International Journal of Engineering Research and General Science Volume 3, Issue 3, May-June, 2015
ISSN 2091-2730

Effect of Double Injection - Combustion Performance and Emissions in HSDI Diesel Engine

K.Bala Showry[1]

[1]*Principal DRK Institute of Science and Technology* gurikini@gmail.com

Abstract

Due to the stringent legislation for emission of diesel engines and also increasing demand on fuel consumption, the importance of detailed Multidimensional modelling and simulation of fuel injection, mixing and combustion have been increased in recent years. In the present work CFD has been used to study the performance and emission by using split injection where in simultaneously reduction of NO_X and PM are possible. No other technique can reduce simultaneously both the emissions. Combustion models for predicting exhaust emissions of diesel engines can be classified into two groups, the first one is multi-zone phenomenological model, and multidimensional fluid dynamics model. In this research work double injection have been carried out for different dwells with 0% exhaust gas recirculation. It was found from the simulations 8°CA to 10°CA was an optimum dwells for double injection. The predicted in cylinder pressure, temperature NO_X and SO_X values have been validated with experimental results. This paper provides an overview of the sub models implemented, which account for liquid spray atomization droplet, secondary break-up, droplet collision, impingement, turbulent dispersion and evaporation. NO_X and Soot emission data were obtained with the use of Zeldowich mechanism and Hiroyasu model Simulations have shown a very good agreement in terms of pressure NO_X and PM.

Keywords: computational fluid dynamics, Multidimensional modelling, simulation, split injection, dwell, pilot injection, main injection,

1. Introduction

Diesel engine is widely used in heavy duty transport applications. Diesel engine is more fuel efficient than spark ignition engine on the other side they have relatively higher emissions and noise levels. Diesel engine manufacturers have to address these problems to meet current and future government regulations which limit particulate and NO_X emissions, while maintaining a quite efficient engine to satisfy the consumers. Particulate matter and NO_X production along with engine noise highly depend on the combustion process Therefore precise control over the fuel injection and spray formation is essential in making improvements to the combustion process .The optimum pressure and optimum nozzle diameter increases the performance and consequently reduces the particulate matter with the better atomization and fuel- air mixing. This in turn unfortunately increases NO_X because of high temperature. To improve the performance and to reduce the NO_X –particulate formation without scarifying the fuel consumption, it is important to understand the relationships between various injection parameters and how they affect the combustion process. Along with the injection pressure and nozzle diameter other injection parameters like such as nozzle hole L/D ratio, rate of injection profile , effect of fuel spray ,spray characteristics , that may affect the droplet size, spray penetration exit velocity and spray cone angle. Use of multiple injections can reduce particulate emissions by as much as a factor of three without increasing NO_X emissions. This will be done by better mixing later in the cycle. Optimizing the injection pressure, injection angle and optimizing the nozzle diameter has proven to be an effective way to reduce particulate emissions and consequently improves the engine performance. Multiple injection strategies have been reported for simultaneous reduction of NO_X and PM in a large bore direct injection diesel engine [1, 2, 3].Small bore diesel engines results shown by Nehmer and Reitz [2] that pulsated injection might provide a method to reduce PM and allow for reduction of NO_X from controlled pressure rise. The effectiveness of double, triple and rate shaped injection strategies to simultaneously reduce NO_X and PM was also evaluated. Numerical simulations were carried out to explore the mechanism of soot and NO_X reduction for multiple injections [4]. Multiple injection strategies have a similar effect to the restarted single injection on NO_X reduction. Reduced emissions are due to the fact that the soot producing rich region is not replenished when the injection pressure is terminated and restarted. Zang investigated the effect of [5] pilot injection on NO_X, Soot emissions and combustion noise in a small diesel engine, soot emission was seen relevant to the pilot flame and reducing the pilot flame at the main injection starting time can reduce soot emissions. By optimizing pilot injection timings and quantity maintaining and dwell between main and pilot injections simultaneous reduction of NO_X and PM was obtained in a HSDI diesel engine [6].It was also shown that simultaneous reduction of combustion noise and emission is possible by the influence of the pilot burned gas through minimizing the fuel quantity by advancing the pilot injection timing [7].Combustion concepts like homogeneous charge compression ignition combustion have been shown to be effective for NO_X and PM reduction. The concept of HCCI was applied initially to spark ignition engines because of its volatility property for better

homogeneous mixture, where as in diesel engines this concept has been delayed as diesel has low volatility. With the concept of multi pulse injection the problem of homogeneous mixture in diesel engines could be solved and the same has been applied for high speed direct injection diesel engines effectively. Hashizume [8] proposed a low soot solution called multiple stage diesel combustion for higher load operating conditions. Although, soothing luminous flame was observed, this luminous flame disappeared quickly and most of the soot was oxidized rapidly smoke and NO_X were reduced. Su W ,Lin T, Pei Y.A have done work[9]on multi pulse HCCI diesel engine ,they used multiple short injection pulses for early injection and followed by main injection near top dead center and they found that for very early injection a great increase in Hydrocarbon emission was seen. Hasegawa and Yanagihara employed two injections called uniform bulky combustion system .The first injection was used to form a pre- mixture. The second injection was used as an ignition trigger. The ignition of premixed gas could be controlled by the second injection when the early injection maintained a low temperature reaction.

2. Methodology and Model formulation

The computer code used in this study was FLUENT. The code can solve unsteady, compressible turbulent flows with combustion and fuel spray, and have been used for the computations of various internal combustion engines The code uses a finite volume methodology to solve discredited Nervier-strokes equations. RNGK-ε was used in this study. It could predict more realistic large scale flame structures compared with the K-ε model. The RNG K-ε model is formulated as

$$\frac{\partial(\rho k)}{\partial t}+\nabla.(\rho uk)=\left[\frac{2}{3}\rho\kappa\nabla.u+\tau.\nabla u\right]+\nabla.(\alpha_\varepsilon\mu\nabla k)-\rho\varepsilon+W^s \quad (1)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\nabla.(\rho u\varepsilon)=-\left[\frac{2}{3}C_1-C_3+\frac{2}{3}C_\mu C_\eta k\nabla.u\right] \quad (2)$$

$$\rho\varepsilon\nabla.u+\nabla.(\alpha_\varepsilon\mu\nabla\varepsilon)+\frac{\varepsilon}{k}\left[(C_1-C_n)\tau\nabla u-C_2\rho\varepsilon+C_sW^s\right]$$

$$C_3=\frac{-1+2C_1-3m(n-1)+(-1)\delta\sqrt{6}C_\mu C_\eta\eta}{3} \quad (3)$$

$\delta = 1$; if $\nabla\cdot u < 0$ $\delta = 0$; if $\nabla\cdot u > 0$ and

$$C_\eta=\frac{\eta\left(1-\frac{\eta}{\eta_o}\right)}{1+\beta\eta^3}-\eta$$

$$S\frac{\kappa}{\varepsilon}S=(2S_{ij}S_{ij})^{1/2}$$

$$S_{IJ}=\frac{1}{2}\left(\frac{\partial u_I}{\partial x_J}+\frac{\partial u_J}{\partial x_I}\right)$$

In equation (1)-(3) k and ε are turbulent kinetic energy and its dissipation rate ρ, u,τ and μ are density, velocity, stress tensor and effective viscosity respectively.η is the ratio of the turbulent to mean strain time scale. S is the magnitude of the mean strain. m =0.5, and n =1.4.The C3 term accounts for the non- zero velocity dilatation which is closed.

Governing equations

The governing equations of gas flow consist of mass, momentum and energy conservation equations turbulence equations, gas state relation equations. To take care of physical modeling k-ε turbulence model is employed. The various equations, which are solved**:**

Continuity $\frac{\partial p}{\partial t}+\nabla(\partial U)=0$

Momentum $\frac{\partial \rho U}{\partial t} + \nabla(\partial U) = -\nabla p - \nabla\left[\frac{2}{3}\partial k\right] + \nabla\sigma + \partial g \quad \sigma = \mu\left[\nabla U + (\nabla U)^r\right] + \lambda\nabla .UI$

Turbulence Model

K-Equation $\frac{\partial(\rho k)}{\partial t} + \nabla(\rho U k) = -\frac{2}{3}\rho k\nabla U + \sigma\nabla U$
$+\nabla.[(\frac{\mu}{pr_k})\nabla k] - \infty\rho\varepsilon$

ε-Equation

$$\frac{\partial(\partial\varepsilon)}{\partial t} + \nabla.(\rho U\varepsilon) = -(2c_{\varepsilon 1}/3 - c_{\varepsilon 3})\rho\varepsilon\nabla.U$$
$$+\nabla.\left[(\frac{\mu}{pr_\varepsilon})\nabla\varepsilon\right] + \frac{\varepsilon}{k}\left[c\varepsilon_1\sigma : \nabla U - c\varepsilon_2\rho\varepsilon\right]$$

The quantities $c\varepsilon_{1,} c\varepsilon_{2,} c\varepsilon_{3,} pr_{\varepsilon,} pr_k$ are constants whose values are determined from experiments and some theoretical considerations, a feature that establishes certain universality. Standard values of these constants are often used in engine calculations as given

$c\varepsilon_1 = 1.44 c\varepsilon_2 = 1.92 c\varepsilon_3 = -1,\ pr_k = 1.0,$
$pr_\varepsilon = 1.3$

3. Mathematical Models

3.1 Figures

Spray model

Spray models used in this study is WAVE break up model suggested by Reitz and could be summarized as follows. [10] Liquid break up is modeled by postulating the new drops are formed (with drop radius r) from a parent drop or blob (with radius a) with stripping.

$$r_{new} = B_{0.}\ A \qquad (4)$$

Where B_0 =0.61 is a constant, the value of which is fixed. The rate of change of drop radius in apparent parcel due to drop breakup is described by using the rate expression;

$$\frac{dr}{dt} = \frac{r - r_{new}}{\tau_{bu}},\ \tau_{bu} = 3.788\frac{r}{\Lambda.\Omega} \qquad (5)$$

The spray wall interaction model used in the simulations is based on the spray wall impingement model descried in [8].The model assumes that a droplet, which hits the wall is affected by rebound or reflection based on the Weber number. The Dukowicz model was applied for treating he heat up and evaporation of the droplet which is described in [11].This model assumes a uniform droplet temperature .I n addition the rate of droplet temperature change is determined by the heat balance which states that that heat convection from the gas to the droplet ether heat up the droplet or supplies heat for vaporization. With higher droplet densities and relative velocities droplet collisions occur. High droplet densities are restricted to the spray kernel. High relative velocities can especially be seen at the tip of the spray, where preceding droplets are decelerated by the gas. Depending on the droplet collision conditions various effects like elastic droplet bouncing, droplet coalescence and droplet atomization are observed.

Ignition and combustion models

The shell auto ignition model was used for modeling of the auto ignition [10].In this mechanism 6 species for hydrogen fuel, oxidizer, total radical pool, branching agent, intermediate species and products were involved. In addition the important stages of auto ignition such as initiation propagation, branching and termination were presented by generalized reactions described in [10]. The combustion model used in this study is of the turbulent mixing controlled verity as described by Magnusson and Heritage[11] .This model assumes that in premixed turbulent flames, the reactions (fuel ,oxygen) are contained in the same eddies and are separated from eddies containing hot combustion products . The chemical reactions usually have time scales that are very short compares to the characteristics of the turbulent transport processes .Thus it can be assumed that the rate of combustion is determined by the rate of intermixing on a molecular scale of the eddies containing reactants and those containing hot products in other words by the rate of dissipation of these eddies.

NO_X and soot Formation Models

The reaction mechanism of NO_X formation is expressed in terms of the extended a Zeldovich mechanism.

$N2+O \leftrightarrow NO+N$ (6)

$N+O2 \leftrightarrow NO+O$ (7)

$N+OH \leftrightarrow NO+H$ (8)

From the fact that in most stoichiometric and fuel –lean flames , the occurring OH concentration very small , the third reaction of the Zeldovich mechanism can be neglected .F or the formation of thermal NO_X, the partial equilibrium approach can be used and the equilibrium of the first two reactions result in one global reaction as follows;

$N2+O2 \leftrightarrow 2NO$ (9)

the chemical species appearing in this global reaction are used in the given single step fuel conversion equation via:

$$\frac{d[NO]}{dt} = 2k_f[N_2IO_2] = 2\text{kf [N2/O2]} \quad (10)$$

Where only the forward reaction is considered and the reaction rate k_f is given as

$$K_f = \frac{A}{\sqrt{T}} \exp\left(\frac{-E_a}{RT}\right) \quad (11)$$

The soot formation model currently implemented in fluent is based upon a combination of suitably extended and adapted joint chemical /physical rate expressions for the representation of the processes of particle nucleation, surface growth and oxidation

$$\frac{dm_{soot}}{dt} = \frac{dm_{form}}{dt} - \frac{dm_{oxid}}{dt} \quad (12)$$

$$\frac{dm_{form}}{dt} = A_f m_{fv} p^{0.5} \exp\left(\frac{-E_a}{RT}\right) \quad (13)$$

$$\frac{dm_{soot}}{dt} = \frac{6M_c}{\rho_s d_s} m_s R_{tot} \quad (14)$$

Numerical model

The numerical method used in this study is a segregated solution algorithm with a finite volume –based technique. The segregated solution is chosen is due to the advantage over the alternative method of strong coupling between the velocities and pressure. This can help to avoid convergence problems and oscillations in pressure and velocity fields. This technique consists of an integration of the governing equations of mass, momentum species, energy and turbulence on the individual cells within the computational domain to construct algebraic equations for each unknown dependent variable. The pressure and velocity are coupled using the SIMPLE algorithm which causes a guess and correct procedure for the calculation of pressure on the staggered grid arrangement .It is more economical and stable compared to the other algorithms. The upwind scheme is always bounded and provides stability for the pressure correction equation. The CFD simulation convergence is judged upon the residuals of all governing equations. This scaled residual is defined as:

$$R^{\phi} = \frac{\sum_{cells} P \left| \sum_{nb} a_{nb}\phi_{nb} + b - a_p\phi_p \right|}{\sum_{cells} P \left| a_p\phi_p \right|}$$

Where Φ_p is a general variable at a cell p, a_p is the center coefficient, a_{nb} are the influence coefficients for the neighboring cells and b is the contribution of the constant part of the source term. The results reported in this paper are achieved when the residuals are smaller than 1.0×10 -4.

$$\sum_{r=1}^{n} r^2 = \frac{n(n+1)(2n+1)}{6} \qquad (1)$$

4. Turbulent Dispersion of Particles

The dispersion of particles due to turbulence in the fluid phase can be predicted using the stochastic tracking model of the particle cloud model. The stochastic tracking (random walk) model includes the effect of instantaneous turbulent velocity fluctuations on the particle trajectories through the use of stochastic methods. The particle cloud model tracks the statistical evaluation of a cloud of particles about a mean trajectory .The concentration of particles within the cloud is represented by a Gaussian probability density function about the mean trajectory. For stochastic tracking a model is available to account for the generation or dissipation of turbulence in the continuous phase. When the flow is turbulent Fluent will predict the trajectories of particles using the mean fluid phase. In this stochastic approach, Fluent predicts turbulent dispersion of particles by integrating the trajectory equations for individual particles, using the instantaneous fluid velocity, along the particle path during integration. By computing the trajectory in this manner for a sufficient number of representative particles termed as "number of tries" the random effects of turbulence on the particle dispersion may be accounted for.

Table 1

Engine Specifications

Engine Type	Caterpillar 3406, Single cylinder Direct injection ,4 valve
Bore	137.2 mm
Stroke	165.2 mm
Compression ratio	15:1
Combustion chamber	Quiescent

Table 2

Fuel specifications

Injector type	Electronically controlled common rail injector
Injection pressure	Variable up to 120 M pa
Number of Nozzles	6
Nozzle hole diameter	0.26 mm
Spray half cone angle	20°
Injection Approach	La grangian
Turbulence model	RNG K-ε

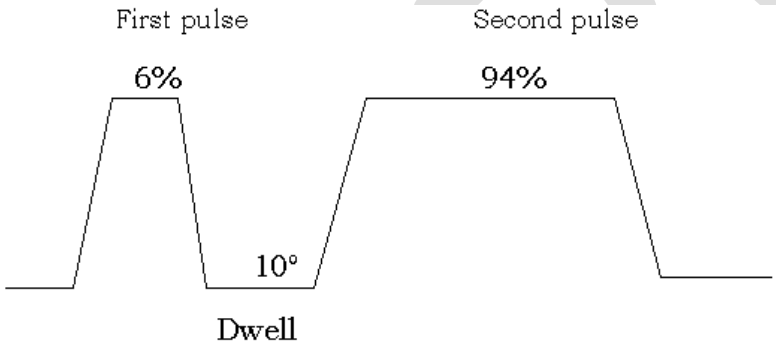


Figure 1 split injection nomenclature

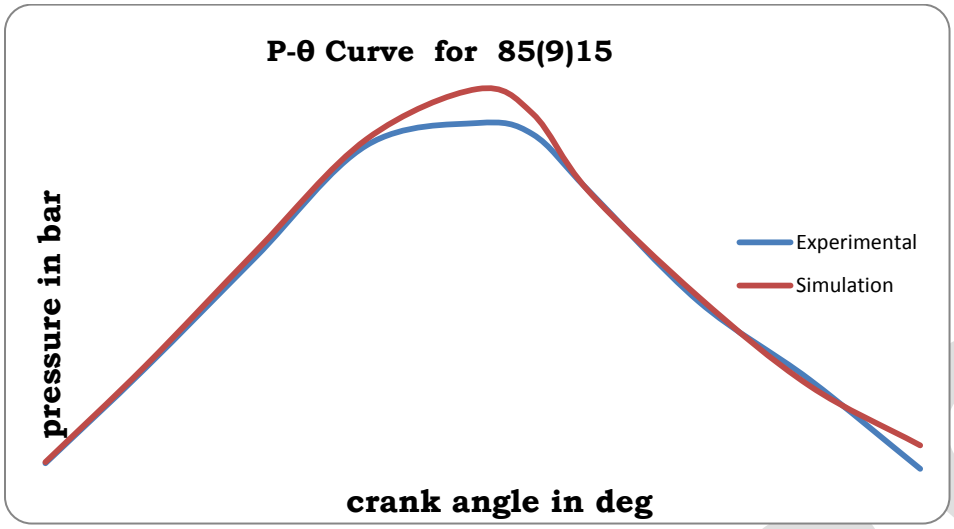


Figure 2 P-ϑ Curve for double injection

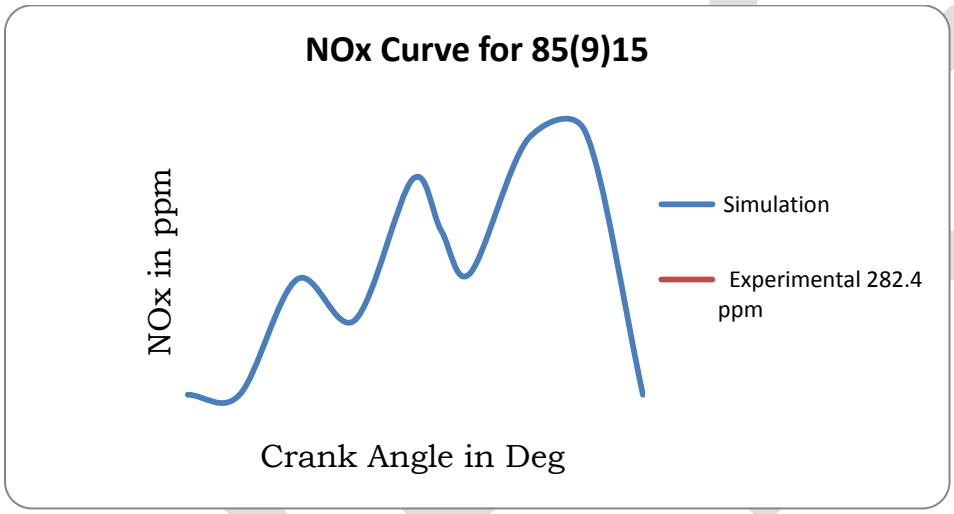


Figure 3- NO_x Curve for double injection

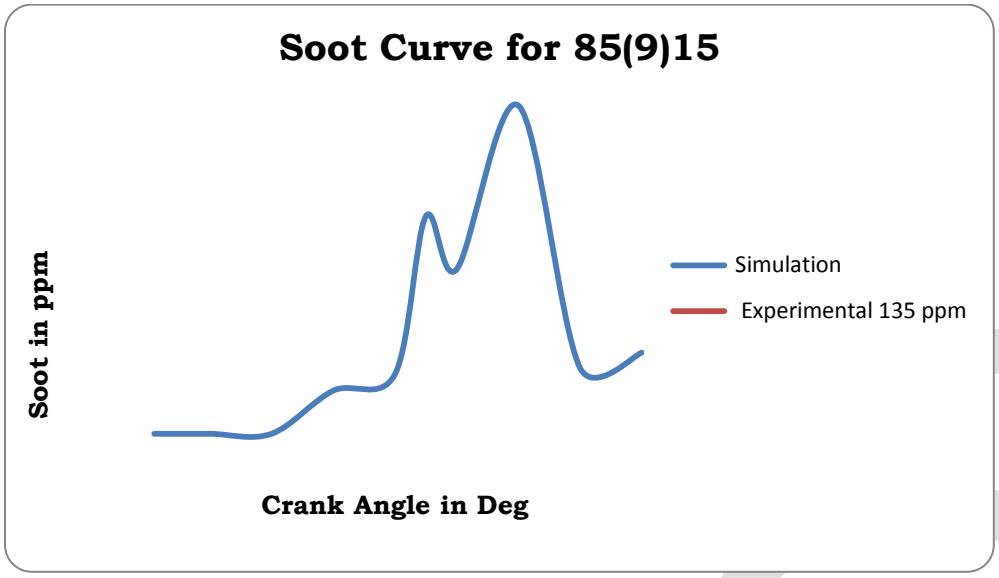


Figure 4 -Soot Curve for double injection

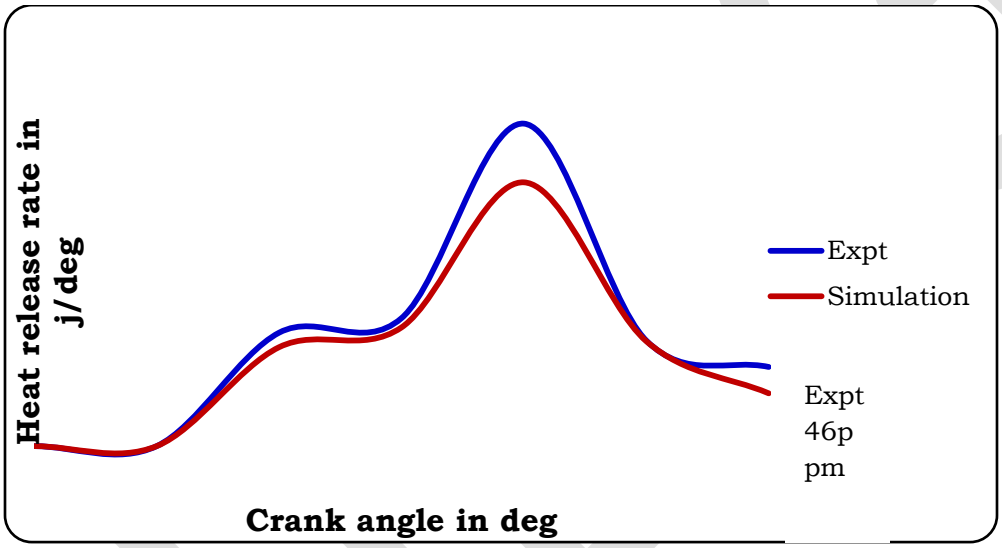
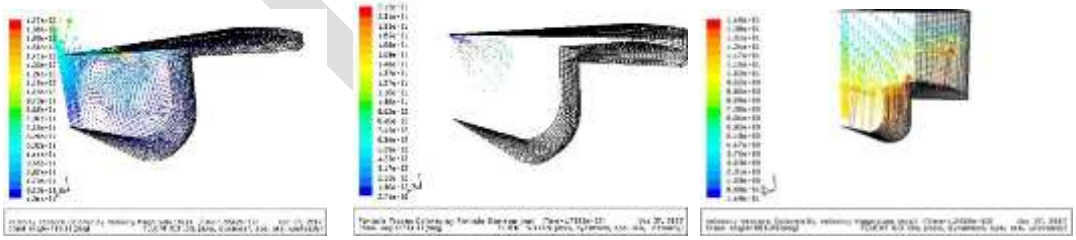


Figure 5- Heat release rate for double injection

CONTOURS OF SIMULATIONS



Velocity vector at 718CA **particle traces at704CA** **static temperature at708CA**

5. Results and Discussions

Fig 2 shows the variations of pressure (experimental and predicted) with variation in crank angle for double injection. It is clear from the fig1that the predicted results are closely following the experimental results. The major deviation of the predicted pressure from the experimental value is noticed near TDC. From the deviations two important aspects can be observed.

- The predicted peak pressure is higher the experimentally obtained pressure. The peak pressure with RNGK-ε model of diesel computational values and experimental for double injection 98 bar and 90bar respectively.

- The other reasons for the difference in peak pressure may be due to the residual gases in the clearance volume during exhaust stroke. The higher temperature residual gases are reduces to the fresh charge entry during suction, as it destroys some vacuum by expanding. Blow by and crevice flows also effect the in cylinder pressures. Both these features are not incorporated in this model.

- The occurrence of predicted peak pressure even before TDC indicates that majority of the fuel is consumed in combustion before the piston reaches TDC. This supports the argument that the ignition delay is more in experimental case. Another important observation that can be made is that the RNGK-ε predicts the pressure variations closer to the experimental results.

- A small injection before the main injection with 0% EGR is not effective in reducing particulate. Thus pilot injection would not be effective in enhancing mixing after the main injection.

- The pilot injection was effective at the 0% EGR condition is that pilot injections are known to reduce the premix burn fraction of burning resulting in lower NO_X production [929461].

- After considering EGR rate also without intercooler there is no additional benefit. Therefore simulations have been carried by considering 10(10)90 and 80(10)20 double injections and they have shown substantial reduction in particulate emissions when compared to single injection.

Fig 3 shows the mass fraction of NO variations with crank angle. The fig reveals that the maximum NO formation takes place between 5° bTDC and 25° aTDC. It is a fact that the combustion generated temperatures during this period will be high. High NO concentration is found in regions with close to stoichiometric mixture fraction and region where the temperature is high. It is understood from the fig the RNG K-ε model prediction agrees well with the measured data. As pilot injection was initiated ignition delay has been reduced hence the reduction in NO_X and Soot as temperature levels got reduced.

The measured value from the experiment was 246.05ppm and where as the computed value from RNG K-ε model is approximately 177.8ppm.

Fig4 shows the soot variations with respect to the crank angle. The soot emission predicted with experimental value is 36.2ppm and 42.6ppm with RNG K-ε model. It is very interesting to note that soot oxidation predominantly takes place in the high temperature regions in which NO_X production is high. The fact that local conditions that favor soot oxidation also favor NO formation is probably major reason for the well known Soot -NO_X trade off typically encountered when optimizing diesel engine. Soot production is given by particle inception rate as a spatial distribution with strong correlation to the mixture fraction field.

Fig5 shows the internal energy of the cylinder contents is the difference between the heat released and the heat transferred from the system. The internal energy increases rapidly due to combustion. Fig shows the variation of total energy with crank angle a sudden increase in internal energy is recorded at crank angle 10° bTDC. This indicates the start of combustion. The peak energy predicted during the operating cycle is 440 j /deg and with RNGK-ε, diesel computational model and experimental value for double injection with 9 deg dwell value is 392.5/deg

REFERENCES:

[1] Tiegang, Fang, Robert.E, Coverdill, Chia-fon F. Lee, Robert A White Effects of injection angles on combustion process using multiple injection strategies in an HSDI diesel engine Fuel 87(2008)3232-3239 2008.
[2] Nehmer DA, Reitz RD. Measurement of effect of injection rate and split injections on diesel Engine, soot and NO_X emissions.SAE Paper 940668 1994.
[3] Tow TC, D.A. Pier Pont, and R.D. Reitz Reducing particulate and NO_X Emissions by using multiple injections in a Heavy duty D.I. Diesel Engine S.A.E Paper 940897 1994.
[4] Zhiyu Han, A. N. Uludogan, Gregory J. Hampson, and Rolf D. Reitz Mechanism of soot and NO_X Emission Reduction Using Multiple-injection in a Diesel Engine SAE Paper 960633 1996.
[5] Zang.L. "A Study of pilot injection in a diesel DI engine" SAE Paper 1999-01- 3493 1999.
[6] Chen.SK. "Simultaneous reduction of NO_X and particulate emissions by using multiple injections in a small diesel engine", SAE Paper 2000-01-3084. 2000

[7] TanakaT.Ando A. Ishizaka K. "Study on pilot injection of DI diesel engine using common rail injection system". JSAE Paper2002:23:297-302 2002
[8] Combustion and Emission Characteristics of Multiple stage Diesel Combustion. SAE Paper 980505 1998
[9] Hasegawa R.Y. Anagihara. HCCI Combustion in DI diesel engine.SAE Paper 2003-01-0745. 2003
[10] Liu, A.B.and Reitz R.D."Modeling the Effects of Drop Drag and Break-up SAE Paper 930072 1993
[11]Dukowicz, J.K "Quasi steady droplet change in the presence of Informal report Los Alamos Scientific Laboratory" LA 7997-MS.
[12]D.A.Pierpont, D.T.Montgomery, and R.D. Reitz Reducing NO_X using multiple Injections and EGR in a D.I. Diesel engine SAE Paper 950217 1995
[13] "Internal combustion Engine fundamentals" by J.B.Heywood McGraw Hill Book Company1988.
[14]Taewon Lee and Rolf D. Reitz "The Effects of Split Injection and Swirl on a HSDI Engine E quipped With a Common Rail Injection System" SAE Paper 2003-01-0349 2003
[15]K.V.Tannin, D.D Wick Man, D.T. Montgomery, S. Das, and R.D. Reitz The influence of boost pressure on emission and fuel consumption of a heavy duty single cylinder diesel Engine SAE paper 1999-01-0840 1999
[16]ChongjinBai and A.D.Gosman "Development of methodology for spray impingement simulation" .SAE paper 950283 1995
[17]Liu, A.B.and Reitz R.D."Modeling the Effects of Drop Drag and Break-up" SAE 930072 1993