Progress in Engineering Application and Technology Vol. 3 No. 2 (2022) 621-629
© Universiti Tun Hussein Onn Malaysia Publisher's Office

PEAT

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/peat
e-ISSN : 2773-5303

Simulation and Modelling. The Spray Behaviour of Ammonia-Biodiesel Fuel Blends and Injector Characteristics in Micro-Gas Turbine

Syahmi Wafiy Mohd Shariff[1], Syabillah Sulaiman[1]*, Muhammad Asri Azizul[1]

[1]Department of Mechanical Engineering Technology, Faculty of Engineering Technology,
Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/peat.2022.03.02.059
Received 27 January 2022; Accepted 07 November 2022; Available online 10 December 2022

Abstract: Ammonia is one of carbon-free fuel that are considered to be alternative fuels. Liquid ammonia is blended with biodiesel fuel in order to further reduce emissions and to achieve more stable combustion. The premix injector design is specially designed to suite the application of the combustion system which is the micro-gas turbine. The aim of this research is to study and analyses the spray performance of the ammonia biodiesel fuel blend on the premix injector nozzle. The method for this research is by using Computational Fluid Dynamics (CFD) simulation approach where ANSYS software was used to perform the simulation. The solver for the simulation is the SST turbulence model as it is one of the most suitable turbulence models for internal flow simulation. Diesel fuel was used first for baseline comparison to flow through the injector. The biodiesel fuel used was B10 Crude Palm Oil (CPO) biodiesel. Then three percentage blends of ammonia-CPO biodiesel were used next to observe and analyses its spray behaviour. The three percentages of ammonia biodiesel blends were 2.00 % ammonia with 98.00 % CPO, 4.00 % ammonia with 96.00 % CPO, and 6.00 % ammonia with 94.00 % CPO. From the simulation results, diesel fuel had the lower velocity flow and turbulence kinetic energy compared to the ammonia biodiesel fuel blends with 2960.0 m/s of fluid flow velocity. The 6.00 % ammonia with 94.00 % biodiesel percentage blends had the highest maximum velocity than other percentage blends with 4870.9 m/s of fluid velocity due to baseline flow of diesel. The study shows that the ammonia element in the ammonia biodiesel fuel influenced the spray performance in premix injector.

Keywords: Injector, Ammonia, Spray

1. Introduction

**Corresponding author: syabillah@uthm.edu.my*

Carbon dioxide, emitted by fuel combustion, is a significant source of (Green House Gaseous) GHGs and has shifted the climate system [1]. New cleaner fuel sources had been studied to substitute the high carbon emission fuels such as standard diesel and petrol. One of the most recent fuels that had been discovered is the ammonia content liquid fuel with the aid of biodiesel to provide cleaner blends [2] [3]. The premix injector that was designed suited the application of the burner system. With new fuel in hand, the performance by using alternative fuels were different that the standard fuels as the fuel composition were different and behave different when flow through the injector system of burner [4] [5]. The use of carbon free fuel blends will lead toward other alternative fuel for the same objective which is to reduce emission from combustion. The CFD simulation method used in this research also will reduce cost and time as it only requires virtual experiment where in this case is done in computers and will save up a lot of time. The simulated conditions were at standard ambient conditions where the temperature is at 25 °C and pressure at 0.9968 atm.

2. Simulation approach

The approach of this study is by using Computational Fluid Dynamics (CFD) method. There are essentially three main stages in running a CFD simulation, which are pre-processing, solver and post processing

2.1 Premix injector geometry

The focus of this study is on the injector nozzle. The premix injector has eight nozzle holes with 45 ° angle in each hole. Instead of using a swirl type multi circular jet plate, this research uses multi circular jet plate with eight identical jet holes [6]. Figure 1 shows the geometry of premix injector and the actual model.

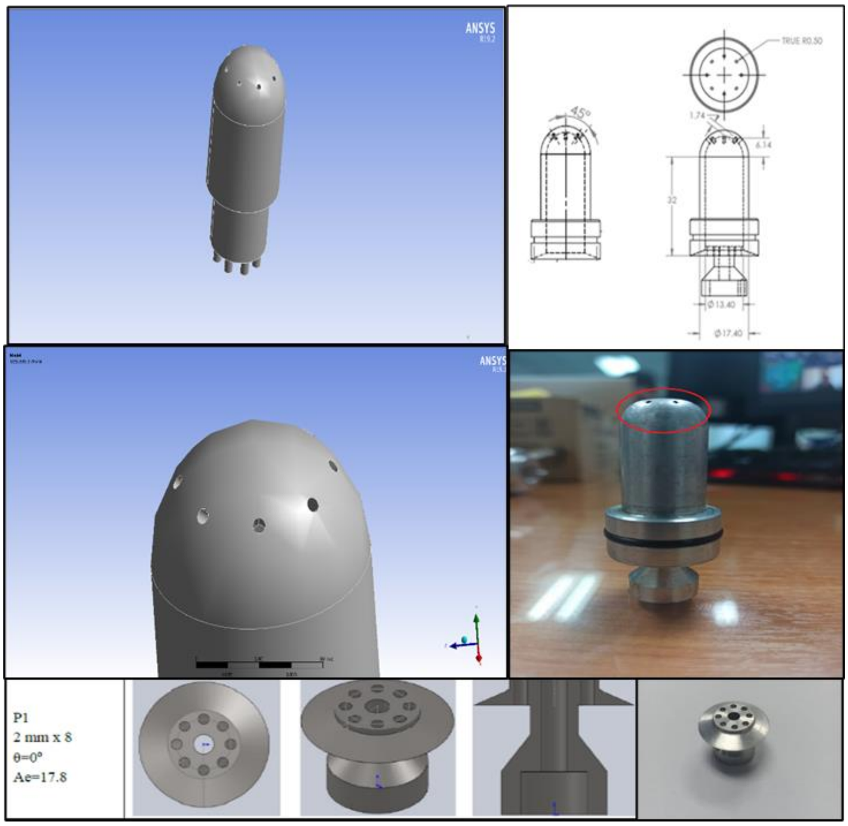


Figure 1: Premix injector model and geometry

2.2 Simulation with ANSYS software

The CFD simulation in used by using ANSYS software as it was one of the most appropriate software solvers for internal flow simulations [7]. The model of the body that are used for analysis its interactions with fluid flow is then calculated by the programmed and limited by the boundary conditions. Figure 2 shows the interface of ANSYS software.

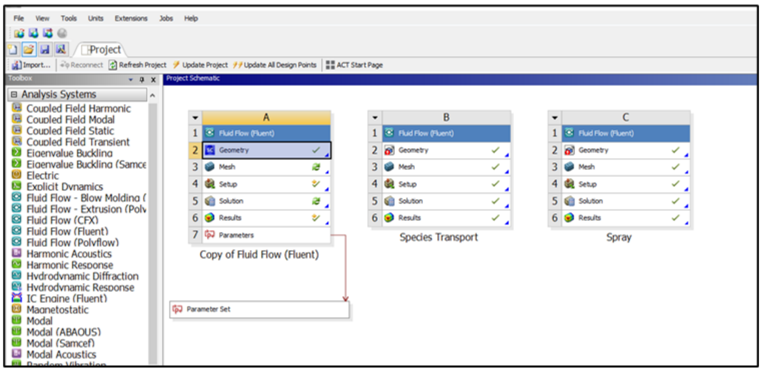


Figure 2: ANSYS software interface

2.2.1 Boundary conditions

The boundary condition of inlet and outlet flow for ANSYS Fluent software is set within the 3D model. Boundary condition of mass flow rate for the inlet is used to determine the velocity and mass flow of the flow at inlet wall boundaries, while the boundary condition of pressure outlet is used to determine the static pressure at the flow outlet. The two phases of materials were air with standard fuel, and ammonia biodiesel percentage blends with air by using multiphase liquid approach. The input for the simulation was mass flow rate for all liquids.

Table 1: Mass flow rate of standard diesel and air

Material phase	Diesel	Ammonia biodiesel			Air
		2% ammonia with 98% biodiesel	4% ammonia with 96% biodiesel	6% ammonia with 94% biodiesel	
Mass flow rate (kg/h)	0.26934	1.6778	1.7186	1.7212	0.00393

The boundary condition that are used is as listed below:

i. Solid wall. Figure 3 shows the solid wall of premix injector.

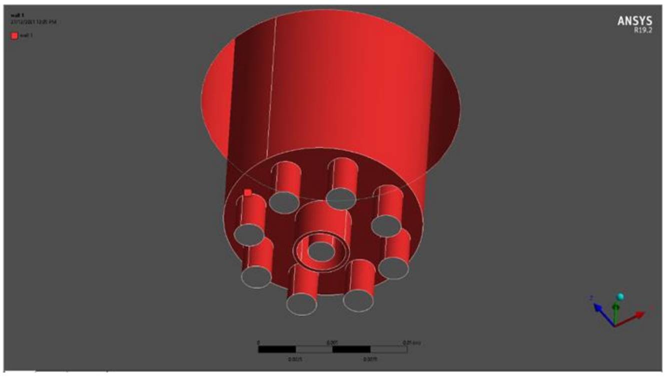


Figure 3: Solid wall of premix injector

ii. Periodic wall. Figure 4 shows the periodic wall for simulation

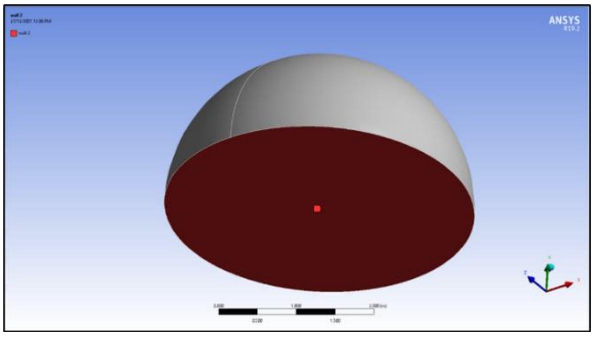
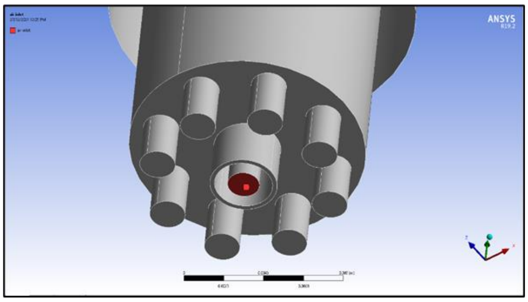
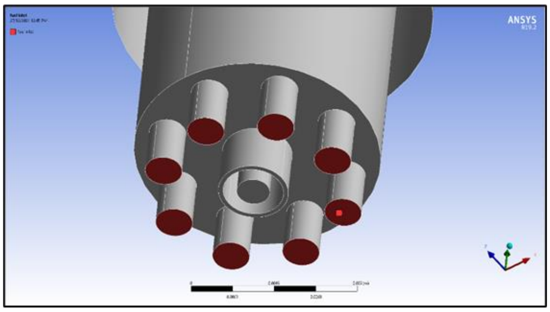


Figure 4: Periodic wall for simulation

iii. Inlet. Figure 5 shows the mass flow rate inlet (a) Air inlet (b) Fuel inlet



(a)



(b)

Figure 5: Mass flow rate inlet (a) Air inlet (b) Fuel inlet

iv. Outlet. Figure 6 shows the outlet domain of simulation

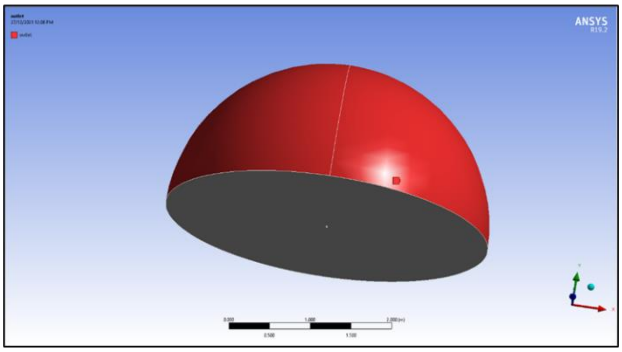


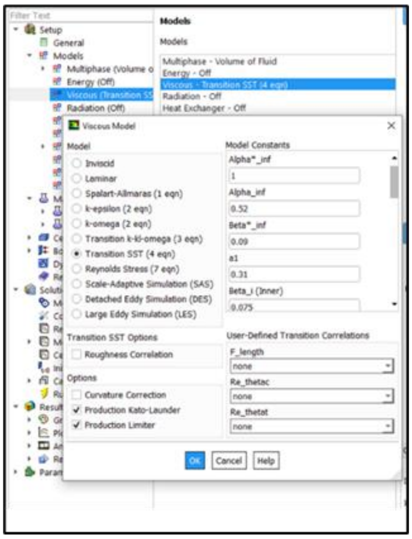
Figure 6: Outlet domain of simulation

2.2.2 Meshing

Meshing is the process of selecting region into multiple locations knowns as elements and nodes to enable calculation to be performed on to the geometry. Smaller region that leads to a greater number of elements and nodes will produce more accurate calculation results. Size of meshing is 0.02m and curvature normal angle is at 18.0°. The mesh elements are 463,262 and the number of nodes is 87,891. Figure 6 shows the Model meshing and meshing details.

2.2.3 Solver

Turbulence model suitable for this simulation is sel ected, which is the Menter's Shear Stress Transport (SST) K-Omega turbulence model. SST K-Omega turbulence model is typical for CFD simulation research to study liquid internal flow for higher accuracy results and less time consuming [7]. Figure 7 shows ANSYS Fluent solver.



Models

Multiphase - Volume of Fluid
Energy - Off
Viscous - Transition SST (4 eqn)
Radiation - Off
Heat Exchanger - Off

Viscous Model

Model
○ Inviscid
○ Laminar
○ Spalart-Allmaras (1 eqn)
○ k-epsilon (2 eqn)
○ k-omega (2 eqn)
○ Transition k-kl-omega (3 eqn)
● Transition SST (4 eqn)
○ Reynolds Stress (7 eqn)
○ Scale-Adaptive Simulation (SAS)
○ Detached Eddy Simulation (DES)
○ Large Eddy Simulation (LES)

Model Constants
Alpha*_inf: 1
Alpha_inf: 0.52
Beta*_inf: 0.09
a1: 0.31
Beta_i (Inner): 0.075

Transition SST Options
☐ Roughness Correlation

Options
☐ Curvature Correction
☑ Production Kato-Launder
☑ Production Limiter

User-Defined Transition Correlations
F_length: none
Re_thetac: none
Re_thetat: none

Figure 7: ANSYS Fluent solver

2.2.4 Post processing

The result of the simulation was observed in the CFD Post sub-software in ANSYS Workbench. In this case of internal fluid in premix injector was observed in CFD Post. The fluid behaviour or the spray behaviour can be observed in CFD Post by displaying the fluid's contour. Figure 8 shows velocity contour of simulation result in CFD Post

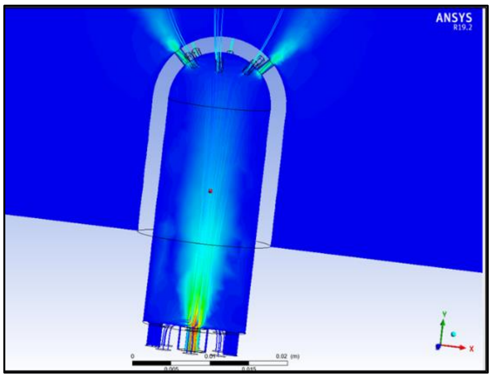


Figure 8: Velocity contour of simulation result in CFD Post

2.3 Fuel

Diesel fuel was used first into the simulation through the premix injector model for baseline comparison. Fluid flow and spray behaviour of the diesel fuel was analysed before switching the fuel.

The fuel was switched with ammonia biodiesel blend fuel. The biodiesel that was used to blend with the ammonia was the B10 Crude Palm Oil (CPO) biodiesel. The ammonia biodiesel fuel was divided into three different percentage blends. The three percentage fuel blends were analysed its spray behaviour to determine its performance effect. Table 2 shows the diesel and ammonia biodiesel percentage blends properties [14].

Table 2: Diesel and ammonia biodiesel percentage blends properties [14]

Fuel	Density (g/cm^3)	Viscosity (mm^2/s)
Diesel	880.0	5.77
2% ammonia with 98% CPO	873.5	4.01
4% ammonia with 96% CPO	846.1	4.08
6% ammonia with 94% CPO	848.9	4.15

3. Results and Discussion

The aim of this study is to study the spray behaviour of premix injector nozzle with different percentage of ammonia-CPO biodiesel fuel blends by using CFD simulation method. This is because different liquid fuel properties will affect atomization process [9]. Diesel fuel was used for baseline comparison with the varying percentage biodiesel blends

3.1 Nozzle flow velocity

The difference of fluid behaviour was due to the difference of properties on flow distribution [11]. The inertia and viscosity pull of the liquid caused the lower velocity flow [13]. The initial low velocity of all fuels was due to cavitation during nozzle entering. Fluid that flows through different opening in the initial stage experience cavitation [10] [12]. In contrast of diesel fuel, ammonia biodiesel percentage blends had lower density and viscosity therefore the higher velocity flow. Velocity of flow reduced after the peak velocity reached due to gravitational force and the loss of energy of fluid's particles [13].

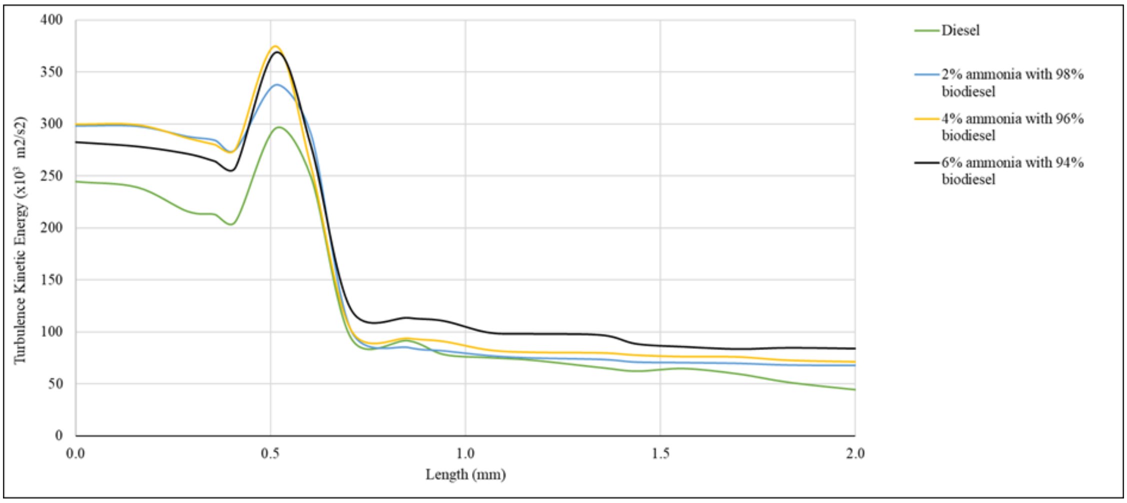


Figure 9: Nozzle flow velocities against length of nozzle of different fuels

3.2 Turbulence kinetic energy

All three ammonia biodiesel fuel blends had higher overall turbulence kinetic energy than diesel fuel. are 2.00 % ammonia with 98.00 % biodiesel, 4.00 % ammonia with 96.00 % biodiesel, and 6.00 % ammonia with 94.00 % biodiesel had similar turbulence kinetic energy beginning of the flow inside injector nozzle and had a sudden decline right after the peaked turbulence kinetic energy, continue the gradual drop of turbulence kinetic energy to the end of the nozzle.

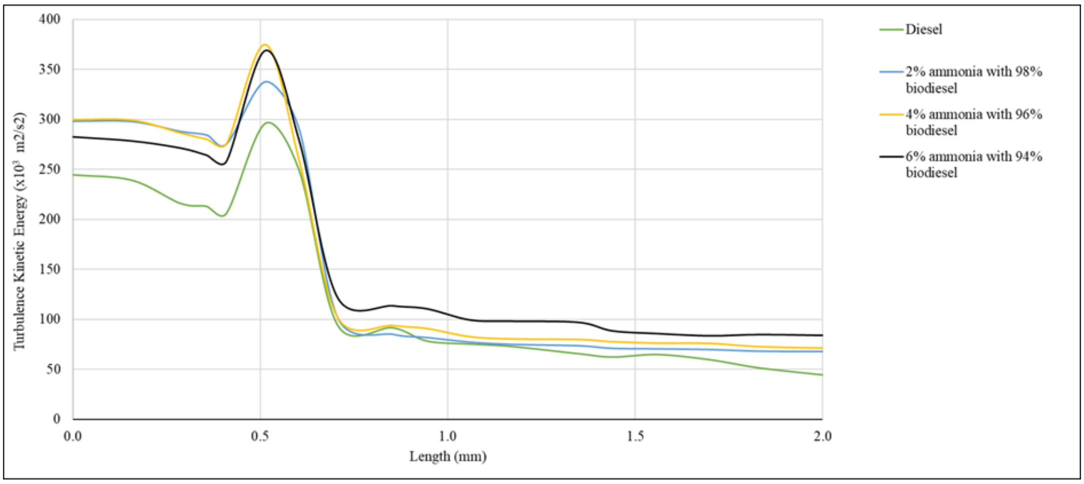


Figure 10: Turbulence kinetic energy of fuels in the injector nozzle

The fuels that flow through the injector nozzle experience turbulence due to reduction during flow entry [40]. Fluid flow through different channel flow involved in energy exchange [8]. The fluctuation of turbulence kinetic energy in the beginning of the fuel indicated that cavitation bubbles were formed in the beginning part of injector nozzle, Cavitation bubbles were formed during that region. Imploding of the cavitation bubble causes the turbulence kinetic energy to rise whereas the fluctuation of turbulence kinetic energy before the 0.5mm region was due to turbulence flow between channels [8] [10].

3.3 Nozzle spray performance

The 6.00 % ammonia with 94.00 % biodiesel fuel blend percentage had the highest maximum velocity compared to other two percentage blends and diesel fuel. The maximum velocity of each fuel was obvious in the velocity contour shown in Figure 11 where the bright green region in the middle part of injector nozzle were present in all the fuel cases. Table 3 indicates the velocity flow of each fuel at specific region length of injector nozzle.

Table 3: Different fuel maximum velocity in injector nozzle

Fuel	Maximum velocity (m/s)
Diesel	2960.0
2% ammonia with 98% biodiesel	4600.7
4% ammonia with 96% biodiesel	4540.6
6%ammonia with 94% biodiesel	4870.9

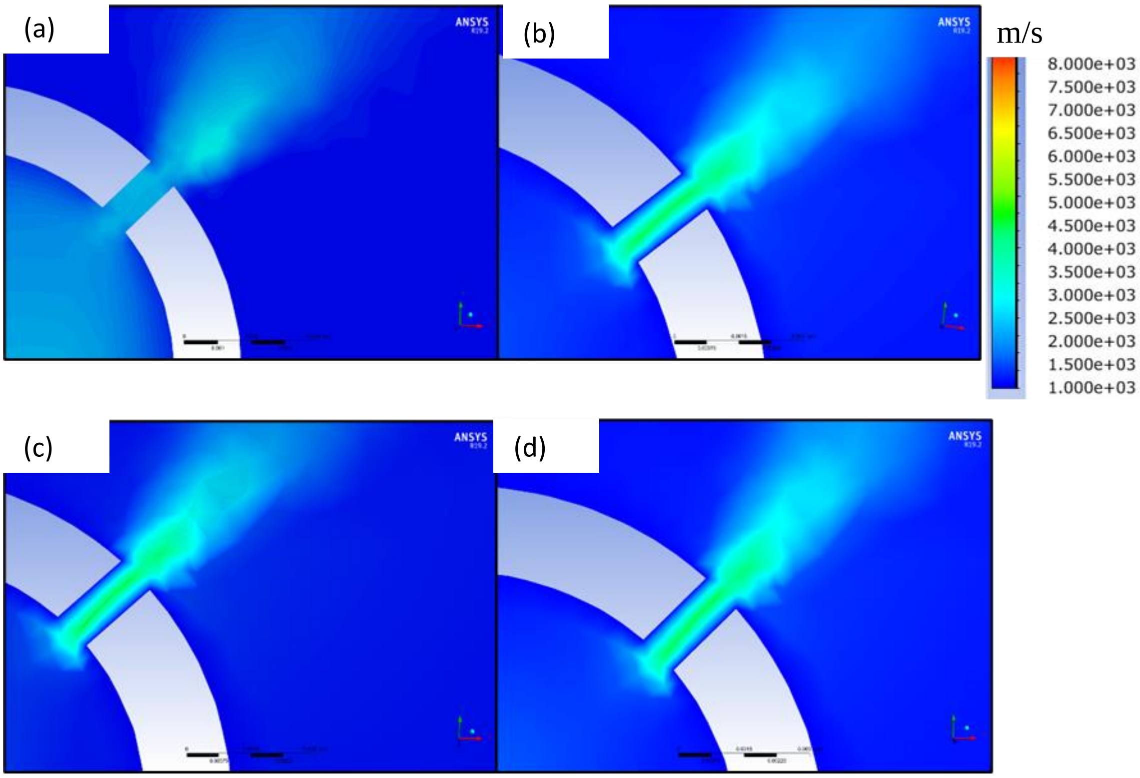


Figure 11: Velocity contour of fuels (a) Diesel (b) 2.00 % ammonia with 98.00 % biodiesel (c) 4.00 % ammonia with 96.00 % biodiesel (d) 6.00 % ammonia with 94.00 % biodiesel

4. Conclusion

Simulation method of ammonia biodiesel fuel blends was constructed with CFD approach by using ANSYS software. Diesel fuel produced lower spray velocity compared to all the ammonia biodiesel percentage blends. The ammonia biodiesel fuel with 6.00 % ammonia with 94% biodiesel percentage produced the highest velocity flow compared to other blend percentages and diesel fuel. Ammonia element contained in the ammonia biodiesel hugely affect fluid behaviour flow in the premix injector.

Acknowledgement

This research was made possible by funding from research grant number K218 provided by the Ministry of Higher Education, Malaysia. The authors would also like to thank the Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] Ridwan, S. N., Zalil, Z., Jalil, S., Amir, K., Syahrunniza, A. H., (2017), Experimental study of the bio-additives effects in biodiesel fuel on performance, emissions and combustions characteristics of diesel engine, ARPN Journal of Engineering and Applied Sciences, Volume 12, Issue 6, Pages 1997-2005

[2] João, S. C., Valter, S., Rodolfo, C. R., Matthew, J., Hall, M. C., Daniela, E., (2021), Ammonia as an energy vector: Current and future prospects for low-carbon fuel applications in internal combustion engines, Journal of Cleaner Production, Volume 296, 126562

[3] Zhijian, W., Youkun, T., Jing, S., Yinghui, Z., Hengzhi, Y., (2021), Ammonia as an effective hydrogen carrier and a clean fuel for solid oxide fuel cells, Energy Conversion and Management, Volume 228, 113729

[4] Muhammad, A. H., Amir, K., Hamidon, S., Norshuhaila M. S., (2017), Effects of Fuel and Nozzle Characteristics on Micro Gas Turbine System: A Review, IOP Conference Series: Materials Science and Engineering, Volume 226, 012006

[5] Hua, X., Augustin, V. M., Philip, J. B., (2017), Modelling Combustion of Ammonia/Hydrogen Fuel Blends under Gas Turbine Conditions, Energy and Fuels, Volume 31, Issue 8, Pages 8631-8642

[6] Shahrin, H. A., Salwani, I., Ronny, Y. S. C., Norani, M., Mas, F., Amir, K., (2015) Effects of Nozzle Diameter on the Spray Characteristics of Premix Injector in Burner System, Applied Mechanics and Materials, Volume 773-774, Pages 570-574

[7] E., Stenmark, (2013), Chalmers On Multiphase Flow Models in ANSYS CFD Software, Chalmers, Department of Applied Mechanincs, Division of Fluid Dynamics, Chalmers University of Technology

[8] A., Nakayama, F. Kuwahara, (2008) A general microscopic turbulence for flows in packed beds, channels, pipes, and rod bundles, Journal of Fluid Engineering, Volume 130, Issue 10

[9] G., Charalampous, Y., Hardalupas, (2016), How do liquid fuel physical properties affect liquid jet development in atomizers, Physics of Fluid, Volume 10, Issue 28

[10] C., Baumgarten., (2006), Mixture Formation in Internal Combustion Engines, Germany: Springer-Verlag Berlin Heidelberg

[11] H., Daikoku, S., Kawanishi, T., Ishikawa, T., Yoshikawa, (2021), Density, surface tension, and viscosity of liquid Si-Cr alloys and influence on temperature and fluid flow during solution growth, Journal of Chemical Thermodynamics, Issue 160

[12] M., Balaji, Y., Wenming, C., Siawkiang, (2014), Cavitation in Injector Nozzle Holes – A Parametric Study, Engineering Applications of Computational Fluid Mechanics, Volume 8, Issue 1, 70-81

[13] Y., Chia-Shun, (1969), Effect of density variation on fluid flow, Journal of Geophysical Research, Volume 64, Issue 12, 2219-2223

[14] R., Saputra Nursal, (2015), Performance and Emission Characteristics of Alternative Biodiesel Fuel on 4-Stroke Marine Diesel Engine (writing)