# AIR & FUEL FLOW ANALYSIS INSIDE THE INTERNAL COMBUSTION ENGINE WITH OVERHEAD CAM

MUHAMMAD AFIQ BIN MOHD NOR AZINAN

**This report is submitted in partial fulfillment of the requirements for the award of a Bachelor in Mechanical Engineering (Automotive) with honours**

**Faculty of Mechanical Engineering**
**Universiti Teknikal Malaysia Melaka**

**2017**

## DECLARATION

"I hereby declare that the work in this thesis is my own except for the summaries and quotations which have been duly acknowledge."

Signature :..........................

Author : MUHAMMAD AFIQ BIN MOHD NOR AZINAN

Date :..16/6/2017........................

**SUPERVISOR DECLARATION**

I have checked this report and the report can now be submitted to JK-PSM to be delivered back to supervisor and to the second examiner.

Signature : ..............................

Supervisor: PROF MADYA. DR. MUSTHAFAH BIN MOHD. TAHIR

Date : ....19/6/2017....

# ABSTRACT

The importance of flow patterns in internal combustion engine is to intensify the performance of the engine in presence of swirl and tumble. Therefore, it is crucial to investigate the flow behavior at the intake valve. In this research, the effects of valve lift to the flow pattern inside the engine are to be investigated. Computational Fluid Dynamic is used to predict the flow behavior in the single cylinder engine. The higher velocity during intake stroke is believed to increased swirl and tumble throughout compression stroke. Besides analyzing the flow behavior, the findings of this project will helps in increasing the performance of engine. Results of the simulation are presented in graphical data in terms of swirl pattern and tumble pattern. Besides, the graph of the Turbulence Kinetic Energy is presented to support the results. A maximum value is compared for all four parameters. The parameters are taken from different valve lift from the lowest to the highest valve lift. From the result, it is found that the lower valve lift results in better swirl, tumble and turbulence kinetic energy value. This gives off better performance to the engine.

# ABSTRAK

Kepentingan corak aliran di dalam enjin pembakaran dalam adalah untuk meningkatkan prestasi sesebuah enjin dengan kehadiran aliran berpusar dan aliran jatuh. Oleh itu, kajian tentang corak aliran pada injap masukan adalah sangat penting. Kajian ini memfokuskan tentang kesan bukaan injap yang berbeza-beza terhadap corak aliran di dalam enjin. Dinamik Bendalir Berkomputer digunakan untuk meramal ciri-ciri aliran pada injap masukan di dalam enjin satu silinder. Aliran berpusar dan aliran jatuh dipercayai akan meningkat semasa lejang mampatan apabila halaju meningkat semasa melalui injap masukan. Selain daripada penganalisaan terhadap ciri-ciri aliran, hasil penemuan projek ini juga akan membantu dalam meningkatkan prestasi enjin. Keputusan kajian simulasi dipersembahkan dalam bentuk grafik di mana aliran berpusar dan aliran jatuh di tunjukkan. Selain itu graf pergolakan tenaga kinetik turut dipaparkan untuk menyokong lagi hasil dapatan simulasi yang telah dijalankan. Nilai maksimum digunakan bagi membandingkan kesemua parameter. Parameter didalam kajian ini melibatkan bukaan injap yang berbeza-beza bermula dari bukaan injap paling kecil sehingga bukaan injap terbesar. Hasil kajian mendapati, bukaan injap yang rendah menghasilkan nilai aliran berpusar, aliran jatuh dan pergolakan tenaga kinetik adalah lebih tinggi. Hal ini membantu meningkatkan prestasi enjin.

# ACKNOWLEDGEMENT

I would like to express my heartfelt gratitude to my supervisor Assoc. Prof. Dr. Musthafah bin Mohd. Tahir, for his encouragement, patience and support to complete my project and thesis writing.

I would also like to thank my senior Mr. Khairil Amri Tajuddin and Miss. Nur Ameelia Binti Rosli who helped me a lot with the understanding of the simulation and share all the knowledge related to my research field. Also my partner Mr. Amarul Afiq Adli Rosli who helped me all the way from the start tills the completion of this project.

I would like to thank Dr. Fatimah Al-Zahrah binti Mohd Sa'at for the suggestion and critics and finally to technician of Computer Aided Engineering Lab, Mr. Mohd Firdhaus bin Md Razali for his cooperation in the process of completing the simulation. Last but not less to all technicians in Fasa B who had been contributes to complete this project.

## TABLE OF CONTENT

## LIST OF TABLES

# LIST OF FIGURES

## LIST OF APPENDIX

## LIST OF ABBREVIATIONS

| | |
|---|---|
| CFD | Computational Fluid Dynamic |
| CNG | Compressed Natural Gas |
| ICE | Internal Combustion Engine |
| NGV | Natural Gas Vehicle |
| PSM | Projek Sarjana Muda |
| FYP | Final Year Project |
| BDC | Bottom Dead Centre |
| TDC | Top Dead Centre |
| 2D | 2-Dimension |
| 3D | 3-Dimension |
| SR | Swirl Ratio |
| AF Ratio | Air Fuel Ratio |
| etc. | etcetera |

## LIST OF SYMBOLS AND UNITS

| | |
|---|---|
| kJ/kg | kilo Joule per kilogram |
| MJ/kg | Mega Joule per kilogram |
| Pa | Pascal |
| m/s | meter per second |
| $Q$ | Flow rate |
| A | Area |
| u | Fluid velocity |
| S | Stroke length |
| N | engine speed per one revolution |
| Ūp | average piston speed |
| rpm | revolution per minute |
| kg/mol | kilogram per mol |
| ω | angular speed |
| $u_t$ | swirl tangential speed |
| $\bar{U}_p$ | average piston speed |
| °C | degree Celsius |
| $NO_x$ | Nitrogen Oxide |
| C-H ratio | Carbon-hydrogen ration |
| CO | Carbon Oxide |
| HC | Hydrogen Carbon |

# CHAPTER 1

# INTRODUCTION

## 1.0 PROBLEM STATEMENT

Compressed Natural Gas (CNG) were been used for nowadays car, due to the CNG advantages such as lower flame speed, cheaper than petrol and diesel, and etc. furthermore by using this CNG it will reduce air pollution because it's have lower greenhouse gas emission compared to the gasoline or petrol.

CNG also consists of its own disadvantages regarding to its performance output. New geometry internal combustion engine are need to be design using suitable software, which is ANSYS Fluent Software. To analyses the CNG air and fuel flow inside the internal engine with overhead cam by using Computational Fluid Dynamic (CFD). This is because to do this project it will cost a lot of money to buy the apparatus or gadget that can detect the movement of air fuel flow inside the combustion engine. Furthermore, this software can helps in giving more knowledge about the flow inside the intake stroke. CFD are needed to demonstrate swirl and tumble behaviors during intake and compression stroke so that improvement can be made to increase the performance.

Furthermore from thr previous study it using a l type engine which is outdated and the engine is uncommonly use in public. Thus we will study on the new engine which is updated to the current era which is engine with overhead cam.

## 1.1 OBJECTIVE

The objectives needed to be done in order to complete this project are listed as follows:

a) To stimulate flows in the engine using ANSYS Fluent for single cylinder engine with overhead cam at intake stroke

b) To investigate the effect of different valve lift to the flow in the engine

c) To analyze the swirl and tumble at intake stroke.

## 1.2 SCOPE OF STUDY

a) To produce geometry and analyze at the intake stroke

b) Design the engine and overhead cam using CATIA CAD.

c) Conduct air and fuel flow simulation

d) Analyze the flow of swirl and tumble from the simulation using ANSYS Fluent Software.

# CHAPTER II

# LITERATURE REVIEW

## 2.0 OVERVIEW

This chapter will cover the review on the Compressed Natural Gas (CNG), internal combustion engine, and computational fluid dynamic and fluid motion within combustion chamber.

## 2.1 COMPRESSED NATURAL GAS

Natural gas is normally happening type of fossil vitality and along these lines renewable. Natural gas happens as gas under pressure in rocks underneath the world's surface or frequently in arrangement with unrefined petroleum as an unpredictable part of petroleum. It is actually hydrocarbon energy shaped in the world's covering by a great many years of natural activity on natural matter (Ramadhas, 2011). In this manner, it doesn't require experiencing refining process as petroleum.

Normal gas is made essentially out of methane (60-98%) with little amount of other hydrocarbon fuel components. It contains different amount of propane, butanes, pentanes and hints of different gasses. Next to that, dynamic mixes additionally/active compounds can be found in the common gas, for example, sulfur and inactive mixes, for example, nitrogen and carbon dioxide. CNG has higher octane number making it reasonable for start ignition engine or a composed gas engine with higher compression ratio.

As been said by Bhandari, K. et al (2005), the higher octane number in Compressed Natural Gas (CNG) empower the engine to work at higher compression ratio (CR) and it might higher efficiency and higher power created. CNG demonstrates higher ignition temperature contrasted with gas. In addition, it is lighter than air density. Because of those properties, CNG is probably going to be more secure than gas. Composition of natural gas in examination with fuel is appeared in Table 2.1.

**Table 2.1**: Composition of natural gas in comparison with gasoline

| Properties | Gasoline | CNG |
|---|---|---|
| **Motor octane number** | 80-90 | 120 |
| **Molar mass (kg/mol)** | 110 | 16.04 |
| **Carbon weight fraction (mass %)** | 87 | 75 |
| **Stoichiometric air fuel ratio (A/F)s** | 14.6 | 16.79 |
| **Stoichiometric mixture density (kg/$m^3$)** | 1.38 | 1.24 |
| **Lower heating value (MJ/kg)** | 43.6 | 47.377 |
| **Lower heating value of stoic. Mixture (MJ/kg)** | 2.83 | 2.72 |
| **Flammability limits (vol% in air)** | 1.3-7.1 | 5-15 |
| **Spontaneous ignition temperature °C** | 480-550 | 645 |

Natural gas is compressed and stored to be utilized as a part of automobile. It is important to store it in minimal shape to diminish the weight of the vehicle. The capacity limit of CNG is in a range from 20 up to 100 liters. The chamber stockpiling/storage must ready to handle pressure of 600 bar. This is a security measure if at any time the storage cylinder is exposed to fire.

CNG is a safe fuel alluding to its physical, substance and burning trademark. Regular gas or methane is a nontoxic gas. It is lighter than air which implies it won't sink to the ground like propane which is heavier than air. Natural gas will go up to the atmosphere and disperse in the air.

Ramadhas, A. S (2011) has inferred that CNG contains the accompanying points of interest;

a) Environmental: CNG vehicles produce far less of all regulated pollutants compared to gasoline or diesel vehicles including $No_x$ and particulate matter. Natural gas has low C-H ratio hence lower CO and HC emissions. CNG vehicles produce far less unregulated air toxics and greenhouse gases. Due to proper combustion of gas-air mixtures, reduced unburned HC emissions will reduce the environmental pollution of visible photochemical smoke.
b) Energy security: Natural gas usage reduces the consumption of gasoline and diesel fuel.
c) Operating cost: Cheaper than gasoline and diesel fuel.
d) Distribution efficiency safety: Natural gas has a higher ignition temperature than gasoline or diesel. Its density that is lighter than air makes it disperses quickly if leakage of fuel is to happen. This shows the safest and most efficient energy distribution system. The explosive limit of natural gas-air mixtures is higher than diesel-air mixtures. Natural gas requires approximately 5% of volume compared to 2% for propane and 1% for gasoline vapor for continuous flame propagation. This means natural gas is safer than other fuels.
e) Flexibility: CNG vehicles can be produces distinctively and bi-fuel versions. CNG vehicles are suitable at the area where natural gas is available. While bi-fuel vehicles can operate both natural gas and gasoline fuel as they have storage tanks on board. This kind of vehicle can operate by using natural and easily switch to gasoline if the area does not provide natural gas supply. It is economical and environmentally friendly.

Be that as it may, research has additionally discovered disadvantages of utilizing natural gas. Because of the engine's low-volumetric efficiency and low-energy density, engine performance diminishes/decrease.

## 2.2 FLOW MOTION IN COMBUSTION CHAMBER

Movement in ignition chamber for all streams into, out of and inside the barrel is turbulence. Panday and Bidesh (2012) said that air flow for the ICE is for the most part in turbulent stream as the speed if ICE is high. Comes about because of the turbulence stream expands the thermal exchange, evaporation, mixing and combustion rates inside the engine. Amid admission/intake, turbulence stream produces rotational movement called swirl noticeable all around fuel blend/air-fuel mixture. Toward the end of the pressure stroke, a rotational movement around a circumferential hub called tumble is framed. Another stream which frames radially towards the centreline is called squish. There are additionally flows in the little crevices of the ignition chamber which are crevices flow and blowby.

Turbulence capacity is specifically corresponding to the engine speed where the expanded of engine speed will expanded the turbulence. Expanding engine speed brings about expanding movement flow rates and also whirls and tumble movement of the flow. Turbulence in the combustion is high amid intake. It diminishes as the flow rate moderates near bottom dead center (BDC). At pressure/compression stroke, the flow increments again as swirl tumble and squish increment near top dead center (TDC). Swirl created the turbulence to be more homogenous completely through the cylinder.

At the point when ignition start to happens, high turbulence close TDC is pined for by combustion since it separates and spreads the fire front quicker than laminar would. Time utilization of air-fuel abbreviates in this way keeping away from self-ignition and knock. As indicated by Pulkrabek (2008), turbulence is enhance by the expansions of the cylinder gasses amid the combustion procedure and the state of the burning chamber is extremely important in producing maximum turbulence and expanding the desired of rapid combustion.

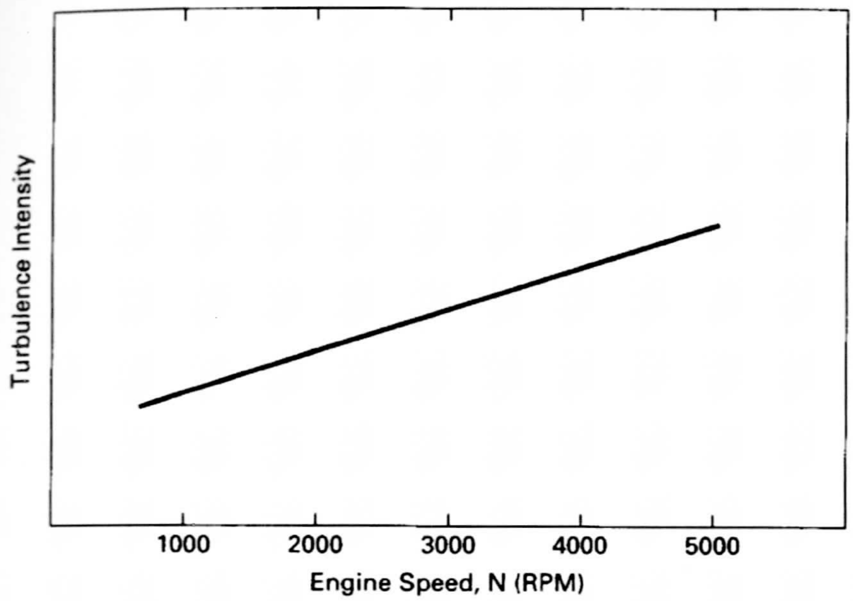


**Figure 2.1**: Turbulence intensity as a function of engine speed.

Swirl (Figure 2.1) fundamentally escalates the blending of air and fuel to give a homogeneous blend in the brief timeframe and quick spreading the front flame over the span of combustion process. To generate swirl, the intake framework was made in an approach to give an extraneous component to the intake flow while entering the cylinder (Figure 2.6).

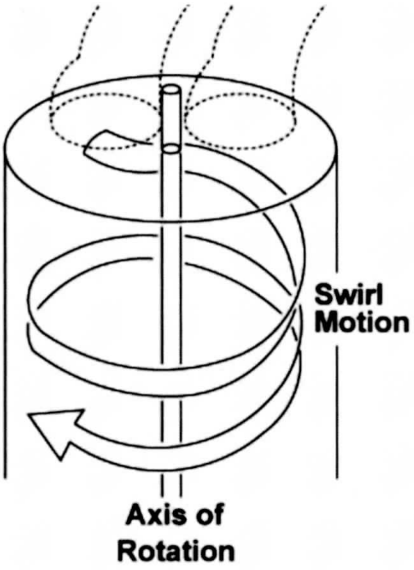


**Figure 2.2**: Swirl motion in engine cylinder

Swirl behaviors changes along the process. It is high amid intake and decline during compression stroke after BDC because of viscous drag with cylinder walls. At power stroke, the combustion expands the gasses and increase swirl. Development of the gasses and viscous drag diminish this maintaining a strategic distance from blowdown. Swirl ratio can be in the scope of 5 to 10 for an advanced engine. The angular momentum is later will be lost from one-forth to 33% at compression stroke.

Toward the end of compression stroke as the piston approaches TDC, the external volume is almost zero constraining it to move radially inward. This will help quickly spread the flame front. This movement is called squish and it regularly happens at around 10° bTDC. The second part of the piston moving upward approach TDC produces another rotational flow known as tumble. Tumble happens around circumferential pivot/axis near the external edge of the piston bowl (Figure 2.3).

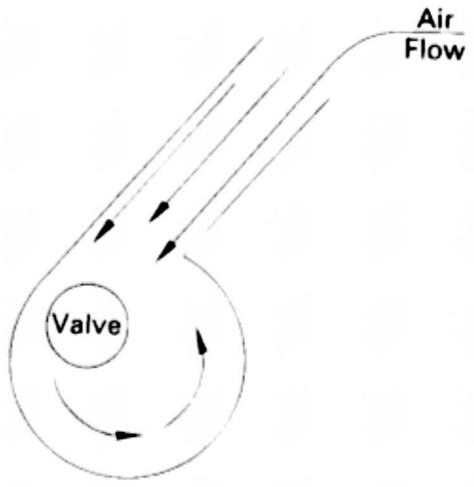


**Figure 2.3**: Air entering the cylinder from tangential direction

Tumble is extremely alluring in modern engine as it builds up the stratification of the air-fuel blend in engines that works with stratified charge combustion. This sort of combustion expects to optimum thermal efficiency while minimizing fuel utilization and discharges or emissions.

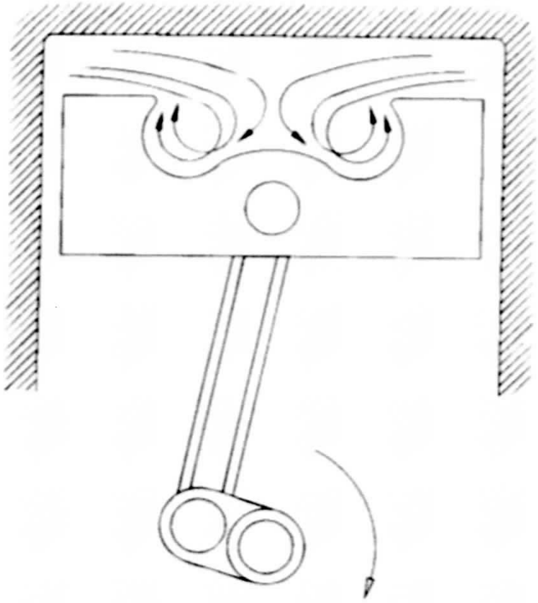

**Figure 2.4**: Tumble occurs around circumferential axis.

## 2.3 Swirl Theories

Twirl is the gigantic mass motion moving rotationally in a cylindrical that produced by developing the intake framework to give tangential component to the intake flow when it enters the cylinder. To measure the rotational movement in the cylinder, a dimensionless parameter called swirl ratio was characterized. There are two methods for calculating the swirl ratio which are;

$$(SR)_1 = \omega / N \quad \mathbf{1}$$

Where

$\omega$ = angular speed

$N$ = engine speed

$$(SR)_2 = u_t / U_p \quad \mathbf{2}$$

Where

$u_t$ = swirl tangential speed