# UNIVERSITI MALAYSIA PAHANG

## BORANG PENGESAHAN STATUS TESIS ◆

**JUDUL: ANALYSIS OF DIESEL SPRAY CHARACTERISTIC AT HIGH PRESSURE INJECTION**

**SESI PENGAJIAN: 2011/2012**

Saya, **NORSYAMSUL SYAZWAN BIN MOHD NUJI (870519-03-5453)**

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

1. Tesis ini 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:

**Lot 355, Kampung Padang Raja,**
**16400 Melor,Kota Bharu**
**Kelantan.**

**DR ABDUL ADAM BIN ABDULLAH**
(Nama Penyelia)

Tarikh: **15 JUN 2012** Tarikh: **15 JUN 2012**

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 SULIT 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).

# ANALYSIS OF DIESEL SPRAY CHARACTERISTIC AT HIGH PRESSURE INJECTION

NORSYAMSUL SYAZWAN BIN MOHD NUJI

This thesis is submitted as partial fulfillment of the requirements for the award of the Bachelor of Mechanical Engineering (Automotive)

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

**UNIVERSITI MALAYSIA PAHANG**

**FACULTY OF MECHANICAL ENGINEERING**

We certify that the project entitled “*Analysis of Diesel Spray Characteristic at High Pressure Injection* “is written by *Norsyamsul Syazwan Bin Mohd Nuji*. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature : ………………………..

Name of Examiner : PROF. K.V SHARMA

Position : LECTURER OF MECHANICAL ENGINEERING

Date : ………………………..

**SUPERVISOR'S DECLARATION**

I hereby declare that I have checked this project report and in my opinion, this project is adequate in terms of scope 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 OF MECHANICAL ENGINEERING

Date : …….......................

## STUDENT'S DECLARATION

"All the trademark and copyrights use herein are property of their respective owner. References of information from other sources are quoted accordingly; otherwise the information presented in this report is solely work of the author."

| | |
|---|---|
| Signature | : ……………………….. |
| Name | : NORSYAMSUL SYAZWAN BIN MOHD NUJI |
| ID Number | : MH09007 |
| Date | : ……………………….. |

## DEDICATION

Specially dedicated to my beloved family, and those who have guided and inspired me throughout my journey of learning

## ACKNOWLEDGEMENT

Throughout the development of this project I have gained chances to learn new skills and knowledge. I wish to express my sincere appreciation and gratitude to my supervisor, Dr. Abdul Adam bin Abdullah for his continuous guidance, concern, encouragement and advices which gave inspiration in accomplishing my final year project.

Special thanks to University Malaysia Pahang for supporting and providing equipment and information sources that assisted my studies and projects.

My sincere appreciation to the lecturers of Faculty of Mechanical Engineering who have put in effort to the lectures and always nurture and guide us with precious advices. Thank you for sharing those experiences.

To all my lovely current and ex roommates and friends who always willingly assist and support me throughout my journey of education, you all deserve my wholehearted appreciation. Many thanks.

Last but not least, my beloved family members who always stand by my side concerning the ups and downs of my life. Home is where I find comfort. Endless love.

## ABSTRACT

In diesel combustion, spray evaporation and mixture formation during ignition delay period play an important role in ignition, combustion and emission production. Spray evaporation begins immediately after start of fuel injection under the condition of high ambient temperature, in particular, at the middle of the spray. Spray atomization is fast promoted at this region, leading to ignition. The ambient temperature and injection pressures affect the droplets size and the number of droplets. In this project, the fuel will be injected at various injection parameters inside spray chamber in order to study the affect of that parameter towards spray characteristics. An analysis study was performed to investigate the macroscopic spray structure and the spray characteristics of high-pressure injector for the diesel engine. The spray structure and microscopic characteristics of high-pressure diesel injector were investigated when fuel was injected at various injection pressures and different nozzle diameter. Spray developing process, spray cone angle and spray tip penetration were obtained by using the software simulation of ANSYS-FLUENT, and the quantitative result of spray characteristics will be analyzes.

# ABSTRAK

Dalam pembakaran diesel, penyejatan semburan dan pembentukan campuran dalam tempoh lengah memainkan peranan yang penting dalam sistem pencucuhan, pembakaran dan pelepasan. Penyejatan semburan bermula serta-merta selepas permulaan suntikan bahan api di bawah keadaan suhu ambien yang tinggi, khususnya, pada pertengahan semburan. Pengabusan semburan terjadi di rantau ini, yang membawa kepada penyalaan. Suhu ambien dan tekanan suntikan mempengaruhi saiz titisan dan bilangan titisan.Dalam projek ini, bahan api akan disuntik pada pelbagai parameter suntikan di dalam kebuk semburan untuk mengkaji kesan parameter tersebut terhadap ciri-ciri semburan. Satu kajian analisis telah dilakukan untuk menyiasat struktur semburan makroskopik dan ciri-ciri semburan suntikan pada tekanan yang tinggi untuk enjin diesel. Struktur semburan dan ciri-ciri mikroskopik pada tekanan tinggi suntikan diesel telah disiasat apabila bahan api disuntik pada pelbagai tekanan suntikan dan muncung diameter yang berbeza . Proses membangunkan semburan, sudut kon semburan dan panjang semburan telah diperolehi dengan menggunakan perisian simulasi ANSYS-FLUENT, dan keputusan ciri-ciri semburan yang pelbagai akan di analisis.

# LIST OF TABLES

# LIST OF FIGURES

## TABLE OF CONTENTS

# CHAPTER 1

# INTRODUCTION

## 1.1 INTRODUCTION

In the diesel engine, combustion and emission characteristics are influenced by fuel atomization, nozzle geometry, injection pressure, shape of inlet port, and other factors. In order to improve fuel–air mixing, it is important to understand the fuel atomization and spray formation processes. So far, to improve the combustion performance and particulate emissions, many researchers have investigated the characteristics of the spray behavior and structure for the high-pressure injector by experimental and theoretical approaches. However, many studies about the detailed information of atomization characteristics and spray developing process of high-pressure diesel spray were still needed.

## 1.2 PROJECT BACKGROUND

In diesel combustion, spray evaporation and mixture formation during ignition delay period play an important role in ignition, combustion and emission production. Spray evaporation begins immediately after start of fuel injection under the condition of high ambient temperature, in particular, at the middle of the spray. Spray atomization is fast promoted at this region, leading to ignition. The ambient temperature and injection pressures affect the droplets size and the number of droplets. In this project, the fuel will be injected at various injection parameters inside spray chamber order to study the affect of that parameter towards spray characteristics. Accurate control of diesel spray parameters (timing, delivery, flow rate, pressure, spray geometry, so on.) is the most effective means to influence fuel and air mixing and to achieve both clean burning and high efficiency. The ANSYS –FLUENT software has been used to investigate the spray characteristic development after injection with various high pressures.

The impingement of diesel spray onto interposed surfaces in an IC engine, equipped either with a direct or an indirect injection system, is a fundamental issue affecting mixture preparation prior to combustion and, therefore, also affecting engine performance and pollutant emissions. In this context, the development of diesel spray systems relies on accurate knowledge of the fluid dynamic and thermal processes occurring during spray. Injection systems however, are very complex and the background physics requires fundamental studies, performed at simplified flow geometries. In particular, the impact of individual droplets and spray characteristic has been extensively used to describe the behavior of spray impact and to predict its outcome, despite the known fact that a spray does not behave exactly as a summation of individual droplets; then, researchers incorporate all the governing parameters. The present paper offers a critical review of the investigations reported in the literature on spray-wall impact relevant to IC engines, in an attempt to address the rationale of describing spray-wall interactions based on the knowledge of single droplet impacts. Moreover, although the review was first aimed at fuel-spray impingement in IC engines, it also became relevant to provide a systematization of the current state of the art, which can be useful to the scientific community involved with droplet and spray impingement phenomena.

## 1.3 PROBLEM STATEMENT

Researchers are exploring ways to reduce pollution formation in the engine by using clean combustion strategies. In the diesel engine, combustion and emission characteristics are influenced by fuel atomization, nozzle geometry, injection pressure, shape of inlet port, and other factors. In order to improve fuel–air mixing, it is important to understand the fuel atomization and spray formation processes. The ANSYS – FLUENT software has been used to investigate the spray characteristic development.

## 1.4 OBJECTIVES

The objectives of this project are:

(i) To investigate the tip penetration of spray at various injection parameters and different diameter of SAC nozzle.

(ii) To investigate the cone angle of spray at various injection parameters and different diameter of SAC nozzle.

(iii) To investigate the spray developments

## 1.5 SCOPES OF WORK

The scopes of the study are:

(i) Measurement of diesel spray cone angle at various parameters.

(ii) Comparison and measurement of spray penetration length at various parameters.

(iii) Comparison of spray geometry including the development of liquid phase and vapor phase area.

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 INTRODUCTION

The literature review had been carry out with reference from sources such as journal, books, thesis and internet in order to gather all information related to the title of this project. This chapter covers about the previous experiment doing by researcher and to go through the result by experimental and numerical. Today, adoption of diesels in the world would decrease the nation's petroleum consumption. However, diesels emit much higher levels of pollutants, especially particulate matter and NOx (nitrogen oxides). These emissions have prevented more manufacturers from introducing diesel passenger cars.

Researchers are exploring ways to reduce pollution formation in the engine by using clean combustion strategies. A key component to the development of clean combustion is controlling the fuel spray and fuel/air mixing. In the diesel engine, combustion and emission characteristics are influenced by fuel atomization, nozzle geometry, injection pressure, shape of inlet port, and other factors. In order to improve fuel–air mixing, it is important to understand the fuel atomization and spray formation processes. So far, to improve the combustion performance and particulate emissions, many researchers have investigated the characteristics of the spray behavior and structure for the high-pressure injector by experimental and theoretical approaches. However, many studies about the detailed information of atomization characteristics and spray developing process of high-pressure diesel spray were still needed.

Spray structure and atomization characteristics of diesel spray have been investigated by Dennis (1998), Maruyama (2001), Ishikawa (1996), Nimura (1996) and Farrell(1996). They reported that the characteristics of fuel spray for the fuel injector obtained by using the shadowgraphs and particle image velocimetry at various chamber conditions. Yeom (2001) and Su (1996) compared experimental results with numerical ones about spray shapes, axial mean velocity, and mean droplet diameter. In original KIVA code, the breakup of droplet is calculated with the Taylor analogy breakup (TAB) model. In order to improve the calculation accuracy, the breakup model of injection spray is modified by introducing Kelvin–Helmholtz and Rayleigh–Taylor (KH–RT) model used by Su and Beale and Reitz .In spite of these studies, it is still needed to improve the breakup model and verify the predicted results. The objective of this work is to investigate the effect of injection pressure and temperature ambient on the macroscopic spray behavior and atomization characteristics of high-pressure spray the common rail type diesel injection system. The test fuel used in this experiment was diesel fuel with the density of 880 kg/$m^3$ and kinematic viscosity of 2.5 x $10^{-6}$ $m^2$/s. The injection pressures selected here were 60, 70, and 80 MPa. Ambient conditions were atmospheric pressure and room temperature for all the test cases. The nozzle hole diameter was 0.3 mm and the depth was 0.8 mm, which makes the nozzle L/D ratio about 2.67 according to Chang Sik Lee and Sung Wook Park,(2002).

In this study, the fuel will be injected at various injection parameters inside spray chamber in order to study the affect of that parameter towards spray characteristics. Macroscopic behaviors of the fuel spray such as process of spray development, spray penetration and spray cone angle were taken in the conditions at various injection pressures. Accurate control of diesel spray parameters (timing, delivery, flow rate, pressure, spray geometry, so on.) is the most effective means to influence fuel and air mixing and to achieve both clean burning and high efficiency.

## 2.2 SPRAY CHARACTERISTIC

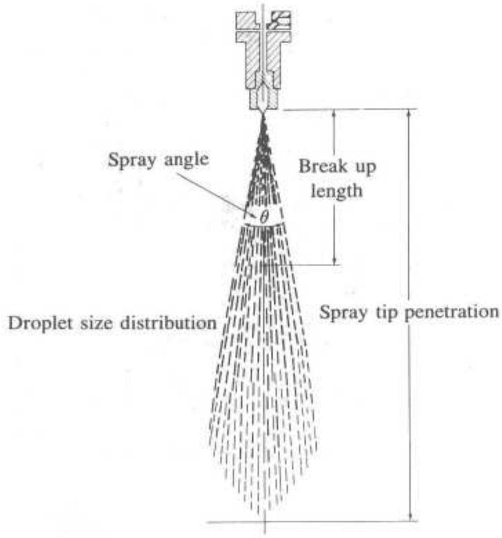



**Figure 2.1:** Physical parameter of a diesel sprays.

[Source: Hiroyasu & Arai, (1990)]

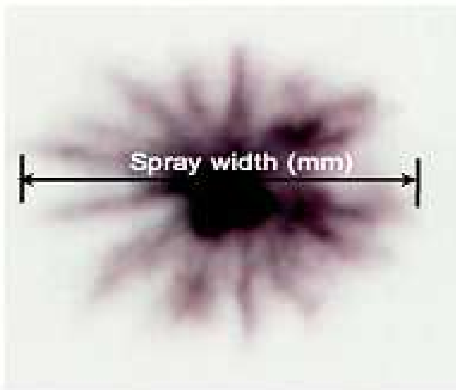



**Figure 2.2:** Spray width.

[Source: S. H. Park et al, (2003)]

## 2.3 FUEL INJECTION SYSTEM

The fuel injection system needs to provide different operating modes for the different loads. Fuel injection pressure is very high. These higher pressure values allow a higher penetration and reduce the mean droplet diameter determining a better atomized spray and a good penetration. Too high injection pressures will enhance atomization but at the same time produce an over penetrating sprays and wall wetting problems, especially when a sac volume is present. For the unthrottled part-load case, a late injection is needed in order to allow stratified charge combustion, with a well atomized compact spray to control the stratification. A well dispersed spray is desirable, with bigger cone angle and a conical shape. As mentioned before the higher injection pressure is necessary to reduce the Sauter mean diameter (SMD) of the liquid spray. To better characterize the spray size distribution the DV90 statistic may also be introduced, which is a quantitative measure of the largest droplets in the spray. It is the droplet diameter corresponding to the 90% volume point, so it gives a measure of the droplet size distribution spread. GDI injectors can either be single fluid or air-assisted (two phase) and may be classified by atomization mechanism (sheet, turbulence, pressure, cavitations), by actuation type, nozzle configuration (that can be either swirl, slit, multihole or cavity type), or by spray configuration (hollow cone, solid-cone, fan, multi-plume).Rossella Rotondi and Gino Bella,( 2005).

## 2.4 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 as shown in Figure 2.1. 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 the 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 tip penetration would be useful for the design of the engine combustion chamber. The dependence of penetration on injection parameters differed significantly from one investigation to another.

Increasing the ambient pressure was shown to decrease the spray tip penetration as well as the break-up length. Changing the ambient temperature had a minor effect on the spray tip penetration, due to a corresponding reduction in ambient density. However, a reduction in the spray angle was observed at higher temperatures, suggesting that the evaporation of the droplets were confined to the region on the periphery of the spray. Hiroyasu and Arai (1990).

The effects of ambient gas density and fuel vaporization on spray penetration were examined by Naber and Siebers (1996). The sprays were generated with an electronically controlled, common-rail, single-hole injector. The injection pressure was 137 MPa and the diameter of the nozzle hole was 0.257 mm. A high-speed camera was used to record the behavior of the sprays in a constant-volume combustion chamber. The ambient density was varied between 3 and 200 kg $m^{-3}$. The most noticeable trends are the decrease in penetration with an increase in ambient density and the decreasing rate of penetration with time. The effect of vaporization was to reduce the penetration. The reduction is as much as 20% at the low density conditions. The effects of vaporization became smaller for longer penetration distance and high gas density.

## 2.5 SPRAY CONE ANGLE

The spray cone angle from Figure 2.1 is defined as the angle formed by two straight lines drawn from the injector tip to the outer periphery of the spray Hiroyasu & Arai, (1990), Lefebvre (1989). According to Naber and Siebers (1996), the definition of the spray angle and the spray penetration are dependent on each other.

The spray cone angle is a qualitative indicator of how well the spray disperses. It is influenced by the nozzle dimensions, the liquid properties and the density of the medium in which the spray is introduced. The influence of the nozzle aspect ratio (L/D) on the spray angle was examined by several researchers. Shimizu (1984) showed that an

L/D of approximately 4 or 5 gave the maximum cone angle and shortest break-up length and also that an increase and decrease of L/D gave respectively a smaller cone angle and longer break-up length. The effects of kinematic viscosity and injection pressure on the spray cone angle were examined by Hiroyasu (1990). They found that spray angles were widened by a reduction in liquid viscosity and an increase in injection pressure, before stabilizing after reaching a maximum value. The increasing of cone angle will be effect the width of spray. From Figure 2.2 the width of spray is measured at horizontal spray by looks below the spray images or chamber at maximum width or distribution of spray. The spray width defines how well the spray disperses. S. H. Park et al, (2003).

## 2.6 DROPLET SIZE DISTRIBUTION

From Figure 2.1 the droplet size distribution (DSD) in sprays is the crucial parameter needed for the fundamental analysis of the transport of mass, momentum and heat in engineering systems. However, the DSD determines the quality of the spray and consequently influences to a significant extent the processes of fouling and undesired emissions in combustion Hiroyasu and Arai (1990). The diameter of the droplets obtained as a result of atomization is based on a series of parameters as follows:

(i) Rate of injection: the diameter of the droplet increases with the rate of injection as an increase in the volume of the injected liquid produces a greater drag of the working fluid, the aerodynamic interaction grows and the critical size of the droplets increases. Apart from this, increasing the numeric population of droplets intensifies de coalescence, resulting in a growth in the geometry of the droplets.

(ii) Density ratio ($\rho^*$): the relation of densities has two opposing effects on the size of the droplets, intensification of atomization and the possibility that there will be coalescence. On increasing the relationship of densities a greater aerodynamic interaction exists, which causes the droplets to slow down and an increase in the numerical population in their field.

(iii) Working fluid temperature (Tg): on increasing working fuel temperature there is an increase on the rate of evaporation, due to which at the beginning of this the droplets with small diameters tend to evaporate completely while those droplets with greater diameters maintain a stable geometry until they evaporate completely.

(iv) Evolution of the diameter of droplets during time: It's generally considered that the medium diameter of the droplets decreases at the point of the spray and increases at the tail, while in areas distant from the injector they maintain a rate of constant values. Generally speaking, the sizes of the droplets tend to diminish at the beginning of the injection and grow at the end. Fausto A. Sánchez-Cruz et al, (2005).

## 2.7 BREAK UP LENGTH

The break up length of the spray as shown in Figure 2.1 is a very important characteristic to define the behavior of the spray in the combustion chamber. This zone of the spray is also called continuous or stationary and it is understood as being from the nozzle exit to the point where the separations of the first droplets occur. To define this zone the use of diverse measurements methods and techniques is of vital importance. In the literature we find some of the most useful measurement methods and techniques in the analysis of the break up length, Hiroyasu & Arai, (1990), Arai et al., (1984)

## 2.8 EVOLUTION PROCESSES OF GLOBAL SPRAY

Figure 2.3 shows the comparison between experimental results and predicted results of developing processes of global sprays with different injection pressure. The computed two-dimensional slice images are also shown in this figure, and a reasonable agreement between photographs and calculations is obtained. It is observed that the droplets near the main spray are dispersed more rapidly with the increase of injection pressure. At the beginning stage, the main stream of the spray shaped like a spiral vortex and the spray shape of the end point is beginning to collapse due to the drag. When the spray moves downstream, the droplets which are positioned outside the main spray breakup into small ones prior to the inside droplets because the relative velocity between spray and the ambient gas in the outside region is large .

This phenomenon is explained more clearly in the spray flow field of Figure 2.4, in which the predicted gas flow field is shown with the droplet size distribution of spray. For this plot, the computed velocity vectors are reproduced with a coarser grid system using post processor. After the injection the gas velocity increased due to droplets with high velocity as shown in Figure 2.4(a), and the relative velocities of droplets injected at later stage are decreased. And droplets cause the gas flow to circulate through the spray as shown in Figure 2.4(b). Chang Sik Lee and Sung Wook Park,(2002).

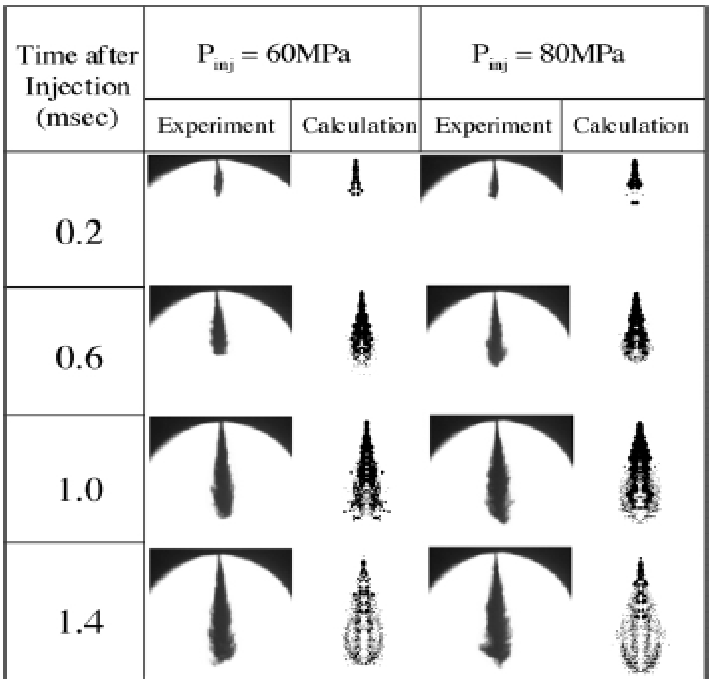
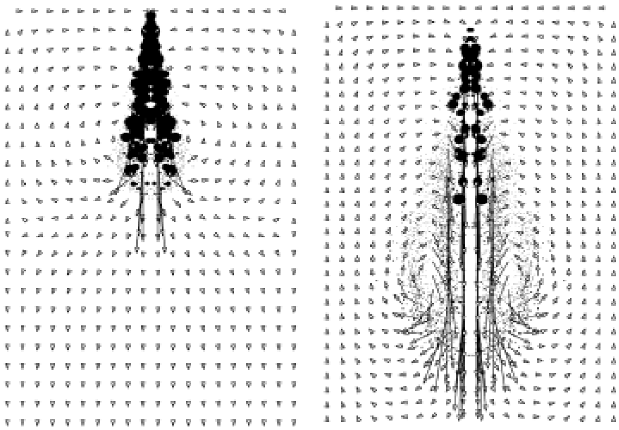



**Figure 2.3:** Spray development of high-pressure diesel injector.

[Source: Chang Sik Lee, and Sung Wook Park,(2002),]



(a)T=0.6 ms (b)T=1.4ms

**Figure 2.4:** Spray induced gas entrainment according to time after injection.

[Source: Chang Sik Lee, and Sung Wook Park,(2002)]

## 2.9 SPRAY SIMULATION

Sprays have always been a challenge for fluid modelers. Sprays that occur within direct injection engines are typically comprised of a very large number of droplets. Each droplet has unique properties and is subject to complex interactions that are a function of those properties. Due to limited computational resources, it is nearly impossible to take into account each individual droplet in a computational simulation. A variety of strategies has been formulated over the years to address this problem. While detail varies from to model, most of these strategies fall into two basic categories: Eulerian-type and lagrangian-type formulation. Sara dailey bauman, (2001).

The Eulerian-type formulation represents the spray using continuous fields on the same computational grid as is used for the ambient fluid. This formulation is often chosen for its simplicity and ease of implementation. Due to the semi-continuous nature of its formulation, spray properties are typically required to remain uniform, such as isothermal droplets and uniform droplet radii, or to follow other simplifying assumptions. Diverse droplet properties can be taken into account by maintaining multiple fields and transport equations. This type is almost appropriate when concerned about macroscopic behavior of the spray on scales much larger than the average droplet spacing or on scales on the order of the spray penetration length. However, the Eulerian approach suffers from numerical diffusion, particularly on coarse grids. Sara dailey bauman, (2001).

The lagrangian-type formulation is based on a fluid-particle model introduced by Dukowicz. The spray is represented by a collection of computational particles. Each particle in turn represents a parcel of spray droplets that are assumed to have identical properties such as position, velocity, density, radius, and temperature. Often referred to as the discrete droplet model or stochastic particle model, this formulation is more resistant to the numerical diffusion inherent in a semi-continuous field representation. If appropriately chosen probability distributions are used to define particle properties, an adequate statistical representation of realistic sprays may be obtained when a sufficiently large number of computational particles are used. In the limit of single droplet per particle and assuming appropriate initial conditions are known, this type of formulation approaches the ideal conditions for simulating the spray. Sara dailey bauman, (2001).

## 2.10 SOFTWARE SIMULATION

Nowadays computational fluid dynamics (CFD) plays a key role for the optimization of the combustion process in direct injection (DI) diesel engines. Despite their great uncertainties compared to the experimental studies, numerical simulations permit carrying out extensive parametric studies, isolating every single variable involved in the general process at any point in time and at any position in physical space. Modeling also allows one to artificially separate specific sub process in example spray atomization, evaporation, diffusive combustion, and emissions from the others that would interact in the real system or to investigate the effects of unnatural boundary conditions on such processes, in order to better understand the combustion process in engines. Basically, engine simulation models can be classified into three categories, depending on their complexity and increasing requirements with respect to the computational power: thermodynamics and phenomenological models, and the multidimensional models used in the so-called CFD codes. ANSYS CFD (FLUENT) is a commercially available 2D-3D computational fluid dynamics (CFD) code. J. M. Desantes et al, (2009).

The thermodynamic codes assume that the cylinder charge is uniform in both composition and temperature, at all times during the cycle. These models are computationally very efficient but cannot provide insight into local processes such as the spatial variation in mixture composition and temperature, essential to predict exhaust emissions. Phenomenological spray and combustion models are more complex than the thermodynamic models since they divide the combustion chamber into numerous different zones, characterized by different temperature and compositions. In the multidimensional CFD-codes the full set of differential equations for species, mass, energy, and momentum conservation are solved on a relatively fine numerical mesh with the inclusion of models to account for the effects of turbulence. As a result, these models are best suited to analyze the various subscale processes of mixture formation and combustion with great detail. J. M. Desantes et al, (2009).

## 2.11 COMPUTATIONAL FLUID DYNAMIC (CFD)

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulations. Computational Fluid Dynamics is basically a tool in the form of a software package which treats the fluid as being broken up into small volumes, and applies a suitable algorithm to solve the Navier Stokes equations of flow motion. Versteeg and Malasekera,( 1995) Computational Fluid Dynamics (CFD) itself emerged as a tool to cut cost and time by doing away with costly experiments to produce better, more efficient engineering designs. In practise, some experimental data or theoretical calculations will still be needed to verify at least the unit or benchmark case, and possibly if time and money permits the final design as well. Experimental data is also often needed for input in CFD simulations, for example in setting the boundary conditions of the model. Theoretical calculations based on simple models are always useful, providing back-of-the-envelope estimates for boundary conditions and sometimes results expected. This shows that an engineer will never do away with experiments and theoretical calculations and totally depend on simulations .Anderson (1995).

This study utilizes a commercial CFD package, FLUENT v6.1.22, which solves conservation equations for mass and momentum. Since the flow involved is gaseous and fuel at a high enough pressure to cause appreciable density change and shock waves, additional equations for energy conservation (using ideal gas) and also species conservation are solved. The species conservation equation is used to represent the different chemical components involved, namely methane, representing the CNG fuel and the surrounding air into which the fuel is injected. Additionally, transport equations are also solved since the flow is turbulent. All the governing equations used are listed in the following section. For all flows; FLUENT solves conservation equations for mass and momentum. For flows involving heat transfer or compressibility, an additional equation for energy conservation is solved. For flows involving species mixing or reactions, a species conservation equation is solved or, if the non-premixed combustion model is used, conservation equations for the mixture fraction and its variance are solved. Additional transport equations are also solved when the flow is turbulent.

# CHAPTER 3

# METHODOLOGY

## 3.1 INTRODUCTION

The methodology had been done right after the motivation and objectives of the project were identified. This methodology functioned as guidance in order to complete the project given. The completed structure of methodology had been illustrated and planned as guideline to achieve the objectives of the project. Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulations.

## 3.2 FLOW CHART OF METHODOLOGY

### 3.2.1 Simulation

Figure 3.1 shows the flow chart of the project. The project starts with literature review and research about title. These is consisting a review of the concept spray process, fuel properties, injection characteristics, software used, and spray modeling. These tasks have been done through research on the book, journal, technical repot and others sources.

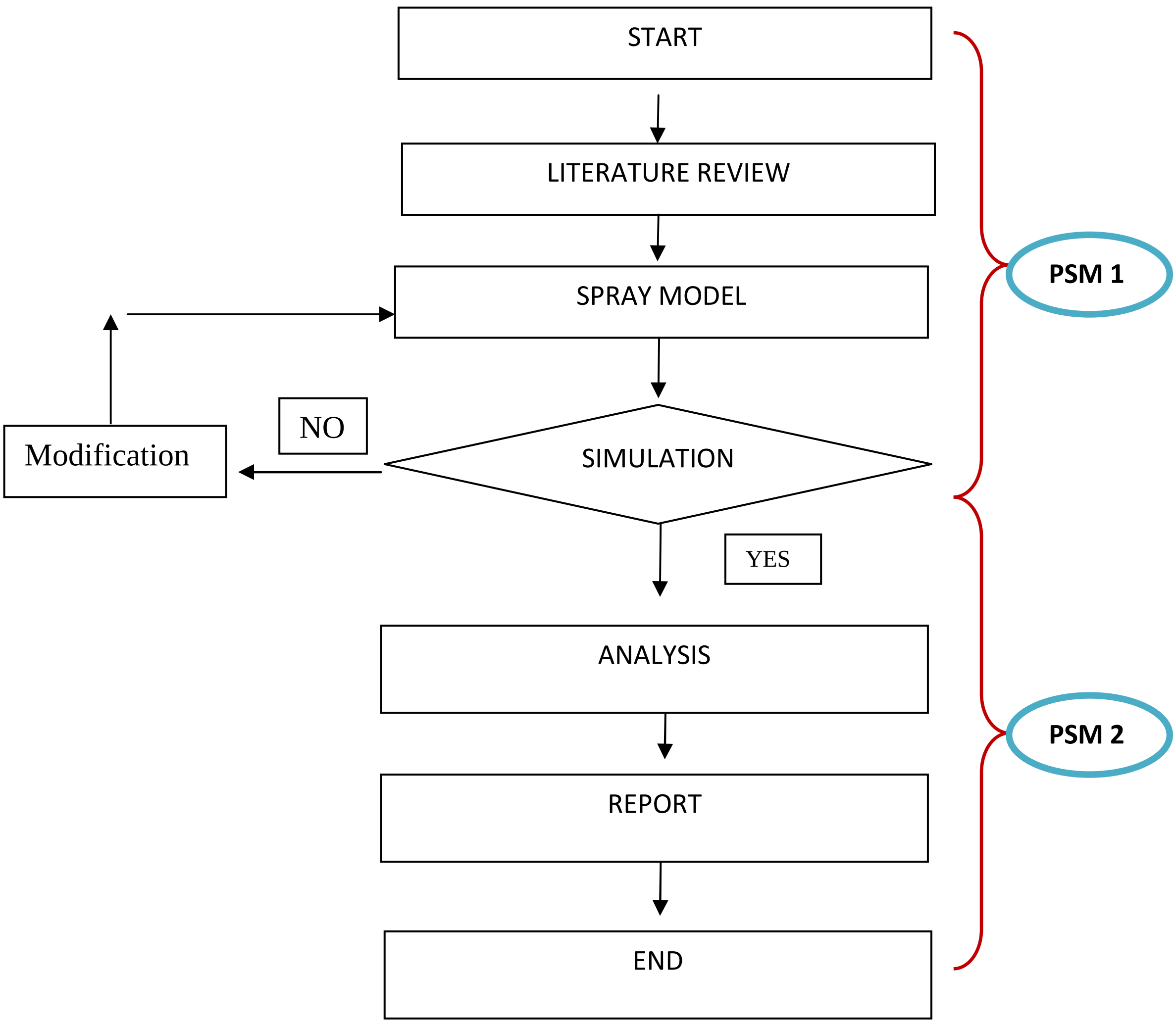



**Figure 3.1**: Flow chart.

After gathering all relevant information in Figure 3.1 above, the project undergoes to spray model. In this step, from the knowledge gather from the review is use to design the nozzle injector and chamber in 2D, and other to complete the system spray. After completing the spray model and meshing of the model, the simulation will be running using ANSYS-FLUENT software. All result will be record. If something errors or problems on this step, the spray model will modified until no error or problems.

The next step is analysis result. Result from simulation will be measure and analyze. The result of simulation includes the liquid atomization, and characteristic of different injection pressure and sac nozzle size like spray angle and spray penetration. The different injection pressure use is 70 MPa and 130 MPa, and the sac nozzle sizes are 0.2mm, 0.4mm and 0.6mm.

All the information will be made into the report, where all materials from beginning will be. The report writing process will be guided by the UMP final year project report writing. This process also included the preparation of slide for the final presentation. The project ended after the submission of the report and the slide presentation has been presented.

## 3.3 INJECTOR

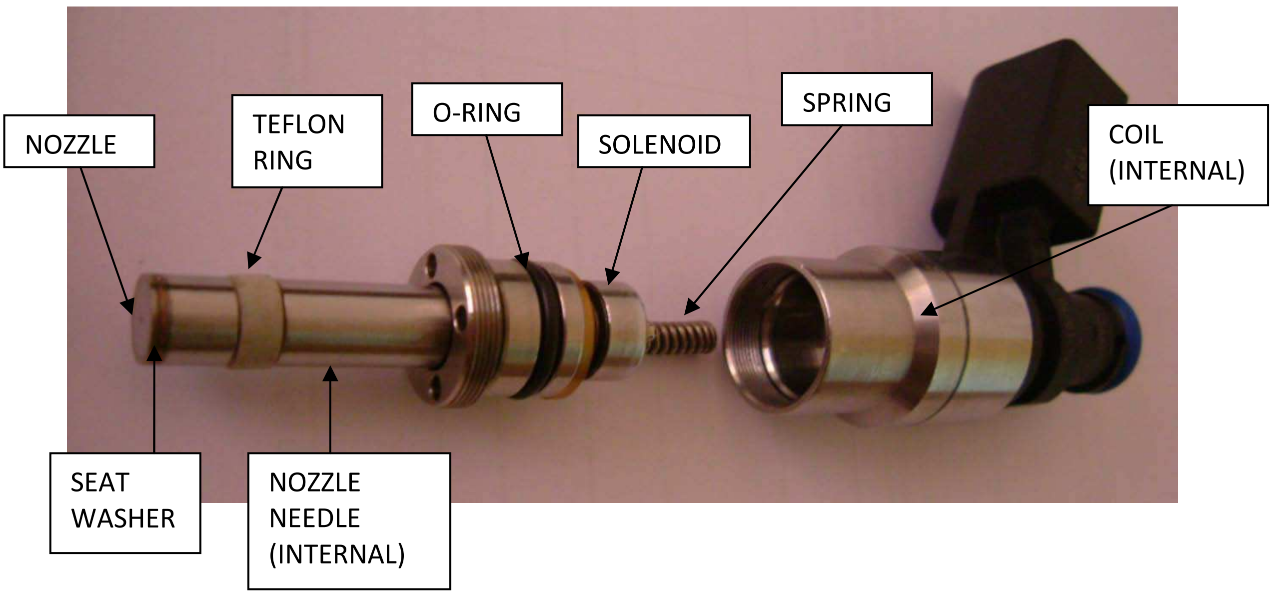



**Figure 3.2:** Injector.

Figure 3.2 show a disassembled of GDI Bosch Injector showing the spring and armature, part of the opening and closing mechanism. The nozzle needle spans from the nozzle seat to the spring.

The injector is controlled electronically. The electronic controller can supply varying pulse widths to the injector solenoid at different time intervals. The pulses are programmed to give single, multi or continuous shots. When an electric current flows through it, the coil generates a magnetic field which then causes a needle valve to lift 0.14 mm away from the nozzle seat, allowing fuel to flow into the combustion chamber. The fuel flow rate is controlled by the Pulse Widths. In the original gasoline engine control system for the HDEV Bosch injector, the pulses can be varied according to input from several sensors in the control system (DI-Motronic). Typically, there are three modes of injection, lean, stoichiometric and homogeneous for light, moderate and heavy load conditions.

One of the objectives of the analysis is to ascertain images flow of injection, spray cone angle and spray penetration from different high pressure of injection and different size of nozzle with CFD simulation.

## 3.4 STEPS OF SIMULATION

### 3.4.1 Geometry

The design of high pressure chamber and injection was design in 2D by ANSYS - fluent software. Figure 3.3 show the design of high pressure chamber injector. The high pressure chamber was designed with rectangular shape with 60mm x 100mm of dimension. This model was generated to geometry in ANSYS Workbench. The material of model was set to fluid. Figure 3.3 below show the model consist with 4 main surfaces, such as pressure inlet, pressure outlet, wall and axis. The straight yellow line was set as pressure outlet and the blue line was set as pressure inlet. The axis for half design was set as green line and the wall on red line. The 2D design in half is used to easy and quick process in running of simulation.

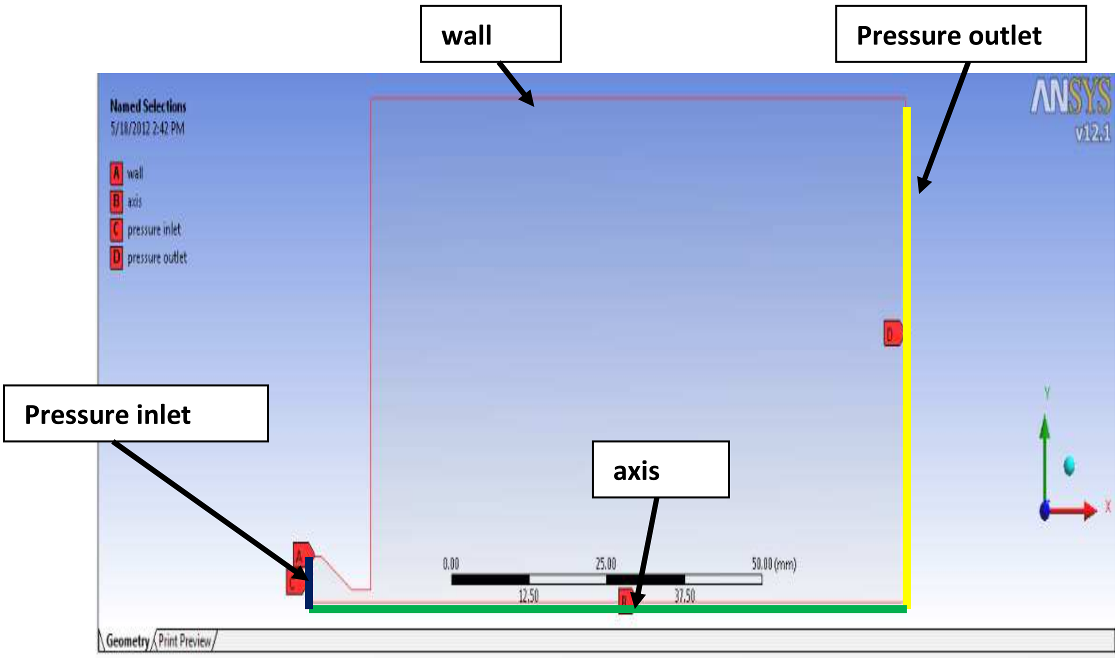



**Figure 3.3**: spray model and geometry.

### 3.4.2 Meshing

The Figure 3.4 shows the mesh of model. The model was meshing with automatic mesh in ANSYS CFD. The enlarged nozzle region is shown below the mesh showing the entire flow domain. Grid sensitivity computations by Li *et al.* (2004) indicated that at least 10 cells across the nozzle orifice were needed for grid independence. This was consistent with the grid recommendations of Abraham (1997).

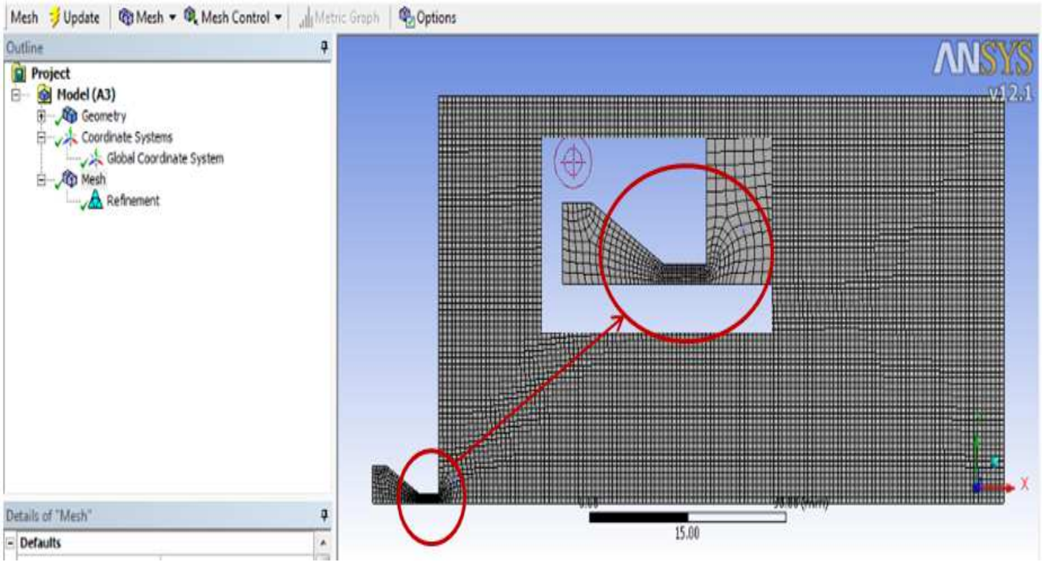



**Figure 3.4**: Meshing.

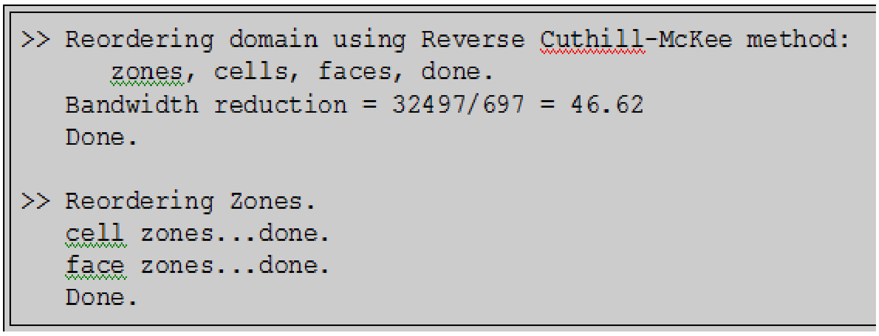
### 3.4.3 Setup

After mesh, Setup was used to launch the appropriate application in ANSYS Fluent. All parameters like load, boundary condition, type of material, and otherwise were insert in this setup.

#### (i) General

a) Check the mesh. ANSYS Fluent was performed various checks on the mesh and report the progress in the console. Ensure that the reported minimum volume was a positive number.

b) Reorder the mesh in Figure 3.5. To speed up the solution procedure, the mesh should be reordered, which substantially reduce the bandwidth. ANSYS FLUENT was reported the progress in the console.



```
>> Reordering domain using Reverse Cuthill-McKee method:
      zones, cells, faces, done.
   Bandwidth reduction = 32497/697 = 46.62
   Done.

>> Reordering Zones.
   cell zones...done.
   face zones...done.
   Done.
```

**Figure 3.5:** Reorder report.

### (ii) Models

a) Enable heat transfer by enabling the energy equation in Figure 3.6.

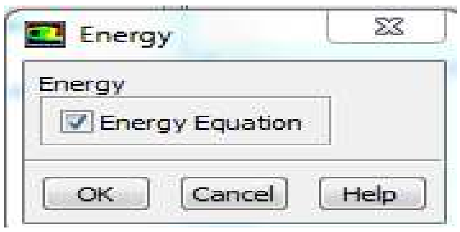



**Figure 3.6 :** Energy dialog box.

b) Enable the realizable $k$- $\epsilon$turbulence model in Figure 3.7. Select k-epsilon (2 eqn) in the Model list. Select Realizable in the k-epsilon Model list. The realizable $k$- $\epsilon$model gives a more accurate prediction of the spreading rate of both planar and round jets than the standard $k$- $\epsilon$model. Retain the default selection of Standard Wall Functions in the Near-Wall Treatment list. Click OK to close the Viscous Model dialog box.

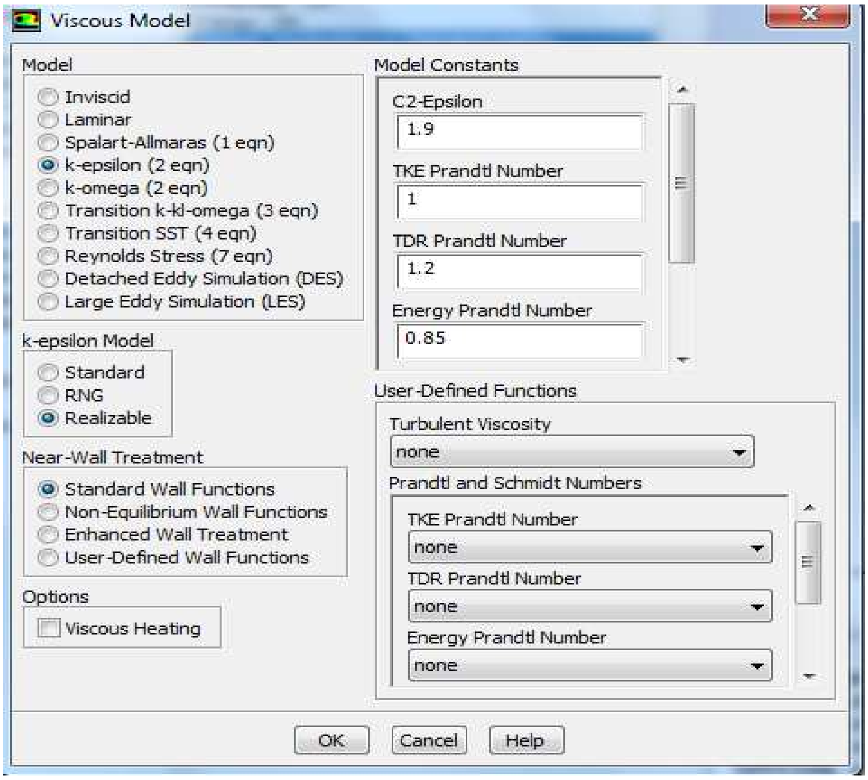



**Figure 3.7:** Viscous model dialog box.

c) Enable chemical species transport and reaction in Figure 3.8. Select Species Transport in the Model list. Select diesel-air from the Mixture Material drop-down list. The Mixture Material list contains the set of chemical mixtures that exist in the ANSYS FLUENT database. The chemical species in the system and their physical and thermodynamic properties were defined by the selection of the mixture material. Click OK to close the Species Model dialog box.

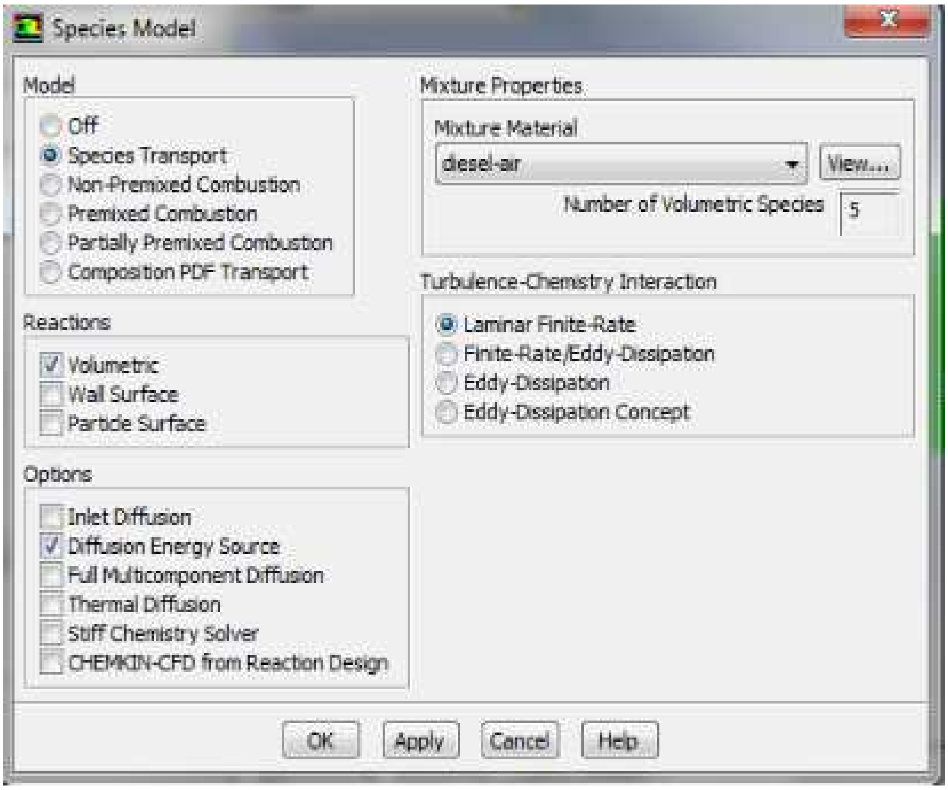



**Figure 3.8:** Species model dialog box.

d) Define the discrete phase modeling parameters in Figure 3.9. Enable Interaction with Continuous Phase in the Interaction group box. This was included the effects of the discrete phase trajectories on the continuous phase. Retain the value of 10 for Number of Continuous Phase Iterations per DPM Iteration. Click the Physical Models tab to enable the physical models. Enable Droplet Collision and Droplet Breakup in the Spray Model group box. Ensure that TAB was enabled in the Breakup Model list. Retain the default value of 0 for y0 and 2 for Breakup Parcels in the Breakup Constants group box.

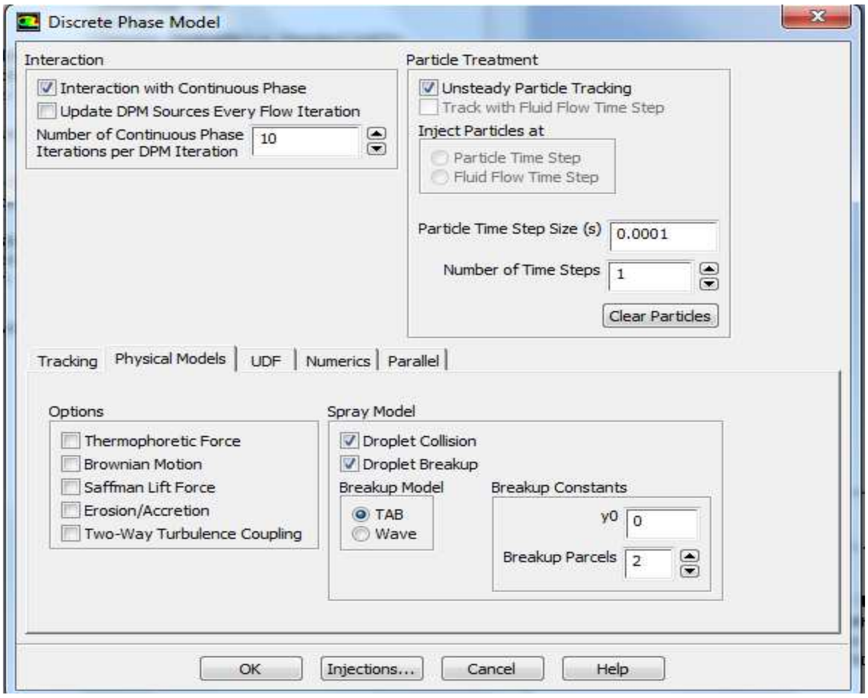



**Figure 3.9:** Discrete phase model dialog box.

e) Click the Tracking tab to specify the Tracking Parameters in Figure 3.10. Retain the default value of 5 for Step Length Factor. Select dynamic-drag from the Drag Law drop-down list in the Drag Parameters group box. The dynamic-drag law was available only when the Droplet Breakup model was used. Retain the Unsteady Particle Tracking option in the Particle Treatment group box. Enter 0.0001 for Particle Time Step Size. Retain the default value of 1 for Number of Time Steps. Click OK to close the Discrete Phase Model dialog box.

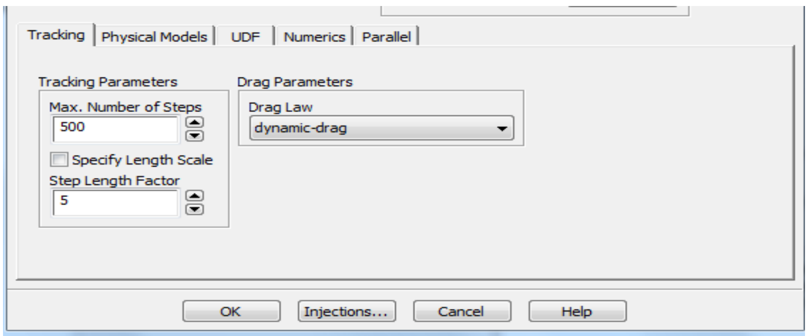



**Figure 3.10:** Tracking tab dialog box.

f) Create the spray injection in Figure 3.11. This step defined the characteristics of the atomizer. Click the Create button to open the Set Injection Properties dialog box. Select single from the Injection Type drop-down list. Select Droplet in the Particle Type group box. Select diesel-liquid from the Material drop-down list. Enter 0, 0, and 0 for X-Velocity, Y-Velocity, and Z-Velocity, respectively, in the Point Properties tab. Enter 263 K for Temperature. Enter 1.785e-3 kg/s for Flow Rate. Retain the default Start Time of 0 s and enter 0.002 s for the Stop Time. For this problem, the injection should begin at $t = 0$ and not stop until long after the time period of interest. A large value for the stop time (e.g., 100 s) ensures that the injection essentially never stops.

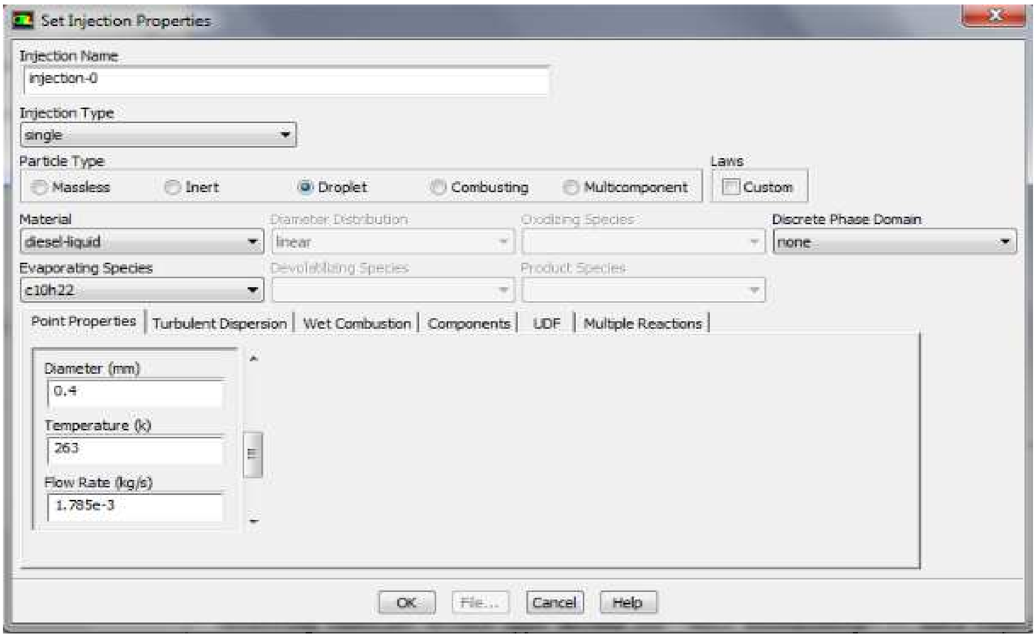



**Figure 3.11:** Set injection properties dialog box.

g) Define the turbulent dispersion. Click the Turbulent Dispersion tab. Enable Discrete Random Walk Model and Random Eddy Lifetime in the Stochastic Tracking group box. These models account for the turbulent dispersion of the droplets. Click OK to close the Set Injection Properties dialog box. Click OK in the Information dialog box to enable droplet coalescence. Close the Injection dialog box.

### (iii) Boundary Conditions

a) Set the boundary conditions for the inlet. Select Pressure outlet and inlet boundary condition from the Type drop-down list. Enter 40, 70 and 130 MPa in Gauge Pressure as shown in Figure 3.12. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list. Enter 2mm for Backflow Hydraulic Diameter and 10% for Backflow Turbulent Intensity. Click the Thermal tab and enter 293 K for Backflow Total Temperature. Click the Species tab and enter 1 for c16h29 in the Species Mass Fractions group box. Click OK to close the Pressure Outlet dialog box.

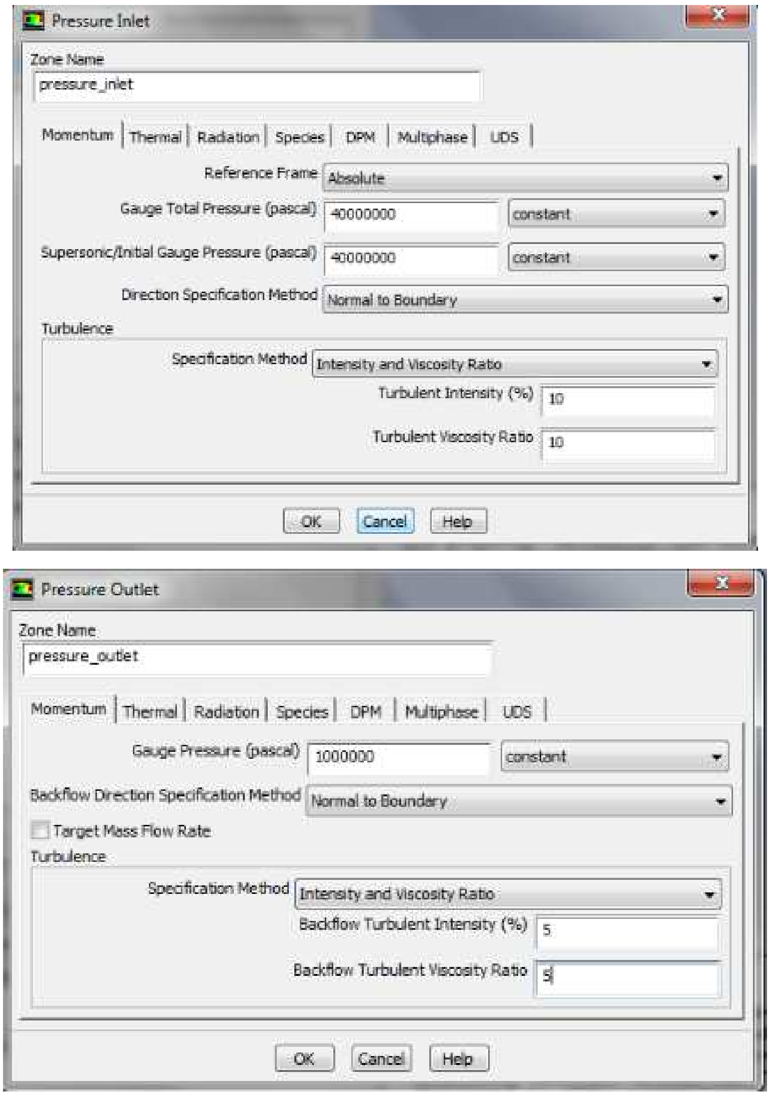



**Figure 3.12:** Pressure inlet and Outlet dialog box.

b) Set the boundary conditions for the outer wall in Figure 3.13. Select Specified Shear in the Shear Condition list. Retain the default values for the remaining parameters. Click OK to close the Wall dialog box.


**Figure 3.13:** Wall dialog box.

**(iv) Solution**

a) Apply second order upwind at Solution method Select Second Order Upwind from drop-down list for all parameter at Spatial Discretization box except Gradient.

b) Decrease the Under-Relaxation Factor for Discrete Phase Sources to 0.1 at solution control.

c) Initialize the flow field at Solution Initialization. Select all zones from the Compute from drop-down list. Initialize to initialize the variables. Click Patch button to set c16h29 at 0. At patch dialog box, select X Velocity from Variable list. At Zones to Patch list select Part_1. Enter 0 at Value space. In a similar manner, patch Y Velocity, Z Velocity and c16h29 at 0. Click OK to close the Patch dialog box.

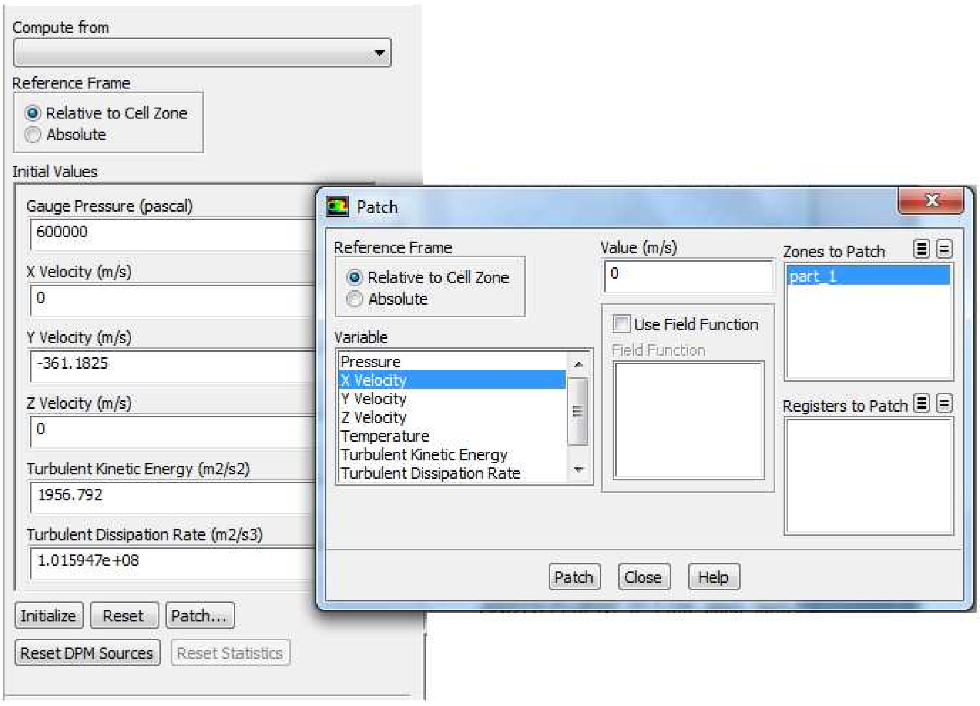



**Figure 3.14:** Patch dialog box.

d) Start the calculation by requesting 120 iterations. Enter 120 for Number of Iterations. Click Calculate. The solution converged in approximately 100 iterations.

# CHAPTER 4

# RESULTS AND DISCUSSIONS

## 4.1 INTRODUCTION

The aim of this chapter is to analyze the result of the project, which includes the diesel spray images or output from result of simulations. The result will also cover the part of fuel spray pattern, spray angle, and spray tip penetration.

## 4.2 RESULTS

After completing the analysis of simulation and get the result, the result taken have been analyzed thoroughly. The fuel pump supplied 40MPa, 70MPa and 130MPa of pressure to the injector has been use in this analysis. The visualization results of the developing spray at various stages by analysis technique using ANSYS-FLUENT are shown in Figure 4.2.

### 4.2.1 Spray characteristic

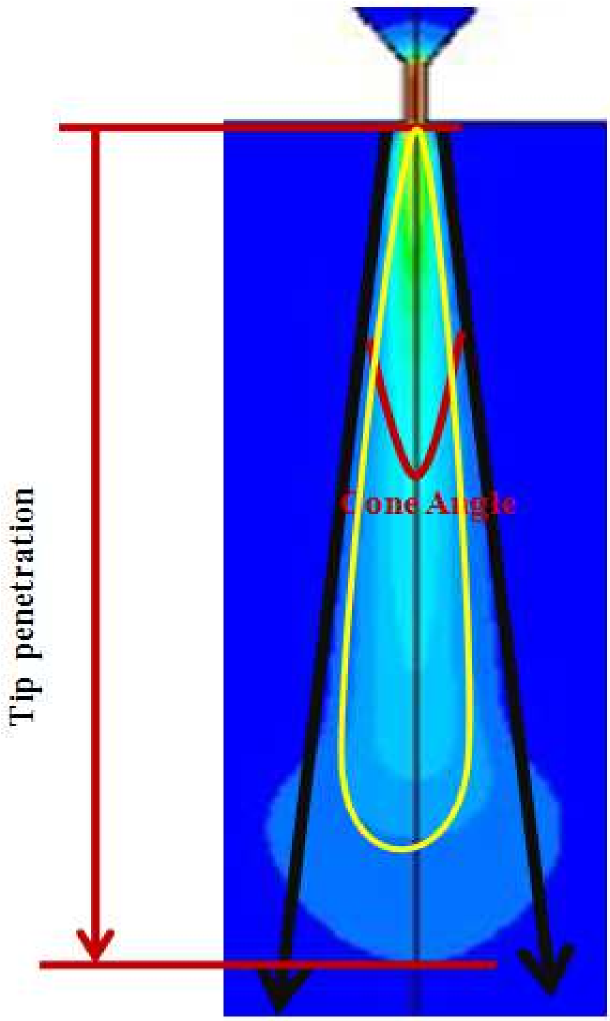



**Figure 4.1**: Characteristic of diesel spray.

The Figure 4.1 above show Spray cone angle (straight black line) is defined as the angle formed by two straight lines drawn from the injector tip at the outer periphery of the spray of Lefebvre (1989). The Spray penetration (straight red line) is defined as the maximum distance of spray from the nozzle tip at any given time. LACOSTE Julien (2006). The Main spray (curve yellow line) is defined as the spray with just fluid phase not vapor phase.

### 4.2.2 Spray development

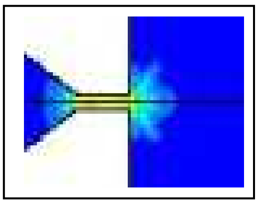
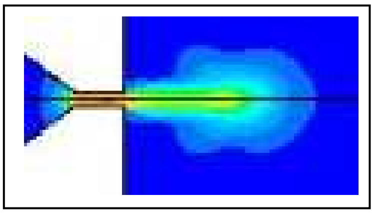
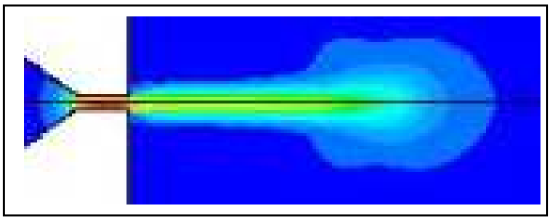
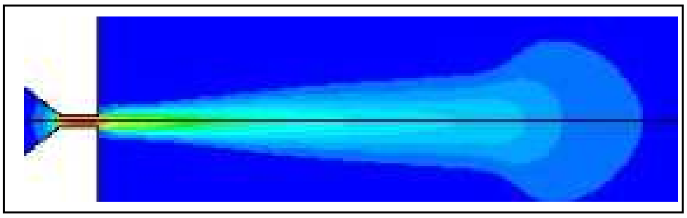
| Pressure injection at 130MPa | |
|---|---|
| **Time after injection (ms)** | **Development of spray** |
| 0.2 |  |
| 0.6 |  |
| 1.0 |  |
| 1.6 |  |

**Figure 4.2**: Result of diesel spray development.

From the Figure 4.2 above show at the beginning stage, the main stream of the spray shaped like a spiral vortex and beginning to collapse due to the drag at the end point. When the spray moves downstream, the droplets which are positioned outside the main spray breakup into small ones prior to the inside droplets because the relative velocity between spray and the ambient air in the outside region is large . After the injection the air velocity increased due to droplets with high velocity, and the relative velocities of droplets injected at last stage are decreased. And droplets cause the air flow to circulate through the spray.

### 4.2.3 Spray shape at various high injections

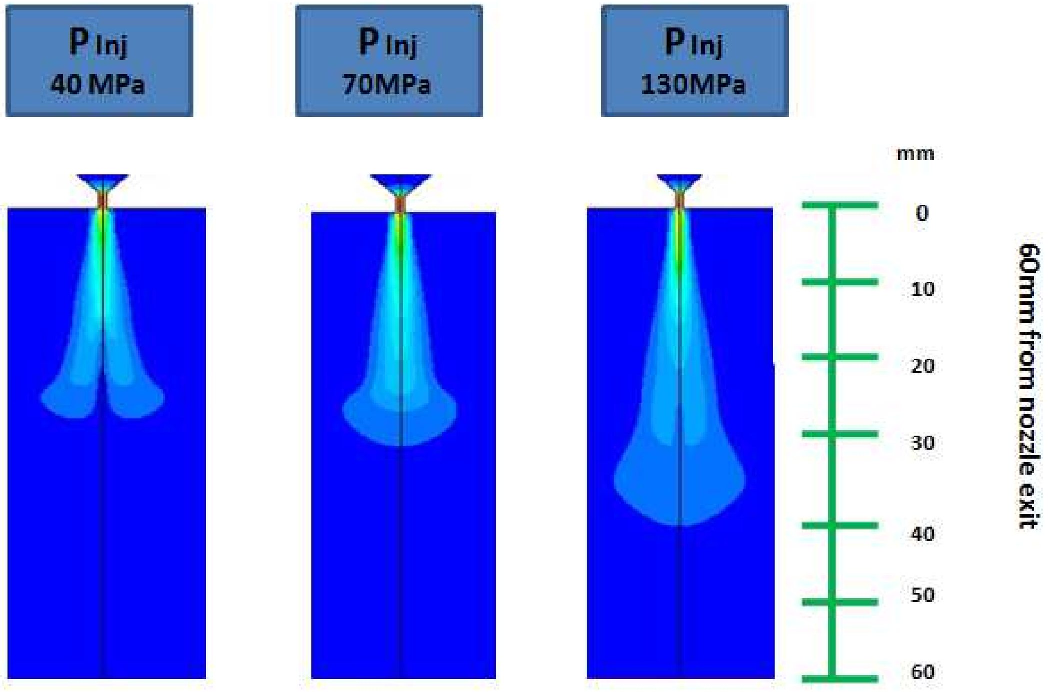



**Figure 4.3:** Spray shape at various high injections.

From this simulation, it can be seen that from Figure 4.3, it was quite difficult to gain the spray cone angle for each .The cone angle decrease, but with very small value when injection pressure increase because initial jet velocity and the mass flow rate of liquid increase . As expected, the spray tip penetration increased with the increase of injection pressure. The effects of vaporization became smaller for penetration distance. This means that large air assistance due to high-injection pressure causes a well-atomized spray. From equation (4.1) by hiroyasu and arai (1980) below show, the spray tip penetration increased with the increase of injection pressure.

$$S = 0.39t_{asoi} \sqrt{(2\Delta P)/ \rho_f} \quad (4.1)$$

Where the S is spray tip penetration, t is time(ms), asoi is after start of injection, P is pressure (MPa), ρ is density ($kg/mm^3$),and f is fuel.

From Figure 4.3 show the spray result at 40MPa as reverse of ‘V’ at maximum distance of penetration because of 1MPa pressure in chamber push the spray shape. At 70MPa and 130MPa is not like that because of high pressure more than 40MPa has been supplied.

### 4.2.4 How to measure the cone angle and tip penetration from simulation result

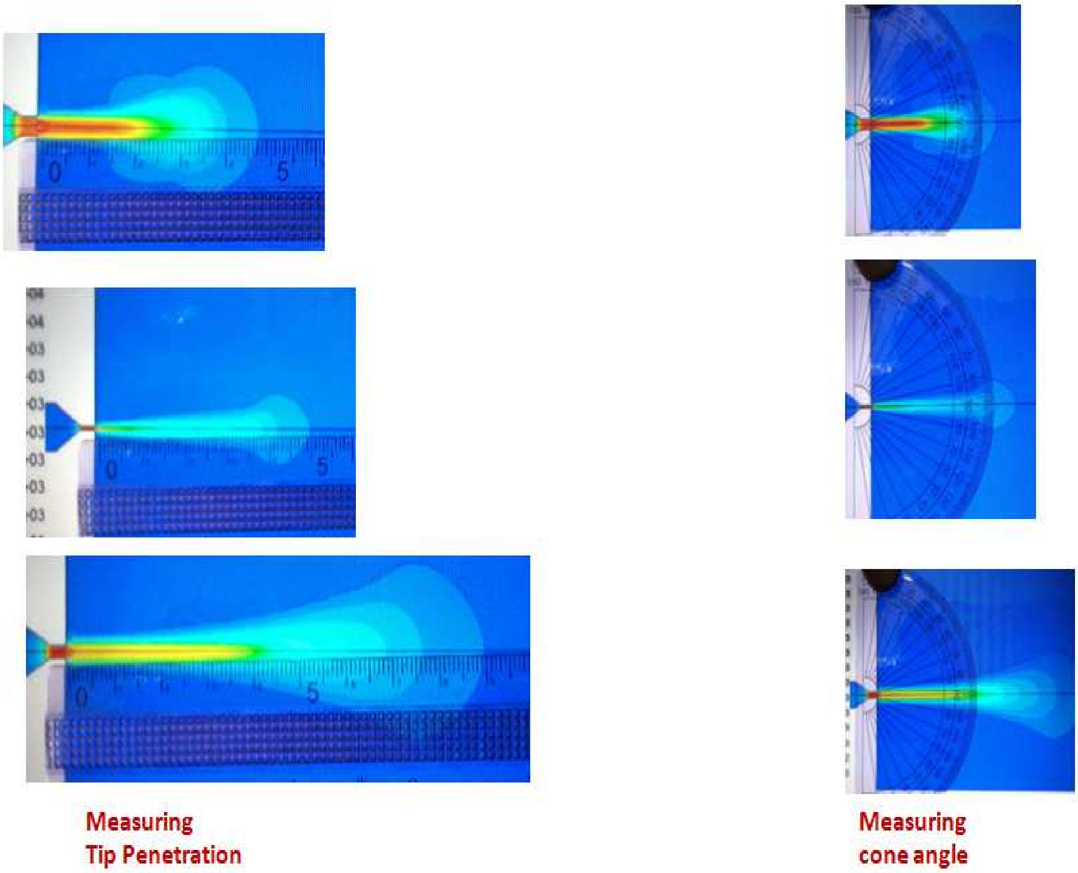



**Figure 4.4:** How to measure the cone angle and tip penetration.

From the Figure 4.4 above show how to measure the cone angle and tip penetration from the simulation result. The protractor will be used to measure the cone angle by reading the value when take the protractor on the images of spray. The ruler will be used to measure the tip penetration from the nozzle tip until the maximum of main spray.

### 4.2.5 Table of results

**Table 4.1:** Spray result of 0.2mm diameter SAC nozzle.

| DIAMETER SAC NOZZLE = 0.2 mm | | |
|---|---|---|
| **INJECTION PRESSURE (MPa)** | **CONE ANGLE (Degree)** | **TIP PENETRATION (mm)** |
| 40 | 15.6 | 31 |
| 70 | 15.4 | 35 |
| 140 | 15.1 | 43 |

**Table 4.2:** Spray result of 0.4mm diameter SAC nozzle.

| DIAMETER SAC NOZZLE = 0.4 mm | | |
|---|---|---|
| **INJECTION PRESSURE (MPa)** | **CONE ANGLE (Degree)** | **TIP PENETRATION (mm)** |
| 40 | 18.4 | 41 |
| 70 | 18.2 | 49 |
| 140 | 18.1 | 54 |

**Table 4.3:** Spray result of 0.6mm diameter SAC nozzle.

| DIAMETER SAC NOZZLE = 0.6 mm | | |
|---|---|---|
| **INJECTION PRESSURE (MPa)** | **CONE ANGLE (Degree)** | **TIP PENETRATION (mm)** |
| 40 | 22.4 | 54 |
| 70 | 22.3 | 55 |
| 140 | 22.3 | 57 |

From the table result 4.1, 4.2 and 4.3 above with three different sizes of diameter nozzle above show the high diameter nozzle produced high tip penetration and small decrease of cone angle due the increase of injection pressure.

### 4.2.6 Graph of results

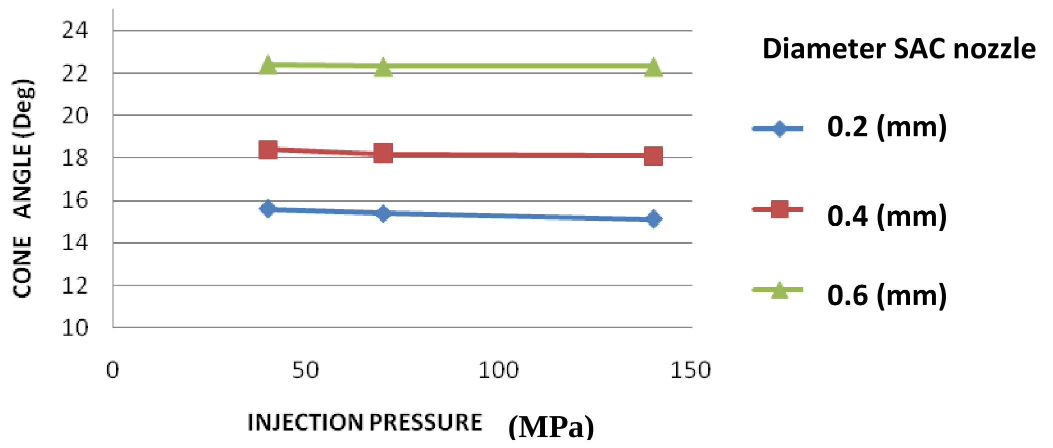



**Figure 4.5:** Cone angle versus injection pressure at different diameter of SAC nozzle.

From the Figure 4.5 above show graph the cone angle increase when the diameter nozzle increase, but the cone angle decrease when pressure increase. The maximum of cone angle will lead shortest break-up length and shortest time of vaporization to happen. The higher diameter of nozzle will produce higher cone angle and the higher cone angle will carry more fuel in spray.

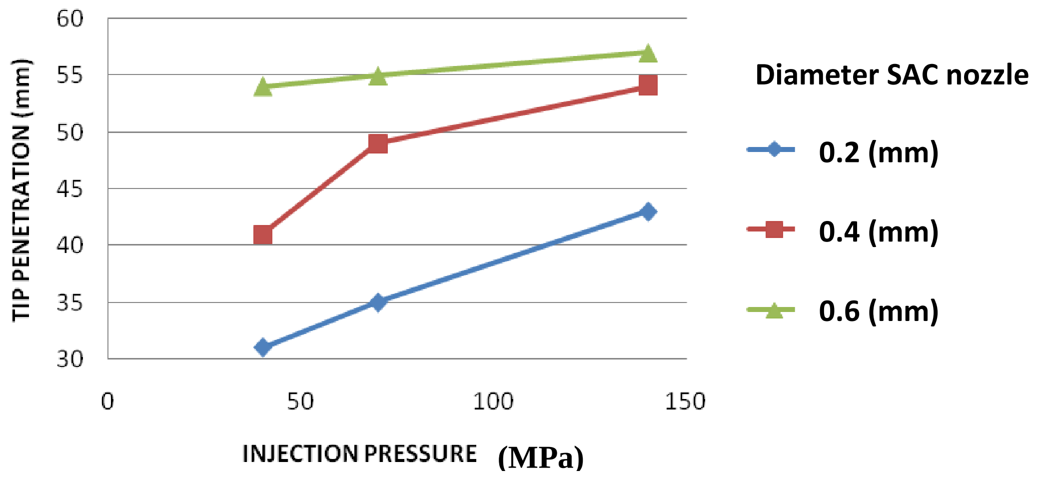



**Figure 4.6:** Tip penetration versus injection pressure at different diameter of SAC nozzle.

From the Figure 4.6 above show the graph the spray penetration increase, when the injection pressure increase. Although the 0.6 diameter is the higher tip penetration and cone angle, but the diameter is too big and will decrease the initial velocity at injection tip and produce longer time to break up happen, then will lead to formation soot of over penetration.

# CHAPTER 5

# CONCLUSIONS AND RECOMMENDATIONS

## 5.1 SUMMARY OF THE PROJECT

This chapter comprises the overview of the study as well as possible suggestion for the future project. Different pressure diesel injectors and different nozzle diameter were tested with using ANSYS fluent to know the spray pattern in high pressure.

## 5.2 CONCLUSIONS OF THE PROJECT

In conclusion, from the result we can conclude the high pressure will decrease the cone angle and increase the tip penetration, the high pressure will increase the vapor phase too .The higher nozzle diameter will increase the cone angle and tip penetration, but when the tip overpenetration happen will lead formation soot because of fuel impingement on cool wall chamber and the unburn fuel will produce. The spray cone angle is a qualitative indicator of how well spray disperses. The maximum cone angle will produce the shortest break up length. The shortest break up length will lead to high efficiency of combustion.

There are only a few results that could be obtained from this analysis due to the lack technological devices used and computer performance. If higher budget and high performance of hardware computer was provided for this project, it will be more successful and meaningful for future references.

## 5.3 RECOMMENDATIONS

This project of fuel spray visualization has been done successfully. However, broader parameters should be considered and analyzed, as more data could be gained with regards of this topic such as size and velocity of droplets, break-up region of spray and time of break-up. For recommendation, this project should proceed with further analysis with simulation and experiment. For instance, this CFD simulation should be in 3D with high performance of computer to get the result of droplets for particle track, with high performance of computer, the finer of meshing can be proceed to get the great result of spray simulation. For the nozzle influence of spray, in this analysis just used the different size of nozzle diameter, for recommendation, further project should be done with different length and diameter of nozzle which makes the nozzle L/D ratio because the nozzle ratio L/D can influence on the spray angle. According shimizu (1994) showed that an L/D of approximately 4 or 5 gave the maximum cone angle and shortest break-up length.

For the experiment should be done in Schlieren’s method in which this would certainly help to provide an effective technique for vapor phase visualization. In addition, it was suggested that the use of Laser Mie scattering and Back-lighting technique should be fully utilized in order to compare the result of spray characteristics which run in higher injection pressure so that it would provide better result in terms of vaporization at the spray tip region.

## 5.4 FUTURE WORK

It was believed that the experiment is necessary for preliminary research and further analysis of internal mixture formation, atomization mechanism and combustion using diesel fuel in diesel engines. This work can be extended with optimization of injector selection and timing strategies by integrating the effects of charge motion in an optical engine. The evaluation of valve train control strategies and the effect on mixing for homogeneous and stratified operation also need to be included to validate the robustness of the computational models.

**REFERENCES**

Anderson, J.D(1995). Computational fluid dynamics- The basics with applications. International Editions Mc-Graw Hill

Arai, M., Tabata, M., Hiroyasu, H., Shimizu, M., 1984. Disintegration process and spray characterization of fuel jet injected by a diesel nozzle, SAE Technical Paper 840275.

Alfred L.wick, 29 Feb (1988) Society of Automotive Engineers international congress and exposition, Detroit, MI, USA,

Hongsuk Kim, Nakwon Sung .November(2003) Combustion and Flame, Volume 135, Issue3, , Pages 261-270

Hiroyasu, H., Arai, M., (1990) "Structures of Fuel Sprays in Diesel Engines", SAE Paper 900475

Dennis LS. SAE Paper, ; 980809.( 1998)

Hattori, H., Naruyima, K., Tsue, M., Kadota, T., (2002) "Photographical Analysis of the Initial Breakup Process of Diesel Spray", THIESEL 2002, Conference on Thermo- and Fluid-Dynamic Process in Diesel Engines

I.V. Roisman , Lucio Araneo 1, C. Tropea ,Darmstadt University of Technology, Chair of Fluid Mechanics and Aerodynamics, Petersenstr. 30, 64287 Darmstadt, Germany Received 17 July( 2006); received in revised form 22 January( 2007)

I. Yamaguchi (2003)- Japan Automobile Research Institute, Inc.

Ishikawa N, Niimura K. SAE Paper, (1996); 960770.

Jeong-Kuk Yeom in Papers 19902002 (2002) .The Structure Analysis of Evaporative Diesel Spray.

J.M. Desantes, X. Margot, J. M. Pastor, M. Chavez, and A. Pinzello, (2009). CFD-Phenomenological Diesel Spray Analysis under Evaporative Conditions. Energy & Fuels 2009, 23, 3919–3929.

Lacoste Julien,( 2006), School of Engineering, University of Brighton In collaboration with Ricardo Consulting Engineers .Title: Characteristics of Diesel Sprays at High Temperatures and Pressures

Maruyama Y, Chiba T, Komori M, Saito M, Arai M. Proceedings of ILASS-Asia, (2001); 2001. p. 241–6.

Mimani, T., Yamaguchi, I., Tsujimura, K., Suzuki, T., (1990) "Analysis of Fuel Spray Characteristics and Combustion Phenomena under High Pressure Fuel Injection", SAE Paper 900438.

Naber, J. D., Siebers, D. L., (1996) "Effects of Gas Density and Vaporization on Penetration and Dispersion of Diesel Sprays", SAE Paper 960034.

Rossella Rotondi, Gino Bella, (2005). Gasoline direct injection spray simulation. International Journal of Thermal Sciences 45 (2006) 168–179.

Su TF, Patterson MA, Reitz RD, Farrell PV. SAE Paper,( 1996); 960861.

Source: Chang Sik Leea,, Sung Wook Park,(2002) , Department of Mechanical Engineering, Hanyang University

Su Han Park, Hyung Jun Kim, Hyun Kyu Suh, Chang Sik Lee,(2003) International Journal of Heat and Fluid Flow

Sidu X, Mingfa Y, Junfeng X. An experimental investigation on the spray characteristics of diesel. SAE technical paper; (2001) (2001-01- 0142)

Su Han Park, Hyung Jun Kim, Chang Sik Lee (2010),Department of Mechanical Engineering, Hanyang University, Republic of Korea

Shimizu, M., Arai, M., Hiroyasu, H., (1984) "Measurements of Breakup Length in High Speed Jet", Bulletin of JSME, Vol. 27, No. 230, pp. 1709-1715.

Sara dailey bauman, 2001. A Spray Model for an Adaptive Mesh Refinement Code. University of Wisconsin-madison. MI 48106-1346 USA 800-521-0600.

Simon, Martínez-Martínez1, Fausto A.Sánchez-Cruz(2005).Liquid Sprays Characteristics in Diesel Engines. *Universidad Autónoma de Nuevo León México*

Versteeg, H. K. & Malalasekera, W. (1995). An introduction to computational fluid dynamics: the finite volume method. Harlow: Longman Scientific and Technical

Yeom J, Senda J, Fujimoto H. Proceedings of ILASS-Asia, 2001; 2001. p. 167–72.

Yeom(2001), Su (1996), Chang. (2002), Hohmann (2003), Toshimi (1991) .Experiments to study the characteristics of diesel spray for the fuel injector. .Department o f Automotive Engineering, Silla University,korea,

# APPENDIX A

| Project Activities | Week | | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 |
| 1.Meeting and briefing with supervisor | ■ | ■ | | | | | | | | | | | | | |
| 2.Do literature review | ■ | ■ | ■ | ■ | ■ | ■ | ■ | | | | | | | | |
| 3. Gathering all relevant information | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | | | | | | | |
| 4.Make injector sketch in 2D and 3D | | | | | ■ | ■ | ■ | ■ | | | | | | | |
| 5.Search suitable parameter | | | | | | | | ■ | ■ | | | | | | |
| 6. Design the nozzle injector and chamber in 2D | | | | | | | | | | ■ | ■ | | | | |
| 7.Prepare a report | | | | | | | | | | | | ■ | ■ | | |
| 8.Prepare for presentation | | | | | | | | | | | | | ■ | | |
| 9.Presentation | | | | | | | | | | | | | | ■ | |

**Figure A.1: Gantt chart for PSM 1**

| No | Project Activities | Week | | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 |
| 1. | Briefing with supervisor for FYP 2 | ■ | | | | | | | | | | | | | | |
| 2. | Discuss the suitable parameter for simulation | | ■ | ■ | ■ | ■ | ■ | | | | | | | | | |
| 3. | Find more journal about simulation ANSYS fluent | | | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | | | | | |
| 4. | Modify and rebuild the spray model | | | | ■ | ■ | ■ | ■ | ■ | ■ | | | | | | |
| 5. | Meshing and simulation the model | | | | | | | | ■ | ■ | ■ | ■ | | | | |
| 6. | Analyze the results | | | | | | | | | | ■ | ■ | ■ | | | |
| 7. | Collect the data and conclusion | | | | | | | | | | | ■ | ■ | ■ | | |
| 8. | Prepare a full report | | | | | | | | | | ■ | ■ | ■ | ■ | | |
| 9. | Prepare presentation slide | | | | | | | | | | | | | ■ | | |
| 10. | Presentation for FYP 2 | | | | | | | | | | | | | | ■ | |
| 11. | Submit the thesis | | | | | | | | | | | | | | ■ | |

**Figure A.2: Gantt chart for PSM 2**