FLAME PROPAGATION IN SPARK IGNITION ENGINE COMBUSTION PROCESS USING COMPUTATIONAL FLUID DYNAMICS (CFD)

HANIF BIN KASMANI

BACHELOR OF ENGINEERING

UNIVERSITI MALAYSIA PAHANG

2010

ABSTRACT

This thesis deals with the flame propagation in spark ignition engine combustion process using Computational Fluid Dynamic (CFD). This study is based on flame propagation inside the combustion chamber which is important as flame propagation affects the engine efficiency, emission and some more. A 3-D model is created based on the Mitsubishi Magma 4G15 that act as a baseline engine. It is then simulated using CFD where its approaches make it a suitable platform to study the internal combustion engine phenomenon. The project simulates only 50^{o} CA starting from the ignition until the completion of the combustion process. The flame radius obtain through simulation is then compared with the experimental data together with the literature review. However, there are discrepancies of the results due to improper boundary condition and inherit limitation of the model. For further simulation of combustion process must consider detail mixture properties, detail boundary condition and model extension for better accuracy data.

ABSTRAK

Tesis ini berkaitan dengan perebakan nyalaan gelora di dalam enjin dengan menggunakan kaedah dinamik aliran berkomputer, Kajian ini berdasarkan perebakan nyalaan di dalam enjin dimana ianya sangat penting kerana ianya mempengaruhi kecekapan enjin, pembebasan bahan api dan beberapa lagi. Kajian ini terdiri daripada pemodelan berangka dengan menggunakan Mitsubishi Magma 4G15 sebagai rekabentuk asas. Simulasi ini menggunakan CFD kerana pendekatan program ini membuatnya menjadi platform yang sesuai untuk mempelajari fenomena pembakaran dalam enjin. Projek ini mensimulasi 50^{O} kitaran bermula daripada proses pengapian sehingga selesainya proses pembakaran. Radius api yang deperolehi daripada simulasi dibandingkan dengan data eksperimen bersama-same dengan kajian sastera. Namun, ada perbezaan keputusan akibat dari keadaan sempadan yang tidak tepat dan keterbatasan model. Untuk simulasi masa hadapan bagi proses pembakaran, penelitian harus dipertimbangkan dari segi keadaan campuran, keadaan sempadan, dan model sambungan untuk ketepatan yang lebih baik.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND OF STUDY

Combustion is the most important process taking place in spark-ignition (SI) engine, in which chemical energy of fuel is transformed into internal energy of the cylinder charge. During this process, a turbulent flame propagates across the combustion chamber and burns the premixed fuel-air mixture. The turbulent flame generally can be assumed in spherical shape (Bayraktar, 2006). The turbulence enhances the mixing process, and hence increases the combustion rate. Thus, combustion can be considered as turbulent flame propagation process.

Theoretically, as the flame propagation become faster, more efficient engine operation can be obtained. Faster burning is essential as it can reduce engine knock means higher compression ratios results to higher efficiency, higher power output and engine can operate in stable condition (Bayraktar, 2006). Moreover, better combustion together with leaner mixtures may also reduce hydrocarbon emissions from the engine. A commonly suggested method to analyses flame propagation is using computer simulation. Computer simulation of the combustion process in spark ignition began several decades ago.

The combustion process in spark ignition engine has been studied very extensively in the past. These studies have came up with, initially the flame is relatively smooth and as it grows, the flame front becomes increasingly influenced by the turbulent flow through it spreading and eventually develops a highly wrinkled and possibly multiply connected structure. The control of these variations could lead to improve engine operations in terms of economy, drivability and exhaust emissions.

1.2 PROBLEM STATEMENT

The purpose of this project is to analyze the flame propagation process and to propose a method to measure the flame radius inside combustion chamber. Flame propagation is useful in internal combustion engine as one can determine the engine efficiency, emission and many more. Therefore, a correct method to study the flame propagation and analyze the flame radius is needed to give better understanding on how flame propagates within combustion chamber.

1.3 OBJECTIVES OF THE PROJECT

The objectives of this research are:

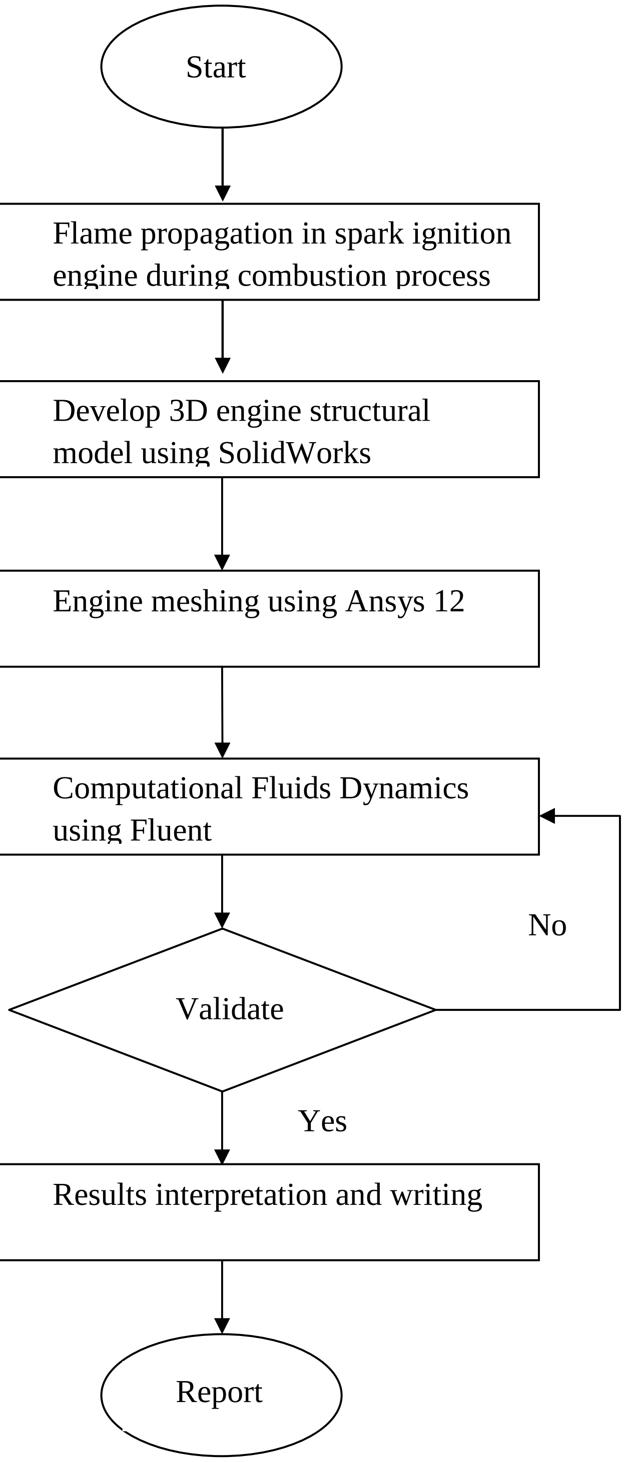
1. To analyze the flame propagation process.
2. To propose a measurement method of flame radius within combustion chamber.

1.4 SCOPES OF THE PROJECT

The scopes of study covered the study of analysis the flame propagation process and to come up with method to measure the flame radius within the combustion chamber. First of all, in order to model the problem, the engine was modeled using based on the carbureted gasoline engine, Mitsubishi Magma 4G15, 12 valve, 1.5 litre engine with pent-roof combustion chamber. The type and the specification of this engine are shown in Chapter 3. The model was created as it can be compared to the reality engine design and it helps to improve understanding of how flame propagates inside the combustion chamber. Next, the model has to undergo meshing process.

The purpose of doing meshing is to analyze the fluid flow inside the combustion chamber. Before running the simulation, points are created inside the combustion chamber for the measurement of flame radius. For this study, the simulation has been carried out at constant engine speed of 2000 rpm. The main interest parameter is cylinder pressure as is used for validation between the simulated and experiment. Other parameters that are discussed are flame propagation in terms of progress variable and lastly flame radius.

1.5 FLOW CHART



:

Figure 1.1: Project flow chart

1.6 ORGANIZATION OF THESIS

This thesis consists of five main chapters which are introduction, literature review, methodology, results and discussion and the last chapter is conclusion and recommendation. For chapter 1, it presents some basic findings that lead to problem statement, objective, scopes and flow chart of the project. While chapter 2 is about literature part that related to the study and acts as basic to the study framework. For chapter 3, it presents the basic dimensioning work on Mitsubishi Magma 4G15 engine, development of 3D model and generation of computational domain. Also, the pre-processing setup is presented in order to attain mesh generation and finally the flame radius measurement method. Chapter 4 discusses the validation of the simulation results against experimental results of the flame radius. Lastly, chapter 5 is about the conclusion and recommendation for future study.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter tells about combustion process in SI engine where generally it is divided into three main sections. Next, it describes about the flame kernel formation together with the affects of variable for example like engine speed, manifold pressure and others. The discussions then continue with classifications of turbulent model and lastly the CFD approach during the simulation.

2.2 COMBUSTION IN SPARK IGNITION ENGINE

Internal combustion engine is a heat engine that converts chemical energy in a fuel into mechanical energy. First, the chemical energy is converted to thermal energy by combustion or oxidation that occurs inside the engine. Consequently, this energy raises the temperature and pressure and the pressure itself expand the mechanical mechanism of the engine. This expansion is converted into mechanical energy by a rotating crankshaft which is connected to transmission and produced the final mechanical energy that rotates the engine (Pulkrabek, 1997). To obtain its maximum power or torque, the combustion event must be properly located relative to top-center. The combined duration of the flame development and propagation process is approximately between 30 and 90 crank angle degrees.

Combustion process in SI engine can divided into three sections which are:

1. Ignition and flame development
2. Flame propagation
3. Flame termination

Basically, the first 5% of the air-fuel mixture process is label as the flame development process. During this period, ignition occurs and the combustion process start, however very little pressure rise hence there is no useful work done. At 80% to 90% of the process, the work is produce in the engine because of the flame propagation period of the combustion process where the bulk of the fuel and air mass is burned.

During this time, the pressure inside the cylinder is greatly increased thus provides the force to produce the work in the expansion stroke. Then, for the final 5% of the process, the pressure quickly decreases and the combustion stops (Pulkrabek, 1997).

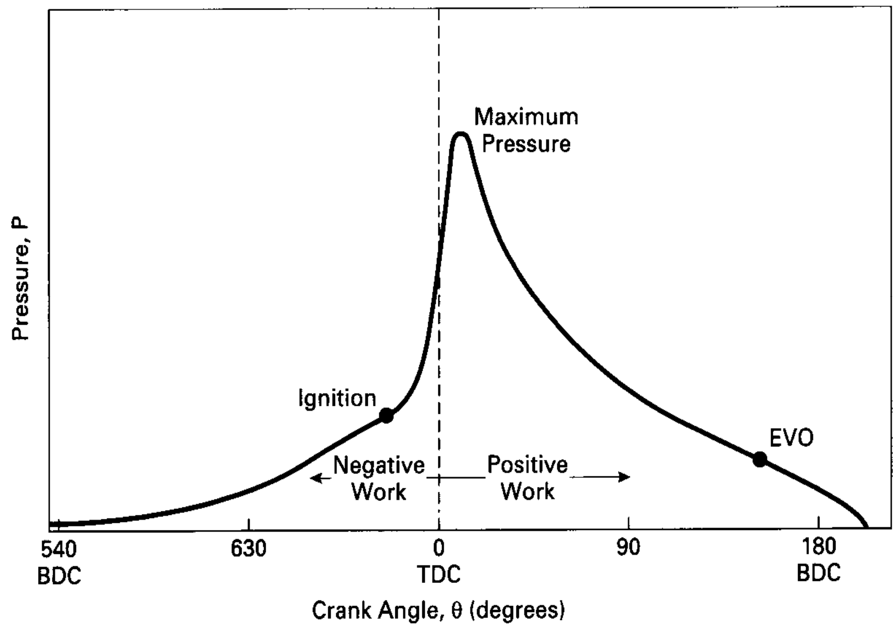


Figure 2.1: Cylinder pressure in the combustion chamber of an SI engine as a function of crank angle.

Source: Pulkrabek 1997

2.2.1 Ignition and Flame Development

Combustion occurs everywhere from 10^o to 30^o before TDC which is initiated by an electrical discharge across the electrodes of a spark plug. The very high temperature called plasma discharged between the electrodes ignites the air-fuel mixture in the immediate vicinity and the combustion spreads outward to the surrounding. Flame can generally detect at about 6^o of crank rotation after the spark plug firing. Several reasons contribute to this action which is the high heat losses to the relatively cold spark plug and the gas mixture.

At the time the spark plug fires, the plasma discharge ignites the air-fuel mixture between the near electrodes causing a spherical flame front created and propagates outward into the combustion chamber. Initially, the flame front propagates slowly as the small size does not generate high energy to heat the surrounding. After 5-10% of the air fuel is burned, the flame velocity increase to higher value corresponding to the fast rise in pressure.

It is important to have a rich air-fuel mixture around the electrodes of the spark plug during ignition because the rich mixture is more ready to ignite, has faster flame speed and provides better start to the overall combustion process. To assure that a rich mixture is obtain, spark plugs are generally located near the intake valves especially when starting a cold engine (Pulkrabek, 1997).

2.2.2 Flame Propagation in SI Engine

After the first 5-10% of the air-fuel mass has been burned, the combustion is well developed and the flame front's speed increase rapidly and it propagates through the combustion chamber. At this time, the speed is calculated about 10 time faster compare to the speed at ignition and flame development process. As the gas mixture burns, the temperature as well as the pressure rises to a high value. It is said that when 30% of the gas mass is burned, the burned gases already occupy almost 60% of the total volume. While the other 70% of the mixture that is still not burned being compresses into 40% of the total volume. This compression of the unburned gases eventually raises their temperature. Ideally the air-fuel mixture should be about two-thirds burned after TDC and almost completely burned at 15^o after TDC. So it is said that the maximum pressure and temperature occur somewhere between 5^o to 10^o after TDC.

Because of the effects of turbulence, swirl and squish, the flame speed depends on the type of the fuel and air fuel ratio. For lean mixture, the flame speed have slower speeds while for the rich mixture, the flame speed is fastest which can be describe in figure 2.2. In another words, flame speed increases with engine speed due to higher turbulence, swirl and squish.

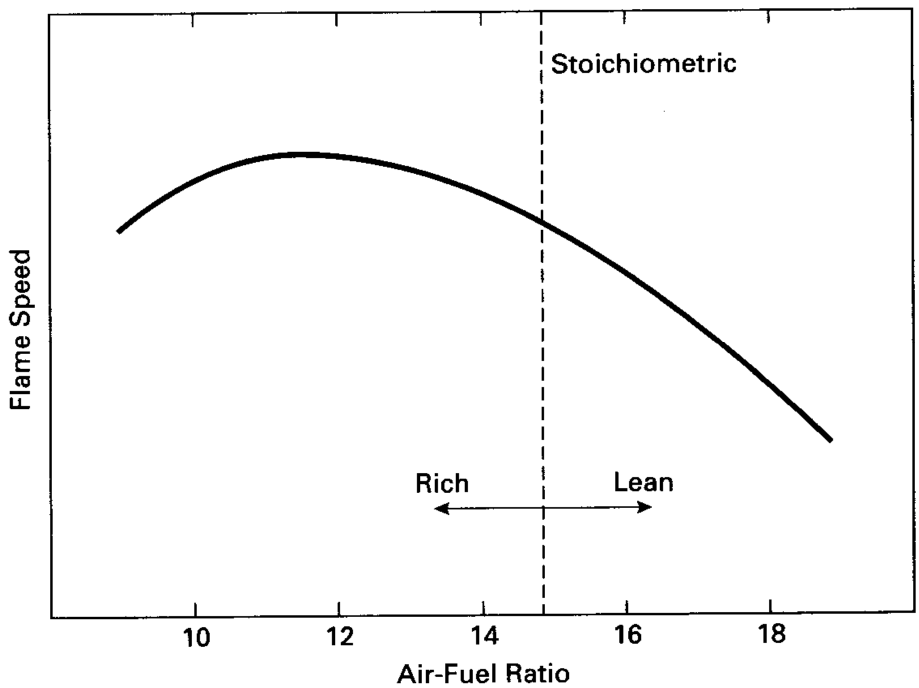


Figure 2.2: Average flame speed in the combustion chamber of an SI engine as a function of the air-fuel ratio for gasoline type fuels.

Source: Pulkrabek 1997

2.2.3 Flame Termination

At a range about 15^o to 20^o after TDC, 90-95% of the air-fuel mass has been combusted while the flame front already reached the extreme corners of the combustion chamber. Although at this point the piston has already moved away from TDC, the combustion chamber volume has only increased about 10-20% from the very small clearance volume. This means that the last mass of fuel-air react in very small volume in the corner of the combustion chamber as well at the wall of the combustion chamber.

Because of the closeness of the combustion chamber walls, the last end gas reacts at a very reduce rate. In addition, the large mass of the metal walls acts as a heat sink and conducts away the energy in the reaction flame. Both of these mechanisms reduce the reaction rate and flame speed and the combustion finally end. Although the additional works delivered by the piston are very little, it is still important because the force transmitted to the piston taper off slowly resulting in smooth engine operation.

Usually, during the flame termination period, self-ignition will occur causing the phenomena called engine knock. However, the resulting knock is usually unnoticeable. This is because the amount of air-fuel mixture left in the combustion chamber is very little and it can only cause slight pressure pulses (Pulkrabek, 1997).

2.3 KERNEL FORMATION

The kernel formation is caused by the combustible mixture in the cylinder which is ignited by the electric discharge between the spark plug electrodes. The spark initially caused by breakdown, discharge the electrical energy through the arc and glow phases. The breakdown process initiates a shock wave that enlarges radically and travels until it becomes a form of acoustic wave (Song et al., 2000).

The plasma is created upon breakdown discharge and the flame kernel is produced from the plasma. Because of that, thermal diffusion dominates the development of the flame kernel. The development is defined to be successful when the power input from the spark is sufficient enough to drive the flame kernel beyond the critical size. Based on (Song et al., 2000), the development of the initial kernel is determined by the heat dissipation during the first several μsecs, and is influential up to 1msec. Both figure 2.3 and 2.4 represent the characteristics of the initial flame kernel when it develops through heat dissipation. Through the energy released from the breakdown nearly doubles, the difference in kernel temperature is only 50-80K. While for the radius, it grows 15-20% faster, suggesting that if breakdown energy increases, it is not the increase in kernel temperature but the increase in kernel radius which accelerates the kernel growth (Song et al., 2000).

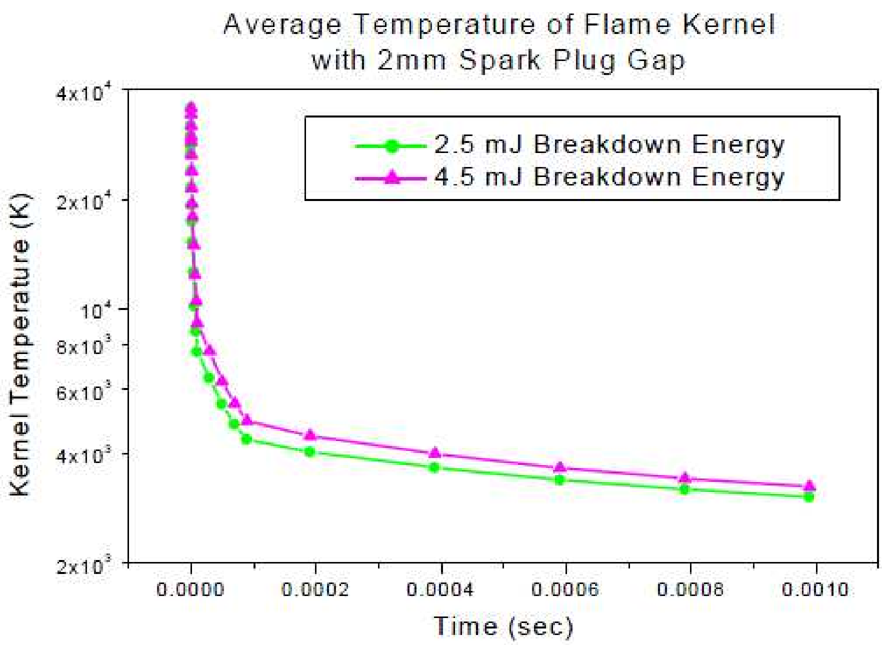


Figure 2.3: Initial flame kernel temperature

Source: Song et al., (2000)

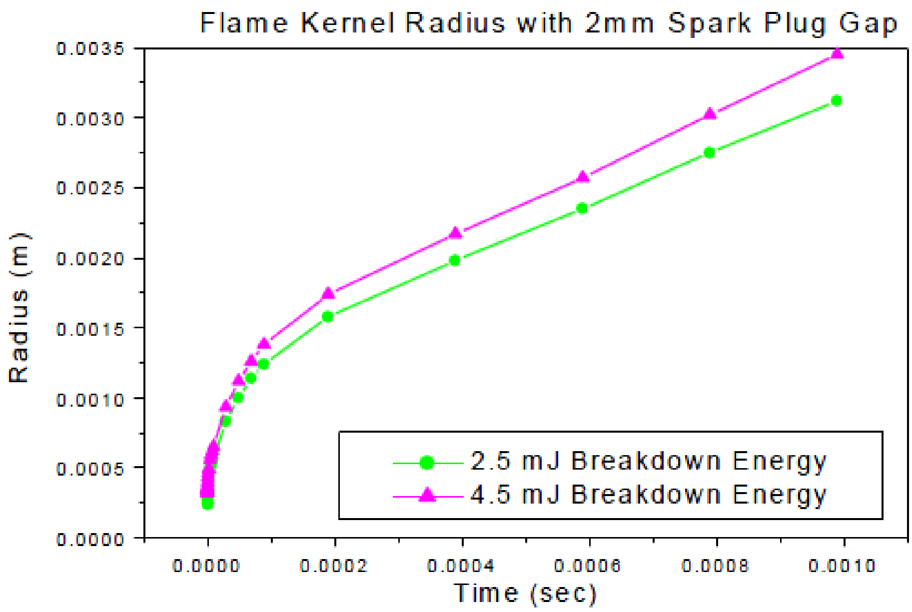


Figure 2.4: Initial flame kernel radius

Source: Song et al., (2000)

2.4 COMBUSTION MODELING USING CFD

The modeling of engine processes continues to develop our basic understanding of the physics and chemistry of the phenomena of interest and as the ability to provide sufficient capability for engine performance prediction. Rapid development in computer technology plays a major role in contributing to this achievement.

In solving the CFD problem, it usually consists of four main components which are:

1. Geometry and grid generation
2. Setting-up a physical model
3. Solving
4. Post processing the computed data

Current approaches to CFD methodology in simulation of engine combustion are based on the governing equation of mass, momentum and energy conservation both two and three dimensional model. In some simulation of complicated internal combustion engine process, the model must consider complex phenomenon like turbulence, ignition, chemical reaction, heat transfer and boundary layer process near the wall (Fadzil, 2008).

However, each of these situations can easily tackle by using different kind of turbulence model. Turbulence is considered as it appears to be dominant over all other flow phenomena.

2.5 CFD APPROACH IN SI SIMULATION BY OTHER RESEACHERS

CFD approach in fluid analysis is not something new especially in SI simulation. Previous study and experiment show that CFD approach in SI engine is a success consider that combustion in engine has been greatly improved by development an enhanced engine from time to time. Since CDF approach is simulating the model, it provides a real insight cylinder to see the fluid flow behavior (Semin et al., 2008). After the simulation has done, one can easily get the data through various forms like graphs, images, tables and even in the form of media graphic.

Through media graphic, researcher can easily compared the results of fluid flows inside the pipe, combustion chamber, duct and a few more by observing and differentiate the result appear. CFD approach also needs the domain to be meshed. The purpose of the meshing is that it provides better accuracy to the results. According to (Payri et al., 2003), hexahedral cells type has become more popular since it provides better accuracy as well stability compare to tetrahedral cell types. However, tetrahedral cell types also have its own advantages and purposes.

This means that the selection of the grid on surface is not easily done. One has to consider the accuracy, sensitivity as well as the computing time before selecting the best cell type. (Bai and Hsiao, 2007) stated that, the selection of the mesh type must consider the shape of the geometry. As for the bend part like intake or exhaust manifold, the tetrahedral cell type is better than the hexahedral cell type.

2.5.1 Effects of Various Parameters into Flame Radius

The Effect of Engine Speed

Turbulence intensity (u') increases simultaneously to engine speed. This also increases turbulent flame velocity while the ignition delay period which is defined as a 0-10% mass fraction burnt period is reduced drastically. The increases in (u') also accelerate burning and boost peak pressure.

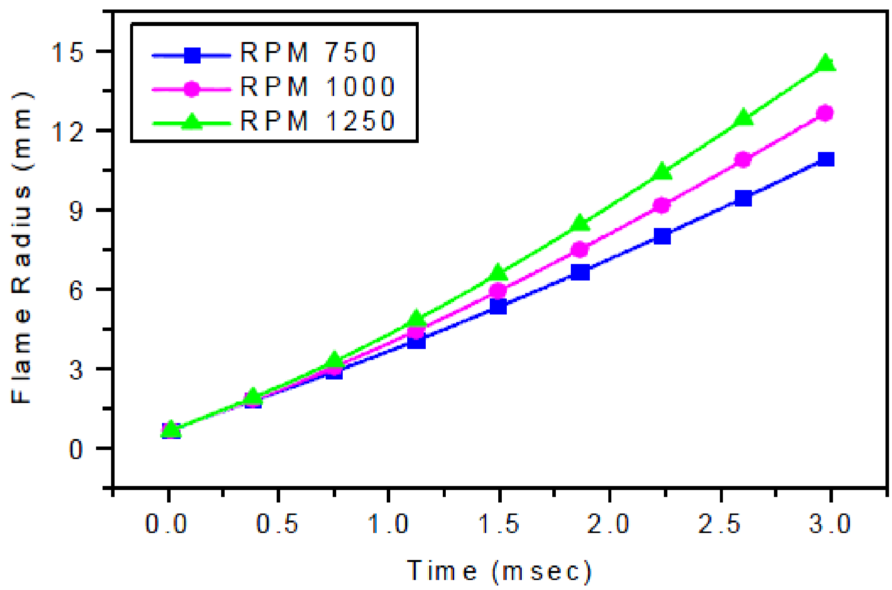


Figure 2.5: Flame kernel radius with various engine speeds

Source: Song et al., (2000)

The Effect of Manifold Pressure

Manifold pressure determined by the opened area at the throttle valve increases as the engine load increases. This process increases the initial pressure in the combustion chamber and increases breakdown energy. Therefore the radius of the initial kernel is enlarged as manifold pressure increases. It also influences the laminar flame and induces faster propagation. Velocity and the amount of intake mixture are likewise increased.

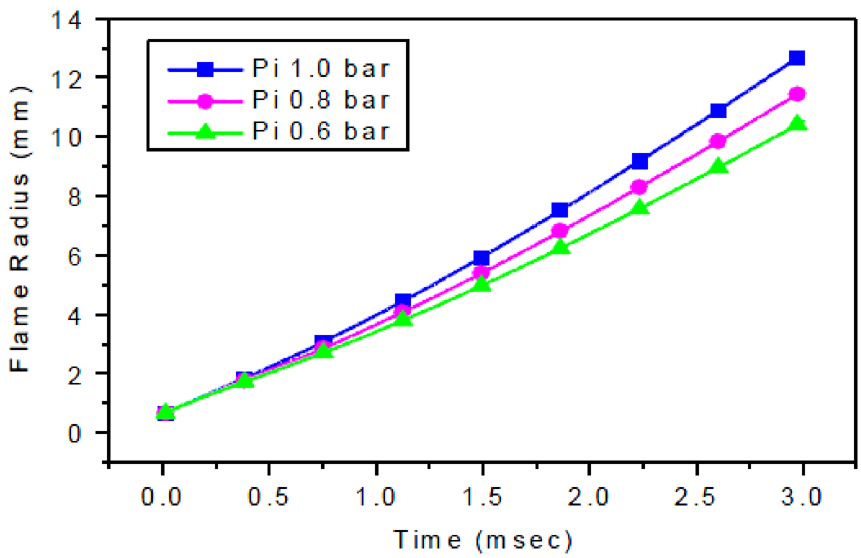


Figure 2.6: Flame kernel radius with various intake manifold pressures

Source: Song et al., (2000)

The Effect of Air Fuel Ratio

The fuel air equivalence ratio Φ is defined as the actual fuel-air ratio divided by the stoichiometric fuel-air ratio. If the Φ is less than 1, the mixture is said to be in lean condition. As the mixture becomes lean, the burnt gas temperature falls and the laminar burning velocity decreases. Mass dissipation is also reduced consequently reducing the turbulence burning velocity.

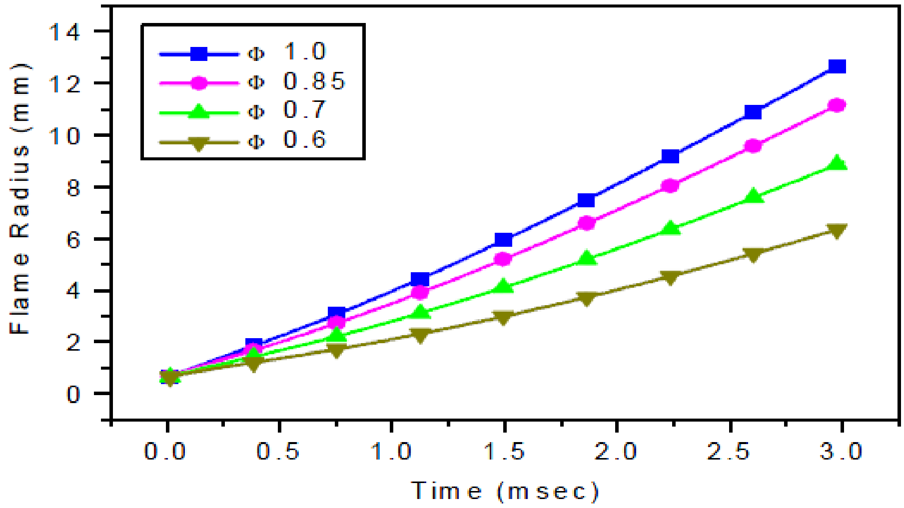


Figure 2.7: Flame kernel radius with various air fuel ratios

Source: Song et al., (2000)

2.6 SUMMARY

This chapter mentioned about the literature findings about the flame propagation process in SI engine. Generally, it is divided into three main categories which are ignition and flame development, flame propagation and flame termination. After that, it discusses some of the kernel formation. Although it only talks about the kernel but one can identify the pattern of the overall flame development once the flame kernel formation has been studied. The effects of various variables such as the engine speed, manifold pressure, air-fuel ratio and the residual gas also has been discuss on this chapter. At the later stage of this chapter, it mentioned about the combustion modeling using CFD. There are various turbulence models available and one must know the usage of each model so that the best model is chosen with the appropriate situation. Last part of this chapter mentioned on the CFD approach which tells about the cell types and also the purpose of each cell types.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter mainly discuss about the numerical modeling setups and procedures. Here, CFD approach for the SI combustion modeling is described for examples mesh generation, initial condition, boundary condition, governing equation involves, numerical analysis and the validation method. Method on how to measure flame radius within combustion chamber also be reveal. Moreover, the engine baseline specifications as well as the important parameters are discussed.

3.2 BASELINE ENGINE SPECIFICATION

In this study, the type of the engine that is used is four-stroke four cylinder SI engine. The engine that is used is Mitsubishi Magma 4G15 1.5L. Below are the engine specifications:

Table 3.1: Mitsubishi Magma 4G15 1.5L engine specification

Parameters	**Size and features**
Number of cylinder and valves per cylinder	4 cylinders with 3 valves per cylinder (two intake valves and one exhaust valve)
Combustion chamber type	Pent-Roof type
Displacement volume	1488
Cylinder bore (mm)	75.5
Piston stroke	82.0
Compression ratio	9.2
Intake valve open/close	15^{o} BTDC/ 63^{o} ABDC
Exhaust valve open/close	57^{o} BBDC/13^{o} ATDC

Form the above table, the bore is 75.5 mm while the stroke is 82.0 mm. The other important parameter is compression ratio. The value for the compression ratio that mention in the table in 9.2 means that the development of the 3D model must have the same or close value follows what shows in the table.

3.3 ENGINE MODELING

After taking all the required dimension of the combustion chamber, the further step is design the model using appropriate software. The model created need to maintain it original dimension so that it can relate with the actual combustion process of Mitsubishi Magma engine. Figure 3.1 below shows the picture of the 3D model. Generally the engine has two intake valves and one exhaust valve per cylinder which the size of each valve is different. The spark plug position is located at the side of the combustion chamber.

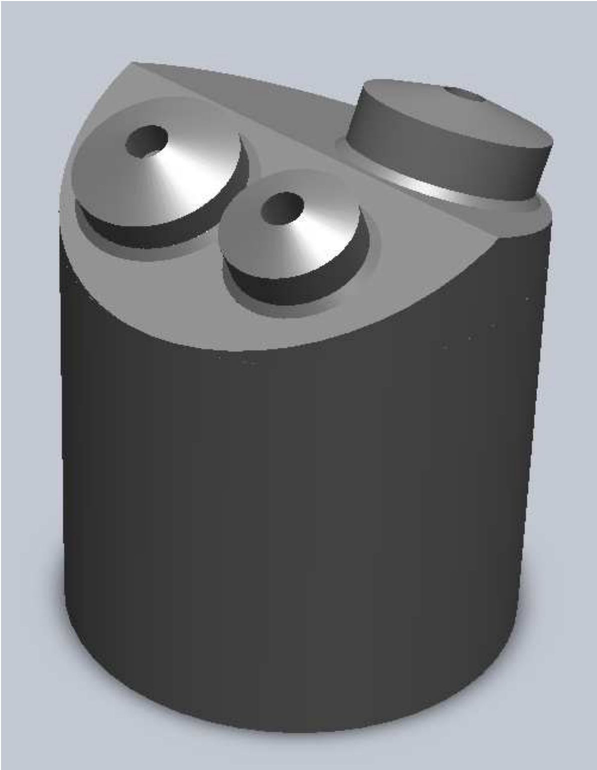


Figure 3.1: Isometric view of engine model

Based on Figure 3.1, some of the engine part has been simplified for example the geometry for both the intake manifolds as well as the exhaust manifold. The reason behind this action is to reduce complexity and ease the meshing process. Although the model has been simplified, it still retains its original dimension of the baseline model.

3.4 THE GOVERNING EQUATIONS FOR CFD

CFD methodology is using partial differential equations of flow variables to calculate and to simulate numerous kind of analysis concerning the fluid flow. Among them are mass, momentums, energy, species concentration, quantities of turbulence and mixture fractions.

3.4.1 Mass Conservation Equation

The continuity equation or the mass conservation equation for any fluid flow is expressed as below (Fluent Inc, 2004);

$$\frac{\partial}{\partial x} + \frac{\partial}{\partial xj}(\rho \mathrm{u}_j) = \dot{\mathrm{m}} \tag{3.1}$$

Where

ρ	=	Fluid density
u_j	=	The *jth* Cartesian component of instantaneous velocity
$\dot{\mathrm{m}}$	=	The rate of mass of the object generated in the system

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

3.4.2 Momentum Conservation Equation

The conservation of momentum in *i* direction for an inertial reference frame can be explained as below (Fluent Inc, 2004):

$$\frac{\partial}{\partial t}(\rho \mathrm{u}_i) + \frac{\partial}{\partial xj}(\rho \mathrm{u}_i \mathrm{u}_j) = \frac{\partial p}{\partial xi} + \frac{\partial \tau ij}{\partial xj} + \rho \mathrm{gi} + \mathrm{Fi} \tag{3.2}$$

Where

ρ	=	Fluid density
$\mathrm{u}_i \& u_j$	=	The *ith* and *jth* Cartesian components of the instantaneous velocity
p	=	Static pressure
τij	=	Stressor pressure
$\boldsymbol{\rho} g_i$	=	Gravitational body force
F_i	=	External body force from interaction with dispersed phase in *i* direction

The stress tensor in given in Equation 3.2 is given as below:

$$\tau_{ij} = \mu(\frac{\partial ui}{\partial xj} + \frac{\partial uj}{\partial xi}) - \frac{2}{3}\mu(\frac{\partial uk}{\partial xk})\delta_{ij} \quad (3.3)$$

Where

μ	=	Fluid dynamic viscosity
δ_{ij}	=	Kronecker delta

Note that the second term on the right hand side of Equation 3.2 describes the effect of volume dilation. By substituting Equation 3.2 into Equation 3.3, another equation is produced that is complete momentum equation.

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial xj}(\rho u_i u_j) = -\frac{\partial p}{\partial xi} + \frac{\partial p}{\partial xj}\{ \mu\left(\frac{\partial ui}{\partial xj} + \frac{\partial uj}{\partial xi}\right) - \frac{2}{3}\mu\left(\frac{\partial uk}{\partial xk}\right)\delta_{ij}\}\rho gi + Fi \quad (3.4)$$

3.4.3 Energy Conservation Equation

$$\frac{\partial}{\partial t}(\rho e) + \frac{\partial}{\partial xi}[u_i(\rho e + P)] = \frac{\partial}{\partial xi}\left[K_{eff}\frac{\partial T}{\partial xi} - \sum_j h_j J_j + u_j(\tau_{ij})_{eff}\right] + S_h \quad (3.5)$$

Where

K_{eff}	=	Effective conductivity
	=	$k + k_i$ (where k_t = turbulent thermal conductivity)
J_j	=	Diffusion flux of species j
S_h	=	Additional volumetric heat sources (example: heat of chemical reaction)
h	=	Sensible enthalpy
e	=	Specific total energy

The first three terms on the right-hand side of Equation 3.5 represent the energy transfer due to conduction, species diffusion and viscous dissipation respectively. From equation 3.5 also, sensible enthalpy, h and specific total energy, e are defined as below (Fluent Inc, 2004):

$$e = h - \frac{p}{\rho} + \frac{ui^2}{2} \quad (3.6)$$

Sensible enthalpy for ideal gas is defined as:

$$h = \sum_j m_j h_j \tag{3.7}$$

Sensible enthalpy for incompressible flow is defined as:

$$h = \sum_j m_j h_j + \frac{p}{\rho} \tag{3.8}$$

Where

m_j = Mass fraction of species j

h_j = $\int_{Tref}^{T} c_{p.j}$ dT with T_{ref} = 298.15K

3.5 MESH GENERATION

The requirements of the computing mesh are (Heywood, J.B. 1988):

1. It adequately fits the topography of the combustion chamber and/or inlet port, including the moving parts.
2. It allows control of local resolution to obtain the maximum accuracy with a given number of grid points.
3. It has the property that each interior grid is connected to the same number of neighboring points.

The first requirement tells that every change is model geometry need to simulate. The second requirement tells that the computing time increases with the number of mesh points while the third requirements tells that the mesh need to be in topologically rectangular shape so that highly efficient equations can be utilized.

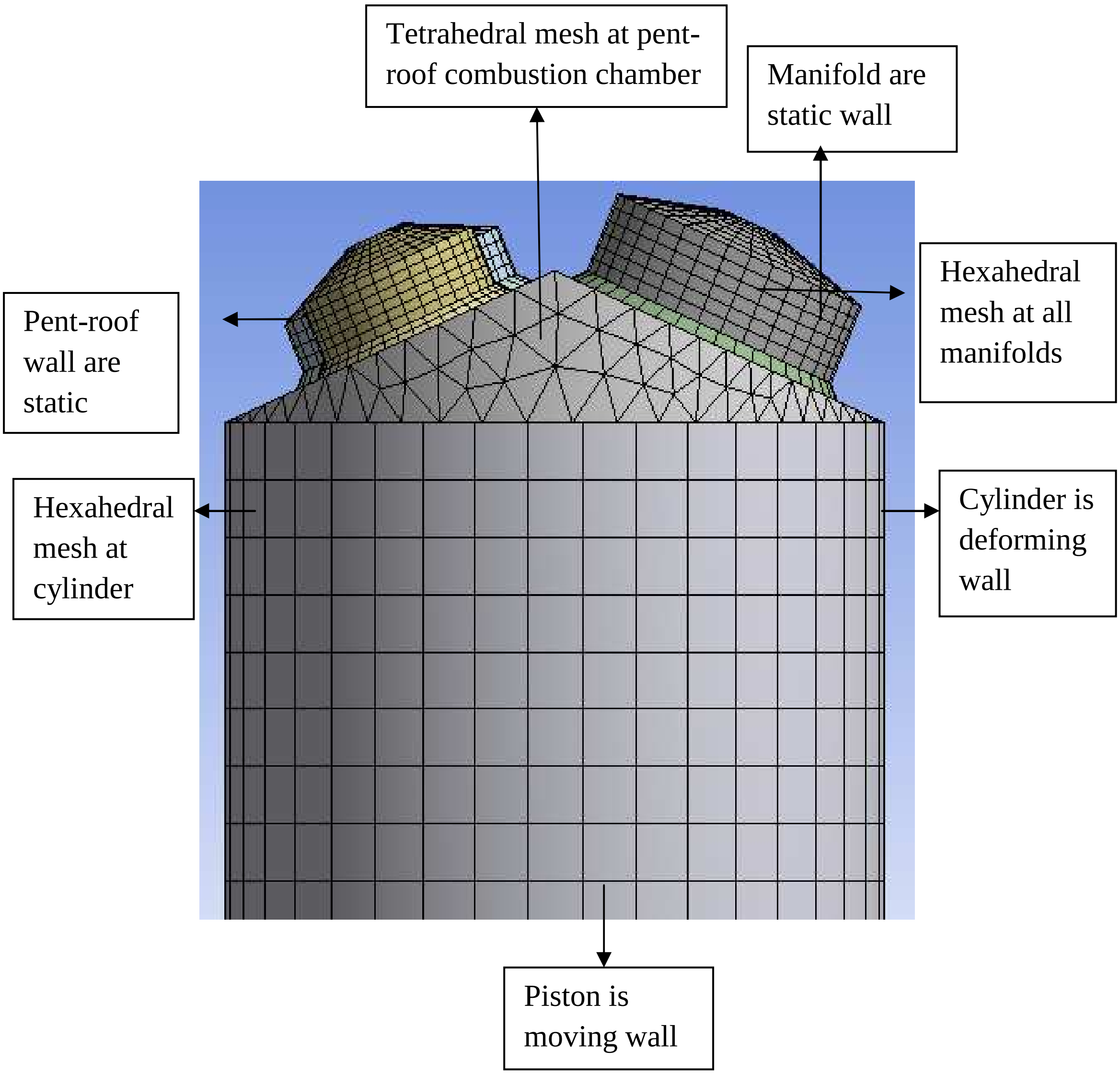


Figure 3.2: Mesh generation

The model is developed using Solid Works based on the engine specification cited in Table 3.1. Next the model was imported into ANSYS to start the mesh generation. As in the Figure 3.2 the tetrahedral mesh is chosen for the combustion chamber. The combustion chamber wall is also set as static wall. As for the cylinder, the mesh is set to be hexahedral mesh and the side wall is set to be deforming wall while the bottom wall is set to be moving wall.

3.6 SOLUTION SETUP

3.6.1 Initial Condition

In simulating the combustion chamber process, several initial conditions need to be known and setup. Initial conditions are important because the value of initial process need to be set up before start simulating the combustion process.

Table 3.2: Initial condition at 2000 rpm

Initial Condition	**Value**
Pressure	1150684 Pa
Temperature	605.1476 Kelvin
Progress variable	0
Engine speed	2000 RPM
Crank angle duration	50^o

From the Table 3.2, pressure and temperature are set as 1150684 Pa and 605.1476 K respectively. The temperature values are set as constant throughout the combustion process. Since the air-fuel mixture still not burn, the progress variable must start with 0. For this study, the constant engine speed was taken as 2000 rpm. The total crank angle for this simulation is 50^o where the simulation starts from the timing of ignition until the completion of the combustion process.

3.6.2 Boundary Condition Setup

For the cylinder wall, it is set to be deforming wall to simulate the compression and power stroke. The pent roof wall as well as the manifolds wall is set to be static wall while the valves wall are set to be moving wall so that all valve can be move up and down according to corresponding axis. The piston is set as moving wall in order to simulate the movement of the piston which follows the real movement of the piston in the engine. Constant engine speed is set at 2000 rpm. The other boundary condition setting is like in Table 3.3.

Table 3.3: Boundary condition at 2000 rpm

Variable	Value	Units
Cylinder Head Temperature	550	K
Piston Face Temperature	573	K
Cylinder Wall Temperature	458	K

3.6.3 Input Data for Premix Mixture

Table 3.4 shows all the input data that are set at the material for premix-mixture properties.

Table 3.4: Input data for premixed-mixture properties

Properties	**Value**	**Units**
Specific heat	0.08207936	J/kg.K
Thermal conductivity	4.10317 x 10-5	W/m.K
Laminar viscosity	27.4547	kg/m.s
Molecular weight	0.4762	kg/kmol
Laminar flame speed	18563	m/s
Critical rate of strain	0.07643	s-1
Lower heating value	4.43 x 107	J/kg

3.7 FLAME RADIUS MEASUREMENT METHOD

In order to measure flame radius within combustion chamber, one has to come up with appropriate method that can provide an accurate result regarding flame radius within combustion chamber. In this study, points have been created in just one plane in which is parallel to the top cylinder surface like in figure 3.3. The reason behind this action is that, the flame radius can be measured from the beginning up till the flame reach the other side of the combustion chamber as corresponding to the flame termination process. The flame radius increase linearly with time as stated by (Tagalian et.al, 1986). However, due to the assumptions that flame propagates