Modeling of Compression Ignition Engine Using Computational Fluid Dynamics: A Review

Vishal C. Momale[1], S.A. Giri[2]

M. Tech Scholar, Department of Mechanical Engineering, Ramdeobaba College of Engineering and Management,

Nagpur, India[1]

Assistant Professor, Department of Mechanical Engineering, Ramdeobaba College of Engineering and Management,

Nagpur, India[2]

Abstract: One of the important problems for direct and in-direct injection diesel engines is meeting the stringent emissions norms. In case of Diesel engine trade-off is present between soot and NOx. Researchers are striving for Improvement in the field of compression ignition engine, current research scenario is mainly focused on enhancing the combustion efficiency. Combustion efficiency can be increased by optimizing the injection timings, by variable compression ratio, increasing the swirl and turbulence in the combustion chamber, testing of different blends of diesel as alternative for diesel, optimizing the shape of piston head and many more. For analyzing these different factors, cost effective and less time consuming way simulation is the. The increasing confidence of using CFD techniques in engine designs depends on continuously improved CFD codes in terms of their fidelity and ease-of-use. Two pioneering works including the KIVA family of CFD codes and the STAR-CD family of CFD codes have made tremendous contributions in bringing CFD techniques to engine simulations. The other main competitors in commercial engine CFD software are FIRE, FLUENT, CFX and VECTIS.

KeyWords: Computational Fluid Dynamics, CI engine.

I. INTRODUCTION

Internal Combustion (IC) engines plays vital role in the field of modernized agriculture sector, propulsion sector, power, transportation of passenger and goods and industry, it becomeindispensable part of industrial growth. As diesel engine has high-thermal efficiency and cost-effectiveness. It is being widely used. it is indispensable source of power for a variety of applications owing to several inherent advantages. Dieselengines in the range of 15–20 HP are used extensively for power generation in the agricultural and industrial sectors. In the past, these engines were not subject to emission legislation. However, the current emission standards imposed are severe, and are posing a challenge to diesel engine manufacturers. The reason for this lies in their high NOx and particulate emissions. Recent advances in multidimensional modelling of flow and combustion processes are significantly aiding in the development of diesel engines with low emissions. This has in turn been aided by the availability of large computing power and memory in recent years. A number of commercial and open-source codes have come up for multidimensional CFD analysis of engines.

II. LITERATURE REVIEW

This section present the detailed study carried out on the models used for turbulence, combustion, spray, and auto-ignition. Review is also carried out to know the different software's which can be used for CI engine simulations, outcomes and accuracy in the previous work carried, methodology for the combustion modeling.

C.S. Sharma et al [1] developed combustion modeling methodology for the analysis in-cylinder processes and combustion for CI engine. Firstly geometry was created from the engine data for the different parts of engine, suction stroke was simulated using commercial code AVL FIRE, and when the valves are closed i.e. during compression and power stroke an open-source code KIVA-3V is used for simulation. To map a generalized 3-D CFD simulation solution from an unstructured mesh used in AVL FIRE to a structured mesh used in KIVA-3V algorithm is first prepared for getting initial boundary conditions for the closed valve simulation i.e. in-cylinder simulation. For combustion modeling of CI engine, an integrated KIVA-3V code is prepared by using two well-validated models in the standard code:

- for simulation of diesel auto-ignition under conditions of high temperature and pressure Shell hydrocarbon auto-ignition model
- For diesel combustion Characteristic-time model is used

The integrated code is validated and calibrated with experimental pressure measurements in a naturally aspirated DI CI engine.

Patil Pradip kailas et al [2] done the geometry modeling in Gambit and combustion phenomena and modeling were done in ANSYS FLUENT. Different compression ratio were used for Analysis. Tilting cylinder mechanism is used to get different compression ratio, Experimentation were carried out on VCM. Numerical modeling is done in CFD and results are obtained and The results show values of peak pressure, peak temperature and volume at each

crank angle which is the compare with experimental values and values are in good agreement. Combustion parameters of a compressed ignition engine can be predicting using the developed model. By varying the compression ratio the peak pressure, and peak temperature were compare with results of another compression ratio results and optimum compression ratio obtained.

S G Karunanidhi et al [3] developed 2D model in GAMBIT 2.4.6 and meshing is also done using GAMBIT 2.4.6 and the combustion modeling were done in ANSYS FLUENT 14.5. Pressure and temperature values obtained from CFD results are comparable to theoretical values. It shows that the combustion modelling of CI engine using CFD can be a reliable tool.

Shaik Magbul Hussain et al [4] carried out Biogas-Diesel dual fuel combustion modeling in CFD using FLUENT, for the analysis of of Biogas substitution on turbulent kinetic energy(TKE) for five compression ratios also same analysis carried out for Turbulent Dissipation Rate, Combustion flame velocity, and NOx formation. RNG $\kappa-\varepsilon$ model were used for modeling Turbulence, for the dual fuel analysis the turbulence model is modified. Meshing is carried out using GAMBIT for the computational domain of the combustion chamber, by tetrahedral element using cooper tool. The analysis were carried out and the effect of compression ratio in dual fuel mode is studied, compression ratios values along with Bio-gas substitutions were varied. Good agreement between the predicted results and experimental results which is the validation of CFD results.

R. Bhoobathi et al [5] CFD simulation was carried out with following methodology Standard k-ε model used for turbulence, Kelvin Helmholtz (K-H) and Rayleigh Taylor (R-T) mechanism used for spray modeling, NOx modeling is also done, Initial boundary condition for pressure 100 kPa and temperature 350 K was used, Motion of valve is not considered, Simulation is carried out for compression and expansion strokes physical modeling of Fuel injector was not used (preset in model).

B. Jayashankara et al [6]CFD simulation was carried out with following methodology Pre-processor GAMBIT is used to create computational domain, STAR-CD is used for the solution of governing equations and post processing the results. RNG k-ε model with standard wall function used for turbulence. Combined laminar and turbulent characteristic time combustion (CTC) model along with for auto-ignition Shell auto-ignition model is used to characterize ignition and combustion. To characterize droplet break-up, the Reitz–Diwaker model is used this also considered rebound boundary condition to increase accuracy at the walls. Emissions modeling is also done. The first order upwind differencing scheme (UD) was used, PISO algorithm was used. When the injection timing is retarded there was increase in pressure, temperature, heat release and Nox.T. Korakianitis et al [7] used many approaches have been proposed to predict emissions engines. They found that each approach has

some good points and limitations. Direct coupling between physical processes and the chemical process is mostly discarded, the pre-integrated approach is used which is firstly developed. Multi-dimensional modeling and detailed chemical kinetics reaction modeling both have some advantages hence they are coupled and used. Thermal dynamics and fluid dynamics processes of engine operation are modeled in KIVA-3V is used to model.

Ajay V. Kolhe et al [8] simulated combustion phenomena of direct injection (DI) diesel engine fuelled with biodiesel blend using CFD code FLUENT. For modeling turbulence, Renormalization Group Theory (RNG) k-ε model were used. The sub models such as for spray modeling Taylor Analogy Breakup (TAB) model and droplet collision model were used. For combustion modeling, finite rate chemistry model and species transport were used. The results obtained from CFD modeling were validated with experimental investigation. The fluid flow in DI diesel has bowel in piston with turbulence and combustion processes modelled with sufficient generality to include spray formation. The model was validated through the comparison of the predicted P-θ curve by the CFD modeling with the experimental P-θ curves and heat release rate. It shows for the combustion modelling of CI engine fuelled with biodiesel blend that CFD can be a reliable tool.

G. De Paola et al [9] developed combustion modeling method for simulation of direct injection heavy-duty CI engine. In his methodology, computational fluid dynamics (CFD) solver coupled with First-order elliptic Conditional Moment Closure (CMC), has been employed. For engine modelling 3-D structured finite difference CMC mesh (grid) has been interfaced to an unstructured finite volume CFD grid (mesh). The use of a moving CMC grid to consider the changes in the control volume or computational domain during the compression and expansion phases which has been obtained using an algorithm element(cell)addition and removal and to consider the additional convection termdue to the CMC cell movement. Special attention has been taken for the wall heat transfer and the boundary conditions. An operator splitting formulation were used to couple the CMC equations accurately. A CMC domain reduction of the 3-D problem to 2- and zero dimensions through appropriate volume integration of the CMC equation has been used in terms of accuracy of solution and time required for the computation. During transient calculations where the probability density function of the mixture fraction changes drastically with time as during fuel injection initialization of the CMC domain in conserved scalar space. At full and half load the results obtained for Pressure from simulation are in good agreement with the experimental pressure traces. Hence analysis can be used for extensive work.

W.M. Yang et al [10] presented work on the performance analysis of IC engine when blend of gasoline and diesel is used. gasoline and diesel are generally used for the IC engine, because it is readily available, hence their

combined effect on the performance of engine is analyzed using CFD which will save time and money for the analysis, simulation are carried out on diesel engine and different blends were used. these blend % are, 0%, 10%, 20%, 30%, 40% at the varying load such as 10%, 50% and 100%. KIVA4-CHEMKIN code for detailed chemistry calculations. In this mechanism, iso-octane and n-heptane was used for the gasoline and diesel respectively for its representation. Comparing the results among different blend conditions, it is found that the ignition delay period is extended by increasing the % gasoline in blend fuels. However, this will cause diverse effects on engine performance when load on the engine were changedand it is observed for pure diesel fuel condition at low loadis best which achieves a better performance; in contrast, a better performance is obtained by blend fuels at medium and high loads, with slightly higher NOx emission level.
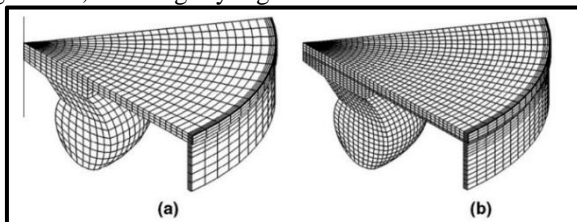

Fig: Medium mesh and fine mesh at TDC

S. Jafarmadar et al [11] worked on the split injection which will make chance to shift the trade-off curve for the soot and NOx closer to origin. The main problems is reduce the trade-off between soot and NOx which are major pollutant emission from DI and IDI diesel engines. Split injection is one of the most powerful tools. For the present work engine specifications of 5.9 kW maximum engine power and engine speed of 730 rpm has been investigated and the combustion process and emissions against split injection are observed for a cylinder IDI diesel engine were analyzed. The analysis of injection timing and split injection scheme parameters, such as the delay dwell and the fuel quantity injected between Injections pulses are analyzed. 3 different split injection schemes, which are 10, 20 and 25% of total fuel injected in the second pulse, were taken. The numerical results show total soot and NOx emissions reduces effectively in IDI diesel engines with 25% of total fuel injected in the second pulse which is accompanied with the 20ºCA delay dwell between injections. The predicted values are in good agreement with the corresponding experimental data. Hence the model can be used as reliable tool that can use for exploring new engine insights.

F. Payri et al [12] developed methodology for the in-cylinder flow simulations. 3-D flow visualizations and Its calculation during the intake and compression stroke of a four-valve DI Diesel engine have been carried out with different combustion chambers shapes. For the validation limited number of calculations of the compression stroke were first performed and then CFD is used for the representation of the in-cylinder flow. laser Doppler velocimetry measurements is used to compare The calculated flow distribution of field in three different combustion chambers. the comparison shows that the 3-D model is produce reasonably accurate results for crank-angles around TDC. It performs better for low swirl combustion chambers while turbulence velocities are less than the predicted. The results conclude that the piston geometry had effect on the in-cylinder flow during the suction stroke and at the start of the compression stroke. However, the bowl shape piston geometry found significant role near TDC and at the start of the expansion stroke it will control both the ensemble-averaged mean and the turbulence velocity fields.

CONCLUSION

The review is carried out on the recent research in the field of CI engine, the major factor for reducing NOx and P.M. is the optimized injection timings. For modeling combustion different codes are available amongst ANSYS FLUENT code is the reliable tool for modeling CI engine combustion. The methodology needs to be developed for combustion modelling, and effect of different factors can be studied using simulation.

REFERENCES

[1]. Sharma C.S. et al; 'A methodology for analysis of diesel engine in-cylinder flow and combustion', Progress in Computational Fluid Dynamics, Vol. 10, No. 3, pp.157–167.
[2]. Patil Pradip kailas et al; Study of combustion in di diesel engine for different compression ratios using experimental and cfd approach. International Journal of Research in Engineering and Technology, Jun 2014 , Volume: 03 pp.1-5
[3]. S G Karunanidhi et al; 'CFD Studies of Combustion in Diesel Engine.   International Journal of Engineering Research and Applications', Aug 2013 , Vol. 3, Issue 4, pp.827-830
[4]. Shaik Magbul Hussain et al; 'CFD analysis of combustion and emissions to study the effect of compression ratio and biogas substitution in a diesel engine with experimental verification', International Journal of Engineering Science and Technology , Feb 2012, Vol. 4 No.02, pp.473-492
[5]. R. Bhoobathi et al; 'Diesel Engine Combustion Simulation using Computational Fluid Dynamics', AMAE International Journal on Production and Industrial Engineering, Dec 2010 Vol. 01, pp. 17-21
[6]. B. Jayashankara, V. Ganesan et al ; '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, March 2010, pp.1835–1848
[7]. T. Korakianitis et al; 'Combustion-response mapping procedure for internal-combustion engine emissions', ELSEVIER, Applied Energy 156, Jun 2015, pp.149-158.
[8]. Ajay V. Kolhe et al; 'Combustion Modeling with CFD in Direct Injection   CI   Engine Fuelled with Biodiesel', Jordan Journal of Mechanical and Industrial Engineering, Feb 2015 , Volume 9 Number 1, pp61- 66
[9]. G. De Paola et al; 'Diesel Engine Simulations with Multi-Dimensional Conditional Moment Closure', Taylor & Francis Group,Combustion Science And Technology 180, pp.883–899.
[10]. W.M. Yang et al; 'Modeling on blend gasoline/diesel fuel combustion in a direct injection diesel engine', ELSEVIER, Applied Energy, Aug 2014, pp.1-7.
[11]. S. Jafarmadar et al; 'Multi-Dimensional Modeling of The Effects of Spilt Injection Scheme On Performance And Emissions of IDI Diesel Engines', International Journal of Engineering, Vol. 25, No. 2, June 2012 pp.135-145.
[12]. F. Payri et al; 'CFD Modeling of The In-Cylinder Flow In Direct-Injection Diesel Engines', ELSEVIER , Computers & Fluids 33, 2004, pp.995–1021