

# Simulation of In-cylinder flow in Direct- Injection Diesel engine through CFD

Balaji M<sup>1</sup>, Hema Chandra Reddy K<sup>2</sup>, Bhanu Priya K<sup>3</sup>

Research Scholar, Department of Mechanical Engineering, JNTU Anantapur, AP, India<sup>1</sup>

Professor, Department of Mechanical Engineering, JNTU Anantapuramu, AP, India<sup>2</sup>

Research Scholar, Department of Mechanical Engineering, JNTU Anantapur, AP, India<sup>3</sup>

**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. 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. 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. 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 Star CCM+.

**KEYWORDS:** CFD, inlet manifold, piston configurations, swirl ratio, Turbulence kinetic energy; Turbulence intensity.

## 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.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 1, January 2017

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.

## II. RELATED WORK

Benajes and Margot *et al.* [3], 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.

Aitaet *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.* [2] 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  $k-\epsilon$  model. Dillies *et al.* [6] 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. Celiket *al.* [4] made a review of computations based on large eddy simulation

## III. ENGINE AND COMPUTATIONAL DETAILS

The engine selected is a single cylinder research DI diesel engine with helical shape inlet manifold. The base engine is same for all three piston configurations CFD analysis. There are different types can be considered like spiral, helical spiral combination, etc. may tested. The detailed specification of the base engine selected for the simulation is given in Table 1.

**Table 1: Geometrical characteristics of the engine**

|                       |                      |
|-----------------------|----------------------|
| Bore                  | 130.0 mm             |
| Stroke                | 150.0 mm             |
| Connecting rod length | 275.0 mm             |
| Displacement          | 1991 cm <sup>3</sup> |
| Intake Valve diameter | 44.4 mm              |
| Intake Valve angle    | 60 <sup>o</sup>      |

## IV. METHODOLOGY

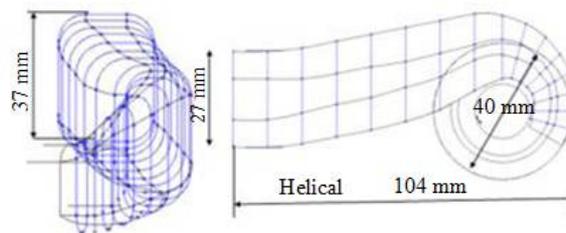
The methodology adopted for the present work is as follows. Flow through the intake manifold is simulated to study the in cylinder flow field during non-reacting conditions, which includes the following steps:

- Solid modelling 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 Star CCM+ 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

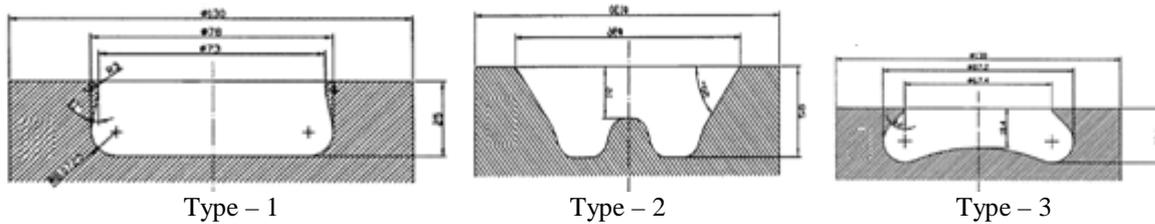
1. Pre-Processor
2. Solver
3. Post Processor

Pre-processor mainly involves the creation of basic 3D model, grid generation and fixing of the boundary conditions. Modeling and meshing is done in Gambit and is exported to Star for completing the mesh.



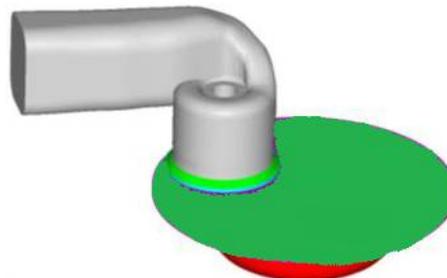
**Fig. 1 Geometric model of IC Engine manifold**

Geometrical model of Internal Combustion Engine manifold by using mesh in Gambit modelling software



**Fig. 2: Shapes of IC Engine manifold of 3 different types**

Different types of piston heads shapes with three different dimensions



**Fig. 3: CFD Model of IC Engine Piston at TDC**

Modelling of engine piston by using Star CCM+ simulation software

## 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".

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 1, January 2017

## 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.

## 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: 0.95 bar and ambient Temperature: 299 K.

Inlet port pressure: 1.013 bar.

## VI. RESULTS AND DISCUSSION

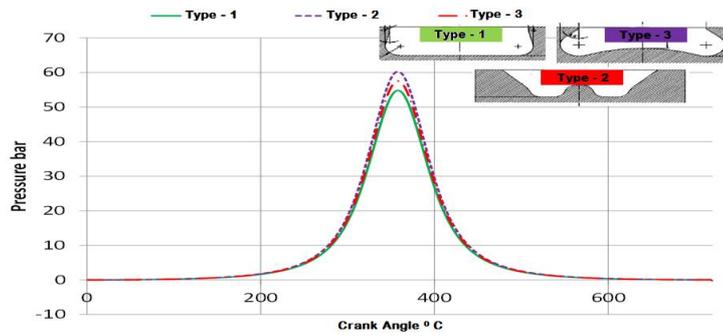
In this chapter, the results from the modeling and CFD simulation using Star CCM+ software are shown and discussed. Results are shown in term of graphs for the simulation results for pressure distribution, temperature distribution and Velocity.

The moving mesh is generated by DYNAMIC MESH, a moving mesh module in Star CCM+. 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  $360^{\circ}$  CA and ends at  $1080^{\circ}$  CA.

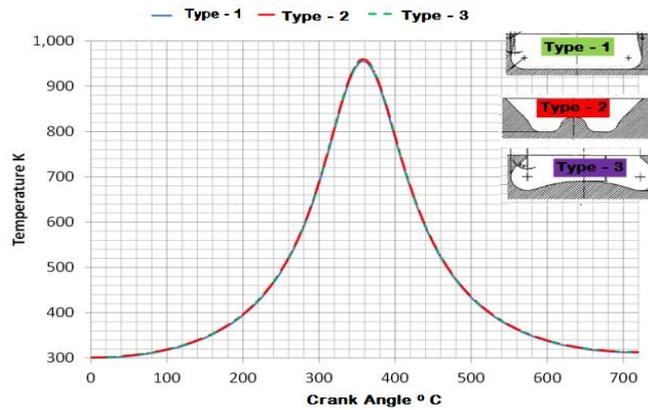
A cold flow analysis is performed for this purpose. Cold flow simulations for IC engines can provide valuable design information to engineers. These simulations allow for the effect on volume efficiency, swirl and tumble characteristics to be predicted based on changes in port and combustion chamber design, valve lift timing, or other parameters.

### A. PRESSURE AND TEMPERATURE

The graphs Fig 4 and 5 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  $0.25^{\circ}$  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- $\theta$  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.



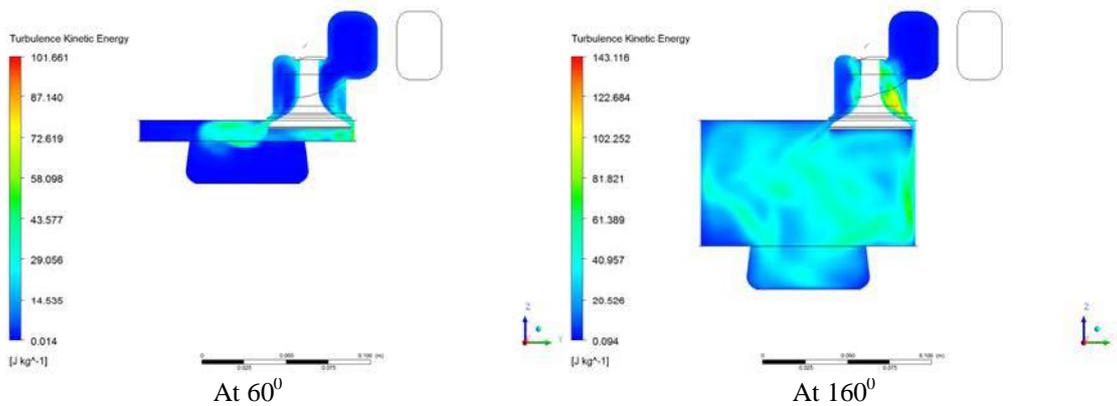
**Fig. 4 Pressure Vs. crank angle**



**Fig. 5. Pressure Vs. crank angle**

**B. TURBULENT KINETIC ENERGY INSIDE THE CYLINDER**

The following figure shows the turbulent kinetic energy level during the suction and compression processes at different Crank Angles for type1, 2 and 3.



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 1, January 2017

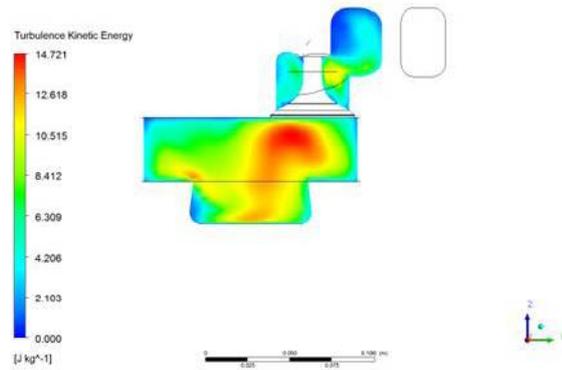


Fig 6 TKE at different crank angles of Type 1

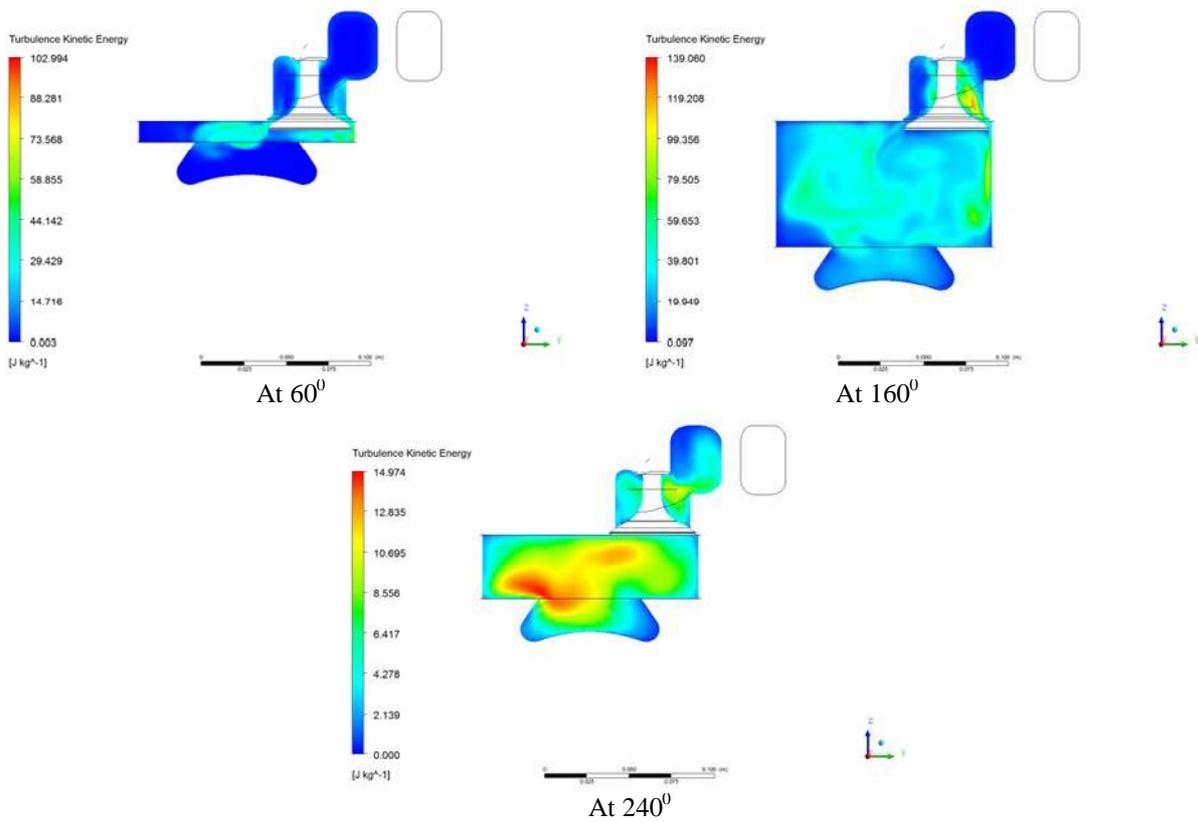


Fig 7. TKE at different crank angles of Type – 2

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 1, January 2017

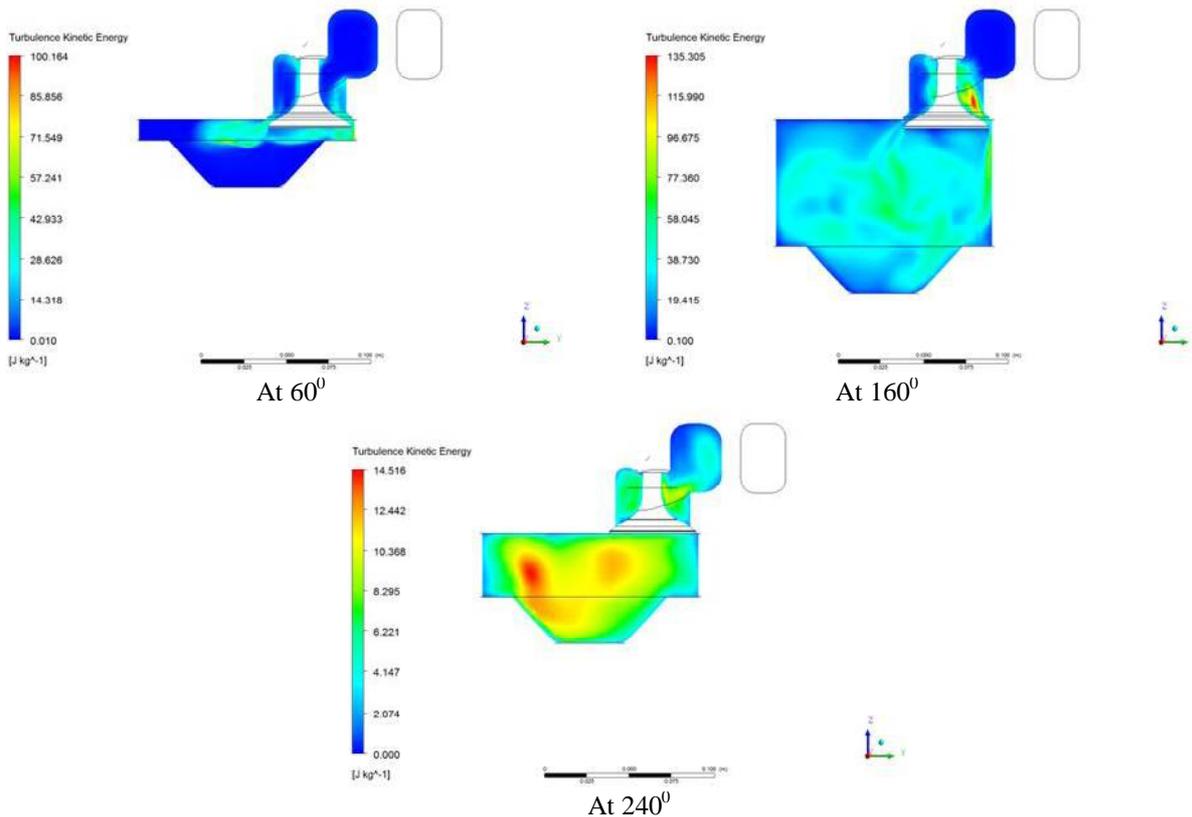


Fig 8. TKE at different crank angles of Type – 3

### C. VOLUMETRIC EFFICIENCY

The Type 1 shows lower volumetric efficiency whereas Type 2 shows higher volumetric efficiency. Turbulence consumes energy; hence swirl produced in the cylinder consumes energy. As Type 3 has maximum swirl ratio volumetric efficiency in this piston is less and therefore volumetric efficiency in Type 2 is maximum. Volumetric efficiency is interrelated with swirl.



Fig 9 Volumetric Efficiency for different piston configurations

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 1, January 2017

## 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- $\epsilon$  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:

- Type 3 creates higher swirl inside the cylinder than Type 2.
- Type 3 provides higher volumetric efficiency. Improvement of volumetric efficiency is achieved by considering piston C.
- Type 2 provides higher Turbulent kinetic energy as well as tumble x and y ratio.
- Type 1 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] Chen A, Veshagh A, Wallace S. Intake flow predictions of a transparent DI Diesel engine. SAE 981020, 1998.
- [3] F. Payri , J. Benajes, X. Margot , A. Gil, CFD modeling of the in-cylinder flow in direct-injection Diesel engines, CMT-MotoresT\_ermicos, Universidad Polit\_ecnica de Valencia, Camino de Vera s/n, 46022 Valencia, Spain.
- [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] 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.
- [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] 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
- [10] K. Nakakita, 1994, "Optimization of Pilot Injection Pattern and Its Effect on Diesel Combustion with High-Pressure Injection," JSME Int. J. Ser. B.37, pp. 966–973. [ISI]
- [11] D. A. Pierpont, and R.D. Reitz, 1995, "Effects of Injection Pressure and Nozzle Geometry on D.I. Diesel Emissions and Performance," SAE Paper No. 950604.
- [12] M. Patterson and R.D. Reitz, "Modeling the Effects of Fuel Spray Characteristics on Diesel Engine Combustion Emissions," SAE 980131, 1998.