UNIVERSITI TEKNIKAL MALAYSIA MELAKA

VALIDATION OF PRESSURE DISTRIBUTION ON GENERIC SIDE VIEW MIRROR BY EXPERIMENTAL AND SIMULATION

This report is submitted in accordance with the requirement of Universiti Teknikal Malaysia Melaka (UTeM) for the Bachelor of Mechanical Engineering Technology (Automotive Technology) with Honours

by

DAPHNE WONG JIE YIN

B071210049

920723-01-6674

FACULTY OF ENGINEERING TECHNOLOGY

2015

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **VALIDATION OF PRESSURE DISTRIBUTION ON GENERIC SIDE VIEW MIRROR BY EXPERIMENTAL AND SIMULATION**

SESI PENGAJIAN: **2015/16 Semester I**

Saya **DAPHNE WONG JIE YIN**

mengaku membenarkan Laporan PSM ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Laporan PSM adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan laporan PSM ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

☐ SULIT (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)

☐ TERHAD (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia sebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972)

☐ TIDAK TERHAD

Disahkan oleh:

_________________________ _________________________

Alamat Tetap:

D 41, Kangkar Pulai,

81110 Johor Bahru,

Johor.

Cop Rasmi:

Tarikh: _________________________ Tarikh: _________________________

** Jika Laporan PSM ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh laporan PSM ini perlu dikelaskan sebagai SULIT atau TERHAD.

DECLARATION

I hereby, declare that this thesis entitled “Validation of pressure distribution on generic side view mirror by experimental and simulation” is the result of my own research except as cited in references.

Signature : ……………………………………………

Name : DAPHNE WONG JIE YIN

Date : ………………………………………………

APPROVAL

This report is submitted to the Faculty of Engineering Technology of UTeM as one of the requirements for the award of Bachelor of Mechanical Engineering Technology (Automotive Technology) with Honours. The following are the members of supervisory committee:

………………………………………………

Supervisor: Mohd Faruq Bin Abdul Latif

………………………………………………

Co-Supervisor: Mohd Suffian Bin Ab. Razak

ABSTRACT

In an age of advanced technology which more towards on green science and green technology, reducing fuel consumption and emission become one of the major goals in automotive industry. Aerodynamics performance around the vehicle gives a great influence in fuel consumption and emission of a vehicle. Side view mirror, a compulsory components that mounted on both side of the automobile. It acts as a device for providing the driver a indirect vision adjacent to the vehicle which cannot be observed by a direct vision. However, the vibration and noise generated from the side view mirror had resulted a distortion, image blurring, and discomfort ambience to the driver and passengers. It is due to the fluctuating pressure on the mirror surface which might led to the aerodynamic effects to the side mirror even thought the effect is considered small if compared to the total drag of a vehicle. Therefore, this aim of this study is to remodel the physical testing using Computer-Aided Design (CAD) software to investigate the pressure distribution on the mirror surface and verifying the simulation result with the physical experiment wind tunnel data. Computational Fluid Dynamics (CFD) software is used as a simulation software in this study. The result obtained are discussed in pressure distribution contour and in term of pressure coefficient, C_p by introducing different parameter of mirror's foot height and foot width.

ABSTRAK

Dalam era teknologi yang semakin menuju ke arah teknologi hijau, bidang perindustrian automotif telah berusaha untuk mengaji penyelesaian yang lebih baik untuk meningkatkan kecekapan penggunaan bahan api, dan mengurangkan pembebasan karbon dioksida. Seretan aerodinamik kereta satu pemangkin yang banyak memberi pengaruh dalam usaha-usaha yang dijalankan ini. Cermin pandang sisi adalah satu daripada penyumbang dalam peningkatan seretan aerodinamik kereta. Walaupun seretan aerodinamik bagi cermin pandang sisi memainkan peranan yang agak kecil berbanding dengan seretan aerodinamik bagi sebuah kereta, ia telah memberi kesan kepada getaran dan bunyi bising dihasilkan daripada seretan aerodinamik. Hal ini disebabkan kewujudan pengedaran tekanan yang tidak rata pada permukaan cermin panding sisi yang boleh menyebabkan seretan aerodinamik berlaku walaupun kesannya adalah kecil berbanding dengan seretan aerodinamik keseluruhan. Oleh yang demikian, kajian ini dijalankan untuk mereka bentuk semula cermin pandang sisi generik menggunakan perisian *Computer-Aided Design (CAD)* demi mengkaji pengedaran tekanan pada permukaan cermin. Akhirnya, data-data tekanan daripada keputusan simulasi perisian akan diesahkan dengan data tekanan daripada *Wind Tunnel Test* yang sebenar. *Computational Fluid Dynamics (CFD)* digunakan sebagai perisian simulasi dalam kajian ini. Keputusan akan dibincangkan melalui kontur pengedaran tekanan dan pekali tekanan, $\boldsymbol{C_p}$

DEDICATION

To my beloved parents, siblings, and friends.

ACKNOWLEDGEMENT

First and foremost, I would like to express my sincere gratitude to my supervisor, Mr. Mohd Faruq Bin Abdul Latif for giving me an opportunity working under his supervision. A lot of thanks to him for his guidance, knowledge, encouragement given to me throughout this project. Not to forget my former supervisor, Mr. Nur Rashid Bin Mat Nuri @ Md Din who gives me the chances to get involved in this project. A lot of extra inputs have been given to me to complete the project within the duration.

I would like to acknowledge my co-supervisor Mr. Suffian Bin Ab. Razak. Thanks to his guidance and supports in assisting me through completing this projects. I am grateful for their assistance and guidance to make this project successful.

Besides, I would like to thank my friends who had been always inspiring me with their brilliant ideas and suggestions to improve my project. Not to forget, my family and siblings for giving me their fully support throughout the project.

TABLE OF CONTENTS

DECLARATION……………………………………………………………………iii
APPROVAL……………………………………………………………………....iv
ABSTRACT……………………………………………………………………..v
ABSTRAK…………………………………………………………………......vi
DEDICATION...vii
ACKNOWLEDGEMENT viii
TABLE OF CONTENTS ix
LIST OF TABLES xiii
LIST OF FIGURES xiv
LIST OF ABBREVATIONS, SYMBOLS AND NOMENCLATURES xvii
CHAPTER 1: INTRODUCTION...1

1.1 Background 1

1.2 Problem Statement 2

1.3 Objectives 3

1.4 Project Scope 3

CHAPTER 2: LITERATURE REVIEW 4

2.1 Introduction..4

2.1.1 History of automotive aerodynamic technology..................................4

2.1.2 Fluid Mechanic 6

2.1.2.1 Reynolds number 8

2.1.2.2 Navier-Strokes equation 9

2.1.2.3 Turbulence and laminar flow 9

2.1.3 Side mirror 10

2.2 Vehicle aerodynamic 11

2.2.1 Internal 11

2.2.2 External 11

2.2.2.1 Aerodynamic drag 12

2.2.2.2 Drag force 12

2.2.2.3 Aerodynamic lift 13

2.2.2.4 Pressure coefficient 13

2.2.2.5 Drag coefficient 14

2.2.3 Bluff Body 14

2.2.4 Vortex shedding 15

2.3 Modeling software 16

2.3.1 Computer-Aided Design (CAD) 16

2.3.2 Computer-aided Three-dimensional Interactive Application (CATIA) 17

2.3.3 Computational Fluid Dynamics (CFD) 17

2.3.3.1 Mesh Generation 18

2.3.3.2 Numerical Simulation Methods 19

2.3.3.3 Turbulence Models 20

2.3.4 Virtual wind tunnel 22

2.4 Experimental 23

2.4.1 Wind tunnel 23

2.5 Pressure measurements on an automobile side rear view mirror 24

2.6 Side mirror aerodynamics analysis 27

2.7 Computational Analysis of an effect of aerodynamic pressure on side view mirror 29

2.8 Designing and optimizing side-view mirrors 32

2.9 The effect of diffuser angle on modified generic side view mirror 35

CHAPTER 3: METHODOLOGY 38

3.1 Introduction..38

3.2 Flowchart 39

3.3 Generic side mirror...40

3.4 Designed Model 40

3.5 Design the side mirror in CATIA 41

3.6 Computational Fluid Dynamics (CFD) Simulation 42

3.6.1 Pre-Processing Configuration in HyperMesh 43

3.6.1.1 Mesh generation 43

3.6.2 Virtual Wind Tunnel 46

3.6.3 Result Interpretation in AcuSolve 47

3.7 Physical Experiment Wind Tunnel Data 48

3.8 Verifying of simulation and experiment result 49

CHAPTER 4: RESULTS...50

4.1 Introduction..50

4.2 Streamline 51

4.3 Pressure contour..52

4.3.1 Pressure contour at 15m/s 53

4.3.2 Pressure contour at 20m/s 55

4.3.3 Pressure contour at 25m/s 57

4.4 Comparison of Simulation and Experimental..59

4.4.1 Pressure Coefficient of Model A 60

4.4.2 Pressure Coefficient of Model B...62

4.4.3 Pressure Coefficient of Model C 64

CHAPTER 5: DISCUSSIONS 66

5.1 Viscosity and pressure 66

5.2 Pressure distribution using CFD simulation 67

5.2.1 Pressure distribution at 15m/s 67

5.2.2 Pressure distribution at 20m/s 68

5.2.3 Pressure distribution at 25m/s 68

5.2.4 Discussion 69

5.3 Comparison of Simulation and Experimental data 69

5.3.1 Model A 69

5.3.2 Model B 70

5.3.3 Model C 71

CHAPTER 6: CONCLUSION AND RECOMMENDATIONS 72

6.1 Introduction 72

6.2 Conclusion 72

6.3 Problems faced during research 73

6.4 Recommendation for future works 74

REFERENCES 75

APPENDICES 78

LIST OF TABLES

Table 2.1 - Designed model 35
Table 3.1 - Parameter of designed model...40
Table 3.2 - Setting of Wind Tunnel's surrounding pressure......................................48

LIST OF FIGURES

Figure 2.1 - History of vehicle aerodynamics in passenger cars. 5
Figure 2.2 - Possible classification of continuum fluid mechanics 7
Figure 2.3 - Incompressible flow over a sphere 7
Figure 2.4 - Streamlines around a car. 8
Figure 2.5 - Laminar flow. 10
Figure 2.6 - Turbulent flow. 10
Figure 2.7 - Examples of bluff body aerodynamics 15
Figure 2.8 - CFD process flow 18
Figure 2.9 - The hole meshed with 16 quads elements 18
Figure 2.10 - Prism layer growth 19
Figure 2.11 - Wind tunnel test 23
Figure 2.12 - Schematic of data representation in relation to mirror geometry and coordinates 24
Figure 2.13 - Wind tunnel test of the mirror with a Ford quarter model. 25
Figure 2.14 - Contour plot of mean C_p,60 km/h 25
Figure 2.15 - Contour plot of mean C_p,120 km/h 26
Figure 2.16 - Pressure distribution..27
Figure 2.17 - Streamline around the side mirror. 28
Figure 2.18 - Half-sphere shape design of side mirror. 29
Figure 2.19 - Sharp end with triangular shape design. 29
Figure 2.20 - Combining rectangular shape with triangular edges design.................30
Figure 2.21 - 100 critical points across the frontal area of side mirror......................30
Figure 2.22 - Pressure coefficient of side view mirror at 17 m/s. 31
Figure 2.23 - Varying the gap..32
Figure 2.24 - Varying the height of the foot..33
Figure 2.25 - Varying the inclination...33
Figure 2.26 - Varying the inner radius. 33
Figure 2.27 - Different housing curvatures. 33
Figure 2.28 - Diffuser angle parameter 35
Figure 2.29 - Drag coefficient ($\boldsymbol{C_D}$) of four models..36

Figure 2.30 - Turbulence separated region around side mirror. 36
Figure 2.31 - Pressure distribution at the mirror. 36
Figure 3.1 - Methodology flow chart...39
Figure 3.2 - Modified side mirror model in CATIA. 41
Figure 3.3 - Parameters used in designing model...42
Figure 3.4 - CFD process flow. 42
Figure 3.5 - HyperMesh Software 43
Figure 3.6 - Hypermesh Geomertry toolbar 44
Figure 3.7 - Altering element density 44
Figure 3.8 - Mesh structure edited by split surf-line and washer split.......................45
Figure 3.9 - Mesh Structure on mirror surface 45
Figure 3.10 - Virtual Wind Tunnel Software 46
Figure 3.11 - Representation of data points on the mirror surface 47
Figure 3.12 - Mirror geometry...49
Figure 3.13 - Representation of data point on mirror surface in Wind Tunnel Test ..49
Figure 4.1 - Streamline along the mirror at 25m/s..52
Figure 4.2 - Contour of pressure distribution at mirror surface with different Model at v=15m/s 54
Figure 4.3 - Contour of pressure distribution at mirror surface with different Model at v=20m/s 56
Figure 4.4 - Contour of pressure distribution at mirror surface with different Model at v=25m/s 58
Figure 4.5 - Position of 30 points on the mirror surface. 59
Figure 4.6(a) - Pressure coefficient of Model A over 30 points on mirror surface in CFD simulation 60
Figure 4.6(b) - Pressure coefficient of Model A over 30 points on mirror surface in physical experiment data 60
Figure 4.6.1 - Pressure coefficient of Model A at 15m/s 61
Figure 4.6.2 - Pressure coefficient of Model A at 20m/s 61
Figure 4.6.3 - Pressure coefficient of Model A at 25m/s 61
Figure 4.7(a) - Pressure coefficient of Model B over 30 points on mirror surface in CFD simulation 62
Figure 4.7(b) - Pressure coefficient of Model B over 30 points on mirror surface in experimental 62

Figure 4.7.1 - Pressure coefficient of Model B at 15m/s 63
Figure 4.7.2 - Pressure coefficient of Model B at 20m/s 63
Figure 4.7.3 - Pressure coefficient of Model B at 25m/s 63
Figure 4.8(a) - Pressure coefficient of Model C over 30 points on mirror surface in CFD simulation...64
Figure 4.8(b) - Pressure coefficient of Model C over 30 points on mirror surface in experimental 64
Figure 4.8.1 - Pressure coefficient of Model C at 15m/s 65
Figure 4.8.2 - Pressure coefficient of Model C at 20m/s 65
Figure 4.8.3 - Pressure coefficient of Model C at 25m/s 65

LIST OF ABBREVATIONS, SYMBOLS AND NOMENCLATURES

2D	-	Two-dimensional
3D	-	Three-dimensional
ANSYS	-	Analysis Software
CAD	-	Computer-Aided Design
CAE	-	Computer-Aided Engineering
CAM	-	Computer-Aided Manufacturing
CATIA	-	Computer-Aided Three-dimensional Interactive Application
CFD	-	Computational Fluid Dynamics
DES	-	Detached-Eddy Simulation
DNS	-	Direct Numerical Simulation
FE	-	Finite Elements
FSI	-	Fluid-Structure-Interaction
FVM	-	Finite Volume Method
FYP	-	Final Year Project
LES	-	Large Eddy Simulation
OSRVM	-	Outside Rear Mirror
RANS	-	Reynolds-Averaged Navier-Strokes
VWT	-	HyperWorks Virtual Wind Tunnel
A	-	Area
C_D	-	Drag Coefficient
C_L	-	Lift Coefficient
C_p	-	Pressure Coefficient
D	-	Aerodynamic Drag
F_D	-	Drag Force
F_L	-	Lift Force
l	-	Length
P	-	Pressure
P_s	-	Free stream total pressure

P_T	-	Total Pressure
Re	-	Reynolds Number
v	-	Velocity
μ	-	Viscosity
ρ	-	Density

CHAPTER 1

INTRODUCTION

1.1 Background

Automobile is becoming one of the necessities in almost every family nowadays. However, in the age of technology which more towards on green science and green technology, reducing the consumption of fuel, minimizing the emission of carbon dioxide and other unwanted emissions in order to achieve perfect combustion to achieve great fuel economy are becoming the major goals in current automotive industry. Automotive aerodynamic is considered as one of the factors which give a great influence in fuel consumption, and the emissions from the vehicles. Therefore, a continuously improvement has been made by automotive industry in order to reduce the aerodynamic performance as well as the noise and vibration generated.

A generic side view mirrors, which give a great influence in contributing the aerodynamic drag, noise and vibration to the vehicles. The function of a side view mirrors is to provide a clearer vision to the driver to observe or know the surrounding condition of the vehicle. However, the vibration or noise generated on the side mirror which mainly caused by the uneven pressure distribution and higher magnitude of fluctuating pressure on the mirror. In addition, the cavities and gaps beside the mirror also act as a aerodynamic noise generator which unintentionally brings discomfort to the driver and passengers. Therefore, the study of pressure distribution of the side mirror helps more in understanding its aerodynamics characteristics to the overall vehicle performance.

1.2 Problem Statement

In today's automotive industry, reducing in fuel consumption and minimizing the emissions of carbon dioxide is becoming a major goals among automobile's manufacturer. (P. S. O. Murukesavan, 2012). In order to encounter the problem, to change the vehicle's propulsion system can be one of the probability, and another way is to reduce the aerodynamic drag of the vehicle. The aerodynamic drag gives a great influences when it comes to velocities over 60km/h. Side view mirror, an accessories attached on the body skin of car which contribute to the overall drag by 2-7 percent.(Kim, Han, Lee, Hwang, & Jung, 2008; Olsson, 2011b). Side mirror primary function is to provide a clear vision of the rear of a vehicle.. However, the aerodynamically mirror vibration causes the several problems such as image distortion which can impair the driver's vision (R Jaitlee, Alam, & Watkins, 2004), and the fluctuations of pressure generated over the mirror housing can also generate aerodynamics noise.(Kim, Han, & Lim, 2009).

Current study and research introduced a lot of methods in order to reduce the aerodynamics effect which focused on side view mirror including the study of (Olsson, 2011b) by varying inclination, gap, height and different housing curvature of the mirror using both actual wind tunnel test and computational fluid dynamic (CFD) method, study of wake flow and wake structure by (Kim et al., 2008), and study of transient flow with variable turbulence model; Detached Eddy Simulation (DES) and Large Eddy Simulation (LES) by (Wang, Gu, Li, & Lin, 2010).

In this study, by introducing parameter changes on the mirror, pressure distribution over the mirror will be study and verify with physical experiment wind tunnel data and computational fluid dynamic (CFD) analysis. Hopefully it will enhance the use of CFD simulation in order to produce important data and information during the design stage(Eduardo, Mello, & Fernandes, 2009), and it would help to reduce the cost and time consuming issues instead of running an actual experiment.

1.3 Objectives

1. To remodel the physical testing using Computer-Aided Design (CAD) software.
2. To verify the physical wind tunnel using Computational Fluid Dynamic (CFD) method.

1.4 Project Scope

There are a few scopes have been drawn in order to achieve the objectives of this study. A generic side mirror is introduced to avoid complexity of the model. The Computer Aided Design (CAD) software is used to remodel back the generic side view mirror. Due to time limitation only the design models with parameter changes in mirror's foot height and foot width are tested and analyzed with different velocity input. The model optimization is not included in this study. With the model created by CAD, it is then evaluate using Computational Fluid Dynamic (CFD) simulation. Steady analysis type is chosen instead of transient type. Due to the limitation of meshing, quads mesh type is used in mesh structure on mirror surface. The physical wind tunnel testing is not covered in this project. The physical experiment wind tunnel data are refer from (Rashid, 2010). The study will focus on the effect of pressure on the mirror surface. Lastly, the result is then verified with the physical testing data.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Automobile aerodynamics' flow processes can be basically subjected into three categories which are air flow through the vehicle's body, air flow around the vehicle, and flow processes within the vehicle's machinery. Aerodynamic drag which is an important criteria to discuss in vehicle aerodynamics. It plays an important role in effect of the performance, fuel economy, emissions, maximum speed, stability and etc of a vehicle. Vehicle aerodynamic comprises much more than drag. For external flow field over the vehicle, the flow around a vehicle is more for the vehicle's directional stability, straight line stability, and dynamic passive steering. It helps to avoid droplets of rain water from accumulating on mirrors and windows. Also, it is responsible in keeping the headlight free of dirt, minimizing the wind noise, and cooling of the engine' s oil pan, brakes, muffler, and more. On the other hand, the internal flow is responsible in accommodating the engine's heat losses and provide a comfortable climate inside the passenger compartment.

2.1.1 History of automotive aerodynamic technology

Aerodynamics and vehicle technology have merged only very slowly, the synthesis of these two elements came to a success only after several tries. One of the reasons for the early repeated failures of aerodynamics with vehicles is that it started far too early, the first automobile were pretty slow. Most of the designs of vehicles from the early era were originally from the shapes of ships and airplanes. However, this turned out to be the wrong approach. (Hucho, 1998)A brief overview of the history of automotive aerodynamics is summarized in Figure 2.1. There are four major periods which distinguished but cannot be separated from each other as strictly as in the schematic. It concentrate on fluid dynamic effects, how it transformed to application, and make clear what is required to convert fluid dynamics into vehicle aerodynamics.

During the first two periods shown in Figure 2.1, most of the aerodynamic development was done by individual, which coming from outside the automotive industry. The discipline of vehicle aerodynamics was taken over by car companies and thus integrated into product development during the remaining two periods as shown in Figure 2.1. (Hucho, 1998)

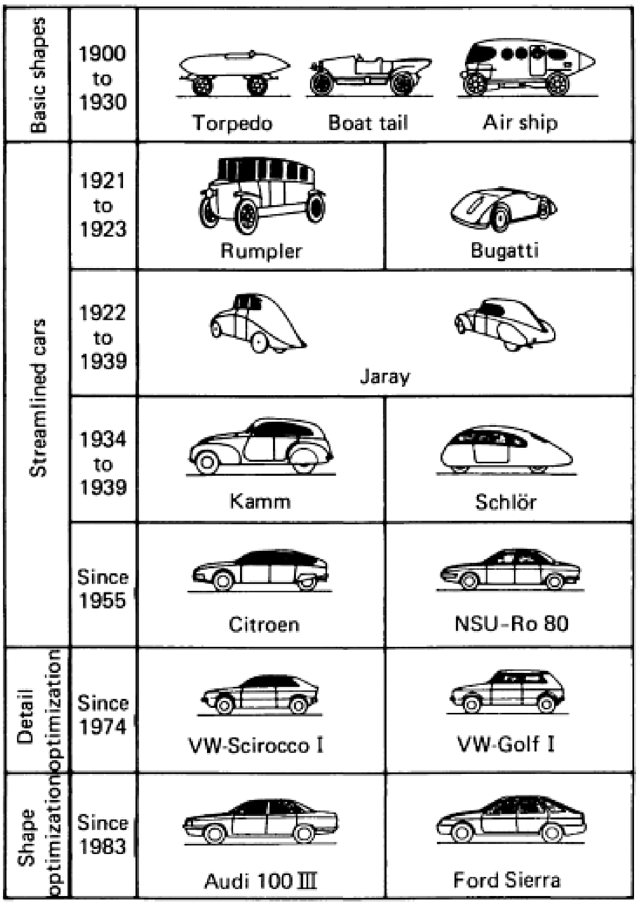

Figure 2.1 : History of vehicle aerodynamics in passenger cars. (Hucho, 1998)

The first automobile develop according to the aerodynamic principles was Torpedo-shaped vehicle which borrow and modify the shape from the marine and airships industry. These vehicles possessed lower drag coefficient, but the exposed drive from the vehicle, free-standing wheels, and undercarriage disturbed its flow properties. As the years passed, the studies and research on aerodynamic effects on automobile had increase and upgrade the designs of vehicle to accommodate the consumers. The automobiles developed with even more smooth bodies, streamlined-design body, integrated fenders and headlamps enclosed in the body. (Hucho, 1998)

2.1.2 Fluid Mechanic

Fluid mechanics study the fluids at either at rest or in motion. It has traditionally been applied in various areas including the aerodynamics of automobiles and sub- and supersonic airplanes. Fluid mechanics can basically break into manageable proportions. It turns out that the two most difficult aspects of a fluid mechanics analysis to deal with are the fluid's viscous nature and it's compressibility. As shown in Figure 2.2, showed the classification in terms of whether a flow is laminar or turbulent, and internal or external. The boundary layer, which is a thin layer of a few millimeters thickness. The viscous effects in the fluid will be restricted to boundary layer if there is no flow separation takes place. Thus, the flow can be regarded as inviscid beyond this layer. The boundary layer is dispersed when the flow separates (Figure 2.3b), now the flow is governed by viscous effects entirely. (Fox, et al., 2012)