Journal of Applied Fluid Mechanics, Vol. 16, No. 12, pp. 2529-2539, 2023.
Available online at www.jafmonline.net, ISSN 1735-3572, EISSN 1735-3645.
https://doi.org/10.47176/jafm.16.12.1928

Transient Analysis of the Flow Behavior under a Small Leakage Accident in Feed Water Pipeline

A. Dahia†, A. L. Deghal Cheridi and M. Boumaza

Nuclear Research Center of Birine, B. P. 180, Ain Oussera, Djelfa, 1720, Algeria

†*Corresponding Author Email:* a.dahia@crnb.dz

ABSTRACT

Feedwater leakages due to excessive loads and cracking caused by corrosion or fatigue failure can affect the reliability of the production facilities. In the present work, a numerical study of a small leakage accident type SB-LOCA on the feed water pipeline was investigated using Computational Fluid Dynamics (CFD) and Relap5 computer codes. The aim is to understand the behavior of the incompressible water flow and its effect on the relevant parameters at the leakage location vicinity, including the mass flow rate, velocity, pressure, and temperature. For this, a mathematical model was developed and validated to evaluate the release of water through the pipe, which is mainly based on the variables that may affect the leakage. The results of CFD show that the leakage has important effects on the distribution of main parameters of the water flow through the pipe, which has an identical outcome from the Relap5 code simulation. The change of fluid velocity only has a little impact on the flow behavior at the leakage region.

Article History

Received April 16, 2023
Revised July 23, 2023
Accepted July 23, 2023
Available online October 8, 2023

Keywords:

Modeling and simulation
CFD
Relap5
Transient analysis
Small break accident

1. INTRODUCTION

Piping has proven to be the simplest, safest, and most economical means for supplying and distributing all types of fluids (Sun et al., 2019; Mutiu et al., 2021). The transport of refrigerant or supply fluids in an industrial installation requires the use of high-quality of pipes with very long service life, especially if the fluid to be transported is subject to excessive temperature or corrosion stresses (Ben-Mansour et al., 2011).

The pipes can be exposed to several problems like cracks and leaks, which are very common phenomena that are mainly due to corrosion, metal fatigue due to external stresses, erosion of welding points, faults in the pipes, joining of pipes or equipment and often a lack of maintenance (Araújo et al., 2013; de Sousa & Oldrich, 2017; Zeng & Luo 2019). Timely, the detection and the identification of leaks can avoid significant loss of water and serious damage to the facility. Hence, it is very important to study these phenomena and analyze their consequences on the performance of the installation.

In the industrial heated plant like boiler several problems can arise during its service, because it works in severe conditions (high temperature, high pressure), corrosive environment and continuous operation. The lack of water is undoubtedly the most serious incident that can occur, because the water level in the balloon drops rapidly and the walls of the tubes are overheated. For this, the water level in the boiler must be constantly monitored and maintained. Generally, among the problems that can occur as a result of the lack of water is the rupture in the tubes. Consequently, thermal shocks cause cycles of restricted expansion, generate stress gradients that will generally initiate on geometric surface defects (Michel & David, 1991). Thus, corrosion or fatigue frequently favor the rupture of the tubes.

To this end, in the case of this incident, interventions during operations must be carried out; if the water leak is small, the level must be maintained and the boiler must be taken out of operation. If the leakage loss is large and the water level cannot be maintained despite the pumping rate by the feed pump being too high, the boiler must be shut down immediately to maintain the water level in the drum and ensure the cooling of the combustion chamber. In this case, the boiler can be drained and a complete inspection of all pressurized parts carried out in order to define the extent of the damage (Paolo, 2006).

In order to ensure the safety of industrial installations, scientists have been able to use computer codes for numerical simulation purpose, to study the thermal-hydraulic behavior and physical phenomena of industrial facilities in normal and accidental operations. Among the accidental transients, we cite the rupture in the pipes. For

NOMENCLATURE

CFD	Computational Fluid Dynamic	g	gravitational force
RMS	Root Mean Square	u	velocity vector
SB-LOCA	Small Break-Loss Of Coolant Accident	ρ	water density
t	time	α	SST model coefficient
T	temperature	D	tube diameter
°*K*	kelvin	d	leakage hole diameter
°*C*	Celsius	μ	dynamic viscosity
p	pressure		

the simulation of water leakage in pipeline after rupture, it is necessary to introduce the size and position of the leak in the pipe, the flow rate of leak, the velocity and pressure of discharge, and subsequently the total safety damage.

Several works have been carried out to investigate the leakage in pipeline flows. Ben-Mansour et al. (2011) conducted a numerical simulations using a 3D turbulent flow model of CFD code on a small leak in a pipe with a rectangular shape leakage. The results show that the presence of a leak causes clear fluctuations of pressure gradient along the pipe. de Sousa et al. (2013) investigated the behavior of a two-phase flow leak in a vertical pipe using ANSYS-CFX code. The obtained results show that the volumetric fraction of the two phases and the velocity of the fluid mixture are the main parameters that influence the drop of the pressure and mass flow at the leakage location. Shehadeh and Shahata (2013) performed a realistic CFD analysis of flow leakage in a pipeline for many geometrical forms and fluid properties. They conclude that leaks at a high mass flow rate are more easily detected than those at a lower mass flow rate. Zhao and Tan (2011) established an experimental device for gas pipe leakage and proposed a computation formula to treat, for validation purposes, a sub-critical flow leakage rate for a large hole. Kostowski and Skorek (2012) made an interesting comparison between experiments and CFD simulation to see the applicability of adiabatic and isothermal models, for different leakage positions and directions. Zheng and Hong (2017) studied the leakage sizes and initial pressure impacts on the leakage rate by using a numerical model simulation for a small leakage hole in pipes, and then experimentally validates the model's correctness. The results show that using the same section size, the leakage increased gradually with the initial pressures. Fu et al. (2014) estimated the velocity distribution in a pipe and use it to input the flow rate and pressure on the leakage hole. Araújo et al. (2014) studied the flow velocity influence on the pressure transient behavior of oil leakage in a pipe through fluid dynamics simulations. The results indicate that the leakage causes a disturbance in the velocity and pressure fields, and the greatest pressure has signed at the leak closest to the pipe entrance. Ong and Siti (2019) investigated the effect of fluid velocity on the leak flow rate, the pressure distribution, and turbulence kinetic for single and double leaks subsea pipelines using the k-ε model at steady-state conditions. They concluded when the fluid velocity increases, the flow rates at both leaks increase. Cheridi et al. (2016) investigate the loss of feedwater in a steam boiler caused by a rupture on a portion of the pipeline using the Relap5 system code at transient. The main objective is to assess various accident scenarios in real plant working conditions. It was demonstrated that in case of control system failure, the heat transfer degradation occurs between the heating wall and the fluid causing a rise in wall temperature, which leads the fusion of the evaporation tubes in the furnace.

Accordingly, this work aims to find the most effective tool to develop a calculation model to simulate, at transient conditions, the loss of water caused by an instantaneous rupture happened at the feedwater line in an industrial installation (Fig. 1) using Computational Fluid Dynamics (CFD) and Relap5 computer codes. This is one of the most frequently encountered accidents, which disturbs the correct operation of the plant. The accident simulation is carried out after the establishment of the steady-state condition in the system.

2. MATERIAL AND METHODS

2.1 Facility Description

The studied installation is a water-tube, radiant, natural circulation type steam boiler. It is installed at the Natural Gas Liquefaction complex (NGL) of SONATRACH Company located in Skikda in Algeria, to produce superheated steam of 374 tons/h to drive a turbine. The facility is composed of three main parts: the main feedwater line, steam generator, and main superheated steam line like it is presented as Fig. 1. The principal steam boiler components are: the combustion chamber, the drum, the downcomers tubes, the economizers, superheaters, centrifugal pump, collection tank, monitoring and control systems, and a set of valves and flappers.

The feedwater is pumped from the collection tank towards the steam generator. Before entering the drum, it is heated through the economizers located in the combustion gases flow exiting the furnace to improve thermal efficiency. Afterward, the feedwater flows to the evaporator tubes where is partially transformed into steam. The water/steam mixture rises towards the drum, and the loop of circulation is carried out by natural circulation due to the density difference between the water in the downcomers' tubes and the water/steam mixture in the combustion chamber. Inside the drum, the steam naturally rises from the water; the steam leaves the drum towards the superheaters, and the water is collected in the liquid compartment. The main functioning parameters of the plant are presented in Table 1.

2.2 Methodology

The feedwater loss is among accidents, which can occur in a steam boiler installation, it can be caused by

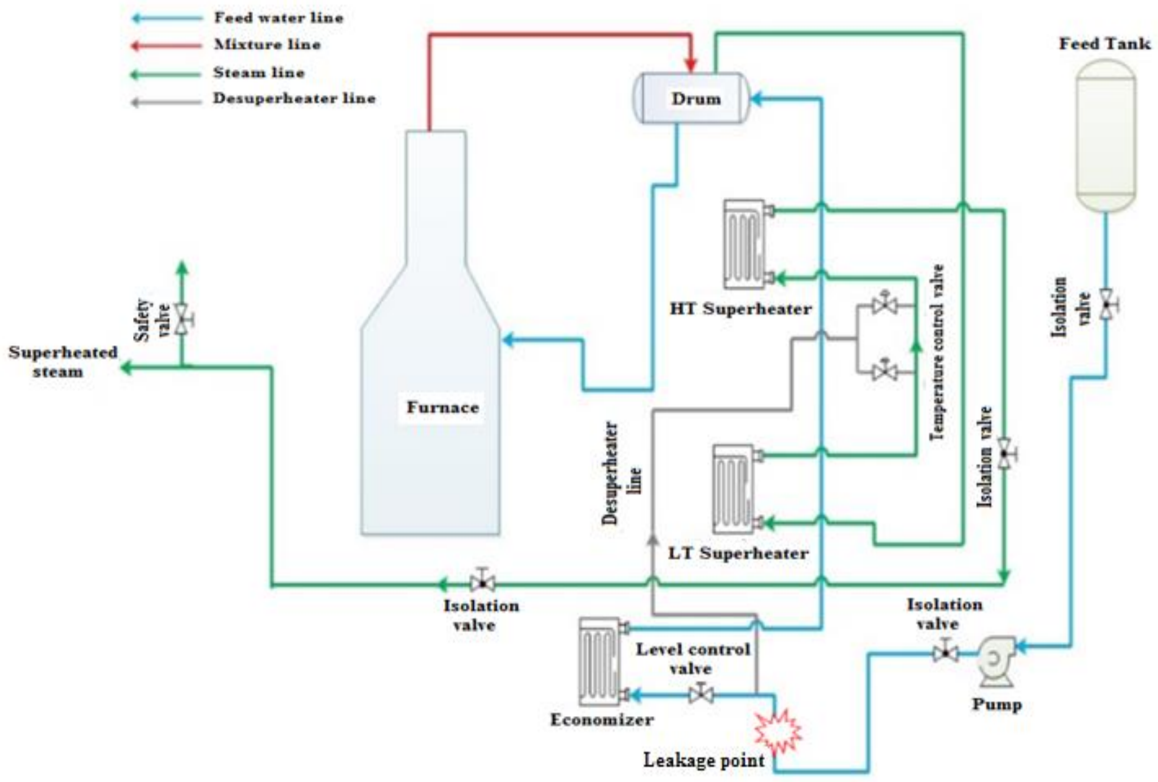


Fig. 1 Diagram of the main components of the facility

Table 1 Main operating parameters of facility (Cheridi et al., 2019a)

Technical parameters	Units	Values
Mass flow rate of feed water	kg s^{-1}	103.88
Feed water temperature	°C	119.0
Superheated steam	°C	487.0
Efficiency of the facility	%	92.0
Drum pressure	bar	76.9
Air flow rate	$Nm^3 h^{-1}$	344.8
Natural gas flow rate	$Nm^3 h^{-1}$	45.69
Fumes mass flow rate	$Nm^3 h^{-1}$	390.49
Air excess	%	1.3

feedwater pump failure, pump power loss, closing the valve of control, or leakage and ruptures in the main pipeline.

In the present investigation, a numerical simulation of water loss caused by a rupture in the main pipeline is carried out using two codes; the first one is Relap5 system code to predict the thermal hydraulic behavior of the all components of the entire installation during the accidental transient, and the second one is 3-D turbulent flow model in Computational Fluid Dynamics (CFD) to perform the local effect of the leak on the ruptured tube with more details, and capture the produced phenomena.

2.2.1 Relap5 Treatment

The thermal-hydraulic analysis code system Relap5 is designed to simulate the transient behavior of Low Water Reactor (LWR) systems under a large variety of supposed accident cases (Cheridi et al., 2019b).

Relap5 code was developed at Idaho National Engineering and Environment Laboratory (INEEL) for the US Nuclear Regulatory Commission (NRC) (Carlson et al. 1990). The code is used to simulate a wide variety of thermal and hydraulic analyses in nonnuclear and nuclear systems involving steam/water mixtures and non-condensable gases (Rahmani et al., 2009; Kaliatka & Valincius, 2012), which is considered the best to perform specific applications such as Loss Of Coolant Accident (LOCA). The hydrodynamic model of Relap5 is based on six equations system (nonequilibrium and nonhomogeneous) for two-phase flow that is solved by a partially implicit numerical method and uses the finite difference method to treat heat transfer problems. It is considered a generic code that has several component models such as pumps, valves, pipes, control systems, heat structures, etc. allowing for the simulation of an installation.

The main motivation to this study is to evaluate through computational tools and numerical simulations the leakage in a pipeline. The influence of the leak on the entire flow parameters and the behavior of the fluid have been widely treated.

The utilized approach to model the facility is to subdivide the system into control volumes connected by flow junctions. So, the nodalization diagram of the studied steam boiler plant is presented in Fig. 2.

To model the break, which is located in the vertical part of pipe 210 before the economizer, we have used Trip-Valve component 666 connecting to the Time-Dependent-Volume component 601 to impose the atmospheric conditions like it is presented in Fig. 2. The opening section of the trip-valve 666 corresponds to the

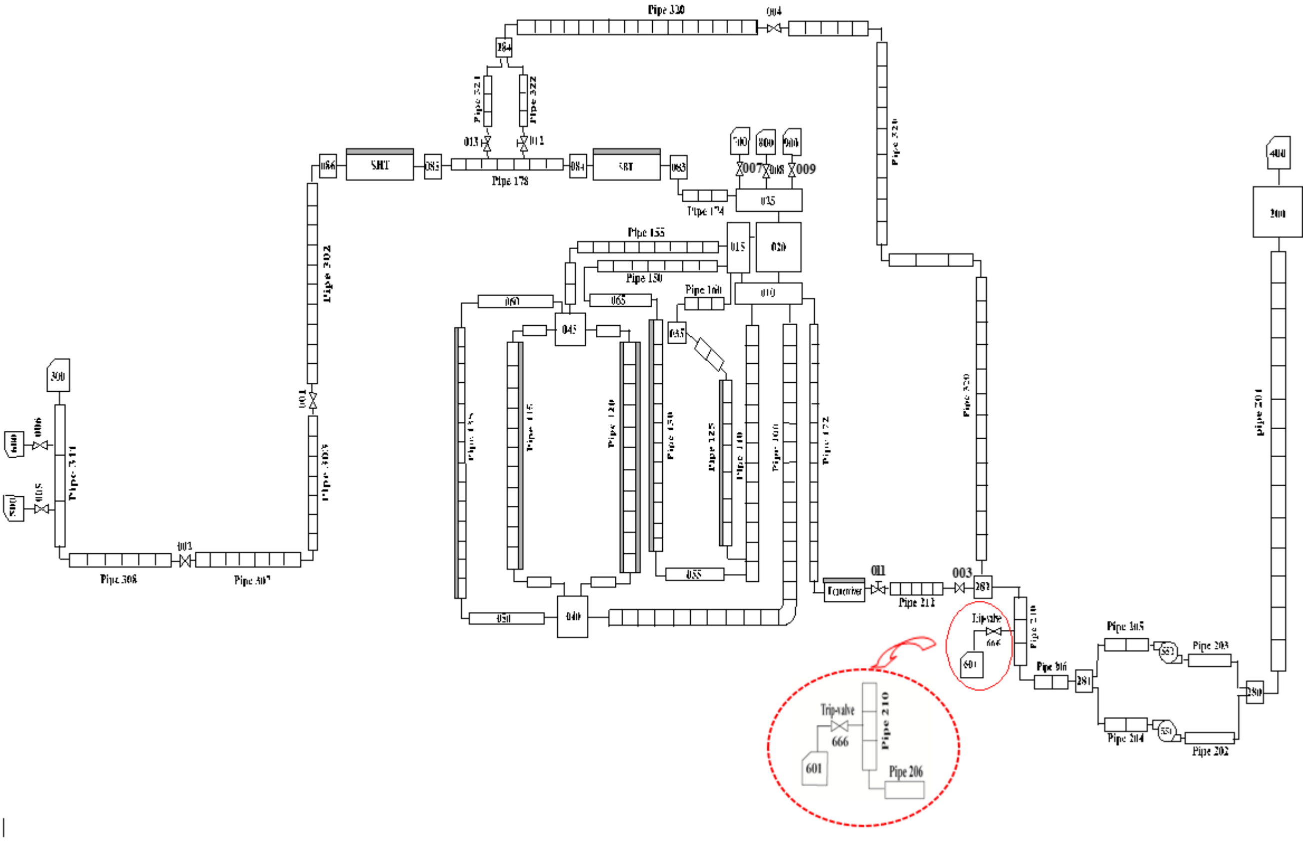


Fig. 2 Nodalization diagram of the plant and the break

Table 2 Imposed sequences of the transient

Time	Events
0 to 100s	Steady state condition
t = 100s	Start of transient
300 s	Feedwater pump shutdown
450 s	End of transient

break flow area, which is about 78.54 cm^2 (Table 3). The time of the simulation was applied in the range of 0-100 seconds for steady-state and 300 seconds for transition conditions. After 100 seconds of the installation steady state operation, the transient scenario starts by opening the Trip-valve 666 with a section corresponding to the break area. Table 2 summarizes the sequences of the accidental transient.

2.2.2 CFD Treatment

Three-dimensional, transient and turbulence model were considered in the calculation using CFD-CFX code. The system mass remains constant over time to all transfers of matter and energy, for this reason the law of mass conservation must be applied. The equation of mass conservation for water flow can be written as (Fu et al., 2020):

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho\vec{u}) = 0 \qquad (1)$$

The law of momentum conservation is governed by Equation 2 as follow consider an unchanged total momentum of the system and all external forces are neglected.

$$\rho\frac{\partial\vec{u}}{\partial t} = \rho\vec{g} - \nabla p + \mu\Delta\vec{u} = 0 \qquad (2)$$

Where ρ is the density (kg m^{-3}), t is the time (s), $\vec{u}$ is the velocity vector (m s^{-1}), $\rho\vec{g}$ is the gravitational force (m s^{-2}), p is the pressure (Pa), and is the dynamic viscosity (Pa s^{-1}).

The selection of an appropriate model scheme of turbulence is very crucial in the flow modeling. The k-ω SST (Shear Stress Transport) model was chosen in this work because it combines the robustness and the accuracy of k-ω model in the region close to the wall and the efficiency of k-ε far from the wall. The model of SST is mainly advised for fluids may encounter unexpected stress changes.

The relation between the two formulations of the two models is made through a transient function which can be presented by the following equation (Menter et al. 1994):

$$SST = \alpha(k-\omega) + (1-\alpha)(k-\varepsilon) \qquad (3)$$

Where α=0 in the flow centerline and α=1 near the wall.

The feedwater that circulates inside the pipeline flows directly into the air at rest through leakage. The leakage point was located at the middle part of the pipe with a total break section of 78.54 cm^2, and a leakage flow rate of 94.5 kg s^{-1} (Table 3). The hole on the pipe, which is source of leaks, was assumed to be circular.

Three parts of the pipe subject of leakage will be considered in the simulation; the first one concern the

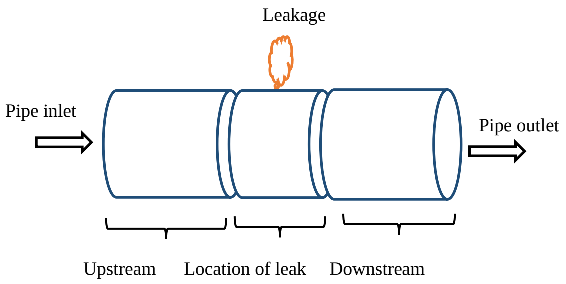


Fig. 3 Physical model and coordinate system (leak center at x = 0, y = D/2, z = 0)

Table 3 Main parameters used in the CFD simulation

Parameters	Units	Values
Inlet pressure	bar	93
Tube length with 3 control volume	m	3.0078
Control volume length	m	1.0026
Tube diameter	m	0.188
Total break section	cm^2	78.54
Flow section	m^2	2.80×10^{-2}
Leakage rate	$kg\ s^{-1}$	94.5

exact location of the leak, the second and third one concern the downstream and upstream of leakage (Fig. 3) (Araújo et al., 2014; Edrisi & Kam 2013).

The considered fluid is the water with monophasic propriety of the density of 998.2 kg m^{-3}, and viscosity of 0.001003 kg m^{-1} s^{-1}. The flow regime is considered turbulent, and the mass flow rate in the inlet of the main pipe is specified to be 103.88 kg s^{-1} (Table 1) with a Reynolds number of 2.02×10^7. The outlet pressure and the pressure at the leak are both atmospheric gauge pressure.

Icem-CFD software was used to establish geometric and meshes. The best quality of mesh could directly affect the computational results. For this reason, and because the flow is turbulent, 557976 (Five hundred fifty-seven nine hundred seventy-six) elements with hexahedron meshes were agreed to make the calculation. To predict correctly the behavior of all parameters during the simulation process, the boundary conditions must be carefully chosen. This alternative pushes us to choose a boundary condition of mass flow rate at the pipe inlet. An open pressure outlet was set at the main pipe outlet and the leakage section, with a very refined mesh for the latest. A boundary condition of no-slip was imposed in the wall, and a fixed Pressure-Outlet boundary condition has been adopted for the free boundary of the calculation domain and set to the atmospheric pressure.

The calculations are made with same time steps of 0.05 s, and residuals variables are calculated less than 10^{-6}. The computation is performed using a pressure-based solver, while the pressure fields are coupled with the velocity fields using SIMPLE pressure-velocity coupling scheme. The calculation is carried out on station work with 82 processors and 32 GB of RAM. Figure 4 shows the layout of all computational domain with a tetrahedral uniform grid, and the small area, which contains the leakage hole with a structured mesh for the smaller and larger grid. The water volume fraction is initially set equal to zero, and the air volume fraction is initially set to one.

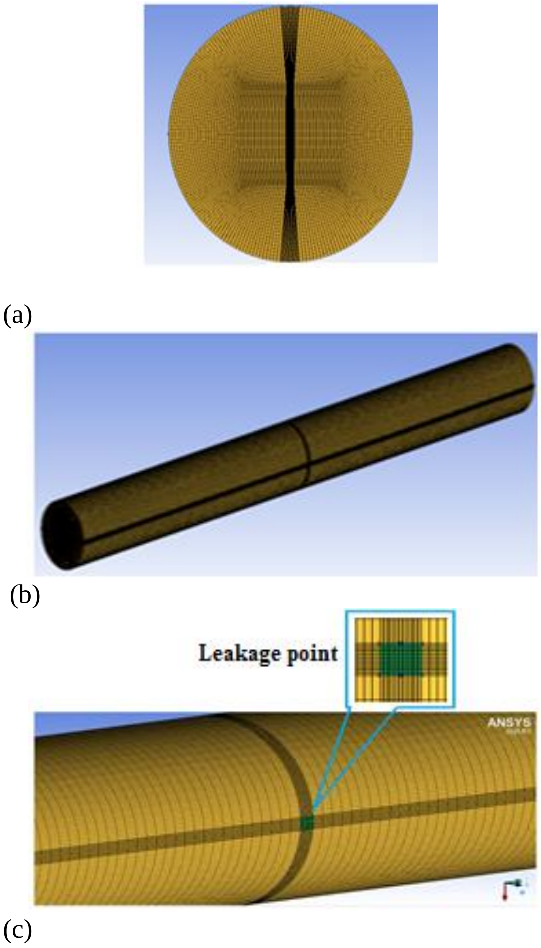


Fig. 4 Geometry and mesh of the pipe with leakage point; a) cross section, b) longitudinal section, c) leakage section

2.2.3 Qualification

Generally, the CFD codes validation should be effectuated using the experimental data before application to transient cases (Fan et al., 2016). In our case, it is difficult to perform the accident of loss of feed water type LOCA. For this, we proceeded to compare and validate the CFD simulation results with the results of Relap5 model of the steam boiler at steady-state condition using the available operating data of the plant. The comparison offers information on the nodalization quality and the good choice of boundary and initial conditions. As is presented in Table 4, the Relap5 results agree well with the installation operating data. From this, the steam boiler model is considered validated at steady-state and it can predict the thermal-hydraulic behavior of the installation under accidental transient conditions. In addition, some parameters like flow rates, temperatures, pressures, and water level are selected to demonstrate the establishment of the steady state (Fig. 5). The total calculation was effectuated for 5000 seconds, and the oscillations observed at the beginning of the calculation are due to search for operating conditions from initial conditions.

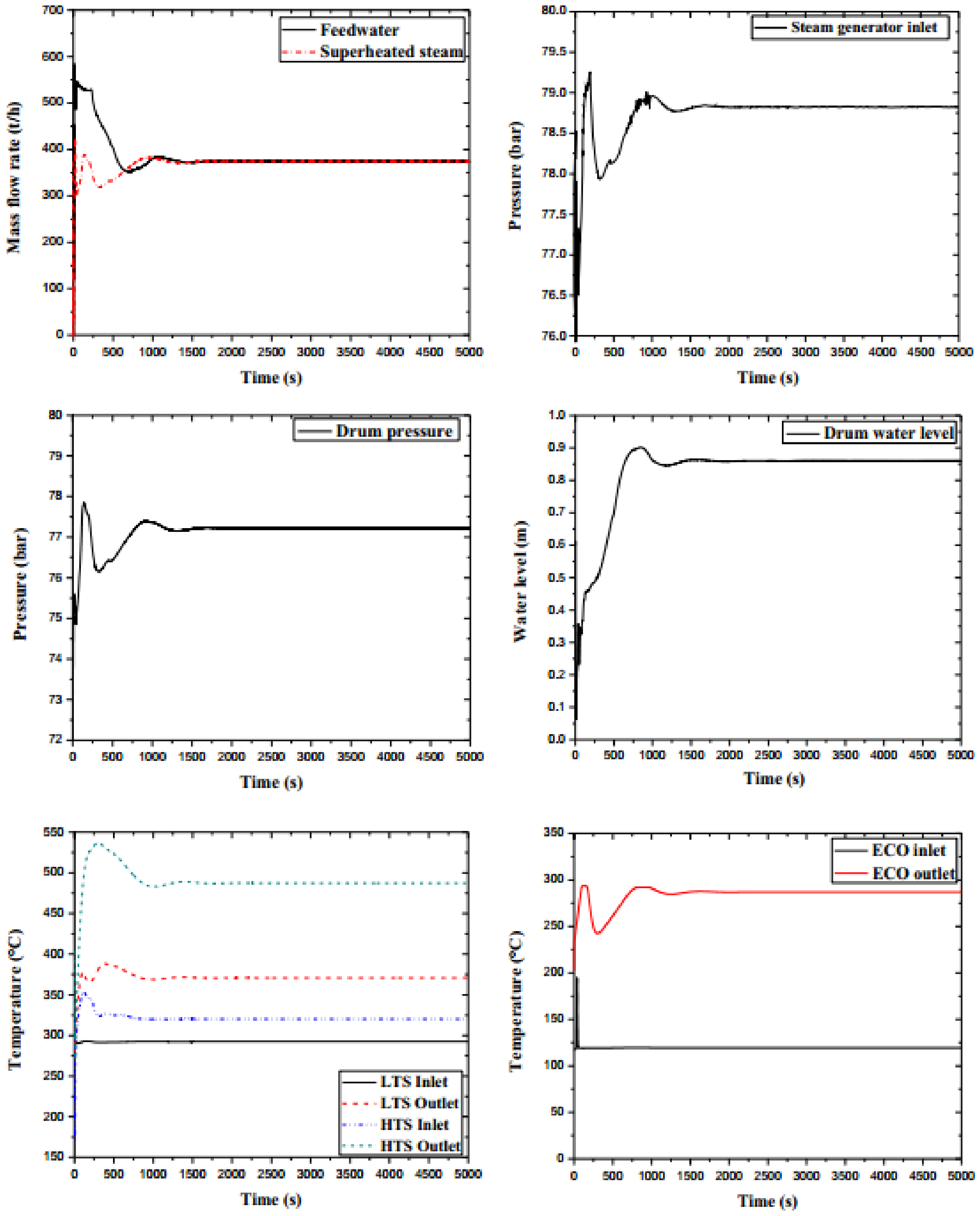


Fig. 5 Thermal-hydraulic parameters of the plant during steady-state operating conditions (Cheridi et al., 2019a)

3. RESULTS AND DISCUSSIONS

The Computational Fluid Dynamics (CFD) software and Relap5 system code are used to simulate the accidental transient of a tube rupture in a feedwater pipeline. The adopted numerical approach has the advantage to describe and analyze the most complex geometry in a very short discretization time (de Vasconcellos et al., 2014). In the CFD simulation, the SST turbulence model was chosen for this purpose. The governing equations and two transport equations of fluid flowing in the pipeline are applied to predict precisely the effects of the turbulence in transient.

The numerical results are shown in Figs 6-16, the numerical simulation was performed for up to 450 seconds including steady state and transient time. During the steady-state operation of the plant, the flow rate of feedwater is constant. After the break opening, a part of the feedwater leaks through the break, which causes the drop of water in the steam drum, the downcomer water flow rate decreases, and the flow rate of steam leaving the plant drops immediately. At this moment, the control

Table 4 Operating parameters of the facility

Boiler parameters	Units	Experimental data	Simulation data
Feed water flow rate	t h^{-1}	374.0	374.1
Steam flow rate	t h^{-1}	374.0	374.5
Desuperheater flow rate	t h^{-1}	25.0	26.1
Inlet Economiser temperature	°C	118.0	119.0
Outlet Economiser temperature	°C	287.0	287.1
Outlet drum steam temperature	°C	292.0	292.4
Inlet LTS temperature	°C	292.0	292.3
Outlet LTS temperature	°C	370.0	370.8
Inlet HTS Temperature	°C	322.0	320.1
Outlet HTS temperature	°C	487.0	487.3
Drum water level	mm	860.0	860.0
Pressure at collection tank	bar	1.89	1.9
Drum pressure	bar	76.9	77.2
Inlet steam generator pressure	bar	82.0	78.2
Outlet steam generator pressure	bar	73.0	73.2
Outlet pump pressure	bar	91.93	94.2

systems of the water level give the signal for opening the main supply valve to maintain the water level at its nominal value of 860 mm, consequently, the flow in break increases. Once the nominal value of the water level is reached, the main feed water valve close gradually, and the feedwater pump shutdown at 300 seconds.

Figure 6 shows typical velocity fluctuations corresponding to Root Mean Square (RMS) recorded for 450 seconds. The RMS visualizes the important flow phenomena associated with the turbulent flow inside the main pipe and outside the leakage hole. Hence, after simulation, the RMS leads to a very smaller value of less than 10^{-4}. This negligible value doesn't affect the final expected results and increases the accuracy of numerical simulation.

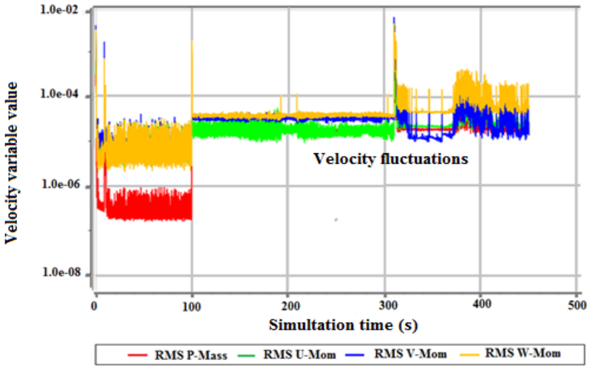


Fig. 6 Velocity fluctuations computed at 450s simulation time

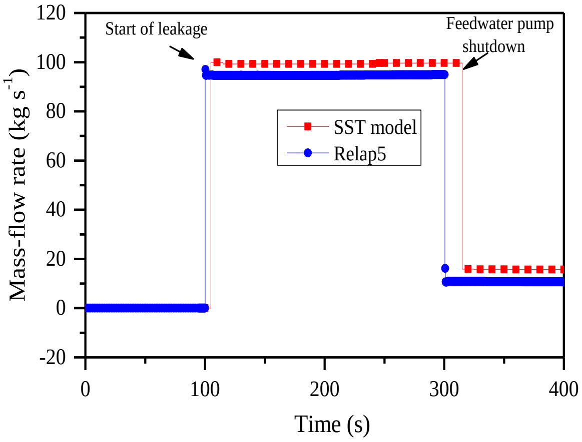


Fig. 7 Mass flow rate of feed water in leakage section

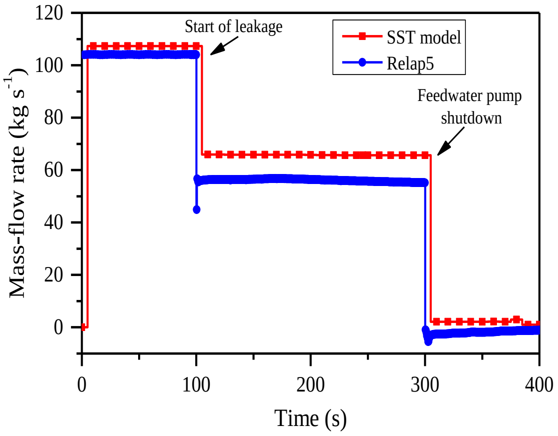


Fig. 8 Mass flow rate of feed water after leakage section

Figure 7 displays the simulation results effectuated by SST and Relap5 codes of the leakage mass flow rate as a function of time. We can see that the profiles are similar. At the steady state, the leakage flow rate is null, and after the occurrence of the accident at 100 seconds, the flow rate increases. However, the inertial forces of the flow become less than the forces caused by the pressure differential between the inside and outside of the pipeline leaking (pressure gradients) (de Sousa et al., 2013). After 300 seconds of transient time, the feedwater pump was stopped, which caused a decrease in the break mass flow rate remained constant until the end of the transient. The feedwater flow rate for both codes after the leakage section is presented in Fig. 8. Before the rupture, the flow rate is constant and equal to 103.5 kg s^{-1}. After that, it decreases rapidly reaching a new value due to loss of water

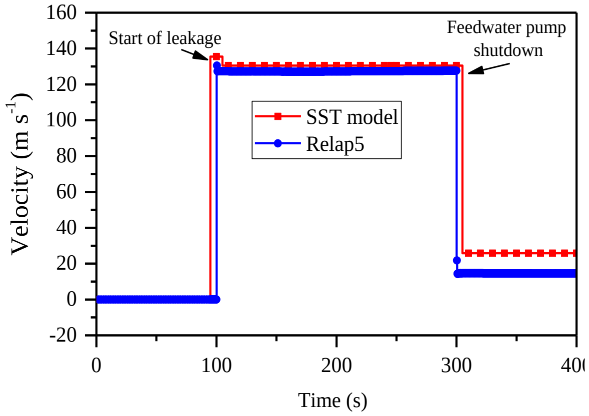


Fig. 9 Velocity of feed water in leakage section

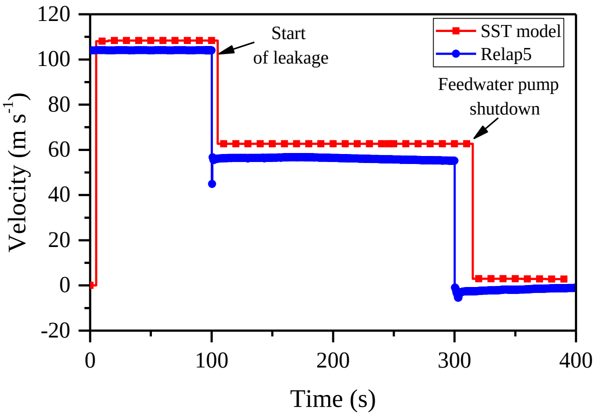
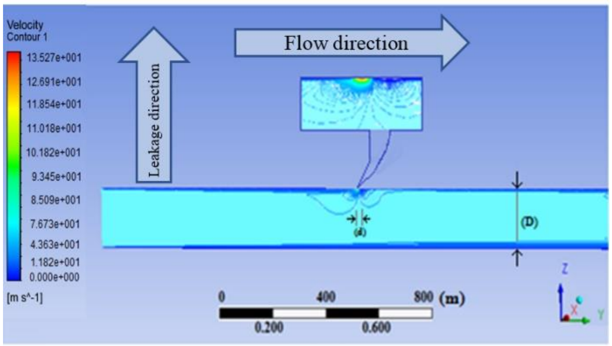


Fig. 10 Velocity of feed water after leakage section

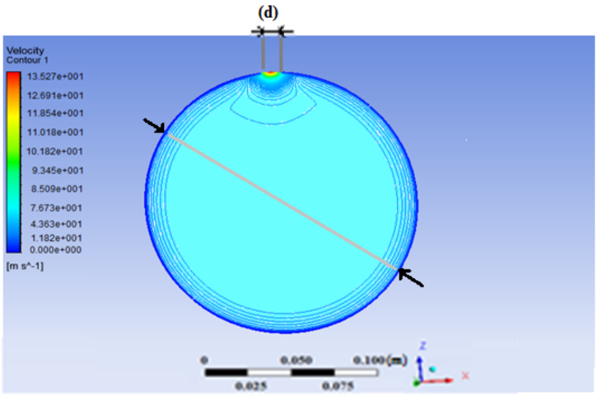
through the rupture, and remains constant until the feed water pump stopped at 300 seconds. At this time, the flow decreases and remains constant until the end of the transient.

Figures 9 and 10 give the evolution of feedwater velocity in and after the leakage section, respectively. It is noticed that the profile of mean velocity obtained by SST numerical simulation is in excellent agreement with those obtained by the Relap5 system code. The same analysis of the mass flow was concluded, which will be reported to the velocity curves, where Figs. 9 and 10 have the same profile trends as Figs. 7 and 8. It can be seen from Figure 9 that the velocity in the leak hole changes with time, where there is a disorder in the velocity magnitude caused by the leakage. Following the rupture occurrence, the velocity begins to rise until reaching a value around 130.475 m s^{-1}, which corresponds to a Reynolds number of 4.25x10^{6}. Afterward, due to the pump stopping, the leakage velocity decreases. After the leakage section, at the steady state, the feedwater velocity is equal to 108.34 m s^{-1}, corresponding to 104 m s^{-1} obtained by Relap5 code, followed by a new permanent regime at 100 seconds with a 60 m s^{-1} (corresponding to 44.9 m s^{-1} for Relap5 code) as a consequence of the leaking through the rupture (Fig. 10). A small decrease in Reynolds number to 1.17x10^{7} at the main pipe accompanied the leakage of the mass flow rate. Then, the fluid velocity undergoes a decrease when the feedwater pump shutdown, which causes loss in feed water pumping energy and explains the main reasons of loss of a volume of fluid through the leakage orifice (de Vasconcellos et al. 2014). At this stage, a sudden decrease in the flow Reynolds number to 5.58x10^{5} was occurs immediately after the leak position.

Figure 11 illustrates the feedwater velocity contours around the leakage orifice. Due to the existence of the leakage diameter, where a vertical momentum of the fluid moving up-stream, a larger velocity occurs at the exit of leakage hole, which leads a maximum velocity gradient of 135.27 m s^{-1}, but causes a reverse flow velocity upstream and downstream at this region with a small value. Thereafter, the velocity gradient along the flow direction can be reduced due to the attenuation of leakage effect (Bapista et al. 2006).



a) Side view



b) Cross section

Fig. 11 Feed water velocity contours around the leakage section; a) Longitudinal side view, b) Cross section

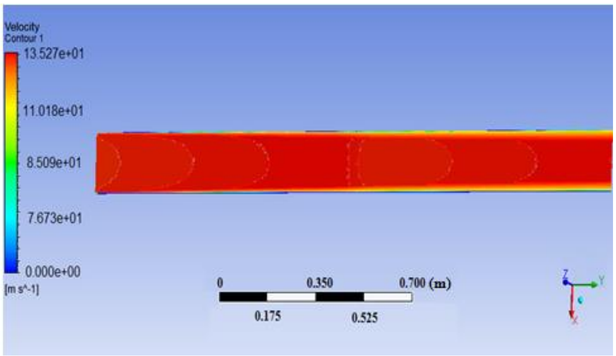


Fig. 12 Feed water velocity profile form inside the main pipeline

Figure 12 shows the profile of feed water velocity inside the main pipeline. From the inlet of the pipe the velocity along the flow direction has a relatively parabolic

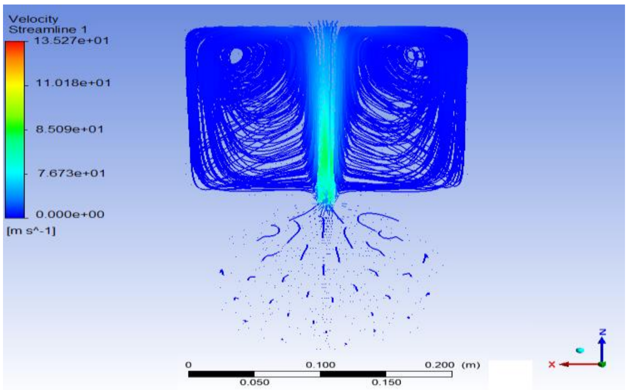


Fig. 13 Velocity streamlines and vectors of feed water flow after the leakage section

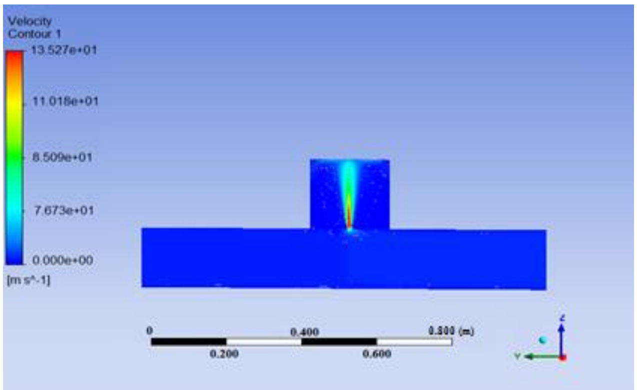


Fig. 14 Overview of feed water flow after the leakage section

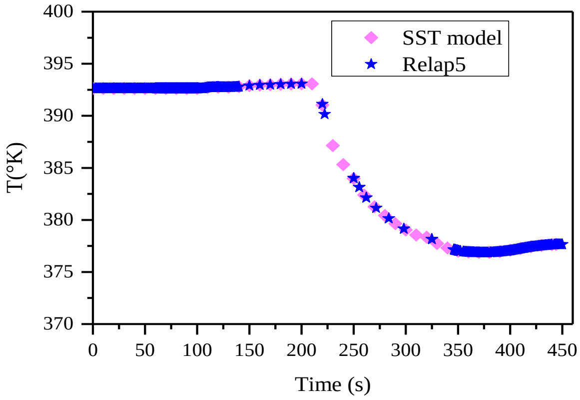


Fig. 15 Temperature distribution of feed water after leakage section

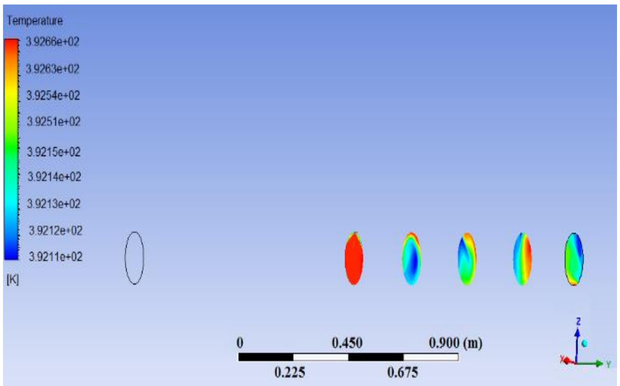


Fig. 16 Cross section of feed water temperature contours in the main pipeline

profile. After that, there is a sudden change in the velocity profile at the leak location, where it becomes flat. This rearrangement is a result of a local increase of the velocity gradient accompanied by the distribution of energy. Then, farther away from the leakage, the velocity profile has established and returns to the initial form.

Figures 13 and 14 show the leakage effect, which is presented by the water velocity streamlines and velocity vectors patterns of the free jet through the leakage orifice. There is a generation of strong vortices created by the ascendant flow on each side of the median axis that propagates in both directions around the leak site directions. The Much higher gradient of velocity is located in the medium section of the flow as detailed in Fig. 14.

Figure 15 gives an overview of the temperature variation during the transient. Before the occurrence of the break, the temperature of the flow inside the main pipeline was about 392.15°K. After the break opening, the temperature decreases rapidly to 377.64°K after 200 s because of the direct contact with ambient air temperature. These values have an identical outcome from the Relap5 system code, which was 377.65°K.

To predict the temperature distribution in the main pipe, a vertical cut planes have been designed (Fig. 16). The time-averaged temperature distribution contours were defined with the SST model. From this figure, we can see the propagation of a high-temperature gradient of liquid along the upper part of the main pipeline near the wall (Bapista, 2006; Ulrich & Paolo, 2017), where the temperature gradient on the lowest part of the pipe becomes more and more significant over time. This last is due to the high degree of mixing inside the main pipeline, which occurs not far from the leakage point, where the temperature ranges from 392.15°K to 392.66 °K. However, the density gradient is the consequence of temperature difference, which has a significant influence on the distribution. Therefore, we can conclude that the SST model has successfully predicted a realistic mixing of the fluid because this model tends to apply statistical average data.

4. CONCLUSION

In this work, a numerical simulation of a small break accident on the feed water pipeline in an industrial installation was investigated. The SST turbulence models was used and compared to Relap5 code data at transient operating conditions. The main objective was to predict any change in the performance of the plant during and after leakage in the feed water pipeline. The main thermal-hydraulic parameters concerned by this study were the flow field, the velocity and the turbulent flow temperature during and after the leakage at the top of the main pipeline. After calculation, we have concluded that the magnitude of velocity reached a maximum value after leakage, where the temperature decreased slightly. Moreover, close to the leakage section, the plant performance thermal load is more significant than the region far from the mixing area.

Also, we conclude that downstream mass flow rate response appears to be the most sensitive indicator serves to detection of leakage.

Consequently, it can therefore be concluded that the SST model considerably agrees well with Relap5 code numerical results for capturing the underlying physics of the turbulent flow. The comparison between the CFD simulation results against Relap5 code demonstrates clearly the ability of the two models to predict properly the thermalhydraulic behavior of the flow at transient condition.

ACKNOWLEDGEMENTS

This work is supported by the Atomic Energy Commission of Algeria (COMENA). The authors thank the Nuclear Research Center of Birine for all the help to get this work.

CONFLICT OF INTEREST

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

AUTHORS CONTRIBUTION

A. Dahia: conceptualization, data-curation, formal analysis, investigation, project-administration, software, supervising, validation, visualization, writing-original-draft, writing-review-editing, revision. A. L. Deghal Cheridi: conceptualization, software, methodology, data-curation, investigation, formal-analysis, validation, revision. M. Boumaza: conceptualization, methodology, data curation, investigation, formal analysis, software, validation, revision.

REFERENCES

Araújo, M. V., Farias Neto, S. R., Lima, A. G. B., & Luna, F. D. T. (2014). Hydrodynamic study of oil leakage in pipeline via CFD. *Advances in Mechanical Engineering, 6,* January - December 2014. https://doi.org/10.1155/ 2014/170178.

Araújo, M. V., de Luna, F. D. T. Barbosa, E. S., de Farias Neto, S. R., & de Lima, A. G. B. (2013). Numerical study of oil flow in tee junction with leaks. *Advances in Petroleum Exploration and Development*, 6(2), 1-11. https://doi.org/10.3968/j.aped. 1925543 820130602.1803.

Bapista, R. M., Quadri, M. B., Machado, R. A. F., Bolzan, A., Nogueira, A. L., Mariano, G., & Lopez, T. J. (2007). Effective Interfacial Tension and Geometrical Parameters Relationship for the Description of Oil Leakages from Submarine Pipelines. *Chemical Engineering Transactions, Edited by Sauro Pierucci*, 11. ISBN: 978-88-95608-00-6.

Ben-Mansour, R., Habib, M., Khalifa, A., Youchef-Toumi, K., & Chatzigeorgiou, D. (2011). Computational fluid dynamic simulation of small leaks in water pipelines for direct leak pressure transduction. *Computers & Fluids*, 57, 110-123. https://doi.org/10.1016/j.compfluid.2011. 12.016.

Carlson, K. E., Riemke, R. A., Rouhani, S. Z., Shumway, R. W., & Weaver, W. L. (1990). Relap5/Mo3.2 Code Manual Volume III: Developmental Assessment Problems. https://www. nrc.gov/docs/ML1103/ ML110330201. pdf.

Cheridi, A. L., & Loubar, A., Dadda, A., & Bouam, A. (2019a). Modeling and simulation of a natural circulation water-tube steam boiler. *SN Applied Science*, 1, 1405. https://doi.org/10. 1007/s42452-019-1452-x.

Cheridi, A. L., Chaker, A., & Loubar, A. (2019b). *Numerical simulation of the accidental transient of an industrial steam boiler*. Chapter in book: Heat and mass Transfer-Advances in science and technology applications, Intech Open Edition.

Cheridi, A. L., Chaker, A., & Loubar, A. (2016). Numerical simulation of a 374 Tons/h water-tube steam boiler following a feedwater line break. *Annals of Nuclear Energy*, 9, 27-35. http://dx.doi.org/10.1016 /j.anucene.06. 021.

de Sousa, C. A., & Oldrich, J. R. (2017). Influence of oil leakage in the pressure and flow rate behaviors in pipeline, *Latin American Journal of Energy Research-Lajer*, 4(1), 17-29. http://dx.doi.org/10.21712/lajer.2017.v4. n1.p17-29.

de Sousa, J.V.N., Sodré, C.H., de Lima, A. G. B., & de Farias Neto, S. R. (2013). Numerical analysis of heavy oil-water flow and leak detection in vertical pipeline. *Advances in Chemical Engineering and Science*, 3, 9-15. http://dx.doi.org/10.4236/aces.2013.31002.

Edrisi, A., & Kam, S. I. (2013). mechanistic leak-detection modeling for single gas-phase pipelines: lessons learned from fit to field-scale experimental data, *Advances in Petroleum Exploration and Development*, 5(1), 22-36. https://doi.org/10.3968/j.aped.19255438 20130501.1027.

Fan, W., Peng, C., Chen, Y., & Guo, Y. (2016). A new CFD modeling method for flow blockage accident investigations. *Nuclear Engineering & Design*, 33, 31-41. https://doi.org/10.1016/j.nucengdes. 2016. 04.006.

Fu, H., Yang, L., Liang, H., Wang, S., & Ling, K. (2020). Diagnosis of the single leakage in the fluid pipeline through experimental study and CFD simulation. *Journal of Petroleum Science and Engineering*, 193, 107437. https://doi.org/10.1016/j.petrol.2020.107437.

Fu, J. M., Zhao, H. X., Chen, G. M., & Zheng, X. Y. (2014). Impact of a hole's geometrical characteristics on the aperture leakage of natural gas pipelines. *Natural Gas Industry*, 34(11), 128-133. https://doi.org/10.3787/j.issn.1000.0967.

2014.11.018.

Kaliatka, A., & Valincius, M. (2012). Modeling of pipe break accident in a district heating system using Relap5 computer code. *Energy*, 44, 813-819. https://doi.org/10.1016.j.energy.energy.2012.05.011.

Kostowski, W. J., & Skorek, J. (2012). Real gas flow simulation in damaged distribution pipelines. *Energy*, 45(1), 481-488. https://doi.org/10.1016/j.energy.2012.02.076.

Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598-1605. https://doi.org/10.2514/3.12149

Michel, F. A., & David, R. H., J. (1991). *Matériaux-Microstructures et mise en œuvre*. 2, 382, 4eme Edition, Dunod. ISBN : 2880744733.

Mutiu, A. A., Aditya, K., & Wai-Keung, F. (2021). Numerical study of pipeline leak detection for gas-liquid stratified flow, *Journal of Natural Gas Science and Engineering*. https://doi.org/10.1016/j.jngse.2021.104054.

Ong, Y. W., & Siti, U. M. (2019). Computational fluid dynamics analysis on single leak and double leaks subsea pipeline leakage, *CFD Letters*, 11(2). 95-107.

Paolo, O. (2006). Boiler Design, Maintenance and Safety Study Guide, Boiler Operation Maintenance & Safety, *Study Guide for MTAA Boiler Operator Trainee Certefication*, SGBO001 306.

Rahmani, A., Bouchami, T., Bélaid, S., Bousbia, S. A., & Boulheouchat, M. H. (2009). Assessment of boiler tubes overheating mechanisms during a postulated loss of feedwater accident. *Applied Thermal Engineering*, 29, 501-508. https://doi.org/10.1016/j.applthermaleng.2008.03.008.

Shehadeh, M., & Shahata, A. (2013). Modelling the effect of incompressible leakage patterns on rupture area in pipeline, *CFD Letters*, 5(4), 132-142.

Sun, Y., Cao, X., & Liang, F. (2019). Investigation on underwater spreading characteristics and migration law of oil leakage from damaged submarine pipelines. *Proc. Saf. Environ. Protect*, (127), 329-347. https://doi.org/10.1016/j. psep.2019.05.030.

Ulrich, B., & Paolo, E. (2017). Numerical analysis of two experiments related to thermal fatigue. *Nuclear Engineering and Technology*, 49(4), 675-691. https://doi.org/10.1016/j.net.2017.01.018.

Zeng, Y., & Luo, R. (2019). Numerical analysis on pipeline leakage characteristics for incompressible flow. *Journal of Applied Fluid Mechanics*, 12(2), 485-494. https://doi.org/10.29252/ JAFM.12.02.28612.

Zhao, J. H., & Tan, Y. F (2011). Experiment study on subcritical flow nozzle model in the gas pipelines. *Journal of Harbin Institute of Technology*, 43, 84-87.

Zheng, M., & Hong, G. Z. (2017). Numerical investigation on impacts of leakage sizes and pressures of fluid convey- ing pipes on aerodynamic behaviors. *Journal of Vibro-engineering*, 19(7), 5434-5447. https://doi.org/10. 21595/jve.2017.17389.