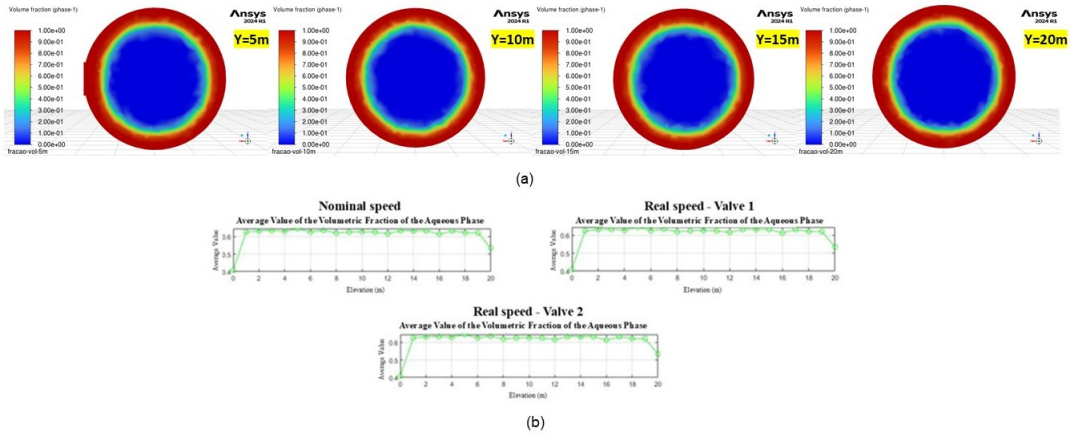


Figure 3. (a) Primary phase volume fraction – nominal velocity: 6.93×10^{-3} m/s; and (b) Average profiles of aqueous phase volume fraction for the three cases.

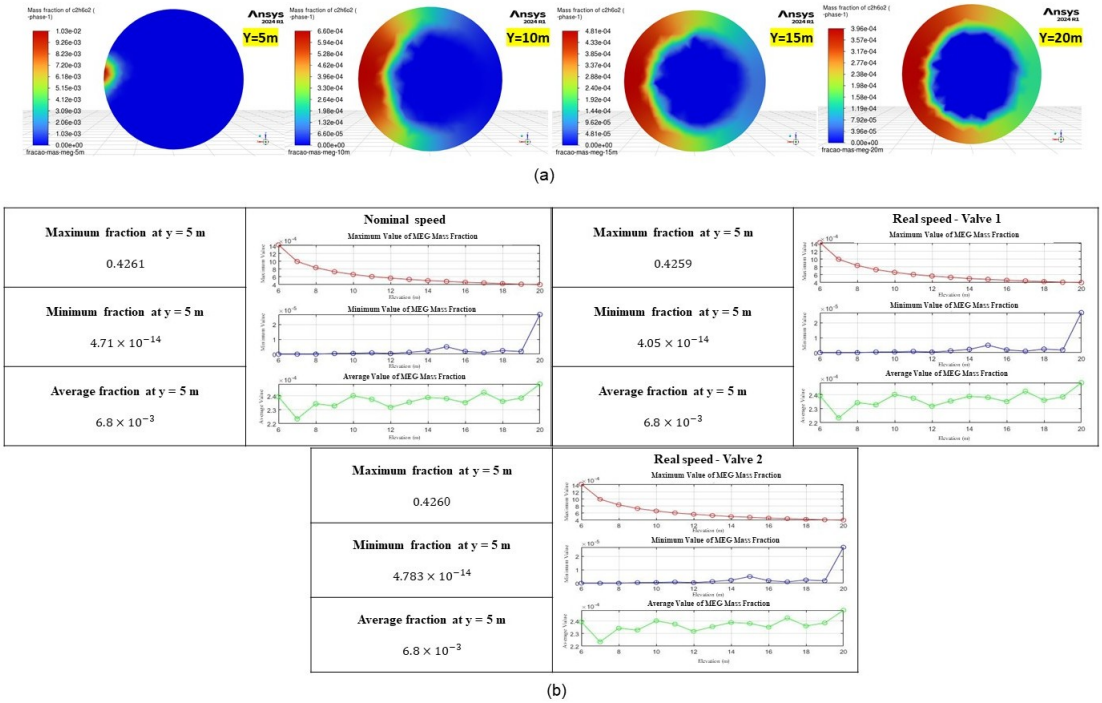


Figure 4. (a) MEG mass fraction – nominal velocity: 6.93×10^{-3} m/s; and (b) MEG mass fraction profiles for the three cases.

4 Conclusions

The study demonstrated that 3D numerical modeling is an effective tool for understanding the behavior of MEG in biphasic flows typical of oil pipelines. It was observed that the distribution of the substance is initially asymmetrical, but tends to homogenize along the flow, regardless of the flow rate used. As a continuation of this work, the simulated pipeline section will be extended, a mesh convergence study will be conducted, and dynamic mesh adaptation techniques will be adopted to improve the accuracy of MEG transport prediction and enhance injection strategies for scale prevention.

Acknowledgements. The authors would like to thank Petrobras, FACEPE, CAPES, CNPQ, UFPE, and i-Litpeg for their financial support in carrying out this research.

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] C. T. Crowe, J. D. Schwarzkopf, M. Sommerfeld, and Y. Tsuji. *Multiphase Flows with Droplets and Particles*. CRC Press, Boca Raton, 2 edition, 2012.
[2] R. G. Silva. *Estudo numérico de movimentação de partículas em escoamento*. Dissertação de mestrado em engenharia mecânica, Universidade de São Paulo, 2006.
[3] O. Shoham. *Mechanistic Modeling of Gas-Liquid Two-Phase Flow in Pipes*. Cambridge University Press, Cambridge, 2006.
[4] F. Pacheco, C. C. P. Cavalcante, J. R. P. Mendes, A. C. Bannwart, and de A. B. S. Serapião. Identificação de padrões de escoamento multifásico e determinação da perda de carga com o auxílio de redes neurais. In *Anais do 4º Congresso Brasileiro de Petróleo e Gás – PDPETRO*, Campinas. ABPG, 2007.
[5] M. J. R. Santos. Detecção de problemas de garantia de escoamento a partir da utilização de ferramentas de aprendizado de máquina, 2020.
[6] N. R. d. Melo, J. A. Ferreira Neto, D. S. d. Oliveira, J. d. C. Teixeira, G. M. L. L. d. Silva, P. R. M. Lyra, A. R. E. Antunes, and D. K. E. d. Carvalho. Analysis of the transport of monoethylene glycol inside oil pipelines using numerical simulation and deep learning modeling. *Proceedings of the XLV Ibero-Latin-American Congress on Computational Methods in Engineering, ABMEC (CILAMCE-2024)*, vol. , 2024.
[7] M. Jordan, N. Feasey, and C. Johnston. Inorganic scale control within meg/methanol treated produced fluids. In *SPE International Symposium on Oilfield Scale*, Aberdeen, 2005.
[8] de N. R. Melo. Simulação numérica do transporte de monoetilenoglicol em dutos do sistema de produção de petróleo, 2022.
[9] M. Bonizzi and R. I. Issa. A model for simulating gas bubble entrainment in two-phase horizontal slug flow. *International Journal of Multiphase Flow*, vol. 29, 2003.
[10] M. Bonizzi, P. Andreussi, and S. Banerjee. Flow regime independent, high resolution multi-field modelling of near-horizontal gas–liquid flows in pipelines. *International Journal of Multiphase Flow*, vol. 35, 2009.
[11] F. N. Silva, T. H. F. Andrade, A. G. B. Lima, and S. R. F. Neto. Estudo numérico do escoamento trifásico (água-óleo pesado-gás) tipo core-flow em uma conexão "t". In *Anais do 6º Congresso Brasileiro de Pesquisa e Desenvolvimento em Petróleo e Gás – PDPETRO*, Florianópolis. ABPG, 2011.
[12] O. Cazarez-Candia and S. Piedra-González. Modeling of heavy oil-water core-annular upward flow in vertical pipes using the two-fluid model. *Journal of Petroleum Science and Engineering*, vol. 150, pp. 146–153, 2017.
[13] C. W. Hirt and B. D. Nichols. Volume of fluid (vof) method for the dynamics of free boundaries. *Journal of Computational Physics*, vol. 39, pp. 201–225, 1981.
[14] ANSYS. *Ansys Fluent Theory Guide, Version 2024 R1*. ANSYS, Inc., 2024.
[15] B. E. Launder and D. B. Spalding. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, vol. 3, pp. 269–289, 1974.
[16] M. Wolfshtein. The velocity and temperature distribution of one-dimensional flow with turbulence augmentation and pressure gradient. *International Journal of Heat and Mass Transfer*, vol. 12, pp. 301–318, 1969.
[17] T. Jongen. *Simulation and Modeling of Turbulent Incompressible Flows*. PhD thesis, EPF Lausanne, Lausanne, Switzerland, 1992.
[18] C. W. Hirsch and B. D. Nichols. *Numerical Computation of Internal and External Flows*. Elsevier, Great Britain, 2 edition, 2007.
[19] T. J. Barth and J. P. P. Reddy. The PRESTO! method for pressure staggering in finite volume simulations. *Computers & Fluids*, vol. 22, pp. 451–478, 1993.