To investigate the variation in Swirl Ratio by changing the Valve Lift of an Internal Combustion Engine

Devang S. Nath¹ and Naveen Kumar¹
¹Delhi Technological University, India
Email Id: devangsath98@gmail.com
Contact Number: 8130279951

Abstract. In today’s scenario, humans are facing the issue of the global pollution crisis and the authorities worldwide have implemented stringent norms. Poor air quality has a hazardous impact on the lives of the people. The major emissions from exhaust pipe contain Particulate Matter (PM), Nitrogen Oxides (NOx), Carbon Monoxide (CO) and greenhouse gases which exploit the environment due to which the air quality index is degrading. Furthermore, these exhaust gases can trigger acute diseases in humans like headaches, nausea, fatigue, etc. A huge amount of research and development is being carried out on reducing the harmful emissions by changing the properties of fuels, and geometry or mechanical changes in the engine itself can further hold on the emissions well within the range. One of the methods is to improve the swirl formed in the combustion chamber. Swirl motion is the intentional spinning of the air that promotes even mixing of fuel and air when fuel is introduced to the cylinder during intake. Better swirl formed helps in efficient combustion which subsequently leads to lesser emissions. This research paper studies the change in swirl ratio by varying the valve lift of an internal combustion engine. The observations were made through simulations done in ANSYS™ 17.0 FLUENT using Cold Flow Simulation in IC ENGINE module. Meaningful curves of swirl ratio obtained for different valve lifts are then compared.

1. Introduction

The development of Internal Combustion engines is one of the most revolutionary moves in the history of mechanical engineering. They have a wide range of applications in numerous industries and fields including automotive, marine and agricultural usage, etc. Although IC engines provide us with the locomotive needs in the automotive industry, it somewhat has an adverse effect on the environment. The harmful emissions from exhaust are one of the major causes of degrading air quality. Thus, it is an essential task for design engineers to have exhaust emissions in compliance with norms set by government globally.

For achieving efficient combustion, the air-fuel mixture should be optimum. Even mixing of the charge and air is dependent upon the swirl formed in the cylinder during the intake stroke. Swirl is the intentional helical motion of air passage from inlet port during intake stroke [1]. Mixture consistency is profoundly dependent upon swirl developed. Thus, better the swirl formed, better is the mixing which subsequently leads to better combustion. Previous researches has shown that swirl influence the heat transfer, combustion quality and emission in addition to affecting the mixing of air-fuel and combustion process[2]. Swirl generation is dependent upon many factors including the inlet port geometry, piston bowl shape, etc [3]. Furthermore, swirl is also dependent on the valve lift. The nature
of the swirling flow in an actual engine is extremely difficult to determine. Thus, Swirl Ratio (Rs) is used to quantify the swirl. Swirl ratio is defined as the solid-body rotating flow, which has equal angular momentum to the actual flow, divided by crankshaft angular speed [1]. Swirl speed or velocity is the angular speed of the charge about the cylinder axis (rad/sec).

2. Methodology

2.1. Cold Flow analysis

For engine CFD purposes, ANSYSTM IC Engine module provides us with Cold Flow analysis [4]. The primary feature is, only air motion is taken into consideration without taking charge into account [5]. This helps in visualization of in cylinder fluid flow characteristics when the inlet and outlet valves are lifted and closed. Results acquired helps in understanding the behaviour of flows and further helps in optimizing the certain characteristics of the engine itself for better performance.

3. Geometry

The 3D pentroof engine was taken from ANSYSTMtutorial model which consists of one inlet and outlet port, two poppet valves and piston bowl. Model was then imported to ANSYSTM Workbench. The computational model of engine while in simulation for analysis includes intake valve, exhaust valve, cylinder and piston bowl.

![Figure 1. ANSYSTM Tutorial Model](image)

![Figure 2. Engine Performance](image)
4. Decomposition and Meshing

For precise visualizations of engine process, the engine model was decomposed into different parts[5]. This helps in better control on the mesh of model. Different decomposed parts are indicated by different colours. Then the meshing of the decomposed model was done using ANSYS™ Meshing tool. The mesh was coarser at the inlet & exhaust port but is finer at the region of valve seat. A total of 378848 nodes and 1197904 elements were there after the meshing was done. Dynamic meshing is used since the combustion chamber expands and compresses as the crank angle changes.

![Meshed geometry](image1)

**Figure 3.** Meshed geometry

![Dynamic geometry](image2)

**Figure 4.** Dynamic geometry

5. Simulation

The simulation was carried out in ANSYS™ Fluent® of IC Engine module. In solver settings of the IC Engine, the number of crank angles to run was set to 720 degrees for simulating full cycle. Temperature was set to 300 K. Standard k-ε turbulence model was used to simulate the turbulence in the engine cylinder. For 1 time step, the crank angle turned was 0.2432 degrees. Thus for simulating 720 degrees, the time step was set to 2960. The simulation was done for three different values of minimum valve lift.

| Table 1. Specifications of engine model used |
|---------------------------------------------|
| Connecting Rod length | 144.3 mm |


### Crank radius

| Parameter      | Value |
|----------------|-------|
| Crank radius   | 45 mm |

| Piston offset | 0 |

| Engine speed  | 2000 RPM |

| Minimum lift  |       |
|---------------|-------|
| (i)           | 0.2 mm |
| (ii)          | 0.4 mm |
| (iii)         | 0.6 mm |

## 6. Result and Discussion

The contours of velocity magnitude is shown in Figure 5. for the lift of 0.2 mm. For the valve lift of 0.4 mm, the contours of velocity magnitude is shown in Figure 6. Similarly for 0.6 mm valve lift, it is shown in Figure 7.

![Velocity magnitude contour for 0.2 mm valve lift](image1.png)

**Figure 5.** Velocity magnitude contour for 0.2 mm valve lift

![Velocity magnitude contour for 0.4 mm valve lift](image2.png)

![Velocity magnitude contour for 0.6 mm valve lift](image3.png)
In above figures, the contours of velocity magnitude can be seen at a crank angle of 460 degrees that is, during the intake stroke for different values of valve lift. The highest velocity magnitude is for the 0.2 mm lift that is 45.9 m/s from the Figure 5. For 0.4 mm, the velocity magnitude was 24.8 m/s from the Figure 6. Similarly, for 0.6 mm, it is 36.2 m/s from the Figure 7. Furthermore, the curves for swirl ratio v/s crank angle were obtained for different values of valve lift. For 0.2 mm lift, the curve shown in Figure 8, for 0.4 mm lift, the curve is shown in Figure 9. And for 0.6 mm lift, the curve is shown in
Figure 9. Swirl ratio for 0.4 mm lift

Figure 10. Swirl ratio for 0.6 mm lift

For the crank angle of 460 degrees, the swirl ratio can be seen from the graphs above. For 0.2 mm lift, the Swirl ratio was 0.8. Then for 0.4 mm lift, the Swirl ratio was 0.25 and lastly for 0.6 mm lift, the Swirl ratio was 0.78.

7. Conclusions

The highest value of swirl ratio was obtained in the case of 0.2 mm lift. Secondly in the case of 0.6 mm lift, the swirl ratio is almost comparable to that of 0.2 mm lift. Lastly minimum swirl ratio was obtained for 0.4 mm lift. This implies that the swirl ratio is dependent upon the velocity magnitude from the inlet port. Since it was highest in the case of 0.2 mm lift, the swirl ratio was highest among all. Secondly, for 0.4 mm lift, it was least since it’s velocity magnitude was least. And finally 0.6 mm lift was the second best.

Figure 11. Swirl ratio comparison

8. References

[1]. Heywood, J.B. (1988). Internal Combustion Engine Fundamentals. McGraw Hill International Editions.
[2] M. Farid and M. Said, “Cold flow analysis on internal combustion engine with different piston bowl configurations,” no. April 2017, 2018.

[3] https://www.researchgate.net/post/Is_swirl_ratio_only_dependent_on_design_of_the_engine_shape

[4] Zunaid, M., Upadhyay, L., and Kushwaha, S., “Cold Flow Simulation for an IC Engine with Different Lengths of Connecting Rod,” 14(1):50–54, 2017, doi:10.9790/1684-1401075054.

[5] ANSYS user’s guide, ANSYS, Inc. Southpointe, Canonsburg, PA 15317, 2012.