COMPUTATIONAL ANALYSIS OF TWO DIMENSIONAL FLOWS ON A CONVERTIBLE CAR ROOF

ABDULLAH B. MUHAMAD NAWI

Report submitted in partial of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUN 2013

UNIVERSITI MALAYSIA PAHANG
FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "Computational Analysis of Two Dimensional Flow On A Convertible Car Roof" is written by Abdullah b. Muhamad Nawi. I have examined the final copy of this report and in my opinion, it is fully adequate in terms of language standard, and report formatting requirement for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

DR. AHMED NURYE OUMER
Examiner   Signature

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this report and in my opinion this report is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature
Name of Supervisor : MUHAMMAD AMMAR BIN NIK MU`TASIM
Position : LECTURER
Date : 26 JUN 2013

STUDENT'S DECLARATION

I hereby declare that the work in this report is my own except for quotations and summaries which have been duly acknowledged. The report has not been accepted for any degree and is not concurrently submitted in candidate of any other degree.

Signature

Name : ABDULLAH B. MUHAMAD NAWI

ID Number : MH08081

Date : 26 JUN 2013

ACKNOWLEDGEMENT

In the name of Allah, the Most Benevolent, the Most Merciful. Alhamdulillah, all praises to Allah, the Almighty, on whom ultimately we depend for sustenance and guidance. All praises to Allah for the strengths and His blessing in completing this report.

Special appreciation goes to my supervisor, Muhammad Ammar b. Nik Mu'tasim whose guidance, careful reading and constructive comments was valuable. His timely and efficient contribution helped me shape this into its final form and I express my sincerest appreciation for his assistance in any way that I may have asked. His invaluable help of constructive comments and suggestions throughout the report have contributed to the success of this project. My sincere thanks also go to Dr. Ahmed Nurye Oumer; I owe special thanks for his consultations especially on this report. Certainly, not forgetting the UMP Faculty of Mechanical Engineering for providing the support and equipment required in order to completing this study.

Sincere thanks to all my friends for their kindness and moral support during my study. Thanks for the friendship and memories.

Last but not least, my deepest gratitude goes to my beloved parents; Mr. Muhamad Nawi b. Abdullah and Mrs. Saadiah bt. Awang and to my brothers and sisters for their endless love, prayers and encouragement. To those who indirectly contributed in this research, your kindness means a lot to me. Thank you very much.

Dedicated to my parents

ABSTRACT

This project describes the study of computational analysis of two dimensional flows on convertible car roof. The objectives of this study are to analyze and study the effect of pressure coefficient (Cp) and Drag Coefficient (Cd) on convertible car roof. The structure of model of convertible car roof was modelled using Solidworks software and analysis was performed using Ansys software. In this experiment, four model car convertible roofs were developed through Solidworks software based on the ratio of 1:20 of the original size of the vehicle. Then the four models tested in three different wind tunnel to get the best results and very similar to the real situation. The task was done by using Computational Fluid Dynamic (CFD) analysis for expected vehicle speed of 120 km/h. A pressure and drag force was obtained based on inputs from CFD analysis. This pressure and force was used to calculate the pressure coefficient and drag coefficient of the model as a whole. Results showed that the model C has the best compared with other models where the minimum and maximum pressure coefficients for model C is the lowest compared to other models of -0.9267 and 0.0488. While the drag coefficient is 0.6526 for model C. Surface design and roof arches were found to affect the results of this experiment.

ABSTRAK

Projek ini menerangkan kajian analisis pengiraan dua aliran dimensi di atas bumbung kereta tukar. Objektif kajian ini adalah untuk menganalisis dan mengkaji kesan pekali tekanan (Cp) dan Pekali seretan (Cd) di atas bumbung kereta tukar. Struktur model bumbung kereta tukar telah dimodelkan menggunakan perisian Solidworks dan analisis telah menunjukkan prestasi yang menggunakan perisian Ansys. Dalam eksperimen ini, empat model bumbung kereta tukar telah dibangunkan melalui perisian Solidworks berdasarkan nisbah 01:20 daripada saiz asal kenderaan. Kemudian empat model diuji dalam tiga terowong angin yang berbeza untuk mendapatkan hasil yang terbaik dan hampir sama dengan keadaan sebenar. Tugas itu dilakukan dengan melakukan Bendalir Komputeran Dynamic (CFD) analisis untuk kelajuan kenderaan dijangka 120 km / h. Satu daya tekanan dan heret telah diperolehi berdasarkan input daripada analisis CFD. Ini tekanan dan daya telah digunakan untuk mengira pekali tekanan dan pekali seretan bagi model secara keseluruhan. Keputusan menunjukkan bahawa model C mempunyai nilai terbaik berbanding dengan model lain di mana pekali tekanan minimum dan maksimum bagi model C adalah yang paling rendah berbanding dengan model lain iaitu -0.9267 dan 0,0488. Manakala pekali seretan adalah 0,6526 untuk model C. Reka bentuk permukaan dan gerbang bumbung didapati mempengaruhi keputusan eksperimen ini.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

ρ	Density
V	Velocity
V_o	Wind velocity (reversed direction)
A	Area
P	Pressure
P_{atm}	Atmosphere pressure
C_D	drag coefficient
C_P	pressure coefficient
F_D	Drag force
μ	Viscosity
U	Dynamic viscosity
Re	Reynolds Number

LIST OF ABBREVIATIONS

CAD	Computational Aided Design
CFD	Computational Fluid Dynamic
km/h	kilometre per hour
m/s	mile per second
mm	millimetre
m	meter
kPa	kilo Pascal
Pa	Pascal

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

The modern day designs of vehicles, especially in the racing industry involve a great deal of air flow study. This study shows that drag force adversely affects the forward motion of the car and that there is a difference in the pressure between the air flowing above and below the car (Chainani, 2008)

The study involves the flow of air against the vehicle roof is one most of the important aspect designing vehicle. However, research on airflow convertible car roofs rarely examined in Malaysia than in European countries and should be implemented. A convertible is a type of automobile in which the roof can retract and fold away having windows which wind-down inside the doors, converting it from an enclosed to an open-air vehicle. Many different automobile body styles are manufactured and marketed in convertible form.

Open cars may have a folding roof but only detachable side screens which snap-on; they do not convert into a fully enclosed car with proper weather-protection. The wind-up windows make the distinction between an open car and an all-weather car now known as a convertible. According to Knight (2009), the designer of convertible vehicles needs to consider the acoustical properties, quality of the interior environment, fatigue life and the aesthetic integrity of the flexible roof. Each of these concerns is affected, to a greater or lesser extent, by the interaction of the flexible roof material and its supporting structure with the aerodynamic loading on the material. For many convertibles, the roof quickly settles into a deformed shape for which the roof's internal forces are in equilibrium with the aerodynamic loading.

1.2 PROBLEM STATEMENT

The problem regarding this project is to analyze the two dimensional flow of convertible car roof using Computational Fluid dynamic. If we can see, there are various forms of roof design made from different car manufacturers but in Malaysia, there are no Malaysian manufacturer's that produces convertible car roof. This is due to climatic factors as the weather in Malaysia as opposed to weather the country that issued convertible car. Materials and design should be reviewed to allow convertible car model produced in Malaysia. Malaysia only imports so far this model and rarely produces models in Malaysia, but only for export to countries that use this vehicle as European countries. In addition, the design factor can also be affecting the total vehicle drag. Drag will cause many problems on the performance of vehicles like instability, noise and vibration, also fuel consumption. In this project, the use of CFD software is very important because it is more economical compared with experiment.

1.3 OBJECTIVE

The objectives of the project are as follows:

i. To analyze and study the effect of pressure coefficient (C_p) and Drag Coefficient (C_d) on convertible car roof.

1.4 SCOPES OF STUDY

The scopes of the project are as follows:

i. To study the effect towards convertible car roof using CFD method (velocity 120km/h) under steady state condition.
ii. Validate with experiment from other researchers

iii. Study the pressure coefficient and drag coefficient on 4 different designs (international market) of convertible car roof.

iv. Boundary condition will be done in a wind tunnel 1:20 (experiment set up)

CHAPTER 2

LITERATURE REVIEW

2.1 COMPUTATIONAL FLUID DYNAMICS

Computational Fluid dynamic (CFD) is a science that with help of digital computers produces quantitative prediction of fluid flow phenomena based on those conservation laws (conservation of mass, momentum and energy) governing fluid motion. These predictions normally occur under those conditions defined in term of flow geometry, the physical properties of a fluid, and the boundary and initial condition of a flow field. The prediction generally concern sets of values of the flow variables, for example, velocity, pressure, temperature at selected location in the domain and for selected times. It may also evaluate the overall behavior of the flow, such as the flow rate or the hydrodynamic force acting on an object in the flow.

2.1.1 General CFD

In fluid dynamics there are three governing equations that describe the behavior of the flow; these are the continuity, momentum and the energy equations. They are derived from basic physics laws such as the conservation of energy, mass and momentum. These equations becomes rather complicated and can't be solved analytic so numerical simulations are required. In a CFD simulations the differential equations are discretizised into large systems of algebraic equations in order to numerical solve them. Since vehicles travels at relatively low speed, $Ma<0.3$, and constant temperature the flow can be assumed incompressible and isothermal, the energy equation can then be neglected (Wolf-Heinrich, 1998)

2.1.2 Governing Equation

2.1.2.1 RANS

The non-linear partial differential equations are not analytically solvable. In order to solve this equation and analyze the flow the simplest approach is the Reynolds decomposition, also called Reynolds Average Navier Stokes (RANS). In the RANS approach the instantaneous velocity and pressure is split into two parts, an average part and a fluctuating part, Eq. (1) and Eq. (2).

$$\bar{u} = \frac{1}{T}\int_0^T u\,dt \tag{2.1}$$

$$\begin{aligned} p &= \bar{p} + p' \\ u &= \bar{u} + u' \\ v &= \bar{v} + v' \\ w &= \bar{w} + w' \end{aligned} \tag{2.2}$$

Inserting Reynolds decomposition into Navier-Stokes equation (x-direction) and in the continuity equation will results in new fuctuating terms

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

$$\rho g_x - \frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x}\left(\mu\frac{\partial \bar{u}}{\partial x} - \rho\overline{u'^2}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial \bar{u}}{\partial y} - \rho\overline{u'v'}\right) + \frac{\partial}{\partial z}\left(\frac{\partial \bar{u}}{\partial z} - \rho\overline{u'w'}\right) = \rho\frac{d\bar{u}}{dt} \tag{2.4}$$

Eq. (2.4) now consists of new unknown terms like $\rho\overline{u'v'}$ also called for Reynolds stresses. Since the number of unknowns is greater than the number of equations a so called closure problem is generated, the extra stress terms has to be modeled to get a closed equation system. This is done by using turbulence models (Wolf-Heinrich, 1998).

2.1.2.2 Turbulence Flow and Turbulence Modeling

The largest difficulty with CFD simulations is to calculate the turbulent flow. A turbulent flow is irregular and varies randomly in time and space. By using so called turbulence models the flow field can be calculated with less computer capacity. Such a model will modify the equations and only consider the average effects of the turbulence. The flow will then be divided into an average term and a fluctuation term; this can be done using Reynolds Average Nervier Stokes (RANS). A turbulence model can never give an exact solution, but a better choice will give a more accurate solution. The choice of model is amatter of computer capacity and required level of accuracy.

To account for the turbulence effect on the flow field, Reynolds time averaging technique was employed on the equation of Navier-Stokes to yield the equation of Reynolds Averaged Navier-Stokes (RANS) which can be mathematically expressed as equation 2.5

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(v \frac{\partial \bar{u}_i}{\partial \bar{u}_j} - \bar{\tau}_{ij}\right) \quad (2.5)$$

Where:

i = 1, 2, 3,;

j = 1, 2, 3,;

The bar on top of the variables implies that the variables are the time-averaged quantities. In Eq. (2.5), ij is the shear-stress tensor. Eq. (2.5) is impossible to resolve due to the appearance of the Reynolds stress. To bring closure to the above equation, the

Reynolds stress term is modeled through the means of k–e turbulence modeling technique (Tsai et al., 2007).

The k–e turbulence model is usually applied to simulate air flow fields in mechanical ventilation system and in other modern engineering applications. In the early stage of research, turbulence model was only applied for incompressible high Reynolds number flows but it was later experimentally proven that air flows next to solid walls were associated with low Reynolds numbers. Therefore, the development and testing of low Reynolds number turbulence models have been a topic for extensive research (Tsai et al., 2007). A remedy to this approach is the introduction of a wall function into the modeling so that the airflow within the entire computational domain can be calculated using equation 2.6 and equation 2.7 at the same time even if the Reynolds number near the walls is low while that far away from the wall is high.

$$\frac{\partial(\rho k)}{\partial t} + u_i \frac{\partial(\rho k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) + G - \rho\varepsilon \tag{2.6}$$

Where:

i = 1, 2, 3,

$$\frac{\partial(\rho\varepsilon)}{\partial t} + u_i \frac{\partial(\rho\varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}G - C_{2\varepsilon}^{*}\rho\frac{\varepsilon^2}{k} \tag{2.7}$$

Where:

i = 1, 2, 3,

The turbulence model used in this work is the RNG k–e turbulence model because of its good prediction of complex flows (Tsai et al., 2007). The complete formulation of the RNG k–e turbulence model is given in Einstein summation convention as follows

$$G = 2\mu_t S_{ij} S_{ij} \tag{2.8}$$

$$C_{2\varepsilon}^{*} = C_{2\varepsilon} + C_{2\varepsilon}' \tag{2.9}$$

$$C_{2\varepsilon}' = \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0)}{1 + \beta\eta^3} \tag{2.10}$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{2.11}$$

$$\eta = S\frac{k}{\varepsilon} \tag{2.12}$$

$$S = \sqrt{2S_{ij}S_{ij}} \tag{2.13}$$

Where S_{ij} is the shearing-rate tensor and g_i is the body force in the x_i direction (Tsai et al., 2007)

2.2 AERODYNAMICS OF VEHICLE

"Aerodynamics" is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics is also a subfield gas dynamics, with much theory shared with fluid dynamics. 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. The use of aerodynamics through

mathematical analysis, empirical approximation and wind tunnel experimentation form the scientific basis.

Aerodynamics and its analysis are basically divided into two major sub-categories, namely the external and internal aerodynamics. External aerodynamics is the study of flow around solid objects of various shapes. Evaluating the lift and drag on an airplane, the shock waves that form in front of the nose of a rocket, or the flow of air over a wind turbine blade are examples of external aerodynamics. On the other hand, internal aerodynamics is the study of flow through passages in solid objects. For instance, internal aerodynamics encompasses the study of the airflow through a jet engine or through an air conditioning pipe (Wolf-Heinrich, 1998).

Apparently, this project concentrates more on the external category of the aerodynamics related to convertible car.

2.2.1 External Flow of Automotive Vehicle

The Fig. 2.1 shows the streamline of an external flow around a stationary vehicle. When the vehicle is moving at an undistributed velocity, the viscous effects in the fluid are restricted to a thin layer called boundary layer. Outside the boundary layer is the in viscid flow. . This fluid flow imposes pressure force on the boundary layer. When the air reaches the rear part of the vehicle, the fluid gets detached. Within the boundary layer, the movement of the fluid is totally governed by the viscous effects of the fluid.

Figure 2.1: Streamline of external flows around a stationary vehicle

Source: http://www.carbodydesign.com/archive/2009/05/14-volkswagen-polo/VW-New-Polo-Wind-Tunnel-Testing-1-lg.jpg (17 March 2013)

The boundary does not exist for the Reynolds Number which is lower than 10^4. The Reynolds number is dependent on the characteristic length of the vehicle, the kinematic viscosity and the speed of the vehicle. Apparently, the fluid moving around the vehicle is dependent on the shape of the vehicle and the Reynolds number. There is another important phenomenon which affects the flow of the car and the performance of the vehicle. This phenomenon is commonly known as 'Wake' of the vehicle. When the air moving over the vehicle is separated at the rear end, it leaves a large low pressure turbulent region behind the vehicle known as the wake. This wake contributes to the formation of pressure drag, which is eventually reduces the vehicle performance (Wolf-Heinrich, 1998).

2.2.2 Drag Coefficient

The drag coefficient is a dimensionless quantity that describes a vehicles aerodynamic resistance and is a useful tool when comparing different vehicle shapes regardless of size and speed. The drag coefficient can be expressed as in Eq. (2.14). The drag coefficient can be divided into two components, a friction and a form component.

$$C_d = \frac{F_D}{\frac{1}{2}\rho U^2 A}$$

(2.14)

Where ρ = air density, U = freestream velocity, F_D= drag force and A= frontal area

When air streams around a body there will be a pressure difference between the upper and lower part, if no separation occurs in the flow field the air on the upper surface will travel a longer path to reach the end of the vehicle. This difference in travel length will create a difference in the speed of the fluid; longer way to travel will give a higher speed, and lower pressure.

2.2.3 Pressure Coefficient

A useful parameter to compare incompressible flows is the pressure coefficient (C_p) see Eq. (13). The pressure coefficient (C_p) describes how the pressure on a surface deviate from the freestream pressure. Every single point in the flow field or on the surface has a unique C_p . To find the stagnation pressure on a surface C_p should be equal to one. If C_p instead is equal to zero this indicates a low pressure region where the risk for separation is high. In Eq. (2.16) the pressure coefficient is expressed in terms of pressure.

$$C_P = \frac{P - P_\infty}{\frac{1}{2}\rho V^2} = \frac{P - P_\infty}{dynamic\, pressure}$$

(2.15)

Where P_∞ the static pressure in freestream and p is the pressure in the specific point.

2.2.4 Aerodynamic Drag For Car Roof

The drag coefficient (C_D) can be reduced by arching the roof in the longitudinal direction; however, if the curvature is too great, C_D again can increase, as can be seen in fig 2.2. The favorable effect of arching depend on maintaining sufficiently large bend radii at the junction between windshield and roof and between roof and rear window, so that the negative pressure peaks at these locations are not large and the corresponding pressure gradient are reasonably small.

However, the design of the roof arch must ensure that the frontal area of a car remain constant; if not, the absolute drag (C_D.A) can increase despite a reduction in drag coefficient (C_D), as shown in the upper graph of fig 2.2. Since a driver's upward viewing angles most not be reduced, the windshield and rear window must be incorporated into the longitudinal arching; as a result, windows get spherical and thus more expensive.

The airflow on the roof run parallel to the direction to travel; the is no flow around the sides of the roof. A drip molding along the roof rail therefore does not disturb the flow and does not increase either drag or wind noise.

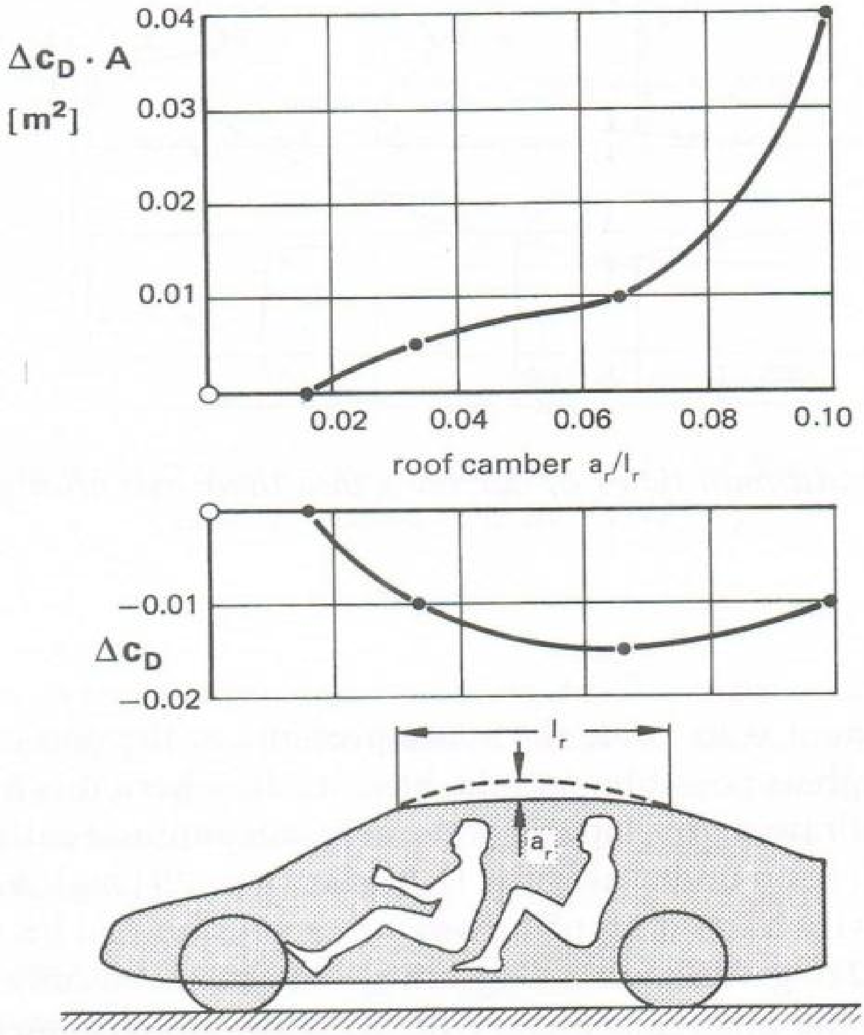
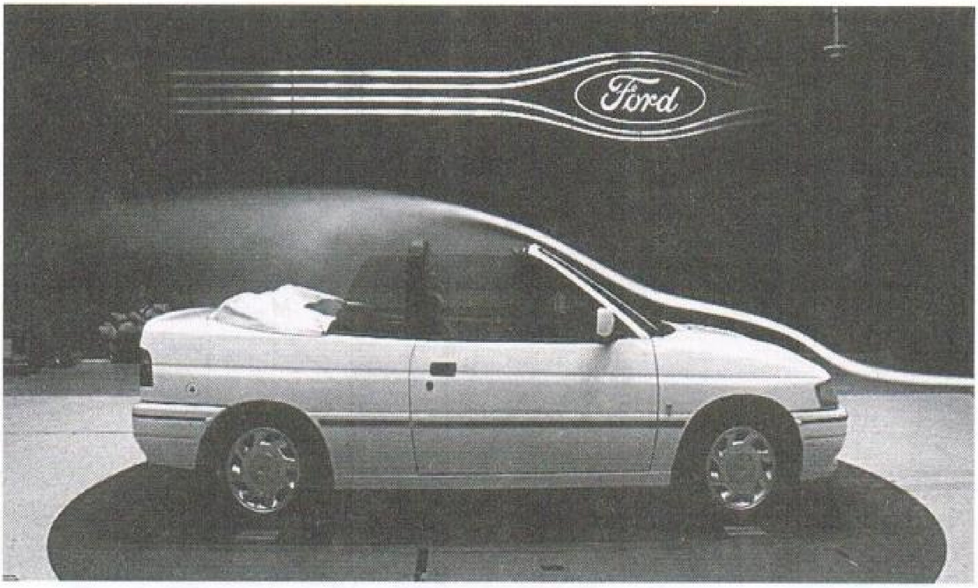


Figure.2.2: Effect of roof camber on drag coefficient (C_D) and absolute drag.

Source: Wolf-Heinrich (1998)

There is major increase in the drag of convertible when the car is open. On the other hand, the drag with closed roof is only slightly greater than that of the associated sedan version. Some data are compiling in fig 2.3. The fact that drag is increased by opening the roof is only of secondary importance because the car is seldom driven in this condition, and more importantly, is then driven slower; the higher fuel consumption in this case is therefore irrelevant.



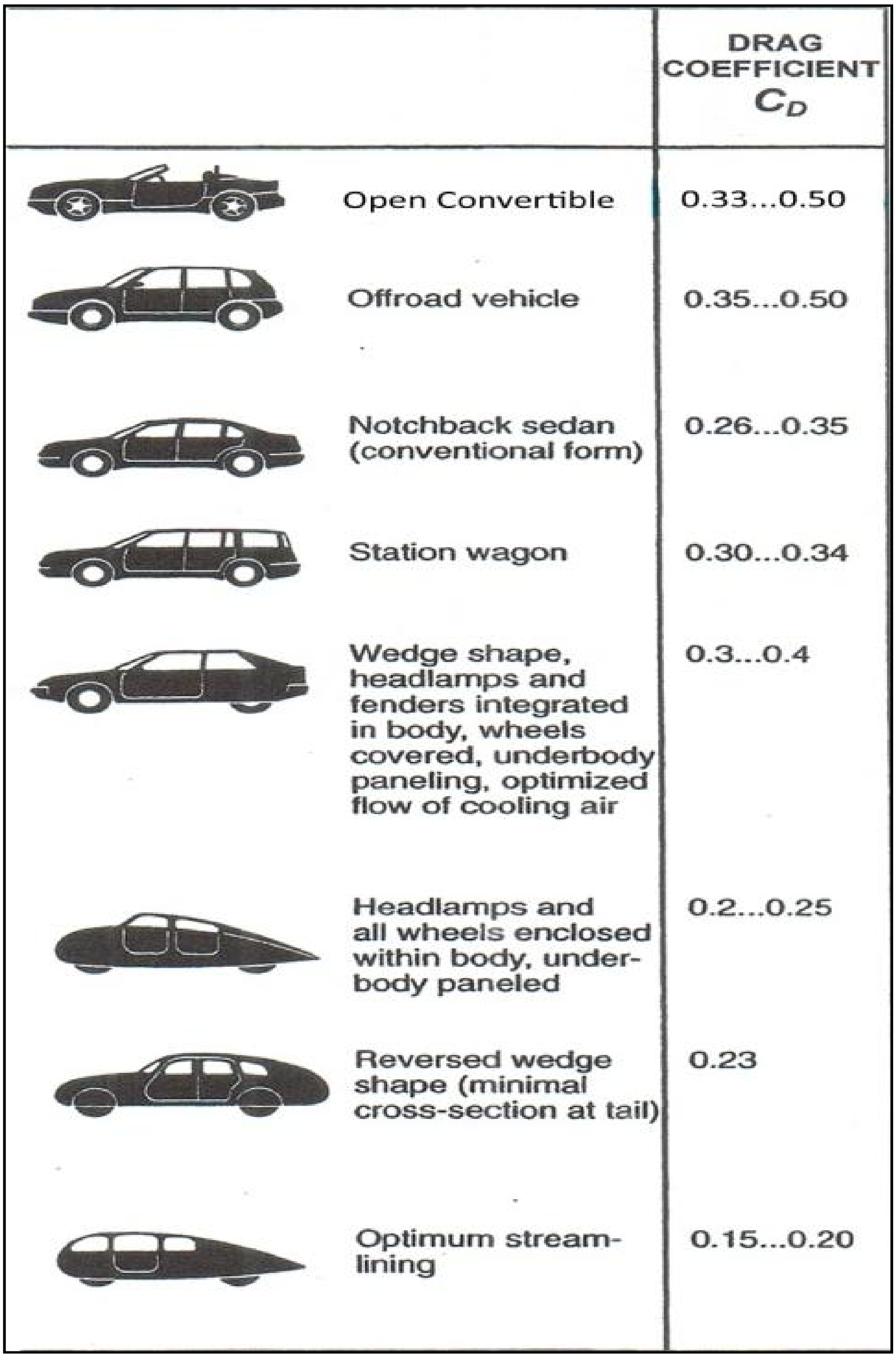
c_D		VW-Beetle	Opel-Kadett 87	Ford-Escort 91	Mercedes-B. W 129
Limousine		0,49	0,32	0,32 [2)]	0,32 [1)]
convertible	closed	0,50	0,34	0,36	0,34
	open	0,68	0,38	0,42	0,41
	+ side windows open			0,43	0,43

1) hardtop

2) notchback

Figure 2.3: Drag coefficients of convertible.

Source: Wolf-Heinrich (1998)



	DRAG COEFFICIENT C_D
Open Convertible	0.33...0.50
Offroad vehicle	0.35...0.50
Notchback sedan (conventional form)	0.26...0.35
Station wagon	0.30...0.34
Wedge shape, headlamps and fenders integrated in body, wheels covered, underbody paneling, optimized flow of cooling air	0.3...0.4
Headlamps and all wheels enclosed within body, under-body paneled	0.2...0.25
Reversed wedge shape (minimal cross-section at tail)	0.23
Optimum stream-lining	0.15...0.20

Figure 2.4: Aerodynamic resistance coefficient for passenger cars of various shapes.

Sources: Wong (2008)

CHAPTER 3

METHODOLOGY

3.1. INTRODUCTION

For methodology in project of computational analysis of two dimensional (2D) flows on a convertible car roof, we noticed that there are several stages in order to complete the research criteria. The stage show the step by step procedure and also including a flow chart as a guide line. The stages are:

a) Stage 1: Prepare 2D Model of convertible car roof
b) Stage 2: Validation
c) Stage 3: Analysis using CFD simulation
d) Stage 4: Compare of simulation result with experiment

All of this stage should be followed to ensure that simulation analysis will be successfully without any error would occur.

3.2 FLOW CHART OF METHODOLOGY

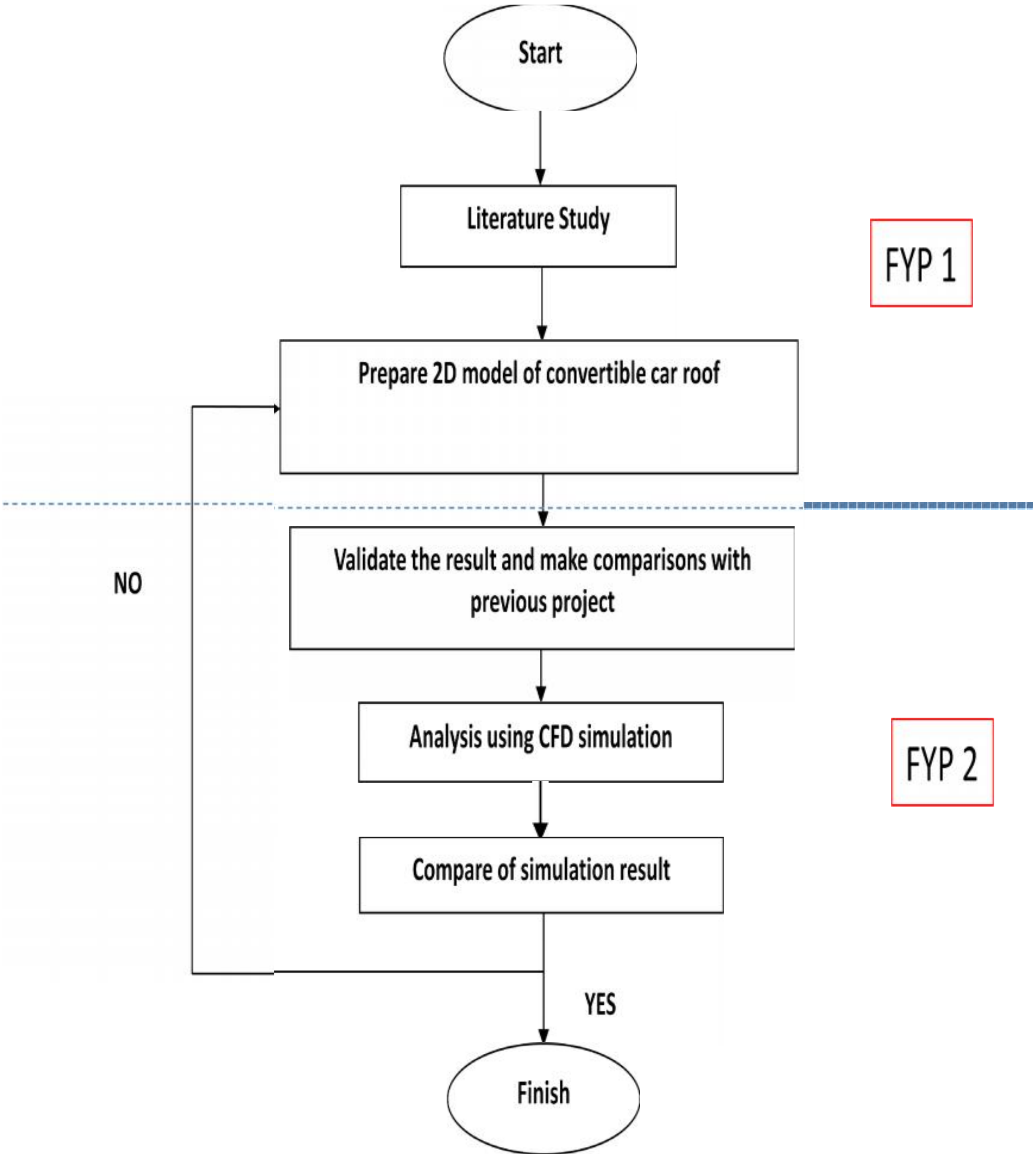


Figure 3.1: Flowchart of methodology

3.3 PREPARE 2D MODEL OF CONVERTIBLE CAR ROOF

3.3.1 Overview

CAE tools will be use for modeling and analyzing the models. First, the models will build up in CAD (Computer-Aided Design) software. Mostly people use CAD software to design and build up the model. For this project, SolidWorks will be use to build up the model and the model will be design according the actual dimension to make sure it can produce an approximately accurate. The design of the convertible car roof not just accurate in dimension, but it also must fix with the base line model that will be use. This precaution step can avoid any errors during analysis and also to make the model of convertible car roof is easily mate with the base line model. The base line model that will be use in this project also must build up according the actual design.

3.3.2 Solidworks Modeling Stage

For this project, Solidworks 2012 was used to create the geometry and the model of convertible car roof. After starting the Solidworks 2012 software, at the menu bar, choose file and select new. A window will appear, and then click OK for the new part. Select top plane at the tree manager, and choose sketch. To get a better view during drawing, draw a centerline starting from the centre of the X-Z axis which is shown in red in colour. Begin sketching. Sketching can be done by using various sketching tools such as the line for straight lines, spline for flexible curvature shapes, circle and arc to construct round shapes geometry, polygon if the geometry has many edges and many more tools that can be found on the sketching tool bar. Note that the drawing should be 2-D sketching mode and model is based on car design in relation to the convertible car. After the drawing process, the geometry is ready to be extruded. Choose feature at the tool bar which is beside the sketch tool bar, select extrude boss/base and click the mouse pointer. The model will appear out. The model is created. Save the file in iges (*.igs) format so that the drawing can be opened in Computational Fluid Dynamics (CFD) software to be analyzed. Here are four sketches of the proposed model.

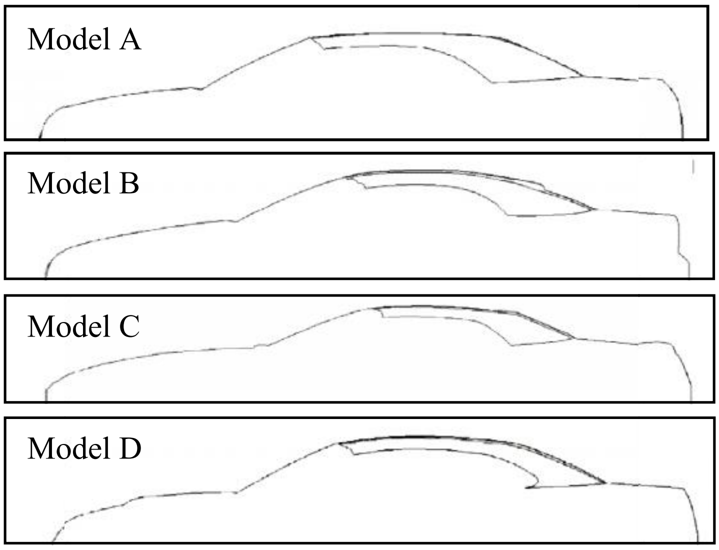


Figure 3.2: Four sketches of 2D model convertible car roof

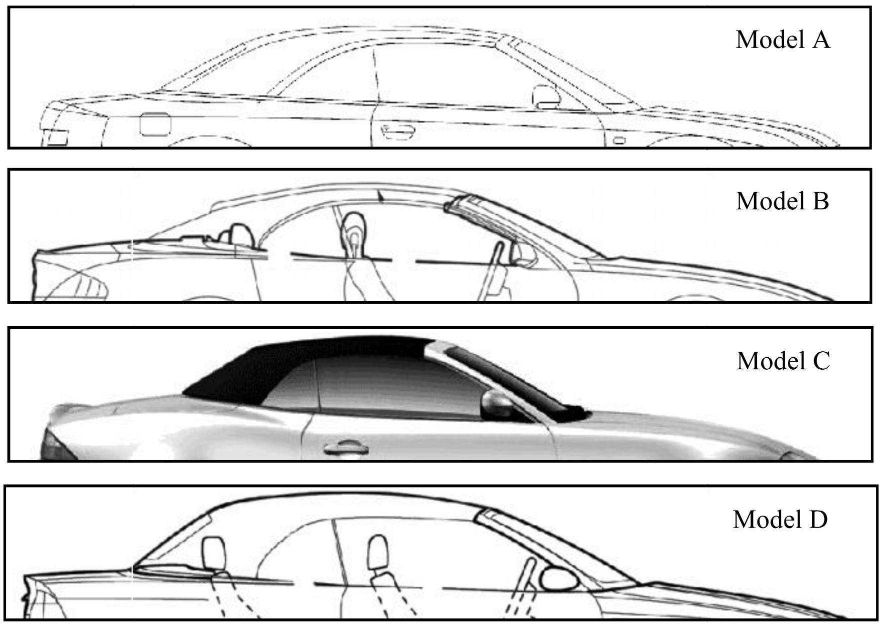


Figure 3.3: Four type of 2D model convertible car roof before sketches

In addition, some data will be collected on a wind tunnel. Four type of wind tunnel has been selected for this project. The goal is to determine which decision is more appropriate for this experiment. This wind tunnel model will be built using Solidworks software and stored in the iges (*.igs) format. This model will be combined with 3D convertible car roof models and used during the analysis using ANSYS software. Boundary condition will be done in a wind tunnel 1:20 (experiment set-up). Here is the data for the model and the wind tunnel.

Table 3.1: Data for Wind Tunnel

No.	Name of Wind Tunnel	High (mm)	Long (mm)
1	POSTECH WT	1500	4300
2	VKI-L1 LOW SPEED WT	3000	5000
3	FLOW SCIENCE WT	2230	5500

In Table 3.2, shows the details of the size of the four actual models and sizes in 1:16 ratio used in the experiment.

Table 3.2: Data for Model of convertible car

Model	Normal Size (mm)			Size in Wind Tunnel (mm)		
	Wide	High	Long	Wide	High	Long
A	2040	1427	4701	102	71.35	235.05
B	2080	1396	4894	104	69.8	244.7
C	1976	1392	4615	98.8	69.6	230.75
D	2028	1329	4794	101.4	66.45	239.7

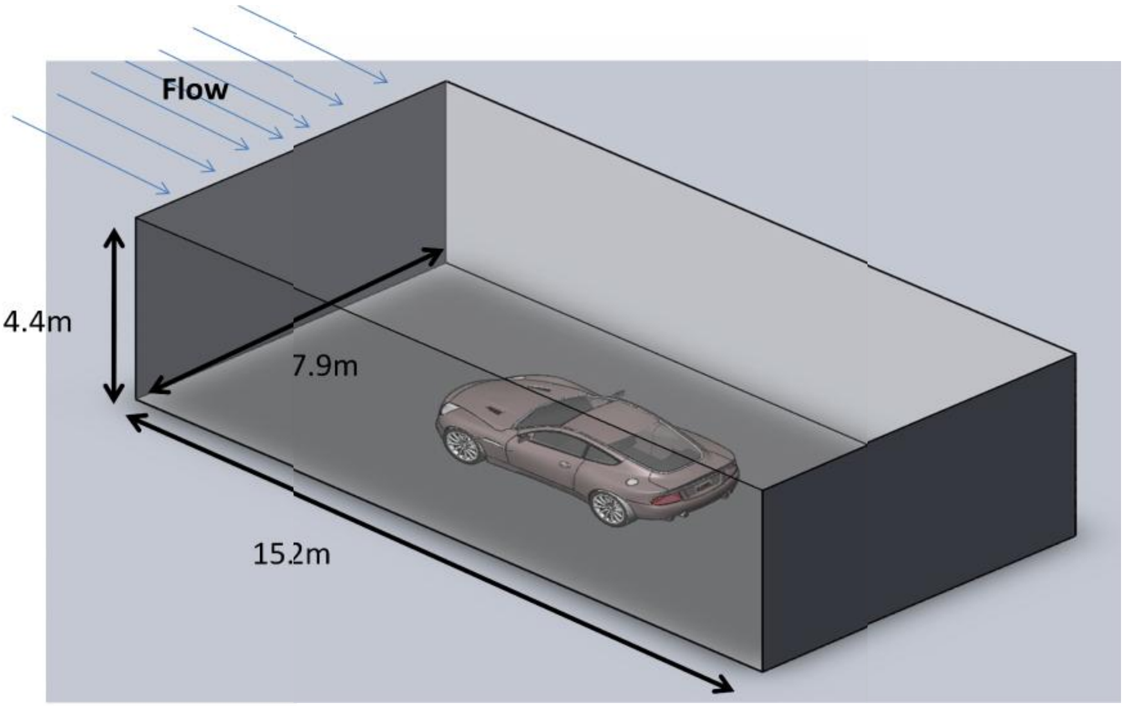


Figure 3.4: Example of Wind tunnel model

3.4 ANALYSIS USING CFD SIMULATION

3.4.1 Overview

For this project, SolidWorks Flow Simulation and ANSYS will be use to analyze the car model with its attachment, which is the convertible car roof. SolidWorks Flow Simulation and ANSYS is the fluid flow analysis tool for designers fully embedded. With this software, it can analyze the solid model directly. The model that has been built up in SolidWorks then will be export into SolidWorks Flow Simulation and ANSYS to analyze the model. Through this software, it can analyze parts, assemblies, subassemblies, and multibodies. Detail steps for use this software is include in its tutorial. The design will analyze, the data will interpret, the result will produce and analysis will summarize and present in form of table, graph, chart or etc.

During the analysis, some errors may be come. Some precaution steps must be notice before analyzing the model, such as the model is must properly build in SolidWorks. Besides, the result that will be got also is not follow as need. Let say that the result get from analysis is differ from the aspect result, known that the value of CD is between 0.3 – 0.5 for passenger car, but result shown the CD from analysis is larger than range. So, refinement is needed by modify the model and analyze again in SolidWorks Flow Simulation and ANSYS.

3.4.2 ANSYS Simulation and Analysis Step

ANSYS is one of the CFD software that can be used for this step. After starting the Workbench application, Unsaved Project window will appear. Under the Analysis Systems at Toolbox, click the Fluid Flow (FLUENT) and drag it into the green colour box in the Project Schematic. A small box will appear, and consist of CFD steps; geometry, mesh, setup, solution and result. Each of this steps must be done is sequence to produce a perfect simulation with accurate readings. They must be double clicked to open their particular window. The name of the project at the bottom of the box can be renamed with suitable project name.

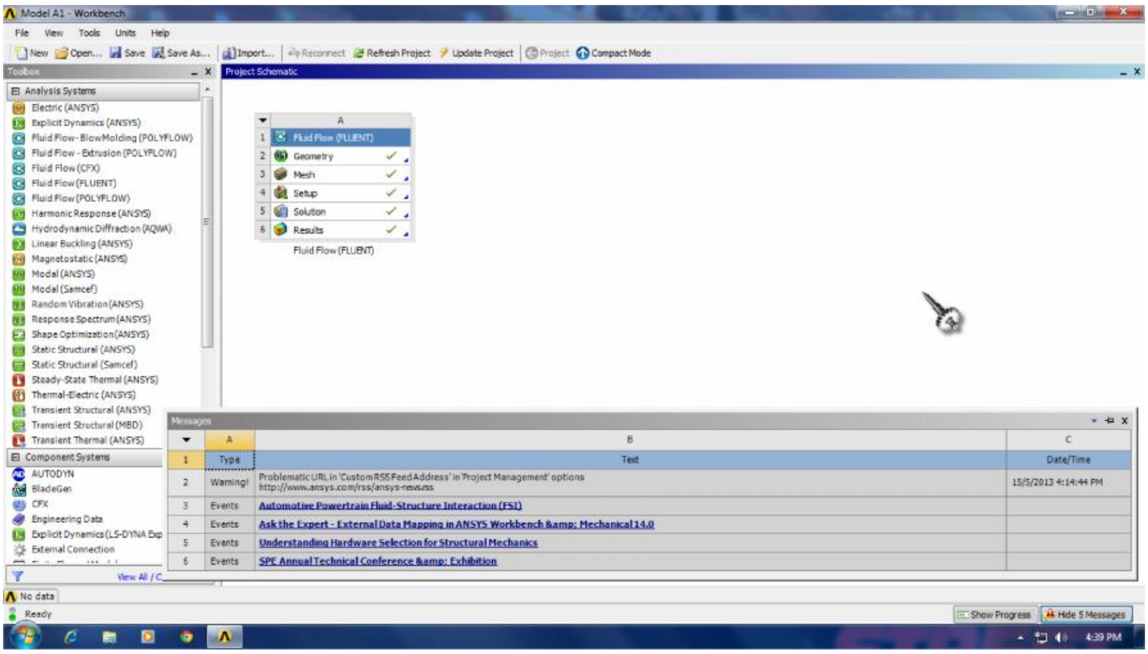


Figure 3.5: Workbench main window (Unsaved Project)

3.4.2.1 Geometry

This is the very first step in CFD analysis. After double click the geometry box, the Design Modeler window will appear. The unit is selected to be in meter. Then, go to file and select Import CAD file. The IGES (.igs) file which was converted with Solidworks 2012 software is browsed and imported into the Design Modeler. Moreover, the Design Modeler has same function with the Solidworks. After importing the design, the geometry is generated by clicking on Generate in the toolbar at the top. After generating without any error, the Design Modeler window is closed.

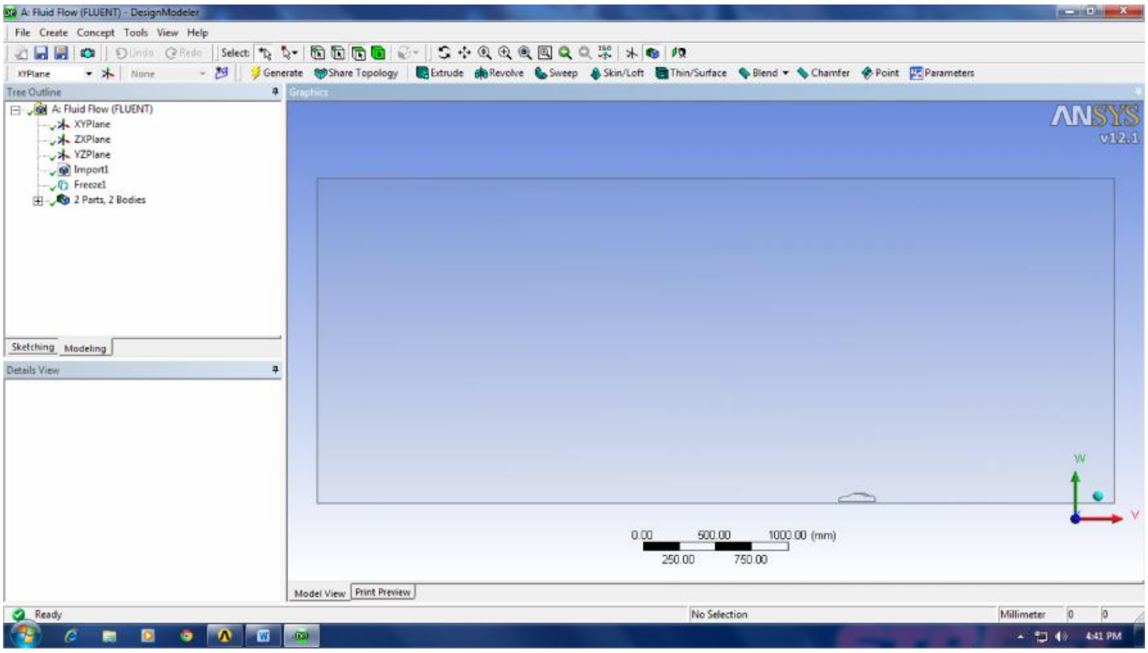


Figure 3.6: Design Moduler

3.4.2.2 Mesh

A key step of the CFD for numerical computation is mesh generation. One is given a domain and must partition it into simple “elements” meeting in well-defined ways. There should be few elements, but some portions of the domain may need small elements so that the computation is more accurate there. The mesh box double clicked and Meshing window appear. The mesh at the outline is selected at the mesh is generated. Then, the relevance center at sizing of the mesh is changed from coarse to fine because the smaller the mesh, the more accurate the calculation and results. Finally, the mesh is updated and the Meshing window is closed.

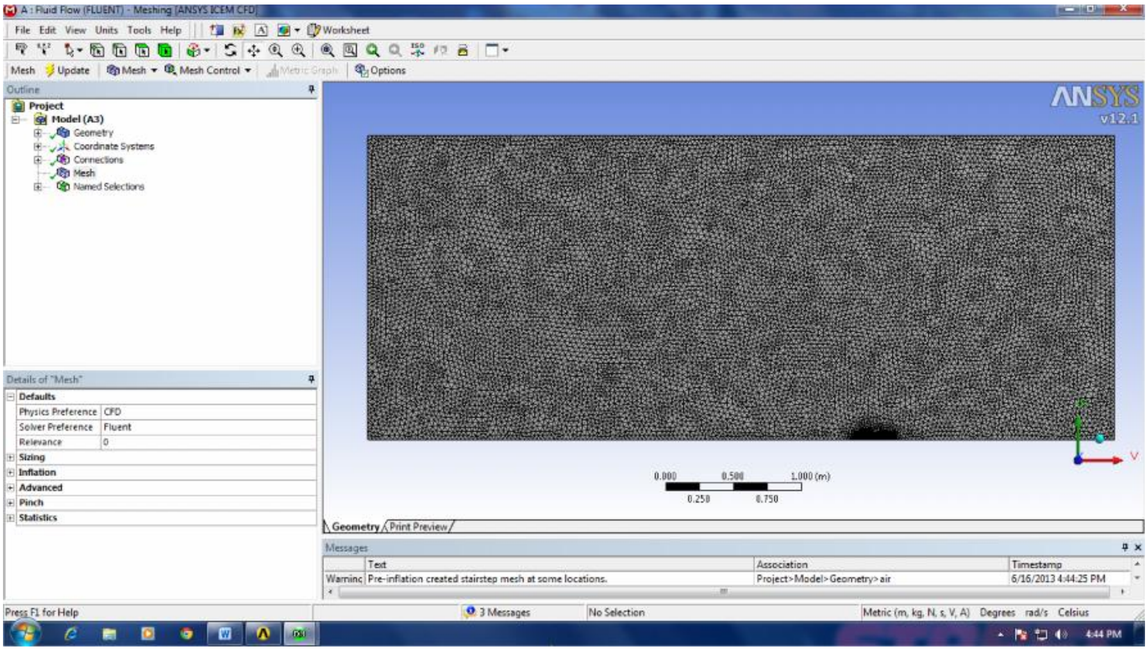


Figure 3.7: Meshing

3.4.2.3 Setup

In this step, the Setup box at Workbench is double-clicked and setup FLUENT window appear. The setup step is used to create the boundary condition of the convertible car roof. This experiment mesh is divided into 2 domains: convertible car roof domain and wind tunnel domain. The wind tunnel is set up as a fluid domain with air as the flowing fluid. The inlet face is chosen and the velocity is set up at 33.33 meter per second by clicking the boundary icon at the top toolbar. The outlet face is chosen and the pressure in set up to be 101.376 Pascal.

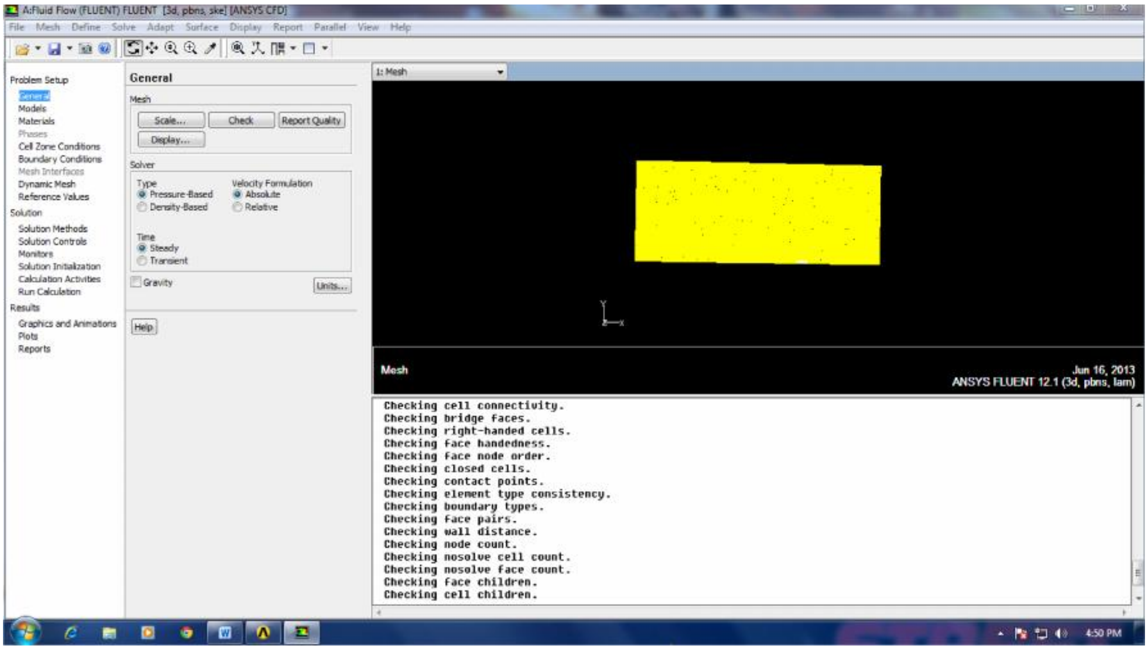


Figure 3.8: Setup Window for FLUENT

After finish the setup all condition, click Run Calculation at solution tree to calculate the solution of the analysis.

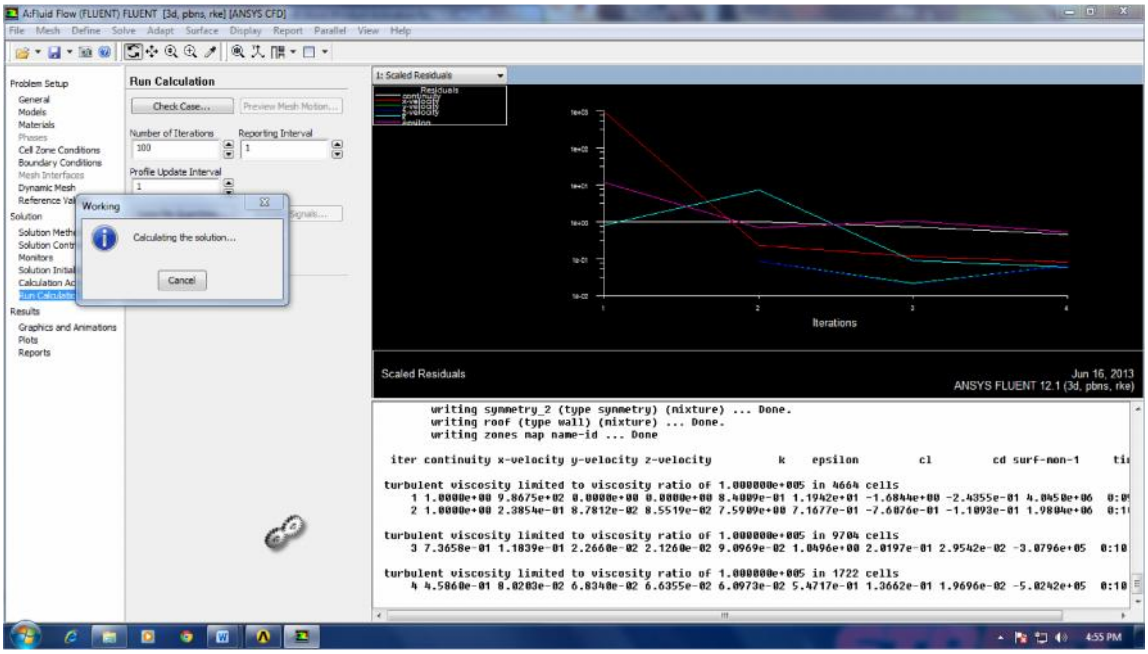


Figure 3.9: Define Run step

3.4.2.4 Results

The last box in the Workbench ANSYS FLUENT is Results. The Results box is double-clicked and FLUENT CFD-Post window appeared. This step is used to show the simulation created through the boundary conditions and calculations. The results can be performed through streamlines, contours and animations to show the change of parameters.

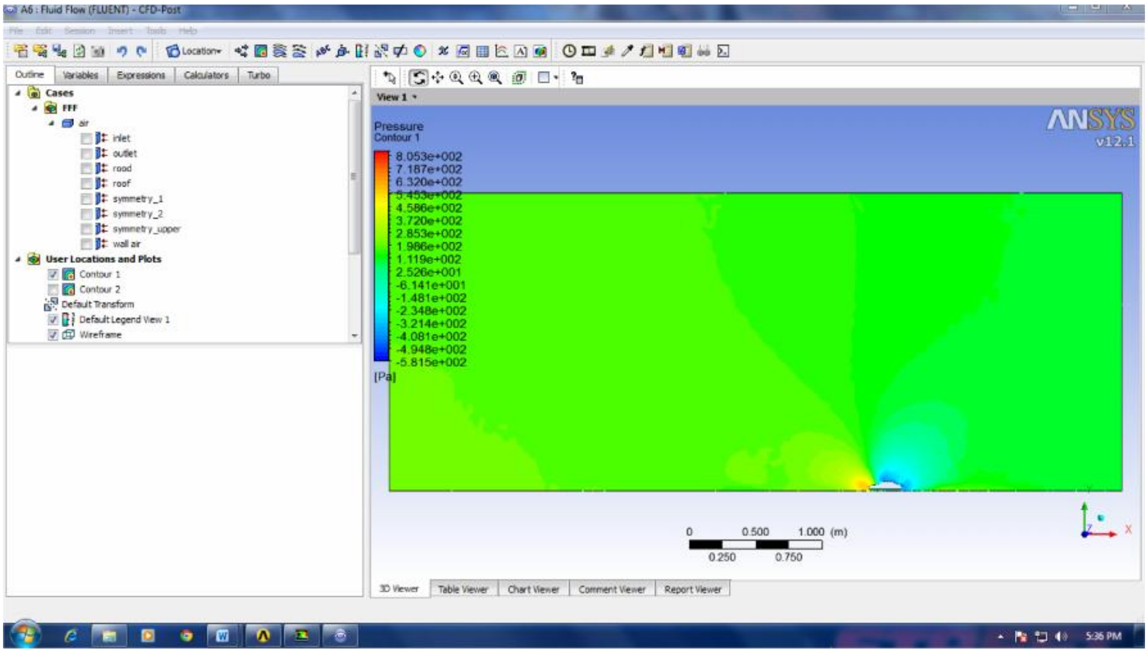


Figure 3.10: Contour of convertible car roof that show the change of pressure

3.5 ANALYSIS OF SIMULATION RESULT

After the use of CFD analysis carried out, all the results will be compared between the two software. Validate the result and make comparisons with previous project. Good engineering practice suggests that prior to using an analysis technique on a new configuration, one should benchmark (validate) the technique against a known (respected) test case similar to the new configuration. If no suitable test case exists, then cross referencing with another analysis technique, such as a wind tunnel, is essential. The benchmark test process is the process of numerical analysis performed on a case which is replica of the real time testing or previous results of numerical simulations.

While performing the benchmark testing, the results of the test will be further compared with other available results.

For CFD, the benchmarking process should result in guidelines for a specific class of problems. The guidelines would describe the preferred boundary conditions, turbulence model and meshing strategy (clustering and growth rate) required to achieve a desired level of confidence and accuracy in the results.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

The main objective of this project is about to analyze and study the effect of pressure coefficient (C_p) and drag coefficient (C_d) on convertible car roof. The study involves the flow of air against the vehicle roof is one most of the important aspect designing vehicle. In automotive design studies, the aerodynamic devices such as roof of vehicle are the important part in designing a vehicle. This aerodynamic device will influence of the stability, performance, fuel consumption and others on the vehicle.

4.2 RESULT AND DISCUSSION

4.2.1 Total Pressure and Pressure Coefficient

Pressure distribution over the top of car body analyzed by setting point global at certain location of car body so the pressure value of that point obtained. The data obtained from the simulation result used to calculate pressure coefficient. Sample of the calculation are as follow:

$$\text{Pressurecoefficien}, C_p = \frac{P - P\infty}{\frac{1}{2}\rho\infty V\infty^2} \qquad (4.1)$$

Where:

Total pressure, P = 100.992 kPa

Upstream pressure, P_∞ = 101.376 kPa

Upstream density, ρ_∞ =1.200 kg/ms

Upstream velocity, V_∞ = 33.33 m/s

Solution:

$$Cp = \frac{(100992 - 101376)\text{Pa}}{\frac{1}{2} \times \left(1.200 \frac{\text{kg}}{\text{m}^2}\right) \times (30.56 \frac{\text{m}}{\text{s}})^2}$$

= -0.5763

In this experiment, three types of wind tunnel were used in all four models to determine which more accurate decisions are. For figure of all models can refer page 19. Here is the data for the wind tunnel

Table 4.1: Data for Wind Tunnel

WT.	Name of Wind Tunnel	High (mm)	Long (mm)
1	POSTECH WT	1500	4300
2	VKI-L1 LOW SPEED WT	3000	5000
3	FLOW SCIENCE WT	2230	5500

Table 4.2 is the result of total pressure and pressure coefficient around convertible car roof for Model A. Fifty points has been located in different location along x-axis.

Table 4.2: Total pressure and pressure coefficient of car roof for Model A.

Point	**WT 1**		**WT2**		**WT3**	
	Pressure (Pa)	**Cp**	**Pressure (Pa)**	**Cp**	**Pressure (Pa)**	**Cp**
1	100991.5	-0.57693	100969.6	-0.60966	100968.7	-0.61108
2	100988	-0.58215	100959.4	-0.62504	100966.3	-0.6147
3	100973.8	-0.60337	100932.7	-0.66509	100954.6	-0.63229
4	100954	-0.63311	100905.5	-0.70583	100920.2	-0.68383
5	100945.2	-0.64639	100887.5	-0.73293	100884.8	-0.73698
6	100953	-0.63462	100894.8	-0.72191	100877.2	-0.74837
7	100985.4	-0.58604	100928.7	-0.67111	100895.5	-0.72091

8	101012.6	-0.54528	100969.3	-0.61012	100935.9	-0.66022
9	101024.9	-0.52668	101003.9	-0.55829	100975.9	-0.60028
10	101029.3	-0.52019	101015.7	-0.54062	100996.2	-0.56988
11	101030.7	-0.51799	101018.7	-0.53602	101000.9	-0.56279
12	101033.5	-0.5138	101022.1	-0.53098	101002.6	-0.56025
13	101038.7	-0.5061	101030.2	-0.5188	101008.5	-0.55139
14	101043.8	-0.49844	101038.4	-0.50654	101015.1	-0.54149
15	101045.8	-0.49542	101044.3	-0.49766	101020.4	-0.53354
16	101046.3	-0.49471	101047.9	-0.49229	101023.1	-0.52943
17	101045.6	-0.49571	101048.6	-0.49118	101023	-0.52961
18	101046	-0.49511	101048.9	-0.4908	101022.9	-0.52977
19	101048.2	-0.49183	101049.9	-0.48925	101023.6	-0.52864
20	101044.9	-0.49677	101050.9	-0.48779	101024.5	-0.52742
21	101051.3	-0.48718	101050.3	-0.48867	101024	-0.5281
22	101048.8	-0.49088	101047.5	-0.4929	101030.4	-0.51852
23	101045.3	-0.49622	101043.9	-0.49819	101017.6	-0.53764
24	101044	-0.49816	101042.2	-0.50082	101015.1	-0.54152
25	101043.4	-0.49894	101041.9	-0.5013	101015.1	-0.54152
26	101039.3	-0.5051	101040	-0.50417	101015.4	-0.54094
27	101030.1	-0.51893	101032	-0.51613	101012	-0.54608
28	101011.7	-0.54653	101010.7	-0.548	100999.8	-0.56439
29	100997.8	-0.56741	101007.1	-0.55352	100987.4	-0.58295
30	100994.1	-0.57303	101002.6	-0.56018	100983.9	-0.58823
31	100999.8	-0.56436	101008.5	-0.55134	100988	-0.58204
32	101017.7	-0.53754	101027.7	-0.52261	101001.5	-0.5619
33	101035.1	-0.51143	101044.4	-0.49755	101017.1	-0.53842
34	101045.4	-0.49603	101059	-0.47566	101029.6	-0.51964
35	101049.6	-0.48969	101063.8	-0.46839	101031.9	-0.51619
36	101051.2	-0.48734	101067.3	-0.46316	101031.5	-0.5168
37	101066.6	-0.46422	101092.2	-0.42578	101041	-0.50262
38	101096.1	-0.41998	101117.5	-0.38779	101066.9	-0.4637
39	101127.5	-0.37285	101156.3	-0.3296	101101.1	-0.41244
40	101165.2	-0.31627	101181.5	-0.29179	101129.7	-0.36948
41	101193.2	-0.27425	101202.9	-0.25963	101171.6	-0.30665
42	101212	-0.24603	101215.3	-0.24104	101193.9	-0.27313
43	101219.9	-0.23426	101218	-0.23706	101201.1	-0.26246
44	101227.2	-0.22321	101230.9	-0.21762	101208	-0.2521
45	101244.6	-0.19718	101262.2	-0.17077	101240.5	-0.20328
46	101278.5	-0.14626	101311	-0.09746	101284.7	-0.13699
47	101309.8	-0.09931	101340.3	-0.05354	101322.5	-0.08022
48	101332.6	-0.06511	101363.1	-0.01934	101342.3	-0.05063
49	101342.2	-0.05075	101379.4	0.005128	101345	-0.04652
50	101401.3	0.037901	101441.7	0.098505	101448.9	0.109307

Figure 4.1 shows the distribution of pressure coefficient around convertible car roof for a model in 3 different wind tunnels. Speed used for this experiment is 33.33m/s or 120km/h. At each Wind Tunnel, a model position in the wind tunnel floor is 2/3 of the overall length of 4300mm wind tunnel in wind tunnel 1 (WT1) , 5000mm and 5500mm at the wind tunnel 2 (WT2) and wind tunnel 3 (WT3). Experimental results found that for WT1 & WT 2, at Point 5 is located next car convertible roof surface gives the lowest pressure coefficient values of 0.6464 and 0.7329. While for WT3, the lowest pressure coefficient is at point 6 of 0.7429. This situation occurs due to the high velocity in the front surface causing downward pressure. The highest pressure coefficient value obtained at point 50 for all wind tunnels where value for WT1 is 0.0379, WT2 is 0.0985 and WT is 0.1093

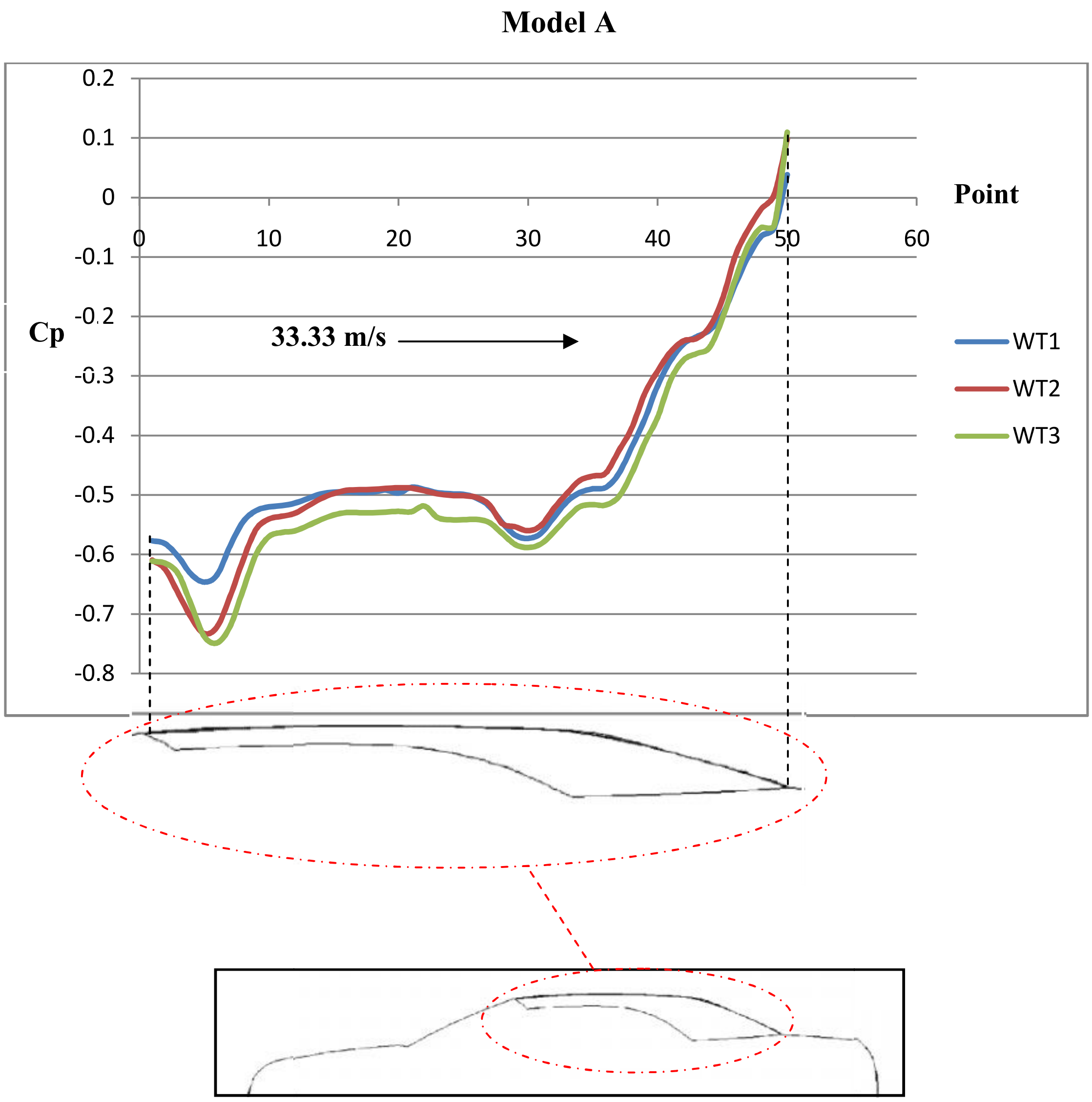


Figure 4.1: Pressure coefficient for Model A in three situation of Wind Tunnel

Table 4.3 is the result of total pressure and pressure coefficient around convertible car roof for Model B. Fifty points has been located in different location along x-axis.

Table 4.3:Total pressure and pressure coefficient of car roof for Model B.

Point	WT 1		WT2		WT3	
	Pressure (Pa)	Cp	Pressure (Pa)	Cp	Pressure (Pa)	Cp
1	100999.9	-0.56433	100968.9	-0.61084	101006.9	-0.55375
2	101000	-0.56408	100967.7	-0.61261	101005.8	-0.55547
3	100999.8	-0.56448	100962.4	-0.62056	101000.8	-0.56296
4	100992.3	-0.57562	100947.5	-0.64292	100987.3	-0.58319
5	100983.5	-0.58881	100924.3	-0.67776	100967.4	-0.61307
6	100969.6	-0.60967	100897	-0.71861	100946.8	-0.64388
7	100961.2	-0.62238	100879.3	-0.74514	100932.8	-0.66487
8	100959.5	-0.62481	100874.1	-0.75301	100929.2	-0.6703
9	100959.9	-0.62423	100880	-0.7442	100932.2	-0.66577
10	100961	-0.62269	100893.6	-0.72373	100940.8	-0.65291
11	100961.2	-0.62227	100910.4	-0.69853	100950	-0.63911
12	100959.6	-0.62476	100923.6	-0.67879	100955.7	-0.63058
13	100956.5	-0.6294	100930.6	-0.66824	100956	-0.63011
14	100950.2	-0.63881	100933.9	-0.66332	100954.5	-0.63242
15	100946.7	-0.64409	100934.2	-0.66277	100953.3	-0.63412
16	100946.9	-0.64385	100934.9	-0.66179	100954.8	-0.63194
17	100951.7	-0.63658	100936.9	-0.65883	100962.6	-0.62019
18	100962.3	-0.62062	100944.9	-0.64672	100974.2	-0.60283
19	100970.4	-0.60848	100951.9	-0.63631	100985.7	-0.58562
20	100976.5	-0.59944	100956	-0.63009	100993.3	-0.57413
21	100978.5	-0.59636	100957.3	-0.62818	100997.1	-0.5685
22	100979.7	-0.59459	100956.7	-0.62902	100998.1	-0.56694
23	100979.2	-0.59539	100956.4	-0.62947	100998.3	-0.56661
24	100981.4	-0.59203	100957.9	-0.62731	101001	-0.5626
25	100987.8	-0.58247	100964.7	-0.61702	101006.9	-0.55371
26	100995.8	-0.57039	100972.9	-0.60478	101014.4	-0.54249
27	101003.9	-0.55831	100983	-0.5896	101020.4	-0.53355
28	101008.9	-0.55078	100988.3	-0.58172	101022.5	-0.5304
29	101010.8	-0.54795	100990.3	-0.5787	101021.4	-0.53202
30	101011	-0.54756	100989.9	-0.57922	101019.5	-0.53479
31	101011.1	-0.54743	100989.3	-0.58012	101019.3	-0.53515
32	101012.6	-0.54519	100991.5	-0.57682	101024.1	-0.52794
33	101020.2	-0.53377	100997.5	-0.56781	101034.5	-0.51233
34	101030.9	-0.51774	101004.5	-0.55742	101043.4	-0.49907
35	101043.5	-0.49885	101003.7	-0.55862	101045.7	-0.49559
36	101052.2	-0.48586	100994	-0.57312	101041.8	-0.50143
37	101056	-0.4801	100979.6	-0.59466	101036.2	-0.50973
38	101056.7	-0.47905	100968.5	-0.61142	101033	-0.5146
39	101057.1	-0.47846	100975.8	-0.60045	101054.5	-0.48228
40	101135.4	-0.36091	101097.5	-0.41776	101160.8	-0.32283
41	101157.8	-0.32738	101175.5	-0.30077	101171	-0.30762

42	101180.7	-0.29306	101198.1	-0.26687	101202	-0.261
43	101210.2	-0.24878	101240.9	-0.20264	101245.6	-0.19562
44	101236.6	-0.20919	101283	-0.13948	101275.6	-0.15059
45	101255.3	-0.18103	101310.1	-0.09888	101285.5	-0.13581
46	101259.9	-0.17418	101314.7	-0.09191	101289	-0.13056
47	101278.9	-0.14566	101333.7	-0.06344	101309.6	-0.09969
48	101316.6	-0.08916	101389	0.019474	101351.9	-0.03613
49	101357.2	-0.0282	101434.7	0.088108	101377.1	0.001631
50	101376	-3.2E-05	101450.3	0.111514	101381.7	0.008514

Figure 4.2 shows the distribution of pressure coefficient around convertible car roof for a model B in 3 different wind tunnels. Speed used for this experiment and a model position in the wind tunnel floor is same with experiment Model A. Experimental results found that for WT1 at Point 15 is located next car convertible roof surface gives the lowest pressure coefficient values of -0.6441. For WT2 and WT3, the lowest pressure coefficient value is -0.753 and -0.6703 at point 8. The highest pressure coefficient value obtained at point 50 for all wind tunnels where value for WT1 is -0.0032, WT2 is 0.1115 and WT3 is 0.0085.

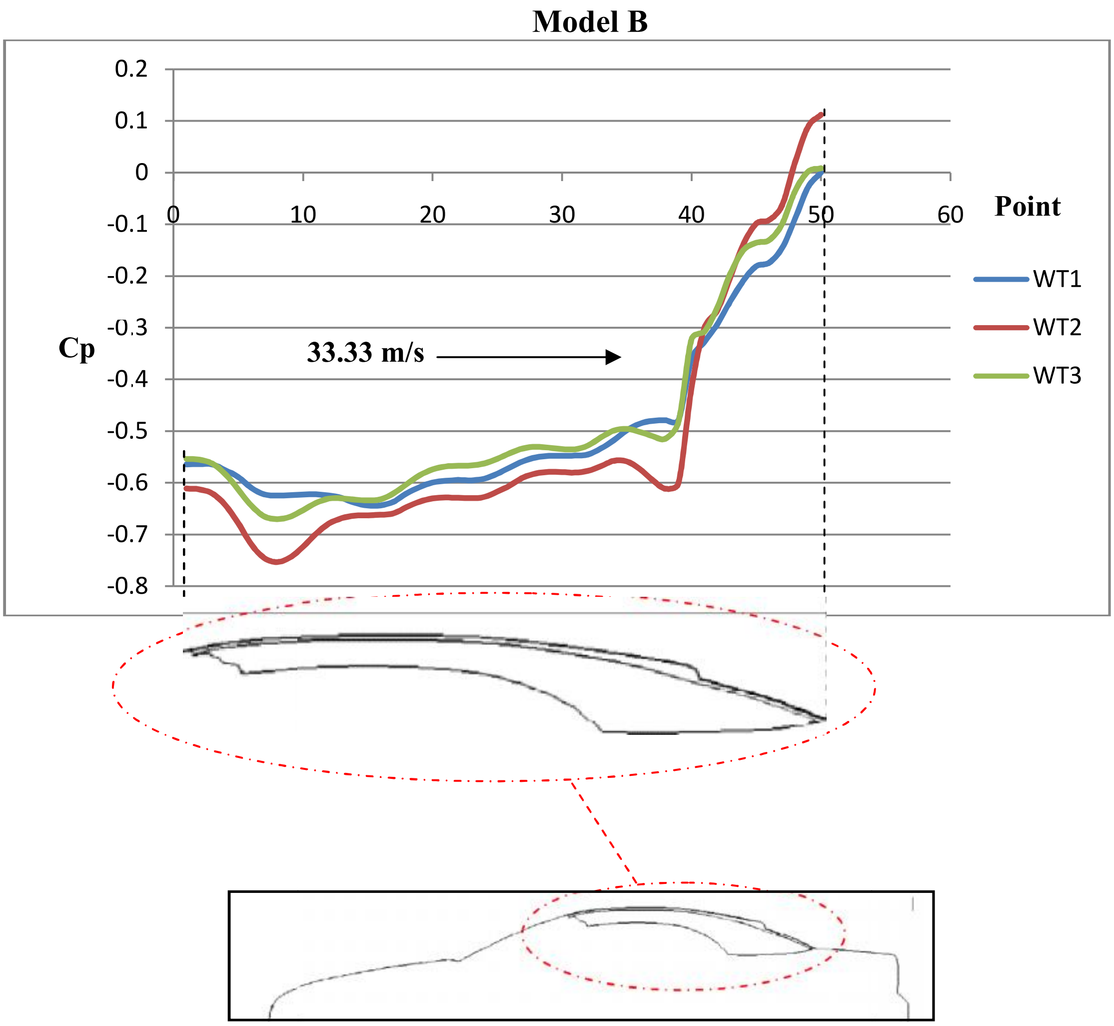


Figure 4.2: Pressure coefficient for Model B in three situation of Wind Tunnel

Table 4.4 is the result of total pressure and pressure coefficient around convertible car roof for Model C. Fifty points has been located in different location along x-axis.

Table 4.4: Total pressure and pressure coefficient of car roof for Model C.

Point	**WT 1**		**WT2**		**WT3**	
	Pressure (Pa)	**Cp**	**Pressure (Pa)**	**Cp**	**Pressure (Pa)**	**Cp**
1	101009.9	-0.54924	101063.3	-0.46912	101026.6	-0.52428
2	100847.2	-0.79338	100910.8	-0.69797	101006.8	-0.55396

3	100786.7	-0.88413	100841.5	-0.80186	100930.5	-0.6684
4	100771.6	-0.90673	100766.1	-0.91506	100887.9	-0.73227
5	100769.2	-0.91042	100758.3	-0.92667	100885.9	-0.73525
6	100776	-0.90025	100770.8	-0.90803	100889.7	-0.72959
7	100788.8	-0.881	100770.9	-0.9078	100883.2	-0.73928
8	100799.9	-0.86433	100784.1	-0.88799	100882.3	-0.74067
9	100804.9	-0.8568	100830.8	-0.81795	100907.8	-0.70237
10	100801.2	-0.86241	100839.6	-0.80473	100885.3	-0.73625
11	100792.5	-0.87544	100830.4	-0.81851	100889.8	-0.72949
12	100782.7	-0.8901	100838.1	-0.80697	100894.5	-0.72242
13	100783.2	-0.88933	100840.9	-0.80276	100897.3	-0.71822
14	100789	-0.88073	100843.1	-0.79955	100898.6	-0.71626
15	100805	-0.85673	100853.6	-0.78376	100905.1	-0.7065
16	100824.4	-0.82763	100871.8	-0.7564	100915.6	-0.69071
17	100842.9	-0.7998	100892	-0.72613	100911.7	-0.69657
18	100856.9	-0.77875	100908.8	-0.701	100919.5	-0.6849
19	100863.8	-0.7685	100912.8	-0.69489	100926.6	-0.67419
20	100866.2	-0.76484	100922.6	-0.68028	100931.4	-0.66704
21	100866.7	-0.76406	100923.5	-0.67892	100933.2	-0.66433
22	100871.5	-0.75686	100925.2	-0.67626	100934.3	-0.66275
23	100884.6	-0.73728	100931.3	-0.66725	100939	-0.65565
24	100890.9	-0.72775	100938.2	-0.65681	100953.5	-0.63381
25	100895.5	-0.72085	100942	-0.6512	100963.4	-0.61907
26	100894.5	-0.72235	100941.5	-0.6519	100972.7	-0.60512
27	100892.3	-0.72571	100940	-0.65418	100970.1	-0.60902
28	100893.7	-0.72359	100939.7	-0.65451	100970.4	-0.60853
29	100901.8	-0.71145	100941.9	-0.65135	100969.9	-0.60923
30	100911.3	-0.69719	100942.7	-0.65012	100962	-0.62107
31	100884.6	-0.73724	100937.7	-0.65761	100956.1	-0.63003
32	100874.3	-0.75268	100929.9	-0.66924	100952.8	-0.63499
33	100870.5	-0.75846	100926.7	-0.67408	100952	-0.63611
34	100878.3	-0.74666	100921.6	-0.68168	100947.9	-0.64233
35	100885.2	-0.73635	100907	-0.7036	100935.8	-0.66041
36	100777.7	-0.89757	100855.7	-0.78055	100919.6	-0.68474
37	100753.6	-0.9338	100826.6	-0.82421	100905.4	-0.7061
38	100752.2	-0.93589	100832.3	-0.81572	100910.7	-0.69812
39	100823.9	-0.82825	100898.1	-0.71705	100949.2	-0.6404
40	100982	-0.59111	100966	-0.61512	100992.9	-0.57478
41	100977.4	-0.59804	100997.8	-0.56739	101005.8	-0.55542
42	100984.5	-0.58743	101012.4	-0.54556	101018.1	-0.537
43	101023	-0.52954	101085	-0.43664	101088.8	-0.43082
44	101132.5	-0.36539	101167.6	-0.31271	101172	-0.30604
45	101176	-0.30008	101203.9	-0.25822	101205.5	-0.25573
46	101185.1	-0.28637	101210.5	-0.24836	101212.4	-0.24544
47	101222.5	-0.23028	101242.9	-0.19973	101240.4	-0.20344

48	101264.2	-0.16772	101284.6	-0.13708	101272.3	-0.15558
49	101284.2	-0.13767	101301.6	-0.11161	101289.5	-0.12984
50	101456.9	0.121412	101408.5	0.04883	101415.7	0.059577

The highest pressure coefficient value is also at point 50 as shown in Figure 4.3 due to the pressure separation region. On the front surface of the convertible car roof, the pressure falls to the point of 8 for most wind tunnels, but its back up to 34 points. Here occurs a sharp decrease involving WT1 where pressure coefficient for WT1 is 0.9338 at point 37 which is the lowest value among the three wind tunnels. For WT2 and WT3, the lowest pressure coefficient value is -0.9267 and -0.7407 at point 5 and point 8.

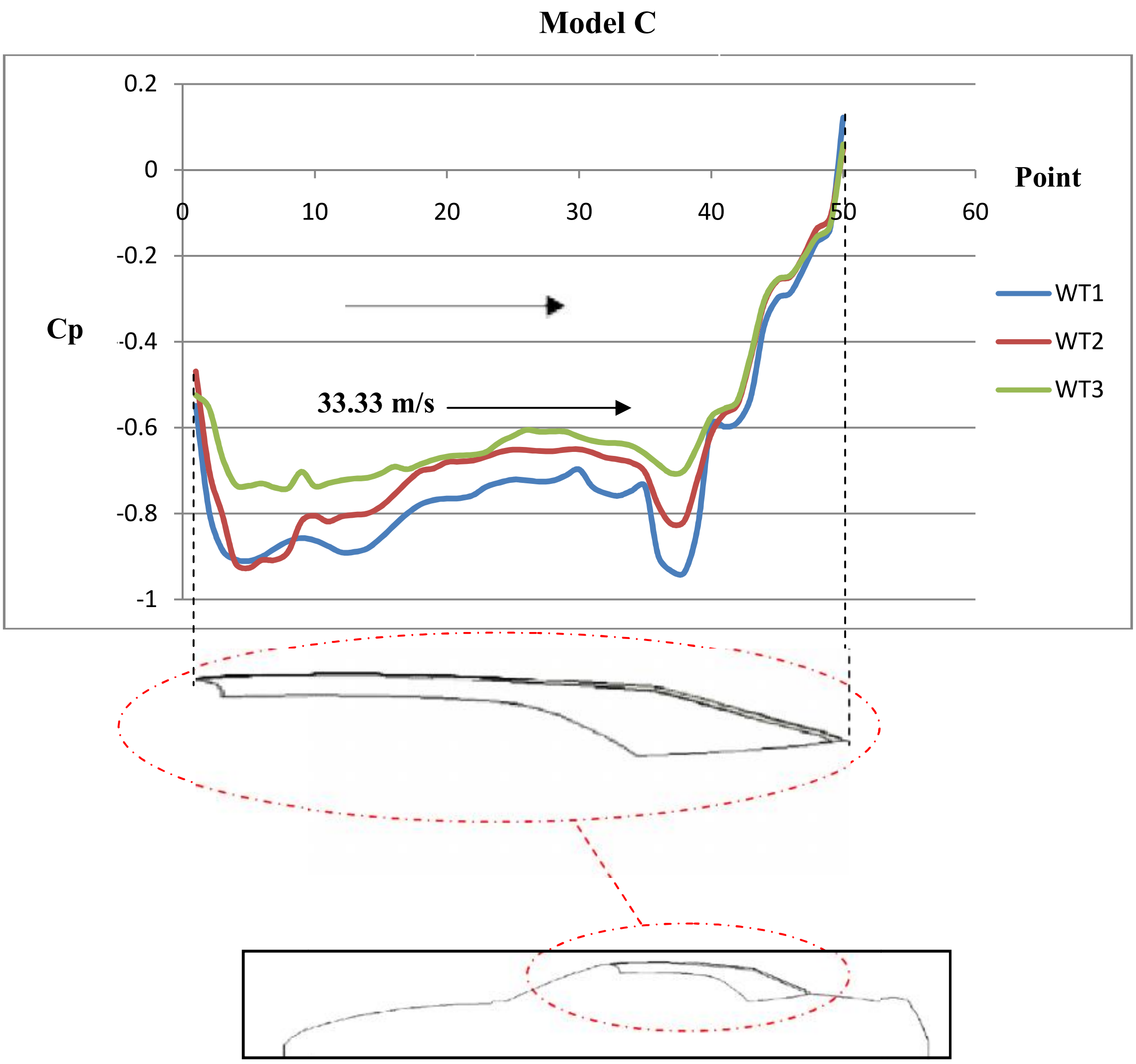


Figure 4.3: Pressure coefficient for Model C in three situation of Wind Tunnel

Table 4.5 is the result of total pressure and pressure coefficient around convertible car roof for Model D. Fifty points has been located in different location along x-axis.

Table 4.5: Total pressure and pressure coefficient of car roof for Model D.

Point	WT 1		WT2		WT3	
	Pressure (Pa)	Cp	Pressure (Pa)	Cp	Pressure (Pa)	Cp
1	101011.9	-0.54633	100955.5	-0.63082	100953.9	-0.63331
2	101008.9	-0.55075	100950.1	-0.63903	100953.7	-0.63356
3	100996.1	-0.57	100932.8	-0.66488	100937.6	-0.65778
4	100970.4	-0.60851	100907.9	-0.70234	100964	-0.61806
5	100925.6	-0.67579	100883.2	-0.73928	100910	-0.69912
6	100914.5	-0.69232	100873.6	-0.75378	100887.1	-0.73354
7	100907.2	-0.70337	100865.1	-0.76648	100883.7	-0.73856
8	100911.1	-0.69743	100868.6	-0.76129	100888.4	-0.73162
9	100917.9	-0.68729	100884.6	-0.73718	100893.6	-0.72374
10	100942.8	-0.64996	100903.2	-0.70929	100909.5	-0.69991
11	100958.1	-0.62699	100930.6	-0.66824	100930	-0.66909
12	100966.4	-0.61448	100952.8	-0.635	100948.1	-0.64203
13	100968.9	-0.61076	100964	-0.6181	100957.7	-0.62754
14	100969.2	-0.61039	100965.9	-0.61524	100959.7	-0.62465
15	100970.8	-0.6079	100969.2	-0.61033	100962.4	-0.62058
16	100974.3	-0.60274	100969.4	-0.61003	100968.9	-0.61072
17	100976.9	-0.59877	100981.7	-0.5916	100976	-0.60006
18	100976.6	-0.59917	100978.3	-0.59672	100969.9	-0.60927
19	100970.6	-0.60824	100981.7	-0.59151	100973	-0.60464
20	100971.7	-0.60655	100983.4	-0.58905	100974.5	-0.60237
21	100971.9	-0.60625	100983.5	-0.58887	100975	-0.60168
22	100971.5	-0.60688	100982.8	-0.58998	100975	-0.60169
23	100980.1	-0.59397	100981.1	-0.59244	100975.8	-0.60047
24	100971.5	-0.60687	100978.9	-0.5957	100977.4	-0.59808
25	100971.7	-0.60661	100977.1	-0.59848	100979.5	-0.59493
26	100975.2	-0.60134	100976	-0.60015	100981.4	-0.59206
27	100971.9	-0.60623	100975.6	-0.60074	100982.3	-0.59063
28	100972.8	-0.60494	100974.6	-0.60219	100981.1	-0.59242
29	100975.4	-0.60105	100968.7	-0.61106	100983.1	-0.58944
30	100973.5	-0.60384	100958.3	-0.62666	100979.9	-0.59426
31	100963.5	-0.61892	100948.7	-0.64106	100971.9	-0.60623
32	100948.4	-0.64159	100943.1	-0.64942	100964.3	-0.61764
33	100948	-0.64208	100942	-0.65118	100961.8	-0.62137
34	100959.7	-0.62451	100948.1	-0.64201	100963	-0.61966
35	100988.9	-0.58073	100967.5	-0.6128	100973.3	-0.60415
36	101014.9	-0.54168	100997	-0.56864	100995.3	-0.57117
37	101026.9	-0.52382	101020.2	-0.53382	101020.8	-0.53284
38	101028.9	-0.5208	101027.3	-0.52314	101021.6	-0.53173

39	101040	-0.50415	101038.2	-0.50687	101040.4	-0.50357
40	101097.4	-0.41802	101085.3	-0.43616	101099.4	-0.41495
41	101155.5	-0.33083	101154.4	-0.33246	101162.5	-0.32039
42	101187.8	-0.28237	101195.6	-0.27072	101194.3	-0.27259
43	101194.3	-0.27268	101202.8	-0.25982	101200.6	-0.26321
44	101222.3	-0.23064	101234.8	-0.21187	101222.3	-0.23063
45	101286.4	-0.13442	101294.2	-0.12272	101276.5	-0.14932
46	101331	-0.06756	101328.6	-0.07105	101316.5	-0.0893
47	101339.3	-0.05504	101335.2	-0.06127	101325.5	-0.07582
48	101366.7	-0.014	101369.4	-0.00997	101375.6	-0.00055
49	101412.1	0.054122	101414.7	0.058017	101442.8	0.100271
50	101427.2	0.076796	101428.2	0.078274	101462.8	0.130273

According Figure 4.4, the distribution of pressure coefficient around convertible car roof for a model D in 3 different wind tunnels. Speed used for this experiment and a model position in the wind tunnel floor is same with all experiment Model. Experimental results found that for all wind tunnels at Point 7 is located next car convertible roof surface gives the lowest pressure coefficient values. The lowest pressure coefficient value is -0.7034 for WT1, -0.7665 for WT2 and -0.7386 for WT3. The highest pressure coefficient value obtained at point 50 for all wind tunnels where value for WT1 is -0.0768, WT2 is 0.0783 and WT3 is 0.1302.

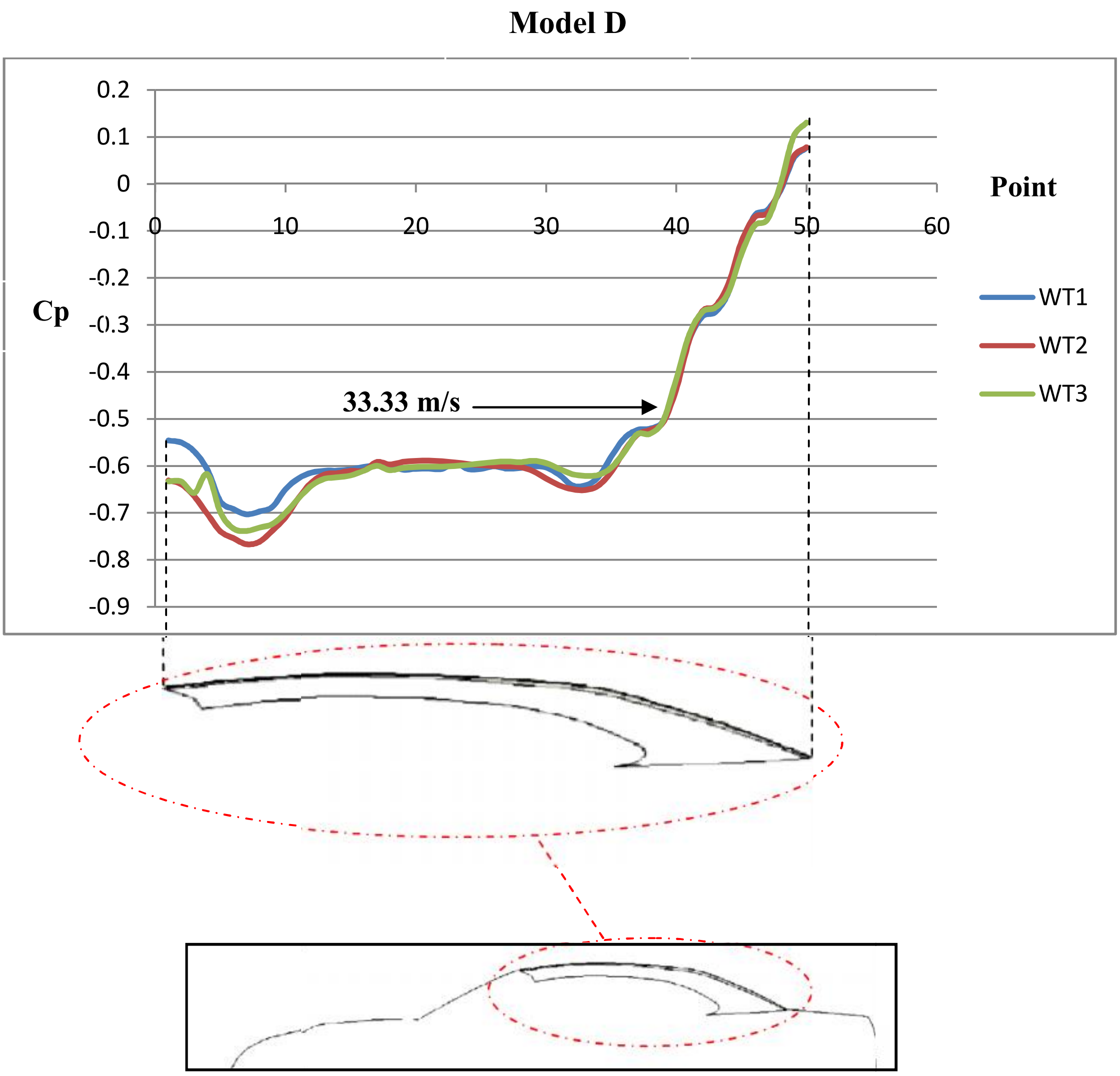


Figure 4.4: Pressure coefficient for Model D in three situation of Wind Tunnel

Experiment result of experiments involving the 3 types of wind tunnel for the four models are designed to get the value of the coefficient of pressure more accurate and approximate the true state of the situation. From this experiment, it was found that the wind tunnel 2 (WT2) has the required characteristics to that of the actual situation. It can be clearly seen that the pressure coefficient is found in the four models used in comparison with other wind tunnel.

4.2.2 Validation with the Experiment

Having found that the value of using a wind tunnel experiment is equal to the actual situation, then the evaluation and comparison of the four models conducted. In addition, comparisons with previous experiments were used to determine the benchmark for this experiment. In Figure 4.5 shows a graph comparing the pressure coefficients between the four models with the previous experiment. Previous experimental procedures are based on the experiment conducted by Knight (2009) using the model Jaguar XK8.

Several factors have been discovered as a result of this experiment, which is speed. Speed used in this experiment is 120km / h or 33.33m / s. result, higher speeds are used, the lower the pressure exerted on the convertible car roof. This situation occurs at the beginning of the convertible car roof. Nevertheless, the high pressure will be created at the rear end of the convertible car roof. This can clearly be seen in all the models in this experiment.

In addition, the design of each model affects the value of the pressure coefficient for each model. Compared between the four models, model C has a low coefficient of pressure compared with other models. Shape and gradient play an important role in the production of a convertible car roof to facilitate the flow of air through the surface.

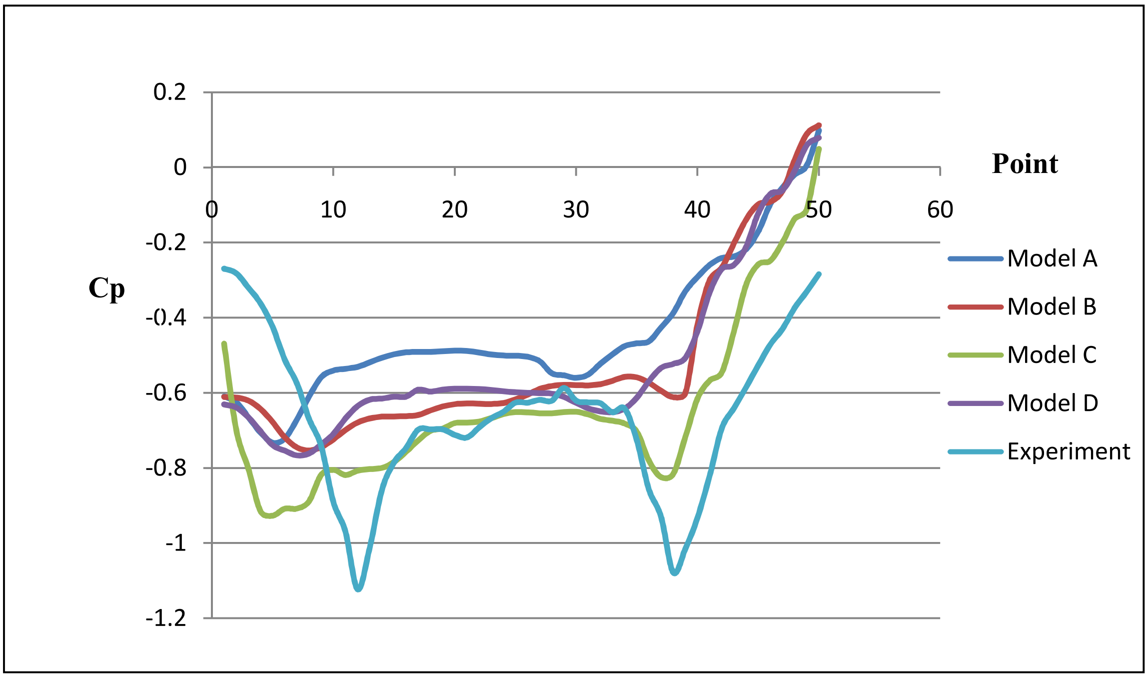


Figure 4.5: Comparison the four models with experiment.

4.2.3 Drag Coefficient

Force that acting along x-axis is drag force. Table 4.6 is the result for drag force for all four model investigated.

Table 4.6: Drag force for all model

Model	Frontal Area (m^2)	Drag Force (N)
A	0.0016435	0.86477
B	0.0018288	0.66796
C	0.0014016	0.61736
D	0.0017121	0.81098

The result from Table 4.6 used to calculate drag coefficient. Drag coefficient calculated using equation 2.14. The samples of calculation are as follow.

Drag Coefficient, C_D:

Drag Force, F_D = 0.86477N

Density, ρ = 1.204 kg/m³

Velocity, V = 33.33 m/s.

Frontal area, A = 0.0016435m^2

Solution:

$$C_D = \frac{0.86477\ \text{N}}{\frac{1}{2} \times \left(1.204 \frac{\text{kg}}{\text{m}^3}\right) \times \left(33.33 \frac{\text{m}}{\text{s}}\right)^2 \times (0.0016435\text{m}^2)}$$

= 0.7868

Table 4.7 shows drag coefficient for all models considered.

Table 4.7: Drag Coefficient.

Model	Frontal Area (M2)	Drag Force (N)	Drag Coefficient
A	0.0016435	0.86477	0.7868
B	0.0018288	0.66796	0.5462
C	0.0014016	0.61736	0.6586
D	0.0017121	0.81098	0.7083

Figure 4.6 is comparison of drag coefficient of all models convertible car roof.. Model B gives lowest drag coefficient value. This is because the curvature of the roof in the longitudinal direction affects the coefficient of drag and good air flow movement. Model A and Model D give higher value of drag coefficient than Model B and Model C. The highest value of drag coefficient produced by roof type of Model A because of its design which alter the main flow of air that is major contributor of the drag force produced by the separation zone at the rear of car roof.

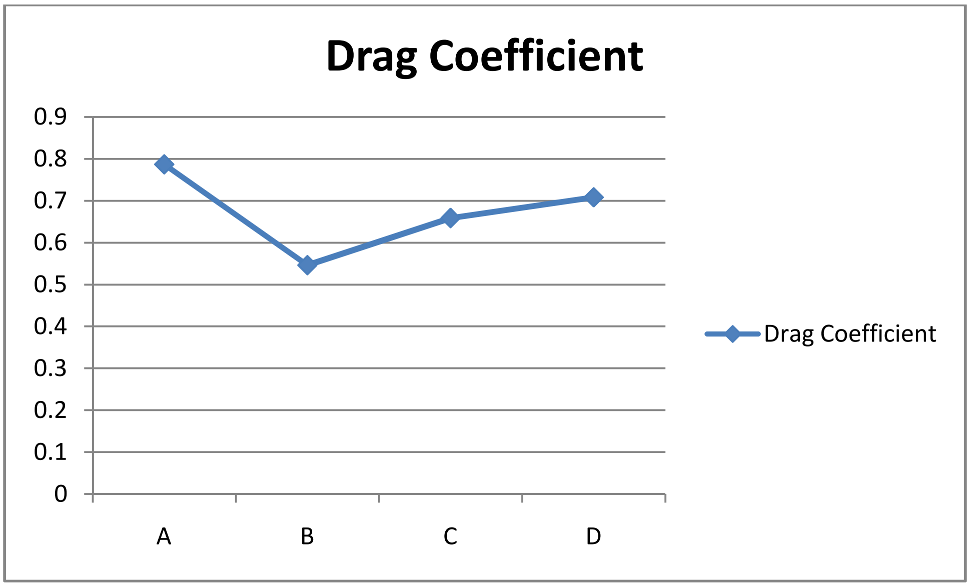


Figure 4.6: Drag Coefficient.

4.2.4 Drag Coefficient vs. Pressure Coefficient.

Figure 4.6, 4.7 and 4.8 is comparison between drag coefficient and pressure coefficient for all models in three critical points. For figure 4.7 shows the graph for comparison between the pressure coefficients with drag coefficient at the midpoint of the 25 points on the surface of the convertible car roof. In Figure 4.8 also involves comparison of the same but at the maximum point on each model is tested. While the figure was 4.9 is a comparison of the minimum point on each model.

Experiment results and comparison found that model A has the highest value in figure 4.7 and figure 4.9, which is the midpoint of convertible car roof surface, and the maximum value of the pressure coefficient. While the figure 4.8, the highest value is model B. By contrast, the lowest value for each figure because test results found that the model C is the lowest value for each figure.

After making all the experiments and analysis, it was found that the model C meets a nice feature to use because it has a low pressure coefficient and low drag coefficient compared to all models used.

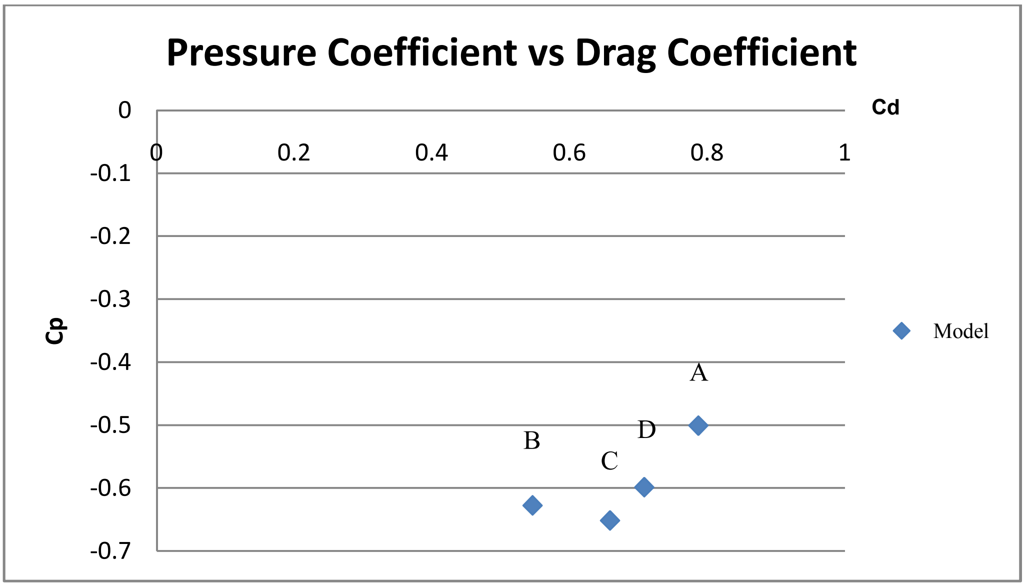


Figure 4.7: Pressure Coefficient vs. Drag coefficient at medium point.

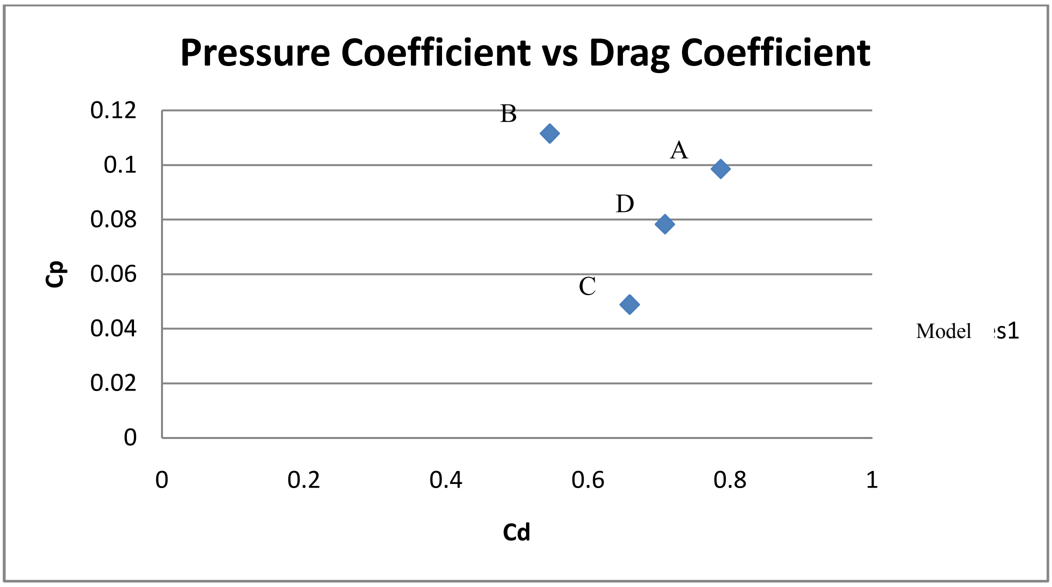


Figure 4.8: Pressure Coefficient at maximum point vs. drag coefficient

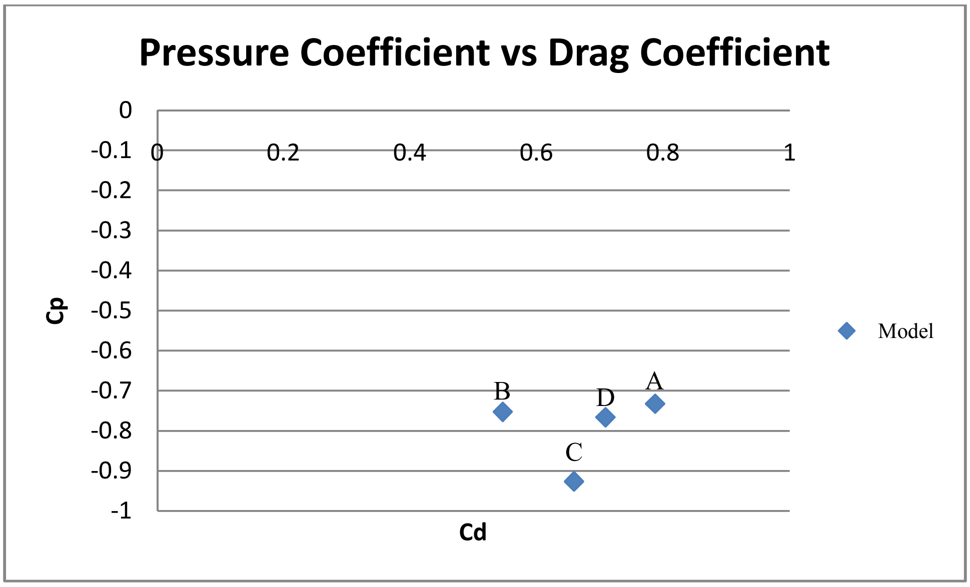


Figure 4.9: Pressure Coefficient at minimum point vs. drag coefficient

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 INRODUCTION

This chapter summarized the conclusion and recommendations for the overall objective of the project based on optimization analysis.

5.2 CONCLUSION

The all model of convertible car roof has been presented. An experimental investigation has been conducted to determine the effect of pressure coefficient on convertible car roof. The pressure coefficient is a parameter for studying the flow of incompressible fluids such as water, and also the low-speed flow of compressible fluids such as air and it is important for this experiment.

From the study, the structural modeling is one of important factors which need to be carefully design before performing the optimization to the convertible car roof. Thus, becoming the initial design of the convertible car roof and were compared with the optimized design based on the objective.

For this experiment, the speed used is 120km / h, the results of the study showed that the model C is better than the other models due to the effect of stress was observed on the C model is so little. Among the slight possibility of pressure is due to the design and shape of the surface that affects the flow of air and produce low pressure.

5.2 RECOMMENDATION

There is still scope for further study to improve the optimization analysis. The recommendations are as follows:

- Verifying the computational result by building and test both models in real situation.

- Increase refinement level (grid mesh size) of model to obtain more accurate result

- Further analysis in 3D analysis with Fluid structure interaction.

.

REFERENCES

Antonello Cogotti. 1996. *Experimental Technique For The Aerodynamic Development of Convertible Car*. Vehicle Aerodynamics. Society of Automotive Engineering. Inc, United State; 179-197.

Cengel, A.Y andCimbala, M.J. 2006.*Fundamentals of Fluid Mechanics*.1st ed. United State: McGraw Hill.

Chanani. A, Perere. N. 2008. CFD *Investigation of Airflow on a Model Radio Control Race Car*. Proceeding of the World Congress on Engineering 2008 Vol. II. WCE 2008. London, UK. July 2-4, 2008.

Choudhury, D.1993.*Introduction to the Renormalization Group Method and Turbulence Modeling, Technical Memorandum TM 107*, Fluent Inc., Lebanon NH.

Heisler, H. 2002. *Advanced Vehicle Technology*. Oxford: Elsevier Butterworth Heinemann.

Ismail, J. 2008. *Design and Analysis of Vortex Generator for a HEV*.B.Eng.Thesis. University Malaysia Pahang. Malaysia.

J.Y.Wong. 2008. Theory of Ground Vehicle. 4th edition. (Canada) John Wilet and Son. Inc.

Katz, J. 1995.*Race Car aerodynamics*. Cambridge: Bentley Publication.

Kim, J.S., Kim, S., Sung, J., Kim, J.S. and Choi, J. 2006. Effect of an AirSpoiler on the Wake of the Road Vehicle by PIV Measurement. *Journal of Visualization.***9**(4): 411-418.

Knight, J. J., Lucey, A. D., and Shaw, C. T. 2010.Fluid–structure interaction of a two-dimensional membrane in a flow with a pressure gradient with application to convertible car roofs. J. Wind Eng. Ind. Aerodynamics.

Knight, J. J., Lucey, A. D., and Shaw, C. T. 2010.A predictive capability for aerodynamic deformation of convertible car roof. Proc. IMechE Vol. 224 Part D: J. Automotive Eng.

Knight, J. J., Lucey, A. D., and Shaw, C. T. 2009.Fluid–structure interaction of the Jaguar XK8 convertible car roof. 18th World IMACS/MODSIM Congress, Cairn, Australia,

Najmudin, M. 2007. Analyze a New Spoiler for KUKTEM HEV.B.Eng.Thesis. University Malaysia Pahang. Malaysia.

Rajamani, G.K. 2006.CFD Analysis of Air Flow Interactions in Vehicle Platoons. B.Eng. Thesis. RMIT University. Australia.

Tsai, C.H., Fu, L.M., Tai, C.H., Loung, Y., Huang. and Leong, J.C.2007. Computational Aero-acoustic Analysis of a Passenger Car with a Rear Spoiler. *Journal of Applied Mathematical Modeling:* 3661-3673.

Watkins, S. and Oswald, G.1999. The Flow Field of Automobile Add-ons-with Particular Reference to the Vibration of External Mirror. *Journal of Wind Engineering and IndustrialAerodynamics.***83**.

Wolf-Heinrich Hucho. 1998. Aerodynamics of Road Vehicles. 4th Edition. Society of Automotive Engineering. Inc, United State.

APPENDIX A

GANTT CHART / PROJECT SCHEDULE FOR PSM 1 AND 2

Title : Computational Analysis of Two Dimensional Flows on a Convertible Car Roof.

No.	***Activities***	***Weeks***													
		1	*2*	*3*	*4*	*5*	*6*	*7*	*8*	*9*	*10*	*11*	*12*	*13*	*14*
1	Literature Study														
2	Identify Problem Statement														
3	Define Objective & Scope of Study														
4	Detail Methodology														
5	Study of CFD software														
6	Sketching the model of convertible car for analysis (Use Solidworks or another software)														
7	Presentation preparation														
8	FYP 1 presentation														

No.	Activities	Weeks															
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
1.	Literature Study																
2.	Analysis the convertible car roof model with CFD to find pressure coefficient.																
3.	Compare the analysis with another result experiment.																
4.	Report writing																
5.	Presentation preparation																
6.	FYP 2 presentation																
7.	Make final report																