AIR FUEL MIXING MODELING FOR DIRECT AND TRANSFER PORT INJECTION IN TWO STROKE ENGINE

TAN YEE HERN

UNIVERSITY SCIENCE MALAYSIA

2013

SIMULASI PENCAMPURAN

UDARA DAN BAHAN API

BAGI PEMANCITAN TERUS DAN

PEMANCITAN LIANG PINDAH

DALAM ENJIN DUA LEJANG

TAN YEE HERN

UNIVERSITI SAINS MALAYSIA

2013

ACKNOWLEDGEMENT

The writing of this master thesis has been made possible with the help and support of the kind people around me. Above all, I am greatly indebted to and truly appreciate my beloved wife, Bee for all her encouragement, support and sacrifices throughout the study. To my parents, brothers and sisters, I would like to say a big thank you for their continuous support and encouragement.

I would like to express my heartfelt thanks and appreciation to my principal supervisor, Dr. Mohamad Yusof Idroas and field supervisor, Prof. Horizon Walker Gitano Briggs for their help, support and patience in providing guidance and advice that has enabled me to complete the research. His unsurpassed knowledge of engines and analytical skills have been my inspiration to overcome all the obstacles in the process of completing of this research work. I also want to thank Prof. Horizon for his financial support during my study that has helped to lighten my financial burden as a part-time student.

I would also like to express my gratitude to my fellow postgraduate coursemates in University of Science, Malaysia, Teoh Yew Heng who has played an important role in promoting a stimulating and welcoming academic and social environment. I really appreciate his assistance in sorting out all the university-related procedures, which was a great help in the beginning of my study.

Last but not least, I wish to express my sincere thanks to all those who have in one way or another helped me in making this study a success.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

DI	Direct Fuel Injection
SOI	Start Of Injection
EOI	End of Injection
BDC	Bottom dead center
TDC	Top dead center
BTDC	Before top dead center
LFL	Lean Flammability Limit
RFL	Rich Flammability Limit
VFR	Volume in Flammability range
PR	Pressure Ratio
PLIF	Planar Laser-Induced Fluorescence

SIMULASI PENCAMPURAN UDARA DAN BAHAN API BAGI PEMANCITAN TERUS DAN PEMANCITAN LIANG PINDAH DALAM ENJIN DUA LEJANG

ABSTRAK

Enjin dua lejang biasanya ditemui pada unit pengangkutan kecil di negara-negara Asia Selatan. Disebabkan oleh kehilangan bahan api yang tinggi semasa proses pemerangkapan, jumlah pencemaran yang dikeluarkan oleh kenderaan-kenderaan dua lejang adalah ketara. Dalam kajian ini, ciri-ciri sistem pancitan terus dengan mengunakan bahan api jenis Gas Petroleum Cecair (GPC) telah disiasat. Satu siri simulasi tiga dimensi dengan teknik Penkomputeran Dinamik Bendalir (PDB) yang mengunakan ANSYS Fluent dijalankan pada pelbagai geometri pancitan, dan nisbah pencampuran udara dan bahan api yang menyebabkan pembakaran dalam silinder telah dinilai. Terdapat dua jenis pendekatan bagi pemasangan pemancit dikaji dalam projek ini. Bagi pendekatan yang pertama, penyuntik diletakkan di kepala enjin untuk pemancitan bahan api jenis gas terus ke dalam kebuk pembakaran semasa omboh hampir menutupi tingkap ekzos. Pelbagai jenis lokasi dan sudut penyuntikan yang berbeza digunakan dalam simulasi ini. Teknik memecah kepulan bahan api melalui pemecah arus juga dikaji untuk memahami penambahbaikan yang boleh dibuat. Pendekatan kedua ialah untuk menempatkan penyuntik di dalam tingkap liang pindah. Muncung penyuntik akan diletakkan di kawasan di mana pemancitan bahan api boleh terus memasuki kebuk pembakaran. Simulasi digunakan untuk mengkaji penyuntikan daripada liang pindah berlainan and orientasi penyuntik terhadap jumlah kebocoran bahan api ke dalam liang ekzos. Beberapa lokasi dan orientasi disimulasikan and keputusan simulasi menunjukkan kombinasi terbaik untuk meletak pemancit ialah ke atas liang pindah tepi dengan orientasi menuju ke kepala enjin. Eksperimen yang dijalankan dengan persediaan enjin berdasarkan panduan simulasi menunjukkan pengurangan sebanyak 80% bagi pelepasan ekzos dan pengurangan sebanyak 35.7% bagi pengunaan bahan api jika berbanding dengan enjin berkarburetor asas.

AIR FUEL MIXING MODELING FOR DIRECT AND TRANSFER PORT INJECTION IN TWO STROKE ENGINE

ABSTRACT

Two-stroke engines are commonly found in small transportation units in many South Asian countries. Due to high fuel losses during the scavenging process, the amount of pollution emitted by these two-stroke vehicles is significant. In this study, the details of a gaseous fuel direct injection system using Liquid Petroleum Gas (LPG) were investigated. A series of three-dimensional, Computational Fluid Dynamic (CFD) simulations in ANSYS Fluent are run on various injection geometries, and the air/fuel mixing of the resulting cylinder charge is evaluated. Two approaches for injector installations are investigated in this study. For the first approach, the injector is placed in the engine's head for direct injection of gaseous fuel into the combustion chamber near the exhaust port closing timing. Many different injector positions and angles are simulated. Technique of splitting fuel plume via flow splitter also being studied to understand the improvement, in which can be done. The second approach is to place the injector in the transfer port window. The injector nozzle is placed in an area where it can directly inject through the transfer port window into the combustion chamber. The simulation is used to understand the injection from different port and the effect of the injector orientation to the amount of fuel leaking into the exhaust port. Several locations and orientations are simulated and the results shows the best combination to install the injector is on the side transfer port with orientation injecting toward the engine head. Experiments had been carried out with the engine setup based on the guideline from simulation and shows 80% decrease in emission and 35.7% decrease in fuel consumption if compare with the baseline carbureted engine.

CHAPTER 1

INTRODUCTION

1.1 Background

Two-stroke engines are commonly used in small transportation units in South Asia. There are over twenty five million two-stroke powered two-wheelers and three-wheelers in South Asia. These vehicles emit substantial quantities of hydrocarbons (HCs), carbon monoxide (CO) and particulate matter (PM), which have significant adverse health effects and cause deterioration in environmental quality [1]. This situation is especially obvious in densely populated areas of South Asia that rely on motorcycles as an essential mode of transportation.

Two-stroke powered two-wheelers are predominantly used by lower wage earners for transportation of goods and for personal transportation purposes. Increasing petrol prices are pushing users to switch to cheaper fuels and more efficient engines. However, buying new vehicles is a significant financial burden for these people. Hence, converting the carbureted two-stroke engine to direct injection engine could be a double-advantage solution where the conversion will improve fuel consumption efficiency and reduce emissions; at the same time, engine conversion incurs lower costs than purchasing a new vehicle.

1.2 Problem Statement

Direct Injection (DI) admits fuel directly into the combustion chamber. A DI system can inject fuel just after the exhaust port closes to avoid fuel leakage through the exhaust port. This feature greatly reduces pollution caused by conventional carbureted two-stroke engines where a large portion of air/fuel mixture is typically short-circuited into the exhaust port during the scavenging process [2][3]. The LPG DI system minimizes modifications of engine system required by removing the fuel pump and air compressor of the gasoline DI system; in replacement, vapor pressure of LPG is used to propel the fuel into the engine.The major problems of the gaseous fuel DI system is the air fuel mixing performance. Computational fluid dynamic (CFD) simulation can help improving the air fuel mixing performance by choosing a correct injector location and orientation.

1.3 Objectives

1) To simulate direct injection gaseous fuel spray in 120cc two stroke engine using CFD software.

2) To investigate the air-fuel mixing pattern of gaseous fuel direct injection and its interaction with combustion chamber wall.

3) To investigate the tumble and swirl motion induction with various injection locations and combustion chamber shapes.

4) To investigate injector location and orientation based on gaseous fuel air-fuel mixing performance to determine the best injector position, orientation and spark plug location.

5) To validate the simulation results on engine emissions and fuel consumption with experiment via urban driving method.

1.4 Thesis Layout

Chapter 1 is an introduction, which covers the background of the project, scope and objectives and also research plan.

Chapter 2 is literature review of the two-stroke direct injection system and the study of LPG as an alternative fuel. The disadvantages of the existing system with the designed LPG direct injection system is being compared.

Chapter 3 discusses the simulation technique used in the two-stroke engine simulation and the technique used in this study to measure the fuel-air mixing performance. This chapter introduces the setup of the first part of the research, which pertains to direct injector placed in the two-stroke engine cylinder head. This includes the two-stroke engine simulation setup with the design of the engine head, injector placement at different areas and flow splitter in front of the injector. The second part of the research is focused on simulation of direct injector placed on the two-stroke engine transfer port. This includes the full engine simulation with scavenging process. The simulation setup includes transfer port, exhaust port, crankcase volume, combustion chamber and injector.

In Chapter 4, the analysis on the air-fuel mixing performance in different scenarios is being studied. An experiment is carried out to validate the design guideline from the simulation against engine emission and fuel consumption. Discussion on in-cylinder flow pattern is also included in this chapter.

Chapter 5 is the summary and conclusion of the whole research. The detailed drawings of the simulation setups, as well as the in-depth information of some topics, are appended in the Appendix.

CHAPTER 2

LITERATURE REVIEW

2.1 Two-Stroke Engine

Two-stroke engines can be found in many applications of lightweight equipment such as chainsaws, outboards, lawnmowers and motorcycles. The reason for having two-stroke engine in these applications is because of its simple design, resulting in low costs and high power to weight ratio. There are plenty of different two-stroke engine designs. The main principles remain the same, but the mechanical details of various two-stroke engines differ depending on the respective designs. The design types vary according to the method of introducing the charge to the engine, the method of exchanging burnt exhaust for fresh mixture and the method of exhausting the cylinder.

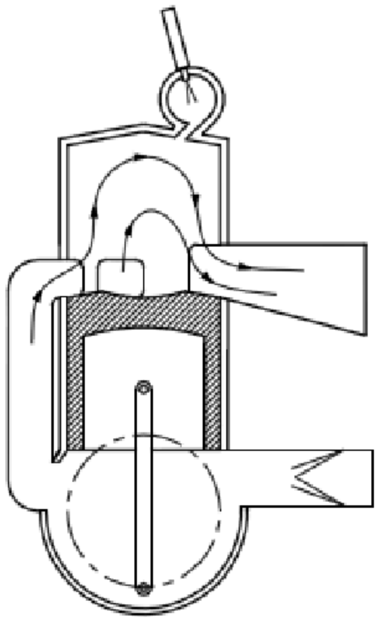

Figure 2-1 - Loop Scavenging Engine [4]

In this project, Loop-scavenged engine (Figure 2-1) is being studied. This method of scavenging uses carefully shaped and positioned transfer ports to direct the flow of fresh mixture towards the combustion chamber as it enters the cylinder. The fuel/air mixture strikes the cylinder head, then follows the curvature of the combustion chamber, and is finally deflected downwards.

The above-mentioned sequence not only prevents the fuel/air mixture from traveling directly out of the exhaust port, but also creates a swirling turbulence which improves combustion efficiency and power generation. Loop scavenging is the most common type of fuel/air mixture transfer used in modern two-stroke engines.

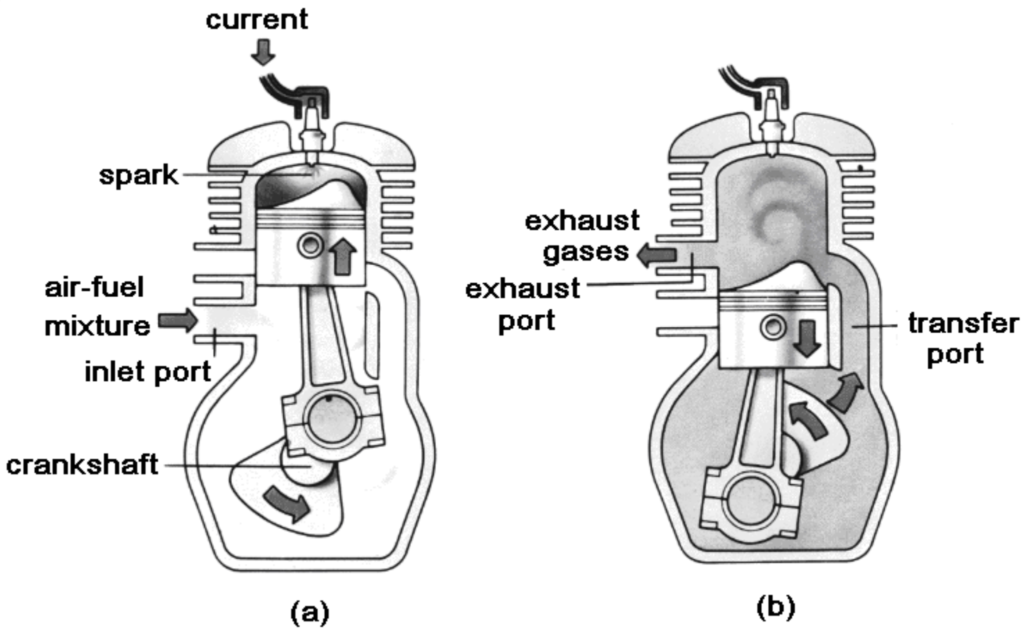

Figure 2-2 – Two-stroke engine cycle.

Two-stroke engines fire once every revolution, and they do not have valves mechanism, which is present in four- stroke engines. Each revolution consists of combustion stroke and compression stroke as shown in Figure 2-2 . For the Combustion Stroke, we supply current to spark plug and it fires as depicted in Figure 2-2(a). The fully compressed air-fuel mixture is ignited and the resulting explosion will drive the piston downwards. When the piston moves downwards, it starts to compress the air-fuel mixture inside the crankcase. As the piston approaches the bottom of its stroke, the exhaust port is opened. The high pressure generated by burned gas in the cylinder drives most of the exhaust gases out of cylinder. With the piston further moving down, the intake port is uncovered. The pressurized air-fuel mixture in the crankcase rushes into the cylinder and displaces the remaining exhaust gases, as shown in Figure 2-2 (b). As a result, the cylinder is filled with a fresh charge of air-fuel mixture, ready for the next cycle of combustion.

For the Compression Stroke, the momentum in the crankshaft starts driving the piston upwards. As the air/fuel mixture in the piston is being compressed in the combustion chamber, a vacuum is created in the crankcase. This vacuum creates a suction force that forces the reed valve to open and suck air/fuel/oil from the carburetor into the crankcase chamber. As soon as the piston makes it to the end of the compression stroke, the spark plug fires again to repeat the cycle.

Two-stroke engines have advantages in terms of power to weight ratio compared with four-stroke engines. As two-stroke engines do not have extra valves or camshaft mechanism, they have fewer numbers of components, which lower the total weight and make them easier to construct compared with four-stroke engines.

However, the lifespan of two-stroke engines is shorter than that of four-stroke engines. This can be attributed to the insufficiency of lubrication system in a two-stroke engine, which makes the two-stroke engine parts wear out a more rapidly. The engine requires lubricant to be added into fuel to keep the piston lubricated while running. In addition, the two-stroke engine design causes some of the fuel to leak out of the chamber through the exhaust port during the gas exchange process. This is the reason why two-stroke engines are mainly used in vehicles or equipment where the weight of the engine is small and is not used continuously for long periods.

2.2 Two-stroke Engine Direct Injection System

Direct Injection (DI) is a system for admitting fuel directly into internal combustion engine. Today, it is the primary fuel delivery system in automotive engines. With the introduction of direct injection technology in two-stroke engines, the waste and pollution caused by conventional carbureted two-stroke engines can be greatly reduced. The primary purpose of direct injection in two-stroke engines is to control the fuel emission timing to avoid the fuel/air mixture flowing out directly through the exhaust port.

There are two direct injection systems commonly used in two-stroke engines, namely the low-pressure air-assisted gasoline injection and high-pressure gasoline injection. Both systems require pressurized fuel, and the air-assisted DI system requires an air pump. EnviroFit developed an air-assisted direct injection retrofit kit for two-stroke engines [5]. The retrofit kit includes a fuel injector and fuel pump as well as an air pump and air injector (Figure 2-3). It also involves extensive modification of the engine to deliver power to the air compressor.

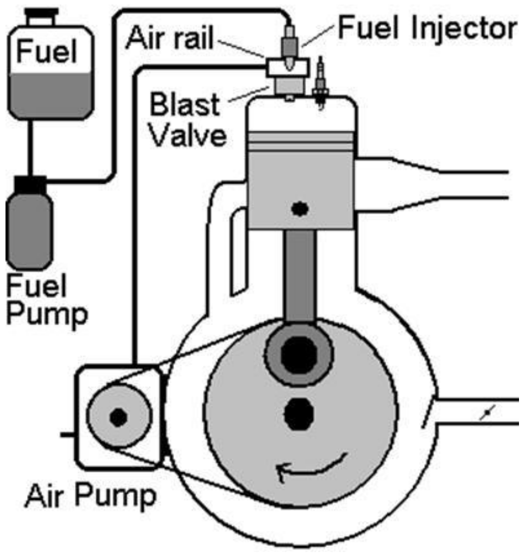

Figure 2-3 - Gasoline DI system setup [6].

2.3 LPG and CNG Fueled Engine

Liquefied petroleum gas (LPG) also known as autogas in some markets, mainly contains propane (C_3H_8) and butane (C_4H_{10}). Compressed natural gas (CNG) is made by compressing natural gas (which is mainly composed of methane, CH_4).Both propane and butane are easily liquefied and are stored in 2-7 bar pressure containers; this enables LPG to be safely transported in cylinders or tanks to end users. CNG require compression to transform it into liquid and stored and distributed in hard containers at a pressure of 200–248 bar pressure.

The normal boiling point of LPG varies from 229 to 273 K [7] and is stored under pressure of 2bar to 7bar below critical temperatures in order to keep it in a liquid state. The flammable fuel-air ratio for pure propane is between 0.021 (LFL - Lean Flammability Limit) and 0.095 (RFL - Rich Flammability Limit) [8]. LPG is widely used in homes as well as in industrial and agricultural sectors. It burns more thoroughly with less carbon build-up and oil contamination [26]. This reduces the wear of LPG-driven engines and increases the lifespan of some components such as rings and bearings compared with that of gasoline-driven engines. The high octane content in LPG also helps minimize the wear from engine knock. Rapid development of LPG technology in vehicle fuel conversion is an indication that LPG may soon be recognized as a premium automotive fuel [9] [10].

LPG is used as a dry gas and contains no fuel additives. It also has higher ignition temperatures than gasoline; this property of LPG increases the importance of maintaining proper control of ignition timing. Poorly designed ignition systems will result in improper combustion and sluggish vehicle performance. The engines with ECU and electronic ignition system can be dealt with by using a spark advance processor (STAP/TAP) to correct the spark advancement signal when using natural gas [11].

LPG and CNG engines deliver almost similar performance as petrol engines and have superior combustion characteristics compared with those of gasoline. The usage of LPG and CNG as an alternative fuel could replace 10% of the current usage of oil and at the same time significantly reduce emissions of CO, CO_2 and harmful greenhouse gases [9]. In the future, LPG and CNG will be more widely available and will potentially gain large market shares among a wide range of vehicles.

Pradeep and Varuna [12] had done the study on LPG transfer port injection in order to reduce engine emission. Experiments were done at 25% and 70% throttle openings with different injection timings and optimal spark timing at 3000 rpm. The experiment had proven that the port injection gaseous fuel engines are feasible with the benefit of emission reduction and satisfactory performance.

Yu Liu [7] and Marek [13] carries out study on CNG fuel DI performance by investigating their flow pattern when it impinges the combustion chamber wall and the flame spreading during combustion. Their study found out that gaseous fuel DI penetration was significantly affected by injection pressure. Due to no evaporation process compare to liquid fuel, the effect of the ambient temperature toward the injector spray penetration and spray angle are not significant as well.

Yew Heng [6] also did the development of the LPG Direct Injection System for Small Two-Stroke Transports. The development of the engine found out the emissions and fuel consumption benefits of DI of LPG can be realized with injection timings shortly before exhaust port closed. The paper suggest that there are strong relationship between injection timing and engine performance due to injector latency and fuel propagation delay

2.4 Two-stroke Engine Simulation

The development of new two-stroke engine model in the early stage is very costly and time-consuming due to the large number of options for improvements. Simulations on engine enable trial-and-error approach to be used, involving minimal time and no material cost. It is also worth noting that visualization in the two-stroke engine is difficult due to the air flow interaction during the scavenging process.

Scavenging process is a complex gas exchange process which involves burned gas being expelled through the exhaust port and replaced by fresh charge from the transfer port. To conduct a 3D CFD simulation on the scavenging process, some simulation techniques are required. A complete two-stroke engine model with moving pistons that open/close the transfer port is needed to allow details of the scavenging process to be visible. In fact, there are quite a few two-stroke engine simulation models that have been developed and proven to match the real engine simulation results.

In the research by María Isabel [14], mass fraction is being used to differentiate fresh air and burned gas to allow it to observe the gas exchange process in two-stroke engine. The technique is useful in detecting problems of short circuiting and gas drag. This technique is similar with the species transport method being used in the thesis.

Various type of scavenging system in two-stroke engine had been investigated by Enrico Mattarelli [4] using CFD simulation. The research found out the loop scavenging engine has bad air-fuel mixing performance compared with uniflow scavenging engine and 4-stroke engine.

Yangbing [17] had done a study on air fuel mixing performance using CFD software, KIVA. The simulation model is concentrate correlated between simulations and experiments. The correlation was made for sprays in quiescent ambient conditions while a good agreement of the spray characteristics was obtained. The Yangbing research concluded longer mixing time does not necessarily warrant better air-fuel mixing and the tumble flow generated by scavenging will brings fuel around to mix with air.

Schmidt [18] develops a model for two-stroke DI engine to optimize the air-fuel mixture preparation. The research experiment shows a clear improvement of the volumetric efficiency by changing the port configuration and port geometry. The experimental investigations also showed good accordance with the results obtained from the CFD simulation.

Hence, it is feasible to use CFD simulation to make a virtual comparison of different geometries in regard to their efficiency and exhaust characteristics.

CFD research on direct injection of gaseous fuel is also one of the popular research areas. Gaseous fuel does not penetrate the combustion chamber as well as liquid fuel, and the air-fuel mixing process is slow and complex. Research on injected gaseous fuel plume has been carried out to further understand how the gaseous fuel plume develops after injection [16]. The results of the study of gaseous fuel direct injection engine provide evidence on the existence of a relationship between combustion chamber shape and mixing performance [13].

2.5 Injection Flow Pattern Correlation

The flow structure and turbulent mixing of the natural gas had been studied by using Acetone-based Planar Laser-Induced Fluorescence (PLIF) technique by Aalto University, Finland [19]. The injection parameter of the study is similar to the parameter being used in the simulation. Table 2-1 shows the setup comparison between the PLIF visualization with the simulation parameter in this project.

Table 2-1 - Direct Injection setup.

	PLIF Visualization	Simulation
Injector	Bosch NG12	Synerject STRATA 1
Injector nozzle exit diameter	5.6mm	6.3mm
Injection Pressure	7 bar	6.5 bar
Chamber Pressure	1 bar	1 bar
Pressure Ratio (PR)	7	6.5
Injection Fuel	Nitrogen with Acetone	Propane

Due to the availability of the injector, Synerject STRATA 1 injector is selected to run both simulation and experiment. The differences on injector nozzle exit diameter are expected to have minimal impact to the injection pattern.

The comparison between the direct injection flows patterns are shown in Figure 2-4. The penetration of acetone based nitrogen is almost aligning with propane injection in the simulation. Since the injection penetration are mainly dependent on to pressure ratio (PR) between the fuel and chamber [20]. The simulation be can concluded that the penetration between the visualization and simulation have correlated penetration depth.

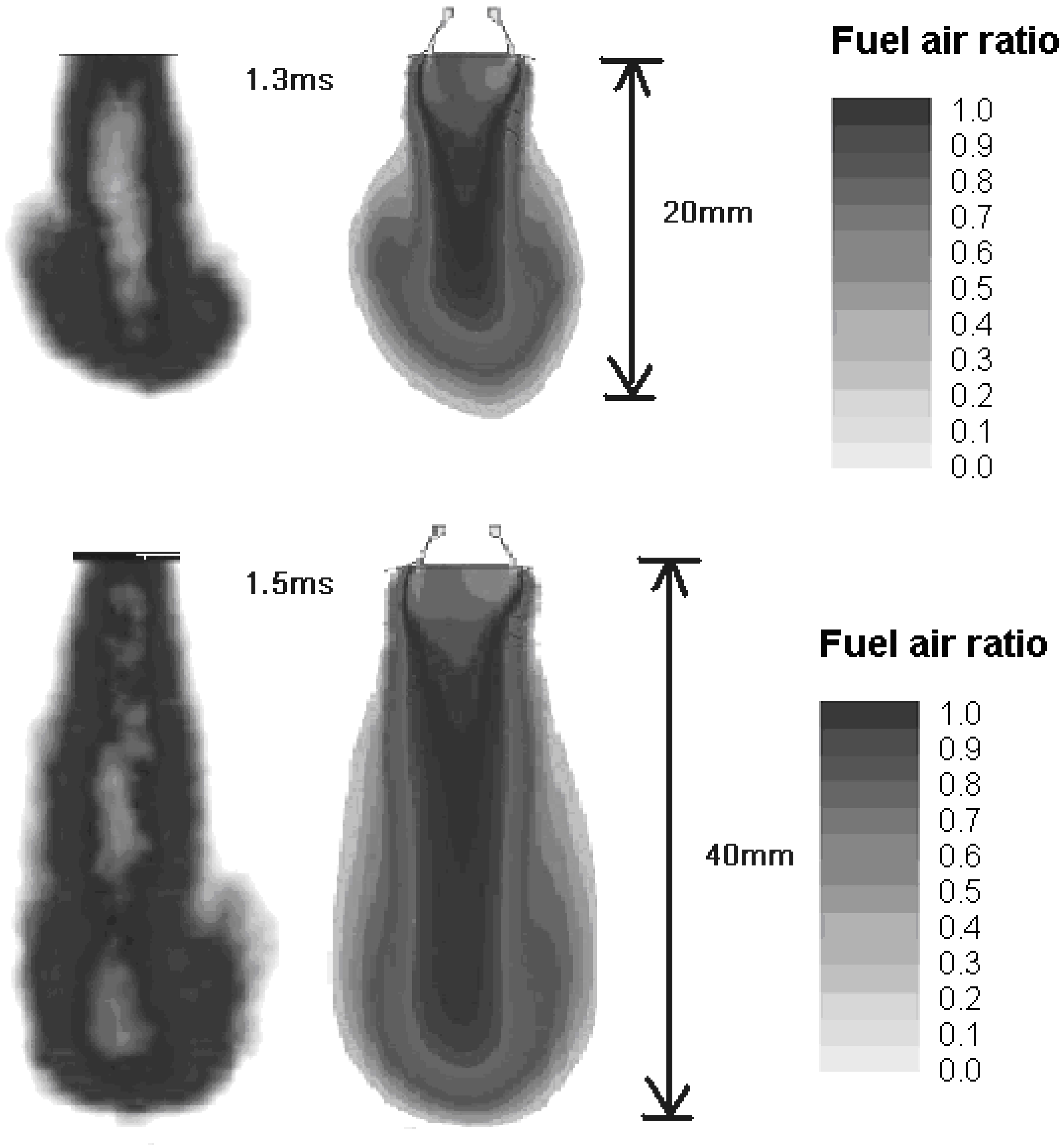

Figure 2-4 - Penetration comparison between PLIF visualization with Nitrogen with Acetone (left) [19] and simulation with pure propane (right).

Besides the penetration depth, the interaction between the gaseous fuel and combustion chamber wall is also one of areas of study. When the fuel plume impinges the combustion chamber wall, the velocity and momentum of the plume are decreased. The accumulated fuel is forced to spread out in disc shape and wall-vortex cores are formed. The same reactions happen to both PLIF visualization and simulation as in Figure 2-5.

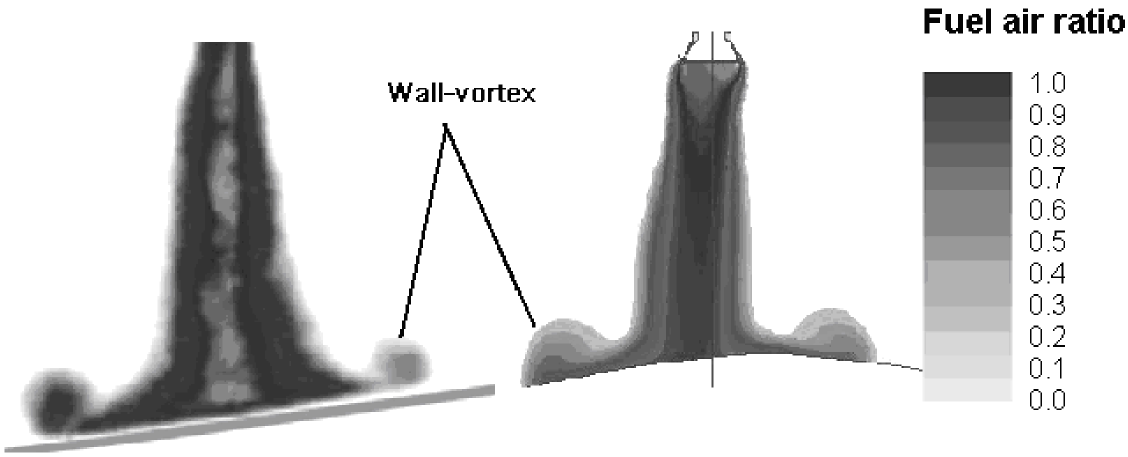

Figure 2-5 - Comparison between DI PLIF visualization with Nitrogen with Acetone (left)[19] and simulation with pure propane (right).

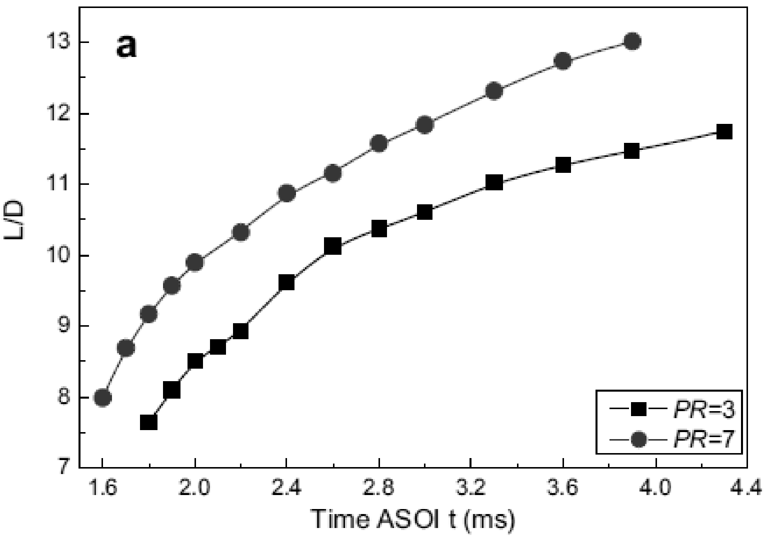

Figure 2-6 - Time evolution of the normalize tip penetration, L/D [19].

Based on the study by Aalto University, the relationship between the fuel penetration and fuel plume disk development against time is being compared between 3 PR and 7 PR injection as in Figure 2-6. The measurement is done in dimensionless unit; L/D where L is the sum of injection penetration and diameter of the plume disk formed after impinges wall and D is the exit diameter of the injector. The 3 PR and 7 PR differences create around 7mm increment of fuel plume disk diameter. By extrapolate this data into our simulation data, the differences between 7 PR and 6.5 PR is around 0.88mm reduction of fuel plume disk diameter. The fuel plume disk diameter differences are small enough to be neglected due to tolerances in picture measurement. Based on this, the simulation will be able to use the 7 PR injection visualization to correlate with 6.5 PR injection simulations.

CHAPTER 3

METHODOLOGY

3.1 Direct Injection of LPG from Engine Head

One of the major problems of direct injection of liquid fuel is fuel pressure. To address this problem, liquid fuel had been replaced with a gaseous fuel, LPG, which has 2-8 bar gas pressure. Replacing the liquid fuel with gaseous fuel simplifies the DI retrofit kit as fuel pump is no longer required. Figure 3-1 shows the LPG DI system components consisting of an LPG tank, a fuel pressure regulator and an injector.

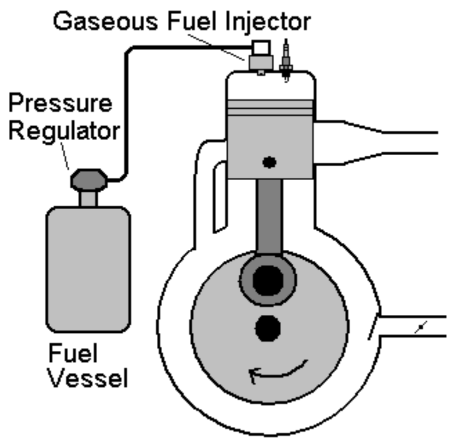

Figure 3-1 - LPG DI system setup [6].

Air fuel mixing is a challenge for direct injection of gaseous fuel. The fuel jet has limited penetration into the combustion chamber due to the relatively low density of the gaseous fuel, resulting in slow fuel air mixing [13]. This situation makes it difficult for a gaseous fuel direct injection system to create a homogenous mixture. Injector position and combustion chamber geometry have significant influence on the mixing process. This project presents numerical analysis of the LPG direct injection system, where the influence of injector position and engine geometry on mixture homogeneity is investigated.

3.2 Direct Injection of LPG from Transfer Port

Direct Injection on engine head requires an injector which can sustain high pressures and temperatures of combustion [6]. In an effort to reduce the cost of such system, the injector is placed in the transfer port (Figure 3-2). The transfer port is exposed to lower pressure and temperature during engine operation. This will put less stress on the injector and give greater flexibility in terms of injection timing. Therefore, a simple low-cost injector can be use to serve this purpose. Placing the injector at the transfer port requires minor modification to the transfer port only. This allows a cheap and simple retrofit kit to be designed with minor modification.

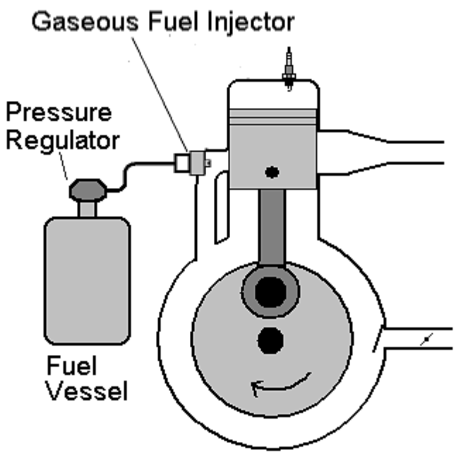

Figure 3-2 - LPG Transfer Port Injection two-stroke engine [6].

3.3 Simulation Technique

3.3.1 Engine Dimension

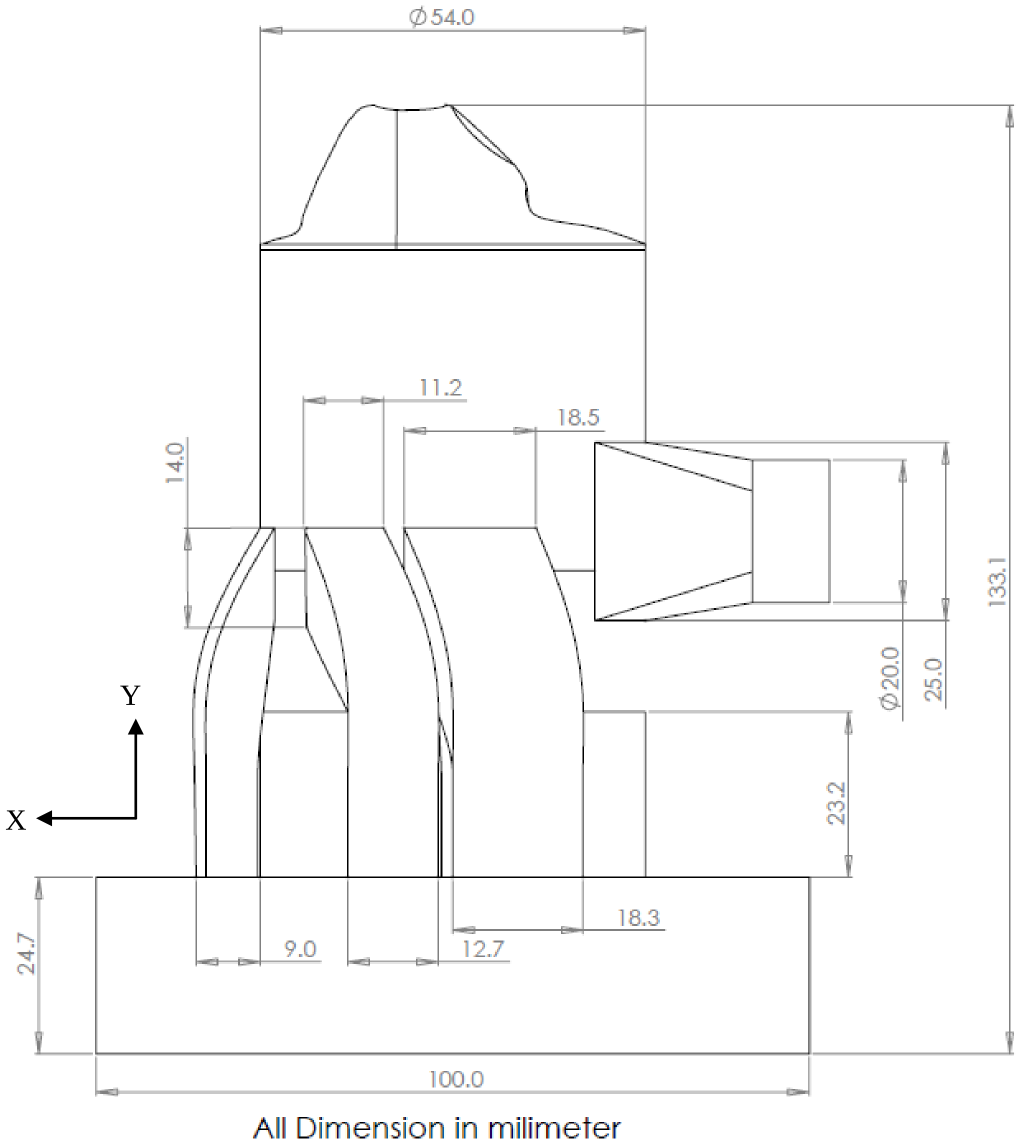

Figure 3-3 - Engine dimension in Right view.

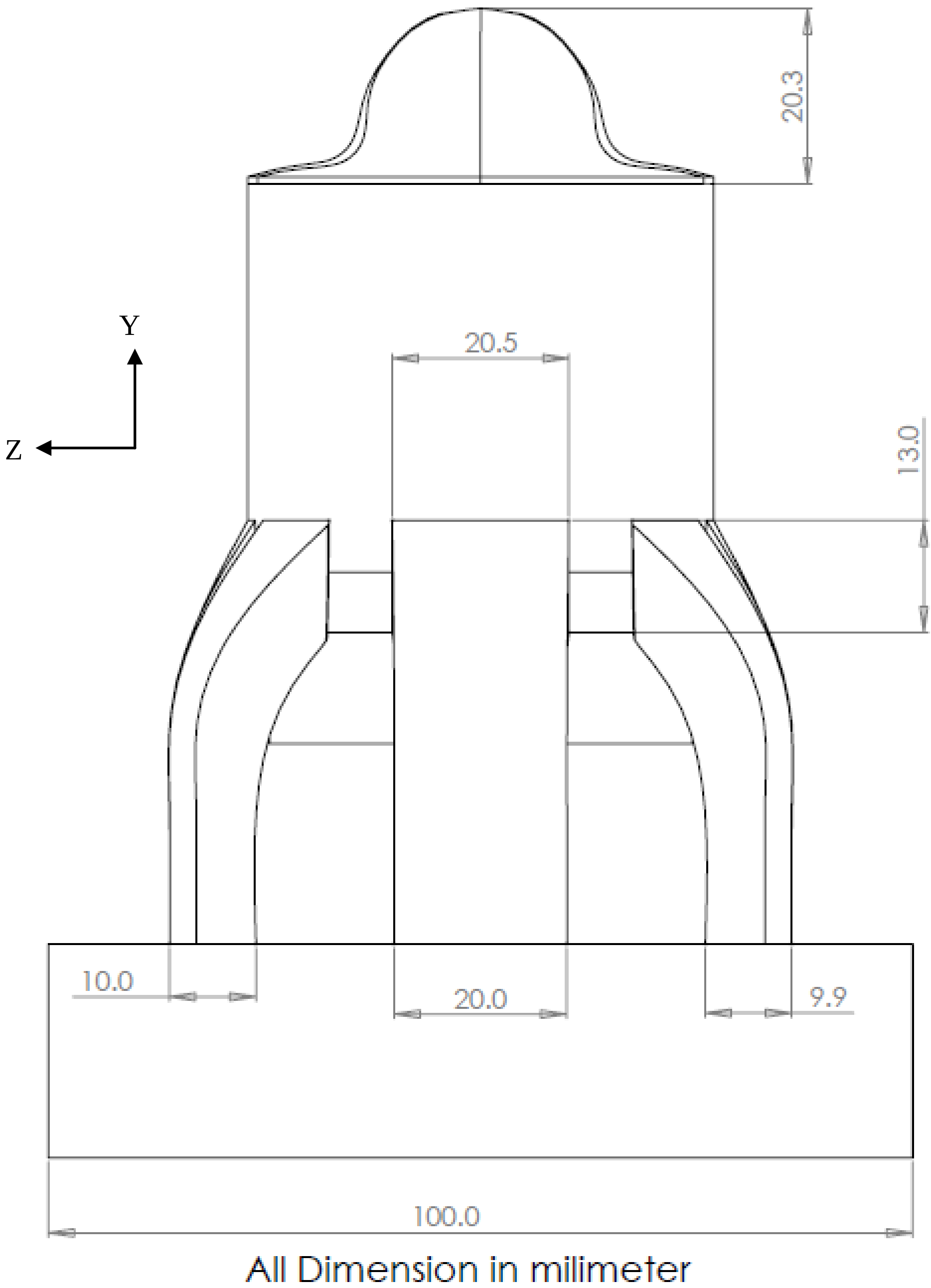

Figure 3-4 - Engine dimension in Front view.

3.3.2 Physical model and Meshed model.

SOLIDWORKS is used to model the physical model of the engine. Figure 3-5 shows the model created in SOLIDWORKS, which contains only the combustion chamber volume. SOLIDWORKS is able to extract the internal surface of the engine part model and stitch it together to form a solid object. The solid geometry has to be exported by neutral files in parasolid (.x_t) format to allow it to be imported into GAMBIT.

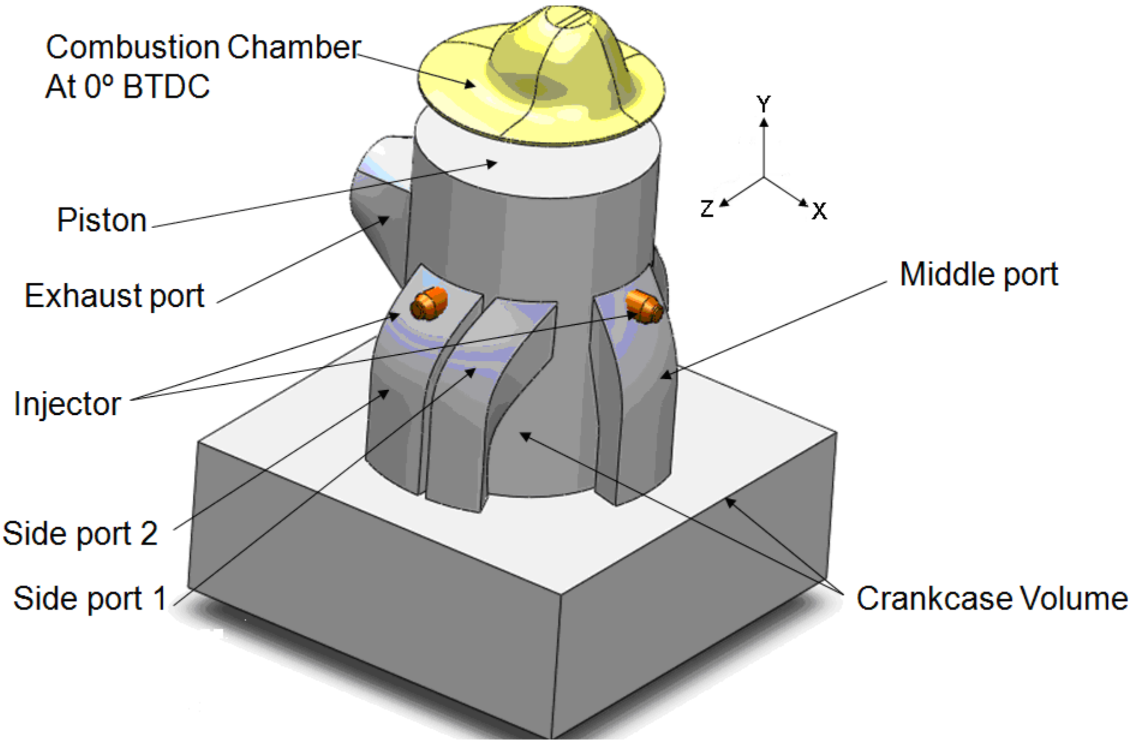

Figure 3-5 - Physical Model build by SOLIDWORKS.

GAMBIT is preprocessing software for CFD simulation. Complex models can be imported from any major CAD system. Using a virtual geometry overlay and advanced cleanup tools, imported geometries are quickly converted into suitable flow domains. A comprehensive set of highly-automated and size function driven meshing tools ensures that the best mesh can be generated. Table 3-1 shows different size function according to the flow complexity on their region.

Table 3-1 - Size Function and Mesh Type on Different Zone

Zone	Size Function (Interval Size)	Mesh Type
Combustion Chamber	0.5	Tet/Hybrid - TGrid
Exhaust port	1.0	Hex - Map
Side Port 1 & Side Port 2	1.0	Tet/Hybrid - TGrid
Middle Port	1.0	Tet/Hybrid - TGrid
Injector	0.2	Tet/Hybrid - TGrid
Crankcase Volume	5.0	Hex - Map

GAMBIT is capable of building structured, multiblock, unstructured or hybrid mesh to fit with different solid models. Figure 3-6 shows the model imported from SOLIDWORKS and meshed in GAMBIT.

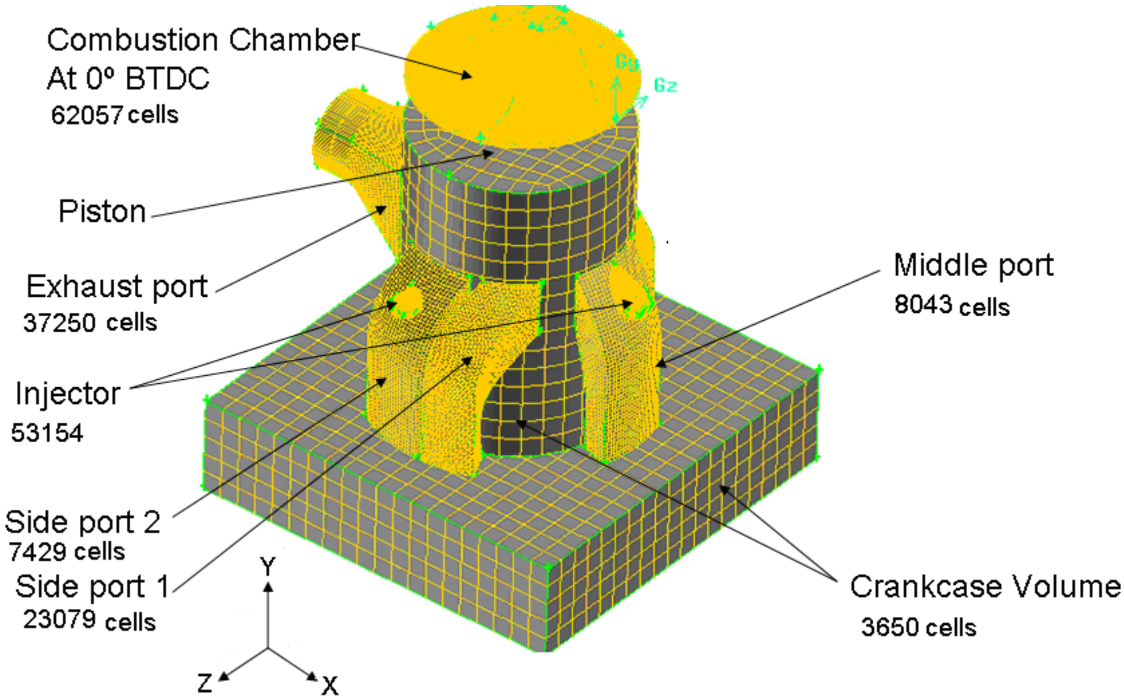

Figure 3-6 – Meshed model by GAMBIT.

Although the geometry can be imported from SOLIDWORKS, some simple geometry is still better when created by GAMBIT itself. GAMBIT can build basic geometry such as cylinders and cubes easily. The geometry created by GAMBIT has more advantages in regard to the meshing option and mesh quality. Because of that, most of the basic geometry such as engine cylinder, crankcase volume and exhaust port has been recreated in GAMBIT (Figure 3-6) to allow higher quality mesh in these areas.

Due to the complex shape of the combustion chamber, Tet/Hybrid meshing is used. The Tet/Hybrid method will automatically mesh the entire volume with unstructured three-dimensional grid according to the geometry of the model. The Tet/Hybrid mesh is composed primarily of tetrahedral elements but may include hexahedral, pyramidal and wedge elements where appropriate. The size function is selected to create uniform cell volume mesh. This will allow us to measure the air-fuel mixing performance which will be discussed in Section 3.4.

3.3.3 Engine Motion - Piston Position

FLUENT provides a built-in function called "In-Cylinder" to calculate the piston location as a function of crank angle. The simulation only needs to set up the parameter of Piston Stroke and Connecting Rod Length. The piston position is calculated based on the equation shown below:

$$p_s = L + \frac{A}{2}(1 - \cos(\theta_c)) - \sqrt{L^2 - \frac{A^2}{4}\sin^2(\theta_c)} \qquad (3\text{-}1)$$

Where p_s is the piston location, L is the connecting rod length, A is the piston stroke, and θ_c is the current crank angle. The current crank angle is calculated from

$$\theta_c = \theta_s + t\Omega_{shaft} \qquad (3\text{-}2)$$

where θ_s is the Starting Crank Angle and Ω_{shaft} is the Crank Shaft Speed.

Table 3-2 shows the engine parameters that are input into FLUENT for piston motion calculation.

Table 3-2 - Engine Parameters

Engine parameter	Value
Crank Shaft Speed - Ω_{shaft}	3000 rpm
Piston Stroke - A	51.8 mm
Connecting Rod Length - L	77.7 mm
Time for each Crank Angle	0.03 ms

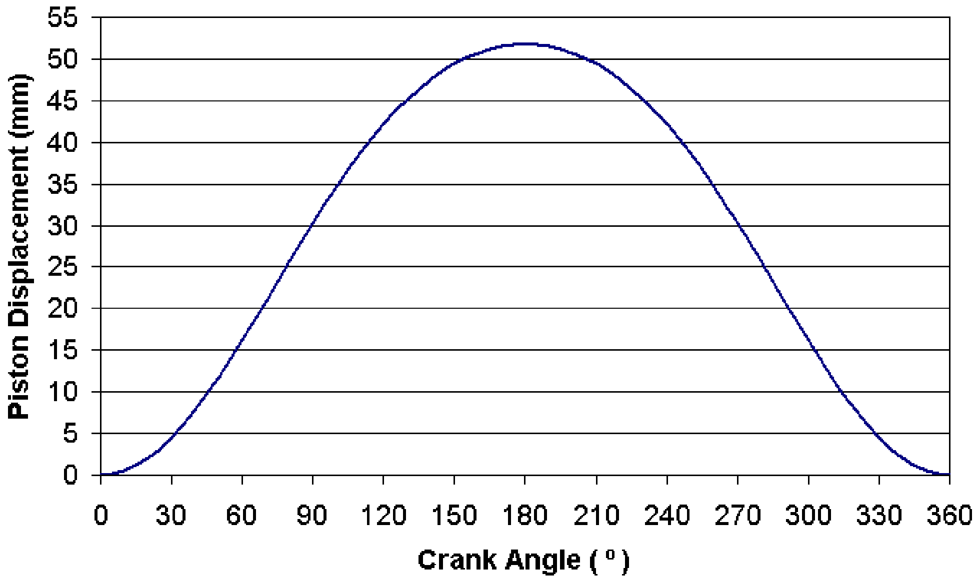

Figure 3-7 - Piston Position as a function of Crank Angle

Piston motion is not simple harmonic motion because the connecting rod adds an extra motion to the simple harmonic motion of the crank; this is why the position equation 3-1 has the square root component added to the cosine. The position curve is not symmetrical as well; it is fatter on the bottom than it is on the top as shown in the Figure 3-7.