

ADAMA SCIENCE AND TECHNOLOGY UNIVERSITY

School of Mechanical Chemical and Materials Engineering



Study of Aerodynamic Drag Reduction of Locally Built Bus Body Using CFD Analysis: A Case Study on Bishoftu Automotive Industry

*A thesis submitted in partial fulfillment of the requirements for the award of the
degree of*

Master of Science

In

Automotive Engineering

By

Lealem Tilahun Tassew

Mechanical System and Vehicle Engineering Program

June, 2018

Adama, Ethiopia

ADAMA SCIENCE AND TECHNOLOGY UNIVERSITY

School of Mechanical Chemical and Materials Engineering

**Study and Aerodynamic Drag Reduction on Locally Built Bus Body
Using CFD Analysis.**

A thesis submitted to Adama Science and Technology University in partial
fulfillment of the requirements for the award of the degree of

Master of Science

In

Automotive Engineering

By

Lealem Tilahun Tassew

Advisor: Mekonen Liben (Asst. Prof.)

Mechanical System and Vehicle Engineering Program

June, 2018

Adama, Ethiopia

Study and Aerodynamic Drag Reduction on Locally Built Bus Body Using CFD Analysis.

By

Lealem Tilahun Tassew

APPROVED BY THE EXAMINERS BOARD

----- Chairman	----- Signature	----- Date
----- Advisor	----- Signature	----- Date
----- Internal Examiner	----- Signature	----- Date
----- External Examiner	----- Signature	----- Date

CANDIDATE'S DECLARATION

I hereby declare that the work which is being presented in the thesis entitled “**Study and Aerodynamic Drag Reduction on Locally Built Bus Body Using CFD Analysis**” in partial fulfillment of the requirements for the award of the degree of Master of Science in Automotive Engineering is an authentic record of my own work carried out from October 2017 to June 2018 under the supervision of Mekonen Liben (Asst. Prof.) Department of Mechanical system and Vehicle Engineering, Adama Science and Technology University, Ethiopia.

The matter embodied in this thesis has not been submitted by me for the award of any other degree or diploma. All relevant resources of information used in this thesis have been duly acknowledged.

Name	Signature	Date
------	-----------	------

Lealem Tilahun	-----	-----
----------------	-------	-------

Student

This is to certify that the above statement made by the candidate is correct to the best of my knowledge and belief. This thesis has been submitted for examination with my approval.

Name	Signature	Date
------	-----------	------

Mekonen Liben (Asst. Prof)	-----	-----
----------------------------	-------	-------

Advisor

ACKNOWLEDGEMENT

First of all, I would like to thank the almighty God, for giving me the knowledge and wisdom and providing me the resources to acquire education.

I would like to express my sincere gratitude for my research supervisor; Mekonen Liben (Ass. Professor) for his excellent guidance, moral support and constructive criticism.

I also want to thank my parents and siblings without their support and guidance I would not have been able to achieve what I have so far in life. And also thank for my all friends, teacher and all society support me by any things.

Finally, I would like to thank, Adama science and Technology University for sponsored us this chance to learn MSc program.

TABLE OF CONTENTS

ACKNOWLEDGEMENT	i
LIST OF FIGURE	vi
LIST OF TABLE	x
NOMENCLATURE	xi
ABSTRACT	xiii
CHAPTER ONE	1
INTRODUCTION	1
1.1. Back ground of the study	1
1.2. Statement of the problem	4
1.3. Objective of the study	4
1.3.1. General objective of the study	4
1.3.2. Specific objectives	5
1.4. Scope of the study	5
1.5. Significance of the study	5
1.6. Structure of the report	5
CHAPTER TWO	6
LITERATURE REVIEW	6
2.1. Previous investigations.....	6
2.2. Bus aerodynamics	8
2.2.1. Flow field around a bus	8
2.2.2. Laminar and turbulent flow	10
2.3. Drag.....	10
2.4. Lift.....	11
2.5. Boundary layer	11
2.6. Bluff and streamlined bodies.....	12
2.7. Impact of aerodynamic drag on fuel consumption.....	15
2.8. Flow separation	16
2.9. Flow control	16
2.9.1. Passive flow control.....	16
2.9.1.1. Effect of VG on aerodynamic drag.....	16

2.9.1.2. Effect of spoiler on aerodynamic drag.....	17
2.9.1.3. Others add-on devices.....	17
2.9.1.4. Design optimization for drag reduction	18
2.9.1.5. Couple flow control methods.....	19
2.9.2. Active flow control.....	19
2.9.2.1. Active flow control with air jets	20
2.9.2.2. Active flow control with steady blowing.....	20
2.9.2.3. Active flow control with suction.....	20
2.10. Airvane as a drag reduction device	20
2.11. Simplified model.....	23
2.12. Computational fluid dynamics (CFD).....	25
2.13. Experiment on wind tunnel	26
2.13.1. Types of wind tunnel	26
2.14. Modeling	28
2.14.1. Governing equation	28
2.14.2. Reynold Averaged Navier Stoke equation	29
2.14.3. Turbulence modeling.....	32
2.14.4. Modeling terminology	33
2.14.5. Two-equation turbulence models	34
2.14.5.1. Shear stress transport k- ω model (SST).....	37
2.14.5.2. Realizable k- ϵ turbulence model (RKE)	39
2.15. Summary of literature.....	40
CHAPTER THREE	41
MATERIAL AND METHODOLOGY.....	41
3.1. Material	41
3.2. Methodology	41
3.2.1. Data collection.....	41
3.2.2. Bench marking the baseline model.....	42
3.2.2.1. SolidWorks Scaled-Down Model	43
3.2.3. Using aerodynamic drag redaction device.....	43
3.2.3.1. Parameter study to find the optimal value.	44

3.2.4. Turbulence modeling	46
3.2.5. CFD/ANSYS Fluent set up and procedure	47
3.2.5.1. Geometry generation.....	47
3.2.5.2. Meshing.....	48
3.2.5.3. Fluent setup and simulation procedure	49
3.2.6. Calibration of fluent simulations	50
CHAPTER FOUR.....	51
RESULTS AND DISCUSSION	51
4.1. Result and discussion for model one.....	51
4.1.1. Result and discussion of scaled residual, drag and lift coefficient, pressure and velocity countour at speed of 80kmph 100kmph and 120kmph for model one.	51
4.1.2. Grid independent test.....	60
4.1.3. Validation of the result	61
4.2. Result and discussion for model two	65
4.2.1. Result of scaled residual, drag and lift coefficient, pressure and velocity countour at speed of 80kmph 100kmph and 120kmph for model two.	65
4.3. Result of model three.	74
4.3.1. Result of scaled residual, drag and lift coefficient, pressure and velocity countour at speed of 80kmph 100kmph and 120kmph for model three.	74
4.4. Result and discussion for model four.....	82
4.4.1. Result of scaled residual, drag and lift coefficient, pressure and velocity countour at speed of 80kmph 100kmph and 120kmph for model four.	82
4.5. Calculation of fuel consumption	92
4.6. Calculation of reduction of fuel exhaust emission.....	93
4.7. Comparison of this work with YisfalgNiguse.....	93
CHAPTER FIVE	95
CONCLUSION AND RECOMMENDATION	95
5.1. Conclusion.....	95
5.2. Recommendation.....	96
5.3. Future work	96
REFERENCE	97
APPENDIX.....	102

APPENDIX A	102
APPENDIX B	105
APPENDIX C	114
APPENDIX D	121

LIST OF FIGURE

Figure 1.1: A Scania truck used as transportation .	1
Figure 1.2: Example of bus in today [huanghai dd serious bus]	2
Figure 1.3: Relation of drag and rolling resistance relative to the speed	3
Figure 2.1: Aerodynamic load on the bus	8
Figure 2.2: Characteristics of Flow around a bluff body (a) far from ground (b) constrained by ground	9
Figure 2.3: Description of laminar and turbulent boundary layer	12
Figure 2.4: Streamlined and bluff body Vs drag coefficient	13
Figure 2.5: Boundary layer separation	14
Figure 2.6: 3D: Bus model with rear spoiler	17
Figure 2.7: Schematic with lateral guide vanes installed for (above) the box and (below) the SUV models	18
Figure 2.8: Airvane kit created by Kirsch with important aspects numbered	21
Figure 2.9: Simplified model geometry	23
Figure 2.10: The original Ahmed body. Top view and side view for the standard (30°)slant angle as presented by Ahmed S.R. and Ramm G.	24
Figure 2.11 : Open circuit wind tunnel layout	27
Figure 2.12: Closed circuit wind tunnel layout	27
Figure 2.13: Mass flows in and out fluid element	28
Figure 2.14: Stress component on three face of fluid element	29
Figure 2.15: Relationship between time-averaged variables, fluctuations and average	30
Figure 3.1: Existing luxury mid bus (baseline) [Bishoftu Automotive Industry]	42
Figure 3.2: Solid Works 1/30 th CAD model one.	43
Figure 3.3: Model two.	45
Figure 3.4: Model three.	45
Figure 3.5: Model four.	46
Figure 3. 6: Bus model in the enclosure box	47
Figure 3.7: Polyhedral mesh created in ANSYS Fluent.	49
Figure 4. 1: Scaled residuals for model one speed of 80 kmph.	51
Figure 4.2: Drag coefficient for model one at 80kmph.	51
Figure 4.3: Drag coefficient for model one at 100kmph	52
Figure 4.4: Drag coefficient for model one at 120kmph.	52
Figure 4.5: Lift coefficient for model one at 80kmph.	53
Figure 4.6: Lift coefficient for model one at 100kmph	53
Figure 4.7: Lift coefficient for model one at 120kmph.	54
Figure 4.8: Drag and lift vs speed of the vehicle	55
Figure 4.9:Aerodynamic pressure and velocity countour for model one at 80kmph.	56
Figure 4.10: Graph of pressure and velocity distribution for model one at 100kmph.	56
Figure 4.11: Graph of pressure and velocity distribution for model one at 120kmph.	57

Figure 4.12: Velocity contour for model one on a streamline.	58
Figure 4.13: Pressure contour of model one at 100 kmph	59
Figure 4. 14: Relation of total pressure with speed of the vehicle for model one.	59
Figure 4.15: Vortex generated at the rear of the bus model.....	60
Figure 4.16: Compression of graph of flow field around the bus from literature and calibration of this simulation.....	61
Figure 4.17: Ahmed body solid work model.	62
Figure 4.18: Polyhedral mesh generate on Ahmed body	63
Figure 4.19: Total pressure on the body	63
Figure 4.20: Calibration of drag coefficient on the body.....	64
Figure 4.21: Calibration of lift coefficient on the body	64
Figure 4.22: Drag coefficient for model two at 80kmph.	65
Figure 4.23: Drag coefficient for model two at 100kmph.	65
Figure 4.24: Drag coefficient for model two at 120kmph	66
Figure 4.25: Lift coefficient for model two at 80kmph.	66
Figure 4.26: Lift coefficient for model two at 100kmph.	67
Figure 4.27: Lift coefficient for model two at 120kmph.	67
Figure 4.28: Drag coefficient for model one and model two at three different speed.	68
Figure 4.29: Lift coefficient for model one and two vs speed.	68
Figure 4.30:Aerodynamic pressure countour for model two at 80kmph.	69
Figure 4.31:Velocity countour for model two at 80kmph	70
Figure 4.32: Graph of pressure distribution on model two at100kmph.	70
Figure 4.33: Graph of velocity distribution of model two at 100kmph.	71
Figure 4.34: graph of pressure distribution for model two at 120kmph.	71
Figure 4.35: Graph of velocity distribution of model two at 120kmph.	72
Figure 4.36: Graph of vortex generated for model two at the rear end of the model.	72
Figure 4.37: Graph of total pressure vs speed of the vehicle for model one and model two.....	73
Figure 4.38: Drag coefficient for model three at 80kmph.	74
Figure 4.39: Drag coefficient for model three at 100kmph.	74
Figure 4.40: Drag coefficient for model three at 120kmph.	75
Figure 4.41: Lift coefficient for model three at 80kmph.	75
Figure 4.42: Lift coefficient for model three at 100kmph.	76
Figure 4.43: Lift coefficient for model three at 120kmph.	76
Figure 4.44: Graph of drag coefficient for model one and model three vs speed of the vehicles	77
Figure 4.45: Graph of lift coefficient for model one and model three vs speed.	77
Figure 4.46:Aerodynamic pressure contour for model three at 80kmph.	78
Figure 4.47:Velocity countour for model three at 80kmph.	78
Figure 4.48: graph of pressure distribution for model three at 100kmph.	79
Figure 4.49: Graph of velocity distribution of model three at 100kmph.	79
Figure 4.50: Graph of pressure distribution for model three at 120kmph.	80

Figure 4.51: Graph of velocity distribution of model three at 120kmph.	80
Figure 4.52: Graph of total pressure vs speed of bus model one and three.	81
Figure 4.53: Drag coefficient for model four at 80kmph.....	82
Figure 4.54: Drag coefficient for model four at 100kmph.....	82
Figure 4.55: Drag coefficient for model four at 120kmph.....	83
Figure 4.56: Lift coefficient for model four at 80kmph.....	83
Figure 4.57: Lift coefficient for model four at 100kmph.....	84
Figure 4.58: Lift coefficient for model four at 120kmph.....	84
Figure 4.59: New bus design concept for model four.....	85
Figure 4.60: Drag coefficient vs speed for model one and model four.	85
Figure 4.61: Graph of lift coefficient vs speed for model one and four.	86
Figure 4.62: Aerodynamic pressure contour for model four at 80kmph.	86
Figure 4.63: Velocity countour for model four at 80kmph.....	87
Figure 4.64: Graph of pressure distribution for model four at 100kmph.....	87
Figure 4.65: Graph of velocity distribution of model four at 100kmph.	88
Figure 4.66: Graph of pressure distribution for model four at 120kmph.....	88
Figure 4.67: Graph of velocity distribution of model four at 120kmph.	89
Figure 4.68: Graph of total pressure vs speed for model one and model four.....	89
Figure 4.69: Graph of drag coefficient vs speed for different model	90
Figure 4.70: Graph of lift coefficient vs speed for different model.	91
Figure 4.71: Graph of total pressure vs speed for different model.	92
Figure A.1: Model one scaled down model view.	102
Figure A.2: Scaled down bus model two.....	102
Figure A.3: Scaled down bus model three.	103
Figure A.4: Scaled down bus model four.	103
Figure A.5: Airvane add on device used as drag reduction device.....	104
Figure B.1: Model one in the enclosure box with converted polyhedral mesh.	105
Figure B.2: Model two when meshing using converted polyhedral mesh.....	107
Figure B.3: Model three when meshing using converted polyhedral mesh.....	109
Figure B.4: Model four when meshing using converted polyhedral mesh.	111
Figure C.1: Total pressure for model one at 80kmph	114
Figure C.2: Total pressure for model one at 100kmph	114
Figure C.3: Total pressure for model one at 120kmph.	115
Figure C.4: Total pressure for model two at 80kmph.....	115
Figure C.5: Total pressure for model two at 100kmph.....	116
Figure C.6: Total pressure for model two at 120kmph.....	116
Figure C.7: Total pressure for model three at 80kmph	117
Figure C.8: Total pressure for model three at 100kmph	117
Figure C.9: Total pressure for model three at 120kmph	118
Figure C.10: Total pressure for model four at 80kmph	118

Figure C.11: Total pressure for model four at 100kmph	119
Figure C.12: Total pressure for model four at 120kmph	119
Figure C.13: Graph of drag coefficient for model one at 80kmph when using fine mesh	120
Figure C.14: Graph of lift coefficient for model one at 80kmph when using fine mesh.....	120

LIST OF TABLE

Table 2.1: Model constants for k- ϵ two equation turbulence model	35
Table 2.2: Closure constants for Wilcox's standard k- ω -equation	36
Table 2.3: Constants for the outer layer in the SST turbulence model	37
Table 2.4: Model constants for the inner layer for the SST turbulence model	38
Table 2.5: Model Constants for the RKE Turbulence Model	39
Table 3.1: Basic geometry of the luxury mid bus [BAI].	41
Table 4.1: Sumery of lift and drag coefficient for model one at different speed.....	54
Table 4.2: Summery of drag and lift coefficient for all models at different speed.....	90
Table 4.3: Total pressure distribution for all models at different speed	91
Table 4.4: Summery of percentage of drag coefficient, lift coefficient redaction and percentage of fuel saving.	92
Table B.1: Mesh result when used converted polyhedral mesh for model one.	106
Table B.2: Result when used converted polyhedral mesh for model two.	108
Table B.3: Result when used converted polyhedral mesh for model three.	110
Table B.4: Result when used converted polyhedral mesh for model four.....	112
Table B.5: Result when used fine mesh for grid dependent test at 80kmph.....	113
Table C.1: Approximate drag coefficient value for various types of vehicle today and for future	121

NOMENCLATURE

Abbreviations

AFC	Active Flow Control
BC	Boundary Condition
BL	Boundary Layer
BAI	Bishoftu Automotive Industry
BEM	Boundary Element Method
CAD	Computer Aided Drafting
CFD	Computational Fluid Dynamics
DNS	Direct Numerical Simulation
FVM	Finite Volume Method
FEM	Finite Element Method
LBM	Lattice Boltzmann Method
LES	Large Eddy Simulation
RAM	Random Access Memory
RANS	Reynolds-Averaged Navier-Stokes
SST	Shear Stress Transport
SUVs	Sport Utility Vehicles
3D	Three Dimensional
GB	Giga Byte
GHz	Giga Hertz

Symbols

A_D	Drag Area
C_p	Specific Heat Capacity
C_L	Lift Coefficient
CPU	Central Processing Unit
C_p	Coefficient of Pressure
C_D	Drag Coefficient
D_p	Pressure Drag

D_f	Friction Drag
F_L	Lift Force
F_D	Drag Force
$kmph$	Kilometer per hour
g	Gravity
K	Coefficient of Thermal Conductivity
ρ	Density
ω	Omega
ϵ	Epsilon
Φ	Viscous Dissipation Function
P_s	Static Pressure
P_d	Dynamic Pressure
P_o	Free Stream Pressure
P	Pressure
T	Temperature
μ	Dynamic Viscosity
ϑ	Kinematic Viscosity
%	Percentage
V	Speed
V_∞	Free Stream Velocity

ABSTRACT

In recent times increase in fuel price is meticulously observed and pertaining to current day's strict government regulation, current buses are on track of being much more inefficient in terms of fuel costs. A large percentage of the usable fuel employed in a commercial bus is utilized to overcome the aerodynamic drag at highway speeds and therefore if reduced could produce a large reduction in the total fuel consumption of the vehicle and reduce the negative environmental effects. This project modified the outer surface of the bus aerodynamically in order to reduce the effect of drag force of the bus which in turn results in reduction of fuel consumption of the bus. Luxury mid bus was modeled in this study with airvanes and front modification simultaneously as drag reduction devices, were employed on the model bus to reduce the drag coefficient of the bus. And the models were entirely designed using the software SOLIDWORK 2017. The models were existing luxury mid bus, existing mid bus with airvane device and airvane device with front faring. The airflow surrounding the bus models were modeled using a computational fluid dynamics software. This work depicts how the modifications could reduce the drag and the lift forces acting on the bus. CFD analysis for redesigned models shows remarkable results for the reduction in drag values and lift values. In this software Realizable k- ϵ (RKE) turbulence models with converted polyhedral meshes were employed to observe the effect of these changes. A drag coefficient of 0.553 was produced which was a reduction of 21.389% in model four from model one and 13.76% and 2.867% drag coefficient reduced in model three and two respectively from model one. Fuel consumption of the bus was reduced by 1.72%, 8.066% and 12.833% from model one in model two, three and model four respectively. This reduction in the drag coefficient lead to a maximum reduction of 13.76% in the fuel consumption, and greenhouse gas emission was reduced by 0.516%, 2.4198% and 3.85% in model two three and four respectively from model one, which if applied to all commercial buses would have a significant effect on the total national fuel consumption and a positive impact on the environment.

Keywords: Luxury mid Bus, Drag and lift force, turbulence modeling Fuel consumption, Emission.

CHAPTER ONE

INTRODUCTION

This chapter describes background of the study, statement of the problem, objective of the study, scope of the study, significance of the study, limitation of the study and structure of the study was studied.

1.1. Back ground of the study

The story of the horseless carriage, as the automobile was originally begins in Europe. In 1860, a French engineer named Etienne Lenoir invented an internal combustion engine that used illuminating gas for fuel and an electric spark for ignition. In 1866, Otto and Lange of Germany improved upon the gas engine by developing the four-stroke cycle, which is still used today. In 1885, Daimler, another German, used the Otto cycle in a gas vapor engine, which he manufactured in quantity. The following year he applied his petrol engine to a motorcar. Daimler's engine was used in France by Emile Levassor, who designed a vehicle that set the basic mechanical pattern for modern automobiles [1].

Due to industrialization, and need of mobile labor mankind started to travel in a modern way in the early 19th century . The horse-drawn buses were followed by steam engine buses. At the end of the 19th century the steam engine was replaced by electric motors and soon came the internal combustion engine. The early truck-based buses had the engine located at the front, as shown in Figure 1.1.

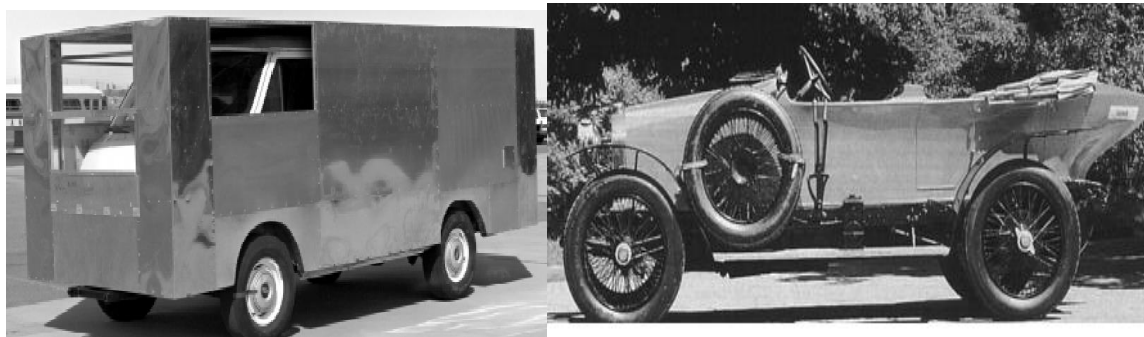


Figure 1. 1: A Scania truck used as transportation [5].

In the 1940s the demand for higher passenger capacity and a larger luggage compartment led to alternative engine locations. The rear mounted engine configuration also enables an easy entrance, especially for disabled persons, which has a high priority in public transport systems. In addition to the customer demands, the bus has to fulfill a number of legal requirements. The customer and legal demands have together led to the bus of today; see an example in Figure 1.2.



(a)



(b)

Figure 1.2: Example of bus in today [huanghai dd serious bus]

Aerodynamics of cars became more and more important with the increase of their velocity. In the beginning of the 20th century, the shape of vehicles was adopted from the field of aviation and ships. Aerodynamics is the branch of fluid dynamics that deals with the motion of air and other gaseous fluids and with the forces acting on bodies in motion relative to such fluids. Automotive aerodynamics is the study of the aerodynamics of road vehicles. The main concerns of automotive

aerodynamics are reducing drag, reducing wind noise, minimizing noise emission and preventing undesired lift forces at high speeds. For some classes of racing vehicles, it may also be important to produce desirable downwards aerodynamic forces to improve traction and thus cornering abilities [2]. Aerodynamics is primarily concerned with flight, its principles are also used in designing automobile and train bodies for minimum drag and in computing wind stresses on bridges, buildings, smokestacks, trees, and other structures.

Wind tunnel is one of the aerodynamicist's basic experimental tools; however in recent years, it has been supplanted by the simulation of aerodynamic forces during the computer-aided design of aircraft and automobiles. Vehicles with an aerodynamic shape use less fuel. Air flows easily over them and less energy is needed to move them forward. At 95Kmph 60-70% of a vehicle's energy is used to move it through the air, compared with only 40% at 50Kmph the rest is rolling and grading resistance [2]. See figure 1.3 drag and rolling resistance relative to the speed of the vehicle.

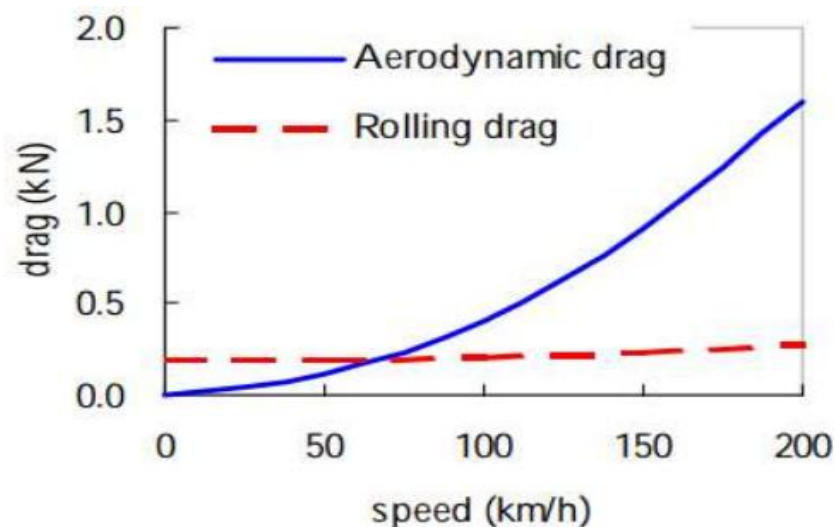


Figure 1.3: Relation of drag and rolling resistance relative to the speed [2].

Installing a sloping front roof on a lorry could save you as much as 7% of your fuel costs [3]. Even small changes to design and shape will make a difference. Vehicles that travel at higher speeds and for longer distances will benefit most from aerodynamic styling, giving you greater savings. Drag is the energy lost pushing through air, and it accounts 26% for most of the fuel used on long-distance journeys, regardless of vehicle type. Overcoming drag uses approximately 60% of fuel used at cruising speeds when loaded, 70% when empty. Sharp corners, racks and parts that stick out will add “parasitic drag”, further reducing fuel efficiency [3].

Buses are the major mode of mass transportation all over the globe, but inefficient in term of fuel consumption, thus in order to decrease the fuel consumption of vehicles, improvement in the aerodynamics of bus shapes will add to the value. More precisely the reduction of their drag coefficient becomes one of the main topics of the automotive research. Decreased resistance to forward motion allows higher speeds for the same power output or lower power output for the

same speeds. Aerodynamics being the aid to form a body shape that maximizes the down force, the negative lifts and minimizes the force that opposes the forward movement and the drag forces. The aerodynamically efficient design of the bus reduces the drag force improving the fuel efficiency. In a moving vehicle, the engine power is used to overcome tractive resistance, which is the combination of rolling and aerodynamic resistance. The rolling resistance will be dominant over the aerodynamic resistance at lower speeds see in fig 1.3 above. Aerodynamic resistance (drag) amounts for more than three fourth of total engine power while operating at higher speeds, since the drag increases as the square of the speed [4]. Thus the maximum power generated by the engine is utilized to overcome the aerodynamic resistance. Due to this the engine load increases substantially which further raises the fuel consumption rate. The external flow analysis of a bus is important as the aerodynamics drag dominate at speed above 65-80kmph [4]. Generally speaking, an incremental decrease in drag can translate into significant fuel savings. Furthermore, buses are used mostly for long-distance travel and the fuel consumption is a big deal for the bus operators.

1.2. Statement of the problem

Increasing fuel price is the main problem in automotive industry. Crude oil prices have gone up in year to year, and also depleting the world proven crude oil reserves, and world oil reserve additions have been less than consumption for the year because of crude oil is a non-renewable energy source. According to [Hucho], the aerodynamic drag of a bus is responsible for a large part of the vehicle's fuel consumption and contributes up to 50% of the total vehicle fuel consumption at highway speeds[25]. The main cause of global warming is the high amount of CO_2 produced by the industry and transportation, so the clever fuel consumption is essential [33]. Bishoftu Automotive Industry (BAI) is an automotive industry which is assembling many varieties of vehicles and exerting its effort in designing and manufacturing of the body and some parts of the vehicle. BAI is built the vehicle body, the same shape with the China manufactures, there has been limited in performing detail aerodynamic analysis yet. There is a considerable scope for study and improving aerodynamic body shape of the luxury mid bus built at BAI to improve fuel efficiency and emission by reducing drag force. Luxury mid bus is new coming bus which has high drag resistance force due to the shape of the body which is almost rectangular blunt as according to Ahmed body of definition of bluff body. It consume 24L/100km fuel at average working speed, according to the standard fuel consumption was high. In this study improve the aerodynamic drag force using airnane device and front faring streamlined simultaneously.

1.3. Objective of the study

1.3.1. General objective of the study

The general objective of this work is to improve and simulate the outer shape of the existing bus (luxury mid bus) aerodynamically in order to reduce the effect of drag force which in turn results in reduction of fuel consumption and carbon dioxide emission of the bus using CFD tool fluent software.

1.3.2. Specific objectives

- To prepare physical model of luxury mid bus using SOLIDWORKS both baseline and improvement.
- To perform flow analyses of the body shapes using CFD both the baseline and improvement.
- To estimate the drag force, lift force, fuel savings, exhaust emission reduction etc.
- To compare the drag and lift force, fuel consumption and emission.
- To prepare the prototype of the modified body shape.

1.4. Scope of the study

This work is delimited to modify the body shape and simulate the air flow around the body using CFD. Wind tunnel test of the prototype is not included.

1.5. Significance of the study

The outcome of this study would give input for bus manufacturers especially during styling process. It would provide some guideline on how to perform the external flow analysis using a CFD tool, especially in terms of the applying a turbulence model. It is also expected to give better understanding about the flow field over the bus and emphasizes more on how to implement an aerodynamic experiment and gathered qualitative and quantitative values.

Apart from that, it can offer the optimization on the bus aerodynamics for the sake of performance and fuel economy. It is true that aerodynamics can be improved by properly designing various component profiles. Computational Fluid Dynamics (CFD) is done to save cost and time of running high cost experiments using experimental wind tunnel. The researches reduced the consumption of fuel and produce less harmful emissions to the atmosphere during the usage of automotive, whereas the result of this paper can be useful as one of the reference for any individual, researcher or manufacturer who are engaged in bus body design, to save foreign currency by reducing fuel consumption.

1.6. Structure of the report

This study is consists of five chapters. The first chapter is introduction about the research study. It includes the background of study, problem statement, objectives, scopes, limitations, significant of the study and the structure of report. Chapter two focuses on the literature review based on the previous aerodynamic experiments done. Chapter three describes materials and methodology of the procedures of this research which was done. The results and discussions are presented in chapter four. The generated data of pressure force, lift force, drag coefficient and lift coefficient are presented in form of tables and graphics for quantitative and qualitative discussion respectively. The contours of flow are also discussed to understand the best design characteristics. The correlation between the experiment result and simulation result is also discussed. The final chapter of this thesis consists of conclusion and recommendation.

CHAPTER TWO

LITERATURE REVIEW

2.1. Previous investigations

Generally, in relative to other heavy vehicles there is somehow shortage in the investigations of buses aerodynamics. Newland [5] develop a transit bus fuel consumption function based upon relationships found in the literature between bus fuel consumption and various bus operating characteristics Especially their variable passenger loads. Diebler and Smoth [6] studied experimentally on a ground research vehicle (GRV) to study the base drag on large-scale vehicles at subsonic speeds. They concentrated on base drag of trucks, buses, motor homes, reentry vehicles, and other large-scale vehicles. They presented preliminary results of both the effort to formulate a new base drag model and the investigation into a method of reducing total drag by manipulating fore body drag.

Roy and Srinivasan [7] investigated on the aerodynamics of trucks and other high-sided vehicles that are of significant interest in reducing road accidents due to wind loading and in improving fuel economy. They summarized that modifying truck geometry can reduce drag and improve fuel economy by concentrated on the associated drag due to the exterior rear-view mirrors. Yamin [8] simulate external flow analysis of a coach used by computational fluid dynamics (CFD) technique. From his results suggested that the steady state CFD simulation can be used to boost the aerodynamic development of a coach. Abdel Gawad and Abdel Aziz [9] studied experimentally and numerically the effect of front shape of buses on the characteristics of the heat transfer and flow field from the rear of the bus in driving tunnels. By using three bus models with flat-, inclined-, and curved-front shapes. They found that the front shape of the bus affects its aerodynamic stability in driving tunnels and the cooling of the inclined- and curved-front vehicles is better than the cooling of the flat-front bus by about 20%. François et al. [10] investigated experimentally the aerodynamics characteristics and response of a double deck bus to cross-wind. They measured pressure distributions over the frontal and lateral part of the bus and also drag and lateral forces related to the position of Centre of gravity.

Yelmule and Kale [11] considered experimentally and numerically the aerodynamics of open window buses where airflow due to motion provides comfort. By modifying the bus exterior body they get the overall drag reduction of about 30% at 100 km/h can be reached. Mohamed-Kassim and Filippone [12] analyzed the fuel-saving potentials of drag-reducing devices retrofitted on heavy vehicles. They considered realistic on-road operations by simulating typical driving routes on long-haul and urban distributions; variations in vehicle weight. Their results show that the performance of these aerodynamic devices depend both on their functions and how the vehicles are operated such that vehicles on long-haul routes generally save twice as much fuel as those driven in urban areas

Bus aerodynamics resistance plays a vital role at higher speeds. When the design of the bus body is very flat the resistance is more so, the researchers gave attention towards the design of the bus

body. It is said that about 60-70 percent of the total wind-averaged drag of a bus is attributed to pressure loads acting on the vehicle forebody making it the principal area for drag reduction [13]. The power required to overcome the drag resistance is very high, hence it is sure that aerodynamics is an important factor to be considered. From many reviews it shows using add on components would reduce the drag force. Arun Raveendran et al [14] investigated on changing the body design by lowering the body of the bus and reduced the drag coefficient up to 0.29 from 0.53. Subrata Roy et al [3] modified the design of the truck they get 30% of drag reduced and fuel consumption is reduced around 35%. Arthur E. Sheridan et al [15] study in the van body trailer adding the drag reduction devices. Here flow vanes are used so that the flow will become smooth and the drag is reduced. Amit Ray et al [5] carried out CFD analysis of AUV bare hull with fluent software. For the analysis they used standard $k-\epsilon$, Realizable $k-\epsilon$, RNG $k-\epsilon$, standard $k-\omega$ and SST $k-\omega$ model to find the drag coefficients with various drift angles. Based on their study, they concluded that Realizable $k-\epsilon$ model predicts better than other models. The computed value of axial force is within 20% for a drift angle greater than 4 degree. He Zhang et al [16] studied numerical simulation of AUV vehicle by CFD software Fluent to evaluate maneuverability of vehicles and the corresponding hydrodynamic coefficients. They concluded that the computer simulation provides a low-cost reliable data for the hydrodynamic tests. M. Karim et al [17] carried out the analysis of 2-D axisymmetric flow around bare submarine hull using both structured and unstructured grid. Shear Stress Transport (SST) $k-\omega$ model is used to simulate turbulent flow past the hull surface. They observed that unstructured grid gives better result and flow visualization than the structured grid. The ultimate aim of the project is to reduce the drag and computational time which has the huge impact on fuel consumption and research and development cost.

Saltzman and Meyer [18] they studied experimentally on reduction of drag of trucks and busses, they proof that drag coefficient could be reduced to 0.242 by tuning the exterior with rounded horizontal and vertical corners, smoothed under body and a boat tail. Ludovico Consano and Davide Lucarelli [19] at IVECO truck building company came up with an aerodynamically efficient truck. The lateral lowered side skirts were added to mask the tanks, rear wheels and axles. To prevent flow detachment, many rounded surfaces were added to the exposed surfaces, such as the roof window, side mirrors, sun visor, etc. The test results a fuel reduction of 8%. Callen et al [6] in their experiments they found, rear view mirror alone decrease the drag of the vehicle by 4.5%. Gilhaus [20] investigated on fine tuning on the front leading edge radii. An optimum front edge radius of 150 mm was selected in his study, and a further increase in the radius was found to have the least influence on the coefficient of drag of the bus. Carr [21] concentrated the effects of streamlining the front end of the rectangular bodies. His study shown a stream lined front end with low leading edge found in a drag coefficient reduction by 30%.

From these previous study conclude that we can reduce the drag force by small modification of the outer shape of the vehicle, and improve the fuel economy, exhaust emission and engine performance. In this study improve the aerodynamic drag using airnane device and front faring simultaneously.

2.2. Bus aerodynamics

The relative motion of the vehicle to the atmosphere generates aerodynamic forces on the vehicle's surfaces. In automotive, aerodynamics analysis is divided into external and internal flow categories [22]. Main concern of this thesis is external flow. External flow characteristics are depend on the geometrical configuration and Reynolds number of the flow. Drag, lift and stability are three major terms that differentiated external flows [23]. A moving vehicle relative to the atmosphere, with a velocity vector is effected with two type of aerodynamic forces, the opposite direction of the motion the force is called drag, while the perpendicular component is defined as lift, the influence had seen at high speed of the vehicle. The shape of the vehicle and atmospheric conditions have a key role to determine the aerodynamic loads. It is a sensitive balance of the lift (pitching moment) and side forces (yawing moment), while the rotating moment has a role only in extreme cases see figure 2. 1 [24].

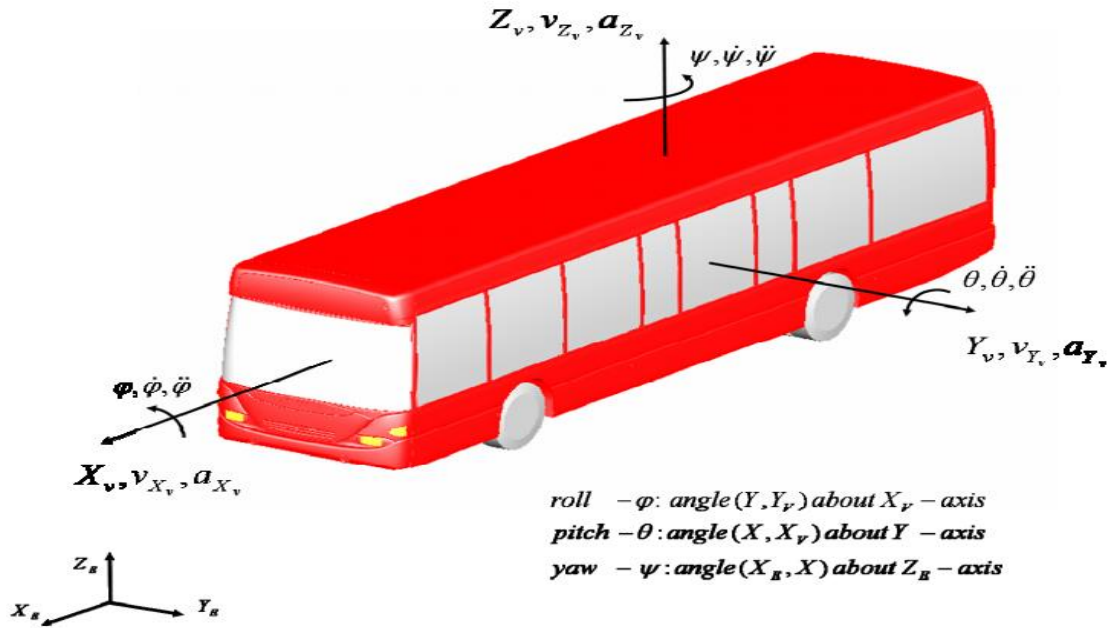


Figure 2. 1: Aerodynamic load on the bus [24].

2.2.1. Flow field around a bus

The aerodynamics of the vehicle is mostly presented by two forces describing the dynamics of the model in the air this is the lift force and the drag force as describe above. Commercial buses is one of blunt-based bluff bodies, which have not an ideal aerodynamic shape and creates an extremely complex flow field, when put in a flowing fluid, as shown Figure 2.2. The complex flow field characterized by a phenomena such as separation and reattachment, high turbulence, large-scale turbulent structures are created around the bus, due to this it has high drag force [25].

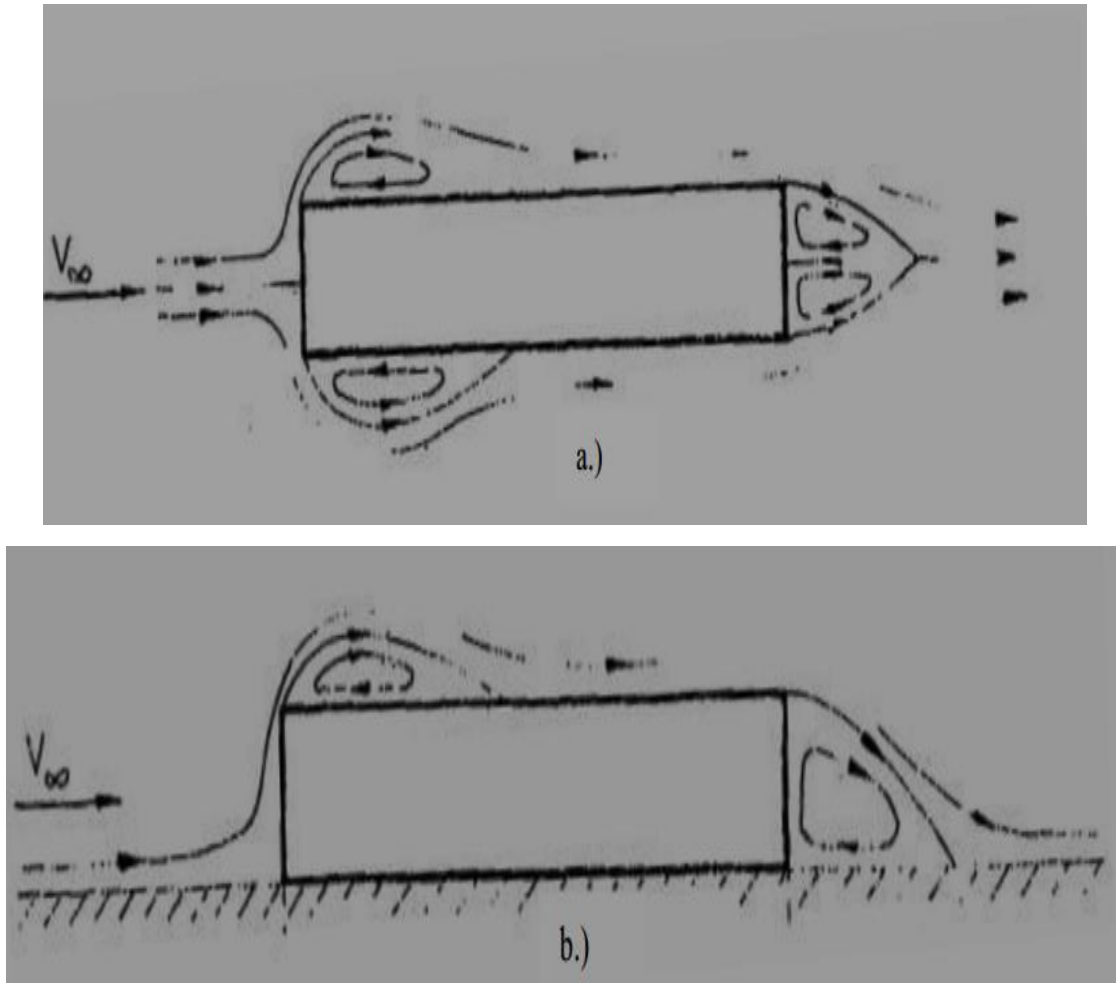


Figure 2. 2: Characteristics of Flow around a bluff body (a) far from ground (b) constrained by ground [25].

From figure observed that, there is a stagnation point at the front of the bus body causing a high pressure zone in front of the vehicle. The air flows around the front corners and creates separation bubbles and reattaches further down the body. Another set of separation bubbles form around the back corners and feed into a turbulent wake at the back of the body formed a low pressure zone which form negative adverse pressure [26].

When a bus or bluff blunt body is placed in a moving fluid, form an axially symmetrical flow pattern see in (Figure 2.2(a)). A stagnation point on the windward face of the bus or bluff body and flow separation around the leading edges. Then reattaches to the side walls and attached until the trailing edges, where another set of separation bubbles form. When the bluff body is constrained by the ground, as in Figure 2.2(b), flow field is similar to the flow field existing on the top side of Figure 2.2(a). The stagnation point is shifted slightly downward and the separation bubble present in the wake is surrounded by a shear boundary layer on all sides, except the bottom, which is constrained by the ground.

Buses are slightly raised off the ground, flow under the bus to be consider as well. Air flow decelerates under the bus and flows out beside the bus laterally. This outward flow displaces the air flowing along the sides of the bus upward and laterally. Air that flows out the rear of the underbody separates on the ground and feeds the separation bubble in the wake as it flows backward [25]. Buses have large frontal areas which create an area of high stagnation pressure on the windward face. A low pressure area is also present on the leeward face of the bus which creating adverse pressure that hindering forward motion called pressure drag force [26]. This Pressure drag forces on buses are accounts for up to 80% of the total averaged aerodynamic drag [27]. Due to a large percentage of pressure drag accounts for such of the overall aerodynamic drag makes it the ideal area to examine for reduction in drag. Increasing pressure in the low pressure zone behind the body would in turn reduce the adverse pressure. Reduction in pressure differential would have a major effect on reducing the total drag force [26]. This flow pattern is very important in understanding the aerodynamics of buses and how drag reduction can be achieved by reducing flow separation.

2.2.2. Laminar and Turbulent flow

In any direction there are constant properties of the fluid in the freestream condition such as viscosity, density, temperature, and pressure. When the moving body contact with the fluid, the fluid velocity and velocity distribution will change as the fluid progresses along the wall. Initially, the velocity distribution looks almost identical to that of the flow before the surface, the flow condition is called laminar flow. Laminar flow is characterized as being smooth and steady, and is the initial section of the boundary layer along a body. If the flow is given a long enough body, the boundary layer will eventually develop into being turbulent. Turbulent flow is characterized by intense cross stream mixing, and has random and chaotic motion.

The laminar and turbulence flow are differentiated by Reynold number (Re)

$$Re = \frac{\rho v L}{\mu} \quad (2.1)$$

Where, $Re < 100000$ the flow is laminar, $Re = 100000$ the flow is transition flow and $Re > 100000$ the flow is turbulence [28].

The region between laminar and turbulent is known as the transition region. It is the region between the critical point and the transition point; the critical point being where the boundary layer becomes unstable ($Re = 100,000$), and the transition point being where the flow becomes fully turbulent [28].

2.3. Drag

The total mechanical forces produced on the surface of a moving object due to the interaction with the fluid in the direction of the motion is called drag. In vehicle aerodynamics, drag is the main concern of investigation, its magnitude is largely affect the vehicle characteristics such as fuel consumption, speed and emission. In case of viscous flow the boundary layer has a strong

influence on the external flow development, like separation where the adverse pressure gradient causes the detachment. So the viscosity and the pressure distribution is related. The drag consist a pressure source term normal to the surface (pressure drag), and a viscosity originated shear stress term parallel to the surface (friction drag). In other approach the aerodynamic drag is an integral of static pressure, friction and turbulence stresses over the surface of the vehicle [29]. The friction drag D_f in a viscous flow is present all over the wall. Due to the molecular friction, shear stress is acting on the surface. For streamlined bodies like airfoils or flat elongated bodies, where separation does not occur, friction drag is the dominant term of the drag [30].

The pressure drag is the dominant source for automotive flows, compared to the friction drag from the shear stresses. Generally the pressure distribution along a bluff body or a vehicle consist a high pressure area in front (stagnation), and a low pressure area at the rear in the separation zone (wake). The flow separation is highly determines the pressure drag, thus it is important to understand the detachment of the flow. A significant pressure loss behind the body is occurring due to the vortex field kinetic energy dissipation in the wake region (separation zone), where depression and low velocities are characterizing the flow. Drag can measured as a dimensionless coefficient, so it is easier to compare different vehicles. This research is aimed to minimize the drag force specially pressure drag. The expression Equation (2.2) contains the ratio of drag force, D where, $D = D_f + D_p$, sum of friction and pressure drag over the multiplication of dynamic pressure and the largest area from the free stream direction A_D .

$$C_D = \frac{D}{0.5 * \rho * U_\infty^2 * A_D} \quad (2.2)$$

Most current bus in the world have a drag coefficient in the range of $0.4 \leq C_D \leq 0.8$, Light vans range 0.35-0.5, Large articulated trucks ranges 0.55-0.8 and Medium-sized cars ranges between 0.28-0.4 further detail is given in appendix c table 5.5.

2.4. Lift

Lift is the resultant force perpendicular to the free stream velocity. Whenever lift is created, so is drag. For this reason, lift must be mentioned and not ignored, however will not be very critical in the analysis of bus aerodynamics due to weight [26]. The equation for the lift force and lift coefficient is shown in equation 2.3 [26].

$$C_L = \frac{L}{A * (\frac{\rho V_o^2}{2})} \quad (2.3)$$

2.5. Boundary layer

When a solid body is stationary and fixed, the fluid, which can be any fluid, passes over the top of this body. The fluid starts from a region where there is no body, which defines the fluid initial properties and reaches the body, which results in an interaction between the solid wall and the moving fluid. The situation is three dimensional in reality, but will be explained as two

dimensional because the properties of the fluid do not change in the direction perpendicular to the direction of fluid flow. The most important part of the fluid-wall interaction is the resulting “no-slip” condition. If a fluid particle is in contact with the stationary wall, slow down it’s moving. Since any Newtonian fluid is a viscous fluid, then the shear stress will be directly proportional to the rate of strain.

$$\tau = \mu * \frac{dV}{dY} \quad (2.4)$$

τ is the shear stress, μ is the dynamic viscosity, and $\frac{dV}{dY}$ is the rate of strain (change in velocity with respect to position). This means that the stationary fluid particle will exert a force on the next fluid particle up, due to this shear stress. As the fluid moves down the solid body, the “no-slip” condition creates a layer within the fluid that has been greatly slowed down by the wall. This region is referred to as the boundary layer, which is the region near the surface where the flow velocity changes from the free stream value to zero at the surface. The thickness of the boundary layer, δ , is the distance from the surface to the point in the fluid where the flow velocity is 99% of the free stream as shown in figure 2.3.

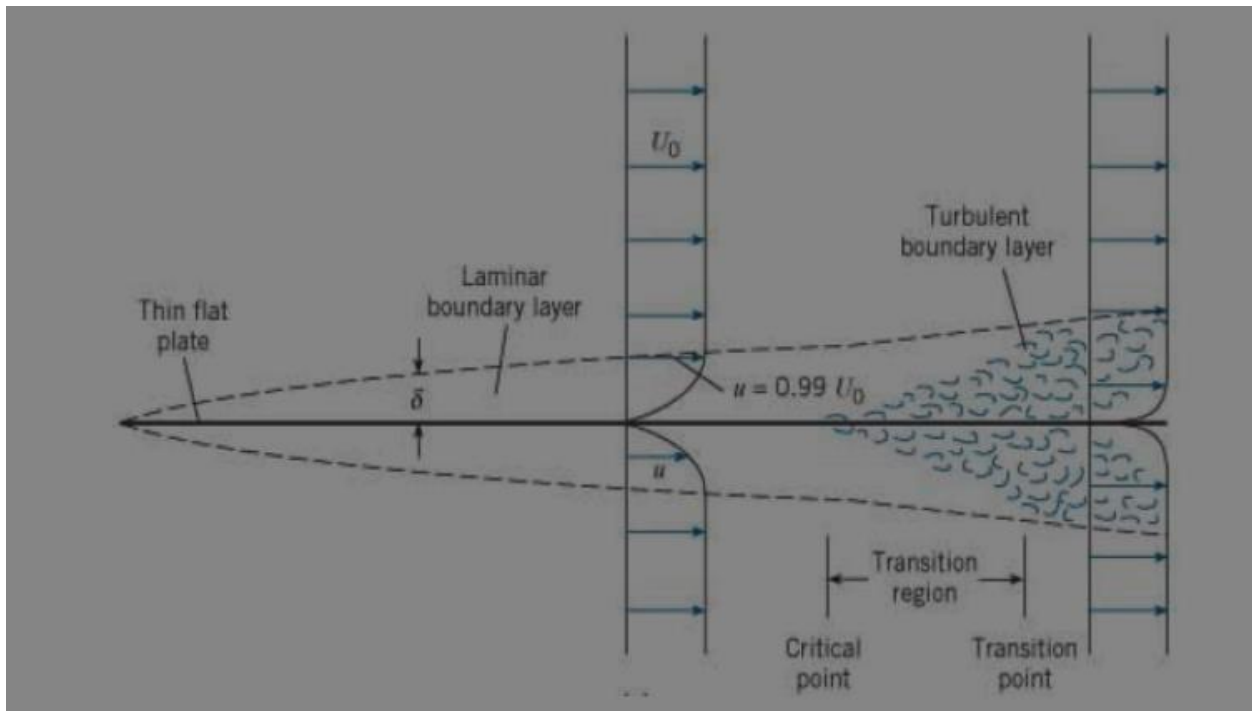


Figure 2. 3: Description of laminar and turbulent boundary layer [58].

2.6. Bluff and streamlined bodies

From external flow point of view aerodynamics of a ground vehicles can be classified as bluff bodies and streamlined bodies. Moving vehicles are influenced by ambient conditions and natural wind. Major contribution for bluff bodies are that the significant drag component originated to the generated pressure drag compared to the friction type. In general if the streamlines are following

the body's surface (streamlined body) due to the D'Alembert paradox the pressure forces are small, while the friction forces are dominating. From his study found that, if the streamlines are follow the surface, applying the Bernoulli equation to determine the flow properties, from the momentum equation we get the final result that the forces acting on the body is zero [31]. In viscous case both pressure and rampant friction forces plays a role. Streamlined bodies such as airfoils can behave as bluff bodies as the angle of attack reaches a certain value [32].

For streamlined bodies to reduce drag advanced flow control methods are available to decrease skin friction show in figure 2.4. For bluff bodies to reach our goal it needs to be considered that pressure drag has to be minimized. Avoid separation of the boundary layer, keep flow attached to the surface as much as possible by using various method. Bluff body flows are characterized by separation of the boundary layer, thus an unsteady velocity field and a low pressure zone evolves in the rear body called wake see in figure 2.4. In 2D bluff body cases these wake regions are generally characterized by periodic vortex shedding. A common example for this phenomenon is the Kármán vortex for lower Reynolds number flows. Due to these conditions the wake has a large energy dissipation rate, thus a high drag arise behind the geometry.

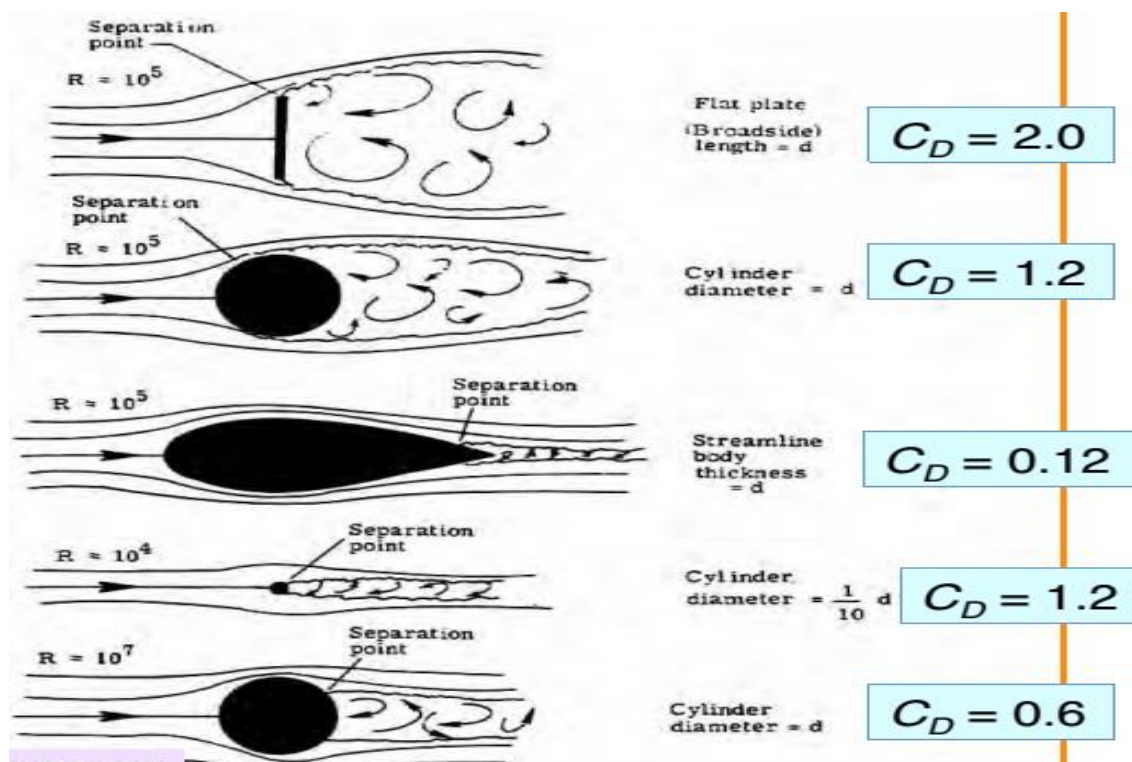


Figure 2.4: Streamlined and bluff body of drag coefficient [32].

In order to reduce the drag it is essential to understand the physics behind it. So the main problem, the boundary layer separation must be investigated. Boundary layer remains thin and does not separate in accelerating flow, that's why in the fore body region near the stagnation point there is no separation. The pressure forces can be related to the boundary layer.

Two basic condition for boundary layer separations is the vicinity of the wall, where the effect of the wall shear stresses arise (viscosity) and the adverse pressure gradient related to stream wise velocity decreasing too. Separation takes place if the deceleration effect of the wall shear stresses and the adverse pressure gradient tops the sum of momentum of velocity distribution. There exist conditions what are influencing the separation, help it to happen or avoid, decrease the possibility. Those conditions which accelerate the process are the following. An obvious case is when the wall shear stress are larger in the turbulent layer when the roughness of the wall is bigger. Larger pressure gradient also affect the separation, it takes place where the flow direction is changing such as cases in front of a concave or behind a convex edge, small rounding up radius. Low kinetic energy (slow velocity) near wall and sudden change in radius also helps the separation. The techniques and methods to prevent or relocate separation are trying to reduce the previous listed cases. Increase the kinetic energy by higher velocity near the wall or blowing in or suction of the boundary layer. For replacing the separation the easiest way is to interfere the laminar-turbulent transition with turbulence generators as shown in figure 2.5 [32].

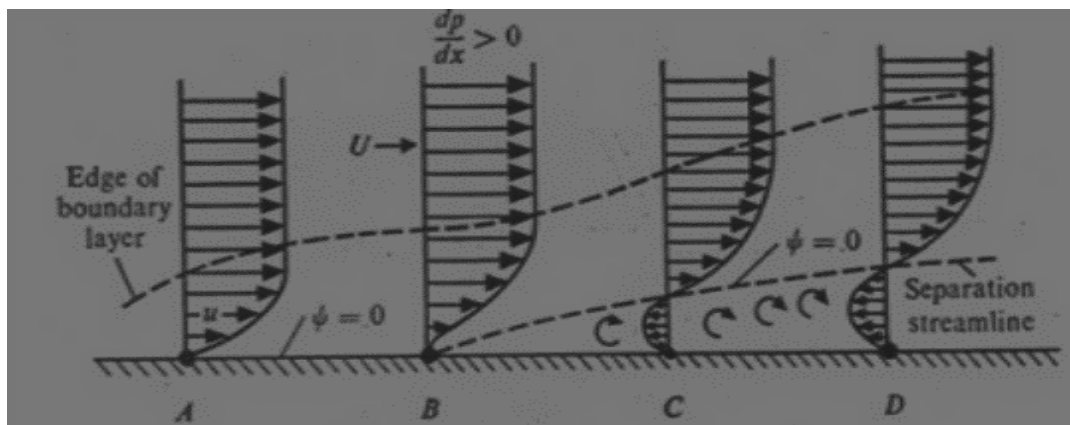


Figure 2.5: Boundary layer separation [32]

Boundary layer separation is a mode of vorticity transport in potential flows by convection. As a result of boundary layer separation, the shear layer enters the flow by creating a closed separation bubble or an open vortex tube. Both cases the characteristic pressure are smaller than the ambient. Separation bubbles and vortex tubes are the typical vehicle aerodynamic flow features in the wake region.

A closed separation bubble development process are containing the following steps. First the boundary layer separation, the formation of shear layer, large velocity gradient across the shear layer and an intense turbulent mixing takes place. Next section is the entrainment, which needs a reverse flow, which has pressure below ambient. It affects the curvature of the streamlines, reattachment or separation bubble closing and the final structure develops [32].

Main features of the closed separation bubble are important to know if the task is to reduce the size effectively. In the bubble compared to the undisturbed motion a relatively slow (20-30% of velocity) reverse flow takes place. The bubble is surrounded by walls and the closed shear layer.

An approximately constant lower pressure compared to the ambient, which is affected by the angle of the undisturbed flow and the tangent of the shear layer at the separation point. An intense turbulent mixing and high turbulence intensity region with a fairly definable passive flow structure[32].

The other structure the open vortex tube is generated when an edge and the flow close a relatively small angle, e.g. Delta wings. The tube's diameter is increasing downwards with length. The pressure and velocity is decreasing as the tube axis is approached. Open vortex tubes are active flow structures, with lower pressure than the ambient. This phenomenon arises at buildings top edges and as well in hatchback cars [33].

Generally the separation's place is fixed in case of sharp edges of the bodies. For rounded, curved shapes where is no sharp edge, the separation depends on the pressure distribution and the position is moving until it reaches the equilibrium. The place of separation can move periodically in time [34].

To sum it up the forces acting on a bluff body is mainly caused by the unbalanced pressure distribution along the geometry. The wall shear stress is playing only a small role compared to streamlined bodies where this force is the dominant, but essential for the boundary layer separation. Bluff body flow are characterized by separation of closed bubbles or open vortex tube depending on the geometry and flow conditions. This low pressure near wake region contains the vortex structure which leads to the high drag [35]. Ground vehicles are categorized as bluff body because of the geometry and aerodynamics flow features. In automotive aerodynamics streamlining is not a straightforward tool to reduce drag, since the shape of the body is determined by other designing constraints and different purposes (heavy vehicles) [36]. Drag reduction can be reached by base flow modifications by active and passive techniques.

2.7. Impact of aerodynamic drag on fuel consumption

Due to power consumption, fuel prices and concern of greenhouse gas emissions, automotive design engineers are focuses on design of vehicles aerodynamically. This is one method of improving performance of the vehicle .The ineffective aerodynamic shape results in excessive drag which leads to increased fuel consumption rates. The main cause of vehicle aerodynamic drag is due to pressure drag or form drag. Pressure drag on vehicles due to flow separation constitutes more than 80% of the total aerodynamic drag [37], while frictional drag constitutes for the remaining 20%. Thus, reducing aerodynamic drag is significant for the fuel consumption efficiency. In United States, the ground vehicles consumed about 77% of all (domestic and imported) petroleum; 34% is consumed by automobiles, 25% by light trucks and 18% by large heavy duty trucks and trailers. It has been estimated that 1% increase in fuel economy can save 245 million gallons of fuel per year. Additionally, the fuel consumption by ground vehicles accounts for over 30% of CO_2 and other greenhouse gas (GHG) emissions. Moreover, most of the usable energy from the engine goes into overcoming the aerodynamic drag (53%) and rolling resistance (32%); only 9% is required for auxiliary equipment and 6% is used by

the drive-train. 15% reduction in aerodynamic drag at highway speed of 55mph can result in about 5–7% in fuel saving [38].

2.8. Flow separation

When fluid flows over a body and assuming a case where flow is attached, ideally the pressure distribution on both upstream side and downstream side of the body remains same and thus, there are no unbalanced forces acting on the body and thus no pressure drag (the only drag force acting on the body will be skin friction). But in most cases the boundary layer does get separated from the body. In this case the pressure in the downstream side is very low and is almost constant in the region. The pressure drop around sharp edges of the body (at these places the velocity of the fluid stream is higher). This pressure is much lower when compared to high pressure formed in the upstream region at the stagnation point. Recirculating flow is caused by the flow separation, the thickness of the boundary layer increases before the velocity reverses indicating a boundary layer separation [38].

2.9. Flow control

Flow control is an active research area in fluid mechanics. Being able to manipulate a flow field in order to achieve desired engineering results seems to be the way to meet today's demands in competitive and efficient solutions for the automotive industry. Based upon whether the methods consume energy to control the flow or not, they are classified into active or passive control methods. Bellow describe in detail.

2.9.1. Passive flow control

The passive flow control systems uses more or less discrete obstacles, added around or on the roof of the vehicle. They can be declined in two groups according to their influence on the flow control. The first group consists on obstacles positioned on the surface of the geometry. The second group consists of the obstacles positioned upstream or downstream of the geometry to be controlled [39].the following are some example of passive flow control methods.

2.9.1.1. Effect of VG on aerodynamic drag

Vortex Generator is an aerodynamic surface which is basically a small vane that creates a vortex. van de Wijdeven and Katz [40] studied the surface pressure distribution and the trailing vortex signature behind the VGs of a generic model in a low speed wind tunnel. In order to delay the flow separation at the rear, bump-shaped VGs at the roof end of a cars is used. VGs themselves create drag, but they also reduce drag by preventing flow separation at downstream. The overall effect of VG was calculated by totaling the positive and negative effects.

Reduction of the aerodynamic forces on a minivan has been achieved using a pair of pockets at the left and the right sides, respectively, of the rear roof end of the vehicle. This increased turbulent kinetic energy induces the flow separation to be delayed further downstream along the vehicle back. Mazyan [41] numerically investigated the effect of VGs on drag. Drag reduction of 4.2% and 10% were attained for the SUV model and Ahmed car model, respectively.

2.9.1.2. Effect of spoiler on aerodynamic drag.

The spoiler is used to minimize unfavorable air movement around the vehicle and can be divided into the front spoilers and the rear spoilers. The added spoiler can diffuse the airflow passing a vehicle, which minimizes the turbulence at the rear of the vehicle, adds more downward pressure to the back end and reduces lift acted on the rear trunk [42]. Hu and Wong [43] studied the flow around a simplified high speed bus car with a rear-spoiler. The standard $k-\epsilon$ model was selected to numerically simulate the external flow field of the simplified Camry model with a rear-spoiler proof that lead less vehicle fuel consumption on the road see in figure 2.6.

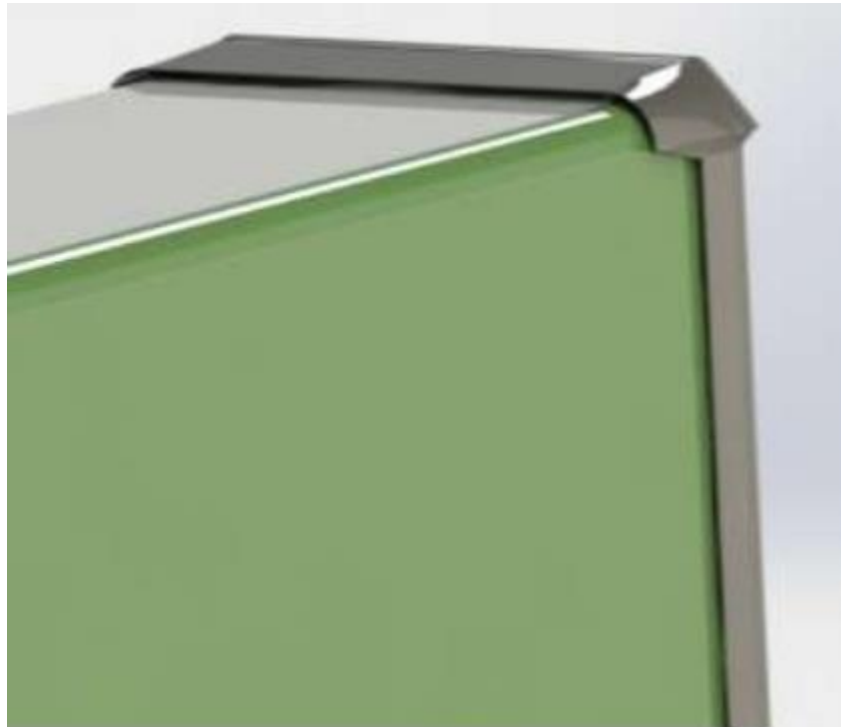


Figure 2.6: 3D: Bus model with rear spoiler [43].

Kodali and Bezavada [44] studied on the aerodynamic drag reduction using rear spoiler on a truck. Results show that there was considerable reduction in the coefficient of lift of around 80% with the presence of rear spoiler and a very insignificant increase in the coefficient of drag of around 3% when the vehicle was fitted with a rear spoiler. In conclusion, the study reveals that rear spoilers have considerable effect on lift, i.e. vehicle stability and moderate effect on drag.

2.9.1.3. Others add-on devices

Wahba et al. [45] numerically investigated the use of lateral guide vanes as a drag reducing device for ground vehicles as shown in figure 2.7. Two types of ground vehicles were considered, a simplified bus model and a simplified sport utility vehicle (SUV) model. The guide vanes were used to direct air into the low-pressure wake region in order to enhance

pressure recovery, which in turn would reduce form drag and hence the overall aerodynamic drag. CFD simulations were used to assess the efficiency of the drag reducing device. The steady-state simulations were based on the RANS (Reynolds Averaged Navier Stock equations). Results of simulations indicated that an overall reduction in the aerodynamic drag coefficient is up to 18% for the bus and SUV models with lateral guide vanes.

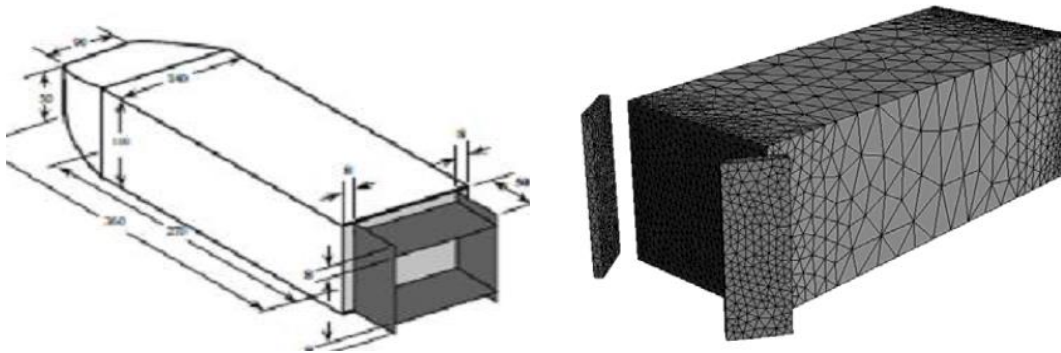


Figure 2.7: Schematic with lateral guide vanes installed for (above) the box and (below) the SUV models [45]

Khalighi et al. [46] investigated several drag reduction devices installed in the rear of a bluff body with square back geometry in ground proximity. The experiments include drag, base pressure measurements and velocity measurements. The results of the drag and base pressure measurements show that significant reductions of the total aerodynamic drag (as high as 48%) was achieved. The results also indicated that models with base cavity have lower drag than model without it.

2.9.1.4. Design optimization for drag reduction

Song et al. [47] investigated on aerodynamically optimized outer shape of a Sedan by using an Artificial Neural Network (ANN), which done on modifying the rear body shapes of the Sedan. To determine the optimization variables, the unsteady flow field around the Sedan driving at very fast speeds was analyzed by CFD simulation, and fluctuations of the drag coefficient (CD) and pressure around the car were calculated. After consideration of the baseline result of CFD, 6 local parts from the end of the Sedan were chosen as the design variables for optimization. Moreover, an ANN approximation model was established with 64 experimental points generated by the D-optimal methodology. After optimized shape of the rear end of the Sedan aerodynamically, the aerodynamic performance was improved by about 5.64% when compared to the baseline vehicle was proposed and also the fuel efficiency of the sedan was improved. Ghani[48] focused on the aerodynamic shape optimization of the rear geometry of passenger cars. The Non-uniform rational Bspline (NURBS) curve was used to represent the rear body of a generic passenger car model (the Ahmed Body) and the NURBS parameters were employed for geometry parameterization. These geometry parameters were systematically modified using design of experiments to obtain different geometries of the simplified car model. Use computational fluid dynamics (CFD) simulations to obtain drag coefficients. design exploration was performed using the response surface model instead of actual

CFD simulations. This technique resulted in a radical simplification of the design process as the behavior of the aerodynamic drag was predicted using a simple polynomial. The proposed methodology was implemented to perform design exploration of a generic fast back model. The response surface model was able to predict the aerodynamic drag coefficients within an error of 5%. Aerodynamic shape optimization was also performed on a generic notch back model using the response surface technique and the optimized geometry parameters for minimum drag were obtained in only 18 iterations. From the result, it can be concluded that the proposed methodology is inexpensive, simple and robust. In vehicle body development, reduction of drag is essential for improving fuel consumption thus protects the global environment and driving performance, and if an aerodynamically refined body is also aesthetically attractive, it will contribute much to increase the vehicle's appeal to potential customers. Islam and Mamun [49] outlined the process taken to optimize the geometry of a vehicle. Vertices and edges were imported into Gambit and a computational domain created. An unstructured triangular mesh was then applied. The CFD program Fluent was used to iterate toward a converged solution with the goal of obtaining a better flow around the car and drag force. The results were analyzed and only the drag force was compared with a recognized journal to validate the results.

2.9.1.5. Couple flow control methods

couple flow control techniques were used to reduce the drag coefficient of the square back Ahmed body. Kim et al. [50] studied on a two-dimensional model vehicle use distributed forcing to reduce drag. The forcing (blowing and suction) was applied at the upper and lower trailing edges, and is steady in time. Both the LES and wind-tunnel experiment were used. LES was performed at the Reynolds number (Re) = 4200, whereas $Re=20\,000$ and $40\,000$ were considered in the experiment. It was shown that a significant base-pressure recovery (i.e. drag reduction) was obtained with the present distributed forcing under LES, together with a substantial suppression of vortex shedding in the wake behind the model vehicle. Similar results were also obtained from the experiment at higher Reynolds numbers, indicating that the present distributed forcing for drag reduction is applicable to a two-dimensional bluff body at various Reynolds numbers.

2.9.2. Active flow control

Active flow control is performed by using actuators that require a power generally taken on the principal generator of energy of the vehicle. The visible part of these systems includes mobile walls, circular holes or slots distributed on the vehicle surface where the flow must be controlled. Their use requires mechanical, electromagnetic, electric, piezoelectric or acoustic systems placed in the hollow parts of the vehicle. Their weights and their overall dimensions must be smallest as possible to reduce their impacts on consumption and habitable volume. Several control solutions have been identified, tested and analyzed for aeronautics. It has been the same for the hydrodynamic and the aerodynamic of the road vehicles. The adopted

solutions generally consist on suction or blowing systems through circular or rectangular slots. [40].The following are some example of active flow control method.

2.9.2.1. Active flow control with air jets

Bideaux et al. [51] studied in a wind tunnel to control the flow separation on rear window of a generic vehicle shape (the Ahmed body with a scale of 0.7 and a slope angle of 35°). This model was equipped with a strip of pulsed jets at the end of the roof to control the flow at a velocity of 30 m/s based on a periodical blowing. A maximum of 20% drag reduction was obtained with a pulsed frequency at 500 Hz and a momentum coefficient $C_{\mu} = 2.75 \times 10^{-3}$. This result confirms the significant of pulsed jets in reducing aerodynamic drag of vehicle.

2.9.2.2. Active flow control with steady blowing

The blunt body geometry of common passenger cars produces a detached, transient flow which induces fluctuating forces on the body, acting especially on the rear axle. This effect may distress dynamic stability and comfort significantly. Hu and Wong [52] studied on a realistic car model by using steady blowing, to observe its influence on the drag and lift forces on the rear portion supplied through continuous slots at positions above and beneath the rear window. The result shown that the reduction of rear axle lift by about 5% was possible with CD changes around 1%. Littlewood and Passmore [53] focused on the influence of steady blowing applied at a variety of angles on the roof trailing edge of a simplified ¼ scale square back style vehicle. Hot-wire anemometry, force balance measurements, surface pressure measurements and Particle Image Velocimetry (PIV) were used to study the effects of the steady blowing on the vehicle wake structures and the resulting body forces. The energy consumption of the steady jet was calculated and was used to deduce an aerodynamic drag power change. Results show that overall gains were achieved.

2.9.2.3. Active flow control with suction

In Harinaldi et al. [54] investigated on a family van was modeled with a modified form of Ahmed's body by changing the orientation of the flow from its original form (modified/reversed Ahmed body). This model was equipped with suction on the rear side. The investigation combines computational and experimental work. Computational approach used a commercial software with standard k-epsilon flow turbulence model, and aerodynamic drag reduction that occurred in the test model. Experimental approach used load cell in order to validate the aerodynamic drag reduction obtained by computational approach. They found that aerodynamic drag reduction is closed to 13.86% for the computational approach and 16.32% for the experimental. The results show that the application of suction in the rear part of the van model gives the effect of reducing the wake and the vortex formation.

2.10. Airvane as a drag reduction device

In this research Airvane will be used as a Drag Reduction Device, which are very similar to lateral air guides. Kirsch submitted patents in 1976 and 1977 studied on bluff bodies which introducing

Airvae used for vehicle with rounded corners like buses and made of two parts bolted together; the air flow guide and the air vane itself. When the air flows through the airvane, if the airvane placed on the leeward face, decelerates into the wake increasing the pressure in the wake zone. This reduces the adverse pressure gradient and the total pressure drag on the body [55].

Based on a standard trailer with a height of 8-13 ft. and a width of about 8 ft. the planar portion of the air vane should be approximately 1 inch above the air flow guide or 2-3 cm. The inlet of the airvane has a larger height than the outlet allowing for a contraction that increases the velocity, the difference between the heights is called the expansion ratio. Kirsch found that the ideal expansion

ratio, b/c , to be 5.5 ± 0.5 while the vehicle travels 70mph or 3.5 when traveling approximately 120 mph, where c is 2-3 cm as mentioned above.

Kirsch was found that the radius of the curved portion be equal to or less than 12% of the distance between the stagnation point and the upper edge of the windward face. The length of the planar portion of the airvane was found to be between $1/2$ and $3/2$ the radius of curvature to ensure proper mixing of the air flow to prevent turbulence. A similar airvane can be placed on the top leeward corner with the one alteration, that the inlet is smaller than the outlet. It was also found that maximum drag reduction was achieved when vertical air vanes were utilized on the vertical windward edges which have identical geometry to the air vane described above. Kirsch et al also wrote a paper in 1973 about airvanes and their ability to reduce the drag utilizing different configurations on both the windward and leeward faces of the bluff body. The geometry of the air vanes was very similar to the patents submitted later, with the addition that the air vanes placed on the side corners on both the windward and leeward faces have an angle of 50° .

The schematic in the top left corner is the standard air vane. The top right schematic is a reversed air vane where the planar portion of the air vane is parallel with the leeward face of the body. The middle left schematic is of stacked air vanes, where the bottom air vane is reverse and the top is a standard positioned air vane. The middle right picture is "piggy-back" air vanes; two reverse air vanes lined up with each other. The bottom most schematic is of a standard airvane placed on the side edges of the body. These different configurations included the standard airvane that was described above, with the planar portion of the airvane parallel with the top face of the bus. The second configuration is the reverse airvane that has the planar part of the air vane parallel with the back leeward face of the body. Airvanes were also stacked on top of each other, where the bottom airvane is positioned similar to the reverse airvane while the top airvane is positioned similar to the standard airvane, making sure that the top airvane is protruding past the bottom airvane. The last configuration was the "piggy-back" formation that contains two reverse airvanes in line with each other.

In his study it was found that while utilizing an air vane on the top windward corner of the body a drag reduction of approximately 10.8% was observed. When 50° air vanes were mounted on the side corners of the windward face a total drag reduction of 8% was attributed to the side air vanes alone. These studies illustrate that airvanes are a successful type of add-on drag reduction device that can be employed on commercial buses [56].

Once the total reduction in drag coefficient was found it could be used to find the reduction in fuel consumption that would be produced by the air vanes employed. The reduction in fuel consumption could be calculated using Equation 2.5 below.

$$\% \frac{\Delta FC}{FC} = \% \frac{\Delta P}{P} = \eta * \left[\frac{\Delta C_D}{C_D} + \frac{\Delta A}{A} + 3 \frac{\Delta U}{U} \right] \approx 3/5 C_D \% \text{ reduction} \quad (2.5)$$

In this equation C_D was the drag coefficient, A as the frontal area and U was the velocity of the vehicle. A property of the diving cycle, η , was used in this equation and it often ranges between 0.5 and 0.7 for vehicles at highway speeds, such as those used in this study [57].

2.11. Simplified model

Companies have to satisfy the customers, fulfill the international standard and maintain the future development. The fossil energy resources are finite and one of the main cause of global warming is the high amount of CO_2 produced by the industry and transportation, so the clever fuel consumption is essential. In case of vehicles one of the solution to the problem is the aerodynamic optimization. The first fossil fuel crisis in the 70's affect largely this sector of the industry. Drag reduction came in to the front that time and since then this aspect is still and getting more important as hybrid technology and solar power driven green vehicles are the options for the prospective future [33].

In the recent time different tools were applied to reach aerodynamic performance optimization goals. From wind tunnel measurements to nowadays CAE techniques the motorsport, passenger and heavy duty vehicles can be managed. There are two ways to done the development process. The first and more applied method is to start aerodynamic concerns in a later phase of design development, the shape and detail optimization. Other approach is to start the full design process with a genuine streamlined body and develop the whole vehicle on that configuration. To test and create such models, simplified or streamlined bodies are used. This is a well approved methodology to investigate the aerodynamic characteristics. From the results of these reduced detailed models relations and conclusion can be drawn to the original vehicles. Studies on these cases show that these simplified bodies can perform realistic flow features as actual cars such as on Figure 2.9.



Figure 2.9: Simplified model geometry [33]

Due to the reduced complexity these models allow to prepare experiments and CAE simulations. The simple geometry has a great potential to equip the measurements cheaper, in simulation point of view the CAD model and CFD mesh are significantly easier to generate. The advantage of this approach appears in the reproducing process, to measure and redo a simulation in order to validation or checking the case. For common models there exist a large range of available data both experimental and numerical, which is a good comparison possibility and a good basis for investigation. Drawback of this approach is that an artificial case is generated and some major

details can be different to the actual vehicle concepts, such as the underbody or the flow conditions. Furthermore this chapter covers one of the most commonly applied simplified bluff body car model the Ahmed body and the actually investigated shape the Windsor model.

Evaluation of flow separation and wake structure around typical geometries of automobiles, Ahmed et al [58, 59] studied on wind tunnels in quarter scale automobile models and concern in the visualization of flow in the wake region on time averaged structure. They also introduced what is now known as the “Ahmed body” geometry. A “generic” geometry consisting of three parts; a rounded up front part, followed by a rectangular box shape and a slant plane in the downstream, shown in figure 2.10.

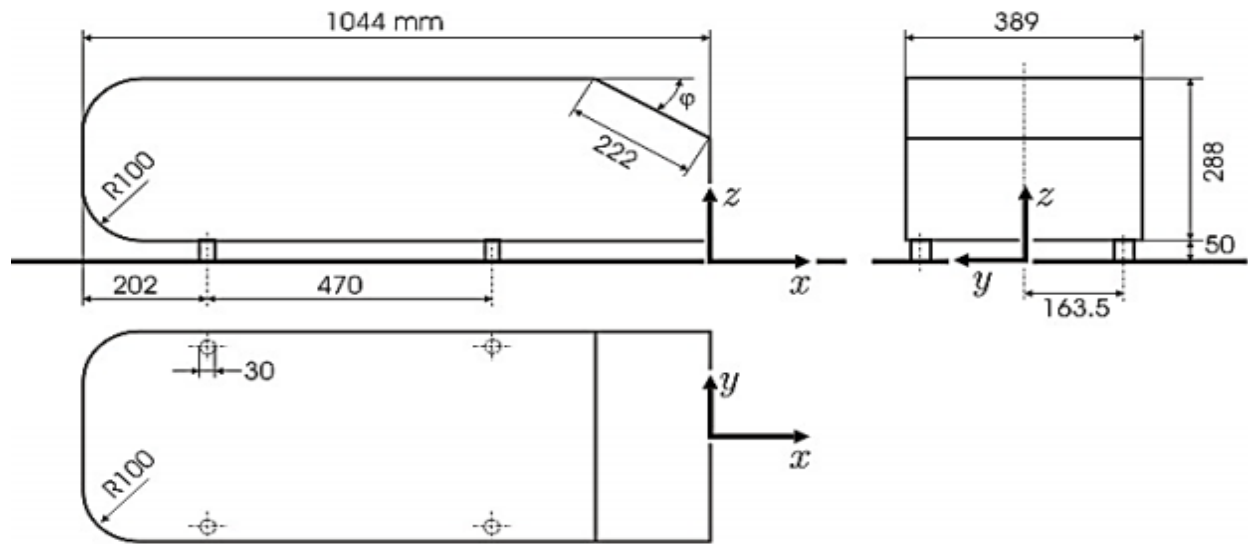


Figure 2. 10: The original Ahmed body. Top view and side view for the standard (30°)slant angle as presented by Ahmed S.R. and Ramm G. [58]

The original dimensions of the geometry introduced by Ahmed S.R. and Ramm G. [58], are the ones seen in Fig.2. 10. The basic characteristics of the geometry which may affect the flow around it, are clear in first sight, however, an extensive analysis to their virtues and phenomena exhibited. The rear slant plane is moveable in order to move in different slant angles for studying flow separation in different angles. The front and the rear part are connected to each other with the rectangular shaped box. The shape of the Ahmed body offers comparatively easy flow simulation environment, but also maintains the great advantage of being a very good scaled replica for exhibiting the aerodynamic features of basic automobile bodies, by having almost the same flow structures. Studies and experiments on Ahmed geometry as in [59] for measurements of pressure, force and wake effect surveys on a combination of different slant angles, it was evident that pressure drag was responsible for a percentage of about 85% of the total aerodynamic resistance, with most of this drag manifestation itself in the rear end of the body. Drag force mostly owes its existence to wake flow behind the Ahmed body with the drag coefficient was

0.262, lift coefficient was 0.159 and total pressure was 630Pa at 40 m/s, so precision in modeling and predicting the flow separation and wake flow is essential in any sort of simulation for this case. An example of the streamlines exhibited in the near wake region of the slant angle, for a Reynolds number of $Re = 768,000$.

2.12. Computational fluid dynamics (CFD)

There are three type of vehicle aerodynamic testing such as road testing, utilizing CFD tools and wind tunnel testing. Three main methods to analyzing fluid dynamic problems exist; theoretical, experimental, and numerical, which can also be referred to as computational fluid dynamics. The theoretical method can also be referred to as the analytical approach, which infers that the problem is solved using known mathematical equations. Since most fluid dynamic problems are complex and cannot be solved directly, assumptions must be made to employ this approach. The main advantage of the theoretical methodology is that “clean” general information is able to be solved for using simple formulas. Reasonable numbers are able to be found and are often are used for preliminary work. Complex flows cannot be solved for without major assumptions that reduce the accuracy of the tests and the solutions found. Along with the fact that it is widely accepted that turbulent flows are mostly not understood, this makes it impossible to solve for theoretically.

Experimental methods are often the primary approach for testing aerodynamic flow fields past buses. Evaluating the performance of a scaled-down model of a prototype device is the main objective of the experimental method. Scaled-down models of the device are relatively inexpensive and therefore it is realistic for companies to use this method. One problem this method faces is that the wind tunnel facility must be able to produce the required free stream conditions which can be difficult to obtain, especially for cases that model large aircrafts or space vehicles. Large scale wind tunnel facilities also require large amounts of energy to run high-speed tests; therefore the run-times must be kept at a minimum for economic reasons. The results of these tests would then be analyzed and a new model would have to be created and tested, this process would continue until an acceptable design was engineered. Experimental methods achieve the most accurate results for most fluid dynamic problems, but the costs of this approach are large and increase every day. In the meantime computational methods are becoming more popular for analyzing complex fluid dynamic problems.

As computers gained popularity a new method of solving complex fluid problems was developed called computational fluid dynamics. The governing equations of fluid motion, including the conservation of mass, momentum, and energy, usually in partial differential or integral form, are solved numerically using this approach to predict the fluid flow. With computational methods few assumptions are needed and a high-speed computer is utilized to solve the resulting governing equations for the fluid dynamic problem. Advantages of this method are that the user is not constrained to linear problems, complicated problems can be solved, and time evolution of flow can be obtained. There are disadvantages to this method which include boundary condition problems and the high costs of a computer that is able to run the complex problems. As

CFD becomes more prevalent in industrial applications the disadvantages decrease due to cheaper computers and turbulence models able to reduce boundary condition issues [60].

Computational fluid dynamics (CFD) software was utilized in this study to determine the effectiveness of specific aerodynamic devices, which are explained in a later section, to reduce the total force drag present on a model bus and in turn reduce the total fuel consumption. There is a wide range of CFD software to choose from today and there are several advantages and disadvantages for each type of software. ANSYS Fluent [61] a widely used CFD software which contains the capability to model flow, turbulence, reactions, and heat transfer problems, was chosen for this study. Advantages of this software are the large selection of turbulence models available for simulation and the high level of accuracy this software is able to model fluid dynamic problems. One disadvantage of this software is the extensive hardware requirements and computing time that is necessary to model the complex problems. Despite this disadvantage ANSYS Fluent was used in this study and the equations behind the simulations were studied.

2.13. Experiment on wind tunnel

Drag force, side force and lift force are the main fluid flow forces acting on a body. Wind tunnel is an instrument which is used to visualize the flow around the body and calculate the magnitude of forces acting on the body. It is basically a tube-shaped structure through which the wind is blown at very high speed under controlled conditions. It is equipped with sensors to determine the air flow change as it passes over and around the models. It is used to imitate the control wind speed of optimal environment facing the model for unlimited time to obtain the aerodynamic steady state experiment results (Mayya 2012). The wind tunnel tries to replicate the real world scenario but it is not possible to check for the specific problems such as vehicles drag when driving through a tunnel or its response to sudden wind gusts, or cross winds and the wakes of the passing vehicle. It can be used to calculate: Force and moments acting on the body, Surface pressure, internal flows and internal flow measurements (ventilation). There is usually some means of measuring and recording the aerodynamic forces and moments on the test model, and these will be corrected for known imperfections in the tunnel [62]. Furthermore, visualization equipment, such as smoke micro tufts, can be used to gain a visual representation of the motion and effect of the flow [63].

2.13.1. Types of wind tunnel

Wind Tunnel can be classified on the basis of construction.

A. Open circuit

In this wind tunnel, there is an intake and an exhaust as shown in figure 2.11. There is no use for corners and long diffusers but needs high power to drive the wind-tunnel because of the loss of energy in the out - flowing air. The open circuit wind tunnel has simple construction. In these tunnels air is expelled directly into the laboratory and typically reinvested after circulating through the lab, though some tunnels utilize instead a compressed gas source. In addition to their low costs, open circuit tunnels are also advantageous because they have relatively

immune to temperature fluctuations and large disturbances in return flow, provided that the volume of the laboratory is much greater than that of the tunnel [64].

These tunnels are so forgiving that exit diffusers can often be completely omitted to allow easier access to test samples and instruments, though the omission often results in a noticeable power loss. Suck down open tunnels are typically more susceptible to low frequency unsteadiness in the return flow than blowers, though some claims have been made that intake swirl is less problematic in these tunnels because it does not pass through the fan before entering the test section [64].

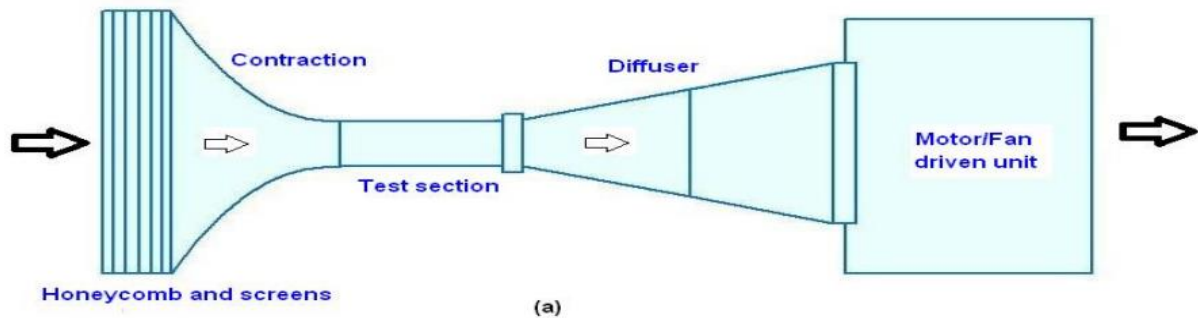


Figure 2.11 : Open circuit wind tunnel layout [64].

B. Closed loop

Closed circuit tunnels form an enclosed loop in which exhaust flow is directly returned to the tunnel inlet. In this wind tunnel, the air is recirculated to improve efficiency for high speed testing. These tunnels are usually larger and more difficult to build. They must be carefully designed in order to maximize uniformity in the return flow. These tunnels are powered by axial fan(s) upstream of the test section and sometime include multistage compressors, which are often necessary to create trans -sonic and supersonic air speeds. Closed circuit wind-tunnels recirculation the air and thus normally need less power to achieve a given low speed, and, above all, facilitate the achievement of well controlled low conditions in the test section. The present, and most low -speed tunnels used for research, are of the closed circuit type [64].see in figure 2.12.

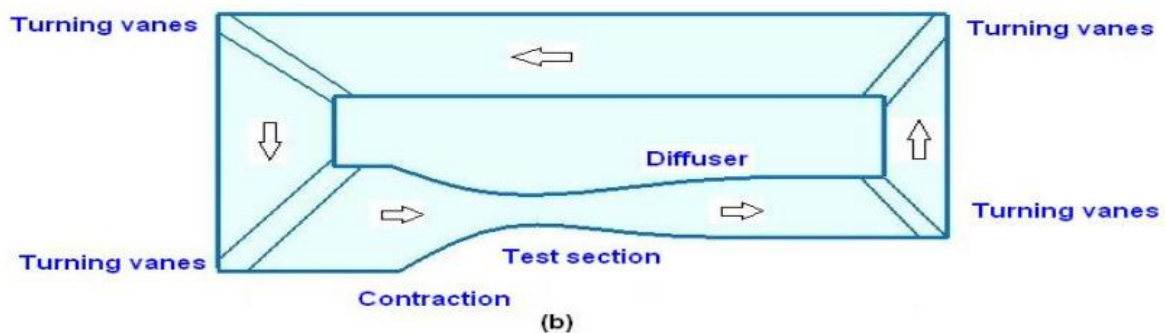


Figure 2. 12: Closed circuit wind tunnel layout [6

2.14. Modeling

2.14.1. Governing equation

The governing equation of the fluid flow represents mathematical statement of conservation law of physics such as mass, momentum and energy are conserved [22]. Consider a small fluid element see in figure 2.13 from this we can model the governing equation.

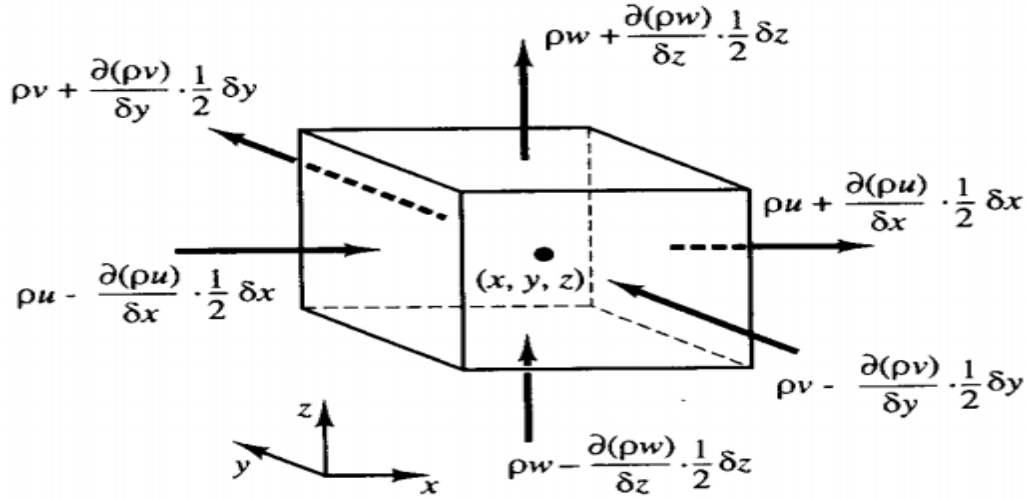


Figure 2.13: Mass flows in and out fluid element [67].

Continuity equation:-states that rate of increase of mass in fluid element = net rate of flow of mass in to fluid element. The continuity equation given by.

$$\frac{\partial \rho}{\partial t} + \partial \frac{(\rho u)}{\partial x} + \partial \frac{(\rho v)}{\partial y} + \partial \frac{(\rho w)}{\partial z} = 0 \quad (2.6)$$

Or more in compact vector form in equation 2.7.

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{u}) = 0 \quad (2.7)$$

Where (u, v, w) is velocity component of in the x, y and z direction, ρ density of the fluid, t time and (x, y, z) is position of the fluid component.

Momentum equation: states that rate of increase of momentum of the fluid particle is equal to the sum of forces on fluid particle shown in figure 2.14 from this solve.

X- Component of momentum equation

$$\frac{\rho Du}{Dt} = \frac{\partial (-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \quad (2.8)$$

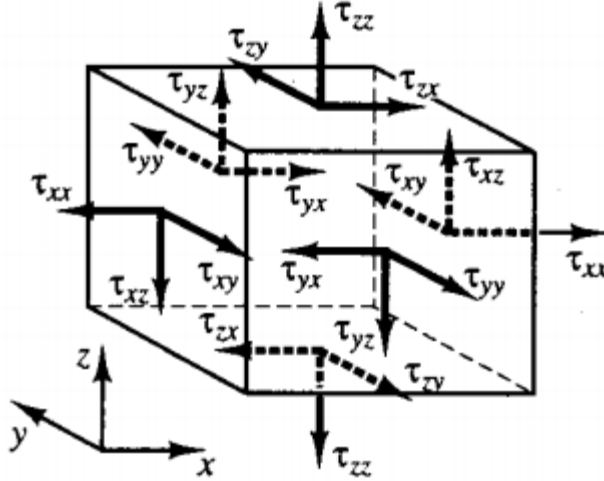


Figure 2. 14: Stress component on three face of fluid element [22]

Y- Component of momentum given by.

$$\frac{\rho Dv}{Dt} = \frac{\partial(-p + \tau_{yy})}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{xy}}{\partial z} + S_{My} \quad (2.9)$$

Z- Component of momentum equation 2.10.

$$\frac{\rho Dw}{Dt} = \frac{\partial(-p + \tau_{zz})}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + S_{Mz} \quad (2.10)$$

Where: τ , stand for viscous stress, ij stand for direction of viscous stress, $S_{Mx,y,z}$ indicates the source term in the x, y, z direction

2.14.2. Reynold Averaged Navier Stoke equation

For the large range of length and time scales of turbulent flows at high Reynolds numbers, it is almost impossible to predict the flow around the body using the Navier Stokes equations. To solve this problem the Navier Stokes equations are derived for a mean state in the turbulent flow [65]. Now a day most turbulent flow are simulated by time-averaged Navier Stokes equations, which are also known as Reynolds Averaged Navier Stokes (RANS) equations. New terms are formed when the equations of motions are time averaged, which can be defined as “apparent” stress gradients due to turbulent flow patterns. To derive the Reynolds equations decomposing the dependent flow variables in the conservation equations to found the time-mean and fluctuating components and then the whole equation is time-averaged. The time averages and fluctuations can be written as shown below.

$$u = \bar{u} + u' \quad v = \bar{v} + v' \quad \rho = \bar{\rho} + \rho' \quad w = \bar{w} + w' \quad p = \bar{p} + p' \quad (2.11)$$

In the above equation u, v, w represent the velocities in the x, y, and z direction respectively, while ρ is the density and p is the pressure. Fluid properties such as viscosity have such small fluctuations as to be neglected in the derivation of the RANS equations. Figure 2.15 illustrates the relationship between the time averages ($\bar{v}$), fluctuations around the average (v'), and the total flow

variables (v). Time averaging of fluctuating quantities will equal zero as illustrated in Equation 2.12.

$$\bar{f}' = \frac{1}{\Delta t} \int_{t_0}^{t_0+\Delta t} f' dt = 0 \quad (2.12)$$

In this study considered an incompressible flow, mass-weighted averaging was unnecessary for velocity components [66]. But in compressible flows the velocity components would be mass weighted averaged to develop more appropriate equations.

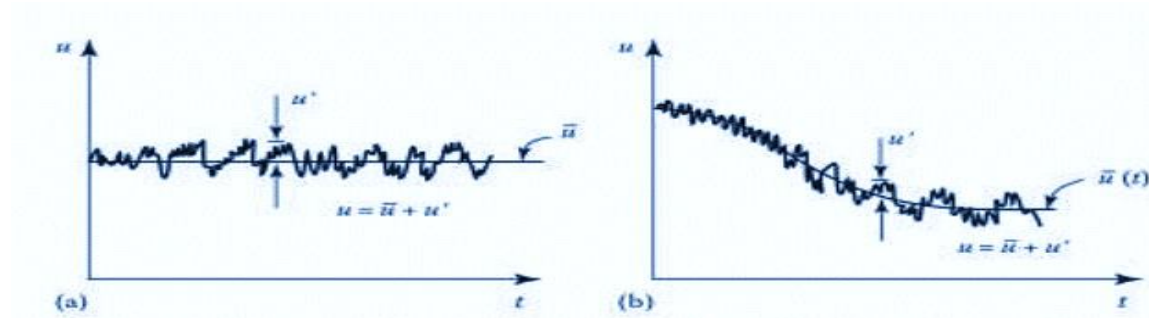


Figure 2.15: Relationship between time-averaged variables, fluctuations and average: (a) steady flow and (b) unsteady flow [66].

The time-averaged variables are denoted by ($\bar{v}$) while the fluctuations are denoted as (v'), the total flow variable (v). It can be observed that in steady flow the time-averaged variables remain constant, but in unsteady flow they do not [66].

There are three fundamental equations of fluid dynamics that are averaged to place them in a Reynolds form; the continuity equation, momentum equation, and the energy equation but energy equation is only necessary for heat transfer problems and not aerodynamic ones. The continuity equation is formed when the conservation of mass law is applied to an inconsequential fixed volume with a fluid passing through shown fig 2.15, in Cartesian coordinates, by Equation 2.13.

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (2.13)$$

After Reynolds decomposition is completed on Equation 2.13 and the variables are in the form shown in Equation 2.11 the entire continuity equation is then time averaged resulting in Equation 2.14.

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}'}{\partial t} + \frac{\partial(\overline{\rho u_j})}{\partial x_j} + \frac{\partial(\overline{\rho' u_j})}{\partial x_j} + \frac{\partial(\overline{\rho u_j'})}{\partial x_j} + \frac{\partial(\overline{\rho' u_j'})}{\partial x_j} = 0 \quad (2.14)$$

Based on the principle shown in Equation 2.12 the second, fourth and fifth terms are equal to zero, leaving Equation 2.15 below.

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial(\overline{\rho u_j} + \rho' u_j')}{\partial x_j} = 0 \quad (2.15)$$

The flow is incompressible flows $\rho' = 0$ therefore the Reynolds form continuity equation becomes Equation 2. 16.

$$\frac{\partial \bar{u}_j}{\partial x_j} = 0 \quad (2.16)$$

To develop the momentum equations in Reynolds form is, starts from the Navier-Stokes momentum equations which are in conservation-law form, consider fluid flows in fixed control volume and apply newton's second law of motion as shown in Equation 2.17.

$$\begin{aligned} \frac{\partial \rho u}{\partial t} + \frac{\partial(\rho u^2 + p - \tau_{xx})}{\partial x} + \frac{\partial(\rho uv - \tau_{xy})}{\partial y} + \frac{\partial(\rho uw - \tau_{xz})}{\partial z} &= \rho f_x \\ \frac{\partial \rho v}{\partial t} + \frac{\partial(\rho v^2 + p - \tau_{yy})}{\partial y} + \frac{\partial(\rho uv - \tau_{xy})}{\partial x} + \frac{\partial(\rho vw - \tau_{yz})}{\partial z} &= \rho f_y \\ \frac{\partial \rho w}{\partial t} + \frac{\partial(\rho w^2 + p - \tau_{zz})}{\partial z} + \frac{\partial(\rho wv - \tau_{yz})}{\partial y} + \frac{\partial(\rho uw - \tau_{yx})}{\partial x} &= \rho f_z \end{aligned} \quad (2.17)$$

Using Equation 2.11 the dependent variables in Equation 2. 17 are replaced with the time averaged variables with the fluctuation and all body forces neglected. The entire equation is then time-averaged. Looking back at Equation 2.12 when linear fluctuation terms become zero when time-averaged, several terms equal zero, while other terms can be cancelled out using the continuity equation, Equation 2.16. All three components of the momentum equations in Reynolds form can be written in k form shown in Equation 2.18.

$$\begin{aligned} \frac{\partial(\bar{\rho}\bar{u}_i + \overline{\rho'u'_j})}{\partial t} + \frac{\partial(\bar{\rho}\bar{u}_i\bar{u}_j + \overline{u_i\rho'u'_j})}{\partial x_j} \\ = -\frac{\partial\bar{p}}{\partial x_i} + \frac{\partial(\overline{\tau_{ij}} - \bar{u}_j\overline{\rho'u'_i} - \overline{\rho'u'_iu'_j} - \overline{\rho'u'_iu'_j})}{\partial x_j} \end{aligned} \quad (2.18)$$

Where: μ

$$\tau_{ij} = \mu \left[\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial \bar{u}_k}{\partial x_k} \right] \quad (2.19)$$

The fluctuations of viscous terms assumed small and can be neglected. These conditions reduce Equation 2.18 to Equation 2.20 and Equation 2.19 to Equation 2.21.

$$\frac{\partial(\rho\bar{u}_i)}{\partial t} + \frac{\partial(\rho\bar{u}_i\bar{u}_j)}{\partial x_j} = -\frac{\partial\bar{p}}{\partial x_i} + \frac{\partial(\overline{\tau_{ij}} - \rho\overline{u'_iu'_j})}{\partial x_j} \quad (2.20)$$

$$\tau_{ij} = \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (2.21)$$

Equation 2.20 or Reynolds form of the momentum equation govern the time-mean motion of an incompressible fluid flow including momentum flux and laminar-like stress terms as well as turbulent stresses shown by the new fluctuation terms. Turbulent stresses can be derived from the Navier-Stokes equations, in particular from the momentum flux terms. This means that the particle acceleration can be related to stress gradients using the equations of mean motion. Knowing the expression for acceleration of the time-mean motion, it can be assumed that any new terms in the equations are the turbulent motion stress gradients. Using the continuity equation, Equation 2.20 can be expressed as shown in Equation 2.22, where the particle derivative is the term on the left-hand side. Equation 2.22 also illustrates the meaning of each term in the Reynolds form of the momentum equation for incompressible flow [66].

$$\underbrace{\frac{\rho D\bar{u}_i}{Dt}}_{\text{particle acceleration of mean motion}} = \underbrace{-\frac{\partial \bar{p}}{\partial x_j}}_{\text{mean pressure gradient}} + \underbrace{\frac{\partial (\bar{\tau}_{ij})_{\text{lam}}}{\partial x_j}}_{\text{laminar like stress gradients for the mean motion}} + \underbrace{\frac{\partial (\bar{\tau}_{ij})_{\text{turb}}}{\partial x_j}}_{\text{apparent stress gradient due to transport of momentum by turbulent fluctuations}} \quad (2.22)$$

Where

$$\bar{\tau}_{ij \text{ lam}} = \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (2.23)$$

$$(\bar{\tau}_{ij})_{\text{turb}} = -\rho \overline{u'_i u'_j} \quad (2.24)$$

The turbulent stresses which is the Reynolds equations cannot be solved as it is due to new unknowns. There are ten unknowns including three velocity terms, a pressure term, and six stress terms compared to the four equations, the continuity equation and the three components of the Navier-Stokes equations. To solve this closure problem, additional equations, including assumptions relating the turbulent quantities and time-mean flow variables or new unknown variables need to be defined. This is done through turbulence modeling [66], [67].

2.14.3. Turbulence modeling

Due to producing accurate numerical solutions for fluid dynamic problems including complex turbulent flows development of turbulence models has greatly increased in recent years. It is the sets of approximate equations CFD software employs to accurately model and simulate flow fields in and around complex objects. There are many turbulence models CFD software can utilize to model different fluid dynamic problems under varying conditions and assumptions. The goal of turbulence models are to find the solutions of the turbulent flows employing fully closed numerical solutions to the Reynolds equations [67].

In the previous study of aerodynamics around bluff bodies to predict the fluid flow turbulence models have been utilized. Pointer (2004) test the drag coefficients around a tractor trailer; the Standard High Reynolds number k-ε model (STD), the Menter SST k-ω model, the (RNG)

Renormalization Group Formation of the k- ϵ model, and the Quadratic k- ϵ model (QUAD) are used and concluded that the SST model was able to most accurately determine the drag coefficient of the vehicle [66]. Kim (2005), investigated on the drag reduction on a large-scale bus used three turbulence models, the STD k- ϵ model, RNG k- ϵ model, and QUAD. In this study it was found that the RNG model was yield a nearly exact solution for the fluid flow when compared to the STD model, it was recommended to not use the STD model. The QUAD model's strength was in modeling the wake behind the vehicle, but its error rate was highly dependent on the discretization scheme [68]. Godwin et al (2014) utilized at commercial buses use different models including Standard k- ϵ model, Realizable k- ϵ two layer model (RKE), SST Menter k- ω model (SST), and AKN k- ϵ model, it was found that AKN k- ϵ model gave the best results, but that the RKE and SST models also gave relatively accurate results [69]. To employ the Standard k- ϵ turbulence model as well as the SST k- ω model. Simulation results illustrated that the SST k- ω model was able to more accurately predict the drag coefficient of the vehicle [70].

Rodi (1997) studied on different k- ϵ models such as Standard k- ϵ model which was found that the Eddy Simulation (LES) method was much more successful in modeling the complex flows involved with bluff bodies [71]. The two turbulence models chosen for this study the Realizable k- ϵ model (RKE) and the Menter Shear Stress Transport (SST) k- ω model. Both models have proven their abilities to provide accurate simulations of fluid flow around bluff bodies as well as having a reasonable computational time. The selection are based on previous study.

2.14.4. Modeling terminology

From the equations of fluid motion, after Reynolds-averaging is used to develop Reynolds Averaged Navier-Stokes equations to relate Reynolds stress term and mean velocity field [66]. To solve Reynolds equations it is necessary to make assumptions involving the apparent turbulent flow. Previous studies have determined that there will be no ultimate turbulence model developed in the near future. Different turbulence models should be used for different problem constraints, but RANS has been proven to be satisfactorily precise for most computational fluids dynamics problems. Two major categories exist for turbulence models that are used to close the Reynolds equations; those that use the Boussinesq assumption and those that do not. The Boussinesq assumption (1877) states that using an apparent scalar turbulent or “eddy” viscosity the apparent turbulent shearing stresses can be related to the rate of mean strain. Equation 2.25 illustrates the Boussinesq assumption.

$$\overline{-\rho u'_i u'_j} = 2\mu_T S_{ij} - \delta_{ij} \left(\mu_T \frac{\partial u_k}{\partial x_k} + \rho \bar{k} \right) \quad (2.25)$$

Where:

Turbulent viscosity, $\mu_T = \rho \nu_T l$

Kinetic energy turbulence, $\bar{k} = \overline{u'_i u'_i} / 2$

The rate of the mean strain tensor S_{ij} is:

$$S_{ij} = 0.5 \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (2.26)$$

Today most turbulence models are turbulent viscosity models or models which use the Boussinesq assumption, also known as first-order models and through experimental tests have been found to be effective in many cases of flow. Reynolds stress models do not employ the Boussinesq assumption to fully close the Reynolds averaged equations. These models are known as second-order equations.

2.14.5. Two-equation turbulence models

Any equations which are used to find the length scale is a partial derivative equation the turbulence model is then considered a two-equation model [66]. In this study chose the Realizable k- ϵ model and the Menter SST k- ω turbulence model. The solved turbulent quantities for all two-equation models are k, the turbulent kinetic energy. The other equation that makes up two-equation models comes from a transport quantity [72]. Variables used as the transported quantity, including ϵ , the dissipation rate, and ω , the specific dissipation rate, where $\omega = \epsilon/k$ and k is the kinetic energy of turbulence [66]. Kinetic turbulent energy is the sum of all the normal Reynolds stresses. The equation to find the length scales can be found using Equations 2.27 and 2.28 using their respective transported quantities.

$$l = C_D \frac{k^{3/2}}{\epsilon} \quad (2.27)$$

$$l = C_D \frac{k^{3/2}}{\omega} \quad (2.28)$$

The k- ϵ turbulent viscosity can be found using Equation 2.29 when, $C_\mu = C_D^{4/3}$

$$\mu_T = C_\mu \rho k^2 / \epsilon \quad (2.29)$$

Using the Navier-Stokes equations to make complete the standard k- ϵ turbulence model by develop a transport partial derivative equation for the turbulent kinetic energy. Assume an incompressible flow Equation 2.30 is the equation that is found.

$$\rho \frac{\partial k}{\partial t} + \rho u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\mu \frac{\partial k}{\partial x_j} - \frac{1}{2} \overline{\rho u'_i u'_i u'_j} - \overline{p' u'_j} \right) - \overline{\rho u'_i u'_j} \frac{\partial u_i}{\partial x_j} - \mu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_i}{\partial x_k} \quad (2.30)$$

To further simplify Equation 2.30 the following substitution can be made to model this part of the equation as a gradient diffusion process:

$$-\frac{1}{2} \overline{\rho u'_i u'_i u'_j} - \overline{p' u'_j} = \frac{\mu_T}{Pr_k} \frac{\partial k}{\partial x_j} \quad (2.31)$$

Where, Pr_k , a purely closure constant, which is turbulent Prandtl number for turbulent kinetic energy. By using the Boussinesq eddy viscosity assumption shown in Equation 2.25. The following can be said:

$$-\rho \overline{u'_i u'_j} \frac{\partial u_i}{\partial x_j} = \left(2\mu_T S_{ij} - \frac{2}{3} \rho k \delta_{ij} \right) \frac{\partial u_i}{\partial x_j} \quad (2.32)$$

Therefore the final turbulent kinetic energy equation can be written as shown in Equation 2.33.

$$\underbrace{\rho \frac{Dk}{Dt}}_{\text{Particle rate of increase of } k} = \underbrace{\frac{\partial}{\partial x_j} [(\mu + \mu_T / pr_k) \frac{\partial k}{\partial x_j}]}_{\text{Diffusion rate for } k} + \underbrace{(2\mu_T \delta_{ij} - \frac{2}{3} \rho k \delta_{ij}) \frac{\partial u_i}{\partial x_j}}_{\text{Generation rate for } k} - \underbrace{\rho \varepsilon}_{\text{Dissipation rate for } k} \quad (2.33)$$

The dissipation rate equation, represented by ε in the k- ε , is found using the same method as was used to find the k equation and is shown in Equation 2.34.

$$\underbrace{\rho \frac{D\varepsilon}{Dt}}_{\text{Particle rate of increase of } \varepsilon} = \underbrace{\frac{\partial}{\partial x_j} [(\mu + \mu_T / pr_\varepsilon) \frac{\partial \varepsilon}{\partial x_j}]}_{\text{Diffusion rate for } \varepsilon} + \underbrace{C_{e1} \frac{\varepsilon}{k} (2\mu_T \delta_{ij} - \frac{2}{3} \rho k \delta_{ij}) \frac{\partial u_i}{\partial x_j}}_{\text{Generation rate for } \varepsilon} - \underbrace{C_{e2} \rho \frac{\varepsilon^2}{k}}_{\text{Dissipation rate for } \varepsilon} \quad (2.34)$$

When Equations 2.33 and 2.34 are used in combination the k- ε turbulence model is created. Table 2.1 contains the typical values for the model constants for the k- ε equations.

Table 2. 1: Model constants for k- ε two equation turbulence model [69].

C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	Pr_k	Pr_ε
0.09	1.44	1.92	1.0	1.3

When using the standard k- ε model further closure is needed due to the fact that this model does not address the damping factor associated with solid boundaries and therefore is not fitting for the viscous sub-layer. To close the system of equations it must be assumed that in the inner region, the law of the wall holds true and by matching a traditional damped mixing-length algebraic model or wall functions, as described by Launder and Spalding, with the k- ε model equations while neglecting the convection and diffusion of ε and k the model can be fully closed. A boundary condition is thus created within the law-of-the-wall region, show in Equation 2.35, at point y_c for k . The final closure solution is shown in Equation 2.36 [66].

$$K(x, y_c) = \frac{\tau(y_c)}{\rho C_D^{2/3}} \quad (2.35)$$

$$\varepsilon = \frac{C_D K^{3/2}}{L} = \frac{C_D [k(y_c)]^{3/2}}{ky} \quad (2.36)$$

Any of the wall functions that allow the k- ϵ model to perform more accurately are also of questionable strength when modeling separating and adverse pressure gradient fluid flows that are present when looking at the flow around a bluff body. Therefore, another transport quantity is more appropriate for these types of flows [65].

The other transport quantity that is often used as the secondary quantity is the specific dissipation rate of the turbulent kinetic energy, ω . Unlike the dissipation rate the specific dissipation rate does not yield an exact equation and in order to use this transport quantity the equation must be developed. There are multiple ways of developing the ω -equation the first being relatively straight forward and very similar to the method shown above for modeling the ϵ -equation. In this method ω/k is multiplied through the k-equation to yield a dimensionally changed secondary transport equation. The second method is more complicated than the first method and is done by replacing ω in the k- and ϵ -equations with ϵ/k . This technique adds several new terms to the transport equation, but provides an equation very similar to the original k- ϵ model with a much improved near-wall behavior. The turbulent eddy viscosity for the ω -equation is modeled in Equation 35, where α^* is a closure constant [72].

$$\mu_T = \alpha^* \rho k / \omega \quad (3.37)$$

Looking at Wilcox's original k- ω model the k-equation was found to be Equation 2.38 while the ω -equation was found to be as shown in Equation 2.39, where β and β^* are closure constants.

$$\frac{\partial(\rho u_j k)}{\partial x_j} = p_k - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.38)$$

$$\frac{\partial(\rho u_j \omega)}{\partial x_j} = \alpha \frac{\omega}{k} p_k - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] \quad (2.39)$$

p_k , also known as the production of kinetic energy of the system, can be defined as shown in Equation 40 for incompressible flows.

$$p_k = \mu_T \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \quad (2.40)$$

Using these turbulence model equations the closure constants are found to be as shown in Table 2.2.

Table 2. 2: Closure constants for Wilcox's standard k- ω -equation [78].

α^*	β^*	A	β	$\sigma_k = \sigma_\omega$
1.0	0.09	0.56	0.075	2.0

The k- ω two-equation turbulence model has been proven to produce more accurate results when modeling adverse pressure gradients and separated flow, both present in fluid flow around a commercial bus, than other two-equation models. Where the k- ϵ model needs a further equation

for near-wall treatment, the k- ω model does not due to the fact that it can be integrated directly to the near wall region. It was found that this model has a tendency to underestimate the near-wall eddy viscosity. Different modifications allow this turbulence model to more accurately predict the viscosity [73].

2.14.5.1. Shear stress transport k- ω model (SST)

This turbulence model combined form of both a k- ϵ model and a k- ω equation. In the SST k- ω model the inner boundary is modeled using a k- ω equation and the outer region of the boundary layer and remain parts of the fluid flow are modeled using a k- ϵ equation. The major disadvantage of the k- ω model is dependent on the free stream value of ω . The k- ω model performs better job of modeling the shear stress in the adverse pressure gradient region and no damping functions are needed in the near-wall boundary layer [74].

Developing a k- ω equation from the k- ϵ equation using a simple change-of-variable, is the first step to creating the SST model, using the relationship shown in Equation 2.41 when $\beta^* = C_\mu$.

$$\omega = \frac{\epsilon}{(\beta^* k)} \quad (2.41)$$

Using this relationship and the chain rule the following k- and ω -equations are found for the outer layer of the fluid flow.

$$\rho \frac{Dk}{Dt} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* k \rho \omega + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_{k2}} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.42)$$

$$\rho \frac{D\omega}{Dt} = \frac{\gamma_2}{v_t} \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta_2 \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_{\omega 2}} \right) \frac{\partial \omega}{\partial x_j} \right] + 2\rho(1 - F_1)\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (2.43)$$

The turbulent viscosity can be defined as shown in Equation 2.44, while Equation 2.45 illustrates the equation for γ_2 in Equation 2.43.

$$\mu_T = \rho \frac{k}{\omega} \quad (2.44)$$

$$\gamma_2 = \frac{\beta_2}{\beta^*} - \frac{k^2}{(\sqrt{\beta^*} \sigma_{\omega 2})} \quad (2.45)$$

Table 2.3 below defines the model constants for the outer layer.

Table 2. 3: Constants for the outer layer in the SST turbulence model [74].

β_2	σ_{k2}	$\sigma_{\omega 2}$	β^*	k
0.0828	1.0	1.168	0.09	0.41

The inner layer uses the original Wilcox k- ω model, shown in Equations 2.38 and 2.39, with modified constants shown in Table 2.4.

Table 2. 4: Model constants for the inner layer for the SST turbulence model [74].

β	σ_k	σ_ω	β^*	k
0.075	1.176	2.0	0.09	0.41

The turbulent viscosity for this part of the model can be defined as shown in Equation 2.46.

Where:

$$\mu_T = \rho \frac{a_1 k}{\max(a_1 \omega, \Omega F_2)} \quad (2.46)$$

$$F_2 = \tanh(\arg_2^2), \quad \arg_2 = \max\left(\frac{\sqrt[2]{k}}{0.09 \omega y}, \frac{500 v}{y^2 \omega}\right) \quad (2.47)$$

After defined the two regions they can be blended smoothly into one set of equations for the whole fluid flow. This theory can be shown on Equation 2.48 with Equation 2.49 illustrating the variables defined, under the assumption that in the inner layer $F_i=1$ and in the outer region F_i approaches zero.

$$F_1 [\rho \frac{Dk}{Dt} + \dots]_{inner} + (1 - F_1) [\rho \frac{Dk}{Dt} + \dots]_{outer} \quad (2.48)$$

$$F_1 \tanh(\arg_1^4), \quad \arg_1 = \min\left[\max\left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500 v}{y^2 \omega}\right), \frac{4 \rho_{\omega 2} k}{CD_{k\omega} y^2}\right] \quad (2.49)$$

Using these equations the final k- and ε -equations for the SST turbulence model come to the following equations:

$$\rho \frac{Dk}{Dt} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* k \rho \omega + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.50)$$

$$\rho \frac{D\omega}{Dt} = \frac{\gamma}{v_t} \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + 2 \rho (1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (2.51)$$

The turbulent eddy viscosity can be defined as shown in Equation 2.52.

$$\mu_T = \rho \frac{a_1 k}{\max(a_1 \omega, \Omega F_2)} = \min\left(\frac{k}{\omega}, \frac{a_1 k}{\Omega F_2}\right) \quad (2.52)$$

Where:

$$F_2 = \tanh(\arg_2^2), \quad \arg_2 = \max\left[\frac{2\sqrt{k}}{0.09 \omega y}, \frac{500 v}{y^2 \omega}\right], \quad \Omega = \sqrt{2 \Omega_{ij} \Omega_{ij}} \quad (2.53)$$

The boundary multiplied by F_2 which is 1 in the near-wall region and nowhere else, while the turbulent eddy viscosity from the original k- ω model is used everywhere else. The SST turbulence model has proven, under adverse pressure gradients, to be able to provide more accurate results in

boundary layers. It has proven to be the most accurate RANS model for aerodynamic fluid flows, making it the best choice for this study [74].

2.14.5.2. Realizable k-ε turbulence model (RKE)

The Realizable k-ε (RKE) turbulence model was developed to overcome the normal stresses and the Schwartz inequality of the shear stresses which are unable to overcome the standard k-ε turbulence model. The governing equations to be adjusted to develop this model. There is many modifications of the standard k-ε model are made to overcome these two constraints and also including a new equation for the turbulent viscosity, a new dissipation rate equation and new model constants, but the equation for the turbulent kinetic energy stays the same as in Equation 2.33. By Deriving a transport equation for the mean-square vorticity fluctuation get the new ε-equation. Another change that is made is the derivation of an equation for C_μ , no longer constant as mentioned above, and is defined as shown in Equation 2.54.

$$C_\mu = \frac{1}{A_o + A_s \frac{U^* k}{\varepsilon}} \quad (2.54)$$

The final k-ε equation becomes the equation shown below.

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \frac{\rho \varepsilon}{\partial x_j} \right] + C_1 S \rho \varepsilon - C_2 \frac{\rho \varepsilon^2}{k + \sqrt{V \varepsilon}} \quad (2.55)$$

Where the variable was defined equation 2.56 and the constant in table 2.5.

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \eta = \frac{sk}{\varepsilon}$$

$$A_s = \sqrt{6} \cos \phi, \phi = \frac{1}{3} \cos^{-1}(\sqrt{6}W) \quad (2.56)$$

$$U^* = \sqrt{\delta_{ij}\delta_{ij} + \Omega_{ij}\Omega_{ij}}, W = \frac{S_{ij}S_{jk}S_{ki}}{\hat{s}}, \hat{s} = \sqrt{S_{ij}S_{ij}}$$

Table 2. 5: Model Constants for the RKE Turbulence Model [75].

σ_k	σ_ε	C_2	A_2
1.0	1.2	1.9	4.0

For the near wall treatment, non-equilibrium wall functions are utilized in ANSYS Fluent For the final closure equation of the different model . This type of near-wall treatment is highly sensitive for adverse pressure gradients and separation fluid fields due to the log-law that is used. For both the viscous sub-layer and the turbulent core region the variables are estimated and used

to determine average dissipation rate and turbulent kinetic energy. The equation used to model the near-wall region is defined by Equation 2.57.

$$\frac{\tilde{u} C_{\mu}^{1/4} k^{1/2}}{\frac{\tau_w}{\rho}} = \frac{1}{k} \ln \left(E \frac{\rho C_{\mu}^{1/4} k^{1/2} y}{\mu} \right) \quad \dots 2.57$$

Where:

$$\tilde{u} = U - \frac{1}{2} \frac{dp}{dx} \left[\frac{y_v}{\rho k K^{1/2}} \ln \left(\frac{y}{y_v} \right) + \frac{y - y_v}{\rho k K^{1/2}} + \frac{y v^2}{\mu} \right] \quad (2.58)$$

In general the RKE turbulence model better than the standard k-ε model, if needed allowing several additional terms to be added, including buoyancy and compressibility terms. This model performs much better for more complex fluid flows including recirculation, rotation, separation, and large adverse pressure gradients, which are largely present in the flow field around a bus making this model more suitable for this study than the standard k-ε model [74].

2.15. Summary of literature

In a moving vehicle, the engine power is used to overcome tractive resistance, which is the combination of rolling and aerodynamic resistance. Aerodynamic resistance increase with increase the speed. The maximum power generated by the engine is utilized to overcome the aerodynamic resistance at high speed. Now a day the main energy source for automotive are gasoline and diesel fuel which is nonrenewable and have greenhouse gas emission. The researcher in this sector focused on producing renewable fuel, modifying the engine and decreasing the aerodynamic drag force. In the literature the researcher focus on flow control are controlled by either modifying the body or use aerodynamic reduction device. In this study focused on reducing the aerodynamic drag resistance to improving the fuel economy and negative effect of exhaust emission by using both method simultaneously.

CHAPTER THREE

MATERIAL AND METHODOLOGY

3.1. Material

The material used in this study was:

SOLIDWORKS: Used to construct the baseline and modified model of bus.

ANSYS fluent: Used to simulate the flow filed around the model bus and calculate the drag and lift coefficient.

Microsoft excel: Used to draw the ANSYS calibration corresponding to the input value.

3.2. Methodology

3.2.1. Data collection

The first task to do in this study was collect the data related to luxury mid bus. The data are shown in table 3.1 bellow and the scale was to fit the computational wind tunnel dimension.

Table 3. 1: Basic geometry of the luxury mid bus [BAI].

Overall baseline dimension		Scaled down 1/30 th dimension
length(mm)	8540	284.66
width(mm)	2455	81.83
Height(mm)	3250	107.80
Interior height(mm)	2030	67.66
Wheel base(mm)	4100	136.66
Trade Front/rear(mm)	1915/1800	63.83/60
Front/rear overhang(mm)	1890/2550	63/85
Min.ground clearance(mm)	222	7.4
Approach/departure angle(°)	10/11	-----
Brake distance(v=30m/h) in mm	≤ 10	≤ 0.333
Fuel consumption (L/100km)	24	-----
Front/rear axle (kg)	3380/7320	112.66/244
curve weight(kg)	8100	270
Total weight(kg)	10700	356.666

Max. speed (km/h)	120	-----
Max. Grade ability (%)	25	0.833
Min. turning diameter (mm)	19000	63.333
Manufacturer	HUANGHAI	-----
Fuel tank capacity (L)	200	6.666
model	Wp5.180E30	-----

3.2.2. Bench marking the baseline model.

From the given data put the baseline model of luxury mid bus shape see in fig 3.1. The dimensions of the body which are almost considered for aerodynamic analysis luxury mid bus was bench marked for the study. It is the latest model, popular in Ethiopia when selecting this research. Engineering parameters of this model was kept as same for the new bus design and was selected as the baseline for studying the aerodynamic performances.



Figure 3. 1: Existing luxury mid bus (baseline) [Bishoftu Automotive Industry]

Criteria for selecting luxury mid bus (baseline model)

Many government and private organizations are using the bus as a service and public transport on Addis Ababa to Adama which is highway road. It is assembled here in Ethiopia in Bishoftu Automotive industry, even if the design and the model of the bus are imported from China. It consumes 24 liters per 100 km. This needs to be improved in total resistance of the bus. Due to these, it was selected to improve the aerodynamic resistance. Which is serious in bus while the speed is above 80kmph and has a significant role in increasing fuel consumption.

3.2.2.1. SolidWorks Scaled-Down Model

Luxury mid bus was chosen for this study, it should be noted that the CAD model used was not an exact replica of the Luxury mid bus, but it is an approximate model.

The luxury mid bus used as the basis for this study was 8540mm from bumper to bumper, had an overall width of 2455mm and an overall height of 3250mm. The distance between the center of front wheels and the center of rear driving wheels was 4100mm, while the front overhang was 1890mm, and the rear overhang was 2550mm. The front overhang was the distance between the front wheels and the windward face and the rear overhang was defined as the distance between the rear driving wheels and the leeward face. These dimensions were then scaled down to create a 1/30th scaled model to fit to the wind tunnel. The dimensions given above give the basic geometry of the bus see in table 3.1 above. SolidWorks scaled down model are shown in figure 3.2 below further detail are on the appendix.

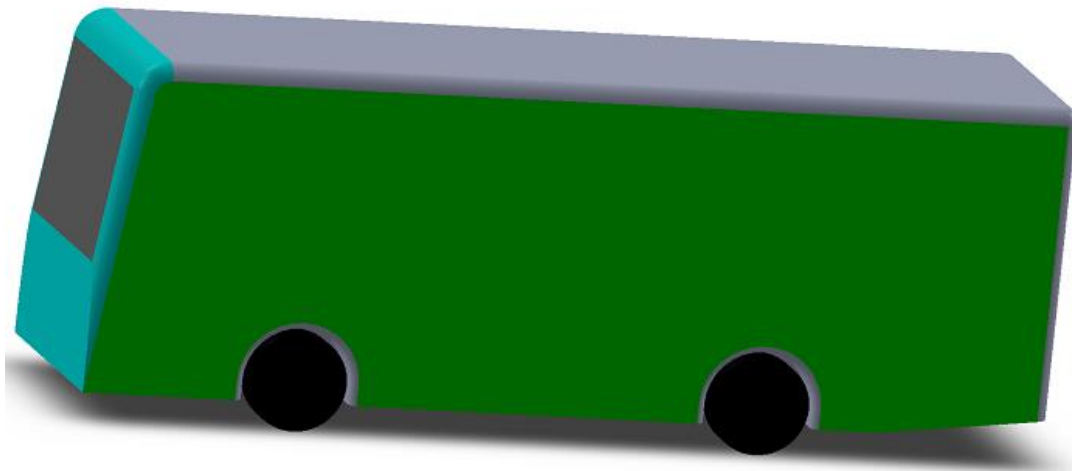


Figure 3. 2: Solid Works 1/30th CAD model one.

3.2.3. Using aerodynamic drag reduction device

In this study Airvane device and front faring used as a Drag Reduction Device. The airvane device which are very similar to lateral air guides. Kirsch submitted patents in 1976 and 1977 studied on bluff bodies which introducing airvanes as a method of drag reduction. Airvanes are nozzle-like devices, which are mounted along the corner edges of both the windward and leeward face of the

bluff body. Front fairing is tapering of the front of the vehicle towards the rear relative to driver visibility and luggage space. Which are used to minimize stagnation area and flow separation at front corner of the vehicle. Both front fairing and airvane device designed by SolidWorks.

After the CAD model was designed and constructed it was imported into ANSYS Fluent where simulations could be performed to determine the drag coefficient of the vehicle at 80m/s, 100m/s and 120m/s. After the model was tested and the drag coefficient for the bus without any drag reduction devices generated, airvanes and front fairing could be designed and added to the model. Parameter studies were implemented for the addition of airvane devices and front fairing on the bus model simultaneously and separately, maximize the reduction in drag.

3.2.3.1. Parameter study to find the optimal value.

Airvanes device and Front fairing was design by SolidWorks and added directly to the bus model and to study the dimensions of the airvanes and front fairing for the parameter studies. As the parameter studies were completed, individual models were made for each parameter examined. In the airvanes there are three loft features, one placed over the flat top face of the bus and one over each curved edge of the top rearward extending face. Two vertical plates which placed between the three lofts used to attaching them to each other and to the bus itself. The dimensions of the three loft was equal, those which were tested in the parameter study.

The airvane dimensions calculated based on Kirch et al.'s [55] study for this thesis. Set the inlet gap 2.5 cm, which for the 1/30th scale was 0.833mm. The curved portion radius of the airvane that extended past the top horizontal leeward corner equal to approximately 12% of the distance between the stagnation point and the top windward edge. Based on this the dimension came of approximately 8.5mm on the 1/30th scale. The airvane planar portion parallel to the top face of the bus was equal to the radius of the curvature. An additional portion of the planar section extended past the leeward face of the bus, before the curved portion began was the same dimension as the inlet gap, 0.833mm.

The dimension of the front fairing was calculated based on the visibility of the driver and luggage space based on M. Kim et al [69] study on model three, which is the windshield angle from the vertical was between "0- 36°" . Increase the angle decrease the drag while the driver visibility and luggage space decrease. In this study the angle select was "26°" for model thee. At 10°, 18° 25°30 ° and 36° was tested [69], the result show that around 25° get good result relative to drag coefficient, driver visibility and luggage space. After studied the parameter, get the bus model shown in figure 3.3 to 3.5 further detail in appendix figure (A-1 to A-5). This model is designed by SolidWorks.

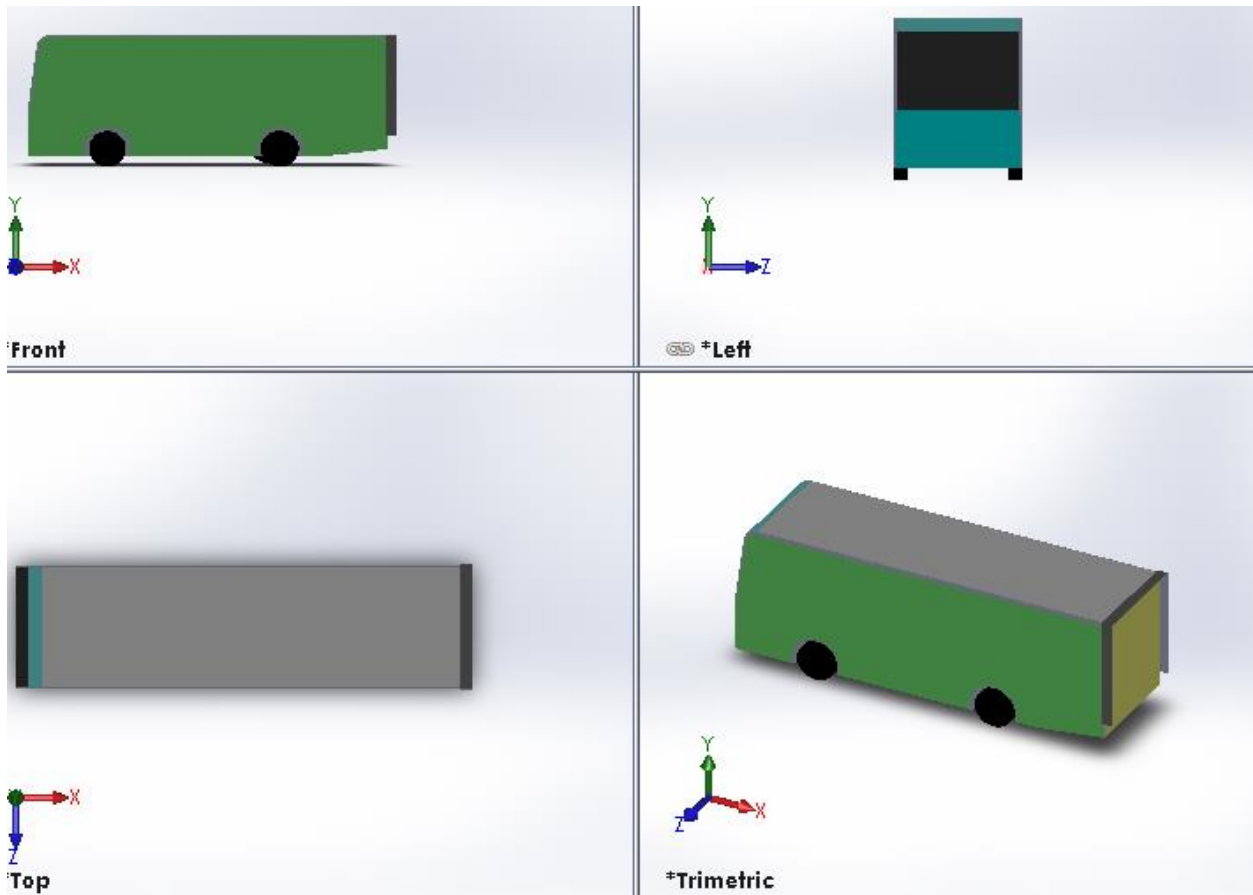


Figure 3. 3: model two

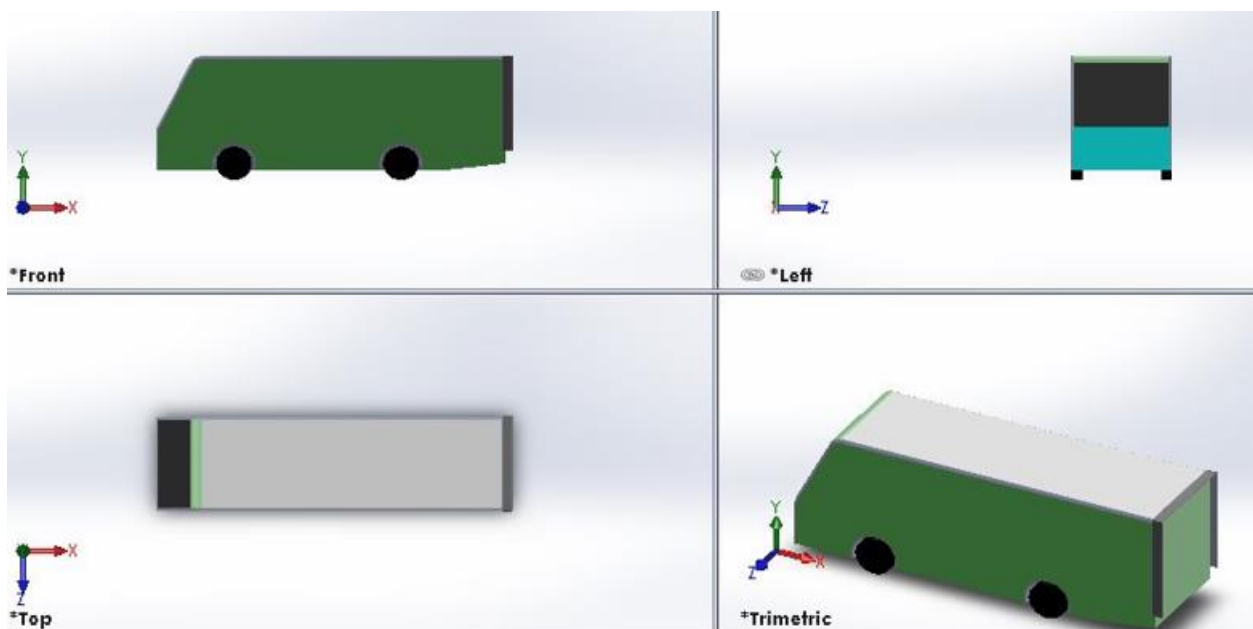


Figure 3. 4: model three

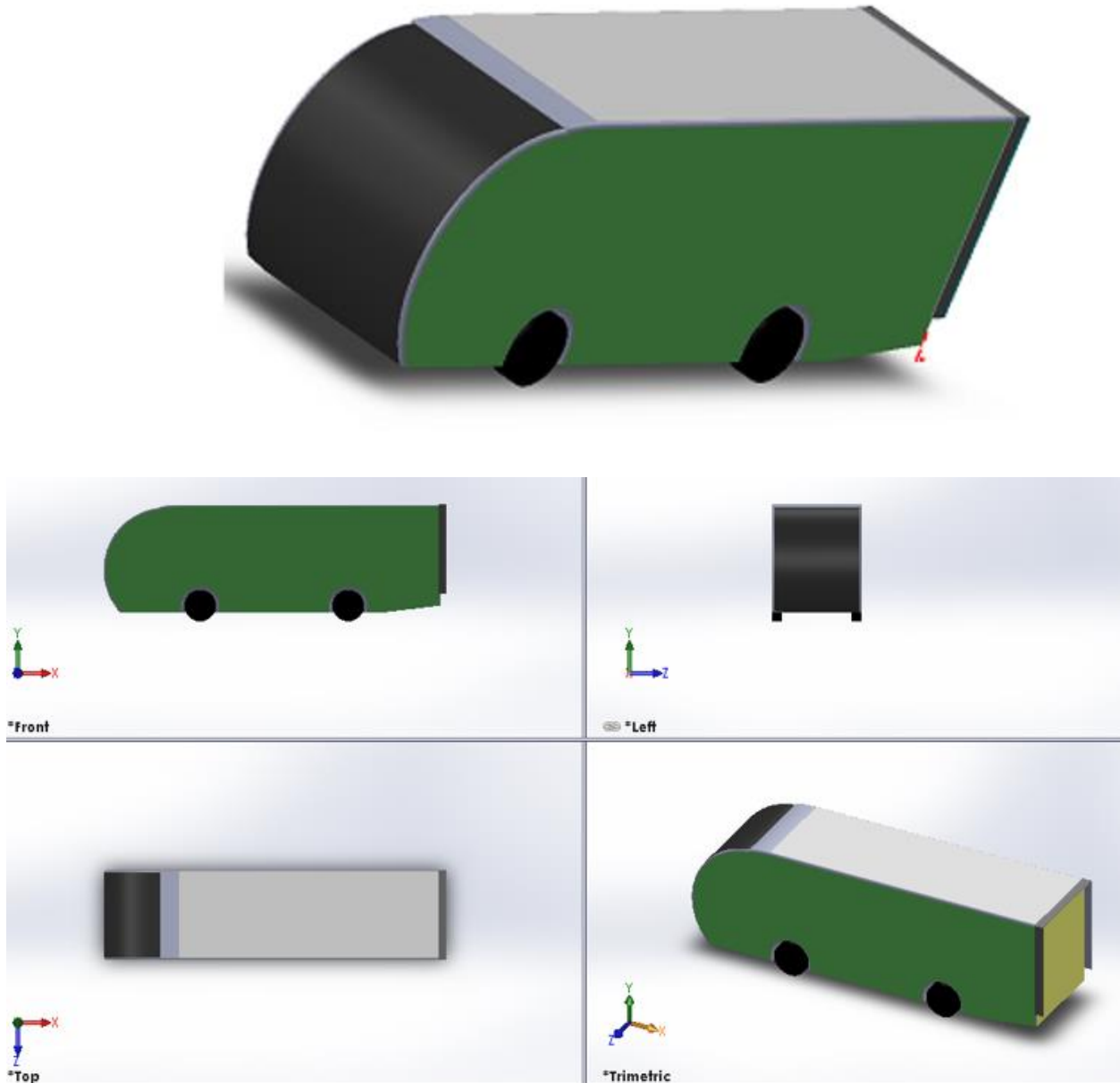


Figure 3. 5: model four.

3.2.4. Turbulence modeling.

In this study use Realizable $k-\epsilon$ Turbulence Model (RKE) described in section 2.14.5.2 and the solver used for the analysis was FLUENT 15.0 and it uses a control-volume-method to solve the governing equations that can be solved numerically. The solver selected was the pressure based implicit solver. In this type the equations of continuity and momentum are solved sequentially.

The flow is considered to be incompressible and steady in nature and the equations are solved using second order upwind method.

3.2.5. CFD/ANSYS Fluent set up and procedure

ANSYS Fluent used as the CFD software, simulate the fluid flow on the SolidWorks CAD models. The simulation process divided into four separate sections such as; Geometry generation, Meshing, Setup, and Solution. For all simulations, for models with and without the airvane and front faring, the process was the same.

3.2.5.1. Geometry generation

The first Step on the fluent setup was to define the geometry that would be modeled by CAD software to represent the air neighboring the bus. After the bus model was uploaded an enclosure was designed surrounding it. For experimental study the enclosure was designed to represent the size of an actual wind tunnel for accurate results. Rodi [71], conclude, in his study of computational methods, the appropriate dimensions of computational wind tunnel were; for the upstream length 3.5 times the overall height of the model bus, for the downstream length 9 times the height, for the width 10 times the height and 6 times the height for the height of the wind tunnel [71]. Dimensions used for the 1/30th scaled wind tunnel model are given below in Table 3.2.

Table 3.2: 1/30th dimension for wind tunnel enclosure designed in fluent.

Upstream length	Downstream length	width	height
0.3796m	1.0833m	0.9750m	0.6500m

After the wind tunnel enclosure was constructed employed the Boolean feature to subtract the volume of the bus from the volume of enclosure to create a new volume that represented only the air that surrounded the bus see in figure 3.6.

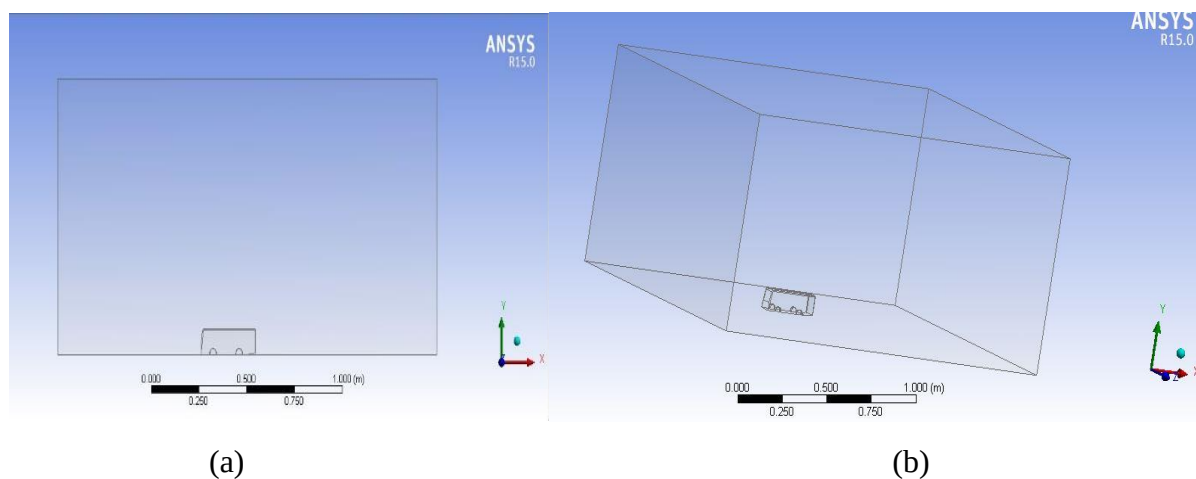


Figure 3. 6: Bus model in the enclosure box

3.2.5.2. Meshing

Once the Boolean was created the Geometry Setup was complete then meshing was started. Polyhedral mesh type was developed in this study. The recommended mesh in the ANSYS fluent software was hexahedral or polyhedral meshes or a hybrid of both. It used for moderately complex geometry similar to the bus model in this study.

fully polyhedral meshes used in this study, which were originally tetrahedral meshes that were converted to polyhedral [76]. Pointer et al. found that an unstructured polyhedral mesh was successful in producing accurate results for external fluid flows [77]. Converted polyhedral meshes rather than constructed in the mesh component of the setup in Fluent which used for reduce the cell count when converting from a tetrahedral mesh to a polyhedral mesh and also required less computational time. The fineness of the entire mesh describe by relevance used 0. Relevance of 100 meant high accuracy and low speed while a relevance of -100 means high speed and lower accuracy. For this study a medium was chosen, which taken a medium amount of simulations time and accuracy. The advanced size function is proximity and curvature, used for the progression from minimum size elements, near and around the curvature of the bus to maximum size elements far away from the bus toward the walls of the wind tunnel box. Put the relevance center at the medium which used that medium entire mesh fineness, which gives appropriate accuracy and taken computational time. Used high Smoothing this increasing the number of smoothing iterations and improving element quality by moving nodes and elements in the vicinity.

The above parameter changes provided a default minimum element size of 1.0507e-003m and a maximum element size of 0.134480m, with the small elements closer to the bus and the larger elements closer to the wind tunnel walls. The default growth rate of 1.2 was used, meaning in each subsequent layer of the mesh the element edge length was increased by 12%. The default node and elements was 178808 and 950809 respectively. The default curvature normal angle, the maximum angle that a single element was allowed to span, of 23.76° was employed. The number of layers used in gaps in the mesh was defaulted at five and the proximity size function source was set to edges in all model but this was for model one see the other in appendix B. Using this setting edge-to-edge proximity would be used when proximity sized function calculations were completed.

Figure 3.7: illustrates the polyhedral mesh that was created when the tetrahedral mesh was converted at model two further detail on the other meshing on the appendix B for all models.

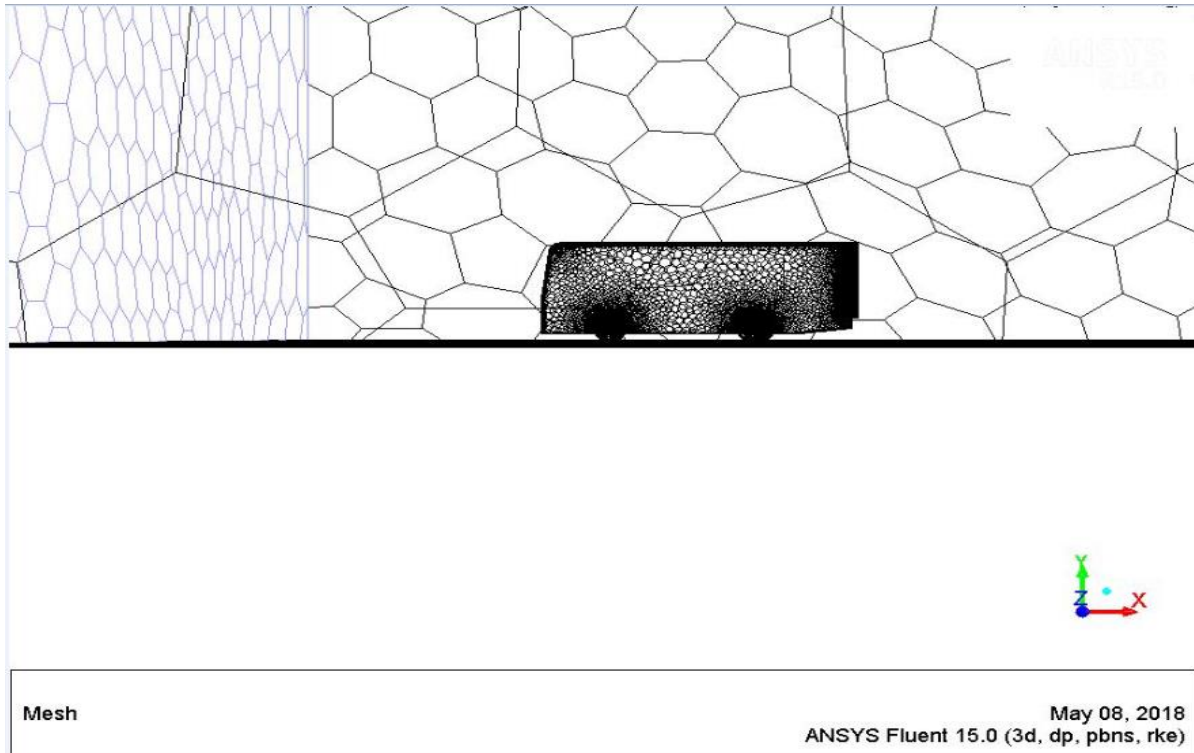


Figure 3.7: Polyhedral mesh created in ANSYS Fluent.

Tetrahedral mesh was converted into a polyhedral mesh increasing accuracy and reducing total cell count, which in turn reduced the total computational time.

Lastly, named sections were created that were referenced in Fluent when performing simulations. The plane in front of the bus was the velocity inlet while the plane behind was the pressure outlet. All other sides were considered as walls. Once the meshes were created the fluid flow simulations could be run.

3.2.5.3. Fluent setup and simulation procedure

Realizable $k-\epsilon$ model was employed in the fluent simulations for all models of the bus with and without the airvane and front faring attached. the Realizable $k-\epsilon$ turbulence model was run with a polyhedral mesh . The simulations were run with a velocity of 80kmph, 100kmph and 120kmph impeding the motion of the bus which is boundary condition. The velocity's origin was set at the named section "Velocity Inlet" in the direction of the named section "Pressure Outlet" in reference to the "Air". The turbulence intensity for the velocity inlet was reduced to 1%. Turbulence intensity can be defined as the root-mean-square of the velocity fluctuations over the mean flow velocity. A turbulence intensity of 1% is relatively low meaning steady air flow and the velocity introduced to the simulation had very low fluctuations and therefore very little turbulence. The area of each specific model was found using Fluent project area feature and was utilized in determining the drag coefficient of the bus model from the force that was

calculated during the simulation. A coupled scheme was chosen for the pressure-velocity coupling. This scheme is more robust and efficient than other schemes for steady-state flows, such as the flow in this study. While employing a coupled scheme the flow Courant number was set to 50 while both the explicit relaxation factors for momentum and pressure to 0.25. The flow Courant number is the ratio between the computational distance step and the distance a disturbance travels in one time step. Reducing this value improves the accuracy and steadiness of the simulation. The reduction in the explicit relaxation factors allowed for a more skewed mesh to be converged easier. They represent the percent of the solution being used from iteration to iteration throughout the simulation. The turbulent viscosity relaxation factor was set to 0.95. In order to monitor the calculation and simulation of the drag coefficient a monitor was created for drag on the bus, due to the velocity impeding the motion of the vehicle. Once the drag monitor was created it could be used to set the convergence criterion for the simulation. Convergence was defined as the drag coefficient residual remaining constant at the 1×10^{-5} [57] value for ten consecutive iterations. Once this convergence was met the simulation was complete and was stopped. The simulations were run for varying times. Simulations employing the Realizable k- ϵ turbulence model were run for 500 iterations and often converged before those iterations were completed. Once convergence was reached the solution could be analyzed to determine the effectiveness of the airvane and front fairing being studied. After the simulations were complete the post-CFD part of Fluent was utilized to fully analyze the solution and the air flow through and around the airvane and bus. Contours of the pressure, turbulence, and velocity magnitudes were created using this software.

3.2.6. Calibration of fluent simulations

When comparing the drag coefficients, once all simulations were run the values found for the drag coefficients were compared and repeatability examined. Similar meshes and turbulence models were compared; meaning the first run of the Realizable k- ϵ model utilizing a polyhedral mesh was compared with the second run of Realizable k- ϵ model utilizing a polyhedral mesh and continuous the 3rd and fourth for four different model at three different speed.

CHAPTER FOUR

RESULTS AND DISCUSSION

4.1. Result and discussion for model one

4.1.1. Result and discussion of scaled residual, drag and lift coefficient, pressure and velocity countour at speed of 80kmph 100kmph and 120kmph for model one.

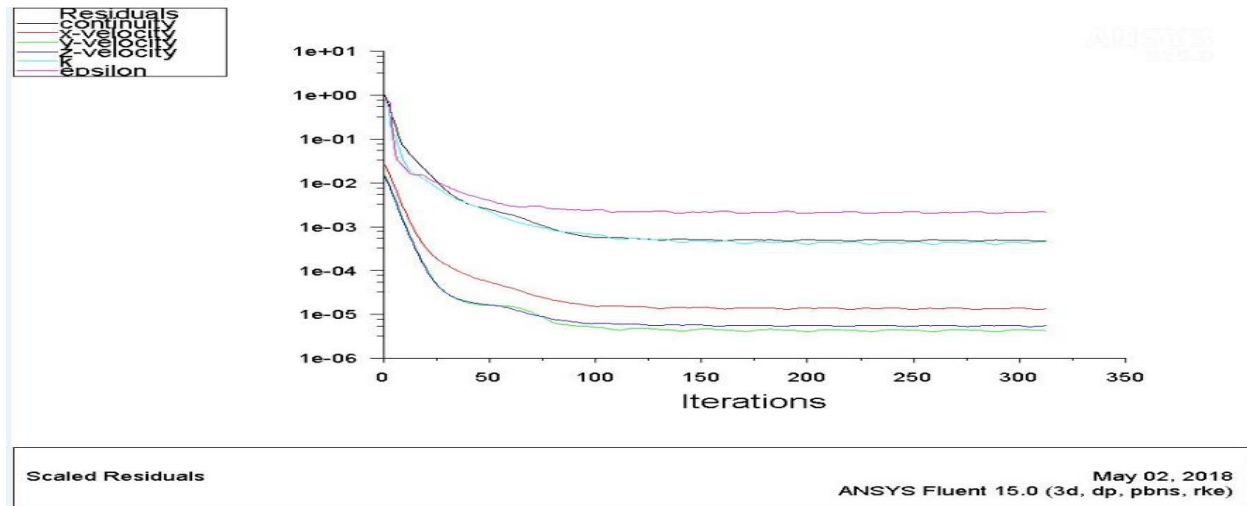


Figure 4. 1: Scaled rsiduals for model one speed of 80 kmph.

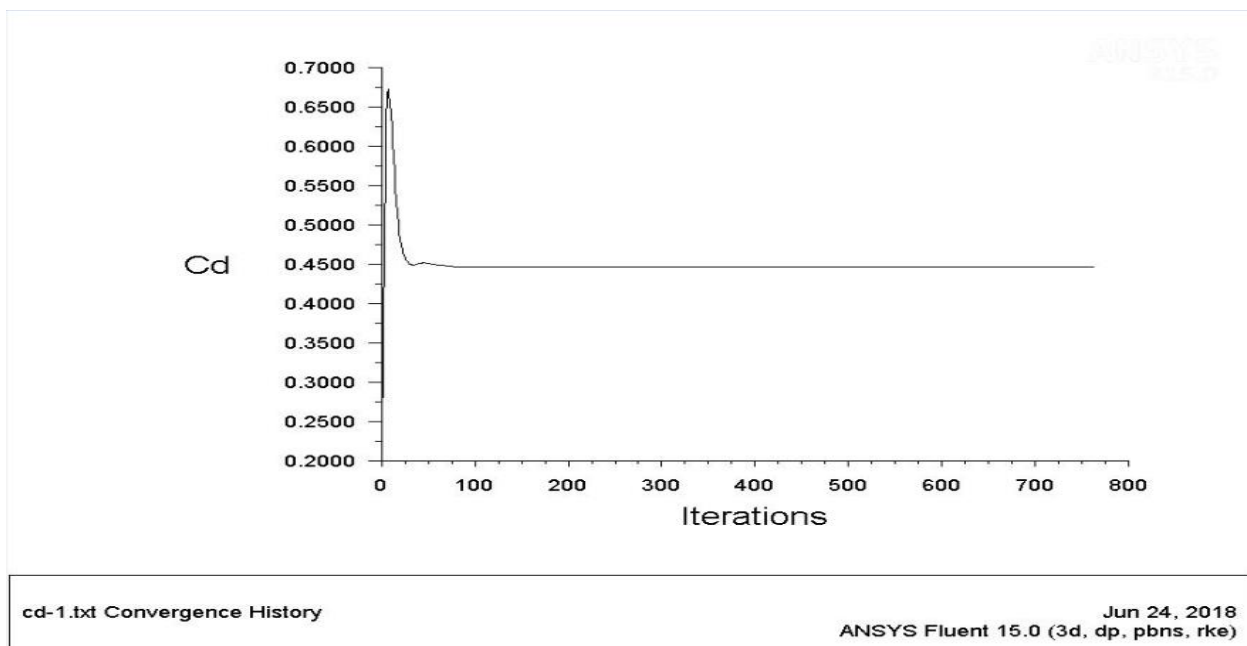


Figure 4.2: Drag coefficent for model one at 80kmph

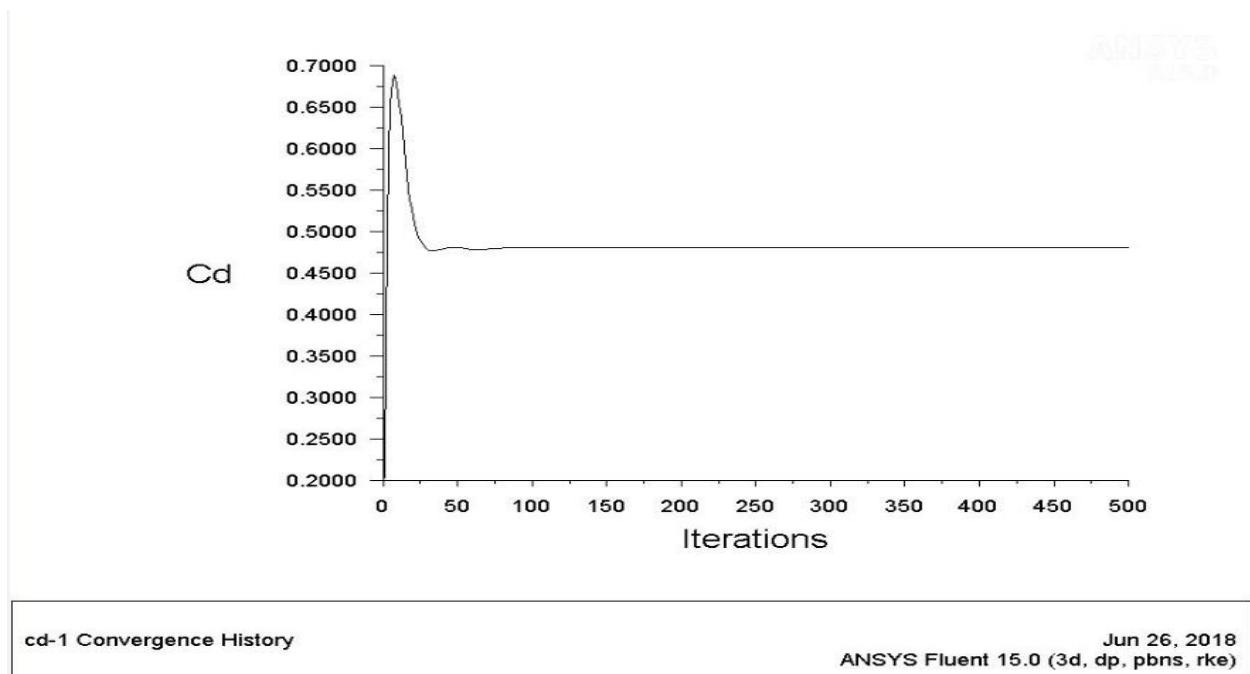


Figure 4.3: Drag coefficient for model one at 100kmph

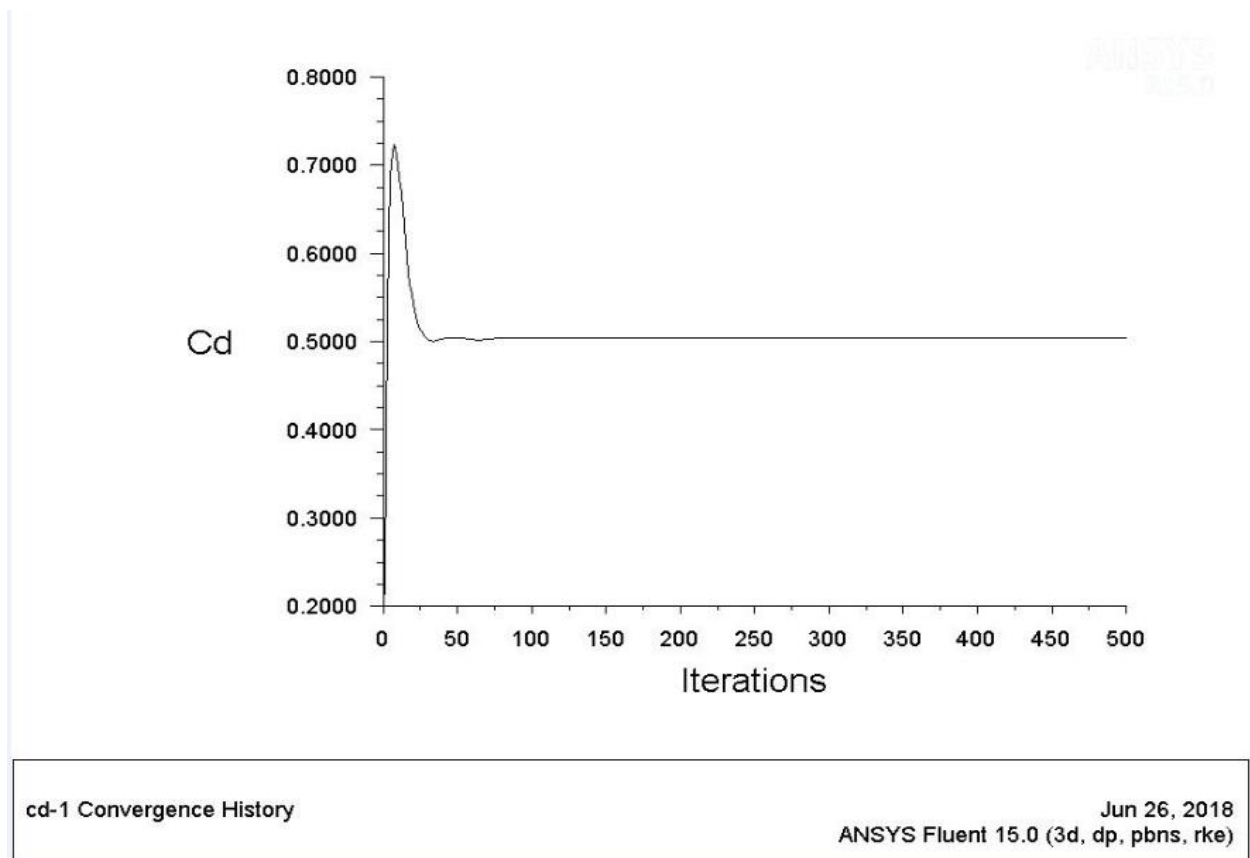
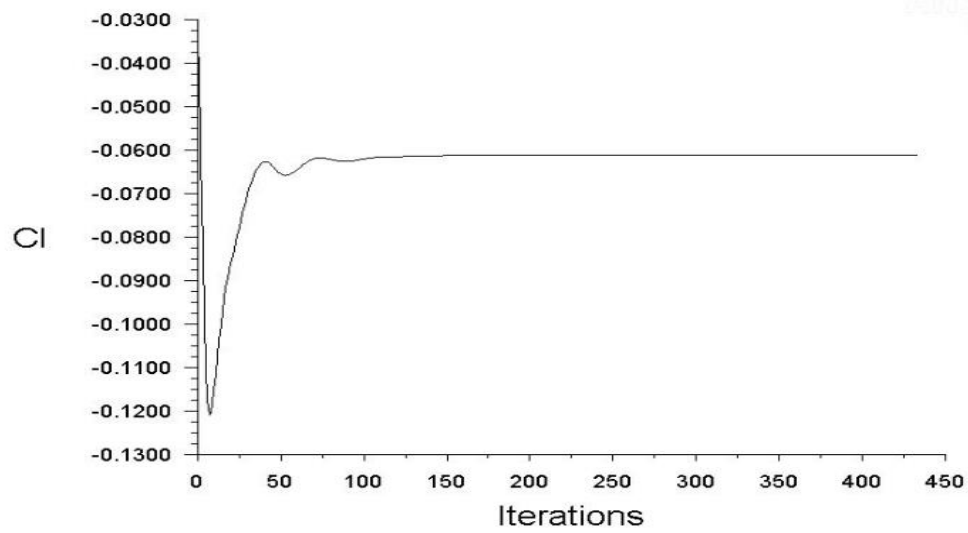


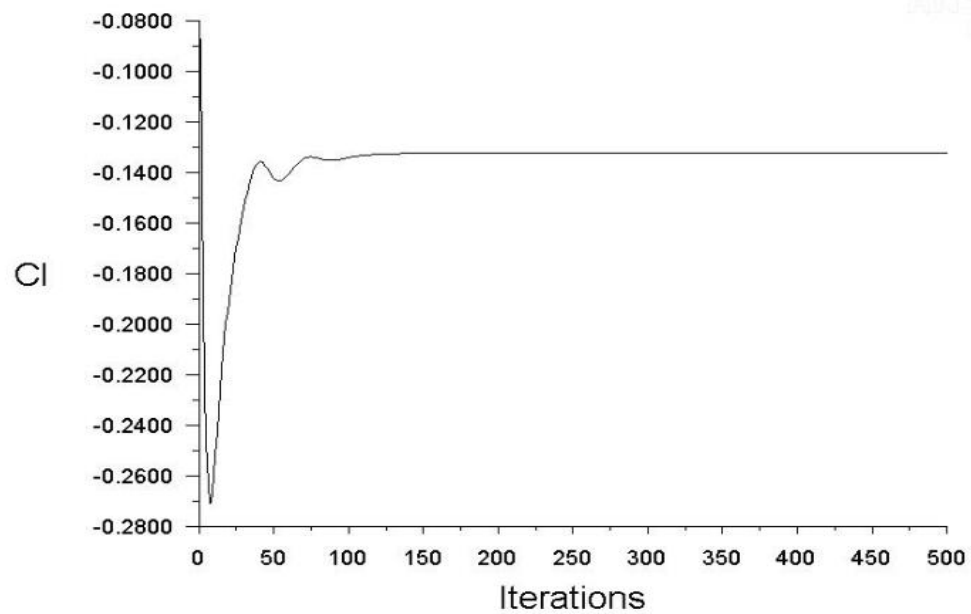
Figure 4. 4: Drag coefficient for model one at 120kmph.



cl-1.txt Convergence History

Jun 30, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

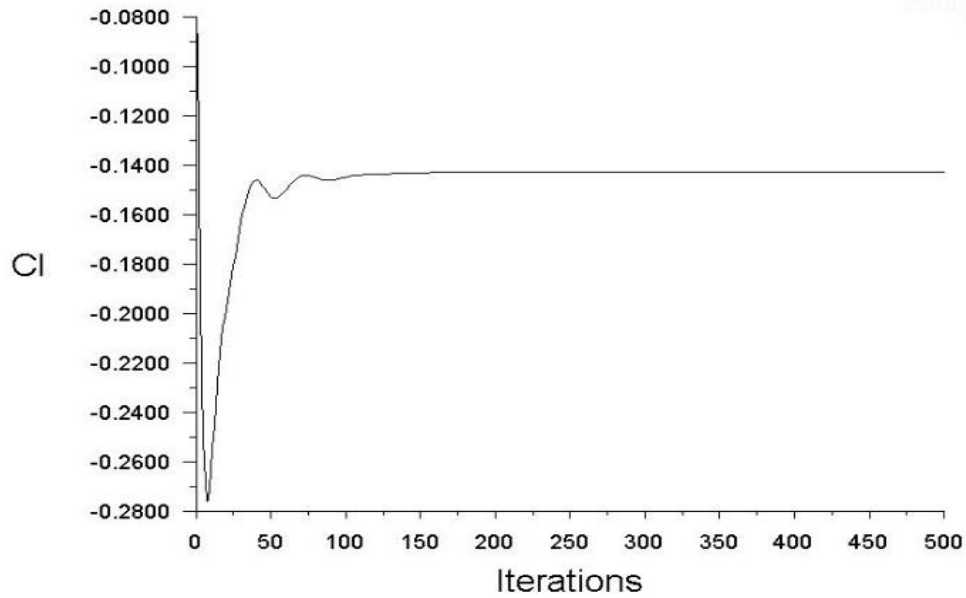
Figure 4. 5: Lift coefficient for model one at 80kmph.



cl-1.txt Convergence History

Jun 29, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure 4. 6: Lift coefficient for model one at 100kmph



cl-1 Convergence History

Jun 26, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure 4.7: Lift coefficient for model one at 120kmph

Table 4. 1: Sumery of lift and drag coefficent for model one at different speed.

Model	Speed(kmph)	Drag coefficient(C_d)	Lift coefficient(C_L)
Model one	80	0.66	-0.120
	100	0.69	-0.132
	120	0.71	-0.142

Table 4.1 shows that the result of the drag and lift at three different speed when simulated the bus model by RKE turbulence model with the converted polyhedral mesh.

To determine the reduction in the drag coefficient from the addition of airvane device and front faring or modifying the front to making streamlined, the drag coefficient, lift coefficient and flow field around the bus without any drag redaction device was examined first. These simulations give the base drag coefficient for the study. After simulate model one the modified model simulate next. After simulate all models compared to the flow field, drag and lift coefficient found in this simulation. To determine the percent reduction of parameter due to the airvane and front faring employed on the bus the drag coefficients were compared. The simulations were completed for all bus model, the polyhedral mesh employing with the RKE turbulence model.

The bus was placed into air flowing with a velocity of 80kmph (22.222 m/s), 100kmph (27.777m/s), and 120kmph (33.333m /s), utilizing a polyhedral mesh and RKE turbulence model the drag coefficient was calculated to be 0.660, 0.69 and 0.71 as shown in figure (4.2, 4.3, 4.4) and the lift coefficient was calculated to be -0.12,-0.132 and -0.142 as shown in the figure above (4.5, 4. 6 and 4.7). These value was the calibration of fluent simulation and high C_D value indicates high drag resistance force and high negative value of the lift coefficient C_L shows the vehicle have high down ward lift which is very necessary for vehicles stability. In general for heavy vehicle the lift is much not affected, due to its mass. The value of drag and negative lift increase as the speed increase as shown in figure 4.8 bellow.

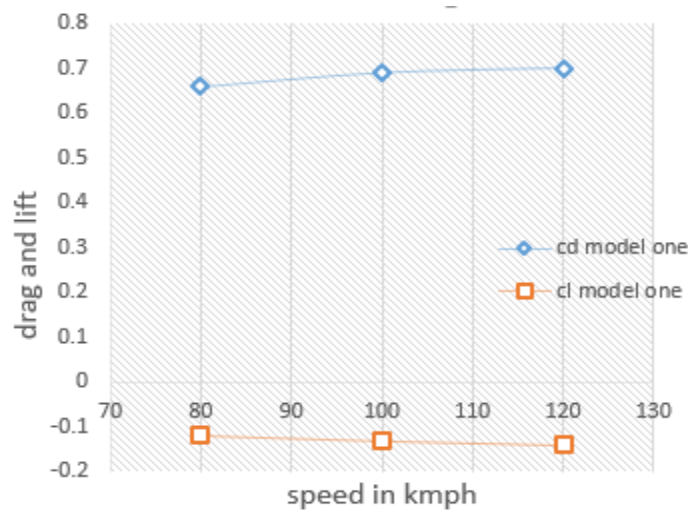
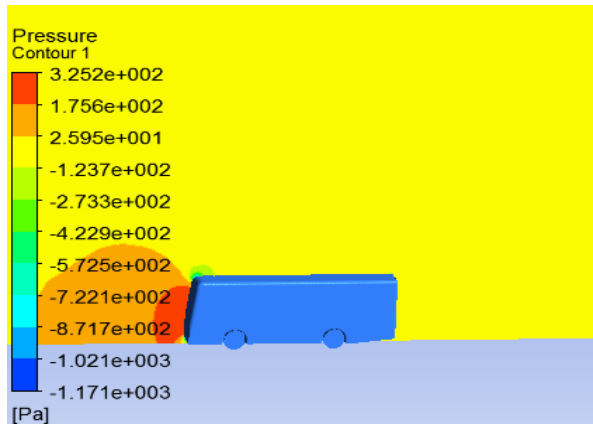
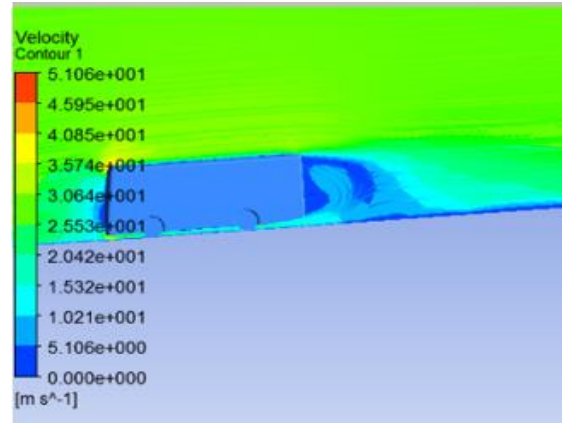


Figure 4. 8: Drag and lift vs speed of the vehicle

As noted in the literature section of this paper the drag coefficients of current commercial buses range from $0.4 \leq C_d \leq 0.8$, therefore the drag coefficient found in this simulation is almost accurate for a Luxury mid bus. Figure 4.69 shows that, as the speed increase the drag coefficient increase, which means the drag resistance force increase and also increase downward lift as shown in the figure 4.69. The fashion of this graph is the same as figure 1.3 which is relation between speeds of the vehicle and drag resistance, we can say that the simulation was almost correct.

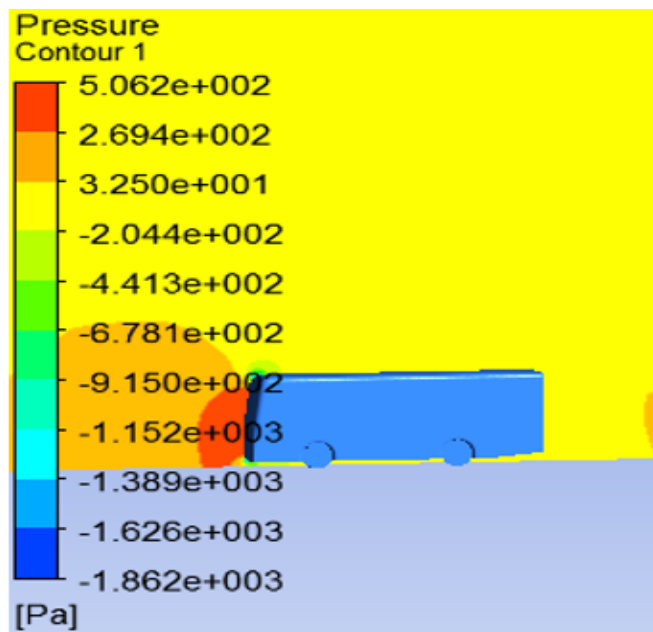


(a)

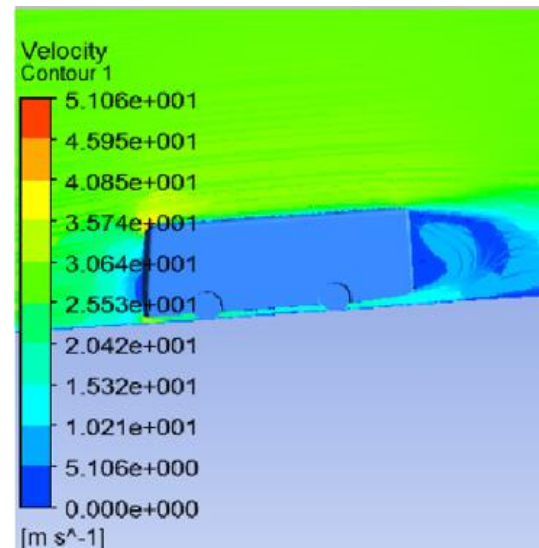


(b)

Figure 4. 9:Aerodynamic pressure and velocity countour for model one at 80kmph.



(a)



(B)

Figure 4.10: Graph of pressure and velocity distribution for model one at 100kmph.

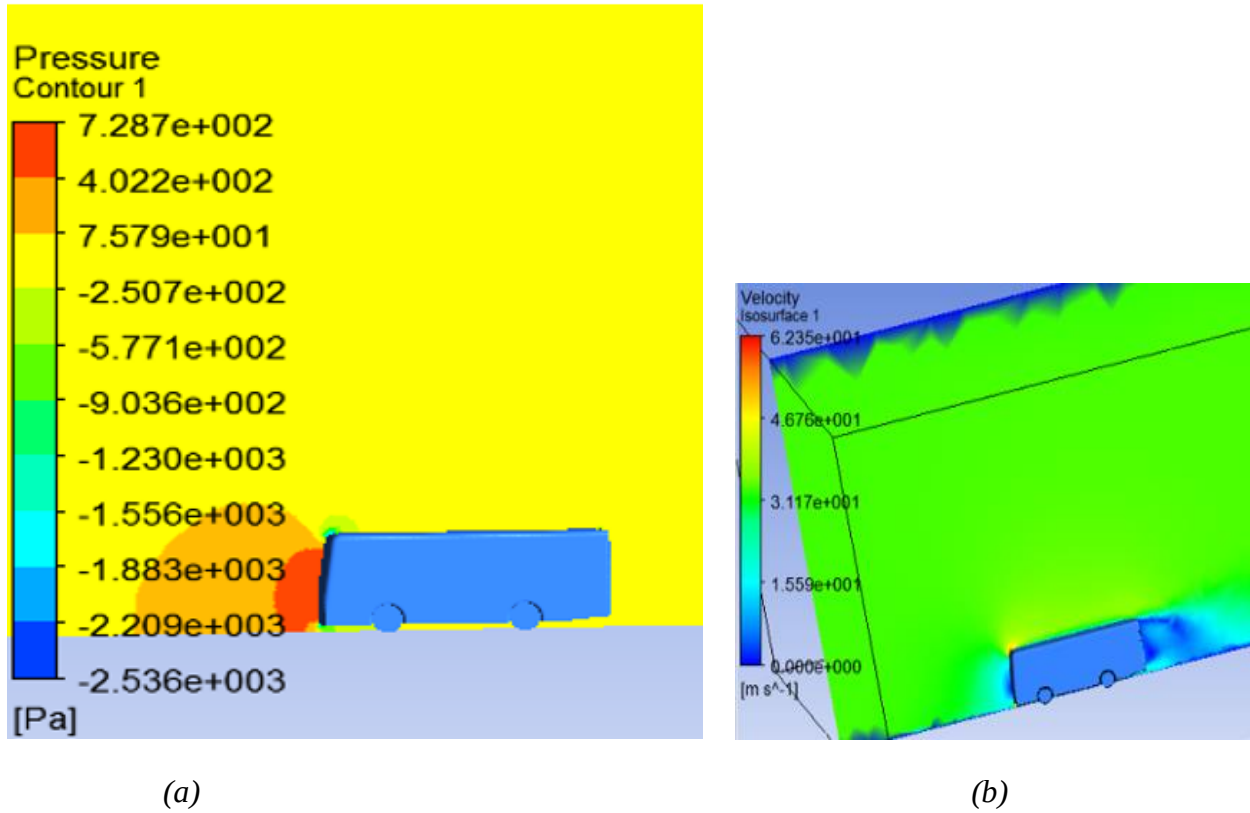


Figure 4.11: Graph of pressure and velocity distribution for model one at 120kmph.

Analyzing the flow field around the bus, due to motion of the vehicle, it was important to look at the velocity field present on the bus. This analysis was able to provide important information about the turbulence and air flow present on the bus. Figure (4.9, 4.10 and 4.11) above illustrates the velocity contour found using Fluent for the bus without any drag reduction device employed at three different speed respectively. A legend can be observed in those Figure, representing the magnitude of the velocity of the air flow in m/s, with colors corresponding to the velocity contour shown. It should be noted that the velocity impeding the motion of the bus was 22.222 m/s, 27.77m/s and 33.33m/s the maximum velocity create for each model in the flow field due to flow separation the value was nearly 51.06m/s, 51.06m/s and 62.35m/s and the minimum velocity was 0m/s respectively it means stagnant air as shown in figure (4.9, 4.10 and 4.11) respectively. The colors indicate the velocity magnitude. The green color in the contour represent the velocities close to the input velocity, meaning that the bus's geometry had no effect on the fluid flow in that area of the flow field.

Sudden changes in geometry created sections of maximum velocities. Such sections can be seen around the top and bottom horizontal edges of the windward face. Red color represents high velocity areas of the contour, were surrounded by layers of slowly decreasing velocities, the orange through yellow colored sections in the contour. This was continued until, farther away from the bus, the velocity evened out at the magnitude of the input velocity. Increase the velocity from the input velocity to the maximum velocity causes separation bubbles along the rearward extending

surfaces, which can be observed in Figure 4.12, below. Separation bubbles, as described in the literature section, are types of turbulence and thus impede motion and increase the drag present on the bus.

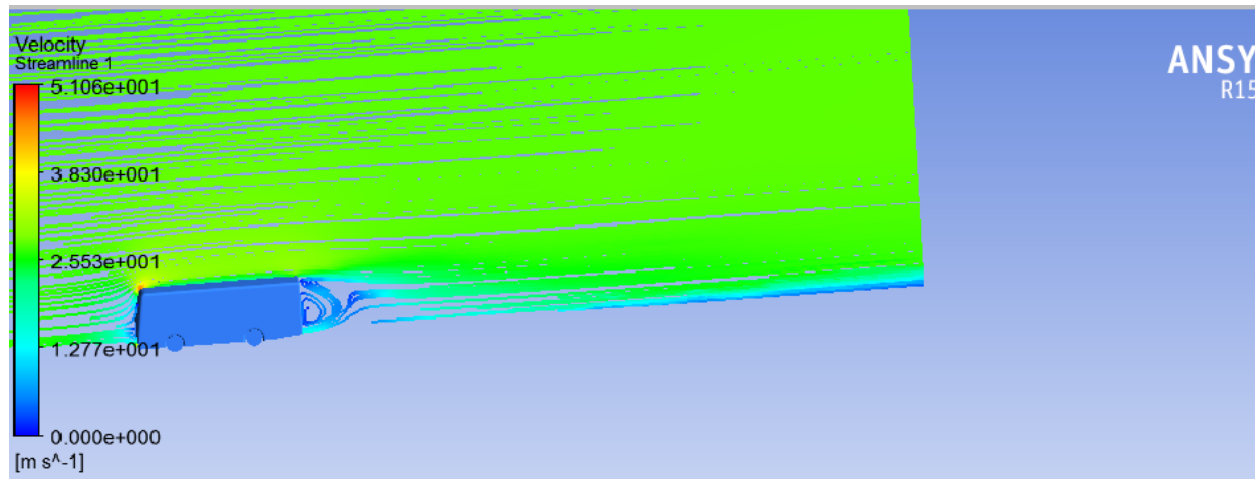


Figure 4.12: Velocity contour for model one on a streamline at 100kmph.

Blue color sections in the contour represent low velocity areas present on the body. Low velocity area on the bus was called the stagnation point and was present on the windward face of the bus. And also the other major area of low velocity is in the wake behind the bus as shown in figure (4.9, 4.10 and 4.11). This low velocity area produce high levels of turbulence and this turbulence increased the drag coefficient present on the bus shown in figure 4.12. Wide area of low velocity zone indicate large wake reign which creates high drag force on the vehicle.

Pressure drag present on the bus constitutes for the largest percent of aerodynamic drag impeding the motion of the bus so we can neglected friction drag force and we assume that the body of the bus was highly smooth. Therefore it was important to analyze the high and low areas of pressure present on the bus and how these sections affected the air flow and coefficient of drag. Figure (4.8, 4. 10 and 4.11) contains the pressure contour that was produced for the model bus without any drag redaction employed when a simulation with a polyhedral mesh and RKE turbulence model was used. The stagnation point which, as described above, was the point on the windward face which had a velocity of 0 m/s can be seen as the area of high pressure, red section. For more clearness illustrates in figure 4.13, below.

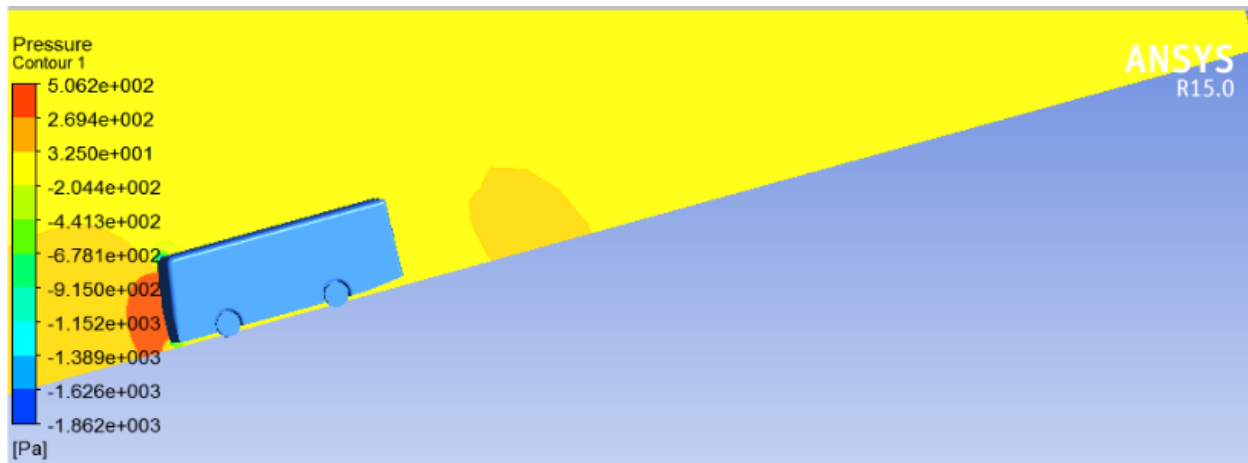


Figure 4.13: Pressure contour of model one at 100 kmph

The magnitude of this high pressure area was approximately 506.2 Pa , with each subsequent layer decreasing in pressure until the ambient pressure, approximately 20 Pa , was reached. The pressure behind the bus, on the leeward face, was at the ambient pressure of 20 Pa , as can be seen by the yellow color. This significant difference in pressure, between the windward and leeward faces of the bus, created a large pressure drag described in the introduction section. This large pressure differential caused a suction-like vacuum behind the bus that caused the pressure drag and thus increased the drag coefficient of the vehicle. These total pressure or relative pressure difference was calculated by maximum pressure on the windward face minus minimum pressure created on the wake region which was 486, 665 and 953 as shown in appendix C figure (C.1, C.2 and C.2) and observe the relation of total pressure with speed in figure 4.14 bellow.

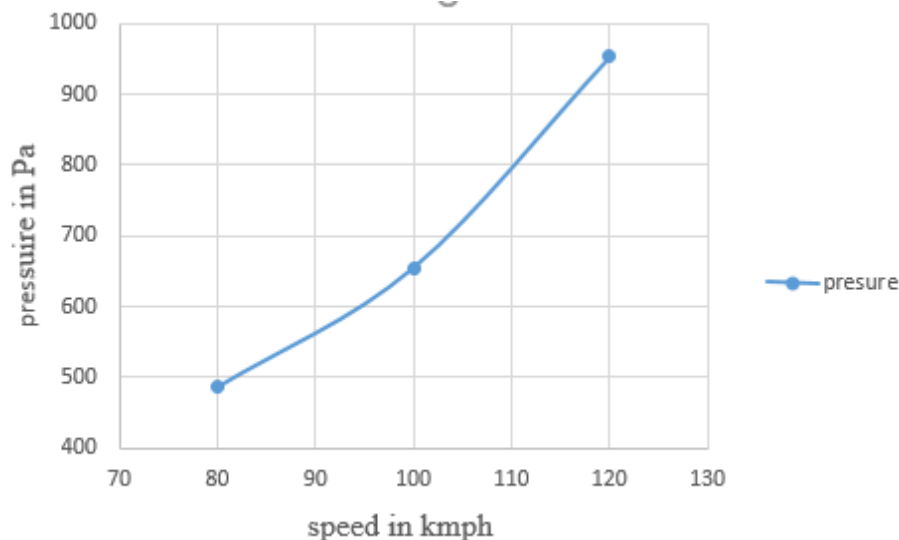


Figure 4. 14: Relation of total pressure with speed of the vehicle for model one.

Figure 4.14 shows that total pressure increase with increase the speed of the vehicle. The pressure present near the separation bubbles, that were caused by the high velocities near the top and

bottom horizontal edges of windward face, were slightly lower than the ambient pressure in the wind tunnel box. The magnitude of this low pressure was approximately -186.2 Pa, as shown by the green sections of the contour in figure 4.12 which creates lift and flow detachment.

The last property that was important to examine was the turbulence kinetic energy. As shown in figure 4.15 below, at the wake region behind the rear of the model creates turbulence due to this create turbulence kinetic energy. The bottom of the bus along with the wheels had further turbulence, increasing at the rear of the bus. This turbulence kinetic energy added to the turbulence in the wake of the bus. The main area of turbulent kinetic energy was found to be in the wake behind the bus. This turbulence kinetic energy produce negative adverse pressure.

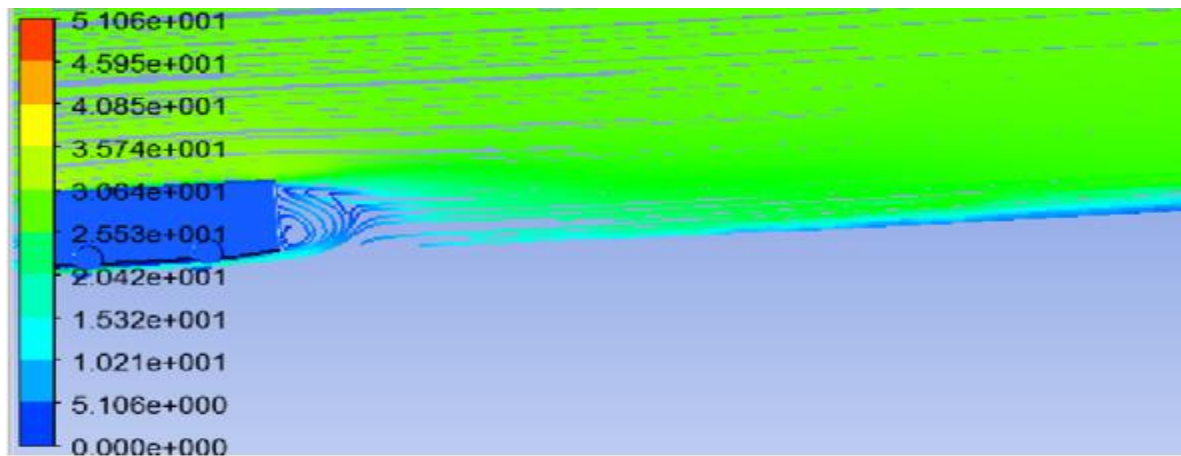


Figure 4.15: vortex generated at the rear of the bus model.

Figure 4.15 shows vortex generated at the end of the bus model which produce low pressure at that point

4.1.2. Grid independent test

Generally grid independent means the result value is not changed when the mesh size decrease. In this study in the relevance center use medium because the result value was not that much vary when simulate by using fine mesh in the relevance center. Generally fine mesh indicates high accuracy and needs high computational time, while coarse mesh needs less computational time and low accuracy result but medium mesh is an appropriate computational time and accuracy.

In model one at a speed of 80kmph simulate using both medium and fine mesh in the relevance center, the result show that almost similar magnitude was calibrated.

When model one simulate at medium mesh size, minimum element size was $1.0507 \times 10^{-3}\text{m}$, maximum element size was 0.13448m , minimum edge length was $3.4725 \times 10^{-4}\text{m}$, number of node was 178802 and number of element was 950809. Further detail in appendix B (Table B-1). The drag and lift coefficient was 0.66 and -0.12 at 80kmph as shown in figure (4.2 and 4.5). And this model simulate at fine mesh the minimum element size was $6.3895 \times 10^{-4}\text{m}$, maximum element size was $8.1785 \times 10^{-2}\text{m}$, minimum edge length was $3.725 \times 10^{-5}\text{m}$, number of node was

226334 and number of element was 1211821. Further detail in appendix B (Table B-5). The drag and lift coefficient was 0.673 and -0.175 at 80kmph. As shown in appendix C (Figure C-13 and C-14). The percent difference calculated to be 1.3% for drag coefficient and 5% for lift coefficient. This difference value was acceptable due to difference standard.

4.1.3. Validation of the result

There are two validation for this study, the first one is from literature, which is drag coefficient at different speed and flow field around the body. Compare the two parameter from the given literature and the calibration of this study. The second one was using [Ahemed] body 30° slant angle was simulate and calculate, then, compare from on this work simulation which has the same setup.

The first one was compare the figure 1.3 in introduction part and figure of 4.69. Figure 2.2 and figure 4.15 flow simulation for this model they are the same fashion as shown in figure 4.16 (a and b) respectively, bellow.

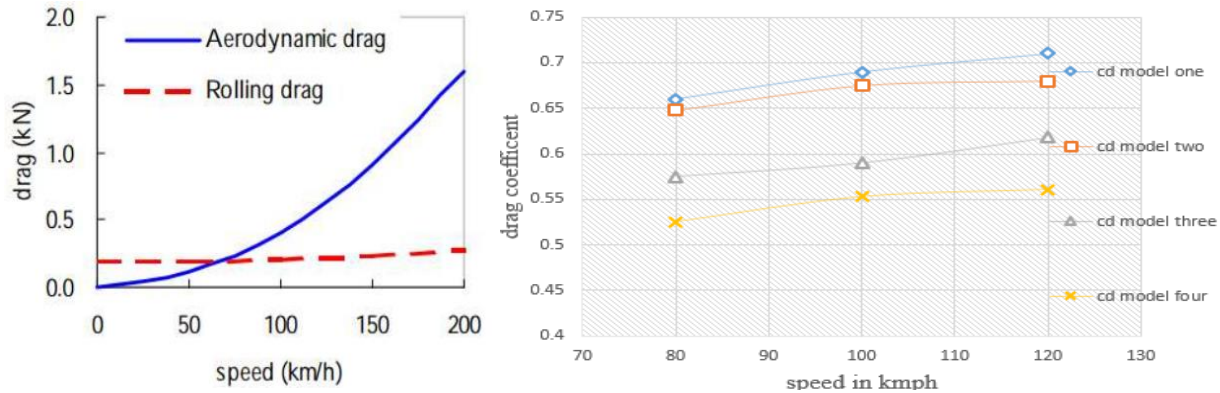


Figure 4.16 (a): Compression of graph of drag vs speed of the bus from literature and calibration of this simulation.

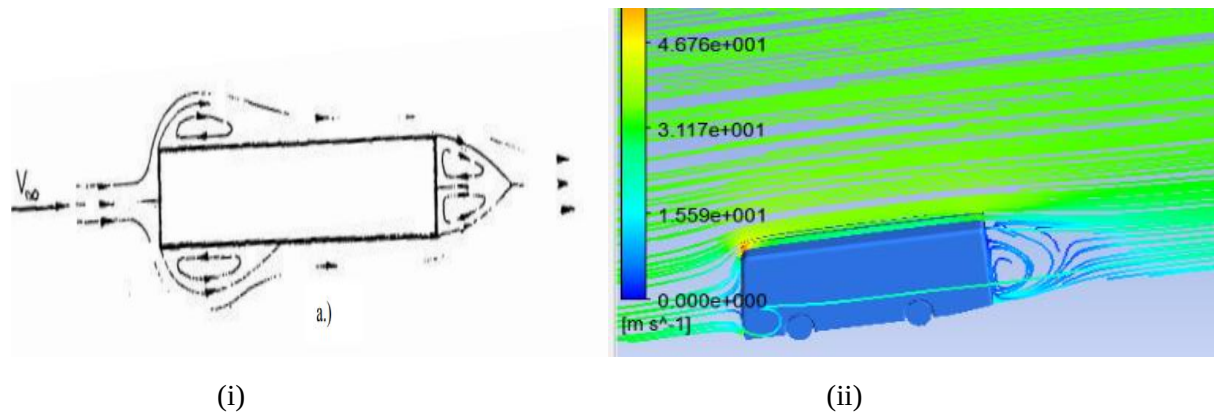


Figure 4.16(b): Compression of graph of flow field around the bus from literature and calibration of this simulation.

In figure 4.19(a) the first two graph was figure 1.3 and figure 4.69 which are drag coefficient vs speed, from the graph observe that they have the same fashion so the simulation was almost correct. And in figure 4.19(b) shows that the flow filed around the bluff body and flow filed around this calibration of the model have the same fashion so the simulation also validated. And also the value of drag coefficient was the range of today bus drag coefficient which was 0.4 to 0.8.

The second part of validation was construct the Ahmed[58] 30° slant angle body import and simulate the same setup like in this work and compare the drag and lift coefficient and total pressure with calibration of this work simulation of this work. The original Ahmed [58] body the drag and lift coefficient was 0.262 and 0.19 when simulate SST K- ω turbulence model with unstructured tetrahedron mesh at a speed of 40 m/s.

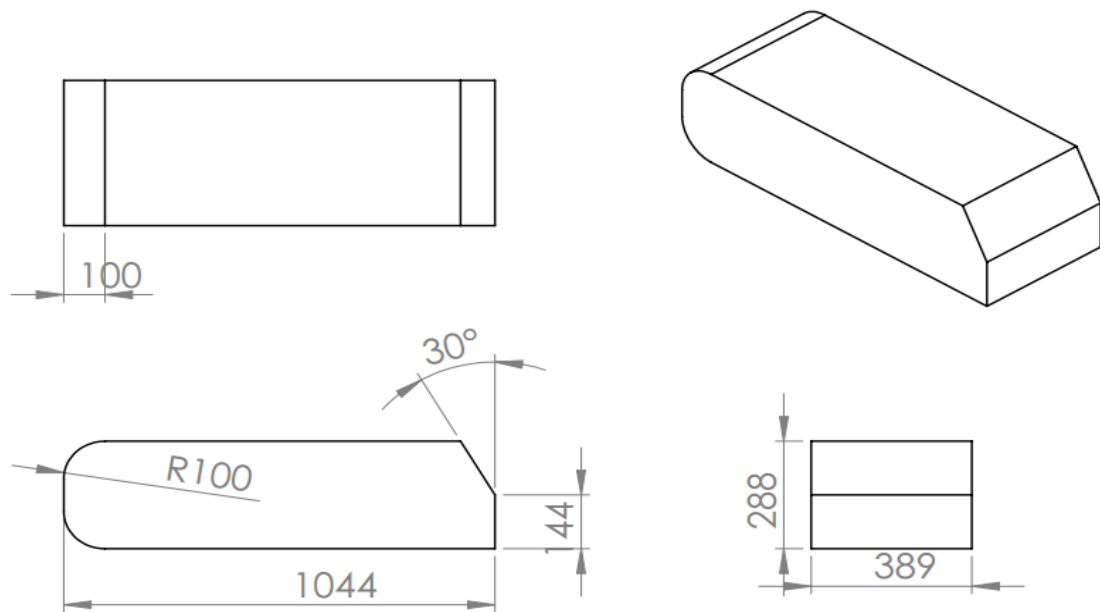


Figure 4. 17: Ahmed body solid work model.

Figure 4.17 was solid work Ahmed [58] body model to simulate for validation purpose.

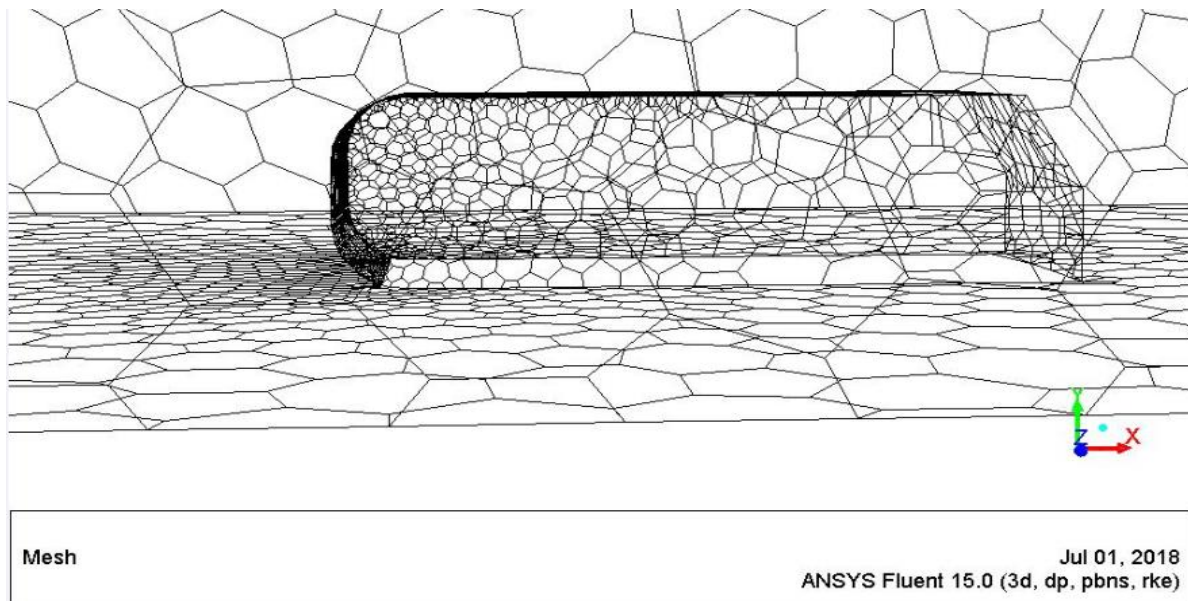


Figure 4. 18: Polyhedral mesh generate on Ahmed body

Figure 4.18 was the result of Ahmed [58] body converted polyhedral mesh generated. All meshing setup was similar with setup simulation of this study.

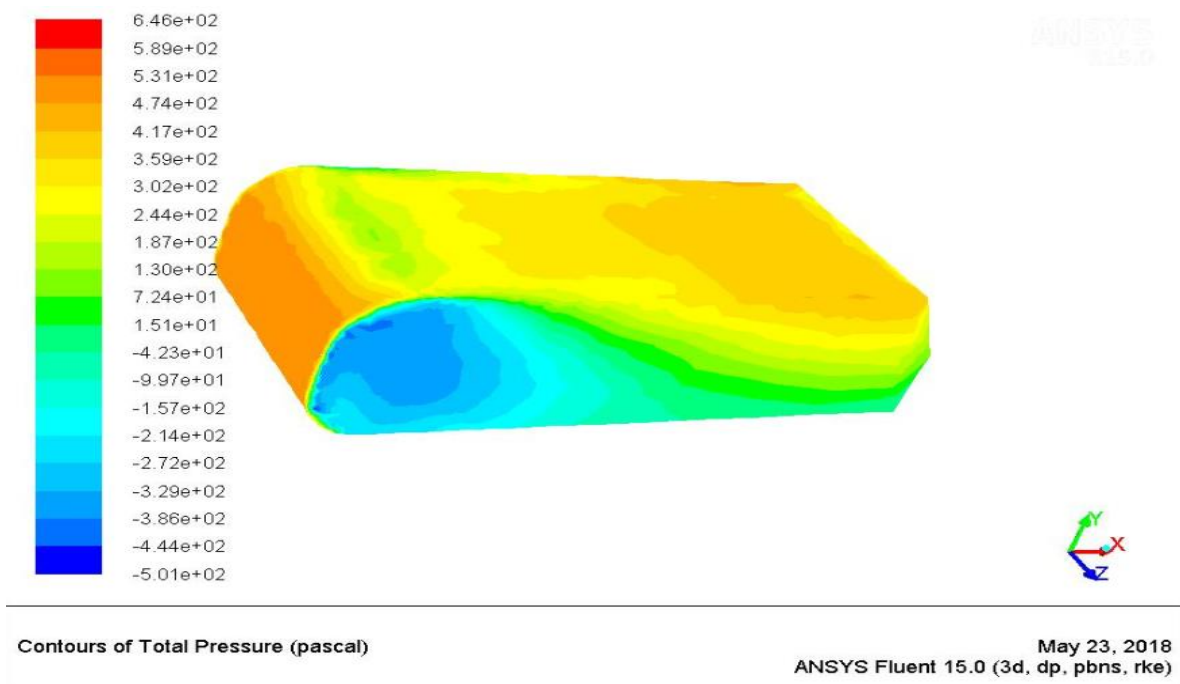
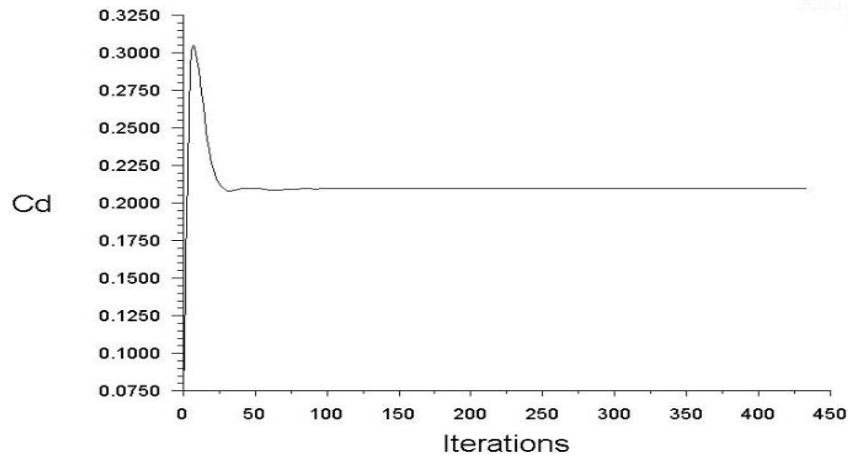


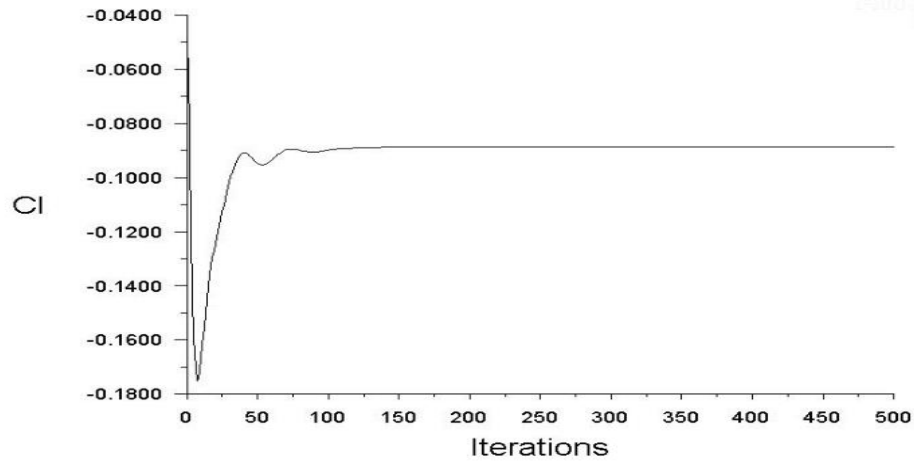
Figure 4. 19: Total pressure on the body



cd-1.txt Convergence History

Jun 30, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure 4. 20: Calibration of drag coefficient on the body



cl-1.txt Convergence History

Jun 30, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure 4. 21: Calibration of lift coefficient on the body

The simulation result shows that use this study setup the drag and lift coefficient was 0.32 and - 0.19 and the total pressure was 646Pa as shown in the figure 4.20, 4.21 and 4.19 respectively at a speed of 40 m/s. The percent difference was calculated to be 5.8% for drag coefficient and 7% for lift coefficient, so the simulation set up was almost correct and acceptable.

4.2. Result and discussion for model two

4.2.1. Result of scaled residual, drag and lift coefficient, pressure and velocity countour at speed of 80kmph 100kmph and 120kmph for model two.

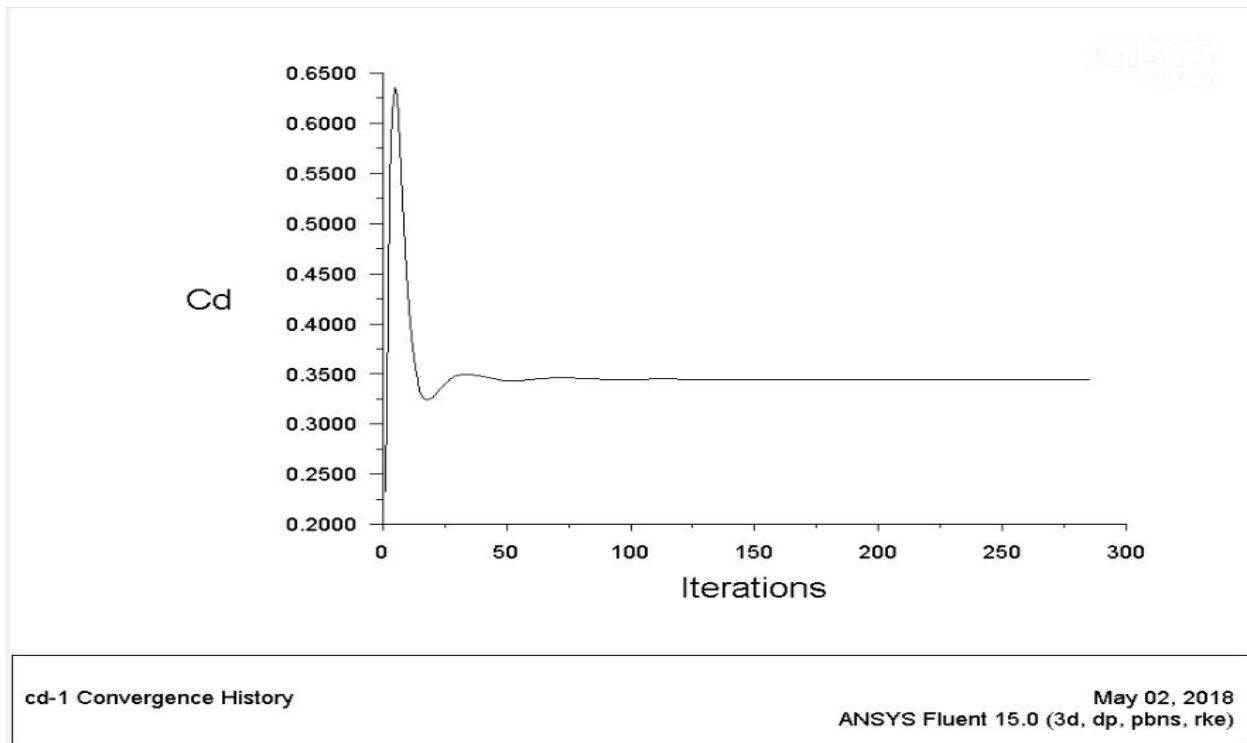


Figure 4. 22: Drag coefficient for model two at 80kmph.

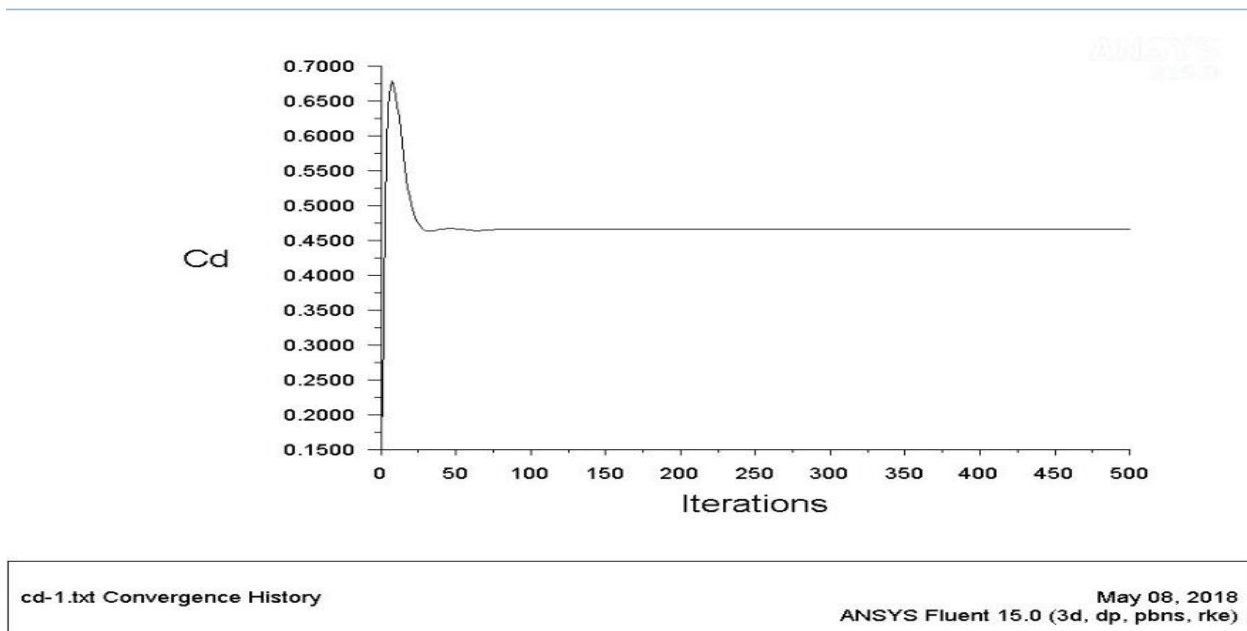


Figure 4. 23: Drag coefficient for model two at 100kmph.

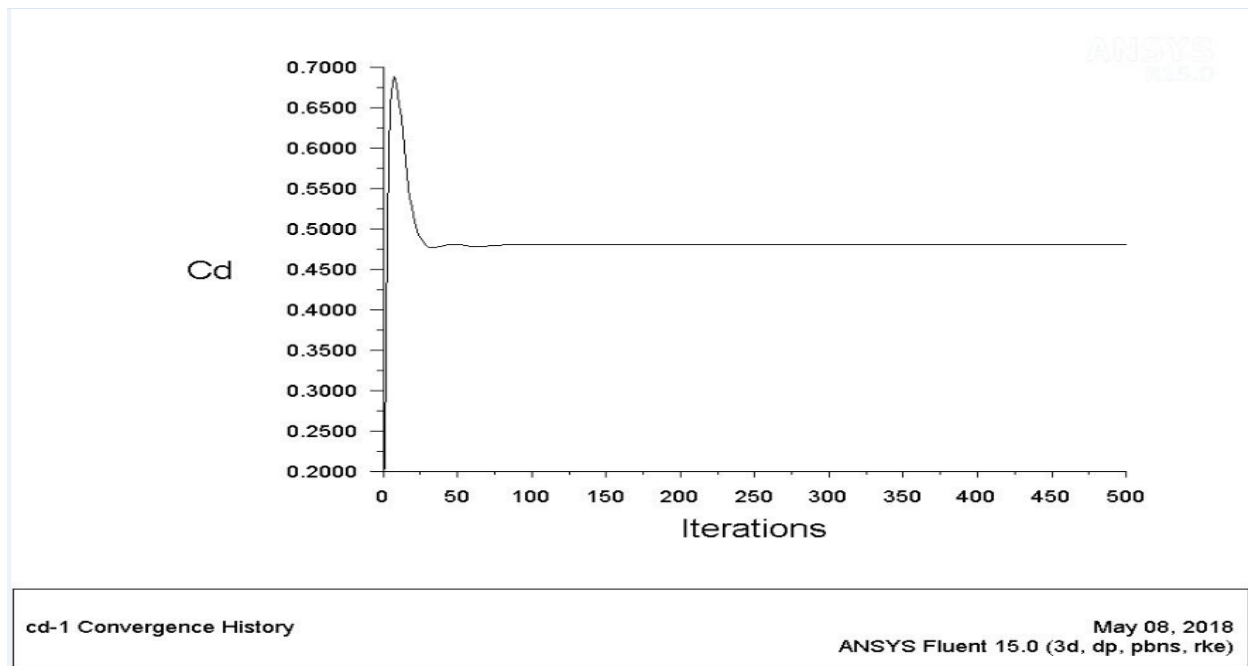


Figure 4. 24: Drag coefficient for model two at 120kmph.

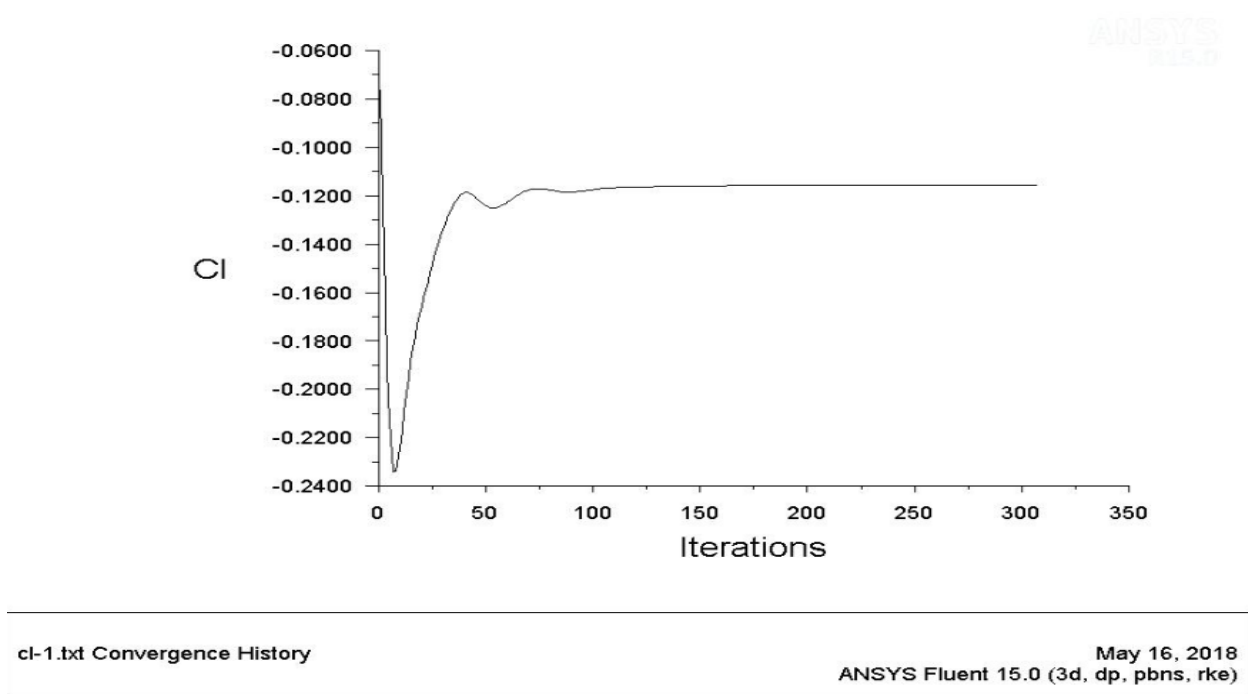
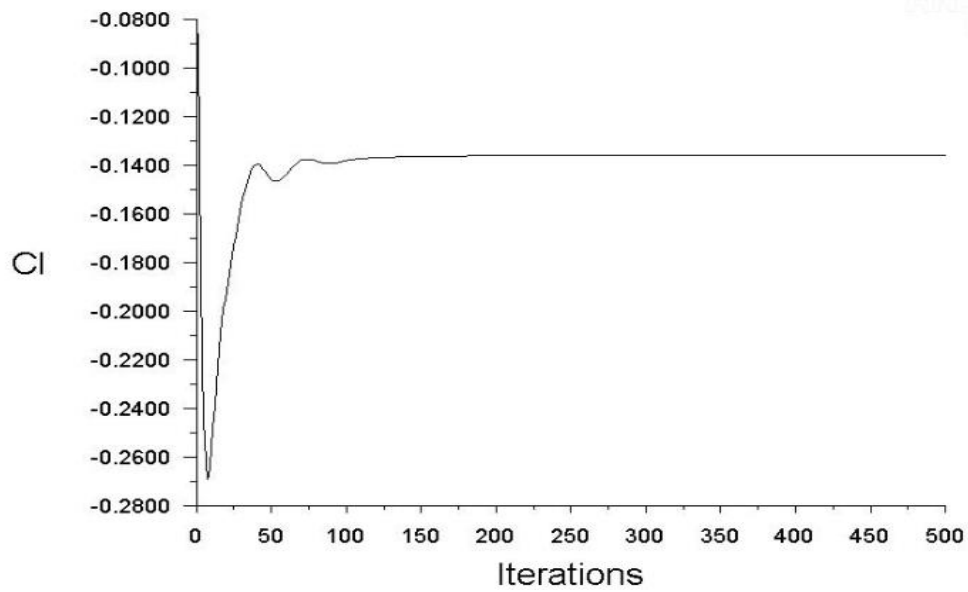


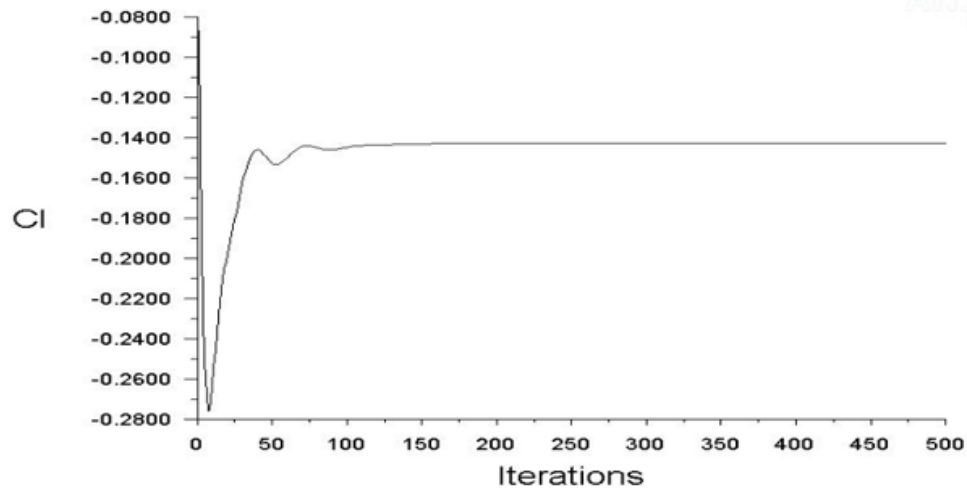
Figure 4. 25: Lift coefficient for model two at 80kmph.



cl-1.txt Convergence History

May 08, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure 4. 26: Lift coefficent for model two at 100kmph.



cl-1 Convergence History

May 08, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure 4. 27: Lift coefficent for model two at 120kmph.

Model two was baseline model attached with airvane device at rearward corner of the model bus using SOLIDWOKS. After construct the model import and calculate the drag and lift coefficient, total pressure and flow file around the bus was tasted at three different speed and see the effect of

the airvane device on these parameter on ANSYS fluent 15.0. The test procedure was the same as model one. The scaled residuals was similar with the baseline model as shown in figure 4.1. The drag and lift coefficients found were compared to model one. The drag coefficients were calculated to be 0.648, 0.675 and 0.68 as shown in figure(4.22, 4.23 and 4.24) and lift coefficients were calculated to be -0.232, -0.269 and -0.275 as shown in figure (4.25, 4.26 and 4.27) at the speed of 80,100 and 120kmph respectively. We can compare the drag and lift coefficient of model two relative to model one as shown in the figure 4.28.

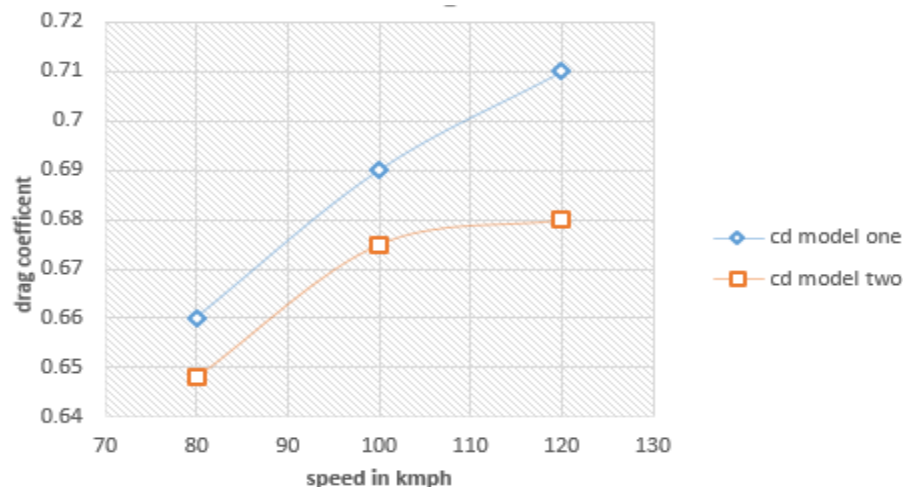


Figure 4. 28: Drag coefficient for model one and model two at three different speed.

In figure 4.28 C_D model one or blue color represent drag coefficient for model one and C_D model two or red color represent drag coefficient for model two. From this observe that the drag coefficient of model two decrease relative to model one which indicates that decrease negative adverse pressure. Redaction of drag coefficient at low speed is low and high at high speed. Percent drag redaction of model two relative to model one were calculated to be 1.6%, 2.174% and 4.225% at 80, 100 and 120kmph respectively and the average reduction was 2.67%. And the downward lift coefficient was increase in general, but the difference of lift coefficient decrease at high speed relative to low speed as shown in figure 4.29, below.

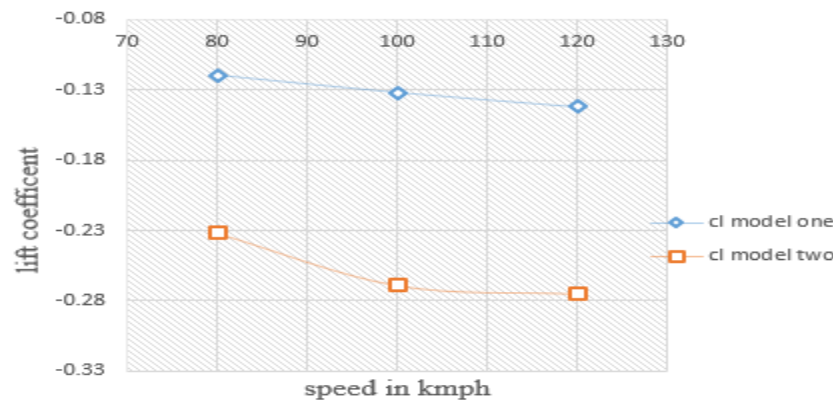


Figure 4.29: Lift coefficient for model one and two vs speed.

In figure 4.29 C_l model one and C_l model two or blue and red color represents lift coefficient for model one and two respectively. Reduction of lift coefficient at low speed is high but small reduction at high speed as shown in figure 4.29 above. The reduction of lift coefficient was -93.3%, -103.78% and -93.66% at 80, 100 and 120kmph respectively. The negative value indicates the downward lift.

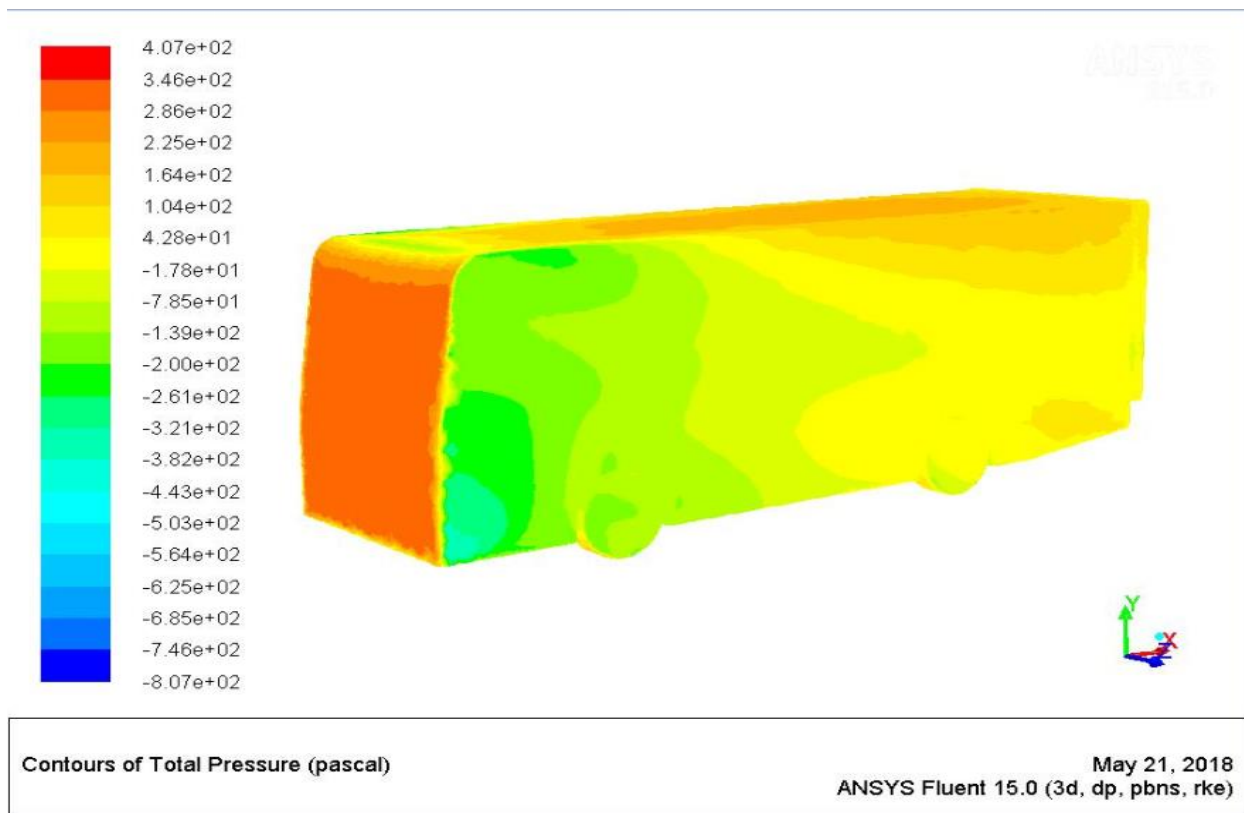


Figure 4. 30:Aerodynamic pressure countour for model two at 80kmph.

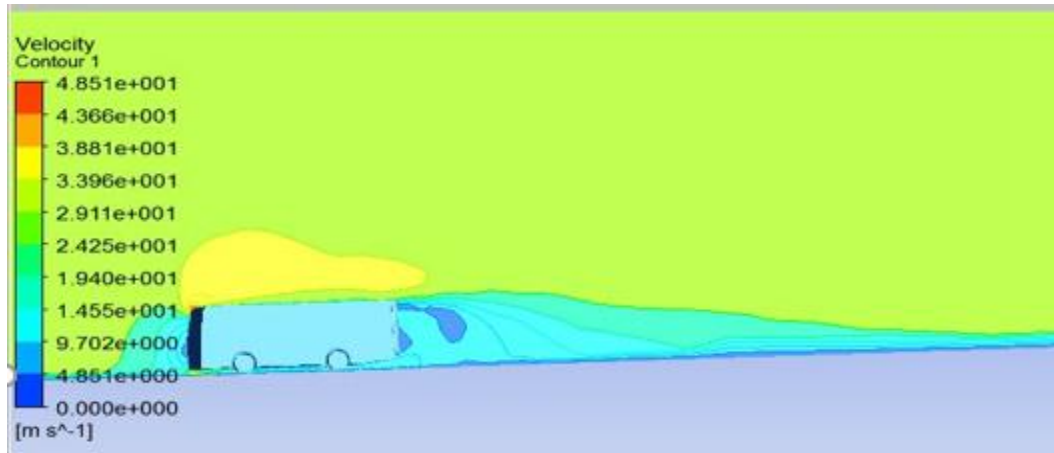


Figure 4. 31: Velocity countour for model two at 80kmph

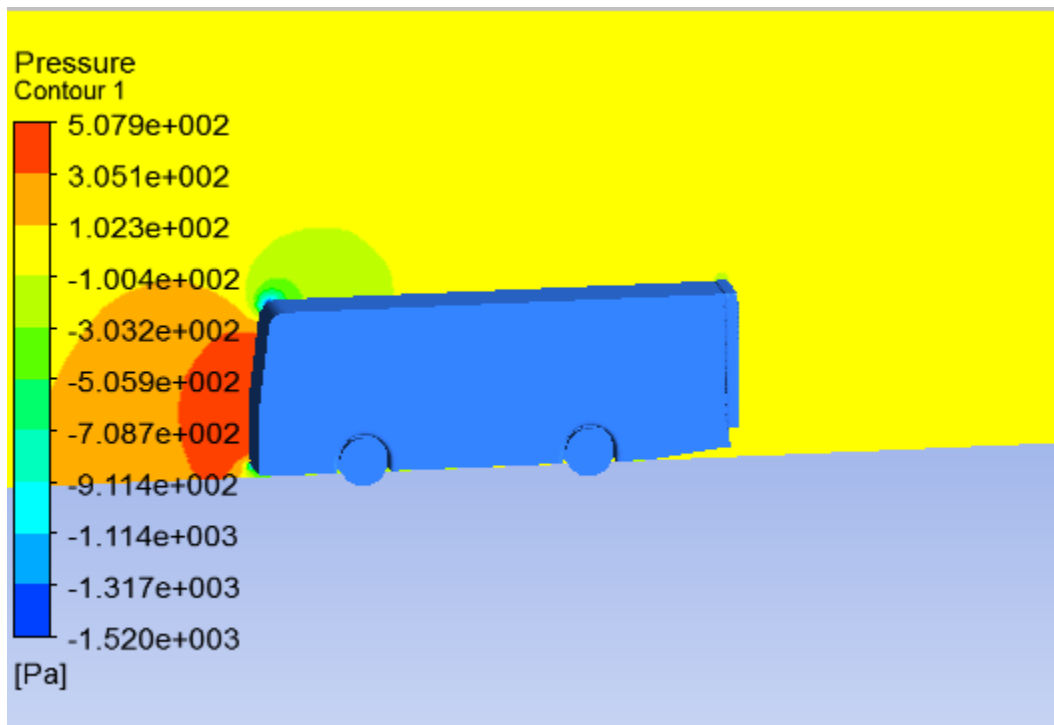


Figure 4. 32: Graph of pressure distribution on model two at 100kmph.

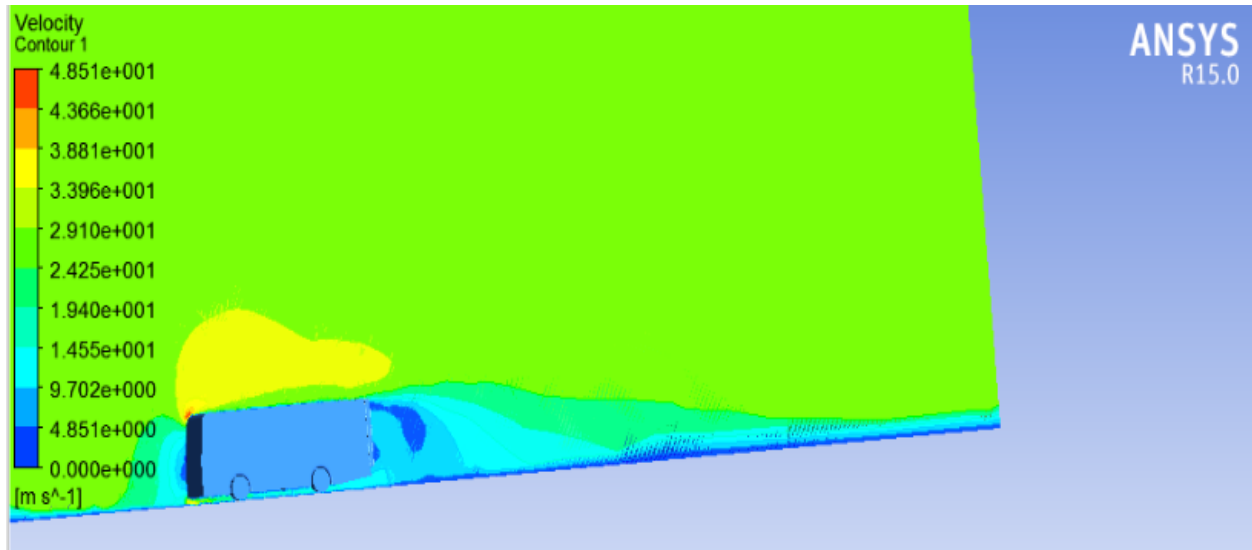


Figure 4. 33: Graph of velocity distribution of model two at 100kmph.

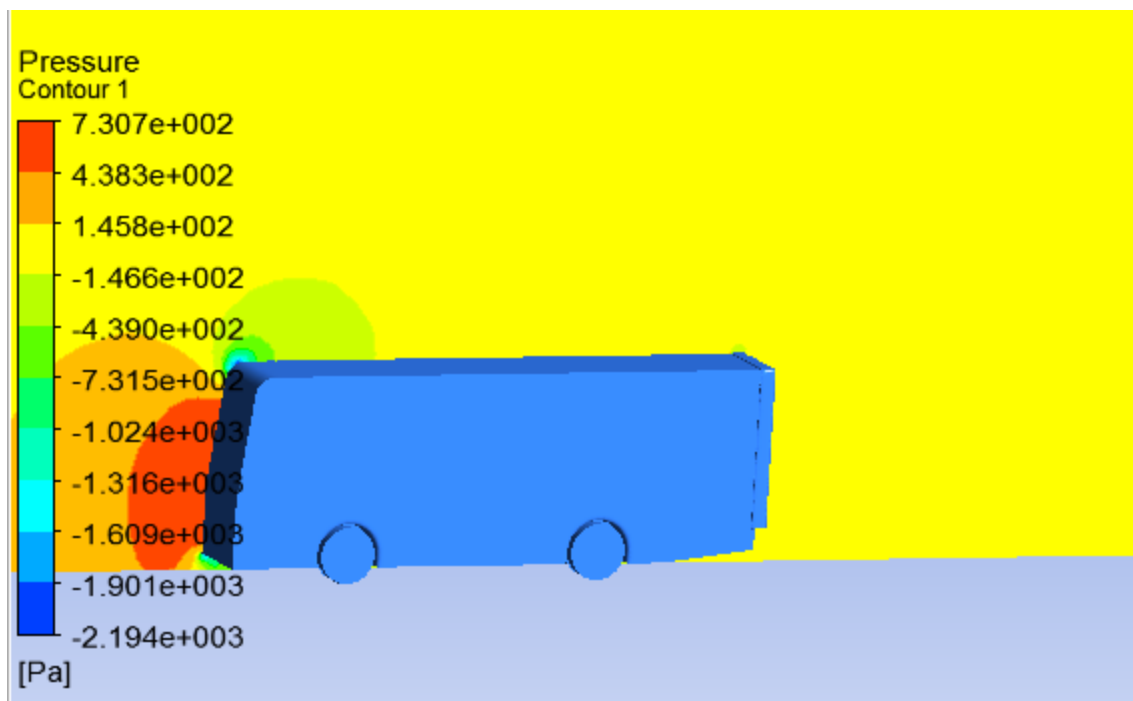


Figure 4. 34: graph of pressure distribution for model two at 120kmph.

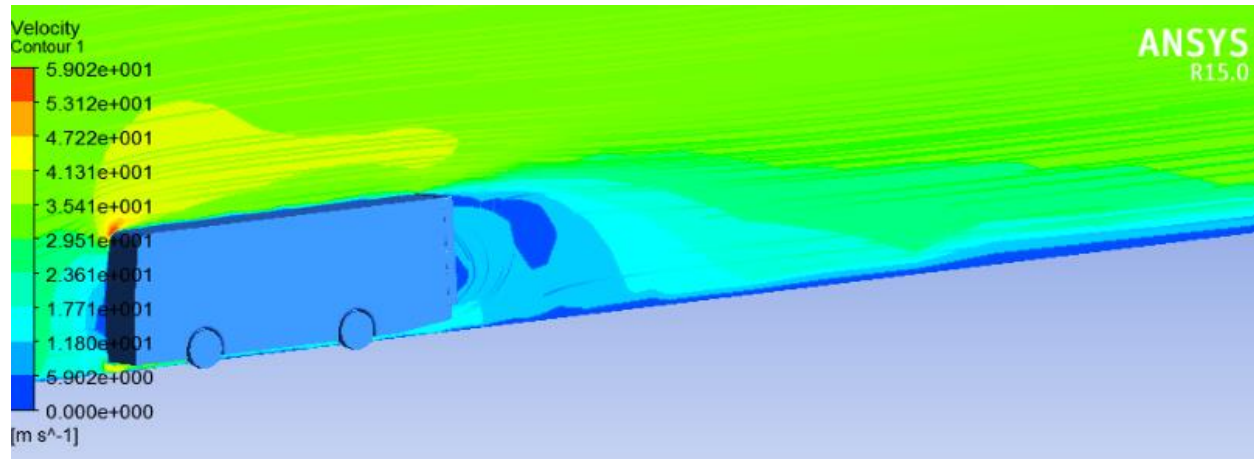


Figure 4. 35: Graph of velocity distribution of model two at 120kmph.

The second parameter to be examined was, how the airvane changed the airflow surrounding the bus to reduce the drag present on the bus, contours of the velocity and pressure were created and are compared to the contours of the airflow around the bus without an airvane attached. Figure (4.31, 4.33 and 4.35) illustrates the velocity contour that was found using the polyhedral mesh with RKE turbulence model for the bus model with an airvane device employed at 80,100 and 120kmph respectively. The different contour shown was at the wake area of the bus due to the fact the velocity contour of the rest of the bus did not change with the addition of the airvane because the airvane device attached on the rearward corner of the bus. This contour can be compared with the velocity contour of the bus model one. From the ANSYS calibration we can observed that minimize the low velocity area at the wake region and delay the turbulence zone away from the bus model. As shown in figure 4.36 and compare with figure 4.15 and figure (4.31, 4.33 and 4.35) with figure (4.9, 4.10 and 4.11).

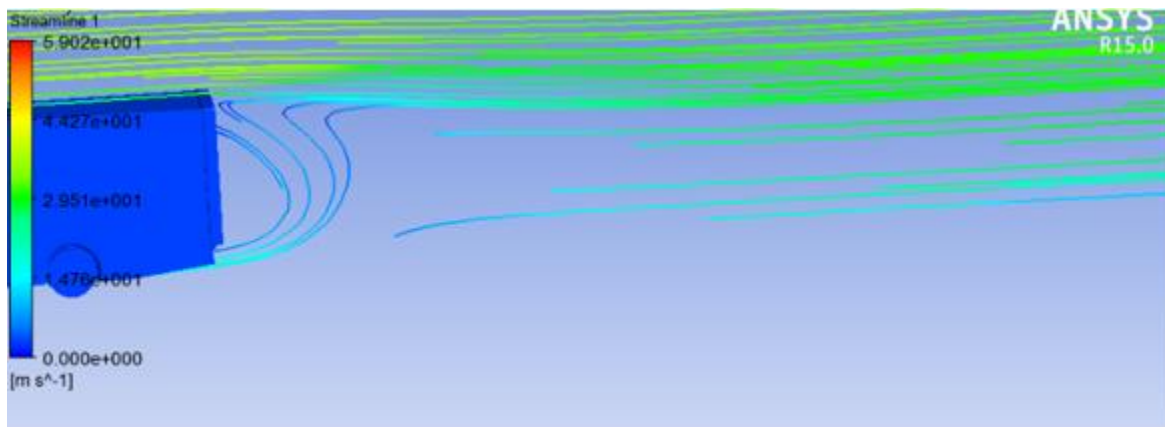


Figure 4. 36: Graph of vortex generated for model two at the rear end of the model.

Due to decrease the wake area to increase the pressure around rear end increase base pressure and delay the separation zone away from the model decrease the total pressure as shown in the figure

(4.30, 4.32 and 4.34) relative to in figure (4.9, 4.10 and 4.11) respectively. The comparison in figure 4.37 bellow.

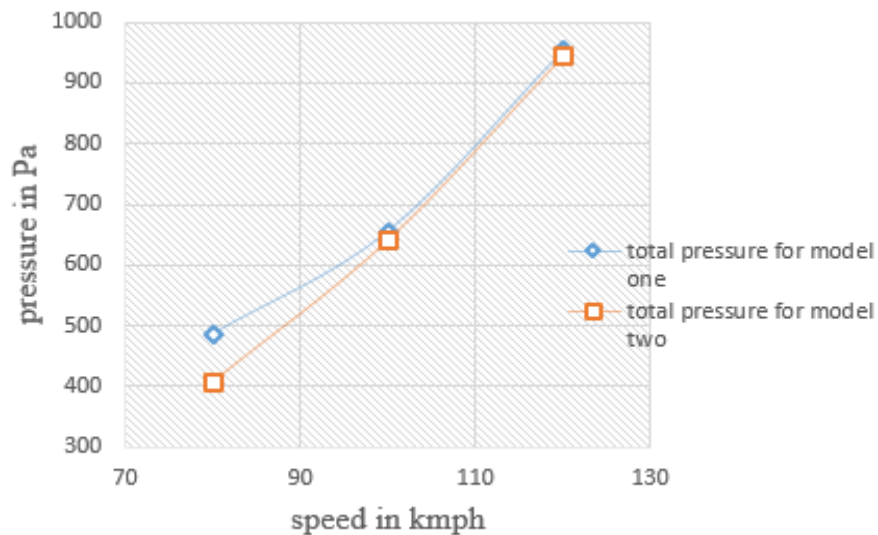


Figure 4. 37: Graph of total pressure vs speed of the vehicle for model one and model two.

In the figure 4.35 total pressure for model one or blue color and total pressure for model two or red color represents total pressure for model one and model two at different speed. From this figure, conclude that decrease the adverse pressure of model two relative to model one. The percent redaction were calculated to be 16.255%, 2.29% and 1.09% at speed of 80,100 and 120kmph respectively. The pressures contour indicates that increase the pressure on the rear end of the bus this increment decrease the adverse pressure or total pressure. This indicated that the turbulence kinetic energy was reduced due to addition of air vane device at the rear end of the model.

4.3. Result of model three.

4.3.1. Result of scaled residual, drag and lift coefficient, pressure and velocity countour at speed of 80kmph 100kmph and 120kmph for model three.

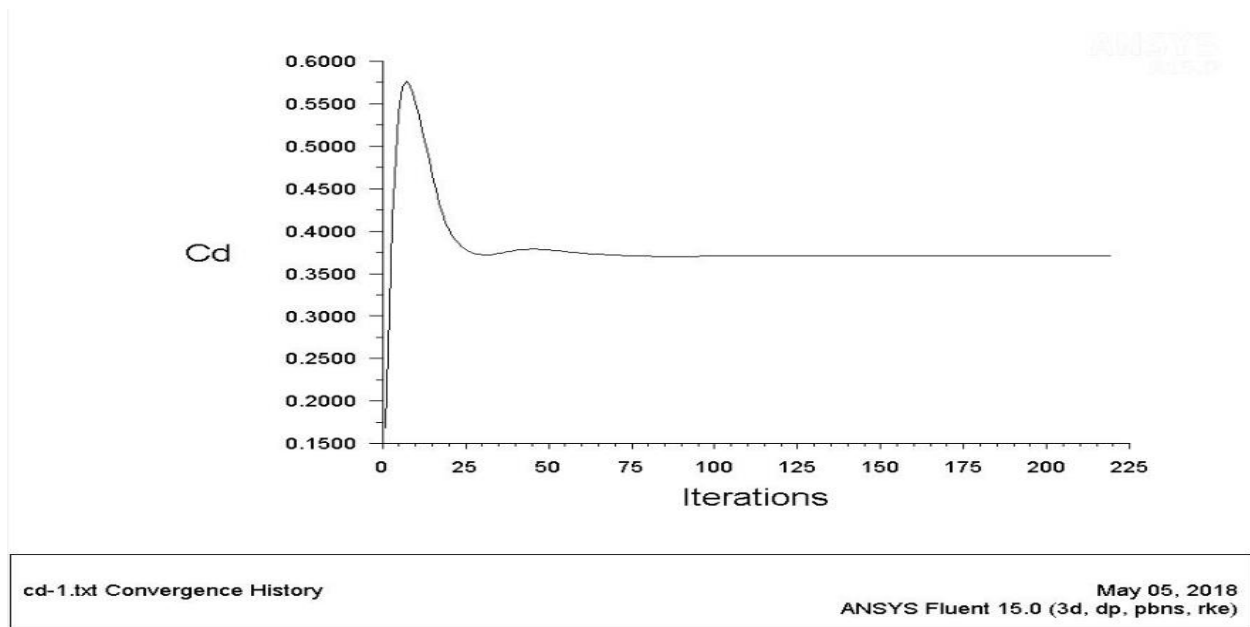


Figure 4. 38: Drag coefficient for model three at 80kmph.

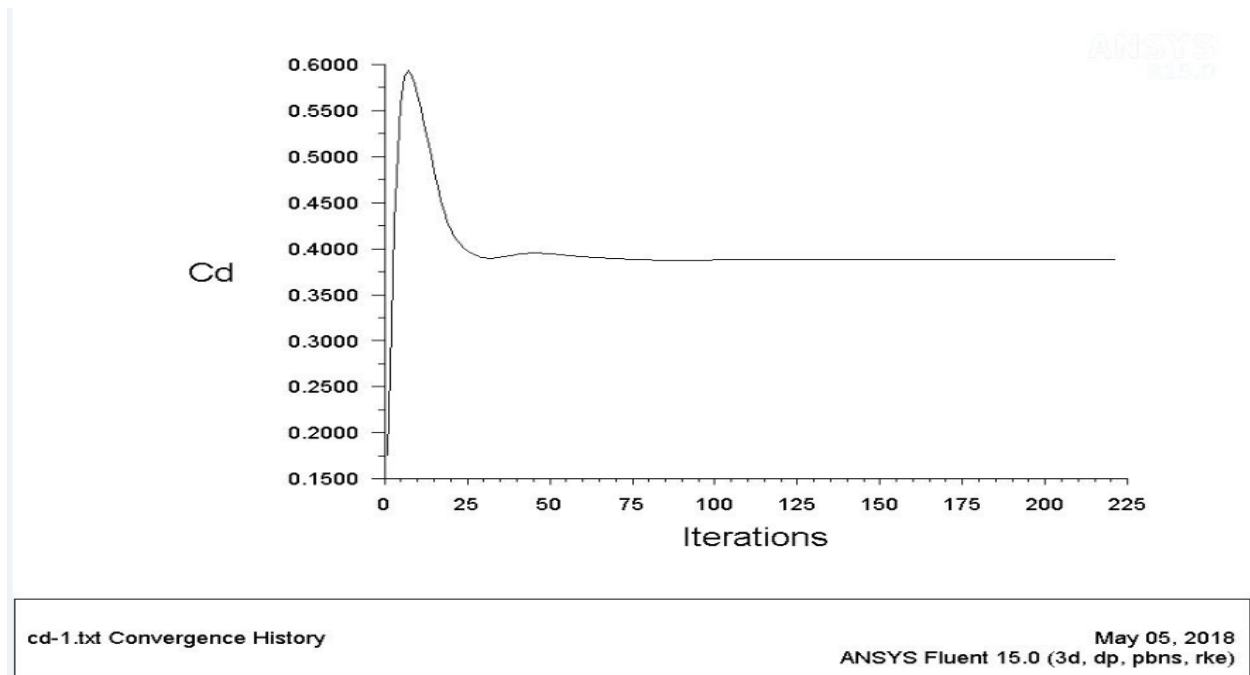


Figure 4. 39: Drag coefficient for model three at 100kmph.

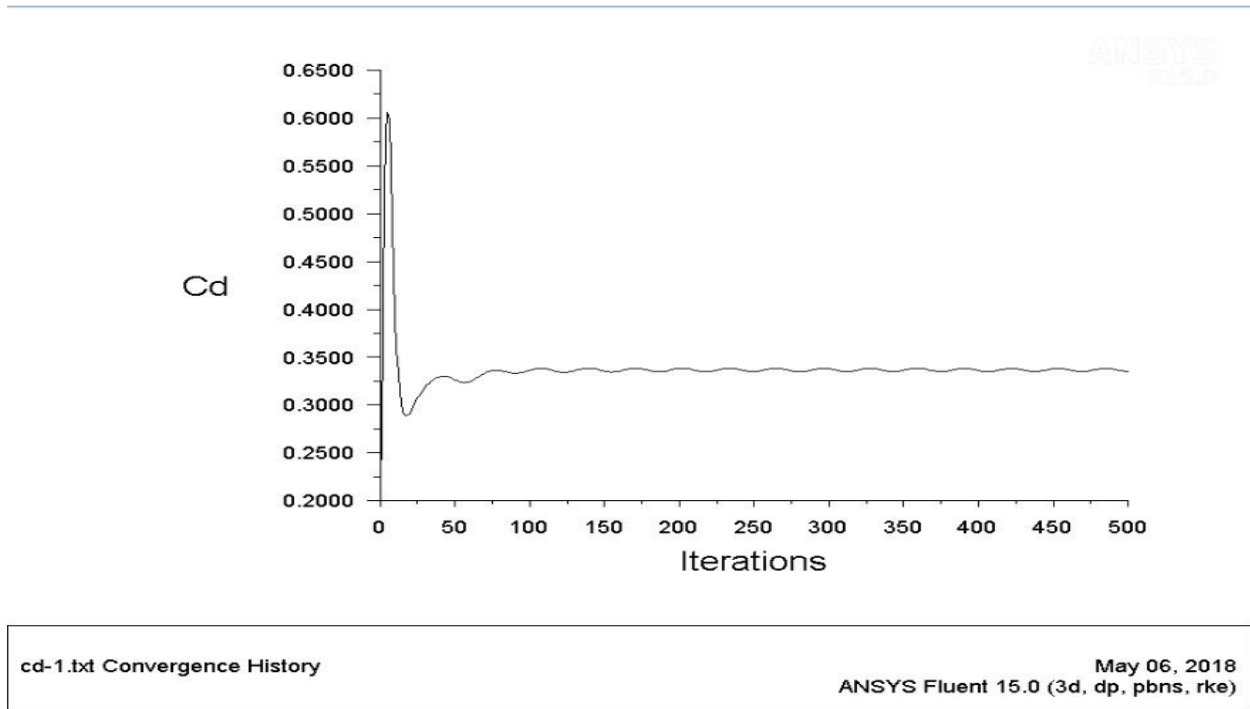


Figure 4. 40: Drag coefficient for model three at 120kmph.

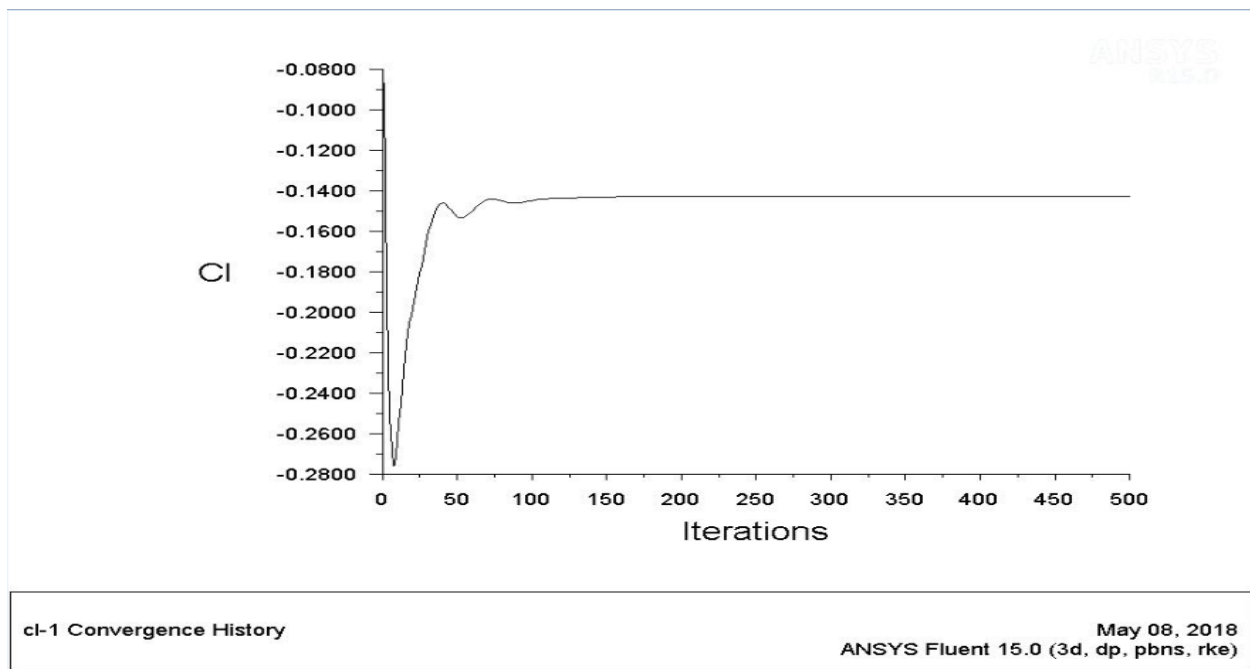


Figure 4. 41: Lift coefficient for model three at 80kmph.

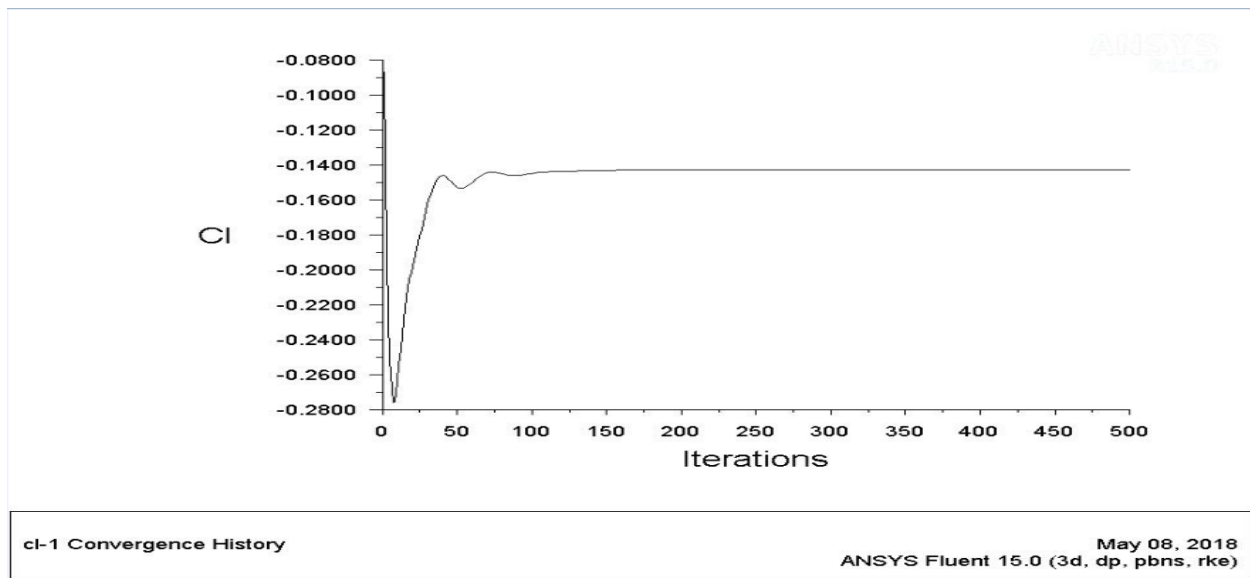


Figure 4. 42: Lift coefficient for model three at 100kmph.

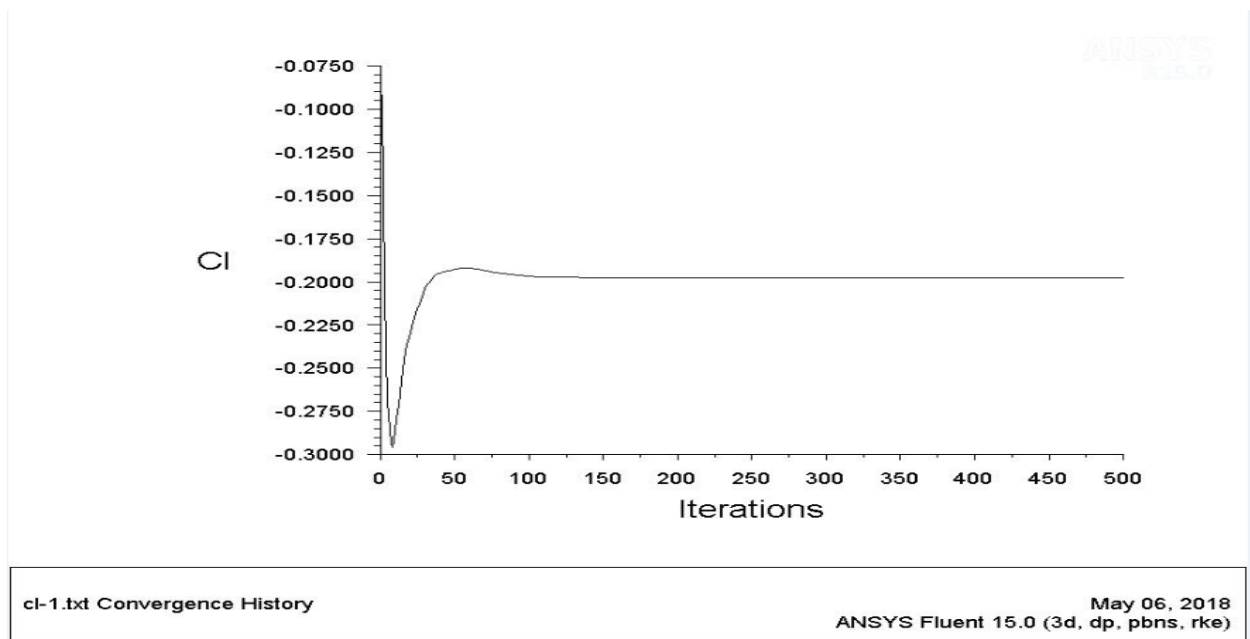


Figure 4. 43: Lift coefficient for model three at 120kmph.

This model is model two with tapering the windward corner towards the rear of bus model using SOLIDWORKS. After construct the model import and calculate the drag and lift coefficient, total pressure and flow file around the bus was tasted at three different speed and see the effect as model one and model two. The test procedure was the same as model one and model two. The scaled residuals was similar with the baseline model and model two. The drag and lift coefficients found were compared to the drag coefficient of the bus of model one and model two. The drag coefficients were calculated to be 0.575, 0.59 and 0.61 as shown in figure (4.38, 4.39 and 4.40) and lift coefficients were calculated to be -0.269,-0.278 and -0.29 as shown in figure (4.41, 4.42

and 4.43) at the speed of 80,100 and 120kmph respectively. We can compare the drag and lift coefficient of model three relative to model one. In figure 4.44 c_d model one and c_d model three or blue and red color represents drag coefficient for model one and model three. From this figure we have observed that decrease drag coefficient of model three relative to model one. The percent of drag redaction of model three relative to model one were calculated to be 12.879%, 14.493% and 12.96% at 80, 100 and 120kmph respectively.

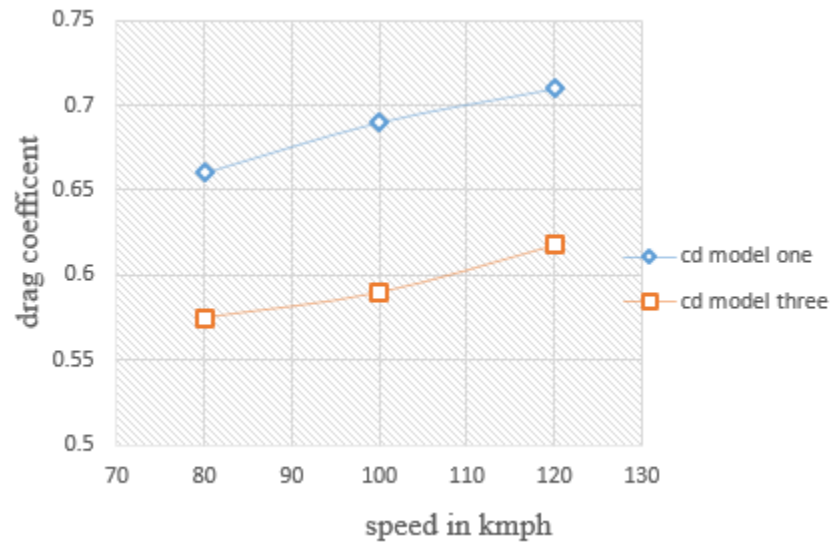


Figure 4. 44: Graph of drag coefficient for model one and model three vs speed of the vehicles

And the downward lift coefficient was increase relative to model one as shown in figure 4.45 bellow.

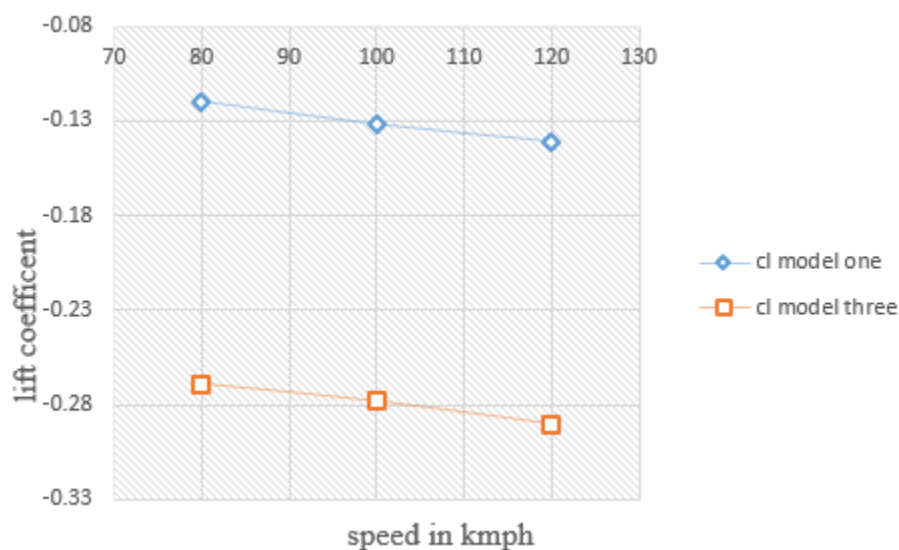


Figure 4. 45: Graph of lift coefficient for model one and model three vs speed.

The percent increment of lift coefficient of model three relative to model one were calculated to be -124.166%, -110.60% and -104.22% at 80, 100 and 120kmph respectively.

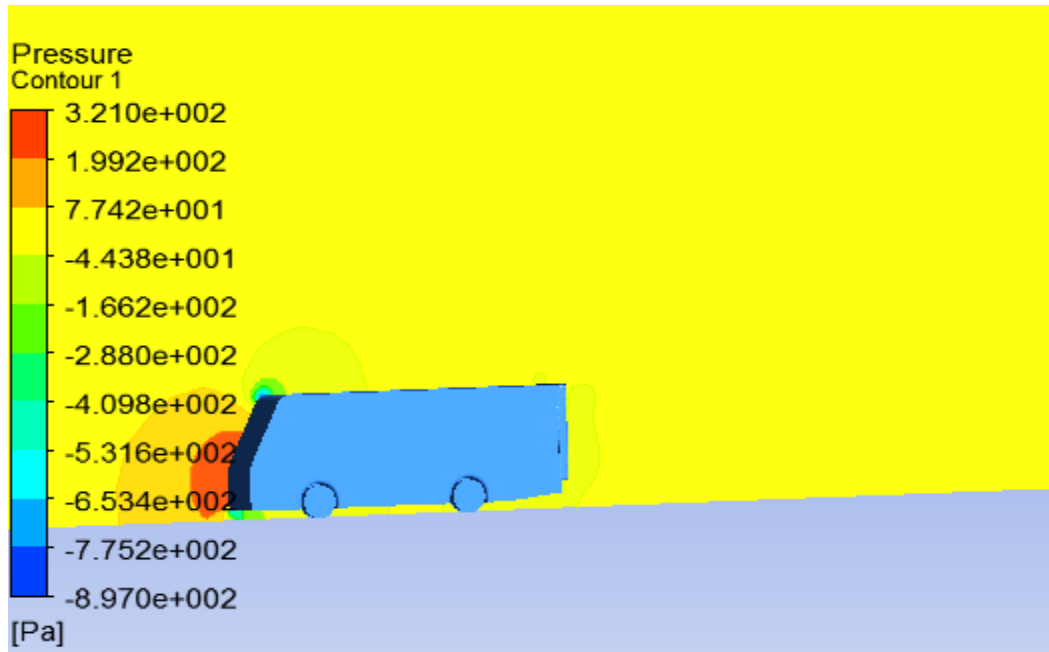


Figure 4. 46:Aerodynamic pressure contour for model three at 80kmph.

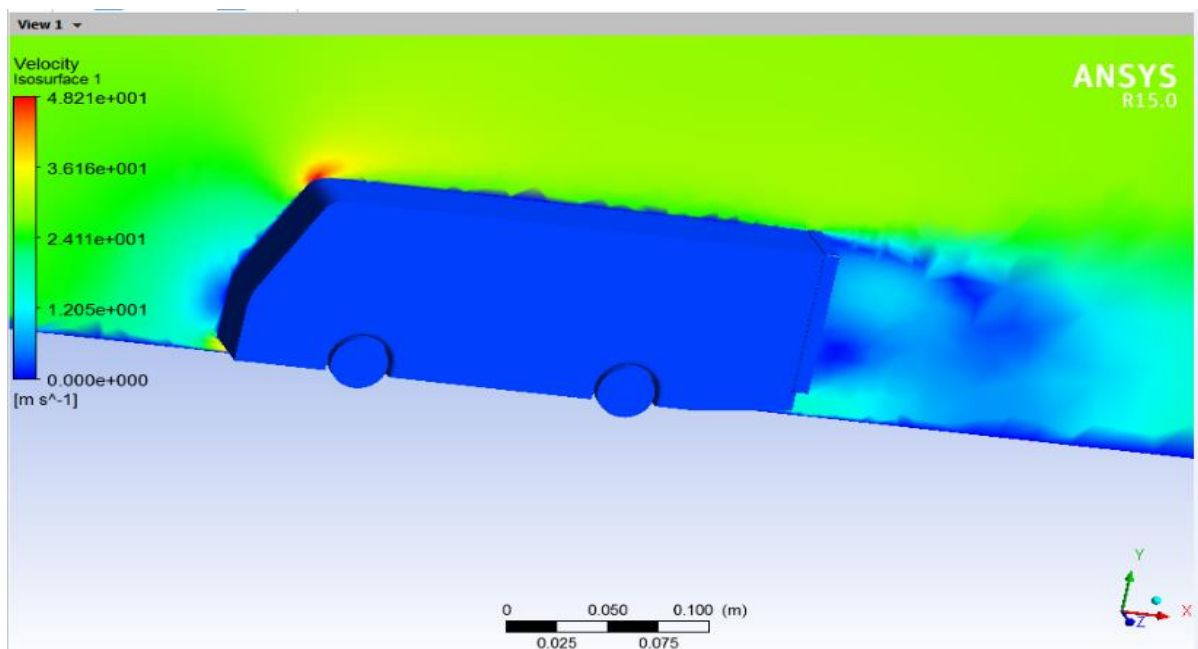


Figure 4. 47:Velocity countour for model three at 80kmph.

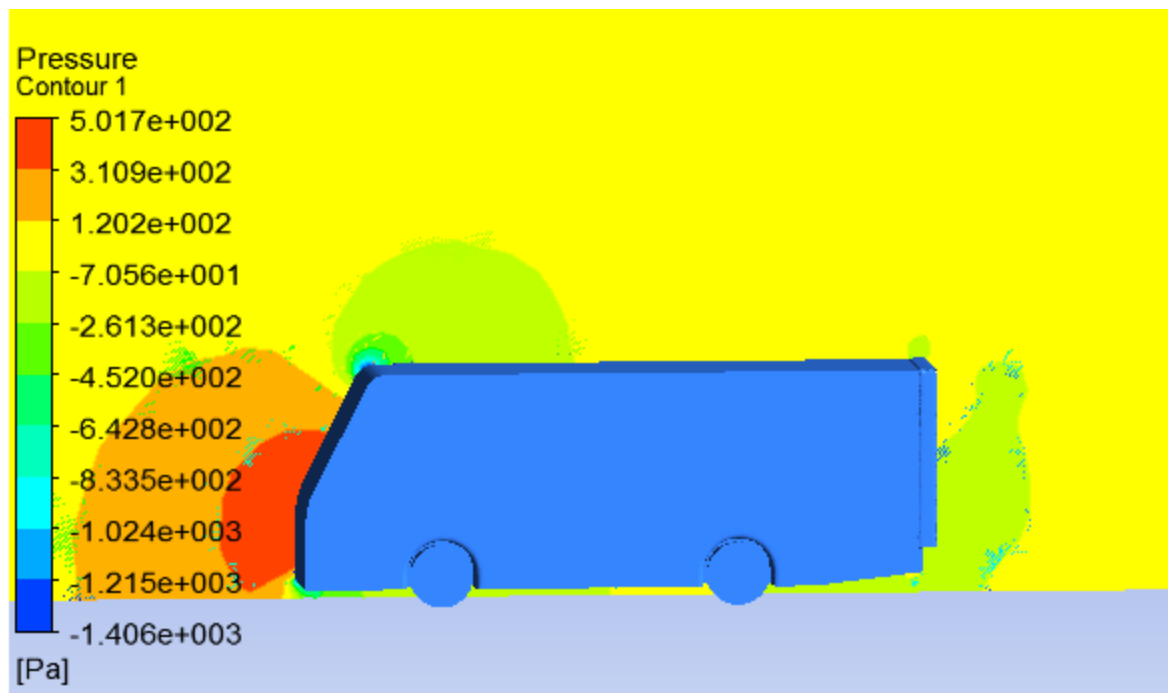


Figure 4. 48: graph of pressure distribution for model three at 100kmph.

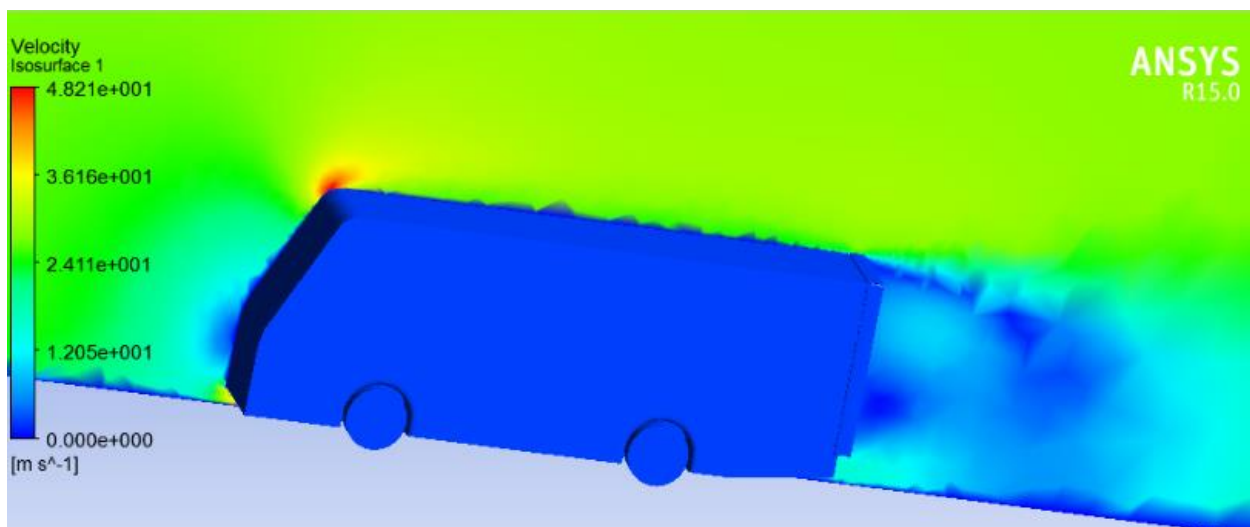


Figure 4. 49: Graph of velocity distribution of model three at 100kmph.

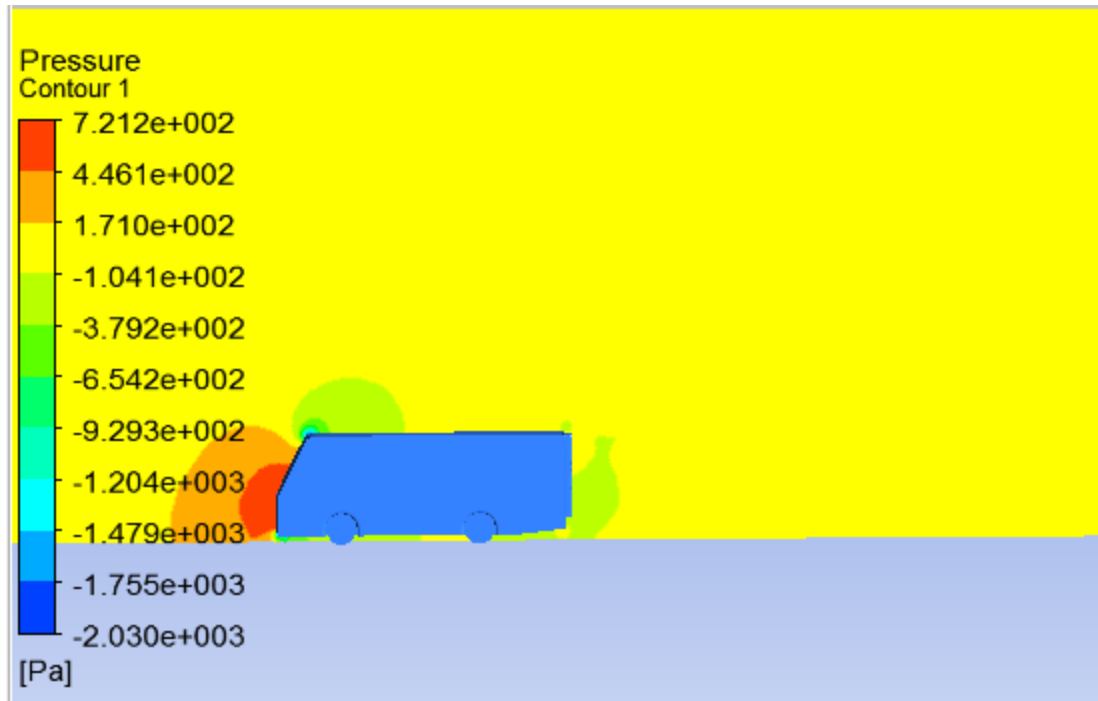


Figure 4. 50: Graph of pressure distribution for model three at 120kmph.

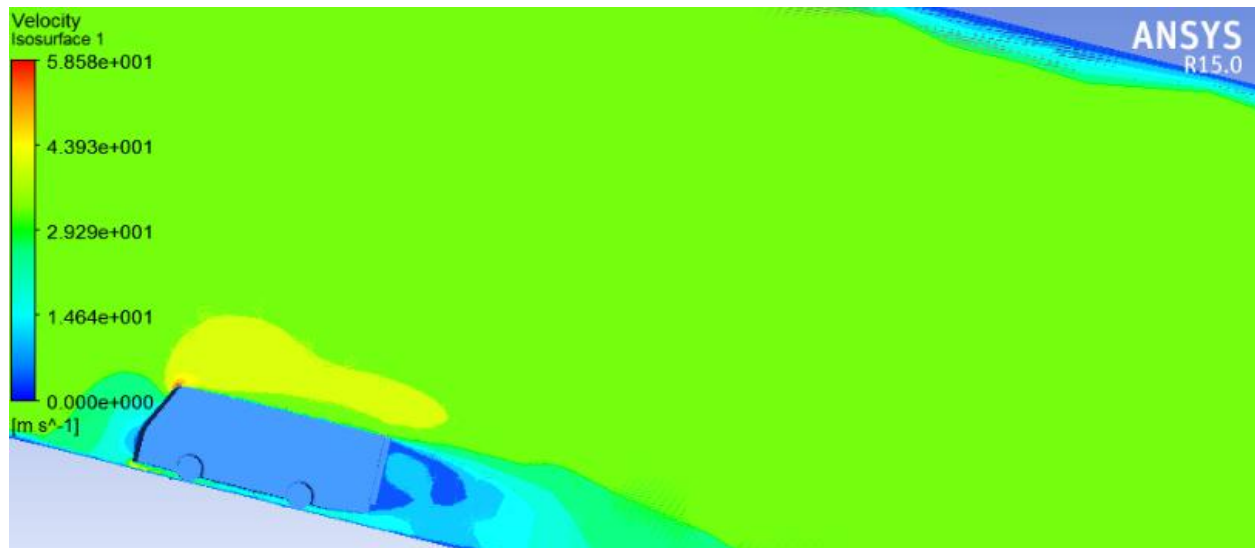


Figure 4. 51: Graph of velocity distribution of model three at 120kmph.

After analyzing drag and lift coefficient analyze velocity and pressure flow field around model bus. To fully examine how the airvane device and front tapering towards the rear changed the airflow surrounding the bus to reduce the drag present on the bus, contours of the velocity and pressure were created and are compared to the contours of the airflow around the bus model one. Figure (4. 47, 4.49 and 4.51) at 80,100 and 120kmph illustrates the velocity contour that was found using the polyhedral mesh with RKE turbulence model simulation. The contour shown was of the wake area of the bus and contour of the rest of the bus had changed with the addition of the

airvane and front tapering towards the rear. This contour can be compared to the velocity contour of the model one for the same speed. Comparing the velocity contours, we observed that the magnitudes found for the different colors of the contour were nearly the same with only minor negligible differences. When examining the wake of the bus was observed that the total area of low velocity in the wake was greatly decreased and more streamlined flow on the bus due to front tapering relative to model one but the difference from model two was only the streamlined flow was increase this decrease front stagnation area to decrease total adverse pressure. The turbulence kinetic energy was also decrease due to decrease low velocity area zone see in figure (4. 47, 4.49 and 4. 51) relative to figure(4.9, 4.10 and 4.11) respectively.

The pressure contour found for this simulation is shown in Figure (4.46, 4.48 and 4. 50) at 80,100 and 120kmph respectively and can be compared to the pressure contour found for the bus model one at the same speed in figure (4.9, 4.10 and 4.11). Comparing the magnitudes present in the legend for the figure above with the legend of the pressure contour of the bus model three it could be seen that there was small differences in the pressures that would provide slight differences in the pressure differential of the bus. higher pressure in the leeward face was created due to the airvane guiding the air flow from the top and side of the bus to the wake and the expansion of the air allowed for the decrease in velocity and in turn the increase of pressure. This increase in pressure decreased the pressure differential or total pressure, since the pressure drag is the largest portion of the aerodynamic drag present on the bus this change significantly reduced the drag coefficient of the bus. From the legend of pressure contour the stagnation pressure was decrease at the front due to front tapering by reducing stagnation area. The total differential pressure were decrease as sown in the figure 4.52 total pressure for model one and total pressure for model three or blue and red color indicates total pressure for model one and total pressure for model three respectively and the percent of pressures reduction were calculated to be 23.045%, 10.229% and 2.518% at 80, 100 and 120kmph respectively. And the average redaction was 11.93%. The kinetic energy was similar with model two.

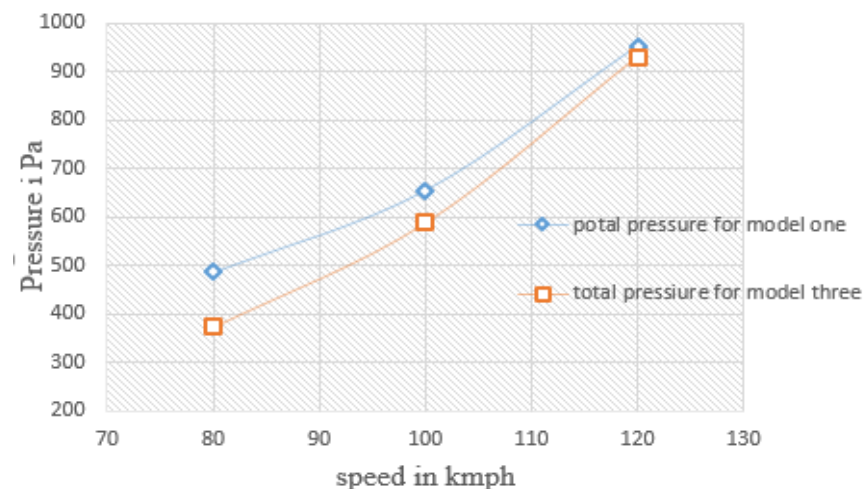


Figure 4. 52: Graph of total pressure vs speed of bus model one and three.

4.4. Result and discussion for model four.

4.4.1. Result of scaled residual, drag and lift coefficient, pressure and velocity countour at speed of 80kmph 100kmph and 120kmph for model four.

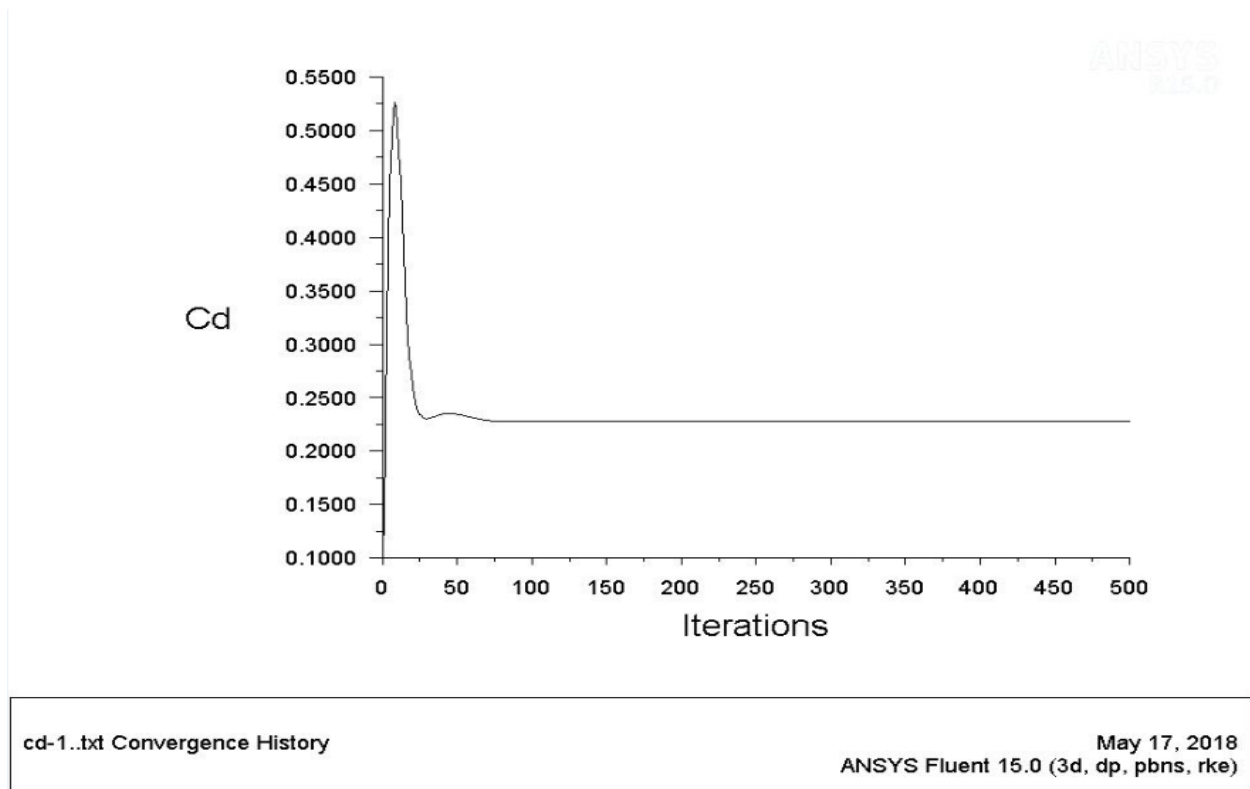


Figure 4. 53: Drag coefficient for model four at 80kmph

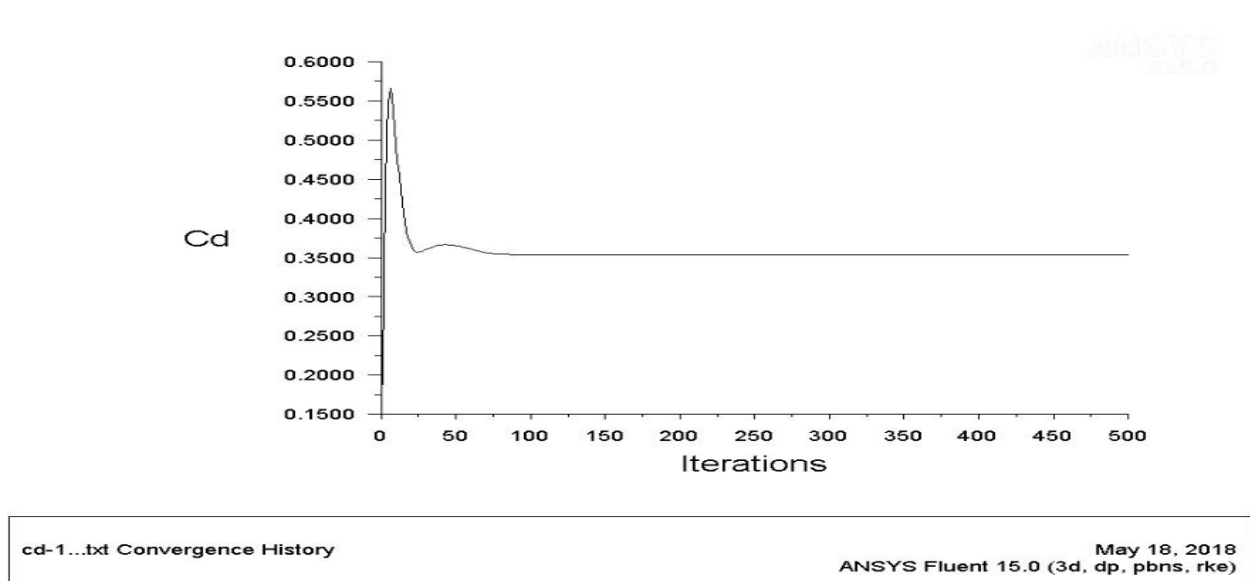


Figure 4. 54: Drag coefficient for model four at 100kmph

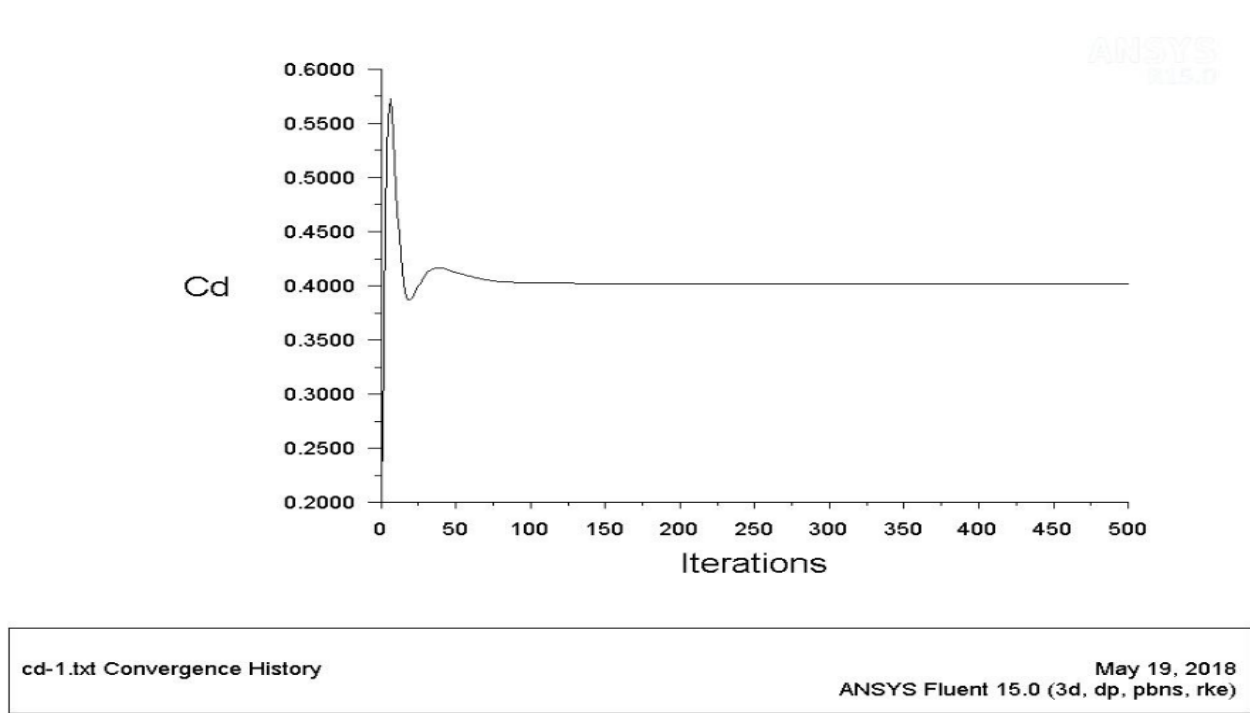


Figure 4. 55: Drag coefficient for model four at 120kmph

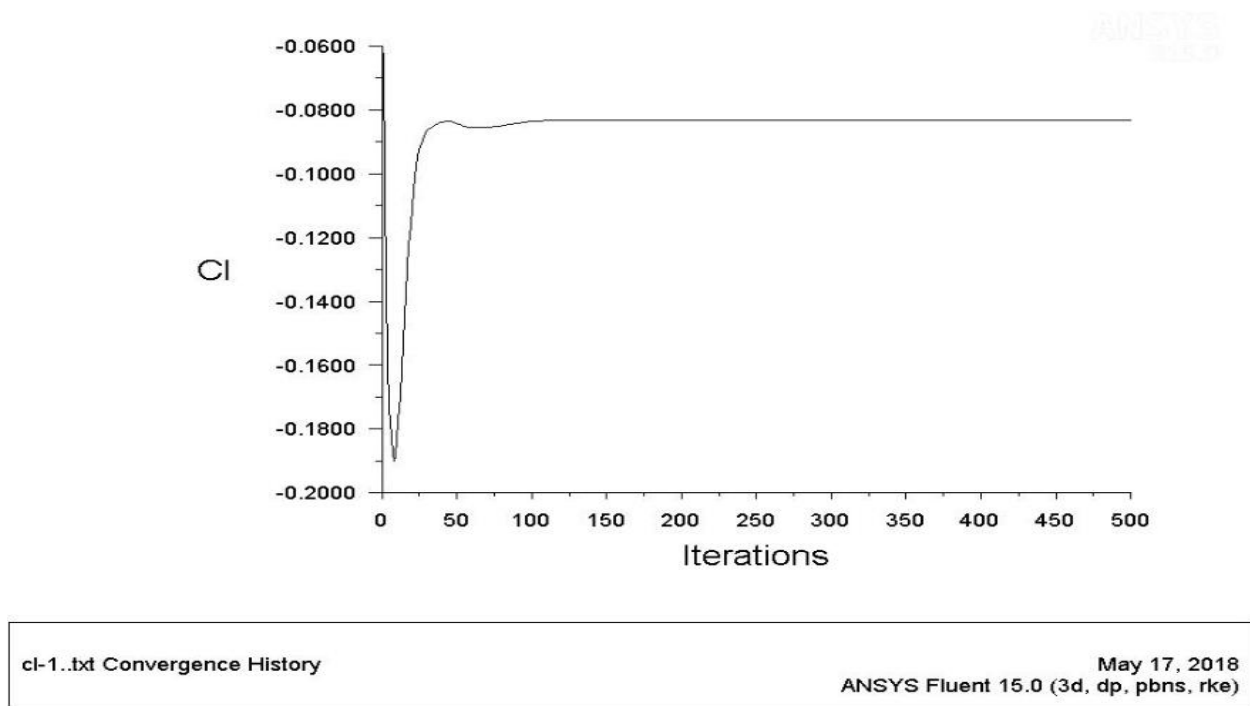
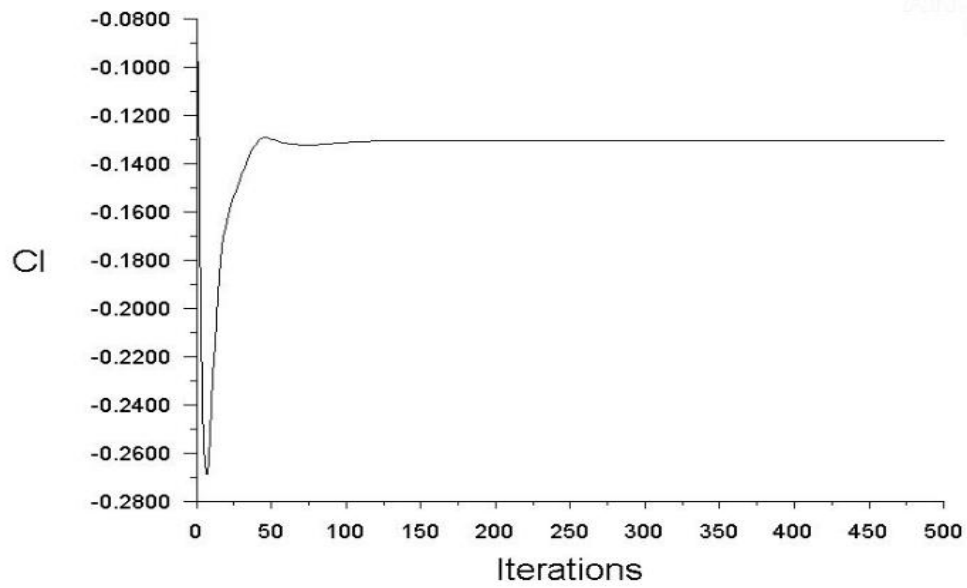


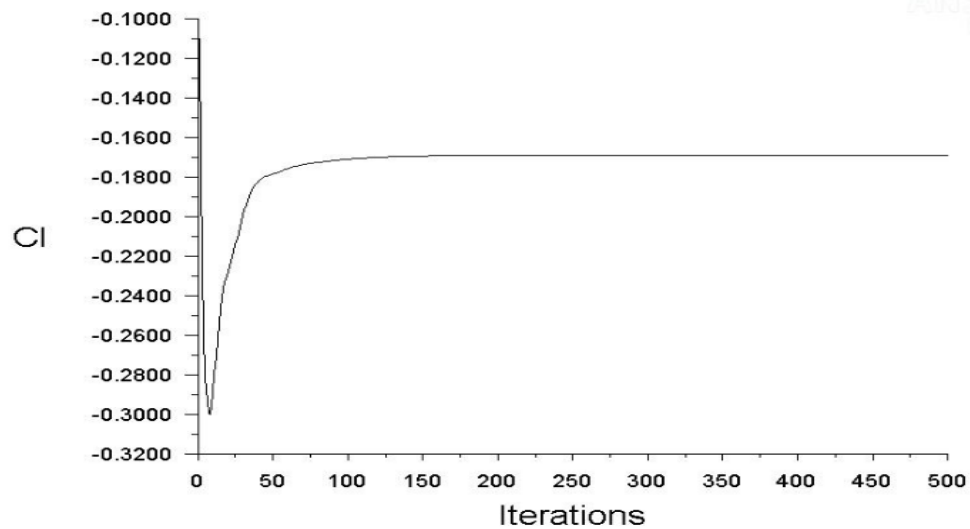
Figure 4. 56: Lift coefficient for model four at 80kmph.



cl-1...txt Convergence History

May 18, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure 4. 57: Lift coefficent for model four at 100kmph.



cl-1.txt Convergence History

May 19, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure 4. 58: Lift coefficent for model four at 120kmph.

This model was the last model which contains the airvane device like model two and three and the front was modified to make more streamlined based on based on the train shape which has high velocity and the train design concept was adapted from in the nature exhibit dynamism and have good aerodynamic shape. Killer whales are big in size and are known for their dynamism in the water it is very fast as shown in figure 4.59.



Figure 4. 59: New bus design concept for model four.

After modeling the bus model import and calculate the parameter like previous model. From the result as shown in the figure (4.53, 4.54 and 4.55) the drag coefficient were 0.525, 0.553 and 0.561 at 80, 100 and 120kmph respectively this value indicates as the drag coefficient increase as speed increase and we compare the drag coefficient of model one. The comparison of drag coefficient of model four relative to model one as shown in figure 4.60.

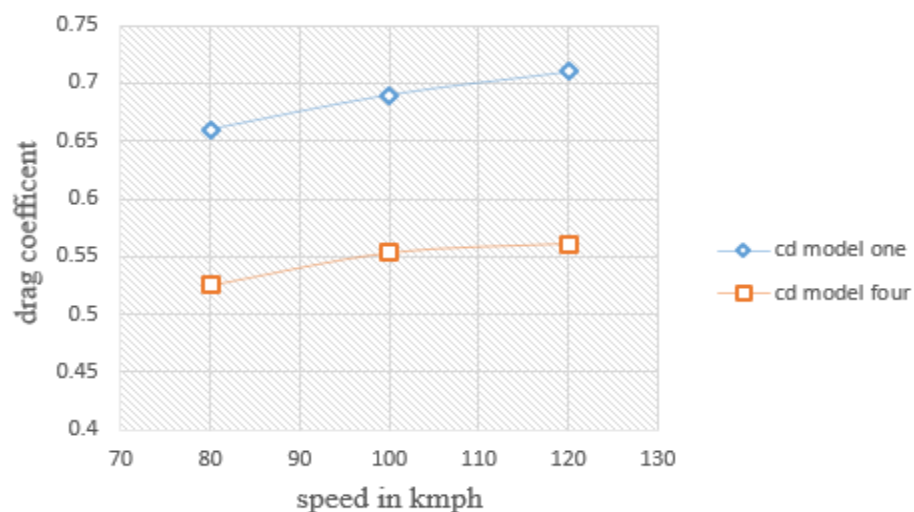


Figure 4. 60: Drag coefficient vs speed for model one and model four.

In figure 4.60 *cd* model one and *cd* model four or blue and red color represents drag coefficient for model one and model four respectively this graph shows that the drag coefficient highly decrease model four relative to model one. The percent redaction were 20.455%, 22.754% and 20.986% at 80, 100 and 120kmph respectively. 22.754% was high drag redaction at 100kmph in model four. The lift coefficient was calculated to be -0.19, -0.269 and -0.3 as shown in figure (4.56,

4.57 and 4.58). Lift coefficient also improved in small value. Lift coefficient reduction in percent between model four relative to model one were calculated to be -58%, -103.78% and -111.26%. This value indicates increase downward lift relative to model one, figure 4.61 shows the comparison of lift coefficient of model four relative to model one.

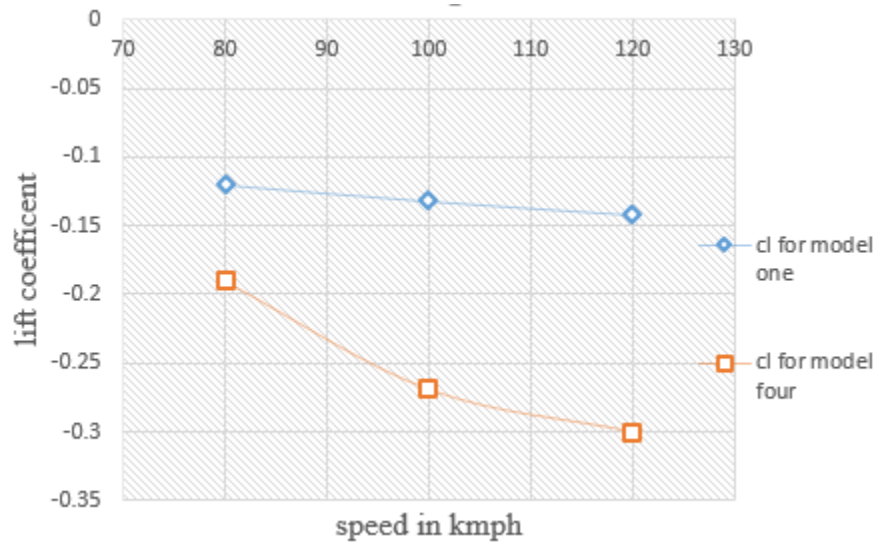


Figure 4. 61: Graph of lift coefficient vs speed for model one and four.

Figure 4.61 show that the downward lift increase model four relative to model one.

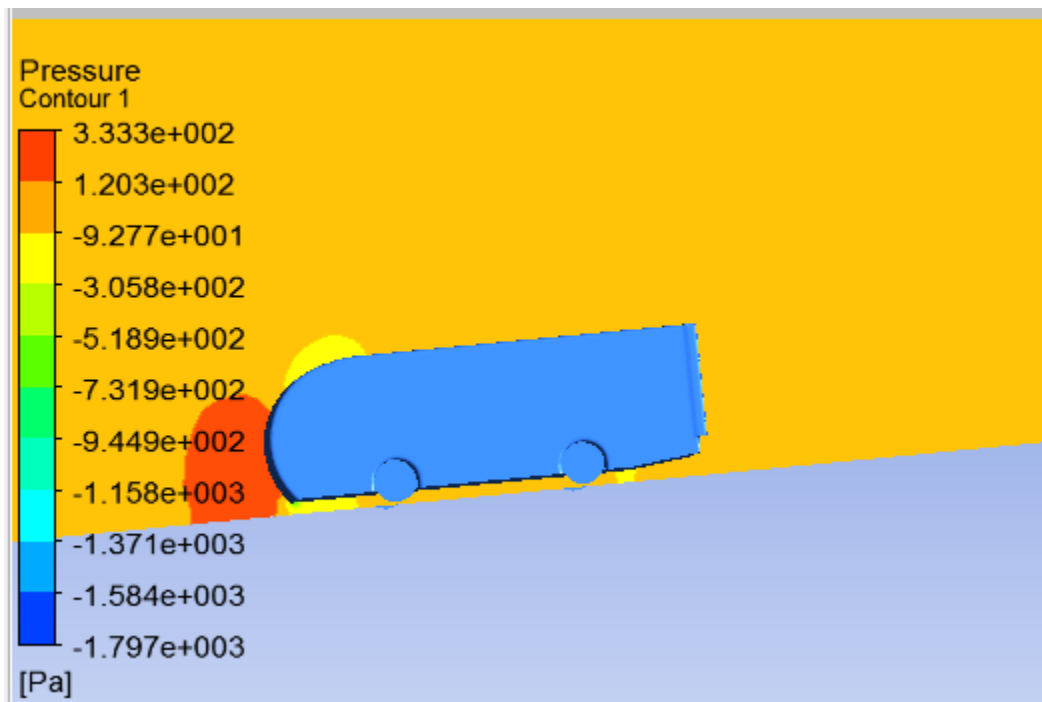


Figure 4. 62:Aerodynamic pressure contour for model four at 80kmph.

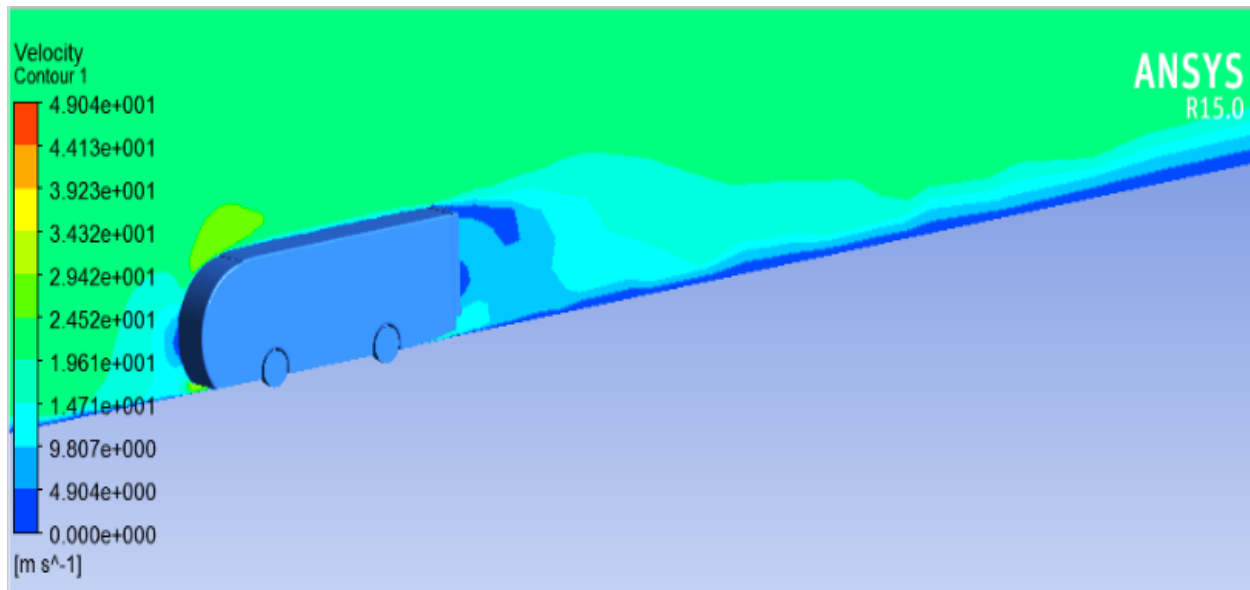


Figure 4. 63: Velocity countour for model four at 80kmph

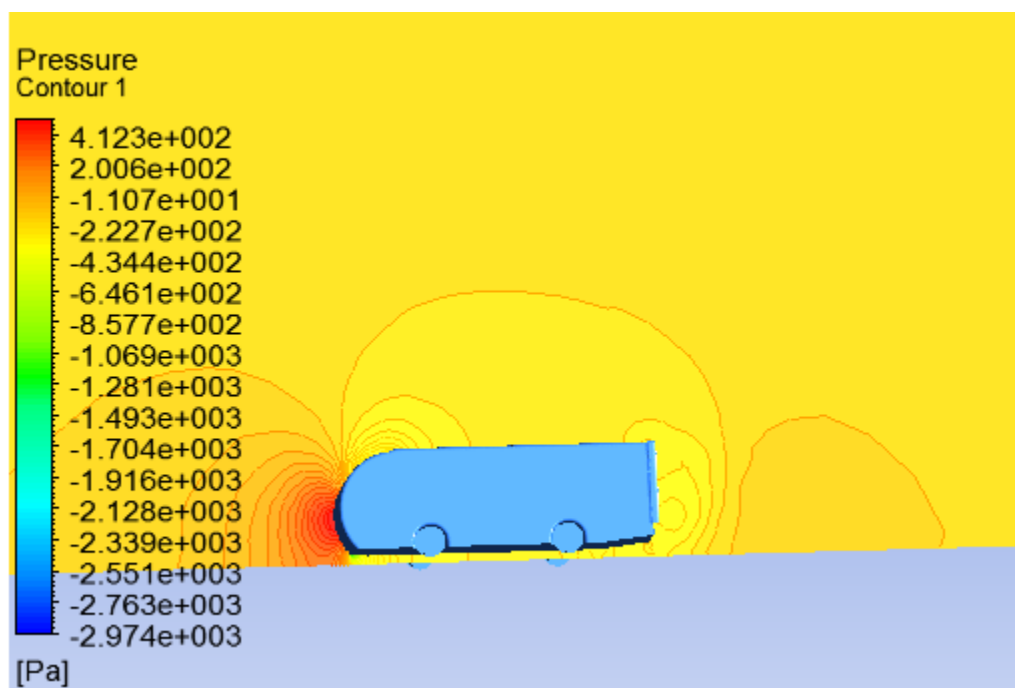


Figure 4. 64: graph of pressure distribution for model four at 100kmph

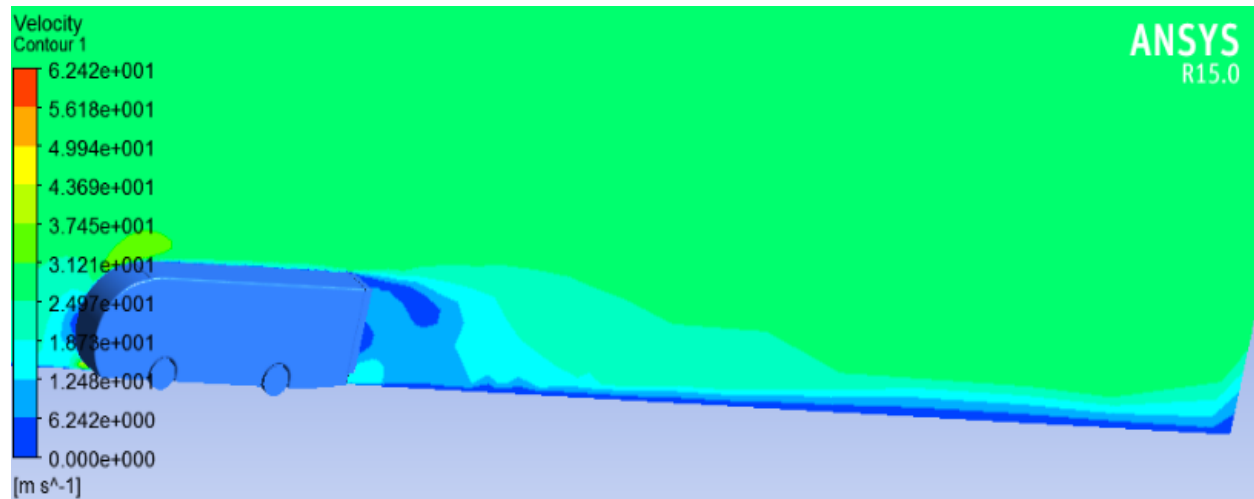


Figure 4. 65: Graph of velocity distribution of model four at 100kmph.

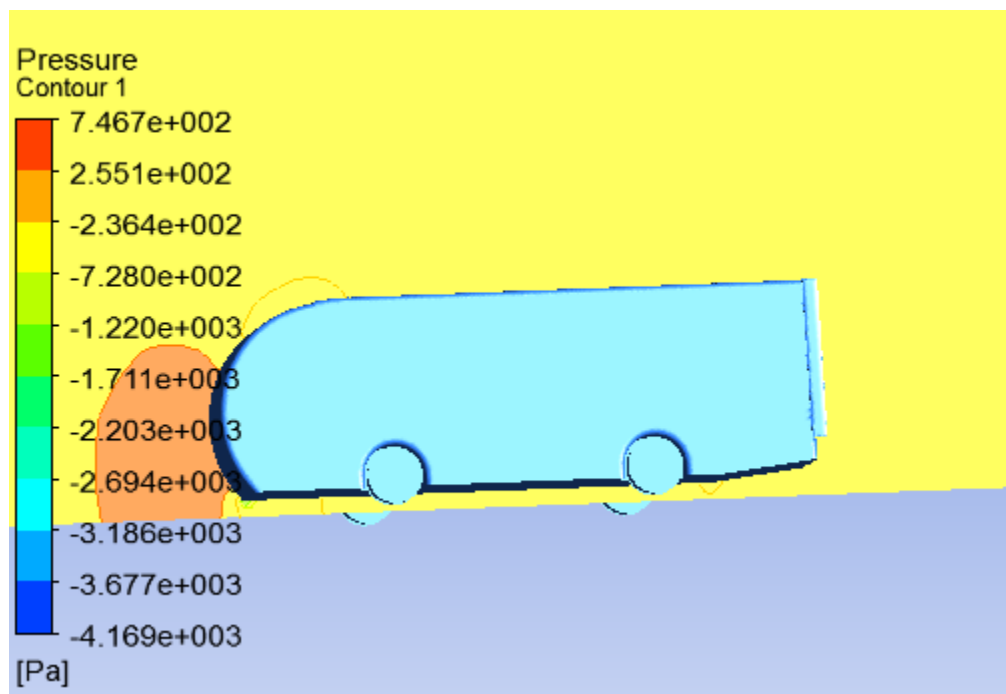


Figure 4. 66: graph of pressure distribution for model four at 120kmph.

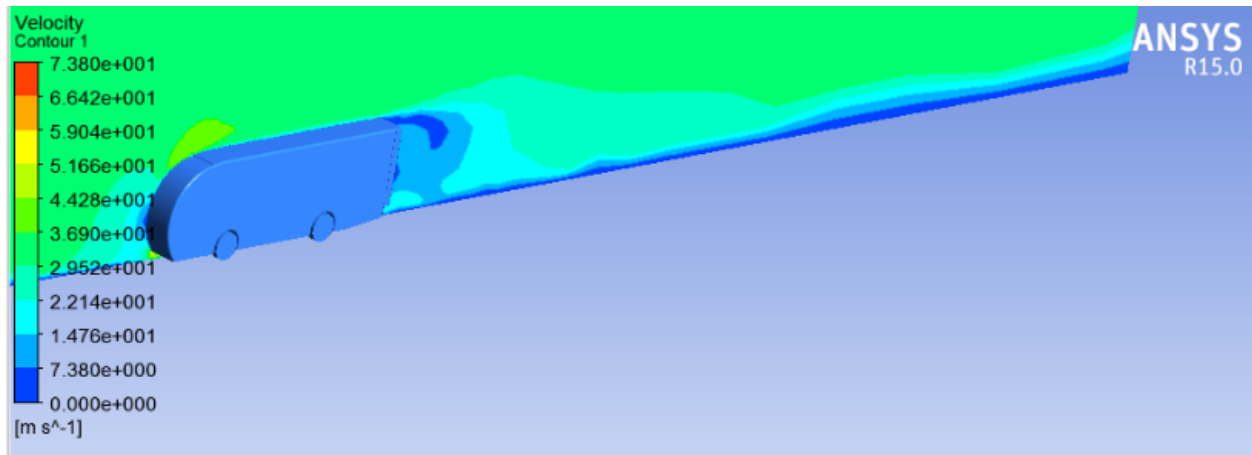


Figure 4. 67: Graph of velocity distribution of model four at 120kmph.

The velocity flow field around the bus model four were observed that the wake area was the same for model three and model two due to have the same aivane device is attached to bus it has high difference for in the stagnation area velocity due to teardrop like shape was design in model four but in relative to model one there is high different velocity contour as shown in figure (4.63, 4.65 and 4.67) relative to figure (4.9, 4.10 and 4.11) at 80, 100 and 120kmph respectively. In model four we have observed there is high streamlined flow in the front and minimize the wake region relative to model one.

The other parameter which analysis the pressure contour around the bus model four and compare with model one. Due to the wake region minimization and makes highly streamlined flow the pressure differential or total pressure were decreased and the value was as shown in the figure (4.62, 4.64 and 4.66) are 424, 583 and 797 pa at 80, 100 and 120 $kmph$ respectively. This value shows that speed increase total pressure also increase. The comparison of total pressure for model four relative to model one shown in figure 4.68 bellow.

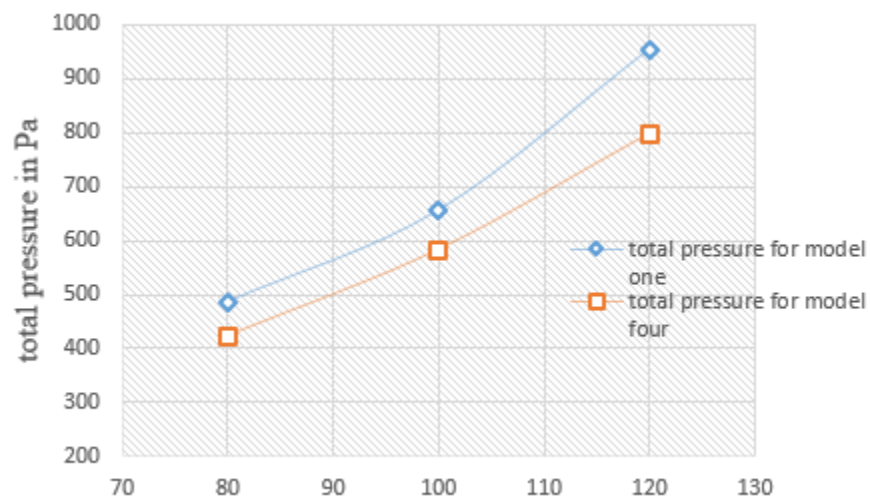


Figure 4. 68: Graph of total pressure vs speed for model one and model four.

Figure 4.68 show that the comparison of total pressure redaction for model four relative to model one, were calculated and we have get the value 12.34%, 10.99% and 16.37% at speed of 80,100, and 120kmph respectively.

SUMMARY

Table 4 2: Summery of drag and lift coefficient for all models at different speed.

Models	Speed in <i>kmph</i>	Drag coefficient(C_D)	Lift coefficient (C_L)
Model one	80 <i>kmph</i>	0.660	-0.12
	100 <i>kmph</i>	0.69	-0.132
	120K <i>kmph</i>	0.71	-0.141
Model two	80 <i>kmph</i>	0.648	-0.232
	100 <i>kmph</i>	0.675	-0.269
	120K <i>kmph</i>	0.68	-0.275
Model three	80 <i>kmph</i>	0.575	-0.269
	100 <i>kmph</i>	0.59	-0.278
	120K <i>kmph</i>	0.618	-0.29
Model four	80 <i>kmph</i>	0.525	-0.19
	100 <i>kmph</i>	0.553	-0.269
	120K <i>kmph</i>	0.561	-0.3

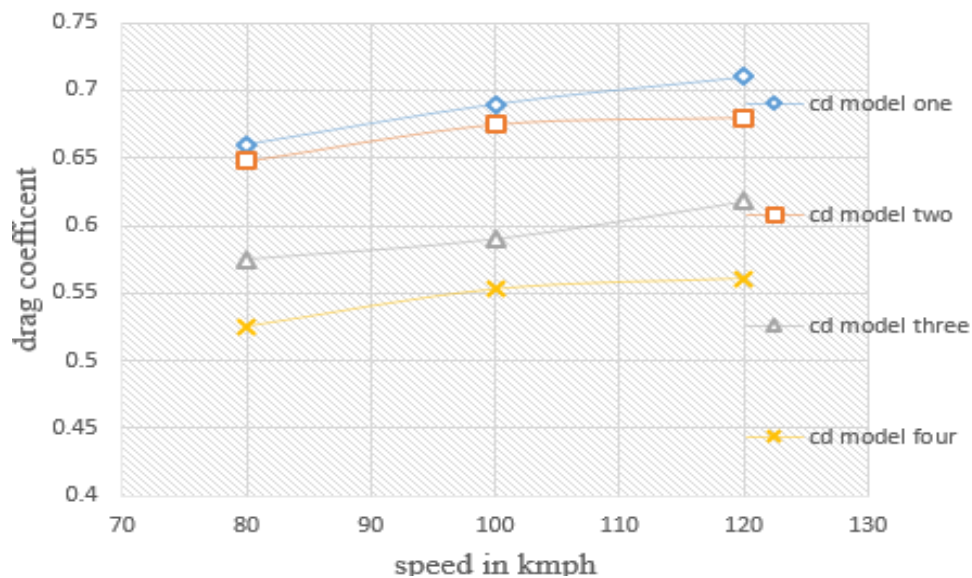


Figure 4. 69:Graph of drag coefficient vs speed for different model .

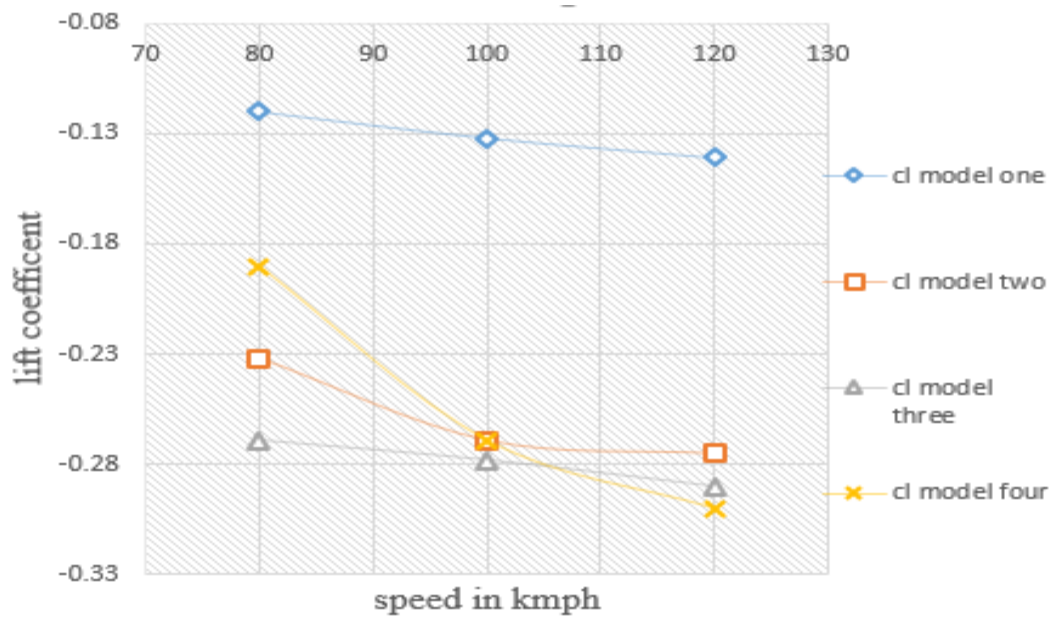


Figure 4. 70:Graph of lift coefficent vs speed for different model.

Table 4.2 , figure 4.69 and 4.70 shows that the ANSYS Fluent result value of lift and drag coefficient of all model at different speed when use RKE turbulence model with converted polyhedral mesh

Table 4. 3: Total pressure distribution for all models at different speed

Model	Speed (<i>kmph</i>)	Total pressure (<i>pascal</i>)
One	80	486
	100	655
	120	953
Two	80	407
	100	640
	120	943
Three	80	374
	100	588
	120	929
Four	80	424
	100	583
	120	797

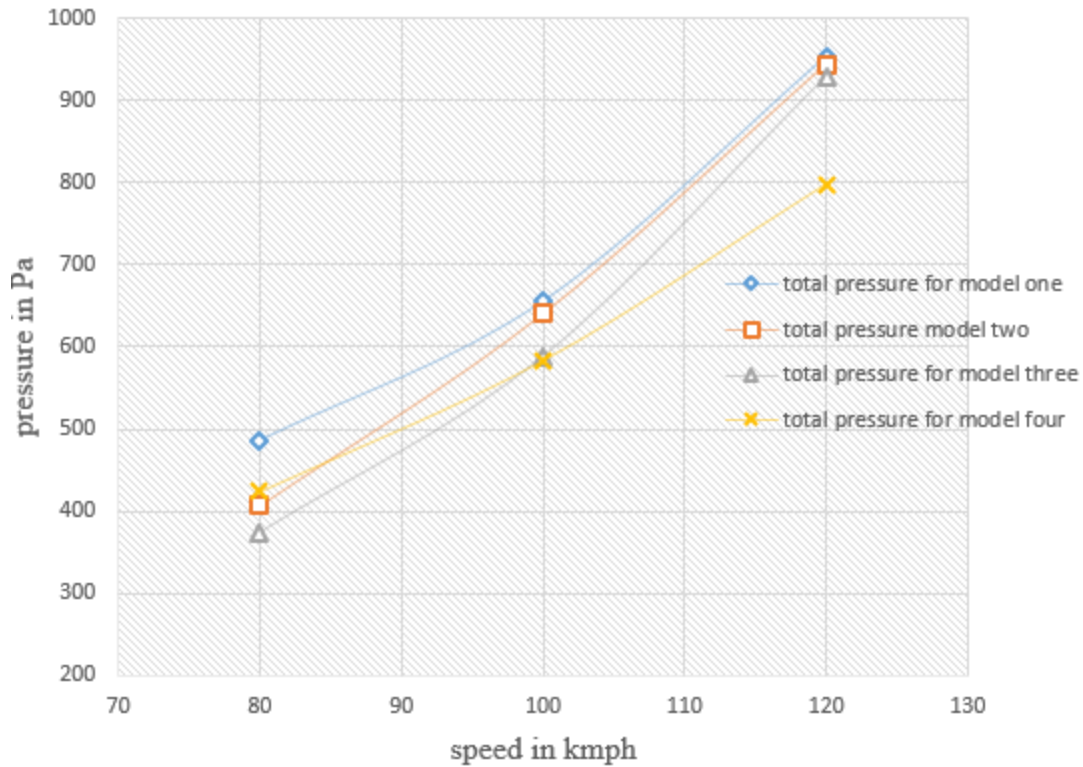


Figure 4. 71: Graph of total pressure vs speed for different model.

Table 4.3 and figure 4.71 shows that the ANSYS Fluent result value of pressure for all model at different speed when use RKE turbulence model with converted polyhedral mesh.

4.5. Calculation of Fuel Consumption

Percentage fuel reduction = $\frac{3}{5}$ (percentage of total drag reduction) [78]. Using this formula we can estimate the fuel saving of the given model relative to baseline model was calculated and the value was given in the table 4.4 bellow and also there is all the summery.

Table 4 4: Summery of percentage of drag coefficient, lift coefficient redaction and percentage of fuel saving.

Model	Speed in <i>kmph</i>	Drag coefficient (C_D)	Percent of drag reduction (%) from baseline model.	Percent of fuel saving in %	Lift coefficient(C_L)	Percent of lift reduction (%) from baseline model.
Model one	80	0.66	...	...	-0.02	
	100	0.69	...	...	-0.142	
	120	0.71	...	...	-0.167	

	Average	0.6867	...	...	0.1096	
Model two	80	0.648	1.88	1.128	-0.232	-93.3
	100	0.675	2.174	1.3044	-0.269	-103.78
	120	0.68	4.223	2.534	-0.275	-93.66
Average	100	0.667	2.867	1.72	-0.2586	96.913
Model three	80	0.575	12.879	7.727	-0.269	-124.166
	100	0.59	14.493	8.6958	-0.278	-110.60
	120	0.618	12.96	7.776	-0.29	-104.22
Average	100	0.594	13.444	8.066	-0.279	-112.99
Model four	80	0.525	20.455	12.273	-0.19	-58
	100	0.553	22.754	13.652	-0.269	-103.78
	120	0.561	20.986	12.59	-0.3	-111.26
Average	100	0.546	21.389	12.833	-0.253	-91.013

Table 4.4 shows a maximum of 13.652% fuel can be saved in model four from model one at 100kmph . 12.833% fuel can be saved in model four at three average speed from Model one. The average fuel consumed by model one per 100 km is 24L (Table 3.1). From this in the bus Model four can save 12.833% of 24L, which is 3.0799 L. Therefore, Bus Model four consumes 20.92 L per 100 km. in model three can save 8.066% 24L , which is 1.9358L and in model two 1.72% of 24L we can save 0.413L move the vehicle the vehicle between 80 and 120kmph.

The downward lift was increase in model two, three and model four relative to model one but increase the speed decrease the downward lift. As shown table 4.4 above.

4.6. Calculation of reduction of fuel exhaust emission.

Due to Gavin Dias et.al study [38] the fuel consumption by ground vehicles accounts for over 30% of CO₂ and other greenhouse gas (GHG) emissions. From this the percent reduction of modified model relative to baseline model was calculated to be 30% of 13.65 which is 4.095% of emissions reduced in model four at 100kmph, and average emission percent reduction was 30% of 12.833 which is 3.85% was reduced, in model three we can save 2.4198% and model two 0.516% greenhouse gas was reduced. This small percent reduction has high significant when using many vehicle

4.7. Comparison of this work with Yisfalgn Niguse

In Yisfalgn Niguse work modeled a new bus body design compared with Yutong cross country bus. He used turbulence model with quadrilateral mesh. The new Bus concept was based on frontal area. Roof tapering and side panel shape. The results show that the drag force reduced by 34.4% from Yutong cross country bus. In this work modify only the frontal part and add on device used

as drag reduction device. Simulate the model using RKE turbulence model with converted polyhedral mesh. The results show that the drag coefficient reduced by 22.754%. From this conclude that design new concept body reduce high amount of drag force but it require high cost and design quality and simply add drag reduction device with same modification reduce to same extent with low cost.

CHAPTER FIVE

CONCLUSION AND RECOMMENDATION

5.1. Conclusion

- In this study airvane, add-on drag reduction devices for model two to four and modify the front shape of front chamfering and making more streamlined in model three and four with airvane device simultaneously. The guidelines pointed out of a general nature that can be implemented in most modern road going vehicles; smooth bus shape, rounded corners, high rake angle for the windscreen, tapered rear end, were employed on a model of a commercial bus to study the effects of this device and this modification on the reduction in drag force.
- Analysis was completed successfully in 3-D by using CFD software using RKE turbulence model with converted polyhedral mesh. In the process of redesigning, exterior styling with improved aerodynamics of existing Luxury mid bus driving on Ethiopian roads, a detailed computational analysis has been done. This computational analysis shows that there is possibility of improving the aerodynamic performance of bus by modifications in exterior design of bus body and add on device which is airvane. These modifications are helpful in reducing the coefficient of drag i.e. C_D which affects the fuel consumption. A three dimensional flow analysis has been performed on the models at various speeds to predict the airflow characteristics around bus. The results provide the flow pattern and associated drag of the bus body.
- The four bus model have been modeled for performing numerical analysis using CFD software. The Bus model one is the existing model, bus model two is the existing model with airvane device added on the rear end, Bus model three is the existing bus with modification at front and adding the airvane device at the rear end and model four was existing bus with airvane device and front modification. The four models have been separately tested for optimizing how each modification is contributing towards the drag force.
- In the fluent analysis at velocity of 80kmph , 100kmph , and 120kmph was used. It was found that the least drag force was acting on Bus model four. Bus model three and bus model two gave an intermediate result as expected. The average drag coefficient of the baseline mode one was 0.6867, model two was 0.667, model three was 0.618 and model four was 0.546. By these modifications the coefficient of drag is reduced by model two, model three and four were approximately 2.867%, 13.444% and 21.389% respectively. Fuel consumption of about 1.72%, 7.776% and 13.652% can be reduced from 80kmph to 120 kmph for model two to model four respectively. Reduction of greenhouse gas emission was 0.516%, 2.4198% and 3.85% can be reduced from 80kmph to 120 kmph for model two to model four respectively. And increase the downward lift in model one to model four by this improvement in fuel efficiency will have a high impact on the reduction

of annual fuel consumption and also reduced to exhaust emission. Hence the aim has been achieved.

5.2. Recommendation

- Bishoftu automotive industry have willingness about this study to build the recommended shape they ask the simulation result and also if any automotive industry willingness to study and check their vehicle aerodynamically we can work together and this cooperation must be motivated to build vehicle aerodynamically.
- Genuine CFD tool (ANSYS Fluent) software is required for better aerodynamic analysis of the bus. Therefore, the department should purchase ANSYS Fluent software.
- The department should review seminar experienced sharing delivery system which is must be programed for every week even not for month or three week.
- In this work all the external accessory like AC at the top, mirror, pillar groove and effect of coating are not included due to this the value of drag force has been changed but the percent reduction was the same, so for the reader recommended that consider this accessory assume as ideally present .

5.3. Future work

Further study to be performed in this paper several different actions could be taken. All dimensions of the airvane were taken from the literature i.e. from “Criseth” study without any optimizing the parameter so for the future study optimized dimensional parameter to get optimal value and can reduce more drag force.

Further turbulence models could be applied to the parameter studies that were run to check the different turbulence model and see the effect and recommended. Large Eddy Simulations and Detached Eddy Simulations could be used to predict airflow around bluff bodies.

Experimental studies, using a wind tunnel, could be run on the final model found in this study to compare the values found to the values produced in this study. This comparison would allow for further analysis of which model and mesh was able to provide a more realistic value for the drag coefficient and how accurate the tests were.

In the future work could be testing of other add-on devices and different modification that reduce the aerodynamic drag present on the bus. Also design and test active drag redaction device to reduce the drag fore on the vehicle.

REFERENCE

- [1] Smith, 2002 Study the History and innovation of Automotive Motor Vehicle Industry.
- [2] CD-Adapco. STARCCM+ Manual, version 4.06.011 edition.
- [3] Alexander Diehl, Rui Miranda, Christoffer Mursu Simu and John Viji, autumn 2002 , “Reducing Drag Forces in Future Vehicles”, Chalmers, University of Technology.
- [4] Ahmad Kamal Matamin, August 2006 "Aerodynamic Study of a Coach Using Computational Fluid Dynamic(CFD) Technique".
- [5] Newland, L.E. July 1980 "A Fuel Consumption Function for Bus Transit Operations and Energy Contingency Planning": The University of Michigan.
- [6] Diebler. C and Smith .M, November 2002 "A Ground-Based Research Vehicle for Base Drag Studies at Subsonic Speeds", NASA Dryden Flight Research Center, California.
- [7] Roy and Srinivasan, 2000 "Aerodynamic Study of Trucks and other High-sided Vehicles", International Truck and Bus Meeting & Exposition.
- [8] Yamin, A. K. M. August 2006, “Aerodynamic Study of a Coach Using (CFD) ", Faculty of Engineering and Computing, Coventry University.
- [9] Abdel Aziz A. A., and Abdel Gawad, F. “Aerodynamic and Heat Transfer Characteristics around Vehicles with Different Front Shapes in Driving Tunnels,” 2006, Sharm El-Shiekh, Sinai, Egypt.
- [10] François D. G., Delnero J. S. Marañón and Camocardi M. E., June 2009 "Experimental Determination of Stationary Aerodynamics loads on a double deck Bus", 11th Americas Conference on Wind Engineering, San Juan, Puerto Rico.
- [11] Yelmule M. and Kale S. R., No. 4, 2009’ ’Aerodynamics of a Buswith Open Windows’’, Int. J. Heavy Vehicle Systems, Vol.16.
- [12] Z. Mohamed-Kassim, Filippone .A, 2010 "Fuel savings on aheavy vehicle via aerodynamic drag reduction", Transportation Research Part D 15, 275-284.
- [13] Jeff Patten et al, (2012), “Review of Aerodynamic Drag Reduction Devices for Heavy Trucks and Buses”, National Research Council of Canada.
- [14] Arun Raveendran et al, (2009), “Exterior Styling of an Intercity Transport Bus for Improved Aerodynamic Performance”.
- [15] Arthur Sheridan and Steven Grier .J, (1978), “Drag Reduction Obtained by Modifying a Standard Truck”.
- [16] Karim .M, Rahman.M and Alim .M. A, January 2009 “Computation of turbulent viscous flow around submarine hull using unstructured grid”, Journal of Ship Technology,.

- [17] Amit Ray, S N Singh 2009 “Evaluation of Linear and Non-linear Hydrodynamic Coefficients of Underwater vehicles using CFD” Honolulu.
- [18] Asltzman and Robert. R, Meyer, (1999), “A reassessment of heavy duty truck aerodynamic design features and priorities”.
- [19] Ludovico Consano and Davide Lucarelli., (2007), “Fuel Reduction on a Tractor-Trailer Truck”, 3rd European automotive CFD conference.
- [20] Gilhaus A., (1998) “Main parameters determining the aerodynamic drag of buses, colloque construire avec le vent, vol 2.
- [21] Callen R. Mc, Salari K., Ortega J., Browand F., M. Hammache, Hsu.T., (2004), “Effort to Reduce Truck Aerodynamic Drag – Joint Experiments and Computations Lead to Smart Design”, AIAA Fluid Dynamics Conference.
- [22] An Introduction to Computational Fluid Dynamics, The finite Volume Method. H K Versteeg and W Ma Lalasefera.
- [23] A Tewari. 2007 “Atmospheric and Space Flight Dynamics”. Lecture Notes in Mathematics. Chapter 10 Elements of Aerodynamics p233-267, Springer.
- [24] J Katz. 2006 “Aerodynamics of Race Cars”. Annual Review of Fluid Mechanics, 2006.38:27-63.
- [25] W. H. Hucho. 1998 “Aerodynamics of Road Vehicles”, 4th Edition. SAE International.
- [26] Lajos T. Preszler L., and Finta L., (1988), “Styling and Aerodynamics of Buses,” Int. J. of Vehicle Design.
- [27] Hilleman T. B., May 2003 “Vehicle Drag Reduction with Air Scoop Vortex Impeller and Trailing Edge Surface Texture Treatment,” U.S. Pat. 7, 192, 077.
- [28] Christopher Michael Mugnaini 2015 “Aerodynamic Drag Reduction of a Tractor Trailer Using Vortex Generators: a Computational Fluid Dynamic Study”, Presented to the faculty of the Department of Mechanical Engineering California State University, Sacramento.
- [29] Gilieron P. Roumeas, M. and A Kourta. 2009 “Drag Reduction by Flow Separation Control on a Car after Body”. Lecture Notes in Mathematics. International Journal for Numerical Methods in Fluids, vol.60, no. 11, pp. 1222-1240. SAE Technical Paper 840302, 1984, 1984.
- [30] Yesfalgn Damissie, 2015 “Aerodynamic Drag Reduction on Locally Built Bus Body Using Computational Fluid Dynamics (CFD): A Case Study at Bishoftu Automotive Industry” Department of Mechanical and Vehicle Engineering. Adama, Ethiopia.
- [31] D’Alambert paradox, Az Aramlastan Alapjai, 4. kiadas. Budapest, Hungary, 2008.”Design streamlined vehicle using CFD.”
- [32] Konstantinidis E. and Bouris E. 2012 “Bluff Body Aerodynamics and Wake Control”. In TechApplied Aerodynamics, Chapter 4.

- [33] P Bearman. 2009 “Bluff Body Flow Research with Applications to Road Vehicles”. The Aerodynamics of Heavy Vehicles II: Trucks, Buses and Trains, Springer, Volume 41, 2009, pp 3-13.
- [34] Mansor S. and Passmore M. A. “Estimation of Bluff Body Transient Aerodynamics Using an Oscillating Model” Southborough University, LE11 3TU, UK.
- [35] Freund J. B. and Mungal. M. G. 1994 “Drag and Wake Modification of Axisymmetric Bluff Bodies” Usinf Coanda Blowing. Journal of Aircraft, May-June.
- [36] Murad, N.M., Naser J. Alam F. and S. Watkins (2004). “Simulation of a Vehicle A-Pillar Aerodynamics Using Various Turbulence Models”.
- [37] Pavan K.V.S (May 2012- July2012); “CFD Modelling of flow around Ahmed Body”.
- [38] Gavin Dias, Nisha R. Tiwari, Joju John Varghese, Graham Koyeerath. 2016 “Aerodynamic Analysis of a Car for Reducing Drag Force”, Dept. of Mechanical Engineering, SJCT, Palghar, Maharashtra, India.
- [39] Bellman, Agarwal M., Naber R., and Chusak,L. (2010) “Reducing energy consumption of ground vehicles by active flow control.” American Society of Mechanical Engineers, 2010.
- [40] Kourta, A., and Gilliéron, P. (2009) “Impact of the automotive aerodynamic control on the economic issues”, Journal of Applied Fluid Mechanics, Vol.2 (2), pp69-75.
- [41] Kotapati, Mittal R. B., R., & Cattafesta III, L. N. (2007) Numerical study of a transitional synthetic jet in quiescent external flow. Journal of Fluid Mechanics, Vol.581, pp287-321.
- [42] Kourta, and Leclerc, C. (2013) “Characterization of synthetic jet actuation with application to Ahmed body wake”.
- [43] Park, J. H., Lee, J., and Kim, K. H. (2013) Aerodynamic drag reduction of Ahmed model using synthetic jet array,SAE International Journal of Passenger Cars-Mechanical Systems.
- [44] Bruneau, Creusé, D., Gilliéron, P., & Mortazavi, I. (2010) “Coupling active and passive techniques to control the flow past the square back Ahmed body”, Computers & Fluids.
- [45] Hahba, T., Springer, M., Lienhart, H., Kniesburgs, and Becker, S. (2012) “passive flow control” on a car model, In 16th Int Symp on Applications of Laser Techniques to Fluid MechanicsLisbon, Portugal.
- [46] Kahailighi, R. and Passmore, 2012 “Aerodynamic drag reduction of a simplified square back vehicle using steady blowing”, Experiments in fluids.
- [47] Song, M., Nagayoshi, T., and Hamamoto, N. (2004) “Research on aerodynamic drag reduction by VGs”, Mitshubishi Motors Technical Review, No. 16, pp11-16.
- [48] Gopal and Senthilkumar, (2012). “Aerodynamic drag reduction in a passenger vehicle using vortex generator with varying yaw angles”, ARPN Journal of Engineering and Applied Sciences.

- [49] Islam and Chen, (2010), "Reduction of aerodynamic forces on a minivan by a pair of vortex generators of a pocket type", International journal of vehicle design.
- [50] Kim and Chen, Mazyan, (2013), "Numerical simulations of drag-reducing devices for ground vehicles", Doctoral dissertation, American University.
- [51] Che Zakem, (2008). "Aerodynamics of aftermarket rear spoiler", Bachelor Thesis, University Malaysia Pahang.
- [52] Hu and Wong, (2011). "A numerical Study on Rear-Rpoiler of passenger vehicle", World Academy of Science, Engineering and technology.
- [53] Littlewood and Passmore (2013), "Numerical study of flow over Ahmed body and a road vehicle and the change in aerodynamic characteristics caused by rear spoiler", International Journal of Fluid Mechanics Research, Vol.40(4), pp354-372.
- [54] Kodali and Bezavada, (2012), "Numerical Simulation of Air Flow Over a Passenger Car and the Influence of Rear Spoiler Using CFD." International Journal of Advanced Transport Phenomena.
- [55] Kirch J. W. and Day E. A., 1977 "Air Drag Reducing Means for Bluff Vehicles and the Like," U.S. March.
- [56] J. W. Kirch, February 1973 "Drag Reduction of Bluff Vehicles with Airvanes," Tech. Rep, Society of Automotive Engineers, New York.
- [57] Agarwal R. K., (2013). "Sustainable Ground Transportation - Review of Technologies, Challenges and Opportunities," Int. J. Energy Environ.
- [58] Ahmed Ramm G. "Some Salient Features of the Time – Averaged Ground Vehicle Wake", SAE Technical Paper, 1984
- [59] Ahmed S.R. (1981) "An Experimental Study of the Wake Structures of Typical Automobile Shapes", J. Wind Eng. Ind. Aerodyn.
- [60] Pletcher R. H., J. C. Tannehill, and, D. A. Anderson, 2013. "Computational Fluid Mechanics and Heat Transfer", CRC Press, New York.
- [61] ANSYS, Fluent, 15.0. 2017 [Download]. ANSYS, Inc., Canonsburg, PA.
- [62] "The 1900 Wright Glider." (30 Nov 2004) Wright Aeroplane Company.
- [63] "The 1901 Wind Tunnel and balances." 2004 Wright Aeroplane Company.
- [64] Fletcher C. A. J. and Stewart G. D. H., 1986 "Bus Drag Reduction by the Trapped Vortex Concept for a Single Bus and Two Buses in Tandem," Journal of Wind Engineering and Industrial Aerodynamics.
- [65] P. K. Kundu and D. R. Dowling, 2012 Fluid Mechanics, 5th ed. Ch. 12 pp. 541-596. Academic Press, Boston.

- [66] Pletcher, J. C. Tannehill, and, Anderson D. A., 2013 "Computational Fluid Mechanics and Heat Transfer", 3rd ed., New York.
- [67] Davidson L., 2014 "Fluid Mechanics, Turbulent Flow and Turbulence Modeling," Lecture Notes, Department of Applied Mechanics, Chalmers University of Technology, Göteborg, Sweden.
- [68] D. Pointer, 2004 "Evaluation of Commercial CFD Code Capabilities for Prediction of Heavy Vehicle Drag Coefficients," AIAA Fluid Dynamics Conference and Exhibit.
- [69] Kim, m., (2005) "Aerodynamic Simulation on the Drag Reduction Device of a Large-Sized Bus Model," Int. J. Heavy Vehicle Systems, 12 207-24.
- [70] John J. G., A. Muthuvel, and Prakash N., (2014). "Numerical Analysis for Drag Reduction in Commercial Buses," IOSR Journal of Mechanical and Civil Engineering, 2320-334.9. E. M. Wahba, H. Al-Marzooqi, M. Shahin, and T. El-Dhmarshawy, "Aerodynamic Drag Reduction for Ground Vehicles using Lateral Vanes,"
- [71] Rodi W., (1997). "Comparison of LES and RANS Calculations of the Flow around Bluff Bodies," Journal of Wind Engineering and Industrial Aerodynamics, 69-71 55-75.
- [72] Bredberg, J. "On Two-equation Eddy Viscosity Models, 2001" Department of Thermo and Fluid Dynamics, Chambers University of Technology, Göteborg, Sweden.
- [73] Peng S., Davidson L., and Holmberg S., 1996 "The Two-Equation $k-\omega$ Model Applied to Recirculating Ventilation Flows," Department of Thermo and Fluid Dynamics, Chambers University of Technology, Göteborg, Sweden, February.
- [74] ANSYS, Inc, ANSYS Fluent, 2014 "Turbulence Modeling using ANSYS Fluent". The Corporation, Canonsburg, Pennsylvania.
- [75] B. Khalighi, S. Zhang, C. Koromilas, S. R. Balkanyi, L.P. Bernal, G. Iaccarino, and P. Moin, "Experimental and Computational Study of Unsteady Wake Flow Behind a Bluff Body with a Drag Reduction Device," SAE Tech. Rept., 2001-01-1042, General Motors R&D Center, University of Michigan, and Cascade Technologies, Inc.
- [76] ANSYS, Inc., ANSYS Fluent. The Corporation, Canonsburg, Pennsylvania, 2014.
- [77] Pointer D., Sofu T., Chang J., and Weber D., 2009 "Applicability of Commercial CFD Tools for Assessment of Heavy Vehicle Aerodynamic Characteristics"; pp. 349-61 in the Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains, Vol. 41, Lecture Notes in Applied and Computational Mechanics. Edited by R. McCallen. Springer Berlin, Germany.
- [78] Bitchtas D. 2012 "Aerodynamic evaluation of heavy trailer upper surface curvature". Master's thesis, Cranfield Universi

APPENDIX

APPENDIX A: SOLIDWORKS 1/30th scaled model of commercial bus used in this study

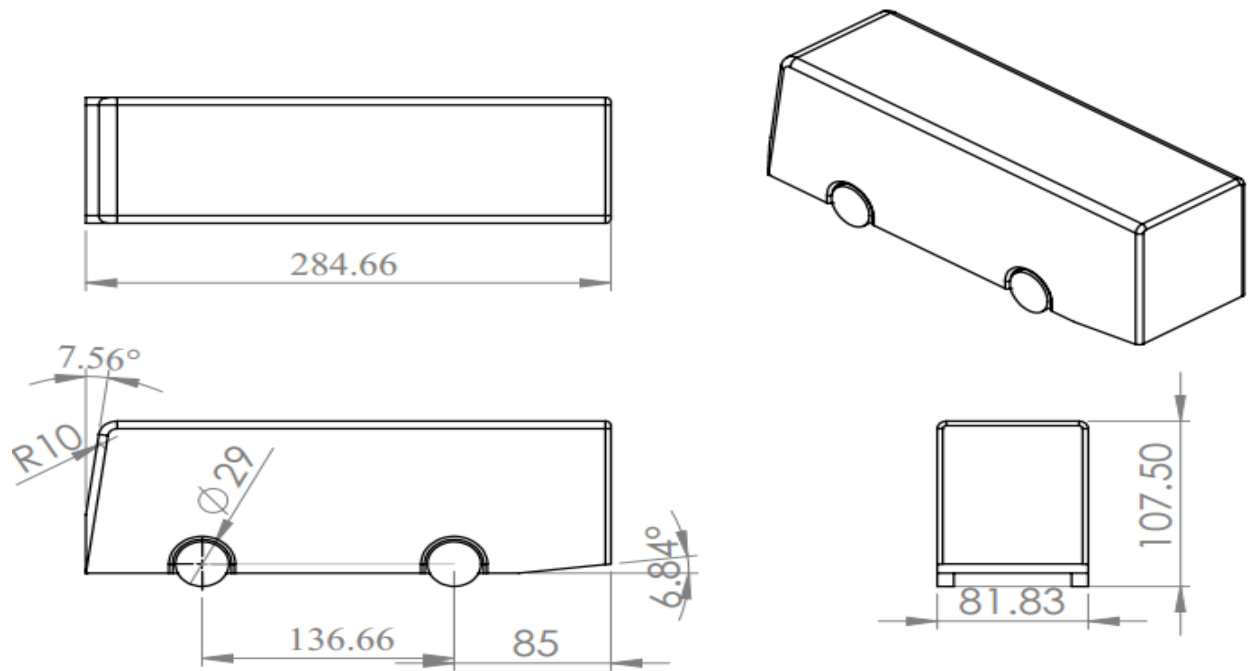


Figure A. 1: Model one scaled down model view.

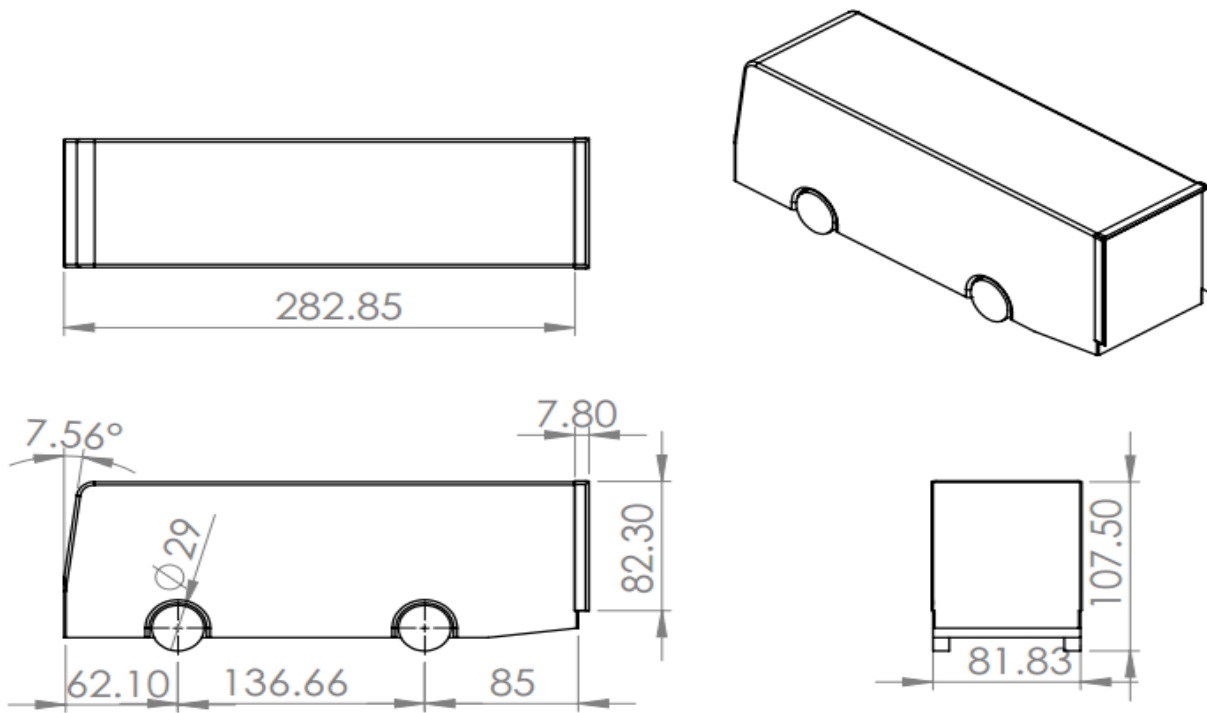


Figure A. 2: Scaled down bus model two

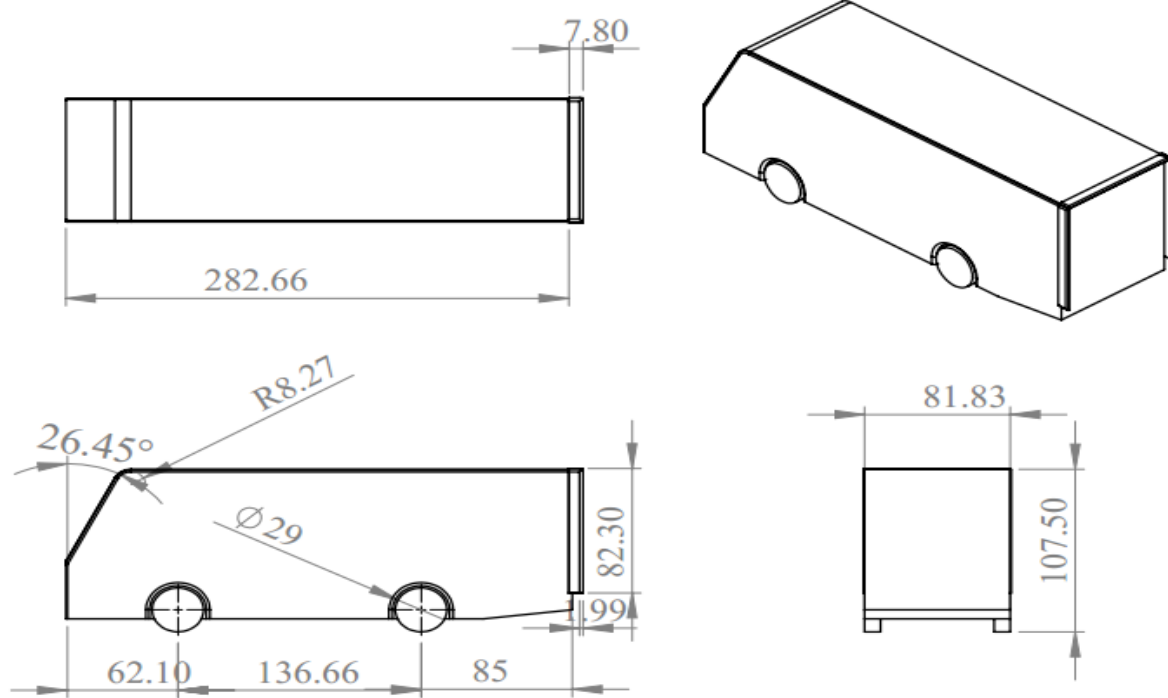


Figure A. 3: Scaled down bus model three.

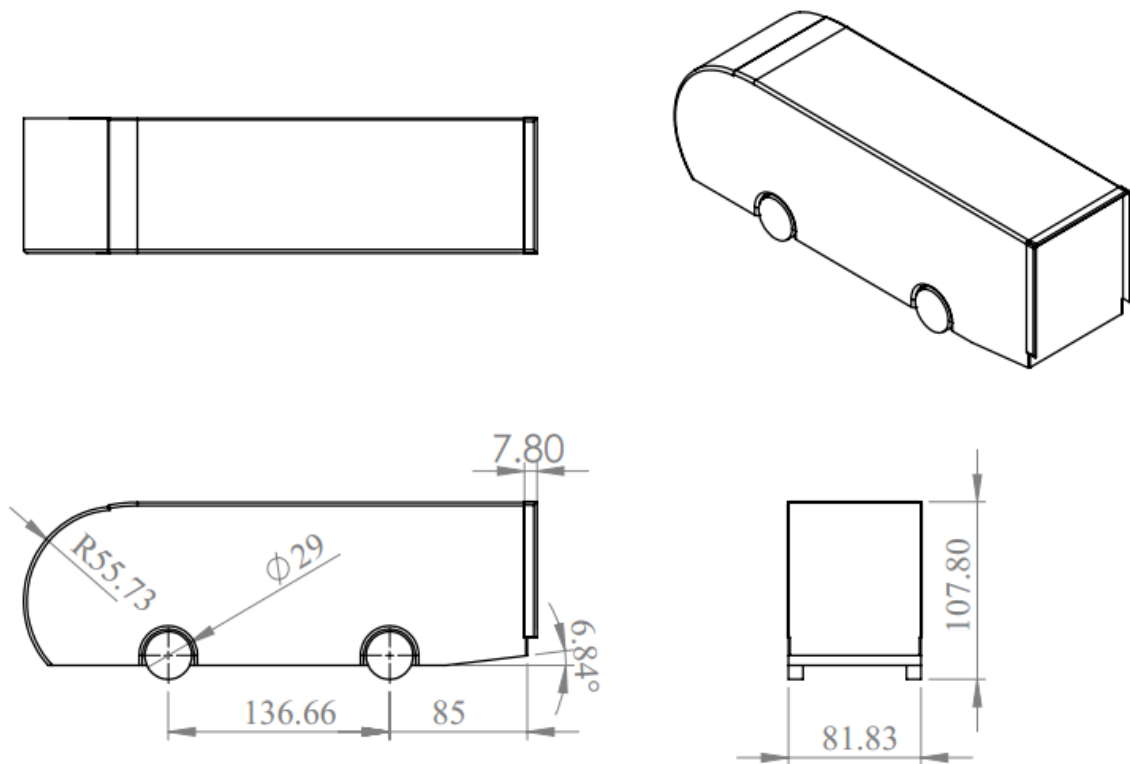


Figure A. 4: Scaled down bus model four.

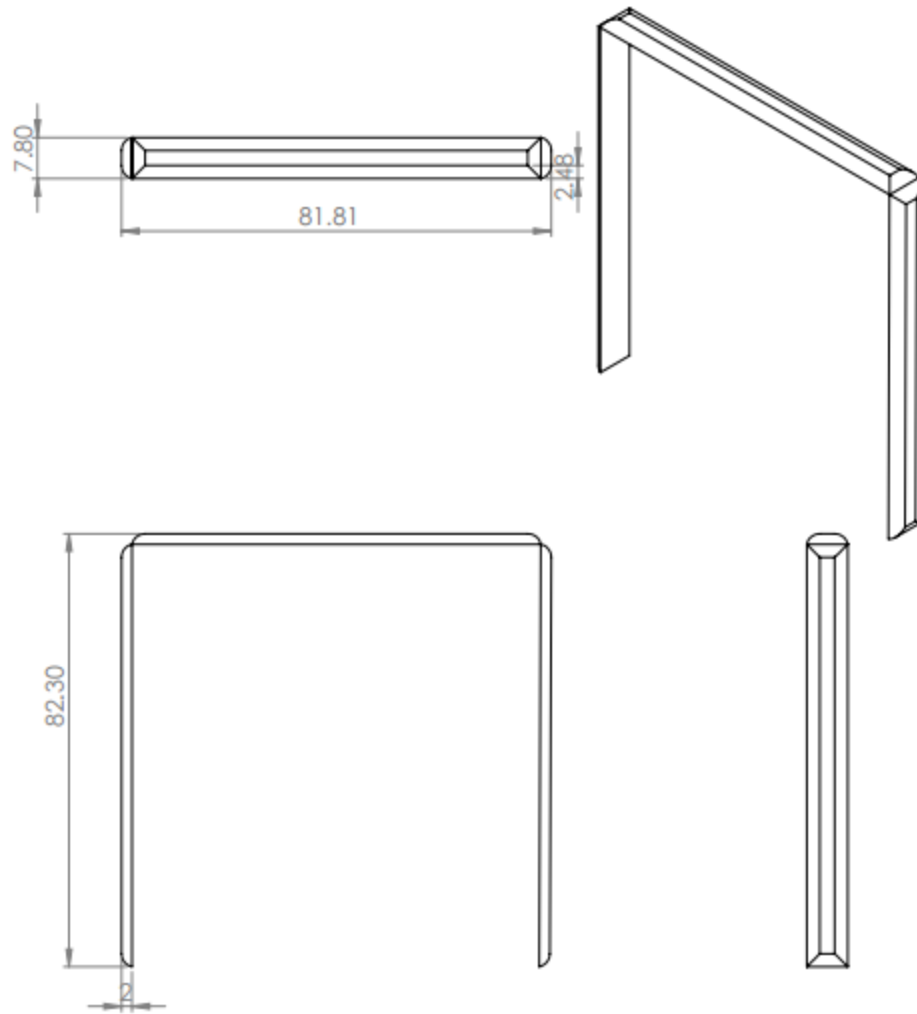
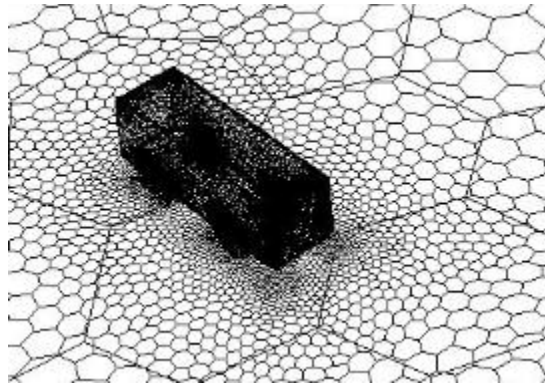
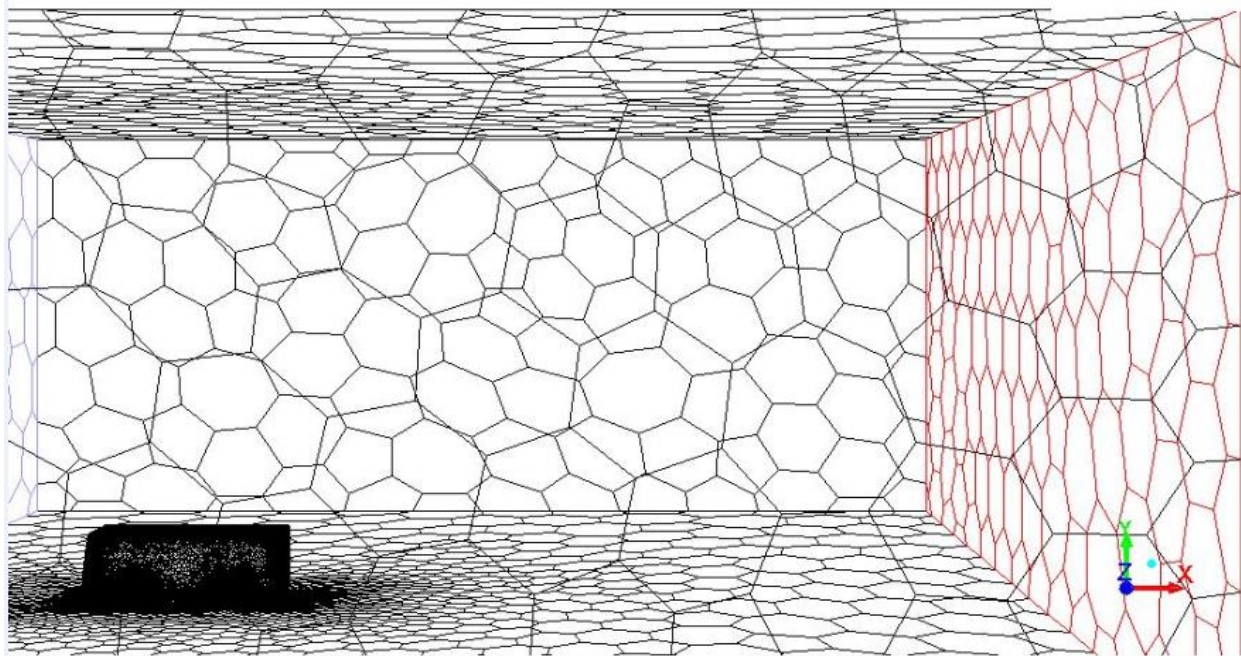


Figure A. 5: Airvane add on device used as drag reduction device

APPENDIX B: SOLIDWORKS 1/30th scaled model of commercial bus in the enclosure box and meshing used in study



(a)



Mesh

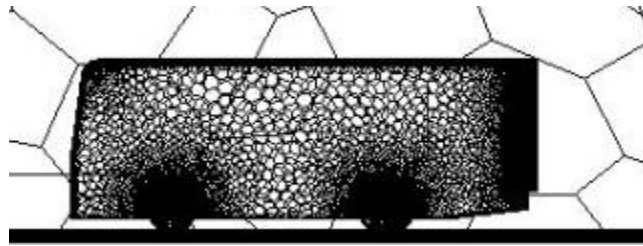
May 20, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

(b)

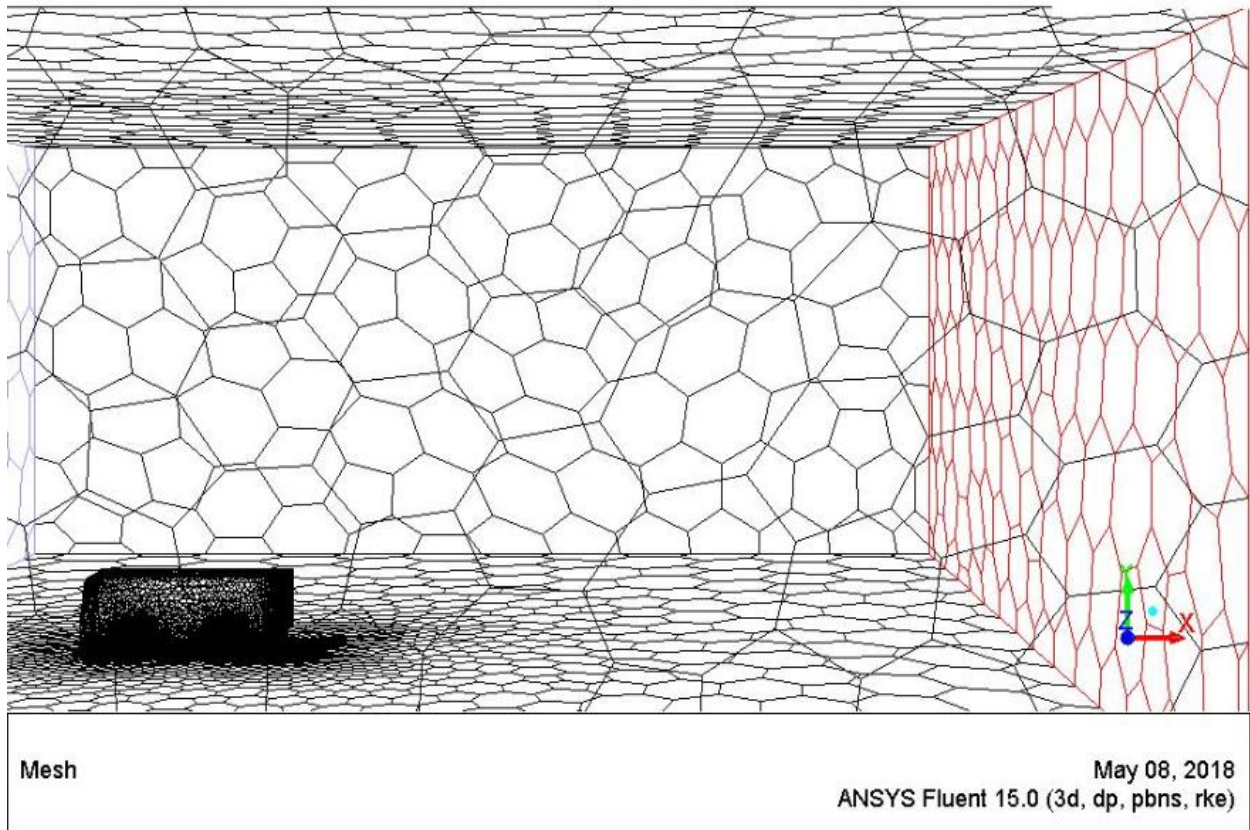
Figure B. 1: Model one in the enclosure box with converted polyhedral mesh.

Table B. 1: Mesh result when used converted polyhedral mesh for model one.

Object Name	<i>Mesh</i>
State	Solved
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Medium
Smoothing	High
Curvature Normal Angle	Default (18.0 °)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Edges
Min Size	Default (1.0507e-003 m)
Proximity Min Size	Default (1.0507e-003 m)
Max Size	Default (0.134480 m)
Growth Rate	Default (1.20)
Minimum Edge Length	3.4725e-005 m
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
View Advanced Options	No
Assembly Meshing	
Method	Tetrahedrons
Feature Capture	Program Controlled
Tessellation Refinement	Program Controlled
Intersection Feature Creation	Program Controlled
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Statistics	
Nodes	178802
Elements	950809
Mesh Metric	None



(a)

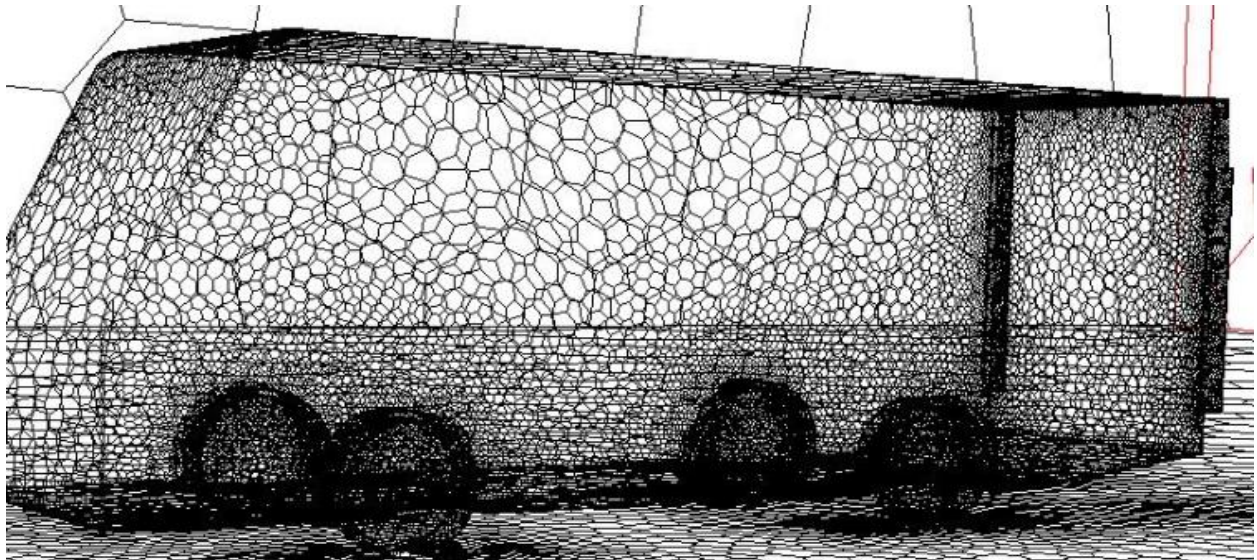


(b)

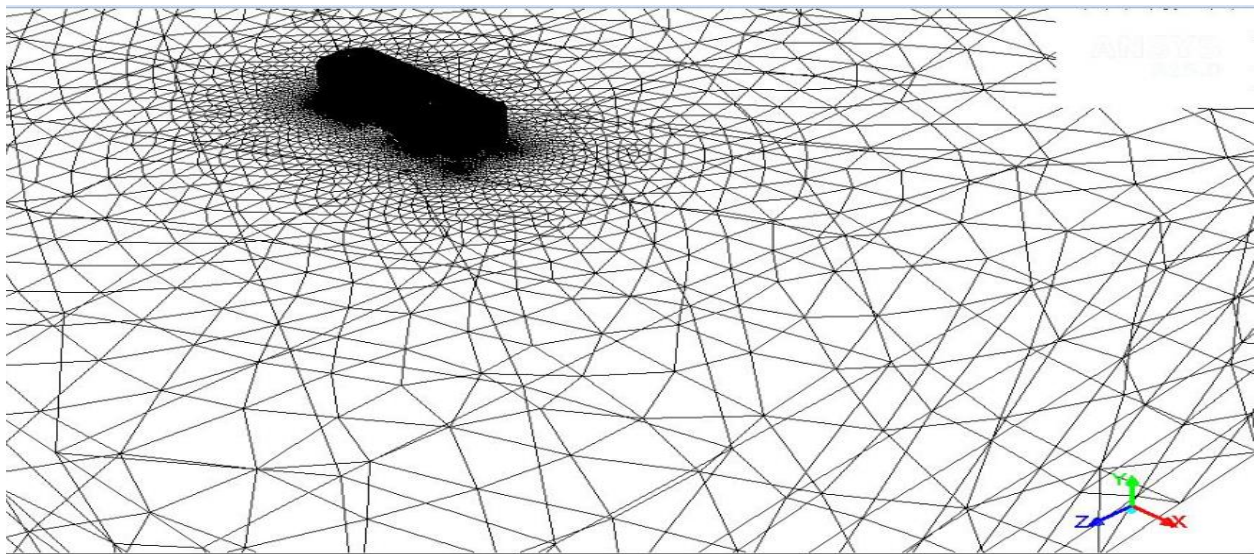
Figure B. 2: Model two when meshing using converted polyhedral mesh.

Table B. 2: Result when used converted polyhedral mesh for model two.

Object Name	<i>Mesh</i>
State	Solved
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Medium
Smoothing	High
Curvature Normal Angle	Default (18.0 °)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Edges
Min Size	Default (1.0934e-003 m)
Proximity Min Size	Default (1.0934e-003 m)
Max Size	Default (0.139960 m)
Growth Rate	Default (1.20)
Minimum Edge Length	3.e-004 m
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
View Advanced Options	No
Assembly Meshing	
Method	Tetrahedrons
Feature Capture	Program Controlled
Tessellation Refinement	Program Controlled
Intersection Feature Creation	Program Controlled
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Statistics	
Nodes	111514
Elements	580166
Mesh Metric	None



(a)



Mesh

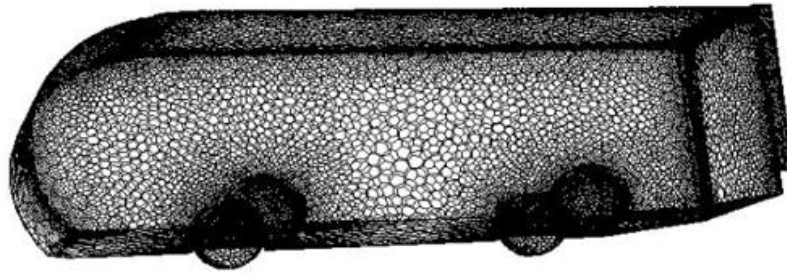
May 12, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, lam)

(b)

Figure B. 3: Model three when meshing using converted polyhedral mesh.

Table B. 3: Result when used converted polyhedral mesh for model three.

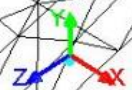
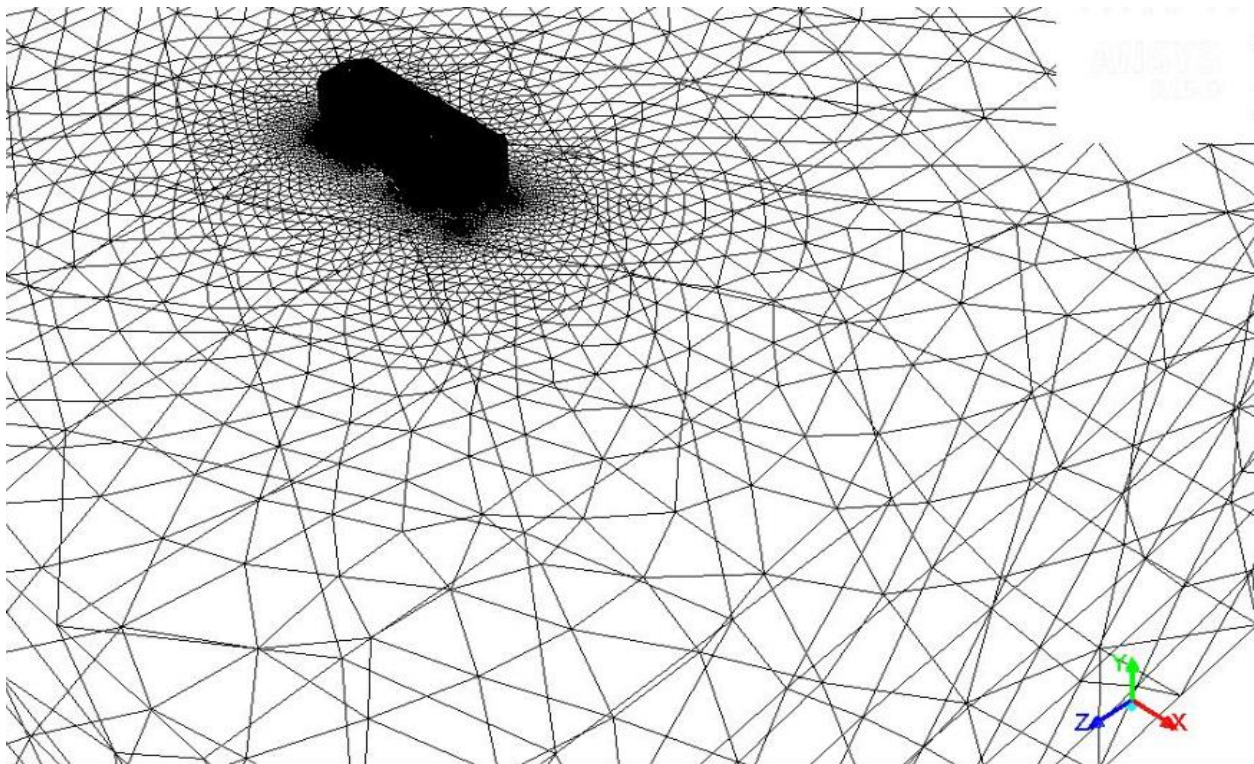
Object Name	<i>Mesh</i>
State	Solved
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Medium
Smoothing	High
Curvature Normal Angle	Default (18.0 °)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Edges
Min Size	Default (1.0934e-003 m)
Proximity Min Size	Default (1.0934e-003 m)
Max Size	Default (0.139960 m)
Growth Rate	Default (1.20)
Minimum Edge Length	3.e-004 m
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
View Advanced Options	No
Assembly Meshing	
Method	Tetrahedrons
Feature Capture	Program Controlled
Tessellation Refinement	Program Controlled
Intersection Feature Creation	Program Controlled
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Statistics	
Nodes	110697
Elements	575776
Mesh Metric	None



Mesh

May 19, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

(a)



Mesh

May 12, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, lam)

(b)

Figure B. 4: Model four when meshing using converted polyhedral mesh.

Table B. 4: Result when used converted polyhedral mesh for model four.

Object Name	<i>Mesh</i>
State	Solved
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Medium
Smoothing	High
Curvature Normal Angle	Default (18.0 °)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Edges
Min Size	Default (1.0993e-003 m)
Proximity Min Size	Default (1.0993e-003 m)
Max Size	Default (0.140710 m)
Growth Rate	Default (1.20)
Minimum Edge Length	7.531e-005 m
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
View Advanced Options	No
Assembly Meshing	
Method	Tetrahedrons
Feature Capture	Program Controlled
Tessellation Refinement	Program Controlled
Intersection Feature Creation	Program Controlled
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Statistics	
Nodes	171200
Elements	905138
Mesh Metric	None

Table B. 5: Result when used fine mesh for grid dependent test at 80kmph

Object Name	Mesh
State	Solved
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Fine
Smoothing	High
Curvature Normal Angle	Default (18.0 °)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Edges
Min Size	Default (6.3895e-004 m)
Proximity Min Size	Default (6.3895e-004 m)
Max Size	Default (8.1785e-002 m)
Growth Rate	Default (1.20)
Minimum Edge Length	3.4725e-005 m
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
View Advanced Options	No
Assembly Meshing	
Method	Tetrahedrons
Feature Capture	Program Controlled
Tessellation Refinement	Program Controlled
Intersection Feature Creation	Program Controlled
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Statistics	
Nodes	226334
Elements	1211821
Mesh Metric	None

APPENDIX C: Calibration of total pressure for all model at different speed.

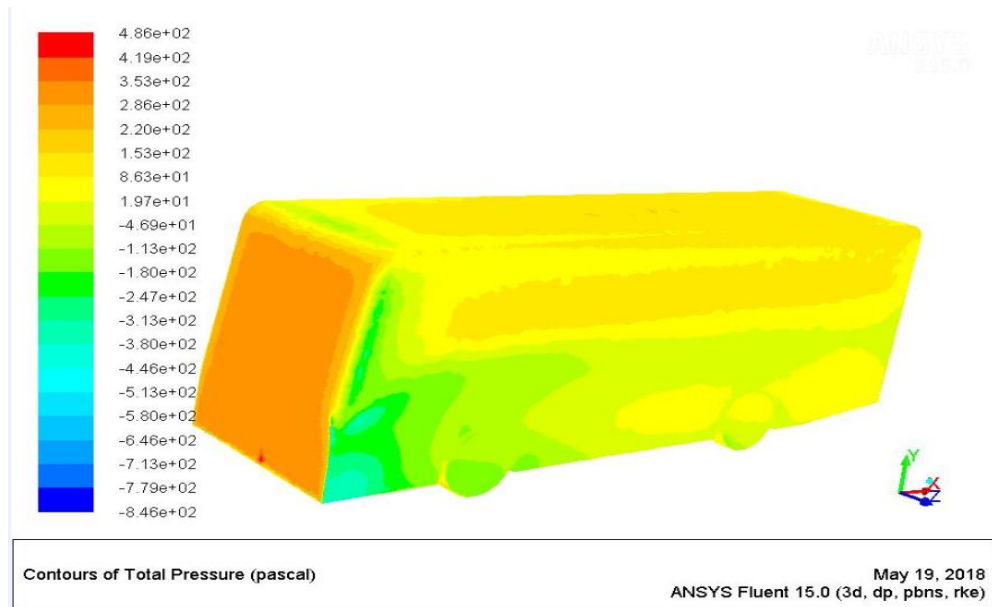


Figure C. 1: Total pressure for model one at 80kmph

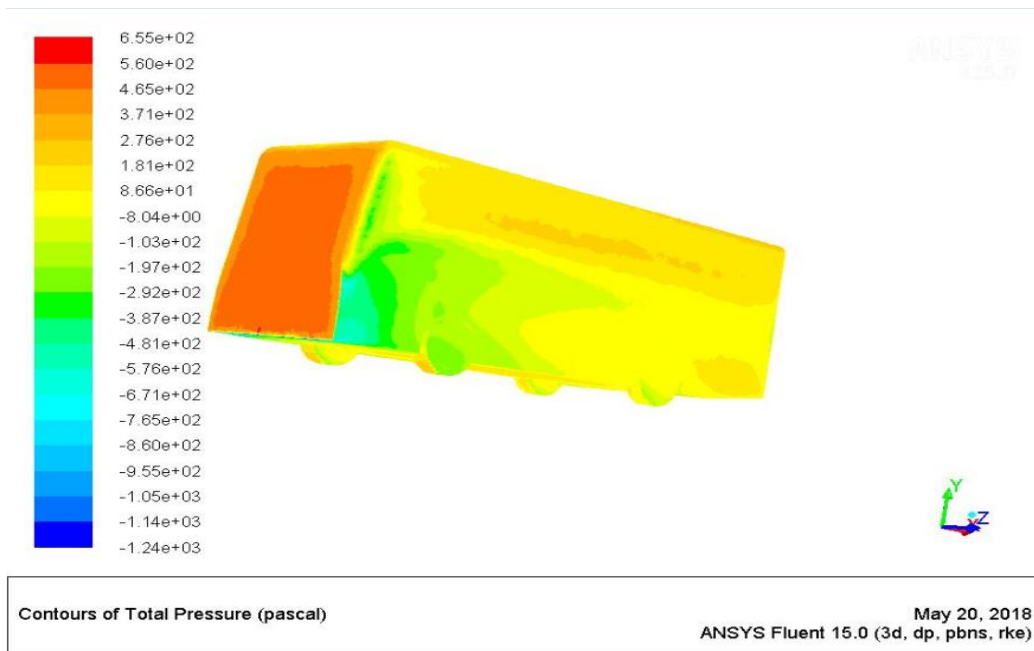


Figure C. 2: Total pressure for model one at 100kmph

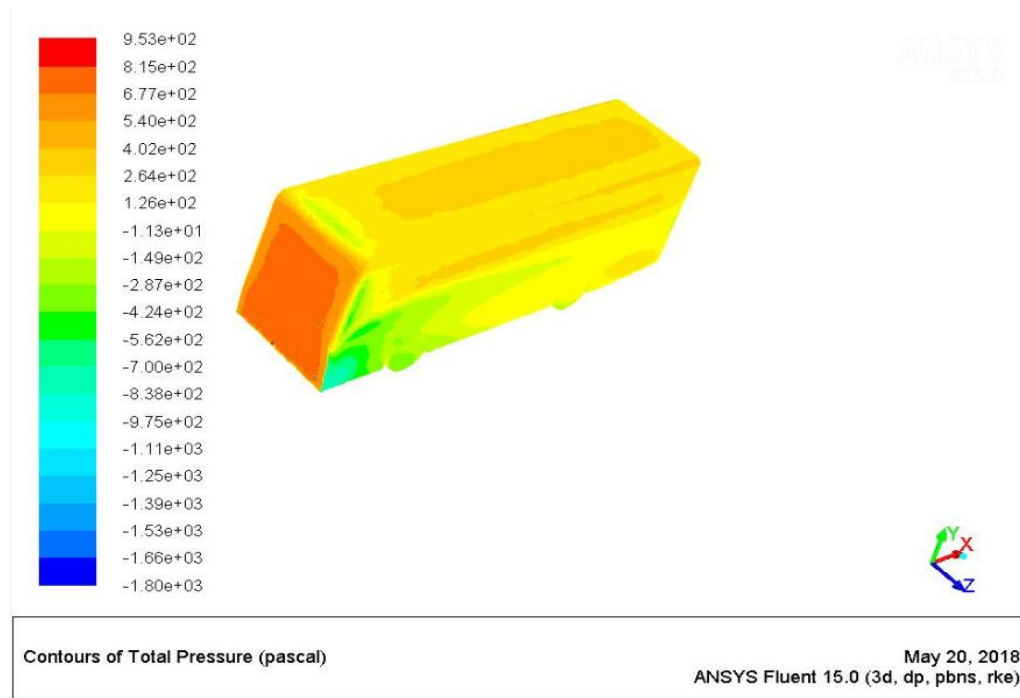


Figure C. 3: Total pressure for model one at 120kmph.

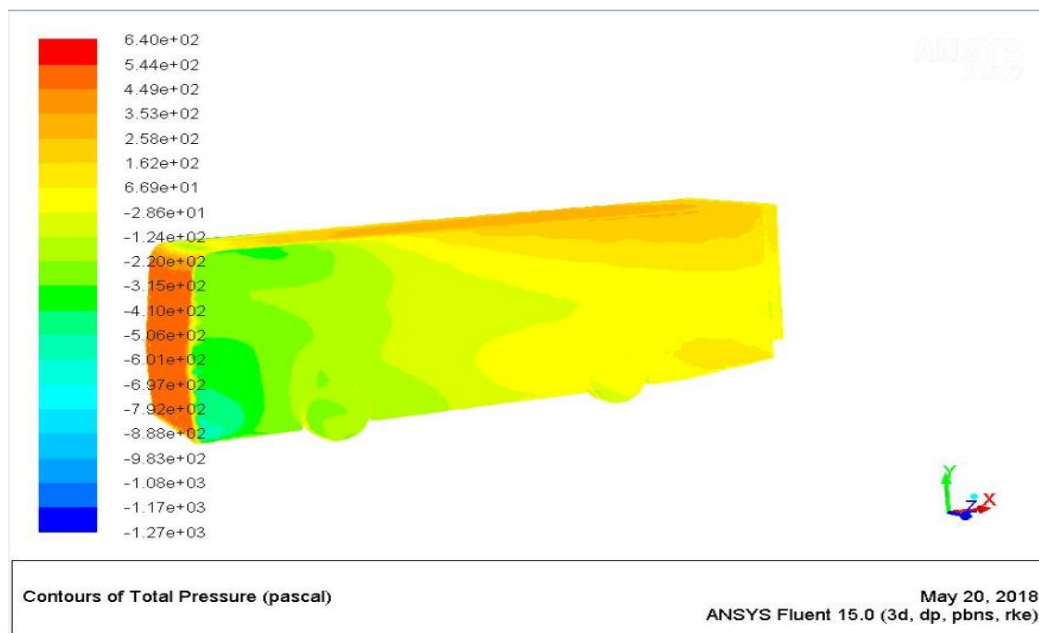


Figure C. 4: Total pressure for model two at 80kmph

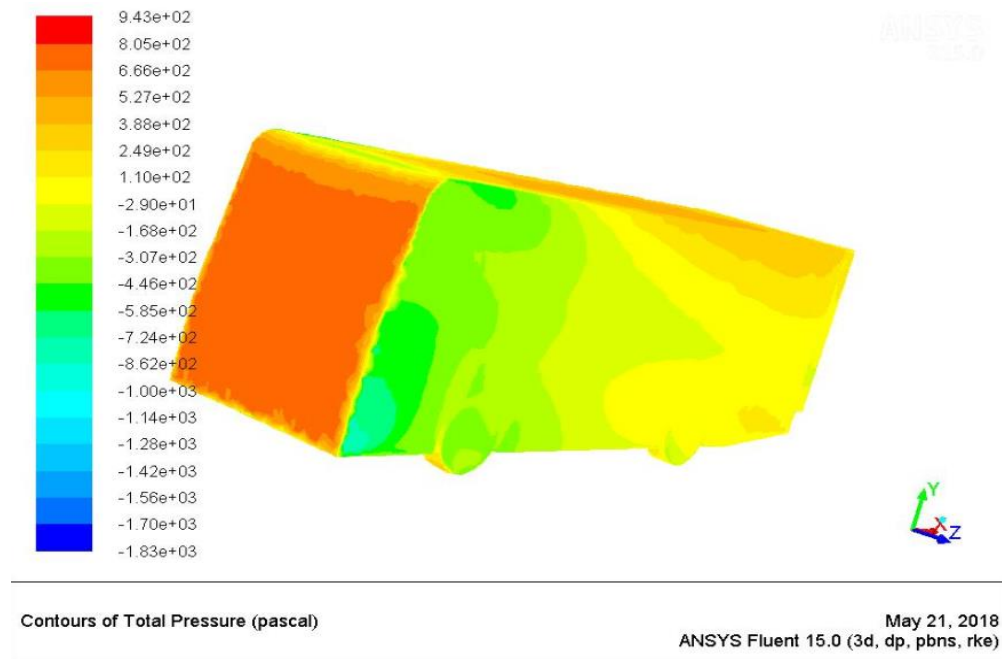


Figure C. 5: Total pressure for model two at 100kmph

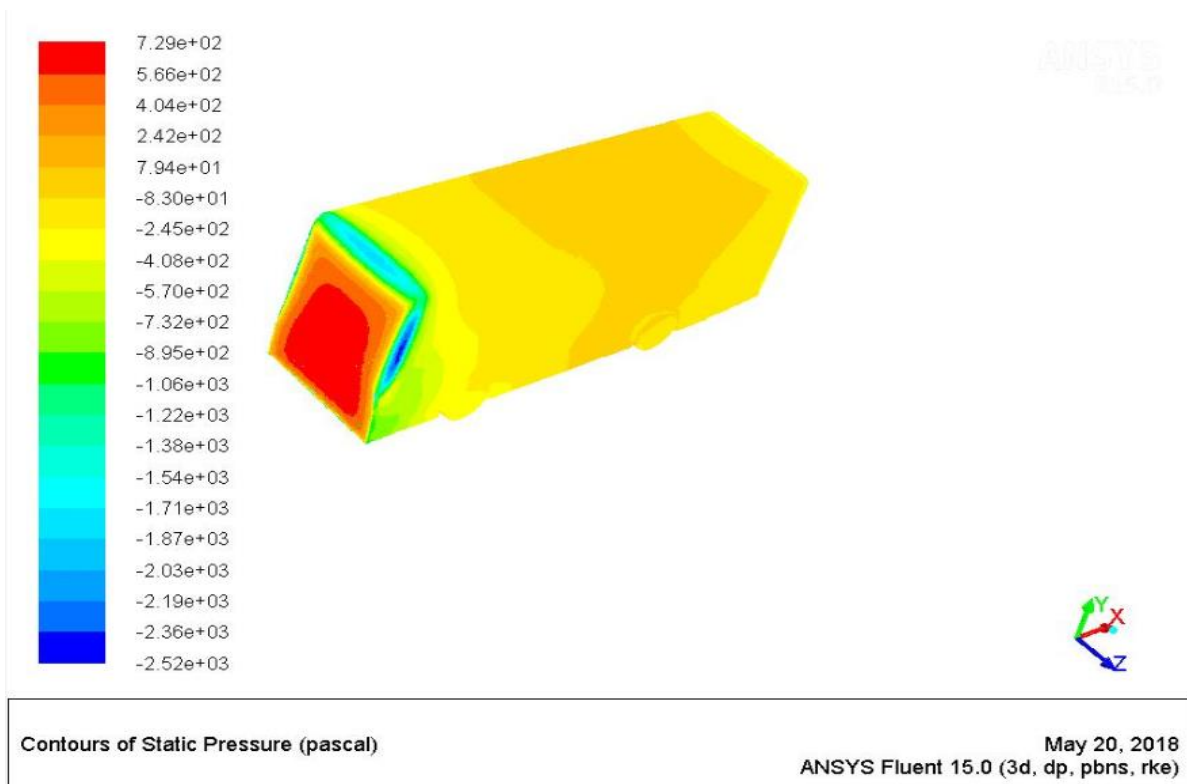


Figure C. 6: Total pressure for model two at 120kmph

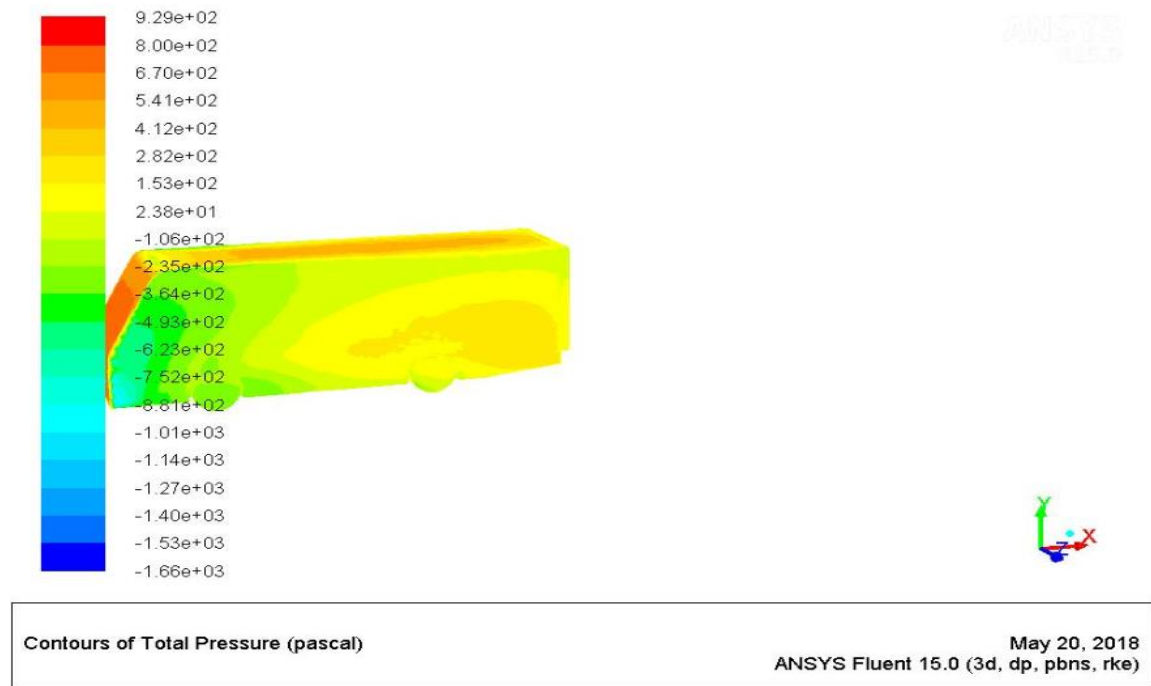


Figure C. 7: Total pressure for model three at 80kmph

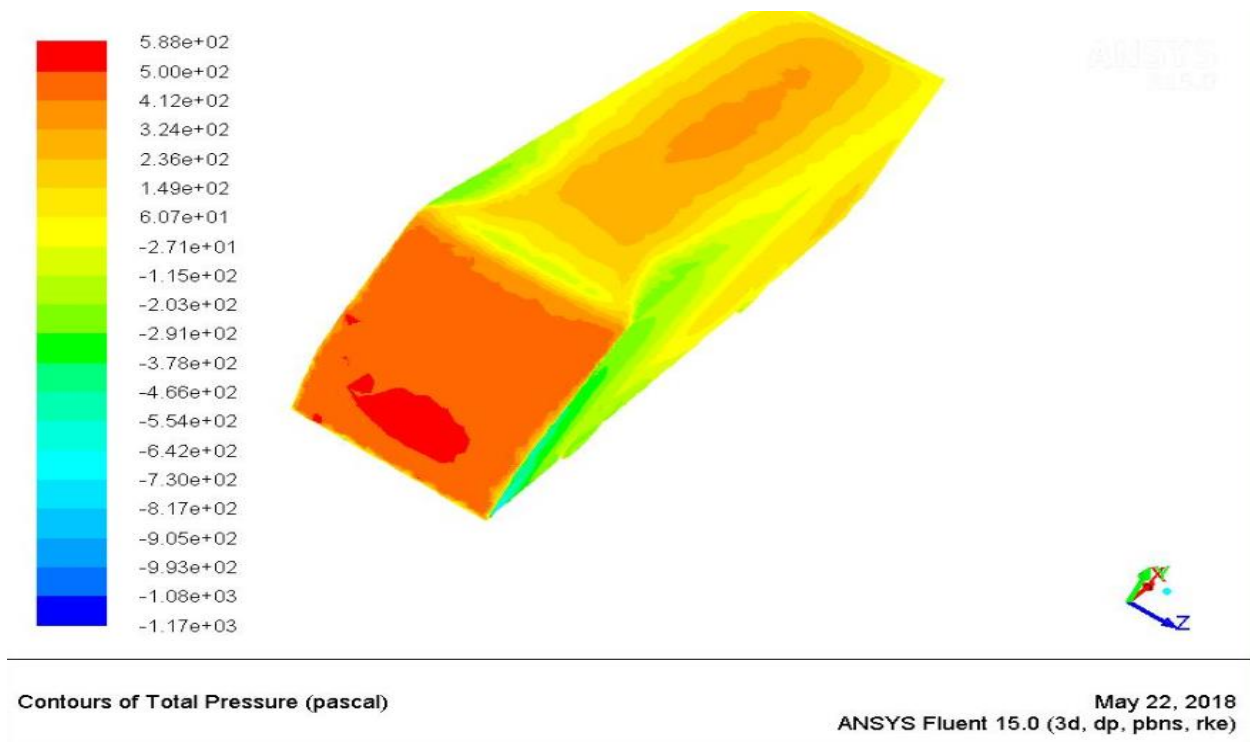


Figure C. 8: Total pressure for model three at 100kmph

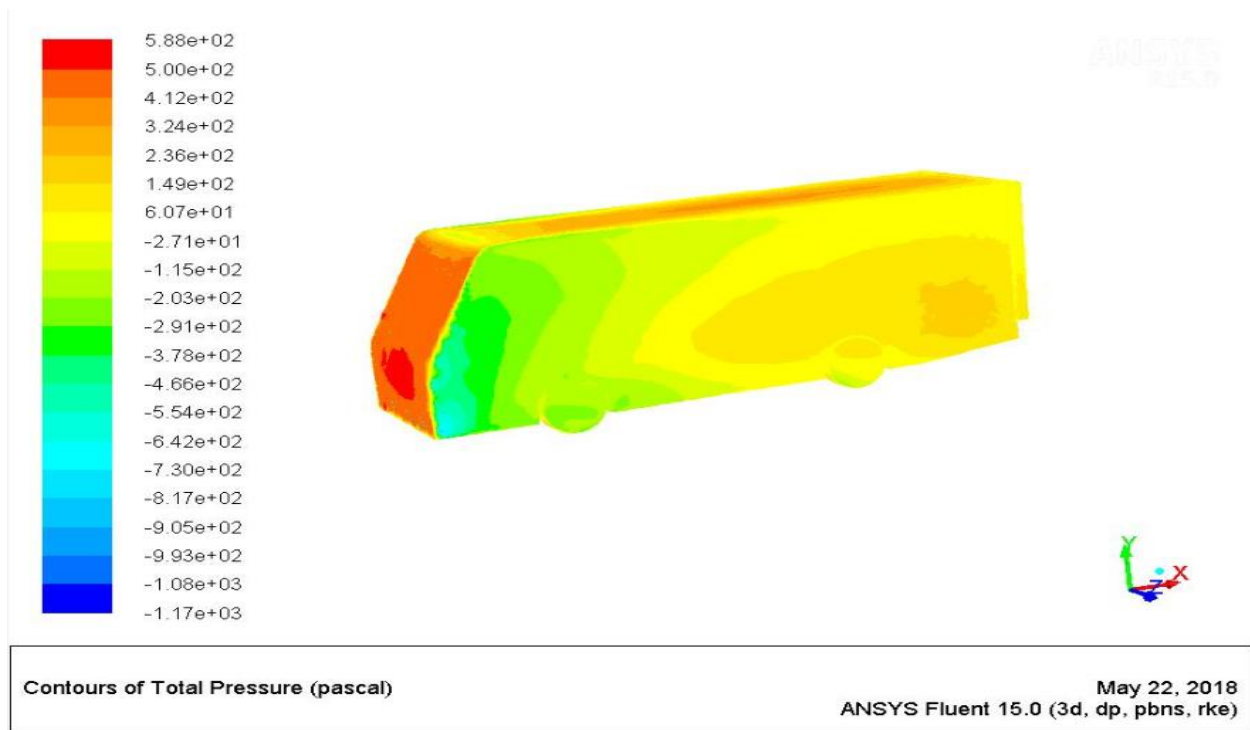


Figure C. 9: Total pressure for model three at 120kmph

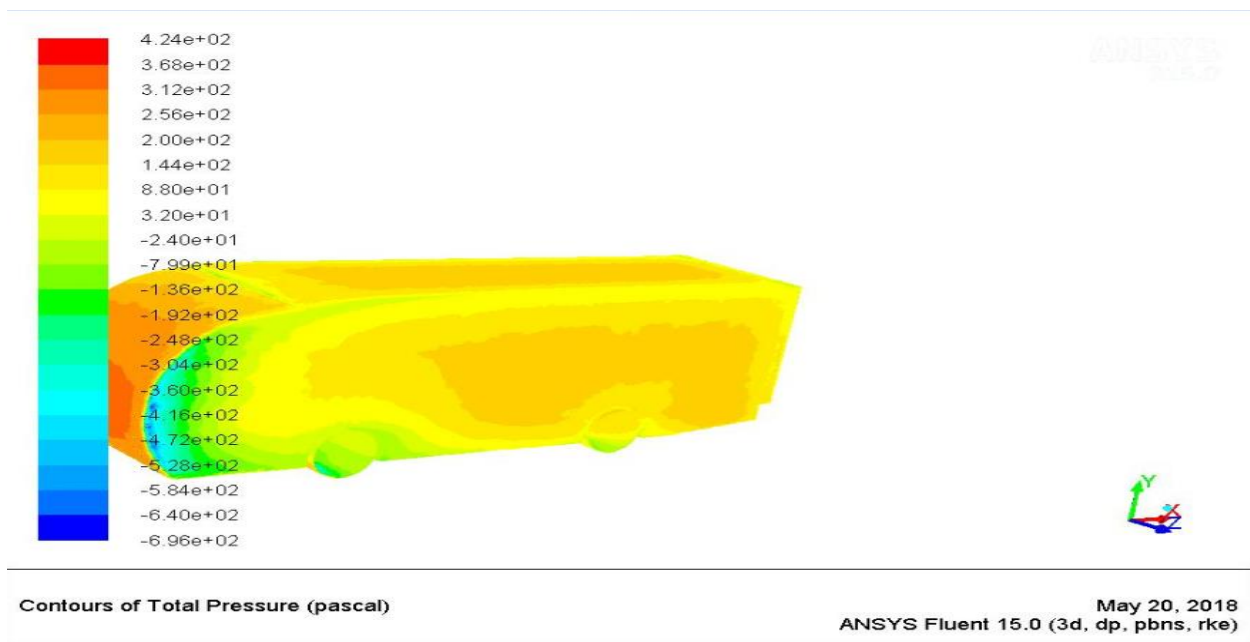


Figure C. 10: Total pressure for model four at 80kmph

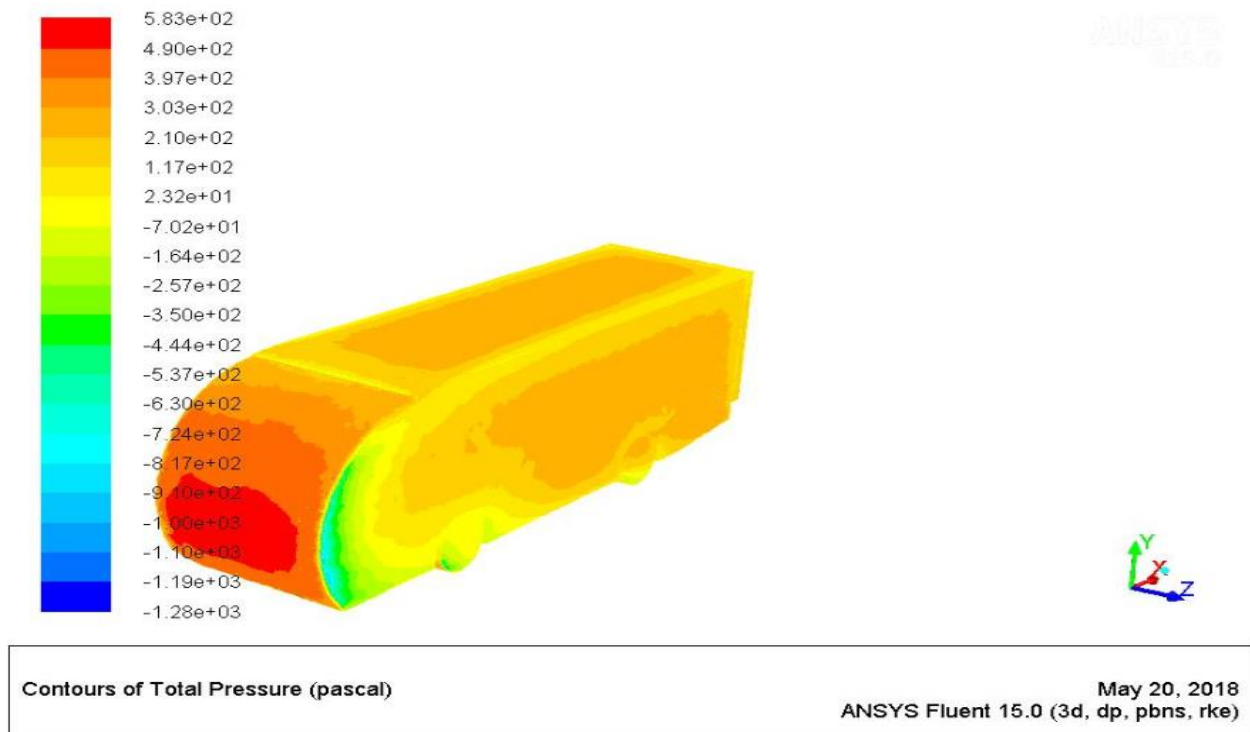


Figure C. 11: Total pressure for model four at 100kmph

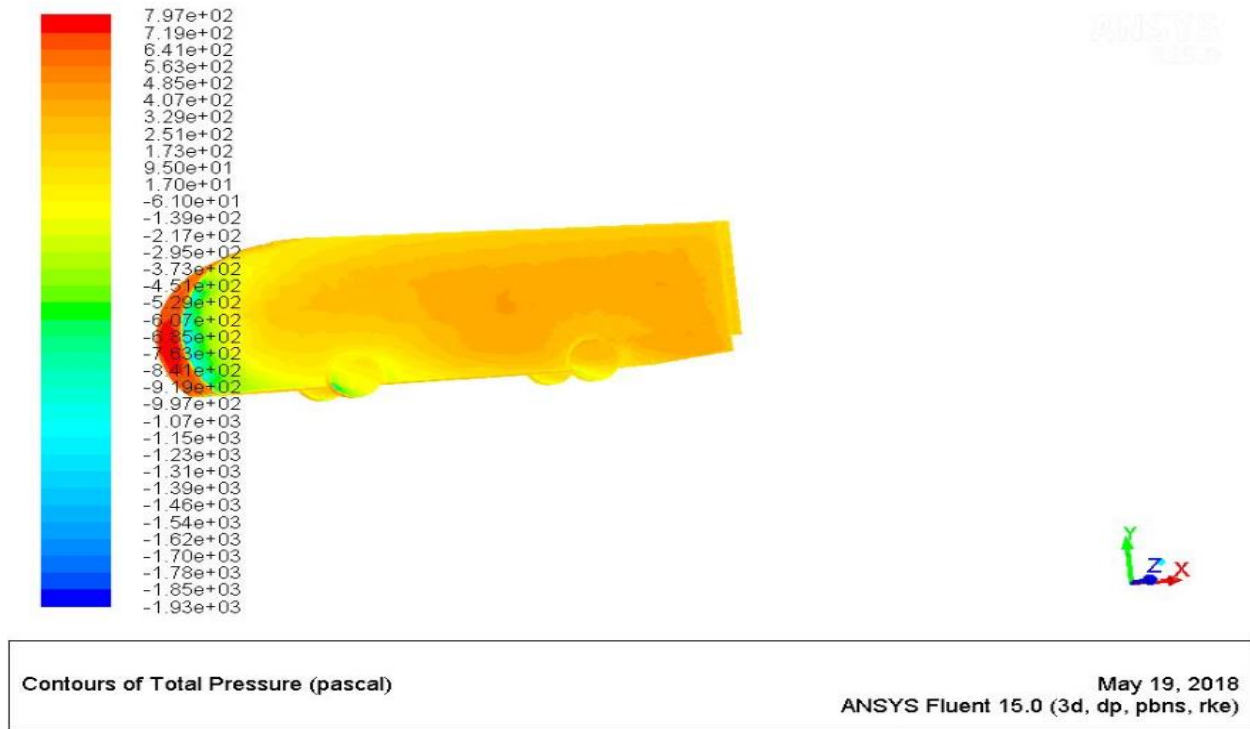
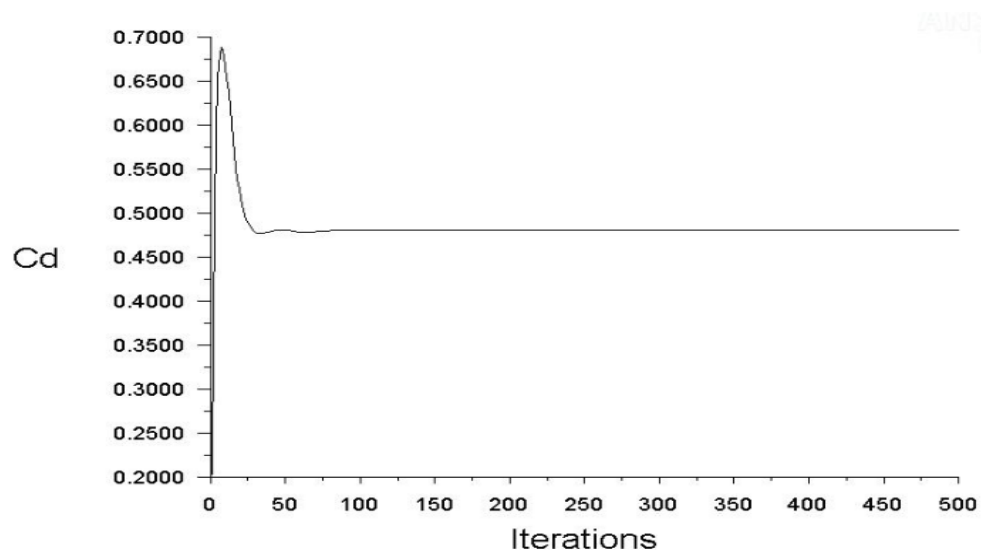


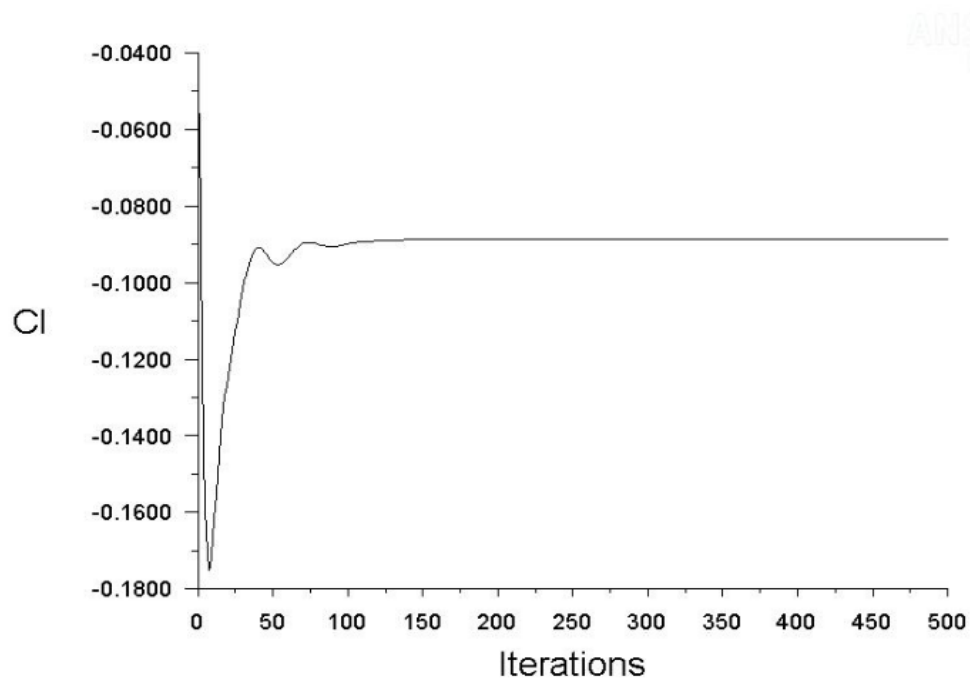
Figure C. 12: Total pressure for model four at 120kmph



cd-1 Convergence History

Jun 26, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure C. 13: graph of drag coefficient for model one at 80kmph when using fine mesh



cl-1.txt Convergence History

Jun 30, 2018
ANSYS Fluent 15.0 (3d, dp, pbns, rke)

Figure C. 14: graph of lift coefficient for model one at 80kmph when using fine mesh

APPENDIX D: Standard drag coefficient for different vehicle.

Table C. 1: Approximate drag coefficient value for various types of vehicle today and for future [Bamard,R.H. 2001]

Vehicle type	Pr-1970	current	Probable near future
Medium-sized cars	0.4-0.55	0.28-0.4	0.25
Light vans	0.4-0.6	0.35-0.5	0.3
buses	0.5-0.9	0.4-0.8	0.3
Light articulated trucks	0.7-0.95	0.55-0.8	0.4
Box truck and drawbar trailer	0.75-1	0.7-0.9	0.5