Simulation of Flow around a Bus with Passive Drag Reduction Attachments

by

Ikhwan Solihin Mat Zuki (5776)

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

June 2009

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

LECTURER: MR. RAHMAT ISKANDAR

CERTIFICATION OF APPROVAL

SIMULATION OF FLOW AROUND A BUS WITH PASSIVE DRAG REDUCTION ATTACHMENTS

By

Ikhwan Solihin Bin Mat Zuki

A project dissertation submitted to the

Mechanical Engineering Programme

University Teknologi PETRONAS

In partial fulfilment of the requirement for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

Approved:

Rahmat Iskandar Khairul Shazi Shaarani

Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

June 2009

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 person

Ikhwan Solihin bin Mat Zuki

ABSTRACT

In this study, a numerical simulation has been carried out for three-dimensional turbulent flows around a bluff-based bus-like body and actual bus body. The first step of this study is to verify the effectiveness of the CFD analysis. In the second step, to reduce the drag of the actual bus model, parameter studies are performed with attention to effective utilization of the rear-spoiler equipped at the roof-end of upper body. . The study is being conducted to get clear view on how the aerodynamics changes when attaching attachments (spoiler) to reduce the aerodynamic drag for the buses. The effect of a rear-spoiler attached at the rear end of the bus body will be investigated.

At the early stage, it is crucial to seek for relevant information regarding the project. Previous experiments report, journals, books and all supporting information will be reviewed for the mean of having clear view on this topic. In the design stage, calculations, modeling, and experiments will takes place. Simulation with computational fluid dynamics (CFD) software will produce initial data before experiments conducted. Similarity between model and prototype must be obtained as it is very important to maintain the accuracy of the data gained from experiments.

Few series of experiments conducted in regards of a different shape of attachments. The drag force primary data as minimizing it result in increasing the aerodynamic performance for buses. All the results will be interpreted and discussed.

Towards the end of this semester the models of bus with passive drag reduction attachments finalized. Final reports and presentations conducted as the research completed. All the objectives are expected to be met during the project.

ACKNOWLEDGEMENTS

The author wishes to take the opportunity to express her utmost gratitude to the individual that have taken the time and effort to assist the author in completing the project. Without the cooperation of these individuals, no doubt the author would have faced some minor complications throughout the course.

First and foremost the author's utmost gratitude goes to the author's supervisor, Mr. Rahmat Iskandar . Without his guidance and patience, the author would not be succeeded to complete the project.

The author would also like to thank all the technical staff of Mechanical Engineering Department especially Mr. Zailan, Mr. Jani and who have been very cooperative in helping and sharing their experience throughout the experimental work.

Here the author would like to thank the author's family, Mr. Alex, Ms Aziewati Satin for the helping, their encouragement and understanding until to the completion of this project

To all individuals that has helped the author in any way, but whose name is not mentioned here, the author thank you all.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 Background of the study

A bus is a road vehicle designed to carry passengers. A bus can generally seat a maximum of anywhere from 8 to 300 passengers. Buses are the most widely used form of public transportation, although they are also used in tourism and as private transport. It needs volume to carry these passengers which become a constraint to design an aerodynamic coach.

Automotive aerodynamics differs from aircraft aerodynamics in several ways. First, the characteristic shape of a road vehicle is bluff, compared to an aircraft. Second, the vehicle operates very close to the ground, rather than in free air. Third, the operating speeds are lower. Fourth, the ground vehicle has fewer degrees of freedom than the aircraft, and its motion is less affected by aerodynamic forces.

Automotive aerodynamics is studied using both computer modeling and wind tunnel testing. For the most accurate results from a wind tunnel test, the tunnel is sometimes equipped with a rolling road. This is a movable floor for the working section, which moves at the same speed as the air flow. This prevents a boundary layer forming on the floor of the working section and affecting the results. The main concerns of automotive aerodynamics in order to reduce the passive drag are as follows:

i. Reducing drag
ii. Reducing wind noise
iii. Minimizing noise emission
iv. Preventing flow separation

1.2 Problem Statement

Nowadays, one of big concern is our world is the petroleum fuel and diesel price hike unexpectedly. Seems the price is slightly growth, so it becomes major problem to the bus company since their profit is declining. One of the solutions to this problem is to increase the efficiency of the bus by reducing the passive drag and aerodynamically designing to the maximum level. It is believed that with this design and solution, it can reduce the fuel consumption and at the same time reduce the operation cost of the bus.

However some of the above aerodynamics design cannot be implement in heavy vehicles such as buses and trucks due to requirement for big volume. This is where this project is significant as the project will look at reducing drag by designing attachments to the bus. The aerodynamic performance of the bus will improve as the result of minimizing the drag. The reduction in drag will help reduce the power required to travel at a set speed, reducing its fuels consumption and hence operating cost. Improving fuel efficiency at highway speeds, where aerodynamic effects represent a substantial fraction of the energy needed to keep the bus moving. About 60% of the power required to cruise at highway speeds is taken up overcoming air drag, and this increases very quickly at high speed [2].

Therefore, a vehicle with substantially better aerodynamics will be much more fuel efficient. Additionally, because drag does increase with the square of speed, a somewhat lower speed can significantly improve fuel economy.

1.3 Objectives

Upon the complition of this project a few objectives need to be achived. The objectives of the study are as follows:

- To study the aerodynamics performance of a bus.
- To investigate the effect of aerodynamics changes with attachments (spoiler) at the rear of bus.
- To design the optimum shape of attachments to typical buses that can reduce the passive drag.

1.4 Scope of Study

The scope of this study will cover the study of aerodynamic performance of the bus with attachments compare to typical bus design. This experiment involve the designing and fabrication of a model of bus, calculation for passive drag and identifying other possible element in order to reduce the drag force as well as upgrading the effectiveness. The literature review that involves this experiment includes laminar and turbulent flows, passive drags, dimensional analysis. The flow that will be visualized around the bus will be analyzed. Similarity between model and prototype must be obtained as it is very crucial by calculation using Reynold's number. The project also concern about the result of testing to validate the simulated data. Finally the project will lead to the planning of fabrication of the attachments for testing on full-scale (refer to Figure 1.1).

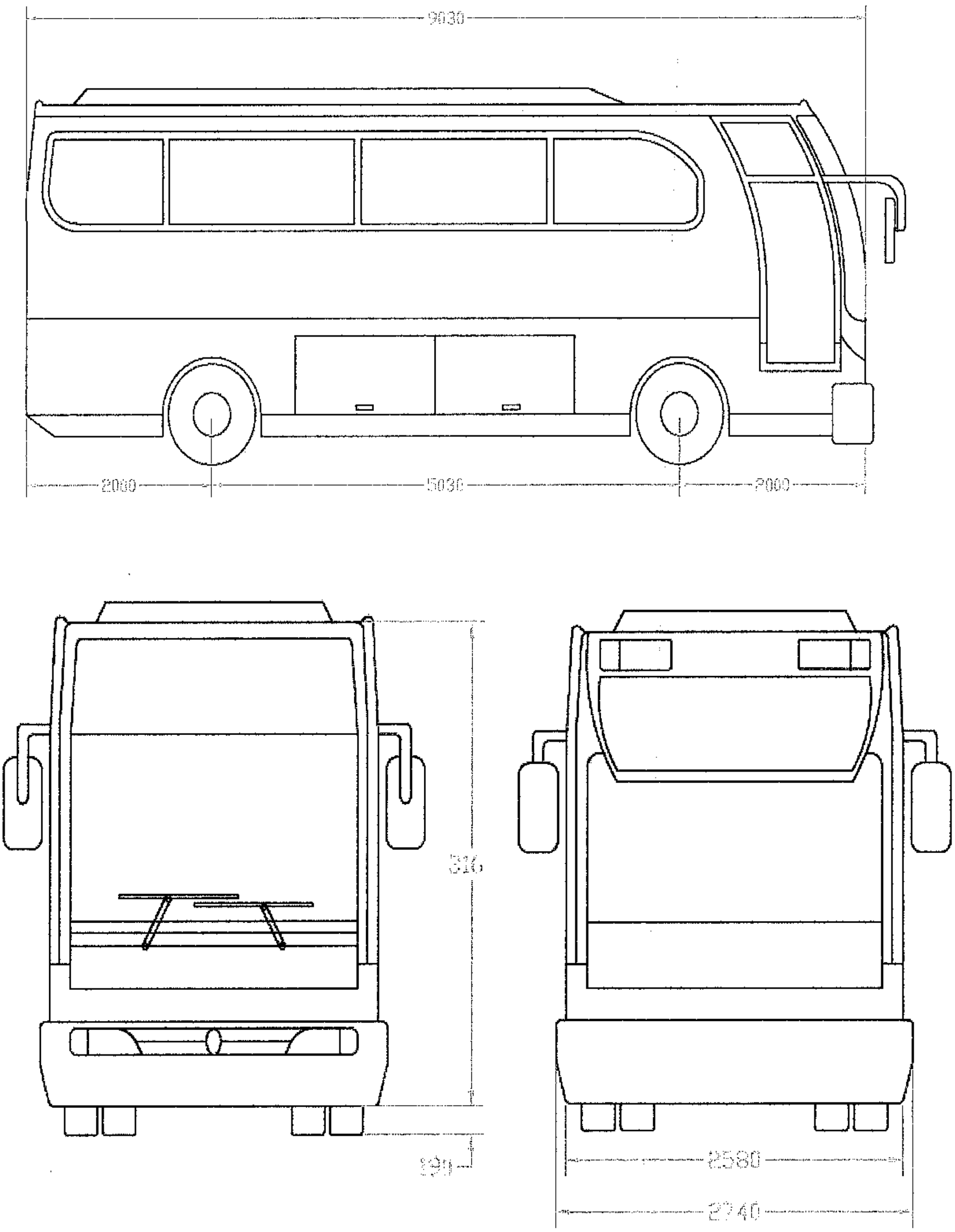


Figure 1.1: Full scale bus (unit in centimeter)

CHAPTER 2

LITERATURE REVIEW AND THEORY

2.1 Bus Drag Reduction

A wind-tunnel test was conducted to lead to an investigation of a 1/20 scale model of a modified Denning Mono Mark II bus indicates that the drag coefficient can be reduced to 0.29 by using optimal radii on the fore body /roof junction, 8 ° tapers behind the fore body and in front of the tail and incorporating a vortex trapping cavity in the tail. Surface pressure distributions and surface-mounted tufts on two fore bodies and two tails provide the detailed flow behaviour. For the vortex cavity tail a vortex is well-defined over the top half of the tail but the under body up wash interferes with the vortex flow over the lower half.

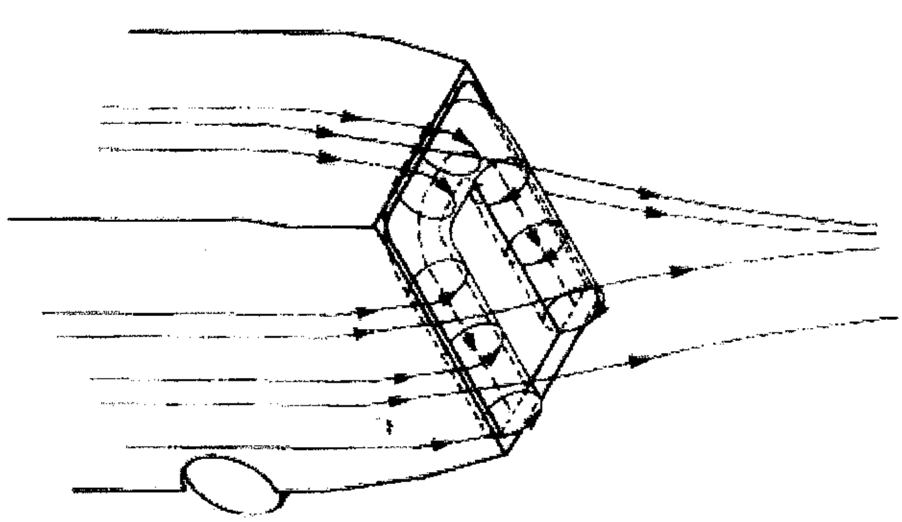


Figure 2.1: Conceptual flow field associated with a trapped horseshoe vortex [13].

The concept is shown in Figure 2.1 and has been discussed previously, in relation to automobile drag reduction. The same idea, in equivalent two dimensional forms, is evident in the appearances of cornices in wind-driven snow-fields [13].

2.2 Flow Visualization Of The Drag Reduction

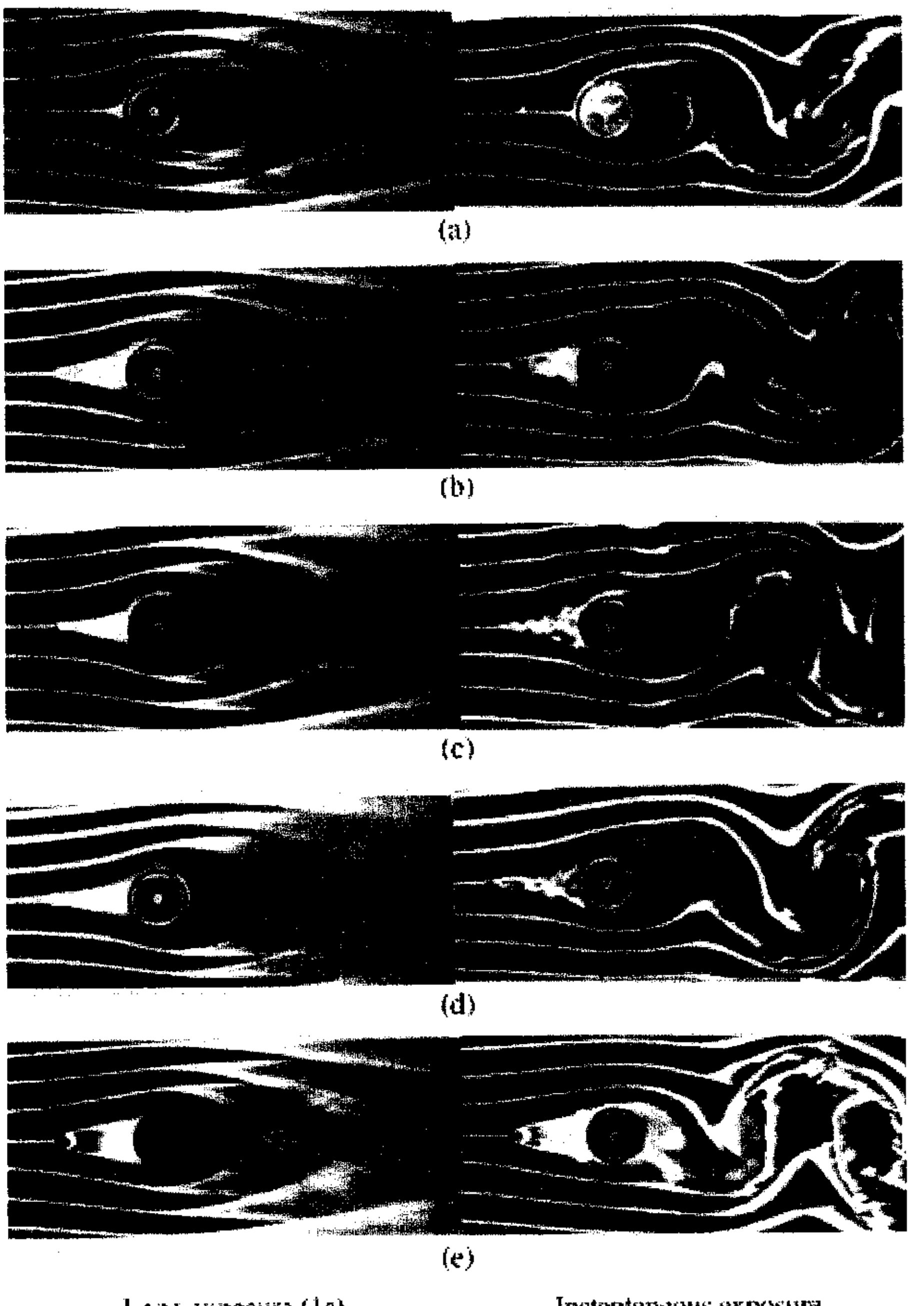


Fig. 2.2: Flow visualization by smoke tunnel [15].

An experiment was conducted lead to the Typical examples of the flow visualization are shown in Figure 2.2 (a)–(e), the left-side and right-side photographs are obtained under long and instantaneous exposures, respectively. Figure 2.2 (a) shows the flow around a circular cylinder without a rod. For the circular cylinder with the rod, two flow patterns A and B corresponding with and without vortex shedding from the rod occurred. Figure 2.2 (c) and (d) are of pattern A with vortex shedding from the rod. The front face of the circular cylinder is exposed to vortices from the rod. Figure 2.2 (b) and (e) are of pattern B, the separated shear layer from the rod reattaches on the front face of the circular cylinder and quasi-static vortex is formed between the rod and the circular cylinder. Figure 2.2 (b) and (c) show that the flow changed from Pattern A to B by increasing Reynolds number. In Figure 2.2 (b) and (d), and Figure 2.2 (d) and (e), the flow changed from pattern B to A and from pattern A to B by increasing L=D and d=D; respectively. These facts indicate that d=D; L=D and the Reynolds number are the dominant factors for the flow patterns [15].

2.3 Aerodynamics on Vehicle

Aerodynamics (shaping of objects that affect the flow of air or gas) is a branch of fluid dynamics concerned with the study of forces generated on a body in a flow. The solution of an aerodynamic problem normally involves calculating for various properties of the flow, such as velocity, pressure, density, and temperature, as a function of space and time [1]. Understanding the flow pattern makes it possible to calculate or approximate the forces and moments acting on bodies in the flow.

2.4 Drag and Lift

Body meets some resistance when it is force to move through a fluid (liquid or gas). A fluid may exert forces and moments on a body in and about various directions. The force a flowing fluid exerts on a body in the flow direction is called drag. A stationary fluid exerts only normal pressure forces on the surface of a body immersed in it. A moving fluid, however also exerts tangential shear force on the surface because of the no slip condition cause by viscous effect. Both of these forces, in general, have components in the direction of flow, and thus the drag force is due to the combined effects of pressure and wall shear forces in the flow direction. The components of pressure and wall shear forces in the direction normal to the flow tend to move the body in that direction, and their sum is called lift [2].

The drag and lift force depend on the density of the fluid ρ, the upstream velocity V, the size, the shape, and orientation of the body.

$$C_D = \frac{F_D}{0.5\,\rho V^2 A}$$

$$C_L = \frac{F_L}{0.5\,\rho V^2 A}$$

2.5 Flow Separation

At sufficiently high velocities, the fluid stream detaches itself from the surface of the body. This is called flow separation. Flow can separate from a surface even it is fully submerge or immersed in the fluid. The location of the separation points depend on several factors such as Reynolds number, the surface roughness, and the level of fluctuations in the free stream, and it is usually difficult to predict exactly where the separation will occur unless there are sharp corners or abrupt changes in the shape of the solid surface [4].

When a fluid separates from a body, it forms a separated region between the body and the fluid stream. This low-pressure region behind the body where recirculation and backflows occur is called the separated region (refer to Figure 2.3). The larger the separated region, the larger the pressure drag. The region of flow trailing the body where the effects of the body on velocity are felt is called the wake.

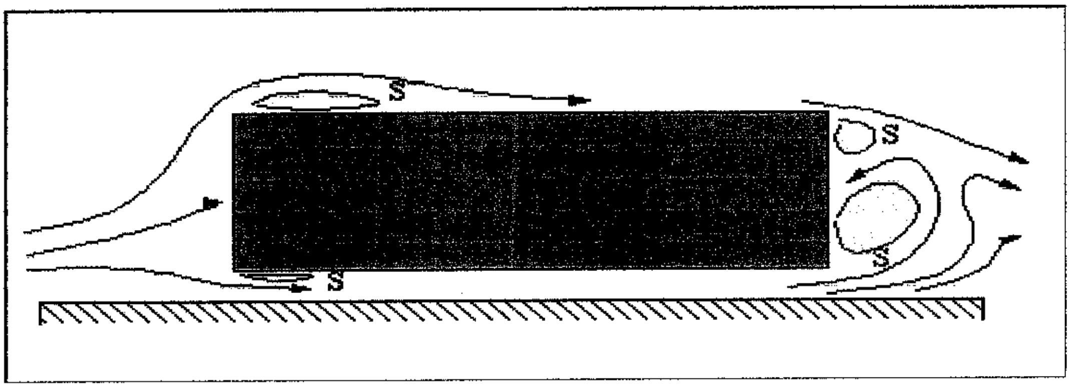


Figure 2.3: Flow separation around a bus

2.6 Spoiler

A wing is a surface used to produce lift and therefore flight, for travel in the air or another gaseous medium (Refer to figure 2.4).The wing shape is usually an airfoil. Down force is the opposite of lift. This is the result of Bernoulli's effect. Bernoulli's effect explains how fluids, in this case the air, will react when traveling over the wing surface. Lift occurs due to a difference in pressures on opposite sides of airfoils caused by this effect. The wings on race cars are essentially wings flipped upside-down so that the lifting force is directed in a downward direction.

From the study of the design, spoilers or inverted airfoils can be use in the opposite way on racing car to avoid lift or generating negative lift to improve traction and control.

The rear wing is designed to provide an enormous down force focused to the rear tires. This gives the vehicle great traction for acceleration and turning. However, by creating more down force by the wing, it also creates more drag. Therefore, it is necessarily to design the wings for the best drag/down force compromise.

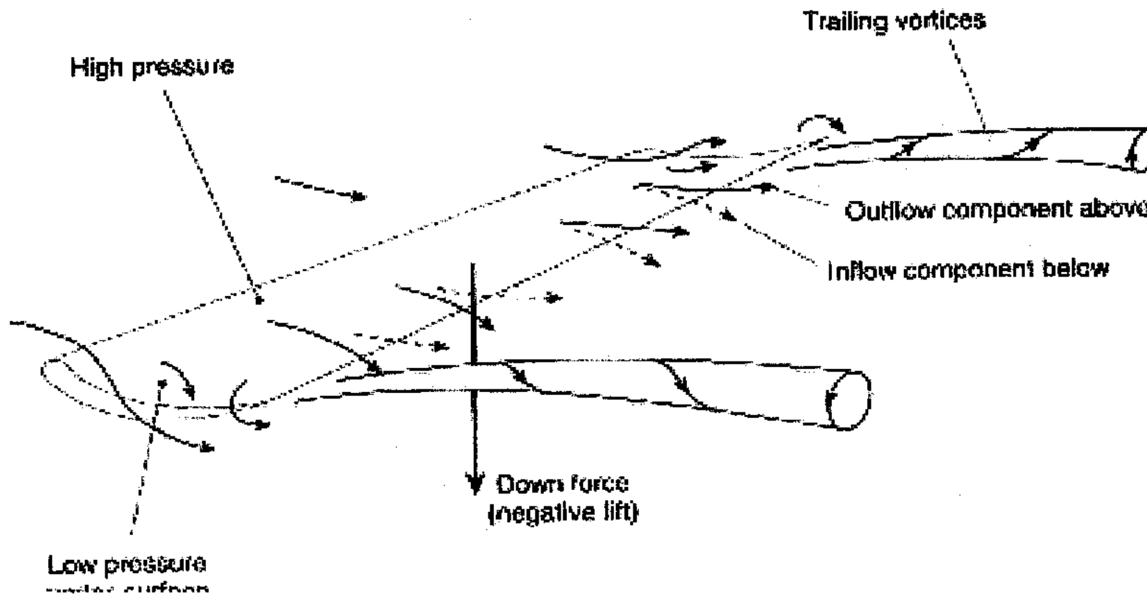


Figure 2.4: Spoiler

The amount of down force produced by a spoiler is determined by its size. The larger the wing, the greater the down force. The length/width ratio is called the aspect ratio. The aspect ratio is the span of the airfoil (the long dimension perpendicular to the airflow) divided by its chord (the dimension parallel to the airflow).

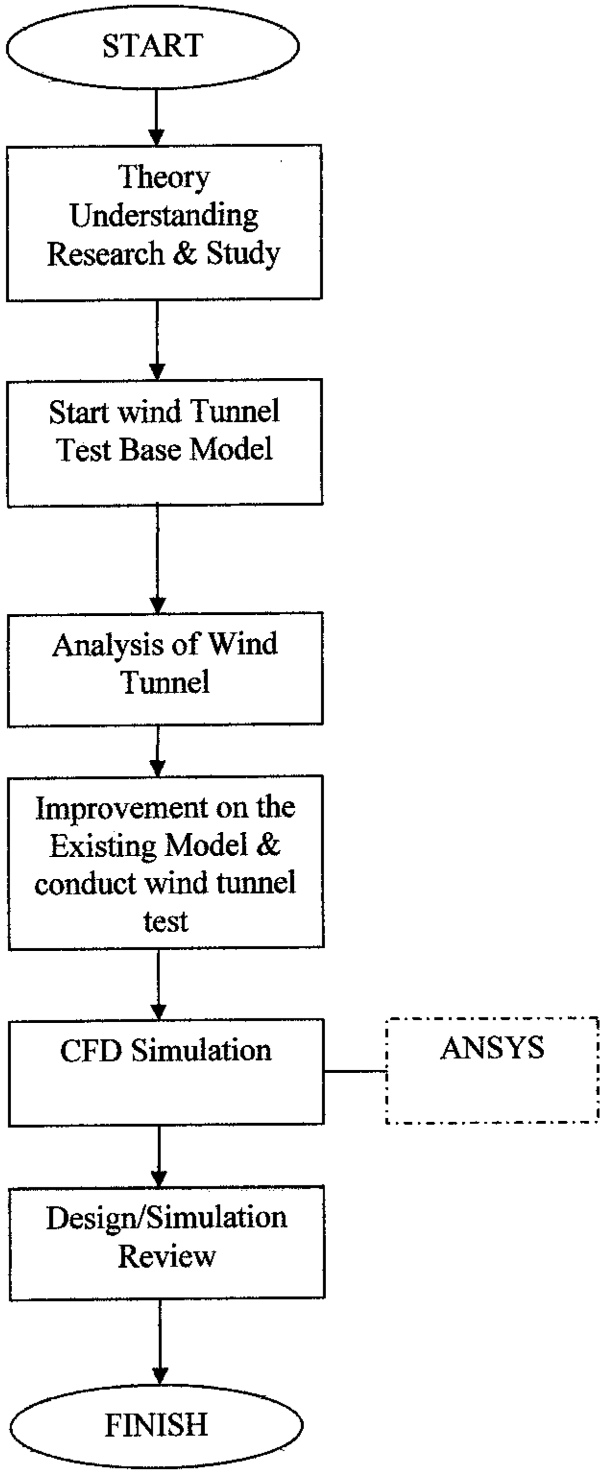
The efficiency of a spoiler is its down force/drag ratio. The amount of down force generated is dependent upon the angle or the tilt of the wing. The angle of attack is greater on a road course rear wing setup than on a speedway setup. The greater the angle of attack, the more down force and drag.

While increasing down force a wing also increases unwanted drag. Drag increases with the angle of attack. The down force generated by the wing works in a vertical, downward direction, while drag acts in the opposite direction.

Front spoilers, found beneath the bumper, are mainly used to direct air flow away from the tires to the underbody where the drag coefficient is less. Rear spoilers, which modify the transition in shape between the roof and the rear and the trunk and the rear, act to minimize the turbulence at the rear of the vehicle. Adding a rear spoiler makes the air "see" a longer, gentler slope from the roof to the spoiler, which helps to delay flow separation. This decreases drag and increases fuel economy.

CHAPTER 3

METHODOLOGY



3.1 This project will be conducted through several steps.

1.) Information gathering regarding the aerodynamic performance of bus.
2.) Information gathering regarding the drag force, passive drag and down force for bus.
3.) Improvement of surface finish for existing models and attachments.
4.) Conducting experiments on the existing models by using wind tunnel to investigate the aerodynamically changes. Wind tunnel testing will investigate the difference in attachments shape and to investigate comparison between before and after improvement of models surface finish. All the similarities will be followed to have accurate data.
5.) All the data will be discussed and interpreted.
6.) Design review and simulate the model.

3.2 Tools required for this project

1) Software:
    a. CATIA software for modelling
    b. ANSYS software for simulation
2) Reading material as references; books, journals, internet web page.
3) UTP wind tunnel.

3.3 Universiti Teknologi PETRONAS (UTP) Wind Tunnel – WT04

The UTP wind tunnel is the equipment that will be used in the experiment steps of this project. The WT04 Subsonic Wind Tunnel is a suction type open circuit wind tunnel designed to conduct teaching and basic research experiment related to fluid mechanics and aerodynamics. Wind tunnel test speeds up to about 60 m/s can be obtained. Tunnel operation, control, data acquisition and processing of data are through a computer based system.

Below are the important features and the specification of the UTP wind tunnel:

Important features and applications:

- Suction type open circuit wind tunnel
- Test velocity range : 0-60 m/s
- Total head variation : within +/- 0.5%
- Mean flow uniformity : within +/- 0.25%
- Longitudinal turbulence : within 0.15%
- Test section: 0.3m x 0.3m x 1.5m long, with glass side window
- Bell mouth entry : 1.5m x 1.5m
- Dust filter: Ashrae dust gravimetric efficiency: 97.5%
- Settling chamber with honey comb (200mm deep and 15mm mesh)
- Contraction, contraction ratio: 25:1

Overall dimensions:

- Length: 6.5 m
- Width : 1.75m
- Height: 2.5m
- Required services: electric supply, 451V, 3 Ph. 50Hz

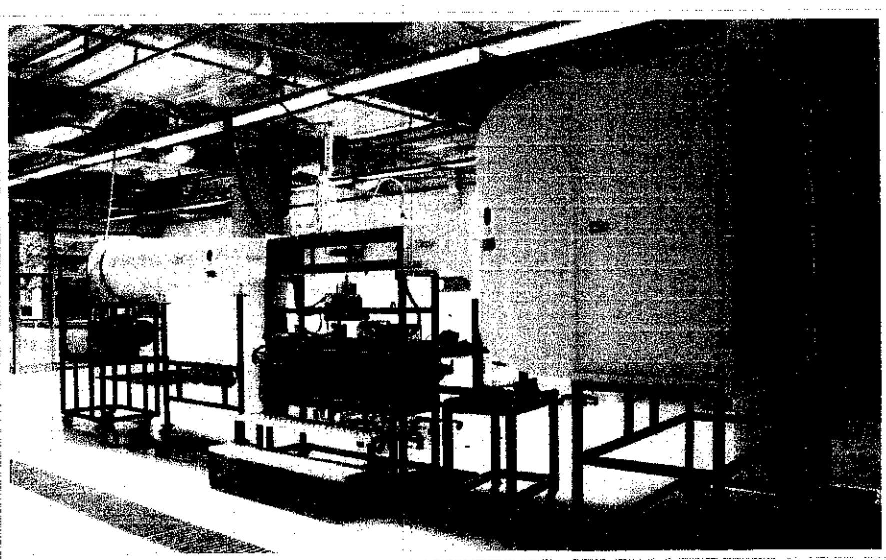
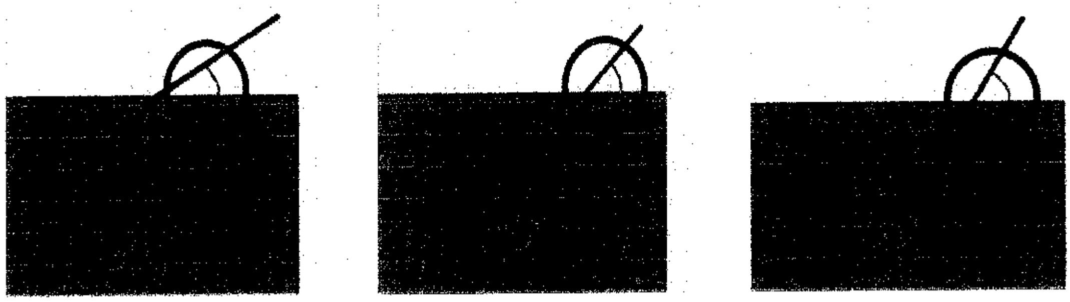


Figure 3.1: UTP wind tunnel (WT04)

3.4 Wind Tunnel Test

Using the improved existing models the author has done the wind tunnel test which is the experiment for this project. The test will help author in better understanding on effect of attachments on improving the aerodynamic of the bus. Through the data obtain author will compare it with previous data to prove the spoiler is a crucial factor in reducing the drag on buses in the same time maintain the similarities with the actual buses.

There is four models will be test in this experiment, Base Model (without attachments), Model 1, Model 2, and Model 3. The author runs the test using five different velocities for each model. The velocities used in this test are around 10 m/s to 50 m/s. in the wind tunnel test; the results obtained are the drag force (F_d) of the each bus model and also the lift force (Fl) of each bus model. However, only the drag force will be consider in this project.



Model 1 (30°) Model 2 (45°) Model (60°)

Figure 3.2: Models different angle of spoiler at the tail of the bus

3.5 ANSYS familiarization

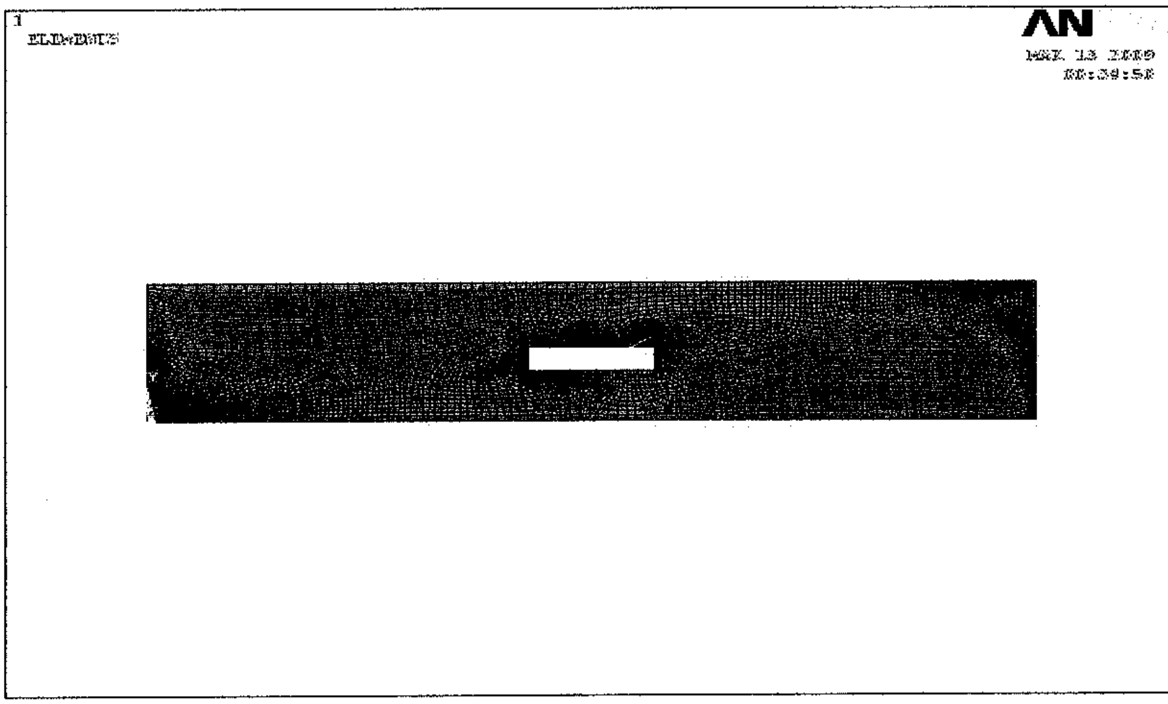


Figure 3.3: Meshing

Before meshing the model, and even before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A *free* mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it [7].

Compared to a free mesh, a *mapped* mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either only quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. In addition, a mapped mesh typically has a regular pattern, with obvious rows of elements. If you want this type of mesh, you must build the geometry as a series of fairly regular volumes or areas that can accept a mapped mesh.

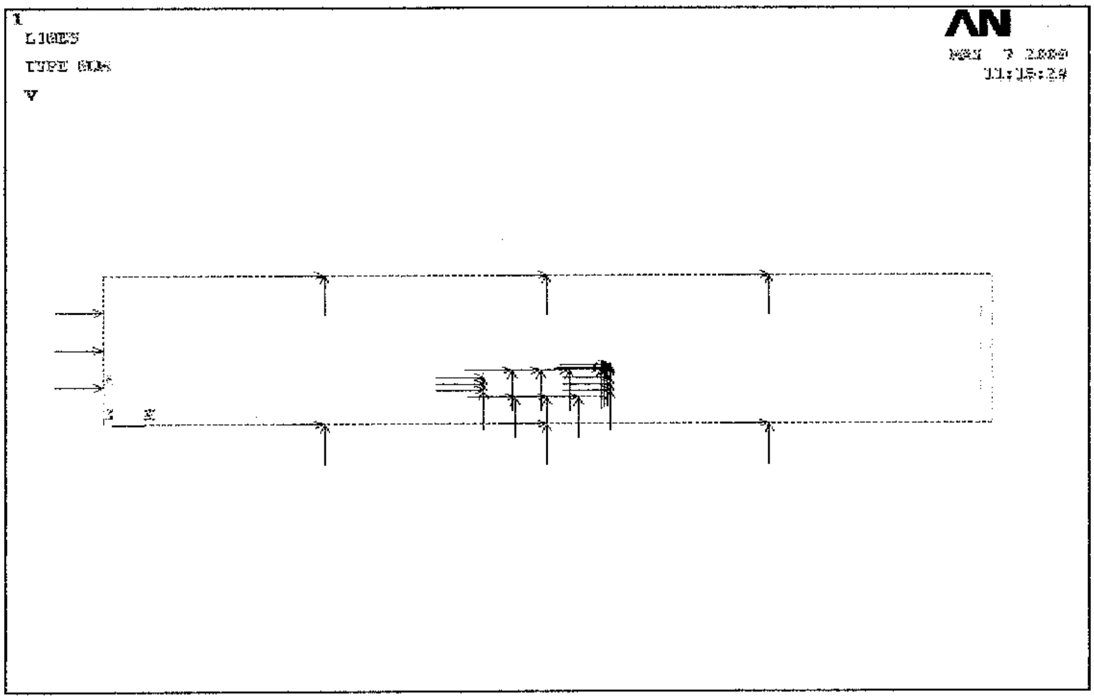


Figure 3.4: Apply load after the meshing

Figure 3.4 shows the load is apply to the model after the meshing. The loads are the fluid velocity and the pressure. As can see there, the velocity 40m/s is appointed at the inlet of the wind tunnel test section and at the wall and at the body of model, the velocity are 0 m/s because no slip condition to that boundary layer. The pressure is behind the test section set it to 0.

The direction of the velocity is from the left to the right as the model is facing to the inlet section. This is important to see the effect of the fluid velocity variation against the spoiler.

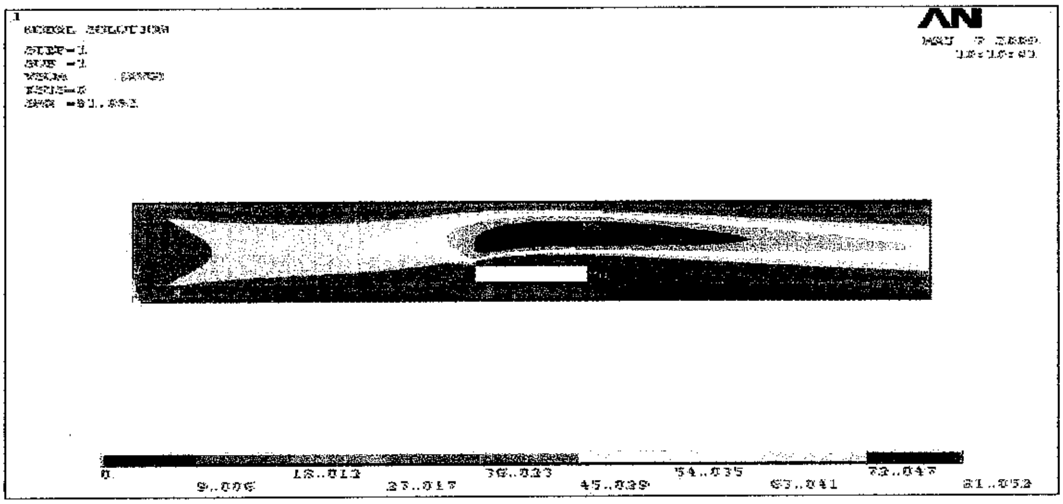


Figure 3.5: Fluid velocity (simulation of the base model)

Figure 3.5 shows the simulation of the base model. This simulation views the variation of the fluid velocity through the base model. The speed of velocity can be view at the inlet section at 40 m/s on the left sight. The red region is indicating the high velocity while the blue region is for the slow velocity.

As the flow moving through the base model, the variation is changing due to the resistance of the model.

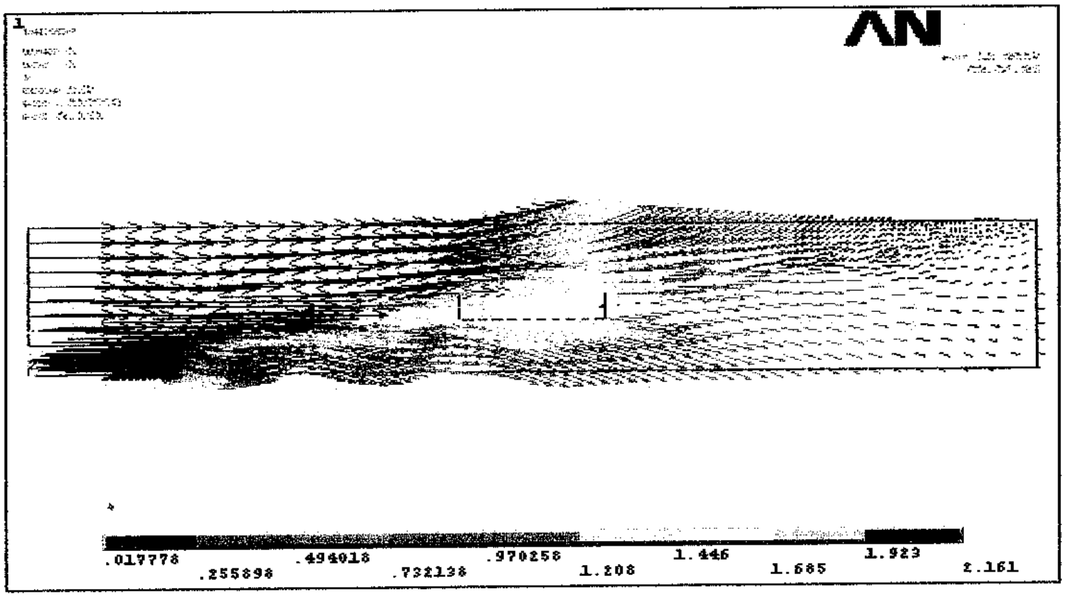


Figure 3.6: Vector plot (simulation of the base model)

Figure 3.6 shows the vector plot of the fluid velocity through the base model. The vector plot is to show direction of the fluid moving through. When the high speed velocity moving through the body, the direction of the fluid a bit reflect upwards due to the resistance of the body

CHAPTER 4

RESULT AND DISCUSSION

4.1 Sensitivity of Results to Reynold's Number.

The objective of this experiment is to investigate the relation of Reynold's Number with Drag Coefficient. The experiment is carried out at 10 m/s, 20 m/s, 30 m/s, 40 m/s, 50 m/s and 57 m/s (maximum) Free Stream Velocity. It is a need to perform similarity calculation between a prototype and a model of a bus to fulfill a few requirements of the experiment. The requirements include the practical speed of actual bus

Similarity equation:

$Re_p = Re_m$ .... (1)

Re_m = Reynold's Number for model

Re_p = Reynold's Number for prototype

$Re_p = [(\rho_{air}) \times (V_p) \times (L_p)] / (\mu_{air})$ .... (2)

$Re_m = [(\rho_{air}) \times (V_m) \times (L_m)] / (\mu_{air})$ .... (3)

ρ_{air} = Density of air in Kg/m^3

V_p = Required Velocity of prototype in m/s

V_m = Required Velocity of model in m/s

L_p = Length of prototype in m

L_m = Length of model in m

μ_{air} = Viscosity of air in Kg/m s

The objective is to find V_m, the required Velocity for model to be tested in wind tunnel. So by re-arranging the equation (1), (2) and (3),

$V_m = [(\rho_{air}) \times (V_p) \times (L_p) \times (\mu_{air})] / [(\mu_{air}) \times (\rho_{air}) \times (L_m)]$.... (4)

$V_m = [(V_p) \times (L_p)] / (L_m)$... (5)

Similarity calculation:

For the similarity calculation, the same density and viscosity are used because both model and prototype experienced the same medium which is air. So, from equation (4), density and viscosity values can be eliminated. Scaled model can be used to define the length of a model and its real prototype. For this experiment, 1: 30 scaled is used, means that 1 unit length of a model is equal to 30 unit length of its prototype. For a standard speed of a bus the author uses 80km/hr which is equal to 22.2 m/s. As the WT04 Sub-sonic wind tunnel only can produce 57 m/s maximum Free Stream Velocity, it is acceptable to use 50 m/s as the in-flight (in wind tunnel) for this similarity calculation.

$80 \times 1000/3600 = 22.2$ m/s

$V_p = 22.2$ m/s

$L_p = 14$ m

$L_m = 0.2$ m

So, from the equation (5), $V_m = 1554$ m/s

Table 1: Model scaled at 1: 30

Model	Prototype
1	30

Table 2: Results obtained from calculations

V_p	22.2 m/s
V_m	1554 m/s

From the results obtained of the similarity calculation, obviously shown that the requirement speed of an bus model V_m, is much more higher than the capability of WT04 Sub-sonic Wind Tunnel which can only produced 60 m/s Free Stream Velocity and it is beyond the capability of WT04 Sub-sonic Wind Tunnel.

The similarity analysis has shown the need of high velocity as requirement to test a model in WT04 Sub-sonic Wind Tunnel Demonstration Unit. The requirement will not be fulfilled due to constraint of capability of the wind tunnel. The solution of this problem is by defining the sensitivity of Reynold's Number on drag and lift coefficient. Reynold's Number increases proportionally with velocity. The study on sensitivity of Reynold's Number will help solving problem due to high requirement of velocity.

4.2 Result and Discussion for Wind Tunnel Testing

The table below shows the drag value for the all models from the wind tunnel testing. The values in the table are the average values for the drag that acting onto the bus models

Table 3 : Drag force value for each model with spoiler different angle.

Velocity (m/s)	Drag Force, F_d (N)			
	Base Model	Model 1 (30°)	Model 2 (45°)	Model 3 (60°)
10	0.85	1.17	2.46	3.1
20	2.14	1.69	5.65	6.7
30	3.13	2.07	5.08	7.82
40	3.82	4.83	6.56	9.02
50	4.56	5.08	6.83	12.17

4.2.1 Improvement of aerodynamic of buses with attachments (spoilers)

The comparison among the results has been done. The drag vs. velocity graph for Model 1, Model 2 and Model 3 has been put in one graph (Figure 4.2). This is to give better understanding on what has happen through the wind tunnel experiment. In the graph below, the drag acting on Base Model is very high compare to Model 1, Model 2 and Model 3. Refer to the graph; the drag has been reduced by making the modification to the bus model. Therefore, the additional spoiler is very useful in reducing the drag acting on the bus. The increase of the degree of spoiler is parallel to the drag coefficient. The ideal angle is about 20-30 degrees.

After comparing with the base model (without spoiler), the author realize that the passive drag is smaller than the model with spoiler. This is because the existence of the spoiler create larger separation region, thus it will have larger number of the passive drag. For bus, the existence of the spoiler is not only to reduce drag but in the same time to maintain the stability of the bus.

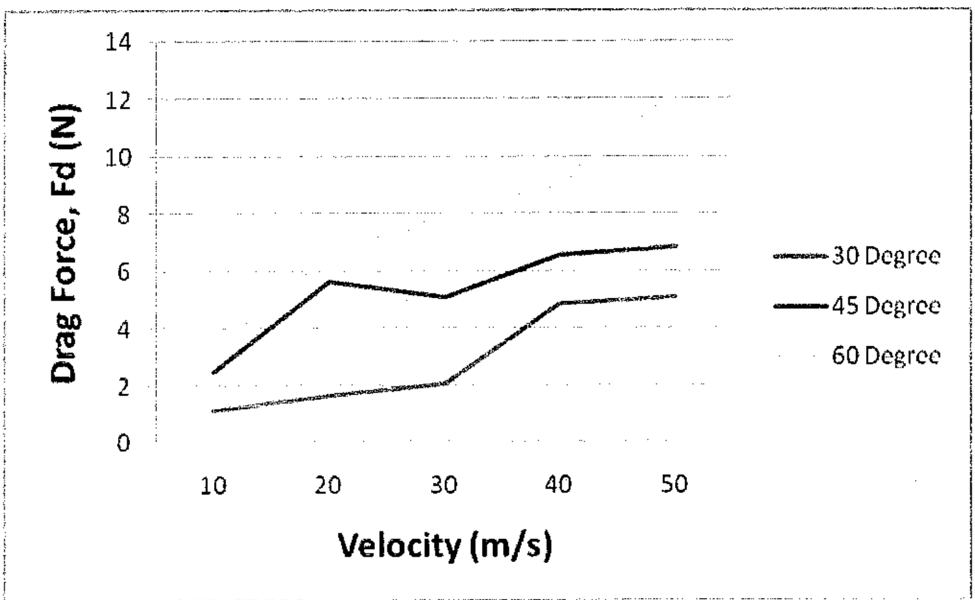


Figure 4.1: Graph for drag force for all models with different angle of spoiler

4.3 Simulation of the models.

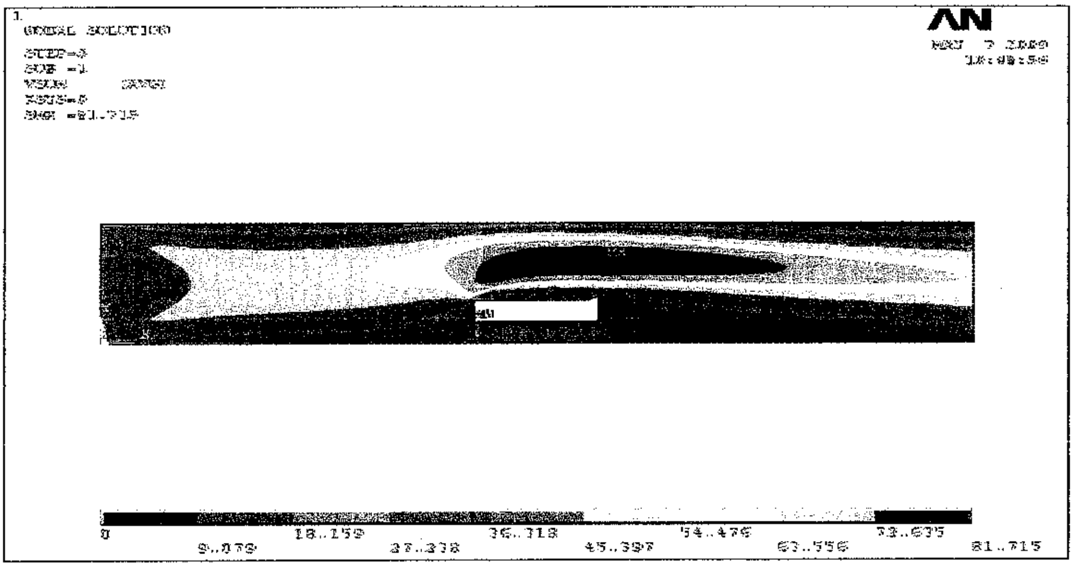


Figure 4.2: Fluid Velocity (simulation of model 1)

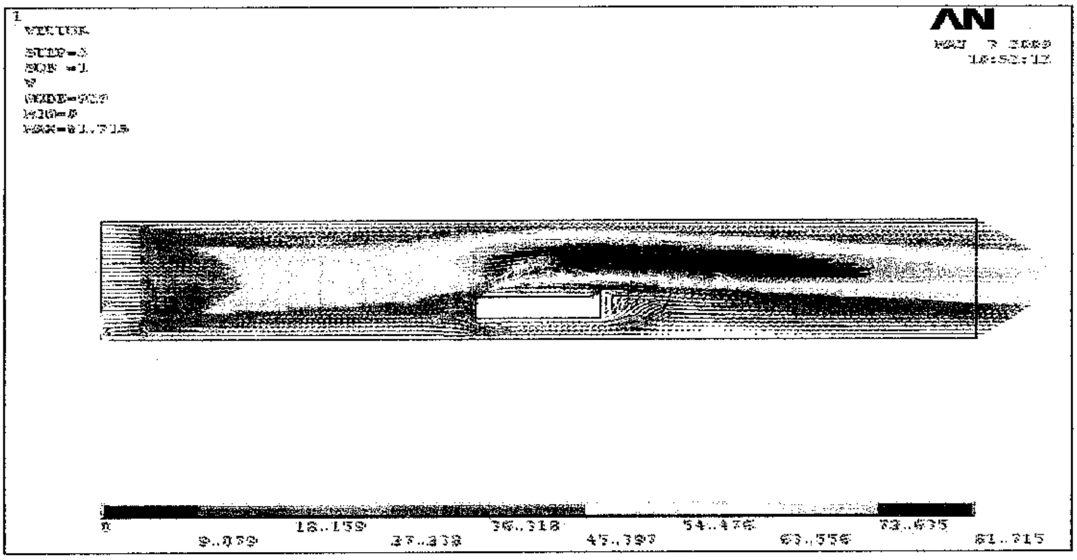


Figure 4.3: Vector plot (simulation of model 1)

The figures 4.2 and 4.3 show the variation of the fluid velocity due to the effect of the 30^0 Degree of spoiler. The maximum speed is about 40 m/s for this simulation. The separation region indicates the aerodynamic effect of the spoiler.

As can see here, the spoiler made the fluid velocity less resistance when the fluid velocity moving through. Compare to the other model, the model 1 is more aerodynamic.

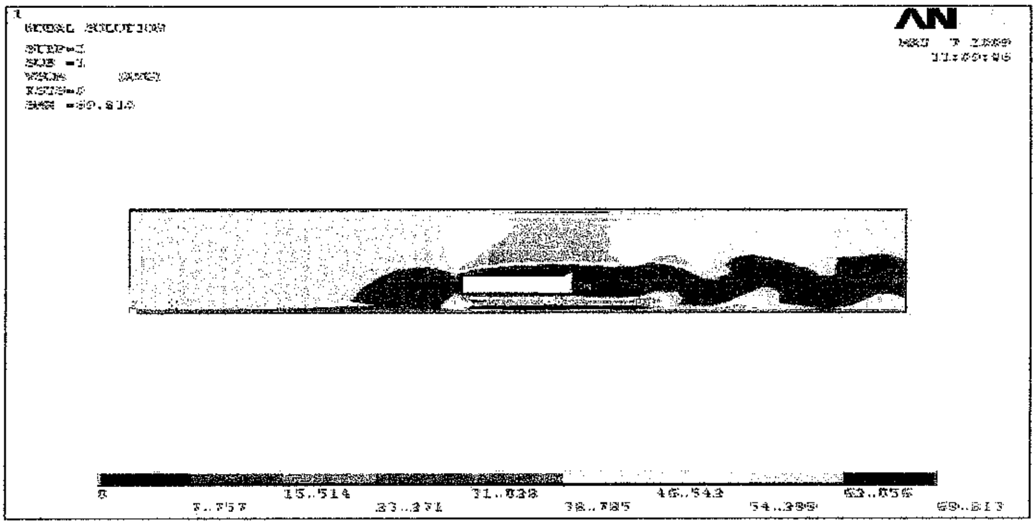


Figure 4.4: Fluid Velocity (simulation of model 2)

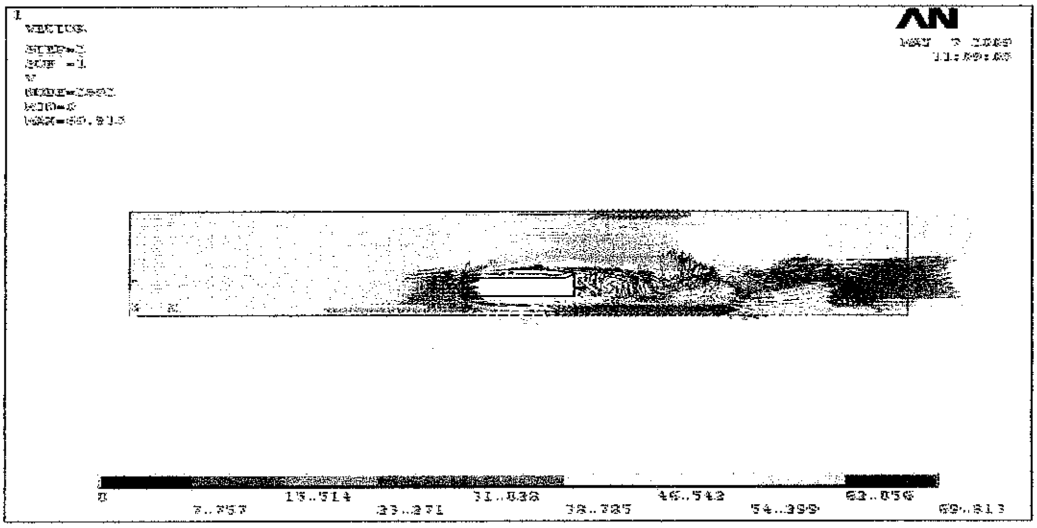


Figure 4.5: Vector Plot (simulation of model 2)

Compare to the Figure 4.2, the Figures 4.4 shows the model is less aerodynamic since the wake region which is indicates the variation of the fluid velocity and the effect of the spoiler. The pressure below the bus is greater than model 1. This is because the variation of the fluid affects the flow towards the angle of the spoiler of the bus.

Figure 4.5 also shows the high resistance of the model and effect the flow variation of the fluid velocity.

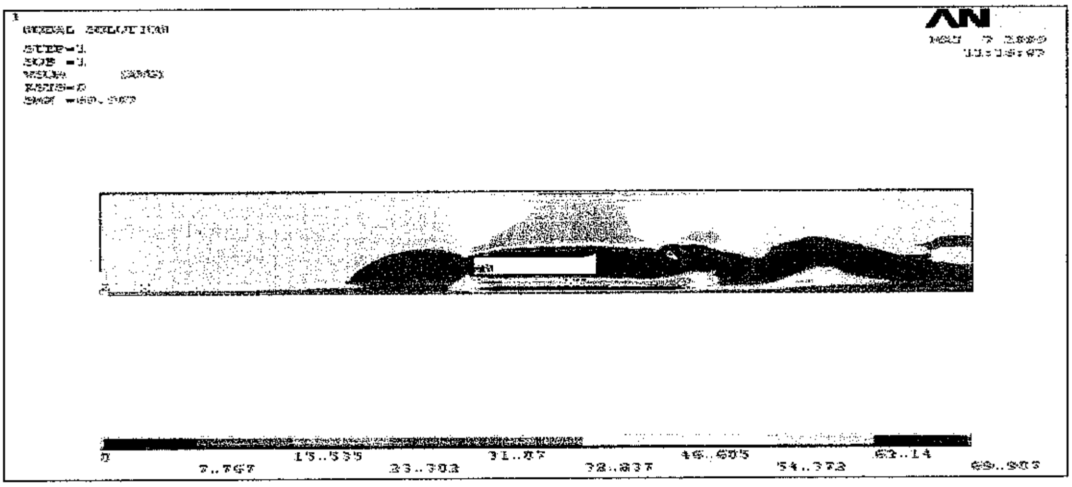


Figure 4.6: Fluid Velocity (simulation of model 3)

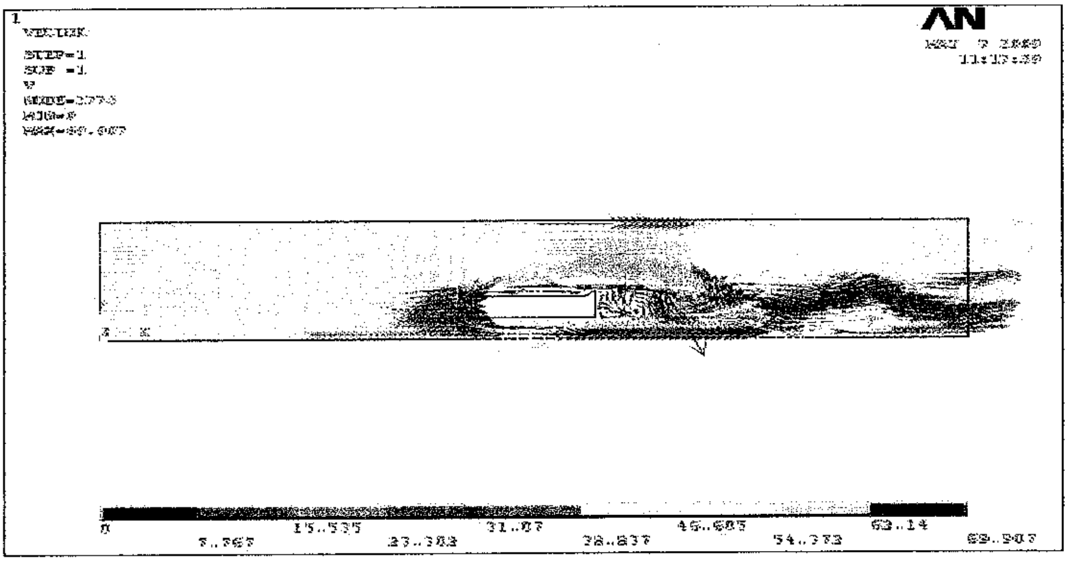


Figure 4.7: Vector plot (simulation of model 3)

Figures 4.6 and 4.7 show the simulation and vector plot of model 3. The model 3 is more resistance compare to the model 1 and model 2. This is because of the 60^0 Degree of spoiler effect.

The higher number of angles will create more resistance of the model. The variation of the fluid velocity shows how the model 3 is not reliable. So, from my observation and analysis the best angle is about $20^0 - 30^0$ degree of the spoiler. The aerodynamic and good spoiler is less resistance when moving through by the fluid velocity.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

The improvement of models and attachments shape is made to verify the effect of attachments to improve the aerodynamic performance of buses as well as to reduce the passive drag. Similarity between model and actual bus must be obtained as it is very crucial. One of important factor to archive the similarity is the angle of the spoiler and it is proven in wind tunnel testing conducted. After comparing with the base model (without spoiler), the author realize that the passive drag is smaller than the model with spoiler. This is because the existence of the spoiler create larger separation region, thus it will have larger number of the passive drag. For bus, the existence of the spoiler is not only to reduce drag but in the same time to maintain the stability of the bus. As the project going on to the end the design and fabrication of optimum shape of attachments to bus that reduce the aerodynamic drag will achieved. Hence it reduces the fuel consumption and operating cost. For the next student who will work on a project like this it is recommended that a wind tunnel experiment must be done to compare the results obtained from the simulation with the experimental results, but the set up in the wind tunnel must be the same as the set up in the CFD simulation which means they must be an account for ground effect.

REFERENCES:

1. R. H. Barnard (1996), "Road vehicle aerodynamics design", Principal Lecturer in Mechanical, Aerospace and Automotive Engineering, University of Hertfordshire.

2. Yunus A. Cengel and John M. Cimbala (2006), "Fluid mechanics: Fundamentals and applications", McGRAW Hill International Edition

3. http://www.inderscience.com

4. White, Frank M ," Fluid Mechanics" , New York : McGRAW-Hill 2008

5. Tu , Jiyuan, "Computational Fluid Dynamics : A practical approach", Amsterdam ; Boston: Butterworth – Heinemann 2008

6. Joseph H , Spurk , "Fluid Mechanics" Berlin : Springe, 2008

7. Prentice – Hall " ANSYS" , New Jersey : 1998

8. Nakasone Y. Yuji , " Engineering Analysis With Ansys Software", Amsterdam ; Boston : Butterworth – Heinemann 2006

9. Jack Zecher , " Ansys Workbench Software Tutorial With Multimedia CD Release II"

10. Fernandon F Griensten , " Implicit Large Eddy Simulation : Computing Turbulent Fluid Dynamics" , Cambridge , New York : Cambridge University Press, 2007

11. Journal : Effect Of Moving Ground Simulation On The Flow Past Bus Models by T. Lajos & L. Preszler

12. Journal : Wind tunnel tests for mean drag and lift coefficients on multiple circular cylinders arranged in-line by Xianzhi Liua,, Marc Levitana, Dimitris Nikitopoulosb

13. Journal: Bus Drag Reduction by The Trapped Vortex Concept For A Single Bus and Two Buses in Tandem C.A.J. Fletcher and G.D.H. Stewart Dept. Of Mechanical Engineering, University Of Sydney, Sydney, N.S.W. 2006 (Australia)

14. WT04 Sub-sonic Wind Tunnel Operating & Experimental Manual, Petronas Univ.of Tech. (UTP).

15. Journal : Drag reduction of a square prism by flow control using a small rod Tamotsu Igarashi Department of Mechanical Engineering, The National Defence Academy, 1-10-20 Hashirimizu, Yokosuka 239, Japan

Appendix A

Data from the Wind Tunnel Testing

Table A.1: Data from the first Wind Tunnel Testing

Velocity	Drag Force, Fd (N)			
	Base Model	Model 1	Model 2	Model 3
10	0.85	0.23	0.24	0.31
15	2.14	0.91	1.08	0.56
20	3.13	1.24	2.63	1.32
25	3.82	1.37	3.04	1.45
30	4.56	2.2	4.43	2.77

Figure A.1: Graph from the data of the first Wind Tunnel Testing

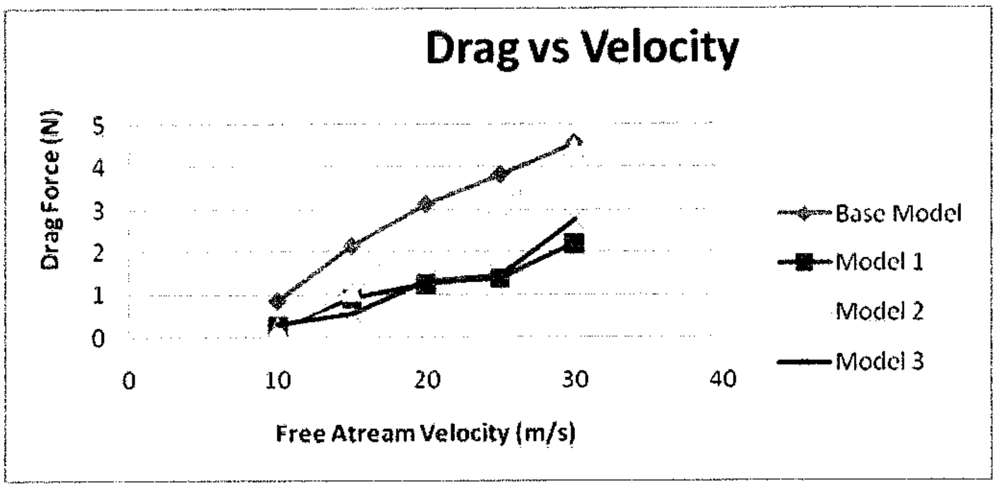


Table A.2: Data from the second Wind Tunnel Testing

Velocity	Drag Force, Fd (N)			
	Base Model	Model 1	Model 2	Model 3
10	1.634	0.51	0.552	0.67
15	2.8	1.018	0.964	1.24
20	3.774	1.346	1.076	1.502
25	4.882	1.962	1.328	1.328
30	5.926	2.658	1.806	1.818

Figure A.2: Graph from the data of the second Wind Tunnel Testing