International Journal of Computational Research and Development (IJCRD)
Impact Factor: 4.775, ISSN (Online): 2456 - 3137
(www.dvpublication.com) Volume 1, Issue 2, 2016

PERFORMANCE IMPROVEMENT OF IC ENGINE BY CHANGING PISTON CONFIGURATION & REDUCING EMISSION BY USING CFD

G. Saravankumar* & S. Arul Angappan**

* PG Student, SSM College of Engineering, Komarapalayam, Tamilnadu
** Assistant Professor, Department of Mechanical Engineering, SSM College of Engineering, Komarapalayam, Tamilnadu

Cite This Article: G. Saravankumar & S. Arul Angappan, "Performance Improvement of IC Engine by Changing Piston Configuration & Reducing Emission by Using CFD", International Journal of Computational Research and Development, Volume 1, Issue 2, Page Number 18-21, 2016.

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 16.0. Three dimensional models of the manifolds, pistons and the cylinder is created in CATIA V5 and meshed using the pre-processor ANSA v15.1.1.

Key Words: Exhaust Manifold, Swirl Intensity, Turbulence & Inlet Manifold

1. 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.

2. Computational 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 16.0 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. Preprocessor mainly involves the creation of basic 3D model, grid generation and fixing of the boundary conditions. Modeling and meshing is done in ANSA v15.1.1. and is exported to ANSYS FLUENT 16.0 for completing the mesh.

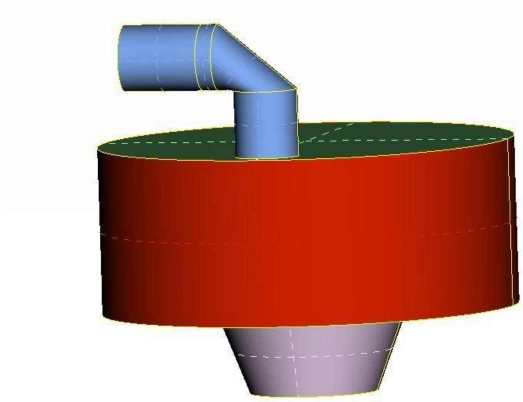


Figure 1: Piston

3. Grid Generation:

Two approaches are employed in Fluent 16.0 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 2: Grid Generation

4. CFD Analyses of IC Engine:

Governing Equation 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).

Transient IC Engine Analyses: To carryout IC engine dynamic analysis meshed model of manifold with combustion chamber is imported into ANSYS Fluent 16.0. 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.

Boundaries and Initial Condition: 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.

Mathematical Model: In this Paper 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.

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

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

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. 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.

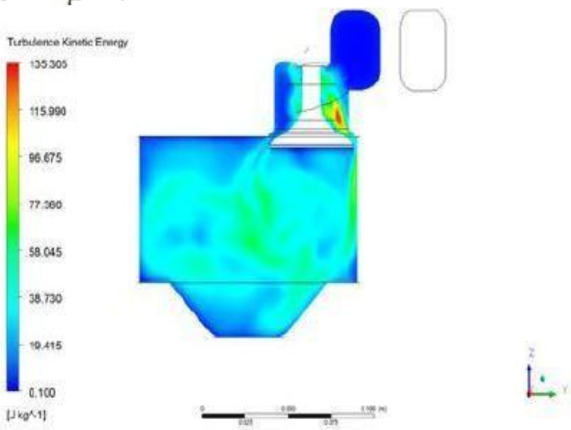
5. Results:

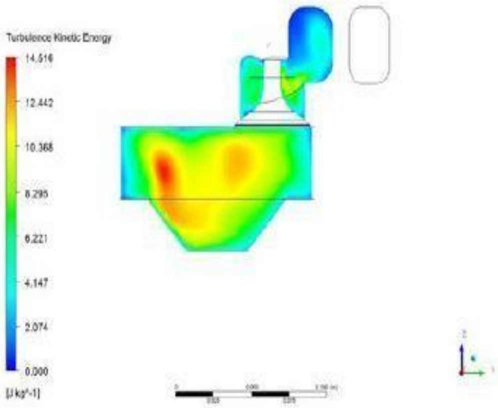
The results from the modeling and CFD simulation using FLUENT software are shown and discussed. Results are shown in term of graphs for the simulation results for pressure distribution, temperature distribution and Velocity. The moving mesh is generated by DYNAMIC MESH ROUTINE, a moving mesh module in 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 360^0 CA and ends at 1080o 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.

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

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

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





6. Conclusion:

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.

7. 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