ANALYSIS OF DIESEL SPRAY CHARACTERISTIC USING SINGLE HOLE SAC NOZZLE AND VCO NOZZLE

MOHD FAZWAN BIN ISHAK

UNIVERSITI MALAYSIA PAHANG

ANALYSIS OF DIESEL SPRAY CHARACTERISTIC USING SINGLE HOLE SAC NOZZLE AND VCO NOZZLE

MOHD FAZWAN BIN ISHAK

Report submitted in partial fulfillment of the requirements

for the award of Bachelor of Mechanical with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS

JUDUL: ANALYSIS OF DIESEL SPRAY CHARACTERISTIC USING SINGLE HOLE SAC AND VCO NOZZLE

SESI PENGAJIAN: 2011/2012

Saya, **MOHD FAZWAN BIN ISHAK (890504-03-6267)**

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:

(MOHD FAZWAN ISHAK)	(TANDATANGAN PENYELIA)
Alamat Tetap: **No 5, Kg. Jakar, Bukit Bunga,** **18500 Tanah Merah,** **Kelantan.**	**DR. ABDUL ADAM BIN ABDULLAH** (Nama Penyelia)
Tarikh: **21 JUN 2012**	Tarikh: **21 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).

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the thesis entitled "Analysis of diesel spray characteristic using single hole Sac and VCO nozzle" is written by Mohd Fazwan Ishak. I have examined the final copy of this thesis and in my opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature:

Name of Panel : MR. GAN LEONG MING

Position : LECTURER OF MECHNICAL ENGINEERING

Date : 21 JUNE 2012

SUPERVISOR’S DECLARATION

We hereby declare that we have checked this project report and in our opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical with Automotive Engineering.

Signature:

Name of Supervisor	: DR. ABDUL ADAM ABDULLAH
Position	: LECTURER OF MECHNICAL ENGINEERING
Date	: 21 JUNE 2012

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 degree and is not concurrently submitted for award of other degree.

Signature:

Name : MOHD FAZWAN ISHAK

Id Number : MH08071

Date : 21 JUNE 2012

ACKNOWLEDGEMENT

Firstly, I want to thank Allah S.W.T, for giving me the source of power, knowledge and strength to finish the project and dissertation for completing my Bachelor of Mechanical Engineering final year project.

I would like to thank my supervisors, Dr. Abdul Adam Abdullah for his guidance and supports throughout my candidature. His experience, wisdom, endurance and encouragement during his supervision period are immeasurable and greatly appreciated. He also was guided me through the dissertation process, never accepting less than my best efforts. His suggestions have helped me improve my technical writing and presentation skills.

Special thank to Mr. Mohd Fadzil Bin Rahim for their kindness in sharing his knowledge about ANSYS Fluent software and doing me a favor in getting additional information for the project. Also thanks to my fellow friends for their co-operation and help.

Lastly I am eternally indebted to my family, my parents for their unconditional support and advice to me to keep looking forward when I’m facing a lot of problems and boundaries in completing my PSM. Saying thank you just doesn’t cut it.

ABSTRACT

This study was focus to investigate the diesel spray characteristics and atomization performance. The influences of injector nozzle geometry and injection pressure conditions on a diesel fuel spray were examined and predicted by using ANSYS FLUENT software. A flow domain and constant volume combustion chamber was designed and temperature was set 540 K at pressure 1 MPa for simulations. Spray penetration length and cone angle of diesel spray were recorded. To investigate the influence of nozzle geometry, Sac and VCO (Valve Covers Orifice) with difference diameter orifice and injection pressure were designed and used in both spray evolution. Comparisons were made between different nozzle geometries and different injection pressures. Differences were observed between VCO and Sac nozzles, with the Sac nozzles showing a higher rate of penetration under the same conditions.

ABSTRAK

Kajian ini adalah fokus untuk mengetahui sifat-sifat semburan diesel dan prestasi pengatoman. Pengaruh rupa bentuk muncung dan keadaan tekanan penyuntikan dalam bahan bakar diesel diuji dan diramal menggunakan perisian ANSYS FLUENT 12.1. Kawasan aliran dan isipadu kebuk yang tetap telah direkabentuk dan ditetapkan suhu 540 K pada tekanan 1MPa untuk simulasi. Panjang penembusan semburan dan sudut kon semburan diesel telah dicatat. Untuk mengkaji pengaruh rupa bentuk muncung, Sac dan VCO (Valve Covers Orifice) dengan berbeza diameter lubang dan tekanan suntikan telah direkabentuk dan digunakan untuk kedua-dua kembangan semburan. Perbandingan telah dibuat antara reka bentuk muncung yang berbeza dan berbeza tekanan suntikan. Perbezaan telah diperhatikan antara muncung VCO dan Sac, dengan muncung Sac menunjukan kadar penembusan yang tinggi di bawah keadaan yang sama.

TABLE OF CONTENTS

		Page
PANEL DECLARATION		**ii**
SUPERVISOR'S DECLARATION		**iii**
STUDENT'S DECLARATION		**iv**
ACKNOWLEDGEMEN		**v**
ABSTRACT		**vi**
ABSTRAK		**vii**
TABLE OF CONTENTS		**viii**
LIST OF FIGURES		**x**
LIST OF SYMBLOS		**xii**
LIST OF ABBREVIATIONS		**xiii**
LIST OF APPENDICES		**xiv**
CHAPTER 1	**INTRODUCTION**	**1**
1.1	Project Background	1
1.2	Problem Statement	2
1.3	Objectives	2
1.4	Work scopes	2
1.5	Flow Chart	3
CHAPTER 2	**LITERATURE REVIEW**	**5**
2.1	Operation Of Diesel Engine	5
2.2	Atomization	7
2.3	Diesel Spray Characteristics	9
2.4	Spray Parameters	11
2.5	Formation of Liquid Spray	12
2.6	Computational Fluid Dynamics (CFD)	13
2.7	Examples of CFD	14

2.7.1 Using Star-CD 14
2.7.2 Using KIVA-3D 16
2.7.3 Using FLUENT 17
2.7.4 Using AVL 18

2.8 Advantages of CFD 20

CHAPTER 3 METHODOLOGY 21

3.1 ANSYS Workbench 21

3.2 Modeling The Flow Domain & Geometry 22

3.3 Meshing 24

3.4 Setup 25

3.4.1 Turbulence Models 26
3.4.2 Boundary Condition 27

3.5 Solution 27

3.6 Results 28

CHAPTER 4 RESULTS AND DISCUSSION 29

4.1 Influent of Injection Pressure 30

4.2 Influent of Nozzle Geometry 32

4.3 Comparison Cone Angle between Two Difference Geometry 34

CHAPTER 5 CONCLUSION 35

5.1 Conclusion 35

5.2 Recommendation and Future Work 36

REFERENCES 37

APPENDICES 39

LIST OF FIGURES

Figure No.		Page
1.1	Flow chart of methodology	4
2.1	Diesel combustion system	6
2.2	Spray regimes	8
2.3	Variable of injection process	9
2.4	Spray Parameters	12
2.5	Flow pattern of a spray formation near the nozzle tip region	13
2.6a	Computational domain	14
2.6b	Numerical result	15
2.7a	Computational domain	16
2.7b	Numerical result	17
2.8	Computational domain and Numerical result	18
2.9a	Computational domain	19
2.9b	Numerical result	19
3.1	ANSYS Workbench	22
3.2	Geometry domain for high pressure chamber (3D)	23
3.3	Flow Domain in Sac nozzle through the high pressure chamber	23
3.4	The mesh showing at enlarged orifice region	24
3.5	Named Selection of flow model	25
3.6	Boundary condition	27
3.7	Simulation result	28
4.1	Comparison of result for difference injection pressure	30

4.2	Comparison of experimental penetration rate for varying injection pressures (VCO, 0.2mm)	31
4.3	Comparison of result penetration rate for difference nozzle sizes and types	33
4.4	Comparison cone angle between Sac and VCO with same parameters	34

LIST OF SYMBOLS

ρ_l	Pressure of liquid injected fluid
ρ_g	Pressure of gas working fluid
μ^*	Relation of viscosities length
μ_l	Kinematic viscosity orifice entrance curvature
μ_g	Kinematic viscosity
v	Velocity
σ	True stress
d	Orifice diameter
l	Length
ΔP	Pressure increasing

LIST OF ABBREVIATIONS

VCO	Valve Covers Orifice
CFD	Computational fluid dynamics
SMD	Sauter Mean Diameter
IQT	Ignition Quality Tester
CAD	Computer-aided design
DNS	Direct Numerical Simulations
RNG	Renormalization-group
Re	Reynolds Number
We	Weber Number
Ta	Taylor Viscosity Parameter
Oh	Ohnesorge Number

LIST OF APPENDICES

Page

GANTT CHART FOR FYP 1 39

GANTT CHART FOR FYP 2 40

DIMENSION OF SAC NOZZLE 41

DIMENSION OF VCO NOZZLE 42

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

Diesel engines are widely used because of their high efficiency and cost effectiveness. Recently, passenger cars have adopted such engines, due to their inherent advantages over gasoline counterparts. Diesel engines have been the favorite power train for heavy-duty vehicles and non-road applications, and their use in light duty vehicles has been increasing. But, diesel engine designers are challenged by the need to fulfill with ever more stringent emissions standards while at the same time improving engine efficiency. Increasingly stringent emissions regulations as well as fuel economy demands means that diesel engines will have to incorporate new fueling technologies to achieve these goals. In order to increase engine efficiency and reduce emissions, great attention has been focused on improving fuel atomization.

The aim of this project was to study the effect of nozzle geometry against the fuel spray behavior. Comparisons were made between different nozzle geometries and different injection pressures. Differences were observed between VCO (Valve Covers Orifice) and Sac nozzles. The physical properties of diesel such as density, viscosity, and surface tension were stated and used in the numerical simulations. Injection process parameters such as injection pressure, nozzle needle lift, injection rate, and volume of injected fuel were controlled on the fuel injection systems in simulation setup.

1.2 PROBLEM STATEMENT

Diesel sprays have been studied for more than a century but were still under research. Through studies by different researchers, it was found spray evaporation and mixture formation during ignition delay period play an important role in ignition, combustion and emission production in diesel engine. Spray evaporation begins immediately after fuel emerged from nozzle. Therefore, the nozzle geometry plays an important role in fuel spray behavior inside spray chamber. In this study, the effect various nozzle geometry such as sac nozzle and VCO nozzle will be investigated.

1.3 OBJECTIVES

i. To study the effect of diesel spray characteristic using difference injector nozzle geometry in diesel engine.
ii. To investigate the influences of diesel spray characteristic using difference injection pressure in diesel engine.

1.4 WORK SCOPES

i. The project was conducted using ANSYS FLUENT 12.1 software on focusing measuring the spray penetration length and spray cone angle.
ii. Two difference type of nozzle which is Sac and Valve Covers Orifice (VCO) was used and the simulation result from these type nozzle was compared.
iii. All the information parameters used taken from previous experimental research.

1.5 FLOW CHART

The project starts with literature review and research about title such as a review of concept of atomization and process, design of chamber and nozzle geometry, injection characteristics, software used, and sprays modeling. These tasks have been done through research on the books, journals, technical reports and other sources.

After gathering all relevant information, the project undergoes to spray model. From the knowledge gathered, the review was used to design the injector, chamber and other to complete the system spray. After completing the spray model, the simulation was run. All results were recorded. When something error or problems arose in this step, the spray model was modified.

The next step was analysis result. Result from simulation was observed and made comparisons between two geometry nozzles. The comparisons were include the spray angle and spray penetration length between Sac and VCO nozzles using difference parameters.

The report was process after complete the analysis. All information like figures, tables and any references were collected to complete the report. Report had been guided by the UMP thesis report writing. This process also included the presentation slide marking for the final presentation of the project. The project ended after the submission of the report. All the procedures and steps for this project was constructing into the flow chart (refer Figure 1.1).

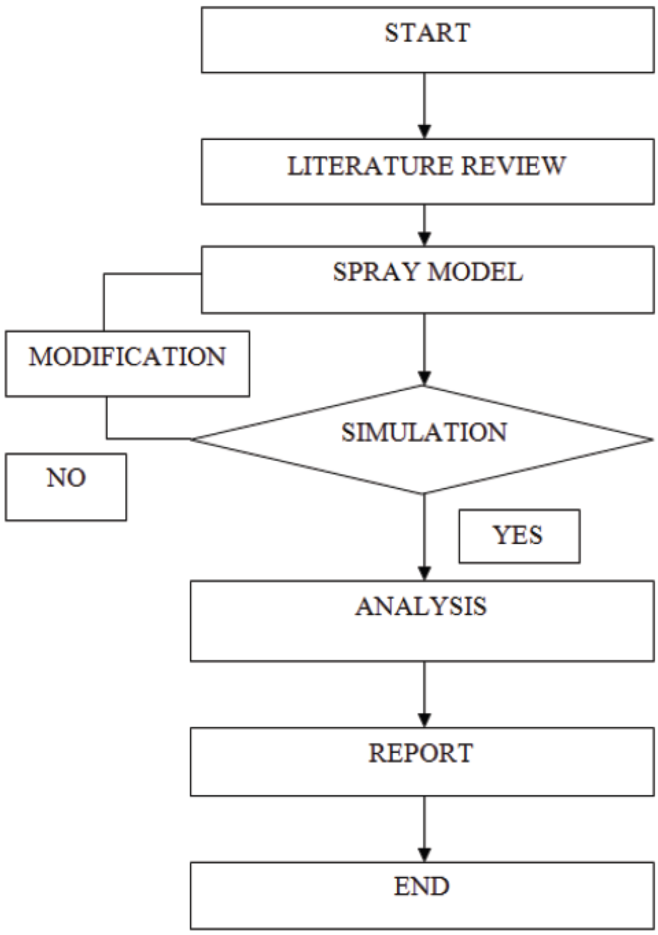


Figure 1.1: Flow chart of methodology.

CHAPTER 2

LITERATURE REVIEW

In this chapter all information related for this project was presented. This chapter will explain in detail about operation of diesel engine, atomization, spray parameters, formation of liquid spray, Computational Fluid Dynamics (CFD) and examples of CFD. Using CFD has a lot of benefit, save money and easy to conduct. At the end of this chapter will explain about advantages of using CFD in this project.

2.1 OPERATION OF DIESEL ENGINE

Figure 2.1 shows the diagram of diesel engine operation system. In a diesel engine, also known as a compression-ignited engine, air enters the cylinder through the intake system. This air was compressed to a high temperature and pressure and then finely atomized fuel is sprayed into the air at high velocity. When it contacts the high temperature air, the fuel vaporizes quickly, mixes with the air, and undergoes a series of spontaneous chemical reactions that result in a self-ignition or autoignition. No spark plug is required, although some diesel engines are equipped with electrically heated glow plugs to assist with starting the engine under cold conditions. The power of the engine is controlled by varying the volume of fuel injected into the cylinder, so there is no need for a throttle.

The timing of the combustion process must be precisely controlled to provide low emissions with optimum fuel efficiency. This timing is determined by the fuel injection timing plus the short time period between the start of fuel injection and the autoignition called the ignition delay. When the autoignition occur, the portion of the fuel that had been prepared for combustion burns very rapidly during a period known

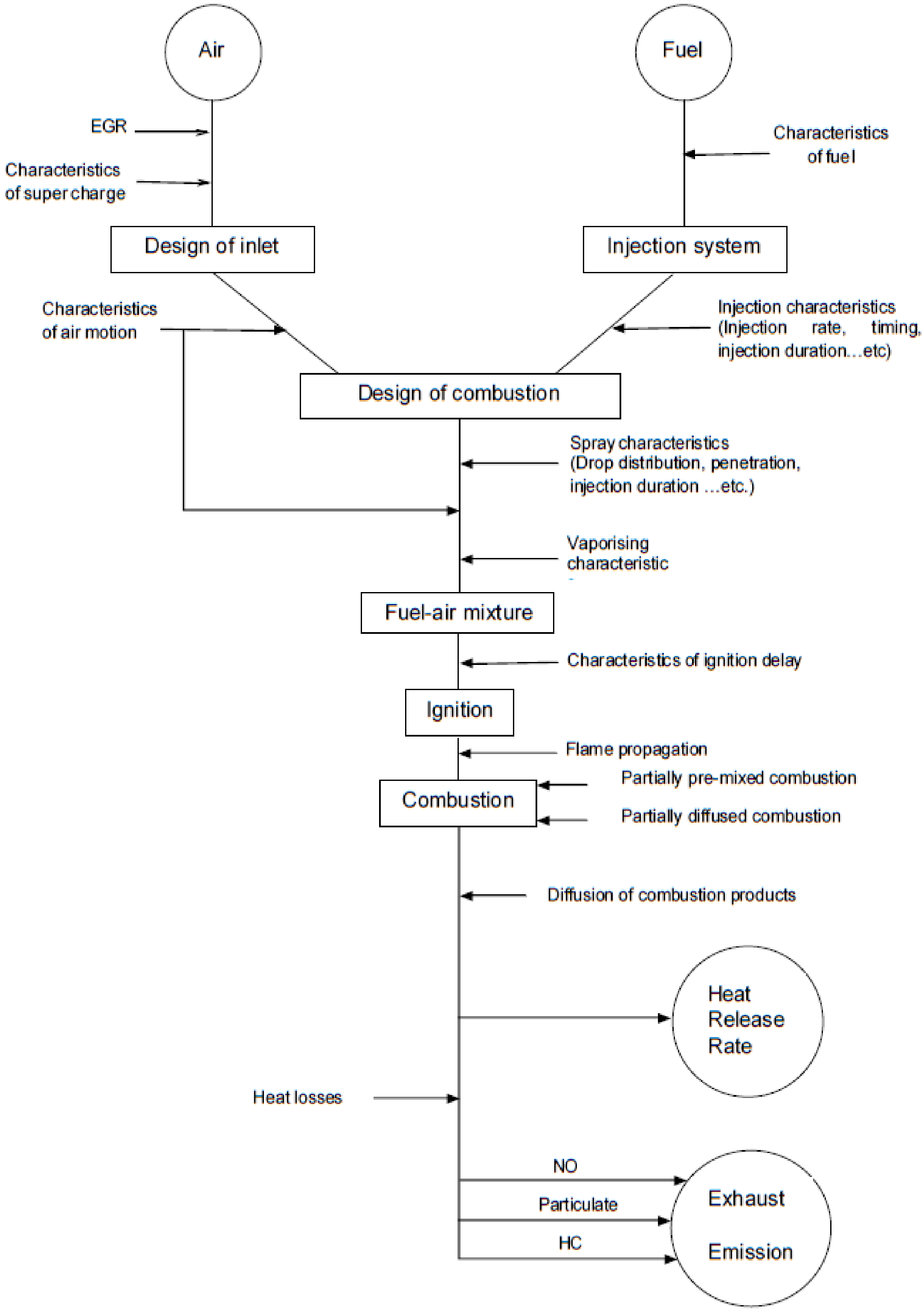


Figure 2.1: Diesel combustion system.

Source: Lacoste et al, (2006)

as premixed combustion. When the fuel that had been prepared during the ignition delay is exhausted, the remaining fuel burns at a rate determined by the mixing of the fuel and air. This period is known as mixing- controlled combustion.

The heterogeneous fuel-air mixture in the cylinder during the diesel combustion process contributes to the formation of soot particles. These particles are formed in high temperature regions of the combustion chamber where the air-fuel ratio is fuel-rich and consists mostly of carbon with small amounts of hydrogen and inorganic compounds.

2.2 ATOMIZATION

The atomization of liquids is a process of great practical importance in diesel engine. It also finds application in many branches of industry such as mechanical, chemical, aerospace, metallurgy, medicine, agriculture and many more. For different applications, the difference operating conditions have may use in order to manipulate the spray by changing the operating conditions to satisfy their own demands. From the Figure 2.2, the sprays are usually categorized into four spray regimes:

i. **Rayleigh regime**: Droplet diameter is larger than jet or spray diameter and liquid break up occurs at the downstream of the nozzle.
ii. **First wind induced regime**: Droplet diameter in the order of the spray diameter. Break up occurs at the downstream of the nozzle.
iii. **Second wind induced regime**: Droplet diameter smaller is than the spray diameter and break up starts some distance downstream of nozzle.
iv. **Atomization regime**: Droplet diameter much smaller than the spray diameter and break up starts close to the nozzle exit.

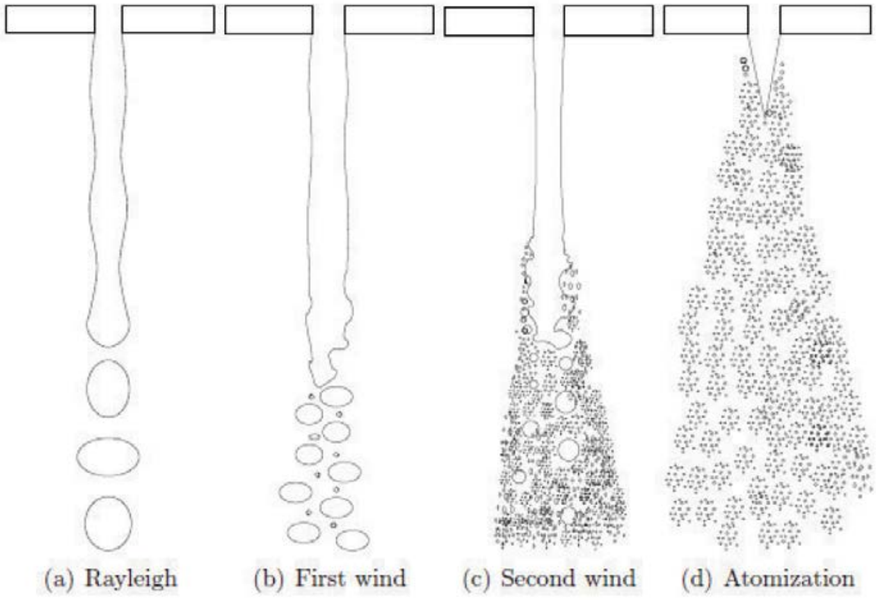


Figure 2.2: Spray regimes.

Source: Bjarke Skovgard Dam, (2007)

Atomization refers to the process of breaking up bulk liquids into droplets. In diesel engine, the nozzle play importance rule by producing spray which is collection of moving droplets that usually are the result of atomization. When high pressure and tiny orifice diameter of nozzle release the spray, the sheets or thin ligaments of liquid become unstable and they will break up into droplets, or atomize. The reason particles are round is due to the liquid’s surface tension. The temperature of liquid will increase due to pressure increase. So, its surface tension generally decreases. This becomes an important factor for fuel in diesel engine to spark and finally burning in high pressure combustion chamber.

There are variety factors affect the droplet size and how easily a stream of liquid atomizes after emerging from an orifice. These factors are usually covers three fluids properties:

i. **Surface tension:** It tends to stabilize a fluid, preventing its breakup into smaller droplets. Everything else being equal, fluids with higher surface tensions tend to have a larger average droplet size upon atomization.

ii. **Viscosity:** A fluid's viscosity has a similar effect on droplet size as surface tension. Viscosity causes the fluid to resist agitation, tending to prevent its breakup and leading to a larger average droplet size.

iii. **Density**: It will cause a fluid to resist acceleration. Similar to the properties of both surface tension and viscosity, higher density tends to result in a larger average droplet size.

2.3 DIESEL SPRAY CHARACTERISTICS

Depending on the mechanism to characterize, diesel spray can be analyzed in a macroscopic or microscopic point of view. With the purpose of understanding in detail this process, the various physical parameters involved during the transition of a pulsed diesel spray will be expressed in this chapter, however it is essential to know the systems that make possible for an injection process to take place. These are the injection nozzle, active fluid to inject (liquid), and the working fluid on which the liquid is injected, as shown in Figure 2.3.

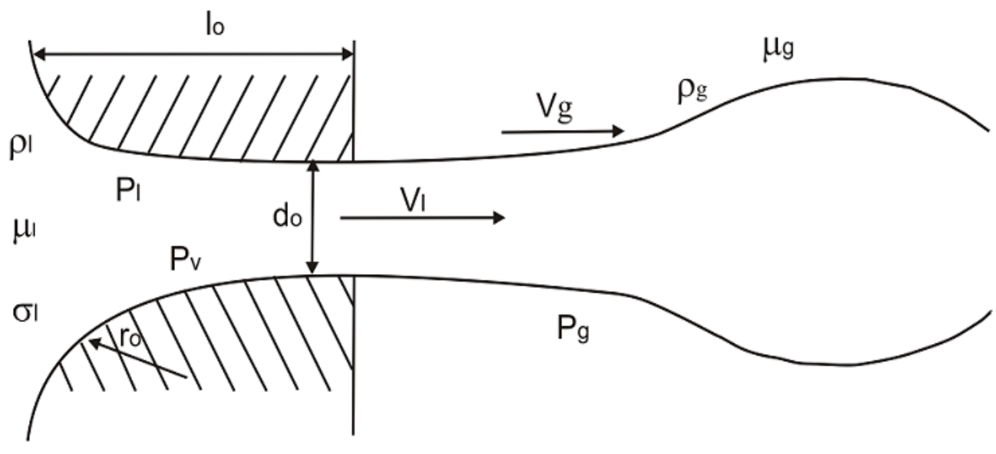


Figure 2.3: Variable of injection process.

Source: Vicente et al, (2010)

All these variables can be, can be fitted into a dimensionless form that allows to have much simpler relations and better defined. The dimensionless variables used in most cases are:

Relation of densities:

$$\rho^* = \frac{\rho_l}{\rho_g} \tag{2.1}$$

Relation of viscosities:

$$\mu^* = \frac{\mu_l}{\mu_g} \tag{2.2}$$

Reynolds Number, relation between inertial and viscous forces:

$$\mathrm{Re} = \frac{\rho d v}{\mu} \tag{2.3}$$

Weber Number, relation between superficial tension force and inertial force:

$$\mathrm{We} = \frac{\rho d v^2}{\mu} \tag{2.4}$$

Taylor Viscosity Parameter:

$$\mathrm{Ta} = \frac{\mathrm{Re}}{\mathrm{We}} = \frac{\sigma}{\mu v} \tag{2.5}$$

Ohnesorge Number:

$$\mathrm{Oh} = \frac{\sqrt{\mathrm{We}}}{\mathrm{Re}} = \frac{\mu}{\sqrt{\rho \sigma d}} \tag{2.6}$$

Length/diameter relation of the Nozzle = $\frac{l_0}{d_0}$

Nozzle radius entrance/diameter relation = $\frac{r_0}{d_0}$

Discharge coefficient of the nozzle:

$$C_d = \frac{vl}{\sqrt{\frac{2\Delta P}{P_l}}} \tag{2.7}$$

Cavitations Parameter:

$$K = \frac{2(P_l - P_v)}{\rho_l v^2} \tag{2.8}$$

Reynolds number, density and kinematic viscosity must be particularized for liquid or gas; furthermore these properties can be evaluated for intermediate conditions between both fluid film conditions. These parameters can be divided into two groups:

i. External flow parameters (relation of densities, Weber number, Taylor parameter), these parameters control the interaction between the liquid spray and the surrounding atmosphere.
ii. Internal flow parameters (Reynolds number, capitations parameter, length/diameter relation, nozzle radius entrance/diameter relation, discharge coefficient): these parameters control the interaction between the liquid and the nozzle.

2.4 SPRAY PARAMETERS

A number of parameters are defined in order to characterize a spray under certain conditions (refer Figure 2.4). The red color represents the spray projection area. Some spray commonly used parameters are:

i. **Penetration**: The penetration length is the distance from the nozzle to the end of spray.

ii. **Spray cone angle**: The spray angle is used to define the size of the spray. It is defined as the quasi steady angle, which is reached after the passing of the spray head.

iii. **Sauter Mean Diameter (SMD)**: The droplet size in the spray is usually characterized with its SMD. SMD is proportional to the surface to volume ratio and has the advantage that even if the droplets are not spheres their surface to volume fraction is equivalent to a sphere and therefore them heat up and evaporate in the same way.

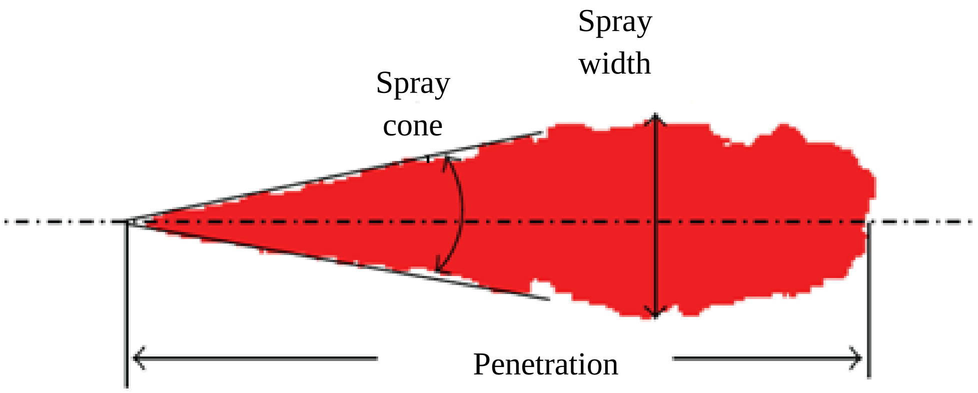


Figure 2.4: Spray Parameters.

Source**:** Metin et al, (2011)

2.5 FORMATION OF LIQUID SPRAY

Spray are known the collection of moving droplets or simply the introduction of liquid into a gaseous environment through a nozzle such that the liquid, through its interaction with the surrounding gas and by its own instability and breaks-up into droplets. The formation of a spray begins with the separating of droplets from the outer surface of a continuous liquid core extending from the orifice of the injection nozzle. From Figure 2.5, during spray formation the liquid core will separate into two difference condition which is primary break-up and secondary break-up.

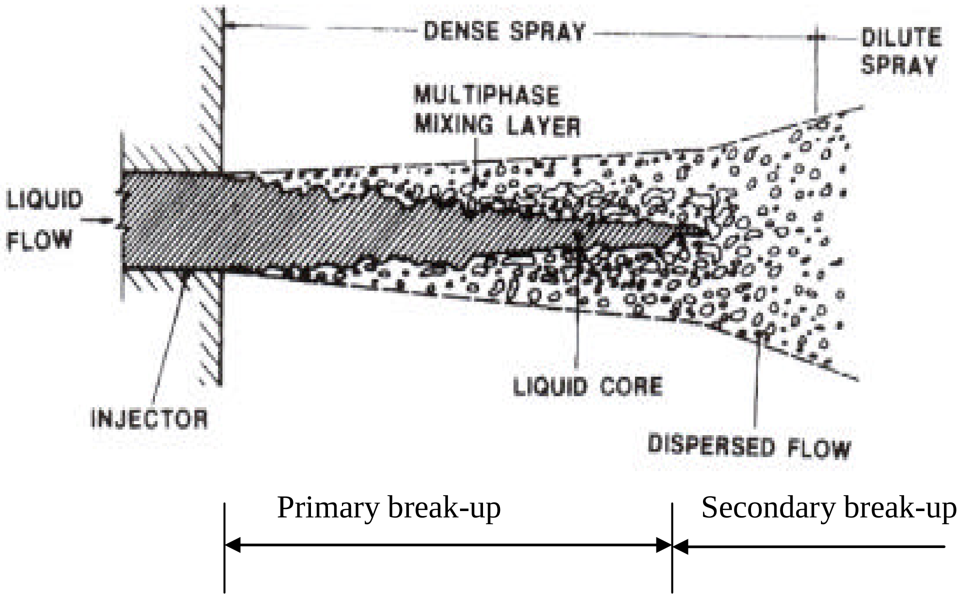


Figure 2.5: Flow pattern of a spray formation near the nozzle tip region.

Source: Faeth et al, (1995)

The separating of the liquid core into ligaments or large droplets is called primary break-up, which involves the action of forces internal to the liquid jet. Then, the liquid will have another process is called secondary break-up. In this process, the liquid ligaments and large droplets will further break-up into small droplets due to the interactions between the liquid and ambient gas or droplet collisions. The near nozzle region, where the volume fraction of the liquid is usually larger than that of the ambient gas is called the dense spray region. And the last region of spray formation is dilute spray region, which formation of downstream region where the volume fraction of the liquid is relatively low.

2.6 COMPUTATIONAL FLUID DYMNAMICS (CFD)

Computational fluid dynamics is part of fluid mechanics that uses numerical method and algorithm to solve and analyze problem that related to fluid flow. This section describes the definition, the examples of simulation and the advantage of using computational fluid dynamics.

CFD is a computer-based mathematical modeling tool that incorporates the solution of the fundamental equations of fluid flow, the Navier-Stokes equations, and other allied equations. CFD incorporates empirical models for modeling turbulence based on experimentation, as well as the solution of heat, mass and other transport and field equations. In order to done the calculations, computers are used to compute such task by using specific software that allows complex calculation for simulation of intended flow process. There are three phases to CFD:

i. Pre-processing or creation of a geometry usually done in a CAD tool.
ii. Mesh generation of a suitable computational domain to solve the flow equations.
iii. Solving with post processing, or visualization of a CFD code's predictions.

CFD is now a widely accepted and validated engineering tool for industrial applications. The results of CFD analysis are relevant in conceptual studies of new design, detailed product development, troubleshooting and redesigning.

2.7 EXAMPLES OF CFD

2.7.1 Using Star-CD CFD

The effect of injection orientation on fuel concentration in diesel engine was investigating using Star-CD software. A single cylinder four stroke DI diesel engine with fuel injector having multi-hole nozzle injector is considered for the analysis. 45° sector is taken for the analysis due to the symmetry of eight-hole injector in the model. The computational mesh when the piston is at Top Dead Center (TDC) is shown in Figure 2.6a. The number of cells in the computational domain at TDC is 10608. For the boundary condition, the initial swirl is taken as 2 m/s and the constant absolute pressure and temperatures as 9.87 bar, 583 K respectively. The turbulent model has the Intensity length scale as 0.1 and 0.001 respectively and it shows no traces of fuel and exhaust gases. The initial surface temperatures of combustion dome region and piston crown regions are taken as 450 K and the cylinder wall region has temperature of 400 K. The fuel was injected using 3 difference orientations which is 95°, 100° and 110°. The result was shown at Figure 2.6b.

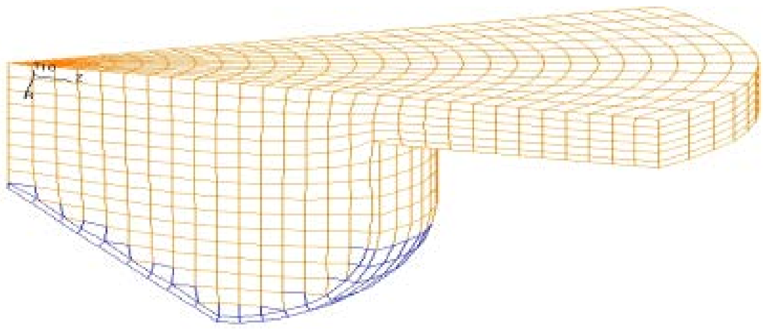


Figure 2.6a: Computational domain.

Source: Manoj et al, (2011)

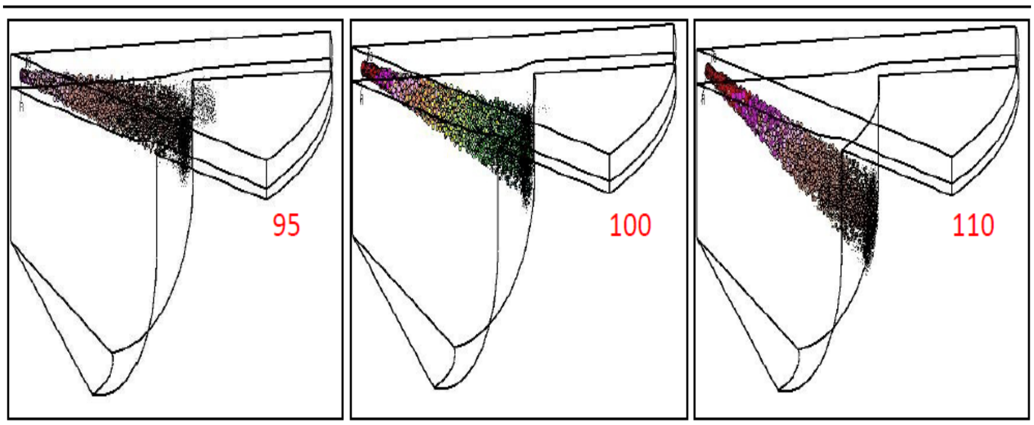


Figure 2.6b: Numerical result.

Source: Manoj et al, (2011)

2.7.2 Using KIVA-3D

KIVA-3V was utilized in developing a three-dimensional CFD model that accurately and efficiently reproduces ignition behavior and temporally resolves temperature and equivalence ratio regions inside the Ignition Quality Tester (IQT). Reacting fuel spray is simulated using a modified version of KIVA-3V software. The code couples Lagrangian particle tracking of liquid spray droplets with Eulerian simulation of three-dimensional fluid flow as governed by the Navier-Stokes equations.

These equations are solved on a structured grid domain of 59,868 cells generated using ANSYS ICEM CFD commercial software. Figure 2.7a shown the computational domain used in KIVA-3D simulation and the numerical result was shown at Figure 2.7b.

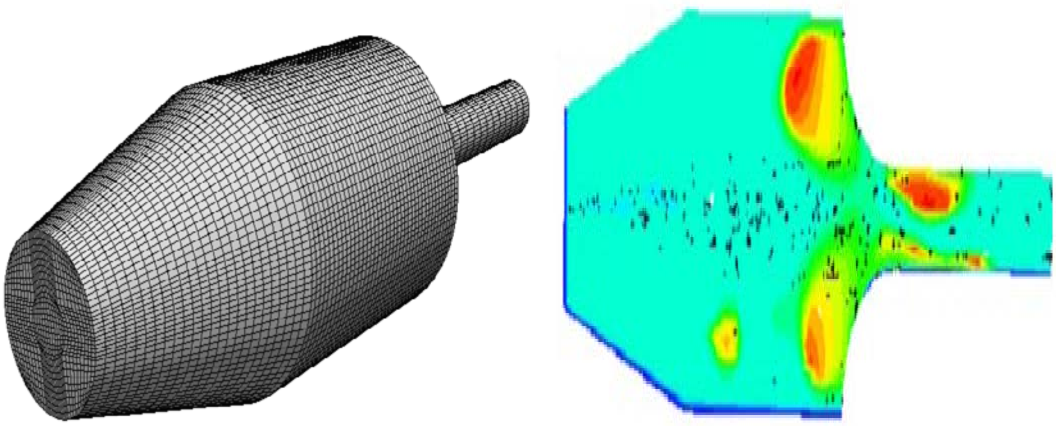


Figure 2.7a: Computational domain.

Source: Gregory et al, (2009)

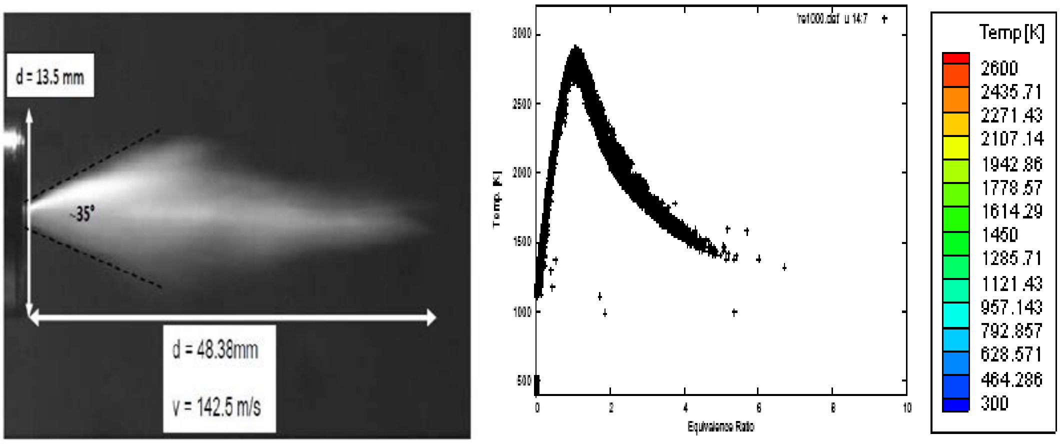


Figure 2.7b: Numerical result.

Source: Gregory et al, (2009)

2.7.3 Using FLUENT

Figure 2.8 show the computational domain and the numerical result used for the CFD simulation project. Three difference nozzles geometry used to study emission characteristic under diesel engine condition. The base nozzle has a cylindrical orifice with exit diameter of 169 µm, K = 0 and r/R = 0, the conical orifice is represented by Dout = 149 µm, K = 2 and r/R = 0, and the hydro ground orifice by Dout = 149 µm, K = 0, r/R = 0.014. The spray was injected in a constant-volume combustion vessel under diesel engine conditions. Spray and combustion simulations were performed using a Euleria–Lagrangian approach in CFD engine software.

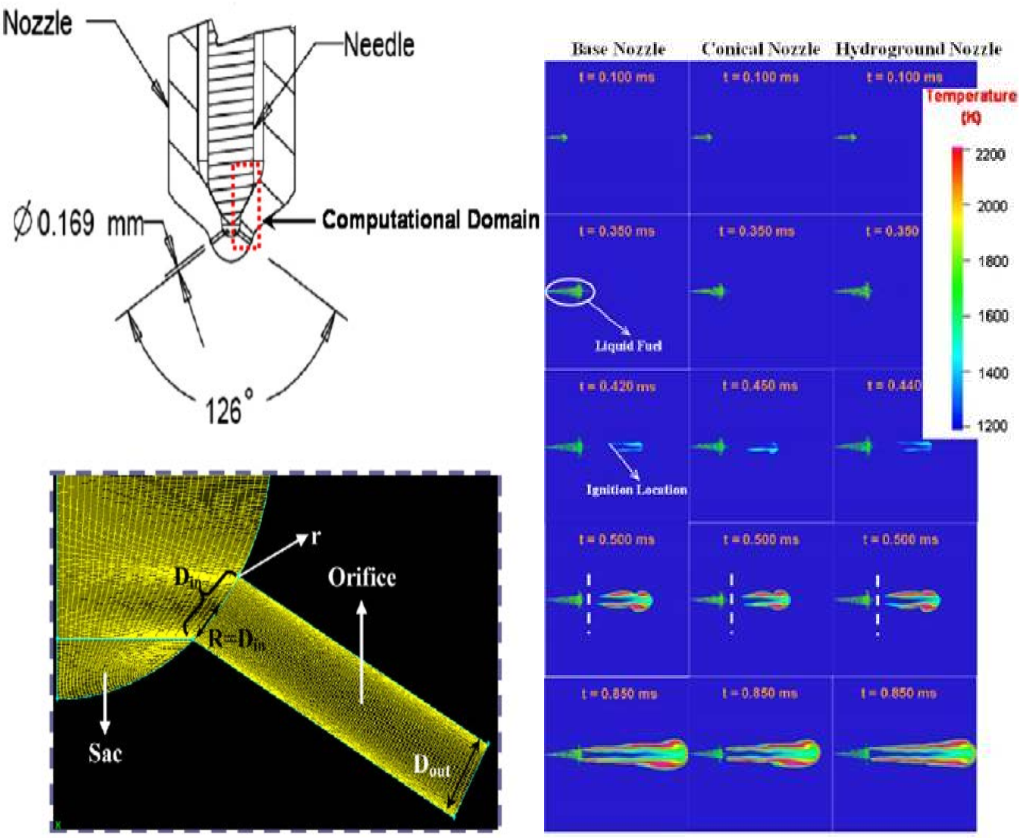


Figure 2.8: Computational domain and Numerical result.

Source: Sibendu Somet al, (2010)

2.7.4 Using AVL

AVL's was used to solve numerical simulations of the dual fuelling process by simultaneous direct injection of liquid and gaseous fuels into a combustion chamber. Injection diesel fuel and CH_4 were injected into a constant volume vessel through separate nozzles at time simultaneously. One major simplification was made to speed up the calculation process: instead of full chamber geometry only a 60 deg sector of the cylindrical chamber containing only two nozzles, one for gaseous fuel and one for liquid was investigated. The simulation results were compared with literature data. The computational domain was divided into 673,000 elements as shown at Figure 2.9a. The Comparison of the diesel fuel injection simulation results presented as spray accumulated view with experimental results obtained by the shadowgraph technique (refer Figure 2.9b).

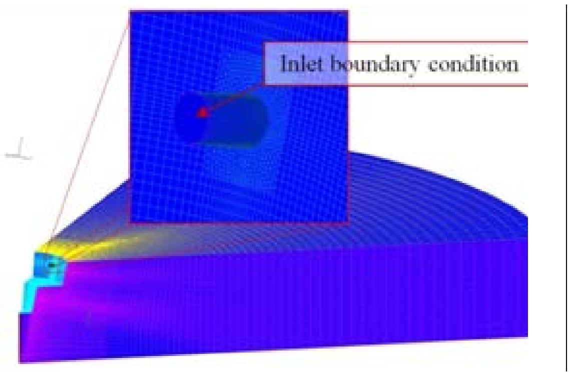


Figure 2.9a: Computational domain.

Source: Lukaszet al, (2012)

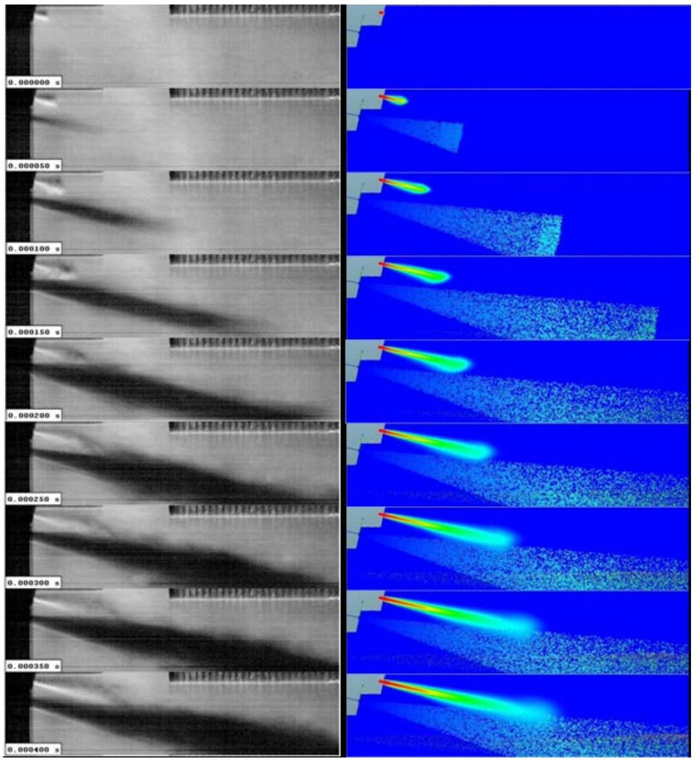


Figure 2.9b: Numerical result.

Source: Lukaszet al, (2012)

2.8 ADVANTAGES OF CFD

This software allows making changes to the analysis at any time during the setup, solution, or post processing phase. This will saves time and enables to refine the designs efficiently. Computer-aided design (CAD) or SolidWorks geometries are easily imported and adapted for CFD solutions.

Solver enhancements and numerical algorithms that decrease the time solution are included in every new release of CFD software. The mature, robust, and flexible parallel processing capability enables to solve bigger problems faster, and has been proven on the widest possible variety of platforms in the industry.

The CFD will post processing provides several levels of reporting. The result will satisfy the needs and interests of all user or researcher. Quantitative data analysis can be as rigorous as required. High resolution images and animations allow communicate the results with impact.

CHAPTER 3

METHODOLOGY

In this project, several methods were used to complete the project. The methods used were literature survey, data collecting, geometry measurement, conceptual design, computational simulation, and analysis. Each of the methods was explained in details and clearly with explanation. In this project, CFD –FLUENT is used which is a software that uses the science of predicting fluid flow, heat and mass transfer, chemical reactions and related phenomena by solving numerically the sets of governing mathematical equations in order to analyze the spray characteristic using two difference nozzle geometry.

3.1 ANSYS WORKBENCH

The simulation of spray chamber was test by using ANSYS FLUENT 12.1 in ANSYS workbench software. The Figure 3.1 shows the selection the Fluid Flow (FLUENT) analysis system in ANSYS Workbench.

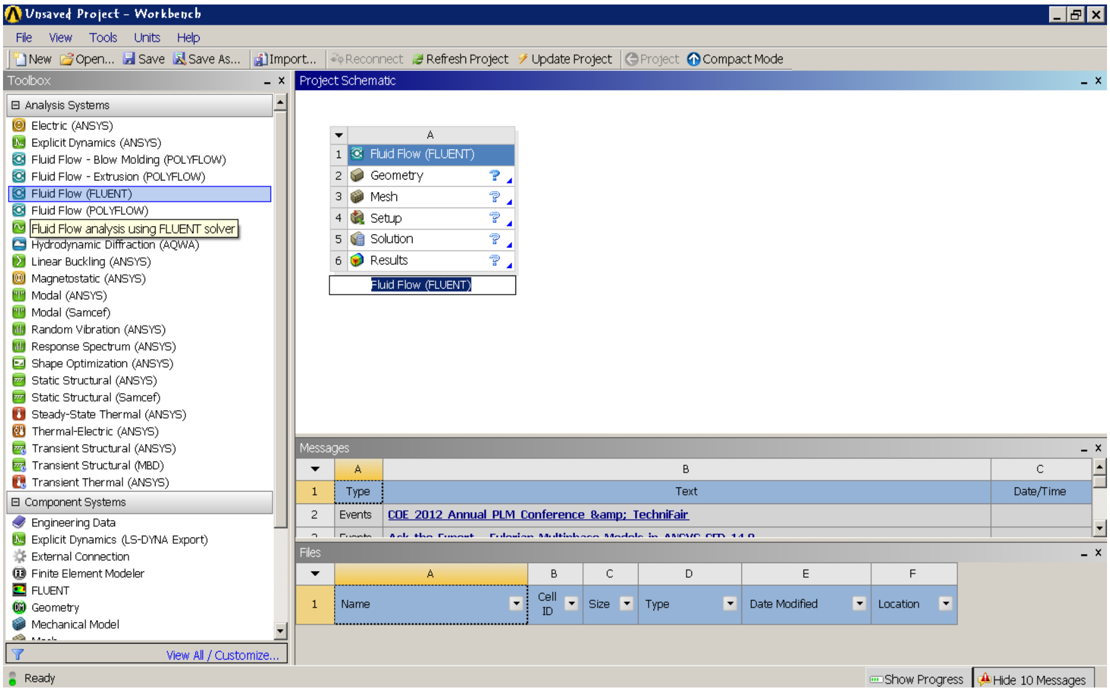


Figure 3.1: ANSYS Workbench.

3.2 MODELING THE FLOW DOMAIN AND GEOMETRY

The geometry nozzle and spray chamber was sketch and design by using ANSYS DesignModeler in Workbench. The geometry was sketch and design ni 2D using “surface from sketches” concept. The material of model was set to fluid. Figure 3.2 show the design of high pressure chamber and also 2D model as shown at Figure 3.3 that used in this simulation. The axisymetry 2D model was used because easy to sketch and much faster to run than 3D in FLUENT simulation.

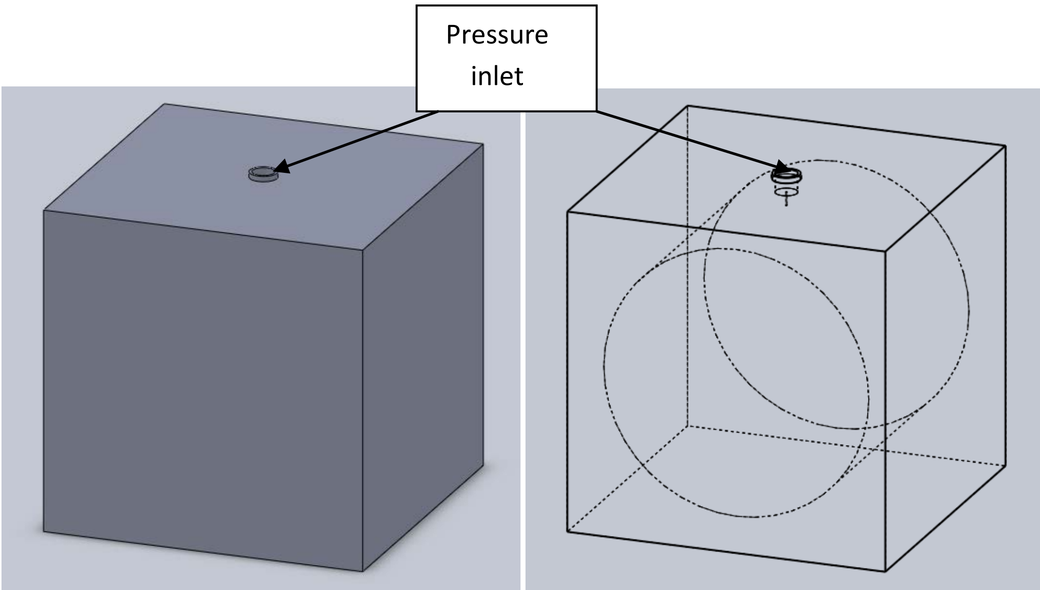


Figure 3.2: Geometry domain for high pressure chamber (3D).

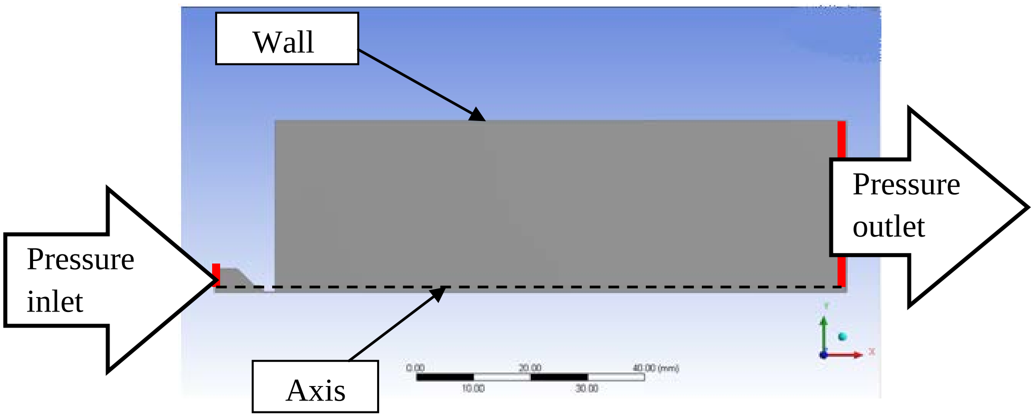


Figure 3.3: Flow Domain in Sac nozzle through the high pressure chamber.

3.3 MESHING

Model was meshing with automatic mesh in ANSYS ICEM CFD. The mesh generated for the first run of cases investigating boundary conditions. The large mesh cell was shown after enlarge at the orifice region at nozzle injector. After "refinement" step at mesh control, the mesh cell become much smaller and that will give more accurate result at the end of simulation. Figure 3.4 show the mesh of model after "refinement" step.

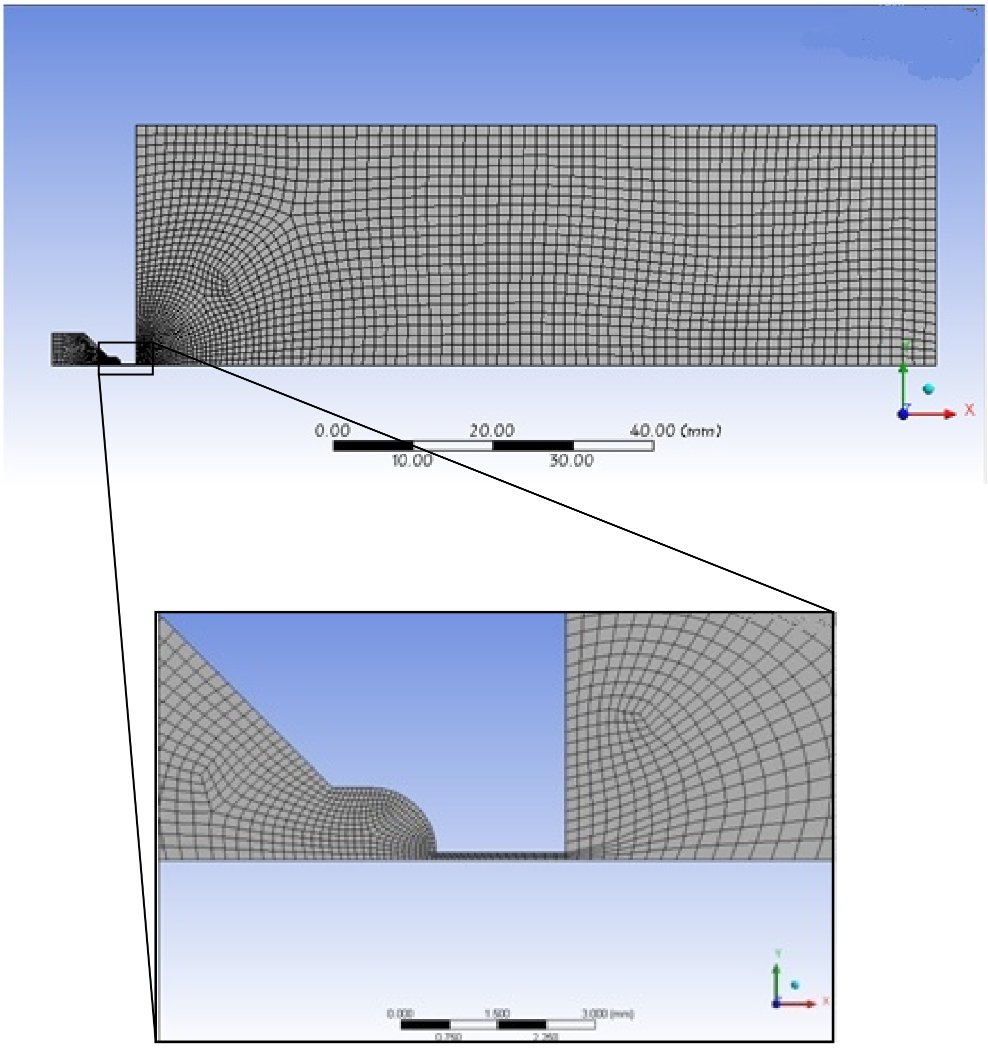


Figure 3.4: The mesh showing at enlarged orifice region.

\

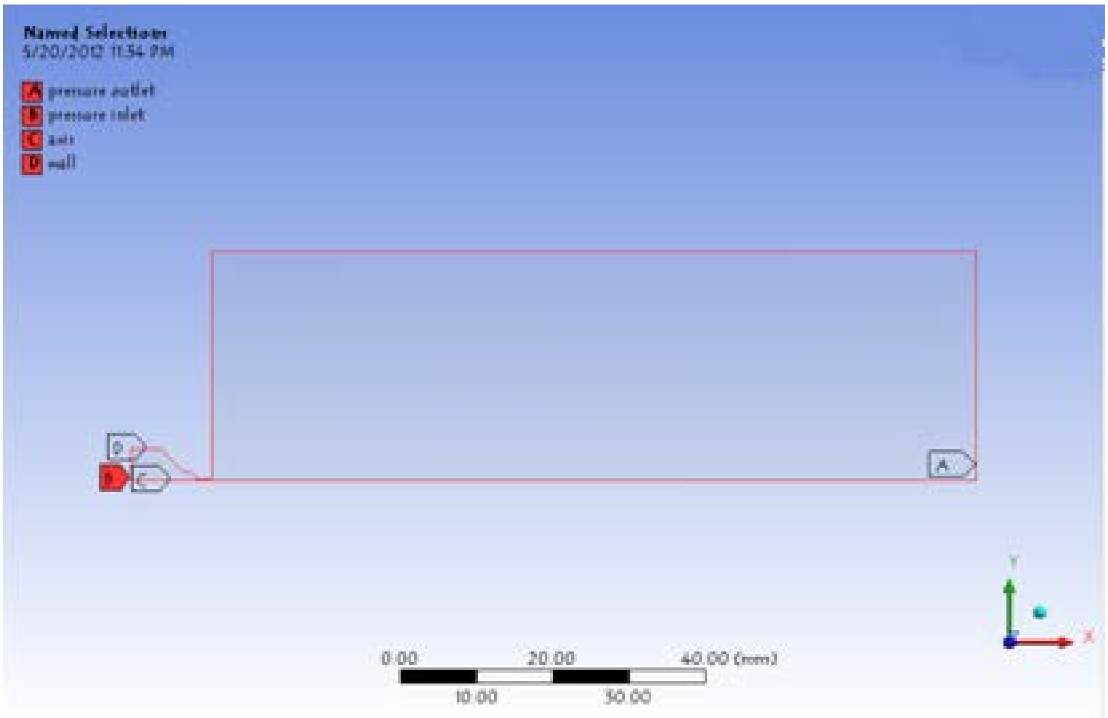


Figure 3.5: Named Selection of flow model.

For the next step is named each surface of model. The names include pressure inlet, pressure outlet, axis and wall (refer Figure 3.5). Every line in flow model must be named in this meshing step. It will help for the next step which is to setup the boundary condition and parameters of flow model such has pressure, temperature, gravity force and any other parameters.

3.4 SETUP

After mesh, Setup was used to launch the appropriate application in ANSYS FLUENT. All parameters like load, boundary condition, type of material, type of mixture, type of fluid, spray model setup, injection setup and otherwise were inserting in this setup.

3.4.1 Turbulence Models

Turbulent flows are extremely complex and time-dependent. Most fluid flows can be described by the full Navier-Stokes equation. However, it is still unfeasible to solve these equations directly via Direct Numerical Simulations (DNS) using current technology, except for low Reynolds numbers in simple geometries. Therefore, a Reynolds averaging method (RANS) is usually used to solve for the mean quantities rather than for all details of turbulence. A turbulence model is then required in order to close the system equations. FLUENT provides the following choices of turbulence models:

i. Spalart – Allmaras model
ii. *k*-ε models
   a. Standard *k*-ε model
   b. Renormalization-group (RNG) *k*-ε model
   c. Realizable *k*-ε model
iii. *k*-ω models
   a. Standard *k*-ω model
   b. Shear-stress transport (SST) *k*-ω model
iv. ν2-*f* model
v. Reynolds stress model (RSM)
vi. Large eddy simulation (LES) model

For this case, the *k*-ε two-equation model was chosen because this form of model is easy to solve, converges relatively quickly, is numerically robust and stable, is able to solve large domains and high Reynolds numbers and requires minimal computational expense, which makes it an attractive choice for many industrial problems.

3.4.2 Boundary Condition

The Fluent User Manual recommends a mass flow or pressure inlet boundary conditions for compressible flows and the pressure are input as gauge pressures relative to the operating pressure defined in the Operating Conditions. The boundaries condition's parameters for Pressure Inlet and Pressure Outlet (inside the chamber) were setup as shown at Figure 3.6. The same wall boundary was applied for the wall adjacent to the injector in all the cases run. Additionally, an axsymmetric boundary condition was applied to the singular axis in the axial direction.

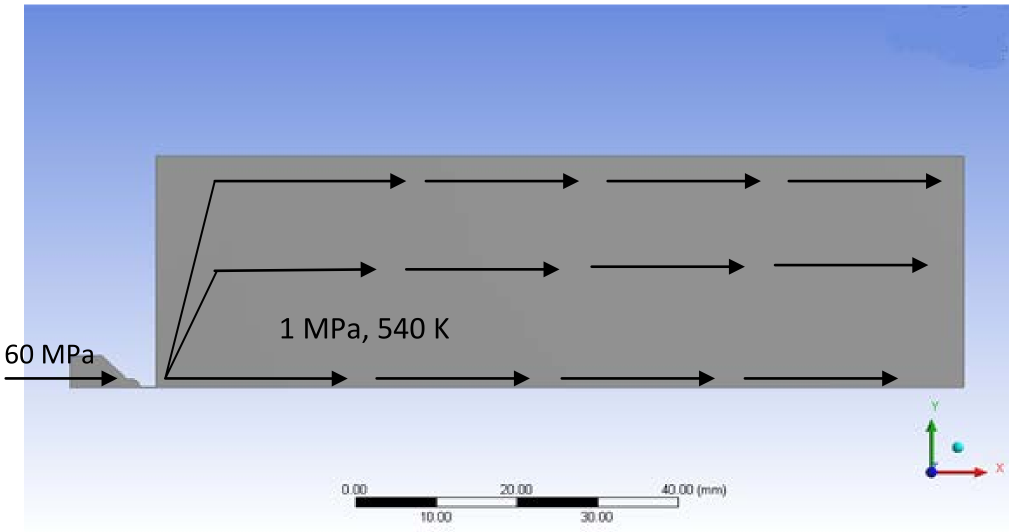


Figure 3.6: Boundary condition.

3.5 SOLUTION

The segregated solver is used extensively in this investigation. The pressure-velocity coupling refers to the way mass continuity is accounted for when using the segregated solver. Three methods are available for pressure-velocity coupling, which are: SIMPLE, SIMPLEC and PISO. SIMPLE is the default and most robust scheme, whereas SIMPLEC allows faster convergence for simple problems. PISO is useful for unsteady flow problems and for meshes with cells of higher than average skew. This study has found the SIMPLE method to be sufficient even for unsteady calculations.

3.6 RESULTS

After the all solution and boundary condition of any parameters was initializing, the simulation was check for the error before run calculation. If there is some error all the parameter, boundary condition, turbulent model, type of solve and mesh need to modify. If there is some incomplete information and need to fill completely. The simulation model was check until there is no error.

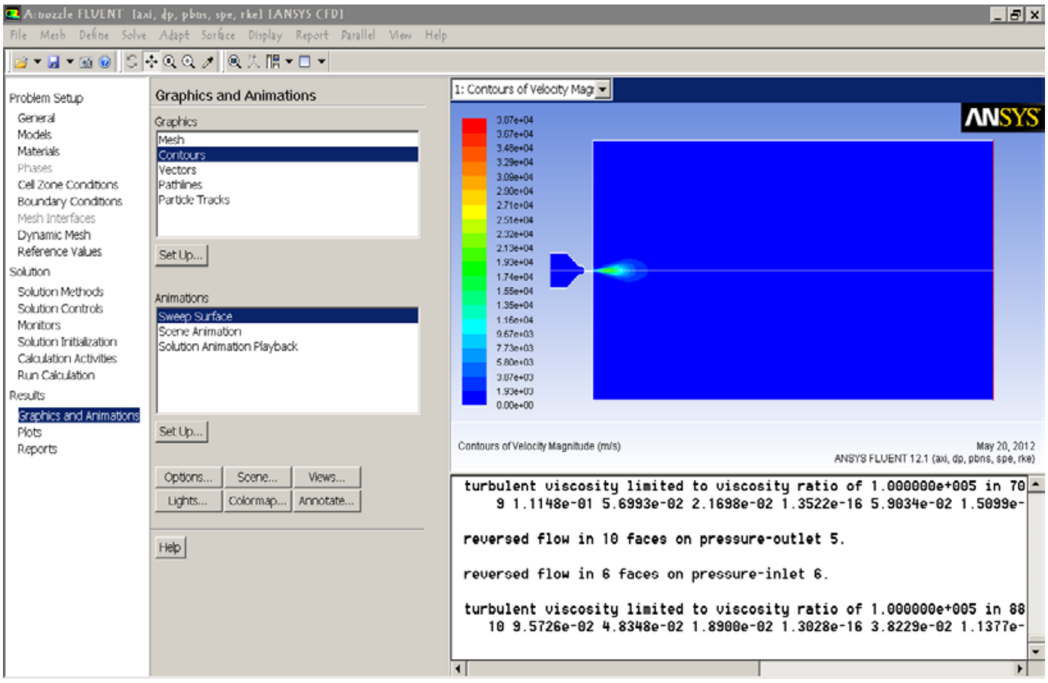


Figure 3.7: Simulation result.

The sample of simulation result was shown at the Figure 3.7. The simulation was running using difference iteration and all the parameter was recorded. Under Graphics and Animation, the spray image was named and saved. Simulation will continue for difference boundary condition and parameters. The spray image for each parameter was saved. Spray cone angle and penetration length was recorded.

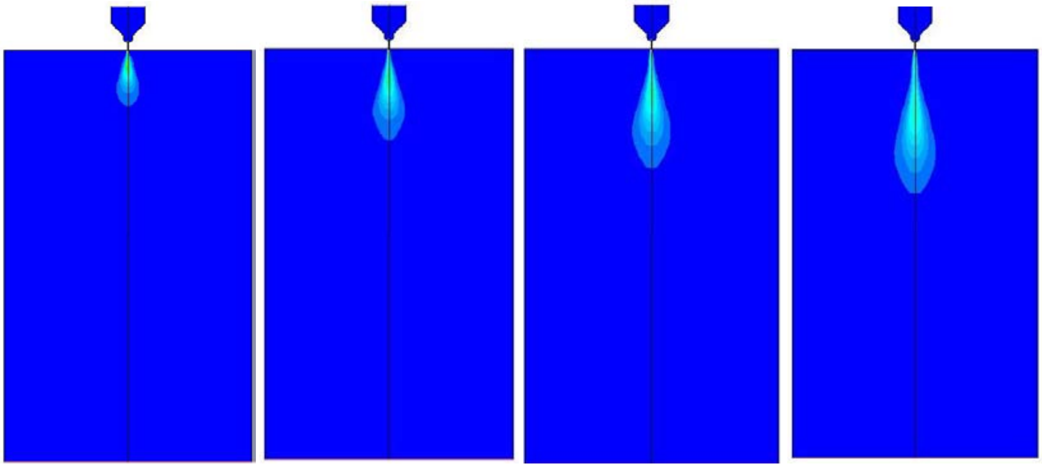
CHAPTER 4

RESULTS AND DISSUSSIONS

The flow model simulation tests were performed at:

i. Fuel pressures between 60 MPa and 160 MPa.
ii. Ambient air pressures (pressure inside combustion chamber) of 1 MPa.
iii. Ambient air temperature (temperature inside combustion chamber) of 540 K.
iv. Single-hole VCO and Sac nozzle with various orifice diameters.

A sequence of images taken 1 ms after the start of injection for a 3.41 ms injection is shown in Figure 4.1. The images were taken at 1 MPa of ambient pressure and four values injection at an ambient temperature of 540 K. The spray can be considered essentially non-evaporating at this temperature. The expected increase in penetration resulting from an increase in injection pressure is observed. The structure of the spray is consistent, with areas of atomized droplets being entrained at the edge of the spray where fuel air interaction takes place. The influence injection pressure can be clearly seen at Figure 4.1 between these four spray images, with the higher injection pressure resulting in a noticeable increasing in spray penetration length.



60 MPa 100 MPa 130 MPa 160 Mpa

Figure 4.1: Comparison of result for difference injection pressure.

4.1 INFLUENCE OF INJECTION PRESSURE

Injection pressure has significant effect on spray liquid penetration. The VCO nozzle was used with 0.2 mm orifice diameter was used to investigate the effect of injection pressure to spray penetration length. As expected, higher rail pressures result in a higher rate of spray tip penetration. The effect of increasing pressure on spray tip penetration is depicted in Figure 4.2. The spray tip penetration gets longer as the injection pressure increases. This result is related to both higher quantity and higher velocity of the droplets at higher injection pressures. When the spray lost its momentum related to lower quantity or lower injection pressure, the difference of the penetration lengths got longer at the downstream region of the spray. Proportional to injection pressure, the spray penetrates faster at higher injection pressures.

It was however found that there was little difference between 160 MPa and 130 MPa injection pressure results. At 160 MPa, the injector has the potential of suppression of flow separation and cavitations. Due to the higher injection pressure, the injector produces higher penetration length. But before 0.8 ms, as fuel lost its velocity in the ambient air, the penetration length of 160 MPa gets similar to the 130 MPa injection pressure. In addition to this, it is well known that spray behavior is a consequence of flow characteristics inside the nozzle. Due to the small geometries of nozzle orifices and the high velocities that take place inside them, physics of this phenomenon are difficult to be studied, especially when cavitations appears. After the 160 MPa injection pressure reach 50 mm penetration length, the penetration rate increase drastically increase due time which reach the maximum penetration length. It mean the cavitations phenomenon was not occur at certain pressure when pressure injection decrease with time (Metin et al, 2011).

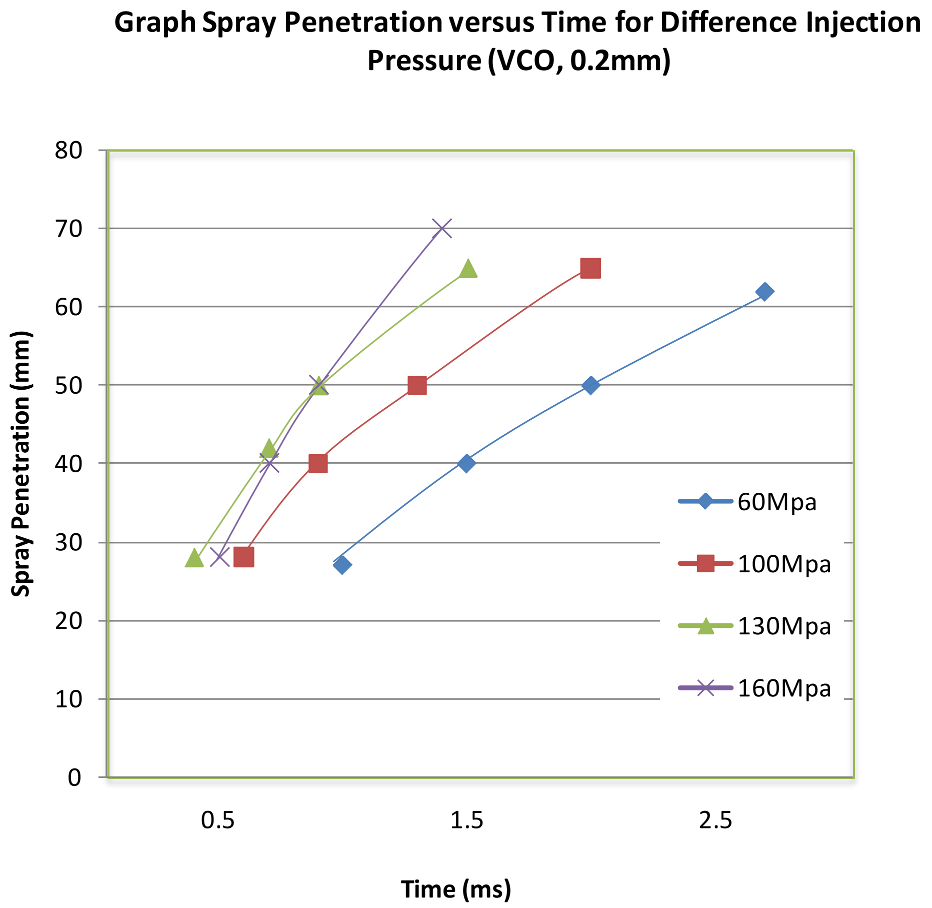


Figure 4.2: Comparison of experimental penetration rate for difference injection pressures (VCO, 0.2mm).

4.2 INFLUENCE OF NOZZLE GEOMETRY

The nozzle type effect on penetration is shown in Figure 4.3**.** Two nozzle designs and three nozzle geometries were tested. The main dimensions are summarized in Table 1. Figure 4.3 shows a comparison of penetration rate at one operating condition ambient pressure = 1 MPa and injection pressure = 100 MPa

Table 4.1: Details of nozzle used in the study.

Nozzle Type	Orifice diameter (mm)
VCO	0.10
VCO	0.15
VCO	0.20
Sac	0.15
Sac	0.20

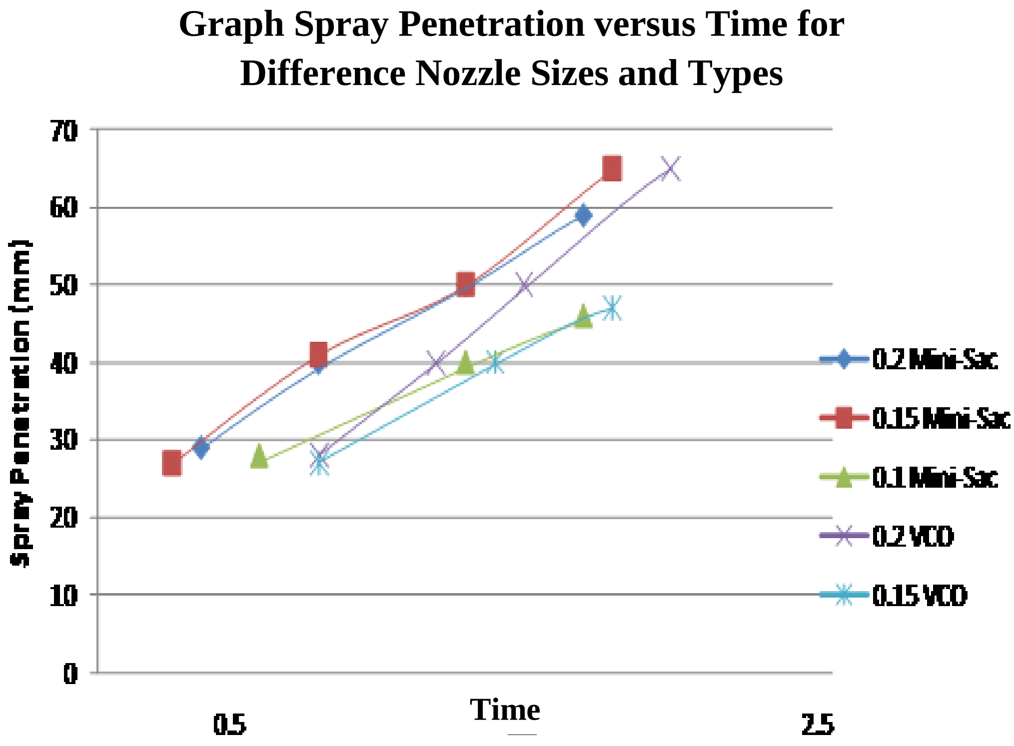


Figure 4.3: Comparison of result penetration rate for difference nozzle orifice diameter and types.

A higher penetration rate is observed for the 0.15 mm nozzle orifice diameters for the Sac compared to VCO nozzle. VCO nozzle with small orifice diameter produces short penetration length and slowest penetration rate. The smaller diameter orifice would be expected to produce a finer more atomized spray. Again, the higher evaporation rate of the smaller droplets could explain the observed result. It has also been shown that VCO nozzle injectors cause a greater pressure loss within the nozzle geometry hence a lower difference pressure between injection pressure and ambient pressure at the nozzle orifice and lower spray penetration.

4.3 COMPARISON CONE ANGLE BETWEEN TWO DIFERENCE NOZZLE GEOMETRY

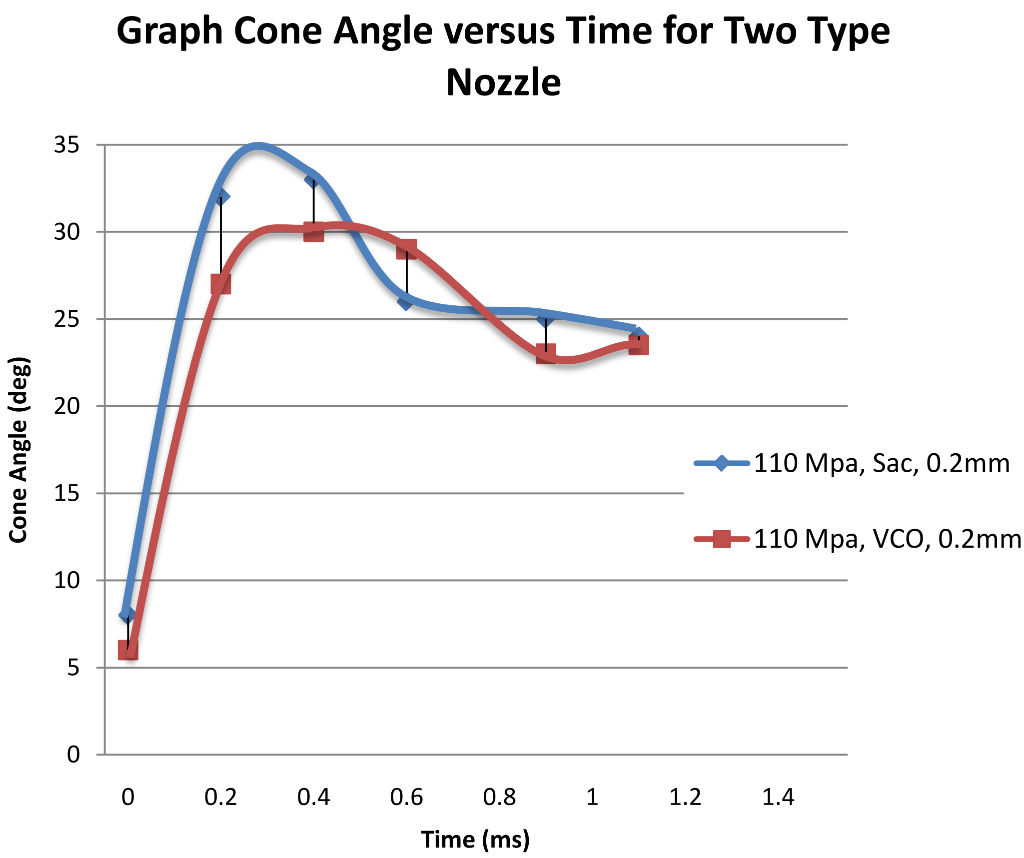


Figure 4.4: Comparison cone angle between Sac and VCO with same parameters.

The Figure 4.4 was show the spray cone Sac nozzle has wider angles at the initial stages of spray evolution (0.3 ms) compare with VCO under same condition. However the cone angle of sac nozzle suddenly decreases as the spray penetration below VCO cone angle at 0.7 ms of injection time. After the spray developed enough at time 1.2 ms, the spray cone angle settles to constant value and both spray cone angle became equal for Sac and VCO nozzle. The combination of difference geometry nozzle only has a minor effect on spray penetration rate.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

This chapter concludes all have been done and achieve in this research and recommendation on how to improve in future research to get a better result.

5.1 CONCLUSION

The influences of nozzle tip geometry, ambient pressure and injection pressure were simulating over a range of conditions representative of a modern diesel engine. From this simulation, differences in spray structure were observed that spray liquid penetration starts from zero velocity at the nozzle exit and reaches maximum velocity at around breakup time.

The expected trends with injection pressure were observed, except at high injection pressure where a lower rate of penetration than expected was observed. The effect on penetration of a change in injection pressure is reduced at higher injection pressures. In addition, the increased rate of evaporation at high injection pressures may also influence the penetration rate of the spray. However, the higher injection pressures may onset of cavitations inside the orifice of nozzle and retard the penetration length.

The VCO nozzles were observed to show a lower rate of penetration than Sac nozzles of the same dimensions. It is proposed that this is due to better atomization of the spray from the VCO nozzle, possibly due to differences in internal flow structure, resulting in a higher rate of evaporation. The rate of injection was also shown to have an effect on the local cooling of the gas adjacent to the penetrating spray, causing a retardation of the vaporizing sprays at higher fuelling rates. This effect, coupled with faster evaporation of the smaller droplets which are produced at higher injection pressures, is suggested as the cause of the apparent reduction in spray tip penetrations.

At beginning of injection, Sac nozzle produce wider cone angle than VCO effect of difference geometry but actually two nozzles have mostly the same cone angle. The combine of difference geometry nozzle only has a minor effect on spray penetration rate. For the range of nozzle type as well as cone angles investigated, the effect of nozzle penetration on the spray characteristics was larger than the effect of nozzle cone angle.

5.2 RECOMMENDATION FOR THE FUTURE RESEARCH

For the future work, upgrade the FLUENT software by upgrading video and sound simulation prediction. Not really an improvement to the model itself, but prediction of spray characteristic certainly has a large effect on the final results as extensively discussed. It wills well if the video of spray droplets can be made from start of atomization process and at the end their start burning. There more result can be concluded.

REFERENCES

Taskiran, O.O. and Metin, E. 2011. Experimental Study on Diesel Spray Characteristics andAutoignition Process.*Journal of Combustio*n.**2011**: 20.

Faeth, G. M., Hsiang, L. P., Wu, P. K. 1995. Structure and BreakupProperties of Sprays.*International Journal of Multiphase Flow*.**22**: 99-127.

Sibendu, S., Anita I. R., Douglas E. L., Suresh K. A. 2011.Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions.*Fuel*.**2011**(90): 1267–1276.

Gregory, E. B. Jr., 2009. Modeling the Fuel Spray and Combustion Process of the I gnition Quality Tester with KIVA-3V. *Presented at the Fall Meeting of the Western States Section of the Combustion Institute Irvine*. National Renewable Energy Laboratory: 26−27 October.

Patrick, V. F. Effect of Injection Parameters and Injection System on Spray Characteristics for HSDI Diesel Engines.*9th International Conference on Liquid Atomization and Spray Systems Sorrento.*University of Wisconsin, Madison: 13 18July.

John, A., Andres, A. and Pedro, B. 2009.Study of diesel sprays using computational fluid dynamics. *Rev. Fac. Ing. Univ. Antioquia*.**49**: 61-69.

Daniele, S. 2009. Diesel Nozzle Flow and Spray Formation: Coupled Simulations with Real Engine Validation. Ph.D Thesis.University of Stuttgart, Italy.

Kourosh, K. 2004. Multiple-Injection Diesel Spray CharacterisationAt Elevated Pressures and Temperatures. Ph.D Thesis.School of Engineering, University of Brighton.

Julien, L. 2006. Characteristics of Diesel Sprays at High Temperatures and Pressures. Ph.D Thesis.School of Engineering, University of Brighton.

Sibendu, S., Anita, I.R., Douglas, E.L. and Suresh, K.A. 2011.Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions.*Fuel 90*: 1267–1276.

Gregory, E.B.Jr., Anthony, D., Chen, J.Y., Gregory, C., Jon, L., Matthew, A.R., Bradley, T.Z. and Anthony, M.D. 2009.Modeling the Fuel Spray and Combustion Process of the Ignition Quality Tester with KIVA-3V. *Presented at the Fall Meeting of the Western States Section of the Combustion Institute Irvine.* University of California, California: 26-27 October.

Han, S.P., Hyung, J.K. and Chang, S.L. 2010. Comparison of experimental and predicted atomization characteristics of high-pressure diesel spray under various fuel and ambient temperature.*Journal of Mechanical Science and Technology 24.*(7) :1491-1499.

Jeong, K.Y. and Hajime, F. 2003. A Study on the Behavior Characteristics of Diesel Spray by Usinga High Pressure Injection System with Common Rail Apparatus.*KSME International Journal.***17**(9): 1371—1379.

Kumar, P.M., Pandurangadu, V., Bharathi. V.V.P. and Deepthi, V.V.N. 2011. Analysis of Fuel Spray with Various Injection Orientations In a Diesel Engine: Through CFD Simulation. *International Journal of Engineering Science and Technology (IJEST).***3**(5): 3746-3750.

Lukasz, J.K. and Andezej, T. 2012. Numerical Simulations of a Simultaneous Direct Injection of Liquid And Gaseous Fuels Into a Constant Volume Chamber. *Journal of Power Technologies* **92**(1): 12-19.

APPENDIX A1

GANTT CHART FOR FYP 1

ACTIVITY	W 1	W 2	W 3	W 4	W 5	W 6	W 7	W 8	W 9	W 10	W 11	W 12	W 13	W 14	W 15
Get the project title	■	■													
Research on the title		■	■	■	■	■	■	■	■	■	■	■	■	■	
Study the simulation method					■	■	■	■	■	■	■	■	■		
Writing the report (Chapter 1,2,3)							■	■	■	■	■	■	■	■	
Final Year Project (FYP) 1 Presentation															■

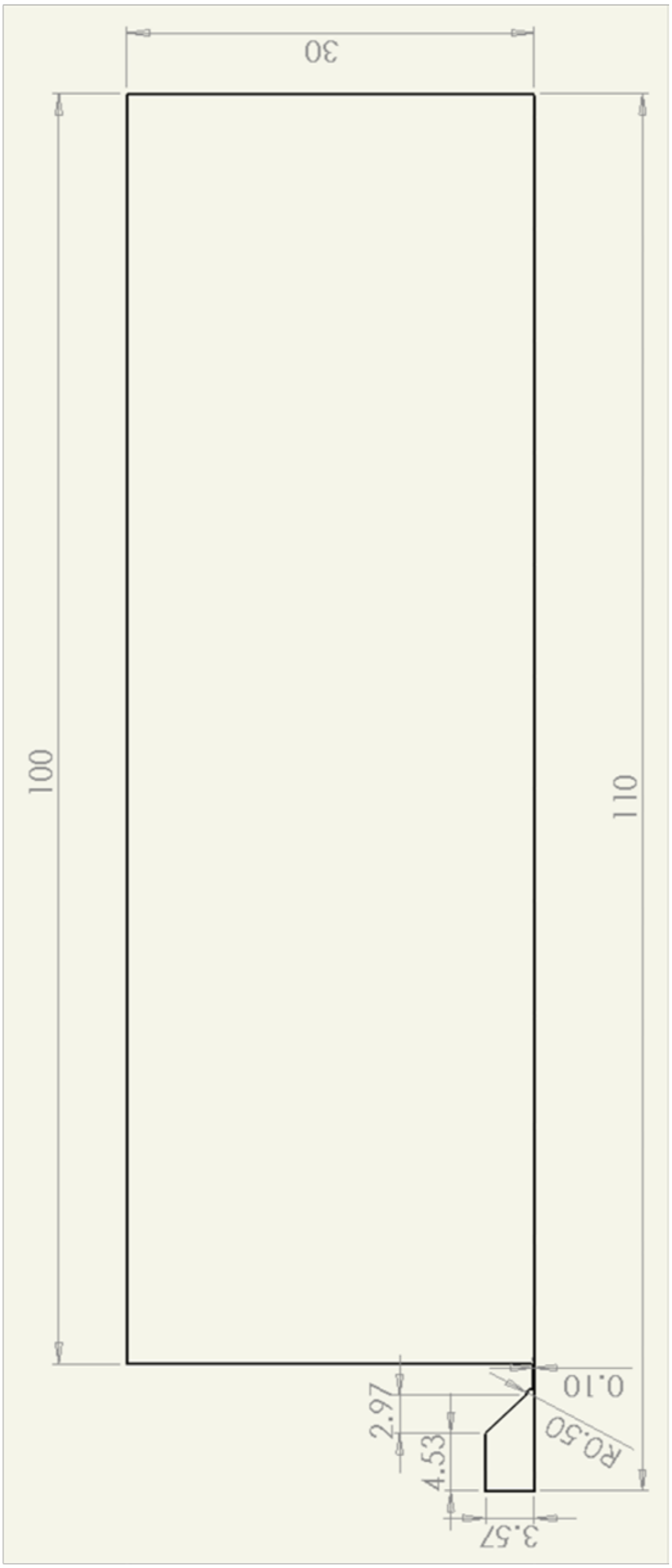
APPENDIX A2

GANTT CHART FOR FYP 2

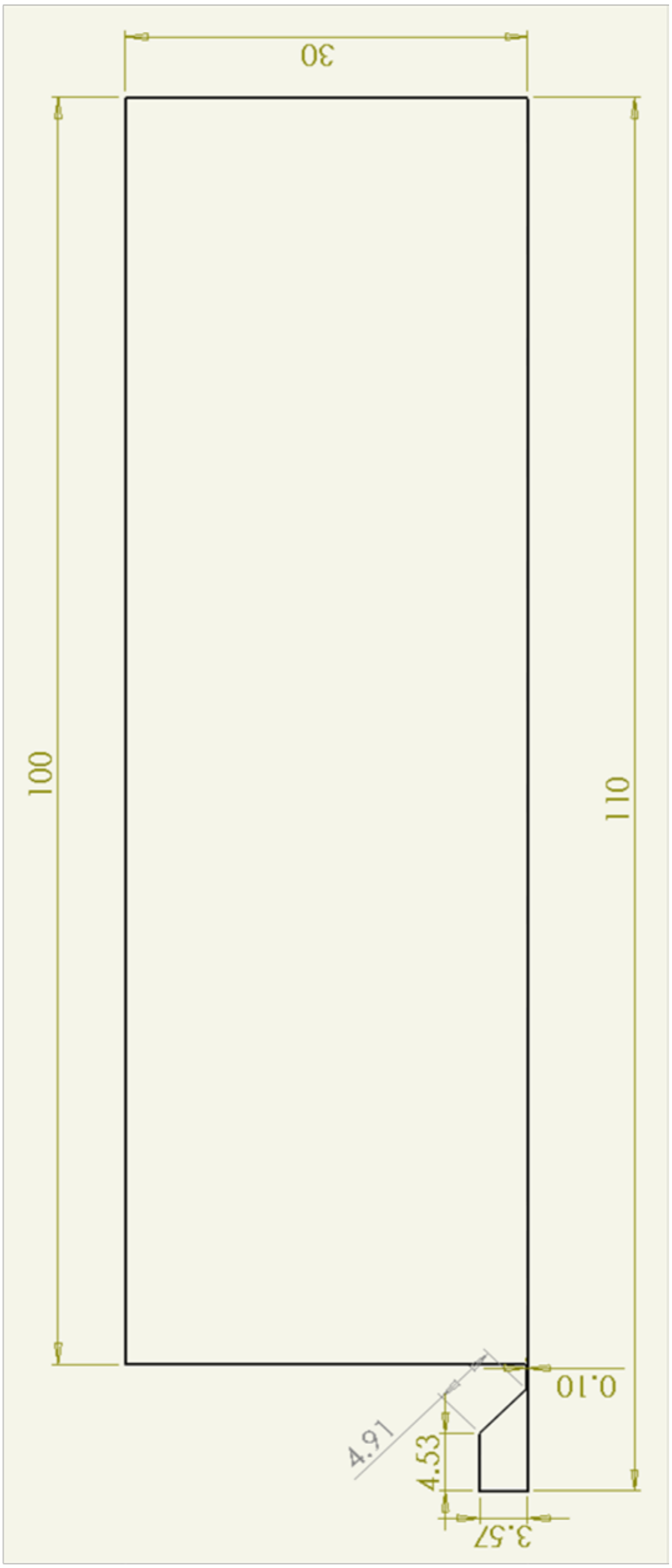
ACTIVITY	W 1	W 2	W 3	W 4	W 5	W 6	W 7	W 8	W 9	W 10	W 11	W 12	W 13	W 14
Discuss the project title with supervisor														
Study the simulation method														
Run the ANSYS FLUENT														
Collect data from simulation														
Writing the report (Chapter 4,5)														
Final Year Project (FYP) 2 Presentation														

APPENDIX B1

DIMENSION OF SAC NOZZLE



APPENDIX B2



DIMENSION OF VCO NOZZLE