23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

NUMERICAL SIMULATION OF THE GAS FLOW IN PIPELINES USING THE FLUX-CORRECTED TRANSPORT METHOD

Carina N. Sondermann
Dept. of Mechanical Engineering, Federal University of Rio de Janeiro (UFRJ – COPPE/PEM)
Av. Athos da Silveira Ramos, 149, Centro de Tecnologia - Cidade Universitária - Rio de Janeiro - RJ - Brasil
carinans@poli.ufrj.br

Rodrigo A. C. Patrício
Dept. of Mechanical Engineering, Federal University of Rio de Janeiro (UFRJ – COPPE/PEM)
Av. Athos da Silveira Ramos, 149, Centro de Tecnologia - Cidade Universitária - Rio de Janeiro - RJ - Brasil
rac.patricio@mecanica.coppe.ufrj.br

Aline B. Figueiredo
Dept. of Mechanical Engineering, Federal University of Rio de Janeiro (UFRJ – COPPE/PEM)
Av. Athos da Silveira Ramos, 149, Centro de Tecnologia - Cidade Universitária - Rio de Janeiro - RJ - Brasil
aline.figueiredo13@gmail.com

Renan M. Baptista
Research and Development Center, PETROBRAS
Av. Horácio Macedo, 950 – Cidade Universitária - Rio de Janeiro – RJ - Brasil
renan.baptista@gmail.com

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
rachid@vm.uff.br

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
gustavo@mecanica.coppe.ufrj.br

Abstract. *The operation of gas pipelines is a very important task, particularly in the oil and chemical industries. Pipeline operations for the transport of gases are usually subject to specific events, such as the opening and closing of valves, the passage of PIGs in the interior of the pipeline and the occurrence of leaks in the system. Because these gas-pipeline flows are very complex, it is necessary to develop a reliable numerical code that calculates the basic flow accurately, under transient and steady-state conditions. Bearing in mind future applications to two-phase flows, we describe in this paper a one-dimensional mathematical model and its associated numerical method to calculate the transient gas flow inside a pipeline. The model is discretized within the finite-volume framework and is solved using the Flux-Corrected Transport (FCT) method, which is second-order in space, coupled to a first-order explicit discretization in time. We formulate the problem as an initial-boundary value problem of the hyperbolic type. In order to validate our model, we simulate the Fanno line and the shock tube flows. Our numerical simulations show results for the pressure, velocity and temperature distributions along the pipe that compare very well to the exact solutions available for these flows.*

Keywords: *gas-flow pipelines, transient flow in pipelines, flux-corrected transport method, numerical simulation, petroleum engineering*

1. INTRODUCTION

Currently, the main interest in studying pipeline flows is related to the oil and natural gas industries. These natural resources are important sources of energy and, particularly, natural gas is a widely used fuel in thermoelectric power plants. In Brazil, most of the electric power generation comes from hydroelectric power plants. However, when the energy demand is higher than production, it is necessary to activate the thermoelectric power plants, especially in periods of shortage of rainfall. These plants continually require the supply of natural gas, which is used as fuel for the gas turbines. Furthermore, the necessary amount of natural gas is provided through pipelines, not requiring the construction of warehouses for storage and other transportation systems.

In order to monitor the flow in gas pipelines in real time and detect transient events, such as gas leak and *PIG* passage, for example, there must be a numerical model able to describe their disposal at any given time so that one can diagnose or predict problems that are present in the product line. Many papers discuss this topic in the literature, and some of them consider that the isothermal model is ideal to describe such flows. However, Osiadacz and Chaczykowski (2001) conducted a numerical study of gas flows in a pipeline comparing an isothermal case with a non-isothermal case. In the non-isothermal case, the temperature distribution is calculated using a mathematical model which includes the energy equation. With this comparison, it was found that there is a non-negligible difference in the pressure profile for the two types of flows.

Abbaspour and Chapman (2008) also studied a non-isothermal flow and performed simulations of one-dimensional compressible flows in pipelines. For convenience, mass, momentum and energy equations were written as a function of the mass flow rate and an implicit finite-difference method was used to find the solution. Their results showed that considering the temperature variation within the framework of a non-isothermal model is very important to process fast transient due to the variation of the gas density with temperature, which directly affects how the steady-state regime is reached.

In this paper, we perform numerical simulations of gas flows in pipelines using a non-isothermal one-dimensional hyperbolic mathematical model, solved with the Flux-Corrected Transport method (Boris and Book, 1973), to calculate the pressure, velocity and temperature profiles along the pipeline. We simulate the Fanno line and the shock tube flows to validate our model, as discussed below.

2. MATHEMATICAL EQUATIONS

We assume that the flow in the gas pipeline is one-dimensional and time-dependent. The cross-section area of the pipe is considered constant and axial heat-conductive effects are neglected. The gas flow is governed by the mass, momentum and energy equations, which may be written as a system of hyperbolic partial differential equations (PDE) according to

$$\frac{\partial(\rho)}{\partial t}+\frac{\partial(\rho u)}{\partial x}=0, \tag{1}$$

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2+p)}{\partial x}=-\rho g\sin(\beta)-\frac{\tau_w S}{A}, \tag{2}$$

$$\frac{\partial(\rho(e+u^2/2))}{\partial t}+\frac{\partial(\rho u(e+u^2/2)+pu)}{\partial x}=q''_w\frac{S}{A}-\rho ug\sin(\beta). \tag{3}$$

In Eqs. (1)-(3), the axial coordinate along the pipe is x, whereas ρ, p, u and e are the density, pressure, velocity and thermal internal energy of the gas. The gas density ρ is related to the pressure p and temperature T by the ideal-gas law, $p = \rho RT$, whereas e depends on T through $e = c_v T$, where R is the gas constant and c_v is the constant-volume specific heat. On the right-hand side of Eqs. (2) and (3), g is the gravity acceleration, β is the pipeline inclination, S is the pipe's perimeter, τ_w is the shear stress on the pipe wall, A is the pipe's cross-section area, and q''_w is the heat flux related to convection through the lateral area of the pipe. In this paper, we consider the pipe horizontal and the flow to be adiabatic, which means that the pipeline inclination and the heat-flux terms are equal to zero.

We assume that the terms on the right-hand side of Eqs. (2) and (3) are given by:

$$\tau_w=\frac{1}{2}f\rho u|u|, \tag{4}$$

$$f=\max\left[\frac{16}{\mathrm{Re}},\frac{0.046}{\mathrm{Re}^{0.2}}\right], \tag{5a}$$

$$\mathrm{Re}=\frac{\rho D|u|}{\mu}. \tag{5b}$$

where f is the wall friction factor, proposed by Taitel and Dukler (1976), Re is the Reynolds number, D is the pipe's diameter (Hirsh, 1990), and μ is the gas dynamic viscosity.

2.1 Initial and boundary conditions

The set of Eqs. (1) – (5) form a closed system of equations, with u, p and T as unknowns. This system is subject to the following boundary and initial conditions:

- Boundary conditions:
  Pipeline inlet: u and T
  Pipeline outlet: p.

- Initial conditions:
  Values of u, p and T set at the boundaries are prescribed throughout the domain at $t = 0$.

3. NUMERICAL METHOD

The FCT is a conservative method of second order in space based on the conservative discretization of the flux terms proposed by Boris and Book (1973), Book *et al.* (1975) and Lezeau and Thompson (1998). This method is used for solving systems of hyperbolic equations, such as those involving shock waves and discontinuities.

3.1 Hiperbolic system of PDE´s

Equations (1), (2) and (3) comprise a hyperbolic system of Partial Differential Equations (PDE's) that is here discretized using the finite difference method. This type of system may be written in the form

$$\frac{\partial \mathbf{Q}}{\partial t}+\frac{\partial \mathbf{F}(\mathbf{Q})}{\partial x}=\mathbf{S}(\mathbf{Q}), \tag{6}$$

with $(x,t)\in(0,L)\times(0,\infty)$.

Equation (6) is a general representation of the hyperbolic system of PDE's. For our model, the non-linear system defined by Eqs. (1), (2) and (3) may be rewritten in the form of Eq. (6), where the vectors **Q**, **F**, **S** are identified as

$$\mathbf{Q}\equiv\begin{bmatrix}\rho \\ \rho u \\ \rho\left(e+u^2/2\right)\end{bmatrix}, \qquad \mathbf{F}\equiv\begin{bmatrix}\rho u \\ \rho u^2+p \\ \rho u\left(e+u^2/2+p/\rho\right)\end{bmatrix}, \qquad \mathbf{S}\equiv\begin{bmatrix}0 \\ -\rho g\sin(\beta)-(\tau_w S)/A \\ (q''_w S)/A-\rho g\sin(\beta)u\end{bmatrix}. \tag{7}$$

3.2 Numerical Discretization

The pipe length is discretized into M computational cells of regular size $\Delta x \equiv L/M = x_{j+1/2} - x_{j-1/2}$, with $x_{j-1/2}=(j-1)\Delta x$, $x_{j+1/2} = j\Delta x$ and $j = 1,\ldots,M$, so that the center of the cell is positioned at $x_j = (j - 1/2)\Delta x$. A general finite-difference approximation to Eq. (6) may be written as

$$\mathbf{Q}_j^{n+1}=\mathbf{Q}_j^n-\gamma\left[\hat{\mathbf{F}}_{j+1/2}-\hat{\mathbf{F}}_{j-1/2}\right]+\Delta t^n \mathbf{S}_j^n, \tag{8}$$

where $\gamma \equiv \Delta t^n/\Delta x$, Δt^n is the time step (considered constant), Δx is the mesh spacing (also considered constant) and $\hat{\mathbf{F}}$ the numerical flux vector. The superscript n and the subscript j refer to the time and space discretization, respectively. Equation (8) provides the value of the vector **Q** at time $t^{n+1}=t^n+\Delta t^n$ evaluated at cell j.

The stepping in time used in Eq. (8) to evolve the flow quantities is explicit. Therefore, the time step Δt is chosen based on the *CFL* (Courant-Friedrichs-Lewy) condition (Hirsh, 1990) which is given by:

$$\Delta t^n=\mathrm{CFL}\frac{\Delta x}{\lambda_{\max}^n}, \tag{9}$$

where *CFL* is a positive number that is, usually, less than or equal to one. The closer to unity, the more efficient the numerical scheme is in terms of stability. For the FCT scheme, Sod (1985) has shown that the CFL value should be less than 0.5. Equation (9) defines the maximum time step that keeps the method stable. The parameter $\lambda_{\max}^n$ is the largest eigenvalue in the flow domain at time level n, and expresses, physically, the largest speed at which pressure waves propagate in the flow. This eigenvalue is numerically close to the speed of sound. For a system of N equations and M mesh cells, $\lambda_{\max}^n$ is given by

$$\lambda_{\max}^{n} = \max_{j}\left[\max_{k}\left|\lambda_{j}^{k}\right|\right], \text{ for } j = 1,\ldots,M \text{ and } k = 1,\ldots,N \cdot \tag{10}$$

3.3 Discretization of the conservative flux

A number of conservative numerical methods of different orders of accuracy (Hirsch, 1990) may be used to calculate the flux term in Eq. (8). This paper is focused on the Flux-Corrected Transport (FCT) technique, which was originally proposed by Boris and Book (1973). This method was further developed by Book *et al.* (1975) and Boris and Book (1976), and later generalized by Zalesak (1979). The FCT technique was the first high-resolution scheme to introduce the concept of limiters in the literature. It may be interpreted as a "predictor/corrector" scheme in which diffusion is introduced in the predictor stage and anti-diffusion is introduced in the corrector stage. The FCT algorithm conserves mass and maintains the positivity of the actual mass and energy densities, such that steep gradients and inviscid shocks are captured accurately (Boris and Book, 1973). The method is second order in space and produces accurate results near sharp gradients and discontinuities, although it is of indeterminate order in these regions (Boris and Book, 1973).

In the context of transient two-phase flows in pipes, Figueiredo *et al.* (2012) investigated the time and space accuracy of a discretization of the type of Eq. (8) using the FCT method and showed that it is first-order in time and second-order in space. Recently, Bueno *et al.* (2014) employed this same formulation to simulate the two-phase flow in a pipeline with a leak and obtained very good results.

Based on Fletcher (1988), the flux and state vectors are given as follows:

- <u>Calculation of the first approximation $\tilde{\mathbf{Q}}_j^*$ from $\mathbf{Q}_j^{n+1}$ generated by the Ritchmyer scheme:</u>

$$\tilde{\mathbf{Q}}_j^* = \mathbf{Q}_j^n - \frac{\Delta t^n}{\Delta x}\left(\hat{\mathbf{F}}_{j+1/2}^{\mathrm{RI}} - \hat{\mathbf{F}}_{j-1/2}^{\mathrm{RI}}\right), \tag{11}$$

where

$$\hat{\mathbf{F}}_{j+1/2}^{\mathrm{RI}} = \mathbf{F}(\bar{\mathbf{Q}}_{j+1/2}^*), \tag{12}$$

and

$$\bar{\mathbf{Q}}_{j+1/2}^* = \frac{1}{2}\left(\mathbf{Q}_{j+1}^n + \mathbf{Q}_j^n\right) - \frac{1}{2}\frac{\Delta t^n}{\Delta x}\left[\mathbf{F}(\mathbf{Q}_{j+1}^n) - \mathbf{F}(\mathbf{Q}_j^n)\right]; \tag{13}$$

- <u>Generation of diffusive fluxes</u>:

$$\mathbf{F}_{j+1/2}^{d} = \nu_{j+1/2}\left(\mathbf{Q}_{j+1}^n - \mathbf{Q}_j^n\right); \tag{14}$$

- <u>Diffusion of the solution</u>:

$$\mathbf{Q}_j^d = \tilde{\mathbf{Q}}_j + \left(\mathbf{F}_{j+1/2}^d - \mathbf{F}_{j-1/2}^d\right); \tag{15}$$

- <u>Generation of anti-diffusive fluxes</u>:

$$\mathbf{F}_{j+1/2}^{ad} = \mu_{j+1/2}\left(\tilde{\mathbf{Q}}_{j+1} - \tilde{\mathbf{Q}}_j\right); \tag{16}$$

- <u>Limitation of the anti-diffusive fluxes</u>:

$$S = \mathrm{sign}(\mathbf{F}_{j+1/2}^{ad}); \tag{17}$$

where

$$\mathbf{F}_{j+1/2}^{cad} = S \cdot \max\left[0, \min\left(S\left(\tilde{\mathbf{Q}}_{j+1} - \mathbf{Q}_j^d\right), \left|\mathbf{F}_{j+1/2}^{ad}\right|, S\left(\tilde{\mathbf{Q}}_{j+2} - \mathbf{Q}_{j+1}^d\right)\right)\right]; \tag{18}$$

- <u>Generation of inter-cell flux</u>:

$$\hat{\mathbf{F}}_{j+1/2}^{FCT} = \hat{\mathbf{F}}_{j+1/2}^{\mathrm{RI}} + \frac{\Delta x}{\Delta t^n}\left(\mathbf{F}_{j+1/2}^{\mathrm{cad}} - \mathbf{F}_{j+1/2}^{\mathrm{d}}\right) \cdot \tag{19}$$

In Eq. (14) and (16) ν and μ are diffusion and anti-diffusion coefficients, respectively. We assume, in our algorithm, that ν and μ are constant and equal to 0.125, as adopted by Omgba-Essama (2004).

4. RESULTS AND DISCUSSION

In order to demonstrate that the results obtained through our simulations with the FCT method are accurate, two classical cases from the literature were chosen: the Fanno Line flow and the Shock Tube flow.

4.1 The Fanno Line Flow

The Fanno Line flow refers to an adiabatic flow of a compressible gas through a constant area duct where the effect of friction is considered. Knowing the initial conditions of a gas at the first section of a control volume it is possible to determine the condition at the second section through a balance of mass, momentum and energy. For this kind of flow that is represented by a Fanno Line, the point of highest entropy is where the flow is sonic, that is, the point where the Mach number is equal to one.

To determine the Mach number at each point of the duct we can relate the Mach number, M, with the length, L, of the duct according to (Fox *et al.*, 2010)

$$\int_0^{L_{max}} \frac{4f}{D} dx = \int_M^1 \frac{1-M^2}{\gamma M^4 \left[1+(\gamma-1)/(2M^2)\right]} d(M^2) \tag{20}$$

where γ is the ratio of specific heats, f is the friction factor and D is the diameter of the duct.

Performing the integration in Eq. (20), we obtain

$$\frac{4fL_{max}}{D} = \frac{1-M^2}{\gamma M^2} + \frac{\gamma+1}{2\gamma} \ln\left[\frac{(\gamma+1)M^2/2}{1+(\gamma-1)M^2/2}\right], \tag{21}$$

which results in

$$\frac{4fL}{D} = \left(\frac{4fL_{max}}{D}\right)_{M_1} - \left(\frac{4fL_{max}}{D}\right)_{M_2} . \tag{22}$$

Considering that the conditions with (*) represents the conditions where the Mach number is equal to one, the parameters charactering the flow are expressed as

$$\frac{T}{T^*} = \frac{\frac{\gamma+1}{2}}{(1+\frac{\gamma-1}{2}M^2)}, \quad \frac{u}{u^*} = \left[\frac{\frac{\gamma+1}{2}M^2}{(1+\frac{\gamma-1}{2}M^2)}\right]^{1/2}, \quad \frac{\rho}{\rho^*} = \left[\frac{(1+\frac{\gamma-1}{2}M^2)}{\frac{\gamma+1}{2}M^2}\right]^{1/2}, \quad \frac{p}{p^*} = \frac{1}{M}\left[\frac{\frac{\gamma+1}{2}}{(1+\frac{\gamma-1}{2}M^2)}\right]^{1/2}, \tag{23}$$

where T is the temperature, u is the velocity, p is the pressure and ρ is the density. Using Eqs. (22) and (23) it is possible to calculate the analytical solution of this flow and compare it to the numerical results obtained with the simulation using the numerical method FCT.

Table 1 shows the values of the flow properties and duct parameters that were used to calculate the analytical solution. From this input data, we obtained the numerical results represented in Figure 1 to validate our code for the Fanno Line flow. As one can see, our numerical results present very good agreement with the analytical solution.

Table 1. Inlet values for the Fanno Line Flow.

M - inlet	0.3
p (Pa) - inlet	101325
T (K) - inlet	273
u (m/s) - inlet	100
ρ (kg/m³) - inlet	1.3
γ	1.4
R (J/kg K) - gas constant	287
P (Pa) - outlet	63210
Number of cells	1000
D (m)	0,15
L (m)	30
f – friction factor	0.005
Heat flux	adiabatic

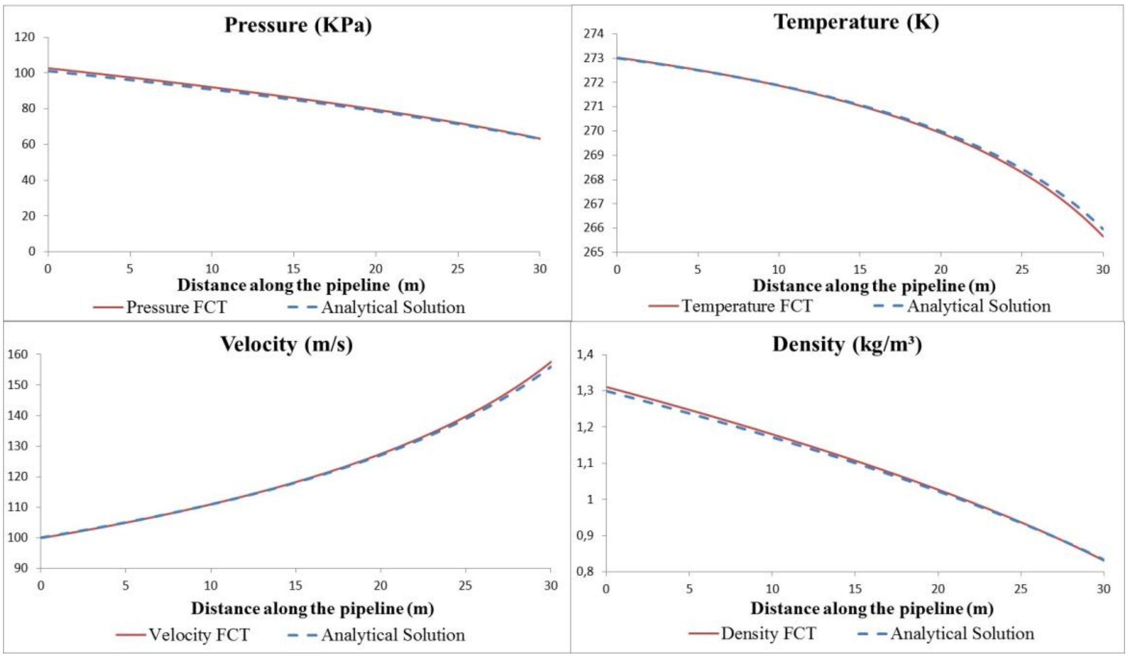

Figure 1. Pressure, temperature, velocity and density variation as a function of the distance along the pipeline after the steady-state is reached for the Fanno Line flow

A study of the numerical relative error due to the number of discretization cells was carried out based on the integration of the mass of gas inside the pipeline in steady-state. Table 2 shows the results of this integration procedure (using the trapezoidal rule) obtained from the numerical solution using FCT compared to the analytical one.

Table 2. Relative errors due to the number of discretization cells.

Number of cells	Error
100	2,37%
1000	0,64%
10000	0,51%

It is evident that the discretization with 100 cells does not produce such an accurate result if compared to the solution obtained with 1000 cells. Furthermore, the simulation with the discretization of 10000 cells presented the smallest relative error, but it took too long to converge, indicating that it is not worth the waist in computational time and memory to refine that much the fluid domain. Thus, based on this study for the Fanno Line flow, we used a discretization of 1000 cells in the validation study with the Shock Tube, which is presented in the next section.

4.2 The Shock Tube Flow

The initial problem of the Shock Tube consists in splitting the domain into two regions using a diaphragm or membrane between them located at a specified position. These two regions are filled with the same fluid, but their temperatures, pressures and densities are different in each domain. At the initial instant of time t = 0, the diaphragm which splits these regions is removed instantaneously, leading to a discontinuity in the fluid properties at the membrane position. The exact solution to this problem was taken from Toro (1999) and compared to the numerical solution obtained from our FCT simulation. Table 3 shows the values that were used to calculate the exact solution.

The results obtained from this validation test for the Shock Tube flow are represented in Figure 2. Again, as one can see, the numerical and exact solutions present an excellent agreement to each other.

According to the results obtained from the validation tests of the FCT method presented in this section, we demonstrated that the method is able to predict and represent accurately the analytical solution for both flows we studied. In addition, the FCT method showed how good it is in determining discontinuities, such as those presented in the Shock Tube flow.

Table 3. Inlet values for the Shock Tube Flow.

Position Diaphragm (m)	5
p (Pa) – left to the diaphragm	10^5
p (Pa) – right to the diaphragm	10^4
ρ (kg/m³) – left to the diaphragm	1
ρ (kg/m³) – right to the diaphragm	0,125
u (m/s) – left to the diaphragm	0
u (m/s) – right to the diaphragm	0
T (K) – left to the diaphragm	348,43
T(K) – right to the diaphragm	278,74
γ	1,4
R (J/kg K) – gas constant	287
L (m)	10
D (m)	1,13
Simulation time (s)	$6{,}1 \times 10^3$
Number of cells	1000
Heat Flux	adiabatic

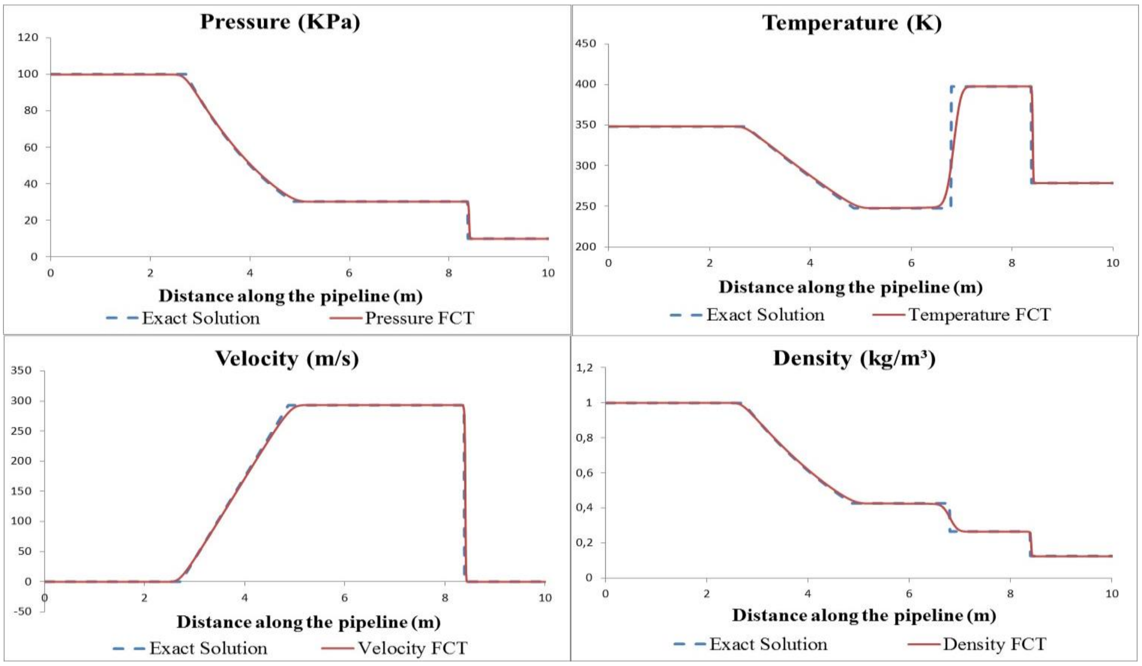

Figure 2. Pressure, temperature, velocity and density as a function of the distance along the pipeline after steady-state is reached for the Shock Tube flow.

5. CONCLUSIONS

Simulations of non-isothermal one-dimensional gas flow in a horizontal pipeline were performed using the *Flux Corrected Transport* numerical method. The model was validated against the Fanno Line and the shock Tube flows because both flows present an exact solution, allowing a precise evaluation of the numerical errors to be carried out. These choices have also practical motivation. The Fanno Line flow is an important test problem for transient flows in ducts and, particularly, it represents the basic flows of many real applications, such as pipeline flows with the passage of a *PIG* and/or with the occurrence of leak. Moreover, the shock tube problem is useful as a simpler representation of flows that present discontinuities in one or more parameters. As an example, we cite the two-phase flow that occurs in gas pipelines that transport a mixture of oil and gas from an offshore platform to a processing unit at the continent, which present discontinuities at the interfaces between the two phases.

Our results indicate that the numerical method is very good in calculating transient flows in ducts, with and without discontinuities, since they reproduce accurately the analytical solutions used for validation.

To conclude, we anticipate that the results presented in this paper will serve, in the near future, as a basis for modeling two-phase flows in pipelines that includes the energy equation. This model, solved with the *Flux Corrected Transport* (FCT) method, will be employed to perform numerical simulation of transient flows in a pipeline with a leak and during a *PIG* passage.

6. ACKNOWLEDGEMENTS

The authors would like to thank CNPq and PETROBRAS S.A. for the financial support of this research project. The authors would also like to acknowledge CAPES 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.

7. REFERENCES

Abbaspour, M., Chapman, K.S., 2008. “Nonisothermal transient flow in natural gas pipeline”. *Journal of Applied Mechanics*, Vol. 75, p. 0310181- 0310188.

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.

Bueno, D.E.G.P., Figueiredo, A.B., Baptista, R.M., Rachid, F.B.F., Bodstein, G.C.R., 2014. “Numerical simulation of stratified two-phase flow in a nearly horizontal gas-liquid pipeline with a leak”. In *Proceedings of the 10th International Pipeline Conference - IPC 2014.* Calgary, Canada.

Figueiredo, A.B., Bueno, D.E.G.P., Baptista, R.M., Rachid, F.B.F., Bodstein, G.C.R., 2012. “*An accuracy study of the flux-corrected transport numerical method applied to transient two-phase flow simulation in gas pipelines*”. In *Proceedings of the 9th International Pipeline Conference - IPC 2012.* Calgary, Canada.

Fletcher, C.A., 1988. *Computational techniques for fluid dynamics: Specific techniques for flow categories*. Springer Verlag, 2nd volume.

Fox, R.W., Pritchard, P.J., McDonald, A.T., 2009. *Introduction to fluid dynamics.* John Wiley & Sons, 7th edition.

Hirsch, C., 1990. *Numerical computation of internal and external flows: fundamentals of numerical discretization*. John Wiley & Sons, 2nd volume.

Incropera, F.P., Dewitt, D.P., Bergman, T.L., Lavine, A.S., 2007. *Fundamentals of heat and mass transfer*. John Wiley & Sons, 6th edition.

Lezeau, P., Thompson, C.P., 1998. *Numerical Simulation of Multi-Phase Flow: Speed, Error Control & Robustness*. Cranfield University Technical Report.

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

Osiadacz, A.J., Chaczykowski, M., 2001. “Comparison of isothermal and non-isothermal pipeline gas flow models”. *Chemical Engineering Journal*, Vol. 81, p. 41 - 51.
Lezeau, P., Thompson, C.P., 1998. *Numerical Simulation of Multi-Phase Flow: Speed, Error Control & Robustness*. Cranfield University Technical Report.
Sod, G.A., 1985. “Numerical Methods in Fluid Dynamics: Initial and Boundary Value Problems”, Cambrigde University Press.

Taitel, Y., Dukler, A.E., 1976. “A Model for predicting flow regime transitions in horizontal and near horizontal gas-liquid flow”. *AIChE Journal*, Vol. 22, p. 47 - 55.

Zalesak, S.T., 1979. “Fully Multidimensional Flux-Corrected Transport Algorithms for Fluids”. *Journal of Computational Physics*, Vol. 31, p. 335 - 362.

8. RESPONSIBILITY NOTICE

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