Study of Omega Shaped Re-Entrant Piston Bowl Geometries on In-Cylinder Flow and Performance in a Direct-Injection Diesel Engine

[1]Raghavendra Reddy NV, [2]Dr. Jayashankara B

[1]AssistantProfessor, [2]Professor
[1]Dept. of Mechanical Engineering
[1]Jain Institute of Technology, Davangere/ VTU, India

Abstract **-Flow fields are generated prior to combustion in internal combustion engines during the induction process in suction stroke and they undergo changes during the compression stroke. It is very essential to understand the fluid motion during suction and compression strokes for developing new engine designs with the best performance and emission characteristics. Therefore, computational studies have been carried out to comprehend the role of in-cylinder flow structure. In this 3D CFD, Ansys Forte V18 is used to study the incylinder fluid motion and performance and emissions of the direct injection compression ignition engines. The optimization of the omega shaped reentrant piston bowl with variation in compression ratio has been carried out by varying the piston height and reentrant radius. The obtained results are plotted during the compression stroke and expansion strokes and are analysed. The results showed that, the bowl shapes plays a significant role in the incylinder fluid motion, performance and emissions of compression ignition engines. It controls the fuel-air mixing and burning rates in compression ignition engines.**

Index terms **-Compression Ignition Engine, Computational Fluid Dynamics (CFD), Swirl Ratio (SR), In-cylinder Flow Structure, Bowl Shape.**

I. INTRODUCTION

The internal combustion (IC) engine has had a significant impact on society and has been the foundation for the successful development of many commercial technologies. It is reported in 2008 [1], approximately 300 million automobiles in worldwide are powered by IC engines. IC engines can deliver power in the range from 0.01 kW to approximately 20X103 kW, based on their displacement. Because of this flexibility, IC engines are used in various applications such as automobiles, trucks, locomotives, marine, aircrafts, and power generation. The IC engine industry has become very large and competitive because of its widespread applications and ages. [2]

As we know the fundamental drivers of the energy demand are Population and economic growth. The overall energy need can be divided into four demand sectors as power generation, transportation, industrial, and residential. As it is reported these four major sectors undergo major growth through 2030 [3]. The power generation will have the highest volume growth as it is anticipated and transportation sector will be next to it. In the transportation sector the IC engine play a vital role in personal transportation, goods transportation, works such as construction and agriculture. The demand for low-emission vehicles is increasing to meet one of the challenges concern to environmental crisis. The scientific and industrial communities have taken a priority to lessen the environmental crisis.

The diesel engine, i.e., compression-ignition (CI) engine has become the prime power source in the transportation sector because of its better fuel economy, power, and durability over the gasoline, i.e. spark-ignition (SI) engines. Diesel engines are the primary power source for non-road equipment including construction and agricultural equipment, marine vessels, and locomotives. Diesel engines are known for their efficiency and durability. They have certain environmental disadvantages over gasoline engines.

The manufacturers of Compression ignition engine inevitably have to focus on the reduction of fuel consumption in order to maintain high performance standards. In this respect, turbocharged direct injection diesel engines represent an appealing solution. The challenge in developing these engines also lies in the reduction of raw emissions along with the above-mentioned factors. Both fuel consumption and emission formation can be addressed by internal engine measures.

The Computational Fluid Dynamics (CFD) methodology has undergone many developments for a period of more than two decades, which resulted in improved ability to analyze the flow field in realistic engine geometries [4]. The overall dynamic characteristics of intake and exhaust flows can usefully be studied with one-dimensional unsteady fluid dynamic computer calculations. Flows within the cylinder and in the intake and exhaust ports are usually inherently unsteady and three-dimensional. Because of this the 3-D unsteady state computation mode is adopted for the present under taken work. Recent increase in computing power coupled with encouraging results with two-dimensional calculations indicates that the useful 3-D calculations are now feasible.

In the present work Multi-dimensional CFD code is used for the simulation of air motion inside the cylinder and combustion process in a DI compression ignition engine. Three-dimensional computational domains employed here constitute the combustion chamber capable of simulating complete engine cycles and can run on any operating conditions. The step by step investigation has been carried out to study the effect of Bowl shapes, swirl ratio on in-cylinder fluid motion and intake pressure on combustion process. The appropriate turbulence model, grid density, engine speed and in-cylinder flow pattern be employed by the investigations carried out on an engine geometry for which experimental measurements are available. Validation of simulated results with experimental data available in the literature under motoring conditions have been carried out and are with good match.

The availability of high performance computers and efficient CAD packages resulted in the creation of 3-D models of the port, bowl shapes etc. Now a day's number of different models can be generated in a short time. Subsequent flow simulation can be carried out using CFD, which can give very useful information regarding the flow pattern, combustion pattern and has the potential to reduce the total development time. Detailed simulation of various features of the in-cylinder fluid motion, including heat transfer and combustion processes has now become a realisable goal.

In the virtual development of future engine combustion processes 3D-CFD is an important tool. Here by, the in-cylinder flow and combustion as well as emission formation is investigated for different bowl geometries.

II. CALCULATION METHODS AND SIMULATION TOOLS

The governing equations in ANSYS Forte follow mainly the Continuity equation, Momentum equation (Navier Stokes equation) and Energy equation to solve computational fluid dynamics problem.

The conservation equation for species is given by

$$\frac{\partial \bar{\rho}_k}{\partial t} + \nabla \cdot (\bar{\rho}_k \tilde{u}) = \nabla \cdot \left[\bar{\rho} D_T \nabla \left(\frac{\bar{\rho}_k}{\bar{\rho}}\right)\right] + \dot{\bar{\rho}}_k^c + \dot{\bar{\rho}}_k^s (k = 1, \ldots, K) \quad (2.1)$$

Where: ρ is the density, subscript k is the species index, $K$is the total number of species, u is the flow velocity vector. Application of Fick's Law of diffusion results in a mixture-averaged turbulent diffusion coefficient D_T. $\dot{\bar{\rho}}_k^c$ and $\dot{\bar{\rho}}_k^s$are source terms due to chemical reactions and spray evaporation, respectively.

The summation of Equation 2.1 over all species gives the continuity equation for the total fluid

$$\frac{\partial \bar{\rho}}{\partial t} + \nabla \cdot (\bar{\rho}\tilde{u}) = \dot{\bar{\rho}}^s \quad (2.2)$$

The momentum equation for the fluid is

$$\frac{\partial \bar{\rho}\tilde{u}}{\partial t} + \nabla \cdot (\bar{\rho}\tilde{u}\tilde{u}) = -\nabla \bar{p} + \nabla \cdot \bar{\sigma} - \frac{2}{3}\bar{\rho}\tilde{k}I + \bar{F}^s + \bar{\rho}\bar{g} \quad (2.3)$$

Energy Conservation Equation i.e. the internal energy transport equation is

$$\frac{\partial \bar{\rho}\tilde{I}}{\partial t} + \nabla \cdot (\bar{\rho}\tilde{u}\tilde{I}) = -\bar{p}\nabla \cdot \tilde{u} - \nabla \cdot \tilde{j} + \tilde{\varepsilon} + \bar{\rho}\tilde{\dot{Q}}_c + \tilde{\dot{Q}}_s \quad (2.4)$$

The standard Favre-averaged equations for k and ε are given in Equation 2.5 and 2.6

$$\frac{\partial \bar{\rho}\tilde{k}}{\partial t} + \nabla \cdot (\bar{\rho}\tilde{u}\tilde{k}) = -\frac{2}{3}\bar{\rho}\tilde{k}\nabla \cdot \tilde{u} + \sigma : \nabla\tilde{u} + \nabla \cdot \left[\frac{(\mu+\mu_T)}{Pr_k}\nabla\tilde{k}\right] - \bar{\rho}\tilde{\varepsilon} + \tilde{W}^s \quad (2.5)$$

$$\frac{\partial \bar{\rho}\tilde{\varepsilon}}{\partial t} + \nabla \cdot (\bar{\rho}\tilde{u}\tilde{\varepsilon}) = -\left(\frac{2}{3}c_{\varepsilon_1} - c_{\varepsilon_3}\right)\bar{\rho}\tilde{\varepsilon}\nabla \cdot \tilde{u} + \nabla \cdot \left[\frac{(\mu+\mu_T)}{Pr_\varepsilon}\nabla\tilde{\varepsilon}\right] + \frac{\tilde{\varepsilon}}{\tilde{k}}\left(c_{\varepsilon_1}\sigma : \nabla\tilde{u} - c_{\varepsilon_2}\bar{\rho}\tilde{\varepsilon} + c_s\tilde{W}^s\right) \quad (2.6)$$

In these equations Pr_k,Pr_ε,c_{ε_1}, c_{ε_2} and c_μ are model constants for the standard and RNG k - ε models.

Table 2.1: Constants in the standard and RNG k-ε models

	c_μ	c_{ε_1}	c_{ε_2}	c_{ε_3}	$1/Pr_k$	$1/Pr_\varepsilon$	η_o	β
Standard k-ε	0.09	1.44	1.92	-1.0	1.0	0.769		
RNG k-ε	0.0845	1.42	1.68	Equation 2-21	1.39	1.39	4.38	0.012

The base engine is same for all the piston configurations for CFD analysis. The specifications of the base engine selected for the simulation is given in Table 2.2.

Table 2.2: Engine specifications

Engine Speed	1500 rpm
Bore	102 mm
Stroke	116 mm
Connecting rod length	174 mm
Number of solid cone injector	1
Mass ofinjected fuel	36mg/cycle
Start of injection(CA)	712 [deg ATDC]
Injection duration(CA)	20 [deg]

A change to the geometry of the combustion chamber is bound to affect the in-cylinder flow structures, as well as have influence on engine performance, emission levels and heat transfer. In the present study,the variation was made to the piston bowl geometry in terms of height (Ph) and re-entrant radius (Pr). Nine piston bowl geometry configurations are used in the study, while the volumes of the combustion bowl, as well as the compression ratios are changed with respect to piston height and reentrant radius. The changes in geometry for the piston bowl is shown in fig 2.1

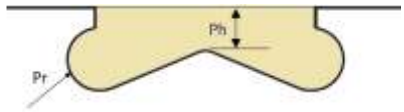

Fig.2.1 Omega-Shaped Re-entrant Piston bowl geometry

ANSYS Forte v18.0 software is used to perform RANS CFD simulations in the present work. A 60° sector of the engine cylinder was modeled corresponding to the sector for one injection hole. The spray-oriented mesh, shown in Figure 2.2 consists of about 58,000 cells at TDC and about 214,000 cells at bottom dead centre (BDC). The FLAME module ESE Diesel mesh tool is used. The turbulent flow was modeled using the RNG k-ε turbulence model. The advanced and recommended version of the k-ε model is derived from Re-Normalized Group (RNG) theory, as first proposed by Yakhot and Orszag. The k equation in the RNG version of the model is the same as the standard version, but the ε equation is based on rigorous mathematical derivation rather than on empirically derived constants. The spray atomization and droplet breakup of solid-cone sprays are modeled by the Kelvin-Helmholtz / Rayleigh-Taylor (KH /RT) hybrid breakup model.

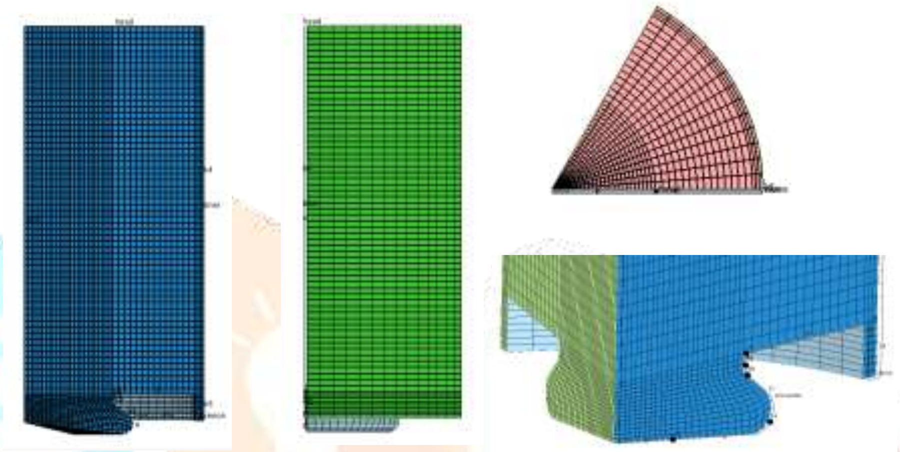

Fig.2.2Sector mesh of in-cylinder fluid domain at BDC

The simulation started at inlet valve closure (IVC), 570° after TDC (ATDC), and ended at exhaust valve opening (EVO), 833° ATDC. The assumption of a swirl ratio was used to enable sector simulation of 1/6 of the total combustion chamber volume.

III. Results and Discussions

A numerical simulation study was carried out to investigate the effect of combustion chamber configuration on air motion inside the cylinder of a DI diesel engine motored at 1500 rpm. Omega Shaped with Re-entrant bowl geometry with ninepiston bowl geometry configurations were modeled for the flow analysis in the first stage. This section describes the results from a comprehensive CFD study on the flow characteristics inside the cylinder of the engine. The flow in the cylinder during only compression and expansion stroke was analyzed, and presented in the following section.

The below figures shows the temperature, velocity and turbulent kinetic energy contour plots for different piston height and reentrant radius of Omega shaped reentrant piston bowl. Fig.3.1 shows the sequence of images for the temperature contour plotsin the mid-plane of the modeled 60-degree cylinder sector, together with the three-dimensional distribution of liquid fuel particles. It is meaningful to consider combustion to occur in multiple stages. The first stage extends along the injection trajectory, with injected fuel evaporation to form a premixed-mixture that ignites as a mass. The second stage involves the liquid droplets that reach the piston bowl wall with evaporation in the arch region of the piston bowl. Fig. 3.2 shows the velocity contour plots in the mid-plane of the modeled 60-degree cylinder sector, together with vector distribution of flow.It shows that the reduced swirl level reduces the velocities in the combustion bowl of the diesel combustion bowls.Fig. 3.3 shows the turbulent kinetic energy contour plots.

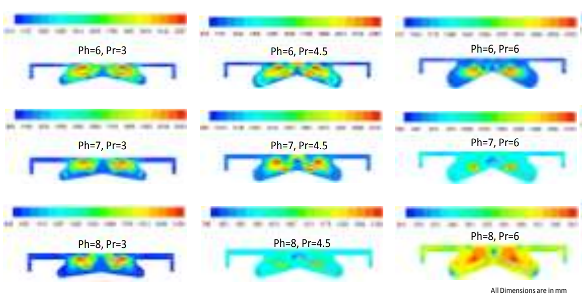

Fig.3.1 Temperature Contours in degree Kelvin

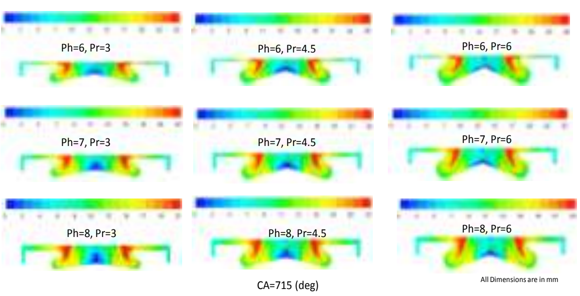

Fig.3.2 Velocity Contours in m/s

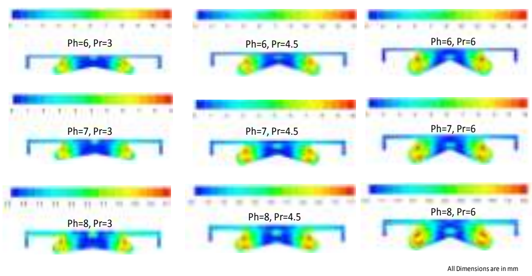

Fig.3.3 TKE Contours in m^2/s^2

The below table 3.1 gives the performance results of the engine for the different geometry of the omega shaped reentrant piston bowl.

Table 3.1: Results Summary of Omega Shaped Re-entrant Piston bowl

P_h	P_r	Gross Indicated Power	IMEP	ISFC	Combustion Efficiency	Thermal Efficiency	Max Pressure	EINOx	SR
mm	mm	kW	MPa	g/kW-h	%	%	MPa	g/kg	
6	3	6.19	0.59	292.88	71.73	30.87	9.85	0.87	2.17
6	4.5	5.85	0.56	311.81	69.80	29.19	8.32	0.67	2.31
6	6	4.91	0.47	376.37	59.99	24.53	6.88	0.40	2.43
7	3	5.99	0.58	304.14	70.15	29.92	7.68	0.52	2.09
7	4.5	5.40	0.52	340.23	63.94	26.96	6.71	0.41	2.20
7	6	4.55	0.44	411.86	56.69	22.70	5.48	0.25	2.31
8	3	5.11	0.49	362.13	61.08	25.51	6.15	0.33	2.04
8	4.5	4.69	0.45	398.26	57.43	23.42	5.34	0.25	2.14
8	6	3.83	0.37	510.30	55.73	19.13	4.26	0.08	2.23

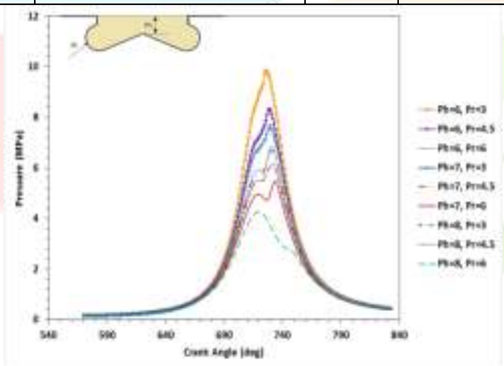

Fig.3.4 PressureVariation Graph

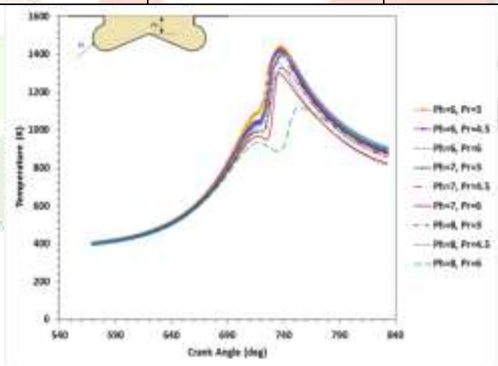

Fig.3.5 TemperatureVariation Graph

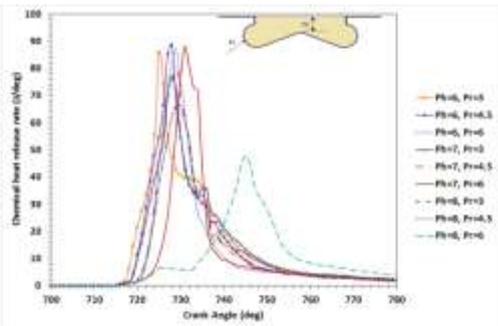

Fig.3.6 Heat Release RateVariation Graph

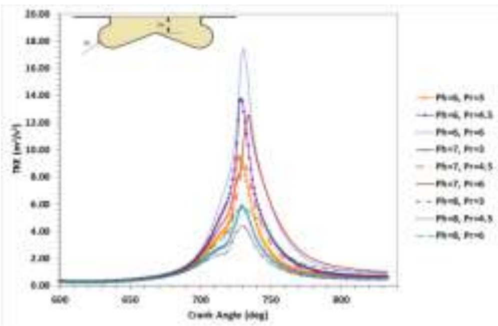

Fig.3.7 TKE Variation Graph

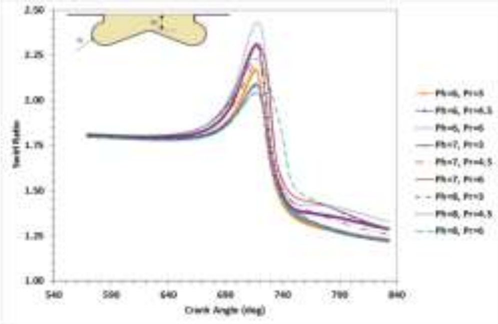

Fig.3.8Swirl Ratio Variation Graph

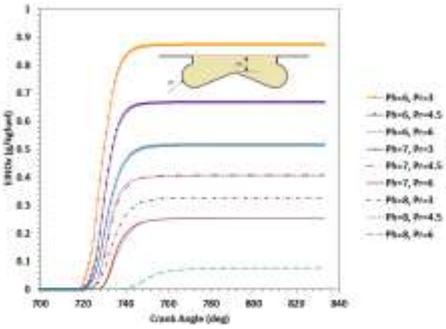

Fig.3.9 NOx Variation Graph

The results from the table 3.1 indicates that keeping piston height constant and as the reentrant radius increases there is a reduction in Indicated mean effective pressure, Indicated power, combustion efficiency and thermal efficiency. Also it indicates that as the increase in reentrant radius the ISFC increases and as the swirl ratio increases which marginally varies the NOx emission.

Figures 3.4 show the effect of compression ratio on the cylinder pressures for diesel engine cases. Clearly, the pressures increase with the increase of compression ratio. This can be easily attributed to the ideal gas law which states pressure is reversely proportional to volume. Very similar trend is found all the parametric analysis case. It shows that higher compression ratio produces higher pressure andtemperature by the end of compression. Then, the higher pressures and temperaturesmake the mixing of fuel and air more rapid and more combustible.The increase of compression ratio produces higher pressure as shownpreviously which means higher power is generated with the same amount of fuel.

Turbulent Kinetic Energy plotis shown in the figure 3.7 at the mid-plane of the modeled 60-degree cylinder sector.The graph indicates that the TKE is maximum for the bowl geometry having more reentrant radius and at moderate piston height.

The degree of swirl in the flow can be quantified by dimensionless parameter.Swirl number (SN) which is defined as the ratio of the axial flux of angularmomentum to the axial flux of axial momentum. Fig. 3.8 shows the effect of swirl ratio on combustion. The case withlow swirl ratio corresponded to swirl number of 1 and the high swirl numbercorresponded to swirl number of 2.5. The results reveal that increasing the swirl makes the combustion faster. In addition,the swirl has marginal effect on emission. The NOX emissions do not changesignificantly by changing the swirl number.Fig. 3.9shows the variation of NOx formation with crank angles at various piston dimensions. It is observed that, the NOx formation increases steadily from the time of spark until for about 20 crank angle degrees after TDC. This is because of increase in the gas temperature beyond 1700 K during combustion. Afterwards their formation is frozen and concentration remains constant up to the EVO. This is because, the gas temperature decreases during the expansion stroke, which inhibits the NOx formation.

There are variations of NOx emissions as a function of the crank angle for different omega shaped with re-entrant geometry configurations shown in figure 3.9. These results show that for higher compression ratio for example Ph=6 mm and Pr=3, 4.5 and 6mm cases had the highest NOx emissions since they have relatively faster and more intensive heat release, which implies higher combustion temperatures. For the case Ph=8 mm and Pr=3, 4.5 and 6mm cases, the combustion was slow and at the low temperatures, NOx emission was low.

IV. Conclusion

In this paper Combustion analysis of single cylinder DI Diesel engine is performed using ANSYS Forte v18.0 software. From this investigation, it is observed that CFD computational study is important to understand in cylinder flow structure during compression stroke on a single cylinder DI Diesel engine with different piston bowl geometries. It shows that piston bowl geometry plays a predominant role in the flow of air pattern inside the cylinder.It is found that, the NOx formation increases marginally with decrease in piston height. This is because of rise in peak in-cylinder pressure which in turn results in the increase in peak in-cylinder temperature. However, it is also observed that, the NOx formation levels are lower at the piston height of 8mm, which is because of slower combustion. The results reveal that increasing the swirl makes the combustion faster. In addition, the swirl has marginal effect on emission. The NOx emissions do not change significantly by changing the swirl number.

References

[1] DOT. 2008; Available from: http://www.fhwa.dot.gov/ohim/onh00/bar4.

[2] Ferguson, C.R. and A.T. Kirkpatrick, Internal Combustion Engines. 2000: John Wiley.

[3] Exxon, M., Annual Energy Outlook. 2005.

[4] Heywood. J.B, "Internal Combustion Engine Fundamentals", McGraw – Hill Book Company, NewYork. (1988)

[5] F. Payri, J. Benajes, X. Margot, A. Gil, "CFD modeling of the in-cylinder flow in direct-injection Diesel engines" Computers & Fluids 33-(2004) 995-1021 ELSEVIER

[6] B.V.V.S.U. Prasad, C.S. Sharma, T.N.C. Anand, R.V. Ravikrishna "High swirl-inducing piston bowls in small diesel engines for emission reduction" Applied Energy 88 (2011) 2355–2367 ELSEVIER