www.arpnjournals.com

EFFECT OF HIGH PRESSURE ON THE FLOW CHARACTERISTICS OF INJECTOR USING COMPUTATIONAL FLUID DYNAMICS (CFD)

Amir Khalid[1], Adiba Rhaodah Andsaler[1], Norrizam Jaat[1]

[1] Combustion Research Group (CRG), Centre of Energy and Industrial Environment Studies (CEIES),
Universiti Tun Hussein Onn Malaysia (UTHM), 86400 Parit Raja, Batu Pahat, Johor,Malaysia
E-Mail: amirk@uthm.edu.my, adiba_rhaodah@yahoo.com,

ABSTRACT

The rapid compression machine was used for the wide observation region. A number of researchers were conducted the experiments to study on the mixture formation and combustion process was performed in Rapid Compression Machine (RCM). This research shows the injector nozzle geometries play a significant role in flow characteristics, atomization and formation of fuel-air mixture in order to improve combustion performance, and reduce some pollutant products from internal combustion engine system. The aim of this research is to determine the effects of high pressure on flow characteristics of the injector by using Computational fluid dynamics (CFD). Multiphase of volume of fluid (VOF) cavitating flow inside nozzles are determined by means of transient simulations and two-fluid approach is used for performing mixing of Coconut palm oil and air. Nozzle flow simulations resulted that cavitation area is strongly dependent on the nozzle conical injector. Conical hole with k-factor of 2 provides higher flow velocity and turbulent kinetic energy. The results show that the premix injector nozzle conical shape gives impact to the flow characteristics and indirectly affects the emission of the internal combustion engine system.

Keywords: Computational fluid dynamics (CFD) ⋅ Rapid Compression Machine (RCM) ⋅ Nozzle flow ⋅

INTRODUCTION

In recent years, biodiesel is seen as a capable alternative to conventional diesel due to its desirable features such as biodegradable, renewable, sustainable and carbon neutral. Rising petroleum prices, increasing threat to the environment from exhaust emission and global warming have caused an intense international interest in developing alternative non petroleum fuels due to gradual running down of the world petroleum reserve. The desire to have suitable replacement, alternative or an entirely different source of fuel from the presently existing fossil fuel has being very imperative [1].

Biodiesel can directly replace petroleum diesel and can be used in diesel engine without the requirement of any major modifications, reducing the country's dependence on imported oil indirectly. Biodiesel produced from either vegetable oil or animal fats which consists of long chain mono-alkyl esters derived through transesterification process. Diesel sprays can be studied by carrying out controlled experiments or deriving mathematical models or sub-models that isolate the relevant sub-processes [1].

Diesel engine is an internal combustion engine that uses the heat of compression to initiate ignition and burn the fuel that has been injected into the combustion chamber and its combustion progresses by natural heterogeneous [2]. The rapid compression machine was applied in order to keep the wide observation region. A number of researchers were conducted the experiments to study on the mixture formation and combustion process was performed in Rapid Compression Machine (RCM). RCM is an instrument designed to stimulate a single compression event of an engine cycle of internal combustion engine. The influence of the injection process in the combustion chamber has to be considered in the analysis of spray formation and propagation. The flame behaviour and turbulence intensity in the combustion field may play an important role, and it is important to clarify these flow characteristics for a better understanding of the mechanism of combustion improvement with high pressure injection. The mixture formation during ignition delay period is important process because ignition is controlled by physical process caused by multi-hole injection and air motion and chemical process of fuel decomposition and oxidation [3].

Computational fluid dynamics (CFD) is a popular approach to study the injection of fuel spray. CFD is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The result of CFD analysis is relevant engineering data such as conceptual studies of new designs, troubleshooting, detailed product development and redesign. CFD analysis complements testing and experimentation is reduces the total effort

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

required in the laboratory [4]. In particular, mixture formation of biodiesel during early stage of burning is important process because ignition is controlled by physical process caused by multi-hole injection, air motion, chemical process of fuel decomposition and oxidation [5-6]. . In addition, velocity components act on the nozzle outlet plane are significant and appreciated since it contribute to the disintegration of the mixing fluids into droplets [7]. Implosions of cavitation bubbles inside the nozzle holes increase when the turbulence kinetic energy increases and contributes to the further break-up of the spray [8-9].

Therefore, the aim of this research is to determine the influence of the pressure by analyzing the nozzle flow characteristics of the injector.

METHODOLOGY

The injector model used in the simulation is based on the injector design in the experimental work. Figure 1 shows the internal geometry of injector in computational domain which is one six of the geometry is used. This study involves one geometry that use conical hole with k-factor 2. This simulation work is conducted using ANSYS Fluent 14.5 software. Generally, the applications of simulation in ANSYS FLUENT 14.5 involve three main stages that have been considered which are pre-processing, solving and post-processing.
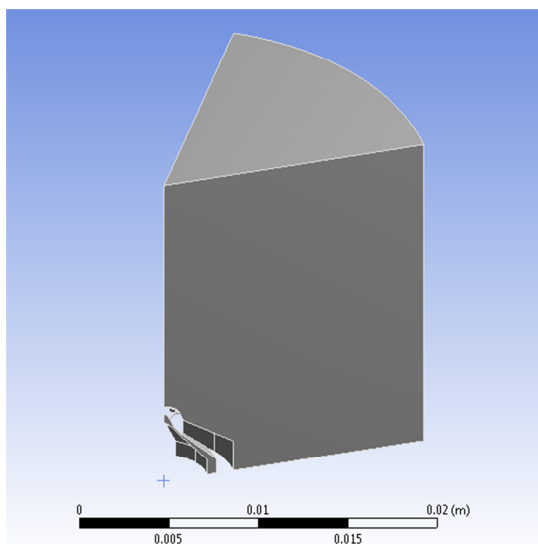

Figure 1: Internal geometry of injector in computational domain

Boundary condition for nozzle flow simulation

A transient multiphase simulation using Volume of fluid model was used in this simulation. The solid wall of the model was assigned as the basic boundary condition of the domain for this fluid flow. The surface of the injector is considered as stationary. Inlet was selected as

pressure inlet and outlet was selected at two sides as pressure outlet. Once the injection pressure enters the domain, the velocity of the fluid, pressure, or temperature can be known. In addition, this case consists of different conductivity for coconut palm oil and air. The pressure output at 4MPa, 3MPa and 2MPa were set at 100MPa injection pressure.

Moreover, the boundary conditions for three different nozzle geometries parameter cases were similar throughout whole simulation process.

RESULTS AND DISCUSSION

Grid independence study

The numerical result of nozzle flow is tested by grid independence study. This focused on the effects of flow along the conical injector nozzle. It is reported that grid size has a significant effect on the convergence and predicted results [10-11]. The number of type of grid sizes according to the number of elements and nodes are shown in Table 1. The table shows the ascending levels of grid sizes from coarse to fine corresponding to the value of elements and nodes. Figure 2 shows the grid sensitivity is tested on the model.

The average values are calculated and required to compare with the original velocities for each set of different grids in order to get the percentage of errors between the simulated velocities and the average velocities. The most preferable grid is selected by attaining the best fit of percentage differences to the average velocities. Therefore, an average percentage of coarse medium which representing the percentage error of 2.072% due to its overall errors are the lowest compared to others grid level. This meshing scheme is selected to be used in further simulations.

Table 1: Type of grid size and number of elements and nodes in each of grid.

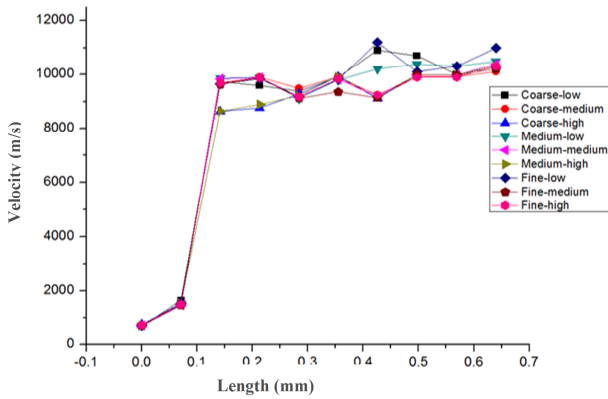| Grid size	Elements	Nodes
Coarse-low	64312	12938
Coarse-medium	64069	12898
Coarse-high	64299	12937
Medium-low	95479	19044
Medium-medium	95990	19144
Medium-high	96230	19172
Fine-low	197173	38389
Fine-medium	198341	38582
Fine-high	198597	38634

www.arpnjournals.com

Figure 2: Velocity inside the nozzle against the
nozzle length with different mesh density

pressure of 4MPa gives the highest turbulence kinetic energy. Implosions of cavitation bubbles inside the nozzle holes will increase when the turbulence kinetic energy increases.

Also, implode of cavitation bubbles give further break-up of the spray and it could leads to finer droplets of fuel which can fasten the process of evaporation of fuel. Figure 4 shows the graph of velocity against the length of nozzle spray with different ambient pressure. From the graph, the velocity obtained by 2MPa ambient pressure is higher than 3MPa and 4MPa ambient pressures.

Nozzle flow simulation

Flow of the spray nozzle is determined by the velocity of spray nozzle. Length of the spray nozzle injector is 0.64mm. Figure 3 shows the flow differences at different ambient pressure in which are 4MPa, 3MPa and 2MPa. From Figure 3(a), it shows that the ambient
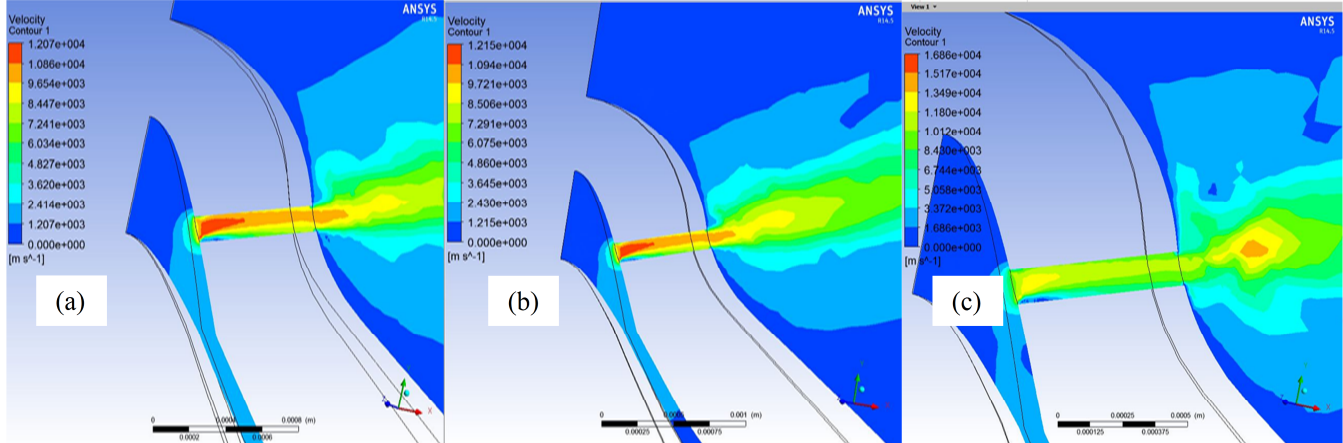

Figure 3: (a) Ambient pressure at 4MPa, (b) Ambient pressure at 3MPa, (c) Ambient pressure at 2MPa.
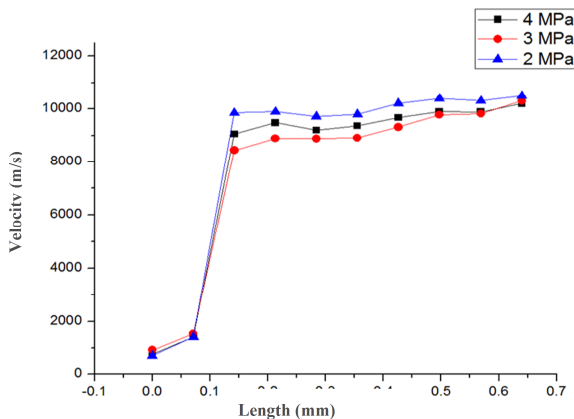
www.arpnjournals.com

Figure 4: Velocity against the length of spray nozzle injector.

Different ambient pressures also give influence on the spray length and spray area indirectly. As the ambient pressure increased, the spray length will decrease. Equation 1 will be used in obtaining the spray area. Table 2 below shows the ambient pressures and value of spray length and spray area obtained.

Table 2 : Ambient pressures with spray length and spray area.

Ambient pressure(MPa)	Spray length (mm)	Spray area(mm^2)
2	24.9	115.12
3	23.8	112.57
4	21.1	103.56

Spray area (mm^2) : $2[\frac{1}{2}(l \times l \tan\frac{\theta}{2})]$ (equation 1)

ACKNOWLEDGEMENTS

CONCLUSION

The analysis shows that high injection pressure strongly influence the flow properties inside nozzle injector. Furthermore, it can be predicted that the spray injected from the nozzle outlet are also affected to spray length and spray area of the nozzle flow characteristics.

It is possible to believe that the concurrent phenomena are taken into account of few investigations. Therefore, the flow velocity encourage fuel-air mixing in the nozzle and rate of fuel injection by increasing fuel volume fraction.

REFERENCE

[1] Darshana S. Gaikwad , Ajay V., Study of Combustion Modeling In CI Engine with Biodiesel as an Alternative Fuel: A Review, 2014, The International Journal Of Engineering And Science (IJES), Volume 3, Issue 1, Pages 25-29,ISSN(e): 2319–1813, ISSN(p): 2319–1805.

[2] A. Khalid And Y. Kidoguchi, "Analysis of Diesel Combustion Using a Rapid Compression Machine and Optical Visualization Technique," Vol. 00, Pp. 1–2.

[3] A Khalid and T. Yatsufusa, "Analysis of Relation Between Mixture Formation During Ignition Delay Period and Burning Process In Diesel Combustion," Sae Technical Paper, 2009.

[4] M. Petkov and M. Roman, "Cfd Training Manual." P. 43.

[5] Amir Khalid, M.D. Anuar, Yusri Ishak, B. Manshoor, Azwan Sapit, Mutalib Leman, Izzuddin Zaman, Emissions characteristics of small diesel engine fuelled by waste cooking oil, MATEC Web of Conferences, Volume 13, 2014, Article number 06006, DOI: 10.1051/matecconf/20141306006

[6] Amir Khalid, Keisuke Hayashi, Yoshiyuki Kidoguchi , Tomoaki Yatsufusa, Effect of air entrainment and oxygen concentration on endothermic and heat recovery process of diesel ignition, (2011) SAE Technical Papers, DOI: 10.4271/2011-01-1834.

[7] J. M. Desantes, R. Payri, F. J. Salvador, J. Manin, And U. P. De Valencia, "Influence On Diesel Injection Characteristics And Behavior Using Biodiesel Fuels," 2009.

[8] Michele Battistoni, C.N.G., Numerical analysis of injector flow and spray characteristics from diesel injectors using fossil and biodiesel fuels. Applied Energy 97 (2012), December 2011, 656–666.

[9] Tingwen Li, A.G., Sreekanth Pannala, Mehrdad Shahnama, Madhava Syamlal, CFD simulations of circulating fluidized bed risers, part I: Grid study. Powder Technology 254, 2014: p. 170–180.

[10] Amir Khalid, N.Tamaldin, M. Jaat, M. F. M. Ali, B. Manshoor, Izzuddin Zaman, Impacts of biodiesel storage duration on fuel properties and emissions, Procedia

www.arpnjournals.com

Engineering, volume 68, 2013, Pages 225–230, Elsevier, 2013, DOI: 10.1016/j.proeng.2013.12.172

[11] J Benajes, JV Pastor, R Payri, AH Plazas, Analysis of the influence of diesel nozzle geometry in the injection rate characteristics, Journal of Fluids Eng, 2004(126): p.63–71

[12] A. Adam, N. Inukai, Y. Kidoguchi, K. Miwa, and S. Miyashiro, "*A study on Droplets Evaporation at Diesel Spray Boundry During Ignition Delay Period.*" 2007.

[13] M. Jaat, A. Khalid, B. Manshoor, S. M. Basharie, and H. Ramsy, "*Review of the investigation of fuel-air premixing and combustion process using rapid compression machine and direction visualization system,*" vol. 465-466, pp. 265-269, 2014.