Numerical Simulation of In-cylinder flow on different piston bowl geometries

[1]Santhosh Kumar Gugulothu, [2]K.Hema Chandra Reddy

[1]Research Scholar, Mechanical Engineering, JNTU Hyderabad, Telangana, India
[2]Registrar, JNTUA Anantapur, Andhra Pradesh,India
Email: [1]santoshgk1988@gmail.com

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 all the efficiency of these engines. Every attempt made to improve these engines tends to attain the maximum efficiency. The study is about the effects of air swirl in the cylinder on its performance. For obtaining different swirl intensities the following design parameters have been changed the piston crown, cylinder head and inlet duct. By changing the piston crown design the enhancement in the turbulence inside the cylinder is achieved. Also grooves are made to achieve the increase in the swirl intensity for better mixing of fuel and air. Numerical analysis is done on different piston bowl geometries i.e toroidal chamber, double hemispherical bowl, Mexican hat, hemispherical bowl. Analysis of pressure and temperature variation inside the cylinder, Turbulence kinetic energy and turbulence intensity variation are analysed for different piston profiles.

Keywords: Direct injection spark ignition engine; Combustion chamber; Turbulence kinetic energy; Turbulence intensity.

I. INTRODUCTION

Internal combustion engines are the engines which burn the fuel inside it and produce energy. Of all the engines the direct injection engines have their own importance because of their higher thermal efficiencies than all the other engines have their own importance because of their higher efficiencies than all other. They can be used for both light-duty vehicles. Fuels are non-renewable resources of energy therefore the maximum usage of energy available from them is to be achieved. In internal combustion engines the efficient burning the fuel i.e.., combustion of the fuel is required to increase the efficiency of the engine. To obtain efficient combustion the fuel injected is to be spatially well distributed throughout the entire space. This requires matching of the fuel sprays with the geometry of the combustion chamber to effectively male use of gas flows. Here air is made to swirl for better mixing of fuel and air which increases rate of mixing and reduces combustion duration. The higher the swirl reduces the soot emission at the cost of higher NOx level.

The in-cylinder fluid motion in internal combustion engines is one of the important factors in controlling the combustion process. It governs the air-fuel mixing and rate of burning in diesel engines. Therefore better understanding of fluid motion is critical for designing the engines with most desirable operating and emission characteristics.

To obtain a better combustion with lesser emission on direct injection diesel engines, it is necessary to achieve a good spatial distribution of injected fuel through entire space. This requires matching of fuel sprays with combustion chamber geometry, fuel injection and gas flows is the most crucial factor for attaining better combustion. Since the flow in combustion chamber develops from the interaction of the intake flow with the in-cylinder geometry, the goal of this work is to characterize the role of combustion chamber geometry on the in-cylinder flow, fuel-air mixing, and combustion and pollutant formation process.

CFD Analysis of in-Cylinder Flow and Air Fuel Interaction in Different Combustion Chamber Geometry in DISI Engine was done by B.Harshavardhan & J.M Mallikarjuna. The models in their analysis were 2D. The piston bowls on which analysis was done were flat piston, flat piston with center bowl, dome piston with center bowl and pent roof piston. CFD analysis has been carried out on in-cylinder fluid flow and air fuel interaction in Direct Injection Spark Ignition (DISI) engine by changing the combustion chamber geometry during intake and compression stroke at an engine speed of 1500 rpm for 4 different piston profiles.

The objective of our present work is to develop a multi dimensional fluid flow which can predict in-cylinder fluid motion behavior by using different piston bowl geometries. All the profiles are 3 dimensional and are designed in CATIA. In order to study the fluid flow characteristics CFD calculation of suction and compression stroke will be performed. Analysis of Pressure and Velocity Distribution, Turbulent Kinetic Energy and Turbulent Intensity is done on 4 piston profiles. Software used for the analysis is ANSYS Fluent.

II. GEOMETRICAL AND COMPUTATION 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.

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°

The engine selected is a single cylinder research DI diesel engine with inlet manifold.

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 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 and is exported to ANSYS FLUENT for completing the mesh.

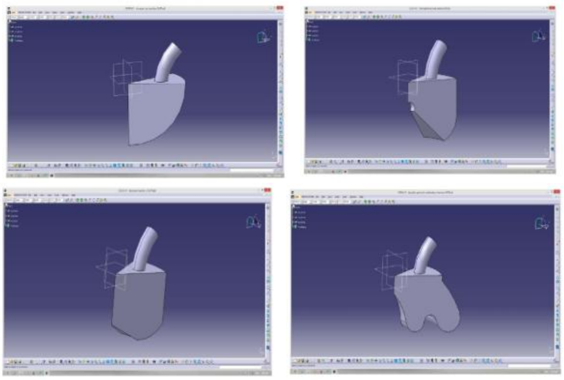
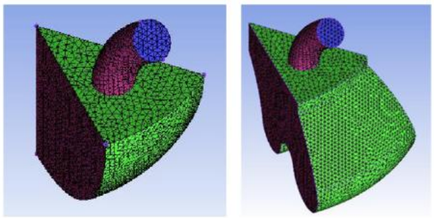


Figure 1: 45^{o} sector of different piston bowl geometries

IV. GRID INDEPENDENCE STUDY:



During analysis of a particular component, we mesh the component with certain number of grids. The more the no. of grids or nodes the more accurate are the results. But doing analysis at each and every node for a large no. of node components is a bit time consuming process. In case of grid independence study we make sure that, the no. node doesn't affect the out coming results. By grid independence study we can get the results accurately and in fewer spans as grid has no affect over the analysis. In our analysis we are using triangular grids. Triangular nodes can give accurate results as they cover each and every small part of the component.

V. CFD ANALYSIS OF IC ENGINE

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".

5.2 Transient IC engine analysis

To carryout IC engine dynamic analysis meshed model of manifold with combustion chamber is imported into ANSYS Fluent. 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

Turbulent intensity: The turbulent intensity is of 5%. When setting boundary conditions for a CFD simulation it is often necessary to estimate the turbulence intensity on the inlets. To do this accurately it is good to have some form of measurements or previous experience. According to CFD the turbulent intensity in case of complex geometries and for the high turbulence the intensity of turbulence ranges from 5-20%.

Mixing length scale: Mixing length scale is equal intervals of length between which the flow of the molecules is analyzed. Mixing length scale which was taken in our analysis is 0.001mm.The small is the mixing length scale more is the tendency to analyze the flow.

Initial pressure at start of computation: 0.99 bar and ambient Temperature: 298 K.

Inlet port pressure: 1.013 bar.

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 figures for the simulation results for pressure distribution, temperature distribution and Velocity.

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. The entire figure shows that turbulence kinetic energy contour plot at end of suction stroke with different piston bowl geometries in the plane passing through centre of the geometry. It can be observed from the figures that TKE, is uniformly distributed all over the combustion chambers for different piston bowls. It is found that toroidal chamber has more TKE(50.98m^2/s^2) compared to other bowls.

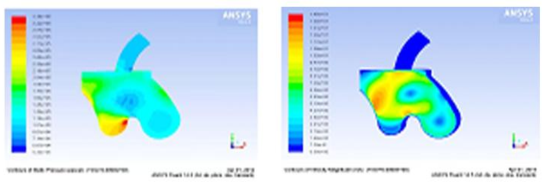
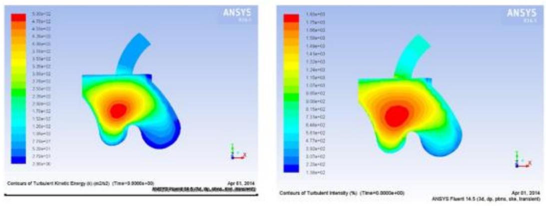


Figure 2 a) Pressure & velocity contour of DSCB



b) Turbulence kinetic energy and turbulence intensity contours of DSCB

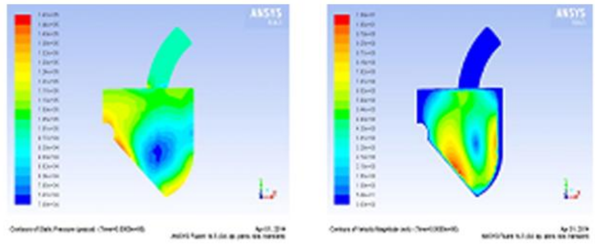
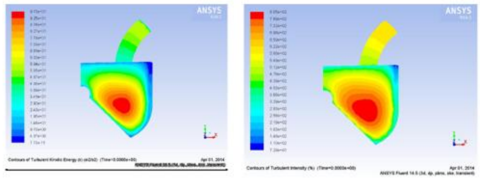
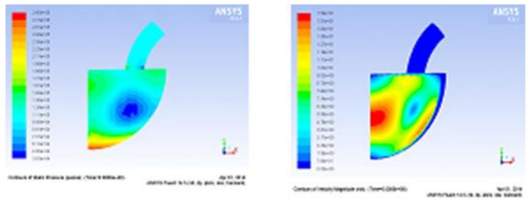


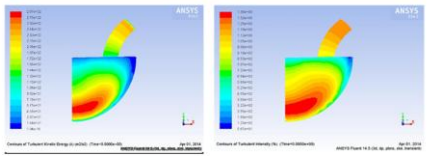
Figure 3 a) Pressure & velocity contour of Hemispherical bowl



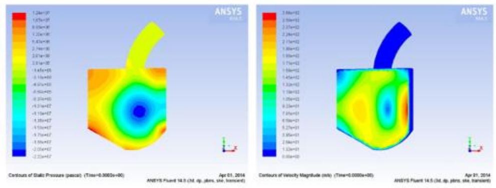
b) Turbulence kinetic energy and turbulence intensity of Hemispherical bowl



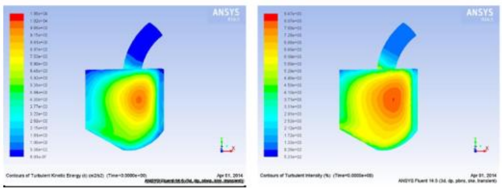
4 a) Pressure and velocity contours of Mexican hat



b) TKE & Turbulence intensity of Mexican Hat



5 a) Pressure & Velocity contours of Toroidal chamber



b) TKE & turbulence intensity of Toroidal chamber

VII. CONCLUSION:

In-cylinder flow is mainly based on the turbulence intensity. The more is turbulence intensity more is the mixing of air and fuel. Turbulent Kinetic energy is another factor that plays a major role in air and fuel interaction next to Turbulence intensity.

From the analysis it is observed that CFD computation study is important to understand in-cylinder flow structure during intake and compression stroke.

Based on the analysis of 4 different piston bowls by sending the diesel into the combustion chamber, more turbulence was found in Toroidal Chamber. The range of intensity of turbulence in case of high turbulence according to CFD analysis is 5-20%. The intensity of turbulence in case of Toroidal Chamber is in between 5-20%. Hence we conclude that Toroidal Chamber is the best piston bowl out of 4 analyzed pistons.

The TKE is distributed for all over the combustion chambers, TKE of toroidal chamber is slightly higher than other combustion chamber. Air-fuel mixture is distributed all over the combustion space all the combustion chambers. Distribution of air fuel is slightly rich at the centre of geometry.

VIII. 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] B. Harshavardhan & J.M. Mallikarjun, “CFD analysis of in cylinder flow and air fuel interaction on different combustion chamber geometry in DISI engine”, IIT Madras.

[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] 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.

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

[6] 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

[7] 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.

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

[9] 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)

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