VOL. 12, NO. 8, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

DISCHARGE AND FLOW COEFFICIENT ANALYSIS IN INTERNAL COMBUSTION ENGINE USING COMPUTATIONAL FLUID DYNAMICS SIMULATION

N. A. Mohamad Shafie[1], M. F. Muhamad Said[1], Z. Abdul Latiff[1] and S. Rajoo[2]
[1]Automotive Development Centre, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, UTM Johor Bahru, Malaysia
[2]Centre for Low Carbon Transport, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, UTM Johor Bahru, Malaysia
E-Mail: mdfarid@utm.my

ABSTRACT

Intake system is one of the crucial sub-systems in engine which can inflict significant effect on the air-fuel mixing, combustion, fuel consumption, as well as exhaust gases formation. There are many parameters that will influence engine performances. Good engine breathing is required to get better air flow rate to the engine. One of the methods includes the improvement of intake system by modifying the intake port design. This paper presents the application of Computational Fluid Dynamics analysis on two engines with different intake port shapes. Dimensionless parameters like discharge coefficient and flow coefficient are used to quantify the changes in intake flow at different valve lifts variation. Results show that when valve lift increases, this inflicted the increase in discharge coefficient because of greater mass flow rate of induction air. Both flow and discharge coefficient is dependent on valve lift. Flow analysis proved the relationship by computing the increase of flow coefficient as valve opening increase. The computed analysis shows that different intake port shapes does bring significant effect on discharge coefficient and flow coefficient.

Keywords: computational fluid dynamics, discharge coefficient, flow coefficient, intake valve.

INTRODUCTION

Improvement of engine system is crucial and has pressured the manufacturers to do thorough investigation on every single factor that can contribute for the development of efficient engine. Due to stringent emission restriction, researchers focus more on improving the engine sub-system that can control the combustion process, and consequently the formation of exhaust gases. Although fuel injection system does bring major influence in controlling combustion, intake system also contributes toward better quality of the combustion and enhances engine performance [1]. Alongside with improving combustion quality, in order to reduce fuel consumption and optimize engine performance, optimization of intake system is deemed as one of the major contributing factor. In particular, efficient intake system is a critical factor in obtaining optimum in-cylinder flow condition which will influence the processes in engine cycle [1, 2, 3].

From years ago until today, the improvement on intake system has been given the special attention by researchers and car manufacturers. There are numerous researches on every single system that can inflict high influence on intake system, which include the modification of intake geometry such as intake manifold, and applying different strategies on valve timing. Different intake manifolds are available in the market which is deemed designed to increase the efficiency of engine intake system. Meanwhile, in modification of valve timing, from decades ago there are many different types of innovative valve train system that has been implemented into the car system. For example, in the purpose of reducing the throttle effect, valve variable timing system has been largely applied into engine system [2]. In the improvement of intake air system by alternative strategies on valve timing, early intake closure and late intake valve closing is also capable in influencing the efficiency of engine system [4].

In the analysis of engine intake system, there are a number of parameters used to quantify the efficiency of induction process. Parameters which are always used include discharge coefficient and flow coefficient [2, 5, 6]. There are different types of research tools that have been used in the approach of investigating intake system. The most common method to determine discharge coefficient and flow coefficient is through Flow Bench Test machine. This machine by fact does give relatively accurate data on the in-cylinder flow condition which does not only include discharge and flow coefficient but also can be used to measure swirl and tumble. However Flow Bench test is not a very powerful tool to be used alone in engine design process. Thus, the advantages proved by the use of Computational Fluid Dynamics (CFD) simulation which emerged from years ago has picked the interest of researchers to apply this analysis tool in investigation of engine system.

CFD simulation has become one of the major interests by development engineer as the analysis method. This three-dimensional simulation does not only provide researchers with detailed and tremendous insight into what occurred during engine process, but also disclose the opportunity and potential area for improvement [7, 8, 9]. CFD simulation can be used to visualize the complex interaction and phenomenon inside engine cylinder involving intake air induction which is related to fluid dynamic, mixture of air and fuel, and thermodynamics changes due to fuel injection or spark ignition [10, 11]. The capabilities of three-dimensional CFD simulation is also proved in analysing other complex process which include chemical reaction and formation of exhaust gases emission. Chemical analysis embedded as a part of CFD

VOL. 12, NO. 8, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

simulation allows for relatively accurate prediction on the thermochemistry of engine system [12, 13].

The present paper aims to investigate the discharge coefficient and flow coefficient at different valve lifts using CFD simulation. Analysis is performed at two different engine models which have different intake port shapes in order to analyse the effect of intake port shapes on discharge coefficient and flow coefficient. Simulation is computed in ANSYS IC Engine where the solver is ANSYS Fluent and results are obtained from moving mesh simulation. This research focuses on investigating the trend of discharge and flow coefficient during the opening of intake valve.

LITERATURE REVIEW

Discharge coefficient

Discharge coefficient is also known as the parameter used to quantify the breathing capacity of engine intake system. When there is minimal difference between geometrical passage area and the effective flow area, this will improve the efficiency of the intake system. Low differences indicate low resistance, thus allow the respective engine to operate close to the desirable condition [2]. Discharge coefficient can be described as

$$C_D = \dot{m}_f / \dot{m}_t \tag{1}$$

Where $\dot{m}_f$ is real mass flow rate and $\dot{m}_t$ is theoretical mass flow rate at intake valve [14, 15]. Another way to determine the discharge coefficient is by effective area. Based on the assumption that in ideal orifice the mass flow rate is proportional to area, thus discharge coefficient is also proportional to area. Relationship between discharge coefficient and area is described as

$$C_D = A_f^e / A_f \tag{2}$$

Where A^e_f is effective area and A_f is the curtain area. Effective area is the area of imaginary frictionless orifice which can produce the real mass flow rate [14]. Equation for compressible flow through a flow restriction is usually referred in order to define the mass flow rate through the poppet valve [16]. Equation 3 shows that

$$C_D = \frac{\dot{m}}{A\frac{p_o}{(RT_0)^{1/2}}\left(\frac{p_T}{p_0}\right)^{1/\gamma}\left\{\frac{2\gamma}{\gamma-1}\left[1-\left(\frac{p_T}{p_o}\right)^{\frac{\gamma-1}{\gamma}}\right]\right\}^{1/2}} \tag{3}$$

Where A is relative area, R is gas constant, T_o is stagnation absolute temperature, P_0 is intake system pressure, P_T is cylinder pressure and γ is specific heat ratio [15].

Flow coefficient

Another parameter used to define the fluid dynamics efficiency of intake system is flow coefficient [2]. Both discharge coefficient and flow coefficient are important for the purpose of determining the capability of intake system to induce flow close to the demanded one. Proper mixing of fuel with air is dependent on intake system. In a more advanced system, engine that can shift between stratified mixture and homogenous mixture need to have a powerful intake system. By means of flow coefficient, the flow at valve is analysed by comparing the actual flow with incompressible constant density flow that is triggered by motion of piston [16]. Flow efficiency is commonly used to quantify the efficiency of the intake system because of its advantages of being easily determined. Relating the condition at valve to piston, equation 4 shows that

$$V_v^i A_f = V_p A_p \tag{4}$$

Where V_p is instantaneous piston velocity, A_p is the area of piston, V_v^i is ideal gas velocity at valve opening, and A_f is actual flow area at valve opening. This equation can be rewrite as

$$V_v^i = V_p\left(\frac{A_p}{A_v}\right)\left(\frac{A_v}{A_f}\right) = V_p\left(\frac{b}{D}\right)^2\left(\frac{A_v}{A_f}\right) \tag{5}$$

Where b is the bore size and D is diameter of valve. In order to obtain the flow coefficient, the equation above is rewrite as

$$V_V = V_p\left(\frac{b}{D}\right)^2\left(\frac{1}{C_v}\right) \tag{6}$$

where V_V is real flow velocity through the valve, and C_v is flow coefficient [14, 16].

METHODOLOGY

Engine modelling

For the analysis, two different engine models have been used to investigate the effect of intake port shape on discharge coefficient and flow coefficient. Figure-1 show both models used in this simulation.

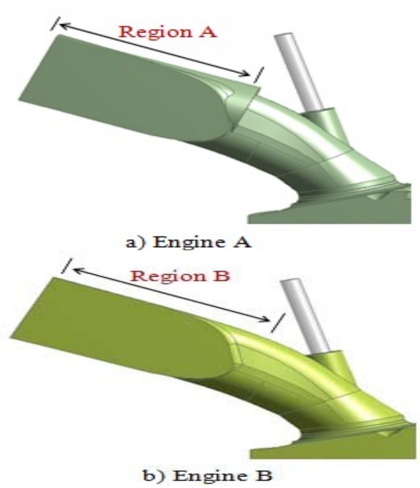


Figure-1. Engine models with different intake port shapes.

ARPN Journal of Engineering and Applied Sciences

Region A of engine A and region B of engine B are the only parts which differ from each other for both engines. Except for these two regions, geometry of other engine parts of engine A and engine B are the same. Engine A has higher inlet diameter compared to engine B, therefore causing higher volume in region A compared to region B. Total volume for intake port of engine model A is 60,108mm^3 while volume for intake port of engine B is 57,389mm^3. Comparing region A and region B, it can be observed that engine B has a smooth intake port shape compared to engine A. The engine bore is 84mm while compression ratio is 7.

CFD simulation

Simulation is computed in ANSYS IC Engine under Cold Flow simulation. In order to compute the result for intake flow analysis, there are steps need to be followed. Initial stage of Cold Flow simulation in ANSYS IC Engine involved inserting the engine parameters. Necessary engine parameters that need to be inserted into the system are shown in Table-1.

Table-1. Engine parameters.

Engine parameters	Value
Connecting rod length (mm)	130
Stroke (mm)	44
Engine speed (rpm)	3000
Minimum valve lift (mm)	0.2

Another required parameters include valves and piston motion profile. For the purpose of this simulation, valves and piston motion profile are obtained from 1-dimensional engine model which operates with similar parameters as this engine as described in Table-1 [17]. Engine model is imported into Design Modeller where engine parameters are automatically assigned to the geometry. In this simulation, analysis is focused on the intake valve opening, thus simulation is computed from 330 cad to 468 cad which is from opening of intake valve until before the intake valve starts to close again. The exact crank angle degree where intake valve starts to open is 338 cad, but for the purpose of simulating by Cold Flow simulation, it is necessary for users to start the simulation at least 5 cad before the intake valve opening begin.

Decomposition and meshing

In Design Modeller, engine is decomposed into different zone mainly intake port, exhaust port, piston and cylinder head. Decomposition of model is important because this allow better control on the quality of the mesh cells. Besides, for simulation of moving mesh, it is crucial for the model to be decomposed into different zones as there are regions of engine model which will move during simulation while other is static. Static region such inlet port and exhaust port are usually mesh with tetrahedral cell, while moving region such as near intake valve, exhaust valve and piston consists of hexahedral mesh. Figure-2 shows the decomposition and meshing of the engine model.

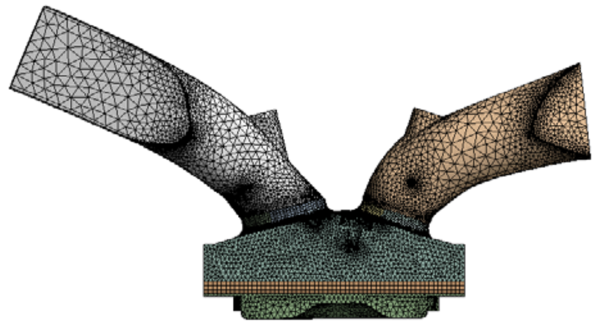


Figure-2. Decomposition and meshing of engine model.

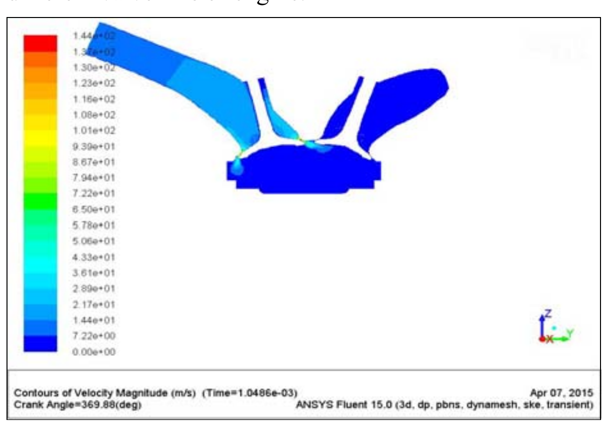
As depicted in Figure-2, it can be observed that different zone has different quality of mesh cell. In order to serve the purpose of analysing the intake flow during intake valve opening, the mesh cell is clearly more refined at the valve region. The total number of mesh cell for both models are around 680,000.

Solver setup

At 3000 rpm, the mass flow rate assigned at the inlet port is 0.0103kg/s and the inlet pressure is 98.9kPa. These values are obtained from the 1-dimensional engine model which operates with the same engine parameters as this model [17]. In this simulation, the turbulent model used is the standard k-epsilon model with Standard Wall Function assigned for Near-Wall Treatment. The computation for this Cold-Flow simulation is based on pressure-correction method and PISO scheme is used to solve the simulation. For gradient, Green-Gauss Node based is selected in the analysis setup. Second order upwind as the spatial discreticationis set for solving the density, momentum and turbulent kinetic energy in order to obtain more accurate result. Initial turbulent intensity is set to 5 % which is sufficient to define turbulent fluid flow in the simulation [1].

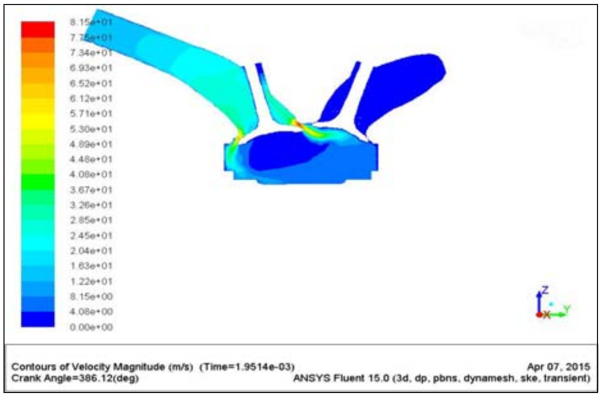
RESULT AND DISCUSSION

Figure-3 shows the velocity magnitude contour at different valve lifts of engine.

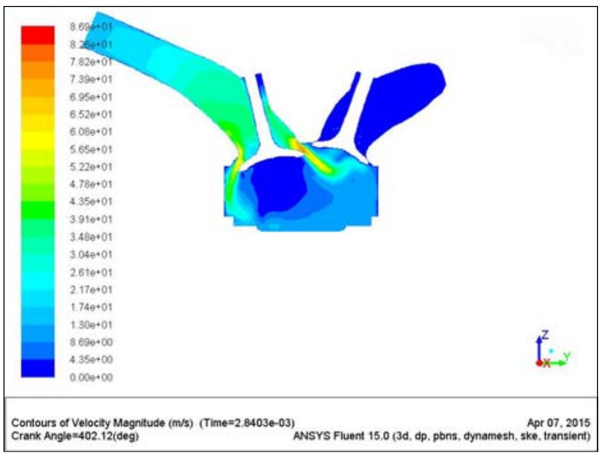


a) Lift = 1mm

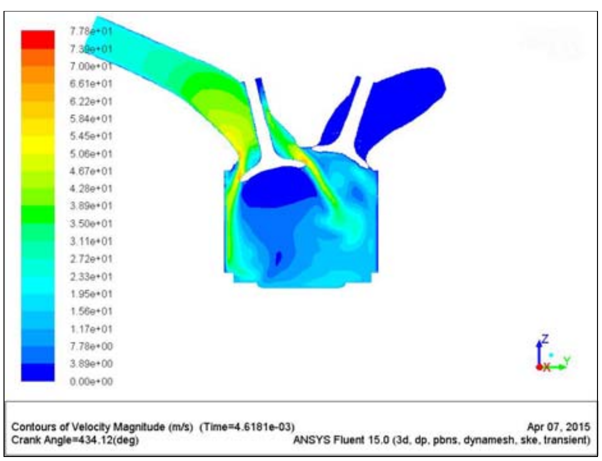
ARPN Journal of Engineering and Applied Sciences



b) Lift = 3mm



c) Lift = 5mm



d) Lift = 8mm

Figure-3. Velocity magnitude contour at different valve lifts.

Figure-3 clearly shows that the velocity of air is higher around the intake valve regions. Small passage for air flow due to the opening of intake valve caused high velocity airstream around intake valve. At 1mm valve lift, high velocity air starts to enter the cylinder. However, due to the small opening of 1mm, there is only small contour of high velocity magnitude around intake valve. At 5mm and 8mm valve lift, results show the flow of high velocity air into engine cylinder. The velocity magnitude is higher near the wall and the middle of engine cylinder due to the position of intake valve. Along the engine intake stroke, high velocity air continues to reach the furthest region from intake valve and the average velocity of air inside cylinder also continues to increase.

Result of intake flow is presented in term of discharge coefficient and flow coefficient. Figure 4 shows the discharge coefficient versus L/D (ratio of valve lift to valve diameter).

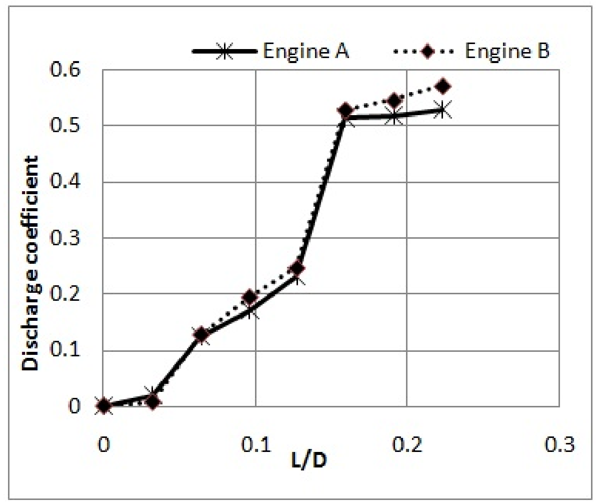


Figure-4. Discharge coefficient vs L/D.

Figure-4 shows that when the L/D value increases, the discharge coefficient value also increases. At low valve lift, discharge coefficient value is low because the flow remains attached to the valve head and seat, thus influencing the flow that pass through the curtain area of valve. Due to the increase in the opening of intake valve along intake stroke, the restriction on the intake flow also becomes less. When the valve lifts continue to increase, the area for the intake air to flow into engine cylinder is higher and mass flow rate of air flowing near the valve also increase, therefore leading to the increase in discharge coefficient. Comparing engine A and engine B, discharge coefficient value at different valve lifts increase with the similar trend. At 1mm valve lift, discharge coefficient for engine B is smaller than discharge coefficient of engine A by 1.1%. As L/D increase from 1mm to 9mm valve lift, engine B continues to have greater discharge coefficient compared to engine A. At maximum valve lift which is 9 mm, the difference becomes larger where discharge coefficient at intake valve for engine B surpass discharge coefficient of engine A by 3.7%. This result reflects the continuous raise in the mass flow rate that enters the engine cylinder.

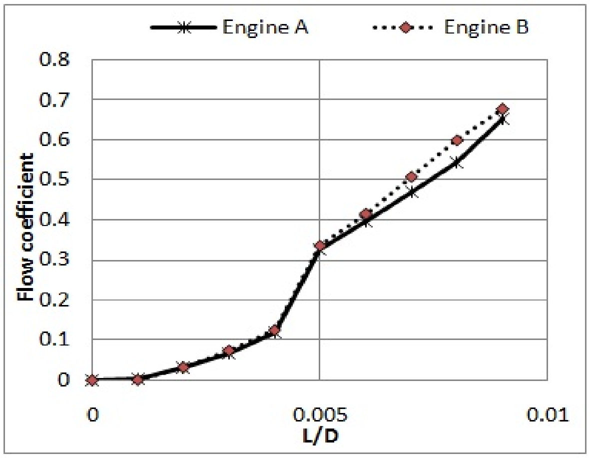


Figure-5. Flow coefficient vs L/D.

Figure-5 shows the relationship between flow coefficient and ratio of valve lift to valve diameter. From the result, it shows that flow coefficient increase when L/D increase. From 0mm to 1mm opening, the increase in flow coefficient is very small. This is because the 1mm valve opening is still small for the induced flow to pass through. At low valve lift, flow remains attached to the valve seat due to viscous condition, thus causing low value of flow coefficient. From 1mm to 9mm valve lift, flow coefficient increases at higher rate until the maximum opening of intake valve. For both engine A and engine B, flow coefficient value increases with the similar trend. The highest difference in flow coefficient is computed at 8mm valve lift where flow coefficient of engine B is higher by 9.61%.

CONCLUSIONS

This research focused on the fluid dynamic behaviour in engine with different intake port shapes. In order to analyse the system, analysis is done by CFD simulation to compute the flow condition by means of discharge coefficient and flow coefficient. From the analysis, it is revealed that discharge coefficient increase significantly with the increase in valve lift. At low valve lift, the computed discharge coefficient is low. This is because of the resistance to the flow due the attachment to the valve seat and valve head. As the valve lift continue to increase, larger intake valve opening allow for greater induction mass flow rate which causes the increase of discharge coefficient. At maximum valve lift, the engine system produce the highest discharge coefficient as the restriction on flow at maximum valve lift is the lowest. From the computational analysis, engine B which has smoother shape of the intake port is computed with larger discharge coefficient value. With a small modification of intake port cross sectional area, this promoted to the increment of discharge coefficient by 3.7%, thus proving the influence of intake port shape on the discharge coefficient.

In term of flow coefficient, the computed data shows that flow coefficient is dependent on valve lift. As the opening of intake valve influence the flow at the valve seat and the curtain area, higher intake valve opening cause the increment in the flow coefficient. Difference in intake port shapes of engine has cause difference in flow coefficient by 9.61%. Even at minor difference of intake port shape, it still influences the intake flow of the engine. Improvement of the intake port shape may allow for greater significant improvement of the induction of flow into engine.

ACKNOWLEDGEMENTS

The authors acknowledge the financial support from Universiti Teknologi Malaysia (UTM) under the research university grant Q.J130000.2409.01G53. Thanks also to the Automotive Development Centre (ADC) UTM for the technical and financial supports.

REFERENCES

[1] Payri F., Benajes J., Margot X. and Gil A. 2004. CFD modelling of the in-cylinder flow in direct-injection diesel engines. Computers and Fluids.33. pp. 995-1021.

[2] Algieri A. 2013. Comparative analysis of the fluid dynamic efficiency of standard and alternative intake strategies for multivalve spark-ignition engines. International Journal of Engineering and Technology. 2(2): 140-148.

[3] Abdalla M. O. and Nagarajan T. 2015. A computational study of the actuation speed of the hydraulic cylinder under different ports' sizes and configurations. Journal of Engineering Science and Technology.10(2): 160-173.

[4] Zhao H. 2010. Advanced Direct Injection Combustion Engine Technologies and Development (1sted.). Cambridge: Woodhead Publishing Limited.

[5] Zenkin V. and Kuleshov A. 2012. Profiling of inlet ports of Z-engine. AumetOy.

[6] Sorian, B. S., Rech C., Zancanaro F. V. and Vielmo H. A. 2012. Steady discharge coefficient in internal combustion engine. 14th Brazilian Congress on Thermal Sciences and Engineering. Brazil, Rio de Janeiro.

[7] Basshuysen R. and Schafer F. 2004. Internal Combustion Engine Handbook: Basics, Components, System, and Perspectives. United States of America: SAE.

[8] Pathak Y. R., Deore K. D. and Maharu P. V. 2014. In cylinder cold flow CFD simulation of IC engine using hybrid approach. International Journal of Research in Engineering and Technology. 3(8): 16-21.

[9] Chiodi M. 2011. An Innovative 3D-CFD-Approach towards Virtual Development of Internal Combustion Engine (1st ed.). Germany: Vieweg+Teubner.

[10] ANSYS, Inc. 2013. Internal combustion engine in workbench. Retrieved March 10, 2015, from http://www.ansys.com.

[11] Banaeizadeh A., Afshari A., Schock, H. and Jaberi F. 2015. Large-eddy simulations of turbulent flows in internal combustion engines. International Journal of Heat and Mass Transfer.60: 781-796.

[12] Balakrishnan V. K., Morton S., Radavich, P., Sivagaminathan N. and Gopalakrishnan N. 2010. Air flow and charge motion study of engine intake port. 37th National and 4th International Conference on Fluid Mechanics and Fluid Power. Chennai, India. pp. 1-9.

[13] Paul B., and Ganesan V. 2010. Flow field development in a direct injection diesel engine with different manifolds. International Journal of Engineering, Science and Technology. 2(1): 80-91.

[14] Lumley J. J. 1999. Engines: An Introduction (1st ed.). Cambridge: Cambridge University Press.

[15] Kumar C. R. and Nagarajan G. 2012. Investigation of flow during intake stroke of a single cylinder internal combustion engine. ARPN Journal of Engineering and Applied Sciences. 7 (2): 180-186.

[16] Heywood J. B. 1988. Internal Combustion Engine Fundamentals (1st ed.). United States: McGraw-Hill, Inc.

[17] Muhamad S. M. F., Abdul A., Abdul L. Z. and Mahmoudzadeh A. A. 2014. Investigation of cylinder deactivation (CDA) strategies on part load conditions. SAE Technical Paper 2014-01-2549.