UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **COMPUTATIONAL STUDY OF FUEL SPRAY STRUCTUE**

SESI PENGAJIAN: **2012/2013**

Saya **MOHD HILMI BIN MOHD ZIN (901123-01-5391)**

mengaku membenarkan tesis (Sarjana Muda/~~Sarjana~~ /~~Doktor Falsafah~~)* ini disimpan di Perpustakaan dengan syarat-syarat kegunaan seperti berikut:

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

[ ]	**SULIT**	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
[ ]	**TERHAD**	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
[√]	**TIDAK TERHAD**	

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap:

PTD 7149, TAMAN HARTA INDAH
JALAN PT SEMERAH
82000 PONTIAN
JOHOR

MR. HAJI AMIRRUDDIN BIN ABDUL KADIR

Tarikh: **25 JUNE 2013**

Tarikh: : **25 JUNE 2013**

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

COMPUTATIONAL STUDY OF FUEL SPRAY STRUCTURE

MOHD HILMI BIN MOHD ZIN

Thesis submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITY MALAYSIA PAHANG

JUNE 2013

EXAMINER'S DECLARATION

UNIVERSITY MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "Computational Study of Fuel Spray Structure" is written by Mohd Hilmi Bin Mohd Zin. I have examined the final copy of this report and in my opinion, it is fully adequate in terms of language standard, and report formatting requirement for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfillment of the requirement for the degree of Bachelor of Mechanical Engineering.

MR. WAN AZMI BIN WAN HAMZAH

Examiner Signature

SUPERVISOR'S DECLARATION

I have examined the final copy of this report and in my opinion, it is fully adequate in terms of language standard, and report formatting requirement for the award of the degree of Bachelor of Mechanical Engineering.

Signature :

Name of Supervisor : MR. HAJI AMIRRUDDIN BIN ABDUL KADIR

Position : Lecturer

Date : 25 June 2013

STUDENT'S DECLARATION

I hereby declare that this thesis, submitted to University Malaysia Pahang as partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering, has not been submitted to any other university for any degree. I also certify that the work described herein is entirely my own, except for quotations and summaries sources of which have been duly acknowledged. This thesis may be made available within the university library and may be photocopied or loaned to other libraries for the purposes of consultation.

Signature :

Name : MOHD HILMI BIN MOHD ZIN

ID Number : MA09037

Date : 25th JUNE 2013

I specially dedicated to my beloved parents
and those who have guided
and motivated me for this project

ACKNOWLEDGEMENTS

First and foremost, I would like to take this golden opportunity to express my sincere gratitude and gratefulness to my project supervisor, Mr. Haji Amirruddin Bin Abdul Kadir for his untiring guidance, support and valuable advice throughout the duration of this project. Without his ideas and guidance through the whole time I would not be able to finish my project. He has always impressed me with his outstanding professional conduct, his strong conviction for science and I acknowledge his help and expertise with gratitude. I am truly grateful for his progressive vision about my training with science and technology, his tolerance of my naïve mistake and his commitment to my future career.

My sincere thanks go to all my lab-mates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to member research group for their excellent co-operation, inspirations and supports during this study.

My sincere gratitude is also extends to my beloved parents and family members for their endless love, prayers, encouragement and personal support in completing this project with a success. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals.

I also would like to express my appreciations to my friends, colleagues and course mates for their support, patience, tolerance and understanding for my tendency to let this research project takes over all else.

ABSTRACT

This thesis deals with the study of fuel spray structure via computational (simulation) method. The main objective of this thesis to perform a computational study of pure gasoline fuel sprays structure development where it covers to parts; to determine the pure gasoline fuel spray angle and spray penetration depth characteristics using sing-hole port fuel injector (PFI) and to determine the impact of different injection pressure on the spray structure of pure gasoline fuel. The spray simulations are done completely by using Computational Fluid Dynamics (CFD) ANSYS CFX software with three nozzle tip diameter; 0.2mm, 0.3mm and 0.4mm. The Computational Aided Design (CAD) model for each nozzle was drawn using the SolidWorks software, the nozzle is attached with 110mm bore and 125mm stroke combustion chamber. In the ANSYS CFX software, the ready CAD model is imported into the design modeler and under goes meshing process with fine relevance center, 4×10^{-5}m min size, 4×10^{-3}m max face size and 8×10^{-3}m max size. There are three types of boundary conditions applied to the meshed geometry model, the first is inlet boundary condition with various injection pressure of 100bar, 150bar, 200bar and 250bar. Opening boundary condition is then place at the combustion chamber with atmospheric pressure value that is 101325Pa and the third boundary condition is wall. The iteration calculation is solved until the convergence approached to the desired residual value and the result is obtained and analyzed. The first comparison made is between penetration depth versus injection pressure and the other is between spray angle versus injection pressure, the results are then compared between nozzle diameter for each injection pressure. The results show that as the injection pressure increased, the penetration depth is also increased as well as the spray angle. The conclusion has shown that the nozzle tip diameter is also effecting the overall spray structure because wider nozzle tip diameter will released more fuel quantity compared to the smaller nozzle tip diameter.

ABSTRAK

Tesis ini adalah berkaitan dengan kajian struktur semburan bahan api melalui kaedah pengiraan (simulasi). Objektif utama projek ini adalah untuk melakukan kajian pengiraan pembangunan struktur semburan bahan api petrol tulen di mana ia meliputi bahagian-bahagian berikut; menentukan ciri-ciri sudut semburan dan kedalaman semburan bahan api petrol tulen dengan satu-lubang port penyuntik bahan api (PFI) dan menentukan kesan tekanan suntikan yang berbeza pada struktur semburan bahan api petrol tulen. Simulasi semburan dilakukan sepenuhnya dengan menggunakan Computational Fluid Dynamics (CFD) perisian ANSYS CFX dengan tiga diameter muncung yang berbeza; 0.2mm, 0.3mm dan 0.4mm. Model Computational Aided Design (CAD) untuk setiap muncung telah dilukis dengan menggunakan perisian SolidWorks, muncung telah dilukis bersama kebuk pembakaran berukuran 110mm diameter dan 125mm strok. Dalam perisian ANSYS CFX, model yang telah siap CAD diimport ke dalam reka bentuk pemodel dan melalui proses penjaringan dengan pilihan pusat relevan yang baik, 4×10^{-5}m saiz minimum, 4×10^{-3}m saiz muka makimum dan 8×10^{-3}m saiz maksimum. Terdapat tiga jenis keadaan sempadan yang digunakan terhadap model geometri, yang pertama adalah keadaan sempadan masuk dengan pelbagai tekanan suntikan seperti 100bar, 150bar, 200bar dan 250bar. Keadaan sempadan pembukaan kemudian meletakkan di kebuk pembakaran dengan nilai tekanan atmosfera, 101325Pa dan keadaan sempadan ketiga ialah dinding. Pengiraan lelaran diselesaikan sehingga penumpuan nilai lelaran mendekati nilai baki yang dikehendaki dan keputusan pengiraan diperoleh dan dianalisis. Perbandingan pertama yang dibuat adalah diantara kedalaman semburan berbanding tekanan suntikan dan perbandigan diantara sudut semburan berbanding tekanan suntikan, keputusan pengiraan jiga dibandingkan diantara diameter muncung dan setiap tekanan suntikan. Keputusan menunjukkan bahawa apabila tekanan suntikan meningkat, kedalaman semburan dan sudut semburan juga meningkat. Kesimpulannya menunjukkan bahawa diameter muncung juga memberi kesan terhadap keseluruhan struktur semburan kerana semburan muncung diameter yang lebih luas akan dikeluarkan kuantiti bahan api yang lebih banyak berbanding dengan muncung diameter yang lebih kecil.

TABLE OF CONTENTS

Page

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

$\sum \dot{Q}$	Net rate of heat addition to the fluid
$\sum \dot{W}$	Net rate of work done by surface forces on the fluid
C_8H_{18}	Gasoline
C_c	Empirical constant
CO_2	Carbon Dioxide
d	Nozzle injector exit diameter
F_i	External body
H_2O	Water
L_c	Liquid core
N_2	Nitrogen
O_2	Oxygen
ρ	Fluid density
ρ_G	Gas densities
ρg_i	Gravitational body force
ρ_L	Liquid densities
t	Time interval
τ_{ij}	Stress tensor
V	Velocity at any point in the flow field
u	Velocity at x-axis direction
v	Velocity at y-axis direction
w	Velocity at z-axis direction

LIST OF ABBEREVIATIONS

CAD	Computational Aided Design
CFD	Computational Fluid Dynamics
PFI	Port Fuel Injector

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

The studies about fuel spray structure for both non-combustion and combustion process have been made multiple times, where most of the research focusing the dilute spray medium which is distant from the nozzle injector exit and the initial dispersed flow of the droplet breakup using common method such as observation, calculation, simulation and modeling are reasonably important due to tiny liquid volume fractions (Faeth, Hsiang and Wu, July 1995). Now a research is made to study the spray structure of a fuel focusing the whole medium, starts from the nozzle injector exit or known as dense spray medium until the dilute spray medium including the droplet features depend on various spray pressure using computational method.

General aspects that need to be include in this studies are divided into four categories, the first aspect is the spray structure in the dense spray medium where the spray liquid still not distributed right before and after the nozzle injector exit in order to investigate and define the initial spray properties such as liquid viscosity, pressure, temperature and volume (Faeth, Hsiang and Wu, July 1995). The second aspect is the properties of primary breakup, it is the initial conditions for the dense sprays medium including both spray structure properties and the hardware properties such as the nozzle injector exit. Every single thing such as pressure, viscosity and more are important and has the potential to influence the structural characteristics of the spray (Faeth, Hsiang and Wu, July 1995).

The third aspect is the properties of secondary breakup which will closely related with the rate controlling process of dense spray medium and structural characteristics of the droplet which also related with the rate controlling process of dilute spray medium. Each characteristics of spray structure like spray distributions, position of spray structure and spray tip penetration are essential to observed in order to gain the perfect outcome (Faeth, Hsiang and Wu, July 1995). The last aspect is the properties of the droplet characteristics at the end of spray distribution such as droplet sizing, where the spray already passed through the nozzle injector exit, dense spray medium and dilute spray medium (Faeth, Hsiang and Wu, July 1995).

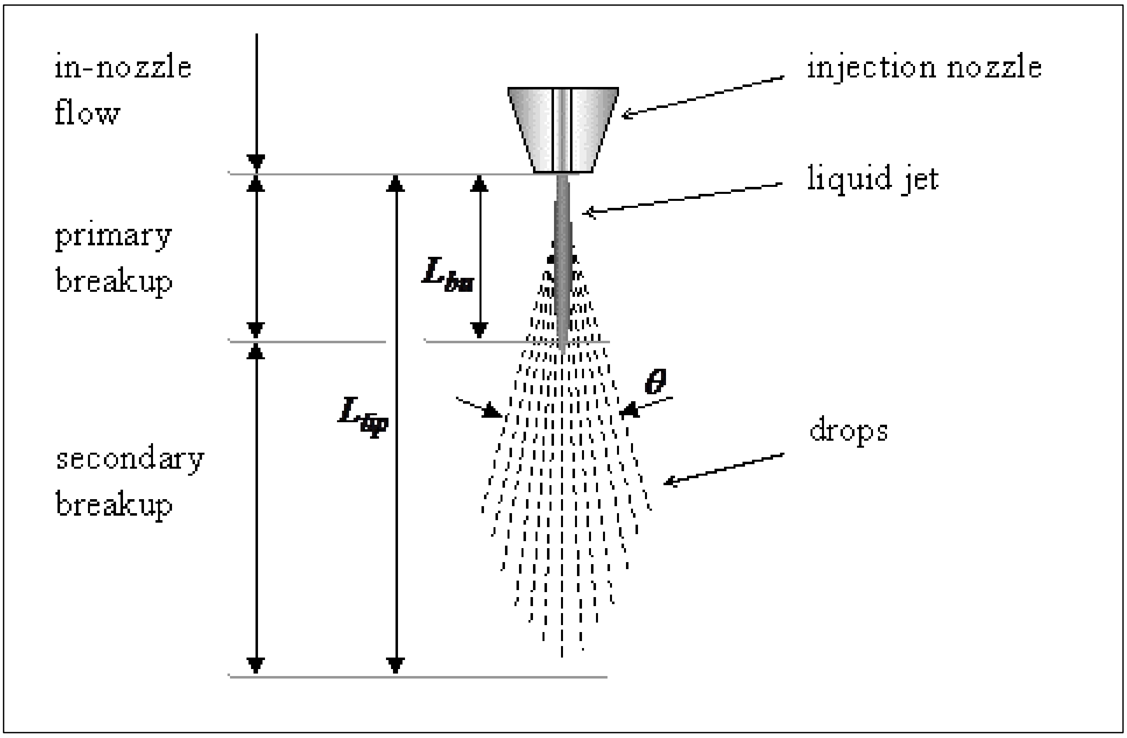

Figure 1.1: Main physical parameters on spray structure

Figure 1.1 shows the main physical parameters on spray structure with the primary breakup and the secondary breakup, it illustrate a measured spray characteristics that are basically been classified into two categories. The first one is called as "Macroscopic Characteristics" that focus on both primary breakup and secondary breakup that containing the spray angle and spray tip penetration. The second one is called as "Microscopic Characteristics" which focus only on the secondary breakup containing droplet distribution (diameter), droplet velocity, air-fuel ratio and so forth. A real experiment would be more reliable because it is exposed to the real world conditions such as pressure, temperature, humidity and any other properties that might be affecting the spray structure development.

1.2 PROBLEM STATEMENT

Nowadays the development in automobile engine is sharply improved with the emergence of bio-fuel engine system, hybrid car system and many more (Anand, Madan Mohan & Ravikrishna, 2012). In order to know how pure gasoline can produce the maximum power output or result that can be matched with the advanced system, a computational study of fuel spray structure is essential. Spray fuel structure of pure gasoline is an important factor to be study and investigate due to differences in characteristics such as injection pressure, droplets size distribution, spray progression, position of spray structure and spray tip penetration may result in different power output of an internal combustion engine (Schmehl, Maier & Wittig, 2000). The competition between advanced fuel and pure gasoline in power output value is vastly intense due to differences chemical substances in each burning fuel.

Different fuel will provide different power output result, hence the computational study of fuel spray structure of pure gasoline is important to identify the characteristics of spray development, droplet size distribution and many more in order to increase the power output of an engine. Any factors that might be affecting the spray structure such as spray angle, depth, type of nozzle and injection pressure will be included in this study. The spray itself must obey air-fuel ratio to generate maximum heat energy that can be transform to mechanical work. Another problem why computational study on spray structure is essential because of incomplete burning of fuel will result in less energy for the mechanical work and at the same time will affect the condition inside the engine (Rossella Rotondi & Gino Bella, 2005).

1.3 OBJECTIVES

The main objective of this research is to perform a computational study of pure gasoline spray structure development where it will cover two parts:

i. To determine the pure gasoline fuel spray angle and depth characteristics using single-hole port fuel injector (PFI).

ii. To determine the impact of different injection pressure on the spray structure of pure gasoline fuel.

1.4 SCOPE OF STUDY

This project is focus on computational study of pure gasoline fuel spray structure development using suitable software that is Computational Fluid Dynamics (CFD). The entire computational study will be performing using several different amount of injection pressure and using a single-hole PFI. To complete this project, the actions are required:

i. Study of spray angle and depth of gasoline fuel by using single-hole PFI.
ii. Study the effect of different injection pressure on spray structure which is in the range of 100bar to 250bar.
iii. Study the result of analysis from the simulation done.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Improving modern internal combustion engine efficiencies by increasing or decreasing the pressure levels of the combustion processes require sophisticated combustion concept and analysis method. The principles of modern internal combustion engine are the strategic characteristics to inject the liquid fuel and to mix it with the flow of compressed air. In order to study and understand the fuel spray structure, computational study is required than depending on previous experiments and researches (Schmehl, Maier & Wittig, 2000).

2.2 DENSE AND DILUTE SPRAY STRUCTURE

Dense spray structure is a part of spray structure where it covers the medium between the nozzle injector exit and the dilute spray medium, a sketch of spray structure near the nozzle injector exit is illustrated in Figure 2.1. It is also a medium where the spray structure will come together and mix with the gas phase inside the combustion chamber. During practical combustion processes, the atomization of the spray breakup is the most important for the rapid mixing of the fuel and the oxygen where both of them existed in liquid and gas phase (Reitz & Bracco, 1982). Dense spray medium consist of two main multiphase flows, the first multiphase is the liquid core where the liquid are mostly does not mix with the oxygen. The second multiphase is the dispersed flow region where the liquid is already mixed with the oxygen and atomization or spray droplet had been developed.

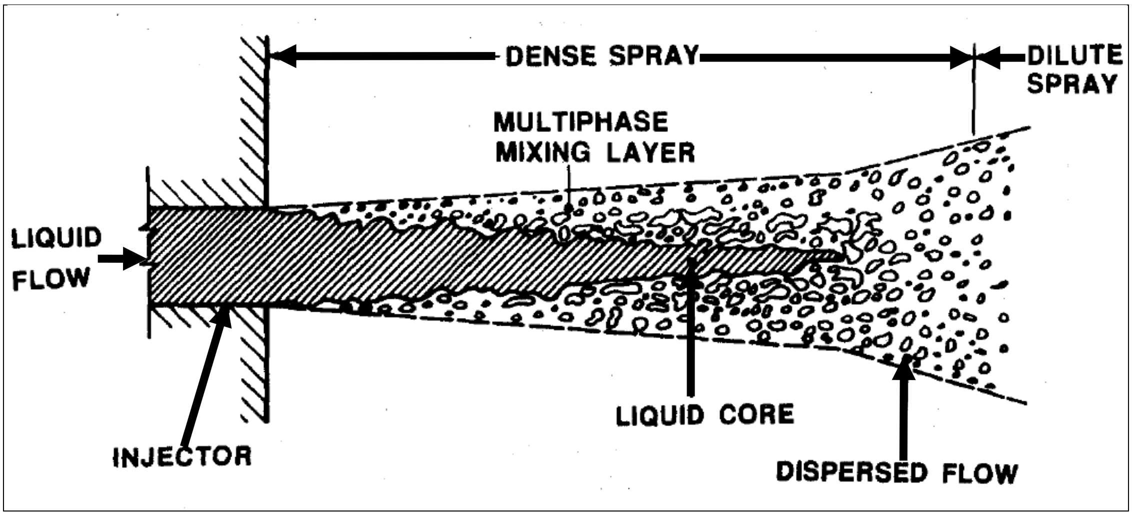

Figure 2.1: Sketch of spray structure near the nozzle injector exit

Source: Faeth, Hsiang and Wu, July 1995

The liquid core is similar to the potential core of a single phase nozzle injector exit although it is generally much longer. The Eq. (2.1) below represents the length of the liquid core, L_c, where d is the nozzle injector exit diameter, ρ_L and ρ_G are liquid and gas densities and C_c is an empirical constant in the range of 7-16. This visualizes that L_c/d in the range of 200-500 for a typical spray at atmospheric pressure, with this ratio generally being inversely proportional to the square root of pressure. Hence, liquid core is most visible feature of round pressure-atomized spray (Chehroudi, 1985).

$$L_c/d = C_c(\rho_L/\rho_G)^{1/2} \tag{2.1}$$

Dispersed flow region take place at the end of the liquid core, it involves a developing multiphase mixing layers between the liquid and the gas, followed by a multiphase layers that evolves into spray droplets in dilute spray flow. It is a region which connected the dense spray medium and dilute spray medium, that shows the mixing of both liquid and gas had been happen. The dense spray medium generally related with the liquid core even though it is not totally accurate because at the end of dense spray medium is a dilute spray medium while the initial liquid core's flow has a large liquid volume fractions. The properties and existence of the dense spray medium are very dependent toward liquid flow properties such as disturbance levels and turbulence levels at the nozzle injector exit.

2.3 SPRAY ANGLE AND SPRAY TIP PENETRATION

In definition, spray angle is the angle of opening of dispersed fluid flow that experienced transformation from laminar flow to turbulent flow under certain condition. Spray angle is determined by many factors such as opening dimension, pressure, viscosity and so forth, it is known that the spray angle does not holds for the entire spray propagation. It tends to collapse or diverge as it moves away from the nozzle tip. An assumption has been made where the spray angle will remain constant throughout the spray distance travelled, but in actual situation the angle will not remain constant throughout the spray distance travelled. Figure 2.2 illustrate the difference between actual and theoretical spray angle. Spray tip penetration or spray depth is the total length of fluid spray structure between the nozzle tip and the end of spray propagation, the distance is determined by the injection pressure and also influenced by the opening dimension.

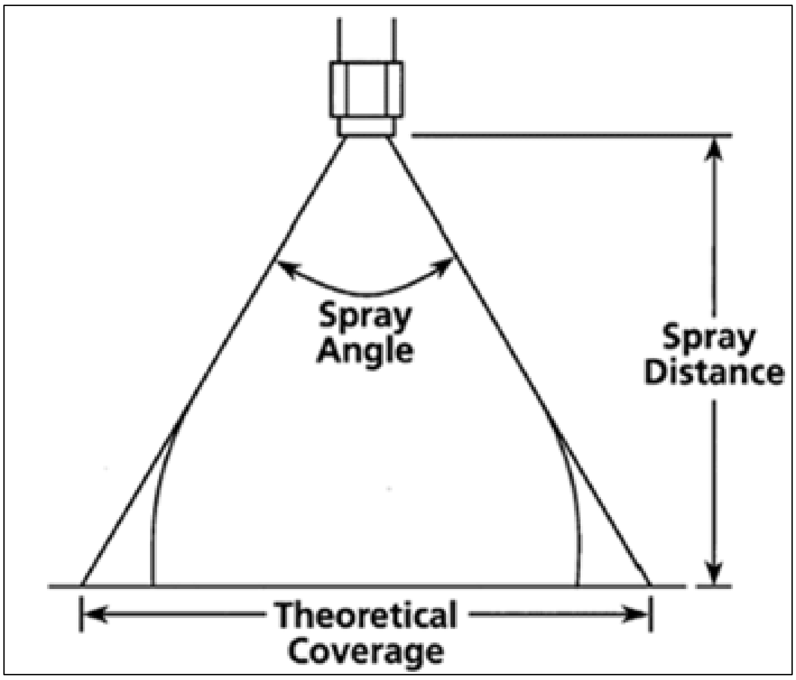

Figure 2.2: Illustration of spray angle

2.4 FLOW STRUCTURE OF DENSE SPRAY MEDIUM

In order to evade the significant effect of the degree of turbulence development at the nozzle injector exit, further information about dense spray properties will be limited to condition where there is fully-developed turbulence flow at the nozzle injector exit (Tseng, 1992). Time-average liquid volume fractions were predicted at various pressure levels, this prediction are based on the turbulence model under the locally-homogenous flow (LHF) approximation. For a complete explanation of this model prediction, the relative velocities between the phases are assumed to be very small in comparison to the mean flow velocities (Ruff, 1989).

Estimated liquid volume fractions near the nozzle injector exit are unity followed by rapid reduction of the liquid volume fractions due to spray breakup development. As the pressure increases, the initial reduction of liquid volume fractions becomes smaller and indicating faster mixing rates at higher ambient gas densities (Ricou & Spalding, 1961). With this condition, LHF predictions generally are good because separated flow effect due to relative velocity differences between liquid and gas are not very prominent when the flow is mostly in liquid phase. Although the variation on liquid volume fractions suggests a relatively short liquid core, this is not completely related in terms of mixture fractions.

2.5 PRIMARY BREAKUP

In primary breakup, the most important process spray structure development is the droplets formation near the liquid surfaces because it initiates the atomization process, controlling the liquid core length and provides the initial condition of the dispersed flow region. Due to problems of observing the primary breakup in the dense spray medium and effects of secondary breakup, the current information and understanding of primary breakup is limited. Other than that, the effects of flow development and liquid disturbances such as turbulence at the nozzle injector exit provide an unusual large impact on the primary breakup properties. With the present of pulsed holography technique, it have provided a chance to observe the properties of dense spray medium for making the progress move forward and at the same time gaining a better understanding of primary breakup process (Wu & Faeth, 1993).

2.5.1 Onset of Breakup

All spray properties that have been established by the past studies including criteria for the onset of breakup are strongly affected by the degree of flow development and the disturbance such as turbulence at the nozzle injector exit. Based on early studies of pressure atomization shows that both mixing rates and atomization quality are not the same for laminar and turbulence flow at the nozzle injector exit (Lee & Spencer, 1933). Further studies conclude that turbulence generated in the flow has a small effect on the spray droplet properties. After that, another studies shows that spray breakup could be suppressed entirely for super-cavitations flows where liquid jet neither separates from the injector route wall near the end of the contraction section and does nor reattach (Karasawa, 1992).

Other studies also have discover that liquid phase flow properties have dominated observations of primary breakup in pressure-atomized spray and the aerodynamics effect does not very crucial at the liquid or gas density ratios at normal pressure and temperature. The breakup of the liquid jet in an air at atmospheric pressure was related with the presence of turbulent boundary layers along the injector route walls near the nozzle injector exit (Hoyt & Taylor, 1977). Other than that, large changes in the aerodynamics environment including both counter-flowing and co-flowing air result a small effect on the breakup properties. In a reality, the actual properties of the turbulent boundary layers along the injector route walls will be ignore in any experimental condition (Hoyt & Taylor, 1977).

2.5.2 Breakup Outcomes

Once the condition for the onset of turbulent primary breakup is determined, the breakup outcomes that cover the variation of spray droplet velocity and size distribution with increasing length from the nozzle injector exit will be review. After the turbulent primary breakup, the spray droplet sizes satisfy the universal root normal distribution and the spray droplet distribution is uniform. According to the previous turbulent primary breakup experiment, there are three types of turbulent primary breakup (Wu & Faeth, 1993):

1. Non-aerodynamics turbulent primary breakup.

2. Aerodynamically-enhanced turbulent primary breakup, observed at the onset conditions.
3. Aerodynamic turbulent primary breakup, which involves unification of turbulent primary and secondary breakup.

The result from the previous experiment shows that the boundaries of these turbulent breakups are fixed by the liquid or gas density ratio, the breakup times used to determine types of turbulent primary breakup were based on the mean diameter of the spray after the primary breakup or after the primary breakup stage of combined primary and secondary breakup for condition outside the onset of breakup.

A major issue still uncover involves primary breakup of non-turbulent liquids and the relevance of the classical primary breakup theories (Taylor, 1963 and Levich, 1962). Results show that it is hard to observe the non-turbulent primary structure. The main obstacles are effects of liquid disturbances such as turbulent, the invasion of secondary breakup and weak aerodynamics effects for most liquid at atmospherics temperature and pressure.

2.6 SECONDARY BREAKUP

Based on the previous considerations of the spray structure of dense spray medium, the secondary breakup is essential with its effect on the spray droplet size distribution as the flow movement approaching the dilute spray medium. As reviewed before, primary breakup at the surface of the liquid core developed spray droplets that ate unstable to the formation and development of the secondary breakup. Other than that, both typical power and propulsion systems of high-pressure combustion involves situation where the surface tension of spray droplets becomes small due to the liquid surface move towards the thermodynamics critical point.

In previous findings, there have two limitations of define disturbances that cause the deformation and spray breakup droplets. The first limitations is the shock wave disturbances that provide changes in the ambient environment of a spray droplets at the end of the primary breakup, while the other limitations is the steady disturbances of freely-falling spray droplets in spray drying processes or in rainstorm. The shock wave disturbances effects have become the major attention and approximate the secondary breakup environment in the dense spray medium.

2.6.1 Deformation and Breakup Movements

Many studies and researches have made an assumption on the conditions and definitions for the onset of various deformation and breakup movements of spray droplets subjected to shock wave disturbances. When the liquid viscosity effects are relatively small, the observed breakup movement at the onset of breakup has been termed as ‘bag breakup’ illustrated in Figure 2.3. This ‘bag breakup’ is the deflection of the spray droplets into a thin disk normal to the flow path and the deformation of the middle of the disk (Wierzba & Takayama, 1988).

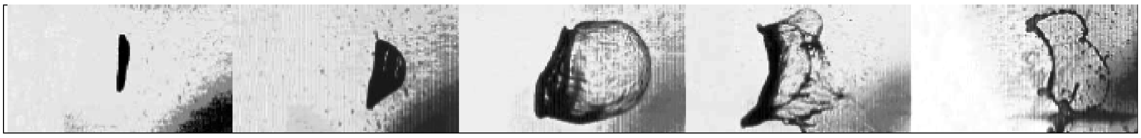

Figure 2.3: Spray droplets deformation and ‘bag breakup’

Source: Schmehl, Maier & Wittig, 2000

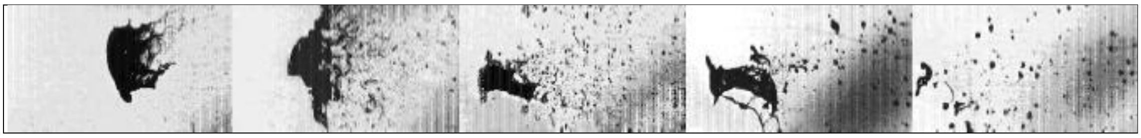

Figure 2.4: Shear breakup

Source: Schmehl, Maier & Wittig, 2000

The observations of shear spray breakup as illustrated in Figure 2.4 have been made at high relative velocities where the shear spray breakup experienced a deflection of the edge of the disk in the downward path, deflection of the middle of the disk and the stripping of spray droplets from the edge of the disk. The conversion between the ‘bag breakup’ and the shear breakup movement is a complex mixture where this complex breakup mechanism can only be observed at high relative velocities and known as ‘catastrophic breakup’ (Reinecke & Waldman, 1970).

2.6.2 Breakup Dynamics

The discussion about deformations and spray breakup movement transitions prioritize the importance of breakup times and identify its characteristics when the liquid viscosity forces are in huge comparison to the force of surface tension. The period before the onset of the spray breakup is the period where the spray droplets experienced significant deformation, the drops are initially drawn into a flat shape because of the presence of the relative motion of the gas phase. Certain researches have summarized a relatively large data base of maximum spray breakup droplets deformations for steady disturbances (Hsiang & Faeth, 1992).

2.6.3 Breakup Outcomes

Secondary breakup can be treated using jump condition with the assumption of spray breakup times and distances are relatively small compared to the characteristics of dense spray medium. To fulfill this approach, information about spray breakup droplets size and velocity distributions right after the secondary breakup take place are essential. Measurement information for the ‘bag breakup’ movement is limited to provide enough guidance about the spray breakup droplets sizes as the result obtained from the secondary breakup (Gel’fand, 1963). Further research used the pulsed holography technique and obtained a complete description of the secondary breakup outcomes for shock wave disturbances conditions (Hsiang & Faeth, 1993).

The secondary breakup in the dense spray medium is not properly represented by jump conditions at the high pressure surrounding of multiple practical spray combustion processes. Under such obstacles, the secondary spray breakup should be assumed as a rate process. Other than the deformations and spray breakup movements, existing information about the secondary spray breakup is still limited and it show clearly that additional study or research is essential in order to gain better understanding of secondary spray breakup properties for practical combustion processes.

2.7 COMPUTATIONAL FLUID DYNAMICS (CFD) SOFTWARE

Computational fluid dynamics (CFD) is a modern analysis process that used numerical and algorithm method which allows a computational model representing the physical system to be built or studied and uses computers to simulate fluid flow dynamics. CFD itself raise the head as a useful tool to reduce cost and time waste by computational method compared with costly and more time consumed experiments to produce a much better result and design. Experimental data is also required for input in CFD simulations for example the flow type and boundary conditions properties. When fluid flow model is applied to this virtual prototype, the CFD software application is capable to predict the outputs of the fluid dynamics.

Other than that CFD also predict the transfer of heat, mass, phase change, chemical reaction, mechanical movement, stress or deformation of related fluid structures and associated phenomena such as chemical reactions by means on computer based simulation (Baris Guler & Rizwan Ali, October 2004). Both compressible and incompressible fluid flows can be combined with specific properties and parameters, all the simulations can be done using 2D and 3D flows (www.cosmol.com). Advantages and benefits of CFD simulation are listed below:

- 3D surface and solid modeling
- Simulation, visualization and analysis of the fluid flow
- Full analysis report including integrated quantities
- Capable to shows result animations and pictures of fluid flow field
- Alterations done in the 3D model are associative with mesh
- Quick recalculation
- Many operating conditions can be calculated with same analysis model

In order to determine the spray structure characteristics such as spray angle and depth of gasoline fuel, CFD CFX software will be used for simulation and get the result. The parameter that will be examined by using this CFD software is different injection pressure range from 100bar to 250bar. This parameter will be the manipulated variable and the final simulation results are capable to determine the spray angle and depth. Another objective that needs to accomplish is to determine the impact of different injection pressure on the spray structure development. From the final simulation result obtained, we will determine the graph that will show details about the simulation and interpret the spray structure result.

2.8 BASIC CONSERVATION EQUATIONS ON CFD

Fundamental of CFD is based on the governing equations of fluid dynamics, derivations of the equations where physical laws are taken into account. The physical laws are mass is conserved for the fluid, Newton's second law-the rate of change of momentum equals the sum of forces acting on the fluid and the first law of thermodynamics-the rate of change of energy equals the sum of the rate of heat addition to and the rate of work done on the fluid.

2.8.1 The Mass Conservation Equation

One conservation law that is pertinent to fluid flow is matter may neither be created nor destroyed. An arbitrary control with volume, V fixed in space and time to be considered, the mass conservation equation or continuity equation can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \left(\rho \mathbf{V}\right) = 0 \tag{2.2}$$

Where ρ is the fluid density, t is the time interval and **V** is the velocity at any point in the flow field where it can be describe as local velocity component *u*, *v* and *w*. On the Cartesian coordinate system the equation can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \tag{2.3}$$

2.8.2 The Momentum Conservation Equation

Newton's second law of motion states that the sum of forces that acting on the fluid element equals to the product between its mass and acceleration of the element. The momentum conservation equation for Newtonian fluids in an inertial (non-accelerating) with u_i reference to the Cartesian coordinates system can be written as:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i \tag{2.4}$$

Where p is the static pressure, τ_{ij} is the stress tensor and the gravitational body force and external body respectively are ρg_i and F_i respectively on the specific coordinate direction.

2.8.3 The Energy Conservation Equation

The equation for the conservation of energy is derived from the consideration of the first law of thermodynamics:

$$\begin{Bmatrix} Time\ rate\ of \\ change\ of\ energy \end{Bmatrix} = \begin{Bmatrix} Net\ rate\ of \\ heat\ added \end{Bmatrix} \left(\sum \dot{Q}\right) + \begin{Bmatrix} Net\ rate\ of \\ work\ done \end{Bmatrix} \left(\sum \dot{W}\right) \tag{2.5}$$

The two terms represented by $\sum\dot{Q}$ and $\sum\dot{W}$ describe the net rate of heat addition to the fluid within the control volume and the net rate of work done by surface forces on the fluid.

2.9 TURBULENCE FLOW

In fluid dynamics contact, a flow regime characterized by the chaotic and stochastic property change is known as turbulence or turbulent flow, most engineering problem will have to deal in turbulent in nature. This type of flow includes high momentum convection, low momentum diffusion and darting modification of velocity and pressure in space and time. A small disturbance in a normal laminar flow might be lead to chaotic and disordered state of motion, Figure 2.5 illustrate the transformation from laminar flow to turbulent flow.

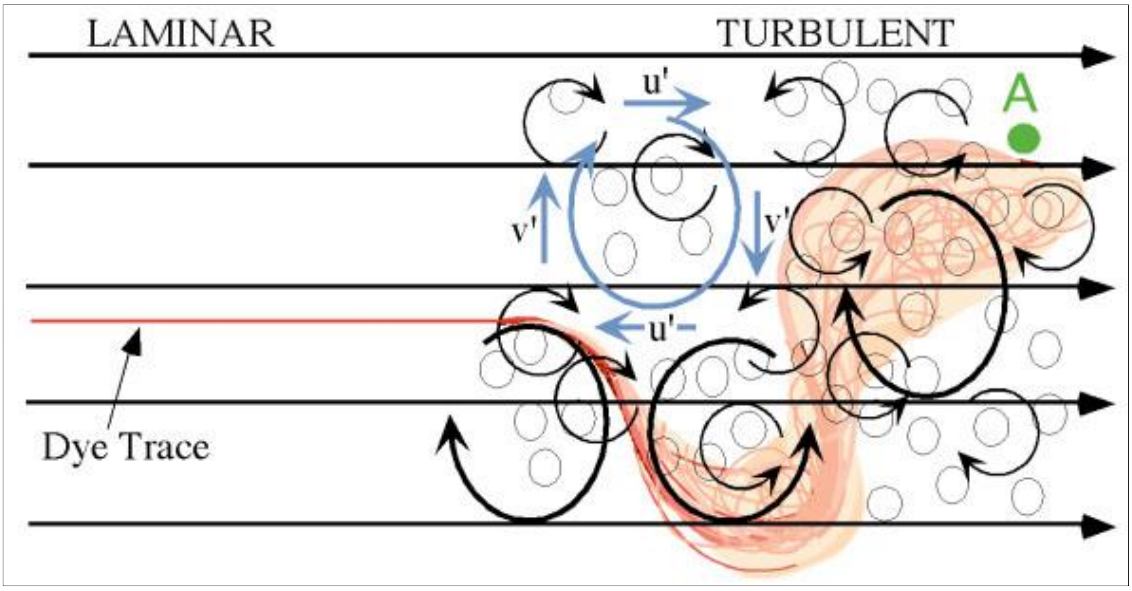

Figure 2.5: Transformation from laminar flow to turbulent flow

2.10 GASOLINE FUEL

Gasoline or generally known as petrol is a colorless liquid that is derived from crude oil, it is widely used as primary fuel options in an internal combustion engines. Compared to the other fuels, gasoline is more volatile than diesel oil or kerosene because of the presence of the base constituents and also additives. Currently the development of vehicles engines is rapid by using other than gasoline as their main fuel and also the emergence of hybrid system might disturb gasoline performance. Only by achieving an ideal mixture of air and fuel ratio or known as stoichiometric air-fuel ratio, than gasoline can experience a complete combustion and hence more power can be generated. Equation 2.6 below shows a chemical reaction of a complete combustion equation of gasoline with 100% air, the combustion products of gasoline are carbon dioxide (CO_2), water (H_2O) and nitrogen (N_2).

$$C_8H_{18} + 12.5(O_2 + 3.76N_2) \rightarrow 8CO_2 + 9H_2O + 47N_2 \tag{2.6}$$

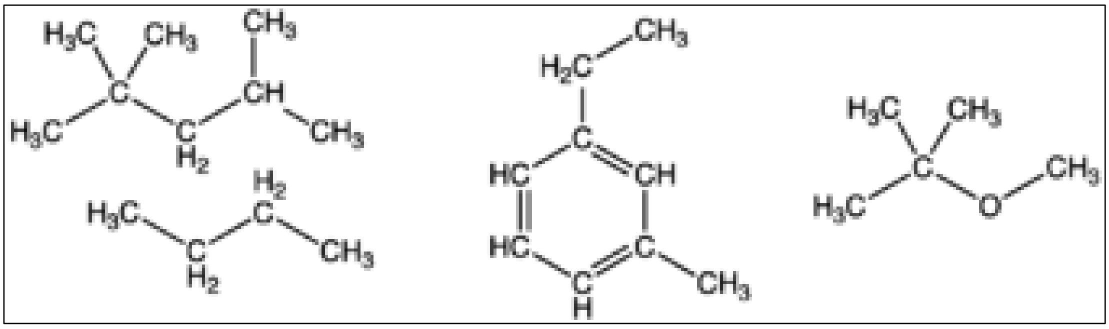

Figure 2.6: Few main component of gasoline base chemical chain

It known that straight-run gasoline from crude oil via distillation does not suitable for modern engines as they do not fulfill the required specification. Due to this complication, gasoline will be blended with other materials to make the fuels have different characteristics to use in modern engines. Figure 2.6 shows few main component of gasoline base chemical chain. The blended fuels still have the same carbon number in their chemical component but only differ in hydrogen number and type on bonds between them. Most gasoline base fuel will have the same properties, the properties of gasoline is listed in Table 2.1.

Table 2.1: Gasoline fuel properties

Property	**Gasoline content**
Chemical Formula	C_4 to C_{12}
Molecular Weight ()	100-105
Carbon	85-88
Hydrogen	12-15
Oxygen	0
Specific gravity, 60° F/60° F	0.72-0.78
Density, lb/gal @ 60° F	6.0-6.5
Boiling temperature, °F	80-437
Freezing point, °F	-40
Flash point, closed cup, °F	-45
Specific heat, Btu/lb °F	0.48
Stoichiometric air/fuel, weight	14.7[b]

CHAPTER 3

RESEARCH METHODOLOGY

3.1 INTRODUCTION

Methodology is a part of important element with the main purpose that is to make sure the development of the study is running smoothly and achieves the expected result. In the other hand methodology is essential to make sure that the study obeys the guideline based on the objectives of the study. Based on the objectives and scope of study, methodology will act as the framework where supervisor can get the overall view of the study flow and development. With this framework, supervisor will be able to maintain and provide sufficient guidance of the study progression in order to make sure all the tasks given can be achieve and complete in the desired time.

Through this section, any problem faced can be identified and discussion can be a huge help to overcome any problems occurred. A methodology consists of a constructed flow chart to provide more clear details about the whole process of the study, there are several steps that must followed in order to ensure that the objective of the project can be achieved.

3.2 FLOW CHART

A flowchart visually displays the formalized sequence of activities, operations or step-by-step progression of this project using connecting lines and conventional symbols. Flowcharts can be used to determine the work flow that is need to be highlight and might be a good help for explaining how a project progression works. Figure 3.1 below shows the flow chart for this project.

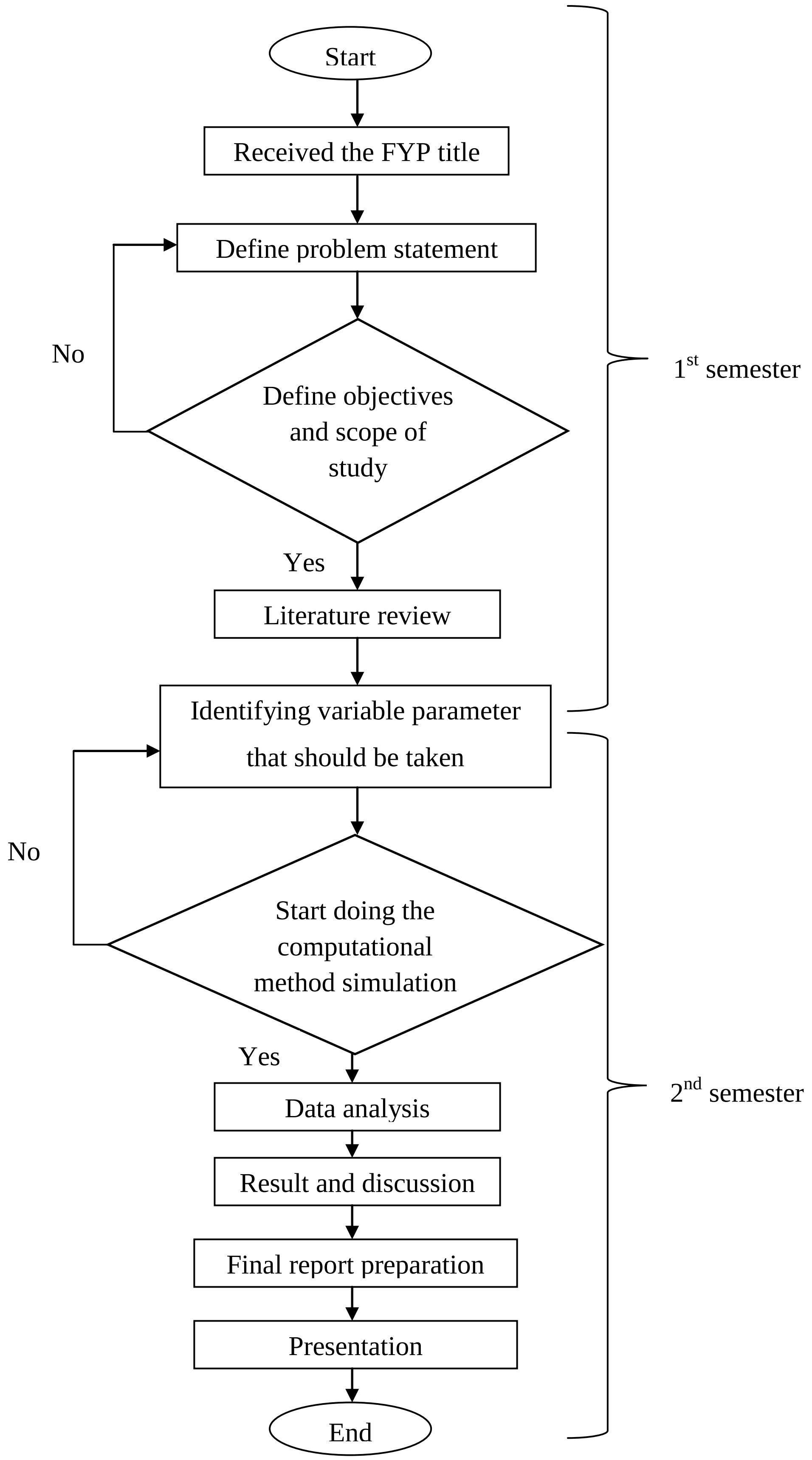

Figure 3.1: Flow chart of the project

3.3 SIMULATION METHODOLOGY

An accustomed process in performing a simulation on CFD analysis has been indicated by the National Project for Applications Oriented Research in CFD (NPARC) Alliance. The simulation of this project will follow the outlined by the NPARC with a complete CFD analysis that consisting three main elements; pre-processor, solver and post-processor. Figure 3.2 shows the connection between the three main elements.

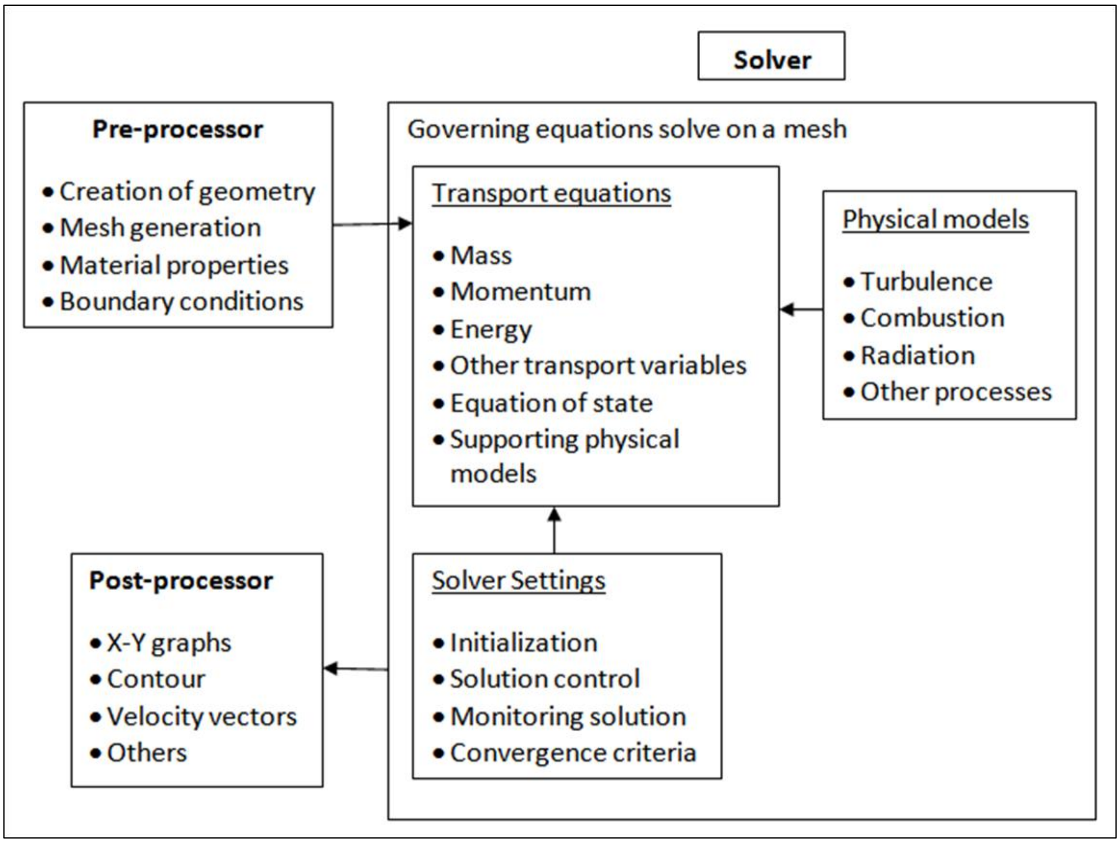

Figure 3.2: Connection between the three main elements in a complete CFD analysis

3.4 NUMERICAL SOLUTION

A control volume based is the fundamental of CFX's numerical approach, the computational (solid geometry) domain will be discrete into multiple smaller control volumes using grid or known as the meshing process. It is proved that the smaller the mesh size will result in better analysis on the fluid flow, but it will take extra time to complete the iteration on each mesh size. Mesh size is determined based on the geometry of Computational Aided Design (CAD) model drawing via SolidWorks software.

Founded on the physical phenomena on the real fluid flows usually need to be solved using iterative solution approach due to complex and unknown consistency within the flow. The discrete values of the fluid flow properties are required for the initialization which is crucial for the solution. Figure 3.3 below shows the flow chart of initialization and solution control in CFD CFX analysis.

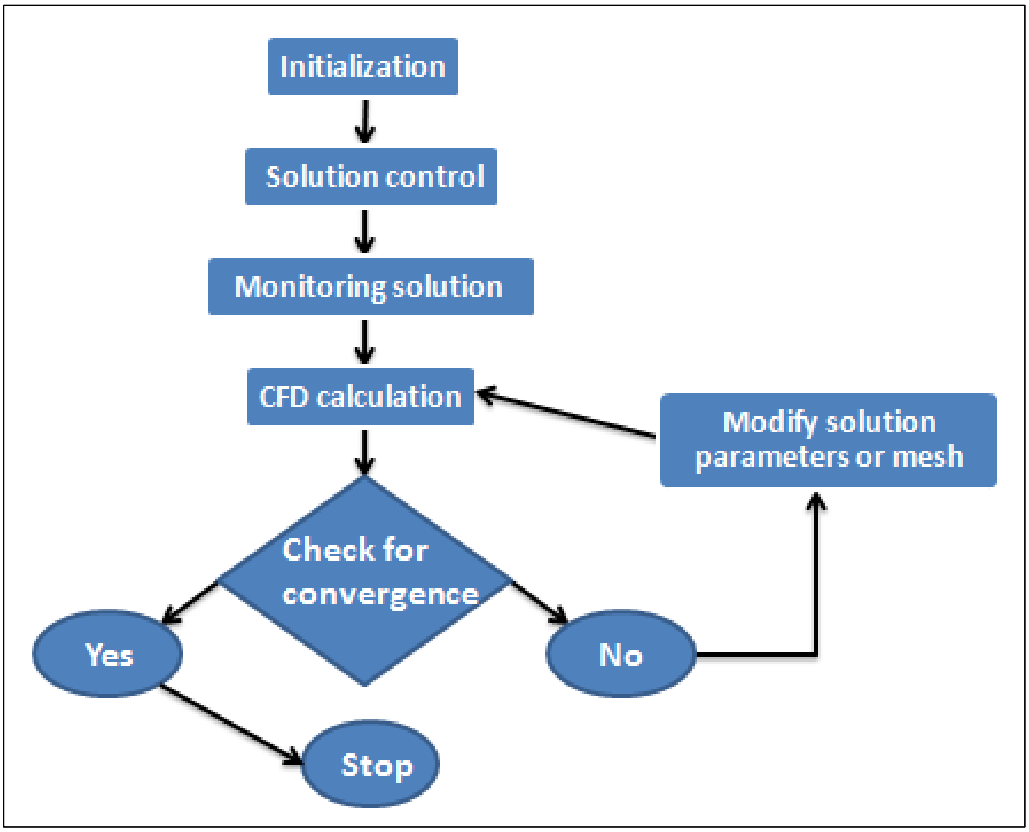

Figure 3.3: Flow chart of initialization and solution control

3.5 FLUID FLOW (CFX) IN CFD ANSYS WORKBENCH

There are five elements in a single standalone system that need to be define in order to perform a perfect ANSYS CFX analysis, each of them has their own properties that must be defined in sequence. Fluid flow (CFX) analysis system is chosen in the project schematic diagram where all five element of a single standalone system will be defined. Figure 3.4 shows the standalone system in the project schematic diagram.

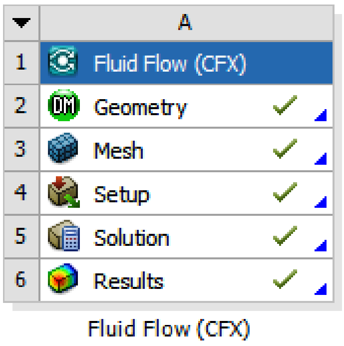

Figure 3.4: A standalone system

3.5.1 Geometry

The first element in a standalone system is the geometry component, the desired length unit for the geometry of the simulation to run is in millimeter. Then, import the external geometry file that is a ready CAD model of fuel injector and combustion chamber into the Design Modeler and generate the geometry file. The injector nozzle CAD drawing with 0.2mm diameter is shown in Figure 3.5, a green tick will appear besides the geometry icon if the process is successful.

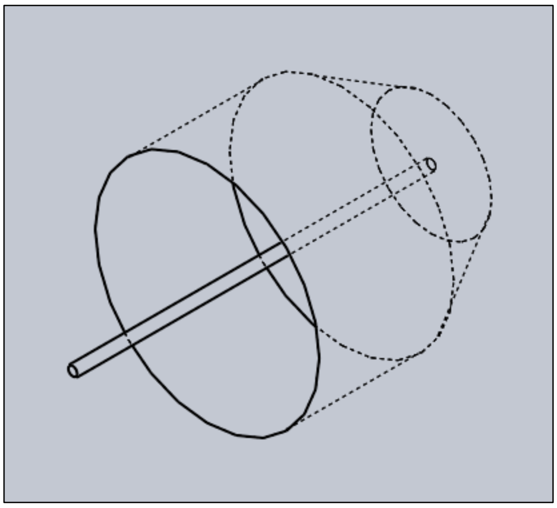

Figure 3.5: Injector nozzle CAD drawing

Dimensions of CAD model:

- Nozzle diameter: 0.2mm
- Bore: 110mm
- Stroke: 125mm

3.5.2 Mesh

The second element in a standalone system is the mesh component, in this component the imported geometry file will go through the meshing process. The sizing of each mesh cell should be suitable with the whole geometry dimension. Under Mesh component in the project outline display, the sizing option will be define as follow:

- Relevance center: Fine
- Min size: 4.e-005m (4×10^{-5}m)
- Max face size: 4.e-003m (4×10^{-3}m)
- Max size: 8.e-003m (8×10^{-3}m)

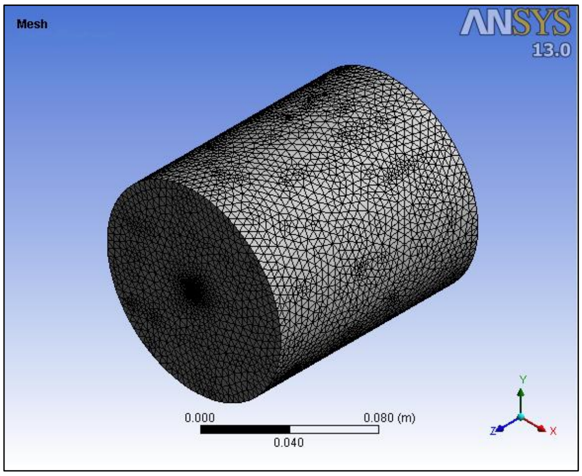

Figure 3.6: Complete mesh generated of the geometry model

After defining the sizing options, click the update button and a window at the bottom will shows the stage at which the Mesher package is in mesh generation process. Figure 3.6 shows the generated mesh of the geometry file.

3.5.3 Setup

The third element in a standalone system is the setup component, in this component all fluid flow properties will be define and inserting all boundary condition under the default domain.

1. The first step is to edit the default domain under Flow Analysis 1, remove the default item under Fluid and Particle Definition and add a new item named as "Gasoline". Then select the new item's material from the extended list as "$C_8H_{18}l$", this is the basic chemical structure of gasoline in liquid state.

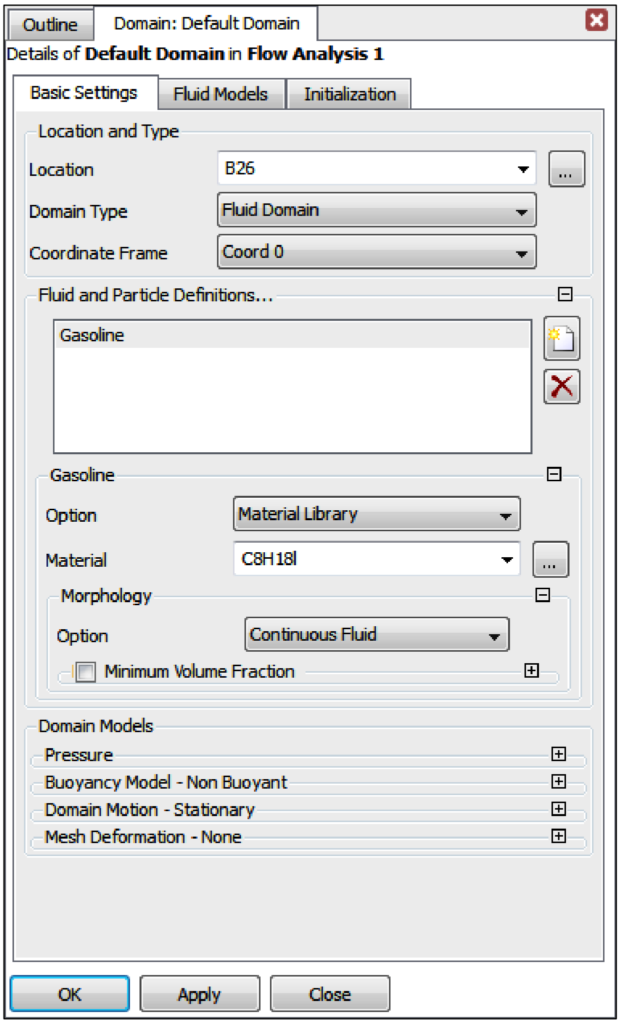

Figure 3.7: Default domain outline

Figure 3.7 shows the default domain outline that has been defined, the entire component under the Domain Models does not being altered as they are in a default options setting. Click Ok button to save and apply the new material properties setting.

2. The second step is to insert the inlet boundary condition by clicking the boundary button and rename it as "Inlet". The boundary type will be define automatically as Inlet and then under Location setting, click the inlet surface on the geometry. Under Boundary Details, the Mass and Momentum option is define as "Total Pressure (stable)" with 100bar relative pressure. Then the Flow Direction option is define as "Acting Normal to Boundary Condition" and the Turbulence option is define as "High Intensity" with 10% vale.

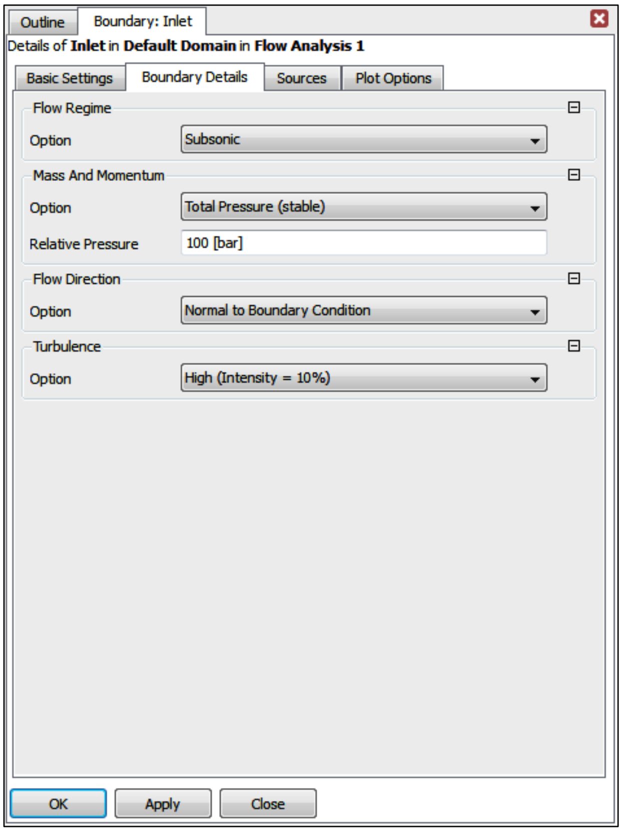

Figure 3.8: Inlet Boundary Condition

Figure 3.8 shows the inlet boundary condition outline, it is define as the fluid flow entering the domain as total pressure in stable condition with 100bar relative pressure value. Click Ok button to save and apply the inlet boundary condition setting.

3. The third step is to insert the opening boundary condition by clicking the boundary button and rename it as “Opening”. The boundary type will be define automatically as Opening and then under Location setting, click the opening surface on the geometry. The opening surface is on the combustion chamber part excluding the surface that is attached with the nozzle tip. Under Boundary Details, the Mass and Momentum option is define as “Opening Pres. and Dirn” with 101325Pa relative pressure, this would apply atmospheric condition to the domain. Then the Flow Direction option is define as “Acting Normal to Boundary Condition” and the Turbulence option is define as “Medium Intensity” with 5% value.

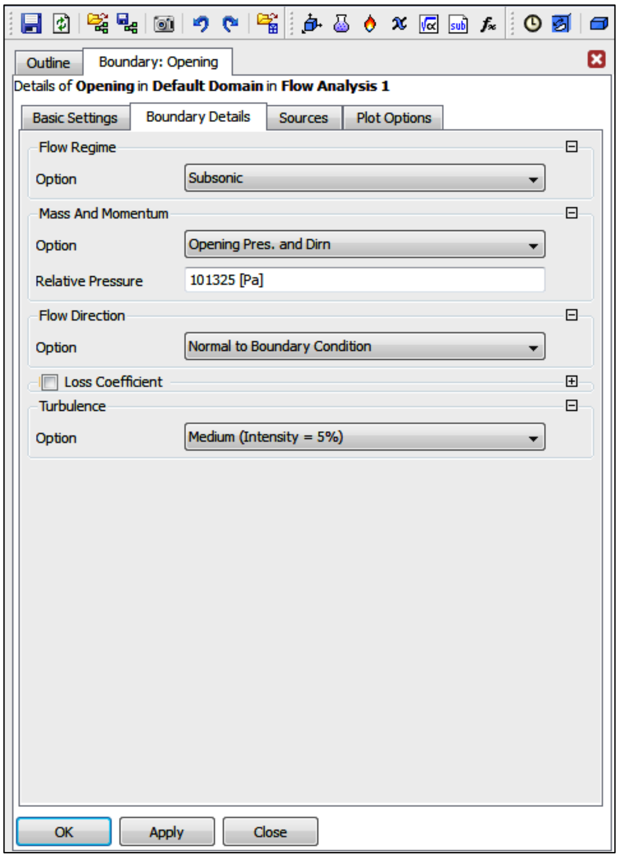

Figure 3.9: Opening Boundary Condition

Figure 3.9 shows the opening boundary condition outline, it is define as the fluid flow will experience atmospheric condition after leaving the nozzle. Click Ok button to save and apply the opening boundary condition setting.

4. The forth step is to insert the wall boundary condition by clicking the boundary button and rename it as “Wall”. The boundary type will be define automatically as Wall and then under Location setting, click the entire surface on the geometry model excluding Inlet and Opening surface. The wall boundary condition is acting as solid barriers to ensure that there is no leaking of fluid and the fluid flow only remains in the control domain. Figure 3.10 shows the wall boundary condition outline.

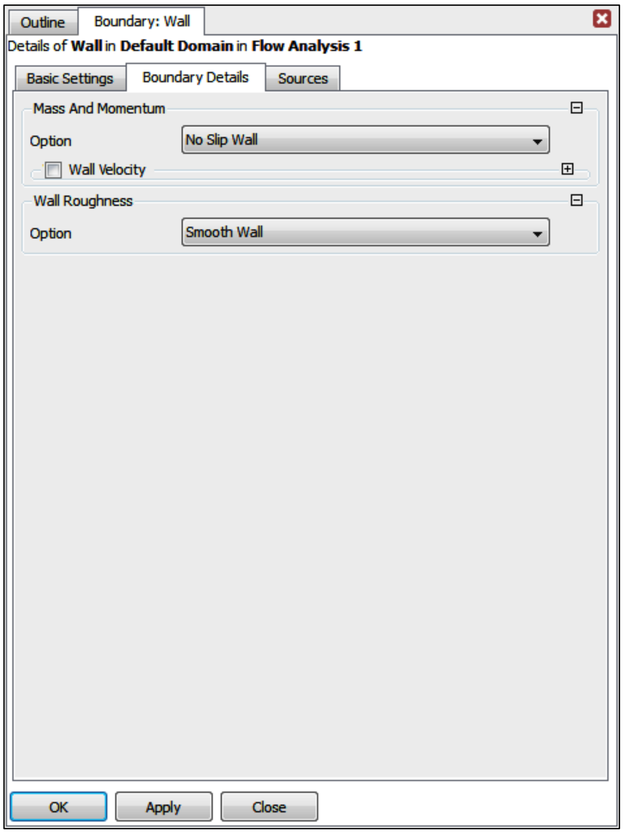

Figure 3.10: Wall Boundary Condition

5. The next step is to change the maximum iterations under Convergence Control in Solver Control component, default setting of maximum iterations is 100 but the convergence solver does not reach the residual target that is 1.e-4 (1×10^{-4}).

The maximum iteration is changed to 150 for better result although it is not reaching the residual target. Figure 3.11 shows the solver control setup in Flow Analysis 1.

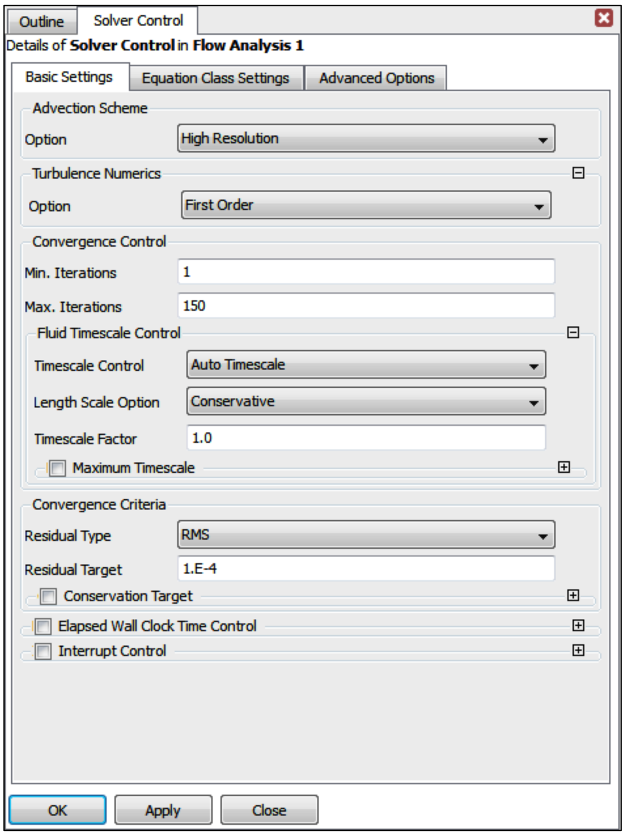

Figure 3.11: Solver Control Setup

6. The last step is to define the Global Initialization component, this component will be defined to determine the initial condition of the simulation. For spray simulation, the velocity type will be defined as "Cylindrical" since the nozzle and combustion chamber are cylindrical in shape while the turbulence option is set to be high intensity with 10% value. Figure 3.12 shows a complete setup definition on the geometry model, check that there is a green tick beside the setup icon. Then proceed to the next element in the standalone system.

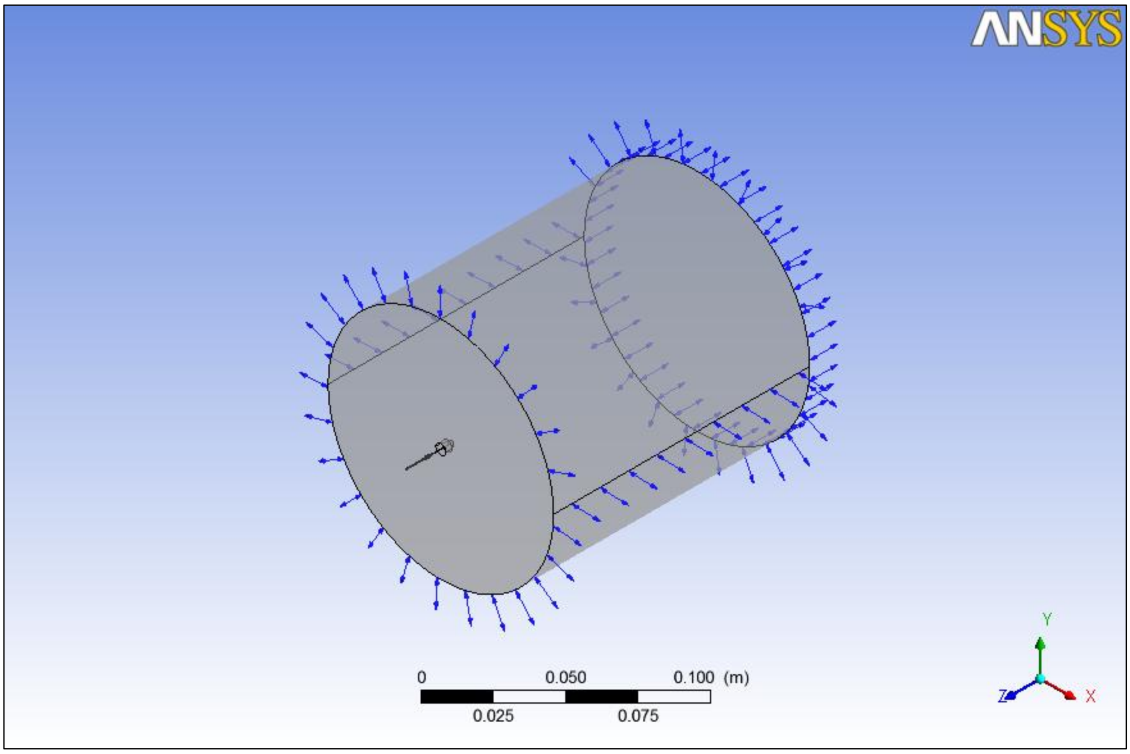

Figure 3.12: Complete setup of the boundary conditions on the geometry model

3.5.4 Solution

The forth element in a standalone system is the solution component, in this component the iteration calculation is solved until it reach a specific residual target but in some cases the iteration calculation can be stop if the convergence is near to the residual target or limited by the maximum iteration number. In the solver, a graph will show as Figure 3.13 where it illustrates the convergence in near to the residual target. Check that there is a green tick beside the solution icon, it means that the iteration calculation has been completely solved. Then proceed to the next element in the standalone system.

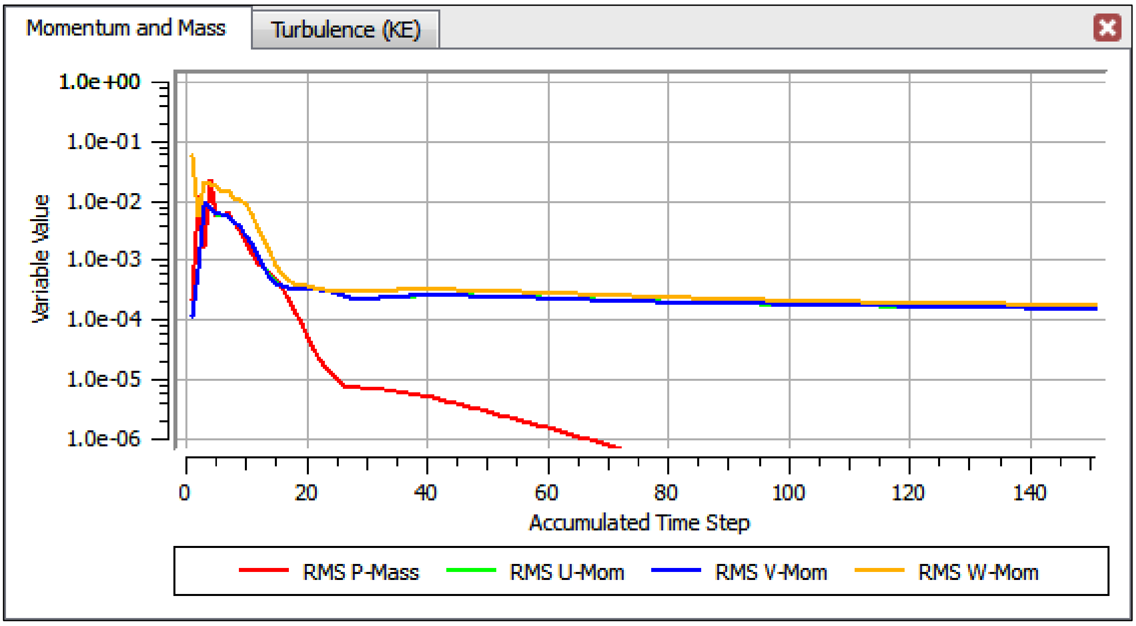

Figure 3.13: A complete iteration calculation

3.5.5 Results

The last element in a standalone system is the results component, it is capable to show the result of iteration calculations in visualized form. With the right method, the fluid flow properties or spray characteristics can be seen in the 3D viewer. There are few steps before the visualized data can be generated, the steps are listed as follows:

1. Inserting plane - Right click User Locations and Plots icon, then go to insert then location and select Plane. Let the default name be “Plane 1” and chose “All Domains” for domain optians and “YZ Plane” for method options then click apply button. The plane will appear according the the Figure 3.14.

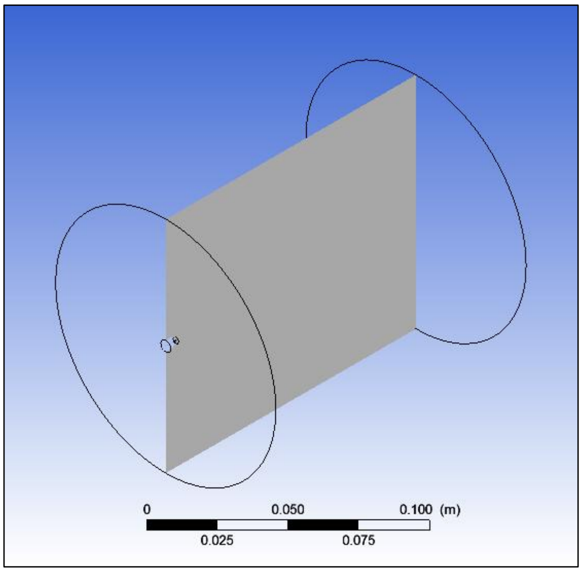

Figure 3.14: YZ plane on the geometry model

2. Inserting contour - Right click User Locations and Plots icon, then go to insert then from the drop down list select “Contour”. Let the default name be “Contour 1” and chose “Plane 1” for locations options and “Velocity w” for variable options. Then go to the Range and select from the drop down list “Global” and “Transparency” for Color Map options. Go to the (# of Contours) and enter 100 into the input cell, the velocity contour will appear according to the Figure 3.15.

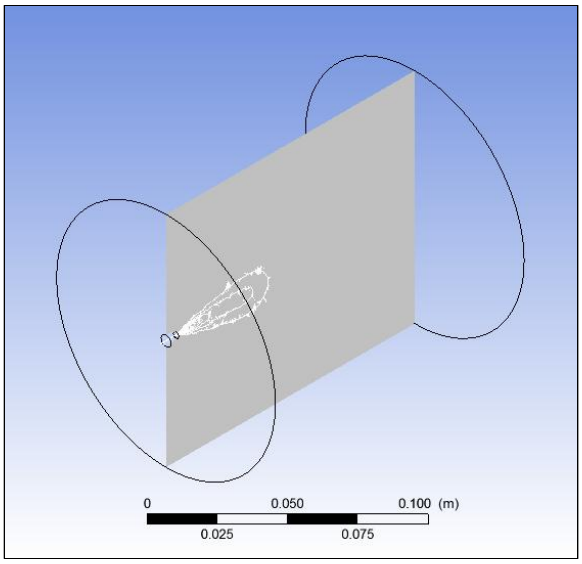

Figure 3.15: Velocity contour on Plane 1

For much clear and better view of the velocity contour, right click on the 3D viewer and go to the Predefined Camera and select "View from +X" from the drop down list. Since the goemetry and mesh setting will be the same, the standalone system can be done in series system as shown in Figure 3.16, only the Inlet pressure will be change to 150bar, 200bar and 250bar. Based on the same procedure, the geometry model for nozzle diameter 0.3mm and 0.4mm are used in to perform the simulation analysis.

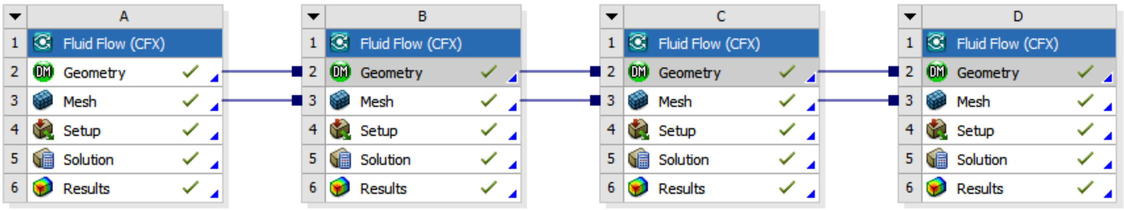

Figure 3.16: Series of standalone system

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

This section will present and discuss the result from all the simulations done for different injection pressure of 100bar, 150bar, 200bar and 250bar on three nozzle diameters of 0.2mm, 0.3mm and 0.4mm. Based on each simulation, what is the effect of various injection pressure to the spray angle and spray tip penetration will be discuss. With the aid of ANSYS CFX software, the simulations have been completed using basic gasoline ($C_8H_{18}l$) properties as liquid phase. Graph of spray angle versus injection pressure and spray tip penetration versus injection pressure for each nozzle tip diameter will be shown.

4.2 VALIDATION

Based on the American Institute of Aeronautics and Astronautics (AIAA), the process of determining the degree to which the computational approach or simulation model is accurate and follow the real world situation is known as validation. In this study the simulations done ware based on basic spray simulation program in which the spray fluid in this case is the gasoline will experienced laminar flow before experiencing turbulent flow as it moves away from the nozzle tip. The high pressure from the inlet will forced the gasoline fuel to burst or dispersed as it move towards a lower pressure medium.

In a validation process, an accuracy of the results are compared to other source of data such as experimental or related study approach. Even though an experimental data is has been used as references, but it still have uneven data due to errors while performing the experiment. Hence there will be unequal data formation between computational approach and real experimental method.

4.3 SIMULATIONS RESULTS

Using ANSYS CFX software, the gasoline fuel spray simulations were done on three nozzle diameters of 0.2mm, 0.3mm and 0.4mm where the same mesh size is used to get the uniform results between each nozzle diameter and injection pressure.

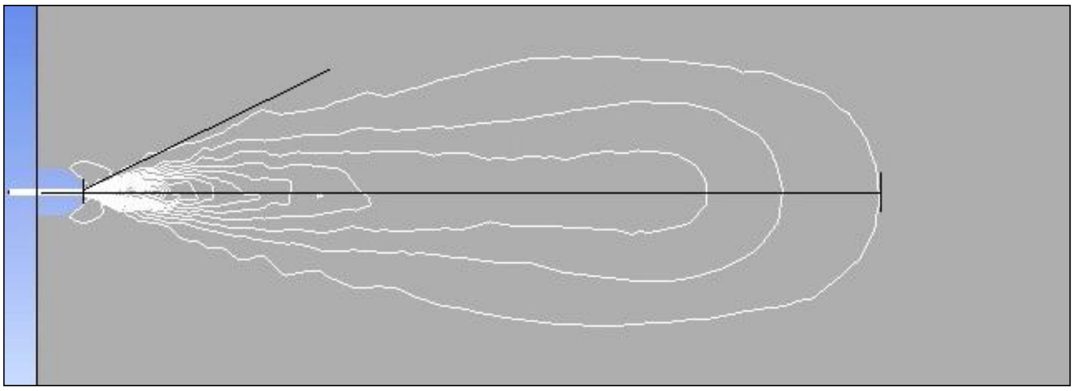

Figure 4.1: Spray tip penetration and spray angle measurement

Table 4.1: The results of different injection pressure on the spray structure

Injection pressure (bar)	0.2mm nozzle diameter		0.3mm nozzle diameter		0.4mm nozzle diameter	
	Spray tip penetration (mm)	Spray angle (°)	Spray tip penetration (mm)	Spray angle (°)	Spray tip penetration (mm)	Spray angle (°)
100	45.2778	22.50	68.9362	23.50	81.8182	24.00
150	46.1972	22.50	77.8571	24.00	83.8983	25.50
200	46.5714	22.75	78.0952	26.50	85.3448	25.75
250	46.8571	23.00	78.0476	27.00	86.5487	26.00

Figure 4.1 illustrates the spray structure, as mention in Chapter 2 the spray tip penetration is measured from the nozzle tip until the end of the spray propagation and

the spray angle is measured form the center line of the spray propagation to the edge of the spray structure. With proper scale and clear simulation's results, both desired parameters are measured manually by using ruler for spray tip penetration and protector for the spray angle. Both parameters are then classified according to their nozzle tip diameter and injection pressure value and included in an appropriate table as shown in Table 4.1. The data has been measured multiple times to avoid parallax error while taking the reading for each simulation result.

4.3.1 0.2mm Nozzle Diameter

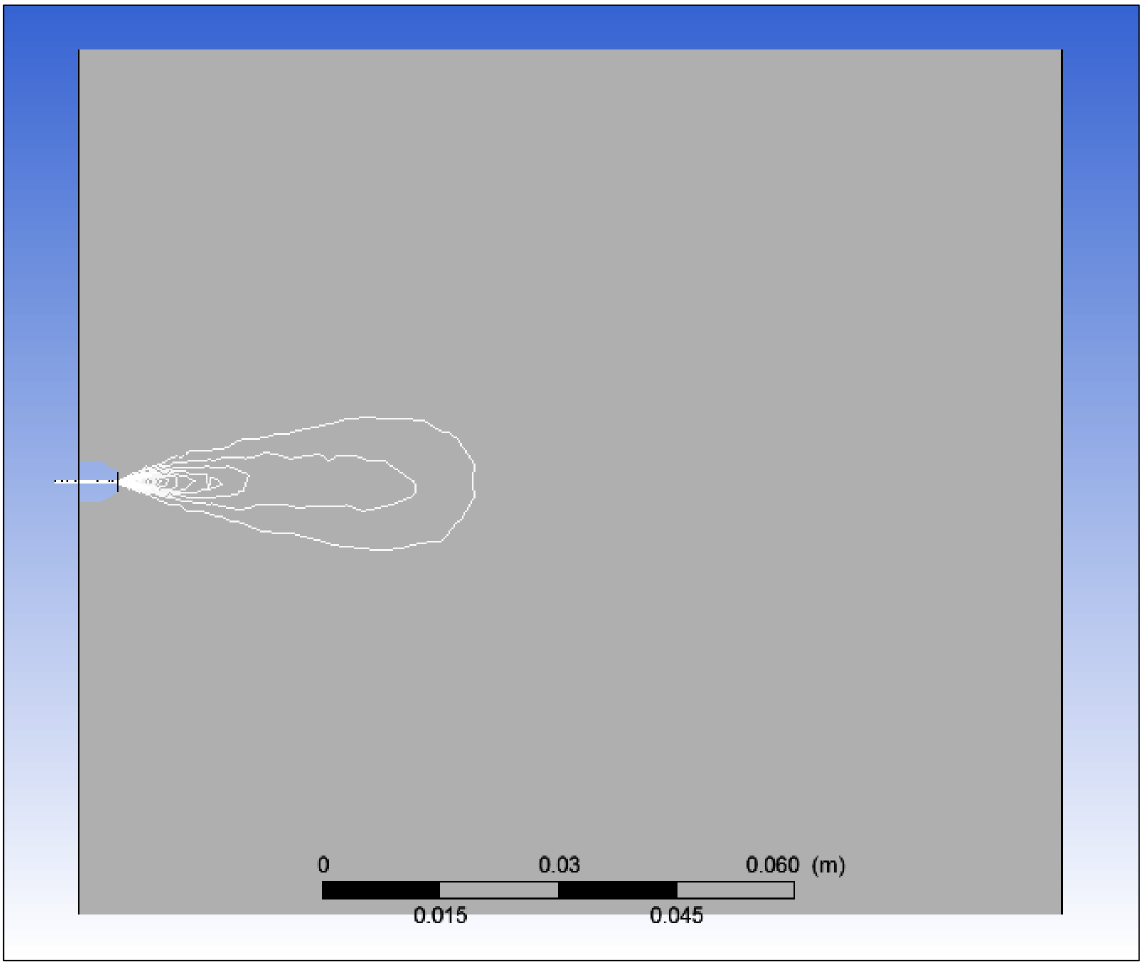

Figure 4.2: 0.2mm nozzle diameter spray structure

Figure 4.2 shows the spray structure of gasoline fuel in 0.2mm nozzle tip diameter inside the combustion chamber. Based on Figure 4.3, the spray tip penetration is increased as the injection pressure increased, the depth difference between injection pressure value of 100bar to 150bar is the largest compared to depth difference of 150bar to 200bar and 200bar to 250bar. The first injection pressure indicates the shortest penetration depth among the other injection pressure while the last injection pressure indicates the longest penetration depth.

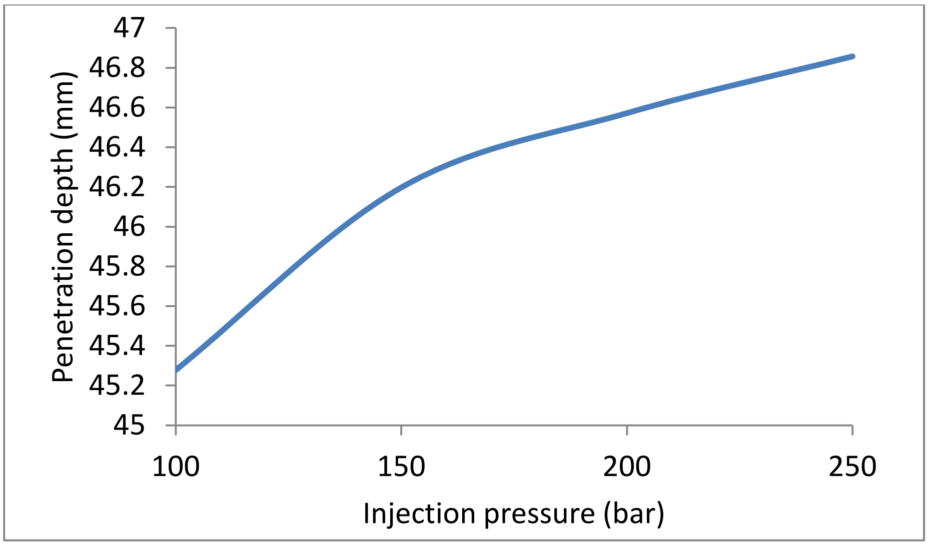

Figure 4.3: Graph of penetration depth versus injection pressure in 0.2mm nozzle tip diameter

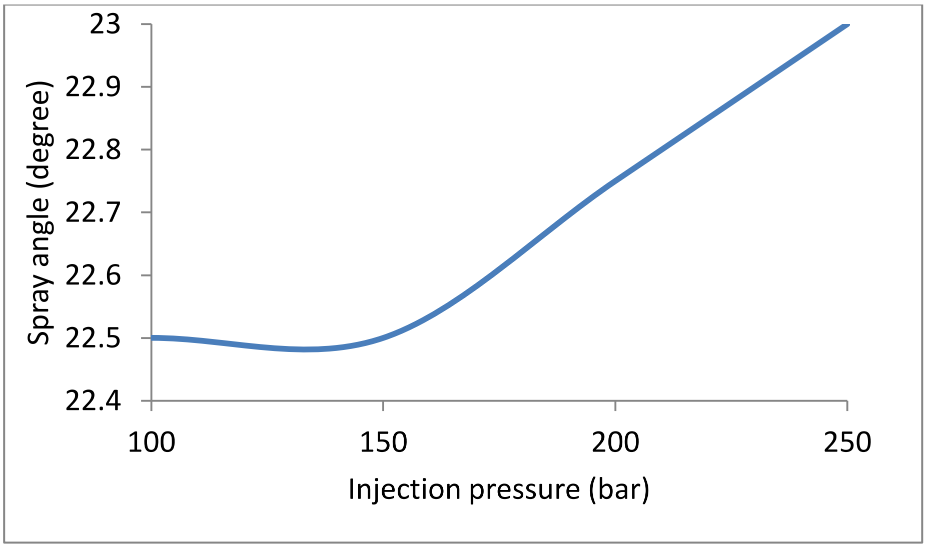

Figure 4.4: Graph of spray angle versus injection pressure in 0.2mm nozzle tip diameter

Based on Figure 4.4, the spray angle is wider on injection pressure of 150bar and above. The angle difference between 100bar to 150bar does not shows any difference while the angle difference between 150bar to 200bar and 200bar to 250bar show uniform enhancement. The spray angle does not changed when until the injection pressure reach 150bar. The injection pressures of 100bar and 150bar indicates the smallest spray angle among the other injection pressure while the highest injection pressure indicates the widest spray angle.

4.3.2 0.3mm Nozzle Diameter

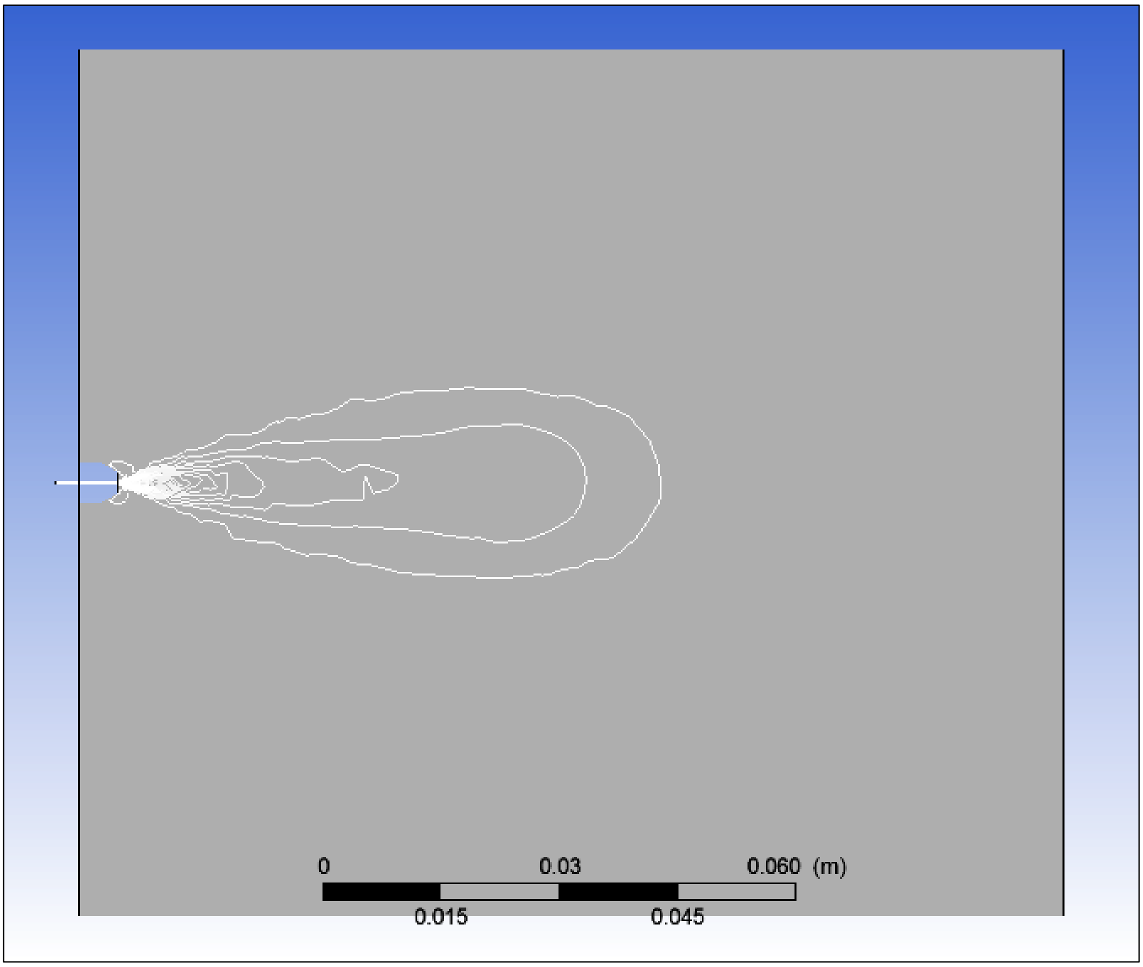

Figure 4.5: 0.3mm nozzle diameter spray structure

Figure 4.5 shows the spray structure of gasoline fuel in 0.3mm nozzle tip diameter. Based on Figure 4.6, the spray tip penetration is increased as the injection pressure increased, we can see that the depth is at its maximum when the injection pressure reach 150bar and becomes slightly constant while the minimum depth is at the minimum injection pressure.

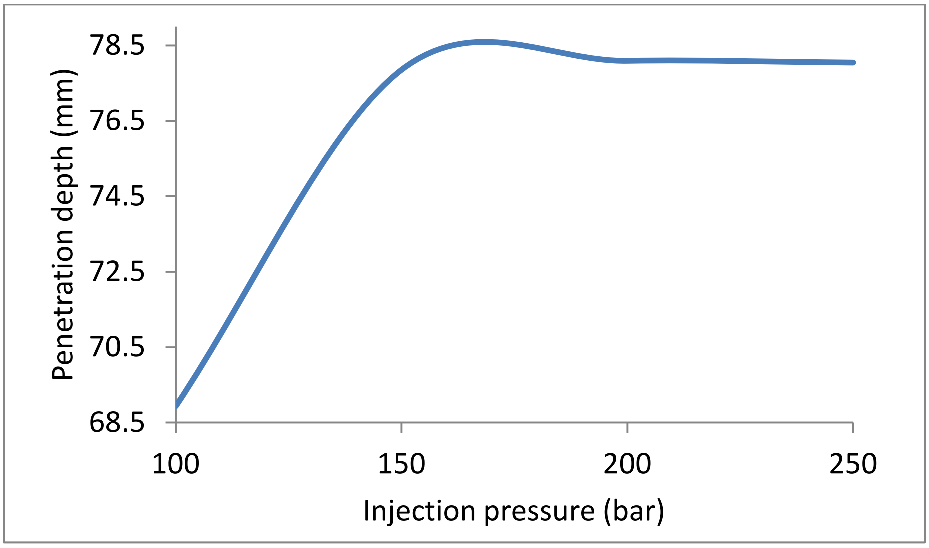

Figure 4.6: Graph of penetration depth versus injection pressure in 0.3mm nozzle tip diameter

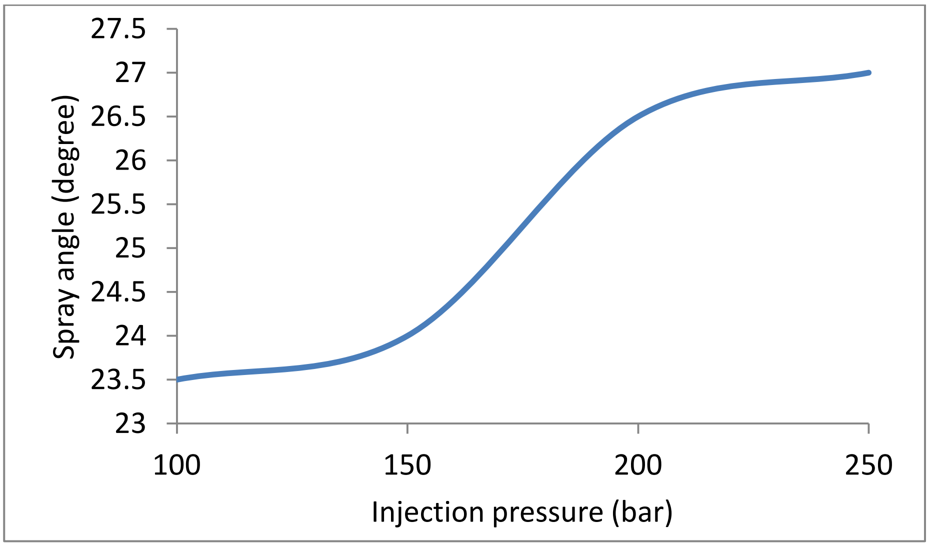

Figure 4.7: Graph of spray angle versus injection pressure in 0.3mm nozzle tip diameter

Based on Figure 4.7, the spray angle is the widest between injection pressure of 150bar to 200bar. It is shown that the differences between the first and second injection pressure has small value, the same condition to the third and forth injection value. These mean that the spray angle is almost the same for the low and high injection pressure, it will experienced major differences for midst of the injection pressure range. As the injection pressure increased the spray angle also getting wider but does not show uniform enhancement.

4.3.3 0.4mm Nozzle Diameter

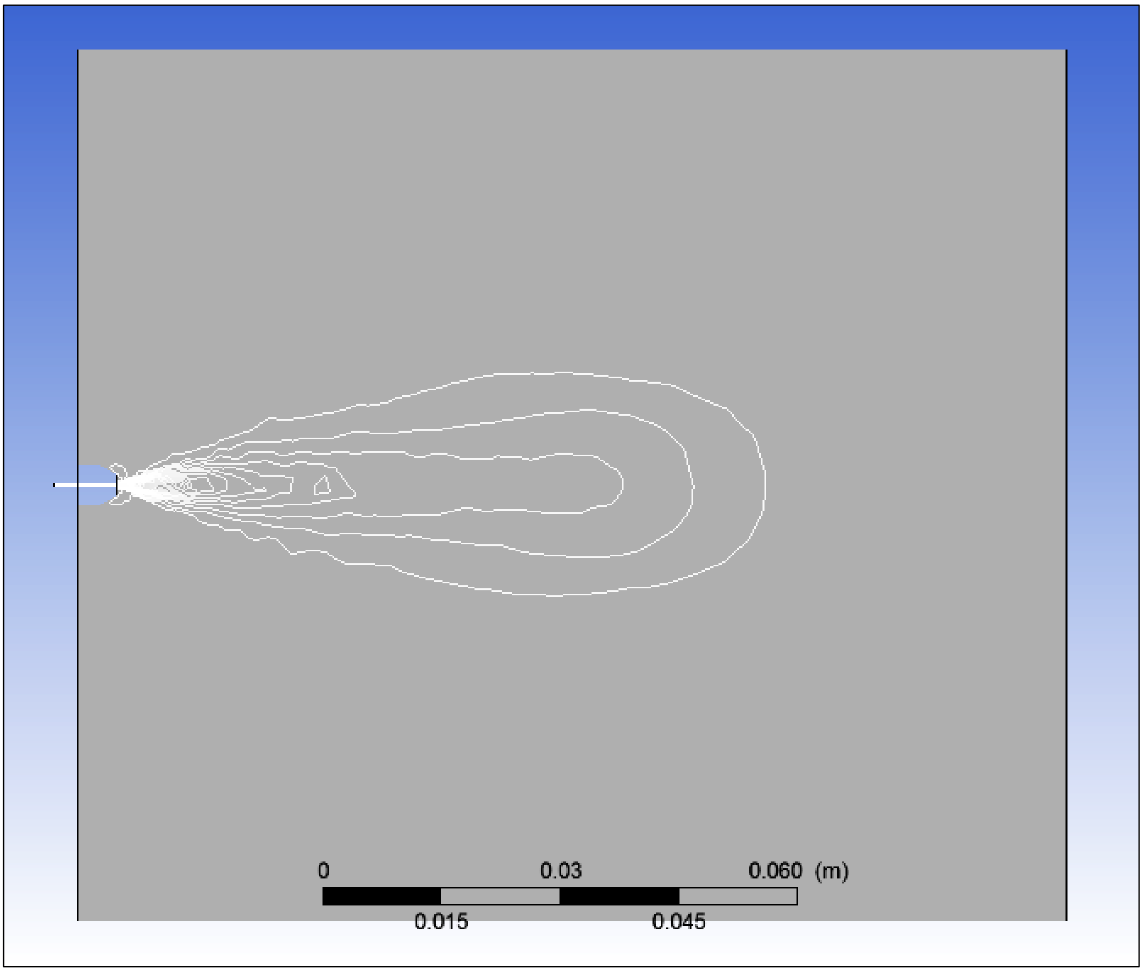

Figure 4.8: 0.4mm nozzle diameter spray structure

Figure 4.8 shows the spray structure of gasoline fuel in 0.4mm nozzle tip diameter. Based on Figure 4.9, the spray tip penetration is increased as the injection pressure increased, we can see that the depth is at its maximum when the injection pressure reach is maximum value. Compared to the other nozzle diameter, 0.4mm nozzle tip shows a uniform enhancement of penetration depth from the lowest injection pressure to the highest injection pressure. The depth differences between each injection pressure are nearly the same.

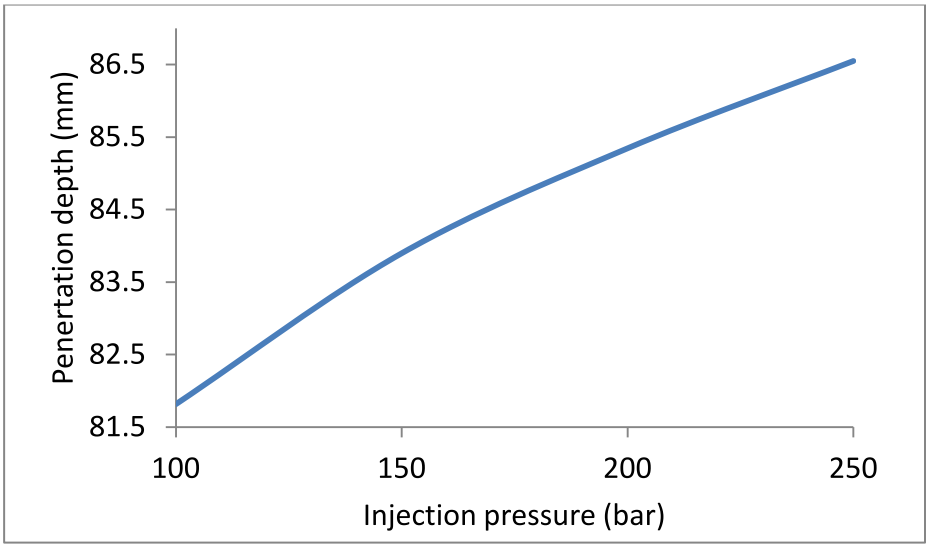

Figure 4.9: Graph of penetration depth versus injection pressure in 0.5mm nozzle tip diameter

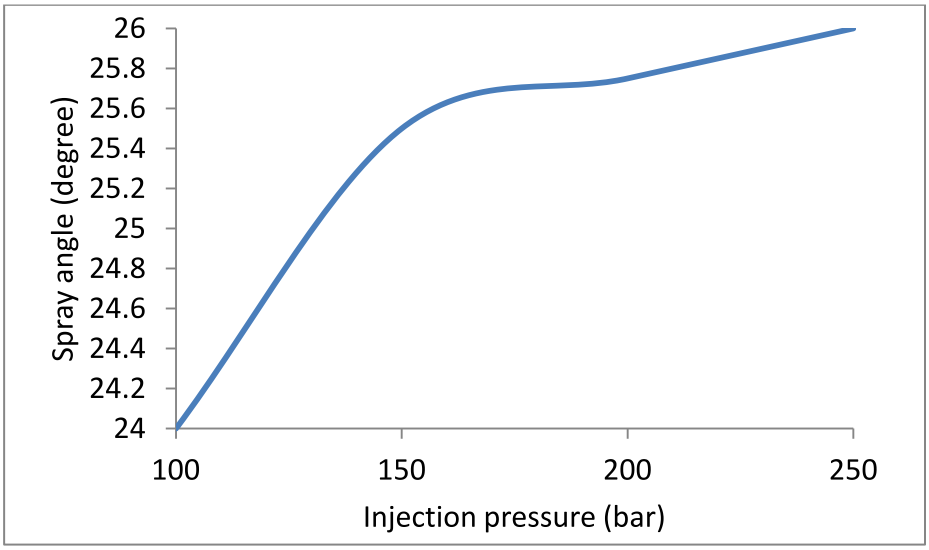

Figure 4.10: Graph of spray angle versus injection pressure in 0.4mm nozzle tip diameter

Based on Figure 4.10, the spray angle is wider as the injection pressure increased, according to the theoretical understanding the spray angle will always increase with the injection pressure and also the nozzle tip diameter. With nozzle diameter 0.4mm, the spray angle will develop but does not the same compared to the other nozzle diameters. The differences of spray angle between the first and second injection pressure is the largest compared to the rest of the injection pressure. after reaching 150bar injection pressure, the spray angle will show small differences as the injection pressure increased.

4.4 DIFFERENCE IN SPRAY STRUCTURE FOR VARIOUS NOZZLE DIAMETER

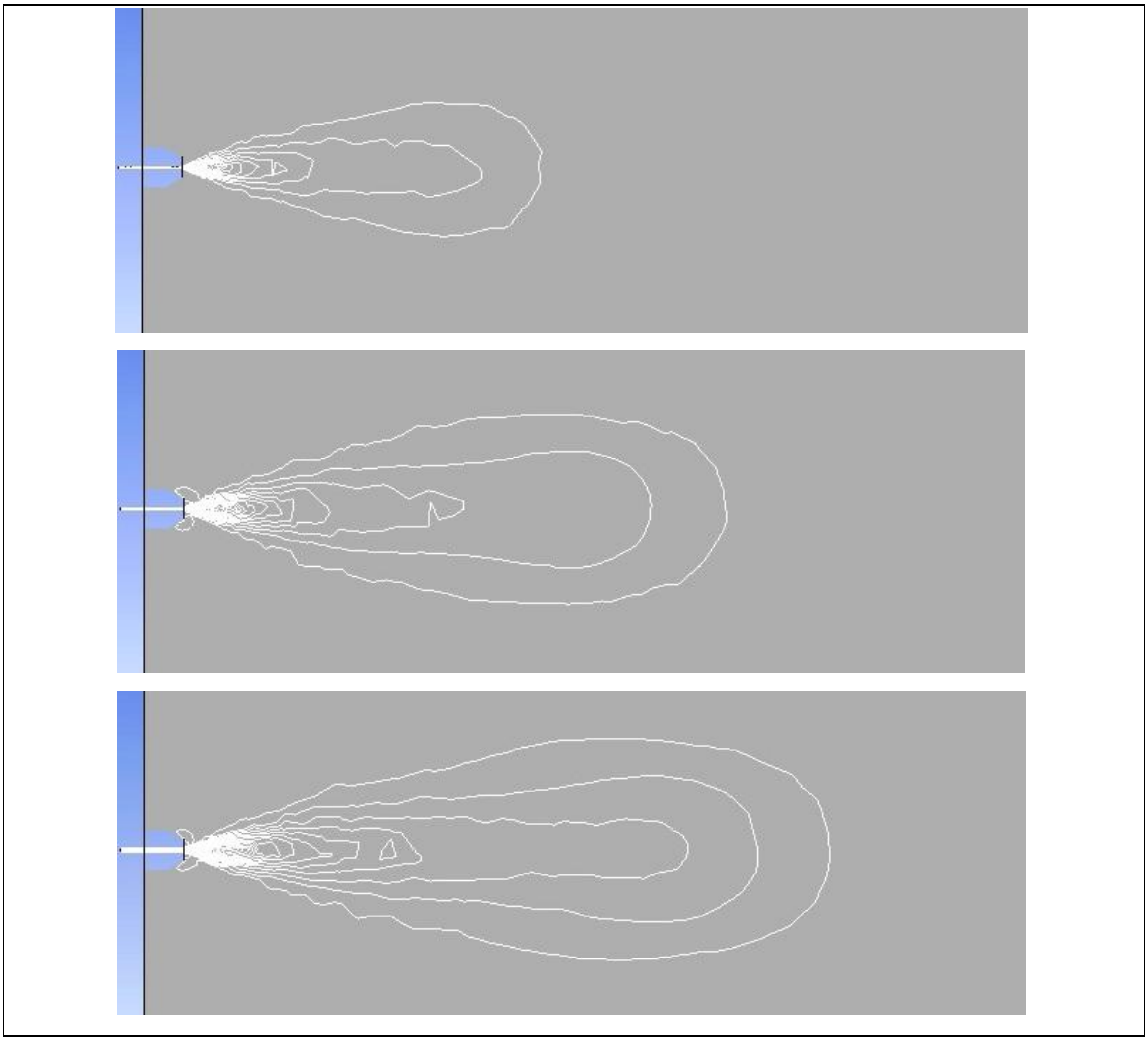

Figure 4.11: Spray structure of gasoline fuel for different nozzle diameter

Figure 4.11 shows the spray structure (penetration depth and spray angle) of gasoline fuel for different nozzle diameter. The penetration depth of gasoline fuel spray is highly influenced by the nozzle tip diameter, the Figure 4.12 shows the difference on penetration depth versus injection pressure for 0.2mm, 0.3mm and 0.4mm nozzle diameter. With small difference in nozzle diameter, the penetration depths are varies because larger opening dimension or nozzle diameter will result in longer spray structure. When the nozzle diameter becomes larger, the percentage of fluid flow is much bigger compared to small nozzle diameter.

We can see from the Figure 4.12 that 0.4mm nozzle diameter has the longest penetration depth for each injection pressure while 0.2mm diameter nozzle diameter had the shortest penetration depth for each injection pressure. Only 0.3mm nozzle diameter will shows non-uniform penetration depth development as the depths are constant after injection pressure reach 150bar. The higher injection pressure resulted in the longer spray tip penetration along the combustion chamber, but the penetration depth is also depends on the nozzle tip diameter.

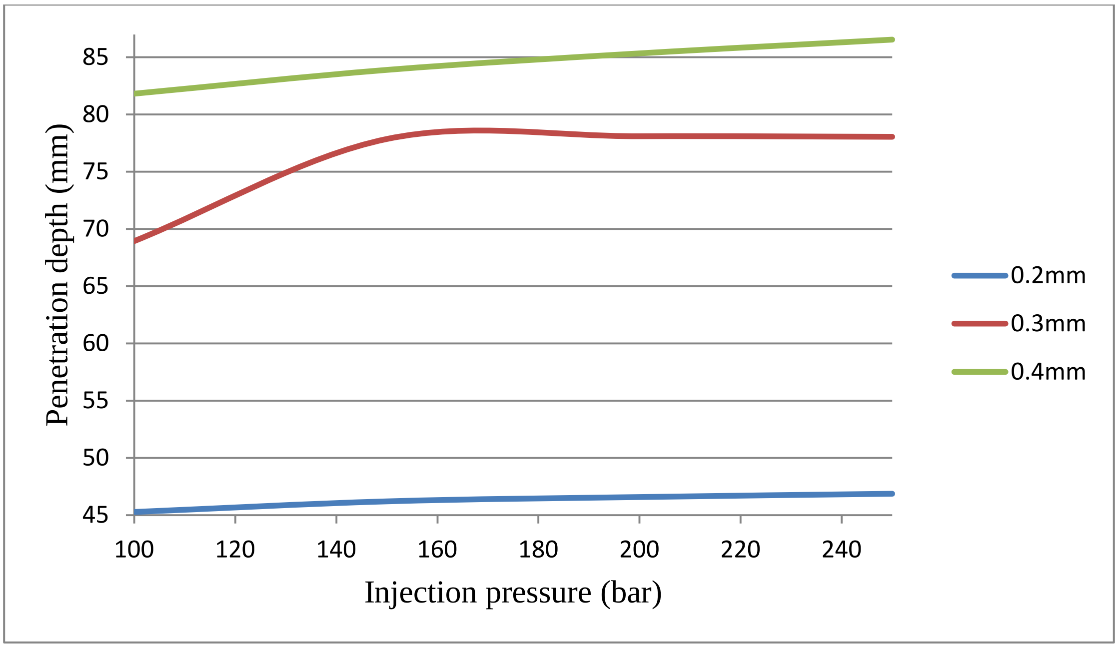

Figure 4.12: Variance of penetration depth between nozzle diameter and injection pressure

The spray angle of gasoline fuel spray is highly influenced by the nozzle tip diameter, the Figure 4.13 shows the difference on spray angle versus injection pressure for 0.2mm, 0.3mm and 0.4mm nozzle diameter. With small difference in nozzle diameter, the spray angle are varies because larger opening dimension or nozzle diameter will result in wider spray structure. Based on Figure 4.13 we can see that 0.3mm nozzle diameter will has the widest spray angle even though it is not the largest nozzle diameter in this study. The simulation results showed a small effect of injection pressure to the spray angle for 0.2mm nozzle diameter, but effect of injection pressure to the spray angle for 0.3mm and 0.4mm nozzle diameter is high. According to the related experimental approach, it is conclude that the spray angle is mainly affected by the liquid viscosity, air-fuel ratio and density.

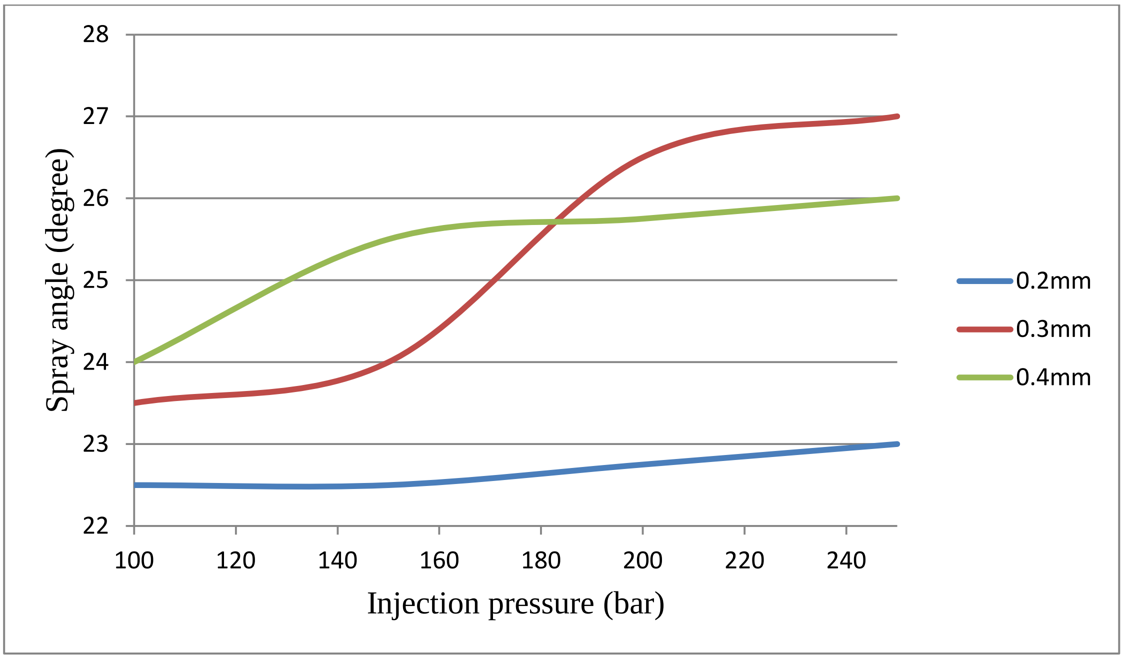

Figure 4.13: Variance of spray angle between nozzle diameter and injection pressure

The overall spray structure in this study is exposed to the atmospheric condition, in a real world situation the spray will occur in the combustion chamber with much higher pressure compared to the atmospheric pressure. The turbulence level of the spray flow will differ if the spray is done in the real world condition and hence, both penetration depth and spray angle will result is different characteristics as it flows into the combustion chamber.

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 CONCLUSIONS

Based on the objectives stated in Chapter 1, the computational study of fuel spray structure has been done completely. The gasoline spray structure on penetration depth and spray angle with various injection pressure have been determined by using ANSYS CFX analysis software. This study has achieved the objectives which is to perform a gasoline fuel spray simulation using computational approach and determine the effect of different injection pressure (bar) on the spray structure. The spray structure behavior and it characteristics at the exit of the nozzle tip and in the combustion chamber for all three nozzle diameter at different injection pressures were studied and compared

It is determined that the effect of different injection pressure on the penetration depth is that the higher the injection pressure, the longer the penetration depth will be. The simulations have been done to three nozzle diameter, each of the nozzle fulfill the theoretical penetration depth criteria. Nozzle diameter of 0.2mm and 0.4mm show uniform penetration depth development as the injection pressure increased, nozzle diameter of 0.3mm shows that the penetration depth is almost constant when the injection pressure reached 150bar. The lowest injection pressure that is 100bar shows the shortest penetration depth for each nozzle diameter, while the highest injection pressure that is 250bar shows the longest penetration depth for each nozzle diameter except for the 0.3mm diameter. There are many factors that might be affecting the penetration depth such as liquid viscosity, density, surface tension, injection pressure and also the outside pressure or ambient pressure.

Other than penetration depth, the effect of different injection pressure on the spray angle has been determine. The effect is the same as the effect on the penetration depth that is the higher the injection pressure, the wider the spray angle will be. Results from the spray simulations does not fulfill the theoretical spray angle coverage criteria as the spray angle will diverge as the spray development flows much further from the nozzle tip. Each nozzle diameter does not show a uniform spray angle development as the spray angle for 100bar and 150bar injection pressure on 0.2mm nozzle diameter are constant. The same situation occurred on 0.3mm and 0.4mm nozzle diameter where the spray angle will be constant at certain injection pressure level.

The lowest injection pressure that is 100bar shows the smallest spray angle for each nozzle diameter while the highest injection pressure that is 250bar shows the widest spray angle for each nozzle diameter. The factors that might be affecting the spray angle development is almost the same for penetration depth, the overall result indicate that the bigger the nozzle tip diameter will result in wider spray angle.

5.2 RECOMMENDATIONS

Based on this study, there are few recommendations can be apply in order to improve the result and understanding of fuel spray structure focused on the penetration depth and spray angle. Even though computational approach is used in solving engineering problems, the result is not totally acceptable for real world situations.

There are still some important aspect that need to be highlight to do a study on the spray structure. In order to obtain much better result in investigating the spray structure including penetration depth and spray angle, a real experiment should be carry out. The spray propagation through the air can be seen clearly with real experiment and with the aid of suitable equipment, the droplet properties on the dilute spray medium can be determined.

Another recommendation that can be apply to improve this study in to increase the injection pressure range, in this study there are only four different injection pressures. The pressure range should be bigger to determine and to obtain a clear understanding about the effect of injection pressure on the spray structure, the spray characteristics for wide range of injection pressure might be different from the results in this study.

More parameters or boundary condition in the analysis can result in different findings about the spray characteristics. Since this study is related to the engine system, the simulation should used appropriate pressure value in the combustion chamber, the real engine condition can be apply to this study. The additional parameters that can be used are temperature in the combustion chamber, pressure in the combustion chamber, type of fuel and different nozzle tip shape.

REFERENCES

A. Heilig, M. Kaiser and F. Dinkelacker. 2011. Spray Analysis and Comparison of Diesel and Biodiesel-Methanol Blends. *24th European Conference on Liquid Atomization and Spray System.*

Ainul Ghurri1, Jae-Duk Kim, Hyung Gon Kim, Jae-Youn Jung and Kyu-Keun Song. 2012. The effect of injection pressure and fuel viscosity on the spray characteristics of biodiesel blends injected into an atmospheric chamber. *Journal of Mechanical Science and Technology*. **26**(9):2941-2947.

Baris Guler and Rizwan Ali. 2004. Understanding the behavior of Computational Fluid Dynamics. *High-Performance Computing*. Dell Power Solution:73-76.

F. Ommi, E. Movahednejad and K. Nekofar. 2008. Study of injection parameters on performance and fuel consumption in a port-fuel injected gasoline engine with experimental and theoretical methods. *Journal of Engineering*:39-50.

G.M. Faeth, L.P. Hsiang and P.K. Wu. 1995. Structure and Breakup Properties of Sprays. *Department of Aerospace Engineering.* **21**:99-127.

Holger Grosshans. Robert-Zoltan Szasz and Laszlo Fuchs. 2011. Full Spray Simulation - Coupled Volume of Fluid and Lagrangian Particles Tracking methods. *24th European Conference on Liquid Atomization and Spray System.*

Jesper Madsen. 2006. Computational and Experimental Study of Sprays from the Breakup of Water Sheets. *Chemical Fluid Flow Processes Ph. D Thesis.*

Lionel Martinez. Adlene Benkenida and Benedicte Cuenot. 2010. A model for the injection boundary conditions in the context of 3D simulation of Diesel Spray: Methodology and validation. *Journal of Fuel*. **89**:219-228.

L. Zigan, I. Schmitz, A. Flugel, M. Wensing and A. Leipertz. 2010. Structure of evaporating single- and multi-component fuel sprays for 2nd generation gasoline direct injection. *Journal of Fuel.* **90**:348-363.

Mingheng Li and Panagiotis D.C. 2006. Computational study of particle in-flight behavior in the HVOF thermal spray process. *Journal of Chemical Engineering Science.* **61**: 6540-6552.

Rossella Rotondi and Gino Bella. 2005. Gasoline direct injection spray simulation. *International Journal of Thermal Sciences.* **45**:168-179.

R. Schmehl, G. maier and S. Wittig. 2000. CFD Analysis of Fuel Atomization, Secondary Droplet Breakup and Spray Dispersion in the Premix Duct of a LPP Combustor. *Eight International Conference on Liquid Atomization and Spray Systems.*

S. Hossaipour and A.R. Binesh. 2009. Investigation of fuel spray atomization in a DI heavy-duty diesel engine and comparison of various spray breakup models. *Journal of Fuel.* **88**:799-805.

Sung Wook Park and Rolf D. Reitz. 2009. A gas jet superposition model for CFD modeling of group-hole nozzle spray. *International Journal of Heat and Fluid Flow.* **30**:1193-1201.

T.N.C. Anand, A. Madan Mohan and R.V. Ravikrishna. 2012. Spray characterization of gasoline-ethanol blends from a multi-hole port fuel injector. *Journal of Fuel.* **102**:613-625.

Zhijun Wu, Zhiyong Zhu and Zhen Huang. 2006. An experimental study on the spray structure of oxygenated fuel using laser-based visualization and particles image velocimetry. *Journal of Fuel.* **85**:1458-1463.

APPENDIX A1

Gantt Chart for Final Year Project 1

PROJECT ACTIVITY	WEEKS													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Received FYP Title and discussed with supervisor	■	■	■											
Gathered related journals and books			■	■	■	■	■	■	■					
1.1 Introduction				■	■									
1.2 Problem Statement					■	■								
1.3 Objectives						■	■	■						
1.4 Scope of Study							■	■	■					
2.0 Literature Review								■	■	■				
3.0 Methodology									■	■	■	■		
Submitted Draft Report												■		
Prepared and Presentation FYP 1												■	■	■

APPENDIX A2

Gantt Chart for Final Year Project 2