I hereby declare that I have read this thesis and in my
opinion this thesis is sufficient in terms of scope and quality for the
award of the degree of
Bachelor Engineering Mechanical Engineering (Thermal - Fluid)

Signature : ..................................................

Name of Supervisor : Yusmady Bin Mohamed Ariffin

Date : May 30, 2006

# THE POTENTIAL LOCATION OF AIR FLOW AROUND CAR'S BODY IN GENERATING ELECTRICITY

MOHD SHAREDZAL BIN JONIT

Thesis submitted to the Faculty of Mechanical Engineering
In partial fulfillment of requirements for the degree of
Bachelor Engineering Mechanical Engineering (Thermal - Fluid)

Faculty of Mechanical Engineering
Kolej Universiti Teknikal Kebangsaan Malaysia

May, 2006

I declare that this thesis entitled "The Potential Location of Air Flow around Car's Body in Generating Electricity" is the result of my own research except as cited in the references.

Signature : ..................................................

Name : Mohd Sharedzal Bin Jonit

Date : May 30, 2006

*Dedicated to my family and friends…*

## ACKNOWLEDGEMENT

Assalammualaikum W.B.T

First and foremost, I would like to thank for Allah blessings. Without Allah help I won't be able to complete this PSM research as required and without the help and support from certain groups and individual it will be impossible for me to actually finish this research.

Not to forget, my supervisor Mr. Yusmady Bin Mohamed Ariffin and other lecturers who had given me endless help, guidance, and support me to mate up with the standard as required as a mechanical engineer student during the research. I believes that without his help is quite impossible for me event to complete with this research.

I also want to thank to KUTKM for giving me opportunity to get more experiences and knowledge during the period of the research. In fact, thanks to Mr. Yusmady Bin Mohamed Ariffin for giving me the guide line to do this report as required.

Last but not least, I would like to express my gratitude to my father and mother, my friends, not to forget for Ms.Lily Marwani Bte Mohd Radzi and all those that have been very supportive to me in finishing this research.

Thank you.

## ABSTRACT

A simulation study is undertaken by using CFX 5.7.1 to analyze the air flow around car's body. This study investigates the profile of air flow around a car. At the end of this study, the specific potential location of air flow in order to generate electricity could be identified. The car design was draw using Solidworks 2005 software which offers great features in order to develop the car model. The car was analyzed in the environments which have air flow velocity of 70 kilometer per hour to 110 kilometer per hour. The various velocity of air flow had been chose to make sure that the air flow profile is accurate. Among the number of turbulence models in existence, the standard K-ε model is most popular and applicable to many complex flows of engineering importance. The model is computationally economical and accuracy is reasonable. A mesh of triangle was chose for the model of the car and wind tunnel. This was done to ensure that an extensive automatic grid generation of the complex geometry could occur. The simulation was then resulted that at the car's roof has the highest air flow velocity. From this finding, the device to generate electricity will be put there and the optimal quantity of energy could be produce.

# ABSTRAK

Kajian simulasi bagi menganalisis pergerakan udara di sekitar kereta dijalankan menggunakan perisian CFX 5.7.1. Kajian ini menyiasat profail pergerakan udara di sekitar badan kereta. Di penghujung kajian ini, lokasi pergerakan udara paling laju pada badan kereta dapat di perolehi. Rekabentuk kereta dilukis menggunakan perisian Solidworks 2005 di mana perisian tersebut menyediakan ciri-ciri yang memudahkan pembuatan lukisan model kereta. Kereta tersebut dianalisa di dalam persekitaran yang mempunyai pergerakan udara 70 kilometer sejam hingga kelajuan udara 110 kilometer sejam. Pelbagai nilai halaju udara digunakan bagi memastikan profail pergerakan udara yang diperolehi tepat. Model bergolak K-ε telah dipilih bagi menjalankan simulasi ini. Model K-ε adalah model yang paling sesuai digunakan dalam kejuruteraan. Sirat segitiga digunakan bagi model kereta dan terowong udara. Keputusan simulasi menunjukkan udara paling laju bergerak di atas bumbung kereta. Dengan keputusan ini, maka alat bagi menghasilkan eletrik tersebut akan diletakkan di bahagian bumbung kereta dan jumlah eletrik yang optima mampu diperolehi.

# TABLE OF CONTENTS

# LIST OF FIGURES

# LIST OF SYMBOLS

| Symbol | Description | Unit |
|---|---|---|
| P | Pressure | Pa |
| ρ | Density | $kg/m^3$ |
| V | Velocity | km/h |
| F | Force | N |
| $l$ | Length | m |
| $u^+$ | Relative Velocity | km/h |
| $y^+$ | Relative Distance | m |
| $K$ | Turbulent Kinetic Energy | J |

## LIST OF ABBREVIATION

| Abbreviation | Description |
|---|---|
| CFD | Computational Fluid Dynamics |
| DNS | Direct Numerical Simulation |

# LIST OF APPENDIX

# CHAPTER 1

# INTRODUCTION

## 1.1 Problem Background

The title of this study is "The Potential Location of Air Flow around Car's Body in Generating Electricity". With this study, it was suggested to draw, using a 3D-modelling package, a car design which could be exported in formats compatible with the CFD packages available. The idea being that, with the 3D car model, it would be possible to spot the locations on a car's body that have a highest air flow velocity.

In this study, the specific potential location of air flow around car's body will be identified. The location is a suitable place to put a generating electricity device in order to generate an additional electric power for the car. We know that since the car is invented, there is always a problem that the car did not start because lacking of battery power. More over, now a day, there were so many other instruments can be install to a car. As an example, a stereo Hi-Fi, a video game, and in a luxury car, there is also a refrigerator. All this stuff consumes a lot of battery power. Although there is an alternator, which will charge the battery power while the car was moving, it is better if we put an extra energy source to the car.

This device basically works as a windmill. Wind can be used to do work. The kinetic energy of the wind can be changed into other forms of energy, either mechanical energy or electrical energy. As an example, when a boat lifts a sail, it is using wind energy to push it through the water. This is one form of work. Farmers have been using wind energy for many years to pump water from wells using windmills. The wind that flow around a moving car will spins the blades. The blades are attached to a small generator that will generate electric. Then, the energy produce will be stored in a battery.

The important thing in this study is an air flow around a car. There are several types of flow. The air flow around a car is a turbulent flow. In fluid dynamics, turbulence or turbulent flow is a flow regime characterized by low momentum diffusion, high momentum convection, and rapid variation of pressure and velocity in space and time. This can be best understood by using a flow over a golf ball as example, considering the golf ball to be stationary, with air flowing over it. If the golf ball was smooth, the boundary layer flow over it would be laminar at typical conditions. However, the boundary layer would separate early, creating a large region of low pressure behind the ball that creates high form drag. To prevent this from happening, the surface is dimpled to perturb the boundary layer and promote transition to turbulence. This results in higher skin friction, but moves the point of boundary layer separation rearward, resulting in lower form drag and lower overall drag. Same thing goes to a car. The shape of a car makes a turbulence flow.

By the end of this study, we should find out the best location at a car's body which has the highest air flow velocity, to put the device that will generate electricity.

## 1.2 Problem Statement

The aim of this study is to find a potential location of air flow around a car's body surface. The location must have the highest air flow velocity so that, it is the most suitable location for attaching the device in order to generate electricity. In this case we are concerned with cars and how their shape affects the flow of air around their body.

Aerodynamics is a branch of fluid dynamics concerned with the study of gas flows, first analyzed by George Cayley in the 1800s. This information was taken from Anderson, J. (1999). The solution of an aerodynamic problem normally involves calculating for various properties of the flow, such as velocity, pressure, density, and temperature, as a function of space and time. Understanding the flow pattern makes it possible to calculate or approximate the forces and moments acting on bodies in the flow. This mathematical analysis and empirical approximation form the scientific basis for heavier-than-air flight. Aerodynamics is concerned with the flow of air around objects. In this case we are concerned with cars and how their shape affects the flow of air around their body. The airflow around an object can create various forces on it, primarily either lift or down force and a resistive force opposing motion called drag.

For this study, an aerodynamic car design have to be choose to make sure that the most fluent air flow could be created around the car's body. But there is a problem with the car that has a great aerodynamic design. A details dimension for that car is protected.

As a result, a Proton Iswara, model 1.5S, 1.5GL our Malaysia's national car has been chose. Proton Iswara is suitable for this project because of a few factors:

1. Not expensive and affordable
2. Widely used
3. Not so hard to design by using a CAD software

Figure 1-1 shows a side view of the Proton Iswara.

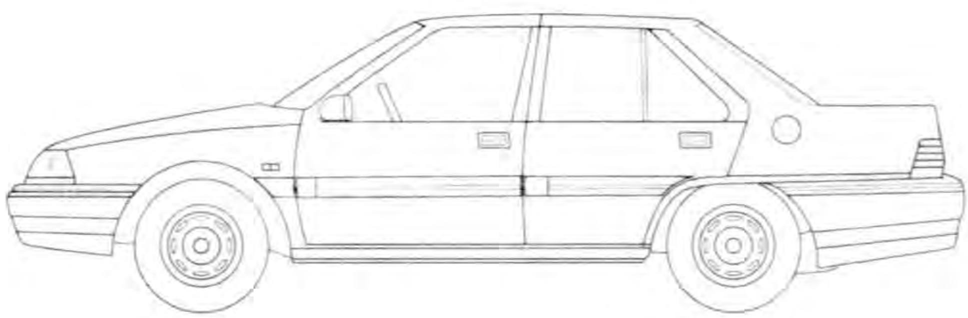

**Figure 1-1: Side view of Proton Iswara**
**(Perusahaan Otomobil Nasional Berhad (1996), *Proton Owner's Manual*, Malaysia: Proton Holdings)**

To locate the potential location of air flow, Computational Fluid Dynamics or CFD is the most suitable software. Computational fluid dynamics or CFD is the software to analyze problems in fluid dynamics.

CFD is a computational technology that enables to study the dynamics of things that flow. Using CFD, a computational car model will be built and it will represent a system that need to be study. After applying the fluid flow physics to this virtual prototype, the software outputs is a prediction of the air flow. CFD is a sophisticated analysis technique. John, A. (1995) stated that CFD is not only predicts air flow behavior, but also the transfer of heat, mass (such as in perspiration or dissolution), phase change (such as in freezing or boiling), chemical reaction (such as combustion), mechanical movement (such as an impeller turning), and stress or deformation of related solid structures (such as a mast bending in the wind).

## 1.3 Objective of the Study

To identify the specific potential location of air flow around car's body in order to generate electricity.

## 1.4 Scope of the Study

- Develop actual car model using CAD or other related software.
- Analyze the model in the environment which has air flow velocity of 70 to 110 km/h.
- Spot the locations that have highest air flow velocity.

## 1.5 Significance of the Study

This study is the first step must be taken before installing a car with the generating electricity device. This study will find out the best location that has the highest velocity of air flow. The device will be put on that place afterward, and as a result, it will work more properly and at optimum levels which produce a great quantity of electricity. Therefore, it will benefit all of the car users. They could travel without hesitating the energy source produce by the car will rundown. As an addition, maybe in the future, there will be no fuel needed to move a car. As we all know, the electric car that has been recently research will extend its battery lifetime by installing this device.

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 Introduction

As we know, the aim of this study is to find a potential location of air flow around a car's body surface. The location must have the highest air flow velocity so that, it is the most suitable location for attaching the device in order to generate electricity. This study will mainly focus more on a flow over a car's body. In this case we are concerned with car and how their shape affects the flow of air around its body.

This chapter will tell us about the pattern of air flow around a car. The laminar flow that hit the car while it's moving will change to the turbulent flow. It is because the unsmooth shape of the car. Other than that, turbulent models will also be explain in this chapter.

## 2.2 Turbulent Flow

Flow is the continuous movement of a fluid, either a liquid or a gas, from one place to another. Basically there exist three types of flows, namely laminar flows, turbulent flows and transient flows. A laminar flow is could be call as a 'simple' flow while a turbulent flow is a 'complicated' flow. A fluid, liquid or gas molecules in a laminar flow move at more or less the same speed, in the same direction. That is why the flow is smooth and not 'complicated'. Meanwhile, a turbulent flow has molecules that tend to move in different directions at different speeds. Transient flow is such a flow where the velocity and pressure changes over time. Transient flows usually occur during the starting or stopping of a pump, the opening or closing of a tank, or simple changes in tank levels.

How easily a fluid becomes turbulent depends to a large extend on its viscosity. Simply speaking, viscosity is the resistance of a fluid either a liquid or a gas to movement. The more viscous a fluid is the less likely it is to become turbulent. Thus, water or air which has a low viscosity can become turbulent relative easily, while honey or syrup, which is very viscous, and tend not to become turbulent.

When the flow is turbulent, the flow contains eddying motions of all sizes, and a large part of the mechanical energy in the flow goes into the formation of these eddies which eventually dissipate their energy as heat. As a result, at a given Reynolds number, the drag of a turbulent flow is higher than the drag of a laminar flow. Also, turbulent flow is affected by surface roughness, so that the increasing of roughness will increases the drag.

Thomas, D. (1992) stated that the external flow over all kind of vehicles such as cars, airplanes, ships and submarines are a turbulent flow.

## 2.3 Air Flow around a Vehicle

The gross flow over the body of a car is governed by the relationship between velocity and pressure expressed in Bernoulli's Equation. The equation is:

$$P_{static} + P_{dynamic} = P_{total} \quad \text{(2-1)}$$

$$P_s + 1/2\rho\, V^2 = P_t \quad \text{(2-2)}$$

This relationship is derived by applying Newton's Second Law to an incremental body of fluid flowing in a well-behaved fashion. As an explanation, 'well-behaved' means that the flow is moving smoothly and is experiencing negligible friction conditions that apply reasonably to the air stream approaching a motor vehicle. In deriving the equation, the sum of the forces brings in the pressure effect acting on the incremental area of the body of fluid. Equating this to the time rate of change of momentum brings in the velocity term.

Bernoulli's equation states that the static plus the dynamic pressure of the air will be constant ($P_t$) as it approaches the vehicle. Visualizing the vehicle as stationary and the air moving (as in a wind tunnel), the air streams along lines, appropriately called 'streamlines'. A bundle of streamlines forms a streamtube. The smoke streams used in a wind tunnel allow streamtubes to be visualized as illustrated in Figure 2-1.

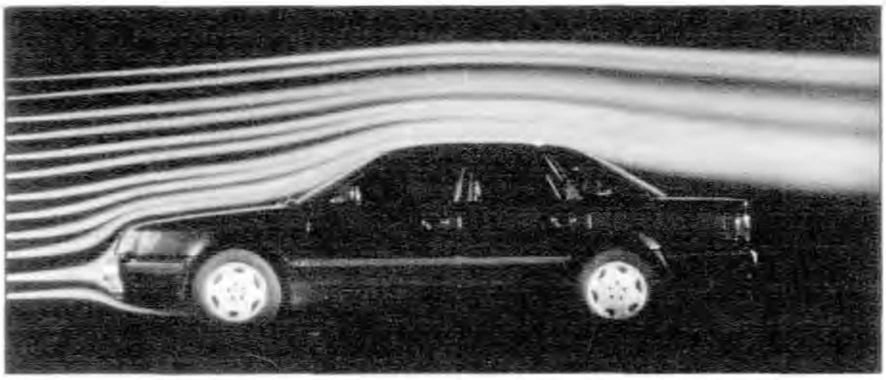

**Fig. 2-1: Streamtubes flowing over an aerodynamic body (Thomas, D. 1992)**

From Thomas, D. (1992), we know at a distance from the vehicle the static pressure is simply the ambient, or barometric, pressure ($P_{atm}$). The dynamic pressure is produced by the relative velocity, which is constant for all streamlines approaching the vehicle. Thus the total pressure, $P_t$, is the same for all streamlines and is equal to $P_s + 1/2\ \rho V^2$.

As the flow approaches the vehicle, the streamtubes split, some going above the vehicle, and others below. By inference, one streamline must go straight to the body and stagnate (the one shown impinging on the bumper of the car). At that point the relative velocity has gone to zero. With the velocity term zero, the static pressure observed at that point on the vehicle will be a total pressure.

Consider what must happen to the streamlines flowing above the hood. As they first turn in the upward direction, the curvature is concave upward. At a distance well above the vehicle where the streamlines are still straight, the static pressure must be the same as the ambient. In order for the air stream to be curved upward, the static pressure in that region must be higher than ambient to provide the force necessary to turn the airflow. If the static pressure is higher, then the velocity must decrease in this region in order to obey Bernoulli's Equation.