EFFECTS OF AERODYNAMIC CHANGES FOR A CAR CAUSED BY ANOTHER IN ITS PROXIMITY: NUMERICAL STUDY

By

Amirullah Bin Isa

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

JANUARY 2009

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

EFFECTS OF AERODYNAMIC CHANGES FOR A CAR CAUSED BY ANOTHER IN ITS PROXIMITY: NUMERICAL STUDY

by

Amirullah Bin Isa

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:

(Dr. Ahmed Maher Said Ali)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
January 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.

AMIRULLAH BIN ISA

ABSTRACT

This project is study about the effect of aerodynamic changed for a car caused by another in its proximity. When the cars are moving near each other like moving back to back, aerodynamic of the fluid surrounding two vehicles are changing. The changes of aerodynamic can generate severe force variation on the vehicles and these forces can have an adverse effect on vehicle handling and stability. Aerodynamic changes are studied on models of vehicle using Computer Fluid Dynamic (CFD). The aim is to validate the CFD models against experiment data which were carried out previously at UTP. The model of the car and simulation will be carried out by using Gambit and Fluent software. The simulations include different experimental cases with varying the separating distance between the car models. Once the CFD simulations are validated, more case can be simulated and extended conclusions can be drawn out. The study is focusing on the drag force and lift force as result of aerodynamic changes when the vehicles in specific position. The results from the simulations show that, the best position for aerodynamically which is less Drag force and safety condition was directly behind the other cars which at one and half width distance (18.75 cm).

ACKNOWLEDGEMENTS

Bless to God for giving the author strength and chance to complete the Final Year Project (FYP) and this dissertation report. The author would like to extend his greatest gratitude to the following people for their contribution and cooperation during the FYP process.

Supervisor:	Dr. Ahmed Maher Said Ali for his supervision plus giving freedom to learn throughout the project. Thank you for your technical expertise advices, information, materials, support and experiences as well.
UTP staff:	All of UTP staffs that have helped a lot in giving information in Wind Tunnel Experiment
Friends:	All people who help the author in making this project possible to finish, including the support, advice and encouragement.
Family:	For their moral support and praying.

The author would like to thank all people who involve in undergoing this project.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1. INTRODUCTION

1.1 Background of the Study

When two vehicles are driven in close proximity, the wind changes can be felt around each other and under certain conditions they can generate severe force. This situation is due to the changes of aerodynamic flow around the vehicles. The aerodynamic changes are much more significance when the size and speed of vehicles increase. As one vehicle passes another on the road, flow fields around the two vehicles will generate transient aerodynamic forces. These forces can have an adverse effect on vehicle handling and stability. For this situation, Computational Fluid Dynamic (CFD) will build a computational model that represents a system or device to analyze and then apply real word physics and chemistry to the model. Beside that CFD will provide the images and data which predict the performance of the design.

1.2 Problem Statement

The forces like drag force and lift force can affect the other vehicle which can alter their road holding and thus result in safety problem. Small size of the vehicle will certainly feel more of the wake from larger heavier vehicles. To simulate the experiment about this project, software Gambit and Fluent will use. Gambit will be used to make geometry setup like a model of a car while Fluent will be used to run the simulation. This project used same dimension and same distance with the real experiment which were carried out previously at UTP. The result will show either same dimension will give same result or not.

1.3 Significant of Project

This project is mainly to investigate the reaction of the aerodynamic forces that react on vehicle in tailing each other. It is significant to the drivers to know the exact distance when they following the other vehicle. Studying the effect of these aerodynamic changes could help in minimizing the risk of an accident.

1.4 Objectives

- To make simulation using Computational Fluid Dynamic (CFD)
    - Model for two vehicles in proximity each other will be simulated using Gambit and Fluent software.
- To study the effect of velocity of vehicles on aerodynamic force.
    - The different speed of the model will show different effects on aerodynamic force.
    - Studying this effect will give more knowledge about the speed for the driver when proximity with another car.
- To study the effect of separating distance on aerodynamic force.
    - Different Drag and Lift forces will get from different distances between two models

1.5 Scope of the Study

- Study the effect velocity on aerodynamic force.
  - When a car is moving in different speed, the aerodynamic force that reacts on the car is different.
- Make a Computational Fluid Dynamic (CFD) simulation for two cars in proximity each other at different distances and analysis the force around the vehicles.
  - The effect of the different distance between two models will be analysis using CFD simulation.
- To validate the CFD model when simulating the experiment.
  - The result from CFD simulation will be compared with result from wind tunnel experiment.

1.6 Feasibility of Study

This project was compared the result between wind tunnel experimental which were carried out previously in UTP with simulation using CFD. Based on the research, the simulation should show the same result because simulation using CFD must use same dimensions and same steps with experimental.

CHAPTER 2

LITERATURE REVIEW

First of all, understanding the basic theory of aerodynamic force is very important. Paper works, journal, engineering books or anything relevant to the project are reviewed to get more knowledge about the project.

2.1 Drag Force

One of the force that come when two car in proximity each others is drag force. When a car is moving, the force a flowing fluid exerts on a body in the flow direction is call drag force. It acts in the opposite direction of the movement of the body. Drag force must to be minimizing because it is undesirable effect like friction that cause some problem when a car is moving. Reduction of drag force in automotive can improved safety and durability of structures subjected to high wind and reduction of noise and vibration.

For this project, the drag force will be discussed when two cars are in proximity each other and this effect can be done using Computational Fluid Dynamic (CFD). Drag force will decrease and increase depends on the velocity and shape of the vehicle. Drag force increases with area. In the context of this model, area is the cross sectional area projected in the direction of motion. Different in area will give different value for the drag force [1].Drag force also increases with speed. An object that is stationary with respect to the fluid will certainly not experience any drag force [1].

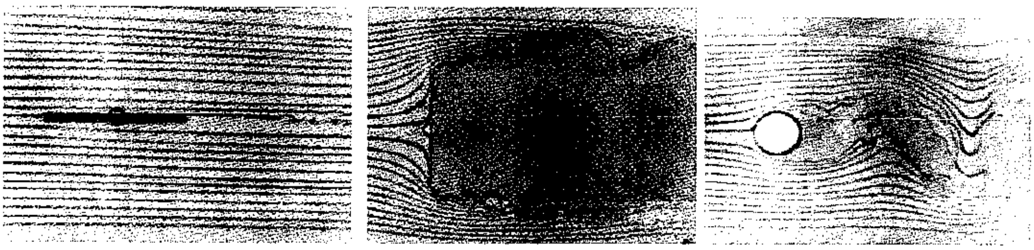

Figure 2.1: Flow of the Drag Force [1].

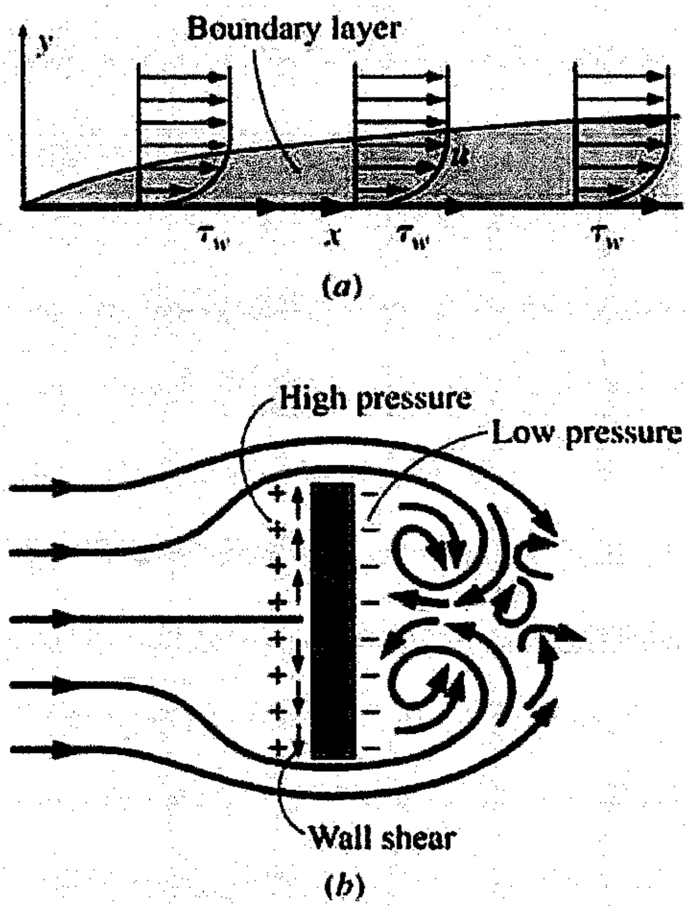

Figure 2.2: (a) The drag force acting on a flat plat parallel to the flow depends on the wall shear [1].
(b) The drag force acting on a flat plat normal to the flow depends on the pressure [1].

2.2 Drag Coefficient

The drag coefficient, in general, depends on the Reynolds number. At higher Reynolds numbers, the drag coefficients for most geometry remain essentially constant [1].This is due to the flow at high Reynolds numbers becoming fully turbulent.

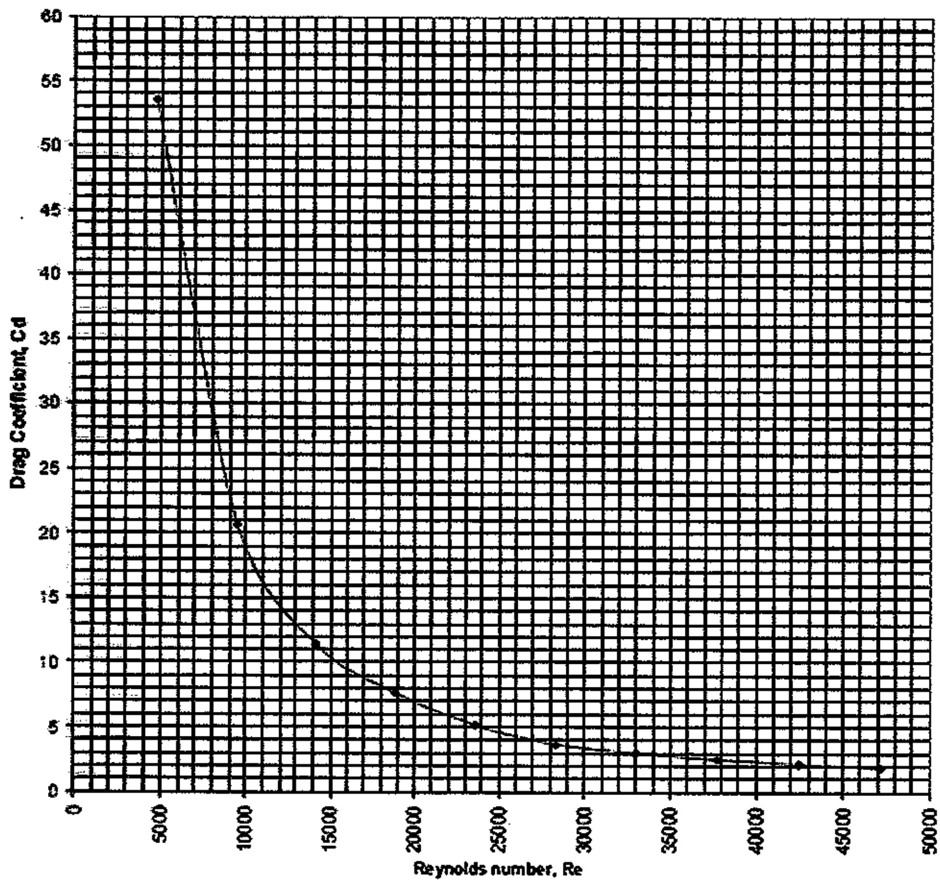

Figure 2.3: Typical graph of Drag coefficient versus Reynolds number [1].

Drag coefficient is inversely proportional to the Reynolds Number. The higher value of the Reynolds Number, drag force becomes smaller. The slope decrease until a certain constant C_D value [1].

Drag coefficient can get from the equation below.

$$C_D = \left[\frac{F_D}{0.5\rho V^2 A_F}\right] \quad \text{(1)}$$

C_D = drag coefficient

F_D = drag force

ρ = density of air

V = freestream velocity of the air

A_F = frontal area of the prototype/model

2.3 Lift Force

In the context of fluid flow relative to a body, the lift force is the component of the aerodynamic force that is perpendicular to the flow direction. It contrasts with the drag force which is parallel to the flow. Lift is generated in accordance with the fundamental principles of physics such as Newton's laws of motion, Bernoulli's principle, conservation of mass and the momentum. In automotive field, lift force are caused by difference in pressure acting on a body. Lift depends entirely on the nature of viscous flow past certain bodies in inviscid flow, there is no lift without imposing a net circulation. When there is no flow, there is no lift and the forces acting on the car are zero.

For the experiment, the lift will not be considered as the wind tunnel testing will not be accurate as the lift is associated with the ground effect. However, there will be no ground or road in the experiment. The models of the vehicle will be supported by a metal rod, connecting it to the balance that will measure the forces acting on the model. For this project, the effect of the lift force will discuss when simulating the model using Computer Fluid dynamic (CFD) [1].

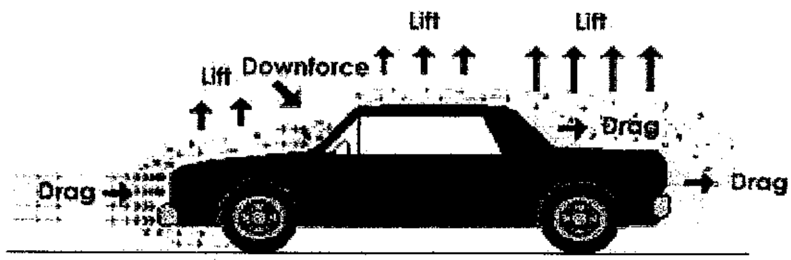

Figure 2.4: Lift force

2.4 Lift Coefficient

The lift coefficient is a number that aerodynamicists use to model all of the complex dependencies of shape, inclination, and some flow conditions on lift. Lift coefficient means the dimensionless quantity that describes the characteristic of the lift on the body. The lift coefficients vary along the surface as a result of the changes in the velocity boundary layer in the flow direction [1].The equation that used to get calculated lift coefficient is:

$$C_L = \left[\frac{F_L}{0.5\rho V^2 A_P}\right] \quad (2)$$

C_L = lift coefficient

F_L = lift force

ρ = density of the air

V = velocity of the air

A_P = top/platform area (parallel to the flow) of the object

2.4 Dimensional Similarity

In order to gain accurate accuracy result, the model that will build from computer must have similarities to the real world. These similarities are the concept of technique called Dimensional Similarity. Dimensional Similarity is introduced so that the test can be done on scale model rather than a prototype. There are three conditions that are needed to complete the similarity which are:

- Geometric similarity - model has the same shape with the size being scaled.
- Kinematics similarity – velocity at any point in the model flow must be proportional to the velocity at the corresponding point in the prototype flow.
- Dynamic similarity – all forces in the model flow are scaled by a constant factor to the corresponding force in the prototype flow.

Size the model that will use in the computer must be same with the model of car that use in wind tunnel. In wind tunnel, to determine the appropriate size, some calculation of dimensional analysis needs to be done [1].

$$\left[\frac{\rho VL}{\mu}\right]_{prototype} = \left[\frac{\rho VL}{\mu}\right]_{model} \quad \ldots\ldots (3)$$

Where:

ρ = density of air

V = freestream velocity of the air

μ = kinematic viscosity

L = Length of the model

2.5 Meshing

Meshing is an integral part of the CAE analysis process. The mesh influences the accuracy, convergence and speed of the solution. More importantly, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from a CAE solution. Therefore, the better and more automated the meshing tools, the better the solution. From automatic meshing to highly crafted mesh, ANSYS, Inc. provides the ultimate meshing solution. ANSYS provides powerful pre- and post-processing tools for mesh generation from any geometry source, to produce almost any element type, for nearly any physics, for virtually any application [5].

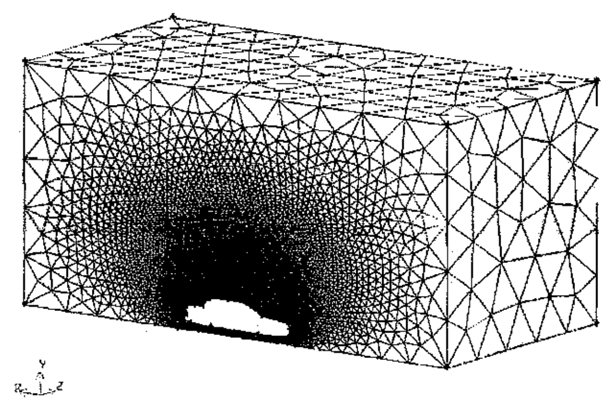

Figure 2.5: Typical portion of the volume mesh [5].

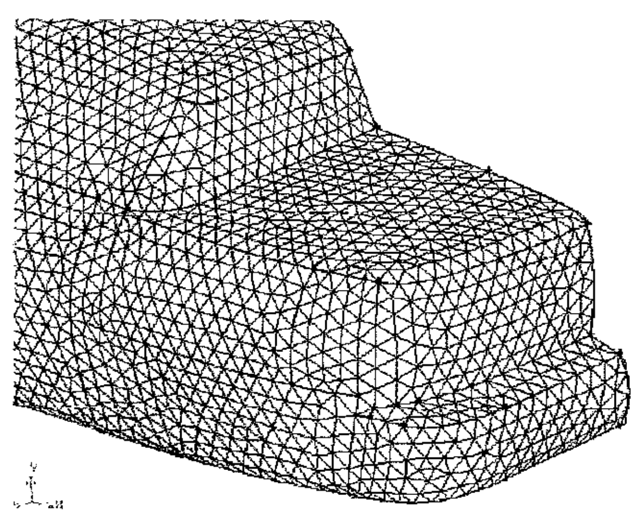

Figure 2.6: Typical surface mesh on rear of car body

CHAPTER 3

METHODOLOGY

For completing this project, there are several methodologies need to be done. The project will start by literature review, analysis of Wind Tunnel Experiment, CFD simulation, calculation for drag and lift forces, and comparing computational result with experimental result. Softwares that used for this project were Fluent, Gambit, Microsoft Word and Microsoft Excel. Figure 3.1 below illustrated the flow chart of the project.

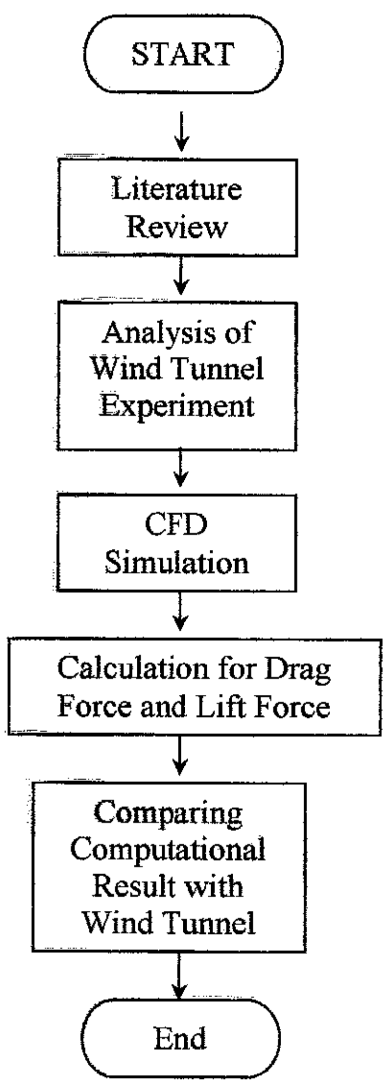

Figure 3.1: Flow chart of the project

3.1 Literature Review

First of all, understanding the basic theory of aerodynamic force is very important. Paper works, journal, engineering books or anything relevant to the project are reviewed.

3.2 Analysis Wind Tunnel Experiment

In order to make CFD simulation for this project, the wind tunnel experiment was reviewed. The purpose of this analysis is to know the value of the velocity, distances between two cars and the dimension of the car that used in experiment. The CFD simulation must use same value with experimental because at the end of this project, both result will be compared.

3.3 Computational Fluid Dynamic (CFD) Simulation

To investigate the force that react when two cars in proximity to each other, some simulation using computer were done. At this stage, there were several steps need to be done before run the simulation.

a) Building the model - Building the model within Gambit software using same dimension with wind tunnel model.

b) Meshing process - The model will separate into small pieces and it will show the force that react on the model. Beside that, the mesh will be examined to check the quality of resulting mesh.

c) Boundary Condition - There are several options at the boundaries through which fluid enters the computational domain (inflow) or leaves the domain (outflow). At a velocity inlet, the velocity that occurs is the velocity of the incoming flow along the inlet face. Beside velocity inlet, there are other boundaries which are pressure outlet and wall.

d) Decide separating distances – The separating distances that used in simulation were 6.25 cm, 12.5 cm, 18.75 cm, and 25 cm.

e) Decide velocity value – the speed of the air in simulation for all cases were same with speed in experiment which are 5m/s,10m/s,15m/s,20m/s,25m/s, 30m/s,35m/s,40m/s,45m/s, and 50 m/s.

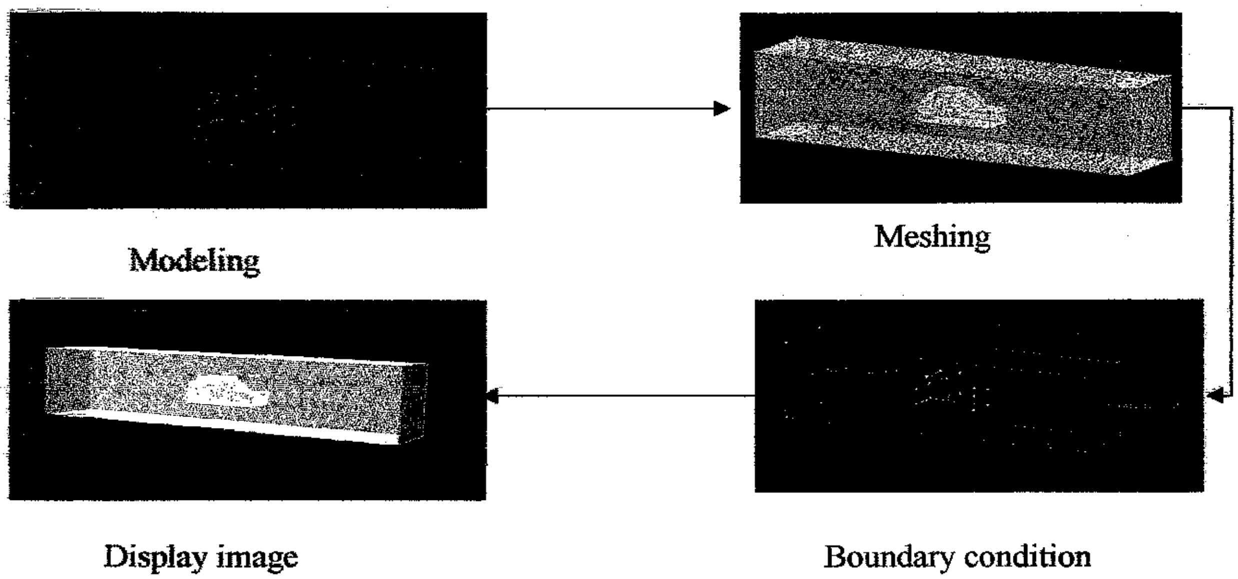

Figure 3.2: CFD procedure

3.4 Calculation for Drag and Lift Force

Drag force and lift force were calculated after known the value of drag coefficient and lift coefficient. From this value also, the Reynolds number will be calculated

3.5 Comparing CFD Result with Wind Tunnel Result

The last method that will use is comparing the result from computer simulation with wind tunnel experiment. Result from the simulation using computer should give more accurate result comparing with wind tunnel result. If the result in the simulation is same, this project will continue with the different value of the distances and various size of the car.

CHAPTER 4

RESULT AND DISCUSSIONS

4. RESULT AND DISCUSSIONS

4.1 Position of Car

Position of car in CFD was same with position in wind tunnel experiment. The dimension car in wind tunnel experiment was measured and applied in CFD. Figure below showed dimension of the model in wind tunnel.

Figure 4.1: Front view of wind tunnel

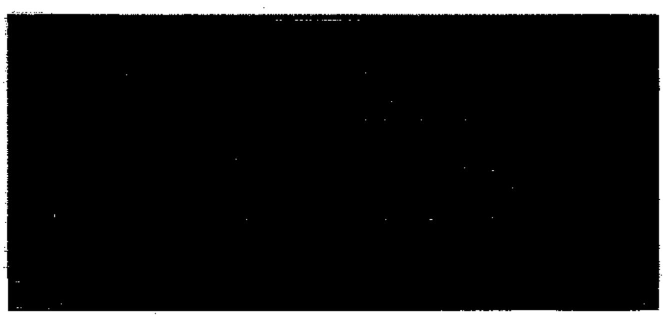

Figure 4.2: Top view of wind tunnel

DIMENSION:

1) A = 10 cm

2) B = 7 cm

3) C = D = 9.2 cm

4.2 Boundary Condition

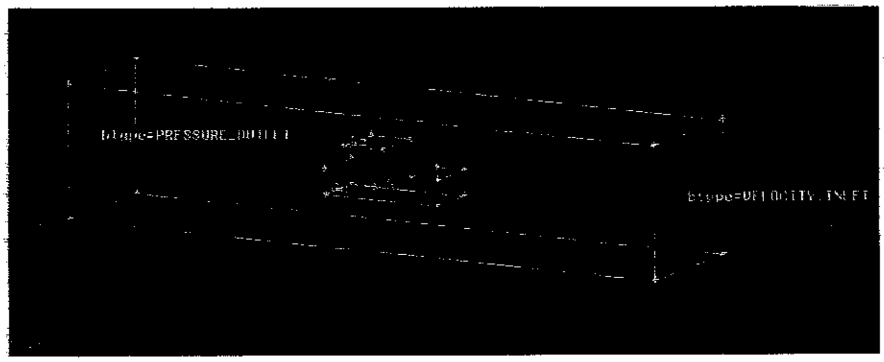

Figure 4.3: Boundary types

The figure shows the outer edges of car geometry those present wall boundaries. For example, the front and the back will show the velocity inlet and pressure outlet.

4.3. Meshing Process

The model of car and wind tunnel were split into two because in the meshing process, only wind tunnel model was meshed. The parameters of the meshing process for single model and double model were same. Figure below showed the meshing process for single model and double model which separating distance were 12.5 cm.

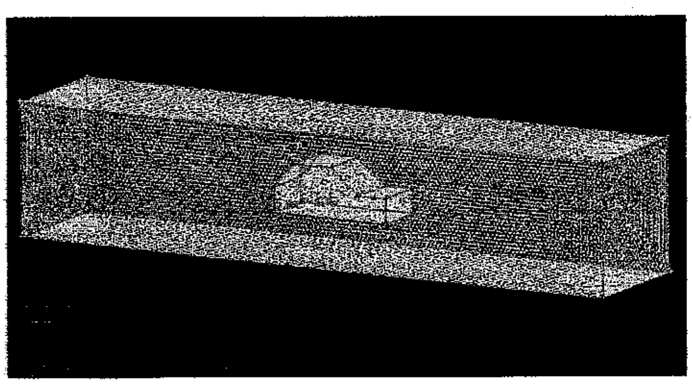

Figure 4.4: Meshing process for single model

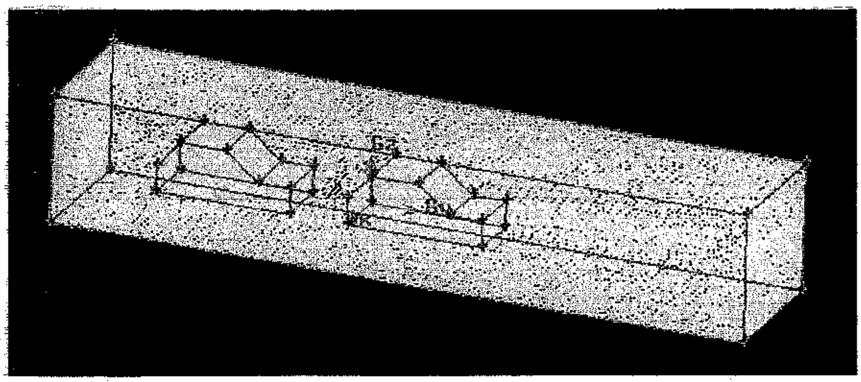

Figure 4.5: Meshing Process for double model (12.5 cm)

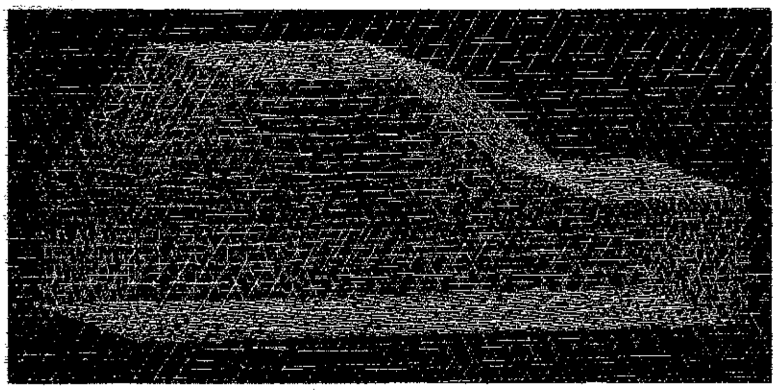

Figure 4.6: Meshing on body of the car

4.4 Postprocessing

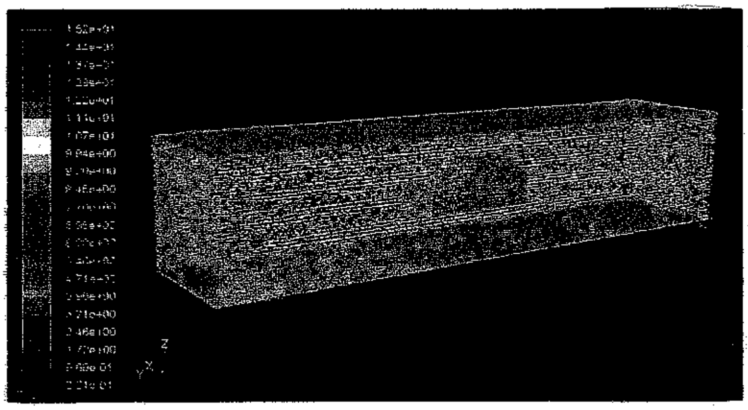

Figure 4.7: Velocity vector in wind tunnel

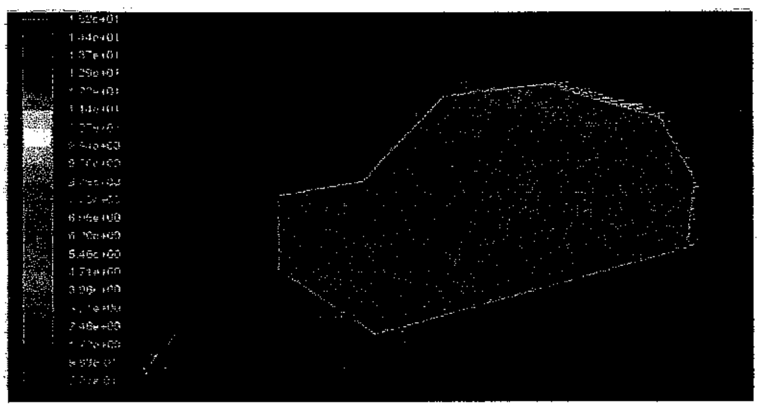

Figure 4.8: Velocity vector in m/s for single car

This figure illustrated the flow of velocity vector when simulation was run. The magnitude of the velocity changes from 5.46 m/s until 9.2 m/s when it flows into the model.

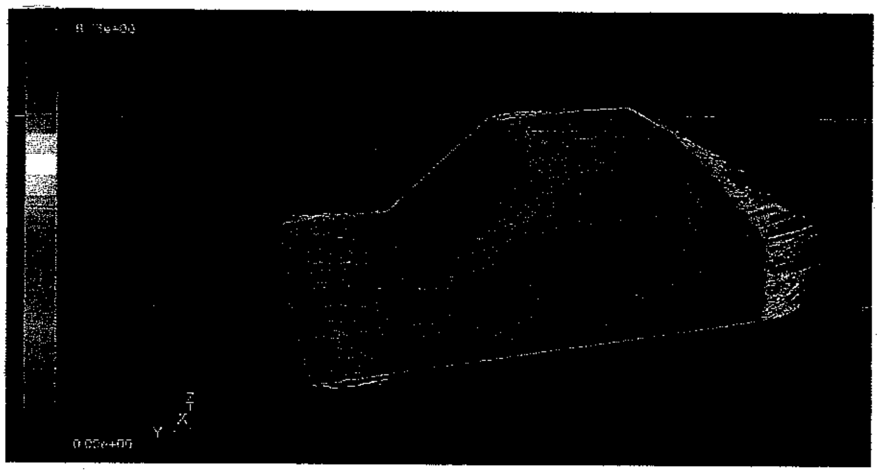

Figure 4.9: Path lines

Pathlines are the lines traveled by neutrally buoyant particles in equilibrium with the fluid motion. This figure shows that pathline that flow into model colored by velocity magnitude. The maximum velocity of air that flow into model is 8.73 m/s.

4.5 Convergence History

For the single model, with steady state condition, the simulation was started with speed 5 m/s. in this case the residuals scale were monitored and it showed that, the residuals have stagnated and do not changed with further iteration.

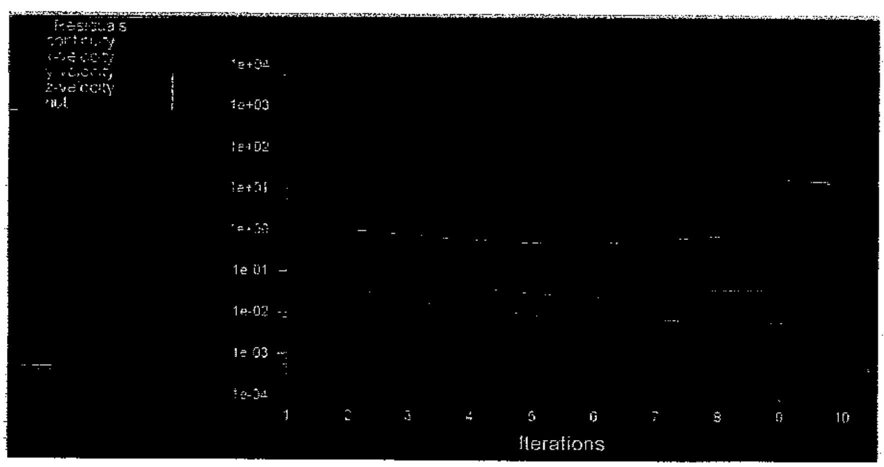

Figure 4.10: Residuals for the 10 iterations.

From the graph, the value of the Drag coefficient was not constant from 0 till 40 iterations but become constant from 50 until 90 iterations. The constant value means that the exact value for Drag coefficient.

Different value of speed will give different value for Drag coefficient. The same iteration was used for each speed and the value of Drag coefficient was used to calculate Drag force.

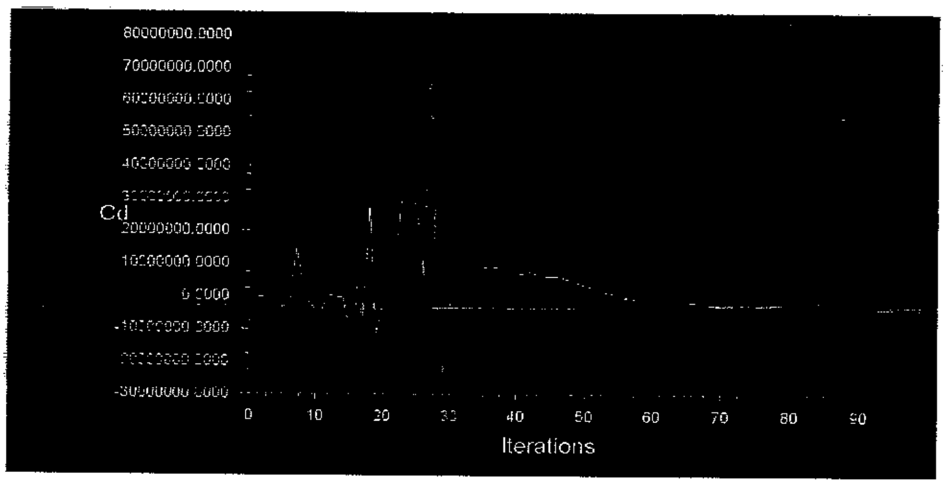

Figure 4.11: Convergence history for Drag coefficient

For double model with different separating distances, the same conditions such as the speed and steady state condition were used.

4.6 Single Model

The simulation was run on different velocity which is from 5 m/s with increment of 5 m/s until 30 m/s. By increasing this value, the drag and lift force can be obtained. Below are the equation that used to get value of drag force, lift force and Reynolds number.

$$F_D = 0.5 \rho V^2 A_f C_D \quad \text{.......... (4)}$$

$$F_L = 0.5 \rho V^2 A_p C_D \quad \text{.......... (5)}$$

$$\mathrm{Re} = \left[\frac{\rho V L}{\mu} \right] \quad \text{.......... (6)}$$

- A_f is the frontal area ($0.01476\,m^2$)
- ρ is the density of the air ($1.185\ kg/m^3$)
- V is the speed of the air
- C_D is drag coefficient
- μ is Kinematics viscosity ($1.572 \times 10^{-5}\ m^2/s$)
- A_P is the top area ($5.085 \times 10^{-3}\ m^2$)
- L is the width of the Car (12.5 cm)

Below are the results of the test for single model:

Table 4.1: Single Model

VELOCITY (m/s)	DRAG FORCE (N)	DRAG COEFFICIENT	LIFT FORCE (N)	LIFT COEFFICIENT	REYNOLDS NUMBER
5	10.5310903	48.168	1.954368582	25.947	4711.354962
10	16.2610108	18.594	3.169832636	10.521	9422.709924
15	20.1531066	10.242	3.941276079	5.814	14134.06489
20	24.1790054	6.912	4.522909185	3.753	18845.41985
25	25.0880794	4.59	4.626626977	2.457	23556.77481
30	26.0679902	3.312	4.88083725	1.8	28268.12977
35	29.1179136	2.718	5.248255832	1.422	32979.48473
40	32.1127416	2.295	5.4231525	1.125	37690.83969
45	34.9048973	1.971	5.490941906	0.9	42402.19466
50	37.3861575	1.71	5.694310125	0.756	47113.54962

The table showed the result for all the force that react on the model of the car like Drag force and Lift force. Based on this result several graphs were plotted to see the relationship between the forces.

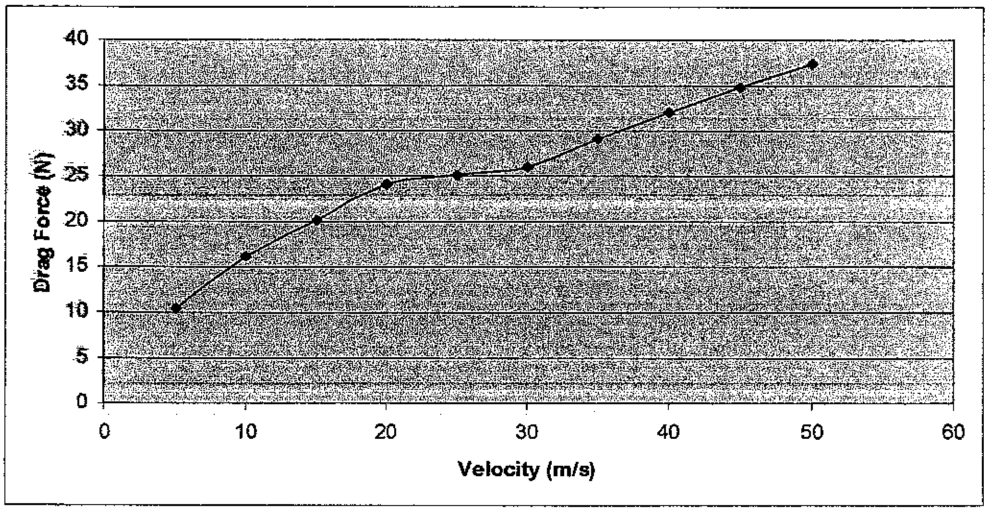

Figure 4.12: Drag force versus velocity

From the graph, the Drag forces showed directly proportional with velocity. The slope increased until maximum drag force which is 37.39 N at 50 m/s.

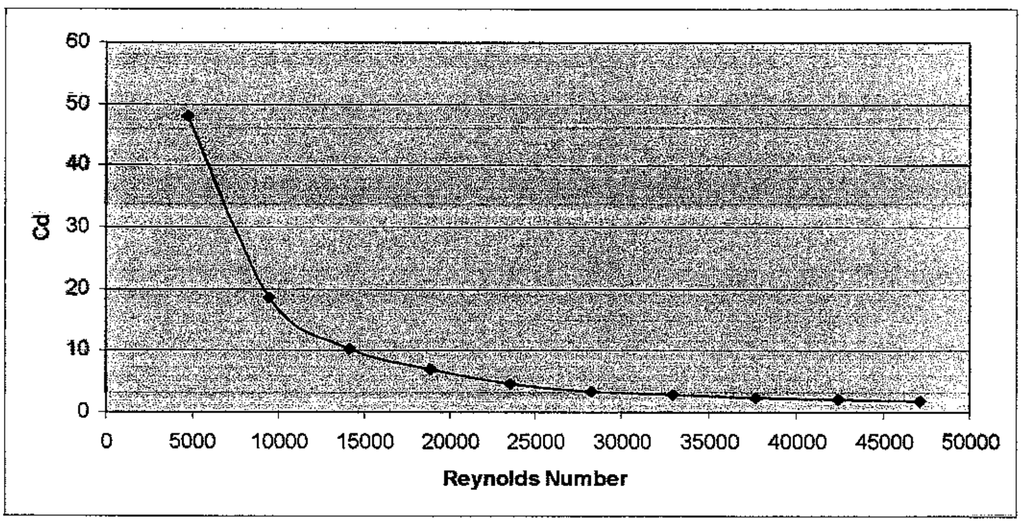

Figure 4.13: Drag coefficient versus Reynolds number

For the Drag coefficient, it showed that drag coefficient inversely proportional with Reynolds number. The slope decrease until certain constant of drag coefficient.

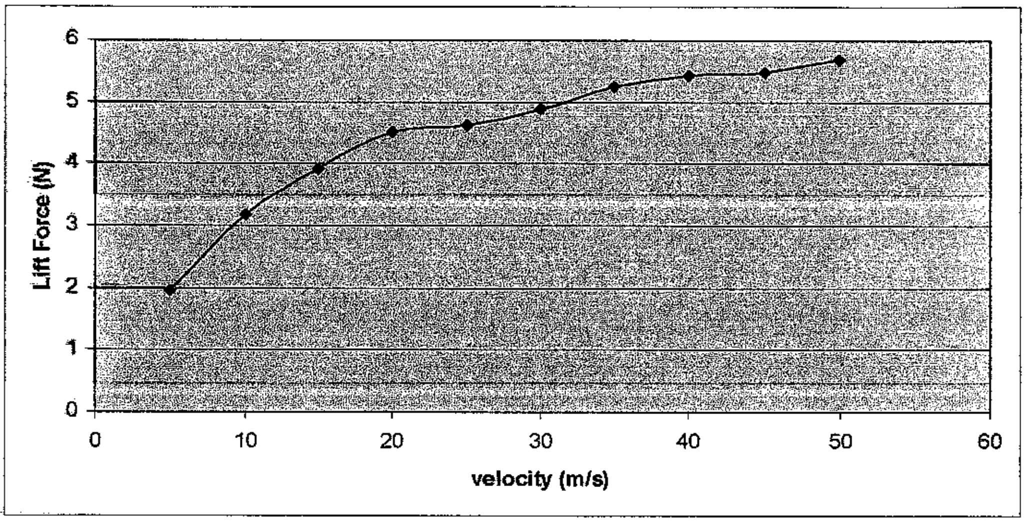

Figure 4.14: Lift force versus velocity

The value of lift force smaller compared to drag force which is 5.69 N when velocity of air is 50 m/s. Plotting Lift force against velocity showed the relationship of two variables which is directly proportional to each other.

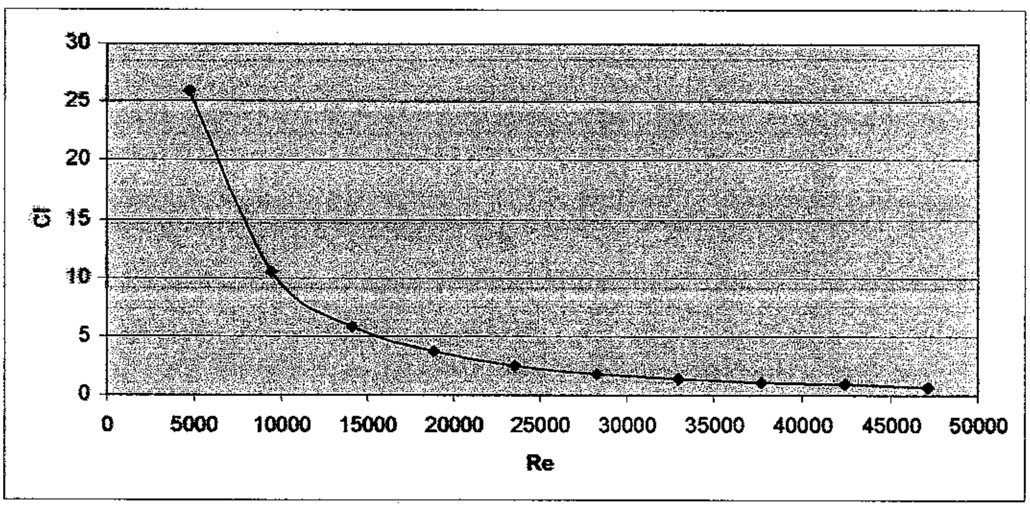

Figure 4.15: Lift coefficient versus Reynolds number

For lift coefficient versus velocity and Reynolds number, it showed that there are indirectly proportional each other. The Lift coefficient was decreased until certain constant value when Reynolds number increases.

4.7 Two Models

The distance between two cars in the simulation must be same with the distance that test in wind tunnel. In the wind tunnel test, the distance was set up according to certain value. Therefore the setups for computational simulation were:

- Half width distance (0.5W) between the models – approximately 6.25cm
- One width distance (1W) between the models – approximately 12.5cm
- One and Half width distance (1.5W) between the models – approximately 18.75cm
- Twice width distance (2W) between the models – approximately 25.0cm

Figure 4.16: Position of two cars

4.7.1 Half Width Distance (6.25cm)

For the first case, the distance was half width separating distance which is 6.25 cm. between two cars. Below was the result for this distance.

Table 4.2: Half Width Distance (6.25cm)

VELOCITY (m/s)	DRAG FORCE (N)	DRAG COEFFICIENT	LIFT FORCE (N)	LIFT COEFFICIENT	REYNOLDS NUMBER
5	0.491923125	2.25	0.648744618	8.613	4711.354962
10	0.88152624	1.008	3.430143956	11.385	1085.496183
15	1.540703228	0.783	10.35957706	15.282	1628.244275
20	1.7315694	0.495	20.75982777	17.226	2170.992366
25	2.262846375	0.414	37.06385787	19.683	2713.740458
30	3.89603115	0.495	62.59673773	23.085	3256.48855
35	5.39934822	0.504	85.30076567	23.112	3799.236641
40	7.30407456	0.522	111.2830893	23.085	4341.984733
45	8.447303903	0.477	148.7496162	24.381	4884.732824

The simulation was started with speed 5 m/s. the speed was increased until 45 m/s with increment 5m/s. The density, area of the car and viscosity were same with single model. Using the same equation, the drag force and lift force were calculated. The graph was plotted to see the relationship between drag force and lift force with velocity.

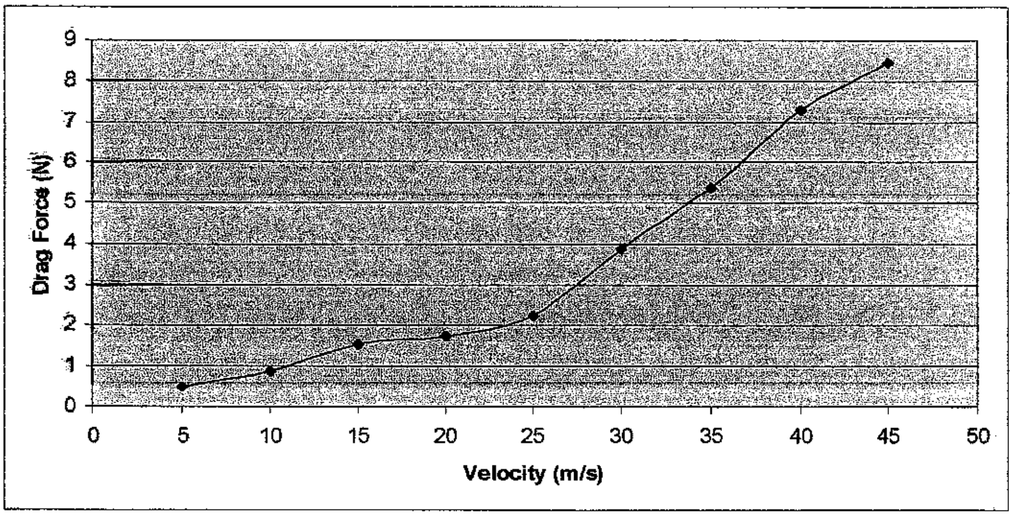

Figure 4.17: Drag Force versus velocity (0.5W)

It is observed from the graph that when velocity of air increase, the drag force increase as well. But in this case the drag force much smaller compare to single model. For the maximum speed which is 45 m/s, the drag force is 8.45N.

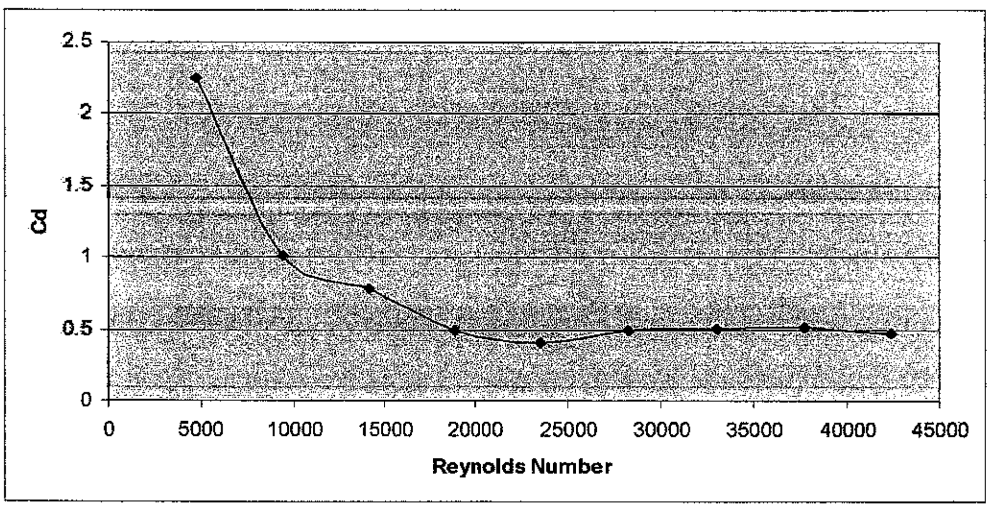

Figure 4.18: Drag coefficient versus Reynolds number (0.5W)

The Reynolds number still same for this case because the density and viscosity same as well. The drag coefficient was decreased until certain value when Reynolds number increased.

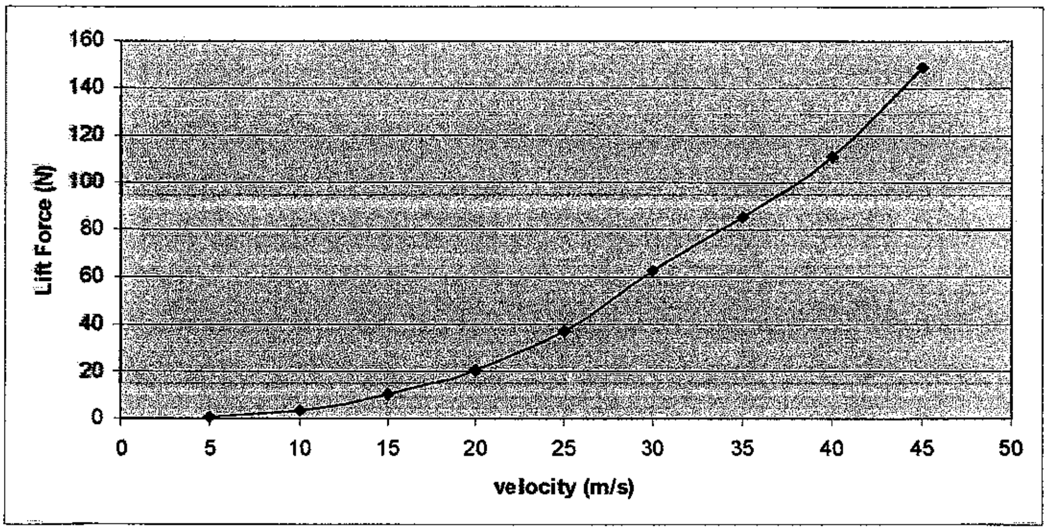

Figure 4.19: Lift force versus velocity (0.5W)

Plotting the lift force against velocity showed the relationship between two variables which is directly proportional to each other. The maximum value of the lift force is 148.75 N when the velocity is 45 m/s.

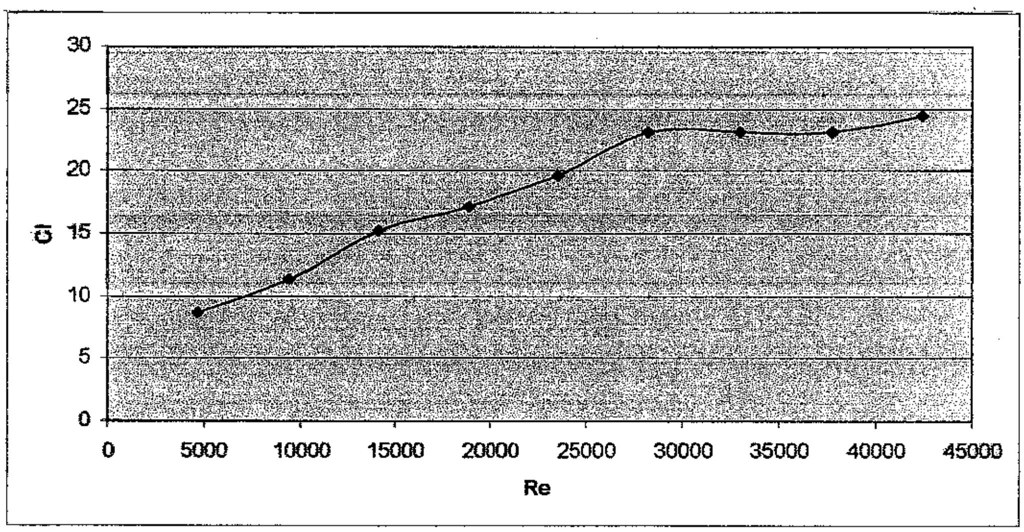

Figure 4.20: Lift coefficient versus Reynolds number (0.5W)

For Lift coefficient, the value is much lower than single model, but the relationship with velocity and Reynolds number still same as single model which is indirectly proportional to each other.

4.7.2 One Width Distance (12.5 cm)

The simulation continued with one width separating distance which is 12.5 cm between two cars. The density, viscosity and temperature still same as single model. Table below illustrate the drag and lift force that get from the calculation.

Table 4.3: One Width distance (12.5 cm)

VELOCITY (m/s)	DRAG FORCE (N)	DRAG COEFFICIENT	LIFT FORCE (N)	LIFT COEFFICIENT	REYNOLDS NUMBER
5	11.54641959	52.812	1.296811342	17.217	4711.354962
10	20.62928817	23.589	6.857576336	22.761	9422.709924
15	25.82006099	13.122	20.71305308	30.555	14134.06489
20	28.20883968	8.064	41.50880924	34.443	18845.41985
25	32.56531088	5.958	74.11076838	39.357	23556.77481
30	36.55185588	4.644	125.1934755	46.17	28268.12977
35	38.66318993	3.609	170.5683145	46.215	32979.48473
40	39.29088384	2.808	231.1998374	47.961	37690.83969
45	42.07713642	2.376	297.4443231	48.753	42402.19466
50	43.09246575	1.971	371.7571039	49.356	47113.54962

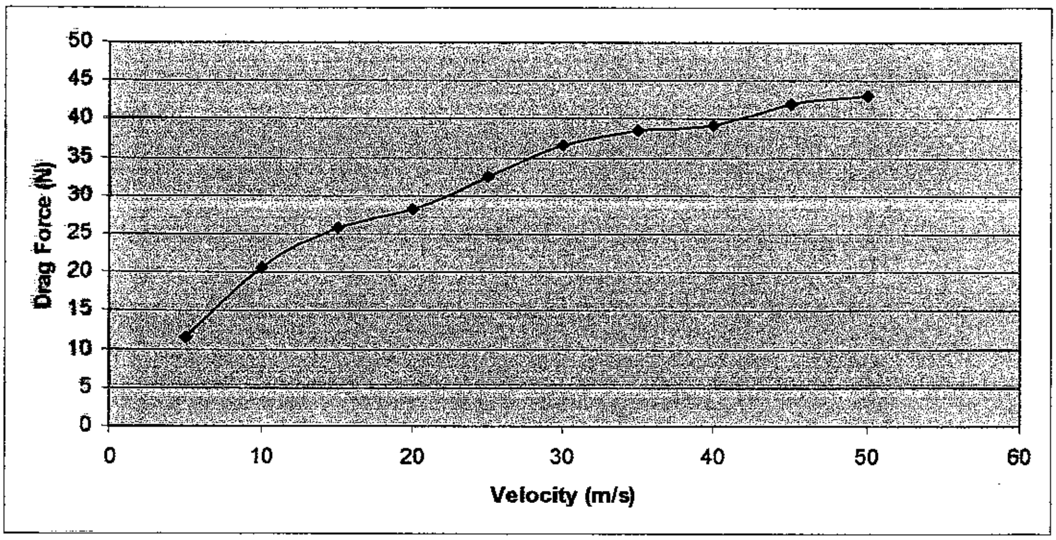

Figure 4.21: Drag force versus velocity (1W)

The speed of the air that flow in the simulation was started by 5 m/s. After that, the speed was increased by 5 m/s until up to 50 m/s. At the maximum speed which is 50 m/s, the drag force is 43.09 N. at this case, the drag value much higher compared to previous case.

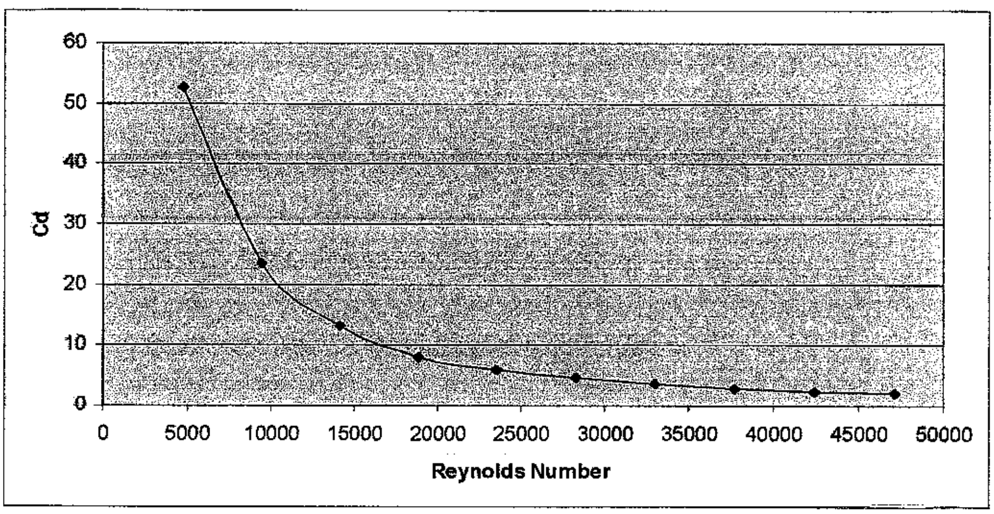

Figure 4.22: Drag coefficient versus Reynolds number (1W)

The relationship between drag coefficient and Reynolds number still same as previous case which is indirectly proportional to each other. The drag coefficient becomes smaller as Reynolds number decrease until certain constant value which is 1.97.

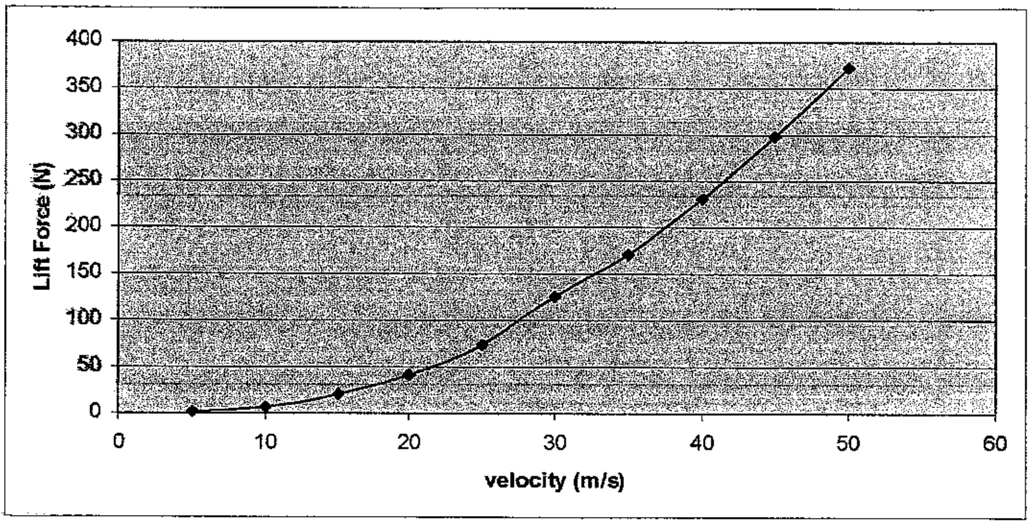

Figure 4.23: Lift force versus velocity (1W)

The lift force also showed same pattern as previous case but the value of lift force much higher. At 50 m/s, the lift force is 371.76 N.

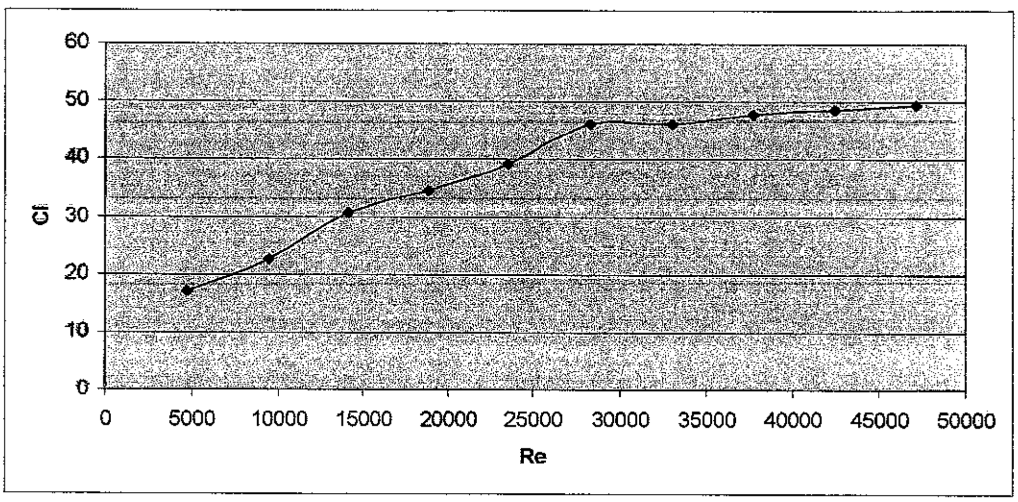

Figure 4.24: Lift coefficient versus Reynolds number (1W)

For the lift coefficient it showed different relationship with Reynolds number if compare with drag coefficient. Lift coefficient increase as Reynolds number increased.

4.7.3 One and Half Width Distance (18.75 cm)

For the third case, the distance was changed to one half width separating distance which is 18.75 cm.

Table 4.4: One and Half Width Distance (18.75 cm)

VELOCITY (m/s)	DRAG FORCE (N)	DRAG COEFFICIENT	LIFT FORCE (N)	LIFT COEFFICIENT	REYNOLDS NUMBER
5	3.31359417	15.156	1.054125267	13.995	4711.354962
10	5.63547132	6.444	6.212221189	20.619	9422.709924
15	7.68580691	3.906	18.52277736	27.324	14134.06489
20	9.88568712	2.826	37.85360445	31.41	18845.41985
25	11.7077704	2.142	67.09456484	35.631	23556.77481
30	13.2465059	1.683	108.8182665	40.131	28268.12977
35	15.4267092	1.44	153.4284411	41.571	32979.48473
40	16.119337	1.152	208.639523	43.281	37690.83969
45	17.3727571	0.981	270.6485266	44.361	42402.19466

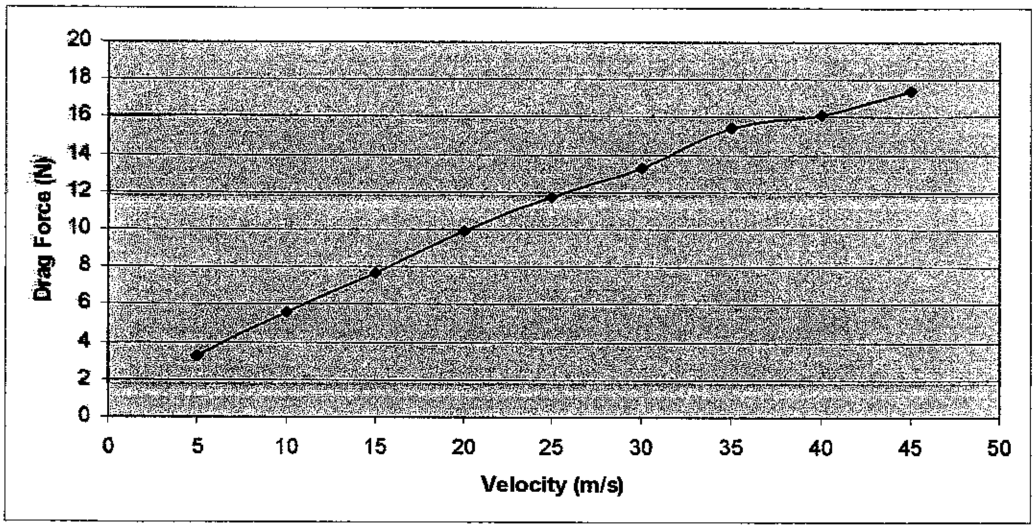

Figure 4.25: Drag force versus velocity (1.5W)

From the graph, it showed that when speed is 5 m/s, the drag force is 3.3 N. drag force increase as velocity increase until 17.37 N when speed is 45 m/s. for this case, the drag force much smaller compared to one width separating distance.

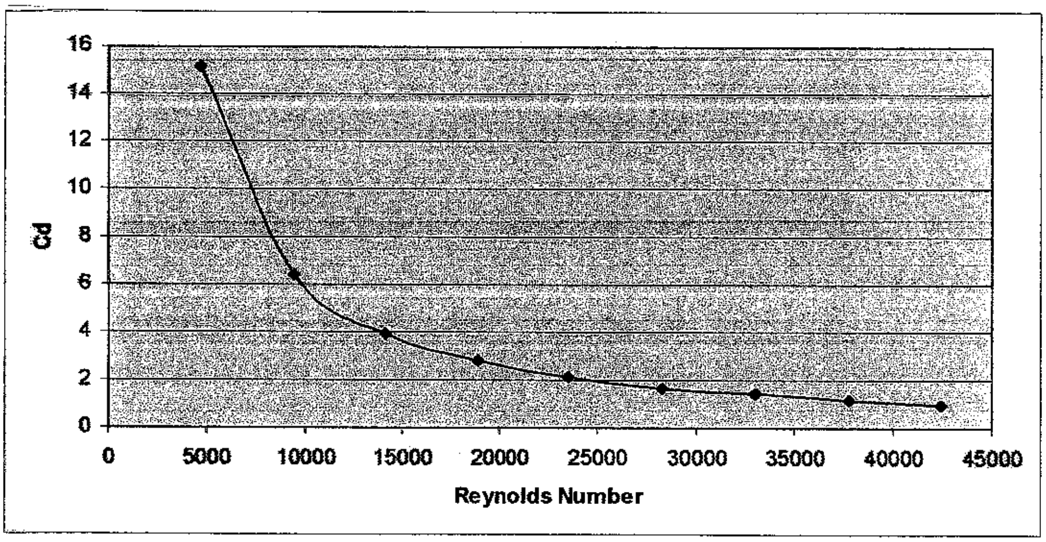

Figure 4.26: Drag coefficient against Reynolds number (1.5W)

Drag coefficient still show same relationship with Reynolds number which is indirectly proportional to each other. The slope decreased until certain value which is 0.96 when Reynolds number increases.

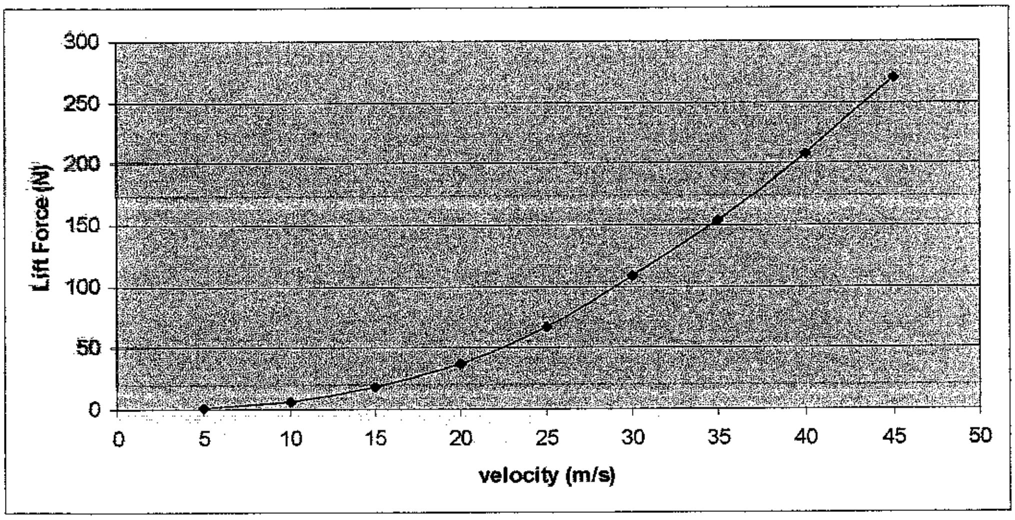

Figure 4.27: Lift force versus velocity (1.5W)

The value of lift force much smaller if compared to previous case. The maximum value of the lift force is 270.6 N when speed is 45 m/s.

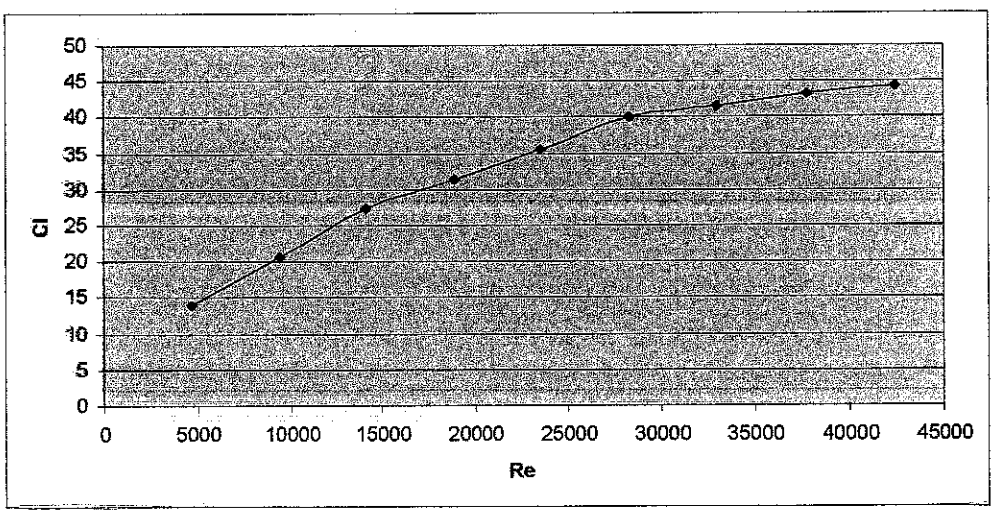

Figure 4.28: Lift coefficient versus Reynolds number (1.5W)

Lift coefficient will increase until certain value when Reynolds number increases. The maximum value for the lift coefficient is 44.36.

4.7.4 Two Width Distance (25cm)

For the last case, the simulation was done with different distance which was two width distance (25 cm). From this distance, the value of Drag force was decrease compare to first case. But, the relationship between velocity and Drag force still same which is directly proportional to each other.

Table 4.4: Two Width Distance (25 cm)

VELOCITY (m/s)	DRAG FORCE (N)	DRAG COEFFICIENT	LIFT FORCE (N)	LIFT COEFFICIENT	REYNOLDS NUMBER
5	7.62874382	34.893	1.523227958	20.223	4711.354962
10	13.789589	15.768	2.611247929	8.667	9422.709924
15	18.7894957	9.549	3.404383982	5.022	14134.06489
20	21.7862914	6.228	4.01313285	3.33	18845.41985
25	23.8582716	4.365	4.507995516	2.394	23556.77481
30	25.5721317	3.249	4.832028878	1.782	28268.12977
35	29.7928321	2.781	5.215039023	1.413	32979.48473
40	31.6090123	2.259	5.46653772	1.134	37690.83969
45	33.7892156	1.908	5.655670163	0.927	42402.19466
50	34.8281573	1.593	5.762099531	0.765	47113.54962

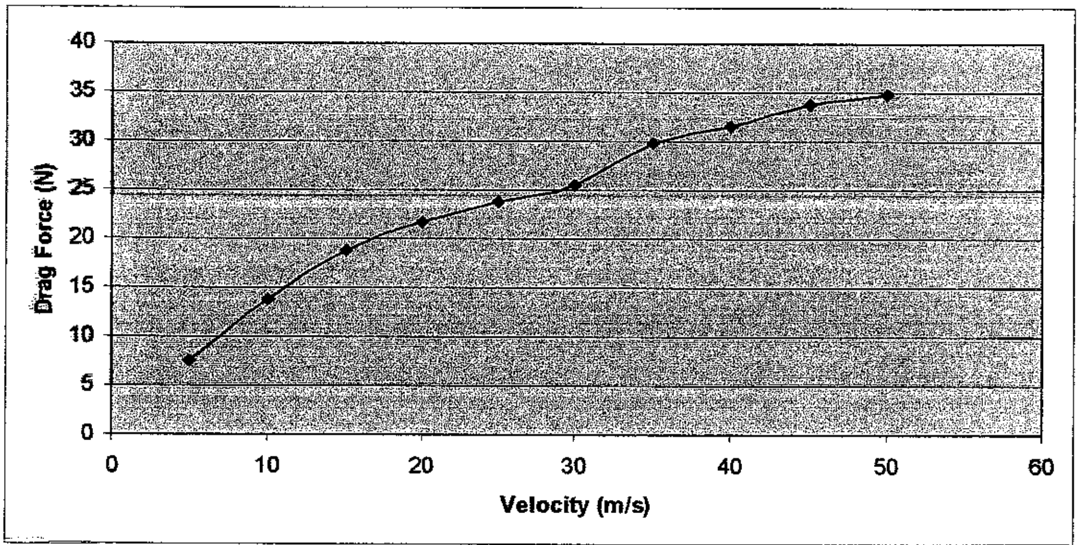

Figure 4.29: Drag force versus velocity (2W)

The graph showed the relationship between drag force and velocity which is directly proportional to each other. The drag force increase as velocity of air increased until maximum value which is 34.83 N at 50 m/s.

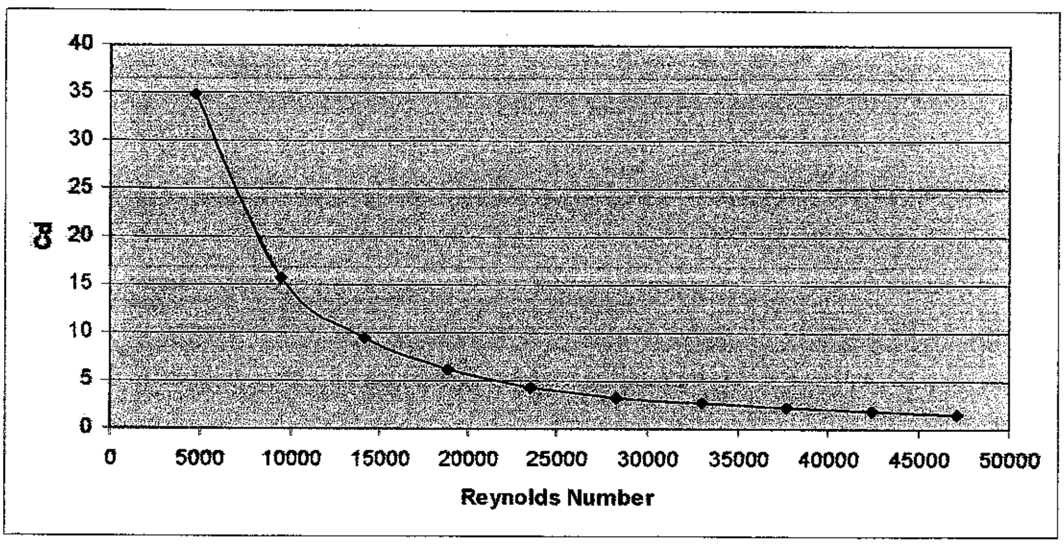

Figure 4.30: Drag coefficient versus Reynolds number (2W)

The value of Reynolds number still same as previous case. The slope decrease until certain value of drag coefficient which is 1.59.

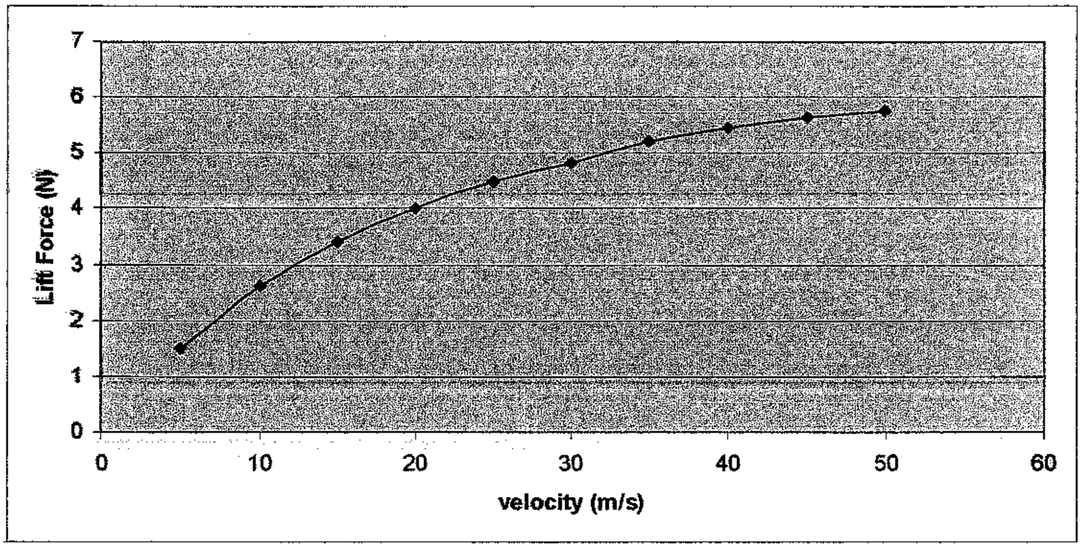

Figure 4.31: Lift force versus velocity (2W)

For the lift force versus velocity, it showed directly proportional to each other. Lift force increase higher when speed of air is 5 m/s to 20 m/s but the increment become lower from 30 m/s to 45 m/s. The maximum value for lift force is 5.76 N at 45 m/s.

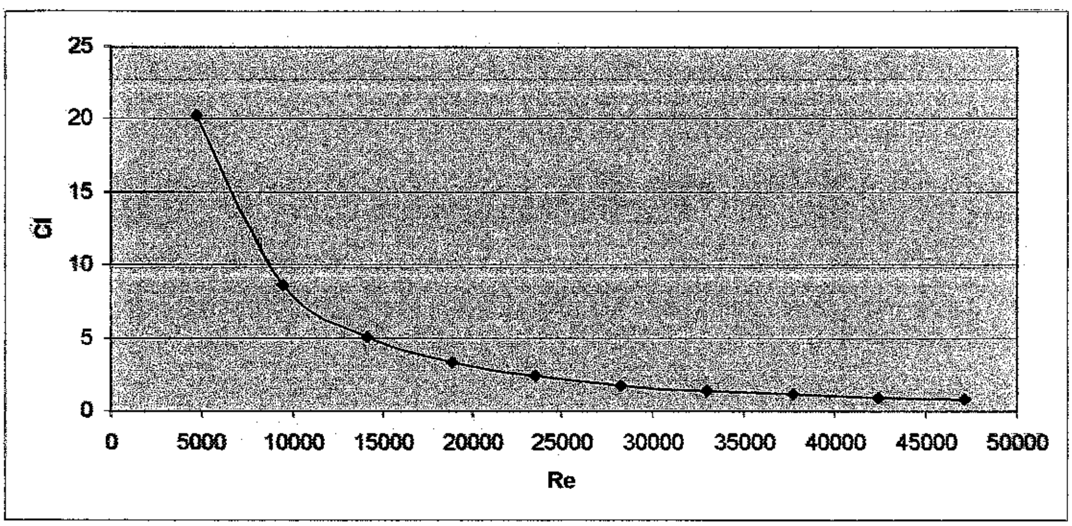

Figure 4.32: Lift coefficient versus Reynolds number (2W)

The slope was decreased until certain value of the lift coefficient which is 0.76. The relationship between lift coefficient and Reynolds number showed that indirectly proportional to each other.

4.8 Comparison between CFD and Experimental Result

After simulation for single and double models were done, the project was continued with comparing computational result with experimental result. The relationship between Drag force and Reynolds number and Drag coefficient with Reynolds number were compared. These comparisons will validate either both situation have same result or not.

4.8.1 Single Model

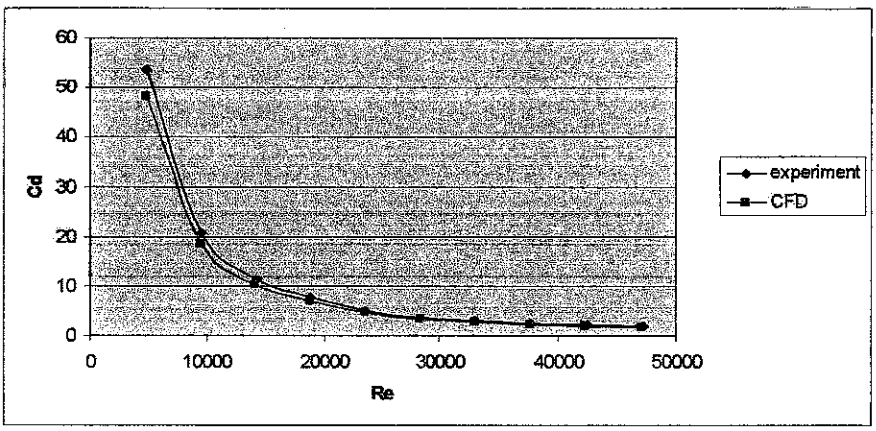

Figure 4.33: Comparison Drag coefficient versus Reynolds number for single model

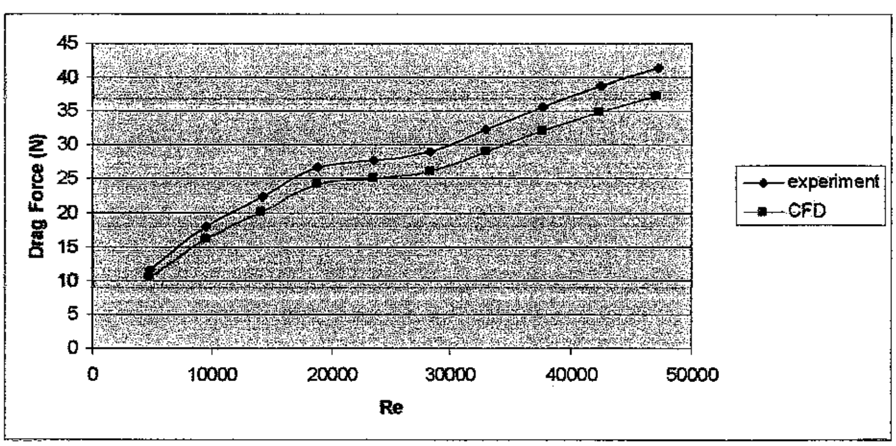

Figure 4.34: Comparison Drag force versus Reynolds number for single model

For single model, there are small different between experimental and CFD in term of Drag force and Drag Coefficient.

4.8.2 Half Width Distance (6.25 cm)

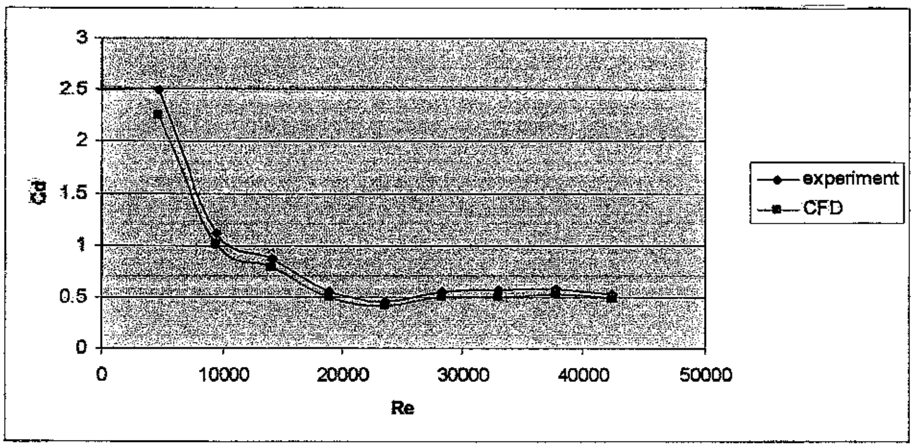

Figure 4.35: Comparison Drag coefficient versus Reynolds number (0.5W)

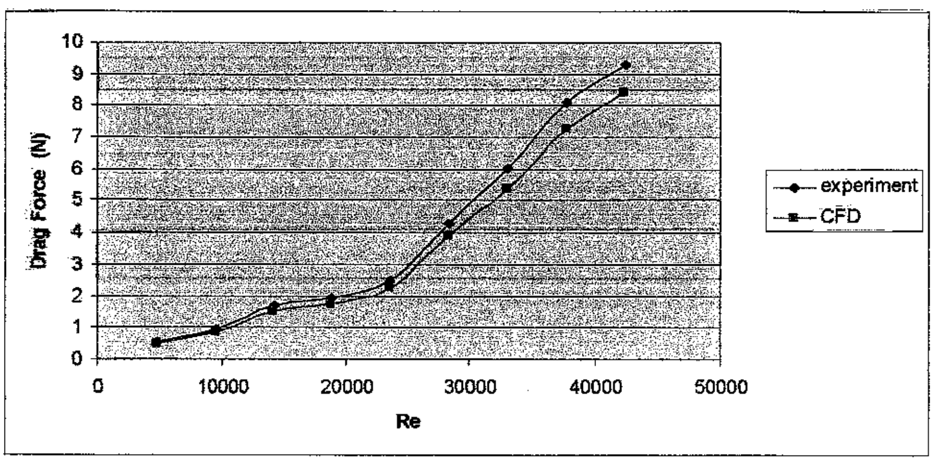

Figure 4.36: Comparison Drag force versus Reynolds number (0.5W)

For the second case which is half width separating distance, it showed that the drag force and drag coefficient almost same for both cases. From figure 4.56, the drag force has same value at early value of Reynolds number but experimental result has higher value compare to CFD result.

4.8.3 One Width Distance

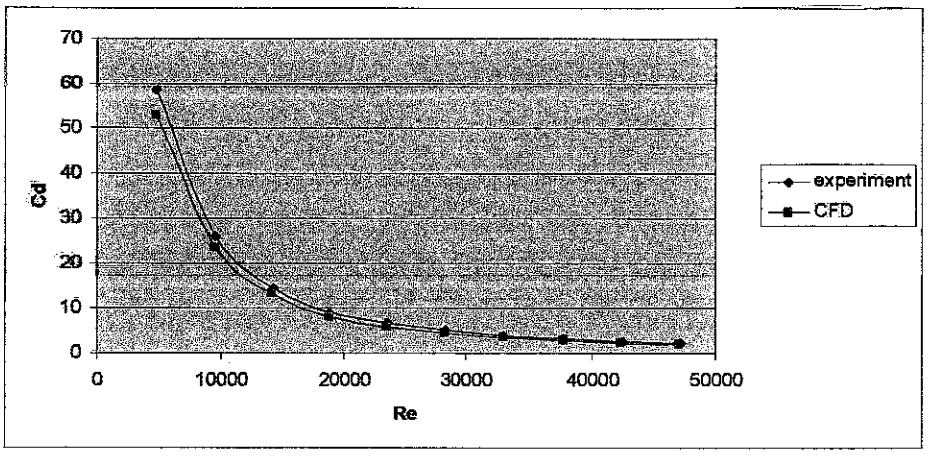

Figure 4.37: Comparison Drag coefficient versus Reynolds number (1W)

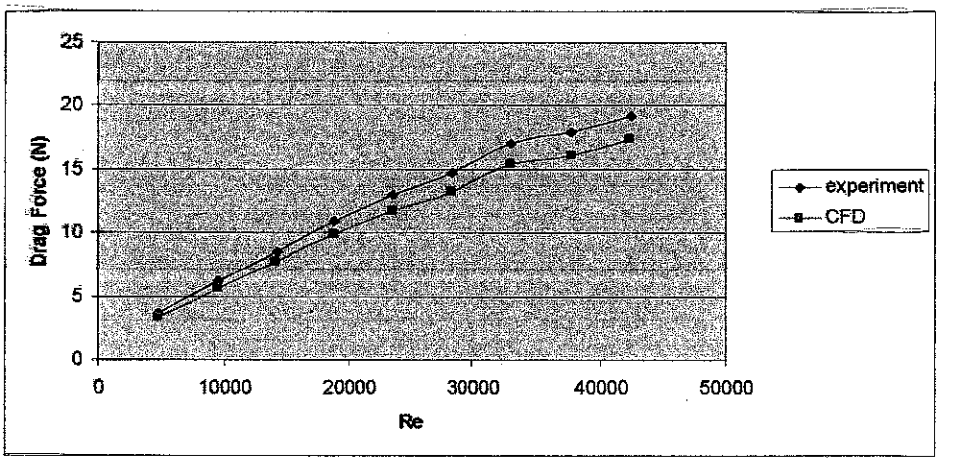

Figure 4.38: Comparison Drag force versus Reynolds number (1W)

From the graph, it showed that the value of drag force and drag coefficient in experimental have higher value compared to CFD result.

4.8.4 One and Half Width Distance

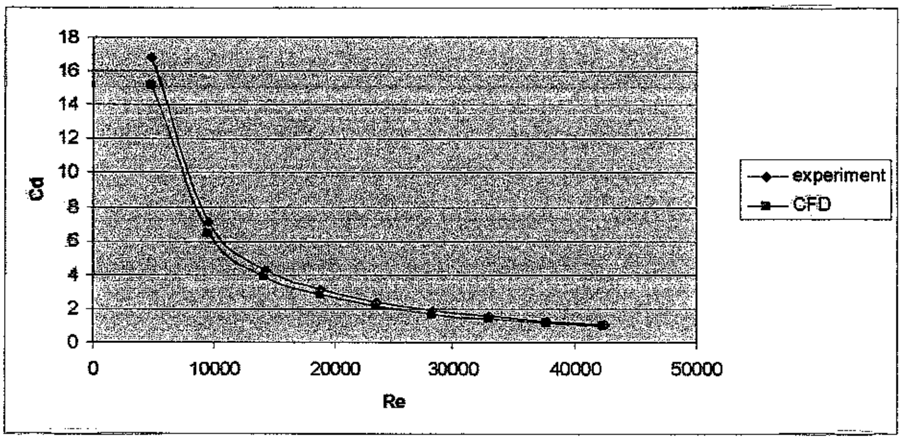

Figure 4.39: Comparison Drag coefficients versus Reynolds number (1.5W)

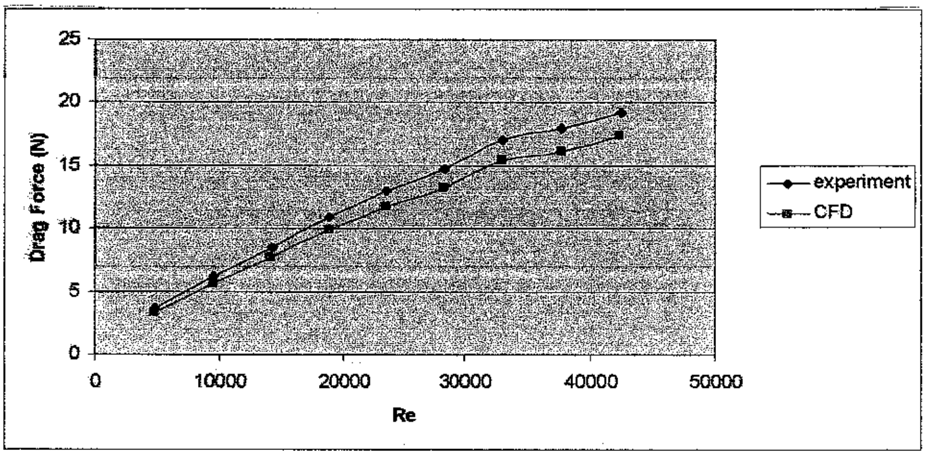

Figure 4.40: Comparison Drag force versus Reynolds number (1.5W)

For this case, experimental result still has higher value for drag force and drag coefficient. The maximum drag force for experimental is 18.45 N while CFD is 17.37 m/s.

4.8.5 Two Width Distance

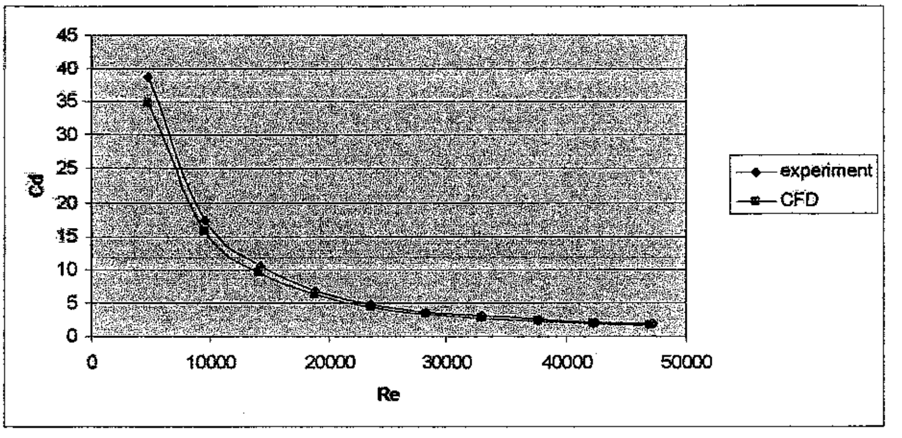

Figure 4.41: Comparison Drag coefficient versus Reynolds number (2W)

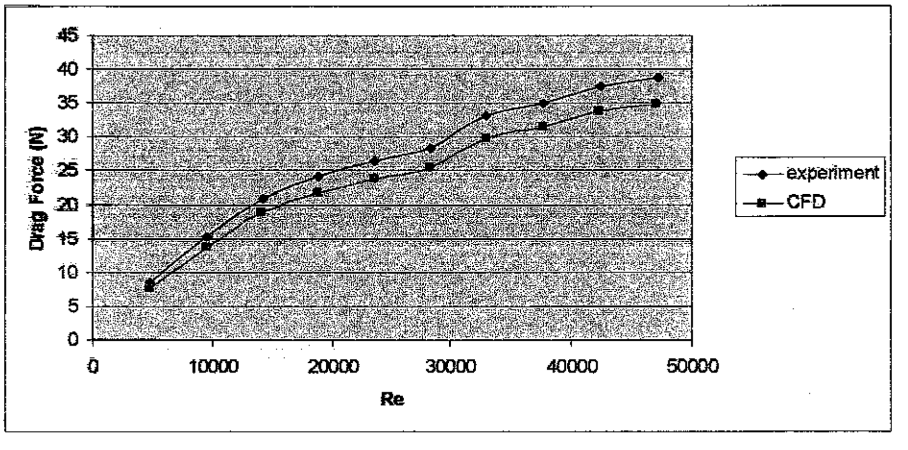

Figure 4.42: Comparison Drag force versus Reynolds number (2W)

For the last case which is two width distances, the comparison showed that the value of the drag force and drag coefficient in CFD are higher than experimental same as previous case. From this figure, it showed that the results in CFD are almost similar to experimental result.

This project was continued with comparison the relationship between Drag force with velocity and relationship between Drag coefficients with Reynolds number for all cases. Figure below illustrate the different value of drag force for each case.

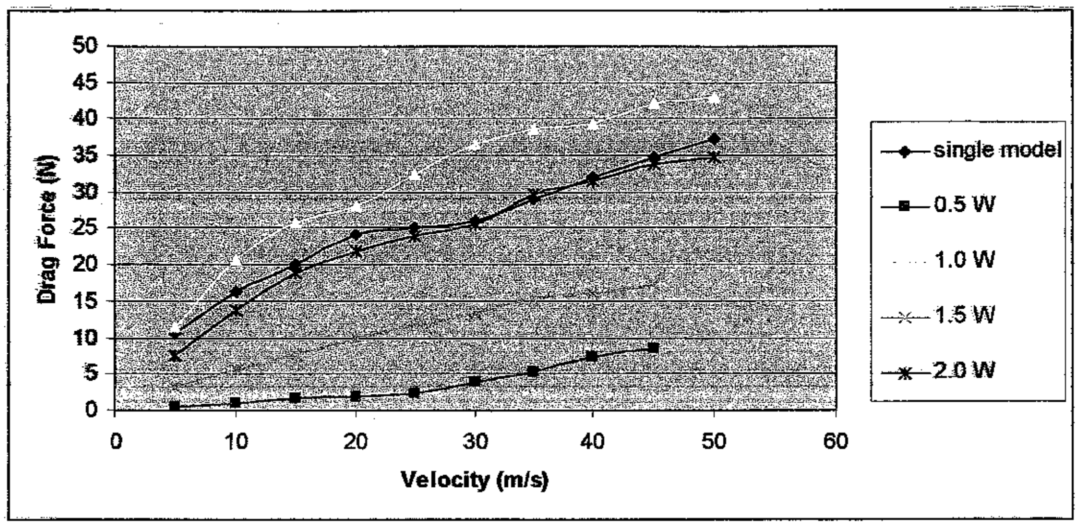

Figure 4.43: Drag Force versus Velocity for all cases

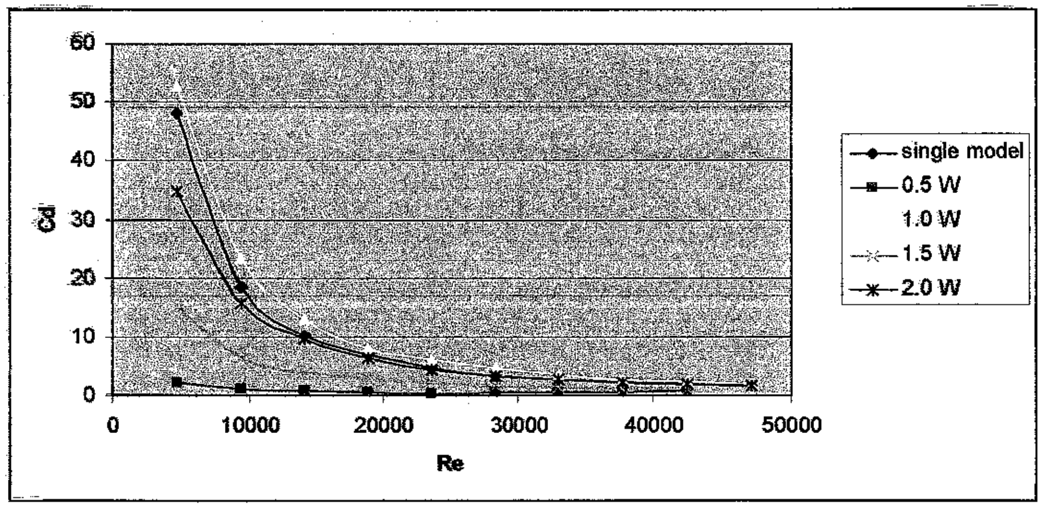

Figure 4.44: Drag coefficient versus Reynolds number for all cases

For half width distances, the second car was very near and directly behind the first car. The Drag force that reacts on second car was lesser because they were considered as one body. Thus the flow continues until the back of second car and as a result almost no Drag force in front of second car.

For the second case which is one width distance, it is observed that the Drag force increase compared to single model. The front car created a turbulent flow that is directly in front of second car which was located at the behind. This turbulent flow from front car has no time to steady down due to short distance and short in different time and resulting higher Drag force when it hit second car

The third case was one and half width distance. This distance basically located between first cases (12.5 cm) and second cases (25 cm).From this setup, the back pressure that caused by second car was caused Drag force to be low.

For the last case which is two width distances, the flow of air hit the first car and turbulent was created at the back of the car. The larger distance gives the flow time to steady itself and become less turbulent. As a result, when it hit second car, the Drag force that acting on the car is lesser. This reading shows that this setup almost the same as single model setup, almost as if there was no second car.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

This project basically wants to validate experimental result with simulation result using Computer Fluid Dynamic (CFD). The model that be used in the simulation was same with the model in the real wind tunnel experiment. The entire dimension on the real model was measured as a reference for the computer model. For single model it found that increasing in speed will give effect on Drag force. The graph show that Drag force and velocity are directly proportional each other.

For double model, different distances between models give different Drag Force. The best position for aerodynamically which is less Drag force was directly behind the other car which at haft width distance (6.25 cm).The Drag force that reacts on second car was lesser because they were considered as one body. In case of safety situation, driving at this position will be dangerous if front car suddenly break, resulting crash from tailing car.

At the one width distance (12.5 cm), it showed higher Drag force leading to harder car control. Increasing Drag force due to aerodynamic changed caused by front car. The turbulent flows that create from front car give more Drag force on back car. This will caused driver to loose control and can caused accident.

For the one half width distance (18.75 cm), the Drag force much less and the distance is safe enough to tailing front car. So the best position for safety and for better aerodynamic condition will be at this distance. From this setup, the back pressure that caused by second car was caused Drag force to be low. At two width distance (25 cm), the Drag force was same as there is no car in front.

5.2 Recommendation

The recommendation for this project is simulation should get the result as same as real experiment for wind tunnel or better than that. If the result is same, the simulation will be continuing with the big scale model which is real size of the car. For further study of this project the distance need to be changed with specific distance to look the effect of this change to the car.

REFERENCES

[1]. C. Noger, C. Regardin, E. Széchényi, 2005, Investigation of the transient Aerodynamic phenomena associated with passing manoeuvres, Journal of Fluids and Structures, 21 (2005) 231-241.

[2]. Muhammad Nazri Bin Dzulkifli,Aerodynamic Changes When Two Vehicles Are in Close Proximity Each Other, Final Year Project 2008, Universiti Teknologi Petronas.

[3]. V. Sumatran, G. Sovron, 1996, Vehicle Aerodynamics; PT-49 SAE International.

[4]. Y. A. Cengel, J. M. Cimbala, 2006, Fluid Mechanics Fundamentals and Applications, Mc Graw Hill.

[5]. <http:/www.cfd-online.com/Wiki/Meshing,9 February 09>

[6]. <http:/ www.engres.odu.edu/Applications/fluent6.2,8 July 08>

[7]. <http:/www.fluent.com,13 September 08>

[8]. <http:/www.mueller-hp.com/windtunnel.htm,4 Jan 09>

APPENDICES

APPENDIX A: Wind Tunnel

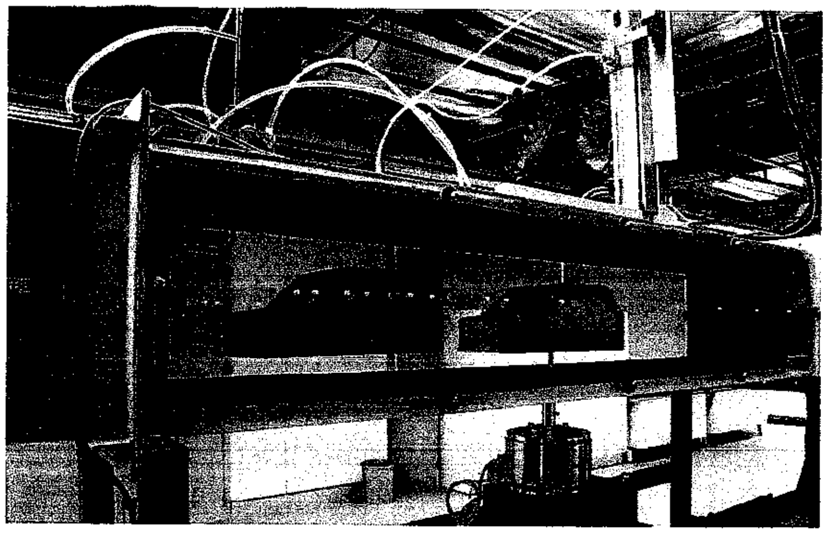

Figure A1: Real Wind Tunnel

Figure A2: Computer Fluid Dynamic Model