EFFECT OF DIFFERENT TURBULENCE MODELS ON COMBUSTION CHAMBER PRESSURE USING COMPUTATIONAL FLUID DYNAMIC (CFD)

MOHD NASRUDDIN BIN AMIR NAZRI

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

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **EFFECT OF DIFFERENT TURBULENCE MODELS ON COMBUSTION CHAMBER PRESSURE USING COMPUTATIONAL FLUID DYNAMIC (CFD)**

SESI PENGAJIAN: **2009/2010**

Saya **MOHD NASRUDDIN BIN AMIR NAZRI (870211-06-5541)**
(HURUF BESAR)

mengaku membenarkan tesis (Sarjana Muda/~~Sarjana~~ /~~Doktor Falsafah~~)* ini disimpan di Perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

[ ]	**SULIT**	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
[ ]	**TERHAD**	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
[√]	**TIDAK TERHAD**	

Disahkan oleh:

______________________	______________________
(TANDATANGAN PENULIS)	(TANDATANGAN PENYELIA)
Alamat Tetap:	
319 LEPAR HILIR 02	**MOHD FADZIL B. ABDUL RAHIM**
26300 GAMBANG	(Nama Penyelia)
PAHANG	
Tarikh: **09 DISEMBER 2009**	Tarikh: : **09 DISEMBER 2009**

CATATAN:

* Potong yang tidak berkenaan.

** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai atau TERHAD.

♦ Tesis dimaksudkan sebagai tesis bagi Ijazah doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled "*Effect Of Different Turbulence Models In Cylinder Pressure During Engine Combustion Process*" is written by *Mohd Nasruddin Bin Amir Nazri*. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Mr. Amirruddin Bin Abdul Kadir

Examiner Signature

EFFECT OF DIFFERENT TURBULENCE MODELS ON COMBUSTION CHAMBER PRESSURE USING COMPUTATIONAL FLUID DYNAMIC (CFD)

MOHD NASRUDDIN BIN AMIR NAZRI

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

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature
Name of Supervisor: MR MOHD FADZIL BIN ADBUL RAHIM
Position: LECTURER
Date:

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurently submitted for award of other degree.

Signature
Name: MOHD NASRUDDIN BIN AMIR NAZRI
ID Number: MH06028
Date:

Dedicated to my parents

Respected father, Mr. Amir Nazri Bin Sambah

Loving mother, Mrs. Sh Radziah Bt Sy Ali

My beloved brother and sister

My precious friends

ACKNOWLEDGEMENTS

In the name of Allah swt, the Most benelovent and Most Merciful. Every sincere gratitude and appreciation in only to Allah swt. By only His Kindness, Guidance and Will that this thesis is finally realized. May this thesis will do some service to humanity. I would like to express my thanks to my supervisor, Mr Mohd Fadzil abdul Rahim for his confidence, supportiveness, and invaluable opinion for me throughout all the stages of this study. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proofreading and correcting my many mistakes. My sincere thanks go to all my labmates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to my classmates for their excellent co-operation, inspirations and supports during this study. I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I acknowledge the sincerity of my brothers and sisters, who consistently encouraged me to carry on my higher studies. I am also grateful to my fellow collegues for their sacrifice, patience, and understanding that were inevitable to make this work possible. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

This thesis deals with the numerical study about the effect of different turbulent models on combustion chamber pressure during the event compression and combustion process using Computational Fluid Dynamic (CFD). The assessment is based on cylinder pressure and computational time predicted by the turbulence models. The vital point for the study is the on effect of different turbulence models on simulating the critical process of combustion in cylinder. The most accurate and time efficient models is k-ω-sst. The predicted results produce 58.2358 % discrepancy in term of cylinder pressure. The model also predicted the shortest convergence time which is 1573 minute. The selection of the models must be right in using numerical modelling approach in order to fulfil three major criteria which are accuracy, computational time and cost. This study consists of numerical modelling by using Mitsubishi magma 4G15 as baseline engine design. Engine speed at 2000 rpm was selected as baseline for initial condition. This project simulates the compression and combustion process right after intake valve closed until exhaust opened. For numerical modelling approach, there were six turbulence models selected which are *k-ϵ-standard, k-ϵ-RNG, k-ϵ-realizable, k-ω-standard, k-ω-SST,* and *RSM-Linear Pressure Strain*. The pressure data for turbulent models validate by compared to the experimental data. However, there are discrepancies of the results due to improper boundary condition and inherit limitation of model. For further simulation of combustion process must consider detail mixture properties, detail boundary condition and model extension for better accuracy.

ABSTRAK

Tesis ini berkaitan kajian berangka tentang kesan daripada pelbagai model aliran gelora dalam ruangan kebuk pembakaran semasa pemampatan dan proses pembakaran berlangsung dengan menggunakan kaedah dinamik aliran berkomputer, Computational Fluid Dynamic (CFD). Penilaian ini berdasarkan pada tekanan silinder dan masa pengiraan yang diramal oleh model aliran gelora. Perkara penting dalam kajian adalah untuk melihat perbezaan pada setiap model aliran gelora mengsimulasi proses pembakaran yang kritikal dalam silinder. Yang paling tepat dan waktu pengiraan yang cepat ialah k-ω-SST. Keputusan ramalan menghasilkan 58.2358% perbezaan tekanan silinder. Model ini juga meramalkan masa konvergen tersingkat iaitu 1573 minit. Pemilihan model haruslah tepat dalam menggunakan pendekatan model berangka untuk memenuhi tiga kriteria utama yang ketepatan, perhitungan waktu dan kos. Kajian ini terdiri daripada pemodelan berangka dengan menggunakan Mitsubishi Magma 4G15 sebagai dasar bentuk mesin. Kelajuan enjin pada 2000 rpm terpilih sebagai garis dasar untuk kondisi awal. Projek ini mensimulasikan proses mampatan dan pembakaran selepas injap masuk tertutup hingga injap ekzos tertutup. Untuk pendekatan pemodelan berangka, terdapat enam model aliran gelora dipilih model iaitu *k-ϵ-standart, k-ϵ-RNG, k-ϵ-realizable,k-ω-standart, k-ω-SST,* and *RSM-Linear Pressure Strain*. Data tekanan untuk semua model aliran gelora disahkan dengan dibandingkan dengan data eksperimen. 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

LIST OF SYMBOLS

A & B	Empirical constant equal 4.0 & 0.5
$D_{i,m}$	Diffusion coefficient for species *ith* in the mixture
∂	Partial
δ_{ij}	Kronecker delta
ϵ	Epsilon
e	Specific total Energy
F_i	External body force from interaction with dispersed phase in *ith* direction
h	Sensible entalphy
h_j	$\int_{Tref}^{T} c_{P.J}$ Dt with T_{ref} = 298.15K
$J_{i,\,i}$	Diffusion flux of species *i*
k	Kinetic energy
k_{eff}	Effective conductivity
$\dot{m}$	Mass flow rate
m_j	Mass fraction of species *j*
ρ	Density

g_i	Garavitational body force
p	Static pressure
Ri	Net rate of production of species *i* by chemical reaction
S_h	Additional volumetric heat sources (example: heat of chemical reaction)
Si	Rate of creation by addition from the dispersed phase
t	Time
τ_{ij}	Stress tensor
μ	Fluid dynamic viscosity
u_i & u_j	The *ith* and *jth* cartesian component of instantaneous velocity
ω	Omega
Y_R	Mass fraction of a particular reactant R

LIST OF ABBREVIATIONS

CFD	Computational fluid dyamic
DNS	Direct numerical simulation
k-ϵ	K epsilon
k-ϵ -realizable	K epsilon realizable
k-ϵ-RNG	K epsilon renormalize group
k-ϵ-standart	K epsilon standard
k-ω	K omega
k-ω-SST	K omega shear stress transport
k-ω-standart	K omega standard
LES	Large eddy simulation
Pa	Pascal
RANS	Reynold averaging Navier-Stokes
RPM	Revolution per minutes
RSM	Reynold stress model
RSM-LPS	Reynold stress model linear pressure strain
TDR	Turbulence disipation rate
TKE	Turbulence kinetic energy
SDR	Specific dissipation rate

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Turbulence is that state of fluid motion which is characterized by apparently random and chaotic three-dimensional vorticity. When turbulence is present, it usually dominates all other flow phenomena. Turbulence can be seen in most cases in daily life such flow at buildings, cars, airplanes, fans, combustion chamber and many more. The successful of turbulence modeling increase in numerical simulation (Sodja, 2007). In these past years, many problems that involve turbulence flows are solve by using CFD for example fluid mixture, internal and external flows and in-cylinder flows. CFD approach provides user for gaining insight into in-cylinder flow (Payri et al., 2003). The view can be one of the result interpretations because the different is significant. The main importance of CFD approach is to attributes of both accurate and computationally fast to solution time (Kulvir et al., 2004). Hence, time consuming is crucial since the standard processor is just average rather that high capability processor that being used in high level or industry. However, that result should be acceptable in order to valid the CFD approach. After all, uncertainty of mathematically modeling turbulence is reflected in the large variety of models available (Kulvir et al., 2004). From here, the problem of choosing the right turbulence models due to right problems in terms of processing time and accuracy is important.

1.2 PROBLEM STATEMENT

From the findings, there are lots of turbulence models that available. But, the problem comes when selecting the right models for the right problems. Therefore, deciding the right turbulence models is not simple. The other concern is to reduce the amount of time that consume during the calculation process. So, the problems are to comparing turbulence models which is suit for in-cylinder flow and combustion study. Particularly, the purposes are to study the effect of turbulence models in term of accuracy to computational time.

1.3 OBJECTIVE

The objectives of this project are:

- To study the effect of different turbulence models on combustion pressure.
- To compare and validate each turbulence model's prediction with experimental data.

1.4 SCOPES

The scope of study covered the study and analysis on the effects of turbulent models and the accuracy due to processing time. Details scopes of this project consist of the following:

- To simulate in-cylinder flow using CFD approach during both valves closed.
- Develop the 2D pent-roof and combustion chamber model based on Mitsubishi Magma 4G15 engine dimension.
- Grid generation and boundary condition setup.
- Simulation of several turbulent models.
- Validate CFD approach by compare pressure data with experimental data.

1.5 ORGANIZATION OF THESIS

This thesis consists of five main chapter, introduction, literature review, methodology, result and discussion and the last part is conclusion and recommendation. For Chapter 1 presents some findings that lead to problems statement, objective, scopes and flow chart of work. Chapter 2 is literatures that related to the study and become basic of study framework. Chapter 3 presents the dimensioning work on Mitsubishi Magma 4G15 engine, development of 2D model and generation of computational domain. The pre-processing setup is presented in order to attain grid generation and imported to the solver to analyze. Chapter 4 addresses the validation of the predicted results against experimental results of the cylinder pressure. Chapter 5 presents the important findings of the study and recommendation for future study.

1.6 FLOW CHART

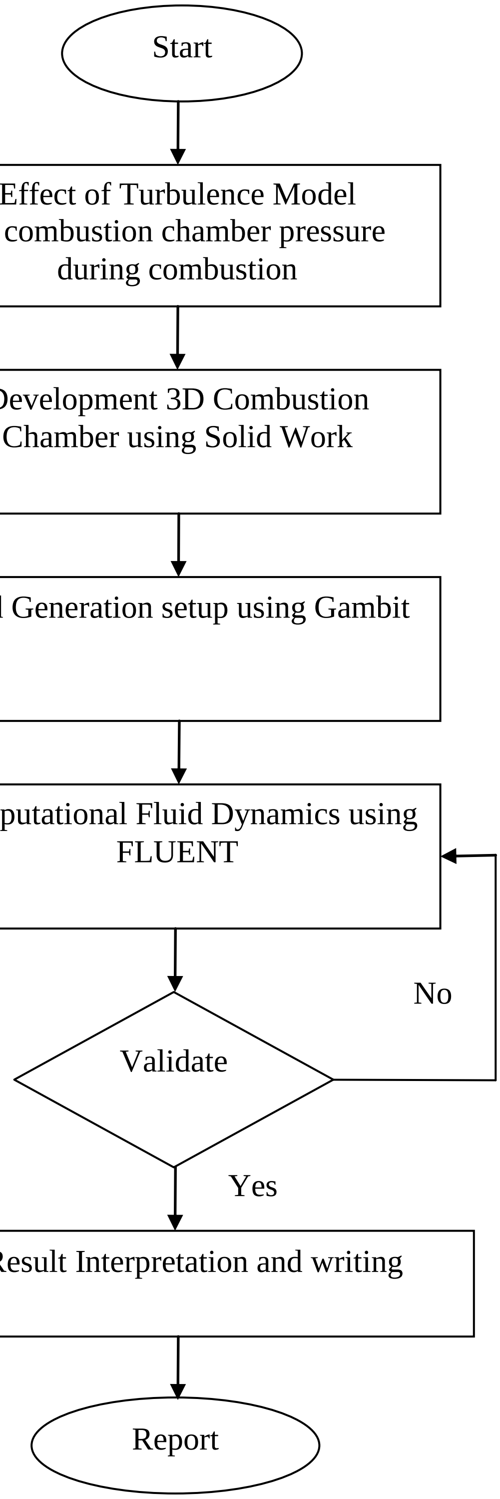

Figure 1.1: Project flow chart

1.7 SUMMARY

The purpose of the study is to acquire the main objective of the study related to the effects of different turbulence models. This chapter has summarized the titles, objective, scope, methodology, and the validation of study.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter deals with definition and characteristic of turbulence. Then, this chapter continues with the application of turbulence flow in in-cylinder flow study and the importance of the study about turbulence model for in-cylinder flow. Lastly, discussions continue with CFD approach for in-cylinder flow modeling and the advantages of CFD modeling for in-cylinder flow study.

2.2 TURBULENCE FLOW

In around 1500, Leonardo Da Vinci once thought about turbulence and draw called “La Turbulenza”. Leonardo describe turbulence as “Observe the motion of the surface of the water, which resembles that of hair, which has two motions, of which one is caused by the weight of the hair, the other by the direction of the curls; thus the water has eddying motions, one part of which is due to the principal current, the other to the random and reverse motion” (Ecke, 2005). So, it is understandable that turbulence has been long time studied and what has Leonardo quote is included in one of turbulence characteristics.

So, turbulence can be described as that state of fluid motion which is characterized by apparently random and chaotic three-dimensional vorticity. When turbulence is present, it usually dominates all other flow phenomena and results in increased energy dissipation, mixing, heat transfer, and drag (Sodja, 2007). If there is no three-dimensional vorticity, there is no real turbulence. There is no specific definition of turbulence model, but it has several characteristic features (Davidson, 2003), (Ziya, 2003), (Uygun et al., 2004) such as:

- Irregularity – As we all know, turbulence is random and chaotic. Turbulence flow is not constant respect to time. The flow consist of different scales of eddies sizes and fluctuate over time.

- Diffusivity – Turbulence flow increase in exchange the increment of momentum. As the turbulence flow increase, it will diffuse and become widely dispersed or spread out. The relation between resistances of friction to the diffusivity is vice versa. When one is increase, the other one is decrease.

- Large Reynolds Numbers – The basic knowledge that turbulence flow only happened only at high Reynolds number. Take fluid flow in pipes for example, transition happen at $Re \approx 2300$ and the turbulence flows start at $Re \approx 10000$.

- Three-Dimensional – This crucial characteristic is very important because turbulence flow is always three-dimensional. The flow is unpredictable and random. Even so, the equation is time averaging so that it can be solve easier.

- Dissipation – Turbulence flows are dissipative, which means the small (dissipative) eddies turns into internal energy. The smaller eddies receive the kinetic energy from larger eddies. The largest eddies get the energy from the main flow. This process that transfer the energy from main flow to the smallest eddies called cascade of energy as shown in Figure 2.1.

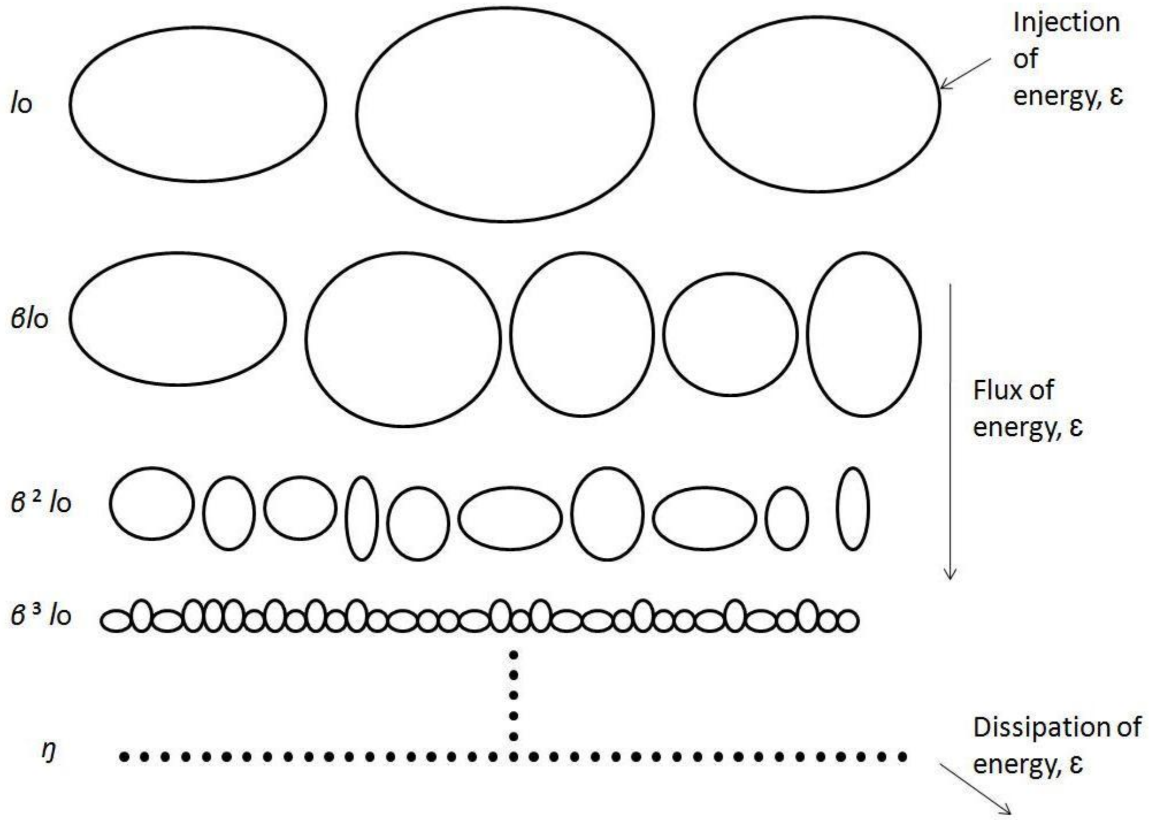

Figure 2.1: Energy cascade of turbulence.

Source: Ecke, (2005)

Since turbulence appears to most in our daily life, the effects of turbulence models are important since it is closer to nature and real cases. By the study the behavior of turbulence flows, the prediction of the desired result acquired by taking any precaution and initial awareness into study. This is important because in any cases such disasters, forecast and internal flow are amongst the need to predict in order to avoid such unwelcome accident. Industry and chemical process also involve fluid flows in packed beds (Gou et al., 2003). The distribution during the process is crucial to fulfill the criteria that demanded. It shows that the wide range of turbulence applications in the new era's.

2.3 TURBULENT MODEL

The most efficient approach to solve turbulence flow is by modeling by based on numerical simulation. By this approach, all fluid motion can be resolve into prediction. Computational on turbulence models can be classified into several models.

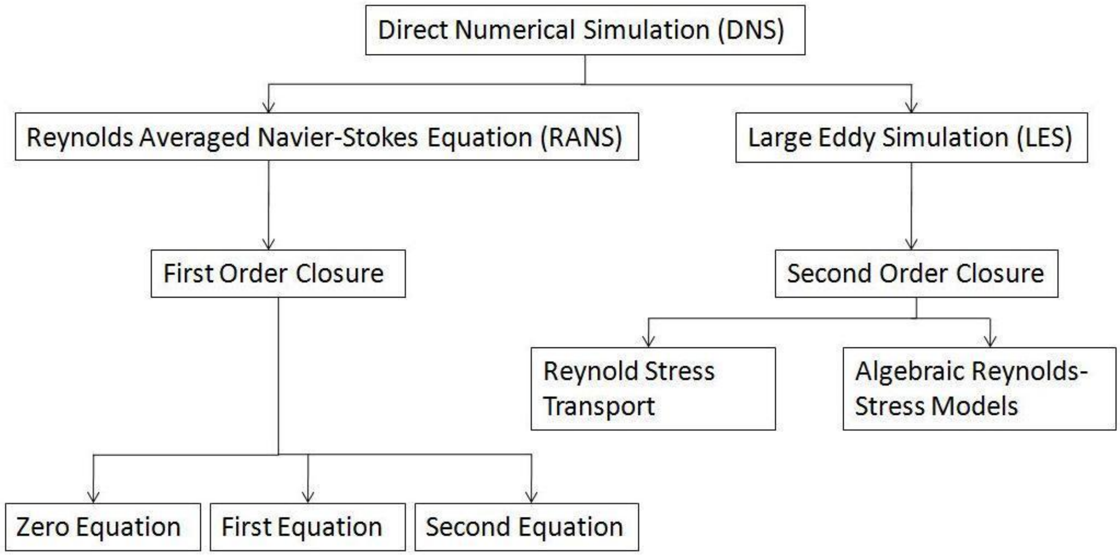

Figure 2.2: Turbulent models classification.

Source: Uygun et al., (2004)

As we can see from Figure 2.2, the turbulence models build from several classes. The classifications were made by previous researcher Uygun et al., (2004) based on result that computed, application, and complexity of the problems. From Figure 2.3, the simplest form of resolving turbulence is only solved the large eddies and modeled the effect of flux energy and dissipation of energy.

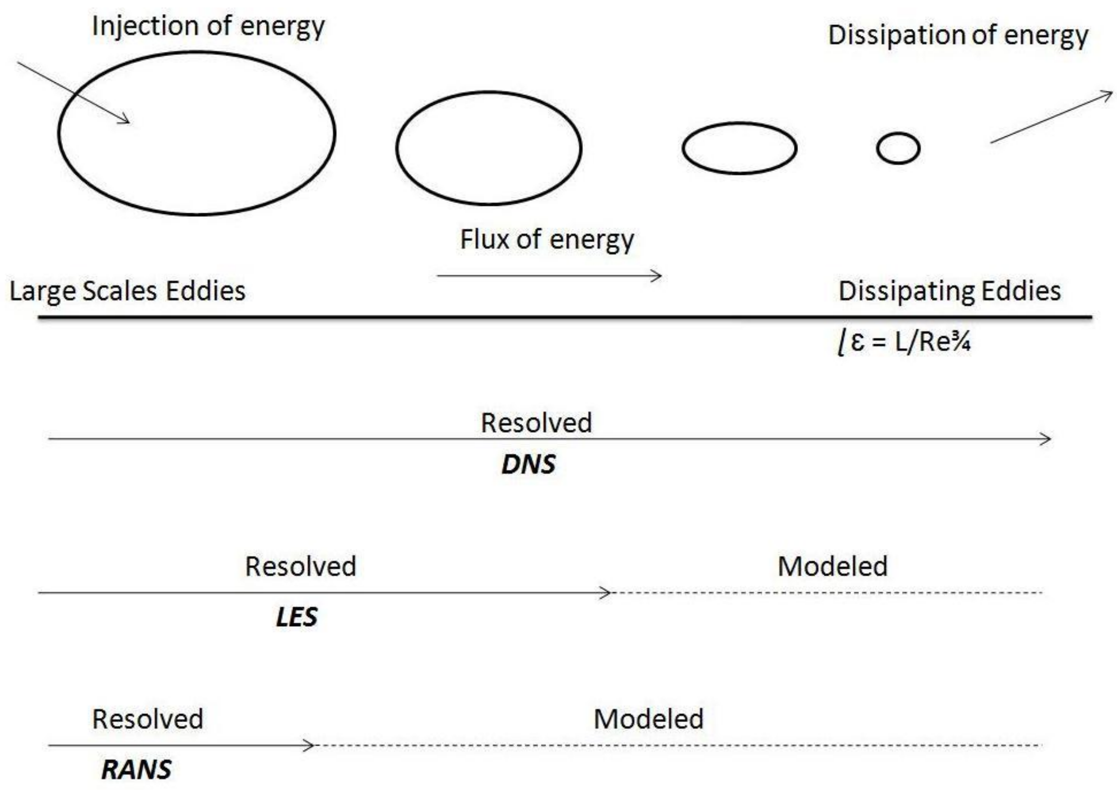

Figure 2.3: Extention to modeling for certain types of turbulence models.

Source: Sodja, (2007)

DNS is the most accurate method to solve turbulence flow (Uygun et al., 2004). This is because DNS does not need time averaging but solve the problem by numerical discretization. Hence all time and length scales are resolved. The solved problem is equivalent to those that attained by experimentally (Vengadesan and Nithiatasu, 2007). So, the accuracy level shown by DNS is idealized since the computed result is accurate as experiment. However, in order to capture all the turbulence scales, the computational domain must be as large as the physical domain or as large as the largest turbulence structure such eddy. It is important because to take into account every turbulence scales, the domain must be very fine grid. Usually, DNS used for simple geometries and to low Reynolds numbers (Vengadesan and Nithiatasu, 2007). From Figure 2.3, DNS solved all turbulence scales. Keeping in mind the relation the cost of a simulation goes up as processing time and grid size goes up. That is why DNS is so demanded method in term of cost and processor. Figure 2.4 show that the different eddies sizes under consideration during turbulence modeling.

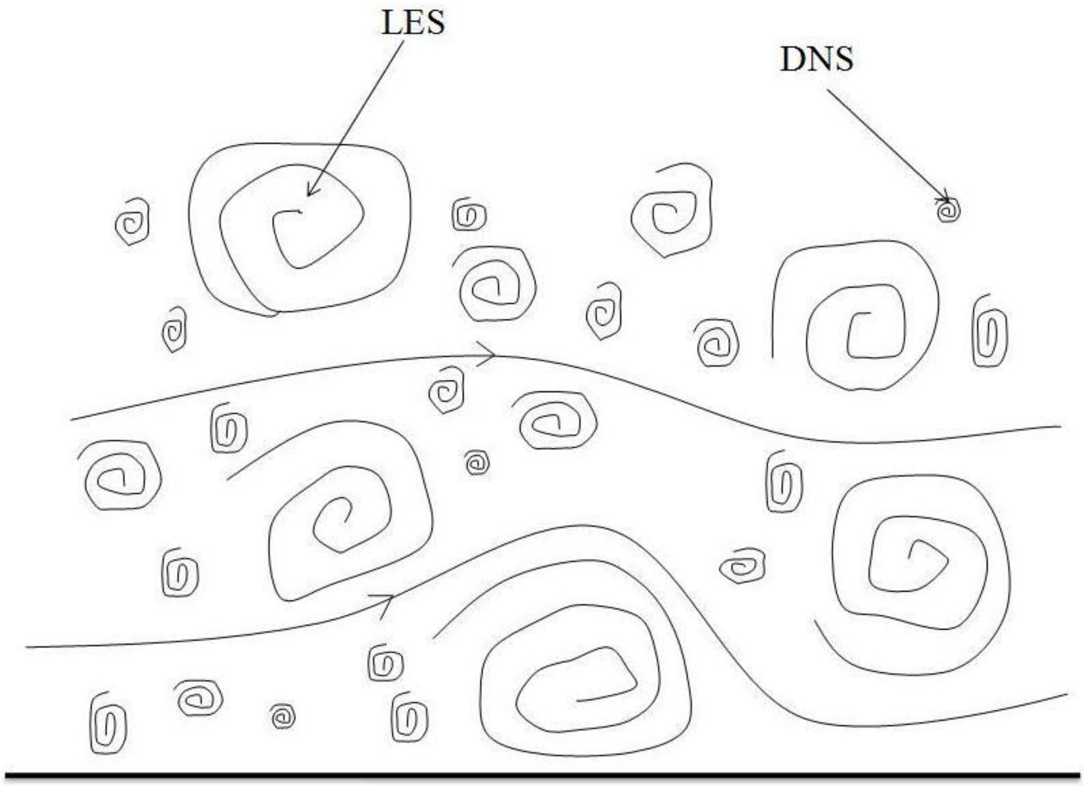

Figure 2.4: Large and small eddies.

Source: Uygun et al., (2004)

For LES, the observation based on large eddies that carries more energy then the smaller (Uygun et al., 2004). The subgrid-scale model used to simulate the energy transfer between the large eddies and the subgrid eddies (Uygun et al., 2004). The energy transport happen during cascade of energy process that continuum until the large eddies turns smaller eddies. That is why the size and energy make them effective for transportation of flow properties through interest. By referring to Figure 2.3, LES solve most of turbulence flow that consists of large scales and medium and modeled the small ones. After certain sizes of eddies, LES modeled the rest of turbulence flows. Even LES is considerable cheaper than DNS, LES still requires higher grid resolution in both the in order to solve the problems. By refer to Figure 2.4, LES solve only the large sizes of eddies that carries more energy, but DNS solve scales and size of all turbulence. That is why DNS far more accurate than LES but required higher cost and processing time.

Based on Figure 2.2, RANS can be divided into two main group, first order closure and second order closure. The discussion will follow those group and focusing on first-order closure.

- Algebraic models: These models contribute to the mixing length model in different ways and their models are the most popular amongst other algebraic models (Ziya, 2003). Examples of algebraic model are Cebeci-Smith model and Baldwin-Lomax model.

- One-equation models: Further improvement from previous models. There some interest in one-equation models of turbulence due to accuracy, simplicity of implementation and less demanding computational requirements (Ziya, 2003). Examples of one-equation model are Sparlat-Allmaras model and Baldwin-Barth model.

- Two equation models: The two–equation models have made truly significant contribution by introducing the famous k-e model. Then, Wilcox have pursued further development and presented successful application of k-e model (Ziya, 2003). Examples of two-equation model are k-ϵ and k-ω.

- Second order closure models: Right after the age of computer merge into new century, most improvements to model were abrupt these model shows some advantageous in sense that automatically accommodate complicating effect such streamlines curvature, rigid body rotation and body forces. However, because of large number of extra partial differential equations, complexity and computational cost is also increase as the demanding computer applicability (Ziya, 2003). Example of second order closure modes are Reynolds-Stress Transport and Algebraic Reynolds-Stress Models.

RANS models based in time-averaging of the dependent variables and the governing equations (Schluter et al., 2005).Technique solves the governing equation by modeling both large and small eddies, taking time-averaging of variables. From Figure 2.3, RANS is modeled the flows, that is why information supplied by these models is the time average of the variable and the fluctuating part. RANS is not represented directly by the numerical simulation, and are included only by means of turbulence models. These models have been extensively used for scientific and engineering calculations during the last decades. There are specially designed for high Reynolds numbers and distinguish separation of time scales related to the fluctuating behavior. Note that from Figure 2.3, the main advantages is the relative low computational cost involved compared DNS and LES since RANS mostly modeled the flows (Uygun et al., 2004). The bottle neck of these models is the difficulty to obtain highly accurate in addition to universally applicable models.

Nowadays, engineer and scientist are move towards to achieve the main objective to complete to the end the unsolved problems. Hence, the most accurate approach to turbulence simulation to directly the governing transport equations without undertaking any averaging or approximation other than the numerical discretization that performed (Tu et al., 2008). Through simulation, those turbulence flow that tested are solved by taking account some parameter to validate even so simulation is just a prediction.

From here, DNS show the most accurate method in CFD but highly cost and need very fine grid. So, LES is overtake by taking large eddies into account since large eddies carries massive energy. Even so LES is cheaper than DNS, but when compared to RANS reliability, LES is quite cost and demanding processor. So, LES modeling has problems with boundaries and is less computationally efficient than RANS techniques. RANS generally, k-ϵ especially is the most efficient in term of computational cost, time processing and processor demand. Even the result that obtained is not exactly same as DNS, but still acceptable and well known in engineering problems (Ziya, 2003).

2.4 TURBULENCE IN-CYLINDER FLOW

In-cylinder process model is simulating the full condition that in charge such thermodynamic cycle that containing spark ignition, turbulent flame propagation, heat dissipation, emission and knocking (Bi et al., 1994). Turbulence flow in-cylinder is important because variety of parameter that affect the consequences to the engine itself such emission, performance, durability, endurance and efficiency. Study showed that piston geometry is important in order to swirl the air-fuel mixture in combustion chamber (Hovart and Hovart, 2003). However, bowl shape plays significant roles near TDC and the early stages of expansion stroke by controlling ensemble-averaging mean and turbulence velocity (Payri et al., 2003).

During the intake stroke, air-fuel mixture is flowing through the intake manifold into combustion chamber. Relationships between flow structures within the runner and cylinder were seen to be strong during the intake stroke but less significant during compression (Justham et al., 2005). The in-cylinder flow diagnostics have been established in these few decades that provides greatly amount of information of flow and it is turbulence characteristics. By study and measure does improve combustion performance and help to understand engine performance. Researcher also noted that turbulence characteristic and intensity does make significant influence on combustion that is why accurate turbulence measurement is really important task (Kaneko et al., 1999).

From previous approach by researcher, turbulence model that used is RANS widely, followed by LES and DNS rarely. For RANS, k-ω model and k-ϵ model are used commonly since both gives inadequate result (Ogor et al., 2006). The requirement of processor to run RANS also lower and the running time is faster than LES and DNS this is another important key points why RANS used widely in CFD analysis (Sodja, 2007). Although RANS is faster and reliable, for high value and very important CFD analysis, DNS and LES usually used in order to achieve the accurate result that idealized for most engineering application (Venayagamoorthy et al., 2003). As far as studied carried on, the selection amongst turbulence model due to condition that went to analysis is still depends on several parameter such processor ability, accuracy, running time and the complexity of geometry.

However, some study combined both method of RANS and LES (Venayagamoorthy et al., 2003) and (Ogor et al., 2006). Researchers attempt to try keeps the computational efficiency of RANS and the potential of LES to resolve larger turbulence structures that build of more coarser grid and with high Reynolds number. Some larger turbulence flows solved by using VLES (Very Large Eddies Simulation) for certain cases. Those cases usually adapted into something massive scale and un-experimental testing that may cause hazardous, cost and damage.

Mostly two-equation model such k-ω model and k-ϵ model used in in-cylinder flow study. Essentially RANS especially k-ϵ model capable to model the cascade of energy process of turbulent kinetic energy, and to resolve the more complex details such as separating and reattaching flow, which is one of the major problems (Mumovic et al., 2004). Hence, no wonder why k-ϵ model is most popular amongst other turbulence model even it has considerable disadvantages such as accuracy which is not comparable to DNS.

2.5 A CFD APPROACH FOR IN-CYLINDER FLOW MODELING

Noted that CFD approach in fluid analysis is not something new especially in-cylinder flows. Previous study show and prove that CFD approach in in-cylinder flow analysis is a success. Years ago, in-cylinder modeling become favorite in CFD analysis because easier and faster (Payri et al., 2003). From previous experimental method, it is understandable that in-cylinder flow analysis does cause high cost and technologies even so, the result is highly precise due data taken based real condition.

Since CFD approach is simulating the problems by modeling, CFD code provides a real insight inside cylinder to see the fluid flow behavior (Semin et al., 2008). After the result is obtained and acceptable, result interpretation is visualize in form of graph, images and table. CFD approach has an impressive graphic as one of data interpretation. Note that CFD does have extra advantages by simulating the problems.

Researcher easily can compared the fluid flow inside the duct, combustion chamber, pipes and few more by observing and differentiate the result appear.

In CFD approach also the domain start to be meshed. These important tasks of this part affect the flow work and the result accuracy. The hexahedral cells type becomes popular since they provide better accuracy and stability than tetrahedral cells (Payri et al., 2003). These means the application and selection of grid on surface is not easily done. The accuracy and sensitivity of the grid must be considered as well as the final outcome. The selection of cells also usually depends on part of domains. Based on Figure 2.5, the hexahedral mesh applied at intake manifold. As the manifold bend, the cells selection changed to tetrahedral and for moving boundary, the mesh selected is wedge cells (Bai and Hsiao, 2007).

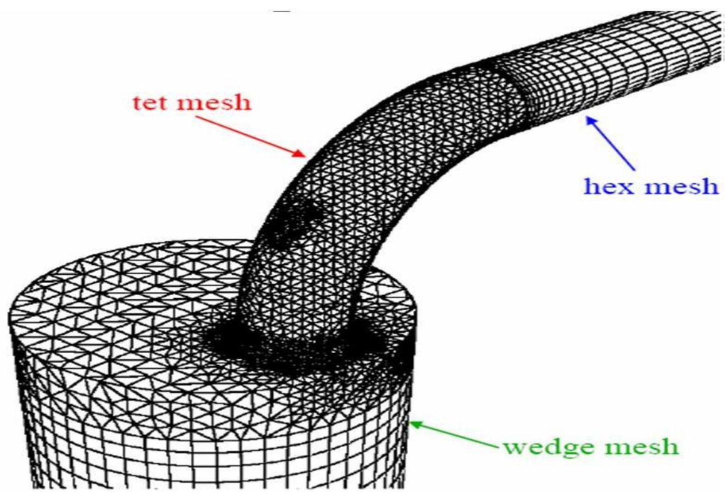

Figure 2.5: Hybrid mesh for IC engine value port.

Source: Bai and Hsiao, (2007)

It is understandable that the selection of mesh is very important due to CFD approach. As researcher expecting the best outcome, the consideration starts from meshing and grid generation. The good mesh selection is half of the CFD analysis done (Jain, 2008). Previous researcher tried good mesh for example hybrid meshed for improving accuracy and applicable of the mesh applied.

Moreover, the application on CFD analysis for in-cylinder flow does not stop in fluid flow only. There are various application such examine the direct injection in diesel engine (Payri et al., 2003), the numerical optimization of in-cylinder process (Colucci et al., 2002), and spatial evaluation (Kaneko et al., 1999). These shows that CFD approach is a success for in-cylinder flow analysis and relevance.

2.6 ADVANTAGES AND DISADVANTAGES OF CFD IN IN-CYLINDER FLOW ANALYSIS

As mention before, even so the experimental method is highly accurate in data measurement and more over experimental method is real analysis that all real parameter taking into account, it is still highly cost and need high technologies. So, CFD approach is the suits the best especially for who are have low budget and lack of high technologies.

Notes that before, CFD methods is capable to do fluid analysis such as in-cylinder, pipelines, vehicle aerodynamics, heat distribution and lot more. The entire situation are considered and made into models to simulate in CFD analysis. In real situation, turbulence flows dominate the fluid conditions. That is why turbulence modeling approach is the most successful in numerical simulations (Sodja, 2007).

CFD analysis also used widely in in-cylinder flow analysis without doubting that experimental method because of the result that obtained is acceptable and reliable. Previous researchers have proven that CFD analysis in in-cylinder flows analysis is compatible and almost perfect to real situation that analysis in-cylinder flow (Payri et al., 2003), (Semin et al., 2008), (Bi et al., 1994), and (Colucci et al., 2002).

CFD analysis simulation provided a real insight into the cylinder flow behavior of the separate fuel and air streams entering the cylinder (Semin et al., 2008). This is one of the reasons why CFD analysis are used in in-cylinder analysis because researcher can see right thru inside the unseen part such combustion chamber, manifold, turbine, compressor and more. Researcher can understand more the behavior of the flow inside based on interest such intensity of pressure or velocity.

Even so CFD used in in-cylinder flow analysis widely, aware that the processing time usually long. So it is crucial due to accuracy of the data measurement. Moreover, the time taken is longer especially complex design and geometry (Payri et al., 2003). Moreover, high performance personal computer or processors are the main requirement to run such detailed and high accuracy geometry. That is why large company that using CFD are using super computer that can run faster and proves such complex geometry.

2.7 SUMMARY

The application of CFD is not limited to developing something, but to simulate real cases that may hazardous and more worst that can take lives such flood, toxic gas, fire, smoke, and more. Focus of the study is on effect of different turbulence models on in-cylinder flows. The model was meshed based on complexity of design. Then, the numerical model set up for solving the turbulence models. Lastly, the data that obtained contain numerous information that need to be extract for proper and neat presentation. All of the data need to interpret and validate based on previous and experiment data so that the CFD analysis is acceptable.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter presents the main outline of the study which contains engine baseline specification, important parameters, initial condition, boundary condition, numerical modeling approach, numerical analysis and the validation method.

3.2 Baseline Engine Specification

Table 3.1: Engine specification Mitsubishi Magma 4G15.

Source: Fadzil, (2008)

Parameter	Size and Feature
Combustion chamber type	Pent-Roof type
Piston bore (mm)	75.5
Piston stroke (mm)	82
Compression ratio	9.2
Intake valve open/closed	15^0 BTDC/63^0 ABDC
Exhaust valve open/closed	57^0 BBDC/13^0 ATDC

From Table 3.1, Mitsubishi magma 4G15 is taken as engine baseline to completing this project. The bore is 75.5 mm and the stroke 82.0 mm. Since the engine combustion chamber type is pent-roof type, the development of top of combustion chamber is pent-roof type. From the table, the other important key of developing the computational domain is the compression ratio. The domain must obey the compression ration which is 9.2 to simulating as it is. The connecting rod for the piston is 129 mm.

3.3 FULL CRANK ANGLE EVENT

Table 3.2: Full crank angle event.

Source: Fadzil, (2008)

Crank angle	Events
-360^0	Start of intake process (intake & exhaust valves already opened) at TDC
-347^0	Intake process (Exhaust valves closed & exhaust manifold deactivation)
-180^0	End of intake process/Start of compression process
-117^0	Compression process (intake valves closed & intake manifold deactivation
-23^0	Timing of ignition (for 2000 rpm)
0^0	End of compression/start of power stroke
123^0	Power stroke (exhaust valves opened & exhaust manifold activation)
180^0	End of power stroke/start of exhaust stroke
345^0	Exhaust stroke (intake valves opened & intake manifold activation)
360^0	End of single cycle

Table 3.2 shows the full crank angle event. Based on the table, four important events that involved in this simulation. The simulation of this project start right after intake valves closed which is -117^0 or 63^0 ABDC. Then, the compression stroke continued until TDC with the crank angle duration 94^0. Spark ignition was set -23^0 BTDC to initiate the spark to continue the next process which is power stroke. After

TDC at 0^0, piston completes its power stroke with 123^0 crank angle right before exhaust valves opened.

3.4 GOVERNING EQUATION FOR COMPUTATIONAL FLUID DYNAMICS

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.

3.4.1 Mass Conservation Equation

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

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

where

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

The equation is valid for the incompressible and compressible flow. Moreover, the rate generated in the system, $\dot{m}$ 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 (Fluent, 2004):

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial xj}(\rho u_i u_j) = \frac{\partial p}{\partial x\mathrm{i}} + \frac{\partial \tau \mathrm{ij}}{\partial x\mathrm{j}} + \rho g_i + F_i \tag{4.2}$$

where

ρ	=	Fluid density
u_i & u_j	=	The *ith* and *jth* 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 dispersed phase in *i* direction

The stress tensor in Equation 4.2 is given as (Fluent, 2004):

$$\tau_{ij} = \mu\left(\frac{\partial u i}{\partial x j} + \frac{\partial u j}{\partial x i}\right) - \frac{2}{3}\mu\left(\frac{\partial u k}{\partial x k}\right)\delta_{ij} \tag{4.3}$$

where

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

Note that the second term on the right hand side of Equation 4.4 describes the effect of volume dilation. By substituting Equation 4.4 into Equation 4.3, another equation is produced that is complete momentum conservation equation (Fluent, 2004):

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial xj}(\rho u_i u_j) = - \frac{\partial p}{\partial x\mathrm{i}} + \frac{\partial p}{\partial x\mathrm{j}}\left\{\mu\left(\frac{\partial u i}{\partial x j} + \frac{\partial u j}{\partial x i}\right) - \frac{2}{3}\mu\left(\frac{\partial u k}{\partial x k}\right)\delta_{ij}\right\} \rho g_i + F_i \tag{4.4}$$

3.4.3 Energy Conservation Equation

$$\frac{\partial}{\partial t}(\rho e) + \frac{\partial}{\partial x i}[u_i(\rho e + p)] = \frac{\partial}{\partial x i}[k_{eff}\frac{\partial T}{\partial x i} - \sum_j h_j J_j + u_j(\tau_{ij})_{eff}] + S_h \quad (4.5)$$

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 entalphy
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:

$$e = h - \frac{p}{\rho} + \frac{ui^2}{2} \quad (4.6)$$

sensible entalphy for ideal gas is defined as :

$$h = \sum_j m_j h_j \quad (4.7)$$

sensible entalphy for incompressible flow is defined as :

$$h = \sum_j m_j h_j + \frac{p}{\rho} \quad (4.8)$$

where

m_j = mass fraction of species j

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

3.4.4 Species Conservation Equation

When choosing to solve conservation equation for chemical species, the prediction of the local mass fraction to each species, m_i, through the solution of a convection-diffusion equation for the *ith* species. This conservation equation takes the following general form.

$$\frac{\partial p}{\partial t}(\rho m_i) + \frac{\partial}{\partial xi}(\rho u_i m_i) = -\frac{\partial}{\partial t} J_{i,\ i} + R_i + S_i \tag{4.9}$$

where

Ri = Net rate of production of species *i* by chemical reaction

Si = Rate of creation by addition from the dispersed phase

$$J_{i,\ i} = -\rho D_{i,\ m} \frac{\partial mi}{\partial xi} \tag{4.10}$$

where

$J_{i,\ i}$ = Diffusion flux of species *i*

$D_{i,\ m}$ = Diffusion coefficient for species *i* in the mixture

The reaction rates that appear as source terms in Equation (4.9) are computed by eddy dissipation model. The reaction rates are assumed to be controlled by the turbulence instead of the calculation of Arrhenius chemical kinetics. The net rate of production for species *i* due to reaction *r* , is given by the smaller of the two expressions below:

$$R_{i,\,r} = \upsilon'_{i,r} M_{w,i} A \rho \frac{\varepsilon}{k} \min\left(\frac{YR}{\upsilon''_{j,r} M_{w,j}}\right) \tag{4.11}$$

where

Y_R = Mass fraction of a particular reactant R

A & B = Empirical constant equal 4.0 & 0.5

3.5 GRID GENERATION AND DOMAIN CREATION

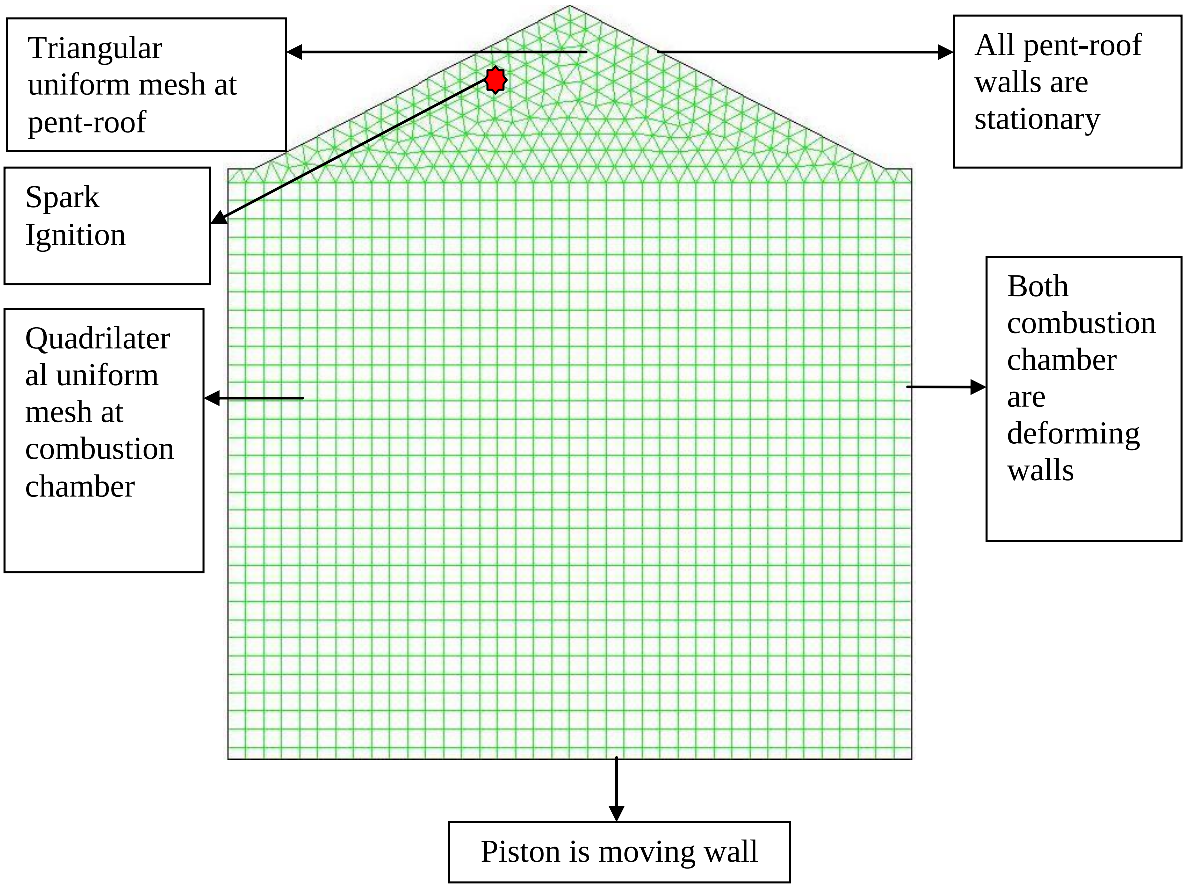

Figure 3.1: Computational domain

The model is developed in Solid Work based on engine specification in Table 3.1. Then, the model was imported to GAMBIT to start the grid generation. The model was meshed as triangular at the pent-roof and as quadrilateral at combustion chamber as shown by Figure 3.1. Mesh size for both pent-roof and combustion chamber is 2mm and uniform mesh is used. The total mesh density per unit area is 24.15 cells/cm^2.

3.6 BOUNDARY CONDITION SETUP

Noted that domain boundary is sets as walls. The materials setting for all walls are aluminum. The walls are created as deforming boundary to simulate the compression and power stroke. The pent-roof walls are set as stationary walls. For both right and left combustion chamber are set as deforming walls which is following the compression and power stroke. The piston is set as moving wall in order to simulate the movement of the piston in combustion chamber during compression and power stroke. Constant engine speed is compulsory for this simulation and 2000 rpm selected as uniform engine speed for all six turbulence models. This is essential because to maintaining the engine speed among turbulence models to see the contradiction in term of pressure. The other main part of boundary condition setting is temperature along the walls. So, temperature setup followed the actual condition as presented in Table 3.3

Table 3.3: Boundary condition at 2000 rpm.

Source: Fadzil, (2008)

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

3.7 SOLUTION SETUP

3.7.1 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 3.4: Initial condition at 2000 rpm.

Source: Fadzil, (2008)

Initial Condition	Value
Pressure	101325 Pa
Temperature	300 Kelvin
Progress variable	0
Engine speed	2000 RPM
Crank angle duration	240^0

Pressure and temperature are set as 101325 Pa and 300 K because to simulate the combustion chamber pressure and temperature at ambient condition by natural aspirated engine right after the intake valves closed. Since the air-fuel mixture is not burn, the progress variable must start with 0 or in other word completely not burn. For this project, the constant engine speed was taken as 2000 rpm for all turbulence models. The total crank angle event for this simulation is 240^0 where the simulation start right after the intake valves closed until exhaust valves opened which involved compression and power stroke.

3.7.2 Input Data For Premix

Table 3.5: Input data for premix-mixture properties.

Source: Fadzil, (2008)

Properties	Value	Units
Specific heat	0.08207936	J/kg.K
Thermal conductivity	4.10317×10^{-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	4.43×10^{7}	J/kg

From the Table 3.5, the value was calculated based on experimental data collection. From Table 3.5 shown, the value was set at the simulation start.

3.8 TURBULENCE SPECIFICATION

Numerical modeling approach consists developing the domain and grid generation that discussed earlier. Now, the numerical modeling approach is more to the main focus where the effect of different turbulence model takes place. In this project, they were six turbulence models that have been used. The models are *k-ϵ-standard, k-ϵ-RNG, k-ϵ-realizable, k-ω-standard, k-ω-SST,* and *RSM – LPS*. Each models give different effect due to each models bring specific and special equation to solve the specific problems.

3.8.1 k-ϵ-standard

Turbulence kinetic energy

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k \tag{4.12}$$

Turbulent dissipation rate

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}\left(P_k + C_{3\epsilon}P_b\right) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon \tag{4.13}$$

3.8.2 k-ϵ-realizable

Turbulence kinetic energy

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k \tag{4.14}$$

Turbulence dissipation rate

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + \rho C_1 S\epsilon - \rho C_2\frac{\epsilon^2}{k + \sqrt{\nu\epsilon}} + C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}P_b + S_\epsilon \tag{4.15}$$

3.8.3 k-ϵ-RNG

Turbulence kinetic energy

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\epsilon \tag{4.16}$$

Turbulence dissipation rate

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}^*\frac{\epsilon}{k}P_k - C_{2\epsilon}\rho\frac{\epsilon^2}{k} \tag{4.17}$$

3.8.4 k-ω-standard

Turbulence kinetic energy

$$\frac{\partial k}{\partial t} + U_j\frac{\partial k}{\partial x_j} = \tau_{ij}\frac{\partial U_i}{\partial x_j} - \beta^* k\omega + \frac{\partial}{\partial x_j}\left[(\nu + \sigma^*\nu_T)\frac{\partial k}{\partial x_j}\right] \tag{4.18}$$

Specific dissipation rate

$$\frac{\partial \omega}{\partial t} + U_j\frac{\partial \omega}{\partial x_j} = \alpha\frac{\omega}{k}\tau_{ij}\frac{\partial U_i}{\partial x_j} - \beta\omega^2 + \frac{\partial}{\partial x_j}\left[(\nu + \sigma\nu_T)\frac{\partial \omega}{\partial x_j}\right] \tag{4.19}$$

3.8.5 k-ω-SST

Turbulence kinetic energy

$$\frac{\partial k}{\partial t} + U_j\frac{\partial k}{\partial x_j} = P_k - \beta^* k\omega + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_k\nu_T)\frac{\partial k}{\partial x_j}\right] \tag{4.20}$$

Specific dissipation rate

$$\frac{\partial \omega}{\partial t} + U_j\frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta\omega^2 + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_\omega\nu_T)\frac{\partial \omega}{\partial x_j}\right] + 2(1 - F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i} \tag{4.21}$$

3.8.6 RSM

Transport equation

$$\frac{\partial}{\partial t}\left(\rho\overline{u_i'u_j'}\right)+\frac{\partial}{\partial x_k}\left(\rho u_k\overline{u_i'u_j'}\right)=-\frac{\partial}{\partial x_k}\left[\rho\overline{u_i'u_j'u_k'}+\overline{p'\left(\delta_{kj}u_i'+\delta_{ik}u_j'\right)}\right]$$
$$+\frac{\partial}{\partial x_k}\left[\mu\frac{\partial}{\partial x_k}\left(\overline{u_i'u_j'}\right)\right]-\rho\left(\overline{u_i'u_k'}\frac{\partial u_j}{\partial x_k}+\overline{u_j'u_k'}\frac{\partial u_i}{\partial x_k}\right)-\rho\beta\left(g_i\overline{u_j'\theta}+g_j\overline{u_i'\theta}\right)$$
$$+\overline{p'\left(\frac{\partial u_i'}{\partial x_j}+\frac{\partial u_j'}{\partial x_i}\right)}-2\mu\overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_j'}{\partial x_k}}-2\rho\Omega_k\left(\overline{u_j'u_m'}\epsilon_{ikm}+\overline{u_i'u_m'}\epsilon_{jkm}\right)+S_{user} \tag{4.22}$$

3.9 VALIDATION METHOD

After the simulation done, the graph plotted. The result compared with experimental data in for validation purpose. The result for turbulence models are plotted along with the experimental data. By do so, the graph is studied based on pressure different among those turbulence models and experimental data. The other interest is turbulence kinetic energy and mass fraction burned also studied in order to see the effect of turbulence models.

3.10 LIMITATION OF STUDY

Since this project is only based on simulation using CFD approach. Several parameters are not achieved at specific and exact event. The design consideration due to limitation of processor capability is crucial. Hence, several main parts in simulation procedure are not considered especially mesh sensitivity and real combustion chamber domain in 3D. In order to do that, the processor must capable to do the grid generation in fine mesh and applied to 3D domain.

The design method also included as limitation of study because the real combustion engine is takes as 2D, not 3D as real combustion engine specification. Hence, the simulation analysis is proceed using 2D instead 3D that more complex but more real. Since it is 2D and mesh sensitivity is not done, the analysis may be a bit slight different than the actual data. Other than that is material of the domain during the simulation. The real combustion engine material is unknown and set as aluminum.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

This chapter presents the result and discussion on CFD analysis. The results are cylinder pressure, turbulence kinetic energy, mass fraction burned, computational time, flame propagation, significant of the result, and summary.

4.2 CYLINDER COMBUSTION PRESSURE

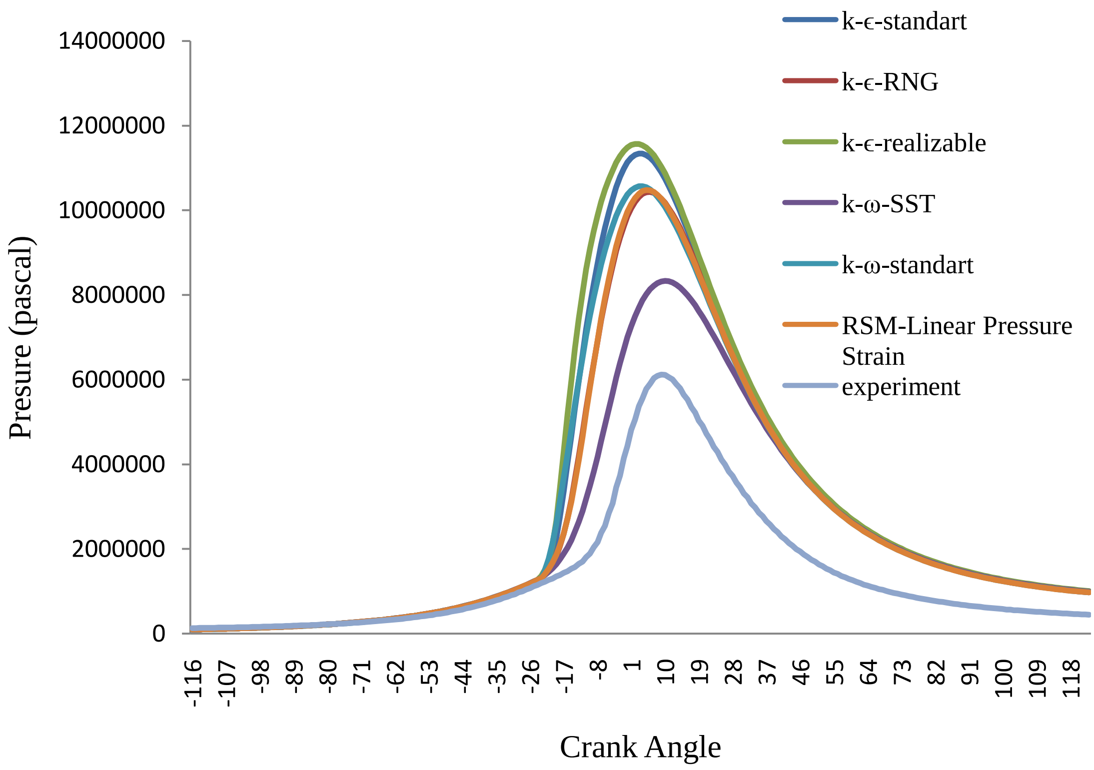

Figure 4.1: Comparison of measured and simulated cylinder pressure.

From the simulation of compression and combustion during both valves closed, most of the turbulence models give identical graph pattern but the discrepancy between instantaneous values at TDC is too big. The k-ϵ-realizable gives the highest peak pressure at 11489265 Pa followed by k-ϵ-standard at 11137987 Pa where those two models are almost likely identical. The other three models, k-ω-standard, RSM–LPS, and k-ϵ-RNG are also shows the identical graph pattern and gives the peak value of 10371732 Pa, 10064353.7 Pa, and 10011213.5 Pa respectively. Even so k-ω-SST is predicted the best results, but the peak pressure value is 8305711.7 Pa still higher than experiment where the peak pressure 6127386.112 Pa. The simulation obeys the actual results but just before the spark ignition start. During TDC, all models predicted higher peak pressure value. In the end of the combustion process, all turbulence models have resulted with almost identically decreasing pressure value due to expanding volume in combustion cylinder.

Table 4.1: Comparison of peak pressure value in simulation.

Turbulence Models	Peak Pressure (Pa)
k-ϵ-standard	11137987
k-ϵ-RNG	10011213.5
k-ϵ-realizable	11489265
k-ω-SST	8305711.7
k-ω- standard	10371732
RSM–Linear Pressure Strain	10064353.7

Table 4.2: Comparison of measured and simulated pressure value at TDC.

Turbulence Models	Pressure different (%)
k-ϵ-standard	151.5703
k-ϵ-RNG	122.7602
k-ϵ-realizable	159.5044
k-ω-SST	58.2358
k-ω- standard	134.2631
RSM–Linear Pressure Strain	125.0617

As the piston continue proceeding to power stroke, the peak value at TDC of each models still resulted with obvious pressure different presented as Table 4.2. This is because the model is only considered an ideal process where several parameters are not specifically followed the actual condition. In addition, the boundary condition for the models did not consider the fluctuating wall's heat flux as well as wall heat transfer process. However, the best prediction peak pressure value is k-ω-SST which gives the lowest percentage at 58.2358% compare to others turbulence models.

4.3 TURBULENCE KINETIC ENERGY, (TKE)

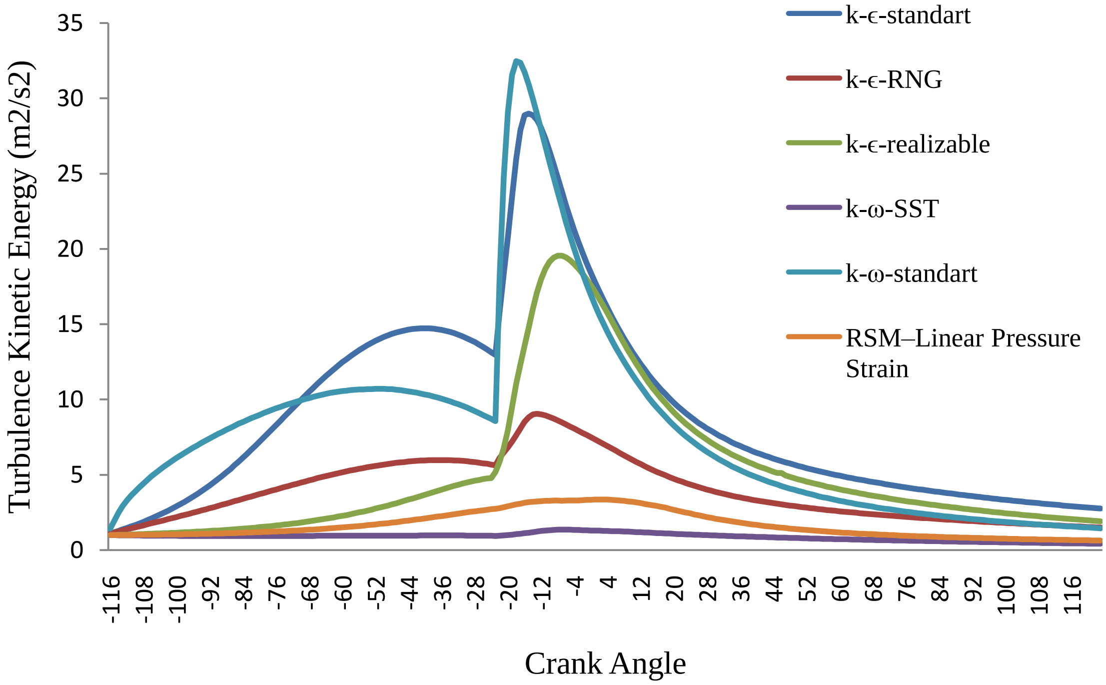

Figure 4.2: Comparison of simulated pattern of turbulence kinetic energy

Based on Figure 4.2, the graph pattern of turbulence kinetic energy (TKE) for all turbulence models are different compared to previous study. In the actual process, the value of TKE is not essentially zero especially during the start of compression. However, in this study, the value of zero is assumed to be valid for the compression and power stroke only. As the Figure 4.2 presented, k-ω-SST gives the lowest TKE while the highest TKE owned by k-ω-standard. k-ϵ-standard and k-ϵ-realizable also have high TKE content. The highest value of TKE content for each models presented in Table 4.3.

Table 4.3: Highest value turbulence kinetic energy in simulation.

Turbulence Models	Turbulence Kinetic Energy (m^2/s^2)
k-ϵ -standard	28.997946
k-ϵ-RNG	9.0630773
k-ϵ-realizable	19.551476
k-ω-SST	1.349863
k-ω- standard	32.48697
RSM–Linear Pressure Strain	3.2905515

Table 4.4: Simulated results of turbulence kinetic energy at TDC.

Turbulence Models	Turbulence Kinetic Energy (m^2/s^2)
k-ϵ -standard	18.427263
k-ϵ-RNG	7.5055436
k-ϵ-realizable	17.587727
k-ω-SST	1.3015189
k-ω- standard	16.954707
RSM–Linear Pressure Strain	3.348337

From Table 4.4, the TKE is evaluated at TDC. This is important because for usual TKE trend, the lowest TKE is at TDC. As the piston pass 23^0 BTDC, all turbulence models fluctuate because of combustion process at high temperature and pressure. Turbulence models which are k-ω-standard, k-ω-standard, and k-ϵ-realizable gives high TKE content after spark ignition. k-ϵ-RNG just only fluctuates just a bit even after the combustion process. However, TKE content for RSM-LPS and k-ω-SST does not seem so high compared to others turbulence models even combustion processes take places.

4.4 TURBULENCE DISSIPATION RATE AND MASS FRACTION BURNED

4.4.1 Turbulence Dissipation Rate, (TDR)

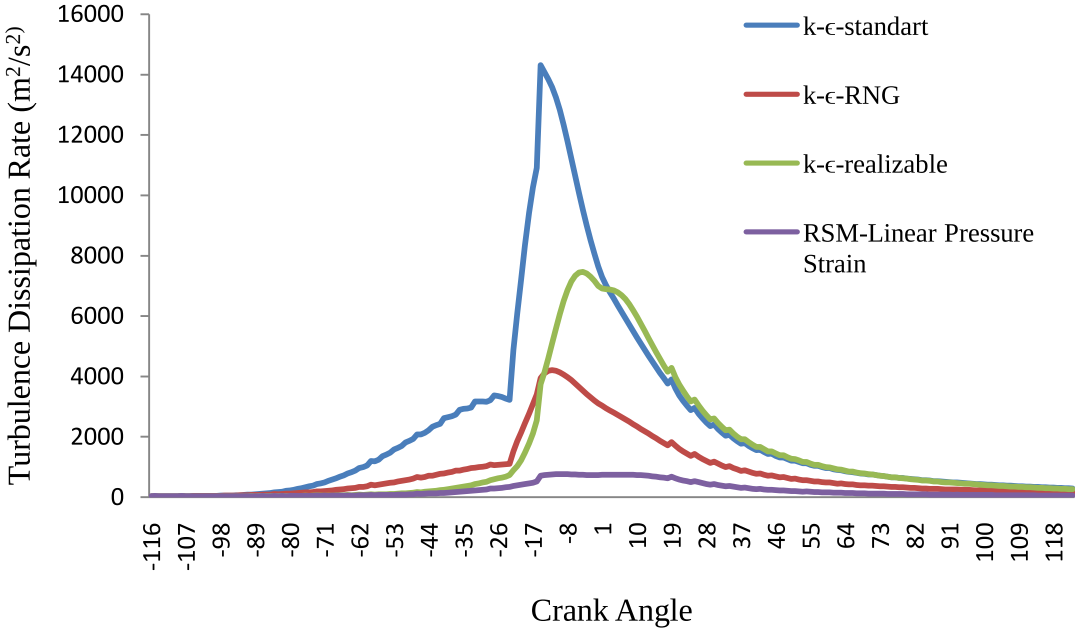

Figure 4.3: Comparison of simulated pattern of turbulence dissipation rate

From Figure 4.3, the turbulence dissipation rate (TDR) is presented for four turbulence models which are k-ϵ-standard, k-ϵ-RNG, k-ϵ-realizable, and RSM-LPS. The result of TDR also show some positive response from k-e-standard model since the peak value is far compared to other models. However, k-ϵ-realizable also has high TDR but the trend is a bit different from k-ϵ-standard and k-ϵ-RNG. Lastly, RSM-LPS show some negative progress in term of TDR. The peak value of TDR is presented in Table 4.5.

Table 4.5: Highest value of turbulence dissipation rate in simulation

Turbulence Models	Turbulence Kinetic Energy (m^2/s^2)
k-ϵ-standard	14304.639
k-ϵ-RNG	4187.0447
k-ϵ-realizable	7465.9188
RSM–Linear Pressure Strain	765.43519

Table 4.6: Simulated result of turbulence dissipation rate at TDC

Turbulence Models	Turbulence Kinetic Energy (m^2/s^2)
k-ϵ-standard	7645.1958
k-ϵ-RNG	3112.6323
k-ϵ-realizable	7003.3807
RSM–Linear Pressure Strain	738.83638

Once again, TDR is compared at TDC where the value of TDR is standardized. As the Table 4.6 showed, the value of TDR for k-ϵ-standard and k-ϵ-realizable is higher than 7000 m^2/s^2 compared to k-ϵ-RNG model which gives about half the value of TDR. RSM-LPS gives TDR value is the lowest among these four models and the graph pattern is not has similarities.

4.4.2 MASS FRACTION BURNED

Table 4.7: Mass fraction burned comparison at TDC

Turbulence Models	Mass Progress Variable
k-ϵ -standard	0.96249192
k-ϵ-RNG	0.91364009
k-ϵ-realizable	0.97545161
k-ω-SST	0.76062283
k-ω- standard	0.94471469
RSM–Linear Pressure Strain	0.91634832

From Table 4.7, the mass fraction burned for all six models is compared to each other to ensure whether which models gives highest mass fraction burned. As the Table 4.7 presented, k-ϵ-realizable gives the highest mass fraction burned followed by k-ϵ-standard, k-ω-standard, RSM-linear pressure strain, and k-ϵ-RNG where the percentages is above 90%. However, k-ω-SST is only gives about 76.06% mass fraction burned and make k-ω-SST is the lowest among these six models.

4.5 COMPUTATIONAL TIME

Table 4.8: Computational time comparison

Turbulence Models	Computing Time (minutes)
k-ϵ -standard	2206
k-ϵ-RNG	1738
k-ϵ-realizable	1723
k-ω-SST	1573
k-ω- standard	1905
RSM–Linear Pressure Strain	1645

Based on Table 4.8, the computational times for each six turbulence models are compared. Generally all six models are taking time to compute more than 1500 minutes. The longest computing time is k-ϵ-standard where the oldest turbulence model is does not have term to calculate the wall, hence the computing is longest. For overall result shows that k-ω-SST is the fastest computing time among six models. The other two k-ϵ-RNG and k-ϵ-realizable shows some slight different in computing time where the different is 15 minutes. In the other hand, k-ω-standard computing time is quite longer that k-ϵ-RNG and k-ϵ-realizable at 1905 minutes. Lastly, RSM-LPS is likely following the k-ϵ-RNG and k-ϵ-realizable computing time at 1645 minutes.

4.6 FLAME PROPAGATION (SPECIES) DURING COMBUSTION PROCESS

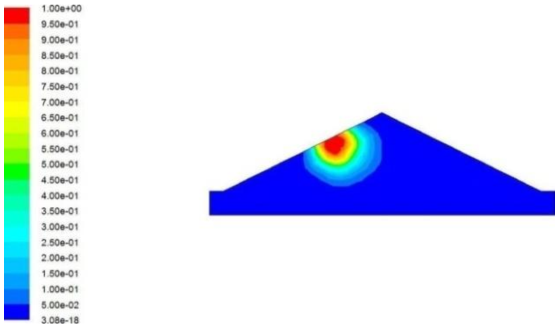

Figure 4.4 (a): CA = -23^0 BTDC

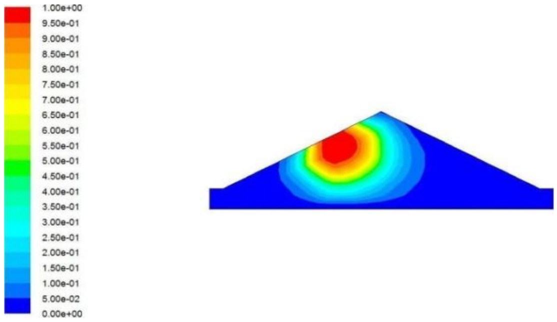

Figure 4.4 (b): CA = -21^0 BTDC

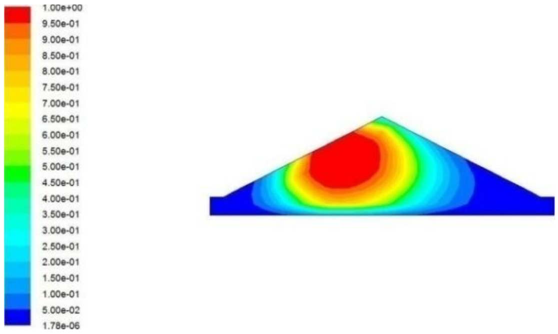

Figure 4.4 (c): CA = -19^0 BTDC

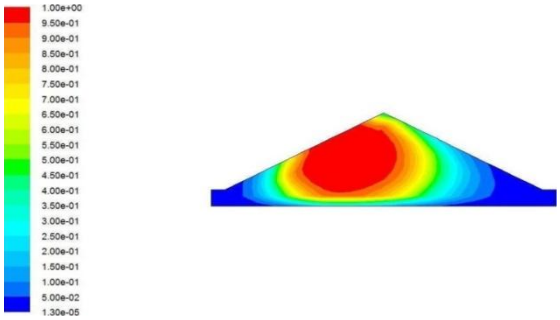

Figure 4.4 (d): CA = -17^0 BTDC

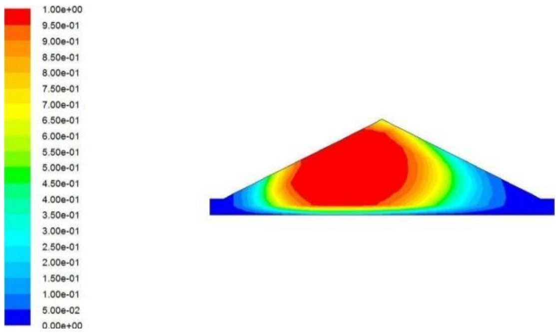

Figure 4.4 (e): CA = -15^0 BTDC

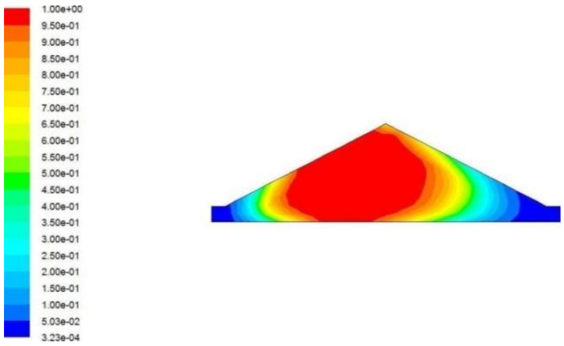

Figure 4.4 (f): CA = -13^0 BTDC

From Figure 4.4, the flame propagation (species) during the combustion process is showed. The flame propagation starts at spark ignite at CA = -23^0 BTDC as the Figure 4.4 (a). As the combustion continued, the flame propagates from the spark location to other sides of combustion chamber uniformly. The flame start to propagate toward to both sides of combustion chamber and continued the burning until the mixture completely burned.

4.7 JUSTIFICATION OF THE RESULT

4.7.1 Input Data Properties

Since this study is only focusing on CFD approach, lots of experimental inputs from previous study are required. The data attained from previous study is not specifically and technically at the exact condition and crank angle as in current simulation. For example, the actual TKE and TDR are not known since the measurement at the time intake valves closed is not carry on. So the exact value for mixture in term of TKE and TDR is unknown and set as zero but in actual condition, TKE and TDR definitely have the value especially right after flowing into combustion chamber by intake manifold through in-cylinder flow. Another important input is mixture properties especially where the compression process starts. Since measurement is not carried on, the previous study data is used but not satisfied the actual condition. Hence, the mixture does experience the compression process twice and contributes to significant result.

4.7.2 Heat Transfer Consideration

The other important factor is the heat transfer process. Noted this simulation is simulated in ideal condition without considering the heat transfer along the walls. However, in the actual condition the heat transfer along the walls is very important in order to avoid from the over heat and engine failure. So, the heat transfer along the walls by convection and conduction is ignored even so the wall temperature is considered. The heat is accumulated inside the combustion chamber and not transfers outside, the value of heat is very high and makes the pressure value for all turbulence models are very high compared to experiment pressure data.

4.7.3 Limitation of Processor

The limitation of processor is one of the important considerations when it comes to simulation especially when deal with very fine mesh, complex geometry, processor capability and turbulence model that being used. At first, this project actually has to be done in 3D model. Since the limitation of processor during the grid generation, the model changed into 2D model. However, the mesh size is not good enough in doing simulation. Finer mesh increasing accuracy but the computational time also increases. By having higher specification of processor may helps the simulation to achieves the accurate result to validate with the experiment data

4.8 Summary

As summary, this chapter has presented the simulation data. The main interest for this project is combustion pressure analysis using CFD prediction. Then, the other data that interest and supported the main objective are computational time, turbulence kinetic energy and mass fraction burned. In addition, figures of flame propagations also included.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

Computational fluid dynamic modeling approach has been done with using six turbulence models which is widely used in range for fluid flow problems. The mesh that has been used is 2 mm and the mesh density is 24.15 cells/cm². Result shows that the turbulence modeling approach using CFD need more detail and specific data at specific event so that the result will be more accurate for validation.

The consideration of three main parameter that has been discussed in chapter 4 where input data properties, heat transfer, and limitation of processor is crucial in order regain the accurate data at the same time the simulation does followed the actual condition in all perspective such heat transfer.

5.2 RECOMMENDATION

For further study, the consideration of input data properties, heat transfer, and limitation of processor need to fully aware so that the percentage of successfulness is higher. Since the CFD modeling approach is compatible, combustion process using CFD is suggested to be continued so that more data are attained so that can be references for another further study.

REFERENCES

Bai, C.J. and Hsiao, F.B. 2007. Introduction to CFD analysis. Fluid aerodynamics laboratory. Department of Mechanical.

Bi, X., Han, S. and Wang, J. 1994. Numerical optimization for in-cylinder processes of a spark ignition engine. The Engineering Society for Advancing Mobility Land Sea Air and Space. Baltimore, Maryland.

Colucci, P.J., Lee, D., Lim, C.K. and Goldin, G. 2002. In-cylinder engine modeling developments at FLUENT, FLUENT Lebanon Incorporated.

Davidson, L. 2003. An introduction to turbulence models. Department of Thermo and Fluid Dynamics.

Ecke, R. 2005. Turbulence problem. Los Alomos, California

Fluent Inc. 2004. *Fluent 6.1 user's guide.* New Hampshire, United States

Hovart, A and Hovart, Z. 2003. Application of CFD numerical simulation for intake port shape design of diesel engine. Department Of Physics, Szechenyi Istvan University

Guo. B., Yu, A., Wright, B. and Paul, Z. 2003. Comparison of several turbulence models applied to the simulation of gas flow in a packed bed. *Third International Conference on CFD in Minerals and Process Industries, CSIRO.* Australia.

Jain, A. 2008. Introduction to basics of grid generation. Faculty of Chemical, Indian Institute of Technologies.

Justham, T., Jarvis, S., Clarke, A., Garner, C.P., Hargrave, G.K. and Halliwell, N.A. 2005. Simultaneous study of intake and in-cylinder ic engine flow fields to provide an insight into intake induced cyclic variations. Wolfson School of Mechanical and Manufacturing Engineering, Loughborough University

Kulvir, K.D., Bailey, C.J. and Pericleous, K.A. 2004. Centre for numerical modelling and process analysis. University of Greenwich, Old Royal Naval College Greenwich, London, United Kingdom.

Mumovic, D.J., Crowther, M. and Stevanovic, Z. 2004. The effect of turbulence models on numerical prediction of air flow within street canyons, Faculty of Mathematics, University of Belgrade.

Ogor, B., Gyllenram, I., Ohlberg, W., Nilsson, E. and Ruprecht, A. H. 2006. An adaptive turbulence model for swirling flow, *Conference Turbulence and Interaction. Porquerolles*, France.

Payri, F., Benajis, J., Margor, X. and Gil, A. 2003. CFD modelling of in-cylider flow direct injection diesel engine. CMT-Motores Termicos, Elsevier

Semin., Nik Izual, N.I., Rosli A.B. and Ismail, A.R. 2008. In-cylinder flow through piston-port engines modelling using dynamic mesh. Automotive Excellent Centre, Faculty of Mechanical, University Malaysia Pahang

Sodja, J. 2007 . Turbulence in CFD, Department of Physics

Schluter, J.U., Pitsch, H. and Moin, P. 2005. Outflow condition for intergrated large eddy simulation/reynolds averaged Navier-Stokes simulation. Center of Turbulence Research, Stanford University.

Tu, J.Y., Yeoh, G.H. and Liu, C. 2008. *Computational fluid dynamics a practical approach*. 1st ed. United Kingdom: Elsevier

Uygun, M., Onbasioglu, S. and Avci, S. 2004. Turbulence modeling for computational fluid dynamics. Aeronautics and Space Technologies, ITU Makina Turkish

Venayagamoorthy, S.K., Kose, J.R., Ferzigery, J.H. and Shih, L.H. 2003. Testing of RANS turbulence models for stratified flows based DNS data, Center of Turbulence Research, Stanford University.

Vengadesan, S. and Nithiatasu, P. 2007. Hybrid LES. Department of Applied Mechanics, Indian Institute of Technology.

Ziya, B.G. 2003. Implementation and comparison of turbulence mc problem using Navier-Stokes solver. Department of Mechanica