Study and Improvement of Drag Coefficient of a Bus

by

Adi Akmal Bin Nasrul Hisham

Dissertation submitted in partial fulfillment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

JANUARY 2008

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Study and Improvement of Drag Coefficient of a Bus

by

Adi Akmal Bin Nasrul Hisham

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

in partial fulfilment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL ENGINEERING)

Approved by,

____

(AP Dr. Chalillulah Rangkuti)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

January 2008

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

ADI AKMAL BIN NASRUL HISHAM

ABSTRACT

As one of the main public transportation, bus is surely essential to peoples that want to travel from one place to another without having to pay much. But as the current design of the body of the bus is not very streamlined, it creates large drag forces. This situation made the operation costs higher and the bus fare also have to be increase to compensate the operation costs. In order to save the usage of fossil fuels, engineers are working to design the vehicles to be more efficient. The main objective of this project is to design the most streamlined body of a bus, so that the fuel efficiency will increase because the drag forces are being reduced. Several shapes will be implemented in the body design of the bus and the design will then be tested using software. From the analysis, it shows that the second modification gives the lowest drag coefficient, which is 0.535. The percentage of reduction for the new shape is 38.86 percent. After the design have been finalized and the drag coefficient is obtained, the design is fabricated and being tested in the wind tunnel. The result of the experiment is 0.574, giving 6.79 percent of difference.

ACKNOWLEDGEMENT

All praise to The Almighty Allah, who have given me the strength and wisdom to accomplish this Final Year Project.

I would like to convey my highest appreciation to my supervisor for this Final Year Project, AP Dr. Chalillulah Rangkuti who have been supporting and guiding me from the start until the end of this project.

I would also like to express my deepest gratitude to FYP I & II coordinators, technicians, general assistance as well as internal and external examiners involves for constructive criticisms and suggestions throughout this project.

Lastly, special thanks to those who are involves directly or indirectly in the path towards completing this project. Thank you all.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 Project Background

The coefficient of drag is an important analysis of a bus design. The drag coefficient is a common metric in automotive design, where designers strive to achieve a low coefficient. Minimizing drag is done to improve fuel efficiency at highway speeds, where aerodynamic effects represent a substantial fraction of the energy needed to keep the bus moving. Indeed, aerodynamic drag increases with the square of speed. Aerodynamics is also of increasing concern to bus designers, where a lower drag coefficient translates directly into lower fuel costs [1].

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering [2]. Using the Computational Fluid Dynamic (CFD), the buses are simulated in a large flow domain assuming it is in the atmosphere. The fixed parameter such as incoming velocity and the temperature entering the inlet of the flow domain is defined; the flow is simulated and the pattern of the velocity vector and pressure contour is observed. Validation of such software is performed using a wind tunnel. A wind tunnel is a research tool developed to assist with studying the effects of air moving over or around solid objects.

1.2 Problem Statement

Nowadays, the global price of fossil fuels has increase significantly due to the fossil fuels are becoming less in the natural reservoirs. Engineers and scientists are working to find replacements for fuels from renewable sources such as the sun or the water. Other method used to save the usage of the fossil fuels is to improve the design of a vehicle by making it more aerodynamically efficient.

Bus is one of the most important means of transportation used by the public everyday. It is also one of the cheapest ways to travel, where they do not have to pay for parking fees or fuels. But the shape of the bus are not very streamlined, making the bus has a lot of drag force exerted on its body. This will affect the efficiency of the bus, specifically the engine that moves the bus. Because of this, the fuel consumptions are very high and the fare might be increase due to the operation cost of the bus is increased.

For a large bluff body that has no ideal aerodynamic shapes, the aerodynamic drag exerted on the body is very large especially when cruising at high speed. When there is large drag, the power to overcome it also must be large. Therefore the fuel consumption is also increase.

Fuel consumption depends on two things:

- Efficiency of the engine and transmission system
- Power to overcome the resistance motion

To calculate power, the general formula below is considered

Power = total resistance x vehicle speed

which means more power is needed when the speed of the vehicle increases. Increases in power generated means fuel consumption also must be high. The price of fuel is kept on increasing nowadays and it contributes to higher operation cost. Thus by reducing the aerodynamic drag of the bluff body, less power will use to overcome the resistance, improved safety and durability of structures subjected to high winds and reduction of noise and vibration [8].

1.3 Objective

This is a computational simulation project in order to find the most optimum design of a bus. Computer simulation is the discipline of designing a model of an actual or theoretical physical system, executing the model on a digital computer, and analyzing the execution output. The objectives of this project are:

1. To design the most streamlined body of a bus based on current bus design.
2. To test using software and experiment using wind tunnel and the results from these tests will be compared.

1.4 Scope of Work

The scope of study is to identify the properties of the actual body of a bus, which is the drag coefficient. Some modification on the design will be made and will be tested. Modification is made to improve the properties of the body, mainly its drag coefficient. By lowering the drag coefficient, drag will also decrease. The knowledge of using software for analysis is essential, as well as the procedure of using the wind tunnel. The scopes of work for this project are as listed below:

1) Perform literature review on the automobile coefficient drag.
2) Perform literature review on coefficient of drag on current bus design.
3) Design and test the new shape of bus using ANSYS software.
4) Fabricate and test the new shape in wind tunnel for experimental data.
5) The analysis data and the experimental data will be compared.

CHAPTER 2
LITERATURE REVIEW

2.1 Reviews on Past Research Work

Research has been done previously by National Aeronautics and Space Administration (NASA) on the shape of vehicles and drag coefficient.

According to NASA, drag is a mechanical force generated by the interaction and contact of a solid body with a fluid (liquid or gas). It means drag is generated when there is the difference in velocity between the solid object and the fluid. Drag also moved in the opposite direction of the moving objects [3].

One of the sources of drag is the skin friction between the molecules of the solid surface of the objects and the air. The magnitude of the skin friction depends on the properties of both solid and air because of the interaction. Along the solid surface, a boundary layer of low energy flow is generated and the magnitude of the skin friction depends on the condition of the boundary layer. For the air, the magnitude depends on the viscosity of the air and the relative magnitude of the viscous forces to the motion of the flow, expressed as Reynolds Number [3].

The study done by the National Aeronautics and Space Administration shows different shapes that produces different drag coefficient (see Fig. 1). The values shown are determined experimentally by placing the shapes in the wind tunnel and measuring the amount of drag, the tunnel velocity and density, and the reference area of the object [4]. The equation used is

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

Below are the results of the experiment.

Table 1: Drag coefficient for experimental shapes

No.	Shape	C_D
1.	Flat plate	1.28
2.	Wedge shape prism (wedge facing downstream)	1.14
3.	Sphere	0.07 ~ 0.5
4.	Bullet	0.295
5.	Typical airfoil	0.045

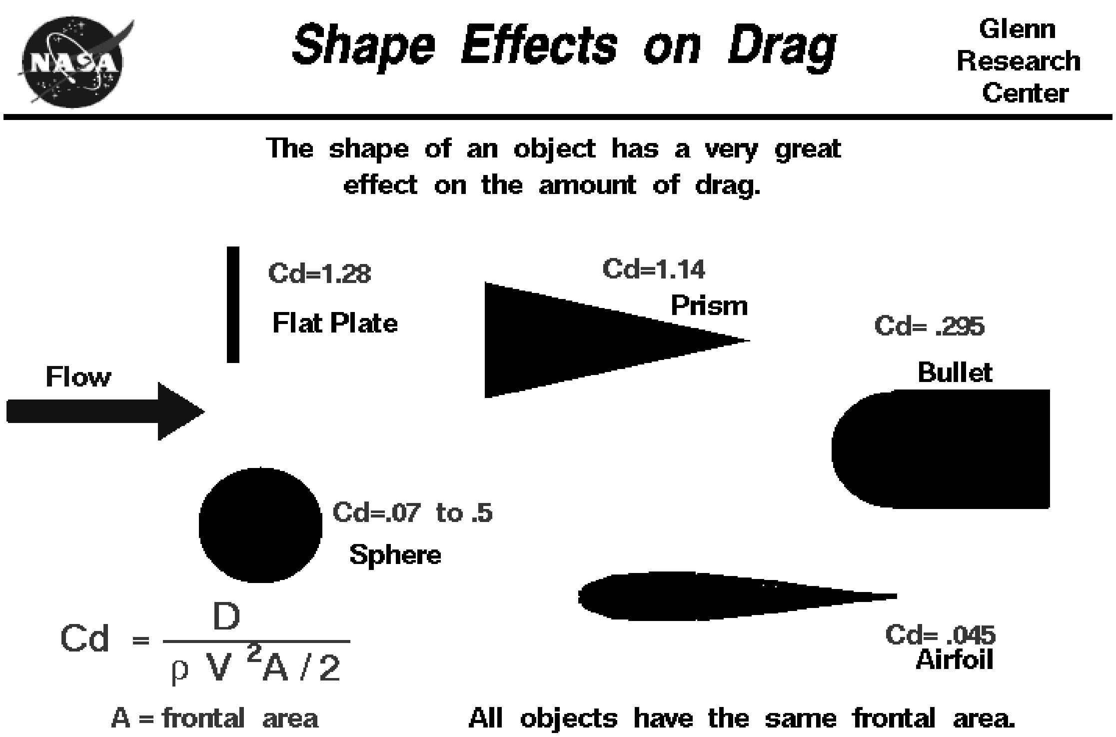

Figure 1: Different experimental shapes that show different drag coefficient [4].

By comparing the results obtained, it shows that the flat plat gives the highest drag and a streamlined symmetric airfoil gives the lowest drag, by a factor of almost 30. It is proven that shape has a very large affect on the amount of the drag produced [4].

2.2 Theory

2.2.1 Force Acting On a Body in Motion

When a body is moving through a fluid, the motion creates forces acting on the body. These forces depend on the properties of the body and the fluid, as well as the relative velocity between the body and the fluid. The properties of the body such as the shape of the body, the size of the body and the orientation of the body will determine the forces acting on the body while the fluid properties such as the density, viscosity and elasticity must be taken account in deriving the equations of the forces or solving it [5].

According to Kuethe/Chow (1998),

> For a body of given shape and orientation that is moving through a flow with speed V, the force may be written in
>
> $$F = f(\rho, V, l, \mu, a)$$
>
> or alternatively,
>
> $$g(F, \rho, V, l, \mu, a) = 0 \qquad (2)$$
>
> By using the method of dimensional analysis, Equation (2) can be written in the equivalent form
>
> $$f_1\left(\frac{F}{\rho V^2 l^2}, \frac{\rho V l}{\mu}, \frac{V}{a}\right) = 0$$

$$\frac{F}{\rho V^2 l^2} = g_1\left(\frac{\rho V l}{\mu}, \frac{V}{a}\right) \qquad (3)$$

Assume μ and a in Equation (2) have no influence on the force F, then the application of dimensional analysis would result to

$$f_1\left(\frac{F}{\rho V^2 l^2}\right) = 0$$

and the solution is

$$F = C_F \rho V^2 l^2 \qquad (4)$$

C_F is called force coefficient and it's a dimensionless constant.

The term $\frac{\rho V l}{\mu}$ and $\frac{V}{a}$ represent Reynolds number and Mach number respectively. If the Reynolds numbers and Mach numbers are equal and the geometries of two flows are similar, the flows are said to be dynamically similar. Dynamically similar means that the flows have the same force coefficient. In order to model a body, these parameters have to be similar to represent the actual body [5].

2.2.2 Aerodynamic Drag Force

The drag force is a combination of both viscous drag and pressure drag. Viscous drag is the force which results from the shear stresses exerted by the fluid as it is deformed by the surface of the object. Pressure drag is the result of the net pressure differences at the front and rear of the object which results from the formation of a wake.

In most applications, moving buses, the pressure drag dominates and streamlining is performed to minimize this force. A perfectly streamlined object experiences no pressure drag (only viscous drag). Streamline is a concept applied in the construction of solid objects to minimize pressure drag imposed on it. A streamlined body allows air to gradually decelerate along the back part the body, thus preventing boundary layers from separating. Separated boundary layers promotes pressure drag, therefore must be eliminated [6].

In order to design a vehicle, there are two forces that must be considered; which is the aerodynamic forces that is acting on the body of a vehicle and rolling resistance which is acting on the drive wheels of a vehicle. The sum of these forces will make up for the total drag forces acting on a vehicle. But in this research, only the aerodynamic force is considered, as it is the only force acting on the body of the vehicle.

When heavy trucks or buses are cruising at a speed of 70 miles per hour, which is the common highway speed today, the aerodynamic drag contributes to 70% of the total drag. Therefore air resistance cannot be a negligible factor anymore for the bluff body vehicle such as the buses. For more efficient in power and fuel consumption, the design of the body of a bus can be altered to decrease the aerodynamic drag [6].

There are 2 types of pressure or form drag in the analysis of the bus design, which are frontal pressure and rear vacuum drag. Frontal pressure is caused by the air attempting to flow around the front of the bus. As the air accumulated in front of the bus, it begins to compress and eventually the air pressure in front of the bus increases. However the air that is traveling along the side of the body of the bus is at atmospheric pressure, a lower pressure compared to the pressure in front of the bus. The compressed air in front of the bus will find a way out from the high pressure zone through the side, top and bottom of the bus [7].

Meanwhile rear vacuum is caused by the ‘hole’ left in the air as the bus passes through it. The blocky shape of the bus punches a big hole in the air with the air rushing around the body. At speeds above a crawl, the space directly behind the bus is ‘empty’. This empty area is a result of the air not being able to fill the hole as quickly as the bus can make it. Even if the air try to fill this area, the bus is always ahead, resulting a continuous vacuum sucks in the opposing direction of the bus. This inability to fill the hole left by the bus is called flow detachment [7].

2.2.3 Power Requirement of a Vehicle

Based on vehicle characteristics such as frontal area, drag coefficient, weight and gear ratio, a vehicle simulation can be used to determine the fuel economy performance of various engine and vehicle combinations [10].

The power requirements for a vehicle are specified by a road load power equation, which includes the effect of aerodynamic drag and rolling resistance [10].

$$\dot{W}_v = \left(C_r m_v g + \frac{1}{2} C_d \rho_o A_v U_v^{\ 2} \right) U_v$$

Where

C_r = coefficient of rolling resistance

m_v = mass of vehicle (kg)

g = gravitational constant, 9.81 m/s^2

C_d = drag coefficient

A_v = vehicle front cross sectional area (m^2)

U_v = vehicle speed (m/s)

2.2.4 Wind Tunnel

Wind tunnels provide experimental information that is useful for solving aerodynamic and hydrodynamic problems. It also can provide the most rapid, economical and accurate means for conducting aerodynamic research and obtaining the aerodynamic data to support design decisions. The reason for this is because:

1) it is possible to use models that are prepared earlier in design process
2) it include the full complexity of real fluid flow
3) it can provide large amounts of reliable data[12]

2.2.4.1 Importance of Similarity

Usually wind tunnel experiments are conducted using scale models. Therefore it is crucial to have the similarity to predict the full-scale behavior. The equations for fluid motion in non-dimensional form can provide a foundation for designing scale experiments and interpreting the resulting data [12] (see Appendix A).

There are three coefficients to be considered in the wind tunnel experiments, which is the Reynolds number, the Mach number and the Froude number. These equations are obtained by introducing non-dimensional variables into the conservation equations [12].

For experiments in which the model is held stationary during data gathering, the Reynolds number and the Mach number are the significant similarity parameters. If a model used in experiment has the same Reynolds number and Mach number as the full-scale application, then the model and the full-scale flows will be dynamically similar. In other terms, it means the non-dimensional functions for the fluid velocity component, pressure coefficient, density, viscosity and temperature will then be same for the model and the full-scale flows [12].

In practice, it is seldom possible to match both Reynolds number and the Mach number to the full scale in a model experiment. Frequently that neither Reynolds number nor Mach number can be matched. It is then choices are made to select one of these coefficients to be used based on the parameters known to be most important for the type of flow situation under consideration. In a low-speed flight region, the Reynolds number will have larger effects rather that Mach number [12].

CHAPTER 3
METHODOLOGY

3.1 Procedure Identification

Procedures involved for the project design development are shown in Figure 2 below. For the analysis, the new design will be created from the dimension similar to the reference model and tested using the software. The shape that produced the lowest drag coefficient will then be tested in the wind tunnel.

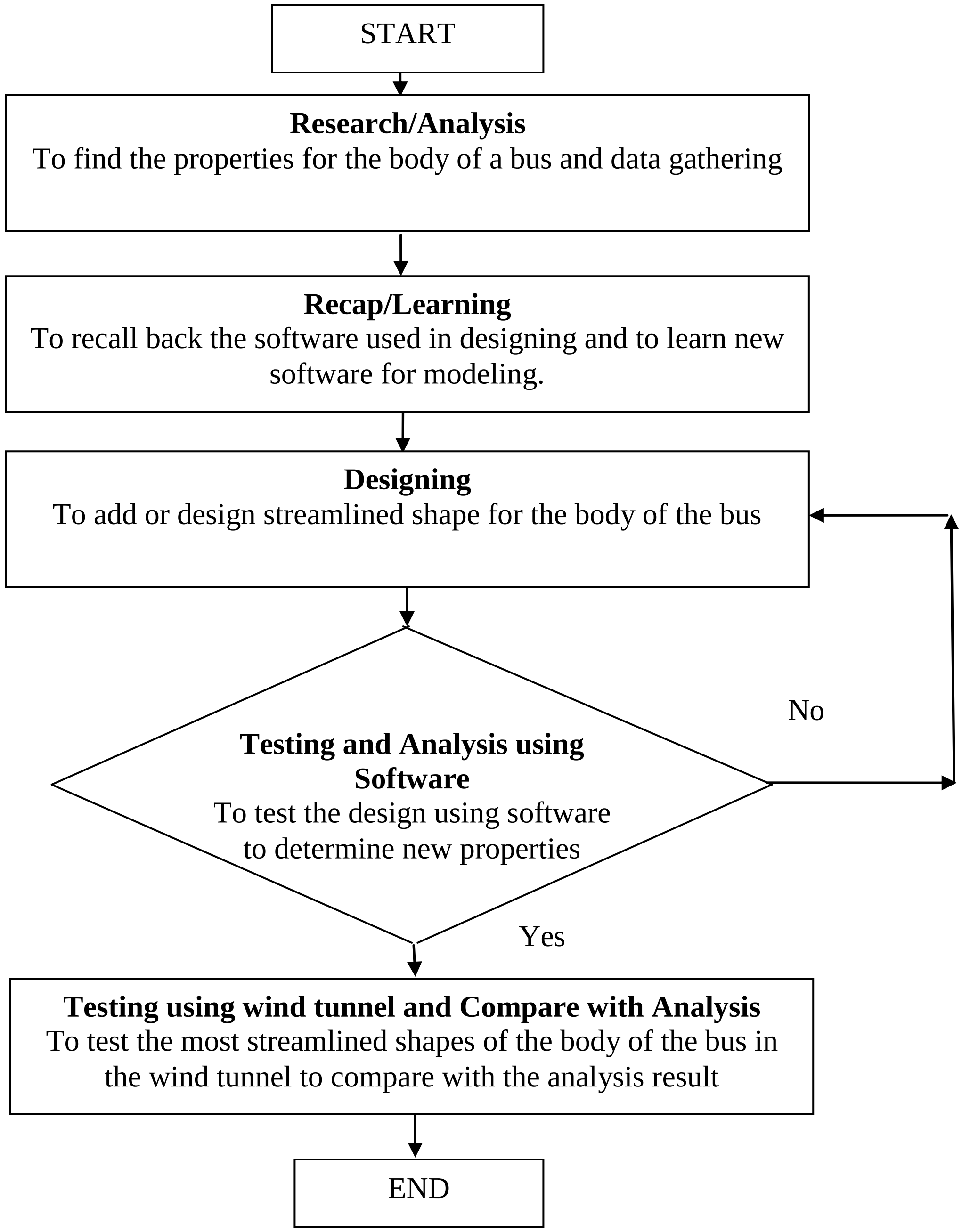

Figure 2: Flow chart for methodology

2.2 Detail Project Procedure

2.2.1 Assumption

A few assumptions have been made through the procedures done for completing this project. They are:

1. *First Assumption*

Flow	Reynolds Number
Laminar	$Re < 1 \times 10^5$
Transition	$1 \times 10^5 < Re < 3 \times 10^6$
Turbulent	$Re > 3 \times 10^6$

$$\mathrm{Re}_x = \frac{\rho V x}{\mu}$$

At temperature, T = 308 K,

The density, $\rho = 1.145$ kg/m^3

The dynamic viscosity, $\mu = 1.895 \times 10^{-5}$ kg/ms

The length, x = 100 m

So, $\mathrm{Re}_x = \dfrac{1.145 x 22.22 x 100}{1.895 x 10^{-5}} = 134258047.5$

The Reynolds number = 1.34×10^8

Therefore the flow is in turbulent region [8].

2. *Second Assumption*

$$Ma = \frac{V}{c} = \frac{22.22}{349} = 0.06366$$

Ma is 0.06366 << 0.3

Therefore the flow is incompressible [8].

3.3.2 Modeling the shape of a bus

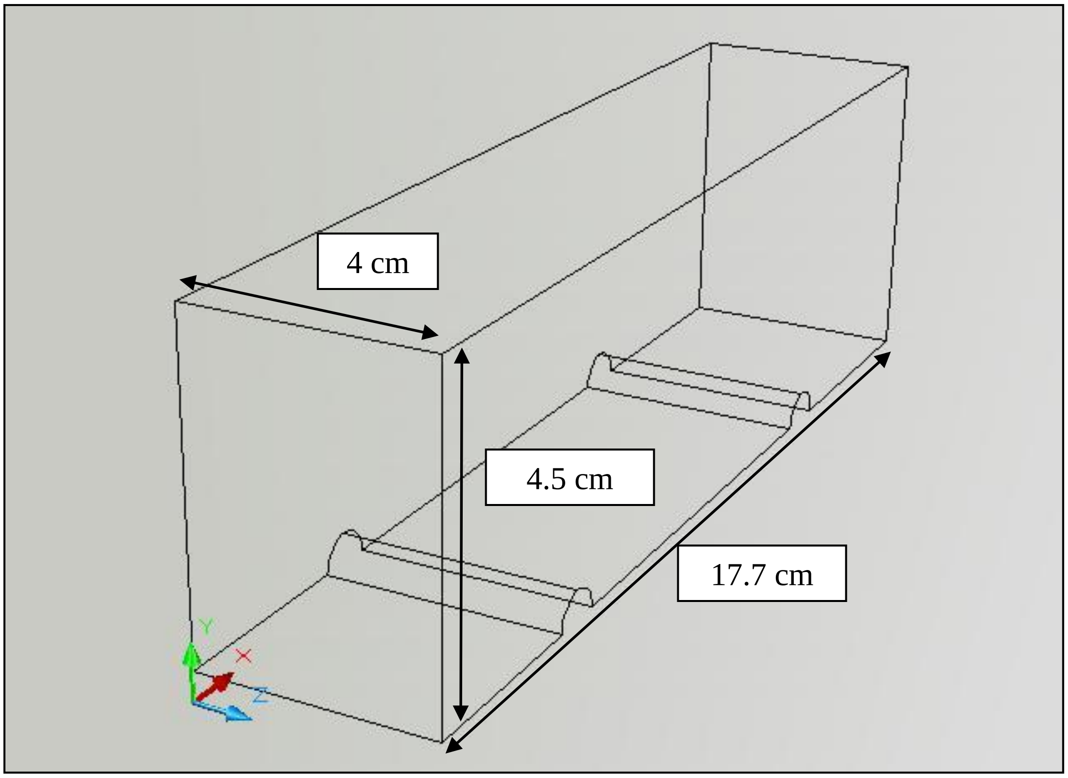

Figure 3: 3-dimensional modeling of reference model

Figure 3 shows the shape of a bus that will act as reference model. This model will also be tested in the ANSYS software to obtain the drag coefficient of the reference shape.

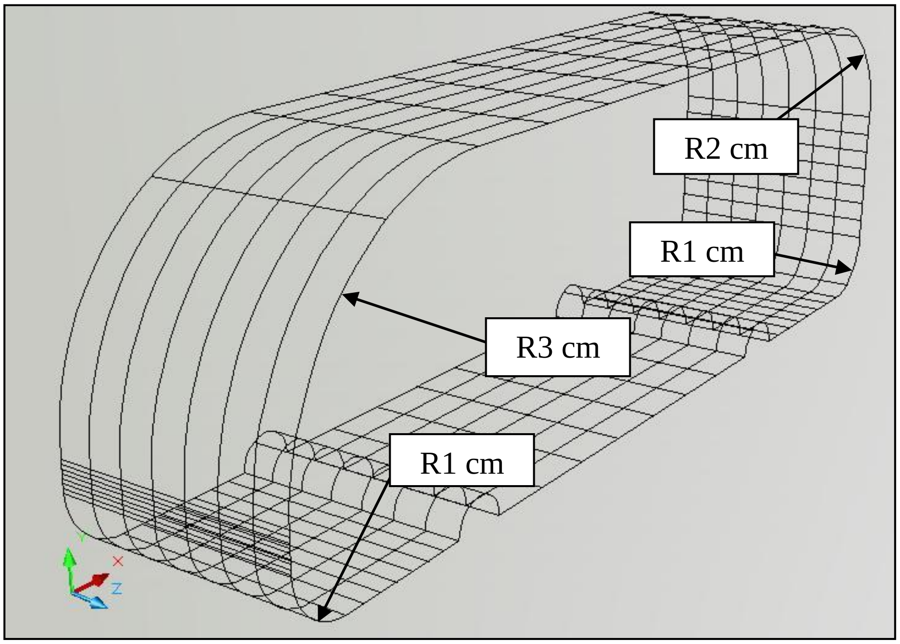

Figure 4: First modification of the shape

The first modification made to the design is to produce more streamlined shape at the frontal area and the rear (Refer Figure. 4). The frontal part of the body is made in rounded shape so that the flow will be smoother. By theory, this would help to reduce the drag force exerted in front of the bus. The streamlined shape at the rear part of the bus will also help in decreasing the vacuum effect produced by the air flow around the body of the bus.

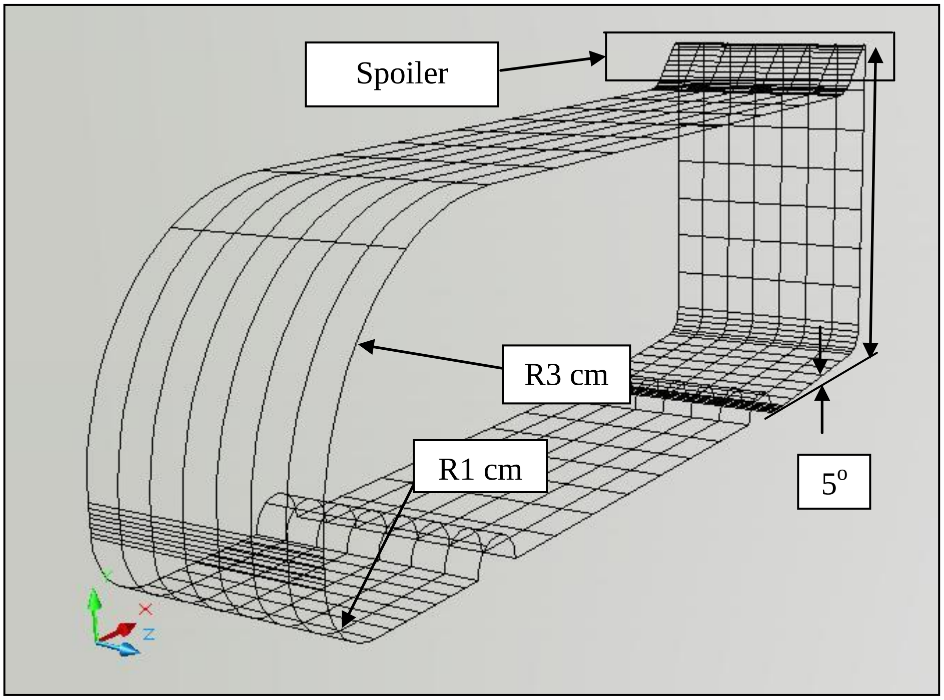

Figure 5: Second modification of the shape

Minor modification made at the rear of the bus (Refer Fig. 5). This shape will serves as a spoiler in order to decrease the vacuum effects at the rear part of the bus and also to provide more traction to the rear tire.

3.3 Tools/Equipment Needed

Equipment needed to:

- Measure and calculate the properties of the body of a bus
- Design and model the new shapes
- Test the new design

Tools required:

- Engineering software
    - AUTOCAD / CATIA for modeling
    - ANSYS for meshing and testing

- Wind tunnel

3.3.1 ANSYS

ANSYS will be employed in this project. ANSYS is a comprehensive general-purpose finite element computer program that contains over 100,000 lines of code. ANSYS is capable of performing static, dynamic, heat transfer, fluid flow and electromagnetism analyses.

There are three processors that are used most frequently [9]:

I. Preprocessor

The preprocessor contains the commands needed to create a finite element model:

a) Define element types and options
b) Define element real constants
c) Define material properties

At this point, physical properties of the material will be defined such as modulus of elasticity, Poisson's ratio, thermal conductivity and etc...

d) Create model geometry
e) Define meshing objects
f) Mesh the object created

II. Processor

The next step involves applying appropriate boundary conditions and the proper loading. The solution processor has the commands that allow applying boundary conditions and loads. It includes velocities, pressures and temperatures for fluid flow problems.

Once the model has been created and the boundary conditions have been applied with appropriate loads, the software will need instructions to solve the set of equations generated by the model.

III. Postprocessor

Postprocessors are available to view the result obtain from the analysis. The result can be in form of:

a) Deformed shape displays and contour displays
b) Tabular listings of the result data of the analysis
c) Calculations for the results data and path operations
d) Error estimations

The solution procedures are shown in Appendix C.

3.3.2 Wind Tunnel

Wind tunnel will be use to obtain experimental data of the drag coefficient. This project will use WT04 Sub-Sonic Wind Tunnel Demonstration Unit, available in the laboratory [13]. The wind tunnel specification is as below:

Table 2: WT04 Sub-Sonic Wind Tunnel Demonstration Unit specifications

No	Item	Specifications
1	Type of Tunnel	Low speed, open circuit, suction type
2	Test Section	300H x 300D x 1500L mm (inside)
3	Air Speed	Up to 60 m/s (continuously variable)
4	Contraction Ration	25:1
5	Drive	Axial flow fan by DC motor with digital type DC drive controller
6	Motor	11Kw, 2800rpm DC motor
7	Overall Size	6.4L x 2.5H x 1.7W meter approx.
8	Power Requirement	AC, 3ph 415 volts, 30 Amps Electrical supply with neutral and earth connection
9	Material of construction	Effuser and Diffuser: Mild Steel Blower and supporting frame: Mild Steel

Figure 6: WT04 Sub-Sonic Wind Tunnel Demonstration Unit

Figure 7: Test section of the wind tunnel

CHAPTER 4
RESULTS AND DISCUSSION

4.1 ANSYS Analysis

In order to find the drag force exerted, the model of the bus is being measured and then being tested using ANSYS analysis. The parameters used are as follows (at 1 atmosphere):

Frontal Area, A	=	18 m^2
Air Velocity, V	=	22.22 m/s
Air Density, ρ	=	1.145 kg/m^3
Air Viscosity, μ	=	1.895×10^{-5} kg/ms
Inlet Pressure	=	1 atm
Outlet Pressure	=	0
Temperature	=	308 K

Note: The result displayed is in meter per second (m/s) for velocity and Pascal (Pa) for pressure.

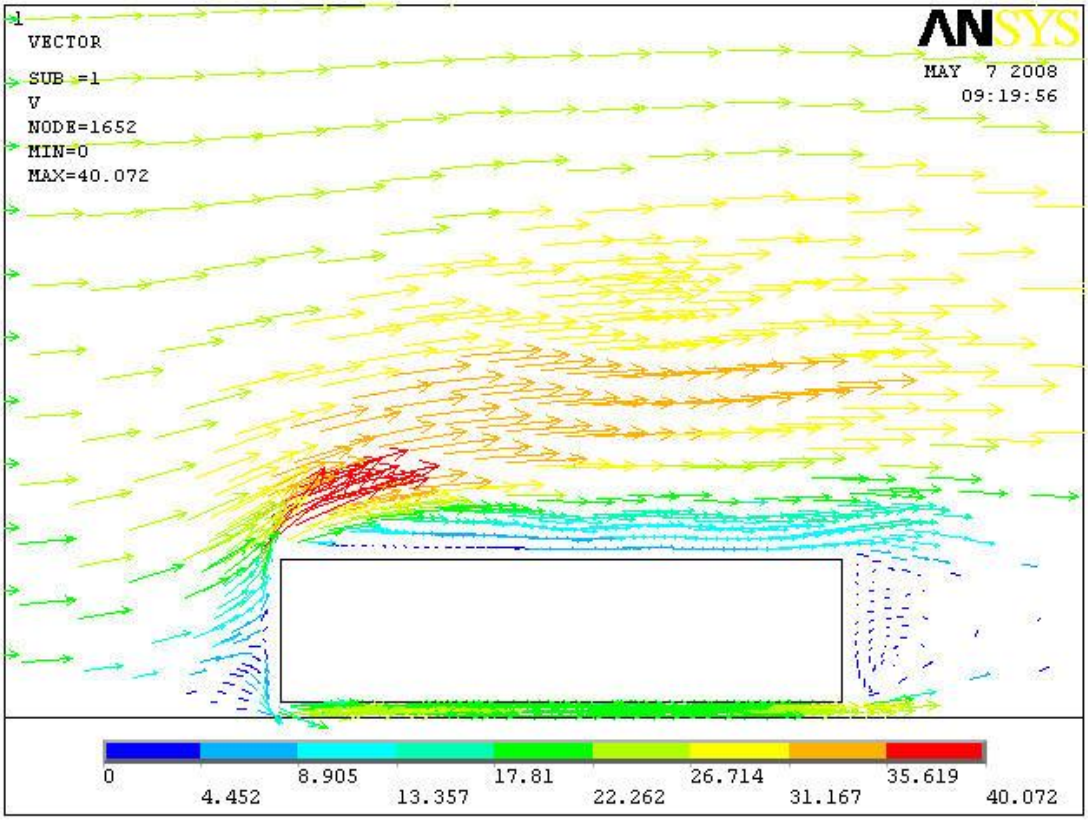

Figure 8: Velocity vector for reference model

As shown in Figure 8, when the air flow at 80 km/h hits the body of the bus, it begins to separate according to the shape of the bus, which is to the top, bottom and the side of the bus. As the flow attempted to get around the body of the bus, it compressed at the front of the bus, causing the frontal pressure to be high (see Figure 9). As it compressed, the flow will try to get around the body because the pressure along the body is at atmospheric pressure, much lower compared to the frontal part of the bus. The flow will then passes along the bus smoothly until it arrived at the rear part of the body. At the rear part, the flow from the top, bottom and side of the body will meet, causing re-circulation of flow or vortex at the rear end of the body, even though it was small. This will cause a wake region at the rear of the body, which is not desirable for vehicles. The flow on top of the bus is accelerating almost twice than the initial value at the inlet. This is due to the sharp edges of the bus.

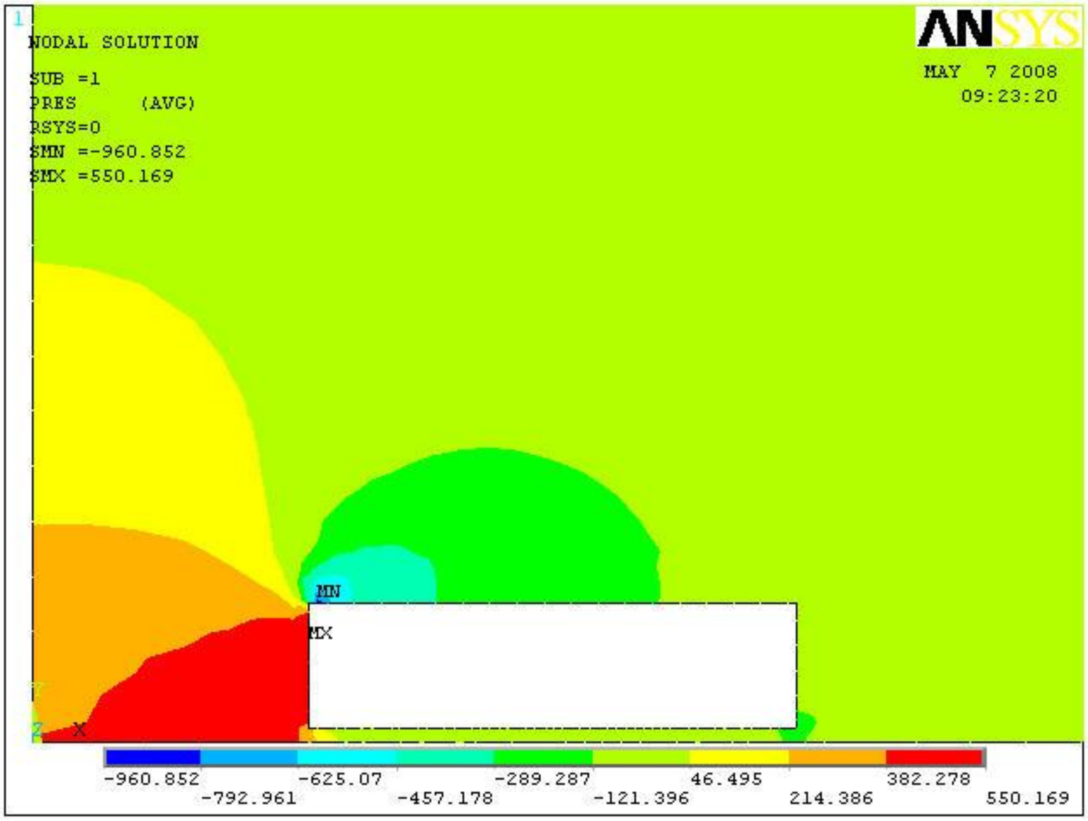

Figure 9: Pressure distribution along the body of the reference model

When the flow in front of the bus is being blocked, it decelerates to almost 0 values before attempting to flow around the body. The effect of this decelerating is the frontal pressure will be increase. This condition will have a pushing effect to the body of the bus, making it to use more power to move forward. This is shown in Figure 9, where the maximum pressure (550.169 Pa) is located at the frontal area.

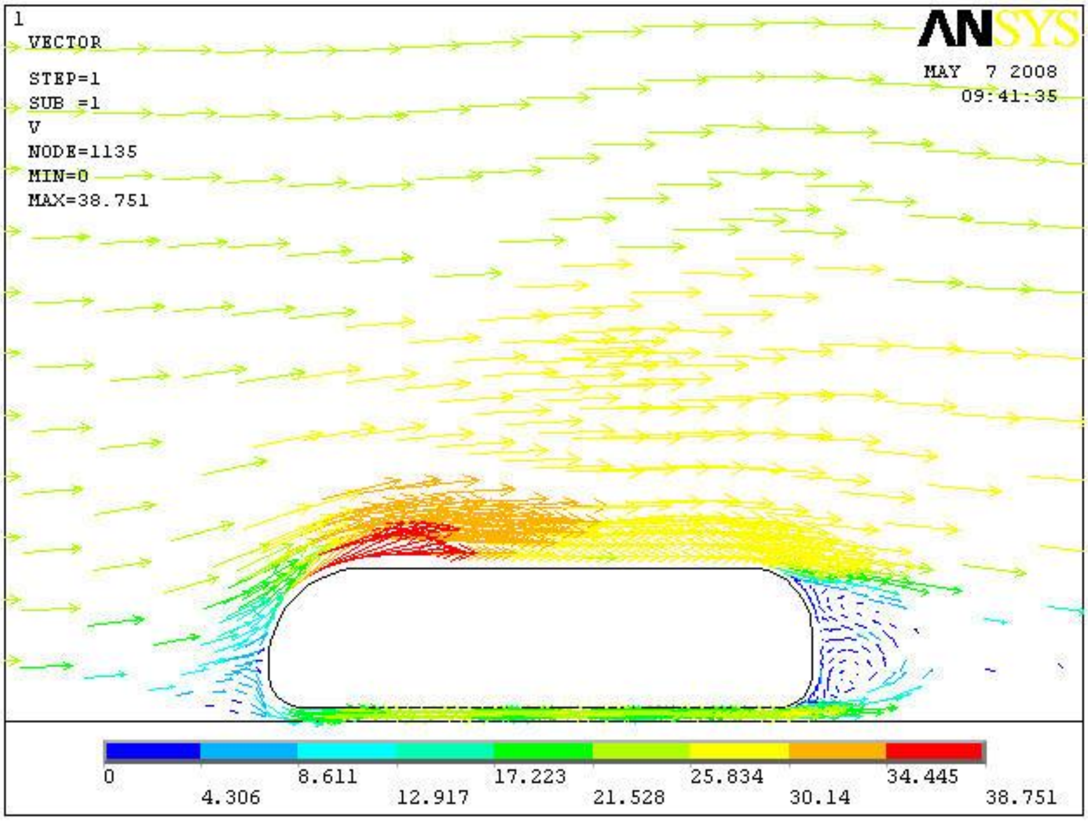

Figure 10: Velocity vector for the first modification

For the first modification shape, the more rounded-shape is applied at the front of the bus. This will ease the flow along the frontal area of the body. The flow will then arrived at the rear part of the body and will then meet with the flow from the side and bottom of the body. Re-circulation will also occur at the rear part of the body, as seen in Figure 10, but not as much as the reference model. The rounded shape helps to provide smoother flow around the body, making it more streamlined. This can be shown in Figure 9, where the frontal pressure is reduced. By comparing to the reference model, the flow at the top area of the body (38.751 m/s) is slightly lower than the reference model (40.072 m/s), thus showing that smoother flow at the top of the body.

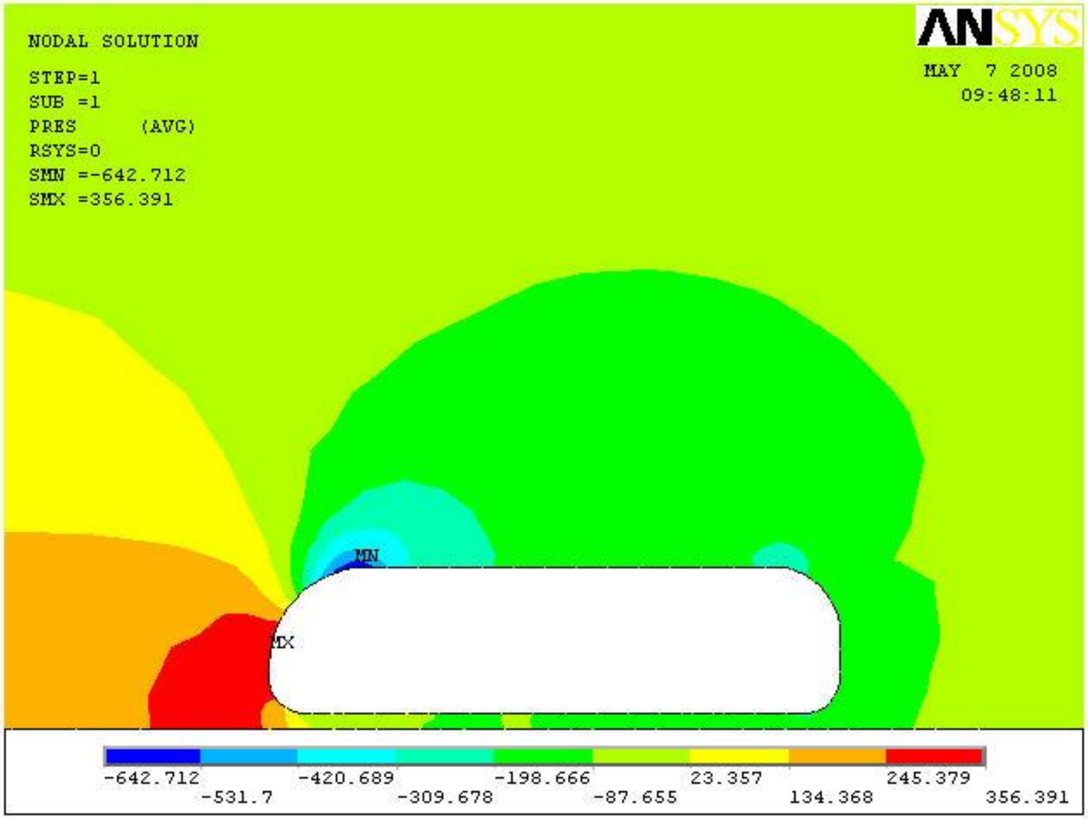

Figure 11: Pressure distribution along the body for the first modification

As describe in the discussion above, the rounded shape of the body will provide smoother flow in front of the bus. This condition will reduce the effect of frontal pressure. Figure 11 shows the maximum pressure (356.391 Pa) at the frontal area is lower than the frontal pressure for the reference model.

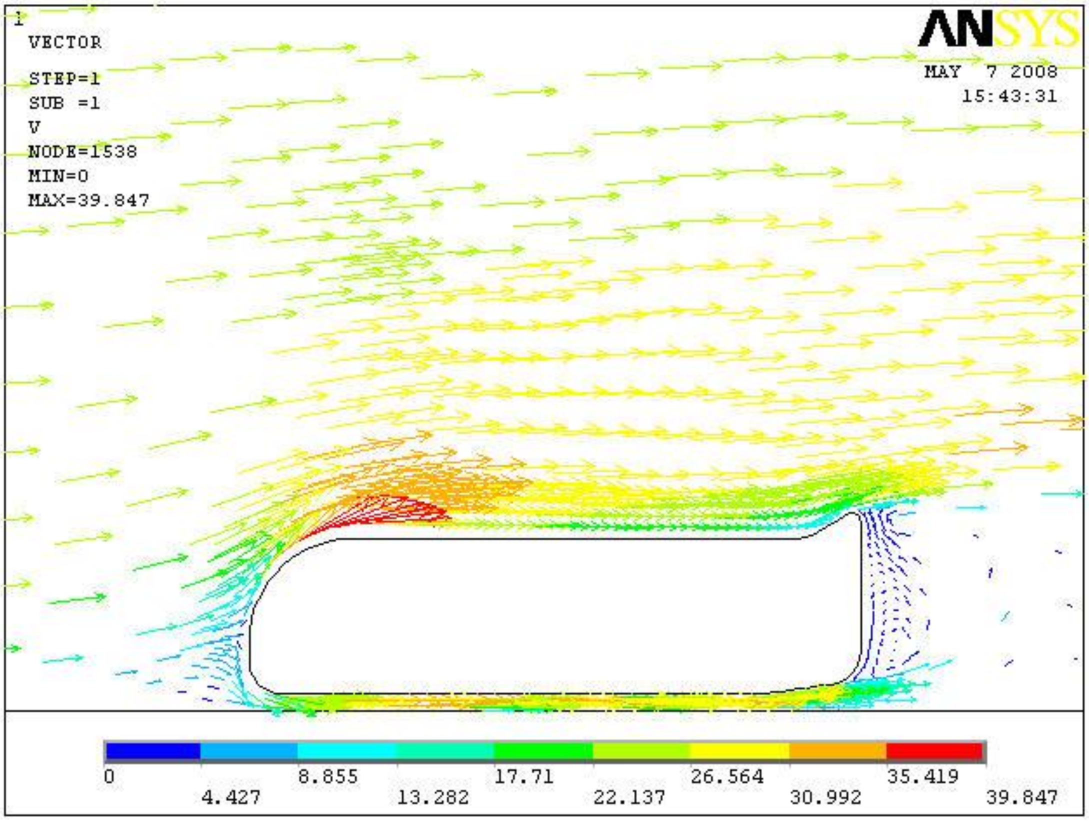

Figure 12: Velocity vector for the second modification

The frontal part of the second modification is the same as the first modification (see Figure 12). The changes made are a rear spoiler is added at the top of the rear body and inclination at the bottom part of the body. As the flow accelerates at the starting point in front of the bus, the inclination will act a diffuser for the flow at the bottom part of the rear body to decrease the velocity of the flow. Meanwhile, the purpose of adding the spoiler is to reduce the vortex effects at the back of the bus. The spoiler will limit the flow from the upper part of the body to meet at the end of the bus, thus decreasing the effect of vortex at the rear part of the bus.

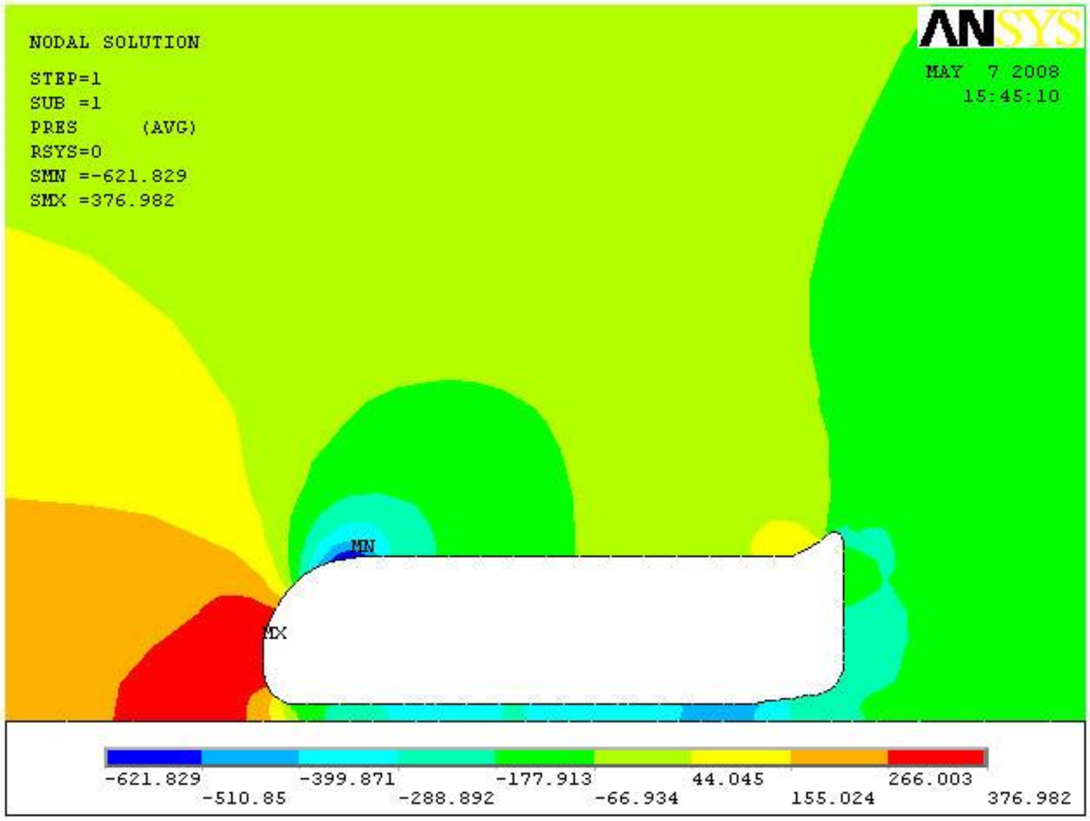

Figure 13: Pressure distribution along the body for second modification

As shown in Figure 13, the pressure around the spoiler is higher than any other part on top of the body. This condition will create more pressure to the rear tire, meaning the traction of the rear tire will increase. Therefore more power can be transmitted from the rear tire to the road. As for the front part of the body, the pressure to the front tire is less causing less traction to the front tire. Therefore the rolling resistance at the front will decrease.

For Figure 8, 10 and 12, the velocity contour is shown in Appendix D, where the velocity covering the whole boundary is shown.

4.2 Calculation

Estimation of Coefficient of Drag

The coefficient of drag, C_d can be estimated by calculating the mean values of pressure coefficient, C_p at the front and rear surface using:

$$C_d = \frac{D_{np}}{\frac{1}{2}\rho A V_\infty^{\ 2}} = \left(\overline{C}_{pf} - \overline{C}_{rf}\right)$$

The derivation of this equation is shown in Appendix B.

4.3 Result of Analysis

Table 3: Result of calculation

Shape	Drag Coefficient (C_d)	% reduction
Reference model	0.875	-
First modification	0.554	36.69
Second modification	0.535	38.86

From the analysis and calculation, the rectangle-shape body with sharp edges will give drag coefficient of 0.875. The second modification of the shape gives the lowest drag coefficient of all shapes, which is 0.535. The percentage of reduction of drag coefficient from the reference model is 38.86 percent.

4.4 Result of Wind Tunnel Experiment

The wind tunnel experiment is being conducted using the model of the second modification, due to the analysis result showing that the shape will give the lowest drag coefficient.

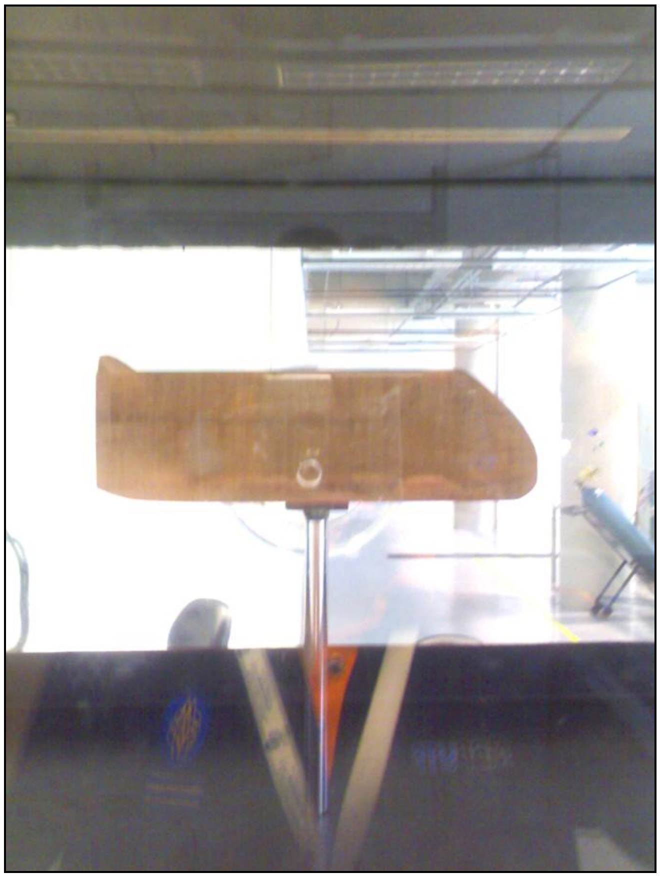

Figure 14: Second modification model in the test section

From the experiment, the second modification shape gives drag coefficient of 0.574. The result obtained is closed to the analysis result, which is 0.535 of drag coefficient. The percentage of difference between the analysis result and the experimental result is 6.79 percent.

The difference of the result between the computational calculation and the experimental results are probably due to the errors in dimension of the model, the surface finish of the model, the wind tunnel setup measurements or tolerance and friction inside the wind tunnel. In reality, the results from the analysis should be the same with the experimental data.

The error of dimensioning the model can be overcome by using precision measurement tools such as vernier caliper and smooth surface finish can be achieved by coating the model with paint.

CHAPTER 5
CONCLUSION AND RECOMMENDATION

5.1 Conclusion

Two new shapes for the body of the bus have been design. The designs are tested using the simulation. The drag coefficient of the current shape of the bus, which is represented by the reference model, is 0.875. By having more rounded shape for the bus, it smoothen the flow around the body of the bus. The rounded shape also decrease the pressure exerted on the frontal area of the body. Also by adding additional spoiler at the rear part of the bus, the vortex effects could be reduced. This is proven by the result, which shown that the second modification has the lowest drag coefficient (C_d = 0.535). The second modification model also showed that the rear spoiler can improve the traction force to the rear tire, which can increase the traction force transmitted to the ground.

The result is compared with the experimental result. The experimental result is obtained by putting a wooden model of the second modification shape in a wind tunnel. It is concluded that the result of the analysis is feasible. The analysis showed that the drag coefficient can be reduced for 38.86 percent from the initial drag coefficient. The experimental result showed that the drag coefficient is 0.574 and the percentage of difference is 6.79 percent.

5.2 Recommendation

It is recommended that the continuation of this project, one should modify the shape of the bus to have a hole through the body of the bus. By theory, this would equalize the pressure exerted at the frontal area to the rear section. The hole also can help reducing the rear vacuum at the rear by filling the rear area with air from the frontal of the bus.

For study of spoiler, the shape of the spoiler can be varied to observe the effect of different shape of spoiler. This can be done in terms of varying the height or the length of the spoiler.

REFERENCE

[1] Wikipedia, Automobile Drag Coefficients, Retrieved on 10th August 2007, http://en.wikipedia.org/wiki/Automobile_drag_coefficients

[2] Wikipedia, Computational Fluid Dynamics, Retrieved on 10th August 2007, http://en.wikipedia.org/wiki/Computational_fluid_dynamics

[3] Glen Research Center, What is Drag? for National Aeronautics and Space Administration. <http://www.grc.nasa.gov/WWW/K-12/airplane/drag1.html>

[4] Glen Research Center, What is Drag? for National Aeronautics and Space Administration. <http://www.grc.nasa.gov/WWW/K-12/airplane/shaped.html>

[5] Kuethe, Arnold M. and Chuen-Yen Chow (1998), Foundations of Aerodynamic: Bases of Aerodynamic Design, Fifth Edition, John Wiley &Sons, Inc. pg 11-13

[6] Preston, Ray. Retrieved on 14th August 2007, <http://142.26.194.131/aerodynamics1/Drag/Page2>

[7] http://media.nasaexplores.com/lessons/02-009/9-12_1.pdf

[8] Yunus A. Cengel, John M.Cimbala, Mc Graw Hill, *Fundamentals and Applications*, Fluid Mechanics,2006, pg 513-515

[9] Moaveni, Saeed. (1999), Theory and Application with ANSYS, Finite Element Analysis, Prentice Hall, pg. 216-234

[10] Colin R. Ferguson, Allan T. Kirkpatrick (2000), Internal Combustion Engine: Applied Thermosciences, John Wiley and Sons, Inc., pg 349

[11] SAE International (2003), Vehicle Aerodynamics 2003, Society of Automotive Engineers, Inc., pg. 341-350

[12] Barlow, Jewel B., Rae, William H., Alan Pope (1999), Low-Speed Win Tunnel Testing, John Wiley & Sons. Pg 19-22

[13] WT04 Sub-sonic Wind Tunnel Operating & Experimental Manual, PETRONAS University of Technology (UTP)

APPENDIX A

Flow Similarity

In principle, it is necessary to find the appropriate solution to the system of partial differential equations with the associated boundary and initial conditions. The complete geometry of the body including any time-dependent motion is required to specify the boundary conditions at the body [12].

To explore the similarity for flows, these set of equations will be considered:

1) Conservation of mass, written in partial differential equation [12]

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho V) = 0$$

2) The Navier-Stokes equation for the case of a viscous compressible fluid with body force of gravitational origin [12]

$$\rho\left(\frac{\partial V}{\partial t} + (V \bullet \nabla)V\right) = \rho g - \nabla\left(p + \frac{2}{3}\mu\nabla \bullet V\right) + 2\nabla \bullet (\mu\dot{S})$$

APPENDIX B

Using pressure coefficient to estimate drag coefficient

$$\bar{C}_p = \frac{1}{A}\int C_p dA\,;$$

where $C_p = \dfrac{p - p_\infty}{\frac{1}{2}\rho v_\infty{}^2}$

From Bernoulli Equation:

$$p_\infty + \frac{1}{2}\rho v_\infty{}^2 = p + \frac{1}{2}\rho v^2$$

Rearranging

$$p - p_\infty = \frac{1}{2}\rho v_\infty{}^2 - \frac{1}{2}\rho v^2$$

Hence

$$C_p = 1 - \frac{v^2}{v_\infty{}^2}$$

Pressure drag is given by

$$D_{np} = \frac{1}{2}\rho A V_\infty{}^2\left(\bar{C}_{pf} - \bar{C}_{rf}\right)$$

Therefore the drag coefficient is

$$C_d = \frac{D_{np}}{\frac{1}{2}\rho A V_\infty{}^2} = \left(\bar{C}_{pf} - \bar{C}_{rf}\right)$$

APPENDIX C

Solution Procedure of Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a field of study devoted to solve fluid flow equations through the use of computer. Modern engineers use both experimental analysis with CFD analysis, thus experimental data are often used to validate CFD solutions by matching the computationally and experimentally determined global quantities. CFD is employed to shorten the design cycle through carefully controlled parametric studies, thereby reducing the required amount of experimental testing [8].

Below are the general steps of using the CFD software to calculate the fluid flow [8]:

1. A computational domain is chosen and a grid (also called mesh) is generated; the domain is divided into many small elements called cells. For 2-dimensional (2-D) domains, the cells are areas, while for the 3-dimensional (3-D) domains, the cells are volumes. Each cell is considered a tiny control volume in which discretized versions of the conservation equations are solved.
2. Boundary conditions are specified on each edge (2-D) or face (3-D) of the computational domain.
3. The type of fluid is specified, along with the fluid properties (density, viscosity). Many CFD codes have a built-in property databases for common fluids, making this step relatively easy.
4. Numerical parameters and solution algorithms are selected. These are specified to each CFD code. The default settings of most modern CFD codes are appropriate for the simple problems.
5. Starting values for all flow field variables are specified for each cell. These are initial conditions, which may or may not be correct, but are necessary as a starting point.

6. Beginning with the initial guesses, descritized form of equations is solved iteratively, usually at the center of the cell. In a CFD solution, the sum is never identically zero, but decreases with progressive iterations. A residual can be thought of as a measure of how much the solution to a given transport equation deviates from the exact, and one monitors the average residual associated with each transport equation to help determine when the solution has converged. Sometimes hundreds or even thousands of iterations are required to converge on a final solution and the residuals may decrease by several orders of magnitude.
7. Once the solution has converged, flow field variables such as velocity and pressure are plotted and analyzed graphically. Users can also define and analyze additional custom functions that are formed by algebraic combinations of flow field variables. Most commercial CFD codes have built in postprocessors, designed for quick graphical analysis of the flow field. There are also stand-alone postprocessor software packages available for this purpose. Since the graphics output is often displayed in vivid colors, CFD has earned the nickname colorful fluid dynamics.
8. Global properties of the flow field, such as pressure drop and integral properties, such as forces (drag and lift) and moments acting on a body, are calculated from the converged solution. With most CFD codes, this can be done "on the fly" as the iterations proceed. In many cases, in fact, it is wise to monitor these quantities along with the residuals during the iteration process; when a solution has converged, the global and integral properties should settle down to constant values as well.

APPENDIX D

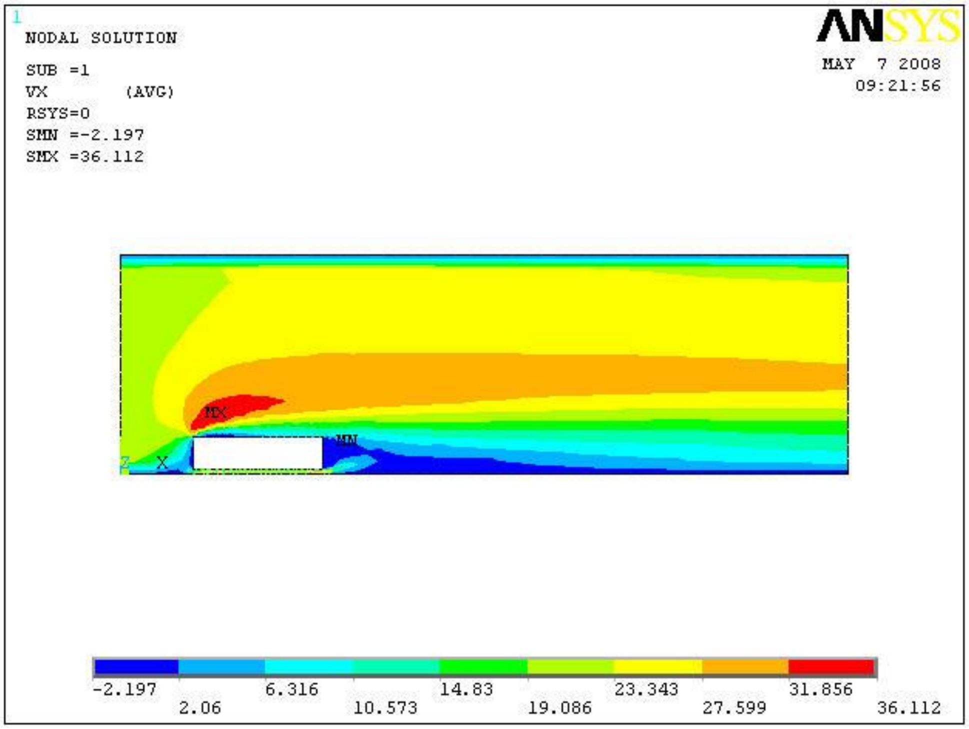

Figure D-1: Velocity contour for the reference model

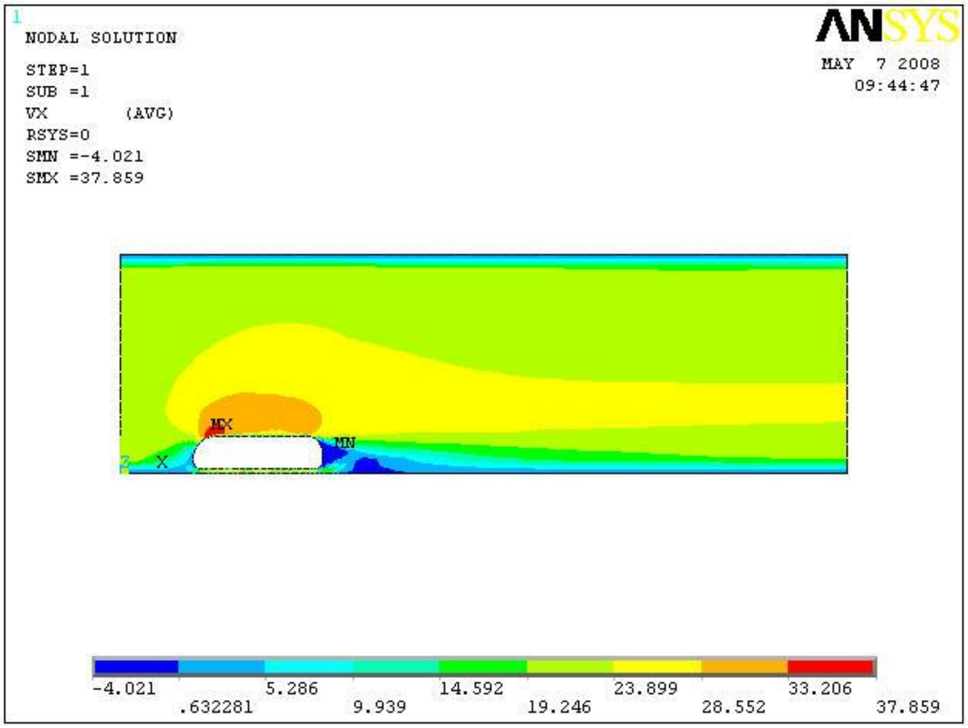

Figure D-2: Velocity contour for the first modification model

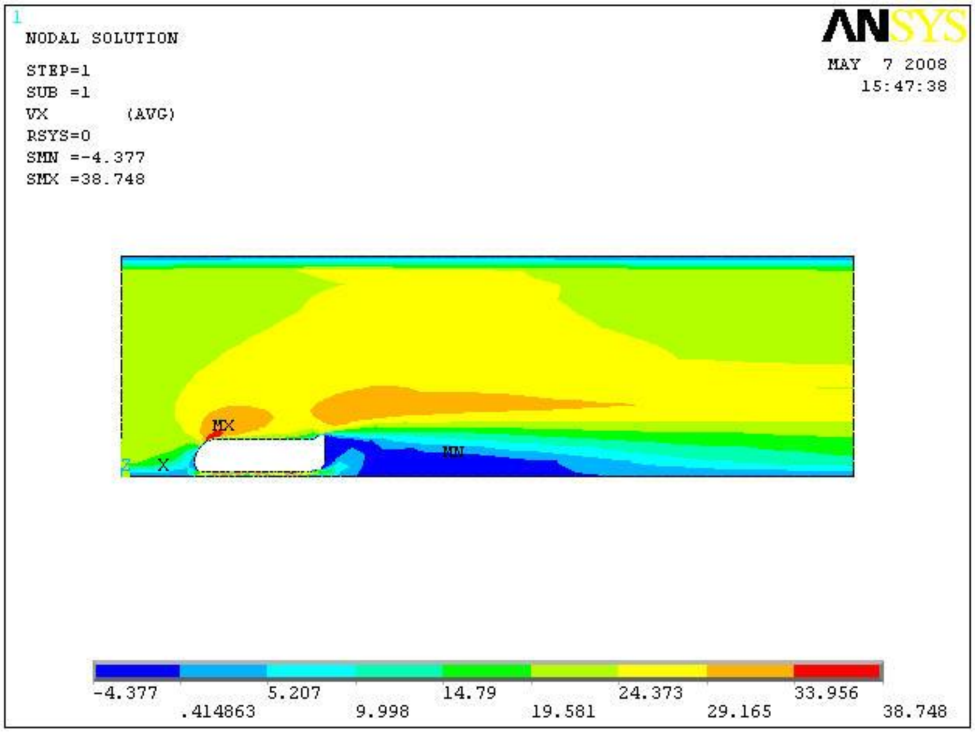

Figure D-3: Velocity contour for the second modification model