Journal of Manufacturing Engineering, March 2019, Vol. 14, Issue. 1, pp 006-009

IN-CYLINDER FLOW ANALYSIS FOR A DIRECT INJECTION DIESEL ENGINE – A CFD APPROACH

https://doi.org/10.37255/jme.v4i1pp006-009

***Ananthakumar S[1], Jayabal S[2] and Thirumal P[1]**

[1]Government College of Engineering, Bargur, Krishnagiri - 635 104, Tamil Nadu, India..
[2]Government College of Engineering, Thanjavur – 613 402, Tamil Nadu, India.

ABSTRACT

A parametric study of the effect of piston bowl configuration on air motion of a direct injection diesel engine motored at 3000 rpm is investigated. Two piston bowl configurations (Mexican-hat and Re-entrant) are modeled for the computational flow analysis. The flow characteristics of these engine bowls are examined under transient conditions using STAR CD, a commercial computational fluid dynamics package. The predicted computational fluid dynamics results of mean swirl velocity of the engine at different locations inside the combustion chamber, at the end of compression stroke were compared with experimental results available in the literature. The results obtained showed very good agreement with the measured data given in the literature. This paper discusses the predicted flow structure inside the combustion chamber at top dead center, with different piston bowl shapes at 3000 rpm. It also compares the radial distribution of mean swirl velocity component in the piston bowl for the two cases. It is observed that the Re-entrant bowl provides a higher swirl ratio at almost all locations than the Mexican hat bowl.

Keywords: *Mean Swirl Velocity, Computational Fluid Dynamics and Flow Analysis.*

1. Introduction

The diesel engine performance is greatly affected by the phenomena occurring inside the combustion chamber, which depends mainly on piston bowl configuration. The piston bowl configuration is closely related to the swirl ratio of the engine [2-5]. In order to maintain the global standards of IC engine performance, multi-dimensional flow simulation is used as an economical tool for the optimum design of an IC engine. For a DI diesel engine, air motion (swirl) inside the engine depends on many parameters like engine speed, port and combustion chamber configuration. Swirl is generated during compression process in DI diesel engines and subsequently it plays a vital role in mixing air and fuel inside the cylinder. It is seen from review of literature, that modeling of combustion chamber system and prediction of in-cylinder flow is essential to achieve better performance of an IC engine.

The main objective of the present study is to analyse the flow inside engine cylinder under transient condition using STAR CD. By using computational fluid dynamics (CFD) simulation, the flow field can be predicted by solving the governing equations viz., continuity, momentum and energy. The renormalization group theory (RNG k-ε) turbulent model was used for analysing the physical phenomena involved in the change of kinetic energy. Two different piston bowls have been considered in this work and in-cylinder flow field investigations have been conducted at 3000 rpm. The engine selected for this simulation work was a single cylinder research DI diesel engine having the geometrical and operating characteristics similar to IVECO direct injection (DI) engine [3]. Present study is aimed at simulating the effect of piston bowl configuration on flow inside a test engine, whose specifications are given Table 1.

Table 1 FIAT Research center engine specifications

Specifications	Value
Bore	93 mm
Stroke	90 mm
Displacement	0.611 litres
Connecting rod length	171 mm
Bowl entry diameter	52 mm
Compression ratio	16
Inlet valve diameter	39.7 mm
Exhaust valve diameter	30 mm
Maximum inlet valve lift	9.1 mm
Inlet valve opening	8 °BTDC
Inlet valve closing	218 °ATDC
Inlet/exhaust valve overlap	16°
Rated speed	3000 rpm
Power	4.5 kW

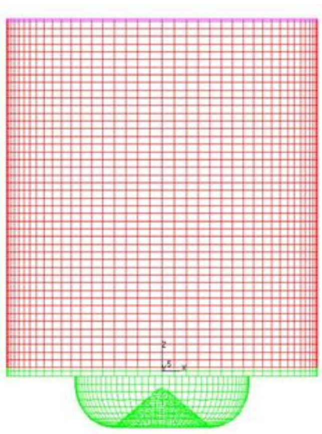
**Corresponding Author - E- mail: ananth202004@gmail.com*

2. Mathematical Model and Governing Equations

The flow inside the cylinder was treated as compressible and by solving momentum, continuity and energy equations and the flow conditions inside the cylinder have been predicted. In the present study, RNG k-ε turbulence model, which is an improved version of k-ε turbulence model, has been used for turbulence modelling. The RNG model was developed using Re-Normalisation Group methods to renormalize the Navier-Stokes equations. It is used to account for the effects of smaller scales of motion. In the k-ε model the eddy viscosity is determined from a single turbulence length

2.1 Pre-Processing and Grid Generation of the Geometry

The pre-processing mainly involves the creation of basic 3D model, grid generation and fixing of the boundary conditions. The creation of the geometry is done in GAMBIT, the mesh generation package of FLUENT. The partially generated grid in GAMBIT is exported to STAR CD for completing the mesh. In this analysis a complete hexahedral structured mesh was created for the piston bowl and cylinder. The overall computational grid for the different piston bowl configuration with cylinder (case1 Mexican-hat, case2 Re-entrant) for the flow simulation is shown in Fig. 1. For performing transient flow simulations it is necessary to use moving grids. The CFD package STAR CD possesses the above capabilities required for the simulation of transient flow cases. Cell layer addition and removal are controlled by event and moving mesh commands. Before starting the flow simulation, the correct piston movement for compression stroke at any crank angle position is verified from the mesh preview.

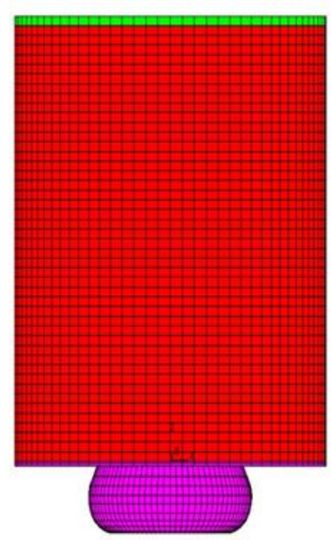


(a) Mexican-hat bowl

2.2. Boundary Conditions

2.2.1 Inlet, Exit and Wall

Pressure boundary was used at top surface of the bowl. The same pressure boundary conditions were adopted for inside the cylinder as well. During the analysis the initial pressure difference between the cylinder and outside was made equal to zero. Attach boundaries were specified on the coincident cell face near the cells in the cylinder and for those non-attached cells wall boundary type is specified to be the alternative boundaries. This attaching and detaching of boundaries were controlled by the event commands. The walls are assumed to be adiabatic with no-slip condition. The temperature inside the cylinder is fixed as 303 K Initial turbulent flow parameters (turbulent intensity and length scale) were assumed as 5% and 1 mm respectively as per normal practice. The simulation was conducted for different piston bowl geometries with the same boundary conditions and for 3000 rpm. At the start of the compression stroke, a swirl ratio of 2.2 is specified as an initial condition [1] for both the Mexican hat and Re-entrant cases.



b) Re-entrant bowl

Fig. 1 Computational grid configuration

3. Validation of the Code

For the purpose of validation, the in-cylinder flow characteristics were compared with the experimental data recorded by Margary *et al* [3]. Good agreement was observed between the simulation and the measurements which were carried out for a single cylinder DI diesel engine (Mexican hat) at 3000 rpm motoring condition. Fig. 2 shows the location of fluid domain selected for the study of the swirl velocity component normalized with piston speed at the end of compression stroke. Also the figure shows the measurement locations for predicting the mean swirl velocity at top dead center (TDC) of compression inside the bowl. These points are 'r' mm radially away from the cylinder axis and 'z' mm from the cylinder head (r =10, 12, 14, 16, 18, 20, and 22 mm, and z = 3, 6, and 9 mm).

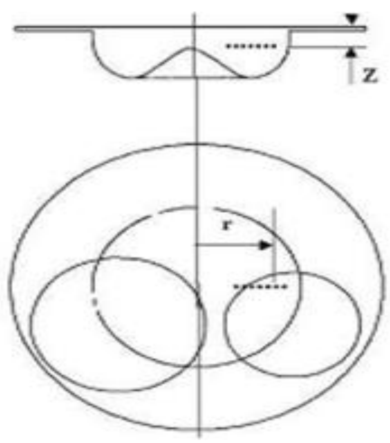


Fig. 2 Location of fluid domain selected for the study of the swirl velocity

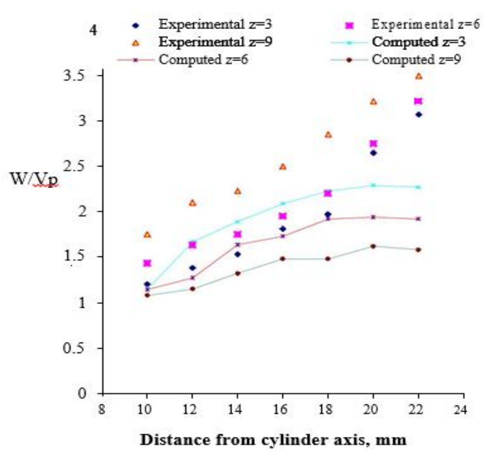


Fig. 3 Radial distribution of mean swirl velocity (W/Vp) in the piston bowl (Experimental Vs. Computed) at TDC for 3000 rpm.

Fig. 3 shows the comparison of experimental mean swirl velocity component (W/Vp) in the piston bowl at TDC of compression for the Mexican-hat bowl. The results from the computations show good agreement with experimental result in the literature [3]. The maximum deviation was 12.72 % at the location of r =18 mm and z = 6 mm.

4. Result and Discussion

Fig. 4 resulted that the swirl ratio for Re-entrant combustion chamber is maximum near to the cylinder head (z =3 mm), at 3000 rpm. The mean swirl velocity component (W/Vp) normalized with piston speed increases as it moves away from the cylinder axis. In all the locations, Re-entrant configuration shows very high swirl mean velocity component than the Mexican-hat combustion chamber. The Mexican bowl has high wall friction due to large area volume ratio and diffusion flow than Re-entrant bowl; hence the swirl velocity decays greatly.

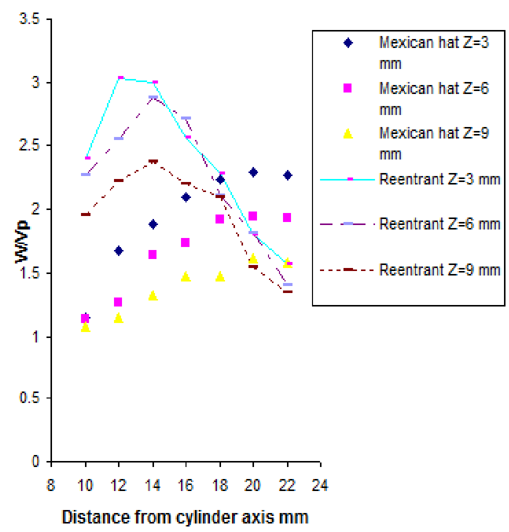


Fig. 4 Radial distribution of mean swirl velocity (W/Vp) in the piston bowl (Mexican hat Vs. Re-entrant) at TDC for 3000 rpm

5. Conclusion

It is concluded that Re-entrant geometry creates higher velocity component (W/Vp) inside the combustion chamber at the end of compression stroke, than the Mexican hat bowl. Therefore, the Re-entrant chamber would result in better mixing of air and fuel leading to better combustion.

References

1. *Benny Paul and Ganesan V (2005), "CFD analysis of the port configurations on air motion inside the cylinder of a DI diesel engine", Proceedings of the 19th National conference on I.C. Engines and Combustion, Annamalai University, India.*

2. *Lin L, Shulin D, Jin X, Jinxiang W and Xiaohong, G (2000), "Effects of combustion chamber geometry on in-cylinder air motion and performance in DI diesel engine", SAE paper 01-0510.*

3. *Margary R, Nino E and Vafidis C (1990), "The effect of intake duct length on the in cylinder motion in a motored diesel engine", SAE paper 900057.*

4. *Payri F and Benajes J (2004), "CFD modeling of the in-cylinder flow in direct-injection Diesel engines", Computers & Fluids, Vol. 33, 995-1021.*

5. *Prasad B.V.V.S.U and Ravikrishna R.V (2005), "CFD Simulation of in-cylinder air flow in a DI diesel engine - Effect of combustion chamber geometry", Proceedings of the 19th National conference on I.C. Engines and Combustion, Annamalai University, India.*

