ISSN (Print 0974-2115) (Online 2349-8552)
www.jchps.com Journal of Chemical and Pharmaceutical Sciences

Analysis of combustion and pollutant formation in diesel engine at various EGR using CFD (neem/diesel blend)

Sebalraj S*, Thirunaukkarasu V, Balaji G
Department of Mechanical Engineering, SRM University, Kattankulathur-603203
***Corresponding author: E-Mail: skitsebal@gmail.com**

ABSTRACT

Exhaust gas recirculation (EGR) is varied in diesel engine (neem/diesel blend fuel) by extended coherent flame model in 3 zones. The three zones are too small to be resolved by the mesh, so they are modeled as sub-grid quantities. To find the appropriate space and time grid and time independent test are carried out. To validate the model preliminary studies are carried out with experiments. By varying the EGR (0% to 3%) in the cylinder present study is conducted. CFD results shows that as EGR increase oxides of nitrogen decreased. This model shows that in the combustion chamber flame temperature is lowered with increase in EGR rate and soot formation, other pollutants are increasing due to lower oxygen concentration. From CFD analysis it shows that as EGR increases the combustion efficiency.

KEY WORDS: CFD analysis, neem/diesel blend, Diesel engine, combustion and pollution.

1. INTRODUCTION

Diesel engine requires high compression ratio produce high temperature to achieve auto ignition, that resulting high expansion ratio. Makes engine discharge less energy in exhaust. Power output and nitrogen oxide emission is achieved by controlled feedback injection timing (Heywood, 1998) in CI engine. The time delay between spray of fuel and actual combustion is referred as ignition delay period. It is expansive to study the process by experiment. Computational techniques are mostly useful to study these processes.

Reitz and Diwakar, 1986 were proposed an eulerian-lagrangian spray and atomization model for diesel spray. Their internal flow study characteristic of a multi-hole fuel injector gives better agreement. Eddy break-up concept model developed by (Magnussen and Hjertager, 1976). This relates combustion rate to eddy dissipation.

Hossai pour and Binesh, 2009 generates a model for spray penetration in combustion chamber by various spray droplets. This is referred us discrete droplet method based on statistical method. To analyze the effect of swirl on combustion, (Prasad, 2011) carried out simulation on various bowl configurations. Their studies indicate that sac-less injector which reduces emission. Spray dynamics plays a vital role on rate of evaporation, flow field and combustion. Combustion efficiency and polluted formation are affected by the atomization of fuel (Colin and Benkenida, 2004).

For optimizing combustion processes various models are studied. By designing the intake port and shaping the piston bowl the swirl is varied for reentered combustion. (Cenksayn and Mustafa, 2009) studied a model for the effect of variation of injection timing in diesel engine.

CFD Models: The modeling for flow field of continuous and dispersed phase, combustion and pollutant formation are conducted in STAR CD, STAR CCM and Ansys fluent CFD package. The 3-D in-cylinder, transient and reaction flow system in a diesel engine is modeled by solving by governing equations from law of conservation of mass, momentum, energy and species.

Modeling of turbulence: The flow field inside the cylinder is turbulent in nature at all speeds and dimensions. To capture the properties of fluid in-cylinder fluid dynamics it is necessary to model the turbulence. The RNG k- ε model (Payri, 1998) is which the turbulence Reynolds number forms of the k- ε equations are used in conjunction with algebraic law of wall representation of flow, heat and mass transfer for the near wall region.

Models of combustion and ignition: The ECFM – 3Z model (Dhuchakalaya and Watkins, 2010) is a combustion model which can simulate turbulent mixing, flame propagation, diffusion combustion and pollutant emission that characterize modern internal combustion engine. This model can be used to for in-cylinder analysis.3z stands for three zones of mixing, namely unmixed fuel zone, mixed gases zone and unmixed air plus EGR zone. To resolve the three zones, they are modeled as sub-grid quantities. The combustion takes place in mixed gases zone which is the result of turbulent and molecular mixing.

Droplet models: Droplets from nozzle enter the combustion chamber in high velocity and get sheared in outer periphery. Hence a model to disintegrate the droplets is given (Kondoh, 1985). Huh's model is based on the criteria of gas inertia and the internal turbulence stresses generated in the nozzle. The spray impingement model is formulated within the frame work of the lagrangian approach in order to reflect the stochastic nature of the impingement process. a random procedure is adopted to determine some of the droplets post impingement quantities. This allows secondary droplets resulting from a from a primary droplet splash to have a distribution of sizes and velocities.

Models of pollutant formation: The oxides of nitrogen plays important role in air pollutant (Balaji, 2015). The NO_x reaction time scale is larger than time scale of turbulence mixing process and the combustion process of hydrocarbons that control the heat releasing reactions. Computation of NO_x be decoupled from main reacting flow field predictions by identifying 3 mechanism to form nitric oxide during combustion of hydrocarbons. The formation and emission of carbonaceous particles called soot is observed during the combustion of hydrocarbons. The first approach to modeling soot, due to is based on laminar flamelet concept. All scalar quantities are related to mixture of fraction and scalar dissipation rate. The rates of soot formation can be correlated with local conditions in diffusion flames or in partially premixed counter flow – twin flames.

2. METHODS & MATERIALS

Computational procedures:

Generation of mesh: CFD simulation starts with the geometry of the piston bowl, referred from Colin (1998). This is shaped from a standard computer aided design. After the generation of bowl, spline is created from bowl profile and used to create in-cylinder mesh. The meshing of the in cylinder fluid domain is performed using es-ICE (Expert system- Internal combustion engine grid generation tool. A 45° sector mesh is considered due to symmetry nature of in-cylinder domain and therefore computational time can be reduced. The in-cylinder grid obtained is checked for negative volumes at all locations between BDC and TDC. The meshed geometry of the flowing fluid domain at TDC i.e, 720 deg crank angle is shown in fig.1 and contains almost hexahedral and tetrahedral cells near cylinder axis.

Boundary condition: To facilitate the solution of energy and momentum conservation equations the boundary conditions in-cylinder fluid domain has to be supplied. The boundary conditions includes wall at the bottom, periodic zones at the sides, cylinder liner wall, cylinder head wall at the top, axis and the injector. Piston top wall is applied with moving wall boundary condition. The velocity of piston wall is calculated by engine speed, crank angle, connecting rod and stroke length details. The different temperatures obtained from experiments at various surfaces are listed in Table.1.

Convergence Criteria: The transient simulation start at 680 CA and else at 800 CA for this study. For every time step the continuity momentum, energy and species equations are solved. Convergence conditions have to be met.

Time and grid Independency Test: Increasing the cells beyond 45000 cells does not do any changes in cylinder peak pressure. Grids are solved by finite volume method. In cylinder average peak measure does not get varied.

Solver details: In the simulation of droplet break up and spray penetration phenomena, Lagrangian multiphase treatment is activated. Experience the varyinn cylinder turbulence dispersion models included for the droplet dispersion. To find collision in every time step collision model (Kondoh, 1985), is included. Over 50 million droplet parcel are considered to study the trajectory, penetration and collision physics.RNG k- ε model is used for euilerian flow field model in the cylinder. The flame surface density equation is achieved by ECFM model.

Figure.1. Computation grid with boundary surfaces at TDC

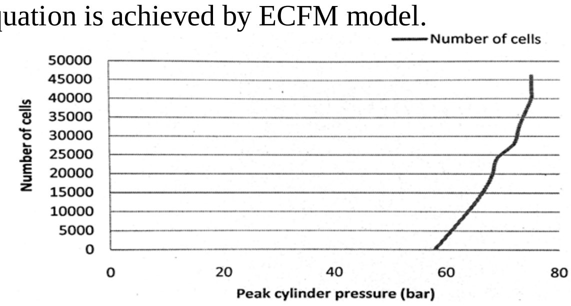

Figure.2. Peak pressure with number of cellsat crank angle interval

Table.1. Boundary conditions

Boundary	Momemtum boundary condition	Thermal boundary condition
Cylinder head	Wall	450 k
Cylinder wall	Wall	400 k
Piston bowl	Moving Wall	450 k
Cylinder side face	Periodic	450 k

Table.2. Codes of Model in CFD package

Phenomena	Model	Phenomena	Model
Droplet breakup	Reitz-Diwakar	Atomization	Huh
Turbulence	RNG k-ε	Boiling	White
Combustion	ECFM-3Z compression	No_x mechanism	Hand, De soete
Liquid film	Angelberger	Soot	Mauss
Droplet wall interaction	Bai	Droplet breakup	Reitz-Diwakar

Post processing: Time steps computations are conducted till the values of the conservation equation of continuity, momentum and energy fall below $10^{-5.}$ At every time steps turbulence, spray model and combustion model and soot emissions are computed. The code output in the cylinder gives the data to an ASCII file for further analysis.

Validation: The pressure data computed is compared with the experimental data. The Colin 45° CFD model shows a set of time and grid independency test are carried out. CA step intervals 0.0250 are observed from these tests. Figure 4, shows the comparison between numerical and experimental data. The numerically simulated pressure values are in good agreement with the data of experiment. Maximum deviation obtained is less than 0.2%.

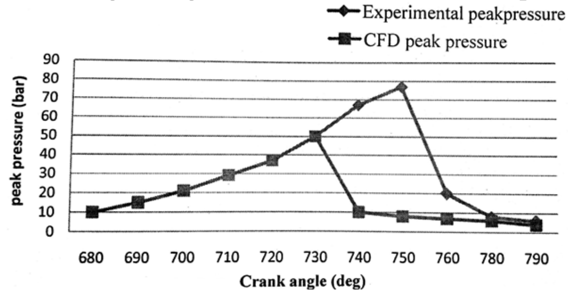

Figure.3. Comparison of computed and experimental

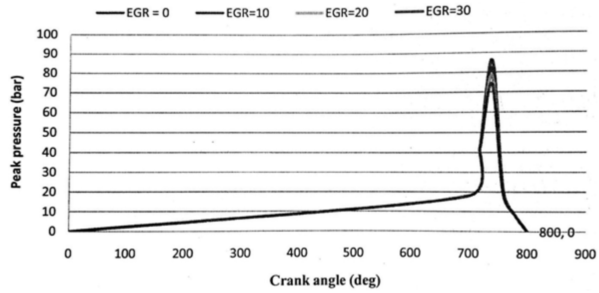

Figure.4. Variation of pressure with crank angle at different EGR

3. RESULTS AND DISCUSSIONS

In cylinder parameters are predicted numerically for same geometry. EGR is varied from 0% to 30 %. Up to 736°CA the in-cylinder pressure increases, afar that it decreases as shown in figure.5. At 0% EGR the peak pressure reaches 84 bar, At 30 % EGR the pressure drops to 75 bar. The EGR reduces the reaction in the cylinder as shown in figure.6.

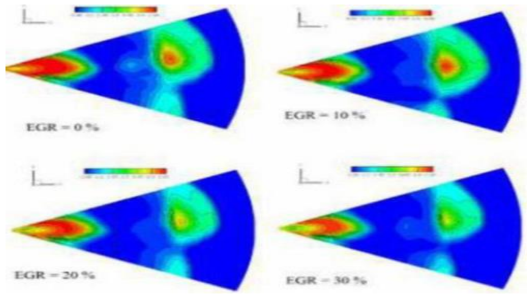

Figure.5. Reaction variable contour

$$\frac{dQ}{d\theta} = \frac{1v\,dp}{\gamma-1} + \frac{\gamma}{\gamma-1} P \frac{dv}{d\theta}$$

First law of thermodynamics gives the heat release rate due to increase in pressure and temperature in above equation. Due to rise in pressure after 716° deg CA heat release rate curve increases. Figure.5, shows heat release rate in different EGR condition. Highest EGR level will reduce peak pressure. The heat release rate is reduced from 41 J/deg CA to 32 J/deg CA with 28% heat release rate reduction. Figure.6, shows the reaction variable for burnt and unburnt location.

Reaction variable 0 indicates complete unburnt fuel and 1 indicates burnt fuel near the injector and above bowl edge. Table.3, shows the ignition delay period of various EGR conditions Increase in EGR regresses the ignition delay which increase the ignition delay from3.07 deg CA to 4.35 deg CA.

Table.3. Ignition delay at various EGR

EGR (%)	Ignition delay (deg CA)	EGR (%)	Ignition delay (deg CA)
30	4.35	10	3.55
20	3.97	0	3.07

Figure.6, shows that in-cylinder temperature at different EGR levels. It is observed that peak cylinder temperature drops from 1750 K to 1500 K as EGR increased from 0% to 30 %. Figure.7, shows the temperature contours and can be observed that cylinder temperature decreased with increase in EGR.

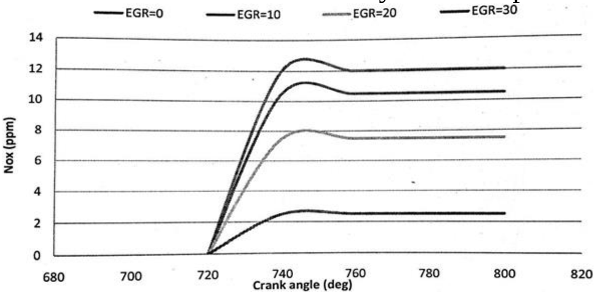

Figure.6. Variation of cylinder NOx for different EGR

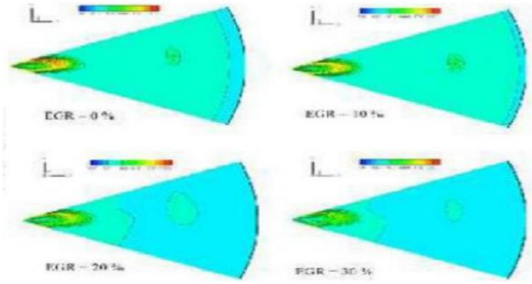

Figure.7. Temperature contour at 720 deg CA at different EGR

Soot and droplet SMD variations for a part of expansion and power stroke are shown in figure.8. Since the combustion regresses soot level increased from 0.17 g/kg to 0.25g/kg. Soot increases before 720 deg CA while NOx later. Soot is formed from unburnt fuel that nucleates from vapour phase. Evaporation of fuel depends on the temperature and, relative velocity. Figure.9, shows CO Contour that exhibit a increasing trend with EGR. As EGR increased Co results from combustion increases.

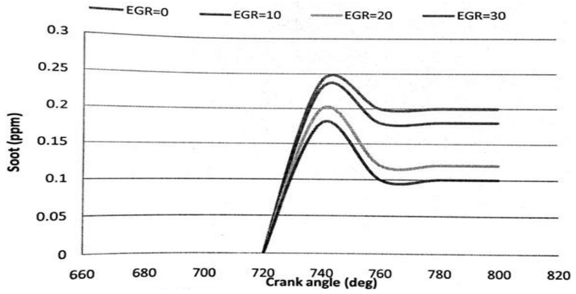

Figure.8. Variation of soot at various EGR

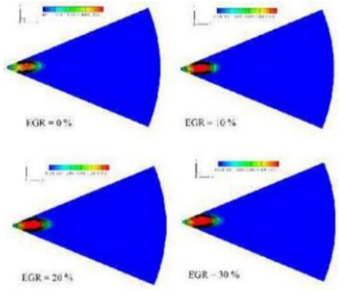

Figure.9. Contours of CO for various EGR condition

4. CONCLUSION

Several models governing neem/diesel injected engine combustion and pollutant formation are studied. Results from simulations are validated. At various EGR presents in cylinder the following conclusions are obtained.

- As EGR increases from 0% to 30%, peak in-cylinder pressure dropped by 9 bar, which results in regressive combustion.
- EGR is decreased from 30% to 0% heat release increases by 28%.
- Peak in-cylinder temperature reduced by 14.3% as EGR is lowered from 30% to 0% and that decreases the NO_x by 75%.
- Soot is increased by 14.2% and droplet SMD is increased by 9.2% as EGR increased from 0% to 30 %.

REFERENCES

Balaji G, Cheralathan M, Effect of CNT as additive with biodiesel on the performance and emission characteristics of a DI diesel engine, International Journal of ChemTech Research, 7 (3), 2015, 1230-1236.

Cenksayn, Mustafa canacki. Effects of engine timing on the engine performance and exhaust emission of a dual fuel diesel engine, energy conversion & management, 50, 2009, 203-213.

Colin O, Benkenida A, The 3-zone extended coherent flame model (ECFM 3-Z) for computing premixed/diffusion combustion, oil and gas science and technology – Rev.IFP, 59, 2004, 593-609.

Dhuchakalaya I, Watkins A.P, Auto ignition diesel spray using the Pdf –Eddy break up model, Applied mathematical modeling, 34 (7), 2010, 1732-1745.

Heywood J.B, International combustion engine fundamentals, Mcgraw hill, Newyork, 1998, 491-566.

Hossainpour S, Binesh A.R, Investigation of fuel spray atomization in a DI heavy duty diesel engine and comparision of various spray break up models, Fuel, 88, 2009, 799-805.

Jayasankara B, Ganesan V, Effect of fuel injection timing and in-take pressure on the performance of a DI engines, A parametric study using CFD, Energy conversion and management, 51, 2010, 1835-1848.

Kondoh T, Fukumoto A, Ohsawa K, Ohkubo Y, An assessment of multidimensional numerical method to predict the flow in internal combustion engines, SAE paper, 850500, 1985.

Magnussen B.F, Hjertager B.H, On mathematical modeling of turbulent combustion with special emphasis on soot formation and combustion, 16th symp. on combustion, The combustion institute, 1976, 719-729.

Payri F, Benajes J, Tinaut F.V, A phenomenological combustion model for direct injection , compression ignition engines, Applied mathematical modeling, 12 (3), 1988, 293-304.

Prasad B.V.V.S.U, Sharma C.S, Anand T.N.C, Ravikrishna R.V, High swirl-inducing piston bowls in small diesel engines for emission reduction, Applied Energy, 88, 2011, 2355-2367.

Reitz R.D, Diwakar R, Effect of drop break up on fuel sprays, SAE technical paper series, 860469, 1986.