# Simulation of Flow around a Bus with Active Drag Reduction System

By

**Nesamari Mpho**

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

CERTIFICATION OF APPROVAL

**Simulation of Flow Around a Bus with Active Drag Reduction System**

by

Mpho Nesamari

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,

(Mr. Rahmat Shazi Iskandar)

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 persons.

MPHO NESAMARI

**ABSTRACT**

A simulation of a bus model dimensions 200 x 50 x 50 mm was conducted using CFD software FLUENT 6.2, in order to investigate the flow around a bus with active drag reduction system. Simulations were carried out for the base model which is the refrence point and two other spoiler models A & B. All models were designed using CATIA & GAMBIT, simulations were carried out for different spoiler angles. The spoiler model B proved to be most efective in reducing drag when the angle of the spoiler is $15^0$. The spoiler model managed to minimize the drag force by 27 %. The active drag reduction system utilized is a spoiler which change angle at different speed (from 0 to 20 m/s angle the spoiler is $5^0$, and from 20 to 45 m/s angle of the spoiler is $15^0$).

**ACKNOWLEDGEMENTS**

Firstly I would like to thank my Lord and Saviour Jesus Christ, my shield, my strength, and my present help in time of need, for seeing me through this project. All I do and will continue doing unto him.

Secondly I would like to thank my sponser PETRONAS for sponsering me and beleaving in me for the 4 years I spent working on my mechanical engineering degree in Universiti Teknologi PETRONAS.

Thirdly I would like to thank Universiti Teknologi PETRONAS for grooming me and providing me with the necessary skilles required to complete my degree and Final year project.

I would like to thank Mr. Rahmat Shazi Iskandar, my Final year project supervisor for his support, advice, mentoring, most importanttly his guidence and iparting knowledge & manegement skills. I would also like to thank Assoc. Prof. Dr. Hussain H. Jaffer Al-Kayiem and Assoc. Prof. Dr. Chalilullah Rangkuti my Final year project exminers for their time and constructive inputs.

I also wish to thank my seniour Lerato for helping me with the simulation software. Not forgetting the previous students who worked on the project before me, theire work provided me with the bases from where I could start on.

Not forgetting my family and friends who never failed to support and motivate me. Thank you all.

# TABLE OF CONTENT

# LIST OF FIGURES

# LIST OF TABLES

# CHAPTER 1
# INTRODUCTION

## 1.1 BACKGROUND

Fluid flow around bodies frequently occurs in practice and this is responsible for numerous physical phenomina such as drag forces acting on the body , lift acting on aircraft wings, vibration and noise generated on the body moving in a fluid.

It is common that a body experience resistance when it is forced to move in a fluid, especially water. Drag is defined as a force a flowing fluid exerts on a body in the flow of direction[1]. There are two types of drag that affect automobiles:

- **Pressure drag:** which is caused by the pressure aplied by the air particles which are compresed upon impact with the moving body before they are forced to move around the body, this pressure differance that is created tends to restrict movement of the body since the pressure in the front is higher thun that at the back of the moving body.
- **Friction drag:** which is caused by the wall shear stresses $\tau_w$, when the fluid move around the body it expireances resistance due to the non-slip conditions on the body surfaces, that resistance causes frictional forces between the body and the fluid flowing around it which we call frictional drag.

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 attachments a sort of tail called a fastback or lift back. It will have a flat and smooth floor to support the Venturi effect and produce desirable downwards aerodynamic forces. A bus aerodynamics is one of the poorer in the automobile family; they usual designed in a box shape because they are one of the public transports and are ment to carry alot of people, this result in a high coefitiant of drag $C_D$, which then translate to higher fuel consumption.

## 1.2 PROBLEM STATEMENT

Crude oil prices have gone down in the past year, however the world proven crude oil reserves are depleting, from 1985 world oil reserve additions have been less than consumption for the year [6]. World crude oil demand grew an average of 1.76% per year from 1994 to 2006, with a high of 3.4% in 2003-2004. World demand for oil is projected to increase 37% over 2006 levels by 2030 (118 million barrels per day from 86 million barrels) [6]. Since there has not been an effective alternative source of energy to raplce crude oil there is a need to improve the fuel consumption of the bus. Also about 45% of the fuel consumption is caused by rolling resistance, 25% by aerodynamic drag and 30% by acceleration and climbing resistance [2].

A need for a more efficient less fuel consuming bus is needed and that can be achived by designing the bus in a more aerodynamical shape and also by introducing proper drag reduction systems on buses. The end results wont only just be drag reduction but also:

i. Reducing drag,
ii. Minimizing noise emission,
iii. Preventing undesired lift forces at high speeds.

## 1.3 OBJECTIVE

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

1. To study the types of drag reduction system applied.
2. To simulate and study the flow around a bus with active drag reduction system comparing base model and different active drag reduction systems applied.
3. To reduce drag using an active drag reduction system.

## 1.4 SCOPE OF STTUDY

This project requires extensive knowllagdge about fluid flow, and understanding the nature of the flow. Knowledge on drag and the effects of drag on a body moving in a fluid is required, also research on the different types of active drag reduction system utulized today needs to be aquired. The bus models will be designed in CATIA and then transfered to GAMBIT for defining boundries and meshing before exporting it to the simulation program. Simulation around the bus will be done by computational fluid program FLUENT, where the bus model will be subjected to wind pressure in the wind tunnel test section, from the simulation the effects of drag on the different models used will be recorded.

# CHAPTER 2
# LITERATURE REVIEW

**Flow separation**

At sufficiently high velocities, the fluid stream detaches itself from the from the surface of the body, that effect is called flow separation. The location of the separation point depends on several factors such as reynolds number, the surface roughness and level of fluctuations in free stream and it is usually difficult to predict exactly where separation will occur unless there are sharp corners or abrupt changes in the surface of the solid surface [1]. When a fluid seperates from a body, it forms a separation region between the body and the fluid stream. The low pressure region behind the body where recirculation and backflows occur is called the separated region, 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. The separated region comes to an end when the two flow stream reattach.

A paper on bus drag reduction by trapped vortex concept for a single bus and two buses in tandem talk about vehicle design paying a great deal ot attension in fuel econnomy, which is achived by strem lining a vehicle so that the flow can move with less unsteadyness (aerodynamic drag reduction), other benefits that come out of this is better handling.

Fletcher and Stewart [2], where able to achive 17% reduction in drag of a MC-7 intercity bus and a corresponding 11.7% reduction in fuel consumed at a steady speed of 88 Km/h by adding aerodynamics in exsisting shapes of buses.

Götz says for a typical bus 45% of fuel consumption is caused by rolling resistance, 25% by aerodynamic drag, and 30% by acceleration and climbing resistance. Fletcher and Stewart [2], where able to reduce the drag coefficient $C_D$ by 0.29 by using optimal radii on the forebody/roof junction, 8 ° tapers behind the forebody and in front of the tail (see figure 2.1).

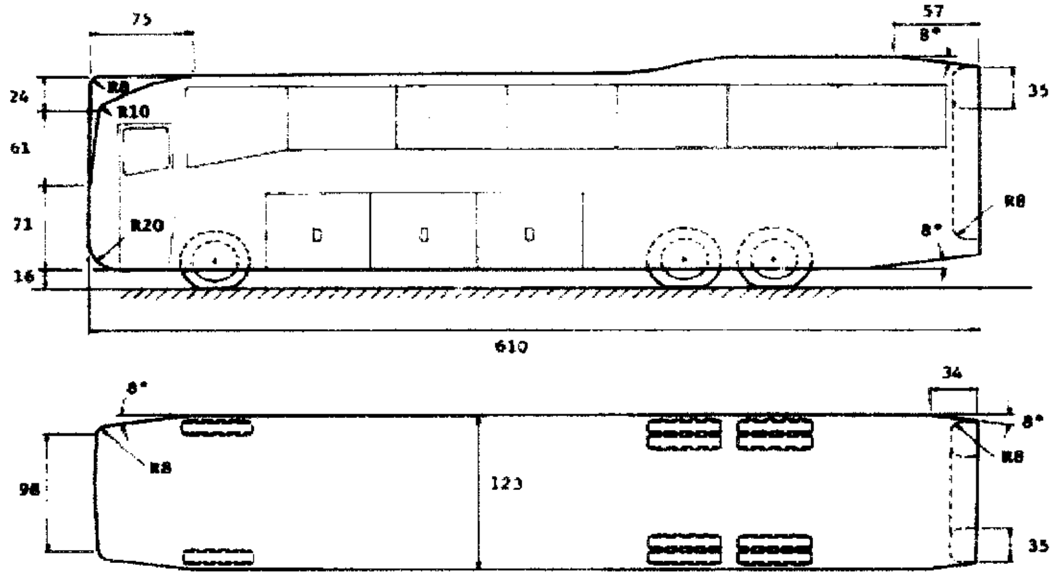



**Figure 2.1: Bus with drag reduction modification [2].**

Bus drag reduction has focused on mainly introducing a sufficient larger radius at the forebody/roof and forebody/side junctions of the buses to prevent flow separation and on reducing small scale obstruction into the flow (e.g. mirrors and window frames).

Less attention has been paid to the rear of the bus to produce a higher pressure recovery and reducing the drag coefficient. A cavity in the rear ward of the buses was introduced to encourage a standing vortex to be produced so that the flow past the bus will be assisted in turning towards the bus wake center line (see figure 2.2). By smoothing the edges and the inclusion of the vortex trapping system they managed to improve the flow separation and where able to streamline the flow at the rear of the bus that way they recovered some pressure at the rear of the bus.

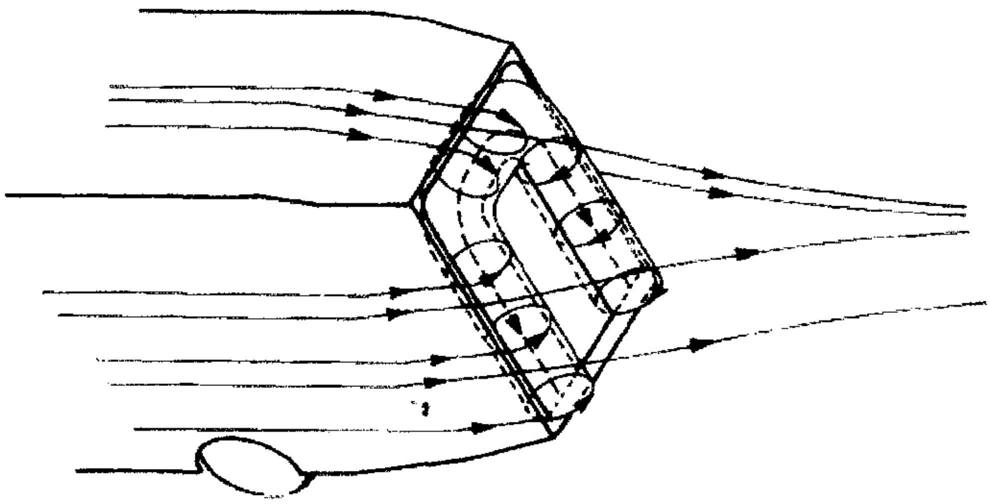


**Figure 2.2: Conceptual flow field associated with a trapped horseshoe vortex [2].**

From all this it can be seen that the main issue when dealing with reducing drag is to make sure that the object is streamlined in hopes to reduce flow separation to prevent unsteady state in the fluid flow.

A paper on drag reduction of an Ahmed car model (Ahmed model is a reference car model with a variable slant angle $\alpha$ controlling the near wake flow structure and the aerodynamic drag) by means of active separation control at the rear vehicle slant talks about the experimental investigations which deal with the reduction of the total aerodynamic drag of a generic car model (Ahmed-Body) by means of periodic forcing. The experiments carried out in this study focus on a unique approach to separation control using fundamental frequencies for local forcing of the shear layer separated from the rear end of the car model. The excitation of large scale vortex structures by periodic forcing intensifies the primary momentum transfer between the separation region and the outer flow, resulting in a substantial reduction of the separation length. A total drag reduction of 27% was achieved using the flow control method described in this study [5].

**Pressure drag**

Drag force is the net force exerted by a fluid on a body in the direction of flow due to the combined effects of wall shear and pressure force. The part of drag due directly to pressure is called pressure drag, also called form drag because if it's strong dependence on the form or shape of the body. When the friction and pressure drag coefficients or forces available, the total drag force can be determined by simply adding them, the pressure drag is proportional to the frontal area and to the difference between the pressures acting on the front and back of the immersed body [1]. The vortex trapping system is one of those systems designed to remove or minimize the effect of pressure drag by recovering pressure at the rear end of the bus.

Another system is found in a paper on the effect of vehicle spacing on the aerodynamics of a representative car shape by Simon Watkins [3] talks about Inter-vehicle spacing on highways which is considered and an analysis of spacing is presented, deduced from data from an instrumented highway. There are many variables in a study of this kind; these include vehicle geometric configuration (e.g. truck or car, including fastback, notchback, etc.), the lateral and longitudinal positioning of vehicles relative to each other and the nature and relative direction of the atmospheric wind. In order to restrict the number of variables, the investigation is limited to a wind-tunnel simulation of representative car geometry in calm conditions (i.e. no yaw angle) and vehicles that are directly aligned (i.e. co-linear) [3].

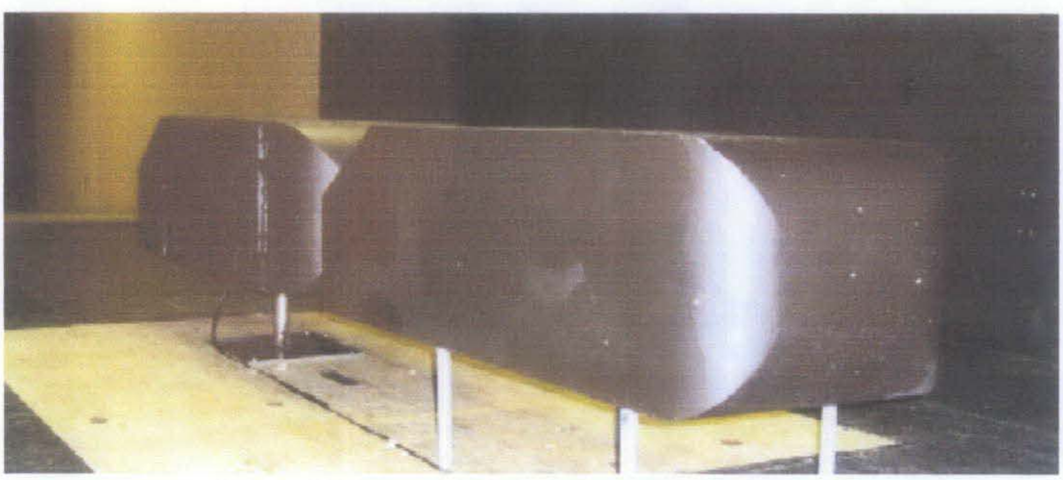


**Figure 2.3: Sample drafting arrangement [3].**

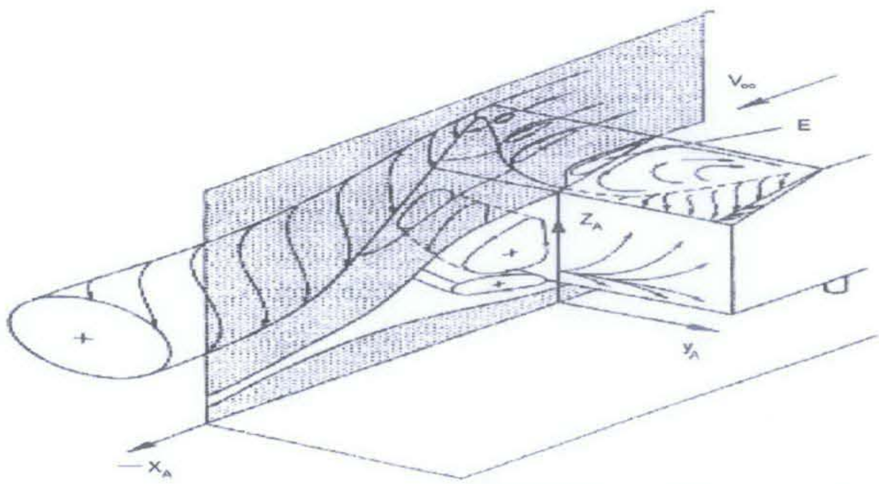
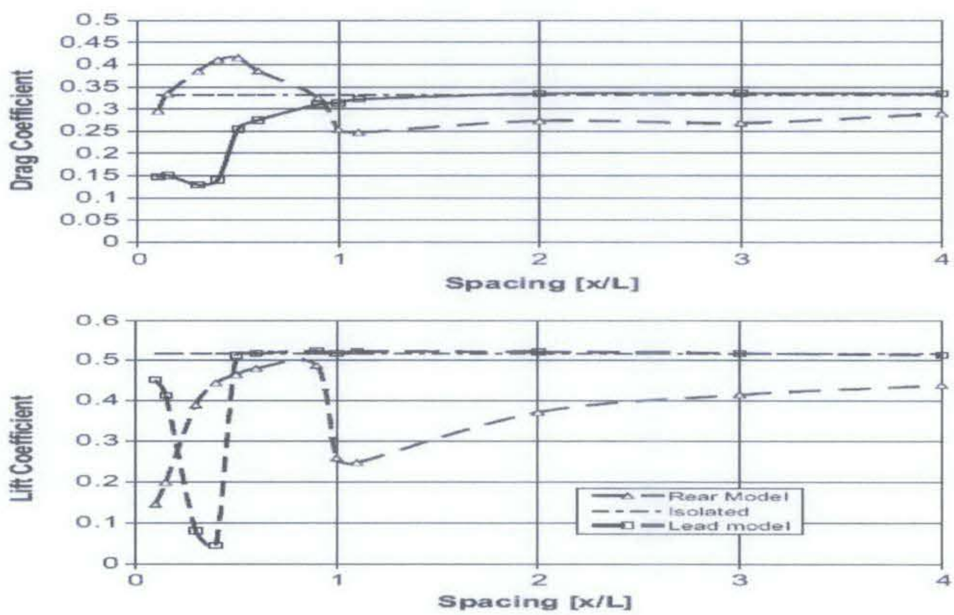



**Figure 2.4: Proposed vortex system from Ahmed et al. (1984) [3].**

Vehicle drag reductions arising from close spacing are discussed and drag and lift data from wind-tunnel tests on two co-linear Ahmed bodies (representative vehicle shapes able to replicate typical car airflow, configured with 30° slant back angles) are given. Inter-body, non-dimensional spacing was varied from 0.1 to 4.0, based on vehicle length. Surprisingly, significant drag increases were found for the rear Ahmed body for spacing of 0.1–1.0, when compared to the drag of the body in isolation. For greater spacing, the drag of the rear body fell below the value of the isolated case, up to the maximum spacing considered. The lift coefficient of the rear body was also found to be very sensitive to spacing [3].




**Figures 2.5: Lift and drag coefficients [3].**

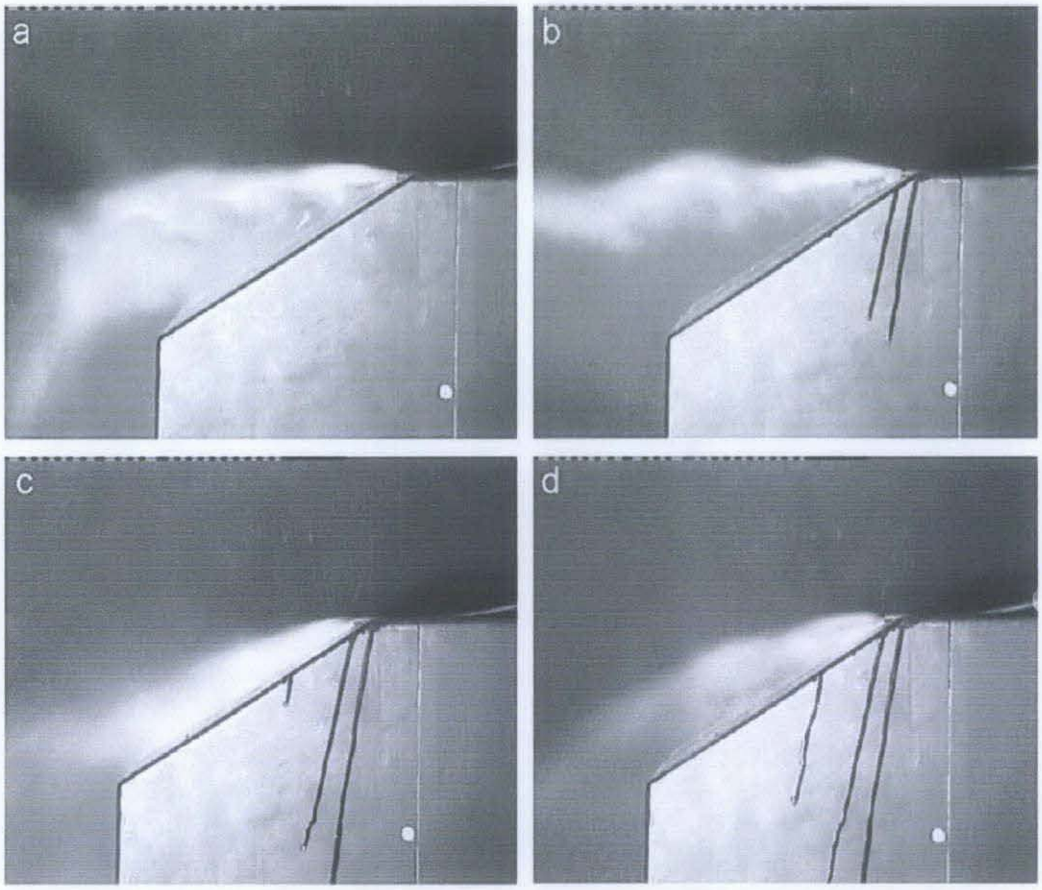



**Figure 2.6: Smoke flow patterns from the trailing edge of leading model. (a) Spacing ¼ 0.15 of body length, (b) spacing ¼ 0.30 of body length, (c) spacing ¼ 0.50 of body length, and (d) spacing ¼ 1.0 of body length [3].**

It was concluded that the effect of the strong vortex system arising from the slant back was the cause of the drag and lift changes of the rear vehicle. Since traffic spacing is likely to reduce with the increasing use of intelligent transport systems (ITS), it is argued that more attention should be paid to understanding these effects.

A paper on drag reduction of motor vehicles by active flow control using the Coanda effect by Geropp D and Odenthal H-J [4], talks about a test facility that has been constructed to realistically simulate the flow around a two dimensional car shaped body in a wind tunnel. A moving belt simulator has been employed to generate the relative motion between model and ground. In a first step, the aerodynamic coefficients $C_L$ and $C_D$ of the model are determined using static pressure and force measurements. LDA-measurements behind the model show the large vortex and turbulence structures of the near and far wake. In a second step, the ambient flow around the model is modified by way of an active flow control which uses the Coanda effect, whereby the base-pressure increases by nearly 50% and the total drag can be reduced by 10% [4]. The recirculation region is completely eliminated. The current work reveals the fundamental physical phenomena of the new method by observing the pressure forces on the model surface as well as the time averaged velocities and turbulence distributions for the near and far wake. A theory resting on this empirical information is developed and provides information about the effectiveness of the blowing method.

**Inverted Wing (Spoiler)**

A spoiler is an aerodynamic device attached to an automobile whose intended design function is to 'spoil' unfavorable air movement across a body of a vehicle of some kind in motion. This is accomplished by increasing the amount of turbulence flowing over the shape, "spoiling" the laminar flow and providing a cushion for the laminar boundary layer [7]. This can result in improved vehicle stability by decreasing drag that may cause unpredictable handling in a vehicle at high speed. Because of air flow separation, the flow of air becomes turbulent and a low-pressure zone is created, increasing drag and instability. Adding a rear spoiler makes the air longer, gentler slope from the roof to the spoiler (see figure 2.7), which helps to delay flow separation. These decreases drag and avoid lift or generating negative lift to improve traction and control.

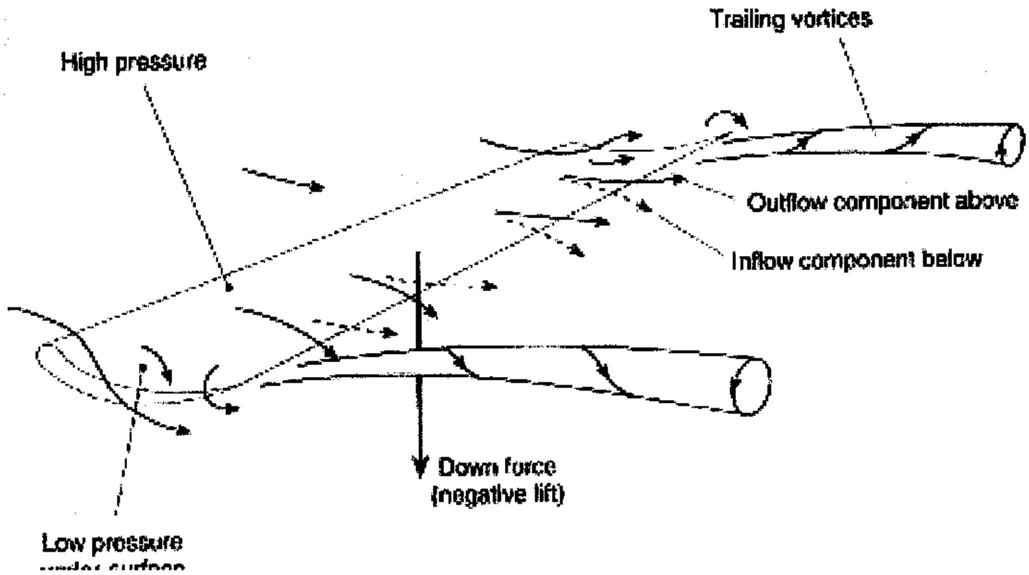



**Figure 2.7: Inverted Wing Spoiler [6].**

In a road test report done by Justin Couture [9], Canadian Auto Press it says Bugatti's engineers came up with a particularly clever solution. They've employed an extremely sophisticated hydraulics system which controls the suspension system in conjunction with the aerodynamic aids, diffuser and power steering system to produce a vehicle whose profile morphs depending on speed. Under normal conditions (up to 220 km/h) the ride height and spoiler are set to normal heights. At speeds above 220 km/h, or at the driver's discretion up to 375 km/h, Handling Mode can be called up, which drops the height by a further 95 mm up front and 80 mm in back, and raises the height and angle of the spoiler to produce 350 kg of down force, increasing stability and grip. But the chase for the final few digits in its top speed required a reduction in drag. The ride height drops a further 65 mm up front and 70 mm out back, but the rear spoiler tucks back down for the most slippery and streamlined shape. Down force is reduced to just 50 kg [9].

# CHAPTER 3
# METHODOLOGY

The steps involved in this project are as follows:

1. First stage of the report is research on different journals on the study that is conducting. Study on the effects of drag and how it is created when a body is moving in a fluid.
2. Familiarize with design software (CATIA & GAMBIT) and CFD software (FLUENT).
3. Design an active drag reduction system using (CATIA & GAMBIT).
4. Simulate the new design using CFD software.
5. Design review and if the results are not satisfying restart from step 3.

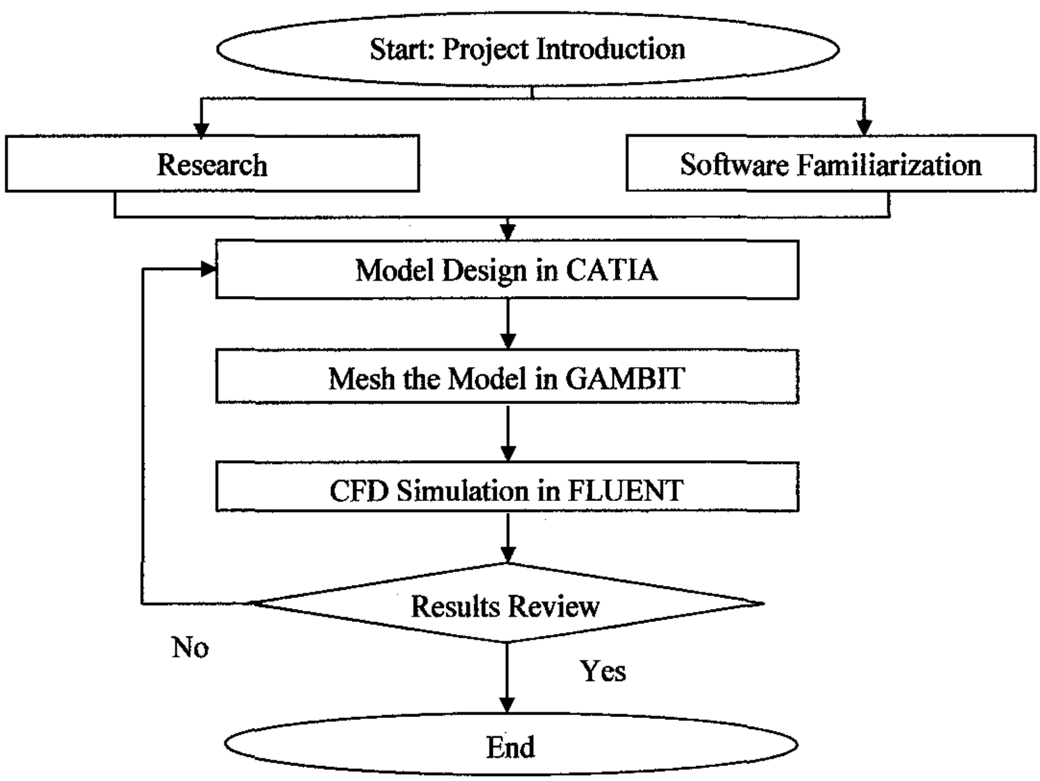



**Figure 3.1: Project flow chart.**

## APPENDIX – B

**Table B.1: Spoiler model A results.**

| | | Down Force (N) | | | | |
|---|---|---|---|---|---|---|
| Speed (m/s) | Re | Base | Spoiler 0 | spoiler 5 | spoiler 10 | spoiler 15 |
| 10 | 1.26E+05 | 0.004 | 0.006 | 0.006 | 0.007 | 0.009 |
| 15 | 1.89E+05 | 0.020 | 0.030 | 0.030 | 0.036 | 0.043 |
| 20 | 2.52E+05 | 0.059 | 0.091 | 0.090 | 0.112 | 0.133 |
| 25 | 3.15E+05 | 0.140 | 0.218 | 0.215 | 0.269 | 0.321 |
| 30 | 3.77E+05 | 0.285 | 0.447 | 0.441 | 0.554 | 0.662 |
| 35 | 4.40E+05 | 0.521 | 0.822 | 0.808 | 1.020 | 1.220 |
| 40 | 5.03E+05 | 0.881 | 1.396 | 1.368 | 1.734 | 2.075 |
| 45 | 5.66E+05 | 1.402 | 2.227 | 2.181 | 2.771 | 3.317 |

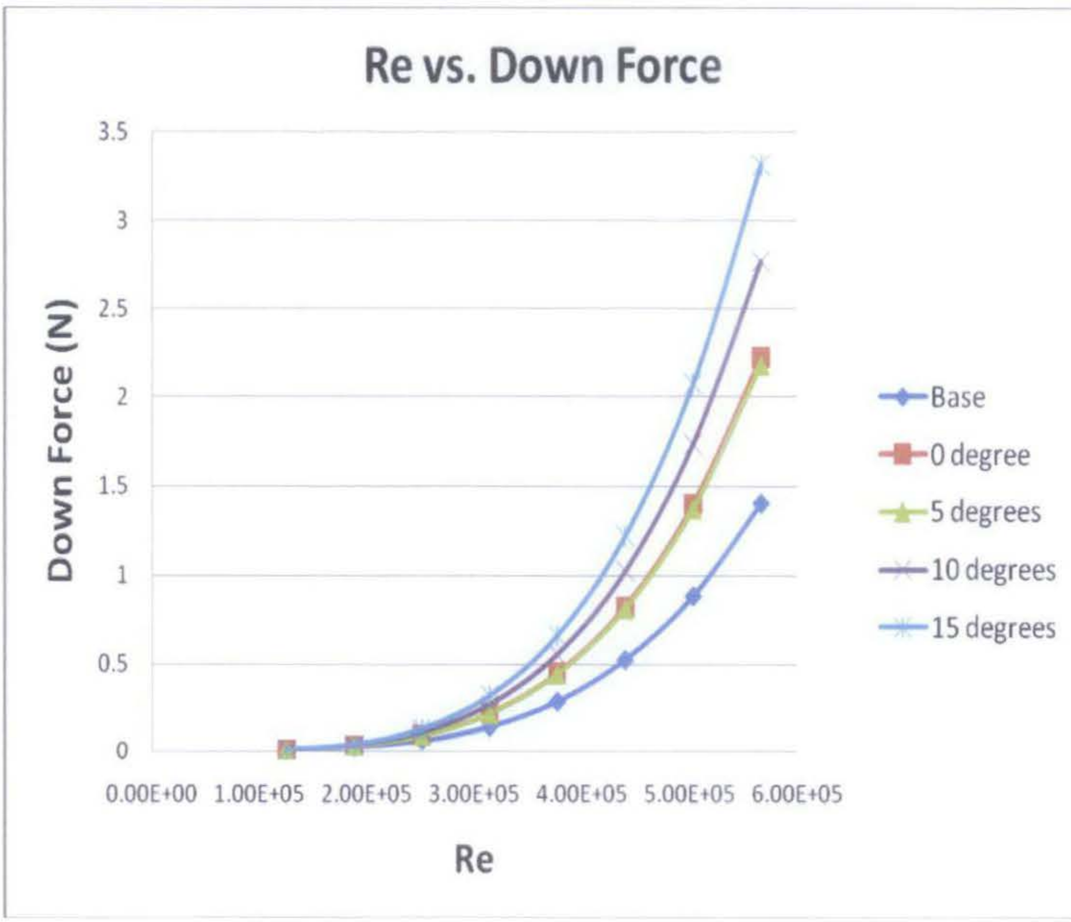



**Figure B.1: Spoiler model A Re vs. Down force.**

Tale B.2: Spoiler model B results.

| | | Down Force (N) | | | | |
|---|---|---|---|---|---|---|
| Speed (m/s) | Re | Base | Spoiler 0 | spoiler 5 | spoiler 10 | spoiler 15 |
| 10 | 1.26E+05 | 0.004 | 0.005 | 0.007 | 0.009 | 0.011 |
| 15 | 1.89E+05 | 0.020 | 0.022 | 0.034 | 0.044 | 0.025 |
| 20 | 2.52E+05 | 0.059 | 0.068 | 0.108 | 0.141 | 0.079 |
| 25 | 3.15E+05 | 0.140 | 0.162 | 0.265 | 0.346 | 0.070 |
| 30 | 3.77E+05 | 0.285 | 0.329 | 0.552 | 0.720 | 0.145 |
| 35 | 4.40E+05 | 0.521 | 0.600 | 1.027 | 1.336 | 0.269 |
| 40 | 5.03E+05 | 0.881 | 1.011 | 1.757 | 2.284 | 0.459 |
| 45 | 5.66E+05 | 1.402 | 1.605 | 2.820 | 3.665 | 0.736 |

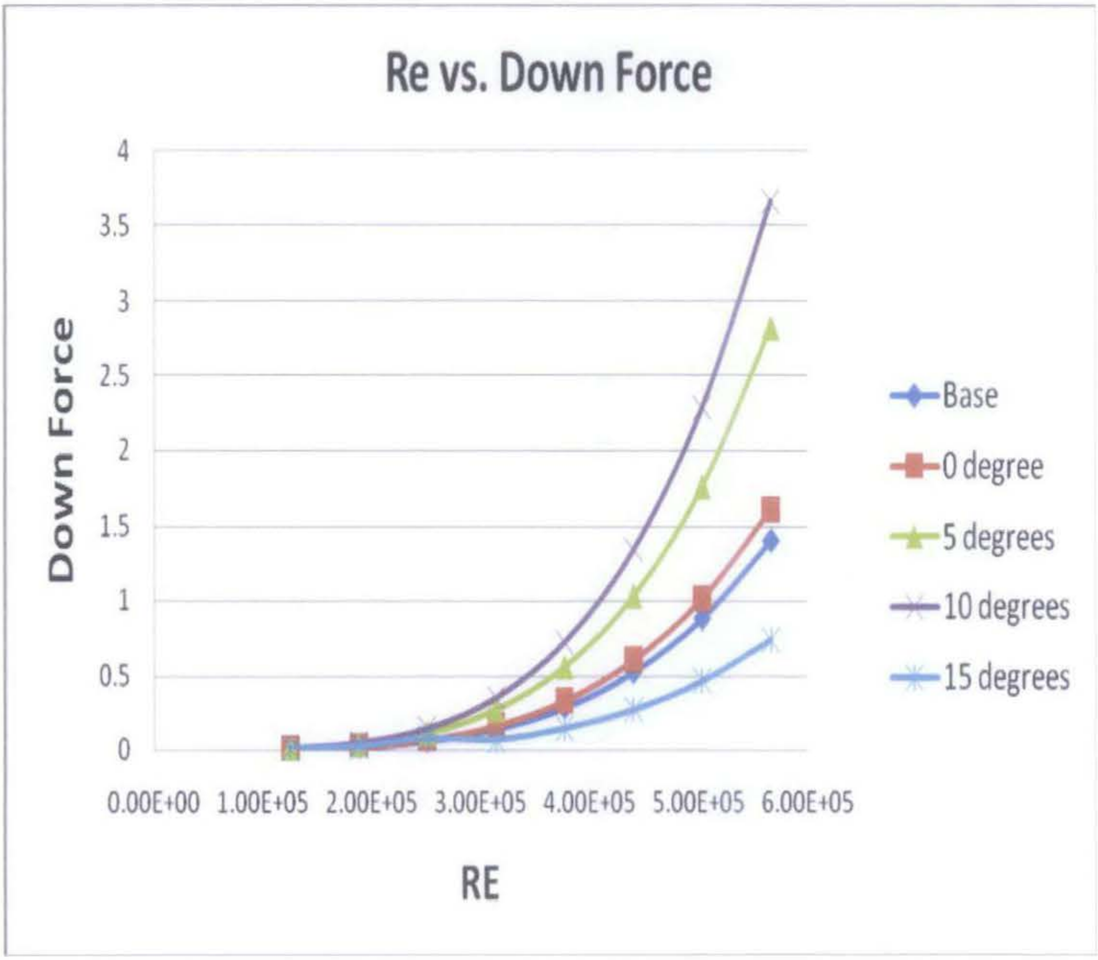



Figure B.2: Spoiler model B Re vs. Down force chart.

## APPENDIX – C

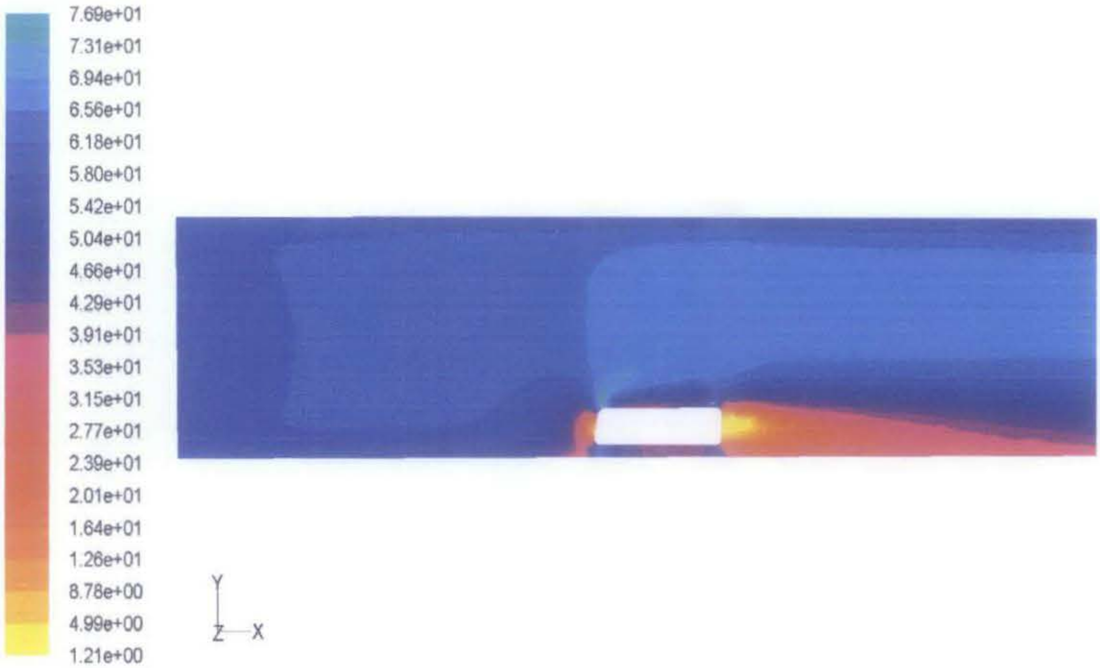



**Figure C.1: Spoiler model B ($5^0$) dynamic pressure contour at 10 m/s.**

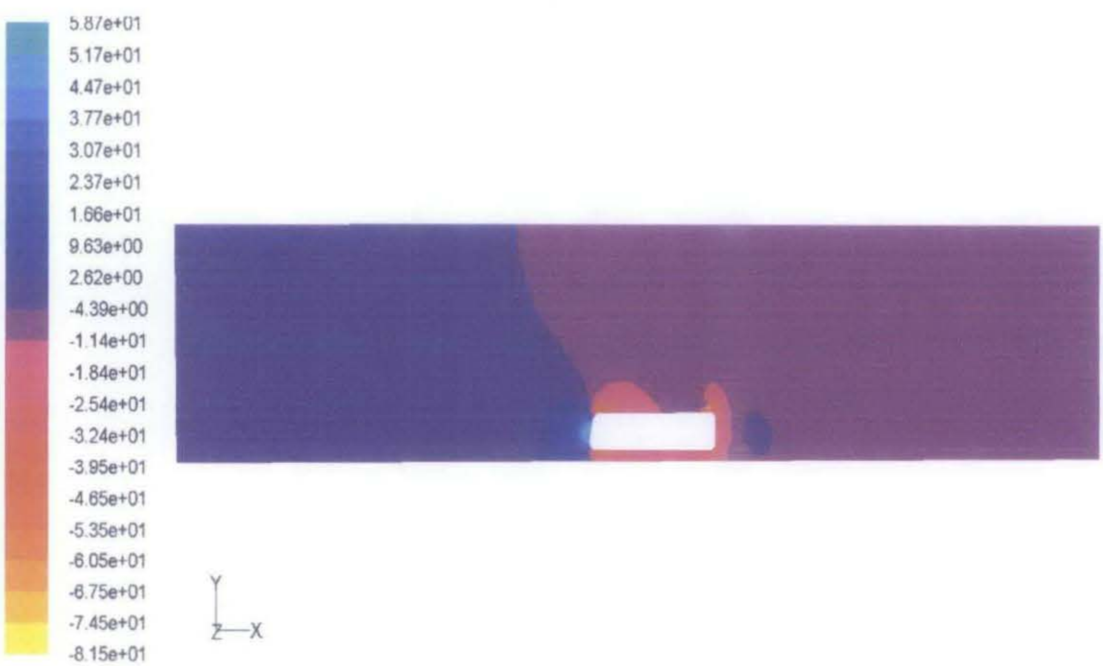



**Figure C.2: Spoiler model B ($5^0$) static pressure contour at 10 m/s.**

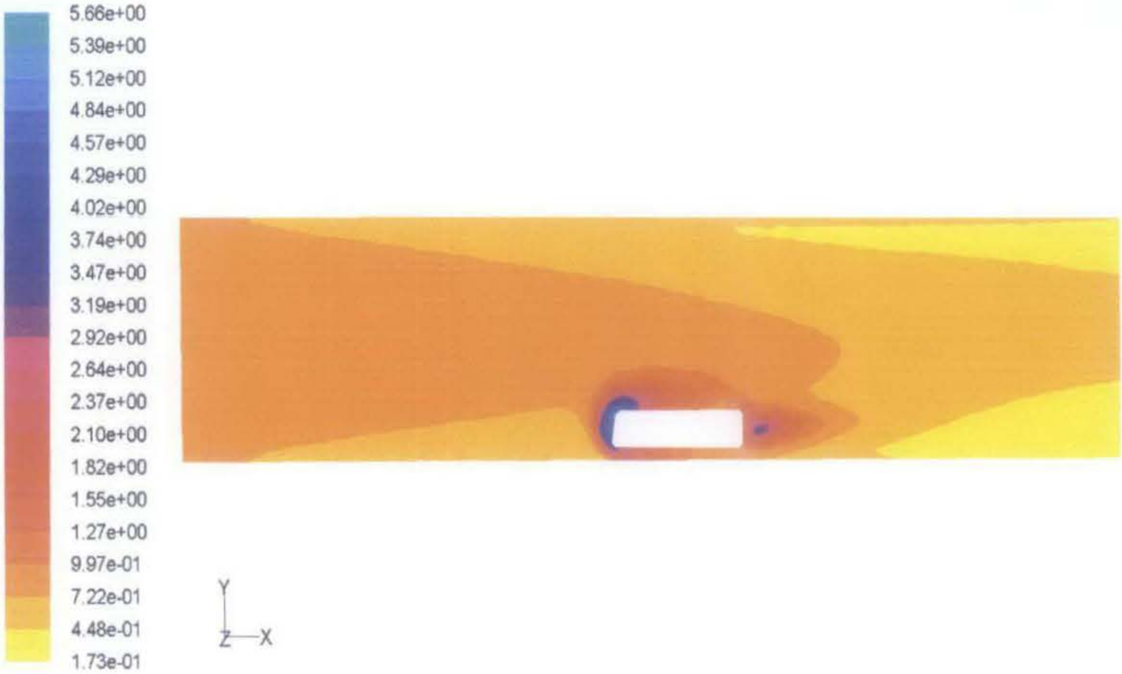



**Figure C.3: Spoiler model B ($5^0$) turbulence contour at 10 m/s.**

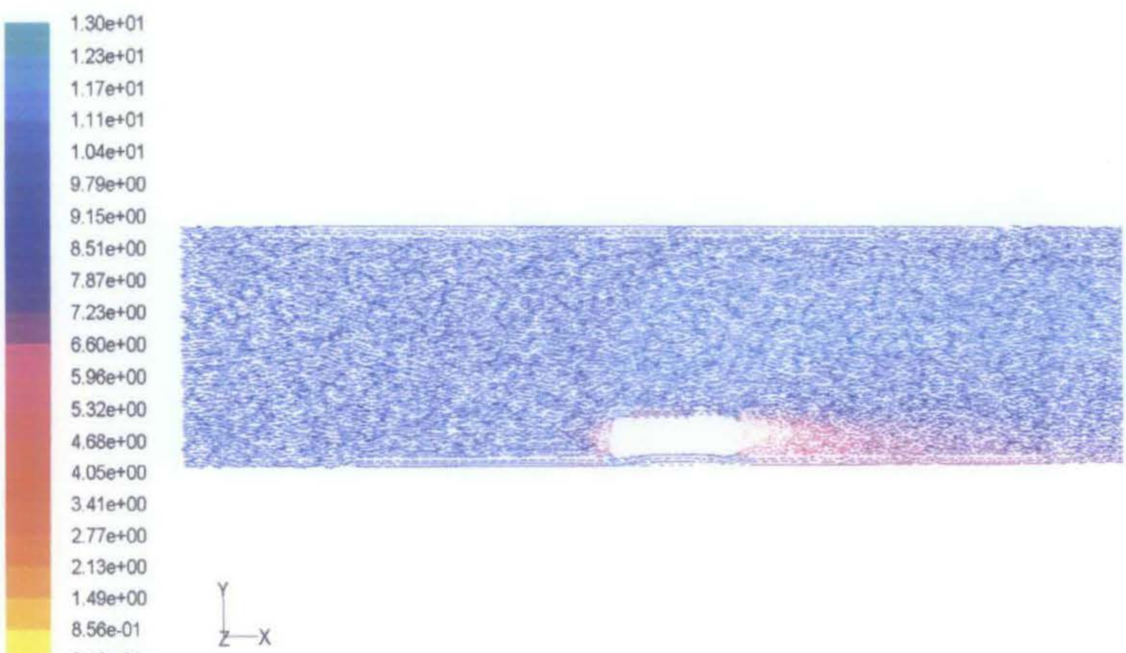



**Figure C.4: Spoiler model B ($5^0$) Velocity vector at 10 m/s.**

**Table 4.2: Spoiler model B results.**

| | | Drag Force (N) | | | | |
|---|---|---|---|---|---|---|
| Speed (m/s) | Re | Base | Spoiler $0^0$ | spoiler $5^0$ | spoiler $10^0$ | spoiler $15^0$ |
| 10 | 1.26E+05 | 0.031 | 0.031 | 0.031 | 0.032 | 0.033 |
| 15 | 1.89E+05 | 0.147 | 0.149 | 0.155 | 0.160 | 0.075 |
| 20 | 2.52E+05 | 0.454 | 0.454 | 0.488 | 0.504 | 0.238 |
| 25 | 3.15E+05 | 1.089 | 1.083 | 1.188 | 1.228 | 0.209 |
| 30 | 3.77E+05 | 2.233 | 2.211 | 2.459 | 2.543 | 0.432 |
| 35 | 4.40E+05 | 4.104 | 4.048 | 4.550 | 4.705 | 0.799 |
| 40 | 5.03E+05 | 6.957 | 6.843 | 7.754 | 8.018 | 1.362 |
| 45 | 5.66E+05 | 11.090 | 10.882 | 12.410 | 12.832 | 2.181 |

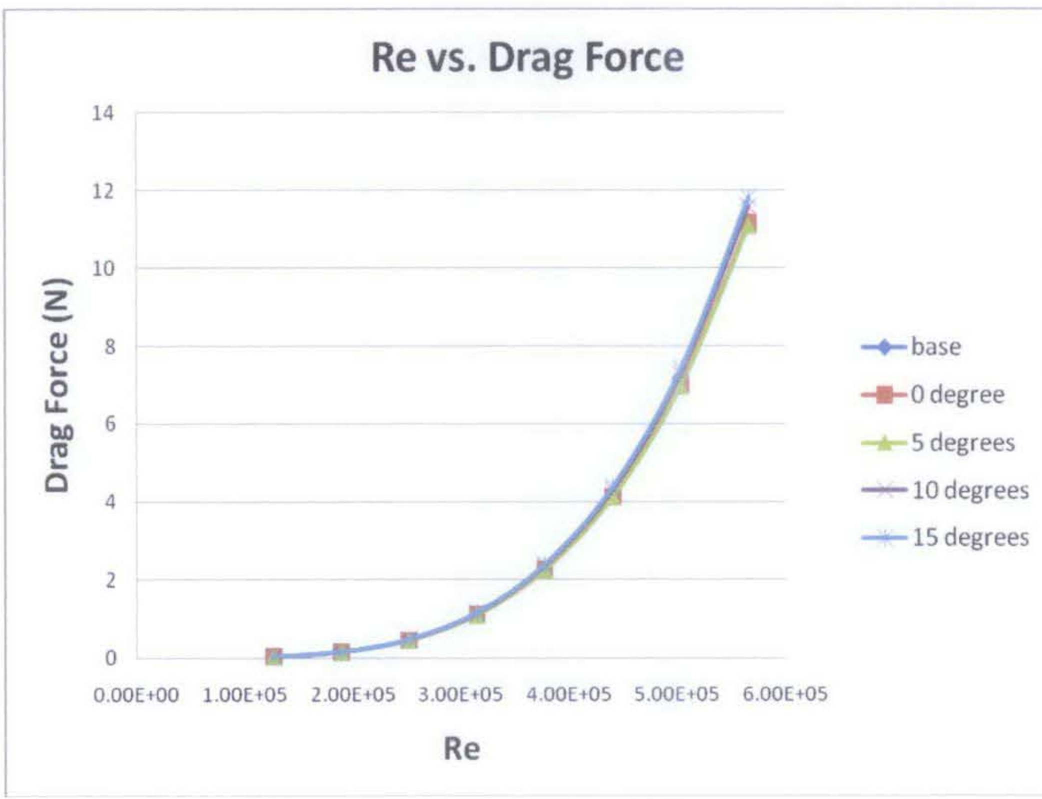



**Figure 4.1: Spoiler model A Re vs. Drag force chart.**

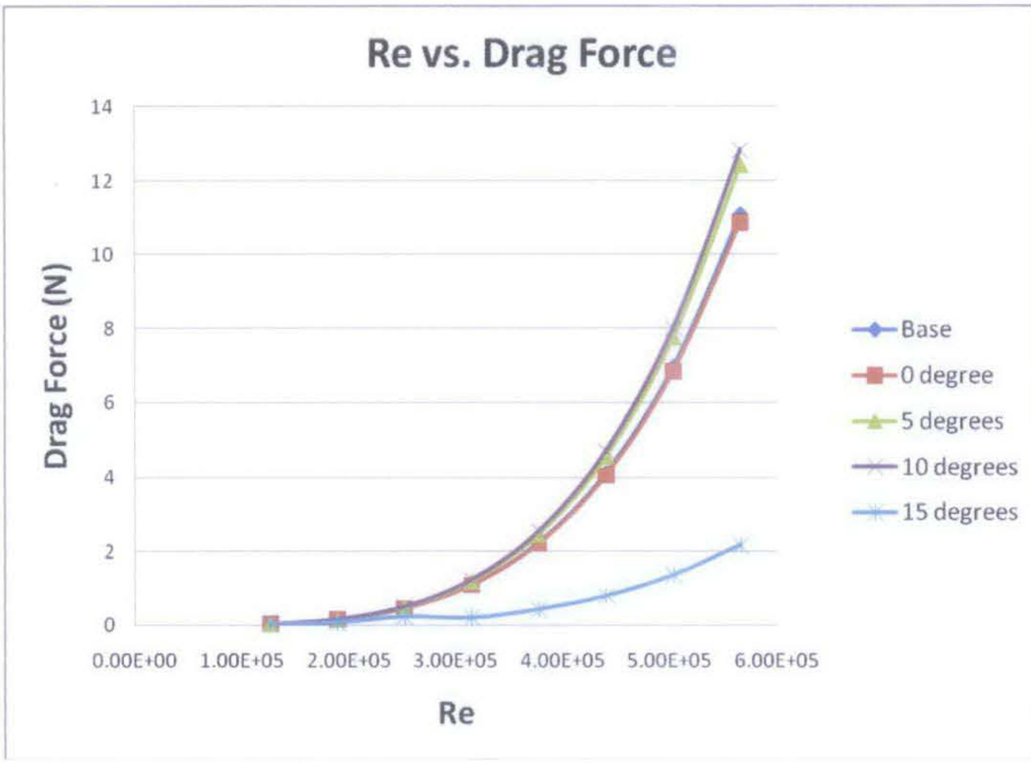



**Figure 4.2: Spoiler model B Re vs. Drag force chart.**

Looking at the results from the chart it is very clear that spoiler model A is not going to work because it actually increased the drag even though it manage to create down force (see appendices B), on the other hand spoiler model B was able to decrease drag by 27 % at maximum speed at $15^0$ spoiler angle and it also had considerable good down force see appendices B. The best spoiler angle for model B is 15 degrees. Above 15 degrees the spoiler creates more drag.

### 4.1.2. Flow visualization

Flow visualization is a tool used to study the flow patens around the model or body being simulated. Flow visualization enables the user to visually see the pressure contours, and be able to identify the areas where the pressure is most concentrated. Also other visualization like turbulence and velocity vectors etc can be seen using this software.

#### 4.1.2.1. Pressure

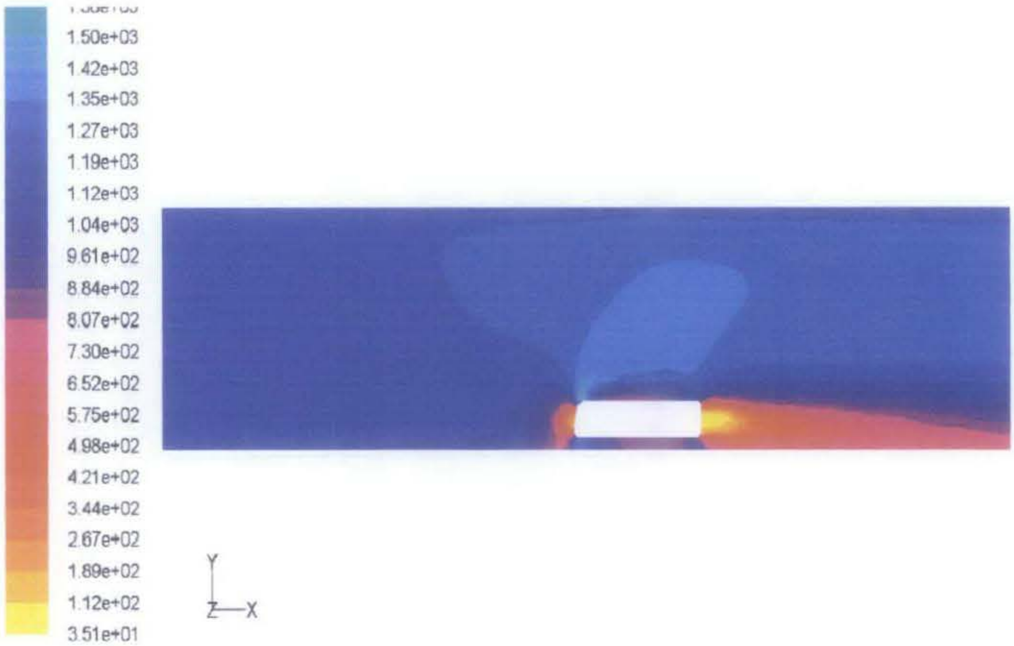



**Figure 4.3a: Base model dynamic pressure contour at 45 m/s.**

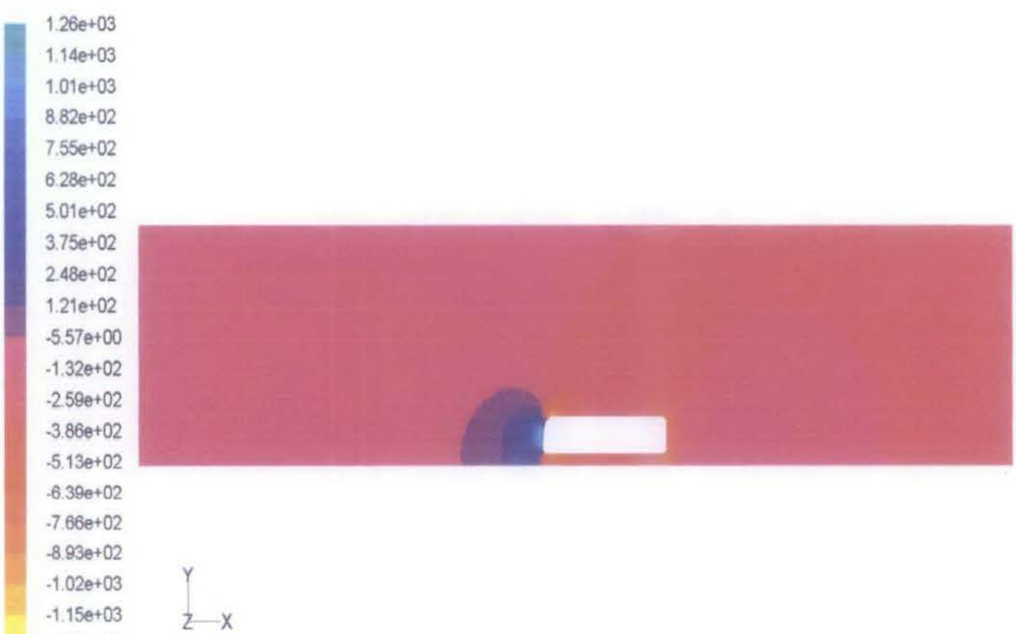



**Figure 4.3b: Base model static pressure contour at 45 m/s.**


**Figure 4.4a: Spoiler model B ($15^0$) dynamic pressure contour at 45 m/s.**

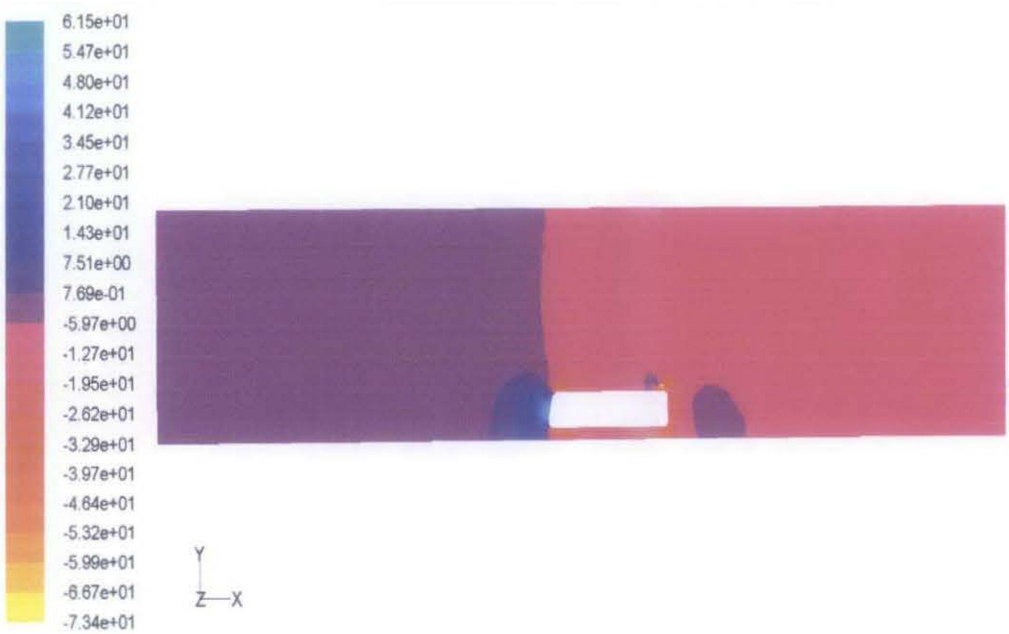



**Figure 4.4b: Spoiler model B ($15^0$) static pressure contour at 45 m/s.**

From the dynamic pressure images shown above it can be seen that the spoiler model managed to decrease some of the pressure that was distributed in the front end of the bus which can be seen by the dens blue color concentration in the front also it manage to increase a bit of pressure at the back of the bus which is seen by the yellow color contour which is more concentrated in the second figure, also the spoiler managed to decrease some of the down force that might create unnecessary down force and slow the bus even more, the differences are also seen in the static pressure images above, the pressure contours changed for all the other angles also (see appendix C), for better pressure recovery at the back an inclusion of a vortex trapping system would be useful but that presented problems with the simulation.

### 4.1.2.2. Turbulence

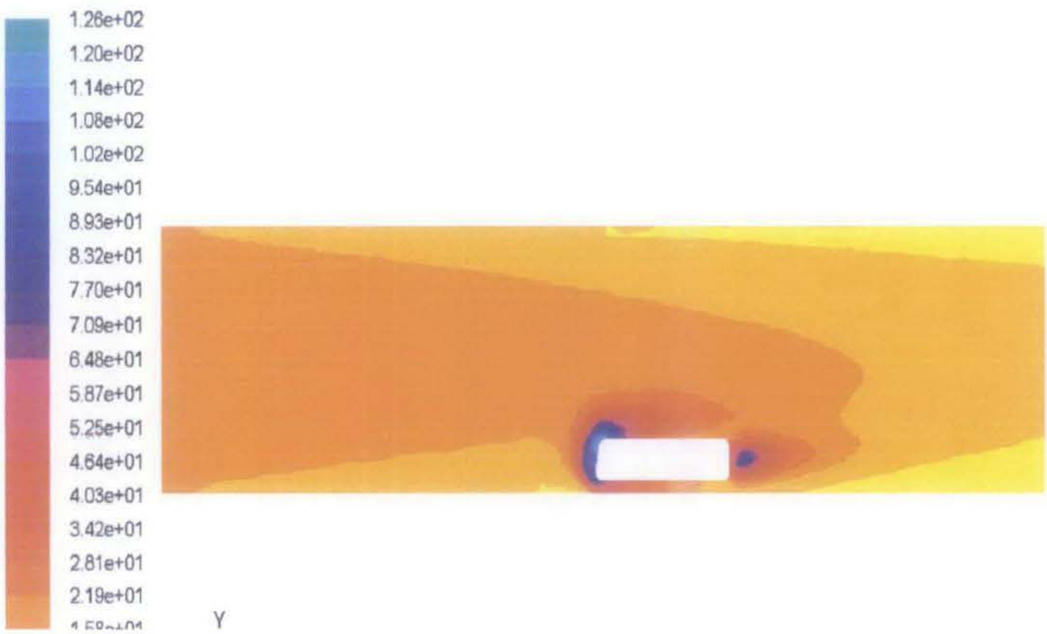



**Figure 4.5: Base model turbulence contour at 45 m/s.**

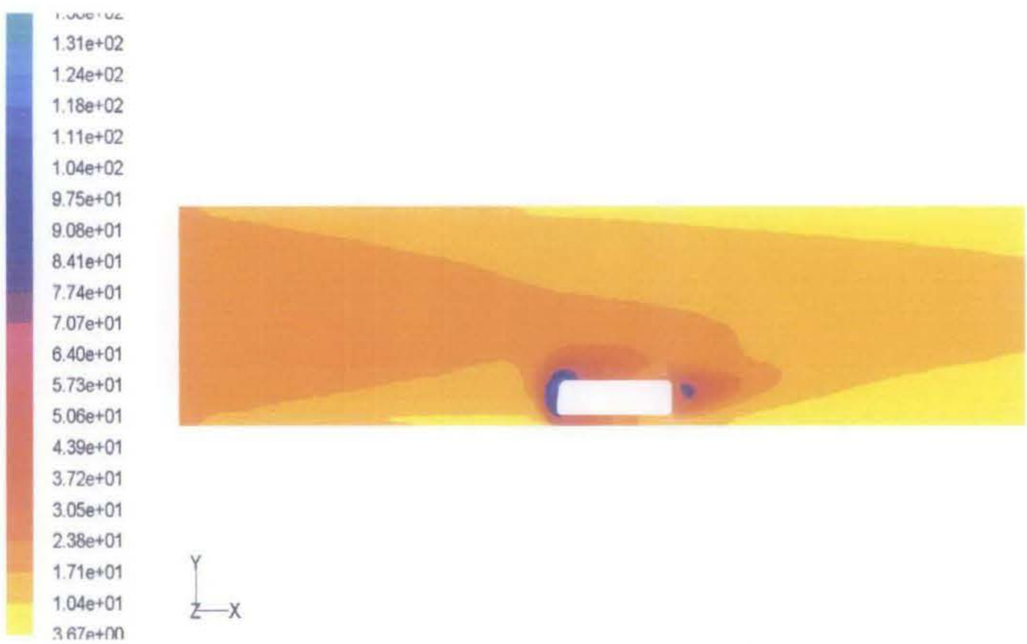



**Figure 4.6: Spoiler model B ($15^0$) turbulence contour at 45 m/s.**

From the two turbulence images above it is easy to see that the spoiler model managed to decrease most of the drag that was concentrated at the front end of the bus which was creating a lot of negative pressure and it also minimized some of the turbulence at the back of the bus with this reduction in turbulence the ride is going to be more stabilized.

### 4.1.2.3. Velocity

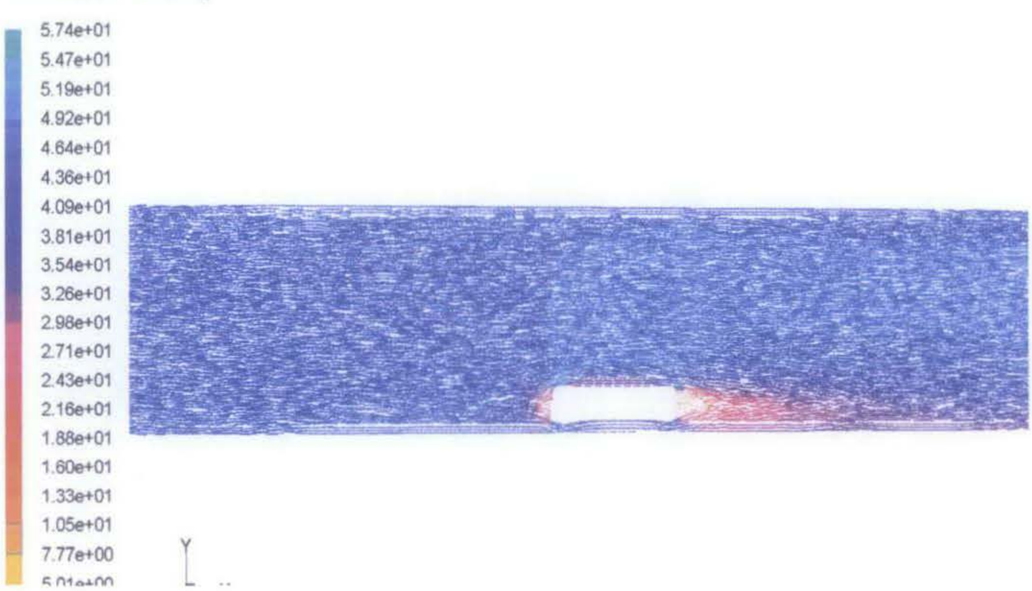



**Figure 4.7a: Base model velocity vector at 45 m/s.**

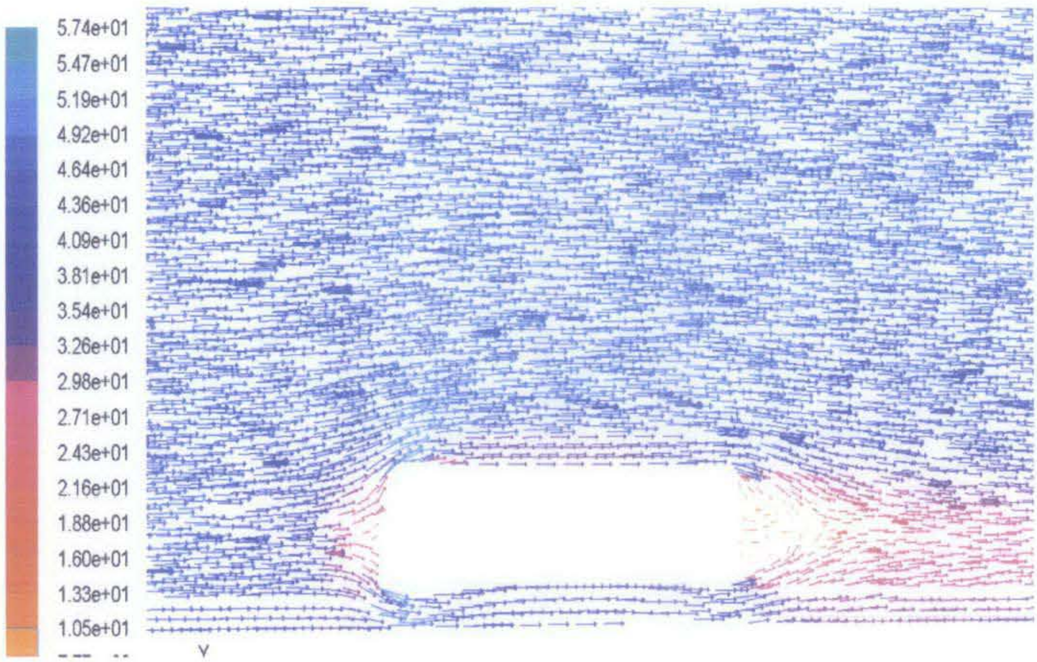



**Figure 4.7b: Base model velocity vector at 45 m/s.**

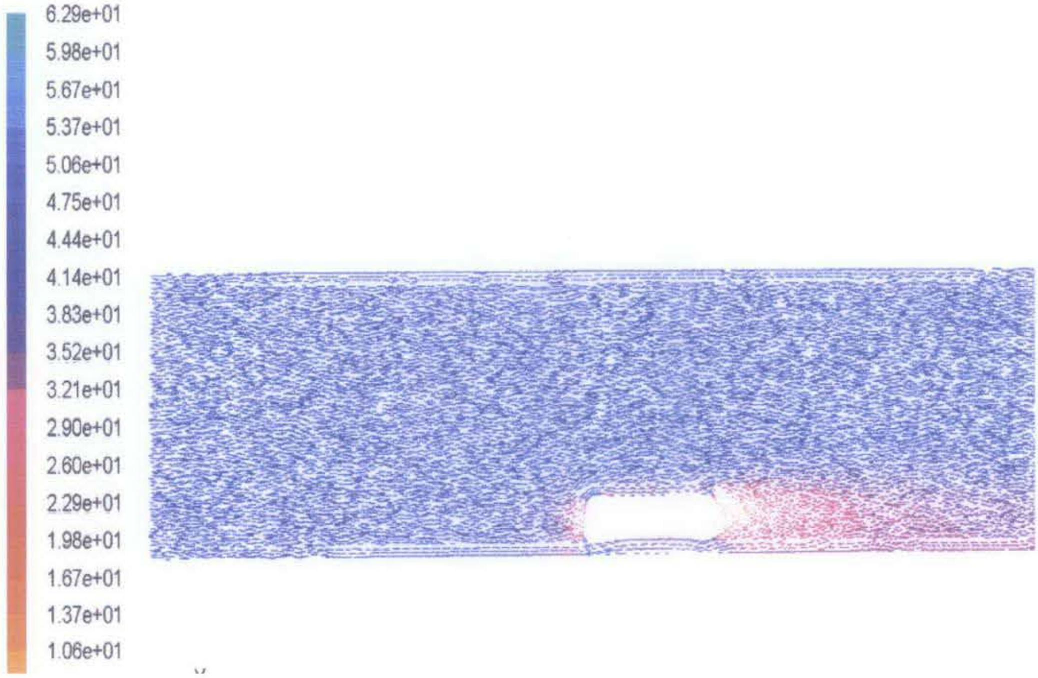



**Figure 4.8a: Spoiler model B ($15^0$) Velocity vector at 45 m/s.**

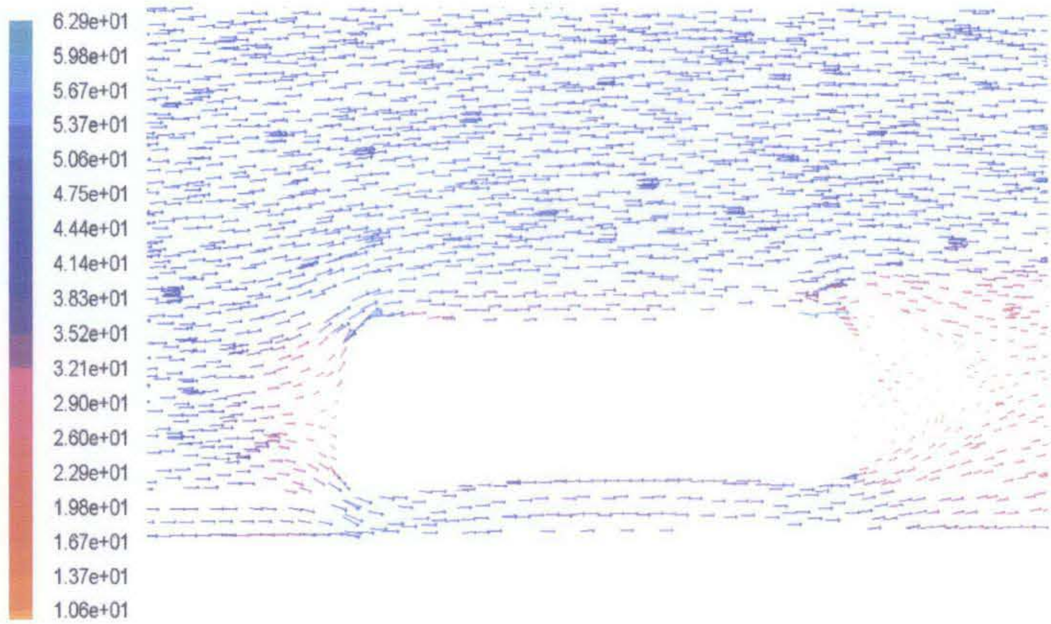



**Figure 4.8b: Spoiler model B ($15^0$) Velocity vector at 45 m/s.**

From the velocity vector images above for both the base model and spoiler model B there is not much difference, the most noticeable difference would be the high velocity vectors at the front edges of the bus for the base model which is caused by the turbulence created at that region compared to those of the spoiler model.

# CHAPTER 5

## 5.1. CONCLUSION

From the results above it is evident that the best spoiler model that reduced the most drag was spoiler model B at an angle of $15^0$, in that case for the active drag reduction system the drag reduction profile has to change at different conditions. In this case the controlling factor will be speed, at different speeds the profile of the system must change. For the purpose of this project the active drag reduction system will be the change in spoiler angle according to different speeds, reading from the Re vs. Drag force chart for spoiler model B when the speed is from 0 to 20 m/s the angle of the spoiler will be $5^0$, then from speeds higher than 20 m/s the angle of the spoiler should change to $15^0$ in order to reduce drag. The drag reduction system can also be used as a breaking system during emergencies by adjusting the angle of the spoiler to generate more drag.

## 5.2. RECOMMENDATION

For improving the drag reduction the inclusion of a vortex trapping system will help especially with streamlining the flow and pressure recovery at the back of the bus, but the inclusion of the vortex trapping system is found to have complicated the geometry by the inclusion of more small angles and faceses which requires new size functions and a finer mesh size, this compromises the file size and it is difficult to simulate the mesh file if the computer is not power full enough to initialize the geometry, another possibility is to change the geometry of the bus to a more streamed line shape that will also have a great effect on turbulence reduction. 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 the must be an account for ground effect.

## REFERENCES:


### Books:

[1]. Fluid mechanics Fundamentals and applications 2006.

### Journals:

[2]. Bus drag reduction by trapped vortex concept for a single bus and two buses in tandem by C.A.J. Fletcher and G.D.H. Stewart.

[3]. The effect of vehicle spacing on the aerodynamics of a representative car shape by Simon Watkins and Gioacchino Vino.

[4]. Drag reduction of motor vehicles by active flow control using the Coanda effect by Geropp D and Odenthal H.-J.

[5]. Drag Reduction of an Ahmed car model by means of active separation control at the rear vehicle slant by A. Brunn and W. Nitsche.

[6]. The International Energy Outlook 2009 report

### Website:

[7]. www.inderscience.com.

[8]. www.sciencelinks.com.

[9]. www.autos.canada.com.

APPENDIX - A

| No. | Detail/ Week | 1 | 2 | 3 | 4 | 5 | 6 | 7 | Mid-semester break | 8 | 9 | 10 | 11 | 12 | 13 | 14 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Research work | ■ | ■ | ■ | | | | | | | | | | | | |
| 2 | Submission of Progress Report 1 | | | | ● | | | | | | | | | | | |
| 3 | Design and simulation | | | | | ■ | ■ | ■ | | | | | | | | |
| 4 | Submission of Progress Report 2 | | | | | | | | | ● | | | | | | |
| 5 | Seminar | | | | | | | | | ● | | | | | | |
| 6 | Simulation | | | | | | | | | ■ | ■ | ■ | ■ | | | |
| 7 | Poster Exhibition | | | | | | | | | | | ● | | | | |
| 8 | Submission of Dissertation (soft bound) | | | | | | | | | | | | | ● | | |
| 9 | Oral Presentation | | | | | | | | | | | | | | ● | |
| 10 | Submission of Project Dissertation (Hard Bound) | | | | | | | | | | | | | | | ● |

● Suggested milestone

■ Process

## APPENDIX – B

**Table B.1: Spoiler model A results.**

| | | Down Force (N) | | | | |
|---|---|---|---|---|---|---|
| Speed (m/s) | Re | Base | Spoiler 0 | spoiler 5 | spoiler 10 | spoiler 15 |
| 10 | 1.26E+05 | 0.004 | 0.006 | 0.006 | 0.007 | 0.009 |
| 15 | 1.89E+05 | 0.020 | 0.030 | 0.030 | 0.036 | 0.043 |
| 20 | 2.52E+05 | 0.059 | 0.091 | 0.090 | 0.112 | 0.133 |
| 25 | 3.15E+05 | 0.140 | 0.218 | 0.215 | 0.269 | 0.321 |
| 30 | 3.77E+05 | 0.285 | 0.447 | 0.441 | 0.554 | 0.662 |
| 35 | 4.40E+05 | 0.521 | 0.822 | 0.808 | 1.020 | 1.220 |
| 40 | 5.03E+05 | 0.881 | 1.396 | 1.368 | 1.734 | 2.075 |
| 45 | 5.66E+05 | 1.402 | 2.227 | 2.181 | 2.771 | 3.317 |

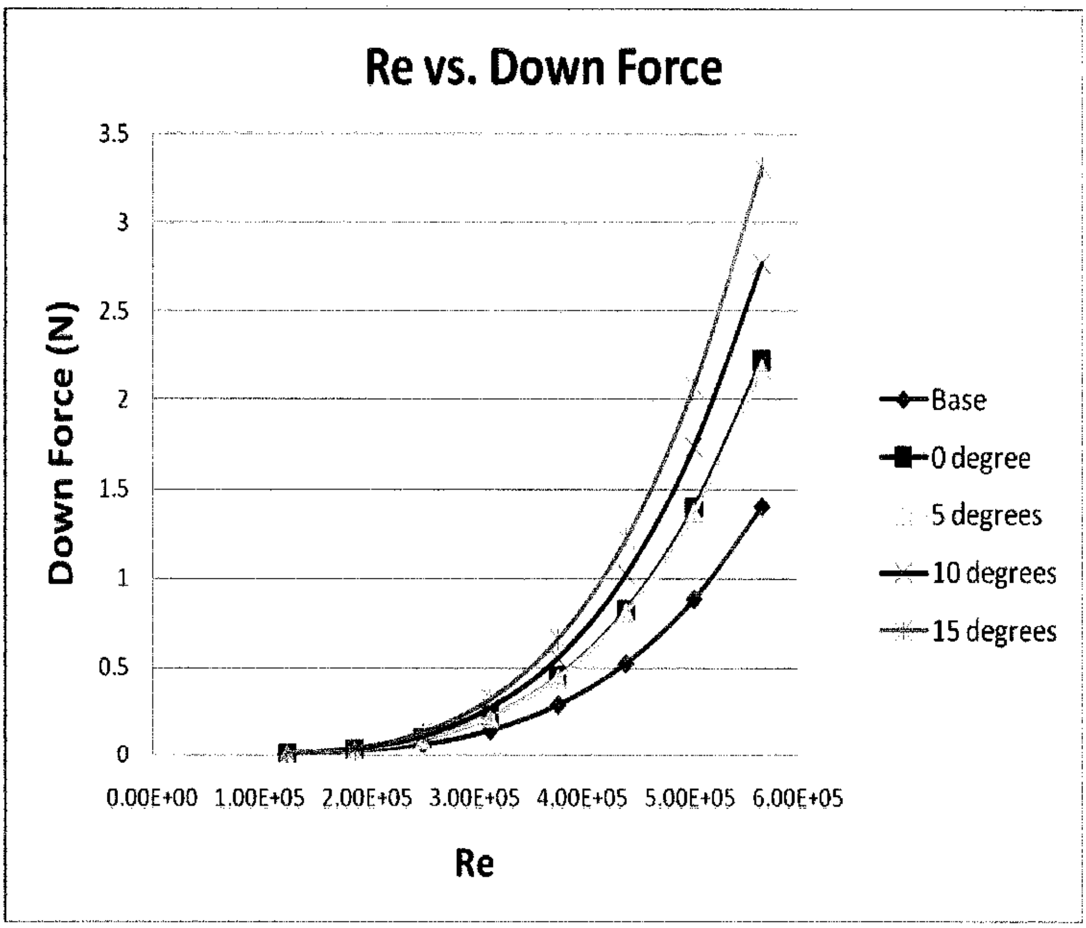



**Figure B.1: Spoiler model A Re vs. Down force.**

**Tale B.2: Spoiler model B results.**

| | | Down Force (N) | | | | |
|---|---|---|---|---|---|---|
| Speed (m/s) | Re | Base | Spoiler 0 | spoiler 5 | spoiler 10 | spoiler 15 |
| 10 | 1.26E+05 | 0.004 | 0.005 | 0.007 | 0.009 | 0.011 |
| 15 | 1.89E+05 | 0.020 | 0.022 | 0.034 | 0.044 | 0.025 |
| 20 | 2.52E+05 | 0.059 | 0.068 | 0.108 | 0.141 | 0.079 |
| 25 | 3.15E+05 | 0.140 | 0.162 | 0.265 | 0.346 | 0.070 |
| 30 | 3.77E+05 | 0.285 | 0.329 | 0.552 | 0.720 | 0.145 |
| 35 | 4.40E+05 | 0.521 | 0.600 | 1.027 | 1.336 | 0.269 |
| 40 | 5.03E+05 | 0.881 | 1.011 | 1.757 | 2.284 | 0.459 |
| 45 | 5.66E+05 | 1.402 | 1.605 | 2.820 | 3.665 | 0.736 |

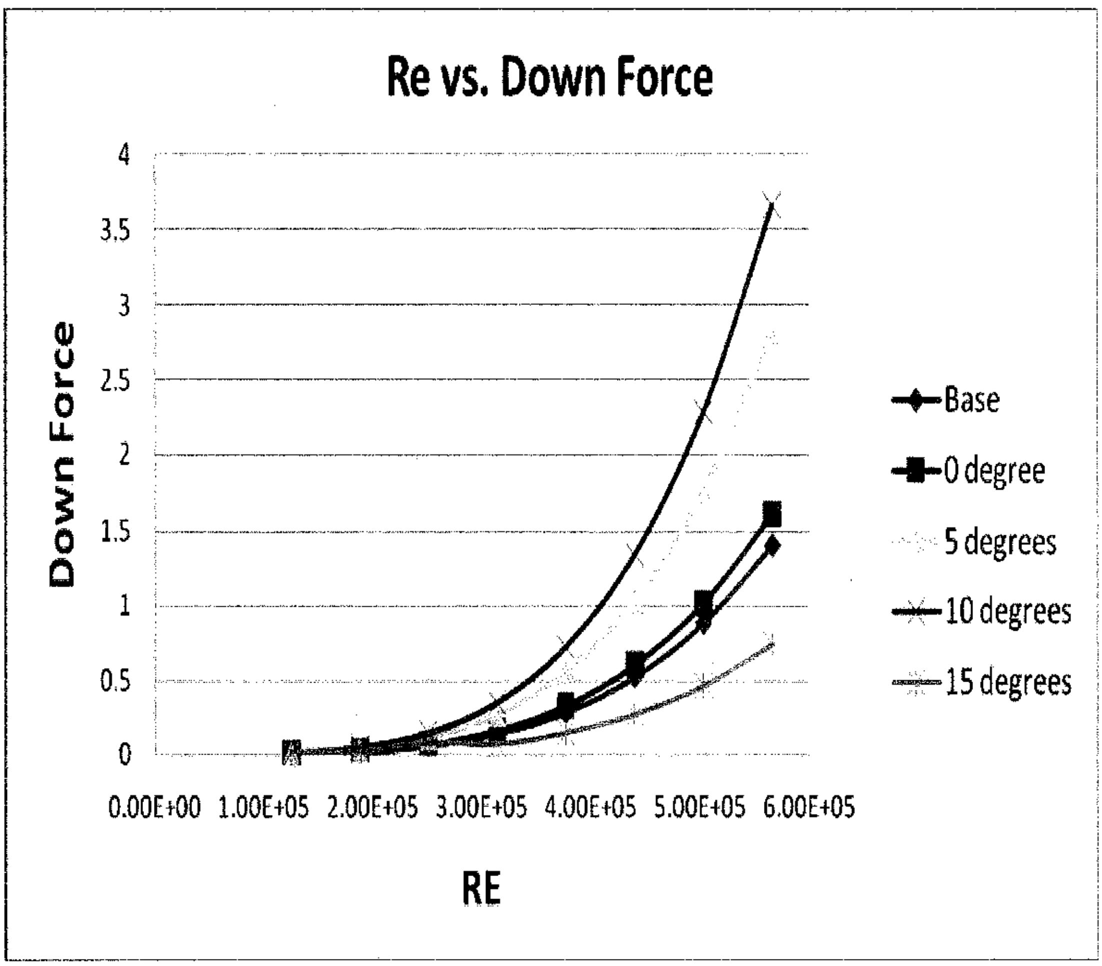



**Figure B.2: Spoiler model B Re vs. Down force chart.**

# APPENDIX – C

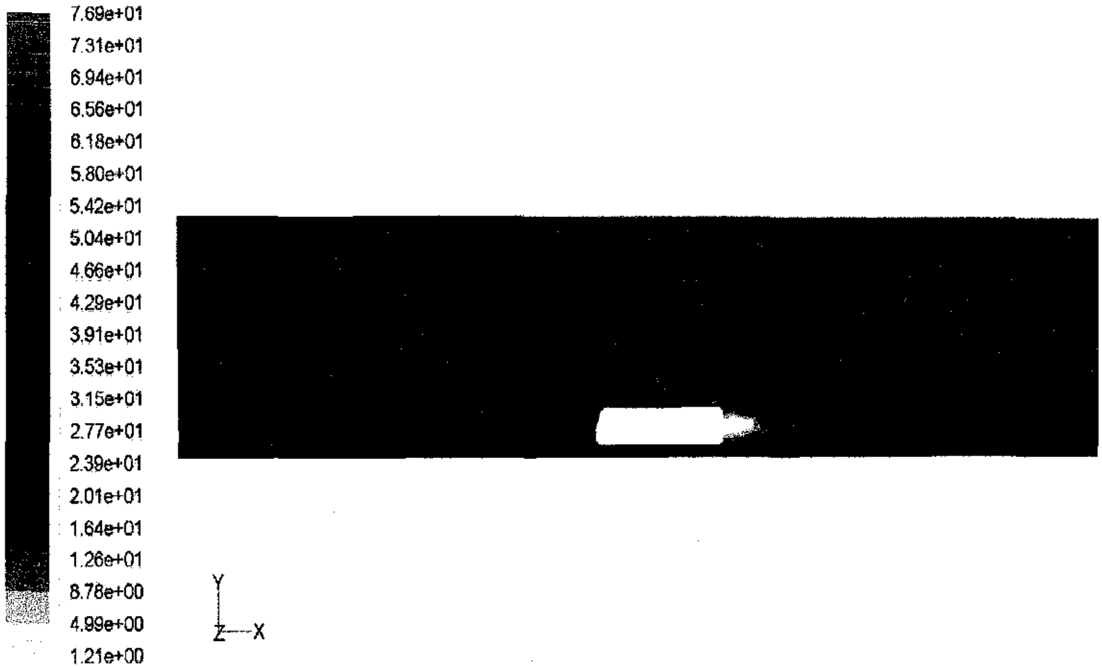



**Figure C.1: Spoiler model B ($5^0$) dynamic pressure contour at 10 m/s.**

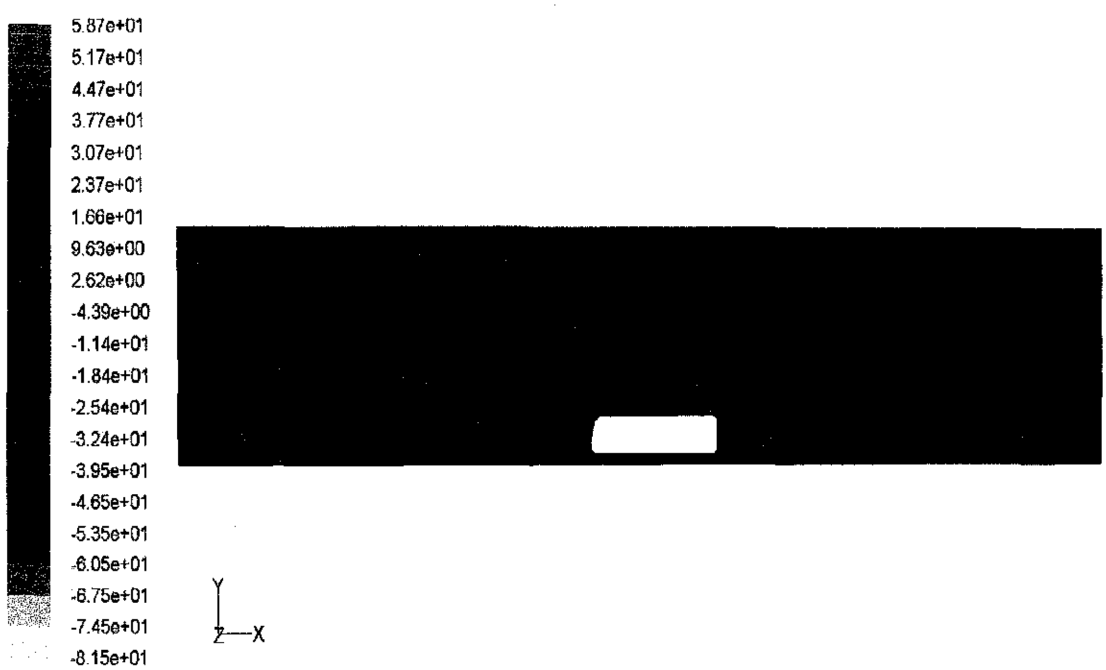



**Figure C.2: Spoiler model B ($5^0$) static pressure contour at 10 m/s.**

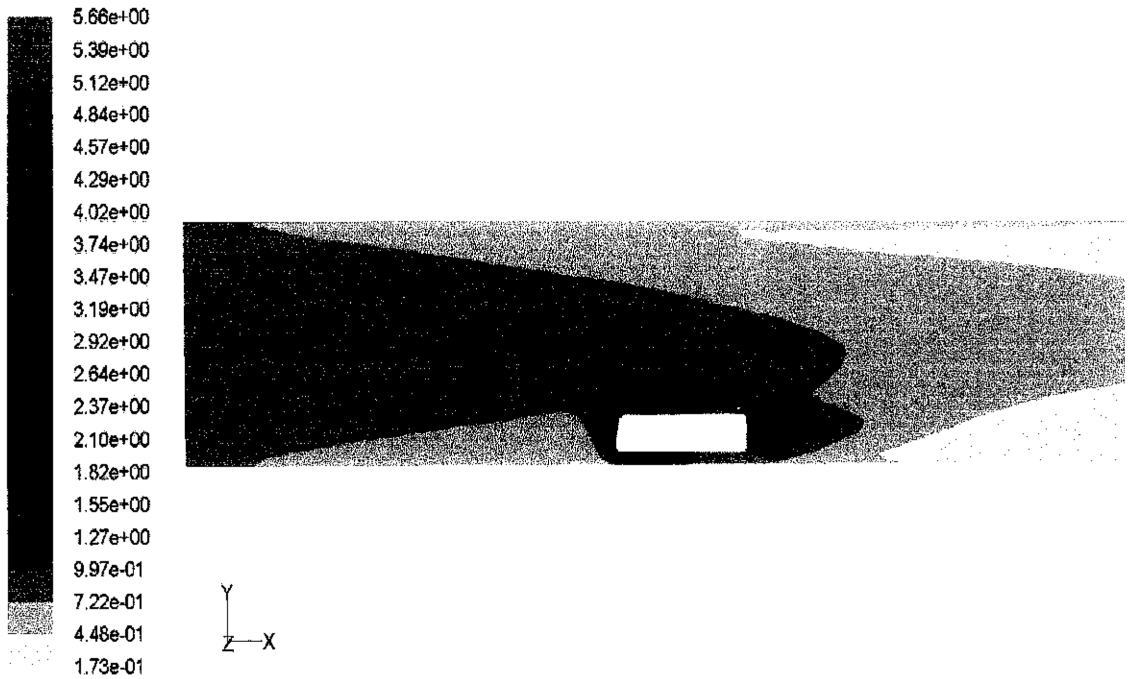



**Figure C.3: Spoiler model B ($5^0$) turbulence contour at 10 m/s.**

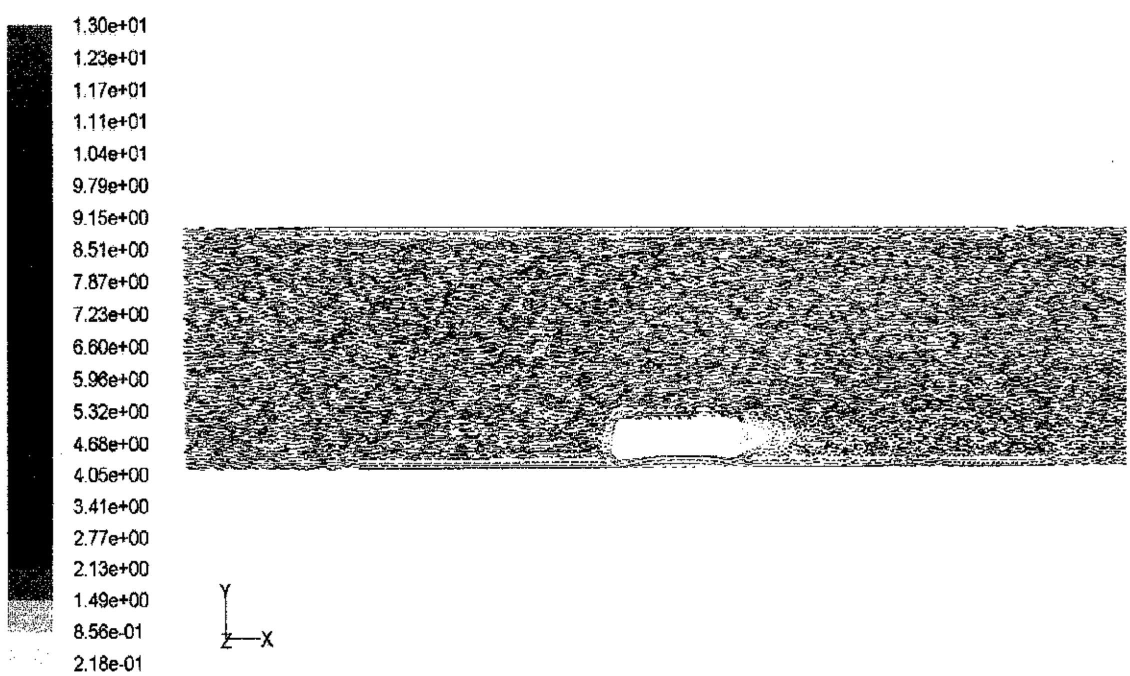



**Figure C.4: Spoiler model B ($5^0$) Velocity vector at 10 m/s.**