

Modeling of In-Cylinder Fuel Sprays Combustion in D.I. Diesel Engine

Bhanu Priya K¹, Hema Chandra Reddy K², Balaji M³Research Scholar, Department of Mechanical Engineering, JNTU Anantapur, AP, India¹Professor, Department of Mechanical Engineering, JNTU Anantapuramu, AP, India²Research Scholar, Department of Mechanical Engineering, JNTU Anantapur, AP, India³

ABSTRACT: D.I. Diesel engines are the commercially used vehicles in daily practice. The performance of D.I. Diesel engine largely depends on the combustion dynamics inside the cylinder. Hence, such combustion is influenced by the spray characteristics, fuel content and then the motion of the piston. The main problem with diesel engines is emissions of nitrogen oxides (NO_x) and particulates. In order to minimize the emissions, it is necessary to design the diesel engine with better in-cylinder flow (air-fuel mixing) and combustion process. Computational Fluid Dynamics (CFD) simulation helps to understand the Diesel engine temperature distribution and NO_x species concentrations with respect to time. A small direct injection (DI) engine was chosen for the study. CFD simulation results were compared with that of engine emission tests. This paper also presents the simulation results of direct injection diesel engine in-cylinder flow (air fuel mixing) and combustion.

KEYWORDS: CFD, Diesel engine, combustion modelling, Turbulent In-cylinder Flow Modeling

I. INTRODUCTION

Direct - injection diesel engines are used both in heavy duty applications and light duty ones due to their high thermal efficiency and low CO₂ emissions [1]. Fuel economy and emission norms are compelling to develop more efficient and cleaner engines. It is necessary to improve the combustion process to reduce exhaust emissions. Computational Fluid Dynamics is popularly used in different stages of engine design and optimization. The combustion system performance can be better understood using CFD simulation. The challenge in using CFD is the complexity of interaction of flow, turbulence, spray and combustion inside the IC engine cylinder. High pressure injection [2-5] and modification of combustion chamber geometry [6-9] were tried to reduce particulate emissions. Exhaust gas recirculation (EGR) [10] is another technique used to reduce NO_x emissions. Supercharging coupled with better injection timing reduces PM emission and NO_x reduction [11]. These methods however, have a trade-off relationship between NO_x and particulate emissions. To optimize the combustion and emissions in diesel engine it is necessary to understand the flow inside the cylinder.

This paper presents the CFD simulation of a diesel engine combustion process to predict the temperature distribution inside the cylinder and NO_x emission with reference to crank angle variation in the expansion stroke.

II. RELATED WORK

Aita et 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- ϵ 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

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 1, January 2017

with the experiments. Celik et al. [4] made a review of computations based on large eddy simulation. 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.

III. ENGINE PARAMETERS

The specifications of the engine are given in Table I. The physical modeling of the injector was not used since preset models of fuel injectors were available in the software used for analysis. The motion of valves also cannot be included in the simulation.

Table I. ENGINE SPECIFICATIONS

| | |
|---------------------------------|------|
| Model No | 5612 |
| Bore (mm) | 82 |
| Stroke (mm) | 68 |
| Displacement (cm ³) | 359 |
| Compression ratio | 18:1 |
| RPM | 3600 |
| HP | 7.5 |

IV. COMPUTATIONAL DOMAIN

The 3-Dimensional model shown in figure 1 is used for the analysis. The simulated engine has a direct injected diesel one with 4 - hole injector and eccentric bowl combustion chamber. For numerical integration the structured (hexahedral) mesh is used. During the solution, as the piston moves, the internal mesh structure deforms automatically to minimize the distortion of each individual cell. Figure 1 and 2 shows the geometry and mesh distribution when piston at a bottom dead center (BDC) and the injection parameters are given in Table II.

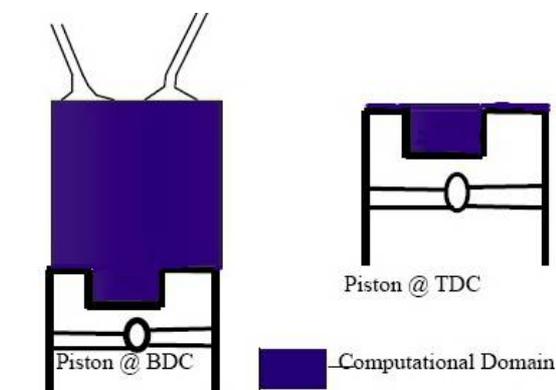


Fig. 1. Computational model of the Combustion chamber

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 1, January 2017

The starting Crank angle when piston @ BDC is 180° . For each stroke corresponding crank angle is 180° , total for 4 stroke 720° CA). Injection starts at 345° CA means, when piston moves from BDC to TDC, the fuel injection starts 15° CA before TDC. Total injection duration 23° CA means fuel injection starts 15° CA before TDC and ends with 7° after TDC).

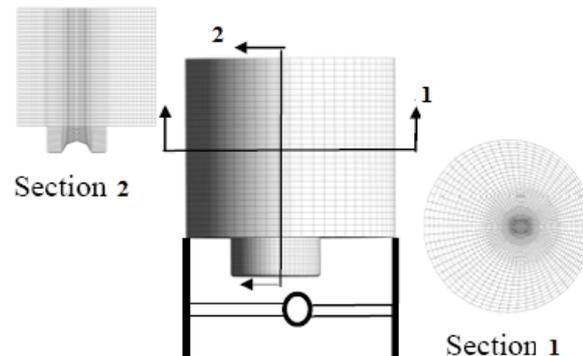


Fig 2. Computational mesh of the Combustion chamber

Table II INJECTION PARAMETERS

| Type | Direct Injection with single central injector |
|---------------------------|---|
| Number of nozzles | 4 |
| Total flow rate (g/cycle) | 0.05 |
| Start time (CA)(deg) | 342 |
| Stop time (CA)(deg) | 365 |
| Mean diameter (m) | 1×10^{-6} |

The numerical simulations in this investigation are performed based on the commercial computational fluid dynamics software. The various models used in the analysis to represent complex processes are explained below.

A. TURBULENT IN-CYLINDER FLOW MODELING

In a DI Diesel engine, the fuel is sprayed directly into the cylinder through the fuel injection nozzle. The fuel is then broken into a number of droplets. These droplets undergo collision and coalescence processes, exchanging momentum and energy with the high temperature and pressure surrounding gases inside the cylinder. Finally the droplets evaporate into vapor and mix with air. The equations governing the in-cylinder flow are species transport equations, continuity equation, momentum equation, energy equation, gas state equation and turbulence equations.

As the spray is the dominant feature of the flow with in a diesel engine combustion chamber, accurately resolving the spray atomization is a prerequisite for diesel engine simulation. Two mechanisms used for breakup of spray are Kelvin-Helmholtz (K-H) inertia instability and Rayleigh- Taylor (R-T) accelerative instability mechanisms. It is assumed that the mechanism responsible for the breakup of the spray is classical fluid dynamic instabilities which act at the interface between two fluids of different densities. For both of the instabilities, the time that the spray droplet breaks up is determined from the growth rate of the fastest growing wavelength predicted by the classical instabilities. For the K-H mechanism, the wavelength of the fastest growing wave is given by Reitz [12]. The R-T break up mechanism [14] is employed to account for the effect of rapid deceleration of the drops on the atomization process. The K-H and R-T mechanisms compete to break up the spray. More details of break up mechanisms are described in Ref.[12] and [14].

B. COMBUSTION MODEL

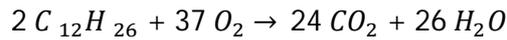
The combustion inside the cylinder is modeled by four kinetic reactions and six equilibrium reactions. The four kinetic reactions include those three elementary reactions for thermal NO and a forward reaction representing the oxidation of the fuel, which is considered to be consisting of $C_{12}H_{26}$ only. The fuel oxidation reaction is given below.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 1, January 2017



C. INITIAL AND BOUNDARY CONDITIONS

The computation started from the end of intake stroke at the bottom dead center corresponding to 180°Crank angle and terminated at the end of expansion stroke corresponding to 540°CA. The pressure and temperature of the air inside the chamber on the starting of the analysis were set as 100kPa and 350K based on the values given by Bedford [17] and Strauss [18]. No slip conditions were used on the walls. Standard wall functions were used for turbulence models. The amount of fuel injected was 0.05 g/cycle. The normal operating parameters were:
Injection duration: 23°CA; Injection start timing: 345°CA

V. RESULTS AND DISCUSSION

Temperature Distribution at various crank angles:

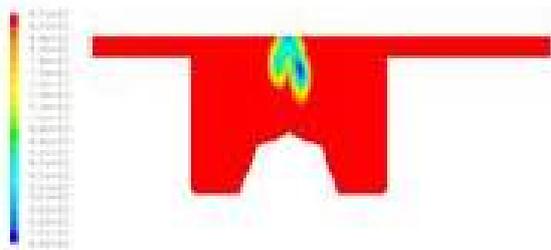


Fig. 3.1 Crank Angle: 350deg

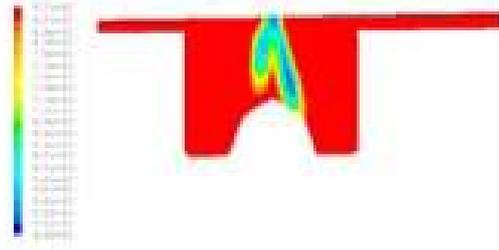


Fig. 3.2 Crank Angle: 355deg



Fig. 3.3 Crank Angle: 360deg

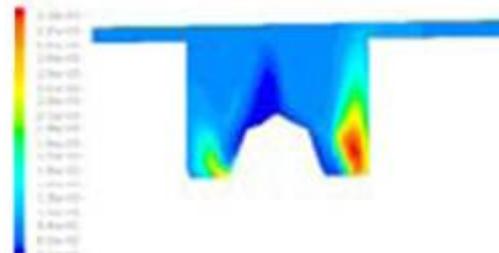


Fig. 3.4 Crank Angle: 365deg

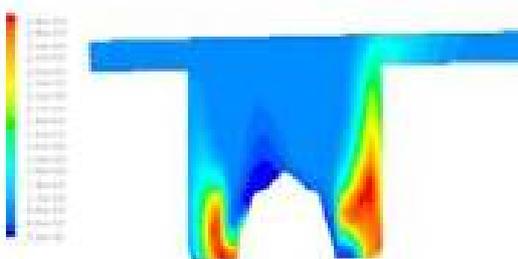


Fig. 3.5 Crank Angle: 370deg

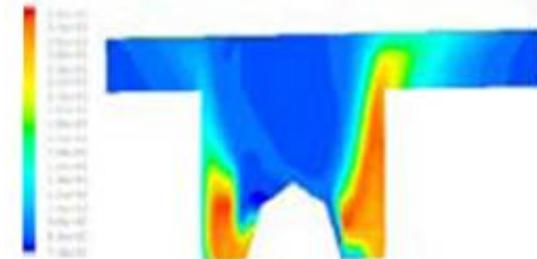


Fig. 3.6 Crank Angle: 385deg

Figure 3(1) to 3(6) presents the in-cylinder temperature distributions from the CFD simulation of base case design. The temperature distributions plotted for different crank angles on a vertical plane. Figure 3(1) show the maximum temperature of

around 950K which is attained from compression stroke. From 343 – 720 crank angle the temperature distribution plotted in Figure 6. Figure 3 (3) shows the initial flame occurs just before compression stroke. The spray and flame reaches the edge of the piston bowl with in short period 7-10 CA deg. The concentration of oxygen in the high temperature zone is very high resulting in a high NO_x rate of formation. Figure 4 and 5 shows the NO formation and Soot formation inside the cylinder.

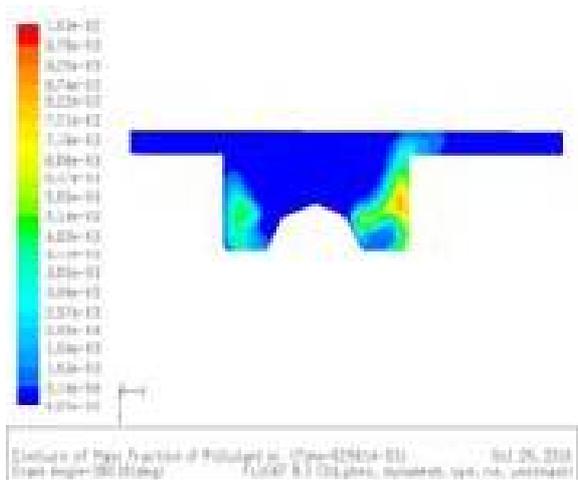


Fig 4 NO formation

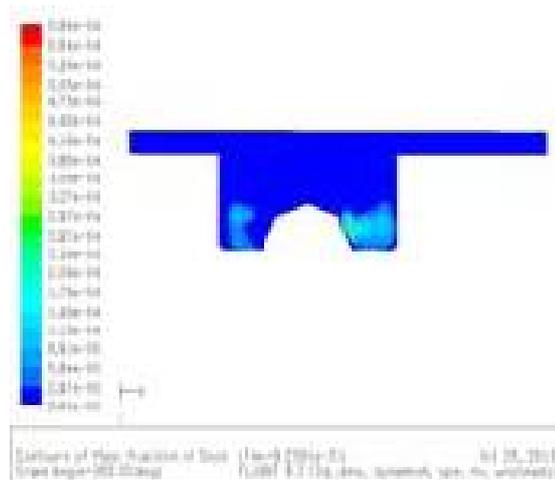


Fig.5 Soot formation

VI. CONCLUSION

In-cylinder physical variations of a single cylinder diesel engine have been investigated using a CFD simulation code and validated with experimental measurements. For reducing NO_x emission it is no doubt that a precise estimation of flame temperature and a good description of reaction chemistry are essential. The detailed time history of spray, fuel mass fraction and temperature distributions provided by the CFD simulation is valuable towards gaining a better understanding of the feature of combustion. Since the adopted methodology for the base case has given a high confidence level for further optimization geometry (bowl shapes) and operating parameters (Injection duration, Start of Injection timings) can be carried out for reducing NO_x emissions through CFD.

REFERENCES

- [1] Y. Kidoguchi, M. Sanda and K. Miwa, "Experimental and theoretical optimization of combustion chamber and fuel distribution for the Low Emission Direct – Injection Diesel Engines," Engg for Gas turbines & Power, Vol.125/351, 2003
- [2] S. Shundoh, T. Kakegawa, K. Tsujimura, and S. Kobayashi, 1991, "The Effect of Injection Parameters and Swirl on Diesel Combustion with High Pressure Fuel Injection," SAE Paper No. 910489.
- [3] 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.
- [4] 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]
- [5] I.D. Middlemiss, 1978, "Characteristics of the Perkins `Squish Lip' Direct Injection Combustion System," SAE Paper No. 780113.
- [6] Y. Daisho, 1986, "Effects of Combustion Chamber Geometry in a Direct-Injection Diesel Engine," Trans. Jpn. Soc. Mech. Eng., Ser. B 52(479), pp. 2768–2773.
- [7] M. Konno, 1991, "Reduction of Smoke and NO_x Emissions by Active Turbulence Generated in the Late Combustion Stage in Diesel Engines — 2nd Report," Trans. Jpn. Soc. Mech. Eng., Ser. B 57(534), pp. 773–777.
- [8] L. Zhang, 1996, "Effect of Chamber Geometry on Flame Behavior in a DI Diesel Engine," Trans. Jpn. Soc. Mech. Eng., Ser. B 62(600), pp. 3213–3219.
- [9] K. Sakata, 1990, "Development of Toyota Reflex Burn (TRB) System in DI Diesel Engine," SAE Paper No. 900658.
- [10] R.S.G. Baert, D.E. Beckman and R.P. Verbeek, 1996, "New EGR Technology retains HO Diesel Economy with 21st Century Emissions," SAE paper No. 960848.
- [11] A.P. Kleemann, A.D. Gosman, and K.B. Binder, "HeatTransfer in Diesel Engines: A CFD Evaluation study,"

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 1, January 2017

International Symposium on Diagnostics and Modeling of Combustion in Internal Combustion Engines (COMODIA 2001).

[12] R.D. Reitz, "Modelling Atomization Processes in High- Pressure Vaporizing Sprays," *Atomization and Spray Technology*, 3, 309-337, 1987.

[13] T.F. Su, M. Patterson, R.D. Reitz and P.V. Farrel, "Experimental and Numerical Studies of High Pressure Multiple Injection Sprays," SAE 960861, 1996.

[14] M. Patterson and R.D. Reitz, "Modeling the Effects of Fuel Spray Characteristics on Diesel Engine Combustion Emissions," SAE 980131, 1998.

[15] J. Warnatz, U. Mass and R.W. Dibble, 1996, *Combustion: Physical and Chemical Fundamentals, Modeling and Simulation, Experiments, Pollutant Formation*, Springer, Berlin.

[16] R.K. Hanson and S. Salimian, 1984, *Survey of Rate Constants in H/N/O Systems*, Combustion Chemistry, Springer-Verlag, New York.

[17] F. Bedford, C. Rutland, P. Dittrich and F. Wirbeleit, "Mechanism of NO_x Reduction from Water Injection in a DI Diesel Engine," Proceedings of 1999 Spring Technical Conference of ASME Internal Combustion Engine, ICE-Vol. 32-1, pp. 13-27.

[18] T.S. Strauss, G.W. Schweimer and U. Ritscher, 1995, "Combustion in a Swirl chamber Diesel engine Simulation by Computation of Fluid Dynamics," SAE Trans. 104, Section: Journal of Engines, pp.519-530.

[19] M.A. Patterson, S.C. Kong, G.J. Hampson and R.D.Reitz, 1994, "Modelling the Effects of Fuel Injection characteristics on Diesel Engine Soot and NO_x Emissions," SAE paper 940523.