

Analysis of In-Cylinder Air Motion in a DI Diesel Engine with Four Different Piston Bowl Configuration - A CFD and PIV Comparison

A Gnana Sagaya Raj, J M Mallikarjuna, and Ganesan Venkitachalam
Indian Institute of Technology Madras

ABSTRACT

Air motion inside the engine cylinder plays a predominant role on combustion and emission processes. An attempt has been made in this investigation to simulate the in-cylinder air motion in a DI diesel engine with four different piston configurations such as dome piston, bowl on dome and pentroof piston and pentroof offset bowl piston. For computational analysis, the commercial general purpose code STAR-CD Es-ice has been used, which works on the method of finite volume. To validate the simulation, qualitative and quantitative comparisons have been done with the PIV results available in the literature. From this study, the best possible piston configuration has been arrived at.

CITATION: Sagaya Raj, A., Mallikarjuna, J., and Venkitachalam, G., "Analysis of In-Cylinder Air Motion in a DI Diesel Engine with Four Different Piston Bowl Configuration - A CFD and PIV Comparison," *SAE Int. J. Engines* 6(4):2013, doi: 10.4271/2013-01-2786.

INTRODUCTION

The combustion chamber design plays a predominant role on fuel-air interaction, combustion and emission processes in DI diesel engines. Particularly, the design parameters involved in the bowl configuration such as pip region, bowl lip area and torus radius must be designed properly to achieve a proper in-cylinder air-fuel mixing and complete combustion [1]. Lu Lin et al [2] simulated the effects of three different types of combustion chambers (central projection type, shallow 'w' type and pataloid type) on in-cylinder air motion in a DI diesel engine by multi dimensioning modeling and found that the central projection combustion chamber exhibited a rapid combustion process and shortened the main combustion duration.

Dolak et al [3] have used a KIVA-CHEMKIN code for the computational analysis on light duty engine with stepped bowl piston geometry and found out that stepped bowl could oxidize more CO emission than conventional bowl. Anttinen et al [4] investigated the flow field measurements in a heavy duty diesel engine with piston bowl by particle image velocimetry technique and their results revealed that the generated swirl was not located symmetrically at the centre of the piston bowl.

Auriemma et al [5] carried out the computation in fluid dynamic code STAR-CD to study the in-cylinder distribution of air velocity in HDDI Diesel engine and reported that there is a formation of two main vortices at the top of the cylinder. Shimoda et al [6] studied the effect of HMMS-III (Hino micro mixing system) combustion chamber configuration on in-cylinder air motion in a DI Diesel engine and proved that the HMMS-III combustion chamber induced a more turbulence, produced four vortex at the four concave cavity, promoted a strong mixing during the diffusion burning period, reduced the combustion duration, fuel consumption and exhaust smoke and also it controlled the position of vortex centre of cavity swirl.

Matsuoka et al [7] compared the analysis of in-cylinder air motion through LDA technique and a mathematical model. Their results revealed that the swirl velocity was accelerated from the upper part of the combustion bowl due to the squish flow in the compression stroke and also it was decelerated from the upper part of the combustion bowl because of the reverse squish in the expansion stroke. Corcione et al [8] analysed the effect of engine speed on in-cylinder turbulent air motion in an optical DI Diesel engine equipped with a re-entrant combustion chamber and motored at an engine speed of 1000-3000 rpm. They proved that the ensemble average tangential velocity showed almost a flat

profile at lower engine speeds and there was a rapid increase in velocity from 310 CAD to 345 CAD during higher engine speeds. Murali Krishna and Mallikarjuna [9] studied the in-cylinder tumble flow in an internal combustion engines with different piston shapes by particle velocimetry technique and proved that at the end of compression, there was an improvement in tumble ratio and turbulent kinetic energy by 41% and 103% compared to that of flat piston. Raj et al [10] investigated the energy efficient piston configuration for effective air motion using a STAR-CD CFD code in a DI diesel engine. They used a four different piston configuration such as flat piston, centre bowl on flat piston, inclined and inclined offset bowl piston. From their investigation, they stated that centre bowl on flat piston can be used for modern DI diesel on the basis of good tumble ratio, turbulent kinetic energy, turbulence intensity and length scale.

In order to investigate the in-cylinder fluid flows, various researchers have adopted the different visualization techniques like meta-aldehyde method, spark tracing method, tuft method, oil film method, hot wire anemometry (HWA), laser Doppler anemometry (LDA), schlieren photography etc. But these methods have some disadvantages like less accuracy, probe type of measurement, difficult to measure three-dimensional flow field, cumbersome, etc. Therefore, CFD codes such as STAR-CD, AVL Fire, and KIVA are extensively used nowadays in fluid flow measurements in many applications. Numerical investigations are essentially important in internal combustion (IC) engines due to complex nature of the in-cylinder fluid flows to reduce cost, time and manpower.

Therefore in this investigation, STAR-CD code has been used to study the in-cylinder flow field in a DI Diesel engine with different piston bowl configurations such as dome piston, central bowl on dome and pent roof piston and pent roof offset bowl piston.

GEOMETRICAL MODELING DETAILS

The CFD predictions were performed using the mesh size of 0.35 millions cells to compare with the PIV results available in the literature [10]. The polyhedral trimming method was adopted for meshing the geometry using CD-Adapco's CFD code STAR-CD Es-ice, version 4.14. It is a commercial general purpose code, which is based on the finite volume method. In this code, the mass, momentum and energy equations are solved to predict the flow. The standard $k-\epsilon$ model is used for turbulence. The differencing scheme used for the analysis is MARS (monotone advection and reconstruction differencing scheme) for the flow variables like U, V and W momentum in order to get a good convergence.

The generated grid was decomposed for 8 processor and it is a replica of a DI Diesel engine with four combustion chambers namely dome piston, central bowl on dome and pentroof piston and pentroof offset bowl piston, which is

shown in Fig.1 with top and isometric view. The detailed specifications of the engine are shown in Table 1. The computational analysis is carried out only for the intake and compression stroke (i.e. 390 CAD to 690 CAD) to study the in-cylinder air motion in a DI Diesel engine and also to validate the predicted results with the experimental results (PIV). The time taken for the computation is about 24 hours in Open Suse 11.3-Linux based Vega cluster consisting of 8 GB RAM and eight 3.0 GHz Intel Xeon processor.

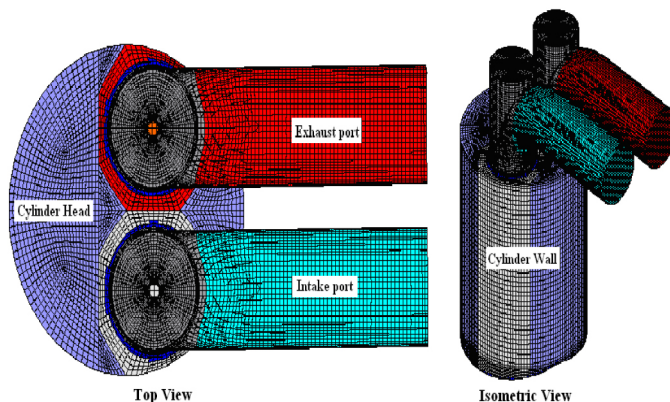


Fig 1. Computational grid of DI Diesel engine

Table 1. Detailed Specifications of the Engine

Bore (mm)	87.5
Stroke (mm)	110
Connecting rod length (mm)	232
Compression ratio	10:1
Maximum intake valve lift (mm)	7.9
IVO/IVC	4.5° bTDC/35° aBDC
EVO/EVC	35° bBDC/4.5° aTDC
Combustion Chambers used	Dome bowl, Central bowl on Dome and Pent roof, Pent roof Offset bowl

GRID REFINEMENT STUDY

The purpose of grid refinement study is to find out the optimal mesh size of the model for predicting almost accurate results to compare with PIV results. Initially the computational grid of the diesel engine with 0.35 million cells is created for pentroof offset combustion chamber configuration. The initial conditions are taken as 1.1 bar and 310 K. The boundary temperatures for cylinder wall, piston crown and cylinder head are taken as 600 K, 550 K and 500