Faculty of Mechanical Engineering

CFD ANALYSIS OF INTAKE FLOW IN THE L-HEAD ENGINE

Abdul Muhaimin bin Mohd Shafie

Master of Science in Mechanical Engineering

2017

CFD ANALYSIS OF INTAKE FLOW IN THE L-HEAD ENGINE

ABDUL MUHAIMIN BIN MOHD SHAFIE

A thesis submitted
In fulfilment of the requirement for the degree of Master of Science
in Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2017

DECLARATION

I declare that this thesis entitle “CFD Analysis of intake flow in the L-head engine” is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ………………………………………

Name : Abdul Muhaimin Bin Mohd Shafie

Date : ………………………………………

APPROVAL

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 Master of Science in Mechanical Engineering.

Signature : ...……………………..…………………………………...…

Supervisor Name : Associate Professor Dr. Musthafah Bin Mohd Tahir

Date : ...……………………………………………………………..

DEDICATION

I dedicate this thesis to my family for nursing me with affections and love and their dedicated partnership for success in my life.

ABSTRACT

The CNG fuel engine usually had lower performance comparing to gasoline fuel. A small single cylinder engine (L-head engine) is used in Green Technology Vehicle laboratory (GTeV) lab, which is low cost and commonly available in the market. It is difficult to experimentally to track the particle in 3D cases. Therefore, commercial numerical simulation is used. This thesis analysed the behaviour of the flow inside L-head combustion chamber for in-cylinder engine without the combustion with three different models were simulated. The objectives are to develop numerical model, to investigate the flow without combustion, to analyse flow with CNG and gasoline and to perform validation on the pressure between experiment and simulation. There are three types of simulation; steady, Port-flow and Cold-flow. The engine parameters and valve lift are measured, and the engine head were scan using ezScan 4.5 software. All of the simulations are simulated using ANSYS software. Only intake stroke is simulated for steady simulation with different crank angle and engine speed. Port-flow simulation is only simulated at intake stroke with introduction of CNG and gasoline as a fuel and three different valve lift. Cold-flow simulated a full cycle engine without combustion process. The Steady simulation is dealing with the static domain. There are only combustion chambers and piston volume involved in the steady simulation. The air inlet velocity was calculated using the standard engine formula for different piston position. The second simulation, Port-flow simulation also deals with the static geometry domain. Inplenum and outplenum was added by the presence of both intake and exhaust valve for the Port flow simulation domain. Three different valve lift was chosen. Gasoline and CNG were used as fuel, which enters the domain through the fuel intake. The last simulation is called Cold-flow where the geometry is moving according to the crank angle. The intake valve and exhaust valve are moving according to the measured valve profile. Meanwhile the piston movement was generated according to the crank angle of the engine. The result of steady flow simulation shows the velocity is high when the piston position is at 45° and engine speed of 4500 rpm. The result of Port-flow simulation shows the mass flow rate and velocity across the domain increase as the valve lift increase. The pressure difference between the intake port and combustion chamber decrease as valve lift increase. The swirl ratio decreases as going down the cylinder. The Cold-flow result shows the turbulence kinetic energy, swirl, tumble, and cross-tumble ratio inside the combustion chamber increase in the middle of the intake stroke. The temperature inside combustion chamber is increasing as the piston reaches TDC due to compression process. The result of Cold-flow simulation is validated by experiment without combustion with 22.73% of percentage difference at peak pressure. The combustion chamber head has been scanned and imported to ANSYS software. The velocity is highest when the piston located at the middle of the stroke and lowest then the piston approaching TDC and BDC. The flow pattern of gasoline and CNG has no significant change. The pressure for experiment and Cold-flow simulation is validated through its pressure pattern.

ABSTRAK

Enjin CNG mempunyai prestasi yang rendah berbanding petrol. Sebuah enjin kecil satu silinder (enjin kepala-L) digunakan, di dalam Makmal Kenderaan Hijau (GTeV) yang mempunyai kos yang rendah dan biasa terdapat di pasaran. Adalah sukar untuk menjejak zarah dalam bentuk 3D. Oleh itu, simulasi berangka komersial digunakan. Tesis ini menganalisis kelakuan aliran didalam kebuk pembakaran enjin berkepala L tanpa pembakaran dengan tiga model yang berlainan. Obejktif adalah untuk membangunkan model berangka, untuk menganalisis aliran dengan CNG dan petrol dan melaksanakan pengesahan pada tekanan diantara eksperimen dan simulasi. Terdapat tiga jenis simulasi; kekal, aliran-rongga, dan aliran-sejuk. Parameter enjin dan valve diukur dan kepala enjin telah diimbas menggunakan perisian ezScan 4.5. Semua simulasi hanya menggunakan perisian ANSYS. Hanya lejang pengambilan diambil kira untuk simulasi kekal. Simulasi aliran-rongga disimulasi pada lejang masukan dengan pengenalan CNG dan petrol sebagai bahan api dengan tiga ketinggian injap yang berbeza. Kitaran penuh digunakan untuk simulasi aliran-sejuk tanpa proses pembakaran. Simulasi kekal adalah berkaitan dengan domain statik. Hanya isipadu kebuk pembakaran dan omboh diambik. Halaju udara masuk dikira menggunakan formula standard yang digunakan untuk pengiraan enjin. Simulasi kedua aliran-rongga yang juga menggunakan domain geometri static. Inplanum dan outplenum ditambah dengan kehadiran kedua-dua injap pengambilan dan ekzos untuk domain aliran-port. Terdapat juga tiga perbezaan kenaikan injap. Petrol dan juga CNG digunakan sebagai bahan api dimana ia masuk ke dalam domain melalui permukaan kemasukan bahan api. Simulasi yang terakhir dipanggil aliran-sejuk dimana geometri akan bergerak mengikut pada sudut engkol. Injap kemasukan dan injap ekzos bergerak mengikut profil injap yang diukur. Sementara itu, pergerakan omboh dihasilkan mengikut sudut engkol engin. Keputusan untuk simulasi aliran kekal menunjukkan halaju tinggi apabila omboh berada pada 45° dan enjin pada 4500 rpm untuk kedua-dua sudut. Keputusan aliran- rongga menunjukkan seluruh kadar aliran jisim dan halaju di dalam domain meningkat apabila injap menaik. Perbezaan tekanan di antara rongga kemasukan dan kebuk pembakaran menurun apabila tinggi injap menaik. Penurunan nisbah pusaran berlaku apabula aliran mengalir menjauhi kedalam silinder. Keputusan aliran-sejuk membincangkan berkenaan pergolakan tenaga kinetic, nisbah pusaran, nisbah tumble, dan nisbah bertentangan tumble di dalam kebuk pembakaran menaik pada pertengahan proses lejang masuk. Suhu didalam kebuk pembakaran menaik apabila omboh menghampiri kedudukan TDC pada lejang mampatan. Hasil simulatsi aliran-sejuk disahkan dengan eksperimen tanpa pembakaran dengan 22.73% perbezaan peratus pada tekanan memuncak. Kepala kebuk pembakaran telah diimbas dan diimport ke perisian ANSYS. Halaju paling tinggi apabila omboh terletak di tengah-tengah lejang dan kembali rendah bila menghampiri TDC dan BDC. Corak aliran untuk petrol dan CNG tidak mempunyai perubahan ketara. Tekanan eksperimen dan simulate aliran-sejuk disahkan dengan corak tekanan yang terhasil.

ACKNOWLEDGEMENT

First and foremost, I would like to take this opportunity to express my sincere acknowledgement to my main supervisor Dr. Musthafah bin Mohd Tahir and my second supervisor Dr. Ahmad Kamal bin Mat Yamin from faculty of Mechanical Engineering University Teknikal Malaysia Melaka (UTeM) for his essential supervision, support, patience, motivation, immense knowledge and encouragement towards the completion of this thesis. I gratefully acknowledge UTeM for their financial support via Short Term Grant Project (PJP) in this research activity. My sincere thanks also goes to Prof. Dato‟ Rosli bin Abu Bakar and Dr. Devarajan a/l Ramasamy from Universiti Malaysia Pahang (UMP) for being my external advisors that provide me with knowledge and chance to use their facilities. Without they solemn support it would not be possible to conduct this research. I thank my fellow partner in UTeM, Muhammad Syahir bin Ali and Ammar Alfaiz bin Mustaffa Albakri for their outstanding collaboration in the work and also for being very good sharing partners during my research. My sincere thanks to my fellow labmates, Fauzi bin Ahmad, Mohamad Zaharudin bin Sariman, Mohd Sabirin bin Rahmat, Hazrin bin Ismail, Ashafi‟e bin Mustaffa, Kamarulhelmy bin Talib, and Abdurahman Dwijotomo. I also would like to thank to all of the technicians from Green Technology Vehicle Laboratory, Nor Izwan bin Junoh who helped me throughout my study. Last but not least, I would like to thank my family, my parents and my sisters for supporting me spiritually throughout writing this thesis and my life in general.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS AND SYMBOLS

A/F	Air Fuel
APEC	Asia-Pacific Economic Cooperation
ATDC	After Top Dead Centre
BDC	Bottom Dead Centre
BTDC	Before Top Dead Centre
CAD	Computational-aided Design
CFD	Computational Fluid Dynamic
CNG	Compress Natural Gas
DAQ	Data acquisition
DES	Detached Eddy Simulation
DNS	Direct Numerical Simulation
EVC	Exhaust Valve Closed
EVM	Eddy Viscosity Model
EVO	Exhaust Valve Open
FDM	Finite Difference Method
FEM	Finite Element Method
FVM	Finite Volume Method
GTeV	Green Technology Vehicle
HP	Horsepower

HWA	hot wire anemometry
ICE	Internal Combustion Engine
IEA	International Energy Agency
IG	Ignition Point
IVO	Intake Valve Open
Ktoe	Kilotonne of Oil Equivalent
LDA	Laser Doppler Anemometry
LDV	Laser Doppler Velocimetry
LES	Large Eddy Simulation
LNG	Liquefied Natural Gas
LPG	Liquefied Petroleum Gas
NG	Natural Gas
NGV	Natural Gas Vehicle
OpenFOAM	Open Field Operation And Manipulation
PIV	Particle Image Velocimetry
PTV	Particle Tracking Velocimetry
RANS	Reynolds Average Navier-Stokes
RNG	Re-Normalization Group
RPM	Revolution Per Minute
RPS	Revolution Per Second
RSM	Reynolds Stress Model
SST	Shear Stress Transport
TDC	Top Dead Center
TKE	Turbulence Kinetic Energy

UDF	User Defined Function
3D	Three-dimensional
Re	Reynolds number
v_{avg}	flow average velocity
V	Flow velocity
D_h	Characteristic length of geometry
D_{hs}	Characteristic length of spherical geometry
D	inner diameter of pipe
S	stroke length
a	crankshaft offset or radius of the crank shaft
$\bar{U}p$	average piston speed
s	distance between the crank axis and the wrist pin axis
r	length of the connecting rod
R	ratio of the connecting rod length to crank is offset
U_p	Instantaneous piston speed
r_c	compression ratio
v_d	Volume Displacement
v_c	Clearance volume
v_{cyl}	Cylinder volume
B	Bore
A	Cross sectional area
Q	Volume flow rate
k_2	Theoretical value of compression ratio

k_1	Simulation value of compression ratio
R_s	Swirl ratio
R_t	Tumble ratio
ω_s	Angular velocity
N	Engine speed
p	Piston position
y	Length of rectangle duct
n	Height of rectangle duct
W	Diameter of circular area
j	Radius of sphere
$a\vec{m}$	Angular momentum per unit mass
$\vec{r}$	Radial coordinate
$\vec{v}$	Velocity
$s\vec{a}$	Swirl axis
SN	Swirl number
$L._{sa}$	Magnitude of angular momentum respect to swirl axis
$I._{sa}$	Moment of inertia of fluid mass about swirl axis
d	Diameter
u	Fluid velocity
Q_{air}	Air flow rate
Q_{fuel}	Fuel flow rate
ρ	Density
μ	dynamic viscosity

π	Pi (3.14159)
θ	crank angle

LIST OF APPENDICES

LIST OF PUBLICATIONS

Mohd Shafie, A. M., Musthafah, M. T., Ali, M. S., Bakar, R. A., Mohamed Arifin, Y., 2015. *Intake Analysis on Four-Stroke Engine Using CFD*. Journal of Engineering and Applied Sciences, 10(17), pp. 7799-7804

Ali, M. S., Musthafah, M. T., **Mohd Shafie, A. M.**,Bakar, R. A., 2014.*Simulation of Single Cylinder Engine Fuel with Alternative Fuel by Using Available Software*. International Review of Mechanical Engineering, 8(4), pp.798-802

Musthafah, M. T., Ali, M. S., Salim, M. A., Bakar, R. A., Fudhail, A. M., Hassan, M. Z., **Mohd Shafie, A. M.**, 2014. *Performance Analysis of Spark Ignition Engine Using Compressed Natural Gas (CNG) as Fuel*. Energy Procedia, 68, pp.335-362

CHAPTER 1

INTRODUCTION

1.1 Overview

Improvement of the efficiency of the combustion engine is vital to diminish the global energy crisis and environmental impact of global warming. Fossil fuels such as gasoline and diesel are non-renewable energy, which keep decreasing as the demand of the energy increases over the year globally. International Energy Agency (IEA) (2014) found that growths of energy demand to the year of 2040 are increasing at about 37% with the average rate of growth of 1.1% per year. Meanwhile, the oil demand is rising at 14mb/d to achieve 104mb/d in 2040, with China as the largest oil-consuming country, despite the new policies all over the world promoting switching the fuel and enhancing the energy efficiency (IEA, 2014).

Devarajan (2015) stated that the reserve of fossil fuel around the world could only provide up to 40-50 years only as it is a finite amount. Due to this limitation of fossil fuel, research has to be more profound in order to utilize the highest efficiency of the system to reduce the consumption of the fuel. According to Deng and Liu (2013), saving one litre of gasoline can reduce the emission of carbon dioxide and carbon emission by 2.3 kg and 0.627 kg respectively.

In Malaysia, the primary energy supply is equivalent to 83 938 kilotons (ktoe) that comes from natural gas (46.0 %), oil (32.1 %), coal (18.9 %), hydro (2.9 %), and renewable energy (0.4 %). Here, the ktoe refers to an energy unit released with the amount of 1000 tons of oil burning, which is equivalent to 42 gigajoules. Total final energy consumption in Malaysia in 2012 was 46 711 ktoe. Transport sector becomes the highest consumption, which is 36.8 % of the total final energy consumption (Malaysia Energy Statistics, 2014).

Environmental issues, such as global warming, have become the major factor to be focused. According to APEC (2013), carbon dioxide is estimated to increase by about 46% in year 2035 compared to the year 2010. Much legislation has been established to control the emission, leading to the reduction of emission, which started with Euro 1 standards in 1992 involving passenger car only.

The used of fuel for the experiment can be minimised by using a single cylinder engine rather than an actual commercial passenger car‟s engine. In Green Technology Vehicle (GTeV) laboratory, instead of using a four-cylinder engine for experiment at one time, the researcher can conduct other experiment using a single cylinder engine. L-head engine which is a small, low-cost and single-cylinder engine that commonly used in basic farm machinery. Furthermore, L-head engine and its spare parts is highly available in the market for a single cylinder engine segment. Due to this condition, the study of CFD simulation will be vital for the research.

1.2 Problem Statement

(i) It is essentially important to study flow occurs in automotive engines. The flow criteria in engines such as swirl, values of turbulent kinetic energy are widely known to affect the engine performance. For example, high turbulent kinetic energy results in better air-fuel mixing, which consequently improves the engine performance. Experimental works are expensive and time consuming. Furthermore, it is difficult to experimentally track flow particle in three dimensional (3-D) cases. Hence, numerical simulations using commercial softwares are the reliable alternative to experiments. There are limited numbers of numerical simulations work that focus on flows in a complete engine stroke. This study presents a fundamental numerical analysis of intake flow in a complete cycle engine. L-head shape engine is utilized as the computational domain. This type of engine head is different from the conventional engine.

(ii) L-head engine simulation can be hardly found due to its combustion chamber unique design. Therefore, the flow study have to be conducted in the GTeV laboratory.

1.3 Objectives

The objectives of this thesis have been set as follows:

(i) To develop numerical model of a Robin EY-20-3 combustion chamber engine incorporate with valve lift profile.

(ii) To investigate the flow inside a combustion chamber of a single cylinder Robin EY-20-3 engine without combustion process.

(iii) To analyse the flow inside L-type in-cylinder Robin EY-20-3 engine using numerical analysis with Compressed Natural Gas (CNG) and gasoline.

(iv) To perform validation on the built-up pressure obtained inside the Robin EY-20-3 engine between experiments and simulation.

1.4 Scopes

The job scopes of this research are as follows:-

(i) The engine parameters and valve lift profiles were measured and only Robin EY-20-3 engine head were scan using ezScan4.5 software. Only ANSYS Fluent are used for the numerical simulation.