



AENSI Journals

Australian Journal of Basic and Applied Sciences

ISSN: 1991-8178

Journal home page: www.ajbasweb.com



## Simulation Study of Characteristic of Premix Injector for Biodiesel Fuel, Air and Water Mixing In Open Burner System

<sup>1</sup>Manshoor Bukhari, <sup>1</sup>Zaman Izzuddin, <sup>1</sup>SIES Mohamad Farid, <sup>2</sup>Leman Abdul Mutalib and <sup>1</sup>Khalid Amir

<sup>1</sup>Centre for Energy and Industrial Environment Studies (CEIES), Faculty of Mechanical and Manufacturing Engineering Universiti Tun Hussein Onn Malaysia 86400 Pt. Raja, Batu Pahat, Johor Malaysia

<sup>2</sup>Faculty of Technology Engineering Universiti Tun Hussein Onn Malaysia 86400 Pt. Raja, Batu Pahat, Johor Malaysia

### ARTICLE INFO

#### Article history:

Received 25 January 2014

Received in revised form 12

March 2014

Accepted 14 April 2014

Available online 25 April 2014

#### Keywords:

Air-fuel mixing, burner, CFD

### ABSTRACT

The works conducted by ARG Research Group are focused on development a new type of mixing devices for air-fuel preventing from ignition of the mixture in the mixing domain. The mixing devices designed mainly for partial oxidation process used in mixing technology. The presented results are initially stage data of simulation supplied with air-fuel-water under environment conditions. The aim of the research is to present some data obtained during the simulation process of an air-fuel-water mixing efficiency in a mixing chamber. Besides that, the data will be use as a preliminary data to develop a new geometry of burner or injector for high stream technology especially in diesel combustion. In this research, an external premix had been simulated using a concept of burner combustion using an injector on air-fuel-water internally rapid mixing. During the simulation, working fluids were divided into three difference of percentage fraction which is 0% water, 10% water and 20% water. The simulation work had been done by using CFD software package ANSYS CFX. Premix injector was modeled using Solidwork software. Simulation work was carried out with different length of mixing chamber, inlets, and spray chamber for each type of Biodiesel fuel. The inlets of air, diesel fuel, and water have been set in velocity. The results showed that high pressure occurred in the combustion chamber are for biodiesel fuel without any addition of water. As the pressure is inversely proportional to the velocity, the velocity in the combustion chamber for different percentage of water showed that the highest velocity occurred for the biodiesel with 20% of water. The spray pattern can be observed in spray chamber and determine the maximum and minimum values of parameters.

© 2014 AENSI Publisher All rights reserved.

**To Cite This Article:** Manshoor Bukhari, Zaman Izzuddin, SIES Mohamad Farid, Leman Abdul Mutalib, and Khalid Amir., Simulation Study of Characteristic of Premix Injector for Biodiesel Fuel, Air and Water Mixing In Open Burner System. *Aust. J. Basic & Appl. Sci.*, 8(5): 478-783, 2014

## INTRODUCTION

Global warming is one of the environmental issues since 1958. Nowadays it becomes more serious since rapidly increasing of emissions from diesel fuel in transportation and manufacturing sectors (Amir Khalid *et al.*, 2011). Basically, the global warming is referring to an increase of temperature of earth surface and it creates green house effect to an environment (K. Hanjo, 1996). The transportation emissions especially from diesel engine can contribute direct warming impact such as carbon dioxide (CO<sub>2</sub>) and indirect impact such as nitrogen oxides (NO<sub>x</sub>), production, and economy (M.F. Seis *et al.*, 2014). One of the solutions that can be proposed to overcome this crisis is by using biodiesel fuel (BDF) as alternative fuel in both sectors (M.F. Seis *et al.*, 2014). Although the BDF can be

An alternative fuel to reduce a usage of diesel fuel, the BDF also had a disadvantage due to low quality of fuel and decreased the performance of vehicle or machines compared to diesel fuel (M. Nazri, 2011). Another issue in BDF is high toxic emission such as NO<sub>x</sub> and soot. In order to decrease the emission to the environment, a water-fuel emulsion was introduced by using an injector. The injector can combine the water-fuel emulsion and air by using suitable ratio in mixing chamber for combustion process.

Various studies on emulsion fuels reveal that they have various benefits, including improvement in combustion efficiency and a reduction in particulate matter (PM) and nitrogen oxide (NO<sub>x</sub>) emissions. Some of the researcher was discussed a variable on fuel properties of two and three-phase biodiesel emulsions (Cherng-Yuan Lin, 2007). The results showed that the burning of neat biodiesel produces the least amount of carbon

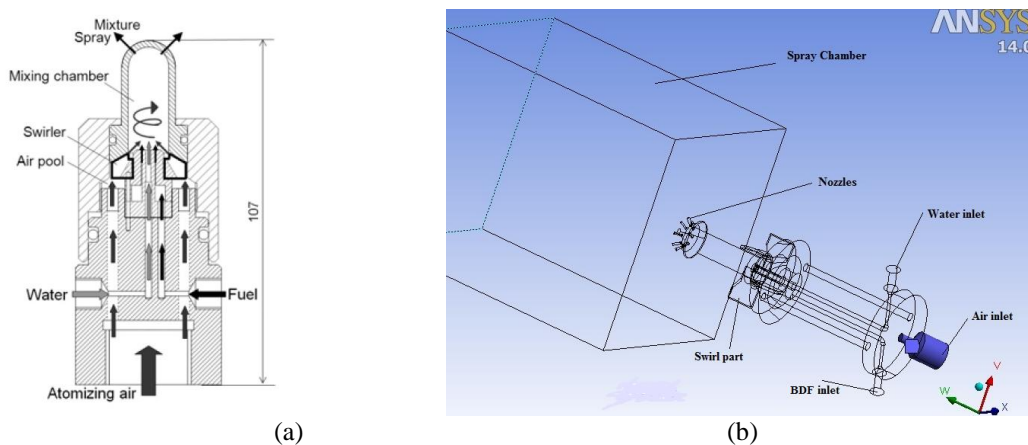
**Corresponding Author:** B. Manshoor, Department of Plant and Automotive Engineering, Universiti Tun Hussein Onn Malaysia, Malaysia.  
E-mail: bukhari@uthm.edu.my

residue. In addition, the existence of water content causes an increase in specific gravity and kinematic viscosity of the biodiesel emulsion. Y. Kidoguci *et al.* (2011) introduced an external premix injector in burner system for rapid mixing. The premix injector involves diesel fuel, water, and air with the different percentage of fraction. Fuel water emulsion is an emulsion of water in the base doped diesel fuel (here used BDF). It consists of water mixed with certain chemical, including surfactants and BDF. Burning emulsion has been widely accepted as an effective method to reduce NO<sub>x</sub> and particulate matter (PM) in BDF exhaust gas emissions (A. Lif *et al.*, 2006). The use of emulsions of water also improves the efficiency of combustion. Besides that, it has been shown in many previous studies where the emulsion can reduce the flame temperature thus reducing NO<sub>x</sub> emissions. The water also improves atomization and mixing caused by the micro emulsion droplets. This is due to the increased momentum jet fuel vaporized give more air to get into the jet fuel (C. D. Bolszo *et al.*, 2010).

### Premix Injector:

Premix injector is a device for mixing fuel with air in an internal combustion engine. It has become the primary fuel delivery system used in automotive petrol engines. The injection process in a diesel based engine can be studied experimentally or by means of simulation work. The initial design of injector is only for fuel-air mixing. Although this design has an advantages in completing combustion in the nozzle chamber, the disadvantages is the combustion were proposed to an unhealthy emission for environment. To overcome the problem, a new design was introduced which is consist of air, fuel and water. Figure 1 (a) shown the concept of injector for this simulation and the design concept it was introduced by Y. Kidoguci *et al.* (2011). For the simulation work, the model was developed and a combustion chamber was added as additional domain for simulation work. The combustion chamber added since the characteristic of the premix injector will study based on mixing results in the combustion chamber.

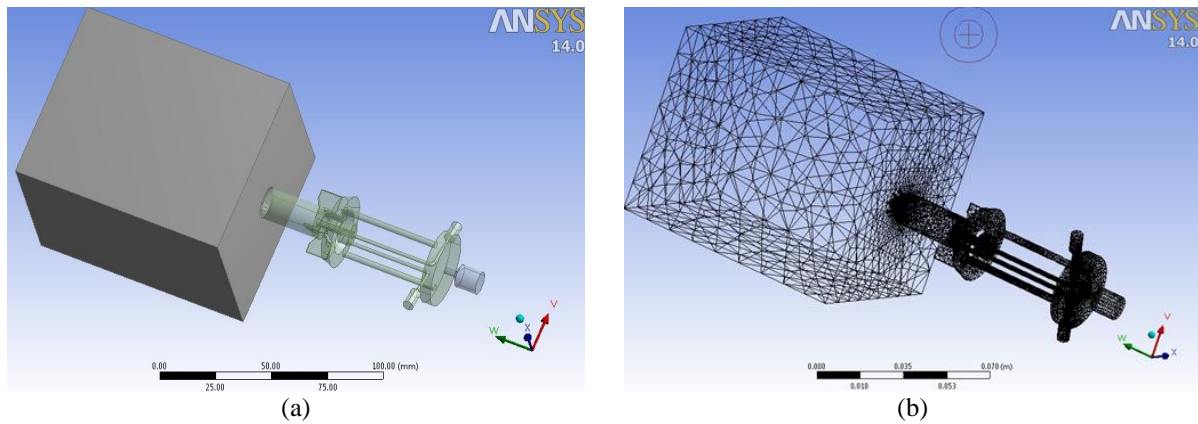
In order to study the characteristics of the premix injector, an external spray pattern using the premix injector will be carried out in this research. Besides that, the scope that will be study here are the mixing of BDF with three compositions of water which is 0% water, 10% water and 20% water. The parameter that will be look further is the velocity in the combustion chamber for each fluid, the pressure occur in the combustion chamber and spray pattern for different configuration of the BDF-air-water. During the simulation work, Reynolds number will set at a  $4.5 \times 10^5$  and the ambient temperature is 25<sup>0</sup>C with density of air,  $\rho = 1.184 \text{ kg/m}^3$ .



**Fig. 1:** (a) Design concept of premix injector (b) Components of premix injector with spray chamber

### Simulation Work:

The simulation work done by using commercial CFD software package ANSYS CFX. An unstructured mesh was adopted for the spatial discretization of the domain, ensuring a better refinement in regions where higher gradients were expected. The model and meshed model were shown in Figure 1. The meshing was meshed in three types which are geometry mesh, connection mesh, and default mesh. For geometry mesh, it divide the solid bodies for premix injector at 11 bodies. The connection mesh is setup between head of mixing chamber and spray chamber and sweep mesh was setup for default mesh. For the mesh, the number of nodes and elements are 81280 and 409449 respectively. The turbulent at all boundaries condition has been set to 2% and back flow turbulent intensity at all boundaries has been set to 5%. The Semi-Implicit Method for Pressure Linked Equation (SIMPLE) method is used to reach a convergent solution set (H.K. Versteeg, 2007). The accuracy of using computational fluid dynamics as a tool for the prediction of flow features depend on the choice of the turbulence model. The standard k- $\epsilon$  model is the most common turbulence model and it is routinely used for general analysis.



**Fig. 2:** (a) Simulation model of premix injector (b) Meshing model

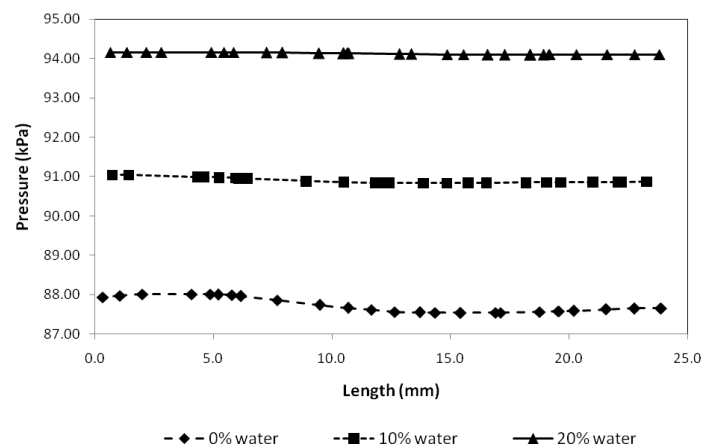
The boundary condition needed to set up before solver. Every inlets, outlet, body and symmetry will be defined in this stage. Pressure, velocity, and turbulent method should be considered in these boundary conditions. The boundary conditions have been selected based in the premix injector specifications. The boundary condition for W0 BDF type was shown in Table 2. The outlet set up as the opening zero Pascal for the flow domain. The walls are set to no slip boundary condition.

**Table 2:** Boundary Condition for inlet, opening, and walls.

Analysis Type	Boundary Condition	
	Steady State	
Inlet	Flow Regime	Subsonic
Air	Mass Dan Momentum	9.55E -4 kg/s
BDF (W0)	Mass Dan Momentum	6.67E -5 kg/s
Opening	Flow Regime	Subsonic
	Mass Dan Momentum	0 Pascal
Wall	Wall Roughness	Smooth wall
Turbulent	K-Epsilon	Scalable

## RESULT AND DISCUSSION

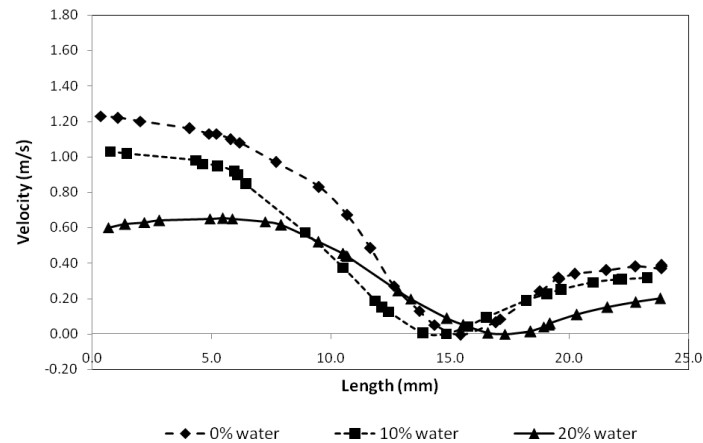
For the premix injector simulation, there are three things that interest to discuss here which is the velocity of each fluid, pressure occurred in the mixing chamber and also the spray pattern. As mentioned before, there are three compositions of water used in the emulsion here which are without water (0% of water), 10% of water and 20% of water. Figure 3 showed the pressure occurred in the combustion chamber during the simulation. As can see in the figure, high percentage of water used gives the highest pressure in the combustion chamber.



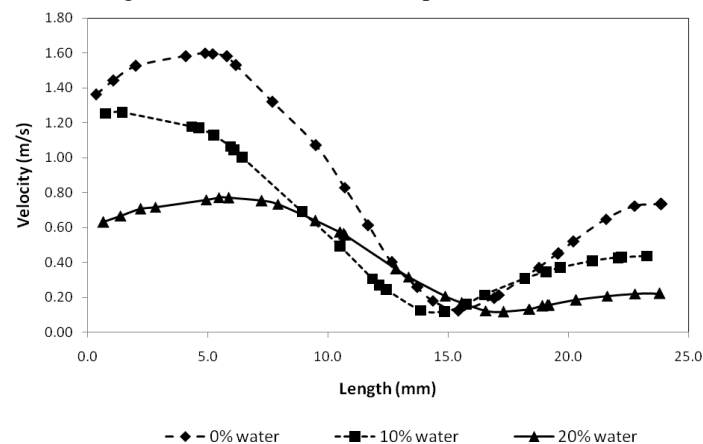
**Fig. 3:** Pressure occurred in mixing chamber for different composition of water.

As the pressure is inversely proportional to the velocity, the result of the velocity for each fluid (air and BDF) in the combustion chamber was shown in Figure 4 and 5. From the figure, it clearly showed that the

pattern of velocity is nearly similar except the velocity for the BDF is higher compare to the velocity of the air. The velocity of the fluids here decrease at the 15mm in the combustion chamber due to orifice effect form the injector. The result is very important since the velocity of both fluids give a view on what is the weak of the injector system here. In order to create a better mixing, hence improve the performance of the injector, the velocity for both fluids need to be control so that the different not so high. As mentioned earlier, this is an initial work to test the capability of the premix injector for BFD-air-water mixing in open burner system. These results also represent an initial data that hopefully can be use for further investigation in premix injector for BFD-air-water mixing in open burner system.



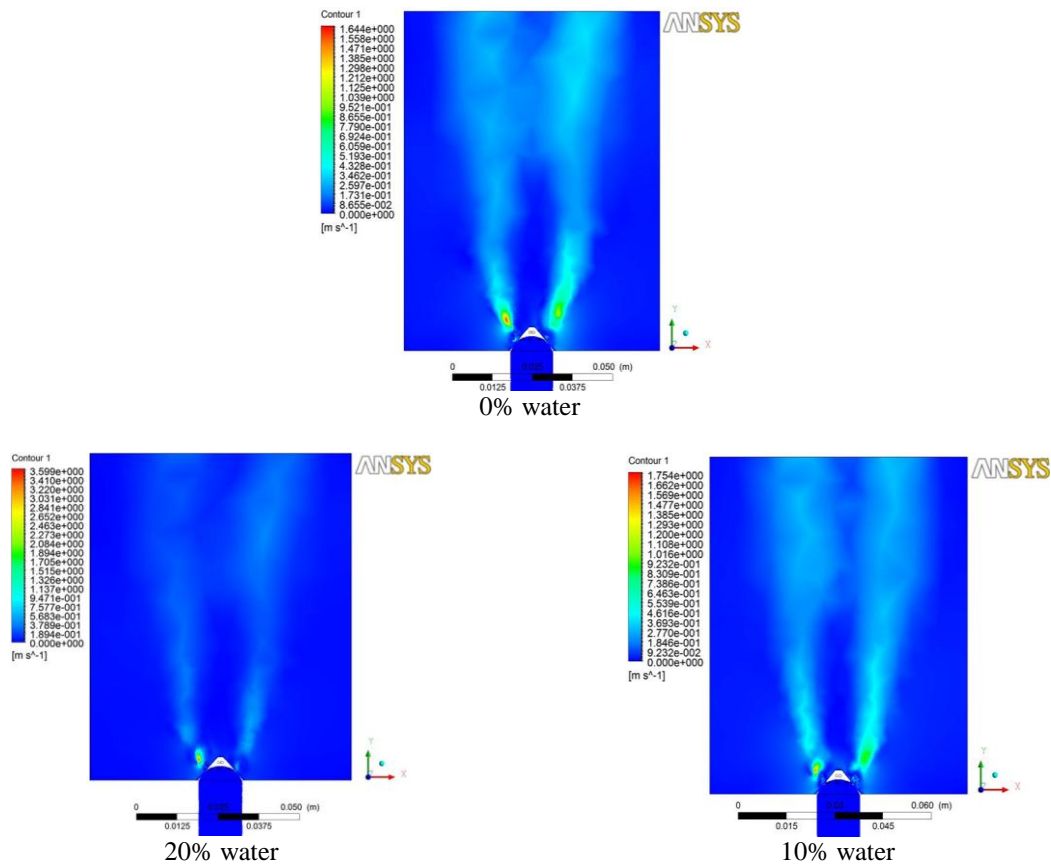
**Fig. 4:** Velocity of air in the mixing chamber for different composition of water.



**Fig. 5:** Velocity of BDF in the mixing chamber for different composition of water.

Figure 6 represent the contour of velocity of BDF in the combustion chamber. Besides that, from the contour also we can see the spray pattern for the different composition of water added into the injector. For the spray pattern, a volume fraction for each type of BDF gives different of fluid flow pattern in the premix injector. The presence of water can affect the shape of spray pattern because of the large value of water which is  $1000\text{kg/m}^3$ . From the figure, it showed that the spray pattern occurred are in the form of hollow cone. Each combination of BDF-water gives a different spray angle and penetration due to the water content.

For the velocity of each BDF studied here, the figure clearly showed that the BDF without water give the higher velocity. It is followed by the BDF with 10% of water and the lowest velocity occurred in the combustion chamber is for the BDF with 20% of water. In this study, as the BDF with 0% water will become as a benchmark for other two BDF, the contour showed that the BDF with 10% water give a nearly similar results with the benchmark, it can be conclude that the method to add the water to the injector did not give much effect to the performance of the injector. However, it will give an effect if the amount of water is increase.



**Fig. 6:** Velocity contour of BDF for different composition of water.

### Conclusion:

The flow mixing in premix injector was investigated to evaluate characteristic of the injector. From the simulation, high pressure occurred in the combustion chamber for the BDF that contain high percentage of water while for the velocity, the composition with the lowest percentage of water give higher velocity in the combustion chamber. For the spray pattern also give a good results where the different spray were occurred for the different composition of water in BDF for the mixing. Last but not least, we noted here that this is only initial work to test the capability of the premix injector for BFD-air-water mixing in open burner system. These results also represent an initial data that hopefully can be use for further investigation in premix injector for BFD-air-water mixing in open burner system.

### ACKNOWLEDGEMENT

The author is grateful to the Universiti Tun Hussein Onn Malaysia (UTHM) and Ministry of Higher Education (MOHE), Malaysia for supporting this research under the MTUN-CoE research grant, vote C010.

### REFERENCES

- Amir Khalid, K. Hayashi, Y. Kidoguchi, T. Yatsufusa, 2011. Effect of air entrainment and oxygen concentration on endothermic and heat recovery process of diesel ignition. SAE/JSAE International Powertrains, SAE Technical Papers No. 2011-01-1834, Japan.
- Honjo, K., 1996. Technology R & D for technology to solve global warming, 59: 218-220.
- Sies, M.F., N. Mustafa, H. Zakaria, H. Salleh, B. Manshoor and Amir Khalid, 2014. Development of the Premixing Injector in Burner System. Applied Mechanics and Materials, 465: 296-301.
- Nazri, M., M. Jaafar, Y.A. Eldrainy and M.H. Asril, 2011. Experimental investigation of spray characteristics of refined bleached and deodorized palm oil and diesel blends using phase doppler particle analyzer. International Journal of the Physical Sciences, 6(29): 6674-6680.
- Cherng-Yuan Lin, S.A.L., 2007. Effects of emulsification variables on fuel properties of two and 3 phase biodiesel emulsion. Fuel Processing Technology, 210-217.

Kidoguchi, Y., T. Yatsufusa and D. Nakagawa, 2011. Improvement of emissions and burning limits in burner combustion using an injector on the concept of fuel-water internally rapid mixing. Proceedings of the European Combustion Meeting.

Lif, A. and K. Holmberg, 2006. Water-in-diesel emulsions and related systems. Advances in colloid and interface science, 126(2): 231-239.

Bolszo, C.D., A.A. Narvaez, V.G. McDonnell and W.A. Sirignano, 2010. Pressure swirl atomization of water-in-oil emulsions. Department of mechanical and aerospace engineering University of California , Irvine Irvine USA.

Versteeg, H.K. and W. Malalasekera, 2007. An Introduction to Computation Fluid Dynamics, The Finite Volume Method 2nd. Edi.