Numerical study of the in-cylinder air flow characteristic generated by GVD for air-fuel mixing improvement using emulsified biofuel

By:

MUHAMMAD ARIFF SOLEHIN BIN RAZAK

(Matrix no.: 125029)

Supervisor:

Dr. Mohammad Yusof Idroas

May 2018

This dissertation is submitted to

Universiti Sains Malaysia

As partial fulfillment of the requirement to graduate with honors degree in

BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

UNIVERSITI SAINS MALAYSIA

School of Mechanical Engineering

Engineering Campus

Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed .... (Candidate)

Date ....

STATEMENT 1

This thesis is the result of my own investigations, except where otherwise stated. Other sources are acknowledged by giving explicit references.

Bibliography/references are appended.

Signed .... (Candidate)

Date ....

STATEMENT 2

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed .... (Candidate)

Date ....

ACKNOWLEDGEMENT

Bismillahirrahmanirrahim

I would like to express my gratitude towards my supervisor, Dr. Mohamad Yusof Idroas for his guidance and support in my final year project. His knowledge and moral support have been an encouragement for me throughout this work.

I am very much thankful to all my colleagues for their kindness and great cooperation during this study. They willing to help and guide me when setting up the simulation besides sharing valuable information and knowledge.

Next, I would like to thank all the School of Mechanical Engineering staff for their contribution in my final year project. Their ideas, cooperation and skills really helps in completing this study. These contributions are very much appreciated.

Finally, I would like to express my special thanks for those who are involve directly or indirectly in completing this project. I appreciate their involvement in this project that had helped a lot as I can finish my final year project successfully. Thank you very much.

Muhammad Ariff Solehin Bin Razak

Mei 2018

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

ABSTRAK

Selepas beberapa kajian telah dilakukan untuk mencari bahan api alternatif untuk enjin diesel, biofuel yang diemulsi telah menjadi salah satu calon yang paling berpotensi yang boleh digunakan dalam enjin diesel konvensional. Walau bagaimanapun, kelikatan yang lebih tinggi dan pengabusan yang lebih rendah dapat menyumbang kepada beberapa masalah enjin seperti, prestasi pencampuran bahan api udara yang lebih rendah, sudut konkrit panjang yang lebih kecil dan konkrit yang rendah. Atas sebab itu, kajian ini menyiasat kesan Panduan Reka Bentuk Vane (GVD) yang dipasang di hadapan manifold pengambilan enjin CI yang dijalankan dengan biofuel yang diemulsi untuk meningkatkan aliran udara dalam silinder untuk mempromosikan proses penyejatan, penyebaran dan pembakaran akhirnya mengurangkan masalah tersebut. Untuk berbuat demikian, satu model asas dinamik cecair pengkomputeran 3D bagi simulasi enjin pembakaran dalaman telah dibangunkan, disahkan dan kemudian simulasi dibawa dengan 3 GVD yang berbeza dari segi variasi ketinggian 0.2R, 0.4R, dan 0.6R. Hasil tenaga kinetik bergolak, kontur halaju, dan kekuatan berputar dibandingkan dengan menentukan ketinggian optimum yang optimum. Dari hasil simulasi, kajian ini mendapati bahawa ketinggian 0.2R vane adalah ketinggian vane optimum dengan sudut 35 °, empat bilah yang disusun secara serentak antara satu sama lain dan panjang vane 3R mm. Lain-lain ketinggian yang berbeza vanes juga menunjukkan peningkatan, tetapi ketinggian 0.2R menunjukkan peningkatan tertinggi. Ini mungkin kerana corak aliran udara dalam bentuk kepala mangkuk dalam piston telah dikuatkan dengan corak aliran udara yang dihasilkan oleh panduan vane 0.2R vane ketinggian. Peningkatan ciri bahan api udara dengan penerapan GVD dijangka menyumbang kepada pencampuran bahan api udara yang lebih baik, pengabusan bahan api dan kecekapan pembakaran enjin menggunakan biofuel yang digemulsikan sebagai bahan api alternatif.

ABSTRACT

After several researches have been done on finding the alternative fuels for diesel engine, emulsified biofuel has become one of the most potential candidate that can be applied in conventional diesel engine. However, its higher viscosity and lower atomization can contribute to several engine problems such as, lower air-fuel mixing performance, higher penetration length smaller cone angle and lower combustion efficiency. For that reason, this research investigated the effect of Guide Vane Design (GVD) installed in front of the intake manifold of a CI engine run with emulsified biofuel to enhance the in-cylinder airflow to promote the evaporation, diffusion, and combustion processes to eventually reduce those problems. In order to do so, a base model of 3D computational fluid dynamic of internal combustion engine simulation was developed, verified and then simulations were carried with 3 dissimilar GVD in terms of height variation of 0.2R, 0.4R, and 0.6R. The results of turbulent kinetic energy, velocity contour, and swirling strength were compared to determine the optimum vane height. From the simulation results, this research found that the 0.2R vane height was the optimum vane height with 35° twist angle, four vanes being arranged perpendicularly to each other and 3R mm vane length. Other different heights of vanes also showed improvement, but 0.2R height showed the highest number of improvements. This could be due to the airflow pattern in bowl-in- piston head shape was amplified by the airflow pattern produced by the guide vane of 0.2R vane height. The improvement of the air fuel characteristic with the application of GVD was expected to contribute to a better air fuel mixing, fuel atomization and combustion efficiency of the engine using emulsified biofuel as an alternative fuel.

CHAPTER 1

INTRODUCTION

1.1 Overview

Nowadays, diesel engine has become one of the main power units especially in utility vehicle sector due to its high thermal efficiency, high reliability, high fuel economy and require less maintenance. The first compression ignition (CI) engine using fuel made from peanut oil have been invented by Dr. Rudolf Diesel in 1990 [1]. But at that time people have more focus on fossil fuel because it have better properties than biodiesel and was abundant. This scenario have cause the development of the CI engine that use biodiesel was slowed down. However, many researchers have been motivated to find and explore alternative fuels to replace depleting petroleum-based fuels and what was proposed by 'the father of Diesel engine' earlier have earned their attention again. The major factor contributing to this research is the unsettling fact that the world is currently experiencing an oil crisis with fuel prices dramatically rising [2]. This problem became critical and clearer when the World Energy Forum have made prediction that the existing fuel supplies would only just survive for the next 10 decades [3]. In the early 1970s, the world have faced the critical oil crisis. In addition, the internal combustion engines (ICEs) have become one of the main cause of environmental pollution because it emits pollutants such as hydrocarbon (HC), Nitrogen Oxide (NOx), Sulphur Oxide (SOx), Carbon Monoxide (CO) and particulate matter (PM) especially in cities with high population densities. As we know, CO can cause ozone depletion while NOx is a toxic gas that are harmful to human body. Thus, it is absolutely necessary to rectify these global problems or at least reduce the dependency on petroleum-based fuels due its detrimental effects.

Therefore, there are two requirements that need to be fulfil in order to find the alternative fuels are that it must be renewable and less polluting. There are several types of biofuels that have these two properties such as crude palm oil (CPO), methanol, ethanol, neat vegetable oil and transesterificaion of vegetable oil to biodiesel. They can be readily available, easy to handle and store, a green energy source and totally renewable[4]. Moreover, it can be used directly in existing CI engines with minor or no modification. However, usually these biofuels need to blend at various proportions with diesel to run in diesel engines due to higher viscosity and lower calorific value [5]. These properties of biofuel due to its large molecular size and weight [6]. As a result, the current trends for research in this area, specifically for the usage, testing and improvement of neat vegetable oil and biodiesel are dramatically increasing.

With the objective to reduce the gap between the biofuel and fossil fuel, researchers today have introduced many techniques in an attempt to replace diesel such as preheat the CPO, alter the chemical composition of biofuel via a transesterification process, emulsify the biofuel (CPO mixed with a certain composition of water) or even modifying the combustion chamber geometry [4]. CPO can be made into a biodiesel via trans- esterification process of triglycerides with methanol. Trans- esterification process have a lot of attention because it is a simple process and is less expensive to produce [4]. In term of viscosity, for CPO to more closely match diesel baseline, it only needs to preheat to 60 ˚C [7]. Consequently, combustion will have high efficiency resulting in less carbon deposit. Previous research shows that biodiesel spray gives a longer injection delay and a smaller cone angle; thus it will result a larger mean diameter due to high viscosity and surface tension [8]. However, the exhaust emissions showed increments of CO and NO emissions of 9.2% and 29.3%, respectively over the entire load range. Another research from Bari et al. [9] proved that by advancing 4°CA injection timing, the engine efficiency increased by approximately 1.6% and CO emission reduced by approximately 9.9% for direct injection diesel engine running on waste cooking oil. The engine however, incurred a considerable production of NO emissions with a 76% increment.

In this paper, to rectify the problems appeared and improving engine in-cylinder air flow characteristic, a Guide Vane Design (GVD) will be installed at the intake runner. A GVD is a set of vanes characterized by three main parameters, vane height, vane angle and vane number. However, this technique is not new and has been widely used for petrol engines [3]. For a CI engine, the baffle type generator is generally used [3]. GVD have more advantageous when compared with baffle type generator because it is simple and the resistance to the air flow can be reduced [3]. A computational fluid dynamic (CFD) solver is utilized to predict the best design of GVD that will be improved the air flow characteristics in term of air fuel mixing, fuel atomization and combustion performance.

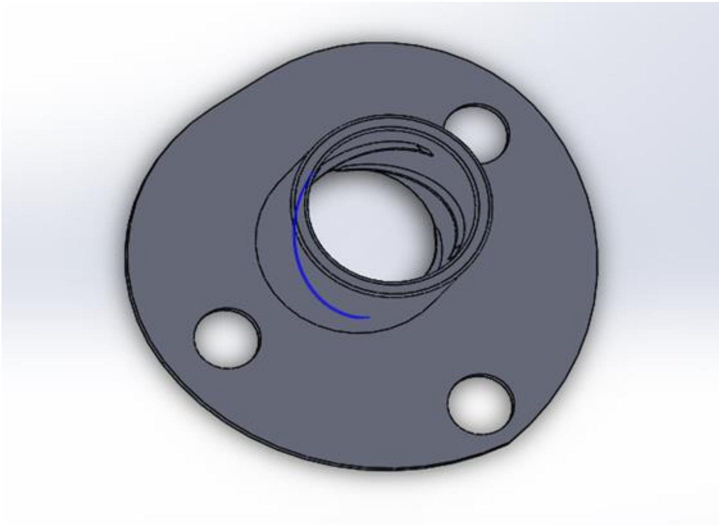

Figure 1: Sample Design of GVD

Figure 2: Yanmar L70 V with GVD installed

Several aims of the project have been established to study the air flow characteristic when using GVD and with the improvement of Turbulence kinetic Energy(TKE) and how to mitigate the issues of emulsified biofuel when it is used in Diesel engine particularly on its high viscosity and volatility. The studies are conducted through computational simulation by using Computational Fluid Dynamics (CFD) approach. In future, the findings of this project can be used as a guideline to researchers and engineers in practically improving the engine performance with emulsified biofuel as a source of fuel.

1.2 Problem Statement

In recent years, researches have more attentions about how to explore the potential application of emulsified biofuel. Generally, emulsified biofuel is formed by adding specific amount of water directly in diesel fuel. Specifically, it is a mixture of palm oil, surfactant and water. The main function of surfactant is to reduce the surface tension between oil and water and then makes them mix together. Water addition in emulsification process elongates the combustible range to higher fuel equivalence ratio side by reduction of PM emission [10]. The water addition also reduces NOx emission drastically[10]. However, when the CPO have been applied directly in conventional diesel engine, there are several issues will come up.

Since emulsified biofuel is widely available and can be mass produced, it has been identified as a best candidate to replace the diesel. However, at the same time, this fuel also has disadvantages and the most notable are higher viscosity and heavier molecule. Due to these two properties, emulsified biofuel will have higher penetration length with lower cone angle then cause the engine performance reduced.

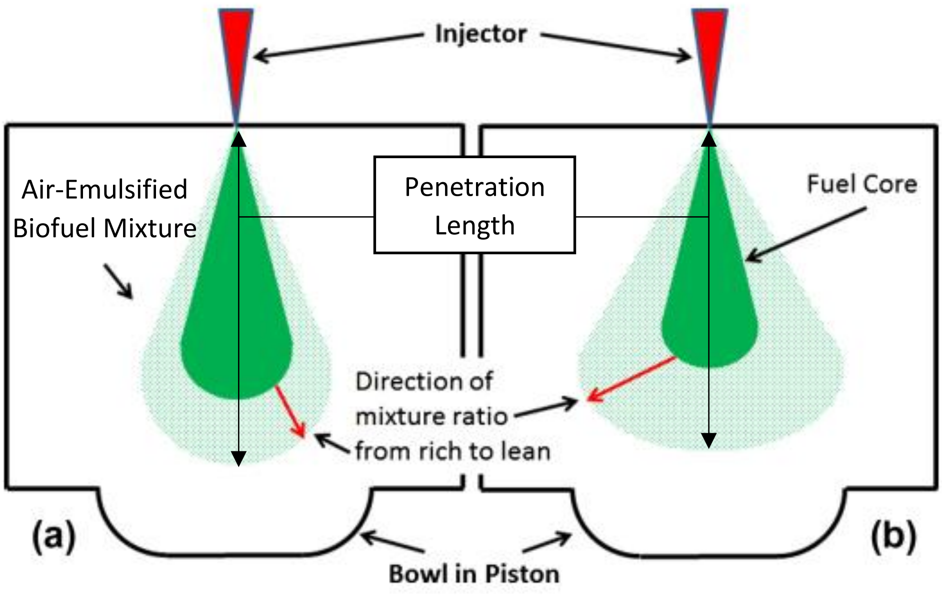

Figure 3: Comparison of penetration length and cone angle: (a)Emulsified biofuel (b) Conventional Diesel [11]

The air-fuel mixing performance can be analysed in terms of swirl ratio, tumble ratio, cross tumble ratio, and Thermal Kinetic Energy (TKE). Swirl is defined as rotational flow about the vertical axis of the cylinder , and tumble is defined as rotational flow about an axis perpendicular to the vertical axis of the cylinder [6]. Cross tumble is defined as rotational flow about an axis perpendicular to both the swirl and tumble axes [6]. Swirl, tumble, and cross tumble can be incorporated into the in-cylinder air flow by installing vanes or shrouds in the air flow prior to the inlet valve. However, when we used emulsified biofuel in Diesel Engine, all of these parameters have been reduced dramatically. These problems arise from the physical and chemical properties of biofuel compared to conventional diesel fuel such as higher viscosity and boiling points. Thus, biofuel operated engines provide lower combustion efficiency, and this can lower the engine performance because the fuel molecules are heavier and this reduces the ability of the fuel to move and mix with the air.

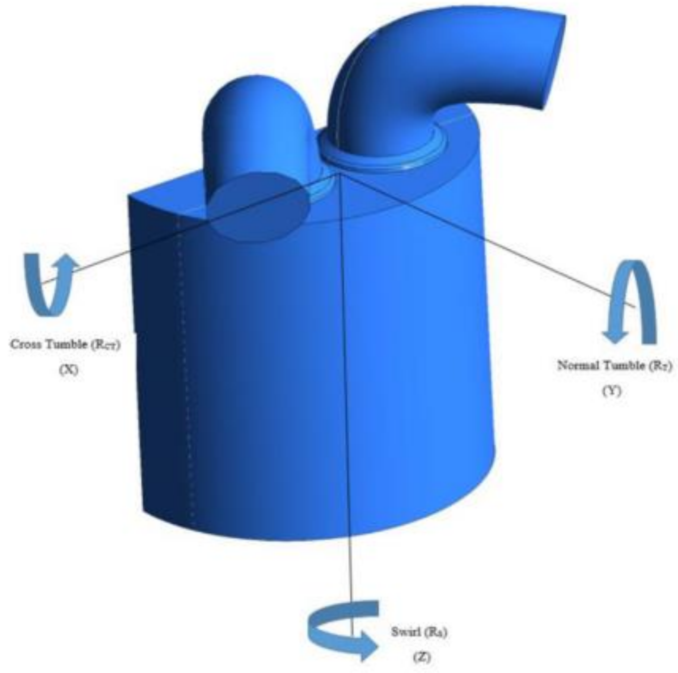

Figure 4: Definition of swirl and tumble axes and direction [4]

1.3 Objectives

The objectives of the project are listed below:

1) To study the in-cylinder air flow characteristic inside the engine using emulsified biofuel which is generated by Guide Vane Design (GVD) for air-fuel mixing improvement using ANSYS simulation.

2) To study the effects of Guide Vane Design (GVD) with varies the height of vane range between 20% to 60% of radius of intake runner with optimum number of vane which is 4 and angle of vane which is 35°.

1.4 Scope of the project

The project is to find what is optimum height of the GVD that will give best performance of the Diesel Engine using emulsified biofuel. Therefore, the project will cover the scopes as follow:

1) Measuring the parts of the Diesel engine that consist of 6 main components: intake runner, exhaust runner, intake valve, exhaust valve, bowl in SCC piston and clearance volume.

2) Designing the geometry of the diesel engine based on Yanmar L70V, engine speed in 2000 revolution per minutes (rpm) using SOLIDWORK 2017.

3) Designing the GVD with varies of height of vane range between 20% to 60% of radius of intake runner with optimum number of vane which is 4 and angle of vane which is 35°.

4) Simulating the performance of the diesel engine using emulsified biofuel with GVD installed at different height of vane by using ANSYS 16.1.

5) Determine the best design and optimum height of vane of GVD based on the diesel engine performance.

1.5 Outline of the reports

This thesis is divided into five main chapters. The first chapter discusses on history using crude palm oil in diesel engine, overview of emulsified biofuel as alternative fuel and the application of GVD as a problem solver to enhance the engine performance. Problem statement regarding the bad properties of emulsified biofuel such as low atomization and high viscosity, and the air-fuel mixing performance in terms of swirl ratio, tumble ratio, cross tumble ratio, and Thermal Kinetic Energy (TKE) are also reviewed in this chapter. This chapter also contains the project scopes, project objectives, and outline of the project report.

In chapter 2, the literature review based on the other parameters of GVD such as vane angle and number of vane will be presented.

In chapter 3, the methodology in terms of developing the model using SOLIDWORKS 17 and setting up the simulation by using ANSYS 16.1 will be presented.

In chapter 4, the results in term of swirl ratio, tumble and cross tumble ratio will be discussed and verified here.

In last chapter, the conclusion of the project will be made in chapter 5. Some suggestions and recommendations are given for improvement in future research.

CHAPTER 2

LITERATURE REVIEW

2.1 Optimum number of Vane

Presently, many researches about the effect of guide vane design (GVD) to the engine performance have been done. However, many researchers they only focus on study about the effect of GVD to the diesel engine that use biodiesel. S. Bari Idris Saad [12] demonstrated that the result of air flow characteristic when they use various design of their devise which is Guide Vane Swirl and Tumble Device(GVSTD) in terms of turbulence kinetic energy (TKE), in-cylinder pressure, and velocity. In their studies, they have developed 10 guide vane models with various numbers of vanes and investigated the optimum number of vane based on the results in terms of in-cylinder turbulence kinetic energy (TKE) and velocity. Based on figure 5, the highest average in-cylinder TKE was generated by the 4-vane model, followed by the 12-vane model, 11-vane model, 8-vane model and the other models. While in figure 6, in term of velocity, the 6-vane model had the highest in-cylinder average velocity with approximately 6.16 m/s, followed by the 5-vane model with approximately 6 m/s and the third highest average in-cylinder velocity belonged to the 4-vane model with 5.97 m/s as compared to the base model of 4.78 m/s at 341°CA. Finally, this research found that the use of guide vanes helped to improve in-cylinder average TKE and velocity and that four (4) vanes was the optimized number of guide vanes for that purpose even in result of velocity it is not been on top placed.

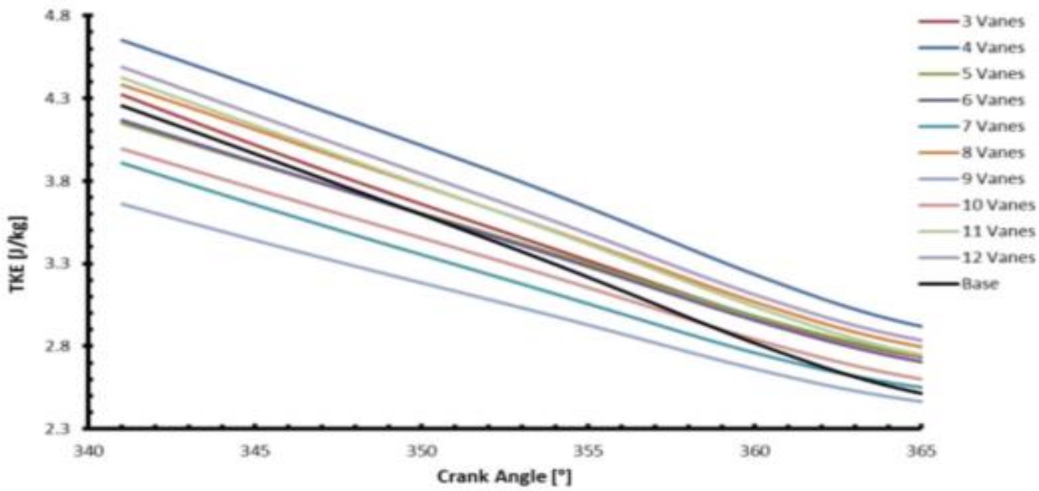

Figure 5: In-cylinder average value of TKE profile: simulation base model and guide vane models.

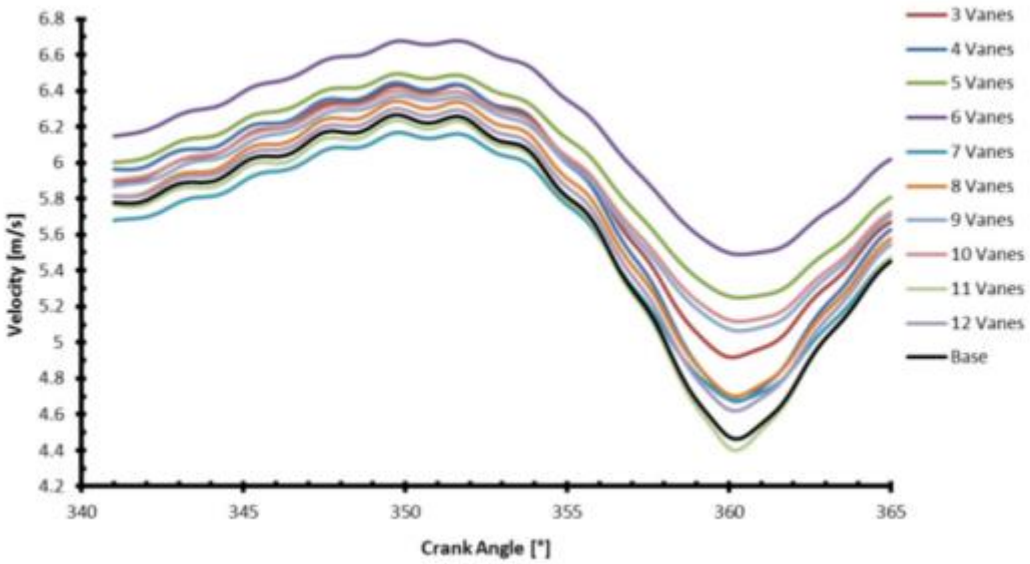

Figure 6:In-cylinder average value of velocity profile: simulation base model and guide vane models.

2.2 Optimum Vane Angle of GVD

Other than previous researches mentioned above, the research about the effect of varying vane angles to the performance and emissions of a compression ignition (CI) engine run with biodiesel also has been done. S. Bari Idris Saad [13], they have fabricated five guide vane models with the vane angles varied between 25° and 45°, installed and tested in generator CI engine run constantly at 1500 rpm [13]. They have demonstrated and discussed the performance of engine in terms of brake-specific fuel consumption (BSFC), engine efficiency, torque, air/fuel ratio and exhaust temperature as well as carbon dioxide (CO_2), oxygen (O_2), nitrogen oxide (NO_x), carbon monoxide (CO) and hydrocarbon (HC) emissions. From the results, the 35° vane angle was found to be the optimum vane angle because it has maximum improvement in BSFC and engine performance [13]. For the other elements, even not placed in first place but the 35° vane angle still among the top positions [13].

2.3 Summary

When we are comparing and go through two researches above and other researches related to this field, it was found that optimum GVD vane angle is 35° while optimum number of vane is 4. It also was found that not many of them was trying to research the effect of GVD on the compression ignition (CI) engine running with emulsified biofuel. So, in this project the in-cylinder air flow characteristic generated by GVD for air-fuel mixing improvement using emulsified biofuel will be investigated.

CHAPTER 3

METHODOLOGY

3.1 Overview

In order to study the effect of guide vanes in modifying the intake airflow up to the injection period, this research prepared the IC engine cold flow simulation. The guide vanes were then constructed and imposed in front of the intake runner. The results of both in-cylinder turbulence kinetic energy (TKE), swirl ratio, tumble ratio, and cross tumble ratio were then compared to study the effect of the guide vanes in modifying both airflow characteristics. Based on various sources, the design of the guide vanes depends on four parameters: number of vanes, vane height, vane length and vane angle which would determine the optimized design of the guide vanes [12] to be used on the CI engine run on emulsified biofuel. However, this paper limits the optimization to the height of vane only. Details of the simulation model and the optimization of height of vanes are presented in the following section.

3.2 Construction of computer modelling

In general, this research has two types of simulation models: a base model and a guide vane model. Both models were prepared in four main steps: drawing the model, meshing the elements, setting up the boundary conditions and computing the airflow characteristics. This research utilized the computer- aided drawing (CAD) program, SolidWorks 17.0, to draw both the base and guide vane models. The rest of the simulation steps were completed using commercial computer fluid dynamics (CFD) software, ANSYS, and, in particular, ANSYS-ICE, which was used to compute the simulation. The details of the base and guide vane models are described in the next section.

3.2.1 Base Model (Yanmar L70V Diesel Engine)

The geometry of diesel engine was designed based on the Yanmar L70V running at speed in 2000 revolution per minutes (rpm) and their specification can be referred in Table 1. The diesel engine has been modelled using SOLIDWORK 2017, which considers the main six components of the model; intake runner, intake port, intake valve, cylinder, exhaust valve and exhaust port as can be seen in figures (7-10). The components were assembled together to become one solid part before exporting to ANSYS- Design Modeller 16.1.

Table 1: Specification of The Engine

<table>
<tr><th colspan="3">ENGINE MODEL</th><th>YANMAR L70V</th></tr>
<tr><td colspan="3">Type</td><td>Vertical cylinder, 4-cycle, air cooled diesel engine</td></tr>
<tr><td>Engine Speed</td><td colspan="2">Revolution Per Minute (rpm)</td><td>2000</td></tr>
<tr><td>Bore</td><td colspan="2">mm</td><td>78</td></tr>
<tr><td>Stroke</td><td colspan="2">mm</td><td>67</td></tr>
<tr><td>Displacement</td><td colspan="2">litre</td><td>0.320</td></tr>
<tr><td colspan="3">Compression Ratio</td><td>19.1</td></tr>
<tr><td colspan="3">Cooling System</td><td>Forced Air by Flywheel Fan</td></tr>
<tr><td colspan="3">Lubricating System</td><td>Forced Lubrication with Trochoid Pump</td></tr>
<tr><td colspan="3">Starting System</td><td>Electric Start / Recoil Start</td></tr>
<tr><td></td><td>Overall Length (L)</td><td>mm</td><td>378</td></tr>
<tr><td>Dimension</td><td>Overall Width (W)</td><td>mm</td><td>422</td></tr>
<tr><td></td><td>Overall Height (H)</td><td>mm</td><td>453</td></tr>
<tr><td>Lubricating Oil</td><td>Dipstic Upper Limit</td><td>litre</td><td>1.10</td></tr>
<tr><td></td><td>Dipstic Lower Limit</td><td>litre</td><td>0.70</td></tr>
<tr><td colspan="2">Fuel Oil Tank Capacity</td><td>litre</td><td>3.3</td></tr>
</table>

Figure 7: Intake and Exhaust Valves

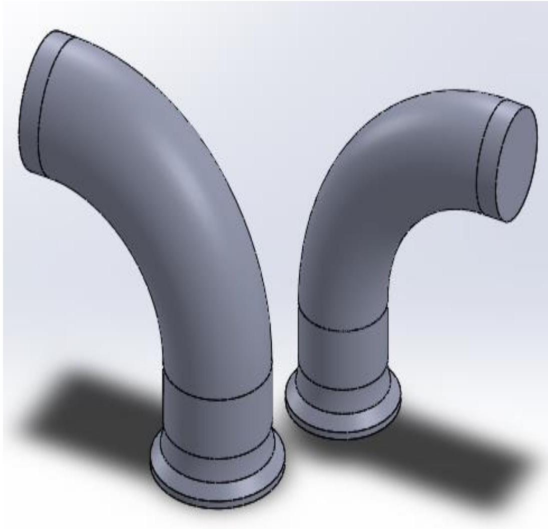

Figure 8:Intake and Exhaust Ports

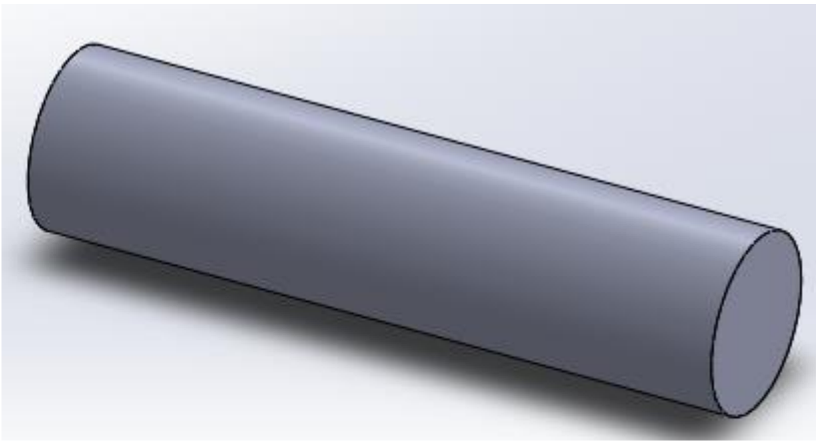

Figure 9: Intake Runner

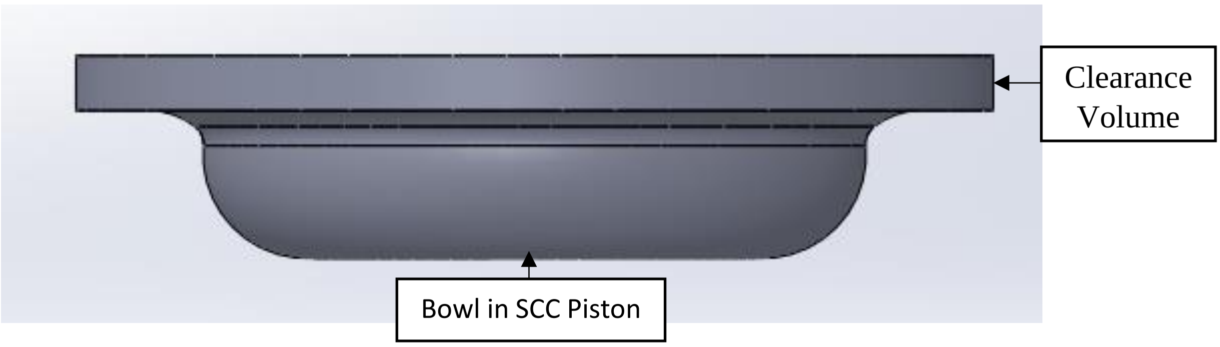

Figure 10: Bowl in SCC piston and Clearance Volume

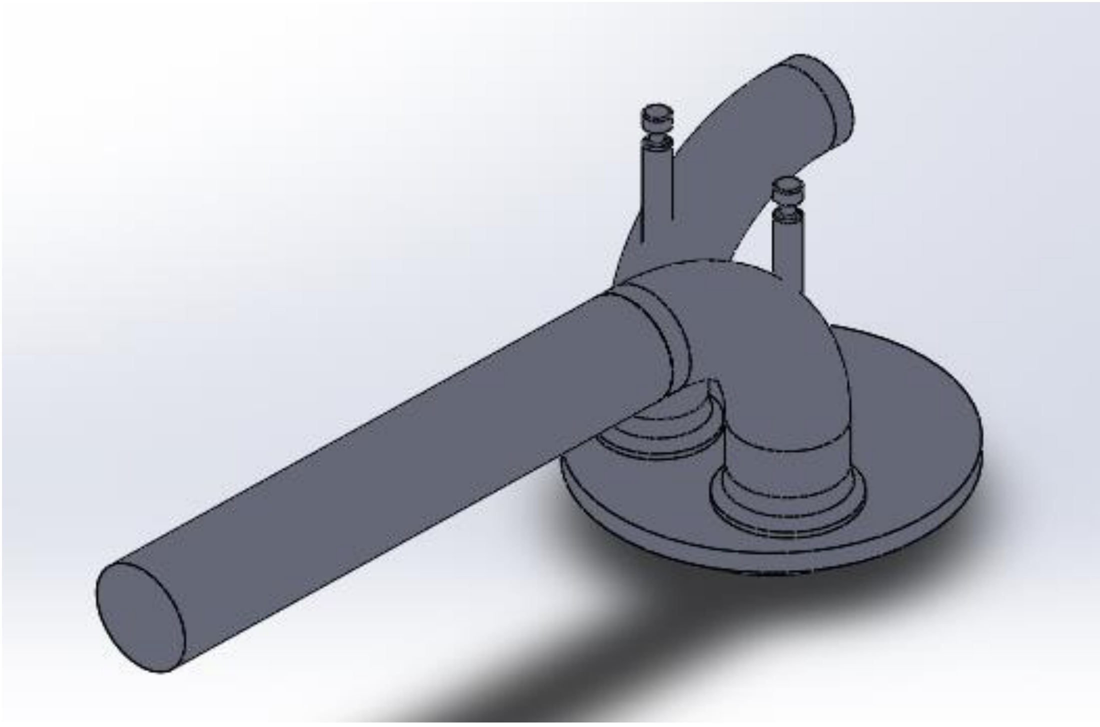

Figure 11: (Base Model)Yanmar L70 V parts in assembled condition

3.2.2 Design of the GVD

As mentioned above, this research sought to optimize the vane height of the guide vane using the parametric optimization method. This meant that the vane height would be varied with each height tested one by one on the base model [14]. According to various sources, the vane height could vary from the very minimum height up to the whole diameter of the intake runner [5]. The minimum height of the vane would produce minimum effect on the rotational flow while the maximum height would produce maximum effect on the rotational flow. However, inserting the guide vane inside the intake manifold would also create an obstacle (more resistance) to the inlet airflow. Hence, the height of the guide vane needs to compromise between generating more turbulence and the extent to which it becomes an obstacle to the inlet air flow. Due to this condition, this research decided to determine the optimum vane height values that will start from 20% of intake radius up to 60% of intake radius. Hence, 3 vane height models were prepared based on the radius of the intake runner (R). For identification purposes, the vane height was multiplied by R and named as, for example, 0.20R, that is, equal to 0.2 times R as shown in Table 2.

While seeking to optimize the height of the vanes, the other parameters (vane number, angle and length) were set respectively as: four vanes, 35° twist angle (TA) and three times R. These values were selected based on various sources in the literature regarding the development of guide vanes to improve in-cylinder airflow characteristics[14], [15]. Moreover, these values were set constant for all 3 guide vanes in order to optimize the height of the guide vanes only. Table 2 summarizes the specifications of the guide vane and figures (12-14) illustrates the sample design of the guide vane and its assembly on the base model.

Table 2: Specification of the guide vane design (GVD).

Parameter	Value
Number of Vane	4
Length of Vane (*l*)	3×intake radius (R=12.5mm) = 37.5mm
Width of Vane	0.5mm
Height of Vane (*Hv*)	0.2R 0.4R 0.6R
Vane twist angle (θ)	35°
Angle of incident	90°

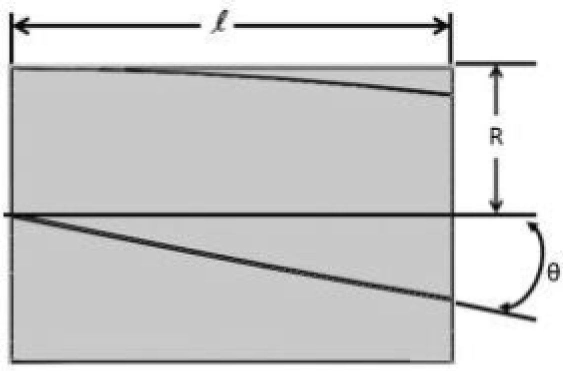

Figure 12: Side View of GVD

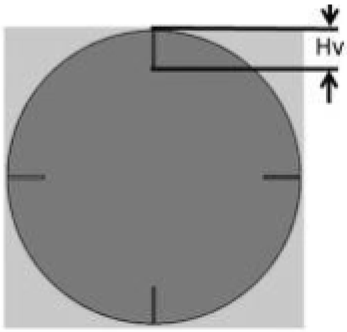

Figure 13: Front View of GVD

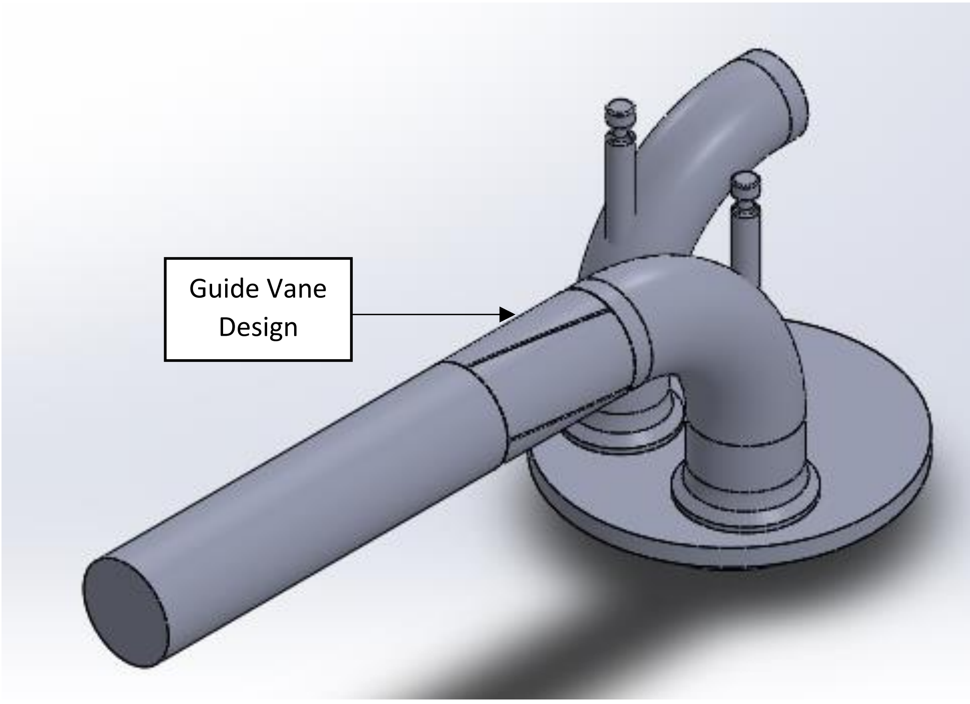

Figure 14: Yanmar L70 V with GVD installed at intake manifold

3.3 Computational Simulation Procedure

3.3.1 Setting the properties and performing decomposition

After the modelling process using SOLIDWORK 2017 have been finished, diesel engine geometry that consist of two types: base model (no GVD) and the other one with GVD installed at intake manifold have been imported to ANSYS Workbench 16.1 (ICE) for meshing, boundary condition setup and simulation process.

In ICE, there are three types of simulation: cold flow, port flow and combustion. In this project, the one that has been applied is cold flow simulation. Cold flow analysis involves modelling the airflow and possibly the fuel injection in the transient engine cycle without reactions. The goal is to capture the mixture formation process by accurately accounting for the interaction of moving geometry with the fluid dynamics of the induction process [16].

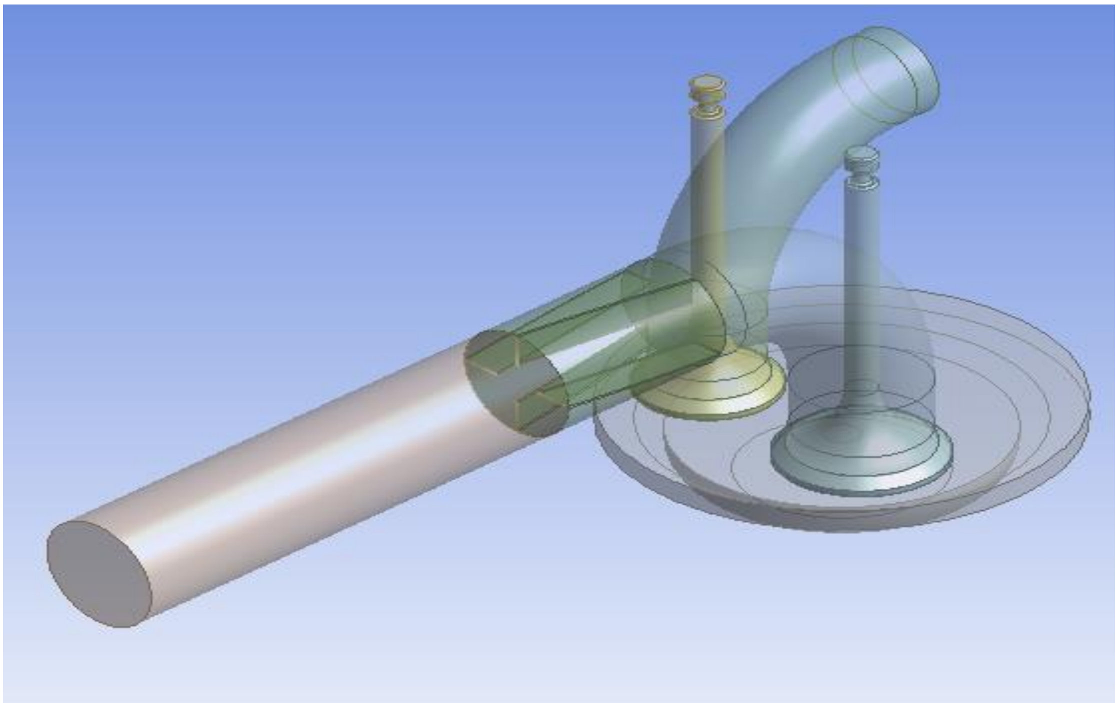

Figure 15:Diesel Engine geometry that have been imported in Design Modeller

In design modeller, intake runner, GVD, intake port, bowl and exhaust port have been defined as a one body by boolean operation. At last, there are only 3 bodies that have been involved: boolean body, intake and exhaust valves. For boolean body, it has defined as a fluid domain while intake and exhaust valves have been defined as solid domain.

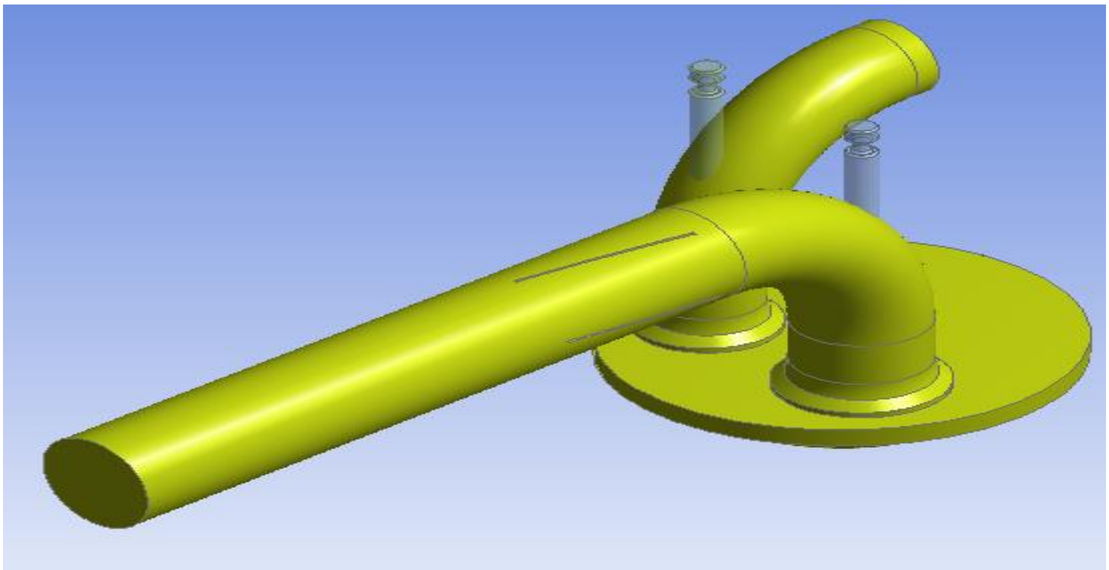

Figure 16: Boolean body (Yellow)

Details View

Details of Body	
Body	Boolean
Volume	...
Surface Area	...
Faces	43
Edges	72
Vertices	36
Fluid/Solid	Fluid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

Figure 17: Details of Boolean Body

Details View

Details of Body	
Body	Intake Valve
Volume	5794.9 mm^3
Surface Area	3077.2 mm^2
Faces	15
Edges	14
Vertices	0
Fluid/Solid	Solid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

Figure 18: Details of Intake Valve Body

Details View	
Details of Body	
Body	Exhaust Valve
Volume	4705.5 mm^3
Surface Area	2649.6 mm^2
Faces	15
Edges	14
Vertices	0
Fluid/Solid	Solid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

Figure 19: Details of Exhaust Valve Body

Next, in Input Manager, the inlet, outlet, cylinder, valve bodies and valve seats have been defined. Other information like compression ratio also have been inserted.

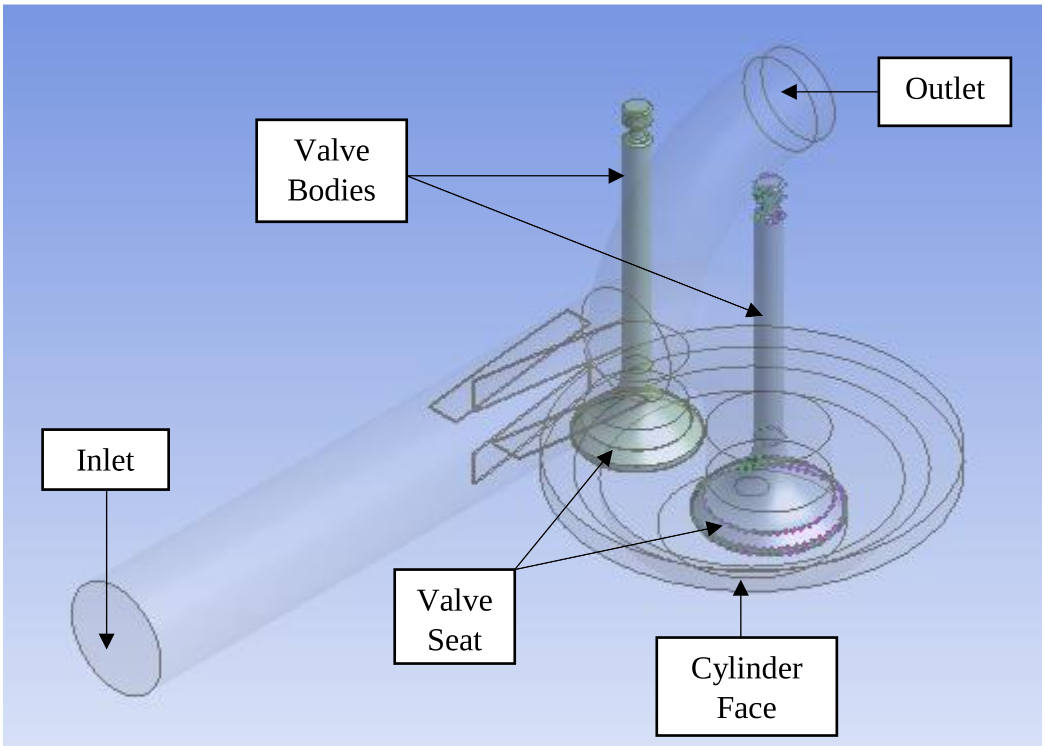

Figure 20: Diesel Engine model with labelled parts

Inlet Faces	1 Face
Outlet Faces	1 Face
Cylinder Faces	1 Face
Symmetry Face Option	No
Topology Option	Full Topology
Crevice Option	No
Validate Compression Ratio	Yes
Compression Ratio	19
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	invalve1
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	exvalve1
IC Animation Inputs (RMB)	

Figure 21: Details of Input Manager

Then the geometry has to undergoes the decomposition process to divide them into different zones.

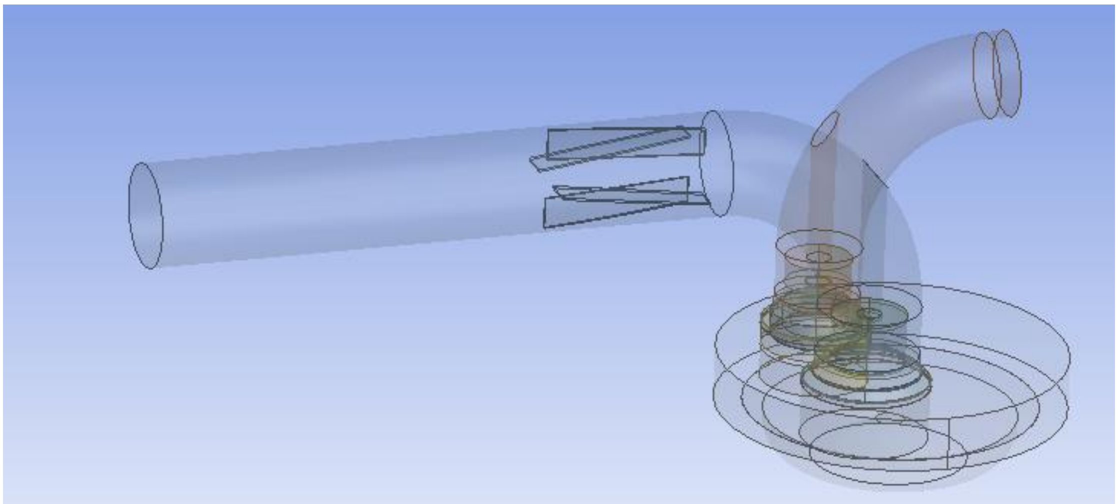

Figure 22: The geometry after decomposition process

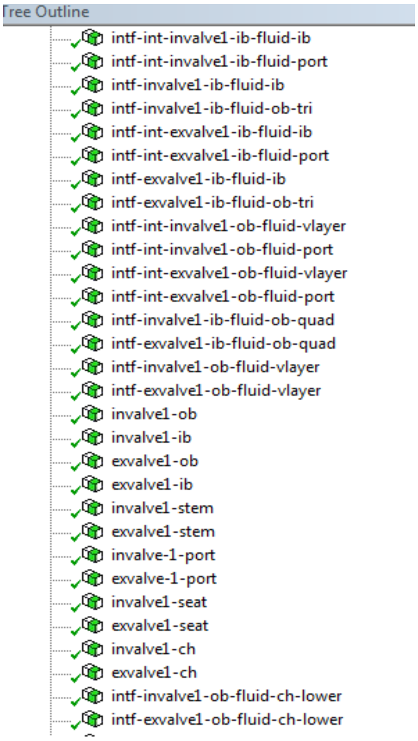

Figure 23: Zones generated after decomposition

3.3.2 Meshing

All parts of the geometry are meshed and the type of the mesh that have been used is coarse type.

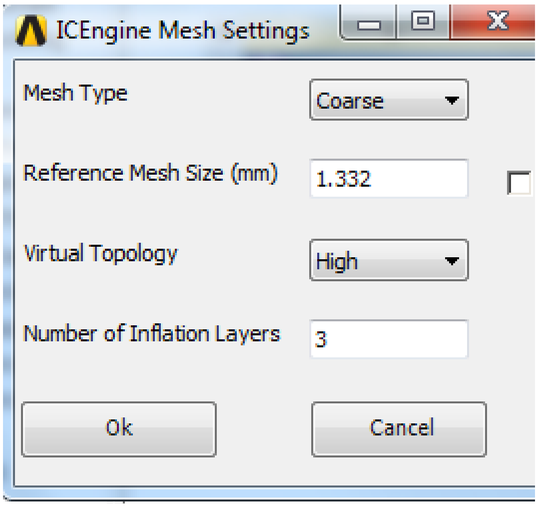

Figure 24: IC Engine Mesh Settings

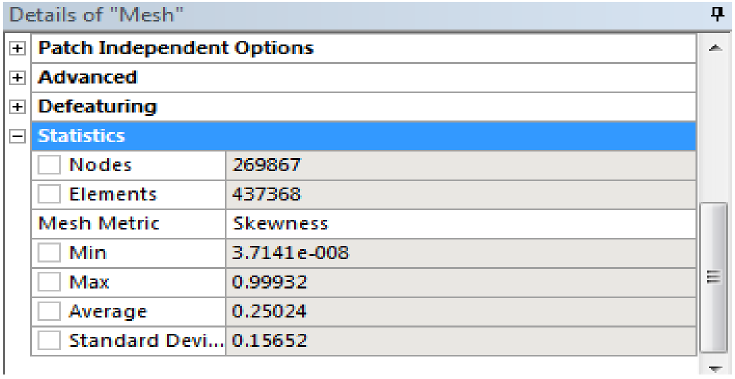

Figure 25: Details of mesh

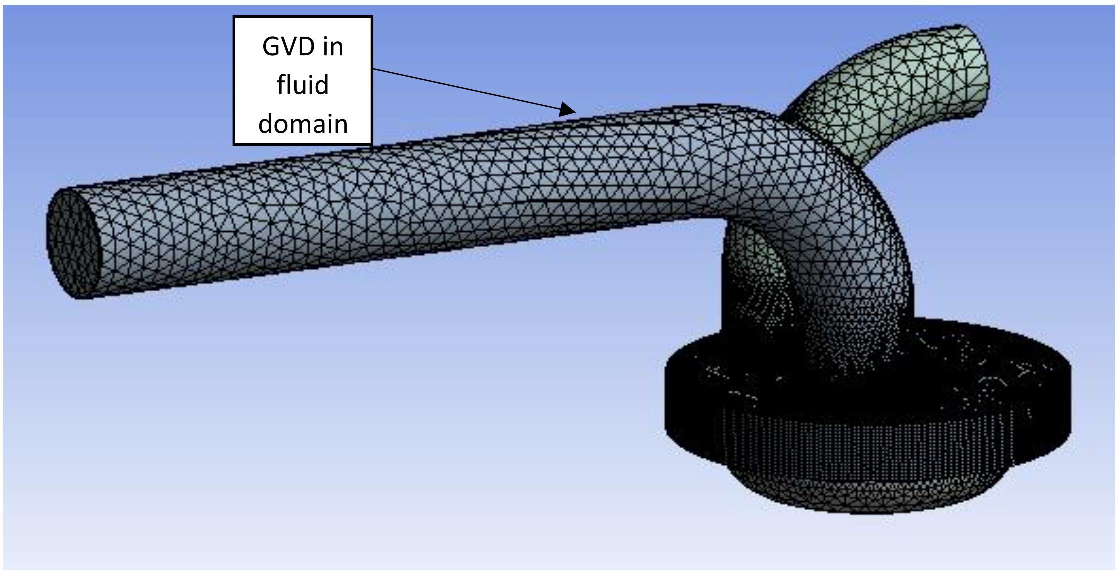

Figure 26: Geometry after meshed

3.3.3 Setting up the simulation

In the basic setting, auto save frequency and engine speed have been set up to 30° and 2000rpm.

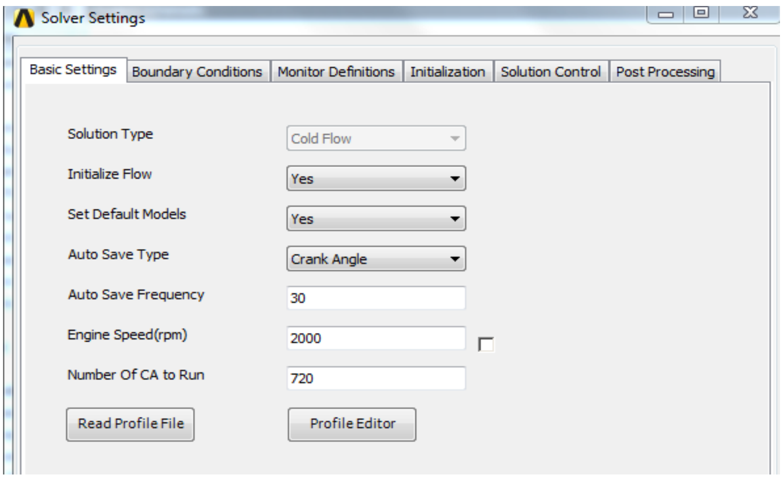

Figure 27: Basic Setting

For boundary conditions, the settings as shown in table 3: -

Type	Zones	Values
Inlet (ice-inlet-invalve)	Ice-inlet-invalve-1port	Pressure=1 atm Temperature=300K
Wall (invalve1)	In valve1-stem, In valve 1-ob, Invalve 1-ch, Invalve 1-ib	Temperature=300K
Wall (exvalve1)	Exvalve1-stem, Exvalve 1-ob, Exvalve 1-ch, Exvalve 1-ib	Temperature=300K
Wall (invalve-port)	Invalve 1-port	Temperature=300K
Wall (exvalve-port)	Exvalve 1-port	Temperature=300K
Outlet (ice-outlet-exvalve)	Ice-outlet-exvalve-1port	Pressure=1 atm Temperature=300K

Table 3: Boundary Conditions

3.3.4 Running the solution

As shown in the picture below, Number of Time Steps is already set to 2940 which has calculated from the number of CA to run in the basic setting tab of solver setting dialog box.

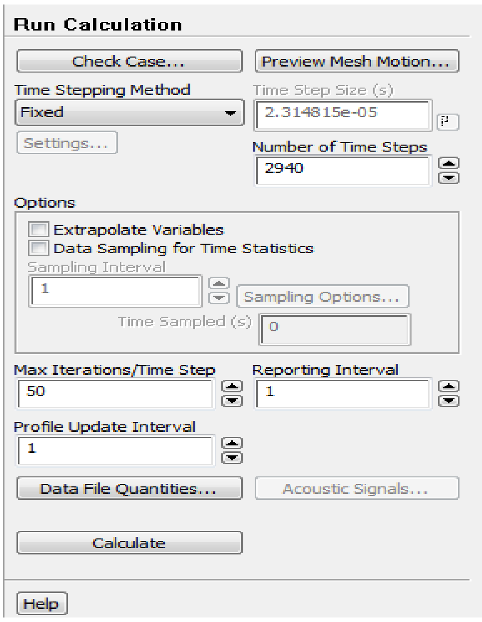

Figure 28: Settings before running the calculation

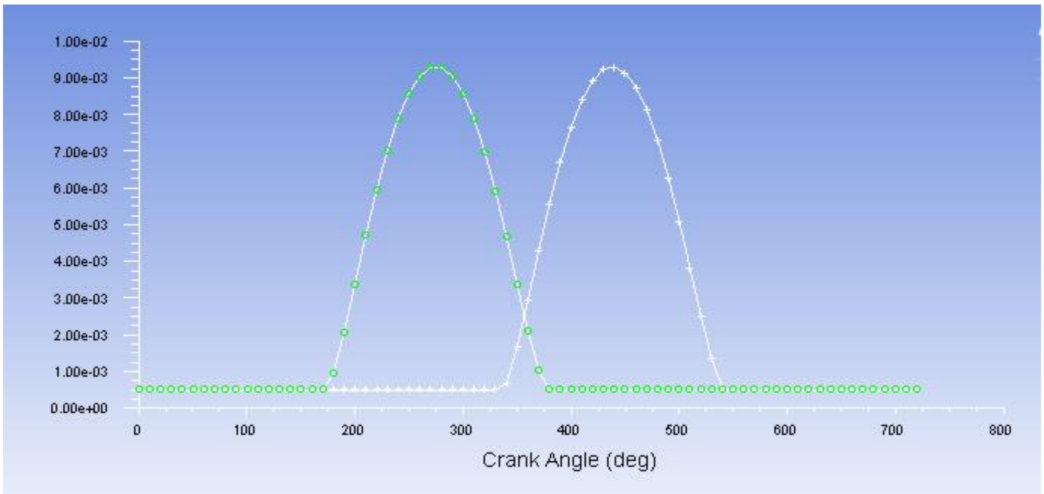

Figure 29: Valve lift profile

In figure 27, the graph shows crank angle vs time plot. The crank angle ranges from 0 to 720 for inlet and exhaust valve. Lastly, the calculation process took around 7 days for 4 CPU machine to solve the problem.

3.4 Governing Equation

Basically, three fundamental physic principles have been applied to compute this simulation, namely conservation of mass, conservation of momentum (Newton's second law) and energy [17]. The equation for vector notation can be referred below:

$$\frac{\partial \rho}{\partial t} + \nabla . [\rho \vec{u}] = 0$$

where $\boldsymbol{\rho}$ is the fluid density and $\boldsymbol{u}$ is the three dimensional flow velocities in the x, y, z directions.

Equation for forces and surface force on the control volume:

$$\frac{\partial (\rho U)}{\partial t} + \nabla . (\rho U \times U) = -\nabla p + \nabla . \tau + S_M$$

Where P, τ and S_M are the fluid pressure, strain rate and momentum source respectively.

Energy Equation can be referred below:

$$\frac{\partial (\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla . (\lambda \nabla T) + \nabla . (U . \tau) + U . S_M$$

Where h_{tot} and λ represent the total enthalpy and thermal conductivity of the fluid respectively.