CFD Analysis of in-Cylinder Flow and Air-Fuel Interaction on Different Combustion Chamber Geometry in DISI Engine

B. Harshavardhan & J. M. Mallikarjuna
I. C. Engine Laboratory, Indian Institute of Technology Madras, Chennai-36, India
E-mail: harshavardhan.ballapu@gmail.com and jmmallik@iitm.ac.in

Abstract – In this investigation, a CFD analysis has been carried out on in-cylinder fluid flows and air-fuel interaction in Direct Injection Spark Ignition (DISI) engine by changing combustion chamber geometry during intake and compression stroke at an engine speed of 1500 rpm for four different types of piston profiles viz., flat piston, flat piston with centre bowl, dome piston with centre bowl and pentroof offset bowl piston. A polyhedral trimmed cell has been taken for meshing of the geometries using STAR-CD Es-ice code. Simulation of in-cylinder flow has been done by solving mass, momentum and energy equations by using SIMPLE algorithm, and simulation of air-fuel interaction has been carried out by solving Reitz-Diwakar droplet breakup model, Bai droplet-wall interaction model and Huh atomization model. From this analysis, the best possible combustion chamber geometry has been obtained.

Keywords – Direct injection spark ignition engine, Combustion chamber, Tumble Ratio, Turbulent kinetic energy.

I. INTRODUCTION

In cylinder flow structure plays important role in internal combustion engines while mixing of air-fuel, combustion process and burning rate. In DISI engine development study of air-fuel interaction is very important because of injecting fuel directly in to the cylinder so analysis of in cylinder flow and air-fuel interaction is important for DISI engine development. Intake flow pattern depends on moving piston also creates large scale rotating flow pattern within the cylinder [1]. In this view combustion chamber shapes studying are important during intake and compression stroke. Limited literature availability on effect of piston shapes like flat, flat central bowl, dome bowl and pent roof offset bowl piston on DISI engine development, so that the study of different combustion shapes is important on in-cylinder flow and air-fuel interaction for DISI engine development. Muralikrishna et al have conducted experiment on DISI engine with different piston crown shapes under motoring condition using Particle Image Velocimetry technique (PIV). They concluded that overall in cylinder tumble flows are more depend on crank angle position irrespective of engine speed. Flat piston is better choice in terms of tumble ratio (TR) and average tumble kinetic energy (TKE) compared to other piston configurations [2]. Huang R F et al have conducted experiment on four valve four stroke engine with different piston crown shapes under motoring conditions using PIV technique. They reported that flat crown piston induces higher TR and turbulence intensity compare to concave piston [3]. Rudolf H et al have conducted experiment on four valves Spark Ignition engine with different speeds under motoring condition using Laser Doppler Velocimetry technique. They observed that the shape of fuel spray was depend on the in cylinder air motion generated by intake air flow [4]. Parrish et al have conducted experiment on motoring engine with Elastic scattering planar sheet imaging. They reported that spray distributions of different injection timings and different engine loads are effected differently by in-cylinder air motion [5]. Terrence Alger et al conducted experiment on four valve spark ignition engine with different speeds and different fuel injection timing under motoring condition to visualize spray pattern by laser sheet of 532nm. They observed that effect of intake flow is differently on fuel distribution and local equivalence ratio based on operating condition [6].

The above literature study indicates that the tumble motion created during intake and compression stroke it depends on piston crown shapes. This tumble motion effects the air-fuel interaction. Therefore this investigation going with the visualization of velocity vector plots, TKE plots at end of suction stroke where fuel is injecting and equivalence ratio (ϕ) plot of air fuel distribution at 30BTDC of compression stroke where the
spark to be ignited in DISI engine with different piston configurations.

II. GEOMETRICAL AND COMPUTATIONAL DETAILS

The engine was used for CFD analysis is shown in table 1; it has straight intake and exhaust port. The numerical prediction are calculated during intake and compression stroke from 360 to 720 CAD for four different piston geometries such as flat piston (FP), flat with centre bowl piston (FBP), dome with centre bowl piston (DBP) and pent roof offset bowl piston (POBP). The governing equations are continuity equation, momentum equation, energy equation and k-ε turbulent model equations were solved for in cylinder flow analysis [7]. For getting air-fuel interactions Reitz-Diwakar, droplet breakup model, Bai droplet-wall interaction model and Huh atomization model are used [8]. Fuel injector was located at the centre of geometry, it was injected at the end of suction stroke 540 CAD with injection timing 20 CAD, injector nozzle hole diameter was 0.2mm and mass flow rate was 0.012kg/s.

Two valve four stroke engine geometry of four different piston shapes surface modeling work has been done with Pro-E wildfire 4.0. Polyhedral grid was generated using STAR-CD Es-ice. Meshing of different geometry cut section passing through the centre is shown in fig.1. Grid independence test was conducted with three different number of cells size these are 1.8Lakhs, 3.6Lakhs and 6.2lakhs, from these test it was found that 3.6 Lakhs cells are good agreement in terms of computational cost and time. For checking correctness of compression ratio of all piston configurations of engine compression pressure was measured and it indicates that the different combustion chamber at 10 compression ratio pressure is almost same.

<table>
<thead>
<tr>
<th>Type</th>
<th>Two Valve, Four stroke</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore</td>
<td>87.5mm</td>
</tr>
<tr>
<td>Stroke</td>
<td>110mm</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>10</td>
</tr>
<tr>
<td>Speed</td>
<td>1000rpm</td>
</tr>
<tr>
<td>Maximum intake valve lift</td>
<td>7.9mm</td>
</tr>
<tr>
<td>IVO : 4.5CAD bTDC</td>
<td>EVO : 35 CAD bBDC</td>
</tr>
<tr>
<td>IVC : 35 CAD aBDC</td>
<td>EVC : 4.5 CAD aTDC</td>
</tr>
</tbody>
</table>

Table 1: Engine Specifications

III. CALCULATION OF TR AND TKE

For CFD analysis, the parameters TKE and TR are calculated on the center plane of the engine geometry. The front view of the measurement plane along with the locations (green dots in Fig.2.) at which data is extracted in CFD analysis is shown in Fig.2. 2D Velocity data at these points is extracted for the calculation of TR. The TKE is obtained on the centre plane by extracting data from TKE contour plot.

TR in CFD analysis is calculated by extracting axial and radial velocity components over the center plane at the locations shown in Fig.2. It is calculated by considering the rotation of radial and axial components of velocity of the flow at selected locations about the center of the plane considered using below equation.

\[
TR = \frac{60T_z}{2\pi R_x R_y} = \sum [u_i(z_i) - w_i(x_i)]
\]
and $T_z = \sum (x_i^2 + z_i^2)$.

Where, $T_z$ represents the sum of angular momentum of fluid particles at location ‘i’ about the axis perpendicular to the plane considered, $x_i$ and $z_i$ are the distances from the centre of the plane to the point $i$ and, $u_i$ and $w_i$ are the $u$ and $w$ component of velocity at the point $i$, $\Omega_c$ is the angular speed of the crankshaft in rev/min.

IV. RESULTS AND DISCUSSION:

Fig. 5. shows the plot of velocity vectors at the end of suction stroke with different piston geometries in the plane passing through centre of the geometry. It is observed from the vector plot that counter clockwise tumble is formed in all four different piston geometries configurations, tumble vortex is almost centre of combustion chamber in flat bowl piston geometry, it may be because of centre bowl. Tumble vortex centre is little bit shifted towards exhaust side to all other three cases. The tumble velocity is little bit higher for flat piston configuration compared to other three piston configurations. From observing fig.5 the velocity scale is not varying so much with different piston configurations. Existing tumble in all the cases helps improving air fuel mixing process. From above observations it is clear that fuel injector can be located in centre of geometry so that better air fuel mixing will be happening.

Fig. 3. shows the variation of tumble ratio with different piston profiles; from fig.3 it is observed that FP has high TR (0.227) compared to other three piston profiles (DBP-0.216, POBP-0.212, and FBP-0.221). It may be uniform space in combustion chamber above flat piston. FP has high TR by about 7.1% as compared to POBP.

Fig.6. shows the turbulent kinetic energy contour plot at the end of suction stroke with different piston geometries in the plane passing through centre of the geometry. It is pointed out from figure that TKE is uniformly distributed all over the combustion chamber in all the four piston configurations. Fig.4. Shows variation of TKE with different piston profiles, from fig.4. It is found that FP has more TKE (56.98$m^2$/s$^2$) compared to other three piston profiles (DBP-51.86, POBP-50.62, FBP-54.57). FP has TKE by about 12.56% as compared to POBP.

Fig.7. shows the equivalence ratio ($\phi$) contour plot during the end of compression stroke with different geometries of piston in the plane passing through centre of geometry. It was observed from the figure that in all the four piston configurations fuel was distributed all over the combustion chamber. It may be due to in all four piston configurations TR and TKE are sufficient for proper mixing of air-fuel. In all the four configurations $\phi$ is slightly rich at the centre, it gives location of spark plug at centre plane of the geometry nearer to fuel injector. It was also observed from fig.7 that $\phi$ was distributed almost uniformly in POBP engine, in case of FBP and DBP $\phi$ was distributed across the combustion, in case of FP $\phi$ was distributed along the axis of engine. FP distribution $\phi$ is required for DISI engine to get gas forces towards the piston and then power output of engine would be increased.
Fig 5. Velocity (m/s) vector plots of in-cylinder at 540CAD (at the end of suction stroke)

- a) Flat piston
- b) Flat piston with centre bowl
- c) Dome piston with centre bowl
- d) Pentroof offset bowl piston

Fig. 6. TKE (m^2/s^2) Contour plots at 540 CAD (at the end of Suction stroke)

- a) Flat piston
- b) Flat piston with centre bowl
- c) Dome piston with centre bowl
- d) Pentroof offset bowl piston
V. CONCLUSION

From this investigation, it is observed that CFD computation study is important to understand in-cylinder flow structures during intake and compression stroke. It affects the air-fuel mixing process in a DISI engine. The following conclusion can be drawn based on above computational study.

- At 540 CAD, tumble vortex was formed all the piston configurations, it is located centre in flat bowl piston other cases it is located slightly away from centre.
- Flat piston has slightly high TR compared to other piston configurations; it has high TR about 7.1% compared to POBP.
- The TKE is distributed all over the combustion chamber; TKE of FP is high about 12.56% compared to POBP.
- The Air-fuel mixture is distributed all over the combustion space in all the piston configurations. Distribution of $\phi$ is slightly rich at the centre of geometry.
- Distribution of $\phi$ is along the axis in case of FP engine, $\phi$ is across combustion chamber in case of FBP and DBP, $\phi$ is uniform in case of POBP.

From this investigation flat piston is suitable for combustion chamber of DISI in terms better TKE, high TR, ignitable air-fuel mixture at the spark plug location, better power output and easy of piston manufacturing.

VI. REFERENCES


