Progress in Engineering Application and Technology Vol. 1 No. 1 (2020) 409-421
© Universiti Tun Hussein Onn Malaysia Publisher's Office

PEAT

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/peat
e-ISSN : 0000-0000

Effect of Different Pipe Diameters to The Flow Behaviour of The Pipeline in UTHM Biodiesel

Stefina Edora Anak Pelias Pog[1], Razlin Abd Rashid[1]*, Zuliazura Mohd Salleh[1], Nik Normunira Mat Hassan[1], Rosniza Hussin[1]

[1]Faculty of Engineering Technology,
Universiti Tun Hussien Onn Malaysia, Pagoh, 84600, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/peat.2020.01.01.043
Received 26 September 2020; Accepted 09 September 2020; Available online 02 Month 2020

Abstract: Pipelines transport flammable or explosive materials, such as natural gas or oil raise specific safety concerns and have encountered various incidents. The aim of this study is to determine the effect of the pipeline's size on the flow behavior and to identify the pipe diameter suitable for fluid properties of B100 in the UTHM Biodiesel Pilot Plant. The study was conducted with three different pipe sizes of its inlet diameter named, Geometry 1 with 120 mm, Geometry 2 with 180 mm and Geometry 3 with 290 mm. There are three types of inlet velocity used which is Velocity 1 with 0.98 m/s, Velocity 2 with 1.1 m/s and Velocity 3 with 1.15 m/s. Geometry 2 and Velocity 2 are the actual parameters of the pipeline in this study. The study was conducted based on UTHM Biodiesel Pilot Plant which focuses on the trans-esterification inlet process. This study was used B100 fluid properties as a raw material in the pipe. The simulation procedure was performed in a structured grid by using ANSYS. Pressure and velocity measurements are made for three types of pipelines. Pressure and velocity are also theoretically studied using 3D-CFD analysis to show the flow behavior of three geometry types and three different inlet velocity values. High velocity value can cause noise, vibration, and higher stress. Geometry 1 achieved a good flow behavior when Velocity 1 was applied. Selecting Velocity 2 for Geometry 2 is recommended. Velocity 3 are suitable for Geometry 3 because of the pipeline size. Overall a good flow behavior in the pipeline will improve the efficiency of pipeline and its lifespan.

Keywords: Diameters, Velocity, Pressure, B100

1. Introduction

Pipelines are one of the fastest and most effective transport of oil and natural gas. Pipeline also has a large volume which it helps complete the production of the oil pipeline. With the rapid construction of pipeline networks around the world, pipeline accidents are bound to occur. These incidents cause damage to property and threaten both safety of employees and the environment. According to statistics,

*Corresponding author: razlin@uthm.edu.my

hazardous liquid was transported through 279,000 km of pipelines until December 2012 and there were 928 accidents that contributed to 218 deaths [1]. Based on studies that examining the cause of accidents there are several factors that lead to pipeline failure, such as external disruptions, corrosion, material defects, structural damage, improper operation and natural disaster [1]. The ability to monitor and evaluate the interior of the pipe is important to maintain the flow behavior of pipeline. The aim of this study is to identify the diameter of pipe that suitable for B100 in UTHM Biodiesel Pilot Plant and to examine the flow behavior in the pipeline by using three different size of geometry and inlet velocity. A 3D-Computational of Fluid Dynamic study by using Ansys Fluent was conducted for the same three pipelines to compare and evaluate flow behavior based on velocity and pressure. Evaluation of different size of geometry and three different inlet velocity in order to propose the best selection in this study.

Biodiesel is a biofuel that is renewable, biodegradable, non-toxic and environmentally friendly. It has been confirmed that biodiesel B100 can be used effectively as an alternative fuel in a small agricultural diesel engine [2]. Chemically, biodiesel is a fuel composed of mono-alkyl ester of long chain fatty acids derived from vegetable oils or animal fats, designated B100, and meeting the requirements of ASTM (American Society for Testing and Materials) D 6751 or EN (European Norm) 14214 [3]. Biodiesel production has now been greatly increased to compete with fossil fuels. Biodiesel production around the world as shown in Figure 1.

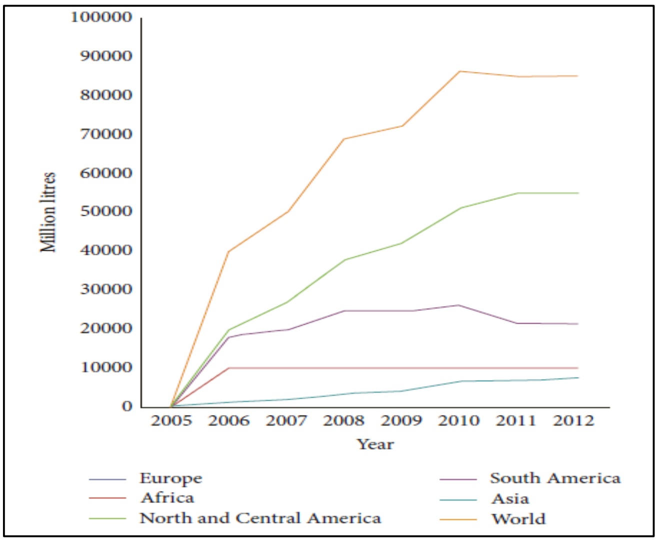

Figure 1: Production of biodiesel in recent years [4]

The pipe parameters do affect the behavior of the fluid in the pipe. When the internal diameter is decreased, the flow area increases and the fluid velocity reduces at a given flow rate. If the inner diameter of the pipe is reduced, the flow area will decrease, the liquid velocity will increase and the friction head loss will increase. Velocity plays a major role in pipeline performance. By increasing the speed of the water inlet, it helps improve the efficiency of the pipeline [5].

1.1 Pipe Losses

Friction and head losses in pipes, bends and valves present a challenge when it comes to transporting fluids in process industry pipe lines [6]. Head loss in the pipe is affected by a number of factors, including the viscosity of the fluid, the size of the inner pipe diameter, the internal roughness of the inner pipe surface, the change in elevation between the ends of the pipe and the length of the pipe along which the fluid travels [7]. Major head loss due to the flow of fluid flowing through the pipe is affected by the viscosity, velocity, roughness and geometry of the pipe. While the minor head loss is caused by the installation of pipes such as connection, reduction of the pipe valve, etc. [19]. According to Bernoulli's theorem, the water raises its velocity when it travels from a high-pressure area to a low-pressure region and the converse is also true, usually when it flows from a high-pressure region to a low-pressure region. Loss of velocity does not always mean that water has moved from a low-pressure region to a high-pressure region, as friction can lead to a loss of velocity [8].

Friction head losses can be examined in straight pipes of different sizes or angles, including laminar, transitional and turbulent flow regimes in smooth pipes over a range of Reynolds numbers from 103 to nearly 105. It is necessary to measure the pressure not only from the total head and pipe diameter, but also from the difference in head loss which is divided into major and minor head losses [8]. The Darcy – Weisbach equation is the most commonly used equation to calculate major tube head losses. According to Darcy, head loss is directly proportional to the velocity and indirectly to the diameter of the pipe [6]. It can be used to work out the pressure drop or the flow rate down such a pipe, and the purpose of the chart was to provide a graphic representation of the function of C.F.Colebrook in collaboration with C.M.White, which provided a practical form of transition curve to bridge the transition zone between smooth and rough pipes, the region of incomplete turbulence[8].

2. Model Development

2.1 Preparation of Model

The inlet for the esterification process in the biodiesel pilot plant of the University Tun Hussein Onn Malaysia was selected for the study. The information of the pipe was collected, including the diameter, length and inlet velocity of the pipe. This research modelled a pipe with a diameter of 180 mm and a length of 1230 mm. Each part of the model was created in a SolidWorks to be analysed in a schematic representation. The fluid properties of the B100 enter at the top of the pipe and exit on the right side of the pipe. Simulation of three types of pipe were carried out for different value of velocity including Velocity 1 with 0.98 m/s, Velocity 2 with 1.1 m/s and Velocity 3 with 1.15 m/s. Three different size of pipeline were categorised as Geometry 1 with 120 mm, Geometry 2 with 180 mm and Geometry 3 with 290 mm. Geometry 2 and Velocity 2 was the actual parameter of the pipeline in this study.

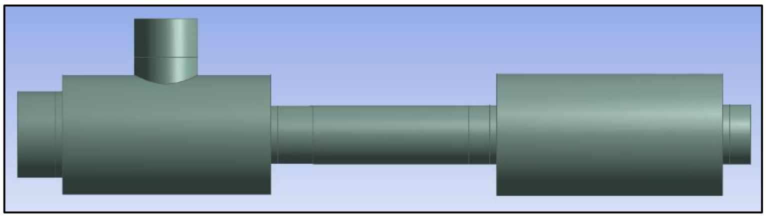

Figure 2: Fluid Domain of Geometry 1 in ANSYS Fluent

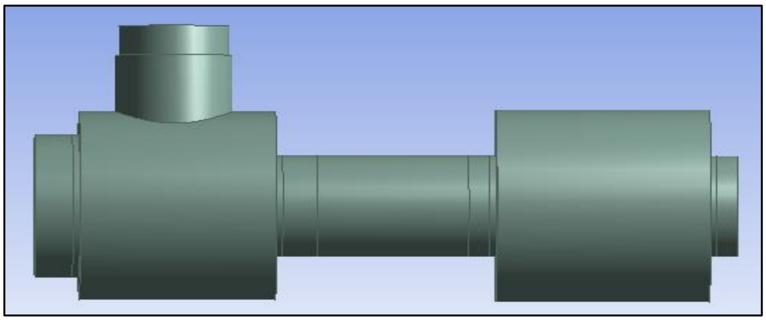

Figure 3: Fluid Domain of Geometry 2 in ANSYS Fluent

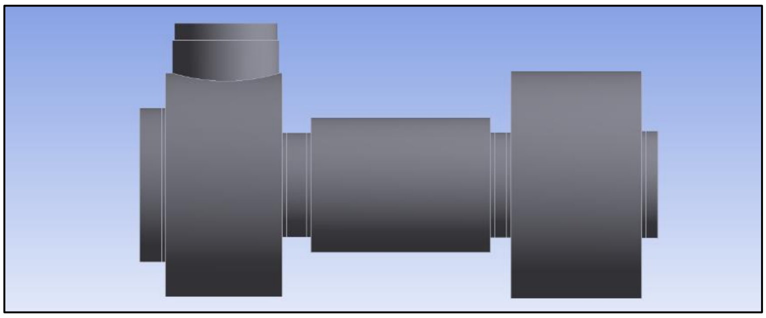

Figure 4: Fluid Domain of Geometry 3 in ANSYS Fluent

2.2 Model Equations

Fluid flow behavior is characterised by Navier-Stokes (NS) equations. The Navier-Stokes equations consist of a continuity equation for the conservation of mass as in Eq.1, momentum equations as in Eq. 3, and energy equations as in Eq. 4. Momentum equations as in Eq. 2 are sometimes referred to as NS equation. The total mass balance is Input – output = accumulation assuming that the mass input = mass output is not stored [5]. The general form of the Navier-Stokes Equation was shown as in Eq. 5

Continuity equation,

$$\frac{D\rho}{Dt} + \rho\frac{\partial U_i}{\partial x_i} = 0 \quad \text{Eq. 1}$$

Momentum equation,

$$\rho\frac{\partial U_j}{\partial t} + \rho U_i\frac{\partial U_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_i} + \rho g_j \quad \text{Eq. 2}$$

$$\tau_{ij} = -\mu\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial U_j}\right) + \frac{2}{3}\delta_{ij}\mu\frac{\partial U_k}{\partial x_k} \quad \text{Eq. 3}$$

Energy equation,

$$\rho c_\mu\frac{\partial T}{\partial t} + \rho c_\mu U_i\frac{\partial T}{\partial x_i} = -P\frac{\partial U_i}{\partial x_i} + \lambda\frac{\partial^2 T}{\partial x_i^2} - \tau_{ij}\frac{\partial U_j}{\partial x_i} \quad \text{Eq. 4}$$

General form of The Navier Stokes equation,

$$\frac{\partial(\rho\Phi)}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho U_i\Phi - \Gamma_\Phi\frac{\partial \Phi}{\partial x_i}\right) = q_\Phi \quad \text{Eq. 5}$$

2.3 Numerical Simulation

Grid independent test was used to define the progress of the results using successively smaller cell sizes for the calculations. The calculation will follow the correct answer as the mesh is smoother. The grid independence study was conducted with three different grid sizes with cell counts ranging from 50 000 to 300 000. It has been observed that the numerical results obtained have become independent of the total number of computational cells above 275 217. The total number of computational cells used to classify the whole geometry was 275, 217. This was found to be ideal for good predictions using the Reynold Stress Method and taking into consideration the computational time required [11]. Numerical simulation is a convenient way to gain insight into some of the explosions and impact problems that are difficult to achieve through theoretical modeling and experimental techniques [14].The operating conditions are defined as the standard atmospheric pressure 101 Pa, which is defined as acting downwards in the main body of the inverted T junction of the pipe.

2.3 Boundary Condition

Computational domain is divided into three sections namely the Geometry 1, Geometry 2 and Geometry 3. Three values of velocity are set at the inlet with three different pipe geometries. The boundary conditions are inlet velocity is 0.98 m/s, 1.1 m/s and 1.15 m/s, the outlet pressure is 101 Pa and the density of B100 is 878 kg/m^3. ANSYS Fluent default meshing was used with inflation on pipe and the mesh size was changed and the results were compared until the results obtained stable and symmetry contours. The maximum number of nodes and elements was 62,597 and 275,217.

3. Results and Discussion

All result was analysed as a comparison of overall performance between pressure and velocity of the pipe in this study. The main purpose of the comparison carried out was to examine the effect of the pipe when the velocity and diameter of pipe changed. When the velocity increased, the pressure of the pipe was recorded.

3.1 Comparison of Velocity and Pressure Graph for Geometry 2 (Actual Pipe)

Based on the simulation data shown in Figure 5 and Figure 6, the maximum pressure of the pipe was 967 Pa and the maximum velocity was 1.40 m/s. From Figure 5 Geometry 2 is suitable for inlet velocity 2 of 1.1 m/s. The unstable flow behavior of velocity shown in Figure 5 is that the shape of the pipe is inverted in T. It causes the trend line of the graph was not stable at the distance until 20 mm and start to rise until distance of 40 mm the inlet of the pipe is at the top where it causes the velocity line is not stable. The different size of the pipeline causes a change in velocity and pressure. At the center of the pipe, where it started to reduce the size of its diameter, the velocity started to increase as well. Trend line of pressure graph shown at Figure 6 was slightly decrease at the centerline of 20 mm to 40 mm, which is because of the fluid travel along the large area of the pipe. Trend line of the graph started to increase at the centerline distance of 40 mm to 80 mm due to the size of the pipeline where it is getting smaller. It is stated in a research that, the pressure in a pipe system is determined by the materials and locations involved [14]. Overall, the maximum pressure for the pipe is suitable for the actual pipe which is Geometry 2. In Figure 7, it showed the pressure contour of the pipe by using Velocity 2. In the figure, there is less color of the red contour. Red contour means the pressure are too high and may affect the pipeline structure. It is very important to record the maximum pressure data as it defined the pipe's strength.

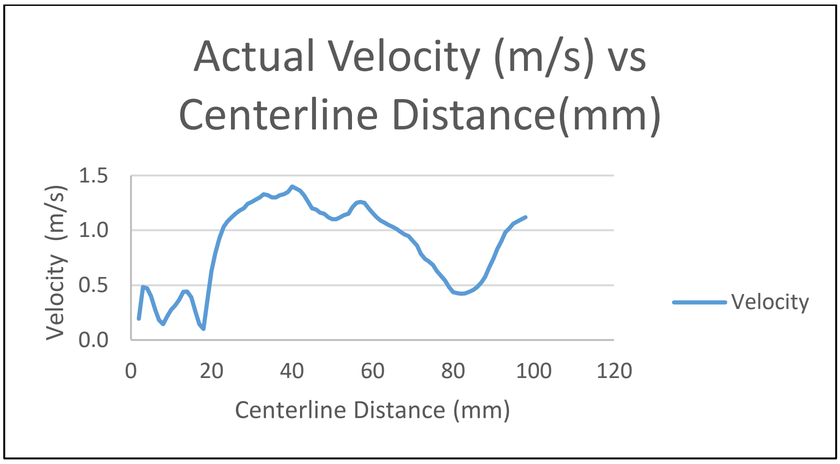

Figure 5: Graph of Actual Velocity and Centerline Distance

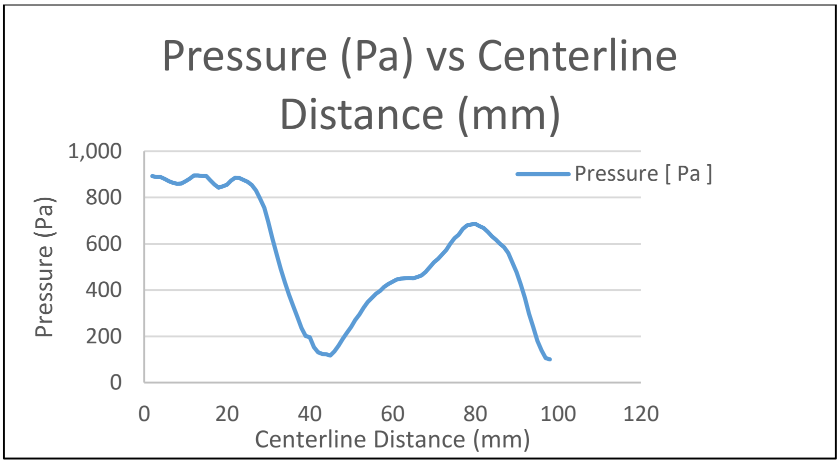

Figure 6: Graph of Pressure and Centerline Distance

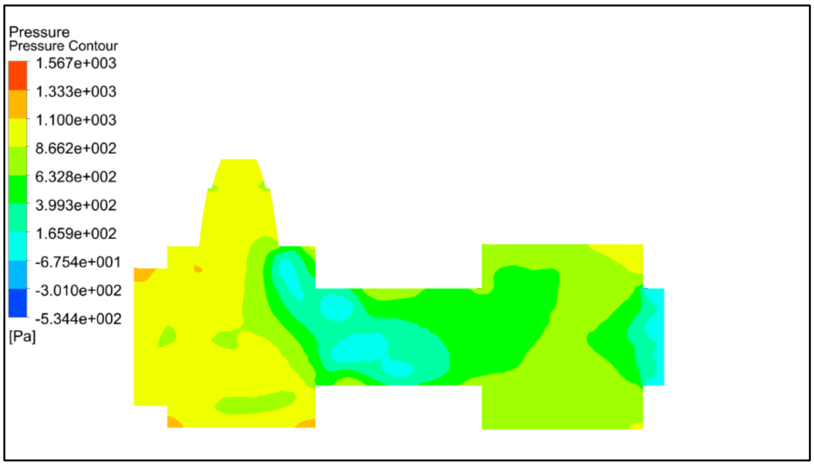

Figure 7: Pressure Contour of Geometry 2

3.2 Comparison of Different Inlet Velocity by Geometry 2 (Actual Pipe)

3.2.1 Velocity at Centerline Distribution by Using Geometry 2

Figure 8 shows the effect of the actual pipe size on the three types of velocity along the centerline of the pipe. It is obtained that the maximum velocity from Geometry 2 with an inlet of Velocity 1 is 1.13 m/s. As the input velocity increase to 1.1 m/s, the maximum velocity reached at 1.40 m/s. By increasing the value of velocity to 1.15 m/s, the maximum velocity obtained is 1.64 m/s. Based on the result achieved, the adequate inlet velocity is 0.98 m/s with a maximum velocity of 1.13 m/s. This has shown at Figure 8 where the inlet Velocity 1 was recorded as the lowest trend line graph. By using Velocity 2 in the pipeline, it also showed a good flow behavior of velocity compare to Velocity 3. From Figure 8, shown that the trend line of inlet velocity of 1.15 m/s produced the highest value of velocity compare to the other inlet velocity. High velocity was not recommended for a pipeline. High velocities in pipelines can lead to a wide variety of operational and maintenance problems, such as noise, vibration, erosion of pipe material, and a combined erosion-corrosion problem. A research approach to the analysis of the velocity profile results of the mean flow of long beam-type fluid conveying pipes [12]. Other than the velocity profile, their values depend on the circumferential wavenumber of the vibration mode structure of the pipe [13]. In a research, low-velocity and high-velocity experimental results are used to evaluate the numerical model, and a reasonable agreement has been reached between the numerical and experimental findings. When a pressure pipe is impacted by high-velocity caused by an explosion, the perforation and fracturing of the pressure pipe can result in a subsequent leaking, fire or explosion leading to an even more serious catastrophe [14].

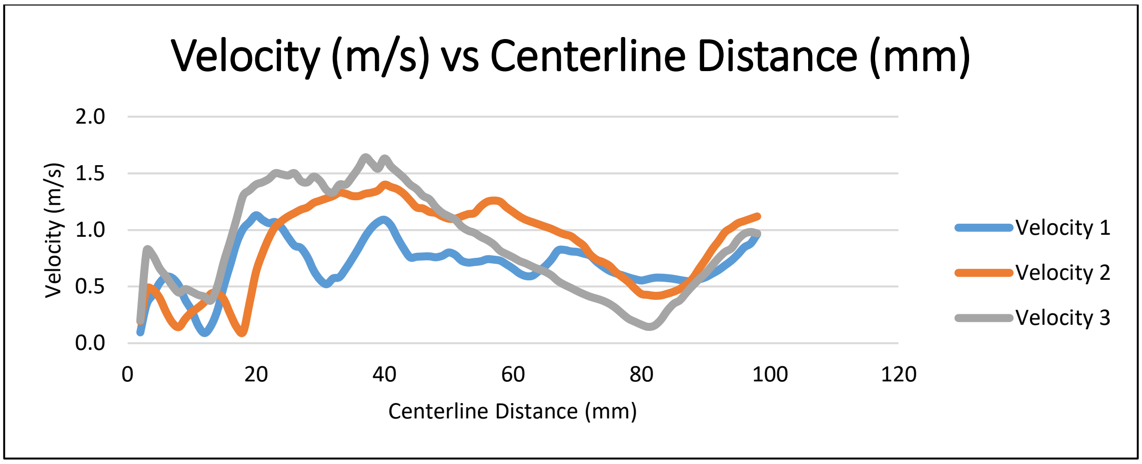

Figure 8: Graph of Comparison of Different Velocity for The Actuals Geometry

3.2.2 Pressure at Centerline Distribution of Three Different Inlet Velocity by Using Geometry 2

Figure 9 shown the graph of comparison of pressure by three different inlet velocity. Actual geometry was used for this simulation. The data was recorded and illustrated by using line graph. There are three types of inlet velocity, where Velocity 1 is 0.98 m/s, Velocity 2 is 1.1 m/s and Velocity 3 is 1.15 m/s. From Figure 9, it was recorded that the minimum pressure when Velocity 1 was set up at the inlet is 103 Pa and maximum pressure occured is 966 Pa. When Velocity 2 was set up for the geometry inlet, data recorded for the maximum pressure is 895.22 Pa and the minimum pressure is 101.08 Pa. Maximum pressure achieved at 1,320 Pa as the Velocity 3 was set up at the inlet.

The highest value of maximum pressure is 1,320 Pa when the Velocity 3 applied, and the lowest maximum pressure is 101.08 Pa with an inlet velocity of 1.1 m/s. It is shown that, actual geometry are suitable for Velocity 2 where it appears as the lowest maximum and minimum pressure. Selecting lowest maximum and minimum pressure was one of the safest way to increase the lifespan of the pipeline. Using inlet Velocity 1 for the actual pipe is also a good selection, this is because the trend line value of inlet Velocity 1 and inlet Velocity 2 was near to each other. Both of this inlet velocity achieved the lowest pressure compare to Velocity 3.

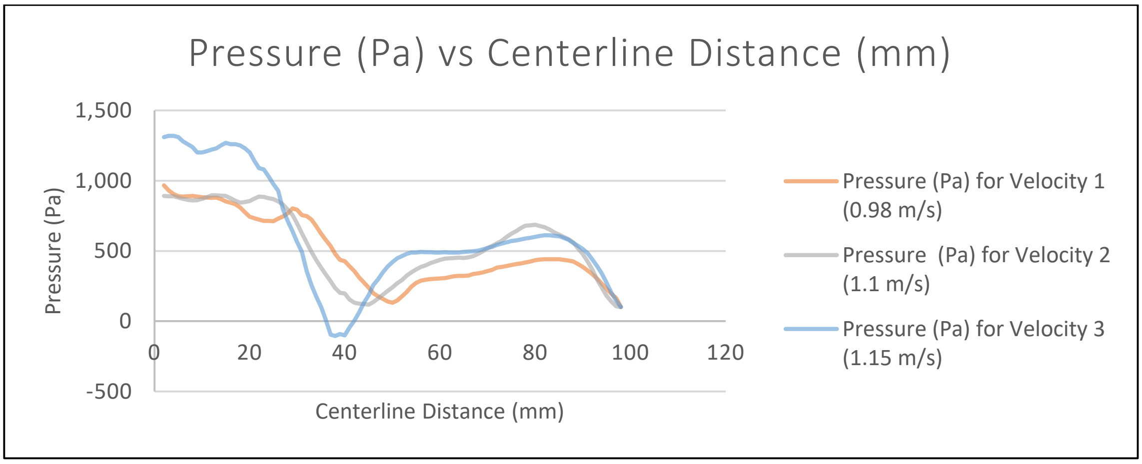

Figure 9: Graph of Comparison of Pressure for Different Inlet Velocity

3.3 Comparison of Different Geometry by Using Velocity 2 (Actual Velocity)

3.3.1 Velocity at Centerline Distribution of Three Geometry

The effect of actual velocity using different geometry was shown in Figure 10. The maximum speed achieved by the actual geometry was 1.40 m/s. For Geometry 1 which is two times smaller than the actual one, it is 1.47 m / s. It is stated from a research that smaller velocities caused permanent inelastic deformation of the pipe, but higher velocities breached their integrity immediately below or at a support [14]. The maximum velocity data obtained by the third geometry shall be 1.88 m/s. The geometry of 1 showed that it produced the highest velocity value compared to the other geometry. This is because the size of the pipeline geometry is the smallest compared to the other geometry. Geometry 3 recorded the second highest maximum velocity value and the second geometry showed the lowest maximum velocity value.

Geometry 3 produce highest velocity because of the pipeline size are not in the same shape and the distance of the inlet from the bottom of the pipe are far. This affects the flow of velocity in the pipeline where the fluid travel from large area and the velocity increase once the fluid travel to the smaller area of the pipeline. Increase of high velocity were not good for a pipeline. Study of the high-velocity impact behavior of pipes, especially those carrying dangerous liquids, is important and has significant implications [15]. The significant implications were that it is involved during high velocity impact in the pipeline is the increase of pressure in the pipe.

An impact of high velocity can cause leakage, fire, and explosion events for pressure pipes, and few papers have appeared in the open literature dealing with such problems. Therefore, numerical simulation is an adequate solution for investigating this research. In this study, peak deflections and permanent deflections are shown to increase with increased velocity [14]. Research [20] has shown that a larger pipe diameter would lead to a greater, critical transition velocity, and increased size of pipe could lead to interface instability.

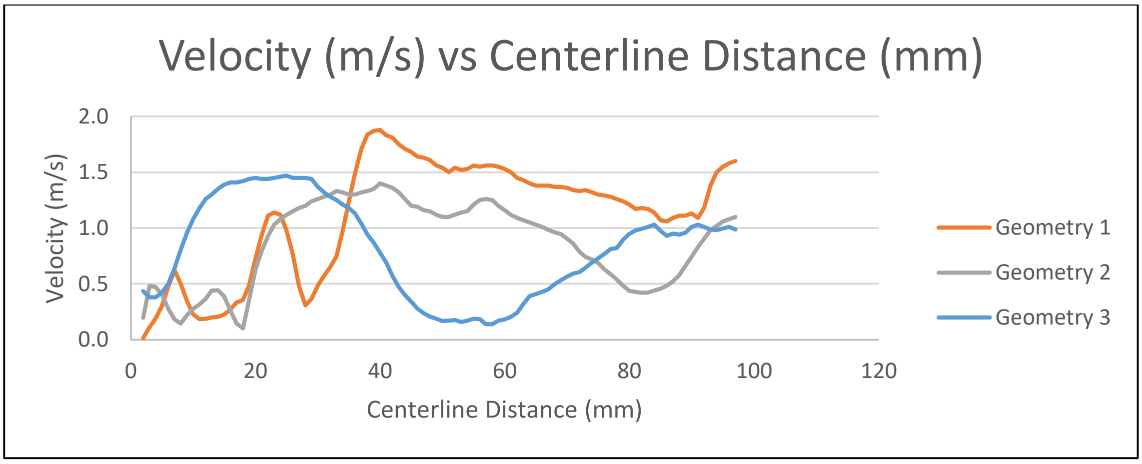

Figure 10: Graph of Comparison of Different Geometry for The Actual Velocity

3.3.2 Pressure at Centerline Distribution of Three Geometry

From Figure 11, the data of pressure for actual inlet velocity effect towards three types of geometry. It was set that diameter of inlet for Geometry 1 was 120 mm, Geometry 2 was 180 mm and Geometry 3 was 290 mm. Inlet velocity was set by using Velocity 2 for all geometry. A trend from the graph shown that the pressure decreases and form a sudden increase at the distance of range 38 mm to 45 mm this is because of the change of pipeline size. It is shown that Geometry 1 achieved the highest pressure in the graph; this is because of the pipeline size. Geometry 3 was the smallest pipeline size compare to the other geometry. It is proven that the decrease in the area will cause an increase in pressure in a pipeline. The smaller the pipeline size, the greater the value of pressure recorded. From Figure 11, it is shown that Geometry 3 was the lowest pressure due to the size of the pipeline. Geometry 3 was the largest pipeline size compare to the other geometry where it affects the pressure value. The increase of area in the Geometry 3 causes the pressure to decrease. It is suitable to use Velocity 2 for Geometry 2 where the value of pressure is not too high and not too low. The trend line of Geometry 2 was at the middle of Geometry 1 and 3 where it means it is safe to use Velocity 2 for this geometry. This is because the maximum pressure for Geometry 2 are lower than Geometry 1.

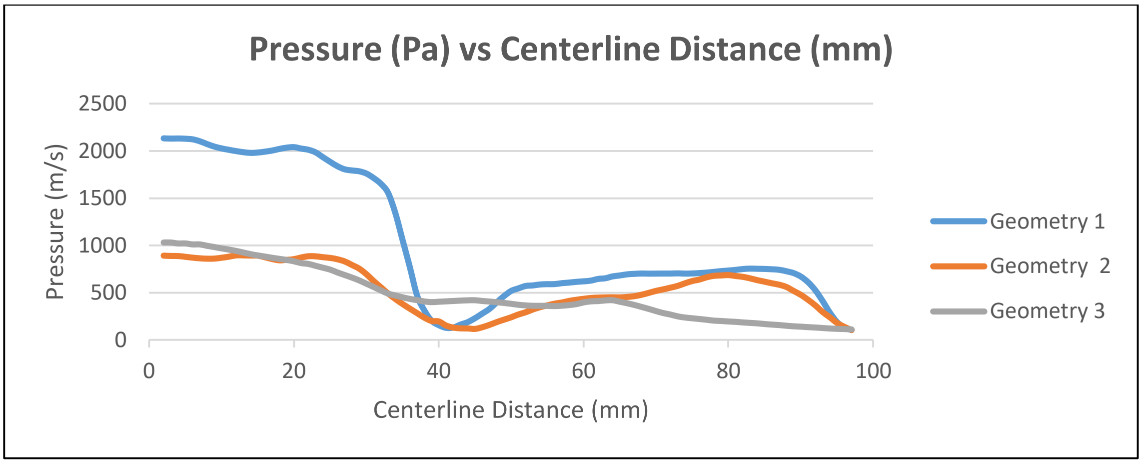

Figure 11: The Pressure of Actual Inlet Velocity in Three Different Geometry

3.4 Comparison of Three types of Geometry and Three Different Velocities

3.4.1 Pressure Contour for Geometry and Three Inlet Velocities

Figure 12 shown a pressure contour from three inlet velocity by using Geometry 1. By using Velocity 1 showed a good pipeline pressure contour where there was no red contour occur in the pipeline. As the Velocity 2 and Velocity 3 apply, it shown that the maximum pressure occurs at a place as shown in Figure 12. Pressure contour of Velocity 2 shown that the red contour of the maximum pressure was caused by the fluid entering from the top and hitting the bottom of the pipe structure directly. In pressure contour of Velocity 3 it is caused by the same result of Velocity 2 but the inlet velocity was increased. Due to the increased of velocity, the pressure increased and the red contour area are larger compare to the other geometry.

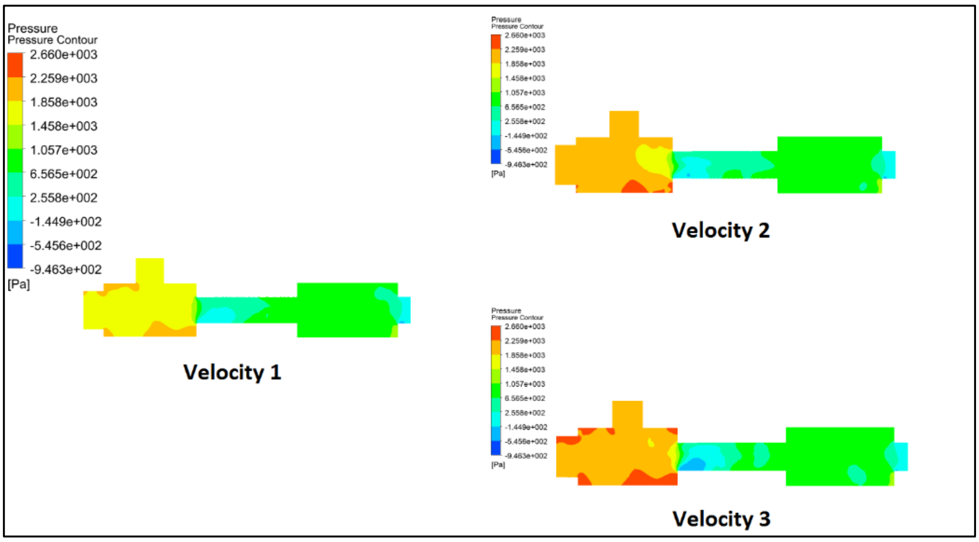

Figure 12: Pressure Contour of Geometry 1 by Using Three Velocity

Figure 13 show pressure contour from three inlet velocity by using Geometry 2. By using Velocity 1 and 2 there was no red contour occur in the pipeline, which mean it is suitable to use Velocity 1 and 2 for Geometry 2. Velocity 3 recorded the highest maximum pressure located as shown in Figure 13. It is not appropriate to use a pipeline with a lot of red contour effect in the pressure contour. Which means an increase in pressure in the pipeline may cause high stress value on the pipeline. It will affect the pipeline structure and it lifespan.

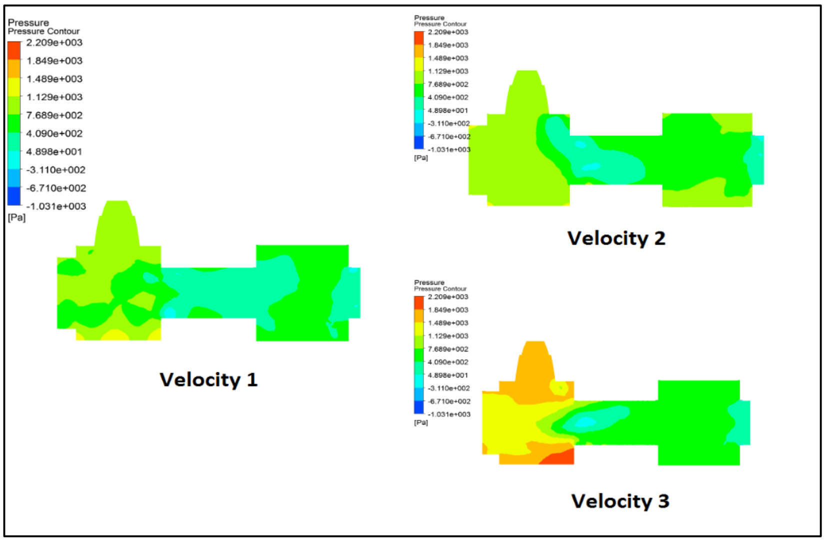

Figure 13: Pressure Contour of Geometry 2 by Using Three Velocity

Pressure contour of Geometry 3 by using three different inlet velocity was shown in Figure 14. Velocity 1 recorded the lowest maximum pressure compare to the other geometry. As the Velocity 2 and Velocity 3 are applied, the peak pressure was shown to occur at a position as shown in Figure 14. Velocity 2 and 3 pressure contours revealed that the red contour of the maximum pressure was caused by the fluid flowing from the top and directly impacting the bottom of the pipe structure.

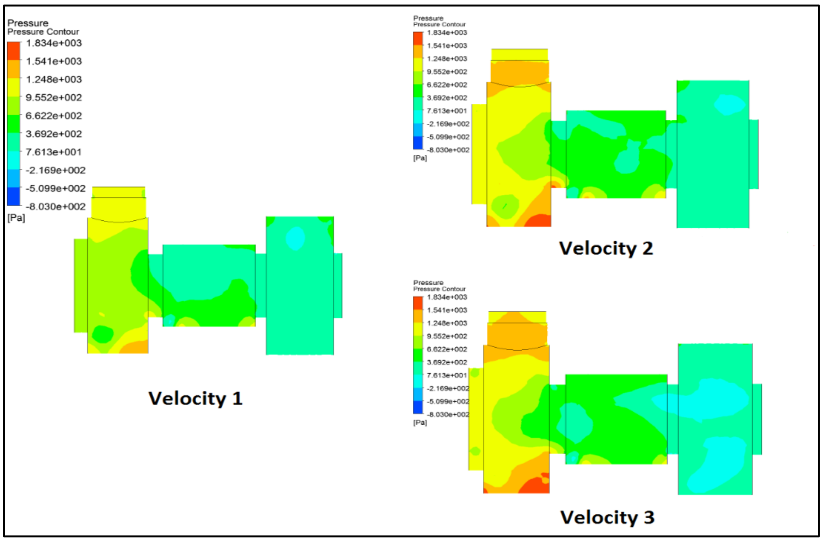

Figure 14: Pressure Contour of Geometry 3 by Using Three Velocity

3.4.1 Comparison of Maximum Pressure of Three Geometry and Velocities

Maximum pressure of three geometry with three different inlet velocity was illustrated in Figure 15. It was stated that Geometry 1 with 120 mm, Geometry 2 with 180 mm, Geometry 3 with 290 mm and Velocity 1 with 0.98 m/s, Velocity 2 with 1.1 m/s and Velocity 3 with 1.15 m/s. From the data obtained, the lowest maximum pressure is at the Geometry 2 with a velocity of 0.98 m/s, where it achieved 1376 Pa. The highest maximum pressure was obtained by Geometry 1 with a velocity of 0.98 m/s. From the Figure 15, it was shown that maximum pressure of Velocity 1 was decrease but then the value was increase once it reached Geometry 3. Same goes to the trend line of maximum pressure of Velocity 2. From Figure 14, it is clearly that the maximum pressure was located at the location where the fluids are exerting pressure on the pipe. The pressure in fluid change with the depth of the surface. The deeper the fluid travels from the inlet, the higher the fluid weight and the liquid pressure will increase. The pressure exerted by the fluid is directly proportional to the height, density of the fluid and its cross-section of the pipe. As the trend line for maximum pressure of Velocity 3 was decreased it is proved that an increase of velocity could lead to decrease of pressure on the pipeline.

For Geometry 1 and Geometry 2 it is suitable to use Velocity 1 as it showed a good flow behavior. It also recorded the lowest maximum pressure in the pipeline. Velocity 1 also suitable for Geometry 3 to prevent high pressure on the pipe that may lead to the higher stress on the structure of the pipeline. Instead of inlet Velocity 1, Geometry 3 also showed a good reading for inlet Velocity 2. This is showed in the trend line of the graph in Figure 15 where the maximum pressure was recorded as the second lowest. It showed that the highest maximum pressure for Geometry 2 was by using inlet velocity of 1.15 m/s. The maximum pressure increases as the inlet velocity increase. Increase of pressure were not good for a pipeline. In a mid-span impact study, it proves that as internal pressure increases, there is a greater chance of pipeline failure will occur and it appears that the local displacements tend to decrease as the pressure is increased [16]. It is clear that the change of geometry size affects the value of velocity and pressure depend on its velocity of inlet.

In other words, as pressure increases more impact energy is likely to be absorbed in the overall deformation mode at the expense of local energy absorption of the pipe. In a study [17] velocity is an important parameter for examining the difference between simulated and experimental results. Release of velocity results indicated that the experimental method led to a greater data uncertainty than the numerical simulation method, and the instability of the working pressure could be the main source of error [18]. As the pressure increases, there will be stress on the pipe and this will lead to the leakage of the pipe and affect the life span of the pipe. When the stream of fluid undergoes a sudden expansion, it will cause a sudden decrease in velocity and increase the pressure [20].

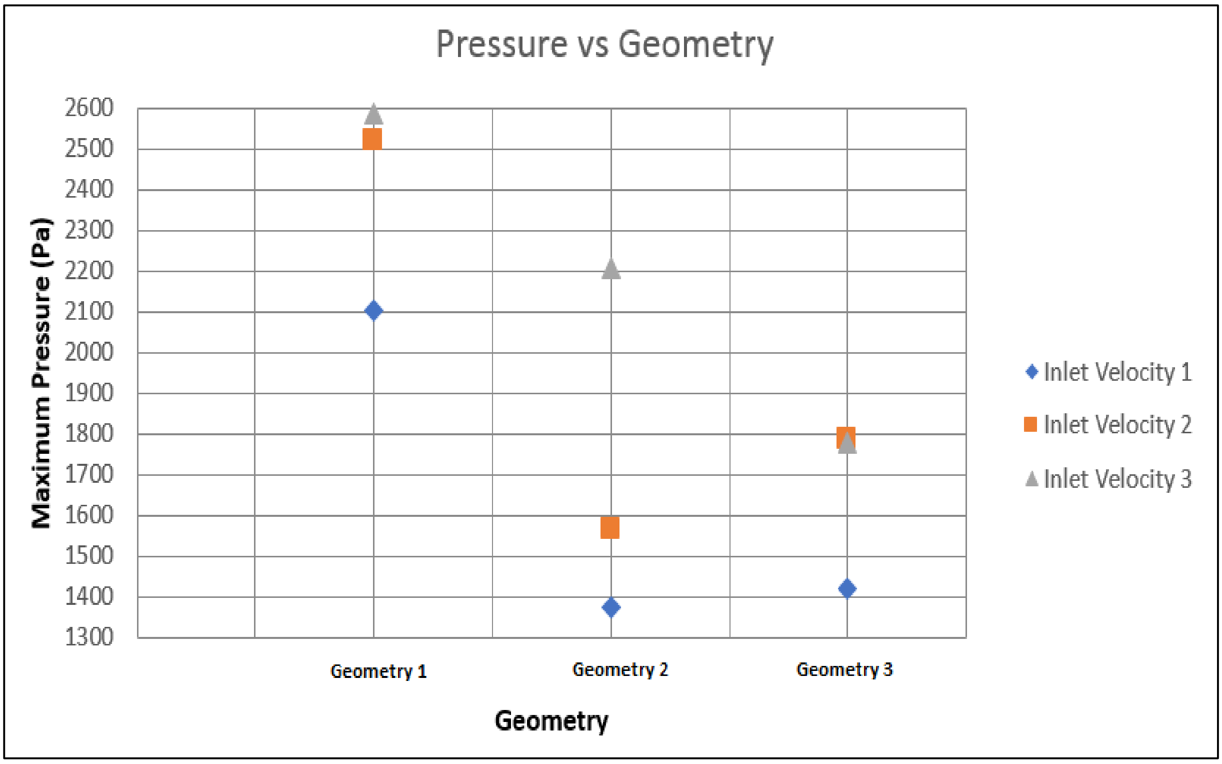

Figure 15: Graph of Comparison of Maximum Pressure, Three Velocity and Three Geometry

4. Conclusion

The CFD method by using ANSYS Fluent analysis has been applied in the simulation of a research of different pipe diameters to the flow behavior of the pipeline. The simulation procedure was performed with different pipe diameters and inlet velocity. Pressure limitation has no effect on the results of the comparison, as all measurements and CFD Ansys Fluent analysis for all lines were carried out under the same outlet pressure of 101 Pa. Geometry 1 achieved a good flow behavior when Velocity 1 was applied. It is recommended to select Velocity 2 for Geometry 2. Velocity 3 is suitable for Geometry 3 due to the size of the pipeline. Overall, a good flow behavior in the pipeline will improve the efficiency and lifespan of the pipeline. Further research for effort is also recommended in particular for expanding the test for higher pressure range and studying the effect of line pressure on the performance of higher input velocity. Further research also should consider in investigating the effect of inlet velocity for bigger size of pipeline by increasing the value of velocity.

Acknowledgement

This research was made possible with the guidance from fellow lecturers and also the support from the Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia.

References

[1] Y. Di, J. Shuai, X.L Wang and L. Shi (2013), Study On Methods For Classifying Oil And Gas Pipelines Accidents, China Safety Science Journal, Vol 23.

[2] T. S. Semsamran, (2018). The Impact of Biodiesel B100 On A Small Agricultural Diesel Engine. Tribology International, 10.

[3] H. Z. Daming Huang, (2012). Biodiesel: An Alternative to Conventional Fuel . Energy Procedia, 12

[4] F. O. Licht(2011), The Global Renewable Fuels Alliance Is A Non-Profit Organisation Dedicated to Promoting Biofuel Friendly Policies Internationally, Global Renewable Fuels Alliance.

[5] S. K. Vinay Sati (March-2015). Hydrodynamic and Thermal Analysis in Pipe. International Journal of Engineering Research & Technology (Ijert), 1.

[6] A. A. Mohamed Elashmawy (2016). Investigation of The Effect of Pipeline Size on The. Engineering, Technology & Applied Science Research, 6.

[7] F. W. Ntengwe, M. C. (October 2015). Evaluation of Friction Losses in Pipes and Fittings of Process Engineering Plants. International Journal of Scientific & Technology Research, 7.

[8] D. S. Prabir Chakraborty, (2016). A Study On Laminar Pipe Flow & Losses. International Journal of Innovations in Engineering and Technology (Ijiet), 7.

[9] Bahrudin, H. S. Alam And T. Haiyunnisa, (2016) Computational Fluid Dynamic Simulation of Pipeline Irrigation System Based on Ansys, International Journal of Technology And Engineering Studies, Vol. 2, No. 6, Pp. 189-193, 2016.

[10] Sökmen, H. A. (2013). CFD Modeling of Thermal Mixing In T-Junction: Effect. The 15th International Topical Meeting On Nuclear Reactor Thermal – Hydraulics

[11] Lee, E. A. (2013). CFD Simulation of a Single-Phase Flow in A Pipe Separator. ARPN Journal of Engineering and Applied Sciences, 7.

[12] C.Q. Guo, C.H Zhang, M.P Païdoussis (2010). Modification of Equation of Motion of Fluid-Conveying Pipe for Laminar and Turbulent Flow Profiles. Journal of Fluids and Structures 26, 793-803.

[13] J. K. Bajsić, (2014). Fluid-Dynamic Loading of Pipes Conveying Fluid with A Laminar Mean-Flow Velocity Profile. Journal of Fluids and Structures, 25.

[14] Zhen-Yu Wang, Y. Z.-W.-G. (2016). A Numerical Study on The High-Velocity Impact Behavior of Pressure Pipes. Journal of Zhejiang University-Science A (Applied Physics & Engineering), 11.

[15] Y Zhao, Z.Y. Wang, Z.G He, 2014. Numerical Study of Cfrp-Bonded Pressure Pipes Subject to Impact Load. Applied Mechanics and Materials, 602-605:432-437.

[16] Norman Jones, R.S. Birch. Low-Velocity Impact of Pressurised Pipelines. International Journal of Impact Engineering, Elsevier, 2009, 37.

[17] Yisheng Zhang 1, B. S. (2018). Experimental and Simulation Investigation On The. Molecular Diversity Preservation International Journal , 13.

[18] G.S. Burillo, R.Delirhasannia, E. Playán, P. Paniagua, B. Latorre, and J. Burguete (2013), Initial Drop Velocity in A Fixed Spray Plate Sprinkler, International Journal of Technology and Engineering Studies, 18.

[19] W. Z. Jieheng Zheng (2017). Cfd Simulation and Experimental Study of Water-Oil Displacement Flow in An Inclined Pipe. International Journal of Heat and Technology, 5.

[20] N. D. Saffari, (2014). Influence of Variation of Pipe Diameter on Pressure Drop Predictions Of The New Modified Three-Fluid Model Inside Condensing Vertical Pipes. Journal of Process Mechanical Engineering.