VOL. 12, NO. 7, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

EFFECTS OF ASYMMETRIC INTAKE VALVE LIFT CONFIGURATION TOWARDS IN-CYLINDER AIR FLOW BEHAVIOR

Mohd Aizad Sazrul Sabrudin, Mohd Farid Muhamad Said and Zulkarnain Abdul Latiff
Automotive Development Centre, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, UTM Johor Bahru, Malaysia
E-Mail: mdfarid@utm.my

ABSTRACT

Air motion in a cylinder of a spark ignition engine affects the air-fuel mixing behavior, combustion quality and the production of the exhaust gas emission. With upcoming stringent market regulations for petrol engines, it is necessary to enhance air-fuel mixing for proper combustion. Air-fuel mixing in an engine combustion chamber is studied by assessing the induced air flow swirl motion. Swirl is a rotational motion of a bulk mass within cylinder. Swirl is generated by shaping and contouring the intake manifold, valve ports and even the piston face. Swirl enhances air-fuel mixing and helps to spread flame-front during combustion. The objective of this paper is to analyze the impact of the asymmetric intake valve lift configurations towards in-cylinder air flow swirl behavior. The study is done on 4 cylinders, 1.3L engine. The engine has 2 intake valves in every cylinder. Computational Fluid Dynamics (CFD) was used as a tool to assess the swirl motion in the case study models. At the end of this paper, the characteristics of the swirl flow motion on every case study models is studied by measuring the swirl ratio value inside the combustion chamber. Also, the pattern of the swirling flow inside the combustion chamber is studied by analyzing the velocity vector and turbulent kinetic energy plots.

Keywords: induced air flow, asymmetric valve lift, computational fluid dynamics, swirl flow.

INTRODUCTION

It is now relatively common for automotive manufacturers to produce a vehicle with an efficient engine to achieve low energy consumption and emission. Many areas have been explored to further enhance the engine performance. For example, Gasoline Direct Injection (GDI) and Turbo-Downsizing engine which are already in the market [1].

In this paper, the characteristics of induced air flow into the combustion chamber are studied to enhance the air-fuel mixture performance inside the combustion chamber. Many ways can be done to enhance the induced air flow characteristics inside the combustion chamber. For example, the induced air flow characteristics can be enhanced by changing the geometry and the arrangement of the intake ports in the cylinder head [2-3]. The induced air flow characteristics can also be improved by shaping the piston crown shape [4]. In this paper, the approach is by adopting the asymmetric intake valve lift configuration during air intake process. This will improve the induced air flow swirling motion which will improve the air-fuel mixture performance thus improve the fuel burning time. By improving the fuel burning time, the fuel consumption and exhaust emission can be reduced significantly.

The asymmetric intake valve lift operates by having one of the valves to operate at higher lift compared to the other one at all time. These will enhance the swirling motion of the induced air flow when the air flow hits the valves. The aim of this asymmetric intake valve lift configurations is to generate turbulence flow inside the combustion chamber and to give homogenous mixture that can enhance the flame front speed to improve combustion process. This is one of the key factors for producing exhausts containing low HC and NOx levels and for decreasing the specific fuel consumption [1].

Computational Fluid Dynamics (CFD) was used to assess the asymmetric valve lift performance towards swirl flow motion characteristics inside the combustion chamber. A steady-state port flow in-cylinder CFD simulation domain was setup to analyze the case study models.

ASYMMETRIC VALVE LIFT CONFIGURATION

In this study, an engine with two intake valves per cylinder is used. The intake valves are named as Valve 1 and Valve 2. The asymmetric configuration is defined when the intake valve opening is not synchronized. In this case, Valve 1 will always have higher lift position to simulate the asymmetric configuration. Valve 2 will remain at a same lift position which is 2 mm lift.

There are 3 case studies have been tested and the details of case studies is shown in Table-1.

Table-1. Asymmetric valve lift configuration of the case studies.

Case name	Valve 1	Valve 2
Base	2mm	2 mm
Case 1	4mm	
Case 2	6 mm	
Case 3	8 mm	

The case study models are visualized as shown in Figure-1.

VOL. 12, NO. 7, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

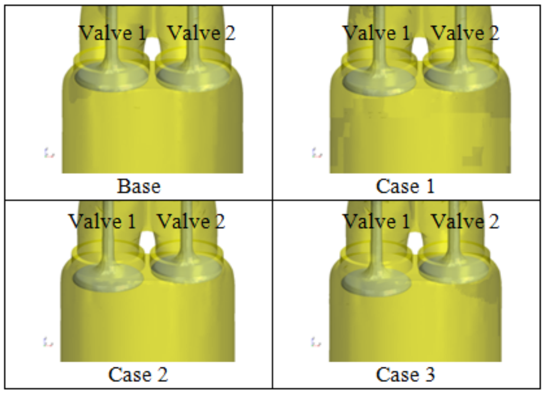

Figure-1. Visualization of the asymmetric intake valve lift configuration of the case study models.

SWIRL FLOW

Swirl is the rotational flow of charge within the cylinder about its axis (Figure-2). Swirl is generated by constructing the intake system to give a tangential component to the intake flow as it enters the combustion chamber. This is done by shaping and contouring the intake manifold, valve ports and even the piston face. Swirl greatly enhances the mixing of air and fuel to give a homogeneous mixture in the very short time available for those in modern high speed engines. It is also a main mechanism for spreading the flame front during the combustion process.

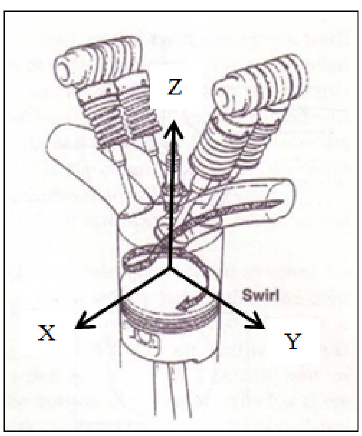

Figure-2. Visualization of swirl flow [5].

In this paper, the swirl value was calculated using an equation provided by the CFD software; namely STAR-CCM+ v9.04. The equation is described in Figure-3.

$$S_z = \frac{M_z}{I_z} = \frac{\sum_{Cells} \rho_i V_i [(X_i - X_m)v_i - (Y_i - Y_m)u_i]}{2\pi \frac{N}{60} \sum_{Cells} \rho_i V_i [(Y_i - Y_m)^2 + (X_i - X_m)^2]}$$

Figure-3. Swirl ratio equation [6].

Where;

v_i	Velocity at cell *i*
N	Rotational speed
M_z	Angular momentum about the z-axes
I_z	Mass moment of inertia about the z-axes
X_i, Y_i	Centroid coordinate of cell *i*
X_m, Y_m	Center of mass/origin of the cylinder

CFD MODELING

In this study, Computational Fluid Dynamics (CFD) is used to study the air flow behaviour inside the combustion chamber with asymmetric intake valve lift configurations. The CFD domain is a steady-state port flow simulation setup that consists of inlet boundary, intake port, intake valve, combustion chamber and the outlet as shown in Figure-4. Mass flow data for inlet boundary was extracted from steady-state flowbench experiment data at engine speed, N=3000 RPM. Inlet mass flow data for all case study models is shown in Table-2. Since this is a steady-state analysis, the outlet boundary is set at piston crown to simulate air flow behaviour inside the combustion chamber. In this study, no piston movement was modelled (steady-state analysis). The simulation only focused on the characteristics of air entering the combustion chamber. Also, the analysis only focused on cold flow analysis which involves modelling the air flow without fuel injection and combustion effect.

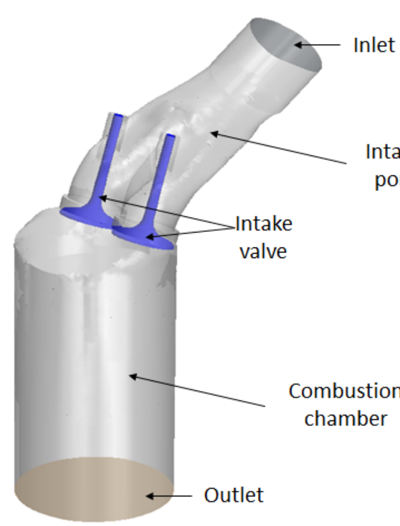

Figure-4. In-cylinder simulation domain.

Table-2. Inlet mass flow rate value for case study models.

Case	Mass flow rate
Base	0.027 kg/s
Case 1	0.0405 kg/s
Case 2	0.0676 kg/s
Case 3	0.1267 kg/s

To model the turbulence flow inside combustion chamber, k-epsilon RANS equation was selected as Physics continua [7]. Cell size of 4 mm was selected as a base mesh size for the simulation domain. Volume refinements area was set especially at high turbulence flow area especially at the small gaps between intake valve and intake port to enhance model accuracy and convergence. The extruded mesh was set at Inlet and Outlet boundary to ensure no reverse flow will exist at Inlet and Outlet boundary. Figure-5 shows the cross-section area of the simulation domain after the volumesh was generated. Eight boundary layers were set to model the flow behaviour near the combustion chamber wall with boundary layer thickness equal to 4 mm. From these local and global mesh settings, the cell count for the case study models is in the range of 493k – 527k which is acceptable for this study. Details of the simulation Physics and Mesh setup is shown in Table-3.

Table-3. Simulation physics and mesh setup.

Turbulence model	k-epsilon RANS
Cell base size	4 mm
No. of boundary layer	8 layers
Boundary layer thickness	4 mm
Cell count: Base Case 1 Case 2 Case 3	 493,530 506,818 516,119 527,981

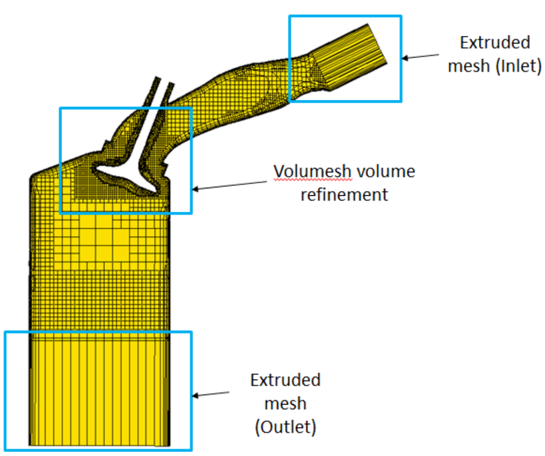

Figure-5. Volumesh result of the simulation domain.

RESULTS

From the simulation results, by having the asymmetric valve lift opening configurations, it helps to increased swirl ratio value significantly as shown in Table-4 below. Case 1 managed to increase the swirl ratio value by 24% from 0.421 to 0.524 compared to Base (non-asymmetric). In Case 2, the swirl ratio value has increased by 29% when compared to Case 1. The highest swirl ratio value increment is found in Case 3 as the swirl ratio value has increased by 86% as compared to Case 2.

Table-4. Swirl ratio result.

Case name	Swirl ratio	Δ%
Base	0.421	-
Case 1	0.524	24%
Case 2	0.676	29%
Case 3	1.259	86%

The increment of the swirl ratio value is corresponded to the increment of the intake valve lift as illustrated in Figure-6. As the valve lift increased, the swirl ratio value also increased.

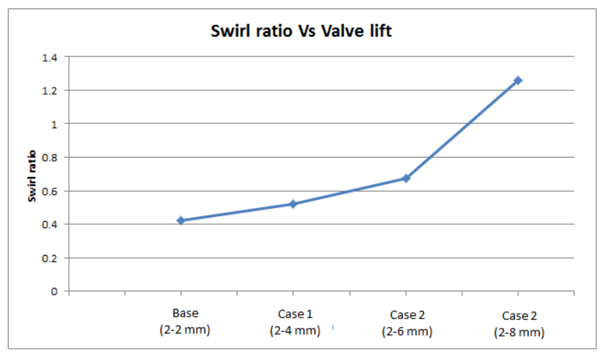

Figure-6. Effects of swirl ratio value on different asymmetric intake valve lift configuration.

DISCUSSIONS

Qualitative observation

Swirl flow behaviour inside the combustion chamber has also been studied qualitatively. This is done by creating a swirl monitoring plane inside the combustion chamber (Figure-7). Two items were monitored; 1) Flow velocity and 2) Turbulent kinetic energy. The flow velocity was illustrated using the velocity vector plot to visualize the swirl motion of the air flow inside the combustion chamber. The turbulent kinetic energy was illustrated using the turbulent kinetic energy scalar plot to observe the characteristic of air-fuel mixture inside the combustion chamber.

VOL. 12, NO. 7, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

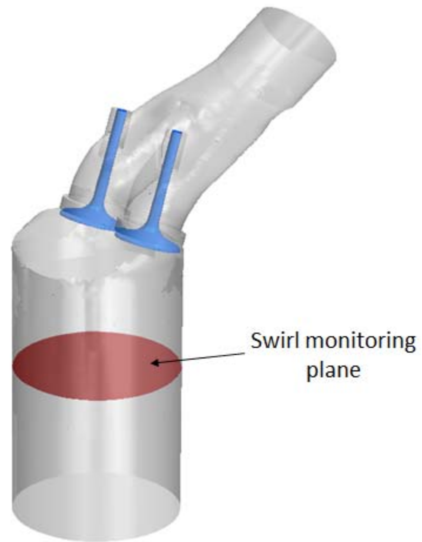

Figure-7. Swirl monitoring plane.

Velocity vector plot

Figure-8 shows the velocity vector plot for Base, Case 1, Case 2 and Case 3. Red colour indicates high velocity flow while blue colour indicates low velocity flow. In Base case, one swirl motion was observed and it is coloured by blue arrows (low velocity). There is also an area where the flow attempted to make a swirl at the right side of the combustion chamber. However, the flow at this area is weak thus resulted an oval shape flow movement. Both behaviour is highlighted in the dotted box labelled 'A' and 'B' below. This flow behaviour shows that the Base case (non-asymmetric) can only produce a weak swirl flow motion. Enhancement in cylinder head design is required to increase flow efficiency. By increasing the Valve 1 lift from 2 mm to 4 mm and maintaining the Valve 2 lift at 2 mm in Case 1, the swirl motion flow is energized and the diameter of the flow has significantly increased (as highlighted by box 'C'). Number of area of the swirl flow also has increased as highlighted by box 'D' below. In Case 2, when Valve 1 lift is increased to 6 mm, the swirl flow pattern becomes more obvious. The asymmetric valve lift configuration effect is more significant since it helps to create more swirling flow when the air flow hits the valve during air intake process. The velocity of the swirl flow also has increased due to the bigger valve lift opening that allows more air to enter the combustion chamber. The swirl flow energy in Case 3 has significantly increased due to bigger valve opening. However, the high energy flow has reduced the number of swirl area when compared to Case 2.

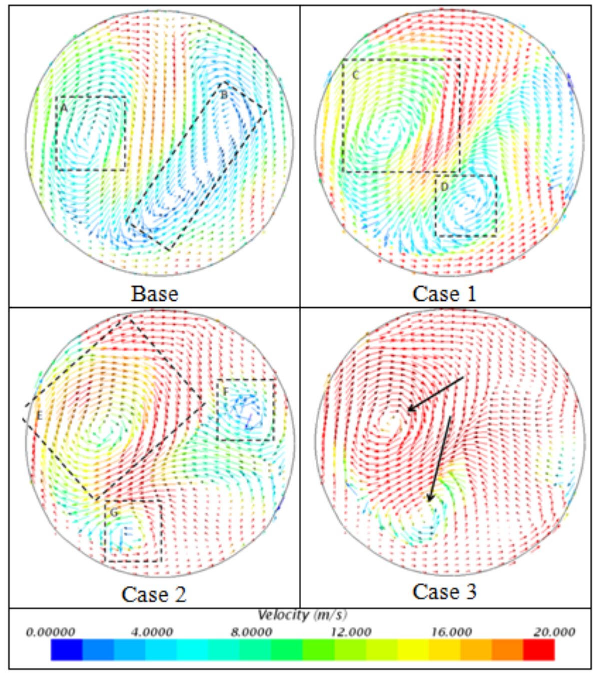

Figure-8. Velocity vector plot on swirl monitoring plane.

Turbulent kinetic energy

Turbulent kinetic energy (TKE) plot is one of the methods that can be used to visualize the flow mixture behaviour inside the combustion chamber. As for this study, the TKE plot is used to assess the effectiveness of the asymmetric valve lift configuration towards the flow mixture performance. The flow in the combustion chamber should have high intensity of turbulent energy to ensure fast burning. Also, the distribution of high turbulent area should be in good uniformity to ensure uniform burning process across the combustion chamber. Figure-9 shows the TKE plot at swirl monitoring plane for all case studies (Base, Case 1, Case 2 and Case 3).

The behaviour of the turbulent flow in all case studies is similar whereby most of the high turbulent flow area is situated at the bottom left of the plot (red shaded area). The turbulent energy increment is parallel with the valve lift increment. The asymmetric valve lift configuration does not give much impact in creating new turbulent flow area in the combustion chamber as in all cases, the high TKE area are situated at the same location. The only difference is the TKE magnitude, where high valve lift case gives bigger TKE value compared to low valve lift case. This may result in an imbalance burning area where there will be an area having good burning process and there will be an area where incomplete fuel burning will occur.

VOL. 12, NO. 7, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

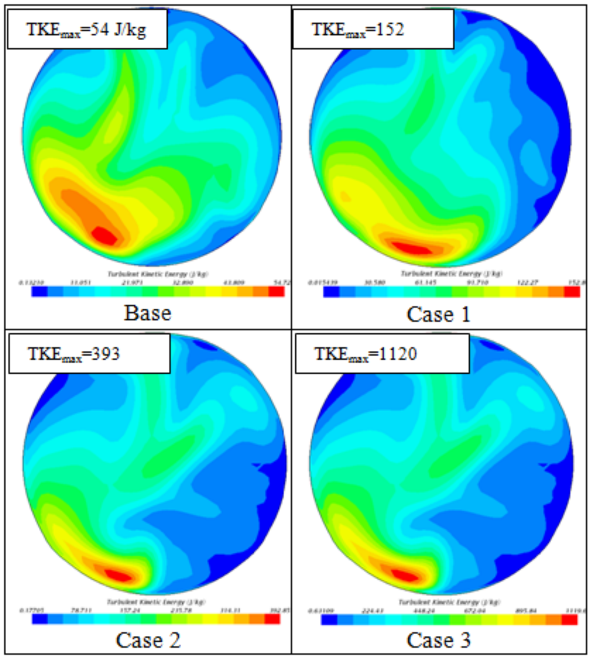

Figure-9. Turbulent kinetic energy plot on swirl monitoring plane.

CONCLUSIONS

The effect of the asymmetric intake valve lift configurations towards in-cylinder air flow behaviour is studied using Computational Fluid Dynamics. The asymmetric valve lift configurations managed to increase swirl ratio value of the air flow that enters the combustion chamber. The improvement is in the range of 24% to 86% where the Case 3 shows highest improvement in terms of swirl ratio value. The qualitative study also been carried out to observe the flow behaviour inside the combustion chamber. From the qualitative observation, the swirling flow is plotted and Case 3 is found to be the best case as it has more swirl flow area as compared to other case studies. The qualitative observation also shows that the asymmetric valve lift configuration does not help in creating new turbulent flow area in the combustion chamber. It only helps to energize the flow by increasing the TKE magnitude (increase flow energy level) which is good for fast fuel burning operation.

ACKNOWLEDGEMENTS

The authors acknowledge the financial support from Universiti Teknologi Malaysia (UTM) and Ministry of Education Malaysia (MOE) under the FRGS grant R.J130000.7809.4F174. Thanks also to the Automotive Development Centre (ADC) UTM for the technical and financial supports.

REFERENCES

[1] Battistoni M. and Mariani F. 2009. Fluid dynamic study of unthrottled part load SI engine operations with asymmetric valve lifts (No. 2009-24-0017). SAE Technical Paper.

[2] Li Y. 2014. A new estimation of swirl ratio from steady flow rig testing (No. 2014-01-2587). SAE Technical Paper.

[3] Sureshkumar J., Ganesan V., Mallikarjuna J. M., and Govindarajan S. 2013. Effect of piston crown shape on in-cylinder flow characteristics in a direct injection engine-A CFD study (No. 2013-01-2797). SAE Technical Paper.

[4] Li Y., Zhao H., Peng Z. and Ladommatos N. 2001. Analysis of tumble and swirl motions in a four-valve SI engine (No. 2001-01-3555). SAE Technical Paper.

[5] https://ieeexplore.org/iee_pilot/articles

[6] Star-CCM+ V7.04 user guide.

[7] Fontanesi S., Cicalese G., and Severi E. 2013. Analysis of turbulence model effect on the characterization of the in-cylinder flow field in a HSDI diesel engine (No. 2013-01-1107). SAE Technical Paper.