Numerical Modeling of the 1-D Two-Phase Flow in Pipelines by Using the Two-Fluid Model and a Very High Order (VHO) Flux Reconstruction (FR) Scheme

Anderson V. do Nascimento[1], Darlan K. E. de Carvalho[2,] Giselle M. L. L. da Silva[3], Paulo R. M. Lyra[2], Alessandro R. E. Antunes[5]

[1]*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*

[2]*Departamento de Engenharia Mecânica, CTG – UFPE, Av. Acadêmico Hélio Ramos, s/n, CEP, Recife, PE 50740-530 Brazil, darlan.ecarvalho@ufpe.br*

[3]*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*

[4]*Departamento de Engenharia Mecânica, CTG – UFPE, Av. Acadêmico Hélio Ramos, s/n, CEP, Recife, PE 50740-530 Brazil, paulo.lyra@ufpe.br*

[5]*Núcleo de Tecnologia, CAA – UFPE, Rod. BR-104 – Km 59 – Nova Caruaru CEP, Caruaru, PE 55002-970, Brazil, alessandro.antunes@ufpe.br*

Abstract. Flow models for 1-D two-phase flows in pipelines are commonly implemented using first-order schemes. Despite being simple and robust, these schemes introduce a large amount of numerical diffusion due to low order truncation errors, causing a high loss of accuracy. In the present work, for the first time in literature, we propose the use of the very high-order (VHO) flux-reconstruction (FR) method to improve the accuracy and efficiency of one-dimensional two-phase flow simulations in pipelines. The FR is implemented to solve the mass and momentum conservation equations of the isentropic four-equation single-pressure two-fluid model. The pressure correction equation is obtained through the mass conservation and a semi-implicit pressure-based method SIMPLE-like is used to perform the coupling. The pressure equation is solved through the Two-Point Flux Approximation (TPFA) finite volume technique. To test and numerically validate our formulation, we present two benchmark problems. For the problems we have solved, our results are very promising.

Keywords: 1-D Fluid Flow in Pipelines, Four-Equation Single-Pressure Two-Fluid Model, SIMPLE algorithm, Flux reconstruction (FR), Finite Volume Method.

1 Introduction

A variety of engineering problems have their resolution dependent on the knowledge of the two-phase flow inside pipelines, as in the case of the nuclear, mechanical, chemical and petroleum industries problems. The flow assurance, for example, involves a series of problems in the oil industry and its management depends on the knowledge of the pressure, velocity and temperature fields of gas–liquid flow inside the pipelines [1]. However, predicting physical conditions of two-phase flows in large pipelines is a task that involves several difficulties.

A very useful tool for the prediction of two-phase flows inside pipelines is the numerical simulation. However, even with this approach due to the high computational cost involved, many simplifications are commonly assumed for numerical simulation to be feasible from a computational point of view. A method to efficiently represent the behavior of two-phase flows within large pipelines is required, and is the use of a simplified one-dimensional two-fluid model together with an appropriate numerical method represents a good option [2]. A two-fluid model consists of a system of non-linear partial differential equations that represent the mass, momentum and energy conservation principles, written for each phase [3].

Various formulations exist for the two-fluid model concerning the derivation of governing equations and their practical computational application [3, 4]. In this research, we seek to solve the common variant of the two-fluid model, that considers that both phases share the same pressure field, i.e., the so called two-fluid four-equation single-pressure model, which is the isothermal case of the two-fluid six-equation single-pressure model [5], same model used in reactor thermal-hydraulics analysis codes RALAP, TRAC, and CATHARE [6, 7]. This model, however, suffers from the Kelvin–Helmholtz instability [8], that arises whenever two fluids are forced to flow together sharing the pressure field. To solve this problem, one of the alternatives is to use the CATHERE model [9], which use an interfacial pressure difference [10].

Usually, it is common to use staggered grid and donor-cell differencing schemes to obtain stable results in two-fluid six-equation model [5, 11], however, the diffusion added by donor-cell differencing schemes reduces the accuracy of the solutions. Therefore, adopting higher-order schemes together with the CATHERE model can also guarantee stable and more accurate results. There are several higher-order schemes in literature that can be used to model such problems, such as the Monotonic Upstream-centered Scheme for Conservation Laws (MUSCL) scheme, proposed by van Leer [12], the spectral volume (SV) method, that was developed in papers of Wang and Liu [13,14,15,16,17], the spectral difference (SD) method, proposed by Liu et al. [18], and the Flux Reconstruction (FR) method, proposed by Huynh [19], which has been recently adapted for the numerical modeling of the multiphase and multicomponent fluid flow in heterogeneous and anisotropic porous media by part of the authors of the present work [20, 21, 22, 23, 24].

In the present work, we implement a semi-implicit algorithm to solve the hyperbolic 1D two-fluid model [25], using the 2nd order MUSCL type finite volume and the very high order (≥ 3) FR method. As far as we know, this is the first work in which the FR is used to model the two-fluid model in pipelines. Even though our implementation of the FR method still presents certain problems for the complete two-fluid model our results for a special smooth problem are very promising showing, the potential of our formulation to model more complex flow assurance problems. The whole formulation was implemented using the Julia computational language [26].

2 Mathematical formulation

The two-fluid four-equation single-pressure model is composed by two continuity equations, one for each phase, and two momentum equations, also one for each phase. The equations were adapted for the 1D, compressible, inviscid and isothermal flow. The continuity equation, for phase $k = L(liquid), G(gas)$, can be written as:

$$\frac{\partial}{\partial t}(\alpha_k \rho_k) + \frac{\partial}{\partial x}(\alpha_k \rho_k u_k) = 0 \quad (1)$$

where α_k represents the volumetric fraction occupied by phase k in the section, ρ_k represents the density and u_k the velocity. Using the same terms, we can write the momentum equation, for phase k, as:

$$\frac{\partial}{\partial t}(\alpha_k \rho_k u_k) + \frac{\partial}{\partial x}(\alpha_k \rho_k u_k^2) = -\alpha_k \frac{\partial p_k}{\partial x} - \left(p_k - p_k^i\right) \frac{\partial \alpha_k}{\partial x} + \alpha_k \rho_k g \sin\theta \quad (2)$$

where p is the pressure shared by the phases, g is the acceleration of gravity, θ is the pipe slope, and p_k^i is the interface pressure in phase k. In this model, the effects of mass transfer, wall friction and interface friction are neglected.

In order to ensure the hyperbolicity of the system of equations and to stabilize the simulation, we have adopted the CATHERE model [9], which uses the artificial interfacial pressure difference term:

$$\left(p_k - p_k^i\right) = \Delta p = \gamma \frac{\alpha_G \alpha_L \rho_G \rho_L}{\alpha_L \rho_G + \alpha_G \rho_L}(u_G - u_L)^2 \quad (3)$$

The constant γ is a value introduced to guarantee hyperbolicity [9]. As recommended by Evje and Flatten [27] and adopted by Wang et al. [10], we have used $\gamma = 1.2$.

3 Numerical Formulation

To discretize the governing equations, we have combined the locally conservative finite volume method (FVM) [28] for the pressure equation based on discretized by the classical Two Point Flux Approximation (TPFA) and the Flux Reconstruction FR method to discretize the transport problem [22, 23, 24]. In Fig. 1, we show the staggered grid strategy that we have used. This grid is used to avoid the odd-even decoupling problem in pressure-correction algorithms [28]. The vector variables (velocities) are defined at the control surfaces, while the scalar variables (pressure and volume fractions) are computed and associated to the control volumes of the primal mesh. The letters w, e and ee used to identify neighboring control surfaces, while W, P, E and F are the control volumes of the primal mesh, and W^S, P^S, E^S and F^S are the control volumes of the dual mesh.

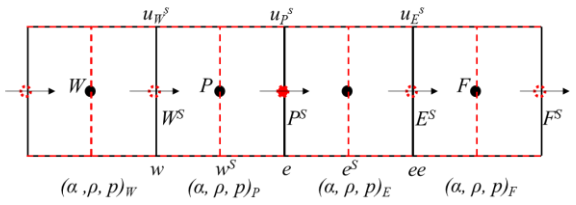

Figure 1. Control volumes and the staggered grid approach (adapted from Wang et al. [10]).

The advective terms are evaluated applying different methods. For comparison purposes we have used the first-order upwind (FOU) or the MUSCL schemes to approximate the fluxes. In this case, we have used the Euler Backward for the time integration. Also, for the first time in literature, we have also implemented the Flux Reconstruction (FR) method to approximate de advective fluxes. For the latter case, to improve robustness and accuracy, time integration was performed using the third-order explicit Total Variation Diminishing Runge-Kutta (TVD-RK) method [29].

For the FOU and MUSCL schemes, the value on the face e (Fig.1) of a generic advected variable can be written [30], as follows:

$$\phi_e = \begin{cases} \phi_P + \frac{\Delta x}{2}\psi(r)\left(\frac{\partial \phi}{\partial x}\right)_w, & \text{if } u_e > 0 \\ \phi_E - \frac{\Delta x}{2}\psi(r)\left(\frac{\partial \phi}{\partial x}\right)_{ee}, & \text{otherwise} \end{cases} \tag{4}$$

where $\psi(r)$ represents the flux limiter function and r is the gradient ratio ($\mathrm{r} = \nabla\phi_{\mathrm{upwind}}/\nabla\phi_{\mathrm{downwind}}$). For the FOU scheme, $\psi = 0$, while for the MUSCL scheme different values of $\psi(r)$ can be adopted giving rise to different slope limiters. Here we choose the Minmod slope limiter [31]

$$\psi(r) = \max[0, \min(1, r)] \tag{5}$$

For FR method, the velocities and volume fractions fields are saved in points named solution points [19], that are localized by a quadrature. Therefore, for the FR, velocities and volumetric fractions are co-localized. The pressure field continues to be balanced at the centers of the CVs. In the Fig. 2, it is showed the grid with solution points for a case that two points are used for to reconstruct first degree polynomials (FR-P1).

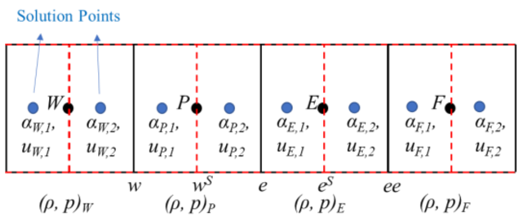

Figure 2. Control volumes for FR-P1.

In the FR method, the continuous flux function F^C is constructed as:

$$F^C = F_i^{\mathcal{P}n} + \left(F_L^{\delta I} - F_L^{\delta D}\right)h_L + \left(F_R^{\delta I} - F_R^{\delta D}\right)h_R \tag{6}$$

and the generic advective term $\frac{\partial F}{\partial x}$ is approximated by:

$$\frac{\partial F}{\partial x} = \sum_{j=0}^{k} F^{\delta D} \frac{dl_j}{dx} + \left(F_L^{\delta I} - F_L^{\delta D}\right) \frac{dh_L}{dx} + \left(F_R^{\delta I} - F_R^{\delta D}\right) \frac{dh_R}{dx} \quad (7)$$

where the terms $F^{\delta D}$ are the discontinuous fluxes, $F_L^{\delta I}$ and $F_R^{\delta I}$ are the left and right fluxes, h is the correction function and l_j are the Lagrange polynomials defined inside each control volume using points named solution points, see [19]. The quadrature used for location of solution points was that of the Lobatto. The Radau polynomials were used for h and the Riemman Solver used was the Local-Lax-Friedrichs (LLF) scheme [32]. These choices are the same ones adopted by Galindo et al. [24].

The semi-implicit pressure-based method SIMPLE [33] was used to couple the momentum and continuity equations. A pressure-correction equation was used to allow the pressure field to be updated. It was derived from the sum of the mass conservation equations for liquid and gas, dividing each equation by the representative values of the corresponding density [10]. The global algorithm uses a predictor-corrector scheme [25]. The resolution steps follow, inside a loop, the following steps, until a certain tolerance is obtained:

1. Solve implicitly (with FOU AND MUSCL) or explicitly (with FR) the momentum equations for velocities, using the pressure field of previous iteration;
2. Solve implicitly the pressure-correction equation, using the new velocity fields;
3. Update explicitly pressure, velocity and density fields, using the pressure variation obtained in the previous step;
4. Solve implicitly (with FOU AND MUSCL) or explicitly (with FR) the continuity equations for volume fractions;
5. Updates the variables and checks if the tolerance has been reached.

We use the three diagonal matrix algorithm TDMA to solve the set of algebraic equations when the matrices are tridiagonal, i.e., solving the pressure problem, and a Gauss elimination solver for the momentum and continuity equations which are not tridiagonal.

4 Results

4.1 Convergency Study

This case is a simple non-linear model, essentially the smooth and inviscid Burger´s problem [34], that can be interpreted as a particular case of Eq. (2) in which $\alpha_k \rho_k = 1$ and the RHS $= 0$. We evaluate the convergence rates of the FR for the following orders of approximation: P1 (2nd order accuracy), P2 (3nd order) and P3 (4nd order) polynomials, similar to what has been done by Galindo et al. [24]. For comparison purposes, we have also included the results for the FOU and the MUSCL methods. We compute the L1 norm of the errors and the numerical convergence rates, for the smooth solution at $t = 0.5$. The results are presented in Tab. 1. Similar as done in Galindo et al. [24], the simulations are made using sequentially refined meshes from 20 to 1,280 CVs.

Table 1. Error and convergence rates for the L1-norm of error of Convergency Study.

	FOU		MUSCL		FR-P1		FR-P2		FR-P3	
CVs	E_{L1}	R_{L1}	E_{L1}	R_{L1}	E_{L1}	R_{L1}	E_{L1}	R_{L1}	E_{L1}	R_{L1}
20	6.10E-03	-	3.16E-03	-	4.26E-03	-	1.17E-04	-	1.10E-05	-
40	3.59E-03	0.765	9.86E-04	1.680	1.13E-03	1.909	1.64E-05	2.840	1.01E-06	3.451
80	1.93E-03	0.894	3.15E-04	1.647	3.22E-04	1.816	2.24E-06	2.872	8.95E-08	3.493
160	1.00E-03	0.943	9.11E-05	1.789	9.00E-05	1.839	2.98E-07	2.905	7.55E-09	3.568
320	5.12E-04	0.971	2.53E-05	1.846	2.48E-05	1.863	3.90E-08	2.936	6.41E-10	3.558
640	2.59E-04	0.985	6.79E-06	1.900	6.86E-06	1.852	5.02E-09	2.959	5.54E-11	3.533
1,280	1.30E-04	0.993	2.39E-06	1.506	1.88E-06	1.868	6.38E-10	2.975	4.99E-12	3.470

For all schemes tested, the convergence behavior is quite close to the formal asymptotic order of accuracy, except for the FR-P3 which as shown a small lost in accuracy for all meshes and the MUSCL scheme for the mesh with 1,280 CVs.

4.2 Discontinuity Moving in the Uniform Two-Phase Flow

This case is a 1D horizontal pipe with 12 m long that has a diaphragm in its midsection and different left and the right states, according to Fig. 2, and based in [35]. The initial state and the state after 0.2 s are presented in the figure. The pipe is assumed to be adiabatic, and the frictional forces are neglected. The air void fraction (α_G) is advected along the computational domain when the flow starts.

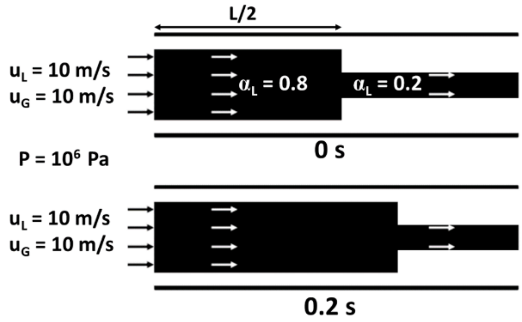

Figure 2. Discontinuity Moving Problem.

The solutions are obtained using 1,000 CVs for time t = 0.2 s. First, in Fig. 3, the result of the air void fraction (α_G) is shown for FOU and MUSCL schemes. Unfortunately, as shown by Fig. 4, from the beginning of the simulation, the FR method, even with P1 and using 100 CVs, still presents spurious oscillations, probably due to some implementation issues.

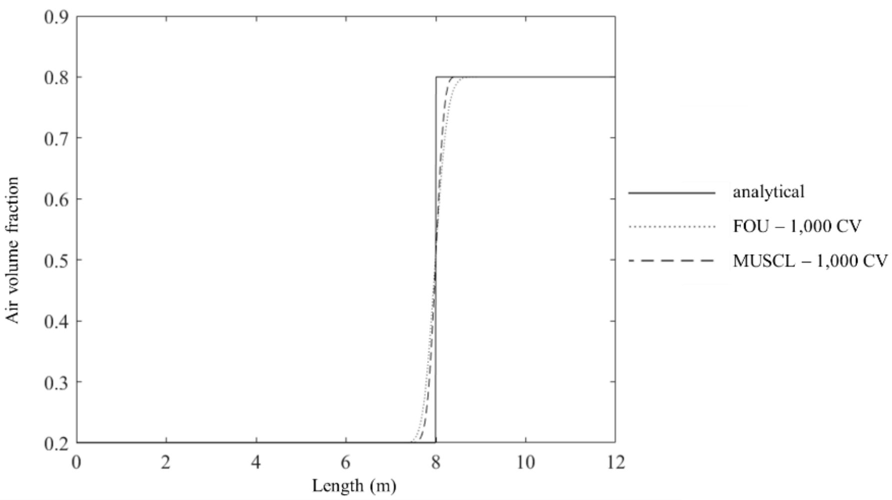

Figure 3. Air void fraction (α_G), t = 0.2 s, FOU and MUSCL.

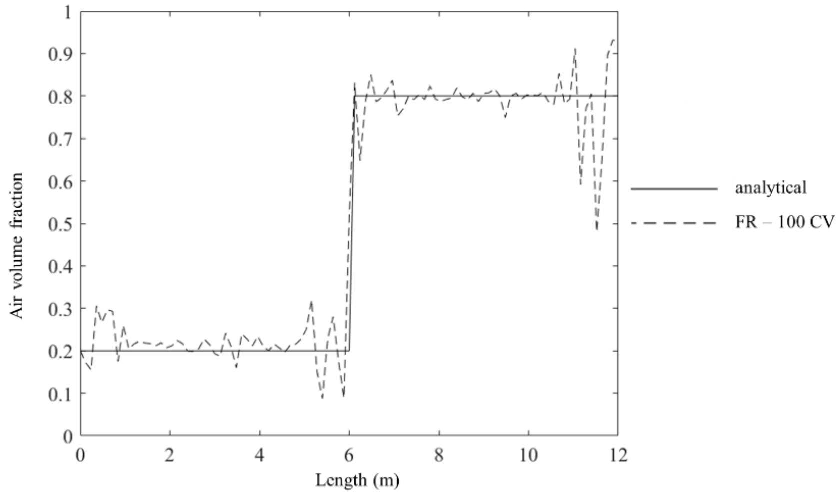

Figure 4. Air void fraction (α_G), t = 0.004 s, FR-P1 method.

The results for the FOU and the MUSCL were satisfactory, the FR implementation still presents spurious oscillations. We are still investigating the source of the problems and, hopefully soon this issue will be addressed.

5 Conclusions

In present work, we have implemented a 1-D semi-implicit numerical algorithm based on the finite volume method and the flux reconstruction formulation. The mathematical model used was the two-fluid four-equation single-pressure model which was stabilized using an interfacial pressure difference term. The convective terms are evaluated applying first-order upwind (FOU) scheme, the MUSCL scheme and the FR method. For the convergency study (example 1), the FR method showed excellent behavior, however, for the Discontinuity Moving Problem (example 2), the FR method still presents some problems and it's not working correctly. Beside the correction of these minor issues, in the near future, we intend to implement the FR successfully for more advanced problems and to include more physics by using the six-equation model to include thermal effects and to incorporate mass transfer terms and frictional forces.

References

[1] J. Su, "Flow Assurance of Deepwater Oil and Gas Production – A Review". In: *22 ed. International Conference on Offshore Mechanics and Arctic Enginnering*, 2003.
[2] M. Jerez-Carrizales, J. E. Jaramillo and D. Fuentes, "Prediction of Multiphase Flow in Pipelines: Literature Review". *Ingeniería y Ciencia*, v. 11, n. 22, pp. 213-233, 2015.
[3] M. Ishii and T. Hibiki. *Thermo-fluid dynamics of two-phase flow*. 2 ed. Springer New York, 2011.
[4] A. Prosperetti, and G. Tryggvason. *Computational Methods for Multiphase Flow*. Cambridge University Press, 2007.
[5] I. Toumi, "An Upwind Numerical Method for Two-Fluid Two-Phase Flow Models". *Nuclear Science and Engineering*, v. 123, pp. 147–168, 1996.
[6] H. Pokharna, M. Mori, and V. H. Ransom, "Regularization of Two-Phase Flow Models: A Comparison of Numerical and Differential Approaches". *Journal of Computational Physics*, v. 134, pp. 282–295, 1997.
[7] M. Pourgol-Mohammad. "Thermal–hydraulics system codes uncertainty assessment: A review of the methodologies". *Annals of Nuclear Energy*, v. 36, pp. 1774–1786, 2009.
[8] M. L. Bertodano, W. Fullmer, A. Clausse and V. H. Ransom. *Two-Fluid Model Stability*. Simulation and Chaos Springer International Publishing, Switzerland, 2017.
[9] D. Bestion, "The Physical Closure Laws in the CATHARE code". *Nuclear Science and Engineering*, v. 124, pp. 229–244, 1990.
[10] Z. Wang, J. Gong and C. Wu. "Numerical Simulation of One-Dimensional Two-Phase Flow Using a Pressure-Based Algorithm". *Numerical Heat Transfer*, Part A, v. 68, p. 369–387, 2015.
[11] H. Städtke. *Gasdynamic Aspects of Two-Phase Flow: Hyperbolicity, Wave Propagation Phenomena, and Related Numerical Methods*. Wiley-Vch, 2006.
[12] 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.

[13] 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.
[14] Z. J. Wang and Y. Liu. “Spectral (finite) volume method for conservation laws on unstructured grids II: extension to two-dimensional scalar equation”. *Journal of Computational Physics*, v. 179, pp. 665–697, 2002.
[15] Z. J. Wang and Y. Liu. “Spectral (finite) volume method for conservation laws on unstructured grids III: extension to one dimensional systems”. *Journal of Scientific Computing*, v. 20, pp. 137, 2004.
[16] Z. J. Wang and Y. Liu. “The spectral difference method for the 2D Euler equations on unstructured grids”. *AIAA Paper No. 2005*, pp. 5112, 2005.
[17] Z. J. Wang and Y. Liu. “Extension of the spectral volume method to high-order boundary representation”. *Journal of Computational Physics*, v. 211, pp. 154–178, 2006
[18] Y. Liu, M. Vinokur, Z. J. Wang. “Discontinuous spectral difference method for conservation laws on unstructured grids”. In: *Proceedings of the 3rd International Conference on Computational Fluid Dynamics*, Toronto, Canada, July 12–16, 2004.
[19] H. T. Huynh. “Flux Reconstruction Approach to High-Order Schemes Including Discontinuous Galerkin Methods”. *AIAA Conference Paper 2007*, pp. 4079, 2007.
[20] F. R. L. Contreras, D. K. E. Carvalho, G. Galindez-Ramirez and P. R. M. Lyra. “A non-linear finite volume method coupled with a modified higher order muscl-type method for the numerical simulation of two-phase flows in non-homogeneous and non-isotropic oil reservoirs”. *Computers & Mathematics with Applications*, v. 92, pp. 120–133, 2021.
[21] 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.
[22] 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.
[23] G. Galindez-Ramirez, F. R. L. Contreras, D. K. E. Carvalho and P. R. M. Lyra. “A very high-order flux reconstruction approach coupled to the MPFA-QL finite volume method for the numerical simulation of oil-water flows in 2D petroleum reservoirs”. *Applied Mathematical Modelling*, v. 106, pp. 799–821, 2022.
[24] M. E. S. Galindo, I. V. Lacerda, G. Galindez-Ramirez, P. R. M. Lyra and D.K.E. Carvalho. “A very high order Flux Reconstruction (FR) method for the numerical simulation of 1-D compositional fluid flow model in petroleum reservoirs.” *Geoenergy Science and Engineering*, v. 229, 2023.
[25] S. Morales-Ruiz, J. Rigola, I. Rodriguez and A. Oliva, “Numerical resolution of the liquid–vapour two-phase flow by means of the two-fluid model and a pressure-based method”. *International Journal of Multiphase Flow*, v. 43, pp. 118–130, 2012.
[26] The Julia Project. The Julia Language, V1.9.2, July 6, 2023.
[27] S. Evje and T., “Flatten. Hybrid Flux-Splitting Schemes for a Common Two Fluid Model”. *Journal of Computational Physics*, v. 192, pp. 175–210, 2003.
[28] C. R. Maliska. *Transferência de Calor e Mecânica dos Fluídos Computacional*. 2 ed. LTC, 2017.
[29] S. Gottlieb and C.-W. Shu. “Total Variation Diminishing Runge-Kutta Schemes”. *Mathematics of Computation*, v. 67, n. 221, pp. 73-85, 1998.
[30] 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.
[31] P. L. Roe. "Characteristic-based schemes for the Euler equations". *Annu. Rev. Fluid Mech*., v. 18, pp. 337–365, 1986.
[32] B. Cockburn and C.-W. Shu. “TVB Runge-Kutta Local Projection Discontinuous Galerkin Finite Element Method for Conservation Laws II: General Framework.” *Mathematics of Computation*, v. 52, n. 186, pp. 411–35, 1989.
[33] S. V. Patankar. *Numerical Heat Transfer and Fluid Flow*. Hemisphere Publishing Corporation, 1980.
[34] J. Hesthaven and T. Warburton. “Nodal discontinuous Galerkin methods”. *Texts in Applied Mathematics*, v. 54, 2008.
[35] G. S. Yeom and K. S. Chang. “Flux-Based Wave Decomposition Scheme for an Isentropic Hyperbolic Two-Fluid Model”. *Numer. Heat Transfer B*, v. 59, pp. 288–318, 2011.

Acknowledgements

The authors would like to thank Petrobras for the financial support (project number 2023/00049-1).