Latin American Journal of Energy Research – Lajer (2017) v. 4, n. 1, p. 17 – 29
DOI: http://dx.doi.org/10.21712/lajer.2017.v4.n1.p17-29

Influence of oil leakage in the pressure and flow rate behaviors in pipeline

Carolina Andrade de Sousa[1*], Oldrich Joel Romero[2]
[1]Undergraduate Student in Petroleum Engineering, Federal University of Espírito Santo – Ufes, campus São Mateus, ES
[2]Professor of the Undergraduate Program in Petroleum Engineering and Energy Graduate Program, Federal University of Espírito Santo – Ufes, campus São Mateus, ES
*Author for correspondence, E-mail: as.carol@hotmail.com

Article history
Received: 25 July 2017
Accepted: 10 August 2017
Available online: 25 August 2017

Abstract: Pipelines are highly used in the oil industry for the transportation of oil, natural gas and products from refineries. One of the main areas of study in pipeline engineering is the flow guarantee. This paper describes the steady state dynamics behavior of oil flow in a pipeline with the presence of a leak. The mathematical and numerical modeling of a single-phase flow in a one meter pipeline with a diameter of 0.15 m in an onshore environment. Three different sizes of leaks are studied based on fluid dynamics simulations using the ANSYS FLUENT® 15.0 commercial software. The interest of this paper is to evaluate the influence of leakage on the pressure, velocity fields and flow rates in a 3D section of pipe in order to identify the influence of the perturbation in these parameters. The results obtained is interpreted in the CFD-Post that is included in the software package. Thus, this work intends to contribute to the development of operations tools in oil and gas industry.
Keywords: Pipeline leakage, three dimensional flow, turbulent flow, ANSYS FLUENT.

1. Introduction

Petroleum has become one of the most important source of energy we have. Every day people use hundreds of products that are derived from oil, but this source of energy mostly has been known as an efficient fossil fuel in the transportation industry. Therefore, the demand for this energy is quite large and is increasing each day (Bahari, 2014). In this context, pipelines are the way of transport most used to connect the oil and gas production fields to the refineries and consumption centers (Araújo *et al.*, 2014).

The use of industrial pipes is very evident nowadays due to the increasing need to transport materials, usually fluids, from one point to another distant one (Sousa *et al.*, 2016), Pipelines constitute a major means of gathering and transporting a variety of commodities ranging from oil, natural gas and chemical to water, are a very important way of collecting and transporting various types of goods, from oil, natural gas, chemicals to water and sewage (Kyriakides and Corona, 2007). From the earliest days to the oil industry, pipelines have proved to be the easiest, safest and the most economical method for the distribution of natural gas and transport of oil in bulk (Stewart, 1933). Their use has expanded with time because they are more energy efficient than competing means of transportation. Although they require significant initial investment, they have a lifespan up to 40 years and require relatively minor maintenance (Kyriakides and Corona, 2007).

According to Sousa *et al*. (2016), one of the main fields of study in the ducts engineering is the guarantee of the flow, with special focus on leakage of fluids that are propitious to happen due to the aggressive environment in which the ducts are generally exposed. Leakage is defined as the accidental admission or escape of fluid through a hole or crack from the pipe, according to the Oxford dictionary. Pipeline monitoring, control, operation and maintenance are very important activities, which have evolved considerably. The detection and behavior of leaks has deserved special attention by different researchers. The loss of fluid to the external environment interferes directly in the fields of velocity, pressure and temperature of the pipe, and especially in the reduction of the total volume transported to the final destination.

Pipelines are used to transport the hydrocarbons from the tanker to the reservoir or vessel for storage and from the tanks to the transportation trucks via the loading racks. There are thousands of connections, joints and valves between pipes and tanks. According to the design specification, those connections have to be tight enough to avoid any leakages but that is not the case all the time. Corrosion, metal fatigue due to external stresses, erosion in welding joints are the problems that brought up since tanks and pipelines are made of metals (Audunsson, 2006). The leak is a very common phenomena and is often subjected to interference from third parties damage, geological disasters, accidents, human errors during operation or may result from several other reasons such as bad workmanship, from any destructive cause due to sudden changes of pressure, corrosive action, cracks, defects in pipes, junction of pipes or equipment and even lack of maintenance. (Ben-Mansour, 2012; Araújo *et al.*, 2013)

Although some previous studies are found in the literature, the whole area of leak detection and how these leaks affect the flow is yet to be mature since several variables have to be analyzed together such as the size, position and number of the leaks present in the pipe, the inlet flow rate, the pressure that the pipe is insert, the causes of leakage appearance and among other things that difficult the real analysis of this problem. Leakages from a pipe can result in damage to the ambient environment depending on the total amount of released hydrocarbon to the atmosphere in an onshore pipeline or in the subsea offshore environment. Accordingly, the deleterious effects associated with the occurrence of leaks may present serious problems to the environment, to the lives in general and to the company reputation. Considering these facts, leak detection and location identification in a timely manner is crucial to reduce or minimize the fluid loss and even to attenuate the economic impact of a hydrocarbon spill to its stakeholders that can be very huge (Jujuly, 2016).

According to Zhang *et al*. (2014), since leaks must be quickly detected, located and repaired, pipeline leak detection technologies have been playing an important role in protecting the safety of pipeline transportation. There are a many leak detection techniques that even estimates the size of those failures along the pipe. Almost all of them use transient flow simulations in order to detect the hole using the mass balance, based on the mass conservation equation and the difference in volume of oil, analysis on the profile pressures considering the drop of the pressure along the pipe or using acoustic methods depend on a sonic leak detection equipment, which identifies the sound of fluid escaping a pipe (Jujuly, 2016; Bahari, 2014; Alonso, 2005,). However, when it is about a very small chronic leak that is below the threshold of current leak detection systems, it could continue undetected for a long period of time, potentially releasing a significant amount of hydrocarbon to the environment (Jujuly, 2016).

The occurrence of a leak divides the pipeline system into three parts: exact location, upstream, and downstream (Fig.1) according to (Araújo *et al*., 2014; Edris and Kam, 2013). Based on the theory of continuous flow in the steady state for incompressible fluids, the profiles of pressure and flow rate over the pipeline show the leak location, since it is possible to observe the drop of both parameters in the graphs (Araújo *et al.*, 2014; Edris and Kam, 2013; Sousa *et al.*, 2016).

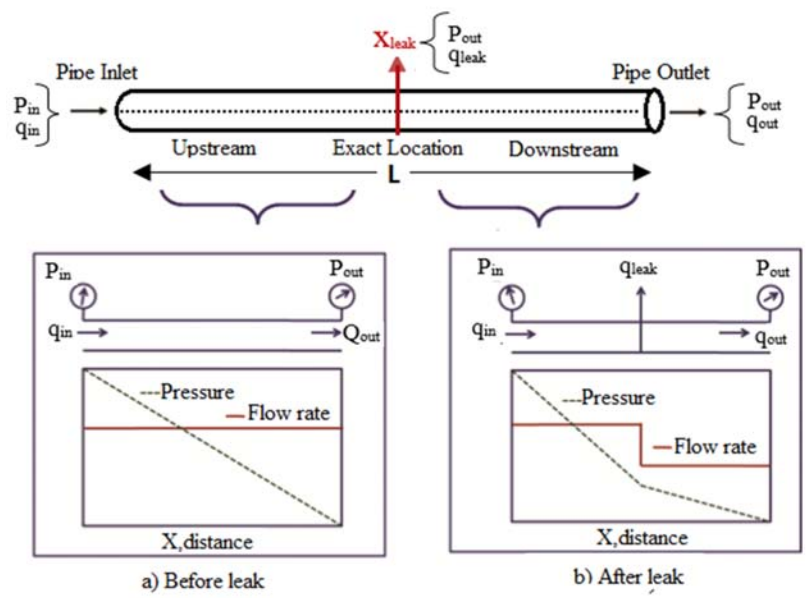

Figure 1. Behavior of pressure and flow rate along the pipe in the steady state. (Adapted from Araújo *et al.*, 2014 and Edris and Kam, 2013).

According to the studies of Araújo *et al.* (2013), it is also possible to affirm that by increasing the diameter of the leak orifice, the volume of fluid leaving the orifice is greater. In other words, the large the magnitude of the hole the higher the flow rate through the leak and the lower the flow rate delivered to the final destination.

1.1 Objective

This paper aims to analyze, in the steady state flow, the behavior of the pressure and velocity field in a leakage pipe in response to different sizes of leaks in order to determine how this perturbation affects the hydrodynamics inside this pipeline.

2. Methodology

A numerical modeling of an onshore pipeline leakage is performed using a 3-D turbulent flow model in computational fluid dynamics (CFD). The primary motivation to this study is the difficulty to evaluate through laboratory experiments the effect of leakage in a pipeline, due mainly their extension. Therefore, it is necessary to use computational tools and a numerical simulation technique in order to improve the analysis.

At first, oil with the same properties of water will be used in a land pipeline to simplify the analysis. The influence of the leak in the entire flow parameters and the behavior of the fluid through the leak were analyzed using velocity vectors and pressure contours. It is important to notice that there are few numerical 3D studies currently available on the literature related to the fluid dispersion from different sizes of leak orifice showing the flow patterns through them.

2.1 Problem definition

The system for the study is defined as an oil flow through a pipeline with 1 m length and 0.15 m diameter. The completely horizontal pipe is located in a land environment, positioned directly on the ground and at ambient temperature of 25 ºC. In this initial part of the study, the fluid flow considered along the pipe is a monophasic liquid oil with constant properties similar to water such as density of 998.2 kg/m3 and viscosity of 0.001003 kg/(m.s). Those theoretical dimensions are considered for simulation purposes and the assumption was made in order to do a basic analysis of the case, understanding the physical phenomenon at first, then improve the analysis in future works. The flow regime is considered turbulent since the velocity magnitude in the inlet zone is specified to be 0.1 m/s. The outlet pressure and the pressure at the leak are both the atmospheric gauge pressure. The physical model is represented in Fig. 2.

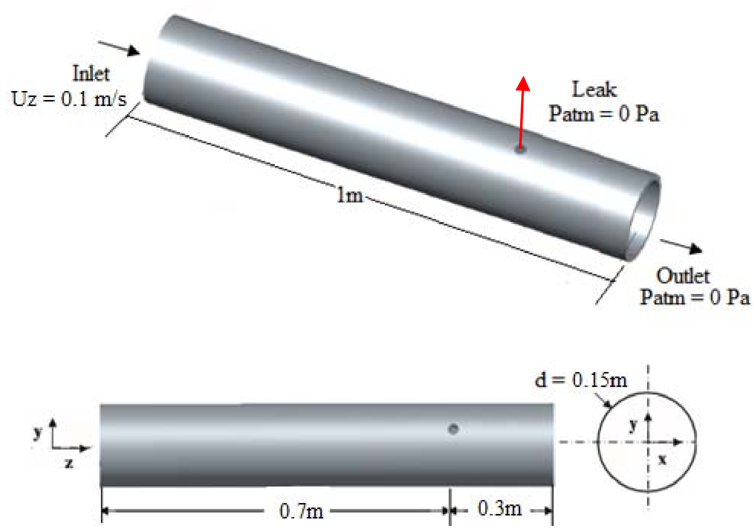

Figure 2. Problem schematic representation of a leakage pipeline with the coordinate system.

In order to evaluate the effect of the perturbation in a pipeline flow, it is assumed that the leak location and its magnitude is already known from other techniques leak detection. In this paper three different sizes of leaks are analyzed comparing the results with a non-damage system. The sizes of the leak diameter (D_{leak}) are based on the percentages of 1%, 5% and 10% of the pipe diameter (D_t) (Macias *et al*., 2005; Romero, 1999).

2.2 Mathematical formulation

The governing equations to describe the phenomenon of the fluid flow in the pipe and leak are the conservation of mass and momentum (Araújo *et al*., 2013). Considering the fluid is Newtonian, incompressible, with constants physical and chemical properties, also the three dimensional flow is isothermal. The equations that describe the flow based on Baptista *et al*. (2007), Jujulu (2016) and Wang *et al*. (2012) are detailed below.

2.2.1 Conservation equations

The mass conservation equation for single-phase flow is

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

where ρ is the density of the fluid, and $\vec{u}$ is the velocity vector. The first term on the left hand side in the previous equation represents the mass rate change in terms of the chosen infinitesimal element and the second term represents the net mass flow rate out of the control surface.

The equation of the linear momentum conservation, or Navier-Stokes equation, to describe the motion of incompressible viscous fluids, neglecting the effect of the gravity force, since the flow is completely horizontal, can be expressed as

$$\frac{\partial(\rho\vec{u})}{\partial t} + \nabla \cdot (\rho\vec{u}\vec{u}) = -\nabla p + \nabla \cdot \bar{\bar{\tau}}, \tag{2}$$

where p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor which can be written as

$$\bar{\bar{\tau}} = \mu_{meff}[\nabla\vec{u} + (\nabla\vec{u})^T], \tag{3}$$

μ_{meff} is defined as the effective viscosity (Eq. 4) and $\nabla\vec{u}$ the velocity gradient.

$$\mu_{meff} = \mu_f + \mu_t, \tag{4}$$

where μ_f is the viscosity of the fluid and μ_t is the turbulent viscosity detailed on turbulence model (2.2.2)

According to the Eq. (2), the first and the second terms in the left hand side of the equation represent the local and convective acceleration, respectively. In the right hand side, the third and fourth terms represents the pressure and the viscous forces, this last one written in terms of the velocity gradient $\nabla\vec{u}$ (Eq. 3).

To simulate the leak considering the mass conservation in the flow diversion, the continuity equation at the leak location takes the following form according to Al-Khomairi (2005)

$$q_{up} - q_{downs} - q_{leak} = 0\,, \tag{5}$$

where q_{up} and q_{downs} is the discharge just upstream and downstream of the leak location, respectively, and q_{leak} is the leak flow rate.

2.2.2 Turbulence modeling

Reynolds number is a dimensionless parameter which classifies the flow as laminar or turbulent based on the following equation

$$Re = \frac{\rho D_t \bar{V}}{\mu}, \tag{6}$$

this parameter expresses the type of motion and velocity of the fluid particles, becoming unstable after a certain number. For low Reynolds number the flow is laminar, on the other hand, for high Reynolds numbers, the flow has turbulent characteristics, a random and chaotic state of motion where velocity and pressure change continuously with time (Wilcox, 1993). Since Reynolds number in this study approximately 15,000 according to Eq. (6), the flow model is turbulent and it is necessary to modeling the conservation equations for this situation.

RANS (Reynold's Average Navier-Stokes) equation based models are used to model steady-state turbulent flow simulations (Jujuly, 2016). The RANS turbulence equation based on the $\kappa - \varepsilon$ standard model, provides fairly reasonable result and is also the most widely used in engineering turbulence modeling for industrial applications, since it is numerically robust and proven to be stable according to Jujuly (2016). This model is based on the eddy or turbulent vscosity concept (μ_t) where the effective viscosity, μ_{meff} (Eq. 4), accountable for turbulence. Therefore, the model assumes that the turbulent viscosity μ_t is related to the turbulent kinetic energy and dissipation through the following ratio

$$\mu_t = \frac{\rho C_\mu \kappa^2}{\varepsilon}, \tag{7}$$

where C_μ is a typical model constant with default value of 0.09 to flow with high Reynolds number according to the Ansys Fluent theory guide.

This model solves two different transport equations. The first one is the turbulent kinetic energy κ, defined by the variation of the velocity fluctuations and can be described as

$$\frac{\partial(\rho\kappa)}{\partial t} + \nabla \cdot (\rho\bar{u}\kappa) = \nabla \cdot \left[\left(\frac{\mu_{eff}}{\sigma_k}\right)\nabla\kappa\right] + P_\kappa - \rho\varepsilon, \tag{8}$$

the second one is the and dissipation rate of the turbulent kinetic energy ε, modeled as

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \nabla \cdot (\rho\bar{u}\varepsilon) = \nabla \cdot \left[\left(\frac{\mu_{eff}}{\sigma_\varepsilon}\right)\nabla\varepsilon\right] + C_{\varepsilon 1}\frac{\varepsilon}{\kappa}(P_k + C_{\varepsilon 3}P_b) - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k}, \tag{9}$$

P_κ e P_b represents the generation of turbulence kinetic energy due to the mean velocity gradient and due to buoyancy respectively. It is important to remind the buoyancy effects on ε are often neglected in the transport equation for ε. The model constants $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_κ and σ_ε have the following default respective values : 1.44; 1.92; 1.0; and 1.3; $\bar{u}$ is the velocity vector average in time (ANSYS FLUENT, 2013).

2.2.3 Boundary conditions

In order to solve the system of equations proposed above, it is necessary to delimit the solution domain, that is, apply the boundary conditions on the studied case, given by

- Constant oil velocity equal to 0.1 m/s in the inlet section (prescribed flow).
- The turbulent kinetic energy κ at the inlet is 0.75 (ANSYS FLUENT, 2013).
- The turbulent dissipation rate ε at the inlet is 0.75 (ANSYS FLUENT, 2013).
- Constant atmospheric gauge pressure equal to 0 Pa in the outlet section;
- Constant atmospheric gauge pressure equal to 0 Pa in the leak section;
- Stationary wall and no slip condition in the wall-solid zone ($Ux = Uy = Uz = 0$ m/s).

2.2.4 Analyzed cases

In this first part of the study, four cases were considered with the goal to determine the effects of the perturbation on the velocity and pressure fields of the system.

Table 1. Analyzed cases

Case	Fluid	D_t, m	Leak magnitude, $\%D_t$	D_{leak}, m	Length, m	Leak position, m
1	Oil at 25ºC	0.15	0	0	1	0.7
2	Oil at 25ºC	0.15	1	0.0015	1	0.7
3	Oil at 25ºC	0.15	5	0.0075	1	0.7
4	Oil at 25ºC	0.15	10	0.015	1	0.7

2.3 Numerical approach

Pipeline hydrodynamics cannot be captured accurately in a small-scale lab environment since the leakage phenomenon occurs in an industrial full-scale pipelines with big extension and large diameters. Thus, a numerical simulation can provide a better understanding, detailed information on the hydrodynamics of the pipeline flow and the consequences of pipeline leaks in different scales, reducing the cost and number of experiments (Jujuly, 2016; Selvarajah, 2014).

The equations that govern the physical phenomenon of a tridimensional, turbulent and monophasic flow inside a pipeline, are solved by CFD (computational fluid dynamic) model and through finite volumes technique discretization implemented in ANSYS FLUENT® 15.0, where the variable of interest is located at the centroid of the control volume. This commercial software package provides an integrated modular design, meshing technology, and large degree of freedom for pre- and post-processing the fluid flow simulation in pipeline with high interactivity which allows the better visualization of the problem and its solution (ANSYS FLUENT, 2013).

The solution method uses the pressure-based type of solver to reach a solution algorithm where the governing equations are solved using the coupled scheme mode. The pressure-based coupled algorithm solves a coupled system of equations comprising the momentum equations and the pressure-based continuity equation according to the solver theory guide of the software. In this mode, the linear momentum and pressure equation are coupled (velocity-pressure coupling) and solves the hydrodynamic equations as a single system using *Rhie Chow* to interpolation. The spatial discretization occurs through the pseudo transient solution approach, where the time depend terms are also evaluated, therefore it is an implicit procedure. The solution, however, is obtained in the steady state regime, where the interest properties of the fluid do not change with time (ANSYS FLUENT, 2013).

Since the governing equations are non-linear and coupled, the solution loop must be carried out iteratively in order to obtain a converged numerical solution. A numerical analysis is proposed to determine the influence of the leak orifice in the flow and the patterns of fluid flow within the pipeline from a particle dynamic study of local parameters- steady flow (i.e. pressure, temperature, turbulence kinetic energy, pressure gradient, velocity vectors, etc.) (Selvarajah, 2014).

The geometry is built using the Design Modeler tool presented in the workbench of the software involving inlet, outlet, leakage and wall domain zones. The spatial discretization is obtained with the Meshing tool, resulting in a certain quantity of elements which have to be analyzed and tested in order to reach a final result independent of the mesh size.

The studied case is solved by iterations simulations with the Fluent-Solver after the determination of the boundary conditions and operational parameters in the setup section. The obtained results are post-processed in the ANSYS CFD-Post to obtain a better visualization of the resulting physical phenomenon. The influence of the leak in the flow dynamics parameters and the behavior of the fluid were analyzed using velocity vectors, streamlines and pressure fields contours.

2.3.1 Mesh test

In order to determine the quality of the discretized domain and how the number of elements influences in the final result, the mesh tests are done by changing the size and quantity of the elements. Four different analysis was made: coarse, medium, fine and a super fine mesh for the case 2 from table 1 with a leak size of 1% of the pipe diameter, as is shown in the Table 2.

Table 2. Mesh information test.

Mesh	Number of elements	Simulation time, min
Coarse	69,173	2.23
Medium	209,519	5.90
Fine	618,959	11.83
Super fine	1,190,237	267.16

The mesh tests were performed in order to evaluate the most efficient mesh to this specific situation. The refinement in the mesh was done varying parameters in the meshing tool on ANSYS software such as relevance, span angle center, minimum size, maximum face size, inflation and proximity minimum size.

The result of this refinement can be observed on Fig. 3 where different numbers of elements show differences in the profile pressure along the pipe.

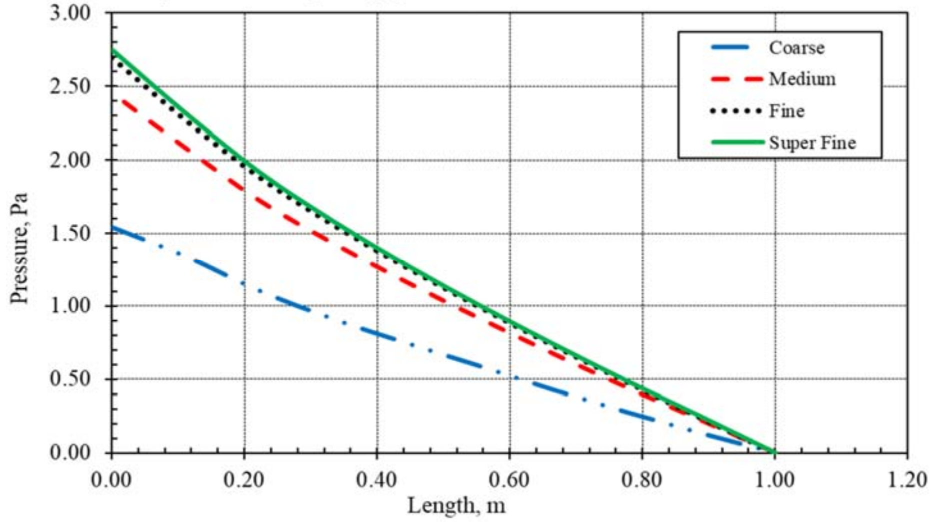

Figure 3. Pressure behavior with the mesh refinement.

Analyzing the curves above and also according to Table 2, the coarse mesh is not the most representative mesh for this study, when comparing to the other ones. Since there is no sufficient discrepancy on the gradient pressure result along the pipe for the medium, fine and super fine meshes, taking into account the simulation time and the computational cost, it can be concluded the medium mesh is the most adequate for this study. The spatial discretization was obtained with the Meshing software, resulting in medium mesh of 209,519 elements as is shown on Fig. 4.

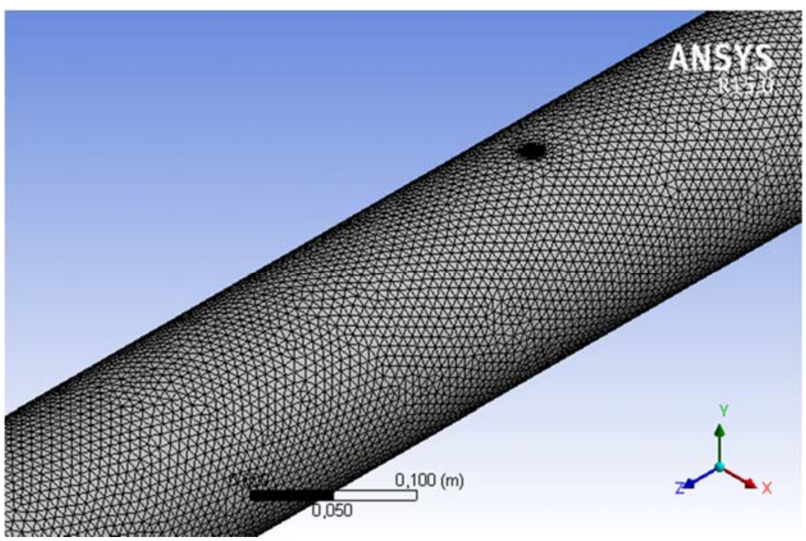

Figure 4. Numerical grid for computational discretized domain of the leakage pipeline (medium mesh, in Table 2).

3. Results and comments

The results are post-processed in the ANSYS CFD-Post, a package included in the software, in order to provide a better analysis of the study. The following figure shows the velocity field along the pipeline and in the outlet section (Fig 5a). This velocity profile is the same in all the cases simulated and means the velocity vectors are null on the wall and have maximum value in the center as can be seen on Fig. 5b, as it was defined in the boundary conditions.

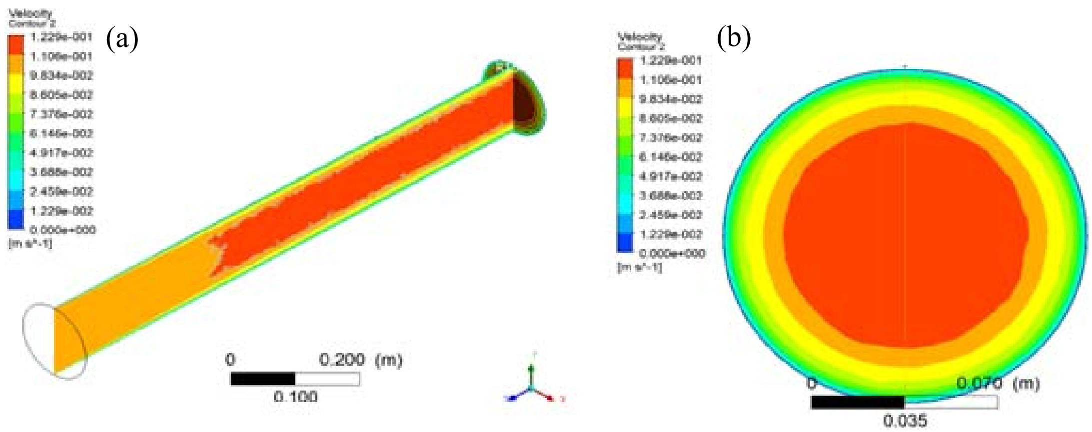

Figure 5. Pattern of velocity fields for all the cases.

Fluent-Solver calculates the mass flow rate in the outlet for each case. The computing results, already converted to unit's field, are shown in Table 3.

Table 3. Mass flow through the pipeline for all cases.

Case	Mass flow rate, kg/s	Inflow rate, bbl/h	Outflow rate, bbl/h	ΔQ_{leak}, bbl/h
1	1.47325	33.395	33.395	0
2	1.47321	33.395	33.391	0.004
3	1.47265	33.395	33.379	0.016
4	1.47111	33.395	33.343	0.052

The profiles of pressure and flow rate along of the pipeline length can also be analyzed for all the cases as is shown on Fig. 6 which presents the results analogously to the theory (Fig. 1).

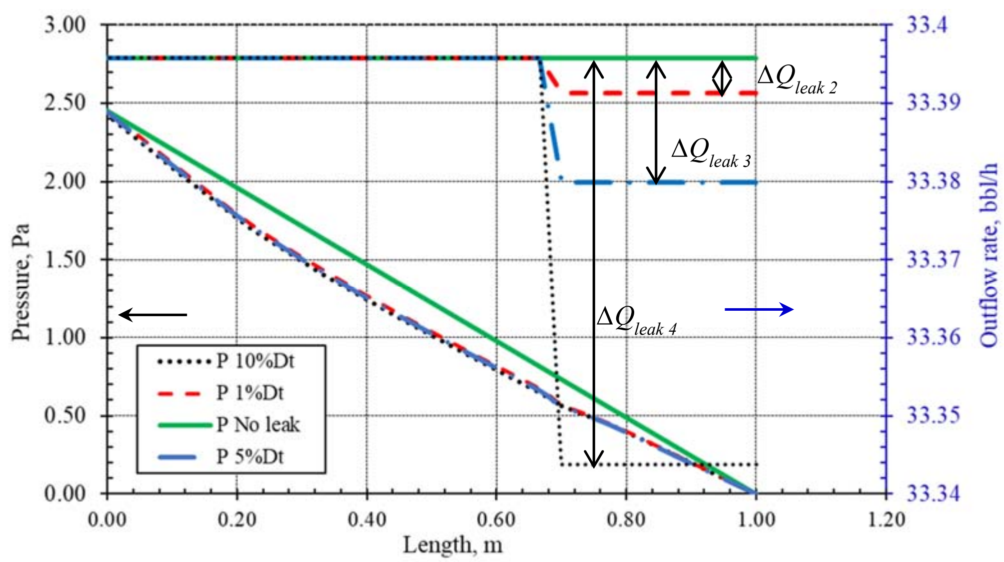

Figure 6. Change of pressure and flow rate along the pipeline.

The pressure gradient is constant when there is no perturbation in the system ("No leak" curve in Fig. 7). Although, when there is even a small leakage, the pressure field changes along the pipeline and not just on the position of the leak. This result is in accordance with the studies of Sousa *et al.* (2016) which affirm that the presence of a leakage influences both the position of the leak and also on its vicinity. The differences between the pressure curves is not so evident, as is shown on the theoretical curve (Fig. 1b), since the ratio between the diameter and length of the duct is small. This effect can be better observed on Fig. 7 that presents the gradient pressure in a larger scale, where it is easier to visualize the differences between the curves. In fact, in the leak region, about 0.70m, the slope in the curve is greater and influences in both upstream and downstream curve. Furthermore, it is possible to realize that the slope of the curve is more evident before the leak position and is softened after it. It happens because the flow is higher before the leak position and after it is lower. Accordingly, once the oil is an incompressible fluid, the velocity upstream of the leak results in higher frictional losses than downstream.

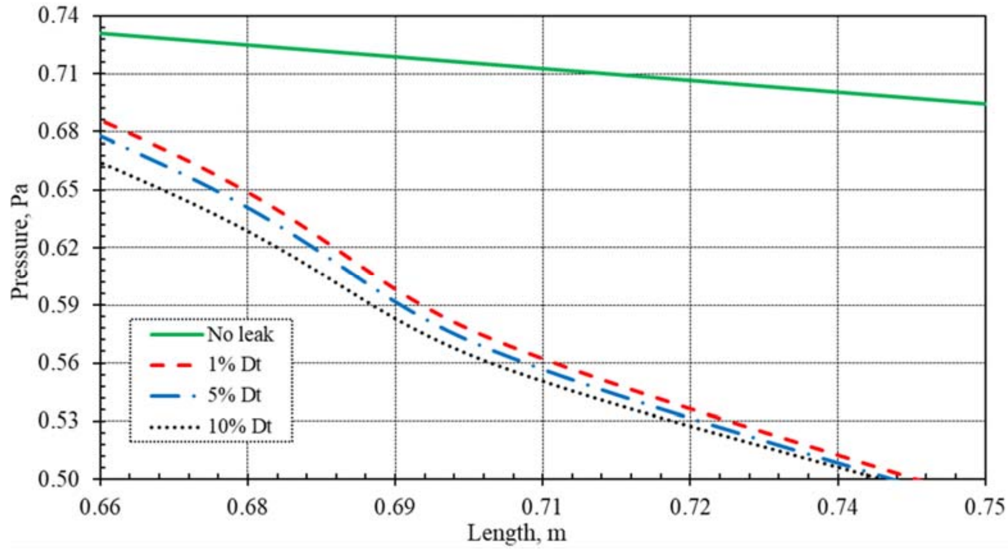

Figure 7. Change of pressure as a response to the leak size diameter.

It is also possible to observe from Fig. 6 the influence of the leak size in the outlet flow rate according to the secondary axis. When there is no leak, by the mass conservation Law, the inflow volume is the same in the outflow ($\Delta Q_{leak} = 0$). The appearance and growth of a leak in a pipeline reduces the outflow since one part of this mass flow is being released in the leakage, as is also possible to take this information from the Table 3 where the flow rate through the leak increases with the increase of the leak diameter ($\Delta Q_{leak2} < \Delta Q_{leak3} < \Delta Q_{leak4}$).

Using the CFD-post to analyze specifically on the position of the leak, it is possible to infer the flow pattern through the different sizes of leak using velocity vectors. On Fig. 8a, it is observed the proportion of the leak size comparing with the pipe length, while on Fig. 8b the leak region is highlighted in the presence of the velocity vectors. On Figs. 8c and 8d is possible to visualize, in different perspectives, the velocity vectors through the volume of control, focused in the leak area to investigate the flow pattern through it. Similar to Fig. 8, Figs. 9 and 10 demonstrate the leak area and its flow pattern for larger diameters, as shown below.

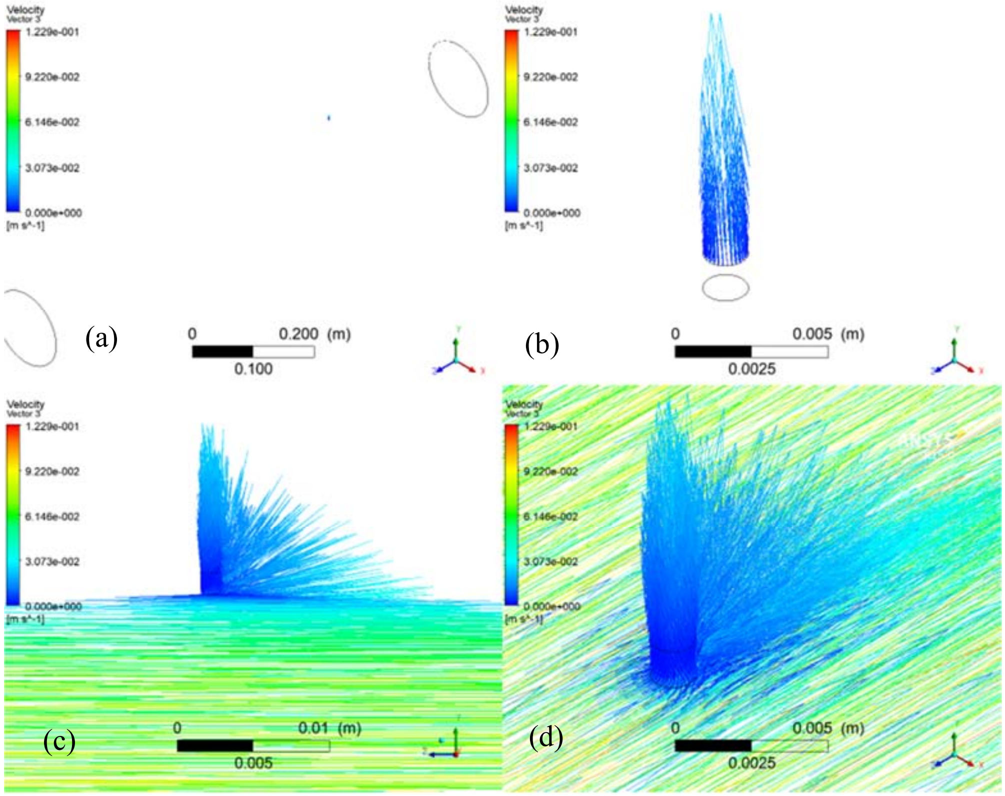

Figure 8 - Flow pattern in the leak of 1%Dt = 0.15 cm.

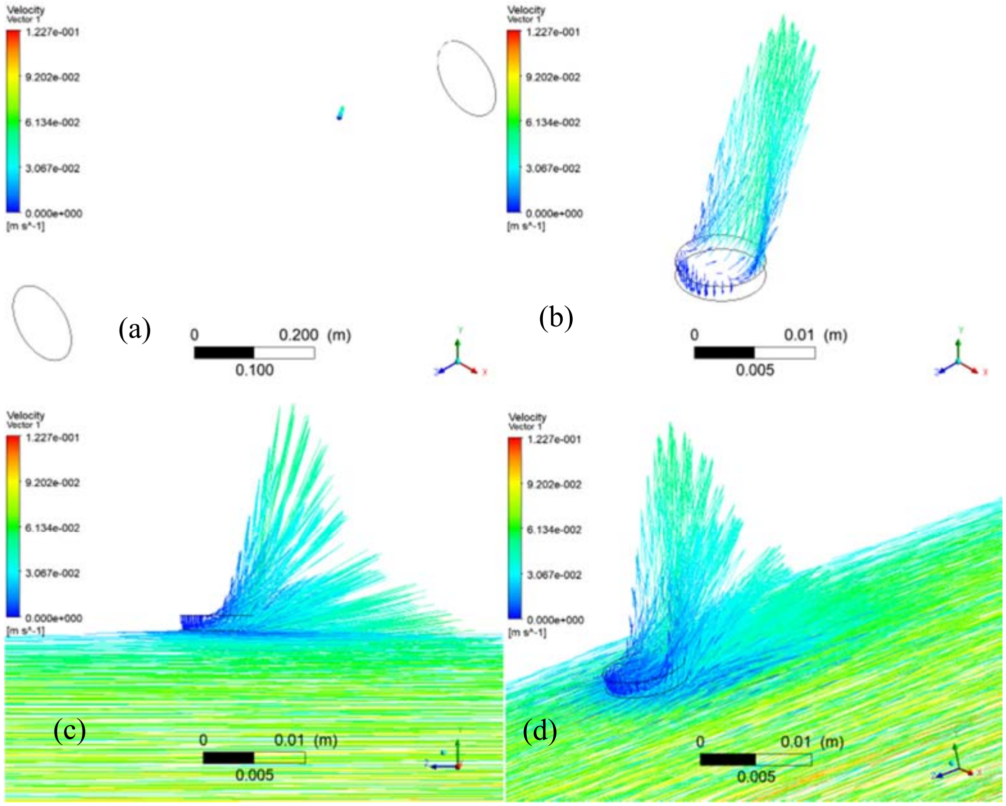

Figure 9. Flow pattern in the leak of $5\%D_t = 0.75$ cm.

As is shown in the figures, the greater the leak diameter, the larger is the flow rate released through it. However, it is also possible to observe an inflow through the leak, acting in this case as a reverse flow since in the setup of the Fluent was modeled just the volume of fluid and no other fluid in the boundaries. In other words, the same fluid coming out of the pipeline is the one that is coming in, but in different quantities. This result was also studied experimentally by Baptista *et al.* (2007) that concluded that the influence of both different geometrical parameters and the direction of the velocity vectors in the leakage flow pattern.

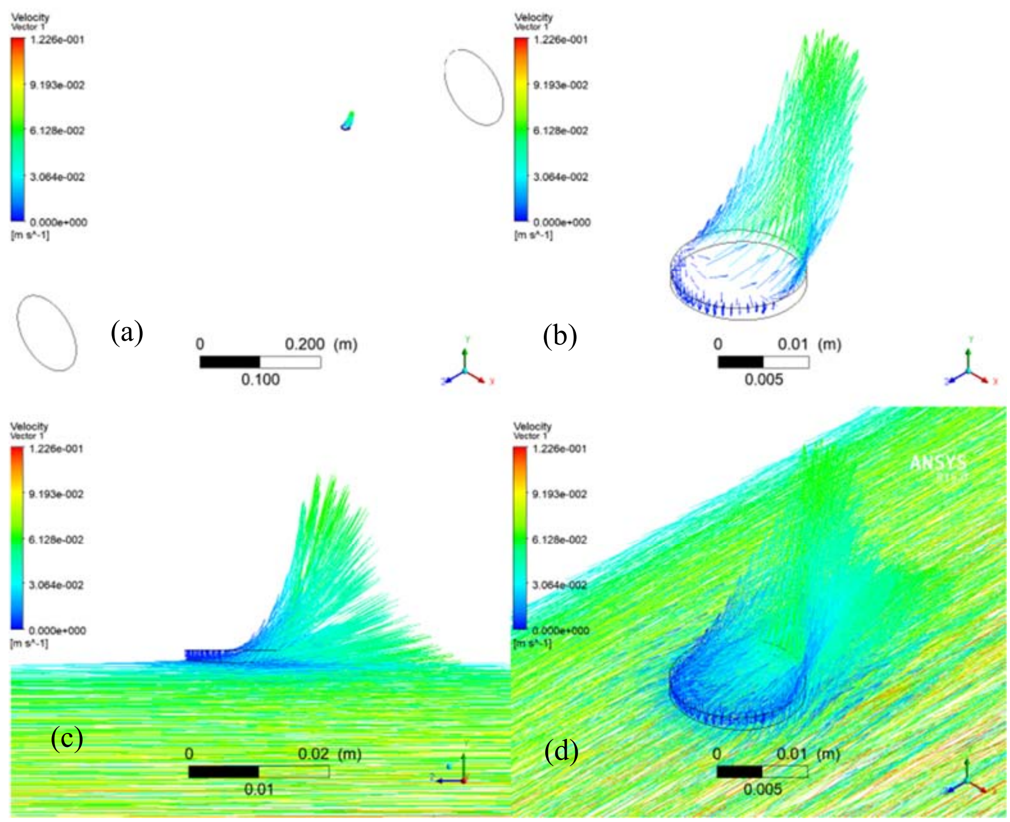

Figure 10. Flow pattern in the leak of 10%D_t = 1.5 cm.

4. Conclusions

The study was developed numerically using the software ANSYS FLUENT® 15.0 package in a monophasic, steady state and turbulent conditions. Is possible to conclude that by increasing the diameter of the leak orifice, the volume of fluid leaving the orifice is greater and this affects the hydrodynamics in the vicinity of the leak inside the pipe. Therefore, monitoring of the pressure and velocity fields along the pipeline is an extremely important tool to identify leaks, since these fields are affected by perturbations both upstream and downstream leak positions. It was also identified the reverse flow through the analysis of the results. In this specific this case, the fluid entering inside the pipe through the leak is the same that is being released, since no other fluid was modeled outside.

All the results obtained is in accordance with the literature and also with other experimental and numerical analysis already done. The real problem still requires further analysis and studies. Finally, it is concluded the importance to use computational tools for optimization purposes in a pipeline, as well as for an analysis and identification of possible problems that can arise with time like the wear of the pipeline.

4.1 Future analysis

(a) Taking this actual study to a more realistic situation in future analysis, the fluid inside the pipe will be changed for different oil viscosity, investigating how this parameter will affect the leakage and its hydrodynamics along the pipeline;

(b) Also, the pipeline length will be increase to greater values such that obtaining the solution be at reasonable computational cost in order to consider the presence of other leaks in the same pipeline.
(c) The simulation of a gas leak will also be explored in transient regime;
(d) According to Baptista *et al*. (2007) and Oliveira *et al*. (2009), the increase in petroleum exploration from shallow and deep and marine water raised the possibility of an oil leakage due to a pipeline rupture, therefore is important to analyze the occurrence of this perturbation considering a submarine pipeline in an offshore environment to study the effects of the hydrostatic pressure from the water blade in the leak. During a submarine leakage, the oil is expulsed from the leak while the water penetrates into the damage pipeline section through the same hole (Baptista *et al*., 2007), thus another point that have to be analyzed is this reverse flow, the effects of this multiphase flow inside the pipeline and how it can be quantified.

The reason for going deeper into this study is because the numerical simulation of pipeline leakage in subsea condition is relatively new and very promising research area.

Acknowledgments

The authors are grateful for the ANSYS *software* licensing, accessible in the Numerical Simulation Laboratory (Labsim) at the Ufes campus in São Mateus, ES, Brazil. We also thank the student chapter SPE/UFES <spe.ufes.br> for enabling the free use of the OnePetro <onepetro.org> platform of SPE - Society of Petroleum Engineers - in our campus.

References

Al-khomairi, A. M. Use of the steady-state orifice equation in the computation of transient flow through pipe leaks. The Arabian Journal for Science and Engineering, v. 30, n. 1B, p. 33-45, 2005.

Alonso, J. R. Technology for detecting and locating leaks in oil or gas pipelines. In: Rio Pipeline Conference & Exposition., Rio de Janeiro, RJ, Brazil, 2005. (in Portuguese)

Ansys Fluent – Solver Theory Guide. Ansys FLUENT Release 15.0. Ansys, Inc.U.S.A, 2013.

Araújo, M. V; Luna, F. D. T.; Barbosa, E. S.; Farias Neto, S.; Lima, A. Numerical study of oil flow in tee junction with leaks. Advances in Petroleum Exploration and Development, v. 6, n. 2, p. 1–11, 2013.

Araújo, M. V.; Farias Neto, S. R.; Lima, A. G. B.; Luna, F. D. T. Hydrodynamic study of oil leakage in pipeline via CFD. Advances in Mechanical Engineering, p. 170-178, 2014. DOI:10.1155/2014/170178.

Audunsson, T. H. Risk analysis of the oil depot in Örfirisey. Ph.D. Thesis, Lund University, ISSN: 1402-3504, 2006.

Baari, M. B. CFD simulation study on the effect of leak size towards oil leakage from submarine pipelines. Bachelor Thesis, University Malaysia Pahang, Pahang, 2014.

Baptista, R. M.; Quadri, M. B.; Machado, R. A. F.; Bolzan, A.; Nogueira, A. L.; Mariano, G. C.; Lopes, T. J. Effective Interfacial Tension and Geometrical Parameters Relationship for the Description of Oil Leakages from Submarine Pipelines. 8º International Conference on Chemical & Process Engineering; v.11, p. 401-406, 2007.

Ben-mansour, R.; Habib, M. A.; Khalifa, A.; OUcef-toumi, K.; Chatzigeorgiou, D. Computational fluid dynamic simulation of small leaks in water pipelines for direct leak pressure transduction. Computers & Fluids, 57, p. 110-123, 2012. Science Direct. DOI: 10.1016/j.compfluid.2011.12.016

Edrisi, A.; Kam S. I. Mechanistic Leak-Detection Modeling for Single Gas-Phase Pipelines: Lessons Learned from Fit to Field-Scale Experimental Data. Advances in Petroleum Exploration and Development, v.5, n.1, p.22–36, 2013.DOI: 10.3968/j.aped.1925543820130501.1027.

Jujuly, M. M. Computational fluid dynamics (CFD) based approach to consequence assessment of accidental release of hydrocarbon during storage and transportation. [M.S. thesis], University of Newfoundland, Newfoundland, 2016.

Khaksarfard, R.; Paraschivoiu, M.; Zhu, Z.; Tajallipour, N.; Teevens, P. J. CFD Based Analysis of Multiphase Flows in Bends of Large Diameter Pipelines. NACE International, 17 March, 2013.

Kyriakides, S.; Corona, E.; Mechanics of Offshore Pipelines: Volume 1: Buckling and Collapse. Austin: Elsevier, 2007. Chapter 1.

Macias, I. B.; Cruz, S. L.; Pereira J. A. F. R. Leak detection in pipelines. In:2nd Mercosur Congress on Chemical Engineering and 4th Mercosur Congress on Process System Engineering, Rio de Janeiro, RJ, Brazil, 2005.

Oliveira, B. Y.; Mariano, G. C.; Quadri, M. B. A CFD study of oil leakage from submerged pipelines. In: Congresso Brasileiro de Engenharia Química em Iniciação Científica, 8., 2009, Uberlândia. Anais... São Paulo: ABEQ, 2009. (in Portuguese)

Pratt, D. B. Reducing Leaks Offshore - A U.K. Regulatory Initiative. Society of Petroleum Engineers, 2002. DOI: 10.2118/73924-MS

Romero, O. J. A conservative method for a transient analysis of gas/liquid in pipes. [M.S. thesis], Pontifícia Universidade Católica do Rio de Janeiro, Rio de Janeiro, 1999. (in Portuguese).

Selvarajah, A. CFD simulation study on the effect of water velocity towards oil leakage from submarine pipelines. Bachelor Thesis, University Malaysia Pahang, Pahang, 2014.

Sousa, C. A.; Miranda, S. M; Benedito, U. M. M.; Romero, O. J. Numerical simulation of transient in pipelines. Latin American Journal of Energy Research v. 3, n. 2, p. 21 – 29, 2016. DOI: 10.21712/lajer.2016.v3.n2.p21-29. (in Portuguese)

Sousa, V. N.; Lima, A. G. B.; Farias Neto, S. R., Numerical analysis of heavy oil-water flow and leak detection in vertical pipeline. Advances in Chemical Engineering and Science, v. 3, n. 1, p. 9–15, 2013.

Stewart, G. Line-pipe. Londres, p.540-541. Society of Petroleum Engineers, 1933.

Tavares, D. P. S. A. Non-isothermal three-phase flow in horizontal ducts with leakage: modeling and simulation [M.S. thesis], Federal University of Campina Grande, Campina Grande, Brazil, 2012. (Portuguese).

Wang, S.; Wu, G.; Wang, Y.; Zhou, Y.; Qi, H.; Li. D.; CFD for multiphase flow transport of buried crude oil pipelines leakage. ResearchGate, p. 861- 874, 2012. DOI: 10.1061/9780784412619.090.

Wilcox, D. C. Turbulence modeling for CFD. La Cañada, CA: DCW Industries Inc., 1993.

Wylie, E.; Streeter, V. Fluid Transients in Systems. Englewood Cliffs: Prentice Hall, Inc., 1993.

Zhang, Y.; Chen, S.; Li, J.; Jin, S. Leak detection monitoring system of long distance oil pipeline based on dynamic pressure transmitter. Measurement, v. 49, n. 1, p. 382–389, 2014.