

Improving the Aerodynamic Performance of Midi Bus Passenger Vehicle Using a Rear End Spoiler (Using CFD)

**By
Lisanework Dilu**



A Thesis Submitted To

Mechanical System and Vehicle Engineering Department
School of Mechanical, Chemical and Materials Engineering

Presented in Partial Fulfillment of The Requirement for The Degree of Master
of Science in Automotive Engineering

Office of Graduate Studies
Adama Science and Technology University

Adama, Ethiopia
October, 2020 GC

Improving the Aerodynamic Performance of Midi Bus Passenger Vehicle Using a Rear End Spoiler (Using CFD)

Lisanework Dilu

Advisor: Dr. Ramish Babu Nallamothu



A Thesis Submitted To
Mechanical System and Vehicle Engineering Department
School of Mechanical, Chemical and Materials Engineering

Presented in Partial Fulfillment of The Requirement for The Degree of Master
of Science in Automotive Engineering

Office of Graduate Studies
Adama Science and Technology University

Adama, Ethiopia
October, 2020GC

APPROVAL OF BOARD OF EXAMINERS

We, the undersigned, members of the Board of Examiners of the final open defense by Lisanework Dilu have read and evaluated his thesis entitled “Improving the Aerodynamic Performance of Toyota Hiace Passenger Vehicle Using A Rear End Spoiler and A Diffuser ” and examined the candidate. This is, therefore, to certify that the thesis has been accepted in partial fulfillment of the requirement of the Degree of Master’s in Automotive Engineering.

_____ Advisor	_____ Signature	_____ Date
_____ Chairperson	_____ Signature	_____ Date
_____ Internal Examiner	_____ Signature	_____ Date
_____ External Examiner	_____ Signature	_____ Date

ADVISOR'S APPROVAL SHEET

To: Mechanical Systems and Vehicle Engineering department

Subject: Thesis Submission

This is to certify that the thesis entitled “Improving the Aerodynamic Performance of Toyota Hiace Passenger Vehicle Using A Rear End Spoiler and A Diffuser” submitted in partial fulfillment of the requirements for the degree of Master's in Automotive Engineering, the Graduate program of the department of Mechanical Systems and Vehicle Engineering and has been carried out by Lisanework Dilu Temesgen Id. No PGR/18239/11, under my supervision. Therefore, I recommend that the student has fulfilled the requirements and hence hereby he can submit the thesis to the department.

Dr. Ramish Babu Nallamothu

Name of major advisor

Signature

Date

DECLARATION

I hereby declare that this MSc Thesis is my original work and has not been presented for a degree in any other university, and all sources of material used for this thesis have been duly acknowledged.

Name:

Lisanework Dilu Temesgen

Signature: _____

This MSc Thesis has been submitted for examination with my approval as thesis advisor:

Name:

Dr. Ramesh Babu Nallamothe

Signature: _____

Date of submission: _____

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 also like to thank my mother and siblings for their support. I would like to thank my teachers Dr. Ramesh Babu Nallamothe and Mekonen Liben (Asst. Prof) for their academic support. I would like to express again my sincere gratitude for my thesis advisor Dr. Ramesh Babu Nallamothe for his excellent guidance, moral support, and constructive criticism, corrections and helps. Finally, I would like to thank my friends and Thermal and Aerospace Engineering Department Laboratory Assistants for their support. At last but not least, I have to give special thanks to Adama Science and Technology University for financial support with research grant number ASTU/SM-R/109/19.

ABSTRACT

When a body is moving through air it faces a resistance force which is acting in the direction opposite to the direction of motion of a moving body, which is the aerodynamic drag. The primary reason of this study is to improve the aerodynamic performance of Toyota Hiace midi bus vehicle by adding a spoiler at different angle at the rear end of the existing car model without changing its body styling which will result in better aerodynamic performance than the existing car model. Both computational fluid dynamics (CFD) and experimental approaches are applied to analyze the turbulence features which will result in a better aerodynamic performance than the baseline model. Solidwork flow simulation is used to perform the computational analysis which is later verified by wind tunnel testing of the baseline model. A 1/10 scaled down generic Toyota Hiace midi bus model that is modeled with solid work was used as the base line model. Improvement in drag coefficient were obtained in models with spoilers adjusted at zero degree but this resulted in an increase in lift coefficient than the baseline model. A drag reduction of 0.255% was achieved in model one but the lift coefficient increases than the baseline model by 3.454%. Model two with a spoiler having a gap of 5 cm adjusted at zero degree was used and it resulted in an increase in aerodynamic drag by 3.5% but it resulted decrement in aerodynamic lift by 46.19% than the baseline model. Model three a spoiler adjusted at 5 degree reduces the aerodynamic lift by 63% which is better than model two and the resulted increase in drag coefficient was 4.95% which is slightly higher than model two. Model four spoiler adjusted at 10 degree reduces the lift coefficient by 99.975% which is higher than all the models but the resulting increase in drag coefficient was almost increase in double than model two which is 7.32%.model two is the preferred model because of it has a better improvement in lift coefficient without affecting significantly the drag coefficient.

Keywords: Add-on Devices, Aerodynamics, Drag and Lift Coefficient, Fuel Consumption, Toyota HiAce, Turbulence Features, Wind Tunnel

TABLE OF CONTENTS

CONTENTS	PAGES
DECLARATION -----	Error! Bookmark not defined.
ACKNOWLEDGEMENT -----	ii
ABSTRACT-----	iii
TABLE OF CONTENTS -----	iv
LIST OF TABLES -----	vii
LIST OF FIGURES -----	viii
ACRONYMS -----	xiii
NOMENCLATURE-----	xiv
1. INTRODUCTION-----	1
1.1 Background of the study-----	1
1.2 Statement of the problem-----	3
1.3 Objectives-----	4
1.3.1 General Objective -----	4
1.3.2 Specific Objectives -----	4
1.4 Significance of the study -----	4
1.5 Scope of the study -----	5
1.6 Limitations-----	5
1.7 Structure of the Thesis -----	5
2. LITERATURE REVIEW-----	7
2.1 Previous Investigations -----	7
2.2 Impact of Aerodynamic Drag on Fuel Consumption-----	10
2.3 Factors Contributing to Flow Field around Vehicle -----	11
2.3.1 Boundary Layer-----	11
2.3.2 Flow Separation-----	12
2.3.3 Friction Drag -----	13
2.3.4 Pressure Drag-----	13
2.4 Drag Reduction Methods -----	13

2.5 Flow Control -----	14
2.5.1 Passive flow control (PFC)-----	15
2.5.2 Active Flow Controls (AFC)-----	17
2.6 Automotive Reference Models -----	18
2.6.1 Bluff Body-----	18
2.6.2 Ahmed Body -----	18
2.7 Computational Fluid Dynamics (CFD)-----	19
2.7.1 Stability -----	20
2.8 Wind Tunnel Testing-----	20
2.9 Turbulence Modeling -----	21
3. CFD ANALYSIS OF AERODYNAMIC PERFORMANCE OF THE BUS-----	26
3.1 Introduction-----	26
3.2. Materials -----	26
3.3 Existing and Modified CAD Models -----	27
3.3.1 Data Collection for Existing Model-----	27
3.4 BASELINE MODELS-----	28
3.4.1 Bench Marking the Baseline Model -----	28
3.5 Spoiler Cad Models and Dimensions -----	29
3.5.1 Design of a rear spoiler -----	29
3.6. ANSYS Fluent Setup and Procedure-----	36
3.6.1 Pre-Processor Stage -----	37
3.6.2 Solver Stage -----	46
3.6.3 Post-Processing -----	47
3.7 Convergence-----	48
4.EXPERMENTAL VALIDATION OF NUMERICAL ANALYSIS-----	50
4.1 Introduction-----	50
4.2 3D Model Preparation-----	50
4.3 Experimental Approach-----	52
4.4 Specification of Conventional Wind Tunnel -----	52
4.5 Testing procedure-----	54
5. RESULTS AND DISCUSSIONS-----	56

5.1 Results and Discussions of CFD Analysis -----	56
5.1.1 Baseline Model Results -----	56
5.1.2 Model One Results -----	66
5.1.3 Model Two Results -----	75
5.1.4 Model Three Result -----	83
5.1.5 Model Four Result -----	91
5.2 Comparison Between Models -----	99
5.3 Results and Discussions of Experimental Validation -----	103
6.CONCLUSIONS AND RECOMMENDATIONS -----	105
6.1 Conclusions -----	105
6.2 Recommendations -----	106
6.3 Recommendations for Future Work -----	106
REFERENCES -----	107
Appendix A: SolidWorks Modelling -----	118
Appendix B: Models Inside the Enclosure and After Meshing -----	123
Appendix C: Mesh Details -----	128
Appendix D: CFD baseline model drag coefficient result at a speed of 12m/s and 17m/s for comparison of experimental validation -----	129

LIST OF TABLES

TABLES	PAGE
Table 3. 1. Basic technical specification of Toyota Midi Bus	27
Table 3. 2. NACA 1412 Coordinate Value	30
Table 3. 3. Dimension for wind tunnel enclosure	39
Table 3.4. Boundary Conditions.....	45
Table 4. 2. Wind tunnel specifications	52
Table 5. 1. Summary of drag and lift coefficients of the baseline model at different speeds	60
Table 5. 2. Summary of drag and lift coefficients of Model one at different speeds.....	70
Table 5. 3. Summary of drag and lift coefficients of Model two at different speeds	79
Table 5. 4. Summary of drag and lift coefficients of Model three at different speeds	87
Table 5. 5. Summary of drag and lift coefficients of Model four at different speeds	95
Table 5. 6. Comparative coefficients of drag and lift for all models	102
Table 5. 7. Percent of C_l reduction from baseline model (%)	103
Table 5. 8. Cd difference (%) between experimental and CFD analysis	104
Table C- 1. Baseline Model Mesh Detail Object Name Mesh.....	122
Table C- 2. Model One Mesh Detail Object Name Mesh.....	123
Table C- 3. Model Two Mesh Detail Object Name Mesh.....	125
Table C- 4. Model Three Mesh Detail Object Name Mesh.....	126
Table C- 5. Model Four Mesh Detail Object Name Mesh.....	127

LIST OF FIGURES

FIGURES	PAGE
Figure 1. History of the drag for road vehicles	2
Figure 1.1. Toyota HiAce Engine 2KD-FTV, Model KDH202L-REMDY.....	3
Figure 2.1. Streamline of external flows around a stationary vehicle.....	12
Figure 2.2. Flow Separations at the Rear of Vehicle.....	13
Figure 2.3. Airfoil as an Example of Streamlined Body.....	14
Figure 2.4. Automobile without and with Spoiler effect (Matt Gartner, 1999, Aerodynamics) .	16
Figure 2.5. Schematic of the Ahmed body (Ahmed et al., 1984)	19
Figure 3.1. Existing Toyota HiAce midi Bus.....	29
Figure 3.2. Baseline model of a Toyota HiAce midi bus 1/10 scaled down.....	29
Figure 3.3. 0° pitch angle spoiler.....	33
Figure 3.4. 0° pitch angle spoiler with a gap.....	33
Figure 3.5. 5° pitch angle spoiler.....	34
Figure 3.6. 10° Pitch angle spoiler.	34
Figure 3.7. Assembly 3D CAD models of vehicle with spoiler configured at 0° pitch angle (model one).....	35
Figure 3.8. Assembly 3D CAD models of vehicle with a wing style spoiler configured at 0° pitch angle (model two).....	35
Figure 3.9. Assembly 3D CAD models of vehicle with spoiler configured at 5° pitch angles (model three).....	36
Figure 3.10. Assembly 3D CAD models of vehicle with spoiler configured at 10° pitch angles (model four)	36
Figure 3.11. An overview of the CFD solution procedure.....	38
Figure 3.12. An Existing model with simplification.....	39
Figure 3.13. Virtual wind tunnel targeted for CFD simulation.....	40
Figure 3.14. Mesh generation with modified sizing settings. (Baseline model)	41
Figure 3.15. Zoom of the mesh after surface sizing (Baseline model)	41
Figure 3.16. Mesh after inflation of the baseline model.....	42
Figure 3.17. Zoom of the mesh after inflation of the baseline model.....	42
Figure 3.18. Virtual car-box orientation.....	43

Figure 3.19. The final mesh.....	44
Figure 3.2.Grid Independence Test for Baseline Model.....	44
Figure 3.2.1. Boundary Condition Overview.....	46
Figure 4. 1. IIP 3D Printer.....	51
Figure 4. 2. The prototype of Toyota HiAce midi bus for wind tunnel test.....	51
Figure 4. 3. Adama science and technology university Educational Wind Tunnel.....	53
Figure 4. 4. Schematic drawing of HM 170 Educational wind tunnel (Garatebau and Barsbuttes, 2010)	54
Fig.4.5. plastic baseline model inside conventional wind tunnel.....	55
Fig.4.6. Plastic baseline model being tested at different speed.....	55
Figure 5.1. Scaled Residuals of the Baseline Model	57
Figure 5.2. Drag coefficient of baseline model at 80 kmph.....	57
Figure 5.3. Drag coefficient of baseline model at 100 kmph.....	58
Figure 5.4. Drag coefficient of baseline model at 120 kmph.....	58
Figure 5.5. Lift coefficient of baseline model at 80 kmph.....	59
Figure 5.6. Lift coefficient of baseline model at 100 kmph.....	59
Figure 5.7. Lift coefficient of baseline model at 120 kmph.....	60
Figure 5.8. Pressure contour of the baseline model at 80 kmph.....	61
Figure 5.9. Pressure contour of the baseline model at 100 kmph.....	61
Figure 5.10. Pressure contour of the baseline model at 120 kmph.....	62
Figure 5.11. Velocity contour of the baseline model at 80 kmph.....	63
Figure 5.12. Velocity contour of the baseline model at 100 kmph.....	63
Figure 5.13. Velocity contour of the baseline model at 120 kmph.....	64
Figure5.14. Surface pressure distribution along the symmetry plane of the baseline model.....	65
Figure 5.15. Scaled residuals of model one	66
Figure 5.16. Drag coefficient of model one at 80 kmph.....	67
Figure 5.17. Drag coefficient of model one at 100 kmph.....	67
Figure 5.18. Drag coefficient of model one at 120 kmph.....	68
Figure 5.19. Lift coefficient of model one at 80 kmph.....	68
Figure 5.20. Lift coefficient of model one at 100 kmph.....	69

Figure 5.21. Lift coefficient of model one at 120 kmph.....	69
Figure 5.22. Pressure contour of the model one at 80 kmph.....	71
Figure 5.23. Pressure contour of the model one at 100 kmph.....	71
Figure 5.24. Pressure contour of the model one at 120 kmph.....	72
Figure 5.25. Velocity contour of the model one at 80 kmph.....	72
Figure 5.26. Velocity contour of the model one at 100 kmph.....	73
Figure 5.27. Velocity contour of the model one at 120 kmph.....	73
Figure 5.28. Surface pressure distribution along the symmetry plane of the model one.....	74
Figure 5.29. Scaled Residuals of Model two	75
Figure 5.30. Drag coefficient of model two at 80 kmph.....	76
Figure 5.31. Drag coefficient of model two at 100 kmph.....	76
Figure 5.32. Drag coefficient of model two at 120 kmph.....	77
Figure 5.33. Lift coefficient of model two at 80 kmph.....	77
Figure 5.34. Lift coefficient of model two at 100 kmph.....	78
Figure 5.35. Lift coefficient of model two at 120 kmph.....	78
Figure 5.36. Pressure contour of model two at 80 kmph.....	79
Figure 5.37. Pressure contour of model two at 100 kmph.....	80
Figure 5.38. Pressure contour of model two at 120 kmph.....	80
Figure 5.39. Velocity contour of the model two at 80 kmph.....	81
Figure 5.40. Velocity contour of the model two at 100 kmph.....	81
Figure 5.41. Velocity contour of the model two at 120 kmph.....	82
Figure 5.42. Surface pressure distribution along the symmetry plane of the model two.....	82
Figure 5.43. Scaled Residuals of Model three	83
Figure 5.44. Drag coefficient of model three at 80 kmph.....	84
Figure 5.45. Drag coefficient of model three at 100 kmph.....	84
Figure 5.46. Drag coefficient of model three at 120 kmph.....	85
Figure 5.47. Lift coefficient of model three at 80 kmph.....	85
Figure 5.48. Lift coefficient of model three at 100 kmph.....	86
Figure 5.49. Lift coefficient of model three at 120 kmph.....	86
Figure 5.50. Pressure contour of the model three at 80 kmph.....	88
Figure 5.51. Pressure contour of the model three at 100 kmph.....	88

Figure 5.52. Pressure contour of the model three at 120 kmph.....	89
Figure 5.53. Velocity contour of the model three at 80 kmph.....	89
Figure 5.54. Velocity contour of the model three at 100 kmph.....	90
Figure 5.55. Velocity contour of the model three at 120 kmph.....	90
Figure 5.56. Surface pressure distribution for model three	91
Figure 5.57. Scaled residuals of model four at 80 kmph.....	92
Figure 5.58. Drag coefficient of model four at 80 kmph.....	92
Figure 5.59. Drag coefficient of model four at 100kmph.....	93
Figure 5.60. Drag coefficient of model four at 120kmph.....	93
Figure 5.61. Lift coefficient of model four at 80kmph.....	94
Figure 5.62. Lift coefficient of model four at 100 kmph.....	94
Figure 5.63. Lift coefficient of model four at 120 kmph.....	95
Figure 5.64. Pressure contour of the model four at 80 kmph.....	96
Figure 5.65. Pressure contour of the model four at 100 kmph.....	96
Figure 5.66. Pressure contour of the model four at 120 kmph.....	97
Figure 5.67. Velocity contour of the model four at 80 kmph.....	97
Figure 5.68. Velocity contour of the model four at 100 kmph.....	98
Figure 5.69. Velocity contour of the model four at 120 kmph.....	98
Figure 5. 70. Surface pressure distribution for model four	99
Figure 5. 71. Drag coefficient comparison of all models at different speed.....	100
Figure 5. 72. Lift coefficient comparison of all models at different speed.....	101
Figure 5.73. Comparison of C_d between experimental and CFD analysis.....	102
Figure A- 1. Baseline Model Multiview.....	111
Figure A- 2. Model One Multiview.....	112
Figure A- 3. Model Two Multiview.....	113
Figure A- 4. Model Three Multiview.....	114
Figure A- 5. Model Four Multiview.....	115
Figure B- 1. Baseline model inside the enclosure box.....	116
Figure B- 2. Baseline model after meshing.....	116
Figure B- 3. Model One inside the enclosure box.....	117
Figure B- 4. Model One after meshing.....	117

Figure B- 5. Model Two inside the enclosure box.....	118
Figure B- 6. Model Two after meshing.....	118
Figure B- 7. Model Three inside the enclosure box.....	119
Figure B- 8. Model Three after meshing.....	119
Figure B- 9. Model Four inside the enclosure box.....	120
Figure B- 10. Model Four after meshing.....	120
Figure D- 1. Drag coefficient of a baseline model at a speed of 12m/s.....	129
Figure D- 2. Drag coefficient of a baseline model at a speed of 17m/s.....	129

ACRONYMS

AFC.....	Active Flow Control
ANSYS.....	Analysis System
BL.....	Boundary Layer
CAD.....	Computer Aided Design
CFD.....	Computational Fluid Dynamics
DNS.....	Direct Numerical Simulation
KMPH.....	Kilo Meter per Hours
LES.....	Large Eddy Simulation
NACA.....	National Advisory Committee for Aeronautics
PDE.....	Partial Differential Equations
PFC.....	Passive Flow Control
RANS.....	Reynolds Averaged Naiver-Stokes
RNG.....	Renormalization Group
SST.....	Shear Stress Transport

NOMENCLATURE

C_d	Drag Coefficient
C_l	Lift Coefficient
P	Pressure
T	Temperature
V	Speed
g	Gravity
K	Coefficient of Thermal Conductivity (W/Mk)

GREEK LETTERS

ε	Epsilon
ω	Omega

SUBSCRIPTS

d	Drag
l	Lift

1. INTRODUCTION

1.1 Background of the study

Saving energy has become one of the main challenges of human beings today. Among energy consumers, transportation represents a significant part, in particular ground transportation (Bruneau.C et al., 2017).

In fact, when road vehicles move in a fluid, they have to resist force acting opposite to their Motion this force is known as the drag. When the fluid is a gas like air, it is specially called Aerodynamic drag. Most of the Power generated by the vehicle engine was consumed to overcome the resistive power which is generated by the drag force thus Improvement on the shape of the vehicle should be done in order to improve the drag. Manufacturer are working on the improvement of vehicle shape and introducing one optimal design having a similar feature from economic, functional and aesthetic consideration. This leaves little room to further improve the shape of the vehicle. Alternative technique has been used without changing the geometry of the existing vehicle this will provide a better solution (Ruiying.Li.,2017).

When fuel supply was not the main concern vehicle was manufactured to have high speed and better cabin space without considering the aerodynamic aspect as well as how much fuel will it consumes. However sudden hike in fuel price and also exhaust emissions into the environment force the researchers to design the car aerodynamically. Drag can be reduced by making the shape of the vehicle more aerodynamic and also by making the vehicle made of lightweight materials. Pressure drag on the vehicle body covers almost all the aerodynamic drag of the vehicle which is 80 % of the resulting aerodynamic drag whereas friction drag constitutes 20 % of the total aerodynamic drag. Flow separation on the vehicles body results in a higher pressure drops and formation of wake behind the vehicle (Abedin. M et al., 2019).

In order to decrease the formation of wake behind the vehicle body the geometry of the vehicle should be designed carefully. However, vehicles that of buses and trucks are made of square back and it's difficult to change into another style. Flow control devices are used as a best solution for making the flow as per our interest (Siniša.K et al., 2011).

Development on the external body features of the vehicle combined with attachments on the vehicle body achieves a remarkable reduction on the drag of a vehicle as illustrated by Figure 1.

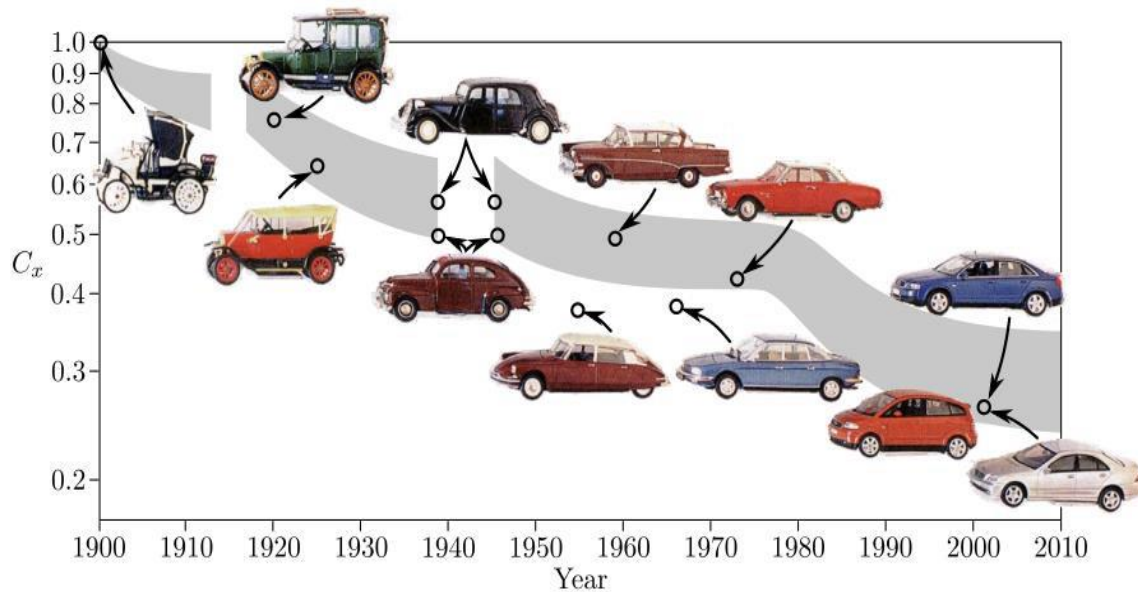


Figure 1. History of the drag for road vehicles

Larger contribution to the aerodynamic drag arises from pressure drag which is happened when there is a sudden pressure drop due to flow separation happened. Improvement in the characteristics of wake results in a better reduction of aerodynamic drag. Vortices are generated when there is flow separation and it's happened in the wake (Bahram. K et al., 2012)

Aerodynamic drag is subdivided in to two forms that are frictional drag and pressure drag. The dominance in friction drag on the vehicle body mean the vehicle body is stream lined whereas the dominance in pressure drag mean the vehicle body is bluff.

Toyota HiAce is a light commercial van produced by the Japanese automotive manufacturing Toyota the external feature of this vehicle is somewhat streamlined at its front and the back of the vehicle is bluff or square like which leads the streamlined air to separate and causing a higher pressure drop to happen and the flow feature becomes turbulent at the back of the vehicle. This will result in a formation of wake region leads to a higher pressure drag.



Figure 1.1. Toyota HiAce Engine 2KD-FTV, Model KDH202L-REMDY

The research developed today in car aerodynamics is carried out from the point of view of the durable development. Some car companies have the objective to develop control solutions able to reduce at least 30% of the aerodynamic drag of the vehicles without constraints on the design, the comfort, the storage or the safety of the passengers. Thus, it is necessary to modify locally the flow, to remove or delay the separation position or to reduce the development of the recirculation zone at the back and of the separated swirling structures (Charles.H et al., 2010)

Low pressure was created in the region where wake is formed this contributes 30 % of the aerodynamic drag which is mostly happened on vehicles made of a square back feature. The resulting low pressure can be reduced by designing a vehicle aerodynamically. Passive as well as active flow control devices can be used to reduce the formation of higher drag on the vehicle body (Littlewood et al., 2011).

1.2 Statement of the problem

Toyota Hiace is a light commercial vehicle produced by the Japanese automotive manufacturing Toyota. Toyota HiAce was available in plenty amount in our country Ethiopia working as transportation and it's clear that this vehicle loses its stability while driven at higher speed and make an accident to happen. Improvement in stability of the vehicle while traveling at higher

speed should be done in order to reduce the vehicle accident due to losing of control of the vehicle. To improve the vehicle tendency of losing stability at higher speed Aerodynamic devices were used. A rear spoiler is designed to change the flow of air over the rear of the vehicle. This change in airflow increases down force on the rear of the vehicle to help improve traction in an effort prevent sliding while cornering. Benefits like maintaining traction, increase braking stability and stylish looking of the vehicle can be improved with the addition of the spoiler at the rear roof end of the vehicle.

1.3 Objectives

1.3.1 General Objective

The general objective of this thesis work is to improve the aerodynamic performance of Toyota Hiace passenger vehicle by adding a spoiler device at the rear roof end of a vehicle.

1.3.2 Specific Objectives

The specific objectives are:

- To prepare models of a midi bus vehicle both existing and improved one using solid work (CAD modeling software).
- To design a spoiler that can be attached to the existing vehicles to reduce the aerodynamic lift of the vehicle.
- To prepare models of a midi bus vehicle an existing baseline performing flow analysis using wind tunnel experiment.
- To perform the flow analysis on the existing baseline and improved models using CFD tool fluent software at different speed (kmph) parameter to analyze the drag and lift forces as well as visualize the flow pattern.

1.4 Significance of the study

Nowadays, the shapes of vehicles are very alike because all the manufacturers are pursuing the shape optimization for drag reduction and tend to follow one unique optimal shape under the similar constraints from functional, economic and aesthetic aspects. This leaves little room to reduce further the drag from the geometry of vehicles. Therefore, alternative technologies

without the geometry modification should be flourished to provide more design liberties. The final outcome of this study would give better contribution for vehicle manufacturers which lead to improve the fuel economy, engine performance and exhaust emission to the atmosphere during the usage of automotive.

The result of this paper can be useful as a reference for individuals, researchers and manufacturers who are engaged in vehicle body design, to reduce the drag and lift forces. It is true that aerodynamics can be improved by properly designing various body profiles and adding add-on devices. It is also expected to give better understanding on wake and turbulent structure of the vehicles.

1.5 Scope of the study

This study is delimited only on improving the aerodynamic performance of Toyota HiAce passenger vehicle by adding a rear end spoiler as well as analyze the airflow around the vehicle using CFD and validate the obtained computational results of the baseline model experimentally using wind tunnel experiment test.

1.6 Limitations

In the automotive industry, aerodynamic analysis carried out, principally, in two ways: Experimentally (Wind-tunnel test) and numerically using ANSYS Fluent software. To perform the numerical analysis, it's necessary to use a high-performance computer to obtain accurate results. Moreover, there are no large-sized powerful Wind-tunnels and high cost of preparing improved 3D model of a Toyota HiAce is the main limitations. The study also focuses on a simplified 3D model of Toyota HiAce body aerodynamic performance excluding parts such as side-view mirrors, windscreen wipers, mud flaps and roof rack while designing the models.

1.7 Structure of the Thesis

This Thesis consists of six chapters. The first chapter is an introduction to the study. It includes the background of the study, statement of the problem, objectives, and scopes, significant of the study, limitation, and structure of the thesis. Chapter two focuses on different kinds of literature that emphasize on study of bus aerodynamics and computational fluid dynamics

analysis. Chapter three describes CFD analysis of aerodynamic performance of the bus and methodology used to complete this work. Also, deals with bus model concepts geometry development and simulation setup. Chapter four presented experimental validation of the computational analysis. Chapter five depicted results and discussions of computational analysis and experimental validation. The drag and lift coefficients are presented in form of tables and graphics for quantitative and qualitative discussions. The final chapter of this thesis consists of conclusions and recommendations.

2. LITERATURE REVIEW

In the design process of a car, we have to consider aerodynamic drag. A car is accepted when it's designed to have less aerodynamic drag and for that many researchers conduct computational fluid dynamics (CFD) techniques to analyze the air flow characteristics around a vehicle body and put a recommended shape that the vehicle should have. Studies that are helpful to addressing various aspects related to this study are collected in this section. At the end of this chapter the research gap is pointed out.

2.1 Previous Investigations

Engidayehu (2019) Adama, Ethiopia, Paper Presentation of CFD Analysis on Aerodynamic Drag Reduction of Pickup Vehicles Using Rear Spoiler. In this paper air flow over a pickup vehicle with spoiler attached on the rear end of the vehicle roof at different inclination angle and spoiler length using computational fluid dynamics (CFD) with the objective of reducing the aerodynamic drag was studied. All numerical simulations were performed using commercial CFD software Fluent and realizable k-epsilon turbulence modeling. A 1/12 scaled down generic pickup model with extended cab modeled with Solid Works was used as the baseline model. Improvements in drag coefficient and lift coefficient were obtained in models with spoiler when compared to the baseline model without any add-on device. A drag reduction of 6.50%, 6.12%, and 0.493% was achieved on model one, model three and model two respectively when compared to the baseline model. Because of the reduction in drag a maximum of 3.90% reduction in fuel consumption was achieved. The lift coefficient of the baseline model was also reduced to 22.47% on model two, 14.55% on model three and 4.30% on model one at a vehicle speed of 120 kmph.

Lealem (2018) Adama, Ethiopia, Paper Presentation of Study and Aerodynamic Drag Reduction on Locally Built Bus Body Using CFD Analysis: A Case Study on Bishoftu Automotive Industry. 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 air vanes and front modification simultaneously as drag reduction devices, were employed on the model bus to reduce the drag coefficient of the bus. 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.

Alamaan et al. (2014) in this study MAN TGX truck series was studied by adding a flap at its rear end with the help of computational fluid dynamics (CFD) software. The flap mounting angle also studied to determine the best angle that the flap should be mounted on the roof. First elliptical shape flap of 1.2 m semi major axis was studied and the resulting drag reduction was 11.1%. a rectangular and triangular flap of having the same area with elliptical flap also studied and the resulting drag reduction was 6.37% and 6.84% respectively. Flap best mounting angel obtained for both was 50°.

Anna et al.(2016) in this study a simplified model which is having a square back was investigated. Tapers are made on the square back of a reference model on its top and also its bottom which in turn results in a change of the wake nature. Tapering of 4 % of the reference model length for both the top and bottom of the square back gives an improvement in the drag reduction obtained.

Alessandro et al.(2019) in this study the integration of vertical and horizontal deflectors on a square back car model was investigated at a different slant angle. The velocity features on the near wake are also studied at a various angle of deflectors. A deflector positioned vertically has played a great roll in the improvement of drag as well as lift when it's positioned at a specific height in combination with a horizontal deflector. The optimal height of vertical deflector was 2/3 of the model height that gives a good reduction in the drag as well as lift and the best mounting angle for both deflectors was obtained 20 from the free stream direction.

Hanfeng et al.(2016) in these study deflectors was used on the 25° slanted Ahmed model to reduce the drag. Deflectors were mounted at leading and side edge of the slant. The length of deflectors mounted at the top edge and side edge of the slant had a height of 1%, 2%, 3% of the length of the Ahmed model respectively. When there is no deflector mounted in the leading edge and side edge of the slant there was a result in pressure reduction which is a source for higher aerodynamic drag to happen. With the increase of deflector height to 2 % and 3 % of the model length the resulting drag reduction obtained was 3.5 % 7.2 % respectively. The drag reduction by increasing the width of a deflector 1%, 2% and 3% of the model length results in 9.3 %, 10.7 % and 10.9 % drag reduction respectively. For Ahmed body 25° slant angle the leading edge of the slant is efficient in reducing the drag than the side edge of the slant.

Ankush et al. (2017) in this study numerical technique was used to analyze the drag on realistic car model. Ahmed body having a deflector at its rear end was used as a representative for the realistic model. A deflector was adjusted at different angle to obtain a considerable drag reduction. First Inlet velocity of 16 m/s and 40m/s was studied at a deflector angle varying from -25° to 60 ° and gives a considerable result than the existing results on the literature. Again, inlet velocity of 20 m/s and 30m/s also studied and the resulting drag is reduced by 7 % than without a deflector.

Grandemange et al.(2015) in this study improvement in the drag coefficient of scaled Ahmed body by chamfering the top and bottom of the square back were obtained. The improved Ahmed body at optimal chamfer angle gives a drag reduction of 5.8 % than the reference Ahmed square back model. Yaw angle effect on the drag also studied, for yaw angle interval of $\pm 0.5^\circ$ drag coefficient becomes unchanged.

Kim et al .(2017) in this study improvement in the drag by the addition of cab roof fairing on the scaled truck model without cab roof fairing was studied. Numerical simulations as well as wind tunnel testing were used to investigate the results. First 2D cab roof fairing was used and the resulting drag reduction was 15.05% than a track without a cab roof fairing and the second test was held by rounding the 2 D cab roof faring in its sides and the resulting drag reduction of 15.97% was obtained which is a little higher than the first investigation. A full modification on a 2D cab roof fairing results in a dreg reduction of 19% which is much better than a 2D cab

roof fairing this is achieved by further rounding of the sides of cab roof fairing to make it more stream lined from the sides.

Kim et al.(2019) in this study roof fairing, side skirts and also boat tail was investigated on tractor having a trailer to make it aero full package (AFP) and the resulting drag reduction was studied. The reference tractor trailer model has a drag coefficient value of 0.693 and the improvement obtained by using aero full package was 26.5% less than the reference model and also a full saving of 13.4% was obtained.

Gaylard et al.(2017) in this study the contamination on the rear of a vehicle due to spray from the tire was investigated. Contamination from the spray on the tire on to the rear of a vehicle happens due to the wake formed at the rear of the vehicle. This was result in reducing vehicle visibility and also vehicles visibility on to the rear of the vehicle due to contamination on the rear of a vehicle. With the help of eddy simulation software, the deposition of contamination due to the turbulence happened on the rear of a vehicle was studied and this were result in an increase in the base pressure.

2.2 Impact of Aerodynamic Drag on Fuel Consumption

The increase in fuel consumption that is the results of ineffective in the aerodynamic shape of a vehicle which is in turn results excessive drag thus improvement in aerodynamic drag is important for the improvement of fuel consumption. 77% of all domestic and imported petroleum consumed by ground vehicles in the United States. Estimations showed that 1 % improvement in fuel economy results in saving of 245 million of gallons of fuel per year. It also accounts for approximately 30% of CO₂ emissions and for the other greenhouse gas emissions. The engine output power is then used to overcome the aerodynamic drag of 53% and also the rest of 32% to overcome the rolling resistance and for drive train 6% only used.15% improvement in aerodynamic drag for a vehicle traveling at high speed will result in improvement of 5-7% fuel consumption (Bellman et al., 2010).

Because of sudden increase in the price of fuel and problem in increase of greenhouse gas emissions automotive engineers are facing a challenge every time in the introduction of better aerodynamic shape vehicle, When the shape of a vehicle is not designed without the

consideration of aerodynamic drag, that results in an increase in aerodynamic drag which further results in the increase in demand for fuel consumption. Causes of aerodynamic drag arise from the pressure of and form drag. Pressure drag arises from flow separation this constituent the major aerodynamic drag that is approximately 80% (Mohd et al ., 2014).

The flow over three-dimensional bluff bodies is of particular interest due to its relevance to automotive vehicles. Such bodies experience large pressure drag due to the large region of separated flow in the wake. For automotive vehicles operating at motorway speeds, this pressure drag is responsible for a significant proportion of the fuel consumption; therefore, its reduction is a topic of key interest (Brackston et al., 2018).

The shape of road vehicles when looking at its back is bluff bodies and the pressure drag constitutes the total aerodynamic drag that is caused by flow separation at its rear end of the vehicle. Flow separation is the main cause for an increase in drag of a vehicle. Improvement in flow separation where subjected in fluid dynamics for many decades to control flow separation with the help of flow control devices (Basara et al., 2013).

2.3 Factors Contributing to Flow Field around Vehicle

The major factors, which affect the flow field around the vehicle, are the boundary layers, separation of flow field, friction drag and lastly the pressure drag (Cakir., 2012).

2.3.1 Boundary Layer

Boundary layer (BL) is one of the major criteria in the design of a vehicle body. For a laminar boundary layer, the flow attachment with the car body is smaller as compared with a turbulent boundary layer. When air is blowing on a flat plat aligned with the direction of flow the friction drag is created on the surface of a flat plat and the resulting drag is because of skin friction and When a flat plate is aligned normal to the flow direction the resulting drag is because of the pressure difference between the front of a plat and its back.

Vehicle is moving at a certain velocity, the viscous effects in the fluid are restricted to a thin layer called boundary layer. Outside the boundary layer is the in viscid flow. This fluid flow imposes pressure force on the boundary layer. When the air reaches the rear part of the vehicle,

the fluid gets detached. Within the boundary layer, the movement of the fluid is totally governed by the viscous effects of the fluid.

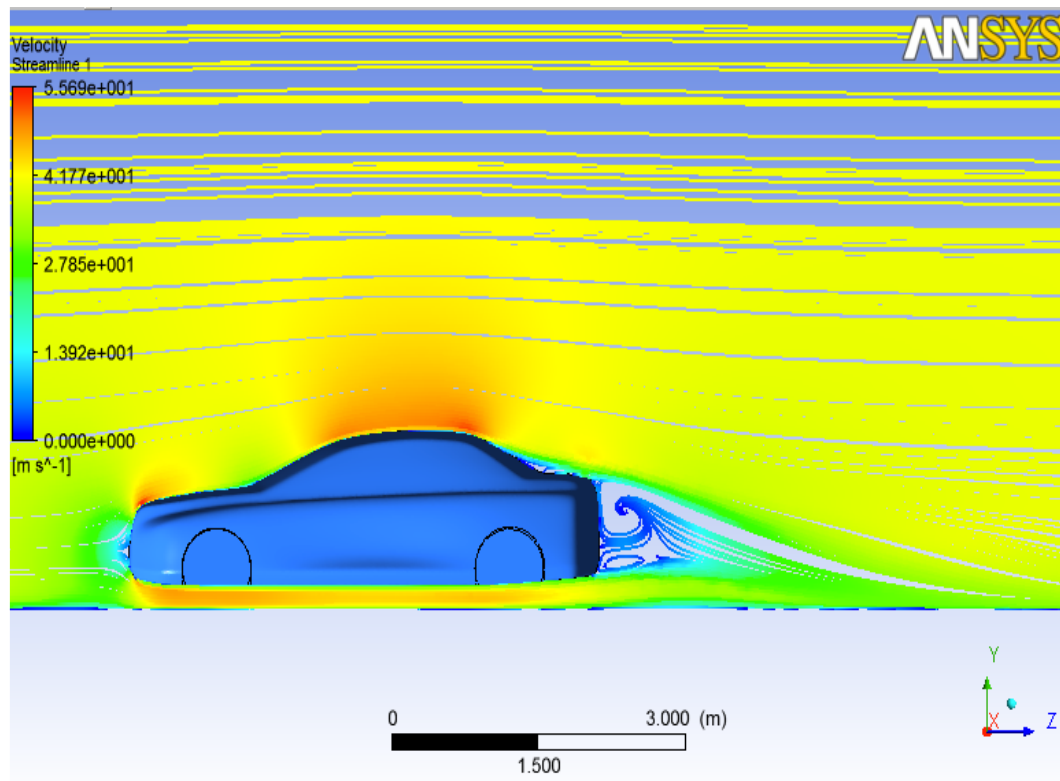


Figure 2.1. Streamline of external flows around a stationary vehicle.

2.3.2 Flow Separation

When air is flowing on a car body it gets separated during this the change in velocity became stall this makes the fluid to move in the reverse to the direction of flow what we call it flow separation. The major existence of flow separation is happened on the rear portion of the vehicle and it's dependent on the pressure difference between the flow of air before separation and after separation. The behavior of flow behind the vehicle changed and flow characteristics become turbulent this is the major challenge in the design of a car body and the resulting phenomenon is known as a wake. Flow separations are bad because it leads to a larger wake and less pressure on the rear surface which reducing pressure recovery. To avoid bad flow separation, the transitions of the airflows from roof to the rear window need to be smoothed. The bad separation also can create more drag. The aerodynamic will be more effective if the flows working in clean

air laminar flow. By improving the aerodynamic of the car can reduce the boundary layer thickness thus avoids worst flow separations.

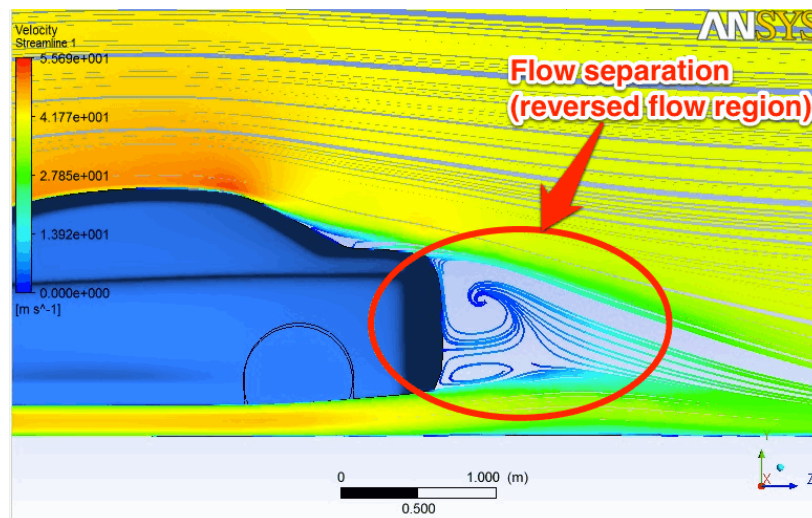


Figure 2.2. Flow Separations at the Rear of Vehicle.

2.3.3 Friction Drag

Every material or wall has a distinct friction, which resists the flow of fluids. Due to molecular friction, a stress acts on every surface of the vehicle. The integration of the corresponding force component in the free stream direction leads to a friction drag. If the separation does not occur, then friction drag is one of the main reasons to cause overall drag.

2.3.4 Pressure Drag

Behind the vehicles, there is a steep pressure gradient, which leads to the separation of the flow separation in viscous flow. The front part of the flow field shows high-pressure value, whereas on the rear part flow separates leading to a high suction in the area. As we integrate the force component created by such high change in pressure, the resultant is called as 'Pressure Drag'. This factor is affected by the height of the vehicle as well as the separation of the flow field.

2.4 Drag Reduction Methods

Drag reduction techniques are proposed for many decades to make a significant change in aerodynamic drag. How and where to fit the flow control device are the basics beside the design

of flow control devices to improve the performance since aerodynamics drag is depend on mainly the velocity of the vehicle (Filippone et al .,2010).

Streamlined bodies are known in that the air flow is attached to the vehicle body in this case the aerodynamic drag is created by the friction drag only and its value become very small. The drag is smaller because of a minimum value of shear stress created between the air flowing on the car body and the car body itself. Airfoil is the only design which results a better streamlining of the air flow over a car body (Lajos ., 2002).

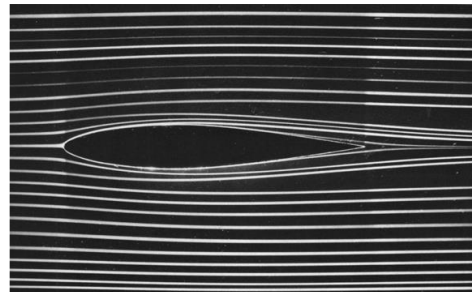


Figure 2.3. Airfoil as an Example of Streamlined Body.

The Various design changes like side tapering at rear, front spoiler and streamlined body with tapering at rear comparing with existing bus model results a minimum drag. A whale shape face design, front face curvature, roof tapering, roof end lowering, side panel tapering and boat tailing also the different techniques applied to vehicle body. Add-on on the roof of vehicle causes the stability issues and the aerodynamic drag increases (Chorage et al., 2020).

2.5 Flow Control

Flow control devices are classified into active and passive devices the classification is based on whether it works with the input of energy or it works by its own. Active flow control devices are work with energy input whereas passive flow control work without any energy input. Due this active flow control mechanism has a better quality of control but the mechanisms they use are complex. During selection of flow control devices include industry standardization weight/size, cost effectiveness fabrication, installation, driver training, and modifications compatibilities, energy/power consumption, and maintenance and vehicle usage (Asrar et al., 2014).

2.5.1 Passive flow control (PFC)

The passive control systems consist of the use of more or less discrete obstacles (attaching add-on devices), added around or on the roof of the vehicle to reduce the aerodynamic drag. They can be declined in two groups according to their influence on the flow control. The first group consists of 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 (Kourta and Gillieron, 2009). This approach includes shape modification on the vehicle body. The following are some examples of passive flow control methods.

2.5.1.1 Effect of spoiler on aerodynamic performance

When a car is traveling at a higher speed the vehicle faces lots of air resistance some of the air flow streamlines travel above the car body while some of the streamline travel below the car body the above air flow streamlines and below air flow streamlines should be laminar throughout the travel of the car length but in actual practice this does not happen. When the flow streamline travel in a straight path the flow is said to be laminar but when the streamline travel not in a straight path the flow is said to be as turbulent. In actual practice the frontal portion has laminar flow but the air flow turns to turbulent when it arrives at the rear end flow turns to turbulent because of sudden decrease in area of a car and also there is a there is a pressure drop in this area. Pressure drop is because of the velocity of air flowing above the car body is high as compared to the velocity of air bellowing the car body. According to bernollious principles for a given volume of air the higher the velocity of air molecules are traveling the lower the pressure became thus as there is low pressure above the car body and there is low pressure bellow the car body the car tends to experience a lift force this lifting force might lead to failure of traction control. Overall there are two external forces which are experienced by the car body when it's moving at a high speed, we call it lift and drag. Spoilers are installed at the rear end of the vehicle to improve vehicle tendency to lift, to give a better traction control, and to improve handling, to make air flow over the car body laminar, and also to improve drag. This are the reasons why spoilers are installed at a vehicles body (Kyei ., 2014).

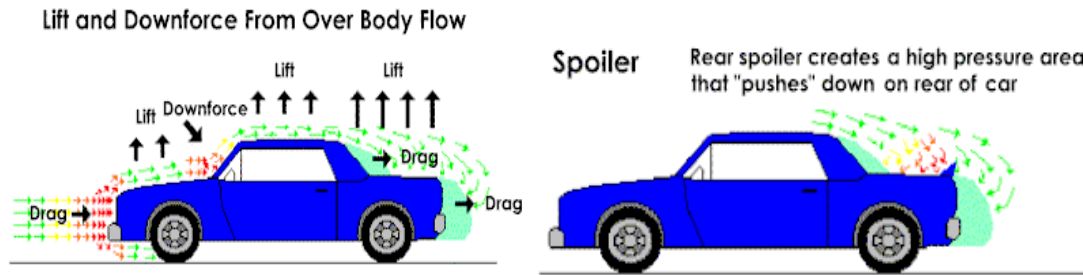


Figure 2.4. Automobile without and with Spoiler effect (Matt Gartner, 1999, Aerodynamics).

As we see that the vehicle at the left side is without a spoiler and its traveling at a higher speed this will create an equal and opposite force from the air and this force opposes the movement of the vehicle towards the front what we call it aerodynamic drag. This shows us that the power output from the engine should be higher in order to overcome the drag thus it has to produce more power this again results in higher fuel consumption and also emission towards the environment also became higher. The vehicle becomes lighter when it travels at a higher speed too.

Invariably there is little stability in the motion car, and this can make braking very difficult as well as negotiating a curve.

Therefore, to minimize the effect of the drag there is a need to create a greater down force, thus the introduction of rear spoilers to an automobile, which is shown on the right-hand side of figure.

In order to minimize the effect of drag we have to create down force this will lead as to the introduction of spoilers at the rear portion of the vehicle and it shown at the vehicle which is at the right side. Spoilers are designed in the shape of airfoil since it creates a pressure difference while the air is moving over it and generating a down force which is important for the vehicle. Spoilers give a better stability, better control of the car during cornering since it creates a greater down force on the car body.

2.5.1.2 Effect of diffuser on aerodynamic performance

Diffuser is installed on a car body to improve the cars lift tendency. Diffuser are based on the principle of that as air travels from a narrower area to a wider area its pressure drops since its speed gets increase this effect suck the air which is present in the under body of a car and the pressure in the under body of a car became lower. This will again create a pressure different between the vehicle upper body and lower body which is a down force creation tendency (ZHAO et al., 2011).

2.5.1.3 Effect of Vortex generator on aerodynamic performance

Vortex generators (VG) are small triangular or vane shaped protuberances that have been used successfully in automotive and aerospace applications to stir the boundary layer and delay flow separation over a wing or body surface (Brownlie et al., 2016).

Moving air reacts with a solid object because air has friction or viscosity and it gets stick. Free stream of air moving along a solid object the air next to that surface gets slow down by it and the that friction propagates upward this area of slow down air is known as the boundary layer. As air blows around a curved surface it would like to follow the curved surface all the way round but in real practice this will not happen because the air moves from an area of lower pressure to area of higher pressure this creates a turbulent boundary layer. The air separated and creates a reverse flow thus installing vortex generator on a car body is to create a swirling bubbles which is having a direction of rotation opposite to that of created on the rear portion of a car and this opposite direction swirling bubbles reduces vortices created at the vehicle rear end thus improvement in aerodynamic drag can be achieved (Imine et al., 2016).

2.5.2 Active Flow Controls (AFC)

Active flow control devices are working with a power supply which is coming out of the vehicle engine. Actuators are used it has a visible part which is having mobile walls, slots which is distributed on the vehicle body in which flow control is required. It requires electromagnetic, mechanical, electrical, piezoelectric or acoustic system are placed on the hollow portion on the vehicle. To reduce impact on habitable volume and consumption their overall dimension and their weight must be small as much as possible.

2.6 Automotive Reference Models

There are different reference models in the automotive industry to study the airflow around a vehicle. All of them are considered as Bluff bodies. Every vehicle reference model has particular geometric shapes to study the fluid flow behavior around the models, under some defined conditions. The comprehension of that behavior on simple shapes helps to understand what will happen with more complex shapes. The most important reference model is the Ahmed model.

2.6.1 Bluff Body

When the length of the body is close to perpendicular to the air flow streamline the body is said to be bluff body. In this case the skin friction is very much lower than the pressure that is acting on the bluff body. When an airfoil is having a larger angle of attack even though it having a streamlined shape it might behave as a bluff body. The characteristics of bluff body are known by creating a disturbed flow behind the wake region this is happened because of flow separation is happened. Many of the experiments and numerical studies had been studied on the formation of vortices and the wake which is near the surface of the vehicle to have better information about the phenomenon and provide better solution (Cooper., 2006).

2.6.2 Ahmed Body

Ahmed body used in automotive industries as a famous benchmark it's categorized under bluff body and its main function is to show the nature of turbulent flow in most of bluff body vehicles. Flow separation in Ahmed body exists at its front body and also at its rear end this contributes a major pressure drop to happen which contributes 80% of the total aerodynamic drag. The frontal body should be rounded to avoid flow separation and the rear body should be studied at a different angle of slant in order to improve aerodynamic drag on bluff body vehicles (Ahmed et al ., 1984).

S.R Ahmed first created the Ahmed body and it became a benchmark for aerodynamic simulation work bench tool. The geometry has a length of 1.044m, height of 0.288m, and a width of 0.389m. The leg of Ahmed body is cylindrical shape and its 0.5m in length and the

rear surface has a slant in its rear of the roof which is 40° from the horizontal (Ashwini et al., 2016).

The principal objective to study such a simplified car body is to tackle the flow processes involved in drag production. Through perceiving the mechanisms involved in creating drag one can be able to design a car to minimize drag and therefore reducing fuel consumption and maximize vehicle performance.

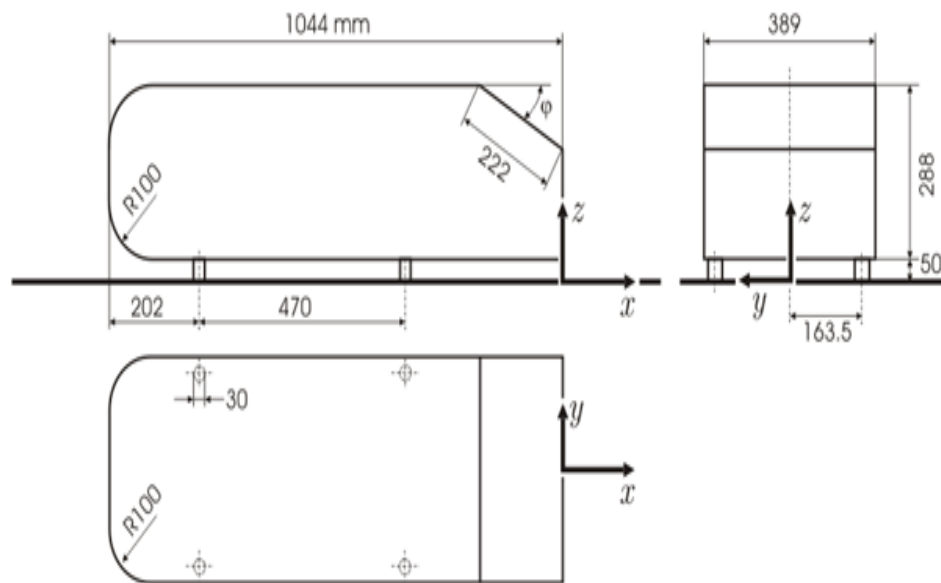


Figure 2.5. Schematic of the Ahmed body (Ahmed et al., 1984)

2.7 Computational Fluid Dynamics (CFD)

Computational fluid dynamics (CFD) is the one that is used to solve problems that are difficult to solve analytically. Before computers are used to solve fluid dynamics problems, we use two approaches to solve for problems that are analytically and experimentally. Analytically we can only solve 1D or 2D geometry problems and also experimental method need resources of electricity, data monitoring, data post processing, and also expensive equipment's thus it's not recommended for most of the small organizations to work on experimentation. CFD solves all the problem and make life easier it works with computers which is modern. This day numerical methods become essential to solve many complex problems and the introduction of CFD makes

problems much easier. Today we can buy supercomputers with a small investment and run CFD code that can work for complex geometry safely.

In general, CFD is better when compared to experimental. In the 20th century problems seem to become easier with this modern technology. In terms of accuracy experimental is the replica of real phenomenon and analytical method uses some assumption and become the second where as CFD the last of all it has a rounding off errors, traction errors, and machine errors in numerical methods.

CFD helps to design jet engine whose body is an example of complex geometry, curvatures and faces can be considered. CFD helps to design after burner mixers which is having a function of providing thrust additionally for a greater maneuverability (Thabit et al., 2018).

2.7.1 Stability

A numerical solution method is said to be stable if it does not magnify the errors that appear in the course of numerical solution process. For temporal problems, stability guarantees that the method produces a bounded solution whenever the solution of the exact equation is bounded. For iterative methods, a stable method is one that does not diverge. Stability can be difficult to investigate, especially when boundary conditions and nonlinearities are present. For this reason, it is common to investigate the stability of a method for linear problems with constant coefficients without boundary conditions. Experience shows that the results obtained in this way can often be applied to more complex problems but there are notable exceptions. The most widely used approach to studying stability of numerical schemes is the von Neumann's method. However, when solving complicated, non-linear and coupled equations with complicated boundary conditions, there are few stability results so we may have to rely on experience and intuition. Many solution schemes require that the time step be smaller than a certain limit or that under-relaxation be used (JAMSHED., 2018).

2.8 Wind Tunnel Testing

Wind tunnel testing has the big advantage that once the vehicle model is produced and rigged in the wind tunnel test section, it can quickly provide highly accurate data. Data for different boundary conditions, such as different wind speeds and yaw angles, can be acquired quickly. If

similar changes in conditions are done on a computer model, the whole simulation has to be run over again for each case (Devaradjane, 2015).

On the other hand, wind tunnel testing can be both highly costly and time-consuming. The wind tunnel itself is a huge investment, and the production of prototypes can be very expensive. Small changes in design will take much more time to implement on a physical prototype than on a computer model. The accuracy of the wind tunnel measurements is affected by several factors, including blockage, scaling effects and the moving road problem, and the reliability and the validity of the data need to be evaluated in each case.

The wind tunnel is used for solving the problem of aeronautical, space, automobile and civil engineering structures, which are best obtained rapidly, economically and accurately by testing the scaled models, and sometimes actual structure in wind tunnels. The size, speed and other environmental conditions of tunnel are determined by actual users, the speed determines the type of the tunnel namely subsonic, near-sonic, transonic, supersonic and hypersonic. While the speeds of these tunnels are obviously named with reference to the sonic, velocity, the low-speed tunnel which is below 300 mph. An alternative definition of the low-speed tunnel would be the tunnel where the compressibility of air is negligible.

2.9 Turbulence Modeling

Wind tunnel testing has the big advantage that once the vehicle model is produced and rigged in the wind tunnel test section, it can quickly provide highly accurate data. Data for different boundary conditions, such as different wind speeds and yaw angles, can be acquired quickly. If similar changes in conditions are done on a computer model, the whole simulation has to be run over again for each case.

Turbulence is a fluid flow phenomenon that can mainly be characterized by unsteadiness, irregular fluid motions on a wide physical scale range and rapid mixing. Flow fields around bluff bodies, such as the buses can be considered to have extremely turbulent flow patterns. Separation bubbles, boundary layers, and wakes present on bluff bodies are examples of turbulence within a flow field.

The largest difficulty with CFD simulations is to calculate the turbulent flows. A turbulent flow is irregular and varies randomly in time and space. With existing computer capacity, it's impossible to solve Navier-Stokes equations exact. By using so-called turbulence models, the flow can be calculated with far less computer capacity. Such a model will modify the equations and will consider only the average effects of the turbulence. The flow will be divided into an average term and a fluctuation term. A turbulence model can never give an exact solution, but an accurate solution close to the real value. For the anticipated future, real-life fluid dynamic problems still require turbulence models in order to solve these. Reynolds-Averaged Navier–Stokes (RANS) equations combined with turbulence modeling provide an adequate model of turbulent flow effects. No single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation (Wilcox., 2003).

There are mainly three categories of turbulence models such as Classical Turbulence Models (also known as Reynolds Averaged Navier-Stokes), Direct Numerical Simulation (DNS) and the Large Eddy Simulation (LES). Algebraic (Zero-Equation) Models, One-Equation Models (for example Spalart-Allmaras), Two-Equation Models (k-epsilon, k-omega, SST, etc.) and Second-Order Closure Models all these four types of models are referred to as RANS turbulence models (Wilcox., 2003).

Each turbulence models have their own advantage and disadvantage. Understanding their strengths and weaknesses is of paramount importance in order to achieve accurate results from CFD simulations. In this study, the two-equation k-epsilon turbulence model has been chosen for reasons that are explained in the following section.

K-Epsilon Turbulence Model

The K- ϵ model is the most common turbulence model approach used in the engineering field due to its robustness and accuracy applicable to a wide range of flow at a low computational cost. In this model two transport equations are used to obtain kinetic energy K and dissipation rate ϵ . ANSYS Fluent offers three turbulence models based on this approach. The Standard K- ϵ was the first to be introduced and its formulation was based on original work presented by

Launder and Spalding (Cable, 2009) following this the RNG K- ϵ and Realizable K- ϵ were then introduced. The last two versions aimed at improving the performance of the Standard K- ϵ .

K- ϵ Model is able to handle various fluid flow conditions. As it has a simple format, strong performance, and also widely validate. In this model, various improvisations have been conducted which results in better performance also make it more capable to handle various problem with simplicity (Aharwal et al., 2018).

Standard K-Epsilon Turbulence Model

The simplest complete models of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k- ϵ model in Fluent falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding (Cable, 2009).

RNG K-Epsilon Turbulence Model

The RNG k-epsilon model was derived using a rigorous statistical technique (called renormalization group theory). It is similar in form to the standard k- ϵ model, but includes the following refinement (Snegirjov, 2009).

- The RNG model has an additional term in its ϵ equation that significantly improves the accuracy of rapidly strained flows.
- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
- The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard K- ϵ model uses user-specified, constant values.
- While the standard K- ϵ model is a high-Reynolds-number model, the RNG theory provides an analytically-derived differential formula for effective viscosity that accounts for low Reynolds-number effects. Effective use of this feature does, however, depend on the appropriate treatment of the near-wall region (Yakhot V and Orszag, 1986)

Realizable K-Epsilon Turbulence Model

The term realizable means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k- ϵ model nor the RNG k- ϵ model is realizable. The realizable k- ϵ model is a relatively recent development and differs from the standard k- ϵ model in two important ways (Shih et al., 1995).

- The realizable k- ϵ model contains a new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate, ϵ , has been derived from an exact equation for the transport of the mean-square vorticity fluctuation.

Both the realizable and RNG k- ϵ models have shown substantial improvements over the standard k- ϵ model where the flow features include strong streamline curvature, vortices, and rotation. Since the model is still relatively new, it is not clear in exactly which instances the realizable k- ϵ model consistently outperforms the RNG model. However, initial studies have shown that the realizable model provides the best performance of all the k- ϵ model versions for several validations of separated flows and flows with complex secondary flow features.

K-Omega Turbulence Model

The k- ω model was first proposed by (Wilcox, 2008) and is also a widely implemented turbulence model. Similar to the k- ϵ model, this is a two-equation model and attempts to predict turbulence by solving PDEs for the kinetic energy k and the specific turbulence dissipation rate ω . It is highly similar to the K- ϵ model but the small differences lead to a large effect on the difference between turbulence predictions. Compared to the K- ϵ model, this model gives superior results for boundary layer flows and for flows with higher pressure gradients and where separation occurs. However, its performance for free shear flows is rather poor.

Standard K-Omega Turbulence Model

The standard K- ω model in Fluent is based on the Wilcox K- ω model (Wilcox, 2008) which incorporates modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. The Wilcox model predicts free shear flow spreading rates that are in close agreement with measurements for far wakes, mixing layers, and plane, round, and radial jets,

and is thus applicable to wall-bounded flows and free shear flows. A variation of the standard $k-\omega$ model called the SST $K-\omega$ model is also available in Fluent, and as described below.

SST K-Omega Turbulence Model

This is a two-equation turbulence model developed by (Menter, 1994) with the aim to combine the accuracy of the $K-\omega$ model in near-wall condition with the behavior of the $K-\epsilon$ in the far-field. In practice, this is achieved thanks to a blending function that allows the model to work as $K-\omega$ close to the wall region and then switch to a modified $K-\epsilon$ model far from the wall region. SST stands for Shear Stress Transport which means that the transport of turbulent shear stress is taken into account when defining the turbulent viscosity.

As has been mentioned the $K-\epsilon$ approach is using wall function while the $k-\omega$ needs a refined boundary layer. This phenomenon is called the near-wall approach. The models separately have different grid refinement in order to perform efficiently. The $K-\epsilon$ model doesn't need too refined mesh only. SST $K-\omega$ model is used to predict flow near wall boundaries as it produces good results in such cases. (SST) $K-\omega$ model is equivalent to $K-\epsilon$ model in for various flow conditions and produce almost similar results (Mishra and Aharwal, 2018).

Summary of review

Analysis done on the improvement of aerodynamic drag and lift was done with different type of add-on devices using CFD/Ansys Fluent. Most of the literature cited in this work focused mainly on the improvement of aerodynamic drag of heavy vehicles which is having a blunt frontal surface and also the improvement was done by changing the vehicles body styling.

Research gap

It is found that lot of work had done to reduce lift of race cars, sedans but very little work had done for midi buses. This study aims to fill this gap by exploring the effect of rear spoiler on aerodynamic performance of the midi bus.

3. CFD ANALYSIS OF AERODYNAMIC PERFORMANCE OF THE BUS

CFD is not a science by itself but a way to apply methods of one discipline (numerical analysis) to another (heat and mass transfer). In retrospect, it is integrating not only the disciplines of fluid mechanics with mathematics but also with computer science.

3.1 Introduction

Aerodynamics analysis mainly carried in two ways such as Computational Fluid Dynamics (CFD) and Experimental analysis by wind tunnel testing. This chapter focused mainly on CFD analysis which was done using ANSYS Fluent and the methodologies employed. A scaled-down model of a Toyota HiAce Midi Bus was designed and constructed, using Computer-Aided Design (CAD) software, Solid Works was employed for this study. Once the CAD models of the bus, were designed they were imported into the Computational Fluid Dynamics (CFD) ANSYS Fluent. ANSYS Fluent was utilized to perform external fluid flow simulations, predicting the drag coefficient of each bus model. To further reduce computational time required for each simulation the CAD model was scaled down to a 1/10th scaled model. The Reynolds-Averaged Navier-Stokes (RANS) approach was used. The turbulence model was the widely used two-equation model, namely, k-epsilon realizable model with enhanced wall treatment. The steady, pressure-based solver was utilized to achieve steady-state simulations. The process that was taken, for both Solid Works and Fluent, is outlined in this section. Also, this study involves an investigation of aerodynamic characteristics and optimization of existing Toyota HiAce Midi Bus using CFD methods. Aerodynamic study on this midi bus was carried out by adding a spoiler at the rear end.

3.2. Materials

The materials used in this study were:

- Experimental scaled Toyota HiAce Midi Bus model: This is used as a model to investigate the drag force on the wind tunnel.
- Wind Tunnel: Used to validate the obtained computational results experimentally.

The software's used in this study were

- **SOLIDWORKS:** Used to construct model of Toyota Hiace both an existing baseline and improvement.
- **ANSYS Fluent:** Used to simulate the flow around a constructed vehicle models as well as analyze drag and lift coefficients.
- **Microsoft Excel:** Used to draw the ANSYS calibration corresponding to the input value.

3.3 Existing and Modified CAD Models

3.3.1 Data Collection for Existing Model

Data were gathered to analyze the shape and dimension of Toyota HiAce Midi Bus from Company's technical specification of vehicles available for sale to construct the SOLIDWORK models with actual dimensions scaled to 1:10. These models can be used in the CFD software to analyze the external shape of the vehicle for the study. Rear spoiler which is externally attached to the rear end of the roof, the spoiler will minimize the amount of swirling air in the rear end of the vehicle. The midi bus dimensions are mentioned in Table 3.1.

Table 3. 1. Basic technical specification of Toyota Midi Bus

Dimensions & Weight	2.5-litre Diesel (2KD-FTV)
Overall Length (mm)	4695
Overall Width (mm)	1695
Overall Height (mm)	1980
Wheelbase (mm)	2570
Tread Front (mm)	1470
Tread Rear (mm)	1465
Overhang Front (mm)	1050
Cargo space Width (mm)	1545

3.4 Baseline Models

3.4.1 Bench Marking the Baseline Model

The Existing Toyota HiAce midi Bus is shown in **Figure 3.1** below. The generic model is scaled down by 1/10 and the length of the model is 469.5 mm, the width of the model is 169.5 mm, and the height of the model is 198 mm.



Figure 3.1. Existing Toyota HiAce midi Bus.

From the given data in Table 3.1, the baseline model of a Toyota HiAce midi bus 1/10 scaled-down model was generated using Solid Works as shown in Figure 3.2. The model was designed with a smooth underbody, enclosed wheel wells, with no side-view mirrors and without openings for cooling airflow.

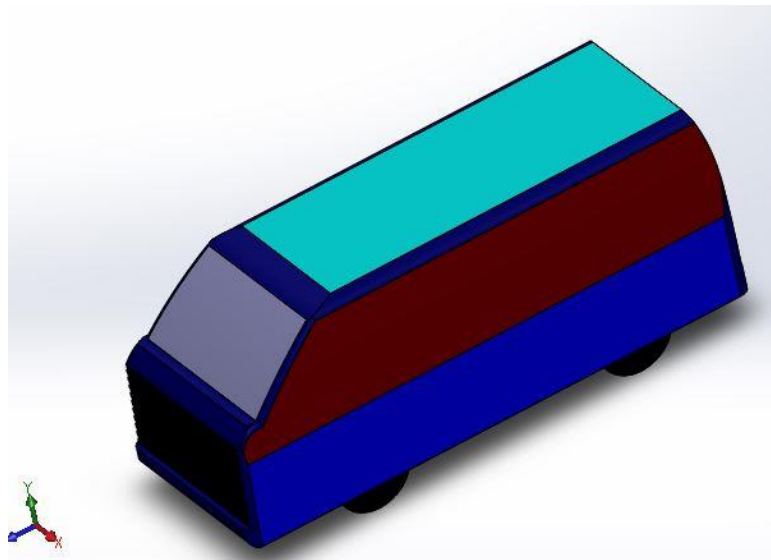


Figure 3.2. Baseline model of a Toyota HiAce midi bus 1/10 scaled down.

Then the designed model is imported into the ANSYS Fluent to do the simulation of the coefficients of drag and lift in the wind tunnel which is generated in the design module of the ANSYS Fluent. Toyota HiAce midi bus was chosen for this study, it should be noted that the Solid Works model used was not an exact replica of the bus, but it is an approximate model.

3.5 Spoiler Cad Models and Dimensions

Adding a rear spoiler could be considered as making the air “see” a longer, gentler slope from the roof to the spoiler, which helps to delay flow separation and the higher pressure in front of the spoiler can help reduce the lift on the car by creating down force. This may reduce drag in certain instances and will generally increase high-speed stability due to the reduced rear lift. Due to their association with racing, consumers often view spoilers as “sporty”.

3.5.1 Design of a rear spoiler

Considerations taken in the design of a rear spoiler is that of it has to give a minimum drag, maximum stability and control that helps the vehicle tire in contact with the ground. Airfoil shapes are the basics in the design of car spoilers since it helps the air flowing on the surface of the spoiler being attached that is flow separation before the trailing edge of the airfoil will not happen. Airfoil should be designed to have a minimum drag and for that there is a catalogue for the design of airfoil that is commonly known as NACA (National Advisory Committee for Aeronautics) airfoil.

NACA airfoils one of the most reliable resources and widely used data base is the airfoils that have been developed by NACA (predecessor of NASA) in 1930s and 1940s. For the design of the car spoiler NACA four - digit airfoil shapes are commonly used. In this series the first digit represents the maximum camber as percentage of the chord of the airfoil and the second digit represents the maximum camber position as in tenths of the airfoil chord length and the last two digit represents the maximum thickness of the airfoil as in percentage of the airfoil chord length.

In this study NACA 1412 was selected it has 12 % thickness to chord ratio and its maximum camber is 10 % of the chord length of the airfoil and the location of the maximum camber is 41 % of the airfoil chord length.

Data from the NACA airfoil catalogue imported into CAD Modeling Software Solidwork this results the airfoil that we want to be.

Table 3. 2. NACA 1412 Coordinate Value

NACA 1412 airfoil database		
x	y	z
1	0.00126	0
0.95025	0.00966	0
0.9004	0.01753	0
0.80058	0.03178	0
0.70061	0.04413	0
0.60051	0.05453	0
0.50029	0.06267	0
0.4	0.06803	0
0.29925	0.0694	0
0.24889	0.06799	0
0.19857	0.06486	0
0.14833	0.05951	0
0.09824	0.05118	0
0.0733	0.04537	0
0.04845	0.03786	0
0.02378	0.02733	0
0.01158	0.01954	0
0	0	0
0.01342	-0.0183	0
0.02622	-0.02491	0
0.05155	-0.03318	0
0.0767	-0.03857	0
0.10176	-0.04242	0
0.15167	-0.04733	0

0.20143	-0.04986	0
0.25111	-0.05081	0
0.30075	-0.05064	0
0.4	-0.04803	0
0.49971	-0.04321	0
0.59949	-0.03675	0
0.69939	-0.02913	0
0.79942	-0.02066	0
0.8996	-0.01141	0
0.94975	-0.00646	0
1	-0.00126	0

For the numerical analysis, two different spoiler styles have been used. The first spoiler was a wing style spoiler, it has a 5 cm gap behind the surface of the vehicle's roof end and the gap span length is 100mm, and the other spoiler was mounted edge of the rear side of the vehicle without leaving a gap between spoiler and the surface of vehicle. The spoiler span is equal in length with the roof width which is 140mm and the length of the spoiler was fixed at 20mm for model one and two and 10mm for model three and four. in the present study after the comparison of the two spoilers the better spoiler is again studied at the various pitch angles, namely, 5°, 10°, All dimensions are in millimeters and it's in 1/10 scale as the vehicle body is 1/10.

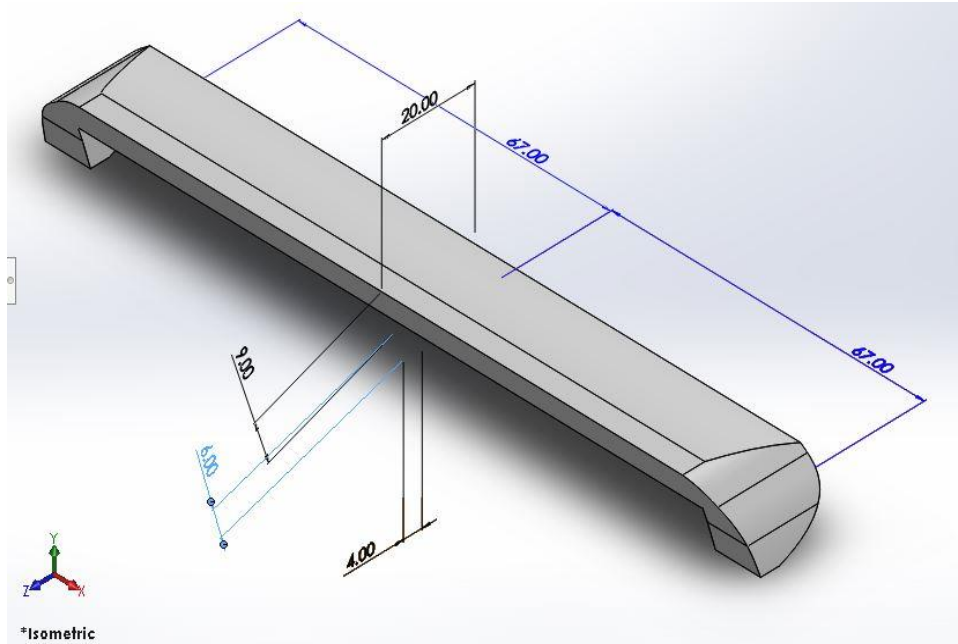


Figure 3.3. 0° pitch angle spoiler. (all dimensions are in millimeter).

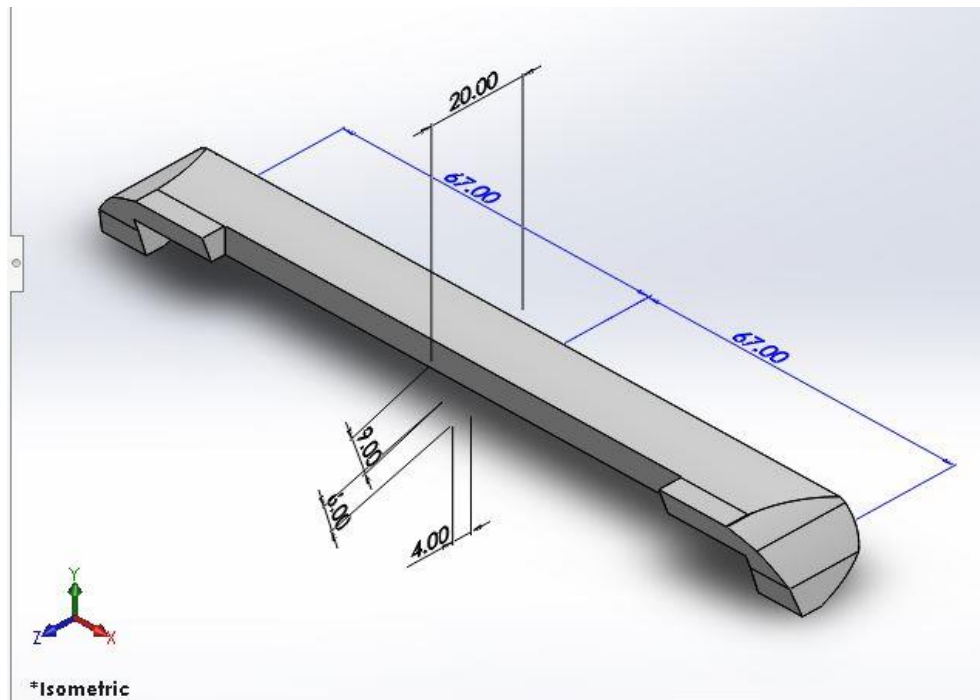


Figure 3.4. 0° pitch angle spoiler with a gap. (All dimensions are in millimeter).

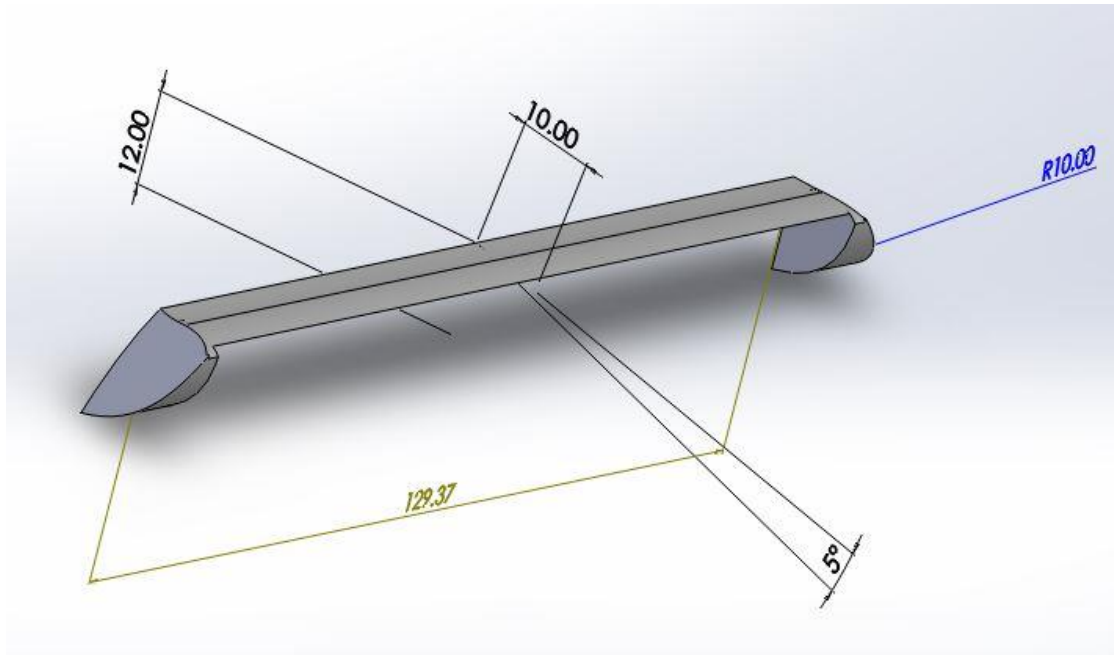


Figure 3.5. 5° pitch angle spoiler. (All dimensions are in millimeter).

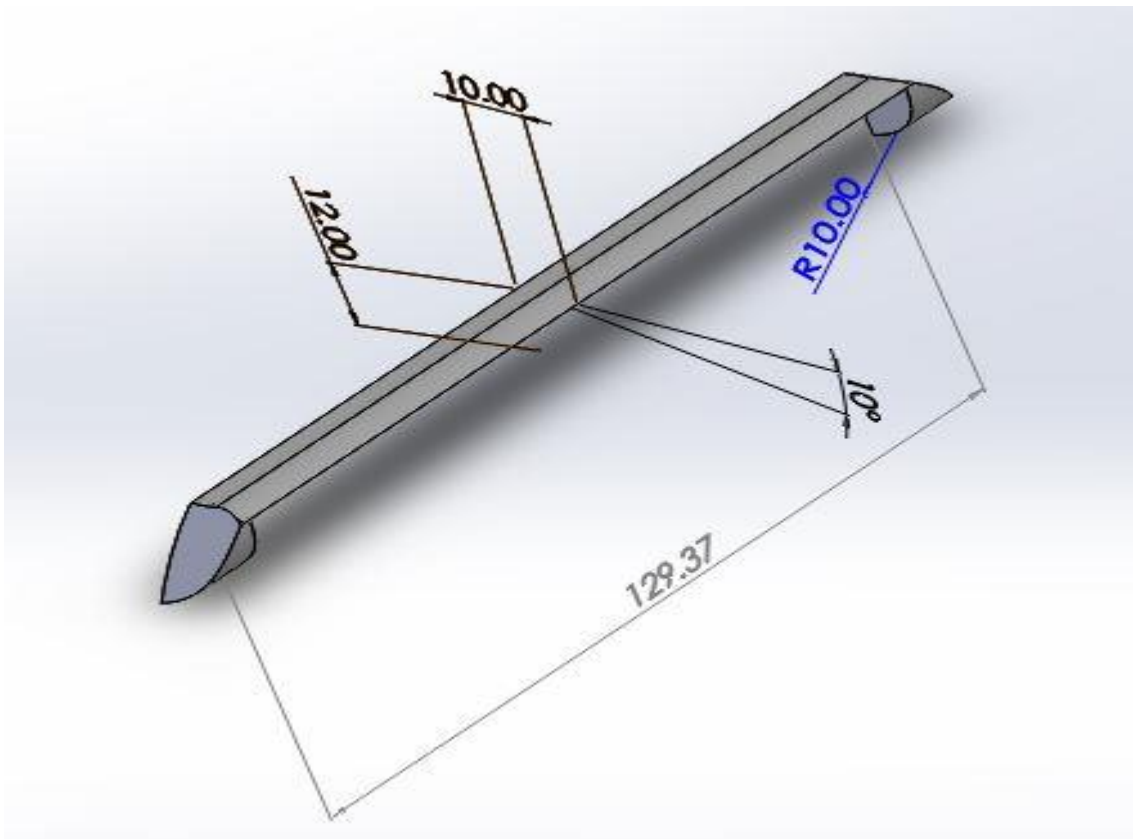


Figure 3.6. 10° pitch angle spoiler (All dimensions are in millimeter).

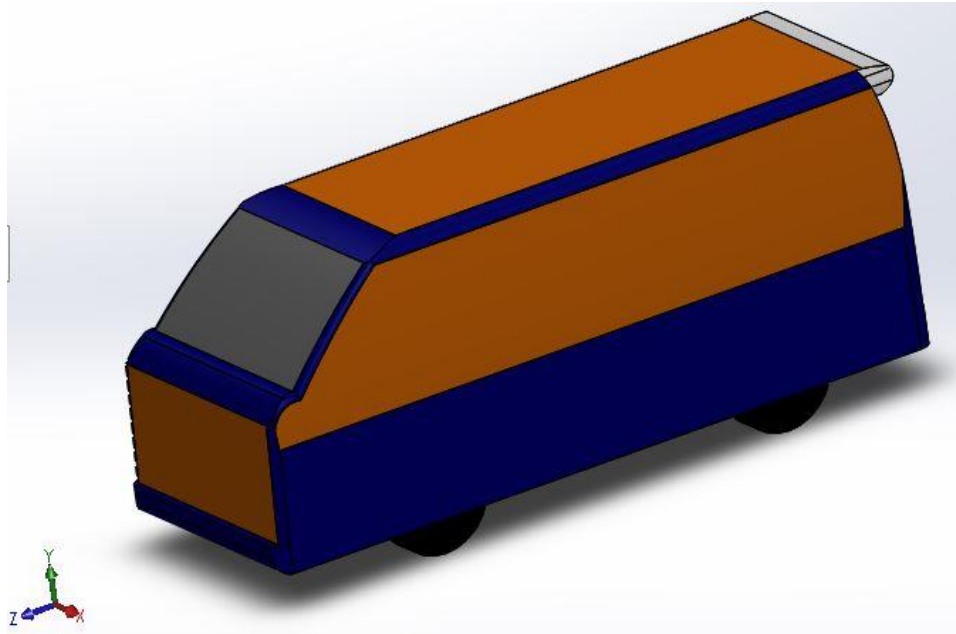


Figure 3.7. Assembly 3D CAD models of vehicle with spoiler configured at 0° pitch angle. (Model one)

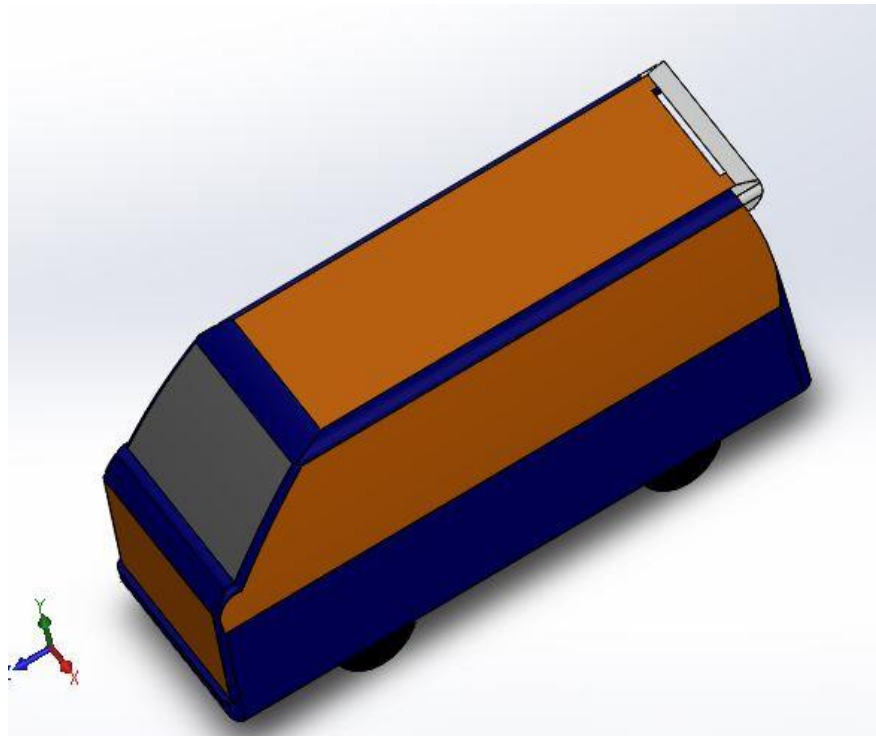


Figure 3.8. Assembly 3D CAD models of vehicle with a wing style spoiler configured at 0° pitch angle. (Model two)

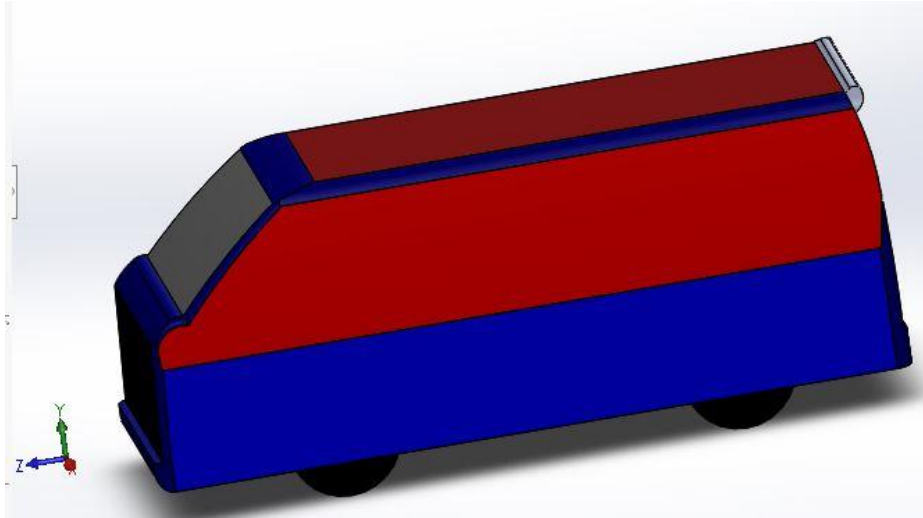


Figure 3.9. Assembly 3D CAD models of vehicle with spoiler configured at 5° pitch angles.
(Model three)

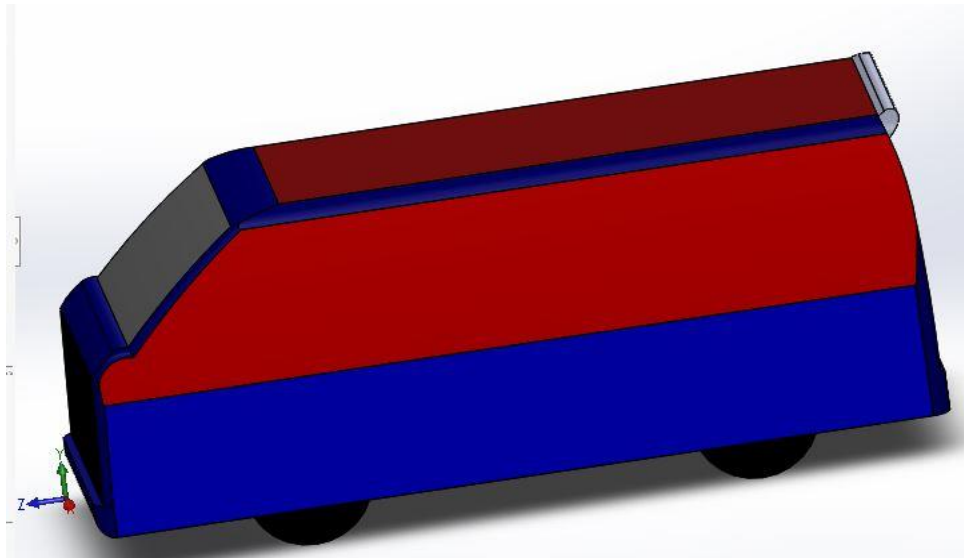
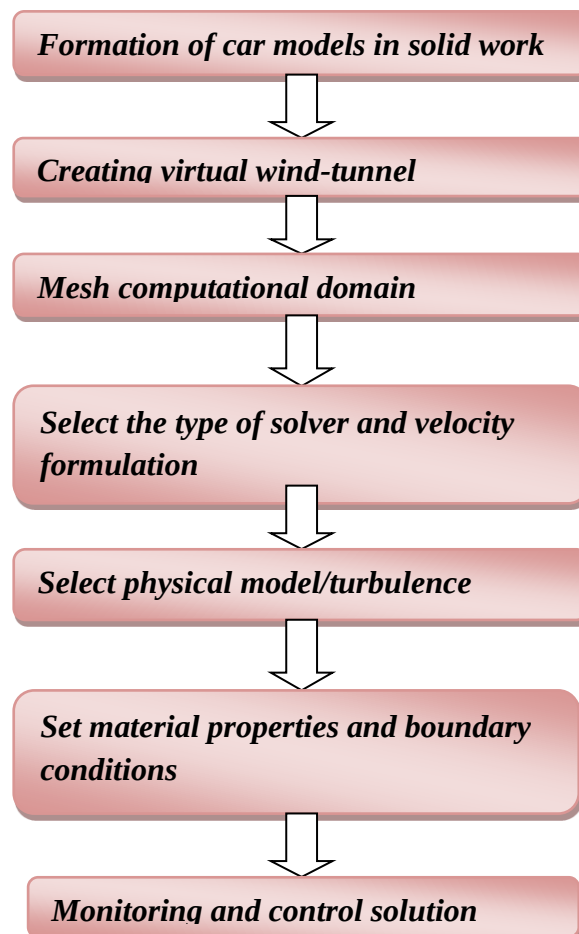


Figure 3.10. Assembly 3D CAD models of vehicle with spoiler configured at 10° pitch angles.
(Model four)

3.6. ANSYS Fluent Setup and Procedure

CFD is a method for solving complex fluid flow and heat transfer problems on a computer. CFD allows the study of problems that are too difficult to solve using classical techniques. Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamic, from aerospace propulsion to weather prediction. CFD is commonly accepted as referring to the broad topic encompassing the numerical solution, by computational methods. These governing equations, which describe fluid flow, are the set of Navier-Stokes equation, continuity equation and any additional conservation equations, for example, energy or species concentrations. The physical characteristics of the fluid motion can usually be described through fundamental mathematical equations, usually in partial differential form, which govern a process of interest and are often called governing equations in CFD.



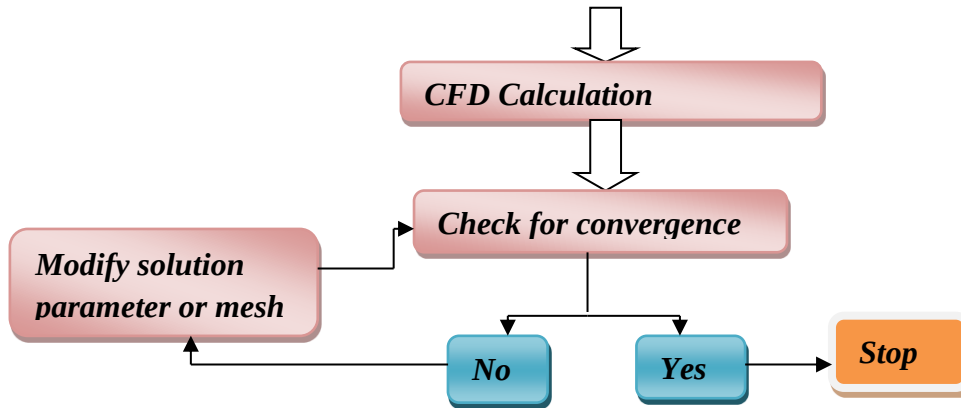


Figure 3.11. An overview of the CFD solution procedure.

All the CFD commercial packages available in the market have three basic elements, which divide the complete analysis of the numerical experiment to be performed on the specific domain or geometry. The three basic elements are:

1. Pre- processor stage
2. Solver stage
3. Post- processor stage

3.6.1 Pre-Processor Stage

The pre-processor contains all the fluid flow inputs for a flow problem. It can be seen as a user-friendly interface and a conversion of all the input into the solver in CFD program. At this stage, quite a lot of activities are carried out before the problem is being solved. In this stage, CFD users are needed to provide sufficient input to the computer in order to obtain the desired output. The pre-processing stage is divided into several steps such as Geometry generation, Boundary condition input, Mesh generation, Selecting flow type (Steady/Unsteady), Turbulence and near wall model input. The accuracy of the CFD solution is governed by the number of cells in the grid and is dependent on the fineness of the grid. The solution of the flow problem such as temperature, velocity, pressure etc. is defined at the node's inside each cell.

3.6.1.1 Geometry Generation

The simplified baseline model of a Toyota HiAce midi bus is designed in SolidWorks as shown in Figure 3.12. The sizes of the midi bus model are 469.5 mm long, 169.5 mm wide, 198 mm high and wheelbase of 257 mm. The main simplifications are eliminating small parts, wheel frames, small details and gaps in the vehicle body for the purpose of the analysis. Thereafter, this model has been analyzed for drag and lift coefficients under the ANSYS (Fluent).

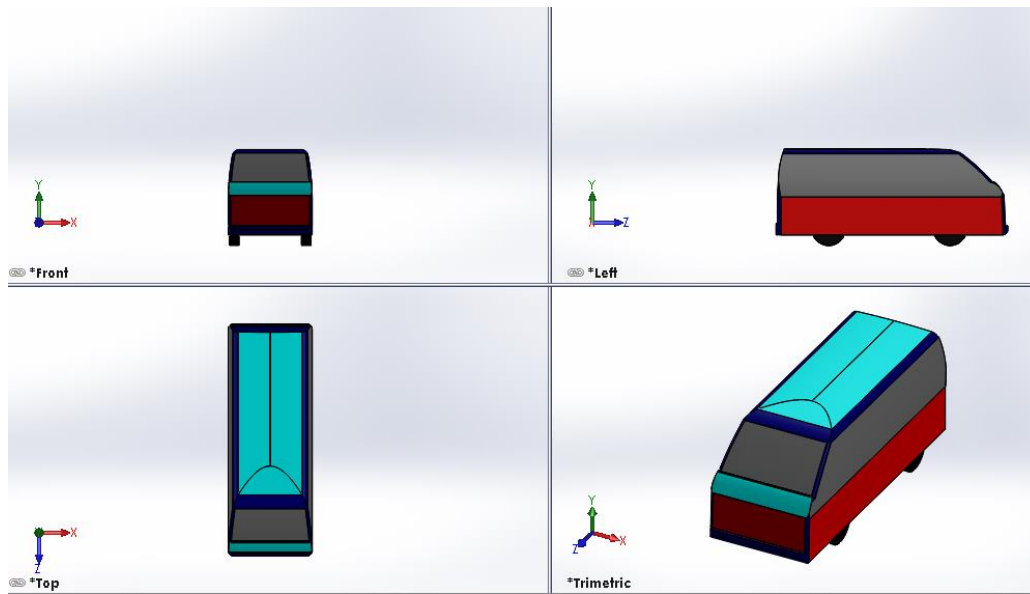


Figure 3.12. An Existing model with simplification.

3.6.1.2. Design of Virtual Wind Tunnel

A virtual air-box has been created around the 3D CAD model (Figure 3.13), which represents the wind tunnel in the real life. Since we are more interested in the rear side of vehicle, which is where the “wake of vehicle” phenomenon occurs, more space has been left in the rear side of the vehicle model to capture the flow behavior mostly behind the vehicle.

Due to the complexity of the simulation with limited computer resources and time, the complete domain was divided to half using a symmetry plane (YZ plane), which means, the simulation would be calculated for just the one side of the vehicle and since the other side is symmetric and YZ plane has been defined as symmetric boundary in the solver to make the boundary condition as “a slip wall with zero shear forces”; the simulation results would be valid for full

model as well. All 6 surfaces of the virtual wind tunnel (air-box) have been named (**Figure 3.13**) so the numerical solver of ANSYS FLUENT would recognize them and apply the appropriate boundary conditions automatically. The measurements of the tunnel are relational to the bus. A virtual wind tunnel contains an inlet, an outlet, wall, and a ground surface. The flow inlet plane was located three car lengths away from the front of the car body, whereas the outlet plane five car lengths from the rear. The side plane was located sufficiently far from the car body side surface (about two car lengths), providing a vast simulation domain.

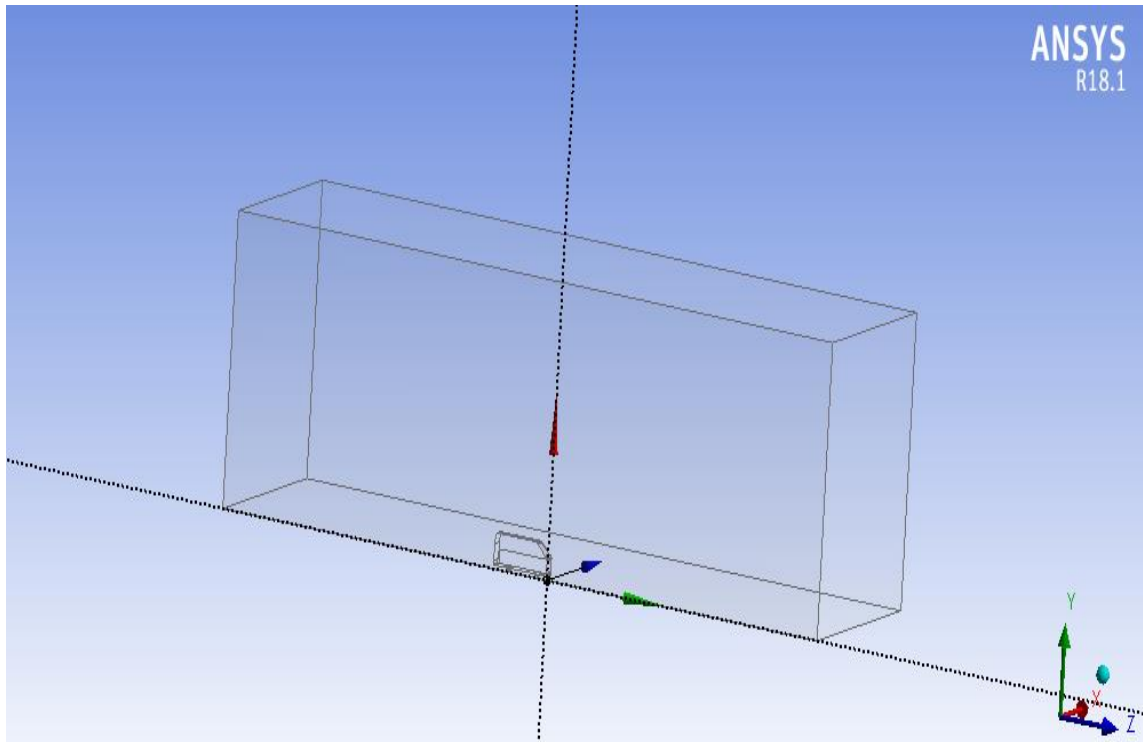


Figure 3.13. Virtual wind tunnel targeted for CFD simulation.

The height of the domain was set to two times the length of the car body Dimensions used for the enclosure. The ground was located at the distance 0.0001 m from the car body bottom surface. The assumptions made in the present simulation have the airflow steady state with constant velocity at inlet and with zero-degree yaw angle, constant pressure outlet, no-slip wall boundary conditions at the vehicle surfaces, and in viscous flow wall boundary condition on the top, sidewalls, and ground face of the virtual wind tunnel.

Table 3.3. Dimension for wind tunnel enclosure

Upstream Length(mm)	Downstream Length(mm)	Width(mm)	Height(mm)
2347.5	2347.5	1408.6	1408.6

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

3.6.1.3 Mesh Generation

The triangular shape surface mesh was used due to its proximity to changing curves and bends. These elements easily adjust to the complex bodies used in automobile and aerospace bodies. With the default settings for mesh generation, ANSYS Meshing has generated the meshes. With the global mesh sizing settings, ANSYS Meshing recognized that there were some curvatures around the vehicle body. But the meshing was very coarse and it was only the initial guess by the software. In order to capture more accurate data through solver we needed to improve the mesh. The first thing to do was changing the mesh sizing parameters. All meshing sizing parameters that have been altered are given in the appendix.

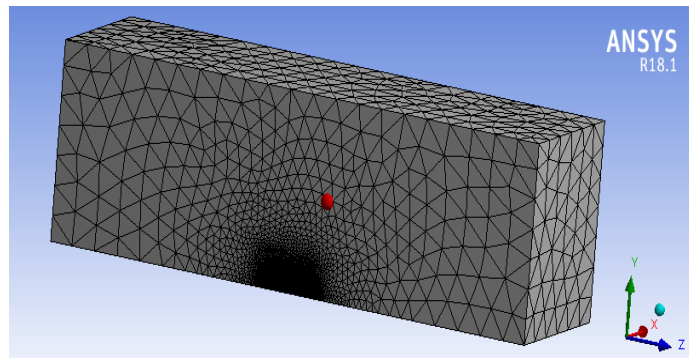


Figure 3.14. Mesh generation with modified sizing settings. (Baseline model).

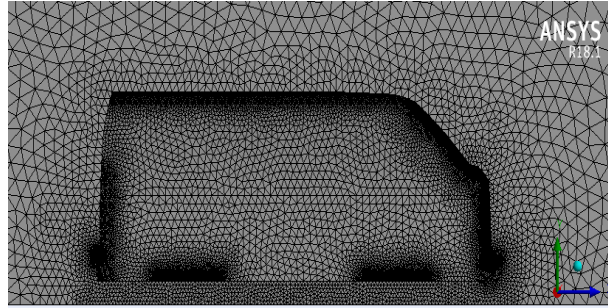


Figure 3.15. Zoom of the mesh after surface sizing (Baseline model).

The new mesh looked more decent but still it was lack of inflation layer around the vehicle body. The inflation layer has been enabled and “Automatic inflation: Program Controlled” has been set to capture the boundary effects of the flow around the body more accurately. The vehicle body itself and the road have been included to “program-controlled inflation” while the other named selections (velocity-inlet, pressure-outlet, symmetry etc.) are excluded. The inflation option has been set as “First Aspect Ratio” instead of “Smooth Transition”.

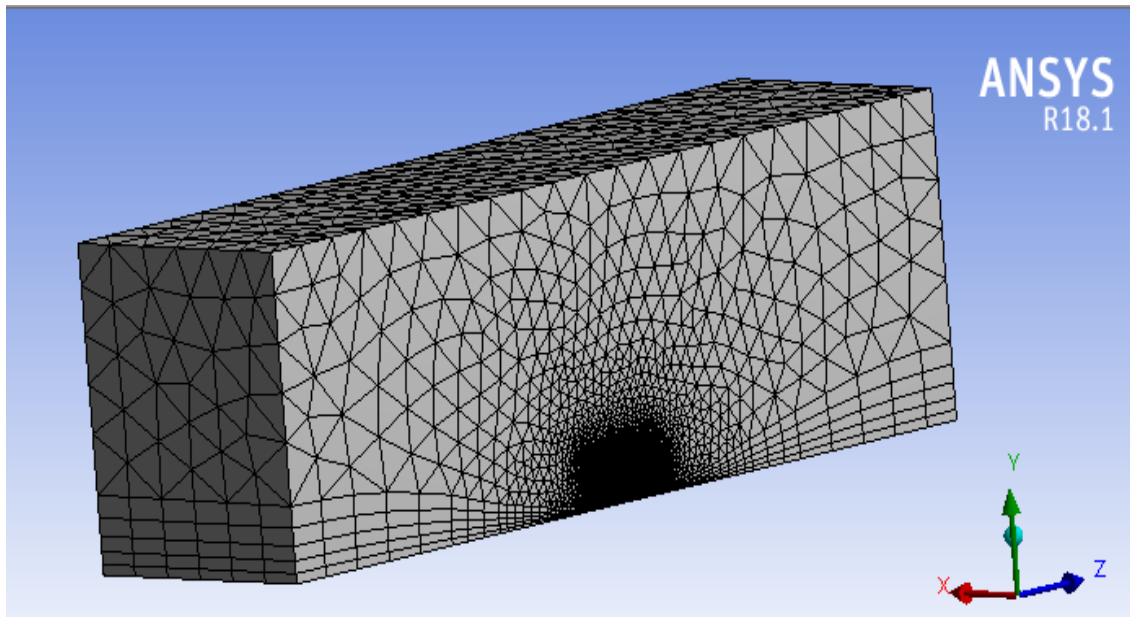


Figure 3.16. Mesh after inflation of the baseline model.

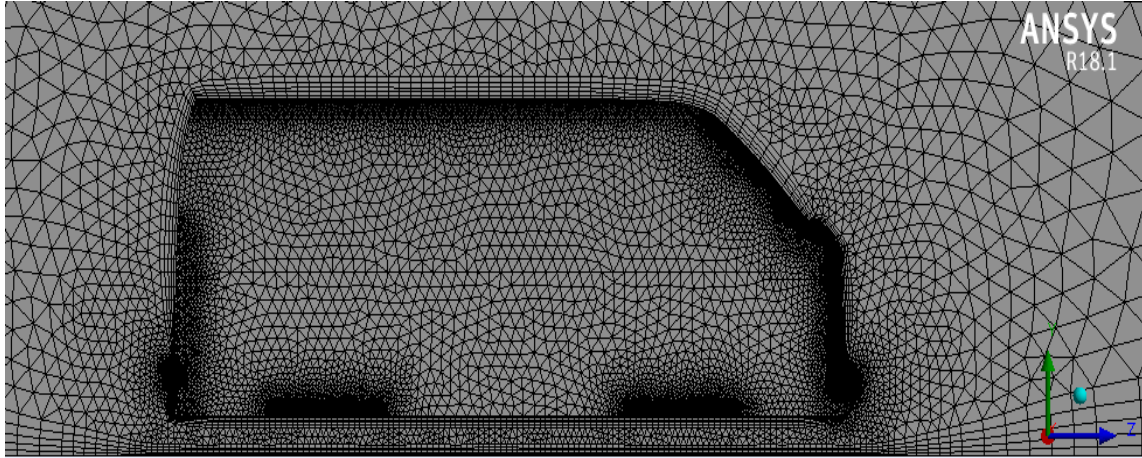


Figure 3.17. Zoom of the mesh after inflation of the baseline model.

Figure 3.17 shows the new mesh after the automatic inflation has been enabled as “program controlled” and the inflation option has been selected as “first aspect ratio”. Another recommendation from the guideline by Fluent Germany is to create a new volume control box around the body where the elements can be limited to a certain size. The advantage of using a control volume and limiting the mesh sizing within the control volume only, lets us to improve the meshing quality only within in the area where we need high resolution mesh; instead of having fine mesh in the entire domain which would eliminate the time consumption to run the simulation and leads us to get decent results quickly. There are several ways doing this, one of them is to create a virtual box around the vehicle that It refines the control volume on the surroundings of all the car surfaces in $0.5L$ on the front of the car, $1L$ on the rear of the car and $0.5L$ on the top of the car, where L is the car length. The orientation of this virtual box (which was called as “car-box”) can be seen in Figure 3.18.

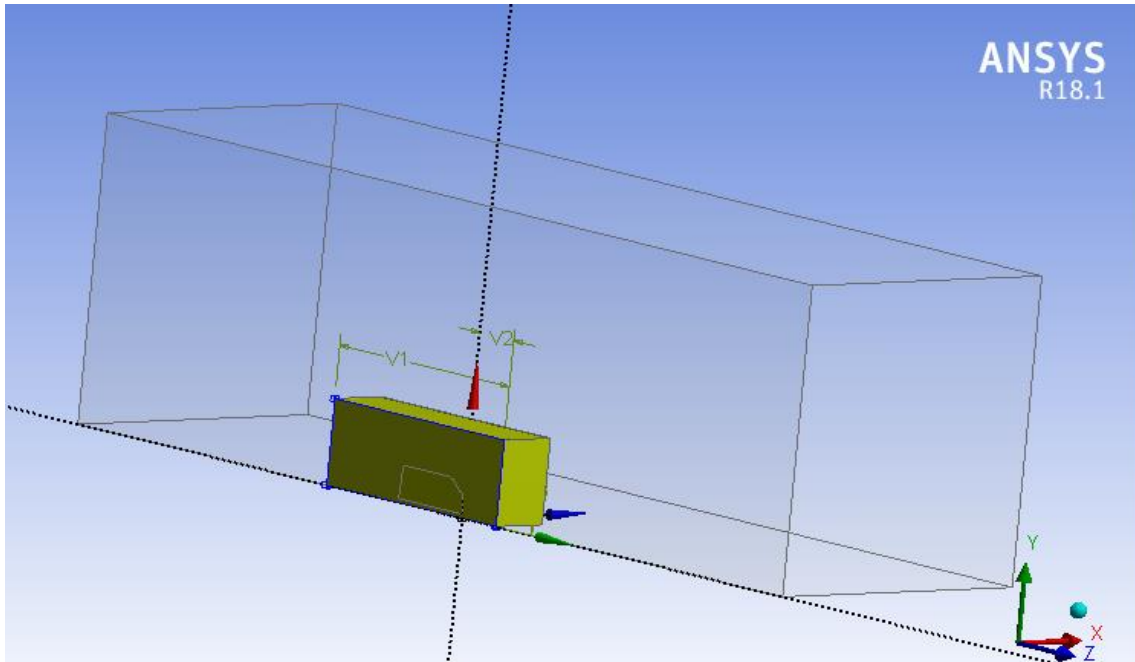


Figure 3.18. Virtual car-box orientation.

The virtual car-box was orientated as seen in Figure 3.18 and it can be noticeable in the figure that there is more space at the backside than the front side of the vehicle. It is because we are more interested in behavior of the airflow at the backside of vehicle after it passes the vehicle. Once the virtual car-box was generated and its mesh sizing was limited to 15mm with the “body sizing” functionality in the software, the new mesh became very detailed and ready to run in the solver. The final meshing can be seen in Figure 3.19.

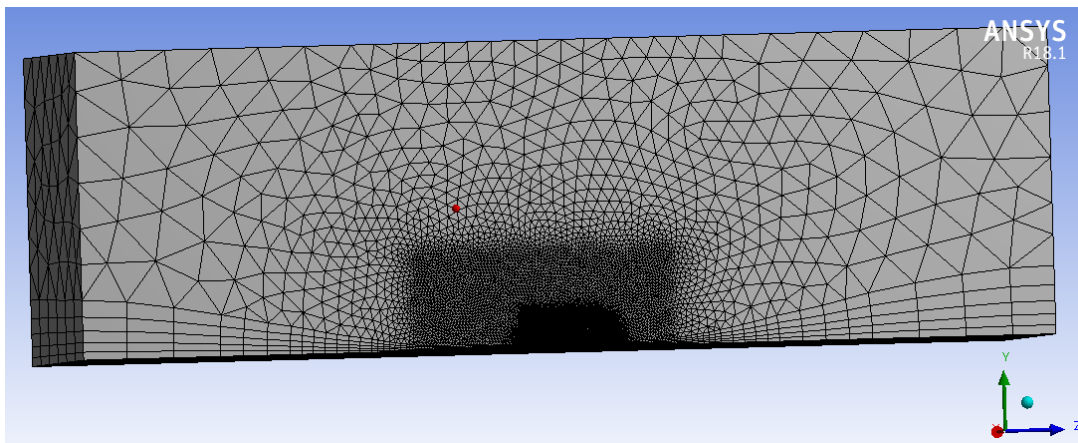


Figure 3.19. The final mesh.

Grid independence test

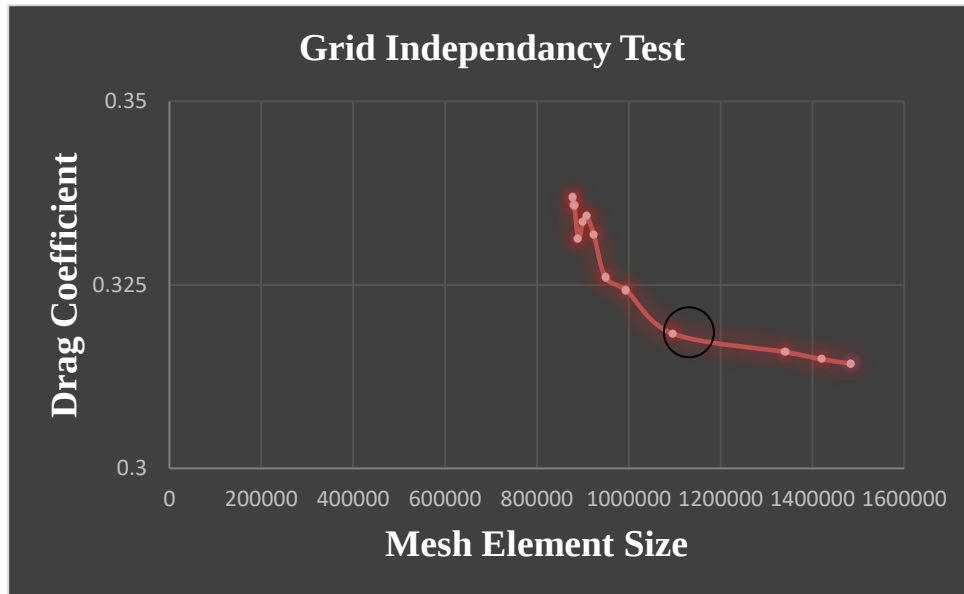


Figure 3.2. Grid independence test for the baseline model

Grid independence test graph which shows a constant line will not give any indication about the better mesh element size, so a comparison was done with another output parameters. The drag coefficient variation is significant with respect to the mesh. Result variation are not significant after the marked cell zone; hence this can be considered as the better mesh for calculation. This mesh size will give better result with less time as compared to finer mesh.

Mesh element size higher than 1,200,000 can be used as a substitute for finer mesh, thus the selected element size for this study is 15mm which is a mesh element size of 1,340,726 and it's acceptable as per the grid independence test. Models mesh element size higher than 1,200,000 can be acceptable as far as this work.

Boundary condition

The distribution of all flow variables needs to be specified. Inlet boundary conditions are mainly for flow velocity and this type of boundary condition is common and specified mostly where inlet velocities as known. In outlet boundary condition the flow tends to fully develop state where no change occurs in the flow direction when the outlet is selected far away from the

geometrical. Distributions in such region an outlet could be outlined in gradient all variable equated to zero in the flow direction except pressure. Wall boundary condition the most common boundary that comes upon in the confined fluid flow problems is the wall. This is also commonly known as no slip boundary condition. This is appropriate for the velocity condition of the wall the normal component set to zero and the tangential component set to the velocity of the wall.

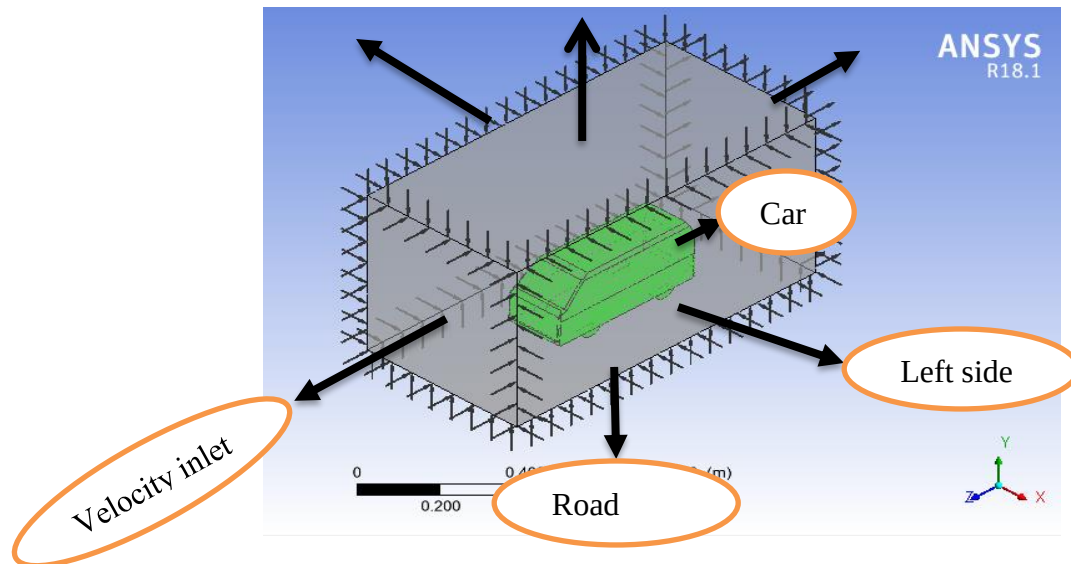


Figure 3.2.1. Boundary conditions over view.

Table 3.4. Boundary conditions

Boundary conditions		
Inlet boundary condition	Speed of car (22.22,27.87,33.33) m/s	
Outlet boundary conditions	Standard pressure(1atm=101.325kpa)	
wall boundary condition	Top wall	Speed of car (22.22,27.87,33.33) m/s
	Left side wall	Speed of car (22.22,27.87,33.33) m/s

	Right side wall	Speed of car (22.22,27.87,33.33) m/s
	Road	Speed of car (22.22,27.87,33.33) m/s
Car boundary condition	Stationary car (zero velocity)	

3.6.2 Solver Stage

After the mesh has been imported in Ansys Fluent, a check has been performed to be sure that there are no negative volumes on the mesh then the solver settings to be completed before starting the simulations. The solver setting includes the type of solver (3D or 2D), the viscous model, boundary conditions and solution controls. The inlet of the wind tunnel was indicated by the term “velocity inlet”, while the outlet of the wind tunnel was termed as “pressure-outlet”. The realizable k- ϵ model was employed in the fluent simulations for all models. The simulations were run with a velocity of 80 kmph, 100 kmph, and 120 kmph in the simulation the airflow from the velocity Inlet towards the pressure outlet with a selected amount of velocity. 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 airflow and the velocity introduced to the simulation had very low fluctuations and therefore very little turbulence (assuming that the flow is quite laminar at the inlet of the control volume). The area of each specific model was found using the fluent project area feature and was utilized in determining the drag coefficient of the vehicle model from the force that was calculated during the simulation.

The bus body is considered stationary for study. Constant velocity inlet condition is applied at the inlet and at the outlet zero-gauge pressure is applied with operating pressure as atmospheric pressure. The flow is considered to be incompressible and steady in nature and the equations are solved using both first and second-order upwind method. A coupled scheme was chosen for the pressure-velocity coupling with Least squares cell-based gradient. It offers more robust and efficient single-phase solutions for Steady flows, which brings the benefit to achieve convergence running less iteration if compared with the other pressure-based schemes such as

Simple, Simplec or PISO. While employing a coupled scheme the flow courant number was set to 50 while both the explicit relaxation factors for momentum and pressure were 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 0.8 during first upwind and changed to 0.95 for second upwind. In order to monitor the calculations and simulations a monitor was created for drag, lift and scaled residuals on the vehicle. Once the monitor was created it could be used to set the convergence criterion for the simulation. The convergence was set to none.

The numerical computation was first started with the first-order scheme to ensure numerical stability. In the first stage, first-order equations were used for the first 100 iterations so as to prevent the solution from diverging. Once sufficient convergence was achieved, the equation order was raised to second order for the rest of the iteration (1000 iterations), so as to get a negligible and accurate drag coefficient. Most of the scaled residuals and the global aerodynamic quantities such as the drag and the lift coefficients converged to 10^{-5} long way before the 1000 iteration. Once this convergence was met the simulation was complete and stopped. The convergence criteria for the monitors are 10^{-5} . It means that the solution will be considered converged when the change on the values on the residual of continuity between two consecutive iterations is equal or smaller than 0.00001. These facts are considered for providing good accuracy and convergence.

3.6.3 Post-Processing

The CFD package provides the data visualization tools to visualize the results of the flow problem. This includes – vectors plots, domain geometry and grid display, line and shaded counter plots, particle tracking etc. Recent facilities are aided with animation for dynamic result display and they also have data export facilities for further manipulation external to the code.

Determining the convergence, whether the solution is consistent and stable for all range of flow variables, is important.

Convergence is a property of a numerical method to produce a solution that approaches the exact solution by which the grid spacing and control volume size are reduced to a specific value or to zero value.

Consistency is to produce the system of algebraic equations that can be equivalent to the original governing equation.

Stability associates with the damping of errors as a numerical method proceeds. If a technique chosen is not stable, even the round-off error in the initial data can lead to wild oscillations or divergence.

3.7 Convergence

When running a well-posed CFD simulation, there are many factors that govern the decision to declare the solution done. Residual reports, force monitors, and overall balances are just a few of the factors that can be used to make this judgment call. More important than the measuring devices is what is to be achieved by running the simulation in the first place. For example, deep convergence is required if it is an aerodynamic problem and want to get accurate predictions of lift and drag coefficients. Rough convergence is acceptable for simple flow problems.

The reason that solution convergence is an issue with all CFD software is due to the iterative nature of the solution procedures used. In particular, iteration is necessary to handle the non-linearity of the equations that govern fluid flow, heat transfer, and related processes. For any given conservation equation, an approximate solution is obtained at each iteration that results in a small imbalance in the conservation statement. During the course of the iterative solution algorithm, the imbalance in each cell is a small, non-zero value that, under normal circumstances, decreases as the solution progresses. This imbalance is called the residual. The total residual for each variable across the entire solution domain is the sum of the absolute values of the individual cell residuals. This total residual is often scaled so that the residuals of different variables can be compared or combined. Scaling factors are taken from the bulk flow quantities or from the error at the start of the calculation.

A truly converged solution is one that is no longer changing with successive iteration. If the residuals for all problem variables fall below the convergence criteria but are still in decline,

the solution is still changing, to a greater or lesser degree. A better indicator occurs when the residuals flatten in a traditional residual plot (of residual value vs. Iteration). This point, sometimes referred to as convergence at the level of machine accuracy, takes time to reach, however, and may be beyond the needs. For this reason, alternative tools such as reports of forces, heat balances, or mass balances can be used instead.

4. EXPERIMENTAL VALIDATION OF NUMERICAL ANALYSIS

4.1 INTRODUCTION

The numerical analysis by Computational Fluid Dynamics (CFD) performed using ANSYS Fluent were discussed on chapter three. The validation of the numerical results is mostly done by comparison with results obtained through direct measurements from experiment. The most reliable information about physical phenomena is usually given by measurements but in certain situations, an experimental investigation which involve full scale model or equipment are either very expensive or difficult to perform or not possible at all. An alternative solution is to perform the experiment on a small-scale model and extrapolate the results obtained to the full scale.

This chapter focused mainly on the experimental validation using wind tunnel to validate the numerical analysis of the baseline model. Experimentation is particularly important in applications such as fluid mechanics, where the behavior of the system is difficult to model analytically. Wind-tunnel testing is thus the preferred method for measuring the aerodynamic effects of fluid flowing past a body. The key flow characteristics such as lift and drag can then be deduced from these experiments. The nature of fluid flow allows engineers to create scale models of the body in order to assess the behavior of fluid flow around it.

4.2 3D Model Preparation

As the wind tunnel testing is essential for the aerodynamic development. The 3D model of a Toyota HiAce Midi Bus was printed with a 50% scale of the baseline model (comparing to the model that was simulated in ANSYS Fluent). The printer is shown in Figure 4.1.

This printer used plastic material of Polyacetide (PLA), which is supplied to the printer in a form of filament (1.75 mm diameter) and then melted in a hot-end at high temperature. The printer head distributes the material layer by layer with a maximum layer thickness of 0.48mm, basing on the input code.



Figure 4. 1. D3MAND 3D Printer.

The positioning of model on the printer table determines the amount of supports, which need to be manually removed once the printing process is finished. This process demands a careful approach, as it is relatively easy to damage the printed object itself. Afterwards, the printing surface needs to be smoothened and all irregularities must be removed this can be done by sandpaper.



Figure 4. 2. The prototype of Toyota HiAce midi bus for wind tunnel test.

4.3 Experimental Approach

In the experimental approach, first a plastic model is prepared using 3D printing machine maintaining the dimensional constraints and the shape of the baseline model concept car. Holes are pierced in the plastic baseline car model to create pressure taps. Pressure data is collected by inserting plastic tubes in the holes and using multi tube manometer. Among the pressure tubes, one remains open to the air which gives atmospheric pressure. The pressure head is obtained from the difference between actual head to the atmospheric head. The experimental investigations were performed on an open circuit suction type subsonic wind tunnel.

The HM 170 Educational wind tunnel (Figure 4.3), is a subsonic, open wind tunnel with a square measurement section profile was developed for experimentation of Toyota HiAce midi bus baseline model. This wind tunnel was used to determine the lift and drag forces on the baseline model.

4.4 Specification of Conventional Wind Tunnel

Table 4. 1. Wind tunnel specifications

Type	Open type wind tunnel
Test Section	335 × 315× 295mm
Speed Controller	For variable seed
Number of Tubes in Manometer	16



Figure 4. 3. Adama Science and Technology University Educational Wind Tunnel.

The air is drawn in from the atmosphere via the streamlined funnel (1). Any transverse flow components are filtered out in the flow straightener (2). It is made of a tubular honeycomb structure. The air exits from the flow straightener as a parallel flow and is accelerated to roughly 3.3 times its original velocity in the jet (3). The static pressure (4) is measured downstream of the jet at the entrance to the measurement section. Based on the assumption of virtually loss free flow, the flow velocity can be determined from the pressure difference with respect to ambient pressure (total pressure at zero velocity). The air then flows through the measurement section (5) which has a constant cross-section as shown in Figure 4.4. After the measurement section the flow is decelerated in a diffuser (6) and part of the pressure drop required to accelerate the air in the jet is recovered. The diffuser angle is designed in a way to prevent flow separation. An axial fan (7) with downstream guide (8) draws the air out of the diffuser and conveys it into the open. A guard (9) in front of the fan prevents damage caused by foreign matter or models, whereas the guard (10) behind the fan prevents contact with the fan when it is in operation.

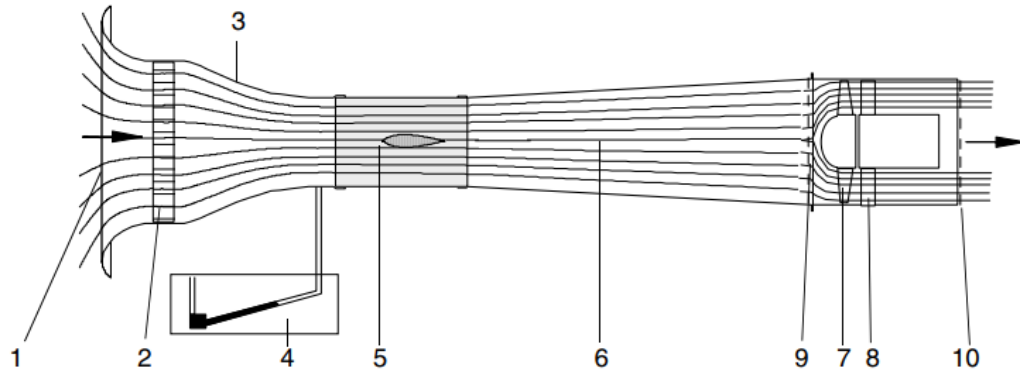


Figure 4. 4. Schematic drawing of HM 170 Educational wind tunnel (Garatebau and Barsbuttes, 2010).

4.5 Testing procedure

The steps that were followed in order to conduct the wind tunnel tests include securing the baseline model of bus in the wind tunnel (Figure 4.5), zeroing the measuring equipment, starting the test at a low speed and working up to faster speeds, taking measurements at each speed increment and repeating the test three times to establish more accurate results. Securing the models was done with the use of model holder. The model holder was attached to the flat rod, which is attached to the equipment that measures the lift and drag forces acting on the model.

In order to zero the measuring instrumentation two radial dials were adjusted until the values of the lift and drag displays were as close as possible to zero. The dials are very sensitive and it was therefore quite hard to have the readout on the displays show exactly zero. Before starting the wind tunnel's motor, the enclosure lid was closed to avoid injury to the operator or damage to the enclosure. Once the motor had been started the wind tunnel's speed was adjusted to a reasonably slow speed. For each test the wind tunnel was adjusted to four speeds and the lift and drag force readings were written down. The test was conducted three times with a baseline model to obtain a reasonable amount of data for analysis allowing for an averaged value to be obtained. Obtaining an average value ensures a good result as it accounts for variables that affect each of the runs.

The prototype was tested at different speeds with values of 12 to 28 m/s in an increment of 5 m/s at zero-degree angle of attack. The values of the normal and axial forces were obtained.

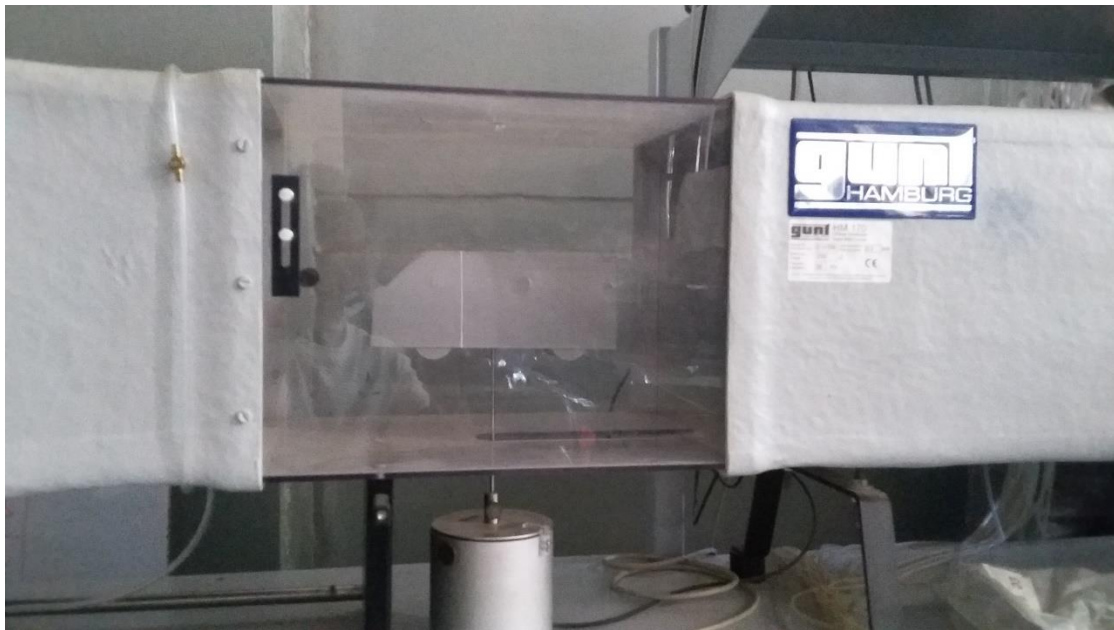


Fig.4.5. plastic baseline model inside conventional wind tunnel.



Fig.4.6. plastic baseline model being tested at different speed.

5. RESULTS AND DISCUSSIONS

This chapter presented the results of both the Computational Fluid Dynamics (CFD) analysis and experimental validation then discussed these results. The CFD analysis was performed using ANSYS FLUENT and the Experimental validation was performed using Wind-tunnel to validate the Computational analysis. The results and discussions of Computational analysis presented in section 5.1, then the results and discussions of experimental validation presented in section 5.3.

5.1 Results and Discussions of CFD Analysis

A 3D steady state, incompressible solution of the Navier-Stokes equations was performed using ANSYS FLUENT. Turbulence modeling was done with the realizable $k-\varepsilon$ model using non-equilibrium wall functions. The computational results for the following cases are presented and discussed: All the results for different cases were obtained with the same meshing resolution, the same $k - \varepsilon$ turbulence model, and also the same boundary conditions. For the first 100 iterations, a first order upwind discretization was used to accelerate the convergence then after 100 iterations second order upwind scheme has been applied and iterations have continued until it reached to the convergence criteria. The convergence criteria were having all residuals below 0.001.

5.1.1 Baseline Model Results

To determine the reduction in the drag coefficient from different models of the bus with modifications, the drag coefficient and flow field of a baseline model examined first. These simulations give the base drag coefficient for the study. After simulating the baseline model, the modified models were simulated next. After the simulation, all models were compared to the flow field, drag and lift coefficient of the baseline model. To determine the percent reduction of the drag coefficient due to the outer body modifications were compared.

Analyzing the residual trends is one of the most fundamental measures in determining whether a solution is converged or not. It directly quantifies the error in the solution of the system of equations. The residuals and the values for the coefficients of lift and drag were monitored during all numerical analyses. The convergence criteria were satisfied when all the values of

monitored residuals dropped below 10^{-5} , which is two orders of magnitude lower than standard convergence criteria (10^{-3}), and the computed values of lift and drag coefficients stabilized over subsequent iterations.

Result of scaled residuals, drag and lift coefficient, pressure and velocity contour at speed of 80kmph, 100kmph and 120kmph of baseline model are presented.

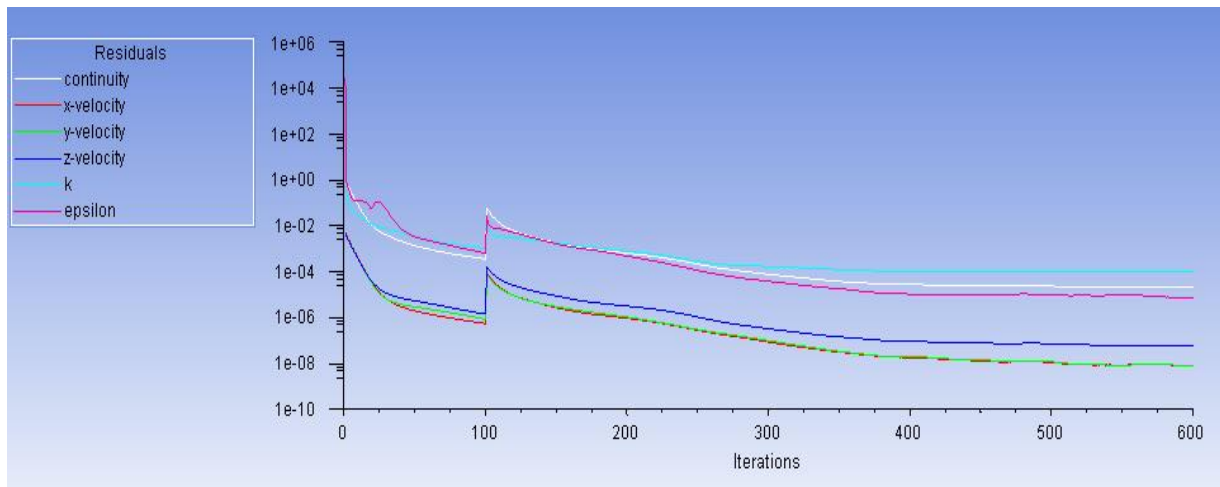


Figure 5.1. Scaled Residuals of the Baseline Model.

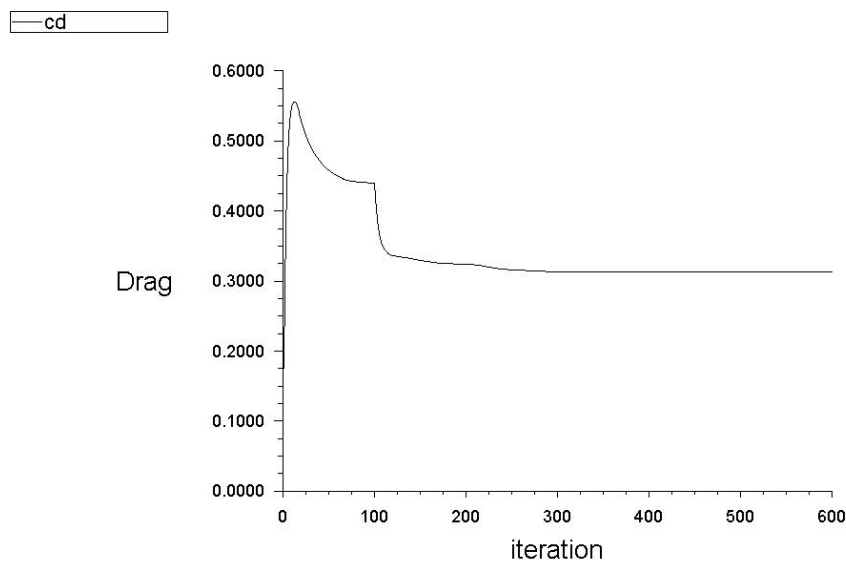


Figure 5.2. Drag coefficient of baseline model at 80 kmph.

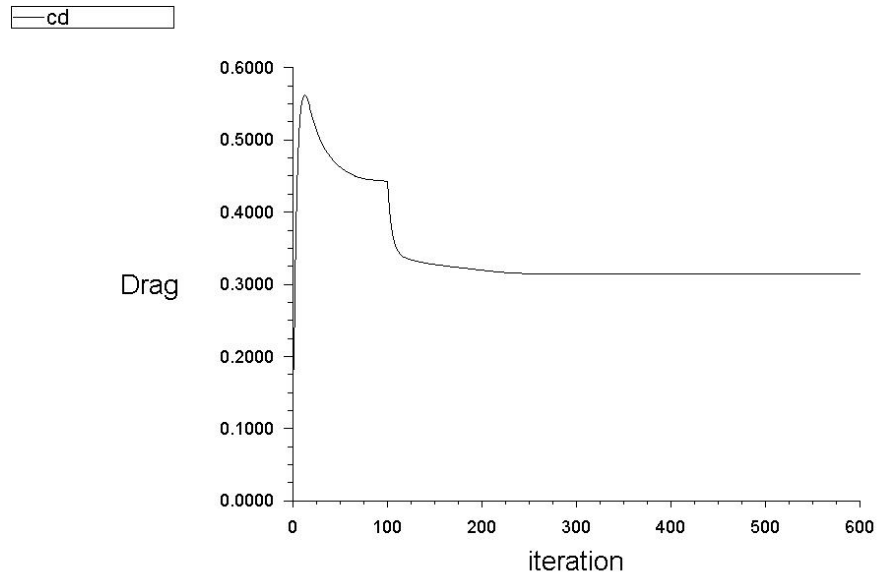


Figure 5.3. Drag coefficient of baseline model at 100 kmph.

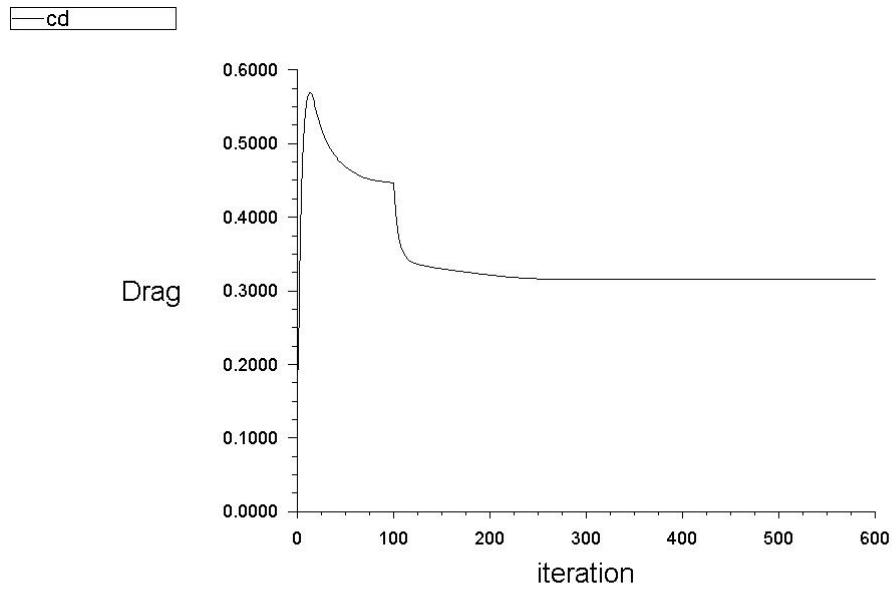


Figure 5.4. Drag coefficient of baseline model at 120 kmph.

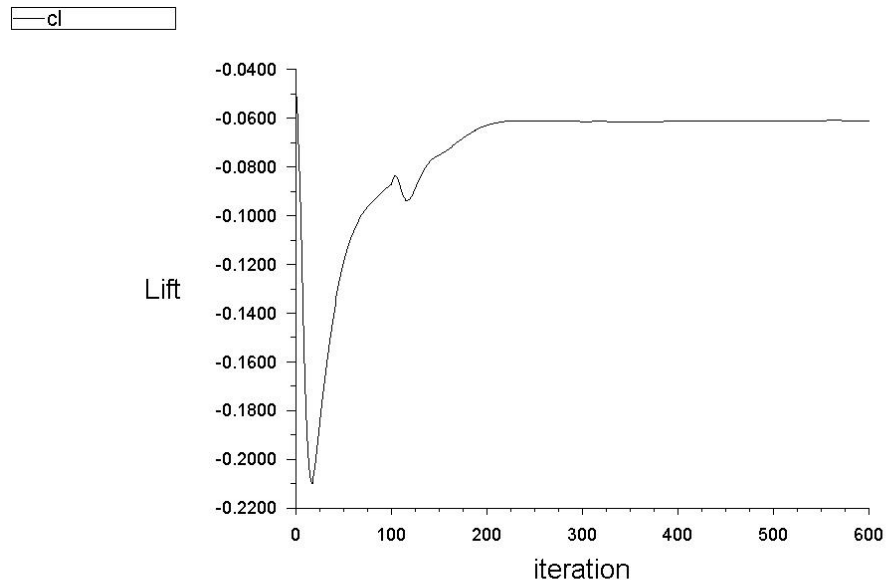


Figure 5.5. Lift coefficient of baseline model at 80 kmph.

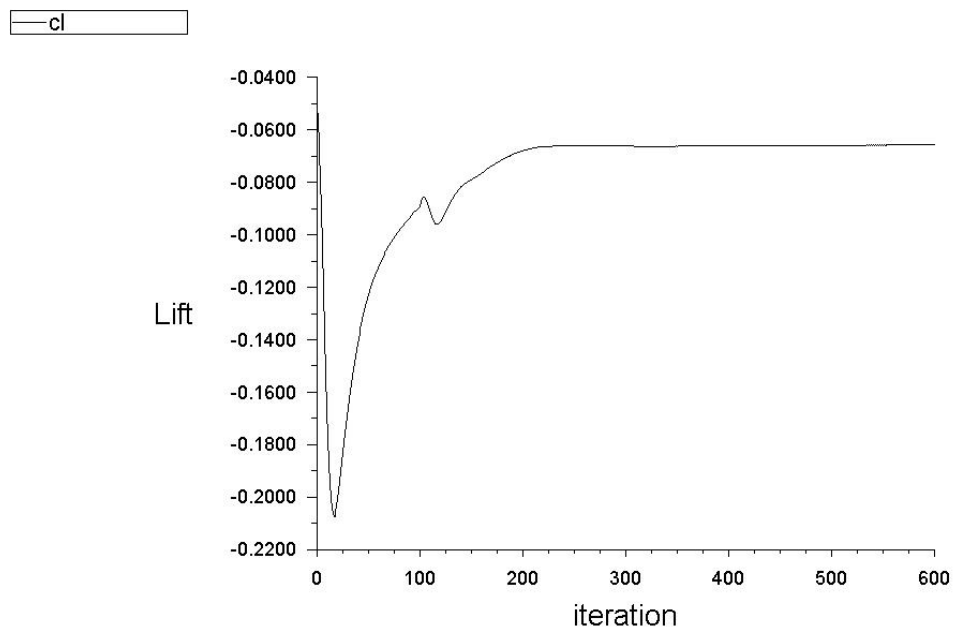


Figure 5.6. Lift coefficient of baseline model at 100 kmph.

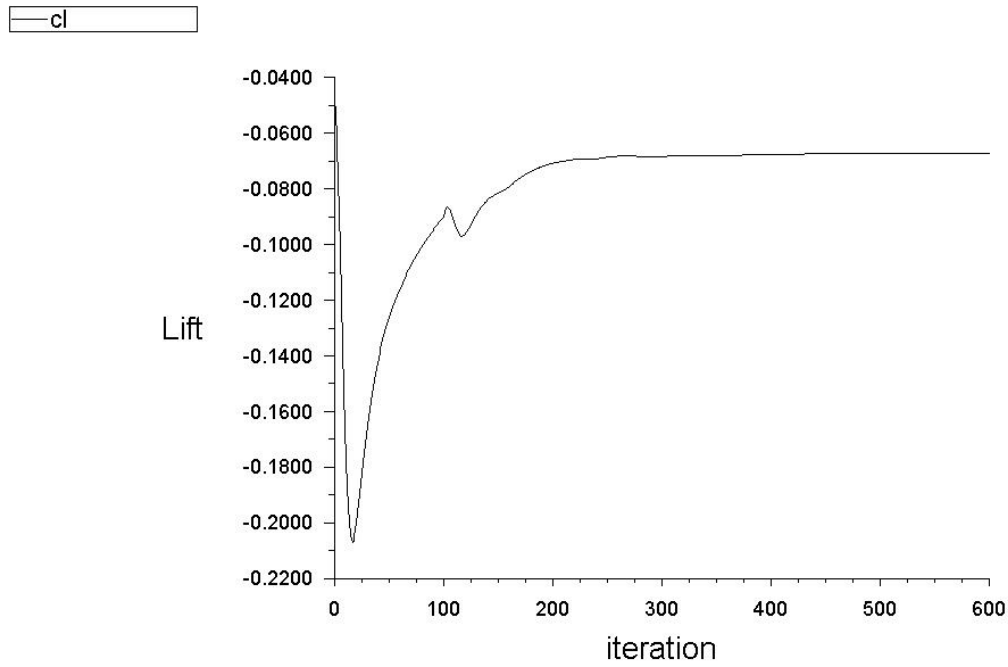


Figure 5.7. Lift coefficient of baseline model at 120 kmph.

The drag and lift coefficients are summarized below in Table 5.1.

Table 5. 1. Summary of drag and lift coefficients of the baseline model at different speeds

Model	Speed (Kmph)	C_d	C_l
Baseline Model	80	0.31238	-0.0609280
	100	0.31315	-0.0631487
	120	0.31585	-0.0670920

As shown in Table 5.1 the drag and lift coefficients were found to be 0.31238, 0.31315, 0.31585 and, -0.0609280, -0.0631487, -0.0670920 at speeds of 80, 100, and 120 kmph, respectively. The drag coefficient of a baseline model increases with increasing speed. However, the lift coefficient of a Baseline model decreases with the increasing speed which results high downward force, which is very necessary for vehicle stability at higher speed.

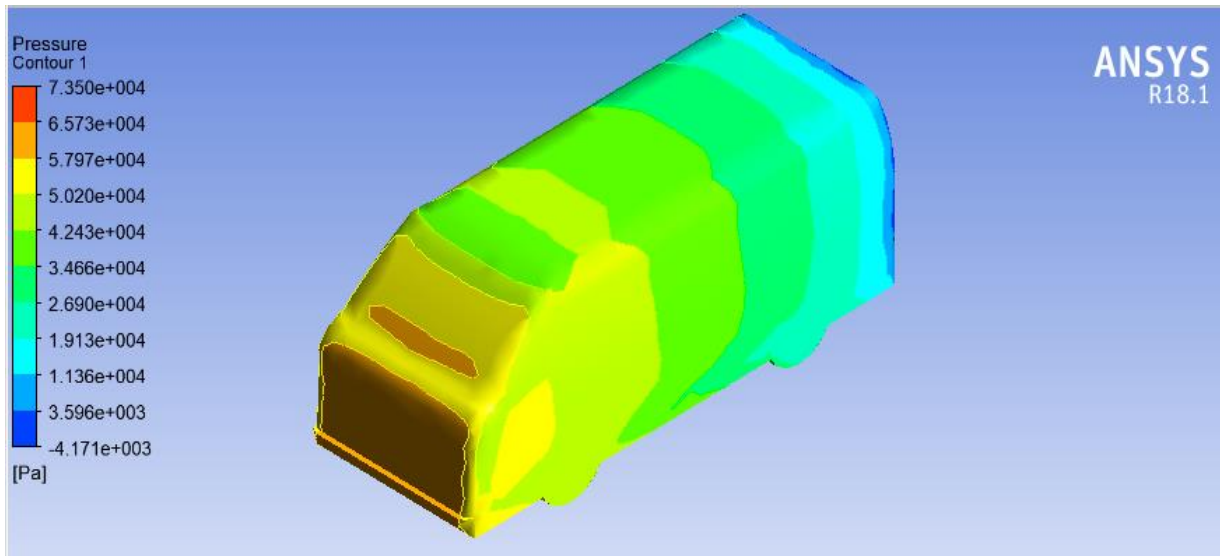


Figure 5.8. Pressure contour of the baseline model at 80 kmph.

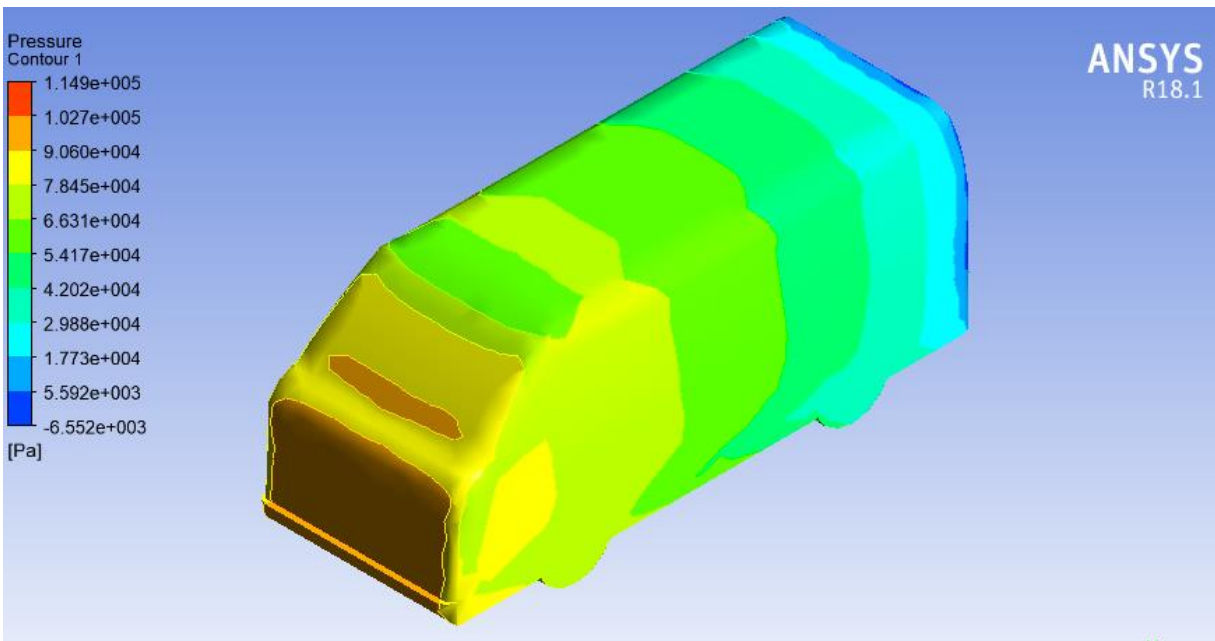


Figure 5.9. Pressure contour of the baseline model at 100 kmph.

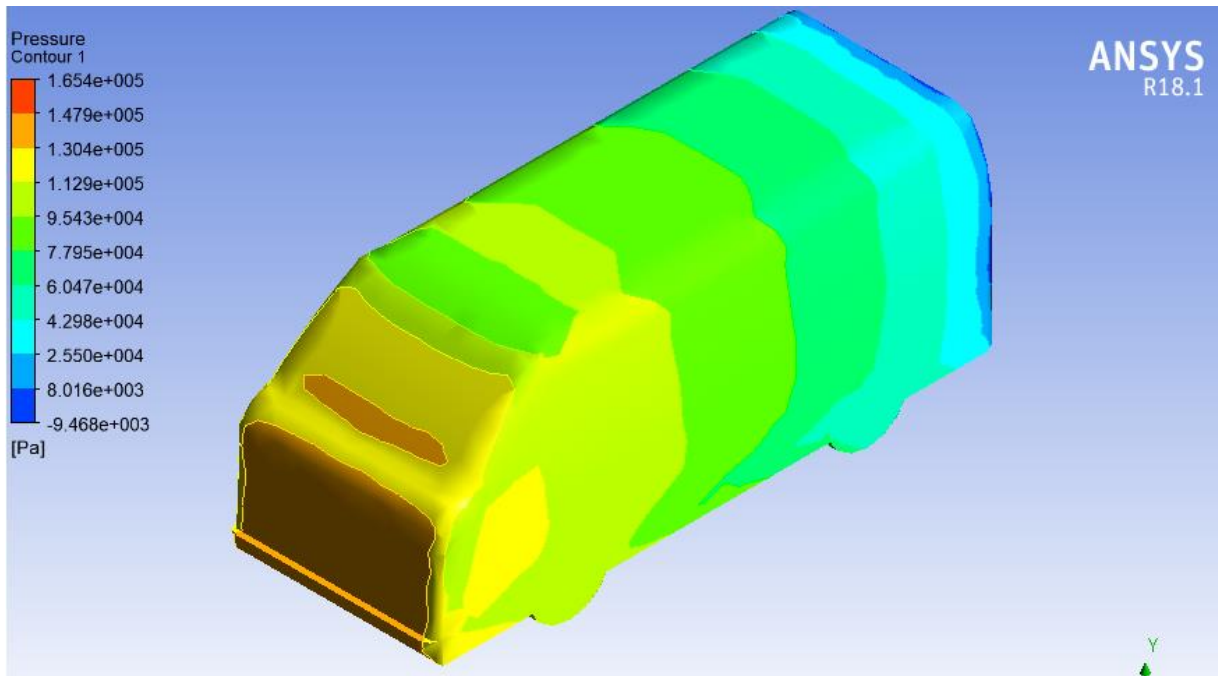


Figure 5.10. Pressure contour of the baseline model at 120 kmph.

The pressure contours plots indicate that the pressure on the front end of the vehicle reaches a maximum as the fluid flowing towards the vehicle comes to a stagnation state. The light blue region indicates atmospheric pressure and red region is the high-pressure region. It is visible that the car frontal area has a larger pressure effect than other regions, so by improving in the design of the car frontal area the pressure effect over the car can be reduced. The frontal areas are too much vertically inclined, for this the air flow strikes the frontal areas which create pressure on car. When the car is in running condition, it has to overcome that extra pressure effect. For this extra pressure a portion of velocity spends to overcome the pressure. For this reason, the velocity of the car reduces. The pressure distribution at a different speed on the body of the baseline model fluctuates a lot on its boundaries. Stagnation is the region that comes in contact with the air in the first instance when the car is in motion. It can be further observed that the pressure of the fluid over the entire body drops to negative values. The rear wall of the bus experiences low pressure compared to the front due to the flow separation and wake formation behind the bus. The high-pressure difference existing from the front to the rear end of the bus leads to a very high “Pressure Drag” this will again result in an increase in the aerodynamic drag of the vehicle. To reduce the pressure difference between the front and the

rear body of a car incorporation of spoilers at the rear end of the vehicle is necessary since it can increase the pressure at the rear end of the vehicle and resulting in reduction of the pressure difference between the front and rear body of a car.

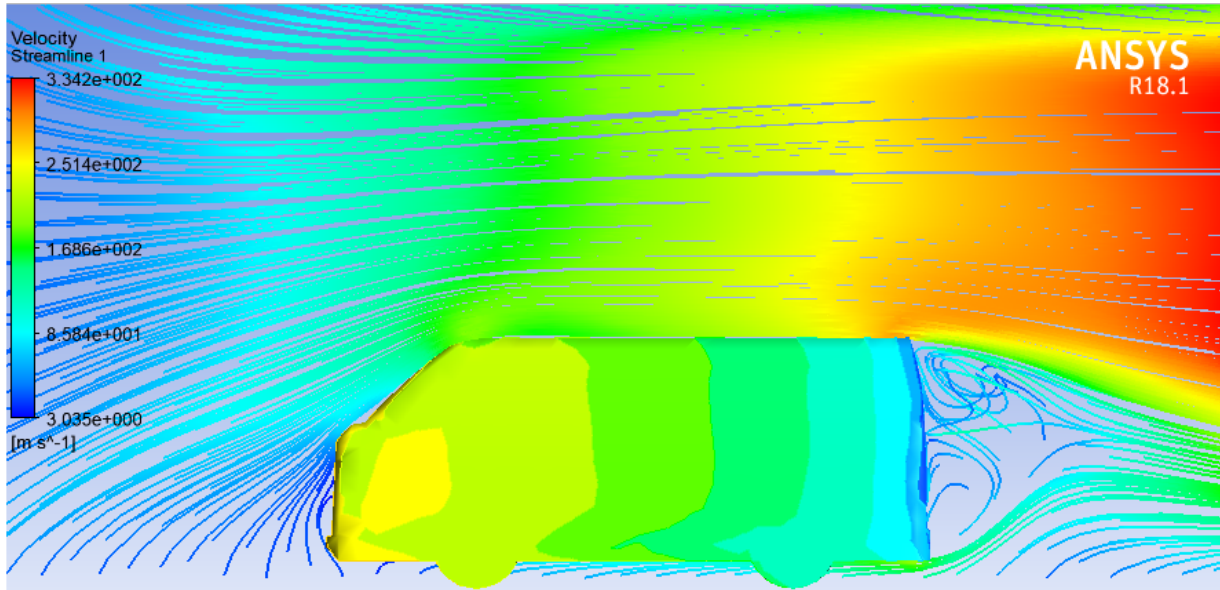


Figure 5.11. Velocity contour of the baseline model at 80 kmph.

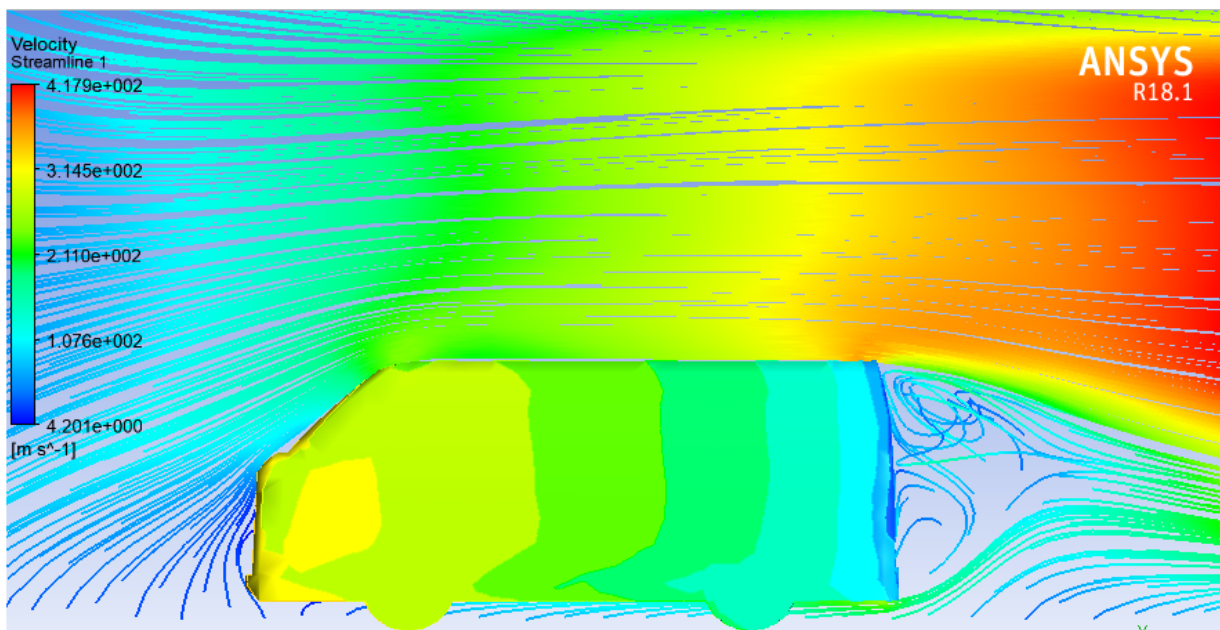


Figure 5.12. Velocity contour of the baseline model at 100 kmph.

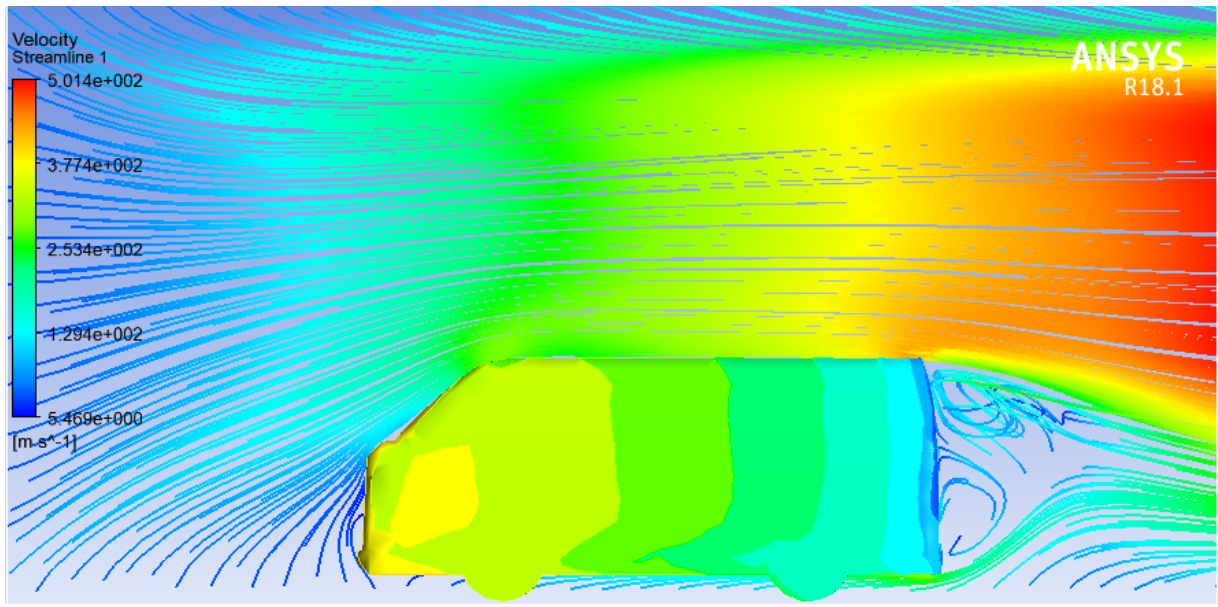


Figure 5.13. Velocity contour of the baseline model at 120 kmph.

Figure 5.11-5.13 shows the velocity distribution over the car body. Visualization of path lines provides important information about the flow structure. The areas of streamlined flow, separation, and recirculation, vortices generation can be determined. This gives a more complete picture of the nature of the flow. Streamlines are the lines that are tangential to the flow velocity. As can be seen, the streamlines are in direction with the car surface and some streamlines tend to bend toward the rear side of the car. Also, the wake region in the back of the car can be observed clearly. The red region shows high velocity region and blue region shows low velocity. The airflow velocity over the roof reduces significantly and remains constant as it flows towards the rear of the car. It should be noted that the velocities were 22.22, 27.78, and 33.33 m/s and the maximum velocities for baseline model in the flow field due to flow separation were nearly 334.2, 417.9, and 501.4 m/s and the minimum velocities were 3.035, 4.201, and 5.469 m/s, respectively. The colors indicate the velocity magnitude. The green color in the contour represents the velocities close to the input velocity and red color represents high-velocity areas of the contour. However, blue color sections in the contour represent low-velocity areas present on the bus body.

From the velocity distribution shown in Figures 5.11-5.13 it can be observed that flow separation occurs on the rear end of the vehicle due to sudden change in the shape. The flow separation

produces a low-pressure zone and it is also seen that vortices are formed due to recirculation. It can also be inferred that the velocity of flow increases over the bus top surface. This can be attributed to the convergence of flow. The high-speed air will not follow the outer contour of the bus instead it gets separated on the rear end, forming a low-pressure zone and inducing pressure drag on the vehicle.

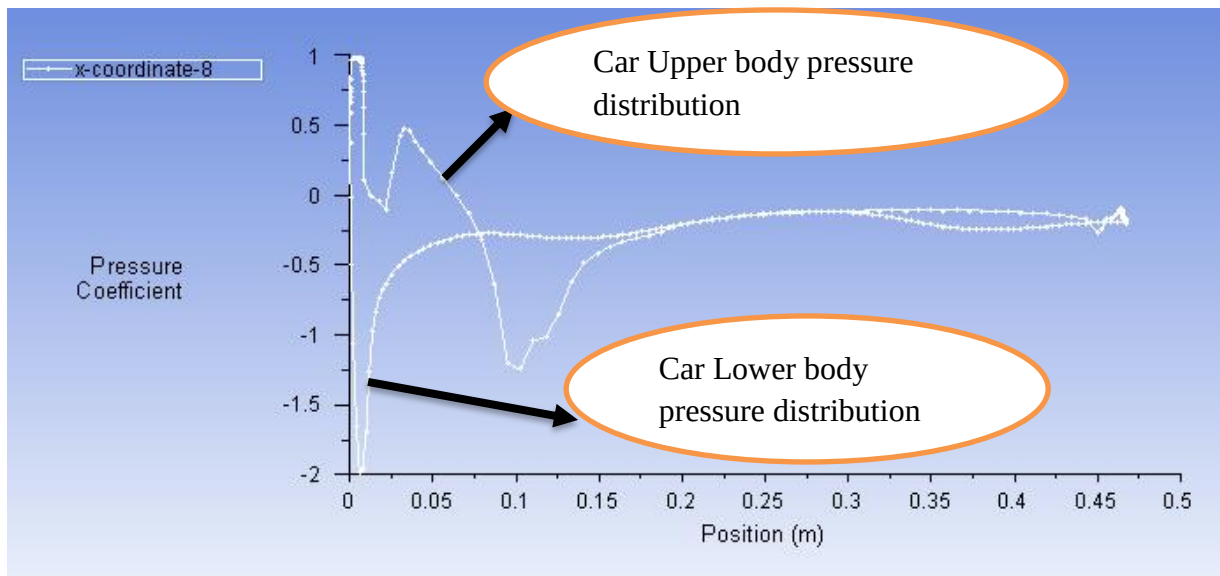


Figure 5.14. Surface pressure distribution along the symmetry plane of the baseline model.

The Pressure Coefficient is a dimensionless characteristic variable that can represent the relative pressure field in the domain. Due to the absence of internal flow, underflow and body roughness, pressure drag occurs to highest percentage. Figure 5.14 shows the surface pressure distribution along the symmetry plane and it indicates that the maximum value is at the front surface of bus where the stagnation point is generated. After the front of the bus roof surface the pressure decreases due to its streamlined shape and then increases gradually to rear roof of the bus. The pressure on the floor of the vehicle is minimum at the front of the bus then gradually increases to Rear of the bus Therefore, it is important to analyze the high and low areas of pressure present on the bus and how these sections affected the airflow and coefficient of drag.

5.1.2 Model One Results (spoiler at zero pitch angles)

Convergence can usually be assessed by progressively tracking the imbalance that is accentuated by the advancement of the numerical calculations of the algebraic equations through each iteration step. This imbalance measures the overall conservation of the fluid properties; they are also commonly known as the residuals.

Analyzing the residual trends is one of the most fundamental measures in determining whether a solution is converged or not. The trends of the baseline simulation were already shown and discussed.

The results of scaled residuals drag and lift coefficients, pressure, and velocity contour at speed of 80, 100, and 120 kmph for model one is depicted from Figure 5.15 to 5.27.

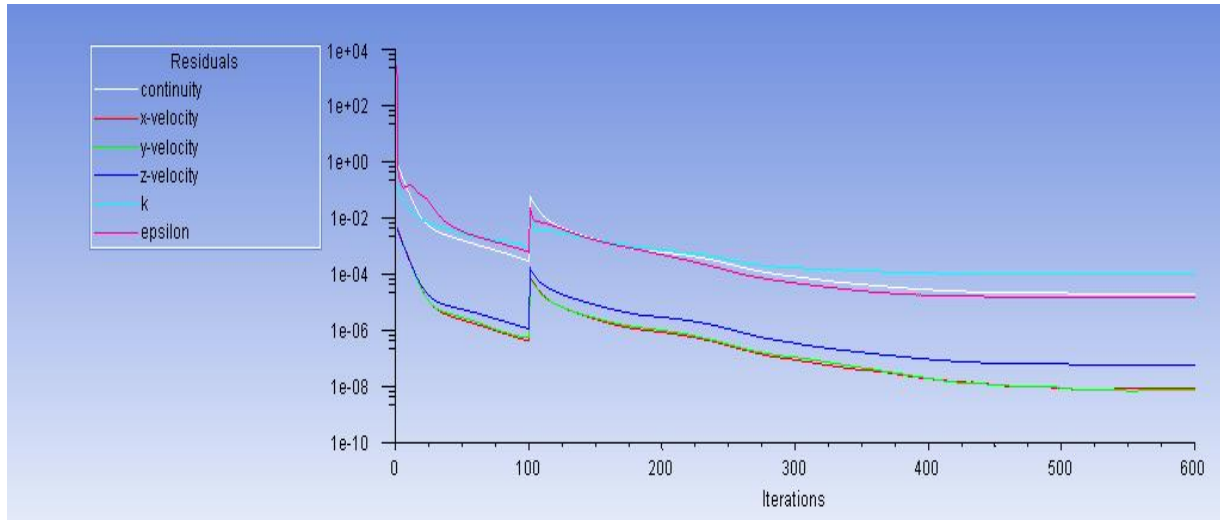


Figure 5.15. Scaled residuals of model one.

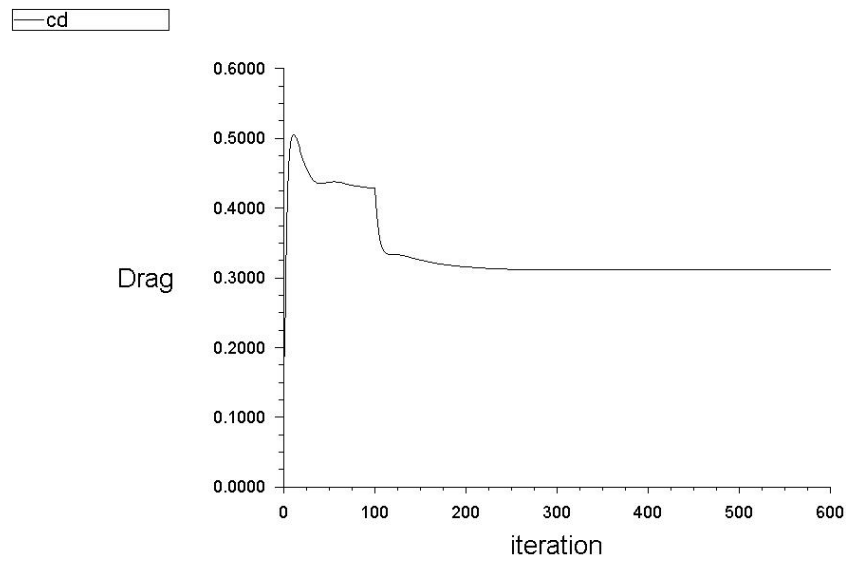


Figure 5.16. Drag coefficient of model one at 80 kmph.

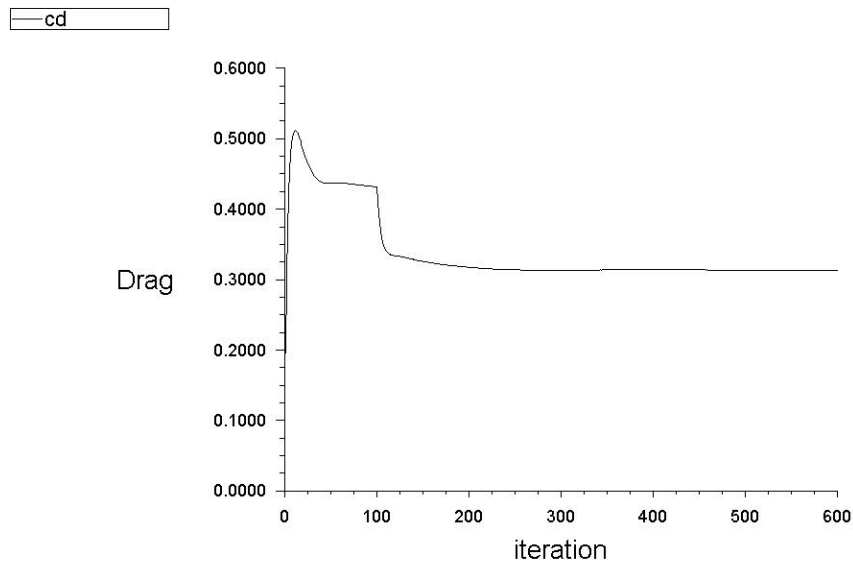


Figure 5.17. Drag coefficient of model one at 100 kmph.

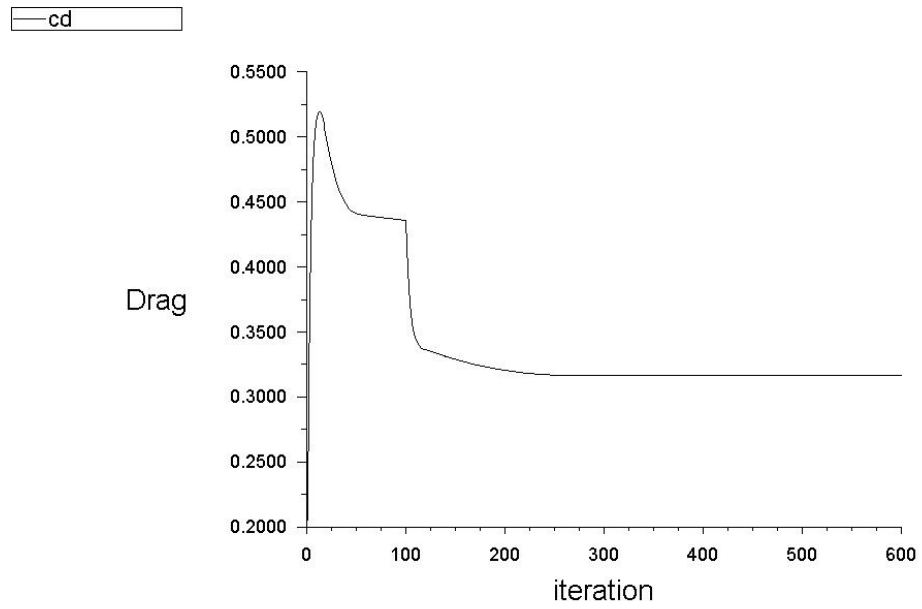


Figure 5.18. Drag coefficient of model one at 120 kmph.

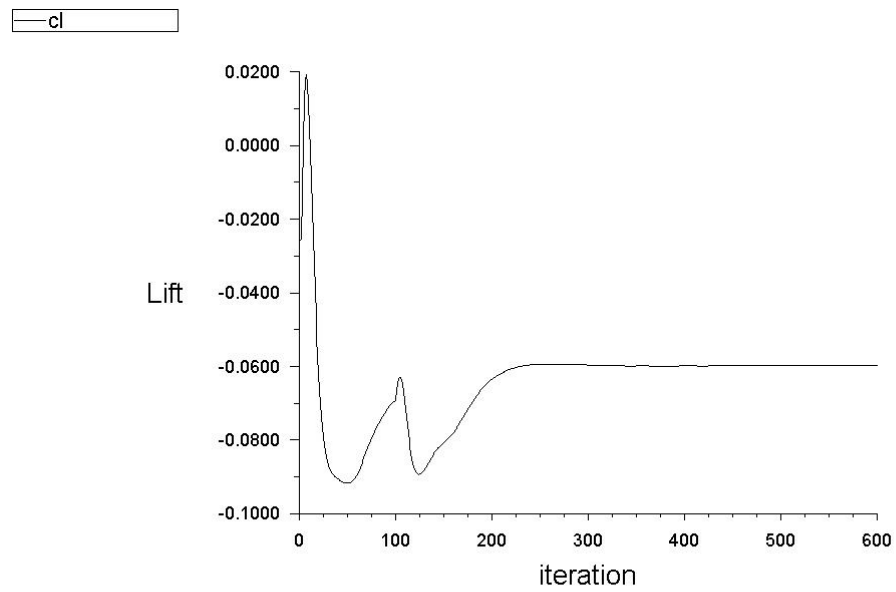


Figure 5.19. Lift coefficient of model one at 80 kmph.

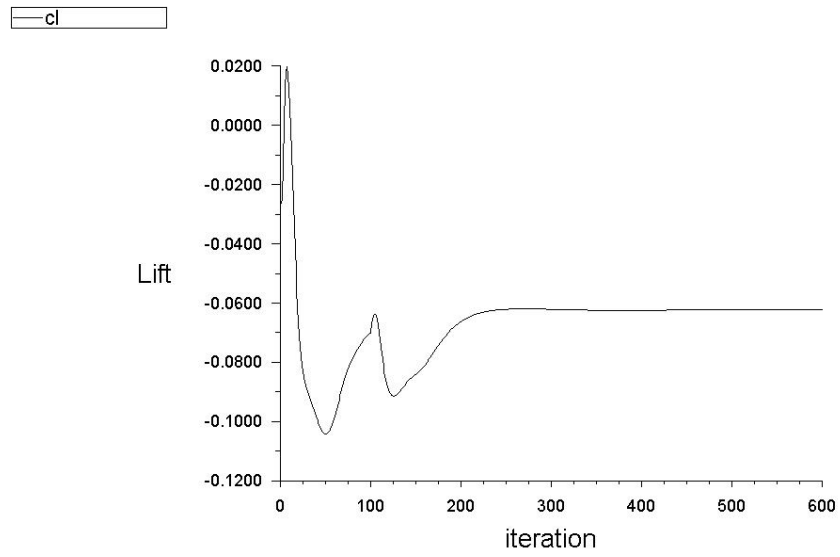


Figure 5.20. Lift coefficient of model one at 100 kmph.

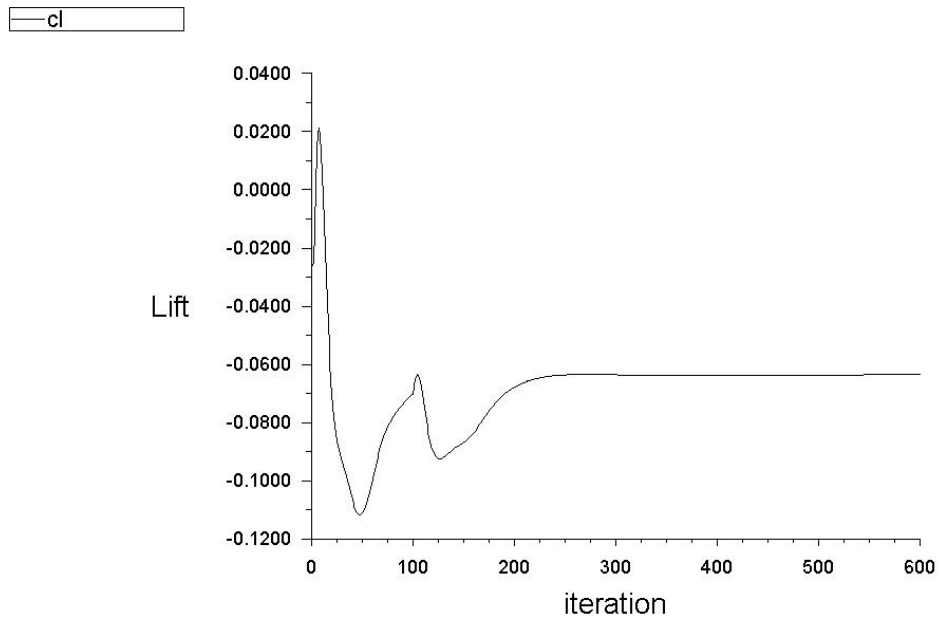


Figure 5.21. Lift coefficient of model one at 120 kmph.

Here the drag and lift coefficients are summarized below in Table 5.2.

Table 5. 2. Summary of drag and lift coefficients of Model one at different speeds

Model	Speed kmph	C_d	C_l
Model one	80	0.31178	-0.059688
	100	0.31246	-0.061289
	120	0.31476	-0.063591

Figures (5.16-5.21) and Table 5.2 showed that the result of the drag and lift coefficients of model one at three different speeds of 80, 100, and 120 kmph. As shown in the Figures 5.16-5.21 both the drag and lift coefficients converge before 500th iteration. The drag coefficient of model one increases with increasing speed. However, the lift coefficient of model one decreases with increasing speed but when we compare it to the baseline model it gives a higher lifting force.

The drag coefficient of model one (Table 5.2) is lower than the baseline model (Table 5.1). The drag coefficient has reduced from 0.31380 (the averaged drag coefficient at 80, 100, and 120 kmph speeds) for the baseline model to 0.313 (the averaged drag coefficient at 80, 100, and 120 kmph speeds) for model one, thus the average reduction in C_d is 0.255 %. This deviation of the drag coefficient of the model one with respect to the baseline model will give a clear idea about how much the drag force is affected by delaying of flow separation. But the reduction in drag is still not satisfactory.

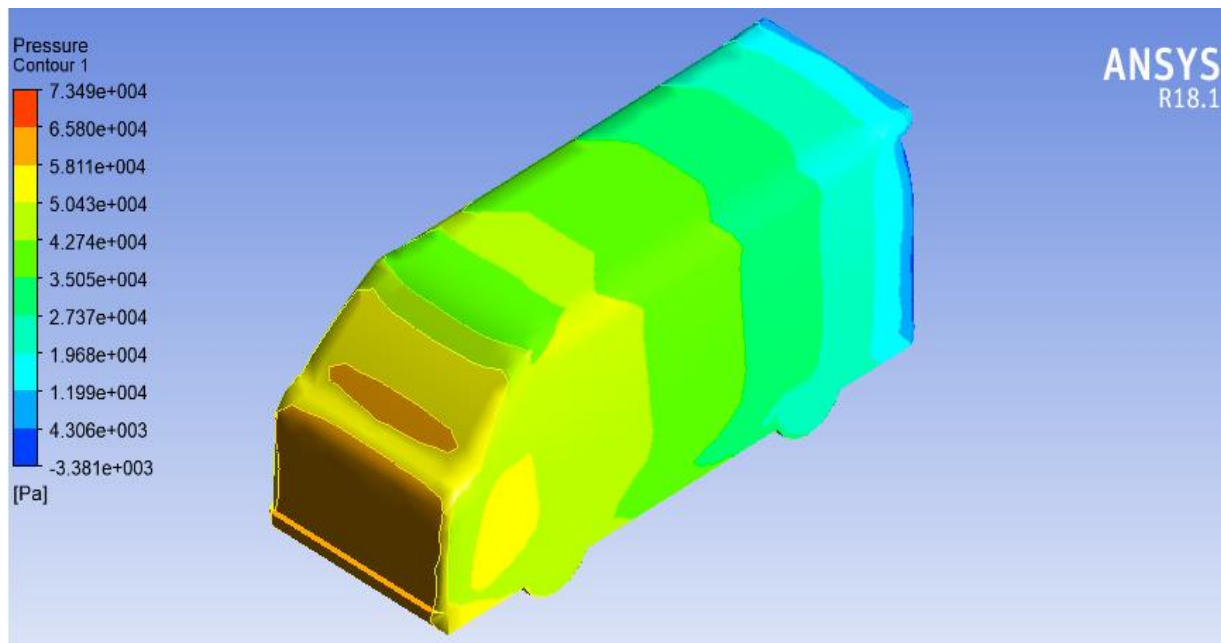


Figure 5.22: Pressure contour of the model one at 80 kmph

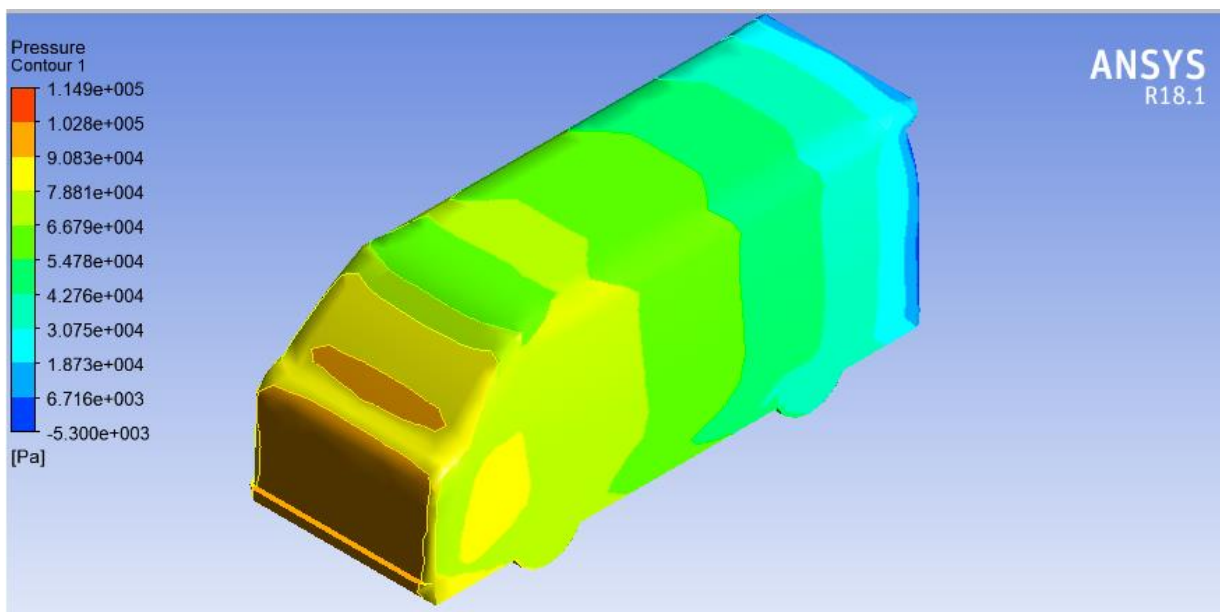


Figure 5.23. Pressure contour of the model one at 100 kmph.

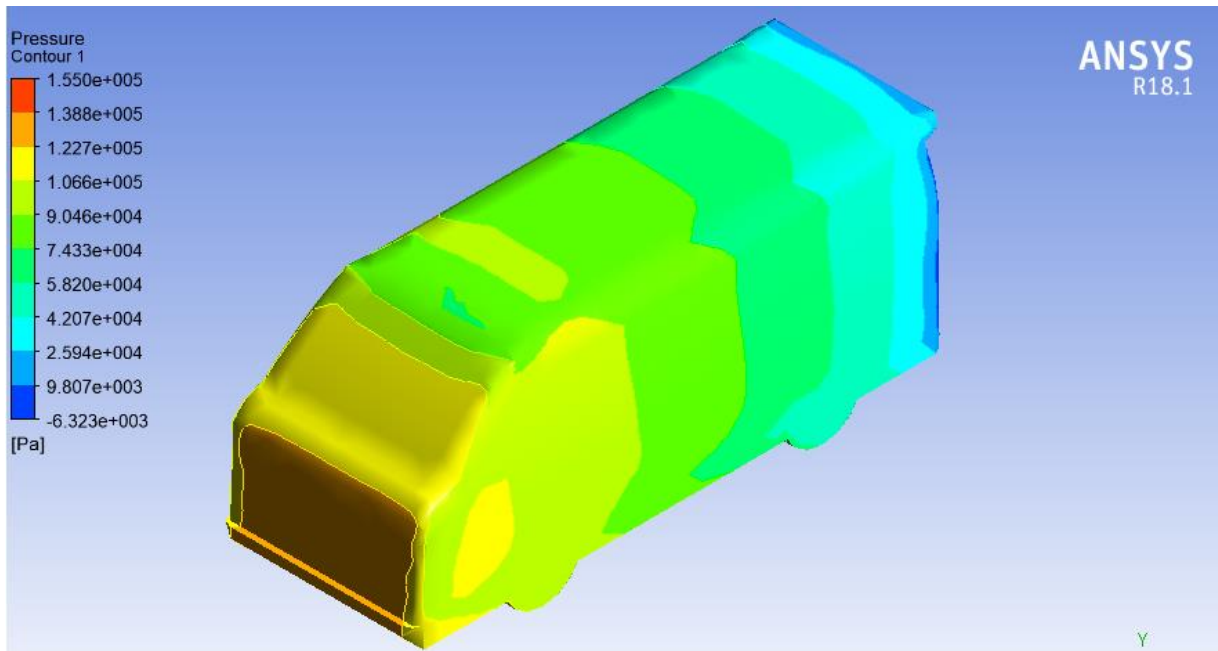


Figure 5.24. Pressure contour of the model one at 120 kmph.

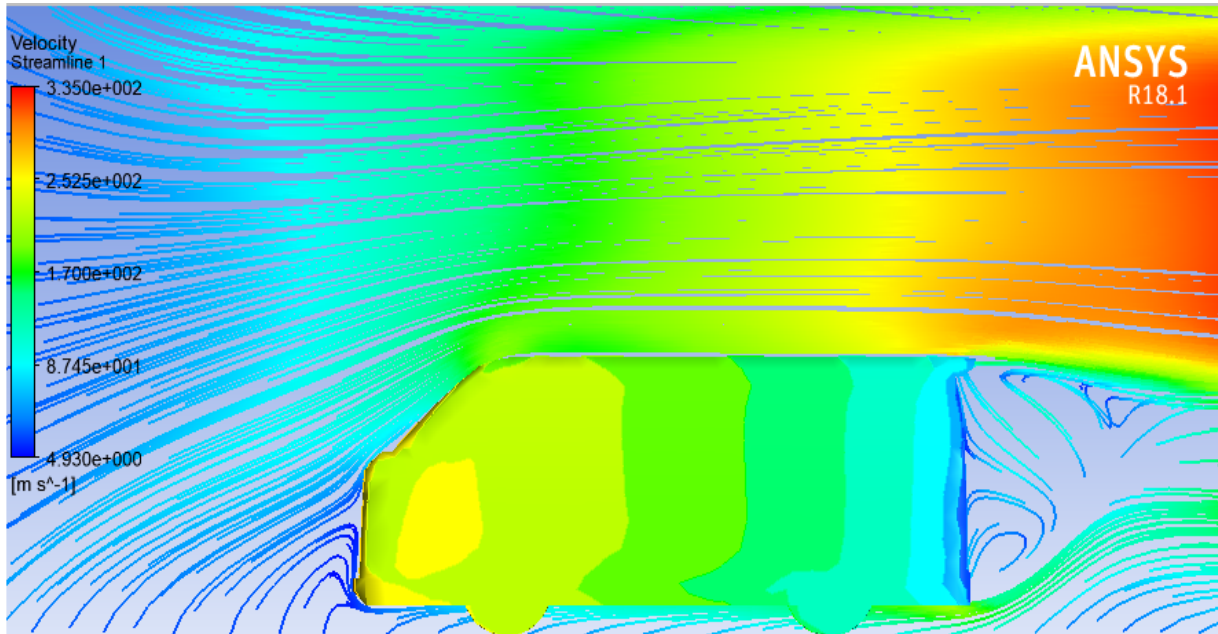


Figure 5.25. Velocity contour of the model one at 80 kmph.

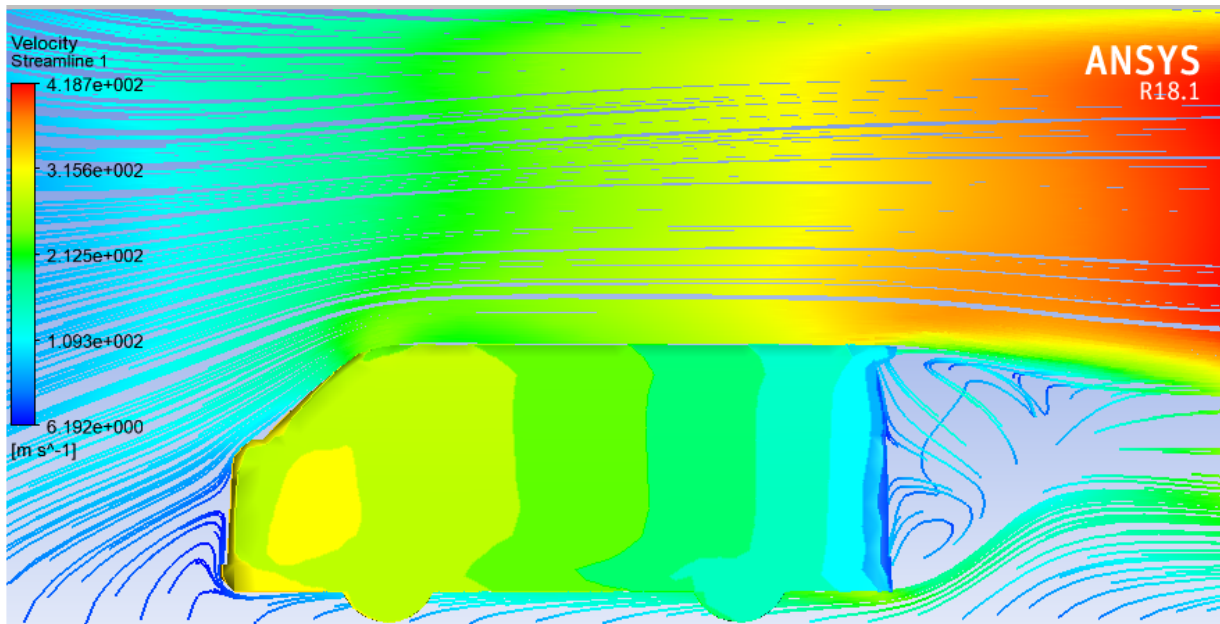


Figure 5.26. Velocity contour of the model one at 100 kmph.

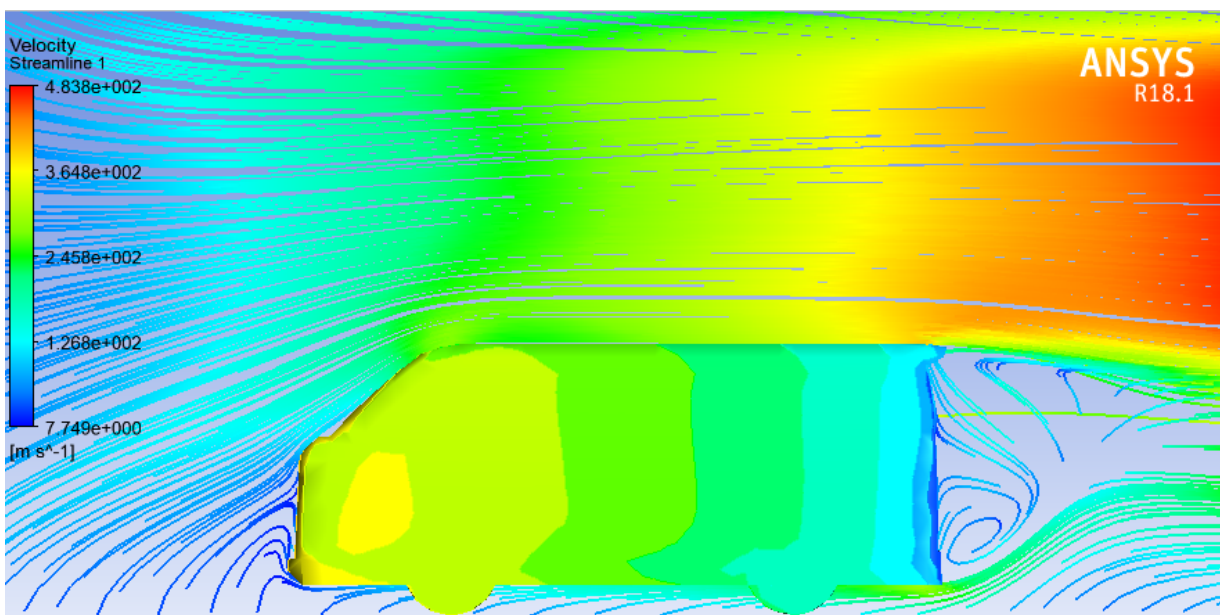


Figure 5.27. Velocity contour of the model one at 120 kmph.

Model one is the baseline model with zero-degree spoiler and the test procedure was the same as baseline model. The flow separation at the rear roof end is made to have a certain delay time behind the car back with the addition of a spoiler and the low-velocity area colored by blue in the wake region is high almost similar to the baseline model. From the velocity streamline

contour (Figure 5.25, 5.26 and 5.27), it can be seen that the wake region gets stronger and wider as the speed increases. Also, the velocity of air is reduced at the front due to the air hitting the bus. Further, it is also reduced behind the bus due to low pressure region created.

In terms of the pressure coefficient, it is not expected to see significant difference compared to the baseline model. Figure 5.28 shows the surface pressure distribution of model one along the symmetry plane and it indicates that the maximum value of pressure is at the front surface of bus where the stagnation point is generated. After the front of the bus roof surface the pressure decreases due to its streamlined shape and then increases gradually to rear roof of the bus. The pressure on the floor of the vehicle is lowest at the front of the bus then increased rapidly after stagnation point. Finally increases gradually to rear of the bus.

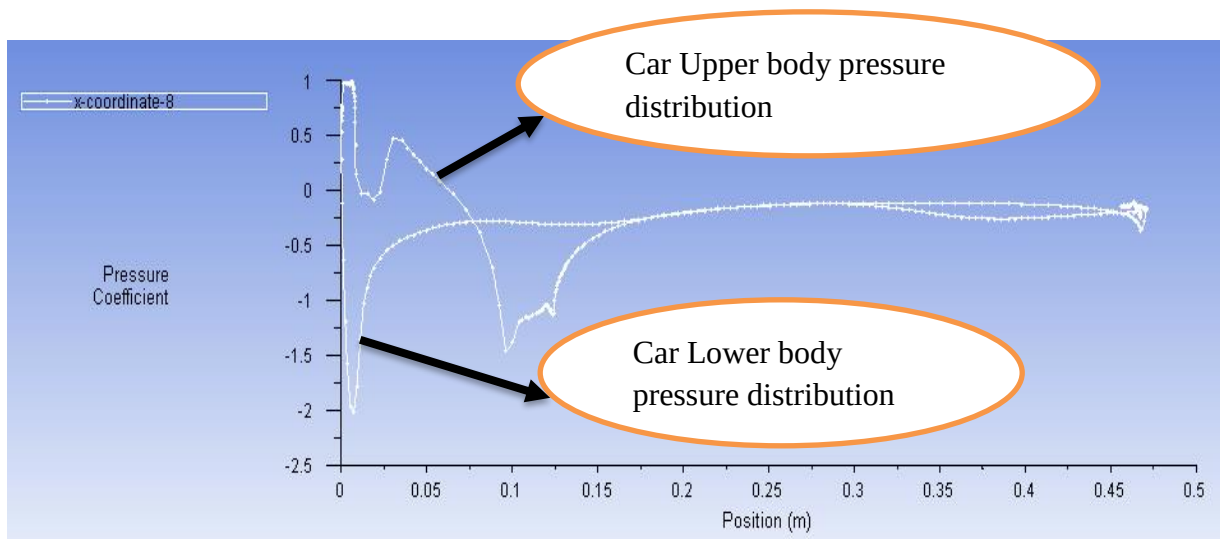


Figure 5.28. Surface pressure distribution along the symmetry plane of the model one.

5.1.3 Model Two Results (zero-degree spoiler with a 5 cm cut)

In this case the spoiler attached at the rear roof end of the baseline model is at zero degree but the difference from model one is it's a wing style spoiler because of it has a cut of 100 cm along its span and 5cm along the spoiler cord length. Result of scaled residuals, drag and lift coefficient, pressure and velocity contour at speed of 80, 100 and 120 kmph for model two are presented below.

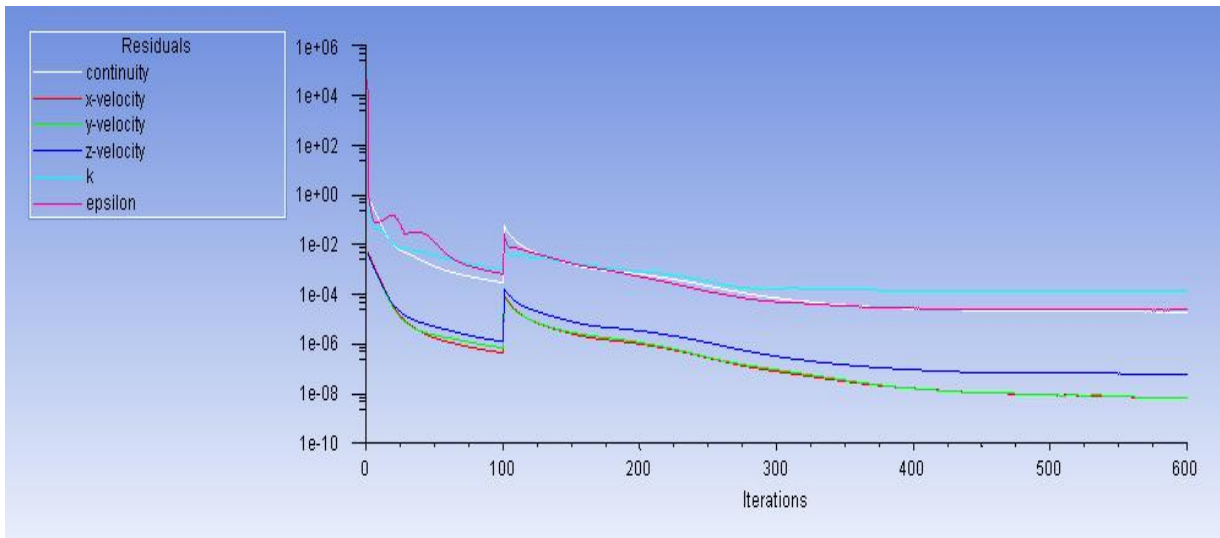


Figure 5.29. Scaled Residuals of Model two.

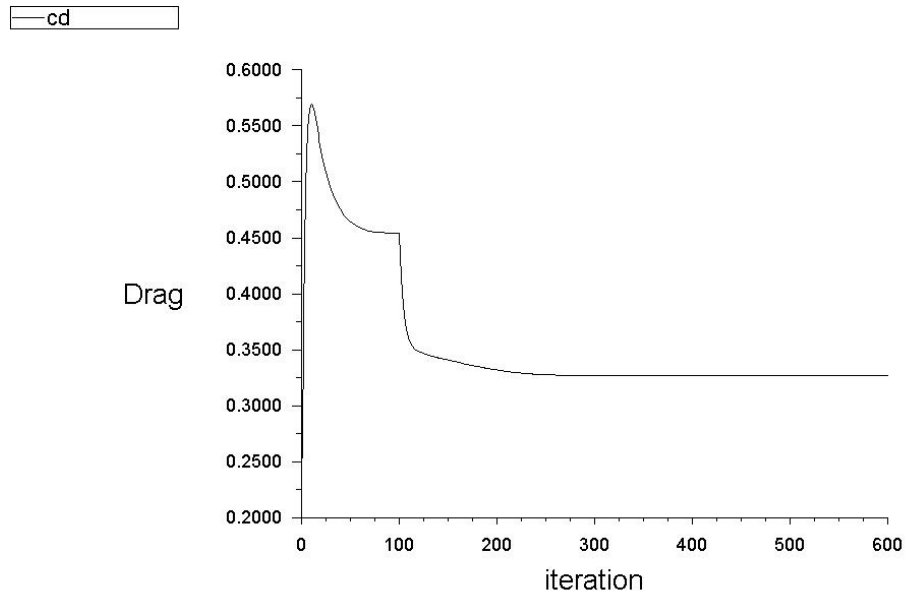


Figure 5.30. Drag coefficient of model two at 80 kmph.

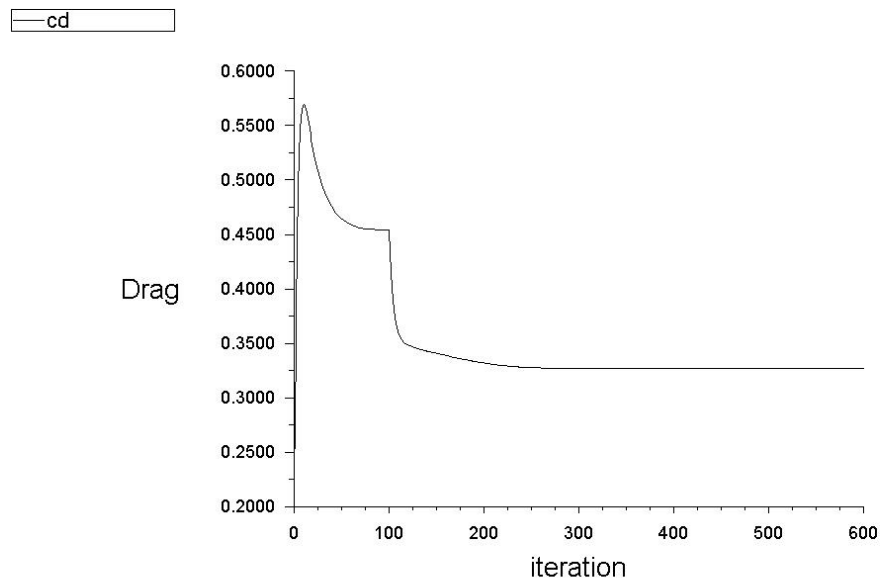


Figure 5.31. Drag coefficient of model two at 100 kmph.

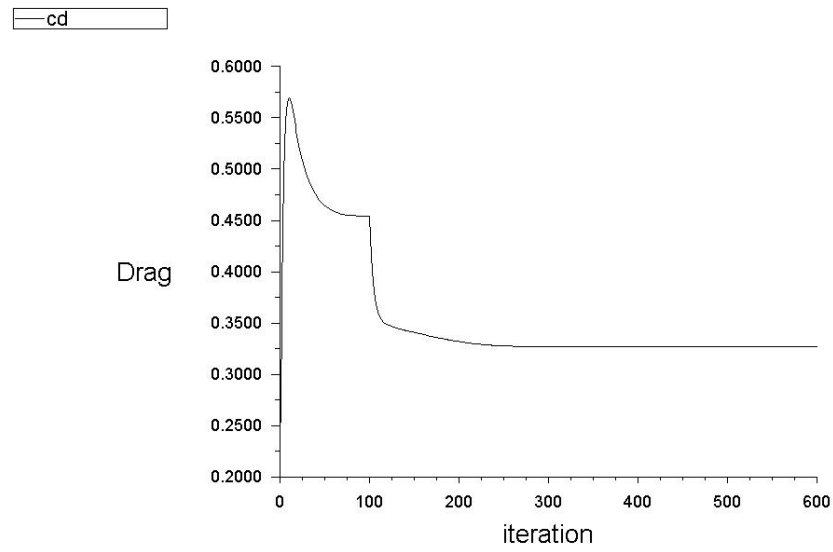


Figure 5.32. Drag coefficient of model two at 120 kmph.

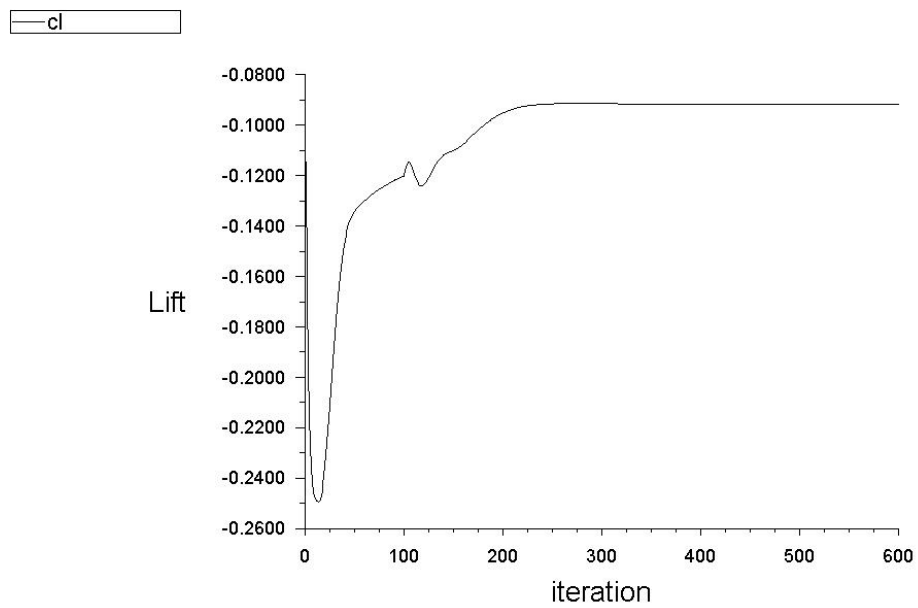


Figure 5.33. Lift coefficient of model two at 80 kmph.

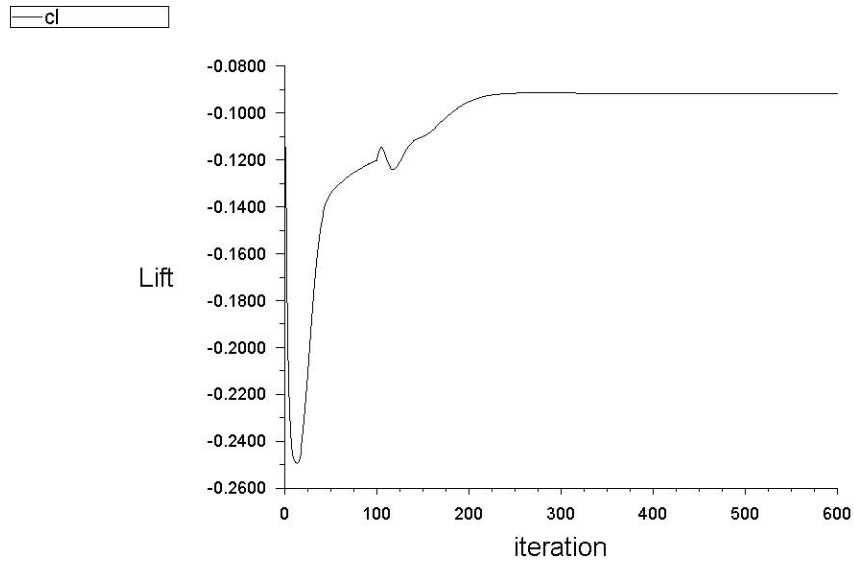


Figure 5.34. Lift coefficient of model two at 100 kmph.

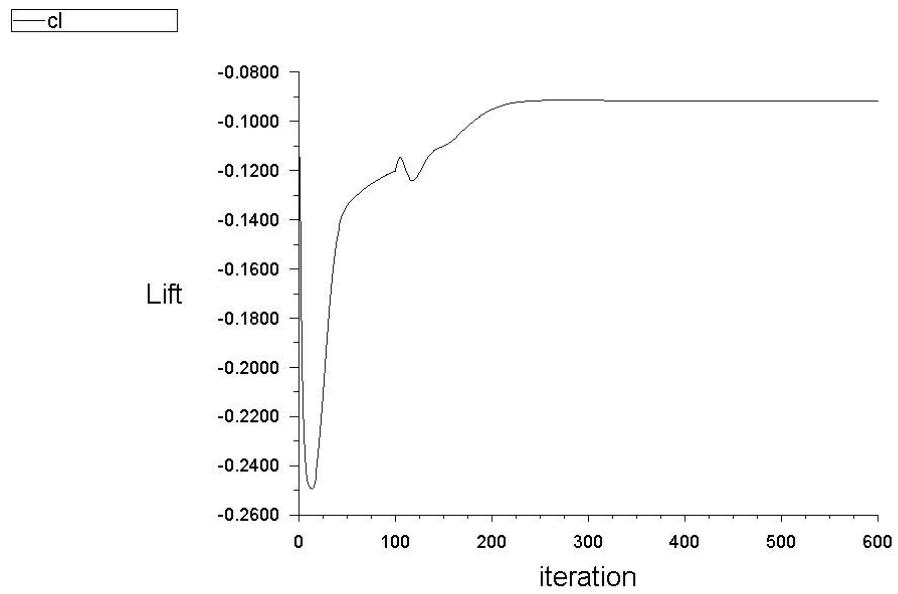


Figure 5.35. Lift coefficient of model two at 120 kmph.

Here the drag and lift coefficients are summarized below in Table 5.3.

Table 5. 3. Summary of drag and lift coefficients of Model two at different speeds

Model	Speed(kmph)	Cd	C _l
Model two	80	0.32310	-0.091519
	100	0.32447	-0.093310
	120	0.32676	-0.094638

The drag coefficient of model two (Table 5.3) is higher than the baseline model (Table 5.1). The drag coefficient has increased from 0.3138 (the averaged drag coefficient at 80, 100, and 120 kmph speeds) for the baseline model to 0.3248 (the averaged drag coefficient at 80, 100, and 120 kmph speeds) for model two, thus the average increase in Cd is 3.5 %. This deviation of the drag coefficient of the model two with respect to the baseline model will give a clear idea about how much the drag force increased if we are using a wing style spoiler but down force for the vehicle gets improved that is important for the vehicle for stability controls.

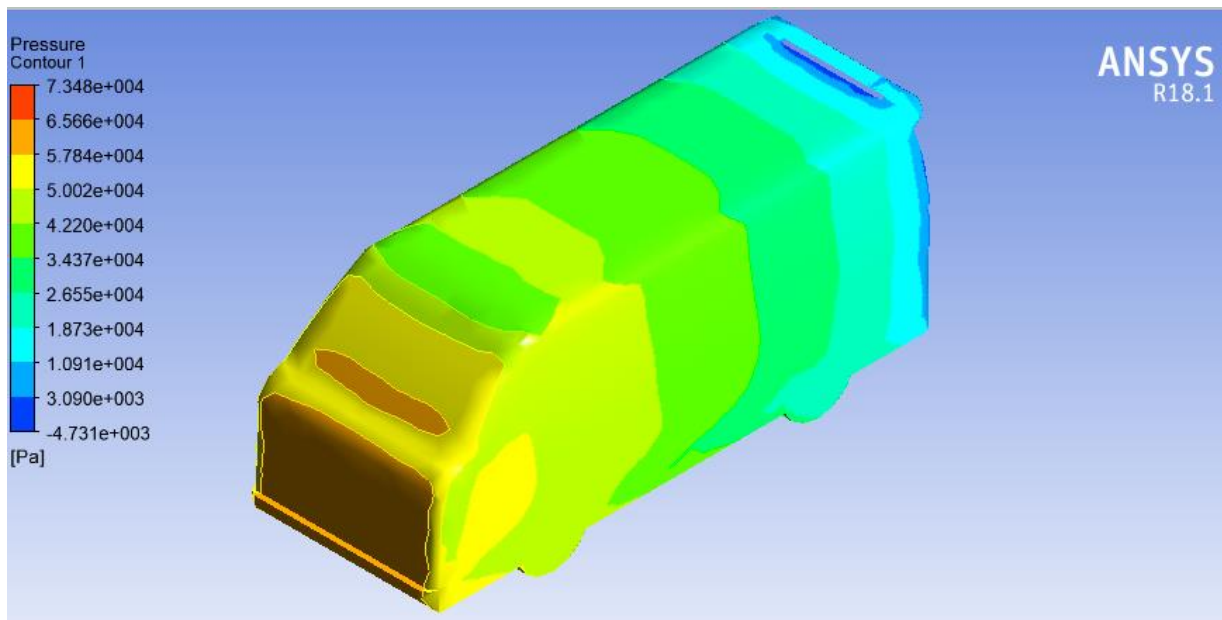


Figure 5.36. Pressure contour of model two at 80 kmph.

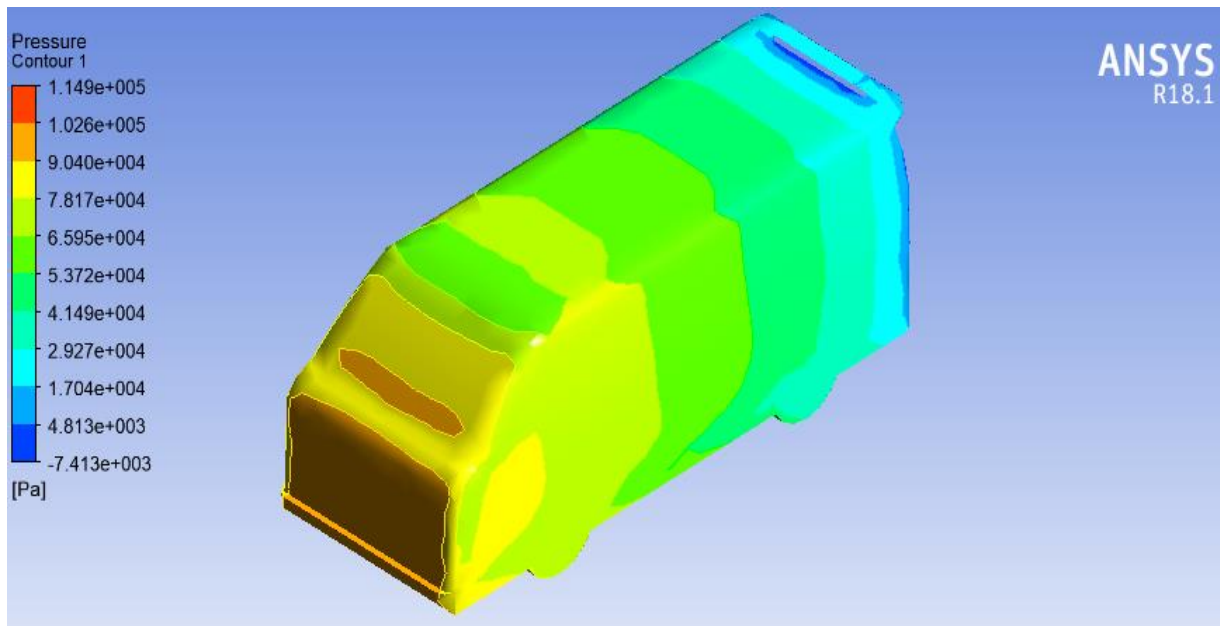


Figure 5.37. Pressure contour of model two at 100 kmph.

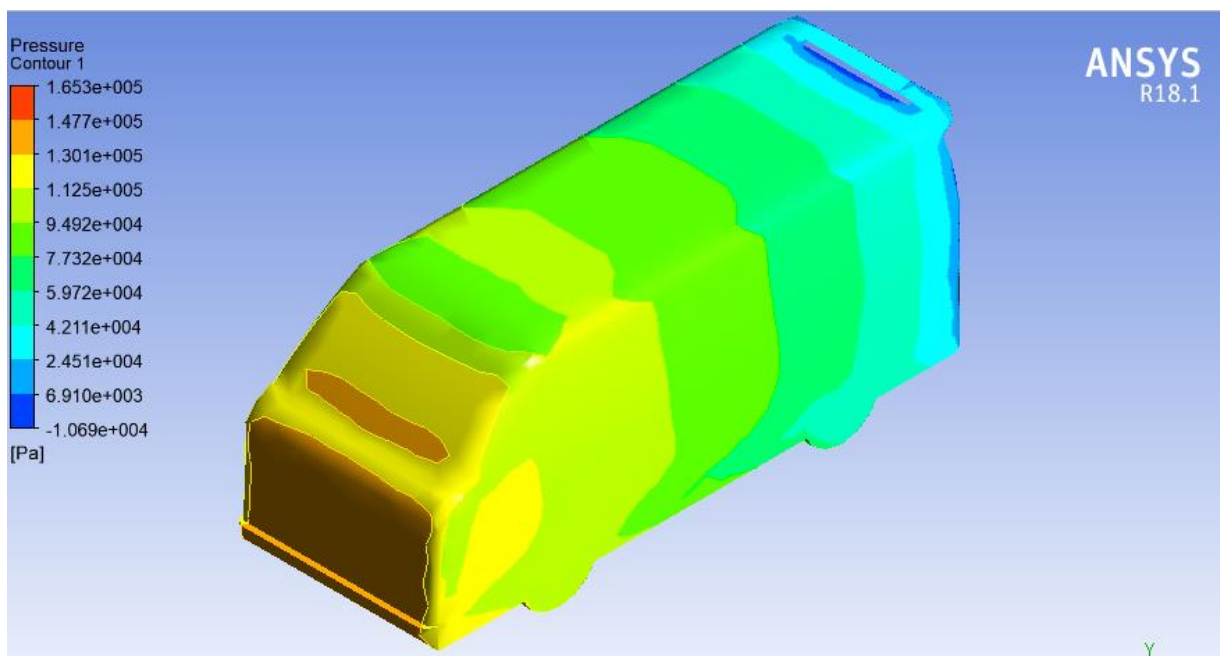


Figure 5.38. Pressure contour of model two at 120 kmph.

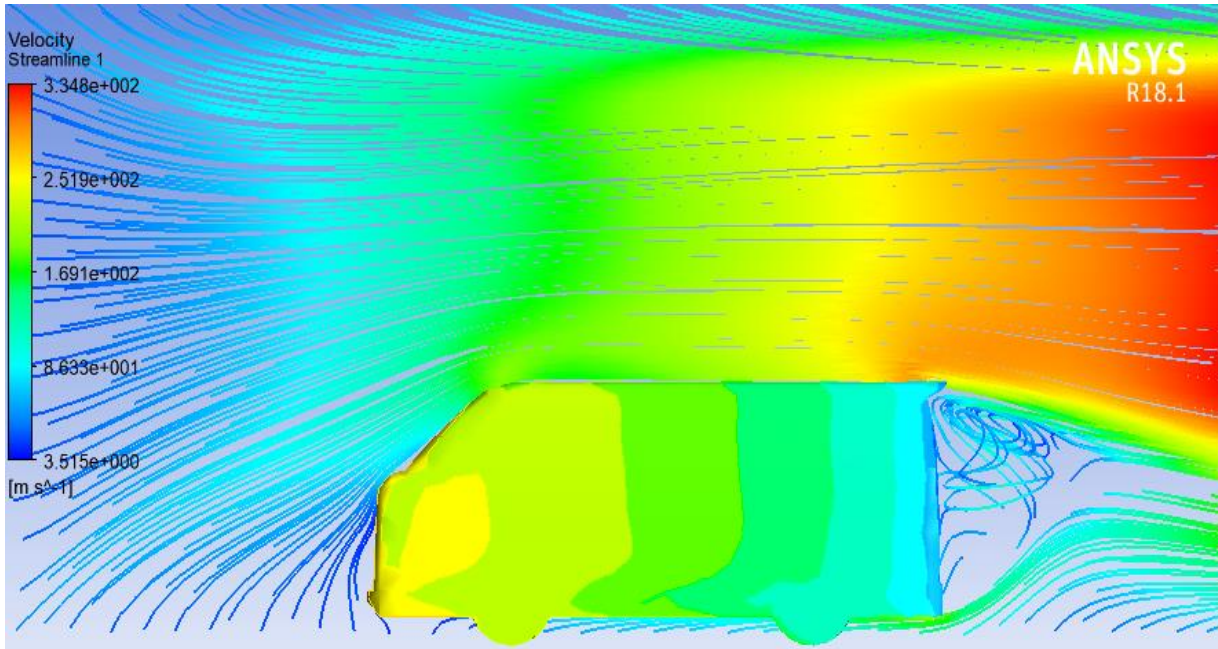


Figure 5.39. Velocity contour of the model two at 80 kmph.

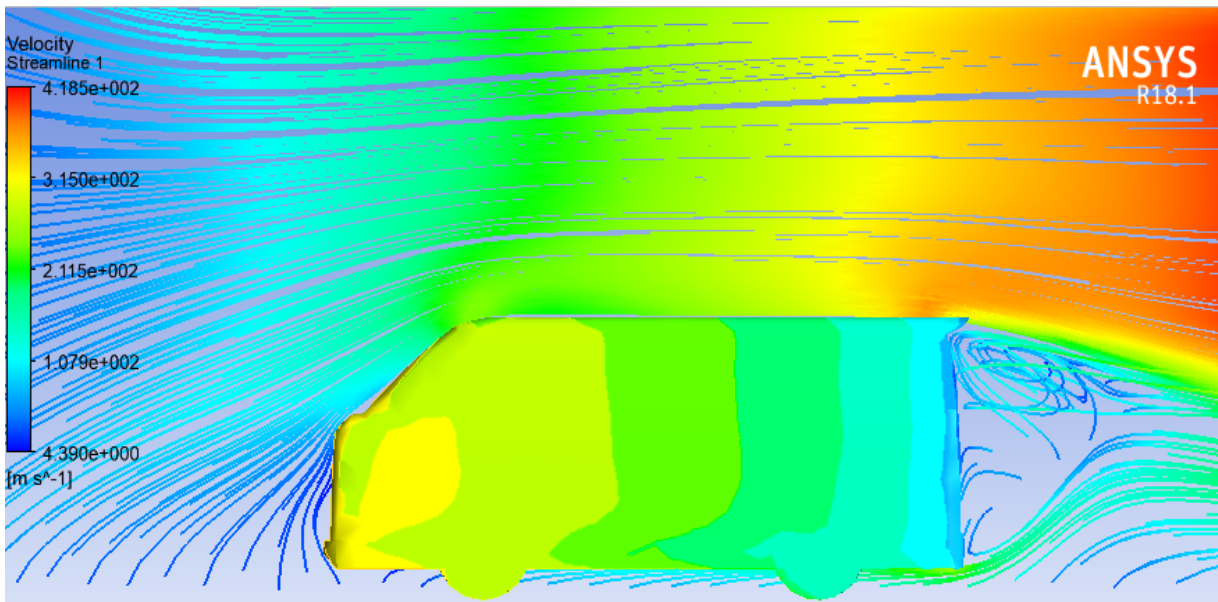


Figure 5.40. Velocity contour of the model two at 100 kmph.

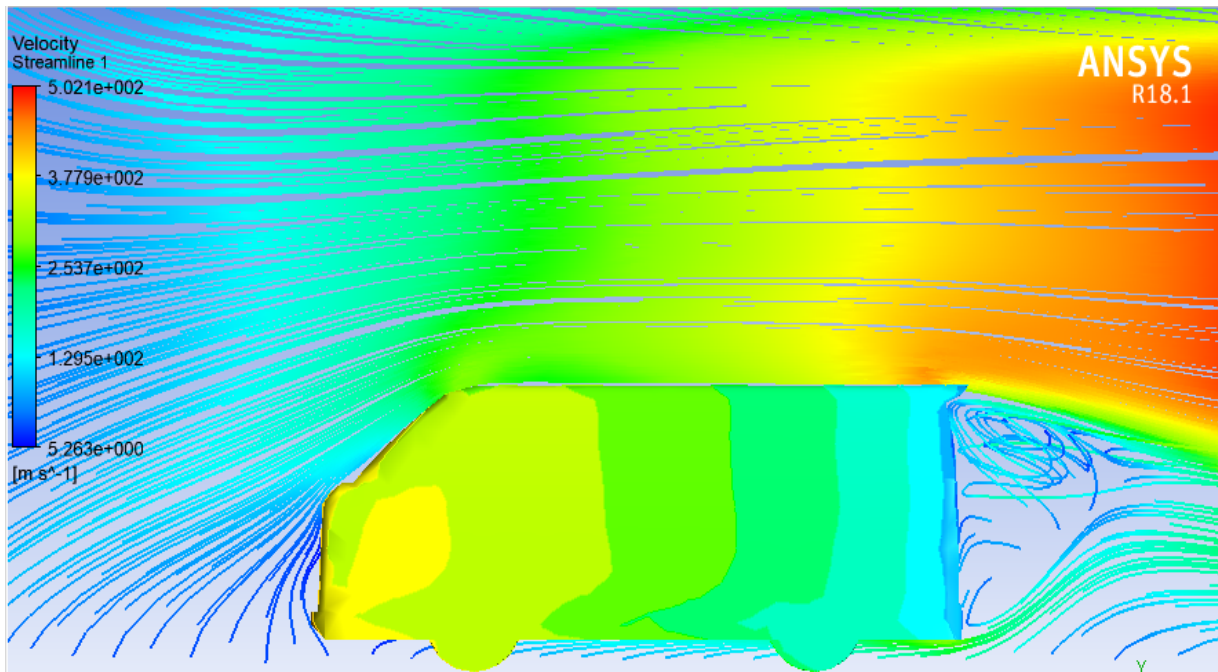


Figure 5.41. Velocity contour of the model two at 120 kmph.

The pressure difference in model two is higher than the baseline model as well as model one this shows as that the drag force is higher and the lifting force is lower as compared with both the models. The velocity distribution also higher and it gets stronger as the speed gets higher and higher.

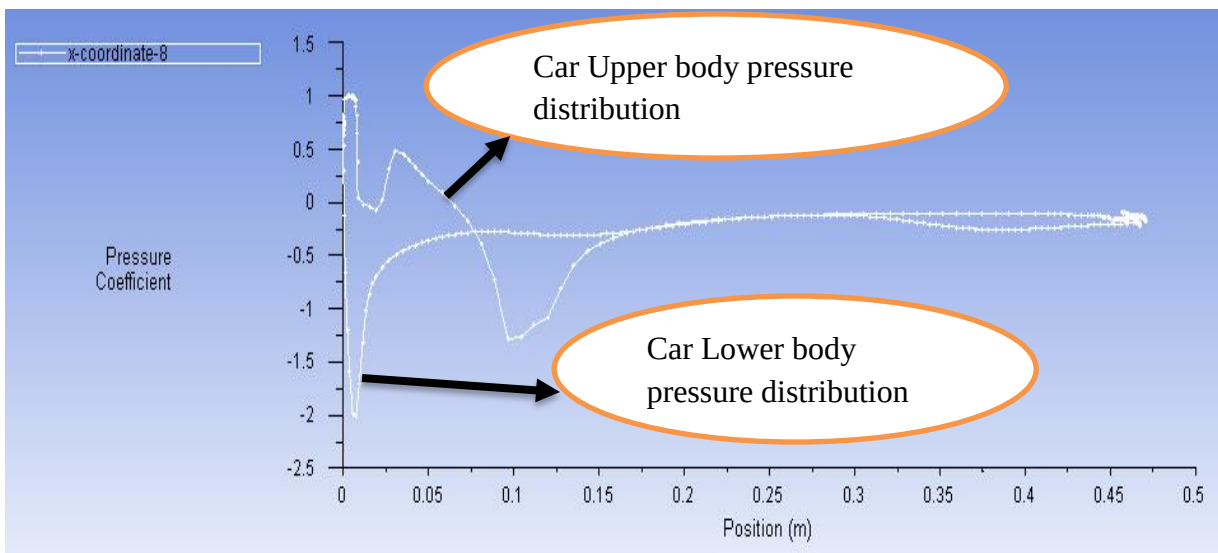


Figure 5.42. Surface pressure distribution along the symmetry plane of the model two.

5.1.4 Model Three Result (spoiler at five degree pitch angle)

In this case the spoiler attached at the rear roof end of the baseline model is attached at five degrees measured from the $+(x)$ axis. Spoilers should be adjusted upward in order to improve the lift force by creating the necessary down force which is important for vehicle stability. Result of scaled residuals, drag and lift coefficient, pressure and velocity contour at speed of 80, 100 and 120 kmph for model three are presented below.

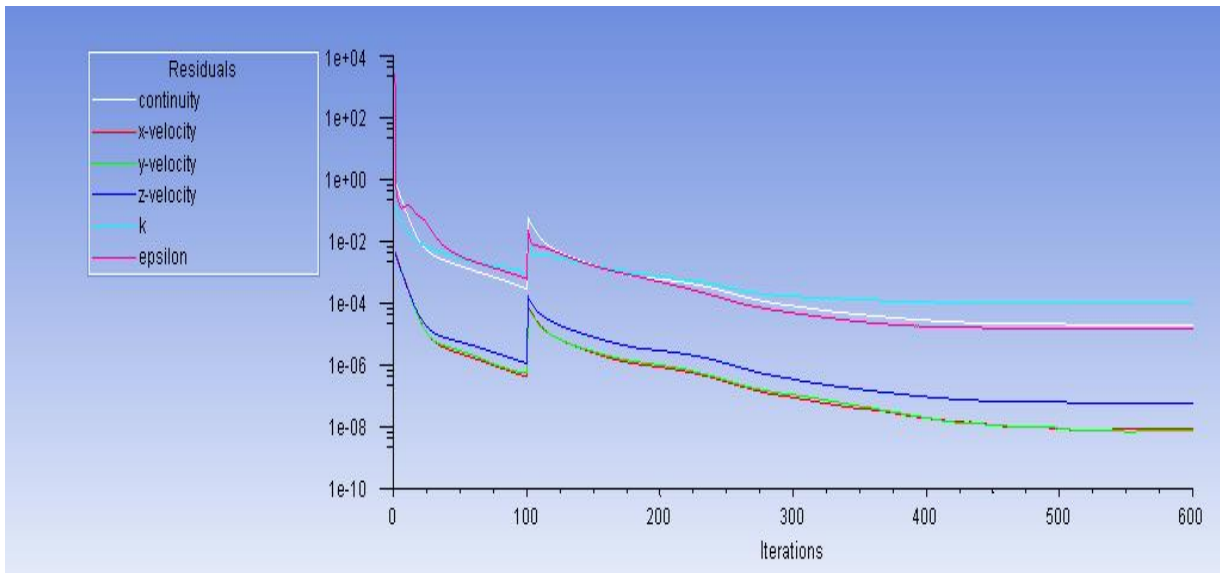


Figure 5.43. Scaled Residuals of Model three at 100 kmph.

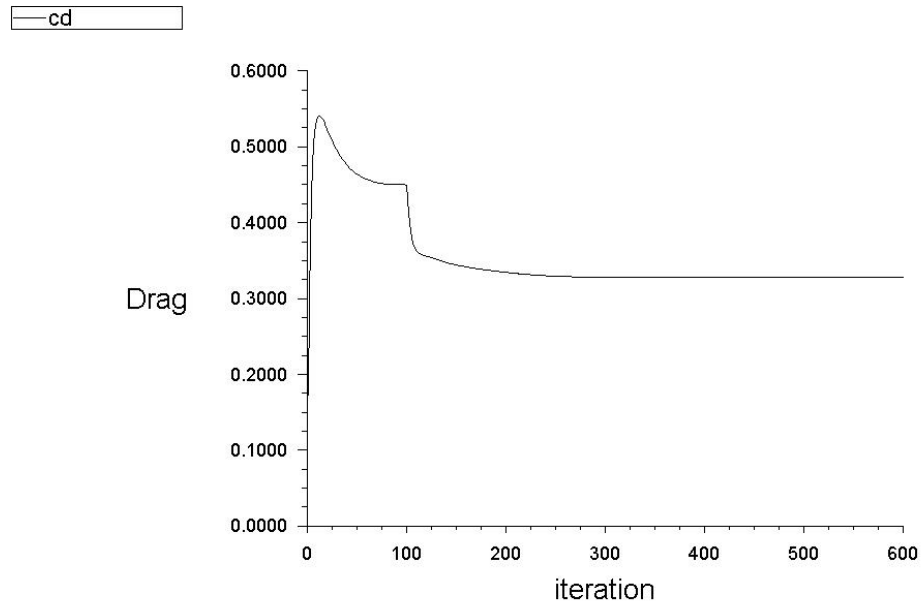


Figure 5.44. Drag coefficient of model three at 80 kmph.

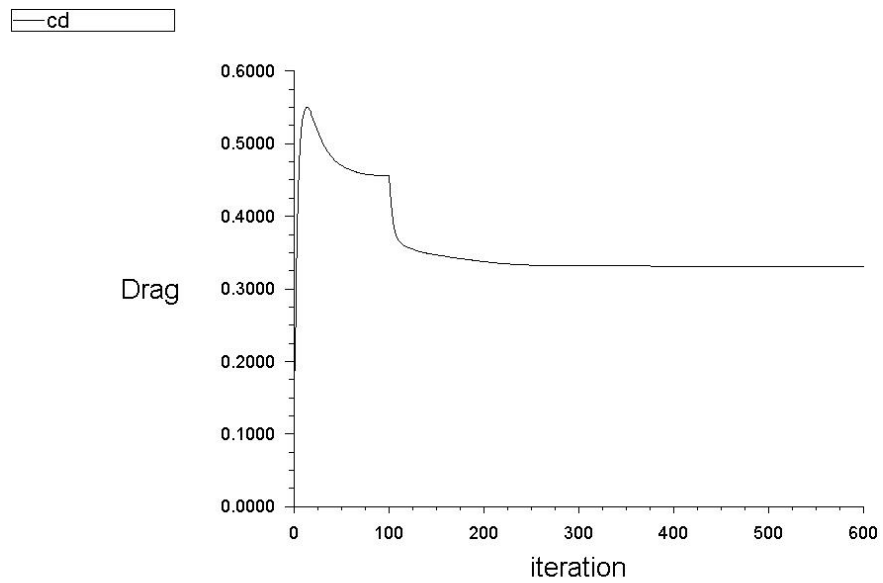


Figure 5.45. Drag coefficient of model three at 100 kmph.

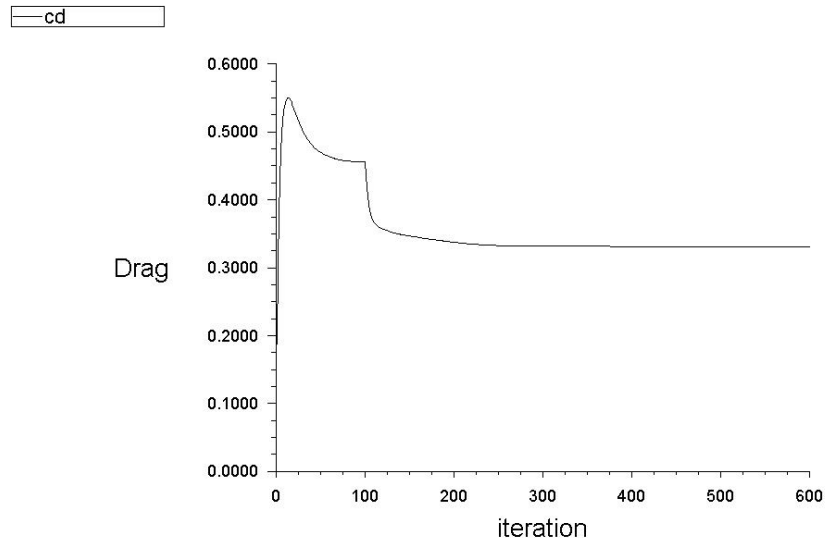


Figure 5.46. Drag coefficient of model three at 120 kmph.

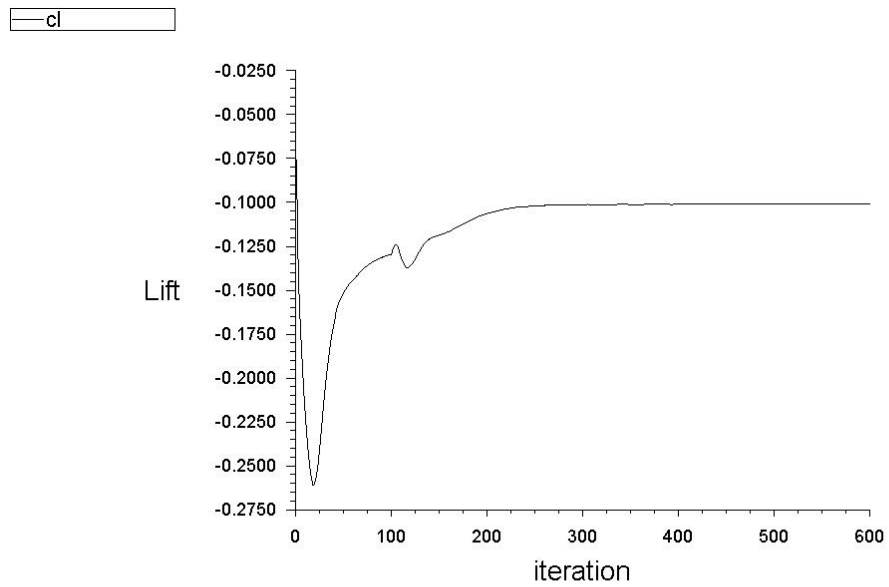


Figure 5.47. Lift coefficient of model three at 80 kmph.

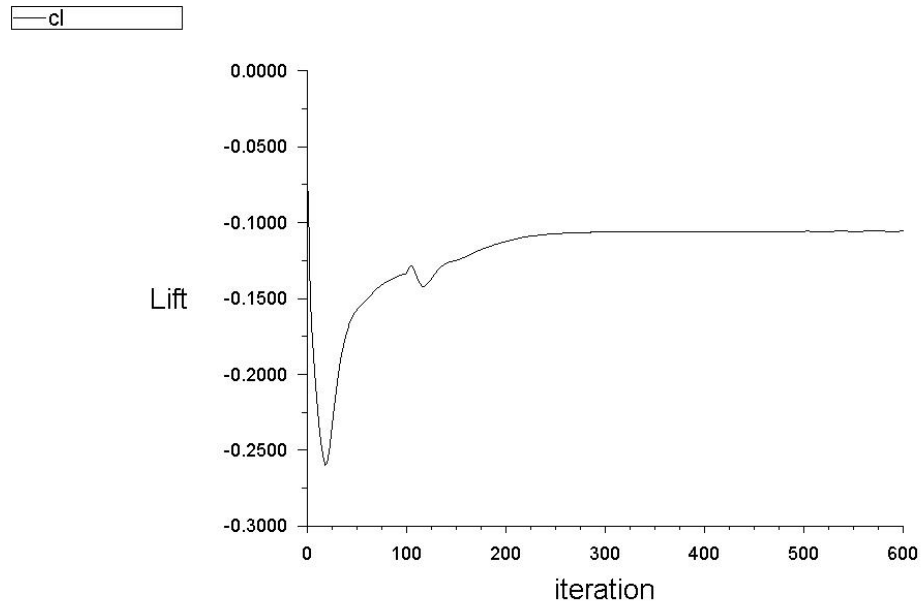


Figure 5.48. Lift coefficient of model three at 100 kmph.

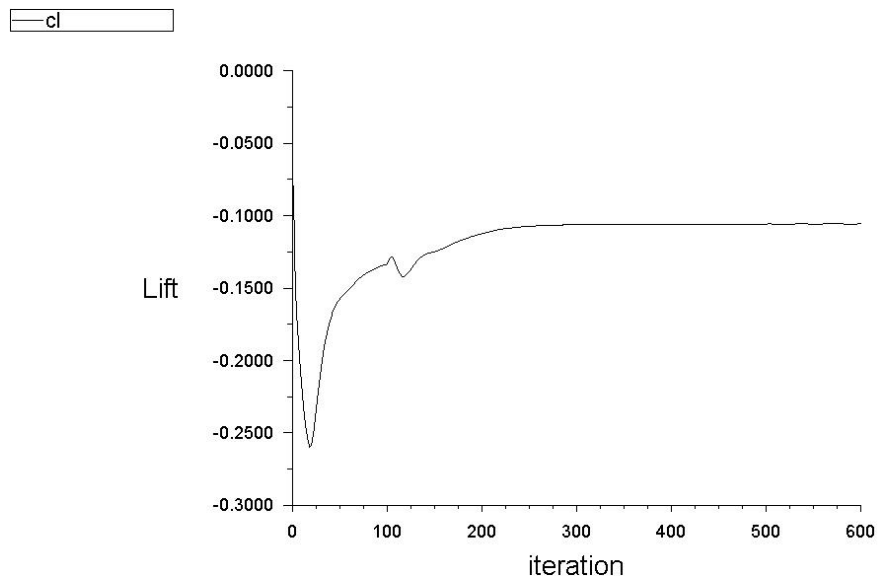


Figure 5.49. Lift coefficient of model three at 120 kmph.

All the drag coefficient graphs shown in Figure 5.44, 5.45 and 5.46 and lift coefficient graphs shown in Figure 5.47, 5.48 and 5.49 of model three at speed of 80, 100 and 120 kmph are summarized in Table 5.4.

Table 5. 4. Summary of drag and lift coefficients of Model three at different speeds

Model	Speed (kmph)	C_d	C_l
Model three	80	0.32740	-0.10070
	100	0.32940	-0.10526
	120	0.33122	-0.10565

Figures 5.44-5.49 and Table 5.4 showed that the result of the drag and lift coefficients of model three at three different speeds (80, 100, and 120 kmph) when simulated the midi bus model by realizable turbulence model in ANSYS Fluent. As shown in the figures, both the drag and lift coefficients converge before the 400th iteration. The drag coefficients were calculated to be 0.32740, 0.32940, and 0.33122 as shown in Table 5.3 and the lift coefficients were -0.10070, -0.10526 and -0.10565 at different speed of 80, 100, and 120 kmph, respectively.

The drag coefficient of model three (Table 5.4) is increased from both the baseline model (Table 5.1) and model one and two (Table 5.2 and Table 5.3) respectively. The drag coefficient has increased from 0.3138 (the average drag coefficient at 80, 100, and 120 kmph speeds) for the baseline model to 0.32934 (the average drag coefficient at 80, 100, and 120 kmph speeds) for model three, thus the average increase in C_d is 4.95%. This deviation of the drag coefficient of the model three with respect to the baseline model will give a clear idea about how much the drag force is affected by upward pitch angle of the spoiler. The lift coefficient of model three decreases with an increasing speed which results high downward force, which is very necessary for vehicle stability at higher speed.

The pressure drag arises from the pressure difference or more generally from non-uniform pressure distribution around the model. The stagnation pressure at the front of the body

represented by the red area is more than that acting on the rear of the body, so resultant force acting on the body in the direction of relative motion exists.

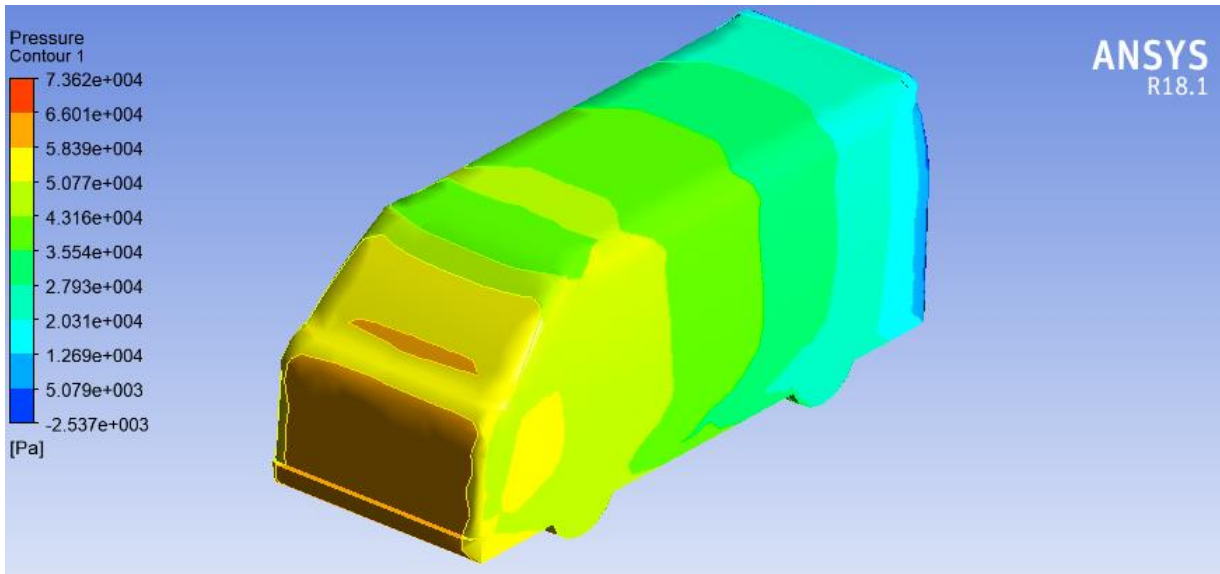


Figure 5.50. Pressure contour of the model three at 80 kmph.

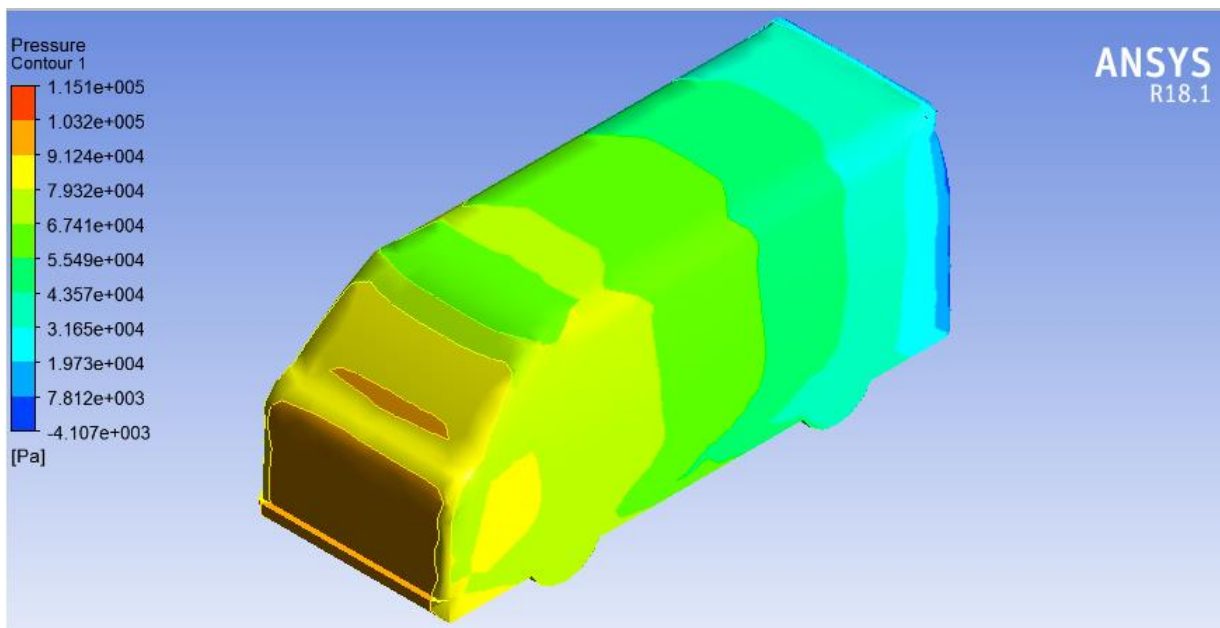


Figure 5.51. Pressure contour of the model three at 100 kmph.

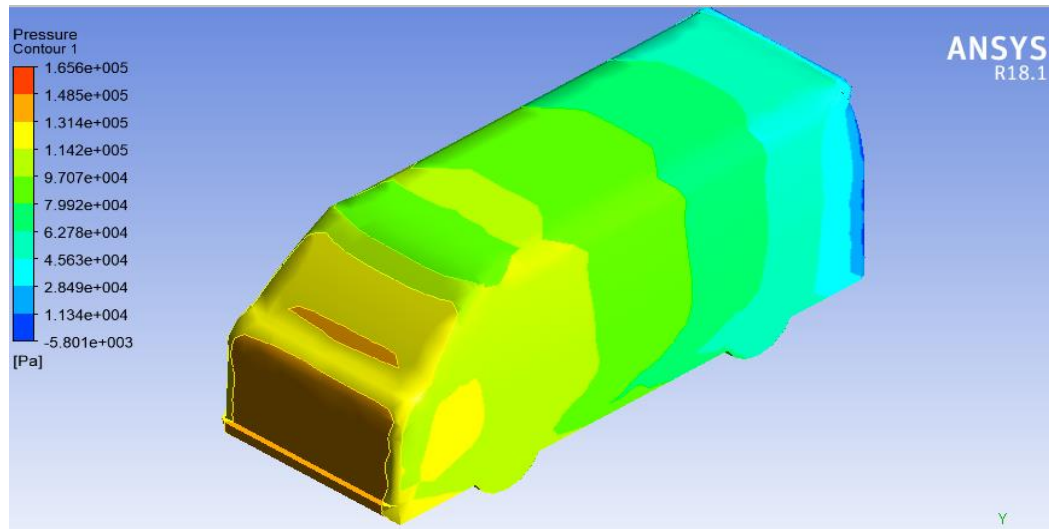


Figure 5.52. Pressure contour of the model three at 120 kmph.

The contour of pressure distribution shown in Figures 5.50, 5.51 and 5.52 indicates the level of pressure effect on the surface of the midi bus. The concept of pressure drags happen when there are two different areas with different level of pressure. In this case, the area on the front of the bus's body (stagnation point colored by red) is relatively higher than the pressure level at the back of the midi bus. The maximum pressure increases with increasing speed. The pressure difference of Model three is higher than the baseline model. This fact suggests that the proposed spoiler pitch angle results in a higher pressure difference than the baseline model.

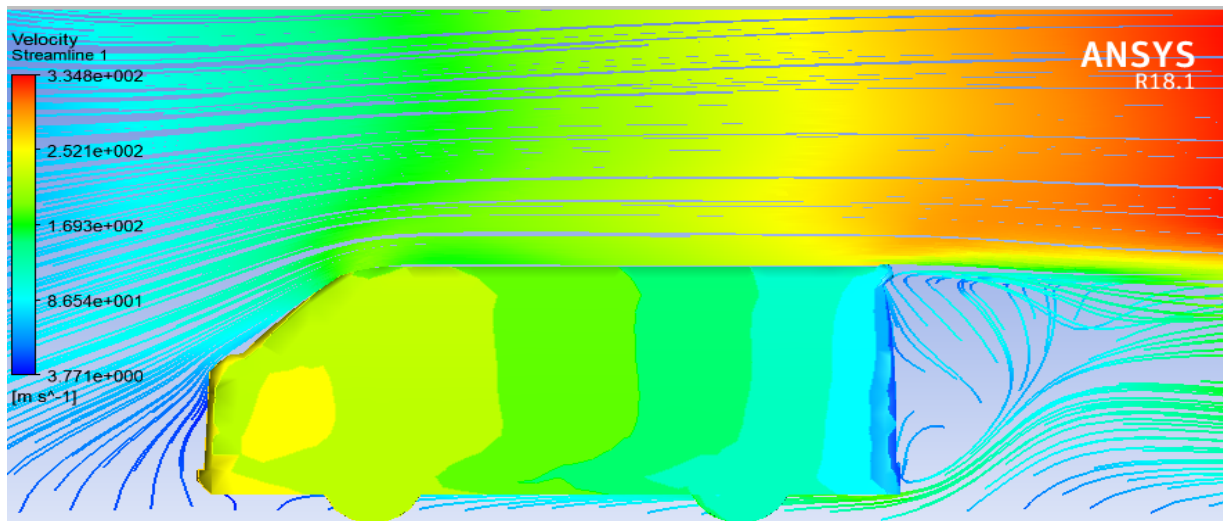


Figure 5.53. Velocity contour of the model three at 80 kmph.

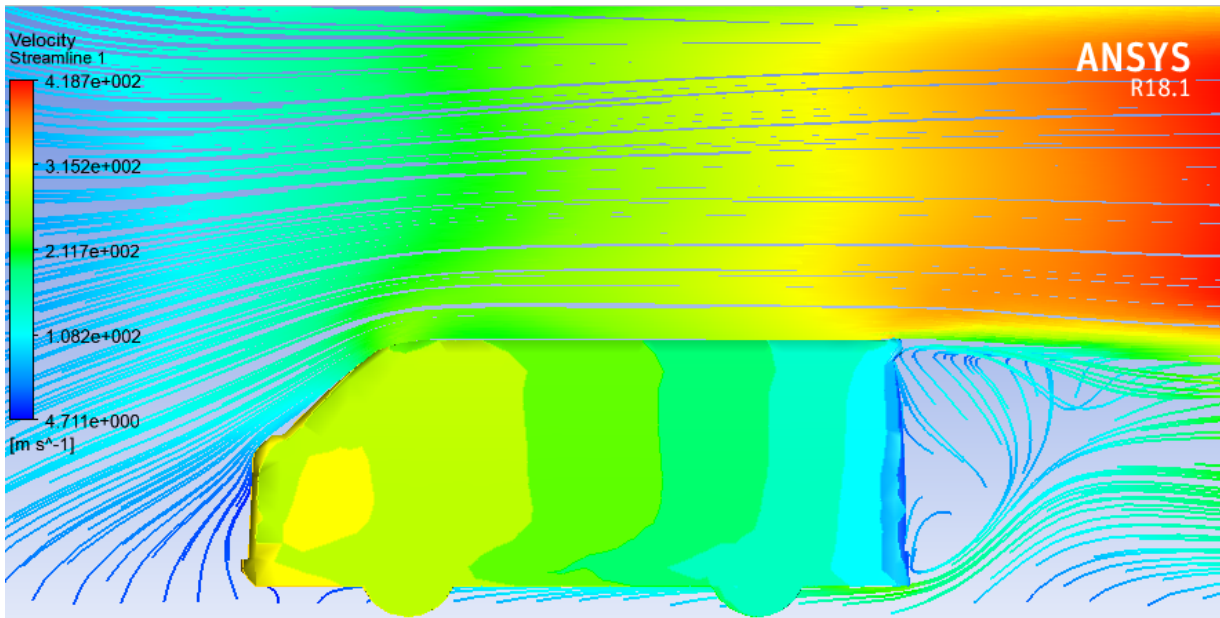


Figure 5.54. Velocity contour of the model three at 100 kmph.

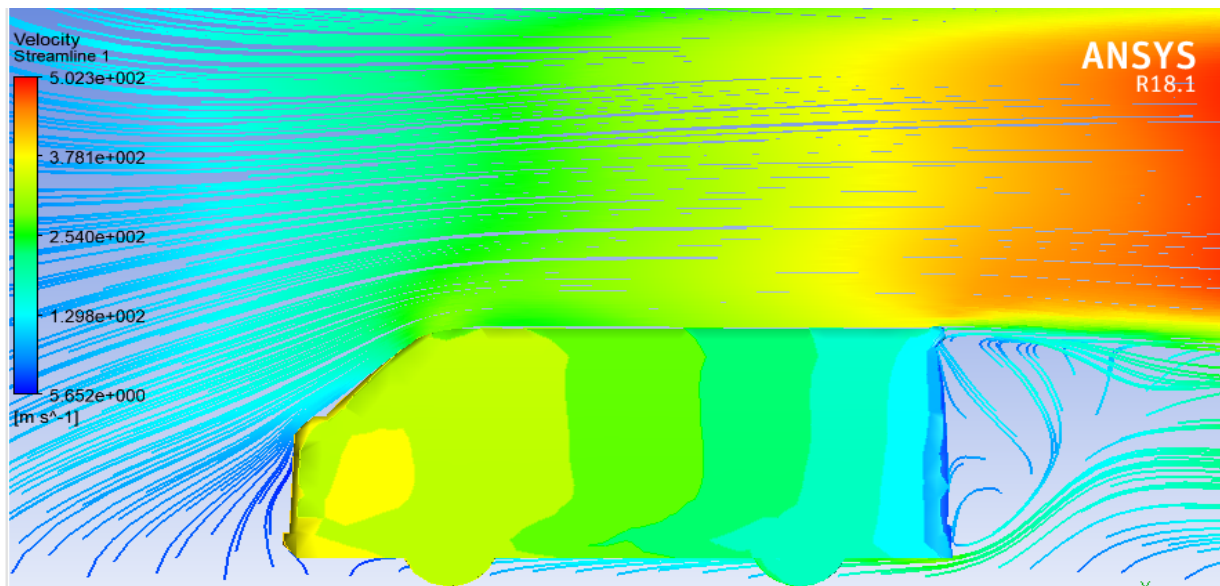


Figure 5.55. Velocity contour of the model three at 120 kmph.

The test procedure was the same as the baseline model. The flow separation at the rear roof end parts of the midi bus has delayed due to spoiler added on the rear roof end. It's clear that the wake region is increased and the turbulence nature became higher. Because of upward

adjustment of the spoiler a large area of turbulence created this increases the drag but it creates the necessary down force.

The pressure coefficient graph (Figure 5.56) shows the surface pressure distribution along the symmetry plane of model three and it indicates that the maximum value is at the front surface of the bus where the stagnation point is generated. It can be noted that the pressure on the front of the bus roof is same as the baseline model and then increases gradually and again it decreases rapidly at the rear roof end then finally increases. The pressure on the floor of the vehicle is lowest at the front of the bus then increased rapidly after the stagnation point. Finally increases gradually to rear of the bus. Therefore, it is important to analyze the high and low areas of pressure present on the bus and how these sections affected the airflow and coefficient of drag.

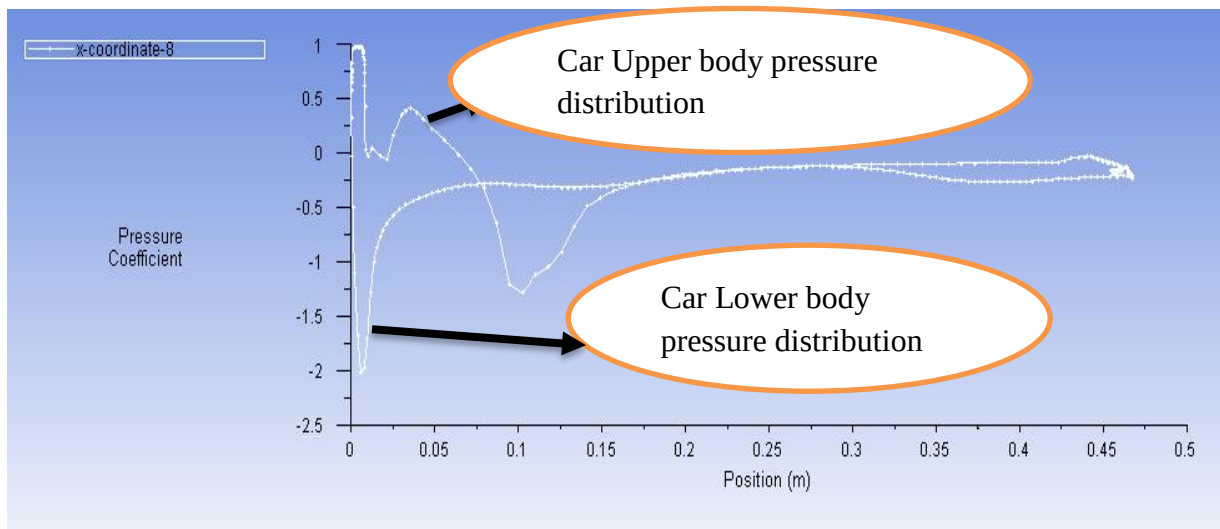


Figure 5. 56. Surface pressure distribution for model three.

5.1.5 Model Four Result (spoiler at 10 degree pitch angle)

The results of scaled residuals drag and lift coefficients, pressure, and velocity contour at speed of 80, 100, and 120 kmph for model four are depicted from Figure 5.57 to 5.69.

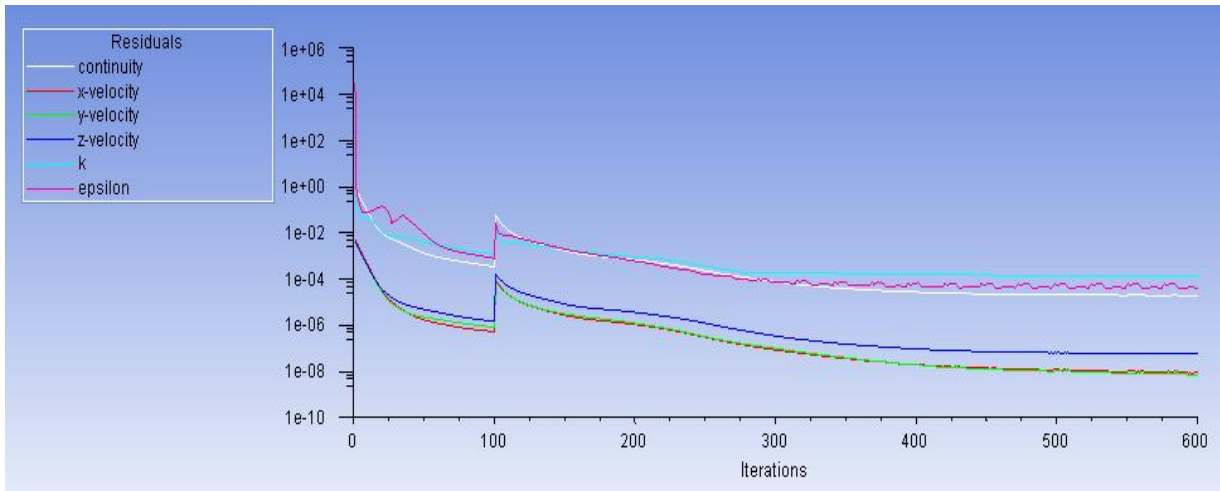


Figure 5.57. Scaled residuals of model four.

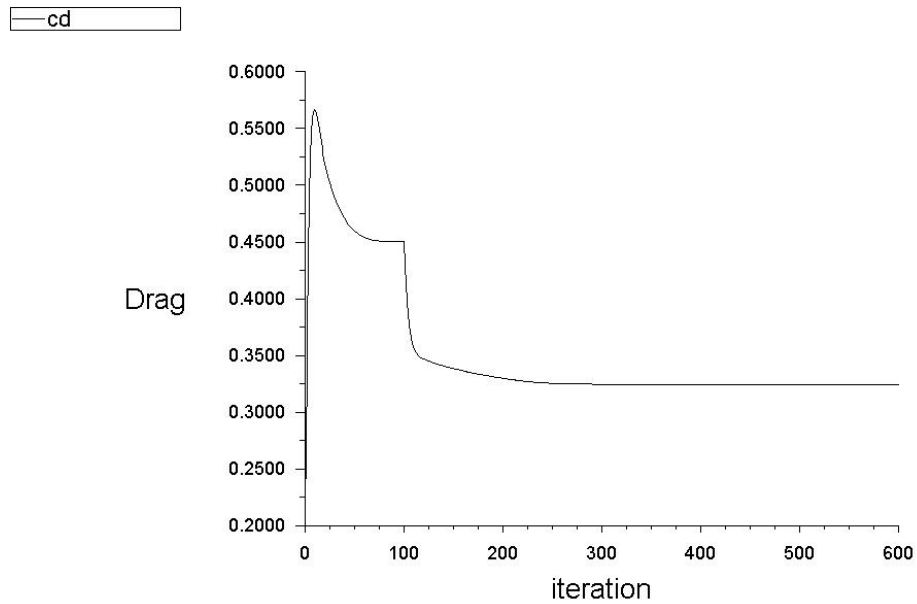


Figure 5.58. Drag coefficient of model four at 80 kmph.

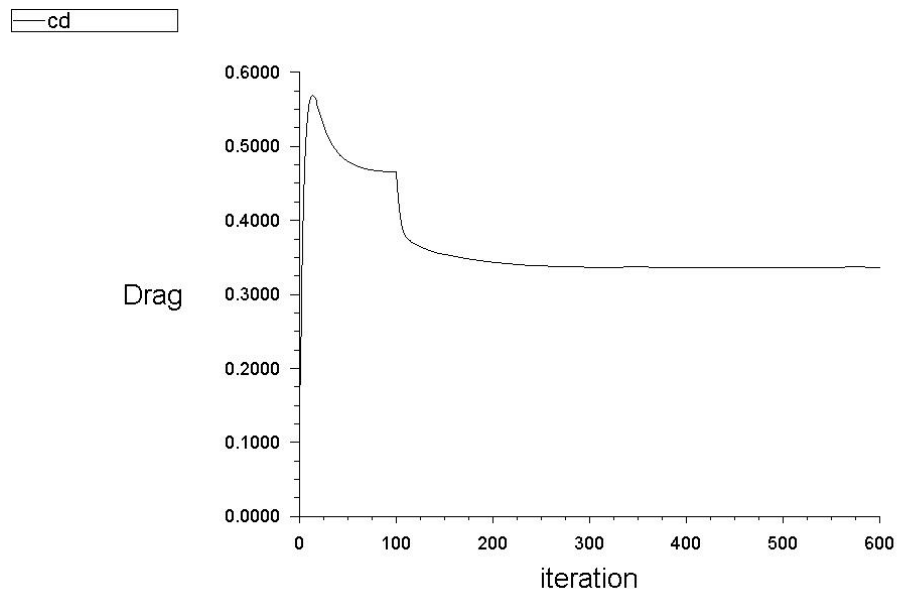


Figure 5.59. Drag coefficient of model four at 100 kmph.

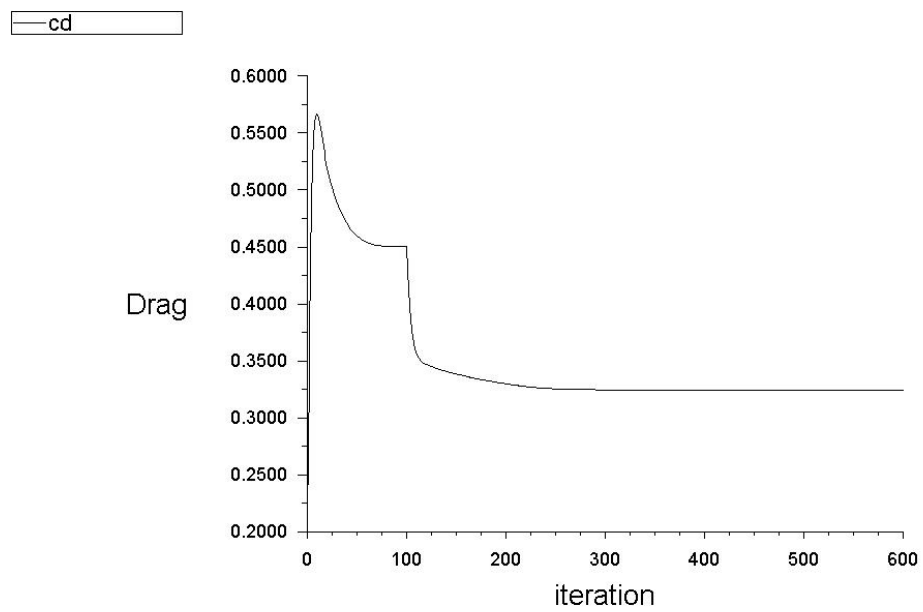


Figure 5.60. Drag coefficient of model four at 120 kmph.

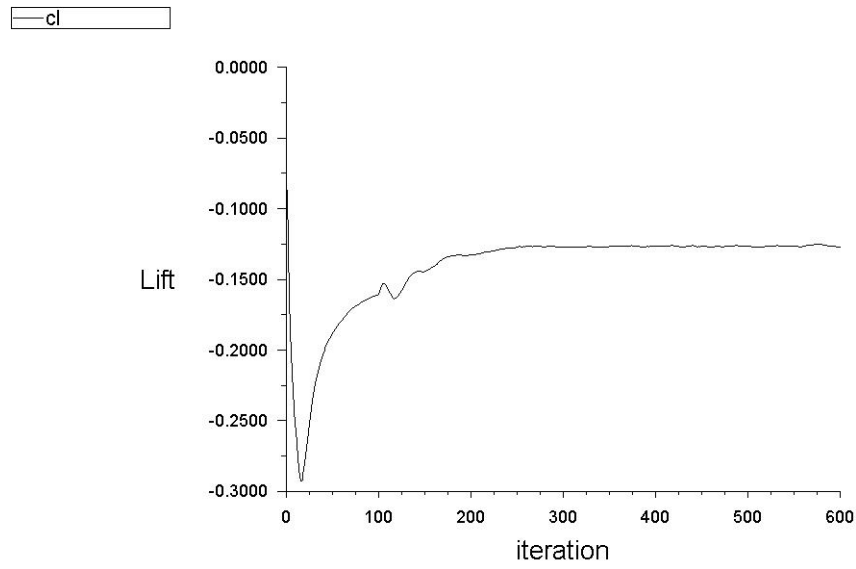


Figure 5.61. Lift coefficient of model four at 80 kmph.

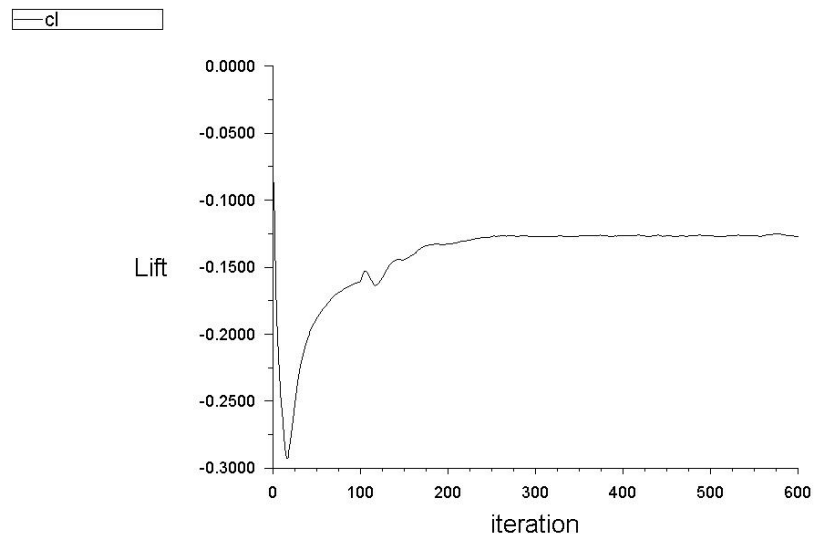


Figure 5.62. Lift coefficient of model four at 100 kmph.

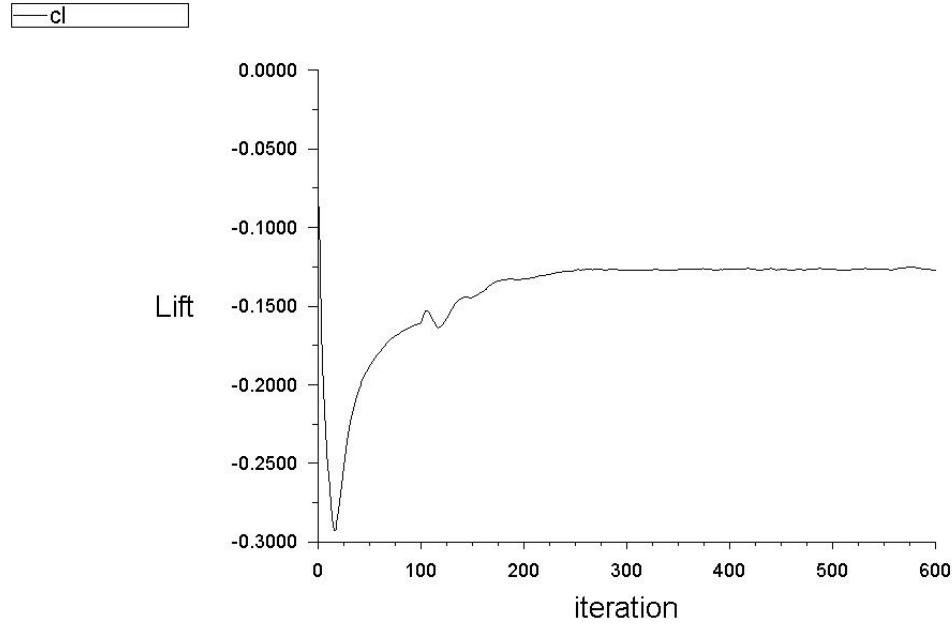


Figure 5.63. Lift coefficient of model four at 120 kmph.

All the drag coefficient graphs shown in Figure 5.58, 5.59 and 5.60 and lift coefficient graphs shown in Figure 5.61, 5.62 and 5.63 of model (4) at speed of 80, 100 and 120 kmph are summarized in Table 5.5.

Table 5. 5. Summary of drag and lift coefficients of Model four at different speeds

Model	Speed (kmph)	C_d	C_l
Model four	80	0.33474	-0.12705
	100	0.33642	-0.12742
	120	0.33914	-0.12781

The drag coefficient of model four (Table 5.5) is increased than both the baseline model (Table 5.1), model one and two (Table 5.2 and Table 5.3) and model three (Table 5.4) respectively. The drag coefficient has increased from 0.31380 (the average drag coefficient at 80, 100, and 120 kmph speeds) for the baseline model to 0.33680 (the average drag coefficient at 80, 100, and 120 kmph speeds) for model four, thus the average increase in C_d is 7.33%. This deviation

of the drag coefficient of the model four with respect to the baseline model will give a clear idea about how much the drag force is affected by increasing upward pitch angle of the spoiler. The lift coefficient of model four decreases with an increasing speed which results high downward force, which is very necessary for vehicle stability at higher speed. And it's clear that the lift coefficient is decreased from the baseline model which is very important for the vehicle stability and traction control.

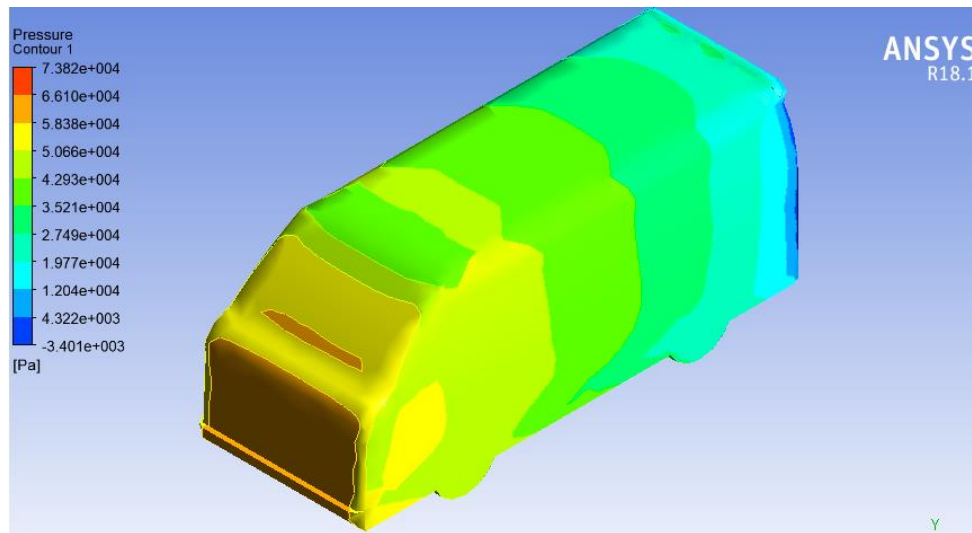


Figure 5.64. Pressure contour of the model four at 80 kmph.

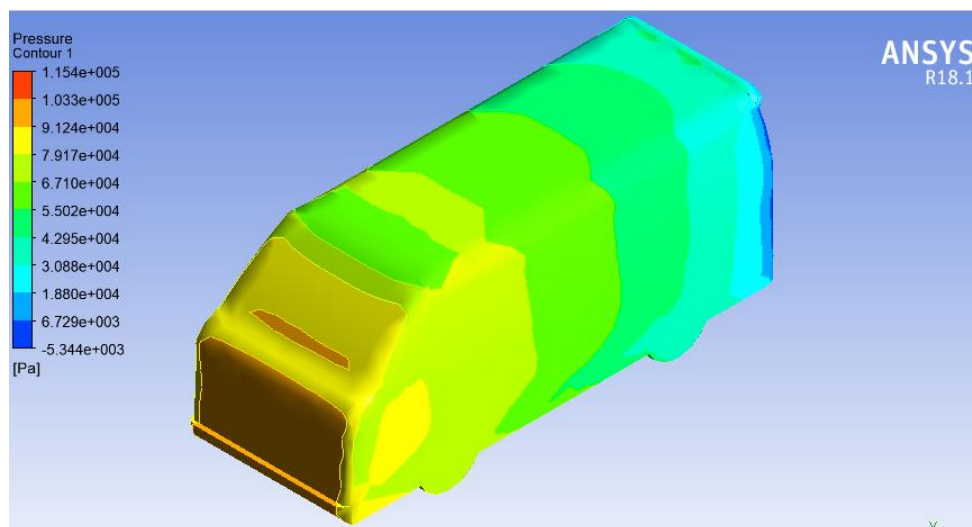


Figure 5.65. Pressure contour of the model four at 100 kmph.

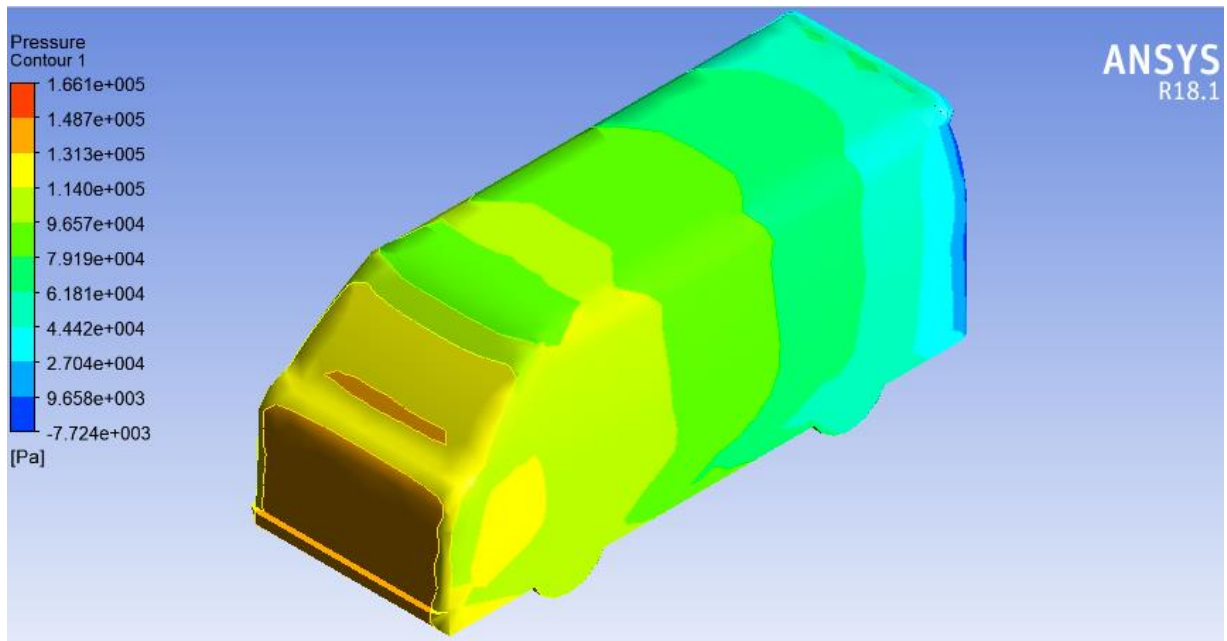


Figure 5.66. Pressure contour of the model four at 120 kmph.

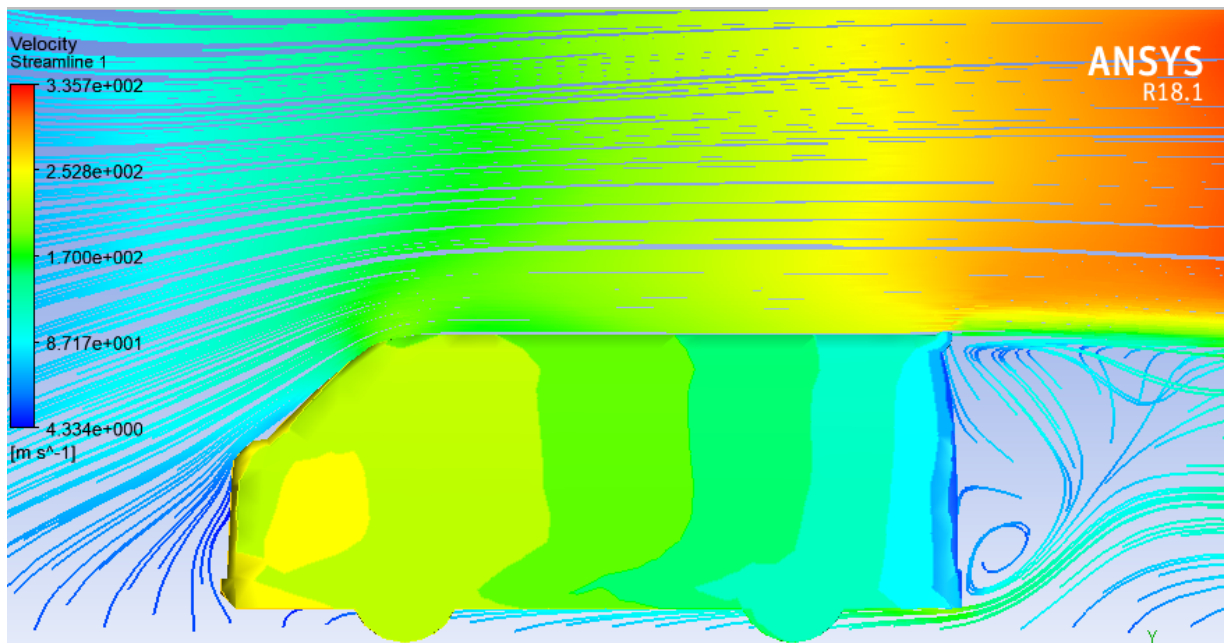


Figure 5.67. Velocity contour of the model four at 80 kmph.

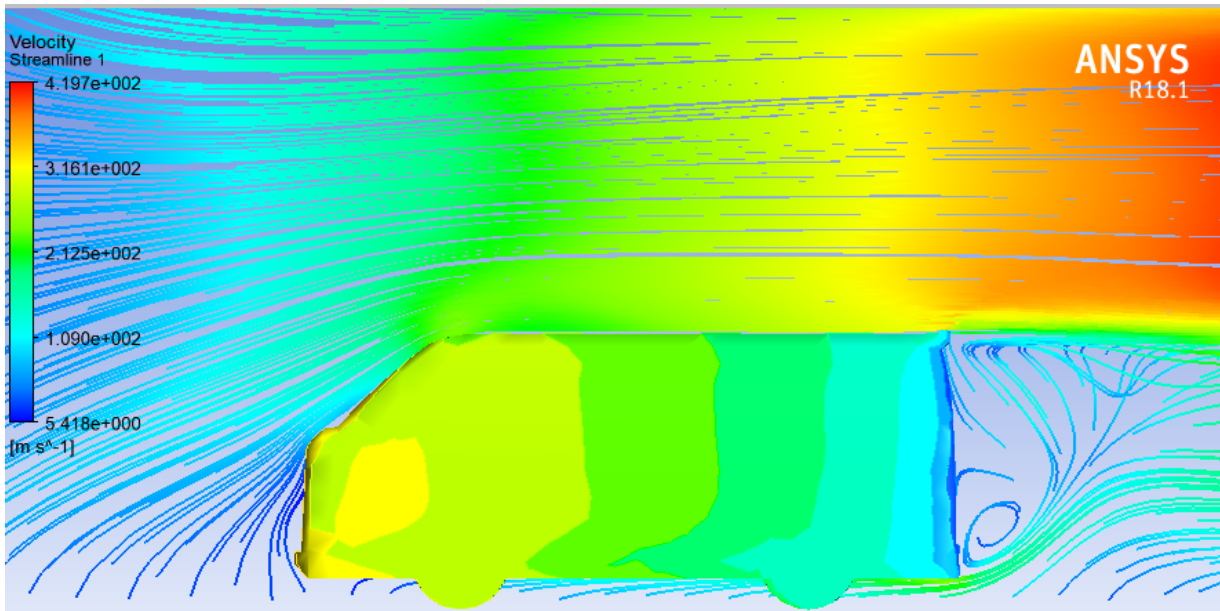


Figure 5.68. Velocity contour of the model four at 100 kmph.

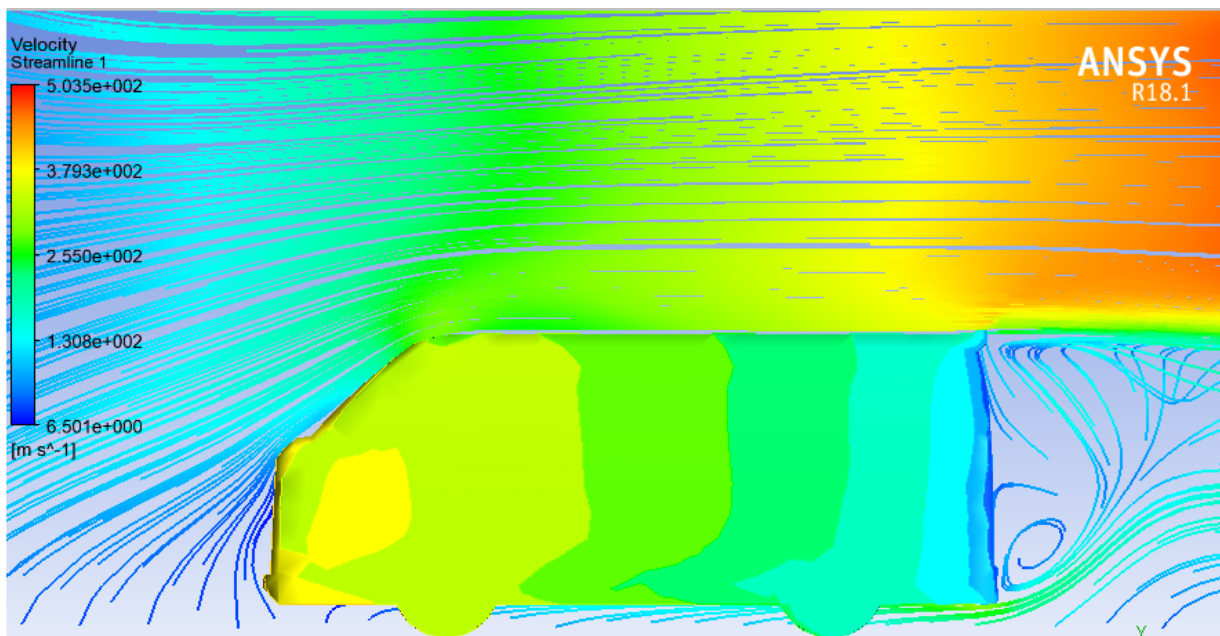


Figure 5.69. Velocity contour of the model four at 120 kmph.

The pressure and velocity distribution are similar to that of all the models but the magnitude is different it's higher at the fore end of the vehicle and lower at the rear end of the vehicle. The surface pressure distribution is similar to that of all the models.

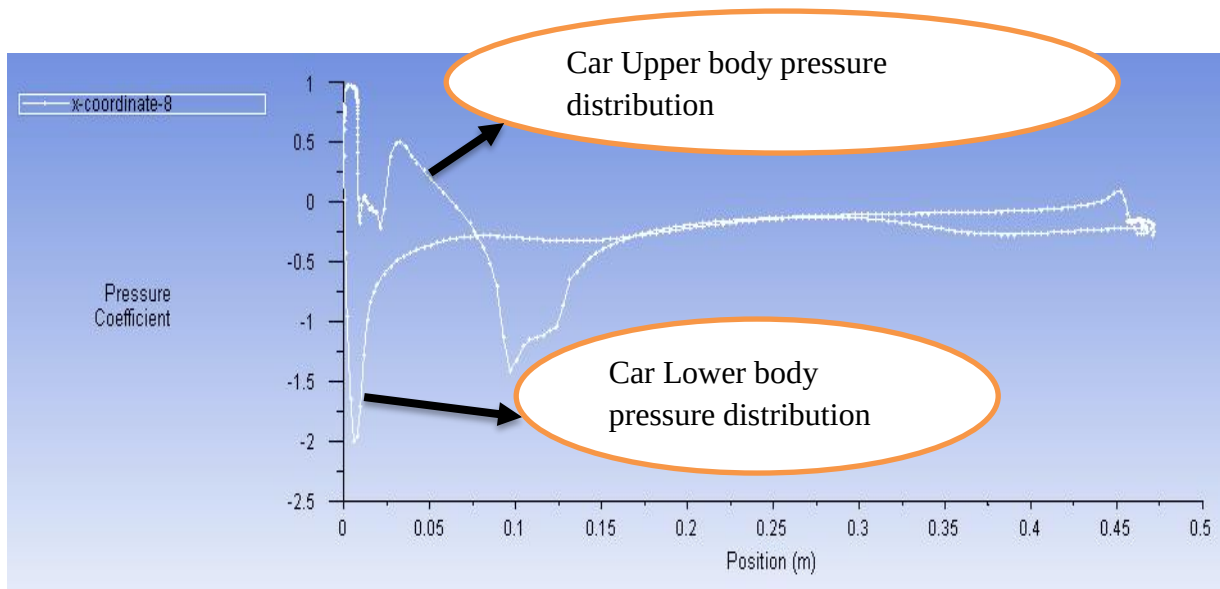


Figure 5. 70. Surface pressure distribution for model four.

The maximum value is at the front surface of bus where the stagnation point is generated. After the front of the bus roof surface the pressure decreases due to its streamlined shape and then increases gradually to rear roof of the bus. The pressure on the floor of the vehicle is minimum at the front of the bus then gradually increases to Rear of the bus Therefore, it is important to analyze the high and low areas of pressure present on the bus and how these sections affected the airflow and coefficient of drag. It's similar to the rest of all models but the magnitude of increment or decrement in pressure value is different.

5.2 Comparison between Models

In order to choose the best aerodynamic model of a Toyota HiAce Midi Bus with lowest drag and lift coefficients the comparison between models are necessary. Figure 5.81 and 5.82 shows the difference in drag and lift coefficient at speed of 80, 100 and 120 kmph between baseline model and all four models.

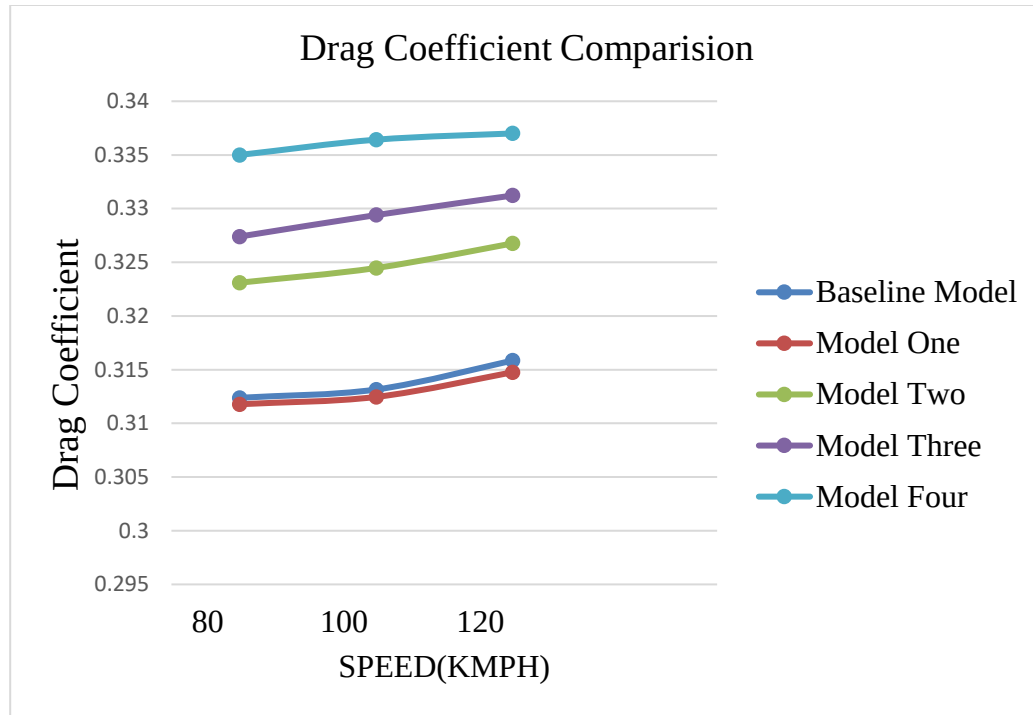


Figure 5. 71. Drag coefficient comparison of all models at different speed.

The drag coefficient of all models increases as their speed increases (Figure 5.71). However, the slope is different for each model. Both at low and high speeds model one has the lowest drag coefficient and model four has the highest drag coefficient. It's clear that the addition of a spoiler with a positive angle of attack can increase the drag coefficient but it will result in an improvement in the lift coefficient which is desirable for the vehicle. The resulting drag coefficient increment for model two, model three, and model four were approximately 3.5%, 4.95%, and 7.33%, respectively and model one is decrease in drag coefficient from baseline model by 0.255%.

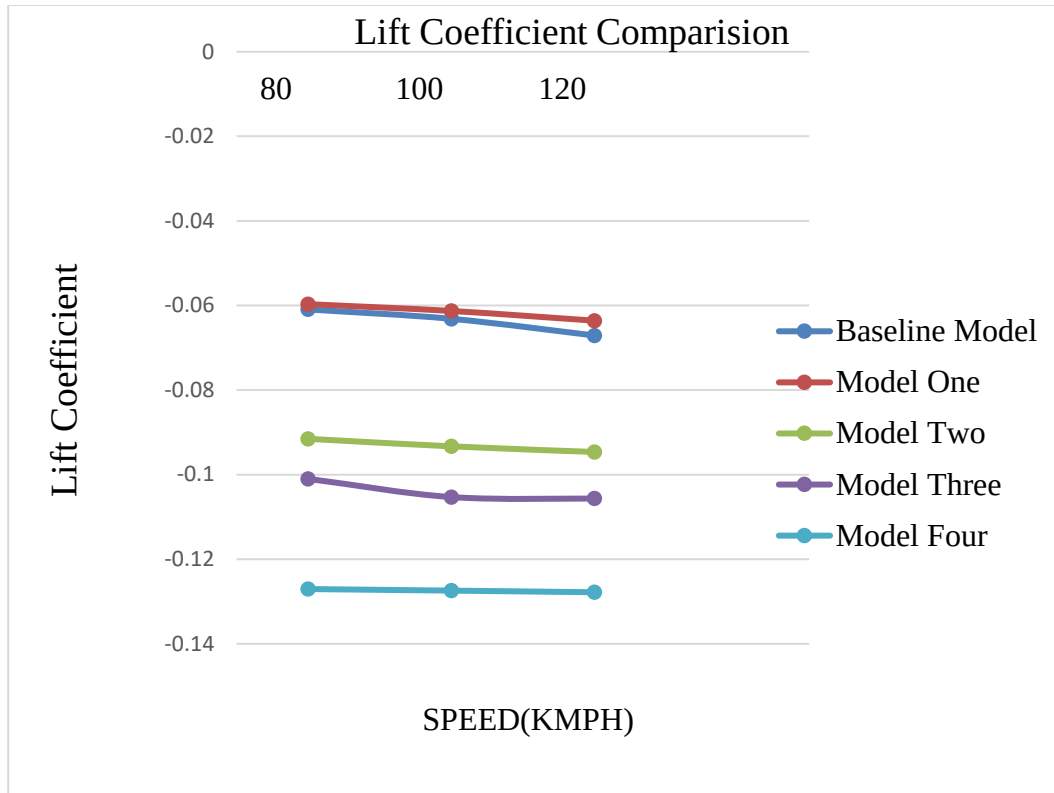


Figure 5. 72. Lift coefficient comparison of all models at different speed.

In order to improve the stability of a Toyota HiAce midi bus at higher speeds the downward force should be high. From the Figure 5.72, model four has the least lift coefficient which makes it the most stable of all model at high speeds. Least stable is model one with the highest lift coefficient. The lift coefficient of all models decreases with increasing speed. Model four as compared to other models that is the baseline model, model one, and model three achieved lowest lift coefficients, which increases the tires' capability to produce cornering force, improves braking performance and gives better traction.

Table 5.6 and Table 5.7. Summarizes the drag and lift coefficients of four Models with the baseline model in percentage difference.

Table 5. 6. Comparative coefficients of drag and lift for all models

Models	Speed (kmph)	Drag coefficient (C_d)	Average C_d Value	Percent of C_d increase from baseline model (%) (Average)
Baseline Model	80	0.31238	0.3138	-
	100	0.31315		
	120	0.31585		
Model One	80	0.31178	0.3130	0.255%(decrease)
	100	0.31246		
	120	0.31476		
Model two	80	0.32310	0.3248	3.5%
	100	0.32447		
	120	0.32676		
Model Three	80	0.32740	0.32934	4.95%
	100	0.32940		
	120	0.33122		
Model Four	80	0.33474	0.33677	7.32%
	100	0.33642		
	120	0.33914		

Table 5. 7.Percent of C_l reduction from baseline model (%)

(Average speed)

Models	Average C_l value	Present of C_l reduction from baseline model at (average speed)
Baseline Model	-0.063723	-
Model One	-0.061522	3.454%(increase)
Model Two	-0.093156	46.19%
Model Three	-0.10387	63%
Model Four	-0.12743	99.975%

Model two has a lower lift coefficient value than the baseline model and when we compare it from the baseline model the decrease in lift coefficient value can be 46.19% which is very important for the vehicle stability and traction control and model three as similar to model two it decreased in lift coefficient of 63%. Model four has a decrease in lift coefficient from the baseline model by 99.975% and again it's important for the vehicle stability and traction control as model two and model three.

5.3 Results and Discussions of Experimental Validation

The results of the CFD analysis must verified on vehicle aerodynamics studies. Some authors compare results with literature data. In this study the computational results verified by wind tunnel test. The determined C_d values for a baseline model both experimentally and numerically are shown in Table 5.9.

Table 5. 8. Cd difference (%) between experimental and CFD analysis

Speed (m/s)	C _d (Wind Tunnel Test)	C _d (CFD Simulation)	C _d (Difference %)
12	0.33512	0.29899	10.78
17	0.34015	0.30314	10.88
22	0.34823	0.31238	10.3
27.8	0.35947	0.31315	12.89

The drag coefficient of the baseline model was determined by both experimentally and numerically. However, there were a difference between the experimental result and CFD result. This difference is acceptable. It can be results from the uncertainty of the wind tunnel, calibration error rate or vibration in high speeds of wind tunnel. The comparison can be seen in Figure 5.82.

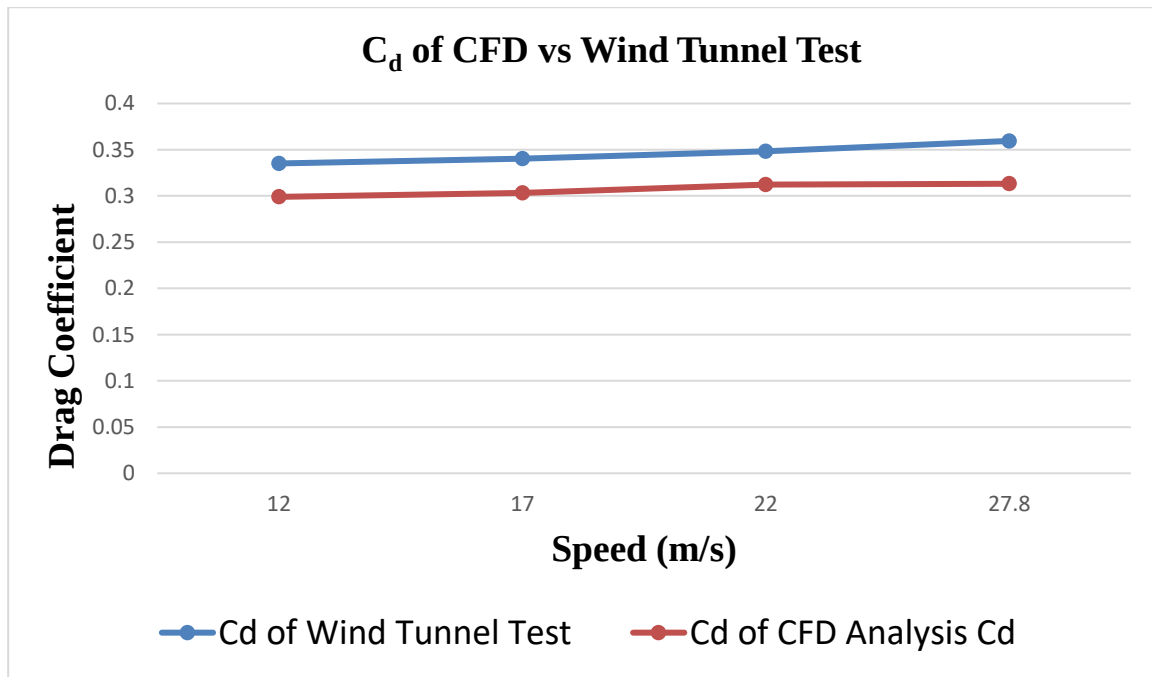


Figure 5.73. Comparison of C_d between experimental and CFD analysis.

6. CONCLUSIONS AND RECOMMENDATIONS

6.1 Conclusions

After the analysis of all models, the model which is having a spoiler adjusted at zero degree angle of attack reduced the amount of swirling air in the back of the midi bus and hence reduced the amount of drag coefficient compared to baseline model but the amount of lift coefficient increases which is not recommended for the vehicle. Model with spoiler adjusted at 0 degree with a gap in it and adjusted at 5 degrees and 10 degrees reduces the lift coefficients and it's the best model which is having aerodynamic improvement on the existing model. It was found that the large circulation region was formed in the back of a Toyota Hiace midi bus baseline model and it created large pressure difference between the front and the rear of the bus, causing high drag in order to improve the aerodynamic performance of the existing bus three models with a spoiler at different angle of attack and different shape were prepared.

The main findings are:

- The results show that there is a possibility of improving the aerodynamic performance of midi bus by adding a spoiler at the rear end of the vehicle body. These improvements are helpful in reducing the coefficient of lift. C_l which improved vehicle stability at higher speed.
- It was found that the least drag force was acting on model one. The average drag coefficient of the baseline model, model one, model two, model three and model four were 0.3138, 0.313, 0.3248, 0.32934 and 0.33677, respectively. Due to the addition of a spoiler at zero degree angle of attack on the baseline model, the coefficient of drag reduced by model one were approximately 0.255% but the amount of lift coefficient increases by an amount of 3.454%. Model two which is a spoiler having a gap with the rear end of the vehicle has an increase in drag coefficient since the swirling motion at the rear end of the vehicle increases and creates a highly negative pressure region which we call a wake but the amount of lifting force reduces which increases the tires' capability to produce cornering force, improves braking performance and gives better traction. The resulting average drag coefficient for model two can be 0.3248 which is 3.5% increase than the baseline model but a higher improvement in lift coefficient than the baseline model can be seen that is 46.19% reduction in lift coefficient. And lastly model three and model four also improves the lift coefficient by a percentage of 63 % and 99.975%

from the baseline model. When comparing the best model which will provide the better improvement in aerodynamic performance of the vehicle model two that is spoiler adjusted at 0 degree and having a gap of 5cm results in an improvement in lift coefficient with less increase in drag coefficient was the best model in the improvement of aerodynamic performance of the Toyota midi bus.

- This work uniquely proves that possibility of improvement in aerodynamics performance of the Toyota HiAce midi bus model using a spoiler at the rear end of the vehicle.

6.2 Recommendations

- Many automotive industries and bus bodybuilding companies have to check the aerodynamics performance of their vehicles body shape and have to give more emphasis on the aerodynamics of vehicles as there is a significant change in their lifting coefficient as demonstrated in this study.
- To obtain more accurate results it's recommended that to use a powerful computer with high processing speed and capacity, which improves the meshing quality and reduces processing time.

6.3 Recommendations for Future Work

The following recommendations can be taken into consideration as future work.

- The turbulence model which is used in this study can be tested in the determination of aerodynamic characteristics of different vehicle body shape types such as trucks, buses, and trains running on Ethiopian roads.
- Experimental studies, using a wind tunnel, could be run on the baseline model found in this study to compare the values found from numerical analysis for validity.
- Using add-on devices and different modifications that reduce the aerodynamic drag present on the bus. Also, design and test active drag reduction device to reduce the drag force on the vehicle.

REFERENCES

- Abedin, &. Z. (2019). Review on aerodynamic drag reduction of vehicles. International journal of engineering materials and manufacturing, 4(1), 1-14.
- Aharwal, p. M. (2018). A review on a selection of turbulence model for cfd analysis of airflow within cold storage. 2nd international conference on advances in mechanical engineering (icame), iop publishing.
- Aharwal, p. M. (2018). A review on a selection of turbulence model for cfd analysis of airflow within cold storage. 2nd international conference on advances in mechanical engineering (icame), iop publishing.
- Ahmed, g. :. (1984). Some salient features of the time-averaged ground vehicle wake. Sae, technical paper 840300.
- Ashwini, d. P. (2016). Aerodynamics study of automobile car ahmed body using cfd simulation to predict the drag coefficient and down forces. International engineering research journal (ierj), special issue page 1140-1145, issn 2395-1621.
- Asrar, a. :. (2014). Passive drag reduction of square back road vehicles. Journal of wind engineering and industrial aerodynamics, 134, 30-43.
- Basara, x. H. (2013). Study of active flow control for a simplified vehicle model using the pans method. International journal of heat and fluid flow, 42, 139-150.
- Bellman, m. :. (n.d.). Reducing the energy consumption of ground vehicles by active flow control. International conference on energy sustainability, pp 785793, american society of mechanical engineers.
- Bruneau, c.-h. (2010). Coupling active and passive techniques to control the flow past the square back ahmed body. Computers & fluids, 39, 1875-1892.
- Bruneau, c.-h. (august 2017). Flow analysis of square-back simpli. International journal of heat and fluid flow, 1-23.
- C., w. D. (2003). Turbulence modeling for cfd. La canada, california: dcw industries.
- Cable, m. :. (2009). An evaluation of turbulence models for the numerical study of forced and natural convective flow in atria. Unpublished thesis submitted to queen's university kingston, ontario, canada p. 148.

- Cakir, m. (2012). Cfd study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle. Mechanical engineering masters theses. Paper 1.
- Capone, a. (2019). Investigation on the effect of horizontal and vertical deflectors on the near-wake of a square-back car model. *Journal of wind engineering & industrial aerodynamics*, 185, 57-64.
- Chorage, c. R. (2020). Modification in commercial bus model to overcome aerodynamic drag effect by using cfd analysis. *Results in engineering*, 6, 100091.
- Devaradjane, p. N. (2015). Analysis of drag and lift performance in sedan car model using cfd. *Journal of chemical and pharmaceutical sciences (jchps)*, special issue 7, issn: 0974-2115 pp:, 429-435.
- Filippone, z. M.-k. (2010). Fuel savings on a heavy vehicle via aerodynamic drag reduction. *Transportation research part d* 15, 275-284.
- Gaylard, a. (2017). Simulation of rear surface contamination for a simple bluff body. *Journal of wind engineering and industrial aerodynamics*, 165, 13-22.
- Grandemange, m. (2015). A study of wake effects on the drag of ahmed's square back model at the industrial scale. *Journal of wind engineering and industrial aerodynamics*, 145, 282-291.
- H., t. . (2018). Computational fluid dynamics, science of the future. *International journal of research and engineering*, 5, 430-433.
- Imine, o. M. (2016). Design optimization of the aerodynamic passive flow control on naca 4415 airfoil using vortex generators. *European journal of mechanics b/fluids*, 56, 82-96.
- J., s. T. (1995). A new-eddy viscosity model for high reynolds number turbulent flows - model development and validation. *Computers fluids*, 24(3), 227-238.
- Jamshed-9780128015674. (2018). Chapter 1 introduction to cfd.
- Khalighi, b. (2012). Simulations and measurements of the unsteady aerodynamic flow around a simplified square-back road vehicle with drag reduction devices. *Fluids engineering division*, 1, 1-8.
- Kim, j. J. (2017). Salient drag reduction of a heavy vehicle using modified cab-roof fairings. *Journal of wind engineering and industrial aerodynamics*, 164, 138-151.

- Kim, j. J. (2019). Considerable drag reduction and fuel saving of a tractor–trailer using additive aerodynamic devices. *Journal of wind engineering & industrial aerodynamics*, 191, 54-62.
- Krajnovic, s., & fernandes, j. (2011). Numerical simulation of the flow around a simplified vehicle model with active flow control. *International journal of heat and fluid flow*, 32, 192-200.
- Kyei, s. M. (2014). Composite car rear spoiler. Degree thesis plastics technology.
- Lajos, t. (2002). Basics of vehicle aerodynamics. University of rome „la sapienza”: lajos, tamás.
- Len brownlie : yuki aihara : jorge carbo : jr., e. (2016). The use of vortex generators to reduce the aerodynamic drag of athletic apparel. 11th conference of the international sports engineering association, isea, 147, 20-25.
- Li, r. .. (2018). Aerodynamic drag reduction of a square-back car model using linear genetic programming and physic-based control. <https://tel.archives-ouvertes.fr/tel-01685306>.
- Littlewood.r. (2011). An investigation into the wake structure of square back vehicles and the effect of structure modification on resultant vehicle forces. *International journal of engines*, 4(2), 2629-2637.
- Menter, f. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *Aiaa journal*, 32(8), 1598-1605.
- Mohd, m. N. (2014). Review of research on vehicles aerodynamic drag reduction methods. *International journal of mechanical & mechatronics engineering*, 14.
- Morrison, r. B. (2018). Modelling and feedback control of vortex shedding for drag reduction of a turbulent bluff body wake. *International journal of heat and fluid flow*, 71, 127-136.
- Perry, a.-k. (2016). Influence of short rear end tapers on the wake of a simplified square-back vehicle: wake topology and rear drag. *Exp fluids*, 57-169.
- R., c. K. (n.d.). ‘full-scale wind-tunnel tests of production and prototype’, second-generation aerodynamic drag reducing devices for tractor-trailers’. Sae, paper no.2006-01-3456, usa.
- Raina, a. (2017). Numerical investigation of flow around a 3d bluffbody using deflector plate. *International journal of mechanical sciences*, 131-132, 701-711.

- Rajkumar, s. K. (2017). Aerodynamic study of state transport bus using computational fluid dynamics. School of mechanical engineering, vit university, iop conf. Series: materials science and engineering, 062052, 263.
- Snegirjov, a. J. (2009). High-performance computing in technical physics. Numerical simulation of turbulent flows. A training manual. Publishing house of st. Petersburg state polytechnical university, st. Petersburg, russia, pp: 143.
- Lealem.T, l. (2018). Study and aerodynamic drag reduction on locally built bus body using cfd analysis: a case study on bishoftu automotive industry.
- Wanghanfeng.(2016). Aerodynamic drag reduction of an ahmed body based on deflectors. Journal of wind engineering and industrial aerodynamics, 148, 34-44.
- Waqarasrar, a. A. (2014). Passive drag reduction of square back road vehicles. Journal of wind engineering and industrial aerodynamics, 134, 30-43.
- Wilcox, d. C. (2008). Formulation of the k-omega turbulence model revisited. American institute of aeronautics and astronautics 11, 2823.
- Engedayehu. W. (2019). Analysis on aerodynamic drag reduction of pickup vehicles using rear spoilers. Not published.
- Yakhot v. : orszag, s. A. (1986). Renormalization group analysis of turbulence: basic theory. Journal of scientific computing, 1(1), 1-51.
- Zhao, x. H. (2011). Influence of different diffuser angle on sedan's aerodynamic characteristics. International conference on physics science and technology (icpst 2011), 22, 239-245.

APPENDIX

Appendix A. SolidWorks Modelling

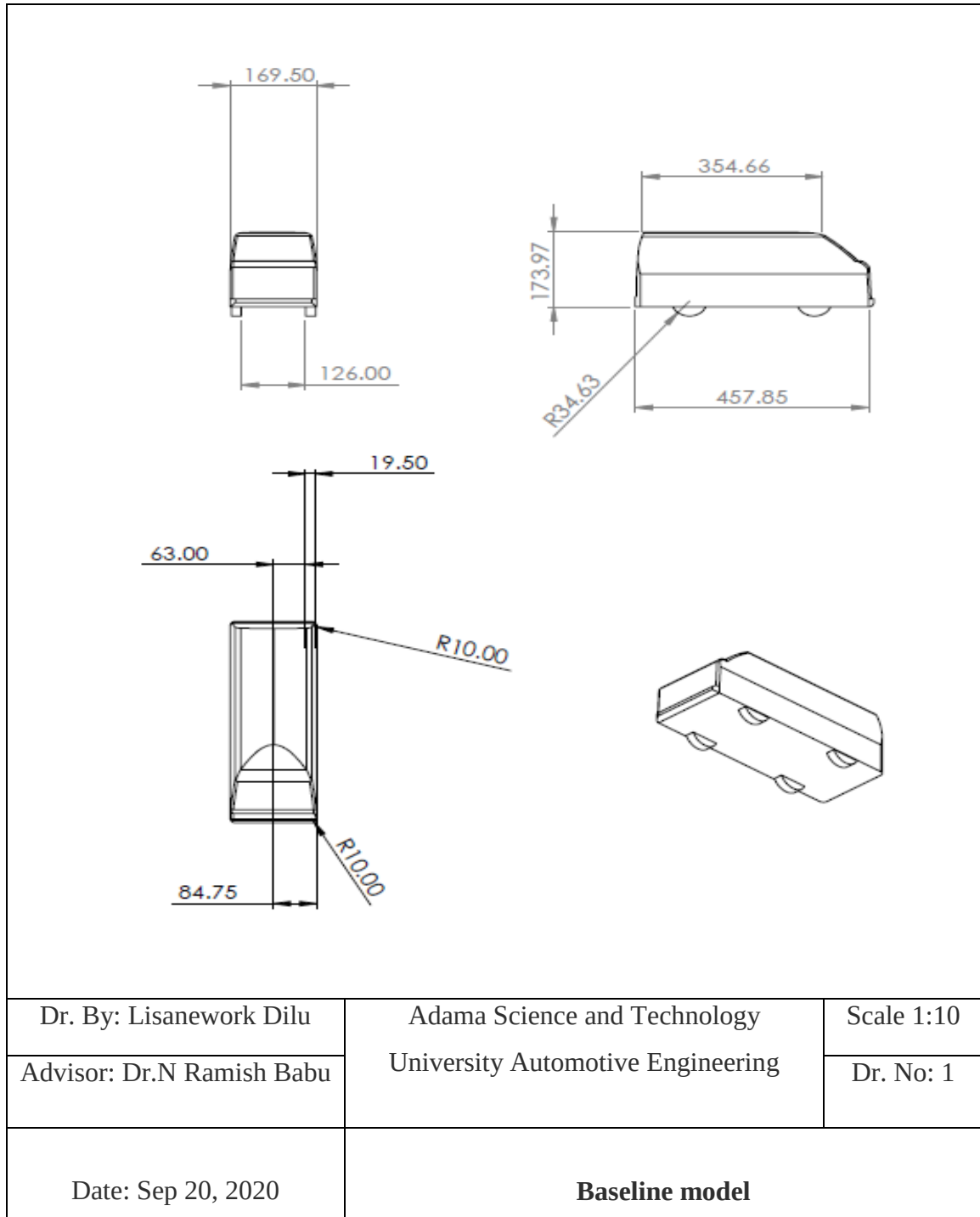


Figure A- 1. Baseline Model Multiview.

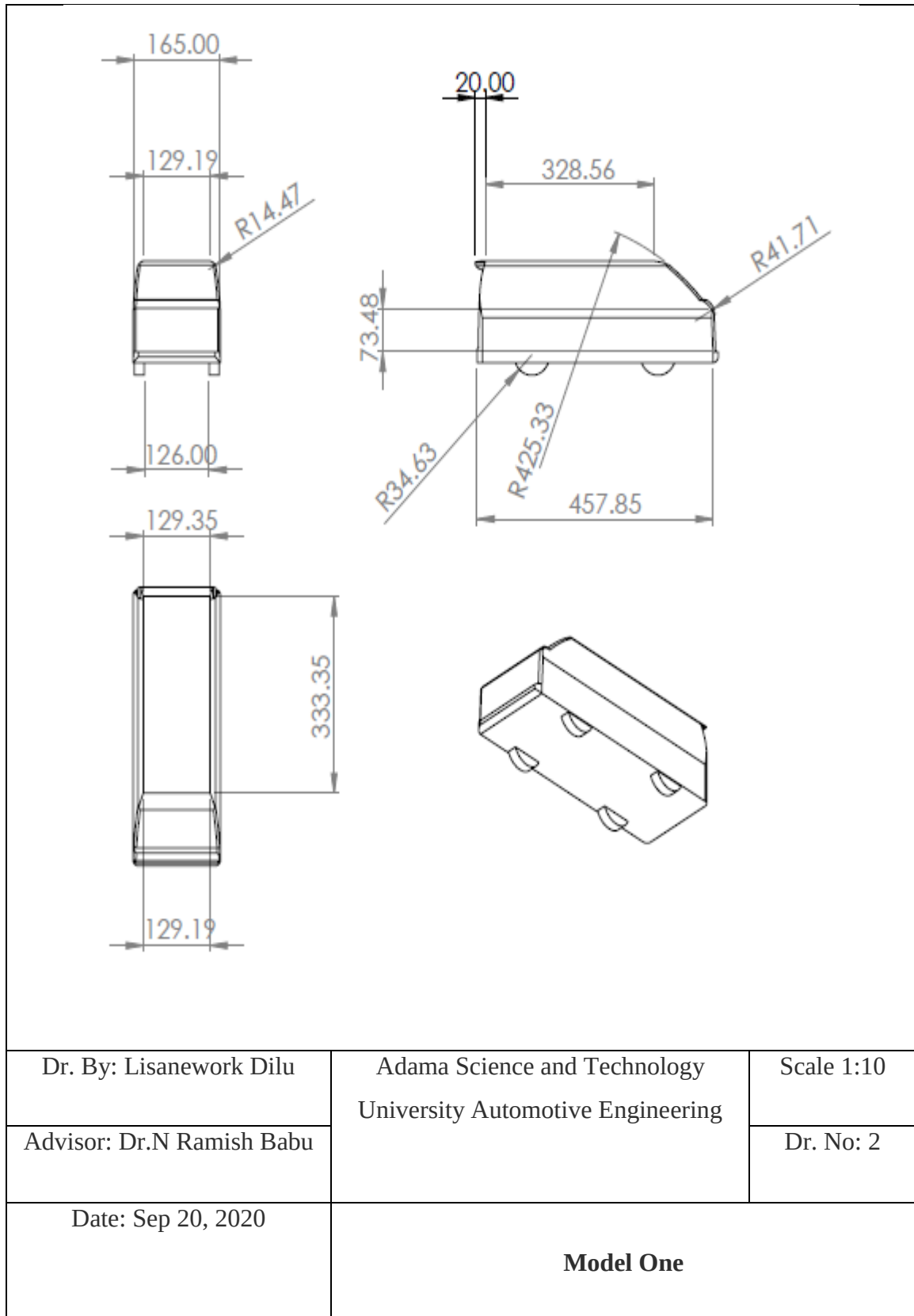


Figure A- 2. Model one Multiview.

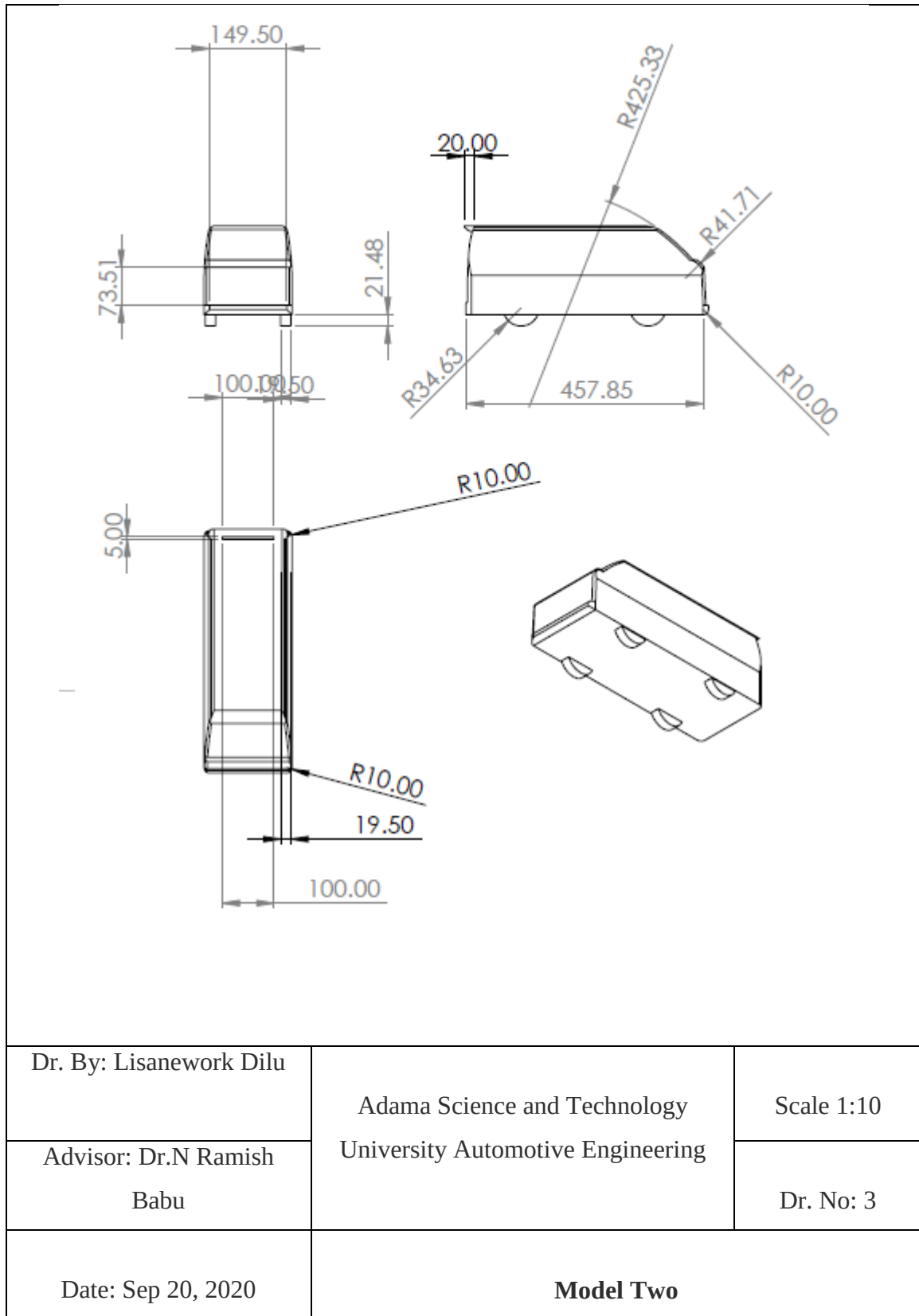


Figure A- 3. Model Two Multiview.

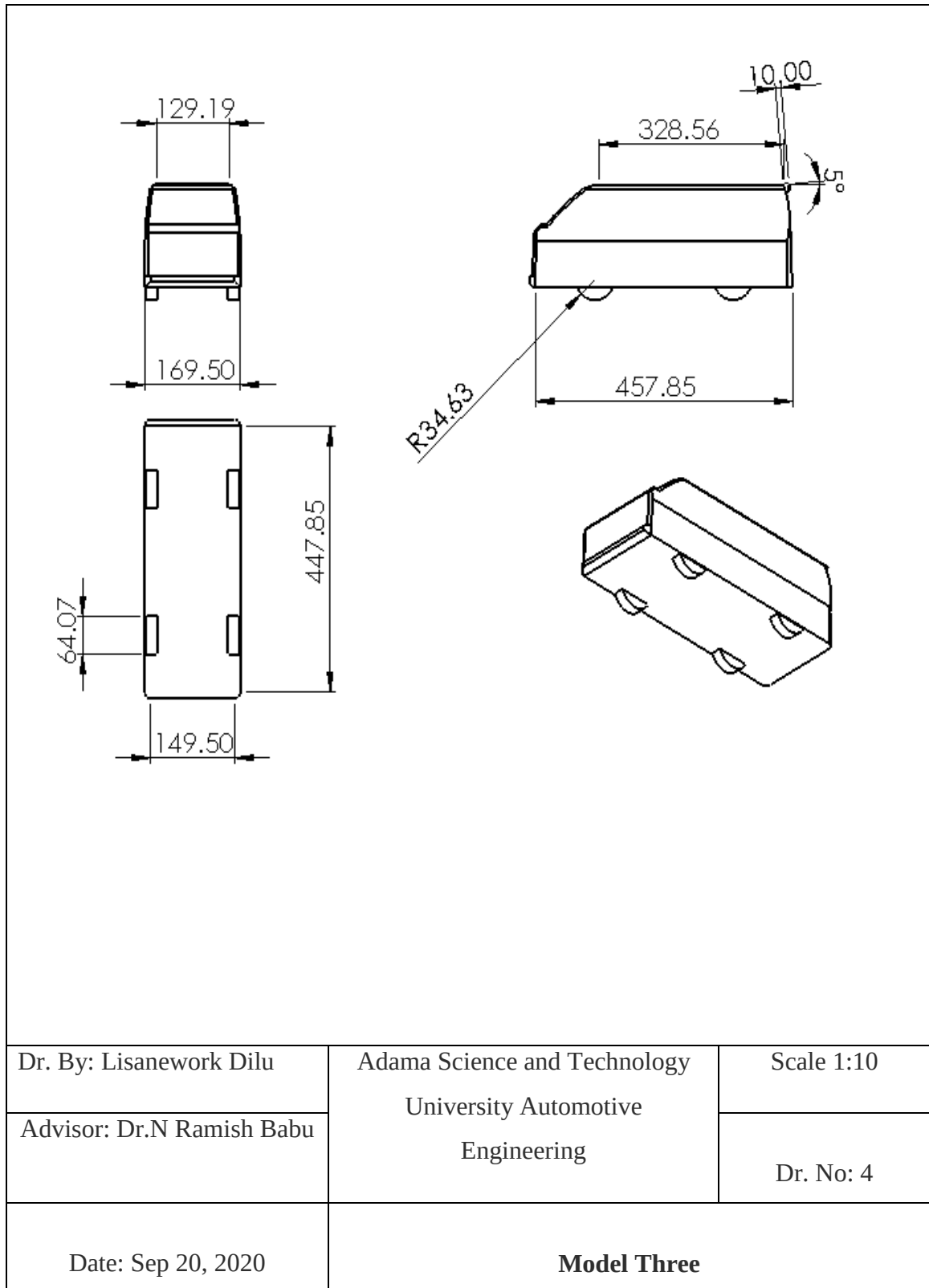


Figure A- 4. Model Three Multiview.

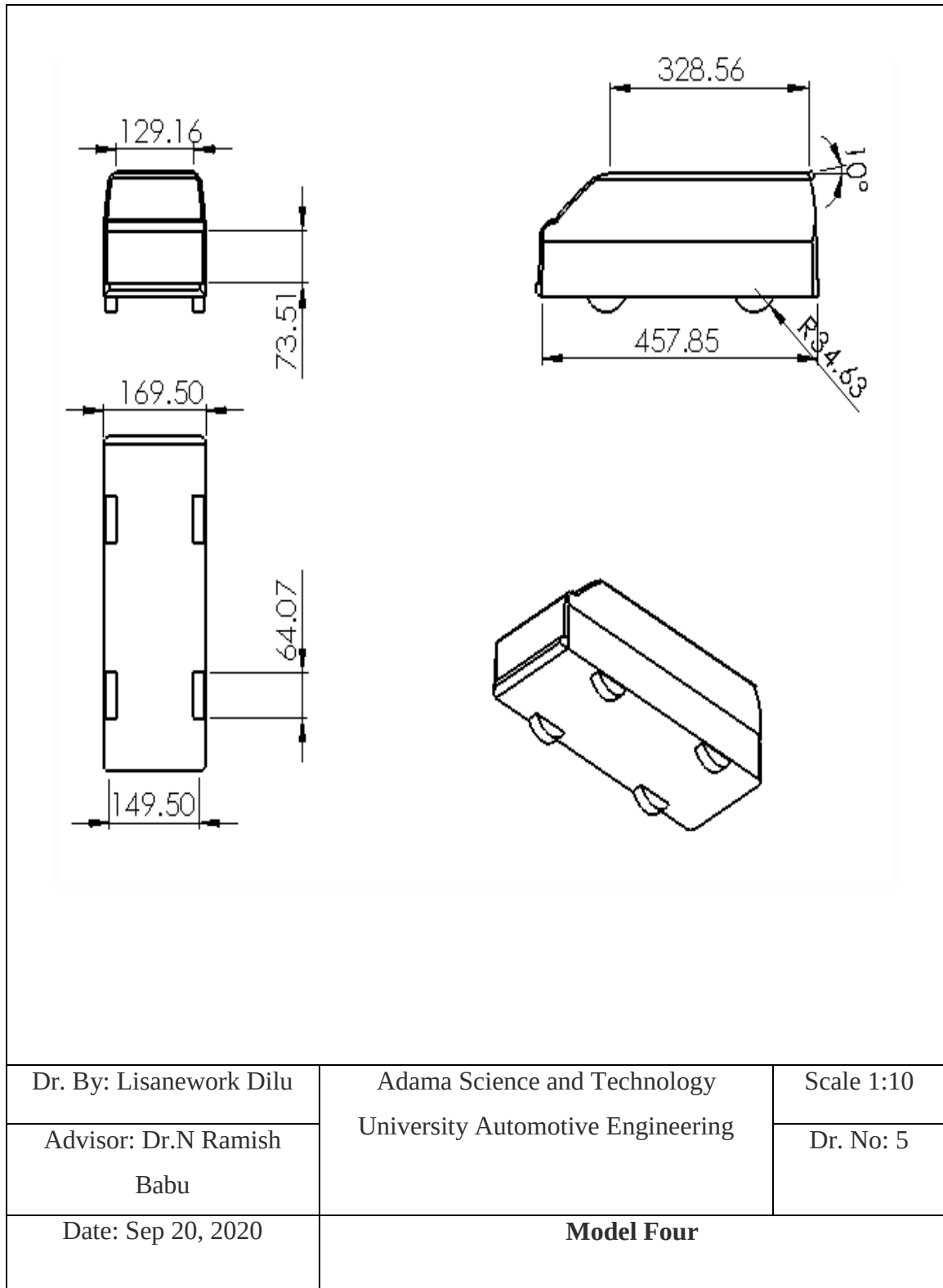


Figure A- 5. Model Four Multiview.

Appendix B. Models inside the Enclosure and After Meshing

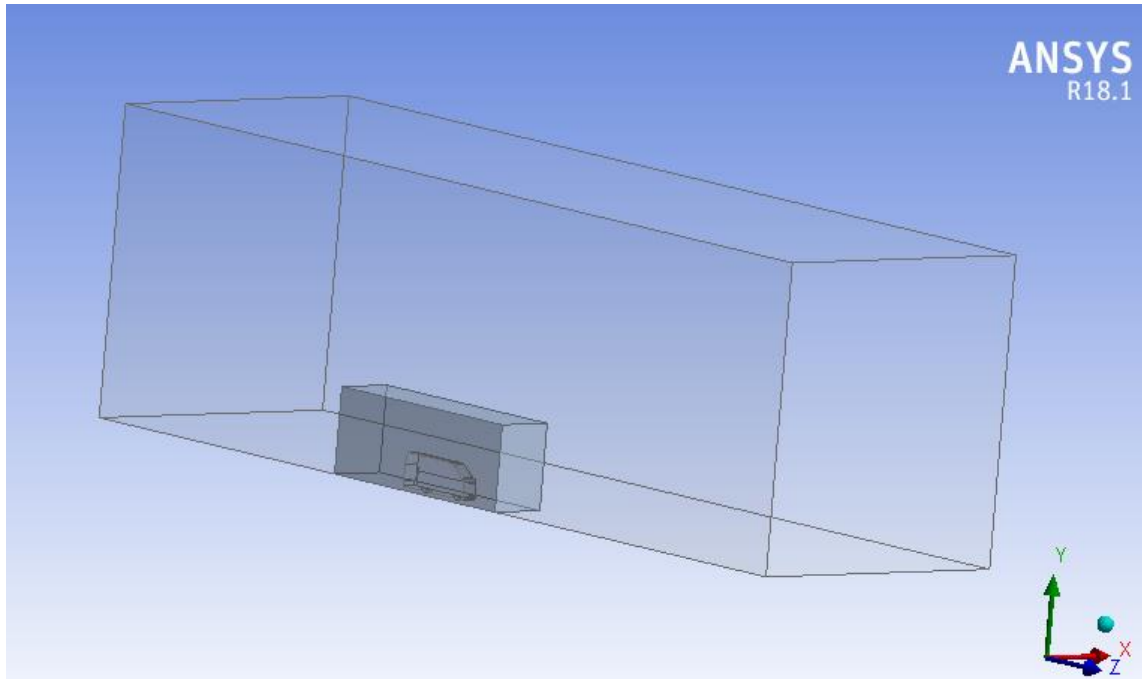


Figure B- 1. Baseline model inside the enclosure box.

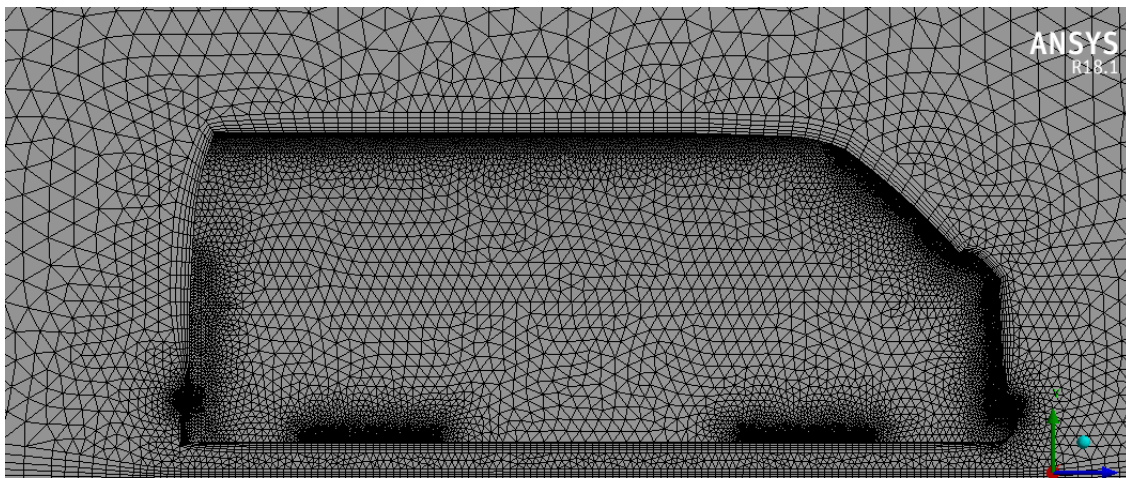


Figure B- 2. Baseline model after meshing.

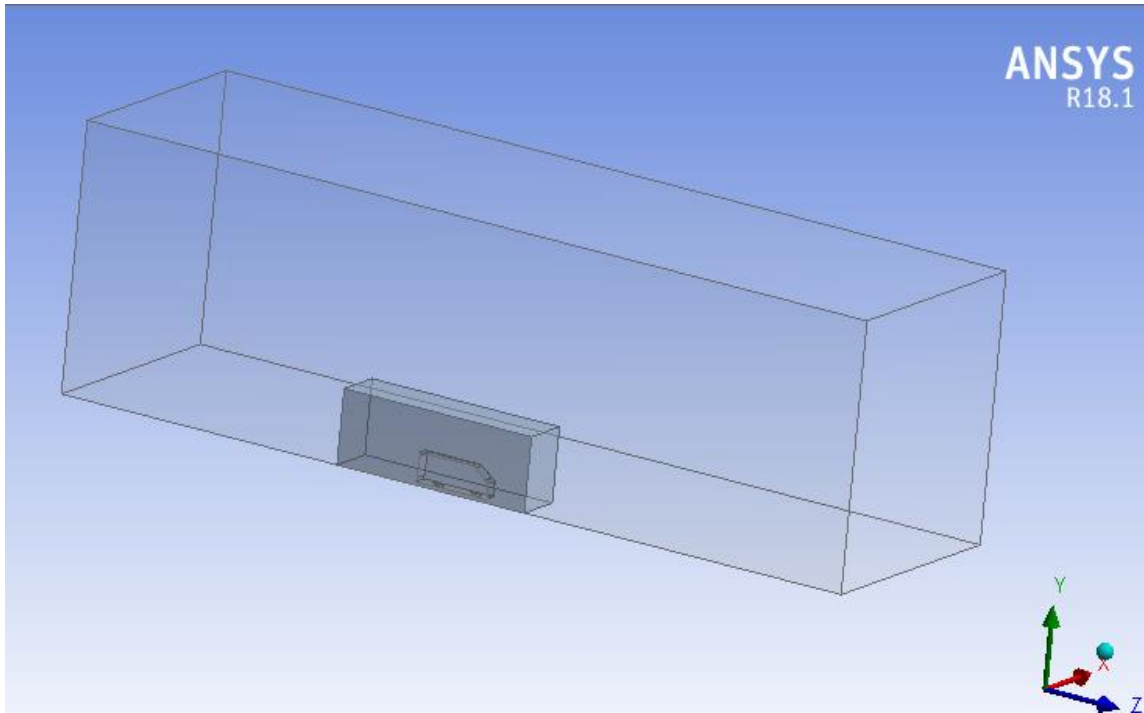


Figure B- 3. Model one inside the enclosure box.

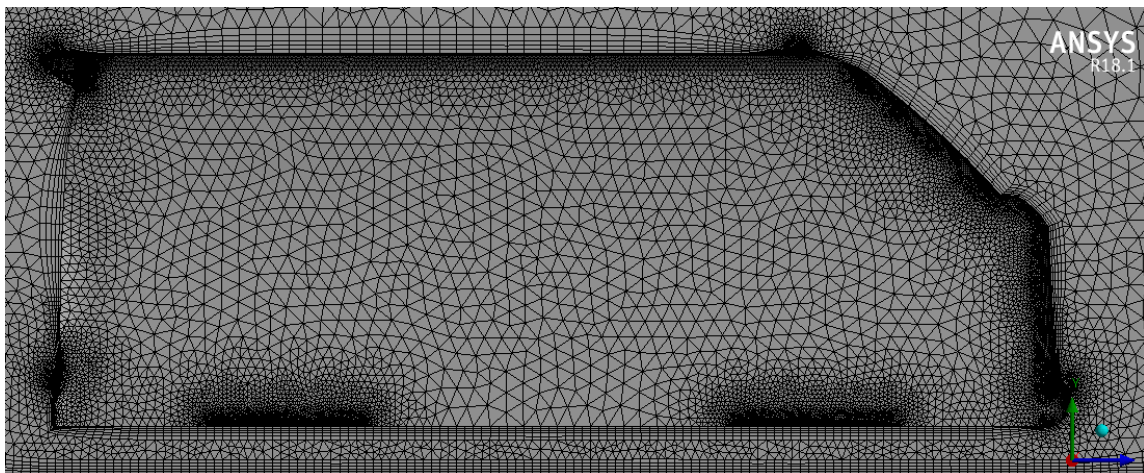


Figure B- 4. Model one after meshing.

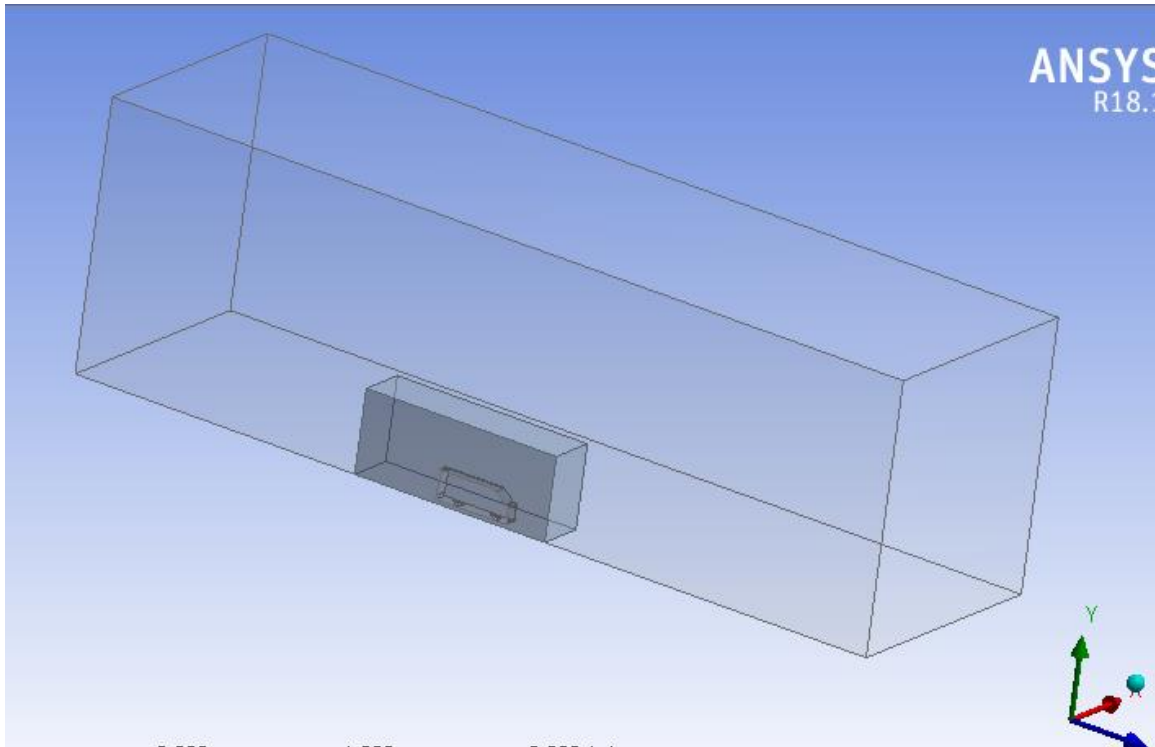


Figure B- 5. Model two inside the enclosure box.

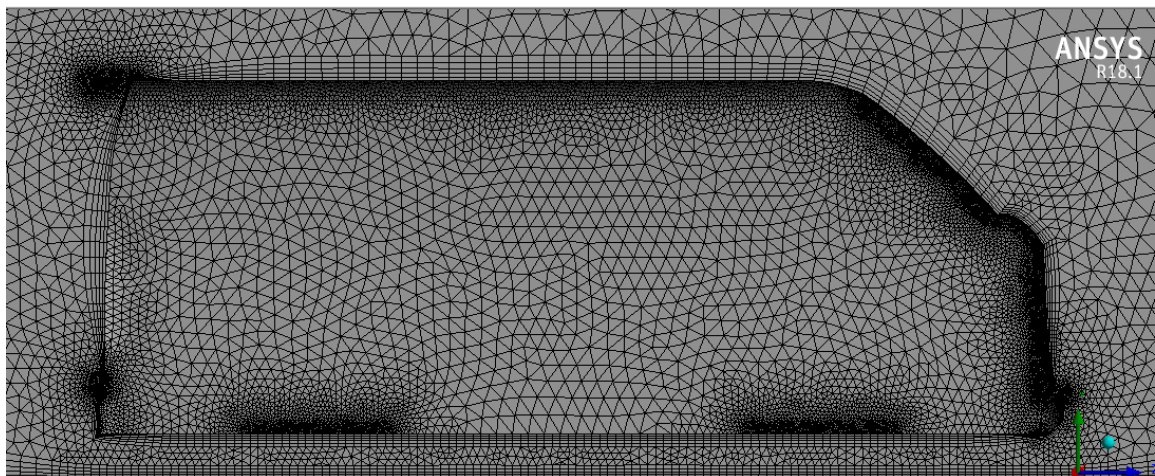


Figure B- 6. Model two after meshing.

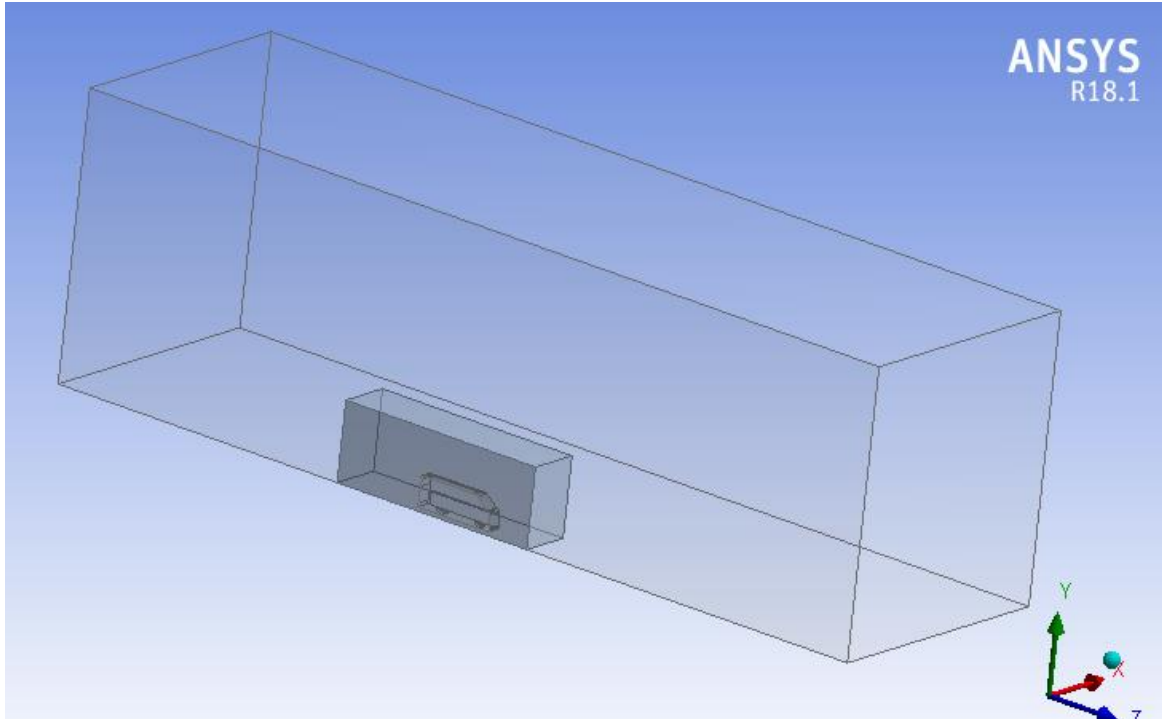


Figure B- 7. Model three inside the enclosure box.

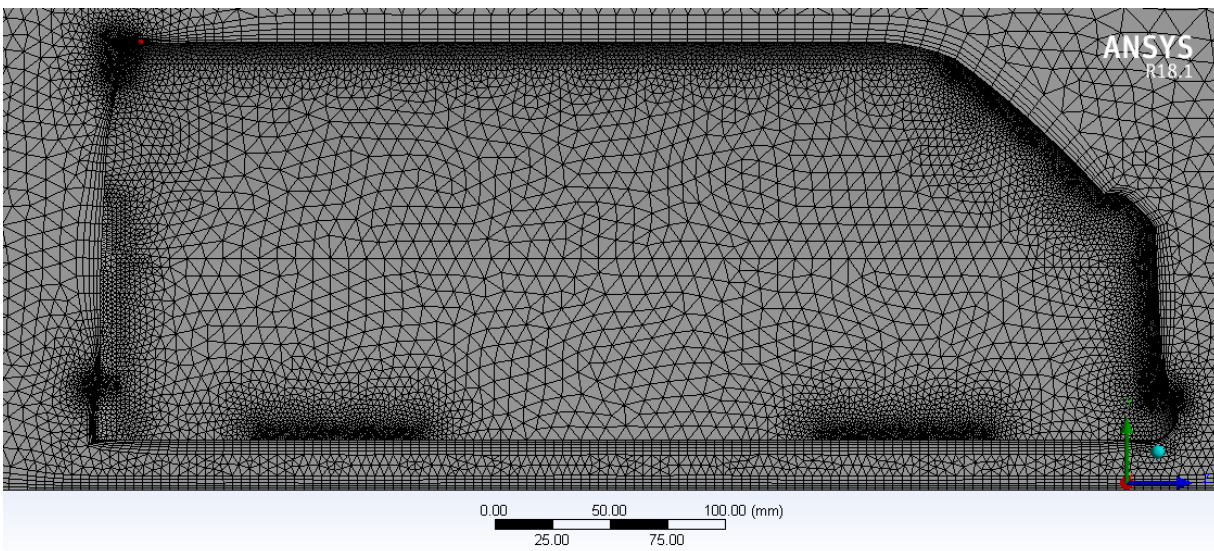


Figure B- 8. Model three after meshing.

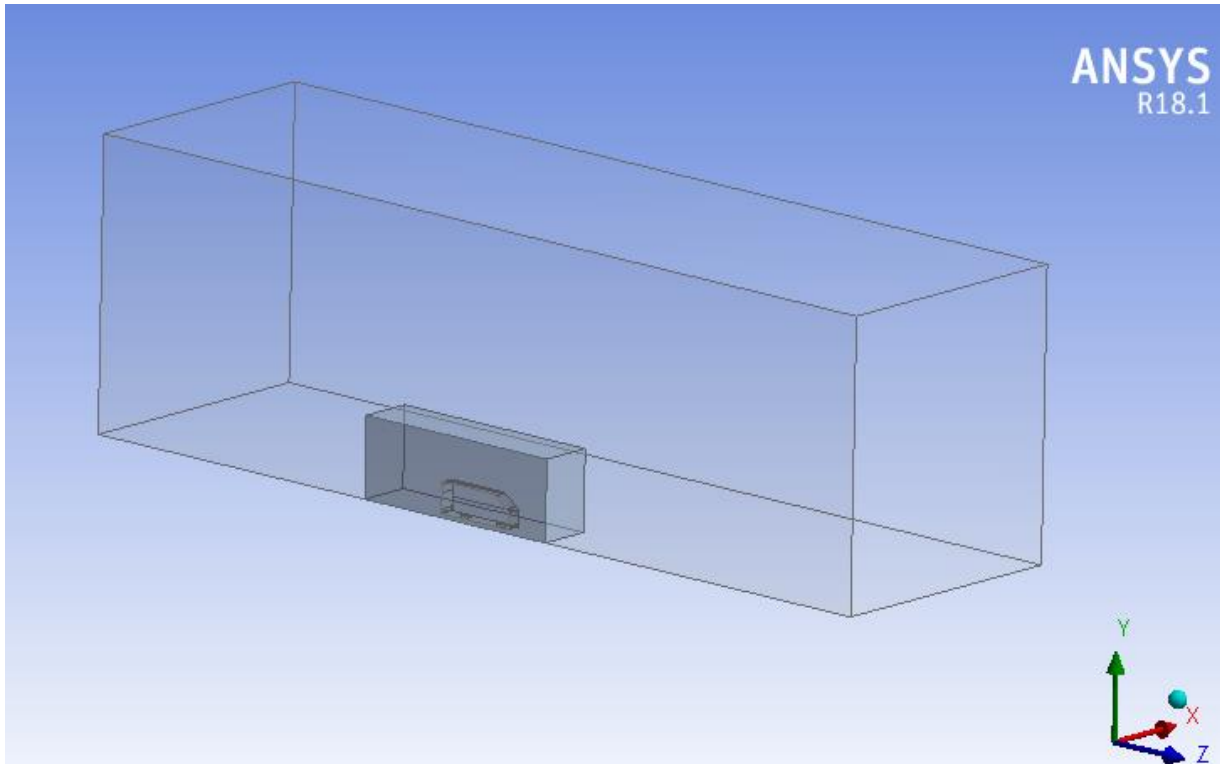


Figure B- 9. Model four inside the enclosure box.

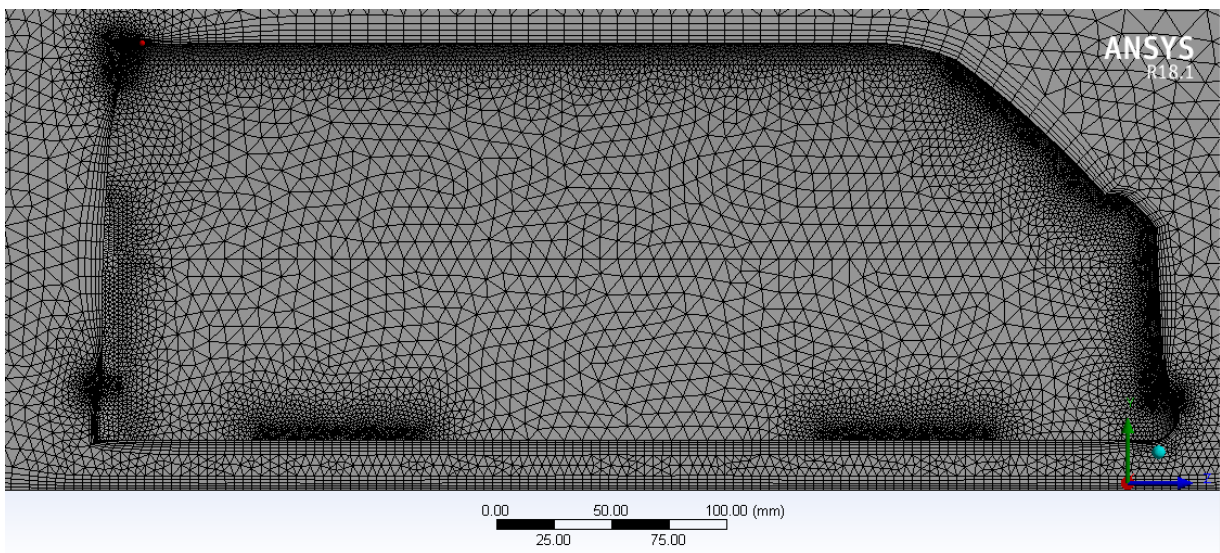


Figure B- 10. Model four after meshing.

Appendix C: Mesh Details

Table C- 1. Baseline Model Mesh Detail Object Name Mesh

Object Name	<i>Mesh</i>
State	Solved
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Export Format	Standard
Sizing	
Size Function	Proximity and Curvature
Relevance Center	Coarse
Curvature Normal Angle	12.0 °
Numb Cells Across Gap	5
Proximity Size Function Sources	Edges
Min Size	1.0 mm
Proximity Min Size	0.50 mm
Max Tet Size	256.0 mm
Growth Rate	Default (1.20)
Minimum Edge Length	0.114190 mm
Quality	
Smoothing	Medium
Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	First Aspect Ratio
First Aspect Ratio	5.
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	347458
Elements	1340726

Table C- 2. Model One Mesh Detail

Object Name	<i>Mesh</i>
State	Solved
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Export Format	Standard
Sizing	
Size Function	Proximity and Curvature
Relevance Center	Coarse
Curvature Normal Angle	12.0 °
Numb Cells Across Gap	5
Proximity Size Function Sources	Edges
Min Size	1.0 mm
Proximity Min Size	0.50 mm
Max Tet Size	256.0 mm
Growth Rate	1.20
Quality	
Smoothing	Medium
Mesh Metric	None

Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	First Aspect Ratio
First Aspect Ratio	5.
Maximum Layers	5
Growth Rate	1.2
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	386690
Elements	1472221

Table C- 3. Model Two Mesh Detail

Object Name	<i>Mesh</i>
State	Solved
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Export Format	Standard
Sizing	
Size Function	Proximity and Curvature
Relevance Center	Coarse
Curvature Normal Angle	12.0 °
Num Cells Across Gap	5
Proximity Size Function Sources	Edges
Min Size	1.0 mm
Proximity Min Size	1.0 mm
Max Tet Size	256.0 mm
Growth Rate	1.20
Minimum Edge Length	0.114190 mm
Quality	
Smoothing	Medium
Mesh Metric	None
Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	First Aspect Ratio
First Aspect Ratio	5.
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	323900
Elements	1281348

Table C- 4. Model Three Mesh Detail

Object Name	<i>Mesh</i>
State	Solved
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Export Format	Standard
Sizing	
Size Function	Proximity and Curvature
Relevance Center	Coarse
Curvature Normal Angle	12.0 °
Num Cells Across Gap	5
Proximity Size Function Sources	Edges
Min Size	1.0 mm
Proximity Min Size	1.0 mm
Max Tet Size	256.0 mm
Growth Rate	Default (1.20)
Minimum Edge Length	3.7791e-002 mm
Quality	
Check Mesh Quality	Yes, Errors

Smoothing	Medium
Mesh Metric	None
Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	First Aspect Ratio
First Aspect Ratio	5.
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	319429
Elements	1246497

Table C- 5. Model Four Mesh Detail

Object Name	<i>Mesh</i>
State	Solved
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Export Format	Standard
Sizing	
Size Function	Proximity and Curvature
Relevance Center	Coarse
Curvature Normal Angle	12.0 °
Num Cells Across Gap	5
Proximity Size Function Sources	Edges
Min Size	2.0 mm
Proximity Min Size	1.0 mm
Max Tet Size	256.0 mm
Growth Rate	Default (1.20)
Minimum Edge Length	0.114190 mm
Quality	
Check Mesh Quality	Yes, Errors

Smoothing	Medium
Mesh Metric	None
Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	First Aspect Ratio
First Aspect Ratio	5.
Maximum Layers	5
Growth Rate	1.2
View Advanced Options	No
Assembly Meshing	
Method	Tetrahedrons
Feature Capture	Program Controlled
Tessellation Refinement	Program Controlled
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Statistics	
Nodes	295463
Elements	1168539

Appendix D: CFD baseline model drag coefficient result at a speed of 12m/s and 17m/s for comparison of experimental validation

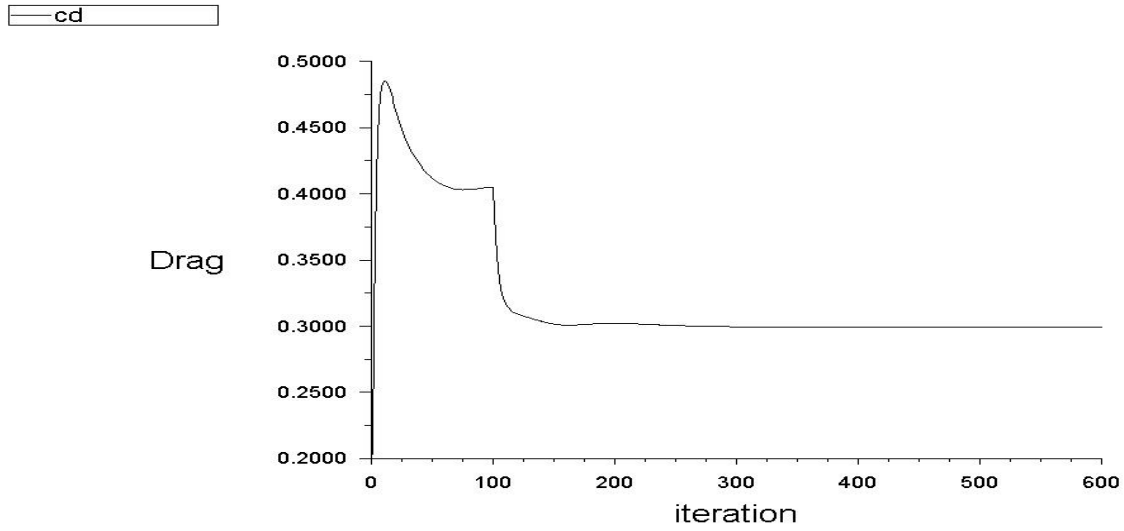


Figure D- 1. Drag coefficient of a baseline model at a speed of 12m/s.

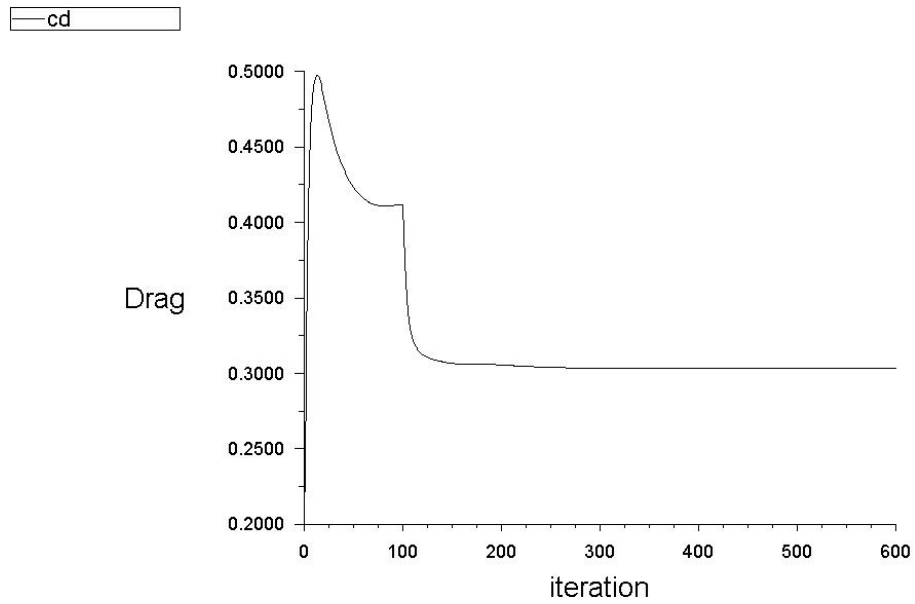


Figure D- 2. Drag coefficient of a baseline model at a speed of 17m/s.

