A Study of Numerical Simulation of In-Cylinder Cold Flow Process in a Di Diesel Engine for a Different Piston Bowl Shapes

Sadashiv Lalasangi 1, Lokesh V2,
1,2Assistant Professors, Mechanical Department, Srinivas Institute of Technology, Karnataka, India

Abstract—The gas motion inside the engine cylinder plays a very important role in determining the thermal efficiency of an internal combustion engine. A better understanding of in cylinder gas motion will be helpful in optimizing engine design parameters. An attempt will be made to study the in cylinder engine processes in a compression ignition engine with different combustion chamber bowl shape.

The piston bowl design is one of the most important factors that affect the air/fuel mixing and the subsequent combustion and pollutant formation processes in a direct injection diesel engine. The bowl geometry and dimensions, such as the pip region, the bowl lip area and the torus radius, are all known to have an effect on the in-cylinder mixing and combustion process. In order to understand better the effect of the pip region, piston bowls with pip designs and lip area and torus radius were investigated using computational fluid dynamics (CFD) software without combustion process.

A turbulent cold flow process will be solved for a different piston 4-stroke diesel engine. The unsteady compressible conservation equations for mass (Continuity), axial and radial momentum, energy, species concentration equations can express the flow field and combustion in engine cylinder. Finally results shows significant differences in the flow field around TDC were observed for the various bowl shapes by comparing each of them.

The widely considered assumption, that the radial distribution of the tangential velocity can be approximated by solid body rotation seems not to be applicable in cases where the swirl–squish interaction is strong. Indeed, strong squish tends to deform significantly the tangential velocity profiles. In particular, in high swirl chambers formed vortex may be considered as uniform only at the core of the upper bowl part and in the bottom of the bowl, in general.

Keywords: Put your keywords here, keywords are separated by comma.

Moreover CFD allows insight into the minute flow details which otherwise are not capture using flow bench tests.

Understanding the nature of the flows and combustion in internal combustion engines are important for improving engine performance. The flows in IC engines can be characterized by swirl, tumble and compression in the cylinder. This flow motion has a strong influence on the engine combustion process and hence on the engine emission of pollutants. Recently simulation results by Computational Fluid Dynamics codes are used in the development and optimization of new engines by car manufacturers (automotive industry). The in-cylinder fluid motion in internal combustion engines is one of the most important factors controlling the combustion process. Swirl and tumble are well known approaches for in-cylinder flow enhancement. Swirl and tumble are generated in the intake stroke as a result of the inlet port shape and orientations.

Multidimensional modeling became as an important tool for investigating flow and combustion in reciprocal engines. In this type of modeling, the physical processes of flow and combustion in-cylinder governed by partial differential equations are solved with suitable boundary conditions. There are many numerical studies in the literature about multi-dimensional modeling of internal combustion engines.

Simulation Driven Product Development (SDPD) has gained huge significance in the last two decades and today plays a very effective role in bridging the gap between conceptual design and mass production. SDPD essentially combines several analysis processes together with simulation and design optimization being two of the many such processes.

The diesel engines is a type of internal combustion engine, more specifically it is a compression ignition engine, in which the fuel is ignited solely by the high temperature created by compression of the air-fuel mixture. The engine operates using the diesel cycle. The parts of an engine vary depending on the engine's type. For a four-stroke engine, key parts of the engine include the crankshaft, one or more cam shaft and valves

I. INTRODUCTION:

Development of any internal combustion engine is driven primarily by fuel efficiency and emission requirements. This requires refinement of the in cylinder flow, mixture formation and combustion processes. The use of Computational Fluid Dynamics (CFD) along with optimization tools can help shorten the design optimization cycle time. Traditional approach of experiments using flow bench testing is very costly as well as time consuming.

II. LITERATURE REVIEW

J. Benajes[3] In the main study, the flow characteristics inside the engine cylinder equipped with different piston configurations were compared. For this, complete calculations of the intake and compression strokes were performed under realistic operating conditions and the ensemble-averaged velocity and turbulence flow fields
obtained in each combustion chamber analyzed in detail. The results confirmed that the piston geometry had little influence on the in-cylinder flow during the intake stroke and the first part of the compression stroke. However, the bowl shape plays a significant role near TDC and in the early stage of the expansion stroke by controlling both the ensemble-averaged mean and the turbulence velocity fields.

Semin, Rosli Abu Bakar[2] The result of the intake and exhaust flow simulation in an engine cylinder shows that the simulation commences at bottom dead Centre (BDC) and continues for a time interval corresponding to 7200 of crank revolution. Inert scalars with the physical properties of air are used to track the intake and exhaust stream through the exhaust induction strokes in engine cylinder. The port boundary pressures are held constant at ambient condition throughout the simulation. Mesh design in problems with moving mesh and changing cell connectivity is dominated by the need to keep the dynamic parts of the grid simple so that they can be easily changed during the transient run. The formulation of mesh motion in such problems is divided into two conceptual steps. The first deals with connectivity changed which are defined by PROSTAR events. The second step is to specify the grid vertex positions as function of time by supplying a set of PROSTAR grid-manipulation commands to be executed at each time step.

Gosman and Harvey [4]-analyzed fuel-air mixing and combustion in an axisymmetric direct injection diesel engine numerically. For this purpose, they improved the multidimensional model and code developed by the authors. They found that the model produces qualitatively realistic results predictions of the major phases of the combustion process, including ignition, premixed burning and diffusion burning.

Y.Takenaka[1] In this study the intake flow is calculated for the steady state and transient condition. A comparison between the experiment and calculation is performed. Details of flow in both cases are clarified from the calculated results. Especially, the difference between the steady state and transient case is investigated.

The conclusions of this study are:

- The numerical simulation for the intake port valve cylinder flow has been achieved both in the steady state and transient condition with moving valve and piston.

- The comparison between the experiment and calculation in a steady state case shows that the agreement is quite good for the flow pattern and absolute values of the velocity. The good agreement is also found for the flow rate.

Benny Paul[5] reported a study on the effect of helical, spiral, and helical-spiral combination manifold configuration on air motion and turbulence inside the cylinder of a Direct Injection (DI) diesel engine motored at 3000 rpm. Three-dimensional model of the manifolds and the cylinder is created and meshed using the pre-processor GAMBIT. The flow characteristics of these engine manifolds are examined under transient conditions using Computational Fluid Dynamics (CFD) code STAR-CD. The predicted CFD results of mean swirl velocity of the engine at different locations inside the combustion chamber at the end of compression stroke are compared with experimental results available in the literature. We also compared the volumetric efficiency of the modeled helical manifold. The results obtained showed reasonably good agreement with the measured data given in the literature. Further, this paper discusses the predicted flow structure, swirl velocity and variation of turbulent energy inside the cylinder with different manifold. Comparisons of volumetric efficiency with different manifold configuration at 3000 rpm speed are also presented. The turbulence is modeled using RNG k-ε model. It is observed that helical-spiral manifold gives the maximum swirl ratio inside the cylinder than helical manifold. But volumetric efficiency observed is less for helical-spiral manifold engine. Swirl inside the engine is important for diesel engine. Hence, for better performance a helical-spiral inlet manifold configuration is recommended.

Summary: - The jet-like character of the intake flow, interacting with cylinder walls, moving piston and bowl shapes creates large scale rotating flow patterns within the cylinder. These flows appear to become unstable, either during the intake stroke or the compression stroke process, and break down into three-dimensional motions.

Multidimensional modeling became as an important tool for investigating flow and combustion in reciprocal engines. In this type of modeling, the physical processes of flow and combustion in-cylinder governed by partial differential equations are solved with suitable boundary conditions. There are many numerical studies in the literature about multidimensional modeling of internal combustion engines. A cold flow analysis can be performed for this purpose. Cold flow simulations for IC engines can provide valuable design information to engineers. These simulations allow for the effect on volume efficiency and/or swirl and tumble characteristics to be predicted based on changes in port and combustion chamber design, valve lift timing, or other parameters.

III. METHODOLOGY

A. Geometric Model Creation

Geometries can be created top-down or bottom-up. Top-down refers to an approach where the computational domain is created by performing logical operations on primitive shapes such as cylinders, bricks, and spheres. Bottom-up refers to an approach where one first creates vertices (points), connects those to form edges (lines), connects the edges to create faces, and combines the faces to create volumes. Geometries can be created using the same pre-processor software that is used to create the grid, or created using other programs (e.g., CAD, graphics). Geometry files are imported into HM to create computational domain. The geometry of combustion chambers is shown in fig.1.

| Table 1: Geometrical characteristics of the engine |
Geometrical characteristics of the engine

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore</td>
<td>130.0 mm</td>
</tr>
<tr>
<td>Stroke</td>
<td>150.0 mm</td>
</tr>
<tr>
<td>Connecting rod length</td>
<td>275.0 mm</td>
</tr>
<tr>
<td>Displacement</td>
<td>1991 cm³</td>
</tr>
<tr>
<td>Intake valve diameter</td>
<td>44.4 mm</td>
</tr>
</tbody>
</table>

Fig: 1 shows Geometry of the combustion chambers

Fig: 2 shows CFD Model of IC Engine Piston at TDC

IV. RESULTS AND DISCUSSIONS

A. Piston Bowl A, B and C

Table: 1 Comparison of pressure for different crank angles of piston bowls A, B and C

<table>
<thead>
<tr>
<th>Crank angle in degree</th>
<th>CASE A</th>
<th>CASE B</th>
<th>CASE C</th>
</tr>
</thead>
<tbody>
<tr>
<td>401</td>
<td>4.84*10²</td>
<td>4.77*10²</td>
<td>4.89*10²</td>
</tr>
<tr>
<td>461</td>
<td>223</td>
<td>52.8</td>
<td>309</td>
</tr>
<tr>
<td>521</td>
<td>218</td>
<td>256</td>
<td>203</td>
</tr>
<tr>
<td>581</td>
<td>5160</td>
<td>5160</td>
<td>5560</td>
</tr>
<tr>
<td>636</td>
<td>70700</td>
<td>71200</td>
<td>71800</td>
</tr>
<tr>
<td>696</td>
<td>1.36*10⁶</td>
<td>1.43*10⁶</td>
<td>1.51*10⁶</td>
</tr>
</tbody>
</table>

Table: 2 Comparison of velocity for different crank angles of piston bowls A, B and C

B. Geometry Decomposition-Mesh generation
CASE A	CASE B	CASE C
Crank angle in degree
Velocity in m/s
401	306	305	303
461	204	203	206
521	32.9	32.8	34.4
581	54.7	55.1	54.07
636	13.7	16.1	15.8
696	12.9	10.9	10.9

Fig: 5 shows pressure for different crank angles of piston bowls A, B and C

Fig: 6 shows velocity for different crank angles of piston bowls A, B and C

Note that each increment of a time step is equals to an increment of 1° of crank angle. The cases mentioned are the comparison between the diesel engine bowl shapes. Piston starts from TDC about 360 degrees and the maximum pressure reaches at 720 degree. At the start of combustion after the ignition delay there is a sudden change of slope of the p-0 curve. The pressure rises rapidly for a few crank angle degrees, and then moves slowly towards a peak value.

For piston A the turbulent velocity decays almost linearly as it approaches TDC. Since the bowl geometry is open to the radial motion is weak and the squish effect is small. It can observe that turbulence dissipation is faster than the turbulence generation rate.

For piston B the turbulence velocity is peak close to TDC. In all the other locations velocity decreases and the pressure increases. As shown in the graph 5 for different cases and for different crank angles the case B is better than case A for the required pressure and turbulence.

For piston C the highest pressure and turbulent velocities appears at TDC for locations close to head plane. During the compression stroke the stratified swirl structure obtained at the end of the intake stroke is maintained. However as the compression advances a gradual increase of the swirl velocity in the top part of the piston.

As per the comparisons shown in graphs 5 and 6 and tables 1 and 2 for the three different bowl shapes It shows case 3 has high pressure compared to other two chambers which improves engine performance by better combustion.

V. CONCLUSIONS

In this chapter, the results from the modeling and CFD simulation using FLUENT software are shown and discussed in terms of graphs for pressure and velocity distributions. The resulting flow field was analyzed for different combustion chamber shapes using CFD software Fluent.

1. The flow field at the end of compression stroke was calculated for three different bowl shapes and the results were analyzed in detail, in particular around TDC measurements for three cases. It was observed that the geometry of the piston had a negligible effect on the flow characteristics during the first phase of the compression stroke.

2. In case C for crank angle degree 696° when the piston reach near to TDC the pressure rise is more compared to piston bowl A and B since the pip region is wide and plane the pressure here in this case for 696° is 1.51*10^6 pa which is comparatively higher than case A (1.36*10^6 pa) and B (1.46*10^6 pa). In this case peak pressure will get and the velocity of flow is 10.9m/s since the turbulent effect is more at TDC. From this we can achieve greater amount of combustion for these flows.

3. Early in the compression stroke, the swirl velocity field becomes more homogeneous in the whole cylinder as the piston moves upwards. The turbulence velocity decays slightly after the closure of the intake stroke (in the case B) and the pressure dropped (52.8pa) at crank angle 461. But the swirl ratios are identical near TDC where the diameters are similar for the 3 piston bowls.

4. It shows case 3 has high pressure compared to other two chambers and higher pressure improves engine performance, better combustion and the efficiency of engine can be improved. The results shows 1% of efficient for case C than A & B.

5. This indicates that combustion chambers of large bowl diameter maintain some memory of the non-
symmetrical flow generated by the intake. The bowl diameter plays an important role in the structure of the flow only around TDC

REFERENCES


[3] Gosman and Harvey has studied “Numerical solution of flow and combustion in an ax symmetric internal combustion engine”.


