Three-phase flow (water, oil and gas) in a vertical circular cylindrical duct with leaks: A theoretical study

Wanessa Raphaella Gomes dos Santos[1,a], Enivaldo Santos Barbosa[1,b], Severino Rodrigues de Farias Neto[2], Antonio Gilson Barbosa de Lima[1,c*], Wanderson Mango Paiva Barbosa de Lima[1,d]

[1]Department of Mechanical Engineering, Center of Science and Technology, Federal University of Campina Grande (UFCG), Zip Code 58429-900, Campina Grande-PB, Brazil.

[2]Department of Chemical Engineering, Center of Science and Technology, Federal University of Campina Grande (UFCG) Zip Code 58429-900, Campina Grande-PB, Brazil.

[1,c*]E-mail: gilson@dem.ufcg.edu.br

[1,a]E-mail: wanessa.raphaella@yahoo.com.br

[1,b]E-mail: enivaldo@dem.ufcg.edu.br

[2]E-mail: fariasn@deq.ufcg.edu.br

[1,d]E-mail: wan_mango@hotmail.com

ABSTRACT

This article describes the fluid dynamic behavior of a three-phase flow (water-oil-natural gas) in a vertical pipe with or without leakage. The studied pipe has 8 meters in length, circular cross-section with 25 cm in diameter and a leak, which hole has a circular shape with 10mm diameter located in the center of pipe. The conservation equations of mass, momentum and energy for each phase (continuous phase - oil, dispersed phases - gas and water) were numerically solved using ANSYS CFX software, in which the Eulerian-Eulerian model and the RNG - turbulence model were applied. Results of the pressure, velocity, temperature and volume fraction distributions of the involved phases are present and analyzed.

Keywords: Three-phase flow, Leakage, Vertical duct, Simulation, Heat

1. INTRODUCTION

Pipelines have been considered one of the best alternatives for the transport of fluids, being a cheap and efficient form of transportation. Despite of this advantage, these products are subject to operational accidents caused by a sudden change in pressure, fractures or corrosion, consequently bringing environmental risks, and financial and social damage.

Leakage monitoring systems in oil pipeline must be capable to detect and locate a leak quickly, reducing risks of environmental and socioeconomics impacts, contributing significantly to operational safety and cost saving in petroleum industry. When a leak occurs, the pressure wave will propagate through the pipeline depending on the position and shape of the leak. For an efficient detection, we must determine and understand the hydrodynamics of flow inside the duct in each instant of time after the occurrence of leak.

With the aim to meet the hydrodynamic behavior of fluids, some works related to multiphase flow with or without leak are being developed being analyzed different fluids, geometries and parameters flow [1–8].

Sarmento et al. [2] report numerical study of the three-phase flow coupled with heat transfer in a curved riser by using CFX 11.0. Different cases were studied, varying the volume fraction of water and gas, maintaining constant oil volume fraction and feeding velocity. It was concluded that the pressure drop in the flow using larger volume fraction of gas is lower than when using lower volume fraction of gas due to the lower density of this phase. It was verified yet that an increase in the volume fraction of gas increases the heat transfer between fluid phases resulting in reduction of difference between the temperatures of the phases.

Cavalcanti et al. [3] analyzed the non–isothermal three-phase flow of petroleum, gas and water in T and Y junctions. For both geometries was maintained constant diameter and length, volume fraction of the phases ($f_{oil} = 0.7$, $f_{water} = 0.2$, $f_{gas} = 0.1$), inlet velocity ($v_{inlet} = 0.03$m/s) and inlet temperature (T = 333K). It was concluded that the software CFX 10.0 was efficient to describe the behavior of flow for the studied geometries and, the pressure drop in the output was similar for both joints.

Barbosa et al. [4] have analyzed the non-isothermal two-phase flow (heavy-oil and water) in a vertical circular duct with leakage (d_{leak} = 6 mm) located at the center of the pipe. The authors verified that the presence of a hole causes large variations in velocity and temperature of fluids inside the duct.

Sousa et al. [5] have realized a numerical analysis of the heavy-oil/water flow and leak detection in a vertical pipe considering effects of gravity and drag forces between the phases for different inlet velocity and volume fraction of the phases. According to the authors it is very difficulty detecting small leaks and oscillations in the total pressure drop is greater as higher volume fraction of water in the mixture is used.

Araújo et al. [6] report a numerical study of the oil flow in T junction with one or two leaks. The geometry was divided into 3 sections: A - main pipe before the branch, B - main pipe after the branch and C - branch. It was concluded that, the leak in the main pipe cause a bigger disturb in the pressure at the section A than the leak in the branch (Section C) of the pipe. Further, the proposed mathematical model was able to predict the flow behavior in this type of geometry with leakage according to number and position of the leak.

As advances in this area, this paper evaluated the fluid dynamic of the three-phase flow (water, oil and gas) in a vertical pipe with and without leakage.

2. METHODOLOGY

2.1 MATHEMATICAL MODELING

The mathematical model used to describe the multiphase flow is the Eulerian-Eulerian inhomogeneous model [9]. In this case, to describe the hydrodynamics of multiphase flow, the following equations are valid:

- Continuity Equation

$$\frac{\partial}{\partial t}(f_i \rho_i) + \nabla \cdot (f_i \rho_i \vec{U}_i) = 0 \tag{1}$$

where f is the volume fraction, ρ is the density, $\vec{U}_i = (u,v,w)$ is the velocity vector, and indices i corresponds to each phase present in the flow, in the case: oil, water and gas.

- Momentum Equation

$$\frac{\partial (f_i \rho_i \vec{U}_i)}{\partial t} + \nabla \cdot [f_i (\rho_i \vec{U}_i \otimes \vec{U}_i)] = -f_i \nabla p_i + \vec{S}_{Mi} + \vec{M}_i + \nabla \cdot \{f_i \mu_{ef,i} [\nabla \vec{U}_i + (\nabla \vec{U}_i)^T]\} \tag{2}$$

where p_i is static pressure of the phase i, $\vec{M}_i$ is interfacial forces (drag) acting in the phase i due to the presence of other forces, $\vec{S}_{Mi}$ describes moment forces due to the external body forces (gravitational force), $\mu_{ef,i}$ is the effective viscosity of the phase i defined as:

$$\mu_{ef,i} = \mu_i + \mu_{t_i} \tag{3}$$

where μ_{t_i} and μ_i are the turbulent viscosity and viscosity of phase i, respectively.

The drag force between two phases α and β can be determined as follows:

$$\vec{M}_i = \frac{1}{8} C_D \rho_\alpha A \left| \vec{U}_\beta - \vec{U}_\alpha \right| (\vec{U}_\beta - \vec{U}_\alpha) \tag{4}$$

where C_D is the drag coefficient, and A represents the interfacial area density.

- Energy Equation

$$\frac{\partial (f_i \rho_i h_{tot_i})}{\partial t} + \nabla \cdot [f_i(\rho_i \vec{U}_i h_{tot_i} - \lambda_i T_i)] = \sum_{\beta=1}^{N_p} (\Gamma^+{}_{i\beta} h_\beta - \Gamma^+{}_{\beta i} h_i) + Q_i + S_i \tag{5}$$

where h_i is the static enthalpy of phase i, T_i is the static temperature (thermodynamics), S_i is the external heat source, λ_i represents the thermal conductivity of phase i, $\sum_{\beta=1}^{N_p} (\Gamma^+{}_{i\beta} h_\beta - \Gamma^+{}_{i\beta} h_\beta)$ corresponds to the heat transfer induced by mass transfer in i, and Q_i is the heat transfer to the phase i due to the interaction with other phases given by:

$$Q_i = \sum_{i \neq \beta} Q_{i\beta} \tag{6}$$

where:

$$Q_{i\beta} = -Q_{\beta i} \Rightarrow \sum_i Q_i = 0 \tag{7}$$

Heat transfer across a phase boundary is usually described in terms of an overall heat transfer coefficient $h_{i\beta}$ which is the amount of heat energy crossing a unit area per unit time per unit temperature difference between the phases. Thus, the rate of heat transfer, $Q_{i\beta}$ per unit time across a phase boundary of interfacial area per unit volume $A_{i\beta}$, from phase β to i, is given by:

$$Q_{i\beta} = h_{i\beta} A_{i\beta} (T_\beta - T_i) \tag{8}$$

- Turbulence Model Equations

The turbulence model used in the simulation is the RNG κ-ε model. The equations describing the continuous phase (oil) in this study are:

a) Equation of the turbulent kinetic energy, ε

$$\frac{\partial (\rho_i k)}{\partial t} + \nabla (\rho_i \vec{U}_i k) = \nabla \left[\left(\mu_i + \frac{\mu_{t_i}}{C_k} \right) \nabla k \right] + P_k - \rho_i \varepsilon \tag{9}$$

where k is the turbulent kinetic energy, ε corresponds to the rate of dissipation of turbulent kinetic energy, P_k is the generation of turbulent kinetic energy, C_k is defined as an empirical constant which value is 1, μ_{t_i} is the turbulent viscosity and is defined for this model as:

$$\mu_{t_i} = C_\mu \rho_i \frac{k^2}{\varepsilon} \tag{10}$$

being C_μ a constant empirical with value 0.09.

b) Equation of the viscous dissipation rate of turbulent, k

$$\frac{\partial(\rho_i\varepsilon)}{\partial t}+\nabla(\rho_i\vec{U}_i\varepsilon)=\nabla\left[\left(\mu_i+\frac{\mu_{t_i}}{C_{\varepsilon1}}\right)\nabla\varepsilon\right]+\frac{\varepsilon}{k}(C_{\varepsilon2}P_k-C_{\varepsilon3}\rho_i\varepsilon) \tag{11}$$

where $C_{\varepsilon1}$, $C_{\varepsilon2}$, $C_{\varepsilon3}$ are constants empirical with value respectively: 1.3, 1.44 and 1.92.

2.2 GEOMETRY AND GENERAL CHARACTERISTICS OF THE STUDIED FLOW

For the construction of the geometry in study was used the software ICEM CFD 12.1 applying the structured method for blocking. The physical problem consists of a vertical pipe with 8 meters in length, with circular cross-section of 25 cm diameter, in which are considered two cases: pipe without hole (Case 1) and with hole (Case 2). For Case 2, the circular hole (d_{leak} = 10 mm) is located at the center of the pipe at the axial position, as shown in Figure 1.

The construction of the mesh was defined to satisfy the physical problem being treated without extending the computational time required. Figure 2 shows the refinement proposed in the hole, entrance region and the exit region of the pipe. The mesh used contains 411656 hexahedral elements.

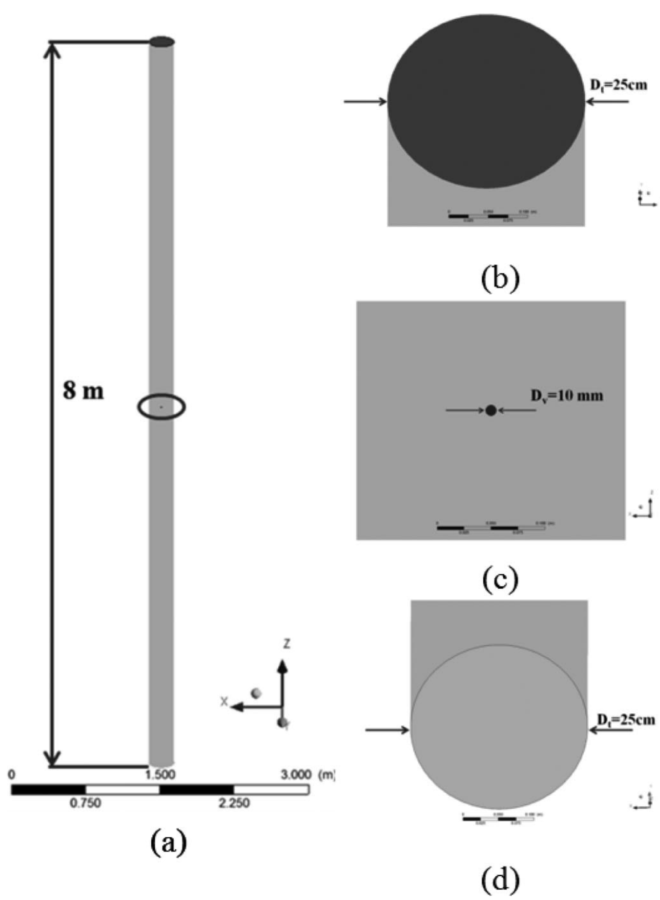


Figure 1: Geometry studied (a) detail of the output region (b), leakage (c) and the input region (d)

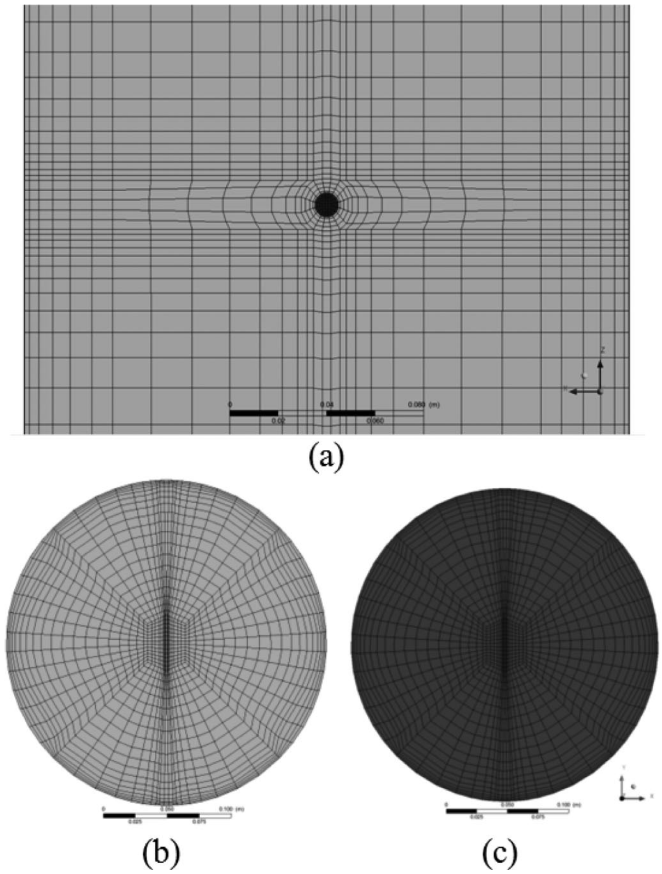


Figure 2: Refinement of the mesh in the hole (a), input (b) and output (c) of the pipe

The flow is three-phase (gas - oil - water) having oil as the continuous phase and water and gas as the dispersed phase.

It was considered following composition of the gas phase: 90% CH_4, 5% C_2H_6, 2.5% C_3H_8 and 2.5% C_4H_{10}. It was used predicted results of a simulated permanent case as an initial condition for simulating the transient case at the same flow conditions. Table 1 and 2 show the initial and boundary conditions, and fluid properties used in this work, respectively.

Some of the characteristics adopted in the resolution of the numerical simulation are given in Table 3.

Table 1: Initial and boundary conditions used

Boundary conditions	**Volumetric flow rate (m^3/h)**	**Pressure (atm)**	**Volume fraction**		
			Oil	**Water**	**Gas**
Inlet	500	–	0.9	0.08	0.02
Outlet	–	0.98	–	–	–
Leakage	–	1	–	–	–
Wall (no slip)	–	–	–	–	–

Table 2: Fluid properties and flow characteristics used in the simulation

Properties	**Water***	**Heavy Oil***	**Natural Gas****
Flow	Dispersed	Continuous	Dispersed
Particle diameter [mm]	8	–	3
ρ[kg/m^3]	997	989	Default
μ [Pa.s]	$\mu_w = 0.00002414x10^{\left(\frac{247.8}{T-140}\right)}$	$\mu_o = 5.187.e\left[-2.3935\left(\frac{T-273}{573-273}\right)\right]$	Default
k [W/m.K]	0.61825	0.1165	Default
T (K)	333	333	333
σ [N.m^{-1}]		0.045 (water/heavy oil) 0.015 (natural gas/heavy oil)	
h [W/m^2.K]		30	

*Produced water viscosity correlation [10, 11] and the heavy oil viscosity correlation [12].
** *Default* are values calculated by the CFX using the library of substances, which are taken into account variations of all greatnesses with temperature.

Table 3: Characteristics of the mathematical model and numerical study treatment

Characteristics	**Considerations**
Flow	Three-phase (water-oil-gas) Three-dimensional incompressible
Chemical reactions	Disregarded
Interface transfer model	Particle models
Minimum volume fraction for area density	10^{-7} kg/s
Pressure interpolation type	Trilinear
Velocity interpolation type	Trilinear
Gravitational effect	Considered
Interfacial force	Drag force
Total time	10s
Time step	0.01s

3. RESULTS AND DISCUSSION

Figure 3 shows the behavior of the total pressure along the pipe with and without leakage. One can observe that the total pressure varies approximately linear along the length of the pipe for the two cases. Also it is seen that in the spill region, there is a peak in the total pressure, near the leak region. The pressure drop in the pipe for the case with leakage was 78170.3 Pa while for case without leakage, this value was 77973.6 Pa. One difference of 196.7 Pa between the studied cases. This result shows the effect of the leak in pressure along the pipe.

Figure 4 shows the of total pressure distribution across the entire length of piping. We can see that the pressure is higher in the inlet region and lower on the outlet region of the pipe.

Figure 5 shows the temperature along the length of the pipe. Was defined as the initial condition that the temperature of the fluids and the wall would be 333K (60°C) and 278 K (5°C), respectively, thus, fluids vary their temperature according to these conditions.

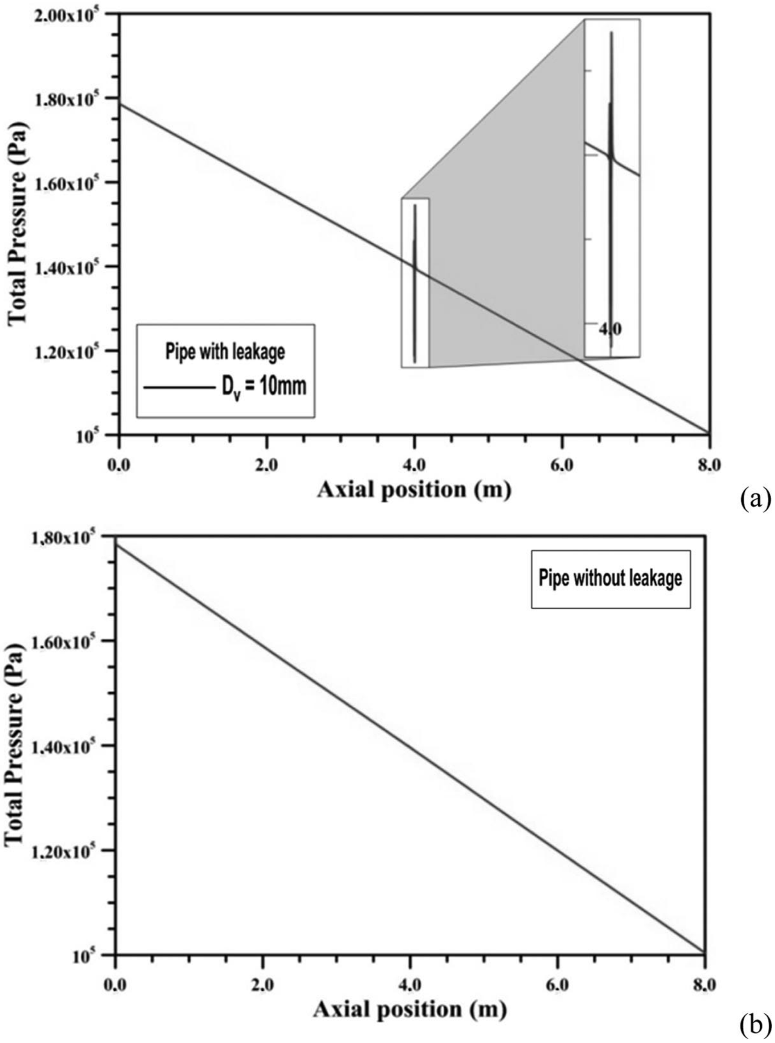


Figure 3: Total pressure along the length of the pipe with (a) and without(b) leakage

It is observed that near the leak (1mm from the leak), water, oil and gas temperature suffers great variation. Along the pipe (considering axial position), the oil temperature is near 35°C, while water and gas are approximately equal to the temperature of the pipe wall, which indicates that the dispersed phase has a higher thermal exchange with the wall more than the continuous phase (oil). It is observed in Figures 6 and 7 the distribution of temperature in the region of the hole in a longitudinal and transversal plan (Z = 4 m), respectively.

From the analysis of these figures we can see that the oil temperature suffers little variation across the pipe, while water and gas have greater variation from the center to the wall of the pipe. Further it is observed that the water is the fluid that remains in lower temperature among the phases present in the flow.

Figure 8 shows the superficial velocity (volume fraction multiplied by the velocity) of the oil, gas and water along the length of the tubing (1 mm from the leak). We see that in leakage area, there is a large variation in the superficial velocity of the phases. After

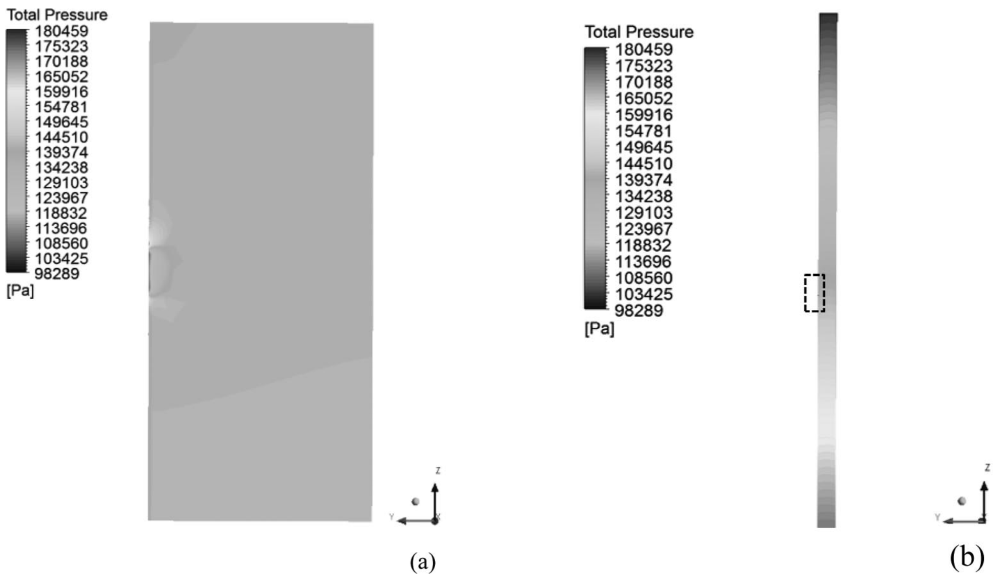


Figure 4: Total pressure distribution near the area of the leak (a) and in the whole pipe (b)

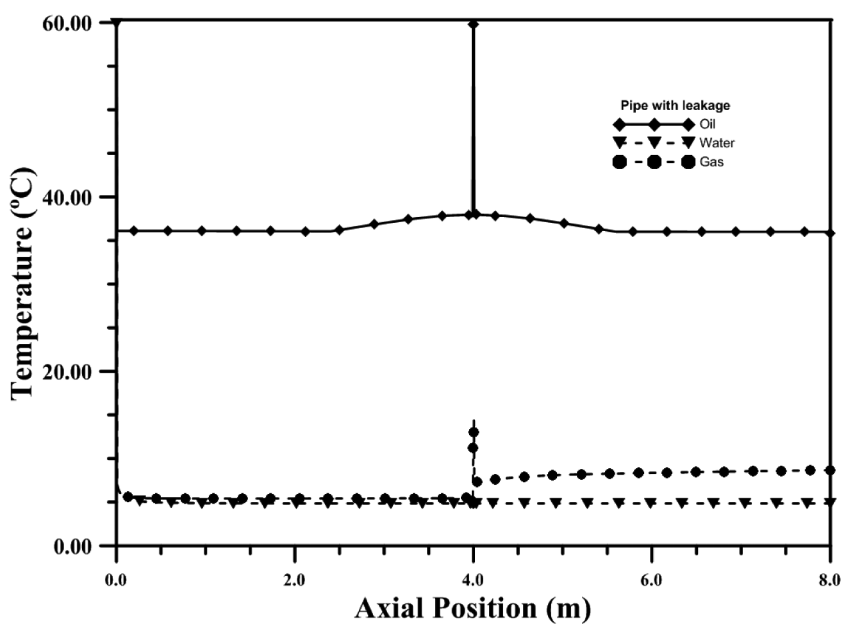
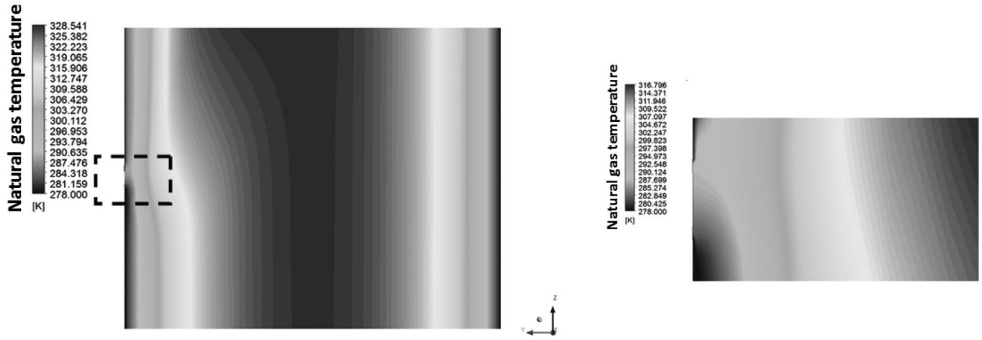
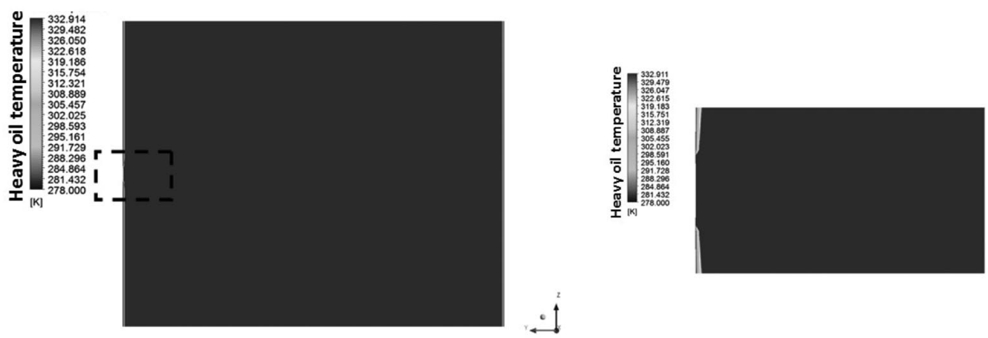


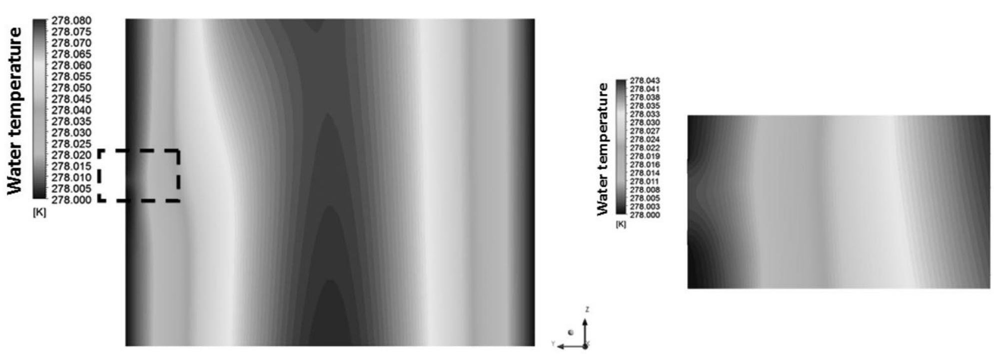
Figure 5: Temperature of the phases along the length of the pipe near the leak (1mm from the leak)



(a)



(b)



(c)

Figure 6: Temperature distribution along the pipe near the area of leak (longitudinal plane)

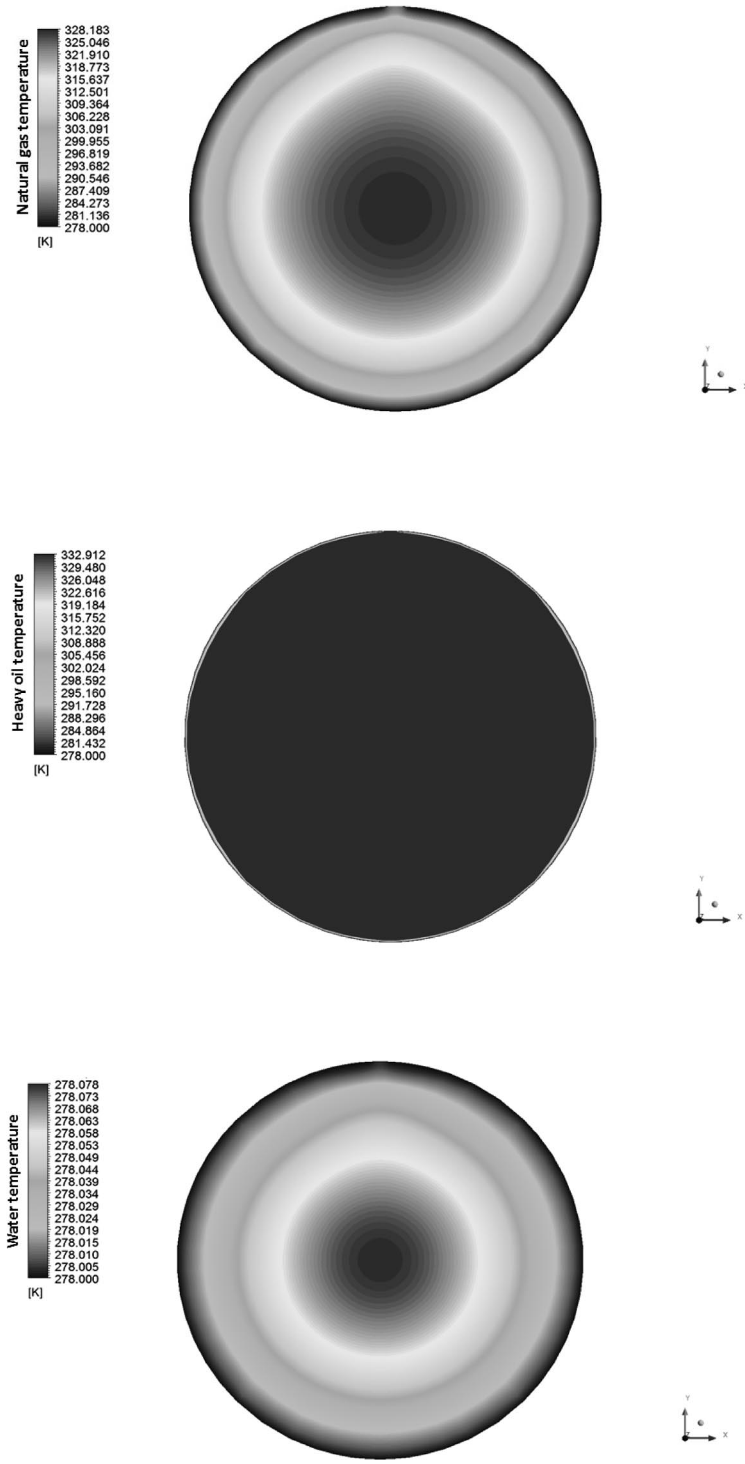


Figure 7: Temperature distribution near the area of the leak (Transverse plane, Z = 4 m)

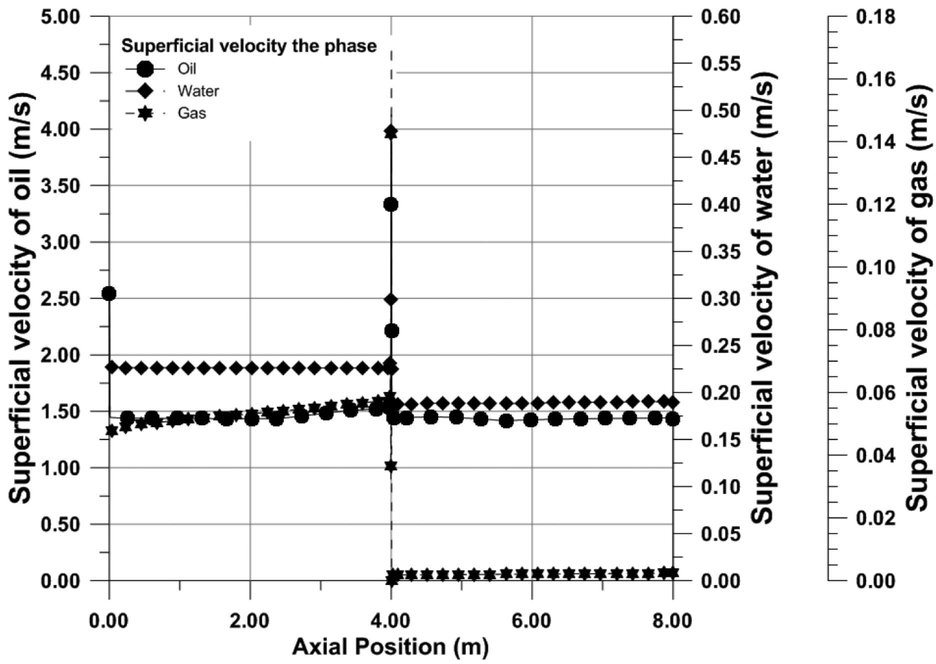


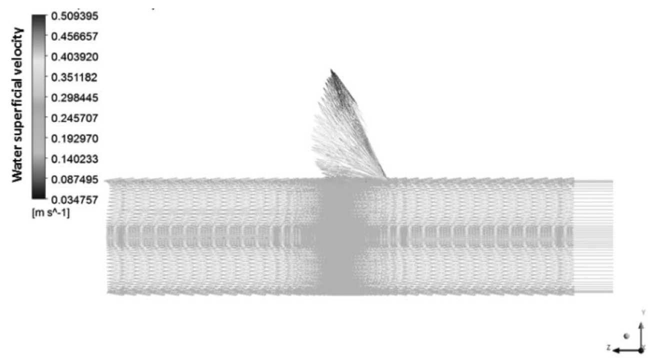
Figure 8: Superficial velocity of the phases along the pipe near the leak (1 mm from the leak)

leakage, the gas superficial velocity approach to zero, probably due to the escape of natural gas of this region through the hole. Further, details of the behavior of this property can be seen in Figure 9 and 10, where it is seen that the superficial velocity of the oil is almost 8 times higher than the superficial velocity of the water and 27 times greater than the superficial velocity of the natural gas.

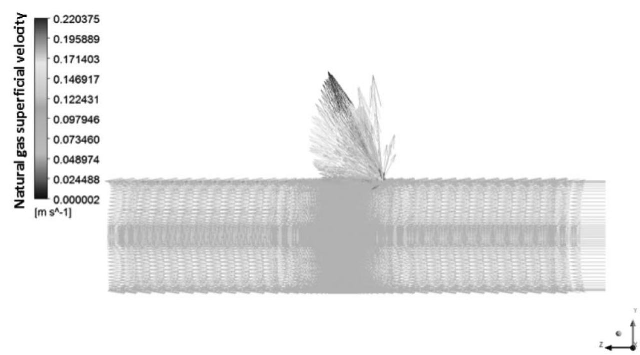
The true velocity vectors of the fluid phases near the leakage are illustrated in Figures 11 and 12. We can see that the variation of the velocity of oil and water are similar while the gas velocity is almost three times greater than the other involved phases due to the difference of density, particle diameter, viscosity, and lower volume fraction (2%) of the gas phase used in the simulation.

Figure 13 illustrates the behavior of the dynamic viscosity of the oil and water in the pipe near the leak (1 mm from the leak). Dynamic viscosity is a measure of the resistance of a fluid shear forces and appears in the momentum equations. This property for these two phases was calculated from correlations in the CFX software that take into account the influence of temperature. One can see that the viscosity of the oil is influenced by presence of the hole (near the leak there is a drop in viscosity due to increases in the oil temperature) while the dynamic viscosity of water keeps without abrupt variations in the hole area.

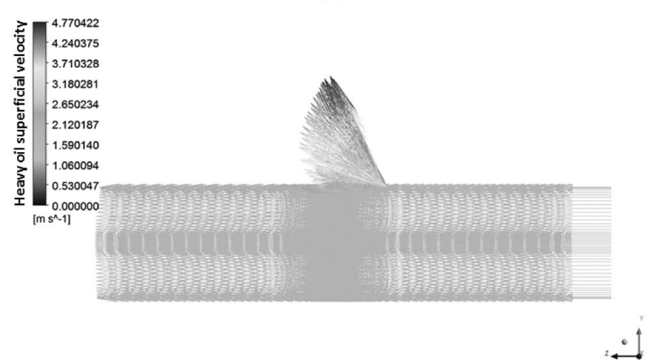
Figure 14 shows the volume fraction of oil, gas and water along the pipe near the leakage area (1 mm from the leak). The volume fraction is defined as the initial condition. It is seen that variations in the volume fraction is perceptible in the leak area. This indicative of discrepancy on this region may facilitate the detection of leakage.



(a)



(b)



(c)

Figure 9: Vectors of superficial velocity of the phase a) water, b) natural gas and c) heavy oil near the leak

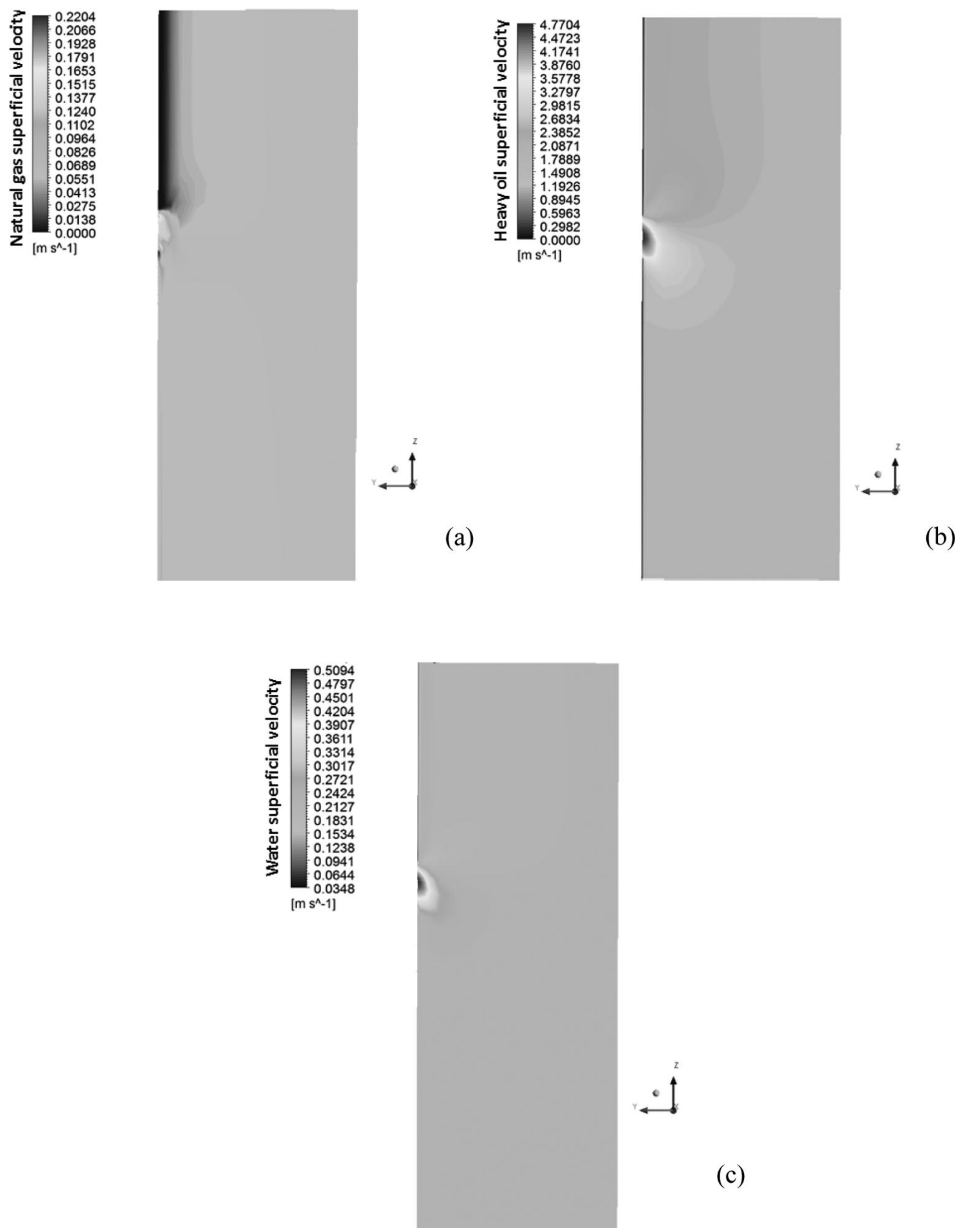
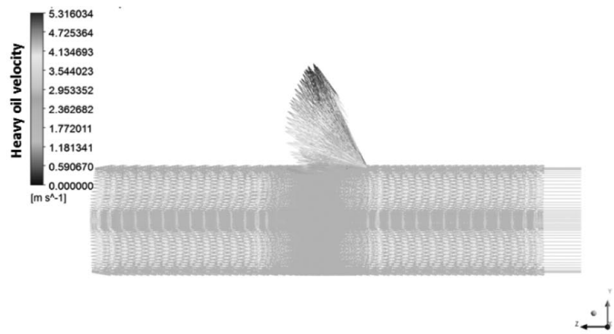
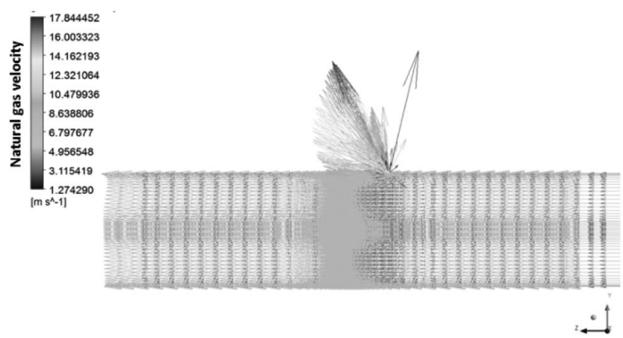


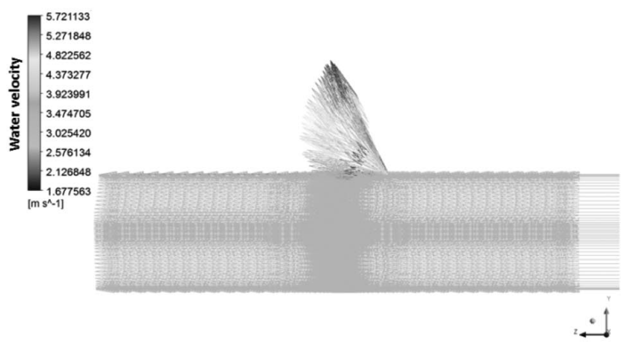
Figure 10: Superficial velocity distribution of the phases a) natural gas, b) heavy oil and c) water near the leak



(a)



(b)



(c)

Figure 11: Velocity vectors of the phases a) heavy oil, b) natural gas and c) water near the leak

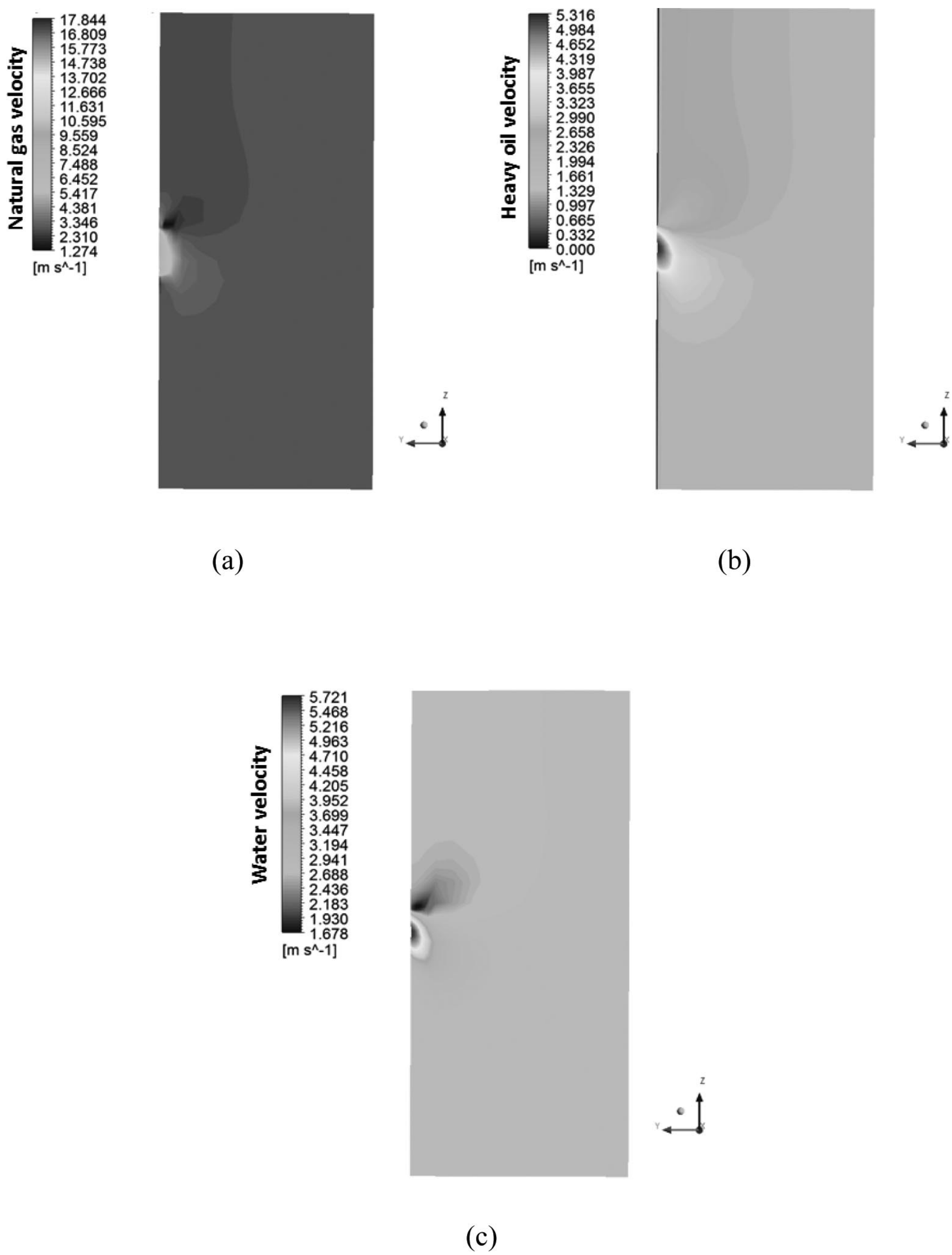
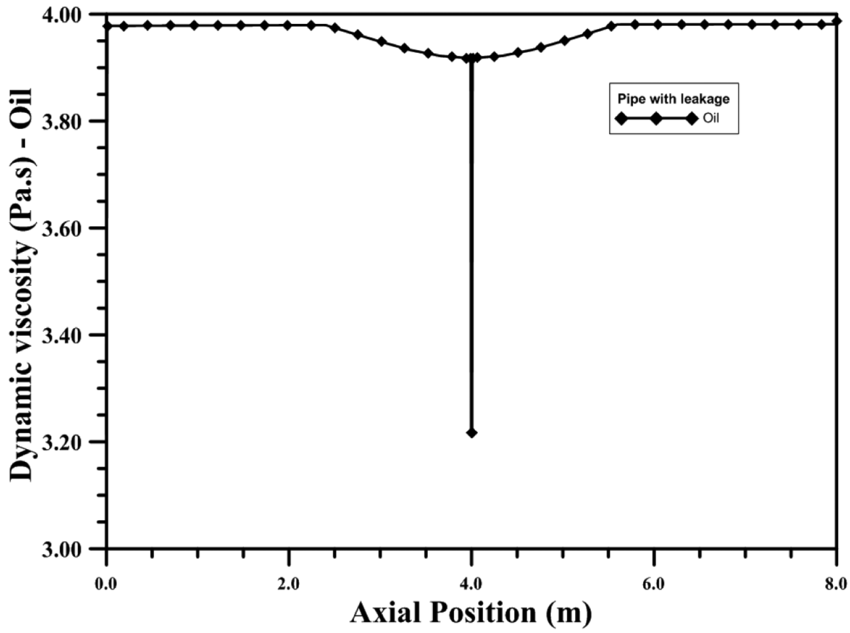
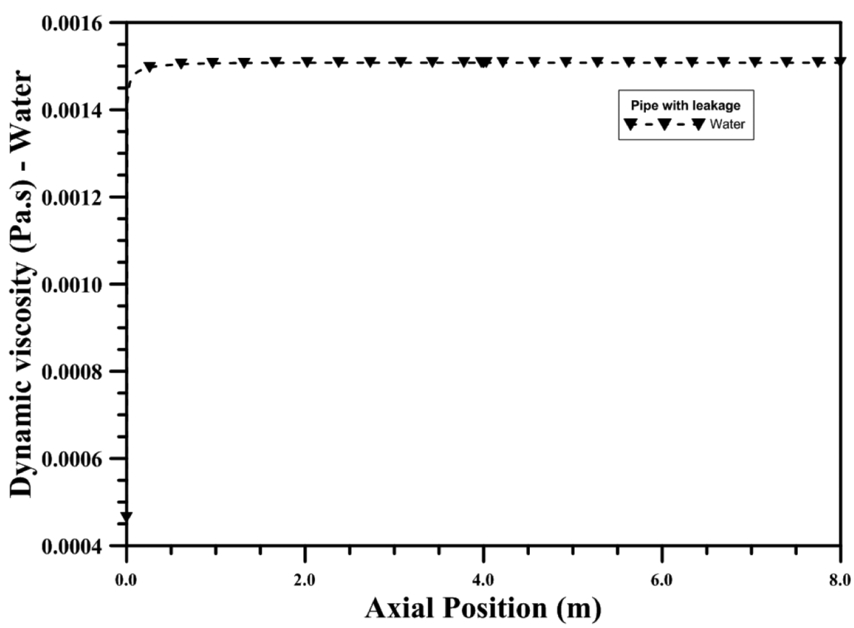


Figure 12: Velocity distribution of the phases a) natural gas, b) heavy oil and c) water near the leak



(a)



(b)

Figure 13: Dynamic viscosity of oil (a) and water (b) along the pipe near the leak (1 mm from the leak)

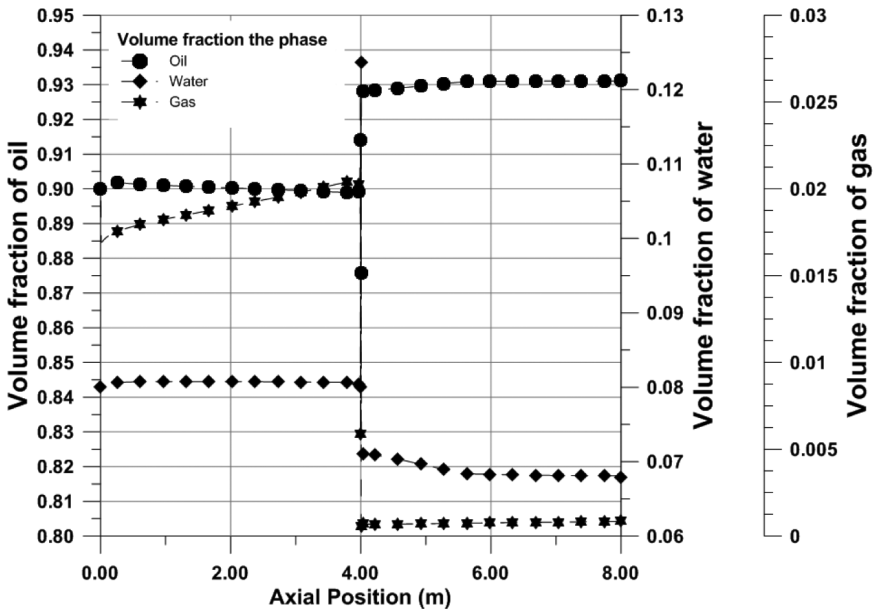


Figure 14: Volume fraction of the phases along the pipe near the leak (1 mm from the leak)

4. CONCLUSIONS

Based on the results of the theoretical study and numerical simulation of the three-phase flow (water, oil and gas) in a vertical circular cylindrical duct with leaks, it can be concluded that:

a) In the region near the hole occur great variations in the volumetric fraction, total pressure and superficial velocity of the phases.
b) The total pressure drop is linear over the whole pipe except in the area near the leakage.
c) The oil temperature suffers little variation across the pipe, while water and gas have greater variation from the center to the wall of the pipe. It is further observed that the water is the fluid that remains in a lower temperature among the phases in the studied flow.
d) The superficial velocity of the oil is about eight times greater than the water and twenty seven times greater than the natural gas due to its higher volumetric fraction used in the studied case. When analyzing the velocity (without taking into account the volume fraction), the highest velocity is attained by the gas phase.
e) In the leakage region, there is an increase in volume fraction of oil and water and a decrease in gas volume fraction probably due to its low viscosity and low density, which facilitates loss of this fluid through the leak hole.

ACKNOWLEDGEMENTS

The authors would like to express their thanks to Brazilian research agencies ANP/UFCG-PRH-25, CNPq, CAPES, FINEP and PETROBRAS S/A for supporting this work, and are also grateful to the authors of the references in this paper, that helped in the improvement of quality.

REFERENCES

[1] Andrade, T. H. F.; Crivelaro, K. C. O.; Farias Neto, S. R.; Lima, A. G. B.; Numerical Study of Heavy oil Flow on Horizontal Pipe Lubricated by Water. In: Andreas Ochsner; Lucas F. M. da Silva; Holm Altenbach. (Org.). Materials with Complex Behaviour II: Properties, Non-Classical Materials and New Technologies. Series: Advanced Structured Materials. Heidelberg: Springer-Verlag, 2012, v. 16, 1ed., p. 99–118.

[2] Sarmento, L. R. B., G. H. S. Pereira Filho, Lima, A. G. B., Farias Neto, S. R., Barbosa, E. S., A. de Lima Cunha. Multiphase Flow and Heat Transfer in Risers. Defect and Diffusion Forum, 2014, v. 348, p. 3–8.

[3] Cavalcanti, W. S.; Belém, E. Z. G.; Lima, W. C. P. B.; Cavalcante, F. P.; Barbosa, E. S.; Farias Neto, S. R.; Lima, A. G. B. (2011). Non-Isothermal Three-Phase Flow of Petroleum, Gas and Water in T and Y Junctions. International Journal of Modeling and Simulation for Petroleum Industry, 5(1), p. 43–52.

[4] Barbosa, L. M. C.; Lima, A. G. B.; Farias Neto, S. R.(2012). Non-Isothermal Transient Three-Phase Flow (Heavy Oil, Water and Gas) in Vertical Pipeline: The Effect of Leakage. International Journal of Modeling and Simulation for the Petroleum Industry, 6(2), p. 23–31.

[5] Sousa, J. V. N.; Sodré, C. H.; Lima, A. G. B.; Farias Neto, S. R. (2013). Analysis of Heavy Oil-Water Flow in Vertical Ducts Leaking via Computational Fluid Dynamics. Advances in Chemical Engineering and Science, 3(1), p. 9–15.

[6] Araújo, M. V.; Luna, F. D. T.; Barbosa, E. S.; Farias Neto, S. R.; Lima, A. G. B.(2013). Numerical Study of Oil Flow in Tee Junction With Leaks. Advances in Petroleum Exploration and Development, 6(2), p. 1–11.

[7] Sarmento, L. R. B., Leite, B. E., Farias Neto, S. R., Lima, A. G. B., Hydrodynamics of Oil-Water with Leakage by CFD. Defect and Diffusion Forum, 2014, v. 348, p. 58–63.

[8] Araújo, M. V., Farias Neto, S. R., Lima, A. G. B., Theoretical Evaluation of Two-Phase Flow in a Horizontal Duct with Leaks. Advances in Chemical Engineering and Science,3, 2013, p. 6–14.

[9] ANSYS, CFX – Theory Manual, 2012.

[10] Babadagli T.; A. Al-Bemani, Investigations on Matrix Recovery during Steam Injection into Heavy-Oil Containing Carbonate Rocks, Journal Petroleum Science and Engineering, Vol. 58, No. 1–2, 2007, pp. 259–274.

[11] Souza, J. S.; Farias, F. P. M.; Swarnakar, R.; Farias Neto, S. R.; Antonio, G. L. Non-Isothermal Separation Process of Two-Phase Mixture Water/Ultra-Viscous Heavy Oil by Hydrocyclone, Advances in Chemical Engineering and Science, Vol. 1 No. 4, 2011, pp. 271–279.

[12] Perry, R. H., Green D.W., Perry's Chemical Engineers' Handbook. Mc Graw-Hill, New York, 2005, 8.ed.