IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology
Vol. 3, Issue 6, June 2016

Compression Ignition Engine Simulation using Computational Fluid Dynamics

V.C. Momale[1], S.A. Giri[2], M.S. Yadav[3]

M. Tech Scholar, Department of Mechanical Engineering, Shri Ramdeobaba College of Engineering and Management, Nagpur, India[1]

Assistant Professor, Department of Mechanical Engineering, Shri Ramdeobaba College of Engineering and Management, Nagpur, India[2]

M. Tech Scholar, Department of Mechanical Engineering, Shri Ramdeobaba College of Engineering and Management, Nagpur, India[3]

Abstract: Combustion efficiency and emission levels are prime concern for the compression ignition engines, which should have high combustion efficiency and meet the stringent emission norms. Combustion is very complex phenomenon which cannot be analyzed easily with the analytical method or different visualization technique. Computational Fluid Dynamics (CFD) has becoming useful tool in understanding the fluid dynamics of IC Engines for design purposes. The methodology for CFD analysis involves use of CATIA V5 for geometry creation. ANSYS FLUENT is used as solver. CFD analysis is able to predict in cylinder combustion, peak pressure and peak temperature hence proposed combustion model can be utilizes for combustion analysis.

Keywords: Computational Fluid Dynamics, CI engine, Combustion modeling.

I. INTRODUCTION

Direct Injection engines are widely used in low, medium and high duty applications. For engine, combustion efficiency and emission levels are prime concern which should meet the stringent emission norms. Out of these combustion is very complex phenomenon which cannot be analyzed easily with the analytical or old technique.

The complexity in the combustion analysis is very high because combustion mainly depends on the many factors, these factors are shown in Figure 1 and these factors affects the combustion and ultimately the output parameters such as peak pressure, peak temperature, emission level, detonation tendency and many more. These factors and their effect on combustion needs to be critically analyzed by developing reliable combustion model. Computational Fluid Dynamics (CFD) has emerged as a useful tool in understanding the flow physics fluid dynamics of IC Engines for design purposes.

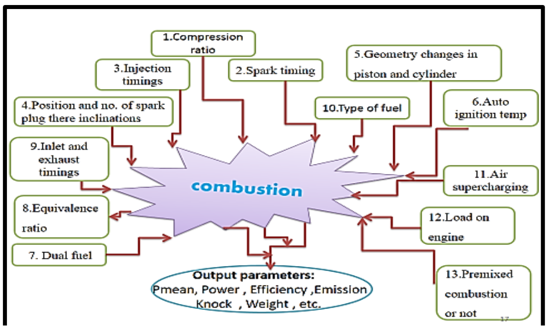

Figure 1: Combustion Phenomena Dependency

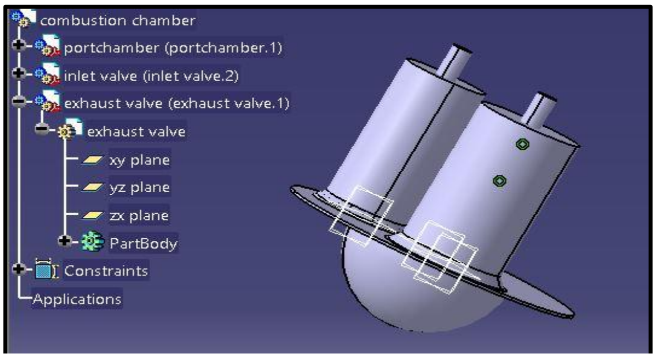

Figure 2: computational domain

T. Korakianitis et al [1] used many approaches which have been proposed to predict emissions engines. 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. 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. W.M. Yang et al [3] presented work on the performance analysis of IC engine when blend of gasoline and diesel is used. 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 is used for modeling. S G Karunanidhi et al [4] developed 2D model in GAMBIT

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
Vol. 3, Issue 6, June 2016

2.4.6 and meshing is also done using GAMBIT 2.4.6 and the combustion modeling were done in ANSYS FLUENT 14.5. S. Jafarmadar et al [6] worked on the split injection which will make chance to shift the trade-off curve for the soot and NOx closer to origin. R. Bhoobathi et al [8] 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 [9] 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 model with Shell auto ignition model is used to characterize ignition and combustion. The Reitz–Diwaker model is used to characterize droplet break-up, with rebound boundary condition at the walls. Emissions modeling is also done. The first order upwind differencing scheme (UD) was used, PISO algorithm was used. C.S. Sharma et al [10] developed combustion modeling methodology for the analysis in-cylinder processes and combustion for CI engine. Suction stroke was simulated using commercial code AVL FIRE, and power stroke an open-source code KIVA-3V is used for simulation. Algorithm is first prepared to map a generalized 3-D Computational Fluid Dynamics solution from an unstructured mesh in AVL FIRE to a structured mesh in KIVA-3V to for getting initial boundary conditions for the closed valve simulation. F. Payri et al [15] developed methodology for the in-cylinder flow simulations. 3-D flow visualizations and It's calculation during the intake and compression stroke of a four-valve DI Diesel engine have been carried out with different combustion chambers shapes.

II. Combustion Modeling Methodology

Methodology starts with geometry (computational domain) creation using CATIA V5 as shown in figure 2, after this Geometry decomposition has done which converts geometry into hybrid topology and different zones are created, then discretization of computational domain into smaller control volume. Sector of 30° is modeled and computational domain is meshed which is shown in figure 3. The mesh parameters such as minimum orthogonal mesh quality 0.265 which should be in the range of 0 to 1 and beyond 0.2 it is acceptable. Maximum aspect ratio is 56.7. It should be below 100 in the near wall range. An engine model to be simulated in ANSYS FLUENT is meshed with a hybrid topology for incorporating dynamic meshing. A multi-block methodology is used to split the domain into four major zones i.e. Chamber, piston layer, ports and valve layer. The zones adjacent to reciprocating boundaries such as the piston and valves are meshed with quadrilateral cells (structured mesh) and layered. Tetrahedral cells (unstructured mesh) are used in the chamber zone because the valves are moving in this zone and cells deform and must be re-meshed. Simulation is carried out in between the inlet valve closing (IVC) angle and exhaust valve opening (EVO) angle. FLUENT is used as solver, First order implicit Pressure based transient formulation is used. Standard k-ε model is used for the turbulence modeling, eddy breakup model and species transport laminar finite rate model for combustion modeling was considered. The governing equations for mass, momentum and energy were used and initial boundary conditions were chosen such that which converge the solution with minimum iteration for the combustion analysis. Under relaxation parameters and residuals are set. Second order upwind scheme is used for the momentum, energy, species, and turbulent kinetic energy. For the turbulent dissipation rate first order upwind scheme is used. Control volume technique is used by the solver for the conversion of differential governing equations into algebraic equation. For the pressure velocity coupling PISO algorithm is used.

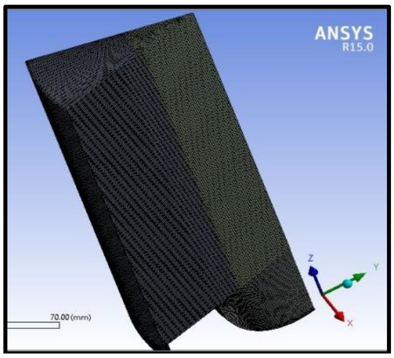

Figure 3: Meshed Geometry

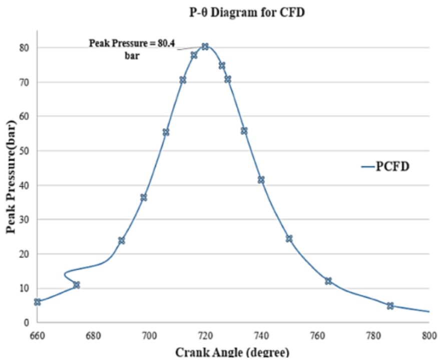

Figure 4: Graph showing P-θ Diagram from simulation

III. RESULT AND DISCUSSION

Figure 5 and figure 6 shows contours of Peak Pressure at 705° and 720° CA respectively. When fuel is injected at 23° BTDC the Peak pressure developed is 52.9 bar and 80.4 bar at 705° and 720° CA respectively. Figure 7 shows how the peak Pressure inside the cylinder increases with further compression of piston and combustion of fuel. Figure 8 shows the Peak temperature developed at various crank angle. It is 994 k at 705°CA and 1720 k at 720° CA. The value of temperature at 705° CA indicate combustion yet to be started and it is started after delay period. Which

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
Vol. 3, Issue 6, June 2016

is indicated by the contour of temperature at 720° CA. Figure 4 shows P-θ Diagram obtained using CFD simulation it shows pressure is rising with compression and after combustion rate of increase in pressure rise is very high.

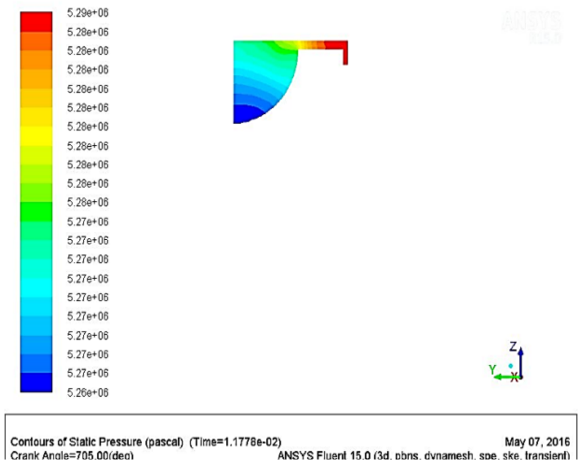

Figure 5: Contour Peak Pressure at 705 crank angle

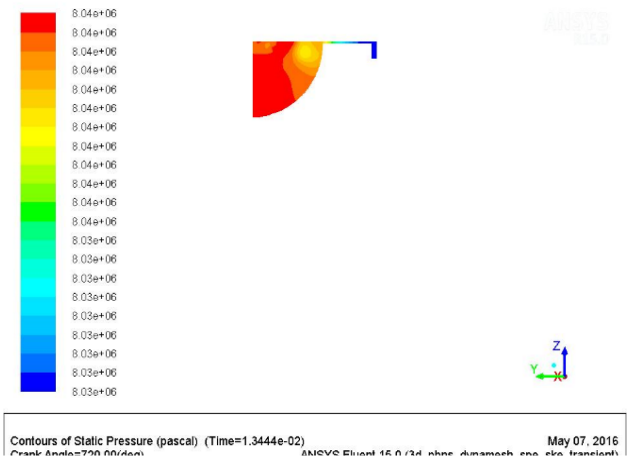

Figure 6: Contour Peak Pressure at 720 crank angle

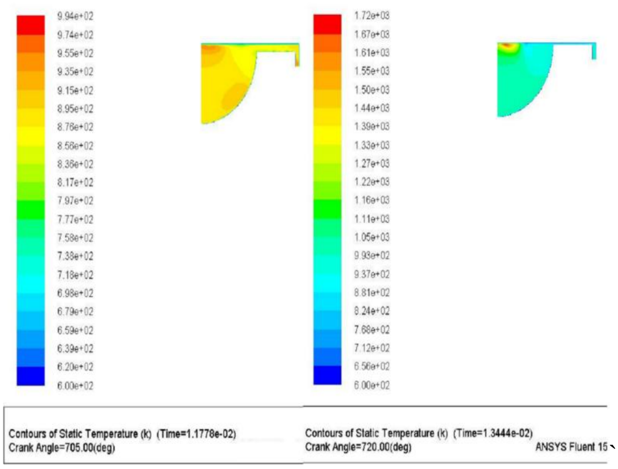

Figure 7: Contour Peak Pressure at various crank angles

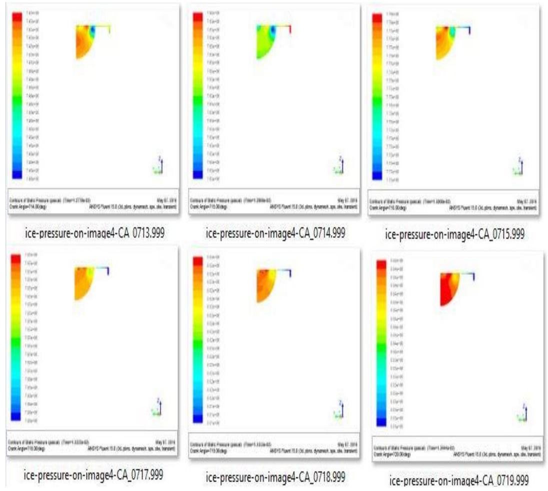

Figure 8: Contour Peak Temperature at various C.A.

IV. CONCLUSION

Compression ignition engine simulation is carried out. The CFD code FLUENT was used to simulate the combustion characteristics of direct injection diesel engine. This model is able to predict in cylinder combustion, peak pressure and peak temperature. Results were obtained at each crank angle during compression and expansion stroke. The reliability of results can be obtained by validating the simulation results with analytical or experimental work.

REFERENCES

[1] T. Korakianitis, S. Imran, N. Chung, H. Ali, D.R. Emberson and R.J. Crookes, "Combustion-response mapping procedure for internal-combustion engine emissions," Applied Energy, 2015, v. 156, pp. 149-158.

[2] P.K. Patil, H.K. Wagh, V.M. Patil, and A.H. Kumbhar , "Study of combustion in DI diesel engine for different compression ratios using experimental and CFD approach," International Journal of Research in Engineering and Technology, 2014, v. 3 pp. 1-5.

[3] W.M. Yang, and S.K. Chou "Modeling on blend gasoline/diesel fuel combustion in a direct injection diesel engine," Applied Energy, 2014, pp. 1-7.

[4] S.G. Karunanidhi, C.R. Melvinraj, K.P. Sarath Das and G.Subba Rao, "CFD Studies of Combustion in Diesel Engine," International Journal of Engineering Research and Applications, 2013, v. 3, Issue 4, pp. 827-830.

[5] S.M. Hussain, B.S. Kumar and K.V. Reddy, "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, 2012, v. 4 n. 2, pp. 473-492.

[6] S. Jafarmadar, A. Zehni, "Multi-Dimensional Modeling of the Effects of Spilt Injection Scheme On Performance And Emissions of IDI Diesel Engines," International Journal of Engineering, 2012, v. 25, n. 2, pp. 135-145.

[7] R. Bhoobathi, A. Shaik, and R. Rudramoorthy, "Diesel Engine Combustion Simulation using Computational Fluid Dynamics," AMAE International Journal on Production and Industrial Engineering, 2010, v. 01, pp. 17-21.

[8] B. Jayashankara and V. Ganesan, "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 , 2010, v. 51, pp. 1835–1848.

[9] C.S. Sharma, T.N.C. Anand and R.V. Ravikrishna, "A methodology for analysis of diesel engine in-cylinder flow and combustion," Progress in Computational Fluid Dynamics, 2010, v. 10, n. 3, pp. 157–167.

IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology
Vol. 3, Issue 6, June 2016

[10] P.A. Lakshminarayanan, and Y.V. Aghav, "Modeling Diesel Combustion," Mechanical Engineering Seriesspringer,2010
[11] B.L.Singhal, "Internal Combustion Engines," Tech-Max Publication, Pune, 2010.
[12] G.De Paola, E. Mastorakos, Y.M. Wright and K. Boulouchos, "Diesel Engine Simulations with Multi-Dimensional Conditional Moment Closure," Taylor & Francis Group,Combustion Science and Technology, 2008, v.180, pp. 883–899.
[13] R.S. Khurmi and J.K. Gupta, " Machine Design" Eurasia Publishing house, 2005.
[14] F. Payri, J. Benajes, X. Margot and A. Gil, "CFD Modeling of The In-Cylinder Flow in Direct-Injection Diesel Engines," Computers and Fluids, 2004, v.33, pp. 995–1021.
[15] D. Veynante and L. Vervisch, "Turbulent Combustion Modeling," Progress in energy and combustion science, 2002, v.28, pp. 193-266.
[16] H.K. Versteeg, and W. Malalasekera, "An Introduction to Computational Fluid Dynamics the finite volume method," Longman scientific and technical, England, 1995.
[17] A. Arnold, F. Dinkelacker, T. Heitzmann, P. Monkhouse, M. Schafer,V. Sick And J. Wolfrum "DI Diesel Engine Combustion Visualized ByCombined Laser Techniques," Twenty-Fourth Symposium (International) on Combustion, Combustion Institute, 1992, pp.1605-1612.
[18] [18] Vishal C. momale and S.A. giri, " Modeling of Compression Ignition Engine Using Computationa Fluid Dynamics: A Review," International Advanced Reasearch journal in science, Engineering And Technology, 2016, v.5, pp. 199-201