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 (Automotive)."

Signature : ....

Supervisor : ....

Date : ....

SIMULATION OF AIR FLOW AROUND VEHICLE FRONT WINDSCREEN

MUHAMMAD YAZID BIN HASNAN

This report is presented in

Partial fulfilment of the requirements for the

Bachelor of Mechanical Engineering (Automotive)

Faculty of Mechanical Engineering

Universiti Teknikal Malaysia Melaka

APRIL 2011

DECLARATION

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

Signature :   ....

Author :   ....

Date :   ....

To my beloved parents, Mr. Hasnan Bin Mohd Zin and Mrs. Kamariah Bt Ngah...

My family...

My Supervisor, Mr. Fudhail Bin Abdul Munir...

My friends...

Universiti Teknikal Malaysia Melaka

April 2011

ACKNOWLEDGEMENT

I would like to express my gratitude to all those who gave me the possibility to complete this project. I want to thank to Allah S.W.T for giving me good health to do the necessary research work. In addition, I would like to thank to my fellow friends who always encouraged me with this project.

I am deeply indebted to my supervisor, Mr. Fudhail bin Abdul Munir from Faculty of Mechanical Engineering whose help, stimulating suggestions and encouragement helped me in all the time of the research and writing of this Final Year Project.

Lastly, I would like to give my special thanks to my parents whose patient love enabled me to complete this project, sacrifice their time and money in show of support towards me. I hope that all results obtained in this research can be used as references for the betterment of science and technology.

ABSTRACT

In this project, simulation of air flow around vehicles front windscreen had been carried out. The models that have been chosen for this research were models from the national car manufacturers. ANSYS CFX software was used to run the simulation. k-ε turbulent model is used to calculate flow around the vehicle model and make a comparison between all models at vehicle speed 25m/s (90km/h). The main objective of the study is to obtain the flow pattern around vehicle front windscreen, to determine factors affecting aerodynamics of a vehicle. Several factors that influence the flow pattern such as flow separation, the effect of pressure coefficient (Cp) and types of turbulent has been studies. Detail velocity variation and pressure distribution plots around the vehicle envelopes have been presented. Proton Saga BLM shows higher pressure distribution at the bottom end of the windscreen compare to other models. Perodua Myvi show higher streamline velocity compares to other models. The Cp variations for all models are very large when close to the edges which are at the top edges of the front bumper, top edges of the windscreen and at the rear end of the vehicle.

ABSTRAK

Dalam projek ini, simulasi aliran udara di sekitar kaca hadapan kenderaan telah dilakukan. Model yang telah dipilih unutk kajian ini adalah model dari pengilang kereta nasional. Perisian ANSYS CFX telah digunakan untuk mensimulasikan model turbulen k-ε untuk mengira aliran di sekitar model kenderaan dan membuat perbandingan antara semua model kenderaan pada kelajuan 25m/s (90km/j). Tujuan utama kajian ini adalah untuk mendapatkan pola aliran di sekitar cermin hadapan kenderaan untuk menentukan factor yang mempengaruhi aerodinamik kenderaan. Beberapa faktor yang mempengaruhi pola aliran seperti pemisahan aliran, pengaruh pekali tekanan (Cp) dan jenis- jenis turbulen telah dikaji. Penelitian kelajuan dan penyebaran tekanan sekitar permukaan kenderaan telah dilakukan. Proton Saga BLM menunjukkan edaran penyebaran tekanan yang lebih tinggi pada bahagian bawah cermin hadapan berbanding model yang lain. Perodua Myvi menunjukkan kelajuan pengaliran udara yang lebih tinggi berbanding model yang lain. Variasi Cp pada semua model kenderaan adalah tinggi pada bahagian tepi atas bumper hadapan, tepi atas cermin hadapan dan pada bahagian belakang kenderaan.

CONTENTS

CHAPTER	TITLE	PAGE
	DECLARATION	
	DEDICATION	
	ACKOWLEDGEMENT	
	ABSTRACT	
	ABSTRAK	
	CONTENTS	
	LIST O FIGURE	
	LIST OF SYMBOL	

LIST OF FIGURE

LIST OF SYMBOL

ρ Density of air at 300K

V Free stream velocity

μ Dynamics fluid viscosity

Re   Reynolds number

L   Characteristic length

Cp   Pressure coefficient

P   Pressure at the car

P∞ Static pressure of the free wind

CHAPTER 1

INTRODUCTION

1.1 OVERVIEW

This research was conducted to study the flow around a vehicle front windscreen. Modern windscreen is made of laminated safety glass, a type of treated glass, which consists of two curved sheets of glass with a layer laminated between them for safety [1].

Windscreen plays an important role in the overall design of a vehicle. It is used to protect the vehicle's occupants from external variables such as wind, extreme temperature and flying object, as well as providing an aerodynamically formed window.

In this study, Computational Fluid Dynamics (CFD) is used to determine the flow pattern around a vehicle front windscreen. CFD software is used to calculate the pressure distribution, drag and lift coefficient in order to define the flow pattern around a vehicle front windscreen.

The aim of this study is to obtain the flow pattern around a vehicle front windscreen and to see the effect of the flow to the driving and comfort characteristic.

Comparison between three national car model which are Proton Saga BLM, Perodua Myvi and Perodua Kenari to investigate the effect of the flow pattern. All car model are chosen based on the average car been used by Malaysian. The result of this study hopefully can be used as a reference and guideline by other researcher.

1.2 PROBLEM STATEMENT

At average speed (90km/h) and above, as driving speed increased, aerodynamic noise on ground vehicle were becoming relatively effect the comfort of the occupants especially at the area of front windscreen and side mirrors. It is vital for the researcher and engineer to study these phenomena in order to rectify the problem for the future models.

1.3 OBJECTIVE

The objectives of this study are:
1. To study flow around a vehicle front windscreen.
2. To obtain the flow pattern around a vehicle front windscreen.

1.4 SCOPE

The scopes of this proposed project are:
1. Simulate flows for different s national car's model which is Proton Saga BLM, Perodua Myvi and Perodua Kenari.
2. To investigate the flow pattern around a vehicle.

1.5 EXPECTED RESULT

At the end of the research, it is expected that the flow pattern around windscreen can be obtained. The flow pattern then will show which national car model is more aerodynamic.

CHAPTER 2

LITERATURE REVIEW

2.1 OVERVIEW

This chapter will define and discuss the information related to the background knowledge of vehicle windscreen, the aerodynamic theory and the used of Computational Fluid Dynamics (CFD) in determining the flow around front windscreen of a vehicle. Reviews on previous studies concerning the flow on frontal area of vehicle are included as well.

2.2 VEHICLE WINDSCREEN

The windscreen of a car is the front window and generally made of laminated safety glass, which consists of two curved sheet of glass with a plastic layer laminated between it for safety purpose [1]. The car windscreen is a very important

part of any car and the unfortunate thing is that they are prone to damage. The windscreen is usually made of glass which is a fragile material by nature, this means when driving extra care should be taken to ensure that the windscreen is in perfect condition each time when turn on the engine to prevent further problems.

Windscreen mainly use to protect the vehicle's occupants from wind, temperature extremes, and flying debris such as dust and rocks, as well as providing an aerodynamically formed window towards the front[1]. Modern windscreens are well made and designed to be dual layered, it also aids in the rigidity of the structure of the car. As with many modern products it goes through intense quality checking to reduce the chance of it being easily damaged. The windscreen though, takes a lot of wear and tear in part due to the wiper blades.

Figure 1: Windscreen of a car

As in some country, there are some problems that occur as the cold weather means we need to scrape the frost off the windscreen which increases the chances of it becoming scratched. There also tends to be more grit and stones on the road than usual and these can easily flick up from the road and chip the windscreen, particularly when driving at high speed for example on the highway. It is really important that you drive with a clear screen because if there are chips or it is dirty, your vision will be impaired which could quickly lead to an accident.

A stone chip is the most common form of damage when it comes to windscreens. There are many road works taking place across the country and this normally means gravel, grit and other debris on the road. When driving around areas with poor road surfaces of which are covered in loose surface materials, take extra care and drive slowly so the stones are less likely to get flicked up on to the windscreen and potentially chipping or cracking it.

Chips in a windscreen can start off as a minor problem but it can quickly turn into a bigger problem as the chip cracks the glass surface. This is why it is very important to get the chip repaired as soon as you notice it. The main reason is cost as getting a chip repaired is relatively cheap and some companies will do it for free if you have the right insurance cover. Buying a new windscreen can cost lots of money depending on the model of the car. It is also more environmentally friendly to get a chip fixed as your original windscreen will not have to be disposed of.

2.3 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problem that involve fluid flows [2]. CFD predicts the fluid problem by describing the fluid flow case mathematics equation and solving it. The mathematical technique includes finite deference method, finite element methods and filter –scheme methods.

CFD can be divided into two major categories, the commercial code and the research code. FLUENT, STAR-CD and CFX are some of the commercial code for computational fluid dynamics. These codes are mostly developed by some commercial corporation and suitable for general applications. The research codes are mainly developed by research institution and these codes are focused on certain cases only. CFX was used in this study as a tool to predict the flow around a front windscreen of a vehicle.

CFD can be applied in many fields of researches; one of the fields that gain the most advantages from the development of CFD methods was the aerodynamics field. By using CFD, the details about the flow field such as shear stress, velocity and pressure profile and flow streamlines can be obtained [3].

2.3.1 Turbulence Model

Turbulence is very complex and details of turbulence motion are too small scale in diameter. The CFD simulations of turbulence flow are much more difficult than laminar flow. Turbulence flow always unsteadies and three dimensional – random, swirling, vertical structure called turbulence eddies of all orientations arise in turbulence [3].

Complexity of different turbulence models may vary depends on the details that want to be observed and investigate by carrying out such numerical simulations. Complexity is due to the nature of Navier-Stoke equation (N-S equation). N-S equation is inherently nonlinear, time-dependent and three-dimensional [13].
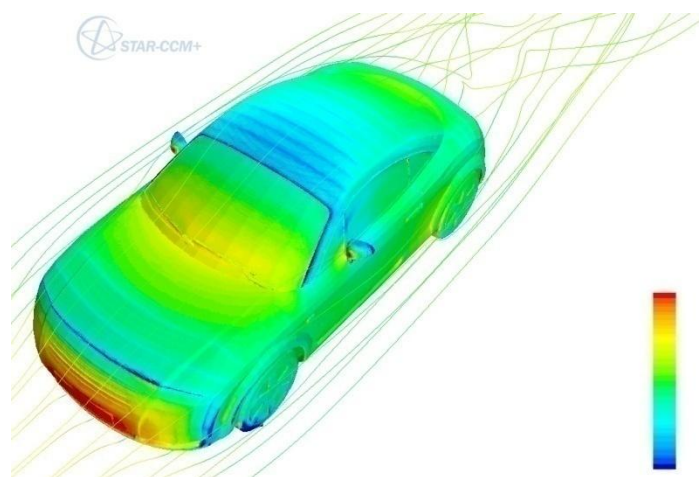

Figure 2: Reynolds Average Navier- Stoke (RANS) turbulence model [15]

Turbulence could be thought of as instability of laminar flow that occurs at high Reynold numbers (Re). Instability of laminar flow forms an interaction between non-linear inertial terms and viscous term in N-S equation. These interactions are rotational and fully three-dimensional. That is why no satisfactory two-dimensional approximations of turbulence flow are available.

Even though, certain properties could be learned about turbulence using statistical methods. These introduce certain correlation functions among flow variables. However it is impossible to determine these correlations in advance.

There are some characteristic features of turbulence flow such as irregularity, random and chaotic. Turbulence flow occurs at high Reynold number, for example, the transition to turbulence flow in pipes occurs at Re= 2300 and in boundary layer at Re=100000. In addition, diffusivity also some of the characteristic of turbulence flow. This means that the spreading rate of boundary layers increase as the flow becomes turbulent.

2.3.2 Mesh Generation.

Mesh generation is the first step in CFD solution, which consists of defining the cells on which flow variables such as velocity and pressure are calculated throughout the process [3]. Meshing is an integral part of the CAE analysis process. The mesh influences the accuracy, convergence, and speed of the solution [10]. The time to create a mesh model is often a significant of the time it takes to get the results from CAE solution. The better and more automated the meshing tools, the better the outcome solution.

An unstructured simplex mesh requires a choice of mesh points (vertex nodes) and a triangulation. The mesh should have small elements to resolve the fine details of the geometry, but larger sizes where possible to reduce the total number of nodes. Furthermore, in numerical applications high quality elements are needed (for

example triangles that are almost equilateral) to get accurate discretizations and well conditioned linear systems [11].
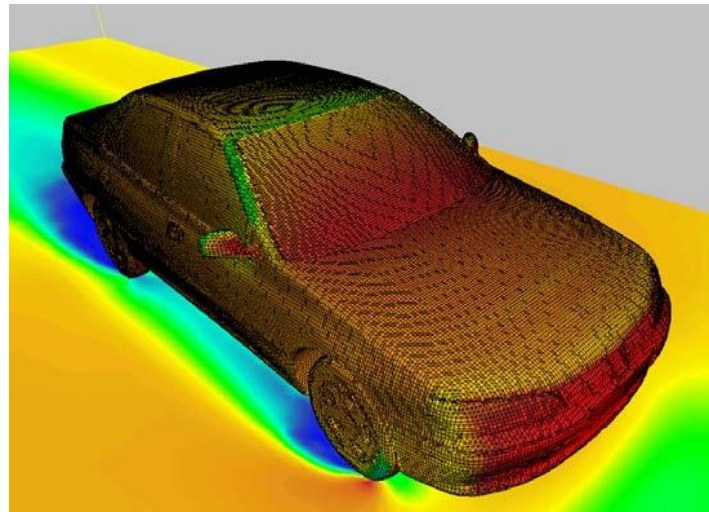

Figure 3: Illustrate the mesh generation [35]

An unstructured mesh is usually applied to the complex geometries. However, there are some advantages unstructured mesh compare to structured mesh, it converges more rapidly and often are more accurately by utilizing the index feature of structured mesh.

2.3.3 Boundary Condition

In a numerical simulation, it is impossible and unnecessary to simulate the whole universe. Generally, region of interest are choose in which will be conducting a simulation. The interesting region has a certain boundary with the surrounding environment. Numerical simulations also have to consider the physical processes in the boundary region. In most cases, the boundary conditions are very important for the simulation region's physical processes [12].

There are several types of boundary condition which is wall boundary condition, inflow/outflow boundary condition, miscellaneous boundary condition and internal boundary condition [3]. Wall boundary condition is the simplest boundary condition, since the fluid cannot pass through a wall and the non-slip condition. Appropriate boundary conditions are required in order to obtain an accurate CFD solution.
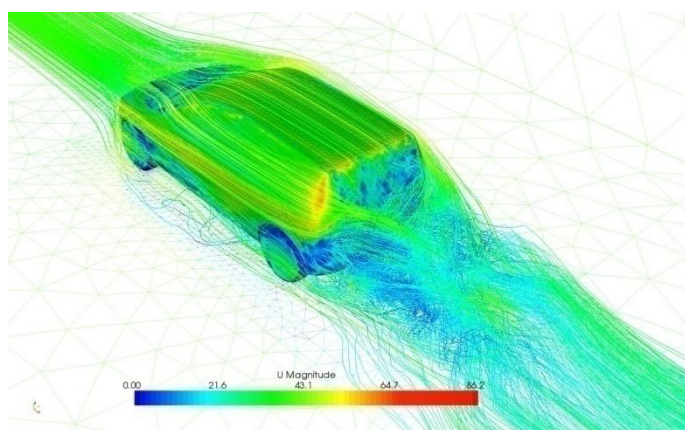

Figure 4: Turbulent flow around a car computed from Navier-Stokes equations with slip boundary condition [14]

Different boundary conditions may cause quite different simulation results. Improper sets of boundary conditions may introduce nonphysical influences on the simulation system, while a proper set of boundary conditions can avoid that. So arranging the boundary conditions for different problems becomes very important. While at the same time, different variables in the environment may have different boundary conditions according to certain physical problems.

Usually the boundaries which fluid enters computational domain will set as inlet or leaves the domain as outlet, which generally categorized as either velocity specified conditions or pressure specified conditions.

2.3.4 Structure of a CFD

ANSYS CFX software has three basic components, which are pre- processor, solver and post-processor. The pre-processor consist of three main components: geometry, meshing and pre-CFX. Pre-processor has various numbers of function that allows user to predefine the domain fluid, known as computational domain and build the geometry of the zone (surface) considered, creating a mesh system and tuning the mesh to improve the computational result and quality. User also may specify the properties of the domain fluid and other material that been used at this stage.

The solver in ANSYS CFX will carry the major computations and providing the numerical solutions. It consists of two components that will define the equation and come out with the solution. The output of the solver is a set of flow variables at the mesh nodes.

The post-processor will then construct a picture of the simulated flow problem by displaying the geometry of the problem, the computational domain and the mesh system [34].
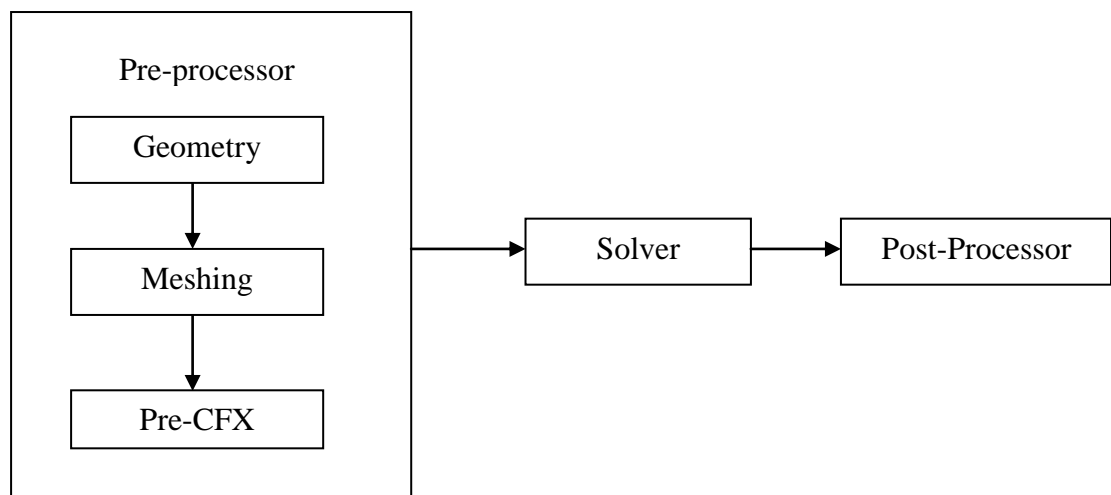

Figure 5: Illustrate ANSYS CFX procedure [34]