A Comprehensive Study On In-Cylinder Ic Engine Due To Swirl Flow

Priscilla *(P.G. Students, Thermal Power Engineering, Department of Mechanical Engineering. The Oxford College of Engineering, Bommanahalli, Bangalore, India)*
Meena. P *(Assistant Professor, Department of Mechanical Engineering, The Oxford College of Engineering, Bommanahalli, Bangalore, India)*

Abstract:

Swirl is an important flow pattern of air motion commonly found in diesel engines; it not only affects the fuel-air mixing and combustion processes, but also has a significant impact on heat transfer, combustion quality, and engine-raw emissions. Swirl is used to describe circulation about the cylinder axis. This study deals with the investigations of the in-cylinder swirl flows in an internal combustion engine before combustion and for the engine with a bowl-in-piston. The bowl shape plays a significant role near TDC and in the early stage of the expansion stroke. In order to achieve the swirl intensities in the cylinder and based on steady state flow, the following engine operating parameters have been considered: velocity, pressure, temperature and turbulence intensity .The flow characteristics of these engine manifolds are examined using Computational Fluid Dynamics (CFD) and turbulence intensity is modelled using SST Model . The purpose of this work is to optimize swirl in the intake systems of diesel engines.

Keywords: Diesel engine, In-cylinder motion, Swirl, Bowl piston, CFD, SST Model.

1. INTRODUCTION

The area with the most potential for technological development concepts is based on the internal-combustion engine. Combustion implicates harmful effect to the environment because of the emissions produced. In order to obtain optimal combustion state in engine developing process, engine designers control the flow characteristics, modifying shape of intake port, valve and combustion chamber shape.

The fluid motion in an internal combustion engine is induced during the induction process. As the intake charge enters the combustion chamber through the intake manifold then the kinetic energy of the fluid resulting in turbulence causes rapid mixing of fuel and air, if the fuel is injected directly into the cylinder. In-cylinder fluid motion controls the fuel-air mixing and premixed burning in diesel engines. Therefore, it is very much essential to understand the in-cylinder fluid motion [9].

There are two types of structural turbulence that are recognisable in an engine; tumbling and swirl. Both are created during the intake stroke [7].

Swirl flow has been used in many different kinds of internal combustion engine because of their effects in increasing efficiency, reducing of noise and other emission pollutants, and improving combustion instability. Swirl is defined as the charge that rotates concentrically about the axis of the cylinder. If the inlet flow is brought into the cylinder with an initial angular momentum, it will create swirl. There are two ways to create swirl, by valve design and by intake design .The design needs a shape that creates turbulence during the intake stroke, and still lasts during the compression stroke.

This study is about creating a suitable intake design producing swirl by developing a helical intake port which would improve the engine efficiency. The idea of a helical port is that the air is brought in rotation prior to entering the cylinder, namely in the inlet port. The rotation is achieved by forcing the air flow around the valve stem, so that there is angular momentum created about the cylinder axis when the charge enters the cylinder [1]. The intake design was developed using CFD techniques, which involve the use of three main software packages; CATIA, Hypermesh and Fluent.

2. GEOMETRY DESIGN

The engine model studied is a typical single-cylinder diesel engine with intake valves and helical port equipped with bowl in a piston shapes. Using CATIA V5R20 the entire 3D-design of intake manifold had been carried out [5].

Table: 2.1. ENGINE SPECIFICATION

BORE	130.0mm
STROKE	150.0mm
CONNECTING ROD LENGHT	275.0mm
DISPLACEMENT	1991cm³

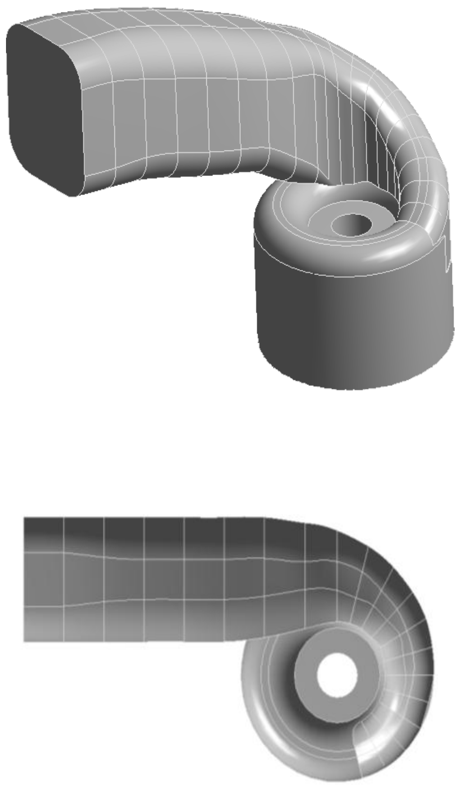

Fig 2.1: Geometric model of helical port

3. MESH GENERATION

The meshes were created in HyperMesh. For the mesh generation special care has been taken to the zones close to the walls. At the valve and valve seat the edges have to be resolved accurately so that the mesh sizes are fine enough to suit the flow characteristics of that area. First, the surface is meshed with triangular element. Finally the remaining region in the domain is filled with tetrahedral cells. Figure below shows conventional and helical port. The mesh has 0.3 millions elements. The quality of the mesh has been tested and the results show that the quality of the mesh is very good.

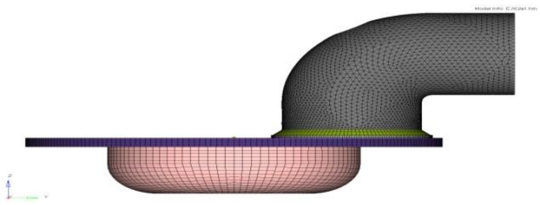

Figure 3.1: Hybrid meshing of conventional port

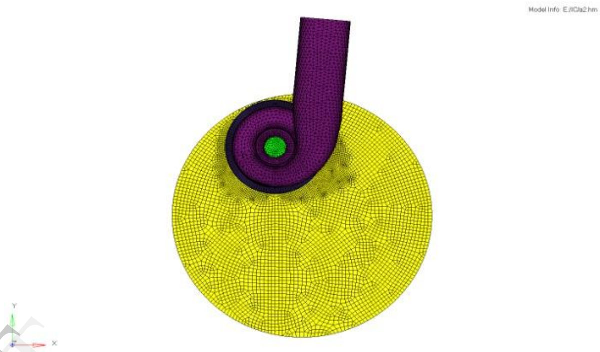

Fig. 3.2: Hybrid meshing of helical port

4. MODELLING: FLUENT

To carryout IC engine dynamic analysis meshed model of manifold with combustion chamber is imported into ANSYS Fluent 14.0. The CFD simulation is carried out for only cold flow without combustion. Inlet condition is given at atmospheric temperature and pressure. During piston motion the layers above the piston will increase when the piston move from top dead center to bottom dead center and the layers will decrease when piston move from bottom dead center to top dead center. This will be helpful to identify the motion of airflow inside the cylinder and this study is completely done under shear stress transport model. The modelled engine is made to run at 1000 rpm for conventional and helical port with a bowl in a piston. The simulation is conducted under transient condition. From this simulation, we can find the variation in parameters like pressure, temperature, velocity and turbulent kinetic energy. We also defined fluid medium, material and boundary condition. The engine parameters are mention in the dynamic mesh settings and valve lift program are uploaded for valve movement and default piston motion is selected. The pressure-velocity coupling is achieved using PISO (Pressure Implicit Splitting of Operators) Algorithm.

5. RESULTS AND DISCUSSIONS

5.1 Variation of Velocity

Velocity will increase at starting crank angle degrees because of small flow passage between inlet manifold and inlet valve. It will act as an air jet and send the air at higher velocity. Velocity will start to decrease when the valve is at fully open position. It shows that flow is maximum at fully opened valve position. Helical port with a bowl in a piston is having higher velocity than conventional port with a bowl in a piston. Figure 5.1 and figure 5.2 shows velocity vector of conventional and helical port.

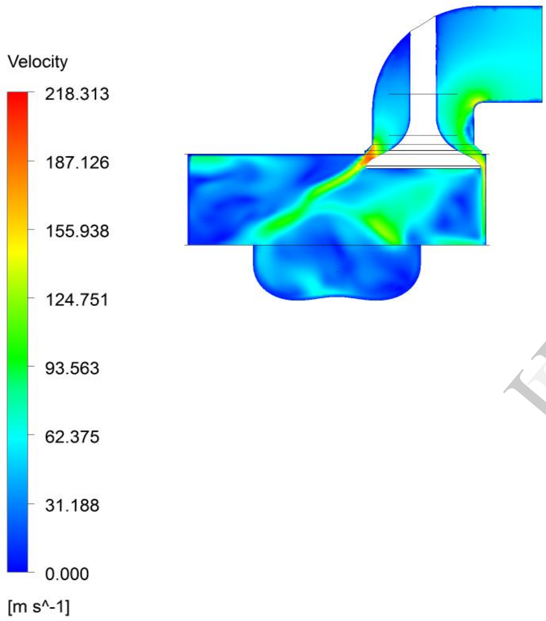

Fig. 5.1: Conventional port velocity contours at crank angle 40^0

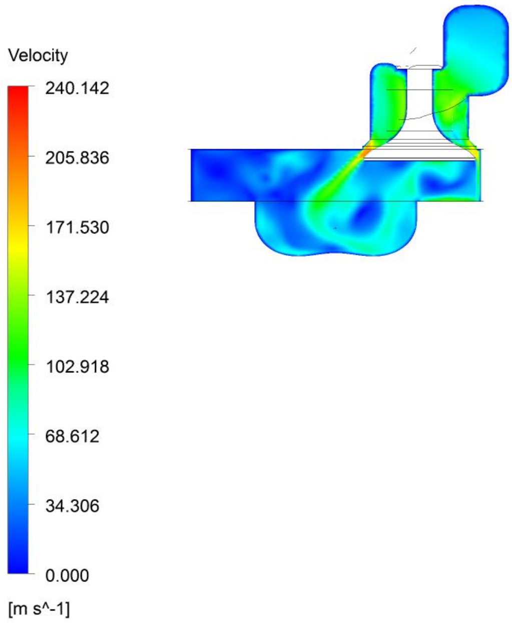

Fig. 5.2: Helical port velocity contours at crank angle 40^0

5.2 Variation of Pressure and Temperature

The graphs for pressure distribution, temperature distribution at various crank angle as shown in figure at intake and compression stroke are plotted against the time step for various cases. Note that each increment of a time step is equals to an increment of 0.25° of crank angle. Piston starts from TDC about 0 degrees and the maximum pressure reaches at 360 degree. At the start of combustion after the ignition delay there is a sudden change of slope of the p-θ curve. The pressure rises rapidly for a few crank angle degrees, and then moves slowly towards a peak value. The Maximum pressure and end of compression stroke is 55 bar and temperature is 951 K.

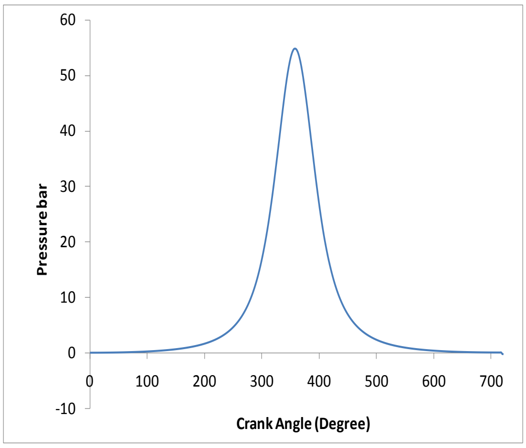

Fig.5.3: Pressure Vs crank angle

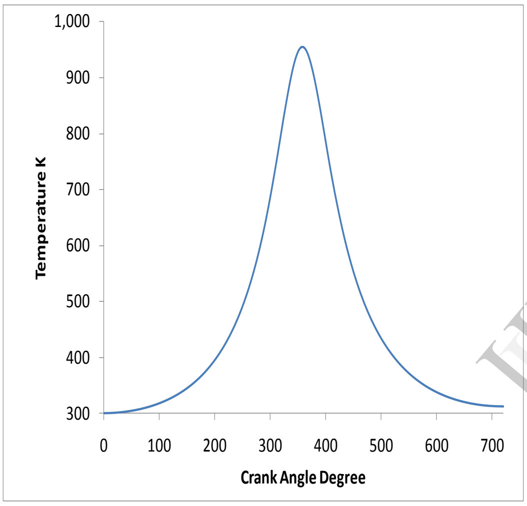

Fig.5.4: Temperature Vs crank angle

5.3 Variation of turbulent kinetic energy

Figure 5.5 shows the variation of Turbulent Kinetic Energy (TKE) with crank angle at 1000 rpm for conventional and helical port. It reaches the peak value during the maximum valve open condition. It is observed that the inlet manifold configuration affects the turbulence of the fluid inside the cylinder.. The dissipation of KE is on account of increased fluid motion. The variation in Turbulent Kinetic Energy is because of deforming volume of the cylinder. During suction stroke, gradual opening of valve will take place, which will provide very small space between manifold and valve for the air to enter. Due to restriction in the flow passage turbulence in the flow tend to increase and piston also will move from top dead center to bottom dead center which has a slightly greater volume than clearance volume .The result shows that higher turbulent kinetic energy was generated due to high swirl ratio for helical port.

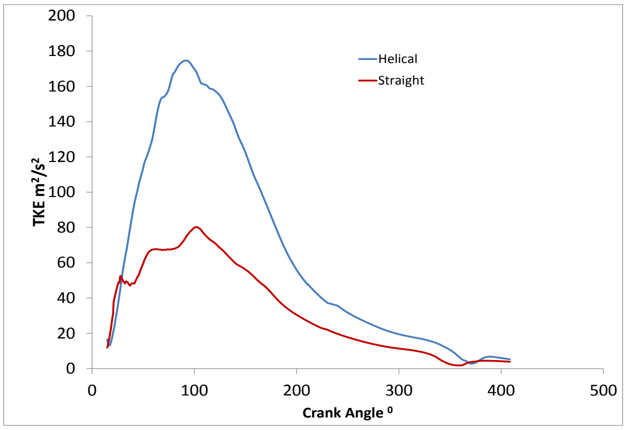

Fig.5.5: Turbulent kinetic energy for different manifold configurations

5.4 Variation of swirl ratio

Swirl ratios are generally defined as the ratio of the angular momentum of the in cylinder flow about each of the three orthogonal axes.

Figure 5.6 shows the variation of Swirl Ratio (SR) inside the cylinder with respect to crank angle for straight and helical manifold configurations at 1000 rpm. During the suction stroke, the swirl ratio increases till the maximum valve lift position and gradually decreases till the end of valve closing and again increases at the end of compression stroke. In the comparison of swirl ratio at 1000 rpm, maximum value is obtained for helical port.

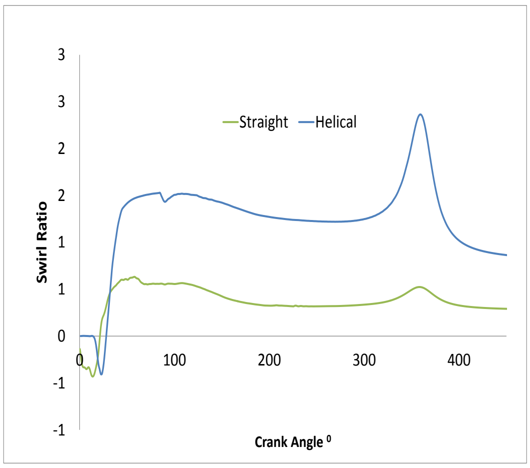

Fig. 5.6: Swirl ratio for different manifold configurations

5.5 Volumetric efficiency

Volumetric efficiency in the internal combustion engine design refers to the efficiency with which the engine can move the charge into and out of the cylinders. More specifically, volumetric efficiency is a ratio (or percentage) of the quantity of air that is trapped by the cylinder during induction over the swept volume of the cylinder under static conditions. VE can be improved in a number of ways; most effectively this can be achieved by compressing the induction charge. From figure the conventional manifold shows lower volumetric efficiency than helical manifold due to the flow restriction.

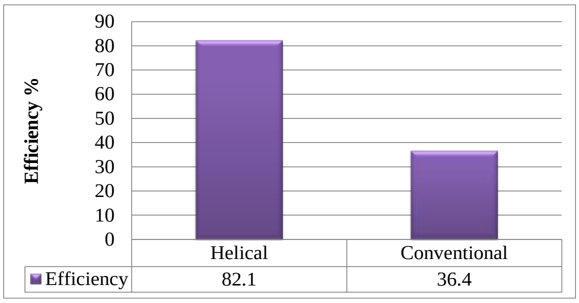

Fig.5.7: Volumetric Efficiency for different manifold configurations

6 CONCLUSIONS

The ability to control the flow in engine components has long been recognized to be the key controlling mechanism to enhance IC engine performance. In particular, control of the flow through the ports and the combustion chamber has become critical for meeting the more stringent emission regulations and fuel economy requirements.

During induction stroke air flows around the inlet valve and gets into the cylinder through the gap between the valve and the valve seat. Air expands into the cylinder and impinges directly on the cylinder wall creating a large angular momentum.

In this present work, the internal flow characteristic in the combustion chamber with two different inlet port with a bowl in a piston are studied using computational fluid dynamics. The overall flow field inside the combustion chamber and various parameters such as pressure, velocity distribution, and turbulence and swirl ratios were examined. The summary of the comparison is as follows:

- Pressure and temperature distribution for helical port is higher than conventional port.
- Helical port creates higher swirl motion than conventional port.
- Helical port creates higher turbulent kinetic energy than conventional port.
- Helical port provides higher volumetric efficiency.

Acknowledgements

The author is thankful to the staff members: Mrs.Meena.P, Mr. Raghunandan. Department of mechanical engineering, The Oxford College of Engineering, Bangalore and Mr.Mallikarjuna, IC engines laboratory, Indian Institute of Technology Madras, Chennai.

REFERENCES

[1] Heywood, J.B. (1988). Internal Combustion Engine Fundamentals. McGraw Hill International Editions.

[2] Flow Characteristics and Influence of Swirl Flow Interactions on Spray for Direct Injection Diesel Engine. Jin-Wook Lee, Kern-Yong Kang, Seung-Hwan Choi, Chung-Hwan Jeon, Young-June Chang.

[3] Benny Paul1, V. Ganesan, Flow field development in a direct injection diesel engine with different manifolds, International Journal of Engineering, Science and Technology Vol. 2, No. 1, 2010.

[4] B. Murali Krishna and Mallikarjuna J. M, "Characteristics of flow through the intake valve of a single cylinder engine using practical image velocimetry", Indian institute of technology madras.

[5] CFD modeling of the in-cylinder flow in direct-injection Diesel engines. F. Payri, J. Benajes, X. Margot, A. Gil.

[6] In-Cylinder Flow Analysis for Production-Type Internal-Combustion Engines. Bas van den Reuvel.

[7] Design of An Inlet Track Of A Small I. C. Engine For Swirl Enhancement. Universidade do Minho, Escola de Engenharia, and Guimarães, Portugal.

[8] Investigation of flow during intake stroke of a single cylinder internal combustion engine. C. Ramesh Kumar and G. Nagarajan CEAR, School of Mechanical and Building Sciences, VIT University, Vellore, India CEG, Anna University, Chennai, India.

[9] A study on combine effects between swirl and tumble flow of intake port system in cylinder head. Jeong-Eue Yun, Jae-Joon Lee, Engine test team, Tech center, Daewoo Motors Co., 199 Chongchon-dong Pupyong-gu, Inchon, 401-714, South Korea.

[10] Extraction and Visualization of Swirl and Tumble Motion from Engine Simulation Data. Christoph Garth, Robert S. Laramee, Xavier Tricoche, Jürgen Schneider, and Hans Hagen.