INFLUENCE OF ORIFICE NOZZLE GEOMETRY ON FLOW AND SPRAY CHARACTERISTICS

NOOR AZUANI BINTI SULIMAN

Report submitted in partial fulfilment of the requirement for the awards of

Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering

UNIVERSITI MALAYSIA PAHANG

JUNE 2013

ABSTRACT

This thesis presented about the effects of nozzle orifice geometry on flow and spray characteristics. The thesis deals with the analysis on the different inlet diameter of nozzle orifice and the different inlet pressure on the flow and also spray characteristics. The objective of this thesis is to compare the final performance of flow and spray characteristics between the different designs of nozzle orifice. Besides that, the purpose of this thesis is to determine the flow and spray characteristics with different inlet diameter of nozzle orifice. This thesis also purposes to study the flow and spray characteristics with different inlet pressure. The geometry of nozzle orifice used is the inlet diameter and the parameter used is the pressure. The data used for analysis is obtained from the simulation by using Solidworks 2012 software. The post-processing method was performed using the simulation with certain parameters that can be chose and graph is plotted by using assistance software such as Microsoft Excel. From the result, it is observed that the performance of 150 µm diameter of nozzle orifice with inlet pressure of 250 bar is better than other combination of inlet diameter and inlet pressure. Future work, this comparison between different inlet diameter and different inlet pressure must do in experimental to support the result. There are many factors of geometry and parameters that required doing the comparison in order to study the flow and spray characteristics.

ABSTRAK

Tesis ini membentangkan mengenai kesan perbezaan muncung geometri orifis pada aliran dan ciri-ciri semburan. Tesis ini berkaitan dengan analisis pada diameter salur masuk yang berbeza lubang muncung dan tekanan masuk yang berbeza pada aliran dan juga semburan ciri-ciri. Objektif projek ini adalah untuk membandingkan prestasi akhir aliran dan cirri-ciri semburan antara reka bentuk yang mempunyai lubang muncung yang berbeza. Selain itu, tujuan projek ini adalah untuk menentukan aliran dan ciri-ciri semburan dengan diameter salur masuk jenis orifis muncung yang berbeza setiap satu. Tesis ini juga bertujuan untuk mengkaji aliran dan ciri-ciri semburan dengan tekanan masuk yang berbeza. Geometri lubang muncung yang digunakan ialah diameter salur masuk dan parameter yang digunakan adalah tekanan. Data yang digunakan untuk menganalisis diperoleh daripada simulasi dengan menggunakan perisian Solidworks 2012. Kaedah pemprosesan dilakukan dengan menggunakan simulasi dengan parameter tertentu yang boleh dipilih mengikut kesesuaian dan graf diplot dengan menggunakan perisian bantuan seperti Microsoft Excel. Dari keputusan yang diperolehi itu, ia membuktikan bahawa prestasi 150 μm diameter orifis muncung dengan tekanan masuk 250 bar adalah lebih baik daripada gabungan diameter salur masuk dan tekanan masuk yan lain. Untuk kerja pada masa depan, perbandingan antara diameter salur masuk yang berbeza dan tekanan masuk yang berbeza perlu dilakukan dalam bentuk eksperimen untuk menyokong keputusan yang diperoleh melalui simulasi yang dilakukan. Terdapat banyak faktor geometri dan parameter yang dikehendaki untuk melakukan perbandingan dalam mengkaji aliran dan ciri-ciri semburan.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF SYMBOLS

µ	micro
m/s	meter per second
cm	centimeter

LIST OF ABBREVIATION

AMR	Adaptive Mesh Refinement algorithm
CFD	Computational Fluid Dynamic
KH-ACT	Kelvin Helmholtz-Aerodynamic Cavitation Turbulence

CHAPTER 1

INTRODUCTION OF THE PROJECT

1.1 BACKGROUND STUDY

The nozzle orifice geometry is one of the parts that important to the flow or to the spray characteristics. Nozzle orifice geometry will give an effect to the flow of liquid or vapor and also the spray characteristics. Various types of geometry can give a various type of flow. There are many patterns of nozzle orifice such as the cylindrical nozzle orifice and the conical nozzle orifice. They are causing the different effect on the flow of liquid or vapor. The nozzle orifice geometry will determine the final result of the flow of the machine or something that using this kind of spraying method.

There have been clear situation in this engineering field that the laws of fluid dynamics predict that the shape or the geometry of an orifice nozzle influences its discharge. For example, a round orifice might be expected to discharge more flow than a slit-like orifice nozzle of the same area, while the nozzle discharges better than an abrupt narrowing. The effects of this orifice nozzle geometry on inner flow and spray characteristics can be examined using the computational fluid dynamic simulation software (Sibendu, 2010).

In order to seek out the best performance from different kind of orifice nozzle geometry, a research is carried out by using a simulation. The type of orifice nozzle is chosen and the parameter such as the inlet size and the pressure of the orifice nozzle is selected to determine the best geometry of orifice nozzle will give the best performances.

1.2 PROBLEM STATEMENT

The final result of the performance of a flow depends on many factors and one of the factors is the geometry of the orifice nozzle itself. Nowadays, there are many types of orifice nozzle that have been used to the spray or something related to the flow characteristics. All that kind of types has their own advantages and disadvantages. But, from all the types of nozzle orifice, it must have the specific parameter that should be taken in order to study the flow pattern and also the spray characteristics.

The size of inlet diameter and the average pressure inside the orifice nozzle right before it comes out of the orifice nozzle, which is injection pressure are two main points that I have focused in this project. Then, the size of inlet diameter and injection pressure will determine the pattern of the flow and spray characteristics.

1.3 OBJECTIVES

The objectives of this project are as follows:

i. To determine the flow and spray characteristics by the different size of the inlet diameter of the orifice nozzle.
ii. To analyze the flow and spray characteristics of the different injection pressure.
iii. To compare which size of inlet diameter and injection pressure that will give the biggest spray pattern.

1.4 PROJECT SCOPES

This project is focusing on the effect of different size of inlet diameter and the different injection pressure on an orifice nozzle on flow and spray characteristics. To complete this project, the actions are required:

i. Study of different orifice nozzle geometry that related to the flow and spray characteristics.
ii. Study the effect of different size of inlet diameter that will lead to different type of flow which is in the range of 100-250µm
iii. Study the effect of different injection pressure that will cause a different type of flow which is in the range of 100-250 bar.
iv. Study the analysis of the simulation done.

1.5 HYPOTHESIS

The influence of flow and spray characteristics will depend on the orifice nozzle geometry and the parameter focused are the size of inlet diameter and the injection pressure of the orifice nozzle. So, the variety the variation of these parameters, the flow pattern and spray characteristics will vary too.

1.6 STRUCTURE OF THESIS

This thesis purposes to study the influence of orifice nozzle geometry on flow and spray characteristics. Besides that, this thesis also aims to analyze the performance of flow and spray characteristics with a different inlet diameter of orifice nozzle and different injection pressure. This thesis consists of five (5) chapters.

First chapter describes overall framework of basic information of this project such as introduction, problem statement, objectives of the project, project scopes and structure of thesis was verified. The main ideas of this project were stated in introduction.

In second chapter, various reviews on theoretical topics which are required as a background study were present. Every important information and theoretical study related to this project is stated in this chapter. Brief explanations about the different types of orifice nozzle, the various result with the related parameter that been study for this project. Some of the explanations give extra information which is useful in conducting this project.

In third chapter, all the method used when conducting the project was described including explanation. The overall methodology sequences are mentioned and in detail.

Otherwise, chapter four is about results and discussions about the project. This chapter explains the result and analysis that got from the graph.

The conclusion of the overall project, recommendation and future works are stated in fifth chapter of the thesis. The conclusion made based from the result obtained, the encountered problem lead to the recommendation to troubleshot the predicaments. The area of improvement will be the source of future projects.

Finally, the references for this project were listed and follow appendices. The related tables are included in appendices for general review.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

The flow of liquid or vapor depending all on the how the nozzle is designed. The effects of orifice nozzle geometry are examined by the final flow of liquid or vapor and also the spray characteristics. Certain orifice nozzle geometry will give a result of slow down the primary breakup, increasing the spray penetration, and at the same time reducing the dispersion too (Sibendu, 2010). The primary breakup is believed to occur in the region which is very close to the orifice nozzle tip as a result of turbulence, aerodynamics and inherent instability caused by the cavitations patterns inside the injector orifice nozzle (Sibendu, 2010b).

The formation of bubbles in a liquid flow will lead to a two-phase mixture of liquid and vapor or gas. It is called cavitations also. In other words, it is the phase transition from liquid to vapor due to the low pressure. There is a pressure depression due to the acceleration of the fluid. If the static pressure falls below the vapor pressure then the phenomenon of this cavitations will take place (Bergwerk, 1959). Cavitations may be gives an advantages to the development of spray because the primary breakup and subsequent atomization of the liquid can be enhanced. In addition, a cavitations will rise up the velocity of liquid at the orifice nozzle exit due to the reduced of exit area available for the liquid. The extended of cavitations patterns from the starting point near the orifice nozzle inlet to the exit will give effect to the formation of emerging spray (Sibendu, 2010b).

The literature review indicates that while the effect of orifice geometry on the injector flow and spray processes has been examined to some extent, its influence on the engine combustion and emissions is not well established (Som, 2010b, 2011). The influence of orifice nozzle geometry on spray and combustion characteristics has also not been studied numerically, mainly due to the complicated nature of flow processes associated. These form a major motivation for the present study which is to examine the effects of orifice nozzle geometry on inner nozzle flow. To characterize the effects of orifice geometry this is the smaller the size or orifice nozzle geometry, the higher the pressure. At the same time it will affect the flow of liquid too (Sibendu, 2012).

An experimental study is conducted to analyze the influence of conical and cylindrical nozzle orifices on the injection rate behavior of a common-rail fuel injection system at maximum needle lift in a cavitations test rig. They observed that compared to a cylindrical orifice a conical orifice reduces cavitations, increases flow efficiency, which is the discharge coefficient and exit velocity, although the fuel injection rate is reduced due to the smaller exit area (Benajes, 2004). The choking conditions were observed with cylindrical nozzles whereas for conical nozzles the mass flow rate had been always proportional to the square root of pressure drop indicating absence of cavitations at the nozzle exit. They also observed an increase in injection velocity due to the presence of vapor at the orifice exit for the cylindrical nozzle (Payri, 2005).

The other researcher is comparing the conical and cylindrical nozzles and found that the primary breakup region is strongly influenced by nozzle geometry. The effect of orifice geometry on spray penetration, liquid length, and cone angle have also been examined (Han, 2002). For non-evaporating sprays the increase in cone results in greater liquid penetration and smaller the cone angle (Payri, 2004 and Blessing, 2003). However, the other person found that spray tip penetration and the cone angle decreased due to increase in cone which is in direct contradiction to studies by Payri et al. and Blessing et al (Bae, 2002). Simulations provided further insights on this issue and confirmed that connect suppresses cavitations and turbulence inside the nozzle, which are known to enhance the primary breakup (Som, 2009).

The effects of injection pressure and orifice diameter on the flame structure and stabilization are well captured by the simulations as shown in Fig 2.1 below. The lift-off length was observed to increase with orifice diameter, a trend well captured by simulations. However, the lift-off length is slightly under predicted by the simulations. This trend can be explained by the fact that the fuel injection rate and hence the total mass injected increases with increase in orifice diameter. Thus to form a conducive fuel–air mixture for ignition, the amount of air to be entrained also increases thus increasing the flame lift-off length. The liquid length is seen to increase linearly with orifice diameter.

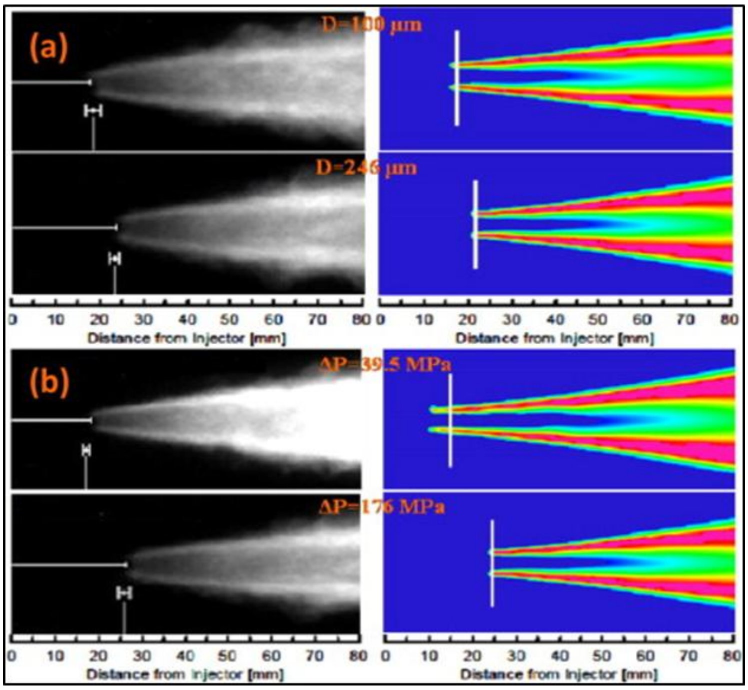

Figure 2.1: The comparison of the flow at different orifice diameter and injection pressure

Source: Sibendu, et al. (2011)

The other study about this influence of orifice nozzle geometry is based on dynamic mesh refinement. In this research they are using the model of breakup spray to simulate engine spray dynamics. The accurate prediction of the spray structure and drop vaporization requires accurate physical models to simulate fuel injection and spray breakup. They are using experimental and also the simulation by software to compare both results. For the experimental set up, they are using the primary breakup as a model to determine the flow and spray characteristics. For the simulation set up, they are using a dynamic mesh refinement algorithm (R. Kolakoluri, 2010).

Thus, the predicted spray structure and droplet distribution using AMR on the coarse mesh are similar to those predicted using the fine mesh. These results indicate that the present AMR algorithm can be used to obtain the same level of accuracy in spray modeling without the need to use a very fine mesh. The Fig 2.2 below shows the results from the Adaptive Mesh Refinement algorithm (AMR). The figure on the next page shows that the predicted result of development of spray from the injector of orifice nozzle after 1.6 meter second (R. Kolakoluri, 2010).

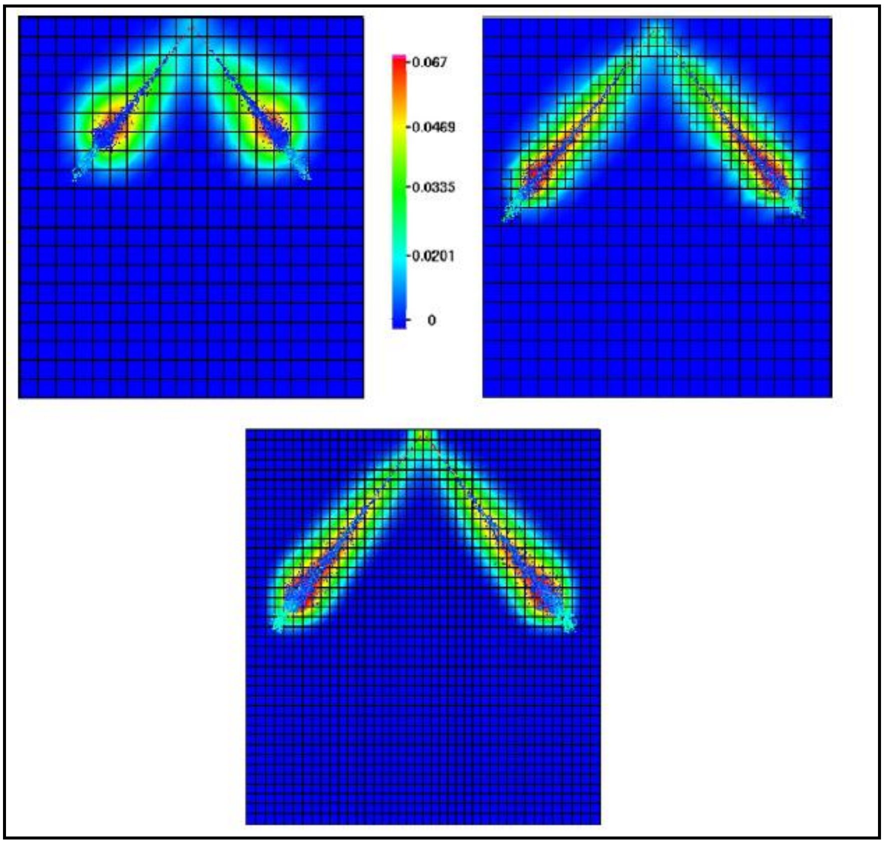

Figure 2.2: Predicted drop and vapor distributions of the gasoline spray at 1.6 ms after injection

Source : R. Kolakoluri et al. (2010)

Figure 2.3 on the next page shows the comparison of the experimental images that set up by Yi (2009) and the predicted drop distributions for injection pressure 100 bar and back pressure 1 bar at 0.7 ms after injection. The spray penetrations predicted by the model are in good agreement with the experimental data. Qualitatively speaking, the predicted droplet distribution is also satisfactory. Figure 2.4 on the next page also shows the comparison at1.6 ms after injection with the same conditions. The spray structure and the droplet distribution are well predicted using the present model.

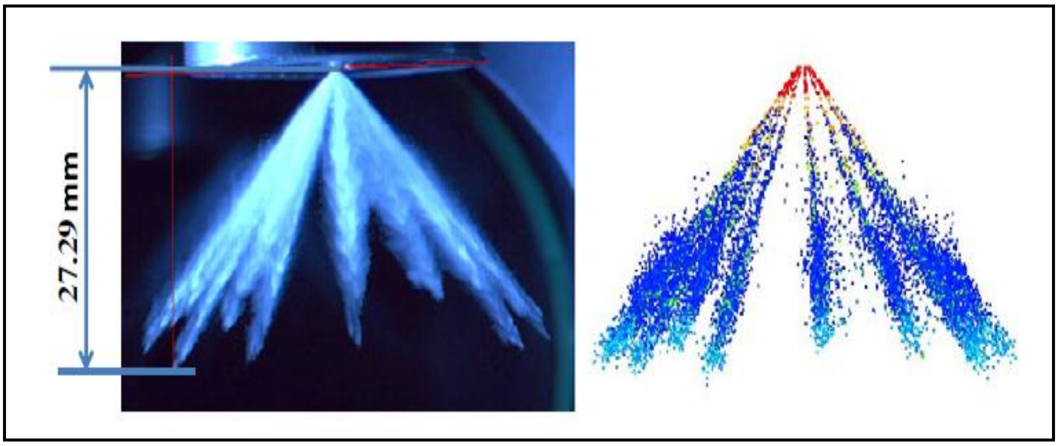

Fig 2.3 : Experimental image and predicted spray structure of the gasoline spray at 0.7 ms after injection

Source : R. Kolakoluri et al. (2010)

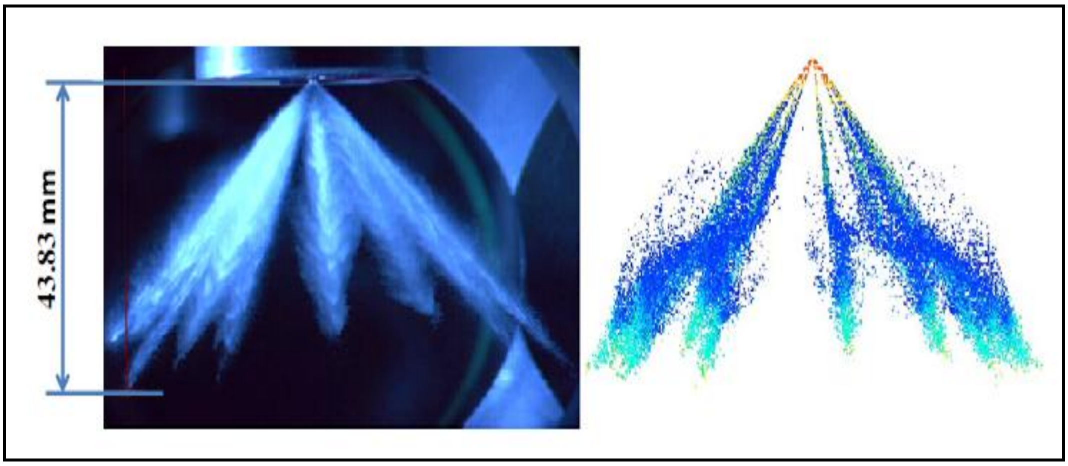

Figure 2.4 : Experimental image and predicted spray structure of the gasoline spray at 1.6 ms after injection

Source : R. Kolakoluri et al. (2010)

The level of back pressure will affect the momentum of the spray. With an increase in back pressure, the drag force experienced by spray particles increases, which will enhance the early breakup of droplets but reduce the spray tip penetration. The injection pressure is maintained constant and the back pressure is varied in this study. In this research search by R. Kolukoluri (2010), it stated that the parameter that they want to determine is the spray structures that coming out of orifice nozzle itself. The other parameters should be the constant variable. It means that the other parameters are constant the same. But each of them gives an effect if they are changing the variable. The other parameters are injection pressure, back pressure, gas identity, gas temperature and also the orifice diameter.

CHAPTER 3

METHODOLOGY

3.1 METHODOLOGY AND FLOW CHART

Methodology is one of the most important things to be considered in order that the project will run smoothly and will get the expected result. It will be discussed in the process of the project due to the flow chart. In this methodology, there are several steps that must follow in order to ensure that the objective of the project can be achieved starting from the literature finding until the submitting the final report. Below are the steps of the project which briefly being shortlisted into the flowchart.

3.2 FLOWCHART

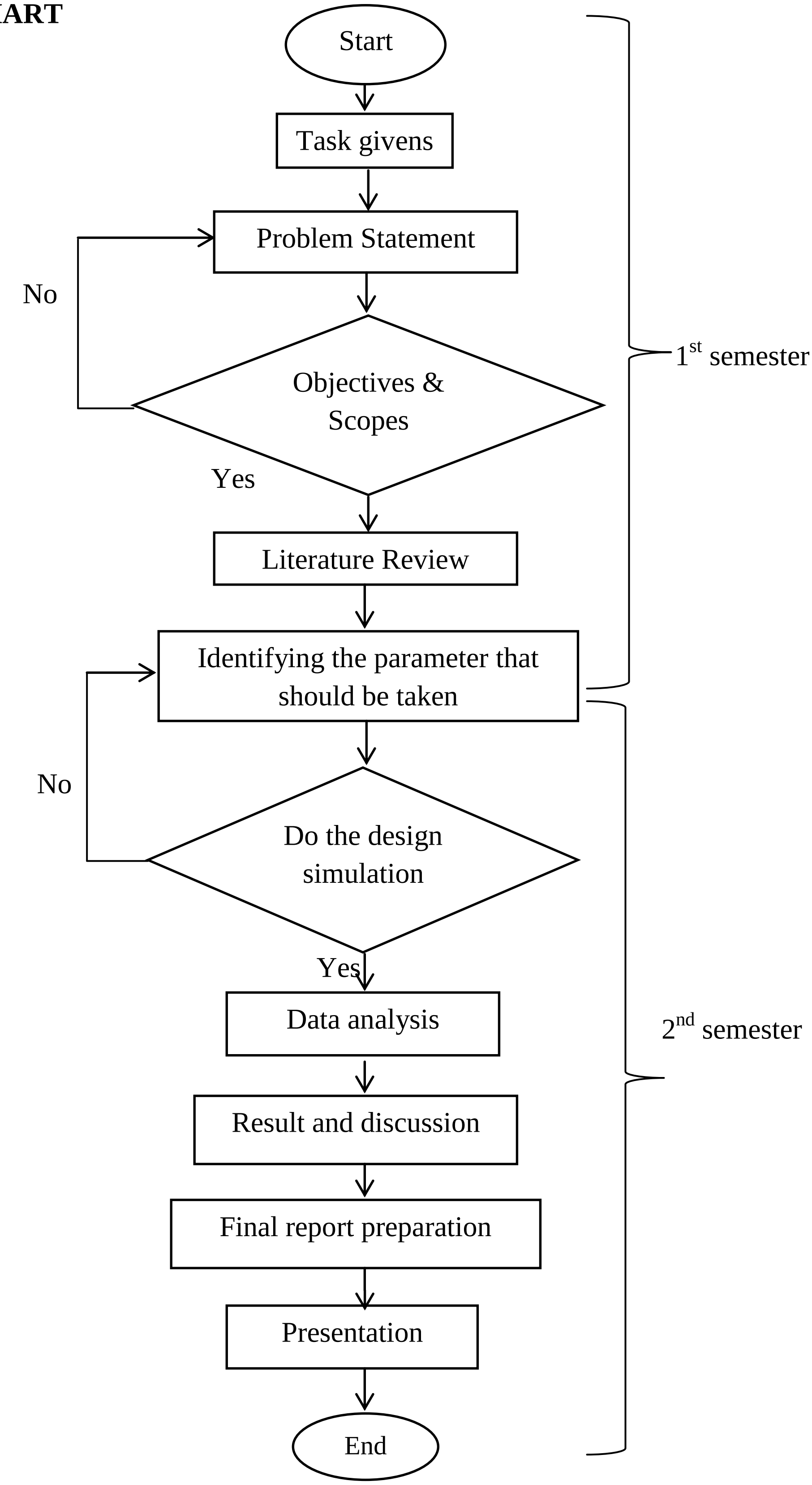

Figure 3.1: Flow Chart of Project

3.3 COMPUTATIONAL FLUID DYNAMIC SOFTWARE

The Computational Fluid Dynamics (CFD) Module is the premier tool in the Comsol product suite for sophisticated fluid flow simulations. Compressible as well as incompressible flows can be combined with advanced turbulence models and forced and natural convection. An important characteristic of the CFD Module is its capability of precise multi physics-flow simulations such as a conjugate heat transfer with non-isothermal flow, fluid-structure interactions, non-Newtonian flow with viscous heating, and fluids with concentration-dependent viscosity. Porous-media flow user interfaces allow for isotropic or anisotropic media, as well as automatically combined free flow and porous domains. Tools for modeling of stirring vessels with rotating parts are available for both 2D and 3D flows.

The Module’s interfaces for homogeneous two-phase flow include a mixture model for fine particle suspensions and a bubbly flow model for macroscopic gas bubble flow. For interface tracking two-phase flow, formulations are provided using the level-set and phase-field methods.

3.3.1 THE PARAMETERS USED IN SIMULATION

The parameters that will be examined by using this CFD software are differential size of the inlet diameter of the orifice nozzle and also the injection pressure at the orifice nozzle. These two parameters will be the manipulated variable and the final result from the simulation will determine the flow of liquid or gas and also the spray characteristics. Since this software can determine the various types of flow, then give the result of my problem statement.

From the final result of the simulation, we will determine the graph that will show details about the simulation. From the graph itself, it can interpret the result of flow and spray characteristics. Then we will compare which of the size of inlet diameter and injection pressure will give better performances for a flow and spray characteristics.

3.3.2 THE PROCESS OF SIMULATION AND MANUAL CALCULATION

First of all, the software that been used in this project is Solidworks 2012. By using this software, the objectives of this project which is to determine the flow and spray characteristics at different inlet diameter of orifice nozzle and different injection pressure can be achieved.

In this project, it is focused on the plain orifice nozzle only. The design of plain orifice nozzle is drawn with three different inlet diameters that have been chose which are 100, 200 and 250 µm. In order to have a spray simulation done, an extra chamber need to be drawn. After the chamber is drawn, it is needed to be assembled together with the plain orifice nozzle that drawn earlier.

Then, the full set of plain orifice nozzle with a chamber can be used to do the simulation. For this project, it is investigating the internal flow of the nozzle. When the set up is carried out, it is ensuring that it has to be the simulation for internal flow. Before proceeding with the set up, one more action should be taken. The full set of plain orifice nozzle need to have a lid, so that the internal flow that supposedly to determine can be determine.

Next, for this project, the liquid used is water. The density of water itself is different with the others. So, the final result will a bit different when using the exact oil for engines. The parameter that needs to be investigated can be adjusted during the set up of simulation. The other parameters that not related with this project will be fixed. The surface boundary needs to be selected too. It is to determine the location of simulation that needs to carry out. The goals at the end of the simulations that related with this project also can be set up.

When the simulation run is done, the result can be observed and analyze. The types of results also can be chose either in cut plots type, surface plots type, flow trajectory type and many more. The goals that set up earlier were easily obtained from the Microsoft Excel. The animation of the flow also can be saved in the correct format.

For the calculation of spray dispersion, it needs to be manually calculated. This means, the flow trajectory obtained earlier, the spray dispersion can be calculated from there. While the flow trajectory is shown, the side view of the whole plain orifice nozzle and a chamber need to be viewed. For more easily, a picture is taken from print screen. The most important is, the size of picture taken must be the same. Then the entire nine pictures taken were printed out. Then, manually calculate the spray dispersion which is taking the furthest point from one side to the other side. After got the result, it needs to be converted in order to synchronize with the design drawn earlier in Solidworks. The magnification factors should used in order to solve this problem.