Journal of Multidisciplinary Engineering Science and Technology (JMEST)
ISSN: 3159-0040
Vol. 1 Issue 4, November - 2014

Experimental validation of combustion with CFD modeling in single cylinder four stroke CI engine fueled with biodiesel

Darshana S. Gaikwad
Heat power Engineering , KITS Ramtek , India
At post-Mansar,Tah-Ramtek,Dist-Nagpur,
City-Nagpur,country-India
Email: gaikwaddarshana08@gmail.com

Ajay V. Kolhe
Mechanical Engineering department, KITS Ramtek,
City-Nagpur,country-India
Email: ajay.kolhe4@gmail.com

ABSTRACT -This describes the development of sub models for combustion analysis in direct injection (DI) diesel engine where we used Pongamia Pinnata biodiesel blend as a fuel by using 50% biodiesl by volume of diesel(PME50). In the present study the Computational Fluid dynamics (CFD) code FLUENT is used to model complex combustion phenomenon in compression ignition (CI) engine. The experiments were performed on single cylinder DI engine diesel engine, with full load condition at constant speed of 1500 rpm. Combustion parameters such as cylinder pressure and heat release rate were obtained from experimental data. The numerical modeling is to be solved by CFD code Fluent, taking into account the effect of turbulence. For modeling turbulence Renormalization Group Theory (RNG) k-ε model is used. The sub models such as droplet collision model and Taylor Analogy Breakup (TAB) model are used for spray modeling. Modeling in cylinder combustion, species transport and finite rate chemistry model is used. The results obtained from modeling are to be compared with experimental investigation. In all, this study has demonstrated the feasibility of integrating a compact multi-component surrogate fuel mechanism with CFD to elucidate the in-cylinder combustion biodiesel fuels. The knowledge acquired here is useful for the development of biodiesel usage strategies in single cylinder four stroke CI engine inorder to use biodiesel blend as suitable alternative for the future use of IC engine.

Keywords-*Biodiesel; CFD; DI; heat release rate;, pressure .*

I. INTRODUCTION

Combustion research is more extensive, diverse and interdisciplinary due to powerful modeling tool like CFD. In CI engine the incylinder multiphase fluid dynamics like fuel spray, chemical reaction kinetics influences the combustion.Fluid flow in an internal combustion engine presents one of the most challenging fluid dynamics problems to model. This is because the flow is associated with large density variations. So, a detailed understanding of the flow and combustion processes is required to improve performance of engine . The simulation carried out in the present work to model DI diesel engine with bowl in piston for better understanding of the in cylinder gas motion with details of the combustion process .An attempt has been made to study the combustion processes in a compression ignition engine and simulation was done using computational fluid dynamic (CFD) code FLUENT, Turbulent flow modeling and combustion modeling was analyzed in formulating and developing a model for combustion process by using Karanja(Pongamia Pinnata) oil 50% by volume of diesel i.e.PME 50. As Karanja is one of the forest-based tree-borne non-edible oil with a production potential of 135,000 metric tons per year in India. which produce seeds containing 30–40% oil.

II. EXPERIMENTAL SET UP

The schematic diagram of experimental set up is shown in figure 1. The engine is of single cylinder four stroke direct injection water cooled diesel engine. The engine has rated output 5.2 kw at speed 1500 rpm with compression ratio 17.5, injection pressure 180 kg/cm^3 and coupled with rope break dynamometer. The detailed specification of engine is given in table no. 1. Performance test are carried out on compression ignition engine using various blends of biodiesel and diesel as fuel.

III. GEOMETRY DEVELOPMENT AND MESHING OF COMPUTATIONAL DOMAIN

In present work Geometry has been modeled in preprocessor by using workbench tool Design modeler and then meshed In ICEM respectively . Figure2 shows the computational domain of two dimensional combustion chamber geometry counting inlet and exhaust ports. Both intake ports have been meshed with same orientation in the flow direction and they are joined with a cylindrical structured mesh in the zone upstream of the valves. The combustion chamber is bowl in piston type, which having a hemispherical groove on piston top. The geometry has been modeled at its zero crank angle position at TDC as shown in figure 2. In ICE it is necessary that for obtaining realistic simulations, computation must include combustion chamber geometry with inlet and exhaust valve. The computations performed on bowl in piston type combustion chamber reveled that, instead of suction stroke at the end of compression stroke the geometry plays important role to access the combustion when both the valves are closed.

IV. MODEL DEVELOPMENT

In this, the problem is to be solved as unsteady first order implicit with turbulence effects considered to simulate the combustion for CI, DI engine. The numerical methodology is segregated pressure based solution algorithm. For solving species, the discrete phase injection with species transport equation and finite rate chemistry reactions are used. The upwind scheme is employed for the discretization of the model equations. FLUENT uses a Control volume based technique to convert the governing equations to algebraic equations that can solve numerically. The governing equations for mass, momentum and energy equations used and appropriate initial boundary conditions were chosen for combustion analysis.

A. TURBULANCE MODEL

Turbulence is distinguished by fluctuation of velocity field. In this work well known RNG kε model is used for modeling turbulence. The RNG kε model was derived using a thorough statistical technique. It is analogous in form to the standard kε model but having an advantage to include effect of swirl, which is important for ICE combustion analysis. Transport equations for the RNG *k*e Model is defined as :

(1)
$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k-\rho\epsilon$$

(2)
$$\frac{\partial}{\partial t}(\rho\epsilon)+\frac{\partial}{\partial x_i}(\rho\epsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right]+C_{1\epsilon}\frac{\epsilon}{k}P_k-C_{2\epsilon}^{*}\rho\frac{\epsilon^2}{k}$$

Where,

$$C_{2\epsilon}^{*}=C_{2\epsilon}+\frac{C_\mu\eta^3(1-\eta/\eta_0)}{1+\beta\eta^3}$$

B. COMBUSTION MODELING

The ignition/combustion model is based on a modified eddy dissipation concept (EDC) which has been implemented into the CFD code. Multiple simultaneous chemical reactions can be modelled, with reactions occurring in the bulk phase (volumetric reactions) and/or on wall surfaces. The conservation equation takes the following general form:

(3)
$$\frac{\partial}{\partial t}(\rho Y_i)+\nabla\cdot(\rho\vec{v}Y_i)=-\nabla\cdot\vec{J}_i+R_i+S_i$$

Where Ri is the net rate of production of species i by chemical reaction and Si is the rate of creation by addition from the dispersed phase.

C. ENGINE IGNITION MODELING

For the present study the Auto-ignition model (Harden- burg model) is the most suitable one for simulating direct injection Diesel engine. The transport equation for an ignition species, *Yig* is given by:

(4)
$$\frac{\partial\rho Y_{ig}}{\partial t}+\nabla\cdot(\rho\vec{v}Y_{ig})=\nabla\cdot\left(\frac{\mu_t}{Sc_t}\nabla Y_{ig}\right)+\rho S_{ig}$$

where Y_{ig} is a "mass fraction" of a passive species representing radicals which form when the fuel in the domain breaks down. Sc_t is the turbulent Schmidt number. The term S_{ig} is the source term for the ignition species which has a form :

(5).
$$S_{ig}=\int_{t=t_0}^{t}\frac{dt}{\tau_{ig}}$$

D. SPRAY BREAK UP MODEL

In the present work TAB model is used. The TAB model is based on the analogy between an oscillating and distorting droplet and a spring mass system. The distorting droplet effect is considered in the present study. The equation governing a damped, force oscillator is,

(6)..... (6)
$$F-kx-d\frac{dx}{dt}=m\frac{d^2x}{dt^2}$$

Where is the displacement of the droplet equator from its spherical position and the coefficients of this equation are taken from Taylor's analogy:

(7)
$$\frac{F}{m}=C_F\frac{\rho_g u^2}{\rho_l r}\qquad \frac{k}{m}=C_k\frac{\sigma}{\rho_l r^3}\qquad \frac{d}{m}=C_d\frac{\mu_l}{\rho_l r^2}$$

,

where ρ_l and ρ_g are the discrete phase and continuous phase densities, u is the relative velocity of the droplet, r is the undisturbed droplet radius, σ is the droplet surface tension, and μ_l is the droplet viscosity. The C_F, C_k, and C_d are dimensionless constants.

E. DROPLET COLLISION MODEL

Droplet collision model includes tracking of droplets; for estimating the number of droplet collisions and their outcomes in a computationally efficient manner. The model is based on O'Rourke's method, which assumes stochastic approximation of collisions. When two parcels of droplets collide then algorithm further establish the type of collision. Only coalescence and bouncing outcomes are measured. The probability of each outcome is calculated from the collision Weber

number (8) and fit to experimental observations. The Weber number is given as:

$$\mathrm{We} = \frac{\rho v^2 l}{\sigma} \quad (8)$$

Where v is the relative velocity between two parcels and l is the arithmetic mean diameter of the two parcels. The state of the two colliding parcels is modified based on the outcome of
the collision.

F. WALL FILM MODEL

Spray wall interaction is an important element of the mixture creation process in diesel engines. In a DI engine, fuel is injected directly into the combustion chamber, where the spray can impinge upon the piston. The modeling of the wallfilm inside a DI engine is compounded by the occurrence of carbon deposits on the surfaces of the combustion chamber. This carbon deposit soak up the liquid layer. It is understood that the carbon deposits adsorb the fuel later in the cycle. The wallfilm model in FLUENT allows a single constituent liquid drop to impinge upon a boundary surface and form a thin film. Interactions during impact with a boundary and the criteria by which the regimes are detached are based on the impact energy and the boiling temperature of the liquid. The impact energy is defined by:

$$E^2 = \frac{\rho V_r^2 D}{\sigma}\left(\frac{1}{\min(h_0/D, 1) + \delta_{bl}/D}\right) \quad (9)$$

where r is the liquid density, Vr is the relative velocity of the particle in the frame of the wall, D is the diameter of the droplet, and s is the surface tension of the liquid. Here, dbl is a boundary layer thickness.

V. FIGURES AND TABLES

1: ENGINE SPECIFICATIONS

Table1. Engine Specifications.

ENGINE SPEED	1500 RPM
NUMBER OF STROKES	4
POWER	5.2 KW / 7 BHP
BORE	87.5 MM
STROKE LENGTH	110 MM
NO. OF CYLINDER	1
DYNAMOMETER	MECHANICAL LOADING
DRUM DIAMETER	35 CM
ORIFICE DIAMETER	20 MM
C_D (COEFFICIENT OF DISCHARGE)	0.6
INJECTION PRESSURE	180-200 BAR

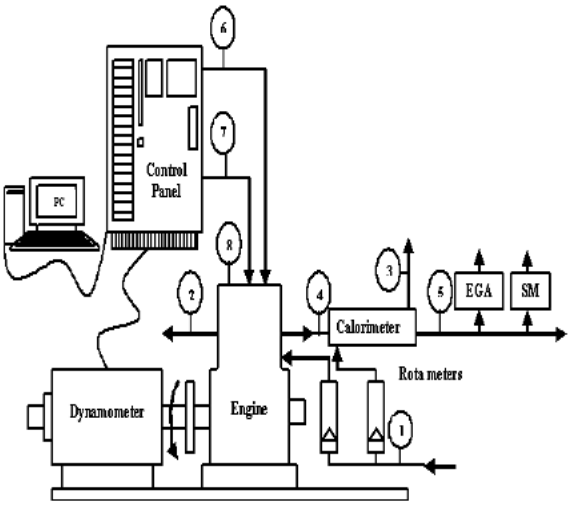

Fig. 1: Schematic diagram of experimental set up.

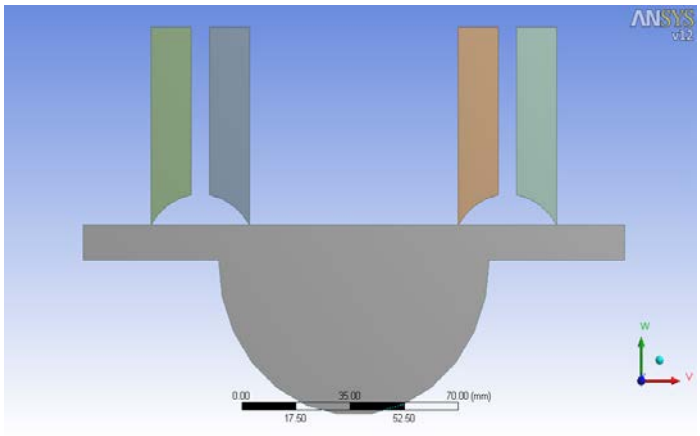

Fig. 2: Geometry of combustion chamber with valves

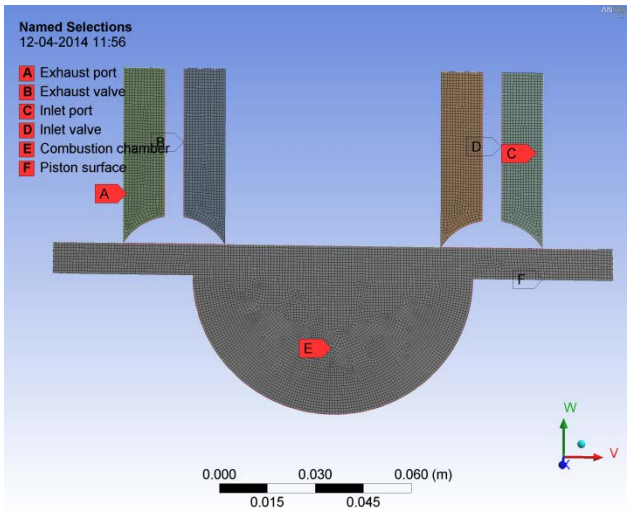

Fig.3: Mesh structure of computational domain for model geometry.

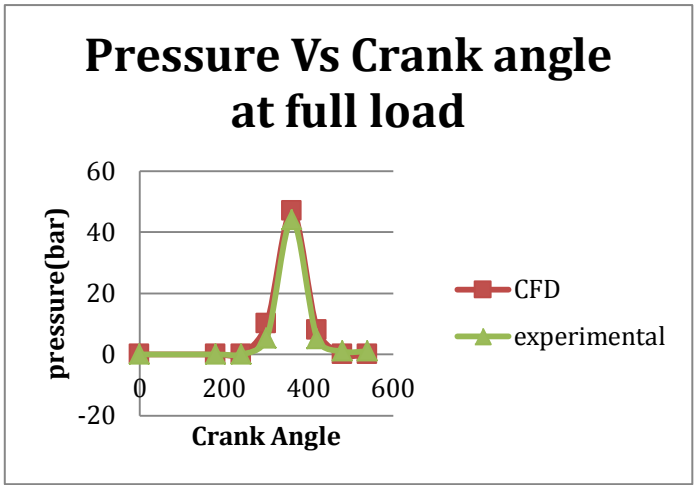

Fig. 4: comparisons between modeling and experimental pressure diagram of Biodiesel blend.

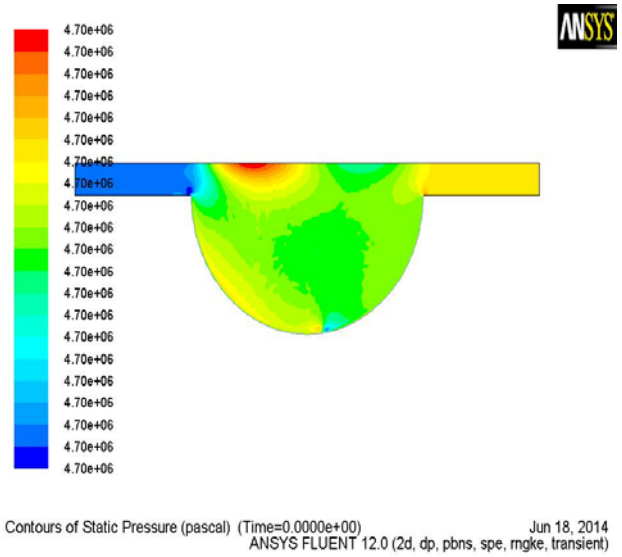

Fig. 5: Pressure distribution in (Bar) CA = 360 (maximum load) for biodiesel blend

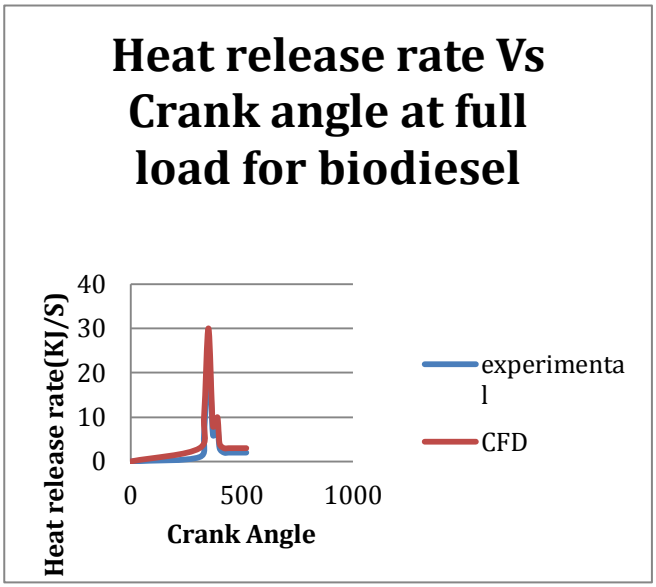

Fig. 6 : Comparison between modeling and experimental heat release rate diagram for biodiesel blend.

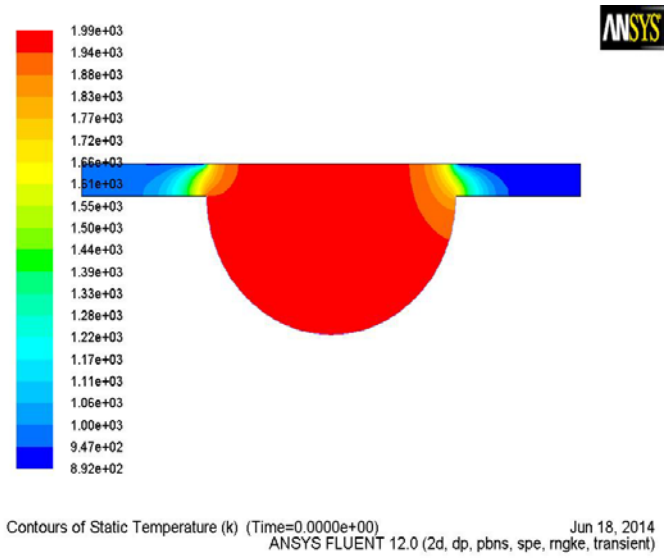

Fig. 7 :Temperature distribution in (K) CA = 360 (maximum load) for biodiesel blend

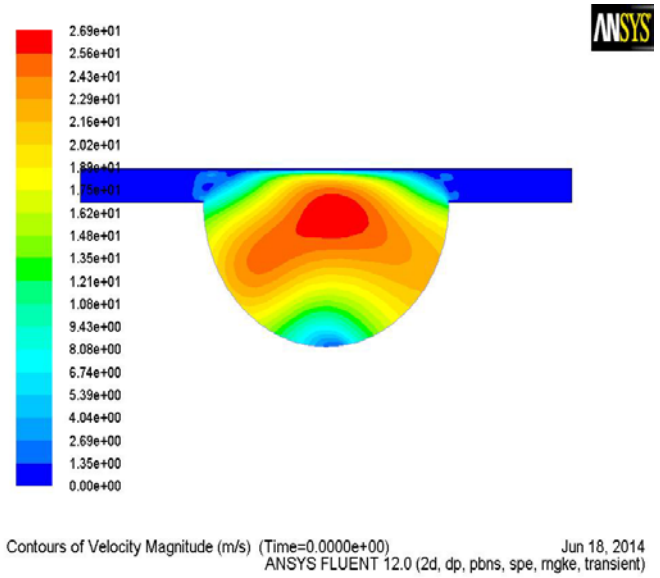

Fig.7: Gas velocity magnitude (m/s) CA = 360 (maximum load) for biodiesel blend.

VI. RESULT AND DISCUSSION

Figure shows modeling and experimental max. in cylinder pressure traces operating at full load condition. The modeled cylinder pressure data shows good agreement with experimental results. The maximum pressure rise depends upon the quantity of fuel vaporized during the delay time and occurs in the state of combustion, some degrees after the beginning of combustion. Note that modeling peak pressure is 47 bar at 360 degree CA, and experimental peak pressure is 45 bar at 360 degree CA for biodiesel. Therefore both scale and timing of occurrence of peak pressure are precisely predicted by the model.

The observed cylinder pressure profiles reflect the effect of in cylinder heat release rate. The heat release rate is determined from pressure data. Figure compares heat release rates computed from modeling and experimental pressure traces. The heat release rate decreases from the start of injection to the start of combustion which is ignition delay period because of the fuel evaporation occurring during this period. The first peak due to premixed combustion strongly depends on the amount of fuel that the prepared for combustion during the ignition delay period. The second peak due to diffusion combustion is controlled by the fuel air mixing rate. Diffusion combustion continues until combustion is completed. Note that, the peak modeling heat release rate is 30 J/s where as experimental peak heat release rate is 25 J/s at 364 degree CA for biodiesel . Figures shows the comparison of the measured and CFD modeled cylinder pressures.. The peak pressure and the pressure gradient over the combustion period produced by CFD simulation match closely with experimental. The peak cylinder pressure is over predicated by about 4%. Also from heat release rate graph it is shown the experimental validation of combustion. Figure shows the temperature contour diagram at maximum loads. This shows that increasing the load which corresponding to increasing the fuel mass flow rate results in increasing the combustion temperature. Figure shows that the temperature of the air at the end of compression is sufficiently high for the droplets of fuel to vaporize and ignite as they enter the cylinder. Max. temperature occurs in case of biodiesel is 1990 k .

Fig. shows velocity magnitude contour for biodiesel and diesel fuel respectively. Velocity magnitude of diesel is 18 m/s while biodiesel has 26 m/s.This difference is due to the variation in viscosity according to change in temperature and pressure of diesel and biodiesel. Also there are changes in velocity contour of diesel and biodiesel because of theire chemical characteristics.

VII. CONCLUSION

The CFD code FLUENT has been used to simulate the combustion characteristics of direct injection diesel engine fueled with biodiesel blend. The fluid flow in DI diesel having bowel in piston with turbulence and combustion processes modeled with sufficient generality to include spray formation, delay period, chemical kinetics and onset of ignition. A good agreement between the modeling and experimental data ensures the accuracy of the numerical predictions collected with this work. The model is validated through the comparison of the predicted p-θ curve with the experimental p-θ curve. It shows that CFD can be a reliable tool for the combustion modelling of CI

engine fueled with biodiesel blend. Also Pongamia Pinnata blend can be a suitable replacement to diesel, hence it can be used as a alternative fuel for a future work.

ACKNOWLEDGMENT

With immense pleasure and great respect I take this opportunity to express my deep sense of gratitude towards my project guide, **Mr. A.V.Kolhe**, Asst. Professor, Department of Mechanical Engineering, K.I.T.S, Ramtek for invaluable guidance, inspiration, constant encouragement and motivation throughout the project work. I am very thankful to all the respected, Professor, Department of Mechanical Engineering, K.I.T.S, Ramtek, who helped me to understand basic methodologies of my project.

REFERENCES

[1] Kongre V. U., Sunnapwar K. V.,2010, "CFD modeling and Experimental Validation of Combustion in Direct Ignition Engine Fueled with Diesel", international journal of applied engineering research, Dindi gul,Volume 1 , No 3, 2010.

[2] Patil D. ,Arakerimath R. R. , " Performance Analysis of Multi-cylinder C.I. Engine by using Karanja Bio-diesel" Vol. 1, Issue 4, pp.1755-1763 .

[3] Mourya D. , Roy V. "Combustion modeling and simulation of combustion emissions for diesel engine operating on the blends of Jatropha biodiesel and diesel". International Journal of Engineering Science and Technology (IJEST) .

[4] Ng H. K., Gan S. , Jo-Han Ng , Pang K.M., 2012.,"Simulation of biodiesel combustion in a light-duty diesel engine using integrated compact biodiesel–diesel reaction mechanism".www.elsevier.com/locate/apenergy.

[5] Jonathon P. McCrady, Valerie L. Stringer, Alan C. Hansen, and Chia-fon F. Lee., 2007, "Computational Analysis of Biodiesel Combustion in a Low-Temperature Combustion Engine using Well-Defined Fuel Properties". 07PFL-655.

[6] Jia M., Xie M., Wang T., Peng Z.,2011, "The effect of injection timing and intake valve close timing on performance and emissions of diesel PCCI engine with a full engine cycle CFD simulation" .Applied Energy 88 (2011) 2967–2975.

[7] Hussain S.M., Dr. Kumar B.S.P, ,Dr. Reddy K.V.K., 2012, "CFD analysis of combustion and emissions to study the effect of compression ratio and biogas substitution in a diesel engine with experimental verification" .Vol. 4 No.02 February 2012.

[8] Tarek M. Belal, El Sayed M. Marzouk, Mohsen M. Osman.,2012, "Investigating Diesel Engine Performance and Emissions Using CFD" Energy and Power Engineering, 2013, 5, 171-180.

[9] T. Mirunalini, R.Anand and N.V.Mahalakshmi,2006, "Jatropha Oil As a Renewable Fuel In a Diesel Engine" BSME-ASME International Conference on Thermal Engineering 20-22 December, 2006.

[10] S. Naga Sarada, M.Shailaja, A.V. Sita Rama Raju1, K. Kalyani Radha,2010," Optimization of injection pressure for a compression ignition engine with cotton seed oil as an alternate fuel". Vol. 2, No. 6, 2010, pp. 142-149.

[11] Lokanadham R., Ravindranath K., "Analysis On Internal Combustion Engine Using Biodiesel As Alter Native Fuel Compared To Conventional Diesel". International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 1, January- 2013 .

[12] Sharma S.C., Anand C.N.T.and Ravikrishn V.R. "A methodology for analysis of diesel engine in-cylinder flow and combustion". Progress in Computational Fluid Dynamics, Vol. 10, No. 3, 2010.

[13] John Agudelo, Andrés Agudelo, Pedro Benjumea, " Study of diesel sprays using computational fluid dynamics"., Rev. Fac. Ing. Univ. Antioquia No 49. pp. 61-69. Sept. 2009.

[14] Helgi Fridriksson,Benget Sunden,Shahrokh Hajireza,and Martin Tuner, "CFD Investigation of Heat Transfer in a Diesel Engine with Diesel and PPC Combustion Modes" JSAE 20119177 ,SAE 2011-01-1838.

[15] Rapalis P.,Lebedeva G.,Gudaityte I, "Comparative Analysis Of Diesel Engine Mathematical Modeling Packages For Practical Use On Transport Diesel Engine Operating On Biodiesel". 2013.

[16] T. Lucchini, G. Montenegro, G. D'Errico,2007, "CFD Simulations of I.C. Engines Combustion, InternalFlows, integrated 1D-MultiD simulations",http://www.engines.polimi.it tommaso.lucchini@polimi.it.

[17] Jayashankara B., Ganesan V.,2009, "Effect of fuel injection timing and intake pressure on the performance of a DI diesel engine – A parametric study using CFD" Energy Conversion and Management 51 (2010) 1835–1848.

[18] Reinhard TATSCHL and Peter PRIESCHING,2009, "3D-CFD Simulation of DI-Diesel engine combustion and pollutant formation". UNIVERSITY OF PITESTI SCIENTIFIC BULLETIN FACULTY OF MECHANICS AND TECHNOLOGY AUTOMOTIVE series, year XVI, no.20 .

[19] Renganathan Manimaran1, Rajagopal Thundil Karuppa Raj2 and Senthil Kumar K.3,2012,"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) Res.J.Recent Sci..

[20] Kong S.C.,Reitz R.D.,2002,"Application of detailed chemistry and CFD for predicting direct injection HCCI engine combustion and emissions",Proccedings of the combustion institute,vol.29,pp 663-669.

[21] Kong S.C.,Senecal P.K. and Reitz R.D.,2000,"Development in spray modeling in diesel and Direct-Injection Gasoline engines".Combustion and flame,vol.121,pp 453-470.

[22] Kong S.C.,Kim H.,Reitz R.D.,Kim Y.,2007,"Comparision of diesel PCCI combustion simulation using a representative interactie flamelet model and direct integration of CFD with detailed chemistry".Transactions of ASME,vol.129,pp 252-260.

[23] C.Hergart,H.Barths,N.Peters,1999 "Modeling the combustion in a small-bore diesel engine using a method based on representative flamelets" vol. SP-1480.

[24] Watanabe T.,Daidoji S.,Keshav S.V.,2000,"Relationship between visibal spray observations and DI diesel engine performance".Transaction of ASME,vol.122,596-602.