Vol.2.Issue.3.2014

http://www.bomsr.com

BULLETIN OF MATHEMATICS AND STATISTICS RESEARCH

A Peer Reviewed International Research Journal

ISSN:2348-0580

RESEARCH ARTICLE

Analysis of the Liquid Fluid Flow Profile and Pressure Drop in the "L" Pipe using the Computational Fluid Dynamic Simulation Method

Dr. K. Jonah Philliph

Assistant Professor, Dept of Science and Humanities, Mother Theresa Institute of Engineering and Technology, Palamaner, Chittoor Dist, A.P

Article Info:
Article received :23/09/2014
Revised on:29/09/2014
Accepted on:30/09/2014

ABSTRACT

A piping system is a system that is widely used to move fluids, whether liquid, gas, or a mixture of liquid and gas from one place to another. The sizes and shapes of pipes' cross sections vary widely. Knowing the pressure drop of the flow in the pipe is crucial when planning a piping system. Pressure drop is highly correlated with a number of pipe characteristics, including pipe roughness, pipe length, pipe diameter, fluid type, flow speed, and flow form. This research aims to find out the results of fluid flow analysis in the L pipe, find out the type of flow pattern formed in the L pipe, and find out how much pressure drop occurs. This research was carried out by varying the flow rate and type of fluid. In this research, with a flow rate of 14 tons/hour using water, the Reynolds number was 35.252, the pressure drop ΔP was 2.7880 Bar. In ammonia fluid, the Reynolds number is 160.72, the pressure drop ΔP is 0.2812 Bar. In diesel fluid, the Reynolds number is 16.999, the pressure drop ΔP is 3.1640 Bar. In gasoline fluid, the Reynolds number is 90.199, the pressure drop ΔP is 1.8423 Bar. In ethylene glycol fluid, the Reynolds number is 1652.26, the pressure drop ΔP is 4.495 Bar. From these results, the greater the viscosity of the type of fluid, the higher the pressure drop and the smaller the Reynol number value.

1. Introduction

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems related to fluid flow. The history of CFD became known in the 60s - 70s which used the CFD concept which uses chemical reactions, but due to developments over time, this CFD application is needed in various other applications [1]. An example is a CAD application that uses the CFD concept to analyze stress in the design being created. The use of CFD in the industrial world is relatively new, first used around the 1990s in the aviation industry.

The field of study known as fluid mechanics examines the stability and motion of fluids and gases, as well as their interactions with surrounding and transiting solids. Fluids are materials that can be pushed or pulled and thus undergo motion. Fluids are malleable and not fixed in any one form. When fluids interact with solids, they take on the characteristics of the shapes they pass through [2].

2. Theoretical Foundation

Computational Fluid Dynamics (CFD): The field of Computational Fluid Dynamics (CFD) analyses fluid flow, heat transfer, and other related phenomena as a whole system. Such as chemical reactions using computer-based simulations (numerical). These calculation controls along with other calculation controls constitute the aforementioned space division or meshing. Later, at each control point the calculation will be carried out by the application with the specified domain boundaries and boundary conditions. This principle is widely used in the calculation process using computer computing assistance. Another example of the application of this principle is Finite Element Analysis (FEA) which is used to calculate the stress that occurs in solid objects[3].

Fluid: Fluids are materials that can be pushed or pulled and thus undergo motion. Fluids are malleable and not fixed in any one form. Different solids are formed by fluids depending on the shapes of the objects they interact with each other. The variables of fluid flow are the pressure, velocity, shear stress, and total pressure [4].

In regions where wall friction is negligible, shear stress can be disregarded, and the fluid behaves like an ideal one: incompressible and viscous at zero. Potential flow is the name for this hypothetical fluid flow. The law of conservation of mass and the principles of Newtonian mechanics holds true for potential flow. There are two primary features of potential flow:

1. There is no circulation or vortices so the potential flow is called irrotational flow.
2. There is no friction so there is no dissipation (release) of mechanical energy into heat

Reynolds Number: The Reynolds number is used to determine the basic properties of flow [5], whether the flow is laminar, transitional or turbulent and its location on a scale that shows the relative importance of turbulent tendencies compared to laminar as shown in the following equation:

$$N_{re} = \frac{D.v.\rho}{\mu}$$

In water fluids, a flow is classified as laminar if the flow has a Reynolds number (Re) of less than 2300. For transitional flows it is 2300 < Re < 4000, also known as the critical Reynolds number. Meanwhile, turbulent flows have a Reynolds number of more than 4000.

Density: Density or specific density (ρ) of a substance is a measure of the form of concentration of the substance and is expressed in mass per unit volume. This property is determined by calculating the ratio of the mass of the substance contained in a certain part to the volume of that part. The relationship can be expressed in the following equation:

$$\rho = \frac{m}{v}$$

Viscosity: Fluid viscosity is a measure of a fluid's resistance to deformation or change in shape. Viscosity is influenced by temperature, pressure, cohesion and the rate of transfer of molecular momentum. The viscosity of liquids tends to decrease as the temperature increases, this is because the cohesive forces in the liquid when heated will decrease as the temperature of the liquid increases, which causes the viscosity of the liquid to decrease. Viscosity is divided into two types, namely kinematic viscosity and dynamic viscosity or absolute viscosity.

1. Laminar: Laminar flow is defined as flow with fluid moving in layers or laminae with one layer sliding smoothly. This laminar flow has a Reynolds number of less than 2000.

2. Transition: Transition flow is a transition flow from laminar flow to flow turbulent. Turbulent flow has a Reynolds number between 2000 and 4000.

3. Turbulent: When the Reynolds number is greater than 4000, the fluid's particles move in a chaotic manner due to mixing and rotation of particles between layers, resulting in the transfer of momentum from one region to another.

Fluid Flow Discharge: Fluid flow discharge is a formula used to calculate fluid flow velocity, which is shown in the equation

$$Q = \frac{v}{t}$$

Then from the continuity equation you will get the formula as shown in the equation

$Q = A.V$, where A = ¼ π D^2
then the flow velocity in a cross section is:

$$v = \frac{q}{A}$$

Continuity Equation: The continuity equation states the relationship between the velocity of fluid entering a pipe and the velocity of fluid exiting (White, 1988). This relationship is expressed by:

$$Q = A\,V$$

Discharge is a quantity that states the volume of fluid flowing each time unit of time.

$$Q = \frac{V}{t}$$

If you substitute equations 6 and 7, you will get the equation: $V = \frac{V}{tA}$
If a fluid moves in a flowing pipe with a different cross-sectional area, then the flowing volume (Tipler.1998):

$$V = A.v.t$$

$$A_1.v_1.t_1 = A_2.v_2.t_2$$

Bernoulli's equation: The ideal Bernoulli equation means that the flow is constant along the path and ignores any losses that occur in the fluid path.

$$\frac{p}{\rho} + \frac{v^2}{2} + gz = \text{constant}$$

Pressure Drop: Pressure drop is a term used to describe the drop in pressure from one point in a pipe or water flow. What we refer to as "Pressure Drop" is actually the result of frictional forces exerted on the fluid as it moves through the tube. Flow resistance is the primary cause of friction. For a given viscosity, fluid density, and pipe diameter, the Reynolds number will determine the pressure drop in a single-phase flow[6].
Friction in pipes: One of the connection components in a piping system is a pipe bend (curved pipe) or elbow. Pipe bend functions to bend the direction of fluid flow in the pipe. However, pipe bends are more difficult to analyze because their surfaces become oval under bending moment loading. This causes pipe bends to have greater flexibility compared to straight pipes of the same size and type of material. This increased flexibility makes the pipe bend function as a thermal expansion absorber. With these various characteristics, pipe bends are a very important component in piping systems and require various considerations in the design process [7].
Types of fluid used

1. Water
2. Ammonia
3. Solar
4. Gasoline
5. Ethylene glycol

Fluid Flow in Closed Channels (Pipes): Pipes are closed channels that usually have a circular cross-section that are used to convey fluids with a full flow appearance. The fluid that flows through the pipe can be liquid or gas and the pressure can be greater or less than atmospheric pressure.
Piping System: A piping system is a system that is widely used to move fluids, whether liquid, gas, or a mixture of liquid and gas from one place to another. The decrease in pressure occurs due to flow turbulence which will cause large friction on the pipe walls, resulting in large head losses.
Various types of pipe connections: Piping connections can be grouped as follows:

1. Connection using welding
2. Connection using threads

Apart from the connections above, there are also special connections using gluing and clamping (for plastic pipes and glass fiber pipes). In refineries, generally low pressure pipes and pipes under 2" only use threaded connections.

Branch connection type

Types of branch connections can be grouped as follows:

- Direct connection (stub in)
- Connection using fittings (connecting tools)
- Connection using flanges (flange-flange).

The type of branch connection can also be determined based on specifications that have been made before designing or can also be calculated based on strength calculations, needs, without forgetting the effectiveness factor.

Branch connection type

Types of branch connections can be grouped as follows:

1. Direct connection (stub in)
2. Connection using fittings (connecting tools)
3. Connection using flanges (flange-flange).

The type of branch connection can also be determined based on specifications that have been made before designing or can also be calculated based on strength calculations, needs, without forgetting the effectiveness factor.

Energy Losses in Piping Systems

1. Longitudinal Losses: Longitudinal loss caused by friction along the pipe circumference. Figure 1 shows the friction factor value on the Moody diagram.

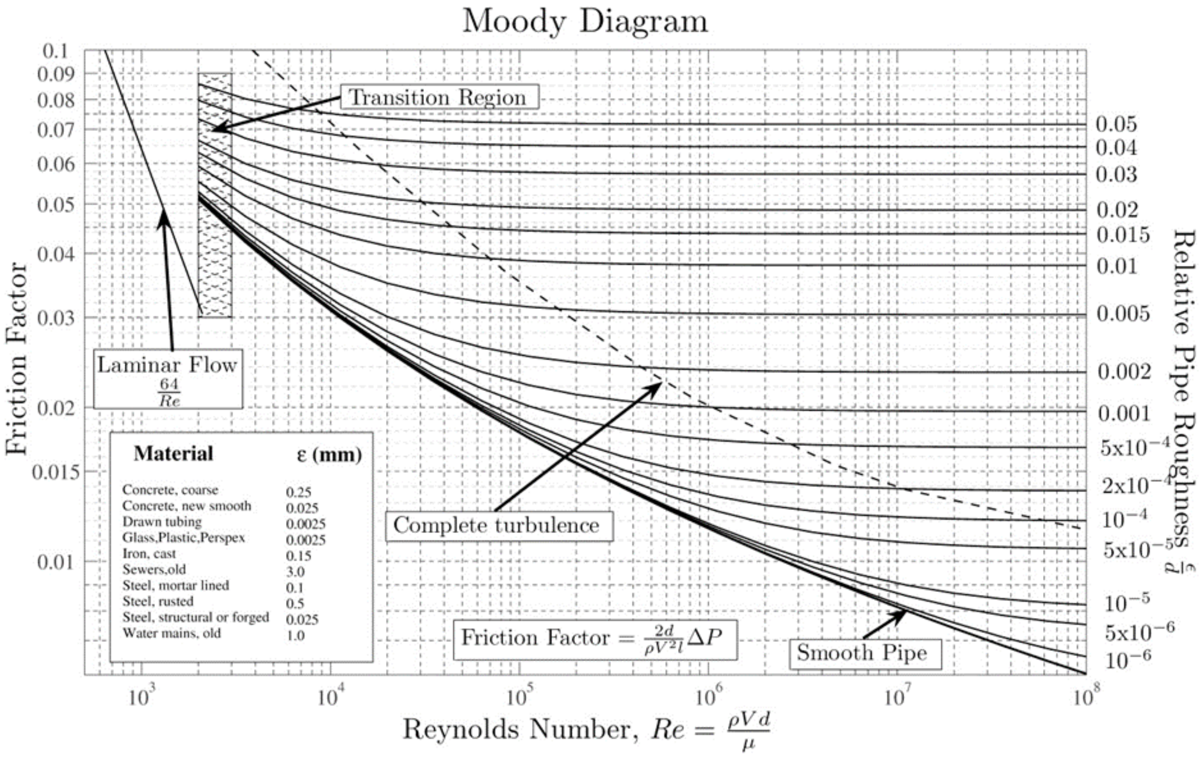


Figure 1: Moody diagram [8]

2. Local Losses: Local losses are head losses caused by connections, bends, valves, enlargement/reduction of cross-section, so that according to Eckert, Michael (2006) [9] which can be seen in the following equation:

$h_1 = h_0 + h_b + h_c$

Simulation Using Computational Fluid Dynamics (CFD) Software

To evaluate and enhance the performance of a dynamic system, simulation can be used to create a model of the system and simulate its behaviour. One more way to define a simulation:

1. A way to reproduce the conditions of a situation, using a model, for learning, testing, training, etc.

2. Modelling of a process or system where the model imitates the response of a real system to every event that occurs at any time.

Computational Fluid Dynamics (CFD)

As a method for calculating the flow of a fluid, Computational Fluid Dynamics (CFD) makes use of computers to perform calculations on each component. The principle is that a space containing fluid that will be calculated is divided into several parts, this is often called cells and the process is called meshing. These divided parts are a control calculation that will be carried out in the application.

Autodesk Inventor: Autodesk Inventor is a specialised software package developed to meet a wide range of engineering requirements, including but not limited to product design, machine design, mould design, and construction design. Since all objects and relationships between geometry in Autodesk Inventor are parametric, changes can be made after the geometry has been finalised without having to redo the entire model.

Autodesk Simulation CFD: Autodesk Simulation CFD is a fluid and thermal flow simulation program to help analyze fluid flow and heat exchange inside and outside buildings. Factors such as aesthetic appearance, thermal comfort, indoor air quality, and security requirements are taken into consideration. Autodesk Simulation CFD also provides realistic simulations to help create energy efficient and sustainable designs.

3. Research Methodology

This research is numerical computing in nature where the initial research begins with drawing the geometry of the L-shaped pipe flow in the liquid ammonia unit using Autodesk Inventor software and then for numerical calculations using Autodesk Simulation CFD software.

Research variable

The fixed variables in this research are pipe length: 10 m, connection type: direct connection, number of bends: 1, pressure: 7.84532 bar, pipe diameter: 6 in, pipe material: stainless steel. The independent variables in this research are:

Flow rate: 10, 11, 12, 13, and 14 tons/hour; fluid types: water and ammonia (liquid phase), diesel (diesel fuel), gasoline (gasoline) and ethylene glycol. The dependent variables in this research are: pressure drop (ΔP), Reynolds number (N_{re}), and fluid flow profile.

Work procedures: Overall, the simulation process for this research has 6 steps, which previously started with geometry formation, which can be seen in the simulation procedure flow diagram in Figure 2.

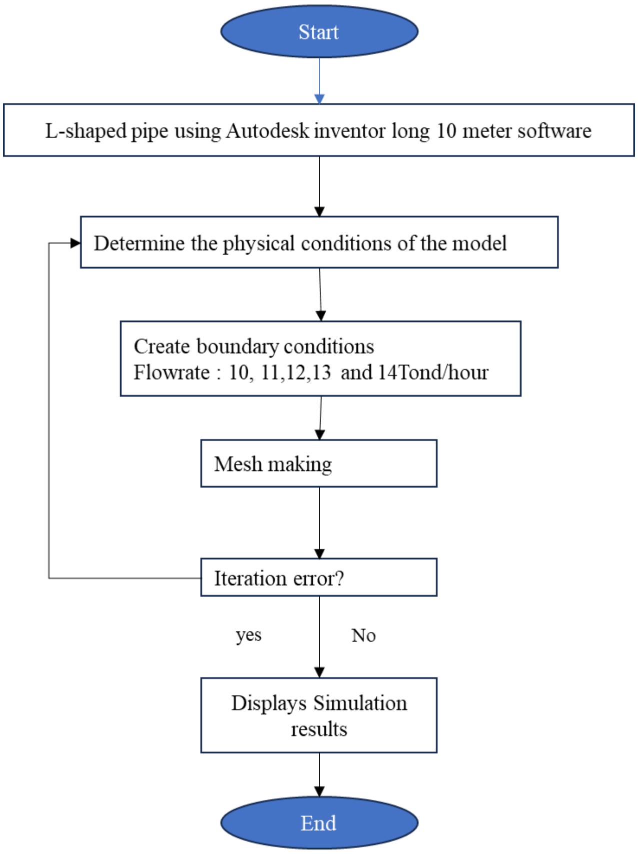


Figure 2 Flowchart for Geometry Creation and simulation process

Depiction of Continuous Pipe Flow Geometry: The first thing that needs to be done before carrying out the simulation process is to create a model of the flow that occurs in pipe L. To create the model, use Autodesk Invertor software.

Modelling Design Stage in Autodesk Invertor: Autodesk Invertor is a computational analysis software to help create images or models to be simulated in the Autodesk Simulation CFD software.

In Autodesk Invertor, you can design the image you want

The simulated image will be in the form of a 3D image like Figure 3.

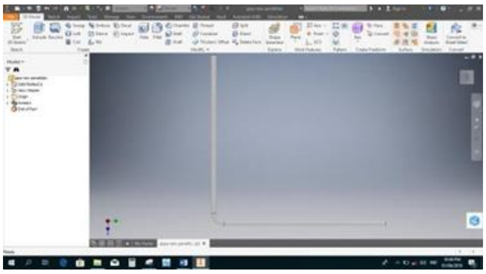


Figure 3 The L pipe in the Autodesk inverter software is 3D in shape

4. Results and Discussion

Research result

Research results on L pipes with variations in fluid type and flow rate.

Table 1 Research Data

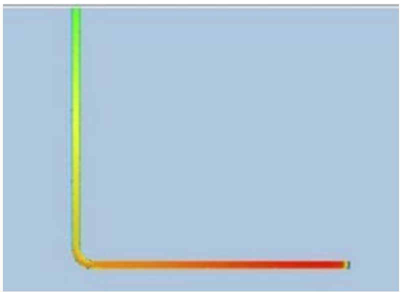
Run	Fluid type	Inlet Pressure (Ton/Hour)	Exit Pressure (Ton/Hour)	Pressure drop (Bar)	Reynold Number
1	water	10	7.8453	5.2065	25.1804
2	water	11	7.8453	5.1895	27.6914
3	water	12	7.8453	5.1320	30.2163
4	water	13	7.8453	5.1632	32.3612
5	water	14	7.8453	5.0532	35.6321
6	Ammonia	10	7.8453	7.5630	114.8
7	Ammonia	11	7.8453	7.5650	126.365
8	Ammonia	12	7.8453	7.5650	137.215
9	Ammonia	13	7.8453	7.5640	149.621
10	Ammonia	14	7.8453	7.5640	160.72
11	Diesel	10	7.8453	4.7763	12.426
12	Diesel	11	7.8453	4.7562	13.24
13	Diesel	12	7.8453	4.7361	14.58
14	Diesel	13	7.8453	4.7160	15.864
15	Diesel	14	7.8453	4.6959	16.365
16	Gasoline	10	7.8453	6.0932	65.389
17	Gasoline	11	7.8453	6.0965	70.8465
18	Gasoline	12	7.8453	6.0998	77.3145
19	Gasoline	13	7.8453	6.1031	83.3155
20	Gasoline	14	7.8453	6.1064	90.1936
21	Ethylene Glycol	10	7.8453	3.5362	1180.18
22	Ethylene Glycol	11	7.8453	3.5264	1298.26
23	Ethylene Glycol	12	7.8453	3.5320	1416.22
24	Ethylene Glycol	13	7.8453	3.3562	1534.24
25	Ethylene Glycol	14	7.8453	3.3503	1652.36

Table 1 is the result of research analysing fluid flow profiles and pressure reduction in water, ammonia, diesel, gasoline and ethylene glycol fluid types.

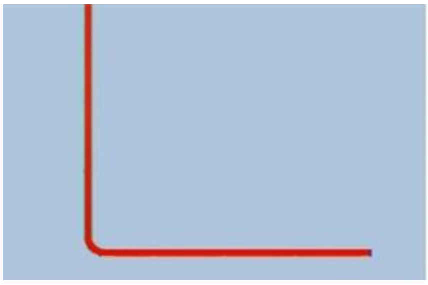
From the results of this research, it can be seen that between these types of fluids there are varying pressure drops, where the highest-pressure drop is found in the ethylene glycol fluid type and the lowest pressure drop is found in the ammonia fluid type.

Results of Fluid Flow Profile: Analysis and Pressure Drop using the CFD simulation method for fluid types Water, Ammonia, Diesel, Gasoline, Ethylene Glycol

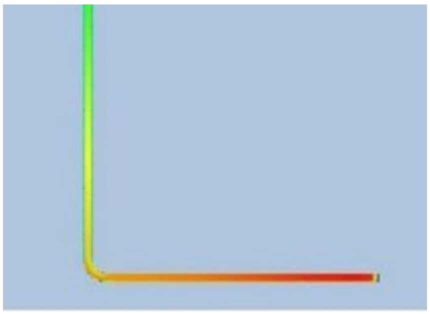
a. Water fluid at a flow rate of 14 tons/hour



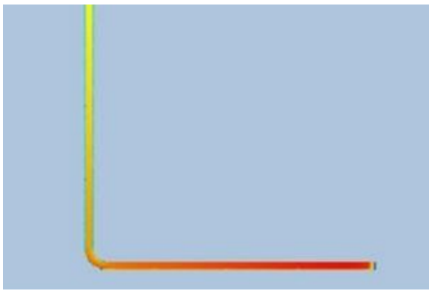
b. Ammonia fluid at a flow rate of 14 tons/hour



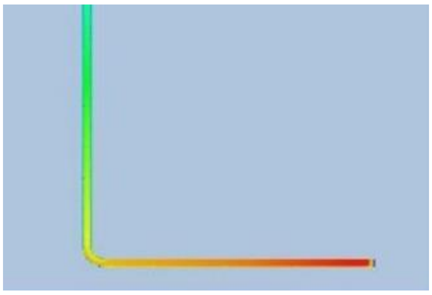
c. diesel fluid at a flow rate of 14 tons/hour



d. Gasoline fluid at a flow rate of 14 tons/hour



e. Ethylene glycol fluid at a flow rate of 14 tons/hour



From the pictures above, you can see the simulation results of the fluid flow profiles of water, ammonia, diesel, gasoline and ethylene glycol and the pressure drop in the pipe using Autodesk CFD simulation software, showing that there is a pressure spread marked by a change in red color to yellow, blue and green. The inlet pressure marked in red has a pressure of 7.845 Bar, however the pressure changes that occur in the pipe are not the same due to the influence of the viscosity of the fluid. Red is the highest pressure, followed by orange, yellow and green. The green color indicates the lowest pressure.

From the five pictures above, it can be seen that the lowest pressure is in the sixth picture because in the sixth picture the fluid used is ethylene glycol which has a very high viscosity, namely 21.4, so the pressure drop that occurs in the pipe is very high, resulting in fluid pressure. in the pipe is very low. This can be seen from the change in color, where the green color is the lowest pressure near the bend in the pipe. It can be seen that the ethylene glycol fluid after passing through the elbow can almost no longer flow.

Then followed by Figure 3 which is of the diesel type with a viscosity of 2.08 cP. This is followed by the first image where the fluid is water with a visosity of 1.003 cP. This is followed by a gasoline type fluid with a viscosity of 0.392 cP. And then followed by ammonia type fluid with a viscosity of 0.22 cP, in the ammonia fluid it can be seen that there is no color change at all, the entire pipe is red, this is because the viscosity of ammonia is very low which results in almost no pressure drop when viewed from 2nd picture, however, if we look at the value, the pressure drop still occurs but with a very small size, namely 0.28127 Bar. With this very small pressure drop value, the image simulation does not show any color variations like in other fluids.

The viscosity value of the fluid greatly influences the decrease in fluid pressure, this is in accordance with the theory which states that the viscosity value of a fluid is directly proportional to the decrease in pressure of the fluid. A large pressure drop greatly affects the performance of a pump to transfer fluid from one place to another. If the type of fluid has a high viscosity, then the pressure generated from the pump must be high so that it can flow the fluid to the desired location.

Results of Fluid Flow Profile Analysis at elbow 900

The following is a fluid flow profile at a 90° bend.

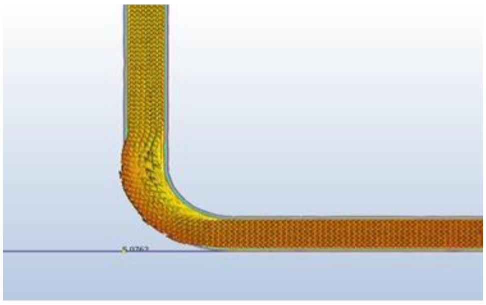


Figure 4 Fluid flow profile at a 90o bend

From Figure 4 it can be seen that the fluid flow profile at the elbow experiences a strong push so that it experiences a strong impact on the pipe wall. From the picture above, it can be explained that elbows in pipes greatly affect pressure loss in the piping system because the fluid flowing in the pipe experiences collisions so that the fluid pressure will automatically decrease and slow down the pressure of a fluid. Apart from that, the presence of elbows in pipes can result in shocks in the piping system so this must be really taken into account when designing a piping system both in everyday life and in the industrial world which can result in major losses such as damage to supports due to the formation of turbulent flow. large due to the presence of the elbow.

The decrease in pressure occurs due to flow turbulence which will cause large friction on the pipe walls, resulting in large head losses.

Discussion

In this research, a variety of fluids with different viscosities were used, namely water, ammonia, diesel, gasoline and ethylene glycol. Where the viscosity is sequentially 1.003, 0.22, 2.08, 0.392, and 21.4. and varied the flow rate, namely 10, 11, 12, 13, 14 tons/hour. Next, the results obtained in this research will be discussed. From Table 1, it can be seen from several types of fluids that the pressure drop becomes smaller along with the viscosity value of the fluid. If the viscosity value is lower, the pressure drop will be smaller, and vice versa, if the fluid viscosity value is higher, the pressure drop that occurs will also be greater.

Based on what is known, if the viscosity of a fluid is high, the resulting pressure drop will also be greater. This is because fluids that have high viscosity have resistance to shear pressure. Shear pressure is the pressure exerted on the fluid to move. Based on Newton's law of viscosity, shear pressure is directly proportional to viscosity [10]. Meanwhile, regarding flow rate, the greater the flow rate, the greater the pressure drop. This can be explained by the fact that if the flow in the pipe is large, the friction that occurs along the pipe between the fluid surface and the inner surface of the pipe is greater, so that the resulting pressure drop is greater. big.

The Reynolds number is also influenced by viscosity and flow rate, if the viscosity is high then the Reynolds number will be small. If the flow rate is high, the Reynolds number will be higher. If the Reynolds number is very high, it is necessary to pay attention to it in the piping system design process which can result in major losses such as damage to supports and pipe leaks.

Analysis of the Relationship between Flow Rate and Fluid Type and Pressure Drop

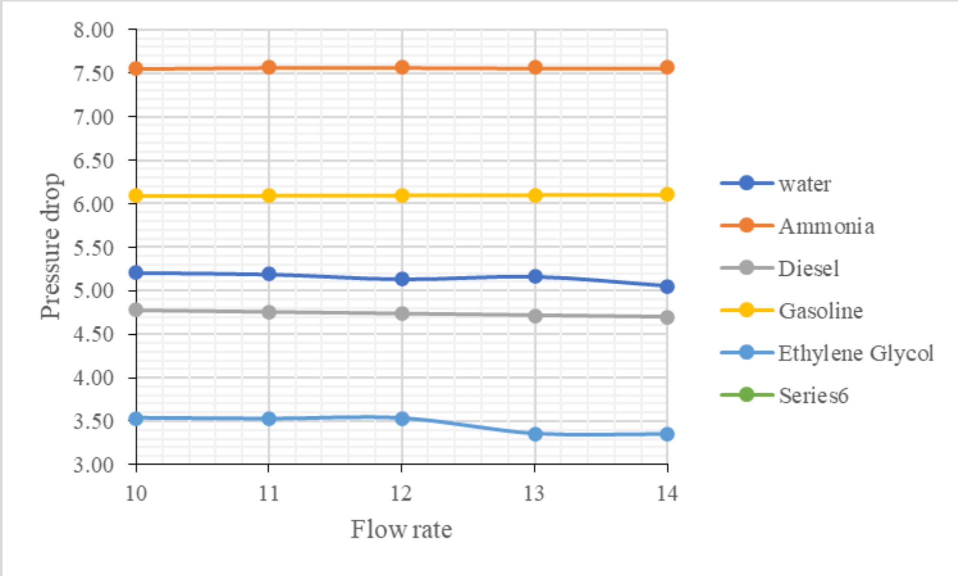


Figure 5 Flow rate relationship

From Figure 5 we can see the comparison of pressure drop and flow rate for the fluid types water, ammonia, diesel, gasoline and ethylene glycol using Autodesk CFD simulation. The highest-pressure drop is found in the ethylene glycol fluid type with a flow rate of 14 tons/hour of 4.49503 bar. Meanwhile, the lowest pressure drop is found in the ammonia fluid type with a flow rate of 10 tons/hour of 0.27 bar. This is due to the difference in viscosity of the type of fluid, if the viscosity is high then the pressure drop is also high. Apart from that, the flow rate also affects the pressure drop, if the flow rate is high then the pressure drop will also be high.

Analysis of the Relationship between Flow Rate and Fluid Type and Reynold Number (Reynol number)

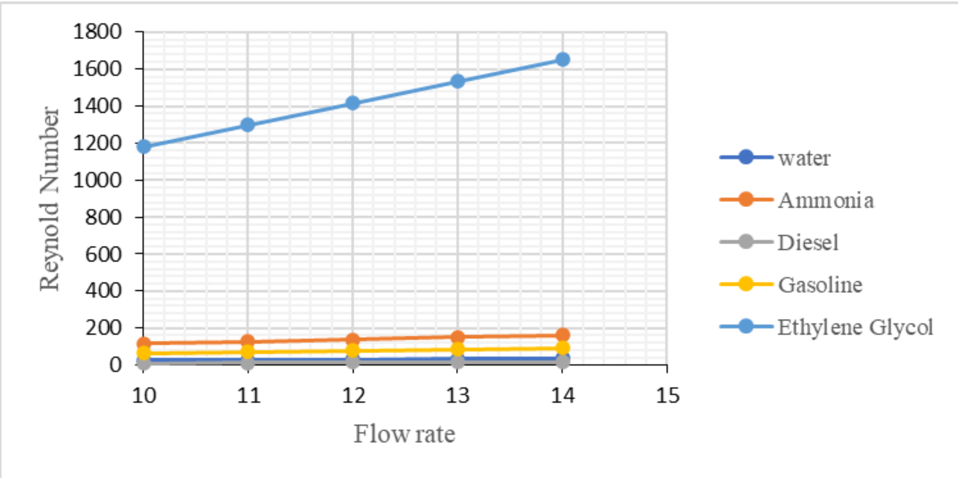


Figure 6 Comparison of Reynolds numbers

From Figure 6, you can see the comparison of the Reynolds number with Autodesk CFD simulation for the fluid types: water, ammonia, diesel, gasoline and ethylene glycol. The highest Reynol number is found in the ammonia type of fluid with a flow rate 14 tons/hour amounting to 160.720. Meanwhile, the lowest Reynolds number is found in the ethylene glycol fluid type with a flow rate of 10 tonnes/hour of 1180.18. This is due to the difference in viscosity of the type of fluid, if the viscosity is high then the Reynolds number will be low. Apart from that, the flow rate also affects the Reynolds number, if the flow rate is high then the Reynolds number is also high.

From Figure 4 & 5 it can be seen that there are two types of flow patterns, namely turbulent and laminar. Laminar flow patterns are found in ethylene glycol fluid types, and turbulent flow patterns are found in water, ammonia, diesel and gasoline fluid types.

5. Conclusions and Suggestions

After analysing the fluid flow profile and pressure drop using Autodesk CFD simulation, several conclusions were obtained, including the following:

1. Results of simulations carried out on L-joint pipes with variations in fluid type and flow rate where the greater the viscosity of a fluid, the greater the pressure drop and the greater the flow rate, the greater the pressure drop.
2. The highest Reynolds number occurs in the ammonia type of fluid with a flow rate 14 tons/hour, namely 160.720 and the lowest occurred in the ethylene glycol fluid type with a flow rate of 10 tons/hour, namely 1,180 tons/hour.
3. The lowest pressure drop obtained from the simulation occurred in the ammonia fluid type with a flow rate of 10 tons/hour of 0.27799 bar, while the highest pressure drop occurred in the ethylene glycol fluid type pipe material with a flow rate of 14 tons/hour of 4.49503 bar.

To develop this research to make it even better, the author provides suggestions for the future to vary the type of material in the pipes used and vary the shapes of other pipes.

References

[1]. Fox, R. W.; McDonald, A. T.; Pritchard, Phillip J. (2004). Introduction to Fluid Mechanics (6th ed.). Hoboken: John Wiley and Sons. ISBN 978-0-471-20231-8.

[2]. Fouz, Infaz "Fluid Mechanics," Mechanical Engineering Dept., University of Oxford, 2001, p. 96

[3]. Dwivedi, P. N. (1977). "Particle-fluid mass transfer in fixed and fluidized beds". Industrial & Engineering Chemistry Process Design and Development. 16 (2): 157–165. doi:10.1021/i260062a001

[4]. Falkovich, Gregory (2018). Fluid Mechanics. Cambridge University Press. ISBN 978-1-107-12956-6.

[5]. Bird, R. Byron; Stewart, Warren E.; Lightfoot, Edwin N. (2006). Transport Phenomena. John Wiley & Sons. ISBN 978-0-470-11539-8.

[6]. White, F. M. (1974). Viscous Fluid Flow. New York: McGraw–Hill. ISBN 0-07-069710-8

[7]. Pope, Stephen B. (2000). Turbulent Flows. Cambridge University Press. ISBN 0-521-59886-9.

[8]. Brown, Glenn. (2002). The History of the Darcy-Weisbach Equation for Pipe Flow Resistance. Proc. Environ. Water Resour. Hist.. 38. 10.1061/40650(2003)4.

[9]. Eckert, Michael (2006). The Dawn of Fluid Dynamics: A Discipline Between Science and Technology. Wiley.

[10]. Sergey P. Kiselev , Evgenii V. Vorozhtsov , Vasily M. Fomin, Foundations of Fluid Mechanics with Applications, Problem Solving Using Mathematica® , Springer Science, 1999