Asian Review of Mechanical Engineering
ISSN: 2249-6289 (P) Vol.1 No.1, 2012, pp.42-46

DOI: https://doi.org/10.51983/arme-2012.1.1.2509

Mixture Preparation in a Four-Stroke Compression Ignition Engine - A CFD Approach

G. Kalivarathan[1] and V. Jaiganesh[2]

[1]*Research Scholar, CMJ University, Shillong, Meghalaya - 793 003, India.*

[2]*Department of Mechnical Engineering, S.A Engineering College, Chennai - 600 077, Tamil Nadu, India.*

E-mail: Sakthi_eswar@yahoo.com

(Received on 07 December 2011 and accepted on 15 January 2012)

Abstract - **A CFD simulation is carried out to predict the effects of piston crown on the fluid flow field, inside the combustion chamber, cylinder of four strokes CI engine. The complete analysis is concentrated on the study of effects of the piston shape. On the fluid flow characteristics, fluid flow dynamics exhibits an important role on charge preparation to attain the better combustion performance and efficiency in the viscosity of swirl and tumble flows as turbulence. The turbulence parameters like swirl and tumble represents the fluid flow behavior occurs inside the combustion chamber, which influence the air streams to the cylinder during intake stroke and reasonably enhances the mixing of air and fuel to have a better mixing during compression stroke. The numerical calculations are preferred in a single cylinder four stroke diesel engine running at full throttle condition by using the CFD mode. Different Bowls are considered with an engine speed of 2200 rpm, for this analysis, to be compared to evaluate the swirl and tumble flows produced during intake and compression stroke. The results obtained from the numerical analysis can be used to predict uniformity in the structure of a charge for a efficient performance and combustion in a compression ignition engine.**

Keyword: **Computational Fluid Dynamics, Turbulence, Charge, piston Geometry, Swirl, Tumbles, Larger Eddies, Diesel**

I. Introduction

In general mixture preparation is considered as a very important criteria in case of diesel engine, because of its heterogeneous combustion. Since high-pressurized diesel fuel is injected into a high compressed hot air, it is necessary to concentrate on the mixture formulation for predicting the turbulence over spray combustion. Because of cycle-to-cycle variation, the incidental flow structure of diesel engines have drawn more attention to the researchers in the existing situations. It is mainly due to the reason that the flow structures accomplished by intake flows is closely related to the performance as well as the design of compression-ignition-engines. The inducement of high turbulence intensity is considered as one of the very important factors for maintaining the combustion process and rapid propagation of flame, particularly in the case of lean-burn combustion. In general, there are three types of turbulence motions are captured in order to create and maintain the flow of turbulence reasonably. These parameters are referred to swirl, tumble and eddies; which are organized rotations in the horizontal and vertical plane of the cylinder of an engine. The improvement of the engine performance is attributed by accelerating the mixing of fuel and induced air therefore it is necessary to predict the overall development of a diesel engine with high compression ratio, high turbulence intensity and lean-burn combustion.

Even though experimental techniques are available unpracticed, it is really an odd task to execute it because the measurements of in-cylinder flows in the diesel engine are characterized by three-dimensional aspects of turbulence and unsteadiness because of the capability of CFD, the detailed investigation for in-cylinder flow is considered as an alterative numerical approach. In the existing practice star-cd CFD code is mostly used as a package for in cylinder flow analysis in the case of diesel engines. Computer codes are generally used to solve the navier-stokes-equations to generate detailed information of the men velocity and the turbulence velocity fields. In the context of diesel engines the CFD modeling is expected to cover the specific problems related to the turbulent flow, complex shape, high Reynolds number and compressible flow. Moreover, the computing are normally costly and it requires large computer memory and high performance computing facilities to minimize the works.

In this investigation, star-cd CFD code is used to predict the effect of the piston crown shape to the fluid flow field. The homogeneity of air structure for fuel mixing preparation that happens inside the cylinder of an engine is also analyzed to predict the feasible piston crown. It is noticed that the fluid flow with different combustion chamber has an influence in mixture preparation and generation o turbulence as well as exhibits more impact on engine performance. Different combustion chambers with reasonable geometry accepted for a four-stroke diesel engine are considered in detail for in-cylinder flow structures during intake and compression stroke in order to predict the effect of the combustion chamber geometry to the fluid flow field, the characteristics of large scale mixing are investigated in detail to represent the behavior of swirl, eddies and tumble flow along the degree of crank angle. The numerical computation is implemented by using the unsteady analysis of intake and compression stroke for different piston crown along with suitable boundary conditions. For a generic context the engine speed is taken as 2200rpm with the constant valve timing and lift. Swirl, tumble and eddies are taken into account to verify the homogeneity of air structure for charge preparation the reasonable air fuel mixture and combustion process can be obtained.

Especially the dissimilarities noticed for the two different piston bowl geometry in the configuration of fluid flow characteristics during the intake and compression stroke are analyzed. From the prediction, the feasible geometry of piston crown can be determined and it is chosen for the improvement of processes in order to obtain the better engine performance and understanding the influence of piston geometry on the characteristics of air structure pattern in diesel engine is normally considered as a very important criteria to take care of the requirements of both performance and emission. Apart from that, this investigation can be treated as a research strategy to provide an alternative way instead of using the conventional experimental techniques.

II. CFD simulation of IC engine process

The computational fluid dynamics simulations of this investigation are performed for intake and compression stroke by using the moving mesh boundary algorithm. Every event represents the different mesh and boundary geometries for every crank angle in each step of engine cycle. Therefore, in order to perform a proper CFD simulation for an IC engine process, the analysis are carried out by using unsteady calculation, moving meshes and boundaries, high compressible Reynolds number, high turbulence intensity, momentum, heat and mass transfer and complex geometries model. The equations used to describe mass, momentum, energy and k-epsilon turbulence model in the vector notations without source terms from spray and chemical reactions under motoring conditions are expressed as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\rho u) + \nabla \cdot (\rho u u) = \nabla P - \nabla\left(\frac{2}{3}\rho k\right) + \nabla \cdot \sigma + \rho g \qquad (2)$$

$$\frac{\partial}{\partial t}(\rho I) + \nabla \cdot (\rho I u) = -P\nabla \cdot u - \nabla\left(\frac{2}{3}\rho k\right) + \nabla \cdot J + \rho \varepsilon \qquad (3)$$

$$\frac{\partial \rho}{\partial t}(\rho k) + \nabla \cdot (\rho u k) = -\frac{2}{3}\rho k \nabla \cdot u + \sigma : \nabla u + \nabla\left[\left(\frac{\mu_t}{Pr_k}\right)\nabla k\right] - \rho \varepsilon \qquad (4)$$

$$\frac{\partial \rho}{\partial t}(\rho \varepsilon) + \nabla \cdot (\rho u \varepsilon) = -\left(\frac{2}{3}c_{\varepsilon 1} - c_{\varepsilon 3}\right)\rho \varepsilon \nabla \cdot u + \nabla\left[\left(\frac{\mu_t}{Pr_\varepsilon}\right)\nabla \varepsilon\right] + \frac{\varepsilon}{k}\left[c_{\varepsilon 1}\sigma : \nabla u - c_{\varepsilon 2}\rho \varepsilon\right] \qquad (5)$$

Where

ρ =Density

$\vec{U}$ =Velocity vector

P = Pressure

$\vec{\sigma}$= Turbulent viscous stress tensor

I = Specific internal energy

$\vec{j}$ = Heat flux vector including turbulent heat conduction and enthalpy diffusion effects

III. Specification of the Engine Model for Simulation

No.of.cylinders	: 4
Bore	: 78 mm
Stroke	: 84 mm
Displacement volume	:15.96 cm^3
Length of the connecting rod	: 135 mm
Crank radius	: 46 mm
Compression ratio	: 16
Intake valve opening	: 14 b TDC
Intake valve closing	: 46 a TDC
Exhaust valve opening	: 44 b BDC
Exhaust valve closing	: 12 a BDC
Maximum intake valve-lift	: 9.2mm
Maximum exhaust valve-lift	: 7.4mm
Combustion chamber	: bowl piston

IV. Engine Geometry and Operating Conditions

A single cylinder engine with a intake and exhaust valve is considered for this investigation. Two piston geometries are considered to investigate the behavior of swirl and considered to investigate the behavior of swirl and tumble flows occurring inside the cylinder in order to obtain the reasonable shape of the piston for combustion process. These two shapes

are representative of the real engine geometry model that is normally operated to obtain the high compression ratio s well as the optimum combustion process in a diesel engine. The specifications and characteristics of operating conditions from engine model practiced can be employed for the computational study and boundary conditions CFD analysis during intake and compression stroke are given in the table.

V. Grid Generation

A grid generation has explored to generate the grid to create the hexahedral cells for the engine model. The computational fluid dynamics for the CFD calculations covers intake ports and valves, the cylinder head and piston bowl. The number of cells varies from 90,000 cells in TDC to around 1,00,000 cells in BDC position, where about half of the cells used to generate the mesh at the cylinder head and piston bowl in the case of considering the grid sensitivity and a reasonable processing time. A fine grid structure is necessary for mesh snapping during the valve movement. The hexahedral cells have been adopted for cell generation, because it provides a reasonable accuracy and also stability as compared to the tetrahedral cells. The requirements of moving meshes and boundaries to accomplish the CFD calculation are normally considered as important criteria for the usage of hexahedral cells. Because of the complexity of the engine valve the computational mesh is divided into four area where each area has been meshed separately. This strategy is implemented to obtain good quality grid and reduce the meshing time significantly both intake part meshes have been generated using a similar specifications, where the cells are oriented in the flow direction and joined with cylindrical structural mesh in the zone, upstream of the valves the grid above the valve in both intake and exhaust have been constructed by the revolution of the structure mesh section during the compression stroke, when intake valves are closed, the sub domain of intake ports re already disappeared from the calculation to reduce the computational time and cost also.

VI. Numeric Methodology

The CFD code of STAR-CD for finite volume method has been used to solve the discretized continuity and Navier stokes equations. This CFD code is normally used for internal combustion engines and it has high potential for solving the transient, compressible, turbulent- reacting flows with, turbulent reacting with sprays on the finite element grids with moving boundaries and meshes. Complete hexahedral of intake and exhaust ports and combustion chamber are considered due to requirements of moving mesh. Valve and piston motion are carried out and supported by vertex motion routines. STAR-CD is competent enough to handle the complex geometry and enables the computational domain to include intake and exhaust ports, valves, valve seats and the combustion chamber with moving piston. Constant pressure boundary conditions carried out at both intake and exhaust ports and lateral walls of the valves are considered as the adiabatic condition. The constant temperature boundary conditions are allocated independently for the cylinder head, the cylinder wall and piston crown that outline walls of the combustion chamber. The temperature on each of these walls will be calculated numerically in the form of iteration for every step.

VII. Results and Discussion

Detailed numerical study carried out to investigate the effect of shape of piston crown to the fluid flow field characteristics for a 4-stroke CI engine under motoring conditions. Pressure vs. crank angle diagram is plotted. Velocity vector field is also investigated due to large scale mixing during intake and compression stroke in the cylinder for two-piston crown. In cylinder air motion is analyzed to predict the behavior of the fluid flow field characteristics between two different shapes of the piston. In general during intake and compression strokes of a internal combustion engine swirls and tumbles flows are always generated due to high turbulence in the cylinder. Swirl refers to a rotational flow within the cylinder about its axis and it is used to accelerate the combustion process quickly, whereas tumble ia a rotational motion about a circumferential axis near the edge of a clearance volume in the piston crown or in the cylinder head which is caused by squishing of the in-cylinder volume as the piston reaches the TDC. Finally, it is evident that the mixture stratification Is the most important factor in the case of in-cylinder injection of an engine hence it is necessary to examine the mixture formation process by changing the shape of the combustion chamber.

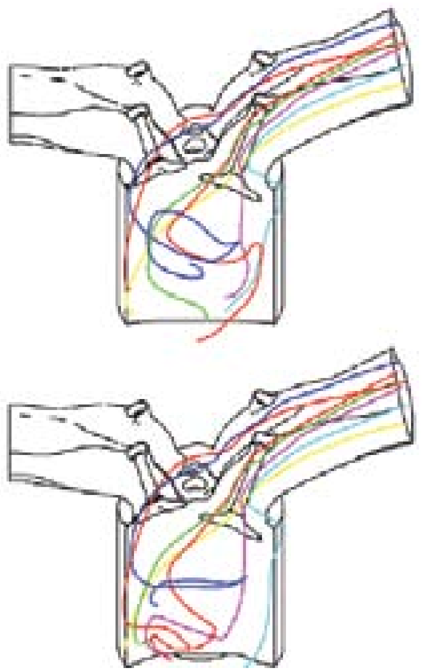

Fig. 1 The computed streamlines at 110 Deg a TDC showing Turbulence Structure as large scale motion

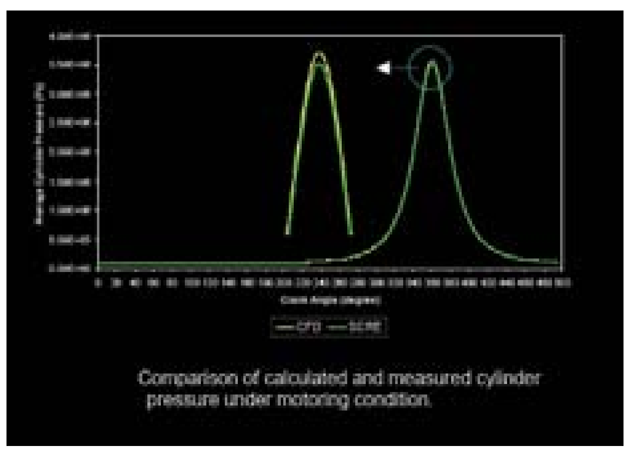

Fig. 2 Comparison of calculated and measured cylinder pressure under motoring condition

Fig.3 Mass Fraction distribution

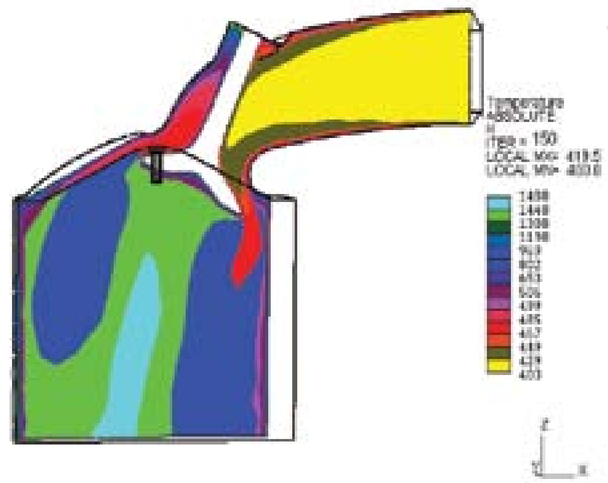

Fig. 4 Temperature Plot

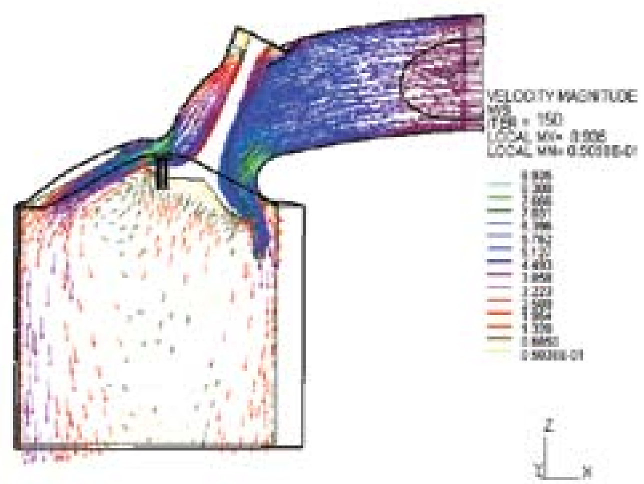

Fig. 5 Velocity Plot

Fig. 6 Generic Pattern of Velocity

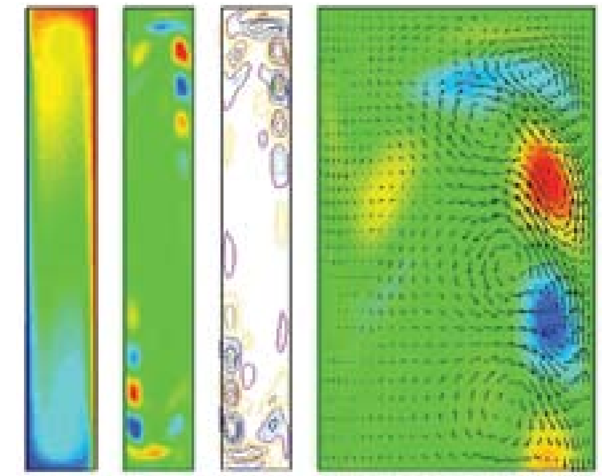

Fig 7 Conventional pattern of mixture fraction with eddies

VIII. Conclusion

In general, this investigation reveals that in-cylinder CFD simulations predicts a reasonable result that propagates improving the aspects of the I-c flow pattern and characteristics during intake and compression strokes instead of using conventional experimental techniques. The cylinder pressure of motoring condition has been validated against the experimental measurement and it shows that the developed engine model and its moving mesh have a capability to approach the condition of an engine. The clear cut understanding of air motion field inside the cylinder for a four stroke diesel engine is represented as an initial step, apart from that CFD can be used as an efficient design tool to develop other IC engine analysis to produce an optimum engine configuration. For further detailed investigation of these highly complex phenomenon of IC engines LES can be used to investigating the same. The investigation and analysis for the characteristics of air motion under motoring condition is numerically carried out for intake and compression stroke by using CFD code with the moving mesh and boundary capability. The predictions are made two shapes of piston crown to analyze the effect of the different shape of combustion chamber to the fluid flow field for the preparation of air-fuel mixture before the fluid injection starts. The CFD analysis for every crank angle thought intake and compression stroke is evaluated and behaviour of those characteristics for overall piston crowns are furnished.

The variation of each characteristic against the crank angle during intake and compression stroke has to be performed to reveal the effect of shape of the combustion chamber within the cylinder of an engine. The shape of the piston crown plays an important role to the developed large-scale fluid motion as the most significant component of mixing parameters during the complete process of engine by characterizing it with the swirl ratio, sideways and normal tumble ratio. In general LES has a good potential to predict the transient phenomenon in the case of flow of an internal combustion engines because of its ability to capture the instantaneous changes within the engine, because of numerical algorithm and hard ware resources the computational cost is seen in the higher side for using LES models. Comparison is finally made for two different geometries of piston crown for better combustion emission as well as performance characteristics.

References

[1] Preussner C., Doring C., Fehler S., and Kampmann S., " GDI: Interaction between mixture preparation, Combustion system and injection performance", *SAE paper* No.98498, 1998

[2] Lee B.S and Lee J.S., 2006 large eddy simulation of tumble and swirl formation in engine in-cylinder flow. *International Journal of automotive Technology,*7: 415-422.

[3] Heywood J.B, 1988 Internal combustion engine fundamentals McGraw-Hill, New York.

[4] Launder B.E, and D.B. Spalding, 1974. The numerical computation of turbulent flow. *Journal of computational methods and applied Mechanics Engineering,* 3:269-289.

[5] Versteeg H.K, and W.Malalasekare, 1995. An Introduction to computational Fluid Dynamics- The Finite Volume Method, Longman Group Ltd, London.

[6] Payri F., J. Benajes, X. Margot and A. Gil, 2004. CFD Modeling of the in cylinder flows in direct injection-diesel engines. *Journal of computational fluids,* 33:995-1021

[7] Suh E.S. and C.J. Rutland, 1999. Numerical study of fuel/air mixture preparation in a GDI Engine, *SAE Paper,* 1999-01-3657.

[8] Soltani S. and Veshagh .A, " CFD analysis of effect of Staggered Intake Valve Timing on Mixute Preparation and Combustion in a Four-Valve SI Engine", 1999 Spring Technical Conference ASME, Paper No. 99-ICE-169, 1999