Comparison On Piston Bowl Shape Effect To Diesel Spray Development

Azwan Sapit\textsuperscript{1,2}, Mohd Azahari Razali\textsuperscript{1}, Akmal Nizam Mohammed\textsuperscript{1,2}, Mohd Faisal Hushin\textsuperscript{1}, Azmahani Binti Sadikin\textsuperscript{1}, Md Norrizam Mohmad Ja‘at\textsuperscript{1}, Amir Khalid\textsuperscript{1}

\textsuperscript{1}FAST Research Group, Universiti Tun Hussein Onn Malaysia, Parit Raja, Johor.
\textsuperscript{2}Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Parit Raja, Johor.

E-mail: azwans@uthm.edu.my

Abstract. Piston bowl geometry plays an important role on the combustion characteristics of diesel engine. There are various design of piston bowl in which each utilize the shape geometry to obtaining the specific required combustion characteristics. This objective of this study is to compare the effect of certain piston bowl shapes, namely Toroidal and Flat Bottom to diesel spray development. Simulation were done using ANSYS FLUENT 16.1 software Computing Fluid Dynamics (CFD). The simulation was performed on different injection pressure of 40 MPa and 100 MPa, with the ambient temperature in the combustion chamber that holding the piston is at 500K and 900K. Results showed that if the pressure and ambient temperature increases, the spray body expand outward from the spray center axis with wider spray cone angle. In addition, the geometry shape of the piston bowl influences the spray velocity distribution and the spray propagation path, indirectly effect the spray area and mass fraction distribution.

1. Introduction

Fuel injection strategy is important to the diesel engine because it influence the performance and emission. During combustion, fuel is sprayed in liquid form into a gaseous environment through a nozzle such that the liquid, through its interaction with the surrounding gas and by its own instability, breaks-up into droplets [1]. Spray characteristics is known to significantly affect the combustion and emission processes in diesel engines. By optimizing spray characteristics, the unwanted emissions from the diesel engine which are mainly NOx and PM can be minimized [2-4]. Thus study on diesel sprays characteristics is surely very beneficial in the effort to reduce dangerous and unwanted emission, and in the work of maximizing efficiency from efficient usage of fuel and combustion[5].

2. Literature Review

There are many studies that deal with diesel engine spray characteristics, due to the nature of the phenomenon; atomization of diesel and fuel mixing are very complex topic. It is understand that the injection pressure has significant effect on spray liquid penetration [6]. The spray tip penetration gets longer as the injection pressure increases, until certain threshold. This result is related to both higher
quantity and higher velocity of the droplets at higher injection pressures [7]. Proportional to injection pressure, the spray penetrates faster at higher injection pressures [8].

The break-up length characterizes a point of discontinuity, where the spray changes from a zone of liquid (bulk liquid, or interconnected ligaments and droplets), to a finely atomized regime of droplets [9]. After the disintegration of the liquid column emerging from the nozzle, the generated droplets may further break-up into smaller ones as they move into the surrounding gas [10]. Development of this deformation leads to break-up into smaller droplets. The forces associated with dynamic pressure, surface tension and viscosity control the break-up of a drop. However, CFD is a tool being used to predict the some aspects of the study that is too difficult to be done through experiment [11]. In this research, the characteristics of diesel spray are investigated focusing on changing ambient temperature and injection pressure with different bowl shape geometry diameter by using Computational Fluid Dynamics.

3. Methodology

This study use Computational Fluid Dynamics software, ANSYS Workbench 16.1. In CFD software, the process for the numerical analysis is divided into three stages which are pre-processing, solver and post-processing. Table 1 show the various parameter uses in this simulation, and Figure 1 show the piston bowl shapes.

| Spray Pressure | Ambient Temperature |
|----------------|---------------------|
| 40 Mpa         | 500 K               |
| 40 Mpa         | 900 K               |
| 100 Mpa        | 500 K               |
| 100 Mpa        | 900 K               |

For this study, 2 type of piston bowl shape were used, namely flat bottom and toroidal. The shape the geometry of the piston bowl is drawn by using Solidwork It is then imported into the DesignModeler (DM) of ANSYS workbench as showed in Figure-2 (a). The geometry is then need to be converted into computational mesh as in Figure-2 (b). Solver will then run the calculation for results.
The last step in ANSYS Fluent software is post-processing where the animation, production of graphics and report can be made through the post-processing and the result of fluid dynamics data can be presented. In addition, the post-processing can also shade the surface of the transparent, path line, vector plots, contour plot, and custom field definition variable and scene construction.

4. Discussion and Results

In result and discussion, simulations conducted at different injection pressure of 40 MPa, 100 MPa, and the temperature at 500 K and 900 K. The aim is to identify differences in the form of spray development, spray areas, and spray droplet velocity on each different piston bowl shape.

![Figure 3. Scale velocity of diesel fuel spray](image)

Spray velocity scale is shown in figure 3 below. Red indicates the velocity of diesel spray over 500 m/s with blue indicates minimum spray movement occurs and the velocity is nearing 0 m/s. Figure 4, show the velocity contour of diesel spray on (a) flat bottom and (b) toroidal shaped piston bowl. Images shows spray development at 0.1 milliseconds to 0.3 milliseconds after the start of diesel fuel injection (SOI).

Results obtained in figure 4 show interesting spray development. Injection pressure, ambient temperature and the shape of the piston bowl clearly affect the spray characteristics and development. In general, at the initial injection time (0.1 millisecond), it can be seen that the spray penetrate the chamber with mostly the spray center (liquid core region) having the maximum velocity and the outer region (droplet region) which rapidly interact with the surrounding air has a lower velocity due to aerodynamic force (air resistance) slowing the spray particle. As time progress, the spray core penetrates longer and impinged the piston wall. This is indicated by the longer maximum velocity contour when compared to 0.1 millisecond. Also, the spray body expand wider across the center axis. This is more significant with the increase in injection pressure and ambient temperature. It can be seen that at 100MPa injection pressure and 900K ambient temperature, the spray body has the widest width when compared to other condition, which indicate that high injection pressure and ambient temperature increase spray cone angle, as seen in the increase of the spray width.

Detailed comparison on spray development between the flat bottom and toroidal piston shape indicate that after the spray impinged to the piston wall, the spray will body will forced to follow the wall curvature. In the case of flat bottom piston, the spray impinged at the piston wall and the spray core continue heading downward along the piston wall. In comparison, in the case of toroidal, after impingement, the spray core seem to divided into 2, with each separated spray heading upward and downward moving along the piston wall. This is clearly the effect of different piston shape geometry.
The spray development characteristics displayed for each piston bowl shape will definitely influence the effect the fuel combustion in the chamber. The flat bottom bowl shape design has longer spray travel distance before wall impingement. Additionally, after impingement, the spray move downward to the piston bowl bottom section. These characteristics are believed to reduce the SOF generation due to reduction of fuel spillage over and outside of the piston bowl. This is especially helpful if the fuel used required long ignition delay.

In comparison, the spray in toroidal has a shorter travel distance before wall impingement. In addition, after impingement, the spray divided into 2, with each moving upward and downward. In general, the spray forced movement due to the piston bowl shape promote turbulence and mixing. This is beneficial for fast and rapid atomization, but the bowl shape also make the spray to move up and spill over the piston bowl, which will increase SOF generation. Thus, the use of toroidal piston bowl shape must be consider thoroughly, and maybe better suitable for small diesel engine that operate in high rpm (fast engine) which, require very rapid fuel atomization paired with fuel that has a short ignition delay.

5. Conclusion

The study has provided significant finding on the insight of spray characteristics when used with the 2 types of piston bowl shape. Each piston bowl shape influence the diesel spray development differently and with considerable effect. The flat bottom bowl shape design has longer spray travel distance before wall impingement. Additionally, after impingement, the spray move downward to the piston bowl bottom section. In comparison, the spray in toroidal has a shorter travel distance before wall impingement. In addition, after impingement, the spray divided into 2, with each moving upward and downward. The usage of each piston bowl design should be paired with a suitable fuel delivery system.
and strategy to promote the desired combustion characteristic optimum for the required engine operation.

References
[1] Adam, A., Inukai, N., Kidoguchi, Y., Miwa, K. Et Al., (2007), "A Study on Droplets Evaporation at Diesel Spray Boundary during Ignition Delay Period," SAE Technical Paper 2007-01-1893.
[2] Sapit, A., Nagayasu, S., Tsuboi, Y., Nada, Y., & Kidoguchi, Y. (2011). “A study on improvement of diesel spray characteristics fueled by rape-seed oil”. SAE International Journal of Fuels and Lubricants, 5(2011-32-0561), 529-539.
[3] Khalid, A., Tamaldin, N., Jaat, M., Ali, M.F.M., Manshoor, B., Zaman, I., (2013), “Impacts of Biodiesel Storage Duration on Fuel Properties and Emissions”, Procedia Engineering, Volume 68, Pages 225-230, Elsevier Ltd, DOI: 10.1016/j.proeng.2013.12.172.
[4] Sapit, A., Yano, T., Kidoguchi, Y., & Nada, Y. (2013). “Effect of Wall Configuration on Atomization of Rapeseed Oil Diesel Spray Impinging on the Wall”. Applied Mechanics and Materials (Vol. 315, pp. 320-324). Trans Tech Publications.
[5] Khalid, A., Hayashi, K., Kidoguchi, Y., Yatsufusa, T., (2011), “Effect of air entrainment and oxygen concentration on endothermic and heat recovery process of diesel ignition”, SAE Technical Papers, Society of Automotive Engineers of Japan, Inc. and SAE International, DOI: 10.4271/2011-01-1834.
[6] Khalid, A., (2013), “Effect of Ambient Temperature and Oxygen Concentration on Ignition and Combustion Process of Diesel Spray”, Asian Journal of Scientific Research, Volume 6, Issue 3, Pages 434-444, Asian Network for Scientific Information, DOI: 10.3923/ajsr.2013.434-444.
[7] Camille Hespel, Jean-Bernard Blaisot, Xandra Margot, S. Patouna, Armelle Cessou, Bertrand Lecordier, (2010), “Influence of Nozzle Geometry on Spray Shape, Particle Size, Spray Velocity and Air Entrainment of High Pressure Diesel Spray”, THIESEL.
[8] V. Roisman, L. Araneo, and C. Tropea, (2007), “Effect of Ambient Pressure on Penetration of a Diesel Spray,” International Journal of Multiphase Flow, Vol. 33, No. 8, Pp. 904–920.
[9] Simon Martinez – Martinez, Fausto A. Sanchez – Cruz, Vicente R.Bermudez And Jose M.Riesco-Avila, (2010), “Liquids Spray Characteristic In Diesel Engine”, In: Daniela Siano (Ed.), Fuel Injection. Intech: Sciyos. 20 – 48 Pages.
[10] Ozg"Ur O’guz Tas Kiran, (2011), “Experimental Study on Diesel Spray Characteristics and Autoignition Process”, Hindawi Publishing Corporation.
[11] Mohammed, A. N., & Ismail, F. (2013). “Study of an entropy-consistent Navier–Stokes flux”. International Journal of Computational Fluid Dynamics, 27(1), 1-14.