APPLICATIONS OF OPEN-SOURCE CAE TOOLS IN MECHANICAL DESIGN

NOOR FARAHANA BINTI ABDUL RAHMAN
B041310127
BMCD
n.farahanaabdulrahman@gmail.com

Final Report
Projek Sarjana Muda II

Supervisor: DR. SYAMSUL ANUAR BIN SHAMSUDIN
Second Examiner: EN. NAZIM BIN ABDUL RAHMAN

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

MAY 2017

APPLICATIONS OF OPEN-SOURCE CAE TOOLS IN MECHANICAL DESIGN

NOOR FARAHANA BINTI ABDUL RAHMAN

This report is submitted in fulfillment of the requirement for the degree of Bachelor of Mechanical Engineering (Design and Innovation)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

MAY 2017

APPROVAL

“I hereby declare that i have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of mechanical Engineering (Design & Innovation)”

Signature :

Name : DR. SHAMSUL ANUAR BIN SHAMSUDIN

Date :

DECLARATION

I declare that this report entitle “Application of open-source CAE tools in mechanical design” is the result of my own research except as cited in the references.

Signature	:	
Name	:	NOOR FARAHANA BINTI ABDUL RAHMAN
Date	:	

DEDICATION

To my beloved my father, Abdul Rahman Bin Ghani, my mother, Siti Alina Binti Ahmad, my younger brother, Muhammad Hafizat Bin Abdul Rahman, supervisor and friends

ABSTRACT

The purpose of this project is to study the applications of open-source Computational-Aided Engineering (CAE) tools in mechanical design. The Computational-Aided Engineering type that has been chosen is Computational Fluid Dynamics (CFD); and the model design that fulfill the criteria to be simulated in this project is the part of aerodynamics model. This aerodynamics design is synonym in industrial designing because of the feedback in terms of its accessibility, convenience and safety. In the creation of model aerodynamic concept, there are a lot of things that need to be identified and calculated as it is hazardous and need to supervise from a professional. As this project research is to study on how work the simulation on certain chosen software that has been applicate on the selected model design. There are two type of software has been selected that are open-source software and commercial-package software. The preferred software is intentionally has been elected in order to see the different in the type of used, the meshing and airflow direction. The software that has been picked for open-source software are OpenFOAM and Gmsh, while for commercial-package are ANSYS-Fluent and SolidWorks. This all software is being using to analyze the CFD simulations and understand the procedure in using it.

ABSTRAK

Tujuan bagi menjalankan projek ini adalah untuk mengkaji aplikasi bagi sumber terbuka (*open-source*) dengan menggunakan alat pengiraan kejuruteraan dibantu (*Computational Aided Engieering*) di dalam reka bentuk mekanikal. Jenis pengiraan kejuruteraan dibantu (*Computational Aided Engineering*) yang telah di pilih ialah pengiraan dinamik bendalir (*Computational Fluid Dynamics*); dan reka bentuk model yang memenuhi kriteria untuk simulasi dalam projek ini adalah sebahagian daripada *aerodinamik* model. Reka bentuk *aerodinamik* ini adalah sinonim dalam prindustrian pemprosesan reka bentuk berdasarkan maklum balas dari segi akses, kemudahan dan keselamatan. Dalam penciptaan konsep model aerodinamik, terdapat banyak perkara yang perlu dikenal pasti dan dikira kerana ia adalah berbahaya dan perlu penyelia daripada pakar. Penyelidikan projek ini ialah untuk mengkaji betapa proaktifnya simulasi pada perisian yang dipilih untuk di gunakan sebagai bahan eksperimen kepada reka bentuk model yang dipilih. Terdapat dua jenis perisian yang telah dipilih iaitu perisian sumber terbuka (*open-source*) dan juga perisian sumber berbayar (*commercial-package*). Perisian yang dipilih adalah dengan syarat yang telah dipenuhi di dalam syarat pemilihan adalah untuk melihat perbezaan dalam bentuk cara penggunaan, gambaran bersirat dan arah aliran udara yang beraksi di reka bentuk model yang dipilih. Perisian yang dipilih untuk sumber terbuka (*open-source*) ialah OpenFOAM dan Gmsh, manakal untuk berbayar (*commercial-package*) ialah ANSYS-Fluent dan SolidWorks. Analisis pengiraan dinamik bendalir (*Computational Fluid Dynamics*) dan cara penggunaan telah dianalisis menggunakan keempat-empat perisian tersebut.

ACKNOWLEDGEMENT

At first, “Alhamdulillah” and without Allah’s will my *Projek Sarjana Muda* (PSM) report would not be completed. Then, I would like to express my special thanks to my parents Mr. Abdul Rahman Bin Ghani and Mrs. Siti Alina Binti Ahmad for their assistance without limits and all the motivations that they have gave throughout my difficulties along completing this project and report. Next, I would like to thank to my supervisor, Dr. Shamsul Anuar Bin Shamsudin that always supporting and guiding me in guidance, stimulating suggestions and constant encouragement that help me to finish my project especially in writing report. I also appreciate the guidance given by En Nazim Bin Abdul Rahman especially during this project presentation of VIVA and report that improved my presentation skills with thier comment and tips. Moreover, I would like to thank my laboratory, who gave me the permission to use all the machienery and all the facilities in order to finish my project report. Last but not least, I wish to avail myself of this opportunity to express a sense of gratitude and love to my family and friends for their help, moral support, and encouragement in everything.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

ANSYS	Analysis of System
CAD	Computer-Aided Design
CAE	Computer-Aided Engineering
CFD	Computational Fluids Dynamics
CLI	Command Line Interface
FEA	Finite Element Analysis
Hex	hexahedra
IGS/IGES	Initial Graphics Exchange Specifications
NASDAQ	National Association of Securities Dealers Automated Quotations
OBJ	Object
OpenFOAM	open source for Field Operation and Manipulation
Split-hex	split-hexahedra
STL	Stereo lithography
STP	Standard for the Exchange Product

LIST OF SYMBOL

m/s	Meter per second
kPa	Kilo Pascal
K	Kelvin

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

This chapter will label about the idea in this investigation. This idea was introducing by targeting to solve the problem in how to decrease the cost spending in engineering software. In this section also will label about the background of this study, the problem statement encouraged to this enquiry, objective and scope take account in this study.

1.2 BACKGROUND

This research is focusing more in Computational Fluid Dynamics (CFD) analysis. Nowadays, engineering field especially in design department does pay attention and rely a lot on simulation analysis before certain products were produce. Simulation analysis is playing an important role in production costing because it will lessen the cost in prototype production. Besides, using simulation analysis is easier in checking the thermal, pressure, and fatigue analysis by using just one software and the design can be optimizing after the fatigue part were analyze.

In order to making this research, little simulation analysis software’s function is being identified and recognize. There are two type of license on software that is open source and commercial packages. Commercial packages are a software that provide pay license while, the other software that is open source does not need paid license because the license can be downloaded directly from the internet. Of course, the paid one will give a lot of function in simulation because it is marketing, everything does not come free and every upgraded version of software will come with expensive prices.

In this research, the simulation analysis was focus just in Computational Fluid Dynamics (CFD). The CFD simulation analyses were focused on both open source and commercial packages license. Every permit will be studied about two examples of software; for open source software: ANSYS-Fluent and SOLIDWORK while for commercial packages: OpenFOAM and OpenFVM.

Most of the commercial package usually can be bought or downloaded directly from internet but needed to pay; it is also coming in the compiled ready-to-run version. The meaning of compiled is the actual program code that the developer created, known as the source code, has run through a special program called a compiler that translates the source code into a form that the computer can understand. It is difficult to modify the compiled version of most applications and nearly impossible to see exactly how the developer created the parts of the coding inside the program. (Initiative, 2013)

The source code is included with the compiled version, so the user can modify and edit the coding to be more useful for long time used. The software developer who developed the open source concept with license that not interfere with the operation of other software are allowing the user to modify the source code are causing the application will be more useful and error-free over the long term. For example, as Red Hat has done with Linux. (Lakhani & Von Hippel, 2003)

1.3 PROBLEM STATEMENT

Currently, the cost of engineering software like ANSYS-Fluent, Computational Aided Design (CAD), and SOLID-WORK have risen (Menter, 2012). In order to low the cost of production, the simulation analysis be present on the rare design. The problem is in a way to complete the analysis, the suitable software needed. Towards fulfill the analysis requirement by technology; the software was come with new version. The difficult quantified is new version comes with luxurious price.

1.4 OBJECTIVES

The objectives of this research are to:

1. Differentiate between open source and a commercial package.
2. Compare the Computational Fluid Dynamics (CFD) analysis of open source and a commercial package.

1.5 SCOPE

The scope of this study is primarily focuses on Computational Aided Engineering (CAE) especially on Computational Fluid Dynamics (CFD) analysis. Principle open source and commercial package software was explored in this research. Besides, the procedures and different between commercial package software and open source software are studied. The simulation is focusing in Computational Fluid Dynamics (CFD) analysis and the other simulation is not included in this research. The coding of open source software is not being studied and does not been changed along this research. This research just focusses on testing the software and get the illustration of the Computational Fluid Dynamics (CFD) simulation.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In this section, the parts of the car that have been undergoes testimony in flow direction are chosen. Further, the open source and commercial packages software capabilities are studied in order to analyze the capabilities in identified the flow direction on the product with approximate reading. Finally, the description of the software is presented.

2.2 COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

Computational Fluid Dynamics (CFD) is one of the fluid mechanics that uses of both numerical analysis and algorithms as converter to real flows simulation to be analyzed. In engineering field especially in aircraft and structure engineering, Computational Fluid Dynamics analysis is widely in used. Based on (Dynamics et al., n.d.), the experiment methods is important in confirming the validity and limits of approximations to the governing equations. Basically, the flow involving governing equations are extremely complicated, for example analytic solutions is possible to be solve for most practical applications. CFD introducing computational techniques as solutions, which replace partial differential equations with systems of algebraic equations which is much easier to solve using computer. In late 1950's, the improvement in computing power solutions become steady and its use is becoming increasingly rampant. (Collins, n.d.)

Next, CFD provides both qualitative and sometimes even quantitative prediction of fluid flows which means of mathematical modeling (partial differential equations, numerical methods (discretization and solution techniques), and software tools (solver, pre- and post-processing utilities). CFD also enables engineering and scientists to perform virtual flow laboratory in computational simulations. (Collins, n.d.)

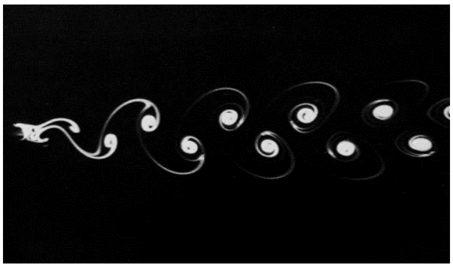

Figure 2-1 Real experiment virtual

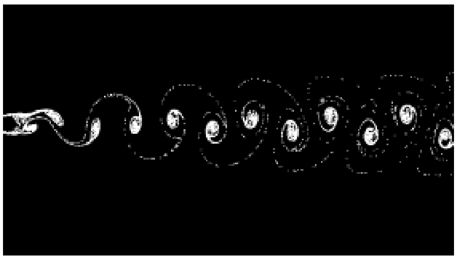

Figure 2-2 CFD simulation virtual

Most of CFD problems are solved using the Navier-Stroke equations that is define single phase which is gas or liquid as fluid flows. These equations can be simplified and lead as new equations as the term in the main equations being removed. When the term describing viscous actions to yield is removed from the main equations, the new equation is defined as Euler equations. Next, by simplify the equations with removing the term related to vortices yield make the equations be the full potential equations. (Kuzmin, n.d.)

2.2.1 PHASE IN COMPUTATIONAL FLUIDS DYNAMICS (CFD) ANALYSIS

In engineering fields, engineers commonly predict the behavior of the systems to facilitate understanding the relationship between the system variables. This method is important to design a better system for optimizing the operations. Usually, engineers perform the experiments directly or construct certain mathematical models that represent the system which contagion concerning to understand the system directly.

Next, another steps to include during the prediction method is constructed a mathematical model based on the understanding of the basic physical phenomena of the system. Understanding of system behavior can assist engineers in given certain conditions to find a mathematical solution by resulting system of equations. This step can be labeled as analytical approach.(Lomax, Pulliam, & Zingg, 1999)

Another approach can be describing is use CFD methods. A CFD method is described as the differential equations that govern the system is replaced with a set of algebraic equations at discrete points. The equations then are solved using digital computers. (Dynamics et al., n.d.)

2.2.1.1 EXPERIMENT TECHNIQUES

Most of reliable experiment data and information are getting from the measurement. In certain experiment involving full-scale equipment can give clue about hoe the equipment can perform under certain conditions. However, most of practical engineering applications does not involve full-scale equipment; full-scale test basically is either very expensive or difficult to handle or not possible at all. Full scale equipment is defined as the scale or size of the testing product are design or create exactly the same as an original product. So, an alternative solution is to perform the experiment on the small scale models. As can relate with this paper is, the original of the model is big and to create with real material and design as the original also high in cost production, so the solution is produce car body frame for same design but in small scale.

In this all advantages stated before, there is also disadvantages in using this experiment method as identify the flow directions manually. Although the model that has to be tested can be created in small size in order to save the cost; the resulting information of the small scale models needs to be extrapolated to the original models. The small scale models can also be models to be tested, but analysis of the testing is limited. Small scale models usually not simulate all the features of the original scale models; this will limit the usefulness of the test result. All of this situations, the most difficult stage in experimental method is when there are significant errors in measuring equipment. It will affect the analysis result and it will cause the major changing of the original design. From this disadvantage, it does not mean that computational models is paramount importance; it should be stress that the validation of the experimental data of numerical models is importance before it is put to good use.(Center, 2015)

2.2.1.2 ANALYTICAL METHODS

An analytical model is the consequences of mathematical models that represent the characteristics of certain system. In certain systems, it is too complex to be translated by drawing; so the function of mathematical model is translating the complex system into model that representing the physical process mainly consists of a set of differential equations. In most practical engineering applications, a lot of assumptions and simplifications are made to enable the analytical solution of differential equations created representing physical situations.

The assumptions and simplifications made in analytical solutions of the differential equations will limit the applicability of these methods. When there are too many assumptions and simplifications in system will limits the applicability of these methods to simple type of problems, although it will limit the validity of the solutions. (Center, 2015)

Despite that, analytical models played significant role in making the engineer or scientist understanding the fundamental of the rules of the system. In addition, this method played role as first stage in the validations of CFD models.

2.2.1.3 COMPUTTAIONAL FLUID DYNAMICS (CFD) TECHNIQUES

CFD technique is one method that involves the advent of digital computers. Computer such high electronic devices, that can calculate or solve high level equations in short time. Using this method, a large number of numerical methods were developed to solve flow problems. The purpose of flow simulations is to analyze the behavior of the flow in a given system for a given set of inlet and outlet conditions. This condition is known as boundary conditions.(Lomax et al., 1999)

Using this method also can identify the flow pattern and temperature distribution within the system; from the analysis, the design improvement need to be made to improve the defect of the system. The fundamental of CFD methods is to fine the values of flow quantities at a large number of points spread around the system geometry. These points are usually connected together and perform as numerical grid or mesh. Based on (Center, 2015), CFD concepts is replaced the system of differential equations with system of algebraic equations; in another words is the flow is converted to interdependency of the flow at those points and another points surrounding.

The development of fast and validated numerical procures, and increasing in computer speed, larger problems are being solved using CFD methods at cheaper cost and in a short time. Based on past study (Lomax et al., 1999), CFD methods are quickly replacing the experimental and analytical methods in many design and analysis applications. In addition, compared to experimental procedures in most engineering applications, CFD methods offer more complete set of information's in speed and reduced cost. It is usually providing all relevant flow information throughout the domain of interest. (Center, 2015)

CFD simulations are enable engineer or scientist to set the data of the flow solutions at true and original scale with the actual operating conditions. Compared to experimental and analytical techniques, CFD methods provide realistic conditions with economically represented and the results can be obtained directly.

2.2.2 THE APPLICATIONS OF COMPUTATIONAL FLUIDS DYNAMICS (CFD) ANALYSIS

Nowadays, the most widespread CFD software such as ANSYS-FLUENT and ANSYS-CFX are used commonly in the industrial applications. In this paper, the CFD can be used to stimulate the flow over a vehicle. For instance, the flow tested can be used to study the interaction between the speed of the car and the car body. The followings figure shows the prediction of the pressure induced by the interaction between the speed of the car and the car body. The models of the car body can be represented with models of varying complexity.

The pressure contour and a cutaway view that present on the result of the tested will be the datum in actual design improvement in order or to embrace the safety requirement being the first factors to be considered. CFD is attractive to industry since it is most cost-effective compared to physical testing. However, the engineering expertise is needed to obtain the validate solutions when the flow simulations are too complex and error-prone. In a large number of areas including engine components, auxiliary systems and modeling the aerodynamics of the car to minimize drag and optimize down force under operating conditions.

2.2.3 STAGES IN COMPUTATIONAL FLUID DYNAMICS (CFD)

In CFD simulations, there are three main stages that are pre-processing, solving and post-processing. In pre-processing stages, the problem faces are studied and identify to get the enough information before any solutions were listed. As the first step, the formulation of the problem such governing equations and boundary conditions are decided. Related to this paper, the boundary conditions a set inlet and outlet of the flow is tested on the model. After boundary conditions being analyzed, construct a computational mesh by setting the constant value of control volumes.

Next, the second stage is about solving the problems by using the data information collected from pre-processing stages. From the data before, the governing equation was analyzed and the algebraic equation was resulting to get the solution. The resulting of algebraic equations then be analyzed and solve using CFD methods.

The last stage in CFD simulations is post-processing. Post-processing is where the result of the analysis is being studied. Calculations of derived quantities such as forces and flow rates is defined and shown. The visualization such as graph and plots of the solutions also being shown in a digital computer those provide CFD simulations. From the visualizations using CFD software such as ANSYS-FLUENT and OPENFOAM, the flow direction shown in many colors with different meaning. The highest velocity in flow directions will label with red colors and the lowest velocity flow will label with dark blue color. From this visualization, the part with highest and lowest in velocity of the flow direction can be identified.(Center, 2015)