Mathematical Modelling of Oil Pipeline Leakages Using Computational Fluid Dynamics - Case of BIDCO Oil Processing Refinery, Uganda.

Ali Wambi Wateya[1], Twaibu Semwogerere, PhD[2*], Richard O. Awichi, PhD[3] and Asaph Keikara Muhumuza, PHD[1].

[1] Department of Mathematics, Busitema University, Tororo, Uganda

[2] Department of Electrical and Computer Engineering, Busitema University, Tororo, Uganda

[3] Department of Mathematics and Statistics, Kyambogo University, Kampala, Uganda

*Correspondence should be addressed to semoge15@gmail.com

Abstract;

The leakage flow phenomena of a refinery oil pipe with a leakage point is numerically studied with the purpose to minimize oil leakage using Computational Fluid Dynamics (CFD) approach. Among consequences of oil pipe leakages are losses as a result of property loss (oil), cost of pipe replacement and also death due to fire or explosion. To understand the leakage phenomena, pipe characteristics at the leakage orifice are necessary. In the simulation, considering a pipe with a leak orifice of 0.002m, diameter 0.06 m and length 10 m, single phased flow was considered. The leakage through the pipe was studied based on fluid dynamics simulations using a Computational fluid dynamic tool ANSYS FLUENT software 17.2 where the Navier-Stokes were solved and for turbulence the standard k-ε was considered. Results from this study show that the leakage flow rate increases with increase in velocity inflow of the fluid. The pressure effect was also studied at the vicinity of the leak and results also show that an increase in velocity increases the pressure drop. Therefore, keeping the inflow velocity range of $0.1 ms^{-1}$ to $2 ms^{-1}$ show minimal leakage rates.

Keywords: Computational Fluid Dynamics (CFD), Mathematical Modelling, Oil Pipeline Leakage, Pressure Drop, Simulations.

Introduction

Pipe failure during transportation and storage of hydrocarbons or flammable materials occur regularly (Jujuly, 2016). Consequently, the threat of the leakage from the pipes cannot be under estimated in oil industries as it may lead to accidents or be costly in terms of material loss. The existence of a leakage hole in a pipe may result into a sudden pressure change, which results into time delay in delivery but also volume flow rate delivered (Silva et al., 1996).

According to (Zhang et al., 2014), different leakage detection measures have be set up in place for proper oil tracking in transportation, but as result of climatic weather changes, corrosion, poor workmanship, over use of pipes and many more factors have continued to cause pipe leakages. Thus, its necessary to quickly detect leaks to avoid oil loss.

The use of CFD to numerically simulate and obtain results of pipeline leakage is a better tool which can portray the behaviour during fluid flow process and the effects of leaks as noted by (De Sousa and Romero, 2017).

In Uganda there are many oil processing factories like BIDCO Uganda Ltd., among others with sophisticated oil processing equipment which consist of multiple pipes, refineries are well known for their composite structure from which leakage occur. The structural interaction between different subsystems makes it difficult the technical staff or operators to observe and also predict failures at the refinery units like leakages (Shaluf et al., 2003).

Globally many incidents arising from refineries due leakage and other causes like explosion have happened for example years ago an explosion at Thailand Oil Refinery at Si Raeha left 7 deaths, 12 injured and property worthy twenty five United states dollars wasted (Chew, 2000). The results of such catastrophic accidents in most cases spread beyond the vicinity of the industry thereby affecting the close neighborhood.

To solve oil leakage problem, its necessary to monitor pipelines routinely for early leakage detection. Different studies have been conducted to study leakage phenomena using CFD;

A study done by Zeng and Luo (2017), conducted a study on water flow using a short pipe with a leak hole. Results indicated that the pressure change towards and after the leak was linear to leakage flow rate while neglecting the pipe length.

The oil leakage effect was studied by (De Sousa and Romero, 2017), how it affected the pressure and flow rate behaviors in a pipe, the results indicated that the increase in the pipe diameter of the leak hole, greatly influences the leakage flow rate affects the hydrodynamics in the vicinity of the leak inside the whole pipe.

Shehadeh and Shahata (2013), using Computational Fluid Dynamics carried out a study on incompressible pipe flow at different diameters and the numerical results showed that leaks occurring due to high Reynolds number flows are easily detected than those at low Reynolds number.

Jujuly (2016), studied the steady state and transient behaviour of the pipe flow of water, crude oil, methane and nitrogen and his findings showed pressure and temperature were observed to be localised at the leak vicinity and in conclusion that leakage influences the pressure gradient.

All these numerical studies summarized above show numerical simulation is important understanding the fluid flow phenomena in existence of a leak. However, with all these studies above there is no study on effect of inflow velocity on flow rate, pressure change and turbulence at the leak orifice which is the primary goal of this current study. The researcher argued that knowing the influence inflow velocity effect on pressure drop, leakage flow rate and turbulence at the leak orifice puts the researcher in a best position to offer alternatives to the problem. Thus, this study discusses how oil pipeline leakages can be minimized based on mathematical methodology and analysis of a pipe with leak using Computational Fluid Dynamics (CFD).

2. Materials and Methods

2.1 Mathematical Model Formulation

Computational Fluid Dynamics (CFD) is a numerical tool used to analyze fluid flows and many more engineering phenomena. Using CFD, a real situation can be predicted and analyzed. CFD ANSYS software was used because of its ability to mesh and discretise the Navier-Stokes equations. To solve a set of differential equations the meshing, divides the volume of interest into finite volumes through discretisation in order to solve the mass and momentum equations hence enabling the solution be obtained numerically. A 3D model of pipe with a leak orifice was employed into Fluent CFD software. The oil of density $891 kgm-3$ flowing within the pipe was considered incompressible and turbulent. The governing equations in this study considered are continuity, momentum, and energy equations for the oil flow in a pipe which are to be solved simultaneously. In this study since the flow was assumed isothermal the energy equation was not solved together with the continuity and momentum.

2.2 Governing equations

The governing equations used to describe the fluid motion in the pipe are conservation of mass and momentum (de Vasconcellos Araujo et al., 2014). The conservation equations also known as Navier-Stokes equations are derived from continuity, momentum conservation are used to describe motion of fluid flows (Esemu et al., 2020). The governing equations, conserve all the laws for each control volume of the domain to be satisfied as observed in the continuity equation (1).

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \tag{1}$$

where ρ is the density of the fluid (oil), u is the velocity vector of the flow which depends on the coordinate x, y, and z. For incompressible flow, see equation (2).

$$\frac{\partial \rho}{\partial t} = 0 \tag{2}$$

Equation of Motion

Conservation of linear momentum leads to the equation of motion as observed in Equation (3). The Navier-Stokes equation below describes the motion of incompressible fluids (Fluent et al, 2018);

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_i)}{\partial x_j} + \frac{\partial \rho}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} - \rho g - F = 0 \tag{3}$$

where, P is the static pressure and τ is the viscous stress tensor, ρ is the fluid density, g is acceleration due to gravity, F is the forces acting externally on the body.

Neglecting the external forces since the flow is inside the pipe and also the effect of the gravitational force, since the flow in the pipelines is completely horizontal in this study, the required equation to by using Reynolds average decomposition reduces to;

$$\frac{\partial\,(\rho u_i)}{\partial t} + \frac{\partial\,(\rho u_i u_i)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{4}$$

Where the tensor term is given in equation (5).

$$\tau_{ij} = \mu_{eff}\left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_i}\right) - \frac{2\partial u_i}{3\partial x_i}\right] \tag{5}$$

Where μ_{eff} is defined as the effective viscosity, expressed in equation (6).

$$\mu_{eff} = \mu f + \mu t \tag{6}$$

Where μ_f is the viscosity of the fluid and μ_t is the turbulent viscosity for the turbulence model. The mass conservation at the leak takes on the continuity equation of the form by AlKhomairi (2005).

$$Q_{in} = Q_{out} + Q_{leak} \tag{7}$$

Where Q_{up} is the discharge just upstream and Q_{down} is the discharge just downstream of the leak location while Q_{leak} is the leak flow rate. The volumetric flow rate Q within the pipe is expressed in equation 8;

$$Q = UA = U\frac{\pi d^2}{4} \tag{8}$$

Where U is the velocity of the fluid in the pipeline, A is the area of the cross section of the pipe and d the internal diameter.

2.3 Turbulence Modelling

Turbulent Model

The Reynold's Average Navier-Stokes (RANS) equations are important in the modeling of steady-state and turbulent flow situations (Jujuly, 2016). The RANS turbulence equation based on the $k-\varepsilon$ standard model, provides fairly reasonable result and has been greatly embraced in the field of engineering to model turbulent flows in industries because of its capability to numerically provide fair results according to Jujuly (2016). This model is based on the eddy or turbulent viscosity concept (μ_t), where the effective μ_{eff} as in Equation (6), accounts for turbulence. The turbulent viscosity μ_t relationship between turbulent kinetic energy and dissipation is given by the ratio in equation (9);

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{9}$$

To cater for turbulent viscosity we consider both the kinetic energy k and its dissipation rate ε. Where C_μ is the turbulence constant given by the default value of 0.09 when the flow with high Reynolds number according to the Ansys Fluent theory guide (Canonsburg, 2017) .

The standard $k-\varepsilon$ model solves two different transport equations. Kinetic energy due to turbulence (k), described as (De Sousa and Romero, 2017) in equation (10) and the dissipation rate in equation (11).

$$\frac{\partial\,(\rho k)}{\partial t} + \frac{\partial\,(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\frac{\mu_{eff}}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + P^k - \rho\varepsilon$$

$$\tag{10}$$

The second one is the dissipation rate of the turbulent kinetic energy ε, modeled as in equation (11).

$$\frac{\partial\,(\rho\varepsilon)}{\partial t} + \frac{\partial\,(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\frac{\mu_{eff}}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right) + C_{\varepsilon 1}\frac{\varepsilon}{k}(P^k + C_{\varepsilon 3}P^b) - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \tag{11}$$

Where P^k and P^b represents the turbulence kinetic energy generation due to the mean velocity gradient and due to buoyancy respectively, σ_k and σ_ε are the turbulent Prandtl number for k and ε respectively (Jujuly, 2016). In this case, the effects of buoyancy in the ε equation is neglected. u is the velocity vector average in time and $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, $C_{\varepsilon 3}$,are constants .

The generation term P^k can therefore be expressed as in equation (12);

$$P^k = -\rho u_i u_j \frac{\partial u_j}{\partial x_i} \tag{12}$$

 The $k - \varepsilon$ model constants values are as shown in Table 1 below (Canonsburg, 2017);

Table 1: The $k-\varepsilon$ turbulence model constants.

Constant	C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	σ_k	σ_ε
Value	0.09	1.44	1.92	1.00	1.30

2.4 Computational Model Geometry

The geometry of the computational domain for simulating pipeline flow of oil with a leakage orifice was designed using Design modeler in ANSYS Fluent 17.2 and imported to the ANSYS workbench. A 3D model of pipe of length 10m, diameter of 0.06m and the leak orifice size was taken to be 0.002m, circular in shape cavity. The shape may vary but for easy generation of results it was considered circular.

2.5   Mesh Generation

The computational geometry boundaries were the inlet, the outlet, leakage orifice (outlet 2) and the pipe wall which was set as no slip condition/stationary.

The meshing of the geometry was done using 3-D triangular meshing utility in ANSYS Fluent. The refinement in the mesh was done by setting the size and relevance parameters while span angle, inflation were set as default in the meshing tool of ANSYS software. It was efficient to use fine mesh in order to capture the leakage region features. The numbers of element generated were 151, 211 . The smaller the grids the more the hydrodynamic features at the near leak region are capture. The mesh for the geometry setup is shown in Figure 1.
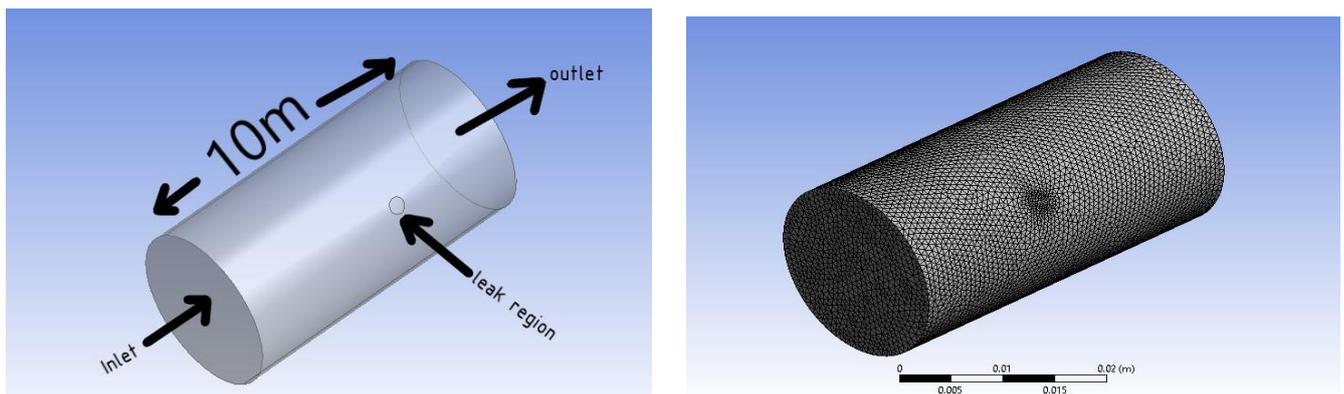

 Figure 1: 3D Computational and Meshed Geometry of a Pipeline with a Leak respectively

2.6 Initial and Boundary Conditions

In order to solve the system of equations proposed above, it was necessary to delimit the solution domain, that is, apply the boundary conditions on the numerical model case; at the inlet of the pipe, velocity was considered (Inflow velocity) ranging from

0.1 ms−1 to 5 ms−1. The leak region and the outlet where set as pressure outlets. The pipe wall was set as a wall boundary and no slip condition (i.e Ux=Uy=Uz=0 ms−1), at room temperature of 25oC. In this study the oil density was assumed to be 891 kgm−3 a hydro static pressure of 17500 pa was applied at the outlet section in order to capture flow features at the leak, constant atmospheric gauge pressure equal to 0 Pa in the leak section. For turbulence the steady state simulations, standard k-ε model was used.

3. Results and Discussion

3.1 Velocity flow field at the leak point

The fluid inside the pipe was studied at varying the inlet velocities so as to observe the flow rate behavior at the leakage hole. The flow velocity at leakage changes significantly showing its effect there. In all the simulations, the velocity flow profiles are observed to be concentrated at the leak hole which is an implication of the leakage effect. Figure 2 shows the simulation of a pipe at inflow velocity 0.1ms−1, 1ms−1, 2ms−1 and 5ms−1 respectively. It is observed that an increase velocity at the inlet increases the velocity at the leakage orifice but also the leakage flow rate. However, at inflow velocity of 5ms−1, the velocity at the leak point is significantly high. Internally there is an increase in the velocity profile within the pipe. It is clear that an increase in velocity at the inlet disturbs the velocity distribution at the leak and whole pipe.
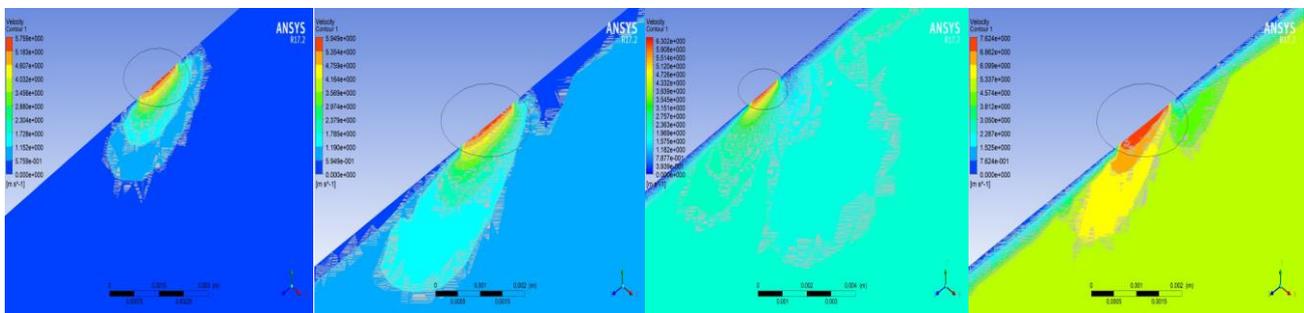

Figure 2: Velocity Field around the Leak Point at inlet Velocity 0.1ms^{-1}, 1ms^{-1} ,5ms^{-1} and 5ms^{-1} respectively

The effect of inlet velocity on the flow rate at the leak point can be seen on a graph in Figure 3, of leakage flow rate plotted against the inlet velocity. It is observed that leakage flow rate increases with inflow velocity.
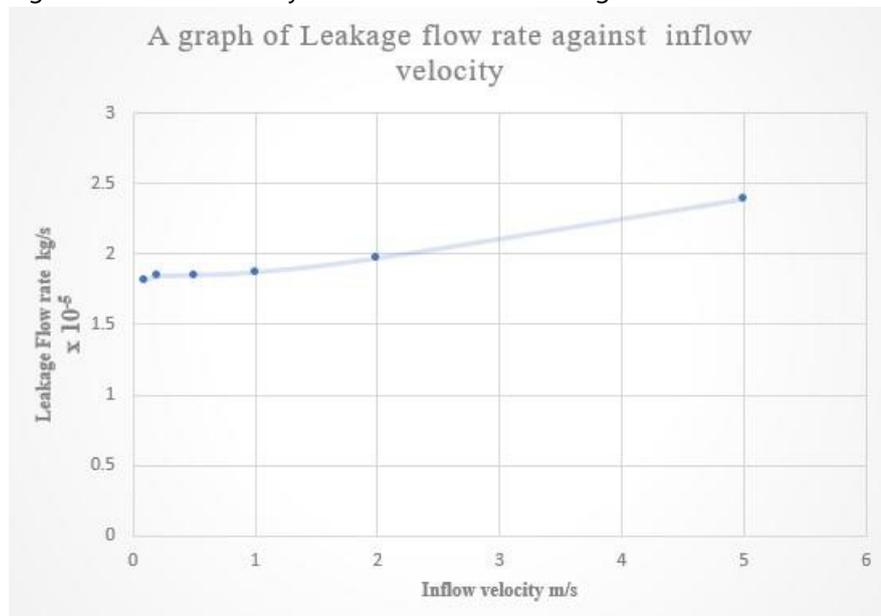

Figure 3: A Graph Showing Leakage Flow Rate Variation to Inflow Velocity

3.2 Pressure variation at the leak point

The pressure flow contours around the leak, show pressure variation around the vicinity of the leak, the pressure is high and dominate within the pipe flow as seen in Figure 4. In all the simulations, pressure drops at the leak

orifice but spreads throughout the pipe domain more as velocity is increased at the inlet as observed in Figure 4 hence yielding into an intermittent flow of a fluid in a pipe. This pressure drop occurs because the molecules collide as the fluid moves in the pipe domain resulting into a decrease in kinetic energy. Thus an increase in fluid flow velocity, results into more collision between molecules yielding more kinetic energy loss and in turn results into greater pressure drop. In conclusion, the pressure drop is proportional to the inlet velo
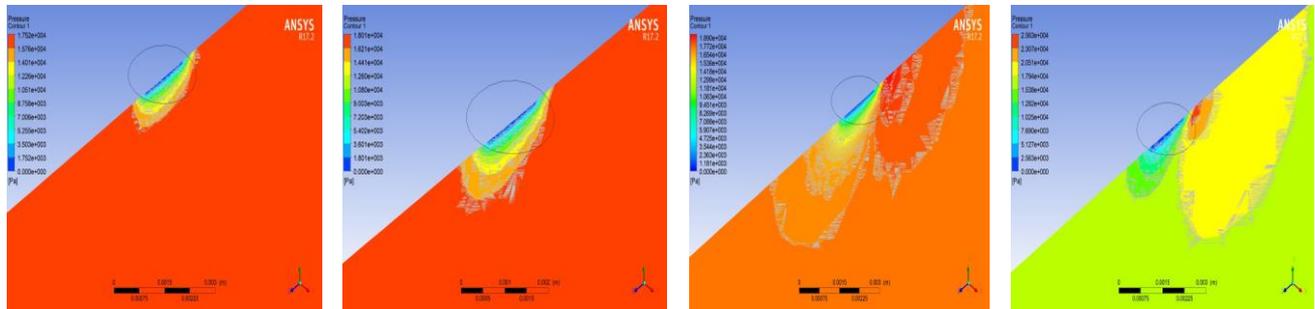
Figure 4: Pressure field around the leak point at inlet velocity 0.1 ms^{-1}, 1ms^{-1} ,5ms^{-1} and 5ms^{-1} respectively**3.3 Turbulence variation at the leak point**

Figure 5, shows the effect of Turbulence at the leakage hole. In this figure, the turbulence of kinetic energy is high at the leak area and decreases to away from vicinity of the leak this decrease is due to rapid collision of oil molecules resulting into the kinetic energy loss. High dissipation rate was observed at the leak hole and the flow field at the leak shows the effect of turbulence to the flow. It is noted that high turbulence perturbs the flow in the pipe and also increases the mass flow rate. The flow field concentration at the leak hole results into pressure change at the leakage yielding noise due to turbulence.
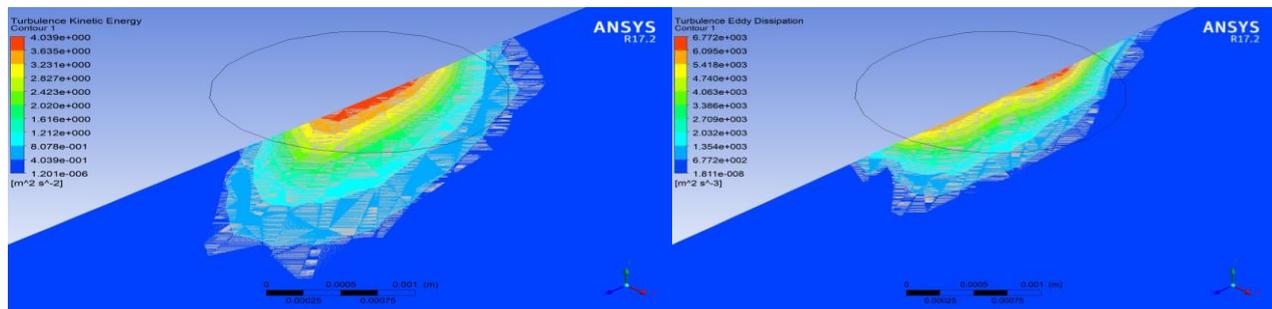

Figure 5: Turbulent Kinetic Energy Field and Turbulence Eddy Dissipation (TED) Contours around the Leak Point

4. Conclusions

In modeling the pipe leakage flow phenomena using CFD, the steady state flow simulation was considered, using a three dimensional geometry with a leak hole in order to determine the effect of velocity on leakage flow rate, pressure and turbulence. In this numerical analysis, turbulence was considered by solving the kinetic energy equation and the dissipation equation using the k-ε model. The Navier Stokes equations were also solved give solutions to both velocity and pressure. Results from the study show that the leakage flow rate increases with increase in velocity inflow of the fluid. The pressure effect was also studied at the vicinity of the leak and results also show that an increase in velocity reduces the kinetic energy and increases the pressure drop at the leak. This sudden change in pressure disturbs the internal pipe flow causing losses in property and also deformation of pipes. The results also showed the pressure drop at the leak hole slows down the flow pressure as it moves. Thus, continuous monitoring of velocity and pressure is important to detect leakage and also minimize it. The study outcomes are in good agreement with theoretical and experimental findings of other researchers.

5. Conflicts of Interest

The authors of this manuscript declare no conflict of interest arising from the information used to generate this manuscript.

8. References

1. Al-Khomairi, A. M. (2005). Use of the steady-state orifice equation in the computation of transient flow through pipe leaks. *Arabian Journal for Science & Engineering (Springer Science & Business Media BV)*, 30.

2. Canonsburg, T. (2017).   Ansys fluent user's guide.   *ANSYS FLUENT User's Guid*, 15317:2498.

3. Chew, S. K. (2000). *Evaluation of Disaster Managemnt and Preparedness of a Petrochemical in Malaysia*. PhD thesis, Universiti Putra malaysia.

4. De Sousa, C. A. and Romero, O. J. (2017). Influence of oil leakage in the pressure and flow rate behaviors in pipeline. *Latin American Journal of Energy Research*, 4(1):17–29.

5. Esemu, N. J., Masanja, V. G., Nampala, H., Lwanyaga, J. D., Awichi, R. O., and Semwogerere, T. (2020). An application of computational fluid dynamics to optimize municipal sewage networks; a case of tororo municipality, eastern uganda. *Journal of Advances in Mathematics*, 18:18–27.

6. Fluent, A. et al. (2018). Ansys fluent theory guide. *ANSYS Inc., USA*, 15317:724–746.

7. Jujuly, M. M. (2016). *Computational fluid dynamics (CFD) based approach to consequence assessment of accidental release of hydrocarbon during storage and transportation*. PhD thesis, Memorial University of Newfoundland.

8. Shaluf, I. M., Ahmadun, F.-R., and Said, A. M. (2003). Fire incident at a refinery in west malaysia: the causes and lessons learned. *Journal of Loss prevention in the Process Industries*, 16(4):297–303.

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

10. Silva, R. A., Buiatti, C. M., Cruz, S. L., and Pereira, J. A. (1996). Pressure wave behaviour and leak detection in pipelines. *Computers & chemical engineering*, 20:S491–S496.

11. Zeng, Y. and Luo, R. (2017). Numerical analysis of incompressible flow leakages in short pipes. In *Proceedings of 12th Pipeline Technology Conference*, pages 1–6.

12. Zhang, Y., Chen, S., Li, J., and Jin, S. (2014). Leak detection monitoring system of long distance oil pipeline based on dynamic pressure transmitter. *Measurement*, 49:382–389.