AERODYNAMIC STUDY OF VEHICLE DRAG BASED ON BLUFF BODY

MOHAMAD KAMARUL BIN SHAMSUDDIN

BACHELOR OF MECHANICAL ENGINEERING WITH AUTOMOTIVE
FACULTY OF MECHANICAL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

2009

AEROL ... DRAG BASED ON BLUFF BODY

MOHAMAD KAMARUL BIN SHAMSUDDIN

A report submitted in fulfillment of the requirements
for the award of the Bachelor of
Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering

UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

ABSTRACT

The Ahmed body represents a simplified geometry of 3D car model that can be used to study the flow characteristic in the wake of vehicles. The main objective of this study is to analyze the drag of the Ahmed body with varying rear slant angle. The model occupied is 25°, 30°, and 35° rear slant angle where the critical remarkable drag occur. The model is simulated at the air velocity, 40 m/s. The result is validated using previous study, (Ahmed, 1984). The CFD proved to be a useful technique since the results compared reasonably well for both the current and the previous simulations. Concept of diffuser is applied to the original model to check the drag coefficient change due to the different angle of diffuser. The tested diffuser angle is 5°, 10°, 15° and 20°. The drag coefficient change is analyzed. Major finding shows that the velocity flow underbody is become higher when adding the diffuser. The flows structure behind the model was found to be somewhat inconsistency in a separated region over the slant and the recirculation flows behind the model. The flow characteristic is influented by velocity change underbody, the observation indicates that the formation of vortices at the rear part is bringing to the drag losses.

ABSTRAK

Ahmed Body mewakili geometri 3D kereta yang di permudahkan supaya ciri-ciri aliran di kawasan yang tidak sekata dapat dianalisa. Objektif utama dalam kajian ini adalah untuk mengkaji daya seretan ke atas *Ahmed Body* dengan mengubah sudut belakang model. Sudut belakang yang akan dikaji adalah 25°, 30° dan 35° di mana berlakunya kenaikan daya seretan yang tidak sekata. Kajian akan dilakukan dengan mengunakan simulasi CFD, iaitu menggunakan perisian FLUENT dan dikaji pada kelajuan angin 40 m/s. Keputusan yang diperolehi akan dibanding dan disahkn dengan keputusan yang diperolehi oleh kajian sebelum ini,(Ahmed,1984). Simulasi CFD membuktikan bahawa keputusan yang diperolehi adalah bersesuaian dengan keputusan semasa dan kajian sebelumnya. Dengan menggunakan konsep *diffuser*, kajian dilakukan keatas perubahan pekali daya seretan jika sudut *diffuser* diubah. Sudut *diffuser* yang akan digunakan dalam kajian ini adalah 5°, 10°, 15° dan 20°. Selepas itu, kajian keatas ciri-ciri aliran sekeliling model akan dianilisa berdasarkan keputusan pekali daya seretan yang diperolehi. Aliran di bawah model meningkat apabila menggunakan *diffuser*. Ciri-ciri aliran di belakang model menunjukkan kawasan pemecahan aliran yang tidak konsisten dan berlaku aliran putaran di bahagian condong. Kajian ini menunjukkan, perubahan aliran halaju udara di bahagian bawah model mempengaruhi pembentukan putaran aliran udara yang menyebabkan kepada peningkatan daya seretan.

TABLE OF CONTENT

CHAPTER 2 **LITERATURE REVIEW**

CHAPTER 3 **METHODOLOGY**

LIST OF FIGURES

LIST OF TABLES

LIST OF GRAPHS

LIST OF SYMBOLS

F_d	Drag Force
C_d	Drag Coefficient
A	Frontal area
ρ	Density of the air
V	Velocity
Re	Reynolds Number
ν	Kinematics Viscosity of air
μ	Dynamic Viscosity of air

LIST OF ABBREVIATIONS

2-D	Two Dimensional
3-D	Three Dimensional
CFD	Computational Fluid Dynamics
CAD	Computational Aided Design
DES	Detach Eddy Simulation
DNS	Direct Numerical Simulation
RANS	Reynolds Averaged Navier-Stokes
RSM	Reynolds Stress Model
TRANS	Transient Reynolds Averaged Navier-Stokes

CHAPTER 1

INTRODUCTION

1.0 PROJECT BACKGROUND

Aerodynamic vehicle development is of fundamental importance to the automotive industry. In vehicle aerodynamics, a number of configurations can be evaluated in a wind tunnel. To compete, numerical simulation must perform with comparable efficiency. For a vehicle, external and internal flow are closely related. The external flow has great influence on the performance characteristics and the directional stability of the vehicle drag(Akiyoshi, 1996). The numerical simulation of external aerodynamics corresponds to an incompressible viscous flow, and it is used turbulence models(Kapadia, 2003). The difficulty of the problem is due to certain factors such as strong vortices in the flow(Kapadia, 2003), wide zones in which a separation of the boundary layer is produced(Ahmed, 1984), and the strong presence of ground effect(Akiyoshi, 1996). In aerodynamic vehicle, CFD is commandly used for measure the flow pattern of the external flow. Ahmed Body has been selected in this study due to its geometric simplicity and availability of the experimental result(Guilmenuel, 2008). The aerodynamic characteristic of this body depended on the afterbody geometry, especially the rear slant angle(Akiyoshi, 1996). Detailed study on the dependence of rear slant angle on coefficient of drag has been performed both experimentally(Akiyoshi, 1996) and numerically(Vino, 2005). By adjusting the rear slant angle, separation can take place at the sharp corner(Guilmenuel, 2008). Prediction of seperation and reattachment of the flow is essential in predicting the true drag trend. In the variation of the rear slant angle, the remarkable drag breakdown occured between 30° and 40°(Akiyoshi, 1996).

1.1 PROJECT PROBLEM STATEMENT

The purpose of the study is to predicy the drag coefficient of the Ahmed Body base on the rear slant angle. The primary aim of this study is to analyze the change in the drag coefficient with different rear slant angle. The angle that use in this study is 25°, 30°, and 35° because at this angle the drag breakdown occur. The analysis is including the study of the drag coefficient at the difference angle by using the concept of diffuser to reduce the drag.

1.2 PROJECT OBJECTIVES

The objectives of this project are:

(i) To study the drag coefficient base on the rear slant angle of Ahmed Body at the selected remarkable drag breakdown angle.

(ii) To make the analysis and comparison between the CFD simulation with the previous experimental result for validation.

(iii) To reduce the drag of the Ahmed Body at the most critical drag occur by using the concept of diffuser.

1.3 SCOPES OF STUDY

The scope of project covered study and analysis the drag coefficient of the Ahmed Body base on rear slant angle using 2D simulation. The scope of this project included modeling of the Ahmed Body using CAD software, SolidWorks model with the difference rear slant angle. The study is involved creation of generic car model (Ahmed Body) using pre-processing tool (GAMBIT) and surface mesh have been created from the same software. The specification of the bondary condition at the domain boundary. Fluent has been used as solver. The FLUENT data is used to analyze the drag coefficient of the Ahmed Body. Then the validation of the prediction drag coefficient is validated by result from previous study.

CHAPTER 2

LITERATURE REVIEW

2.0 INTRODUCTION

Since the start of the 20^{th} Century, designers had been aware of aerodynamic forces acting on cars, but it wasn't until the 1930s that the materials and processes became available for the designers to be able to use aerodynamic principles cost effectively when designing cars. An aerodynamically well-designed rear end turned out to be effective only if the flow around the front of the car remained attached. A steep windshield can be very unfortunate for an otherwise well-designed body and can result in high drag. However, if the drag of the model is already high, for instance because of the seperation at the rear, the influence of windshield slope on drag is only minor (Huncho, 1998).

The remaining counter measure for reducing the drag of the vehicle is via the shape. Figure 2.1 show the developement of the car shape history on reducing the drag coefficient.

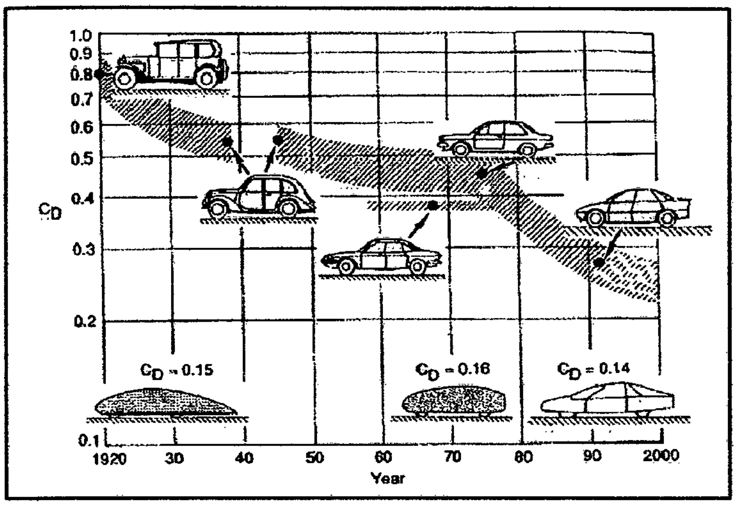

Figure 2.1 : History of Drag Coefficient

(Source; Huncho, 1998)

The study of three-dimensional flow around a ground vehicle has become a subject of significant importance in the automobile industry. One obvious way of improving the fuel economy of vehicles is to reduce aerodynamic drag by optimizing the body shape. Execution of good aerodynamic design under stylistic constraints requires an extensive understanding of the flow phenomena and, especially, how the aerodynamics are influenced by changes in body shape(Guilmineau, 2008). The flow region which presents the major contribution to a car's drag is the wake flow behind the vehicle(Ahmed, 1984). The location at which the flow separates determines the size of the separation zone, and consequently the drag force. Clearly, a more exact simulation of the wake flow and of the separation process is essential for the accuracy of drag predictions.

2.1 AERODYNAMIC

Aerodynamics is closely related to fluid dynamics and gas dynamics, with much theory shared between them. Aerodynamics is often used synonymously with gas dynamics, with the difference being that gas dynamics applies to all gases. Understanding the motion of air (often called a flow field) around an object enables the calculation of forces and moments acting on the object. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. By defining a control volume around the flow field, equations for the conservation of mass, momentum, and energy can be defined and used to solve for the properties. External aerodynamics is the study of flow around solid objects of various shapes.

Aerodynamic considerations are fundamental in car design. A car's aerodynamics will affect its handling, performance and appearance. Conversely, the aerodynamics will be affected by the cars required function, safety regulations, economy and desired aesthetics.

The main aerodynamic forces acting on a car are drag and downforce. The only desirable aerodynamic force is downforce. Both forces are related to each other and an increase in downforce will normally result in an increase in drag as well. This means that when designing a car compromises constantly have to be made. In general when designing a car aerodynamic drag wants to be reduced as much as possible, whilst downforce wants to be as high as possible. This becomes more important as the cars travel at a higher speed forces due to airflow increase with the square of the speed, so although aerodynamics have to be considered when designing road cars, they become a lot more important when designing racing cars.

2.1.1 EXTERNAL FLOW

In fluid mechanics, external flow is such a flow that boundary layers develop freely, without constraints imposed by adjacent surfaces. Accordingly, there will always exist a region of the flow outside the boundary layer in which velocity, temperature, and/or concentration gradients are negligible.It can be defined as the flow of a fluid around a body that is completely submerged in it.

The external flow around a vehicle is shown in Figure 2.2. In still air, the undisturbed velocity is the road speed of the car. Provided no flow seperation takes place, the viscous effects in the fluid are restricted to a thin layer of a few millimeters thickness. Beyond this layer the flow can be regarded as inviscid and its pressure is imposed on the boundary layer.

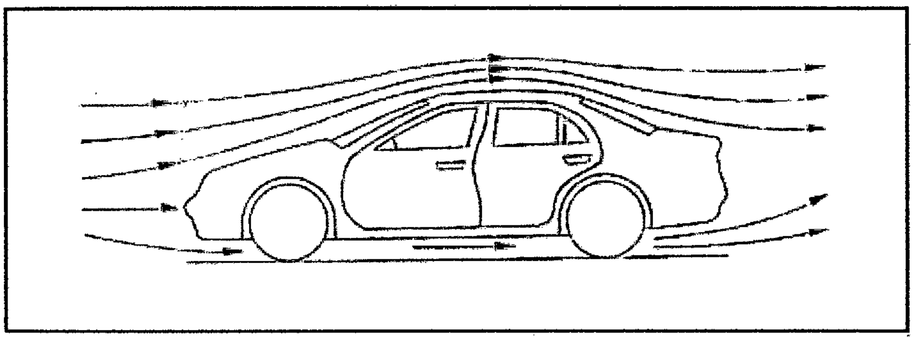

Figure 2.2 : External Flow Around Vehicle
(Source; Huncho, 1998)

2.1.2 BOUNDARY LAYER

When air flows over a solid surface a thin boundary layer is formed between the main airstream line and the surface. Any relative movement between the main airstream line flow and the surface of the body then takes place within this boundary layer via the process of shearing of adjacent layer of air.

A thicker boundary layer creates more viscous friction and also lead to flow seperation which lead to additional drag and a loss of downforce. There are two type of bondary layer, laminar and turbulent. The variation of boundary layer thickness along the plate may be seen in Figure 2.3.

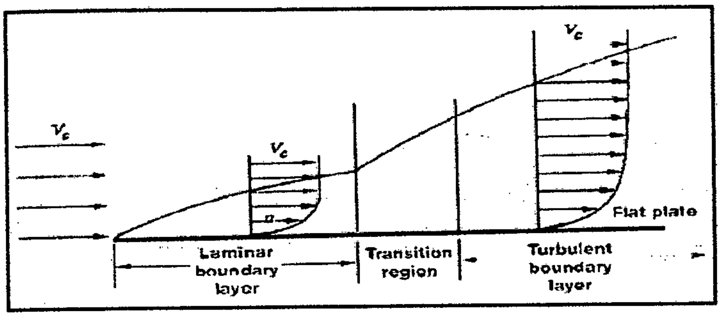

Figure 2.3 : Boundary Layer along a thin plate
(Source; Huncho, 1998)

2.1.3 FLOW SEPARATION

The flow near the surface may actually be reversed by the action of the pressure as shown in Figure 2.4. The point where the flow stops is known as the separation point. At this point, the main stream is no longer attached to the body but able to break free and continue in more or less straight line. It tries to entrain air from the region behind the body, the pressure in this region drop below the ambient. Vortices form and the flow is very irregular in this region(Hucho, 1998).

The phenomenon of the separation prevents the flow from simply proceeding down the back side of the car. The pressure in the seperation region is below that imposed on the front of the vehicle and the difference in these overall pressure force is responsible for drag form. The drag forces is arising from the action of the viscous friction in the boundary layer on the surface of the car is friction drag(Gillespie, 1992) .

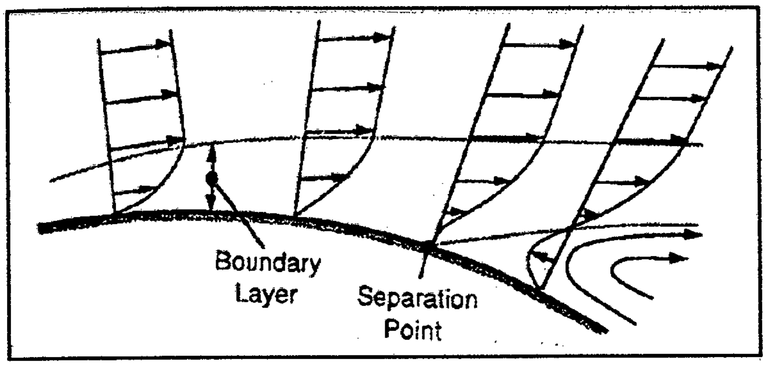

Figure 2.4 : Flow Separation in a adverse pressure gradient
(Source; Heisler, 2002)

2.1.4 AERODYNAMIC DRAG

In car design, drag is the force that slows down the car. The lower the drag, the faster the car will go while using the same amount of power. This means that top speeds are directly influenced by drag, along with fuel economy and emissions. The drag that a car produces wants to minimised, this will allow a higher top speed along with a better fuel economy. The way to do this is to get the air past the car with as little disturbance and change in direction as possible.

The equation shows that the drag of a vehicle is determined by its size, which is quite well defined by its frontal area (Figure 2.5), and its shape, the aerodynamic quality of which is defined by the drag coefficient. Also it shows that drag increases with the square of the speed. This means that at higher speeds drag reduction becomes a lot more important. Normally the size of a car, and hence its frontal area, is determined by design requirements. This means that efforts to reduce drag are concentrated on reducing the drag coefficient by proper shaping of the body.

The equation for aerodynamic drag is:

$$F_d = \frac{1}{2}\rho v^2 C_d A \tag{2.1}$$

Where:

F_d = Drag Force

C_d = Drag Coefficient

A = Projected frontal area of the vehicle

ρ = Density of the ambient air

V = Velocity of the vehicle

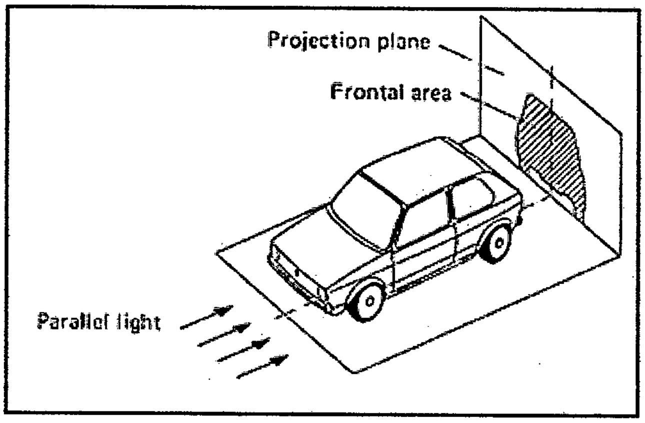

Figure 2.5 : Definition of Frontal Area

The drag coefficient (C_d) is a dimensionless quantity which is used to quantify the drag or resistance of an object in a fluid environment such as air or water. It is used in the drag equation, where a lower drag coefficient indicates the object will have less aerodynamic drag. The dimensionless drag coefficient is a ratio between two numbers in the same units of area. The reference area chosen for comparison depends on what type of drag coefficient is being measured. The drag coefficient of an object varies depending on its orientation to the vector representing the relative velocity between the object and the fluid. C_d is not a constant but varies as a function of speed, object length, fluid density and fluid viscosity.

2.2 REYNOLDS NUMBER EFFECT

The flow characteristics of wind passing across bluff body are depend on magnitude of inertial to viscous within the flow (the parameter are called Reynolds Number). The Reynolds Number is defined as:

$$Re = \frac{VD}{\nu} \text{ or } Re = \frac{\rho VL}{\mu} \tag{2.2}$$

Where V is the wind velocity, D, L is the lateral dimension of the body, ν is kinematics viscosity of air, μ dynamic viscosity of air and ρ is density of air.

2.3 BLUFF BODY

A bluff body is define as bodies that when subjected to a stream of fluid, suffers separation of large part of its surface or defined as bodies that are not streamline shape so that separation occurs (Williams, 2002). The occurrence of separation on a two dimensional bluff-body flow causes the creation of two vortices in the rear region of the body (Meneghini et al., 2002).

Nebres and Villafranca (1992) stated that "bluff body flows may be more complicated such as flow around bridges or automobiles and becomes more complex when the body geometry or the flow field is three-dimensional". Bluff bodies may be contrasted to streamlined bodies such as an airfoil which are shaped in such a way as to avoid regions of significant flow separation. Airfoils however becomes bluff bodies when they stall and at some operating conditions, such as an airplane wing at high angles of attack.

As highlighted in a review by Le Good and Garry (2004), the use of the simpified forms of pasenger vehicles has proven extremely useful in therm of understanding the fundamental flow characteristics associated with more complex passenger cars. Although many types of simplified passenger vehicle geometries have been investigated, one of the most popular has been the Ahmed Body model.

The influence of rear slant angles on drag coefficients was initially studied by Morel (1978). Morel found from his studies, that the rear slant angle of a simplified passenger vehicle significantly influences the drag coefficients of the vehicle. When the rear slant angle was increased from 0° to 60°, it was accompanied with a corresponding increase of drag coefficients. Beyond 60° the model experienced a significant increase of drag coefficients which almost doubled his previous value.

2.4 AHMED BODY

A real-life automobile is very complex shape to model or to study experimentally. However, the simplified vehicle shape employed by Ahmed (1984), generates fully three-dimensional regions of separated flow which may enable a better understanding of such flows. Ahmed's body is 1044mm long , 288mm high and 389mm width. The slant part is 222mm long, whatever the angle. The bottom surface of the Ahmed body is located at 50mm above the ground. This geometry is represented in (Figure 2.6) . The flow around this body is strongly influenced by the angle of the rear slant surface, which indicates that the large portion of aerodynamic drag is generated by the development of three-dimensional vortex separation from the rear slant surface.

The Ahmed body was first defined and its characteristics described in the experimental work of (Ahmed, 1984). Two configurations with slant angles of 25°and 35°are considered as a test case. In the experiment perform by the (Ahmed, 1984), flow velocity was taken 40 m/s, Reynolds number was 4.29 million based on model length.

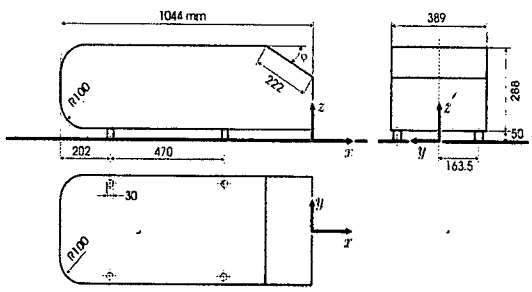

Figure 2.6 : Ahmed Body Dimension in mm
(Source; Williams, 2002)