INTAKE VALVE DESIGN FOR HIGH LEVEL SWIRL INDUCTION IN CARBURETED SPARK IGNITION ENGINE

Pandey K. M.* and Roy Bidesh
Department of Mechanical Engineering, N.I.T Silchar, Assam (INDIA)

Received May 05,2012 Accepted September 10,2012

ABSTRACT
The air standard efficiency for SI engine is approximately 60% under full load condition but the actual brake thermal efficiency under full load condition is approximately 32.6% which is due to the various losses that occur. One of the primary loss is burning time loss which is approximately 4% and occurs due to finite time combustion of the charge. This lose can be reduced to some extent by generation of a higher degree of swirl which will increase turbulence intensity with in the engine cylinder. Keeping the above points in view, in this paper, an analysis is performed in a carbureted SI engine using Computational Fluid Dynamic (CFD) code FLUENT to determine the level of swirl induced. The computation is performed again on the same engine for another three different types of intake valve. Finally on the basic for the data achieved from the four computation, a new type of intake valve has been designed and the CFD analysis is also carried out for it at different crank angle during the suction stroke of the engine and at corresponding valve lift and intake pressure and thus, a detail investigation of the newly design intake valve in terms of the level of swirl is carried out.

Key Words : Swirl, Turbulence intensity, Swirl ratio, Computational Fluid Dynamic (CFD), SI engine

INTRODUCTION
The in-cylinder flows of Internal Combustion Engine (ICE) have drawn much attention to the automotive researchers and scientist in the present times. It is due to the fact that the flow structure generated by intake flows is related closely to the design and performance of the Internal Combustion Engines. The production of turbulence of higher intensity is one of the most important factors for stabilizing the ignition process, fast propagation of flame, especially in case of lean-burn combustion In general, two type of vortices are utilized in order to generated and preserve the turbulence flows efficiently. These vortices are usually known as swirl and tumble flows, which are organized rotations in the horizontal and vertical plane of the engine cylinder, respectively. They contribute to the improvement of engine performance. Hence, it is indispensable for the development of an ICE with high compression ratio to realize high turbulence intensity and lean burn combustion. Many researchers worked in this area via experimental as well as computational to explorer the phenomenon of the in-cylinder flow of Internal Combustion Engine. Some of them are cited here. B. Reveille and A. Duparchy worked on 3D CFD analysis of an abnormally rapid combustion phenomenon in downsized gasoline engines. This paper has focused on a particular abnormally rapid, yet non destructive and seemingly stable combustion phenomena which have been identified
on low speed mid to high load operating points when performing aggressive downsizings on various engines. Franz X. Tanner & Seshasai Srinivasan worked on CFD-based optimization of fuel injection strategies in a diesel engine using an adaptive gradient method. A gradient-based optimization tool has been developed and, in conjunction with a CFD code, utilized in the search of new optimal fuel injection strategies. The approach taken uses a steepest descent method with an adaptive cost function, where the line search is performed with a backtracking algorithm. Vijaya Kumar Cheeda, R. Vinod Kumar and G. Nagarajan worked on design and CFD analysis of a regenerator for a turboshift helicopter engine. In this paper a continuous heat transfer regenerator for a turboshift helicopter engine is designed suitably. The regenerator effectiveness is assessed by the CFD tool CFX and evaluated the effectiveness and the pressure drop. The predicted CFD results are in good agreement with experimental results. L. Li, X.F. Peng, and T. Liu worked on combustion and cooling performance in an aero-engine annular combustor. The investigation was conducted to understand the characteristics of the flow, combustion, cooling performance and their interaction in an aero-engine combustor. The conservation equations and Eddy-dissipation combustion model were employed for solving the flow, heat transfer and combustion in the entire combustor. The reliability of the simulation was demonstrated by comparing calculated combustor exit temperature distributions with profiles of the rig-test measurements. Christian Hasse Volker Sohm, and Bodo Durst worked on Numerical investigation of cyclic variations in gasoline engines using a hybrid URANS/LES modeling approach. The study investigates the feasibility of using the SST DES model to predict cycle to cycle variations in internal combustion engines and the effect of cyclic variations in engines and their root causes including the major flow patterns. Wendy Hardyono Kumiawan, Shahrir Abdullah and Azhari Shamsudeen worked on CFD study of cold-flow analysis for mixture preparation in a motored four-stroke direct injection engine. In this study, the CFD simulation to investigate the effect of piston crown to the fluid flow field inside the combustion chamber of a four-stroke direct injection automobile engine under the motoring condition is presented. The analysis is focused on study of the effect of the piston shape to the fluid flow characteristics the result obtained from the analysis could be employed to examine the homogeneity of air-fuel mixture structure for better combustion process and engine performance. Andras Kadocsa, Reinhard Tatschl and Gergely Kristof worked on analysis of spray evolution in internal combustion engines using numerical simulation. This paper summarizes results of research about a new approach of spray formation calculations. Using a primary breakup model for separately describing the initial liquid disintegration of injected liquid based on the flow properties stemming from a previous calculation of injector nozzle flow gives a better prediction capability and suits the new needs of advanced combustion systems such as HCCI engines or various forms of split injection. Toyoshige Shibata Hideo Matsui, Masao Tsubouchi and Minoru Katsurada worked on evaluation of CFD tools applied to engine coolant flow analysis. This paper presents the results of test application of some automatic mesh generation tools to the CFD calculation of coolant flow, and compares the functional characteristics and features of these tools. The paper also discusses coolant flow items that can be evaluated by CFD analysis and the merits of applying CFD to these items. Semin, N.M.I.N. Ibrahim, Rosli A. Bakar and Abdul R. Ismail worked on in-cylinder flow through piston-port engines modeling using dynamic mesh. This paper presents numerical study of three-dimensional analysis of two-stroke spark-ignition cross loop-scavenged port.

AIMS AND OBJECTIVES

The objective of this study is to investigate the in-cylinder characteristics at motored transient condition. The pressure on in-cylinder and intake port were collected and applied for validation with numerical results for 1400 rpm.

MATERIAL AND METHODS

The three-dimensional modeling analysis was performed utilizing dynamic mesh method. The prediction of distribution of in-cylinder pressure and mass fraction of gases function of crank angle were discussed. The results shown that the
relative error between experimental and numerical less than 2%. Helmut Doleisch\textsuperscript{10} worked on simvis: interactive visual analysis of large and time-dependent 3d simulation data. In this paper the major new technological concepts of the SimVis approach are presented and real-world application examples are given. SimVis is a system for the graphical analysis of simulation data, built on a new, cutting-edge technological approach for interactive visual analysis of large, multi-dimensional, and time-dependent data sets resulting from CFD simulation. S. M. Jameel Basha, P. Issac Prasad and K. Rajagopal\textsuperscript{11} worked on simulation of in-cylinder processes in a DI diesel engine with various injection timings. In this paper an attempt has been made to study the combustion processes in a compression ignition engine and simulation was done using computational fluid dynamic (CFD) code Fluent. An Axisymmetric turbulent combustion flow with heat transfer is to be modeled for a flat 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 axisymmetric engine cylinder. Turbulent flow modeling and combustion modeling was analyzed in formulating and developing a model for combustion process. W.H. Kurniawan, S. Abdullah and A. Shamsudeen\textsuperscript{12} worked on turbulence and heat transfer analysis of intake and compression stroke in automotive 4-stroke direct injection engine. This paper shows that in-cylinder CFD predictions yield a reasonable result that allows improving the knowledge of the in-cylinder flow pattern and characteristics during the intake and compression strokes instead of using the experimental test. R. Rezaei, S. Pischinger, P. Adomeit and J. Ewald\textsuperscript{13} worked on evaluation of CI in-cylinder flow using optical and numerical techniques. In this paper different port concepts for modern compression-ignition engines, usually quantities as the swirl level and the flow coefficient are evaluated, which are measured on a stationary flow test bench. As additional criterion, in this work, the homogeneity of the swirl flow is introduced and defined quantitatively. Different valve lift strategies are evaluated using three-dimensional particle imaging velocimetry in a stationary flow configuration and transient in-cylinder CFD simulation using both the reynolds averaged Navier stokes equation and the large Eddy simulation approach. M.M. Noor1, K. Kadigama1, R. Devarajan, M. R. M. Rejab, N. M. Zuki N. M. and T. F. Yusaf\textsuperscript{14} worked on development of a high pressure compressed natural gas mixer for a 1.5 Litre CNG-diesel dual engine. In this paper Computational Fluid Dynamics (CFD) analysis software was used to study the flow behavior of Compressed Natural Gas (CNG) and air in a CNG-air mixer to be introduced through the air inlet of a CNG-Diesel dual fuel stationary engine. Yasar Deger, Burkhard Simperl and Luis P. Jimenez\textsuperscript{15} worked on coupled CFD-FE-analysis for the exhaust manifold of a diesel Engine. This paper aims to investigate the thermo-mechanical behaviour of an exhaust manifold which has an active cooling system, the full water flow, partial water flow (by 50% reduced cooling flow) and vapour flow three cases of cooling analyzed. Fluid flow, thermal heat transfer and stress analysis are coupled for each case using a one-way coupling approach. Selected results given in form of temperature, stress and displacement distribution plots in this paper. The investigation was focusing on potential structural optimization measures. Therefore some suggestions for design improvements are presented also, which are presumably effective to reduce the temperature peaks and temperature gradients and to ensure a longer service life for the exhaust manifold. Kihyung Lee, Choongsik Bae, and Kernyong Kang\textsuperscript{16} worked on the effects of tumble and swirl flows on flame propagation in a four-valve S.I. engine. The effects of in-cylinder flow patterns, such as tumble and swirl flows, on combustion were experimentally investigated in a four-valve S.I. engine. Tumble flows were generated by intake ports with entry angles of 25, 20 and 15. Inclined tumble (swirl) flows were induced by two different swirl control valves. The initial flame propagation was visualized by an ICCD camera, the images of which were analyzed to compare the enflamed area and the displacement of initial flames. The combustion duration was also calculated by the heat release analysis. B. Murali Krishna and J. M. Mallikarjuna\textsuperscript{17} worked on tumble flow analysis in an unfired engine using particle image velocimetry.
This paper deals with the experimental investigations of the in-cylinder tumble flows in an unfired internal combustion engine with a flat piston at the engine speeds ranging from 400 to 1000 rev/min., and also with the dome and dome-cavity pistons at an engine speed of 1000 rev/min., using particle image velocimetry and It is suggested in the paper to use the flat piston rather than dome, dome-cavity pistons which are rather difficult to manufacture as far as tumble flows are concerned. B. Khalighi worked on study of the intake tumble motion by flow visualization and PTV. The purpose of this work is to characterize the in-cylinder tumbling flow generated by an engine head during the induction process using flow visualization and PTV. The study was carried out for a 4-valve engine head with shrouded intake valves in special single cylinder transient water analog. This shrouded intake valve configuration was used to obtain a prototypical pure tumble flow suitable for fundamental combustion studies.

**Specification of the SI engine**

The engine considered for the computation analysis is a single-cylinder carbureted four stroke naturally aspirated SI engine with cylindrical combustion chamber and single intake port and exhaust port. The computation analysis is performed at WOT maximum power condition. The specification of engine is listed in Table 1.

<table>
<thead>
<tr>
<th>Engine specifications</th>
<th>Calculation conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore x stroke</td>
<td>96mm x 99mm</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>9:1</td>
</tr>
<tr>
<td>Piston cavity</td>
<td>Flat</td>
</tr>
<tr>
<td>Max power at WOT</td>
<td>13.5 BHP at 4800 RPM</td>
</tr>
<tr>
<td>Intake valve diameter</td>
<td>42mm</td>
</tr>
<tr>
<td>Maximum intake valve lift</td>
<td>12mm.</td>
</tr>
<tr>
<td>Exhaust valve opening</td>
<td>65° BBDC</td>
</tr>
<tr>
<td>Exhaust valve closure</td>
<td>5° ATDC</td>
</tr>
<tr>
<td>Intake valve opening</td>
<td>5° BTDC</td>
</tr>
<tr>
<td>Intake valve closure</td>
<td>60° ABDC</td>
</tr>
<tr>
<td>Fuel</td>
<td>C_{h_{18}}</td>
</tr>
<tr>
<td>BSFC</td>
<td>0.459 lbm/hp-hr</td>
</tr>
</tbody>
</table>

**Modified intake valve for computational investigation**

For the study three new types intake valve has been modeled by making simple modification in the geometry of the Poppet intake valve in order to analysis the effect produce by the change in geometry on the level of intake swirl generation within the engine.

**Poppet intake valve**

A Poppet intake valve is used in the SI engine in which the computational analysis is performed. The dimensions of the Poppet intake valve are shown in the Fig.1

**Intake valve Model-1**

Intake valve Model-1 is the first modification of Poppet intake valve for the CFD analysis. The dimensions of Intake valve Model-1 are shown in the Fig. 2 below

**Intake valve Model-2**

Intake valve Model-2 is the second modification of Poppet intake valve for the CFD analysis. The dimensions of Intake valve Model-2 are shown in the Fig. 3 below

**Intake valve Model-3**

Intake valve Model-3 is the third modification of Poppet intake valve for the CFD analysis. The
dimensions of Intake valve Model-3 are shown in the Fig. 4 below:

**Computational domain and boundary conditions**

The numerical formulation of the problem is incomplete without prescribing boundary conditions, which correspond to the specific physical model. The specification of mathematically correct boundary conditions that ensure the uniqueness of the solution, while being compatible with the physics at the boundaries, is not always straightforward. Before arriving at the boundary conditions at various boundaries, we have to first identify the solution/computational domain of the problem. The physical domain and computational domain usually differ. However, the computational domain largely depends on the geometry of physical domain. The computational domain boundary (truncated from the real boundary) along with appropriate boundary
conditions should be chosen in such a way that there is negligible change in the results with further increase in its size.

The computational domain shown in the Fig. 5 is a generalized one since, the analysis is performed at different crank angle during the suction stroke of the engine as result the distance of the piston from the engine head shown in the Fig. 5 by “B” also varies corresponding to the engine crank angle. The boundary conditions shown in Fig. 5: are as follows:

- **Inlet boundary on the inlet port of the engine:** The inlet boundary condition is assigned as mass flow inlet.
- **Solid surface of the cylinder of the engine:** It is assigned wall boundary condition i.e. no slip condition on the solid surface of the cylinder.
- **Outlet Boundary on the piston of the engine:** Outlet boundary is assigned the pressure outlet boundary condition.

![Fig. 5: Computational domain and boundary conditions (All dimension in mm)](image)

### RESULTS AND DISCUSSION

**Poppet Intake Valve at 72 crank angle**
The nature of swirling flow in actual operating engine is very difficult to determine. Swirl ratio is a dimensionless parameter used to quantify swirling flow within the cylinder as shown by the equation below

\[
SR(\theta) = \frac{|v(\theta) \times 60|}{2\pi r}
\]

From the equation 1, it is clear that tangential velocity plays a vital role in determining the intensity of swirl within the engine.

From the results of the computation analysis carried out at 72 crank angle with poppet intake valve, intake model-1, 2 and 3 on the above mention SI engine, we can see that the tangential velocity produced by the incoming charge on using the poppet intake valve high than the other intake valve considered in the study. Hence, poppet intake valve is the better design compare to the other intake valve considered for the analysis.

From the above computational results it is also seen that the surface at 9.18 mm from engine cylinder head which is closer to the valve shows higher tangential velocity at various location compared to the surface at 28.8 mm from engine cylinder head which is at higher distance from the intake valve.

From the results of the computation analysis carried out at 123 crank angle with poppet intake valve, intake model-1, 2 and 3 on the above mention SI engine, we can see again that the tangential velocity produced by the incoming charge on using the poppet intake valve is high than the other intake valve considered. Hence, poppet intake valve is the better design compare to the other intake valve considered for the analysis.
From the above computational results it is also seen that the surface at 15.3mm from engine cylinder head which is closer to the valve shows higher tangential velocity at various location compared to the surface at 61.2mm from engine cylinder head which is at higher distance from the intake valve. Hence the computational result for 123° crank angle is similar in nature to that of the computational result for 72° crank angle although the magnitude of tangential velocity varies for both the cases. (Table 2 to Table 9)

Table 2: Contour plot of tangential velocity (m/sec)

- For surface located at 9.18mm from Engine cylinder head.
- For surface located at 28.8mm from Engine cylinder head.

1) Poppet intake valve at 72° crank angle

Table 3: Contour plot of tangential velocity (m/sec)

- For surface located at 9.18mm from Engine cylinder head.
- For surface located at 28.8mm from Engine cylinder head.

2) Intake valve model-1 at 72° crank angle

Table 4: Contour plot of tangential velocity (m/sec)

- For surface located at 9.18mm from Engine cylinder head.
- For surface located at 28.8mm from Engine cylinder head.

3) Intake valve model-2 at 72° crank angle

Table 5: Contour plot of tangential velocity (m/sec)

- For surface located at 9.18mm from Engine cylinder head.
- For surface located at 28.8mm from Engine cylinder head.

4) Intake valve model-3 at 72° crank angle
Table 6: Contour plot of tangential velocity (m/sec)

For surface located at 15.3mm from Engine cylinder head.

1) Poppet intake valve at 123° crank angle

For surface located at 61.2mm from Engine cylinder head.

Table 7: Contour plot of tangential velocity (m/sec)

For surface located at 15.3mm from Engine cylinder head.

2) Poppet intake model 1 at 123° crank angle

Table 8: Contour plot of tangential velocity (m/sec)

For surface located at 15.3mm from Engine cylinder head.

3) Poppet intake model 2 at 123° crank angle

For surface located at 61.2mm from Engine cylinder head.

Table 9: Contour plot of tangential velocity (m/sec)

For surface located at 15.3mm from Engine cylinder head.

4) Poppet intake model 3 at 123° crank angle

For surface located at 61.2mm from Engine cylinder head.
High swirl induction intake valve

From the previous section it is seen that the poppet valve is more effective than any other intake valve in terms of producing high level of intake swirl. So, in order to further increase the intake swirl generation capacity of the poppet intake valve, two curve blades are incorporated on its neck. Curve blades on the neck of the poppet valve are preferred over the conventional shrouded poppet valve since; it will provide lesser blockage to the incoming charge and hence, will result in higher volumetric efficiency the conventional shrouded poppet valve. (Fig. 6 and Fig. 7)

Computational result for high swirl induction intake valve

Computational result at 72 crank angle for high swirl induction intake valve.
The computational result for high swirl induction intake valve at 72° crank angle shows that the tangential velocity produced by the incoming charge is higher compared to the tangential velocity produced by the incoming charge on using the poppet intake valve at same crank angle. From the above computational it is also seen that the surface at 9.18mm from engine cylinder head which is closer to the valve shows higher tangential velocity at various location compared to the surface at 28.8mm from engine cylinder head which is at higher distance from the intake valve. Computational result at 123 crank angle for high swirl induction intake valve.
The computational result for high swirl induction intake valve at 123 crank angle shows that the tangential velocity produced by the incoming charge is higher compared to the tangential velocity produced by the incoming charge on using the poppet intake valve at same crank angle. From the above computational it is again seen that the surface at 15.3mm from engine cylinder head which is closer to the valve shows higher tangential velocity at various location compared to the surface at 61.2mm from engine cylinder head which is at higher distance from the intake valve. (Table 10 and Table 11).
CONCLUSION

From this study the following point can be concluded.
Poppet intake valve is the better design in term of intake swirl generation within the engine out of the three new types intake valve has been modeled by making simple modification in the geometry of the Poppet intake valve. By incorporated two curve blades are on its neck the poppet intake valve the intensity intake swirl generation within the engine increased significantly.
From the computational analysis it is seen that in all the cases the surface which is closer to the valve shows higher tangential velocity at various location compared to the surface at which is at higher distance from the intake valve i.e. the intensity of swirl decreases along the stroke length of the engine cylinder.
The intensity intake swirl generation within the engine decreased with the increase of the valve lift for all the cases.

REFERENCES

5. Christian H. , Volker S. and Bodo D., Numerical investigation of cyclic variations in gasoline engines using a hybrid URANS/LES modeling approach, Computers and


