Effect of Piston Bowl Geometry on Various Fluid Flow Patterns

P. Krishna Kaushik[1], D. Sai Swaroop[2]
1, 2. Department of Mechanical Engineering
GITAM University
Hyderabad, India

***Abstract*—This paper presents about a CFD analysis which has been carried out on in-cylinder flow and air fuel interaction simulated on 4 different piston bowls of Diesel engines. There are many ways to optimize the usage of an IC engine, one of them is by achieving high turbulence inside the cylinder. For obtaining different swirl intensities, piston bowl design has been changedin each case. By changing the piston bowl design, turbulence enhancement inside the cylinder is achieved. Four solid piston bowls that were designed in CATIA (Computer Aided Three Dimensional Interactive Application) are Hemispherical Bowl, Mexican Hat, Double Spherical Combustion Chamber and Toroidal Chamber. Triangular meshing is done to all piston bowls in ICEM CFD. Analysis of in-cylinder flow is done in ANSYS Fluent 14.5. From this analysis, the best possible piston bowl has been obtained.**

Keywords*—Diesel engine; air swirl; Turbulence; Turbulent Intensity

I. Introduction

Internal Combustion Engines are the engines where combustion process occurs inside the engine and produces energy out of it. In IC engine, combustion of fuel in the presence of an oxidizer (oxygen) occurs in the cylinder. The pressure due to the expansion of gases produced by the combustion of the fuel applies force on some component of the engine. The force due to expansion of gases is generally applied on piston. The component moves to a particular distance and converts the thermal energy produced by combustion of the fuel to the required mechanical energy.

IC Engines are classified into many types, based on number of cylinders (4 cylinder, 6 cylinder, 8 cylinder, 12 cylinder, 16 cylinder), number of strokes (2 stroke, 4 stroke, 6 stroke), type of thermodynamic cycle employed (Otto cycle, Diesel cycle) and based on the fuel used (gasoline, Diesel). Irrespective of the type of IC engine the basic principle of combustion process i.e., conversion of thermal energy which is produced by the combustion of the fuel to the required mechanical energy is same.

IC engines are majorly employed in many areas like automobiles, locomotives, power plants, generators. The applications of IC engines have escalated drastically in the 20th century. This made the leading manufacturers not only to increase the production but also made them to shift their way of thinking to make optimum usage out of it as fuels that are used to feed any IC engine are non-renewable.

There are many ways to optimize the "power output" of an IC engine. One of the ways is by concentrating on the in-cylinder flow. The best the in-cylinder flow, the better can be the combustion of the fuel. The turbulence in the cylinder has its impact on the combustion process. More the turbulence, more is the mixing of air and fuel. It is ultimately the geometry of combustion chamber that aids the in-cylinder flow. So, there lies the need to analyze the in-cylinder flow with respect to the combustion chamber geometry. The in-cylinder flow doesn't necessarily translate greater efficiency but promotes the engine to run smoothly.

DI Diesel engines, having the evident benefit of a higher thermal efficiency than all other engines, have served for both light- duty and heavy-duty vehicles.

II. Literature Survey

B. Harshavardhan and J. M. Mallikarjuna[1] work on "CFD analysis of in-cylinder flow and air-fuel interaction on different combustion chamber geometry in DISI Engine". They carried the work by fixing the engine speed and crank angle. The engine speed which they took was 1500 rpm. They have concluded that flat piston has slightly high Tumble Ratio when compared to the other piston configurations i.e., dome piston with centre bowl, Pentroof Offset Bowl Piston (POBP), flat piston with centre bowl,the Turbulent Kinetic Energy (TKE) is distributed all over the combustion chamber; TKE of Flat Piston (FP) is high about 12.56% when compared to the POBP. They didn't analyze the contours of turbulent kinetic energy and turbulent intensity of all four geometries.

V. V. PrathibaBharathi and G. prasanthi[2] work on "Influence of in-cylinder air swirl on diesel engines performance and emissions" have concluded that lesser smoke can be achieved by the complete combustion, lesser carbon deposits in the combustion chamber, piston crown and exhaust system occur due to the controlled complete combustion, better fuel economy can be achieved by the improved and complete combustion, there will be raise in the cylinder pressure due to the effective combustion. But Pressure and velocity contours have not been analyzed.

Jinou song, Chunde Yao, Yike Liu and Zejun Jiang[3] work on "Investigation on flow field in simplified piston bowls for Direct Injection diesel engine" have concluded that squish flow plays an important role in the turbulence generation process near the TDC during compression, the coupling among the swirl, squish, bowl shape and turbulence is much more pronounced for the flow fields in the combustion chambers and the piston bowl configurations should be designed to coincide with the contour lines of the turbulence.

C. Morley, R. J. Price, N. P. Tait and C. R. McDonald [4] work on "Understanding how fuels behave in engines" concluded that differences in the fuel composition can sometimes play a significant role on the performance of the engine as they lead to the formation of combustion chamber deposits. Fuel components considerably differ in their deposit forming tendency, combustion chamber deposits in gasoline engines can increase NO_x and hydro carbon emissions.

III. Piston Bowl Specifications

Bore	43.75 mm
Depth	27 mm
Fuel	Diesel
Number of cylinders	one
Number of strokes	4 stroke

Table 1.1 Piston bowl specifications

IV Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the best suited software for the analysis of flow patterns which involve dynamic parameters of the flow. Study of moving particles involves many complex calculations and large variables. But their analysis can be easily carried with the help of CFD.

CFD is based on the finite volume method. The area of the flow is first modelled in any of the modelling software. This is known as Computational domain. The area that has been modelled is discretized into many small elements. The process of discretizing the computational domain into small elements is known as meshing. Hence each volume is considered to be a control volume. The control volume has certain properties like mass, momentum, energy, turbulence quantities and material properties. Based on the flow we pick up the properties of control volume. The theoretical flow in the control volume is represented physically with the help of numerically solvable partial differential equations. These equations govern the flow of fluid in the computational domain. These equations also undergo discretization process for flow analysis.

There are three equations that are applied to all fluid dynamic problems. Those are conservation of mass, momentum and energy equations.

Continuity equation:

$$\frac{\partial y}{\partial x} + \nabla(\rho\, u) = 0$$

Momentum equation:

$$\rho\left(\frac{dv}{dt} + v.\nabla v\right) = -\nabla \mathrm{p} + \nabla \mathrm{T} + \mathrm{f}$$

Energy equation:

$$\frac{\partial}{\partial t}(\rho \mathrm{h}) + \frac{\partial}{\partial x_j}(\rho \mathrm{u_j h}) = -\frac{\partial q_j}{\partial x_j} + \frac{\partial \rho}{\partial t} + \mathrm{u_j}\frac{\partial p}{\partial x_j} + \tau_{ij}\frac{\partial u_i}{\partial x_{ij}}$$

K-ε Turbulence model was used in the analysis, where k represents kinetic energy and ε is the turbulence. In this project, 45^0 degree sector of each piston bowl has been designed in CATIA, as the piston bowls are axi-symmetric and analysis of 360^0 sector would rather take much more span for analysis.As number of nodes is directly proportional to the time for analysis, 45^0 sectors are being designed. Inlet duct of same dimension has been attached to all the piston bowls. Exhaust wall is not designed as we are concerned with the flow but not on the emissions of each piston bowl. Triangular meshing has been done to all piston bowls in ICEM CFD, because triangular grids can cover each and every small part of the component thus resulting in accurate results. Material of bowls was taken as aluminium as it is light in weight and acceptable strength.

IV. Importance of In-Cylinder Flow

As fuels are necessary to run any IC engine, proper usage of fuels must be achieved as they are non-renewable. In IC engines, efficient burning of the fuel can be achieved by proper combustion process. To obtain efficient combustion of the fuel injected in the cylinder of an IC engine with less emissions, fuel must be spatially distributed within the cylinder. To achieve spatial distribution, the fuel sprays must match with geometry of the combustion chamber to effectively make use of gas flows.In other words, matching the combustion chamber geometry, fuel injection and gas flows is the most crucial factor for attaining a better combustion.This can be contributed as a major factor why different swirl and turbulent intensities are obtained for different combustion chamber geometries. Here air is made to swirl for proper mixing with fuel which increases rate of mixing. The higher the swirl reduces the soot emission at the cost of higher NO_xlevel.

The in-cylinder fluid motion in Internal Combustion Engines is one of the most important factors in controlling the combustion process. It governs the air-fuel mixing and rate of burning in diesel engines.The fluid flow prior to combustion in Internal Combustion Engines is generated during the induction process and developed during the compression stroke.So, proper understanding of fluid motion in the cylinder plays an important role in case of designing the engines with most desirable operating and emission characteristics.

Swirl interaction with compression induced squish flow increases turbulence levels in the combustion bowl, promoting mixing. Since the flow in the combustion chamber develops from interaction of the intake flow with the in-cylinder geometry, the goal of this work is to characterize the role of combustion chamber geometry on in-cylinder flow, thus the fuel-air mixing.

It is known that the influence of geometry has a negligible effect on the airflow during the intake stroke and early part of the compression stroke. But when the piston moves towards Top Dead Centre (TDC), the piston bowl geometry has a significant effect on air flow thereby resulting in better combustion.

VI. OBJECTIVE OF THE WORK

The main objective of the work is to do "pressure and velocity distribution" and "Turbulent intensity" analysis on four different piston bowls and choosing the best piston bowl out of four, based on the turbulent intensity.

VI. BOUNDARY CONDITIONS

- Constant pressure boundary conditions at the the inlet port.
- Turbulent intensity -5%
- Mixing length scale-0.001mm
- Initial pressure at start of computation- 0.99 bar and ambient temperature -298 K
- Inlet port pressure- 1.013 bar.

In case of complex geometries where high turbulent intensity is achieved, turbulent intensity ranges from 5-20%. In case of medium complex geometries like pipes, the turbulent intensity ranges from 1-5% and in case of flow that is originated from the stationary fluid, least turbulent intensity is below 1%.

VII. RESUTS AND DISCUSION

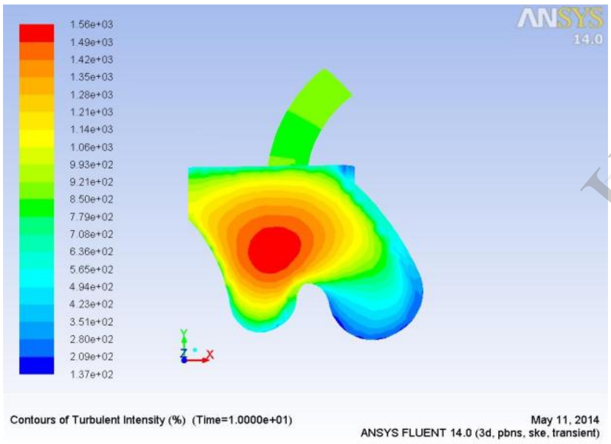

Fig 1 Turbulent Intensity of DSCB Chamber

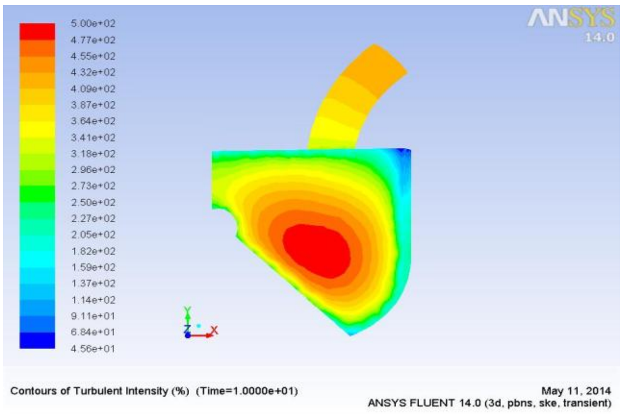

Fig 2 Turbulent Intensity of the Hemispherical Bowl

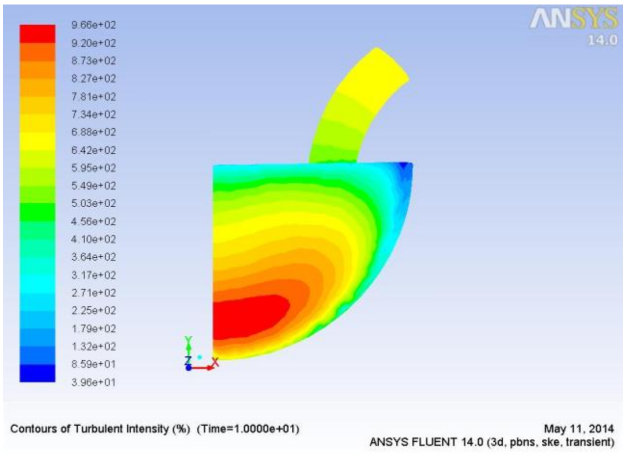

Fig 3Turbulent Intensity of Mexican Hat

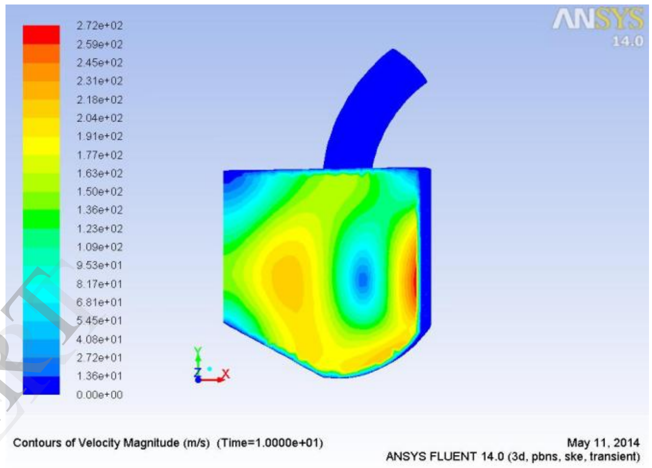

Fig 4 Turbulent Intensity of Toroidal chamber

The above images gives the Turbulent Intensity of 4 piston bowls i.e., Toroidal chamber, Mexican hat, Double Spherical Combustion Chamber and Hemispherical Bowl. In case of in-cylinder flow, Turbulent Intensity plays a major role as it is main cause for the origination of different magnitudes of pressure and velocity in different directions i.e., X,Y,Z directions. Turbulent Kinetic energy plays a major role next to turbulent Intensity in the matter of in-cylinder flow.

The more the turbulence inside the cylinder, the greater is the extent of mixing of air-fuel mixture. Permissible variation in turbulence at the instant of ignition leads to high flame speeds i.e., speed with which the flame produced in the diesel engine is covering the entire combustion chamber thus making the combustion process to complete. The more propagation of flame, more is the fuel burnt and more can be the smoothness of engine. Higher turbulence i.e., the magnitude of turbulence above the permissible value leads to increase in the heat transfer to the cylinder walls. Heat produced in the engine should be converted to the mechanical work, which is the primary principle of an IC engine. If the heat transfer rate to the cylinder walls has increased, heat is being absorbed by the cylinder walls thus reducing the heat content in combustion chamber. If the heat content in the combustion chamber is low, there is a drop in the conversion of heat energy to the desired mechanicalenergy thus reducing the thermal efficiency of the engine. Hence, the turbulent intensity must vary between 5-20% in case of complex geometries where permissible high turbulence is mandatory.

In case of above analysis, the areas with red spots have turbulence more than 20% which is highly undesirable, so, the piston bowl with minimum red spot is considered as the best piston bowl in terms of in-cylinder flow and air-fuel interaction. Toroidal chamber is the best as it has minimum area with red spot and the turbulent intensity in this chamber lies between 5-20%.

VIII. CONCLUSION

Turbulent Intensity must lie between 5-20% according to CFD analysis in case of complex geometries. And high turbulence i.e., turbulence above the permissible variation leads to the rapid heat transfer to the cylinder walls, thus a decrease in Thermal efficiency of the engine. Based on the permissible variation in the Intensity of Turbulence Toroidal chamber has the Turbulent Intensity between 5-20%.

REFERENCES

[1] B. Harshavardhan, J. M. Mallikarjuna, "CFD analysis of in-cylinder flow and air-fuel interaction on different combustion chamber geometry in DISI Engine," I. C Engine Laboratory, IIT Madras,IJTARME, vol. 2, Issue-3 , 2013.

[2] V. V. Prathiba Bharathi, and G. Prasanthi, "Influence of in-cylinder flow air swirl on diesel engine performance and emission",IJAET, vol. 1, pp.113-118. October-December 2011

[3] Jinou Song, Chunde Yao, Yike Liu, and Zejun Jiang, "Investigation od flow field in simplified piston bowls for DI diesel engine," State Key Laboratory of Engines, Tianjin University, EACFM, vol. 2, No. 3, pp. 354-365, 2008

[4] C. Morley, R. J. Price, N. P. Trait and C. R. McDonald, "Understanding how fuels behave in engines," Shell Research and Technology Center at Thornton, United Kingdom, The Fourth International Symposium COMODIA 98 , 1998