In-Cylinder Flow through Piston-Port Engines Modeling using Dynamic Mesh

Semin, N.M.I.N. Ibrahim, Rosli A. Bakar and Abdul R. Ismail

Automotive Excellent Center, Faculty of Mechanical Engineering, University Malaysia Pahang, Locked Bag 12, 25000 Kuantan, Pahang, Malaysia.

Abstract: This paper presents numerical study of three-dimensional analysis of two-stroke spark-ignition (SI) cross loop-scavenged port. 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. 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 that 2%.

Key words: Computational fluid dynamics, in-cylinder flow, modeling, two stroke engines

INTRODUCTION

The internal combustion (IC) engines started when Nicolaus A. Otto first developed the spark-ignition engine and Rudolf Diesel invented the compression-ignition engine\(^1\). Since then, many researches have focused on the combustion process. Contemporary engine research has attempted to maximize the power produced from fuel combustion while minimizing pollutants and other regulated compounds. The difficulty is that there exit many variables that affect these treatments; the piston speed, the piston chamber geometry, inlet and exhaust port design, fuel composition, in-cylinder fluid dynamics and ignition devices used. A comprehensive and detail study is needed to embrace all the factors. Two-stroke technology was widely studied by several well-known engine manufacturers. The advantages of two-stroke engines over four-stroke engine are evident. A compact size and a low weight with respect to engine output and potentially smaller mechanical losses are the main advantages\(^2\).

The application of multidimensional calculation methods for detailed investigation of the in-cylinder flow processes in the loop-scavenged two-stroke engine more popular. The research laboratories of General motors model the multidimensional modeling of two-stroke engine scavenging\(^3\), Blair\(^4\), and Heywood and Sher\(^4\) were reported multidimensional numerical simulations of the loop-scavenging process in two-stroke engine. In their extensive study Blair\(^4\) was employed the multidimensional method to investigate the scavenging characteristics on scavenging process.

The author has been gained experience from use of CFD coded called PHOENICS and StarCD. Those CFD coded are developed for the simulation of wide variety of fluid flow processes. Detailed study of type of experimental explained by Heywood and Sher\(^4\) measured on motored and fired engine. Ahmadi-Befrui\(^4\) used multidimensional computational method to investigate the details of the in-cylinder flow and gas exchange process in a loop-scavenged SI engine. The inlet and outlet boundary conditions were obtained from a gas dynamic calculation. Results shows the in-cylinder flow structure early during the scavenge phase comprise of a three dimensional. Kang\(^7\) modified version of KIVA-II code to solve the scavenging flow simulation of a four-poppet-valve two-stroke engine. The standard \(k-\varepsilon\) turbulence model is used with no slip on the wall. Grid is generated through direct interface with three dimensional CAD data using a commercial CAE package. Results show reasonable trends for variation of the velocity field and fresh air mass fraction distribution with crank angle.

The new subroutines are developed by Xiaofeng\(^8\) to handle various valve shrouds. Visualization of scavenging flow on a modified two-stroke transparent cylinder engine is used for validation and compared well in flow pattern. Raghunathan\(^9\) was simulated flows within a motored two port loop-scavenged two stroke engine. The simulation are carried out using the Star-CD CFD code and employs a multi-block approach to simulate the flow within the transfer duct, cylinder and exhaust duct. A moving mesh with cell layer activation-deactivation was used to present the reciprocating piston motion. Eligott\(^10\) simulated the
simulation of the stratified scavenging using computational fluid dynamics (CFD) code Vectis, showed good correlation with measured results. The simulation provided a real insight into the cylinder flow behaviour of the separate fuel and air streams entering the cylinder. Tulus[11] simulated of in-cylinder pressures and temperatures in the combustion chamber of two-stroke linear combustion engine incorporating combustion and kickback chambers. The problem is a combination of compressible and incompressible fluid flow, ignition of mixture, and heat transfer problem. The problem is analyzed transiently using a finite volume method to a three-dimensional model of a system that consists of a combustion chamber, intake manifold, and exhaust port. The fuel used in the analysis is gasoline.

This paper aims to provide some insight on ability of a CFD code to simulate the flow within a loop-scavenged two-stroke engine under dynamic conditions under motored conditions. The validation under motored conditions is necessary step before attempting to validate under firing engine.

MATERIALS AND METHODS

A single cylinder piston-ported two-stroke engine loop-scavenged crankcase compressed is used in this research. The specification of engine with flat form piston surface is shown in Table 1.

Three-dimensional structural model of TANAKA-328A engine was developed using CAD software for the transient flow analysis. The specifications of the engine are mentioned in[12]. Figure 1 is shows the loop-scavenged engine design presented in half-cut model. The engine has been used in earlier studies by Bakar[12].

Due to the symmetry of the cylinder port layout, it is only necessary to model half of the geometry[9,13,14]. The mesh was constructed using three separate blocks representing the intake port, cylinder and exhaust port of the engine. Their cell faces at the interface between each other matched exactly setting in the TGRID stage. The overall mesh structure for the piston at the TDC is shown in Figure 2. There are overall comprising 7964 cells for the whole components at the TDC position. The volume is divided and the mesh to fulfill the requirement of the dynamic mesh transient analysis. Layered hexahedral meshed is specified for moving part and unstructured tetrahedral element was for stationary region

Table 1: Engine specifications

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Size / Feature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder type</td>
<td>Single cylinder, piston ported</td>
</tr>
<tr>
<td>Compression type</td>
<td>Crankcase compression</td>
</tr>
<tr>
<td>Displacement</td>
<td>30.5 cm³</td>
</tr>
<tr>
<td>Maximum output</td>
<td>0.81 kW / 6000 rpm</td>
</tr>
<tr>
<td>Scavenging concept</td>
<td>Multi port-Loop scavenged</td>
</tr>
<tr>
<td>Bore × Stroke</td>
<td>36 × 30 mm</td>
</tr>
<tr>
<td>Exhaust port opening / closing</td>
<td>101CA ATDC/ 259 CA ATDC</td>
</tr>
<tr>
<td>Scavenged port opening/ closing</td>
<td>140CA ATDC/ 220 CA ATDC</td>
</tr>
</tbody>
</table>

Fig. 1: The inner geometry of engine

Fig. 2: Volume grid

The lowest layer of vertices in the cylinder mesh was always maintained to be at the piston crown level. Additional layers of cells within the cylinder were activated until achieved at the BDC. A layer of vertices was also required to be moved in the scavenged port to ensure that the cell faces of the cylinder and scavenged matched at the port also exhaust itself.
In each case, a datum mesh was read in at the start of each time step and mesh motion program modified the datum mesh to that required at the end of time step. In this case, that setting of time step mesh motion start at TDC and until to BDC for one and half cycle. The starting moving at TDC as shown in Figure 4 below:

The dynamic mesh model in FLUENT can be used to model flows where the shape of the domain is changing with time due to motion on the domain boundaries. The update of the volume mesh is handled automatically at each time step based on the new positions of the boundaries. Using FLUENT, it provided a built-in function to calculate the piston location as a function of crank angle. Using this function, it was needed to specify the piston stroke and connecting rod length. The piston location is calculated using the following expression,

\[ p_s = L + \frac{A}{2} \left( 1 - \cos \left( \frac{\theta_c}{2} \right) \right) - \sqrt{L^2 - \frac{A^2}{4} \sin^2 \left( \frac{\theta_c}{2} \right)} \]  

where \( p_s \) is piston position (0 at TDC) and \( A \) at BDC, \( L \) is connection rod length, \( A \) is piston stroke, and \( \theta_c \) is current crank angle position.

The current crank angle is calculated as

\[ \theta_c = \theta_i + t \Omega_{\text{shaft}} \]  

where \( \theta_i \) the starting is crank angle and \( \Omega_{\text{shaft}} \) is the crankshaft speed. The three mesh update procedures available are dynamic (i) layering, (ii) local remeshing and (iii) spring smoothing. In this case, only dynamic layering method is applicable since the grid, which was built, is based on layered hexahedral element. Stationary zones were maintained intact for update.

The integral form of the conservation equations\(^{(18)}\) for a general scalar, \( \varphi \) on an arbitrary control volume, \( V \), whose boundary is moving can be written as

\[ \frac{d}{dt} \int_V \rho \varphi dV + \int_V \rho \left( \vec{u} - \vec{u}_g \right) \cdot \vec{n} dA = \int_V \nabla \varphi \cdot \vec{d} A + \int_S \varphi S dV \]  

where \( \rho \) is the fluid density, \( \vec{u} \) is the flow velocity vector, \( \vec{u}_g \) is the grid velocity of the moving mesh, \( \Gamma \) is the diffusion coefficient and \( S \) is the source term if \( \varphi \). Here is used to represent the boundary of the control volume is \( V \). The time derivation term in Equation (3) can be written using a first-order backward difference formula as:

\[ \frac{d}{dV} \int_V \rho \varphi dV = \frac{(\rho \varphi V)^{n+1} - (\rho \varphi)^n}{\Delta t} \]  

Fig. 3: Mesh motion for expansion\(^{(9)}\)

Fig. 4: Mesh motion preview start from TDC to BDC to verify the mesh motion step
where \( n \) and \( n+1 \) denote the respective quantity at the current and next time level. The \((n+1)\) th time level volume is calculate using:

\[
V^{n+1} = V^n + \frac{dV}{dt} \Delta t
\]  

(5)

\[
\frac{dV}{dt} = \sum_j \vec{u}_g \cdot \vec{A}_j = \sum_j \vec{u}_{g,j} \cdot \vec{A}_j
\]  

(6)

where \( n \) is the number of faces on the control volume and \( \vec{A}_j \) is the \( j \) face area vector. The dot product \( \vec{u}_{g,j} \cdot \vec{A}_j \) on each control volume face is calculated from

\[
\vec{u}_{g,j} \cdot \vec{A}_j = \frac{\delta V_j}{\Delta t}
\]  

(7)

\( \delta V_j \) is the volume swept out by the control volume face over the time step \( \Delta t \).

The computational domain is produced based on the actual engine geometry\(^{[10]}\). The finite volume enclosed the domain and starts from the scavenged/intake ports and ends at the exhaust outlet. The analysis is limited to in-cylinder process only.

Fluid passing through the engine was set to be air and was allowed to be compressible by the inclusion of the solution of enthalpy; the density was calculated through the use of the ideal gas law. The calculation utilized the standard version of the \( k - \varepsilon \) turbulence model. Here, the species transport model is activated with two phase conditions being defined as burned and unburned. The properties of two species could be different, it is simply because both are given the same properties as those of fresh air.

Boundaries through which flow can enter and leave the solution domain exit at the entry to the intake ports from the crankcase and exit of the exhaust port to atmosphere. The boundary condition at the intake ports were measured from experimental using complete test-rig show clearly explained from\(^{[12]}\). The boundary condition was selected from the experimental data at the scavenged / inlet port. From the pressure profile function of crank angle data applied with average value.

In this research, the simulation is run over 360 time step of 1° CA each and completed 1 cycle for two-stroke engine started at TDC and finally finished at BDC positions. Based segregated solver (with 1° Order implicit scheme) was employed to solve the governing equation of Navier Stokes. Pressure interpolation was done using Pressure Staggering Option (PRESTO) scheme. PRESTO is highly recommended for high swirl number, high speed rotating flows and flow in a strong curved domain. PISO Algorithm is chosen for pressure-velocity coupling method. PISO algorithm is specially developed for transient flow calculation. Correction factor is set to 1 and 0 factor for skewness correction. Second Order Upwind Scheme based\(^{[20]}\) is chosen for discretisation of the momentum equation. For other variables such as \( k, \varepsilon \), density, and species concentration, Second Order Upwind Scheme is selected as the discretization method. The Second Order Upwind is useful for reducing numerical diffusion since the meshes contained tetrahedral elements and the flow will be as a result not aligned with the grid. The \( k - \varepsilon \) model is activated to capture turbulence\(^{[21]}\). It is the simplest model for which only the initial and boundary conditions are needed for turbulence modeling. Transient analysis of in-cylinder processes is very complicated to study. Instead of moving boundary, compressibility effects, it involves chemical reaction (fired) also influenced by time consuming. In these cases, one complete cycle simulation took about 20 hours to calculate from TDC to BDC for 540° crank angles for one speed. Time step calculation was performed using a Pentium XION with 1 gigabyte memories. At the early stage of simulation condition, crank angle is iterated 50 times. After the iteration converge, the static pressure can be obtained by calculating the static pressure difference between pressure inlet and outlet of in-cylinder model.

RESULTS AND DISCUSSIONS

Cylinder Pressure Prediction: The experimental results shown that the higher rpm mode produced the highest cylinder pressure compare other lower rpm as shown in Fig. 5. The higher speed may reach nearly 9.04 bar and at the speed 1100 rpm and 1400 rpm showed the value of 8.43 bar and 8.73 bar, respectively. All data is taken at motoring condition which is for flow process without firing. The pressures started increase at the -10° BTDC for all speeds until achieve the maximum values when exhaust port started close (EPC). From this experiment also shows that when pressures reach at the 88° ATDC which is exhaust port started open (EPO), the pressures drop to negative values cause of residual form exhaust port enter to cylinder called backflow.

Fig. 6 shows that the intake pressures at the different of speed function of crank angle degree. The trend of profile show different compared with cylinder pressure. At the lower speed of engine can achieve the maximum pressure at the 1100 rpm and reduced for the 1400 rpm also 1700 rpm. The profile explained of process happen at the crankcase area.
Fig. 5: The effects of cylinder pressure versus crank angle degree.

Fig. 6: The effects of intake pressure versus crank angle degree.
The crankcase pressure is similar with the intake port value of pressure. From the overall figure, concluded that the minimum pressure occur when the fresh charge enter to crankcase through the transfer port and shown that at -45° BTDC the pressures was dropped. When the intake port started open (IPO) at 140° ATDC, the pressure of intake port was increased and can reach maximum values. At this time the fresh charge enter to cylinder at the high velocity and reduced when piston reaches at BDC. Form this pressure profile can be concluded that the higher speed can produce minimum pressure compared the lower speed of engine. These data will used as boundary condition for simulation approach.

Contours of Mass Faction of Burned Gases: A gas species concentration for a cross section in the plane of symmetry through exhaust port (on left), for a loop scavenged engine is shown in Fig. 7. The engine has two main inlet one exhaust ports. The shading level varies from red ‘old gas’ (the burnt gas in the cylinder at exhaust port opening) and others color ‘new gas’ (the fresh scavenging flow). Three locations selected at 150°, 160° and 180° ATDC to show the scavenging characteristic at port started open to bottom dead center (BDC) on engine cylinder. It is shows that at 1400 rpm the fresh air entering the cylinder and more can seen at BDC location. The increasing of engine speed, gave the slow of fresh air entering the cylinder for difference on engine speeds.

Conclusion: The experimental results are shown that the higher rpm mode produced the higher cylinder pressure than lower rpm. Based on simulation result, the cylinder pressure results is a function of crank angle degree for the different speed. The combustion chamber, transfer volume, intake and exhaust ports were derived optimally using these techniques leading to a short downtime period of engine prototyping and testing. The future work is need to improve the AMG solver setting to enable the simulation using accurate fuel characteristics be performed.

ACKNOWLEDGMENTS

We would like to acknowledge University Malaysia Pahang for providing the support this research project.

REFERENCES


