SUPERVISOR DECLARATION

"I hereby declare that I have read this thesis 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: ....

Supervisor: DR. CHENG SEE YUAN

Date: ....

© Universiti Teknikal Malaysia Melaka

DECLARATION

"I hereby declare that the work in this report is my own except for summaries and quotations which have been duly acknowledged."

Signature: ....

Author : CHO HON MENG

Date: ....

© Universiti Teknikal Malaysia Melaka

ACKNOWLEDGEMENT

During the course of this project, I have acquired an impressive indebtedness. It is virtually impossible to acknowledge everyone who has helped me to perform this study, but I wish to express my sincere gratitude to several people in particular. I am thankful to my supervisor Dr. Cheng See Yuan for his guidance and advice throughout the whole project. He spent his time in guiding me with his expertise and also encouraging me with his support. I have gained a lot of knowledge and learned a lot from conducting this project. Next, I would like to acknowledge and thank to the panel for this project for their comment and feedback. Last but not least, I would like to express a huge thank to all my friends for their understanding, encouragement and support when I am facing problems.

© Universiti Teknikal Malaysia Melaka

ABSTRAK

Industri automobil semasa perlu diperbaiki dalam reka bentuk kenderaan mereka disebabkan oleh perkembangan dalam teknologi setiap tahun. Dengan menentukan ciri-ciri dan corak aliran bendalir pada kenderaan yang menggunakan simulasi, industri dapat mengubah reka bentuk kenderaan mereka untuk mengurangkan daya seret pada kenderaan yang disebabkan oleh pergolakan. Walau bagaimanapun, memilih model gelora yang betul akan mempengaruhi keputusan simulasi yang diperolehi. Objektif utama kajian ini adalah untuk mencadangkan model gelora yang paling sesuai dengan membandingkan keputusan simulasi dan keputusan eksperimen. Kajian ini menggunakan rujukan Ahmed model kenderaan sebagai badan geometri yang dipermudahkan untuk simulasi dan aliran dihasilkan sekitar model Ahmed itu. Kajian ini menggunakan skala 1: 1 dengan model Ahmed asal dengan ketinggian model 288mm dan sudut condong 35 darjah. Aliran sekitar model Ahmed disiasat oleh sembilan model gelora yang terdapat di ANSYS FLUENT 15.0 pakej CFD. Keputusan berbanding menggunakan beberapa kriteria yang merupakan ciri aliran dan corak model, profil halaju untuk U-halaju dan pekali seretan model. Simulasi keputusan dibandingkan dengan baik berdasarkan nilai eksperimen diberikan dalam kesusasteraan. Model pergolakan yang meramalkan paling baik berbanding yang lain adalah model k-omega (standard) dalam kajian ini. Model k-omega (standard) meramalkan yang baik bagi majoriti profil halaju dan kesilapan menurunkan peratusan untuk perbandingan pekali seretan. Perbandingan antara saiz gelembung pemisahan dan pekali seretan tidak tepat. Beberapa cadangan dinyatakan untuk memperbaiki keputusan kajian ini.

© Universiti Teknikal Malaysia Melaka

ABSTRACT

The current automobile industries need improvement on their vehicle design due to the development in technology every year. By determining the fluid flow characteristics and pattern on the vehicle using simulation, industries are able to alter their vehicle design to minimize drag force on the vehicle which is caused by turbulence. However, selecting the correct turbulence model will influence the results of the simulation obtained. The main objective of the study is to propose the most suitable turbulence model by comparing simulation results and experimental results. The study uses the reference Ahmed vehicle model as the simplified geometric body for the simulation and flow is generated around the Ahmed model. The study uses a scale of 1:1 with the original Ahmed model with the model height of 288mm and a slant angle of 35 degrees. The flow around Ahmed model is investigated by nine turbulence models available in the ANSYS FLUENT 15.0 CFD package. The results are compared using several criterions which are the flow features and pattern of the model, velocity profiles for U-velocity and the drag coefficient of the model. The simulations results compared well based on the experimental value given in the literature. The only turbulence model that predicted the most above the others are the k-omega (standard) model in this study. The k-omega (standard) model predicted well for the majority of the velocity profiles and the lowest percentage error for the comparison of drag coefficient. The comparison between the size of separation bubble and drag coefficient are not accurate. Several recommendations are stated to improve the results of this study.

© Universiti Teknikal Malaysia Melaka

TABLE OF CONTENTS

© Universiti Teknikal Malaysia Melaka

© Universiti Teknikal Malaysia Melaka

CHAPTER	CONTENT		PAGES

© Universiti Teknikal Malaysia Melaka

LIST OF TABLES

© Universiti Teknikal Malaysia Melaka

LIST OF FIGURES

© Universiti Teknikal Malaysia Melaka

© Universiti Teknikal Malaysia Melaka

© Universiti Teknikal Malaysia Melaka

LIST OF SYMBOLS

μ	=	Kinematic Viscosity
F	=	Body Forces Acting on the Fluid
H	=	Body Height
k	=	Specific Turbulent Kinetic Energy
L	=	Length
m	=	Meter
m/s	=	Meter Per Second
m^2/s	=	Meter Square Per Second
mm	=	Millimeter
P	=	Pressure
Re	=	Reynolds Number
t	=	Time
$\bar{u}$	=	Mean Velocity
U	=	Bulk Velocity
u’	=	Fluctuating Velocity
V	=	Inlet Velocity
W	=	Width
ε	=	Turbulent Dissipation Rate
ν	=	Dynamic Viscosity
ω	=	Specific Dissipation Rate
ρ	=	Density
∂	=	Partial Differential

© Universiti Teknikal Malaysia Melaka

LIST OF ABBREVIATION

CAD	=	Computer Aided Design
CFD	=	Computational Fluid Dynamics
DNS	=	Direct Numerical Simulation
EVM	=	Eddy-Viscosity Models
FRSM	=	Full Reynolds Stress Models
LES	=	Large Eddy Simulation
RANS	=	Reynolds Averaged Navier-Stokes
RNG	=	Renormalization Group
RSM	=	Reynolds Stress Model
SST	=	Shear Stress Transport

© Universiti Teknikal Malaysia Melaka

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND STUDY

Analysis and prediction of turbulence has been an area of interest for researchers over the years as we encounter turbulence in our daily life. Among all fluid flows that occur, turbulence is one of the key phenomena in fluid dynamics. The major difficulty arises in predicting and solving turbulence is due to its random and chaotic behavior. Engineers and researchers have made an effort on getting the exact interpretation of flow behavior using various equations and numerical solutions. Turbulent flow calculations can be solved by Computational Fluid Dynamics (CFD) approach. A better understanding of flow physics especially turbulent flow can be achieved by conventional analysis and experimental techniques.

In the past decades, the technology of CFD evolved greatly becoming more and more important in development and industrial researches. Besides that, it has the methodology in analysis of predicting turbulent flow by solving governing equations using three approaches. The approaches consist of Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and a model based Reynolds Averaged Navier-Stokes (RANS) equations. This study only focuses on the turbulence modeling for closure of RANS equations where RANS-based turbulence model is used. With the emergence of advanced computers in the modern era, the development of turbulence model has

© Universiti Teknikal Malaysia Melaka

increased and improved the numerical simulation capabilities (Huang et al., 1997). Therefore, it becomes a significant importance to the automotive industry.

With the increase in applications of CFD every year, automotive industry makes great advances when it comes to aerodynamics and automotive designs of vehicles. Moreover, an execution of a good aerodynamic design states its importance as CFD has the ability of prediction for the performance of new designs before they are manufactured or carried out (Schaldach, 2000). Bluff body aerodynamics perhaps is one of the most distinguished research areas in wind engineering where turbulence will often occur in bluff body.

Bluff bodies have flow fields that can be characterized by random fluctuations, separation, stagnation, circulation and vortices. Mostly every phenomenon that is considered to be difficult to resolve in fluid mechanics is present in these flow fields (Murakami, 1997). With the help of RANS-based turbulence models, flow fields are modeled out with the help of detailed experimental data. However, validation and testing of various turbulence models are necessary to understand the capabilities and limitations of these models to accurately predict the complex flow of turbulence. This study is a part of the same venture to predict the flow behavior on bluff body using turbulence modeling which will be compared with experiments and to acquire the best turbulence model for automotive industries vehicle development.

1.2 PROBLEM STATEMENT

Turbulence modeling is commonly used in most automobile industries during the analysis stage of a vehicle to predict the effects of turbulence in a fluid flow. A type of simulation software named Computational Fluid Dynamics (CFD) is put to use for simulating and generating the results of turbulent flow. Among all DNS, LES and RANS in the CFD package, RANS is more preferable and widely used in the industries due to its simplicity and less computational time.

© Universiti Teknikal Malaysia Melaka

Most automobile industries required modeling and simulating the flow around a vehicle inspired bluff body as these industries need statistical aerodynamic properties to manufacture a vehicle. However, RANS has a variety of turbulence models to utilize and most industries do not know which turbulence models are more suitable for their simulation. Based on previous research, researches have perform a study on a cube-based body by comparing which RANS-based turbulence model achieves a better results based on experimental values. However, not all bluff bodies exhibit the same results as some bodies have different shapes and angles which affect the flow characteristics. Therefore, a vehicle-inspired bluff body has been chosen in this study to verify which turbulence model performs better. The comparison between the simulation values has to be compared with the experimental values to prove which turbulence model performs better and provide the best turbulence model for industries.

1.3 OBJECTIVE

The objective of this study is:

a) To conduct vehicle-inspired bluff body flow simulation using different RANS-based turbulence models.
b) To access the performance of each turbulence model by comparing numerical results to experimental data.
c) To propose the most suitable turbulence model for automotive aerodynamic simulation.

© Universiti Teknikal Malaysia Melaka

1.4 SCOPE

This study will consider all RANS-based turbulence model simulation using a simplified Ahmed vehicle-inspired bluff body where a scale of 1:1 of the original dimension of the Ahmed body model will be simulated. The rear slant angle used in this study is 35 degrees with length of slanting section of 222 mm. The Reynolds number of the flow simulated was 768,000 based on the velocity, U of 40 m/s and body height, H of 288 mm. The simulation software for this study is ANSYS FLUENT 15.0 package.

© Universiti Teknikal Malaysia Melaka

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Most fluid flow of engineering interest exhibits turbulence and this causes researchers to understand more about the characteristics and how turbulence may occur in a fluid flow. Furthermore, turbulence has the need to validate its flow features using various tools or software. This chapter describes the characteristics of a bluff body and introduces the methods of turbulence modeling using CFD.

2.2 AERODYNAMICS FEATURES ON BLUFF BODY

In aerodynamics, a bluff body commonly refers to one which has a frontal surface close to perpendicular of the flow direction. The front part of a bluff body is not in the same direction with the flow direction causing a disturbance for the fluid flow to pass through the bluff body. A bluff body is an exact opposite of a streamlined body where a bluff body will influence the fluid flow causing drag while a streamlined body will not influence the fluid flow as much as a bluff body. There are some research done by Nakamura (1993) where the researcher has performed an experiment to determine the aerodynamics effects on various bluff bodies and inspecting how turbulence is formed by these bodies. Figure 2.1 shows bluff bodies that are being tested and the arrow shows

© Universiti Teknikal Malaysia Melaka

the fluid flow direction. Most examples of bluff bodies are cube-shaped body, building, vehicles and much more. However, for streamlined body such as airfoils, there is no particular need to describe as this study is only for bluff body.

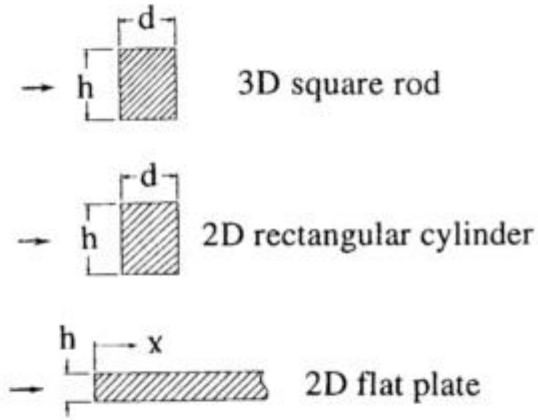

Figure 2.1: An example of bluff bodies

(Source: Nakamura, (1993))

One of the main features that characterized a bluff body in aerodynamics are a large region of separated flow where also a high amount of pressure drag occurs following vortex shredding phenomenon. The flow separations are determined mostly by sharp-edged contours such as a square shaped object that has a sharp corner.

A study performed by Lakehal and Rodi (1997) to determine the flow around a surface-mounted cube. Streamlines of the fluid flow is shown using computational software where this paper compared flow field experimental results with simulation results, this will be discussed later on this study. Figure 2.2 shows streamlines of the fluid flow around a cube and clearly the separation of flow began at the top front corner of the cube. The fluid flow is separated from the surface of the cube and turbulence began to occur. Turbulence phenomenon that occurs will cause various effects to the bluff body such as the increase of pressure drag follow by the increase of drag coefficient. Therefore, various methods of computation have been provided for researchers to perform simulation to capture every detail and characteristics of the fluid flow.

© Universiti Teknikal Malaysia Melaka

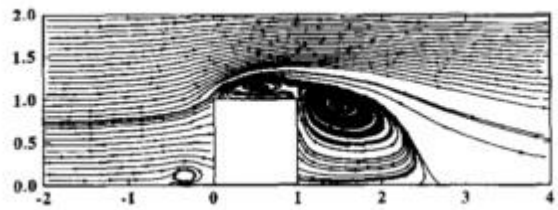

Figure 2.2: Streamline flow on a surface mounted cube
(Source: Lakehal and Rodi, (1997))

2.3 METHODS OF CFD NUMERICAL SIMULATION

In CFD, there are three methods of simulation where it includes levels of resolving turbulence which are fully resolved, partially resolved and unresolved.

Direct Numerical Simulation (DNS) is one of the methods that attempt to fully resolve all the arrangements of the flow by solving the Navier-Stokes equations. Turbulence and empirical models are not required for this method because DNS resolves the governing equations at all length and time scales. In a turbulent flow which consists of large and small eddies, DNS method is used to calculate and solve all large and small eddies in the flow where no turbulence modeling is used. However, the problem with this method is that it can consume tremendous computational resources and cause numerical difficulties. For now, DNS is just a research tool and is only feasible for simple flows at lower Reynolds number. Researchers agreed that DNS is not an option in the future but at least useful for general engineering problems and industrial applications. Moin and Mahesh (1998) issued a useful review of contributions of DNS in turbulence physics.

Large Eddy Simulation (LES) is one of the methods that attempts to partially resolve turbulent flow. For this technique, the larger scales of eddies which are more dependent on specific flow conditions and geometry are resolved by governing equations and smaller scales of eddies are modeled using turbulence model which are

© Universiti Teknikal Malaysia Melaka

LES-based models. Comparing LES with DES, LES provide a considerable computational savings over DNS at the cost of modeling and a more complex numerical algorithm. The LES method will probably become the mainstream for CFD. However, for most engineering applications, particularly aerospace, LES is not yet feasible on this era of technology.

By discovering all these methods, the most compatible computational method for this study of dealing with turbulence flow is by using the Reynolds Averaged Navier-Stokes (RANS) equations. For this method, turbulent flow that consist of large and small eddies are completely unresolved where modeling is done for both large and small scales. This is accomplished by the time averaging or Reynolds averaging governing Navier-Stokes equations. In the RANS method, only the mean flow field is predicted as flow is statistically averaged. This offers a huge savings for the computational cost when RANS is compared to both DNS and LES. Therefore, the present study is an investigation and comparison of turbulence modeling using the RANS equation.

2.4 BASIC TURBULENCE MODELING CONCEPT

A turbulence model is defined as a set of equations whether they are algebraic or differential which determine the turbulent transport terms in mean flow equations. They do not simulate the details of turbulent motion but only the effect of turbulence on the mean flow behavior. The concept of Reynolds averaging and the averaged conservation equations are some of the main concepts forming the basic of turbulence modeling.

The solution of the general equations of viscous fluid motion is required to solve any numerical fluid mechanics problem, in example the continuity equation and the Navier-Stokes equation. These types of equations come in sets of nonlinear partial differential equations with appropriate boundary conditions. The continuity equation and general form of the Navier-Stokes equations in tensor notation are given by:

© Universiti Teknikal Malaysia Melaka

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u_i)}{\partial x_i}=0 \tag{2.1}$$

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial\left(\rho u_i u_j\right)}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]+F \tag{2.2}$$

In equation (2.2), the left side of the equation is the instantaneous acceleration term whereas the right side of the equation is the convection term. The first term on the right side is the pressure gradient term and then followed by the viscous dissipation term whereas F represents the body forces acting on the fluid. For fluid flows which are incompressible, the density, ρ is denoted as a constant term where ρ = 0 and therefore the equation can be simplified to equation (2.3). For a three-dimensional flow, the equation is further classified into three components which are x, y and z components (Cartesian coordinate system).

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{2.3}$$

For equation (2.2), which is the Navier-Stokes equation (momentum equation), it can be simplified when the flow is classified as incompressible flow and three-dimensional flow in Cartesian coordinates. Therefore, equation (2.2) can be simplified to equation (2.4) where equation (2.4) shows the x-direction of the simplified Navier-Stokes equation. The time factor is removed due to a steady flow including the force acting on the fluid and the equation is divided with density, ρ.

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial P}{\partial z}+\nu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{2.4}$$

Thus, for three-dimensional flow, there are three Navier-Stokes momentum equation and continuity equation where there are four coupled differential equations for four unknowns which are u, v, w and P. All four equations can be solved using simultaneous equations for four unknowns. Therefore, no modeling required for a

© Universiti Teknikal Malaysia Melaka

laminar flow of a fluid with constant properties. The derivation of Reynolds Averaged Navier-Stokes (RANS) equation starts here where the fluid has turbulent properties and requires turbulence modeling.

2.4.1 Reynolds Averaged Navier-Stokes Equations (RANS)

There are various fluctuating flow parameters such as the unsteadiness in the flow that can be averaged based on the averaging concepts suggested by Reynolds in the year 1895. The three most common forms of averaging technique which are most distinguished in turbulence modeling research, depending on the type of flow being analyzed are the ensemble averaging, spatial averaging and time averaging. These average forms of the Navier-Stokes equations are referred to as the Reynolds Averaged Navier-Stokes (RANS) equations.

For a fully developed turbulent flow, the technique of time averaging and ensemble averaging is being utilized. The models based on RANS equation calculate mean quantities and also the fluctuating quantities based on the additionally modeled variables. Therefore, these equations are able to model transport of large eddies using RANS-based turbulence model where computational is required. An example of turbulence models available are from the two-equation models to the stress-transport models that researcher use during simulation of fluid flow.

2.4.2 Reynolds Stresses (Fluctuating Component)

The knowledge of Reynolds stresses starts with the understanding of the instantaneous velocity where at any point, the velocity is assumed to consist of a mean component (denoted by the top bar) which varies slowly with time and a fluctuating component (denoted by the prime). Equation (2.5) shows the instantaneous velocity that is described as the addition of the mean velocity and the fluctuating velocity.

© Universiti Teknikal Malaysia Melaka