ISSN 2395-1621

Investigation of Emissions in Inline Diesel Engine Combustion using ECFM- 3Z Model

#1Mr. Vinayak V. Kumbhar, #2Dr. Suresh M. Sawant

#1Research Scholar, Department of Mechanical Engineering,
Rajarambapu Institute of Technology, Rajaramnagar,
Sakharale, Tal.Walwa, Dist.Sangali, Maharashtra, India.

#2Professor, Department of Mechanical Engineering,
Rajarambapu Institute of Technology, Rajaramnagar,
Sakharale, Tal.Walwa, Dist.Sangali, Maharashtra, India.

ABSTRACT

Diesel engine combustion contains Complex phenomenon which includes Heat transfer, Turbulence flow, two phase flow Evaporation and mixing Spray, wall impingement dynamics, Film chemistry, Turbulence chemistry interaction, Real gas effect, High speed flow radiation etc. Diesel sector model is used in the simulation which allows to reduction of time and experimental task. Innovative Combustion concepts are used in three dimensional CFD computations in order to understand behaviour of combustion process. The objective of the paper is to present preliminary results of investigation for the effect of variation in injector nozzle hole diameter on diesel engine emissions. The graphs of pressure variation during motoring and firing conditions, temperature and heat release with respect to crank angle degree are drawn. The numerical result using Simulation of Turbulence flow in Arbitrary Region Computational Dynamics code is validated with experimental results. PISO algorithm is used for solving Navier Stoke equation. It is found that the pressure variation with crank angle degree obtained from simulation is close agreement with experimental results. The results show that current methodology can be used for the analyzing the parameter of the diesel combustion as a useful tool.

Keywords: Combustion, ECFM 3Z, STAR CD.

ARTICLE INFO

Article History

Received: 25th March 2017

Received in revised form :

25th March 2017

Accepted: 25th March 2017

Published online :

4th May 2017

I. INTRODUCTION

The CFD tool is used for simulation of diesel engine combustion which provides better justification in understanding the heat release rate, peak pressure, peak temperature and formation of the reactant process. Nowadays there is tremendous progress happen in the computational analysis which causes the reduction in experimental analysis. Diesel engine widely used in the various field such as good transport, power generation, pump output etc. For computational analysis STAR-CD software is used .Diesel fuel is directly injected into the cylinder under high pressure which helps to atomization process of fuel. The mixture in the diesel combustion is heterogeneous in nature. The fluid fuel is injected before top dead centre position is atomized into fine droplet forms combustible mixture which auto ignite under high compression ratio. Diesel engine operated at lean mixture resulting higher thermal efficiency. The fluid fuel is injected with various range of air fuel ratio from stiochiometric to rich mixture in order control desirable combustion process.

Experimental investigation required expensive measuring instrument and high level of skill in order to get accurate results.[1] Advance computational technique reduces time and gives better understanding of invisible combustion process. The advances in combustion modeling in Simulation of Turbulent flow in Arbitrary Regions-Computational Dynamics (STAR-CD) applied to diesel combustion analysis.[2] Model validation was done with experimental data of two different engines with different bowl design, initial swirl and injector nozzle diameter. Combustion analysis for diesel inline cylinder engine with different percentage of EGR conditions was analyzed by R. Manimaran et al [3-4]. Swirl ratio is varied by changing piston bowl profile .Optimum swirl in combustion chamber helps in rapid mixing of fuel and air in order to form combustible mixture. R. Manimaran et al. observed that the Peak pressure and heat release is increased significantly by varying the swirl ratio and injection timings [4-5]**.** CFD-3D tool Simulation of Turbulent flow in Arbitrary regions-Computational Dynamics (STAR-CD) was use to study the influence of different start of injection timing. Invisible combustion process analyzed to improve fuel economy and emission reduction using Extended Coherent Flame 3 Zone Model. Effect of different injection pressure, start of injection and injection spray angle nozzle on diesel combustion was studied by Maddalena Auriemma et al [6]. They reported that the nozzle geometry affect the fuel spray penetration and air fuel mixture homogeneity within the combustion chamber. They also noted that the injection pressure has positive impact on the thermal efficiency with considering effect of higher pressure on atomization and mixing rate. Automization of fuel is very important parameter in diesel combustion.

II. METHODOLOGY

A spline is generated from the solid model by using trimming operation and used for the creation of in-cylinder mesh. In this paper 51.42 degree sector mesh is used in order to reduce computational time. Computational fluid domain meshing was done by using ES-ICE (Expert System – Internal Combustion Engine). The inline cylinder grid thus obtained is checked for negative volumes at all locations between BDC and TDC. The meshes are placed very fine to the wall and thereby both the hydrodynamics boundary layer and thermal boundary layer phenomenon consideration [3]. Grid independence test and time independent test is carried out. The total number of cells in the moving domain amounts more than 25000 at TDC. The cell count beyond 25000 cells does not alter the in-cylinder peak pressure and other process variables. It can be observed that in-cylinder averaged peak pressure does not get varied even the crank angle step interval is reduced below 0.025 deg. Hence the optimum crank angle step interval is maintained at 0.025 deg for all simulations in this study.

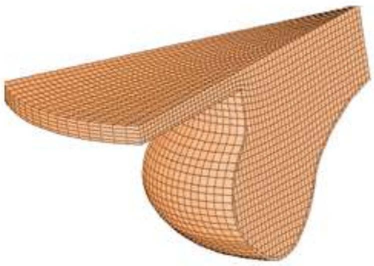

Figure 1: Computational sector mesh at TDC position

Solver Details

STAR-CD (Simulation of Turbulent flow in Arbitrary Regions-Computational Dynamics) CFD code which uses continuity, momentum, energy and species equation in the respective solver to understand the combustion phenomena. The computational domain consists of finite volume grids with moving boundaries and meshes. For simulation purpose 60 radial cells, 16 azimuthal cells and 130 axial cells in a 51.42 degree sector of engine cylinder was used. [3] The ECFM-3Z model is a general purpose combustion model capable of simulating the complex mechanisms of turbulent mixing, flame propagation, diffusion combustion and pollutant emission that characterize modern internal combustion engines. '3Z' stands for three zones of mixing, namely the unmixed fuel zone, the mixed gases zone, and unmixed air plus EGR zone.[8]

ECFM-3Z model used for simulating combustion regimes with n dodecane fuel. Standard k-ε model turbulence model was used for the predictions of turbulent flow and turbulence viscosity. The program is based on PISO (Pressure Implicit Splitting of Operators) algorithm with first order upwind differencing scheme. Lagrangian multiphase approach used for the simulating of fuel spray regime. For consideration of collision phenomena 500000 parcels are used to capture precise physical process. Gravitational force effect is also considered. [3-4]

Table 1: Boundary conditions

Boundary	**Momentary boundary condition**	**Thermal boundary condition**
Cylinder head	Wall	650K
Cylinder wall	Wall	600K
Piston bowl	Moving wall	500K

Reitz and Diwakar spray model is used for droplet breakup phenomena. A droplet breakup phenomenon occurs in two modes i.e. bag breakup and stripping breakup. In 'bag breakup' process, non-uniform pressure field around the droplet causes it to expand in the low-pressure wake region and eventually disintegrate when surface tension forces are overcome. In 'Stripping break-up' process, liquid is sheared or stripped from the droplet surface. N-Tetradodecane fuel is selected from NIST table because it gives correct liquid fuel density. The flame propagation phase is modelled by the flame surface density transport equation incorporating the theoretical flame speed. Huh's atomization model is based on the gas inertia and the internal turbulence stresses generated in the nozzle. Initial turbulence set at 10% of mean flow and initial turbulence scale set as 0.001m.For interpolation vetex data is used and under relaxation factor for langrangian source at 0.5. Droplet trajectory maximum file size set as 400mb.Fuel mass flow rate is used to specify the injection timing and corresponding distribution of mass flow rates. This spray impingement model is formulated within the framework of the Lagrangian approach, on the basis of literature findings and mass, momentum and energy conservation constraints. In order to reflect the stochastic nature of the impingement process, a random procedure is adopted to determine some of the droplet post-impingement quantities. This allows secondary droplets resulting from a primary droplet splash to have a distribution of sizes and velocities. The various models used for the combustion analysis are shown in table II.

Table II: Models associated with STAR-CD Code

Name of the model	**Model**
Combustion model	ECFM-3Z
Turbulence model	I-L
Wall function model	Angel-berger
Breakup model	Reitz
Droplet model	Bai
Automation model	Huh
No_x mechanism	Hand,De Soete
Soot model	Mauss

III. RESULT AND DISCUSSION

The 51.42 degree sector model was used for analysis purpose. To verify the results from the CFD code, experimental engine specification shown in table III. CFD sector of diesel engine model is modelled by using standard cad model package. A series of time and grid independency test is carried out. Crank angle step of 0.025CA and mesh with more than 25000 cells at TDC are the key parameter for further simulation from these test. Engine specification and simulation parameter are shown in the table III. The result obtained from the simulation data shows good agreement with experimental result data shown in figure 2.

Table III: Engine specification

Description	Detail
Bore	85 mm
Stroke	96 mm
Connecting rod length	149 mm
Compression ratio	18
Engine speed	3750 rpm
Fuel	n-Dodecane
Start of injection	10.0 CA bTDC
Injection duration	20 deg
Valves/Cylinder	4
EGR	0 %
Injector nozzle diameter	150 micron

In-cylinder cylinder combustion parameter such as pressure, temperature, heat release rate etc parameter are predicted numerically for the same geometry and compared with experimental data. Equation for the heat release rate due to increase in pressure and temperature is given by the first law of thermodynamics as below

$$\frac{dy}{dx} = \frac{1}{\gamma - 1} V \frac{dp}{d\Theta} + \frac{\gamma}{\gamma - 1} P \frac{dV}{d\Theta}$$

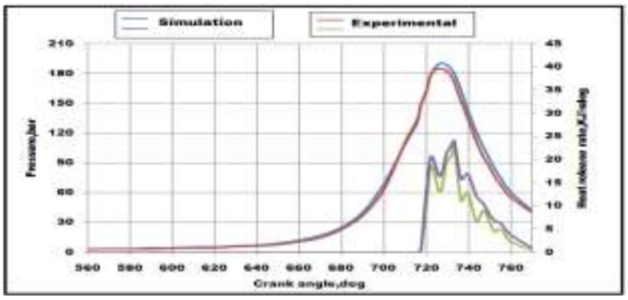

Figure 2: Pressure and heat release rate vs crank angle

Both premixed and diffusion type combustion occurs in diesel engine. Initial peak in heat release rate curve is due to premixed combustion. Second peak in the heat release rate curve is due diffusion combustion. The results obtained from simulation shows good agreement with experimental results in terms of auto ignition timing onset, pressure rise and peak pressure. Maximum deviation in peak pressure obtained from simulation with respect to experimental data is less than 0.3%.

Effect of variation in injector hole diameter

The peak pressure depends on the combustion rate and how much fuel is taking part in rapid combustion. The uncontrolled phase is governed by the ignition delay period and by mixture preparation during the ignition delay period. Therefore mixture preparation during the ignition delay period is responsible for the variation of peak pressure and maximum rate of pressure rise. Faster combustion and rapid pressure rise is due to better fuel atomization and mixing.

Pressure, bar vs. crank angle, deg graph shown in figure. 3. It shows that maximum pressure is developed in injector having diameter 135 micron is more compare to injector having diameter 150 micron. Pressure is developed engine which having injector with a diameter 135 micron and peak pressure developed was 195 bar. Temperature vs crank angle, as in figure 4.

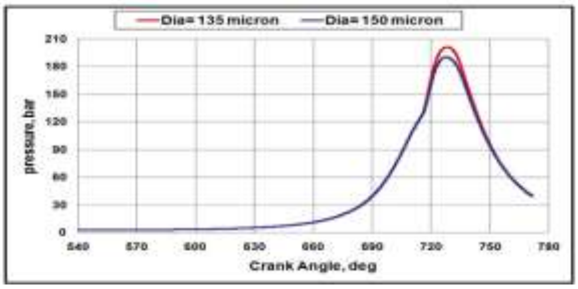

Figure 3: Pressure vs crank angle

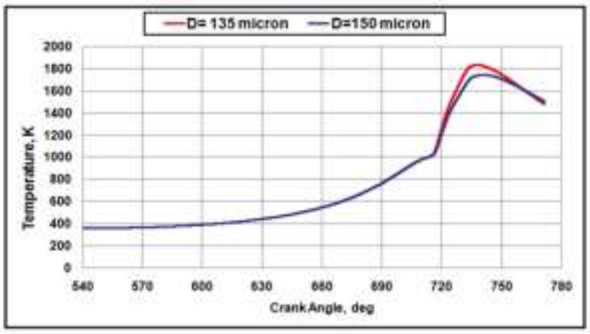

Figure 4: Temperature vs Crank angle

The soot and NO_x emission obtained from different injector nozzle diameter is shown in figure 5 and 6.No_x was reduced with increase in injector nozzle diameter whereas soot was increased. Greater peak temperature developed in the cylinder causes higher NO_x emissions in diesel engine. The increased in the injector hole diameter leads to reduction in oxidation of soot because the soot oxidation takes place later in expansion stroke when the temperature of gas is lower. Further we can analyze EGR analysis in order to reduce NO_x emission.

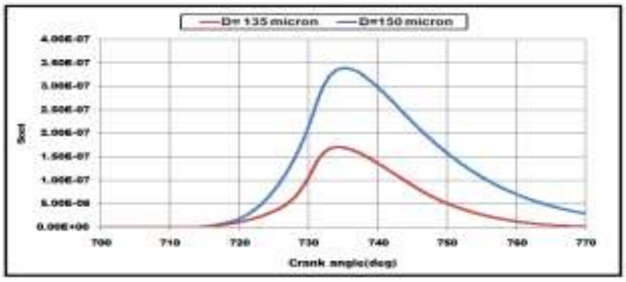

Figure 5: Soot vs Crank angle

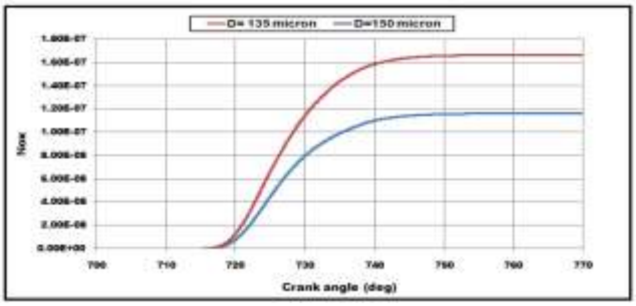

Figure 5: No_x vs Crank angle

IV. CONCLUSION

In the present work, an advanced CFD tool was used to explore the capability of Extended Coherent Flame Model - 3 zone model for predicting emissions by combustion analysis of inline cylinder diesel engine combustion. The results obtained from simulation shows good agreement with experimental results in terms of auto ignition timing onset, pressure rise and peak pressure.

From the present study, it is found that

- Peak pressure occurs at 8 degree aTDC and peak temperature occurs at 19 degree aTDC.
- Reducing injector nozzle hole diameter causes 9% increased in peak firing pressure with reduction in emissions.
- The numerical simulation pointed out injector nozzle hole diameter plays major role in controlling performance and No_x-smoke trade-off.
- Due higher temperature the No_x emission was increased by one-third whereas soot emission decreased by doubled from 712 Crank Angle.

REFERENCES

[1] R. Manimaran, R. Thundil Karuppa Raj, K. Senthil kumar, “Premixed charge compression ignition in a direct injection diesel engine using Computational Fluid Dynamics”, WSEAS TRANSACTIONS on HEAT and Mass TRANSFER, Jan 2013,Vol 8 Issue 1,P17.

[2] D. Abouri, G. Desoutter, A. Cano, F. Ravet, “Advances in combustion modelling in STAR-CD: a new technique for automatic grid and mesh motion generation applied to diesel combustion and emissions analysis ” ,19th International Multidimensional Engine User’s Meeting at the SAE Congress 2009,April,18,2009 Detroit, MI.

[3] R. Manimaran, R. Thundil Karuppa Raj, “CFD analysis of combustion and pollutant formation phenomena in a direct injection diesel engine at different EGR conditions”, International conference on Design and Manufacturing ,IConDM 2013,procedia Engineering 64(2013) ,497-506.

[4] R. Manimaran, R. Thundil Karuppa Raj ,“Numerical Investigations of Spray Droplet Parameters on Combustion and Emission Characteristics in a Direct Injection Diesel Engine using 3-Zone Extended Coherent Flame Model”, Advanced Materials research ,volume 768,226-230.

[5] Raouf Mobasheri, “Analysis the ECFM-3Z combustion model for simulating the combustion process and emission characteristics in a HSDI diesel engine.”, International journal of spray and combustion dynamics Volume 7, Number 4, pages 353 – 37.

[6] Maddalena Auriemma, Stefano Iannuzzi, “Injection system assessment to optimize performance and emissions of a non-road heavy duty diesel engine: experiments and CFD modeling.”, Journal of KONES Powertrain and Transport, Vol. 19, No. 3 2012.

[7] Renganathan Manimaran, Rajagopal Thundil Karuppa Raj, Senthil Kumar K., “Numerical analysis of direct injection diesel engine combustion using Extended Coherent Flame 3-Zone Model.” , Research Journal of Recent Sciences, ISSN 2277-2502, Vol. 1(8), 1-9, August 2012.

[8] R. Manimaran, R. Thundil Karuppa Raj, “Effect of Swirl in a Constant Speed DI Diesel Engine using Computational Fluid Dynamics.”, Vol. 4 (4) – December 2012.

[9] O. Colin and A. Benkenida, “The 3-Zones Extended Coherent Flame Model (ECFM3Z) for Computing Premixed/Diffusion Combustion.”, Oil & Gas Science and Technology – Rev. IFP, Vol. 59 (2004), No. 6, pp. 593-609.