UNIVERSITI MALAYSIA PAHANG
BORANG PENGESAHAN TESIS

JUDUL: **CFD SIMULATION AND VALIDATION OF THE IN-CYLINDER WITHIN A MOTORED TWO STROKE SI ENGINE**

SESI PENGAJIAN: 2007/2008

Saya **MOHD FAUZANI BIN MOHD NAWI**
(HURUF BESAR)

mengaku membenarkan tesis (PSM/Sarjana/Doktor Falsafah)* ini disimpan di Perpustakaan Universiti Malaysia Pahang dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Pahang.
2. Perpustakaan Universiti Malaysia Pahang dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (4)

[ ] SULIT (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)

[ ] TERHAD (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/bahan di mana penyelidikan dijalankan)

[√] TIDAK TERHAD

Disahkan oleh

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap:
76 KG GONG PAK CHING
22000 JERTEH BESUT
TERENGGANU DARUL IMAN

MR KUMARAN A/L KADIRGAMA
Nama Penyelia

Tarikh: ____________ Tarikh: 5/5/08

CATATAN:
* Potong yang tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.
υ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM)

CFD SIMULATION AND VALIDATION OF THE IN-CYLINDER WITHIN A MOTORED TWO-STROKE SI ENGINE

MOHD FAUZANI BIN MOHD NAWI

A report is submitted in partial fulfillment of the requirements for the award of the degree of Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

MAY 2008

SUPERVISOR DECLERATION

I hereby declare that I have read this thesis and in my opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature : ....

Supervisor : Mr. Kumaran a/l Kadirgama

Date : 6/8/08..

STUDENT DECLERATION

I declare that this thesis entitled "*CFD Simulation And Validation of The In-Cylinder Within A Motored Two Stroke SI Engine*" is the result of my own research except as cited in the reference. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any degree.

Signature : ....

Name : Mohd Fauzani Bin Mohd Nawi

Date : ....06/05/08..................

Dedicated to my beloved
Mother,
Brothers and Sister

ACKNOWLEDGEMENTS

In the name of ALLAH, the Most Beneficent, the Most Merciful. First of all I am very grateful to the One and Mighty Creator, for giving me strength to complete this project successfully. It is only through His mercy that this project could be complete. Perhaps this project Praise be to God.

In completion this project, I was in contact with many people that contributed toward my understanding and problem solving. In particular, would like to express my sincere gratitude to my supervisor Mr. Kumaran a/l Kadirgama for his support, valuable ideas, exact guidance and full of observation in making this research possible. I am also deeply thankful to everyone who assisted me on this project especially to all lecturer and FKM laboratory staffs for their support and guidance.

Besides that, I also like to dedicate my appreciation to all my fellow friends especially to the mechanical engineering student batch 04/05 especially to my best friends for their support, suggestions and co-operation throughout my study.

Not inadvertently omitted to my family especially to my mother, brothers and sister for their faith in me to be a good son and to attain my goal. I can not describe my gratitude to them for giving me lots of supports in the aspects of moral, social and financial during my study.

ABSTRACT

The two stroke engine is the current engine of choice in the motocross bike because it of simplicity of design, size to power ratio, lightweight, low number of moving parts, ease of maintenance and excellent power and torque characteristics. However, the major disadvantage to a two stroke engine that negates all the previously mentioned positive qualities is the emission of substantial hydrocarbons, carbon monoxide and fine particulate matter. These pollutants have significant adverse health effects on environmental quality

The objective of the research is to study about the in-cylinder flow of the two stroke spark ignition (SI) engine. This research is carried out from the experimental analysis data that were set as the boundary condition in the numerical analysis. The experiments were based on the motored condition where there is no combustion inside the cylinder. The numerical analysis is carried out using Computational Fluids Dynamic (CFD) Fluent code.

From the research, the result from the both experimental analysis and numerical analysis were validated so that the major characteristic of the in cylinder flow can be understand. Beside the validation, other relating characteristic such as cylinder pressure and burned mass fraction can be determined.

ABSTRAK

Enjin dua lejang adalah pilihan masa sekarang untuk tunggangan motorcross kerana mempunyai model yang mudah, saiz perbandingan kuasa, ringan, bahagian bergerak yang sedikit, penjagaan yang mudah dan keunggulan kuasa dan tenaga putaran. Bagaimanapun, keburukan utama kepada enjin dua lejang yang meniadakan kesemua kualiti bagus yang disebutkan sebelum ini adalah pengeluaran hidrokarbon yang banyak, karbon monoksida dan benda-benda yang halus. Bahan tercemar ini mempunyai bahan yang memudaratkan kualiti persekitaran.

Tujuan kajian ini dihasilkan ialah untuk mengkaji tentang aliran di dalam enjin dua lejang. Kajian ini dilaksanakan berdasarkan dua cara iaitu melalui ujikaji dan juga simulasi komputer. Hasil kajian ujikaji akan dijadikan sebagai "Boundary Condition" semasa simulasi komputer. Ujikaji ini dijalankan dengan mengunakan motor untuk menggerakkan enjin iaitu tiada pembakaran berlaku di dalam enjin. Simulasi komputer pula dilakukan menggunakan perisian "Computational Fluids Dynamic" Fluent.

Berdasarkan hasil kajian, keputusan ujikaji dan juga simulasi komputer akan dibandingkan supaya karektor aliran bendalir di dalam enjin dua lejang dapat difahami. Selain itu, parameter-parameter lain seperti tekanan di dalam enjin dan pecahan pembakaran jisim dapat ditentukan.

TABLE OF CONTENTS

CHAPTER	TITLE	PAGE
	DECLARATION	iii
	DEDICATION	v
	ACKNOWLEDGEMENTS	vi
	ABSTRACT	vii
	ABSTRAK	viii
	TABLE OF CONTENTS	ix
	LIST OF TABLES	xi
	LIST OF FIGURES	xii
	LIST OF SYMBOLS	xiv
	LIST OF ABBREVIATIONS	xv
1	**INTRODUCTION**	
1.1	Introduction	1
1.2	Problem statement	2
1.3	Objective	3
1.4	Project scopes	3
2	**LITERATURE REVIEW**	

	2.1 Introduction	4
	2.2 Fundamental Method of Operation of a Simple Two-Stroke Engine	4
	2.3 Method of Scavenging the Cylinder	7
	2.3.1 Loop Scavenging	8
	2.3.2 Cross Scavenging	9
	2.3.3 Uniflow Scavenging	10
	2.4 Comparison between two-stroke and four-stroke	11
	2.5 Computational Fluid Dynamic (CFD)	11
	2.6 Application of CFD by other researchers in combustion engine	12
3	**METHODOLOGY**	
	3.1 Introduction	14
	3.2 Experimental	14
	3.2.1 Geometry	16
	3.2.2 Cylinder Design	17
	3.2.3 Test-rig	18
	3.2.4 Pulley	19
	3.2.5 Experiment	21
	3.3 The Numerical Analysis	23
	3.3.1 The Engine Dimensioning	23
	3.3.2 Grid Generation (GAMBIT)	24
	3.3.3 Dynamic Mesh Setup	25
	3.3.4 Fluids Properties	26
	3.3.5 Boundary Conditions	27
4	**RESULTS AND DISCUSSION**	29
	4.1 Introduction	29

	4.2 Experimental Analysis Result	29
	4.3 The Numerical Analysis Result	31
	4.4 Validation and Discussion	33
5	**CONCLUSION**	35
	5.1 Conclusion	35
	5.2 Recommendation	36
REFERNCES		37

LIST OF TABLES

TABLE NO.	TITLE	PAGES
3.1	The Engine Specification	16
3.2	Engine Operation Condition	22
3.3	The In-Cylinder Parameters	26
3.4	The Average Pressure at Scavenged Port	27

LIST OF FIGURES

FIGURE NO.	TITLE	PAGES
2.1	Intake	5
2.2	Compression	6
2.3	Ignition	6
2.4	Exhaust	7
2.5	loop scavenged	8
2.6	cross scavenged	9
2.7	Uniflow Scavenged	10
3.1	Flowchart Methodology	15
3.2	Cylinder	17
3.3	Cylinder Design	18
3.4	Test-rig	19
3.5	Motor pulley	20
3.6	Engine pulley	20
3.7	Schematic diagram	21
3.8	Experimental Testing	22
3.9	The In-Cylinder Volume Generated by COSMOS FloWork	24
3.10	The overall mesh structure at TDC	25
3.11	The experiment data as the pressure inlet in simulation for 1060 rpm	28

4.1	The experiment data as boundary condition in numerical analysis for 1061 rpm speed	30
4.2	The in-cylinder pressure from the experimental analysis for 1061 rpm speed	30
4.3	Burned mass fraction for a= 0 CA, b= 36 CA	31
4.4	Burn Mass Fraction for a= 72 CA, b=108 CA, c= 144 CA, d= 180 CA, e= 216 CA, f= 252 CA	32
4.5	Burn Mass Fraction for a= 288 CA, b= 324 CA, c= 360 CA	33
4.6	The validation of the in cylinder pressure for the 1061 rpm speed	33

LIST OF SYMBOLS

%	-	Percentages
°	-	Degree
cm^3	-	Centimeters Cube
mm	-	Millimeter
x	-	Times
Hz	-	Frequency
K	-	Kelvin
Bar	-	Pressure
+	-	Positive
-	-	Negative
ε	-	Epsilon

CHAPTER 1

INTRODUCTION

1.1 Introduction

The two-stroke engine is used for the lightweight power units which can be employed in various attitudes as handled power tools such as the chainsaws, brush cutter, concrete saws and etc. These tools are manufactured with a view to lightness and high specific power performance.

The first two-stroke engine was a gas engine invented and built by Etienne Lenoir in 1860 and followed the two-stroke diesel engine by Sir Dugald Clark in 1878 [1]. Both design of the early two-stroke engine used a similar cylinder head as the fourstroke diesel engine and a supercharger. The gasoline two-stroke engine, and the cylinder ports on which it depends were invented by Joseph Day in 1889 [1]. These cylinder ports were subsequently incorporated into diesel two-stroke engines, replacing either just the inlet valve or both inlet and exhaust valve. A great deal of development of the two-stroke engine was done between the two world wars, particularly by Germany companies such as DKW and Fitchel & Sachs [1].

Internal combustion engine performance and exhaust emissions are governed by unsteady fluid dynamic process. The objective of these processes is fundamental to improving engine characteristics. Computational Fluid Dynamic (CFD) is able to produce detailed information on the flow within an engine.

1.2 Problem Statement

The applications of small loop-scavenged, single cylinder, two- stroke spark-ignition engines are widespread due to the specific advantages of this engine concept. Today's engines have to face stringent pollutant and noise emission regulations and fuel consumption reductions. Advanced technical solutions will have to be applied even in the most compact and light weight high output power units.

The two stroke engine is the current engine of choice in the motocross bike because it of simplicity of design, size to power ratio, lightweight, low number of moving parts, ease of maintenance and excellent power and torque characteristics. However, the major disadvantage to a two stroke engine that negates all the previously mentioned positive qualities is the emission of substantial hydrocarbons, carbon monoxide and fine particulate matter. These pollutants have significant adverse health effects on environmental quality.

According to the EPAs own data, a two stroke engine dumps 25 to 30 % of its fuel directly into the air and onto the ground. This external dumping process is due to the fact that a large fraction of the air or fuel mixture passes through the engine and into the exhaust system without being combusted. This is due to what is known as an inefficient scavenging process of the two stroke engine.

1.3 Objective

The objective of the project is to evaluate and analyze a Computational Fluid Dynamic (CFD) simulation of the flow a motored loop scavenged two-stroke engine using commercial software FLUENT and compare with the experimental data.

1.4 Project Scopes

The scopes of the research are:

1. Set up the motored experimental purpose
2. Build model for simulation purpose
3. Predict flow within the cylinder and the transfer port

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter will discuss about the two-stroke engine operation and information about the scavenge process in the two-stroke engine. It also provides comparison between two stroke engine and four stroke engines. It also will provide information about the computational fluids dynamic (CFD).

2.2 Fundamental Method of Operation of a Simple Two-Stroke Engine

The intake stroke begins with the piston near the top of its travel. As the piston begin its descent, the exhaust valve close fully, the intake valve opens and the volume of combustion chamber begins to increase, creating a vacuum. As the piston descents, an air/fuel mixture drowns from the carburetor into the cylinder through the intake manifold. The intake stroke ends with the intake valve closed just after the piston has begun it upstroke. The process is shown in Figure 2.1.

As the piston ascends, the fuel/air mixture is forced into the small chamber machined into the cylinder head. This compress the mixture until it occupies ½th to 1/11th of the volume that it did at the time the piston began its ascent. This compression raises the temperature of the mixture and increases its pressure. Increasing the force generated by the expansion of gases during the power stroke. The process is shown in Figure 2.2.

The fuel/air is ignited by the spark plug just before the piston reaches the top if its stroke so that a very large portion of the fuel will have burned by the time the piston begins descending again. The heat produced bye combustion increases the pressure in the cylinder, forcing the piston down with great force. The process shown in Figure 2.3

As the piston approaches the bottom of its stroke, the exhaust valve begins opening and the pressure in the cylinder begins to force the gases out around the valve. The ascent of the piston then forces nearly all the rest of the unburned gases from the cylinder. The cycle begins again as the exhaust valve closes, the intake valve opens and the piston begins descending and bringing a fresh charge of fuel and air into the combustion chamber. The cycle shows in Figure 2.4.

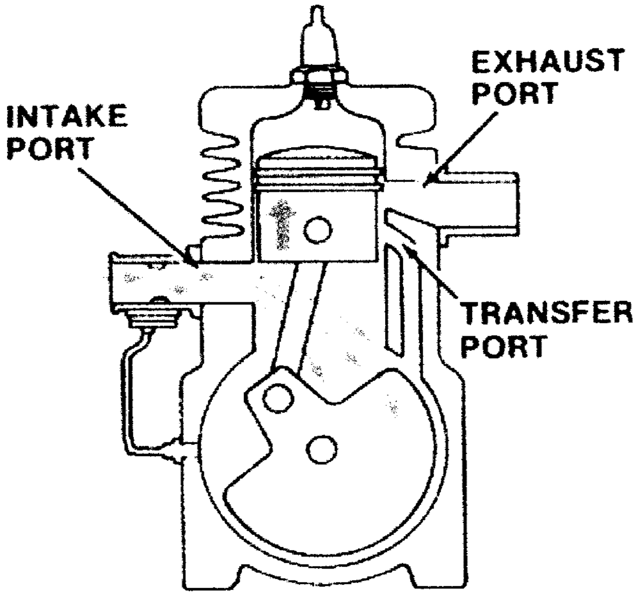


Figure 2.1: Intake

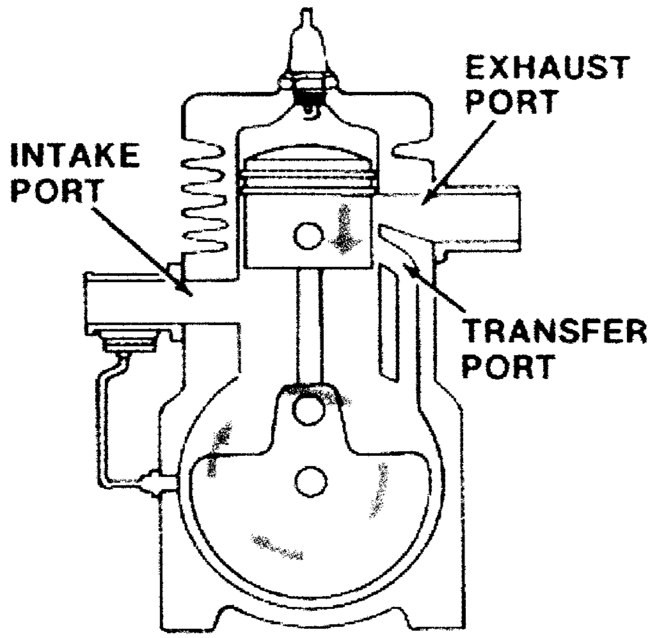


Figure 2.2: Compression

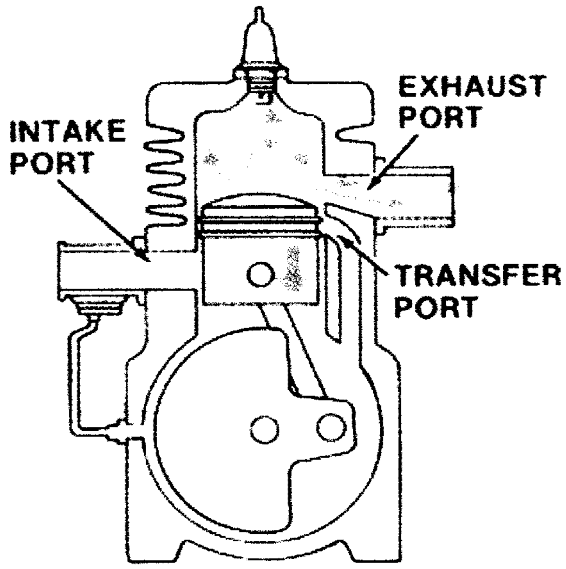


Figure 2.3: Ignition

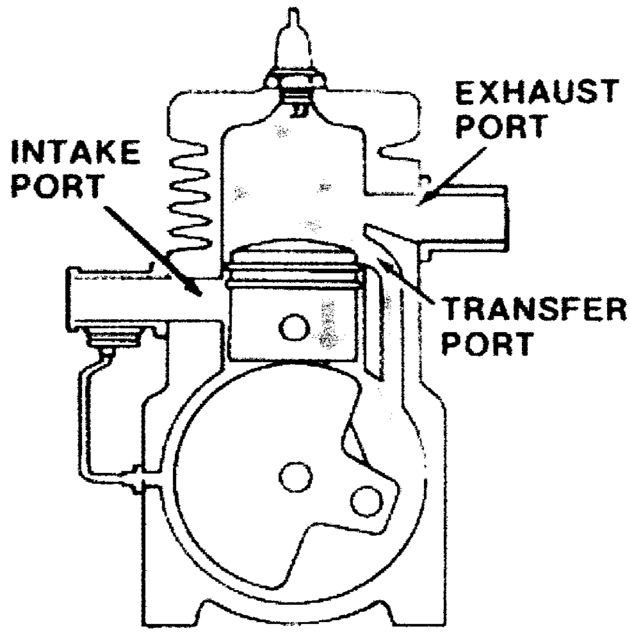


Figure 2.4: Exhaust

2.3 Method of Scavenging the Cylinder

There are three types of scavenging in the two-stroke engine. The scavenging processes are categorized as:

1. Loop scavenging
2. Cross scavenging
3. Uniflow scavenging

2.3.1 Loop Scavenging

In loop scavenging, the fresh air first sweeps across the piston top, moves up and then down and finally out through the exhaust. The system avoids the short-circuiting of the cross scavenged engine and thus improves upon its scavenging efficiency. The Schnuerle type of loop scavenged shown in Figure 2.5.

Loop scavenged engines have a high resistance to the thermal stresses and are thus much suited to higher supercharge. In a loop scavenged two stroke engine, the major mechanical problem is that of obtaining an adequate oil supply to the cylinder wall consistent with reasonable lubricating oil and cylinder wear.

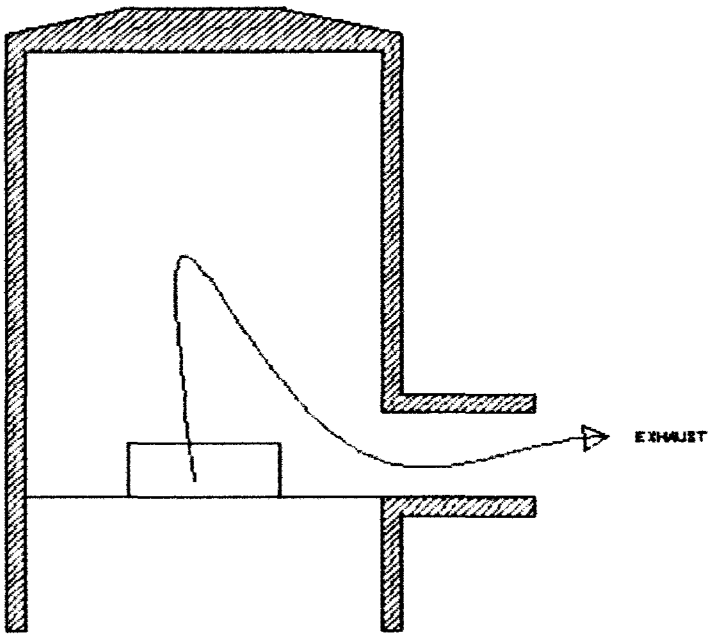


Figure 2.5: Loop Scavenged

2.3.2 Cross Scavenging

In this system the inlet and exhaust ports are located on opposite sides of the cylinder as shown in Figure 2.6. The incoming flow is directed upwards by the deflector on the piston, and the cylinder head reverses the direction of flow, so that exhaust gases are forced through the exhaust port.

In this type of arrangement the engine is structurally simpler than that with the uniflow scavenging. The main demerit of this system is that scavenging air is not able to get rid of the layer of the exhaust gas near the wall resulting in poor scavenging. A small portion of fresh charge goes directly into the exhaust port.

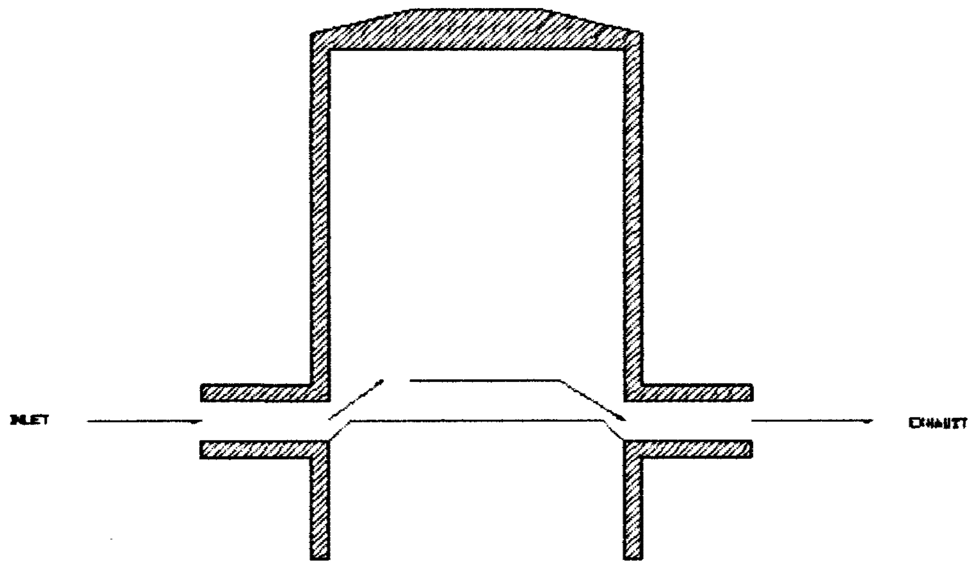


Figure 2.6: Cross Scavenged

2.3.3 Uniflow Scavenging

The fresh charge is admitted at one end of the cylinder and the exhaust escapes at the other end. The air flow is from end to end, and little short-circuiting between the intake and exhaust openings is possible. The uniflow scavenging is shown in Figure 2.7. Uniflow system permits unsymmetrical port timing and supercharging.

Due to absence of any eddies or turbulence it is easier in a uniflow scavenging system to push the combustion product out of the cylinder without mixing with it and short circuiting. Thus the system has the highest scavenging efficiency [2].

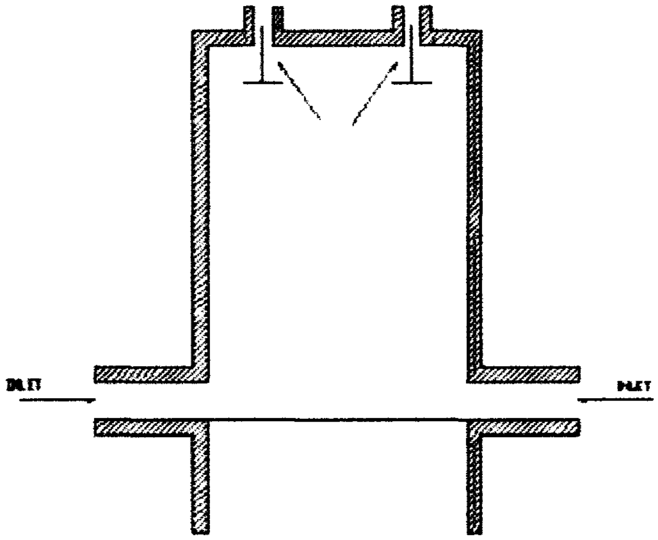


Figure 2.7: Uniflow Scavenged

2.4 Comparison between two-stroke and four-stroke

Two-stroke engines have several marked disadvantages that have largely precluded their use in automobiles and are reducing their prevalence in the applications. Firstly, they require much more fuel than a comparably powerful four-stroke engine due to less efficient combustion. The burning oil, and the less efficient combustion, makes their exhaust far smellier and more damaging than a four-stroke engine, thus struggling to meet current emission control laws.

They are noisier, partly due to the more penetrating high-frequency buzzing and partly due to the fact that muffling them reduces engine power far more than on a four-stroke engine (high-performance two-stroke engine exhausts are tuned by determining the resonant frequency of the exhaust systems and exploiting it to top-up the fuel air charge just before the cylinder port closes) Finally, they are considered less reliable and durable than four stroke engines.

There are more elaborate possible two-stroke engine configurations, but these often have enough complications that they do not out perform comparable four-stroke engines. New two-stroke designs rely on electronically-controlled fuel injection, oil injection and other design tweaks to reduce pollution and increase fuel efficiency.

2.5 Computational Fluid Dynamic (CFD)

Computational Fluid Dynamic is the analysis of systems involving fluids flow, heat transfer and associated phenomenon such as chemical reactions by means computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

CFD analysis is a solution of fluid motion equation usually partial differential equations (PDE) type using numerical approximation. This process which is called discretization turns the PDEs into simultaneous algebraic equation which in turn are solved using algebraic solution techniques, usually iterative method.

The approximations are applied to a numbers of small domains in space and times called grid. So, the final results are spatially and temporally resolved. Using CFD, you can build a computational model that represents a system or device that you want to study. Then you apply the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena. Therefore, CFD is a sophisticated computationally-based design and analysis technique.

2.6 Application of CFD by other researchers in combustion engine

According to B.D.Raghunathan and R.G.Kenny [3], A CFD simulation of the flow within a motored two-port loop-scavenged two-stroke engine is described. The simulation is carried out using the STAR-CD CFD. A moving mesh with cell layer activation deactivation is used to represent the reciprocating piston motion. Prediction of the flow within the cylinder and at the transfer port are presented over the open cycle and are compared to an existing measured velocity field for an engine speed of 600rpm and a delivery ratio of unity.

The results shows the in-cylinder flow have a highly complex structure dominated by recirculating flow feature. The predictions were compared to an existing set of measured data. From comparisons of the measurements and predictions they make the conclusion which are the model was able to predict global quantities such as the flow velocity at the transfer port and the cylinder pressure to a high level of accuracy. Also the correlation between the measured data predicted flow within the cylinder was poor over the port opening phase of the cycle. The flow conditions and profile at the transfer port were well predicted provided the flow within the duct remained attached to the inner surface of the duct at the 90° bend. Flow separation within the duct was not well predicted.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In this topic is about how the research is carried out. The research is to study about the scavenging characteristic in two-stroke spark ignition (SI) engine by the experimental and simulation method. All the data from the experiment will be compared with the data collected from the simulation. The flow chart of the methodology is as shown in Figure 3.1.

3.2 Experimental

Before we run the experiment, there are few things that must be considered such as the engine specification, the test rig, the crank angle encoder and the pulley to mount the crank angle encoder and the crankshaft. These considerations are important to make sure the experiment run smoothly.

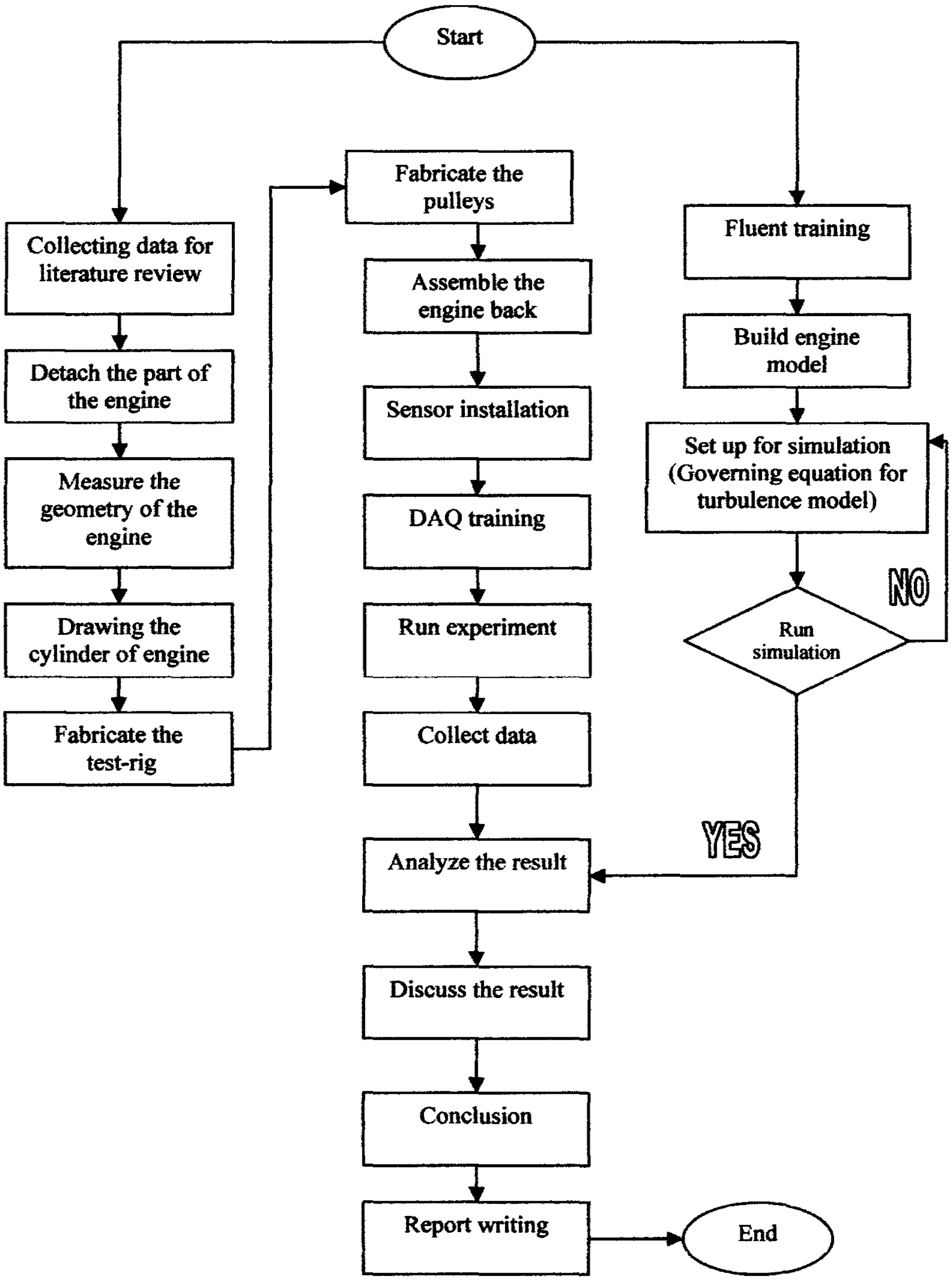


Figure 3.1: Flowchart Methodology

3.2.1 Geometry

The Figure 3.2 below show the cylinder after separated from the body. After the engine separated, the dimension of bore, stroke, transfer port, exhaust port was measured. The measurement done using the automatic vernier caliper as it is can increase the accuracy of the dimension. From the measurement, the data collected as shown in Table 3.1 below:

Table 3.1: The engine specification

Parameter	Size / Feature
Cylinder Type	Single Cylinder, piston ported
Compression Type	Crankcase compression
Displacement	30.5 cm^3
Scavenged Concept	Multi-port Loop scavenged
Bore x Stroke	35 x 30 mm
Exhaust port opening / closing	101.20 CA/ 259.2 CA ATDC
Scavenged port opening/ closin	241.40 CA/ 155.20 CA ATDC

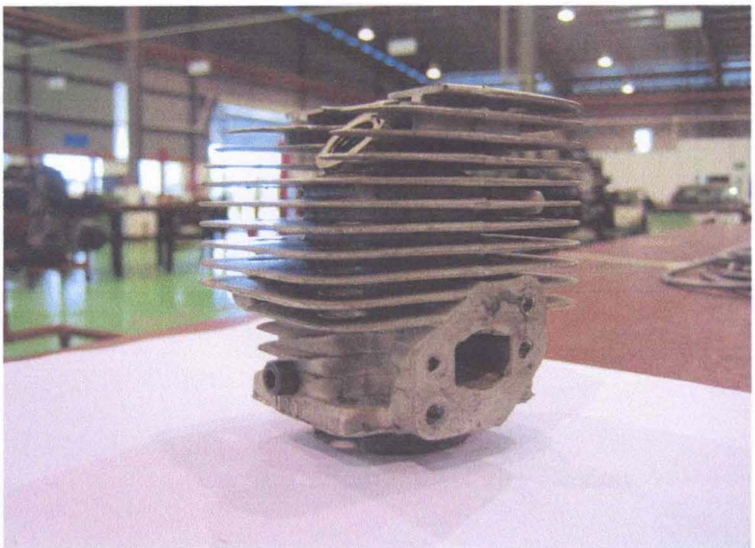


Figure 3.2: Cylinder

3.2.2 Cylinder Design

From the data measured, the inner cylinder draws using SOLIDWORK. The design of the inner cylinder shown in Figure 3.3 below:

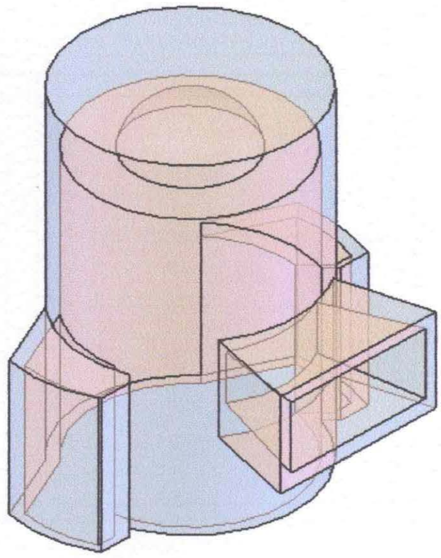


Figure 3.3: Cylinder Design

3.2.3 Test-rig

To run the experiment, it needs the test-rig to place the engine, motor and etc. The test-rig produces using the hollow metals as it can pierce to screw the motor and the engine on it. The Figure 3.4 below shows the test-rig to run experiment.

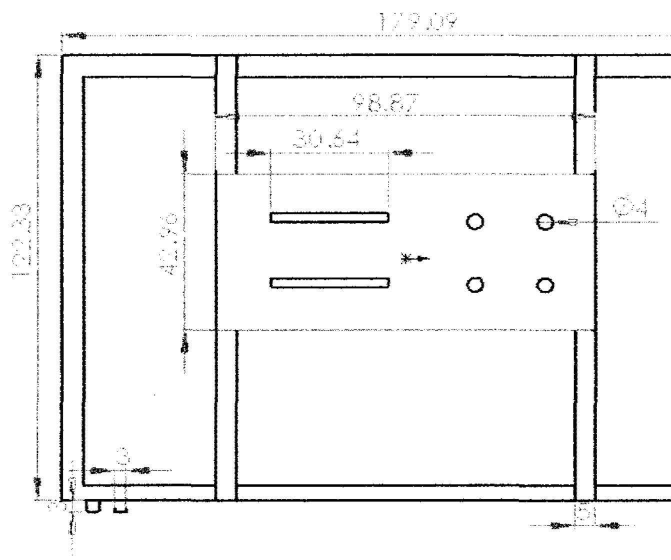


Figure 3.4: Test-rig

3.2.4 Pulley

The other necessary to run the experiment is the pulleys. The work piece used is the aluminum alloy as it is not heavy, suitable with the engine used. The first pulley placed at the motor meanwhile the second pulley placed at the engine. The pulleys connected using the belt as it can transfer the energy from the motor to the engine. The motor pulley machined using CNC LATHE machine and EDM WIRECUT machine. Meanwhile the engine pulley machined using CNC LATHE machine and CNC MILLING machine. The motor and engine pulley are shown in Figure 3.5 and Figure 3.6 respectively.

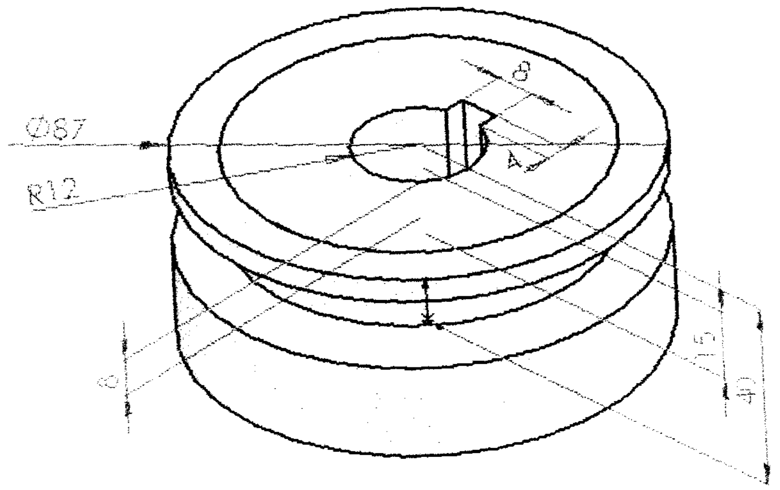


Figure 3.5: Motor pulley

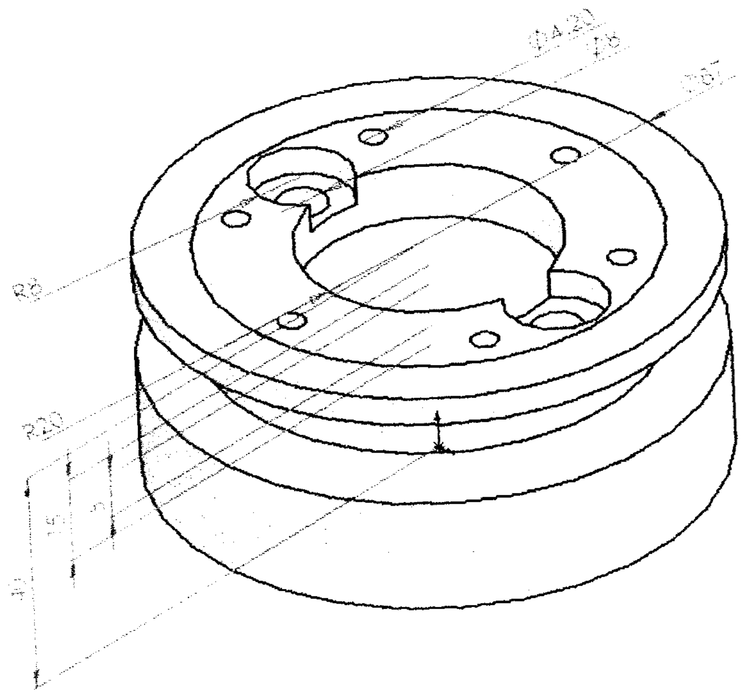


Figure 3.6: Engine pulley

3.2.5 Experiment

Figure 3.7 show the schematic diagram layout of the experimental testing. Measurement for pressure data and crankangle data are using Kistler Pressure Transducer and Crank Angle Encoder respectively.

On stage numerical analysis, experiment data totally utilize to study the behavior of in-cylinder analysis. The measurement location has chosen to determine the main features of the pressure data at the intake port and exhaust port. Pressure transducer (Kistler M8x0.75) was used to measure in-cylinder pressure mounted in the injector hole and (Kistler M5x0.8) mounted in the intake port and exhaust port. The cylinder head was drilled using CNC Milling in order to make a hole and thread to place the pressure transducer.

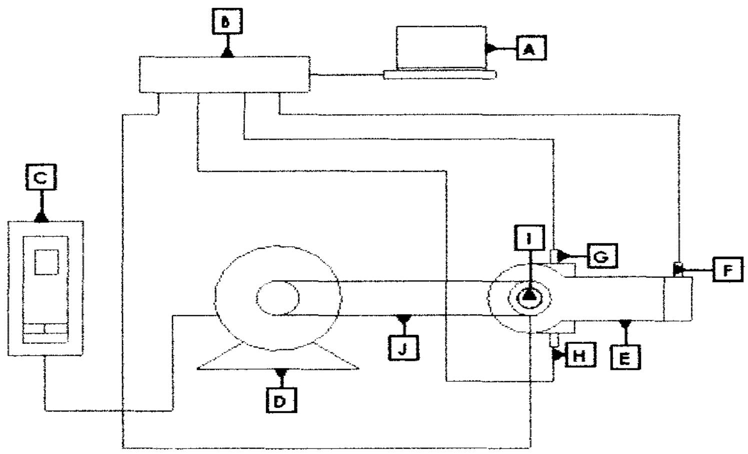


Figure 3.7: Schematic diagram

A: Computer, B: DAQ, C: Inverter, D: Motor, E: Engine Tested, F: Transducer M8, G/H: Transducer M5, I: Crank Angle Encoder, J: Belt

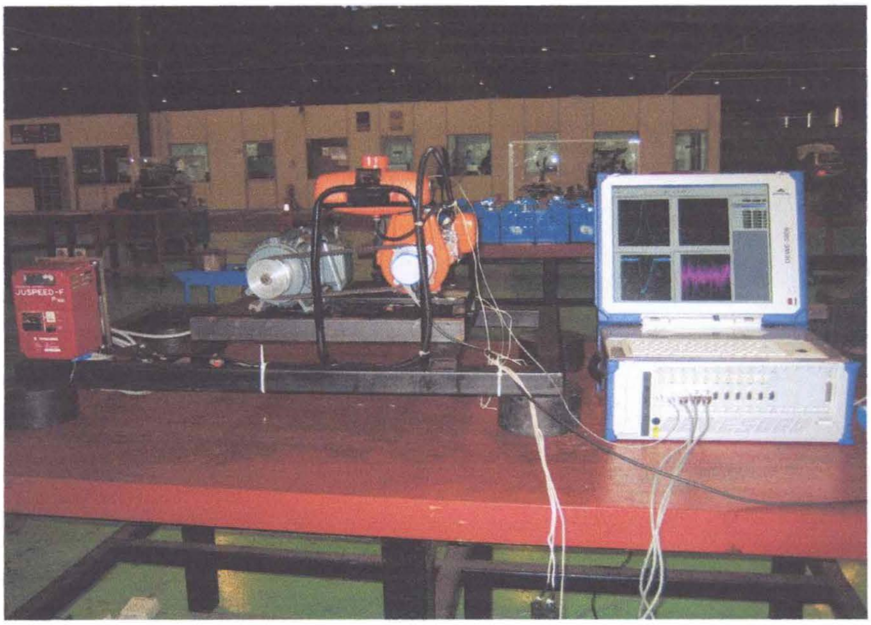


Figure 3.8: Experimental Testing

The experiment is carried out using the motored method which the engine combustion is generated using motor and without firing. The data is collected at different engine speed from the engine idle speed to maximum engine speed as shown in Table 3.2.

Table 3.2: Engine Operation Condition

Frequency (Hz)	**Engine Speed (rpm)**
20	682
25	754
30	788
35	939
40	1061

45	1239
50	1384
55	1535
60	1655

3.3 The Numerical Analysis

The numerical analysis is carried out using the Computational Fluid Dynamics (CFD), Fluent. Fluent is capable of generating meshes automatically. That why the Fluent software used. At the same time giving its user control over density, skewness and other parameter of mesh quality. In addition to these characteristic it has well endorsed models for combustion, swirl, flow and other fluids properties. It also allows complex geometries to be generated easily and support mesh types for 2D and 3D simulation.

3.3.1 The Engine Dimensioning

First of all, it is important to build the Computer Aided Design (CAD) model based on the Tanika BG-328A engine geometries. The geometries are carried out using Coordinate Measuring Machine (CMM) and vernier caliper to make sure the geometry of the model is accurate to the actual engine geometries. This is very important to get the accurate result of the numerical analysis.

The In-Cylinder volume as shown in Figure 3.9 below is generated by COSMOS FloWork and divided into half of the geometry. The In-Cylinder volume is divided to three volume which are Scavenged port volume, Exhaust port volume and Cylinder Clearance volume and saved as .ACIS file. The volume meshed using gambit 2.16 Pre-processor and saved in TGrid to transfer into FLUENT software.

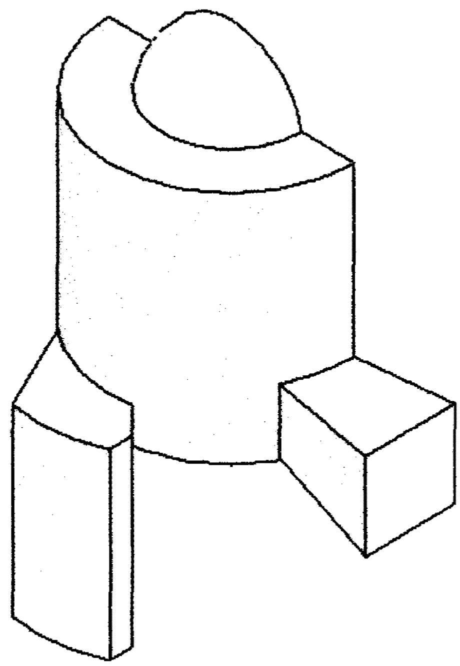


Figure 3.9: The In-Cylinder Volume Generated by COSMOS FloWork

3.3.2 Grid Generation (GAMBIT)

The mesh was constructed using three separated blocks that represent the scavenged port, exhaust port and cylinder. 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 3.10. The volume is divide and the meshed to fulfill the requirement of the dynamic mesh method. Layered hexahedral meshed is specified for moving part and unstructured tetrahedral element is for stationary region using Gambit 2.16 Pre-processor and TGrid.

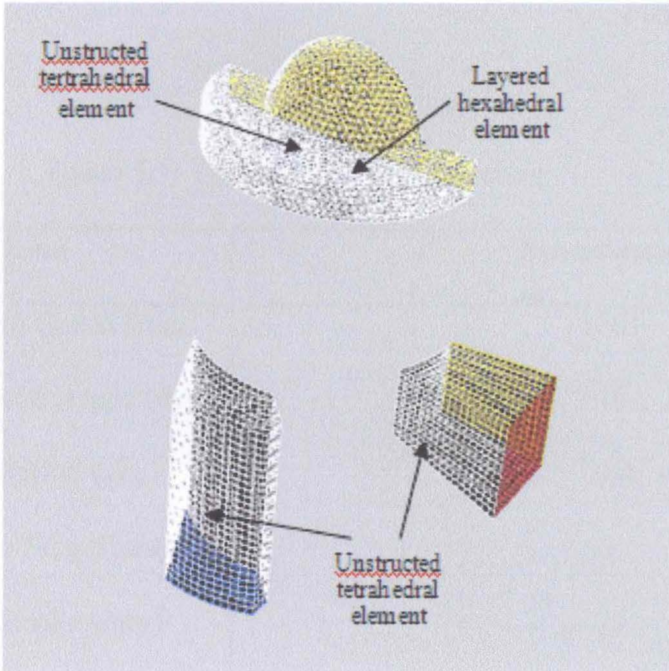


Figure 3.10: The overall mesh structure at TDC

3.3.3 Dynamic Mesh Setup

The dynamic mesh model in FLUENT can be used to model flow where shape of the domain is changing with time due to motion on the domain boundaries. In this case, dynamic mesh is set to calculate the piston location that is proportional to the crank angle. The update of the volume mesh is handled automatically by FLUENT at each time step based on the new positions of the boundaries.

There are three dynamics mesh methods applicable in FLUENT that were Smoothing, Layering and Remeshing. Under in-cylinder, the parameters were set following the data as shown in Table 3.3.

Table 3.3: The in-cylinder parameters

Item	Specification
Crank Shaft Speed (rpm)	1061
Starting Crank Angle (deg)	0
Crank Period (deg)	360
Crank Angle Step Size (deg)	1
Piston Stroke (mm)	30
Connecting Rod Length (mm)	57.10
Piston Stroke Cutoff (mm)	10.000
Minimum Valve Lift (mm)	0.100

3.3.4 Fluids Properties

The 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 two phase conditions being defined as burned and unburned. The properties of two species could be different, it is simplicity both given the same properties as those of fresh air.

3.3.5 Boundary Conditions

Boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are important in the FLUENT simulation and to be specified appropriately during simulations.

The boundary condition (BC) at the intake ports were measured from experimental method. The average scavenged pressure for speeds that is 40 Hz (1061 rpm) is used as the boundary condition in the simulation. The average pressure for the scavenged port is shown in Table 3.4.

The flow at the exit of the exhaust port to atmosphere was set to constant static pressure equal to atmospheric pressure [3]. The cylinder was initialized with an inlet pressure used data as shown in Figure 3.11. The wall temperature was fixed at 300 K. The turbulence intensity and scale at both scavenged port entrance and the exhaust port exit was set at 10 % of mean velocity and 2 mm, respectively.

Table 3.4: The average pressure at scavenged port

Frequency/ Speed	Average Scavenged Pressure
40 Hz / 1061 rpm	2.182 Bar

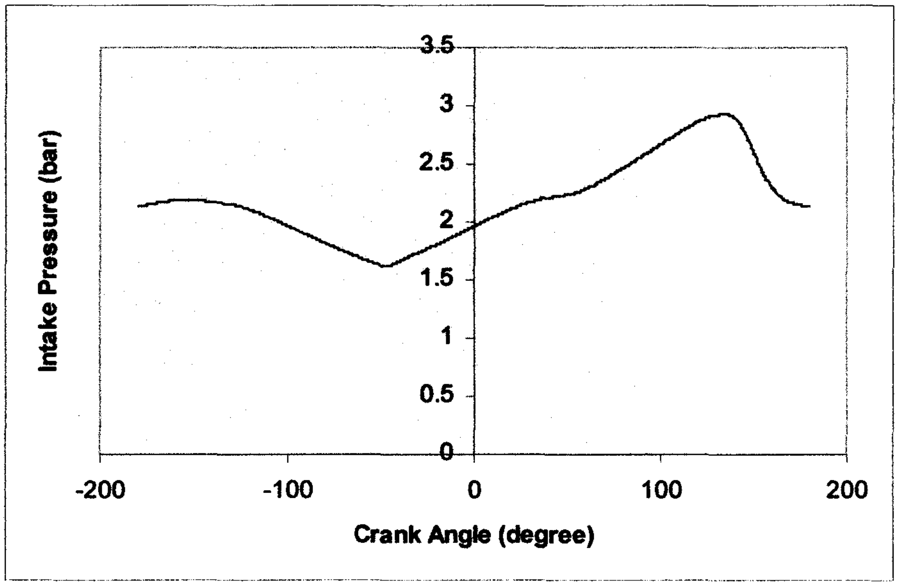


Figure 3.11: The experiment data as the pressure inlet in simulation for 1060 rpm

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

In this chapter, the result from the experiment and simulation will be discussed. The research is about the in-cylinder of two stroke spark ignition (SI) engine. The research is carried out using the experimental analysis and the numerical analysis using Computational Fluid Dynamic (CFD). The experiment was carried out according to the condition as mentioned in Table 3.2 to get data result that will be used as the boundary condition in the numerical analysis

4.2 Experimental Analysis Result

From the experimental analysis, a set of data is collected when the engine is running at the specific speed. The data will be used as the boundary condition in the numerical analysis. The data result from the experimental analysis for the 1061 rpm speed is shown in Figure 4.1. The cylinder pressure data was measured from the experiment is shown in Figure 4.2. The data from the experiment will be compared with the simulation data for the validation.

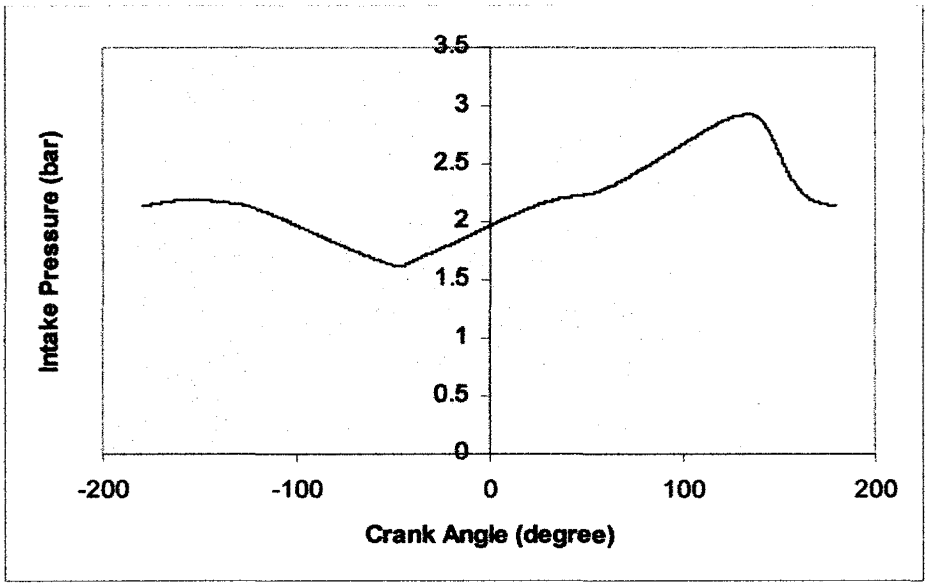


Figure 4.1: The experiment data as boundary condition in numerical analysis for 1061 rpm speed

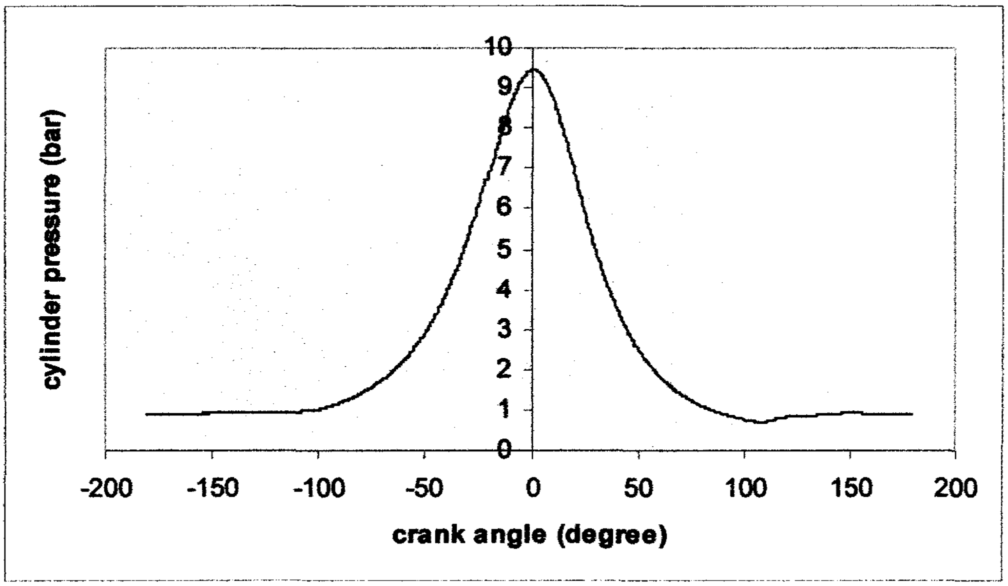
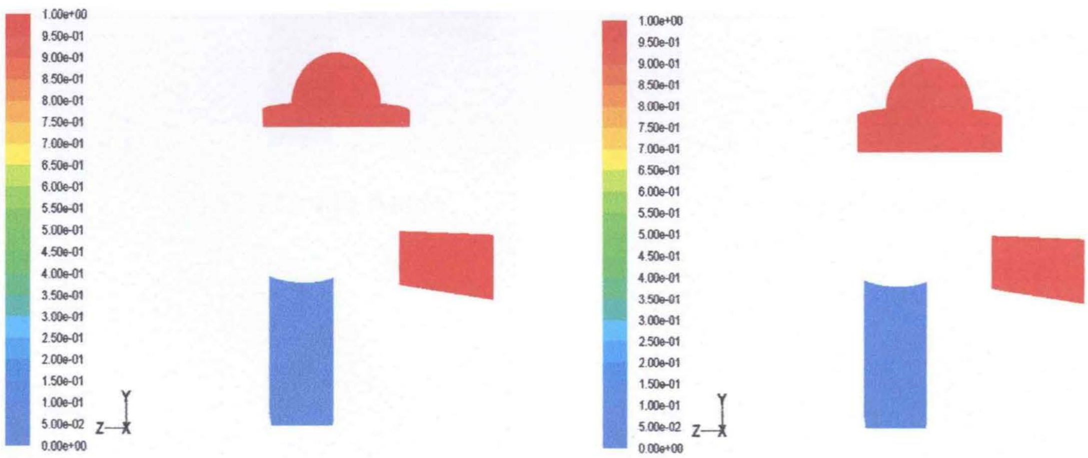


Figure 4.2: The in-cylinder pressure from the experimental analysis for 1061 rpm speed

4.3 The Numerical Analysis Result

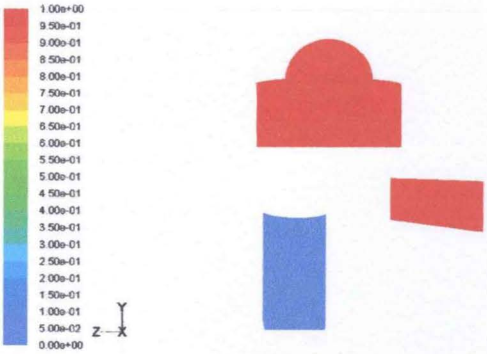
From the numerical analysis, the prediction of the burned and unburned mass fraction will explained the in-cylinder characteristic happen inside the cylinder. These characteristics also play the important role to the performance of the two stroke engine.

The two-stroke problems such as the short circuiting, unburned mixture and irregular combustion can be predicted by these characteristics. From the simulation, the effects of the speed to the burned and unburned characteristic can be predicted as shown in Figure

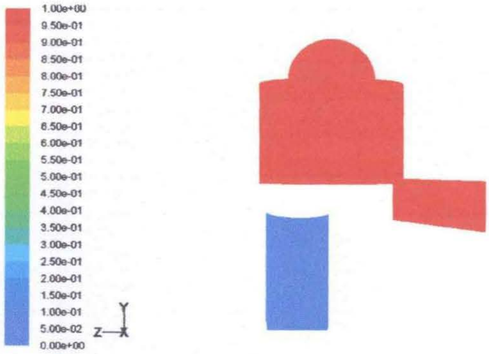


(a) 0 Crank Angle (b) 36 Crank Angle

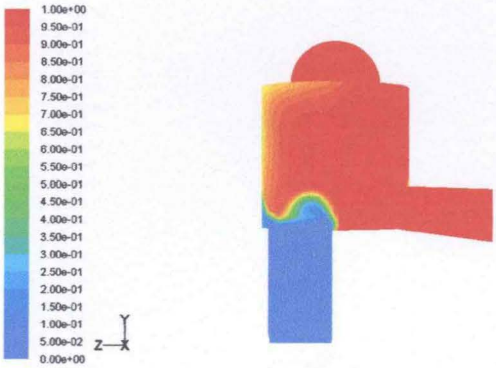
Figure 4.3: Burned mass fraction for a= 0 CA, b= 36 CA



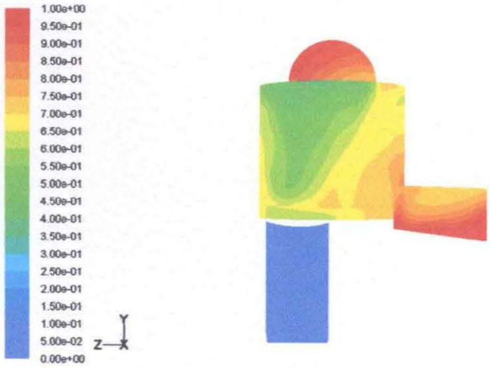
(a) 72 Crank Angle



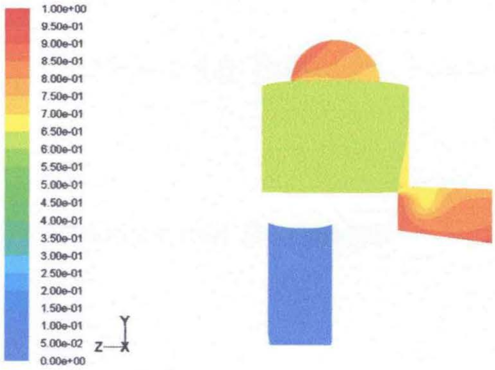
(b) 108 Crank Angle



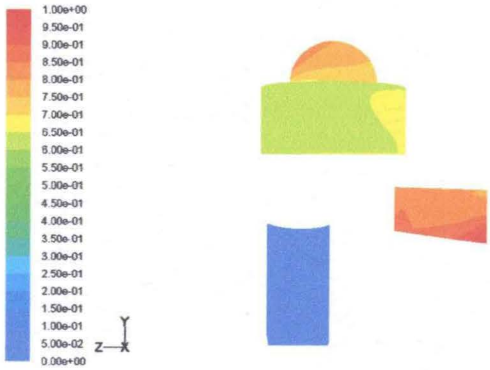
(c) 144 Crank Angle



(d) 180 Crank Angle

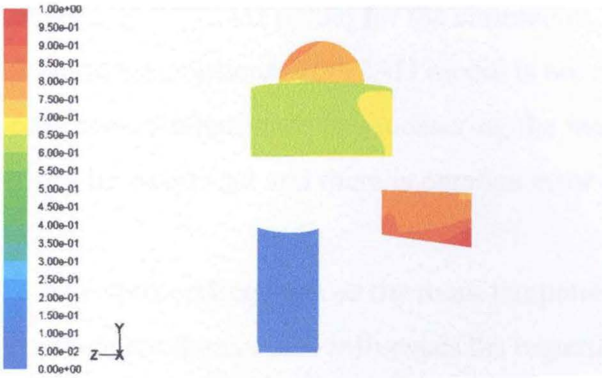


(e) 216 Crank Angle

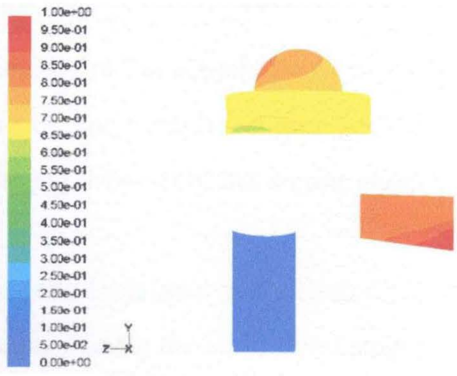


(f) 252 Crank Angle

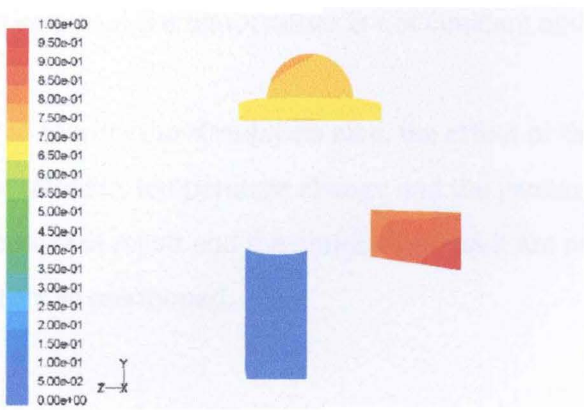
Figure 4.4: Burn Mass Fraction for a= 72 CA, b= 108 CA, c= 144 CA, d= 180 CA, e= 216 CA, f= 252 CA



(a) 288 Crank Angle



(b) 324 Crank Angle



(c) 360 Crank Angle

Figure 4.5: Burn Mass Fraction for a= 288 CA, b= 324 CA, c= 360 CA

4.4 Validation and Discussion

From the experiment, the maximum pressure for the 1061 rpm is 9.282 Bar but for the simulation, the maximum pressure value is 9.571 Bar. There is 3.02 % relative error for the experimental method and the numerical method. The in-cylinder pressure graph for both experimental and simulation is shown in Figure 4.14.

The relative errors are caused by the several factors such as the geometries dimension of the CAD model for the simulation, the experiment condition and the simulation assumptions. The CAD model is not accurate to the actual engine because there is some limitation during measuring the model. Some complex engine geometry needs to be simplified and there is parallax error during measuring the engine model.

The surrounding such as the room temperature and improper method handling the experiment equipment also influences the experiment. During the simulation stages, there is much assumption that is made such as the temperature and the pressure inlet of the simulation is set to constant. This is different to the experiment, which is the pressure, and the temperature is not constant and influenced by the surrounding.

During the simulation also, the effect of the surrounding such as the radiations, heat transfer, temperature change and the pressure changed is neglected. Therefore, the experiment result and the simulation result are not accurate because of the several factors as mentioned.

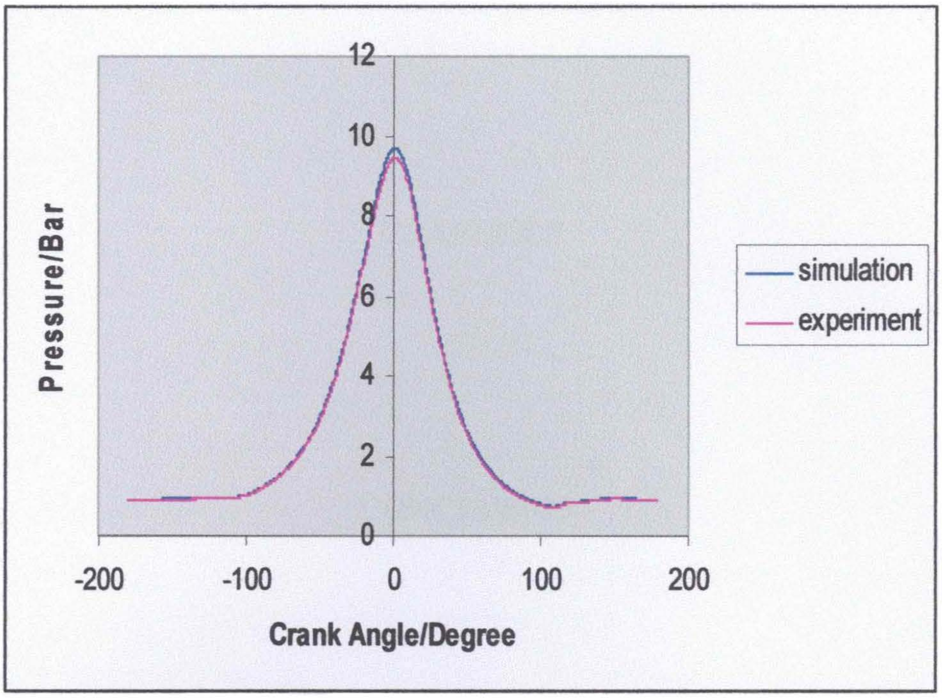


Figure 4.6: The validation of the in cylinder pressure for the 1061 rpm speed

CHAPTER 5

CONCLUSION

5.1 Conclusion

The aim of the research is to do numerical analysis to predict the in-cylinder flow of two-stroke spark ignition engine. The simulation is carried out using computational fluid dynamics (CFD) Fluent.

The operating and boundary conditions are measured using the experiment using the motored method, which is no combustion inside the cylinder. The experiment data then used in the simulation that is three-dimensional and unsteady condition. Then the result from the experiment and the simulation is validated.

The study of in-cylinder characteristic is important to understand the scavenging process of the two-stroke engine. This is important to design new two-stroke engine to produce low emission that will meet current and future emission regulation.

5.2 Recommendation

There are few improvements need to do for the future research. This is to improve the accuracy of the predicted in-cylinder characteristic. Some of the recommendations are:

i. The research is carried out using the firing method to understand the actual combustion of the two-stroke engine
ii. The research about the effect of the combustion chamber design, scavenged port design, exhaust port design, spark plug location and other parameter of the two-stroke engine to understand the scavenging process of the two-stroke engine.
iii. The dimensioning of the actual engine model is carried out using three-dimensional scanner to get the accurate dimension of the engine.

REFFRENCES

1. Blair, G. (1996). *The Design and Simulation of Two-Stroke Engine*. Society of Automotive Engineers Publishers, Ins.Warrendale, PA.

2. R. K. Rajput, "*A Textbook of Internal Combustion Engine*," M.E. (Heat Power Engg.) Hons.-Gold Medallist; Grad. (Mech. Engg. & Elect. Engg.); M.I.E (India); M.S.E.S.I. ; M.I.S.T.E. ; C.E. (India) PATIALA, 2005

3. B. D. Ragunathan and R. G. Kenny, "*CFD Simulation and Validation of the Flow within a Motored Two-Stroke Engine*," The Queen's University of Belfast, SAE Technical Paper Series 970359, 1997

4. J.G Smyth, R.G Kenny, G.P Blair, "*Steady Flow Analysis of the Scavenging Process in a Loop Scavenged Two-Stroke Cycle Engine Theoretical and Experimental Study*," SAE paper No. 881267, Society of Automotive Engineers, Warrendale, Pa.,1988.

5. R.G Kenny. J.G Smyth, R. Fleck, G.P Blair, "*The Scavenging Process in the Two-Stroke Engine Cylinder*," Dritte Grazer Zweiradtagung, Technische Universitat, Graz, 13-14 April, 1989.

6. Gordon P. Blair, "*design and Simulation of Two-Stroke Engines*," The Queen's University of Belfast, SAE Order No. R-161, 1994.

7. Marco Nuti, "*Emission From Two-Stroke Engine,*" Senior Staff Scientist Mercury Marine, SAE Order No. R-223, 1998

8. Heywood, J.B.and Sher, E. (1999). *The Two-Stroke Cycle Operations, and Design*. New York, McGraw-Hill.

9. Heywood, J.B.(1988). *Internal Combustion Engine Fundamental*. New York, McGraw-Hill

10. Grasso, I.F, Wey, M.J, Bracco, F.V and Abraham, J.(1987), *Three Dimensional Computational of Flows in a Stratified Charge Rotary Engine*. SAE 870409

11. Thornhill, D. Douglas, R. Kenny, R and Fitsimons, B.(1999). *An Experimental Investigation into the Effect of Bore/Stroke Ratio on a Simple Two-Stroke Cycle Engine*. SAE 1999-01-3342

12. Bozza, F. Tuccillo, R. and Domenico, D.F.(1995). *A Two-Stroke Engine Model Based on Advanced Simulation of Fundamental Processes*. SAE 95239

13. Yu, L. Cambell, T. and Pollock, W.(1997). *A Simulation Model for Direct-fuel Injection of Two-Stroke Gasoline Engines*. SAE 970366

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

15. Amsden, A.A. O'Rourke, P.J. and Butler, T.D.(1992). *Comparison of Computed and Measured three-dimensional Velocity Fields in a Motored Two-Stroke Engine*. SAE Paper 920418