CFD Studies of Combustion in Diesel Engine

S Gavudhama Karunanidhi, Melvinraj C R, Sarath Das K P, G Subba Rao
Assoc. Professor, Department of Mechanical Engg MES College of Engg Kuttipuram, Kerala
PG Scholar, MES College of Engineering, Kuttippuram,Kerala, Calicut University
PG Scholar, MES College of Engineering, Kuttippuram,Kerala, Calicut University
Principal, Geethanjali Institute of Science and Technology, Nellore.

ABSTRACT

Increasing computational power of modern computers, multi-dimensional Computational Fluid Dynamics (CFD) has found more and more applications in diesel engine research, design and development . Various successful applications have proven the reliability of using multi-dimensional CFD tools to assist in diesel engine research, design and development. By using CFD tools effectively it is easy to predict and analyse various details that are technically difficult like in cylinder process of diesel combustion, temperature & pressure contours, emission etc. prior to experimental tests to reduce the number of investigated parameters as well as time and thus costs. A multidimensional model was created and analysis of combustion was done using FLUENT, ANSYS 14.5 package and the 2D geometry was modelled and meshed using GAMBIT 2.4.6.

Keywords**:** CFD, CO_2 emission, Combustion modelling, Diesel combustion, k-ε model

I. INTRODUCTION

Reduced fuel consumption and improved efficiency are always the primary area of consideration in internal combustion engine design sector. They were generally done by building prototypes and testing them. But this traditional process always had several limitations. They were time consuming, cost consuming and also a prototype made for a single purpose didn't had the complete versatility of using the same for other purpose. This difficulty can be overcome by using CFD studies [1]. With the increasing advancement in computational power of modern computers, CFD has found its application in diesel combustion. This is now widely used by many automobile industries not only for design and analysis of engine but also for the whole vehicle analysis. Of the many types of models for engine combustion process, multidimensional computational fluid dynamics (CFD) models is gaining momentum due to its capability to predict the gas flow patterns, fuel spray structure etc[2].

II. COMPUTATIONAL PROCEDURE

The combustion simulation of compression ignition engine was developed using Fluent software (ANSYS 14.5 package) and the various equations of the multi-dimensional model were solved by the software automatically . The main inputs include engine speed, injection details, bore, stroke, connecting rod length, initial pressure and temperature. The program concerning the simulation model predicts the cylinder pressure, cylinder temperature, heat release rate, emission etc. The results including graphs and various contours (temperature , pressure etc) were generated by Fluent software as outputs to the program for given inputs.

III. MODEL DEFINITION AND MESHING

A 2D cylinder geometry with centrally located injector was considered .The mesh was created using GAMBIT 2.4.6.The engine geometry details and specification details are given below. Mesh generation plays an important role in obtaining accurate results. A quadrilateral mesh was created uniformly throughout the area and analysed using FLUENT, ANSYS 14.5 package. The complete meshed geometry contains 10600 cell, 21403 faces and 10804 nodes. Fig.1 shows the meshed geometry.

Connecting rod length	:140 mm
Bore	:80 mm
Crank radius	:55 mm
Crank shaft speed	:1500 rpm

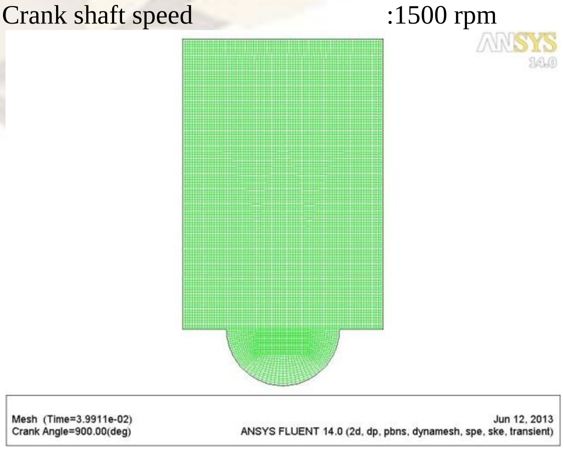

Fig.1 Meshed geometry of cylinder

IV. GOVERNING EQUATIONS AND MODEL

The basic approach of in-cylinder diesel combustion models are typically compressible turbulent flow. Apart from the complexity of turbulent model, the high pressure spray and resulting spray penetration , evaporation, and involvement of multiphase , multi-component nature only increases its complexity. Even then the nature of fluid is still governed by the basic equations including continuity (mass conservation), momentum (Navier-Stokes equation), energy and turbulence (k-ε model) equations. Of the three combustion modelling (thermodynamic, multidimensional and phenomenological modelling) the software use multidimensional modelling, i.e. CFD modelling[1]

V. COMBUSTION SIMULATION

In engine environments, the combustible mixture is subject to a turbulent flow and, once mixed, undergoes subsequent elementary reactions which convert the fuel vapour to complete and incomplete combustion products with the accompanying release heat. So, in diesel engine simulations, combustion modelling mainly deals with two processes: first, low-temperature chemistry, which leads to auto ignition and produces intermediate species, and second, these intermediate species trigger high-temperature reactions that contribute the main heat release, as well as further complete and incomplete combustion products. Another important task for combustion models is to properly account for the significance of the effect of turbulence on the combustion processes. The combustion of diesel fuel was considered which is simulated using FLUENT, ANSYS 14.5 package. A single cylinder, single zone, multi-dimensional model was considered in analysis process. The mixture material, diesel-air, properties were copied from the Fluent material database for combustion. Using species transport model of fluent the analysis is done. While using this model conservation equation for chemical species , ANSYS FLUENT predicts the local mass fraction of each species through the solution of convection diffusion equation for the i^{th} species . This conservation equation has the form:

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla.(\rho \vec{v} Y_i) = -\nabla.\vec{J_i} + R_i + S_i \qquad (1)$$

where R_i is the net rate of production of species i by chemical reaction. S_i is the rate of creation by addition from the dispersed phase plus any user-defined sources. An equation of this form will be solved for N−1 species where N is the total number of fluid phase chemical species present in the system. Since the mass fraction of the species must sum to unity, the Nth mass fraction is determined as one minus the sum of the N − 1 solved mass fractions. To minimize numerical error, the Nth species should be selected as that species with the overall largest mass fraction, such as N_2 when the oxidizer is air.[3]

VI. COMPUTATIONAL METHODOLOGY AND BOUNDARY CONDITIONS

After modelling the geometry, for analysis viscous standard k-e model is enabled for considering eddy dissipation. Since geometry is subjected to motion of piston, dynamic meshing is enabled subjected to suitable boundary condition for piston, cylinder , walls etc. Combustion in a diesel engine involves the transient injection of finely atomized liquid fuel into the air at high temperature and pressure[1]. The fuel injection parameters like location of the injector, size of the injector, injection temperature and pressure, mass flow rate etc are having significant effect in diesel combustion modelling. The injection parameters and specifications are given below

X-position	:113.68 mm
Y- velocity	:468 m/s
Diameter	:0.286 mm
Temperature	:341 K
Flow rate	:0.001044 kg/S
Start crank angle	:710 deg
Stop crank angle	:725 deg

VII. RESULTS AND DISCUSSIONS

1. Cylinder pressure and Cylinder temperature

The modelled cylinder pressure data is shown in fig.2. From the graph the peak pressure generated by the software was approximately equal to 30 bar which is in agreement with the theoretical value . Also the maximum cycle temperature obtained was near 1700K(Fig.3). The static pressure and static temperature contours at the end of expansion was also generated by the software (shown in fig.4. and fig.5.)

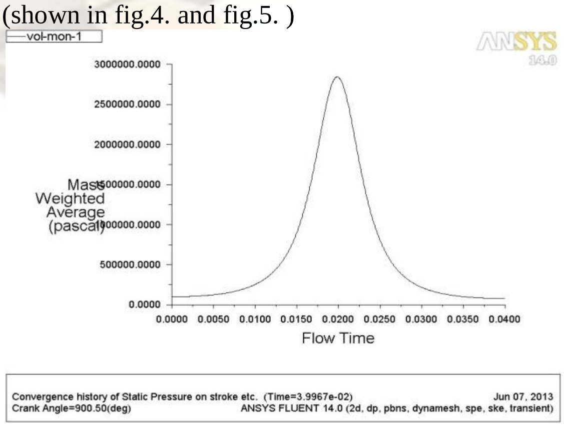

Fig. 2. Pressure v/s Flow time

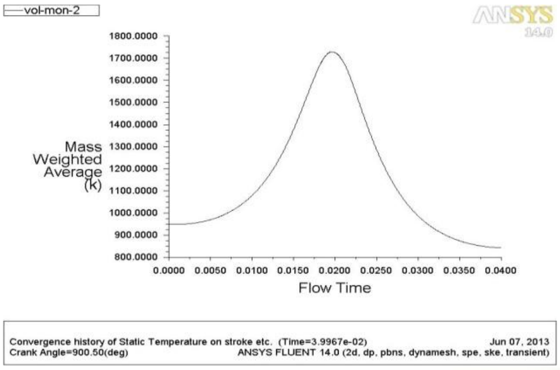

Fig.3. Temperature v/s Flow Time

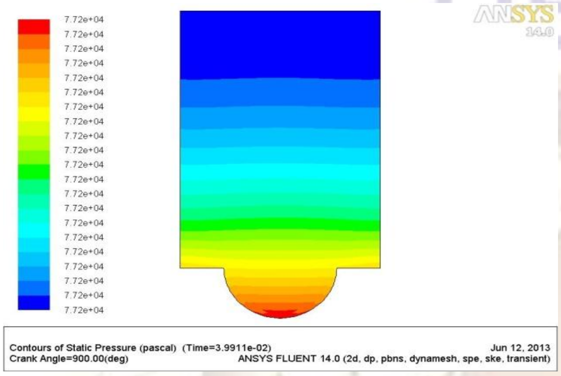

Fig.4. Contours of static pressure

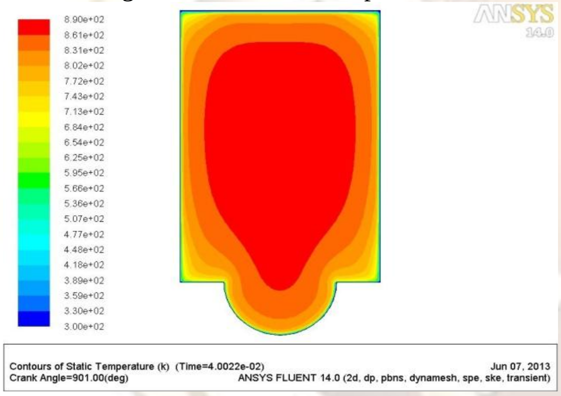

Fig.5. Contours of static temperature

2 CO_2 Emission

The mass fraction of CO_2 was generated by the simulating software and was found to be near 0.0048 at the end of expansion. (fig.6).

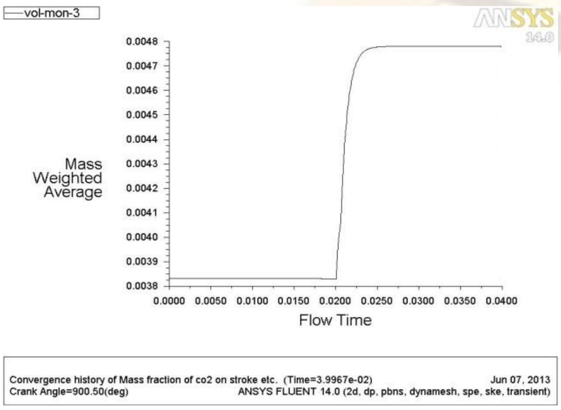

Fig.6. Mass fraction of CO_2

VIII. CONCLUSION

The model was created using Gambit , and the combustion phenomenon was analysed using Fluent Ansys 14.5. The results shows values of pressure and temperature comparable to theoretical values. Hence the developed model is suitable for predicting the combustion characteristics of a compressed ignition engine. By changing the injection timing, the peak pressure, temperature etc can be varied and can be used to find the optimal injection timing as well as the combustion

REFERENCE

[1] Lakshminarayanan, P. A. et. al, *Modelling diesel combustion (Springer, 2009)*

[2] Dr. Ibtihal Al-Namie et. al., *" Study the Effect of Ceramic Coating on the Performance and Emissions of Diesel Engine", Journal of Engineering , August 2012, Volume 18, Number 8,935-942.*

[3] Krishnan, D.B. et. al., *" Performance of an Al-Si graphite particle composite piston in a diesel engine", Transactions of Wear, 1980,60(2), 205-215.*

[4] Murat Ciniviz et.al., *" Ceramic Coating Applications and Research Fields for Internal Combustion Engines", Ceramic Coatings - Applications in Engineering, Feb. 2012,195-234.*

[5] Alkidas, A.C.,*" Performance and emissions achievements with an uncooled heavy duty, single cylinder diesel engine", SAE, vol. 890141, 1989.*

[6] Uzun, A., Cevik, I. and Akcil, M, *"Effects of thermal barrier coating material on a turbocharged diesel engine performance", Surf. Coat. Technology,1999, 116–119.*

[7] Spaniel, M., Macek, J., Divis, M. and Tichanek, R., *''Diesel engine head steady state analysis'', International Journal of Middle European Construction and Design of Cars, 2003,2(3), 34–41.*

[8] M. Fadaei a, et. al., *"New thermo-mechanical analysis of cylinder heads using a multi-field approach", Scientia Iranica, November 2010, 18 (1), 66–74.*

[9] Parlak, A.," *Experimental Investigation of Injection Advance and Compression Ratio of a Supercharged Ceramic Coated Diesel Engine", PhD Thesis, Sakarya University, Turkey (in Turkish), 2000.*

[10] Gataowski, J.A., *'Evaluation of a Selectively Cooled Single Cylinder 0.5L Diesel Engine", SAE, Paper 900693,USA, 1990.*

[11] Afify, E.M. & Klett, D.E. (1996), *"The Effect Of Selective Insulation On The Performance, Combustion, And NO_x Emissions Of A DI Diesel Engine", International & Congress and Exposition, Detroit, Michigan February 26-29.*

[12] Rasihhan, Y. & Wallce F.J., Piston-Liner "*Thermal Resistance Model for Diesel Engine Simultion", Proc. Inst. Mech. Engrs., 1991, Vol.205.*

[13] J.Blazek,*" Computational Fluid Dynamics: Principles and Applications", Elsevier Science Ltd,2001,1-39*

[14] Büyükkaya, E.; Yaşar, H.; Çelik, V. & Ekmekci, M.,*" Effects of Thermal Barrier Coating to Exhaust Emissions of a Turbocharged Diesel Engine", International Combustion Symposium, Bursa, Turkey, 21-23 July 1997 , 21-23.*