International Journal of Mechanical Engineering and Technology (IJMET)
Volume 10, Issue 2, February 2019, pp; 1301-1318 Article ID: IJMET_10_02_136
Available online at http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=10&IType=2
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

INTAKE VALVE LIFT EFFECT ON AIR-FUEL MIXING OF DUAL FUEL ENGINE

Muhammad Badrus Zaman
Head, Department of Marine Engineering
Institut Teknologi Sepuluh Nopember, Surabaya. Indonesia.

Semin
Professor, Department of Marine Engineering,
Institut Teknologi Sepuluh Nopember, Surabaya. Indonesia.

Nilam Sari Octaviani
Doctoral Program, Department of Marine Engineering,
Institut Teknologi Sepuluh Nopember, Surabaya. Indonesia.

Nurhadi Siswantoro
Laboratory of Marine Operation & Maintenance, Department of Marine Engineering,
Institut Teknologi Sepuluh Nopember, Surabaya. Indonesia.

Rosli Abu Bakar
Faculty of Mechanical Engineering
University Malaysia Pahang, Pekan, Pahang, Malaysia

ABSTRACT

The addition of CNG fuel in conventional diesel engines has increased the efficiency of engines and reduces diesel fuel consumption. The CNG fuel injected into the combustion chamber through inlet air system. The objective of this paper is to investigate of how airflows and CNG fuels are mixed and enter to the engine cylinder. This paper method is based on CFD simulation. The simulation of incoming airflow and CNG fuel on modified diesel engines that are developed with the addition of an inlet valve opening are expected to provide an overview of the effect of inlet valve lift on a dual fuel engine. The structural stress approach by varying the opening of the inlet valve during the default condition of 7.28 mm, 3.64 mm, 10.92 mm, and 14.56 mm. The investigation results shown that at the time of opening at 14.56 indicates the average increase air speed and increase the mass of airflow and fuel. Result shown that the power from the valve opening 14.56 mm has been increase.

Keywords: air fuel mixing; compressed natural gas; dual fuel engine; valve lift.

Cite this Article; Muhammad Badrus Zaman,Semin,Nilam Sari Octaviani,Nurhadi Siswantoro And Rosli Abu Bakar Intake Valve Lift Effect on Air-Fuel Mixing of Dual Fuel Engine *International Journal of Mechanical Engineering and Technology* (IJMET) 10(2), pp;1301-1318
http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=10&IType=2

1. INTRODUCTION

The importance parameters in diesel engine performance are geometric components for efficiency. Engine efficiency such as thermal efficiency, brake heat efficiency, mechanical efficiency, volumetric efficiency and relative efficiency [1-10]. A wide variety of port entrance geometry patterns are used to achieve this over the diesel size range.

The dual fuel technology has been more successfully used in engines with slow variation of load, where the engine speed varies in a small range. The engine ratings typically indicate the highest strength at which manufacturers expect their products to deliver robust strength, economy, reliability, and durability under service conditions [9-15].

Air-fuel mixing is very important in the internal combustion engine. The excellent in the air-fuel mixing has been increase combustion in the engine cylinder. The first effect has been increased the engine performance such as power, torque and efficiency. The second, the air-fuel mixing has been reduced the emissions [16-37]

The objective of this paper is to investigate the air-fuel flow and mixing in the intake system of duel fuel engine. The dual fuel is mean CNG and diesel fuel. The parameters to investigate is using CFD simulation. The numerical simulation is using GT-Power software to collect the internal boundary data. GT-Power is powerful numerical simulation software to investigate the internal combustion engine [17-21]. The CFD simulation is using 3D model. The software of CFD namely ANSYS. The detail investigation strategy is shown in research method.

2. RESEARCH METHOD

The research method the steps of the research investigation conducted. The first step is engine specification to collect the data of the engine for the investigation. The second step numerical modelling to collect the data of engine parameters for the internal boundary condition. The third is CFD modelling to investigate the air-fuel flow and mixing.

2.1 Engine Specification

Duel fuel engine is based on the principle of spark ignition engine, where the use of diesel engine to ignite gas fuel as a step ignition or ignition. In the gas engine mixture of gas and clean air enter the combustion chamber first when the suction step. Then when the compression before reaching TDC gas fuel is injected into the pre-chamber, inside the gas pre-chamber is switched on by diesel fuel then burns first before entering to the main chamber or combustion chamber.

In this step, the data collection and specification of Yanmar TF85 MH diesel engine. Which has been be used to load the modelling appropriately. In this paper data taken from measurement is valve, combustion chamber, intake port of Yanmar TF85 MH. The measurements covered the overall dimensions of the Yanmar diesel inlet valve TF85 MH. Data input in this research is obtained from manual measurement and engine specifications, the dimensions are presented in Table 1. Another data input is taken from engine simulation software, GT Suites. Data consists of internal boundary conditions is shown in Table 2.

Table 1 Engine Specification

Parameters	Value
Cylinder Bore	85 mm
Cylinder length	87 mm
Intake valve diameter	32.5 mm
Intake valve length	84.1 mm
Exhaust valve diameter	27.5 mm
Length of exhaust valve	84.1 mm
Compression ratio	1:16

Table 2 Interal Boundary Condition

Parameters	Value
Airflow rate	68.26 m / s
Pressure inside cylinder	68.232 pa
Overlap valve	0.1 mm

2.2 Numerical Modelling

Duel fuel engine is based on the principle of spark ignition engine, where the use of diesel engine to ignite gas fuel as a step ignition or ignition. In the gas engine mixture of gas and clean air enter the combustion chamber first when the suction step. Then when the compression before reaching TDC gas fuel is injected into the pre-chamber, inside the gas pre-chamber is switched on by diesel fuel then burns first before entering to the main chamber or combustion chamber.

Numerical modelling is suing GT-Power software such as shown in Figure 1. The duel fuel engine is based on the principle of spark ignition engine, where the use of diesel engine to ignite gas fuel as a step ignition or ignition. In the gas engine mixture of gas and clean air enter the combustion chamber first when the suction step. Then when the compression before reaching TDC gas fuel is injected into the pre-chamber, inside the gas pre-chamber is switched on by diesel fuel then burns first before entering to the main chamber or combustion chamber.

The modelling process is carried out to conduct research and investigate the airflow and camshaft stress when variable valve lift is used on a single cylinder engine. Modelling is done by creating internal combustion model and camshaft of gas engine in 3D with Autodesk Inventor such as shown in Figure 2. Model making is based on engine data obtained from GT-Power. The drawing is done by drawing Bore diameter, intake valve depiction, exhaust valve, intake pipe diameter and drawing of exhaust pipe [38]. In this process has been adjusted at the time of modelling depiction and has been modified valve lift that has been be used in the calculation process later that is 3.64 mm, 7.28 mm, 19.92 and 14.56 mm which later this study has been be compared the speed and mass of air flow at the valve opening.

The calculations were numerically calculated and solved by transient analysis of the intake and compression of stroke for the piston crown below engine speed at 2000 rpm and completed its boundary conditions. The five parameters for the fluid flow and turbulence characteristics obtained from the simulation has been be considered to verify the homogeneity of the air structure for mixed preparation so that better fuel and combustion mixtures can be achieved. In this work, there is no simplification of the built-in geometry model and the intake calculation involves the intake port and the moving valve so that the flow field can be fully investigated. Specifically, the observed differences for two different piston bowl forms in the fluid flow parameter configuration and turbulence characteristics during the inlet pressure and compression are discussed and some conclusions can be drawn out. In general, this study provides insight into the effect of the piston bowl shape on the characteristics of air structure patterns for direct injection.

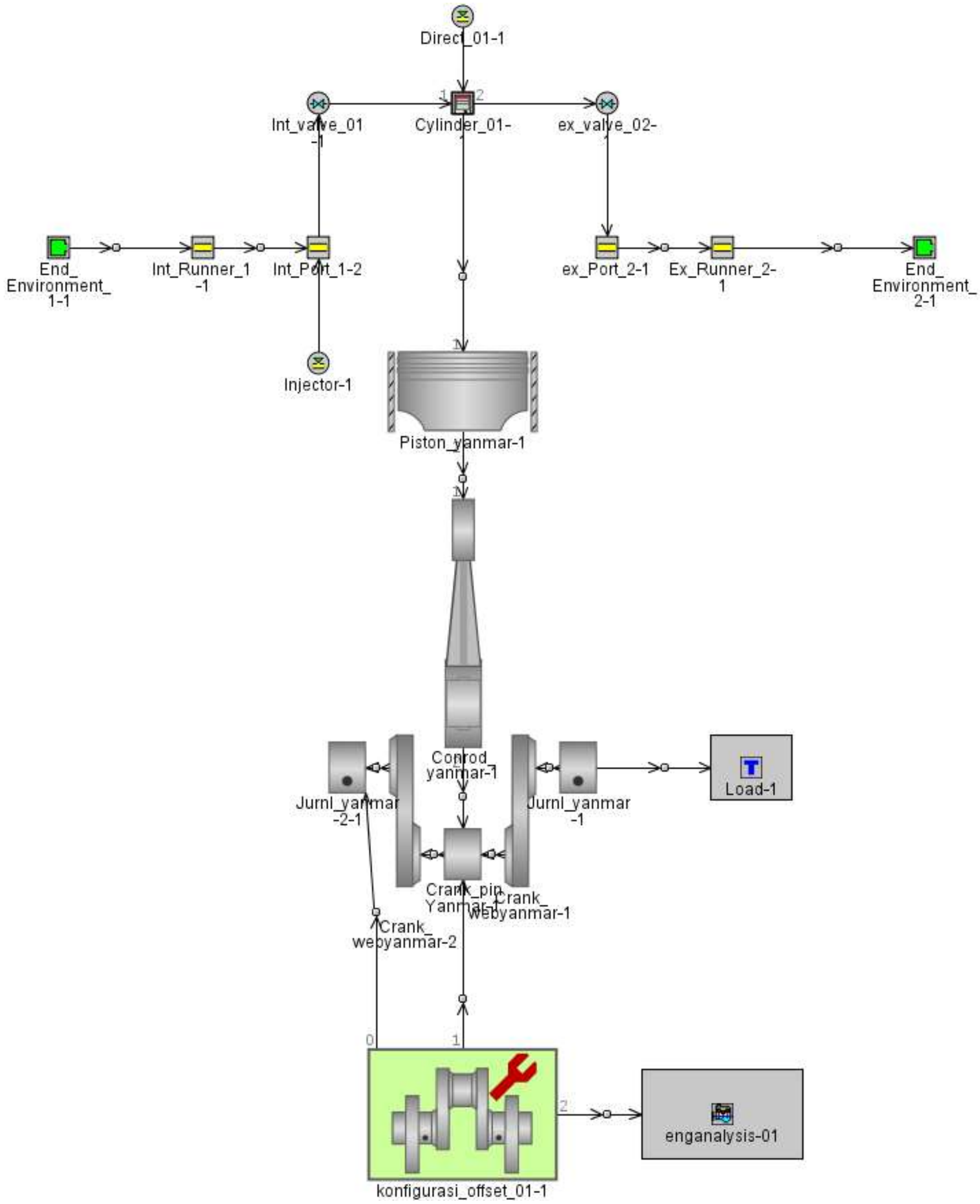

Figure 1. Numerical Modelling of Yanmar TF85 MH

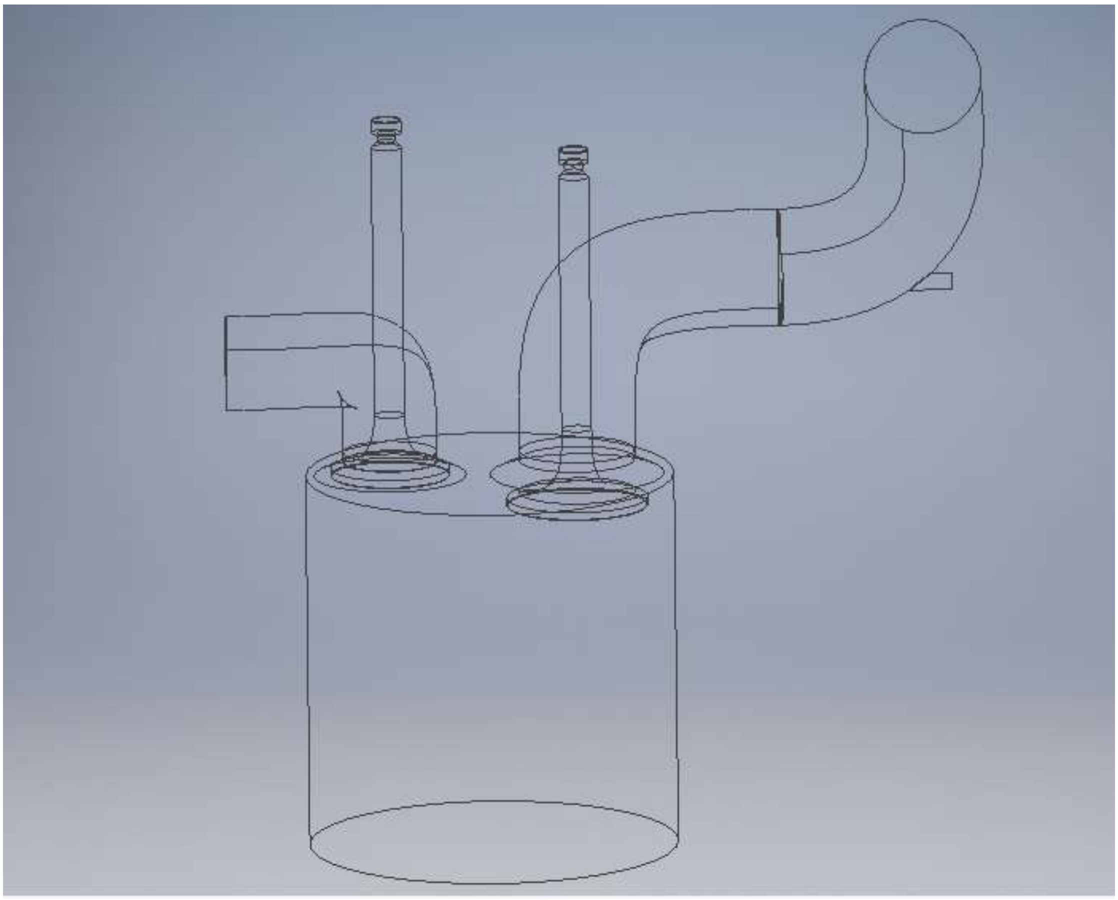

Figure 2. Engine Modelling Process of Yanmar TF85 MH

2.3 CFC Modelling

Air-fuel flow simulation using Fluent is a fluid analysis step which can be measured more than one port to be analyzing at the condition of air entry and other fluid. Fluent flow simulation is selected because it can include two fluid inputs to be analyzing step to input geometry and compose geometry. In this step is determined where the fluid has been entered, where air has been come out, and choose valve for intake and exhaust [18].

Internal combustion model of the machine that has been made in the previous step then simulated with the software CFD namely ANSYS is shown in Figure 3 – Figure 5. The purpose of CFD is to know the spread of mixing speed when air and fuel are fed into the model. Is an input parameter where the state is adjusted to the actual engine condition. The input parameters used are: Pressure inlet from the fuel that has been be input on the simulation in accordance with the results have been modelled on GT power. This parameter is derived from GT Power data on a single-cylinder gas engine. Setup is the process of calculation that has been be processed by the software. In the parallel process determines the number of processors in the computer, the more processors the faster running is done. Conversely, if running using a serial or a processor has been needed long time.

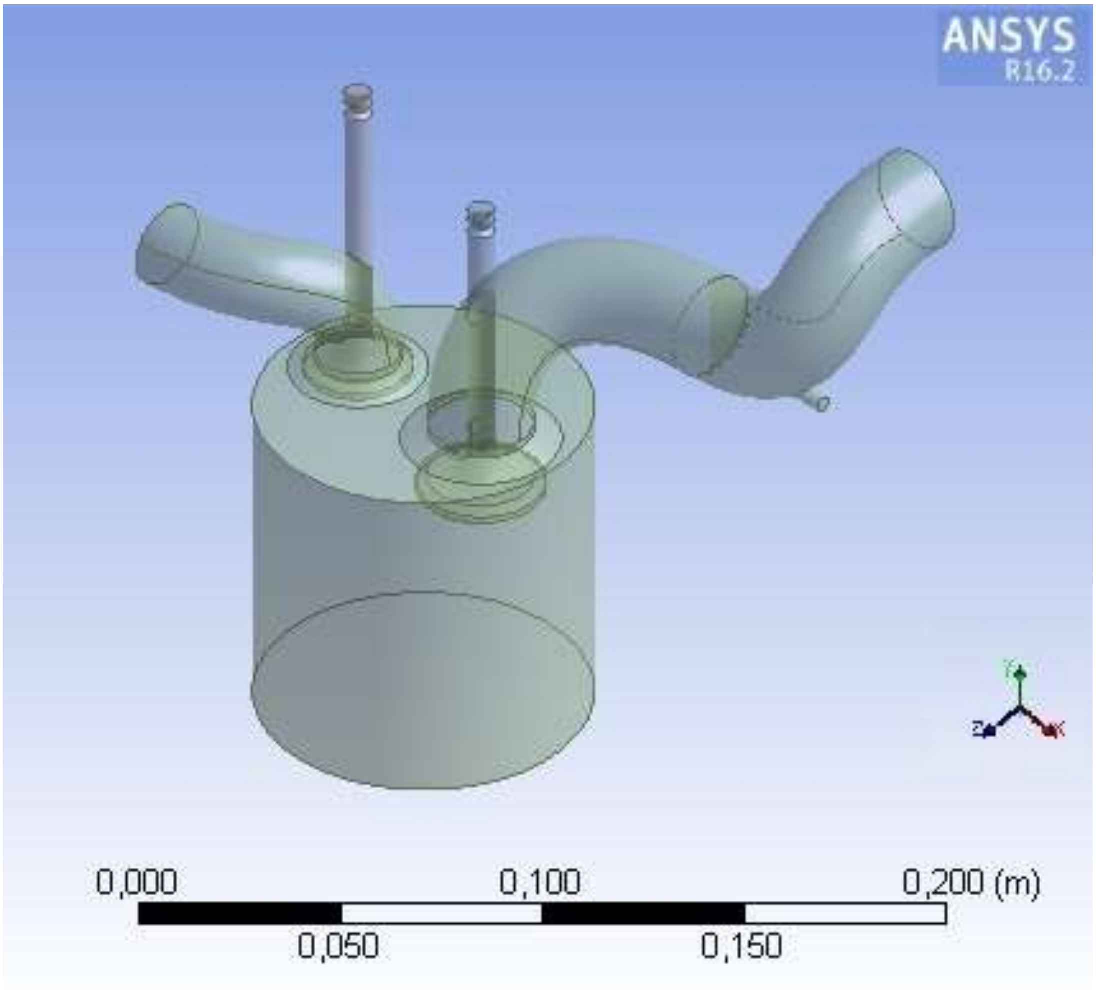

Figure 3. Engine Modelling

Post stage aims to display the results of data processing that has been done on the running process. The results obtained can be either numerical data or visual data. The data obtained has been be used as an analysis of the velocity and mass of the airflow when air enters in accordance with the valve opening variation. From the simulation it has been show the Contour Velocity or the speed contour showing how the flow has been occur in modelling flash, it has been be in know how the pattern of airflow and fuel during combustion. Then to know the result of average velocity and mass of air flow that happened can be seen at calculator function to know the quantity of airflow density and mass that happened. The data obtained is the average velocity and average airflow mass in meshing step aims to create area extents that has been be used as a process of calculation analysis by software. The smaller the area given the more detailed the calculation. However, the longer the meshing process has been done. CFD solver process aims to perform data processing with computer numerical calculation of all parameters specified in the domain and boundary condition. At this stage, the parameters used are maximum iteration 3000 iterations, and using automatic timescale. The iteration is used to obtain convergence, ii the matching between input and output. The smaller the difference in convergence then the results have been more accurate. The boundary condition parameters where conditions are adjusted to the actual conditions.

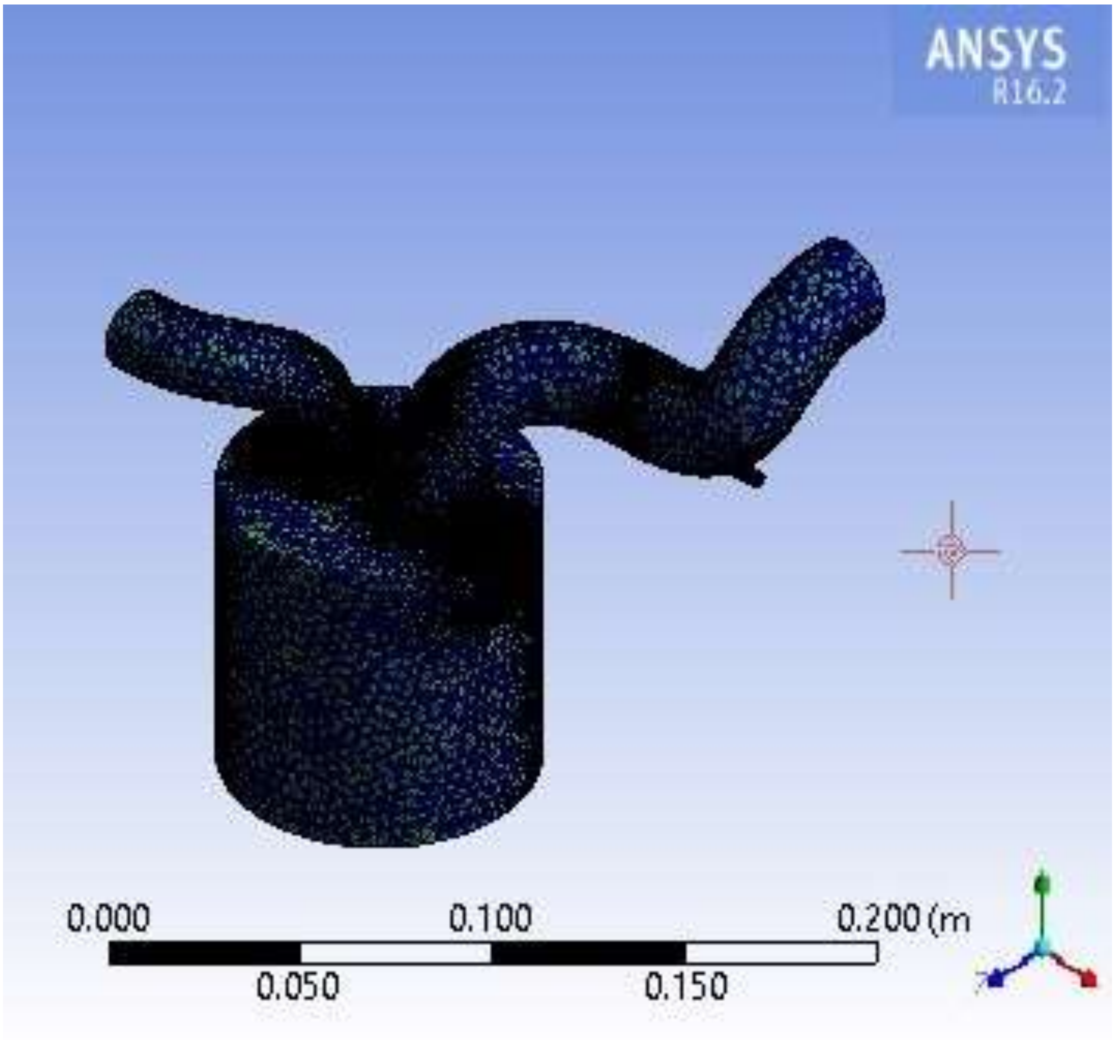

Figure 4. Messing Process

The input parameter used is the result of the air intake simulation and the fuel that has been be inputted to the simulation in accordance with the results that have been modelled on GT power. This parameter is obtained from GT Power data on cylinder gas engine. The process of calculation to be processed by the software. In the parallel process determines the number of processors in the computer, the more processors the faster running is done. Conversely, if running using a serial or a processor has been needed long time. In this simulation, the incoming air material is the ideal gas with the density of 1,225 kg / m3. Internal Combustion model of the machine that has been made in the previous step then simulated with the software CFD namely ANSYS. The purpose of CFD is to know the spread of mixing speed when air and fuel are fed into the model. Pressure inlet from the fuel that has been be input on the simulation in accordance with the results have been modelled on GT power. This parameter is derived from GT Power data on a cylinder gas engine.

Setup is the process of calculation or calculation that has been be processed by the software. In the parallel process determines the number of processors in the computer, the more processors the faster running is done. Conversely, if running using a serial or a processor has been needed a long time.

Post stage aims to display the results of data processing that has been done on the running process. The results obtained can be either numerical data or visual data. The data obtained has been be used as an analysis of the velocity and mass of the airflow when air enters in accordance with the valve opening variation. From the simulation it has been show the Contour Velocity or the speed contour showing how the flow has been occur in modelling flash, it has been be in know how the pattern of airflow and fuel during combustion. Then to know the result of average velocity and mass of air flow that happened can be seen at calculator function to know the quantity of airflow density and mass that happened. The data obtained is the average velocity and average airflow mass [18-21, 38].

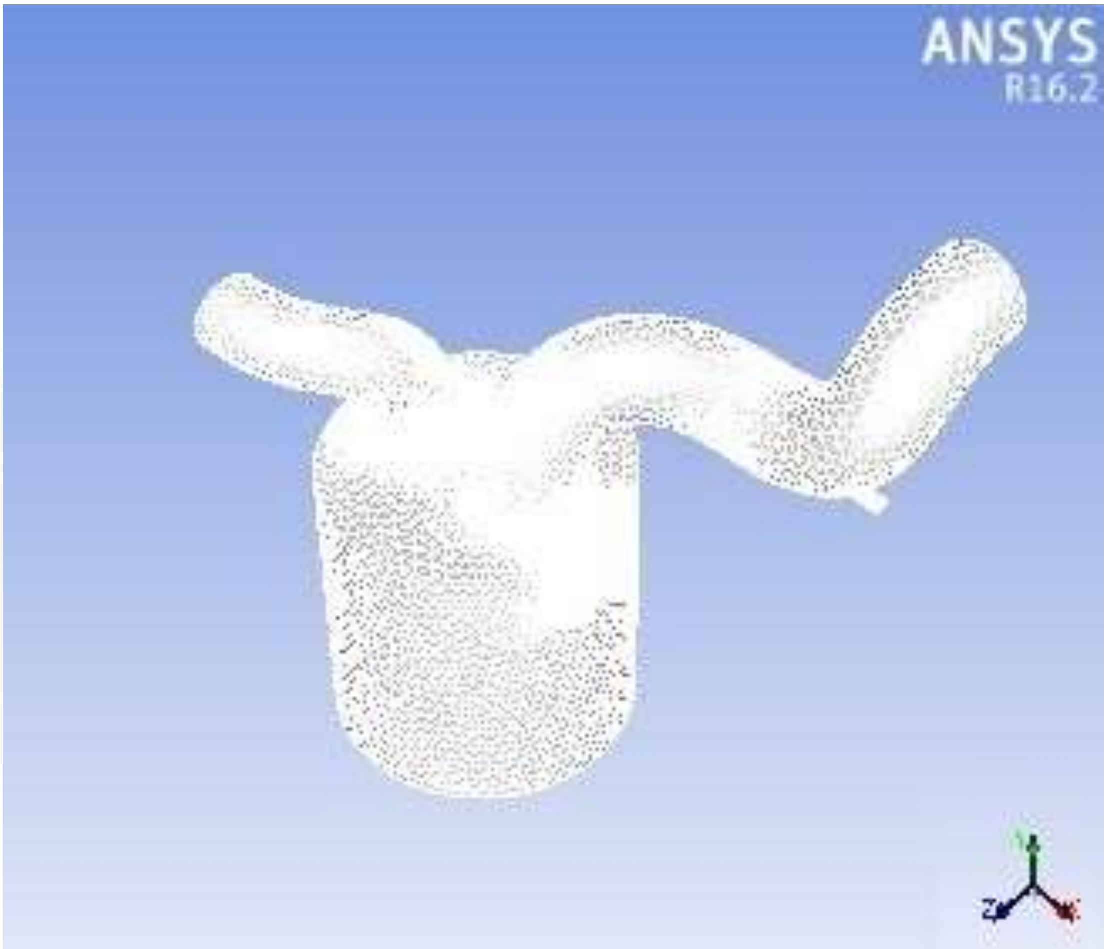

Figure 5. Setup Data.

3. RESEARCH METHOD

The research method the steps of the research investigation conducted. The first step is engine specification to collect the data of the engine for the investigation. The second step numerical modelling to collect the data of engine parameters for the internal boundary condition. The third is CFD modelling to investigate the air-fuel flow and mixing.

In this paper, an intake valve lift investigation was opened at half open from the maximum intake valve intake valve opened at 7.28 and has been at 1.5x openings and 2x valve openings at 14.56 mm. The maximum exhaust valve lift in this experiment opens at 0.1 mm. The airflow performance of the intake valve and exhaust valve of small diesel engines in the results of this investigation is shown in the Figure 6.

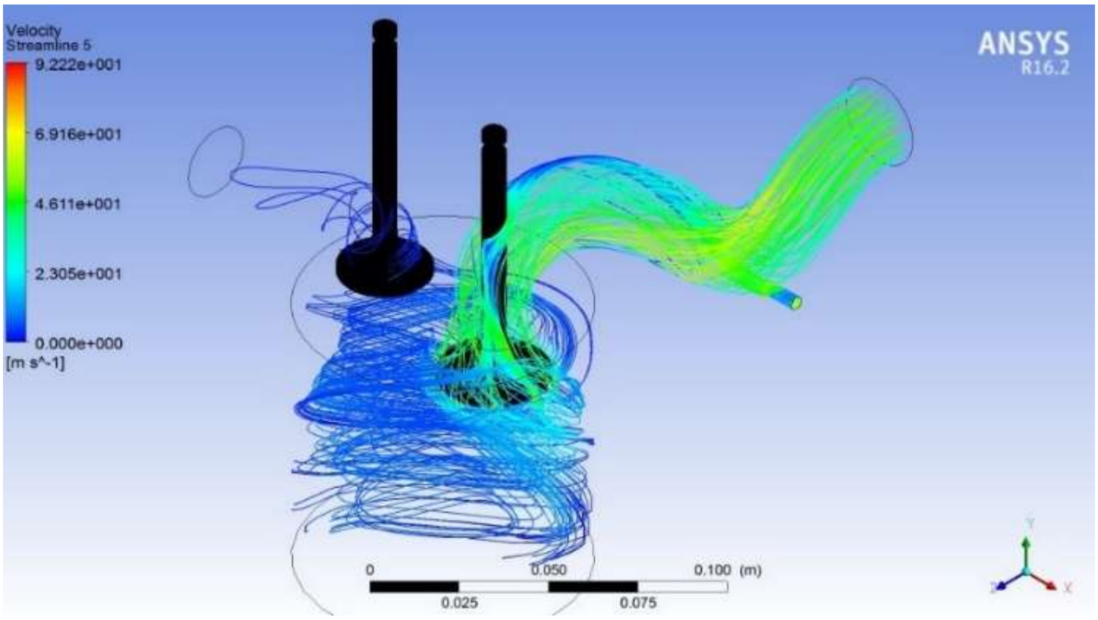

Figure 6. Flow of intake and exhaust valve open.

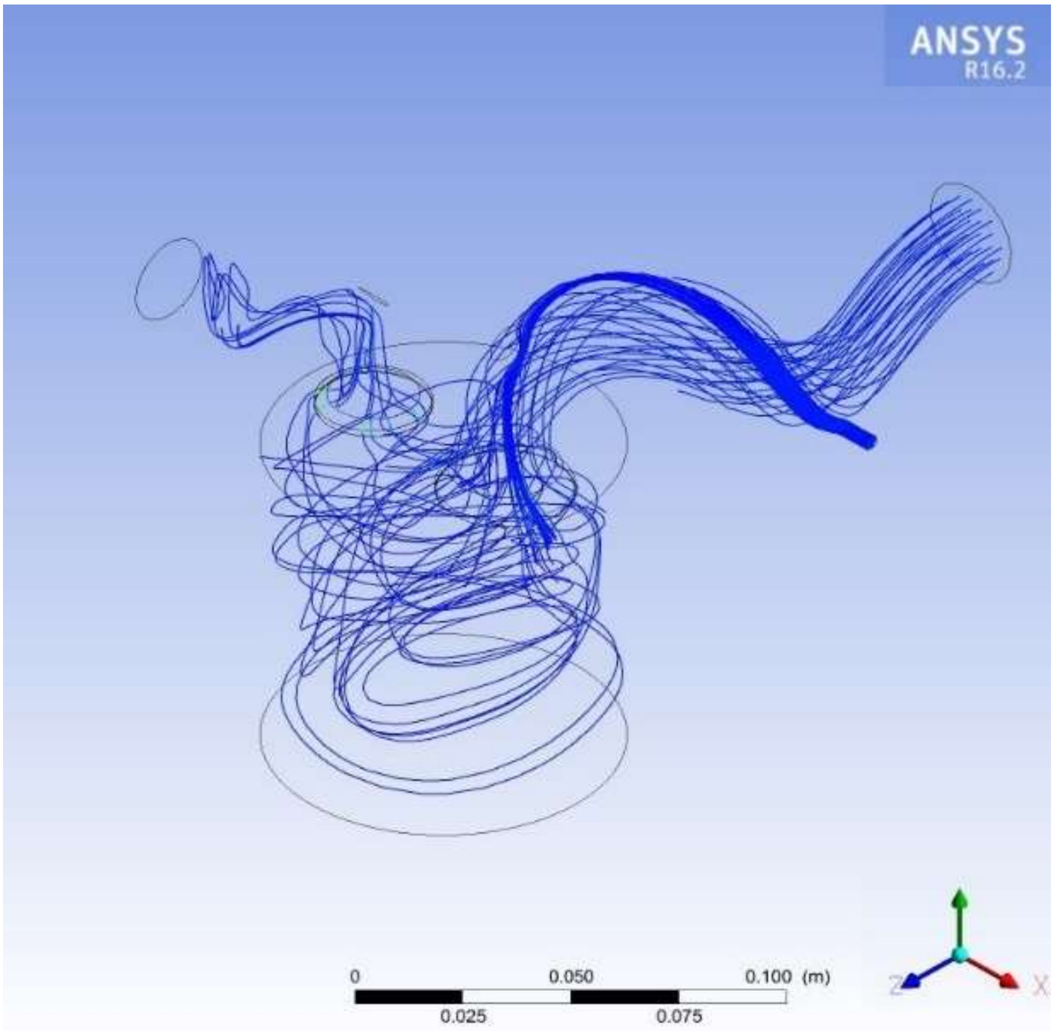

Figure 7. Baseline condition flow in 2000 rpm

From the simulation results of 3.64 mm openings at 2000 rpm Contour Velocity or contour velocity displays airflow with fuel material that has entered the combustion chamber. Figure 7 shows that the flow of baseline and fuel is very little because of this opening the valve opens and close very briefly. From contour velocity or speed contour displays colour where dark blue to light blue are low fluid velocities, green colour is medium speed, and orange to red is high speed or maximum that can be achieved. From the simulation results of 3.64 mm openings at rpm 2200 Contour velocity displays air flow with fuel entering the combustion chamber is shown in Figure 8.

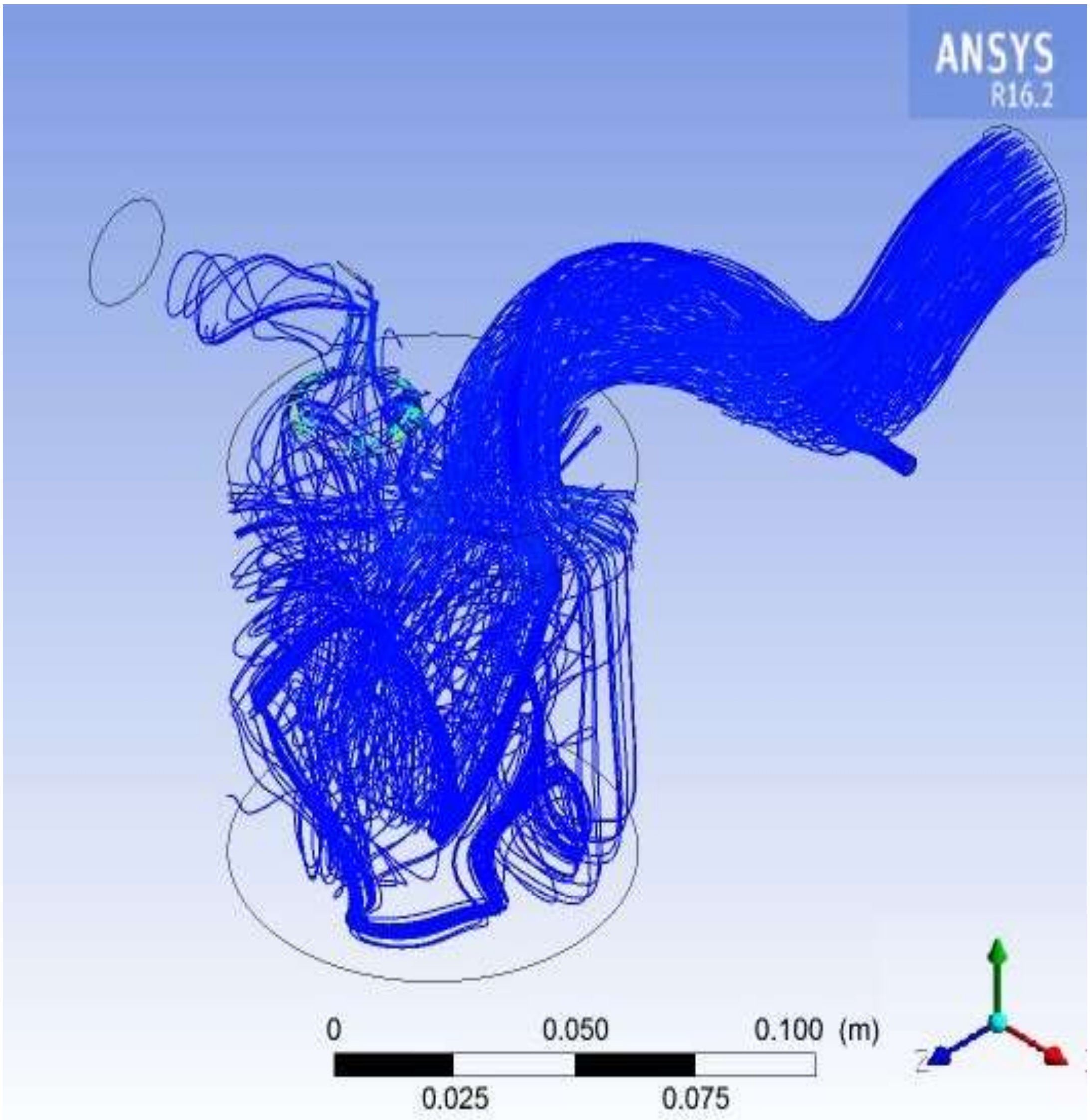

Figure 8. Baseline condition flow in 2200 rpm

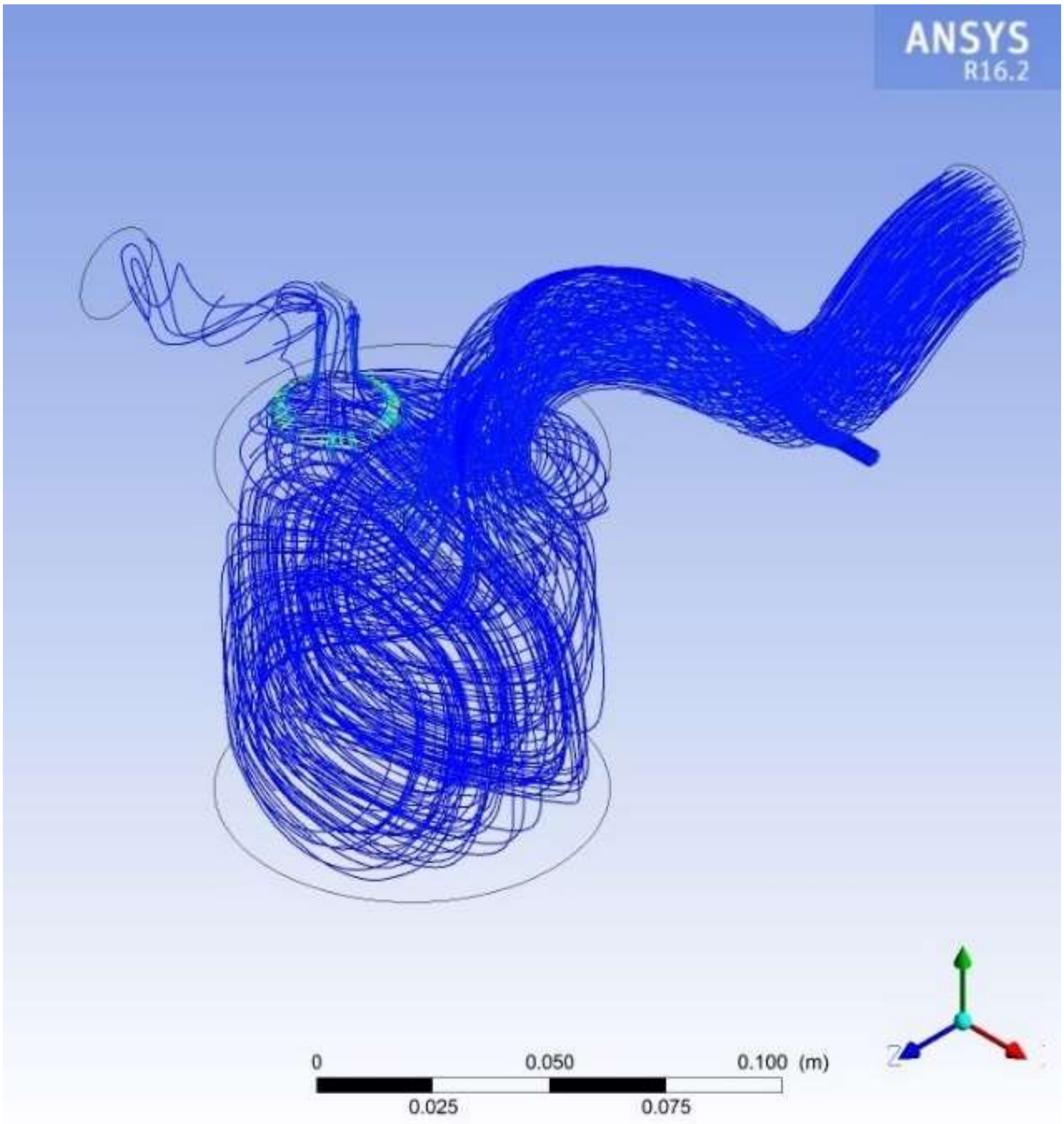

Figure 9. Flow of 7.28 mm intake valve opening in 2000 rpm

From the simulation results of 7.28 mm openings at 2000 rpm Contour Velocity or contour speed displays airflow with baker material that has entered the combustion chamber, From Contour Velocity or contour speed displays color where dark blue to light blue is low fluid speed, green color medium, and orange to red is the maximum or maximum speed that can be achieved. It can be seen from Figure 9 that the flow on this faucet is predicting more tumble flows when air and fuel enter to cylinder.

From the simulation results of openings of 7.28 mm at rpm 2200 Contour Velocity or contour velocity displays airflow with fuel that has entered the combustion chamber. It can be seen from Figure 10 that air and fuel flow show the flow pattern that has a better turbulence level compared with the flow at the opening of the valve at 3.59 because at this opening the valve opens and closes according to the original state. From Contour Velocity or speed contour displays colors where dark blue to light blue are low fluid velocities, green color is medium speed, and orange to red is high speed or maximum that can be achieved.

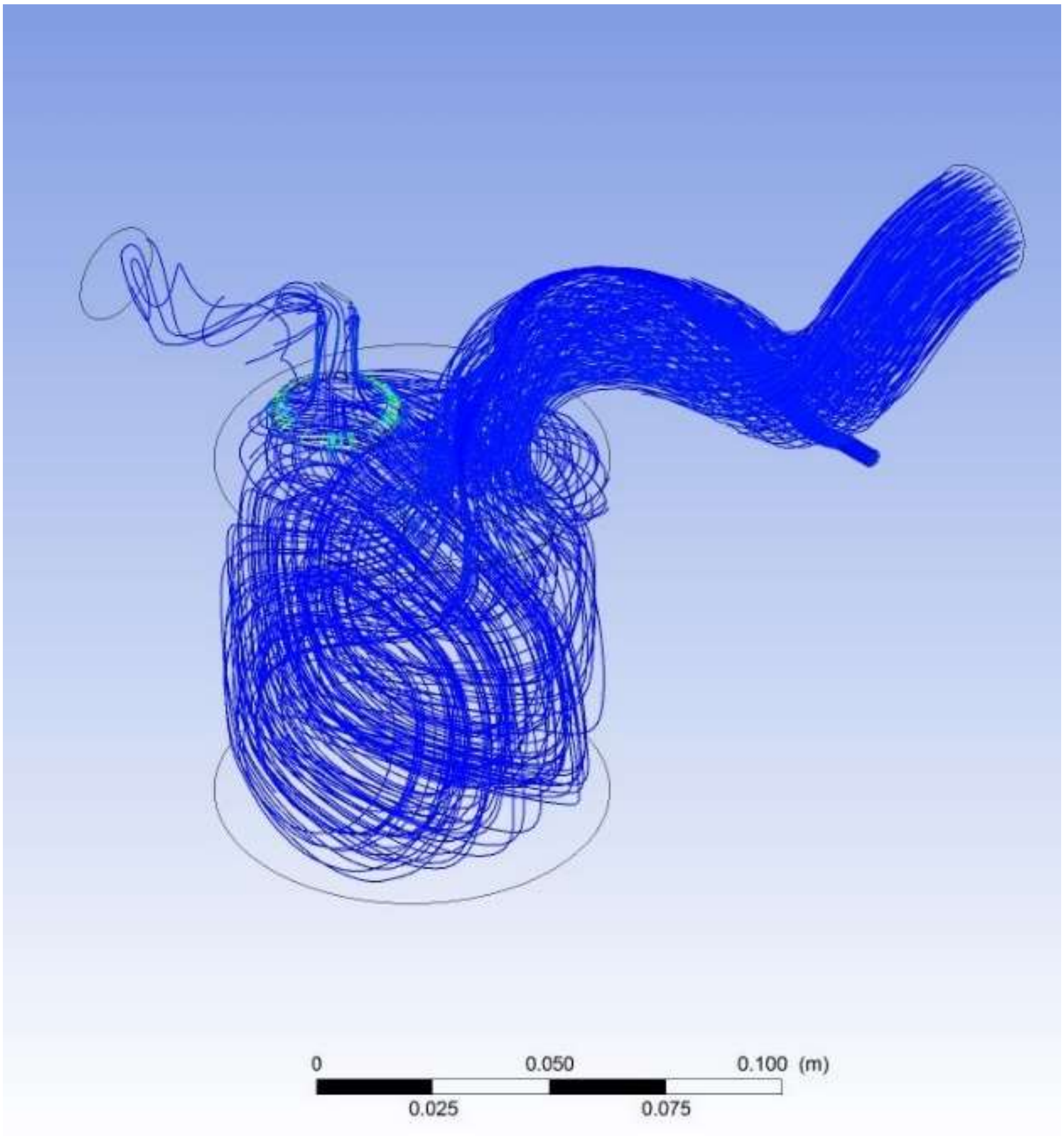

Figure 10. Flow of 7.28 mm intake valve opening in 2200 rpm

Figure 11 and Figure 12 shown the result about the influence of valve openings that deepen affect the maximum speed that can be achieved by air and fuel. The higher the valve aperture the maximum air velocity has been decrease. This is in accordance with the law of continuity A1.v1 = A2 v2. Where a deeper valve aperture produces larger open area openings that can decrease airflow velocity. From the Figure 11 and Figure 12 its can be seen that the addition of valve openings deeper effect the air velocity of entry. The more in the valve opening the faster the average airflow that goes into the chamber. At open valve lift 3.64 mm can reach average air speed 74 m/s. However, it decreased at 14.56 mm valve openings to 62 m/s.

Figure 11. Average air flow velocity at 2000 RPM

Figure 13-14 shown the influence of valve openings that deepen affect the maximum speed that can be achieved by air and fuel. The higher the valve aperture the maximum air velocity has been decrease. This is in accordance with the law of continuity A1.v1 = A2.v2. Where a deeper valve opening produces a larger opening area that can decrease airflow velocity.

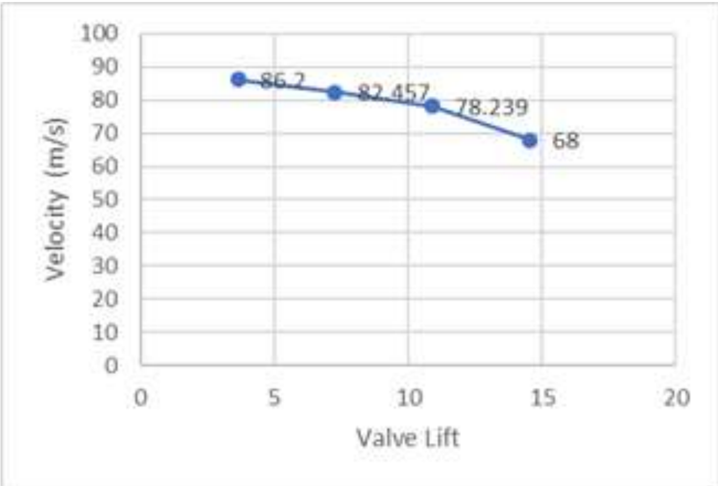

Figure 12. Average air flow velocity at 2200 RPM.

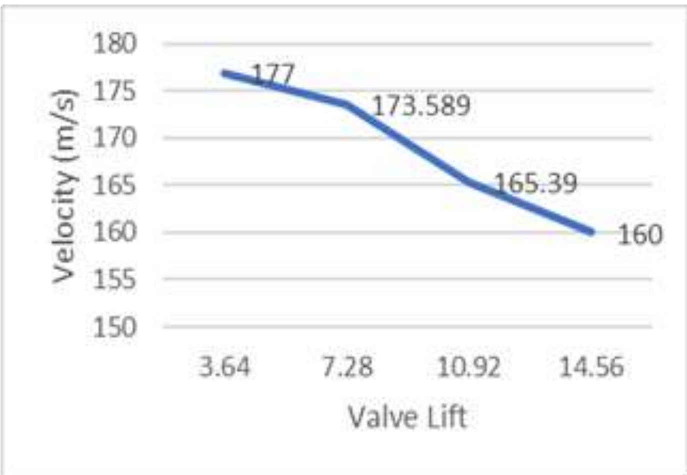

Figure 13. Max Air Flow Velocity at 2000 RPM

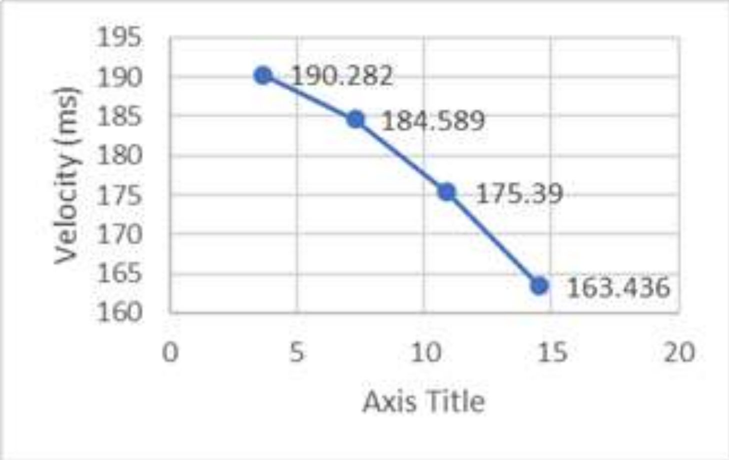

Figure 14. Max Air Flow Velocity at 2200 RPM

The results from Figure 15-16 clearly shown that the average of kinetic energy turbulence of the simulation result is quite increased with valve openings. Which is the highest average kinetic energy at intake valve opening 10.56 mm with 11478 J / kg at RPM 2000 and at RPM 2200 with valve lift 10.56 mm get the value. In contrast, the lowest average velocity is a valve with a 14.56 mm opening with 9713 J / kg. For comparison, the average speed at 4 intake valves was 10927.8 J / kg in 3.64mm and 11003.8 J / kg at 7.28 mm and 11478 J / kg in 10.92 mm and 9713 J / kg at 14.56 mm at RPM 2200. And 8467.4 J / kg in 3.64mm and 9316J / kg at 7.28mm and 9464.5J / kg in 10.92mm and 9113J / kg at 14.56mm at RPM 2000.

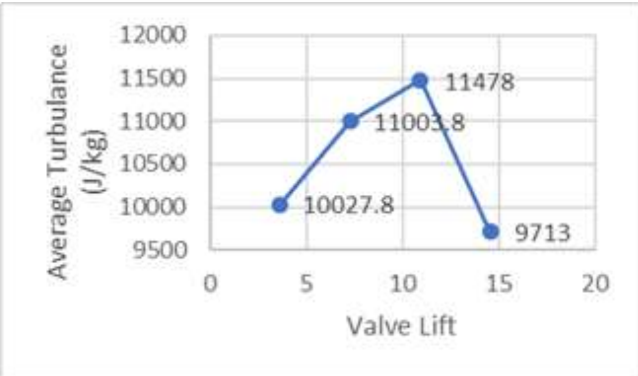

Figure 15. Max Air Flow Velocity at 2000 RPM with valve lift 10.56 mm

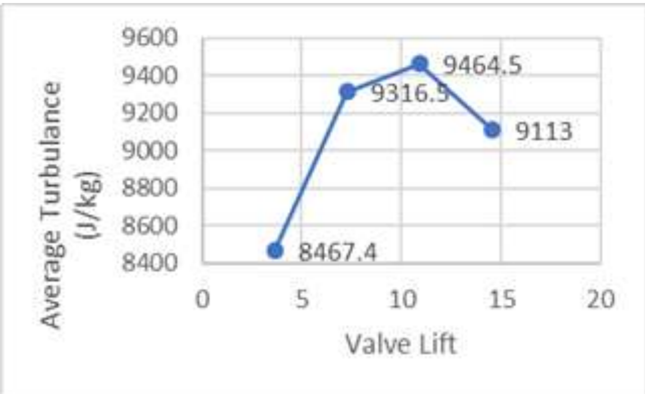

Figure 16. Max Air Flow Velocity at 2200 RPM with valve lift 10.56 mm

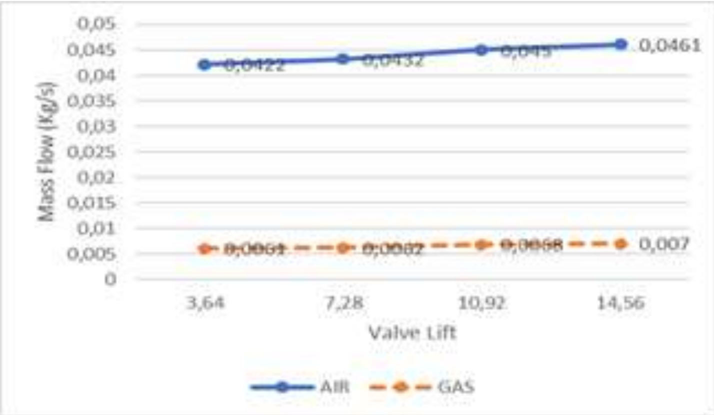

Figure 17. Air and CNG Mass Flow at 2000 RPM

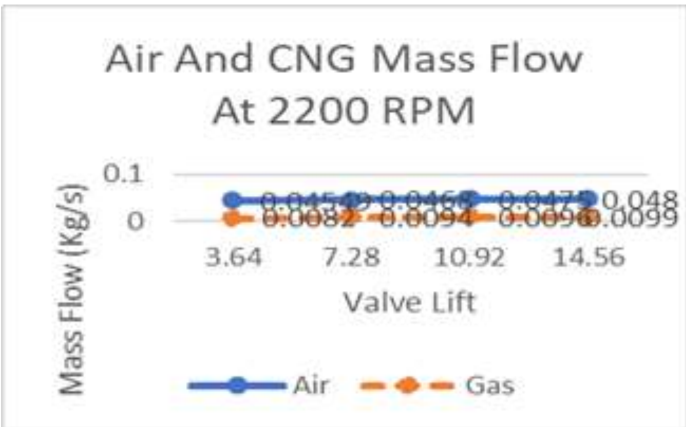

Figure 18. Air and CNG Mass Flow at 2200 RPM

Figure 17-18 shown that valve openings effect in the mass of airflow and fuel coming into the chamber. The more in the valve opening the incoming air mass has been be higher also. Air mass has the greatest value at 14.56 mm valve opening with air flow mass of 0.048 kg / s and Gas at 0.0099 kg/s.

4. CONCLUSION

The highest velocity and the highest turbulence occur at 10.92 mm valve openings and mass air most at 14.56 mm valve openings. Turbulent flow intensity has been increased in the duration of combustion with certain turbulence intensity.

The most appropriate combination of valve openings is the 10.92 mm open aperture which is 1.5 x valve opening at 7.28 mm. Because the valve openings of 10.92 mm have a high air flow mass between other valve openings and have the highest turbulence level, when the air and fuel enter has been be mixed more leverage at this opening.

ACKNOWLEDGMENT

The authors very express acknowledge to the AA, Afif, Azka, Azzam, Linggar from the Laboratory of Marine Power Plant the DMarE Institut Teknologi Sepuluh Nopember (ITS), the Ministry of Research, Technology, and Higher Education of the Republic of Indonesia, Research Institute and Public Services of ITS, Surabaya Indonesia for PTUPT research grant in batch 2019.

REFERENCES

[1] Al-Khishali, K. J. M., Najjar, Y. S. H. and Ghazal, O. H, “Effect of valve lift at different IVO, IVC and OVERLAP angles on “SI Engine performance’, The 7th Jordanian International Mechanical Engineering Conference, (September),2010, pp. 27–29.

[2] Imran S., Embrson D.R., Ihracska B., Wen D.S., Crookes R.J., Korakianitis T., “Effect of pilot fuel quantity and type on performance and emissions of natural gas and hydrogen-based combustion in compression ignition engine”, International Journal Hydrogen Energy 39 (10), 2014, pp 5163-5175.

[3] Lavhale, Y. K. and Salunke, J., “Overview of failure trend of inlet &amp: exhaust valve”, International Journal of Mechanical Engineering and Technology, 976(3), 2014, pp. 976–6359.

[4] Yerrennagoudaru, H. and prasad Desai, S., “Generation of Air Swirl through Inlet Poppet Valve Modification and To Enhance Performance on Diesel Engine”, IOSR Journal of Mechanical and Civil Engineering, 12(6), 2015, pp. 2278–1684. doi: 10.9790/1684-12665465.

[5] Ehsan Md, Bhuiyan, “Gas-Oil fuel performance of small diesel for application with less frequent load”, International Journal of Mechanical and Mechatronics Engineering (IJMME). Volume 9 (10), 2009, pp 30-39.

[6] Zoltowski A., “Investigation of combustion process in Gas-Oil fuel diesel engine”, Journal of KONES Powertrain and Transport, Volume 21, No 2, 2014, pp. 303-309.

[7] Wei, L. and Geng, P., “A review on natural gas/diesel Gas-Oil fuel combustion, emissions and performance”, Fuel Processing Technology. Elsevier B.V., 142, 2016, pp. 264–278. doi: 10.1016/j.fuproc.2015.09.018.

[8] Semin, Cahyono B. Amiadji. Bakar R.A. (2014), “Air-fuel mixing and fuel flow velocity modeling of multi holes injector nozzel on CNG marine engine,” Procedia Earth and Planetary Science 14, 2014, pp. 101-109.

[9] Shrirao P.N., Sambhe R.U., “Effect of Swirl Induction by Internally Treaded Inlet Manifolds on Exhaust Emissions on Single Cylinder (DI) Diesel Engine”, International Journal of Science and Research, Vol 3 (7), 2012, pp 1718-1722.

[10] Zhang, H. and Mastorakos, E, “Modelling local extinction in Sydney swirling non-premixed flames with LES/CMC”, Proceedings of the Combustion Institute. Elsevier Inc., 36(2), 2017, pp. 1669–1676. doi: 10.1016/j.proci.2016.07.051.

[11] Sahoo B.B., Sahoo N., Saha U.K., “Effect of engine parameters and type of gaseous fuel on the performance of dual-fuel gas diesel engine – A critical riview”, Renewable and Sustainable Energy Riviews, Volume 13, 2009, pp 1151-1184.

[12] Semin. A.R. Ismail. and R.A. Bakar., “Investigation of GAS Engine Intake Port Gas Flow Temperature Based on Steady-State and Transient Simulation”, European Journal of Scientific Research 22 (3), 2008.

[13] Semin. A.R. Ismail. R.A Bakar and I. Ali, “Heat Transfer Investigation of Intake Port Engine Based on Steady-state and Transienttion”, American Journal of Applied Sciences 5 (11), 2008.

[14] Abdul Rahim Ismail. Rosli Abu. Bakar. Semin. and Ismail Ali, “Computer Modelling for 4-Stroke Direct Injection Diesel Engine,” Advanced Materials Research, Volumes 33-37, 2008.

[15] A.R. Ismail. R.A. Bakar and Semin. “An Investigation of Valve Lift Effect on Air Flow and CD of Four Stroke Engines Based on Experiment.”, American Journal of Applied Sciences 5 (8). 2008.

[16] Semin. R.A. Bakar and A.R. Ismail., “Investigation of Diesel Engine Performance Based on Simulation”, American Journal of Applied Sciences 5 (6), 2008.

[17] Semin. R.A. Bakar and A.R. Ismail. “Computational Visualization and Simulation of Diesel Engines Valve Lift Performance Using CFD”. American Journal of Applied Sciences 5 (5). 2008.

[18] Semin. A.R. Ismail and R.A. Bakar, “Comparative Performance of Direct Injection Diesel Engines Fueled Using GAS Based on GT-POWER Simulation”, American Journal of Applied Sciences 5 (5), 2008.

[19] R.A. Bakar. Semin and A.R. Ismail, “Fuel Injection Pressure Effect on Performance of Direct Injection Diesel Engines Based on Experiment”, American Journal of Applied Sciences 5 (3), 2008.

[20] R.A. Bakar. Semin. A.R. Ismail and I. Ali, “Computational Simulation of Fuel Nozzle Multi Holes Geometries Effect on Direct Injection Diesel Engine Performance Using GT-POWER”, American Journal of Applied Sciences 5 (2), 2008.

[21] Semin. R.A Bakar. and A.R. Ismail., “Compressed Natural Gas as an Alternative Fuel for Internal Combustion Engines: A Technical Review”, International Review of Mechanical Engineering Vol. 3 (2), 2009.

[22] Semin. A. Idris. and R.A Bakar, “Effect of Port Injection GAS Engine using Injector Nozzle Multi Holes on Air-Fuel Mixing in Combustion Chamber”, European Journal of Scientific Research 34 (1), 2009.

[23] Semin. A. Idris. and R.A. Bakar, “An Overview of Compressed Natural Gas as an Alternative Fuel and Malaysian Scenario”, European Journal of Scientific Research 34 (1), 2009.

[24] Semin. A.R. Ismail. and R.A. Bakar, “Gas Fuel Spray Simulation of Port Injection Compressed Natural Gas Engine using Injector Nozzle Multi Holes”, European Journal of Scientific Research 29(2), 2009.

[25] Semin, A. Idris. R.A. Bakar. A.R. Ismail., “Study of the Engine Cylinder Fluid Characteristics for Diesel Engine Converted to GAS Engine”, European Journal of Scientific Research 26 (3), 2009.

[26] Semin, A.R. Ismail. and T.F. Nugroho., “Experimental and Computational of Engine Cylinder Pressure Investigation on the Port Injection Dedicated GAS Engine Development”, J. Applied Sci.10 (2). pp: 107-115, 2010.

[27] Semin, “Injector Nozzle Spray on Compressed Natural Gas Engines: A Technical Review”, International Review of Mechanical Engineering 6. (5), 2012.

[28] Semin, RA Bakar, Simulation and experimental method for the investigation of compressed natural gas engine performance”, International Review of Mechanical Engineering 7 (7), pp. 1427 7 (7), 1427, 2013.

[29] Semin and R.A. Bakar, “Computational Modelling the Effect of New Injector Nozzle Multi Diameter Holes on Fuel-Air Mixing Homogeneous of GAS Engine”, International Journal of Applied Engineering Research. Volume 9 (21). pp. 9983, 2014.

[30] Semin, “Analysis of Biogas as an Alternative Fuel for Electric Generator Engine in Bawean Island – Indonesia”, International Journal of Applied Engineering Research 10 (16). pp. 35313-35317, 2015.

[31] Semin, “Investigation the Effect of Injector Nozzle Multi Holes Geometry on Fuel Spray Distribution Flow of GAS Engine Based on Computational Modeling”, International Journal of Applied Engineering Research 10 (15). pp.36087-36095, 2015.

[32] Semin, B Cahyono, Amiadji, RA Bakar, “Air-fuel Mixing and Fuel Flow Velocity Modeling of Multi Holes Injector Nozzle on CNG Marine Engine”, Procedia Earth and Planetary Science 14, 101 – 109, 2015.

[33] Semin., Gusti. A.P., Octaviani, N.S., and Zaman, M.B., “Effect of New Injector on the Torque Performance Characteristics of Gas engine”, International Journal of Applied Engineering Research, 11(11), 2016, pp.7467-7471.

[34] Semin., Octaviani. N.S., Gusti, A.P. and Zaman, M.B., “Power Performance Characteristics Investigation of Gas Engine using New Injector”, International Journal of Applied Engineering Research, 11(11), 2016. pp.7462-7466.

[35] Semin., Iswantoro, A. and Faris, F., “Performance and NOx Investigation on Diesel Engine using Cold EGR Spiral Tube: A Review”, International Journal of Marine Engineering Innovation and Research, 1(3). 2017.

[36] A. P Gusti, Semin, “Effect of Ship Speed on Ship Emissions”, Asian Journal of Scientific Research, 11 (3), 2018, 428-433.

[37] Semin, A.P Gusti, “The Effect of Ship Speeds on Fuel Consumption: A Review”, Journal of Engineering and Applied Sciences, 12 (22), 2017, 6052-6056.

[38] Gamma Technologies, “GT-POWER User’s Manual Version 6.1,” Gamma Technologies Inc., 2004.