CFD modeling of the in-cylinder flow in Direct-injection Diesel engine

H.Sushma and Jagadeesha.K.B

Department of Mechanical Engineering, B.T.L.INSTITUTE OF TECHNOLOGY AND MANAGEMENT, Bangalore-560099, Karnataka, India

Abstract- Internal combustion engines in now a days is the best available reliable source of power for all domestic, large scale industrial and transportation applications. The major issue arises at the efficiency of these engines. Every attempt made to improve these engines tends to attain the maximum efficiency. The performances of the diesel engines are enhanced by proper design of inlet manifold, exhaust manifold, combustion chamber, piston etc. The study is about the effect of piston configurations on in- cylinder flow. Here a single cylinder direct injection diesel engine is used for study. For obtaining swirl intensity helical-spiral combination inlet manifold is used. Increase in swirl intensity results in better mixing of fuel and air. Swirl Velocities in the charge can be substantially increased during compression by suitable design of the piston. In the present work, a study on the effect of different piston configuration on air motion and turbulence inside the cylinder of a Direct Injection (DI) diesel is carried out using Computational Fluid Dynamics (CFD) code Fluent 13. Three dimensional models of the manifolds, pistons and the cylinder is created in CATIA V5 and meshed using the pre-processor Hypermesh 10.0.

Index Terms- CFD, inlet manifold, piston configurations, swirl ratio, tumble ratio, volumetric efficiency.

I. Introduction

As engines have evolved over the years, pistons have evolved with them. They're getting shorter and lighter, and use smaller skirts — the cylindrical "body" of the piston. Newer pistons are often made of aluminum alloys comprised of more silicon than in the past. This improves resistance to heat and reduces thermal expansion.

One of the biggest advancements in piston technology is the use of different piston "tops" or "crowns," the part that enters the combustion chamber and is subjected to combustion. While older piston tops were mostly flat, many now feature bowls on top that have different effects on the combustion process. The piston bowl is primarily used in diesel engines. Diesels don't have an ignition phase, so the piston crown itself may form the combustion chamber. These engines often use pistons with differently shaped crowns, although with direct injection becoming increasingly popular, gasoline engines are starting to use them as well.

The shape of the piston bowl controls the movement of air and fuel as the piston comes up for the compression stroke (before the mix is ignited and the piston is pushed downward.) The air and fuel swirl into a vortex inside the piston bowl before combustion (or compression) takes place, creating a better mixture.

By affecting the air/fuel mixture, you can achieve better and more efficient combustion, which leads to more power. The bowls have a variety of different shapes; some are also designed to optimize fuel economy. With direct injection becoming the hottest new technology for gasoline engines, expect uniquely-bowled pistons to become more and more popular. In high-speed direct-injection Diesel engines, the flow conditions inside the cylinder at the end of the compression stroke, near top dead center (TDC), are critical. for the combustion process

These are determined by the air flowing into the cylinder through the intake valves during the induction process and by its evolution during the compression stroke.

Many researchers had been studied on piston geometry effecting the flow distribution of diesel engine. This chapter reviews the previous published literatures, which lays the foundation and basis for further work in this project. This helps to give a better understanding about the topic and also acts as a guideline for this thesis.

Benajes and Margot *et al.* [6], studied the flow characteristics inside the engine cylinder equipped with different piston configurations were compared. For this, complete calculations of the intake and compression strokes were performed under realistic operating conditions and the ensemble-averaged velocity and turbulence flow fields obtained in each combustion chamber analyzed in detail. The results confirmed that the piston geometry had little influence on the in-cylinder flow during the intake stroke and the first part of the compression stroke. However, the bowl shape plays a significant role near TDC and in the early stage of the expansion stroke by controlling both the ensemble-averaged mean and the turbulence velocity fields.

Aita *et al.*[1] studied the swirl motion in the cylinder during the intake and compression strokes on a real geometry with one intake valve, but presented little validation of their calculations. Chen *et al.* [4] performed calculations of the full intake and compression processes and presented some comparisons with experimental data. Their results showed that calculations significantly under predicted the turbulence velocity. They explained the differences by errors in the experimental data and the limitations of the standard k–ε model. Dillies *et al.* [5] also presented similar calculations of a Diesel engine with one intake valve for one combustion chamber, and in this case results compared reasonably well with the experiments. Celik *et al.* [3] made a review of computations based on large eddy simulation

(LES) and concluded that this method has great potential in this kind of application; however its computational cost is still too high for engine design.

From the review of literature, it can be noted that, design of inlet manifold configuration and piston geometry is very important in an IC engine. Hence, this study looks up on the effect of helical-spiral combined configuration with different piston configurations on the induced mean swirl velocity in the piston bowl at TDC, swirl ratio during suction and compression stroke, turbulent kinetic energy variation and volumetric efficiency at engine speed 1000 rpm.

Objective of the present study is:

- Perform CFD Simulation of the IC engine with inlet valve, intake manifold (Helical-spiral combination) and piston using dynamic mesh approach.
- Study the effect of different piston head configurations on the in-cylinder flow-(only intake and compression stroke).
- Compare effect of different piston head configurations on volumetric efficiency, turbulence, swirl and tumble ratio in the engine.

By using the CFD code, flow field can be predicted by solving the governing equations viz., continuity, momentum and energy. The renormalization group theory (RNG k-ε) turbulent model is used for analyzing the physical phenomena involved in the change of kinetic energy. In this work, three pistons (piston A, piston B and piston C,) and in-cylinder flow field investigations are carried out for an engine speed 3000 rpm.

II. Engine and Computational Details

The base engine is same for all three piston configurations CFD analysis. The detailed specification of the base engine selected for the simulation is given in Table 1. The engine selected is a single cylinder research DI diesel engine with Helical-spiral combination inlet manifold.

Table 1: Geometrical characteristics of the engine

Bore	130.0 mm
Stroke	150.0 mm
Connecting rod length	275.0 mm
Displacement	1991 cm3
Intake valve diameter	44.4 mm
Intake valve angle	60°

III. Methodology

The methodology adopted for the present work is as follows. Flow through the intake manifold is simulated to study the in cylinder flow field during non-reacting conditions, which includes the following steps:

• Solid modeling of the intake manifold and cylinder geometry with valves.

• Mesh generation. Solution of the governing equations with appropriate boundary conditions.

• Comparison of the simulated results with the various piston configurations

The study is expected to explore the potential of using CFD tool for design and optimization of engine piston geometry. The commercial CFD code ANSYS FLUENT 13.0 is used for the analysis of flow. The CFD package includes user interfaces to input problem parameters and to examine the results. The code contains three elements

1. Pre-Processor
2. Solver
3. Post Processor

Preprocessor mainly involves the creation of basic 3D model, grid generation and fixing of the boundary conditions. Modeling and meshing is done in Hypermesh 10.0 and is exported to ANSYS FLUENT 13.0 for completing the mesh.

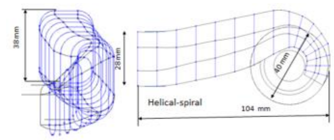
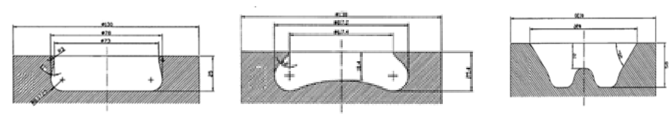


Figure 1 Geometric model of IC Engine manifold



Piston A Piston B Piston C

Figure 2: Geometric model of IC Engine manifold

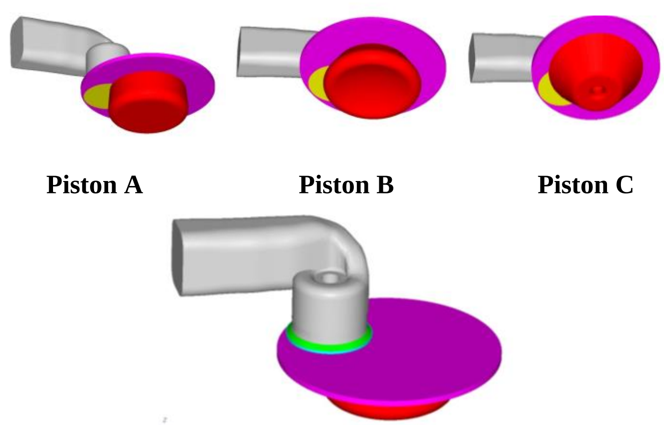


Figure 3: CFD Model of IC Engine Piston at TDC

IV. Geometry Decomposition and Mesh Generations

Two approaches are employed in Fluent 13.0 to solve in-cylinder problems, namely, hybrid approach and layering approach. While the hybrid approach is used for engines with canted valves like most SI engines, the layering approach is typically used for engines with vertical valves like most diesel engines. For either approach mentioned above, in-cylinder problems solved in Fluent consist of three stages. The first stage is to decompose the geometry into different zones and mesh them properly. By breaking up the model into different zones, it is possible to apply different mesh motion strategies to different regions in a single simulation. The second stage is to setup the engine case inside Fluent with the help of a setup journal. The third stage is to perform a transient in-cylinder simulation. The decomposition process is shown in Fig 4. The computational domain includes intake ports and valves, the cylinder and the piston bowl. The number of cells varies from 0.5 million cells in TDC, to 1.6 million cells in bottom dead center (BDC). Since the computational domain is very complex, composed of different zones with different topologies, each zone has been meshed separately (see Fig. 4). This strategy is very useful to obtain a good quality grid and to reduce significantly the meshing time. The connectivity of the various sub-domains is ensured by means of arbitrary interfaces that connect common faces of adjacent zones. Both intake ducts have been meshed following a similar topology, the cells are oriented in the flow direction and they are joined with a cylindrical structured mesh in the zone upstream of the valves. The mesh above the valves has been constructed by revolution of a structured mesh section. During the compression stroke, once the intake valves are closed, the intake ports sub-domains are disconnected from the calculation, so that only the combustion chamber is considered.

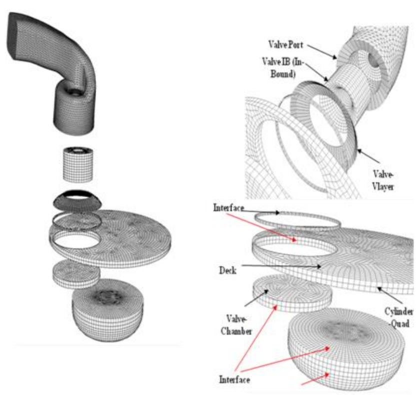


Figure 4: Mesh Boundary and Cell Zone Nomenclature

V. CFD ANALYSIS OF IC ENGINE

5.1 Governing Equations in CFD

There are mainly three equations we solve in computational fluid dynamics problem. They are Continuity equation, Momentum equation (Navier Stokes equation) and Energy equation. The flow of most fluids may be analyzed mathematically by the use of two equations. The first, often referred to as the Continuity Equation, requires that the mass of fluid entering a fixed control volume either leaves that volume or accumulates within it. It is thus a "mass balance" requirement posed in mathematical form, and is a scalar equation. The other governing equation is the Momentum Equation, or Navier-Stokes Equation, and may be thought of as a "momentum balance".
The Navier-Stokes equations are vector equations, meaning that there is a separate equation for each of the coordinate directions (usually three).

5.2 Transient IC engine analysis

To carryout IC engine dynamic analysis meshed model of manifold with combustion chamber is imported into ANSYS Fluent 13.0. The CFD simulation is carried out for only cold flow without combustion. The procedure for solver settings selection of turbulence model and applying boundary conditions to simulate analysis is discussed in this section.

5.3 Boundaries and initial conditions

Constant pressure is used as boundary condition at both the intake and the exhaust manifolds. Attach boundaries are specified on the coincident cell face near the cells above / below the valve. No slip wall boundary condition in conjunction with logarithmic law of wall is used. Walls are considered to be adiabatic.

5.4 Mathematical model

In this thesis RNG k- ε model because, in this model *'k'* is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of (L^2 T^{-2}), e.g. m^2/s^2. 'ε' is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) and has dimensions of *k* per unit time (L^2 T^{-3}), e.g. m^2/s^3.

The turbulent kinetic energy equation as modeled has a number of simplifications from the rigorous equation.

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \tau_{tij} \frac{\partial u_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j}\left[V + v_t/\sigma_k \frac{\partial k}{\partial x_j}\right]$$

$$\tau_{t,ij} = -\overline{u_i u_j} = 2v_t S_{ij} - \frac{2}{3} k \delta_{ij}$$

The first term on the RHS is the production of '*k'*, the second term (*ε)* is the specific dissipation per unit mass. The last terms describe the transport of '*k'* by molecular and turbulent diffusion.

The standard k-*ε* model is the default turbulence model in Fluent. Rather than solving for a length scale it solves a second transport equation for the dissipation rate.

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \tau_{tij} \frac{\partial u_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j}\left[V + v_t/\sigma_k \frac{\partial k}{\partial x_j}\right]$$

$$\frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_j} = C_{z1} \frac{\varepsilon}{k} \tau_{tij} \frac{\partial u_i}{\partial x_j} - C_{z2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left[V + v_t/\sigma_z \frac{\partial \varepsilon}{\partial x_j}\right]$$

$$v_t = C_\mu k^2/\varepsilon$$

This model was derived and tuned for Flows with high Reynolds numbers. This implies that it is suited for flows where the turbulence is nearly iso-tropic and is suited to flows where the energy cascade proceeds in local equilibrium with respect to generation.

VI. RESULTS AND DISCUSSION

In this chapter, the results from the modeling and CFD simulation using FLUENT software are shown and discussed. Results are shown in term of graphs for the simulation results for pressure distribution, temperature distribution and Velocity.

The moving mesh is generated by DYNAMIC MESH ROUTINE, a moving mesh module in FLUENT. In engine operation, valves and the piston move, so the mesh should move according to the real engine in order to simulate the charge of valve and piston position with crank angle. Piston and piston bowl movement are decided by the stroke, connecting rod and crank angle. Calculation starts at 360^o CA and ends at 1080^o CA. A cold flow analysis is performed for this purpose. Cold flow simulations for IC engines can provide valuable design information to engineers. These simulations allow for the effect on volume efficiency, swirl and tumble characteristics to be predicted based on changes in port and combustion chamber design, valve lift timing, or other parameters.

6.1 Pressure and temperature

The graphs Fig 5 and 6 shows pressure distribution, temperature distribution 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 60 bars and temperature is 980 K.

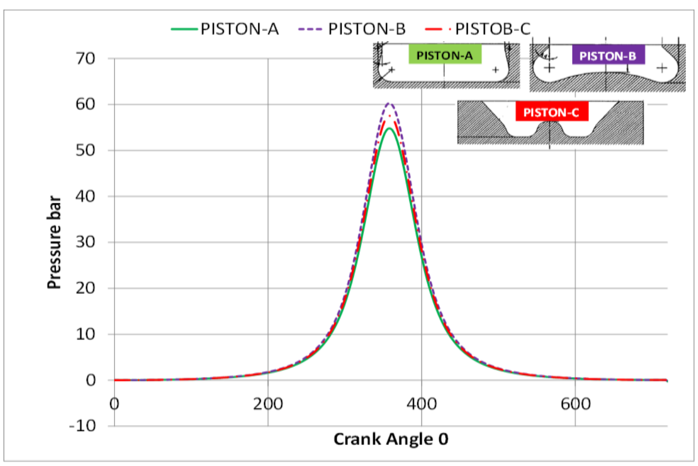


Figure 5 Pressure Vs. crank angle

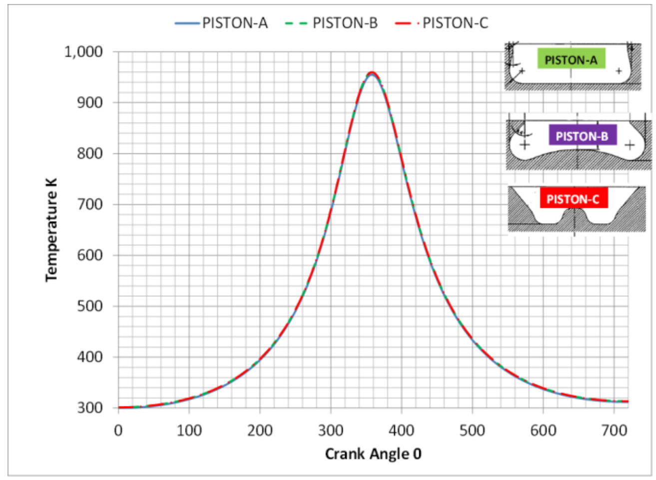


Figure 6. Pressure Vs. crank angle

6.2 Swirl and Tumble ratio inside the cylinder

Swirl and tumble ratios are generally defined as the ratio of the angular momentum of the in cylinder flow about each of the three orthogonal axes. It is normalized against the same gas rotating as solid body the same axes at crank speed. The usual method for determining these ratio is, first the centre of the combustion chamber is determined. Then the X, Y and Z are dined with the origin of the centre of the mass. The Z axis is defined as being parallel to the line of piston motion. The Y axis is defined as perpendicular to Z and parallel to central axis of the inlet manifold. Finally, the X axis is defined as perpendicular to Z and Y.

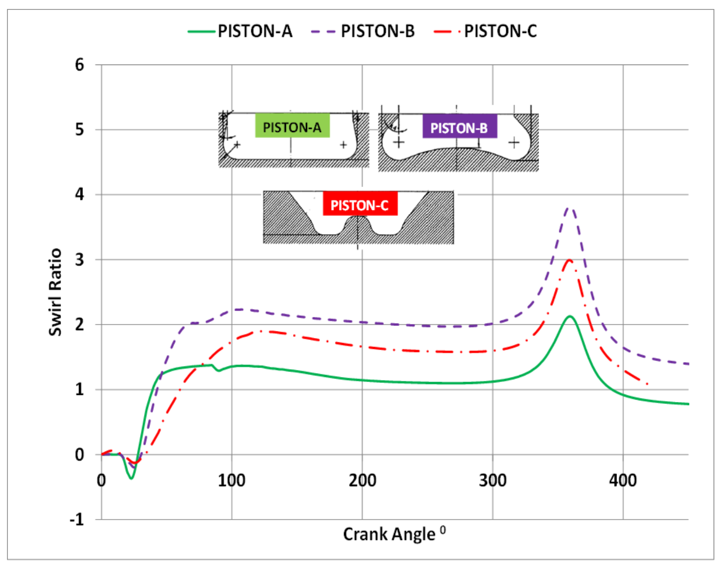


Figure 7 Swirl ratios for different piston configurations

Figure 6.9 shows the variation of Swirl Ratio (SR) inside the cylinder with respect to crank angle for different piston 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 piston B configuration over the other two pistons.

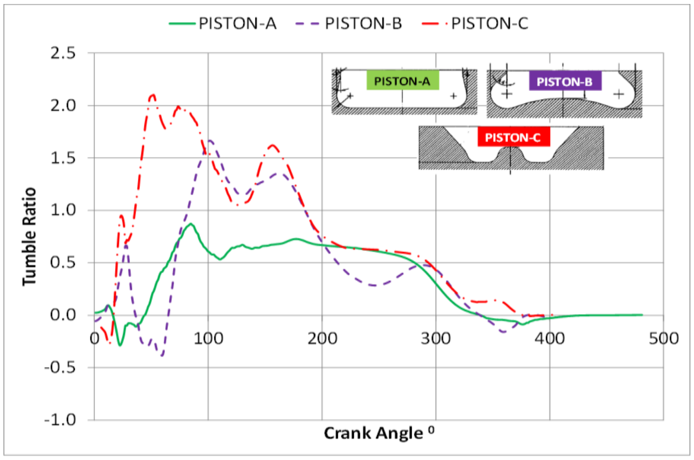


Figure.8 Tumble ratio Y for different manifold configurations

Fig. 8 and 9 shows the variation of the TR with CADs at 1000 rpm engine speed considered in this study during suction and compression strokes. It is observed that the TR ratio changes its magnitude (positive to negative or vice versa) indicating overall air movement changes its direction during entire cycle with CADs. The reasons for this could be: (i) change in the overall tumble flow pattern due to low pressure and bifurcation zones, (ii) change in piston speed with CA, and (iii) change in the direction of the piston movement during suction and compression strokes. With an increase in engine speed, variation in TR is marginal at all the CA. However, at 50 to 100 CAD, TR is maximum.

It shows Stronger the tumble motion (more TR), more the turbulent kinetic energy released during its break down at the end of compression stroke. Also, it helps in higher turbulence levels at the time of ignition. Among the intake manifold shapes considered in this study, piston A results in higher TR compared to the piston B and piston C .

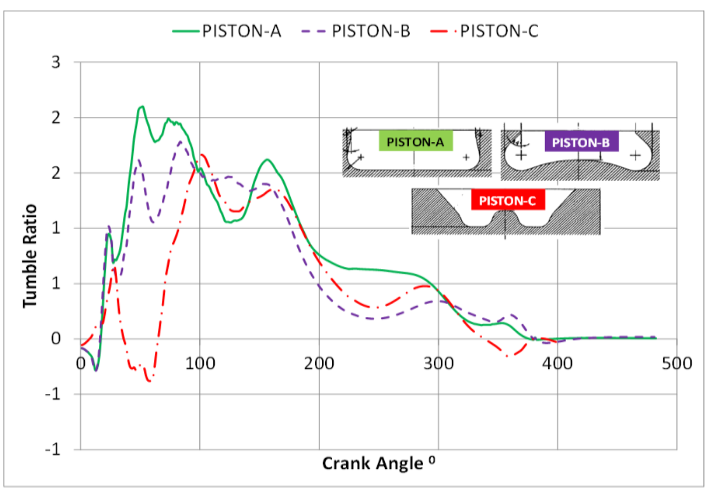
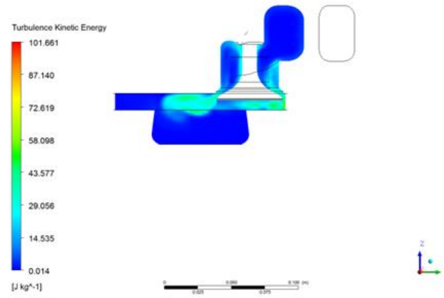


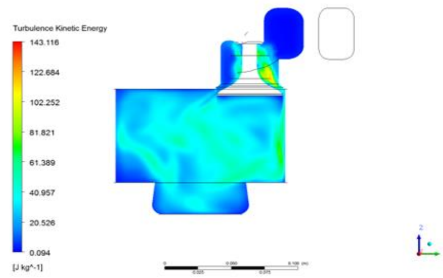
Figure.9 Tumble ratio X for different manifold configurations

6.3 Turbulent kinetic energy inside the cylinder

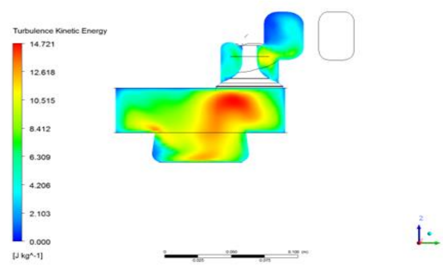
The following figure shows the turbulent kinetic energy level during the suction and compression processes at different CAs for piston A, B and C.



At 60°

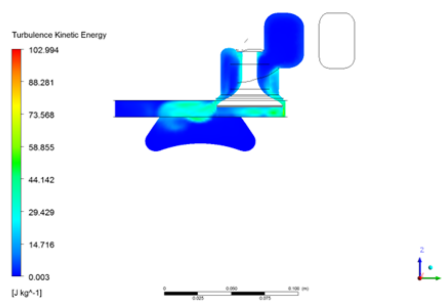


At 160°

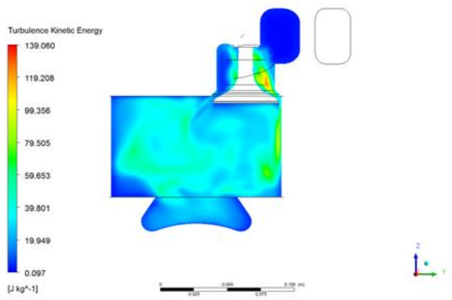


At 240°

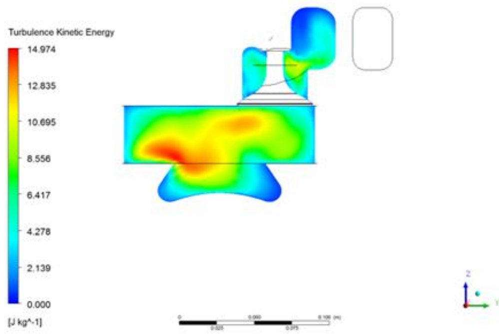
Figure 10 TKE at different crank angles of piston A



At 60°

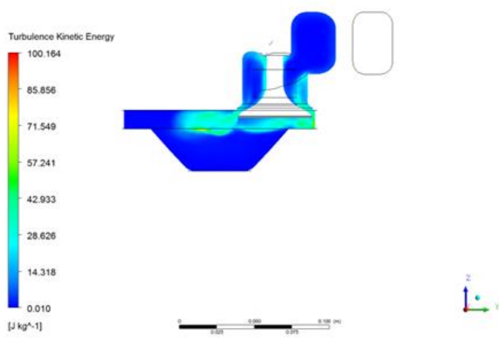


At 160º

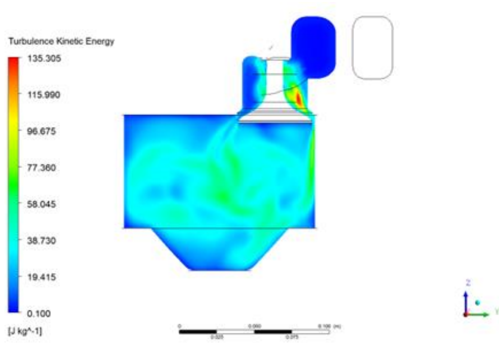


At 240º

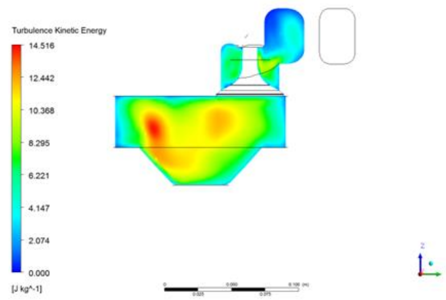
Figure 11. TKE at different crank angles of piston B



At 60°



At 160 º



At 240°

Figure 12. TKE at different crank angles of piston C

Figure 13 shows the variation of Turbulent Kinetic Energy (TKE) with crank angle at 3000 rpm for different pistons. It is observed that the piston configuration affects the turbulence of the fluid inside the cylinder. It reaches the peak value during the maximum valve open condition. The variation of TKE is probably due to different level of air induced through the inlet manifold. The dissipation of KE is on account of increased fluid motion. Due to this, high SR is observed for piston B than other pistons and low level of TKE as shown in Figure 7, while piston A shows low SR as well as low TKE

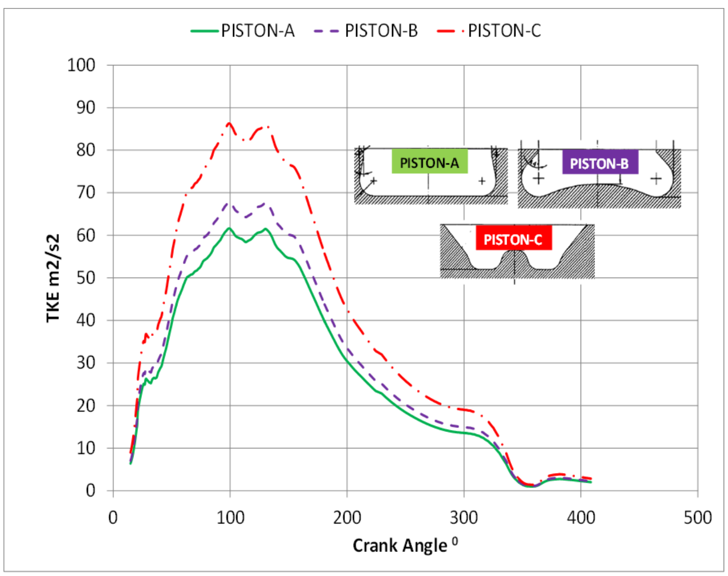


Figure 13 Turbulent Kinetic energy for different piston configurations

6.4 Volumetric efficiency

The piston A shows lower volumetric efficiency whereas Piston C shows higher volumetric efficiency. Turbulence consumes energy; hence swirl produced in the cylinder consumes energy. As piston B has maximum swirl ratio volumetric efficiency in this piston is less and therefore volumetric efficiency in piston C is maximum. Volumetric efficiency is interrelated with swirl.

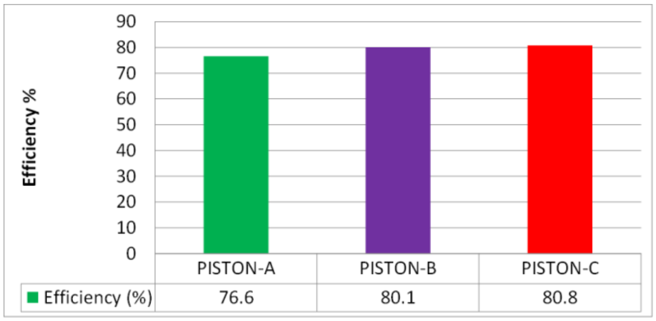


Figure 14 Volumetric Efficiency for different piston configurations

VII. Conclusions

Development of any internal combustion engine is driven primarily by fuel efficiency and emission requirements. This requires refinement of the in-cylinder flow, mixture formation and combustion processes. Design optimization of the intake/exhaust port, valves and piston bowl is essential to realize the above mentioned requirements. The use of Computational Fluid Dynamics (CFD) along with optimization tools can help shorten the design optimization cycle time. Traditional approach of experiments using flow bench testing is very costly as well as time consuming. Moreover CFD allows insight into the minute flow details which otherwise are not capture using flow bench tests. Air motion inside the intake manifold is one of the important factors, which govern the engine performance and emission of multi-cylinder diesel engines. Hence from the literature study a Helical-spiral combination intake manifold is considered. The geometry of the piston amplifies the air motion at the end of compression stroke. In this project work, the internal flow characteristic in the combustion chamber of a diesel engine is investigated computationally for the different piston configurations. The governing equations for unsteady, three-dimensional, compressible, turbulent flow are solved with the two equation RNG k-ε model to consider the complexity of the geometry and fluid motion. The overall flow field inside the combustion chamber and various quantities, such as pressure, velocity distribution, swirl and tumble ratios were examined for all three types of pistons.

Swirl and tumble ratios can be obtained quite accurately for both production and research engines using computational fluid dynamics. The summary of the comparison is as follows:

- Piston B creates higher swirl inside the cylinder than piston C.
- Piston B provides higher volumetric efficiency. Improvement of volumetric efficiency is achieved by considering piston C.
- Piston C provides higher Turbulent kinetic energy as well as tumble x and y ratio.
- Piston A provides comparatively low swirl ratio, low turbulent kinetic energy and low volumetric efficiency.

References

[1] Aita S, Tabbal A, Munck G, Montmayeur N, Takenaka Y, Aoyagi Y, *et al.* . Numerical simulation of swirling portvalve- cylinder flow in Diesel engine. SAE 910263, 1991

[2] 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.

[3] Celik I, Yavuz I, Smirnov A. Large eddy simulations of in-cylinder turbulence for internal combustion engines: A Review. Int J Engine Res 2001;2(2):119–48.

[4] Chen A, Veshagh A, Wallace S. Intake flow predictions of a transparent DI Diesel engine. SAE 981020, 1998.

[5] Dillies B, Ducamin A, Lebrere L, Neveu F. Direct injection Diesel engine simulation: a combined numerical and experimental approach from aerodynamics to combustion. SAE 970880, 1997

[6] F. Payri , J. Benajes, X. Margot , A. Gil, CFD modeling of the in-cylinder flow in direct-injection Diesel engines, CMT-Motores T_ermicos, Universidad Polit_e cnica de Valencia, Camino de Vera s/n, 46022 Valencia, Spain.

[7] IC Engines by V Ganesan-Tata McGraw-Hill Education, 2002

[8] J. David Rathnaraj and Prof. T. Michael. N. Kumar, "Studies on variable swirl intake system for DI diesel engine using CFD", ISSN 0973-4562 volume 2, number 3 (2007)

[9] Patankar VS. Numerical heat transfer and fluid flow. Washington: Hemisphere Publishing Corp; 1980

Authors

H.Sushma, doing master's degree in Thermal power engineering at B.T.L.INSTITUTE OF TECHNOLOGY AND MANAGEMENT, Bangalore which affiliated to VISVESVARAYA TECHNOLOGICAL UNIVERSITY, Belgaum-590014, India,

Jagadeesha.K.B, Lecturer, Department of mechanical engineering, in B.T.L.INSTITUTE OF TECHNOLOGY AND MANAGEMENT, Bangalore.