SEMARAK ILMU PUBLISHING
202103268166(003316878-P)

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Computational Fluid Dynamics: Flow Analysis on the Effect of Different Jet Orifice Angle Multi Circular Jet for Fuel and Air Mixing

Ridzwan Kamaruddin[1], Syabillah Sulaiman[1,*], Amir Khalid[2], Mohammad Arafat[1], Norrizam Jaat[1], Shaiful Fadzil Zainal Abidin[1], Norirda Mohamed[1], Mohd Fuad Yasak[1]

[1] Department of Mechanical Engineering Technology, Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Malaysia
[2] Department of Mechanical Engineering, Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia, Malaysia

ARTICLE INFO

Article history:
Received 29 May 2023
Received in revised form 25 June 2023
Accepted 21 July 2023
Available online 5 December 2023

Keywords:
Multiphase flow; internal flow; fuel mixing; turbulence generator

ABSTRACT

affecting the combustion performance and emissions of the internal mixing air-assisted atomizers. Air-assisted atomizers are introduced to counter the low-pressure differential of a simplex nozzle, which reduces the atomization quality. The present study aims to determine the effects of Multi Circular Jet (MCJ) plates on the geometrical configurations of internal flows in mixing chamber and the internal flow of plate 3 using different properties of fuel. In this study, the realizable k-ε turbulence model, specifically designed for strongly swirling flows, is validated through numerical simulations. The turbulence model selected is a type of Reynolds averaged Navier-Stokes (RANS) model called the k-ε model. The MCJ plates provide the primary air entrance into the mixing chamber. Additionally, it acts as a turbulence generator and can be adjusted to alter the flow of fuel and air mixtures in a mixing chamber. The study compares several MCJ geometries in terms of pressure, speed, turbulent kinetic energy, and volume fraction and compares the performances of diesel and Crude Palm Oil (CPO) B30 biodiesel fuels. The findings imply that CPO B30 biodiesel has superior atomization and mixing due to its higher density and turbulent kinetic energy. CPO B30 biodiesel was compared to Diesel in terms of maximum pressure, average speed, turbulent kinetic energy per unit mass, and volume fraction. The results indicate that CPO B30 has lower pressure and higher velocity than Diesel, suggesting better fuel atomization and mixing. The higher density of CPO B30 leads to increased turbulent kinetic energy, improving fuel-air mixing inside the combustion chamber. The study demonstrates that the use of MCJ plates can enhance mixing in a mixing chamber. In addition, MCJ plates show the ability to control the spray and atomization. The findings of this study contribute to a better understanding of the relationships between geometry and fuel-air mixing, as well as the characteristics of the internal mixing air-assisted atomizer, which will lead to future burner system improvements.

1. Introduction

Premix injectors are a sort of indirect injection in which the fuel is pumped into a sub-chamber to be swirled with compressed air before being sprayed into the combustion chamber area [1]. Many

* *Corresponding author.*
E-mail address: syabillah@uthm.edu.my (Syabillah Sulaiman)

https://doi.org/10.37934/cfdl.16.1.121137

studies have suggested factors that influence the combustion and emissions of the burner. The physical and chemical properties of the fuel [2], spray characteristics [3], and fuel-air mixing [4, 5], were discovered to have a significant impact on the performance of the burner.

The size, geometry, and physical properties of the dispersed phase (liquid) and continuous phase (pressurised air) determine the performance of an air-assisted atomizer. The air-assisted atomizer is designed to improve fuel-air mixing prior to spraying. It has advantages due to its excellent control for a variety of applications and has been extensively explored by many researchers [6-8]. It also has advantages over external mixing of air-assisted atomizers, such as good spray quality at low pressure, easy-to-control spray performance, and low primary air consumption. The atomizer nozzle geometry plays an important role in the characteristics of spray atomization and the formation of fuel-air mixtures [9], which contribute to enhancing the combustion and reducing the emissions [10, 11].

In the work of Rasha Abdulrazzak Jasim., *et al.,* [12] the volume of fluid model (VOF), coupled with the turbulent model, has been applied. The two-dimensional Reynolds-averaged Navier-Stokes equations are wrongly solved by the second-order upwind method. The simple method developed using control volumes is used to apply the numerical technique. Calculations were made for a wide variety of Reynolds numbers (Re), which correlate to different flow patterns. The findings show that the impact of the gas phases on the liquid phases grows as the Reynolds number rises. While the height of the recirculation zone rises downstream from the obstruction, the boundary layer thickness downstream from the obstruction decreases with increasing Re. In the pressure field, this article found an inversion of the pressure gradient, which caused the boundary layer to separate; this is known as the "back flow phenomenon".

Two-phase flow refers to the interactive flow of two distinct phases with common interfaces in a channel, with each phase representing a mass or volume of matter. The two phases can exist as combinations of solid, gas and/or liquid phases. In fluid mechanics, two-phase flow is a flow of gas and liquid, which is a particular example of multiphase flow. The simulation of isobutene flow via adiabatic capillary tubes by Yonghui Shu., *et al.,* [13] included a comparison of the separated flow model with the homogeneous flow model. Due to the larger vapor phase mass percentage and higher vapour velocity in the separated flow model compared to the homogeneous flow model, which caused the separated flow model to forecast a lower mass flow rate, the separated flow model offered more dramatic pressure drop and temperature decrease.

Previous studies have simulated the turbulence flow of a two-phase system using Reynolds averaged Navier-Stokes (RANS), which is beneficial for turbulence mixing process simulation [14, 15]. The results of the Renormalization Group (RNG) k-model and the conventional k-model were compared to published Particle Image Velocimetry (PIV) data. In addition, the constants in the transport equations were changed to show how the flow was moving on average over time. It was discovered through a multi-dimensional analysis that the RNG k-model performed worse than the conventional k-model in predicting the liquid-liquid two-phase mixing process. This insight can be valuable in selecting the suitable turbulence model [16].

To optimize swirl intensities, it is necessary to consider several in-cylinder variables such as velocity, pressure, temperature, and turbulence intensity. The intake system can be altered, or the valve design can be changed, to produce a swirl. Manifold designs with bend angles of 15°, 30°, 45°, 60°, and 75° were employed, and Ansys Fluent was used for all aspects of the numerical analysis. The study's findings showed that 75° was the best-optimized design (in terms of turbulent kinetic energy) for achieve superior swirl [16]. Furthermore, this research provides insight into how to develop novel methods for enhancing mixing by increasing turbulent kinetic energy [16, 17].

In 2014, another study conducted by Farid Sies. M., *et al.,* [18] introduced the same fuel-water emulsion premix injector into the application of open burner. The study used a mixture of crude palm oil (CPO) biodiesel and regular diesel fuel, along with standard diesel fuel and diesel fuel mixed with water as the working fluids. The spray formations were discovered to be dependent on the equivalent ratio or mass flow rate, with CPO biodiesel having longer penetration duration and spray area than diesel but a smaller spray angle. Furthermore, they revealed that high-density fuel with additional water content would result in longer spray penetration and a small spray angle. It is also to improve the optimization process of vegetable fuels in the combustion chamber of the engine, due to the turbulence causes by the atomizer. Density and viscosity are depending on the temperature which is increase in temperature under the influences of additional vortices, changes the physical parameters of the fuel that will affect the spraying process [18]. The spray penetration will increase as the water content in the fuel increases, while the spray area will decrease [19].

This theoretical analysis focuses on modifying fuel injector components, specifically using spiral elliptical ducts in the atomizer, to enhance the injection parameters and combustion process of a modern compression ignition engine powered by biofuel. The combustion processes of compression ignition engine and micro gas turbine share some similarities, which make them a useful reference for each other. The objective of the injector modification is to improve the fuel injector process.

The study by Amirnordin Shahrin *et al.,* [20] compares the spray properties, swirling motion, and flame formation for multi-circular (MCJ) jet and swirl plate using two plates. When compared to MCJ, swirl plates create entirely distinct properties. The additional ability of MCJ plates to regulate spray and atomization can affect the combustion and emission of the internal mixing air-assisted atomizer.

The flow recirculation created by the MCJ plates within the mixing chamber increases the liquid fuel and air interaction by having a longer retention time, which influences spray combustion. As a result, combustion can take place under any operating state. The use of MCJ plates in the atomizer allows for more precise control of spray combustion properties. This is critical in applications of the atomizer in the burner system that can fulfil industry standards. The present study aims to determine the effects of Multi Circular Jet (MCJ) plates on the geometrical configurations of internal flows in mixing chamber. The atomizer unit consists of components such as the atomizer plate with air holes, mixing chamber, nozzle orifice, air ducts, and fuel. Three different jet hole angle geometries are considered in this study. The function of this part is to mix the fuel and air before it is sprayed from the nozzle. Ansys Fluent software was used to simulate and determine the results in terms of pressure, average velocity distribution, turbulence kinetic energy and volume fraction.

2. Methodology

Design of Multi-Circular Jet (MCJ) is referred to the test rig performed by Yasufusa and Kidouchi [4]. The injector uses swirl to generate turbulence. It is located at the center of the chamber as in Figure 1. The input or fuels are supplied through the bottom of section of the injector.

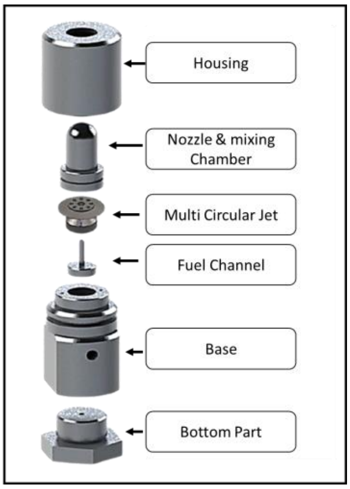
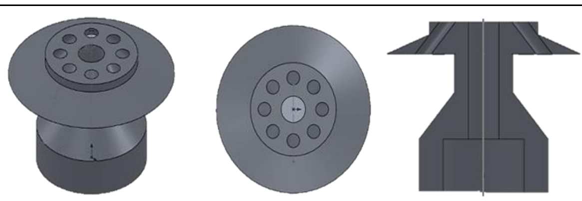


Fig. 1. Turbulence generator configuration

This injector is constructed with air and fuel inlet, mixing chamber, nozzle orifice air ducts. There are 3 geometries with different inlet angle are considered in this study. The specifications of each MCJ geometries are included in Table 1. MCJ plays the role of turbulence generator for the mixing of fuel and air before injected into the combustion chamber. Spray characteristics in the combustion chamber were influenced by the geometrical and physical structure of this turbulence generator. Injector plates deliver the fuel and air into the mixing chamber for the mixing process before discharge the mixture into the combustion chamber.

Table 1
MCJ geometries and specifications

Specifications		Plate 1
Diameter, D_1	2 mm x 4 holes	 **P1**
Hole angle, ϑ	0°	
Open area, mm^2	26.70	
Open area ratio, % Ae	18.94	
Specifications		**Plate 2**
Diameter, D_1	2 mm x 8 holes	 **P2**
Hole angle, ϑ	30°	
Open area, mm^2	25.13	
Open area ratio, % Ae	17.8	
Specifications		**Plate 3**
Diameter, D_1	2 mm x 8 holes	 **P3**
Hole angle, ϑ	45°	
Open area, mm^2	25.13	
Open area ratio, % Ae	17.8	

2.1 Turbulence Model

Turbulence plays a crucial role in computational fluid dynamics (CFD), characterized by its random and non-constant nature, making it challenging to predict using theoretical approaches. To account for turbulence, it becomes necessary to make assumptions about the transport process and establish connections between different physical quantities within the time-averaged flow field [21]. Therefore, turbulence models are introduced.

Turbulence parameters refer to the characteristics of turbulent flow, which is fluid motion characterized by chaotic changes in pressure and flow velocity. Turbulent kinetic energy (TKE) is another parameter used to describe turbulence, which is the energy associated with the random motion of fluid particles in a turbulent flow. The choice of turbulence parameters can affect the accuracy and efficiency of the simulation, and different parameters may be more appropriate for different types of flows. When a fluid is moving in relation to a surface, the Reynolds number may be defined for a variety of different scenarios. These definitions typically incorporate the fluid's density and viscosity, as well as a velocity and a characteristic length or characteristic dimension. The Reynolds number is an important parameter in all viscous flows where a numerical model is selected according to pre-calculated Reynolds number.

In this study, the realizable k-ε turbulence model, specifically designed for strongly swirling flows, is validated through numerical simulations. The turbulence model selected is a type of Reynolds averaged Navier-Stokes (RANS) model called the k-ε model [22, 23]. This model is widely used and preferred for numerical simulations in technical applications due to its numerical stability, robustness, and popularity.

2.2 Simulation Setup

Ansys Fluent software was used for the flow analysis process that involves injecting fuel and air into a chamber. Diesel and CPO biodiesel fuels are injected into the chamber to simulate the actual fluid flow inside the mixing chamber before they are released into the replica micro gas turbine combustion chamber using the multi circular jet modelling from SolidWorks.

The process of meshing, which involves dividing complicated geometries into elements that may be utilized to discretize a domain, is a critical one in the simulation process. The accuracy, convergence, and speed of the simulation are all influenced by the mesh quality, and several software packages provide different meshing procedures. Unstructured meshing is used for complicated geometries where it would be impossible to build a structured mesh. Structured meshing is utilized for these simulation geometries. Structured meshes are also easier to generate, as the grid cells have a regular and structured arrangement that makes them easier to map by simple indices.

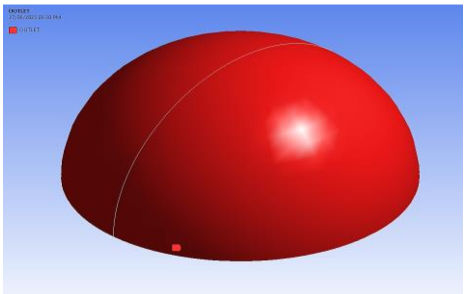
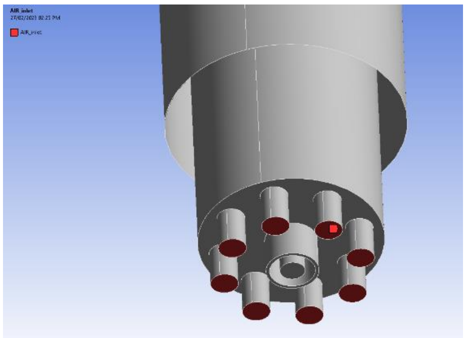
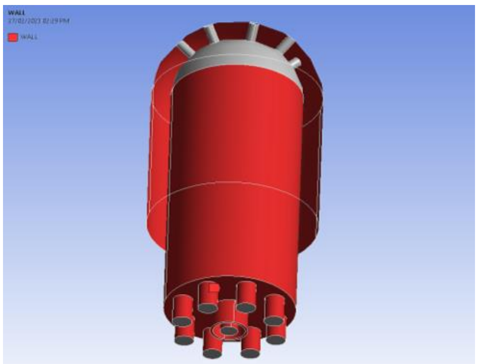
A more exact simulation is achieved by using cutcell meshes, which can compute precise intersections with the geometry and delete the piece of the mesh that is outside the geometry. Create a body-fitted mesh with a mostly hexahedral cell structure using the meshing process known as cutcell meshing. The simulation incorporates cutcell meshing, which can result in a body-fitted mesh with a mostly hexahedral cell structure. Better results, a body-fitted mesh, a primarily hexahedral mesh, and quicker mesh production are all benefits of cutcell meshing.

In general, selecting the appropriate inlet conditions for a multiphase flow simulation requires a good understanding of the physics of the system being modeled and the goals of the simulation. Objectives of these models is to act as turbulence generators which produce a mixing flow of two different phase. The air inlets selection is based on the outer inlet which will produce turbulence to helps in the mixing with fuel. The fuel flow in the middle inlets provides the collision of molecules from the air and fuel for mixing and turbulence to occur.

Import this into a computing program after that to solve equations iteratively and determine the values of the parameters of each mesh node. The segregated solution method was used to solve the governing equations and the turbulence model. The velocity-pressure linked to a multiphase model (VOF) was simulated using the SIMPLE (Semi Implicit Method for Pressure Linked Equations) method, while the convective and diffusive elements were modelled using a second-order upwind technique.

When simulating the fluid and air flow in an atomizer, boundary conditions need to be specified to determine how they enter and exit the system. These conditions include variables such as temperature, intake pressure, and fluid flow velocity, and it is important to consider additional boundaries such as the smoothness of the mixing chamber's walls and the heat transfer requirements of the injector's walls. In this study, 3-Dimensional model (3D) defines the boundary conditions for the Ansys Fluent software's inlet and outlet channels. Table 2 displays the boundary conditions that were employed in this simulation

Table 2
Summary of boundary conditions

Description	Figure
OUTLET The outlet is replicated as a semi sphere which is parallel to the surface nozzle and the outlets pressure is set to atmospheric pressure which is 1 bar. The flow from mixing chamber will flow into the combustion chamber through the nozzle orifice.	
INLETS The velocity of the fluid or mass flow rate can be determined by the inlet selection in form of mass flow rate. In this case consists of two different inlets with different flow rate.	
WALL Solid wall is the basic boundary condition that exists in the domain of fluid flow. The wall injector, nozzle, and spray chamber are no wall motion, and it remains stationary. Aside from that the fluid flow is non-slip due to the shear condition of the wall.	

For this simulation study, three different MCJ plates were tested with three different fuel mass flow rates. The boundary conditions for three simulations were kept constant throughout the procedure. Table 3 shows the mass flow rates of intake diesel fuel and air used in the simulation. The air flow for the simulation is incompressible as the density is constant throughout the simulation, where the velocity of the air is at low speed.

Table 3
Mass flow rate of inlet

Inlet	Density (kg/m^3)	Mass Flow Rate (kg/m)
Diesel	833.7	2.06×10^{-3}
Crude palm oil	839.5	1.37×10^{-3}
Air	1.255	0.0393

2.3 Grid Independence Study

In numerical simulations, a grid independence study is often conducted to ensure that the results obtained are not influenced by the grid resolution used. It is essential in CFD simulations to minimize the discretization errors and ensure accurate results. Some of the literature has reported that grid size has a significant effect on the convergence and predicted results [24, 25]. In this study, a grid independence study was performed to ensure that the grid was sufficiently refined with respect to different sizes. Grid sensitivity was tested on the model with different grid sizes such as coarse, medium, and fine mesh.

For comparison purposes, the graph of velocity produced along the mixing chamber by three different mesh levels is shown in Figure 2. 8 (eight) planes were drawn inside the chamber so that velocity data could be extracted from all the planes and graphed. As can be seen in Figure 2, the values of the velocity variable and graph pattern were not significantly changed when the grid size increased from coarse to fine.

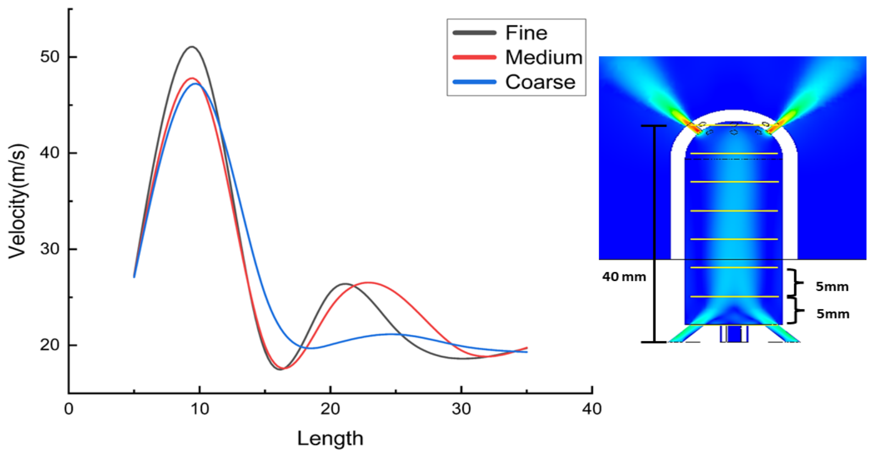


Fig. 2. Velocity Profile at different mesh levels

The sudden increase in velocity at the area of 5- 10 mm from the fuel inlet is believed to be where the rapid fuel-air mixing occurs. After 10 mm, the velocity started to decrease along the mixing chamber. It is because of the energy loss brought on by the quickly forming mixture of diesel fuel and colliding air. The best velocity graph fit among the three mesh levels is used to choose the optimum grid. Because it lies between the coarse and fine grid levels, the medium mesh level is chosen to represent the mesh element of 4112970. It is decided to use medium mesh resolution for the rest of the simulations.

3. Results and Discussion

This section discusses the computational work involved in the analysis of the internal mixing air-assisted atomizer. This is to meet the objective of getting information about the influences of changes in geometries on the internal flow of the mixing chamber. Another factor to be considered is the influence of MCJ plate geometry on primary air entering the mixing chamber and the mixing process between the primary air and liquid fuel. Two factors need to be considered: the effects of the open area ratio and the jet-hole angle. As a result, the mixing chamber, as shown in Figure 3, is divided into eight planes with equal distances from the fuel entrance hole. Data such as velocity, turbulent kinetic energy, and volume percentage are collected and analyzed in each plane.

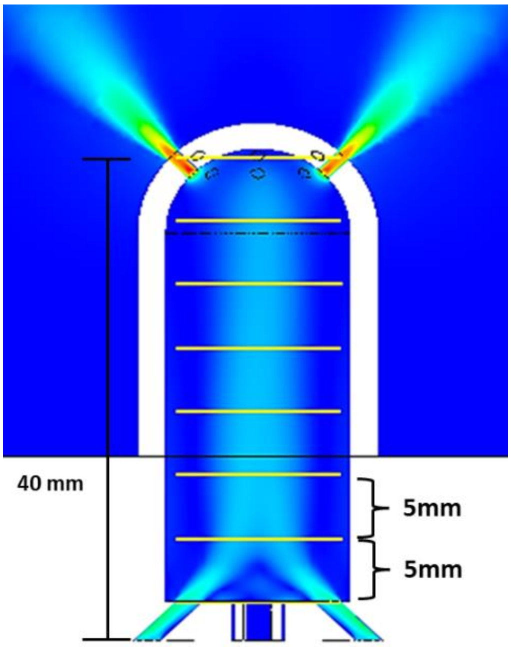


Fig. 3. Mixing chamber plane division

3.1 Effects of Different Jet-hole Angle on the Flow Behavior

The simulation analysis for given the fuel used is in stoichiometric condition. The parameters in the flow analysis represent the physical conditions that occur in the mixing chamber before it is sprayed out. A velocity streamline is a line that connects the velocity vectors and shows the direction of fluid flow. In this study, it represents the flow of fuel and air mixture from the fuel inlet to the nozzle along the mixing chamber. Fuel and air enter the chamber through two different inlets and travel vertically through the chamber before exiting through the nozzle at the top. In the chamber, parameters such as velocity and pressure were calculated numerically.

3.1.1 Velocity streamline and vector

Figure 4 shows the velocity streamline and vector of the different MCJ geometries. P1 represents the flow of fuel and air mixtures from the inlets, which travels vertically straight up to the nozzle of the chamber shown in Figure 4. The mixing process at the bottom of the chamber takes place slower as the fuel enters the chamber through fuel inlets and does not directly meet the air flow into the chamber. Vertical flows of air into the chamber do not produce enough tangential velocity components to produce turbulence that promotes mixing.

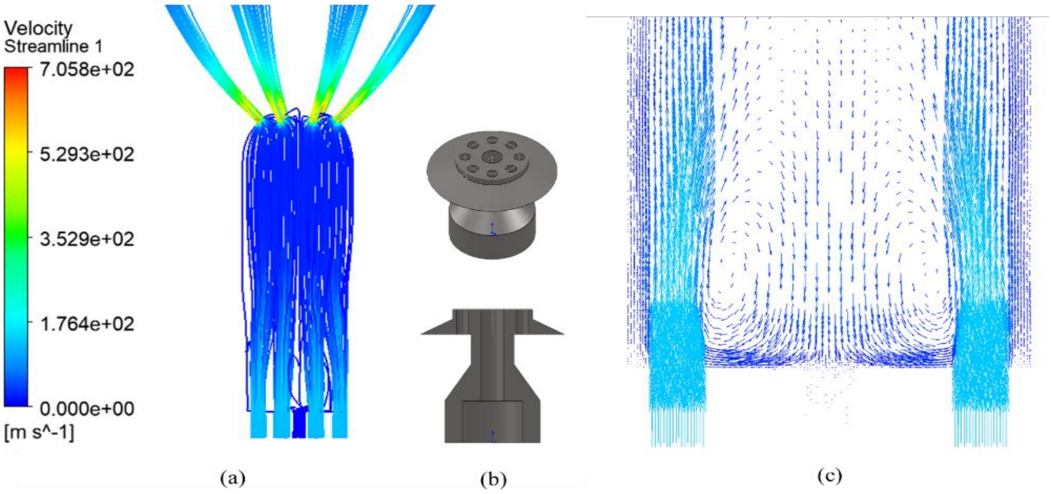


Fig. 4. Flow characteristics for the P1; (a) velocity streamline (b) plate geometry (c) velocity vector

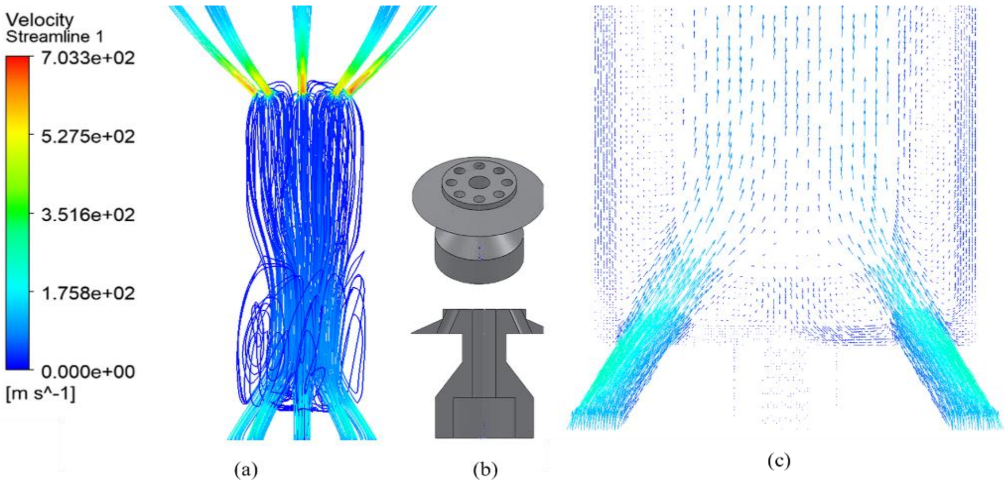


Fig. 5. Flow characteristics for the P2; (a) velocity streamline (b) plate geometry (c) velocity vector

The conditions for P2 and P3 are, however, different (Figure 5 and 6). The air enters at an angle and attacks the fuel as it flows through the inlets, allowing the fuel and air to mix. This causes the molecules to come together and promotes mixing in this region. The angle of the air inlets increases the length of the pathway and duration residence, enabling the stream to travel further away. The procedure allows for the development of a homogeneous mixture in the chamber.

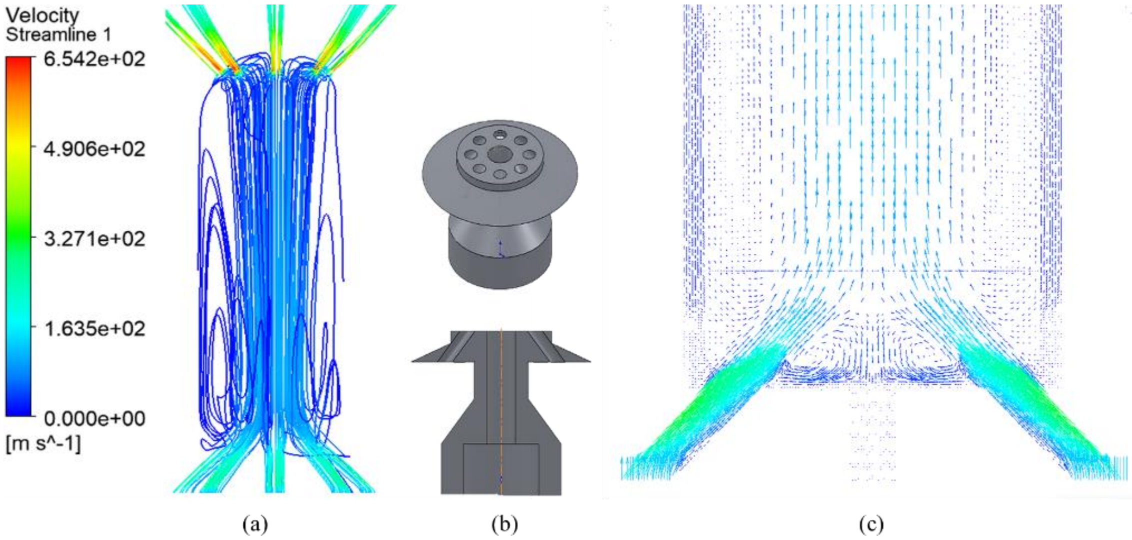


Fig. 6. Flow characteristics for the case, P3; (a) velocity streamline (b) plate geometry (c) velocity vector

3.1.2 Velocity characteristics

The bottom of the mixing chamber exhibits high velocity for P2 and P3. For these plates, its velocity decreases before increasing as it approaches the nozzle. However, the velocity for P1 decreases gradually along the mixing chamber. This is due to the changes in fuel and air volume from a smaller area (inlet) to larger area (mixing chamber). Besides, it is found that the average velocity for both plates decrease along the mixing chamber as the fuel mass flow rate increase as shown in Figure 7.

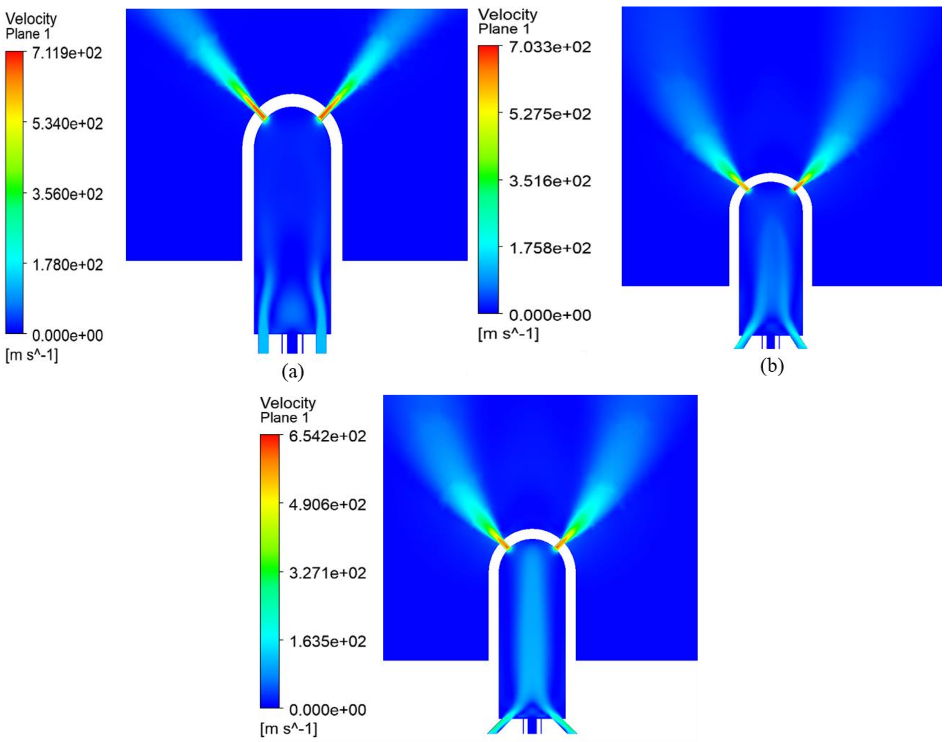


Fig. 7. Velocity contour for the different jet-hole angle; (a) 0°, (b) 30° and (c) 45°

In the comparison from Figure 8, it is clearly seen that the average velocity on the higher inlet angle plates is higher compared to the others. P3, with the largest angle at 45°, exhibits the highest velocity for all planes. At 45°, the air enters the mixing chamber and directly hits the fuel, which is vertically entering the chamber. The momentum produced has increased the kinetic energy and produced high velocities in the mixtures at the bottom of the chamber. The air entrance at 30° is slightly lower in terms of velocity. The trend is consistent at all planes in the chamber except at the top section, where it is not applicable due to the large distance from the recirculation area. This is due to the existence of turbulence motion in the mixing chamber of the plate, which can increase the velocity. As a result, it is possible to deduce that the velocity profile has a considerable impact on fuel mixing and atomization.

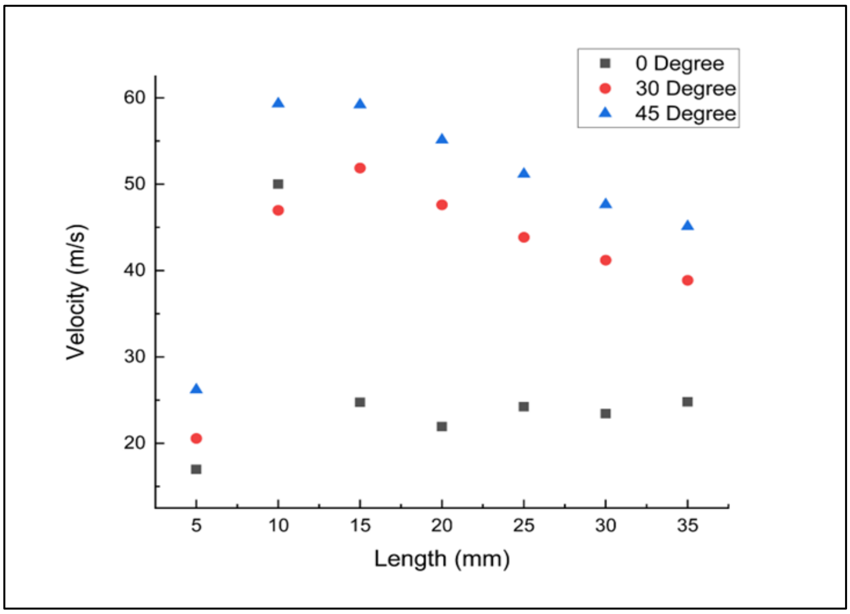


Fig. 8. Velocity profile for MCJ plates in the mixing chamber

3.1.3 Turbulence kinetic energy

Turbulence kinetic energy (TKE) is the mean kinetic energy per unit mass, which is related to the eddies in turbulent flow. Figure 9 depicts the contour of the turbulent kinetic energy of fuel and air mixing inside the mixing chamber. Figure 9 describes the TKE inside the mixing chamber for the MCJ plates with different inlet angles. For P1, the result shows that the TKE decreases gradually as the fluid flow approaches the mixing chamber. This is due to the high flow velocity generated by the inlet hole, which led the inlet air to flow into the center of the mixing chamber. The presence of flow recirculation from the angular air direction gives a higher impact compared to the vertical direction. The collision of air and fuel molecules inside the chamber assists the high production of TKE and is effective at the middle planes (20 mm and above) from the bottom of the chamber, as in Figure 10.

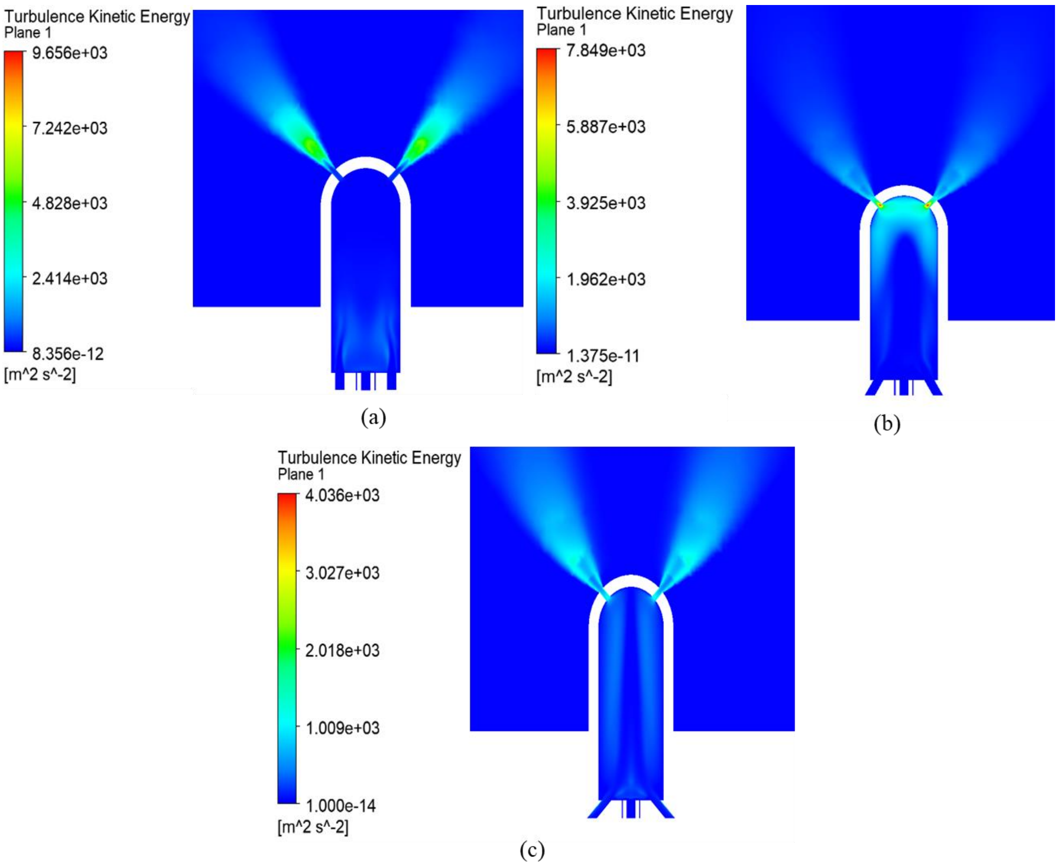


Fig. 9. (a) P1 (2) P2 (c) P3 turbulence kinetic energy contour

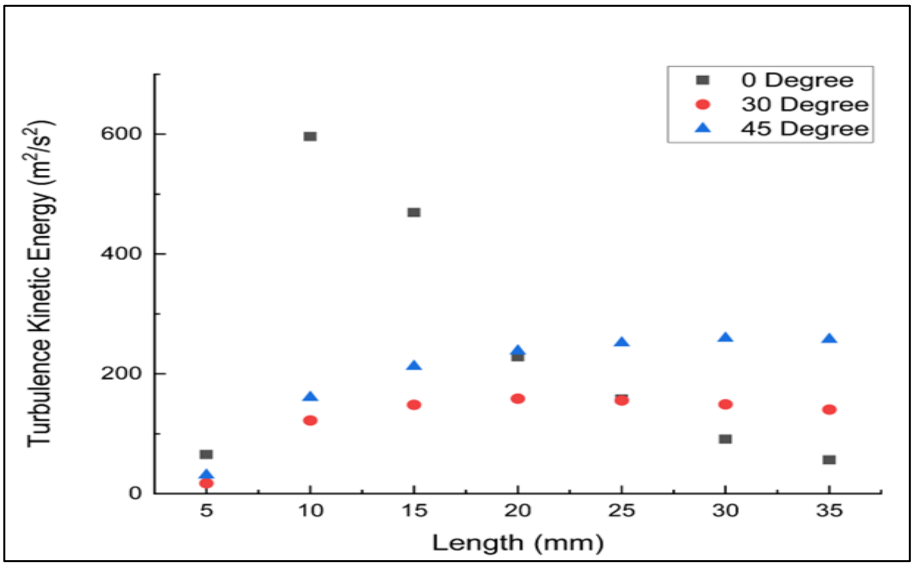


Fig. 10. Turbulence Kinetic Energy for MCJ plates inside the mixing chamber

The comparison of the plates shows that angular jet hole plates can produce higher TKE than vertical jet hole flows. Turbulent kinetic energy is one of the crucial upstream parameters to be determined because it is related to the spray breakup and minimizes the cavitation inside the orifice.

3.1.4 Volume fraction

The volume fraction represents the composition percentage of the diesel fuel and air mixture within the mixing chamber. Figure 11 illustrates the volume fraction along the mixing chamber for MCJ plates. The results showed that the volume fraction of 0-degree angle plates (P1) exhibits a higher value compared to P2 and P3 at the bottom of the mixing chamber. However, the volume fraction drops at higher planes. This is due to the mixing that has taken place throughout the mixing chamber.

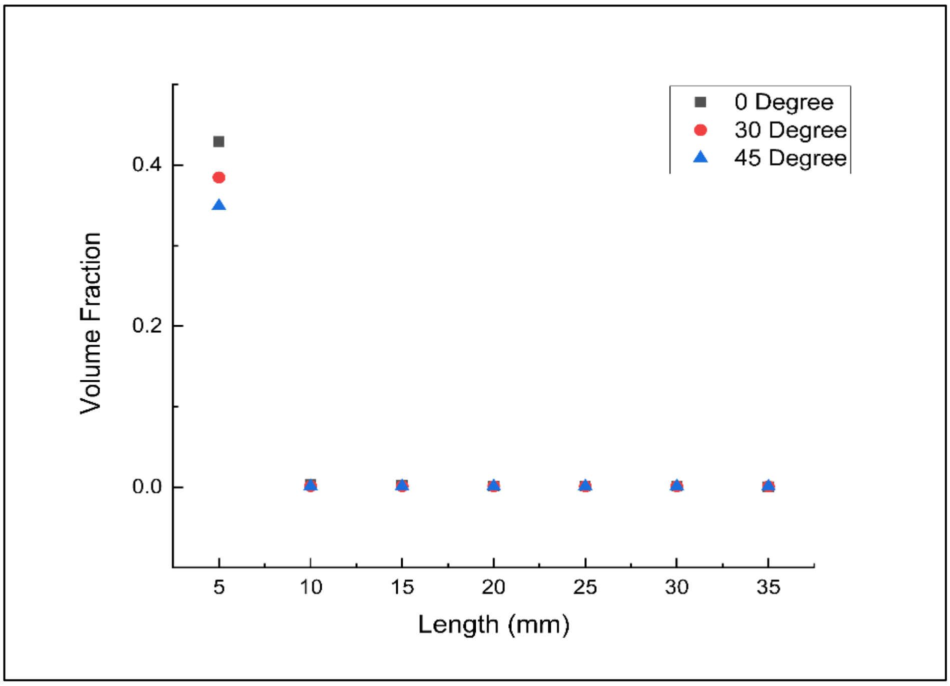


Fig. 11. Volume Fraction for MCJ plates inside the mixing chamber

3.1.5 Pressure characteristics

The pressure that is developed in the mixing chamber determines the pump power required to overcome the pressure of the atomizer. Figure 12 exhibits the pressure developed inside the mixing chamber of P1, P2 and P3. It is observed that the pressure of P2 and P3 is higher compared to P1. The pressure increases the kinetic energy and turbulence inside the mixing chamber to provide a better mixing condition.

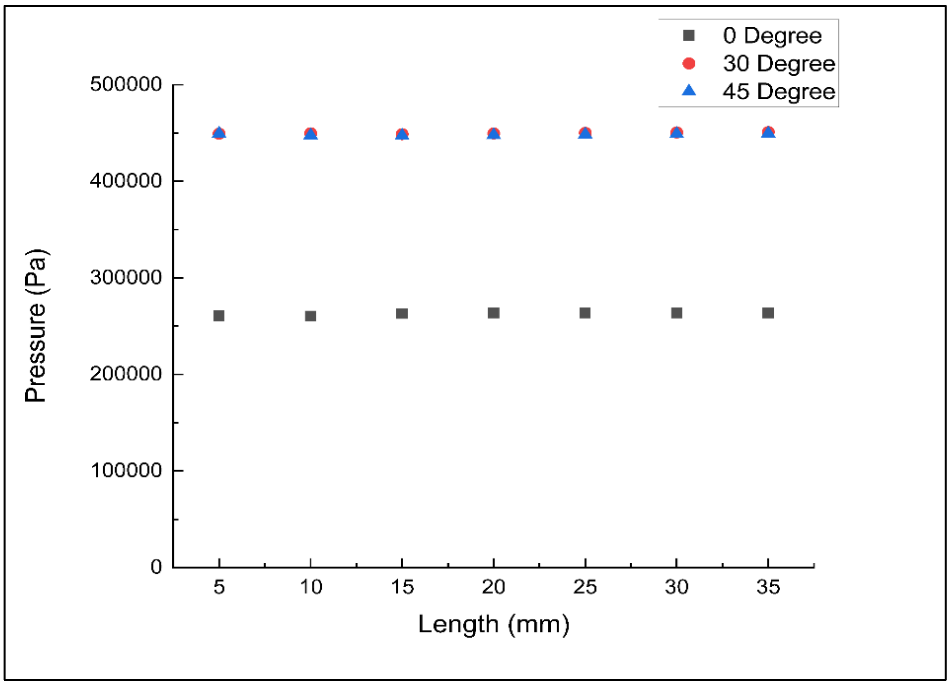


Fig. 12. Pressure for MCJ plates inside the mixing chamber

3.2 Comparison of 45° MCJ (P3) with Different Fuel Properties

This section discusses further the internal flow in the mixing chamber of P3 plate with different types of fuels. The discussion is on how the fuel and air mix with different density and viscosity of a fuel. Fuel compare are diesel and CPO B30 biodiesel as mentioned in subchapter 2.1.

Figure 13 (a) displays the average velocity for the fuels. Both fuels show that the average velocity decreases in the vertical direction of the mixing chamber. However, diesel shows a lower average velocity due to its lower density and higher flow rates. Fuels with a lower density have a lower mass per unit volume, which means the same volume of liquid contains less mass. This reduction in mass will require less force to move the fluid through the inlet at the same velocity, resulting in an increase in flow rate. It is found that CPO B30 has a higher velocity compared to diesel, which shows that CPO B30 has a better mixing and atomization process.

Figure 13 (b) depicts the contour of turbulent kinetic energy per unit mass inside the mixing chamber, where it is related to the eddies in a turbulent flow. CPO B30 with a higher density has higher turbulence kinetic energy due to the increasing Reynolds number. Reynolds numbers influence the turbulent intensity of certain fluids. Thus, increasing the Reynolds number improves the turbulent flow, which leads to better mixing of fuel and air inside the chamber. A study of the turbulence kinetic energy of both fuels demonstrates that CPO B30 biodiesel has a better mixing process than diesel.

Figure 13 (c) depicts the maximum pressure obtained for diesel and CPO B30 biodiesel. CPO B30 produces less pressure than diesel. A low backpressure indicates low pumping power, which can be related to the low power input required to run the burner system. This is related to production cost savings.

Figure 13 (d) shows the volume fraction along the mixing chamber for both fuels. The volume fraction of diesel is higher compared to CPO B30 at the entrance of the mixing chamber (5–10 mm). However, at planes 3 and above, the value is 0, which is the same for both fuels. The volume fraction indicates the composition percentage of the diesel fuel and air mixture inside the mixing chamber. The low volume fraction indicates the swirling flow has assisted in mixing fuel and air, especially at planes 1 and 2. The swirl mixes the liquid and air at a faster rate, as seen from the high velocity of flow inside the mixing chamber. Both fuels undergo a complete mixing process since the volume fraction is zero at the upper level of the plates.

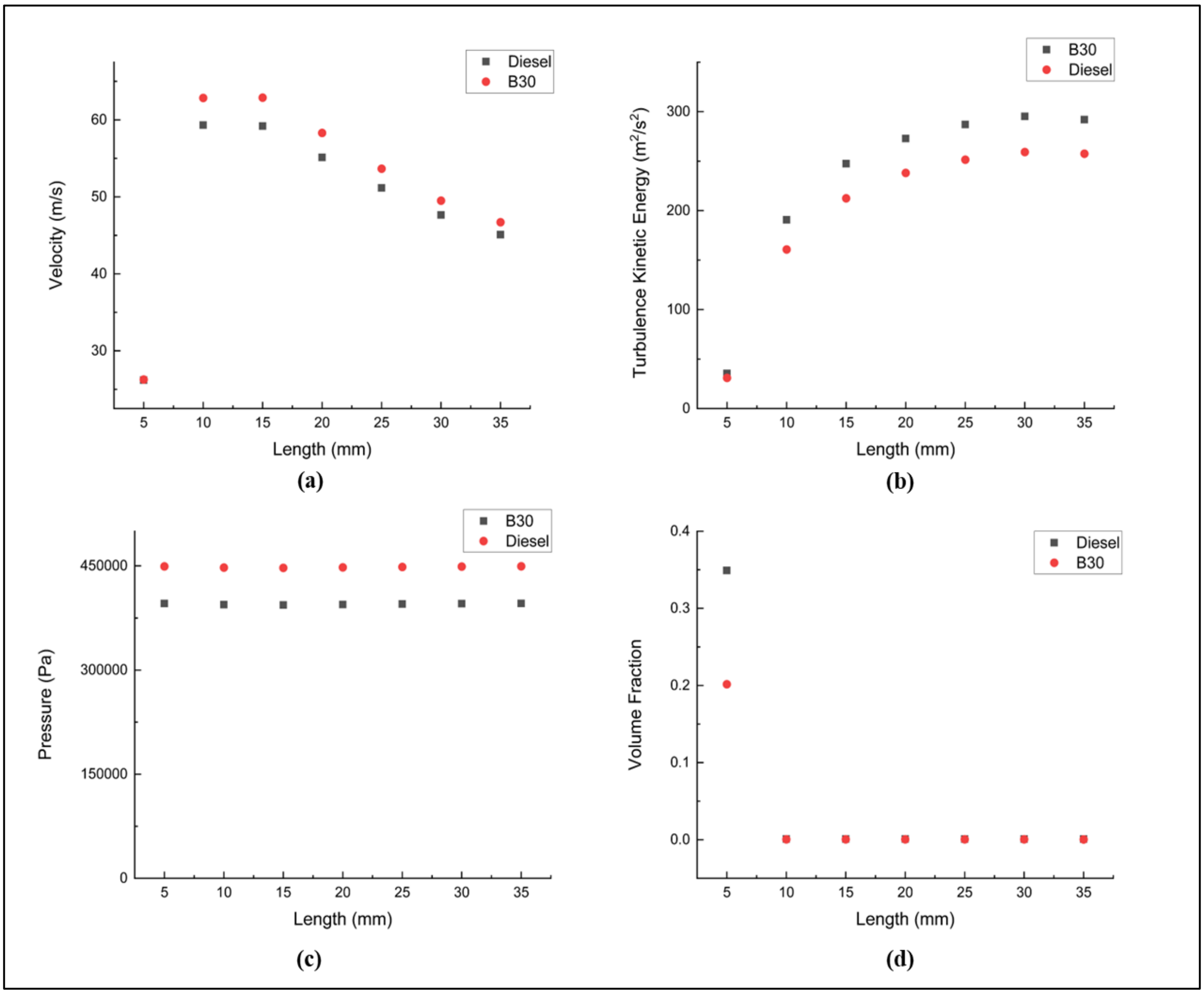


Fig. 13. (a) Velocity (b) Turbulence Kinetic Energy (c) Pressure (d) Volume fraction for P4 between diesel and CPO B30 fuel

4. Conclusions

In this paper, flow behavior in terms of velocity and pressure of fuel and air in a mixing chamber is predicted using CFD analysis. At the bottom of the mixing chamber, MCJ plates act as turbulence generators, achieving a high turbulence level at various jet hole angles.

The plates enable the generation of an intense turbulent field, and the geometry can be manipulated to control the flow. This is used to control the spray and atomization characteristics. The analysis compares various MCJ geometries and shows that P1 geometry causes slower mixing at the bottom of the chamber because there is no direct contact between fuel and air at the inlets and not enough tangential velocity components of the vertical air flows to encourage turbulence and mixing.

Diesel and CPO B30 biodiesel were compared in terms of maximum pressure, average speed, turbulent kinetic energy per unit mass, and volume fraction. The results show that CPO B30 has lower pressure and higher velocity than diesel, implying a better atomization and mixing process. The increased turbulent kinetic energy caused by the higher density of CPO B30 improves the mixing of fuel and air inside the chamber. The volume percent shows that both fuels fully mix, but the swirl in the mixing chamber has helped to further mix air and fuel, particularly at planes 1 and 2.

Acknowledgement
The authors would like to thank the Ministry of Higher Education (MOHE) for supporting this research under the Fundamental Research Grant Scheme No. FRGS/1/2019/TK10/UTHM/02/7 Vot K218 and Universiti Tun Hussein Onn Malaysia (UTHM).

References

[1] Hisham Amirnordin, Shahrin, S. A. L. W. A. N. I. Ismail, Ronny Yii Shi Chin, N. O. R. A. N. I. Mansor, M. A. S. Fawzi, and A. M. I. R. Khalid. "Effects of Nozzle Diameter on the Spray Characteristics of Premix Injector in Burner System." *Applied Mechanics and Materials* 773 (2015): 570-574. https://doi.org/10.4028/www.scientific.net/amm.773-774.570.
[2] Yasin, MF Mohd, R. S. Cant, C. T. Chong, and S. Hochgreb. "Discrete multicomponent model for biodiesel spray combustion simulation." *Fuel* 126 (2014): 44-54. https://doi.org/10.1016/j.fuel.2014.02.020.
[3] Som, Sibendu, Anita I. Ramirez, Douglas E. Longman, and Suresh K. Aggarwal. "Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions." *Fuel* 90, no. 3 (2011): 1267-1276. https://doi.org/10.1016/j.fuel.2010.10.048.
[4] Yatsufusa, Tomoaki, Yoshiyuki Kidoguchi, and Daisuke Nakagawa. "Improvement of emissions and burning limits in burner combustion using an injector on the concept of fuel-water internally rapid mixing." *Journal of Energy and Power Engineering* 8, no. 1 (2014): 11.https://doi.org/10.17265/1934-8975/2014.01.002
[5] Khalid, Amir, and Bukhari Manshoor. "Effect of high swirl velocity on mixture formation and combustion process of diesel spray." *Applied Mechanics and Materials* 229 (2012): 695-699. https://doi.org/10.4028/www.scientific.net/AMM.229-231.695
[6] Liu, Fengshan, Ahmet E. Karataş, Ömer L. Gülder, and Mingyan Gu. "Numerical and experimental study of the influence of CO2 and N2 dilution on soot formation in laminar coflow C2H4/air diffusion flames at pressures between 5 and 20 atm." *Combustion and Flame* 162, no. 5 (2015): 2231-2247. https://doi.org/10.1016/j.combustflame.2015.01.020
[7] García, J. A., A. Lozano, J. Alconchel, E. Calvo, F. Barreras, and J. L. Santolaya. "Atomization of glycerin with a twin-fluid swirl nozzle." *International Journal of Multiphase Flow* 92 (2017): 150-160. https://doi.org/10.1016/j.ijmultiphaseflow.2017.03.010
[8] Yu, Shenghao, Bifeng Yin, Weixin Deng, Hekun Jia, Ze Ye, Bin Xu, and Huaping Xu. "An experimental comparison of the elliptical and circular nozzles spray and mixing characteristics under different injection pressures." *Fuel* 236 (2019): 1474-1482.
[9] Gad, H. M., E. A. Baraya, T. M. Farag, and I. A. Ibrahim. "Effect of geometric parameters on spray characteristics of air assisted pressure swirl atomizer." *Alexandria Engineering Journal* 61, no. 7 (2022): 5557-5571.https://doi.org/10.1016/j.aej.2021.11.010
[10] Yatsufusa, T., T. Kumura, Y. Nakagawa, and Y. Kidoguchi. "Advantage of using water-emulsified fuel on combustion and emission characteristics." *Fuel* 5, no. 60 (2009).
[11] Shriram Sathishkumar, Bommisetty Sambasiva Rao, Sidharth Pradeep, Solai Sairam R. M., Balaji Kalaiarasu, & Prabhu Selvaraj. "Modelling and Validating the Spray Characteristics of a Co-axial Twin-Fluid Atomizer Using OpenFOAM. " *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 91, no. 1 (2022): 35–45. https://doi.org/10.37934/arfmts.91.1.3545
[12] Rasha Abdulrazzak Jasim, Qasim Hussen, W., Abdullah, M. F., & Rozli Zulkifli. "Numerical Simulation of Characterization of Hydraulic Jump Over an Obstacle in an Open Channel Flow. " *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 106, no. 1 (2023): 1–15. https://doi.org/10.37934/arfmts.106.1.115
[13] Shu, Y., Hu, P., Adnan, I., Zhi, L., & Chen, Q. (2022). Comparison between homogeneous and separated flow models of isobutane flowing through adiabatic capillary tubes. JUSTC, 52(7), 5. https://doi.org/10.52396/JUSTC-2021-0130
[14] Du, Zhao-bo, Wei Huang, and Li Yan. "Parametric study on mixing augmentation mechanism induced by air injection in a shock-induced combustion ramjet engine." *Energy* 186 (2019): 115895. https://doi.org/10.1016/j.energy.2019.115895
[15] Zhu, Zhaoyou, Bin Qin, Shuhua Li, Yigang Liu, Xin Li, Peizhe Cui, Yinglong Wang, and Jun Gao. "Multi-dimensional analysis of turbulence models for immiscible liquid-liquid mixing in stirred tank based on numerical

simulation." *Separation Science and Technology* 56, no. 2 (2021): 411-424. https://doi.org/10.1080/01496395.2020.1715436

[16] Abidi, Shahim Haider, and M. M. Hasan. "Effect due to Variation in Bend Angles of Intake Manifold on Turbulent Kinetic Energy for Diesel Engine." *In International Journal of Innovative Technology and Exploring Engineering (Vol. 9, Issue 5, pp. 1979–1982). Blue Eyes Intelligence Engineering and Sciences Engineering and Sciences Publication - BEIESP.* https://doi.org/10.35940/ijitee.e2973.039520

[17] Narayana Rao, K. V. L., B. V. S. S. S. Prasad, C. H. Kanna Babu, and Girish K. Degaonkar. "Influence of inlet swirl on pattern factor and pressure loss in an aero engine combustor." *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 236, no. 5 (2022): 2631-2645. https://doi.org/10.1177/09544062211027209

[18] Eliasz, Jacek, Tomasz Osipowicz, Karol Franciszek Abramek, and Łukasz Mozga. "Model issues regarding modification of fuel injector components to improve the injection parameters of a modern compression ignition engine powered by biofuel." *Applied Sciences* 9, no. 24 (2019): 5479. https://doi.org/10.3390/app9245479

[19] Sies, Mohamad Farid, Norrizal Mustaffa, Hanis Zakaria, Hamidon bin Salleh, Bukhari Manshoor, and Amir Khalid. *Development of the premixing injector in burner system*. Vol. 465. Trans Tech Publications Ltd, 2014. https://doi.org/10.4028/www.scientific.net/AMM.465-466.302

[20] Amirnordin, Shahrin Hisham, Amir Khalid, Norshuhaila Mohamed Sunar, Mas Fawzi Mohd Ali, Azwan Sapit, Muhammad Ridzwan Kamaruddin, Syabillah Sulaiman, and Dr Hasan Koten. "Comparison of Spray Characteristics, Swirling Motion and Flame Development for Multi-Circular Jet and Swirl Plates." *Swirling Motion and Flame Development for Multi-Circular Jet and Swirl Plates*. *SSRN Electronic Journal*. https://doi.org/10.2139/ssrn.4055166

[21] Hou, Xingxia, Shaolin Wang, Pengfu Xie, Qing Gao, Chunqing Tan, and Jianwen Wang. "CFD Simulation of Combustion in Gas Turbine Engine." In *2017 5th International Conference on Frontiers of Manufacturing Science and Measuring Technology (FMSMT 2017)*, pp. 890-896. Atlantis Press, 2017. https://doi.org/10.2991/fmsmt-17.2017.170

[22] Zhang, Yang, and Maarten Vanierschot. "Modeling capabilities of unsteady RANS for the simulation of turbulent swirling flow in an annular bluff-body combustor geometry." *Applied Mathematical Modelling* 89 (2021): 1140-1154. https://doi.org/10.1016/j.apm.2020.07.037

[23] Wan Rosli, W. M. A., & Padzillah, M. H. "Investigation on the Flow Angle of a Mixed Flow Turbocharger Turbine Under Steady State Operating Conditions." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 99, No.1 (2022): 104–113. https://doi.org/10.37934/arfmts.99.1.104113

[24] Du, Bingxuan, and Zhenfeng Zhao. "Numerical prediction of the spray from an air-assisted fuel injection system via Eulerian–Lagrangian approach." *Energy Reports* 7 (2021): 6718-6732. https://doi.org/10.1016/j.egyr.2021.09.128

[25] Hou, Xingxia, Shaolin Wang, Pengfu Xie, Qing Gao, Chunqing Tan, and Jianwen Wang. "CFD Simulation of Combustion in Gas Turbine Engine." In *2017 5th International Conference on Frontiers of Manufacturing Science and Measuring Technology (FMSMT 2017)*, pp. 890-896. Atlantis Press, 2017. https://doi.org/10.2991/fmsmt-17.2017.170