Jurnal Teknos-2k Vol. 15, No. 2, Juli 2015

Flow Analysis of Intake Port in Internal Combustion Engine

Mohd Zahin Mohd Shahrin, Mohd Farid Muhamad Said*, Nur Adila Mohamad Shafie
Automotive Development Centre, Faculty of Mechanical Engineering,
Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Phone: +6075535449,
Fax: +6075535811
E-mail: mdfarid@utm.my

Abstract

The purpose of this study is to analyse the development of swirl and tumble motion inside internal combustion engine by using computational fluid dynamic (CFD) modelling. Computational fluid dynamic software, FLUENT was used to simulate the swirl and tumble motion inside combustion chamber during intake stroke. Three different intake port design geometries were used to investigate the flow inside the engine cylinders. The original combustion chamber of 1.6L SI engine was used as benchmark and compared with the calculated swirl and tumble ratio in the other two modified intake port design. Analysis works of the intake stroke were performed. Here, the flow analysis using static model illustrates similarity compared to dynamic model. The effect of intake port geometries and intake valve position was analysed. Results were compared between different intake port geometries and intake valve position in term of swirl and tumble ratio. It is discovered that with similar parameters of the engines, intake port geometries with the biggest diameter give the lowest value of swirl and tumble ratio, while by increasing the position of intake valve, value of swirl and tumble ratio increases. The simulation results confirmed that different intake port designs affect the development of swirl and tumble inside internal combustion engine based on calculated swirl and tumble ratio.

***Keyword**: swirl, tumble, intake port, intake valve position*

1. Introduction

In most of passenger vehicle, internal combustion engine is one of the important factors that attributes to better vehicle performance. The process occur in this engine is basically the combustion of fuel and air mixture to provide movement for piston where the engine typically function on gasoline or diesel fuel. Fluid motion within the cylinder of internal combustion engines has fundamental effect on combustion of air-fuel mixture and hence on engine performance (Pulkrabek, 2004). Since, the internal combustion engine is one of the keyway that is related in improving the vehicle performance, there are many aspects that have been studied in order to improve the performance of internal combustion engine. One of the techniques for producing a strong intake airflow pattern is the design of port and valve to create either a strong tumble motion or a strong swirl motion inside the cylinder, which will be sustained well into the end of the compression stroke (Selvaraj, 2011). In this project, parameter that has been considered is intake port design in order to indicate generation of swirl and tumble behaviour based on the modification of intake port design.

Basically, swirl and tumble are types of fluid motion that are created in combustion chamber and it is important for enhancing air-fuel mixing and increasing combustion speed and efficiency (Selvaraj, 2011; Jasmi, 2012). So, by doing the modification on intake port geometry, it will aid in determining on how the development of swirl and tumble occur, and its behaviour in combustion chamber. Swirl and tumble ratio will be determined and studied based on swirl and tumble behaviour created. These ratios are dimensionless parameter used to quantify rotational motion within the cylinder (Shuisheng, *et al*., 2012; Federico, *et al*., 2014; Nagarajan and Kumar 2012).

The main goal of this project is to study the swirl and tumble flow analysis of Internal Combustion Engine and its effect on intake port design. The study also focuses on spark-ignition engine. The swirl and tumble ratio obtained based on different intake port design will be compared.

2. Methodology

The methodology used in performing the present work is as follows:

1. Intake Port Design – There are modifications on intake port geometry that have been made in order to study the effect of intake port design on swirl and tumble behaviour inside combustion chamber. Several designs of intake port have been create in this study to show the comparison between these designs in term of flow behaviour inside cylinder. The parameters that are highlighted in these intake port design study are variation of intake port diameter and variation of valve lift.

2. Variation of Intake Port diameter – Intake Port diameter will be created for different size and dimension. By changing this parameter, it will affect the amount of air that is allowed to enter the intake port. 3D CAD model will be constructed for these types of models and simulation will be conducted on these designs. The results from simulation will be compared with each other while swirl and tumble ratio also will be calculated.

3. Variation of valve lift – There are two intake valves provided in the cylinder and both of this valve will be taken as variable in their position of valve lift. The investigation on position of intake valve lift will be conducted by redrawing 3D CAD model of the intake port design. Then, result of swirl and tumble behaviour will be compared.

4. Intake port modification – In order to increase the flow that enters the cylinder head, modification through the wall of intake port is done.

Figure 1 below shows the hierarchy of this project:

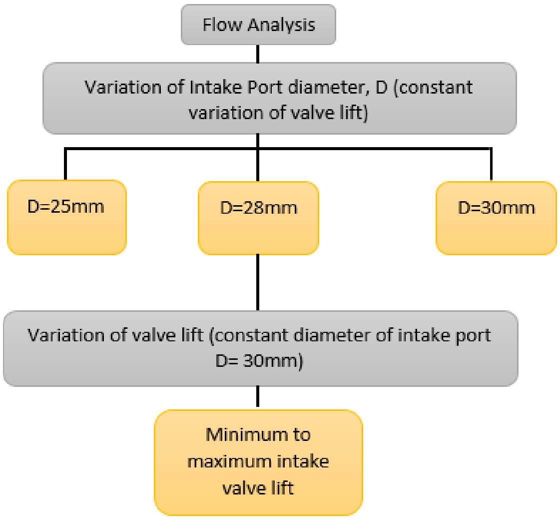

Figure 1. Hierarchy of the project

Based on the chart above, there are two case study that are conducted in this research which are variation of Intake Port diameter, D (constant variation of valve lift) and variation of valve lift (constant diameter of intake port, D= 30 mm).

2.1 Engine description

All of the engine parameters are referred to 1.6L SI engine as a reference in this study. Table 1 below show the engine cylinder parameters including bore and stroke dimension. Figure 2 show the illustration of bore and stroke in engine cylinder.

Table 1. Engine specification

Parameter	**Specification**
Bore diameter	76 mm
Stroke length	88 mm
Peak engine torque	148 Nm @ 4500rpm
Peak engine power	82 kW @ 6000rpm

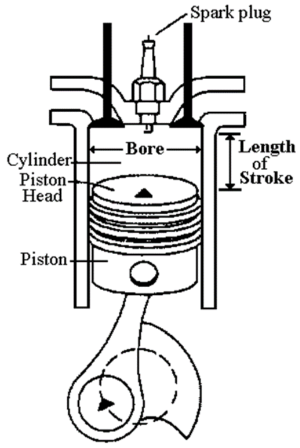

Figure 2. Bore vs length of stroke position

2.2 Modelling setup

The model is developed in SolidWorks software for both case studies which is variation of intake port diameter and intake valve position as illustrated in Figure 3 below:

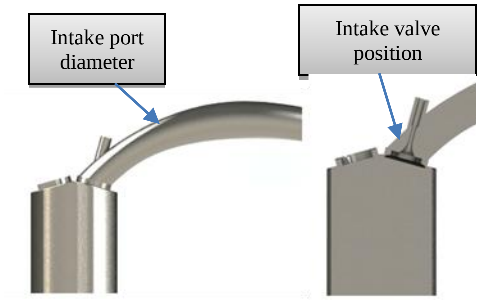

Figure 3. Simulation model

2.3 Meshing study

When the CAD modelling of intake port is complete, the model is imported directly to CFD ANSYS Workbench software for meshing study. For this study, static mesh is used to compute the simulation of flow streamline in the cylinder. Three name of selection face is assigned to this model which is inlet, cylinder and piston. Figure 4 below show the mesh for 30mm diameter intake port in ANSYS Mesh Modeller.

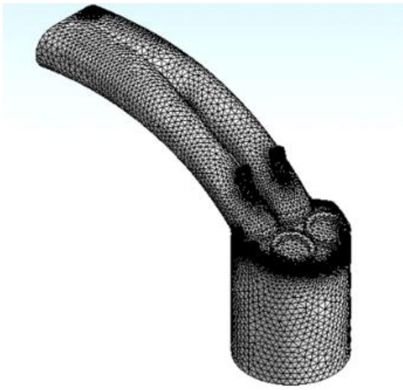

Figure 3: Mesh of 30 mm intake port diameter model

Study on meshing process is important to give better result of analysis in this research. Based on that, mesh independence test is done to analyse the best meshing number required and give an optimum number of element in meshing process. Figure 5 below shows the graph of tangential velocity against number of element.

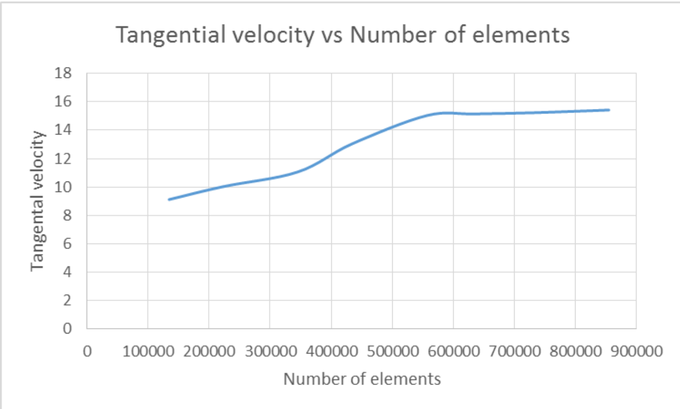

Figure 5. Graph of tangential velocity vs. number of elements

Based on the graph above, value of tangential velocity become constant at 500,000 number of elements. Thus, the same meshing size which indicates its number of elements for all geometry design which is about 500000 will be used in meshing study.

2.4 Boundary condition

After completing the meshing step in ANSYS Mesh Modeller, the geometry is imported to ANSYS Fluent to set up all the boundary condition needed in the analysis. This step is important before the analysis or calculation is computed because some important parameters inside cylinder are need to be identified first as preferred in the study.

In viscous model, RNG k- epsilon model is used with standard wall function because of the effect of swirl and tumble flow will enhance its accuracy compared to standard K-epsilon model (Basha and Gopal, 2009). Table 2 below show the boundary condition applied in this study.

Table 2. Boundary conditions

Parameter	Value
Inlet velocity	8.9 m/s
Inlet pressure	98 kPa
Engine speed	3000 rpm@12 m/s

At cylinder wall, the analysis assumes that there are no shear force occurs along it, thus rigid wall is set on cylinder wall.

2.5 Analysis of data

After the mesh and setup boundary condition complete, analysis of the data is conducted by selecting Hybrid initialization. 500 number of iteration is applied to run the calculation for every intake port design. Basically the calculation step needed to be run until the result is converged. Generation of velocity streamline is observe when the calculation is finish**.**

3. Result and Discussion

3.1 Case study 1: Variation of intake port diameter

There are three models which differ in term of their intake port diameters which are 25 mm, 28 mm and 30 mm. All of the pressure contours and velocity streamline are discussed for every model to observe the difference on the pressure and velocity generated. There are 3 plane shown to be discussed in one model to observe the pressure and velocity development on different plane. The position of valve lift for every model in these cases is at maximum valve opening for both intake valves.

3.1.1 Pressure contour

The pressure generated for every model is shown in Figure 6 below by comparing pressure contour on different planes for every intake port designs. The constraint used in this study is the position of piston which is at the end of intake stroke.

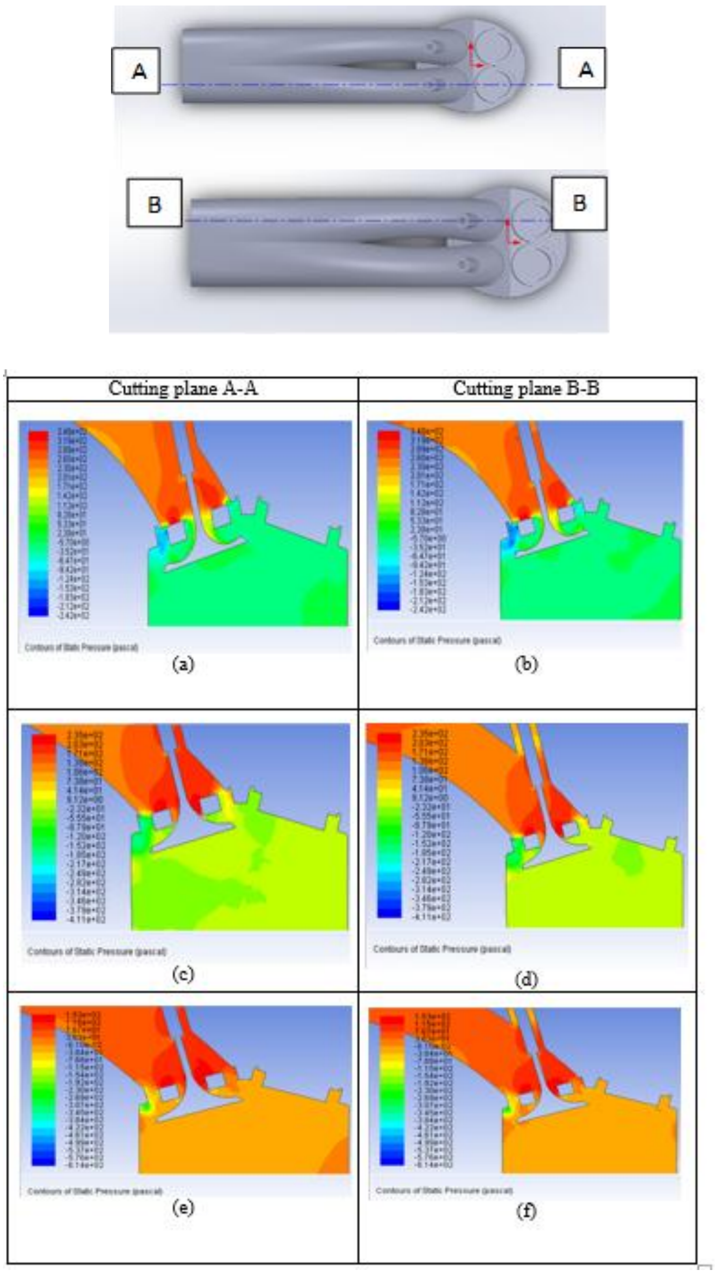

Figure 6. Pressure contours on cut section A-A and B-B at (a) (b) 25 mm intake duct, (c) (d) 28 mm intake duct, & (e) (f) 30 mm intake duct

By comparing the value of maximum pressure localized for every designs, it shows that the smaller the intake port diameter, the higher the maximum pressure.

This is because of smaller intake port diameter has smaller area and thus there will be more distributed force received on that surface area that results in high pressure.

3.1.2 Velocity vectors

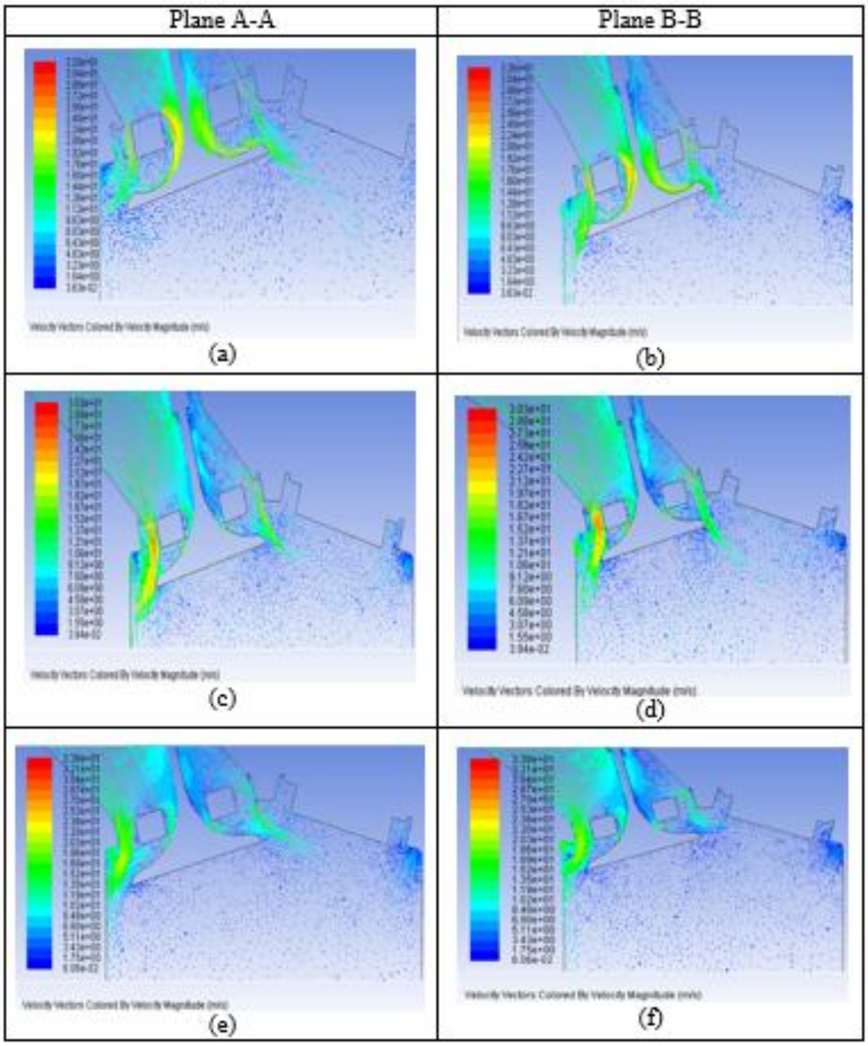

Figure 7. Velocity vector on cut section A-A and B-B at (a) (b) 25 mm intake duct, (c) (d) 28 mm intake duct & (e) (f) 30 mm intake duct

As can be seen in the Figure 7, the distribution of the flow around the valve gap is irregular for all cases, causing a reduction of discharge coefficient. It also detected recirculation zone in the intake duct caused by the sharp edge shape of the duct. This phenomenon causes a pressure drop in the intake system and decreases the discharge coefficient.

3.2 Case Study 2: Variation of intake valve lift

30 mm intake port diameter is analysed with different valve lift constraint. This analysis is to observe the effect of valve lift position on its pressure and velocity generated. One of the intake valves will be at maximum opening while another intake valve will be changing on its valve lift position.

3.2.1 Pressure contours

Figure 8 below shows the static pressure generated in 30 mm of intake port design on the cut plane of A-A and B-B.

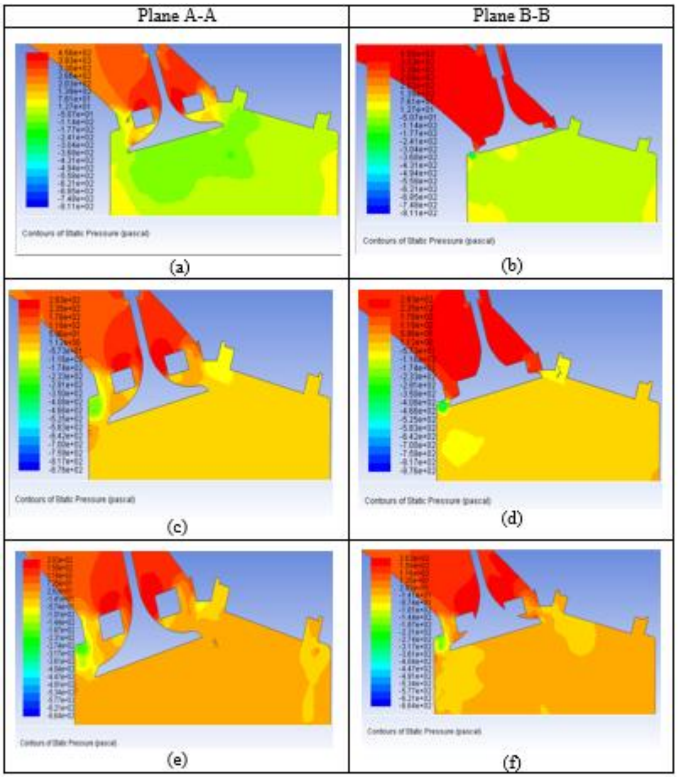

Figure 8. Pressure contours on cut section A-A and B-B at (a) (c) (e) 7.5 mm valve lift (b) 0 mm valve lift (d) 2 mm valve lift & (f) 5mm valve lift

Pressure generated at the intake duct is reduced when increasing the intake valve lift as shows in Figure 8a, 8d and 8f. This happens because of the flow that enters combustion chamber become smooth due to the increase of area that allows the smooth movement of fluid while entering combustion chamber. Thus, there will be lower force that applied on the surface of intake duct and creates lower pressure.

3.2.2 Velocity vectors

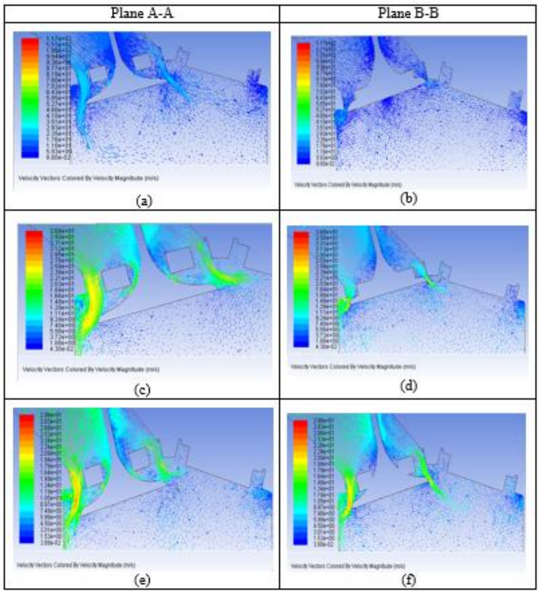

Figure 9. Velocity vectors on cut section A-A and B-B at (a) (c) (e) 7.5 mm valve lift (b) 0 mm valve lift (d) 2 mm valve lift & (f) 5mm valve lift

By observing the results for different valve lift at plane B-B, it shows that velocity is increased when valve position increased. This is due to the higher distribution of the flow that can be passes through the intake valve when valve lift is increased.

3.3 Swirl and tumble ratio

3.3.1. Effect of variation intake port diameter to the swirl ratio

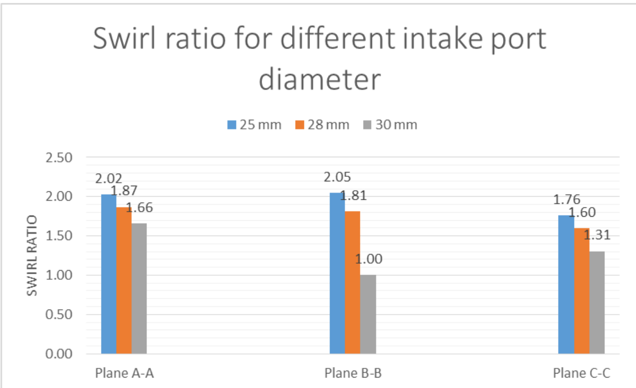

Figure 10. Swirl ratio for variation of intake port diameter at plane A-A, B-B and C-C

Figure 10 shows the swirl ratio computed at the same plane for engine with different intake port diameters. The position of both intake valves are the same which is at maximum lift which is 7.5 mm. Based on the graph above, results of swirl ratio for every plane decrease as the intake port diameter increased. This happens because the cancellation of the high swirl velocities under the valves due to the colliding flows from the two intake valves. Thus, development of tangential velocity is reduced when intake port diameter is bigger and which bring significant effect to the swirl ratio.

3.3.2. Effect of variation intake port diameter to the tumble ratio

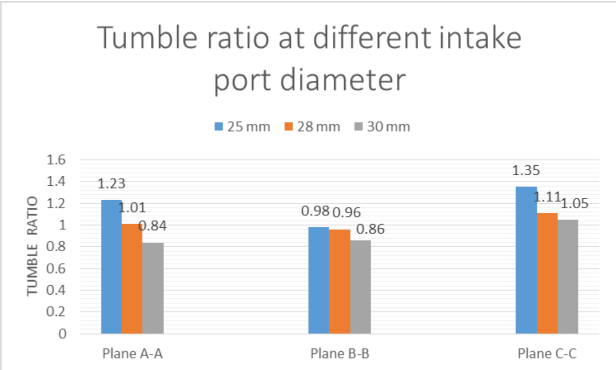

Figure 11: Tumble ratio for variation of intake port diameter at plane A-A, B-B and C-C

Figure 11 shows the tumble ratio calculated inside the combustion chamber at different intake port designs. All of the graphs at plane A-A, B-B and C-C shows the same pattern where its tumble ratio decreased as the intake port diameter increased. Tumble ratio is calculated based on angular velocity of tumble and it value is decreased when the intake port diameter increased. This is due to more fluid flow in intake duct that will cause the fluid to collide each other, thus reduce the tumble velocity.

3.3.3. Effect of variation intake valve lift to the swirl ratio

Figure 12 illustrates the swirl ratio at different intake valve lift in the same intake port design which is 30 mm of its diameter. One of the intake valve positions is keep to be constant at 7.5 mm while other intake valve is changed as represent in the graph above. The swirl ratio increases at all plane when the intake valve position is increased. Based on the results, by increasing the intake valve position from 0 mm to its maximum lift, the swirl ratio also increased because more fluid can enter the combustion chamber as the area of intake valve become wider which increase the tangential velocity. The percentages of swirl ratio increase at plane A-A, B-B and C-C is 45%, 69%, 37%. From this, it can be deduce that the highest percentage of swirl is located at plane B-B. The highest percentage of swirl ratio located in this case at plane B-B.

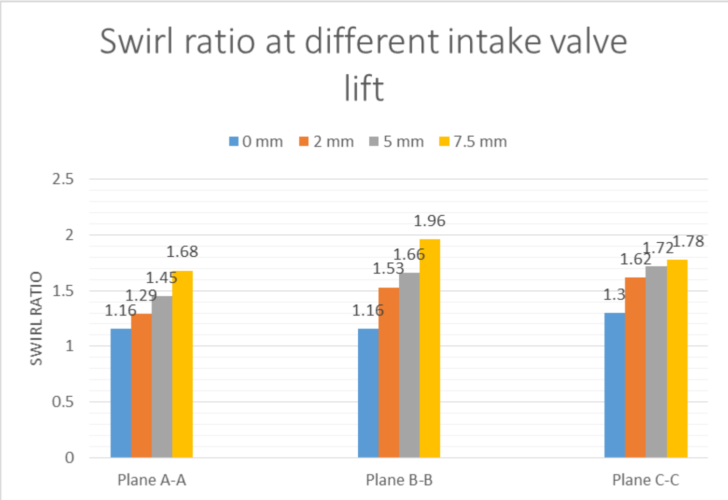

Figure 12: Swirl ratio for variation of intake valve lift at plane A-A, B-B and C-C

3.3.4. Effect of variation intake valve lift to the tumble ratio

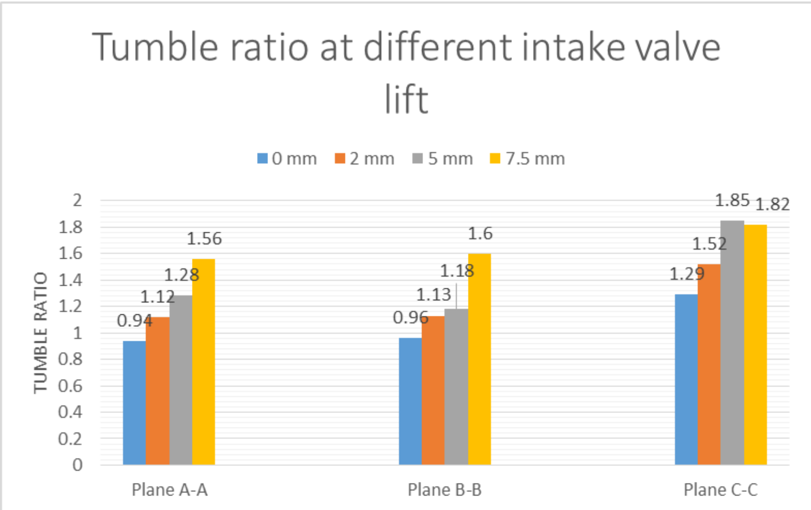

Figure 13: Tumble ratio for variation of intake valve lift at plane A-A, B-B and C-C

From the graph in Figure 13, the tumble ratio increases when the intake valve increases due to its area of intake valve become wider which allow more fluid entering combustion chamber that increased its tumble velocity. The percentages of tumble ratio increase at plane A-A, B-B and C-C is 66%, 67%, and 41%. The highest percentage of tumble ratio is located at plane B-B.

4. Conclusion

Case study 1: Variation of intake port diameter

The variation of intake port diameter and intake valve lift does affect the development of swirl and tumble inside internal combustion engine. The bigger the intake port diameter designs, the smaller the swirl and tumble ratio. The design of intake port diameter should not be too small because it bring effect on the high pressure development in intake duct. Thus, 30 mm of intake port diameter is suitable to be selected as best intake port diameter due to lowest pressure and acceptable swirl and tumble generation.

Case study 2: Variation of intake valve lift

Maximum intake valve lift gives the highest swirl and tumble ratio which is at 5 mm lift. Swirl and tumble development should not to be too high because it can affect the volumetric efficiencies of engine.

Acknowledgments

The authors acknowledge the financial support from Universiti Teknologi Malaysia (UTM) under the research university grant Q.J130000.2509.08H93.

References

Basha, S.A. and Gopal, K.R. (2009). In-Cylinder Fluid Flow, Turbulence and Spray Models - A Review. *Renewable and Sustainable Energy Reviews*. 1620-1627.

Federico, B., Stefania, F. and Giulio, C. (2014). Tumble motion generation in small gasoline engines: a new methodological approach for the analysis of the influence of the intake duct geometrical parameters. *Energy Procedia. ATI 2013 - 68th Conference of the Italian Thermal Machines Engineering Association.* 997-1006.

Jasmi, M. N. A. (2012). *Effect of Intake Port Design on Swirl Behavior of IC Engine.* Malaysia: Universiti Teknologi Malaysia.

Nagarajan, G. and Kumar, C. R. (2012). Investigation of Flow during Intake Stroke of a Single Cylinder Internal Combustion Engine. *ARPN Journal of Engineering and Applied Sciences*. 180-186

Pulkrabek, W. W. (2004). *Engineering Fundamentals of the Internal Combustion Engine* (*2nd edition*). Pearson Prentice Hall.

Selvaraj, B. (2011). Effects of Intake Port Geometry on the Performance of an SI Engine. SAE 2011-32-0356

Shuisheng, J., Siwei, .Z., Hua, W. and Shuping, H. (2012). Parameter Analysis of Diesel Helical Intake Port Numerical Design. *Energy Procedia. International Conference on Future Energy, Environment, and Materials.* 558-563.