A STUDY OF THE SPRAY CHARACTERISTIC FOR VALVE COVERED ORIFICE DIESEL NOZZLE INJECTOR USING CFD

AHMAD AIZUDDIN FAIZ BIN CHE JAAFAR

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **A STUDY OF THE SPRAY CHARACTERISTIC FOR VALVE COVERED ORIFICE DIESEL NOZZLE INJECTOR USING CFD**

SESI PENGAJIAN: **2012/2013**

Saya **AHMAD AIZUDDIN FAIZ BIN CHE JAAFAR (850113-11-5295)**
(HURUF BESAR)

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**	

Disahkan oleh:

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap:

24, LORONG PASIR PUTEH,
PASIR PANJANG,
21100 KUALA TERENGGANU.
TERENGGANU

Dr. ABDUL ADAM BIN ABDULLAH
(Nama Penyelia)

Tarikh: **27 JUNE 2013** Tarikh: : **27 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).

A STUDY OF THE SPRAY CHARACTERISTIC FOR VALVE COVERED ORIFICE DIESEL NOZZLE INJECTOR USING CFD

AHMAD AIZUDDIN FAIZ BIN CHE JAAFAR

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

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2013

EXAMINERS APPROVAL DOCUMENT

UNIVERSITI MALAYSIA PAHANG
FACULTY OF MECHANICAL ENGINEERING

I certify that the report entitled 'A Study of The Spray Characteristic for Valve Covered Orifice Diesel Nozzle Injector using CFD' is written by Ahmad Aizuddin Faiz Bin Che Jaafar. 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 with Automotive Engineering. I herewith recommend that it be accepted in fulfillment of the requirements for the degree of Bachelor of Engineering.

Signature:
Name of Examiner: MR WAN ANUAR WAN HASSAN.
Date:

SUPERVISOR'S DECLARATION

I hereby declare I have checked this report, which written by Ahmad Aizuddin Faiz Bin Che Jaafar, and in my opinion, this report project is adequate in terms of scopes and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature:
Name of Supervisor: DR. ABDUL ADAM BIN ABDULLAH
Position: LECTURER
Date:

STUDENT'S DECLARATION

I hereby declare that the work in this report is my own, except for quotations and summaries which have been duly acknowledged. The report has not been accepted for any other degree and is not concurrently submitted for award of other degree.

Signature:

Name: AHMAD AIZUDDIN FAIZ BIN CHE JAAFAR

ID Number: MH08083

Date:

ACKNOWLEDGEMENTS

Praise to Allah S.W.T that I am so grateful in preparing this project report. This project had taught me in many ways, something I couldn't get in classes. I was able to contact with many important people, researcher, lecturers, and engineers. They have contributed towards my understanding and thought in accomplishing this project. In particular, I am indebted and I want to express my sincere appreciation to my project supervisor Dr. Abdul Adam Bin Abdullah for encouragement, guidance, comments, and knowledge that he had taught to me.

I am also indebted to University Malaysia of Pahang for providing me with all the utility and equipment that I needs towards the completion of this project and report. Also to librarians at UMP, all the founders of web pages and Book author for giving me relevant literature and assistance. Not to forget to my sincere appreciation goes to all my colleagues that always supported me and assisting me in completing this project. Their view and guidelines are valuable and very useful without a doubt. A very thank you wishes that I can give for those who helping me for I wish I could put the entire name in this limited space. Last but not least, I am very grateful to all my family members for their support, advises and motivation for encouraging me not giving up whenever I feel down and hopeless.

This project wouldn't be succeeding with one man alone. I do hope for whom that does help me with this project may achieve success in their future life. Thank you.

ABSTRACT

Diesel engine performance and emissions are strongly coupled with fuel atomization and spray processes, which in turn are strongly influenced by injector flow dynamics. Modern engines employ micro-orifice with different orifice designs. It is critical to characterize the effects of various designs on engine performance and emissions. Spray characteristic of diesel fuel injection is one of the most important factors in diesel combustion and pollutant emissions where the interval between the onset of combustion and the evaporation of atomized fuel is relatively short. Therefore, this project is to study the spray simulation of diesel fuel using valve covered orifice (VCO) nozzle injector in the closed chamber. Three main components are focused on this paper, first is the relation between the spray characteristic influences of the various ambient temperature, T_{amb}. The second focus is the influences of the injection pressure, P_{inj} to the spray characteristic and the third focus is relation between the various diameter of nozzle hole size to the spray characteristic. Good spray characteristic leads to the good drivability, high combustion efficiency and stoichiometric air-fuel mixture. Therefore, Computational Fluid Dynamics (CFD) method using ANSYS Fluent simulation software is used for this purpose. The simulation of injection spray in chamber is conducted by using diesel fuel with the single and double-hole Valve Covered Orifice (VCO) nozzle, injection pressure, P_{inj} were various in range 5 KPa – 150 MPa, the ambient pressure, P_{inj} at atmosphere pressure at 101.325 Pa, the ambient temperature, T_{amb} was various in range of 273 K – 1000 K and at the same time iteration.

ABSTRAK

Prestasi dan hasil hasil pembakaran bagi enjin diesel adalah sangat berkait rapat dengan proses perubahan bahan api dari bentuk cecair kepada semburan yang sangat halus dan proses semburan yang sangat dipengaruhi oleh kadar semburan pemancit. Enjin moden menggunakan lubang mikro dengan reka bentuk lubang yang berbeza. Dimana ia penting untuk mencirikan kesannya kepada prestasi dan hasil pembakaran enjin. Sifat dan bentuk semburan bahan api merupakan salah satu faktor penting dalam pembakaran dan penghasilan bahan tercemar dimana tempoh diantara bermulanya pembakaran bahan api adalah agak singkat. Oleh itu, projek ini adalah untuk mengkaji semburan bahan api iaitu diesel secara simulasi dengan menggunakan muncung pemancit jenis VCO di dalam ruang yang tertutup. Tiga komponen utama ditumpukan dalam projek ini iaitu pertama adalah kesan semburan dalam suhu ruang yang berbeza. Yang kedua adalah kesan semburan terhadap perbezaan tekanan bahan api yang dipancitkan oleh pemancit dan yang ketiga adalah kesan semburan bagi perbezaan saiz diameter lubang pemancit. Bentuk semburan yang bagus akan menyumbang kepada pemanduan yang lancar, menghasilkan tahap kecekapan pembakaran yang tinggi. Oleh itu, Dinamik Bendalir Berkomputer (CFD) dengan mengunakan perisian ANSYS Fluent digunakan untuk tujuan ini. Simulasi semburan dijalankan dengan menggunakan diesel sebagai bahan api, simulasi antara satu dan dua lubang bagi muncung pemancit jenis VCO turut dijalankan. Tekanan semburan bahan api adalah antara 5 KPa sehingga 150 MPa, dan tekanan di dalam ruang tertutup adalah menyamai tekanan udara iaitu 101.325 Pa, bagi suhu di dalam ruang tertutup adalah diantara 273 K sehingga 1000 K dan dijalankan dalam kadar masa yang sama bagi setiap semburan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

NO_X	Nitrogen Oxide
H_2O	Water
B20	2-% of Biodiesel
KPa	KiloPascal
MPa	MegaPascal
K	Kelvin
Mm	Milimeter
%	Percentage
º	Degree
θ	Teta
L	Length
VS	Versus

LIST OF ABBREVIATIONS

VCO	Valve Covered Orifice
CFD	Computational Fluid Dynamics
HSDI	High Speed Direct Ignition
3D	3 Dimension
cc	Centimeter Cubic
hp	Horse Power
rpm	Revolution per Minute
PLN	Pump Line Nozzle
P_{inj}	Injection Pressure
P_{amb}	Ambient Pressure
P_{atm}	Atmosphere Pressure
T_{amb}	Ambient Temperature

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

A diesel engine also known as a compression-ignition engine is an internal combustion engine that used the heat of the compression to initiate ignition to burn the fuel, which is injected into the combustion chamber. This is in contrast to spark-ignition engine such as a petrol engine (gasoline engine) or gas engine (using a gaseous fuel as opposed to gasoline), which uses a spark plug to ignite an air-fuel mixture.

The engine was developed by Rudolf Diesel in 1893. The diesel engine has the highest thermal efficiency of any regular internal or external combustion engine due to its very high compression ratio. Low speed diesel engines as used in ships and other applications where overall engine weight is relatively unimportant can have a thermal efficiency that exceed 50 percent. Diesel engine is manufactured in two-stroke and four-stroke versions. They were originally used as a more efficient replacement for stationary steam engines. Since the 1910s they have been used in submarines and ship. Use in locomotives, trucks, heavy equipment and electric generating plants followed later. In the 1930s, they slowly began to be used in a few automobiles. Since the 1970s, the use of diesel engine in larger on-road and off-road vehicles.

Nowadays, the depleting reserves of fossil fuel, increasing demand for diesel and uncertainty in their availability have been a matter of global concern. From the automotive view, to minimize the fuel consumption rate in the diesel engine is by improving the engine performance to reduce the energy lose in the combustion. In the diesel engine, combustion and emission characteristic are influenced by the fuel atomization, nozzle geometry, injection pressure, shape of inlet port and other factors. In order to improve the fuel-air mixing, it is important to understand the fuel atomization and spray formation process. So far, to improve the combustion performance and particulate emissions, many researchers have investigated the characteristics of the spray behavior by experimental and theoretical approaches.

Generally, this study will be focus on the atomization and spray characteristics of diesel fuel, recently, most of the diesel engines applying direct injector method. The injected fuel into the combustion chamber undergoes atomization process. Atomization is the chemical reaction process between injected fuel and the compressed air in the combustion chamber, where the high pressure injected fuel breaking up into very fine mist or droplet. This process is important in order to produce the combustion ready mixture fuel. The smaller the droplet exiting the injector, the faster it will vaporize and ignite when it is injected into the engine cylinder. The effective atomization process will reduce the HC and NO_x emission production. There are numerous factors that influence the atomization process, such as injection pressure, the temperature in the combustion chamber, the piston surface geometry and the nozzle injector geometry.

Under direct injection method, nozzle injector geometry plays an important role in producing an effective injection for atomization process. There are many type of nozzle injector used in diesel engine, but three standard type of nozzle that commonly used at the diesel vehicle by most heavy duty vehicle manufacturer is sac nozzle, micro-san nozzle and valve covered orifice nozzle that also known as VCO-nozzle. The sac-nozzle has as sac volume space at the nozzle tip. After the injection process, a volume of fuel trapped in the tip of the injector. This volume of fuel would be wasted, leads to incomplete combustion and contribute in the HC and NO_x emission. in order to reducing the pollution, the micro-sac nozzle have been introduces, both sac-nozzle and micro-sac nozzle have a similar design, but the sac volume space size of the micro-sac nozzle is smaller compares to the sac-nozzle. The sac volume hole size have been reduced depending on the requirement fuel after injection process. Although the volume of fuel trapped in the sac volume hole of the micro-sac nozzle is less than sac nozzle, incomplete combustion still occurs. Then the valve covered orifice has been used widely. This type of nozzle has removes the sac volume hole at the nozzle tip. For the

VCO-nozzle configuration there is almost no residual fuel to vaporize. In addition, control of injection timing and quantities are such improved with VCO-nozzle configuration, since no time is required to fill the sac volume. The major drawback is the unequal pressure distribution of fuel through individual nozzle holes when multi-hole nozzles are utilized. This is due to random oscillations of the needle in the transverse direction during needle lift and fall. This eccentricity of the needle transforms the axial symmetry of the flow around the needle and can result in asymmetry of the spray from each individual hole.

Spray characteristic of fuel injection is another factor in diesel combustion and pollutant emissions. An investigation into various spray characteristics from different holes of VCO-nozzle will be performed and compared. In this study the spray characteristic such as a spray tip penetration also known as spray penetration length and spray cone angle were measured from the simulation result by ANSYS Fluent software.

Spray Cone Angle is the angle that formed by the cone of fuel injection leaving a nozzle orifice. For any given flow rate, the wider the spray angle is the smaller the droplet size will be. Larger angle spray simply has more space to distribute the droplet and so there is less chance of recombination and a greater opportunity to atomize.

1.2 PROBLEM STATEMENT

The depleting reserves of fossil fuel, increasing demand for diesel and uncertainty in their availability have been a matter of global concern. From the automotive view, to minimize the fuel consumption rate in the diesel engine is by improving the engine performance and to reduce the energy lose in the combustion.

1.3 OBJECTIVES

Generally, the objectives to be achieved in this study are stated below:

i) To study the effect of the spray behavior based on the various temperature and pressure, single and double hole of valve covered orifice nozzle.

ii) To study spray penetration length and spray cone angle.
iii) To determine the effective nozzle type for better spray atomization.

1.4 SCOPE

The project is focused on:

i) Literature review.
ii) Design the 3D constant chamber model using Solidworks software.
iii) Simulate the model by using ANSYS Fluent software.
iv) Result comparing the spray characteristic by different boundary condition, different nozzle hole size and between single and double hole nozzle injector.

1.5 METHODOLOGY

i) Stage 1: Literature study
Make review on literature study involving project title
ii) Stage 2: 3D modeling
3D modeling of the injector and engine geometry.
iii) Stage 3: Boundary condition setting simulation.
Set up boundary condition for simulation analysis.
iv) Stage 4: Simulating process.
Simulation analysis by using ANSYS Fluent software.
v) Stage 5: Analysis the results.
Analyse the simulation results.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter covers the recent review of diesel engine powered with diesel fuel. Diesel fuel properties, spray combustion chamber, injector nozzle parameter, and the software involved in this study are presented here. All the studies are mainly focus on the spray characteristic such as spray penetration length and spray cone angle on intention of various temperature, pressure, nozzle hole size, and single and double hole spray characteristic.

2.2 DIESEL ENGINE

The history of the diesel engine was started by Dr. Rudolf Diesel (1858 to 1913). In 1982, he present his diesel engine which the ignition of fuel by compression process. Compared to gasoline engine and steam engine, this engine had number of advantages. It is less fuel consumption and could be dimensioned for higher power outputs. In 1922, Robert Bosch decided to develop a fuel-injection system for diesel engines. Those Bosch fuel-injection pumps were a stepping stone in achieving higher running speeds in diesel engine. In 1936, Mercedes Benz 260D (2580cc, 50 hp) was the first volume-production car to be fitted with diesel engine. In diesel engine, the spark plug and carburetor are replaced by a fuel injector. This due to the air when being compressed to a temperature that is above the auto-ignition temperature of the fuel, and combustion creates on contact as the fuel is injected into this hot air.

In diesel engines, only air is compressed during the compression stroke, which can eliminate the possibility of auto-ignition. (Yunus and Micheal, 2007). Therefore the diesel engine can operate on much higher compression ratios, between 12 and 24. The principle operation of diesel engine relies on the heat within the compressed air to cause an ignition of the fuel charge. The chemical energy stored in the fuel is then converted into mechanical energy, which can be used to power tractors, locomotive and freight trucks. The diesel engine also burn fuel more completely compared to gasoline engine since they work on lower revolutions per minute (rpm) and having higher air-fuel mass ratio. It is more efficient then the spark ignition engine because they operate at much higher ratios (Yunus and Micheal, 2007). Lower fuel cost and higher efficiency become the reason why they have been used in large ships and emergency power generation units.

2.3 VISCOSITY

Viscosity is a measure of the fuel's adhesive or cohesive property and is the key factor in estimating the required temperature for pumping, injection, storage and transfer of the fuel. A viscosity comparison of diesel and other oils is shown in Table 2.1.

Table 2.1: Comparison of Viscosity among Diesel, Biodiesel and Vegetable Oils

Type	**Heat of Combustion (MJ/Kg**	**Kinetic Viscasity (mm^2/s)**	**Flash Point (°C)**
Diesel	427	1-4	80
Biodiesel	372	4-6	100
Jatropha Oil	396	757	340
Rapeseed Oil	376	74	317
Sunflower Oil	371	66	316
Soya Oil	371	63.5	359

Source: Knothe. 1997

2.3.1 Diesel Fuel

Diesel is processed from crude oil, a fossil fuel with broad variations in colour, from clear to tar-black, and viscosity, from that of water to almost a solid. Crude oil contains a complex mixture of hydrocarbons comprised of differing chain length and physical and chemical properties. Hydrocarbon containing up to four carbon atoms are gaseous in nature, those with 5 to 19 carbon atoms are usually found in liquid form, and those with a carbon composition greater than 19 are found as solid as shown in Table 2.2.

Table 2.2: Hydrocarbon Contents in Crude Oil

HYDROCARBONS	GENERAL FORMULA	CHAIN TYPE	STATE (Room Temp)	EXAMPLES
Paraffins (Aliphatic)	C_nH_{2n+2} (n:1 to 20)	Linear or Branched	Gas or Liquid	Methane, Propane Hexane
Aromatic	C_6H_{5-Y}	One or More Benzene Rings wt Long Chains Ys	Liquid	Benzene Napthalene
Napthenes (Cycloalkanes)	C_nH_{2n}	One or More Cycloalkane Rings	Liquid	Cyclohexane Methyl Cyclohexane
Alkenes (Olefin)	C_nH_{2n}	Liner or Branched One or More Double Bond	Gas or Liquid	Ethylene Butene Isobutene
Dienes and Alkynes	C_nH_{2n-2}	Triple Bond	Gas or Liquid	Butadiene Acetylene

Source: ATSDR, 1995; OTM, 1999

2.4 SPRAY COMBUSTION PROCESS

The combustion performance and emission are mainly influenced by the atomization of the liquid fuel, the motion and evaporation of the fuel droplets and mixing of the fuel with air. The dynamics of spray and its combustion characteristics are extremely important in determining, for instance, the flame stability behavior at widely varying loads, the safe and efficient utilization of energy, as well as the mechanisms of pollutants formation and destruction. Figure 2.1 show the consecutive phase's occurring during the operation of the diesel engine.

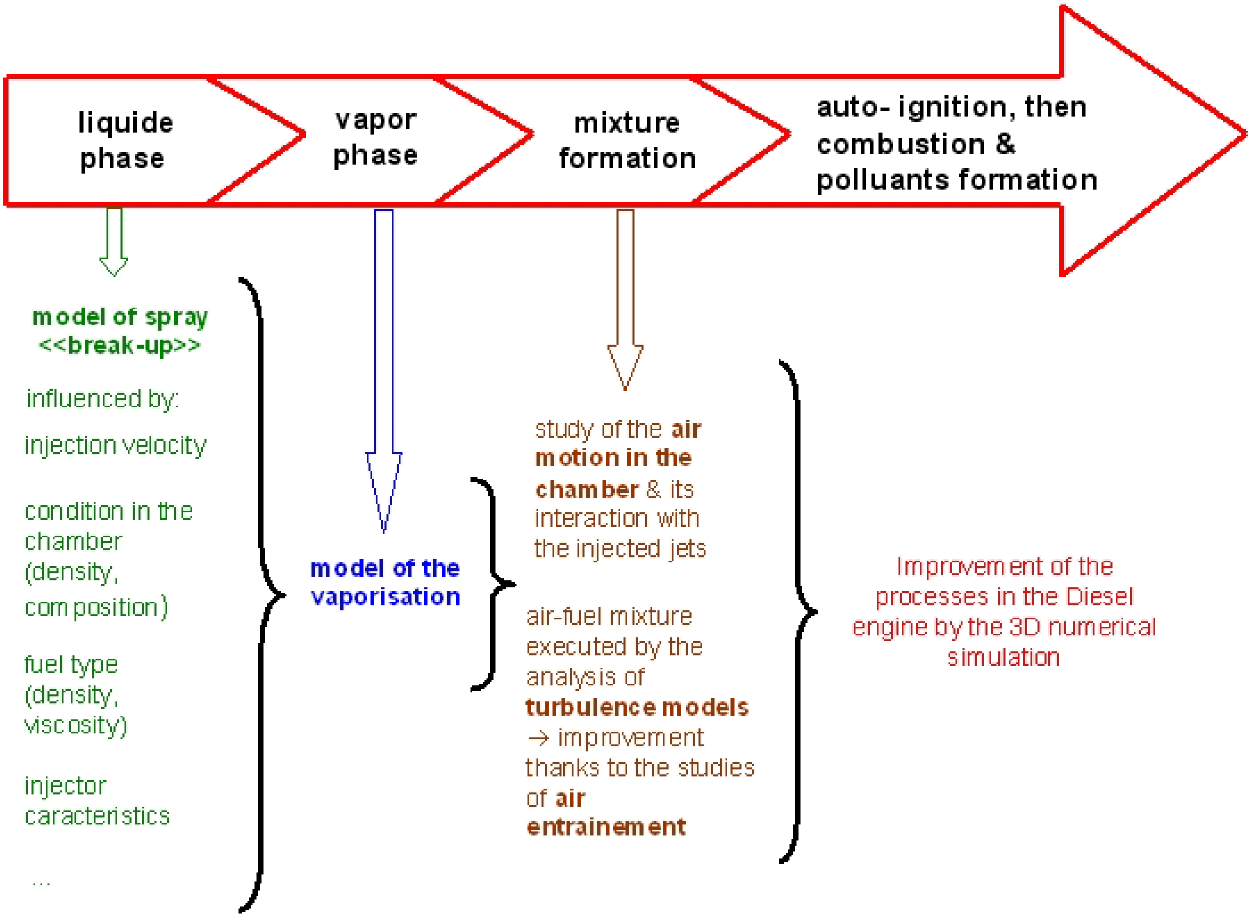


Figure 2.1: The Consecutive phase's during the operation of a Diesel engine.

Source: Olga WOJDAS, 2010

Understanding and controlling atomization and spray combustion is becoming an essential part of the industrial applications, which have been driven by increasingly urgent demands to improve fuel and energy efficiencies, and to drastically reduce the emission of pollutants. The spray combustion process may be divided into several elements, such as atomization, liquid transport, vaporization, and combustion. In general, liquid fuel is injected through a nozzle system into the combustion chamber and is atomized to form a spray of droplets before gas-phase combustion take place in the vaporized fuel. In the atomization region, the liquid dominates the flow and the liquid fuel disintegrates into ligament and droplets. Large liquid blobs which are bulks of continuous liquid present in the atomization region.

The dense spray region has lower but still significant liquid volume fraction and includes secondary breakup of drops and ligaments as well as drop interactions, such as collisions and coalescence. Liquid ligaments normally present in the atomization and dense spray regions, which are non-spherical liquid sheets, sheared off the liquid jet column. In the dilute spray region, spherical droplets are well formed and have a strong interaction with the turbulent airflow. In general, the spray structure depends on the injection pressure difference, injector size, fuel viscosity and fuel density. (X.Jiang, G.A Siamas, K.Jagus, T.G.Karayiannis, 2010)

2.5 SPRAY REGIME

Diesel engine sprays are usually of the full-cone type. This means that in the idle mode the fuel is blocked from the upstream side of the nozzle and during injection the core of the spray is denser than the outer regions. The liquid spray can be characterized by distinguishing spatial regimes. Starting from the nozzle exit there is an intact liquid core. A few nozzle diameters further downstream in the so-called churning the liquid consist of ligaments (blobs). These liquid parts are like large droplets with sizes comparable to the nozzle diameter. Then the ligaments are breakup into liquid phase is high. Further downstream the breakup process of droplets goes on and in the same time more and more of the surrounding gas is entrained into the spray area.

This results in diverging behavior with a characteristic spray angle. The regimes after the thick zone are the thin zone (low volume but still high mass fraction of liquid) and the dilute zone (negligible volume and low mass fraction of liquid), respectively.

In a hot environment the position at which all liquid evaporated is called the liquid length. From (automotive) experiments this length is found to be approximately constant after a short development time. From that point on the evaporated fuel continues to penetrate the surrounding gas and is denoted as vapor length. In atypical diesel injection timeframe (few milliseconds) the vapor length does not reach a steady state.

2.6 SPRAY PENETRATION

The spray penetration is defined as the maximum distance from the nozzle to the tip of the spray at any given time and is one of the most important characteristics of the combustion process. If the spray penetration is too long, there is a risk of impingement on the wall of the combustion chamber, which may lead to fuel wastage and formation of soot. This normally occurs when the chamber wall is cold and where there is limited air motion. However, a short penetration will reduce mixing efficiency hence resulting in poor combustion. Hence the information of spray penetration length would be useful for the design of the engine combustion chamber.

Some of the studies on spray atomization spray formation and spray characteristics were reviewed. The main findings of individual investigations on Diesel sprays are summarized in Table 2.3. Many of the studies concerned with spray atomization and spray characteristics dealt with either steady spray or low injection pressure sprays and at these conditions the spray no longer represented that found under real engine conditions.

Table 2.3: Summary of Diesel Spray Characteristic.

Author and Year	Technique used	Main results
Hiroyasu and Arai (1990)	Constant volume spray chamber. Electric resistance to measure breaks- up length. Photography and light interception method for spray penetration and cone angle. Fraunhofer diffraction	Increasing the ambient pressure decreases the break-up length. Increasing the ratio L/D leads to a longer break -up. Penetration is linear with time until spray reaches break-up. Increasing the ambient gas pressure decreases the spray tip penetration. Spray

	technique to determine droplet size distribution.	penetration is not significantly affected by temperature changes. Increasing injection pressure decreases SMD. Increasing the viscosity of fuel increases SMD. Equations developed for spray tip penetration, SMD, spray angle and break-up length and time.
Hosoya and Obokata (1993)	Atmospheric spray from four different types of nozzle (measurement restricted to one hole). Droplet velocity and size measured using LDA and PDA.	Difficulties to measure the spray (unsteady under condition of high temperature and pressure). Different spray characteristics between single-hole nozzle and multi-hole nozzle. The number and the position of the holes do not affect multi-hole basic spray characteristics.
Naber and Siebers (1996)	Study of vaporizing and non-vaporizing spray. High-pressure fuel injection system (common-rail). Constant volume combustion chamber. Schlieren imaging system and high speed film camera.	Spray dispersion increase with an increase in gas density. A reduction in penetration length is observed with an increase of ambient gas density. Comparison of the data with spray penetration correlation from the literature. Development of a theoretical penetration correlation for non-vaporising spray.
Warrick et al. (1996)	Multi-hole Diesel Injector. Heated spray chamber. Nd:YAG laser and 35mm	Increasing the temperature of the spray environment decreases the hole-by hole variations.

	camera used for spray visualization. Two experiments: Static cold and warm tests.	Small effects of the spray temperature on the spray tip penetration. Increasing the chamber temperature increasing the cone angle (vaporization of the droplets situated on the edge of the spray plumes).
Ficarella et al. (1997)	High injection and vessel pressure. Five-hole VCO nozzle (analysis of the spray produced by each hole). Sizes and velocity of the spray measured by Malvern method and PDA. Photographic technique used to analyze the spray from every hole.	Spray cone angle decreases with time. Increasing the feeding pressure decreases SMD. No effect of the injected quantity on SMD. Variation of the shape of the spray from hole-to-hole but size distribution remains constant. Limitation of the PDA technique to analyze dense Diesel spray
Levy et al. (1997)	Low injection pressure and high pressure chamber. Sizes and axial velocities measured with PDA. KIVA-II code for numerical simulations.	Increase of SMD with distance from the nozzle (because of the coalescence of droplets at low temperature and evaporation of the small ones at higher temperature). Increase of axial velocity with the distance from the nozzle exit. Fastest velocity at highest temperature.
Bae and Kang (2000)	Different VCO nozzle discharging in high-pressure chamber. CCD camera and shadowgraph technique were used to observe the spray.	Hole-to-hole variation only observed for sac nozzle in term of spray angle. Negligible hole-to-hole variation of spray for VCO nozzle. Droplets formed from the ligaments around the spray

		surface at the early stage.
Jimenez et al. (2000)	Single hole Diesel Injector. Spray into a flow speed air cross flow at atmospheric pressure. PDA and CCD camera used for the measurements.	Correlation for spray penetration under atmospheric ambient conditions. Observation of two stages in the temporal evolution of the spray tip. SMD increased with air temperature, during the early stage of the injection.
Kernaid et al. (2002)	High speed video camera and Schlieren technique. High injection pressure common-rail system. Single, three and five hole VCO and mini-sac nozzle. High pressure spray rig.	No orifice variation of penetration for the multi-hole nozzle. Dependence of the vapour penetration on both injection pressure and in-cylinder density (an increase in injection pressure and a decrease in in-cylinder density resulted in a better vapour penetration). Faster liquid spray penetration at higher injection pressure.

Source: Lacoste J. 2006.

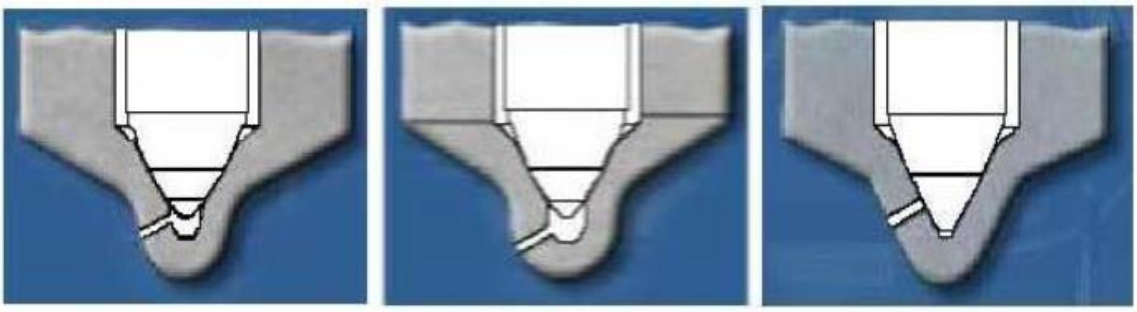
2.7 NOZZLE PARAMETERS

2.7.1 Valve Covered Orifice Nozzle Injector

Fuel injection nozzle is the key component in the successful delivery and combustion of fuel. Nozzles are basically closed valve that are opened by high pressure fuel delivered from the injection pump assembly. Pump Line-Nozzle (PLN) design by the manufacturer such as Bosch, Delphi Automotive and Caterpillar, electronic unit pump, as well as distributor pump type system are couple to nozzles. Within this timing, high fuel pressurization and fuel quantity are performed in the injection pump, while fuel atomization occurs at the nozzle spray tip. The high pressure fuel is delivered through a small bore steel line from the injection pump to each nozzle.

Conversely, the term injector is normally applied to both Mechanical Unit Injectors and Electronics unit Injectors, where the timing, atomization, metering and high fuel pressure are created within the body of the injector. There are three basic types of injector nozzle as shown in Figure 2.2 sac-nozzle, micro-sac nozzle and valve covered orifice (VCO) nozzle. Each type has its own advantages and disadvantages.

With VCO nozzle, the volume space of sac has been reduced to such an extent that the spray holes branch off bear the needle seat. Therefore, the opening and closing of the nozzle(s) is carried out by the needle itself.



(a) Micro-Sac (b) Sac-Nozzle (c) Valve Covered Orifice

Figure 2.2: The Basic Type of single-hole Injector Nozzle; (a) Micro-Sac nozzle, (b) Sac-nozzle, and (c) Valve Covered Orifice nozzle.

Source: Bosch, 2003.

For the VCO configurations there is almost no residual fuel to vaporize. In addition, control of injection timing and quantities are much improved with VCO configuration, since no time is required to fill the sac volume space. The major drawback is the unequal pressure distribution of fuel through individual holes when multi-hole nozzles are utilized. This is due to random oscillations of the needle in the transverse direction during needle lift and fall. This results in asymmetry of the spray from each individual hole. The injector type and nozzle configuration used in the current study are summarized in Table 2.4.

Table 2.4: Parameter of the Nozzle and Injection Strategy

Manf.	**Type**	**Diameter (mm)**	**L/D_n**	**Number of Holes**	**Injection Strategy**	
					Spray Characterization	**Rate Measurement**
Bosch	VCO single guided	0.2	5	1, 3, and 5	Single	Single (for 1 and 3 hole nozzle)
Bosch	VCO Double guided	0.2	5	1	Single	

Source: Kourosh Karimi, May 2007

2.8 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics or known as CFD is defined as the process and set of methodologies that enable the computer to provide numerical simulation of fluids flows. Computational fluid dynamics is a technology that is used to analyze the dynamics of anything that can flow regardless in liquid or gaseous state. It is also a software tool that can model or simulate o flow or phenomena of any system or device under analysis (Hirsh, 2007). CFD is computed using a set of partial different equations to predict the flow behavior. Besides that, it is also used for analyzing heat transfer model, mass flow rate, phase change, chemical reaction such as combustion, turbulence model, mechanical movement, deformation of solid structure and many more.

The word simulation is to indicate that the usage of computer in solving numerically the laws that govern the movement of fluid in or around a material system, where its geometry is also modeled on the computer. Hence, the whole system is transform into virtual environment or virtual product. This can be opposed to an experimental investigation, characterized by a material model or prototype of a system in measuring the flow properties in a prototype of an engine.

CHAPTER 3

METHODOLOGY

3.1 GENERAL METHODOLOGY

This chapter describes flow of the project and the project progress. It also describes how the project is organized in order to complete this project. The methodology can be divided into five parts, which are:

i) Literature Study.
ii) Project Flow Chart
iii) 3D Design Modelling
iv) ANSYS Fluent Simulation Setup:
    Step 1: Geometry.
    Step 2: Computational Meshing.
    Step 3: Simulation Setup.
    Step 4: Graphics and Animations.
v) Measurement Method

3.2 LITERATURE STUDY

Literature study is very important before entering the simulation and analysis stage. It will help improves the understanding of the project and give the general idea of the project. Literature study was done in order to get the previous research result and then can easily decide the parameter that will be used in the project such as the injection pressure, P_{inj}, ambient temperature, T_{amb}, ambient pressure, P_{amb}, and the injector nozzle size etc.

After the parameter had been decided, the apparatus was designed before the experiment of fuel spray can be operated. The design of the apparatus was referring to the parameters that had been decided before. After the apparatus was designed, the set up was run and all the data could be collected.

The simulation and analyse was about to get the image of the fuel spray particle and the result will be analyses. If the result was not acceptable, experiment will run again until get the acceptable result. During these work progress, thesis writing had be done until the presentation that present the progress of whole work.

3.3 FLOW CHART/ PROJECT FLOW

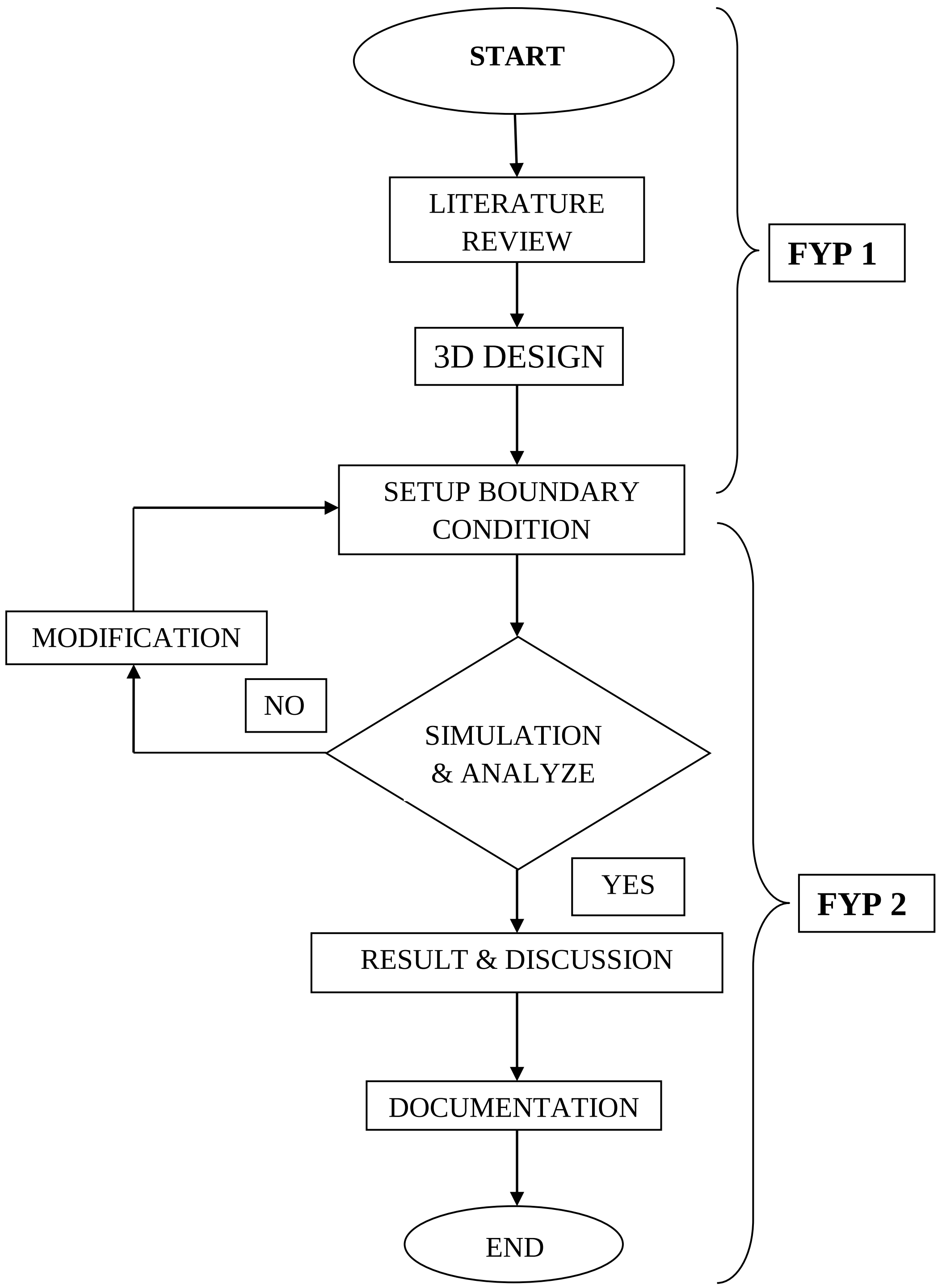
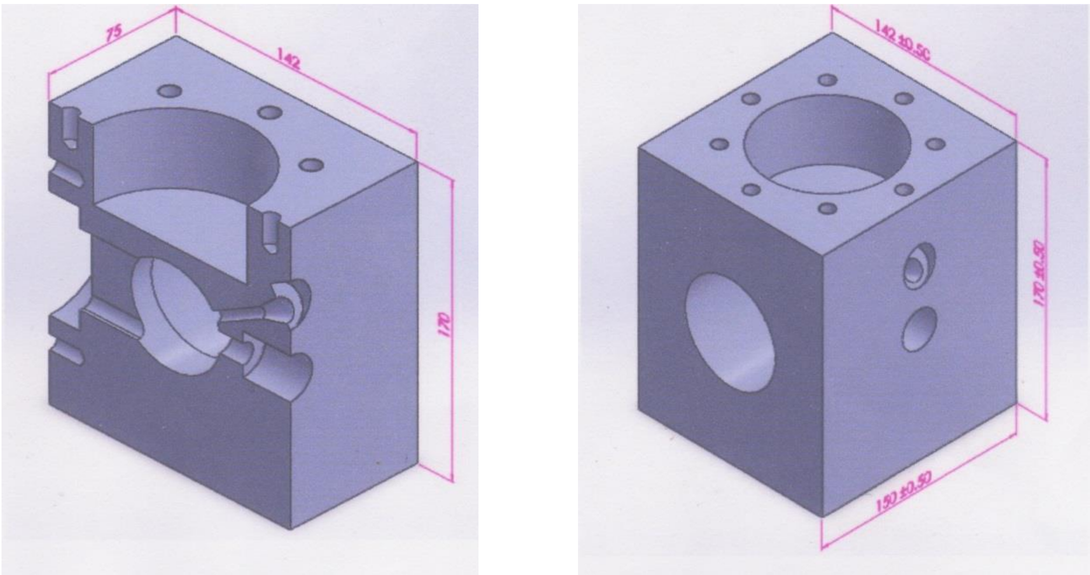


Figure 3.1: Flow Chart.

3.4 3D DESIGN MODELING

The closed chamber complete attached with the nozzle injector system was designed in 3D design using SolidWorks software. Figure 3.2 shows the 3D design of combustion chamber.



(a) Offset View. (b) Complete View.

Figure 3.2: 3D Design of Combustion Chamber.

Based on the objective, the main part that needs to be focus on is the volume of the nozzle injector hole and volume of the closed chamber itself, where the spray proses is happen. The closed chamber volume that complete with the nozzle injector hole was design based on the closed chamber parameter and geometry. Figure 3.3 shows the Closed Chamber Volume complete with the nozzle injector hole in isometric view. Meanwhile Figure 3.4 shows the dimension of the closed chamber with nozzle injection volume.

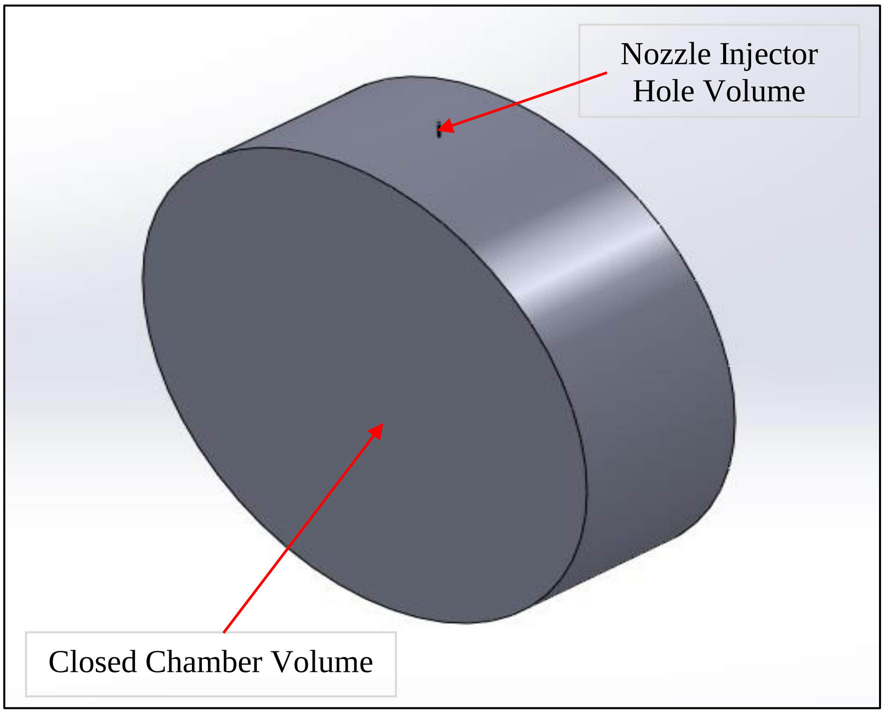


Figure 3.3: Closed Chamber Volume with the Nozzle Injector Hole in Isometric View.

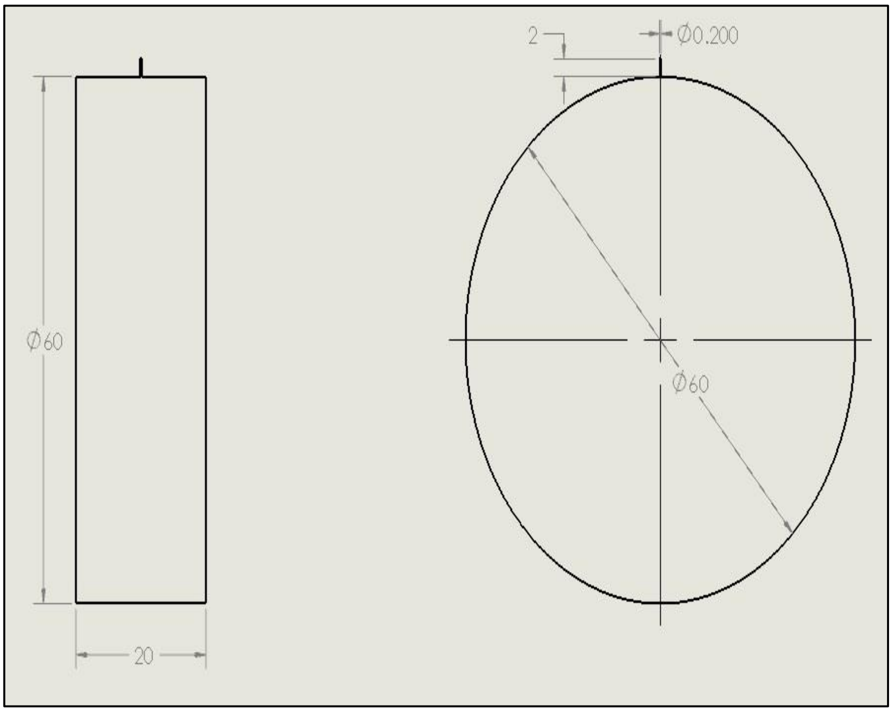


Figure 3.4: Closed Chamber with Nozzle Injector Volume Dimension.

3.5 ANSYS FLUENT SIMULATION SETUP

To predict the behaviour of an evaporating fuel in constant chamber that have been design in SolidWorks software, the simulation tools was used. The most common educational purpose ANSYS Fluent was selected for this study as simulation tools. Figure 3.5 shows the ANSYS Fluent software layout.

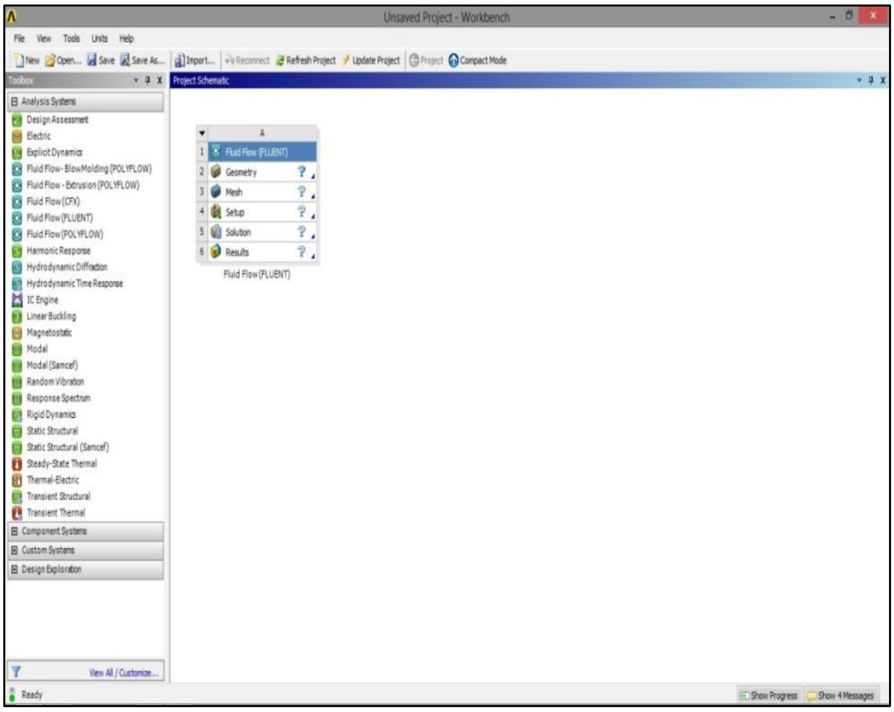


Figure 3.5: ANSYS Fluent Software Layout.

3.5.1 Geometry

Import the closed chamber with nozzle injector design in .IGS or .IGES format to the geometry. Update the project to process the design geometry. Select the injection surface at design geometry and right click, click the Named Selection to declare the injection surface as the Inlet for injection system. Then select the nozzle cone and nozzle hole surface with the whole surface of the constant chamber, name them as Wall as shown in Figure 3.6. Next select the surface at the end edge of the chamber, then name it as Output. Every time after name the surface, the geometry must to be Update or Generate. After all surfaces were named, close the geometry window.

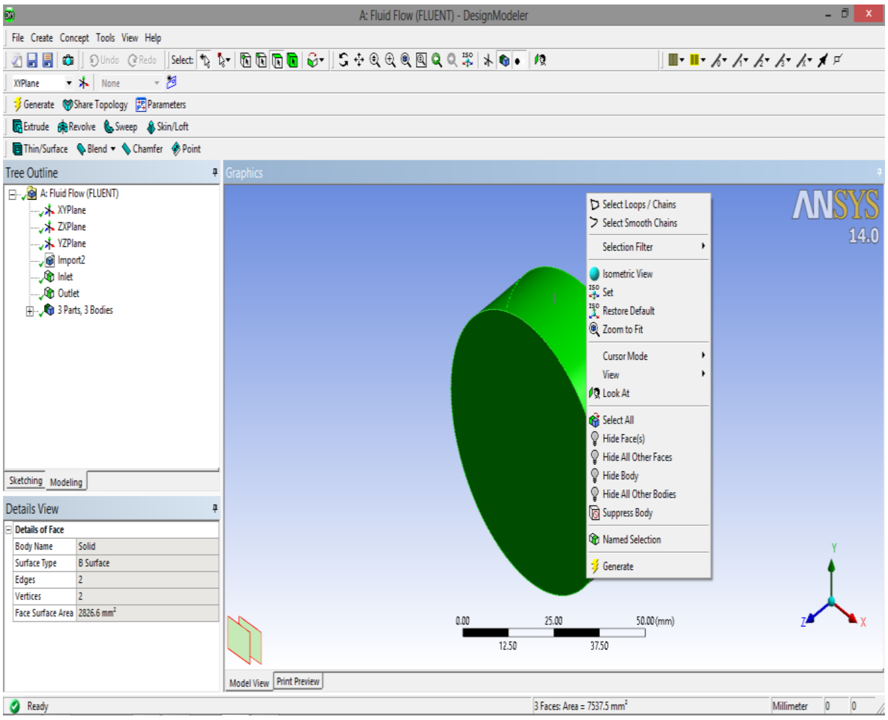


Figure 3.6: Named Selection Surface Processes in Geometry.

3.5.2 Computational Meshing

Computational Meshing is important procedure in numerical analysis of complex processes. With the aid of simulation tools, one can analyse the flow problems. To get the reliable and high accuracy results, the mesh refinement method need to be applied. The refinement operation and number of refinement point closely related in simulation time. Higher level of refinement will take longer simulation time. However the lower refinement level may cause the result to be inaccurate. In this simulation, the higher level of refinement was used to get the clear and accurate result as show in Figure 3.7. After complete the meshing process then close the meshing window.

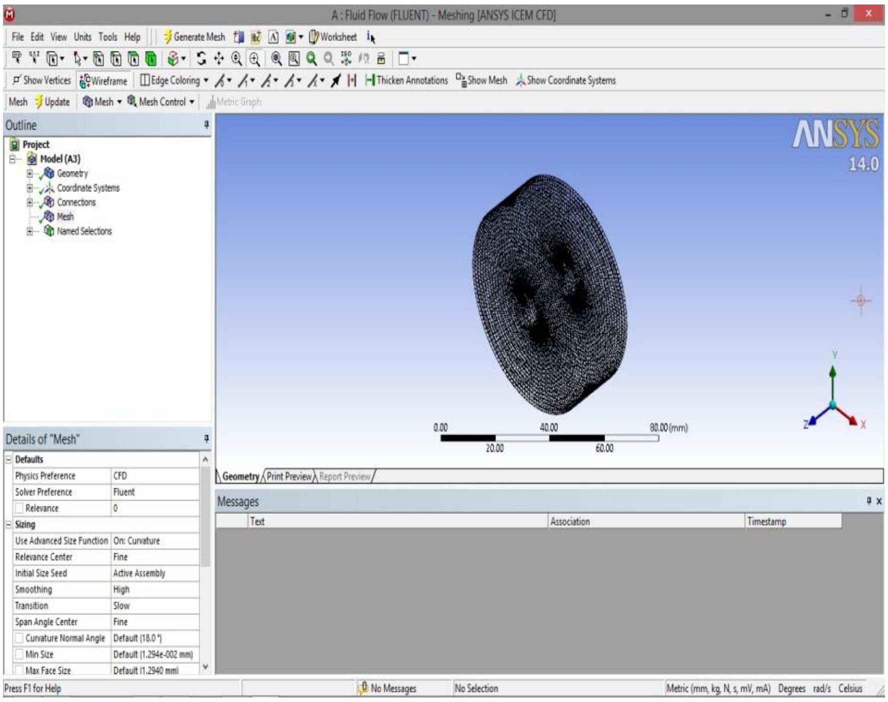


Figure 3.7: The Meshing Process with the higher Refinement Level.

3.5.3 Simulation Setup

Start the 3D version of Fluent.

General Setup

Check the grid and report the quality of the meshed design. Fluent will perform various checks on the meshed design and report the progress in the console. Figure 3.8 shows the General Setup option button. Make sure that the minimum volume report is in a positive number.

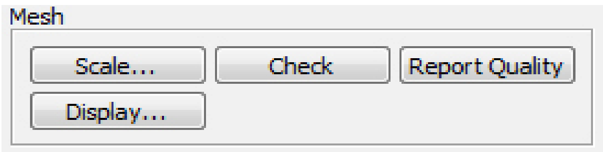
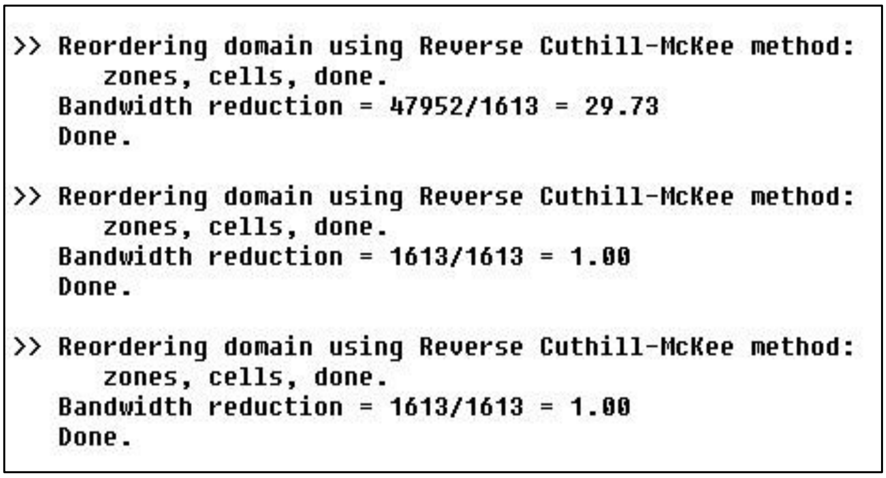


Figure 3.8: General Setup Option Button.

Reorder the domain until the value is constant. This mesh reorder process is to speed up the solution procedure, which will substantially reduce the bandwidth. The Figure 3.9 shows the step to reorder and Figure 3.10 shows the report of the progress in the console.

Figure 3.9: Reorder Step.



```
>> Reordering domain using Reverse Cuthill-McKee method:
      zones, cells, done.
   Bandwidth reduction = 47952/1613 = 29.73
   Done.

>> Reordering domain using Reverse Cuthill-McKee method:
      zones, cells, done.
   Bandwidth reduction = 1613/1613 = 1.00
   Done.

>> Reordering domain using Reverse Cuthill-McKee method:
      zones, cells, done.
   Bandwidth reduction = 1613/1613 = 1.00
   Done.
```

Figure 3.10: The Report of the Reorder Progress.

Retain the default for the Solver setting.

Model

In the model setup, there were many option of setting. The setting in the Model Setup is shown as Figure 3.11.

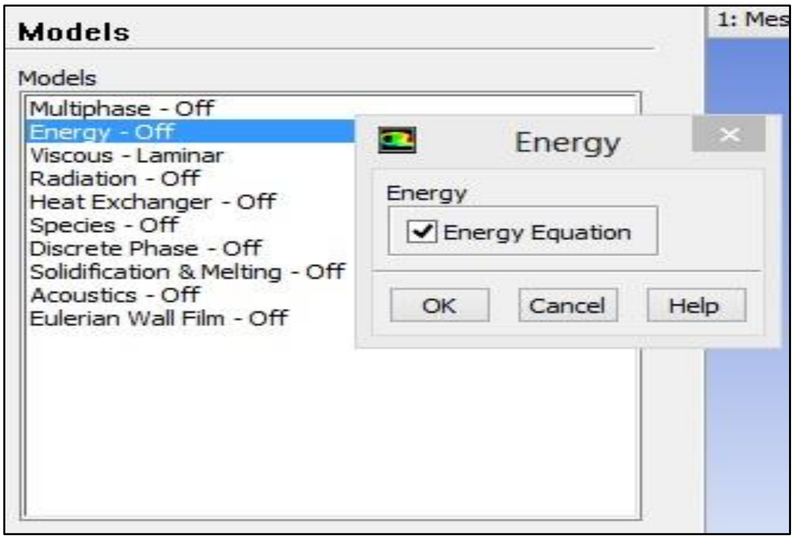


Figure 3.11: The setting in the Model Setup.

i) Enable the heat transfer by enabling the Energy Equation as shown in Figure 3.11.

ii) Enable the realizable k-epsilon turbulence model at Viscous Model as shown in Figure 3.12. The Realizable k-epsilon model gives a more accurate prediction of the spreading rate of both planar and round jets than the standard k-epsilon.

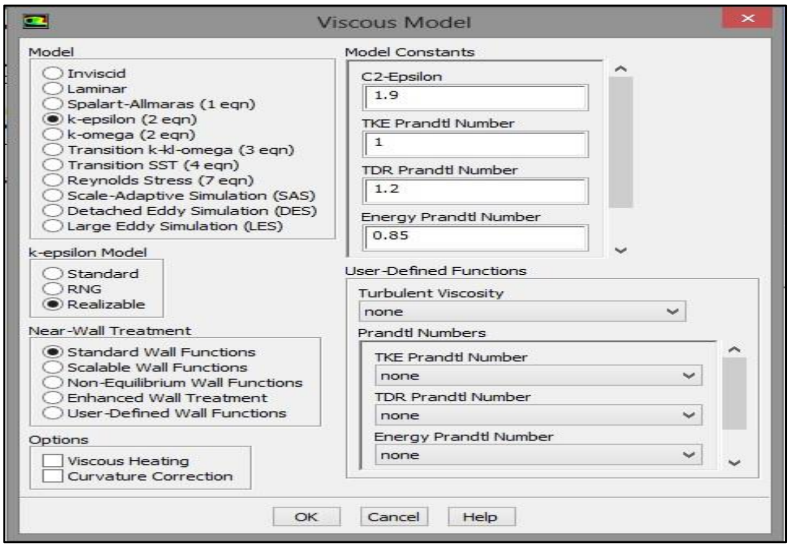


Figure 3.12: Viscous Model Setting.

iii) Enable the chemical Species Transport and Reaction. Refer to Figure 3.13.

a) Select Species Transport in the Model List
b) Select diesel-air in the Mixture Material drop-down list. The Mixture Material list contains the set of chemical mixture that exist in the fluent database. The complete description of the reacting system can be access by selecting one of the pre-defined mixtures. The chemical species in the system and their physical and thermodynamics properties are define by the selection of the mixture material. Mixture material selection or properties can be alter and modify by using the material panel.
c) Enable volumetric in the Reaction and for the Option, enable Inlet Diffusion and Diffusion Energy Source.
d) Click OK to close the Species Model setting. Fluent will list the properties that are required the models that have been enabled. An information dialog box will open, reminds to confirm the property values that have been extracted from the database. Click OK in the information dialog box to continue.

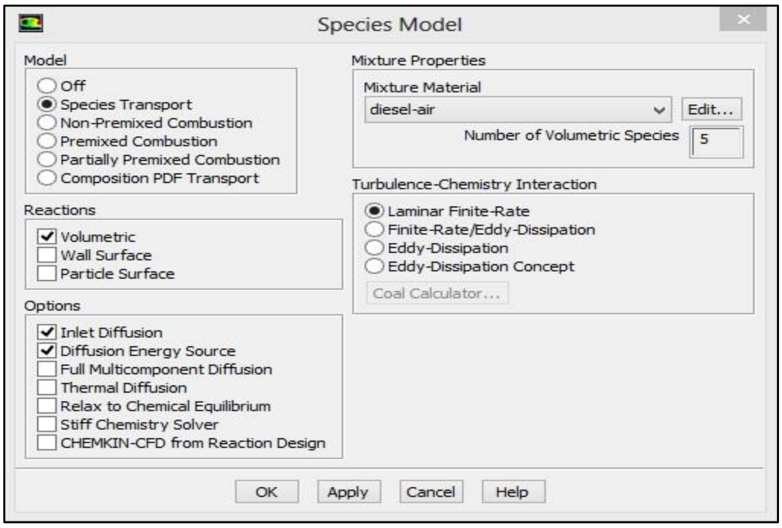


Figure 3.13: Species Model Setting.

iv) Define the Discrete Phase Modeling parameter. Refer to Figure 3.14.

a) Enable interaction with Continuous Phase in the Interaction group box. This will include the effect of the discrete phase trajectories on the continuous phase.

b) Set Number of Continuous Phase Iteration per DPM Iteration to 10.

c) Click the Physical Model tab to enable the Physical Models.

i) Enable Droplet Breakup in the Spray Model group box.

ii) Retain the default selection of TAB in the Breakup Model list.

iii) Retain the default value of 0 for y0 and 2 for the Breakup Parcels value in the Breakup Constant group box. This parameter is the dimensionless droplet distortion at t = 0.

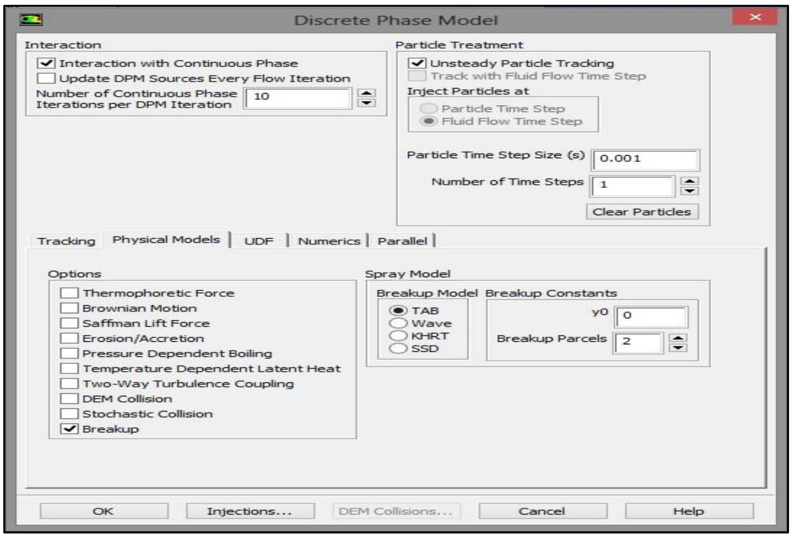


Figure 3.14: Discrete Phase Model Panel, Physical Model Tab Setting.

d) Click the Tracking tab to specify the tracking parameter. Refer to Figure 3.15.

i) Retain the default value of 5 for Step Length Factor.

ii) Select dynamic-drag in the Drag Law drop-down list in the Drag Parameters group box. The dynamic-drag law is available only when the Droplet Breakup model was used.

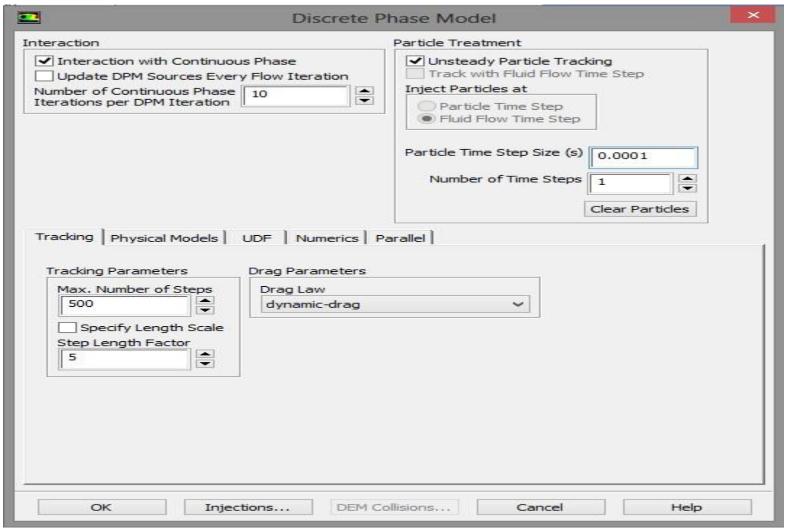


Figure 3.15: Discrete Phase Model Panel, Tracking tab setting.

e) Retain the Unsteady Particle Tracking option in the Particle Treatment group box.

f) Enter the value of 0.0001 for the Particle Time Step Size.

g) Retain the default value of 1 for Number of Time Steps.

h) Click OK to close the Discrete Phase Model panel. An information dialog box will open, reminds to confirm the property values before continuing.

Boundary Conditions

Set the boundary condition for the inlet and operating condition. Determining the boundary conditions is crucial because the variable setting will influence the result of the analysis. Errors in defining the boundary conditions may cause invalid simulation results.

i) Inlet Zone

a) Declare the Inlet zone as the Pressure-Inlet type.
b) Under Momentum tab, select Normal to Boundary from the Direction Specification Method dropdown list.
c) Set the value of Gauge Total Pressure and Supersonic/Initial Gauge Pressure in range 5 KPa ~ 150 Mpa in each simulation depend on the boundary condition decided.
d) Retain the default value and setting for Turbulence group box.
e) Click the Thermal tab and set the value at 273 K for the Total Temperature. Change the value of the temperature in range 273 K ~ 1000 K for each simulations depend on the boundary condition decided. All of the settings for the Pressure Inlet panel are shown in Figure 3.16.

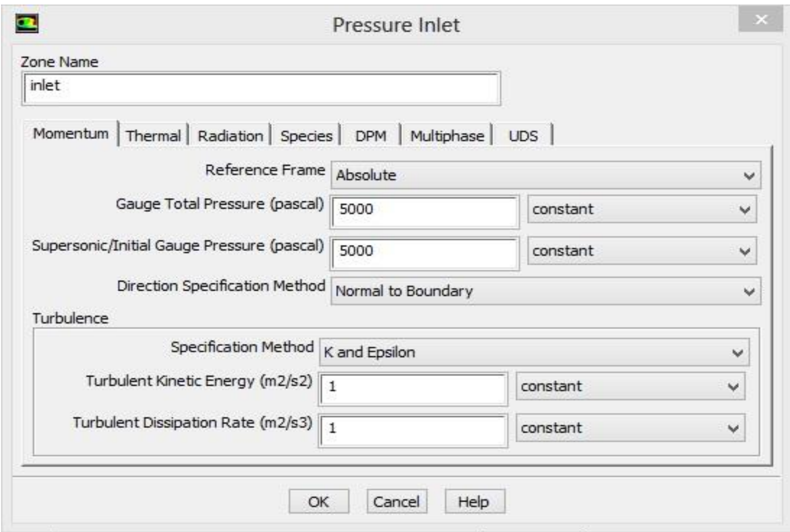


Figure 3.16: Pressure Inlet Panel setting.

ii) Outlet Zone

a) Declare the Outlet zone as the Pressure-Outlet type.
b) At Momentum tab, retain all the value including the Gauge Total Pressure at 0.

c) Click the Thermal tab and set the value at 273 K for the Backflow Total Temperature. The temperature value at the outlet zone must be same as the Inlet zone for each simulation. Change the value of the temperature in range 273 K ~ 1000 K for each simulations depend on the boundary condition decided. The Pressure Outlet panel setting are shows in Figure 3.17.

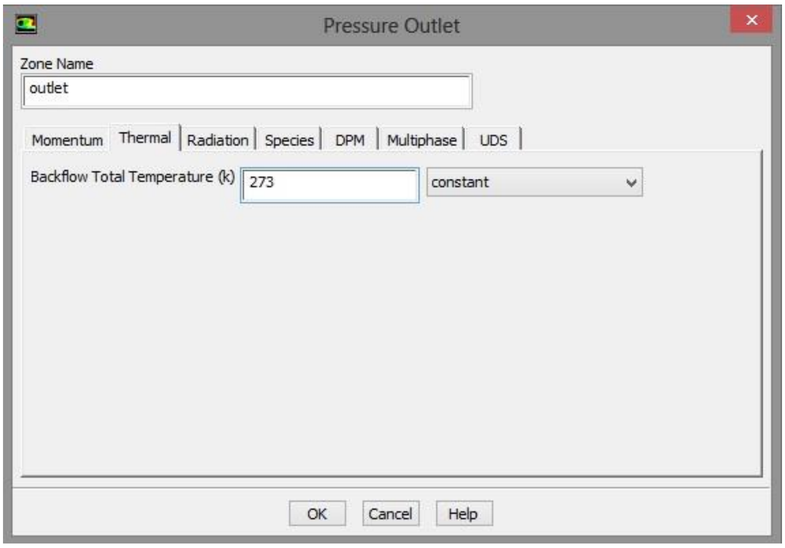


Figure 3.17: Pressure Outlet Panel setting.

iii) Operating Conditions.

a) Click the Operating Condition to open the Operating Condition Panel.
b) Retain the value of Operating Pressure at atmosphere pressure, 101325 Pa as shows in Figure 3.18.
c) Click OK to close the Operating Condition Panel.

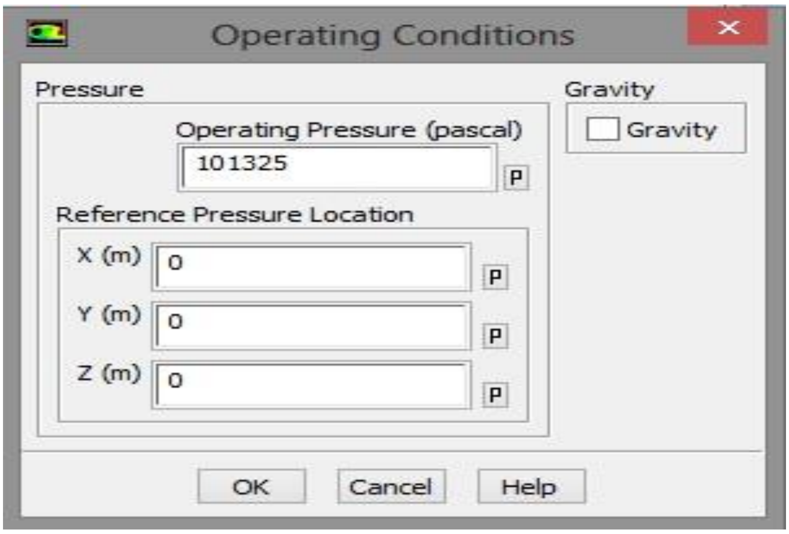


Figure 3.18: Operating Condition Panel setting.

Solution Setup

Solution Methods

Retain the SIMPLE for Scheme in Pressure-Velocity Coupling. Change all method to Second Order in Spatial Discretization except for the Pressure that retains Standard and Least Squares Cell Based for Gradient as shows in Figure 3.19.

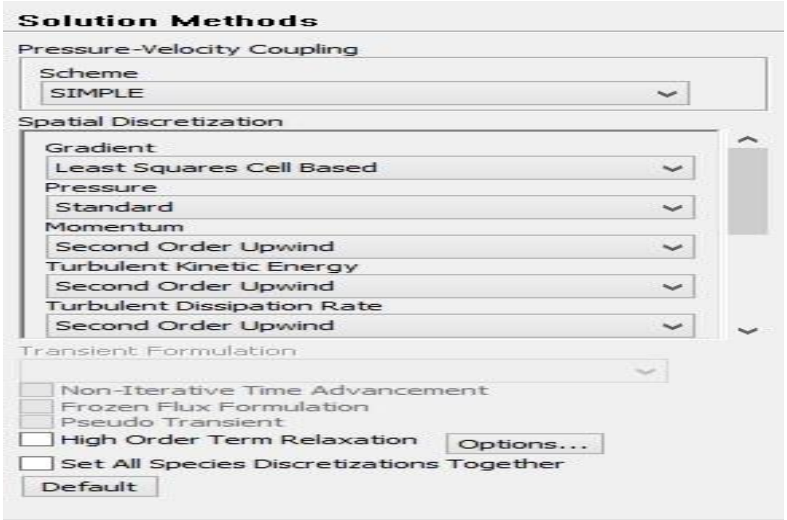


Figure 3.19: The Solution Methods setup.

Solution Initialization

Select Standard Initialization under Initialization Method group box. Then select Inlet from the Compute From dropdown list. Click Initialize button to start initialize the variables as shows in Figure 3.20. For any changes in the setting, must to be initialized before proceed to Run Calculation process. Close the Solution Initialization Panel.

Figure 3.20: Solution Initialization Panel setup.

Run Calculation

Start the calculation by requesting 350 iterations as shown if Figure 3.21. Run the simulation again after change the temperature and pressure at both Inlet zone and Outlet zone at the Boundary Condition. The same procedure was repeated on the different nozzle hole size, single and double hole of nozzle injector.

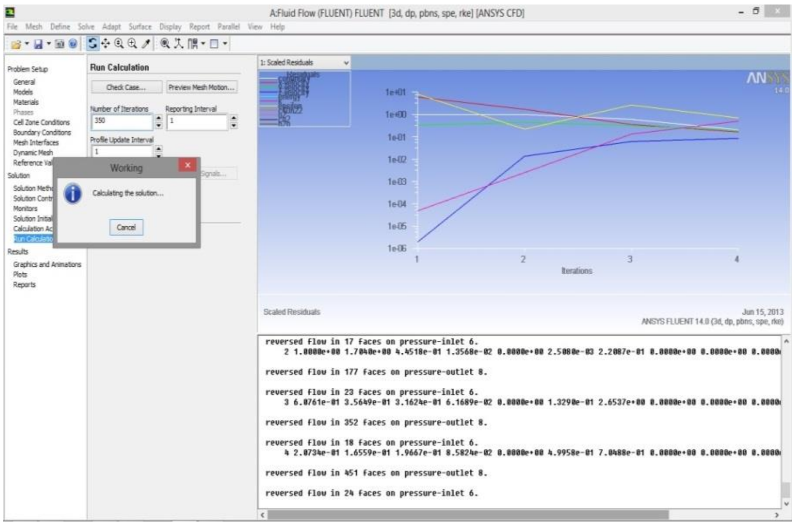


Figure 3.21: The Simulation process by 350 iteration.

3.5.4 Graphics and Animations

Graphics and Animations is the function panel that shows the result of the simulation that have been done. Under Graphics group box, there were many options that can be choosing to show the results. Select the Contours under Graphics group box and then click Set Up button. The Contours Panel will appear. Enable the Filled under Option group box and then at the Contour of dropdown list, select the Velocity. Select the surface that need to be show under Surface group box as shown in Figure 3.22 and then click the Display button. The result of the simulation will appear.

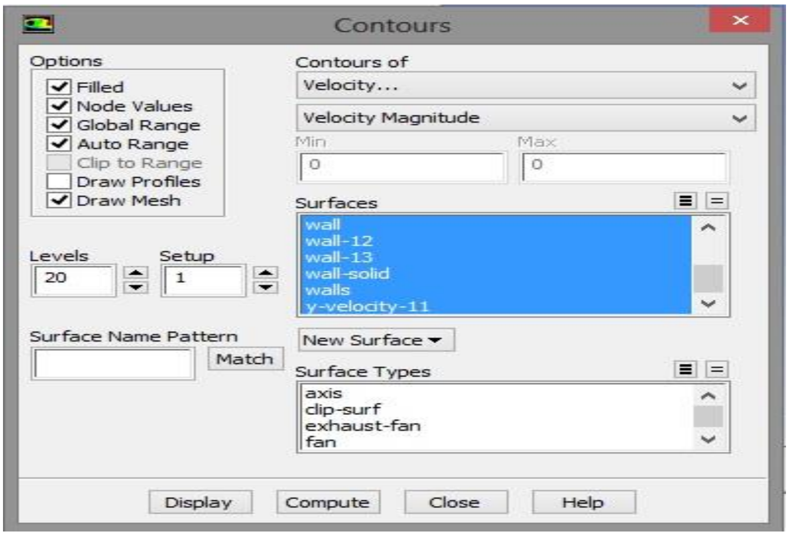


Figure 3.22: The Contour Panel setting.

3.6 MEASUREMENT METHOD

For the measurement method in order to get the value of Spray Penetration Length and Spray Cone Angle there were no standard method have been used. The study or experiment that are related to this topic that have been conducted by other researchers before have used their own method to get the value of spray penetration length and spray cone angle.

Therefore, in this study the scale method was used to measure the spray penetration length and the spray cone angle, where there were a line that was set at 10mm in the spray image. The 10mm line is the real measurement of the design that is used in the simulation. As the spray image of the result was print out, the spray penetration length and spray cone angle of the spray image was measured by using the steel ruler. To determine the ratio for the real measurement, the real value of design which is 10mm will be divided to the print out measurement of the same point. The ratio value then will be multiply with the print out measurement value of the total spray penetration length and spray cone angle for each results. Figure 3.23 shows the scale value for real measurement that was set at 10 mm, then the figure also show the spray penetration length and spray cone angle line measurement. This method are been applied to all result that are obtained in the simulation.

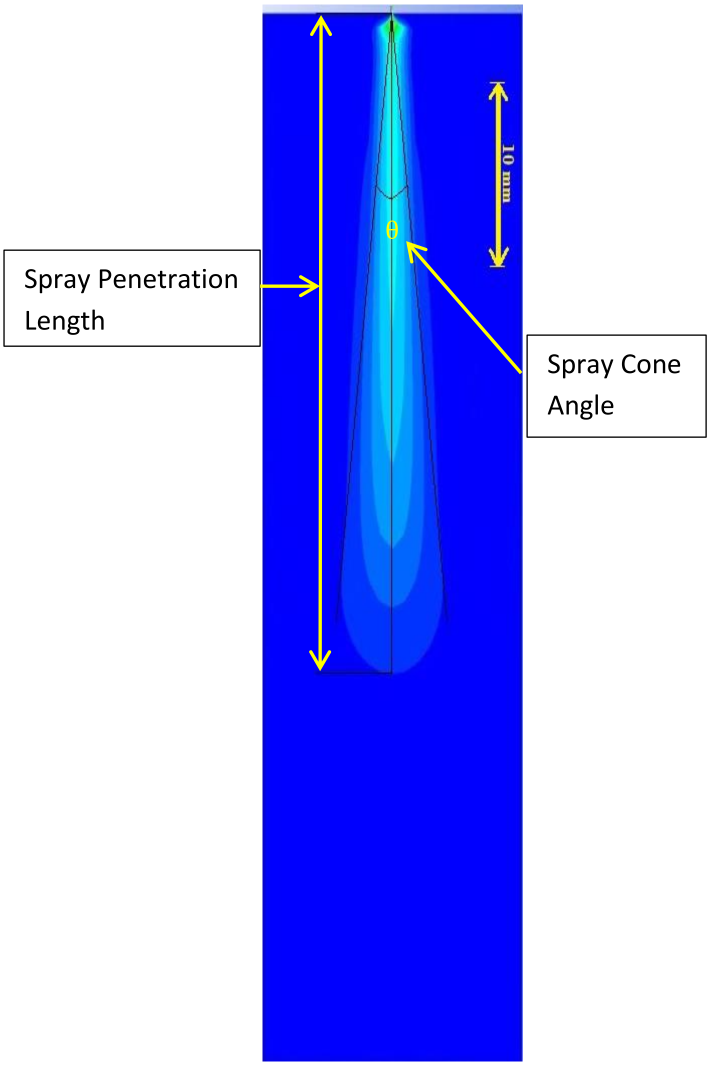


Figure 3.23: The Measurement Scale.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

In this chapter it will discuss about the result obtained from the simulation analysis on the various boundary condition that has been conducted using ANSYS Fluent analysis. The result is presented by the spray image, table and graph to show the different spray physical and characteristic affect. The fuel that has been tested in the simulation is diesel fuel with different ambient temperature and injection pressure and different nozzle holes diameter size.

4.2 SPRAY IMAGE

From the simulation using ANSYS Fluent software, the spray images were obtained in contour line type of spray. In complex flow, it is often to track the flow of discrete particles through the flow field, but the contour line also can be used to determine the flow properties.

Contour lines are the lines of constant magnitude for a selected variable (isotherms, isobars, etc.). A profile plot draw these contours projected off the surface along a reference vector by an amount proportional to the value of the plotted variable at each point on the surface. Figure 4.1 shows the spray image in contour line type.

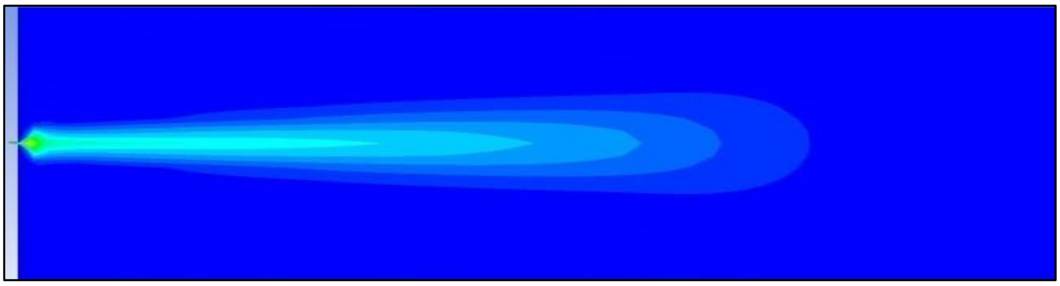


Figure 4.1: Spray Image Contour Line Type.

4.3 VARIOUS TEMPERATURE AND PRESSURE

In order to analyze the spray characteristic changes due to temperature and pressure, the various temperature and pressure was set in this simulation. The nozzle injector diameter hole size and the hole length is remain constant at 0.2 mm and 2.0 mm respectively. The ambient temperature, T_{amb} was set in range of 273 ~ 1000 K. For the injection pressure, P_{inj} it was set in range if 5 Kpa ~ 150 Mpa of changes.

4.3.1 Temperature at 273 K

For the first boundary condition, the temperature was set at very low temperature which is 273 K and then simulate by various pressure that are mention above, but for the result only shows the selected pressure which is 10 KPa, 100 KPa and 70 MPa. The results for 273 K are shown in Figure 4.2.

Figure 4.2 below shows the result of the spray characteristic for 273 K at temperature for 10 KPa, 100 KPa and 70 MPa of injection pressure respectively. From the figure, the spray penetration length and spray cone angle can be determined. Its show that the spray penetration length for the 10 KPa is the shortest that is about 38.8597 mm compare to the other two pressures. For the 100 KPa the spray penetration length was slight increased to 39. 1737 mm, but at the 70 MPa, the spray length is 39.5415 mm. It showed a highly significant of spray penetration length at the higher injection pressure.

The 10 KPa spray has a biggest value of spray cone angle compare to the other two sprays which is 15.93°. The value of spray cone angle slightly decreased when the injection pressure was increase to 100 KPa at about 13.68° and it continue decreased to 11.42° for 70 MPa. This is because, at a very high injection pressure, the spray has more energy to form a long spray so that the spray cone angle will decreased by increase of length of the spray.

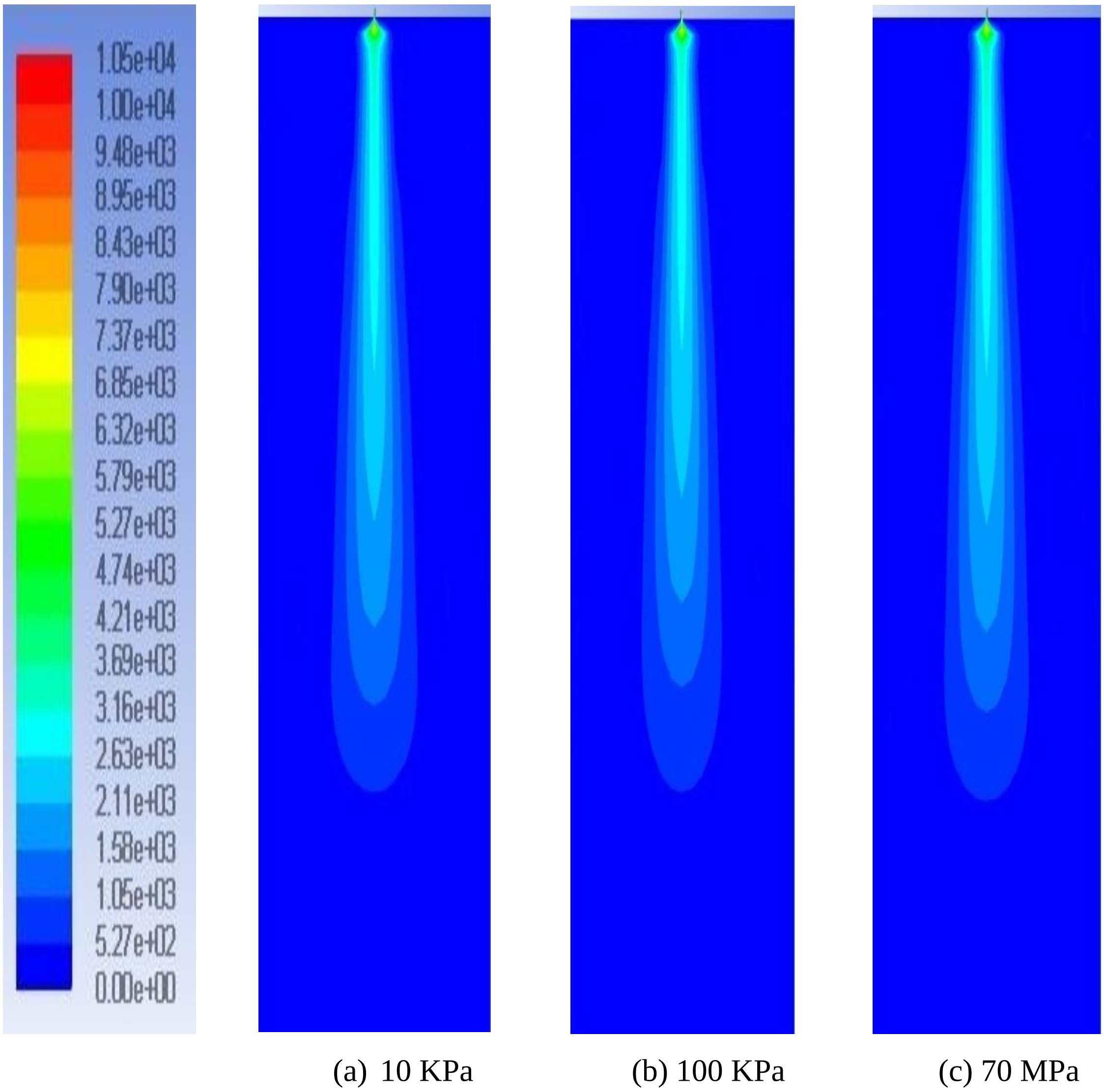


Figure 4.2: The Spray Contour at 273 K; (a) 10 KPa, (b) 100 KPa and (c) 70 Mpa

4.3.2 Temperature at 300 K

Second boundary condition, the temperature was increased to 300 K and then been simulate by the same pressure as before. The Figure 4.3 shows the results for 300 K of temperature.

Figure 4.3 below shows the spray image in contour line type for the simulation at 300 K with various injection pressures which is 10 KPa, 100 KPa and 70 MPa respectively. From the results, its show that by increasing the injection pressure, the spray penetration length will also increases. A slight increment of spray penetration length value when the injection pressure was tested from 10 KPa to 100 KPa at about 38.5690 mm to 39.0201 mm, but when it tested at 70 MPa, the spray penetration length was highly increased to 39.2457 mm.

For the spray cone angle at 10 KPa, the angle is 13.68^o. The angle show a slight decrease for every increased of injection pressure. For 100 KPa, the angle is 11.42^o and 10.96^o for the 70 MPa. This is because at the low injection pressure, the spray did not have enough force to form a long spray pattern so that the spray will easily spread to the chamber surrounding and cause the spray cone angle become higher.

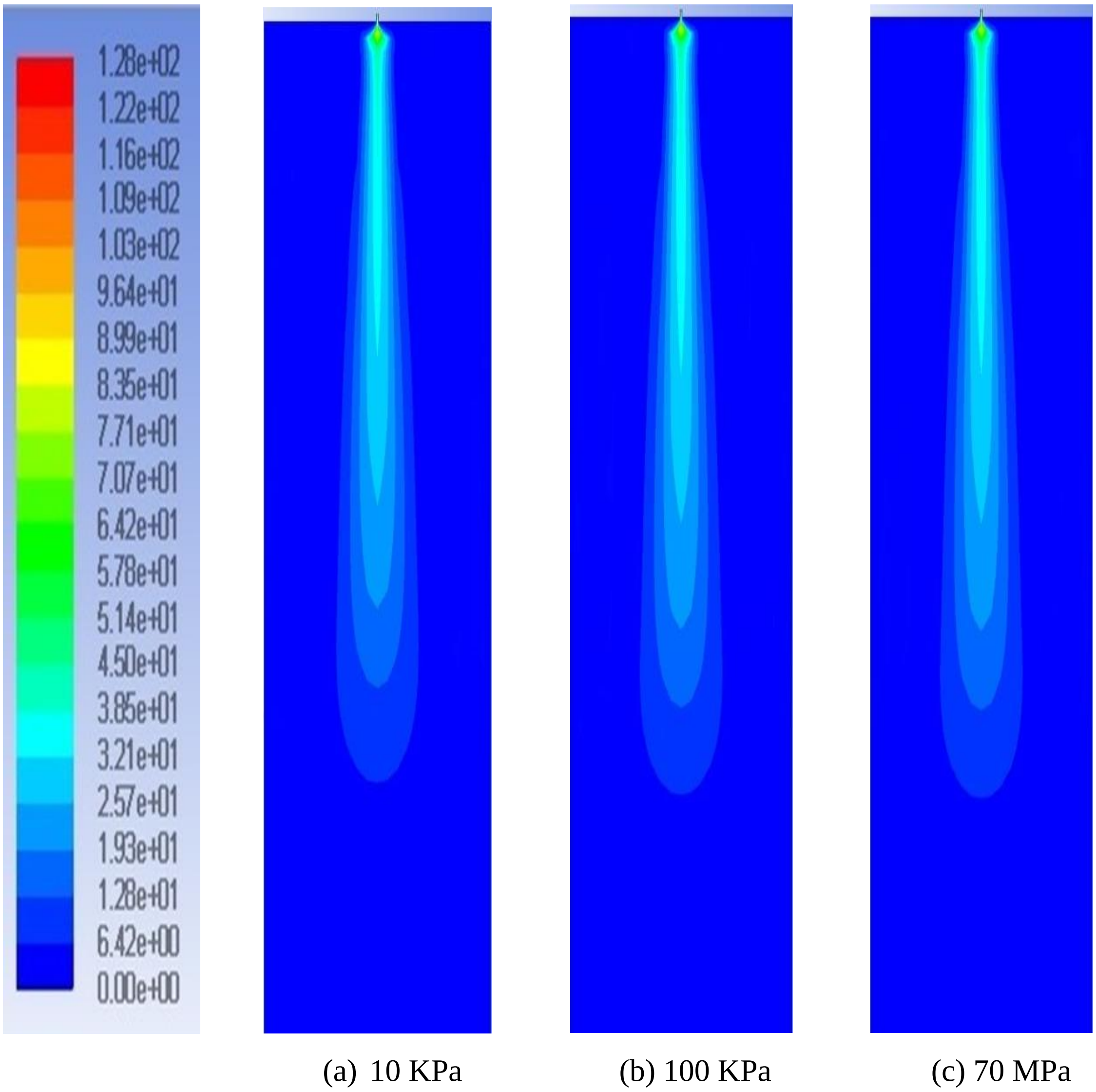


Figure 4.3: The Spray Contour at 300 K; (a) 10 Kpa, (b) 100 KPa and (c) 70 MPa.

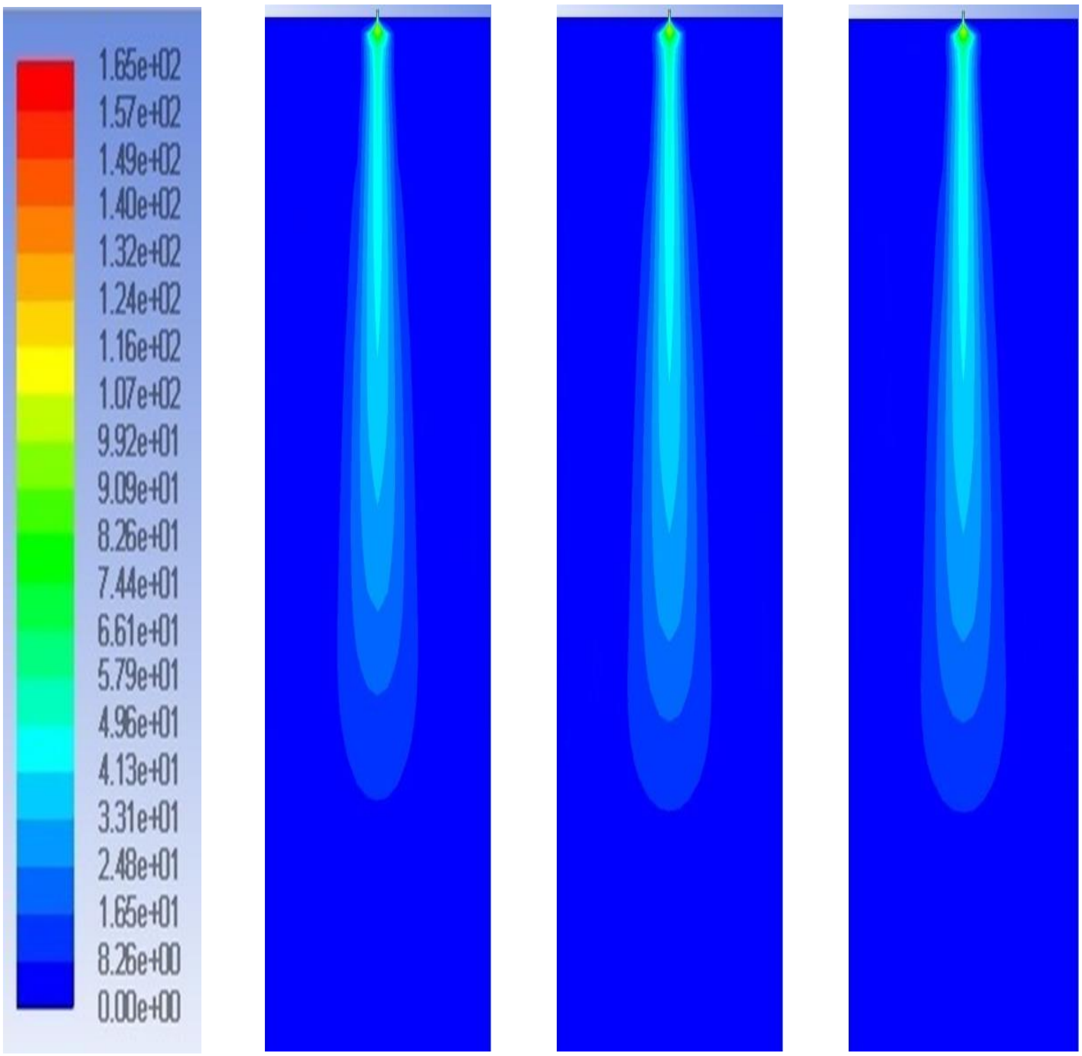
4.3.3 Temperature at 500 K

The simulation then was simulated for the third boundary condition where the temperature was set at 500 K and the pressure is remains at 10 KPa, 100 KPa and 70 Mpa. The spray contours result for this simulation is shown in Figure 4.4.

From the Figure 4.4 below, the spray contour image show the result for the spray at temperature 500 K. The patterns of increment in spray penetration length value when the injection pressures are increased are showed in the result. At 10 KPa, 100 KPa and

70 MPa the spray penetration length is 38.7946 mm, 39.2457 mm and 39.4712 mm respectively.

Meanwhile, for the spray cone angle the value showed the decrement pattern when the injection pressure are increased. For the 10 KPa, the angle is about 13.68^o, at the 100 KPa of injection pressure the angle is 13.23^o and for the 70 MPa the spray cone angle is 11.42^o. This pattern of decrement is similar to the previous result at 273 K and 300 K.



(a) 10 KPa (b) 100 KPa (c) 70 MPa

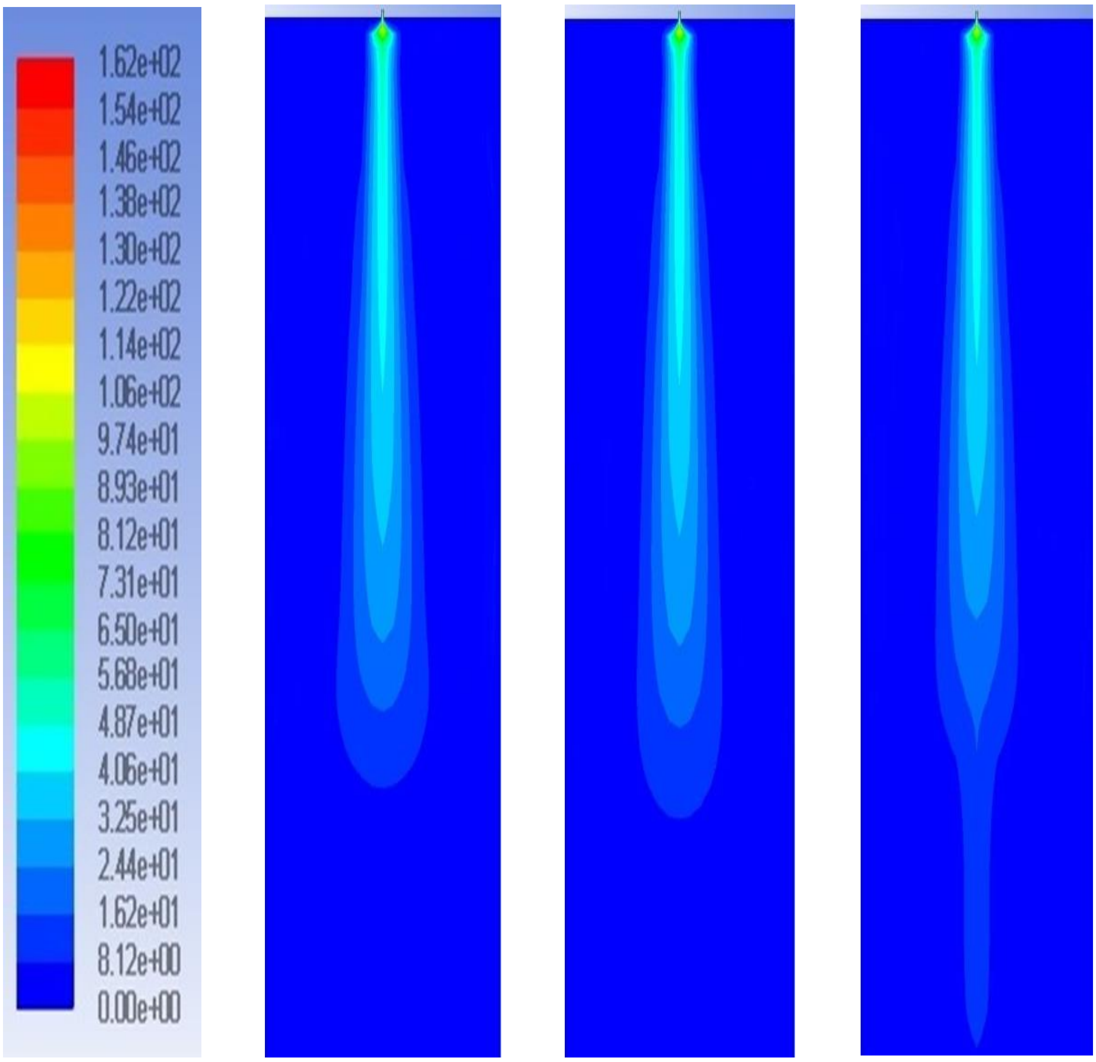
Figure 4.4: The Spray Contour at 500 K; (a) 10 Kpa, (b) 100 KPa and (c) 70 MPa.

4.3.4 Temperature at 1000 K

For the last boundary condition, the simulation was set to simulate at an extreme temperature which is 1000 K. The pressure was various from 10 KPa, 100 KPa, and 70 MPa. Figure 4.5 shows the result in contour line for simulation at 1000 K.

The Figure 4.5, (a), (b), and (c) below showed the result of the spray image for 1000 K of ambient temperature with various injection pressures which is 10 KPa, 100 KPa and 70 MPa respectively. The similar pattern of increment in spray penetration length and decrement in spray cone angle values are also obtained in this 1000 K result. The spray penetration length from 10 KPa to 100 KPa show significant different at about 38.0352 mm to 39.4712 mm, but for the 70 MPa result it showed tremendous different in spray length value which is about 50.7136 mm. This is because when the fuel is applied to the high injection pressure the bond between the fuels is easier to break up to droplet and make it less resistance to move forward compare to the fuel with lower injection pressure that have high resistance to break up.

For the spray cone angle, at 10 KPa, 100 KPa and 70 MPa, the spray cone angle is 13.68^{o}, 13.23^{o} and 11.87^{o} respectively. It showed that when the injection was increased the spray cone angle will decrease.



(a) 10 KPa (b) 100 KPa (c) 70 MPa

Figure 4.5: The Spray Contour at 1000 K; (a) 10 Kpa, (b) 100 KPa and (c) 70 MPa.

The spray penetration length and spray cone angle was measured and plotted in graphs. Figure 4.6 shows the Spray Penetration Length vs Ambient Temperature (0.2mm).

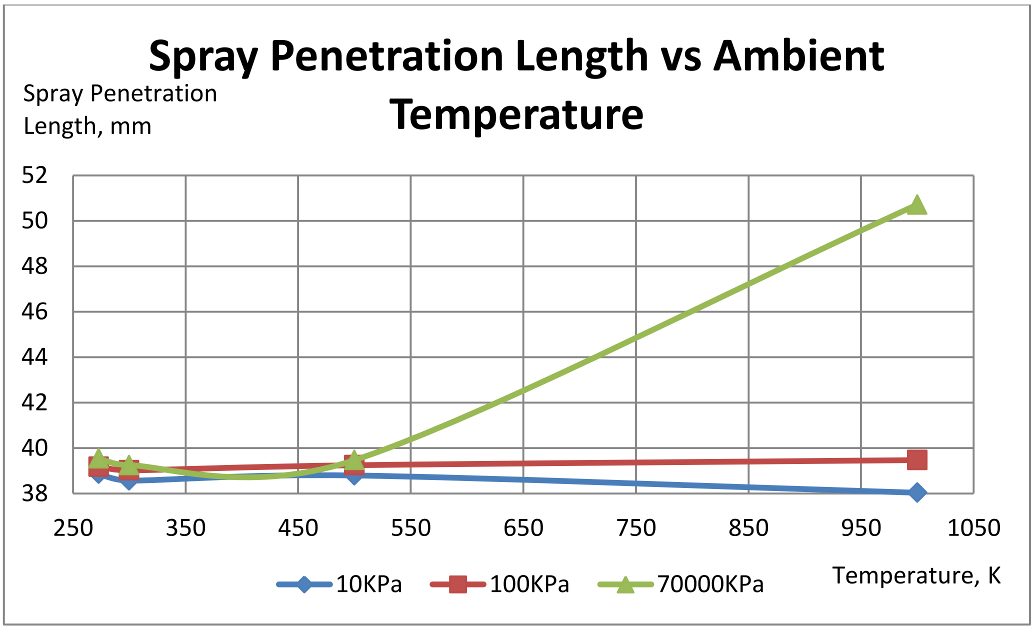


Figure 4.6: Spray Penetration Length vs Ambient Temperature Graph (0.2 mm).

Figure 4.6 shows the Spray Penetration Length vs Ambient Temperature (0.2 mm) graph. The blue line and circle marker represent the spray length for the 10 KPa, meanwhile the red line and square marker represent the spray length for the 100 KPa, and the green line and triangle marker represent the spray length for the 70 MPa.

For the 10 KPa at 273 K, the value of the spray penetration length is 38.8597 mm, it decrease to 38.5690 mm at 300 K of ambient temperature. The angle value at the 500 K is show the increment at about 38.7946 mm and decrease again at about 38.0352 mm for 1000 K. For the 100 KPa of injection pressure, at the 273 K the spray penetration length is about 39.1737 mm, it decreased to 39.0201 mm at 300 K and increased for the 500 K of ambient temperature at about 39.2457 mm and it continue the increment for 1000 K at 39.4712 mm. At the pressure of 70 MPa, for 273 K of ambient temperature, the spray penetration length is about 39.5415 mm and decreased at about 39.2457 mm at 300 K. The spray length then increased back at about 39.4712 mm for 500 K and extremely increased at 1000 K to be 50.7136 mm.

The pattern of spray length in intention of various ambient temperature show a slight decrement at the low temperature, but it increased again when the ambient temperature was increased to the higher temperature. Except for the spray length of 70 MPa at 1000 K of ambient temperature, all of the spray length value is lower than the first temperature which is 273 K. This pattern is equal and same as the result from the research which concluded that the increase in chamber temperature will cause the decrement of spray length - R.Morgan and J.Way.

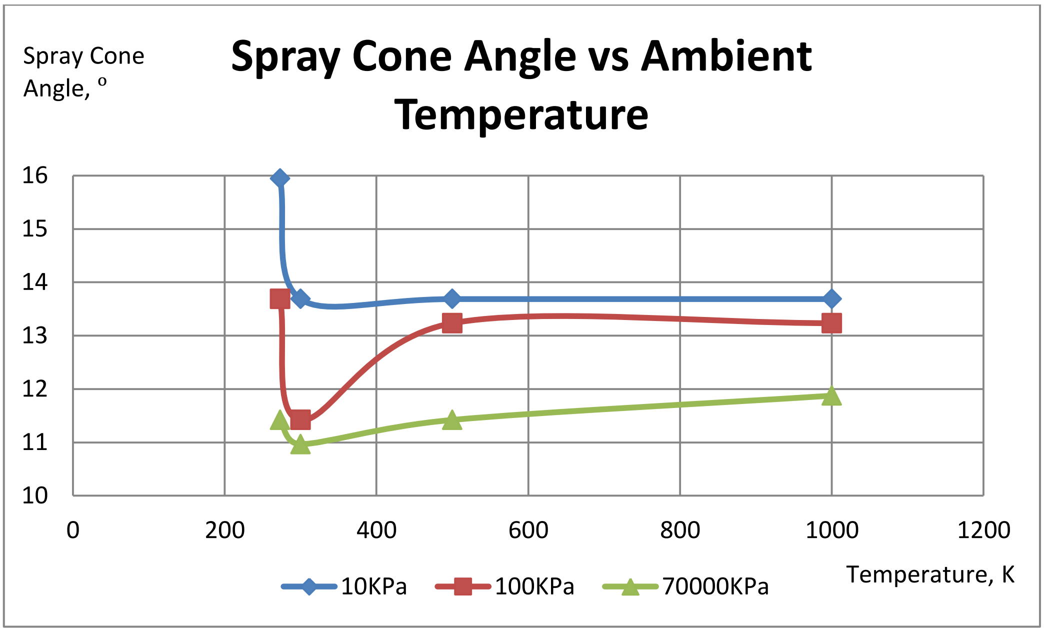


Figure 4.7: The Spray Cone Angle vs Ambient Temperature Graph (0.2 mm).

Figure 4.7 shows the Spray Cone Angle vs Ambient Temperature graph (0.2 mm). The blue line and circle marker represent the spray length for the 10 KPa, meanwhile the red line and square marker represent the spray length for the 100 KPa, and the green line and triangle marker represent the spray length for the 70 MPa.

The graph show the plotted result for the spray cone angle in intention of various ambient temperatures. For the 10 KPa of injection pressure, the spray cone angle showed the decrement from 15.93° at 273 K to the 13.68° at the 300 K and the values are stay constant at 13.68° even the temperature was increased to 500 K and 1000 K. Then, for the 100 KPa, at 273 K it still shows the wider spray angle which is about 13.68° compared to the other temperature at the same injection pressure. By the increment of the temperature at 300 K, the angle decreased to 11.42°, but it showed the increment for 500 K at about 13.23^o and became constant when the temperature was increased to 1000 K. At the 70 MPa of injection pressure, 11.42^o was measured at 273 K and the value was decreased to 10.96^o at the temperature of 300 K, but when the temperature was increased to 500 K, the spray angle are increased back to 11.42^o, and continue to increase at about 11.87^o for the 1000 K of ambient temperature.

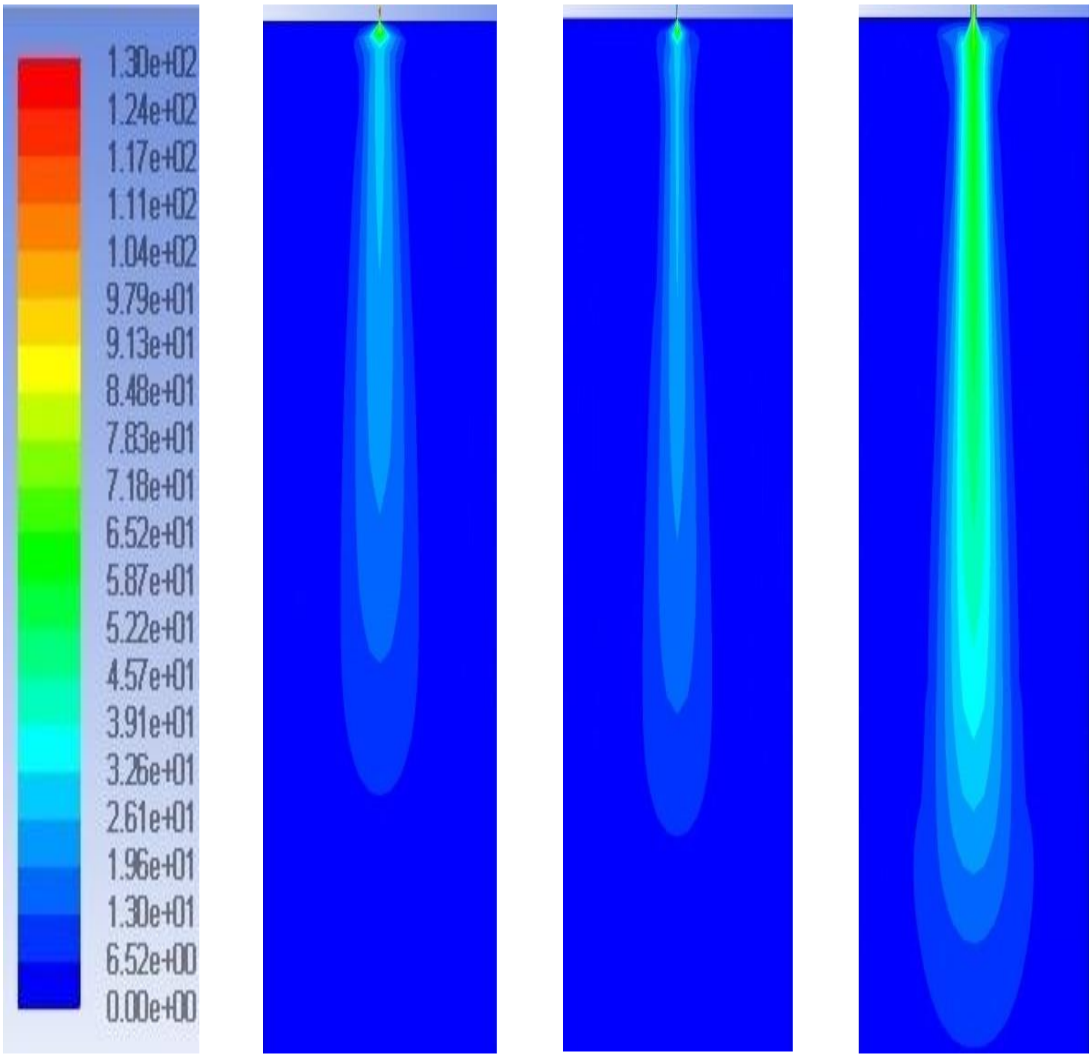
The pattern of the spray cone angle in this study showed that, at the lower temperature, the spray cone angle will be the widest angle for each injection pressure. The values are decreased in term of increased ambient temperature. This is because at the lower injection pressure, the sprays injected into the combustion chamber have less force to form a long spray. Adding with the lower ambient temperature, the sprays have a lower rate of evaporation, so that the sprays are more attracted to spread widely into the chamber. Compare to the higher injection pressure and the higher ambient temperature the spray injected has more force to form a long spray rather than spread widely into the chamber and at the very high of the evaporation rate, the spray will rapidly transform into the vapor phase. So that the spray image form the small angle of spray cone angle.

4.4 VARIOUS DIAMETER NOZZLE HOLE SIZE.

The different nozzle diameter hole size also influence the changes in the spray characteristic. Next simulation the focus is on the spray characteristic that using the different size of nozzle diameter. The nozzle diameter hole size was various from 0.1 mm (Nozzle 1, N1), 0. 2 mm (Nozzle 2, N2) and 0.5 mm (Nozzle 3, N3). For this simulation the injection pressure, P_{inj} and ambient temperature, T_{amb} was set as constant which is 100 KPa and 300 K respectively. Figure 4.8 shows the result of the spray characteristic with various nozzle diameter hole size. The spray penetration length and spray cone angle was measured and plotted into the graph.

Figure 4.9 show the Spray Penetration Length vs Nozzle Diameter graph. From the graph, it is showed that the spray penetration length for 0.1 mm nozzle diameter hole size is the shortest which is about 38.5690 mm and the spray length increase when the 0.2 mm nozzle diameter hole size was used at about 41.6868 mm. For the 0.5 mm nozzle diameter hole size, it's clearly show the longest of spray penetration length, 45.0928 mm which has the less resistance in chamber to move forward to form longest spray compare to the small nozzle diameter hole size. This shows that the spray characteristic are been influence by the different nozzle diameter hole size. This is because, the different nozzle diameter hole size will affect the injected fuel volume and also the fuel flow rate of the spray. The velocity of the spray also changes according to the initial boundary condition.

Figure 4.10 show the Spray Cone Angle vs Nozzle Diameter graph. From the graph, it show the decrement in the value when the nozzle diameter hole size are increased. Usually the short spray will form the widest spray cone angle compare to the long spray length. For the 0.1 mm, the angle is about 18.18^{o}, the angle decrease at about 14.81^{o} for 0.2 mm of nozzle diameter hole size and 11.42^{o} of angle for 0.5 mm of nozzle hole diameter.



(a) N1 (b) N2 (c) N3

Figure 4.8: Spray Characteristic of the Various Nozzle Diameter hole; (a) N1, (b) N2 and (c) N3.

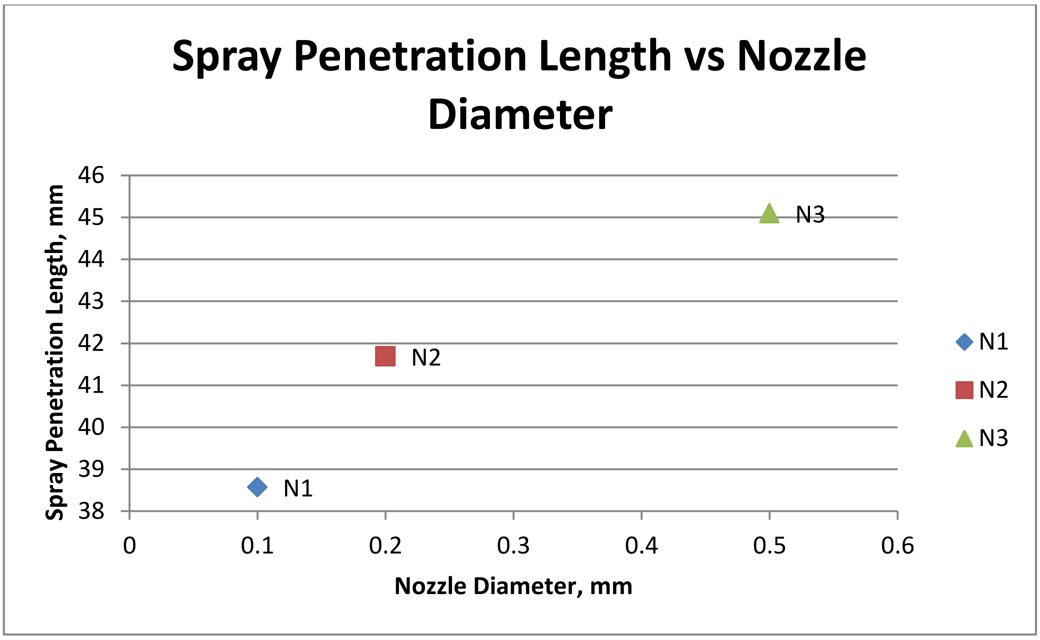


Figure 4.9: Spray Penetration Length vs Nozzle Diameter (P_{inj} = 100 KPa, T_{amb} = 300K).

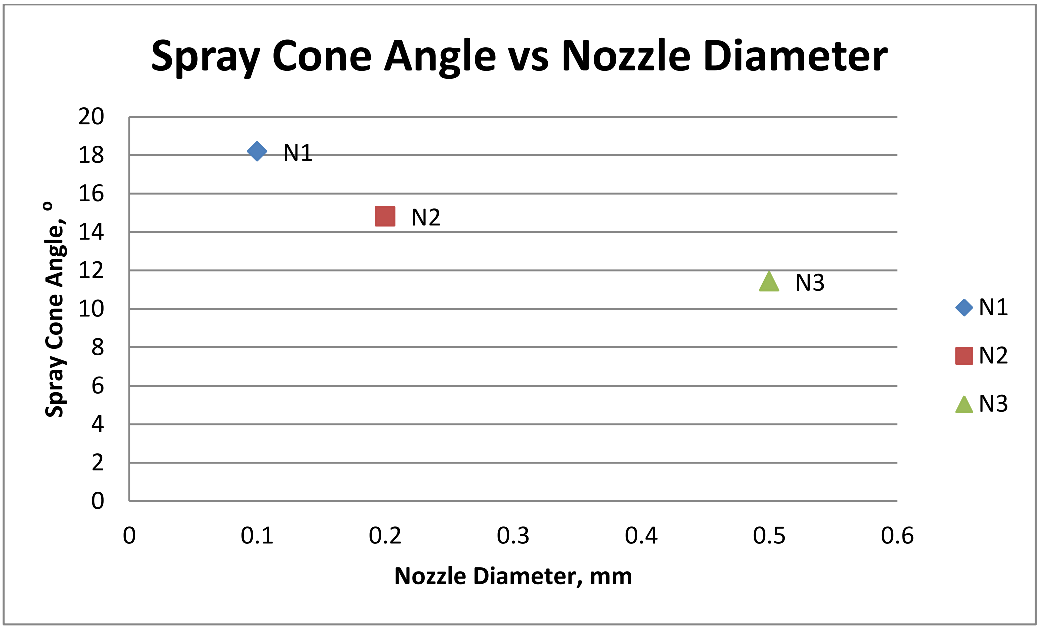


Figure 4.10: Spray Cone Angle vs Nozzle Diameter (P_{inj} = 100 KPa, T_{amb} = 300 K).

4.5 DOUBLE HOLES NOZZLE INJECTOR

In this study, the spray characteristic is also been focus for the double holes of nozzle injector. The simulation setup was set as same as single hole spray simulation setup includes the procedure and also the boundary condition. The intention of the simulation for the double hole spray characteristic is also same as simulation for single hole spray characteristic which is divided into three parts, the first part is the intention due to the changes of temperature, where the temperature was various between 273 K ~1000 K. The second part is on the intention of the changes of pressure, where the pressure was various from 5 KPa ~ 150 MPa. For the third part is the intention in changes of the nozzle diameter hole size that was simulated using 0.1 mm, 0.2 mm and 0.5 mm.

From the journal that have been reviewed along this study, for the multi-hole, the spray image supposed to be approximately as shown in Figure 4.11. The research on the spray pattern of VCO-nozzle under low needle lift condition has been performed with large-scale transparent nozzle and confirmed that eccentricity of the needle tip and partial hydraulic flip are responsible for different spray pattern which is directed more bias to one side and has a small spray angle because a side of fuel flow boundary inside hole may be remaining detached till the exit – Choongsik Bac and Jinsuk Kang.

Unfortunately, due to missing code in the procedure setup for double hole in ANSYS Fluent software, the result that obtained from the simulation is not as expected as shown in Figure 4.12. The result show the weird spray pattern of straight double hole so the result for the double hole are undefined.

Figure 4.11: The Spray Pattern for Multi-Hole of Nozzle.

Source: P.G. Aleiferis, Z.R. van Romunde. 2013.

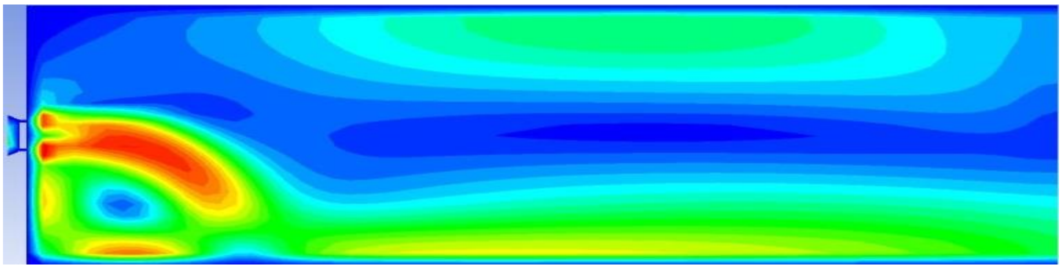


Figure 4.12: The Simulation Spray Pattern for Double Hole of Nozzle.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 CONCLUSION

The objective of this study is to determine the effective nozzle type for better spray atomization in different injection pressure, ambient temperature and nozzle diameter hole size effect to the spray characteristic in engine chamber. By using the ANSYS Fluent, the simulation gives a positive result on spray characteristic. Spray characteristic that are considered in this study is spray penetration and spray cone angle.

Spray penetration length are increased as the injection pressure are increased, but spray cone angle are decreased. In term of increased of the ambient temperature, the spray penetration decreased and the spray cone angle results value is increased. The longest spray penetration length for all tested injection pressure and ambient pressure is 50.7136 mm which is at 70 MPa and 1000 Kelvin. But this result cannot be accepted because it is too long and are out of the range of normal spray penetration. If the spray penetration is too long, there is a risk of impingement on the wall of the combustion chamber, which may lead to fuel wastage and formation of soot. Then the spray penetration at 39.5415 mm for 70 MPa and 273 Kelvin are recognized to be the longest spray length. For the spray cone angle, at 10 KPa of injection pressure and 273 Kelvin of ambient temperature are recognize as the widest angle of spray which is at 15.93^{o}.

The same pattern of spray characteristic are also shown in intention of different diameter nozzle hole size, where the spray length are increased while the nozzle diameter are increased and spray cone angle decreased while the spray length is increased. The longest spray length is show at 0.5 mm of nozzle diameter at about

45.0928 mm. but it is too long and cannot be acceptable. So the spray lengths of 41.6868 mm for 0.2 mm are determined as the acceptable spray length. It can be conclude that the 0.2 mm diameter nozzle hole size can give the effective spray characteristic. Spray characteristic play important part in engine combustion. The higher spray penetration length and spray cone angle give the better fuel spread in the engine chamber. This could contribute to better combustion thus will affect the engine performance and will decrease the emission from exhaust as well.

For the double-hole spray characteristic, the unexpected result was obtained and cannot be accepted. This is because of the missing code in the procedure setup for double-hole in ANSYS Fluent software. The future deep study on double-hole simulation needs to be done.

5.2 RECOMMENDATIONS

In this present study, a lot of the problem was faced when doing the experiments. For continuing this project or further research, some of the recommendations had been made and should be considered for better analysis and accuracy results. The better meshing for the project must be consider as it affect the accuracy of the result, the smaller and accurate meshing will produce the better and accurate result. To get the smaller and better meshing, the high performance of computer needed to support it. For the future study, the deep study and understanding on the ANSYS Fluent software must to be make in order to obtain the good result for multi-hole spray characteristic and having good instrument devices was a must for taking the best and effectively data have.

REFERENCES

Bae, C. H., Kang, J. 2000. "*Diesel Spray Characteristics of Common-Rail VCO Nozzle Injector"*. THIESEL. Korea Advanced Institute of Science and Technology, Korea.

Ficarella, A., Laforgia, D., Starace, G., Damiani, V. 1997. *"Experimental Investigation of the Sprays of an Axi-Symmetric Nozzle of a Common-Rail High Pressure Electro Injector"*. SAE Paper 970054. Polytecnic of Bari, Italy.

Haeyoung J., Kihyung L., Yudi Ikeda. 2007. *''Investigation of the Spray Characteristics for a Secondary Fuel Injection Nozzle Using a Digital Image Processing Method''*. Journal. Department of Mechanical Engineering, Hanyang University, Korea.

Hiroyasu, H., Arai, M. 1990. *"Structures of Fuel Sprays in Diesel Engines"*. SAE Paper 900475. University of Hiroshima, Japan.

Hosoya, H., Obokata, T. 1993. *"Effect of Nozzle Configuration on Characteristics of Steady-State Diesel Spray"*. SAE Paper 930593. Japan Electronic Control System Co. Ltd and Gunma University, Japan.

Jimenez, J., Castro, F., Tinaut, F., Gimenez, B. 2000. *"The Tip Evolution of an Evaporative Intermittent Fuel Spray"*. THIESEL. Thermofluidynamic Processes in Diesel Engines, pp. 67-74.

John Agudelo, Andres Agudelo, Pedro Benjumea. 2009. *''Study of Diesel using Computational Fluid Dynamics''*. Journal. University of Antioquia, Medellin, Colombia.

Jungmo Oh, Kihyung Lee, Haeyoung Jeong. 2008. *''Study on the Spray Behavior and Diesel Fuel Distribution Characteristic of a Second Injector for a Lean 〖NO〗_x Trap Catalyst''*. Journal. Hanyang University, Korea.

Kennaird, D. A., Crua, C., Lacoste, J., Heikal, M., Gold, M. R., Jackson, N. S. 2002. *"In-Cylinder Penetration and Break-Up of Diesel Sprays Using a Common-Rail Injection System"*. SAE Paper 2002-01-1626. University of Brighton, United Kindom.

Koo, J. H., Hong, S. T., Shakal, J. S., Goto, S. 1997. *"Influence of Fuel Injector Nozzle Geometry on Internal Flow Characteristics"*. SAE Paper 970354. Hankuk Aviation University, Korea.

Kourosh Karimi. 2007. *''Characterization of Multiple-Injection Diesel Sprays at Elevated Pressure and Temperatures''*. Ph.D Thesis. University of Brighton in collaboration with Ricardo, United Kindom.

Lacoste Julien. 2006. *‘‘Characteristic of Diesel Sprays at High Temperatures and Pressures’’*. Ph.D Thesis. University of Brighton in collaboration with Ricardo Consulting Engineers.

Levy, N., Amara, S., Champoussin, J. C., Guerrassi. 1997. *“Non-Reactive Diesel Spray Computations Supported by PDA Measurements”.* SAE Paper 970049. Ecole Centrale de Lyon, France.

Naber, J. D., Siebers, D. L. 1996. *“Effects of Gas Density and Vaporization on Penetration and Dispersion of Diesel Sprays”.* SAE Paper 960034. Sandia National Labs.

R. Morgan, J.Wray, D.A Kennaird, C.Crua, M.R. Heikal. 2001. *‘‘The Influence of Injector Parameters on the Formation and Break-Up of a Diesel Spray’’*. SAEPaper 2001-01-0529. University of Brighton, United Kindom.

Raul Payri, S.Molina, F.J Salvador, J.Gimeno. 2004. *‘‘A Study of the Relation between Nozzle Geometry, Internal Flow and Spray Characteristics in Diesel Fuel Injection Systems’’* International Journal. University Polytechnic of Valencia, Spain.

R.J.H Klein-Douwel, P.J.M Frijters, X.L.J. Seykens, L.M.T. Somers, R.S.G.Beart. 2008. *‘‘Gas Density and Rail Pressure Effects on Diesel Spray Growth from a Heavy-Duty Common Rail Injector’’*. Journal. Eindhoven University of Technology, Eindhoven, The Netherlands.

Tiegang F., Robert E. Coverdill, Chia-fon F Lee, Robert A. White. 2008. *‘‘Effect of Injection Angle on Combustion Processes using Multiple Injection Strategies in an HSDI Diesel Engine’’*. Journal. Department of Mechanical and Aerospace Engineering North Carolina State University, United State. Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, United States.

Warrick, C. B., Su, T. F., Farrel, P.V. 1996. *“Temperature Effects on Fuel Sprays from a Multi-Hole Nozzle Injector”.* SAE Paper 962005. University of Wisconsin, Madison.

APPENDICES

APPENDIX A

Spray Penetration Length for Various Injection Pressure and Ambient Temperature.

Pressure / Temperature	10	100	70000
273	38.8597	39.1737	39.5415
300	38.569	39.0201	39.2457
500	38.7946	39.2457	39.4712
1000	38.0352	39.4712	50.7136

APPENDIX B

Spray Cone Angle for Various Injection Pressure and Ambient Temperature.

Pressure / Temperature	10	100	70000
273	15.9392	13.6855	11.4211
300	13.6855	11.4211	10.9671
500	13.6855	13.2334	11.4211
1000	13.6855	13.2334	11.8748

APPENDIX C

Spray Penetration Length and Spray Cone Angle for Various Nozzle Diameter.

Pressure / Temperature	1000 KPa	1000 KPa
0.1mm	38.569	18.1805
0.2mm	41.6868	14.8138
0.5mm	45.0928	11.4211

APPENDIX D

Gantt Chart / Project Schedule for FYP 1

No.	Activities	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14	W15	W16
	Project Progress																
1	Meeting with Supervisor																
2	Briefing on the FYP title, problem statement, objective and scope																
3	Searching for journal and reference books																
4	Literature review																
5	CAD drawing																
6	Learn Fluent simulation software																
7	Submission of Draft 1 with logbook																
8	Presentation arrangement																
9	FYP submission																
	Thesis Progress																
1	Writing report: Chapter 1																
2	Writing report: Chapter 2																
3	Writing report: Chapter 3																
4	finalizing																
6	Final preparation																

APPENDIX E

Gantt Chart / Project Schedule for FYP 2

No.	Activities	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14	W15	W16
	Project Progress																
1	Learn about spray in engine	■	■	■	■	■	■	■	■	■	■						
2	Perform simulation						■	■	■	■	■	■					
3	Literature review on previous research				■	■	■	■	■	■	■	■	■				
4	Obtain full result												■	■			
5	Presentation arrangement														■		
6	Submission of Thesis															■	■
7	FYP Presentation															■	
	Thesis Progress																
1	Writing report: Chapter 4				■	■							■	■	■	■	
2	Writing report: Chapter5													■	■	■	
6	Finalize thesis															■	■