STUDY OF SPRAY BREAKUP AND MIXTURE IN A GASOLINE DIRECT INJECTION ENGINE BY USING SIMULATION

MUZAMMIL BIN MOHAMAD ALIAS

Report submitted in partial fulfillment of the requirements
For the award of
Bachelor of Mechanical Engineering with Automotive Engineering

FACULTY OF MECHANICAL ENGINEERING
UNIVERSITY MALAYSIA PAHANG

JUNE 2013

ABSTRACT

The project is to study the spray breakup and mixture in Gasoline Direct Injection (GDI) The spray breakup and fuel air mixture in the injector system really important to improve the fuel efficiency of Gasoline Direct Injection (GDI) Engine. Engine by using simulation. By using the ANSYS Design Modeler, the design of the injector with different inlet size and combustion chamber has been done. Then, by using Computational Fluid Dynamic (CFD), ANSYS Fluent the flow simulation has been run. The results extracted from the simulation are spray cone angle and penetration length. The simulation is done based on different size of nozzle which are 0.2, 0.3, 0.4 and 0.5 mm. While for another variable is injection pressure which are 3, 6, 10, 15 and 20 Mpa. From the result, the spray cone angle is decreasing as the pressure increase which means the spray cone angle is inversely proportional to the injection pressure. While for another results, the penetration length is directly proportional to the injection pressure. The penetration length is increase as the injection pressure is increase. But, as the nozzle diameter increase with the same pressure, the penetration length is decreases.

ABSTRAK

Pemecahan semburan dan campuran udara dan minyak di dalam sistem suntikan adalah penting untuk meningkatkan kadar efesiensi minyak enjin gasoline suntikan terus. Projek ini dijalankan adalah untuk mengkaji pemecahan semburan dan campuran udara dan minyak dalam enjin gasoline suntikan terus dengan menggunakan simulasi. Dengan menggunakan perisian 'ANSYS Design Modeler', reka bentuk penyuntik dengan berlainan saiz saluran masuk dan ruangan pembakaran telah dilukis. Kemudiannya, dengan menggunakan perisian Computational Fluid Dynamic (CFD), ANSYS Fluent simulasi arus akan dijalankan. Maklumat yang akan diambil setelah simulasi dijalankan adalah sudut kon semburan dan panjang semburan.Simulasi dijalankan berdasarkan saiz corong berlainan yaitu 02., 0.3, 0.4 dan 0.5 mm. Pembolehubah seterusnya adalah tekanan suntikan dengan 3, 6, 10, 15 dan 20 Mpa tekanan digunakan. Keputusan yang diperolehi daripada simulasi ini menunjukkan sudut semburan semakin mengecil sekiranya tekanan penyuntik semakin meningkat.manakala bagi lagi satu keputusan menunjukkan panjang semburan bergerak balas secara lansung dengan tekanan penyuntik. Sekiranya tekanan punyuntik ditingkatkan nescaya panjang semburan juga akan meningkat. Namun, apabila tekanan semburan dikekalkan, keputusan menunjukkan semakin membesar saiz diameter corong penyembur, semakin pendek jarak semburan.

TABLE OF CONTENTS

CHAPTER 4 RESULT AND DISCUSSION

CHAPTER 5 CONCLUSION

APPENDIX

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

ICE	Internal Combustion Engine
GDI	Gasoline Direct Injection
PFI	Port-fuel injection
ECU	Engine Control Unit
3D	Three dimension
2D	Two dimension
SI	Spark Ignition
DISI	Direct Injection Spark Injection

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Internal combustion engine is an engine that widely use in automotive industry. The ICE transforms the chemical energy which is the fuel into the useful mechanical energy. These engine works almost in all automotive vehicles like car, motorcycle and many more. Moreover, for the high power to weight ratios required transportation kind of engine appear in the form gas turbine.

There are many type Internal Combustion Engines produce until today and each of them have different kind of design. Each of them has their own strength and also the weakness. As there have different kind of engine there must also have different kind of fuel used to generate it. There are many kind of fuel that such as diesel, gasoline, alcohol, natural gas and many more.

For most of the ICE will produce air pollution emissions. The chemical equation for the complete combustion of fuel is $CH_4 + 2\ O_2 \rightarrow CO_2 + 2\ H_2O$ + energy. But, for the some reason the complete combustion is almost impossible to achieve. The incomplete combustion will produce the dangerous products such as nitrogen oxides and sulfur. By increasing the amount of the air intake during the combustion will reduce the emissions from the incomplete combustion.

Gasoline Direct Injection (GDI) is one of the Internal Combustion Engine examples which use the fuel injection system to two and four stroke engines. The advantages using this engine are increase the fuel efficiency and high power output. The emissions level from the fuel combustion also can be controlled by the GDI system.

1.2 PROBLEM STATEMENT

Automotive industry nowadays realize that the decreasing of the fossil fuels will continually and someday if there are no precaution step taken there could be even worse. Gasoline, diesel and others will remain decreasing while the price will remain increasing. Furthermore, the environmental issues had been serious lately while the government and the NGO keep on campaigning about the important of the environment. This make the automotive manufacturer and automotive company try to find the alternative and better way of fueling. One of the main challenges for it is to make sure an ignitable air fuel mixture at the spark plug for all kind of engine operating points for the usage of different worldwide fuels (Sauer et al., 2008). Then, nowadays most of the automotive engineers and company tend to move on and research more about the GDI engine instead of other engine because of the better fuel consumption (Saito et al., 2011). For the gasoline engine itself the manufacturer try to minimize the pollution and increase the fuel consumption. And one of the solutions is to study about the spray breakup and the mixture.

1.3 PROJECT OBJECTIVES

The main objectives for the project are:

1. To study the spray breakup for different parameter and geometry of the nozzle injector with different pressure.
2. To develop 2- dimensional simulation model for Gasoline Direct Injection engine.

1.4 SCOPE OF THE PROJECT

The scope of the project is to study the different type of injector for better air fuel mixing in the Gasoline Direct Injection Engine. To conduct this analysis or simulation, the software that can be used are ANSYS Fluent and Solidwork. First, the design of the injector will be design by using Solidwork. The design then will be import into the ANSYS Fluent for the spray breakup simulation. For the validity, the spray breakup simulation results that had been done then will be compared to the previous research.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Jafarmadar and Heidarpoor, (2011) stated that for the past 20 years, the spark ignition engines especially fuel system had been developed fluently from carburetion to throttle body injection, then to simultaneous fire port fuel injection (PFI) and lately phased sequential fire PFI Then, nowadays many automotive engineers tend to move on and research more about the GDI engine instead of gasoline PFI engine because of the better fuel consumption (Saito et al., 2011). Leipertz et al. (2010) stated that the mixture formation appeared for modern engines are different compared to the gasoline since the application of modern regenerative and synthetic fuels. It changes basically because the evaporation behavior is affected by the boiling points of the fuels.

Furthermore for the SI engine, one of the most important processes is the spray and mixture formation process. It is because, it will affect the whole combustion process for the engine performance since the fuel will atomized and blend with the air to provide the combustible mixture before the combustion take part(Celik et al., 2010). The Direct Injection spark Ignition (DISI) engine categorized into three types as shown in figure 2.1 according to their system which are Spray Guided, Wall Guided and Flow Guided.

From the entire GDI engine concept, the spray guided technologies with charge stratification will result the most possibility to reach the highest fuel efficiency. Homogeneous mixture in the cylinder will happen when early injection at the inlet stroke for full load operating points during the ignition time point (Park et al., 2009).

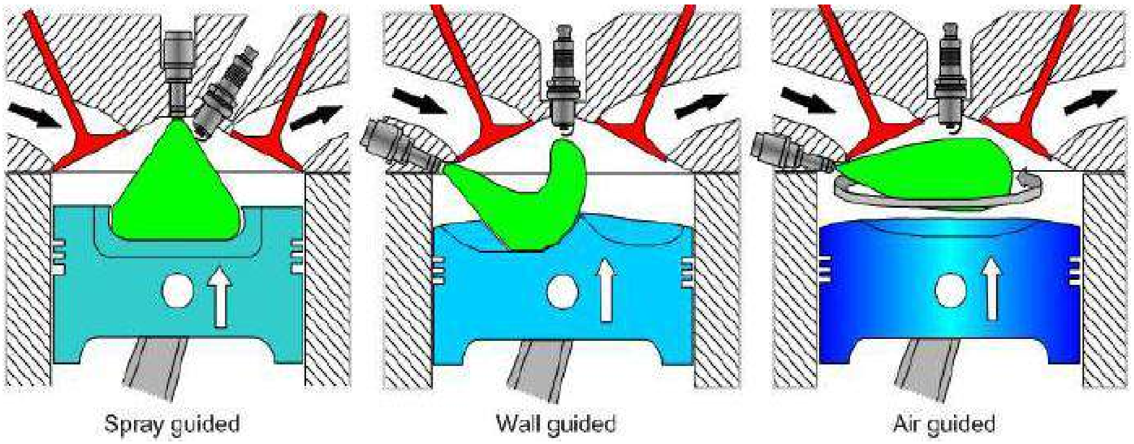


Figure 2.1: Type of injection

Zigan et al. (2010) investigated that for the partial load conditions, to increase the fuel efficiency the charge stratification is more preferable. For the late injection operating points, the pressurized fuel will be injected close to the spark plug where then will be mix with the excess air to form the mixture. In case to ensure a stable ignition of the air fuel mixture, it requires a very fast and reproducible fuel atomization and evaporation.

Quick fuel atomization and evaporation is need for SI engine with a reproducible placement of fuel vapor close to the spark plug to make sure a steady ignition of the air fuel mixture. For the GDI engine, in order to avoid problems related to the fuel film, the fuel is injected into the cylinder directly. Furthermore, GDI engine need less fuel to start compare to the PFI engine. As the result it will reduce the HC emission and also other harm gasses in the standard operating conditions (Abani et al.2007)

Montanaro et al., (2011) stated that in theory the GDI engine, offers a lot of opportunities for achieving important improvements in fuel consumption and emissions reductions because it does not have any limitations. Moreover, the evaporating process of the engine offer the air cooled by the fuel droplets hence allow the compression ratio higher and lower the octane requirement for the fuels. In addition, the volumetric efficiency also can be improvised if the injection occurs during the induction event.

In the GDI engine, the higher injection pressure and the shorter time for the needle open is much needed in order to get the fuel spray atomization magnitude finer (Rotondi and Bella, 2006). For the operating modes, there have three which is stratified for the lean mixture, homogeneous for the lean mixture and homogeneous for stoichiometric mixtures. The engine is operated with the stratified, homogeneous lean and homogeneous stoichiometric modes; at low load and speed, at medium load and speed and at high load and speed.

Som et al., (2008) stated that the design of the nozzle is the most importance factor in a injection system since it resulting the turbulence flow that effect the spray atomization. Instead of nozzle, the fuel property also is another criterion that should be considering in this system. The fuel properties such as boiling point and volatility will affect the droplet size and evaporation process inside the chamber. As example, the fuel with high volatility will evaporate faster while the fuel with high boiling points will remain in the droplet.

In the modern era, with the new type of synthetic fuels and regenerative fuels, the study about the nozzle parameter and spray characteristic should be more advance as the world needs to optimize the fuel usage. One of the main challenges for it is to make sure an ignitable air fuel mixture at the spark plug for all kind of engine operating points for the usage of different worldwide fuels (Sauer et al., 2008). Zigan et al., (2010) also stated that large differences in microscopic spray behavior close to the nozzle were detected at valve opening and closing conditions. The radial spray width at nozzle exit as well as the microscopic cone angle decrease by trend with higher Reynolds- and Weber numbers at early injection phases.

With the stable spray cone angle and a short penetration length also under increase back pressure, the SG –DISI injector is the best reproducible spray created. Although it comes with the best specification and advantages, it also has their own weakness. The injector is very complex compare to the others and the price is also expensive. Hence, the new technology with the less expensive multihole injectors is created. In order to optimize the combustion concepts, optical measurement techniques and spray modeling have become important tools (Iyer and Yi, 2009).

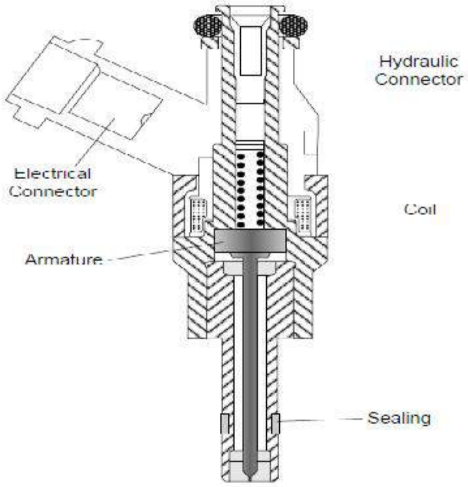


Figure 2.2: The high pressure injectors.

Source: Celik, M. B., 2010

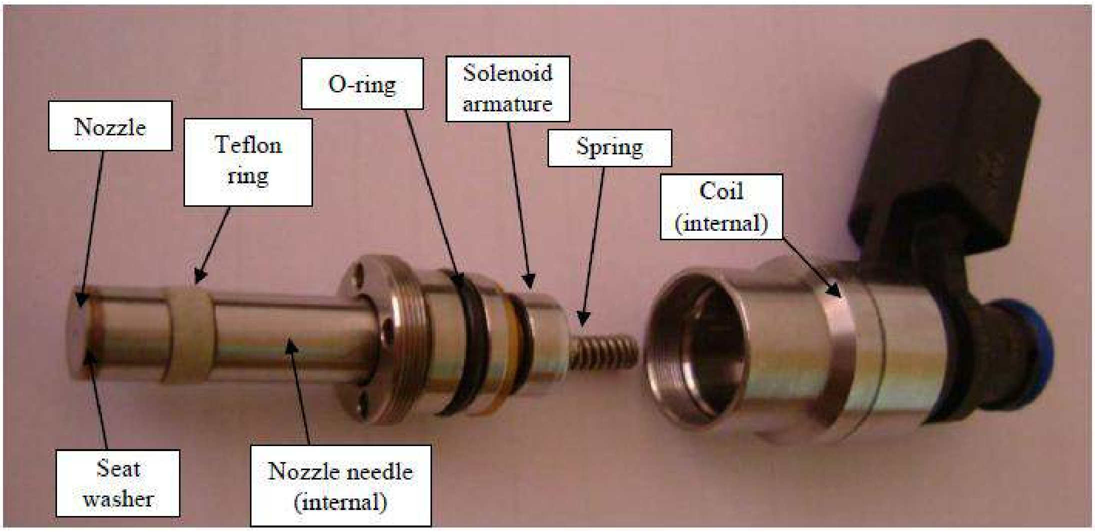
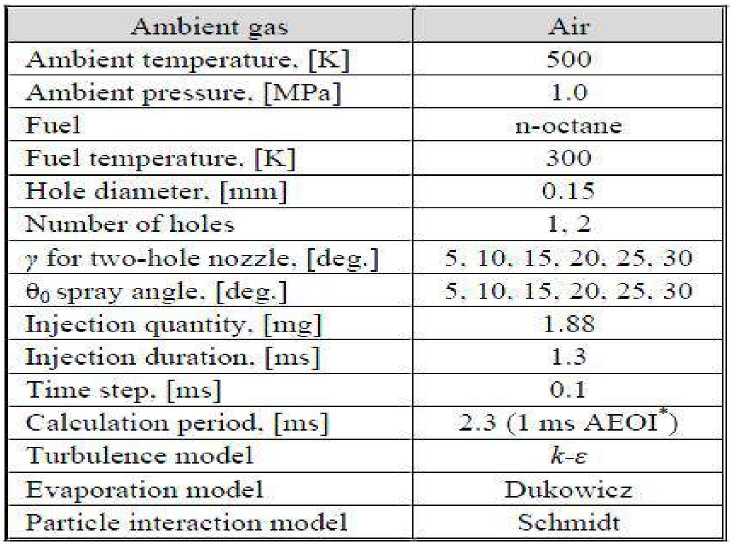


Figure 2.3: An opened GDI Bosch injector

Source: Celik, M. B., 2010

Both of the figure 2.2 and figure 2.3 shows the internal system for the GDI injector which the system is usually controlled electronically. The electronic controller can supply varying pulse to the injector solenoid at different time interval.

The figure 2.3 shows one of the computational conditions examples that is used in flow simulation. The setup for each research computational study is different with the others if the parameter that been used is different. Hemdal et al., (2011) stated that the parameter that should be consider while doing the simulation or experiment are the position of injector and spark plug, the combustion chamber geometry, piston geometry and others. These parameters are important criteria for DI engine since the different size or dimension will give the different results.



Ambient gas	Air
Ambient temperature, [K]	500
Ambient pressure, [MPa]	1.0
Fuel	n-octane
Fuel temperature, [K]	300
Hole diameter, [mm]	0.15
Number of holes	1, 2
γ for two-hole nozzle, [deg.]	5, 10, 15, 20, 25, 30
θ_0 spray angle, [deg.]	5, 10, 15, 20, 25, 30
Injection quantity, [mg]	1.88
Injection duration, [ms]	1.3
Time step, [ms]	0.1
Calculation period, [ms]	2.3 (1 ms AEOI*)
Turbulence model	k-ε
Evaporation model	Dukowicz
Particle interaction model	Schmidt

Figure 2.4: The computational conditions.

Source: Cao et al., (2008)

The efficiency of a DI engine depends on the mixture preparation and its distribution inside the chamber. This phenomenon is based on the interaction of the in-cylinder air motion, either tumble or swirl, and the spray characteristics from the high pressure swirl injector which are generally employed for these engines (Chan et al., 2012). With the high probability to improve the fuel consumption, coupled with the advantages such as quick start that gives the transient response and the accurate control of the air fuel mixture ratio make this engine worldwide interest as an automotive power plant.

The effect of the gas phase on liquid phase is achieved by considering relative terms and gradients for mass, momentum and energy. The transfer of thermal energy is towards the droplet from the surrounding gas phase. With this gain in thermal energy, the droplet temperature increases. At the droplet temperature, the fuel-vapor concentration at the droplet surface is considered saturated.

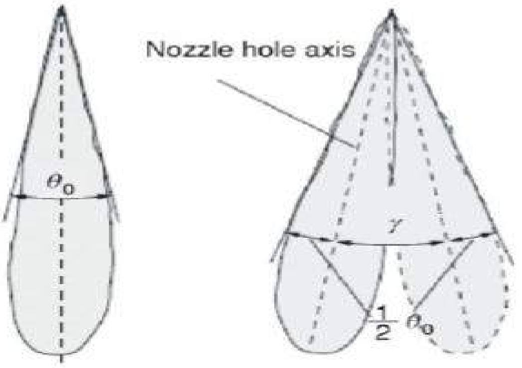


Figure 2.5: The spray angle (θ) and the hole-axis-angle (γ)

Source: Cao et al., (2008)

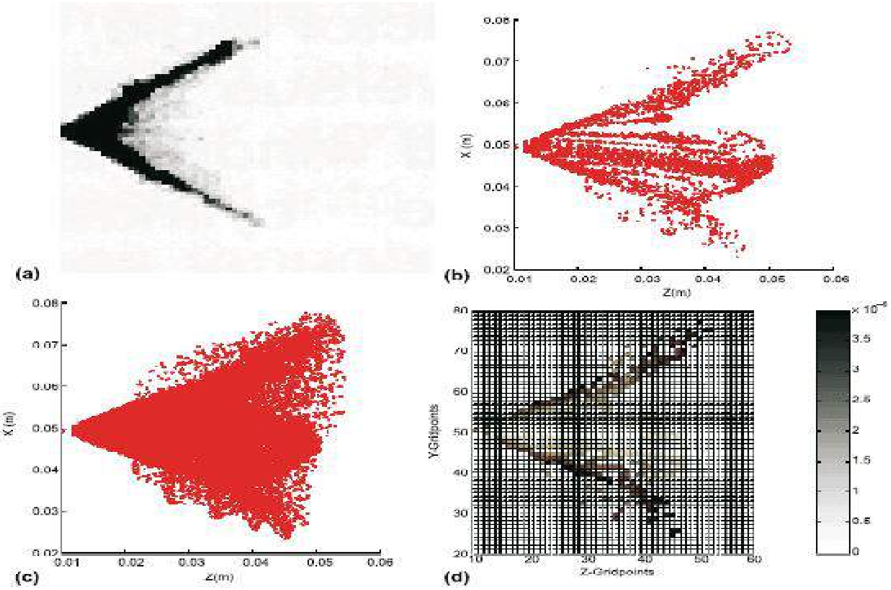


Figure 2.6: Examples of spray breakup in experimental result.

Source: Cao et al., 2008

Computational Fluid Dynamics, CFD is a computer based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow which in a special form over a region of interest, with specified conditions on the boundary of that region.

There are various type of turbulence model existed for solving the computational fluid problem, one of the common turbulence model is the K-Epsilon model. The equation of the model as follows,

For turbulent kinetic energy, k,

$$\frac{\delta}{\delta t}(\rho k) + \frac{\delta}{\delta xi}(\rho k u_i) = \frac{\delta}{\delta xi}\left[\left(\mu + \frac{\mu t}{\mu k}\right)\frac{\delta k}{\delta x_i}\right] + Pk + Pb - \rho\epsilon - YM + Sk \qquad \text{(Eq. 1)}$$

For dissipation energy, ε epsilon,

$$\frac{\delta}{\delta t}(\rho\epsilon) + \frac{\delta}{\delta xi}(\rho\epsilon ui) = \frac{\delta}{\delta xj}\left[\left(\mu + \frac{\mu t}{\sigma\epsilon}\right)\frac{\delta\epsilon}{\delta xj}\right] + C^1\epsilon\frac{\epsilon}{k}(Pk + C^3\epsilon Pb) - C^2\epsilon\rho\frac{\epsilon^2}{k} + S\epsilon \qquad \text{(Eq. 2)}$$

The turbulence viscosity is modeled as,

$$\mu t = \rho C\mu\frac{k^2}{\epsilon} \qquad \text{(Eq. 3)}$$

The production of k is modeled as,

$$Pk = \mu t S^2 \qquad \text{(Eq. 4)}$$

S is the modulus of the mean rate-of-strain tensor, defined as,

$$S \equiv \sqrt{2SijSij} \qquad \text{(Eq. 5)}$$

The model constant for k- ε

$$C_1\epsilon = 1.44 \; C_2\epsilon = 1.92 \; C\mu = 0.09 \; \sigma k = 1.0 \; \sigma\epsilon = 1.3 \qquad \text{(Eq. 6)}$$

The reason of listing the k-epsilon turbulence model is due the consideration of the injector where the flow is in turbulent and compressible. The k-epsilon turbulence model a model that usually use in ANSYS when the flow is related to turbulent and compressible

The flow through the injector nozzle really related to the Bernoulli's equation. Bernoulli equation is concern about the conservation of kinetic, potential and flow energies of the fluid stream. The equation considers the relationship between pressure, velocity and elevation. Thus, the flow through the injection nozzle also is the same situation. The fuel is supply from fuel tank to the injector with the high pressure at large volume. When the injector ready to spray in the fuel inside the chamber, the injector needle move up with the high speed created a space for the fuel to move through the narrow space of nozzle then spay into the chamber. From the situation we can use the equation where when the pressure is high, the velocity is low and when the pressure is low, the velocity will be increase.

According to Millo et al.,(2011) stated that the analysis for spray atomization and characteristic is important for the optimization for bore and stroke for the GDI engine. As the engine is small, the fuel that needed for combustion process is also reduce. In addition, as the engine is downsizing the emission that created from the engine also will be decrease. Pollutant and soot emissions are closely influenced by the parameter of the engine. That parameters are injector geometry (injector type), combustion chamber design and operating engine conditions (pressure injection). There have many benefits when the optimization of the engine is done.

Kiura et al., (2008) stated that there were many research that had been made about the effect of the spray breakup and characteristic form the result it can conclude that downsizing the nozzle hole diameter, increasing nozzle hole number and injection pressure will lead to a better fuel atomization and spatial distribution, which will result in a better engine performance and a lower emission level. The result will be different if the parameter that had been use if different such as sometimes the short penetration tip is suitable to avoid the wall impingement while in the other case sometimes the long penetration tip is more suitable since it will give better air utilization. The vaporization process of the spray is also very important because it will affect not only the ignition stability, but also the flame propagation (Yamamoto et al., 2005)

According to Chan et al, (2012) among other technologies, such as a diesel engine, turbocharging, and downsizing, the GDI engine was considered as one of the technology paths for meeting future emission regulations for reducing greenhouse gases. Furthermore, these engines have potential to achieve greater fuel economy compared with a diesel engine at partial loads and give better performance than port fuel injection (PFI) spark ignition engines at high loads(Moore et al. 2010). So, in order to get the best efficiency for this GDI engine there must have serious investigation, experimental, simulation and other way in order to successfully get it.

CHAPTER 3

3.0 Methodology

3.1 Introduction

In this chapter, we will be discussing about the method that use for completing this project. The method used for obtaining the result and conclusion for this project is based from the other research that had been made by previous researcher. The method that usually use is collecting the parameter that use from the other research as a guide for our project. Most of the research made used Computer Aided Design, CAD model for interpreting their model design. Later the CAD model will be transfer to the analysis software for further investigation. Since the analysis contains the interaction with fluid behavior, ANSYS FLUENT software is used for this type of analysis. Inside the software, it contains various type of function for obtaining the desired result. Simulation on the fluid behavior also can be obtained from this advance software.

3.2 Flow Chart

The flow chart in Figure shows the sequence of method used for the project progress. For the Semester 1, Final Year Project, the progress set up is more likely towards the design stage of the injector before proceeding with simulation. The design stage covers from the rough sketch of the injector until producing the CAD model.

The expected results are determined with the help from literature review reading and also supervisor advice. After the CAD model completed, the model is transfer into the analysis software, ANSYS to be mesh.

The meshing condition also is also according the idea form literature review reading together with supervisor advice. If the model fail on for meshing process, means there is an error occur on the CAD model and correction towards the injector model will be done. On the same time also each main component that needed to be insert inside the ANSYS later will be determine as expected parameters, boundary condition and type of meshing.

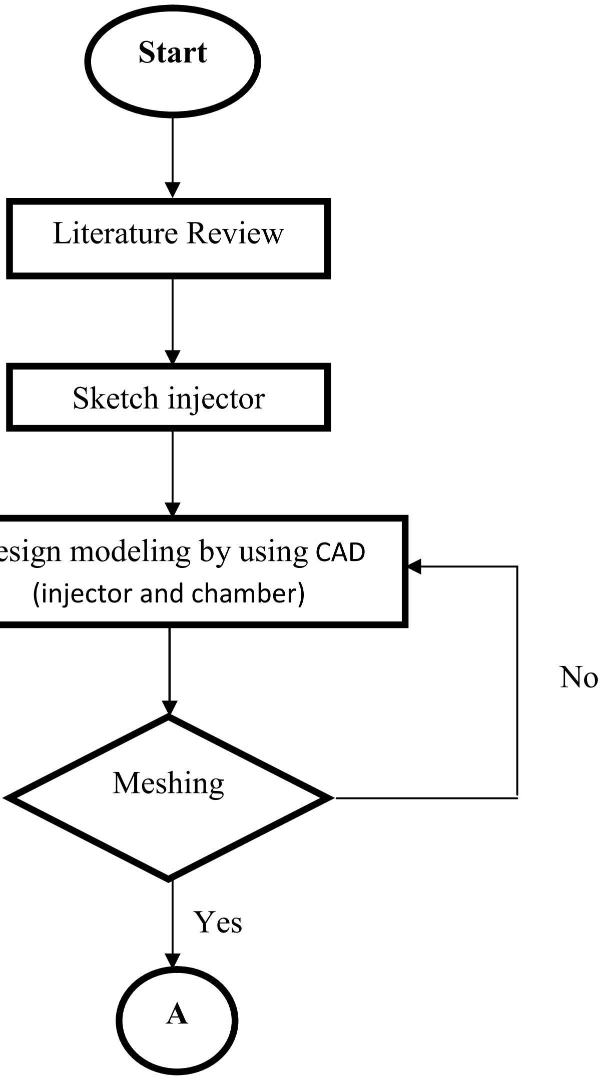


Figure 3.1: Flow chart for Semester 1 FYP progress

For Figure 3.2 shows the Semester 2, Final Year Project, the progress is more likely towards analysis stage. From the expected parameters, boundary condition and also turbulence model, the most suitable component is chosen for the whole analysis process. The analysis result will be taken for conclusion making. Final report writing will finish for this final year project analysis and also final presentation.

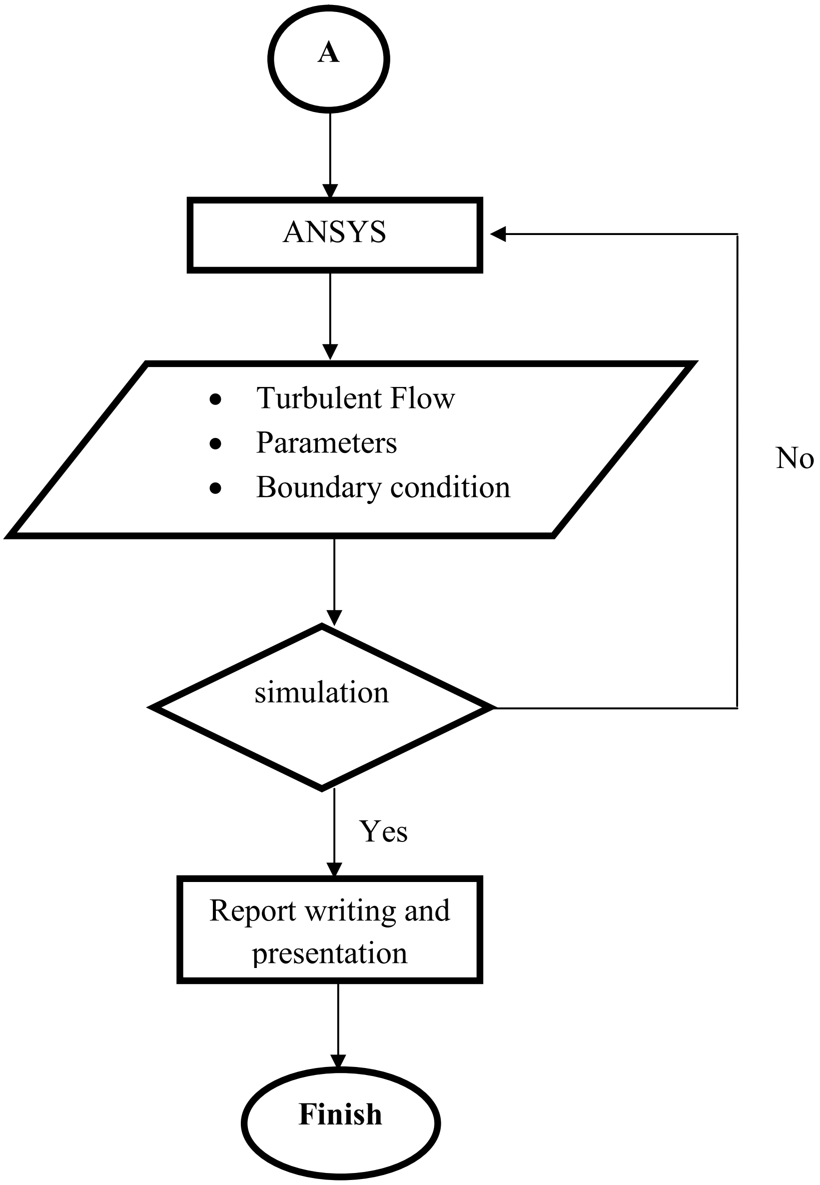


Figure 3.2: Flow chart for Semester 2 FYP progress

3.3 Design Modeling

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulations (Versteeg & Malasekera, 1995).

Computational Fluid Dynamics (CFD) itself emerged as a tool to cut cost and time by doing away with costly experiments to produce better, more efficient engineering designs. In practise, some experimental data or theoretical calculations will still be needed to verify at least the unit or benchmark case, and possibly if time and money permits the final design as well. Experimental data is also often needed for input in CFD simulations, for example in setting the boundary conditions of the model. Theoretical calculations based on simple models are always useful, providing back-of-the-envelope estimates for boundary conditions and sometimes results expected. This shows that an engineer will never do away with experiments and theoretical calculations and totally depend on simulations (Anderson, 1995).

Since experimental results available are of gaseous jets issued from the Bosch Gasoline Direct Injector into free space, a sufficiently large flow domain is required to simulate this. A schematic of the flow domain for the benchmark case is shown in Figure 3.3 and the setting shown in table 3.1.

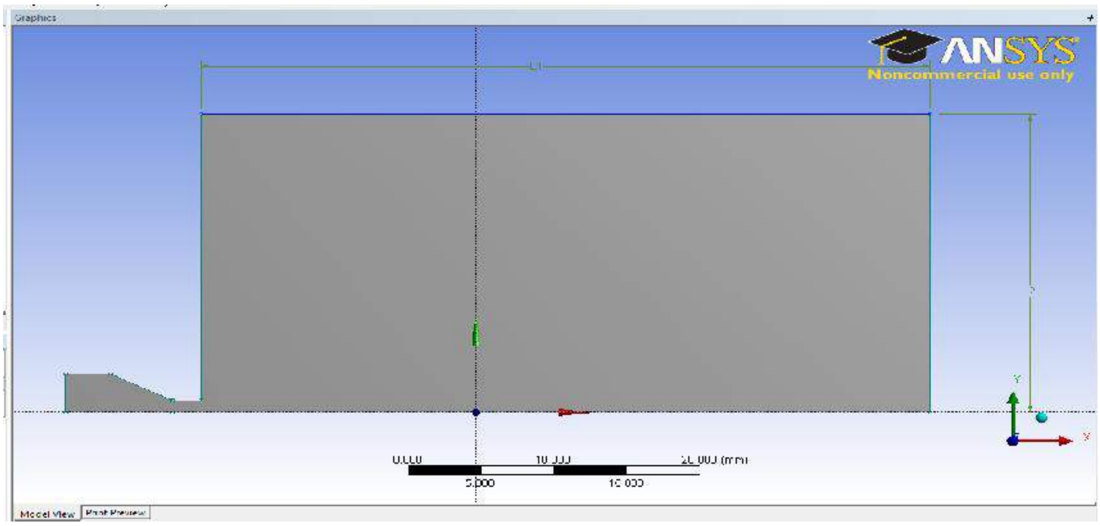


Figure 3.3: Modeling