University of Southern Queensland

Faculty of Engineering and Surveying

The Effect of Compression Ratio on the CNG-Diesel Engine

A dissertation submitted by

LIM Pei Li

in fulfillment of the requirements of

Courses ENG4111 and 4112 Research Project

towards the degree of

Bachelor of Engineering (Mechatronic)

Submitted: October, 2004

ABSTRACT

The objective of this study is to investigate the effects of different compression ratio on the performance of a diesel engine operating on a dual fuel system using Compressed Natural Gas (CNG) as the main fuel. The original diesel engine is a four-stroke, Ford diesel engine with a compression ratio of 22.9:1. In order to accommodate CNG in diesel engine, the compression ratio has to be reduced to prevent knock. Therefore, Computational Fluid Dynamics (CFD) method using FLUENT simulation software is used for this purpose. The engine performance will be investigated in terms of the mixing quality of CNG and air before injection of diesel fuel, temperature and pressure distribution, and stream function of the mixture during the compression stroke. Based on the simulation results, the optimum compression ratio chosen to operate the CNG-diesel engine without knock is 16.6:1. At this compression ratio, the engine can operate until the normal operating load condition where the wall temperature is 373 K before the engine was knocking.

University of Southern Queensland

Faculty of Engineering and Surveying

ENG4111 and ENG4112 Research Project

Limitations of Use

Certification

I certify that the ideas, designs and experimental work, results, analyses and conclusions set out in this dissertation are entirely my own effort, except where otherwise indicated and acknowledged.

I further certify that the work is original and has not been previously submitted for assessment in any other course or institution, except where specifically stated.

Lim Pei Li

Student Number: 0050012455

____

(Signature)

____

(Date)

ACKNOWLEDGMENTS

I like to express my gratitude to all my supervisors, Dr. Talal Yusaf, Dr. Fok Sai Cheong and Dr. Ahmad Sharifian for their guidance and support towards the end of this project. Besides that, many thanks to Miss Suhaila, a local researcher in University of Tenaga Nasional (UNITEN) for her guidance in the usage of Computational Fluid Dynamics (CFD) simulation software FLUENT. Last but not least, I like to thank my family and friends for their support especially during the bad times.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

Φ	-	Equivalence ratio
ϕ	-	Scalar quantity
Φ_f	-	Value of Φ convected through face f
δ_{ij}	-	Kronecker delta
ε	-	Dissipation rate
Γ_Φ	-	Diffusion coefficient for ϕ
σ_k	-	Turbulent Prandlt number for turbulent kinetic energy, k
σ_ε	-	Turbulent Prandlt number for dissipation factor, ε
$\nabla\Phi$	-	Gradient of ϕ
$(\nabla\Phi)_n$	-	Magnitude of $(\nabla\Phi)$ normal to face f
A	-	Surface area
A_f	-	Area of face
a	-	Speed of sound
b		Bore diameter
CO	-	Carbon monoxide
CO_2	-	Carbon dioxide
CH_4	-	Methane
C_2H_6	-	Ethane
C_3H_8	-	Propane
d	-	Piston stroke
e	-	Specific total energy
F_i	-	External body force from interaction with the dispersed phase in i direction
g_i	-	Gravitational acceleration
G_k	-	Generation of turbulent kinetic energy due to mean velocity gradients
G_b	-	Generation of turbulent kinetic energy due to buoyancy
h	-	Clearance or squish height

h	-	Sensible enthalpy
$h_{j'}$	-	Sensible enthalpy of species j'
$J_{j'}$	-	Diffusion flux of species j'
k	-	Turbulent kinetic energy
k_{eff}	-	Effective conductivity
k	-	Turbulent thermal resistance
L	-	Length scale
$\dot{m}$	-	Rate of mass of the object generated in the system
$m_{j'}$	-	Mass fraction of species j'
N_{faces}	-	Number of faces enclosing cell
N_2	-	Nitrogen
n_f	-	Number of faces of the control volume
NO_x	-	Nitrogen oxides
O_2	-	Oxygen
p	-	Pressure
$\Pr_t$	-	Turbulent Prandlt number for energy
ρ	-	Fluid density
S_Φ	-	Source of ϕ per unit volume
S_h	-	Additional volumetric heat sources i.e. heat of chemical reaction
τ_{ij}	-	Stress tensor
t	-	Plate thickness
u_i and u_j	-	The *jth* Cartesian component of the instantaneous velocity
μ	-	Fluid dynamic viscosity
u	-	Component of flow velocity perpendicular to the gravitational vector
$\vec{u}$	-	Flow velocity vector
$\vec{u}_g$	-	Grid velocity of the moving mesh
U_s	-	Squish velocity
U_p	-	Instantaneous piston speed
V	-	Total clearance volume

V_S	-	Swept volume
V_{plate}	-	Plate volume
v	-	Component of flow velocity parallel to the gravitational vector
v	-	Velocity vector
v_f	-	Mass flux through the face
V_{bowl}	-	Volume of piston bowl or cup
V_{cup}	-	Volume of piston bowl or cup
Y_M	-	Fluctuating dilatation in compressible turbulence to the overall dissipation rate
$Y_{C_N H_{2N}}$	-	Mass fraction of the hydrocarbon
M_t	-	Turbulent Mach number

GLOSSARY OF TERMS

2D	-	Two dimensional
3D	-	Three dimensional
BDC	-	Bottom dead center
CAD	-	Computer-Aided Drafting
CFD	-	Computational fluid dynamics
CNG	-	Compressed natural gas
CR	-	Compression ratio
ECU	-	Electronic control unit
EGO	-	Exhaust gas oxygen
HC	-	Hydrocarbon
LCA	-	Lobe Centerline Angle
LEL	-	Lower explosive limit
LES	-	Large eddy simulation
NGV	-	Natural gas vehicle
RAPIER	-	Rapid analysis of products by integrated engine routine
RNG	-	Renormalization group
RSM	-	Reynold’s stress number
Rpm	-	Revolution per minute
TDC	-	Top dead center
UEL	-	Upper explosive limit
USA	-	United States of America
VOC	-	Volatile organic compound

CHAPTER I

INTRODUCTION

1.1 Introduction

It is important to understand the function and the operation of the diesel engine before the design stage begins. Therefore, a brief explanation of the operation for a conventional diesel engine is explained below and the main focus is on the compression stroke. The details of the operation in the diesel engine will be explained further in Chapter 3.

- 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, p.412).

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 dual fuel CNG-diesel engine. Since conventional diesel engine operates at a high compression ratio, the compression ratio of the dual fuel system is an area of major concern.

1.2 Objectives and Scope

The main objective of the project is to investigate and simulate the effects of different compression ratios on the engine performance for Compressed Natural Gas (CNG)-Diesel engine. The engine performances mentioned above are in terms of the mixing quality of CNG and air, degree of turbulence in the combustion chamber, pressure and temperature distribution.

As specified in the Project Specification under Appendix A, the sub-objectives are explained below:

- To research on the information related to the combustion process, heat and mass transfer, emission gases, compression ratio, knocking effects, turbulence model and piston structure.

- Modify the dimensions of the piston in the combustion chamber to get different compression ratio for the investigation.
- Using CFD simulation software such as FLUENT to simulate a 2D model of the piston in the combustion chamber during compression stroke to investigate the pressure and temperature distribution as well as the stream function.
- Select the optimum compression ratio that delivered optimum combustion efficiency without knock.

1.3 Project Methodology

The execution of the research project is planned in several stages. After the project is allocated and approved by the examiner and staff of ENG4111, the next step is to specify the details of the project such as the objectives, the requirements and the plan for the project workload. At the same time, research and literature survey are carried out to find the requirements for the design of the piston in the combustion chamber and other information related to the project.

The literature survey undertaken is mainly from the books, journal papers and information from the Internet. After that, a review is written to explain, summarize and critically report on all other relevant information found in the materials mentioned. The review should summarize the basic principles used for the design of the piston in dual fuel engine. Besides that, information on the compression ratio, properties and composition of CNG and emission control are equally important to make use of their advantages to the design project.

Then, analysis of the conventional design of piston used in diesel engine is set as a reference for the design of the piston in CNG-diesel engine. Although CNG has high resistance to knock but it has a tendency to cause knocking in very high pressure when it is compressed. The method of redesigning the piston dimension by insertion of plates on the piston head is employed as a measure to reduce the compression ratio of the CNG-diesel engine, thus minimizing the possibility of knock to occur. Analysis in terms of simulation would be performed for both the diesel and mixture of CNG-diesel fuel to see the comparison.

Modeling and simulation procedures are carried out by using computational fluid dynamics (CFD) simulation software such as FLUENT 6.1 and the modeling software like GAMBIT 2.1. Simulation is carried out for different compression ratio by using different thickness of plate on the piston head to increase the clearance volume in the combustion chamber. The objectives are to simulate and analyze the result of the pressure distribution, temperature distribution and the stream function in the combustion chamber when piston is compressed by using dynamic meshing method in FLUENT.

All the simulation will be on the analysis of the mixing quality of CNG and air, and pure air also. If the mixture is homogeneously mixed, then the combustion efficiency would be better. Finally the optimum compression ratio is chosen after all the results are obtained and analyzed. The optimum compression ratio for the CNG-diesel engine must provide a similar or better performance than the conventional diesel engine.

The order of the project methodology as explained earlier is illustrated in the schematic diagram, Figure 1.1 in the following page.

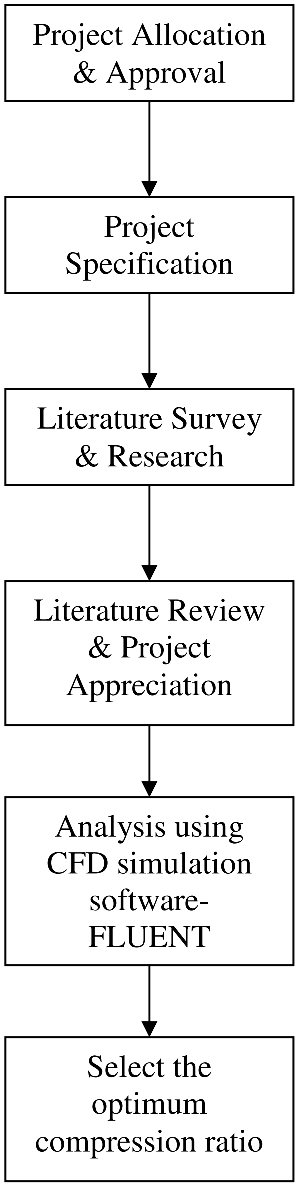


Figure 1.1 - Project Methodology

1.4 Dissertation Overview

This dissertation is divided into seven chapters. Chapter 1 is the current chapter, which introduce the research topic to the reader and explained on the objectives, scope of the research and the project methodology. The following Chapter 2 will summarize the details on the literature reviewed earlier before commencing on the design stage. Among the topics reviewed and covered in this chapter are about the characteristic and properties of natural gas, operation of diesel engine and dual-fuel engine, and compression ratio.

Next, in Chapter 3, the piston design and the types of piston dimensions are explained. Besides that, the operation of the engine is discussed further in this chapter as well as the introduction to the swirl amplification. After the explanation on the piston design, the following Chapter 4 will focus on the theoretical explanation of the method used in the computational fluid dynamic analysis such as the continuity equations of mass, momentum and energy.

The actual model and the modification of the piston design are discussed in Chapter 5. Moreover, this chapter will give all the relevant calculations and the boundary conditions that are required for the simulation of the piston model. After the actual planning in Chapter 5, all the simulation results will be shown and discussed in Chapter 6. Finally, the conclusion will be made in Chapter 7 by concluding the overall results obtained as well as the knowledge learned from the project.

CHAPTER II

LITERATURE REVIEW

2.1 Background Information

2.1.1 The History on the Usage of 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 17^{th} century in Northern Italy, where it was used as a fuel to provide lighting and heating (Tiratsoo 1979, p.8). 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 20^{th} 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, p.8) 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, p.101).

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, p.102). 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, p.102). By 2003, Argentina overtook Italy and became the leading country in the world to convert vehicles to use CNG (*Year-end 2003 Refining Margins Dip, then Recover* 1 April 2004 <http://www.hydrocarbonprcessing.com>).

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, p.103).

Hence, from the history of natural gas, the expansion of the usage of natural gas in automotive industry becomes clearer. Next, by learning the composition of natural gas, it will lead to a better understanding of its effects and its importance in the dual fuel engine conversion.

2.1.2 Composition of Natural Gas

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 2.1.

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

Table 2.1 - Typical Composition of Natural Gas in Malaysia

(Yusaf et al., cited in Heath 1996, p.20)

It is very important to know the composition of the natural gas used for the analysis because different composition has different effect on the combustion process in the diesel engine. Unfortunately, there is no standard reference for the design of a standard CNG-diesel engine because the natural gas composition varies in different countries. This posed a problem to the engineer in designing the fuel feeding system and the injection system for the CNG-diesel engine.

Moreover, 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, p.740).

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.

Component	**Mole percentage**
Methane	88% + 0.5%(*)
Ethane	8% + 0.3%
C_3 and more complex HC_S	4% + 0.2%
C_5 and more complex HC_S	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 2.2 - Natural Gas Composition Recommended by the NGV Coalition (U.S.A) for Emission Test Certificate (Bassi 1990, p.742)

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

Table 2.3 – Natural Gas Composition (U.S.A & Europe) (Bassi 1990, p.741)

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.

2.1.3 Properties of Natural Gas

Physically, natural gas is colorless, tasteless, relatively non-toxic (Yusaf et al. 1996a, p.19) and not a volatile organic compound (VOC) (*Green Mission-Compressed Natural Gas a Fact File* 5 February 2004 <http://www.gasmancng.com>). It exists in our environment at normal temperature and pressure, which gave it its name. To use natural gas as fuel in vehicles, it has to be compressed at a high pressure of about 18-20MPa at normal temperature in vessels before it can be supplied to the engine's combustion chamber. Generally, natural gas is lighter than air with a vapour density of 0.68 relative to air (*Why Natural Gas: Economic, Environmental Pollution, Safety, Properties, Composition and Availability 1995*, p.10). Therefore, if leaking happens, it will not cause explosion but instead it will disperse to the atmosphere.

Natural gas has a high auto-ignition temperature compared to gasoline or diesel, which is the lowest temperature for it to ignite through heat alone and without any spark or flame (*Clean Air Program - Assessment of the Safety, Health, Environmental and System Risks of Alternative Fuel* 2 April 2004 <http://ntl.bts.gov/DOCS/afrisks.html#toc>). Higher ignition temperature means that natural gas is more difficult to ignite. This can significant reduce the fire hazard, and constitute anti-knocking ability especially when it is compressed in a very high pressure in the combustion chamber. This property is certainly useful for the design of a dual-fuel engine. The ignition temperature for natural gas is about 900 K. (*Compressed Natural Gas* 11 March 2004 <http://www.mckenziecorp.com/dehydration.htm>).

Other physical properties such as the flammability limits range, octane rating, Wobbe Index and flash point also play an important role in the analysis of compression ratio and combustion efficiency of the engine. The Wobbe Index is a measure of the fuel interchangeability with respect to its energy content and the air-fuel ratio (*CNG Engine Performance* 1995, p.2). Flash point is the minimum temperature for an ignition. The flash point for natural gas is approximately 180°C at normal pressure as shown in Table 2.4 in next page.

The flammability limit range is the concentration of natural gas in air to cause an explosion. This is between the lower explosive limit (LEL) of 5% to the upper explosive limit (UEL) of 15% (*Clean Air Program - Assessment of the Safety, Health, Environmental and System Risks of Alternative Fuel* 2 April 2004 <http://ntl.bts.gov/DOCS/afrisks.html#toc>). If the concentration of natural gas is more or less than this range, an explosion would not occur. This will certainly reduce the risk of explosion of CNG in air due to leaking because natural gas can only burn in air when the concentration of CNG is high. With this wide range of limits shown in Table 2.4 in the following page, a lean mixture of CNG and air can be used for the CNG-diesel engine to promote better exhaust emission properties.

The octane rating is an important property in determining the compression ratio of the engine. For natural gas, the octane number is approximately 130 as shown in Table 2.4. This is much higher than gasoline with an octane number of 96 (Yusaf et al. 1996a, p.21). This property is important as it determines the time needed for the natural gas and air to mix homogeneously in the combustion chamber to minimize knocking or detonation. Table 2.4 shows the comparison for the general properties of natural gas, diesel and gasoline. Other properties such as the energy content, boiling points, density, combustion energy and vaporization energy have been included in Table 2.4 below:

Properties	**Natural Gas**	**Diesel**	**Gasoline**
Boiling range (K @ 101325Pa)	147	433 - 655	300 - 489
Density (kg/m^3)	128	785 - 881	689 - 785
Auto ignition Temperature (K)	900	477 - 533	505 - 755
Flash Point (K)	124	325	266
Octane Number	130	n/a	96
Flammability Limits Range	LEL = 5 UEL = 15	LEL = 0.7 UEL = 5	LEL = 1.4 UEL = 7.6
Net Energy Content (MJ/kg)	49.5	43.9	43.5 - 44.4
Combustion Energy (kJ/m^3)	24.6	36.0	32.6
Vaporization Energy (MJ/m^3)	215 - 276	192	293

Table 2.4 – Properties of Natural Gas, Diesel and Gasoline Fuel (*Compressed Natural Gas* 11 March 2004 <http://www.mckenziecorp.com/dehydration.htm >)

The chemical properties of the natural gas are important for the selection of natural gas as the vehicle fuel. Very often, natural gas is said to be a clean fuel because it is lead free, has a lower emission for carbon dioxide (CO_2), carbon monoxide (CO), nitrogen oxides (NO_x) and unburned hydrocarbon compare to the emission produced by diesel engines (Yusaf et al. 2001b, p.58). All the emission gases mentioned are a major hazard to human being and the environment.

High concentration of carbon dioxide in the atmosphere causes global warming. Emissions of carbon monoxide can deteriorate the health of human being because it will combine with the hemoglobin of the red blood cell in the blood stream to form a stable red carbonxy-hemoglobin, which is not able to carry oxygen. Nitrogen oxides cause smog and acid rain to the environment. It causes health hazards like bronchitis, lung irritation, respiratory infections and pneumonia. It also reduces the strength and quality of textiles and corrosion in metals (Yusaf et al. 2001b, p.58).

The given properties of natural gas, which is mainly consists of methane is a big influence to the CNG-diesel engine conversion process. For a better understanding on the reason for choosing CNG in the dual fuel engine conversion, its advantages and the disadvantages of dual fuel engine are presented in the next section.

2.2 Advantage and Disadvantage of Using CNG-Diesel Engine

The advantage of converting conventional diesel engine to dual fuel CNG-diesel engine is to convert the engine back to 100% diesel operation easily. It does not require a spark ignition or an electrical system to start the combustion process (Shamsudin & Yusaf 1995, p.104). Instead like spark ignition engine operating on premixed combustion process, the dual fuel engine works in a diffusion combustion process with a high pressure.

Besides that, conversion of diesel engine is more economical than conversion of spark ignition engine due to less modification on the original diesel engine (Shamsudin & Yusaf 1995, p.104). Since dual fuel engine operates at low compression ratio such as 16:1, the original diesel engine only needs minimum modification to suit the CNG-diesel operation. Therefore, the cost of conversion for diesel engine is lower compared to spark ignition engine.

Another advantage of using CNG-diesel engine is the increase in power output compared to the original diesel engine, which must be within the range of up to 3000rpm (Yusaf et al. 2001b, p.64). If more than 3000rpm, the power output of CNG-diesel engine would decrease slowly due to knocking in the rapid combustion process. Since CNG has high octane rating and high ignition temperature, the CNG-diesel engine has a higher resistant to knock than the conventional diesel engine. However, mild knock still will occur; hence a careful consideration for the compression ratio is necessary to minimize the knock.

The CNG-diesel engine can be used at a low substitution level (Shamsudin & Yusaf 1995, p.104) to lower the emission gases. It is more environmental-friendly and has a good energy security because it produces less hazardous emissions such as carbon monoxide, nitrogen oxides, sulphur oxides and particulate matters. Moreover, the benefits and the good properties of methane as the main component in natural gas reduce the risk of explosion when leakage occurs.

However, there still exist some limitations to the CNG-diesel engine. For instance, the fuel control system is more complicated compared to the spark ignition system (Shamsudin & Yusaf 1995, p.104). Adjustment to the mixing ratio of CNG-diesel and the control of the injector pump are difficult to set correctly for a homogeneous mixture. The variation in the composition of natural gas around the world creates difficulties to certificate the engine performance and exhaust pollution characteristic in the fuel control system.

Another important disadvantage of CNG-diesel engine that caused significant withdrawal from vehicle buyers is the supply system. Normally, dual fuel vehicles need to have separate tanks to store the natural gas in compressed tanks, which are in liquid form. This increases the weight of the vehicle (Weaver, P.E & Lit 2002, p.2) causing reduced power output, limited storage space and posing other drivability problems to the vehicle owner.

The refueling time is longer approximately twice as much as the refueling time for normal gasoline vehicles. There are lesser refueling station for natural gas compared to other conventional fuel such as diesel and gasoline (Weaver, P.E & Lit 2002, p.2). Moreover, it has shorter driving range due to lower energy density of natural gas. Lastly, it has a low efficiency when operating in part load or no load condition (Shamsudin & Yusaf 1995, p.104) but this disadvantage is not significant because the vehicle would be in idling position.

After reading up on the dual fuel engine’s advantages and disadvantages, the following section will provide an insight to the type of diesel engine conversion process with the utilization of natural gas.

2.3 Conversion of Diesel Engine

There are two methods of converting diesel engine to utilize natural gas as the main vehicle fuel in diesel engine. The first method is to utilize CNG fully in diesel engine known as the CNG dedicated conversion. The second method is the dual fuel conversion using CNG and diesel fuel to ignite the flame in the combustion chamber.

2.3.1 CNG Dedicated

The diesel engine for this type of conversion used 100% CNG to operate it. Therefore, only CNG is injected into the combustion chamber to mix with the air and hence the name, CNG dedicated. This type of conversion is basically meant for heavy duty vehicles like busses and trucks that have a lot of problems with the large cylinder unit displacement (Bassi 1990, p.756).

Conversion to CNG dedicated engine needs greater modification than the dual fuel conversion especially on the combustion chamber design and other components also. Basically, there are three main modifications such as the gas supply system, the ignition system to reduce knock and the electronic control unit (ECU) to arrange the two subsystems (Shamsudin & Yusaf, cited in Alimoradian & Allen 1990, p.105).

Firstly, the original diesel engine's cylinder head is modified. The injection system is removed and substituted with a spark plug and an injector pump to promote a sparks ignition system like gasoline engine. The inlet and exhaust valves were replaced to accommodate CNG with special seating angle while the piston is milled to reduce the compression ratio from 14:1 to 12:1 to promote flame propagation (Shamsudin & Yusaf 1995, p.105).

Next, the air manifold is redesigned to accommodate the gas flow mixer, the control valve and the governing unit. The control valve is used to control the pressure at high engine load and speed. The governing unit is used to control the high speed inlet air. The ECU is programmed to receive signals from the engine speed, throttle piston, inlet air temperature and gas pressure sensor (Shamsudin & Yusaf 1995, p.105). It is programmed to control and determine the gas flow rate and the ignition system under a certain engine rpm.

Other components to be considered are the CNG cylinders, filler valve, pressure regulators, carburetor, ignition system and end speed governor. The numbers of CNG cylinders required depend on the vehicle but the cylinder must be a high pressure tank of 200 bar with typical volume of 50 liters to store the CNG. Normally, there are two pressure regulators used to reduce the pressure of 200 bar to just above the atmospheric pressure (*Green Mission – Compressed Natural Gas (CNG) a Fact File* 5 February 2004 <http://www.ashokleyland.com/cng.html>).

By using CNG dedicated engine, the torque and power output obtained are higher than the original diesel engine. It will promote better exhaust emission than the original diesel engine also. However, it will better to maintain the original power output and torque of the diesel engine to reduce the modification process. The high compression ratio of the original diesel engine needs to be reduced to prevent knocking in the combustion chamber (Bassi 1990, p.756). Besides that, if the valve stems are not fitted with oil seals, the oil might enter the combustion chamber causing blue smoke emission which might damage the spark ignition system (Shamsudin & Yusaf 1995, p.109).

2.3.2 Dual Fuel (CNG-Diesel)

Like CNG dedicated, there are a few components to be considered in the conversion to dual fuel CNG-diesel engine. The numbers of cylinders and their capacity depend on the engine and the vehicle. The refueling valve or the filler valve has to be adjusted to accommodate CNG into the cylinder. Just like the previous case in CNG dedicated, the pressure capacity of the tanks used in the dual fuel engine is about 200 bar and 50 litres of volume. However, to obtain the optimum mixer pressure, multiple pressure regulators are used to reduce the pressure of 200 bar to below the atmospheric pressure (*Green Mission – Compressed Natural Gas (CNG) a Fact File* 5 February 2004 <http://www.ashokleyland.com/cng.html>).

Other new components used for dual fuel CNG-diesel engine are the pneumatic or electronic speed control, rack limiter, venturimeter and the linear load valve. For electronic or pneumatic speed control, it uses the safety valve to close the gas supply when the engine's rpm is beyond the specified limits. As for the rack limiter, it controls the diesel fuel flow into the combustion chamber with respect to the engine's rpm, load and the speed. It allows 100% diesel flow up to a certain engine's rpm and reduced to pilot value when beyond the specified speed (*Green Mission – Compressed Natural Gas (CNG) a Fact File* 5 February 2004 <http://www.ashokleyland.com/cng.html>).

Besides that, the linear load valve, which is connected to the accelerator, is used to control the CNG flow as per engine load. The CNG-air mixer consists of the venturimeter and the metering device are fitted to the air manifold to control the gas mixture (*Green Mission – Compressed Natural Gas (CNG) a Fact File* 5 February 2004 <http://www.ashokleyland.com/cng.html>). The size and shape of the venturimeter depend on the types of diesel engine used, the fuel-air ratio and other secondary factors (Shamsudin & Yusaf 1995, p.108). Moreover, the diesel fuel pump is adjusted and fitted to the tune control system, usually an actuator to reduce the diesel supply at idle rate during the dual fuel operation. This mechanism will be supported by the linkage between the pedal and both the fuel pump and mixer (Bassi 1990, p.755).

If necessary, the compression ratio has to be reduced to prevent knock in the combustion chamber due to the high pressure and self-ignite characteristic of CNG. Theoretically, increasing the compression ratio can improve the thermal efficiency by promoting better mixing for the CNG and air (Shamsudin & Yusaf 1995, p.110). However, this theory is not linear and there is no formula to calculate the compression ratio to access the correct proportion used for the CNG and diesel in the combustion process.

Therefore, various parameters have to be considered to prevent knocks. Increasing the compression ratio will increase the fuel consumption causing vibration to the vehicles especially busses. Hence, air-fuel ratio and the ignition timing are reduced to promote a knock free operation (Shamsudin & Yusaf 1995, p.110). Besides that, crank angle of the dual fuel engine has to be larger than the original diesel engine, but shorter than the CNG dedicated engine (Bassi 1990, p.755).

It is concluded that correct proportion of CNG composition and diesel fuel are supplied to the combustion chamber. Besides that, all the relevant analysis related to the crank angle, pressure distribution, temperature distribution and the heat transfer in the combustion chamber are carried out to analyze their effects on the engine performance. Engine performances such as the efficiency of the combustion process and the mixing quality of the gas mixture are investigated to minimize the existence of knocking prior to the start of the combustion process.

In order to analyze the engine performance as mentioned above, the role of compression ratio in engine performance and how it affects the dimensions of the piston design is presented in the next section.

2.4 Compression Ratio (CR)

Compression ratio (CR) is the ratio of the total volume of the combustion chamber when the piston is at the bottom dead center (BDC) to the total volume of the combustion chamber when piston is at the top dead center (TDC). Theoretically increasing the compression ratio (CR) of an engine can improve the overall efficiency of the engine by producing more power output. Indeed, to increase the CR, there are many aspects concerning the operation of the engine that has to be considered to check for the parts compatibility.

For instance, shorter cam duration can improve the effectiveness of increasing the CR. During the compression stroke, more air is allowed to trap above the piston prior to the inlet valve closure. Moreover, the lesser the opening duration for the cam; the shorter is the distance for the piston to move up to the bore on the compression stroke (Vizard 2003, p.2).

Besides cam duration, the cam's Lobe Centerline Angle (LCA) and the cam advance are important to increase the CR. A wider LCA (numbers getting larger) promotes greater increase in CR than a tight LCA (numbers getting smaller) (Vizard 2003, p.4). Moreover, engine with more advance cam, about 2-4 degrees of advance (Vizard 2003, p.4) promotes a faster intake closure. Hence, the engine power output will be less sensitive to the valve closure and the compression timing combination.

For every ratio increased, the peak cylinder pressure will increase with approximately 100-110 psi (Vizard 2003, p.5). Consequently, there will be thermal stresses on the component parts of the engine such as the head gasket, connecting rods, crank and blocks. Usually, peak pressure will occur earlier in the power stroke and the rate of cylinder pressure decay is much faster due to higher rate of volume increased in the combustion chamber.

With these limitations, situation such as knocking will occur and the preventive methods are to find a fuel that has a high octane rating about 115 or higher (Vizard 2003, p.8), a better design on the piston bore or the combustion chamber to improve the swirling action and the injection timing. Since higher octane rating increases the temperature, it is important to keep the induction system as cool as possible to avoid any knocking.

Besides that, the piston to head clearance at TDC has to be optimized to improve the swirling action. The crevices and sharp corners within the combustion chamber (Vizard 2003, p.9) are minimized to set a tight quench/squish clearance. This 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, p.11).

It is concluded that to achieve a high power output it is not an easy task. If the design of the piston head or the geometry of the combustion chamber is not well done, it will lead to failure of the engine operation. Therefore, it is wiser to maintain the efficiency and performance of the original diesel engine in the CNG-diesel engine with a possibility of reducing the CR to prevent knock.

Besides the compression ratio, as mentioned before the natural gas composition is another important criterion for the engine conversion. Therefore, the types of engine used for the conversion depending on the variation of fuel composition will be discussed in the section.

2.5 Types of Engine Used in term of the Fuel Composition

Basically, engine operates on different fuel-air ratio is divided into two types of engine. They are the stoichiometric burn engine and the lean burn engine.

2.5.1 Stoichiometric Burn Engine

Stoichiometric burn engine is characterized by its ability to operate without excess air remaining after the combustion process. It operates on a stoichiometry fuel-air ratio, which means the equivalence ratio (Φ) equals to 1. This type of engine can be found in light duty vehicles that have clean exhaust emission properties. A three-way catalyst exhaust after treatment technology is installed in the light duty vehicles to meet the exhaust emission standards (*CNG Engine Performance* 1995, p.1).

Light duty engine operates on a closed loop system that has a feedback control on the fuel-air ratio to accommodate the variation in the fuel composition. The feedback control is equipped with standard stoichiometric exhaust gas oxygen (EGO) sensor to assist the computer in measuring the oxygen content (Clark et al. 2003, p.1). This helps to adjust the engine operation according to the fuel composition. Hence, stoichiometric burn engines are said to be more tolerant to the variation in fuel composition.

However, the disadvantage of this engine is its high combustion temperature causing knocking or detonation to the components in the combustion chamber. The engine has no excess air to dilute the combustion products after the combustion process, thus the combustion temperature increased. Moreover, the variation in fuel composition is a burden to the engine operation. The fuel consumption rate has to adjust to a stoichiometric fuel-air proportion that eventually reduced the fuel economy.

Nevertheless, to accommodate CNG in this type of engine is still acceptable because CNG has a high octane rating. Since the engine can cause knocking, high octane rating of CNG can resist knock or detonation prior to the combustion process.

2.5.2 Lean Burn Engine

Lean burn engine operates under excess air condition in the combustion process, where the equivalence ratio (Φ) is less than 1. The mixture of fuel and air is known as lean mixture and there will be some leftovers of oxidizer in the combustion product because all the fuel is burned with excess oxygen in the process.

Lean burn engine is used in medium to heavy duty vehicles because it is more fuel-efficient and has a low combustion temperature. Due to the excess air condition, all the fuel is consumed and the combustion products are able to be diluted. Unlike stoichiometric burn engine, it does not have a catalyst exhaust after treatment technology. It relies on the advantage of low combustion temperature to minimize the hazardous exhaust emission in order to meet the emission standards.

The disadvantage of using lean burn engine is its inability to operate in a more stable condition when fuel composition changes. Due to the lean mixture of fuel and air, both the ignition delay and combustion duration increased during the combustion process, thus causing detonation to occur. Besides that, its power density will be reduced unless it is turbocharged and operates at higher energy level to ignite the lean mixture (Clark et al. 2003, p.2).

In order to cope with the variation in fuel composition, a special wide range or universal exhaust gas oxygen (UEGO) sensor and the control algorithm (Clark et al. 2003, p.1) are installed to the engine. With this installation, the lean burn engine has to operate under a closed loop control system to control the exhaust emission when fuel composition changes. Besides that, a knock sensor is installed in the feedback control system to protect the engine from knocking as well as to prolong the wear life of the engine.

Since CNG has knock resistance due to its high octane rating of 130, its application to both the lean burn engine and the stoichiometric burn engine is acceptable. However, lean burn engine with a closed loop control system will be more acceptable to be used in this project due to its more advanced features and its advantages.

2.6 Summary

The literature review written on various aspects of natural gas and the criterion to develop a suitable design for a dual fuel CNG-diesel engine conversion. Section on the background of natural gas provides the basic knowledge on its history of development, its composition and the general properties. After this section, the following sections written on the design criterion such as the CR and the types of engine to use for conversion by considering the aspects on fuel composition and the natural gas conversion process.

All the reviews will provide the required information related to the project methodology such as the piston modification, the effects on the heat and mass transfer system and prevention of knocking. By setting the correct proportion of natural gas composition and experimenting with different CR using various dimensions on the piston, the engine performance is evaluated. The evaluation is based on the mixing quality of the turbulence model in the combustion chamber when piston is compressed, the stream function, heat and mass transfer system, and temperature and pressure distribution using CFD simulation.

CHAPTER III

PISTON DESIGN WITH INSERTION OF PLATE

3.1 Introduction

3.1.1 Structure of Piston

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 3.1 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.

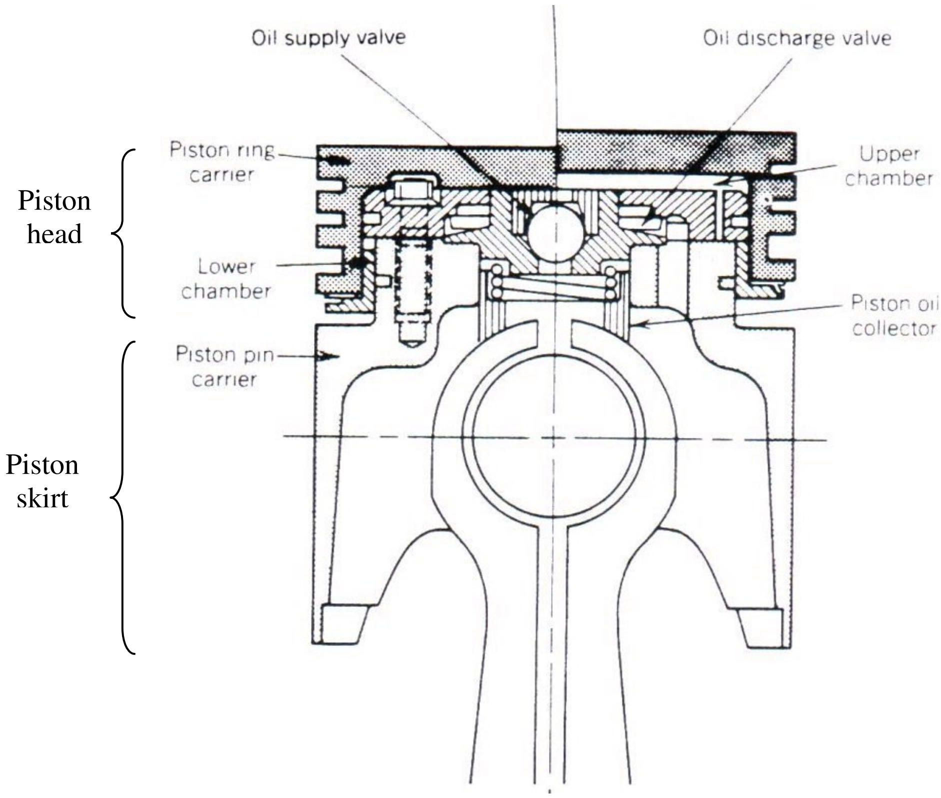


Figure 3.1 - Structure of Piston (Ferguson 1986, p. 46)

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, p.21). 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 3.2 in the following page. However, to simplify the problem, a flat piston head (Figure 3.2(1)) is used for the design purpose.

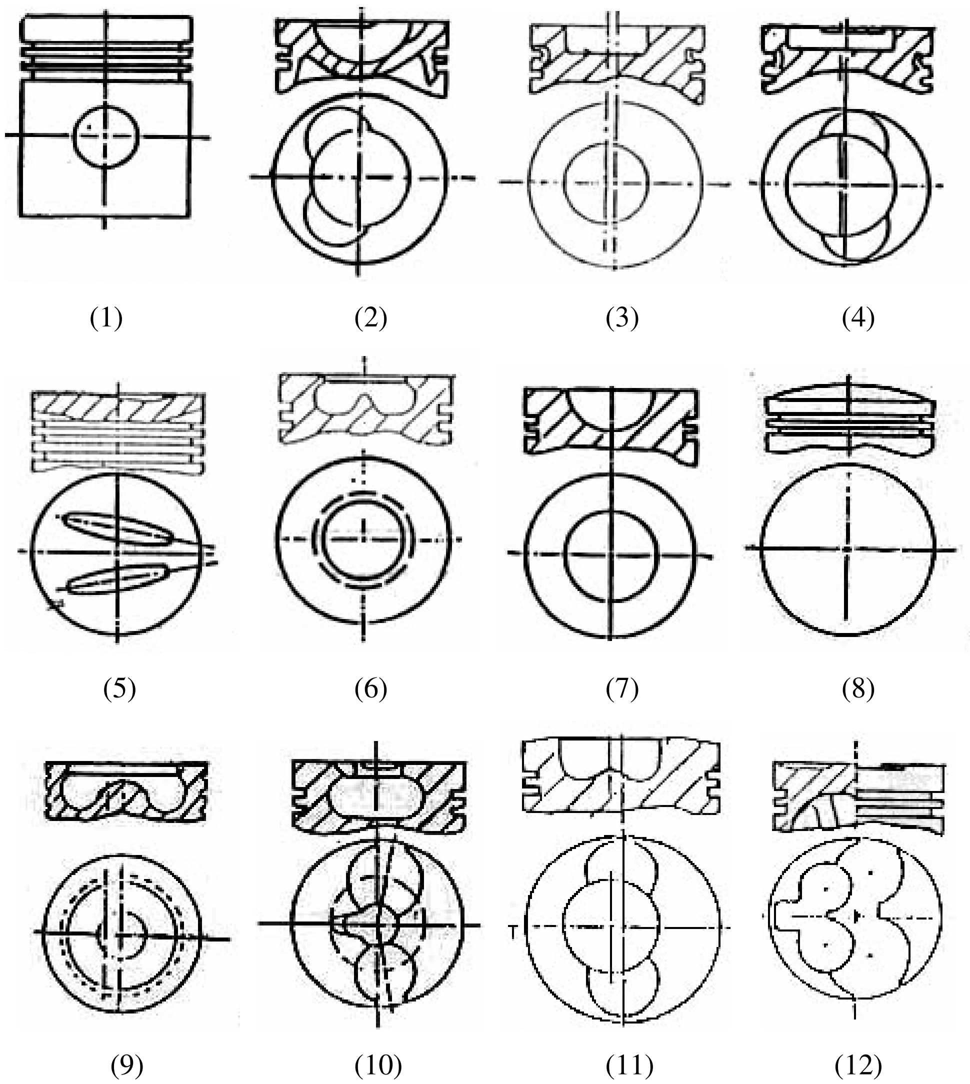


Figure 3.2 - Various Shapes of Piston Crown (KVR BMC Pistons Limited 2004 <http://www.kvrbmc.com/shape_pis.htm>)

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, p.16-17).

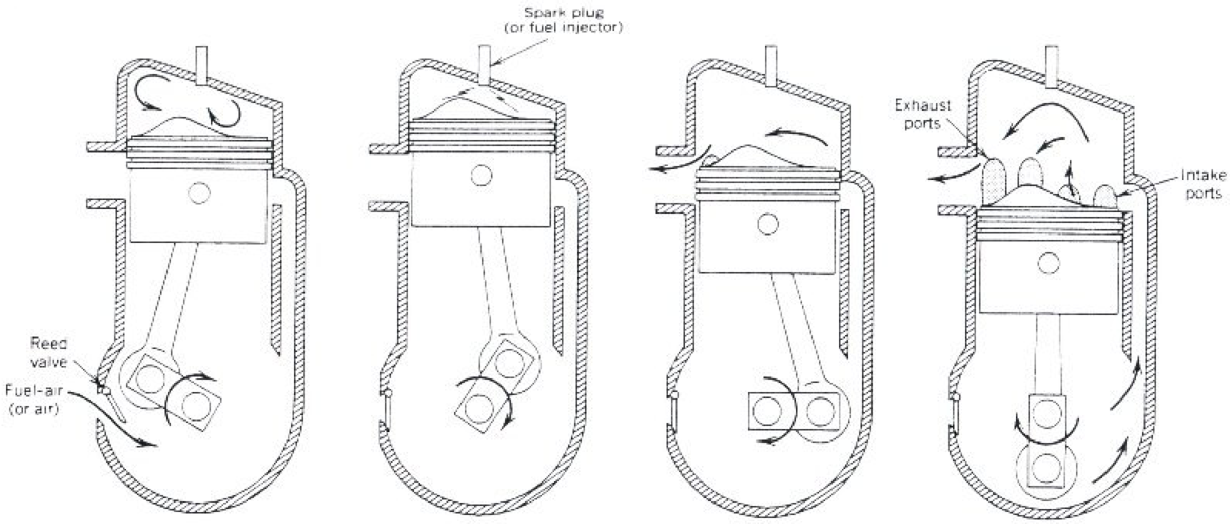
Next, the following section will explain on the mechanism of piston and its role in the engine. With a proper understanding on how the piston works in the engine, the design process will be made easier.

3.1.2 Mechanism of Piston in Internal Combustion Engine

Piston acts as an energy converter in an internal combustion engine. It is linked to a connecting rod and placed inside a cylinder, which is commonly known as combustion chamber. In any typical engine, the piston will transmit the gas pressure generated from the combustion process to the crank pin through the connecting rod.

From this mechanism, it converts the potential energy of the gases into kinetic energy that turns the crank shaft. Since, the piston moves up and down in the cylinder and rotates with the corresponding crank angle, its movement is said to be in a cyclic motion. Besides that, it also works as a movable gas-tight plug that seals the cylinder and to ensure that the combustion process operates in the cylinder itself (Ferguson 1986, p. 41).

The mechanism of piston in 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, p. 41). 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 illustrated in Figure 3.3 in the following page.



Induction stroke | Compression stroke | Expansion (Power) stroke | Exhaust stroke

Figure 3.3 - Four-Stroke Cycle (Ferguson 1986, p. 42)

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 at the TDC level. Next, the following section will explain on the effect of swirl and squish in the combustion chamber.

3.2 Swirl and Squish Action

Swirl is the rotational speed of the fluid within the cylinder about its axis that has the same angular momentum as the actual flow when normalized by the engine speed (Ferguson 1986, p.287). In diesel engine, swirl is important because it promotes rapid mixing between the injected fuel and air inducted through the intake valve. Therefore, if swirl does not occur in diesel engine, the engine is said to be quiescent and the intensity of the fuel injected into the cylinder is instead relied upon to mix the fuel and air (Ferguson 1986, p.287).

The swirl level at the end of compression stroke depends on the swirl generated during the intake stroke and the intensity of its amplification during the compression stroke. Swirl amplification in piston bowl occurs during the compression stroke is illustrated in Figure 3.4 in next page. The piston has a squish height, *h* of 0.05 inches (1.27mm) and CR of 10:1. The 75% squish bip means that the cross sectional area of the bowl-in piston is 25% of the piston area. The same principle applies to the 60% squish bip in the figure also (Ferguson 1986, p. 297).

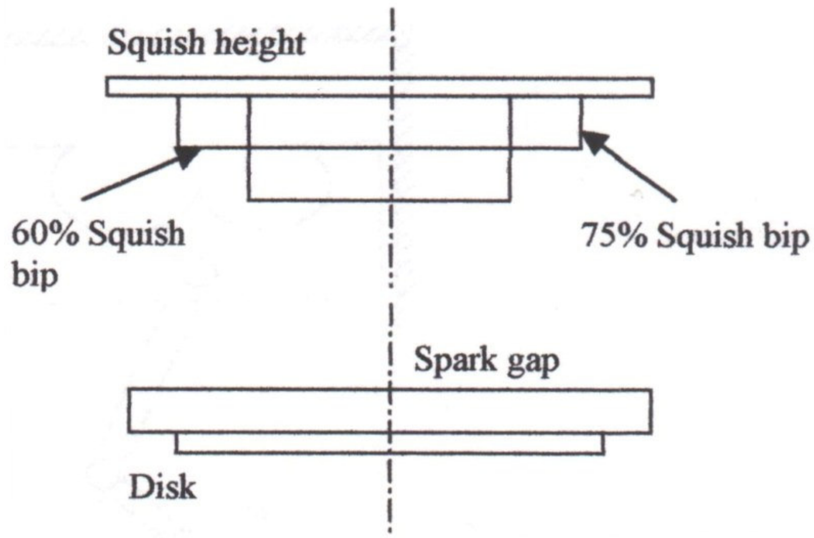


Figure 3.4 – Piston Cup for Swirl Amplification (Ferguson 1986, p. 297)

Since the original piston design for conventional diesel engine is a flat shape piston head, the swirl amplification still applies to the clearance volume of the piston when it is compressed adiabatically to the TDC. Although there is no bowl-in structure for the flat piston head, swirling of the mixture of fluid in the combustion chamber will still occur but might not be so significant compared to the bowl-in piston head.

By the way, swirl is proportional to the angular momentum but is inversely proportional to the moment of inertia (Ferguson 1986, p.298). However, the angular momentum within the cylinder is reduced or decayed by the fluid friction. It can only be increased during the intake stroke because it is convected in during that period. For other strokes, at any other time, the angular momentums will always decaying due to the fluid friction (Ferguson 1986, p.298).

Therefore, at the TDC, the swirl will increase and decrease at a short period of time due to these two reasons. In the case for moment of inertia, it goes to the minimum level at the TDC, which is dependent on the design of the piston clearance volume. At a constant CR, the larger the clearance volume, the greater is the change and the swirl amplification. Hence, by referring to Figure 3.4 again, the 75% squish bip will has a larger swirl than the 60% squish bip.

Besides that, there is another phenomenon, which occurs in the clearance volume of the piston in the combustion chamber called the squish motion. It is a fluid motion that occurs toward the end of compression stroke when a portion of the piston surfaces and the cylinder head approach closely to each other (Ferguson 1986, p.287). It is also commonly associated with direct-injection engines and can be applied to the flat piston head design by inserting a plate between the engine block and the piston head.

Usually, if the combustion chamber has a piston bowl located within the piston, a byproduct of the swirl amplification known as squish motion is used to amplify the swirl generated during the intake stroke (Ferguson 1986, p.299). Therefore, it is more significant to be seen in piston with bowl-in or groove geometry. For example, the squish motion that used to generate turbulence and swirl about a circumferential axis in piston bowl is shown in Figure 3.5 below.

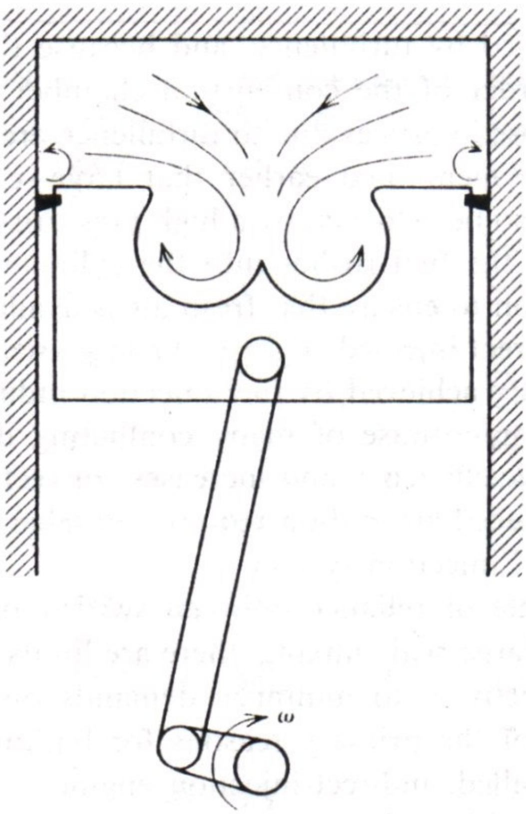


Figure 3.5 – Swirl Generated by Squish Motion about a Circumferential Axis in Piston Bowl (Ferguson 1986, p.299)

Besides that, the details on how the piston bowl can generate squish motion and swirl is illustrated in Figure 3.6. During the compression stroke, the density within the cylinder at any instant of time is almost uniform (Ferguson 1986, p.298). Therefore, the mass within zones 1, 2 and 3 is directly proportional to the volume in these zones at any time. When the piston moves upward to the TDC, zones 1 and 2 will get smaller while zone 3 remains the same. Hence, the mass in zones 1 and 2 must flow to zone 3.

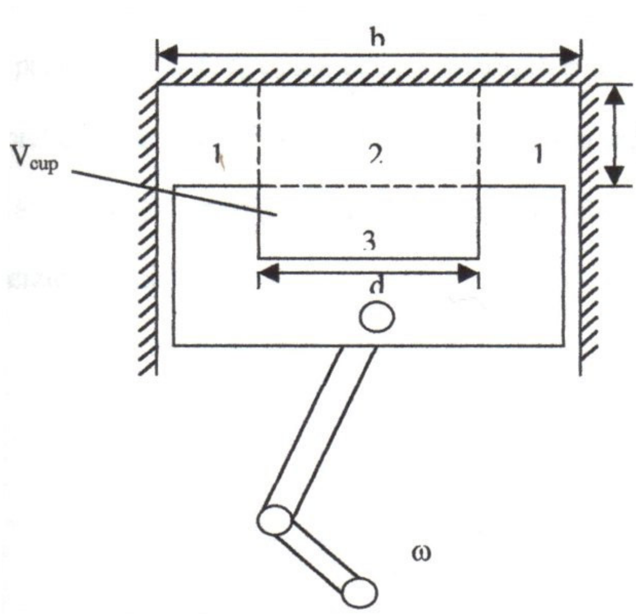


Figure 3.6 – Generation of Squish and Swirl in a Control Volume of a Piston Bowl (Ferguson 1986, p. 298)

The velocity of gas flows between zones 1 and 2 is called squish velocity and zone 1 is known as squish zone (Ferguson 1986, p. 298). Since the density is uniform, the squish velocity depends only on the cylinder geometry and the rotational speed. When equation of continuity is applied to the control volume in Figure 3.6, it yields an expression for the ratio of squish velocity to instantaneous piston speed as follows:

$$\frac{U_s}{U_p} = \frac{b^2 - d^2}{4dh}\frac{V_{bowl}}{V} \tag{3.1}$$

where

U_s = squish velocity

$U_p = \dfrac{dh}{dt}$ = instantaneous piston speed

b = piston bore

d = piston stroke

h = clearance height or squish height

$V_{bowl} = V_{cup}$ = volume of piston bowl/cup

$V = V_{bowl} + \dfrac{\pi}{4} b^2 h$ = Total clearance volume

Next, in the following section, knocking effect will be discussed since it is a very important characteristic to determine the compression ratio in the engine. By understanding the effects of knocking, a better design for the piston in the combustion chamber can be made to prevent it.

3.3 Knocking Effect

Knocking occurs when the fuel self-ignited before reaching the TDC due to high pressure and high temperature inside the combustion chamber during the compression stroke. It is not preferred and should be avoided because it can cause failure to the piston and damage the whole cylinder block in a short period of time.

It is important to consider the knocking effect on the CNG-diesel engine and understand the factors that promote this phenomenon. There are a few factors that need to be considered such as the octane rating and cetane number of the fuel, the equivalence ratio, the ignition delay period, compression ratio and other operating parameters (*CNG Engine Performance* 1995, p.2).

Octane rating is the ability of the fuel to resist detonation during the combustion and it is an important characteristic to determine the compression ratio. Fuel with higher octane rating can perform better without detonation and knocking in the combustion chamber when operating in a higher compression ratio engine. Therefore, CNG can be used in a diesel engine with high compression ratio because it has a high octane rating of 130 (Yusaf et al. 1996a, p.21).

Cetane number is the ability of the fuel to self-ignite. Therefore, the cetane number and the octane rating are inversely correlated. Fuel with low cetane number tends to have higher ignition temperature and vice-versa. For CNG, the ignition temperature is about 900 K, which is capable to resist the high pressure and high temperature in the combustion chamber to avoid knocking (*Compressed Natural Gas* 11 March 2004 <http://www.mckenziecorp.com/dehydration.htm>).

However, practically CNG is unable to operate in high pressure and temperature during the compression stroke. As the CNG enters the combustion chamber, it will be compressed up to the design compression ratio of the engine. If the compression ratio is very high, then the CNG will be self-ignited due to high pressure inside the combustion chamber i.e. compression effect (Yusof 2000, p.55). Therefore, it is expected that the compression ratio for the engine needs to be reduced to accommodate CNG in the dual fuel engine system.

Besides that, another factor to prevent knocking is the ignition delay period. It is important to make sure that the fuel injection time is longer than the ignition delay time. If the ignition delay time is longer than the fuel injection time for a fixed operating condition for the engine, there would be more fuel injected into the combustion chamber and accumulated there before the fuel ignites (Yusof 2000, p.54). Thus, knocking will occur due to large amount of fuel accumulated inside the high pressure and high temperature combustion chamber when being compressed.

Hence, by considering all the factors mentioned above, preventive measure can be taken to avoid such a phenomenon to occur. This is one of the objectives in this project that is to design a CNG-diesel engine without knocks.

3.4 Piston Design

The conventional diesel engine with a compression ratio (CR) of 22.9:1 is modified to accommodate CNG in diesel engine. Since the original diesel engine operates under a high compression ratio, the conversion to a dual fuel engine system might promote knocking in the combustion chamber due to self-ignition characteristic of CNG.

Although CNG has a high octane rating of 130, it will still self ignite when operates in a high pressure combustion chamber due to compression effect of the compression stroke. Therefore, the compression ratio of the engine needs to be modified and reduced to obtain the optimum performance for the dual fuel CNG-diesel engine. The compression ratio can be reduced by three different methods, which are listed below (Yusaf et al. 1996, p.21).

- Modifying the piston groove or bowl
- Modifying the length of the connecting rod
- Insertion of plate onto the piston head

The first method is usually constructed by milling the piston head to create a recessed bowl shape. The size of the bowl depends on the size of the piston. It is usually suitable for large piston because small piston with recessed bowl can cause thermal stress to build up in the piston head and piston skirt. This will cause failure to the piston. The second method is to reduce the length of the connecting rod. However, this method is very costly and complicated to be constructed. Improper design will cause vibration and thermal stress to build up in the piston (Yusaf et al. 1996, p.21).

The last method is chosen for the design of the piston in the combustion chamber to reduce the compression ratio. A plate with a thickness t is added between the piston head and the cylinder block and act as a seal between the engine block and the piston head (Yusaf et al. 1996, p.22). The shape of the plate will follow the shape of the top of the piston head. It is chosen because of its lower construction cost and easier to be built compared to the other two methods explained earlier. Besides that, the design is much simpler and does not require any complicated calculation. The calculation related to the implementation of plate on the piston head will be shown in Chapter 5.

When modifying the piston to reduce the compression ratio, it is important to make sure that the operation temperature and maximum pressure during the compression stroke are appropriate so that no pre-ignition occur in the combustion chamber. Besides that, the size and weight of the piston after modification should be the same as the original piston. This is to ensure that no vibration and inertia loading occur on the bearings in the piston. Lastly, the effects of the friction force between the piston and the cylinder block should be considered also (Yusaf et al. 1996, p.21).

In the next section, the details on the design criteria for the addition of plate onto the piston head to reduce the compression ratio are explained further.

3.5 Design Criteria for the Plate

The process to design a piston with an insertion of plate onto the piston head must follows a few considerations in order to accomplish the objectives of the project as mentioned earlier in Chapter 1. The considerations are listed below:

i. Role of piston in the internal combustion engine.
ii. Condition and parameters to be considered for the plate.
iii. The analytical methods used to study the combustion efficiency.
iv. Piston materials.
v. Software used for the modeling and simulation.

For part (i), the piston is used to compress the fluid within the cylinder until ignition occurs due to high pressure. The fuels used in this project are mixture of CNG and air; and air alone with diesel fuel acting as the igniter for both conditions. The results obtained from these two fuel conditions are used to determine whether the engine efficiency of the dual fuel condition is similar to the pure diesel-air fuel condition.

After the objectives of the project are determined, the conditions and parameters that are used for plate to be inserted are considered. Those parameters are the distribution of temperature, pressure and stream function through the piston wall and combustion chamber wall under normal load condition. However, there might be other load conditions to be used during the analysis for improvement of the initial results obtained.

Next, the types of analysis to be used in the modeling of the piston are divided into two methods. There are the steady state analysis and the transient analysis. The first one will be used when the model needs to consider all the mechanical parameters such as power output, torque, engine speed, mechanical load and thermal load. This analysis is further executed by following the Rapid Analysis of Products by Integrated Engine Routine (RAPIER) used in the FLUENT software (Yusof 2000, p.34).

This routine consists of three steps:

i. The proposed piston needs to consider the objectives of the design, the criteria to accomplish the objectives, constraints encountered, appropriate recommendations and the usage of standard CAD application.

ii. Application of loads to test the piston. For examples, the firing condition and non-firing condition. Firing condition is the loads exerted to the TDC by the piston motion and its stresses, temperature and displacement.

iii. Sub-modeling by continuously repeating the modeling process for any minor changes to improve the piston design.

For transient analysis, it focused on the possible failure of the piston to determine the appropriate design criteria for the improvement in the piston design. Usually, load conditions such as constant load and variable load are applied to the piston to compare both results for improvement. The application of load on the piston follows the steps shown in next page (Yusof 2000, p.35):

i. No load steady state condition.

ii. Linear ramp from no load condition to full load condition until the temperature stabilized to detect a transient response.

iii. Full load steady state condition.

iv. Linear ramp from full load condition to no load condition until the temperature stabilized to detect a transient response.

After executing tasks from part (i) to part (iii), selection of the most suitable material is necessary to make a piston. The criteria for a good quality of piston should have high quality in terms of strength, durability, thermal conductivity and wear resistance as well as lightness in weight. Therefore, aluminium is chosen as the most suitable material for the piston in this project.

Lastly, the computational fluid dynamics softwares to be used are GAMBIT version 2.1 and FLUENT version 6.1 for modeling and simulation work respectively. Besides these two softwares, there are other softwares such as ANSYS, CFD-ACE, Pro-Engineer and Solidworks to execute the tasks. However, these two softwares are chosen due to their compatibility and the excellence function of FLUENT in analyzing flows of fluid in various conditions such as combustion, reacting and non-reacting flows, turbulence model and etc.

It is important to follow the steps of the design process as mentioned above for accuracy and effectiveness of the product designed. These design steps can be proven by the accomplishment of piston produced using these steps such as the conventional standard piston, aluminum gallery piston, reentrant bowl piston, fiber-reinforced piston and the fiber-reinforced aluminum gallery piston (Yusof 2000, p. 36).

Since most of the design process is complicated, usually computer aided softwares are used for convenience, accuracy and economical. Therefore, the next section will guide the readers to the theoretical part of the software used, which is the FLUENT software using computational fluid dynamic method.

CHAPTER IV

COMPUTATIONAL FLUID DYNAMICS (CFD)

4.1 Introduction

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 the CFD analysis in FLUENT follows the simple steps below:

i. 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.

ii. Next, in FLUENT, the saved mesh or grid file of the model is read, checked and scaled for the required working unit.

iii. 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.

iv. The model is solved by setting the required parameters in the solution panel and then iterated for convergence.

v. Results can be obtained from the graphic display and report in FLUENT. Results can be displayed in terms of contour, velocity vector, particle track and path line. Any calculation required can be performed in FLUENT also.

vi. Finally, the results and all the data can be saved for future references by writing the files.

Next, the following section will discuss on the governing equations used in FLUENT when computing and analyzing the fluid flow behavior. All the equations will give the details on how the CFD works in order to simulate the result for certain problems.

4.2 Capability of FLUENT Solver

This software has various modeling capabilities that can be used in numerous kinds of analysis and application. Among its capabilities are listed below (*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.

- Inviscid, 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.

4.3 Governing Equations

The CFD methodology in FLUENT is using partial differential equations of flow variables to calculate and to simulate numerous kinds of analysis concerning the fluid flow. Among the flow variables that are commonly used in analysis are mass, momentum, energy, species concentration, quantities of turbulence and mixture fractions. Therefore, the governing equations to be used in this analysis are the conservation of mass, momentum, energy and turbulent equations.

4.3.1 Mass Conservation Equation

The continuity equation or the mass conservation equation for any fluid flow is expressed as (*FLUENT Manual – Mass Conservation Equation* 2004):

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = \dot{m} \tag{4.1}$$

where

ρ = fluid density

u_j = the j^{th} Cartesian component of the instantaneous velocity

$\dot{m}$ = the rate of mass of the object generated in the system

This equation is valid for the incompressible and compressible flows. Moreover, the rate of mass generated in the system, $\dot{m}$ can be defined as the mass added to the continuous phase from the dispersed second phase such as the vaporization of liquid droplets and any other user-defined sources.

4.3.2 Momentum Conservation Equation

The conservation of momentum in *i* direction for an inertial reference frame can be explained as (*FLUENT Manual – Momentum Conservation Equations* 2004):

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i \tag{4.2}$$

where

ρ = fluid density

u_i & u_j = the i^{th} and j^{th} Cartesian components of the instantaneous velocity

p = static pressure

τ_{ij} = stress tensor

ρg_i = gravitational body force

F_i = external body force from interaction with the dispersed phase in *i* direction

The stress tensor in Equation 4.2 is given as (*FLUENT Manual – Momentum Conservation Equations* 2004):

$$\tau_{ij} = \mu(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) - \frac{2}{3}\mu(\frac{\partial u_k}{\partial x_k})\delta_{ij} \qquad (4.3)$$

where

μ = fluid dynamic viscosity

δ_{ij} = Kronecker delta

Note that the second term on the right hand side of Equation 4.3 describes the effect of volume dilation. By substituting Equation 4.3 into Equation 4.2, another equation is produced that is the Navier-Stokes Equation as below (*FLUENT Manual – Momentum Conservation Equations* 2004):

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\{\mu(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) - \frac{2}{3}\mu(\frac{\partial u_k}{\partial x_k})\delta_{ij}\} + \rho g_i + F_i \qquad (4.4)$$

4.3.3 Energy Conservation Equation

The partial differential equation for energy conservation is expressed as below (*FLUENT Manual – Energy Equation* 2004):

$$\frac{\partial}{\partial t}(\rho e) + \frac{\partial}{\partial x_i}[u_i(\rho e + p)] = \frac{\partial}{\partial x_i}[k_{eff}\frac{\partial T}{\partial x_i} - \sum_{j'} h_{j'} J_{j'} + u_j(\tau_{ij})_{eff}] + S_h \qquad (4.5)$$

where

k_{eff} = effective conductivity

= $k + k_t$ (where k_t = turbulent thermal conductivity)

$J_{j'}$ = diffusion flux of species *j'*

S_h = additional volumetric heat sources i.e. heat of chemical reaction

h = sensible enthalpy

e = specific total energy

The first three terms on the right-hand side of Equation 4.5 represent the energy transfer due to conduction, species diffusion and viscous dissipation respectively.

From Equation 4.5 also, sensible enthalpy, h and specific total energy, e are defined as below:

$$e = h - \frac{p}{\rho} + \frac{u_i^2}{2} \tag{4.6}$$

Sensible enthalpy for ideal gas is defined as:

$$h = \sum_{j'} m_{j'} h_{j'} \tag{4.7}$$

Sensible enthalpy for incompressible flow is defined as:

$$h = \sum_{j'} m_{j'} h_{j'} + \frac{p}{\rho} \tag{4.8}$$

where

$m_{j'}$ = mass fraction of species j'

$h_{j'} = \int_{Tref}^{T} c_{p,j'} \, dT$ with $T_{ref} = 298.15\text{K}$

4.3.4 Turbulent Equation

Turbulent flow is characterized by its fluctuating velocity fields. They mix the transported quantities such as momentum, energy and species concentration to fluctuate as well. For FLUENT, the turbulent flow can be modeled in several ways depending on the problems encountered. Following are the types of turbulent model available in FLUENT (*FLUENT Manual – Turbulent Model* 2004):

- Spalart-Allmaras model

- Standard k- ε model

- Renormalization-group (RNG) k- ε model

- Realizable k-ε model

- Reynolds stress model (RSM)

- Large eddy simulation (LES) model

For this project, the simulation is using the Standard k- ε model as the preferred turbulent model. Basically, this model is using two model transport equations in partial differential form to govern the transport of turbulent kinetic energy, *k* and its dissipation rate, ε. It is assuming that the flow in the system is fully turbulent and the effects of molecular viscosity are negligible. Therefore, the Standard k-ε model is only valid for fully turbulent flows.

The transport equations for the turbulent kinetic energy, *k* and its dissipation rate, ε are defined below (*FLUENT Manual – Transport Equations for the Standard k-epsilon Model* 2004):

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_i}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_i}] + G_k + G_b - \rho\varepsilon - Y_M \qquad (4.9)$$

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i}[(\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_i}] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad (4.10)$$

where

G_k = generation of turbulent kinetic energy due to mean velocity gradients

$$= -\rho\overline{u_i' u_j'}\frac{\partial u_j}{\partial x_i} \qquad (4.11)$$

G_b = generation of turbulent kinetic energy due to buoyancy

$$= \beta g_i \frac{\mu_t}{\Pr_t} \frac{\partial T}{\partial x_i} \quad (4.12)$$

For ideal gas,

$$G_b = - g_i \frac{\mu_t}{\rho \Pr_t} \frac{\partial \rho}{\partial x_i} \quad (4.13)$$

$\Pr_t$ = turbulent Prandtl number for energy

= 0.85 (default)

$\beta = -\frac{1}{\rho}(\frac{\partial \rho}{\partial T})_p$

Y_M = fluctuating dilatation in compressible turbulence to the overall dissipation rate

$$= \rho \varepsilon 2 M_t^2 \quad (4.15)$$

$M_t = \sqrt{\frac{k}{a^2}}$ is the turbulent Mach number (4.16)

a = the speed of sound

$C_{1\varepsilon}$ & $C_{2\varepsilon}$ = constant

= 1.44 & 1.92 by default respectively

$$C_{3\varepsilon} = \tanh\left|\frac{v}{u}\right| \quad (4.17)$$

v = the component of the flow velocity parallel to the gravitational vector.

u = the component of the flow velocity perpendicular to the gravitational vector

σ_k & σ_ε = turbulent Prandtl numbers for k and ε respectively

= 1.0 & 1.3 by default respectively

4.4 FLUENT Solver

There are two numerical methods to be chosen in the FLUENT solver to solve the governing integral equations such as conservation of mass, momentum, energy and other scalars like turbulence and chemical species. They are the segregated solver and the coupled solver (*FLUENT Manual – Choosing the Solver Formulation* 2004).

The segregated solver is chosen as the most appropriate solver for this project because it operates by solving the governing equation sequentially until the solution converged. The solver will iterate the solution loop according to the user specification of the number of iterations to be performed in order to get the final solution, which will converge at the end of the iteration. The steps on the solution loop are illustrated below (*FLUENT Manual – Segregated Solution Method* 2004).

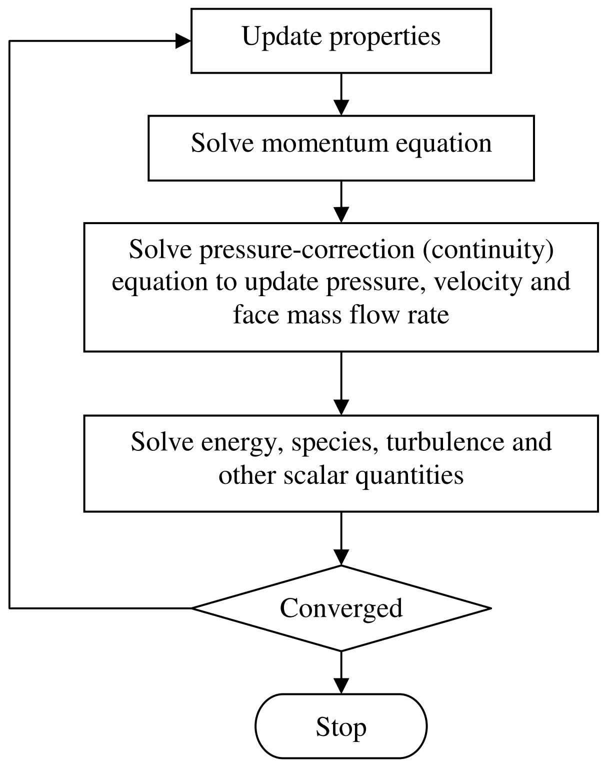


Figure 4.1 – Segregated Solution Loop

(*FLUENT Manual – Segregated Solution Method* 2004)

As for the coupled solver, this method solves all the continuity equations simultaneously or coupled together. The procedure is similar to the segregated method, where several iterations are solved for the coupled governing equations before a converged solution is obtained.

As mentioned earlier, segregated solution method is chosen for the simulation task in this project, therefore next section will discuss on the discretization method used for this solver.

4.5 Discretization Method

FLUENT uses a control-volume-based technique as the discretization method to convert the governing equations to algebraic equations so that the solution can be solved numerically in the segregated solver. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis. The discrete quantities are usually the mass, momentum, energy, turbulence and other scalars (*FLUENT Manual – Discretization* 2004).

The discretization of the governing equations is applied to each control volume or cell in the computational domain. It can be illustrated most easily by considering the steady-state conservation equation for transport of a scalar quantity Φ, which is written in integral form for an arbitrary control volume V as follows (*FLUENT Manual – Discretization* 2004):

$$\oint \rho \Phi v.dA = \oint \Gamma_{\Phi} \nabla \Phi .dA + \int_{v} S_{\Phi} dV \qquad (4.18)$$

where

ρ	= density
v	= velocity vector
A	= surface area
Γ_{Φ}	= diffusion coefficient for Φ
$\nabla\Phi$	= gradient of Φ
S_{Φ}	= source of Φ per unit volume

Discretization of the Equation 4.18 for a given cell is given in the following page (*FLUENT Manual – Discretization* 2004):

$$\sum_{f}^{Nfaces} v_f \Phi_f A_f = \sum_{f}^{Nfaces} \Gamma_\Phi (\nabla\Phi)_n A_f + S_\Phi V \qquad (4.19)$$

where

N_{faces}	= number of faces enclosing cell
Φ_f	= value of Φ convected through face f
v_f	= mass flux through the face
A_f	= area of face f, |A|
$(\nabla\Phi)_n$	= magnitude of $(\nabla\Phi)$ normal to face f
V	= cell volume

4.6 Solution Method

There are three methods use for to solve the problems modeled in FLUENT, which are the SIMPLE, SIMPLEC and PISO methods (*FLUENT Manual – SIMPLE, SIMPLEC and PISO).* The solution for the problems in this project is based on the SIMPLEC method.

The SIMPLE and SIMPLEC algorithms use a relationship between velocity and pressure corrections to enforce mass conservation in order to obtain the pressure field. For PISO algorithm used with neighbor correction, it is highly recommended for all transient flow calculations. It allows you to use a larger time step, as well as an under-relaxation factor of 1.0 for both momentum and pressure (*FLUENT Manual – SIMPLE, SIMPLEC and PISO* 2004).

SIMPLEC means SIMPLE-Consistent is preferred over the SIMPLE method because it can accelerate convergence in problems where pressure-velocity coupling is the main deterrent to obtaining a solution. Besides that, it can use a higher value for under-relaxation factor to get a faster convergence for solution, which the SIMPLE method is not able to do (*FLUENT Manual – SIMPLE & SIMPLEC* 2004).

It is especially useful for complicated flows involving turbulence and/or additional physical models because the SIMPLEC method can improve convergence when it is being limited by the pressure-velocity coupling. Therefore, the turbulent model used for this project should be solved by using the SIMPLEC method.

4.7 Phenomena Simulated

4.7.1 Turbulent Flow

As mentioned earlier, the problem simulated in this project is a turbulent model. Therefore, the model requires additional transport equations, which are k and ε transport equations. These two equations are used to predict the turbulent kinetic viscosity in regions of flow separation and stagnation.

The accuracy in specifying the turbulent characteristic of inlet flows requires three important quantities namely turbulent kinetic energy, k, its dissipation rate, ε and the length scale, L. Usually, the inlet diameter or height of the inlet is used to specify the length scale, L for internal flow. Therefore, if the dissipation rate, ε is unknown, the length scale, L can be the substitute for the specification of turbulent characteristic (*FLUENT Manual – Turbulent Models* 2004).

The turbulent kinetic energy, k can be calculated using the formula below (Yusof 2000, p.48):

$$k = \frac{1}{2}(u^2 + v^2 + w^2) \tag{4.20}$$

where

u, v & w = perturbation velocities about the mean velocity in X, Y and Z for Cartesian coordinate system.

The turbulent dissipation rate, ε can be determined using the following formula (Yusof 2000, p.49):

$$\varepsilon = \frac{C_\mu^{3/4} k^{3/2}}{0.014L} \quad (4.21)$$

where

C_μ = constant, by default is 0.09

k = turbulent kinetic energy

L = length scale (can be inlet height or diameter)

This formula assumed that the length scale, L for the turbulent eddies is approximately 0.3% of the size of the inlet.

4.7.2 Dynamic Mesh

The dynamic mesh model in FLUENT 6.1 is used to model flows where the shape of the domain is changing with time due to motion on the domain boundaries. The motion can be prescribed motion or unprescribed motion. Prescribe motion is set by the user to move according to the linear and angular velocities about the center of gravity of a solid body with time. As for unprescribed motion, it is not defined by the user; hence the linear and angular velocities are calculated from the force balance on a solid body (*FLUENT 6.1 User Guide – Chapter 9:Modelling Flows in Moving and Deforming Zones* 2004, p. 9-44).

Dynamic mesh can be generated by initiate a proper mesh for the model and describing the motion of the moving zones. If there are moving and non-moving regions exist in the model, it is important to identify these regions and set up any sliding interfaces such as valves opening and closing. For the purpose of this project, the model simulated is based on the in-cylinder flow motion that relies on the crank angle and the piston stroke.

The dynamic mesh model follows the integral form of the conservation equation for a general scalar, Φ on an arbitrary control volume, V is expressed as (*FLUENT 6.1 User Guide – Chapter 9: Modelling Flows in Moving and Deforming Zones* 2004, p. 9-45):

$$\frac{d}{dt}\int_V \rho\Phi dV + \int_{dV} \rho\Phi(\vec{u} - \vec{u}_g).d\vec{A} = \int_{dV} \Gamma\nabla\Phi.d\vec{A} + \int_V S_{\Phi} dV \tag{4.22}$$

where

ρ	= fluid density
$\vec{u}$	= flow velocity vector
$\vec{u}_g$	= grid velocity of the moving mesh
Γ	= diffusion coefficient
S_Φ	= source term of Φ
dV	= boundary of the control volume

The time derivative in the first term of equation 4.22 can be re-written using the first order backward difference formula below (*FLUENT 6.1 User Guide – Chapter 9:Modelling Flows in Moving and Deforming Zones* 2004, p. 9-45):

$$\frac{d}{dt}\int_V \rho\Phi dV = \frac{(\rho\Phi V)^{n+1} - (\rho\Phi V)^n}{\Delta t} \tag{4.23}$$

where

$n+1$ and n are the next time level and current time respectively.

The $n+1$th time level for volume V^{n+1} is computed from (*FLUENT 6.1 User Guide – Chapter 9:Modelling Flows in Moving and Deforming Zones* 2004, p. 9-45):

$$V^{n+1} = V^n + \frac{dV}{dt}\Delta t \tag{4.24}$$

where

$\frac{dV}{dt}$ = the volume time derivative of the control volume, which can be written as in the following form:

$$\frac{dV}{dt} = \int_{dV} \vec{u}_g.d\vec{A} = \sum_j^{nf} \vec{u}_{g,j}.\vec{A}_j \tag{4.25}$$

where

n_f = number of faces of the control volume

$\vec{A}_j$ = j face area

and the dot product of $\vec{u}_{g,j}.\vec{A}_j = \dfrac{\delta V_j}{\Delta t}$

δV_j = volume swept out by control volume face j over the time step Δt.

Generally, there are three methods to update the mesh when simulating the moving or dynamic mesh (*FLUENT 6.1 User Guide – Chapter 9: Modelling Flows in Moving and Deforming Zones* 2004, p. 9-46. There are spring based smoothing method, dynamic layering method and local remeshing method. In this case, all the methods are turned on so that the meshes in the model are updated according to the type of mesh topology used in the model.

Usually, spring based method is used for non-tetrahedral cell zones in 3D and non-triangular cell zones in 2D that are moving or deforming. For dynamic layering method, it is used to update moving cell zones and to split or merge any cell zones adjacent to the moving boundary in the forms of quadrilateral in 2D and hexahedral in 3D (*FLUENT 6.1 User Guide – Chapter 9: Modelling Flows in Moving and Deforming Zones* 2004, p. 9-84).

Lastly, local remeshing method is only applicable on cell zones with triangular or tetrahedral mesh when the moving and deforming boundary violates the usage of the other two methods due to larger boundary displacement compared to the local cell sizes.

4.7.3 Non-Reacting Flow

It is assumed that there are no reaction occurs between the mixture of the CNG and air in the combustion chamber. The simulation is based on the time when compression stroke occurs in the combustion chamber. The piston is simulated in such way that the piston head reached the TDC using the dynamic mesh method mentioned earlier.

Hence, the behavior of the mixing between the CNG and air is monitored just before reaching TDC to detect any possible knocks.

4.7.4 Incompressible Flow

The flow modeled in this problem is an incompressible flow, which means the density of the fluids is not constant. Hence, it activates the pressure-correction equation and the density for each of the fluid can be calculated based on the pressure and/or species mass fractions, where the mass fractions are the values entered as the initialization condition.

CHAPTER V

DESIGN METHODOLOGY

5.1 Engine Specification

The simulation work started with the original diesel engine. Later on, modification is made to find the optimum compression ratio. The following are the engine specification of the model that is used for the modeling and simulation purposes.

- Model : Direct injection diesel engine
- Type : Four-stroke
- No. of cylinder : Four cylinders (but only one is considered for this project)
- Bore : 60 mm
- Stroke : 109.5 mm
- Compression Ratio : 22.9:1
- Piston clearance : 5 mm
- Connecting rod : 160 mm
- Engine speed : 1000 RPM (minimum) - 3000 RPM (maximum)
- Wall temperature : 300 K

In the following section, the modification of the piston to change the compression ratio will be explained further.

5.2 Piston Modification

The model simulated for this project is based on a 2-D control volume of a combustion chamber when the piston is at the BDC. The model is simulated during the compression stroke where the dynamic mesh is used to enable the mesh of the model to move for every subsequent crank angle of 1° up to 180°.

The original model used in the simulation is illustrated in Figure 5.1, which shows the dimension also.

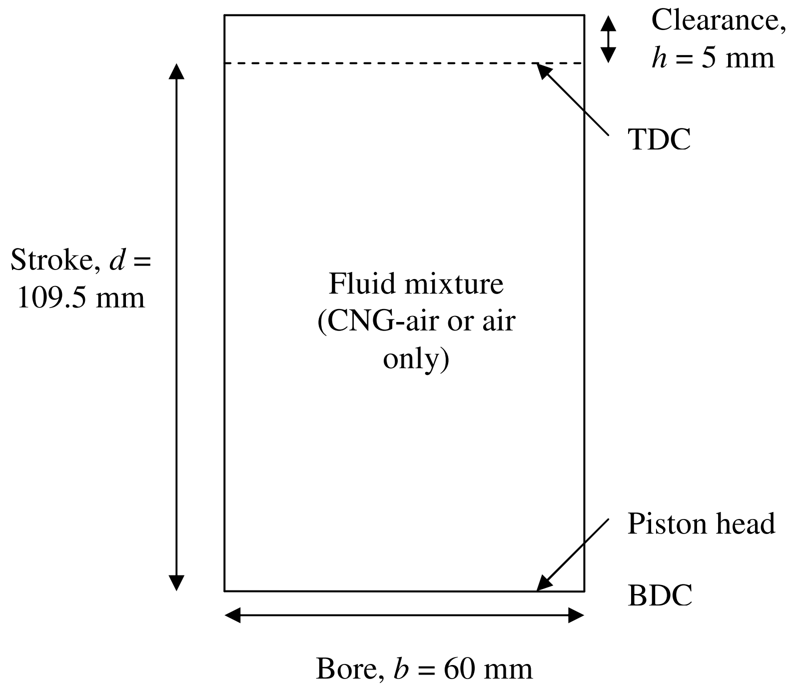


Figure 5.1 – Simulation Model

Initially, the original design for the diesel engine at compression ratio of 22.9:1 is simulated to monitor the pressure distribution, temperature distribution and stream function of the mixture of CNG and air. If the result shows that knocks occurred when the mixture of CNG and air is used for a diesel engine with compression ratio of 22.9:1, modification to the piston has to be made.

Compression ratio is reduced to monitor the pressure distribution and the temperature distribution so that knocking will not occur. If the maximum temperature and pressure obtained in the simulation are higher than the mixture's ignition temperature and pressure, knocking is said to occur in the combustion chamber during the compression stroke.

Therefore, the point where knocking occurred in the temperature and pressure distribution has to be recorded, to ensure that the following design values of compression ratio do not have knocking. The compression ratio can be reduced by adding a plate between the piston head and the cylinder block as introduced earlier in Chapter 3, which also shown in Figure 5.2.

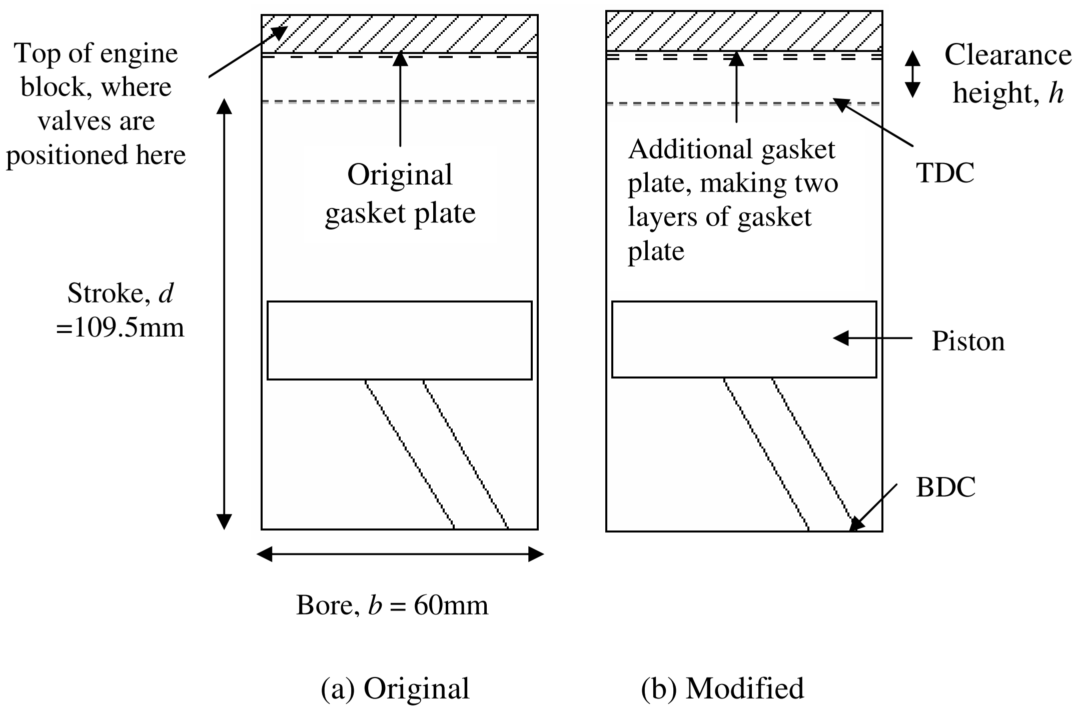


Figure 5.2 – Additional Plate to Increase Clearance Volume

This plate is actually a gasket plate that normally exists in every engine block and acts as a seal between the piston head and the engine block. Therefore, additional gasket plate is added for the purpose of this project to increase the clearance volume, which is shown in Figure 5.2. The plate thickness used to reduce the compression ratio from 22.9:1 to 19.3:1 and 16.6:1 is 1 mm and 2 mm respectively. Calculation to obtain the compression ratio for each case will be explained in the following section.

Besides considering the pressure and temperature distribution, the stream function of the mixture of CNG and air needs to be analyzed and simulated also. Therefore, the temperature of the wall is varied between 27˚ C (300 K) to 150˚C (423 K) to see the differences in temperature distribution in the combustion chamber as well as the stream function.

In the next section, the theoretical calculation to find the compression ratio for different piston design and mass fraction for each species are performed.

5.3 Theoretical Calculation

5.3.1 Compression Ratio for Different Piston Design

The addition of plate on the piston head will reduce the compression ratio of the diesel engine. Firstly, plate thickness of 1 mm is added to the piston head, which increases the clearance volume and thus reduces the compression ratio.

Compression Ratio (CR) is defined as the total volume when piston at BDC divided by the total volume when piston at BDC. Hence,

$$\text{CR} = \frac{V_S + V_C}{V_C} \tag{5.1}$$

where

V_S = Swept volume

V_C = Clearance volume

Therefore, the original diesel engine compression ratio is calculated as below:

Swept Volume, $V_S = \frac{\pi}{4} b^2 S$

$$= \frac{\pi}{4} \times 60^2 \times 109.5$$

$$= 309603.96 \text{ mm}^3$$

Clearance Volume, $V_C = \frac{\pi}{4} b^2 H$

$$= \frac{\pi}{4} \times 60^2 \times 5$$

$$= 14137.17 \text{ mm}^3$$

$$\text{CR} = \frac{V_S + V_C}{V_C}$$

$$= \frac{309603.96 + 14137.17}{14137.17}$$

$$\underline{= 22.9 : 1}$$

Addition of plate creates an extra volume, $V_{Plate} = \frac{\pi}{4} b^2 t$

where

b	= bore diameter (mm)
d	= stroke (mm)
h	= clearance height (mm)
t	= plate thickness (mm)

The new CR = $\frac{V_S + V_C + V_{Plate}}{V_C + V_{Plate}}$ (5.2)

Therefore, the calculation for the compression ratio when a plate with thickness, t=1 mm is added onto the piston head is shown next page:

Volume of the plate, $V_{Plate} = \frac{\pi}{4} B^2 t$

$$= \frac{\pi}{4} \times 60^2 \times 1$$

$$= 2827.43 \text{ mm}^3$$

$$\text{CR}_{\text{New1}} = \frac{V_S + V_C + V_{Plate}}{V_C + V_{Plate}}$$

$$\text{CR}_{\text{New1}} = \frac{309603.96 + 14137.17 + 2827.43}{14137.17 + 2827.43}$$

$$= 19.25 : 1$$

$$\underline{\approx 19.3 : 1}$$

When plate thickness is 2 mm, the compression ratio is:

Volume of the plate, $V_{Plate} = \frac{\pi}{4} \times 60^2 \times 2$

$$= 5654.87 \text{ mm}^3$$

$$\text{CR}_{\text{New2}} = \frac{309603.96 + 14137.17 + 5654.87}{14137.17 + 5654.87}$$

$$\underline{= 16.6 : 1}$$

5.3.2 Mass Fraction of the CNG-Air Mixture

As mentioned in Chapter 2, the composition of the CNG consists of gases such as methane, ethane, propane, iso-butane, hexane and other gases. To simplify the problem, CNG will be based on three main gases namely methane (CH_4), ethane (C_2H_6) and propane (C_3H_6). By approximation, CNG consists of 85% of methane (CH_4), 13% of ethane (C_2H_6) and 2% of propane (C_3H_8) by volume. As for the air, it will consist of 21% of oxygen (O_2) and 78% of nitrogen (N_2) by volume.

Since there is no combustion process in the simulation and to simplify the problem, equivalence ratio, $\phi=1$ is assumed for the mixture of CNG and air. Hence, there are approximately 0.15 kmoles of C_2H_6 and 0.02 kmoles of C_3H_8 for every 1 kmole of CH_4 in CNG. For air, there are about 3.76 kmoles of N_2 for every 1 kmole of O_2 as calculated from the percentage given above. The number of mole for CNG and air that enter the combustion chamber is expressed as below:

$$CH_4 + 0.15C_2H_6 + 0.02C_3H_8 + O_2 + 3.76N_2$$

The calculation for the mass fraction of each species in the combustion chamber is shown below:

- Mass Fraction of methane (CH_4), Y_{CH4}

$$Y_{CH4} = \frac{N_{CH4}MW_{CH4}}{N_{CH4}MW_{CH4} + N_{C2H6}MW_{C2H6} + N_{C3H8}MW_{C3H8} + N_{O2}MW_{O2} + N_{N2}MW_{N2}}$$

$$= \frac{1(16.043)}{1(16.043) + 0.15(30.069) + 0.02(44.096) + 1(31.999) + 3.76(28.013)}$$

$$= 0.1011$$

- Mass Fraction of ethane (C_2H_6), Y_{C2H6}

$$Y_{C2H6} = \frac{N_{C2H6}MW_{C2H6}}{N_{CH4}MW_{CH4} + N_{C2H6}MW_{C2H6} + N_{C3H8}MW_{C3H8} + N_{O2}MW_{O2} + N_{N2}MW_{N2}}$$

$$= \frac{0.15(30.069)}{1(16.043) + 0.15(30.069) + 0.02(44.096) + 1(31.999) + 3.76(28.013)}$$

$$= 0.0284$$

- Mass Fraction of propane (C_3H_8), Y_{C3H8}

$$Y_{C3H8} = \frac{N_{C3H8}MW_{C3H8}}{N_{CH4}MW_{CH4} + N_{C2H6}MW_{C2H6} + N_{C3H8}MW_{C3H8} + N_{O2}MW_{O2} + N_{N2}MW_{N2}}$$

$$= \frac{0.02(44.096)}{1(16.043) + 0.15(30.069) + 0.02(44.096) + 1(31.999) + 3.76(28.013)}$$

$$= 0.0055$$

- Mass Fraction of oxygen (O_2), Y_{O2}

$$Y_{O2} = \frac{N_{O2}MW_{O2}}{N_{CH4}MW_{CH4} + N_{C2H6}MW_{C2H6} + N_{C3H8}MW_{C3H8} + N_{O2}MW_{O2} + N_{N2}MW_{N2}}$$

$$= \frac{1(31.999)}{1(16.043) + 0.15(30.069) + 0.02(44.096) + 1(31.999) + 3.76(28.013)}$$

$$= 0.2016$$

- Mass Fraction of nitrogen (N_2), Y_{N2}

$$Y_{N2} = \frac{N_{N2}MW_{N2}}{N_{CH4}MW_{CH4} + N_{C2H6}MW_{C2H6} + N_{C3H8}MW_{C3H8} + N_{O2}MW_{O2} + N_{N2}MW_{N2}}$$

$$= \frac{3.76(28.013)}{1(16.043) + 0.15(30.069) + 0.02(44.096) + 1(31.999) + 3.76(28.013)}$$

$$= 0.6634$$

5.4 Steps for Simulation in FLUENT

The steps taken to simulate the pressure distribution, temperature distribution, stream function and heat transfer in the combustion chamber during the compression stroke are listed below.

1. Read and check the files.

    i. Read the mesh file of the model.

    ii. Check the grid of the model for errors.

    iii. Set the scale of the model according to the unit used for the dimensions. For example, change the default unit in meter to millimeter.

    iv. If necessary, smooth and swap the mesh of the model.

2. Define the model.

    i. Next, define the model's solver, which is an unsteady segregate solver that is used for dynamic meshes.

    ii. Enable the energy options to determine the temperature distribution and heat transfer.

    iii. Select the $k-\varepsilon$ equations in viscous model to model turbulence.

    iv. Specify the species used for the simulation such as fluid, solid or mixture. In this case, the model consists of a mixture. Therefore, the species in the mixture must be defined, which are methane, ethane, propane, oxygen and nitrogen.

v. Then, the properties of each of the species must be defined or used the default values in FLUENT.

vi. Set the operating condition. In this case, the operating condition is set to floating operating pressure, so that the pressure will change accordingly with the solutions.

vii. Set the boundary condition. For example, the temperature of the wall boundary is set to 300 K for the light load condition, which is the ambient temperature also.

3. Define the dynamic mesh.

   i. Next, the operating parameters for the dynamic mesh model are defined. As mentioned in Chapter 4, the dynamic mesh is using all the three methods to update the meshes in order to generate the moving mesh. Therefore, the default values for these three methods are used and only the parameters such as starting crank angle, piston stroke, crank shaft speed and connecting rod length need to be specified in the in-cylinder model.

   ii. Then, the zones of the dynamic mesh need to be defined according to the mesh topology. For example, the piston head is defined as a rigid body while the mixture inside the combustion chamber is defined as a deforming body. If the boundary condition is not defined, the dynamic mesh will automatically define them as stationary body.

4. Saving the file.

   i. After specifying all the parameters, the file is saved by writing the file and save the file for a case and data file under a file name.

ii. To read the file or run the file again, go to the FILE on the toolbar of FLUENT and scroll down to READ and select ‘Case and Data’ to run the file created earlier.

5. Initialize the solution.

   i. Specify the zone for the solution initialization and defined all the appropriate values for parameters such as the mass fraction of each species, temperature, pressure and $k-\varepsilon$ turbulent equations. Some of the unknown parameters can use the default values also.

   ii. After that, iterate for about 180 times that where the time step for the iteration is equal to crank angle.

6. Obtain the graphical solution.

   i. There are a few options to display the results. The results can be displayed in contour, vectors or path lines form that can only displayed the diagram of the combustion chamber at the end of the compression stroke, which is the clearance volume.

   ii. Else, the solution can be obtained in these forms also but with animation. The animation will show the changes every time the piston head compressed up to the TDC at a rate of 1 time step, which is equals to 1° of crank angle. This only applies to solution with dynamic mesh.

   iii. Besides that, the results can be plotted by monitoring the surface of the model according to the zones. Overall, all the zones are selected to displayed the whole meshes. For example, if a plot of pressure versus crank angle is required, then the plot is defined to display pressure with respect to crank angle and save the graphical file under a file name.

7. Lastly the FLUENT file is saved again as a case and data files so that they can be retrieved again. This will be the end of the simulation process by selecting exit to close the program.

CHAPTER VI

RESULT AND DISCUSSION

6.1 Introduction

In this chapter, the results from the modeling and CFD simulation using FLUENT software are shown and discussed. Results are shown in term of graphs for the simulation results for pressure distribution, temperature distribution and stream function.

Firstly, the graphs for pressure distribution, temperature distribution and stream function are plotted against the time step for various cases. Note that each increment of a time step is equals to an increment of 1° of crank angle. The cases mentioned are the comparison between the original diesel engine when operating with pure air (diesel engine) and CNG-air (dual fuel engine), comparison for CNG-air (dual fuel engine) for different CR and cases for different load operations for dual fuel engine with CR = 19.3:1 and CR = 16.6:1 by using different wall temperatures as the load conditions.

6.2 Results and Discussion from the CFD simulation

6.2.1 Effects on Using Different Working Medium in Diesel Engine

The working medium for an original diesel engine with compression ratio of 22.9:1 is air. However, when the diesel engine is using mixture of CNG and air as the working medium in the combustion chamber, the performance of the engine will be different. If knocking occurs in the combustion chamber prior to the piston reaching the TDC, a different compression ratio has to be design.

Hence, the results show the pressure distribution, temperature distribution and stream function of an original diesel engine with compression ratio of 22.9:1 when operating with two different working mediums. One is using air as the original design while the other one is using mixture of CNG and air for a dual fuel system. Comparison is made to analyze the probability of knocks in the combustion chamber. Note that time step in the x-axis for each graph is equivalent to increment of 1° of crank angle as the piston compressed up from BDC to TDC i.e. from 180° crank angle to 360° crank angle.

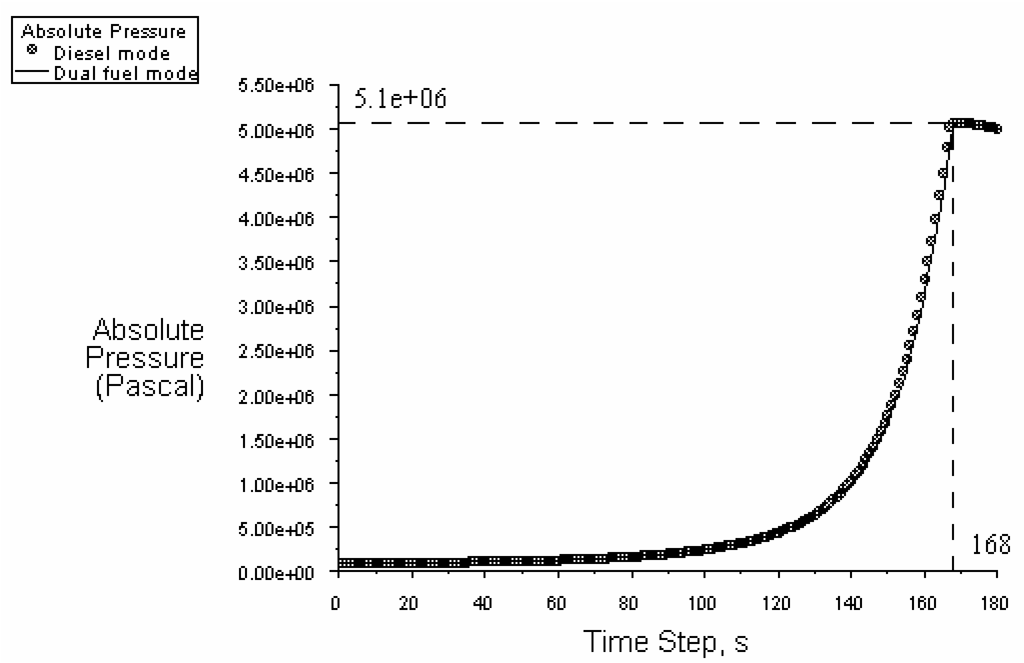


Figure 6.1 – Pressure Distribution for Diesel Mode and Dual Fuel Mode

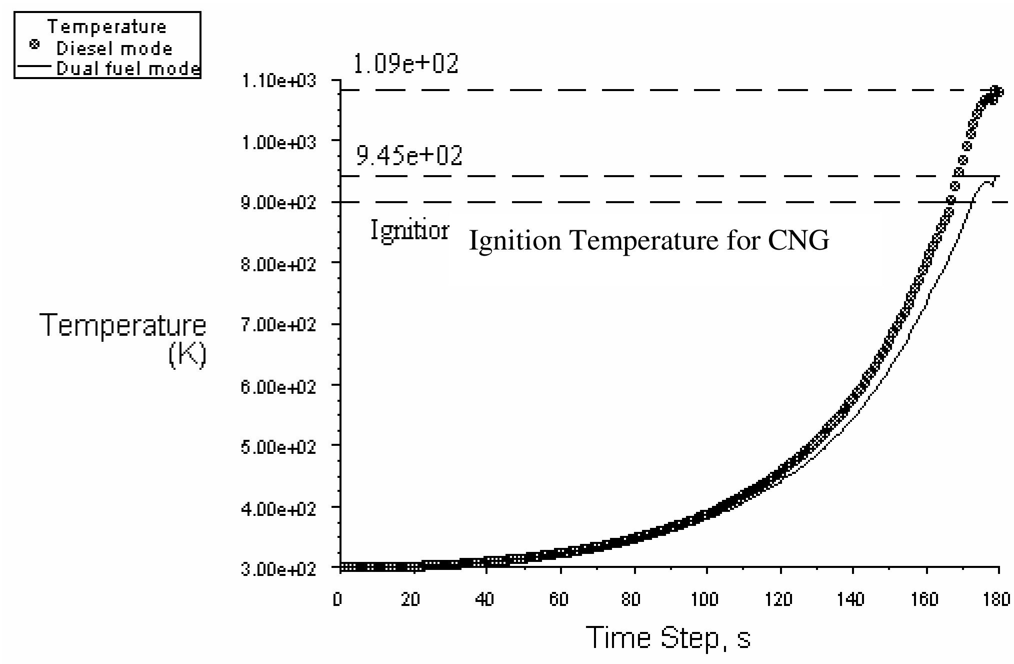


Figure 6.2 – Temperature Distribution for Diesel Mode and Dual Fuel Mode

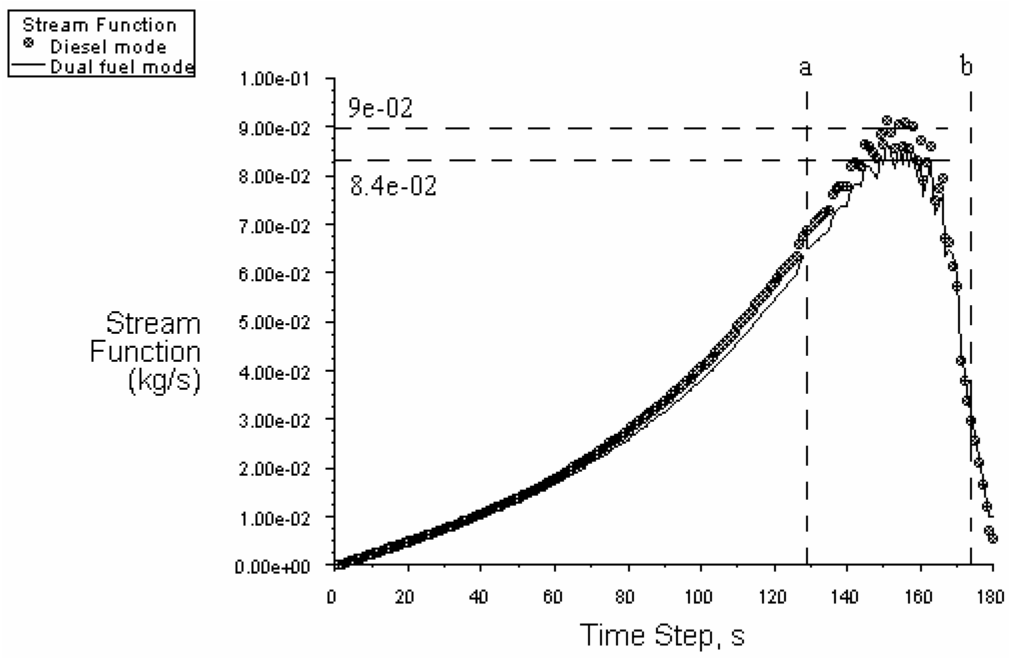


Figure 6.3 – Stream Function for Diesel Mode and Dual Fuel Mode

In Figure 6.1, the pressure distribution for both cases is very similar with air slightly higher than CNG-air after the crank angle pushing up for about 168° from the BDC, which is at a crank angle of about 348°. The peak pressure for both cases is about 5.1 MPa, which is acceptable at this compression ratio. Note that there is a slightly decrease in the pressure from 348° crank angle to the TDC at 360°. This might be due to the reaction of the compressed gas that tried to push back the piston downwards as they compressed up to the TDC.

Meanwhile, the temperature distribution for both cases in Figure 6.2 is significantly different from each other. The peak temperature for diesel mode is about 1090 K while duel fuel mode is about 900 K. Although the working medium consists of a mixture of CNG and air, there is no proper formula to determine the actual ignition temperature. Therefore, only the ignition temperature for CNG is considered in this case. Since CNG ignites at about 900 K, this result shows that CNG cannot operate at this high compression ratio because knock might occur near the TDC.

Next, the stream function for both cases in Figure 6.3 shows that air has higher values than the CNG-air medium as shown in the region between line 'a' and 'b'. Since stream function describes the degree of turbulence of fluid and the quality of mixing, air as a working medium for diesel engine shows better turbulence than the CNG-air. From Figure 6.3, the value for stream function of air is approximately 0.09 kg/s while CNG-air is about 0.08 kg/s, which is very low for both cases. This maybe due to air as the only medium in diesel engine does not require any mixing except during the injection of fuel. For CNG-air, the quality of mixing is poor because of smaller clearance volume for mixing.

Hence, to clearly understand the mixing quality of CNG-air in the combustion chamber, different compression ratios are used to investigate their stream function behavior.

6.2.2 CNG-Diesel Engine For Different Compression Ratio

The results show the pressure distribution, temperature distribution and stream function for different compression ratios of CNG-diesel engine with CNG and air as the working medium inside the combustion chamber. Compression ratio of the original diesel engine (22.9:1), design values of 19.3:1 and 16.6:1 are investigated.

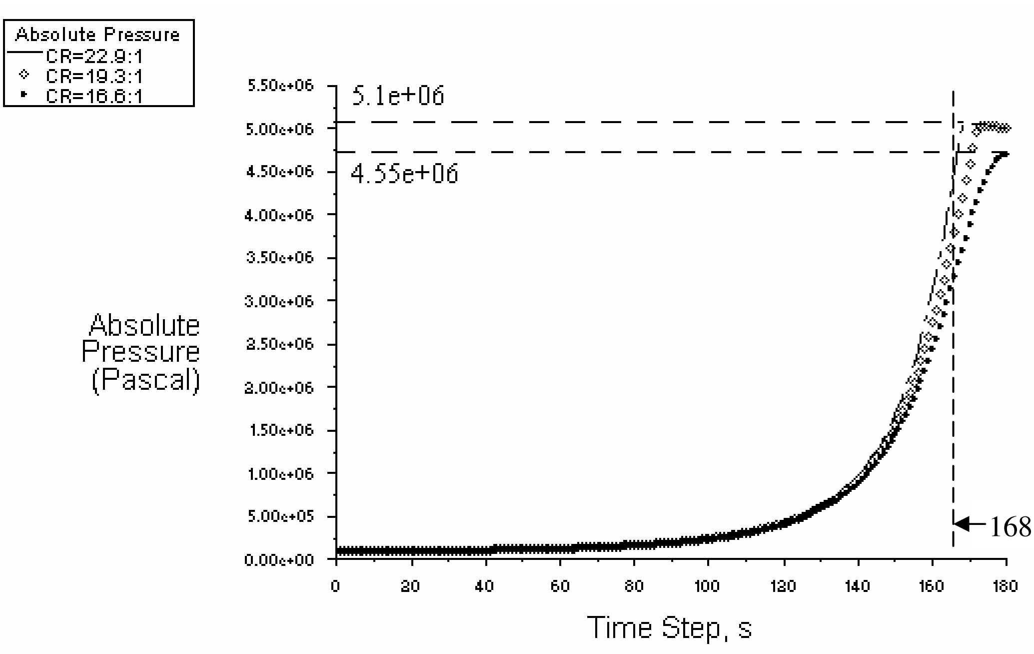


Figure 6.4 - Pressure Distribution for Different Compression Ratios (CR)

Referring to Figure 6.4 above, the peak pressure for CR = 22.9:1 and CR = 19.3:1 are about 5.1 MPa and for CR = 16.6:1 is about 4.55 MPa. All the three curves differ at crank angle of 348°, which is 168° from the BDC. The curve for the original diesel engine shows the highest pressure distribution among the other two lower compression ratio curves after crank angle of 348°. Therefore, Figure 6.4 clearly shows that pressure increases with increasing compression ratio. Note that, the pressure distribution for CR = 16.6:1 does not decrease like the other two CRs. This is because the pressure of the compressed gas at this CR is steadier compared to the other two CRs and the engine does not knock.

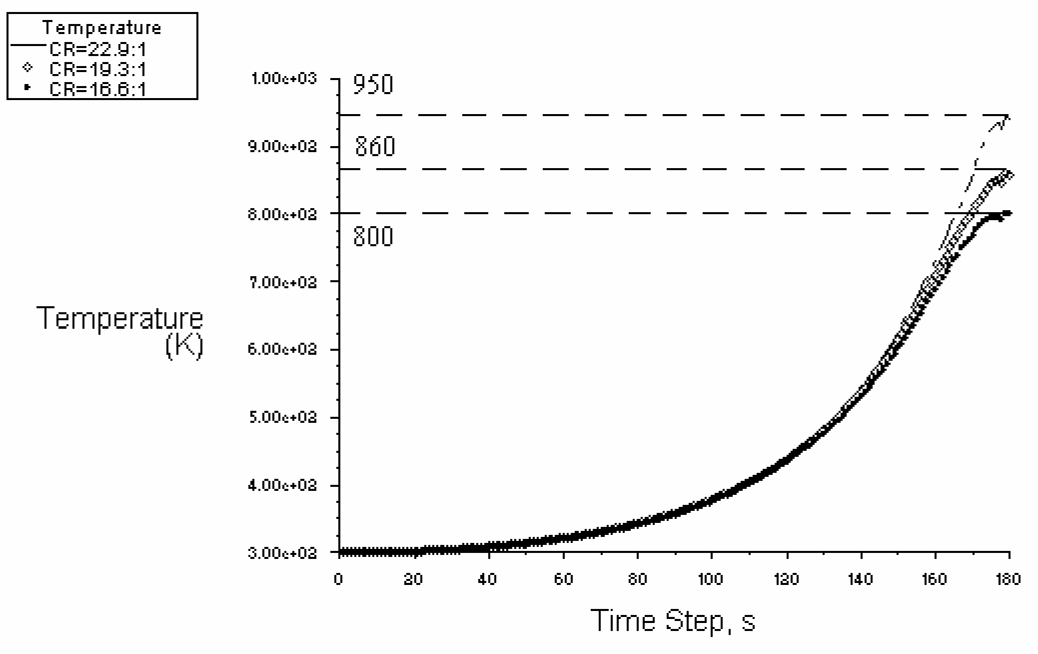


Figure 6.5 - Temperature Distribution for Different Compression Ratios (CR)

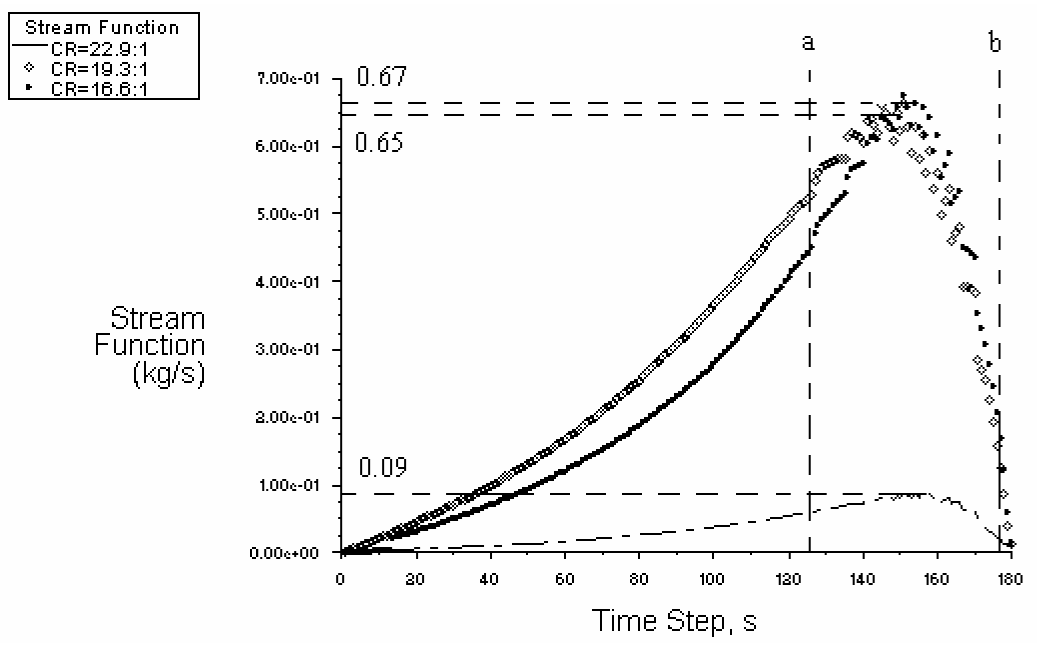


Figure 6.6 – Stream Function Distribution for Different Compression Ratios (CR)

Meanwhile, the temperature distribution for the three different compression ratios in Figure 6.5 differs significantly. The peak temperature for CR = 22.9:1 is about 950 K, for CR = 19.3:1 is about 860 K and for CR = 16.6:1 is around 800 K. This clearly shows that temperature like pressure also will increase with increasing compression ratio. However, CNG has a low ignition temperature about 900 K, which can cause knocking near the TDC in the engine with high compression ratio of 22.9:1. Therefore, lower compression ratio such as 16.6:1 for CNG-diesel engine should be used to ensure that knocking does not occur in the combustion chamber at all time.

In Figure 6.6, the stream function for all the cases focused on the region between line 'a' and 'b'. By comparing the stream function for all the three cases, it is noted that the lowest compression ratio of 16.6:1 shows the maximum value of stream function, which is about 0.67 kg/s. This is followed by compression ratio of 19.3:1 and lastly CR = 22.9:1, which are about 0.65 kg/s and 0.09 kg/s respectively. The higher values of stream function for CR =16.6:1 is due to the additional gasket plate between the piston head and the engine block that increased the clearance volume. This concept applies to engine with CR = 19.3:1 but its plate thickness is smaller than the former one, thus its stream function is lesser than the former one also.

Hence, the degree of turbulent is the highest for CR = 16.6:1 and the mixing between CNG and air is the best among the other two cases. With these two factors, the combustion efficiency for the CNG-diesel engine will be more efficient and indirectly produces more power output. However, if smaller CR is used, the pressure will be too low for ignition to occur, which is not efficient to provide power output.

6.2.3 Different Load Conditions for CNG-Diesel Engine with CR = 19.3:1

There are a few load conditions applied to the CNG-diesel engine with compression ratio of 19.3:1 to investigate the engine performance in terms of pressure distribution, temperature distribution and stream function. The load conditions refer to the wall temperature, which normally varies between 25°C to 200°C. If the wall temperature is higher than 200°C, the engine might experience failure due to expansion of the piston that causes instability when compressed upward and downward.

In this case, the load conditions are listed below:

i. Light load operating condition - 27° or 300 K
ii. Normal load operating condition - 50°C or 323 K
iii. Medium load operating condition - 100°C or 373 K
iv. Heavy load operating condition - 150°C or 423 K

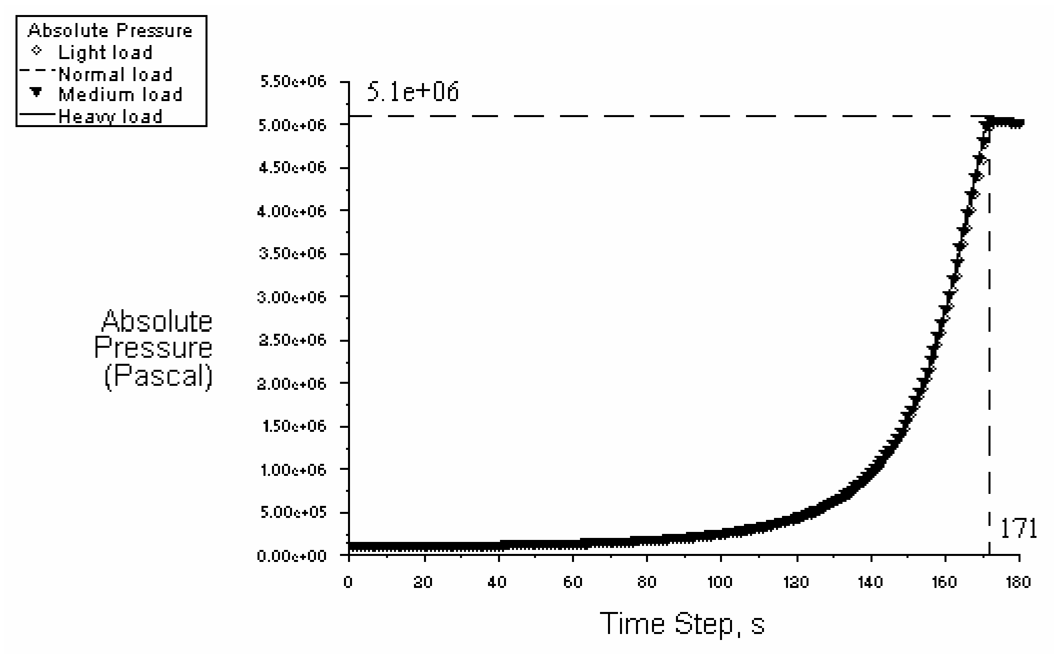


Figure 6.7 - Pressure Distribution for Different Load Conditions for CR=19.3:1

In Figure 6.7 above, the pressure distribution for all the load conditions is very similar. There is slightly difference among all the load conditions, which is between the heavy load condition and the light load condition. The former load condition is the highest among all the other load conditions and the latter is lesser than the former one. This means that pressure will increase with heavier load condition and the peak pressure for all the load conditions is approximately 5.1 MPa that occur after 171 time steps, which is equals to 351° crank angle.

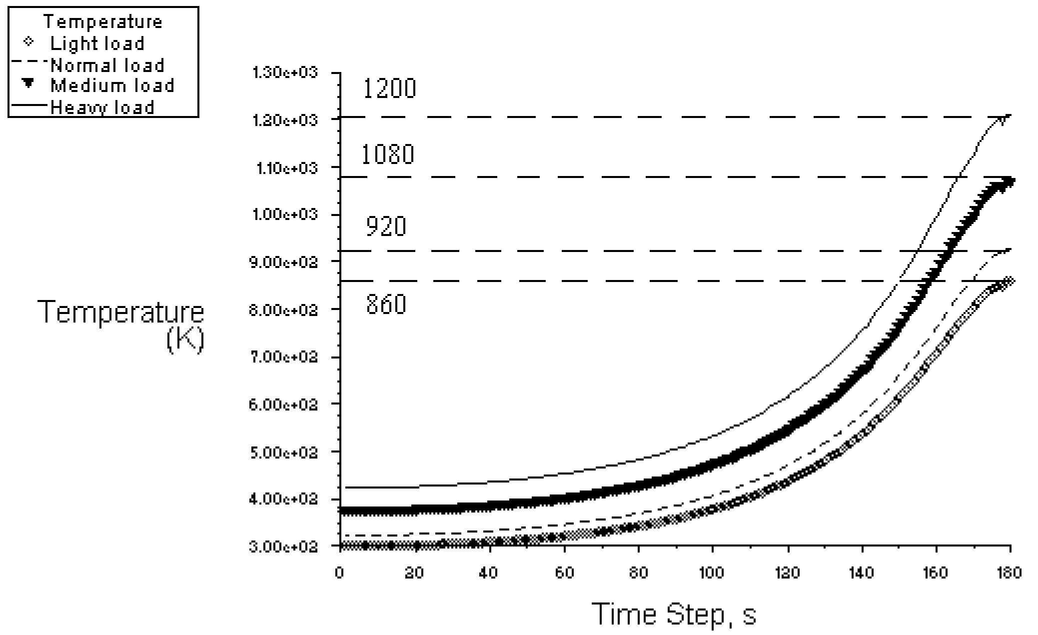


Figure 6.8 - Temperature Distribution for Different Load Conditions for CR=19.3:1

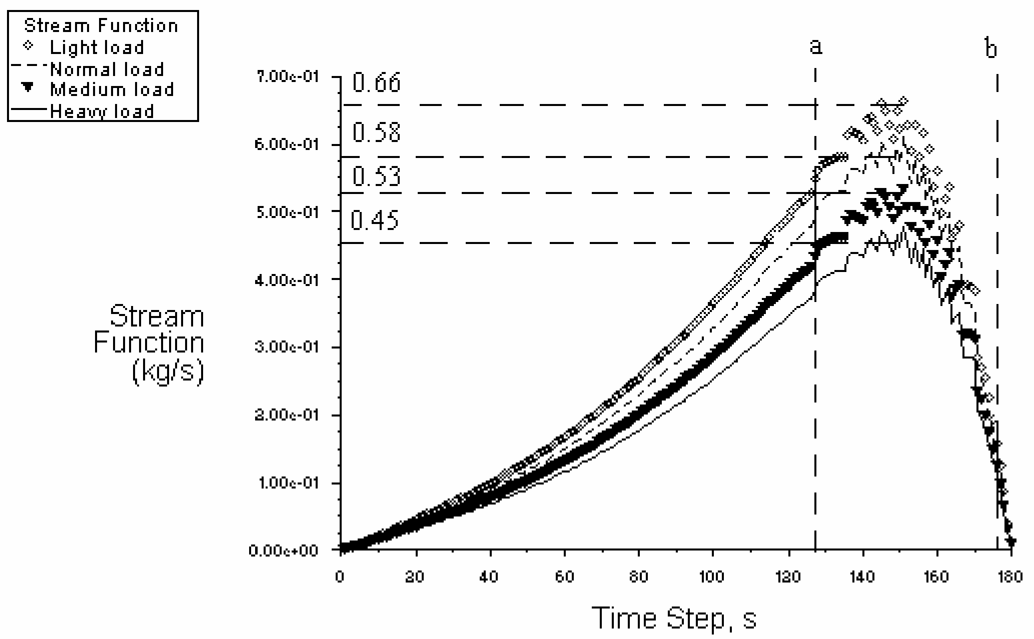


Figure 6.9 – Stream Function Distribution for Different Load Conditions for CR = 19.3:1

The most significant result is shown in Figure 6.8 for the temperature distribution under various load conditions. With increasing temperature on the wall, the temperature distribution for all the load conditions starts at different temperature also. Therefore, the peak temperature for light condition is about 860 K, for normal condition is about 920 K, for medium operating condition is about 1080 K and lastly for heavy load condition is about 1200 K.

For the stream function in Figure 6.9, it clearly indicates that different load conditions have different set of stream function curves. As in the previous stream function curves, only the region between line 'a' and 'b' is focused. The highest value for stream function under the same compression ratio is the light load condition and vice-versa for heavy load condition. The value can be as high as 0.66 kg/s for light load condition and the lowest value for the stream function are about 0.45 kg/s for heavy load condition.

6.2.4 Different load conditions for CNG-Diesel Engine with CR = 16.6:1

Same as the previous section 6.2.3, the load conditions are divided into 4 categories namely light, normal, medium and heavy load condition. The wall temperature varies from 300 K to 423 K to determine the engine performance for CNG-diesel engine with lower compression ratio of 16.6:1.

Hence, engine performance in terms of pressure distribution, temperature distribution and stream function for CNG-diesel engine with compression ratio of 16.6:1 are illustrated below. Note that the values for the pressure, temperature and stream function in this case are different from the previous case for engine with compression ratio of 19.3:1. This helps to determine the optimum compression ratio that can accommodate CNG as a dual fuel system in a diesel engine so that its performance is comparable to the original diesel engine.

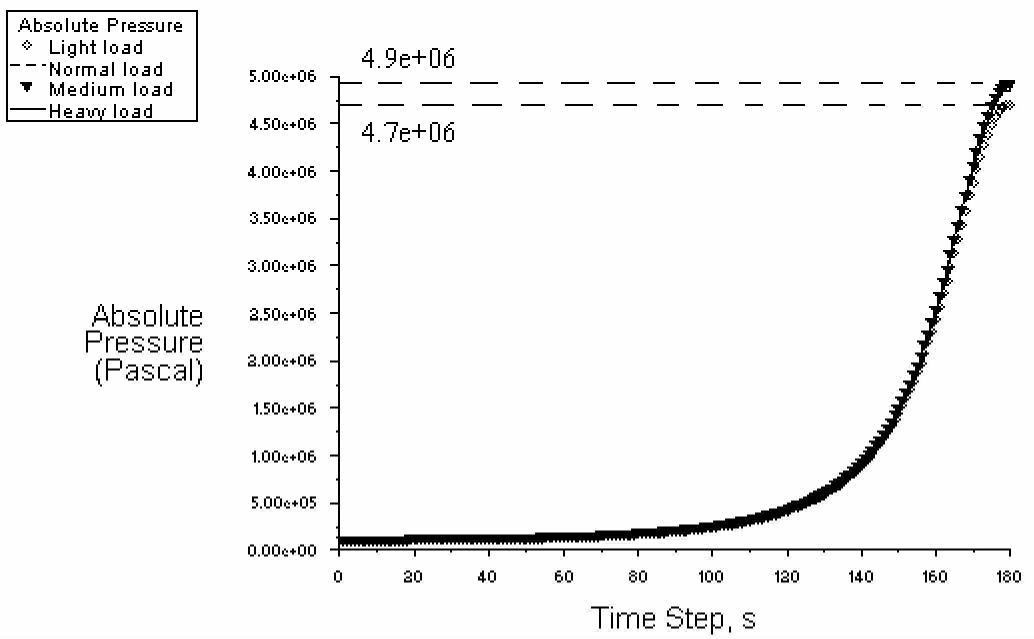


Figure 6.10 - Pressure Distribution for Different Load Conditions for CR = 16.6:1

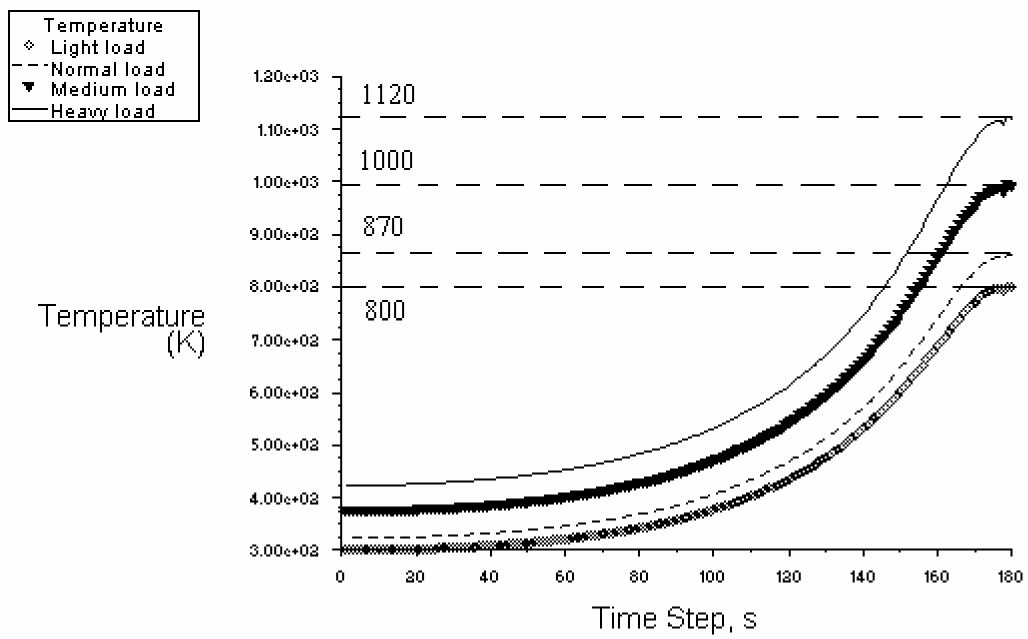


Figure 6.11 - Temperature Distribution for Different Load Conditions for CR = 16.6:1

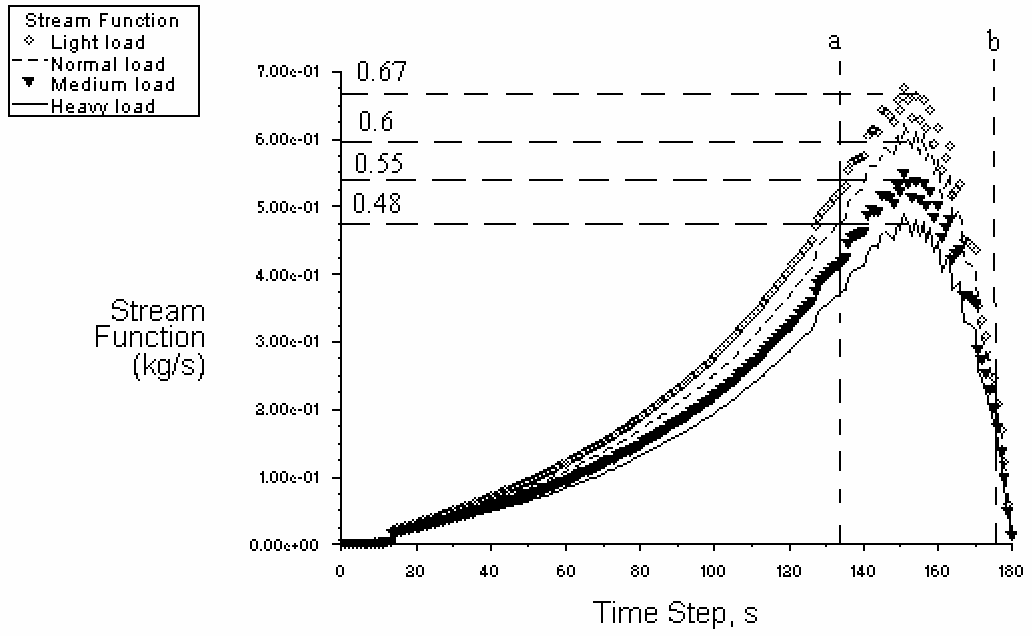


Figure 6.12 – Stream Function Distribution for Different Load Conditions for CR = 16.6:1

Referring to Figure 6.10 for the pressure distribution in the combustion chamber, it is found that the peak pressure for light condition is the lowest among the other three load conditions. The peak pressure for light condition is about 4.7 MPa and the pressure for the other load conditions are in the range of 4.9 MPa. This result shows that pressure increases with at higher load condition. Therefore, for a low compression ratio of 16.6:1, the CNG-diesel engine can still maintain a high pressure without knocking in the engine even though during the heavy load condition.

Next, for temperature distribution in Figure 6.11, the graph shows that there is a big difference between each load condition. The temperature increases when heavier load is applied to the engine. The difference between heavy load condition and medium load condition is about 120 K while the difference between light load condition and normal load condition is about 70 K.

However, knocking will occur when medium or heavy load condition is applied to the engine because the peak temperature for these two conditions exceeds the ignition temperature for CNG. The peak temperature for heavy load is about 1120 K and the peak temperature for medium load condition is about 1000 K, which exceeds the ignition temperature for CNG at 900 K.

Lastly, the stream function shown in Figure 6.12 indicates that the mixing quality for the light load condition remains the highest compared to the other load conditions. At the region between line ‘a’ and ‘b’, the highest value recorded for light load condition is about 0.67 kg/s and the lowest value for heavy load condition is about 0.48 kg/s. This clearly indicates that heavy load condition is not favour because the mixing of CNG with air is poor and prone to knocking phenomena.

Therefore, based on the results obtained, the optimum compression ratio for the dual fuel engine to operate without knocks is 16.6:1 and the maximum loading condition it can operates is until the normal load condition, which is about 373 K.

CHAPTER VII

CONCLUSION

7.1 Achievement of the Objectives

Overall, the objectives of this project as listed in the Appendix A of the Project Specification or in Chapter 1 were achieved. Different compression ratios (CR) of 16.6:1, 19.3: and 22.9:1 were used to simulate the combustion chamber and to investigate their effects on the engine performance, which is in terms of pressure distribution, temperature distribution and stream function.

Based on the results obtained in Chapter 6, it is concluded that the pressure distribution is lower as the CR reduced. This is true because when compression ratio decreased, the compression pressure decreased also. Although the compression pressure reduced to about 4.5 MPa, it is high enough to ignite the CNG-diesel mixture at the BDC. As for the temperature distribution of the mixture in the chamber, it recorded a peak temperature below 900 K, which is the ignition temperature for CNG, for CR of 16.6: and 19.3:1. Therefore, the CNG-diesel engine can operate under these two CRs.

To select the optimum CR, stream function for all the CRs is simulated to study the turbulence flow in the mixture. The stream function for CR of 16.6:1 recorded the highest value compared to the other two CRs. This is due to the increased in clearance volume that promotes better turbulence in the flow and thus better mixing quality of CNG and air mixture. Hence, the optimum compression ratio selected based on the performances as explained above is 16.6:1 that promotes higher combustion efficiency for the CNG-diesel engine and thus better power output also.

The operating load conditions that varied between light load conditions to heavy load condition were simulated and the performances were investigated for CR of 19.3:1 and 16.6:1. Both results that based on the same concept as described above show that the CNG-diesel engine cannot operate under medium and heavy load conditions. Therefore, the maximum operating load condition for the CNG-diesel engine is normal load condition with wall temperature of 50°C or 373 K.

Although the project successfully carried out the research and simulation work to achieve the objectives of this project, there are still a lot of improvement and adjustment to be made on the design of the CNG-diesel engine. The current design in this project is not perfect yet and contains flaws due to time constraint. Therefore, the following section will explained on the possible recommendation and further work to improve the design of the CNG-diesel engine.

7.2 Recommendation and Future Work

This project researches on the capability of CNG to use with diesel fuel in dual fuel engine is still in its infancy to get the concept realized for a commercial use. Therefore, more research has to be executed to improve the design of the CNG-diesel engine to gain better performance.

As explained in Chapter 5, there are different methods used to reduce the compression ratio of an engine. Besides addition of plate, the other two methods are modifying the connecting rod and modifying the piston to contain a piston bowl shape. Therefore, future researchers can do research on the effects of these two methods to accommodate CNG in the diesel engine to operate as a dual fuel engine.

Besides that, the engine performance can be further research and simulated in more detail such as the heat transfer in the combustion chamber, gas emission and the combustion process. The heat transfer model can be used to simulate the heat losses from the compressed gas to the wall while the combustion process can be used to simulate the possible residual left after the combustion process and the possible emission gases also.

LIST OF REFERENCES

1. Alimoradian, B. & Allen, R.S. 1990, '*Dedicated natural gas conversion of a Cummins N41 diesel engine using an electronic fuel control system'*, Conference Proceedings of *NGV-New life in transportation naturally*, Buenos Aires.

2. Bassi, A. 1990, '*Natural gas vehicles for a clean environment - an overview and the trends for the future'*, in Carvallo, Maria da Graca, Fiveland, Woodrow A., Lockwood, F.C. & Papadopoulos, Christos (eds), *Clean combustion technologies*, vol 2, Part B, pp.739-762.

3. Clark, N.N., Atkinson, C.M., Lyons, D.W., Mott, G.E., Atkinson, R.J. & deJong, R.J. (n.d.), '*Development of a closed loop fuel management system for a lean burn natural gas engine*' [online], pp.1-10. Retrieved: March 2, 2004, from http://www2.cemr.wvu.edu/~englab/Projects/Hercules/hercpap.pdf.

4. 'Clean air program-assessment of the safety, health, environmental and system risks of alternative fuel' (n.d.). Retrieved: April 2, 2004, from http://ntl.bts.gov/DOCS/afrisks.html#toc.

5. 'CNG engine performance' (n.d.), pp.1-7. Retrieved: February 5, 2004, from http://www.arb.ca.gov/regact/cng-lpg/appe.doc.

6. 'Compressed natural gas' (n.d.). Retrieved: March 11, 2004, from http://www.mckenziecorp.com/dehydration.htm.

7. 'Effects of using CNG to propel your car' (n.d.). Retrieved: April 2, 2004, from http://www.gasmancng.com.

8. Ferguson, C.R. 1986, *Internal combustion engines - applied thermosciences*, John Wiley & Sons, Singapore.

9. 'Green mission - CNG a fact file' (n.d.). Retrieved: February 5, 2004, from http://www.ashokleyland.com/cng.html.

10. Heath, M. 1996, '*Alternative transportation fuel: natural gas, propane, methanol, ethanol compared with gasoline and diesel*', Canadian Energy Research Institute.

11. KVR BMC Pistons Limited (n.d). Retrieved: 2 August 2004, from http://www.kvrbmc.com/shape_pis.htm.

12. Selim, M.Y.E. 2003, '*Sensitivity of dual fuel combustion and knocking limits to gaseous fuel composition*', Energy Conversion and Management, pp. 411-425.

13. Shamsudin, A.H & Yusaf, T.F. 1995, '*A review on the use of natural gas in internal combustion engine*', ASEAN Journal Science Technology Development, pp. 101-113.

14. Tiratsoo, E.N. 1979, *Natural gas - fuel for the future? A world survey,* 3rd edn, Gulf Publishing Company Houston, Texas.

15. Vizard, D. 2003, '*High compression*'. pp. 1-11.

16. Weaver, C.S., P.E. & Lit, M.L. 2002, '*Issues and potential benefits of natural gas vehicles'*, presented in Better Air Quality Workshop (BAQ) 2002. Retrieved: March 3, 2004, from http://www.cse.polyu.edu.hk/~activi/BAQ2002/BAQ2002_files/Proceedings/Subworkshop4/sw4b-3Chan_Final.pdf.

17. 'Why natural gas: economic, environmental pollution, safety, properties, composition and availability' 1995, pp. 1-14.

18. 'Year-end 2003 refining margins dip, then recover' 2004, *Hydrocarbon Processing*. Retrieved: April 1, 2004, from http://www.hydrocarbonprocessing.com.

19. Yusaf, T.F., Shamsudin, A.H., Ali, Y. & Ahamd, F.I. 1996, '*Design modification of high speed diesel engine to accommodate compressed natural gas*', RERIC International Energy Journal, vol. 18, no.1, pp. 19-26.

20. Yusaf, T.F., Ali Al-Atabi, M.T., Ali, Y. & Shamsudin, A.H. 2001, '*A dual fuel system (CNG-diesel) for a single-cylinder, stationary engine used in power generation*', Brunei Darussalam Journal of Technology and Commerce, vol. 2, no.1, pp. 58-64.

21. Yusof, A.F 2000, '*Effect of compression ratio on engine performance for CNG-Diesel engine*', Department of Engineering, University of Tenaga Nasional (UNITEN), Malaysia.

BIBLIOGRAPHY

1. Clark, N.N, Atkinson, C.M., Atkinson, R.J., McDaniel, T.I. & Park, T. (n.d.), 'Optimized emission reduction strategies for dual fuel compression ignition engines running on natural gas and diesel'. Retrieved: March 3, 2004, from http://www2.cemr.wvu.edu/~englab/Projects/navistar/navistar.html.

2. 'CNG - an alternative fuel' (n.d.). Retrieved: April 2, 2004, from http://www.leeric.lsu.edu/bgbb/7/ecep/auto/j/j.htm.

3. 'CNG for vehicles - technological and safety measures' (n.d.). Retrieved: February 5, 2004, from http://www.saeindia.org/saeconference/cngvehicles.htm.

4. 'Compressed natural gas (CNG)' (n.d.). Retrieved: February 5, 2004, from http://www.altfuels.org/backgrnd/altftype/cng.html.

5. Davies, J.G. & Sulatisky, M.T., '*Demonstration of CNG as a fuel in agricultural tractors*', Society of Automotive Engineering (SAE) Paper No. 891669.

6. Dong, S.J., Sung, W.S., Seung, M.O. & Kyo, N.C., '*Development of mechanical CNG-diesel dual fuel supply system*', Society of Automotive Engineering (SAE) Paper No. 931947.

7. Ikegami, M., Shioji, M., Zhu, Q.M., Hotta, Y. & Endo, H. 1993, '*A study of direct-injection high-compression engines fueled by CNG*', Japan Society of Automotive Engineering (JSAE) Review, vol 14, no.2, pp. 87-90.

8. Johnston, B., McGoldrick, T., Funston, D., Kwan, H., Alexander, M., Alioto, F., Culaud, N., Lang, O., Mergen, H.A. & Carlson, R. (n.d.), '*The continued design and development of the University of California, Davis Future Car*' [online], pp.1-14. Retrieved: March 11, 2004, from http://www.team-fate.net/reports/1997TechReport.pdf.

9. Katz, D.L. & Lee, R.L. 1990, *Natural gas engineering - production and storage*, McGraw-Hill Publishing Company, Singapore.

10. McCaw, D.L. & Horrell, W.A. 2001, '*On-road development of John Deere 6081 natural gas engine'* [online], pp.1-54. Retrieved: March 11, 2004, from www.afdc.doe.gov/pdfs/deere6081.pdf.

11. Meyer, R., Meyers, D., Shahed, S.M. & Duggal, V.K., '*Development of a heavy duty on-highway natural gas-fueled engine*', Society of Automotive Engineering (SAE) Paper No. 922362.

APPENDIX A, PROJECT SPECIFICATION

University of Southern Queensland

FACULTY OF ENGINEERING AND SURVEYING

ENG4111/4112 Research Project
PROJECT SPECIFICATION

FOR:	**LIM Pei Li**
TOPIC:	The Effect of Compression Ratio on the Engine Performance for CNG-Diesel Engine.
SUPERVISOR:	Dr. Ahmad Sharifian
ASSOCIATE SUPERVISORS:	Dr. Fok Sai-Cheong (USQ) and Dr. Talal Yusaf (Malaysia)
PROJECT AIM:	The project aims to investigate and simulate the effect of different compression ratios on the engine performance for Compressed Natural Gas (CNG)-Diesel engine.
PROGRAMME:	**Issue A, 19 March 2004**

1. Research information on the CNG-diesel engine combustion process, related equations on the heat and mass transfer system, combustion and emissions, theory related to compression ratio and the effect on the engine performance.

2. Develop a MATLAB program to calculate the heat and mass transfer in the combustion chamber.

3. Develop a 2-D CFD simulation model using FLUENT to investigate the affect of pressure strength and the combustion temperatures distribution on the knock using CNG-diesel system.

4. Evaluate the CNG-Diesel engine efficiency with respect to the quality of mixing, pressure distribution and temperature distribution in the combustion chamber for different compression ratios.

5. From 3 and 4 an optimum compression ratio will be selected to deliver optimum combustion with no knock.

AGREED: _______________ (Student)
LIM Pei Li

___/___/_____

____________________, ___/___/_____
Dr. Ahmad Sharifian
(Supervisor)

____________________, ___/___/_____
Dr. Talal Yusaf
(Associate Supervisor)

____________________, ___/___/_____
Dr. Fok Sai-Cheong
(Associate Supervisor)