DESIGN AND ANALYSIS OF INTAKE VALVE OF SI ENGINE

BULASARA SIVA KUMAR¹ & M. BALAJI²

¹Scholar, Department of Mechanical Engineering, V. R. Siddhartha Engineering College, Vijayawada,
Andhra Pradesh, India

²Associate Professor, Department of Mechanical Engineering, V. R. Siddhartha Engineering College, Vijayawada,
Andhra Pradesh, India

ABSTRACT

This research aims to explore the possibilities of improving the SI engine's swirl characteristics by means of different change in the geometry of the intake valve. This work aims at improving air-fuel mixing through the replacement of traditional valve with swirling vane surfaces built into the intake valve and a comparison of the proposed geometry. We designed four different types of modified valves, each with different vane angles, such as Model-1 inlet angle 75° and outlet angle 35°, Model-2 inlet angle 75° and outlet angle 45°, Model-3 inlet angle 65° and outlet angle 30°, Model-4 inlet angle 65° and outlet angle 40°, respectively. The geometry of the intake valves is modelled using CATIA V5 and imported into ANSYS CFD FLUENT code for the simulation of flows. Simulation of the flow is carried by assuming the location of three valve lifts 8mm, 10mm and 12mm respectively. The results obtained are strong, and show that the changed intake valve has great influence on the characteristics of the swirl.

KEYWORDS: Flow Simulation, Intake Valve, CFD, SI Engine, Valve Lift, Catia v5, ANSYS & Design

Received: Jun 08, 2020; **Accepted:** Jun 28, 2020; **Published:** Sep 17, 2020; **Paper Id.:** IJMPERDJUN20201298

INTRODUCTION

Although engines have improved rapidly over the past few decades, efficiency in the sense of fuel production and exhaust emissions still lags behind. Inefficient use of air in the engine results in poor combustion due to improper atomization of the air-fuel mixture, which affects the performance characteristics of the engine in terms of fuel economy and emissions under part load conditions [1].

With respect to the axis of the cylinder, the vortex is the coordinated rotational movement of the mass of fluid. Swirl significantly improves the mix of fresh air and fuel to quickly produce a homogeneous mixture. It is also a major cause of the high flame front dispersion during the combustion process. The intake valve and therefore the combustion chamber can produce eddies in the compression process, in the intake stroke inside the engine. The vortex force increases the tangential velocity component of the air inside the engine cylinder, which tends to mix with the fuel and air and thus affects the combustion and emission characteristics of the engine. The deterioration of the environmental situation has raised the concern of the manifold due to the emissions of various energy conversion devices. This concern has forced scientists and engineers to focus their attention on creating such processes / devices in research, this can minimize fossil fuel consumption and harmful environmental pollution. Compared to the spark-ignition (SI) engine, a reduction in fuel consumption and exhaust emissions can be achieved to some extent by creating a higher degree of vortex which increases turbulence within the engine cylinder. The high intensity turbulence performance is one of the most important factors to stabilize the ignition process, the rapid spread of the ignition, especially in the case of lean combustion. In general, two types of eddies are used to efficiently produce and maintain

turbulent flows. These eddies are commonly called swirling and tumble flows, which are synchronized rotations in the engine cylinder. They play a role in improving engine performance and exhaust emissions.

PROBLEM DEFINITION

Many researchers worked in this field, both experimentally and computationally, to explore the phenomenon of the internal combustion engine's cylinder flow, but there has never been any attempt to improve the fuel economy and reduce emissions in the SI Engine by using different intake valve geometry configurations. This project work aims to tackle this issue and aims to provide a solution to boost fuel efficiency and minimize emissions by regulating swirl movement. This work provides a comparative analysis of both valves to determine efficacy in terms of air movement inducing swirl movement. The analysis is carried out with variable valve lift for both valves with a phase size of 2 mm each.

RESEARCH OBJECTIVES

- To research the current configuration and three-dimensional modeling of the inlet valve.
- Modify valve configuration using CATIA V5 software with directed vane structure and three-dimensional modeling.
- Using CFD simulation, to investigate the airflow motion within the turbine.
- For both valve designs to perform Swirl Ratio analysis, velocity, and turbulent kinetic energy.

SPECIFICATION OF THE SI ENGINE

The engine considered for the calculation review is a naturally aspirated four-stroke single-cylinder carbureted SI engine with a cylindrical combustion chamber and single intake port. Engine requirements are specified in the below table 1.

Table 1: Engine Specifications

Bore x Stroke	96mm x 99mm
Intake valve diameter	42mm
Max Intake valve lift	12mm

LITERATURE SURVEY

[1] A. Raj Kumar et al. "Authors have been researching devices that create swirls in a single-cylinder diesel engine. The present study consists of swirl-type modeling guidance vane generating devices in the intake manifold. The swirl generating devices are three, four five, and six versions with a varying number of vanes. The vane angle is maintained from 0° to 40°. The vane configuration should be such that it can disperse the maximum amount of air without impacting the engine's volumetric performance. CFD analyses are carried out using standard K-Epsilon Model with an inlet speed of 35 m/s and 100Kpa pressure. The unit of the six vane type produces optimum swirl as opposed to other versions". [3] M D Raj Kamal et al. "In this study, they adjust an intake valve and measure varying vortex and ignition points. To model the airflow in the cylinder, the design factor used Ansys cfx. Turbulence velocity and volume flow were determined from the data. Several model prototypes were built before an acceptable design was completed. The valve guides have been placed inside the intake manifold. The movement of air inside the cylinder is provided by the inlet valve. The movement of air plays a very important role in the air-fuel mixture, combustion and emission formation. The swirling movement of air generally occurs due to the design of the intake port. A good intake design can create a higher vortex and help burn fuel". [4] Dhanush Dineshan et al. "Work on the CI engine were investigated using three modified inlet valves, namely flat vane — curved vane 1 and curved vane 2 — and 3-dimensional numerical analysis of in-cylinder flow conditions was carried out using CFD (Computational

Fluid Dynamics) technique. Each concept is tested at different parts of the cylinder head for the swirl characteristics, followed by experimental validation of the analysis using a steady-state airflow bench test to analyze the swirl generation. In changed intake valve geometries the swirl parameters were evaluated and compared with the reference engine valve”.

METHODOLOGY

Figure 1: Imported Design Model.

With the support of CATIA V5 the intake valve and the cylinder engine are modeled. The first move is to use a traditional intake valve to model the current geometry. The second step is to model the new modified intake valve for computational investigation for the study of four different types of inlet valves by making a simple modification of the Poppet intake valve geometry. In the ANSYS CFD FLUENT the traditional valve geometry and changed valve geometry is then imported.

BOUNDARY CONDITIONS

- The initial boundary conditions are established considering the inlet as a velocity 35m/s and the outlet as a pressure outlet.
- And the base of the wall is selected as outlet.
- Wall state, wall temperature is an adiabatic, non-slip, and non-filtration boundary and wall temperature is 300K.

RESULT AND ANALYSIS

Figure 2: Velocity Vector(m/s) Base Model at 8mm Valve Lift

Figure 3: Velocity(m/s) Base Model at 8mm Valve Lift

Figure 4: Velocity Vector(m/s) Model-1 at 12mm valve Lift .

Figure 5: Velocity(m/s) Model-1 at 12mm Valve Lift.

Seen in figure 2, that the two incoming jets on either side of the valve produce two powerful vortices, one on the left side of the valve and one below the valve plate while the liner wall is restricted to the flow. The movement of these two vortices interferes with one another under the valve plate causing stagnation of the flow velocity which is also evident in Figure 3. It is also evident that the flow velocity on the right side of the wall is higher, allowing the jet on the right side of the valve to control more than the left side. From Figure 4, displays the flow field and velocity magnitude of the modified inlet valve 10 mm valve raise, it is obvious that the same flow structure as shown in Figure 5, still exists with the two powerful vortices close to the valve. The distinction here is the presence of a horizontally directed flow field on the cylinder's right side. This flow is induced by a kind of swirling motion created by the jet perpendicular to the two jets. When this perpendicular jet reaches the cylinder liner it produces a rotational motion of the flow around the liner which later appears on the left side of the cylinder as this horizontally guided flow. From Figure 5, it also demonstrates how the two opposite flow fields of the cylinder's left and right side converge in the centre and are guided upwards towards the valve that creates these two vortices. But more importantly, it prevents the formation of a strong movement for mixing and turbulence intensity purposes which is critical in S.I engines.

Table 2 : Velocity Magnitude (m/s) Values at Different Valve Lift Position

Model	8mm	10mm	12mm
Base Model	163.9	111.54	85.4
Model-1	212	153.3	117.2
Model-2	151	105.9	89.36
Model-3	164.23	109.2	87
Model-4	173.6	106.13	84.64

Table 3: Turbulence Kinetic Energy (m^2/s^2) Values at Different Valve Lift Position

Model	8mm	10mm	12mm
Base Model	118.7	96.12	68.23
Model-1	586.3	327.8	304.5
Model-2	145.46	88.98	58.84
Model-3	154.11	89.2	59
Model-4	104	82.22	60.07

The above tables, display the Velocity (m/s) and Turbulence Kinetic Energy (m^2/s^2) of both models versus 2 mm, 10 mm and 12 mm valve lifts respectively. Due to the narrow flow passage between inlet manifold and inlet valve, speed is increased at 8 mm valve lift. It will act like an air jet and at higher speed blast out the air. When the valve is fully open, the velocity will begin to decrease. Modified Model-1 inlet valve displays the higher velocity than Base Model. Also it is found that the configuration of the inlet valve influences the fluid turbulence within the cylinder. The result shows that higher turbulent kinetic energy was created for Model-1 because of the high swirl ratio. Also it is found that the configuration of the inlet valve influences the fluid turbulence within the cylinder. The result shows that higher turbulent kinetic energy was created for Model-1 because of the high swirl ratio.

CONCLUSIONS

- The flow of fuel within the engine would be influenced in large part due to Airflow motion by the use of the swirl driven vanes.
- Which will contribute to improved swirling motion and turbulence, which ensures better mixing of air and fuel.
- The effect of air flow velocity and turbulence in the intake manifold and inlet valve will ensure the engine will perform well.
- A better mixture of air and fuel also decreases exhaust emissions.

REFERENCES

1. Raj Kumar, G. Janardhana Raju and K. Hemachandra Reddy, (2016) Comparison of swirl turbulence generating devices in compression ignition engine. Archives of applied science research, 8 (7): 31-40.
2. Athanasia Kalpakli Vester, Yu Nishio & P. Henrik Alfredsson "Unravelling tumble and swirl in a unique water analogue engine model" 23 January 2018.
3. Madhusudan Barot, Abhishek Shah & Mit Patel "CFD Analysis of Single Cylinder Four Stroke Gas-Fueled Engine for Prediction of Air Flow Rate during Suction Stroke" IJEDR | Volume 5, 2017.
4. M D Raj Kamal, S. Kaliappan, S. Socrates & G. Jagadeesh Babu "CFD Analysis of Single Cylinder IC Engine Inlet Swirl Valve" International Journal of Latest Engineering Research and Applications (IJLERA), Volume – 02, Issue – 08, August – 2017.
5. Dhanush Dineshan, Akhif Rasheed, Mihir Kulkarni, T. Venugopal, "Investigation on Swirl Motion of CI Engines through Intake Valve configurations" International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 12, December

2017.

6. Ms. Mamta .R. Zade "Optimization Of Inlet Valve Of IC Engine Using FEA" International Research Journal of Engineering and Technology (IRJET), 10, Oct-2016.