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 A VEHICLE SIDE MIRROR

NUR ATIQAH BTE SHAHARI

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

JUNE 2012

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. Shahari Che Bakar and Mrs.Norasiah Abdul Karim...

My family...

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

My friends...

Universiti Teknikal Malaysia Melaka

June 2012

ACKNOWLEDGEMENT

First of all, I would to say Alhamdulillah. Praise to Allah with His bless,I am very thankful for giving chances and ability to complete this project. In addition, I would like to thank to my fellow friends who always encouraged me with this project.

Noted with thank to my helpful supervisor, Mr. Fudhail bin Abdul Munir whose help, stimulating suggestions and encouragement helped me in all the time of the research and writing of this Final Year Project.. All of the support and supervision from him is completely useful and it will not be forgotten. The opportunity he had willingly given to me is a very substantial key for me to serve better for the industry and educational field particularly.

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 vehicle side mirror had been investigated. The side mirror models that have been chosen for this research were models from the national car manufacturers. ANSYS FLUENT 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 research is to study and obtain the flow pattern around vehicle side mirror. 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. Simulation and results of this study showed the airflow pattern for each selected national car model and the effect in terms of fuel efficiency had been discussed. Perodua Kenari shows higher pressure distribution compared to other models. Perodua Myvi show higher streamline velocity compared to other models.

ABSTRAK

Dalam projek ini, simulasi aliran udara di sekeliling cermin sisi kenderaan telah dijalankan. Model cermin sisi yang telah dipilih untuk kajian ini adalah model daripada pengeluar kereta nasional. Perisian ANSYS FLUENT telah digunakan untuk menjalankan simulasi. k-ε model digunakan untuk mengira aliran di sekitar model cermin sisi kenderaan dan membuat perbandingan antara semua model kenderaan pada kelajuan 25m / s (90km / h). Objektif utama kajian ini ialah untuk mengkaji dan mendapatkan corak aliran di sekeliling cermin sisi kenderaan. Beberapa faktor yang mempengaruhi corak aliran seperti pemisahan aliran, kesan pekali tekanan (Cp) dan jenis daripada yang bergelora telah kajian. Perincian perubahan halaju dan plot taburan tekanan di sekeliling cremin sisi kenderaan telah dibentangkan. Simulasi dan keputusan kajian ini menunjukkan corak aliran udara bagi setiap model kereta nasional yang dipilih dan kesan dari segi kecekapan bahan api telah dibincangkan. Perodua Kenari menunjukkan taburan tekanan yang lebih tinggi berbanding dengan model-model lain. Perodua Myvi menunjukkan lebih tinggi memperkemaskan halaju berbanding dengan model-model lain.

CONTENTS

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
$P\infty$	Static pressure of the free wind

CHAPTER 1

INTRODUCTION

1.1 OVERVIEW

This study has been carried out in order to study the flow around a vehicle side mirror. A side mirror also known as wing mirror is a mirror found on the exterior of motor vehicles. It is used to help the driver see rear areas and to the sides of the vehicle, outside of the driver's peripheral vision in the 'blind spot' [1].

Most modern cars mount their side mirror on the doors, normally at “A” pillar. The side mirror can be manually adjusted or by using remote for vertical and horizontal adjustment to produce enough coverage depends on driver height differences and seated position. As windscreen plays an important role in the overall design of a vehicle, side mirror also has its own role.

This project deals with Computational Fluid Dynamics (CFD), which is used to establish the flow pattern around a vehicle side mirror. In this case, CFD software is used to calculate the pressure distribution, drag and lift coefficient in order to obtain the flow pattern around a vehicle front windscreen.

Two main objectives can be defined from this study. The first is to study the airflow characteristic around a vehicle side mirror and to obtain the flow pattern around a vehicle side mirror using Computational Fluid Dynamics (CFD). Comparison made between three national car models which are Proton Saga BLM, Perodua Myvi and Perodua Kenari to investigate the effect of the flow pattern. The result of this study hopefully can be used as a reference and guideline by other researcher.

1.2 PROBLEM STATEMENT

As the main concern of automotive aerodynamics is reducing drag in order to reduce the roll resistance, drivers are always curious to see the effect in terms of fuel efficiency. In this study, the airflow pattern around a side mirror will be investigate and define. Comparison between three national car models which are Proton Saga BLM, Perodua Myvi and Perodua Kenari are made to investigate the effect of airflow pattern. Simulation and results of this study will show the airflow pattern for each selected national car model and the effect in terms of fuel efficiency will be discussed.

1.3 OBJECTIVE

The objectives of this study are:

1. To study airflow characteristics around a vehicle side mirror.
2. To obtain the flow pattern around a vehicle side mirror using Computational Fluid Dynamics (CFD).

1.4 SCOPE

The scopes of this proposed project are:

1. Simulate airflow for different national cars model which is Proton Saga BLM, Perodua Myvi and Perodua Kenari.
2. To investigate the flow pattern around a vehicle.
3. Simulation will be done using ANSYS 12.1.

1.5 EXPECTED RESULT

At the end of the research, it is expected that the flow pattern around side mirror can be obtained and the parameter that affects the fuel efficiency at vehicle will be discussed.

CHAPTER 2

LITERATURE REVIEW

2.1 OVERVIEW

This chapter will discuss about interests related to the basic and background knowledge of side mirror, the aerodynamic theory and the used of Computational Fluid Dynamics (CFD) in defining the airflow around a vehicle side mirror. Reviews on previous studies concerning the airflow around a vehicle side mirror are included as well.

2.2 Aerodynamic Effects on an Automotive Rear Side View Mirror by By Alam, R. Jaitlee and S. Watkins.

A flat rear mirrored face defined the function of a rear view mirror during vehicle handling. The resulting bluff body generates unsteady base pressures which generate unsteady forces, leading to movement of the mirror surface and potential image blurring. This study has been carried out due to the objective is to experimentally determine the fluctuating base pressure on a standard and modified mirror using a half of the full-size vehicle, fixed to the side wall of RMIT Industrial Wind Tunnel as well as in isolation (without the half car)[2].

Driver's vision can be impairs by pressure fluctuations from the wake of the side mirror view. Vibration of mirror glass due to the pressure fluctuations cause the potential of blurring image will consequently affect the safety of the vehicle and its occupants. Two major contributors to the driver's vision are the location and functionality of vehicle side view mirrors. Some previous study has been carried out in order to define the structural input (engine, road/tyre interaction etc) and also the aerodynamic input to mirror vibration.

The mirror location in the vicinity of the A-pillar vortex provide complex problem. The strength and intensity is very high even the size of the A-pillar vortex is not very large at this location. Thus, the main objective of this project was to measure the aerodynamic pressure (mean and fluctuating) on the mirror surface in order to understand the aerodynamic effects on mirror vibration. Moreover, the mirror was modified by shrouding around the external periphery (24mm, 34mm, and 44mm extensions) to determine the possibility of minimization of aerodynamic pressure fluctuations and to see if this method would attenuate the fluctuating base pressures[2].

The mean and fluctuating pressures were measured using a production rear side view mirror fitted to a Ford AU Falcon in the RMIT Industrial Wind Tunnel. The car was cut along its plane of symmetry, thus only replicated the case of zero yaw. The car segment was used as compared to a complete car was to reduce the blockage effect(~15%). Smoke and documented visualized the airflow around the mirror. The RMIT Industrial Wind tunnel is a closed return circuit wind tunnel with the maximum speed of 150 km/h. The tunnel's working section is rectangular with dimension of 3 m wide, 2 m height and 9 m long. The car segment and test section is shown in Figure 1.

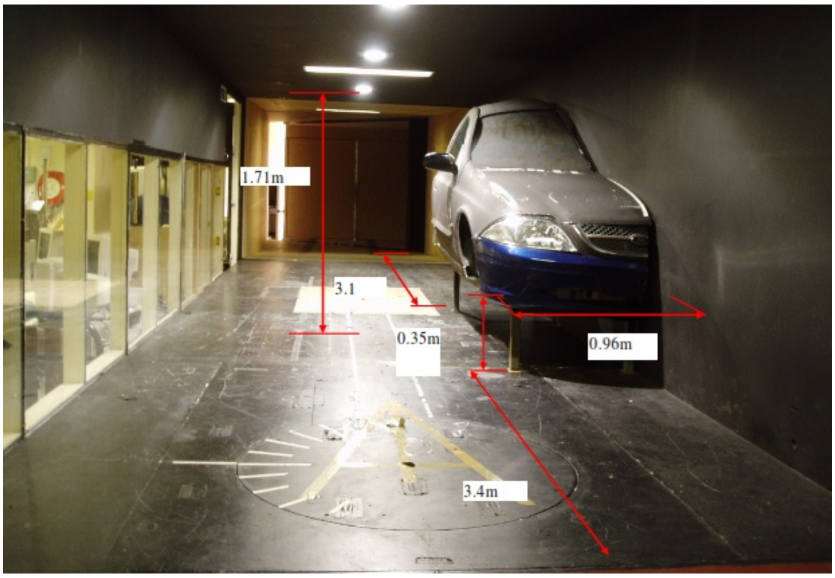

Figure 1: A quarter car with side mirror in the test section

A Dynamic Pressure Measurement System (DPMS) developed by Turbulent Flow Instrumentation (TFI) was used in order to measure the mean (time-averaged) pressures and fluctuating pressures (time-dependent) on the mirror.The DPMS is a multi-channel pressure measurement system that can accurately measure the fluctuating pressure up to 1000Hz depending upon tubing diameters and lengths[2].

In this investigation, the glass of the mirror was then replaced with an Aluminium plate (2.4 mm thickness) and the mirror case was slightly modified in order to hold the aluminium plate and allow exit of the pressure tubing. There were 51 holes made in the aluminium plate in a rectangular grid pattern. The outer diameter and inner diameter of the tubing was 2.4 mm 0.9 mm respectively. The distances between the two adjacent holes were 25 mm horizontally and 13 mm vertically. The silicon rubber tubing was connected to four pressure sensor modules, each having 15 channels. All pressure sensor modules were connected to an interface box that provided power and multiplexed the inputs to the data acquisition system. Figure 2 shows the schematic layout of the pressure measurement of the rear view mirror.

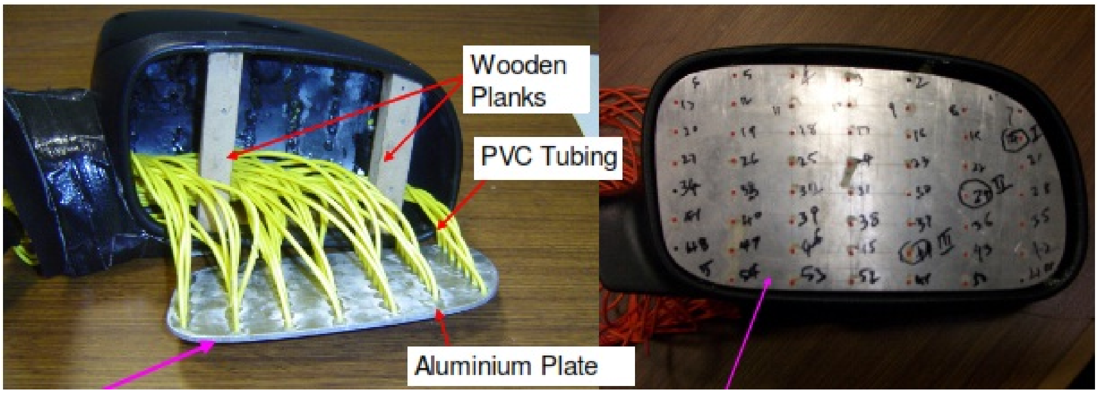

Figure 2: Schematic layout of the pressure measurement of the rear view mirror

.

Mean, rms (standard deviation), minimum and maximum pressure value of each pressure port on mirror was provided by the DPMS data acquisition software Dimensions (diameter and length) of the tubing used was entered , then the data were linearized to correct for tubing response to obtain dynamic pressure measurements. The sampling frequency of each channel was1250 Hz. It may be noted that the peak energy of fluctuating pressure on mirror surface was well below 500 Hz. The mean and fluctuating pressures were measured at a range of speeds (60 to 120 km/h with an increment of 20 km/h) at zero yaw angles. The mirror was tested as standard and then modified by adding 24mm, 34mm and 44mm shrouding on the mirror periphery.

Fluctuating pressure coefficients ($C_{p\ rms}$) of three dimensional (3D) was plotted for the standard mirror as shown in Figures 3 and 4 for the speeds of 60 km/h and 120 km/h.

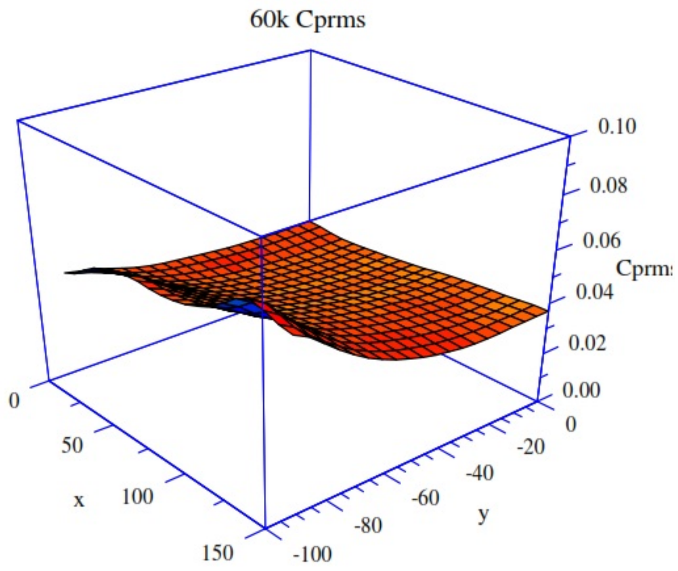

Figure 3: Fluctuating Pressure Coefficients (Cp rms) -3D - 60km/h

As the origin of the plot is located at the top left hand corner position, the x-distance is horizontal and y-distance is vertically down. The 3-D plots clearly determine that the fluctuating pressure is distributed non uniformly on mirror surface. It is most concentrated at lower central part of the mirror surface. While at low speeds, the maximum fluctuating pressure coefficients were measured at the bottom right part of the mirror surface. However, the magnitude of fluctuating pressure coefficients decreases with an increase of speeds. The maximum fluctuating pressure shifts towards the bottom central part of the mirror surface.

From the whole experiment, it should be noted that the results are for specific vehicle and mirror geometry. The experiment defined that the fluctuating aerodynamic pressure are non-uniformly distributed over the mirror surface. The central bottom sections of the mirror surface showed the highest magnitude of fluctuating pressure for the standard mirror. Moreover, the magnitude of the base pressure fluctuating can be reduced by extending the outer periphery of the typical automotive rear view mirror. This condition at the same time will change the pressure distribution across the mirror face.

The writer also discussed about further work recommended, where the phase of the pressure fluctuations need to be understood, in order to clarify the aerodynamic inputs to mirror vibration. As the yaw angle is also known to affect mirror noise and vibration, this should also be considered in future work.

2.3 Simulation of air flow around an Opel Astra vehicle with FLUENT by Andreas Kleber

In this study, investigation has been done by using OPEL ASTRA as an example to show an efficient process for the simulation of the flow field around a vehicle. Individual steps involved in this simulation process include a special emphasis on mesh generation and adaptions are discussed. Applications of aerodynamic simulation in various areas are reviewed.

CFD flow simulation used to define characteristics about flow circulation around car even models or prototypes are not available. Comprehensive information provided to designers about the entire flow field could drive to the process of aerodynamic design. Creation of the model geometry, discretization of the physical domain, and choice of a suitable numerical computing scheme are significant factors that can determine the level of success of such an effort. The simulation process for the OPEL ASTRA that is described in this article is based on practical experience, primarily in the field of mesh generation that has been developed through the use of software from Fluent Inc. from Lebanon, NH, for various automobile projects [3].

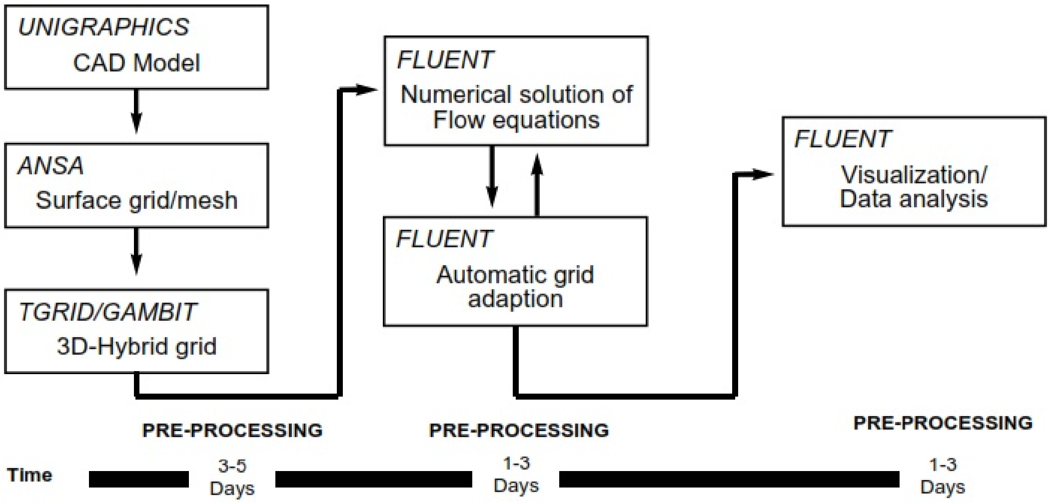

Figure 4: The simulation process

The simulation process is divided into following steps: CAD surface preparation, mesh generation, CFD solution of the fluid flow, mesh adaption, and visualization of the results. The software packages used for these steps include, UNIGRAPHICS (CAD), ANSA (CAD/mesh generation), TGRID and GAMBIT (additional mesh generation), and FLUENT (solver and post processing).

At the beginning of the process, the CAD body shell data for the ASTRA is downloaded from a common CAE database at OPEL, from which all of the vehicle parts and components can be accessed. For initial concept studies, the vehicle exterior data was made by the design department for initial concept studies. In this study, the air cooling vents and the engine space above the subframe are closed. ASTRA in this study is constructed as a full model with an asymmetric geometry (where the asymmetries are mostly confined to the underbody). This is done to enable detailed predictions of the flow field around and under the vehicle. The wind tunnel geometry around the model is a rectangular enclosure. It is of such a dimension that the adverse pressure effects between the vehicle and the wall are minimized. The three CAD components, namely the outer body of the vehicle, the underbody with tires, and the wind tunnel together constitute the CFD model (Figure 2) [3].

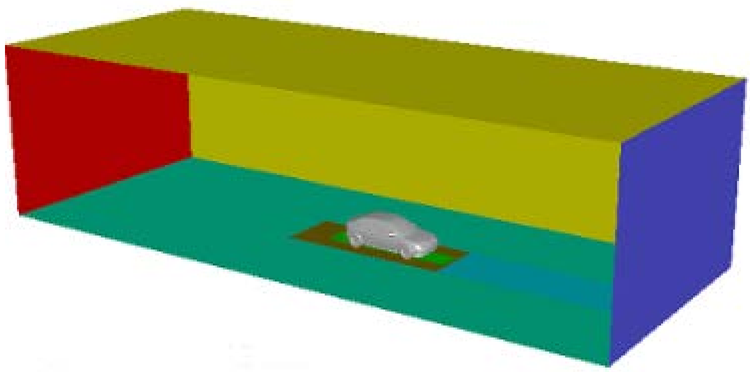

Figure 5: The CAD model[12]

Since the wind tunnel and the underbody have already been meshed, the description of the task of grid creation is limited to the styling surfaces. The window frames are covered with regular quadrilateral elements so as to close the stage of the window later using separately inserted prism blocks.

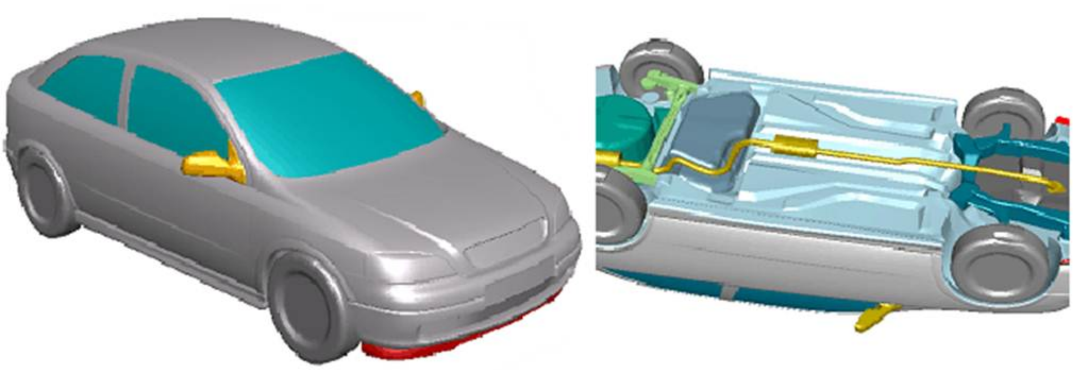

Figure 6: Shaded surface mesh of the CFD model for vehicle outer skin[12]

For numerical solution of flow equation and grid adaption,a 3D steady state incompressible solution of the Navier-Stokes equations was performed using FLUENT 5. Turbulence modeling was done with realizable k-□□ using none equilibrium wall functions and the free stream velocity was set to be 140km/h.During this project, a total of 1600 iterations on an eight processor SGI Origin 2000 were performed.