DESIGN AND ANALYSIS REAR DIFFUSER FOR HEV MODEL

MOHD SYAZRIN BIN SOPNAN

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS

JUDUL: DESIGN AND ANALYSIS REAR DIFFUSER FOR HEV MODEL

SESI PENGAJIAN: 2008/2009

Saya **MOHD SYAZRIN BIN SOPNAN (860904-33-5541)**

mengaku membenarkan tesis (Sarjana Muda / ~~Sarjana~~ / ~~Doktor Falsafah~~)* ini disimpan di perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh Organisasi/badan dimana penyelidikan dijalankan)
√	**TIDAK TERHAD**	

Disahkan oleh

____
(TANDATANGAN PENULIS)

____
(TANDATANGAN PENYELIA)

Alamat Tetap:
KG BN 38/7, KM 16,
KG BUKIT NIBONG, PAYA DALAM,
75460, MELAKA

Nama Penyelia:
DEVARAJAN A/L RAMASAMY

Tarikh: **7 NOVEMBER 2008**

Tarikh: **7 NOVEMBER 2008**

CATATAN: * Potong yang tidak berkenaan
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD
Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM)

DESIGN AND ANALYSIS REAR DIFFUSER FOR HEV MODEL

MOHD SYAZRIN BIN SOPNAN

A report is submitted in partial fulfillment
of the requirements for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

NOVEMBER 2008

SUPERVISOR DECLARATION

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

Signature	:	
Supervisor	:	DEVARAJAN A/L RAMASAMY
Date	:	NOVEMBER 2008

Signature	:	
Panel	:	EN. MOHD YUSOF BIN TAIB
Date	:	NOVEMBER 2008

STUDENT DECLARATION

I declare that this thesis entitled "Design and Analysis Rear Diffuser for HEV Model" is the result of my own research except as cited in the reference. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ....
Name : MOHD SYAZRIN BIN SOPNAN
Date : NOVEMBER 2008

DEDICATION

I would like to show my expression and gratitude to Allah Subhanahu wa Ta'aalaa whose guidance, help and grace was instrumental in making this humble work become a reality. To my beloved parent, Mr Hj Sopnan Bin Hj Ponin and Pn. Siti Zawiah Binti Hj Habib and to all by sibling and friends. Also to all staff in Faculty of Mechanical Engineering from University Malaysia Pahang especially to my supervisor Mr. Devarajan A/L Ramasamy.

ACKNOWLEDGEMENTS

Syukur Alhamdulillah, endless thank you to the Almighty for the strength that has been given to the author as to complete this work successfully. First of all I would like to express my gratitude and very high appreciation to all those who give me possibility to complete this report.

A special thanks to my supervisor, Mr. Devarajan A/L Ramasamy for his guidance, suggestion, continuous encouragement and spent a lot of time for me in order to complete my project and writing this report for the past one year. Also to other lecturers that have shared their opinions and experiences in making this project possible.

Last but not least, my sincere gratitude to all my friends especially to my mate, housemate and also to all my course mate who spending their time in giving me a motivation, moral support and help whenever I need during the process for my project.

Thank you very much to all of you.

ABSTRACT

The performance and handling of automobile are significantly affected by its aerodynamic properties. One of the main causes of aerodynamic is about drag force and lifting force. This will influence all the aspect of the vehicles such as overall performance, fuel consumption, safety and stability. The ground effect uses the airflow between the rod surface and the vehicle underbody to create a downforce. Rear diffuser will produce low pressure in causing overall lift to be lowered down to negative values. The negative value is occurring depends on the diffuser cross-section area. Rear diffuser will reduce the flow separation at the rear that causing the turbulent airflow. The wake region also will be reduced and this will make the drag force that produce at the rear vehicle reduced. By the lower drag force will contribute to the lower fuel consumption. The task was done by doing a Computational Fluid Dynamic (CFD) analysis for expected vehicle speed of 40-140 km/h. A drag force was found based on Finite Element Method (FEM) based on inputs from CFD. This force was calculated to produce the drag coefficient of the model as a whole. The approach needed to justify the amount of drag that can be reduced by addition of a rear diffuser as compared to the model without a diffuser. In an aerodynamic field, the main important thing to get the stability, performance and good fuel consumption is to design a vehicle with low C_D. The reduction of lift and flow separation is the key results that will be a point of discussion. This project is to get an overall comparison of the velocity and pressure distribution before and after the rear diffuser is added.

ABSTRAK

Ciri-ciri aerodinamik adalah sangat mempengaruhi akan prestasi dan kawalan sesebuah kenderaan. Salah satu kesan penyebab akan aerodinamik adalah geseran atau “drag force” dan daya tujahan. Ini akan mempengaruhi kesemua prestasi, penggunaan minyak, keselamatan, dan kestabilan sesebuah kenderaan. Kesan dasar pengaliran udara diantara permukaan jalan dengan bawah badan kenderaan akan menghasilkan daya tujahan ke bawah. Diffuser akan menghasilkan tekanan yang rendah dan ini akan menyebabkan tujahan ke atas menjadi rendah sehingga menjadi nilai negatif. Nilai negatif ini terhasil berdasarkan luas permukaan diffuser tersebut. Diffuser juga akan menghasilkan peyebaran pengaliran udara yang kurang di belakang kerana ini akan menghasilkan pegaliran udara yang bergelora. Kawasan olak di belakang juga akan berkurangngan dan ini akan menjadikan daya geseran yang terhasil di bahagian belakang kenderaan berkurangan. Dengan nilai daya geseran yang rendah, ia akan membantu dalam penggunaan minyak yang rendah. Tugasan ini dimulakan dengan menggunakan kelajuan yang telah ditetapkan pada 40 km/j hingga 140 km/j dengan menggunakan analisis Computational Fluid Dynamic (CFD). Daya geseran akan didapati apabila menggunakan perisian Finite Element Method (FEM) berdasarkan maklumat daripada CFD. Nilai daya ini akan digunakan untuk mengira pekali geseran keseluruhan model kereta tersebut. Nilai pengurangan geseran yang terhasil daripada diffuser diperlukan untuk menbenarkan pembezaan di antara model tanpa diffuser. Di dalam aspek aerodinamik, kestabilan, prestasi dan penggunaan minyak amat penting untuk menghasilkan kenderaan yang rendah C_D. Pengurangan tujahan dan peyebaran udara adalah kunci utama di dalam perbincangan. Projek ini akan mendapatkan perbezaan berdasarkan pegaliran angin dan tekanan sebelum dan selepas diffuser dipasangkan.

TABLE OF CONTENTS

	Page
TITLE	i
SUPERVISOR DECLARATION	ii
STUDENT DECLARATION	iii
DEDICATION	iv
ACKNOWLEDGEMENTS	v
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	xi
LIST OF FIGURES	xii
LIST OF SYMBOLS	xv
LIST OF ABBREVIATION	xvi
LIST OF APPENDICES	xvii

CHAPTER 1 INTRODUCTION

1.1	Background	1
1.2	Problem Statement	2
1.3	Objectives	2
1.4	Scopes of Study	2

CHAPTER 2 LITERATURE REVIEW

2.1	Automotive Aerodynamics	3
2.2	Aerodynamic Force	4
	2.2.1 Forces	4
	2.2.2 Aerodynamic Lift	5

2.3	Aerodynamic Pressure	6
2.4	Aerodynamic Drag	8
2.5	Drag Coefficient	8
2.6	Air Flow around the Vehicle	10
	2.6.1 External Flow	11
2.7	Rear Diffuser	12
2.8	Dynamic Fluid Properties	14
	2.8.1 Air Density Properties Related To Vehicle	14
	2.8.2 Air Viscosity Properties Related To Vehicle	15
2.9	Effected of Viscosity	16
	2.9.1 Separation Point	16
	2.9.2 Friction Drag	17
2.10	Reynolds Number	18
2.11	Fuel Consumption	19

CHAPTER 3 METHODOLOGY

3.1	Introduction	21
3.2	Problem Solving	24
	3.2.1 Literature Study	24
	3.2.2 Identify Project Objectives	24
	3.2.3 Measurement the Rear Diffuser	25
	3.2.4 Dimension of the HEV Model	26
	3.2.5 3-D Car Modeling	27
	3.2.6 Sketching Applying For Model Improvement	28
	3.2.7 CFD and FEM Simulation	29

CHAPTER 4 RESULT AND DISCUSSION

4.1	Introduction	35
4.2	Drag Coefficient	36
4.3	Aero Power	39

4.4 Percentage Reduction 41

4.4.1 Percentage of Drag Coefficient Reduction 42

4.4.2 Percentage of Aero power Reduction 43

4.5 Simulation Result 43

4.5.1 Simulation of External Pressure Distribution 43

4.5.2 Simulation of External Velocity Distribution 46

4.5.3 Simulation of External Velocity Rear Region 48

4.5.4 Simulation of External Velocity Underbody 51

4.6 Underbody Analysis 54

4.7 Discussion 55

CHAPTER 5 CONCLUSION AND RECOMMENDATION

5.1 Conclusion 56

5.2 Recommendation 57

REFERENCES 58

APPENDICES 60

LIST OF TABLES

Table No.		Page
2.1	Forces acting on moving vehicle	5
4.1	Drag coefficient of vehicle with and without rear Diffuser	37
4.2	Aero power of vehicle with and without rear diffuser	40
4.3	Average values of C_D and P	42

LIST OF FIGURES

Figure No.		Page
2.1	Aerodynamic of bluff bodies	4
2.2	Arbitrary forces and origin of the forces acting on the vehicle	5
2.3	Pressure and velocity gradients in the air flow aver the body	7
2.4	Drag coefficients of various shapes	9
2.5	Flow around a vehicle	12
2.6	Lift and drag coefficient variation with ground clearance for a generic model with underbody diffuser	14
2.7	Distribution of velocity and temperature in the vicinity of a wall	15
2.8	Flow separation of the boundary layer flow at a surface	16
2.9	Effect of separation point at rear car	17
2.10	Determination of the drag of a body (two-dimensional flow)	18
2.11	Graphic depicting representative horsepower requirements versus vehicle speed for a heavy vehicle tractor-trailer truck	19
3.1	Methodology flow chart for PSM 1	22
3.2	Methodology flow chart for PSM 2	23

3.3	Basic view of the rear diffuser from side	25
3.4	Basic view of the rear diffuser from bottom	26
3.5	Rear diffuser on the model	26
3.6	(a) manual drawing from the data sheet, (b) 3-Dimensional drawing	27
3.7	(a) manual drawing from the data sheet, (b) 3-Dimensional drawing	28
3.8	(a) manual drawing from the data sheet, (b) 3-Dimensional drawing	28
3.9	Model view with rear diffuser	29
3.10	(a) CFD boundary condition with streamlines, (b) Computational Domain size	30
3.11	Simulation of analysis type	30
3.12	(a) Project wizard, (b) FloWork input data for the project analysis	31
3.13	(a) Run startup for simulation, (b) Velocity analysis solver for each speed	32
3.14	(a) Second stage of simulation, (b) Load and restraint applying on model	32
3.15	(a) Mesh surface at the model, (b) COSMOSWorks result solver for each speed	33
3.16	CFD Pressure Result Simulation on model	34
4.1	Drag coefficient vs. Speed	38
4.2	Aero power vs. Speed	41

4.3	Comparison of pressure distribution between with and without rear diffuser	43-45
4.4	Comparison of velocity distribution between with and without rear diffuser	46-47
4.5	Comparison of velocity rear region between with and without rear diffuser	48-50
4.6	Comparison of velocity underbody between with and without rear diffuser	51-52
4.7	Underbody pressure analysis	54

LIST OF SYMBOLS

F_L	Lift force
ρ	Density
V	Velocity
V_o	Wind velocity (reversed direction)
A	Area
P	Pressure
P_{atm}	Atmosphere pressure
C_D	Drag coefficient
F_D	Drag force
$^{\circ}$	Degree of angle
l	Length
P_r	Prevailing pressure
T	Temperature
μ	Viscosity
U	Dynamic viscosity
Re	Reynolds Number
%	Percentage
θ	Angle
P	Aero power

LIST OF ABBREVIATIONS

HEV	Hybrid Electrical Vehicle
CFD	Computational Fluid Dynamic
FEM	Finite Element Method
CW	COSMOSWork
FW	COSMOSFloWork
km/h	kilometer per hour
m/s	mile per second
mph	mile per hour
mm	millimeter
kW	kilowatt
N	newton

LIST OF APPENDICES

Appendix		**Page**
A	Gantt Chart For PSM	60
B	Plane Analysis Profile Visualization	62
C	3-D Modeling (Without Rear Diffuser)	64
D	3-D Modeling (With Rear Diffuser)	65

CHAPTER 1

INTRODUCTION

1.1 Background

Aerodynamics is the branch of dynamics that deals with the motion of air and other gaseous fluids and with the forces acting on bodies in motion relative to such fluids. For some classes of racing vehicles, it may also be important to produce desirable downwards aerodynamic forces to improve traction and thus cornering abilities. Most everyday things are either caused by aerodynamic effects or in general obey the aerodynamic laws. For aerodynamic bodies is simplifies procedure may then be devised for the evaluation of the aerodynamic loads. A car driven in a road is affected by aerodynamic forces created. In all these categories, the aerodynamics of such cars is of vital importance. They affect the cars stability and handling. They influence both performance and safety.

1.2 Problem Statement

Most of moving vehicle produces drag especially at rear because of the turbulent air of airflow separation. These will produce the drag force and the reduction of drag is essential for improving performance and fuel consumption. In every vehicle movement, the air flow will go through to the rear underbody of the vehicle. The air flow also will make the drag and friction occur because of the underbody parts and rear side of the boundary effect. This rear diffuser will reduce the drag and friction of air flow at the rear and will increase the down force of the car when moving in high velocity. This project will reduce the drag and increase down force by adding the diffuser at the rear of vehicle in order to reducing fuel consumption and achieve the objectives.

1.3 Objectives

1) To analyze the effect of rear diffuser on vehicle in term of velocity and pressure.
2) To estimate percent reduction and compare the drag coefficient and aero-power of vehicle between with and without rear diffuser.

1.4 Scopes of Study

1) Study on aerodynamics drag reduction by rear diffuser.
2) Redevelop the existing model of rear diffuser with SolidWork.
3) Simulate the model by using Computational Fluid Dynamic (CFD) and Finite Element Method (FEM).
4) To compare the drag for both with and without rear diffuser.

CHAPTER 2

LITERATURE REVIEW

2.1 Automotive Aerodynamics

Aerodynamics is the branch of dynamics that deals with the motion of air and other gaseous fluids and with the forces acting on bodies in motion relative to such fluids. Automotive aerodynamics is the study of the aerodynamics of road vehicles. The main concerns of automotive aerodynamics are reducing drag, reducing wind noise, minimizing noise emission and preventing undesired lift forces at high speeds. For some classes of racing vehicles, it may also be important to produce desirable downwards aerodynamic forces to improve traction and thus cornering abilities [1]. An aerodynamic automobile will integrate the wheel and lights in its shape to have a small surface. It will be streamlined, for example it does not have sharp edges crossing the wind stream above the windshield and will feature a sort of tail called a fastback or Kammback or liftback. It will have a flat and smooth floor to support the venturi or diffuser effect and produce desirable downwards aerodynamic forces. The air that rams into the engine bay, is used for cooling, combustion, and for passengers, then reaccelerated by a nozzle and then ejected under the floor. Most everyday things are either caused by aerodynamic effects or in general obey the aerodynamic laws. For aerodynamic bodies is simplifies procedure may then be devised for the evaluation of the aerodynamic loads.

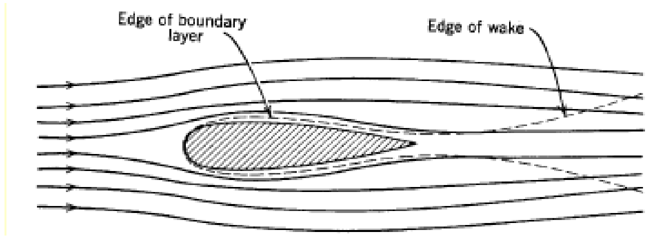


Figure 2.1: Aerodynamic of bluff bodies [1]

A car driven in a road is affected by aerodynamic forces created. The aerodynamics of such cars is of vital importance. They affect the cars stability and handling. They influence both performance and safety.

2.2 Aerodynamic Force

2.2.1 Forces

A body in motion is affected by aerodynamic forces. The aerodynamic force acts externally on the body of a vehicle. The aerodynamic force is the net result of all the changing distributed pressures which airstreams exert on the car surface [3]. Aerodynamic forces interact with the vehicle causing drag, lift, down, lateral forces, moment in roll, pitch and yaw, and noise. These impact fuel economy, and handling. The aerodynamic forces produced on a vehicle arise from two sources that are form (or pressure) drag and viscous friction. Forces and moment are normally defined as they act on the vehicle. Thus a positive force in the longitudinal (x-axis) direction on the vehicle is forward. The force corresponding to the load on a tire acts in the upward direction and is therefore negative in magnitude (in the negative z-direction). The forces also corresponding to the shape on the vehicle part in aerodynamic shape. Figure 2.2 below shown of the vehicle most significant forces [2].

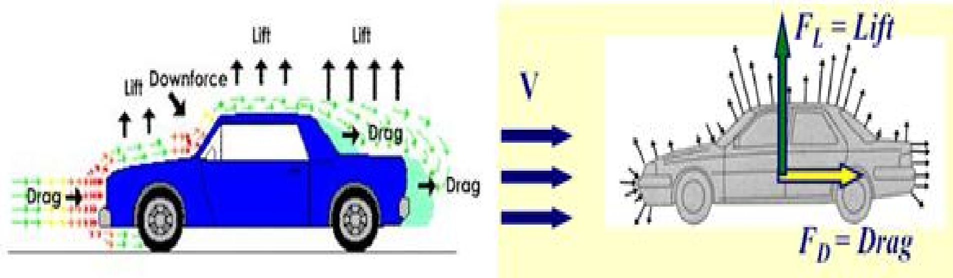


Figure 2.2: Arbitrary forces and origin of the forces acting on the vehicle [2] [3]

Table 2.1: Forces acting on moving vehicle

Direction	Force	Moment
Longitudinal (*x*-axis, +*ve* rearward)	Drag	Rolling moment
Lateral (*y*-axis, +*ve* to the right)	Sideforce	Pitching moment
Vertical (*z*-axis, +*ve* upward)	Lift	Yawing moment

The focus in cars is on the aerodynamic forces of downforce and drag. The relationship between drag and downforce is especially important. Aerodynamic improvements in wings are directed at generating downforce on the car with a minimum of drag. Downforce is necessary for maintaining speed through the corners. [3]

2.2.2 Aerodynamic Lift

The other component, directed vertically, is called the aerodynamic lift. It reduces the frictional forces between the tires and the road thus changing dramatically the handling characteristics of the vehicle. In addition to geometry, lift F_L is a function of density ρ and velocity V. Lift is the net force (due to pressure and viscous forces) perpendicular to flow direction. The aerodynamic drag coefficient equation is [2]:

$$C_F = \frac{F_L}{\frac{1}{2}\rho V^2 A} \tag{2.1}$$

F_L = lift force [N]

ρ = density of the air [kg/m^3]

A = area of the body [m^2]

V = velocity of the body [m/s]

Aerodynamic lift and its proper front-and-rear-axle distribution is one of the key aspects in terms of on-road stability [4]. As long as driving speed is low, below say 100 km/h, lift and pitching moment have only a small effect on the directional stability of a car, even in crosswind. However, at higher speeds this is no longer true, and so recent developments are directed at controlling them.

2.3 Aerodynamic Pressure

The gross flow over the body of a vehicle is governed by the relationship between velocity and pressure expressed in Bernoulli's Equation. Bernoulli's Equation assumes incompressible flow which is reasonable for automotive aerodynamics [2].

$$P_{static} + P_{dynamic} = P_{total} \tag{2.2}$$

$$P_s + \frac{1}{2}\rho V^2 = P_t$$

ρ = density of air [kg/m^3]

V = velocity of air (relative to the car) [m/s]

In equation above, the sum of the forces brings in the pressure affect acting on the incremental area of the body of fluid. The static plus the dynamic pressure of the air will be constant (P_t) as it approaches the vehicle. At the distance from the vehicle the static pressure is simply the ambient, or barometric, pressure (P_{atm}). The dynamic pressure is produced by the relative velocity, which is constant for all streamlines approaching the vehicle. As the flow approaches the vehicle, the streamlines split, some going above the vehicle and others below. By inference, one streamline must go straight to the body and stagnate (impinging on the bumper of the vehicle). At that point the relative velocity has gone to the zero. This will make the static pressure observed at that point on the vehicle. Figure 2.3 below showing flow over a cylinder that it affects is most same to the vehicle [2].

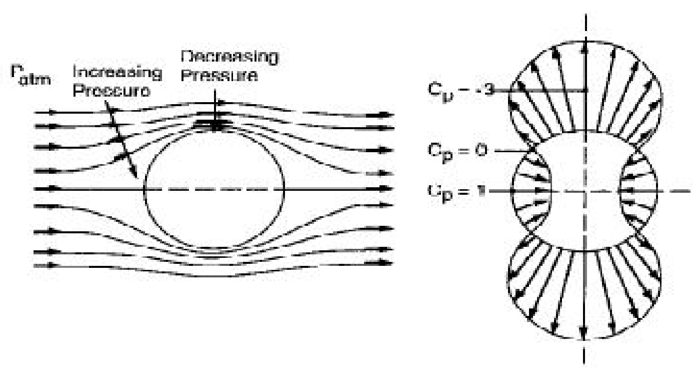


Figure 2.3: Pressure and velocity gradients in the air flow aver the body [2]

The static pressure will distribute along the body of a car. The pressures are indicated as being negative or positive with to the ambient pressure some distance from the vehicle. A negative pressure is developed at the front edge of the hood as the flow rising over the front of the vehicle attempts to turn and follow horizontally along the hood. Near the base of the windshield and cowl, the flow must be turned upward, thus the high pressure is experienced. Over the roof line the pressure goes negative as the air flow tries to follow the roof contour [2].

2.4 Aerodynamic Drag

The component of the resultant aerodynamic force which opposes the forward motion is called the aerodynamic drag. The aerodynamic drag affects the performance of a car in both speed and fuel economy as it is the power required to overcome the opposing force. In order to explain the aerodynamic drag, there have two forces that are the frontal pressure and the rear vacuum [5].

Frontal pressure is caused by the air attempting to flow around the front of the car. As millions of air molecules approach the front part of the car, they begin to compress, and in doing so raise the air pressure in front of the car. Rear vacuum or wake is caused by the "hole" left in the air as the car passes through it. This empty area is a result of the air molecules not being able to fill the hole as quickly as the car can make it. The air molecules attempt to fill in to this area, but the car is always one step ahead [5].

In every moving vehicle, the drag will produce in every surface of the vehicle. The drag is due in part to friction of the air on the surface of the vehicle, and in part to the way the friction alters the main flow down the back side of the vehicle. Drag is the largest and most important aerodynamic force encountered by passenger cars at normal highway speeds. The overall drag on a vehicle derives from contributions of many sources. For the vehicle, the drag produced from the body (forebody, afterbody, underbody and skin friction). The major contributor is the afterbody because of the drag produced by the separation zone at the rear. It is in area that the maximum potential for drag reduction is possible. [2]

2.5 Drag Coefficient

The aerodynamic drag is the focus of public interest in vehicle aerodynamics. It is and even more so it's non-dimensional number of C_D, the drag coefficient has almost become a synonym for the entire discipline. Performance, fuel economy, emissions, and top speed are important attributes of a vehicle because they represent decisive sales arguments, and they all are influenced by drag. Drag coefficient (C_D) is a commonly published rating of a car's aerodynamic smoothness, related to the shape of the car. Multiplying C_D by the car's frontal area gives an index of total drag. The result is called drag area, and is listed below for several cars. The width and height of curvy cars lead to gross overestimation of frontal area. The aerodynamic drag coefficient equation is [2]:

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A} \qquad (2.3)$$

F_D = drag force [N]

ρ = density of the air [kg/m^3]

A = area of the body [m^2]

V = velocity of the body [m/s]

The drag coefficient varies over a board range with different shapes. Figure 2.4 below shows the coefficients for a number of shapes. In each case it is presumed that the air approaching the body has no lateral component. The simple flat plate has a drag coefficient of 1.95. This coefficient means that the drag force is 1.95 times as large as the dynamic pressure acting over the area of the plate.

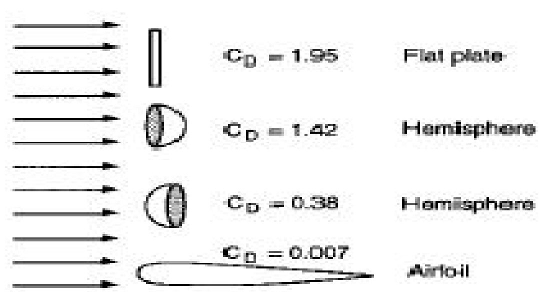


Figure 2.4: Drag coefficients of various shapes [2]

In contrast, with the mush better aerodynamic design of cars, their drag coefficient is not as sensitive to yaw angle because the flow will not separate so readily. Normally, the drag coefficient increase by 5% to 10% with yaw angles in the range typical of on-road driving for passenger cars. The different of yaw angle will influence on the drag coefficients of several different types of vehicles [2].

2.6 Air Flow around the Vehicle

Under calm conditions and no traffic, vehicles travel through still air, hence the relative air flow they experience has no turbulence, is unsteadily relative to the vehicle and has the same magnitude as the vehicle speed relative to the road. If an atmospheric wind or air flow is present, generally a yaw angle is created because the flow is not aligned with the centerline of the vehicle and thus the air speed of a vehicle experiences is not the same as the road speed [6]. The flow processes to which a moving vehicle is subjected fall into two categories:

1) Flow of air around the vehicle.
2) Flow of air through the vehicle's body.

The two categories of these flow fields are closely related. For example, the flow of air through the engine compartment depends on the flow field around the vehicle. Both flow fields must be considered together. On the other hand, the flow processes within the engine and transmission are not directly connected with these two categories of flow. They are not called aerodynamics, and are not treated here [8]. Flow separations may appear in different locations on vehicles with more angular geometries, and fairing dominated flows can exist on a variety of road vehicles. The main aspect of this flow field is the formation of two concentrated side edge vortices which dominate the nearby flow field. Those two vortices induce a large velocity on the plate creating strong suction forces which considerably increase the lift of the flat-plate wing. Typical pattern of flow-separation frequently found on three-box-type sedans. In this case a separated bubble, with locally recirculation flow, is observed in the front, at the break point between the bonnet and the windshield. The large angle created between the rear windshield and trunk area results in a second, similar flow-recirculation area [9] [10].

2.6.1 External Flow

The flow around a vehicle is responsible for its directional stability as well as straight line stability, dynamic passive steering, and response to crosswind depend on the external flow field. Furthermore, the outer flow should be tuned to prevent duplets of rain water from accumulating on windows and outside mirrors, to keep headlights free of dirt, to reduce wind noise, to prevent the windshield wipers from lifting off, and to cool the engine's oil pan, muffler, and brakes, etc.

The external flow around a vehicle is shown in Figure 2.5 below. In still air, the undisturbed velocity, V is the road speed of the car. Provided no flow separation takes place, the viscous effects in the fluid are restricted to a thin layer of a few millimeters thickness, called the boundary layer. Beyond this layer the flow can be regarded as in viscid, and its pressure is imposed on the boundary layer. Within the boundary layer the velocity decreases from the value of the in viscid external flow at the outer edge of the boundary layer to zero at the wall, where the fluid fulfills a no-slip condition. When the flow separates, the boundary layer is "dispersed" and the flow is entirely governed by viscous effects. The character of the viscous flow around a body depends only on the body shape and the Reynolds number. For different Reynolds numbers entirely different flows may occur for one and the same body geometry. Thus the Reynolds number is the dimensionless parameter which characterizes a viscous flow [8] [9].

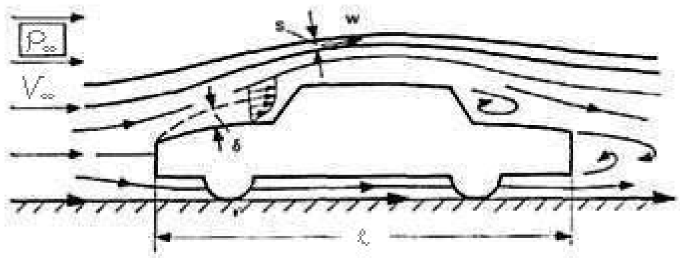


Figure 2.5: Flow around a vehicle [8]

Flows around geometrically similar bodies are called "mechanically similar if the Reynolds number has the same value for different body lengths, l air speeds, V and fluid properties. For models smaller than the original vehicle it is necessary to increase the velocity, V but the value must remain in the low subsonic regime. This means that it is not possible to perform tests on very small models in supersonic [8] [11].

2.7 Rear Diffuser

The aft part of a car underbody can be a diffuser. It tries to connect the underbody to the back without producing turbulences as shown in figure 2.10 below. Diffuser is a shaped section of the car underbody which improves the car's automotive aerodynamics properties. Rear diffuser is known as a venture. It designed to create a low pressure or vacuum area under the rear of the car using a physics principle known as the venturi effect. In essence, a diffuser is like an air channel that is designed to accelerate the air out from underneath the back of the car and help minimize underbody wind turbulent and create negative lift at the rear of the car. It will enhance the transition between the high-velocity airflow underneath the car. To help channel the high speed air flow coming from underneath the car, diffusers redirects the high speed air flow to transition smoothly transition into the low pressure vacuum in the rear.

It works by providing a space for the underbody airflow to decelerate and expand so that the boundary between the car's airflow and "external" airflow is less turbulent, and it also provides a degree of "wake infill" (the wake being a turbulent area of low pressure that is caused by the passage of the vehicle through the air; this can cause pressure drag). Diffusers also work in conjunction with other aerodynamic components of a car to help produce downforce. Because a car displaces air as it moves forward, the air molecules in the rear of the car get disrupted causing a vacuum of low pressure air. The side and the roof end in a sharp edge, so that their pressure does not increase. Because the pressure in the back tends to equilibrate, the pressure below the car is lower than on the side and the roof of the car otherwise make the downforce of the car increase. A diffuser also serves to eject air out from the underside of the car. This pulling action increases the velocity of the air below the car, so that the more slowly moving air above the car will push the car into the ground. Aerodynamically, the diffusers achieve that is minimizing pressure under the car.

It works when the air enters towards the front of the car it accelerates and reduces pressure. The diffuser then eases this "high velocity" air back to normal velocity and also helps fill in the area behind the car making the whole underbody a more efficient downforce producing device by reducing drag on the car and increasing downforce. A rear diffuser helps drive the under-car flow by exposing it to the turbulent low-pressure wake region behind the car, using this low pressure to suck the flow out. The downforce generated by the underside is strictly related to the velocity and the mass flow of the air blowing underneath the car. A bumpy or irregular, i.e. not flat, underbody induces flow separation and a high thickness boundary layer which slow down the flow. This means that in order to make the most of its downforcing capability, the underbody of a car needs to be flat or regular. Then diffusers added at the rear end of the car help to increase air speed and mass flow under the car still further. While at low speeds, typically in cornering, downforce does more than drag to improve cornering speed while the opposite happens at high speeds (straight line), where drag low is needed to improve top speed performance and reduce fuel consumption.

Diffuser is effective in generating large amounts of downforce by increasing air speed underneath, thereby reducing pressure. Since this low-pressure region acts on a large surface area, plenty of downforce can be generated. The goal of a diffuser is to provide a smooth transition from low pressure back to atmospheric, ideally without separating or stalling the flow.

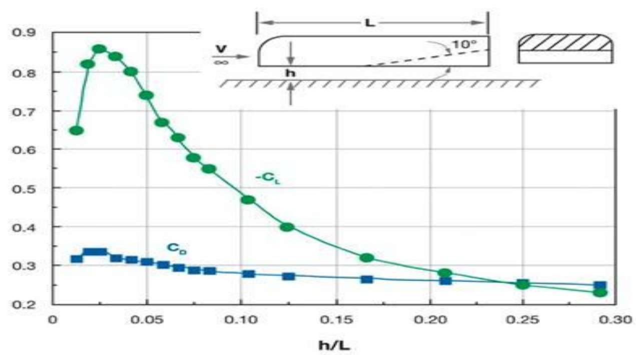


Figure 2.6: Lift and drag coefficient variation with ground clearance for a generic model with underbody diffuser [Cooper et al. (1998), Re_L=0.83x106, rolling ground]

2.8 Dynamic Fluid Properties

2.8.1 Air Density Properties Related To Vehicle

The air density is variable depending on temperature, pressure, and humidity conditions. The air density must be expressed as mass density, obtained by dividing by the acceleration of gravity. Density at other conditions can be estimated for the prevailing pressure, P_r and temperature,T_r conditions by the equation below [2].

The highest speeds achieved by land-vehicles during record attempts are on the order of the speed of sound which is for air, = 330 m/s = 1225 km/h = "65.6 mph. In the flow field of a body exposed to such a free stream the compressibility of the air is very important. On the other hand, most vehicles including racing cars are operated at speeds which are lower than one-third of the speed of sound. For this speed range the variations of pressure and temperature in the flow field are small as compared to free-stream values, and therefore the corresponding changes of density can be neglected. Thus the fluid can be regarded as incompressible [2].

2.8.2 Air Viscosity Properties Related To Vehicle

The air will have its own viscosity when the car is moving through the air surrounding. Viscosity is caused by the molecular friction between the fluid particles. It relates momentum flux to velocity gradient, or applied stress to resulting strain rate. According to Newton's law for the flow parallel to a wall, the shear stress, τ is proportional to the velocity gradient *du/dy*. The constant factor *U* is a property of the fluid called dynamic viscosity. In general its value depends on the temperature. Often the quotient $\nu = \mu/\rho$ is used, which is called kinematics viscosity and which depends on pressure and temperature. For incompressible fluids, only temperature dependence exists for ν and μ. The viscosity of a real fluid is the physical reason for the occurrence of a friction drag in the presence of a velocity gradient at a wall. It is same case in surface contact of the vehicle as when the vehicle is moving in any velocity.

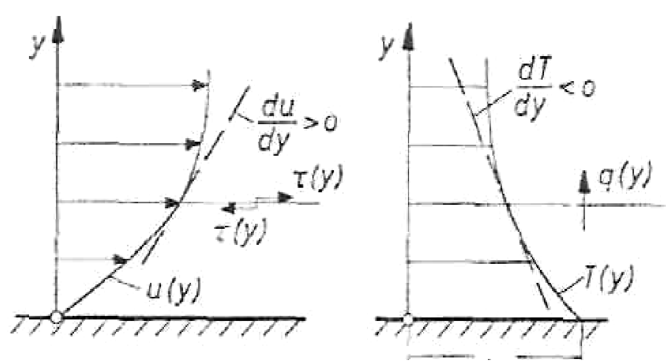


Figure 2.7: Distribution of velocity and temperature in the vicinity of a wall [12]

2.9 Effected of Viscosity

The layer of reduced velocity in fluids, such as air and water, that is immediately adjacent to the surface of a solid past which the fluid is flowing. The layer of fluid immediately adjacent to a body in a fluid. Adhesion between fluid particles and the body surface create viscous stresses which increase drag. The boundary layer is subjected to shear forces. A range of velocities is established across the boundary layer, from zero to maximum. Flow in boundary layers is more easily described mathematically than is flow in the free stream. Boundary layers are thinner at the leading edge of an car fairing and thicker toward the trailing edge such boundary layers generally have laminar flow in the leading (upstream) portion and turbulent flow in the trailing (downstream) portion. The boundary layer distorts surrounding nonviscous flow. It is a phenomenon of viscous forces. This effect is related to the Reynolds number. Despite the thinness of the boundary layer at the wall, the viscous flow within it has a strong influence on the development of the whole flow field. The occurrence of drag in two-dimensional incompressible flow can be explained only by these viscous effects.

2.9.1 Separation Point

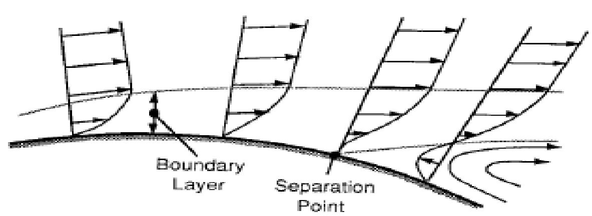


Figure 2.8: Flow separation of the boundary layer flow at a surface [2]

As the flow turns again to follow the body, the pressure again increases. The increasing pressure acts to decelerate the flow in the boundary layer, which causes it to grow in thickness. Thus it produces what is known as an “adverse pressure gradient”. At some points the flow near the surface may actually be reversed by the action of the pressure as illustrated in figure 2.8 above. It can be seen that between forward and reverse flow a dividing streamline leaves the wall. This phenomenon is called separation. The point where the flow stops is known as the “separation point”. At this point, the main stream is no longer ‘attached’ to the body but is able to break free and continue in a more or less straight line. Because it tries to entrain air from the region behind the body, the pressure in this region drops below the ambient [2].

Laminar and turbulent boundary layer flow strongly depends on the pressure distribution which is imposed by the external flow. Turbulent boundary layers can withstand much steeper adverse pressure gradients without separation than laminar boundary layers. This is because the turbulent mixing process leads to an intensive momentum transport from the outer flow towards the flow adjacent to the wall. For a pressure decrease in flow direction there exists no tendency to flow separation.

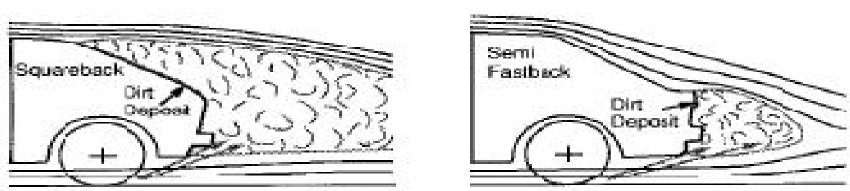


Figure 2.9: Effect of separation point at rear car [2]

2.9.2 Friction Drag

The pressure in the separation region is below that imposed on the front of the vehicle, and the difference in these overall pressure forces is responsible for ‘from drag’. The drag forces arising from the action of viscous friction in the boundary layer on the surface of the car is the “friction drag”. In the boundary layer, the velocity is reduced because of friction [2]. In a viscous fluid a velocity gradient is present at the wall. Due to molecular friction a shear stress acts everywhere on the surface of the body as indicated in Figure 2.10 below .The integration of the corresponding force components in the free-stream direction leads to the so-called friction drag. In the absence of flow separation, this is the main contribution to the total drag of a body in two-dimensional viscous flow.

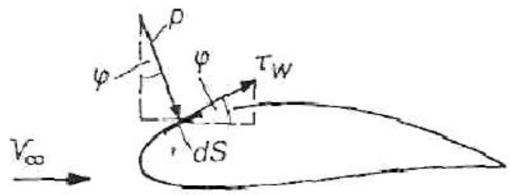


Figure 2.10: Determination of the drag of a body (two-dimensional flow)

2.10 Reynolds Number

$$Re = \frac{\rho\, l\, \nu}{\mu} \tag{2.4}$$

ρ = density of the fluid

l = characteristic of length

ν = velocity of the body in the fluid

μ = viscosity of the fluid

The general factors of drag, aerodynamic friction, are density and viscosity of the fluid, air being considered a fluid. The definition of Reynolds Number is shown at above [9]. The Reynolds Number for a body that is large in size and slow in velocity could produce an equivalent Reynolds Number of a very small object that travels with a high velocity. This seems like it isn't possible logically until we evaluate the definition Reynolds Number and see that it is. At high Reynolds numbers, typical of full-sized car, it is desirable to have a laminar boundary layer. This results in a lower skin friction due to the characteristic velocity profile of laminar flow. At lower Reynolds numbers, such as those seen with model car, it is relatively easy to maintain laminar flow. This gives low skin-friction, which is desirable. However, the same velocity profile which gives the laminar boundary layer its low skin friction also causes it to be badly affected by adverse pressure gradients. As the pressure begins to recover over the rear part of the rear hood, a laminar boundary layer will tend to separate from the surface. Such separation causes a large increase in the pressure drag, since it greatly increases the effective size of the rear hood section [9].

2.11 Fuel Consumption

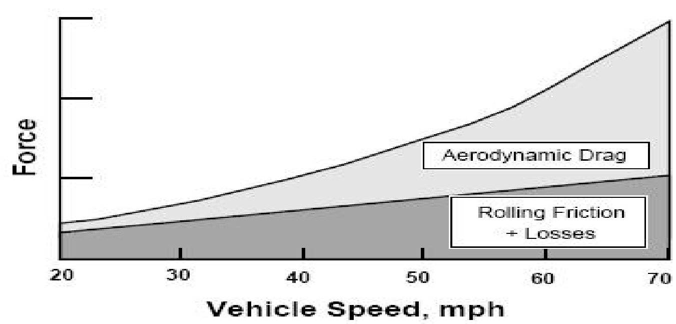


Figure 2.11: Graphic depicting representative horsepower requirements versus vehicle speed for a heavy vehicle tractor-trailer truck [13]

Aerodynamics devices will reduce the drag and also will improve the fuel economy of moving vehicle. Drive-train losses, rolling friction, and aerodynamic drag are contributed for the primary resistance forces that influenced of energy usage. The chart of figure 2.11 shows that as vehicle speed is increased the force required to overcome both aerodynamic drag and rolling friction increases. However, the rate of increase in aerodynamic drag with increasing vehicle speed is much greater than that for rolling friction such that at approximately 50 mph the force directed at overcoming aerodynamic drag exceeds that required to overcome rolling friction [13]. These data do not take into account several operational and environmental factors that can have a dominating effect on the aerodynamic drag of moving vehicle. The chart also shows that if the average speed approaches 30 mph then it is nearly impossible to achieve a 10% improvement in fuel economy through aerodynamic drag reduction. It shows that in higher speed will achieve the 10% of improvement in fuel economy.

A number of additional factors such as interference from other vehicles, atmospheric effects, and road conditions are the factors that must be addressed when developing technologies to improve the fuel economy of vehicles. These efforts of adding the aerodynamics devices have produced reductions in the aerodynamic drag of 30%, for an operating speed of 60 mph, with corresponding improvements in fuel economy approaching 15%. An assessment of the aerodynamic drag indicates that the reduction in aerodynamic drag at 60 mph would be closer to 20%, which corresponds to a 10% improvement in fuel economy.

This improvement in fuel economy correlates to an equivalent drag reduction of approximately 30% with a corresponding drag coefficient of 0.45. Note, the aerodynamic drag reduction and associated fuel savings also result in a measurable reduction in exhaust emissions that is equivalent to the percent reduction in fuel usage. The application of the subject aerodynamic drag reduction technologies to trucks and similar high drag vehicles offers additional synergistic benefits such as the ability to use of alternate lower-energy fuels and the use of alternate power sources.

CHAPTER 3

METHODOLOGY

3.1 Introduction

The objective of the project can be achieved by setting the methodology in well. This will make the project is cable of complete on time. This chapter will explain the detailed about the methodology progress during Final Year Project 1 and 2. This chapter will list all the method that included and relevant to complete the project. The title was given by the supervisor in the beginning of this semester with the title "Design and Analysis Rear Diffuser for HEV Model ". The detailed related literature review was informed acquired the important things in the chapter 2.

The project will analyze the affect of rear diffuser on the car based on the pressure, velocity and drag. Computational Fluid Dynamic (CFD) in COSMOSFloWorks and Finite Element Method (FEM) in COSMOSWorks software will be used to analyze the drag coefficient reduction by rear diffuser. This software will make the objective to be achieved successfully. To solve all the problem, first thing to do is to determine all the flow works with the duration of time, the Gantt chart is a recommended method to use. So that all the flow work with the description of works were carried out to meet the date line.

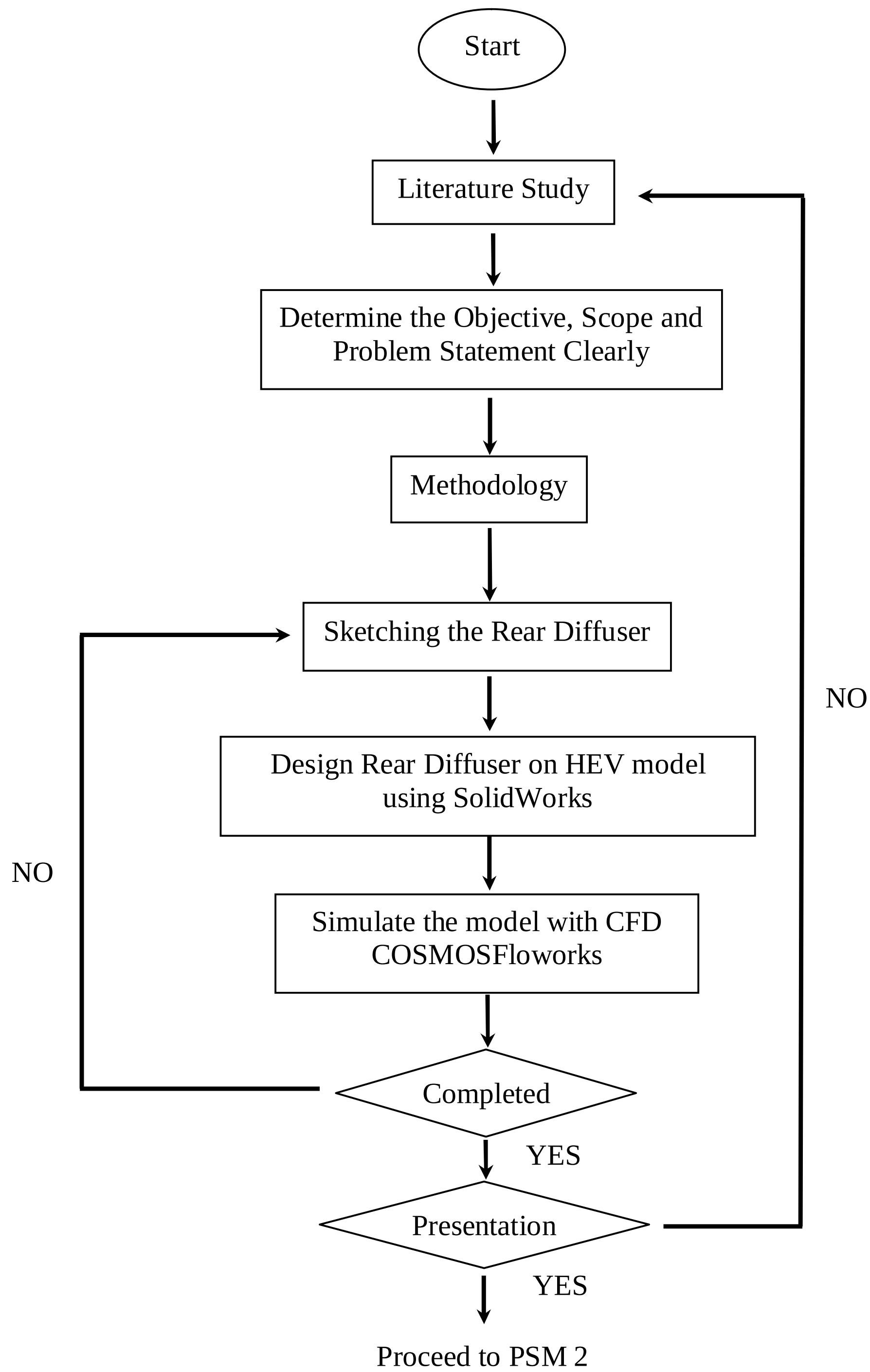


Figure 3.1: Methodology flow chart for PSM 1

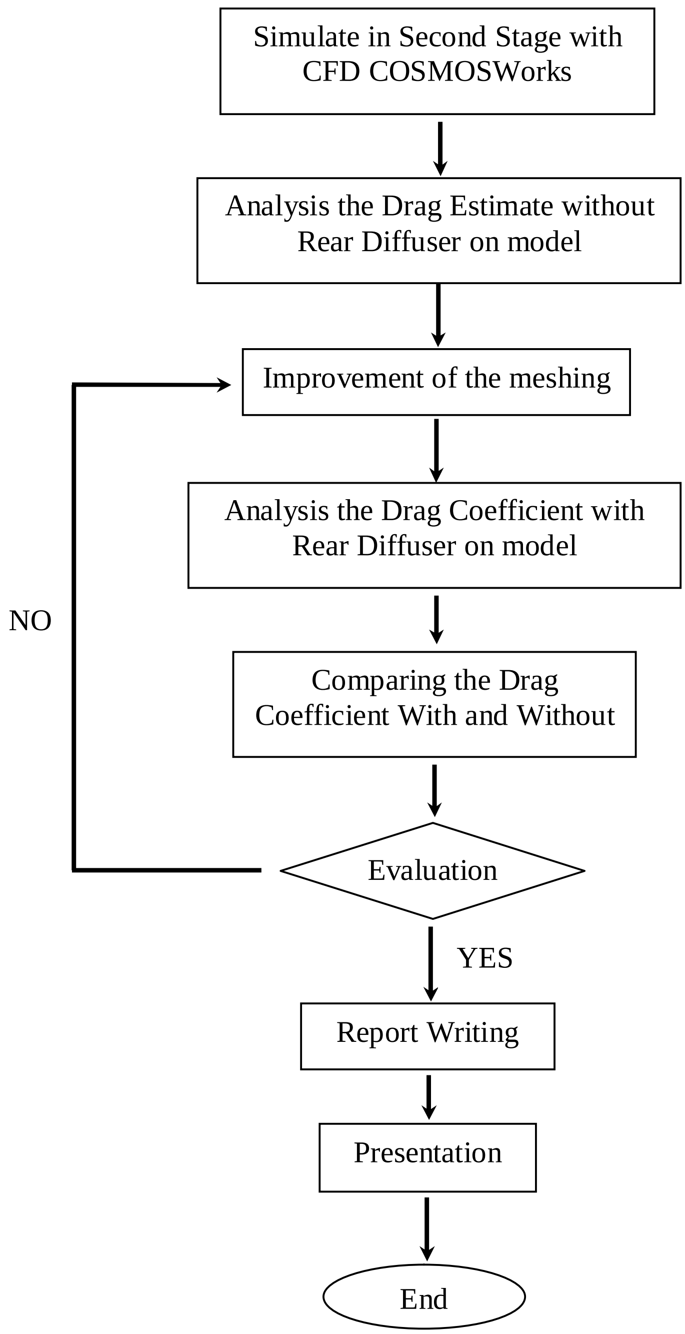


Figure 3.2: Methodology flow chart for PSM 2

3.2 Problem Solving

By referring to this project, the main problem solving is by using the Computational Fluid Dynamics to analyze the flow velocity and pressure distribution around the car body, to complete that there must be a problem solving method or flow to complete that. So, the works are regarding to this solving problem must be more organize. With referring to the methodology flow chart, the detail for each activity can be referring to next sub topics.

3.2.1 Literature Study

This project is start with literature review and research from the internet, company websites, market survey, books and journal about the title. This stage is very important to make a literature study about the basic of the aerodynamics such as a flow characteristic around the vehicles , drag coefficient, lift , pressure distribution and others fluid dynamics requirement. In this part, the detail about the design of the rear diffuser and all the effect about the external flow of the HEV model will be explained.

The literature study was continued from the beginning of this project so that all the latest information will be updated from the time to time. This part also can give the individual to understand what the important things needed before proceed to the next stage.

3.2.2 Identify Project Objectives

Objective is the most important part in this project. By the determination of the objective, the project will clearly see what will be doing from the beginning. The problem that will occur at the rear end of the car must be analyzed. This is very important so that the target objective from the starting can be achieved. The scopes of this project can be done after determine the objective. This will help the project to progress smoothly and can be success.

3.2.3 Measurement the Rear Diffuser

Before the analysis of this project, the rear diffuser model is requiring to get the dimension. From the literature study, the rear diffuser is installing at the underbody of the model. The dimension of the rear diffuser are limited because the space of the boot at the rear model. Figure 3.3 and figure 3.4 below are the dimension of the rear diffuser. By using the calculation, the angle degree that gets for the rear diffuser is about 19.36°.

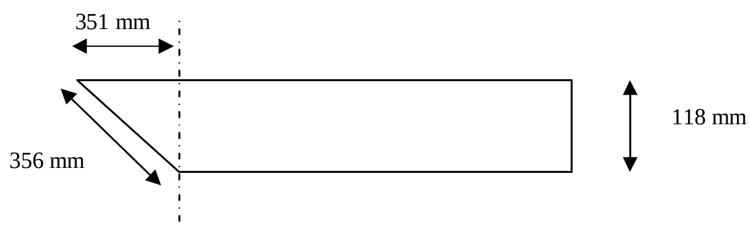


Figure 3.3: Basic view of the rear diffuser from side

$$\frac{sin\,90}{356} = \frac{\sin\theta}{118} \tag{3.1}$$

$\theta = 19.36°$

Equation at above is the mathematic solver for the value of rear diffuser angle. From tip to the tip, the measurement is 848 mm and the chord length is about 351 mm. Span is measured tip to tip and chord is defined as the leading edge to the trailing edge.

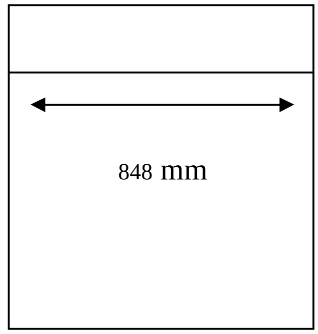


Figure 3.4: Basic view of the rear diffuser from bottom

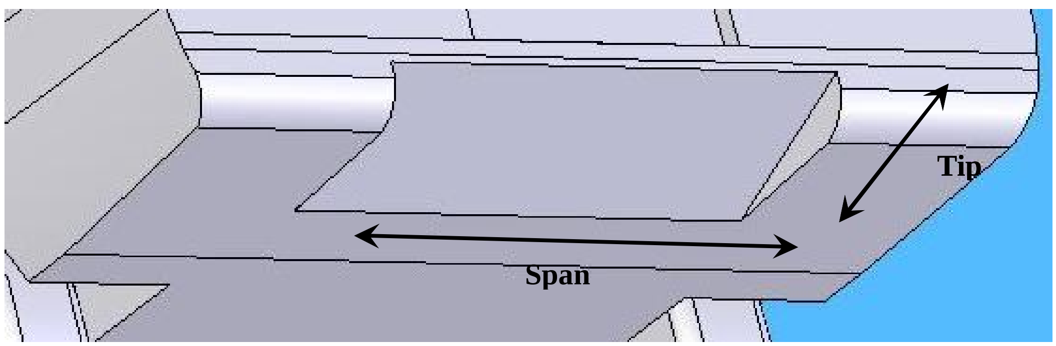


Figure 3.5: Rear diffuser on the HEV model

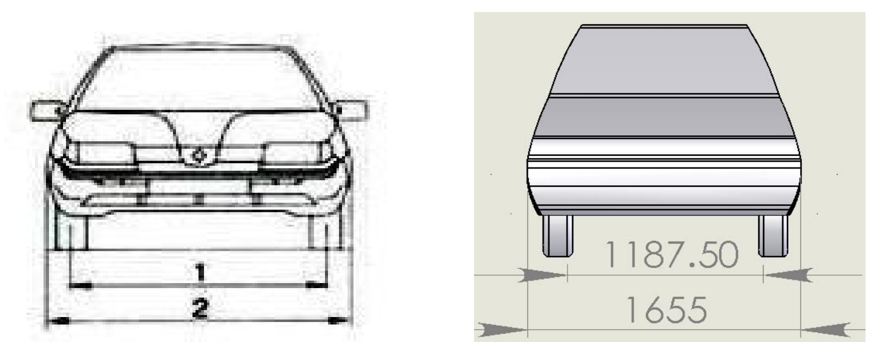
3.2.4 Dimension of the HEV Model

The car model that will be use on this project is basically a Proton Iswara Aeroback. The car model are get from the supervisor, all the dimension are already taken by the previous student that doing the aerodynamic devise on this car model. The dimensions are taken from the Proton Manuals Book and from that dimension, the car was transferred into a 3-D modeling by using the SolidWorks software.

3.2.5 3-D Car Modeling

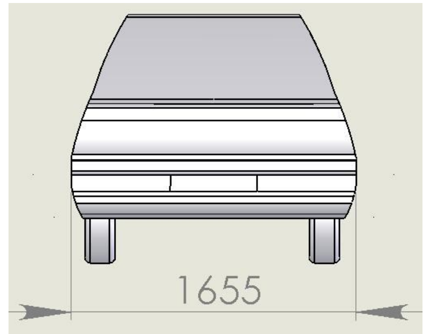
HEV model was modeling and transferred into a 3-D modeling using the 3-D modeling simulation (SolidWork). The model is modeling by referring the true dimension from the Proton Manuals Book. The model of rear diffuser is added to this 3-D model at the rear bumper to complete the CFD simulation. The model dimension and actual vehicle dimension is not quite match. This is because some detailing such as a fender, underbody, front and rear bumper of the vehicle were made by using the assumption that will only fits the model. This is the important things to carry out for the simulation result.

All the part that has been mention at above, the part is modeled together with the body because the project is focusing on the external flow only. The dimension is not accurate from the real measurement from the Proton Manual Book.



(a) Front View (b) Front View

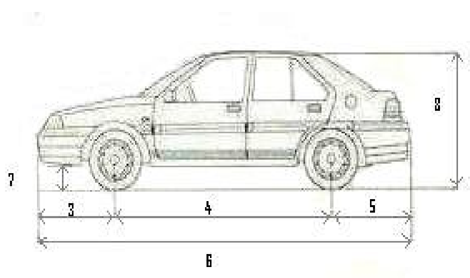
Figure 3.6: (a) manual drawing from the data sheet, (b) 3-Dimensional drawing

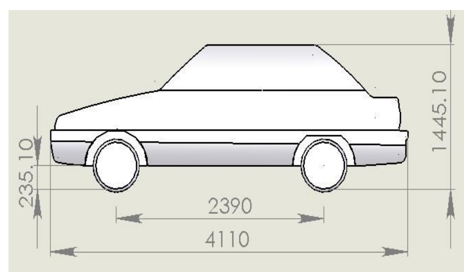


(a) Rear View

(b) Rear View

Figure 3.7: (a) manual drawing from the data sheet, (b) 3-Dimensional drawing





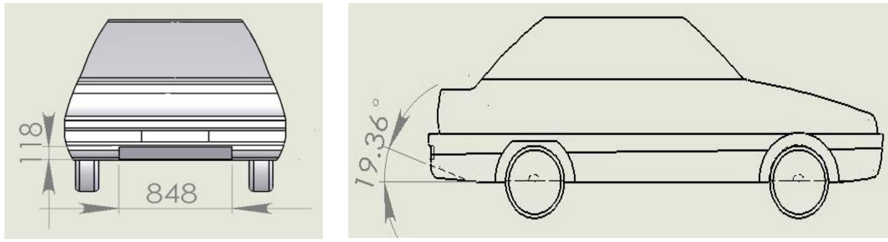
(a) Side View

(b) Side View

Figure 3.8: (a) manual drawing from the data sheet, (b) 3-Dimensional drawing

3.2.6 Sketching Applying For Model Improvement

The dimension of the rear diffuser is installing directly into the HEV model by using the Solidwork software. Figure below is the HEV model with the rear diffuser at the rear side of the model.

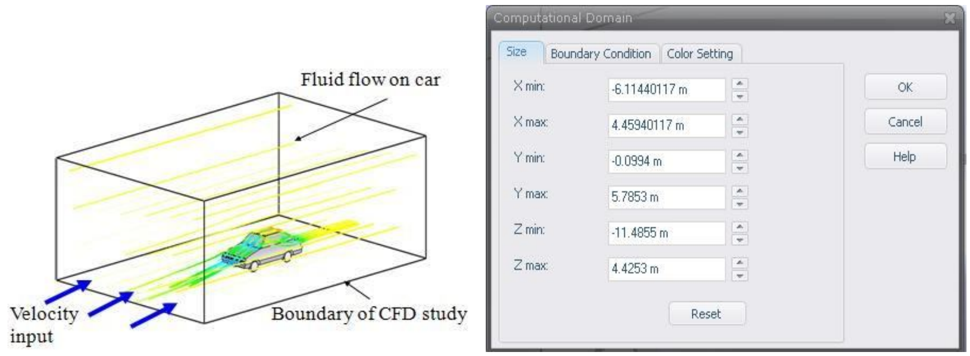


(a) Back View (b) Side View

Figure 3.9: Model view with rear diffuser

3.2.7 CFD and FEM Simulation

After the HEV car modelling on Solidwork, the model will be transfer into a CFD Simulation known as Computational Fluid Dynamics Simulation. The process will make an analysis about the flow velocity and the pressure distribution over the car model body. In this simulation also can determine the laminar and turbulent flow over the car body. The drag can be determined after the amount of the drag force get from the first stage analysis. The HEV car model with rear diffuser is used as the model for the all analysis. With this model, the simulation will be running by using the CFD software. Figure 3.10 below show the boundary condition of the HEV car model with rear diffuser on CFD.



(a) Boundary Condition (b) Computational Domain

Figure 3.10: (a) CFD boundary condition with streamlines, (b) Computational Domain size

This CFD simulation will be used for determine the flow over the car before and after adding the rear diffuser to see the difference. This method of CFD simulation is depending on the model, every little aspect will affect the result. This simulation is focusing on the external flow of the vehicles and the effect of the velocity and pressure distribution after given a desired input. The boundary condition and computational domain of FEM is same with the boundary condition of CFD.

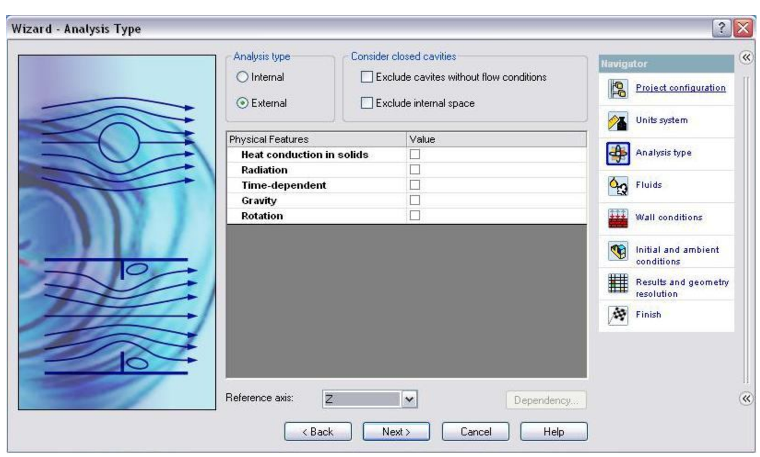
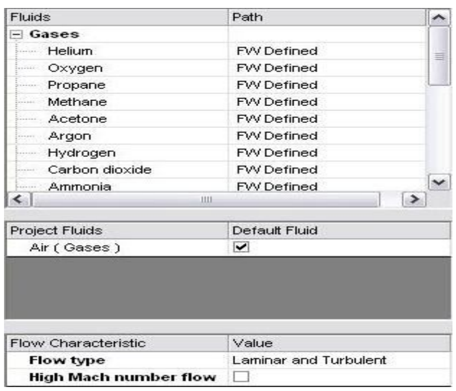
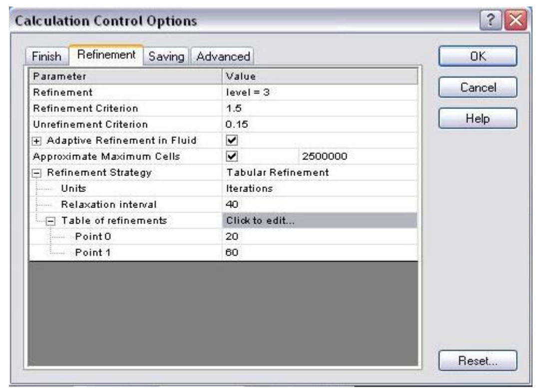


Figure 3.11: Simulation of analysis type

The location of the most laminar and turbulent flow of the vehicles and the most pressure distribution will be shown when a different velocity is given to the vehicles according to the changes from the lower velocity to the maximum velocity. The setting velocities are set 40, 60, 80, 100, 120 and 140 km/h. During this simulation also, all the data will be recorded and the graph of the analysis will be compared after all the simulation is archive.



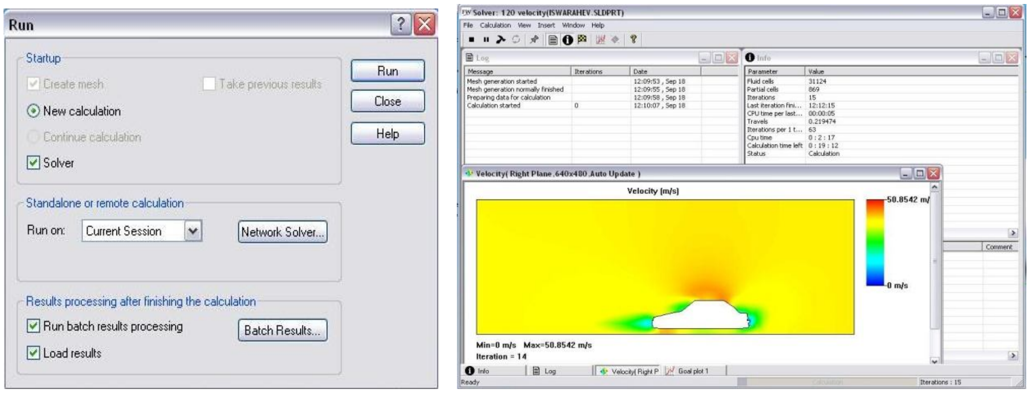
(a) Analysis Fluids



(b) Refinement

Figure 3.12: (a) Project wizard, (b) FloWork input data for the project analysis

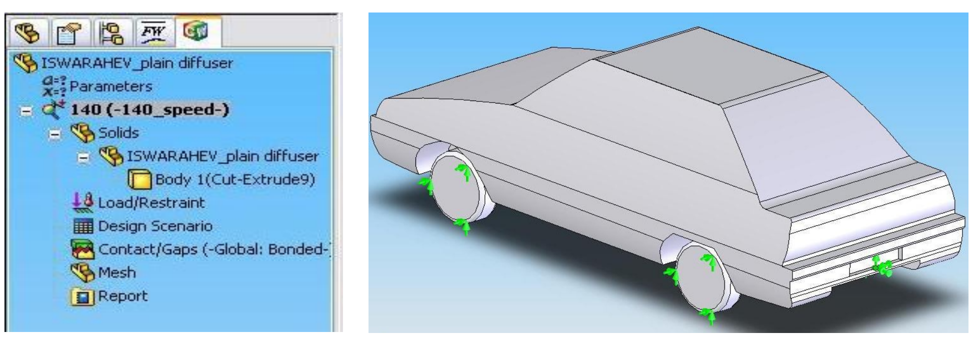
Air is taking as a fluid for project analysis that surrounding the whole car model in the simulation. The flow characteristic value is a laminar and turbulent, these types of flow will be considered for the simulation. For the refinement, the level taking is level 3 and the two points that taking for the table of refinement is 20 and 60. The unit in refinement is iterations. The mesh display is needed to show at COSMOSFloWorks parameter before the running of simulation. The mesh taken is about between course and fine value.



(a) Run Simulation (b) Analysis Solver

Figure 3.13: (a) Run startup for simulation, (b) Velocity analysis solver for each speed

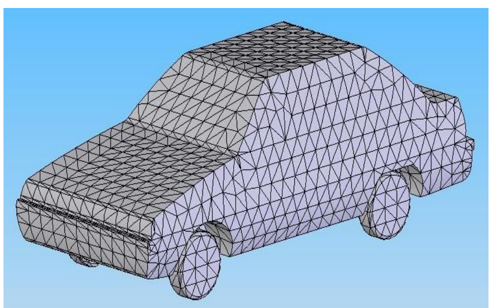
The result from the COSMOSFloWorks will be export into the COSMOSWorks software to obtain the drag force, F_D. From this drag force, the drag coefficient can be estimate.



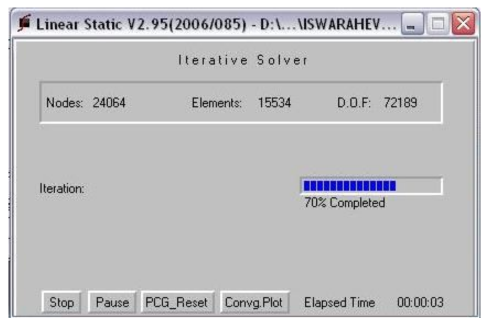
(a) COSMOSWorks study (b) Load and Restraint

Figure 3.14: (a) Second stage of simulation, (b) Load and restraint applying on model

For the second stage analysis using COSMOSWorks, the CW study are been added from the add-ins from the tools. The simulation study in this stage are taken for solid mesh. The porperties of flow or thermal effect are show in term of fluid pressure. This term are be loaded from the COSMOSFloWorks result in first stage. The load and restraints are apply at the model surface. Four flat surface on the model is been restraint in four side of the wheels. Tires are restrict from moving to vertical & sideways and one point is fixed support on horizontal to make body support by longitudinal only. The component are been taken in resultant reaction force with Newton units. The plot type is a fringe. One point at the rear of the model are been fixed for the load simulation.



(a) Mesh Model



(b) Iteration Solver

Figure 3.15: (a) Mesh surface at the model, (b) COSMOSWorks result solver for each speed

Before the COSMOSWorks solver, the model are been meshing with the global size of 150 fine. This created mesh are approaching to the fine meshing for the detail result. The all data that been choose will be export to the COSMOSWorks result before the iteration solver runing. The drag force are estimate from the back point that have been fixed for the load simulation.

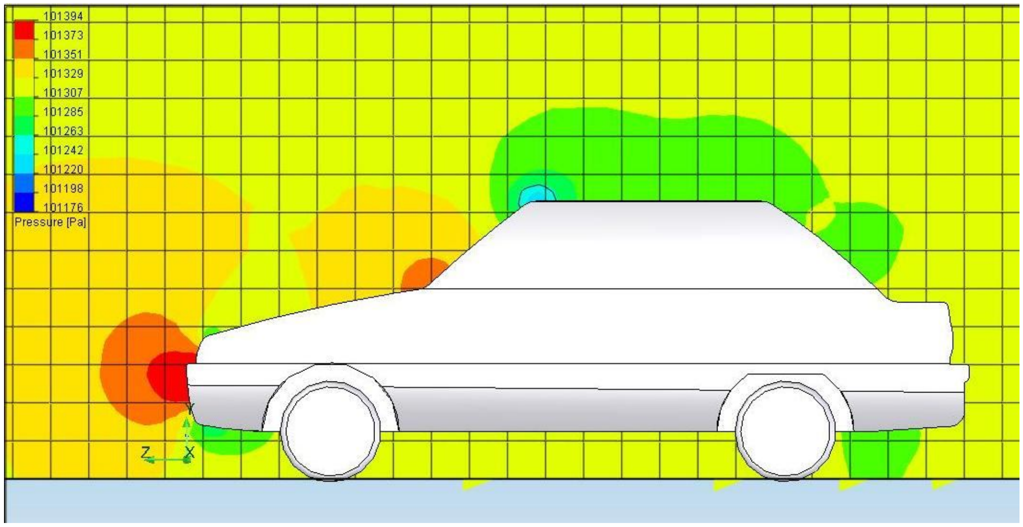


Figure 3.16: CFD Pressure Result Simulation on model

From the figure above, the pressure distribution is show around the model in velocity of 60 km/h. There are different of the pressure at the rear of the car model. The upper side of the rear car pressure is higher than the below side of the rear car. As state at the previous chapter that different pressure can be obtained referring to different velocity. By referring with different velocity, the changing of the laminar and turbulent flow will appear at the back side of the car body. After the analysis of the model in different velocity, the result of the drag coefficient will be different too. By this different of the value, the comparison in term of drag coefficient between model with rear diffuser and without rear diffuser can be done. By the reducing drag coefficient with rear diffuser, the project is complete and the objective are achieve.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

The main objective of this project is about to study the effect of rear diffuser base on pressure and velocity distribution. The effect will be simulated before and after the adding of rear diffuser using the CFD simulation. In automotive design studies, the aerodynamic devices such as rear diffuser are the important part in designing a vehicle. This aerodynamic device will influence of the stability, performance, fuel consumption and others on the vehicle.

Nowadays, there are many new vehicle model are install the rear diffuser device. In this project analysis will show the result when adding the rear diffuser at the rear vehicle and will compare with the vehicle model without the rear diffuser. The adding of rear diffuser is the main characteristics that were taken into consideration in the analysis. By mean, this is also one of the ways how a designer of the most leading world performance car manufacturer produce this rear diffuser as their standard part for the vehicles as their standard kits. This CFD simulation analysis will determine there is any changing and improvement before and after the rear diffuser device are install at the vehicle model.

4.2 Drag Coefficient

The drag coefficient can be calculated by using the drag force equation. From the value of drag force that obtains in FEM COSMOSWork analysis, the drag coefficient, C_D can be determined. The drag coefficient will be calculated with the equation 2.3. The velocity used is from 40 km/h until 140 km/h.

Determine the C_D for model with rear diffuser at velocity 140 km/h. From the data given,

F_D = 584.6 N

ρ = 1.225 kg/m^3

A = 1.833 m^2

V = 38.89 m/s

Solution,

$$C_D = \frac{584.6\ N}{1/2\left(1.225\frac{kg}{m^3}\right)(38.89\frac{m}{s})^2\ (1.833\ m^2)}$$

$$= 0.34421573$$

Determine the C_D for model without rear diffuser at velocity 140 km/h. From the data given,

$F_D = 627.3$ N

$\rho = 1.225$ kg/m^3

$A = 1.833$ m^2

$V = 38.89$ m/s

Solution,

$$C_D \quad = \frac{627.3\ N}{{}^1\!/_2\left(1.225\frac{kg}{m^3}\right)(38.89\frac{m}{s})^2\ (1.833\ m^2)}$$

$= 0.369358$

Table 4.1: Drag coefficient of vehicle with and without rear diffuser

	Drag Force, F_D [N]		Drag Coefficient, C_D	
Velocity Speed, km/h	With Rear Diffuser	Without Rear Diffuser	With Rear diffuser	Without Rear Diffuser
40	39.44	41.96	0.28454868	0.302730
60	95.3	101.8	0.30540032	0.326230
80	169.8	187.4	0.30626499	0.338010
100	268.2	301.3	0.30948676	0.347682
120	391.2	441.1	0.31359996	0.353602
140	584.6	627.3	0.34421573	0.369358

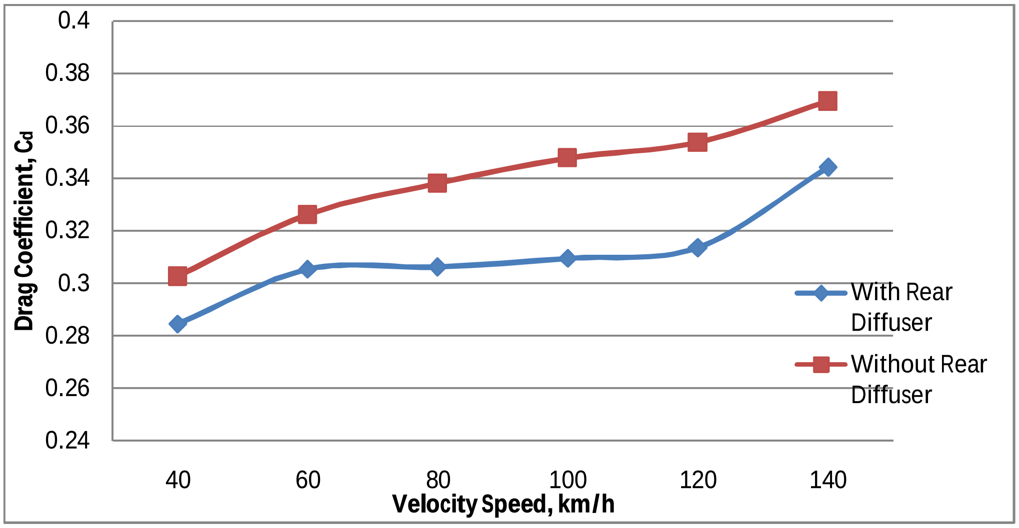


Figure 4.1: Drag coefficient vs. Speed

The figure show the drag coefficient related with the vehicle speed. The different lines of the drag coefficient with and without rear diffuser are plotted. When the changing of the velocity, the higher C_D is obtained and the flow contour also change. By this analysis, increasing value of speed will make the drag of the vehicle increase too. By adding the rear diffuser, the maximum drag at high speed will be reduced for reducing the fuel consumption usage. The drag coefficient of model without rear diffuser is higher than model with rear diffuser. By compare with and without the rear diffuser, the drag coefficients of model with rear diffuser are decrease. This is because rear diffuser will reduce the wake region area at the back of the model. Drag will be reduced when the wake region are decrease. At the high speed only will show the effect of adding the rear diffuser to the model. This has shown at the graph above. From the graph of drag coefficient versus speed above, the increasing of C_D is slightly proportionally with the increasing of speed. The higher speed of vehicle will give higher drag coefficient due to higher drag force to overcome as the vehicle moving. The minimum C_D obtained at the speed of 40 km/h is 0.28454868 and the maximum value of C_D obtained at the speed of 140 km/h is 0.369358. The average C_D for model without rear diffuser is 0.339602 and for the model with rear diffuser is 0.310586.

4.3 Aero Power

The aero power can be calculated by using the aero power equation from the study. From the result value of drag coefficient that obtains in drag coefficient equation, aero power, *P* can be determined. The aero power will be calculate with the equation at the below.

$$P \quad = \frac{\rho}{2} C_D A V (V + V_o)^2 \tag{4.1}$$

P = Aero power [W]

ρ = air density [kg/m^3]

C_D = drag coefficient

A = vehicle section area [m^2]

V = velocity [m/s]

V_o = wind velocity [m/s]

Determine the *P* for model with rear diffuser at velocity 140 km/h. From the data given,

ρ = 1.225 kg/m^3

C_D = 0.34421573

A = 1.833 m^2

V = 38.89m/s

V_o = 0 m/s

Solution,

$$P = \frac{1.225\ kg/m^3}{2} (0.34421573)\ (1.833\ \text{m}^2)\ (38.89\ \text{m/s})\ (38.89\ \text{m/s} + 0\ \text{m/s})^2$$

$$= 22.7373\ \text{kW}$$

Determine the P for model without rear diffuser at velocity 140 km/h. From the data given,

ρ = 1.225 kg/m^3

C_D = 0.369358

A = 1.833 m^2

V = 38.89m/s

V_o = 0 m/s

Solution,

$$P = \frac{1.225\ kg/m^3}{2} (0.369358)\ (1.833\ \text{m}^2)\ (38.89\ \text{m/s})\ (38.89\ \text{m/s} + 0\ \text{m/s})^2$$

$$= 24.3981\ \text{kW}$$

Table 4.2: Aero power of vehicle with and without rear diffuser

	Aero Power, kW	
Velocity Speed, km/h	With Rear Diffuser	Without Rear Diffuser
40	0.4382	0.4662
60	1.5888	1.6972
80	3.7733	4.1644
100	7.4513	8.3709
120	13.0400	14.7033
140	22.7373	24.3981

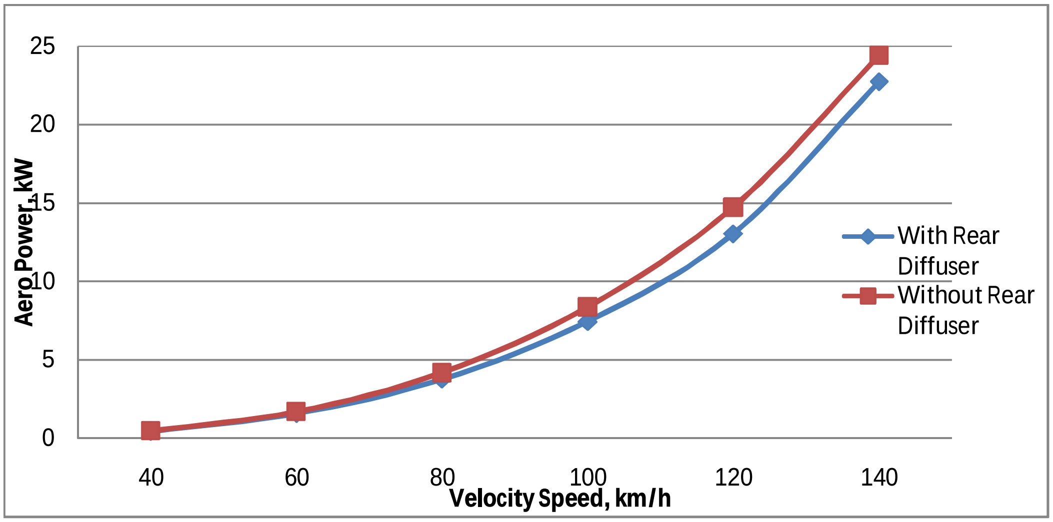


Figure 4.2: Aero power vs. Speed

The figure show the aero power related with the vehicle speed. Vehicle with high speed will use the higher power to make the vehicle move. By using the higher power, the amounts of the fuel used also are higher. The higher fuel consumption will make the usage of vehicle are not economical. At the low speed, the power are used is still lower compare when at high speed. It will show the difference at 100 km/h and above of vehicle speed in term of power usage. Rear diffuser device will reduce the amount of power usage at high speed. By reduced the amount of power usage, the fuel consumption usage also will lower. This is the economical of fuel usage when adding the rear diffuser on the vehicle. From the graph, model without rear diffuser are show the higher of power that is 24.3981 kW compare to the model with rear diffuser that is 22.7373 kW.

4.4 Percentage Reduction

From the adding of rear diffuser on the model, the reduction of drag coefficient, C_D and aero power, P will obtain. This reduction showed the good benefit of adding rear diffuser on the vehicle. The reductions are calculated in percentage. The average values are used to calculate the percentage of reduction. The percentage reduction will be calculate with the equation at the below.

$$\% \ reduction = \frac{Average\ without - Average\ with}{Average\ without} \ x\ 100\% \qquad (4.2)$$

Table 4.3: Average values of C_D and P

	Average Value	
	With Rear Diffuser	Without Rear Diffuser
Drag Coefficient, C_D	0.310586	0.339602
Aero Power, P	8.171489 kW	8.966691 kW

4.4.1 Percentage of Drag Coefficient Reduction

$$\% \ C_D \ reduction = \frac{0.339602 - 0.310586}{0.339602} \ x\ 100\%$$

$$= 8.54\%$$

The range for C_D reduction is estimate to be up than 8.54%.

4.4.2 Percentage of Aero power Reduction

$$\%\ P\ reduction = \frac{8.966691\ kW - 8.171489\ kW}{8.966691\ kW}\ x\ 100\%$$

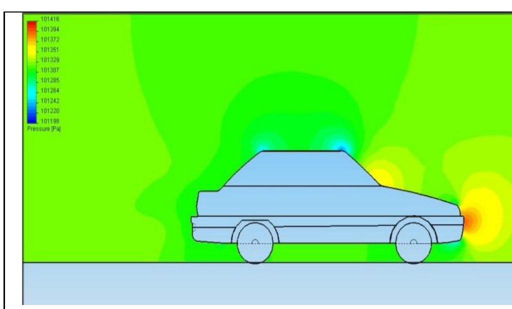
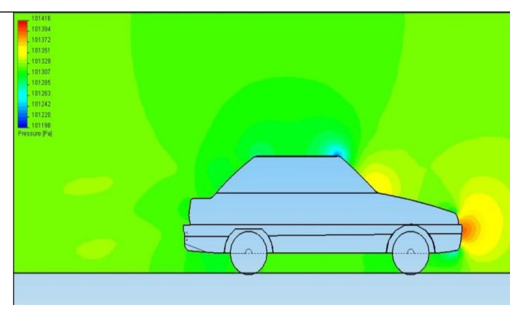
$$= 8.87\%$$

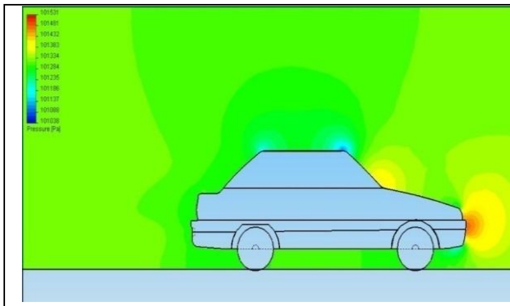
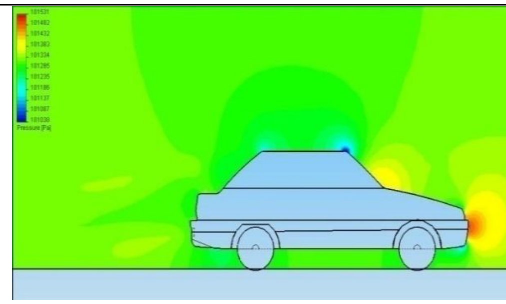
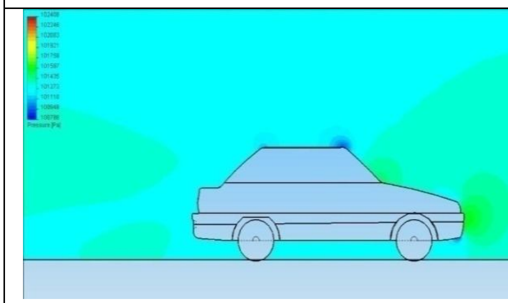
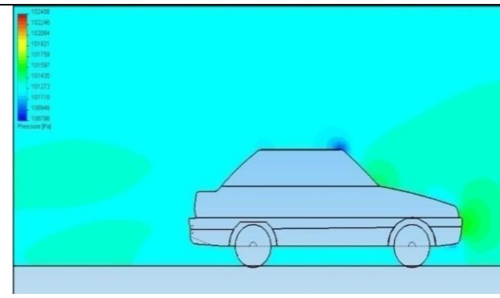
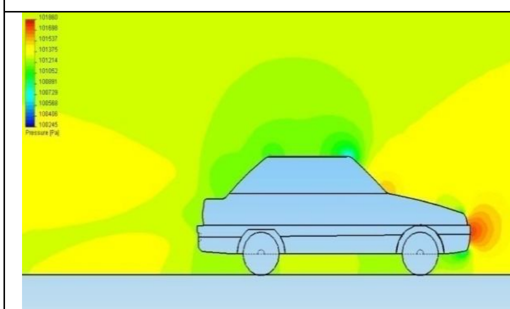
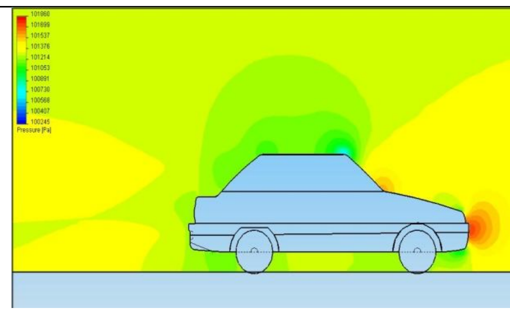
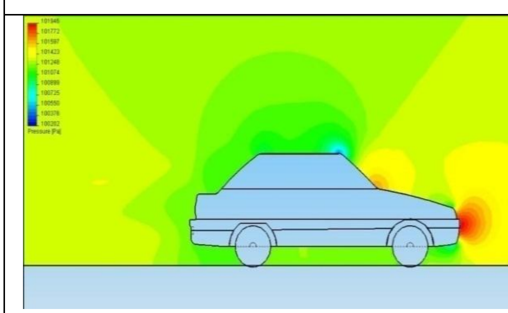
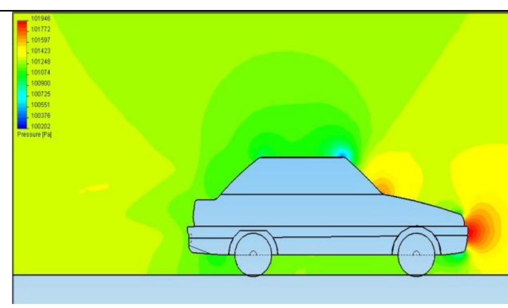
The range for *P* reduction is estimate to be up than 8.87%.

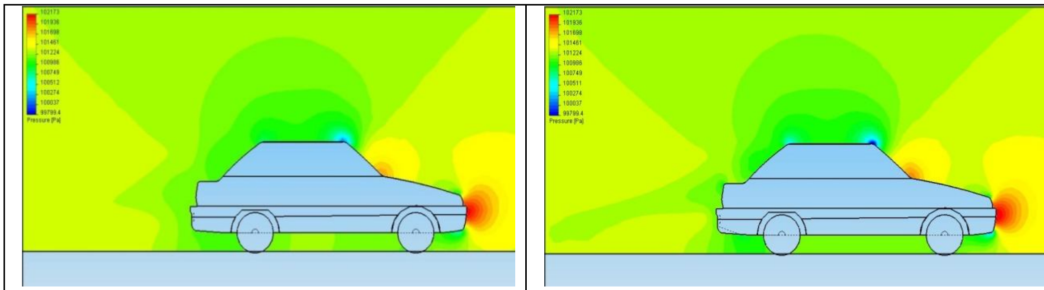
4.5 Simulation Result

The simulation result shows the final result for both conditions. The first condition is where the HEV model is in a standard condition without an addition of rear diffuser. The second condition is where the HEV model is an addition of rear diffuser. Any aerodynamic devices such as front spoiler, vortex generator, rear spoiler, rear diffuser and more do have an effect on overall flow of the vehicles. The different velocity will give the different result of simulation because of the changing of *Re.*

4.5.1 Simulation of External Pressure Distribution

	
40 km/h without Rear Diffuser	40 km/h with Rear Diffuser

 60 km/h without Rear Diffuser	 60 km/h with Rear Diffuser
 80 km/h without Rear Diffuser	 80 km/h with Rear Diffuser
 100 km/h without Rear Diffuser	 100 km/h with Rear Diffuser
 120 km/h without Rear Diffuser	 120 km/h with Rear Diffuser



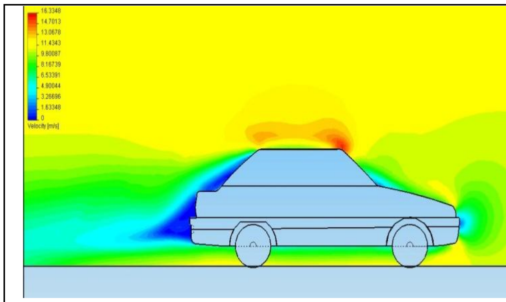
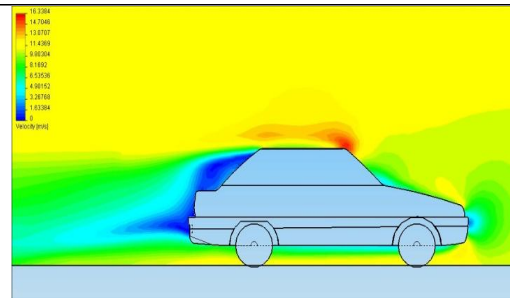
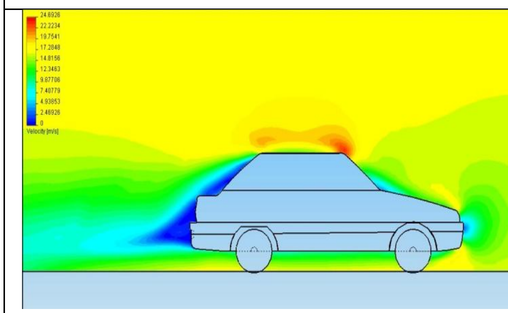
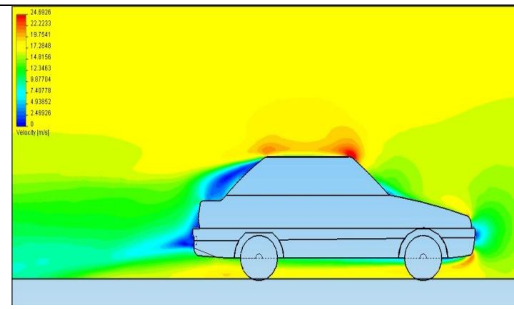
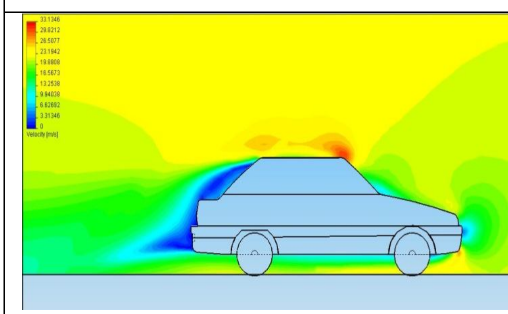
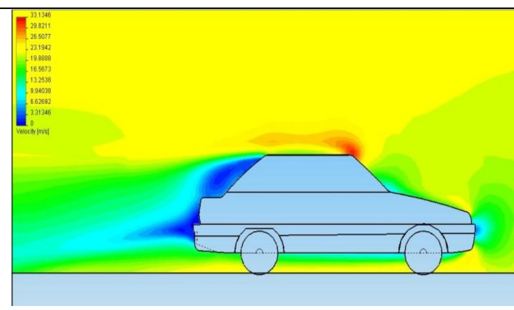
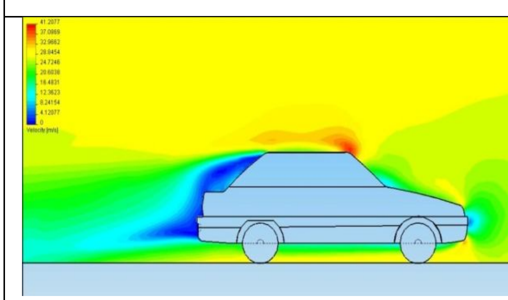
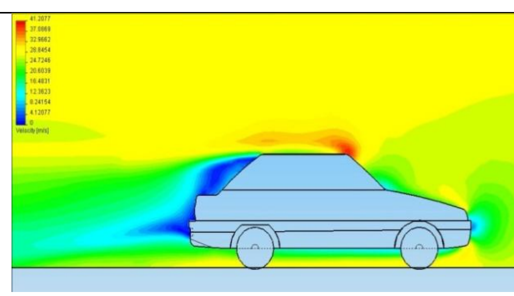
140 km/h without Rear Diffuser	140 km/h with Rear Diffuser

Figure 4.3: Comparison of pressure distribution between with and without rear diffuser

The simulation result was taken on the symmetric plane. When the diffuser device is installed on the model, the pressure at upper area of the rear boot will increase due to the increasing of velocity. This will make the model not easily slip or lift. The pressures at upper end profile of the model with diffuser are higher compared with the model without rear diffuser. The surrounding pressure of the model also higher compare with the model without rear diffuser. As shown in figure, the pressure distributions are lower at the bottom of the underbody compared at the upper as the speeds are increase on model with rear diffuser. Different pressure distribution at rear will influence for the model stability. Higher pressure also is located at the front bumper which the location where the stagnation point of the air flow to the model body. The spot between the front windscreen and front hood also contributed to the higher pressure. However there got a several location which is has the lowers pressure distribution at both the front and rear end of the underbody. From the scale, the higher value is 102173 Pa and the lowest is 99799.4 Pa.

These lower pressure locations are important for the stability of the vehicle. The high pressure are refer as a positive pressure and the lower as a negative pressure. By comparing from these two conditions, the upper pressure distributions of rear diffuser are higher than the model without rear diffuser. These states the vehicle with rear diffuser is better in term of stability. This is because rear diffuser will reduce the curved of streamlines at the rear. These will reduce the pressure at the bottom profile and will increase the downforce at the rear.

4.5.2 Simulation of External Velocity Distribution

 40 km/h without Rear Diffuser	 40 km/h with Rear Diffuser
 60 km/h without Rear Diffuser	 60 km/h with Rear Diffuser
 80 km/h without Rear Diffuser	 80 km/h with Rear Diffuser
 100 km/h without Rear Diffuser	 100 km/h with Rear Diffuser

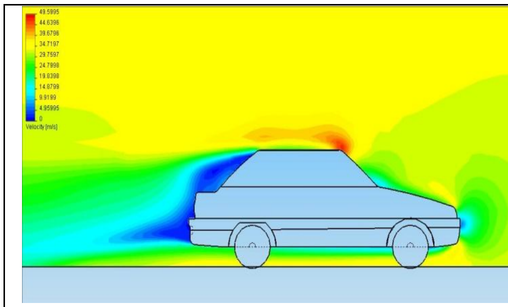
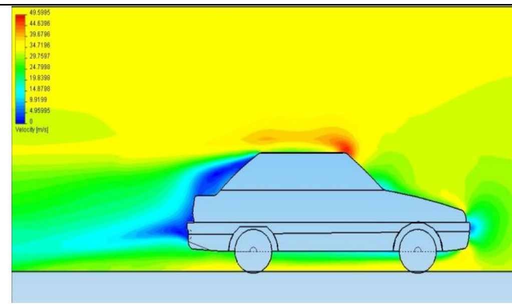
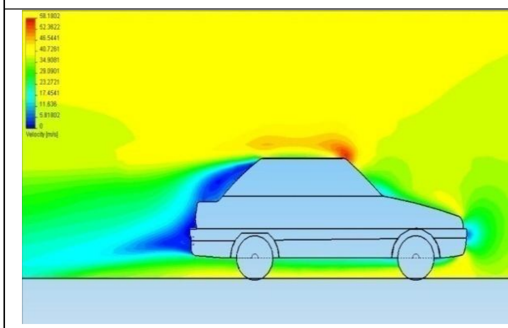
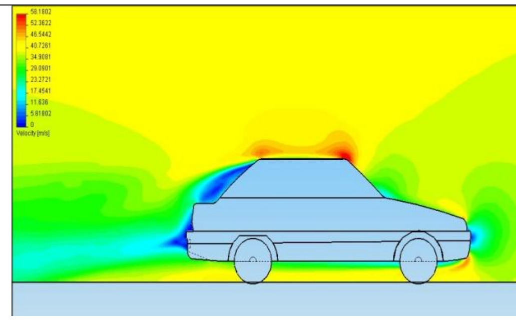
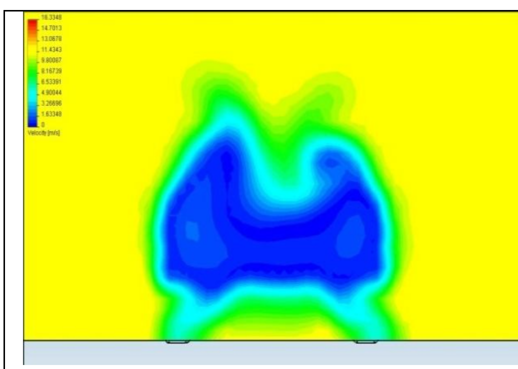
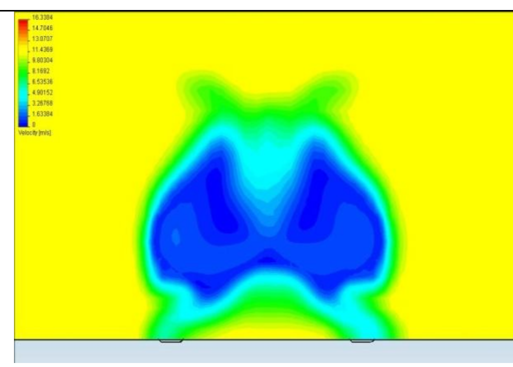
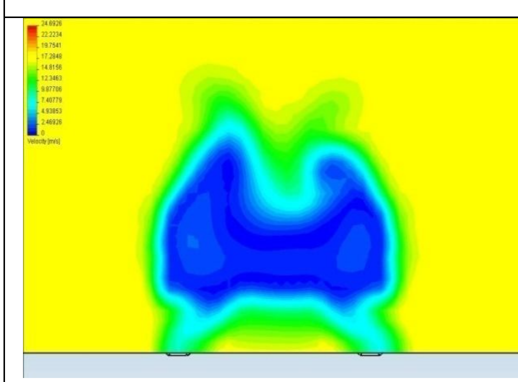
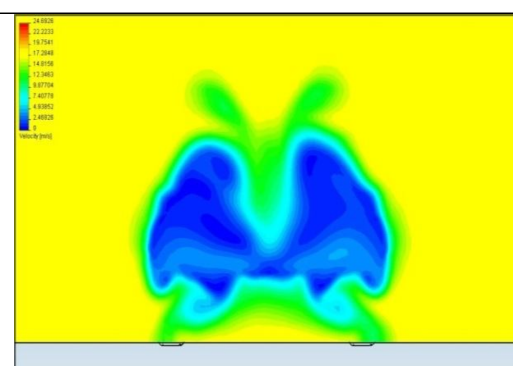
	
120 km/h without Rear Diffuser	120 km/h with Rear Diffuser
	
140 km/h without Rear Diffuser	140 km/h with Rear Diffuser

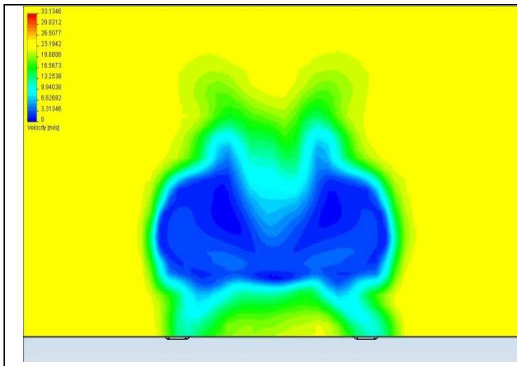
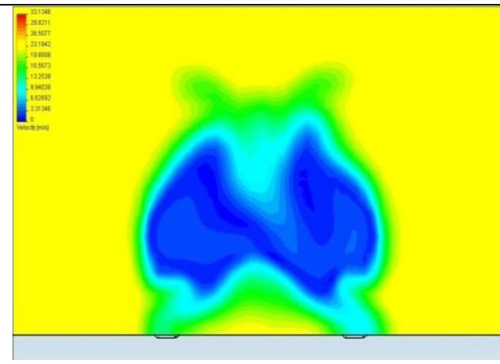
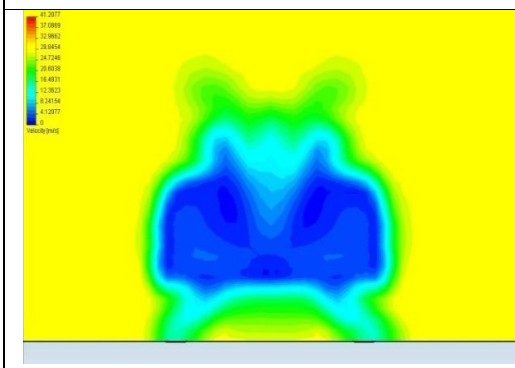
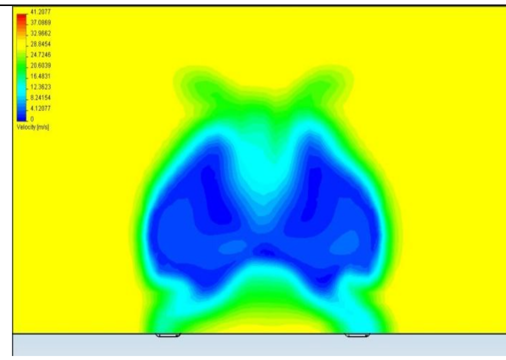
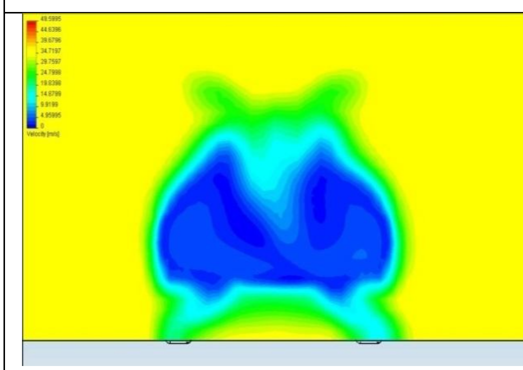
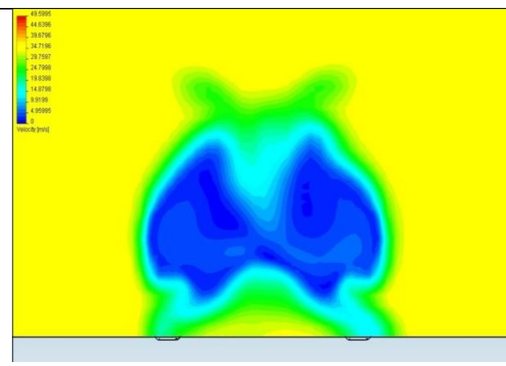
Figure 4.4: Comparison of velocity distribution between with and without rear diffuser

The simulation result was taken on the symmetric plane. From the figure shown above, there are air flow stream that will change due to the increasing of the velocity. The flow also is difference between model with and without rear diffuser. This is because of the flow separation at rear area that been helped by rear diffuser. The flow stream will become slightly smaller that showed the flow separations at the rear model are became smaller too. But, the flow separation of the model with rear diffuser device is smaller than model without rear diffuser device. The wakes areas are produce when the vehicle is moving as show as the blue color regions. This region produces at the rear model area. These regions are called as reversed flow of the velocity. When the speed is increase, the rear flow separation will curved upward as the higher velocity. By installing the rear diffuser device, the wake region will be reducing.

These reducing of wake region will improve the reduction of vehicle drag. So, the drag of the vehicle will reduce due to fuel consumption. At the speed of 140 km/h is clearly showed the reducing of the wake region at the rear area with the angle of the rear diffuser is 19.36°. Low drag is required to attain high speed. Less of wake region will contribute the lower drag of moving body. These have shown at the model with rear diffuser that is the wake region area is less than the model without rear diffuser. These reducing of wake region happen because of the reducing flow separation at the back. From the scale, the higher value is 58.1802 m/s and the lowest is 0 m/s.

4.5.3 Simulation of External Velocity Rear Region

 40 km/h without Rear Diffuser	 40 km/h with Rear Diffuser
 60 km/h without Rear Diffuser	 60 km/h with Rear Diffuser

	
80 km/h without Rear Diffuser	80 km/h with Rear Diffuser
	
100 km/h without Rear Diffuser	100 km/h with Rear Diffuser
	
120 km/h without Rear Diffuser	120 km/h with Rear Diffuser

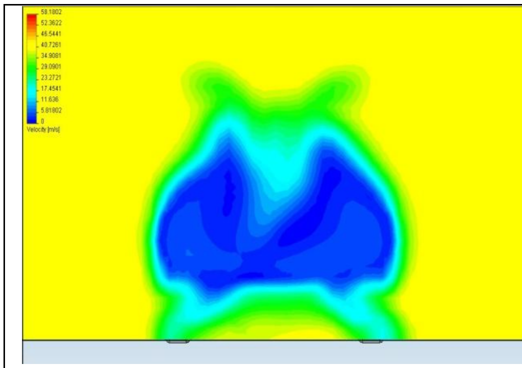
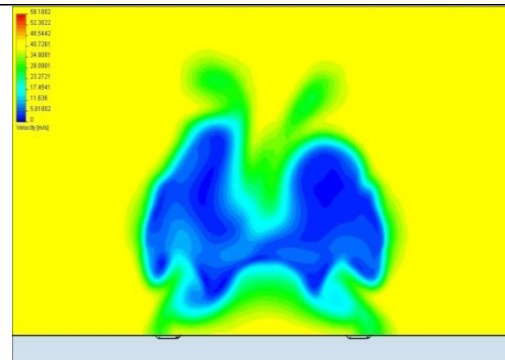
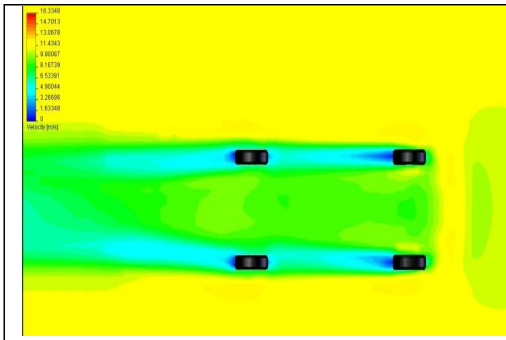
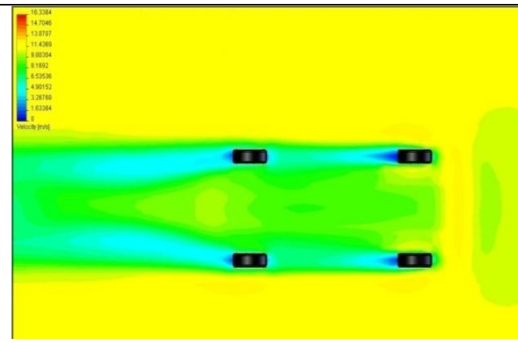
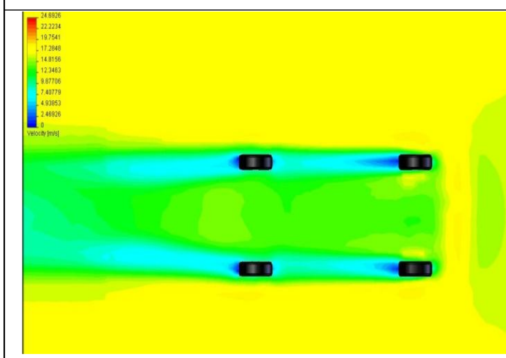
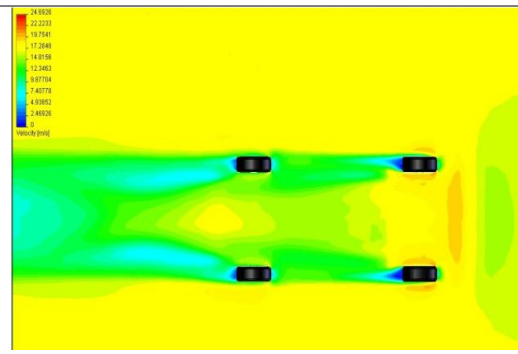
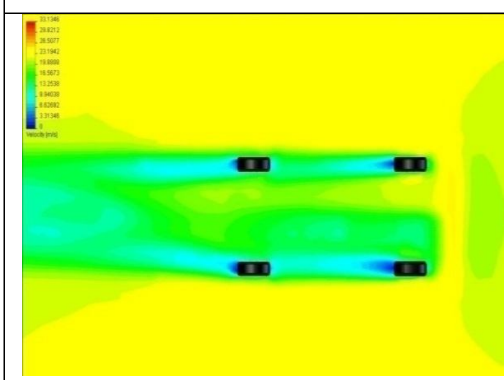
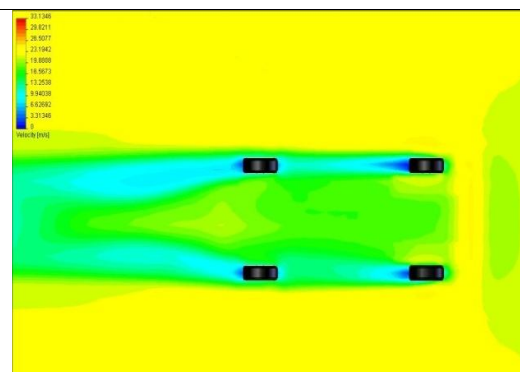
	
140 km/h without Rear Diffuser	140 km/h with Rear Diffuser

Figure 4.5: Comparison of velocity rear region between with and without rear diffuser

The simulation result was taken on the Iso-*x* plane. For the model with rear diffuser device, the wake region at the back will be reducing due the increasing of the vehicle speed. The wake region for model without rear diffuser are decrease too when the speed are increase but the decreasing of the wake region are lower. Velocity 140 km/h are showing the difference wake region area between models with and without rear diffuser. From the scale, the higher value is 58.1802 m/s and the lowest is 0 m/s.

The wake region areas of model with rear diffuser are lower compare with the model without rear diffuser. This are showed the drag of with aerodynamic device are decrease. This will reduce the fuel consumption along the vehicle movement. The lowest of wake region area will reduce the drag of the vehicle. Vehicle with smaller wake region at the rear area that is freestream are better for the fuel consumption.

4.5.4 Simulation of External Velocity Underbody

	
40 km/h without Rear Diffuser	40 km/h with Rear Diffuser
	
60 km/h without Rear Diffuser	60 km/h with Rear Diffuser
	
80 km/h without Rear Diffuser	80 km/h with Rear Diffuser

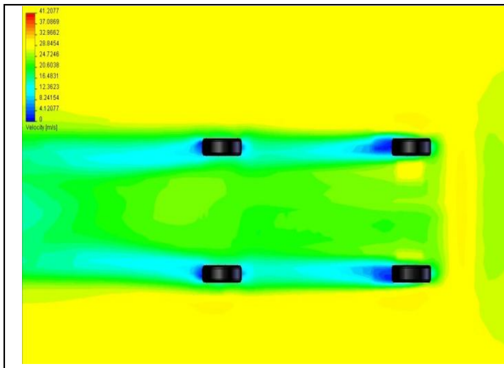
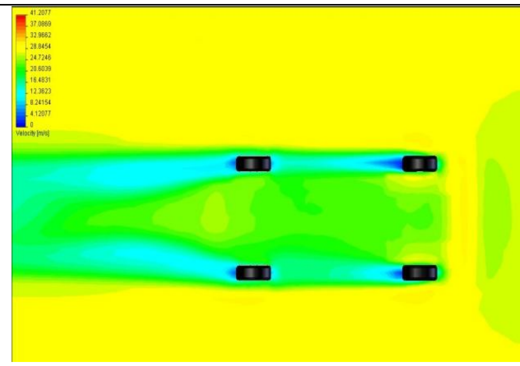
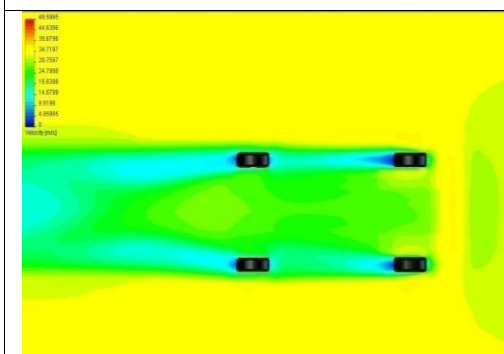
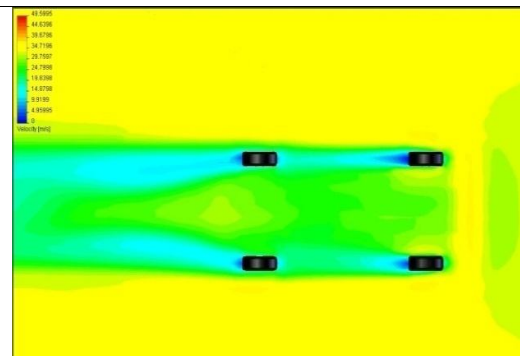
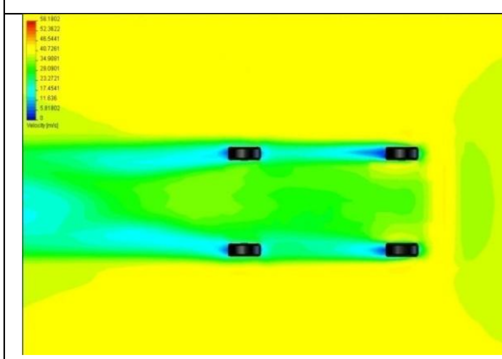
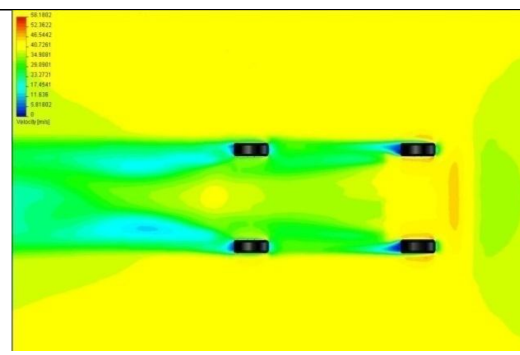
	
100 km/h without Rear Diffuser	100 km/h with Rear Diffuser
	
120 km/h without Rear Diffuser	120 km/h with Rear Diffuser
	
140 km/h without Rear Diffuser	140 km/h with Rear Diffuser

Figure 4.6: Comparison of velocity underbody between with and without rear diffuser

The simulation result was taken on the horizontal plane. At the underbody view, the flow separation will not to separate to each other when rear diffuser is installing. The function of rear diffuser is to pushing the flow from the right and left flow to become in the center at the rear area. There is more separation of the flow at the rear area of the car without diffuser device. As the more flow separation, the wake region at the rear area will more occur. From the scale, the higher value is 58.1802 m/s and the lowest is 0 m/s.

State that the rear diffuser will reduce the curved of streamlines of air flow at underbody to the rear end of the model. By reduce and minimize the streamlines curved, the turbulent air flow also will be reduced. This will contribute to the wake region effect. As for the result, the wake region will be decrease and this will make the drag of the model are decreased too. The aerodynamic drag has been reduced by adding the rear diffuser to the rear end of underbody model in order to direct the air flow. That will provide drag reduction, increased fuel economy, and improve operational performance.

As the installing the rear diffuser device, the frontal pressure are also decrease compare with the model without rear diffuser. It can be concluding that only small negative pressure distributions occur as rear diffuser is added.

4.6 Underbody Analysis

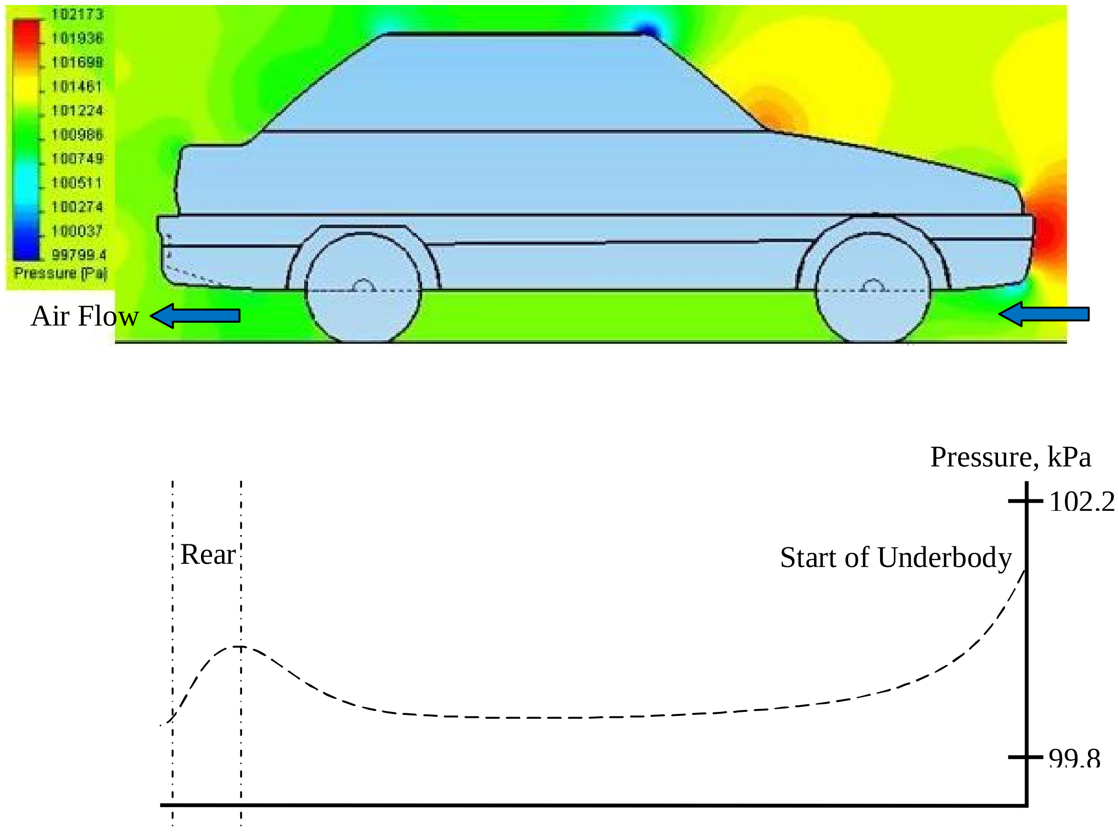


Figure 4.7: Underbody pressure analysis

The figure above are show the along underbody pressure. Rear diffuser is functional to help better air flow at the rear side. This smoothly of air flow will help to control the vehicle center of pressure at the underbody area. This will reduce the lift force when moving in high speed. At the starting of underbody, the pressure is higher because of the stagnation point of the air flow to the vehicle body. Along the underbody area, the pressure is lower. When the air flow reaching at the rear diffuser area, the pressure are increase due to increase the air flow. Higher velocity of air flow will contribute the lower pressure.

4.7 Discussion

As the pressure contribution at underbody, rear diffuser will make a difference in performance at high speed. High speed will cause enough downward thrust to help maintain stability of the car when with rear diffuser. The pressures at above of the vehicle body are lower than vehicle body with rear diffuser. Also the pressures at underbody are higher than vehicle body with rear diffuser. Rear diffuser will make the underbody pressure lower than upper vehicle body. These will make the car not easily to slip and lift. Rear diffuser will reduce the drag and also with the pressure drag. At the upper rear boot, rear diffuser will increase the static pressure of separation region. These will reduce the magnitude of the separation region. It also will reduce the overall pressure drag on the vehicle by introducing high pressure air into the low pressure wake region at the rear area.

By the functional as reduce the streamlines curved at the back, the turbulent air flow will reduce. The wake region will be occurring in less of area at the back profile. By the reducing of the wake region, the drag of moving body also will be lower. As the result of decreasing the wake region, the reduction of the drag can be obtained. The C_D reduction effect for rear diffuser were all equivalent to - 0.0290 and the P reduction effect for rear diffuser were all equivalent to – 0.7952 kW. Pressure distribution from rear diffuser will help vehicle to more stable when high speed and rear diffuser will reduce the drag of vehicle especially in high velocity and fuel consumption.

From this simulation, there are several things can be carried out:-

- Any aerodynamic devices such as rear diffuser, vortex generator, front and rear spoiler do have an effect on overall flow of the vehicles.
- Different velocity will give different result to the simulation because of the changing pressure and velocity distribution.
- The vehicles need to be modeled completed and choosing the fine value of meshing to get the better simulation data result.
- Combination of CFD and FEM gives a good approximation for the drag and drastically reduces cost for the development stage of vehicle body.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

Drag force is the main issues for the road vehicles over the years. By reducing the drag force is the one of the solution to reduced fuel consumption, stability and handling particularly at high speed. It is well known that a large proportion of the drag is the result of pressure drag produced at the rear end of the vehicle and the understanding derived from the study of the wake structure is crucial in improving the road vehicle aerodynamic performance. For the road vehicles, it is basically three-dimensional bluff bodies in proximity to the ground. The trailing vortices and interactions with the near-wake region are influenced the aerodynamic drag. By reducing the wake-region area at the back, it will reduce the total aerodynamic drag for the moving vehicles on the road.

By adding the aerodynamic devices like rear diffuser, the result will decrease the pressure distribution at the rear of the vehicles. From the result, it is shown an improvement by adding the rear diffuser as one of the aerodynamic devices. This simulation by using CFD and FEM also proved that what the previous researcher, journalist and aerodynamics designer that to produced high performance vehicles in a same time can also improved the stability, and fuel consumption that is one of the important devices to include in their design is rear diffuser.

This project is started by adding the rear diffuser device on the HEV model by using SolidWork. There is two different method are using to analyze this project. The first stage is simulating the model using the CFD COSMOSFloworks. The second stage will be using the FEM COSMOSWorks. Before simulate in the second stage, the data from first stage must be convert to FEM. In this stage, the drag force will be obtained. All the data were taken as it will compare to the data after the model is added the rear diffuser. From the analysis that had been finished, there is a different before and after the simulation is complete. This different can be seen clearly even by using an input data for both condition.

From the result, the reductions of aero-power are showed of reducing of fuel consumption. After all the simulation data and analysis is completed, all the objective had archived, all the scope has successfully fulfill. This project become one of the research of the aerodynamics to proven that doesn’t care what ever devices; it will affect the overall body flow characteristic.

5.2 Recommendation

As for the future researches, the different length and different angle of rear diffuser can be compared by simply attached to the same model to see the different of flow and also to determined is the design of the rear diffuser also influent the characteristic of the flow. Some other recommendation can be carried out:-

- Used the Computational Fluid Dynamic software (Fluent) to do the same analysis to see the different on data.
- Further analysis on other parts on the model such as side mirror, side skirt, vortex generator, and others beside the well-known aerodynamic devices.
- Determined the design of rear diffuser also influent the characteristic of the flow.

REFERENCES

[1] T. Morel (1976), *The Effect of Base Slant on the Flow Pattern and Drag of Three-Dimensional Bodies with Blunt Ends*, Aerodynamic Drag Mechanisms of Bluff Bodies and Road Vehicles

[2] Thomas D. Gillespie, Fundamentals of Vehicle Dynamics, Society of Automotive Engineers Inc.

[3] Bruce R. Munson, Donald F. Young, Theodore H. Okiishi, Wiley-Fundamentals of Fluid Mechanics, 5th Edition

[4] M. Boleslav. (July 2006) Sustainable Development: *Aerodynamics Solves Noise and Vehicle Soiling*

[5] B. Fred (October 10-11), *Reducing Aerodynamic Drag and Fuel Consumption Journal*, Global Climate and Energy Project Workshop on Advanced Transportation

[6] Simon Watkins, Greg Oswald, *Journal of Wind Engineering and Industrial Aerodynamics 83 (1999)*, The Flow Field Of Automobile Add-ons-with particular reference to the vibration of external mirror. Pg 541-554

[7] Boris Epstein, Sergey Peigin, Shlomo Tsach, *Aerospace Science and Technology (2006)*, A New Efficient Technology of Aerodynamic Design Based on CFD Driven Optimization, pg 100-110

[8] R.H. Barnard, *Road Vehicle Aerodynamic Design* - An Introduction – (Second Edition), Mechaero Publishing

[9] M.A.Leschziner, Fluid Dynamics Research 38 (2006), *Modelling Turbulent Separated Flow in the Context of Aerodynamic Applications*, pg 174-210

[10] Soo-Jin Jeong, Woo-Seung Kim, Sang-Jin Sung, Medical Engineering & Physics 29 (2007), *Numerical Investigation On The Flow Characteristics And Aerodynamic Force of The Upper Airway of Patient With Obstructive Sleep Apnea Using CFD*, pg 637-651

[11] S. Yongling (2003), *Numerical Simulation of the External Flow Field Around a Bluff Car*, Wu Guangqiang, Xieshuo Automotive Engineering Department Shanghai Tongji University Shanghai, China. pg 1-6

[12] Michele Ferlauto, Roberto Marsilio, Computer & Fluids (2006), *A Viscous Inverse Method for Aerodynamic Design*, pg 304-325

[13] Richard M. Wood, Steven X. S. Bauer, 2001, *Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor-Trailer Trucks*.

APPENDIX A

GANTT CHART / PROJECT SCHEDULE FOR PSM 1

Project Activities	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14
Literature study	14 WEEKS													
Identify problem statement		3 WEEKS												
Define objective and scope of study				2 WEEKS										
Detailed methodology						4 WEEKS								
Sketching the diffuser and apply it at SolidWork							3 WEEKS							
Simulate the rear diffuser with CFD software COSMOSFlowork								3 WEEKS						
Proposal preparation								3 WEEKS						
Presentation preparation											3 WEEKS			
FYP1 presentation														

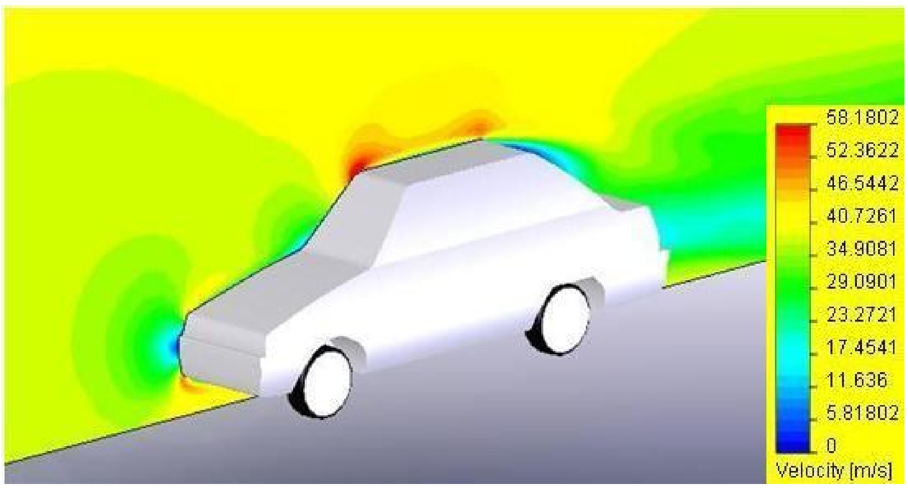
GANTT CHART / PROJECT SCHEDULE FOR PSM 2

Project Activities	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14
Literature study								14 WEEKS						
Analysis the drag estimate without rear diffuser on car model			4 WEEKS											
Analysis the drag estimate with rear diffuser on car model							5 WEEKS							
Compare the drag estimate without and with rear diffuser											2 WEEKS			
Conclude the down force and drag force of the project												2 WEEKS		
Report writing												4 WEEKS		
Presentation preparation												3 WEEKS		
FYP 2 presentation														

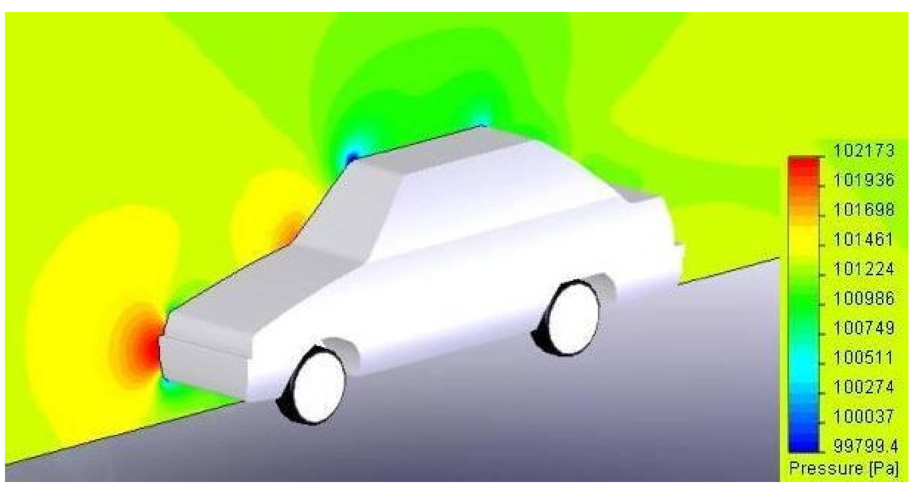
APPENDIX B

PLANE ANALYSIS PROFILE VISUALIZATION

Plane 1

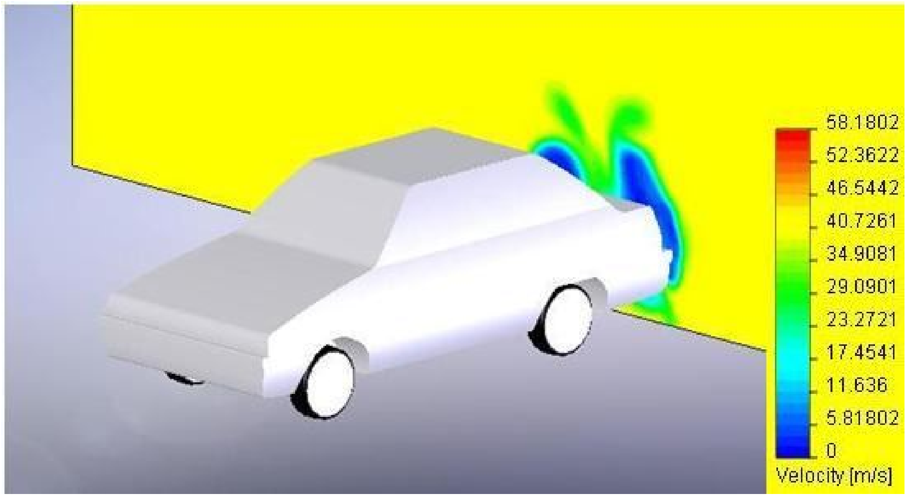


Symmetric plane at the center of the model body



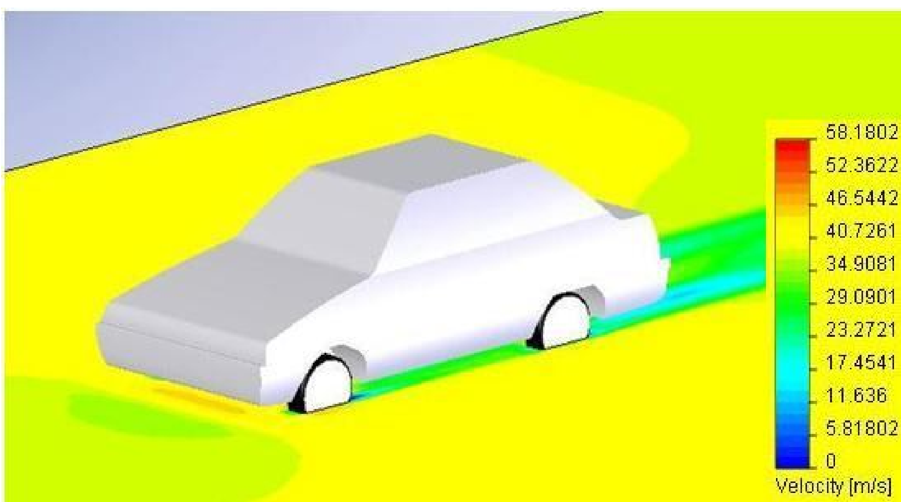
Symmetric plane at the center of the model body

Plane 2



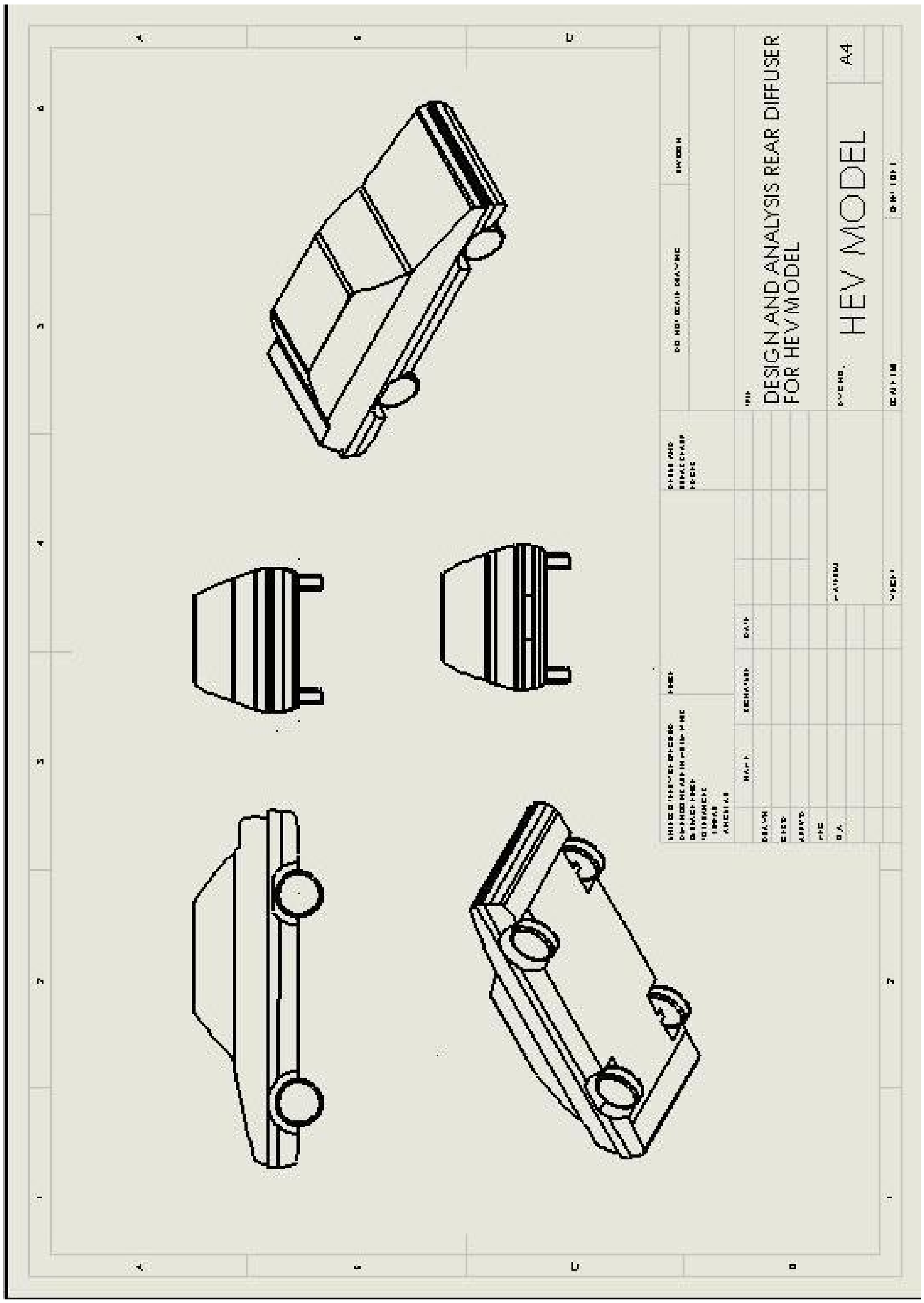
Iso-*x* plane at the freestream behind the model body

Plane 3

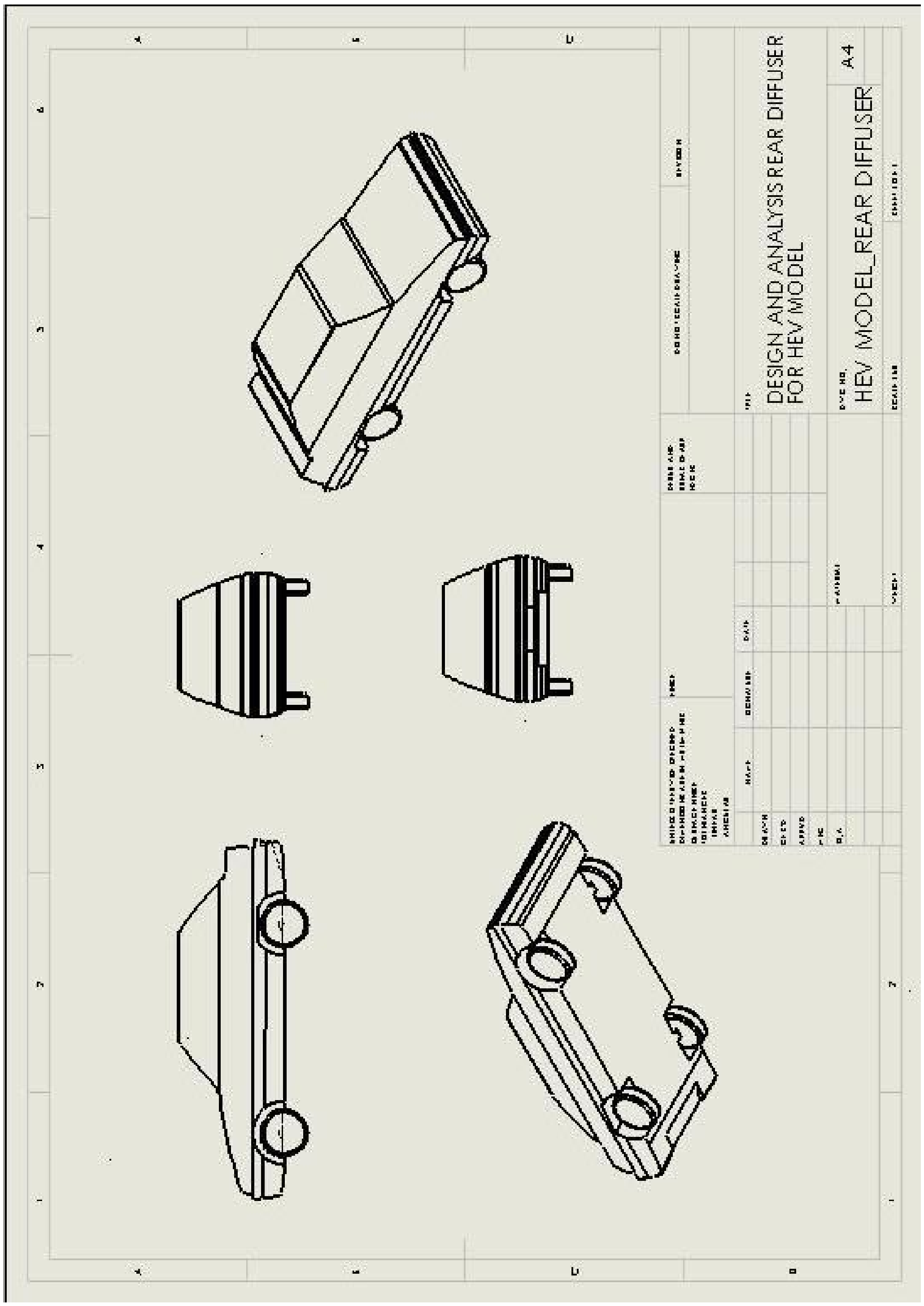


Horizontal plane at the underbody of the model

APPENDIX C


DESIGN AND ANALYSIS REAR DIFFUSER FOR HEV MODEL
HEV MODEL
A4

APPENDIX D


DESIGN AND ANALYSIS REAR DIFFUSER
FOR HEV MODEL
HEV MODEL_REAR DIFFUSER
A4