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

Numerical Simulation of Non-Isothermal Two-Phase Flows in Vertical Pipelines with the Two-Fluid Five-Equations Model Using a 2nd Order MUSCL-Type Scheme

Eduardo O. Reis[1], Anderson V. do Nascimento[2], Giselle M. L. L. da Silva[3], Tiago C. Freitas[4], Rosane A. Fontes[5], Jussara de M. Silva[6], Monica T. da Silva[7], Alessandro R. E. Antunes[8], Darlan K. E. de Carvalho[9], Paulo R. M. Lyra[10]

[1]Programa de Pós-Graduação em Engenharia mecânica (PPGEM), CTG – UFPE, Av. Acadêmico Hélio Ramos, s/n, CEP, Recife, PE 50740-530 Brazil, eduardo.oreis@ufpe.br
[2]Programa de Pós-Graduação em Engenharia Civil (PPGEC), CTG – UFPE, Av. Acadêmico Hélio Ramos, s/n, CEP, Recife, PE 50740-530 Brazil, anderson.viananascimento@ufpe.br
[3,4,5,6,7] Petróleo Brasileiro S.A. (PETROBRAS) - Petróleo Brasileiro - Rio de Janeiro - Matriz, Petróleo Brasileiro – Rio de Janeiro. Avenida Melício Machado, s/n km 1 - Atalaia 49037445 - Aracaju, SE, Brasil, gisellelopes@petrobras.com.br, ticfreitas@petrobras.com.br, rafontes@petrobras.com.br, jussarams@petrobras.com.br, monicats@petrobras.com.br
[8]Núcleo de Tecnologia, CAA – UFPE, Rod. BR-104 – Km 59 – Nova Caruaru CEP, Caruaru, PE 55002-970, Brazil, alessandro.antunes@ufpe.br
[9,10]Departamento de Engenharia Mecânica, CTG – UFPE, Av. Acadêmico Hélio Ramos, s/n, CEP, Recife, PE 50740-530 Brazil, darlan.ecarvalho@ufpe.br, paulo.lyra@ufpe.br

Abstract. In the oil industry, problems frequently arise that depend on the understanding of multiphase flow in pipelines, such as in flow assurance management. To determine flow parameters such as velocities, pressures, and temperatures, numerical simulations of multiphase flow within wells and oil risers can be employed. Due to the great length of these pipes, 2D or 3D models are often computationally prohibitive, and one-dimensional models are preferred, such as the drift-flux model and the two-fluid model. The latter has the advantage of solving a momentum equation for each phase. The non-isothermal two-fluid five-equation model is a version of these models that includes a global energy equation, allowing the prediction of a shared temperature. However, this model is typically solved using low-order numerical schemes, as commonly reported in literature and in commercial code technical manuals. Therefore, the objective of this work is to numerically model and solve the two-fluid five-equation model, which also includes heat transfer effects, using high-order numerical schemes, such as the second-order MUSCL (Monotonic Upstream-centered Scheme for Conservation Laws) method for spatial discretization and the Backward Euler method for time integration. For the pressure-velocity coupling, a SIMPLE-like algorithm (Semi-Implicit Method for Pressure Linked Equations) is used, and the implementation is done using the Julia programming language. A vertical pipe is considered with core-annular flow, formed by either a liquid and a gas or two immiscible liquids. These models include heat transfer through the pipe wall and account for frictional effects. Our results were very promising and were compared with those from the commercial 1D simulator (ALFAsim).

Keywords: Two-Fluid Five Equations Model, 1-D Non-Isothermal Flows, High-Order Schemes, MUSCL-Type Scheme.

1 Introduction

Engineering plays a crucial role in optimizing costs and developing technologies to improve the operational efficiency of oil and gas exploration, production, refining, and decommissioning in Brazil [1]. As a result of the

discovery of the pre-salt layer, there has been a major advance in deepwater oil exploration, representing an important milestone for the Brazilian oil industry [2]. With the discoveries in the Santos Basin, Brazil has established itself as one of the leaders in offshore oil production [2].

Deepwater oil and gas exploration poses major challenges, such as high costs resulting from challenging physical conditions and the great lengths of pipelines, as well as hydrocarbon separation, treatment, and disposal [3]. Among these challenges, the physical conditions and long pipelines can cause significant production problems, including the formation and deposition of scale, paraffins, asphaltenes, and other solids, which can obstruct pipelines and/or cause pressure and temperature drop in the production line [4]. Under these conditions, understanding the different flow patterns is crucial, as they depend on pipeline inclination, phase velocities, and other parameters [5]. In annular flow, a detailed assessment of these conditions is essential to mitigate such problems, since water is in direct contact with the pipeline wall, which facilitates scale formation [5].

Efficient flow management is essential, but it requires knowledge of various factors, such as flow regimes, flow patterns, and the physical conditions present in the pipeline. This knowledge is fundamental for identifying areas prone to the formation of compounds that accumulate on internal pipe surfaces. Computational Fluid Dynamics (CFD) is a viable tool for determining these physical conditions [4].

Numerical modeling of deepwater oil and gas production lines is computationally expensive due to the complexity of multiphase and turbulent flow over long distances [6]. To reduce computational costs, it is necessary to use simplifications and efficient numerical methods. One such simplification is the adoption of a one-dimensional model to two-phase flow, which is widely applied by specialized software such as ALFAsim (ESSS, n.d.) [7].

Various models have been developed to describe two-phase flows, such as the drift-flux model and the two-fluid model [4]. In this work, we adopted the two-fluid five-equation model, which consists of a system of partial differential equations representing the conservation of mass, linear momentum, and energy [4]. Specifically, the model comprises five coupled equations: two for mass conservation (one per phase), two for linear momentum conservation (one per phase), and one for energy conservation assuming a shared temperature [4, 8, 9]. A staggered grid was used to discretize the problem using the finite volume method, coupled to the CATHARE model framework [10]. The high-order MUSCL (Monotonic Upstream-Centered Scheme for Conservation Laws) scheme was adopted to minimize the artificial numerical diffusion introduced by the classical upwind discretization commonly adopted [11]. Sophisticated high-order methods, such as Flux Reconstruction (FR), are used to obtain better accuracy in solving partial differential equations, alongside mesh adaptation strategies, such as h-p refinement techniques [12, 13].

Therefore, in the present work we study and implement, for the first time in literature, the 2nd order MUSCL scheme in the context of the Finite Volume method and a SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm to numerically discretize the Two-Fluid Five-Equation 1D model with heat-transfer effects applied to study the 1D vertical annular flow pattern in pipes. Our analysis was carried out for one-dimensional (1D) flow without mass transfer between phases in deepwater oil production lines, in order to support studies on incrustation formation tracking. This is done through an in-house simulator developed using the Julia programming language [13]. To verify the results, we made a comparison using an oil-water case with the results obtained from the commercial software ALFAsim.

2 Mathematical formulation

The Two-Fluid model is a formulation used to solve the conservation equations separately, treating each phase as an interpenetrating continuum. The Two-Fluid Five-Equation Model consists of two mass conservation equations, two linear momentum conservation equations, and one energy equation. In this article, the conservation equations have been adapted considering one-dimensional, compressible, no mass transfer, and inviscid flow. The mass conservation equation for $k = 1, 2$ is written as [15]:

$$\frac{\partial(\alpha_k \rho_k)}{\partial t} + \frac{\partial(\alpha_k \rho_k v_k)}{\partial x} = 0, \tag{1}$$

where α_k, ρ_k and v_k are the volume fractions, densities, and velocities of phase k, respectively. In this context, the linear momentum equation for each phase k can be written as follows [15]:

$$\frac{\partial \alpha_k \rho_k v_k}{\partial t} + \frac{\partial \alpha_k \rho_k v_k^2}{\partial x} = -\alpha_k \frac{\partial p_k}{\partial x} - \left(p_k - p_k^i\right)\frac{\partial \alpha_k}{\partial x} + \alpha_k \rho_k g sin\theta + \frac{\tau_{k,i} S_i}{A_k} + \frac{4\tau_{k,w}}{D\alpha_k}, \tag{2}$$

where p_k is the pressure of phase k; g is the acceleration due to gravity; θ is the duct inclination angle; $\tau_{k,i}$ is the interfacial friction for phase k; S_i is the interface perimeter; A_k is the cross-sectional area of phase k; D is the duct diameter; and $\tau_{k,w}$ is the wall friction for phase k. The term $\left(p_1 - p_1^i\right)$ is the CATHARE model term used to introduce an artificial diffusion into the problem [9], acting as an additional interfacial pressure difference term, which is given as:

$$\left(p_k - p_k^i\right) = \Delta p = \gamma \frac{\alpha_1 \alpha_2 \rho_1 \rho_2}{\alpha_1 \rho_2 + \alpha_2 \rho_1} (v_1 - v_2)^2. \tag{3}$$

Finally, the energy equation is expressed as the sum of the two conservation equations, as follows [7]:

$$\frac{\partial \left(\alpha_1 \rho_1 \left(h_1 + \frac{v_1^2}{2}\right) + \alpha_2 \rho_2 \left(h_2 + \frac{v_2^2}{2}\right)\right)}{\partial t} + \frac{\partial (\alpha_1 \rho_1 v_1 \left(h_1 + \frac{v_1^2}{2}\right) + \alpha_2 \rho_2 v_2 \left(h_2 + \frac{v_2^2}{2}\right)}{\partial x} = \frac{\partial p_k}{\partial t} + (\alpha_1 \rho_1 v_1) g sin\theta + (\alpha_2 \rho_2 v_2) g sin\theta + \frac{\overset{"}{q}_{W,m} S}{A}, \tag{4}$$

where h_1 and h_2 are the enthalpies of phases 1 and 2; $\overset{"}{q}_{W,m}$ is the average heat flux into the duct; and S is the perimeter wetted by phase 2, which is in direct contact with the duct.

So far, we have seven primary variables to be determined in the solution: α_1, α_2, p, v_1, v_2, h_1, and h_2. However, only five partial differential equations are available to solve for them. Therefore, two additional closure equations are required to close the system. The closure relations adopted are the saturation restriction and the assumption of thermal equilibrium between the phases, as expressed by eq. (5), which allows the use of specific enthalpy as a thermodynamic variable, leading to eq. (6).

$$\sum \alpha_k = 1, \tag{5}$$

$$h_1 = \frac{cp_1}{cp_2} h_2, \tag{6}$$

where cp is the specific heat.

The equations presented so far cover all immiscible flow patterns without mass transfer and with different inclinations [7, 15]. For the different flow patterns, variations occur only in the empirical terms: in the linear momentum conservation equation, $\tau_{k,i}$ and $\tau_{k,w}$ change, whereas in the energy conservation equation, the variation occurs in the term $\overset{"}{q}_{w,m}$ [7, 8, 15].

3 Numerical Formulation

A hybrid approach was adopted, combining the locally conservative finite volume method (FVM) to discretize the governing equations of the model [16]. For the pressure equation, the discretization was based on the classical Control Volume Finite Difference method a.k.a. the Two-Point Flux Approximation (TPFA) [17, 18]. We used the staggered grid shown in Fig. 1, which is designed to avoid odd-even decoupling [15]. In this scheme, the velocities are computed using finite volumes on the dual mesh (in black), while the scalar variables (volume fraction, density, pressure, and enthalpy) are computed using finite volumes on the primal mesh (in red).

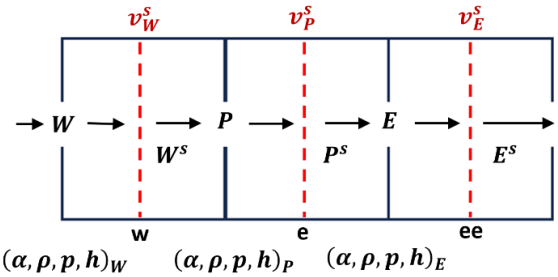


Figure 1. Staggered grid configuration used for the numerical discretization (adapted from Wang et al. [10]).

In Fig. 1, $W, P,$ and E correspond to the center points of the primary mesh, while W^S, P^S and E^S correspond to the center points of the dual mesh. Finally, w, e, and ee represent the control surfaces of the primary mesh.

The temporal terms were discretized using the Backward Euler method, while the numerical fluxes were evaluated at the interfaces between cells [19]. For the MUSCL scheme, the interfacial value ϕ_e of the advected variable ϕ at face e is given by [20]:

$$\phi_e = \begin{cases} \phi_P + \dfrac{\Delta x}{2}\psi(r)\left(\dfrac{\partial \phi}{\partial x}\right)_P, \text{if } u_e > 0 \\ \phi_E - \dfrac{\Delta x}{2}\psi(r)\left(\dfrac{\partial \phi}{\partial x}\right)_E, \text{Otherwise} \end{cases}, \tag{7}$$

$$r = \frac{\phi_P - \phi_W}{\phi_E - \phi_P}, \tag{8}$$

where ϕ is the advected variable can be the velocity v_k, the volume fraction α_k, or the enthalpy h_k. $\psi(r)$ is the limiter function, and r is the gradient ratio. In this work, the Minmod limiter was employed to achieve higher-order accuracy without generating unphysical oscillations near discontinuities, and it is defined as [19]:

$$\psi(r) = max\ (0, min(1, r). \tag{9}$$

We employ a SIMPLE-like algorithm for the coupling between the pressure and velocity fields [21]. The overall structure of the implemented code and the solution sequence of the problem variables are illustrated in Fig. 2.

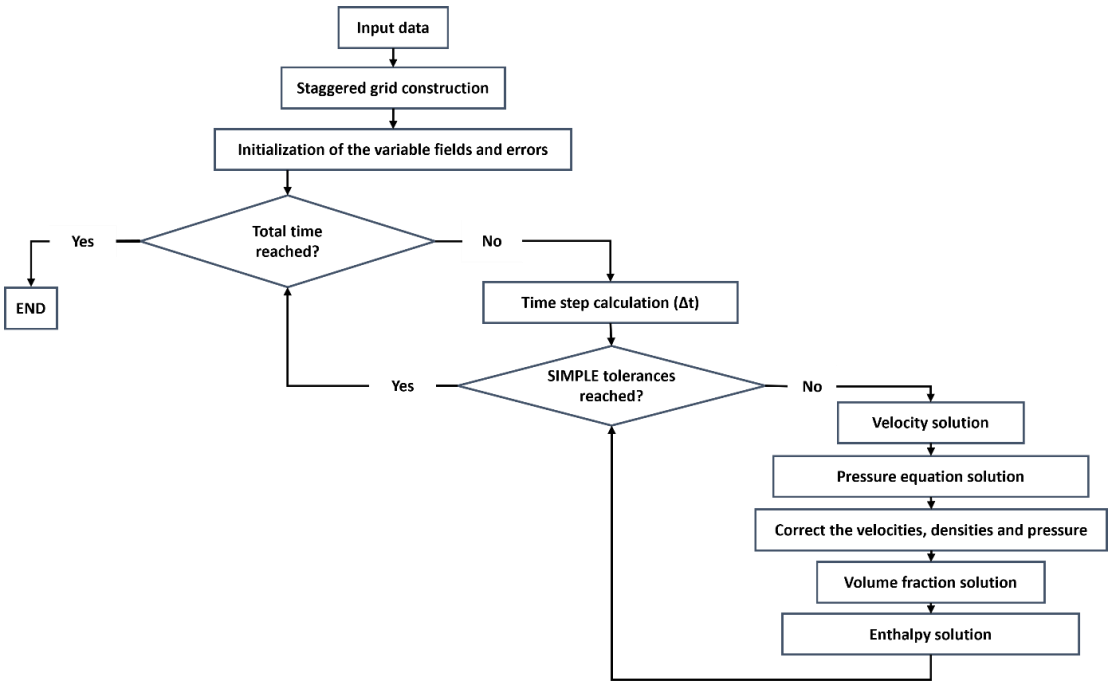


Figure 2. Algorithm flowchart of the in-house simulator, indicating the main steps of the numerical procedure.

4 Results

To verify our in-house code, we devised a test case and applied both the First-Order Upwind (FOU) method and the high-resolution MUSCL scheme, comparing the results with ALFAsim [7], considering a vertical downward annular flow, where the phases are oil (core) and water (annulus), in a pipeline 100 meters long, with an internal diameter of $D_{int} = 0.1$ m and an external diameter of $D_{ex} = 0.14$ m. In this problem, we account for shear stress at the interface between the fluids and between the fluid in the annulus and the pipe wall. Additionally, heat transfer is considered between the inner and outer surfaces of the pipe, with an external convective heat transfer coefficient of $h_{conv} = 500$ W/m^{2}K as illustrated in Fig. 3.

The two phases are injected into the upper part of the pipeline with the following parameters: phase volume fractions $\alpha_o = \alpha_w = 0.5$; phase velocities $v_o = v_w = 3$ m/s; densities $\rho_o = 819.867$ kg/m^3 and $\rho_w = 988.4935$ kg/m^3 and viscosities $\nu_o = 2.58$ mPa.s and $\nu_w = 0.547$ mPa.s for oil and water, respectively. In addition, the initial temperature is $T_i = 50$ °C throughout the pipeline. A temperature boundary condition of $T_{inp} = 60$ °C is imposed at the inlet, with an external temperature of $T_\infty = 60$ °C along the pipe, and a pressure boundary condition of $P_{out} = 2.0$ MPa at the outlet. The previously described parameters were used in both the in-house model and the reference simulation using the ALFAsim software.

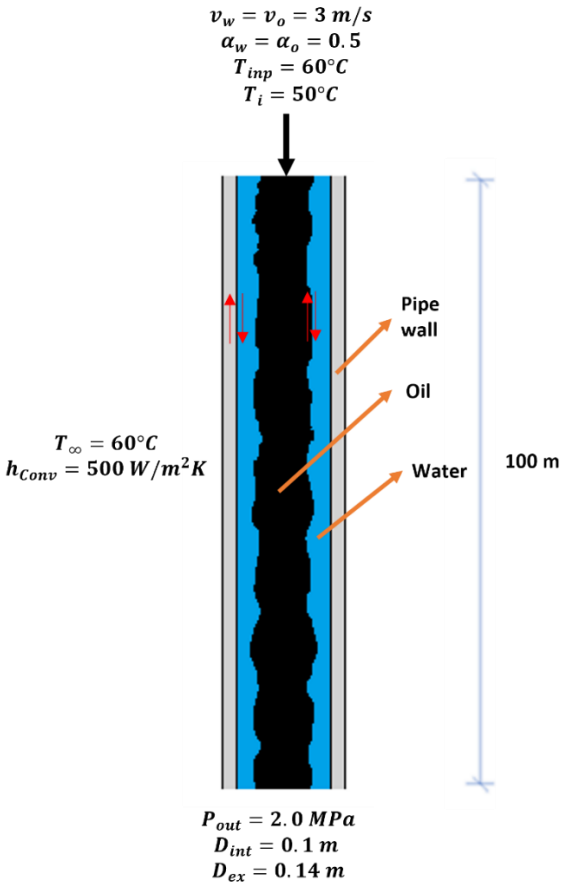


Figure 3. Schematic of the annular water–oil flow and pipe geometry, showing the phase distribution, flow direction, and applied boundary conditions.

Solutions were obtained using 16, 32, 64, and 128 control volumes (CV) up to t = 5 s. In Fig. 4, the temperature at t = 5 s is presented for ALFAsim, FOU, and the second-order MUSCL scheme, using as reference the converged ALFAsim result obtained with a 1,024 CV grid. After 5 s, temperatures are expected to be near 60 °C up to about 15 m, decreasing to roughly 50 °C along the remaining length, as both fluids flow at 3 m/s. To ensure stability, a very small time step corresponding to CFL = 0.01 was required for both the in-house simulator and ALFAsim; higher CFL values led to oscillations due to the strong nonlinearity of the energy equation. Convergence was verified when the residuals of all governing equations fell below a tolerance of 10^{-3}.

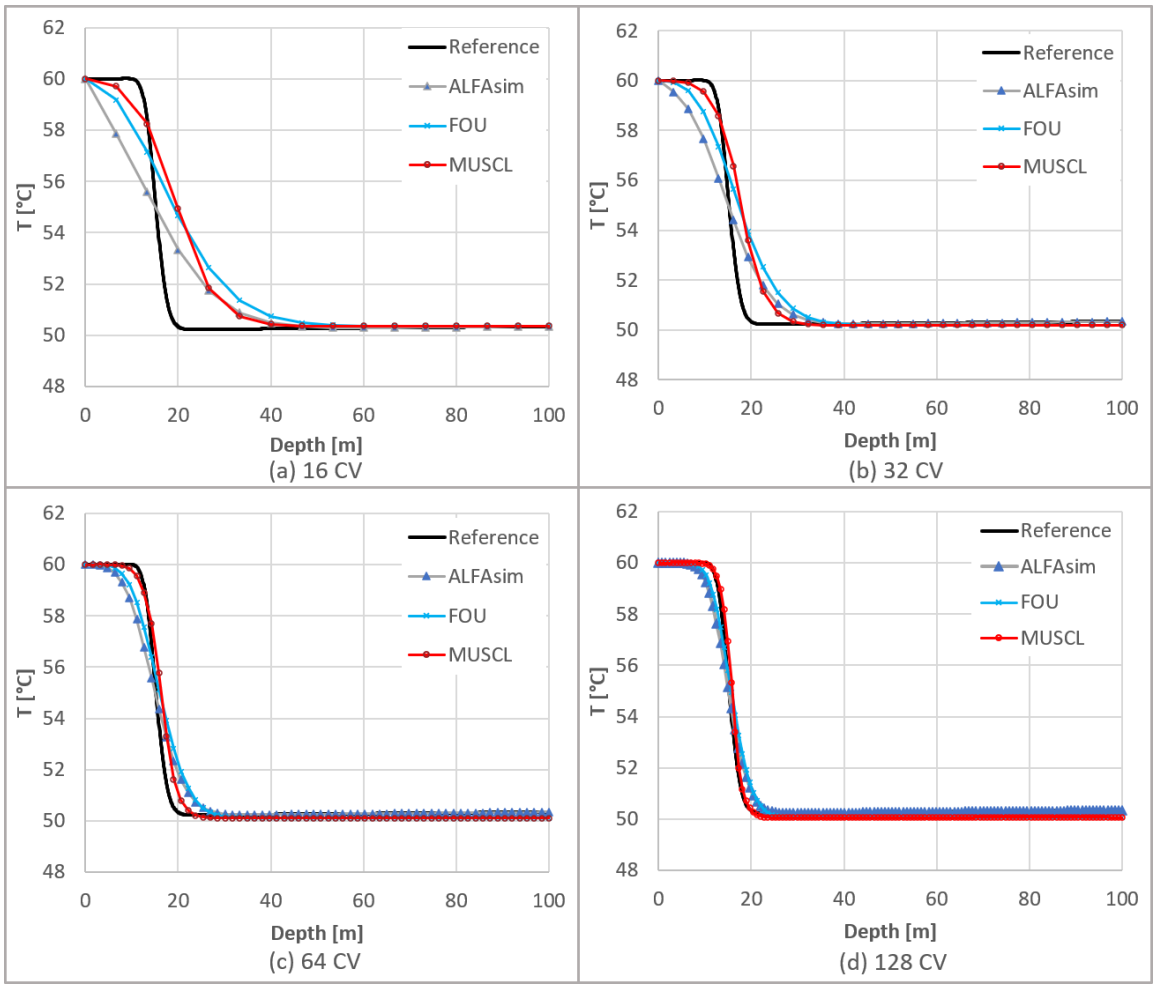


Figure 4. Temperature profiles for the vertical downward 1D annular oil–water flow at t = 5 s, comparing results from ALFAsim, FOU, and MUSCL.

In Fig. 4(a), corresponding to the mesh with 16 control volumes (CVs), it is observed that the worst performance is presented by ALFAsim, which shows greater numerical diffusion compared to the other methods. Next, the FOU scheme presents an intermediate response, while MUSCL provides the best result among the three, being closer to the reference solution.

In Fig. 4(b), with a mesh of 32 CVs, there was a significant improvement in all cases, maintaining the same order of quality observed for the coarser mesh: ALFAsim with greater diffusion, followed by FOU, and MUSCL again as the best performing method.

Analyzing Fig. 4(c), referring to the mesh of 64 CVs, all results showed further improvement. In this scenario, the results of the ALFAsim and FOU schemes were very close to each other, demonstrating reduced numerical diffusion. The MUSCL method, in turn, maintained its superiority, delivering a more accurate and consistent solution compared to the others.

Finally, Fig. 4(d), which corresponds to the mesh with 128 CVs, reveals that the curves obtained by the ALFAsim and FOU methods almost overlap, indicating convergence between these two schemes, although still not fully reaching the reference solution. In contrast, the MUSCL curve already converges to the reference solution, demonstrating its higher accuracy. Thus, the mesh of 128 CVs proves satisfactory for the proposed analysis, ensuring reliable results especially when used together with the MUSCL scheme. Also as expected, the temperatures in the duct are close to 60 °C at the 15 m position, as seen in the figure corresponding to the 128 CV grid for the MUSCL method.

Based on the L_1 norm in the upstream shock region, MUSCL error to the reference solution reduced by factors of ~4.5 in comparison to the corresponding FOU error for 64 CV mesh and by a factor of ~26.3 for 128 CV mesh, respectively, at runtime increases of only 17% and 2%, respectively, evidencing a favorable cost–benefit balance for the use of higher-order schemes.

5 Conclusions

In this work, the MUSCL and FOU methods were implemented in the two-fluid five-equation model with a single temperature, using an equation of state that accounts for variations in pressure and temperature to update the phase densities. The methods are used to solve the conservation equations of mass, linear momentum, and energy. The results show that the second-order MUSCL method provides considerable improvements in accuracy when compared to FOU and ALFAsim.

In the near future, we intend to compare the computational performance of the different alternatives tested and to implement very high-order schemes to further improve accuracy. At this stage, the very high-order FR method with h-p adaptation will be considered for comparison with the other approaches [13,14].

Acknowledgments. The authors would like to thank PPGEM-UFPE, CAPES, Petrobras (project number 2023/00049-1), FACEPE (IBPG-0199-3.05/24) and CNPq (process number PQ-306508/2024-9 and PQ-310145/2021-0) for their financial support.

References

[1] Agência Nacional do Petróleo, Gás Natural e Biocombustíveis (ANP). “Brazil’s national average oil and gas production hits record in 2023”. Brasília, DF: ANP, 2024.
[2] Agência Nacional do Petróleo, Gás Natural e Biocombustíveis (ANP). “Statistical Yearbook of Oil, Natural Gas and Biofuels 2023”. Brasília, DF: ANP, 2023.
[3] Instituto Brasileiro de Petróleo e Gás (IBP). “Pre-salt gas: opportunities, challenges and perspectives”. 2017.
[4] A. Kumar. “Perspectives of Flow Assurance Problems in Oil and Gas Production: A Mini review”. Energy & Fuels, vol. 37, pp. 8142–8159, 2023.
[5] KUMAR, Asheesh. Perspectives of Flow Assurance Problems in Oil and Gas Production: A Mini-review. *Energy Fuels*, 37, 8142−8159, 2023.
[6] J. Gong, Q. Kang, H. Wu, X. Li, B. Shi, S. Song. “Application and prospects of multi-phase pipeline simulation technology in empowering the intelligent oil and gas fields”. Journal of Pipeline Science and Engineering, v. 3, 100127, 2023.
[7] ESSS. “About ESSS”. 2025.
[8] C. N. Sondermann, R. M. Baptista, F. B. F. Rachid, and G. C. R. Bodstein. “Numerical simulation of non-isothermal two-phase flow in pipelines using a two-fluid model”. Journal of Petroleum Science and Engineering, vol. 173, pp. 298–314, Feb. 2019.
[9] E. F. Simões, J. N. E. Carneiro, and A. O. Nieckele. “Numerical prediction of non-boiling heat transfer in horizontal stratified and slug flow by the Two-Fluid Model”. International Journal of Heat and Fluid Flow, vol. 47, pp. 135–145, 2014.
[10] Z. J. Wang. “Spectral (finite) volume method for conservation laws on unstructured grids: basic formulation”. Journal of Journal of Computational Physics, v. 178, pp. 210, 2002.
[11] B. van Leer. “Towards the Ultimate Conservative Difference Scheme, V. A Second Order Sequel to Godunov's Method”. Journal of Computational Physics, v 32, pp. 101–136, 1979.
[12] L. HUANG, Z. JIANG S. LOU, X. ZHANG, C. YAN. “Simple and Robust h-Adaptive Shock-Capturing Method for Flux Reconstruction Framework”. Chinese Journal of Aeronautics, 2023.
[13] H. FU, J. XIA, X. MA. "Shock capturing with the high-order flux reconstruction method on adaptive meshes based on p4est". Engineering Reports, v. 4, n. 10, p. e12501, 2022.
[14] The Julia Project. “The Julia Language”, Version 1.11.6, July 9, 2025.
[15] O. Bratland. *Pipe Flow 2: Multi-phase Flow Assurance*. Self-published, Bangkok, 2010.
[16] C. R. Maliska. “Transferência de Calor e Mecânica dos Fluídos Computacional”. 2 ed. LTC, 2017.
[17] G. Galindez-Ramirez, D. K. E. Carvalho and P. R. M. Lyra. “Numerical simulation of 1-D oil and water displacements in petroleum reservoirs using the correction procedure via reconstruction (CPR) method”. Computational Geosciences, pp. 1–15, 2019.
[18] G. Galindez-Ramirez, F. R. L. Contreras, D. K. E. Carvalho and P. R. M. Lyra. “Numerical simulation of two-phase flows in 2-D petroleum reservoirs using a very high-order CPR method coupled to the MPFA-D finite volume scheme”. Journal of Petroleum Science and Engineering, 107220, 2020.
[19] P. L. Roe. "Characteristic-based schemes for the Euler equations". *Annu. Rev. Fluid Mech*., v. 18, pp. 337–365, 1986.
[20] P. Wu, F. Chao, D. Wu, J. Shan and J. Gou, “Implementation and Comparison of High-Resolution Spatial Discretization Schemes for Solving Two-Fluid Seven-Equation Two-Pressure Model”. Science and Technology of Nuclear Installations, pp. 1–14, 2017
[21] S. V. Patankar. Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation, 1980.