STUDY AND ANALYSIS ON THE EFFECT OF COMBUSTION CHAMBER CONFIGURATION ON IN-CYLINDER AIR MOTION IN A DIRECT INJECTION TURBOCHARGED DIESEL ENGINE – COMPUTATIONAL FLUID DYNAMICS

Krishan Kant Meel[1], MD Aslam Khan[2]

[1&2]Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology,
Email: kmeel24@gmail.com.

Abstract-

In the present investigation due to the stringent legislation for emission of diesel engines and also increasing demand on fuel consumption, the importance of detailed 3D simulation of fuel injection, mixing and combustion have been increased in the recent years. In this regard, Computational Fluid Dynamics (CFD) technique has been identified as a very helpful tool through which the in-cylinder process can be simulated and studied before proceeding to prototyping and experimental studies.

In this study, CFD approach was focused to investigate the effect of combustion chamber configuration on in-cylinder air motion in a DI diesel engine. Two different shapes of diesel engine combustion chamber were modeled and simulated under transient conditions using a commercial CFD code STAR-CD. In this paper, results of detailed 3-D simulations of the two configurations of combustion chamber are presented. It has been observed that effect of combustion chamber shape on in-cylinder air motion was quite significant near TDC of compression which is important period with respect to combustion and pollutant formation.

I. INTRODUCTION

Direct injection (DI) diesel engines are widely used in heavy duty transport applications. With the increasingly stringent regulations on emissions from diesel engines and continual pursuit for more efficient engines, a better understanding of the combustion process in a diesel engine is necessary in order to optimize further the engine performance and emissions.

In combustion tuning of a DI diesel engine, the mixture formation of the injected fuel jet and air is very important. Mixture formation mainly depends on combustion chamber configuration and distribution of fuel jet, because bowl configuration can control air motion within the combustion cavity (swirl and squish) and its air motion can affect the distribution of fuel jet. The shaping of piston cavities has attracted considerable attention of design/development engineers during development of low emission engines. Here, CFD can play a very important role to reduce the number of experiments and to provide a detailed insight on combustion and related issues.

II. CFD MODEL SETUP

CFD analyses are carried out on a six cylinder direct injection turbocharged diesel engine, focusing on the influence of the combustion chamber shape on in-cylinder air motion. The calculations are carried out for full load at rated speed. The engine specifications are listed in Table.1

TABLE. 1 ENGINE SPECIFICATION

Power output	165 kW
Speed	2500 rpm
Bore	108 mm
Stroke	
	6.48 liters

The baseline configuration of the combustion chamber “Configuration-1” is shown in Fig.1. and Fig.2 shows the modified combustion chamber “Configuration-2”. Volume of both configurations was kept same to maintain the same compression ratio.

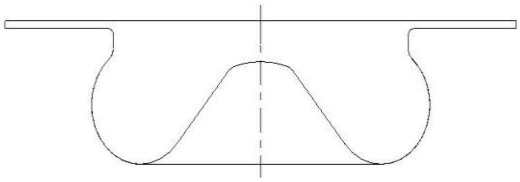

Fig.1 Configuration-1 (Baseline)

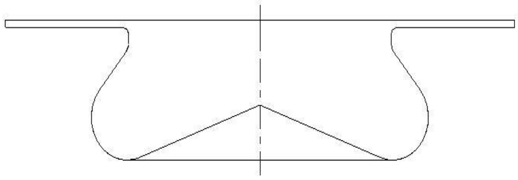

Fig.2 Configuration-2

Compression and expansion strokes are simulated in each of the above two cases. During the simulation of the closed valve part of the cycle, cylinder and combustion chamber have been considered as computational domain.

III. NUMERICAL METHODOLOGY

Commercial CFD software, STAR-CD has been used to perform the in-cylinder air flow calculations in the present investigation. Any general flow field is described by the Navier-Strokes (N-S) equation. As the air inside is subjected to high compression, density variations are required to be considered during solution procedure. To solve Navier-Strokes equation numerically, the SIMPLE algorithm has been implemented on an unstructured grid. To model the turbulence, k-ε model has been chosen for the current study. The computational grid that has been generated for closed valve part of the cycle is shown in Fig.3.

IV. RESULTS AND DISCUSSION

CFD simulations of in-cylinder air motion give 3-D distribution of different quantities of interest like velocities, pressure, Turbulence Kinetic Energy (TKE) etc. Study of these results leads to clear understanding of effect of any change made to the geometry of the bowl on flow and turbulence.

A convention of treating the TDC of compression as 360° CA is followed in figures and discussion. Swirl number and TKE are identified as important with regard to in-cylinder air motion of DI diesel engines.

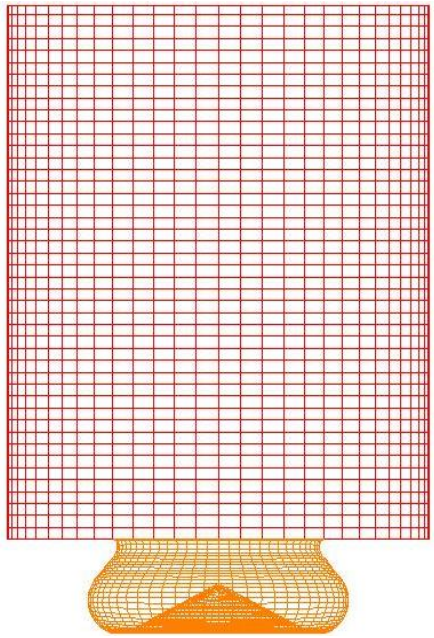

Fig.3 Grid Used for the Closed Cycle Simulation

Variation of mass averaged swirl number during the closed valve part of the cycle for the combustion chambers Baseline and Configuration-2 are shown in Fig.4. During initial part of compression, there is a negligible difference in swirl ratio of Baseline and Configuration-2. But from 300° CA (60° CA before TDC) to 390° CA (30° CA after TDC), which is important period with regard to combustion, the swirl ratio of Configuration-2 was higher than Baseline combustion chamber. The peak value of the swirl is about 23% higher than that of the Baseline chamber.

The central projection in Baseline seems to obstruct the squish flow. As the reduction in neck radius (Throat radius) is only 12%, the squish flow was not strong enough to overcome the obstruction from the central projection. These may be the reasons for the lower swirl level in Baseline chamber.

In Configuration-2, squish flow is penetrating much deeper towards the axis of the cylinder, which is the main reason for higher swirl intensification.

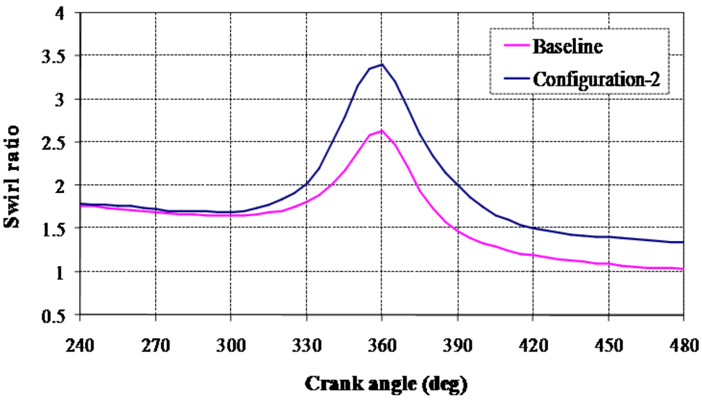

Fig.4 Average Swirl Ratio During Compression and Expansion

Variation of TKE during the compression and expansion strokes is shown in Fig.5. It can be observed that around TDC, the TKE intensification of Configuration-2 chamber is as high as 32% more than the Baseline chamber. Since Configuration-2 was provided with more re-entrancy (reduction in throat radius is 23%) with minimum central projection, TKE values are much higher.

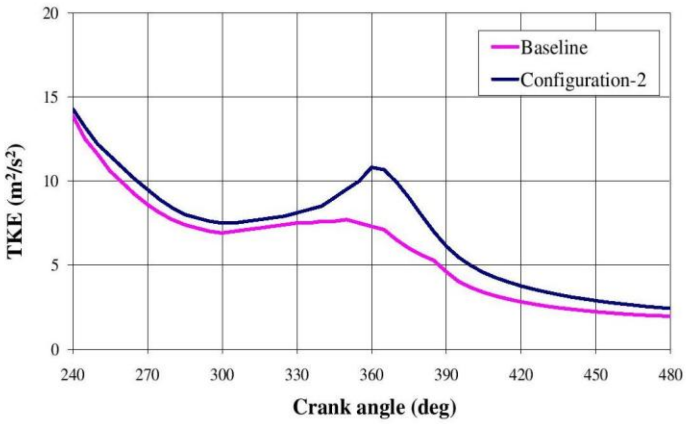

Fig.5 Average TKE Variation During Compression And Expansion

It is expected that, using the combustion chamber Configuration-2 will definitely help in promoting better combustion and reduction of emissions. Starting of Injection for this engine is at around 8° CA before TDC. Higher velocities and turbulence help in faster break-up and mixing of the fuel with air. Better mixing will enhance fuel combustion which is helpful in reducing significantly the CO, HC and Particular matter emissions of the selected engine.

Better combustion leads to higher temperature of the gas in combustion chamber. As NO_X is highly sensitive to temperature, considerable rise in NO_X emission is possible. Strategy of retarding the injection can be used to reduce the NO_X emission.

Use of Configuration-2 with appropriate injection timing, may help in promoting the turbulence, thereby enhance the rate of combustion and reduce the emissions of the selected engine.

V. SUMMARY AND CONCLUSION

CFD simulations of in-cylinder air flow of a DI turbocharged diesel engine have been carried out to investigate the effect of bowl geometry on the in-cylinder flow. Two different configurations of combustion chamber, Baseline & Configuration-2 have been studied with respect to in-cylinder air flow and turbulence. The Configuration-2 was found to be better than baseline bowl.

Swirl and TKE levels of Configuration-2 are much higher than the baseline bowl. This is due to more re-entrancy of bowl with minimum central projection. Configuration-2 with appropriate injection timing shows a high potential towards significant reduction of emissions of the engine.

REFERENCES

[1] Corcione. F. E, Annunziata Fusca, and Gerardo Valentino, “Numerical and Experimental Analysis of Diesel Air Fuel Mixing”, SAE 931948.

[2] Ikegami. M, Fukuda. M, Yoshihara. Y and Kaneko. J, “Combustion Chamber Shape and Pressurized Injection in High- Speed Direct –Injection Diesel Engines”, SAE 900440.

[3] Lu Lin, Duan Shulin, Xiao Jin, Wu Jinxiang and Xiaohong, “Effect of Combustion Chamber Geometry on In-Cylinder Air Motion and Performance in DI Diesel Engine”, SAE2000-01-0510.

[4] Payri. F, Benajes .J, Margot .X, Gil. A, “CFD Modeling of the In-Cylinder Flow in Direct –Injection Diesel Engines”, Computers and Fluids 33 (2004) 995-1021.