THE BEST COMPRESSION RATIO OF MODIFIED PISTON FOR COMPRESSED NATURAL GAS (CNG) USAGE

IZAR FARHAN BIN NOR KHIRI

A report submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

ABSTRACT

Compression ratio can be defined as the pressure different between top dead center (TDC) and bottom dead center (BDC) of the piston in the combustion chamber. Compression ratio takes importance element to determine the power of the engine. Therefore, the objective of this study to determine the best compression ratio of modified piston for compressed natural gas (CNG) usage. The original engine is single cylinder four-stroke diesel engine. Compression ratio for this diesel engine is 20.28:1. The original piston of this engine is bore a bowl on the piston head to get certain compression ratio. Therefore, Computational Fluid Dynamics (CFD) method using FLUENT simulation software was used for this purpose. Model of combustion chamber with modified piston construct by using GAMBIT software, than export the model to FLUENT for dynamics analysis. The maximum pressure and maximum temperature in the combustion chamber model determined. Base on the simulation result, the highest pressure before the spark plug spark the flame is 15.5:1 of compression ratio. Meaning that, the best compression ratio of modified piston for compressed natural gas is 15.5:1.

ABSTRAK

Nisbah mampatan boleh definisikan sebagai tekanan yang berbeza antara pusat mati atas (TDC) dan pusat mati bawah (BDC) piston di dalam ruangan pembakaran. Nisbah mampatan merupakan elemen penting untuk menentukan kekuatan enjin. Oleh itu, objektif kajian ini adalah untuk menentukan nisbah mampatan terbaik bagi piston yang telah diubahsuai untuk kegunaan gas alam mampat (CNG). Enjin asal ialah enjin diesel empat lejang, satu silinder. Nisbah mampatan enjin diesel ini adalah 20.28:1. Piston asal enjin ini ditebuk mangkuk pada permukaan piston untuk mendapatkan nisbah mampatan tertentu. Dengan itu, perisian Computational Fluid Dynamics (CFD) yang digunakan untuk tujuan simulasi ini ialah perisian FLUENT. Model ruangan pembakaran bersama piston yang telah diubahsuai dibuat dengan menggunakan perisian GAMBIT sebelum model dieksport untuk dianalisis dengan menggunakan analisis dinamik. Tekanan maksimum dan suhu maksimum dalam model ruangan pembakaran akan menentukan. Berdasarkan keputusan simulasi, tekanan tertinggi sebelum percikan busi api ialah nisbah mampatan 15.5:1. Kesimpulan, nisbah mampatan yang terbaik bagi piston yang telah diubahsuai untuk kegunaan gas alam mampat (CNG) ialah 15.5:1.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

BDC	Bottom Dead Center
CA	Crank Angle
CFD	Computational Fluid Dynamics
CNG	Compressed Natural Gas
CR	Compression Ratio
FYP	Final Year Project
LCA	Lobe Centerline Angle
NGV	Natural Gas Vehicles
RPM	Revolution per Minute
TDC	Top Dead Center
2-D	Two-Dimentions
3-D	Three-Dimentions

CHAPTER 1

INTRODUCTION

1.1 Introduction

It is important to understand the function and the operation of the diesel engine before the modification stage begins. Therefore, a brief explanation of the operation for the conventional diesel engine is explained below and the main focus on the compression stroke. The detail of the operation in the diesel engine will be explained further more in Chapter 2.

- Air flows into the internal combustion engine through the inlet/intake valve during the induction stroke of the piston.
- Then, it is compressed adiabatically to the top dead centre (TDC) of the combustion chamber during the compression stroke. Before the piston reaches the TDC at the end of the compression stroke, fuel (diesel) is injected into the combustion chamber. The mixture of fuel and air would auto-ignite soon after the fuel is injected.

However, in reality, the diesel engine might not work as in theoretically. Normally, the mixture of fuel and air would burn very rapidly. This causes the pressure to rise rapidly and produces excessive knocking that can damage the engine. Therefore, the higher the pressure rise rates in the combustion process, the noisier is the diesel engine compared to the gasoline engine (Selim 2003).

Besides that, it produces gases like carbon monoxide, nitrogen oxides, unburned hydrocarbon, smoke, soot and other forms of black carbon as well as particulate matter such as lead. All the gases are harmful to the environment and human kind. They can cause greenhouse effect, acid rain, ozone thinning and air pollution to the environment. Due to these effects, human will get all kind of diseases such as lung cancer, breathing difficulties, poison and skin cancer.

Therefore, to overcome the above problems, researchers had been investigating into a new engine development to replace the conventional diesel engine. The aim is to develop a more environmental-friendly engine with similar if not better performance as the conventional diesel engine. One such development being investigated is the CNG-diesel engine. Since conventional diesel engine operates at a high compression ratio, the compression ratio of the CNG-diesel engine system is an area of major concern.

1.2 Problem Statement

Converting diesel engine to CNG-diesel engine give a huge change to the design of the combustion chamber. It is because of the engine need to reduce the compression ratio to the optimum level to get the best engine performance. There is several ways to reduce the compression ratio in the combustion ratio; one of them is by boring a bowl on the piston head. Therefore, this study is carried out with purpose to investigate and simulate the best compression ratios on the engine performance for Compressed Natural Gas (CNG)-diesel engine using CFD method

1.3 Objectives of the Study

In order to achieve the project, the objective of this project must be stated as a guide for the project. The objective of this project is stated below:

i) The main objective of the project is to find the best compression ratios on the engine performance for Compressed Natural Gas (CNG)-diesel engine by simulate the compression process of the engine.
ii) To investigate maximum temperature at certain compression ratio

1.4 Scopes of the Study

For this study, the scope that the project limited to is stated below:

i) To research on the information related to the combustion process, piston design, compression ratio and natural gas.
ii) Modify the piston design in the combustion chamber to get different compression ratio for the investigation.
iii) Using CFD simulation software such as FLUENT to simulate a 2D model of the piston in the combustion chamber during compression stroke.
iv) To analysis compression ratio between 13.5:1 to 15.5:1 at speed from 200 RPM to 4000 RPM with increment of 200 RPM.
v) Select the best compression ratio that delivered highest maximum pressure before the spark plug spark the flame.

CHAPTER 2

LITERATURE REVIEW

2.1 FUNDAMENTAL OF FOUR-STROKE ENGINE

The engine can be divided into two categories. One is the four-stroke cycle and the other one is the two-stroke cycle (Ferguson 1986). For the purpose of this project, only the four-stroke cycle will be discussed in this section. Basically, the four stroke cycle consists of the intake, compression, power and exhaust strokes as shown in Figure 2.1 below:

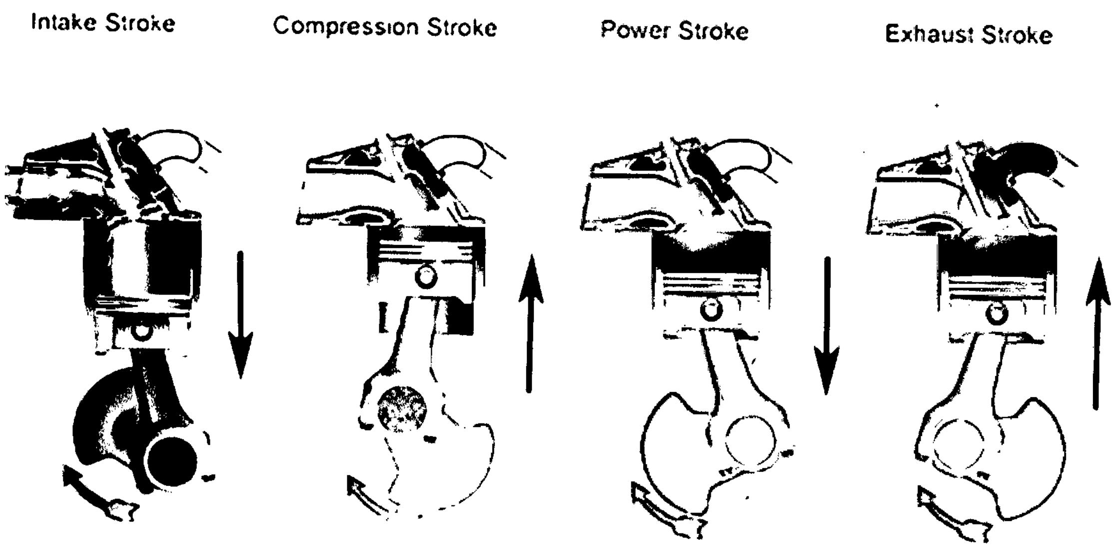

Figure 2.1: Four-Stroke Cycle

Each of the strokes is explained in more details as in the following:

- Intake stroke is the first stroke for the Diesel cycle, where the piston moves down to the bottom dead center (BDC). During this stroke, the inlet valve will open at the start of the stroke until it ends to let the air to flow into the combustion chamber while the exhaust valve is closed.
- Next, the piston will move up to the top dead center (TDC), which is also known as the compression stroke. Here, the air is compressed until just before the piston reaching the TDC, fuel is injected into the combustion chamber to ignite the combustion process through diffusion between fuel and air.
- Due to the high pressure gases compressed in the compression stroke, the piston is pushed down by the gases and forces the crank to rotate. Therefore, the piston will move from the TDC to the BDC again. This stroke powers the engine and thus it is known as the power stroke.
- For the fourth stroke, the piston will move up again to the TDC. As it approaches the TDC, the exhaust valve will open and allows the combustion products and the remaining burned gases to exit through it. Therefore, this stroke is known as exhaust stroke and the cycle will repeat again.

From the explanation above, the information will provide the basic knowledge for the design of piston in the combustion chamber. The most important part among the strokes mentioned above is the compression stroke, which is the time when the combustion will start to ignite before TDC level.

promotes better mixing quality of the fuel and air that improves the efficiency and faster combustion process. To lower the peak pressures and widening the knock limits, high compression engine needs a high performance ignition system with a gross overkill mode and cam's advance reduced by 2-3 degrees.

Lastly, seeking a high compression engine to produce maximum power output through the effectiveness of the combustion process means that the engine must operate at a higher rpm also. Therefore, combustion chamber compacted by lengthening the piston stroke, modified the piston crown to be flatter and bringing the intake and exhaust valves to a more vertical position approximately 18 degrees (Vizard 2003).

2.3 PISTON MODIFICATION

Generally, piston is one of the components in the internal combustion engine that is connected to a connecting rod to control its movement. The structure of the piston can be divided into two sections, which are the top section and the lower section as shown in Figure 2.1 in the following page. The top section is known as the crown or head while the lower section is called the skirt.

For the purpose of this project, the skirt is neglected and only the crown will be focused. The crown of the piston is the top surface where the explosive force is exerted when the piston moved up and down in the combustion chamber. Therefore, it is usually thick to resist the high gas pressure and to provide a smooth heat flow from the crown to the combustion rings.

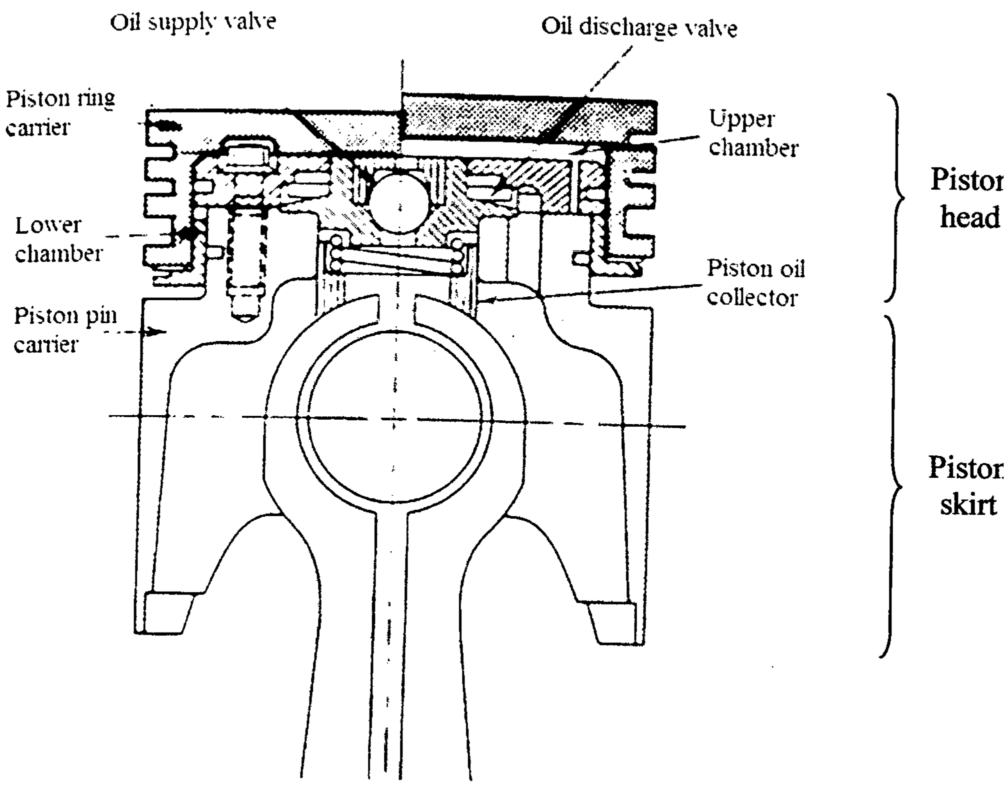

Figure 2.2: Structure of Piston (Ferguson 1986)

The shape of the piston crown depends on the design of the combustion chamber. There are various shapes such as concave, flat, cup, dome, hump and contour to promote turbulence in combustion or to control the combustion process (Yusof 2000). For example, a piston with an offset of a non-annular bowl is contoured to increase the spray plume length in order to avoid impingement. The shapes of the piston crown are shown in Figure 2.2 in the following page.

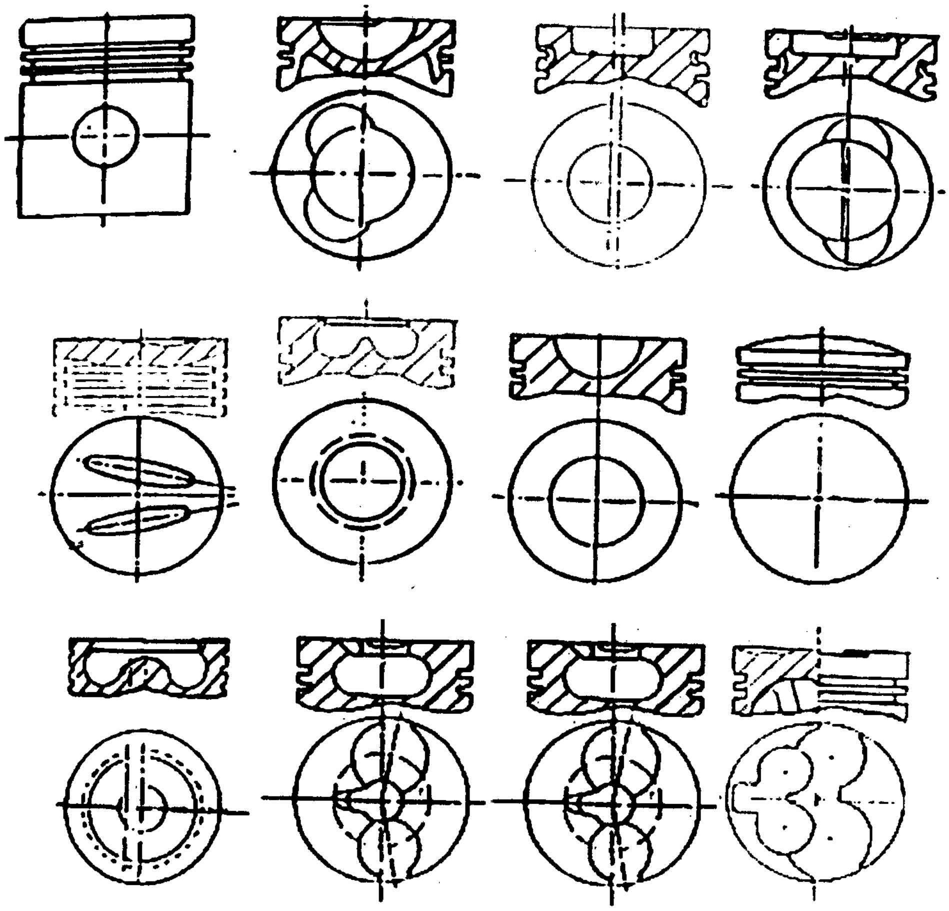

Figure 2.3: Various Shapes of Piston Crown

Besides that, it is important to select a proper material for the piston. The material must be light in weight, strong, economical and has high heat conductivity and wear resistance as well as can expand slightly when heated to achieve good combustion efficiency. Therefore, piston used in internal combustion engine is usually made of aluminum for its lightness and good heat conductor; or cast iron for greater heat and wear resistance (Yusof 2000). With a proper understanding on how the piston works in the engine, the design process will be made easier.

2.4 NATURAL GAS

Natural gas was first used as fuel in China during the Shu Han dynasty in AD221-263. The gas was obtained from shallow wells near seepages and was distributed locally through piping made of hollowed-out bamboos. Since then, there are no records on the usage of natural gas until the early 17th century in Northern Italy, where it was used as a fuel to provide lighting and heating (Tiratsoo 1979). As the time moves on, the usage of natural gas spread to North America, Canada, New Zealand and Europe. The usage was limited to domestic and industry heating.

When the world turned into the 20th century, the usage of natural gas expanded to most part of Western Europe and USA. Exploration for the natural gas source was more active after the post-war years. It became a commercial item in the form of liquefied natural gas (Tiratsoo 1979) for exports and imports. The gas fields or the natural gas resources are mainly found in Asia and Middle East countries. These include Malaysia, Brunei, Algeria, Libya, Saudi Arabia, Kuwait and Iran. By 1980s, these countries became the main exporters of natural gas.

The usage of natural gas as a vehicle fuel was discovered back in the early 1920s in Italy. The usage was not popular then due to the fact that natural gas was more commonly used in domestic and industry heating as well as to generate electricity. However, after the World War 2, there is growing interest on the usage of natural gas as vehicle fuel. This interest had led to establishment of approximately 1200 refueling stations and 1500 sub stations for natural gas in Italy by the early 1950s (Shamsudin & Yusaf 1995).

In 1991, Italy became the leading country in the research of natural gas vehicles. Italy had about 235 000 gasoline vehicles and 20 diesel vehicles that were converted to natural gas (Shamsudin & Yusaf 1995). Natural gas is compressed in a high pressure tank of 18-20MPa to form compressed natural gas (CNG). The country with the second highest natural gas vehicles is Argentina, which has about 100 000 gasoline vehicles and 10 converted diesel vehicles (Shamsudin & Yusaf 1995).

The trend towards converting gasoline and diesel vehicles to use CNG is still quite unpopular in Malaysia, compared to more developed western countries. However, due to the limitation of the crude petroleum oil reserves, which should last for another 15 years, Malaysia has since resolved to do more researches and experiments to use alternative fuels like natural gas. This is because the country natural gas reserves would last for about 80 – 90 years. Review showed that up until December 1994, there were about 900 vehicles converted to use CNG as fuel in Malaysia (Shamsudin & Yusaf 1995).

Generally, natural gas is one of the hydrocarbon families, made up of carbon and hydrogen atom. There are different compounds in natural gas such as methane, ethane, propane and iso-butane as well as other non-hydrocarbon compounds such as carbon dioxide and nitrogen. The natural gas found in Malaysia and used in this project is assumed to consist of mainly methane, ethane and propane. Their respective composition percentage of the typical natural gas found in Malaysia is shown in table below;

Table 2.1: Typical Composition of Natural Gas in Malaysia
(Yusaf et al., cited in Heath 1996)

Components	**Mole (%)**
Methane	83.44
Ethane	10.55
Propane	1.13
Iso-butane	0.13
Normal butane	0.07
Iso-pentane	0.01
Normal pentane	-
Hexane	0.01
Carbon dioxide	4.17
Nitrogen	0.31

The variation in the natural gas composition brought difficulties in the improvement of engine performance and minimization of the exhaust gas pollution. Since the proportion of methane in natural gas is the largest compare to other gases like propane and ethane, the main characteristic of natural gas can be directly related to the characteristic of methane.

To configure this problem with variation of natural gas composition, the Natural Gas Vehicles (NGV) Coalition in USA has recommended a general guideline of natural gas composition used for the emission test certificate. This test is carried out to help the certification of the engine's performance and its exhaust gas pollution characteristics that are affected by the gas composition (Bassi 1990).

This guideline of the natural gas composition is shown in Table 2.2 in terms of mole percentage. The data provided in Table 2.2 is based on the test carried out during the absence of liquid over the whole range of temperatures and pressures encountered in the engine and in the fuel supply system. Moreover, it is based on the average natural gas composition in USA and Europe as shown in Table 2.3.

Table 2.2: Natural Gas Composition Recommended by the NGV Coalition (U.S.A) for Emission Test Certificate (Bassi 1990)

Component	**Mole percentage**
Methane	88% + 0.5% (*)
Ethane	8% + 0.3%
C3 and more complex HCS	4% + 0.2%
C5 and more complex HCS	0.5% max
Total unsaturated HC	0.5% max
Hydrogen	0.1% max
Carbon Monoxide	0.1% max

Note: (*) expressed as % of total present organic carbon.

Table 2.3: Natural Gas Composition (U.S.A & Europe) (Bassi 1990)

Component	USA (%)	Italy (%)	Holland (%)	Russia (%)
Methane	92.21	99.63	89.44	93.27
Ethane	3.78	0.07	3.25	3.32
Propane	0.91	0.04	0.69	0.83
Butanes	0.47	-	0.29	0.37
Pentanes	0.10	-	0.09	-
Hexane and higher	0.04	-	-	-
Carbon Dioxide	0.59	0.01	0.70	1.00
Oxygen	0.05	-	-	-
Nitrogen	1.84	0.25	5.51	0.91

From the information given above, the composition of natural gas affects its properties no matter is physically or chemically. This is the reason to the difficulties faced by engineers all around the world to configure their engine design since every gas field in every nation has its own natural gas composition.

2.5 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics (CFD) is a technology that is used to analyze the dynamics of anything that can flow regardless in liquid or gaseous state. It is a software tool that can model or simulate a flow or phenomena of any system or device under analysis.

CFD is computed using a set of partial differential equations to predict the flow behavior. Besides that, it is also used for analyzing heat transfer model, mass flow rate, phase change such as solidification, chemical reaction such as combustion, turbulence model, mechanical movement such as rotating shaft, deformation of solid structure and many more (*FLUENT Manual – Introduction* 2004).

It is always a preferred method over the conventional design method because it is cheaper and save a lot of time. Before there is such technology, usually engineers need to build a real model for testing and redo the model again until the optimum result is obtained. Such a long procedure would consume more money and time. With the aid of CFD software, engineers can simulate different set of parameters for testing to get the optimum result before working on the real prototype without any additional cost.

The procedure for most of the CFD analysis divided by three phase. They are pre-processing, processing and post data who are the final step of analysis. In FLUENT software, the steps of three phases are shown below:

i. Pre-processing: The model used for the analysis is drawn, meshed and the boundary layers are determined. This is done using the GAMBIT software, which is the compatible modeling software for FLUENT. All the files for the geometry and meshing of the model are saved as mesh or grid file. Next, in FLUENT, the saved mesh or grid file of the model is read, checked and scaled for the required working unit.
ii. Processing: The model is defined for the type of solver and boundary conditions to be used. The model is defined according to the type of analysis required in the research project. The model is solved by setting the required parameters in the solution panel and then iterated for convergence.
iii. Post Data: Finally, results can be obtained from the graphic display and report in FLUENT. Results can be displayed in terms of contour, velocity vector, and particle track and path line. Any calculation required can be performed in FLUENT also. The results and all the data can be saved for future references by writing the files.

This software has various modeling capabilities that can be used in numerous kinds of analysis and application. Among its capabilities are listed in the following page. (*FLUENT Manual – Program Capabilities* 2004):

- Flows in 2D or 3D geometries are using unstructured solution-adaptive triangular/tetrahedral, quadrilateral/hexahedral, or mixed (hybrid) grids that include prisms (wedges) or pyramids.
- Incompressible or compressible flows.
- Steady-state or transient analysis.
- In viscid, laminar, and turbulent flows.
- Newtonian or non-Newtonian flow.
- Convective heat transfer, including natural or forced convection.
- Coupled conduction/convective heat transfer.
- Radiation heat transfer.
- Inertial (stationary) or non-inertial (rotating) reference frame models.
- Multiple moving reference frames, including sliding mesh interfaces and mixing planes for rotor/stator interaction modeling.
- Chemical species mixing and reaction, including combustion sub-models and surface deposition reaction models.
- Arbitrary volumetric sources of heat, mass, momentum, turbulence, and chemical species.
- Lagrangian trajectory calculations for a dispersed phase of particles/droplets/bubbles, including coupling with the continuous phase.
- Flow through porous media.
- One-dimensional fan/heat-exchanger performance models.
- Two-phase flows, including cavitation.
- Free-surface flows with complex surface shapes.

All the capabilities mentioned above are useful in providing a better approach for the analysis in applications such as process equipment, aerospace and turbo machinery, automobile, heat exchanger power generation in oil/gas industry and material processing. Therefore, with the availability of such capabilities, the analysis for the purpose of this research project can be carried out in a more accurate and user friendly way.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

Methodology is underlying principles and rules that govern a system while method can be defined as systematic procedure for a set of activities. Methodology is starting with piston modification by choosing a design from several idea and concept. Than, model of combustion chamber with modified piston is build up using the modeling software, GAMBIT and so with the grid generations. Before running simulation of the combustion chamber with modified piston, mesh sensitivity analysis is carried out to get the best grid generation. From the best grid generation, simulation procedures are carried out using computational fluid dynamics (CFD) simulation software, FLUENT.

3.2 PROJECT FLOWCHART

To achieve the objectives of this project, a methodology has been constructed (see figure 3.1). The methodology flow chart is purposed to give guidelines and directions to successfully accomplish the main goal of this project. The following is the summary methodology flow chart. Start with searching the information for literature review. Then select the piston design and start modeling. Grid selection is done after mesh sensitivity analysis. The best grid generation is setting up for analysis. All data from the analysis is collected and interpreted into table format. Lastly, discuss the obtain result.