24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-0210
NUMERICAL SIMULATION OF THE TWO-PHASE WATER-FAUCET FLOW USING A SINGLE PRESSURE FOUR EQUATION MODEL ASSOCIATED WITH THE FLUX-CORRECTED TRANSPORT METHOD

Raphael V. N. Freitas
Carina N. Sondermann
Rodrigo A. C. Patrício
Aline B. Figueiredo
Gustavo C. R. Bodstein
Dept. of Mechanical Engineering, Federal University of Rio de Janeiro (UFRJ – COPPE/PEM), Rio de Janeiro, RJ, Brasil
Av. Athos da Silveira Ramos, 149, Centro de Tecnologia - Cidade Universitária - Rio de Janeiro - RJ - Brasil
raphael_viggiano@poli.ufrj.br
carinans@poli.ufrj.br
rac.patricio@mecanica.coppe.ufrj.br
aline.figueiredo13@gmail.com
gustavo@mecanica.coppe.ufrj.br

Felipe B. F. Rachid
Dept. of Mechanical Engineering, Fluminense Federal University (PGMEC/UFF), Rio de Janeiro, RJ, Brasil
Rua Passo da Pátria, 156, Campus da Praia Vermelha - São Domingos – Niterói - RJ - Brasil
rachidfelipe@gmail.com

Abstract. *Numerical Simulation of two-phase flow in pipelines is an important tool to predict the behavior of the flow quantities, especially in the real-life engineering problems. A one dimensional single-pressure four-equation model was used in previous works to simulate specific types of flows, such as transient stratified-pattern two-phase flows in gas pipelines, with or without a PIG passing through the pipeline, and in the occurrence of a leakage in the system. The model is discretized within a finite-difference framework and is solved using the Flux-Corrected Transport (FCT) method, where the eigenstructure stays within the hyperbolic region. This work uses the common test case of gravity dominated flow, also known as the water faucet problem, to promote further testing of the numerical method and the single-pressure four-equation model in question. We chose this test case as benchmark in this work because it offers an analytical solution and the particular presence of a discontinuity in the volume fraction before reaching the steady state.*

Keywords: *two-phase flow, flux corrected transport method, numerical simulation, gas pipeline, petroleum engineering.*

1. INTRODUCTION

Pipelines are commonly used by various industries to transport fluids. Specifically, in the oil and gas industry, it is very common the presence of multiphase flows in the ducts The operation of this components are critical specially in the occurrence of interferences in the line, such as the operation of valves and a passage of a pig through the line, or in non-routine events, such as the occurrence of a leak in the system or the accumulation of solid material in the line. Modeling of two-phase flows that is solved with a numerical method is a reliable way to predict the behavior of the flow quantities along the pipeline. A one dimensional single-pressure four-equation model alongside with the Flux-Correct Transport (FCT) method can be used to simulate two-phase flows of specific gas-pipeline cases. To support this approach, we propose a common benchmark, a gravity dominated flow, also known as the water faucet case problem, to use as a validation test of the numerical model. This numerical model has already been used on previous work for the simulation of a *PIG* passing through a pipeline (Patricio, 2016) and in the occurrence of a leakage in the system (Figueiredo *et al.* 2017).

2. MATHEMATICAL MODEL

For the mathematical modeling of the conservation laws, we assume a one dimensional isothermal time-dependent flow. Geometrical parameters, such as the pipe's cross-section area and shape, are considered to be constant throughout the pipeline length. The pressure is supposed to be equal for both phases at any section of the pipe and there is no mass transfer between the two phases. Based on these hypotheses we use the single-pressure four-equation two-fluid model, which consists in a system of four nonlinear partial differential equations comprised of the mass-conservation equation and momentum-balance equation for each phase separately, that is,

$$\frac{\partial \alpha_G \rho_G}{\partial t} + \frac{\partial \alpha_G \rho_G u_G}{\partial x} = 0\,, \tag{1}$$

$$\frac{\partial \alpha_L \rho_L}{\partial t} + \frac{\partial \alpha_L \rho_L u_L}{\partial x} = 0\,, \tag{2}$$

$$\frac{\partial \alpha_G \rho_G u_G}{\partial t} + \frac{\partial \left(\alpha_G \rho_G u_G^2 + \alpha_G p \right)}{\partial x} = p_I \frac{\partial \alpha_G}{\partial x} - \rho_G \alpha_G g \text{ sen}\, \beta + T_I + T_{Gw}\,, \tag{3}$$

$$\frac{\partial \alpha_L \rho_L u_L}{\partial t} + \frac{\partial \left(\alpha_L \rho_L u_L^2 + \alpha_L p \right)}{\partial x} = p_I \frac{\partial \alpha_L}{\partial x} - \rho_L \alpha_L g \text{ sen}\, \beta - T_I + T_{Lw}\,, \tag{4}$$

or in the canonical form

$$\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{Q})}{\partial x} = \mathbf{H}(\mathbf{Q}) \frac{\partial \mathbf{W}}{\partial x} + \mathbf{S}(\mathbf{Q})\,, \tag{5}$$

where **Q** is the conservative variables vector, **W** is the primitive vector, **F** is the flux vector, **H** is the matrix of non-conservative term and **S** is the source term vector. This terms can be defined as

$$\mathbf{W} = \begin{bmatrix} p & \alpha_G & u_G & u_L \end{bmatrix}^T \tag{6}$$

$$\mathbf{Q} = \begin{bmatrix} \alpha_G \rho_G \\ \alpha_L \rho_L \\ \alpha_G \rho_G u_G \\ \alpha_L \rho_L u_L \end{bmatrix} \tag{7}$$

$$\mathbf{F} = \begin{bmatrix} \rho_G \alpha_G u_G \\ \rho_L \alpha_L u_L \\ \rho_G \alpha_G u_G^2 + \alpha_G p_G \\ \rho_L \alpha_L u_L^2 + \alpha_L p_L \end{bmatrix} \tag{8}$$

$$\mathbf{S} = \begin{bmatrix} 0 \\ 0 \\ -\rho_G \alpha_G g \text{ sen}\, \beta + T_I + T_{Gw} \\ -\rho_L \alpha_L g \text{ sen}\, \beta - T_I + T_{Lw} \end{bmatrix} \tag{9}$$

$$\mathbf{H} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & p_I & 0 & 0 \\ 0 & -p_I & 0 & 0 \end{bmatrix} \tag{10}$$

Using the subscript $K \in \{G,L\}$, where G denotes the gas phase and L the liquid phase, α_K is the volume fraction, u_K is the fluid velocity and ρ_K is the density. The terms p is the single pressure for the gas and liquid phases and and p_I the interfacial pressure. The term that accounts for the interfacial friction is T_I, the wall friction at each phase T_{Kw}, the pipe inclination β and, finally, the gravitational field g. It is also important to mention the relation between the phases' void fractions, provided by

$$\alpha_G + \alpha_L = 1 \tag{11}$$

For the closure of the system we consider that the liquid phase is incompressible and the gas phase is governed by the ideal gas law. The interfacial-pressure model used is the one proposed by Bestion (1990), where the pressure correction term is written as

$$\Delta p_I = p - p_I = \delta \frac{\alpha_G \alpha_L \rho_G \rho_L}{\alpha_G \rho_L + \alpha_L \rho_G} \left(u_G - u_L\right)^2 . \tag{12}$$

The choice for this pressure model relies on the mathematical prospect of keeping the system of non-linear differential partial equations hyperbolic throughout the simulation of the test case addressed in this paper. Other correlations may be applied in order to model the interfacial pressure depending on the flow regime of interest. For more information regarding the hyperbolicity analysis of this model see Sondermann *et al.* (2015).

3. NUMERICAL METHOD

In order to get an approximate numerical solution for the vector of conservative variables $\mathbf{Q}(x,t)$, the one-dimensional system of equations given by Eq. (5) is discretized within a finite-difference framework into N uniform cells as follows

$$\mathbf{Q}_j^{n+1} = \mathbf{Q}_j^n - \frac{\Delta t}{\Delta x}\left[\mathbf{F}_{j+1/2}^n - \mathbf{F}_{j-1/2}^n\right] + \Delta t \left(\mathbf{H}\frac{\partial \mathbf{W}}{\partial x}\right)_j^n + \Delta t \mathbf{S}_j^n . \tag{13}$$

where $\Delta x = L/M$ is the cell length and L is the total length of the pipe. To number the cells in the domain the subscript j is used from j=1 to j=M. The subscripts j-1/2 and j+1/2 denote when a variable is discretized at the inter-cells boundaries between two adjacent cells. The current time step is represented by the superscript n and the next time step is represented by n+1.

The intercell flux vector $\mathbf{F}_{j+1/2}^n$ is obtained using the Flux Corrected transport method, which was first proposed by Boris and Book (1973) and later by Book *et al.* (1975). The FCT method may be interpreted as a predictor/corrector scheme, where diffusion is introduced in the predictor stage and anti-diffusion in the corrector stage. The method is expected to be first-order accurate in time and second-order accurate in space. The advantages of its use are the relatively easy implementation if compared to methods that relies on the Riemann problem, and its ability to be tuned under diffusive behavior. Moreover, the non-conservative term is solved using a second-order min-mod scheme. It is important to note that for this method the *CFL* condition must be respected, according to:

$$CFL = \frac{\Delta t}{\Delta x} \lambda_{j,\max}^n , \tag{14}$$

where $\lambda_{j,\max}^n$ represents the maximum eigenvalue in absolute value for the PDE system of equation for a given cell in a specific numerical time

$$\lambda_{\max}^n = \max_j \left[\max_k \left| \lambda_j^k \right| \right], \text{ for } j = 1,\ldots,M \text{ and } k = 1,\ldots,N , \tag{15}$$

where the local eigenvalue can be obtained locally through the characteristic polynomial of the model

$$P_{4E1P}^{BESTION}(\lambda) = \frac{\alpha_G \rho_G \alpha_L \rho_L}{c_G^2} \Bigg[-\alpha_G \left(u_G - \lambda\right)^2 \left(u_L - \lambda\right)^2 + \frac{\alpha_G \Delta p_I}{\rho_L}\left(u_G - \lambda\right)^2 + \\ + \frac{\rho_G \alpha_L c_G^2}{\rho_L}\left(u_G - \lambda\right)^2 + \alpha_G c_G^2 (u_L - \lambda)^2 - \frac{\Delta p_I c_G^2}{\rho_L} \Bigg]. \tag{16}$$

For the simulation performed in this paper, the time and space steps are treated as constants and the maximum *CFL* limiting value of 0.5 is respected. With this in mind, the maximum eigenvalue is set as the gas sound speedy, considered as an upper bound value.

3.1 Discretization of the conservative flux

a) First approximation of the do conservative variables vector $\tilde{\mathbf{Q}}_j$, using a Ritchmyer scheme, as detailed in Toro (1999):

Firstly the intermediate solution vector $\bar{\mathbf{Q}}_{j+1/2}$ is obtained,

$$\bar{\mathbf{Q}}_{j+1/2} = \frac{1}{2}\left(\mathbf{Q}_{j+1}^{n} + \mathbf{Q}_{j}^{n}\right) - \frac{1}{2}\frac{\Delta t}{\Delta x}\left[\mathbf{F}(\mathbf{Q}_{j+1}^{n}) - \mathbf{F}(\mathbf{Q}_{j}^{n})\right]. \tag{17}$$

Followed by the calculation of the Richmyer's flux vector $\hat{\mathbf{F}}_{j+1/2}^{\mathrm{RI}}$,

$$\hat{\mathbf{F}}_{j+1/2}^{\mathrm{RI}} = \mathbf{F}(\bar{\mathbf{Q}}_{j+1/2}). \tag{18}$$

Finally $\tilde{\mathbf{Q}}_j$ can be obtained,

$$\tilde{\mathbf{Q}}_j = \mathbf{Q}_j^n - \frac{\Delta t}{\Delta x}\left(\hat{\mathbf{F}}_{j+1/2}^{\mathrm{RI}} - \hat{\mathbf{F}}_{j-1/2}^{\mathrm{RI}}\right). \tag{19}$$

b) Computation of the diffusive fluxes:

$$\mathbf{F}_{j+1/2}^{\mathrm{d}} = \nu_{j+1/2}\left(\mathbf{Q}_{j+1}^{n} - \mathbf{Q}_{j}^{n}\right), \tag{20}$$

in which $\nu_{j+1/2}$ is the diffusive coefficient.

c) Computation of the diffusion of the solution:

$$\mathbf{Q}_j^{\mathrm{d}} = \tilde{\mathbf{Q}}_j + \left(\mathbf{F}_{j+1/2}^{\mathrm{d}} - \mathbf{F}_{j-1/2}^{\mathrm{d}}\right). \tag{21}$$

d) Computation of the dispersive fluxes:

$$\mathbf{F}_{j+1/2}^{\mathrm{ad}} = \mu_{j+1/2}\left(\tilde{\mathbf{Q}}_{j+1} - \tilde{\mathbf{Q}}_{j}\right). \tag{22}$$

in which $\mu_{j+1/2}$ is the dispersive coefficient.

e) Computation of the first difference between the conservative variables, $\mathbf{Q}_j^{\mathrm{d}}$:

$$\Delta\mathbf{Q}_{j+1/2}^{\mathrm{d}} = \mathbf{Q}_{j+1}^{\mathrm{d}} - \mathbf{Q}_{j}^{\mathrm{d}}. \tag{23}$$

f) Limitation of the dispersive fluxes:

$$\mathbf{F}_{j+1/2}^{\mathrm{cad}} = S\max\left[0,\quad \min\left[S\Delta\mathbf{Q}_{j-1/2}^{\mathrm{d}},\quad \left|\mathbf{F}_{j+1/2}^{\mathrm{ad}}\right|,\quad S\Delta\mathbf{Q}_{j+3/2}^{\mathrm{d}}\right]\right], \tag{24}$$

where,

$$S = \operatorname{sgn} \mathbf{F}^{\text{ad}}_{j+1/2} \quad . \tag{25}$$

g) Computation of the inter-cell fluxes $\mathbf{F}^{n}_{j+1/2}$:

$$\mathbf{F}^{n}_{j+1/2} = \hat{\mathbf{F}}^{\text{FCT}}_{j+1/2} = \hat{\mathbf{F}}^{\text{RI}}_{j+1/2} + \frac{\Delta x}{\Delta t^n}\left(\mathbf{F}^{\text{cad}}_{j+1/2} - \mathbf{F}^{\text{d}}_{j+1/2}\right). \tag{26}$$

The diffusive and dispersive coefficients are estimated in many ways in the literature. The objective of this values is to minimize the dispersive and the diffusive effects in the simulation, as presented by Boris and Book (1976). In this work, these coefficients are calculated according to the correlation presented in Fletcher (1998), as follows:

$$\nu_{j+1/2} = \eta_0 + \eta_1\left(\frac{\Delta t}{\Delta x}\lambda_{\max, j+1/2}\right)^2, \tag{27}$$

$$\mu_{j+1/2} = \eta_0 + \eta_2\left(\frac{\Delta t}{\Delta x}\lambda_{\max, j+1/2}\right)^2, \tag{28}$$

where η_0, η_1 and η_2 are constant that are typically $\eta_0 = 1/3$, $\eta_1 = 1/6$ and $\eta_2 = -1/6$ in order to minimize the diffusive and dispersive influences. The value of $\lambda_{\max, j+1/2}$ is calculated based on the arithmetic average of the its value in the cells located upstream and downstream, as presented below:

$$\lambda_{\max, j+1/2} = 0,5\left(\lambda_{\max, j} + \lambda_{\max, j+1}\right) \tag{29}$$

3.2 Discretization of the non-conservative

For the discretization of the non-conservative term that appears in Eq.(13), $\mathbf{H}\,\partial\mathbf{W}/\partial x$, a particular numerical technique is used. This numerical technique is a second-order minmod scheme that was proposed by Harten (1989), showing the discretization as:

$$\left(\mathbf{H}\frac{\partial \mathbf{W}}{\partial x}\right)^n_j = \frac{\mathbf{H}^n_j}{\Delta x} m(\mathbf{x}, \mathbf{y}, \mathbf{z}), \tag{30}$$

where $\mathbf{H}^n_j = \mathbf{H}(\mathbf{W}^n_j)$ and the $m(\mathbf{x}, \mathbf{y}, \mathbf{z})$ is the minmod function defined as

$$m(\mathbf{x}, \mathbf{y}, \mathbf{z}) \equiv \begin{cases} s \min\left\{|\mathbf{x}|, |\mathbf{y}|, |\mathbf{z}|\right\}, & \text{if } \operatorname{sgn}(\mathbf{x}) = \operatorname{sgn}(\mathbf{y}) = \operatorname{sgn}(\mathbf{z}) = s \\ 0, & \text{otherwise} \end{cases}, \tag{31}$$

with

$$\mathbf{x} \equiv 2(\mathbf{W}^n_{j+1} - \mathbf{W}^n_j), \quad \mathbf{y} \equiv \frac{1}{2}(\mathbf{W}^n_{j+1} - \mathbf{W}^n_{j-1}), \quad \mathbf{z} \equiv 2(\mathbf{W}^n_j - \mathbf{W}^n_{j-1}) \ . \tag{32}$$

It is worth mentioning that the calculations performed with $m(\mathbf{x}, \mathbf{y}, \mathbf{z})$ and s are to be interpreted component-wise.

4 THE WATER FAUCET PROBLEM

The gravity dominated flow described by Ransom (1987) is one of the most common cases used as numerical benchmark for two-phase flow simulations. This case was chosen due to its well-known analytical solution that can be used to validate the simulation and also for having an increasing discontinuity with time, until it reaches the steady state. This problem can be used to check the capability of the method to capture discontinuities in two-phase flows. The water faucet problem was simulated in previous works, such as Trapp and Riemke (1986), Quin (1992), Coquel *et al.* (1997), Saurel and Abgrall (1999), Paillère *et al.* (2003), Evje and Flåtten (2003), Essama (2004), Guillard and Duval (2007), Munkejord *et al.* (2009), Munkejord (2007, 2010) and Ansari and Daramizadeh (2012)

It is an initial-boundary value problem that can be interpreted initially as a uniform flow field with no influence from gravity, where the gas phase is static and the liquid phase has a constant velocity along the domain. The influence of gravity is then introduced, as the angle $\beta = \pi/2$ (Eq. (3) and (4)), and the flow field starts evolving. For this benchmark, the drag forces at the interface and at the wall are neglected. The boundary conditions are set equal to the initial condition for all variables, in a way that the velocities and void fraction are set at the inlet and pressure is set at the outlet. The analytical solution, according to Coquel *et al.* (1997) is:

$$\alpha_G(x,t)=\begin{cases}1-\dfrac{(\alpha_G u_L)_{x=0}}{\sqrt{2gx+(u_L^2)_{x=0}}}, & \text{se } x\le (u_L)_{x=0}+\dfrac{1}{2}gt^2\\ 1-(\alpha_L)_{x=0}, & \text{otherwise.}\end{cases}, \tag{33}$$

$$u_L(x,t)=\begin{cases}\sqrt{(u_L^2)_{x=0}+2gx}, & \text{se } x\le (u_L)_{x=0}+\dfrac{1}{2}gt^2\\ (u_L)_{x=0}+gt, & \text{otherwise.}\end{cases}. \tag{34}$$

The analytical solution presumes that the pressure is constant along the domain throughout the time. Figure 1 shows the evolution of the water faucet problem with time.

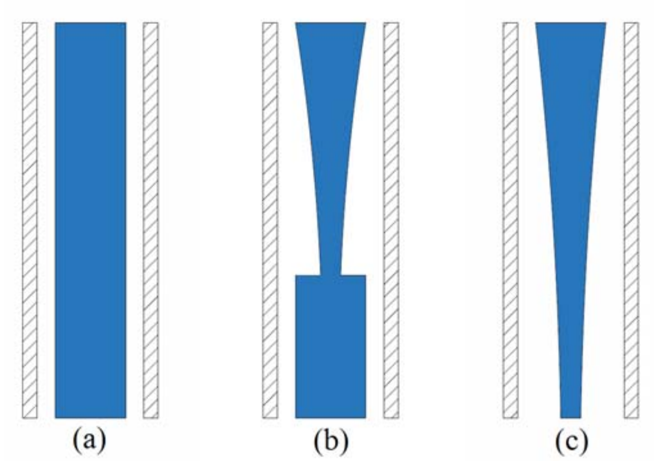

Figure 1. Representation of the water faucet problem evolution with time (a) Initial uniform flow field, (b) Flow field at a given time $t = t_o+\Delta t$, (c) Steady state.

5 RESULTS AND DISCUSSION

A numerical simulation of the water-faucet problem is carried out using the Flux-Corrected Transport method. For this simulation, a 2500 cells grid is implemented with a *CFL* value of 0.45. The results obtained represent the numerical solution in comparison with the analytical one for time t=0.6s. The parameters used for the simulation are presented in the following Table 1. Table 2 shows the initial and boundary conditions of the uniform flow field.

Table 1. Constants values for the water faucet test case.

Constants	Value
g (m/s²) - gravity	9.81
T (ºC) - Temperature	273
ρ_L (kg/m³) - Liquid density	1000
γ_G - specific heat ratio	1.4
R (J/kg K) - gas constant	287
L (m) - pipe length	12

Table 2. Constants values for the water faucet test case.

Variable	Boundary Condition Position	Value
p (Pa) - Pressure	Outlet	10^5
u_G (m/s) - Gas velocity	Inlet	0
u_L (m/s) - Liq. Velocity	Inlet	10
α_G (-) - Void Fraction	Inlet	0.2

The following graphs represent the solution obtained with FCT. Figure 2 shows the gas volume fraction distribution along the pipe plotted against the analytical solution. The comparison shows that the method was capable of capturing the discontinuity and the volume fraction profile correctly. Figure 3 shows the liquid velocity distribution for the simulation, plotted against the analytical solution. Figure 4 shows the gas velocity and Figure 5 the pressure distributions for the simulation. The numerical results for liquid velocity present an excellent agreement with the analytical solution, and the gas velocity shows the correct physical behavior. There is no analytical solution available for the gas velocity and pressure to be used for comparison.

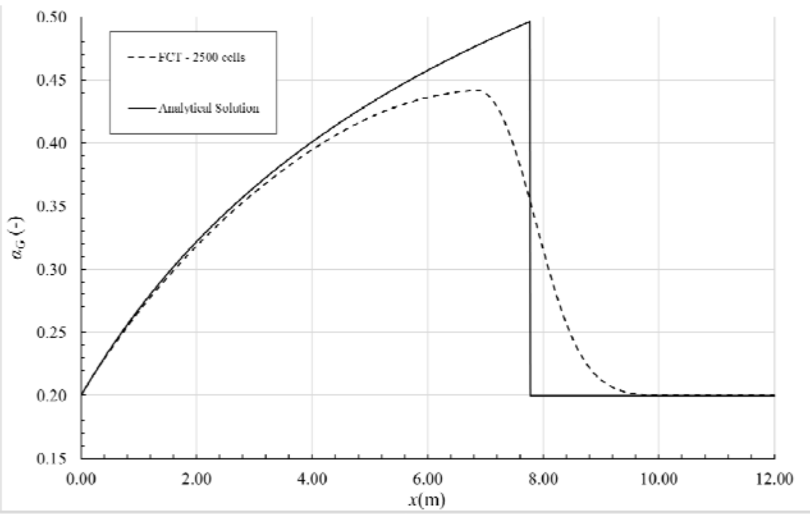

Figure 2. Gas volume fraction distribution.

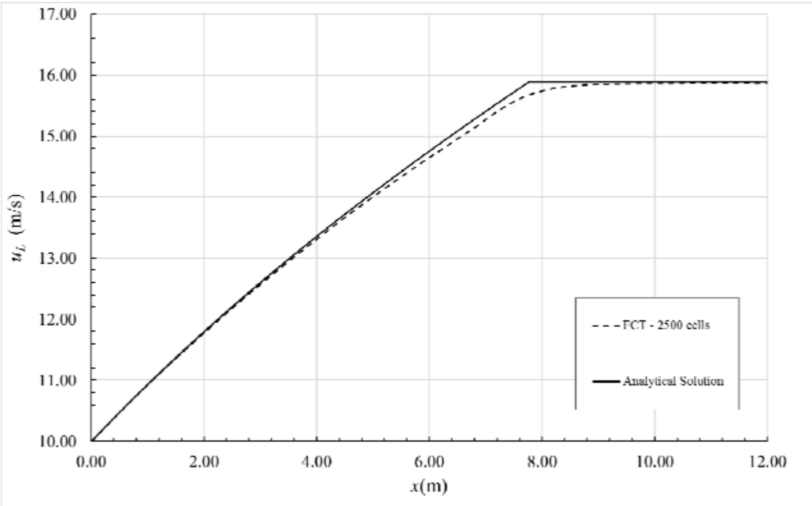

Figure 3. Liquid velocity distribution

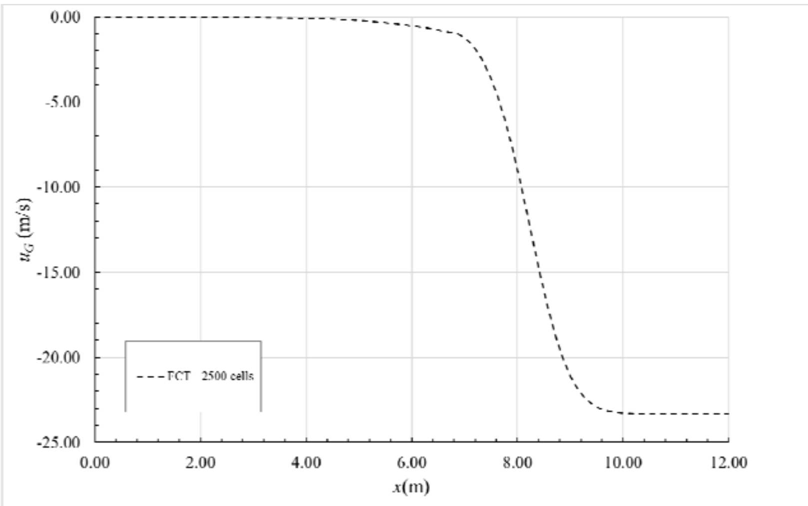

Figure 4. Gas velocity distribution.

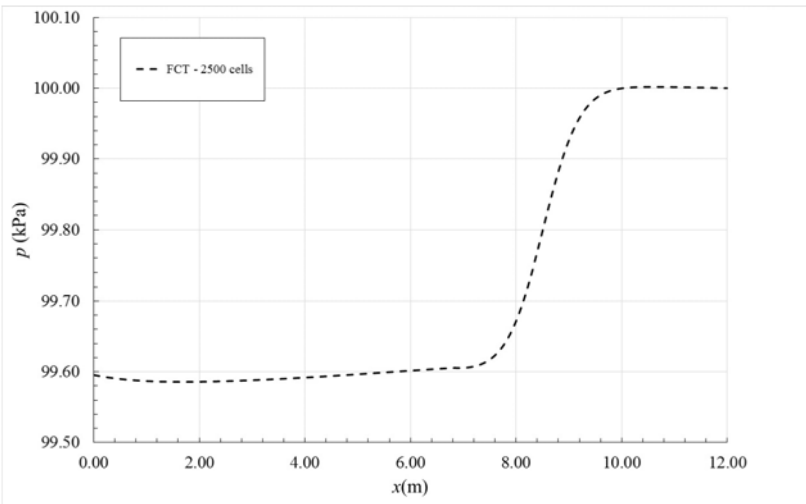

Figure 5. Pressure distribution.

The results obtained from the comparison of the FCT with the analytical solution for the water faucet problem show that the numerical method is able to predict the analytical solution used for validation. In the graph of the variation of void fraction along the domain the discontinuity presented in the solution was captured by the numerical method. The pressure distribution presents small variations, as expected for this case. In addition, the gas velocity assumes a negative value, which was also expected, due to the fact that the gas phase must flow from the outlet towards the inlet, until it reaches the position where the void fraction discontinuity is present, respecting the principles of conservation of mass.

6 CONCLUSIONS

Simulation of the water faucet two-phase flow problem was performed employing a one-dimensional two-fluid model. The results obtained indicate that the coupling of the flour-equation single-pressure model with the FCT method to simulate this problem produces accurate results. The implementation of the Bestion (1990) pressure correction term was crucial for this particular case, placing the system of equations in a hyperbolic region. The results show good agreement between the FCT simulation and the analytical solution, indicating that the numerical technique is efficient in predicting discontinuities present in the solution. The results are encouraging, showing that the numerical model can deal with sharp gradients and discontinuities giving support for the results from the previous works of Patricio (2016) and Figueiredo *et al.* (2017), that focus their simulations on pipeline cases.

7 ACKNOWLEDGEMENTS

The authors would like to thank CAPES and PETROBRAS S.A. for the financial support of this research project. The authors would also like to acknowledge CNPq and FAPERJ, research sponsoring agencies of the Brazilian and Rio de Janeiro State governments, for the continuous support of all research activities of this group over the years.

8 REFERENCES

Ansari, M.R., Daramizadeh, A., 2012, "Slug type hydrodynamic instability analysis using a five equations hyperbolic two-pressure, two-fluid model", *Ocean Engineering*, v. 52, pp. 1-12;

Bestion, D., 1990, "The physical closure laws in the CATHARE code", *Nuclear Engineering* and *Design*, Vol.124, pp. 229–245;

Book, D.L., Boris, J.P., Hain, K., 1975. "Flux corrected transport II. - Generalizations of the method Private Communication". *Journal of Computational Physics*, , Vol. 18, p. 248-283;

Boris, J.P., Book, D.L., 1973. "Flux correct transport I - SHASTA - A fluid transport algorithm that works". *Journal of Computational Physics,* Vol. 11, p. 38 – 69.

Boris, J.P., Book, D.L., 1976. "Solution of continuity equation by the method of flux-corrected transport". *Journal of Computational Physic*, Vol. 129, p. 85 - 129.

Coquel, F., El Amine, K., Godlewski, E., Perthame, B., Rascle, P., 1997, "A numerical method using upwind schemes for the resolution of two-phase flows", *Journal of Computational Phys*ics, v. 136, pp. 272-288;

Essama, C., 2004. *Numerical Modelling of Transient Gas-Liquid Flows (Application to Stratified & Slug Flow Regimes)*. Ph.D. thesis, Cranfield University, United Kingdom.

Evje, S., Flåtten, T., 2003, "Hybrid flux-splitting schemes for a common two-fluid model", *Journal of Computational Physics*, v.192, pp. 175–210;

Figueiredo, A.B., Baptista, R.M., Rachid, F.B.F., Bodstein, G.C.R., 2017, "Numerical simulation of stratified-pattern two-phase flow in gas pipelines using a two-fluid model", *International Journal of Multiphase Flow*, v. 88, pp. 30-49;

Fletcher, C.A. J., 1988, *Computational Techniques for Fluid Dynamics: Specific techniques for Flow Categories*, Volume 2. Springer-Verlag;

Guillard, H., DuvaL, F., 2007, "A Darcy law for the drift velocity in a two-phase flow model", *Journal of Computational Physics*, v.224, pp. 288–313;

Harten, A., 1983, "High resolution schemes for hyperbolic conservation laws", *Journal of Computational Physics*, v. 49, pp. 357-393;

Munkejord, S.T., 2007, "Comparison of Roe-type methods for solving the two-fluid model with and without pressure relaxation", *Computer & Fluids*, v. 36, pp. 1061–1080;

Munkejord, S.T., 2010, "A numerical study of two-fluid models with pressure and velocity relaxation", *Advances in Applied Mathematics and Mechanics*, v. 2, pp. 131-159;

Munkejord, S.T., Evje, S., FlåtteN, T., 2009, "A MUSTA scheme for a non-conservative two-fluid model", *SIAM Journal on Scientific Computing*, v.31, pp. 2587-2622.

Paillère, H., Corre, C., García CascaleS, J.R., 2003, "On the extension of the AUSM+ scheme to compressible two-fluid models", *Computers & Fluids*, v.32, pp. 891–916;

Patricio, R. A. C., 2016, *Simulação de escoamento bifásico em gasodutos na presença de um pig utilizando o método numérico FCT*, M.Sc. Dissertation, COPPE/UFRJ, Rio de Janeiro, RJ, Brasil;

Quin, H.Q., 1992, "Comparisons of predictions with numerical benchmark test no2.1: Faucet flow", *Multiphase Science and Technology*, v. 6, pp. 577-590;

Ransom, V.H., 1987, "Numerical benchmark test no.2.1: Faucet flow", *Multiphase Science and Technology*, v. 3, pp. 465-467;

Saurel, R., Abgrall, R., 1999, "A Multiphase Godunov Method for Compressible Multifluid and Multiphase Flows", *Journal of ComputationalPhysics*, v. 150, pp. 425–467;

Sondermann, C.N., Patricio, R.A.C, Figueiredo, A.B., Baptista, R.M., Rachid, F.B.F., Bodstein, G.C.R., 2015, "Hyperbolicity analysis of a two- fluid two-phase flow one-dimensional model for stratified pattern", *Proceedings of the 2015 International Mechanical Engineering Congress & Exposition*, Houston, Texas, USA, November;

Toro E.F., 1999, *Riemann Solvers and Numerical Methods for Fluid Dynamics*. Springer;

Trapp, J.A., RiemkE, R.A., 1986, "A nearly-implicit hydrodynamic numerical scheme for two-phase flows", *Journal of Computational Physics*, v.66, pp. 62–82;

9 RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.