National Conference on software Engineering & Computer Systems 2007 (NaCSES'07), 20-21 August 2007, Universiti Malaysia Pahang.

Numerical Analysis of in-Cylinder Process for Small Two-Stroke Spark-Ignition (SI) Engine

R.A.Bakar[1], N.M.I.N.Ibrahim[2], M.F.A.Rahim[3] and I.Ali[4]
[1,2,3,4]Automotive focus Group,
Faculty of Mechanical Engineering,
Universiti Malaysia Pahang,
Log Bag 12, 25000 Kuantan, Pahang.

ABSTRACT

This paper presented the results carried out from numerical simulation using CFD commercial code software of three-dimensional analysis of two-stroke spark-ignition (SI) cross loop-scavenged port. Objective for this study is to investigate the in-cylinder and velocity characteristics while the scavenging process for motored condition. Based on the experimental data, the boundary condition was defined from the pressure to apply at the pressure inlet and pressure outlet defined as atmospheric pressure. Using the transient flow analysis, unsteady compressible flows and applied with the dynamic mesh method, the three-dimensional modeling solved numerically. The results show the prediction of distributions of in-cylinder pressure as a function of crank angle at different of speed. The relative errors show 1.86 %, 0.22 % and -1.01 % compared between experiments. As overall can be concluded that through the numerical solving, the prediction of in-cylinder pressure can be solved using the CFD applied by dynamic mesh method.

1. INTRODUCTION

Two-stroke technology has been 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 [1]. 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 [2] model the multidimensional modeling of two-stroke engine scavenging. Blair [3] and Heywood and Sher [4] have reported multidimensional numerical simulations of the loop-scavenging process in two-stroke engine. In their extensive study Blair [3] employed the multidimensional method to investigate the scavenging characteristics on scavenging process. Much experience has been gained from use of general-purpose CFD coded called PHOENICS and StarCD. These 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 *et al.* [5] 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 *et al.* [6] modified version of KIVA-II code to solve the scavenging flow simulation of a four-poppet-valved 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 *et al.* [7] 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 *et al.* [8] 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.

Elligott *et al.* [9] 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.

Tumble flow has been recognized as an important and positive enhancement of combustion for SI engines [10]. This tumble flow usually investigates through numerical simulation using CFD software. Its effect of turbulence and can be modeled using a Two-Equation Model ($k-\varepsilon$ model).

2. METHODOLOGY

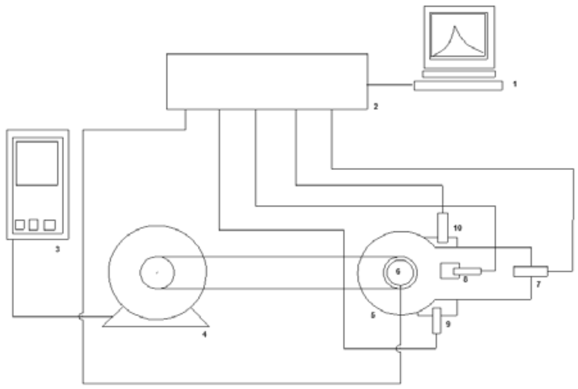
Using the TANAKA-328A engine, model was modeled on three-dimensional for the transient flow analysis. The specifications are shown in Table 1.

Table 1 Engine specifications

Parameter	Size / Feature
Cylinder type	Single cylinder, piston ported
Compression type	Crankcase compression
Displacement	30.5 cm^3
Maximum output	0.81 kW / 6000 rpm
Scavenging concept	Multi port-Loop scavenged
Bore × Stroke	36 × 30 mm
Exhaust port opening / closing	101 CA ATDC / 259 CA ATDC
Scavenged port opening/ closing	140 CA ATDC / 220 CA ATDC

2.1 Engine Test-Rig and data measurement

Figure 3.1 illustrated the detail schematic diagram for motored test powered by inverter. Two-stroke cross-loop scavenged port installed with data acquisition system (DAQ) to measure the in-cylinder pressure data through experimental. Pressure transducer sensor and crank angle encoder were installed and collect the data at the different of speed which are at 1100 rpm, 1400 rpm and 1700 rpm. The higher speed important to understand the in-cylinder characteristics for the turbulence flow condition. The new of test-rig was developed refer to combination from the Behrouz *et a.l* [11] and Yasou *et al.* [12]. Behrouz *et al.* [11] explained the in-cylinder pressure measurement for motored test and Yasou *et al.* [12] introduced the locations for pressure sensors installation.



1:Computer,2:DAQ,3:Inverter,4:Motor,
5:Engine,6:Crank Angle Encoder,
7:Pressure Transducer M8,
8/9/10:Pressure Transducer M5

Figure 3.1 The engine schematics for motored testing

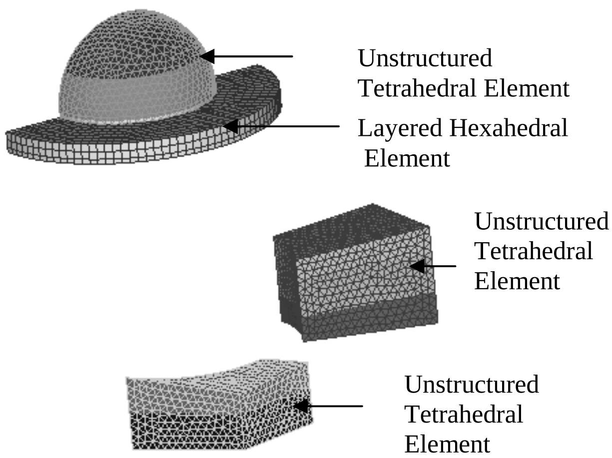


Figure 3.2 Volume grid

2.2 Numerical Solving

The performance tests for simulation were carried out with three operating condition as shown in Table 2 taken form experimental test.

Table 2 Boundary conditions

Inlet Pressure (Average)		
1100 rpm	1400 rpm	1700 rpm
1.168827	1.117891	1.029336

The model is meshed using unstructured element at the stationary region and structured element or hexagonal at the isotropy motion region shown on Figure 3.2. Both type of mesh are selected to fulfill the requirement of dynamic mesh. The segregated computational fluid dynamic (CFD) solver with implicit formulation is used to solve the unsteady, compressible, 3-dimensional Navier-Stokes equation and energy conservation equation. Commercial CFD software is employed for its easy approach and user-friendliness. Table 3 illustrated the attributes of numerical solving using CFD.

Table 3 Numerical Model Attributes

No.	Parameter	Attribute
1.	Material definition	Air, ideal gas density
2.	Operating condition	101325 bar, 300 K
3.	Boundary condition	Pressure inlet and pressure outlet
4.	Turbulence Model	Two equation $k-\varepsilon$ model
5.	Turbulent specification	Intensity and hydraulic diameter

2.3 Dynamic mesh set-up

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

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

Where p_s is piston position (0 at TDC) and A at BDC, L is connection rod length, A is piston stroke, and θ_c is current crank angle position. The current crank angle is calculated as

$$\theta_c = \theta_s + t\Omega_{shaft} \quad (2)$$

Where θ_s the starting is crank angle and Ω_{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.

3. RESULTS AND DISCUSSION

3.1 Prediction of in-Cylinder pressure

Cylinder pressure data was measured in experimental, and introduce in simulation stages to study the characteristics of in-cylinder behavior. The cylinder pressure data provided a more than satisfactory result thus increasing the confident level to explore more details on the next stage of this study which is scavenging analysis studies. Figures 3.3-3.5 shows the cylinder pressure results as a function of crank angle degree for the different speed. Figures 34.3-3.4 clearly shows that the simulation higher than the experimental and opposite for Figure 3.5. The experimental results show the values for 1100 rpm, 1400 rpm and 1700 rpm gave 8.43 bar, 8.73 bar and 9.04 bar, respectively. On other side, the simulation results gave 8.59 bar, 8.75 bar and 8.95 bar for the same speed as above. As theoretically explained that, the simulation results must always give higher than measured values because refer to ideal cycle and not consider factor such as heat losses (Yasou *et al.* [12]). Figure 4.3 give relative error is 1.86 % and followed by Figure 4.4 with 0.22 %. Comparatively, results for the Figure 4.5 shows that the experimental higher than simulation at 1 % of relative error compared based the experimental. The trends of Figures show that when the speed increased, the simulation results will be decrease at the small of discrepancy. When the exhaust port started open (EPO) at 88^0 and 101^0 give at negative value for measured and simulation, respectively. The values seen that small for simulation and show clearly at high value for experimental. It explained that influenced of this modeling process. The simulation stage is complexity of geometries which is modeled based on approximation values especially at the angle port geometries. Due to the same reason, the engine port timing was also change, since two-stroke engine timing is port controlled. The approximation of geometries is measured using the Coordinate Measurement Machine (CMM) measurement and manual measurement. But CMM measurement gives an accurate measurement on plane only. The difficulty of getting accurate geometry profile occurs since the actual geometry is three dimensional. Complex geometries at the intake, exhaust ports angles, dome and volume clearance high of the engine are approximated. Manual measurement of actual geometries slightly reduced the actual compression ratio (CR) which influences the increasing of pressure in cylinder.

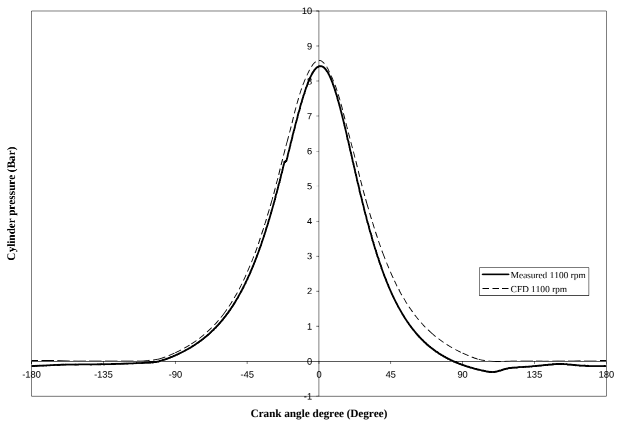


Figure 3.3 The effects of cylinder pressure versus Crank angle degree at 1100 rpm

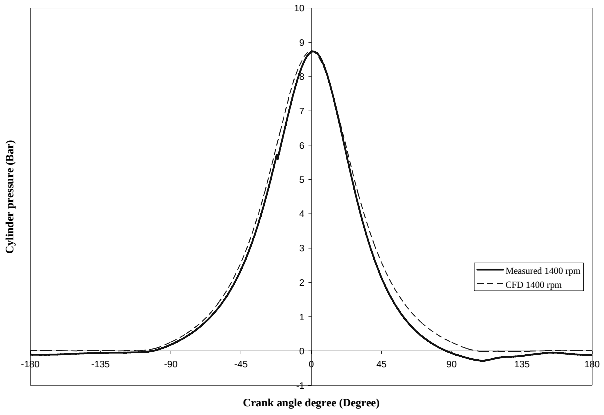


Figure 3.4 The effects of cylinder pressure versus Crank angle degree at 1400 rpm

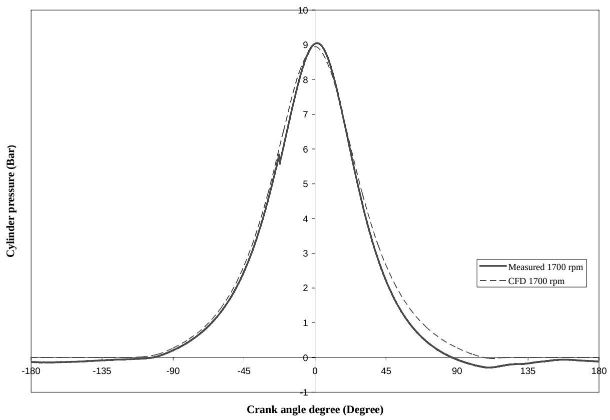


Figure 3.5 The effects of cylinder pressure versus Crank angle degree at 1700 rpm

4. CONCLUSION

Through this study, it were found that using the different variations of pressures, the simulation results show that gave significant values compared than the experimental and the relative error which are 1.86 %, 0.22 % and -1% at 1100 rpm, 1400 rpm and 1700 rpm. Applied with pressure average as boundary condition for every speed found that small discrepancy compared experimental results. It shown that through the average pressure, the prediction of in-cylinder process give confident. For next papers, the profile pressure will be build-in with CFD software.

ACKNOWLEDGEMENT

The authors would like to thank to Automotive Laboratory associates and final year undergraduate students Mr.Mohd Hafiz and Mr.Fauzani co-operated on engine test data.

NOMENCLATURE

CAE	Computer Aided Engineering
CFD	Computational Fluid Dynamics
PHEONICS	CFD code commercial software
StarCD	CFD code commercial software
KIVA	In-house CFD code
$k-\varepsilon$	Two-Equation Model for Turbulence Flow
DAQ	Data Acquisition System
IPO	Intake Port Open
IPC	Intake Port Close
EPC	Exhaust Port Close

REFERENCES

[1]. Jukka, T. Ari, S. Tore, G. and Martti, L.(2000). "Novel Two-Stroke Engine Concept, Feasibility Study", *SAE 2003-01-3211*

[2]. Ravi, M.R and Marathe, A.G.(1992)."Effect of Port Sizes and Timings in the Scavenging characteristics of a Uniflow Scavenged Engine", *SAE 920782*

[3] Blair, G. (1995). *The Design and Simulation of Two-Stroke Engine*. Society of Automotive Engineers Publishers, Ins.Warrendale, PA.

[4] Heywood, J.B.and Sher, E. (1999). *The Two-Stroke Cycle Engine: Its Development Operations, and Design*. New York, McGraw-Hill.

[5]. Ahmadi-Befrui, B. Branstatter, W. and Kratochohwill,H.(1989).Multidimensional Calculation of the Flow Processes in Loop-Scavenged Two-Stroke Engine. *SAE 890841*

[6]. Kang, Y.H. Ku, K.K. Chang, R.C. Soo, C.P. Sun, M. and Kwi, Y.L.(1993). Scavenging Flow Simulation of a Four-Poppet-Valved Two-Stroke Engine. *SAE 930500*

[7]. Xiaofeng, Y. Atsushi, O and Yoshihisa, T.(1999). Numerical Study of Scavenging Flow in Poppet-Valved Two-Stroke Engines. *SAE 1999-01-1250*

[8]. Raghunathan, R.D. and Kenny, R.G.(1997). CFD Simulation and Validation of the Flow within a Motored Two-Stroke Engine. *SAE 970359*

[9]. Elligott, S.M. Douglas, R. and Kenny, R.G. (1999). An Assessment of a Stratified Scavenging Process Applied to a Loop Scavenged Two-Stroke Engine. *SAE 1999-01-3272/JSAE 9938027*

[10]. Wen, D. Charles, E.N. and George, C.D.(1996). "Prediction of In-Cylinder tumble Flow and Combustion in SI Engines wit ha Quasi-Dimensional Model", *SAE 961962*

[11]. Behrouz, C., and David, S. (1995). Intake-Port Behavior in a Motored and Fired Two-Stroke Research Engine, *Experimental Thermal and Fluid Science 1995*, 10:86-100.

[12]. Yasou, M. Kazuhiro.M and Koji, M. (1999)."Numerical Analysis of Gas Exchange Process in a Small Two-Stroke Gasoline Engine, *SAE 1999-01-3330 / JSAE 9938085*