# GRID SENSITIVITY STUDY OF IN-CYLINDER ENGINE MESHING APPROPRIATE FOR FLOW AND COMBUSTION ANALYSIS USING COMPUTATIONAL FLUID DYNAMICS (CFD)

UMAR BIN SHAFEE

Thesis submitted in fulfillment of the requirements for the
award of the degree of Bachelor of Mechanical
Engineering With Automotive

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

4 DECEMBER 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.

Signature

Name of Supervisor : MOHD FADZIL ABDUL RAHIM

Position : LECTURER

Date : 4 DECEMBER 2009

## 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 concurrently submitted for award of other degree.

Signature

Name : UMAR BIN SHAFEE

ID Number : MH06024

Date : 4 DECEMBER 2009

**Dedication to my beloved parents, siblings and friends**

## ACKNOWLEDGEMENTS

In the name of Allah, the Most Benevolent and the most Merciful. Every sincere appreciation and gratitude is only to God. Only by His Kindness and Guidance that this thesis is finally realized.I am grateful and would like to express my sincere gratitude to my supervisor Mr Mohd Fadzil Abdul Rahim for his ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a degree program is only a start of a life-long learning experience. 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, his tolerance of my naive mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proof reading and correcting my many mistakes. My sincere thanks go to all my members who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to member in my group research with the same supervisor 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 cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. Special thanks should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

## ABSTRACT

This thesis deals with grid sensitivity study where it analyze the grid size as an input parameter. The change of grid size will give effect on the accuracy of result and computing time taken for analysis. The key objective for this project is to determined the suitable or acceptable grid size by considering the accuracy of result and computing time. This thesis describe on technique to tackle the objective from the start of modelling until finish the project. The analysis is focus on spark ignition (SI) combustion of the baseline engine design, Mitsubishi Magma 4G15 1.5L. The domain of in-cylinder engine was developed from CAD software and analysis using Computational Fluid Dynamics (CFD). 2-dimensionall (2D) domain was used in order to reduce the time consume instead of 3-dimensional (3D). Numbers of grid size is generated before it is analyzed. The speed of engine is fixed at 2000 revolution per minute (rpm) and the turbulence model is k-epsilon (k-ϵ) model. The iteration number is set to 2000 iteration per time step. There are six different cases analyzed with different grid size. The grid size was set from 1mm to 4mm. The analysis is to predict the behavior of in-cylinder engine's combustion process. Different grid size give different result of accuracy and computing time. The accuracy test is based on cylinder pressure and the simulation data is validated with experiment data. Smaller grid size will resulted the better accuracy while consume more computing time. Case 3 of grid size 2.5mm has given the best as and most acceptable result.

# ABSTRAK

Thesis ini membentangkan tentang analiysis sensitiviti bagi grid di mana analisis saiz grid sebagai parameter penting. Sebarang perubahan kepada saiz grid akan memberi efek kepada ketepatan data dan juga masa untuk melengkapkan sesuatu analisis. Objektif utama untuk projek ini adalah untuk mencari grid saiz yang paling sesuai yang mengambil kira ketepatan sesuatu data dan masa untuk analisis. Dalam thesis ini juga menerangkan cara-cara untuk mencapai objektif bermula dari membuat model sehingga projek ini selesai. Analisis memfokuskan proses pembakaran ke atas enjin palam pencucuh, Mitsubishi Magma 4G15 1.5L. Domain untuk silinder enjin telah dibuat dengan menggunakan perisisan CAD dan analisis akan menggunakan Dinamik Aliran Berkomputer. 2-dimensi (2D) domain telah digunakan kerana ia lebih menjimatkan masa berbanding 3-dimensi (3D) domain. Pelbagai saiz grid telah dibuat sebelum ia dianalisis. Kelajuan enjin telah ditetapkan kepada 2000 pusingan per minit (ppm) dan juga model gelora k-epsilon (k-ϵ). Bilangan lelaran pula ditetapkan kepada 2000 sela setiap masa. Enam kes telah dianalisa untuk projek ini dengan menggunakan saiz grid yang berbeza. Saiz yang telah ditetapkan adalah dari 1mm kepada 4mm. Analisis ini adalah untuk meneka sikap untuk dalam silinder enjin semasa proses pembakaran. Saiz grid yang berlainan akan memberikan keputusan yang berlainan untuk ketepatan dan masa. Ketepatan sesuatu ujian data adalah berdasarkan tekanan silinder dan data akan di validasi dengan data eksperimen. Semakin kecil saiz grid akan memberi keputusan bacaan data yang semakin tepat tetapi mengambil masa yang panjang. Kes 3 iaitu saiz grid 2.5mm telah dipilih sebagai saiz yang paling sesuai dan diterima.

# TABLE OF CONTENT

## LIST OF TABLES

## LIST OF FIGURES

# LIST OF SYMBOLS

| | |
|---|---|
| $C_{1\varepsilon}$ | Model constant for of ∈ equation |
| $C_{2\varepsilon}$ | Model constant for of ∈ equation |
| $C_{3\varepsilon}$ | Model constant for of ∈ equation |
| $g$ | Gravitational acceleration |
| $k$ | Turbulent kinetic energy∈∈ |
| $P_r$ | Prandtl number |
| $P_k$ | Production of $k$ |
| $P_b$ | Effect of buoyancy |
| $S$ | Modulus of the mean rate-of-strain tensor |
| $T$ | Temperature |
| $\tilde{u}_j$ | Velocity component in x, y and z direction |
| $u'$ | Root-mean square velocity |
| $Y_M$ | Mass fraction |
| $\epsilon$ | Turbulent dissipation rate |
| $\beta$ | Coefficient of thermal expansion |
| $\sigma_\epsilon$ | Standard deviation of distribution of ∈ |
| $\rho$ | Density |
| $\mu_T$ | Turbulent molecular viscosity |

# LIST OF ABBREVIATIONS

| | |
|---|---|
| ABDC | after top dead center |
| ATDC | after bottom dead center |
| BBDC | before bottom dead center |
| BDC | bottom dead center |
| BTDC | before top dead center |
| CA | Crank angle |
| CAD | Computer aided design |
| CFD | Computational fluid dynamics |
| k-ϵ | k-epsilon |
| OHV | Over head valve |
| PISO | Pressure-Implicit Splitting of Operators |
| rpm | revolution per minutes |
| SA | Sensitivity Analysis |
| TDC | top dead center |
| SI | Spark ignition |
| 2D | 2-dimensional |
| 3D | 3-dimensional |

# CHAPTER 1

# INTRODUCTION

## 1.1 BACKGROUND

Spark ignition (SI) engine is the most common engine that was used in passenger vehicle all over the world. In the past few years SI engines have been greatly improved in terms of efficiency and pollutant emissions. This trend must continue in the near future. However, the combustion process also depends highly on an efficient fuel–air mixture. The core constraint that controls the SI engine performance is the combustion process inside the engine cylinder. The airflow structures generated inside the cylinder are known to significantly influence the air-fuel mixing and mixture transport during induction, and the turbulent flame speed during combustion. Accurate measurement of these structures is necessary to optimize the engine in terms of power output, fuel economy and emissions control. The method for combustion analysis such as laser diagnostic tools with optical window are relatively expensive and need huge financial support. Another method than can be used is Computational Fluid Dynamics (CFD). Flow and combustion analysis is very complex. As stated before, it required huge investment to do the experimental analysis. Using CFD this study is covered about the grid analysis of flow of in-cylinder SI engine. In order to obtain accuracy, finer grid are required. However, finer grid will result with increase in time computing. Thus, optimization of grid size is required to provide an acceptable solution accuracy.

## 1.2 PROBLEM STATEMENT

The purpose of this project is to analyze the effect of using different grid size in CFD analysis. Besides having accurate result, the computing time also play a important role in the choice of the grid size. Since using a long time, it is equal to rising the cost of the study in industrial application. Hence, a good balance between the accuracy and the computing time by choosing the appropriate grid size is the best options to optimizes the analysis.

## 1.3 OBJECTIVE OF THE SYUDY

1. To determine an acceptable grid size which promoting accuracy
2. To minimize the computing time taken to do the analysis

## 1.4 SCOPE OF THE STUDY

Firstly, in order to model the problem, the engine dimensioned based on four stroke four cylinder SI engine. The type and specification of this engine is presented in Chapter 3. The dimension of the engine, modeled using CAD software, is based on the specification given. Then it is meshed using Gambit application. The model then translated for used in Fluent 3D solver. In order to reduce the time consuming, computational domain developed in 2D. The mesh is set to multiple sizes so that a comparison of accuracy among them could be observed. The last part of the study is finalization of grid size.

## 1.5 Flow Chart of The Study

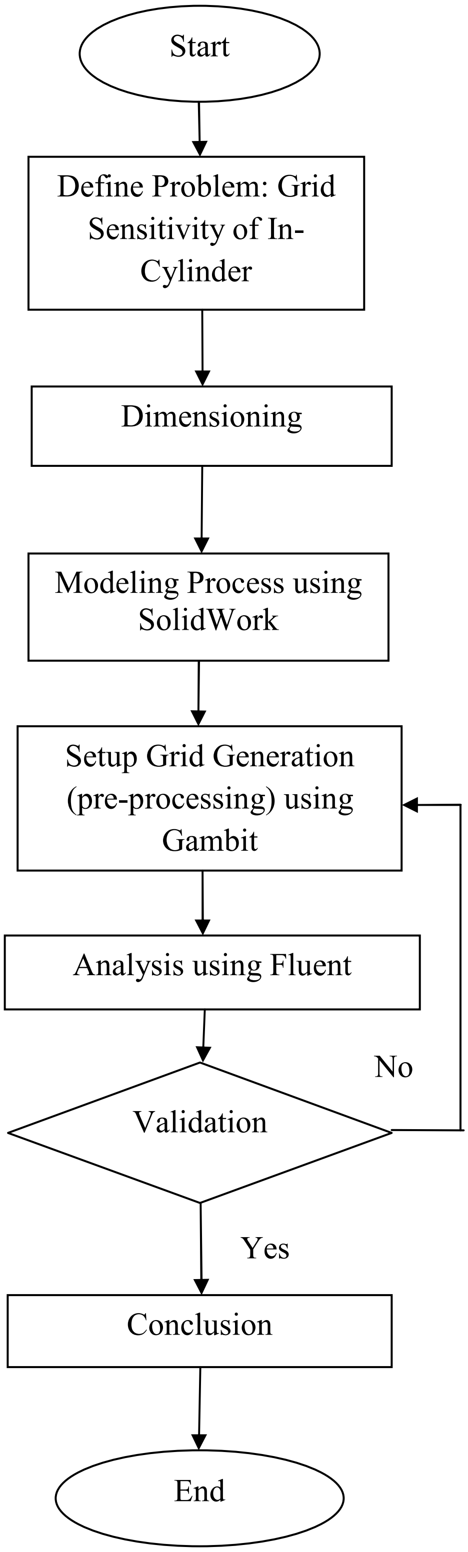



**Figure 1.1**: Flow chart of the overall study

## 1.6 Organization of Thesis

This thesis has been written with necessary details to the understanding of this project. Excluding the introduction, which is the first chapter of this thesis, the main point of the project is devided into four chapter. Chapter 2 consist of a literature review of several studies on grid generation and grid sensitivity analysis. Chapter 3 elaborate the numerical setup and methodology  used to achieve the objective of the study. This include the specification of the engine based on manufacturer data sheet, and the choice of grid parameter used for CFD analysis. The result of simulations are reported in Chapter 4. Finally, Chapter 5 present the conclusion of overall study and the future work recommendation.

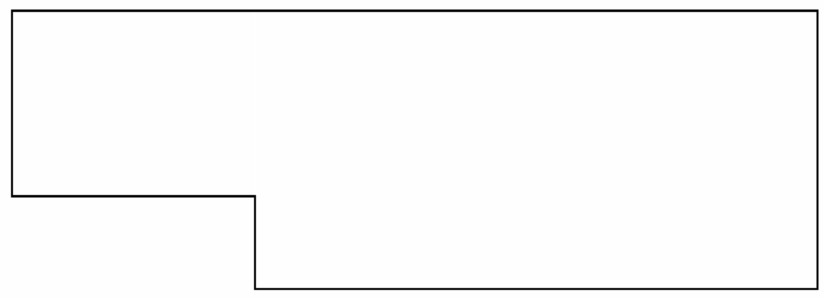
# CHAPTER 2

# LITERATURE REVIEW

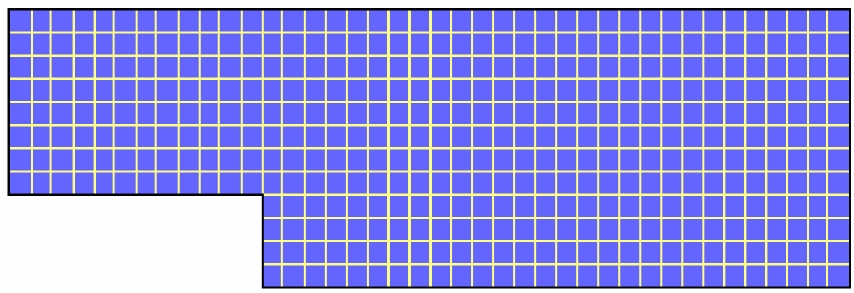
## 2.1 INTRODUCTION TO GRID GENERATION

Numerical grid generation arise from the need to compute solutions to the partial differential equation of fluid dynamics on physical regions with complex geometry. By transforming a physical region to a simpler region, one removes the complication of the shape of physical region from the problems (Patrick, 1994). An important element of the numerical solution of partial differential equation by finite-element or finite difference methods on general region is grids which represent the pysical domain in a discrete form (Liseikin, 1999). Except for very simple cases, for fluid flow and heat transfer, the partial differential equations that used for it are not usually done by analytical solutions. Therefore, in order to analyze fluid flows for complex case, numerical solution is used. The flow domains are split into smaller subdomains such as hexahedral and tatrahedral in 3D, and quadrilaterals and triangles in 2D and discretized governing equations are solved inside each of these portions of the domain. There are three method to solve the problem. The method are finite volumes, finite elements, or finite differences. Typically, one of three methods is used to solve the approximate version of the system of equations.

Each of these portions of the domain also can be called as elements or cells, and the combination of all elements is known as grid or mesh. For 2D analysis, a domain split into elements resembles a wire mesh. An example of a 2D analysis domain or flow over a backward facing step and its mesh are shown in Figure 2.1 .



a) Before grid generation



b) After grid generation

**Figure 2.1**: Example of 2D domain a) before grid generation and b) after grid generation

The process of obtaining an appropriate grid is called grid generation. This process was consider as the most important and most time consuming part of CFD simulation. The quality of the grid plays a direct role on the quality of the analysis

## 2.2 THE IMPORTANT OF GRID GENERATION

From section before, it is determined that grid generation play the important role in CFD. The grid generation is important because it is used to transform a physical region to simpler region and removes the complication shape of the physical region from a problem. In order to do the analysis, the physical region must be separated into small element called grid so that the results can be determined. The CFD simulation is about numerical calculation one by one of the element. It is important for people that will use CFD analysis to know and understand all of the various grid generation methods. Only by knowing all the methods they can select the right tool to solve the problem at hand.

## 2.3 TYPES OF GRID

### 2.3.1 Structured Grid

Structured grid methods take their name from the fact that the grid is present in a regular repeating pattern called a block. The arrangement of grid is alligned properly. These types of grids can utilize quadrilateral elements in 2D and hexahedral elements in 3D in a computationally rectangular array. The element topology for structural grid is fixed, but the grid can be shaped to be body fitted through stretching and twisting of the block. Really good structured grid generators utilize sophisticated elliptic equations to automatically optimize the shape of the mesh for uniformity. It is known that hexahedral meshes provide more accurate and better representation of the surface gradients, but on the other hand, requires more expertise and effort (Luis et al, 2004). Structured grid lack of required flexibility and robustness for handling domin with complicated boundaries, or the grid cells may become too skewed and twisted, thus prohibiting an efficient nemerical solution (Liseikin, 1999). Figure 2.2 shows the 3D domain structured grid where the grid is generated in multiple block. The upper part of the domain is generated  by stretching the block grid. Figure 2.3 shows an example of 2D overset mesh. The points from the red mesh will be cut  away to match the outer boundary of blue mesh

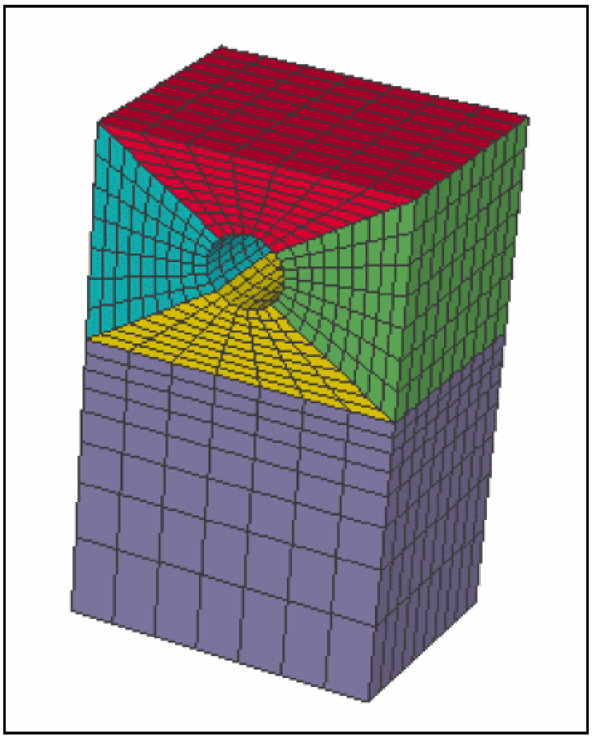


**Figure 2.2**: An example of structured grid in  multiblock mesh

Source: Wyman (2001)

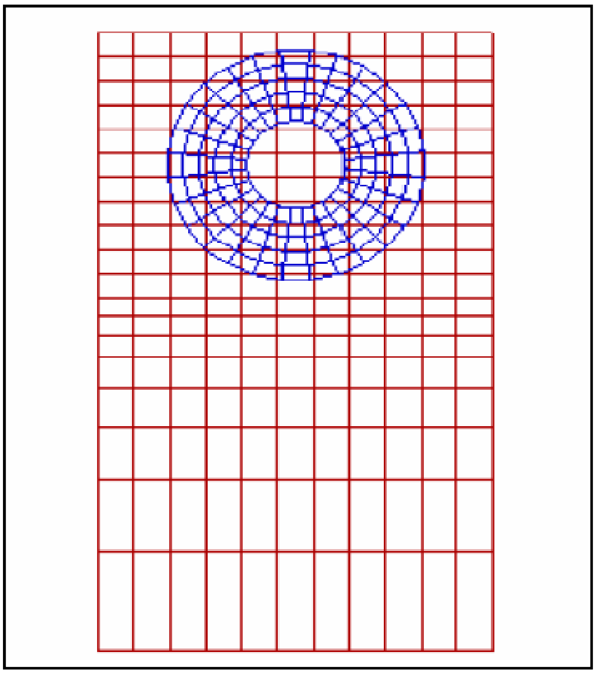


**Figure 2.3**: An example of a 2D overset structured mesh.

Source: Wyman (2001)

### 2.3.2 Unstructured Grid

Unstructured grid methods utilize an arbitrary collection of elements to fill the domain. Because the arrangement of elements has no specific pattern, the mesh is called unstructured. These types of grids typically utilize triangles in 2D and tetrahedral in 3D. The triangles cells are the simplest 2D element and the tetrahedral cells are the simplest 3D element (Liseikin, 1999). While there are some codes which can generate unstructured quadrilateral elements in 2D, there are currently no production codes which can generate unstructured hexahedral elements in 3D.

Unstructured grid has irregularly shaped domains with greater efficiency and less effort. Besides that, it has similarity with structured grids that is the elements can be stretched and twisted to fit the domain. These methods have the ability to be automated to a large degree. Given a good CAD model, a good mesher can automatically place triangles on the surfaces and tetrahedral in the volume with very little input from the user. The automatic meshing algorithm typically involves meshing the boundary and then either adding elements touching the boundary or adding points in the interior and reconnecting the elements. It is called delaunay trianglulation. Figure 2.4 shows the example of 3D unstructured grid. It is consist of triangle and tetrahedral elements.

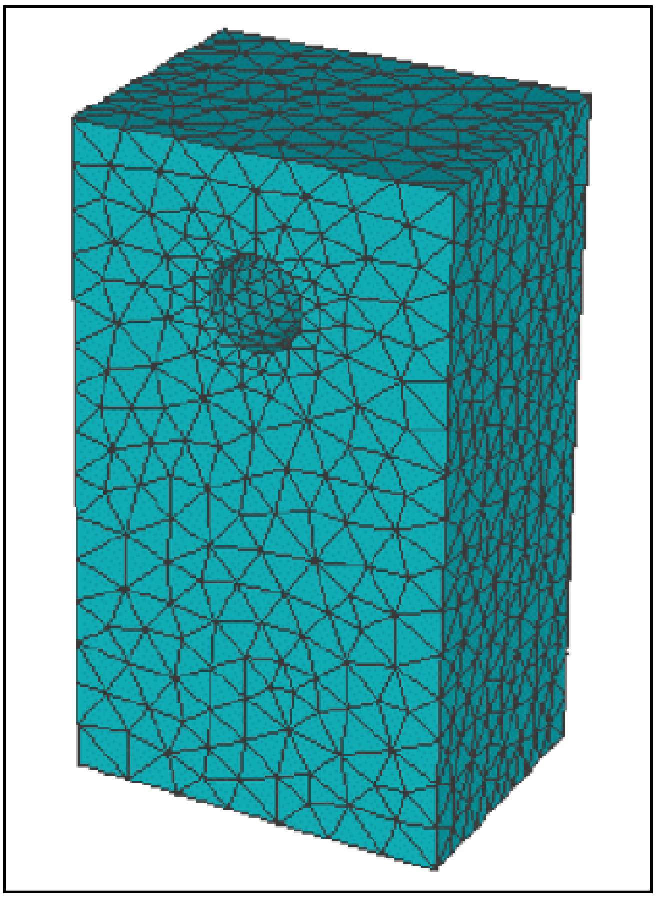


**Figure 2.4**: An example of unstructuresd grid
Source: Wyman (2001)

### 2.3.3 Hybrid Grid

Hybrid numerical grid are mesh which are obtained by combining both structured and unstructured grids (Liseikin, 1999). The design of hybrid grid methods is used to take advantage of the positive aspects of both structured and unstructured grids. Hybrid grids can utilize some form of structured grid in local regions while using unstructured grid in the bulk of the domain. Hybrid grids can contain hexahedral, tetrahedral, prismatic, and pyramid elements in 3D and triangles and quadrilaterals in 2D. According to their strengths and weaknesses, various element are used in order to get good grid generation. Elements such as hexahedral are excellent near solid boundaries, where flow field gradients are high and also give the user a high degree of control. But, it consume more time to generate it. Prismatic elements are useful for