raghavendra Reddy Nv Journal of Engineering Research and Application www.ijera.com
ISSN : 2248-9622, Vol. 8, Issue5 (Part -IV) May 2018, pp 56-61

RESEARCH ARTICLE OPEN ACCESS

CFD Simulation Of Direct Injection CI Engine With Flat Piston Bowl And W Shape Toroidal Piston Bowl Combustion Process

[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
Corresponding Auther: raghavendra Reddy Nv

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 with flat piston bowl and W shape toroidal piston bowl. The validation of the CFD results with the experimental results available for flat piston bowl. The optimization of the bowl geometry for W shape toroidal piston bowl with constant compression ratio has been carried out by varying the central pip height and toroidal 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.

Date of Submission: 07-05-2018 Date of acceptance: 22-05-2018

I. INTRODUCTION

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 [1]. 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 codeANSYS Forte is used for the simulation of fluid motion inside the engine cylinder and combustion process in a DI compression ignition engine of two different bowls shapes one with Flat bowl and another W-shape reentrant piston bowl. Complete simulation of engine cycles and run on any operating conditions. The step by step investigation has been carried out to study the effect of Bowl shapes onswirl ratio on in-cylinder fluid motion, combustion and emission process. 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 has been carried out and iswith good match.

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

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

Where: ρ is the density, subscript k is the species index, Kis 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

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

The momentum equation for the fluid is

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

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

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

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

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

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

cε2ρε+csWs (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_0	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		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	136 mm
Stroke	160 mm
Connecting rod length	304 mm
Number of injector hole	8
Mass ofinjected fuel	0.68mg/st
Start of injection(CA)	-20° [deg BTDC]
Injection duration(CA)	7.5 [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,two different piston bowl shapes are considered. The flat piston bowl and W shape toroidal piston bowl in which the variation was made to the piston bowl geometry in terms of centre pip (Ph) and re-entrant radius (Pr). Nine piston bowl geometry configurations are used in the study, withthe same compression ratio. The geometry of the flat piston bowl and W- shaped Piston bowlare shown in fig 2.1 and 2.2

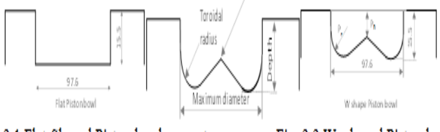

Fig.2.1 Flat-Shaped Piston bowl geometry

Fig. 2.2 W- shaped Piston bowl geometry

Diesel engine emissions are mostly affected by in-cylinder combustion parameters. Diesel piston bowl parameters such as spray impingement position i.e. lip area, toroidal radius in which main combustion occurs, pip height and pip inclination which changes flow pattern and swirl rate in combustion chamber are main factors affecting diesel engine combustion and emissions. Hence by optimizing piston bowl parameters emissions can be reduced to greater extent which is more cost effective.In this piston central pip (Ph) is varied from 6 to 12mm and toroidal radius (Pr) varied from 7.5 mm to 12.5mm.

a. Maximum bowl diameter:

The maximum bowl diameter is defined as the largest diameter parallel with the piston face at any position through a section of the piston bowl. The ratio of the maximum bowl diameter to the bowl depth defines the piston bowl aspect ratio. The piston bowl total volume and thus compression ratio is largely controlled by the maximum bowl diameter. This is one of the first parameters to be set when designing a new piston bowl shape.

b. Toroidal radius (Pr):

The majority of combustion occurs in the main toroidal radius volume.

c. Central Pip (Ph):

The central pip is used to occupy a volume in the Centre of the piston bowl, where the air velocity is low. Low air velocity in the Centre of the swirling flow-field results in poor air/fuel mixing rates. The central pip allowed this volume to be redistributed further from the Centre of rotation, resulting in a higher mean airflow velocity and better air/fuel mixing

ANSYS Forte v18.0 software is used to perform RANS CFD simulations in the present work. Chemkin with 173 species was chosen as the chemistry model. In addition, the spray properties were obtained using the radius of influence model for the droplet collision model, and the radius of influence was set at 0.2 cm. The fuel properties of nC12H26 (dodecane) were chosen, and the discharge coefficient was accepted as 0.7. The boundary conditions, such as piston temperature, head temperature, and liner temperature, were set at 500K. A sector angle of 45° was chosen, with multi-selected boundaries on which the boundary condition was applied. A sector mesh is tremendously useful in constituting a much smaller mesh size. The sector mesh, which is a fine mesh size at top dead center (TDC) is shown Figure 2.3 and at bottom dead center (BDC), is illustrated in Figure 2.4. Injector location is at the centre of the cylinder top. In this simulation RNG k-epsilon turbulence model is used.

The experimental data was obtained by Singh et al.'s study [2] where the test engine is equipped with a non-production, high-pressure, electronically-controlled, common-rail fuel injector. A vaporization model for the fuel surrogate has a composition of 51% n-tetradecane, 35.5% ndecane, and 15.5% 1-methylnaphthalene.

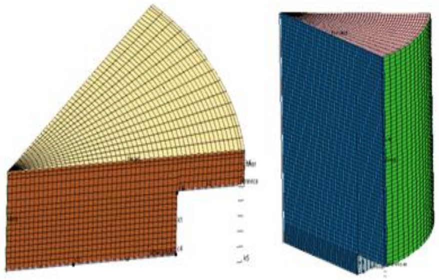

Fig.2.3Sector mesh of in-cylinder fluid domain at TDC Fig.2.4Sector mesh of in-cylinderfluid domain at BDC

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. Flatshaped bowl geometry 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 incylinder pressure and rate of heat release values arevalidated with the experimental results available for flat piston bowl. The results of CFD simulation and experiments with respect to Pressure and Heat release rates are illustrated in Fig.3.1. The difference between CFD and experiment in terms of pressure values was approximately 1.95%. The graph shows a good agreement of the results between the CFD predicted and experimentally measured in-cylinder pressure curves and the rate of heat release curves.

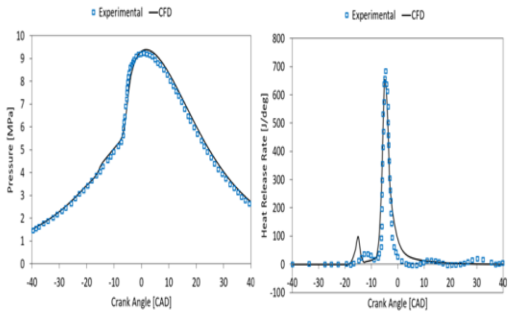

Fig.3.1comparison of cylinder pressure and Heat release rate CFD and experimental values

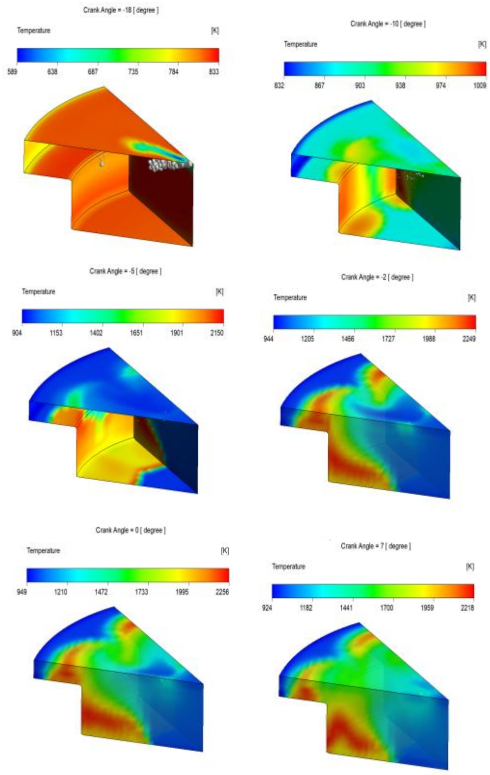

Fig.3.2 Fuel particles and in cylinder temperature contours of flat piston bowl

Fig. 3.2 illustrates the fuel particles and in-cylinder temperature contour from the start of injection to the piston location at the TDC. The fuel particles vaporization, ignition, and combustion are clearly shown. Initially, the fuel particles temperature started to rise from the beginning of the injection and then the fuel particles started to vaporize. As a result of the fuel particles vaporization, gases in the cylinder and fuel vapor constitute homogenous mixture. Ultimately, as a consequence of the fuel and gas mixing, ignition is carried out and combustion begins.

Table 3.1: Results summary of flat and W-Shaped toroidal piston bowl

Table 3.1: Results summary of flat and W- Shaped toroidal piston bowl

Piston Type	Ph	Pr	Max Pressure	CO	NOx	EINOx	Soot	Turbulence Velocity	SR
	mm	mm	[MPa]	(g/kgfuel)	(g/kgfuel)	(g/kgfuel)	(g/kgfuel)	cm/s	
Flat Shape	15.5	0	9.39	531.75	1.39	2.27	0.0518	536.4091	1.14
	6	7.5	9.33	490.96	0.95	1.59	0.0097	571.3642	1.11
	6	10	9.32	484.94	0.98	1.64	0.0102	551.4553	1.12
	6	12.5	9.30	491.64	1.03	1.74	0.0127	549.8734	1.12
	9	7.5	9.29	493.58	0.96	1.61	0.0126	553.8414	1.19
W Shape	9	10	9.33	491.47	0.99	1.66	0.0110	563.3878	1.19
	9	12.5	9.31	484.52	0.98	1.63	0.0084	565.9865	1.18
	12	7.5	9.35	508.95	1.06	1.74	0.0159	602.3021	1.28
	12	10	9.34	476.43	1.01	1.67	0.0108	574.8025	1.22
	12	12.5	9.35	494.19	1.08	1.79	0.0134	586.3098	1.24

Making 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 rate. In the section, a variation was made to the W shape toroidal piston bowl geometry in terms of central pip height (Ph) and toroidal radius (Pr). Nine different parametersare used in the study, while keeping the compression ratios constant with respect todifferent central pip height and toroidal radius.

The following are the graphs drawn with the post processing of the results of Flat and W shape toroidal combustion chamber.

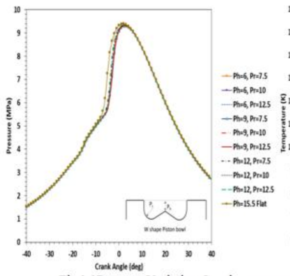

Fig.3.3Pressure Variation Graph

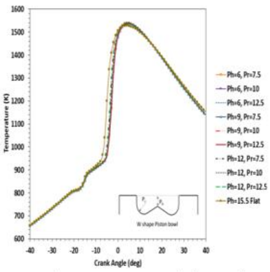

Fig.3.4Temperature Variation Graph

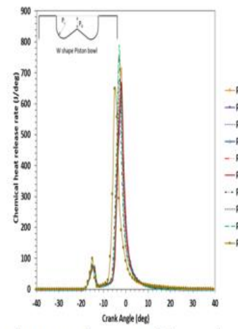

Fig.3.5Heat Release Rate Variation Graph

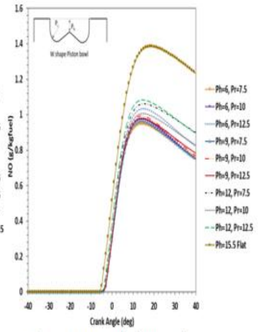

Fig. 3.6 NOx Variation Graph

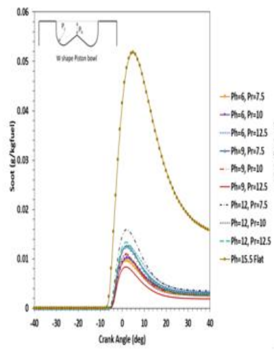

Fig.3.7Soot Variation Graph

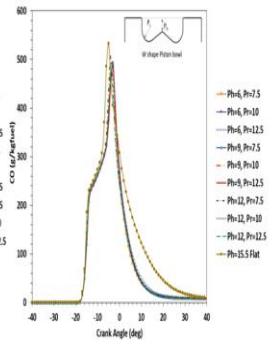

Fig. 3.8 CO Variation Graph

The results from the Fig.3.3 and 3.4 shows the in-cylinder pressure v/s crank angleand Temperature v/s crank angle respectively for W shape toroidal piston bowl configurationsfor variation in Ph and Pr and the Flat piston bowl. It shows that no significant change in the in-cylinder pressures and temperature with variation in piston bowl geometry and also compared to flat piston.

Figure 3.5 shows rate of heat release rate versus crank angle for variation in combustion chamber geometry. From the graphfor Ph=12mm and Pr=12.5mmit shows higher heat release rate compared to other chambers geometries. It may be because of swirl ratio which is around 1.24 which helps in good mixing of air with the injected fuel.

The NOx emission plots are shown in figure 3.6, which shows variations of NOx emissions as a function of the crank angle of W shapetoroidal geometry configurations. These results show that for Ph=12 mm and Pr=12.5 and 7.5mm cases, highest NOx emissions since they have relatively faster and more intensive heat release rate which causes higher combustion temperatures. For the cases Ph=6 and 9 mm and Pr=7.5 and 6mm cases, the combustion was slow hence NOx emission is low. It shows that as central pip (Ph) increases NOx emission increases.

The Soot emission graph is shown in figure 3.7which compares the soot emissions from thedifferent bowl geometry including the flat bowl. Note that Ph=9 and Pr=12.5 geometry chamber provides better soot oxidation compared to others. The soot formed inW shape toroidal geometry is lower than flat piston bowl. This is due to better air–fuel mixing and improved combustion. Also in W shape toroidal geometry it can be seen that soot emissions were observed to be higher for Ph=12 and Pr=10 due to improper mixing fuel and air and also lower in-cylinder temperature leading to in-complete oxidation of soot particles.

Fig. 3.8 shows the CO emissions v/s crank angle in which the formation of CO in the flat chamber is more compared to toroidal chamber due to lower swirl ratio in turn no proper mixing of fuel and air reduces the oxidation.

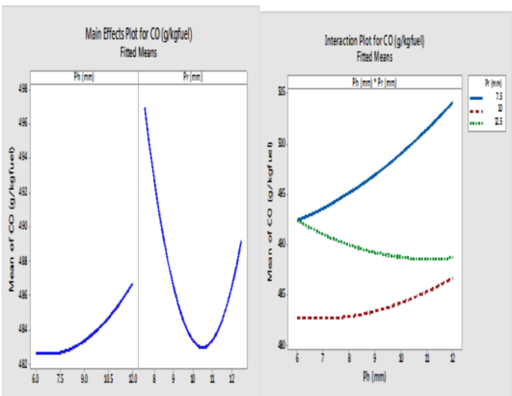

Fig. 3.9 Main Effects and Interaction plots of CO emission

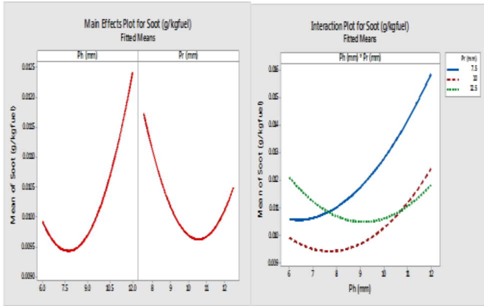

Fig. 3.10 Main Effects and Interaction plots of Soot emission

Fig. 3.10 Main Effects and Interaction plots of Soot emission

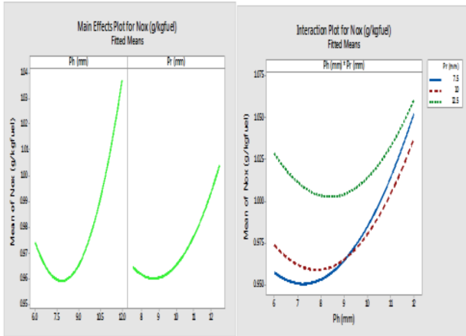

Fig. 3.11 Main Effects and Interaction plots of NOx emission

The above figures show the graphs of optimization ofW shape toroidal piston bowl geometries with respect to emissions. The graphs in figure 3.9 give the effects of CO emission for different Ph and Pr values of W shape toroidal geometry. From the graph we can see that for Ph = 6 to 8 mm and Pr = 10 mm gives less CO emission. The figure 3.10 gives the soot emission and from the graph it can be seen that with Ph= 7.5 to 8.5 mm and Pr= 10 mm gives the less soot formation. The figure 3.11 gives the effects of NOx emissions. The graph shows that Ph=7.5 mm and Pr = 7.5 mm gives the lower NOx emissions.

IV. CONCLUSION

In this paper Combustion analysis of single cylinder DI Diesel engine of flat piston bowl is performed using ANSYS Forte v18.0 software and validation is done for the incylinder pressure and the heat release rate with the available experimental results. There is a good agreement between the experimental and the CFD simulation results. The CFD simulation is performed for the W shape toroidal piston for different geometry values by varying the toroidal radius and the central pip height from the surface of the piston. The results are plotted and optimization of W shape toroidal piston bowl geometry is done by plotting the available results from the CFD analysis. 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, CO and Soot emissions reduces with the small central pip height from the top surface of the piston (Ph) and moderate toroidal radius (Pr). NOx formation increases as the increase in Ph and the Pr value. This is because of rise in peak in-cylinder pressure which in turn results in the increase in peak in-cylinder temperature. The results reveal that increasing the swirl makes the combustion faster.

REFERENCES

[1]. Heywood. J.B, “Internal Combustion Engine Fundamentals”, McGraw – Hill Book Company, New York. (1988)Ferguson, C.R. and A.T. Kirkpatrick, Internal Combustion Engines. 2000: John Wiley.

[2]. Singh, S., Reitz, R., and Musculus, M., “Comparison of the Characteristic Time (CTC), Representative Interactive Flamelet (RIF), and Direct Integration with Detailed Chemistry Combustion Models against Optical Diagnostic Data for Multi-Mode Combustion in a Heavy-Duty DI Diesel Engine,” SAE Paper No. 2006-01-055, Detroit, Michigan, April 2006

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

[4]. 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

[5]. 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

raghavendra Reddy Nv "CFD Simulation Of Direct Injection CI Engine With Flat Piston Bowl And W Shape Toroidal Piston Bowl Combustion Process"International Journal of Engineering Research and Applications(IJERA) , vol. 8, no.5, 2018, pp.56-61