Asian Review of Mechanical Engineering
ISSN: 2249-6289 (P) Vol.1 No.1, 2012, pp.35-41

DOI: https://doi.org/10.51983/arme-2012.1.1.2513

Fluid Dynamic Simulation Studies in a Four Stroke Compression Ignition Engine

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

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

[2]*Department of Mechanical Engineering, S.A. Enggineering 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 - Fluid Dynamic based models are always referred to "Multidimensional models", due to their ability to provide complete geometric information on the flow field, based on the solution of the governing equations. It is well known that over the last one decade, computers were extensively used for simulation in modeling purposes and building powerful, integrated-database systems. In general, computer based simulations have been built by the theoretical foundations and it is used in a wide range of applications. Engine flow simulation normally represents turbulence to predict various types of motions of turbulence like swirl, squish, tumble and eddies. Basically turbulent flow deals with the lateral and longitudinal motion, which is due to eddy motion. Eddy is a large group of fluid particles, which moves laterally, and longitudinally in the flow field. During this type of motion, it can change its shape or stretch and rotate or breaks into two or more eddies. Eddies are generally generated in the region of high shear in the mean flow field, near the boundary in a pipe or channel flow or in the vicinity of interface between two streams flowing at different velocities and parallel to one another. The size of large eddy will be the size of flow basically or the diameter of the pipe in which flow is analyzed. The eddies of different sizes are embedded in each other and it is impermanent in nature. The larger the eddied, which are continuously formed are breaks into smaller and smaller eddies, until they are dissipated through viscous shear finally. Large Eddy Simulation is a viable option for simulating the turbulent reacting processes that occur within the diesel environment and it has high potential to represent engines unsteadiness. LES has been developed to address the large-scale unsteady phenomenon and it is assumed for engine flow simulation with promising results, since it concerns the smaller part of spectrum. Normally, it is predicted that RANS can be used for preliminary design explorations, whereas LES can be used for detailed investigations. In this paper, engine flow simulation with LES is reviewed to represent the unsteady phenomenon in a diesel engine. In recent years, since the availability of high speed computing systems, the simulation techniques have become faster, easier and more accurate also. Therefore, the systems, which were upto now, not responsive for simulation techniques, have come under the purview of simulation studies. The modeling of the engine process continuous to develop as the understanding of physic sand chemistry of the phenomenon of interest steadily expands, and as the capability of the computers to solve complex equations, continues to increase.

Keywords: **Computational Fluid Dynamics, Simulation, Turbulence, Large Eddy, Combustion, Fuel Spray**

I. Introduction

A. Engine Flow Simulation

Diesel engines are characterized with more fuel economy and relatively low levels of undesirable emissions such as nitrous oxides, unburnt hydrocarbons and particulates. In order to predict the combustion, emission and performance characteristics it is necessary to analyze the engine flow simulation because of the in-cylinder flow structures.

The general characteristics of in-cylinder flow structures normally includes, turbulent flow with heat and mass transfer, unsteady as a result of the reciprocating motion of the piston and highly turbulent flows with a wide range of time and length scales, and flows which are subjected to "cycle-to-cycle variations" is also considered as a characteristics of an in-cylinder flow structures.

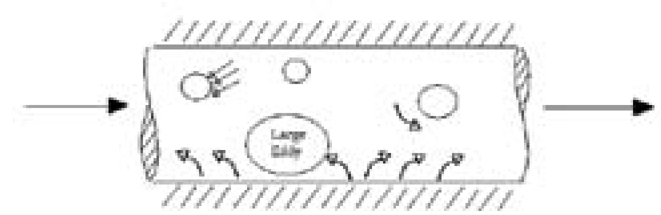
LES is a viable option for simulating the turbulent reacting processes that occur within the diesel environment. It has good potential to predict engines unsteadiness. It deals with improvement in productivity. It addresses the large-scale unsteady phenomena. It concerns with the smaller part of the spectrum. It is also close to engineering applications. LES has been developed and assessed for engine flow simulation with promising results and it has become practical and affordable. As far as this paper is concerned, a complete review of LES in diesel engine is carried out to represent the turbulence as an engine flow simulation.

B. Eddies and Sizes

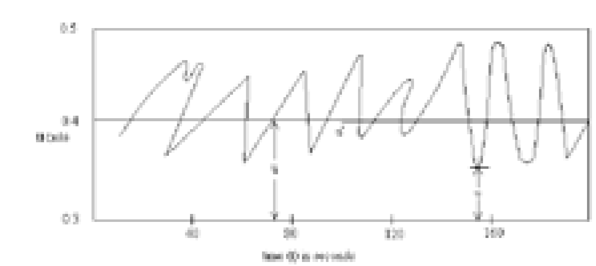
Eddy is generally considered as a large group of fluid particles, which moves laterally and longitudinally in the flow field. During this type of motion, it can change its shape or stretch and rotate or break into two or more eddies. Eddies are generated in the region of high shear in the mean flow field, near the boundary in a pipe or channel flow or in the vicinity of interface between two streams flowing at different velocities and parallel to one another. At high Reynolds number or at high shear, the size of large eddies

are governed by the size of flow and its geometry. Generally it is predicted that, at any time the flow contains eddies of various sizes. The largest eddy will be the size of the flow depth in a channel. The smallest eddy is normally seen in the order of millimeters. However the smaller eddy will be sufficiently large compared to the size of the molecules of fluid. The eddies of different sizes are embedded in each other and impermanent in nature, larger the eddies, which are continuously formed, breaks into smaller and smaller eddies, until they are finally dissipated through viscous shear.

The passage of the smallest, small, and largest eddies through a point in a flow field would induce velocity fluctuations of smaller magnitude large frequency and of large magnitude and small frequency. This would yield a fluctuating velocity field.



Instantaneous Eddy representation in a pipe



Velocity variation in quasi- steady state

u, v, w :- Instantaneous values
$\bar{u}, \bar{v}, \bar{w}$: - averaged values
u', v', w' :- fluctuating components

$$U = \bar{u} + u'$$
$$V = \bar{v} + v'$$
$$W = \bar{w} + w'$$
$$P = \bar{p} + p'$$

II. RANS Vs LES

LES and RANS techniques are viewed in such a way that they address the present impossibility to resolve all the scales present in engine flows and especially those related to turbulence, combustion and liquid jets. RANS simulations are based on a statistical averaging to solve only the mean flow. This implies that modeling concerns the whole spectrum of scales, which in turn makes the productivity of RANS simulation dependant on quality of the models used. The statistical averaging also extremely complicates addressing the unsteady phenomena. In LES, a spatial or temporal filtering is used to represent the large turbulent scales of the flow, which are directly resolved, while the small scales are modeled. In LES modeling therefore concerns a much smaller part of the spectrum, which leads to an improvement of productivity as compared to RANS. LES inherently allows to address large-scale unsteady phenomena, and therefore it has a good potential to predict engine unsteadiness. In LES and RANS, the effect of the modeled part of the turbulent spectrum on the resolved part is assumed to be "diffusive" and it is often taken into account by introducing a turbulent viscosity. The level of turbulent viscosity directly depends on the amount of modeled energy leading to high levels for RANS and far less important levels for LES. This explains the different requirements of LES and RANS in terms of numerical schemes. In RANS, the main requirement is to be robust and stable on distorted meshes. Numerical simulation is normally considered as a second-order aspect. Most of the RANS use upwind scheme, known to be very stable, but it is dissipative. Inversely in LES, vortices above and at the resolution limit must be accurately resolved, with as few numerical dissipation as possible. This implies the use of precise and energy-conserving numerical schemes, as Finite-Volume Centered-Differencing (FVCD) schemes. LES and RANS not only differ by the requests imposed on numeric, they also imply a different simulation procedure and exploitation of results.

A. LES Theory

Large eddy simulations involves a three-dimensional, time dependent corruption of the large-scale turbulent motions responsible for turbulent mixing. Which those with scales smaller than the computational-grid (the sub grid stresses) are modeled. The main difference between conventional turbulence modeling approaches and large eddy simulations is the " averaging " procedure used to derive the equation of motion. The LES techniques does not involve the use of ensemble averages, rather it consists in applying a spatial filter to the Navier-strokes equations. If Φ (x, t) is generic instantaneous flow variable, the corresponding filtered variable is defined as,

$$\Phi\,(t, x) = \int\int\int G(x-x', = \Delta\ \Phi(x,t)\, dx'$$

Where G (x, Δ) is a suitably defined function, characterized by a width Δ , which can D be in general a function of space and it is defined as $\Delta = (V_{cell})^{1/3}$. In the presence of variations of density, a density-weighted filtering operator is introduced, which shares the same properties as the unweighted one, i.e.

$$\langle\Phi\rangle = [\ \overline{pq}\ /\overline{p}]$$

Large Eddy simulation can be used to close the equations governing turbulent reacting flows. LES equations are basically different from RANS equations, in that spatial averaging are used in lieu of ensemble averaging. Accordingly, an LES allows unsteady flow to be resolved, while a RANS simulation averages out many transient phenomena. Another characteristic of LES is that, three-dimensional simulation must be performed, as turbulence is fundamentally three dimensional in nature. Conversely, RANS simulations arouse isotropy and can be run in two-dimensions as the geometry allows. Due to these requirements, large eddy simulations are typically more expensive computationally than RANS simulations. However, as computational resources increases, it is agreed that LES models may become more widely used.

STAR-CD offers a wide variety of turbulence modeling capabilities, such as eddy viscosity models, Reynolds stress models, large eddy simulation models, and detached eddy simulation models. It is generally recognized that all the existing turbulence models, are inexact representations of the physical phenomena of turbulence. The degree of approximation in a given model depends on the nature of the flow to which it is being applied, based on experience.

Commonly used filtering fluctuations include the box and Gaussian filters. The resolved components is directly determined by solving the filtered equations for mass, momentum, energy and scalars. The effect of the sub-grid on the resolved field must be modeled, i.e., determined as a function of known quantities. It is noticed that since the LES equations are spatially averaged rather than ensemble or time averaged, LES results fluctuated in time or ensemble averaged when compared to ensemble averaged experimental measurements.

Another advantage of LES models over RANS models is the "grid Sealing". Many sub-grid LES models contain a length scale in their formulation, commonly taken to be proportional to the grid size. Therefore, as grid density increases, the sub-grid components automatically tend to zero and valid DNS result will be obtained. In other words, the sub-grid LES models turns themselves off, when grid resolution is adequate to resolve all the length scales present in the flow. RANS models on the other hand, do not have this feature. Therefore when the flow field is well resolved with a fine mesh, the RANS simulation results are polluter by the model and therefore valid DNS results will not be obtained. However, one disadvantage result of the grid length scale being in the LES model is that a non-uniform grid, which can lead to erroneous results.

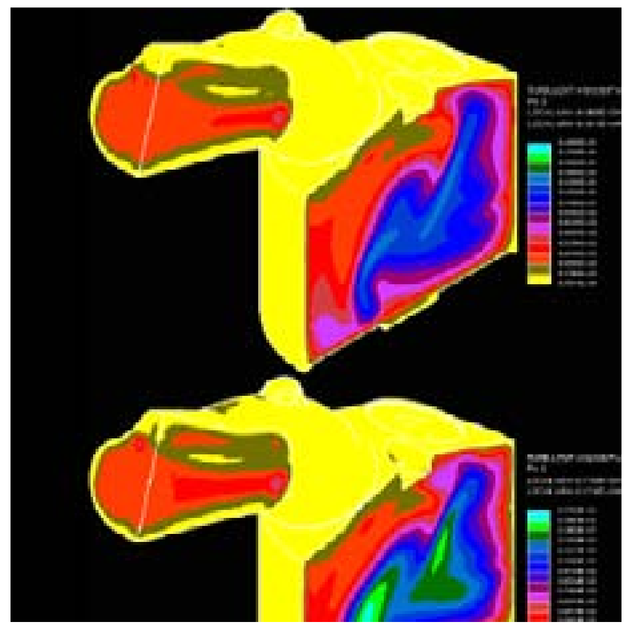


Fig. 1 Turbulence Kinematic Viscosity during Intake stroke at 95 deg aTDC

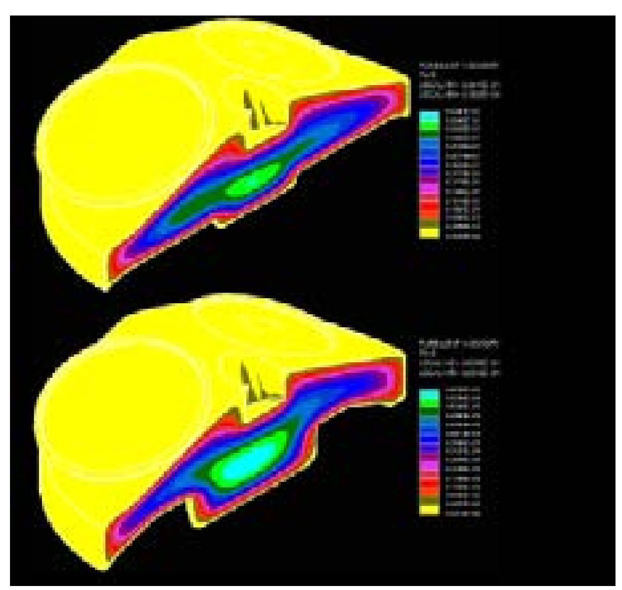


Fig. 2 Turbulence Kinematic viscosity during compression stroke(32 Deg b TDC)

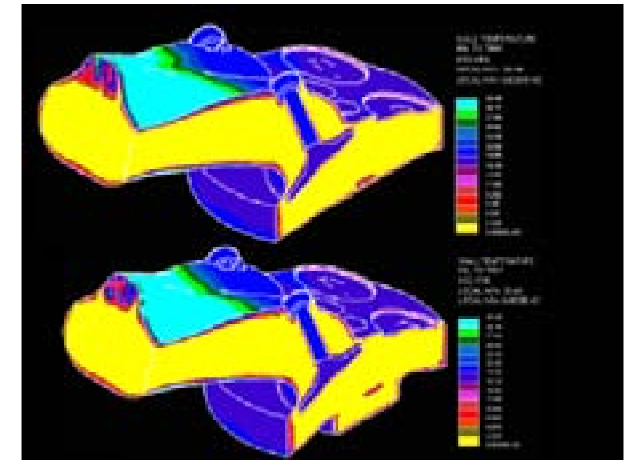


Fig. 3 Wall Temperature during early part of intake stroke

B. Combustion Modeling

Basically, ignition and combustion are represented as two separate processes in the current version of STAR-CD. Generally, there are two recommended models for diesel combustion such as Magnussen model and laminar-turbulent characteristics-time model.

Magnussen model is based on eddy-break-up concept, which relates the rate of combustion to the rate of dissipation of eddies and expresses the rate of reaction for the mean mass fraction of reacting spices, the turbulence kinetic energy and the rate of dissipation of this energy.

C. Droplet interactions

Along their trajectory the droplets undergo various interactions with the continuous and with each other. They exchange momentum with the background flow, with the interface forces computed by a dryer co-efficient. Star-CD has built in correlations for each of this transfer co-efficient or the users can code in their own functions through user sub routines.

If the background flow is turbulent a random velocity component is added to the droplet velocity. The droplets may increase in size by agglomeration due to collision, and decrease by breaking up (one drop of collision model and three different breakup models are built into STAR-CD). Thus, even an initially uniformed distribution of droplet properties quickly evolves into a distribution of properties.

Droplets exchange mass with the continuous phase by evaporation or condensation. There is also a boiling model available to deal with drop. Which reach the boiling or critical temperature. Droplets may consist of up to 10 components and each component can be associated with a scalar species in the continuous phase. Therefore multiple fuel droplets can be simulated very easily.

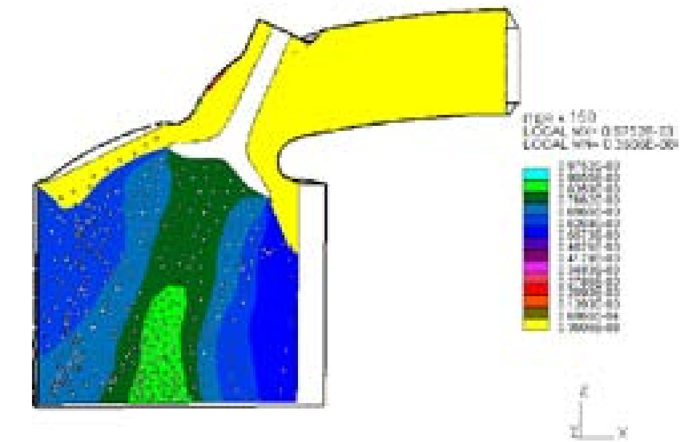


Fig. 4 Fuel Droplet Tracking

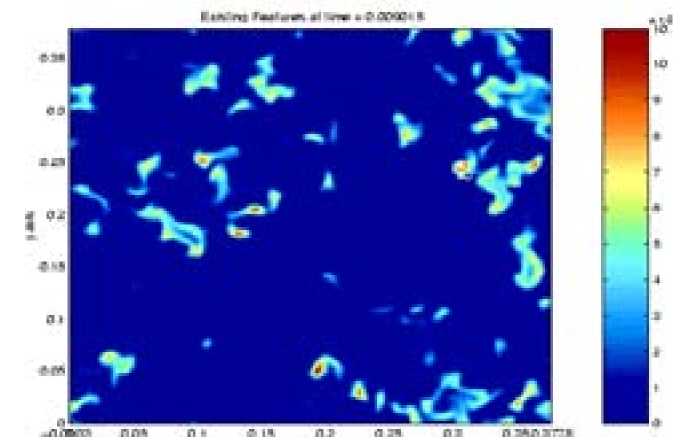


Fig. 5 Fuel Spray pattern

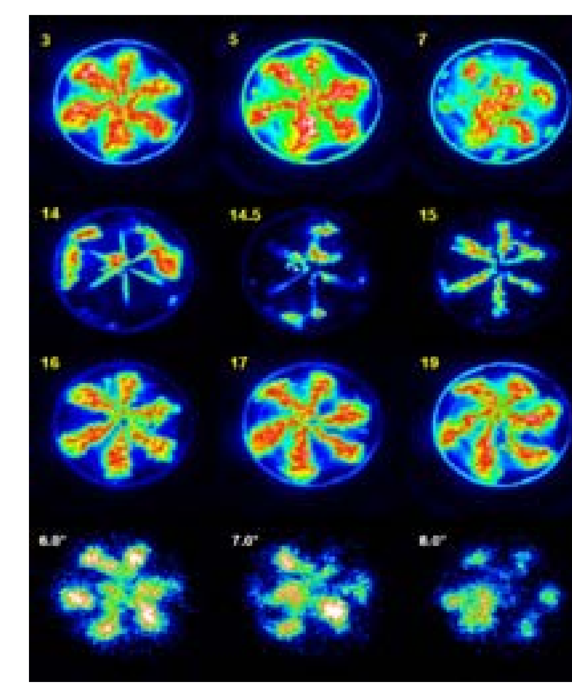


Fig. 6 Simulated Fuel Spray

III. Compression Stroke Turbulence

The fluid flow, heat transfer and turbulence characteristics during the compression stroke normally depends on the shape of the piston bowl on the characteristics of air structure pattern for a direct injection engine. CFD simulation for compression stroke is normally performed by using the moving mesh boundary algorithm. Every contained event represents the different mesh and boundary geometries for every different crank angle in each step of engine cycle. Hence, in order to perform the proper CFD simulations for an I.C. engine process, the analysis and calculations by using the unsteady conditions, moving mesh and boundaries, high compressible Reynolds number, high fluid dynamic characteristics or turbulence intensity, momentum, heat transfer, mass transfer, complex geometries model and chemical thermal dependent as well.

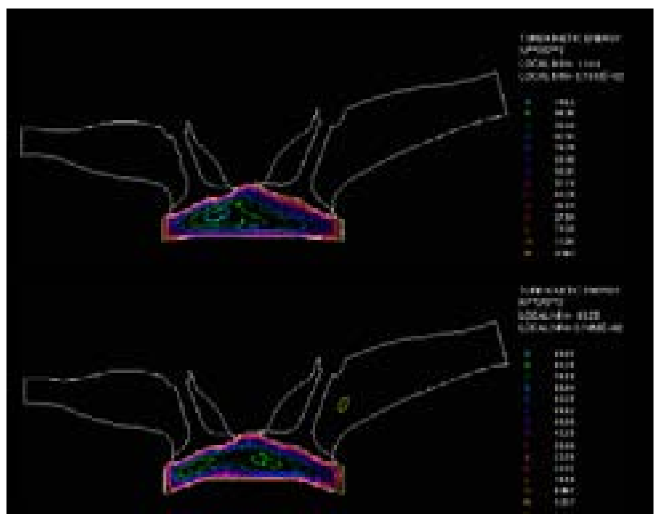


Fig. 7 Turbulence Kinetic energy during Compression stroke (32 Deg b TDC)

IV. Turbulence Models

Since turbulence directly affects spray, mixing and combustion in a an engine, adequate prediction of turbulence behavior is necessary for better understanding in order to improve engine performance and emissions. In most of the multidimensional computational codes developed so far many important characteristics of velocity length and time scales are directly related to the corresponding turbulence scales in models for the fuel spray combustion, heat transfer and so on. Therefore, whether these process are accurately modeled is also dependent on the accuracy of the turbulence modeling and prediction the typical turbulence parameter consist of turbulence viscosity, turbulent integral length scale and turbulence intensity. The turbulence kinematics viscosity term is defines as

Turbulence Viscosity =

$$\text{Co-efficent of turbulence viscosity (0.09)} \times \text{Density} \times \frac{\text{Turbulence Kinematic Energy}}{\text{Turbulence dissipation rate}}$$

$$\mu = F_{\mu} \times \rho \times (\text{T.K.E} / \text{T.D.R})$$

$$\text{Turbulence integral length scale} = \frac{F_{\mu}^{3/4}\,(\text{T.K.E})^{3/2}}{\text{K}.\varepsilon}$$

Where K= Vonkarman constant

$$\text{Turbulence intensity} = \sqrt{\frac{2 * \text{T.K.E}}{3}}$$

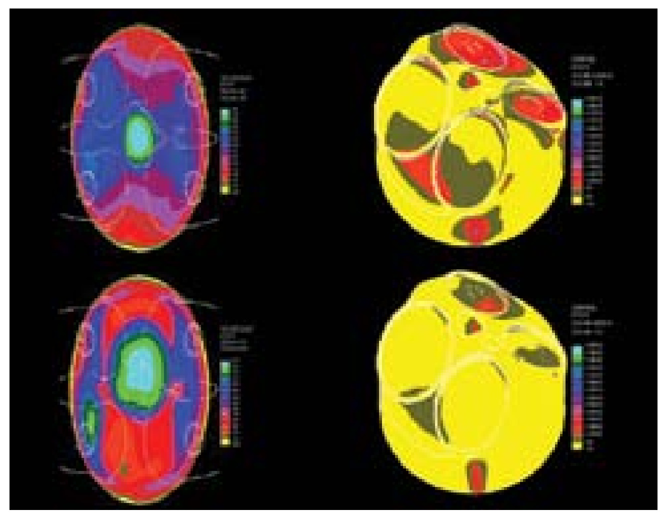


Fig 8 (a) Turbulence Kinetic energy at 15 mm below the cylinder head at 22 Deg bTDC. Fig 8 (b) Turbulence dissipation rate during Compression stroke at 32 Deg bTDC

V. Engine simulation results

These dimensional computations on engine geometry are able to capture the generic features of fluid flow in a reciprocating engine and are consistent with some of the experimental observations available in the literature. In the presence of high shear zones near to the edge of the bowl during the squish/reverse squish is evident more so in the case of the high compression ratio with the bowl-in-piston geometry. The standard k-ε model predicts trends in turbulence intensity around top dead center that are different from the experimental data available in the literature. This is thought to be due to the limitation of the k-ε model in capturing the unsteady effects during the squish/reverse squish period and therefore not attributable to the difference

in shape of the combustion shape geometry. These motored turbulence parameter, newly the turbulent intensity and length scale, have been used as input for zero dimensional thermo dynamic modeling by considering a simple rapid distortion crosses the account for reaction condition in earlier publications. However, at optimum ignition setting the predictions are in accurate. When the reverse squish effect becomes dominant it appears that the enhanced fluid momentum during the reverse squish period, significantly modifies the burn rate. However it might be possible to gain insight into some of these aspects more precisely by adapting computational techniques such as LES.

Turbulence in reciprocating engines is considered as important to cross of its large influence on combustion characteristics in order to find optimum conditions in a combustion systems is essential to obtain a good approximation of in cylinder turbulence from this view point multi dimensional modeling using CFD codes pursued an involved simulation with and without combustion. One of the major challenges of recent times has been that several models need to be considered in order to simulate a complete engine cycle. The k-ε model is the most commonly used turbulence model in CFD even though its difficulties are known. Indeed the k-ε model is still considered as the best compromise between computational time and precision. However in recent times second order closure models have been pursued in engine modeling with increased validity of the model. However complex numerical approach is such as DNS are difficult to implement in engine like situation because of enormous pre requisites high mesh density and exorbitant CPU time. This is also in the case of large eddy simulations, where accurate sub grid models are required where the research in this field is still progress. For implementation of CFD models mesh density assumes primary importance.

LES models allow for the turbulent transfer coefficients for both momentum and scalar flux to be determined independent of each other. The turbulent viscosity, μ_v is determined as a function of the sub grid turbulent kinetic energy, which is in turn determined from a one-equation model. The formulation for the scalar transfer co efficient, μ_s is similar to that of the turbulent viscosity, yet is made to be consistent with scalar transport. Results for the LES momentum transfer co efficient are compared to experiments for a backward facing step. This model in conjunction with the LES flux model is verified by comparing with experimental data for a non-reacting turbulent jet. Finally, these models are implemented in conjunction PDF combustion model to model the diesel combustion process. These results show the potential of using the LES models to model the turbulent reacting process that occurs with the diesel environment.

It is predicted that RANS can be used for initial design exploration whereas LES can be used for engine turbulence analysis in detail.

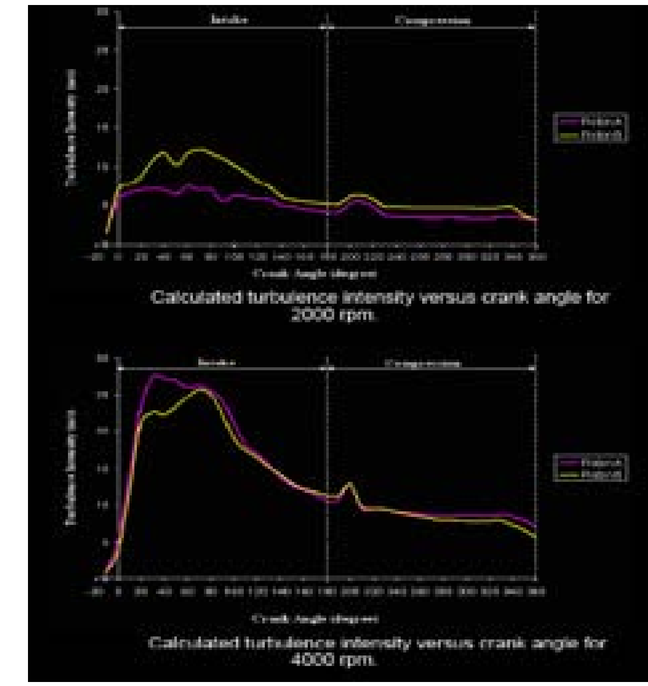


Fig. 10 Calculated turbulence intensity versus crank angle for 4000 rpm

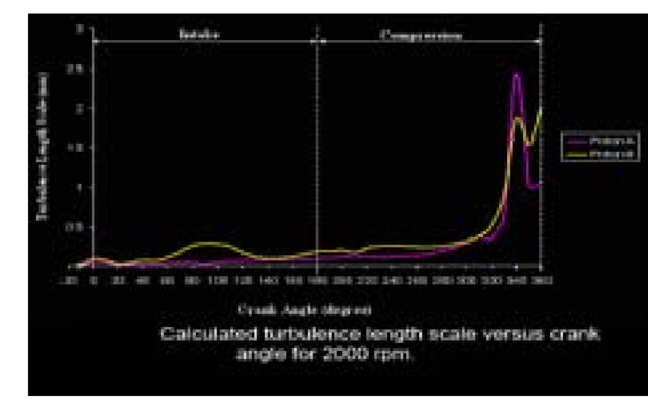


Fig. 11 Calculated turbulence length scale versus crank angle for 2000 rpm

VI. Conclusion

In this paper, turbulence analysis is completely predicted to represent turbulence through large eddy simulation in a four-stroke compression ignition engine in detail. It is predicted that large eddy simulation is a viable option for simulating the turbulent reacting process that occur within the diesel environment and it has high potential to represent the engine unsteadiness. LES has been developed to address the large scale unsteady phenomena and it is assessed for engine flow simulation with best results, since it concerns the smaller part of the spectrum. Therefore an engine flow simulation with LES is reviewed and analyzed to represent the unsteady phenomena in a diesel engine. The complete turbulence analysis for compression stroke, combustion and fuel spray in a diesel engine is analyzed. In order to predict the combustion, emission and performance characteristics, it is necessary to analyze the engine flow simulation because of the in-cylinder-flow structures. Since the basic turbulence flow deals with lateral and longitudinal motion due to eddies it is necessary to predict the engine turbulence by using large eddy simulation. In recent year since the availability of high speed computing systems the simulation techniques have become faster, easier and more accurate also. The modeling of the engine process continues to develop as the understanding of physics and chemistry if the phenomena of interest steadily expand and as the capability of computers to solve the complex equations continues to increase.

The development of CFD methodology for I.C.Engine design represents a particular challenge due to the complex physics and mechanics, perhaps more than with any other widely used mechanical devise. Improved understanding is essential to explore new solutions, reduced cost and improve efficiency. Although substantial advances have been made in most of the areas there are numerous additional requirements to be met for it to become a design tool. Moreover it is necessary to predict the turbulence in engines, because in most of the times it is essential to make sure that how far the air is well contained in the fuel, to improve the overall performance through complete combustion.

References

[1] J.M.Duclos, M.Zolver,T.Baritand, " 3D modeling of combustion for DI-SI Engines", *Oil gas science and technology,* Vol. 54, 1999.

[2] D. Adloph, R. Rezaei, S. Pishinger, P. Adomeit, T. Korter,A. Kolbeck, M. Lamping, D. Tatur, and D. Tomazic, " Gas exchange optimization and the efforts on emission reduction for HSDI diesel engines", *SAE Paper,* pp.1-653, 2009.

[3] D. Adomeit, S. Pischinger, M. Becker, H. Rohs, A. Greis, and G. Grunefeld, "Potential soot and co reduction for HSDI diesel combustion systems", *SAE paper*, pp.1-1417, 2006.

[4] Z. Han and R.D. Ritz, "Turbulence modeling of internal combustion Engines using RNG K-1 model", *Combution Sci. tech,*Vol.105, pp. 267-295, 1995.