CFD Analysis on Performance and Emission in a Direct Injection Diesel Engine at Different Piston Bowl Re-Entrant Angle

Jinuchandran
PG Scholar
Department of Mechanical Engineering
Mar Athanasius College of Engineering, Kothamangalam

Jyothis. S
PG Scholar
Department of Mechanical Engineering
Mar Athanasius College of Engineering, Kothamangalam

Prof. Biju Cherian Abraham
Associate Professor
Department of Mechanical Engineering
Mar Athanasius College of Engineering, Kothamangalam

Prof. Jijo Johnson
Assistant Professor
Department of Mechanical Engineering
Mar Athanasius College of Engineering, Kothamangalam

Prof. Dr. Shajan Kuriakose
Professor
Department of Mechanical Engineering
Mar Athanasius College of Engineering, Kothamangalam

Abstract

The present works investigates the influence of re-entrant angle in a piston bowl geometry on both engine performance and combustion efficiency in a direct injection (DI) diesel engine using STAR-CD. The analysis was done by varying the reentrant angle of piston bowl between 16.35˚ to 25.35˚. All the other relevant parameters namely, compression ratio, bowl volume, squish clearance, engine speed and the mass of fuel injected were kept constant. It is observed that the in-cylinder pressures and temperatures are higher for 19.35˚ reentrant angle. NOx emissions were observed higher for 22.35˚ reentrant angle. Lower soot emissions were observed for 19.35˚ reentrant angle when compared with the other reentrant angles. The simulation analysis revealed that 19.35˚ reentrant angle is the optimum reentrant angle yielding best performance and lower emissions for the chosen engine geometry. Higher turbulent energy and velocity magnitude levels are obtained with 19.35˚ reentrant angle indicating efficient combustion emphasizing the need for optimized combustion chamber geometry for better performance and emissions.
Keywords- CFD (Computational Fluid Dynamics), STAR-CD, Re-Entrant Angle, Piston Bowl Geometry, Nox, Soot

I. INTRODUCTION

Researchers are forced to devise to devise new strategies for improving the emission and combustion performance of internal combustion engines with ever increasing pollution awareness and the consequent enforcement of stringent emission norms throughout the world. Diesel engines, in particular, due to its higher thermal efficiency and reliability appears to have been the focus of numerous scientists and engineers and at the same time it has a potential to cater to a wide cross-section of applications such as marine, automobiles, stationary power plants etc. Any strategy to enhance the emission and combustion characteristics of such engines must meet the new emission standards [1].

The simultaneous reduction in the soot and NOx to meet increasingly strict regulations, while maintaining reasonable fuel economy is the current focus of engine research. It is very difficult to reduce them simultaneously, since the factors leading to NOx and soot emissions are totally different. Major challenge is faced is the fact that traditional methods for reducing one of these emissions are inclined to increase the other. It is necessary to explore the vast design space using new analysis techniques, in order to overcome this trade-off feature between NOx and soot emissions for future diesel engines. Optimization of current engine systems is the most economic and feasible way to achieve this aim, because it minimizes the required modifications to the whole vehicle system. Important design parameters include the injection timing, injection pressure, injection rate shape, nozzle design, swirl ratio, combustion chamber design, turbo charging, and exhaust gas recirculation (EGR) [2].

Jaichandra et al. [3] carried out an experimental investigation on combined effect of injection timing and combustion chamber geometry on the performance of a biodiesel fueled diesel engine. It was reported that re-entrant piston bowl geometry could result in improvement of engine performance and increase in NOx emission. Shi and Reitz [4] numerically investigated the effects of piston bowl geometry, swirl ratio as well as injection strategy, which include spray angle and start of injection, on the combustion process and emissions formation under high and low load conditions in a heavy-duty diesel engine. They discovered that for different piston bowl geometries, different injection strategies as well as swirl ratios had to be used for optimal performance

and lower emissions. Moreover, it was found that the shape of the piston bowl affected combustion under high load condition to a greater extent than under low load condition. Also, through numerical simulations using the AVL FIRE software. Prasad et al. [5] carried out the numerical simulation in various swirl inducing piston bowls in small diesel engines and reported that; along with air motion, spray characteristics such as injection pressure, spray angle, injector-hole diameter and injection timing also have a significant effect on diesel engine combustion. Bapu et al. [6] carried out experiments as well as numerical simulations to investigate the effects of a modified hemispherical combustion chamber on the performance and emissions in a conventional diesel engine. It was found that the modified hemispherical combustion chamber gave superior combustion characteristics when compared to the normal hemispherical combustion chamber due to better squish generation.

Park [7] studied the optimum combustion chamber geometry for a diesel engine using a micro genetic algorithm. Through this optimization, a shallow chamber geometry was developed; this led to a 35% improvement in the gross indicated specific fuel consumption. Li et al [8], performed numerical simulation on a hemisphere (HCC), shallow (SCC) and omega (OCC) combustion chamber by using the KIVA-4 CFD software at a constant compression ratio of 18.5. Results indicated that, SCC and OCC generate higher NOx emission at low and high engine speed respectively. The combination of spray and improved piston bowl geometry has positive impact over emission reduction. Jesús et al [9], carried out experimental analysis with the single and double injection strategy on three different piston bowl geometry, and results have achieved low NOx and soot emission with low and partial load conditions at double injection strategy.

The aim of this paper is to present the results of a comprehensive CFD study on the performance characteristics of a heavy-duty, DI diesel engine. The approach followed is divided in two parts: validation of the predicted results and analysis of the performance of engine under consideration. For the purpose of validation, calculations of the suction and compression strokes are performed and the predicted results are compared with experimental data obtained from literature Mobasheri et al. (2013). In the second part, CFD predictions of the combustion process have been discussed for different re-entrant angle in a piston bowl geometry. The combustion prediction includes cylinder pressure, temperature, heat release, Indicated mean effective pressure, brake specific fuel consumption, NOx and soot emissions, so that the combustion process could be analyzed in detail. In particular, differences observed for the different injection timing and different intake pressure are discussed and appropriate conclusions are drawn. These provide an insight on the influence of injection timing and intake pressure on the combustion characteristics.

II. ENGINE GEOMETRY AND COMPUTATIONAL DETAILS

A single cylinder Caterpillar 3401 heavy-duty DI diesel engine has been used. The engine specifications are given in Table 1 with a constant engine speed of 1600 rev/min. The fuel delivery system was an electronically controlled, common rail fuel injection system. In all the injection cases studied, the same amount of fuel is injected in each engine cycle. The main characteristics of the injection system are listed in Table 2. After the piston bowl is generated and used for the creation of in-cylinder mesh. The meshing of the in-cylinder fluid domain is performed using es-ICE (Expert System - Internal Combustion Engine) grid generation tool. Because of the symmetrical location of the injector at the center of the combustion chamber, the CFD calculations are performed on 60˚ sector meshes. Exhaust and intake ports are not included in the computational mesh by concentrating this simulation on in-cylinder flow and combustion processes. Calculations begin at Intake Valve Closure (IVC) and end at Exhaust Valve Opening (EVO) i.e. 580˚CA to 800˚CA. 720˚CA corresponds to TDC. The same initial and boundary conditions are used for all the computations. Figure 1 shows computational grids at TDC. Figure 2 shows computational grids at 100˚BTDC.

Table 1: Engine specifications

Engine type	*Caterpillar3401*
Bore	*13.719cm*
Stroke	*16.51cm*
Compression Ratio	*15.1:1*
Displacement	*2.44l*
Connecting rod length	*26.162cm*
Squish clearance	*4.14mm*
Piston Shape	*Mexican Hat style*

Table 2: Injector fuel system specifications

Injector type	*Electronically controlled, common rail*
Injection pressure	*Variable(upto120MPa)*
Number of nozzle holes	*6*
Nozzle hole diameter	*0.26mm*
Included Spray Angle	*125˚*
Start of injection	*9˚BTDC*
Injection duration	*21.5˚CA*

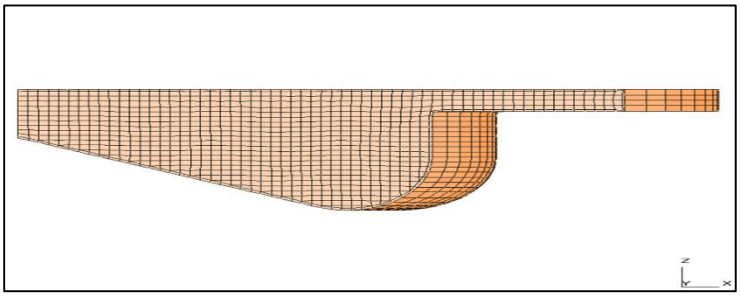

Fig. 1: Computational grids at TDC

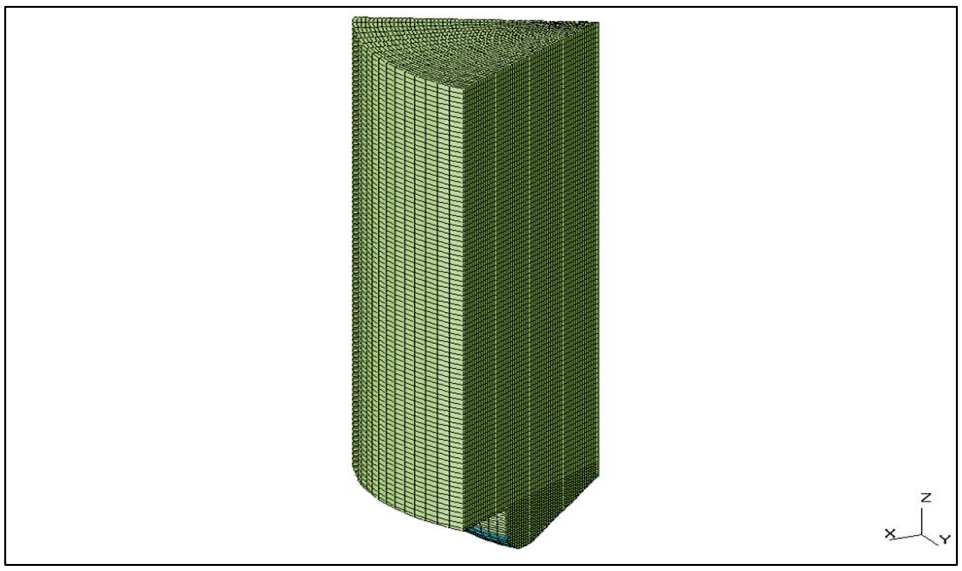

Fig. 2: Computational grids at 100˚BTDC

III. MODELING STRATEGY

The STAR-CD used in the present study has integrated several sub models such as turbulence, fuel spray and atomization, wall function, ignition, combustion, NOx, and soot models for various types of combustion modes in CI as well as SI engine computations. As initial values of k and ε are not known therefore the turbulence initialization is done using I-L model. This practice will ensure that k and ε and the turbulent viscosity μt, will all scale correctly with V∞, which is desirable from both the physical realism and numerical stability point of view. Moreover the turbulent intensity is defined using the same velocity vector magnitude as that of stagnation quantities. The combustion is modeled using ECFM-3Z. As far as fluid properties are concerned, ideal gas law and temperature dependent constant pressure specific heat (Cp) are chosen.

IV. VALIDATION

The diesel engine used for the model validation is a single-cylinder version of a Caterpillar 3401 heavy-duty truck engine. The experimental results for this part of study have achieved from the Mobasheri et al. (2013). A comparison between experiments and simulation is presented, in order to assess the accuracy of the subsequent predictions. As visible, Figure 3.10 and Figure 3.11 computational in-cylinder pressure and heat release rate agrees fairly well with the experimental trace. The computed and experimental in-cylinder peak pressures are 102.5 bar and 105.8 bar, respectively. The peak pressure discrepancies between experiment and computation are about 3%.The computed and experimental peak heat release rate are 49.5 J/deg and 52.5 J/deg, respectively. The peak heat release rate discrepancies between experiment and computation are about 6%. The trend predicted by the model is reasonably close to experimental results, although there are still some differences. These discrepancies could be related to experimental uncertainties in input parameters to the computations such as the precise injection duration, start of injection timing and gas temperature at IVC.

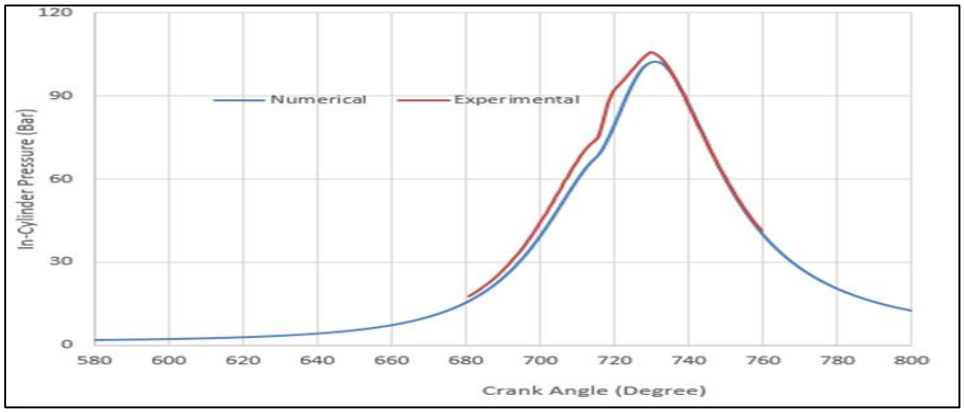

Fig. 3: Comparison of numerical and experimental in-cylinder pressure

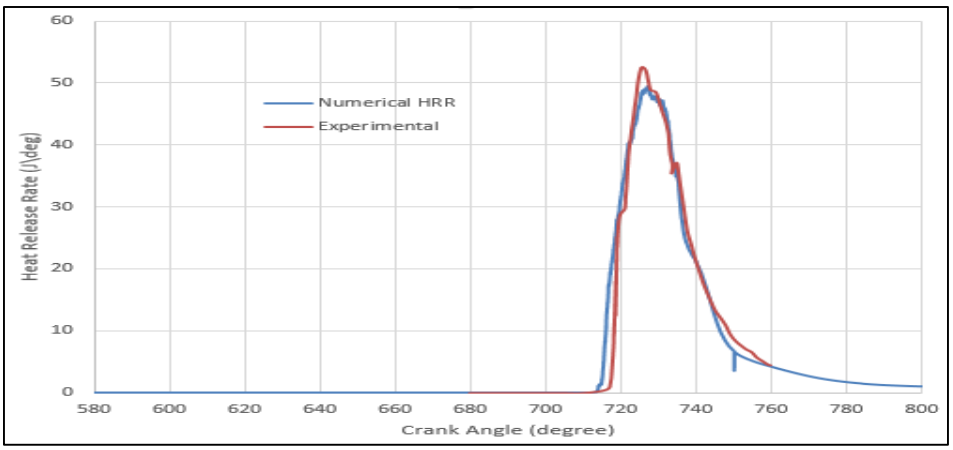

Fig. 4: Comparison of numerical and experimental heat release rate

V. RESULT AND DISCUSSIONS

Based on the confidence gained from validation, the study is extended to evaluate the effect of piston bowl geometry by varying re-entrant angle from 16.35˚ to 25.35˚ on diesel engine performance and emissions. A comprehensive study of the effect of re-entrant angle in a piston bowl geometry with respect to performance and emissions are presented below. Figure 5 shows In-cylinder pressure variations versus crank angle at different re-entrant angle. No significant change in the in-cylinder pressures can be observed with variation in reentrant angle from 16.35˚-25.35˚, but reentrant angle 19.35˚ is showing higher in-cylinder pressures than the remaining. It was observed that the in-cylinder pressures first increased when the reentrant angle varied from 16.35˚ to 19.35˚ and the decreased till 25.35˚ reaching the lowest. With variation in reentrant angles the charge motion inside the combustion chamber changes this will affect the turbulent kinetic energy and dissipation rates inside the combustion chamber. At reentrant angle 19.35˚ the higher in-cylinder pressures are observed because of the better charge motion inside the combustion chamber leading to a better combustion when compared with the other reentrant angles.

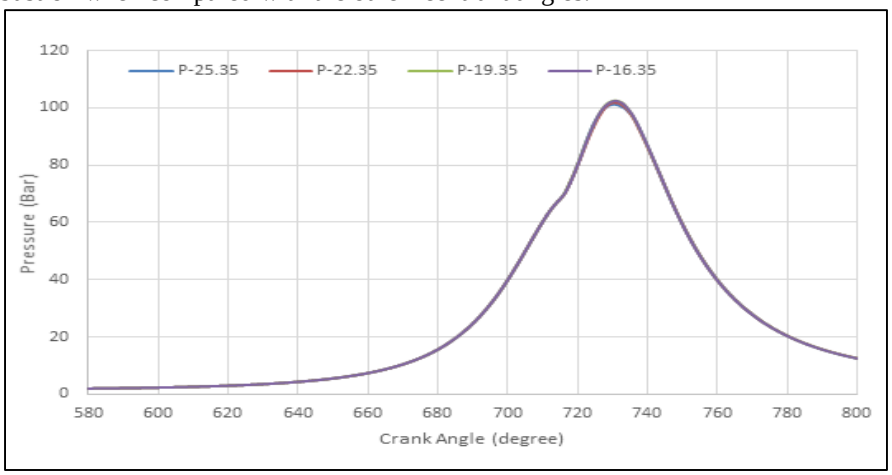

Fig. 5: In-Cylinder pressure at different reentrant angles

Figure 6 shows In-cylinder temperature variations versus crank angle at different re-entrant angle. It was observed that the in-cylinder temperature first increased when the reentrant angle varied from 16.35˚ to 19.35˚ and the decreased till 25.35˚ reaching the minimum. A peak temperature of 1683.14 K was observed with 19.35˚ reentrant angle. The increase in in-cylinder temperature at 19.35˚ is due to better combustion because of increased charge motion inside the combustion chamber. This shows that the geometry of the combustion chamber plays a vital role improving the combustion efficiency without having any modifications to the operating parameters. Figure 7 shows the variation of velocity magnitude with crank angle at different reentrant angles. The velocity magnitude at a particular region should be optimum and uniform to have a better mixing of fuel – air mixture, to draw out the trapped charge from crevice regions and to avoid higher heat transfer losses. From Figure 7 it is clear that the velocity magnitude is uniform throughout the combustion chamber with reentrant angle 19.35˚ having the least difference between the local maximum and minimum velocity magnitudes when compared with other reentrant angles. Maximum variation in minimum and maximum velocity magnitudes was observed with 16.35˚ reentrant angle. Velocity magnitude was observed to be increasing with crank angle with all reentrant angles, this phenomena is attributed to higher combustion temperatures increasing the kinetic energy of charge particles in the combustion chamber.

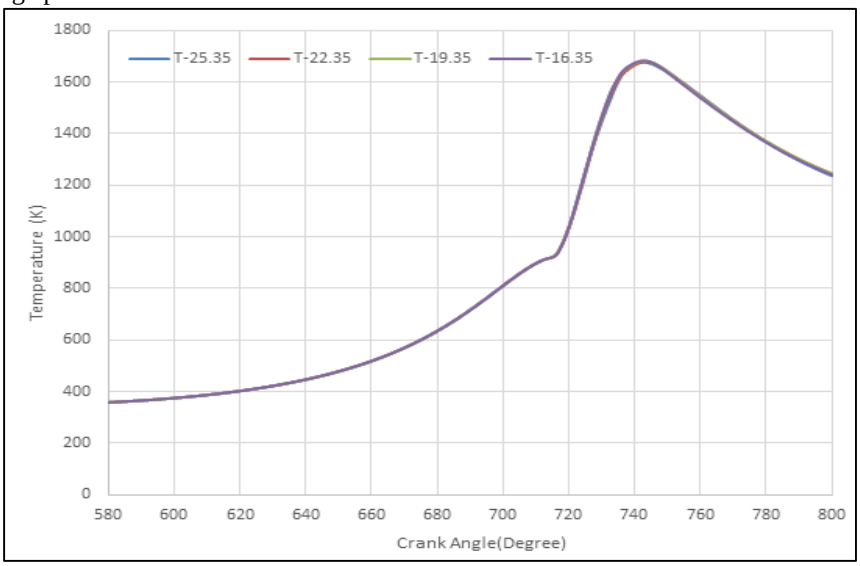

Fig. 6: In-Cylinder temperature at different reentrant angles

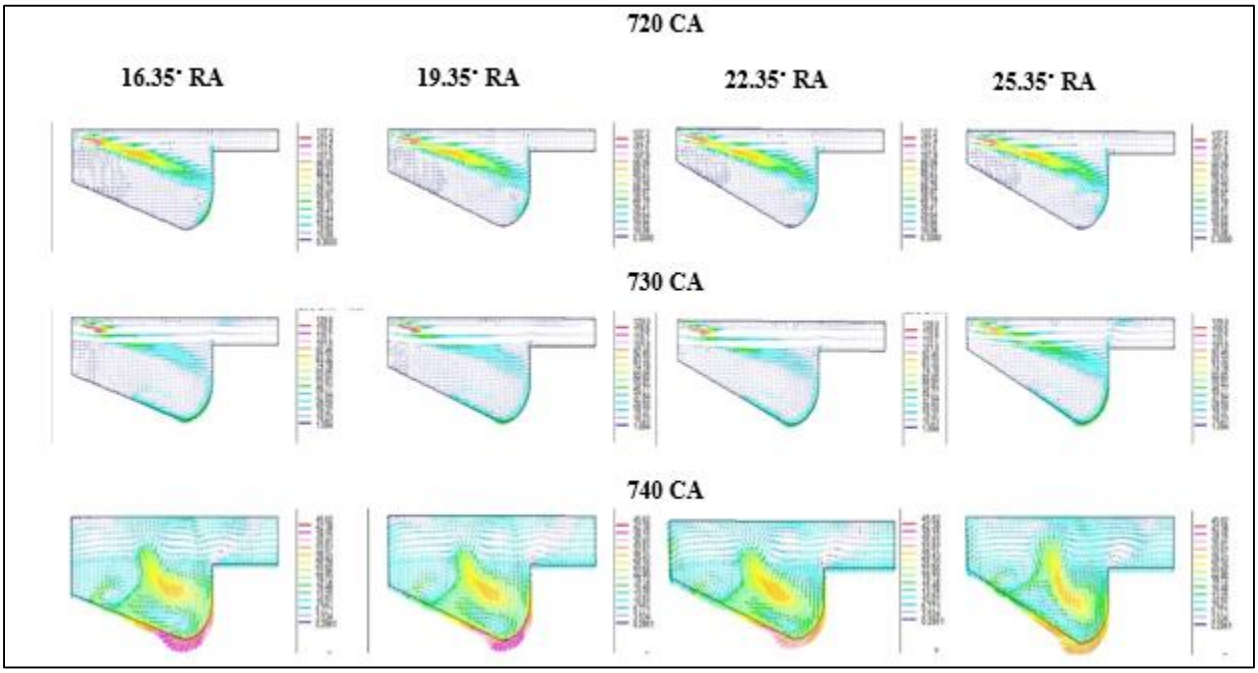

Fig. 7: Variation of velocity magnitude with crank angle at different reentrant angles

Figure 8 shows the variation of turbulent kinetic energy with crank angle at different reentrant angles. Burn time and flame speed at a particular region can be determined by Turbulent Kinetic Energy (TKE). Uniform Turbulent kinetic energy distribution in the combustion chamber lead to efficient and complete combustion. From Figure 8 it is clear that the difference

between local minimum and maximum values of TKE of 19.35˚ reentrant angle was observed to be lowest compared to other reentrant angles reveling uniform distribution of TKE throughout the combustion chamber. The uniform distribution of TKE represents the efficient and complete combustion conditions. This shows that 19.35˚ reentrant angle is optimum reentrant angle for the engine specification chosen. Increase in turbulent kinetic energy with crank angle can be seen with all the reentrant angles.

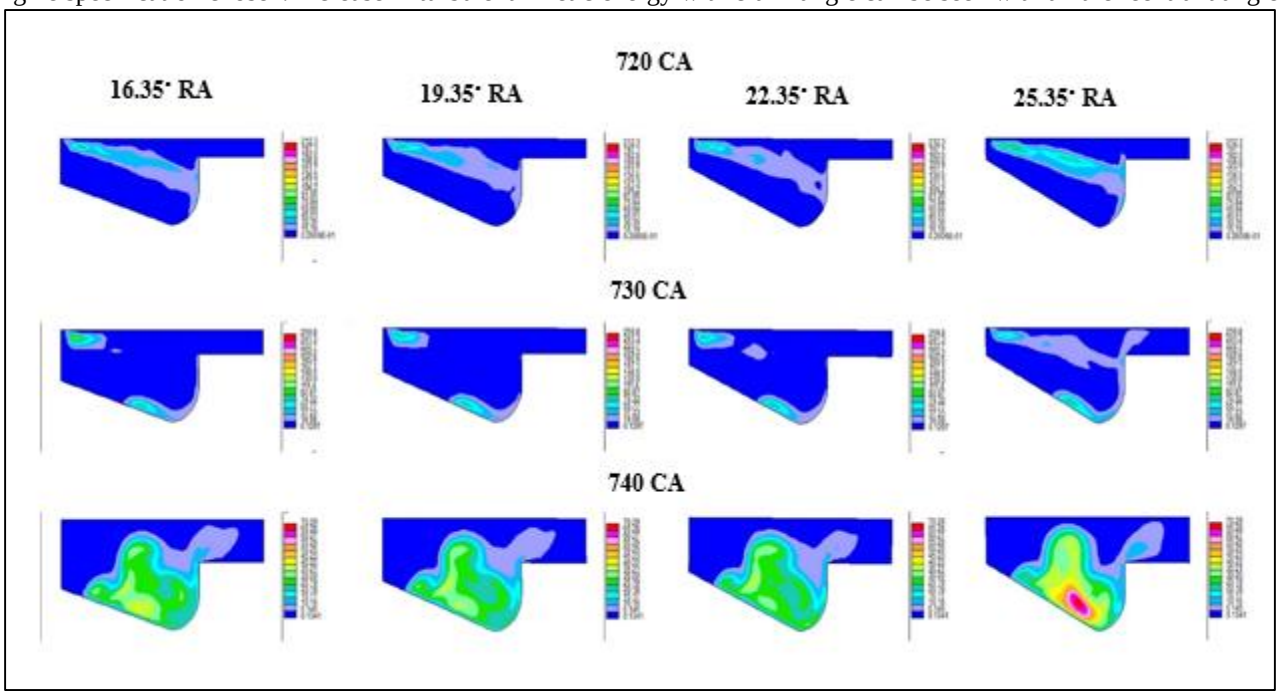

Fig. 8: Variation of turbulent kinetic energy with reentrant angle at different crank angles

Figure 9 shows NOx emissions variations versus crank angle at different reentrant angles. Figure 10 shows the variation of NOx emissions contours with crank angle at different reentrant angles. The oxides of nitrogen in the exhaust emissions contain nitric oxide (NO) and nitrogen dioxide (NO_2). The formation of NOx is highly dependent on the in-cylinder temperature, oxygen concentration and residence time for the reaction to take place. Form Fig. 9 it can be seen that NOx emissions were observed to be higher for 22.35˚. Though with 19.35˚ reentrant angle the in-cylinder temperatures are high the higher wall heat transfer losses and efficient combustion reduces the availability of oxygen for the N_2 oxidation. Figure 11 shows soot emissions variations versus crank angle at different reentrant angles. Figure 12 shows the variation of soot emissions contours with crank angle at different reentrant angles. Form Fig. 9 it can be seen that soot emissions were observed to be higher for 25.35˚ due to improper mixing fuel and air and also lower in-cylinder temperature leading to in-complete oxidation of soot particles.

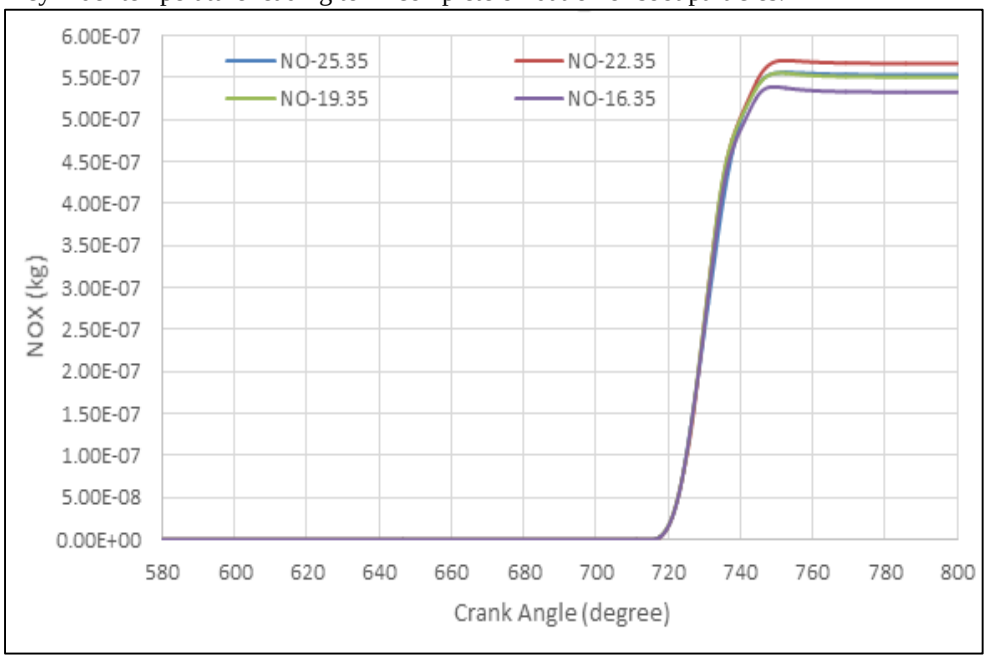

Fig. 9: NOx emission at different reentrant angles

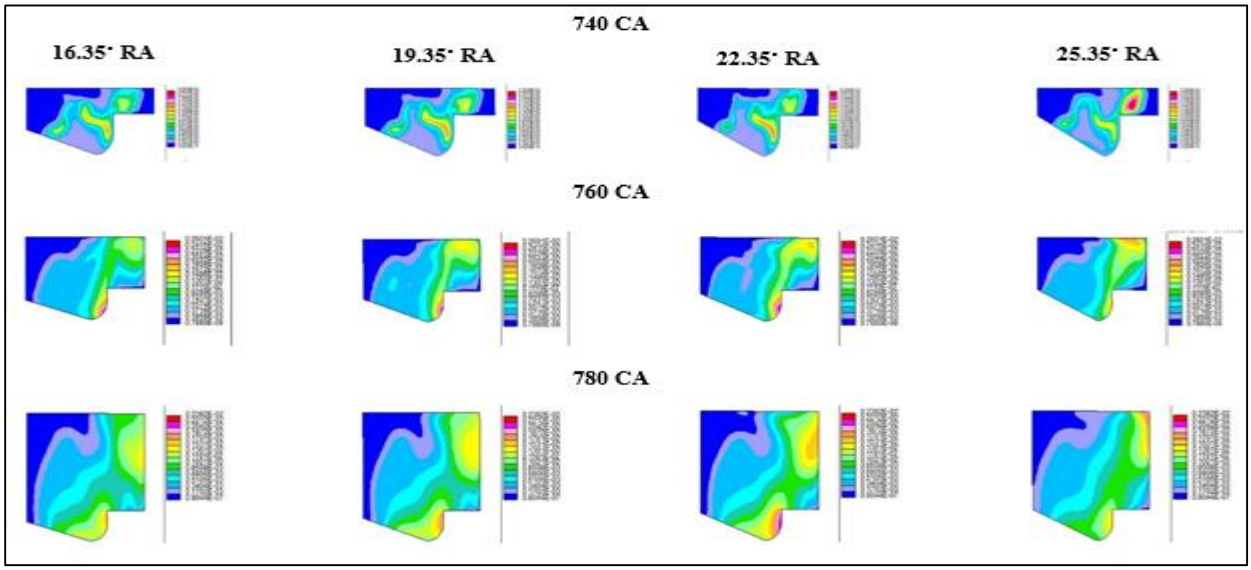

Fig. 10: NOx contours at different reentrant angles

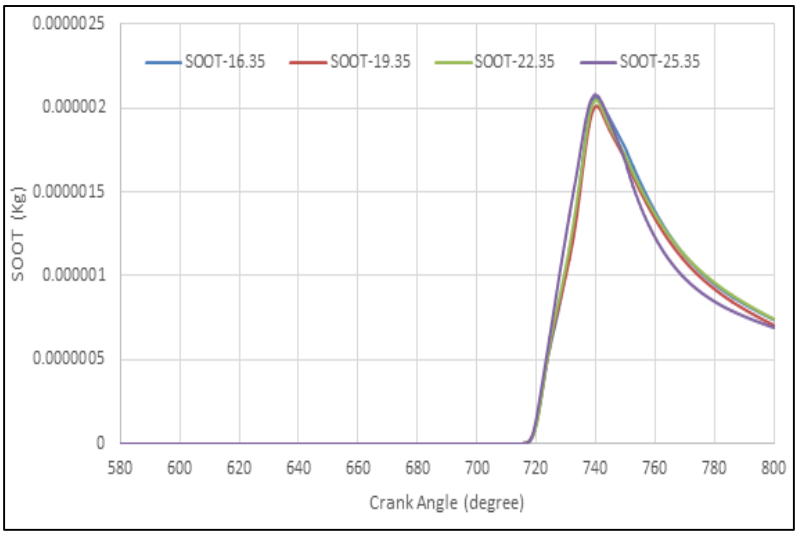

Fig. 11: Soot emission at different reentrant angles

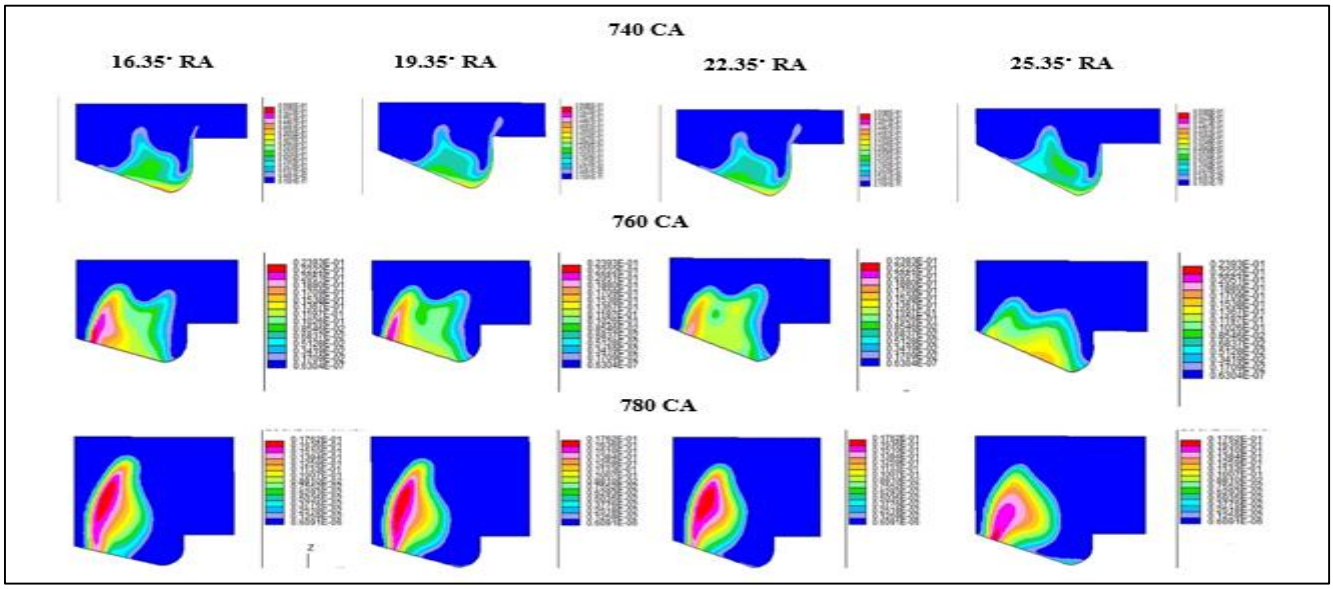

Fig. 12: Soot contours at different reentrant angles

VI. CONCLUSIONS

In this study, an advanced CFD simulation was performed to demonstrate the effects change in re-entrant angle in piston bowl geometry on a Direct Injection (DI) diesel Engine (Caterpillar 3401). Re-entrant angle of piston bowl was varied between 16.35˚ to 25.35˚ i.e. study was conducted at 16.35˚, 19.35˚, 22.35˚ and 25.35˚ angles. From the current study, the following conclusions can be drawn. The in-cylinder temperatures and pressures with 19.35˚ reentrant angle were observed to be comparatively higher. NOx emission with 22.35˚ reentrant angle was observed to be higher when compared with other reentrant angle. Soot emissions were found higher for 25.35˚reentrant angle since it has the lowest in-cylinder temperature and soot emissions were lower at 19.35˚ reentrant angle. Results demonstrated that, the variation of reentrant angle from 16.35˚ to 25.35˚ showed that 19.35˚ reentrant angle is the optimum reentrant angle for the chosen engine specifications.

REFERENCES

[1] Abdul Gafoor C.P., Rajesh Gupta, "Numerical investigation of piston bowl geometry and swirl ratio on emission from diesel engines," Energy Conversion and Management 101 (2015) 541–551.
[2] Raouf Mobasheri, Zhijun Peng," CFD Investigation of the Effects of Re-Entrant Combustion Chamber Geometry in a HSDI Diesel Engine," World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol: 7, No:4, 2013.
[3] Jaichandar S, Senthil Kumar P, Annamalai K.," Combined effect of injection timing and combustion chamber geometry on the performance of a biodiesel fueled diesel engine" Energy 2012;47:388–94.
[4] Shi Y, Reitz RD.," Optimization study of the effects of bowl geometry, spray targeting, and swirl ratio for a heavy-duty diesel engine operated at low and high load," Int J Engine Res 2008;9(4):325–46.
[5] Prasad BVVSU, Sharma CS, Anand TNC, Ravikrishna RV.," High swirl – inducing piston bowls in small diesel engines for emission reduction," Appl Energy 2011;88:2355–67.
[6] Bapu BR, Saravanakumar L, Prasad BD.,"Effects of combustion chamber geometry on combustion characteristics of a DI diesel engine fueled with calophyllum inophyllum methyl ester," J Energy Inst 2015.
[7] Park SW.," Optimization of combustion chamber geometry for stoichiometric diesel combustion using a micro genetic algorithm," Fuel Processing Technology. 2010;91(11):1742-52. [8] D.A. Zumbrunnen, M. Aziz, Convective heat transfer enhancement due to intermittency in an impinging jet, Journal of Heat Transfer 115 (1) (1993) 91–98.
[8] Li J., Yang WM., An H., Maghbouli A., Chou SK.," Effects of piston bowl geometry on combustion and emission characteristics of biodiesel fueled diesel engines," Fuel, 120:66–73. http://dx.doi.org/10.1016/j.fuel.2013.12.005.
[9] Jesús B., José VP., Antonio G., Javier MS.,"An experimental investigation on the influence of piston bowl geometry on RCCI performance and emissions in a heavy-duty engine," Energy Conversion and Management, 103:1019–1030.
[10] Pasupathy Venkateswaran, S., and G. Nagarajan, 2010, "Effects of the Re-Entrant Bowl Geometry on a DI Turbocharged Diesel Engine Performance and Emissions—A CFD Approach," Journal of engineering for gas turbines and power 132.12 (2010).