Uluslararası İleri Doğa Bilimleri ve Mühendislik Araştırmaları Dergisi
Sayı 8, S. 370-395, 7, 2024
© Telif hakkı IJANSER'e aittir
Araştırma Makalesi

International Journal of Advanced Natural Sciences and Engineering Researches
Volume 8, pp. 370-395, 7, 2024
Copyright © 2024 IJANSER
Research Article

https://as-proceeding.com/index.php/ijanser
ISSN: 2980-0811

Leak Flow Analysis Through Narrow Cracks In High-Pressure High-Temperature Pipe

Rahul Karmakar[1], Dr. Prasun Bhattacharjee[2*]

[1]*Department of Mechanical Engineering, JIS College of Engineering, India*
[2]*Department of Mechanical Engineering, Jadavpur University, India*

[*]*(prasunbhatta@gmail.com*

(Received: 27 August 2024, Accepted: 29 August 2024)

(5th International Conference on Engineering and Applied Natural Sciences ICEANS 2024, August 25-26, 2024)

ATIF/REFERENCE: Karmakar, R. & Bhattacharjee, P. (2024). Leak Flow Analysis Through Narrow Cracks In High-Pressure High-Temperature Pipe. *International Journal of Advanced Natural Sciences and Engineering Researches*, 8(7), 370 -395.

Abstract **–** This research examines the behavior of high-pressure pipelines with narrow cracks under various pressure conditions relevant to transportation systems in power plants. The focus is on the detection and measurement of leaks, crucial for safety analysis, particularly in nuclear reactors where radioactive substances are transported. The study investigates crack formation and growth under sustained mechanical and thermal loads, emphasizing the importance of leak detection in Leak Before Break (LBB) analysis. Various numerical techniques, including the Finite Element Method (FEM), are used to evaluate crack growth and through-wall crack opening under high-pressure leak flow conditions. Material selection is also emphasized, with an analysis of different alloys, such as carbon steel and stainless steel. The research covers both rectangular and circular crack geometries, using Computational Fluid Dynamics (CFD) simulations to analyze mass flow rate and velocity under different pressure scenarios. Findings indicate that mass flux increases consistently with pressure for both crack types, highlighting the need for precise analysis and continuous monitoring to maintain pipeline integrity and prevent hazards. Inlet pressures ranging from 70 bar to 180 bar are considered to assess their impact on mass flux and leak flow velocity through the cracks. The study observed that mass flux and leak flow velocity both increased consistently with rising inlet pressure for rectangular cracks, while for circular cracks, mass flux showed similar growth, but leak flow velocity exhibited uneven growth with increasing inlet pressure. Pressure and velocity contour images are provided to visualize flow behavior through cracks of different geometries. Overall, this research offers valuable insights into crack behavior and leakage in high-pressure duct systems, contributing to the development of improved safety measures and maintenance strategies for power plant pipelines.

Keywords – Crack Growth, Finite Element Method, Leak Detection, Pipeline Integrity, Pressure Contours.

I. INTRODUCTION

The pipeline in a power plant is a significant part that deals with the transportation of fluids. High-pressure system design requires a mandatory safety analysis in the form of a Leak Before Break. This has been proven effective and cost-efficient. The formation and growth of cracks are scrutinized in this analysis before the entire system experiences a disruption. The primary system pipelines in nuclear reactors carry radioactive substances, and safety is a prime concern. The formation of cracks occurs at a stable rate due to the sustaining mechanical and thermal loads. Therefore, leak detection and measurements are considered to be a primary requisite in LBB analysis, which also helps in gaining valuable information regarding the probable size and geometry of the formed crack. This has been proven effective and cost-efficient. Therefore, the material taken into consideration must have the characteristics to prevent the mentioned deteriorating factors. Pipeline leakage is a common phenomenon in various oil or gas transporting pipelines and it occurs due to metal erosion, corrosion, cracks, defects, and lack of maintenance [1]. Leakages can adversely affect human and marine life, if not monitored closely [2]. According to a survey conducted by the American National Standards Institute, material failure and leakage cause an economic loss of 4.2% of the gross national product in the USA, which amounts to around one hundred billion US dollars. [3]. To determine and evaluate fracture mechanics, several parameters need to be taken into account which include fracture toughness and nil ductility transition temperature. When it comes to cracks that have complex geometric shapes and continue to grow in various engineering structures, they are primarily assessed through the calculation of the Stress Intensity Factor using numerical techniques. The application of the Finite Element Method in studying fracture mechanics has been well documented in various literature discussing pipeline integrity and the interactions caused by corrosion defects [4-6]. Advanced numerical techniques have been employed to accurately calculate the extent of the growth of the crack that occurs as a result of stress corrosion cracking [7]. Additionally, these techniques have also been used to analyze the crack opening through the wall that may occur under high-pressure and high-temperature leak flow conditions [8]. Before discussing further, it is important to understand the concept of a loss of coolant accident.

A. Loss of Coolant Accident (LOCA)

Loss of Coolant Accident (LOCA) is a significant event in the operation of nuclear reactors, particularly in pressurized water reactors (PWRs) and boiling water reactors (BWRs). It refers to the unintended loss of reactor coolant, typically water, from the primary cooling system. There is a strong need to understand the characteristics of the LOCA to predict the behavior of the protection and safeguard systems designed to withstand any break size and break location in the primary circuit of the Nuclear Power Plant (NPP). The NPP’s supplier is obligated to prove that the plant parameters during LOCA should not violate the acceptance criteria [9].

Nuclear reactors rely on a continuous flow of coolant to remove heat generated by the nuclear fission process. This coolant circulates through the reactor core, where it absorbs thermal energy and transfers it to a secondary cooling system for electricity generation. The integrity of the coolant system is crucial for safe and stable reactor operation.

B. Leak Before Break (LBB)

By utilizing the Leak Before Break (LBB) method, researchers and engineers can gain critical insights into the behavior of materials subjected to extreme environments and develop effective strategies to mitigate the risk of catastrophic failures. Some numerical techniques have been used to determine crack growth from stress corrosion cracking and through-wall crack opening under high-pressure leak flow conditions. Various materials based on their unique properties are applicable for the high-pressure pipelines used in several industries. For instance, despite cast iron being highly resistant to corrosion for a short period, it becomes extremely prone to internal and external damages over time. Its use is, therefore, limited to applications that do not require long-term durability.

Carbon steel, on the other hand, can withstand high temperatures and is widely used in various industries. However, it is susceptible to corrosion when exposed to a high level of moisture, chemicals, and acids. To mitigate this, protective coatings are applied to the surface of the pipelines. On the other hand, stainless steel is regarded as one of the ideal materials used in various pipelines. It is highly resistant to corrosion, even in acidic and high-moisture environments. This means that it can be used in applications that require long-term durability, such as in the oil and gas industry. Despite the associated cost, its durability and corrosion-resistant properties make it a preferred choice in various industries. Stainless steel is a type of metal used in pipelines. Austenitic stainless steels containing 17-18% chromium are resistant to corrosion while duplex stainless steels containing higher chromium [10] are stronger and more resistant to wear. Stainless steel is stronger than carbon steel at high temperatures. Inconel 625 is another material that is strong at high temperatures and resists corrosion. Scratches, cracks, and gouges can harm the integrity of a pipeline. These types of damage are often found inside dents in pipelines [12]. Pipelines often have a type of damage called a dent crack, which looks like a semi-ellipse. Several crack detection techniques using sensors have been reported in the literature [13]. Several studies have focussed on the determination of critical leak flow evaluations through predefined cracks or slits through theoretical and experimental investigations [14]. CFD simulations have emerged as a popular alternative for the prediction of leakage mass flux under a variety of boundary conditions [15]. However, there is a lot of scope to simulate various analytical models to evaluate the effect of stagnation pressure and crack morphology on pipe leaks causing severe accidents in key power stations. This paper aims to analyze the behavior of a high-pressure pipeline with a predefined circumferential crack of rectangular and circular profile under various pressure conditions related to the transportation pipes. Mass flow rate and velocity conditions for various metal alloys in response to intermediate and high-pressure ranges at a given fixed condition have been investigated.

In the applications of high-pressure pipelines used in several power plants and chemical industries, various materials are used based on their unique properties. Carbon steel can withstand high temperatures but it faces corrosion when exposed to moisture, chemicals, and acids [16]. To mitigate this, protective coatings are applied to the surface of the pipelines. Stainless steel, on the other hand, is regarded as one of the ideal materials used in various pipelines. It is highly resistant to corrosion, even in acidic and high-moisture environments. Despite the associated cost, its durability and corrosion-resistant properties make it a preferred choice in various industries inclusive of Nuclear Power Plants. Pipelines often have a type of damage called a dent crack, which looks like a semi-ellipse. Several crack detection techniques using sensors have been reported in the literature. Several studies have focused on the determination of critical leak flow evaluations through predefined cracks or slits through theoretical and experimental investigations. CFD simulations have emerged as a popular alternative regarding the prediction of leakage mass flux under a variety of boundary conditions. However, there is a lot of scope to simulate various analytical models to evaluate the effect of stagnation pressure and crack morphology on pipe leaks causing severe accidents in key power stations. This paper aims to analyze the behavior of a high-pressure pipeline with a predefined circumferential crack of rectangular and circular profile under various pressure conditions. Mass flow rate and leakage flow patterns have been studied for various boundary conditions through CFD simulation techniques to address a sustainable solution regarding loss of coolant accidents especially faced in nuclear power plants [17].

"Leak before brake" is a safety principle often used in hydraulic systems. It means that in case of a failure, such as a leak in the hydraulic system, the fluid should escape before the brakes fail. This design approach prioritizes preventing sudden and catastrophic brake failure, allowing for a gradual loss of hydraulic pressure instead. It enhances safety by providing a warning through fluid leakage before a potential brake system failure occurs. The accurate prediction of pipe break flow is pressurized water reactor in primary loop pipeline in loss of coolant accident. The leak flow rate from the wall crack is crucial for the leak-before-break method. It is designed to avoid the radioactive hazard economically. The effect of friction has been decreased due to the very opening channel in the wall crack. It can estimate the two-phase maximum flow rate, where boundary condition and geometry are known for the unknown crack geometry. Subcooled coolant has been choked by the vapor phase in the flow with a decrease of local sonic velocity in a two-phase mixture. In the nuclear power plant, the Leak-before-break is a very important factor in the pipelines for safety. The basic two-phase flow in the pressurized water reactor depends on the heat and mass transfer. Boiling and flashing phenomena are very important factors for nuclear reactor systems. The bubble growth in boiling is inhibited by the heat transfer at the interface. The bubble growth in the flashing is limited in the surrounding water. The flashing process is more explosive, damaging, and violent than the boiling. Flashing develops in the flowing liquid when pressure decreases sufficiently small in some regions of the flow. It is reaching a metastable state when the local temperature is more than the saturated temperature. The superheated stream releases very fast and produces a pure bubble or liquid-vapor mixture at high velocity. This expansion also affects the geometry of the nuclear power plant. In a two-phase blowdown superheated liquid condition, bubble collapse is not an important factor. The flashing in the superheated liquid affects the critical flow rates [18].

Many types of nuclear reactor applications depend on convective nucleate boiling for dispersing high heat from the heated surfaces. Nucleate boiling is a very effective heat transfer mechanism. The solid surface is important for the constant latent heat supply in the phase change. This surface is to modify the flow pattern and characteristics of this multiple flow. Departure from Nucleate Boiling is to play an important role in degradation performance for the crud deposition problem. The two-phase flow phenomena that happen in the light water reactor (LWR) include changes in the coolant phase and the multiple flow rule which directly impact the reactor performance and the coolant reaction with fuel assembly. Pressure is decreased when a loss of coolant accident, is caused by the leak, crack, or break of the piping system. The result is changes in physical quantities in the leak area like pressure and temperature in the pressure retaining boundary. The postulated leak has been calculated by the leak mass flow and structural mechanic. The fluid temperature and pressure change by the thermal hydraulic, which allows the determination of the time-dependent change of size of the leak for the altering parameters in the Finite Element (FE) analysis method. For the thermal-hydraulic estimations, the practice has been to put the initial leak sizes with an equivalent diameter, which is constant for the time at a particular position in the cooling circuit. A compressible fluid is flowing from one container to another container through a flow passage, then the pressure initially increases at the inlet side area and pressure decreases at the outlet side area. The velocity of sound reaches the tightest position of the passage flow. A two-phase discharge flow and a critical mass flow have been observed in the discharge flow of an incompressible fluid. The pressure becomes smaller as the temperature of the fluid compares to saturation pressure. The fluid dynamic and thermal dynamic non-equilibrium process mainly depend on the flow condition and geometry of flow passage at inlet flow [19].

High-pressure pipelines sustain the carbon dioxide in liquid or dense phase or superheated phase, which is an important element for the development of enhanced oil recovery, storage technologies, and carbon capture. If a large amount of CO_2 is transported in the pipelines, then carbon capture and storage is deployed on an industrial scale. So, it is very important for a safe and cost-effective implementation of transportation in the pipelines. Accidents happen from pipelines, the result is high concentrations due to a significant amount of CO_2 being involved, which is leading to negative effects on the environment and threats to nearby residents. Excessive stress, corrosion, and valve cracks are other reasons for the many types of pipeline failures. To find and prevent the risk of these types of accidental failures in primary stages. It is difficult to know and estimate the leakage flow characteristics including the temperature and flow pressure, the mass outflow rate, and the exit velocity. This data is essential for the development of a leakage detection system. It is vital to modeling the CO_2 atmospheric dispersion because it gives important input data, and serves as source terms for leakage strength estimation [20].

The ability to estimate the discharge rates for the single and two-phase fluid from pipelines, orifices, and nozzles are very important factor for the safety test of water-cooled nuclear reactors. The heat transfer has been controlled by the discharge flow rate in the core and the rate of system depressurization. This is also influencing the design of emergency cooling systems and containment. The theoretical models to predict the single component two-phase discharge flow rates, also assume the homogeneous flow with phases in the thermodynamic equilibrium, non-equilibrium between two-phase and separated flow in the thermodynamic equilibrium. When a fluid expands from a compressed state to random conditions in passage through the outlet. The flow rates are always less than a certain maximum or critical value. After this critical condition has been reached reductions in receiver pressure. This pressure leaves the flow rate, serving from a steep pressure gradient at some location in the outlet passage. The pressure gradient in the outlet section becomes infinite, which is referred to as the exit plane. The Flow properties have attained critical values, being virtually independent of receiver pressure in the exit plane. The degree of inter-phase heat, momentum, and mass control the critical flow rate of a one-component two-phase mixture. The discharge of high-pressure and temperature fluid in the reactor coolant system is frequently encountered during the safety analysis of a pressurized water reactor [21].

The pressurizer is filled with nitrogen gas, water, and a stream during operation. It is connected to three gas cylinders by the three penetrating pipes. If one of the pipes is broken, the mixture of nitrogen gas, water, and stream has been discharged through the broken pipe. It is very important for predicting the critical flow rate of discharge water with non-condensable gas. The major components of the non-condensable gas two-phase critical flow test facility are a test section, a suppression tank, a pressure vessel, and a nitrogen gas supply system. The major circulation mechanism has the functions of setting which maintains the stagnation pressure and temperature for the required experiment conditions. This has been possible by pressurizing and heating the coolant. The pressure vessel is circulated in a circulation loop, which is connected to a pump, a heat exchanger, and an electrical heater. The coolant discharge mechanism has a function, which produces the experimental data by discharging the coolant through the test section to the suppression tank [22].

II. MATERIALS AND METHOD

The current CFD analysis had been conducted within the flow domain of a circular stainless-steel pipe. The location of the leakage had been considered at the middle of the pipe domain. The study had been done by designing leak locations of various rectangular dimensions and varying the pressure conditions paired with thermal condition. This physical problem for CFD analysis represents the actual life scenario of crack formation in the high-pressure pipelines of a duct system of a nuclear power plant.

In the current work, the hydraulic condition of fluid flow has been analysed for a pipeline at high-pressure and high-temperature conditions. The CFD analysis had been conducted using open-source software. Water had been used as a fluid and stainless-steel had been used as the material of the pipeline carrying the fluid in the simulation. The type of flow being simulated is turbulent.

A.1.Two-dimensional simulation of turbulent flow in a pipeline

For the two-dimensional simulation, a geometry of a 2D pipeline is drawn to analysis pipeline system. Various boundary conditions are provided to create pressure conditions. The project involved an in-depth analysis of the hydraulic conditions of fluid flow in a pipeline operating under high-pressure conditions. In simulations using water as the fluid medium and steel as the material for the pipeline. A turbulent flow model to simulate the fluid flow in the pipeline and used ANSYS software to create a 2D figure of the pipe for visualization and analysis. This allowed us to gain a better understanding of the fluid dynamics and to identify potential issues that could impact the pipeline's performance.

- Geometry

The dimensions of the pipe whose flow domain is taken into account are 90 mm length and 30 mm width. The geometry is designed using open-source design modeller tool. The geometry of the flow domain of the pipe is shown in Fig.3.

- Mesh

Mesh of the geometry is generated taking element size as 0.001 m. The various sections of the geometry are named before generating the mesh, namely, inlet, outlet, pipe wall. After the generation of mesh, the mesh quality is checked through mesh metrics. For better meshing of the element, refinement level 1 has been applied in geometry.

- Computational Setup and solution

The computational fluid dynamics simulation has been performed using Fluent code in ANSYS platform. The 2D analysis has been conducted using double precision setting. The viscous model used for the setup is k-epsilon (standard). The material selected for the fluid flow domain simulation is water in liquid state. The boundary condition given for the simulations are pressure inlet and pressure outlet at the outlet of the flow domain and crack opening. The pressure set at the inlet ranges from 5 bar to 20 bar and the pressure set at the outlet ranges are 1 bar. The calculation is run for 1000 iterations.

A.2. Three-dimensional simulation of turbulent flow in a pipeline

For the three-dimensional simulation, a geometry of a 3D pipeline is drawn to analysis pipeline system. Various boundary conditions are provided to create pressure conditions. The project involved an in-depth analysis of the hydraulic conditions of fluid flow in a pipeline operating under high-pressure conditions. In simulations using water as the fluid medium and steel as the material for the pipeline.A turbulent flow model to simulate the fluid flow in the pipeline and used ANSYS software to create a 3D figure of the pipe for visualization and analysis. This allowed us to gain a better understanding of the fluid dynamics and to identify potential issues that could impact the pipeline's performance.

- Geometry

The dimensions of the pipe whose flow domain is taken into account are 800 mm (length) x 40 mm and 45 (inner and outer diameter). The geometry is designed using open-source design modeller tool. The geometry of the flow domain of the circular pipe is shown in Fig.5.

- Mesh

Mesh of the geometry is generated taking element size as 3 mm. The various sections of the geometry are named before generating the mesh, namely, inlet, outlet, pipe wall. After the generation of mesh, the mesh quality is checked through mesh metrics. For better meshing of the element, refinement level 1 has been applied in geometry. The Fig. 6, showcase the generated mesh of the pipe flow domain.

- Computational Setup and solution

The computational fluid dynamics simulation has been performed using Fluent code in ANSYS platform. The 3D analysis has been conducted using double precision setting. The viscous model used for the setup is k-epsilon (standard). The material selected for the fluid flow domain simulation is water in liquid state. The boundary condition given for the simulations are pressure inlet and pressure outlet at the outlet of the flow domain and crack opening. The pressure set at the inlet ranges from 5 bar to 20 bar and the pressure set at the outlet ranges are 1 bar. The calculation is run for 1000 iterations. After completed the meshing of element, many conditions have been applied to the geometry.

A.3. Three-dimensional simulation of turbulent flow in a high-pressure pipeline with crack formation

For the three-dimensional simulation, a geometry of a circular pipeline is drawn to replicate a real-life pipeline system. Various boundary conditions are provided to create high-pressure and high temperature conditions. The project involved an in-depth analysis of the hydraulic conditions of fluid flow in a pipeline operating under high-pressure conditions. Our team conducted simulations using water as the fluid medium and steel as the material for the pipeline. We opted for a turbulent flow model to simulate the fluid flow in the pipeline and used ANSYS software to create a 3D figure of the pipe for visualization and analysis. This allowed us to gain a better understanding of the fluid dynamics and to identify potential issues that could impact the pipeline's performance.

- Geometry

The dimensions of the pipe whose flow domain is taken into account are 500 mm (length) x 100 mm (diameter) having 5 mm thickness. Three circular-shaped minute cracks of diameter 1 mm, 1.5 mm, and 2 mm and three rectangular-shaped narrow cracks of length 1 mm, 2 mm, 3 mm, and breadth 0.5 mm each have been taken into consideration for the analysis. The geometry is designed using open-source design modeller tool. The geometry of the flow domain of the circular pipe is shown in Fig. 3.5. Similarly, the close-up of the minute circular crack and the narrow rectangular crack designed on the middle position of the pipe domain can be seen in Fig.7 and Fig. 8 respectively. The thickness of the pipe is taken into consideration while designing the crack openings.

- Mesh

Mesh of the geometry is generated taking element size as 3 mm. The various sections of the geometry are named before generating the mesh, namely, inlet, outlet, pipe wall, crack outlet, and crack wall. After the generation of mesh, the mesh quality is checked through mesh metrics. The Fig.9 and Fig.10 showcase the generated mesh of the pipe flow domain, circular crack and rectangular crack respectively.

- Computational setup

The computational fluid dynamics simulation has been performed using Fluent code in ANSYS platform. The 3D analysis has been conducted using double precision setting. The viscous model used for the setup is k-epsilon (standard). The material selected for the fluid flow domain simulation is water in liquid state. The boundary condition given for the simulations are pressure inlet and pressure outlet at the outlet of the flow domain and crack opening. The pressure set at the inlet ranges from 70 bar to 180 bar and the pressure set at the outlet ranges from 69 bar to 179 bar respectively (the pressure difference between the inlet and outlet is kept 1 bar for all the test cases). The pressure at the crack opening is set at 1 bar which is the atmospheric pressure under normal conditions. The calculation is run for 2000 iterations.

Thermal condition had also been added in the later computational setup. 523K temperature was set as the initial temperature for the fluid flowing in the pipe flow domain and viscous model used for the setup was k-epsilon (realizable). The later simulation with thermal condition was conducted in multiphase to simulate the phase change from liquid to vapour during the leak flow process and sudden depressurization when the high-pressure and high-temperature fluid flowing in the pipe comes out of the crack into the atmospheric pressure condition

- Computational Model

The two-equation models are the simplest “complete models” of turbulence in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k- ϵ model in ANSYS FLUENT falls within this class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [29]. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

The standard k- ϵ model [29] is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ϵ). The model transport equation for k is derived from the exact equation, while the model transport equation for ϵ was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart.

In the derivation of the k- ϵ model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k- ϵ model is therefore valid only for fully turbulent flows.

- solution settings

The simulation had been conducted using steady flow and transient process at a later stage and was completed after 1500 iterations initially as the initial simulations where less complex in nature due to lower inlet pressure and simple conditions provided in the open-source software used for the simulation. Gradually the number of iterations taken to complete the calculation increases to 2000.

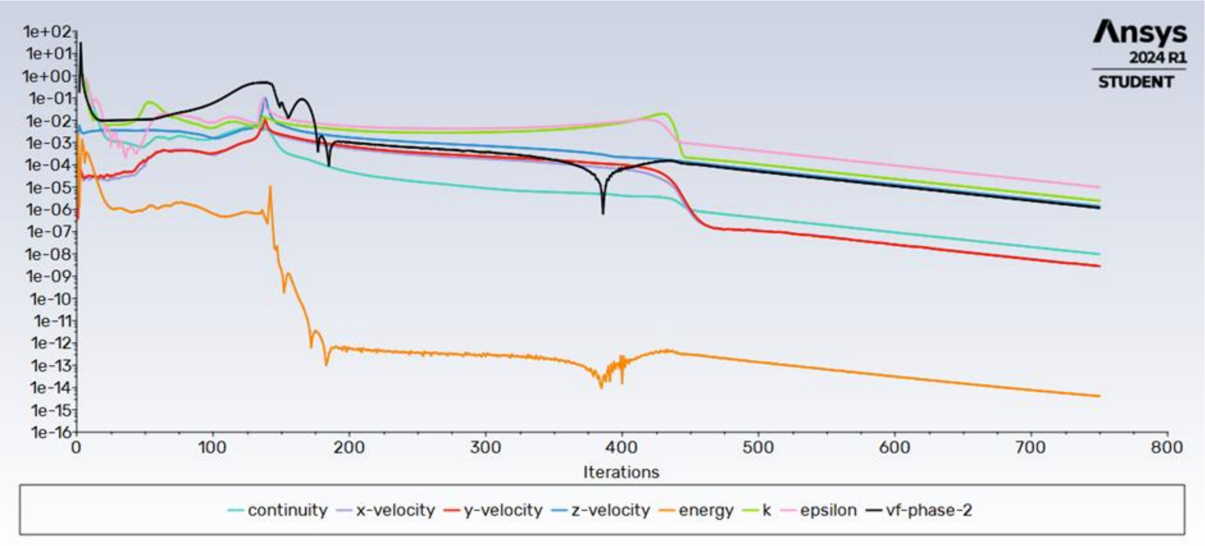

Figure 1: Residual scale of the simulation

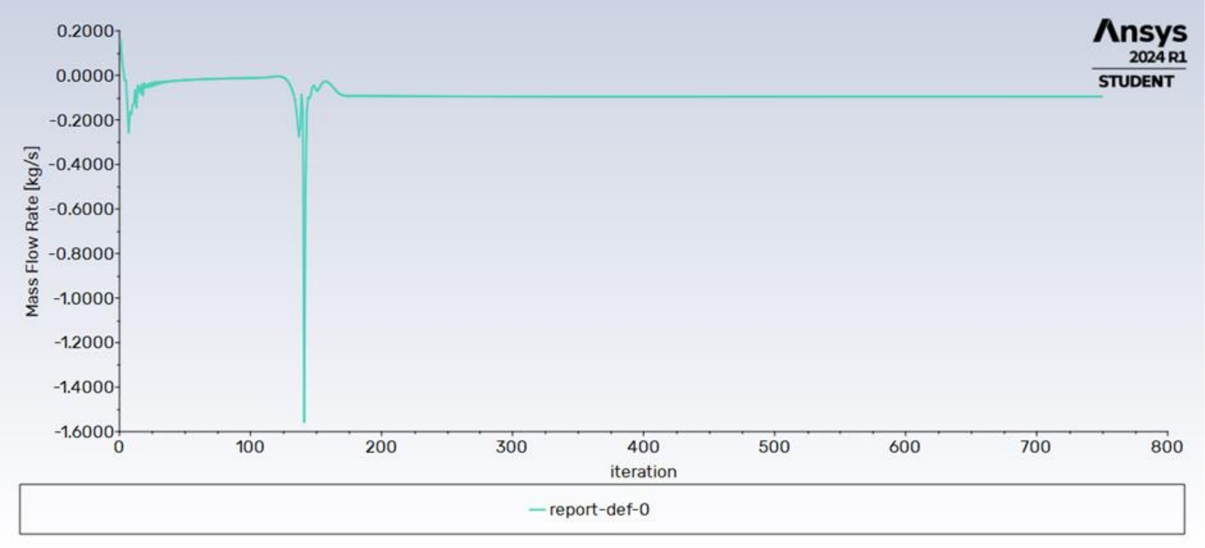

Figure 2: Mass flow rate graph of the simulation

The Mass flow rate vs iterations graph and residual graph were plotted as the iteration cycles were completed as shown in Fig. 1 and Fig. 2 above. The Pressure Implicit with Split Operator (PISO) algorithm was used for the iterations. PISO is a pressure-velocity coupling algorithm in Ansys FLUENT that uses predictor-corrector steps to satisfy mass conservation. The PISO algorithm involves one predictor step and two corrector steps. The main idea of the PISO algorithm is to move the repeated calculations required by SIMPLE and SIMPLEC (SIMPLE-Consistent) inside the solution stage of the pressure-correction equation. The PISO and COUPLED schemes are generally more aggressive and converge faster than SIMPLE per iteration cycle. SIMPLE is the default algorithm, and SIMPLEC is similar to SIMPLE but uses a different expression for face flux correction. SIMPLEC can be beneficial for many problems because of the increased under-relaxation that can be applied. SIMPLEC is said to help with convergence in some cases, and is seen to converge faster than the SIMPLE algorithm

B. Figures and Tables

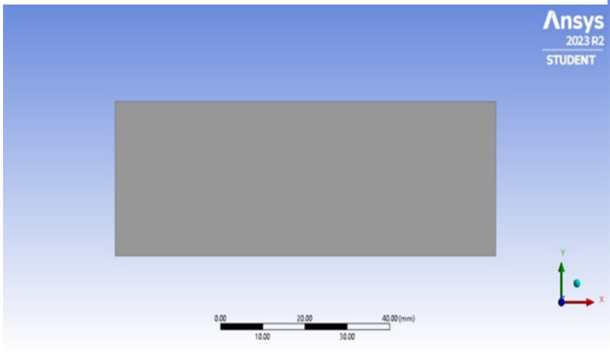

Figure 3: geometry of the pipe in 2D

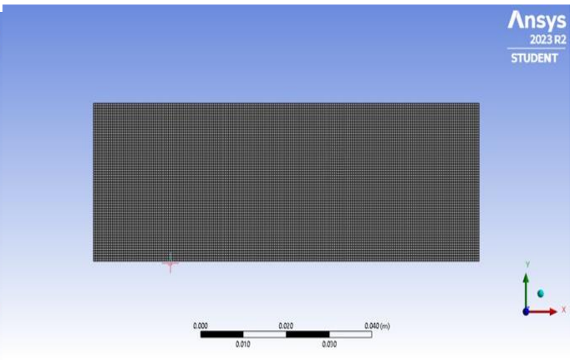

Figure 4: mesh of the pipe in 2D

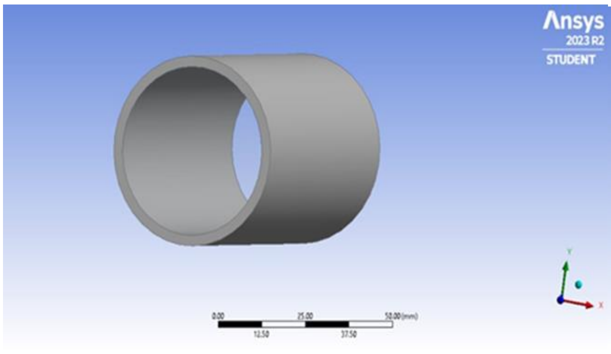

Figure 5: geometry of the pipe in 3D

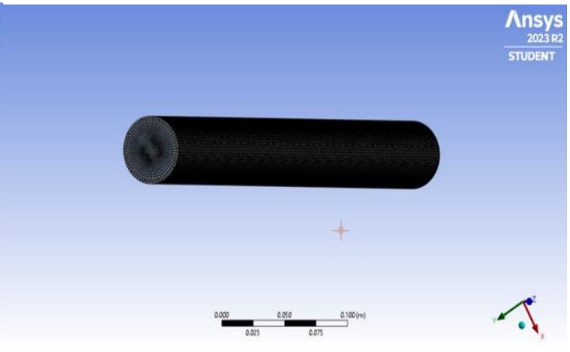

Figure 6: mesh of the pipe in 3D

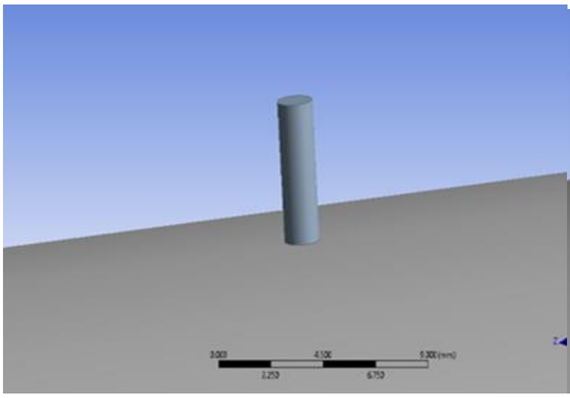

Figure 7: The circular crack geometry

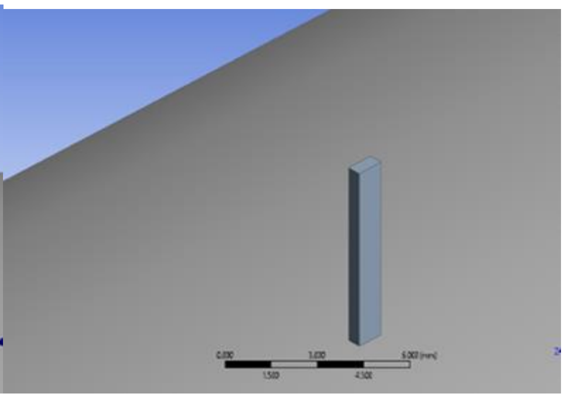

Figure 8: The rectangular crack geometry

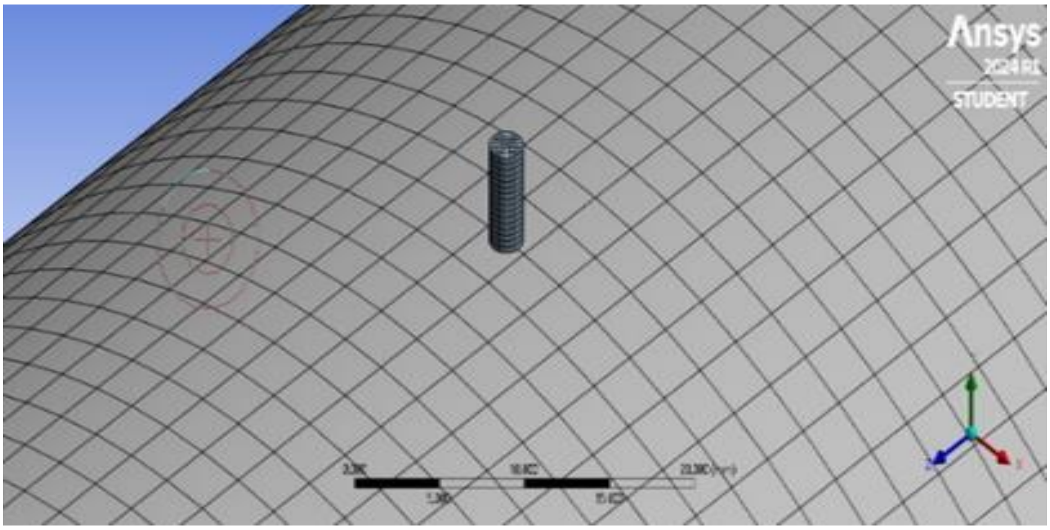

Figure 9: mesh of the circular crack

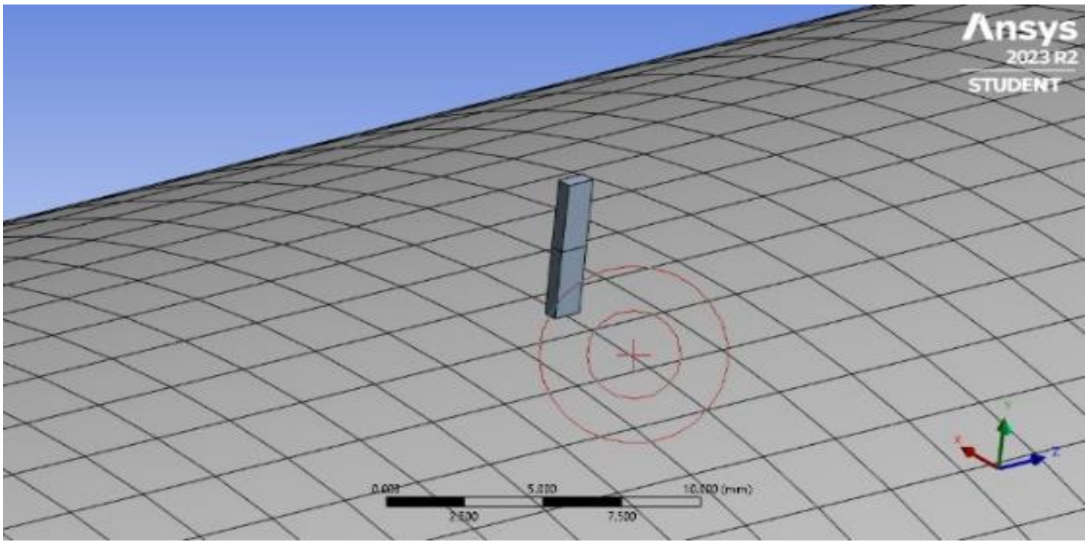

Figure 10: mesh of the rectangular crack

Table 1: Setup configuration for 2D and 3D pipe

Viscous	K-epsilon (2 equation)
K-epsilon model	Standard
Near wall function	Standard wall function
Materials (fluid)	Water-liquid
Materials (solid)	Steel
Cell zone conditions (fluid domain)	Water-liquid
Prevent reverse flow (inlet)	on
Residual	0.00001
Initialization	standard
Compute from	inlet
Run calculation (no. of iterations)	1000

Table 2: Boundary conditions for 2D and 3Dpipe

	Boundary condition	
Sl.no	Inlet pressure(bar)	Outlet pressure(bar)
1	5	1
2	10	1
3	15	1
4	20	1

Table 3: Boundary conditions for circular and rectangular cracked pipe

Boundary conditions					
Sl.no	Crack area (m^2)	Crack area(m^2)	Inlet pressure(bar)	Outlet pressure(bar)	Crack outlet pressure(bar)
1	7.06x10^-4	0.5x10^-4	70	69	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
2	7.06x10^-4	0.5x10^-4	80	79	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
3	7.06x10^-4	0.5x10^-4	90	89	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
4	7.06x10^-4	0.5x10^-4	100	99	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
5	7.06x10^-4	0.5x10^-4	110	109	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
6	7.06x10^-4	0.5x10^-4	120	119	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
7	7.06x10^-4	0.5x10^-4	130	129	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
8	7.06x10^-4	0.5x10^-4	140	139	1
	12.56x10^-4	1x10^-4			

	19.63x10^-4	1.5x10^-4			
9	7.06x10^-4	0.5x10^-4	150	149	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
10	7.06x10^-4	0.5x10^-4	160	159	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
11	7.06x10^-4	0.5x10^-4	170	169	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			
12	7.06x10^-4	0.5x10^-4	180	179	1
	12.56x10^-4	1x10^-4			
	19.63x10^-4	1.5x10^-4			

Table 4: Vaporization pressure vs saturated temperature

Sl.no	Vaporization pressure(bar)	Saturated temperature(k)
1	10	457.27
2	20	488
3	30	508.85
4	40	524.99
5	50	538.38
6	60	549.83
7	70	559.93
8	80	569.04
9	90	577.30
10	100	584.89
11	110	591.92
12	120	594.87
13	130	604.61

III. RESULTS AND DISCUSSION

The present work aims to contribute to the data and analysis of the fluid flow behaviour within a high-pressure and high-temperature pipeline system used in industries and power plants. Another objective is to showcase the result of leak flow occurring through narrow circumferential cracks that develop on stainless steel pipe surfaces overtime due to various natural factors such as corrosion, LBB phenomenon or human dependent factors such as improper maintenance of the duct systems. The present work intends to show the leak flow properties under various boundary conditions at a high temperature. This data will help in the safety analysis which is required for the high-pressure and high-temperature fluid carrying duct systems across industrial and technical fields.

A. Result for two-dimensional analysis

Here steel pipes have been used to analyse the relation between the mass flow rate and the area of the crack. For the 2d pipe, under various pressures, the change of mass flux is shown in table 9. Here we observed that mass flux increases with the increase of pressure. Throughout the process, the temperature is considered constant. The mass flow rate analysis under various pressures gives the change of mass flux.

Table 5: Data table of mass flux and velocity of the outlet through 2D pipe for various pressure conditions

Sl. No.	Pressure (Bar)		Mass flux (kg/s/m²)	Velocity (m/s)
	Inlet	Outlet	Max	Max
1	5	1	28257.6	28.3086
2	10	1	42386.9	42.4633
3	15	1	52866	52.9613
4	20	1	61587.1	61.6982

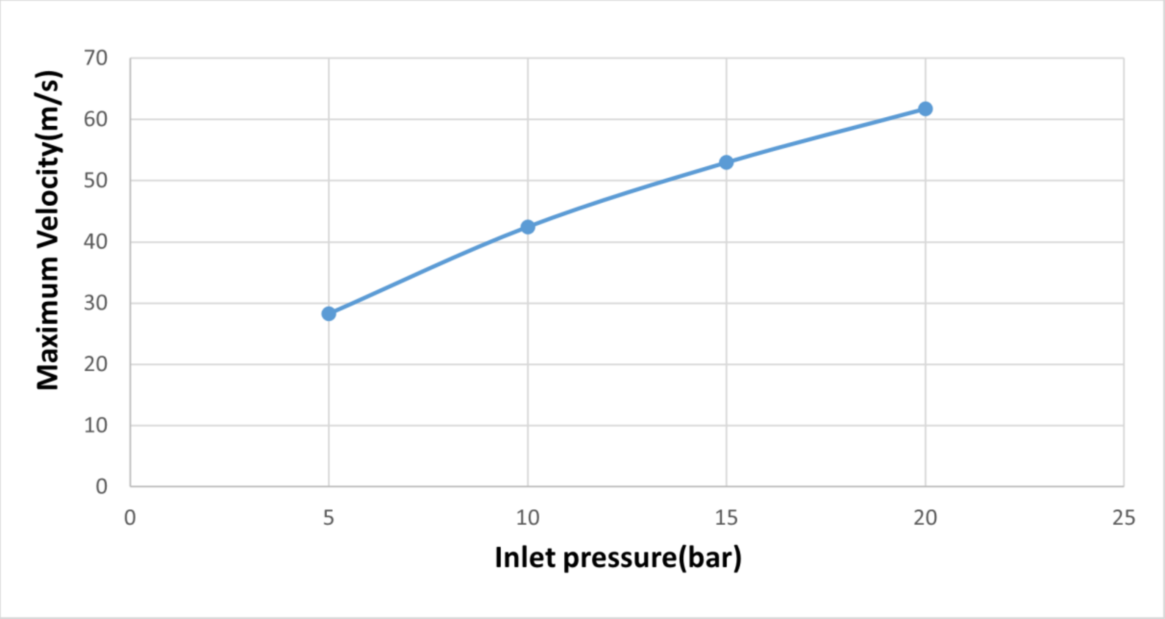

Figure 11: Variation of maximum flow velocity with pressure for 2D pipe

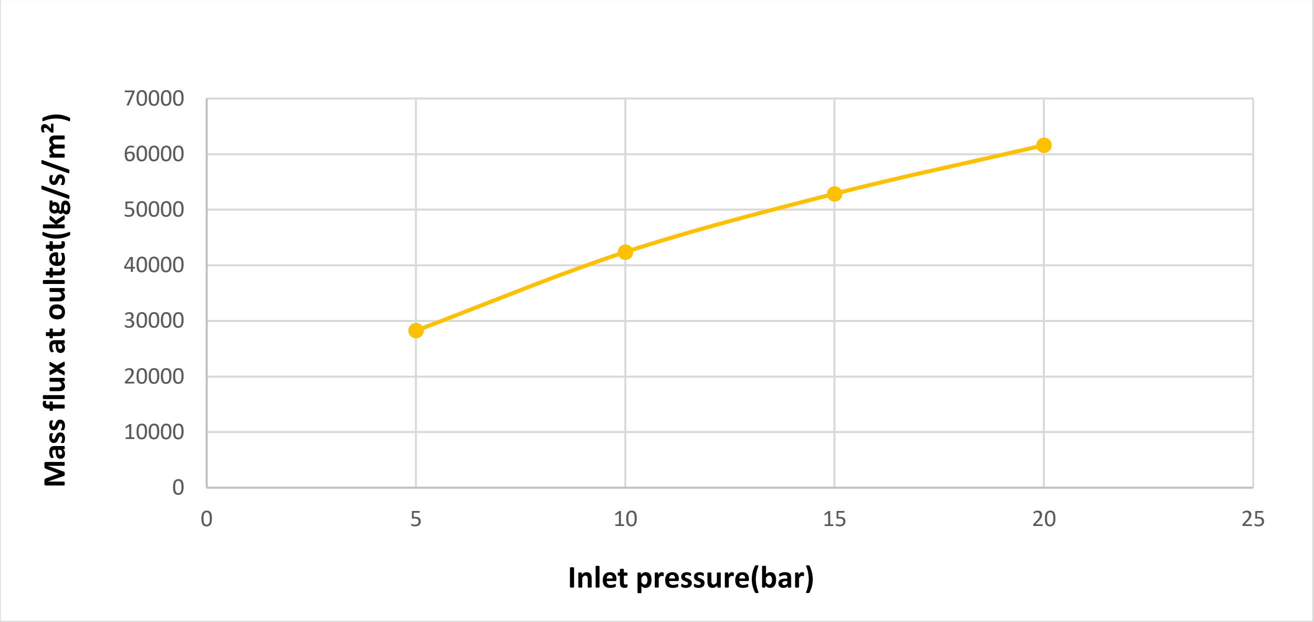

Figure 12: Variation of maximum mass flux with pressure for 2D pipe

The variation of the maximum flow velocity following the inlet pressure can be depicted in the graph in fig. 11. The maximum leak flow velocity increases with the increase in the pressure at the pipe inlet. Also, the variation of the mass flux with the inlet pressure is shown in the graph in fig. 12. It is seen that the mass flux increases with the increase in the inlet pressure.

B. Result for three-dimensional analysis

Here steel pipes have been used to analyse the relation between the mass flow rate and the area of the crack. For the 3d pipe, under various pressures, the change of mass flux is shown in the table 10. Here we observed that mass flux increases with the increase of pressure. Throughout the process, the temperature is considered constant. The mass flow rate analysis under various pressures gives the change of mass flux.

Table 6: Data table of mass flux and velocity of the outlet through 3D pipe for various pressure conditions

Sl. No.	Pressure (Bar)		Mass flux(kg/s/m²)	Velocity (m/s)
	Inlet	Outlet	Maximum	Maximum
1	5	1	28252.6	28.3035
2	10	1	42381.4	42.4578
3	15	1	52859.8	52.9551
4	20	1	61580.8	61.6918

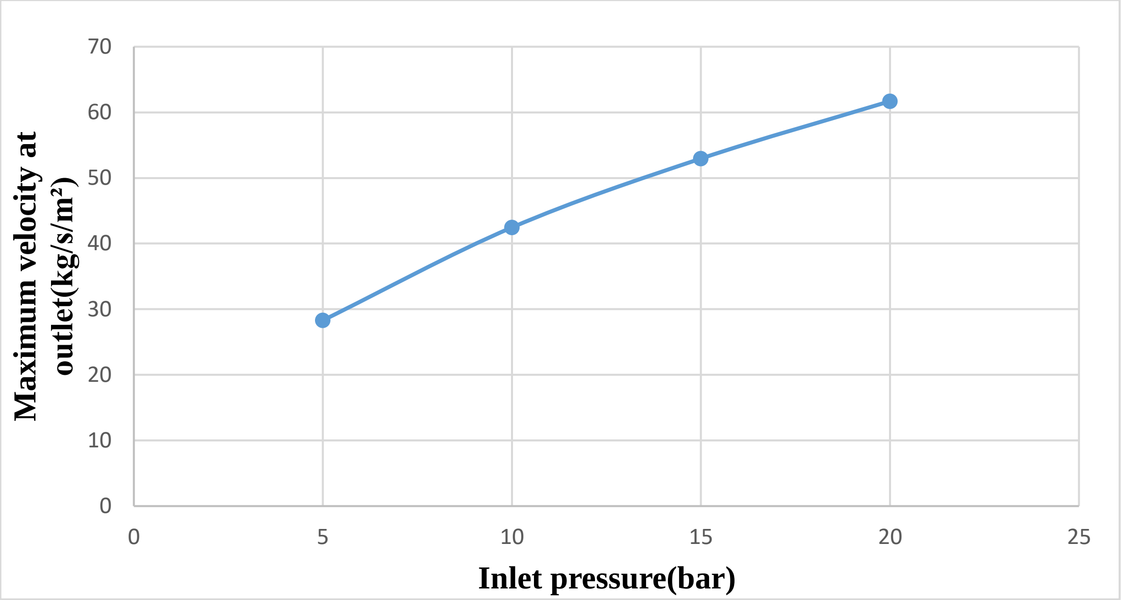

Figure 13: Variation of maximum flow velocity with pressure for 3D pipe

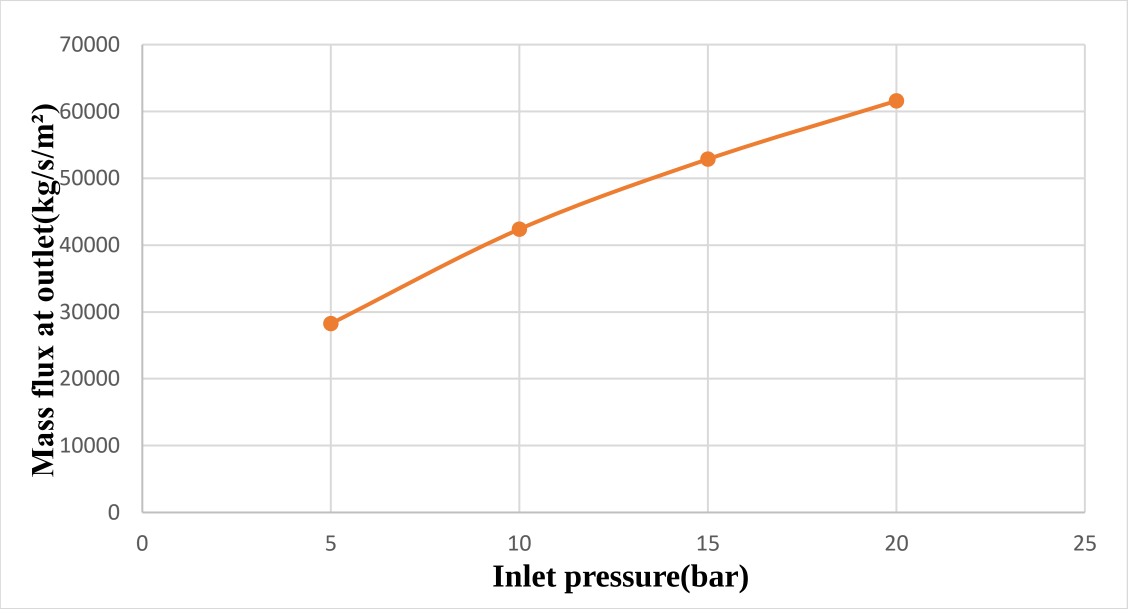

Figure 14: Variation of maximum mass flux with Pressure for 3D pipe

The variation of the maximum flow velocity following the inlet pressure can be depicted in the graph in fig.13. The maximum leak flow velocity increases with the increase in the pressure at the pipe inlet. Also, the variation of the mass flux with the inlet pressure is shown in the graph in fig.14. It is seen that the mass flux increases with the increase in the inlet pressure.

C. Result of circular crack analysis

The analysis of circular cracks was conducted to determine the relationship between pressure, mass flux, and velocity of the crack. The results of the analysis are presented in Table 11, which shows the changes in mass flux due to variations in pressure and Area. The analysis was conducted under the assumption of constant temperature. The results of the analysis revealed that an increase in pressure led to a corresponding increase in mass flux. The fluid flow rate through the crack was influenced by the pressure gradient and the crack size. Overall, the analysis provides valuable insights into the physics of fluid flow through circular cracks, and it underscores the importance of understanding the underlying mechanisms that govern fluid flow to optimize the design and operation of fluidic systems.

Table 7: Data table of the mass flux and velocity through circular crack opening at various pressure conditions

Sl. No.	Inlet Pressure (bar)	COA x10^{-6}(m^2)	Mass Flux (kg/m^{2}s)	Leak flow Velocity (m/s)
1	70	7.06	0.021	99.259
		12.56	0.022	98.274
		19.63	0.023	100.441
2	80	7.06	0.023	105.334
		12.56	0.022	104.298
		19.63	0.023	107.552
3	90	7.06	0.023	109.421
		12.56	0.024	110.651
		19.63	0.024	114.457
4	100	7.06	0.024	116.278
		12.56	0.025	115.336
		19.63	0.026	119.548
5	110	7.06	0.026	121.745
		12.56	0.027	121.275
		19.63	0.028	126.354
6	120	7.06	0.028	126.648
		12.56	0.028	126.357
		19.63	0.029	132.025
7	130	7.06	0.028	147.185
		12.56	0.028	148.357
		19.63	0.029	148.474
8	140	7.06	0.029	152.414
		12.56	0.030	153.528
		19.63	0.030	153.564
9	150	7.06	0.031	140.213
		12.56	0.032	141.243
		19.63	0.033	147.684
10	160	7.06	0.032	144.388
		12.56	0.033	145.840
		19.63	0.034	152.538
11	170	7.06	0.033	148.377
		12.56	0.034	150.306
		19.63	0.035	157.245
12	180	7.06	0.034	152.261
		12.56	0.035	154.655
		19.63	0.036	161.812

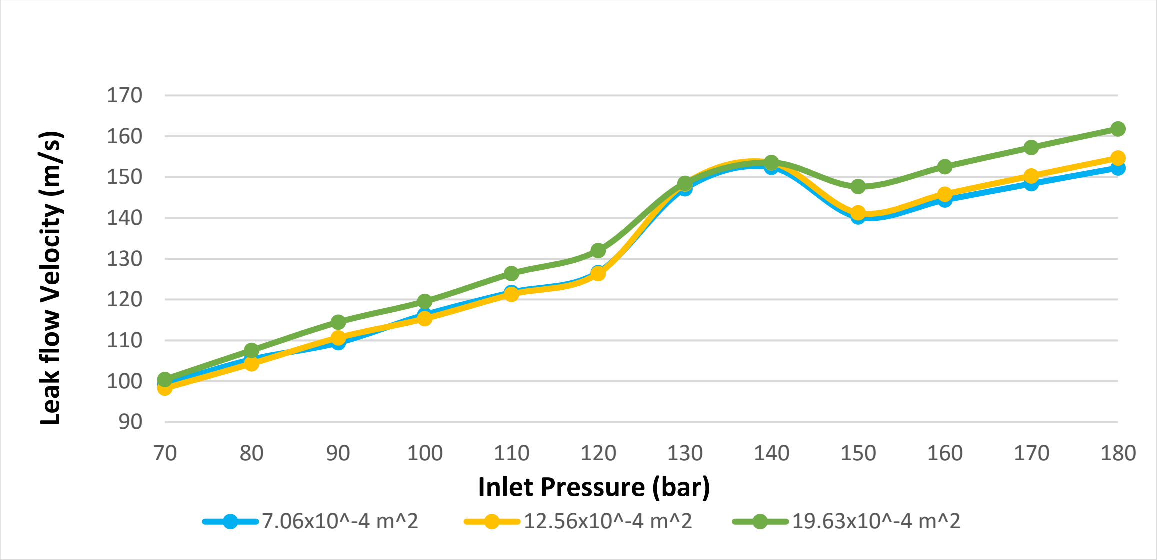

Figure 15: Variation of maximum leak flow velocity with pressure for circular cracks

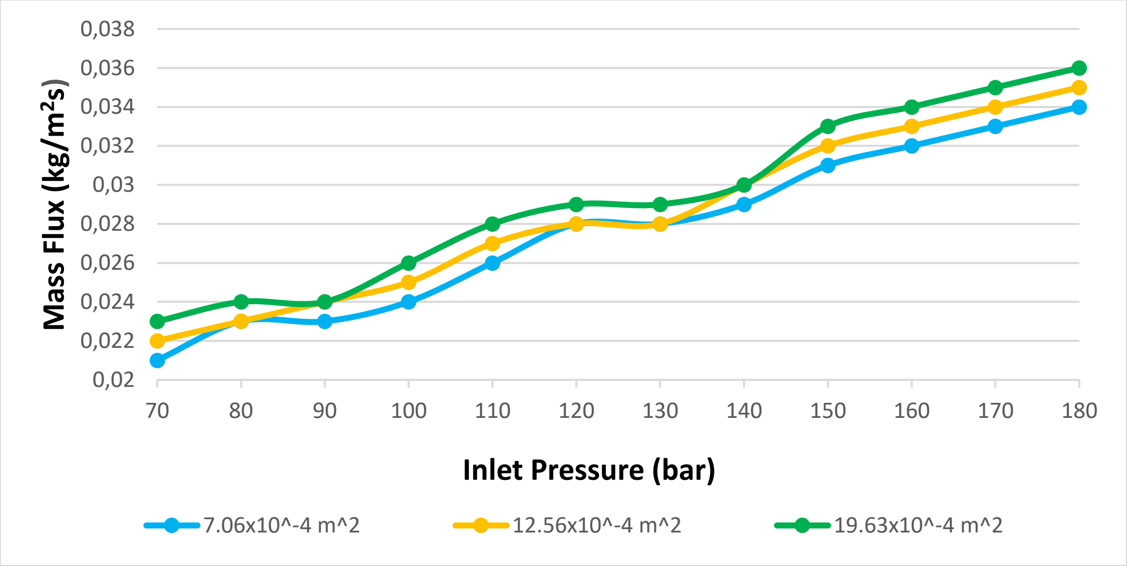

Figure 16: Variation of maximum mass flux with Pressure for circular cracks

In the case of circular cracks, the variation of the maximum leak flow velocity following the inlet pressure can be seen in the graph in fig.15. The maximum leak flow velocity increases with the increase in the pressure at the pipe inlet of the pipe flow domain. Also, the variation of the mass flux with the inlet pressure is shown in the graph in fig.16. It is seen that the mass flux increases with the increase in the inlet pressure for most of the scenarios of leak flow with some uneven variations as reported in some experimental works.

D. Result for rectangular crack analysis

Here we have used the steel pipes to analyse the relation between the mass flow rate and the area of the crack. For the rectangular crack under various pressures with various crack areas, the change of mass flux is shown in Table 12. Here we observed that mass flux increases with the increase of pressure. Throughout the process, the temperature is considered constant. The mass flow rate analysis under various pressures gives the change of mass flux.

Table 8: Data table of the mass flux and velocity through rectangular crack opening at various pressure conditions

Sl. No.	Inlet Pressure (bar)	COA x10^{-6} (m^2)	Mass Flux (kg/m^2s)	Leak flow Velocity (m/s)
1	70	0.5	0.078	86.241
		1	0.081	87.559
		1.5	0.082	88.483
2	80	0.5	0.084	92.358
		1	0.086	93.754
		1.5	0.087	94.682
3	90	0.5	0.089	98.224
		1	0.092	99.673
		1.5	0.093	100.642
4	100	0.5	0.094	103.774
		1	0.097	105.275
		1.5	0.098	106.276
5	110	0.5	0.099	109.193
		1	0.102	110.762
		1.5	0.104	111.829
6	120	0.5	0.104	114.236
		1	0.107	115.854
		1.5	0.108	116.942
7	130	0.5	0.108	118.918
		1	0.111	120.555
		1.5	0.113	121.621
8	140	0.5	0.113	123.727
		1	0.115	125.435
		1.5	0.117	126.540
9	150	0.5	0.117	128.218
		1	0.12	129.965
		1.5	0.122	131.071
10	160	0.5	0.121	132.562
		1	0.124	134.345
		1.5	0.126	135.448
11	170	0.5	0.125	136.772
		1	0.128	138.587
		1.5	0.130	139.691
12	180	0.5	0.129	140.861
		1	0.131	142.704
		1.5	0.133	143.809

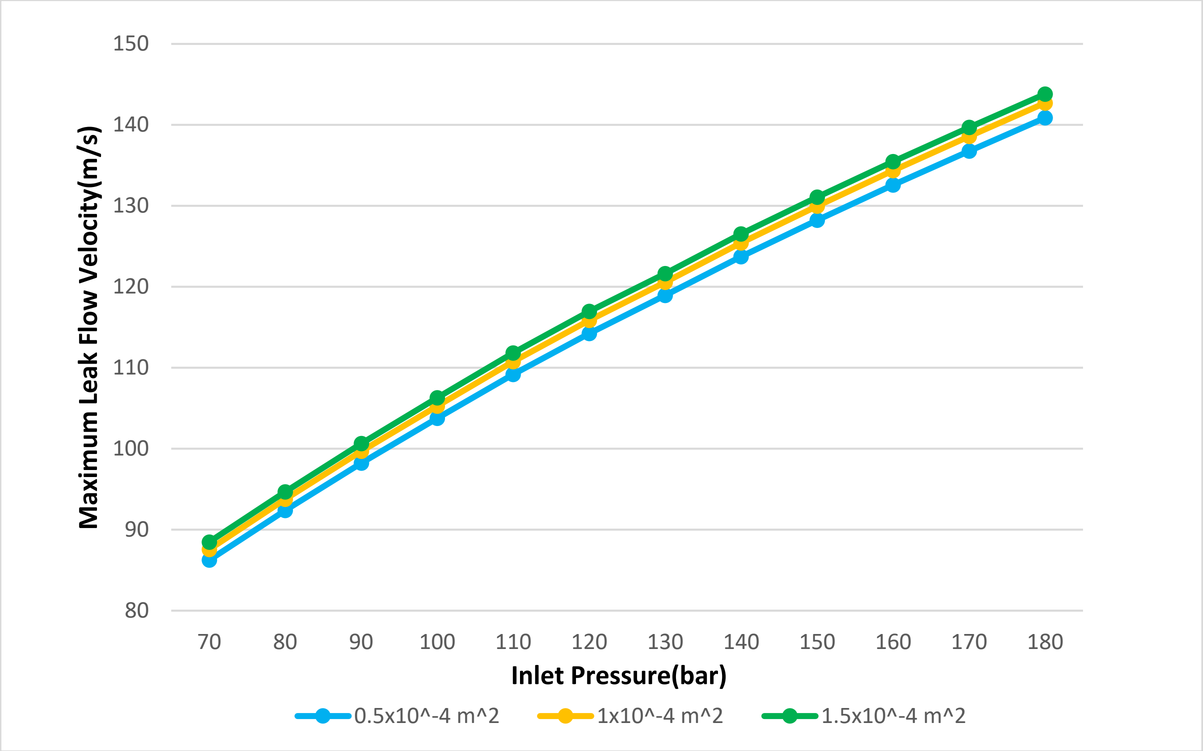

Figure 17: Variation of maximum leak flow velocity with pressure for rectangular cracks

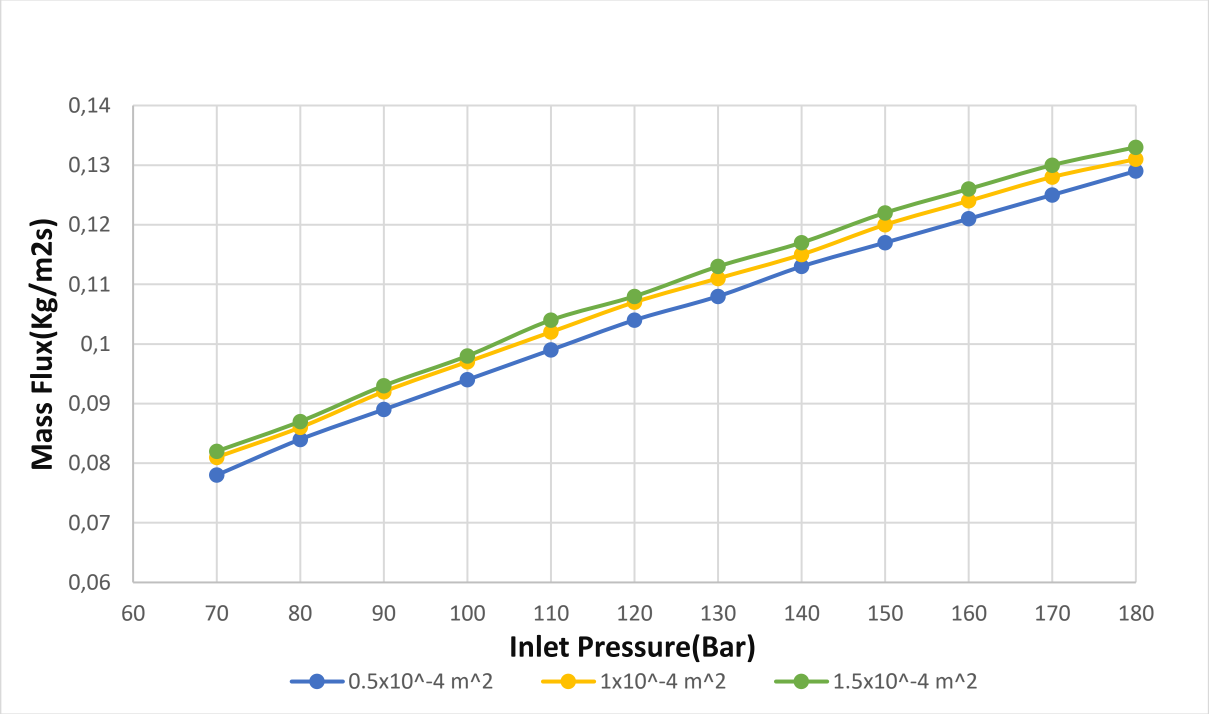

Figure 18: Variation of maximum mass flux with Pressure for rectangular cracks

The variation of the maximum leak flow velocity following the inlet pressure can be depicted in the graph in fig.17. The maximum leak flow velocity increases with the increase in the pressure at the pipe inlet. Also, the variation of the mass flux with the inlet pressure is shown in the graph in fig.18. It is seen that the mass flux increases with the increase in the inlet pressure.

E. Result for rectangular crack analysis with thermal condition

Here we have used the steel pipes to analyse the relation between the mass flow rate and the area of the crack. For the rectangular crack under various pressures with various crack areas, the change of mass flux is shown in the table 13. Here we observed that mass flux increases with the increase of pressure. Throughout the process, the temperature is considered constant. The mass flow rate analysis under various pressures gives the change of mass flux.

Table 9: Data table of the mass flux and velocity through circular crack opening at various pressures and temperature condition

Inlet/Stagnation Pressure(bar)	Saturation Temperature(k)	Inlet/Stagnation Temperature (K)	Subcooling	Crack/leakage Mass flow rate (kg/s)	Leak flow velocity(m/s)	Dimension (mms)
60	548.73	523	25.73	0.02238	188.420	1x0.5
60	548.73	523	25.73	0.02419	386.885	2x0.5
60	548.73	523	25.73	0.07085	241.415	3x0.5
70	559.95	523	36.95	0.00949	101.709	1x0.5
70	559.95	523	36.95	0.01978	119.647	2x0.5
70	559.95	523	36.95	0.06260	237.594	3x0.5
80	569.04	523	46.04	0.01275	109.41	1x0.5
80	569.04	523	46.04	0.10352	366.093	2x0.5
80	569.04	523	46.04	0.06089	335.848	3x0.5

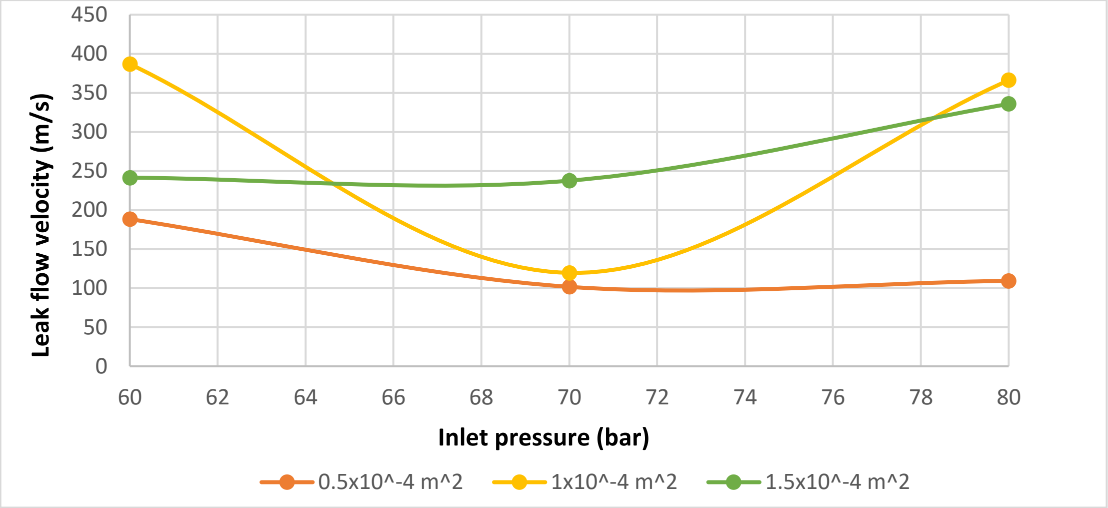

Figure 19: Variation of maximum leak flow velocity with pressure and temperature for rectangular cracks

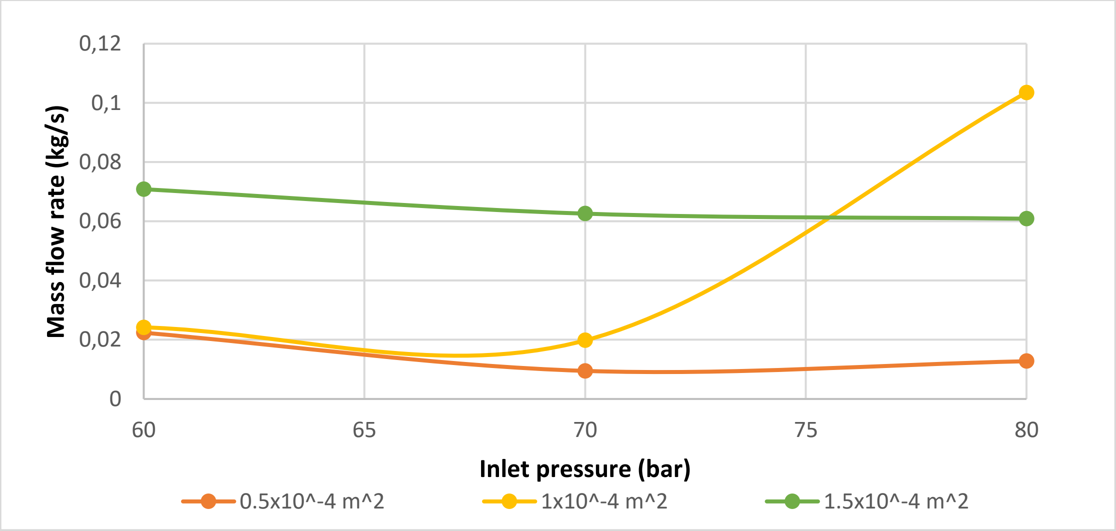

Figure 20: Variation of maximum mass flow rate with pressure and temperature for rectangular cracks

The variation of the maximum leak flow velocity following the inlet pressure can be depicted in the graph in fig.19. Also, the variation of the mass flux with the inlet pressure is shown in the graph in fig.20.

IV. CONCLUSION

In this study, leak flow prediction was carried out for predefined narrow circular and rectangular circumferential cracks over steel pipes using computational simulations. The results of the analysis are as follows:

1. The analysis revealed a range of mass flux during the leakage phenomenon for changing boundary conditions.

2. Both two-dimensional and three-dimensional configurations were studied, and circular crack scenarios were considered.

3. The analysis of both configurations revealed distinct patterns in the relationship between pressure increments and flow characteristics.

4. In the two-dimensional and three-dimensional setups, there was a clear trend of increasing flow velocity and mass flux at the outlet with pressure increments.

5. Conversely, for crack outlets, especially circular ones, while there was an overall increase in mass flux and leak flow velocity with rising pressure, there were notable inconsistencies in the growth pattern.

6. These findings underscore the importance of considering geometric factors, such as crack shape, in understanding fluid dynamics under varying pressure conditions.

ACKNOWLEDGMENT

The authors acknowledge the support of JIS college of engineering and Jadavpur University for the current work.

REFERENCES

[1] Han-Sun Huang; Fracture Characteristics Analysis of Pressured Pipeline with Crack Using Boundary Element Method; Advances in Materials Science and Engineering, 1-13, 2015

[2] J G Chen, F G Zang, Y Yang, K K Shi, and X L Fu; Fracture mechanics analysis of cracked structures using weight function and neural network method; (Kitakyushu City) 372,10-13, 2018

[3] Rached Ben-Mansour; Computational fluid dynamic simulation of small leaks in water pipelines for direct leak pressure transduction; Computers & Fluids 57, 110-123, 2012

[4] C N Amos and V E Schrock; Critical Discharge of Initially Subcooled Water through Slits; NUREG-CR--3475, USNRC, 1983

[5] Jun-Young Nama, Duck-Hee Seob, Sang-Yun Leeb, Woong-Ki Hwangb, Bo-Young Lee; The effect of residual stress on the SCC using ANSYS; Procedia Engineering 10, 2609–2614, 2011

[6] R C C Silva, J N C. Guerreiro, A F D. Loula; A Study of Pipe Interacting Corrosion Defects Using the FEM and Neural Networks; Advances in Engineering Software 38, 868-875, 2007

[7] S Ghosh and S K Saha; Modelling and Simulation of Subcooled Coolant Loss Through Circumferential Pipe Leakage; ASME J of Nuclear Rad Sci. 7(3): 034503, 2021. https://doi.org/10.1115/1.4048477

[8] H Zhu; A CFD simulation for oil leakage from damaged submarine pipeline; Energy 64, 887-899, 2014

[9] L Gardner; The use of stainless steel in structures Progress in Structural Engineering and Materials 7, 44-55, 2005

[10] H Ghaednia; Safe burst strength of a pipeline with a dent-crack defect: Effect of crack depth and operating pressure Engineering Failure Analysis 55, 288–299, 2005

[11] A Kundu, S Ghosh, S Mondal, A Ghosh and S Roy; A Comparative Analysis of Sensor Based Pipe Crack Detection System; In: Applications of Networks, Sensors and Autonomous Systems Analytics. Studies in Autonomic, Data-driven and Industrial Computing. Springer, Singapore, 71-78, 2022. https://doi.org/10.1007/978-981-16-7305-4_7

[12] S Ghosh, D Mukhopadhyay, S K Saha; An experimental analysis of subcooled leakage flow through slits from high pressure high temperature pipelines; International Journal of Pressure Vessels and Piping 88, 281-289, 2011. https://doi.org/10.1016/j.ijpvp.2011.05.008

[13] D W Sallet; Critical mass flow rates through pressure relief valves; College Park, Maryland, USA, Warme und Stofffibertragung 26, Springer-Verlag, 315-321, 1991

[14] G L Sozzi and W A Sutherland; Critical Flow of saturated and subcooled Water at High Pressure, General Electric Company, NEDO-13418, 1975

[15] S Manna, A Kundu, K Dey, S Roy, R Bhuit and S Ghosh; CFD simulation of circumferential crack in low pressure water pipelines; Materials Today Proceedings 80, Part 2, 806-812, 2023

[16] S Revankar, B Wolf and J Riznic; Flashing flow of Subcooled liquid through small cracks; Procedia Engineering 56, 445-461, 2013

[17] M Zhou, R Kulenovic, E Laurien, M Kammerer and X Schuler; Investigation On Temperature Fluctuations in Thermal Mixing Pipe Flows by Micro-Thermocouple Measurements; The 11th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, Operation and Safety, 2016

[18] K Ardron and R Furness; A study of the critical flow models used in reactor blowdown analysis; Nuclear Engineering and Design, 39, 257-266, 1976

[19] S Ghosh; Thermal-hydraulic Code prediction of Primary loop leakage in NPP; Journal of Emerging Technologies and Innovation Research 6(5), 162-167, 2019

[20] S Ghosh and S K Saha; Experimental validation of RELAP5 predictions for subcooled leak flow through narrow circumferential slits; Annals of Nuclear Energy 75, 507-515, 2015

[21] J Sievers, K Heckmann, H Grebner, G Moner and G Lerchl; Structural mechanical and thermal hydraulic aspects on the behaviour of crack like leaks in piping; Towards Convergence of Technical Nuclear Safety Practices in Europe, 2013

[22] G Palau-Salvador, P Gonzalez-Altozona and J Arviza-Valverde; Numerical modeling of cavitating flow for simple geometries using FLUENT V6.1; Spanish Journal of Agricultural Research 5(4), 460-469, 2007

[23] S K Choi, H Y Nam and J C Jo; Numerical Analysis of Evolution of Thermal Stratification in a Curve Piping System, Journal of the Korean Nuclear Society 32(2), 169-179, 2000

[24] X Zhou, K Li, R Tu, J Yi, Q Xie and X Jiang; A Modelling Study of the Multiphase Leakage Flow from Pressurized CO$_2$ Pipeline, 2016

[25] P T Williams and S Yin; Prediction Of The Leakage Rate For Cracked Pipes In Nuclear Power Plants; Transactions, 2013

[26] H S. Park, N H. Choi, S K. Chang, C H. Chung, S J. Yi, C K. Park and M K. Chung; Experimental study on a two-phase critical flow with a non-condensable gas at high pressure conditions; International Journal of Multiphase Flow 33, 1222-1236, 2007

[27] S Majumdar, K Kasza, S Bakhtiari, J Park, J Oras, J Franklin, C Vulyak and W Shack; Steam Generator Tube Integrity Issues: Pressurization Rate Effects, Failure Maps, Leak Rate Correlation Models, And Leak Rate in Restricted Areas, Nuclear Engineering Division Argonne National Laboratory; U.S.NRC, 2009

[28] J Zhang, H Yu, M Wang, Y Wu, W Tain, S Qiu and G Su; Experimental study on the flow and thermal characteristics of two-phase leakage through micro crack; Applied Thermal Engineering 156, 145-155, 2019

[29] B. E. Launder and D. B. Spalding; Lectures in Mathematical Models of Turbulence; Academic Press, London, England, 1972.

[30] T.H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu; A New k- ϵ Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation; Computers Fluids, 24(3):227-238, 1995.

[31] W. C. Reynolds; Fundamentals of turbulence for turbulence modelling and simulation; Lecture Notes for Von Karman Institute Agard Report No. 755, 1987.