Analysis of Influence of Masks Flow on Intake Valve of Gas and Oil Fuel Engine Based Simulation

Semin[1], Mohammad Azzam Satriawansyah[2], Beny Cahyono[3], Nilam Sari Octaviani[4]

***Abstract*—Turbulent flow in the combustion chamber affects the combustion process which also affects the engine performance. The turbulent flow intensity can increase the duration of combustion, but too much turbulent turbulent flow leads to a tap. In addition, turbulent flow also contributes to the gas-water mixing process. There are two types of turbulent flow that occur inside the machine, which is a current vortex and a falling current. The current whirl is better known as the swirl flow and the falling current is better known as the tumble flow. The fluid flow in the combustion chamber can be adjusted by optimizing the engine components and developing the engine design. A review has been made of the vortex and Tumble flow on a luminous spark machine and its effect on turbulence and fire propagation. From several journals available to create tumble and swirl streams in the combustion chamber, they use modifications to the intake valve by adding vin to create a swirl stream and adding a mask to create a tumble stream. to know the experiment was successful or not, first modeling and then perform simulation using CFD Ansys Fluent. for the variables choose 3mask, 4mask, and 5mask (for addition of mask) which later dilih air flow formed by the addition of mask.**

I. INTRODUCTION

After the discovery of dual fuel engines, now many studies using dual fuel engine is. the main advantages of dual fuel gas diesel engines reduce the emissions of NOx and PM and reduce fuel costs due to the cost of natural gas is much higher than the diesel fuel injection. The advantage of other dual fuel engines is to increase thermal efficiency, which is load dependent, and reduce the transport impact of fuel and use locally available natural gas. Since large amounts of diesel are displaced by natural gas, up to 70%, carbonization is reduced in dual fuel engines [1].

In recent years, increased interest in improving engine efficiency has led to the development of new engine technology The accurate determination of heat transfer across the combustion chamber is highly relevant for performing a valid thermal equilibrium while evaluating the potential of the new concept engine. Some work related to heat transfer acceleration considering rotation is found in the literature; However, there is a shortage of work related to the heat transfer relationship that takes account of the effects of falling motion. In this work, the heat transfer model calculates the fall of motion presented. HSDI machine two strokes HSDI with high fall and no vortex usually perform theoretical studies, model development and the last calibration. Initially, a theoretical analysis of the phenomena of gas movement was based on CFD results and then, a model developed and calibrated based on skip-fire testing techniques. Finally, a sensitivity study focused on evaluating model toughness performed. The results confirm an average RMSE reduction of 70% with respect to the Woschni model, as a consistently qualitative increase is evidenced in the instantaneous heat transfer of evolution [1].

Natural gas contains a mixture of methane, ethane, propane, butane, pentane and other hydrocarbons. It's formed from animals and plants that decomposed millions of years ago, awakened in thick layers and trapped beneath the earth's surface. Over time, intense heat and pressure turn this fossil into black oil, coal and natural gas. Natural gas is extracted from underground formation through wells with other liquid hydroca rbons and non-hydrocarbons, which are then filtered from these components and sent through a pipeline network for distribution [1].

Turbulent flow in the combustion chamber affect the combustion process which also affect the engine performance. Turbulent flow intensity can improve the duration of combustion, but too much turbulent turbulent flow intake leads to knocking. In addition, turbulent flow also contributes to the gas-water mixing process. There are two types of turbulent flow that occurs inside the machine, which is the current vortex and falling current. The current vortex is better known as the swirl flow and the falling current is better known as the tumble flow. Fluid flow in the combustion chamber can be

Semin, Department of Marine Engineering, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia, Email : seminits@yahoo.com
Mohammad Azzam Satriawansyah, Department of Marine Engineering Institut Teknologi Sepuluh Nopember, Surabaya 60232, Indonesia, Email : azzamsatriawansyah@gmail.com
Beny Cahyono, Department of Marine Engineering, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia, Email : benyjtsp98@gmail.com
Nilam Sari Octaviani, Department of Marine Engineering, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia, Email : nilam.octaviani@gmail.com

adjusted by optimizing engine components and developing engine design. A review has been made from a swirl and a Tumble stream on a luminous spark machine and its influence on turbulence and flame propagation. The rotating flow can significantly increase the intensity of turbulence during the burning period. This in turn can lead to reduced burning periods and increased thermal efficiency [2].

Another flow generated by large-scale vertical flow patterns is the falling motion or the so-called tumble flow. The axis rotates a normal vortex to the cylinder axis, resembling a rotation of the barrel. It is formed around a circular axis near the edge of the clearance volume in the piston crown or in the cylinder head, caused by squishing the cylinder volume as the piston approaches the TDC. Thus, the falling motion is also called a vertical swirl or

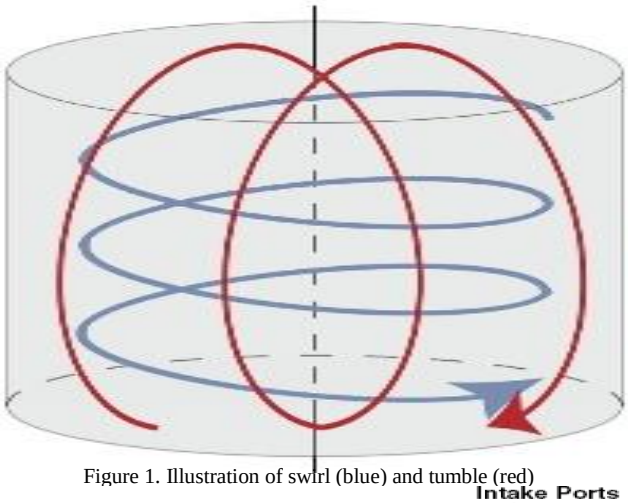

Figure 1. Illustration of swirl (blue) and tumble (red)

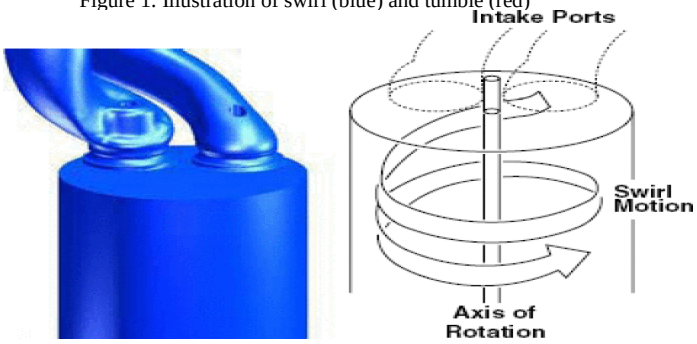

Figure 2. Illustration of swirl motion

The swirl stream is defined as a large-scale swirl in a cylindrical fluid with a rotational axis parallel to the piston axis, which is considered a solid two-dimensional body rotation. The occurrence of swirl flow is caused by the addition of vin in the intake valve where the movement of the air flow rotates downward like a vortex. Creating a whirlpool rotating inside the cylinder has been recognized as a way to increase turbulence levels during compression. Swirl increases turbulence during compression stress through the following methods: Turbulence generated by shear on the wall is transported along the largest flow by the backflow of diffusion and the rotating current or protruding objects that do not exist on the rotation axis of the vortex will make turbulence through shear and vortex shedding or swirl vortex combined with squish flow will cause acceleration of vortex rotation speed as the piston approaches TDC to preserve angular momentum. This will increase turbulence at the end of compression pressure [3].

swirl barrel. To produce pure fall motion for a single intake valve cylinder, the directional jet intake vector must be on a plane defined by the cylinder axis and the intake axis of the intake valve. The flow falls with radial motion and axial motion is expected to occur in cylindrical coordinates. If the intake jet has only tangential components and axial components in cylindrical coordinates, a purely vortex flow is generated. This whirl is compressed during stroke compression and increases the rotation rate to preserve angular momentum. With increasing compression, the vortex becomes less circular. The whirl reaches a critical point beyond the decomposed vortex into smaller vortices. This vorticity turns into a similar turbulent structure, increasing turbulent levels [3].

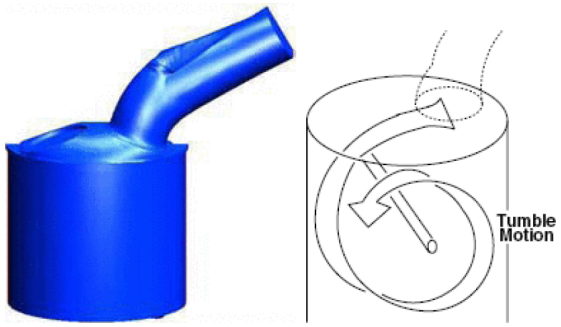

Figure 3. Illustration of tumble motion

II. AIR INTAKE VALVE

Valve settings in the engine control the movement of in and out of charge and exhaust gas in the cylinder associated with the piston position in the hole. now it is located in the cylinder head on all machines. Among the commonly used sleeves, sliding, rotary, and valve types are popets, the most common popet valves because these offer reasonable weight, good strength and good heat transfer characteristics. The most popular form of popet valve. for car applications using a small cup at one end of the rod. The valve rods are placed in manual holes centrally made in a circular section on the cylinder head. The valve disk head opens and closes the porting portion towards the cylinder during in and out of the movement of the rod.

particularly important at idle and low part loads when low maximum lifts are to be used for improving the fuel economy or for achieving the required power [4-8].

Modelling was done for early intake valve closure angle in comparison to original closure angle. The engine was simulated as a naturally aspirated one and for the cases such indicated parameters as indicated efficiency, mean indicated pressure, fuel consumption were calculated. During the modelling ignition, timing and air-fuel ratio were fixed. For better comparison for two cases of early intake valve closure angle the engine was modelled as supercharged one where mean indicated pressure was fixed at the same level as for the naturally aspirated engine working with original

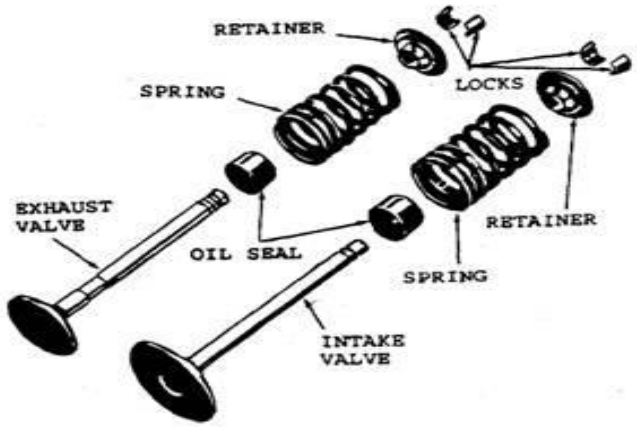

Figure 4. figure engine valve

Reducing fuel consumption is a prime objective in the automotive industry in order to meet regulatory and customer demands. Variable valve actuation offers many opportunities for improving the spark ignition engine's performance in areas such as fuel economy and pollutant emissions. Our studies revealed that the ability to control maximum intake valve lift does indeed offer the ability to control intake air mass, but also has the added benefit that it improves the fuel-air mixing process thanks to an increased turbulence, caused by the increased intake flow velocity. This is

valve timing and indicated parameters were calculated and compared with in parameters determined from this naturally aspirated engine [9-31].

III. VALVE INTAKE MASK

According to research conducted Yerrennagoudaru and Desai masking is the process of building a small piece of metal in the valve head without disturbing the seat of the valve, small pieces called masks. Peda research conducted by

Yerrennagoudaru and Desai have used 2-masked valves, 4-masked valve and 6- masked valve.[4]

Angle of mask: 1200
Width of mask: 2mm
Thickness of mask: 2mm
Outer dia of mask: 28mm
Inner dia of mask: 24mm
Hence these masks are split into 2 , 4 and 6 pieces
angularly i.e.,
1200/2 = 600 masks in 2 pieces
1200/4 =300 masks in 4 pieces
1200/6 = 200 masks in 6 numbers

but substantially reduce volumetric efficiency, leading to limited applications in production machinery . Strong cargo movements have been generated by shrouding valves and valves masking. For example, increases in turbulence during combustion by shrouding valves and masking. Correlated the intensity of turbulence with 10% - 90% burn angle by using masked valves.The authors also point out that for the geometry of this combustion chamber, falling air movement is a more effective way to produce turbulence at 19
TDC temperatures than the vortex. The effects fall on duration of burn and stability of combustion with a veiled valve [5].

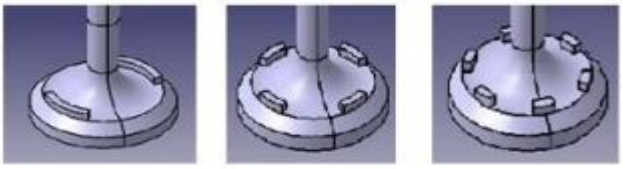

Figure 5. Example of Mask Modeling

To analyze the vortex, turbulence, air velocity as well as the distribution pressure within the cylinder during their suction stroke using CFD (Computational Fluid Dinamic) and analyzed at different inlet valve lift position in comparison with base model. The intensity of the vortex is very less on medium and high valve lift. By using CFD we can see that the type of valve to be given the vortex intensity is better [4].

Designing and testing swirl locations and ignition variables. Aspects of design used Ansys cfx to model airflow inside the cylinder. The velocity and the volumetric flow rate are calculated from the results. Many design iterations occur before the appropriate design is achieved. Guide. and valve seat mounted on the head. The inlet valve creates an air rotation inside the cylinder. Air movement plays a very important role in the mixing of air fuel, combustion and emission formation. The rotating

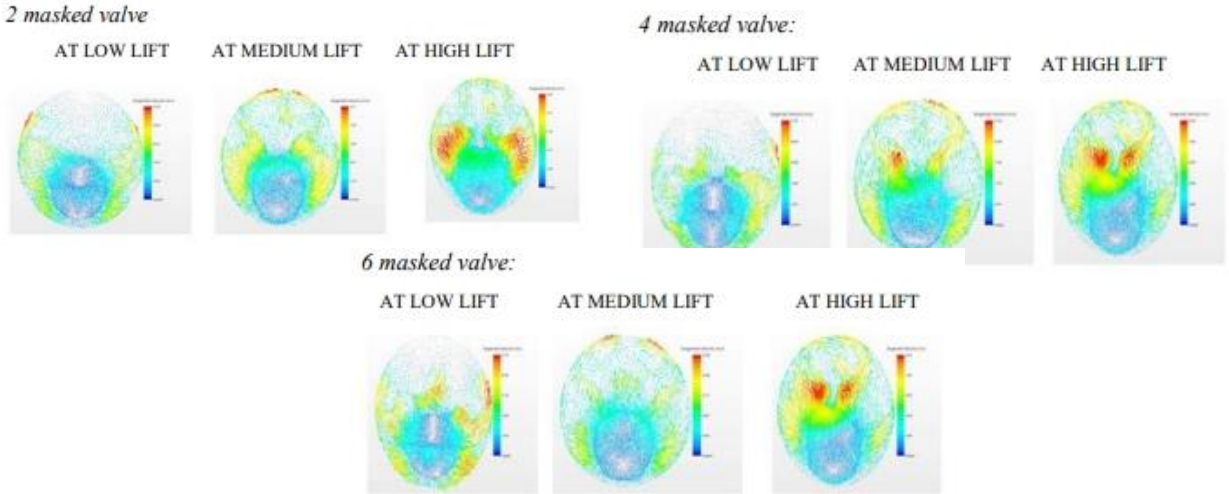

Figure 6. intensity of vortex on CFD

Both the cover and the mask partially block the flow through the inlet valve cover is part of the valve while the mask is attached rigidly to the seat of the valve. Intake valve shrouding and masking can bring intense spinning streams into the cylinder

motion in the air is usually generated due to the intake port design. A good intake port design will produce a higher swirl and help improve Combustion. This helps reduce pollutant emissions and better fuel economy. These machines tend to

achieve maximum efficiency. CFDs enable numerical experiments (ie computer simulations) in "virtual flow labs". They contribute to improved engine performance [6].

The flow pattern is clicked by using a state breaker called steady state. Domain used for parameters on certain machines. Persecuting the CFD ANSYS FLUENT problem solver, the CFD movement is gained for four very different geometries. The geometry consists of Horizontal, Vertical, curve and airborne arcs. In this simulation, only the stroke intake is sectional. From these results indicate that high airflow velocity during sappu stroke intake occurs. This condition produces more speed, thus increasing the speed in different areas of the cylinder engine's clearance volume. This will start production falling and spinning in a cylinder engine [32].

Experimental results by (R.GOBINATH, at al. 2017) for engine valves with a horizontal blade structure (Mask) using Ansys Fluent. The first design has modified the horizontal blade structure (mask) where it helps know and improve the results and takes more steps in valve modification.

IV. CONCLUSION

1. Gas and oil fuel engines reduce NOx and PM production.
2. Turbulent flow intensity can increase the duration of combustion with certain turbulence intensity.
3. Intake valve mask can make tumble motion in combustion camber
4. Intake valve vin can make swirl motion in combustion camber

Controlling the intake valve lift has been improved the air mixing process due to increased turbulence, caused by an increase in the flow rate of the intake.

Table 1. Velocity and Pressure for engine horizontal blade (mask)

CONTENT	MINIMUM	MAXIMUM
Velocity (m/s)	1.286	5.143
Pressure (pa)	-5.392	8.199

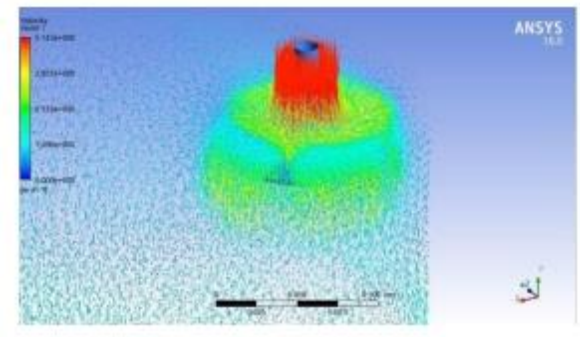

Figure 6. The flow velocity above the valve with a horizontal blade (mask)

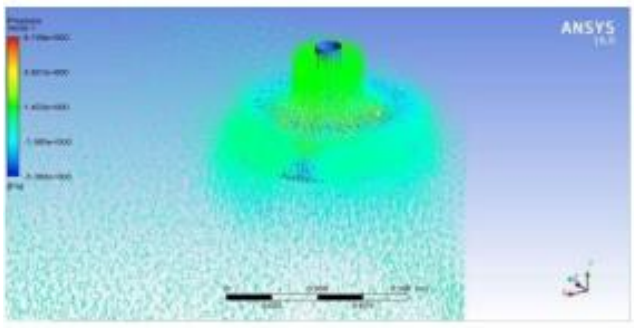

Figure 7. Distribution of pressure due to Fluid flow

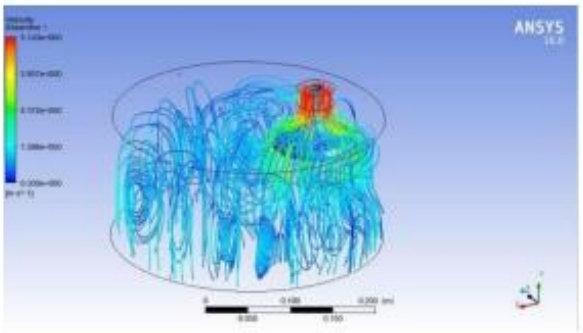

Figure 8. Flow shape of the horizontal valve blade design (mask)

REFERENCES

[1]Wan Nurdiyana Wan Mansor , Dual Fuel Engine Combustion And Emissions An Experimental Investigation Coupled With Computer Simulation , Disertation, Colorado State University, 2014

[2]P.G. Hill , D. Zhang, The effect of swirl and tumble on combustion in spark ignition engines., Department of Mechanical Engineering, The University of British Columbia, 1994.

[3] Abhilash M Bharadwaj , K Madhu, Seemanthini J, Vismay K G, Anand M Shivapuji & Aravind T6 , Study of Swirl and Tumble Motion using CFD, 2013

[4] Yerrennagoudaru H., Desai S.P., Effect of inlet air swirl on four stroke single cylinder diesel engine performance, International Journal of Recent Development in Engineering and Technology, Vol 2 (6), 2014.

[5] Yuesheng He, M.S., Effect Of Intake Primary Runner Blockages On Combustion Characteristics And Emissions In Spark Ignition Engines, Dissertation Graduate School Of The Ohio State University 2007.

[6]M D Raj Kamal, S.Kaliappan, S.Socrates, G.Jagadeesh Babu, CFD Analysis of Single Cylinder IC Engine Inlet Swirl Valve, Department of Mechanical Engineering, Velammal Institute of Technology, Chennai601204, India, 2017.

[7]Pablo Olmeda, Jaime Martín , Ricardo Novella, Ricardo Carreño, An adapted heat transfer model for engines with tumble motion, Universitat Politécnica de Valéncia, Camino de Vera, 2015.

[8]Adrian Clenci, Adrian Bîzîiac , Pierre Podevin , Georges Descombes , Michael Deligant and Rodica Niculescu, Idle Operation with Low Intake Valve Lift in a Port Fuel Injected Engine, University of Pitesti, str. Tg. din Vale nr. 1, Pitesti 110040, Romania, 2013.

[9]Karol Grab-Rogaliński, Stanisław Szwaja, Influence Of Intake Valve Closure Angle On Ic Engine Indicated Parameters, Czestochowa University Of Technology, Institute Of Thermal Machinery Armii Krajowej Street 21, 42-201 Czestochowa, Poland, 2015.

[10]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.

[11]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.

[12]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.

[13]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.

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

[15]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.

[16]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.

[17]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.

[18]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.

[19]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.

[20]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.

[21]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.

[22]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.

[23]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.

[24]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.

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

[26]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.

[27]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.

[28]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.

[29]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.

[30]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.

[31]H Prastowo, Semin, MB Zaman, Amiadji, TB Musrijadi, A Santoso, "Investigation of Fuel Flow Velocity on CNG Engine using New Injector", Asian Journal of Engineering and Technology 4 (2), 38 – 43, 2016.

[32].Gobinath,G.Mathiselvan,R.Kumarasubramanian, Analysis Of Effect Of Inlet Swirl In Four Stroke Single Cylinder Diesel Engine With Different Inlet Valve Geometries Using Cfd, Assistant Professors, Department Of Automobile Engineering, Sathyabama University, 2017.