



JIMMA UNIVERSITY
SCHOOL OF GRADUATE STUDIES
JIMMA INSTITUTE OF TECHNOLOGY
FACULTY OF MECHANICAL ENGINEERING
AUTOMOTIVE ENGINEERING STREAM

AERODYNAMIC ANALYSIS OF BISHOFTU PLUTUS DOUBLE CABIN
PICK-UP VEHICLE WITH VORTEX GENERATORS

BY
BELETE ERBELO

Jan, 2024
Jimma, Ethiopia

JIMMA UNIVERSITY
SCHOOL OF GRADUATE STUDIES
JIMMA INSTITUTE OF TECHNOLOGY
FACULTY OF MECHANICAL ENGINEERING
AUTOMOTIVE ENGINEERING STREAM

AERODYNAMIC ANALYSIS OF BISHOFTU PLUTUS DOUBLE CABIN
PICK-UP VEHICLE WITH VORTEX GENERATORS

A thesis Submitted to The School of Graduate Studies of Jimma University, Jimma
Institute of Technology, in Partial Fulfillment of The Requirements for The Degree
of Masters of Science in Automotive Engineering

BY
BELETE ERBELO

ADVISOR: Dr. N. RAMESH BABU (Ph.D.)

CO-ADVISOR: Mr. ESHETU TADESSE (MSc)

DECLARATION

I declare that this thesis work entitled “**Aerodynamic Analysis of Bishoftu Plutus Double Cabin Pick-up vehicle with Vortex generators**” is my work and has not been submitted to any university for a similar purpose. The references used in this thesis are duly recognized by proper citations.

Name of Student

Signature

Date

Belete Erbelo



18/12/2023

APPROVAL PAGE

A thesis entitled “**AERODYNAMIC ANALYSIS OF BISHOFTU PLUTUS DOUBLE CABIN PICK-UP VEHICLE WITH VORTEX GENERATORS**” has been approved by the following assessment boards: Advisors, Co-Advisor Examiners, Chairman, in partial fulfillment of the requirement for the degree of Master of Science in Automotive engineering.

Name

External Examiner

Gertachew Shukri (Dr.-Ing.)



/

Jan. 29, 2024

Name

Signature

Date

Internal Examiner



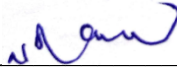
/

Main advisor

Signature

Date

Dr. N. Ramesh Babu



/

Co-advisor

Signature

Date

Eshetu Tadesse



/

19/12/2023

Chair Person

Signature

Date

/

ABSTRACT

The science of aerodynamics determines how gaseous fluids such as air, flow as well as the forces, exerted on moving objects such as automobiles. Due to the significant increase in fuel prices and vehicle speeds since the beginning of the 20th century, it has become crucial to minimize aerodynamic drag. This thesis work explains the aerodynamic study of a Bishoftu double cabin pickup truck equipped with delta and triangular-shaped vortex generators at the roof end of the truck to reduce the vehicle's aerodynamic drag coefficient. Pickup vehicles have a higher aerodynamic drag coefficient than the other types of road vehicles compared with Sport Utility Vehicles (SUVs). The shape of the Bishoftu Plutus double cabin pickup vehicle is a direct effect and it can be influenced by the drag coefficient result. The main objective of this thesis work is to examine the aerodynamic performance of the Bishoftu Plutus double cabin pickup vehicle with different configurations of vortex generators. To validate the numerical simulation results with experimental tests. It was also conducted under various parameter settings. This thesis work examined the flow field structures of adding vortex generators to the model's roof end to demonstrate the reduction mechanism numerical simulation results of pressure, velocity, and vortex structure provided by contrasting the baseline model with experimental data. Using ANSYS fluent software for computational analysis and conducted experimental tests on wind tunnels. Two different types of passive flow drag reduction devices were investigated in this work, and each had a significant impact on the development of the separation flow. Using vortex generators on the roof end of the model has a maximum drag reduction of 6.50% Lift coefficient reduction of 8.69%, and 14% average fuel save achieved.

Keywords: Drag coefficient, Lift coefficient, Vortex generator, Computational fluid dynamics.

ACKNOWLEDGMENTS

To begin with, I want to express my sincere gratitude to my Lord and God for all of his blessings and provisions during my life and the writing of my thesis. Additionally, I want to express my gratitude to Kuleni Diro, chair of the automotive department, as well as my entire family, for their encouragement and support. My sincerest appreciation goes to my advisor, Associate Professor Dr. N. Ramesh Babu, for all of his beneficial guidance and constructive feedback. I also want to thank Mr. Eshetu Tadesse my co-advisor, for all of the encouragement he provided me. Lastly, I want to express my gratitude to my friends for their helpful advice and for supporting me through all of the requests they made.

TABLE OF CONTENTS

DECLARATION	I
APPROVAL PAGE	II
ABSTRACT.....	III
ACKNOWLEDGMENTS	IV
TABLE OF CONTENTS.....	V
LIST OF TABLES	VIII
LIST OF FIGURES	IX
LIST OF ACRONYMS	XIII
CHAPTER ONE	1
INTRODUCTION	1
1.1 Background of the study	1
1.2 Statement of problem.....	4
1.3 Objectives	4
1.3.1 General Objective	4
1.3.2 Specific Objectives	4
1.4 Significance of the Study	5
1.5 Scope of the Thesis	5
1.6 Limitations	6
1.7 Organization of the thesis	6
CHAPTER TWO	7
LITERATURE REVIEW	7
2.1 Pickup Truck History.....	7
2.2 Pickup Truck Aerodynamics.....	16
2.3 Drag.....	23
2.4 Lift Force	26
2.5 Computational Fluid Dynamics	27
2.6 Advantages behind the reduced drag of a vehicle	28
2.7 Drag Reduction of Pickup Vehicle by Vortex Generators.....	29
2.8 Passive flow Control System	31

2.9 The Boundary Layer over a pickup truck	32
2.10 Boundary Layer Transition	33
2.11 Separation	35
2.12 Stream-wise Mean Flow Field	36
2.13 Pressure drag	37
2.14 Friction drag	37
2.15 Flow Fields and Pressure	37
2.16 Verification of Vortex generators mechanism	38
2.17 Design Considerations	38
2.18 Influence of the vortex generators on the aerodynamic forces	38
2.19 Summary	40
2.20 Research gap	41
CHAPTER THREE	42
MATERIALS AND METHODOLOGY	42
3.1 Introduction	42
3.2 Materials and Equipment	42
3.3 Methodology	43
3.3.1 Model Specifications of Bishoftu SG Plutus double cabin pickup	43
3.3.2 Bishoftu Double Cabin Pickup Model	45
3.3.3 Vortex Generators	46
3.3.4 Computational Domain	49
3.3.5 Mesh Generation	49
3.4 Pre-Processing	54
3.5 Solver	54
3.6 Numerical methods	54
3.7 Turbulence Modeling	56
CHAPTER FOUR	57
RESULTS AND DISCUSSIONS	57
4.1 Drag and Lift Coefficient Reduction in Optimization Method	94
4.2 Experimental Setup and Results	100
4.3 None-Dimensional Analysis	108
4.2 Calculations of fuel saved for modified model	110

CHAPTER FIVE	112
CONCLUSIONS AND RECOMMENDATIONS	112
5.1 Recommendations	113
5.2 Future work	113
REFERENCES	114
APPENDEX.....	124

LIST OF TABLES

Table 1 Basic Dimension of Bishoftu double cabin pickup	43
Table 2: Bishoftu pickup specification for SG Plutus Double Cabin Pick up 4JB1TI Model.....	44
Table 3: Drag and Lift Coefficient of Baseline at Different Speeds.....	59
Table 4: Drag and Lift Coefficient of Pickup with Delta VGs at Different Speeds	71
Table 5: Drag and Lift Coefficient of Pickup with Triangular VGs at Different Speeds.....	80
Table 6: Drag and Lift coefficient result of all models [95]	88
Table 7: Input parameters with delta-shaped vortex generator.....	94
Table 8: Averaged results of experimental test in wind tunnel	105
Table 9: Result summary of fuel saved for modified Bishoftu pickup models [95].....	110

LIST OF FIGURES

Figure 1: History of the drag for road vehicles [11]	2
Figure 2: Bishoftu Plutus double cabin pickup truck structure [12]	3
Figure 3: Ford Model T pickup in 1922 [14]	7
Figure 4: Chevrolet Model ER pickup in 1947 [16]	8
Figure 5: Dodge B-3-B pickup 1950S [16]	9
Figure 6: International Travelette pickup model [14]	10
Figure 7: Dodge D100 Utilized pickup [16]	11
Figure 8: Dodge Adventurer pickup model [17]	12
Figure 9: Ford's F-Series Pickup Truck in 1980 [18]	13
Figure 10: Hilux pickup Truck in 1987 [17]	14
Figure 11: Ford Single cab Pickup Truck in 1990s [20]	15
Figure 12: External Flow Field over the Car Body [36]	20
Figure 13: Flow field above a single (a) and extended cabin (b) pickup truck [39]	21
Figure 14: Proposed nomenclature for pick-up parts [42]	22
Figure 15 : The drag force due to air flow acting on a car [44]	24
Figure 16: Drag, Lift and down force from over body flow [48]	26
Figure 17: Schematic of velocity profile around rear end [6]	32
Figure 18: Function of Vortex Generators [74]	33
Figure 19: Flow separation and wake [80]	35
Figure 20: Bishoftu Double Cabin Pickup Model without Vortex Generator	45
Figure 21: Bishoftu Pickup with delta type vortex generator	46
Figure 22: Design of delta type vortex generator in 15mm, 20mm and 25mm	47
Figure 23: 3D design of delta type vortex generator with SOLIDWORKS	47
Figure 24: Bishoftu Pickup with triangular type vortex generator	48
Figure 25: 3D design of triangular type with SOLIDWORKS	48
Figure 26 : Wind Tunnel Dimensions	49
Figure 27: Mesh size in Body element	50
Figure 28: Mesh size in car element	51
Figure 29 : Velocity inlet boundary condition as baseline model	52
Figure 30: Pressure outlet boundary condition in baseline model	52
Figure 31: Walls boundary condition as baseline model	53

Figure 32: Symmetry boundary condition as baseline model.....	53
Figure 33: Classification of Turbulence model [95].....	56
Figure 34: Scale Residual of Baseline Model at 80 km/h.....	58
Figure 35: Drag Coefficient of Baseline Model at 80 km/h	58
Figure 36: Lift Coefficient of Baseline Model at 80 km/h	59
Figure 37: Pressure Contour in Symmetry Plane at 80 km/h.....	60
Figure 38: Velocity Contour in Symmetry Plane at 80 km/h	60
Figure 39: Scale Residual of Baseline Model at 100 km/h.....	61
Figure 40: Drag Coefficient of Baseline Model at 100 km/h	61
Figure 41: Lift Coefficient of Baseline Model at 100 km/h	62
Figure 42: Pressure Contour in Symmetry Plane at 100 km/h.....	62
Figure 43: Pressure Contour in car at 100 km/h	63
Figure 44: Velocity Contour in Symmetry Plane at 100 km/h	63
Figure 45: Velocity Contour in car at 100 km/h	64
Figure 46: Scale Residual of Baseline Model at 120 km/h.....	64
Figure 47: Drag Coefficient of Baseline Model at 120 km/h	65
Figure 48: Lift Coefficient of Baseline Model at 120 km/h	65
Figure 49: Pressure Contour in Symmetry Plane at 120 km/h.....	66
Figure 50: Pressure Contour in car at 120 km/h	66
Figure 51: Velocity Contour in Symmetry Plane at 120 km/h	67
Figure 52: Velocity Contour in car at 120 km/h	67
Figure 53: Drag Coefficient with delta type vortex generator at 80 km/h.....	68
Figure 54: Lift Coefficient with Delta type vortex generator at 80 km/h	68
Figure 55: Drag Coefficient with Delta type vortex generator at 100 km/h.....	69
Figure 56: Lift Coefficient with Delta type vortex generator at 100 km/h.....	69
Figure 57: Drag Coefficient with Delta type vortex generator at 120 km/h.....	70
Figure 58: Lift Coefficient with Delta type vortex generator at 120 km/h.....	70
Figure 59: Scale Residual with Delta type vortex generator	71
Figure 60: Pressure Contour in car with delta type vortex generator	72
Figure 61: Pressure Contour in Symmetry with delta type vortex generator Plane at 80 km/h ...	72
Figure 62: Velocity contour in symmetry plane with delta type at 80 km/h	73
Figure 63: Velocity Contour in car with delta type vortex generator at 80 km/h.....	73
Figure 64: Pressure Contour in Symmetry with delta type vortex generator Plane at 100 km/h .	74

Figure 65: Velocity contour in symmetry plane with delta type at 100 km/h	74
Figure 66: Velocity Contour in car with delta type vortex generator at 100 km/h.....	75
Figure 67: Pressure Contour in Symmetry with delta type vortex generator Plane at 120 km/h .	75
Figure 68: Velocity Contour in car with delta type vortex generator at 120 km/h.....	76
Figure 69: Velocity Contour in car with delta type vortex generator at 120 km/h.....	76
Figure 70: Drag Coefficient with Triangular vortex generator at 80 km/h.....	77
Figure 71: Lift Coefficient with Triangular vortex generator at 80 km/h.....	77
Figure 72: Drag Coefficient with Triangular vortex generator at 100 km/h.....	78
Figure 73: Lift Coefficient with Triangular vortex generator at 100 km/h.....	78
Figure 74: Drag Coefficient with Triangular vortex generator at 120 km/h.....	79
Figure 75: Lift Coefficient with Triangular vortex generator at 120 km/h.....	79
Figure 76: Scale Residual in application of Triangular vortex generator	80
Figure 77: Pressure contour in symmetry plane with triangular vortex generator at 80 km/h	81
Figure 78: Pressure contour in car with triangular vortex generator at 80 km/h	81
Figure 79: Velocity contour in symmetry plane with triangular vortex generator at 80 km/h	82
Figure 80: Velocity contour in car with triangular vortex generator at 80 km/h.....	82
Figure 81: Pressure contour in symmetry plane with triangular vortex generator at 100 km/h ...	83
Figure 82: Pressure contour in car with triangular vortex generator at 100 km/h	83
Figure 83: Velocity contour in symmetry plane with triangular vortex generator at 100 km/h ...	84
Figure 84: Velocity contour in car with triangular vortex generator at 100 km/h.....	84
Figure 85: Pressure contour in symmetry plane with triangular vortex generator at 120 km/h ...	85
Figure 86: Pressure contour in car with triangular vortex generator at 120 km/h	85
Figure 87: Velocity contour in symmetry plane with triangular vortex generator at 120 km/h ...	86
Figure 88: Velocity contour in car with triangular vortex generator at 120 km/h.....	86
Figure 89 : Velocity vector effect on the pickup model with triangular vortex generator	87
Figure 90: Mesh grid independence test of baseline model.....	89
Figure 91: Drag coefficient and speed comparisons of baseline model	89
Figure 92: Lift coefficient and speed comparisons of baseline model	90
Figure 93: Drag coefficient and speed comparisons with delta type vortex generator.....	90
Figure 94: Drag coefficient and speed comparisons with triangular type vortex generator	91
Figure 95: Lift coefficient and speed comparisons with delta type vortex generator.....	91
Figure 96: Lift coefficient and speed with triangular vortex generator	92
Figure 97: Drag comparisons of baseline model with delta type vortex generator	92

Figure 98: Drag coefficient comparisons of baseline model with triangular vortex generator	93
Figure 99 : Drag coefficient of baseline with delta and triangular vortex generator	93
Figure 100: Contour of optimization with delta-shaped vortex generator.....	94
Figure 101: The 3D surface of delta-shaped vortex generator	95
Figure 102: Predicted and Actual of drag coefficient results with delta vortex generator	95
Figure 103: Contour of optimization with triangular-shaped vortex generator	96
Figure 104: The 3D surface of triangular-shaped vortex generator.....	96
Figure 105: Predicted and Actual results with triangular-shaped vortex generator.....	97
Figure 106: Contour and 3D surface of lift reduction with delta-shaped vortex generator	97
Figure 107: Predicted with actual in delta-shaped vortex generator.....	98
Figure 108: Contour and 3D surface of lift reduction with triangular-shaped vortex generator ..	98
Figure 109: Predicted with actual in triangular-shaped vortex generator	99
Figure 110: Wind tunnel structure in thermo fluid laboratory.....	100
Figure 111: The 1/50 th scale model of Bishoftu double cabin pickup	101
Figure 112: Adjusting model holder before installing.....	101
Figure 113: installing the model into wind tunnel machine	102
Figure 114: Structure of installed model in the wind tunnel	102
Figure 115: Digital manometer for different measurements.....	103
Figure 116: Digital amplifier for drag force (N) and lift force (N) measurements.....	103
Figure 117: Experimental set-up in the wind tunnel.....	104
Figure 118: Scaled model during experimental test inside the wind tunnel test	104
Figure 119: Coefficient of drag of baseline 1/50th scaled model	105
Figure 120: Distribution of static pressure.....	106
Figure 121: Distribution of velocity magnitude (km/h).....	106
Figure 122: Pressure contour of scaled model in 77.4 km/h.....	107
Figure 123: Velocity contour of 1/50 th scaled model in 77.4 km/h	107
Figure 124 : The relationship between drag coefficient and Reynolds number	108
Figure 125: Numerical and experimental coefficient of drag comparisons.....	109
Figure 126: Amount of drag force (N) fuel saved	111
Figure 127: Amount of drag force reduction (Fd) and speed	111

LIST OF ACRONYMS

APG	Adverse pressure gradient
BL	Boundary layer
BPV	Bishoftu pickup vehicle
CAD	Computer-aided design
CD	Coefficient of drag
CFD	Computational fluid dynamics
DOE	Design of experts
GHG	Greenhouse gas
METEC	Metals and Engineering cooperation
RANS	Reynolds averaged Navier-Stokes equations
RWD	Rear-wheel-drive
SUV	Sport Utility Vehicle
VG	Vortex generators

CHAPTER ONE

INTRODUCTION

1.1 Background of the study

The science of aerodynamics determines how gaseous fluids such as air, flow as well as the forces, exerted on moving objects such as automobiles [1]. Due to the significant increase in fuel prices and vehicle speeds since the beginning of the 20th century, it has become crucial to minimize aerodynamic drag [2]. The most vital factor in the automotive industry is aerodynamics, which is closely connected to the green economy and fuel consumption [3]. There are major global issues that have been mentioned in energy conservation and emission reduction. In the automotive sector, reducing aerodynamic drag is regarded as an effective way to improve fuel efficiency and save energy [4].

The aerospace industry was one of the first to investigate in coefficient of drag issues after the automotive industry became conscious of its potential impact on fuel efficiency [5]. This situation encouraged the manufacturers to improve the appearance of their vehicles. Some aspects that improve a vehicle's engine power, are a comfortable layout and aerodynamic design [6]. The boundary layer is one of the interrelated factors that interact to produce the pressure distribution around a vehicle. The drag caused by this effect largely depends on where flow separation occurs, a plate held perpendicular to the flow will cause separation around the leading face producing a wide wake with a high coefficient of drag [7].

Drag is the result of the air molecule's interaction with the body of the vehicle due to the relative velocity of the air and vehicle, which produces a force acting in opposite motion. Aerodynamic drag must be reduced for the vehicle to perform better. When the vehicle's speed rises, the issue of aerodynamic drag becomes more significant. Any gas that passes by an object causes aerodynamic forces on it known as drag and lift forces, which have an impact on how it moves [8].

Many governments have committed to significantly reducing greenhouse (GHG) emissions to address climate change across the globe. Since almost all types of ground vehicles currently depend on liquid fossil fuels, The transportation sector makes a significant contribution to greenhouse gas emissions [9]. Describe the historical context of reference models for vehicle aerodynamics that date back to the 1970s in detail. None-production models were divided into two categories, such as basic car shapes and simple bodies. The initial energy scarcity of the 1970s encouraged a clear perspective on reducing the coefficient of drag as a way to achieve improved fuel consumption, which is an essential issue presented in their review [10]. The history of drag for road vehicles is shown in Figure 1.

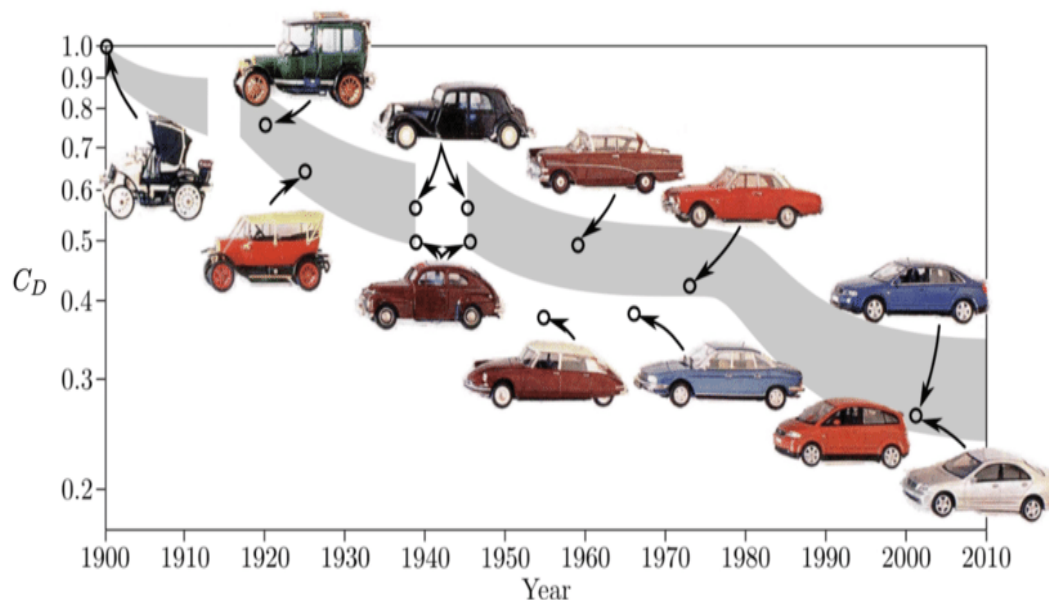


Figure 1: History of the drag for road vehicles [11]

In Ethiopia, numerous motor vehicles are available. The majority of these vehicles are imported from other countries. However, some of them have locally manufactured bodies without engine compartments and chassis designs. The majority of vehicle body-building firms did not consider aerodynamics in and out of factor while modifying the outer surface shape of the car.

In the 1950s, customers started buying pickup vehicles for lifestyle reasons instead of as a tool for the job. In the 1990s, less than 15% of owners stated that they utilized their pickup vehicles primarily for work [3].

Bishoftu Plutus double cabin pickup vehicle is fabricated by China automobile manufacturer SG Automotive and assembled in Ethiopia Metals and Engineering Cooperation (METEC) in the Bishoftu automotive industry. Governmental and private organizations are widely using Bishoftu Plutus double cabin pickup for several purposes in the country.



Figure 2: Bishoftu Plutus double cabin pickup truck structure [12]

Many studies have been conducted on the rear side of pickup vehicles' aerodynamics influence with add-devices to quantify aerodynamic analysis of pickup vehicles. To evaluate the aerodynamic characteristics of a pickup truck manufactured in Bishoftu, Ethiopia, this study uses a systematic approach called "on the roof end of the Bishoftu pickup vehicle." Figure 2 shows a pickup truck from Bishoftu (BPV). Usually, they have double-capped and four wheels (two on the front and two on the rear). It could also carry five people: the driver properly placed in the front left side of the cabin, one in the front right, three in the back, and there is also a luggage compartment in the back of the car. Its maximum speed is 120 km/h, and its maximum gross weight (vehicle plus occupant) is approximately 2495kg [12]. In the automotive industry, determining the airflow around a vehicle and how it affects drag and lift forces are vital research topics. Vehicle performance and fuel efficiency suffer from high drag. The shape of the vehicle and flow controls are related to reducing drag. The subject of improving vehicle performance has attracted a lot of research interest. Flow separation on the vehicles usually leads to considerable pressure-based drag [13].

1.2 Statement of problem

Pickup vehicles have a higher drag coefficient than the other types of road vehicles compared with Sport Utility Vehicles (SUVs). The shape of the Bishoftu Plutus double cabin pickup vehicle is a direct effect and it can be influenced by the drag coefficient result. The effects of fluid motion and complexity around the car model are the reason for the high aerodynamic drag coefficient and, the high aerodynamic drag coefficient is related to the pickup vehicle's fuel consumption, performance, and design of the body. The geometry of the Bishoftu Plutus double cabin pickup vehicle is related to it and has a direct impact on motion flow, including drag and lift coefficient. The researchers did not consider the number of both delta and triangular-shaped vortex generators on the roof end of pickup vehicles with different configurations, heights, and parameters. The existing studies only focused on the use of delta and triangular-shaped vortex generators and only investigated the rear end of the pickup vehicle. Using computational fluid dynamics, the length of the Bishoftu Plutus double cabin pickup vehicles affects the reverse flow. However, the result of the highest aerodynamic force on pickup vehicles is obtained with the action of a downward flap on the body.

1.3 Objectives

1.3.1 General Objective

The general objective of this thesis work is to analyze the aerodynamic performance of the Bishoftu Plutus double cabin pickup vehicle with two types of vortex generators.

1.3.2 Specific Objectives

-  To model the body of the pickup with and without vortex generators
-  To conduct aerodynamic analysis using CFD software varying vortex generator.
-  To compare the results obtained with the aerodynamic analysis of Bishoftu Plutus double cabin pickup with and without vortex generators.
-  To reduce the drag coefficient of the Plutus double-cabin pickup vehicle.
-  To analyze the effects of vortex generators in the Bishoftu Plutus double cabin pickup.
-  To conduct the Experimental validation of the baseline model in wind tunnel test to evaluate the drag coefficient of the model.

1.4 Significance of the Study

The findings of this thesis work have significance in the aerodynamics analysis of the Bishoftu Plutus double cabin pickup vehicle with two various types of vortex generators analyze them with different configurations to become a popular factor for reducing the aerodynamic The pickup truck's drag and lift coefficients. Due to the increased back pressure of the vehicle with triangular and delta-shaped vortex generators in the roof end of the vehicle. Furthermore, improving the fuel efficiency of the Bishoftu Plutus double cabin pickup vehicle with the mathematical application. This work can also serve as a reference for academic and industry researchers in their future work on the design and development of car models for better aerodynamic shape and performance.

1.5 Scope of the Thesis

In this thesis, the main focus is on reducing the aerodynamic drag of the pickup vehicle by applying add-on devices such as vortex generators increasing the back pressure with Triangular and Delta wing shape vortex generators, and analyzing numerical analysis of CFD modeling simulation. Implementing the same shape of delta and triangular-shaped vortex generators with different dimensions and configurations at the roof end of the Bishoftu Plutus double cabin pickup vehicle for balancing flow separation and conducting experimental validation in wind tunnel test of a baseline car model to compare the results of drag coefficients with the numerical analysis.

1.6 Limitations

Aerodynamic analysis is mostly performed with two methods in the automotive industry, numerically, using ANSYS Fluent software, and experimentally, utilizing wind tunnel tests. A high-performance computer is required to simulate the numerical analysis to get precise results, taking time to print a 3D model of Bishoftu plutus double cabin pickup is the main limitation. The study also focuses on a simplified 3D model of the Bishoftu double cabin pickup body aerodynamic performance with delta-shaped and triangular-shaped vortex generators designed on the models.

1.7 Organization of the Thesis

This research work comprises five chapters. The first chapter is an introduction to the study, which contains the background of the study, a statement of the problem, the significance of the study, the objectives of the study, the scope of the study, limitations, and the structure of the study. Chapter Two contains various relevant literature reviews. Chapter three explains the materials and methodology used to accomplish this research work by using computational fluid dynamic analysis of aerodynamic force and experimental test material. Discussion and comparison of the results are presented in chapter four. The results obtained from the coefficients of lift and drags are illustrated in the form of graphics and tables for both quantitative and qualitative arguments. For a better understanding of airflow on the vehicle body, the contours of flow are discussed. Chapter five consists of a conclusion of the whole of this thesis work and recommendations that help other researchers to know and recognize for better future work.

CHAPTER TWO

LITERATURE REVIEW

For a better understanding and significance of the current study, on related major areas, various relevant literature were reviewed.

2.1 Pickup Truck History

Automobiles were initially marketed only for the chassis, with the bodywork being added by other firms. Max and Morris Grabowsky established the Rapid Motor Vehicle Company in Pontiac, Michigan, in 1902 to produce trucks with a ton of carrying capacity. The Galion Allsteel Body Company, an innovator in the development of the pickup and dump truck, constructed and mounted towing boxes on Ford Model T chassis that had been slightly modified in 1913, and on Model TT chassis starting in 1917. To get some of this market share, Dodge unveiled a 3/4-ton pickup in 1924 that consisted of a wood-only cab and body. Ford followed up in 1925 with a half-ton featuring a steel body, an adjustable tailgate, and powerful rear springs. It was advertised as the "Ford Model T Runabout with Pickup Body" and cost \$281; 34,000 units were produced. The Model A was equipped with a three-speed transmission, roll-up side windows, a safety glass windshield, and a closed cab, took its place in 1928 [14]. A 1922 Ford Model T pickup is shown in Figure 3.



Figure 3: Ford Model T pickup in 1922 [14]

Pickup trucks have existed for well over a century. A pickup truck's open back and modified truck cab define its basic design. It is simple to load and move goods to and from owing to the open back, or "bed." Due to its application as a vehicle to transport and carry heavy loads, the term "pick up" was created.

The Ford Model T Runabout with a pickup body was the first widely used pickup truck. Henry Ford manufactured this car in a plant in 1925 [15]. During World War II, the three major automotive manufacturers of the 1940s—Chevrolet, Ford, and Dodge—launched their postwar models in 1949. Many pickup enthusiasts think that the Dodge B-Series pickup trucks' different, never-before-seen cab design helped them get the advantage. A chair-height seat, a wider, taller windshield, and optional rear quarter cab windows to reduce blind spots were all available on the Dodge half-ton pickup. Chevrolet created advanced design light-duty pickup trucks with upgraded features and a new, post-war design in the late 1940s. From 1948 to 1952, Ford developed the First Generation of F-Series pickup trucks [16]. Chevrolet ER model pickup in 1947 is shown in Figure 4.



Figure 4: Chevrolet Model ER pickup in 1947 [16]

Light-duty trucks for both GMC and Chevrolet that were intended for private ownership were first launched by GM in 1931. The Chevrolet Master served as the basis for these pickup trucks. The dedicated light-truck platform, known as the AK series by GM, was first produced in 1940, separately from passenger cars. Ford Australia developed the first Australian "ute" in 1932, and Ford North America preserved their pickup body available type regarding the Ford Model 51. Ford introduced the Ford F100 light-duty truck chassis in 1940, and It was changed to become the Ford F-Series in 1948 after World War II [15]. The US approved the Interstate Highway System in 1956 to support both private and business trucks. Growing numbers of Americans flocked to the suburbs during the decade, which aided in the development of the vehicle and the pickup truck.

Ford, Chevrolet, and Dodge continued to produce newer truck vehicles with modern features and body designs, maintaining their position as market leaders [16]. A 1950 Dodge pickup model is shown in Figure 5.



Figure 5: Dodge B-3-B pickup 1950S [16]

Ford, Chevrolet, and Dodge began to manufacture better truck vehicles with modern features and body designs and maintained their position as market leaders. The first contemporary V-8 engine with overhead valves debuted in the 1955 Chevrolet pickup truck, giving more horsepower at a faster speed. In 1957, the International Harvester debuted the first crew cab, which had three doors. The vehicle featured a fourth door in 1961 [16]. Customers began to buy pickups in the 1950s for aesthetic rather than economic reasons. Trucks with smooth sides and no fenders were launched, including the Chevrolet Fleetside, Chevrolet El Camino, Dodge Swept line, and Ford's specially designed Style side in 1957. Comfort features like power choices and air conditioning started to appear on pickups [14]. Pickup truck designs can vary significantly depending on the region, make, and model of the vehicle. The pickup truck has been referred to be a half-ton vehicle as it has developed over time. The frame and cab of early pickup trucks, which were most common in the 1960s, could hold up to half a ton (1000 pounds) of cargo.

Trucks are capable of carrying heavier loads due to the advancement of technology. Although it might seem difficult to compete with the technological advancements of the 1960s, Dodge succeeded with their "Lifestyle" pickup trucks. Many Americans were keen to travel in the 1970s. [14].

Pickup truck aesthetics can differ significantly depending on the region, make, and model of the vehicle. The pickup truck has been referred to be a half-ton vehicle as it has developed over time. The combined capacity of the bed and cab of early pickup trucks, which were most common in the 1960s. A 1961 Travelette pickup model is shown in Figure 6.



Figure 6: International Travelette pickup model [14]

Many Americans were keen to travel during the 1970s. New family-friendly cars like SUVs, station wagons, and pickup trucks were created to fill this new requirement. The Dodge D200 Camper Special had a slide-on camper body when it was released in 1972. GMC focused on people wishing to move huge loads like heavy equipment whereas Dodge was devoted to people looking to travel. In addition to introducing its first crew cab, GMC also started an ambitious initiative to upgrade all of its earlier models. The GMC series of pickup trucks was redesigned with cushioned inside. [15].

At the beginning of the decade, GMC unveiled new truck designs that featured a full-width hood, jet pod grilles, and a pinched-waist body crease. Following the arrival of the Dodge factory-built crew cab in 1963, Ford enlarged their pickup truck cab and developed their first crew cab vehicle in 1965. In the 1960s, Japanese automakers Datsun and Toyota revolutionized the North American pickup lifestyle through the development of the lightweight pickup truck. Although pickups were now built to cover greater distances at a faster rate of speed while carrying a bigger load, pickup truck transmissions significantly improved throughout the industry in the 1960s [16]. A 1964 Dodge D100 utilized pickup model is shown in Figure 7.



Figure 7: Dodge D100 Utilized pickup [16]

When Dodge released the Lifestyle pickup trucks in the 1970s, they raised the bar once more. Americans were ready to take journeys in the 1970s for vacations, cross-country road trips, and visits to relatives. When carrying campers and travel trailers, suitable for families vehicles such as station wagons, SUVs, and pickup trucks were popular. With a slide-on camper body, the 1972 Dodge D200 Camper Special satisfied the requirement. GMC, meanwhile, introduced its first crew cab to respond to a new client base determined to carry greater loads and equipment in pickup trucks. All models were modified by GMC, and the interiors were made better for carrying passengers. Along interior truck surfaces, cushioned material was utilized in place of metal fittings [16].

The concept of an extended cab pickup truck first appeared in the 1980s; GMC introduced full-size extended cab pickups in 1988, while Chevrolet introduced S-Series extended cab versions in 1983. In the meantime, to compete with more compact foreign models, the major American pickup truck producers Ford, Chevrolet, and Dodge launched smaller pickup trucks to the US market. Starting in 1987, GMC launched their new aerodynamic truck series at the end of the 1980s. The updated pickups became the benchmark for contemporary GMC pickup design and production. A 1970 Dodge Adventure pickup model is shown in Figure 8. The Ford F-Series has been manufactured continuously since 1948, and had a big decade in the 1990s; the eighth generation of the Ford F-Series was produced from 1987 to 1991.



Figure 8: Dodge Adventurer pickup model [17]

During the 9th generation, which spanned 1992–1996, For the first time since 1980, the 10th generation 1997 Ford F150 experienced a comprehensive redesign. Larger interior space, improved fuel efficiency, and improved aerodynamics were among the updates. In 1993, Dodge also introduced their new Ram trucks, also known as the T-300. The Dodge Ram had a roomier cab with additional storage space inside. In September 1978, the third-generation Hilux went on sale. Its model featured comfortable rides together with specs and equipment similar to sedans. Three standard-length variants and four long-body models were among the variations. The inside space was increased with the new top-of-the-line Super Deluxe model's extended cab, which was 90 mm longer than a conventional cab. The previous generation's 1.6-liter 12R-J engine was carried over [17].

The 3rd generation F-series truck is essentially identical to the previous version, although it has a new grille, parking lamps above the headlamps, and a concave body-length groove that contains side-marker lamps. In 1974, the Club Cab debuts, featuring either an inadequate seat with a movable bottom cushion or a pair of center-facing jump seats [18].

The extended cab pickup truck was invented when Chevrolet introduced the S-Series extended variation in 1983. In 1988, GMC began offering a full-sized extended cab pickup truck for sale. At the same time, Ford, Dodge, and Chevrolet, the "big three," had to contend with fending off foreign competition by introducing small trucks to the American market. By the end of the decade, GMC introduced a new series, more aerodynamic and streamlined. In 1987, a new season of this show started. Like how the Japanese release of the small pickup truck revolutionized the appearance of the industry, this new revolutionary line accomplished the same. It established standards for upcoming GMC pickup truck models [19]. A 1980 Ford series pickup model is shown in Figure 9.



Figure 9:Ford's F-Series Pickup Truck in 1980 [18]

As "the first new truck of the 1980s," Ford refers to its seventh generation F-series, which is built with an emphasis on aerodynamics and luxury interior embellishments. Ford offered 173,050 F-150s in 1980, while 133,590 consumers can still be found for the entry-level F-100. The 300-cubic-inch inline-six engine with a one-barrel carburetor and 117 horsepower and 223 pound-feet of torque that Ford is known for is still used by 73% of those people. At the end of the 1983 model year, the F-150 replaced the F-100 as the entry-level model in the F-series [18].

A small space for living while camping can be provided by fitting pickup trucks with camper shells. However, with slide-in truck campers, a pickup truck may have all the benefits of a compact RV while nevertheless providing the driver the ability to remove the trailer and utilize the vehicle independently. In some regions of Southeast Asia and Africa, Passengers get transported by pickup trucks. The majority of songthaews in Thailand are modified flatbeds and pickup trucks. Press taps are also modified pickup vehicles in Haiti [14].

The 1987 edition of the F-150 celebrated its 50th anniversary with a minor update that includes a new grille, flush headlamps, and rounded wheel arches. Power brakes, power steering, and rear anti-lock braking are now included as standard features. Fuel injection boosts the standard 300-cubic-inch six-cylinder's output to 145 horsepower and a robust 265 pound-feet of torque [18].



Figure 10: Hilux pickup Truck in 1987 [17]

The 5th-generation Hilux, which was released in September 1988, embodied the fundamental ideas of strength, durability, and comfort. The interior equipment was further improved to further enhance its functionality as a multi-purpose vehicle (called an RV or recreational vehicle in Japan at the time). There was just one basic body variety left, and it had pressed doors with built-in window sashes. A 1987 Hilux pickup truck model is shown in Figure 10. Wheel arch flares were a key distinction between rear-wheel-drive (RWD) and four-wheel-drive (4WD) vehicles: RWD versions had fenders without flares, while 4WD models had robust-looking over fenders with flares [17].

The Ford F-Series, which has been produced continuously since 1948, had a big decade in the 1990s; the 8th generation was produced from 1987 to 1991, and the 9th generation from 1992 to 1996. For the first time since 1980, the tenth-generation 1997 Ford F150 experienced a comprehensive redesign. Larger interior space, improved fuel efficiency, and superior aerodynamics were among the updates. Dodge also launched their new Ram pick-ups in 1993; also called the T-300, the Dodge Ram featured a more spacious cab with extra interior room and storage [16]. A 1990 Ford single-cab model is shown in Figure 11.



Figure 11:Ford Single cab Pickup Truck in 1990s [20]

Similar to a dragon and its hoard of gold, dependability and a pickup truck fit hand in hand. After all, it's one of the main things that attract customers to truck dealerships in the first place. Pickups are the essentials of the present day, so it only makes logical that they should be able to withstand any punishment to them. To do this, they need to be reliable as well as have a lot of power [20] However, in the 2000s, GMC launched Duramax diesel engines in the Sierra HD. By releasing the GMC Sierra 1500 Denali pickup in 2007, GMC further extended to carve out a niche in the luxury pickup industry. For luxury buyers, the Denali featured a crew cab, a deep bumper cover, and a big metal grille. Ford unveiled the 12th generation of F150s, which will be produced between 2009 and 2014. Ford upgraded the full-size truck platform for the series, introduced distinctive Super Duty grilles and headlamps, shrunk regular cabins to have only two doors rather than four, and did away with the manual transmission [16].

2.2 Pickup Truck Aerodynamics

Although pickup trucks are the most common vehicles on the market. Today, pickup trucks have attracted little consideration in the literature on car aerodynamics. Whenever a vehicle is moving numerous forces, including drag force and downforce are exerted on the vehicle. The greater drag force is more accountable for a vehicle's higher fuel consumption and reduced maximum speed [21]. Even though today pickup trucks represent a significant portion of the overhaul vehicle market, few studies are concerned with aerodynamics in the available literature. Due to rising costs and fuel as well as the fact that pickup trucks have higher drag coefficients than Sport Utility Vehicles (SPV) and sedan cars [22]. Concerns regarded vehicle fuel economy have developed as a result of the increased incidence in gasoline prices. Today one of the most crucial aspects of fuel consumption is vehicle aerodynamics since it is possible to increase fuel efficiency during the results of aerodynamics drag [2].

Pickup trucks are a category of vehicles that are capable of carrying loads ideal for off-road enthusiasts due to their unique qualities for such a type of road surface. while moving on a road has an impact on the vehicle [23]. The basic shape of a pickup truck with a huge frontal area does not seem to be the most advantageous among vehicles in terms of aerodynamics which is why pickup trucks have difficulty especially when compared to their emissions [24]. The last few decades have seen significant improvements in automotive aerodynamics with a focus on reducing drag. Most existing approaches to minimize vehicle drag on commercial road vehicles and aerodynamic developments on automobiles during the last decades have focused on drag reduction. Most existing approaches to minimize vehicle drag on pickup vehicles [25].

A flow field around a pickup truck or any Sport Utility Vehicle (SUV) indicates that the bed geometry is one of the most significant components of a pickup truck in consideration of aerodynamic drag [3]. Each manufacturer attempts a balance between performance, safety, and design. The solution for better vehicle aerodynamics is finding an appropriate consideration between those areas which is directly influenced by the economic and productive factors. Constant development of the producers is necessary due to the variety of models, and complexity of the flow associated with each other. Movements of human occupation, and transportation requirements [2].

Around motor vehicles, the flow field is extremely separated unstable, turbulent, and three-dimensional. A car with a bluff body interrupted the flows in the back, causing losses in pressure recovery and a vortex to form in the wake are the most contributors of aerodynamic drag [26]. Pickup truck reduction in weight and engine modification are less efficient and effective than reducing aerodynamic drag, such as pressure and skin friction drag, by modifying the pickup truck shape and flow, aerodynamic drag can be reduced [27]. The conventional applications Using computational fluid dynamics (CFD) to vehicle aerodynamic wake flow structures has involved the creation and verification of models, a generic understanding of flow behavior to interpret results from Computational Fluid Dynamics (CFD) in the wake flow structure. More recently, the range of CFD has increased beyond the possibility of providing a prospective design tool. These simulations are becoming more common throughout the design cycle of the majority of significant vehicle developments globally. A comprehensive review of the Computational Fluid Dynamics to model the vehicle exterior aerodynamics [28].

Road vehicle aerodynamics is the most common area of implementation in Computational Fluid Dynamics (CFD) was developed on the importance of application. Due to the growing competitive and regulatory pressure being experienced by the automotive industry, this trend utilizes CFD to concerns related to vehicle aerodynamic design is appropriate and relevant. The establishment of an aerodynamic flow model takes place in an environment where errors in turbulence modeling and numerical computations are present.

A thorough analysis of the CFD results and measurements of the aerodynamic flow structure over the Bishoftu Plutus double cabin pickup truck geometry is conducted to determine the accuracy of the Computational Fluid Dynamics flow [29]. Aerodynamic drag in pick-up-type vehicles is one of the main issues concerning their high fuel consumption. External devices attached to their body frame, such as vortex generators, have been developed to reduce the drag. The aerodynamic resistance of the vehicle can be controlled if the values of the drag coefficient of the vehicle change. External devices, such as vortex generators (VG's), allow to reduce drag and can be attached to the vehicle without modifying the body frame [30].

Bishoftu Plutus Double Cabin Pickup truck is a representation of a multi-purpose vehicle in Ethiopia culture that can be used for passenger transport, agriculture, also in law enforcement, the military, and fire services since it offers a better towing capacity. Pickup trucks are more

durable than most other vehicle types. It is easy to transport just about anything with a pickup truck due to its rear bed which has made it unique, compared to other road vehicles. The rear beds are separated from the cab and used for carrying and pulling heavier loads than any other type of passenger vehicle [3].

Although the size of the pickup truck is smaller than the usual truck, it is also necessary to pay attention to the efficiency of fuel consumption, safety, and comfort when using a pickup truck. This can be influenced by the aerodynamic factors of the vehicle [31]. While the rear box has the advantage of giving space for transporting goods and pieces of stuff, at the same time it is responsible for the low fuel efficiency of the pickup trucks. The recirculation flows over the bed and the reverse flow in the wake have significant effects on the aerodynamic drag. The vortex inside the bed is a recirculation flow, which is detached from the rear edge of the roof and enters into the rear box cavity [3].

Along with the public's interest in pickup trucks, many people change the shape without knowing the feasibility of a pickup truck, such as changing the bumper, roof, the body cover. This can affect the aerodynamics of the pickup truck. A study states that controlling aerodynamic forces on a pickup vehicle can also improve the performance of the vehicle [32]. Controlling aerodynamic forces on a pickup vehicle can also improve the performance of the vehicle. The aerodynamic body shape of the pickup truck can reduce drag engine power to become the driving force for vehicle traction, maintaining vehicle stability, and saving fuel [31].

The vehicle body design is designed by considering various aerodynamic aspects, such as drag and lift forces that affect the pressure, speed, and coefficient of drag generated by the vehicle body. The aerodynamic aspect of a vehicle is one of the most important parameters in automotive design because it relates to the emergence of drag on the vehicle and will affect the amount of electricity or fuel consumption used, the stability of the vehicle direction, and free flow dynamic pressure. and vehicle surface area [32]. One of the aerodynamic characteristics is the airflow pattern that occurs due to the vehicle body blocking the airflow so that there will be a change in pressure. Airflow patterns are influenced by the shape of the pickup truck body itself [31]. However, in pick-up-type vehicles, the aerodynamic drag is higher due to the shape of the body frame.

The formation of low-pressure zones in the box and behind the box drives a significant increase in drag. Several studies have focused on the add-on devices to reduce drag resistance in this type of vehicle [30]. The understanding of the flow field over pickup trucks is a lot less than the understanding of sedans or SUVs for which a lot of effort has been spent on the aerodynamics [33].

The flow of moving air relative to the car body has a direct effect on the stability, driving characteristics, operation, fuel consumption, and safety of a car. The aerodynamic force plays an important role in the performance and stability of the pickup truck when it takes high speed [34]. The analysis of vehicle aerodynamics using computational methods has been widely utilized by automobile manufacturer as the reference to their bodywork design to reduce drag and increase the downforce for stability during maneuver [35].

As the relevant literature survey has revealed, the rear bed is one of the most important parts of a pickup truck in terms of aerodynamic drag and it is aimed to investigate the aerodynamic performances of some modified pickup trucks by changing their bed designs in terms of pressure contours, flow structure and other non-dimensional numbers such as friction, pressure and drag coefficients [8]. The streamline of an external flow surrounding a stationary vehicle is shown in Figure 12. The boundary layer is a small layer in which the fluid's viscous effects are contained while the speed of the vehicle has a limited distribution. The adhesive flow takes place outside the boundary layer. The boundary layer is under pressure due to this fluid flow. The fluid separates when the air reaches the vehicle's rear. The viscous effects of the fluid control the movement of the fluid within the boundary layer [36].

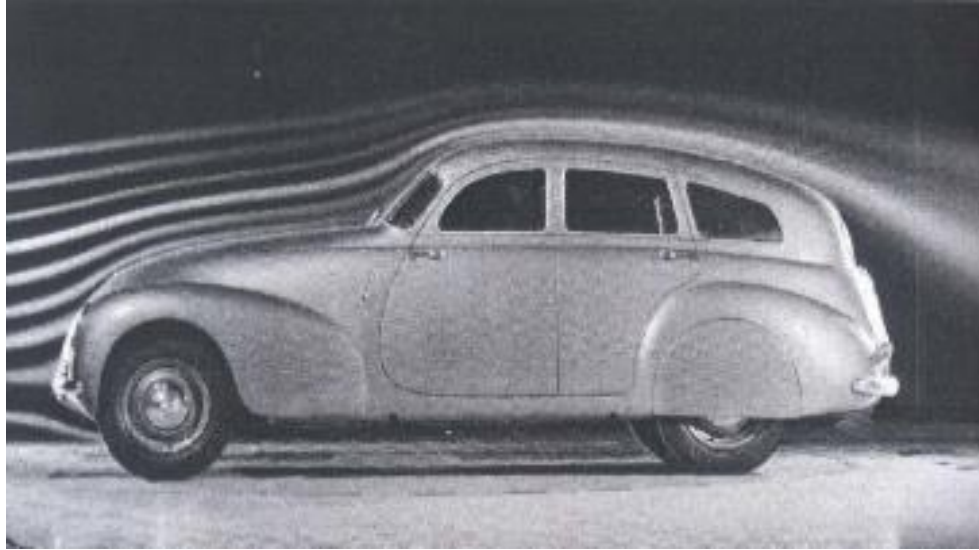


Figure 12: External Flow Field over the Car Body [36]

In the present-day automobile industry, understanding the flow pattern of mechanical vehicles, such as a pickup truck is important. An open trunk creates a special flow topology that is of particular importance for manufacturing investigations. Furthermore, academic works are now being created. In the current market, improving pickup truck's aerodynamic performance is a significant approach [37]. Most studies that analyze the exterior flow field of moving objects conduct thus to assess the aerodynamic drag of pickup vehicles [38]. When utilized in a positive direction, the vertical force pressing on the body lifts the vehicle into the air. If the force applies in a negative direction, however, it may generate an excessive amount of wheel-down force. Nevertheless, to prevent too much downforce or lift, the researchers work to keep this value within the specified limit. Despite the wide range of tasks that aerodynamics can do, also this study focused on external add-on devices that influence the flow of around double cabin pickup vehicle body and to decrease the resistance of the vehicle under typical operating conditions [36]. Pickup truck bodies are primarily created to meet functional, economical, and visual requirements. The body form selected to suit the functional, financial, and aesthetic design limitations is frequently based on the results of aerodynamic drag. Despite considering the vehicle's fundamental design elements, this can pick-up trucks' aerodynamic drag can be reduced by adding modifications to the roof end of the car., and it is an important factor in examining the flow of a pickup truck [10].

The investigation of conducting aerodynamic analysis of the Bishoftu plutus double cabin pickup truck with vortex generators is one of the most important goals of this study. The primary objective of this research is to implement various types of vortex generator designs to reduce the (BPV) aerodynamic drag and better understand pickup truck's aerodynamics. Due to their rectangular bodies, trucks have significantly higher aerodynamic drag than other ground vehicles, making it necessary to reduce drag in pickup trucks since it is directly related to fuel consumption. Figure 13, demonstrates For a pickup truck with a combination at highway cruising speed [39].

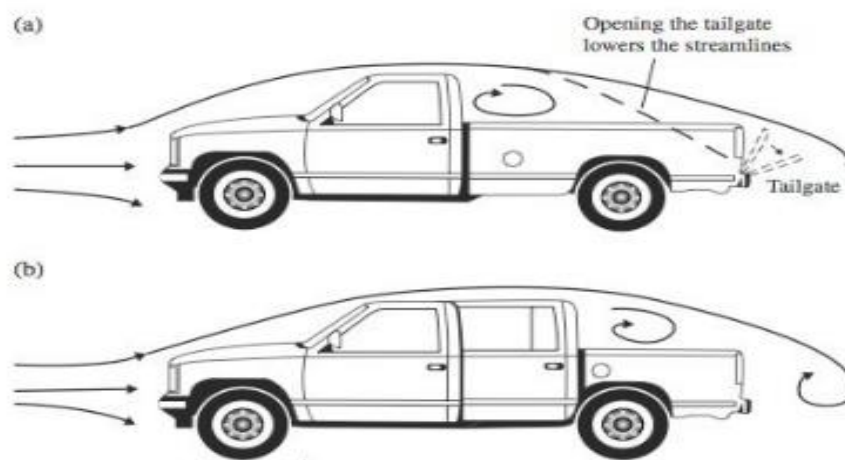


Figure 13: Flow field above a single (a) and extended cabin (b) pickup truck [39]

The impacts of the pickup truck configurations on the flow structure surrounding the vehicle and on the produced aerodynamic drag force were numerically evaluated. The majority of findings suggested that the aerodynamic drag is controlled by an adequate separation space behind the cabin [40]. For the Bishoftu Plutus Double Cabin Pickup truck to get better fuel economy, which leads to fewer emissions, aerodynamic drag reduction is crucial. However, adding drag-reduction devices on the pickup's roof will allow you to reduce the aerodynamic drag of a truck [41]. Due to the impact of flow from the bottom, top, and sides of the body, ground vehicles' aerodynamics is difficult. The open trunk of pickup trucks, which is simultaneously impacted by all three of its individual components, creates a divergence in the study of pickups [42].

The system of terminology indicated in Figure 14 is going to be used to provide the following description and the remaining discussions in this text. The front end of the car is its frontal portion. The distance between the front end and the frontal wheelbase is known as the front overhang, while the space between the rear of the pickup and the rear wheelbase, makes up the underbody. The hood, front windshield, and superior surface of the cabin are considered to be parts of the cab. The specified trunk takes into consideration the tailgate, interior (bed), and lateral walls of the trunk as well as the back surface of the cabin. The pickup's overall length (L), bed length (l), and tailgate height (h) are the reference measurements.

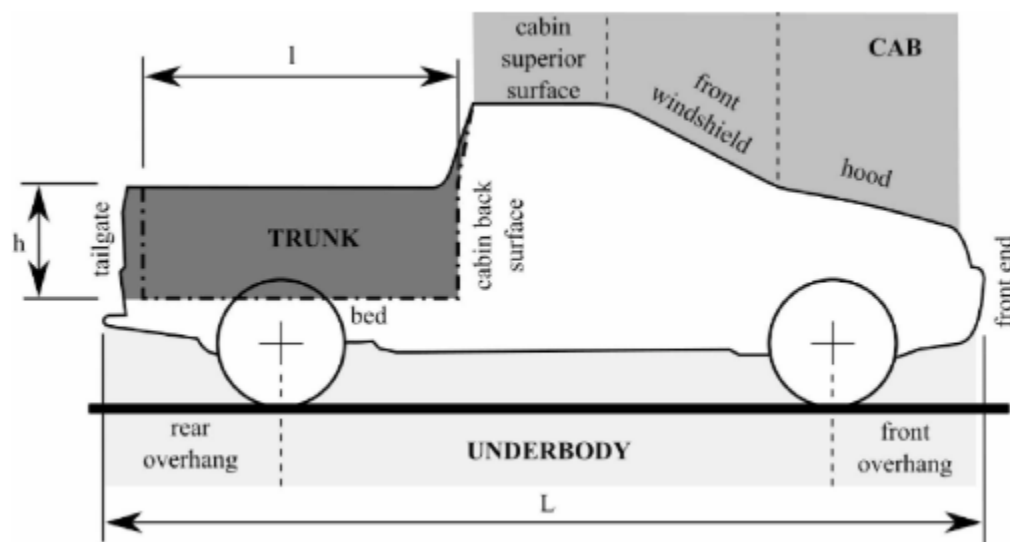


Figure 14: Proposed nomenclature for pick-up parts [42]

A large recirculation zone is created inside the luggage compartment by a major sphere composed of air that has been diverted from the cabin's roof. Due to the angle, flow from the trunk's lateral also deviates to the bed, forming significant vertical structures both inside and outside the trunk.

According to a review of the appropriate literature, a pickup truck's bed is among its most crucial components concerning reducing aerodynamic drag. Therefore, the purpose of the current study is to examine the aerodynamic capabilities of several modified pickup trucks using the concepts of pressure contours, flow structure, and other non-dimensional quantities such as friction, pressure, and drag coefficients [8].

A highly turbulent, unstable, separated, and three-dimensional flow field surrounds road vehicles. Turbulent flows in the rear of a bluff body type vehicle, which cause losses in pressure recovery and the emergence of vortices in the wake are the main contributors to aerodynamic drag. [26]. Since the ideal parameters for vehicle shape and dimensions are well understood, the aerodynamic performance of pickup trucks introduced over the last five years is well optimized for the majority of vehicle models [10].

As a result, Steady State was done using the Reynolds averaged Navier-Stokes equations (RANS) method. The realizable k-model was used to model turbulence because of its success in forecasting fluid flow in related studies from the literature [30].

The drag, lift, side forces, pitching, diving, and rolling moments are some of the aerodynamic forces at work on the vehicle. The geometry has to be changed or redesigned. It is expected that by improving or rebuilding the geometry, it will be possible to provide a smaller drag force, enabling the vehicle to move steadily while consuming less fuel [32].

The body shape is designed to produce the best possible aerodynamic properties. Utilizing a wind tunnel and the computational fluid dynamic (CFD) approach, it is possible to examine the aerodynamics of a vehicle. Free stream static pressure, free stream dynamic pressure, static pressure on the contours of the vehicle model, pressure coefficient, and airflow pattern are the dependent variables in this study [30].

2.3 Drag

Drag is a force that is applied to a surface by a flow that opposes the motion of a moving vehicle. Overall surfaces exposed to the stream, the drag is the sum of the local stream-wise components of the normal (pressure) and tangential (skin friction) surface forces. Direct evaluation of drag involves the integration of the pressure distribution throughout the multidimensional surface of the vehicle and accurate measurement of the pressure distribution, a combination that is extremely difficult to obtain. However, after the CFD setup has been sufficiently validated, detailed surface pressure distribution for a flow over an object can be easily obtained [13].

Forces are applied to the surface of an object when it is placed in a flow or when it is moving through a fluid. With appropriate limitations, researchers can compare these forces' attempts to slow down motion as a result of shear stress, energy is lost. Heat energy is created from kinetic energy loss of the drag force [43]. The airflow's drag force on an automobile as shown in Figure 15



Figure 15: The drag force due to airflow acting on a car [44]

All objects that go through motion, whether it be internal or exterior, are impacted by drag, a fundamental force. Science benefits from comprehending the way drag influences items since most of the things that we utilize daily are affected by drag forces. [45].

Drag force (N) = F_d

Density of air (kg/m^3) = ρ

Frontal area (m^2) = A_f

The velocity of fluid flowing (m/s) = V

Coefficient of drag = C_d

Drag in fluid dynamics is a force that opposes an object's relative motion to the fluid it is moving relative to. Between the fluid and a solid surface, this exists. Drag forces depend on velocity, in contrast to other resistive forces that are almost independent of velocity [44].

Airflow over a vehicle determines drag forces, which in turn influence a vehicle's performance. Testing equipment has been designed to assess both the vertical and horizontal components of air resistance on a model car. The vertical component of the downforce is just negative [46].

The stability, driving qualities, operation, fuel economy, and safety of the vehicle are all directly impacted by the flow of moving air about the car body. When a vehicle is traveling at a high speed, the aerodynamic force is crucial to its performance and stability. On a vehicle, various aerodynamic forces such as side forces, lift and drag are created. When the vehicle travels at a high rate of speed, there is more drag [34]. Although a car will move more smoothly through the air if its drag coefficient is low, every manufacturer will make every effort to create a vehicle with the lowest feasible drag coefficient. Also, it enhances the ability to turn quickly while maintaining adequate traction on the road. The creation of favorable downward aerodynamic forces is essential, particularly for racing and high-performance vehicles [23]. Determining the forces and moments acting on an object requires understanding how the air is moving around it. Velocity, pressure, density, and temperature are common parameters that are determined for a flow field as functions of position and time. Equations for the conservation of mass, momentum, and energy can be created by specifying a control volume surrounding the field of flow, and then used to solve for the properties [36]. External and internal aerodynamics form the two main sub-categories of aerodynamics and its analysis. The study of flow around solid objects with different shapes is known as external aerodynamics.

Some instances of external aerodynamics include analyzing the lift and drag on an airplane, the waves of impact that are generated in front of a rocket's nose, or the flow of air over a wind turbine blade. Internal aerodynamics, on the other hand, is the study of flow via perforations in solid objects. In this case, the study of the airflow through a jet engine, an air conditioning pipe, and other internal flow conditions falls under the broad heading of internal aerodynamics [36].

Along with the general interest in pickup trucks, numerous individuals change the shape of the vehicle by changing the bumper, roof, and body cover without considering when similar changes are feasible. The pickup truck's aerodynamics might be impacted by this. According to a study, reducing aerodynamic pressures on a pickup truck can improve the vehicle's performance [32].

External devices, such as spoilers and vortex generators (VGs), enable reducing drag and can be mounted to the vehicle without modifying the body structure, permitting the driver to regulate the vehicle's aerodynamic performance if the values of the drag coefficient of the vehicle are different. However, as a consequence of the design of the body frame, pick-up-style vehicles have higher aerodynamic drag. A large increase in drag is caused by the formation of low-pressure areas inside and behind the box [30].

To understand the lift and drag coefficients occurring on vehicles, several researchers have utilized both experimental and/or computational methods to investigate the flow field surrounding the vehicle [47].

2.4 Lift Force

The proportion of aerodynamic force that is perpendicular to the flow stream velocity is known as aerodynamic lift. It is mostly caused by the pressure difference between a vehicle's top and bottom surfaces. Driving stability is strongly influenced by aerodynamic lift, and it is essential to avoid harming it to keep the vehicle stable. The wheels of the vehicle will have less traction force with the road if the aerodynamic lift increases too much. This will make the vehicle more unstable and enhance the possibility of a rollover [48]. Drag and lift force are illustrated in Figure 16.

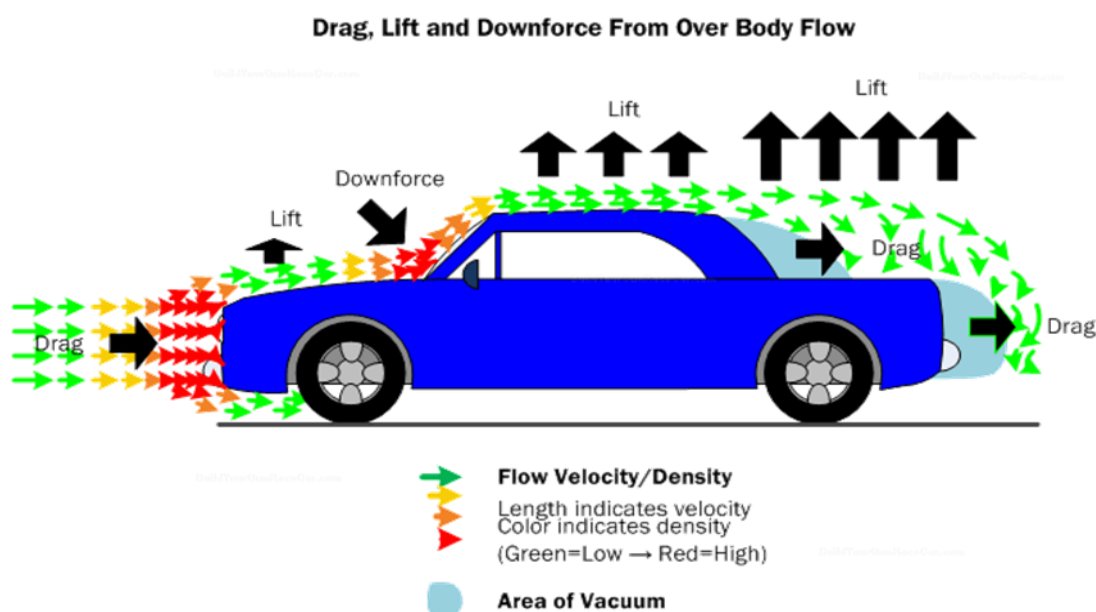


Figure 16: Drag, Lift, and downforce from over-body flow [48]

The sum of the components of the pressure and wall shear forces acting in the direction perpendicular to the flow tends to move the body in that direction. The body could rotate as a result of the fluid forces creating moments. a two-dimensional body's resistance to pressure and viscous forces, as well as the resulting lift and drag forces [49].

2.5 Computational Fluid Dynamics

Recently, a numerical method such as Computational Fluid Dynamics (CFD) has been conducted to investigate the airflow characteristics around the object and then to predict the coefficient of drag of the car [50]. The complexity of the flow makes drag prediction tools, including CFD-based methods, unreliable. The main goal of this thesis work is to conduct a better understanding of pickup vehicle aerodynamics analysis using detailed flow field measurements.

Computational Fluid Dynamics (CFD) has gained popularity as a tool for many airflow situations including road vehicle aerodynamics. This trend, to identify CFD on vehicle aerodynamic design issues, is appropriate and timely because of the increasingly competitive and regulative pressures being faced by the automotive industry [51].

CFD is a technique under fluid dynamics used to obtain real flow simulation with the numerical solution, obtained by solving governing equations i.e. Navier- Stokes equations. For some practical applications, solving flow governing equations became very complicated and sometimes impossible using analytical methods. CFD techniques replace the analytical method i.e. governing partial differential equations with algebraic equations, which is very easy to solve with the help of computers also testing conditions is very much easier with the help of computational techniques while it is extremely difficult with the experimental and analytical methods. Thus, CFD is the best and cheapest technique for testing fluid flows compared to experimental and analytical methods [52].

While experimental measurements typically need either scaling up or down, CFD simulations also offer flow solutions at the real dimensions of the engineering systems with the actual operating circumstances. Most of the time, it is not possible to cheaply show realistic situations, hence results must be extrapolated. In CFD simulations, this particular problem is removed [53]. In general, the influence of lift is regarded as a comfort requirement in that an automobile appears more pleasant to drive when it has good lift characteristics.

The influence can, however, occasionally become a safety concern. Cars have gradually incorporated aerodynamically improved under flooring, mostly to reduce drag. Lift is also decreased, however, the distribution of lift is determined by the specific floor design, and when the motor compartment is compressed, cooling airflow significantly affects lift [54].

2.6 Advantages behind the reduced drag of a vehicle

Extremely equivalent to wading through an endless sea of atmosphere while driving an automobile. The last few years have seen a significant increase in demand for automotive manufacturers to find workable solutions to this issue due to the declining quality of the air and the scarcity of natural resources, particularly oil. High-speed cars used to be completely dependent on an engine's horsepower to sustain the performance portion of the vehicle. However, subsequent developments have seen design engineers adopt aerodynamic principles to increase the vehicle's efficiency. Approximately half of the energy used by the vehicle is utilized by fuel consumption as a result of aerodynamic drag. As a consequence, one of the key strategies used by manufacturers is to reduce drag. Optimizing for low drag becomes a crucial step in the design process and can be achieved by shaping the vehicle's body and adding several add-on components. Drag force is primarily influenced by the body's speed, frontal area, and coefficient of drag [55].

The effective frontal surface area of the vehicle, the vehicle's shape, and the speed at which the vehicle is moving are all factors that have an impact on how much energy is needed to overcome aerodynamic drag. Lower drag results from a vehicle having a smaller frontal surface area. A vehicle with a rounded or stream-lined shape will also have lower aerodynamic drag since there will be a lower degree of airflow separation (which is a smaller wake). The rectangular shape of the vehicle's speed, distance traveled, and specific fuel consumption determines how much power is needed to overcome aerodynamic drag [56]. Nowadays vehicle designers in the automotive business are under a lot of pressure to develop a vehicle that meets requirements for performance, fuel efficiency, aerodynamic shape, etc. Manufacturers thus want to discover new methods to acquire knowledge about the latest developments that reduce fuel consumption and improve vehicle economy. Different techniques are utilized to reduce aerodynamic drag to manage the flow separation at the back.

These methods, which include modifying the design of the vehicle or adding components to reduce aerodynamic drag, seem to be the simplest to put into procedure, although they should only be used for specialized applications [57].

2.7 Drag Reduction of Pickup Vehicle by Vortex Generators

To save energy and to protect the global environment, fuel consumption reduction is the primary concern of the automotive industry. In vehicle body development, reducing the aerodynamic force of a pickup vehicle is crucial for enhancing driving performance, fuel economy, and vehicle stability. If the vehicle's aerodynamically improved body is also visually appealing, this will further increase the car's attractiveness to potential users [58]. Drag can change in certain velocity regimes or crosswind scenarios. Reducing the vehicle drag coefficient can be done by modifying its shape or applying active or passive flow control systems. Even with controlled movement, platooning drag, and fuel consumption reduction can be achieved [59].

Applying add-on devices as principles of aerodynamics, it is possible to reduce the coefficient of drag (CD) of a vehicle and thus contribute to its fuel efficiency. Most of the research on the benefits of VGs has been conducted in the field of aviation. Some research has also proven that Vortex generators can produce measurable improvements in drag reduction on vehicles. In addition to having the potential to reduce the coefficient of drag CD on the pickup vehicle, there are other benefits to using VGs on a pickup vehicle [60].

To decrease the aerodynamic drag on a vehicle, the sources of drag must be analyzed. As mentioned previously, aerodynamic drag is the force that opposes the direction of thrust of a car and is not desirable. Given a set of pickup vehicle conditions, the drag force can be calculated. Drag is a function of the frontal area of the vehicle, the density of the air, the coefficient of drag of the vehicle, and the vehicle speed squared [61].

The purpose of adding VGs is to supply the momentum from the higher region to the lower region. This allows the separation point to shift further downstream. So it enables the expanded airflow to persist proportionately longer. It reduces drag increasing the back pressure of the vehicle body. It provides a dual advantage in drag reduction [6].

One of the important targets of the boundary layer control is to delay or eliminate the flow separation concerning overall drag reduction, or better yet, Lift/Drag ratio improvement. To decrease the drag, one must act on the nature of the airflow in the near wall [62].

In pickup vehicles, body development reduction of aerodynamic drag is essential for improving fuel consumption and if an aerodynamically well-designed body is also aesthetically attractive, it will contribute much to increasing the vehicle's appeal to customers [63]. Three factors contribute to the aerodynamic drag: the shape of the vehicle, the cross-sectional area perpendicular to the free stream, and the speed of the vehicle [64]. The design criteria of add-on devices are only limited to aerodynamic considerations. Car drivers usually install add-on devices that successfully reduce the lift and improve traction leading to maneuverability and stability. However, the aerodynamic performance corresponding to the add-on devices could deteriorate severely [65].

Controlling the flow separation; a significant increase in system performance can be obtained resulting in energy conservation as well as weight and space savings. A correct understanding of the physical phenomenon that connects a car body with the related aerodynamic forces can provide a breakthrough for the better design of a car body [66]. Although the purpose of implementing VGs to control flow separation at the pickup truck's roof end is quite similar to utilizing VGs on aircraft [67].

One of the important targets of boundary-layer control is to delay or eliminate the flow separation concerning overall drag reduction, or better yet, Lift/Drag ratio improvement. To decrease the drag, one must act on the nature of the airflow in the near wall [62]. The modern-day design of vehicles, especially in the automobile industry involves a great deal of aerodynamic design study to analyze the airflow. The aerodynamic drag force adversely affects the forward motion of the vehicle, which in turn reduces the efficiency. If the vehicle is redesigned to optimize the aerodynamic forces, it could produce better results but requires a huge capital to change the complete design. Here in this paper, we are going to use these vortex generators for sedan-modeled lightweight compact cars with various profiled designs and CFD [68]. For any high-performance vehicle, the aerodynamic properties are significant when attempting to optimize performance. One of the main sources of aerodynamic drag is the flow separation which occurs near the vehicle's rear end [63].

2.8 Passive Flow Control System

Passive flow control is considered a way to improve the aerodynamic performance of a pickup vehicle [62]. A large contribution to the aerodynamic drag of a vehicle arises from the failure to fully recover pressure in the wake region, especially on square back configurations. A degree of base pressure recovery can be achieved through careful shape optimization, but the principle of automotive aerodynamics to implement significant shape changes is limited by a variety of additional factors such as styling, ergonomics, and loading capacity. passive flow control technologies present the potential to create flow field modifications without the need for external shape changes [69]. The low-pressure zone behind the car minimizes the effect of negative pressure with the application of passive flow control. It was observed that with a decrease in the exhaust velocity the coefficient of drag also was found to reduce [21]. The passive control systems consist of the use of more or less discrete obstacles, added around or on the roof of the vehicle. They can be divided into two groups according to their influence on flow control. The first group consists of obstacles determined on the surface of the geometry. The second group consists of the obstacles positioned upstream or downstream of the geometry to be controlled. A VG is an aerodynamic device and part of a passive flow control system It is a little vortex-producing vane [70].

Vortex generators are widely used in the aerospace industry, mainly to control boundary layer transition and to delay flow separations and CFD analysis confirms that the use of VGs reduces both the drag and lift coefficient [69]. During the flow over the surface of the vehicle, there are some points when the change in velocity comes to stall and the fluid starts flowing in the reverse direction. This phenomenon is called the 'Separation' of the fluid flow. This usually occurs at the rear part of the vehicle. This separation is mostly dependent on the pressure distribution, which is imposed by the outer layer of the flow [71]. Numerical simulation on a pickup vehicle to optimize the aerodynamic drag attaching passive flow control Vortex generator device. this helps in turbulent airflow around the top of the vehicle thereby delaying the separation point and hence obtaining a smaller wake and less drag force result [21]. However, this thesis work aims to develop an actively using vortex generator device to reduce the aerodynamic drag of the pickup vehicle.

2.9 The Boundary Layer over a pickup truck

Initially, a specific circumstance will be investigated. A flat plate is fixed and stationary. This flat plate is passed over by the fluid, which could include any fluid. The initial characteristics of the fluid are determined by the region that does not contain a plate. Once the fluid has reached the plate, it interfaces with the boundary condition and the flowing fluid. Although the scenario is three-dimensional, it will be described as being two-dimensional because of the fluid's characteristics. Consider adjusting in a direction that is opposite to the way the fluid is flowing. The resulting "no-slip" condition is the most significant aspect of the fluid-wall interactions. The suitable position for the Vortex generator line is upstream from where the boundary layer naturally separates. When the VGs are placed downstream of the natural separation line, a large drag reduction is also obtained. The best boundary layer enhancement of disturbances is closely linked to its stability characteristics. Determining the correct fluctuations that the boundary layer will enhance and then properly modifying the boundary layer's properties to control the separated flow are indeed the two difficulties that must be resolved when the purpose is to control a separated flow [72]. Figure illustrating the velocity profile around the back end as shown in Figure 17.

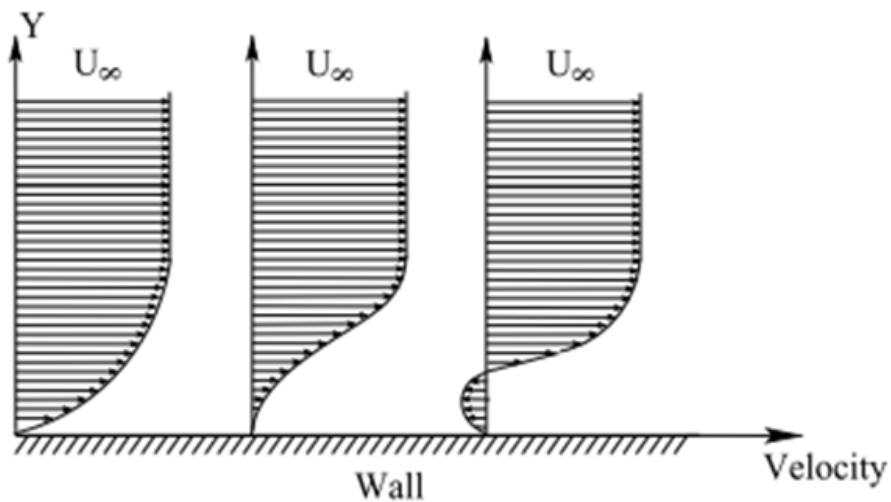


Figure 17: Schematic of velocity profile around rear end [6]

The wall shear stress increases as momentum is transferred toward the body, or when the velocity in the inner zone is increased, limiting the thickness of the boundary layer. This usually occurs when the boundary layer is close to the wall.

The vertical shear stress for turbulent boundary layers is negligible or nearly zero. Additionally, demonstrated how sub-boundary layer vortex generators can reduce drag and improve lift [73].

2.10 Boundary Layer Transition

Laminar flow is advantageous because it decreases friction drag, but it is disadvantageous since it does not carry much momentum and will consequently separate when the wing increases its angle of attack. A wing should minimize separation because it causes stalling. When the wing fails to produce lift, the situation is considered to be in a stall. Although turbulent flow delivers more resistance to flow separation, it also causes higher friction drag [74]. From the top trailing tip of each vortex generator, a very small spinning vortex is produced that transfers momentum and energy into the boundary layer. A wing can be moved to a greater angle of attack without stalling mainly due to the turbulence that is produced by this energy, which delays separation [75]. Flow separation of vortex generator as shown in Figure 18.

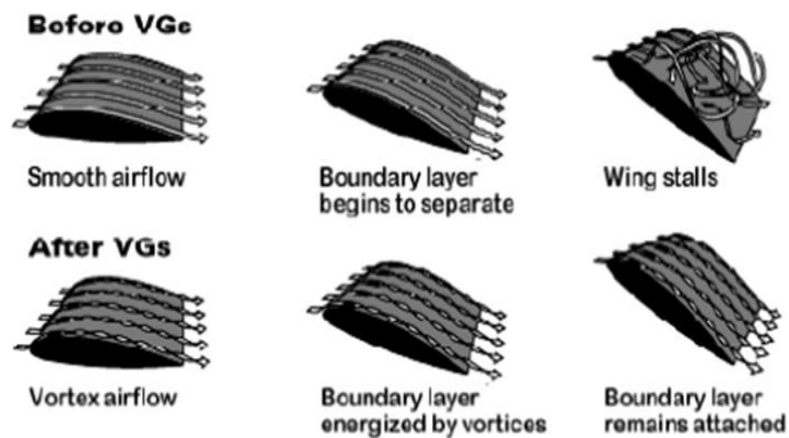


Figure 18: Function of Vortex Generators [74]

The boundary layer is a small layer in which the fluid's viscous effects are retained while the vehicle is moving at an unclaimed velocity. The fluid flow is visible outside the boundary layer. The boundary layer is under pressure due to this fluid flow. The fluid becomes separated when air passes the vehicle's rear. The turbulence of the fluid completely controls the fluid's motion within the boundary layer [76].

The Reynolds number is determined by the vehicle's characteristic length, kinematic viscosity, and speed. It appears that the fluid's ability to circulate the vehicle depends on both its shape and its Reynolds number. Another significant phenomenon also has an impact on the performance and flow of the vehicles. This phenomenon is generally referred to as the vehicle "Waking." The wake is a significant low-pressure turbulent region that is left behind a vehicle when the air flowing over it is separated at the back end. This wake helps to create pressure drag, which eventually affects the performance of the vehicle [77]. One of the most popular systems is the categorization of boundary layer control techniques into active and passive groups. Among the many devices used for the passive flow control technique are the vortex generator, dispersed roughness, streamlining, and uniform blowing and suction. On the other hand, a few active flow control techniques include surface element movement, periodic blowing and suction, heated walls, and synthetic jets [78].

There is a condition in which neither is dominant, between the two extremes of active ideal flow at high Reynolds numbers and slow creeping flow at low ones. With a high Reynolds number, the flow will be almost perfect in most areas, except for those close to solid limits, where the no-slip condition and the necessity demand that the flow rate of the fluid corresponds with the wall's movement. Transition layers are going to develop here since Rapid fluctuations in stream velocity are seen in the wall's velocity to the fluid's overall velocity. When compared to the radius of curvature of the solid walls, boundary layers are usually small.

The flow characteristics in a boundary layer consequently shift from perfect far from the boundary to advancing close to it. These transition zones usually become related to the most interesting and challenging phenomena. However, humans are usually oblivious to their surroundings since they live in almost perfect flows of air and water with boundary layers that are only millimeters thick and Reynolds numbers in the millions. Smaller species struggling to survive at the surface of a Riverstone might be far more conscious of the fancies of boundary layer physics and how they might affect their internal organ systems and body forms [79].

2.11 Separation

There is a region in the flow across the vehicle's surface when the velocity change occurs, and the fluid through the reverse direction. The fluid flow phenomenon concern is known as "separation." and frequently occurs at the back of the vehicle. The pressure distribution produced by the flow's outer layer has a significant impact on this separation. Compared to laminar flow, the turbulent boundary layer can tolerate significantly higher pressure by separating [77]. This separation influences the flow's behavior behind the vehicle, which has an impact on the flow field surrounding it. When analyzing the vehicle's wake, this phenomenon should be taken into consideration as the main factor [76]. Flow separation of aerodynamic characteristics is shown in Figure 19.

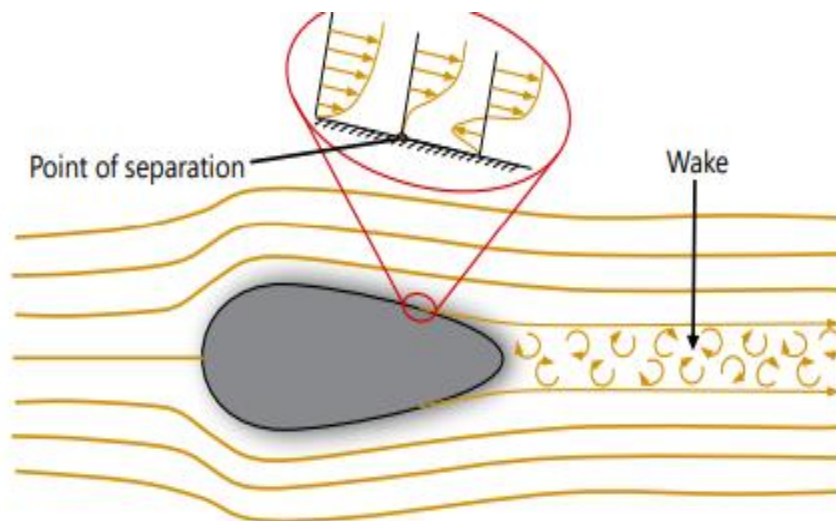


Figure 19: Flow separation and wake [80]

Boundary layer separation can be caused by significant changes in surface geometry or by a large adverse pressure gradient (APG). Examples of the latter usually appear in regions with strong curvatures or sharp edges, including bluff bodies (common truck designs) and backward-facing steps. However, the probability of separation depends additionally on the local pressure gradient but also the local boundary layer state for strong adverse pressure gradient flows along flat or kindly curved surfaces [81]. In pickup trucks, boundary-layer flow separation is a common issue. The ejection of fluid from the boundary layer toward the flow away from the wall is known as flow separation.

This fluid ejection is linked to a flow back-flow zone. There are two kinds of factors for the flow separation: a significant negative pressure gradient and an important wall geometry curvature. [82].

The potential theory of flow explains many kinds of fluid mechanics events, however, it is insufficient for the majority of boundary layer circumstances, such as the flow field surrounding an airfoil. As long as the pressure gradient stays zero along the surface, the boundary layer over an extremely hard surface is going to keep expanding. The boundary layer thickens significantly under negative pressures. The boundary layer's momentum is decreased by the negative pressure and shear (viscous) forces; if any of these are effective across a significant portion of the surface, the boundary layer's formation is interrupted. Researchers refer to this phenomenon as separation. The drag force reduces to a minimal value if separation over an airfoil can be avoided because the boundary layer remains lightweight and the pressure drop in the trailing edge is minimized [78]. In pickup truck aerodynamics, the effects of flow separation are often unwanted and can have adverse consequences. Additional losses are caused by the flow separation from an energy perspective. For the work exchange, the energy used in flow separation is negligible [82].

2.12 Stream-wise Mean Flow Field

The previous study investigated the average pickup vehicle rear bed streamlines from various tests. The streamlined pattern outside the rear bed and the streamlined pattern inside the rear bed are similar. Although there is a recirculating flow area behind the cabin, there may not be one behind the tailgate. Instead, at the symmetry plane, the mean flow produces a very high downstream. The attached flow behind the rear bed creates a situation in which the mean pressure is higher on the exterior of the body rather than on the interior. As a result, there is less aerodynamic drag on the pickup vehicle's body due to the force flowing in the forward direction. According to the results of the cross-flow planes the downstream is thought to be induced by the presence of strong stream-wise vortex shedding behind the rear bed. These measurements for the rear bed flow indicate that the flow enters the rear bed in the back and exits around the cab base, causing a recalculation region in the front of the vehicle [83].

2.13 Pressure drag

Pickup vehicles, including truck bodies, have various drag characteristics. There is an exceptionally considerable pressure gradient on the rear of these vehicles, which causes the flow to separate in viscous flow. In comparison to the rear of the flow field, where flow separates and creates a strong pressure, the front of the flow field demonstrates high pressure values. Pressure drag is the term utilized to describe the force contribution produced by such a massive shift in pressure. The separation of the flow field and the height of the vehicle both have an impact on this factor [84].

2.14 Friction drag

Each wall surface or layer has a unique friction that prevents fluids from flowing through it. Every surface of the vehicle generates stress as a result of particle friction. This friction drag results from the integration of the corresponding force component in the flow velocity direction. Friction drag is one of the major causes of an overall drag if the separation does not occur [85].

2.15 Flow Fields and Pressure

The variation in pressure between the front and rear of the car ends is the main cause of drag for aerodynamic vehicles. More than 70% of the total drag originates from this, with the other resulting from viscous drag. Designers will examine the flow field and the regional pressure distributions to understand how the additional devices affect drag reduction. It was observed that the top plate of the pickup vehicle and the partial truck rear bed frame are the main causes of the drag coefficient. Any combination of these two components can result in a beneficial drag reduction [86].

2.16 Verification of Vortex generators mechanism

According to many studies, VGs have the effect of shifting the flow separation region's separation point downstream. Thus, the flow field was investigated to verify the accuracy of this prediction. High velocities are found when there is high overall pressure when Adding Vortex generators. However, a high-velocity region increases downward, indicating that the flow separation region is squeezed [87].

2.17 Design Considerations

The Following design considerations will be helpful for the drag reduction of a vehicle

- ✚ Aerodynamic drag is reduced by precisely designing the flow of streamlines.
- ✚ A vehicle's ideal drag coefficient is considered.
- ✚ The rear of the vehicle seems to have the greatest influence on total drag
- ✚ Designers aim to make displays pleasant and adjustable for better results

Most of the parameters were fixed, which reduced the degrees of freedom, according to analysis and sources from previous researchers. The delta wing and triangular-shaped vortex generators are selected. Triangular vortex generators are ideal for attaching to the body of a vehicle due to their simplicity and extensive use of the low-drag device rather than any other type. Aeronautical wings most commonly implemented delta-shaped vortex. The thickness of the boundaries was determined concerning height under the concept that the vortex's ideal height would be about equal to the thickness of the boundary layer [68].

2.18 Influence of the vortex generators on the aerodynamic forces

Since the optimum fluctuation can sometimes be predicted numerically or through a stability analysis, it is necessary to conduct to parametric investigation to determine the flow's sensitivity. The pickup vehicle body's top inclination is covered in vortex generators, resulting in a wavelength k (or span-wise spacing) of 0.02 m. (2mm) Measurements of drag and lift are performed for various curvilinear configurations of the Vortex generators on the top of the pickup vehicle body. The difference in drag and lift for the location of the vortex generator [72].

It is commonly recognized from previous studies that the distance between the vortex generators' 3D fluctuations has a significant impact on how efficient they are in producing forced flows. The ideal boundary layer enhancement of perturbation is closely related to its stability characteristics. Identifying the correct instabilities that will be magnified by the boundary layer is the first issue when trying to manage a separated flow. The second challenge is determining how to adjust the boundary layer's properties to control the separated flow. These are currently unresolved issues with a lot of research work. To find the critical spacing for a particular configuration of Vortex generators, parametric analyses remain the only viable solution [88]. Numerous experimental and numerical research works demonstrated the benefit of utilizing VGs to improve heat transmission. VGs with a delta form have been determined to be more effective per unit area than other types in early investigations. The highest performance is provided by the delta wing, and triangular shape Vortex generators [43].

Vortex generators (VG) are passive flow control devices that influence the boundary layer progress by generating free stream velocity or momentum transfer from upper layers into the inner flow area closer to the wall. By enhancing the lift coefficient, VGs attempt to delay or, in extreme situations, prevent the boundary layer from separating [89]. However, VG implementations have the possibility of increasing drag, which is terrible for flow control devices. The boundary layer separation starts when the component that is nearest to the wall or the leading edge of a blade reverses the flow direction. The boundary layer is reenergized and the separation is delayed as a result of momentum transfers that occur between the free stream velocity and the fluid near the wall as a result of the VGs' design. During this energy transfer, the boundary layer thickness decreases while the inner boundary layer (BL Velocity) increases, delaying the flow's separation. Additionally, it was found that the sub-boundary layer of the vortex generator induced an increase in lift and a reduction in drag force. Basic studies on the use of VGs as flow control devices on pickup trucks have produced effective flow separation control. Investigations were conducted on the stream-wise vortexes produced by a rectangular VG in a flat plate and the significant flow separation pressure gradient stream. To decide on the possibility of an exact recreation of the characteristics of the wake behind VGs using CFD simulations. Due to the installation of VGs, a massive increase in the maximum lift coefficient was noted [50].

As the impact angle increased, higher lift and drag coefficient values were observed than those reached in steady-state conditions. According to the use of VG, near-stall conditions show a significant increase in the lift coefficient. It was found that with various Vortex generator geometries, a normal stress boundary layer (BL) control was able to significantly reduce the total separation area. Similarly to this, the construction of sub-boundary layer vortex generators successfully delayed the separation [51]. A Vortex generator is a type of aerodynamic surface that can take the form of a rectangle, triangle, or delta wing shape that tends to stir up the fluid flow. Initially, the aircraft industry utilized VGs extensively, mostly to regulate boundary layer transition and delay flow separations. On road cars, many types of VG are used to control the flow over and under the vehicle [52]. The existing research investigated the highest drag reduction of 1.15% and lift reduction of 1.60% by the implication of a vortex generator at the rear end of the vehicle [85]. Furthermore, another literature investigated 0.5% of drag reduction result formed with the installation of vortex generators drag is reduced [52].

2.19 Summary

The literature presents the aerodynamic analysis of pickup vehicles as the result of the optimization of the flow field in the vortex generators. Fuel saving and environmental issues are the most important factors in the Automotive sector. However, reducing the drag coefficient of pickup vehicles is a significant concern for vehicle performance. Furthermore, momentum from a higher position to a lower position is a result of adding vortex generators at the roof end and the separation point moves further downstream. Previous studies claim that identifying the design selection of vortex generators before implementation is the first aspect of the design process. However, comparing triangular and delta shape vortex generators triangular type is more effective than delta wing shape for installing on the roof end of the pickup vehicle and delta wing shape vortex generator due to its simplicity and frequent usage. More literature's investigated how fixing the vortex generators on the center edge of the pickup vehicle is performed to reduce the aerodynamic drag force. The effects of vortex generators are applied in all vehicles with different application methods for the function of drag reduction. Finally, many literatures conclude using vortex generators with various shapes of design is considered the effective method for aerodynamic analysis and flow separation of the vehicle.

2.20 Research gap

Aerodynamics analysis of pickup vehicles by applying two types of vortex generators is described in the literature review. The following research gap was observed after reviewing

- ✚ The effect of delta and triangular-shaped vortex generators at the roof end of pickup vehicles with related comparison was not studied.
- ✚ The researchers did not consider the number of both delta and triangular-shaped vortex generators on the roof end of pickup vehicles with different configurations, or heights.
- ✚ The effect of drag and lift coefficient of the car model with delta-shaped and triangular-shaped and without vortex generators are not examined.
- ✚ The existing studies only focused on the use of delta and triangular-shaped vortex generators and only investigated the rear end of the pickup vehicle.
- ✚ The Fuel efficiency effect of pickup vehicles concerning the implication of both delta and triangular-shaped vortex generators is not employed in the existing studies.
- ✚ A comparison with an experimental test of the baseline model with delta and triangular-shaped vortex generators was not studied.
- ✚ Both delta and triangular-shaped vortex generators on the roof end of the pickup vehicle with different parameters in the combined method were not studied.

CHAPTER THREE

MATERIALS AND METHODOLOGY

This chapter explains the processes for this thesis work and the materials that were employed in the methodology used to work on the thesis.

3.1 Introduction

This chapter mainly focused on numerical modeling commercial CFD analysis packages. ANSYS Fluent should be used for this investigation and SOLIDWORKS 2015 was used to develop the model of the Bishoftu Plutus double-cabin pickup truck. The flow over a pickup truck and, its coefficient of drag was thoroughly studied using ANSYS Fluent tools. This paper uses CFD techniques to analyze the aerodynamics of the baseline Bishoftu double cabin pickup model as well as with and without vortex generators. However, this interstate pickup's exterior body can improved using aerodynamic technology to conduct experimental tests with an educational wind tunnel to examine the comparisons over the numerical approach [90].

3.2 Materials and Equipment

To attain the objective formulated for this work here the listed equipment and software were required and utilized.

PERSONAL COMPUTER: A laptop that has eight processors was required and utilized for the required iteration process.

SOLIDWORKS: Utilized to construct the model of baseline pickup truck and modified delta and triangular type vortex generators

ANSYS FLUENT: Utilized to simulate and analyze the airflow around various modeled configurations of the pickup vehicle and determine the lift and drag coefficients as well.

MICROSOFT EXCEL: Used in generating the graphs and charts using ANSYS output results as input data

WIND TUNNEL: To test the experimental validation of the baseline model

3.3 Methodology

In this study, numerical simulations of different model configurations are performed. By using the same method the results of the baseline simulation and the configurations of interest can be identified for simulation of the three new models.

For every scientific research, there should be a broad review of previous literature review, therefore intense works on the basic concept of aerodynamics, air flow around vehicles, drag, lift coefficient, and CFD analysis.

3.3.1 Model Specifications of Bishoftu SG Plutus Double Cabin Pickup

Table 1: Basic Dimension of Bishoftu Double Cabin Pickup

Plutus Double Cabin Pick up dimensions in mm	Dimensions
Length	5350
Width	1725
Height	1690

All the specified data was collected from the Federal Democratic Republic of Ethiopia Metals and engineering corporation Bishoftu automotive industry, and for more detail clarity and understanding of the model taken, is manufacturer specification data set is given in Table 1.

Table 2: Bishoftu pickup specification for SG Plutus Double Cabin Pick up 4JB1TI Model

		
Property	Unit	
Displacement	L	2.771
Rated power	hp/rpm	84/3600
Curb weight	Kg	1730
Outlined dimensions	Mm	5350x1725x1690
Cargo Box Dimension	Mm	1715×1455×476
Wheelbase	Mm	3380
Max. speed	km/h	120
Max. Torque	Nm (r/min)	
Approaching angle/departure angle	(°)	26/24
Seating capacity	(persons)	5
Average fuel consumption under limited conditions	l/km	7 /100
Drive Type		4x4

Bishoftu pickup specification for SG Plutus Double Cabin Pick up 4JB1TI Model all specific data set as given in Table 2.

3.3.2 Bishoftu Double Cabin Pickup Model

The approximate outer profile of the (Huangai Plutus) Double Cabin pickup car model without any aerodynamic devices by using 3D modeling software is shown in Figure 20.

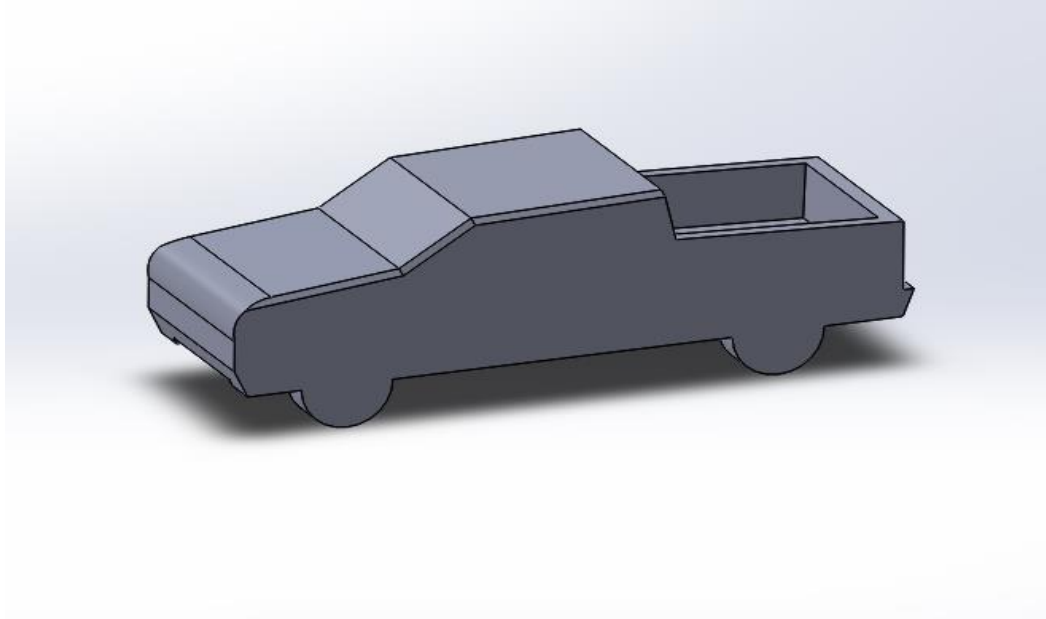


Figure 20: Bishoftu Double Cabin Pickup Model without Vortex Generator

The model represents a full-scale of the Bishotu Plutus Double Cabin pickup vehicle. Schematics of the pickup truck model with the pertinent dimensions are depicted in Figure 20. The baseline model of the Bishoftu double cabin pickup car is designed in SOLIDWORKS. The full size of the pickup vehicle is 5350 mm in length, 1725mm in width, and 1690mm in height. This model has been analyzed for drag coefficient and forces under the ANSYS (Fluent) module and values of drag coefficient and lift coefficient.

3.3.3 Vortex Generators

The vortex generator (VG) was introduced by Taylor in the 1950s. A vortex generator is an aerodynamic device that typically consists of a small vane attached to a lifting surface. Relative to the air, the vortex generator creates a vortex that prevents local flow separation and aerodynamic delaying by removing some of the slow-moving boundary layers from contact with the vehicle's surface [48]. Many factors, including fuel efficiency, aerodynamic characteristics, and aesthetic considerations, should be considered in the vehicle design. Recently, designers have been experimenting with various methods to reduce the vehicle's drag [52]. Throughout their development, vortex generators have been used in a wide range of applications. These involve flow separation delays on a variety of surfaces [91].

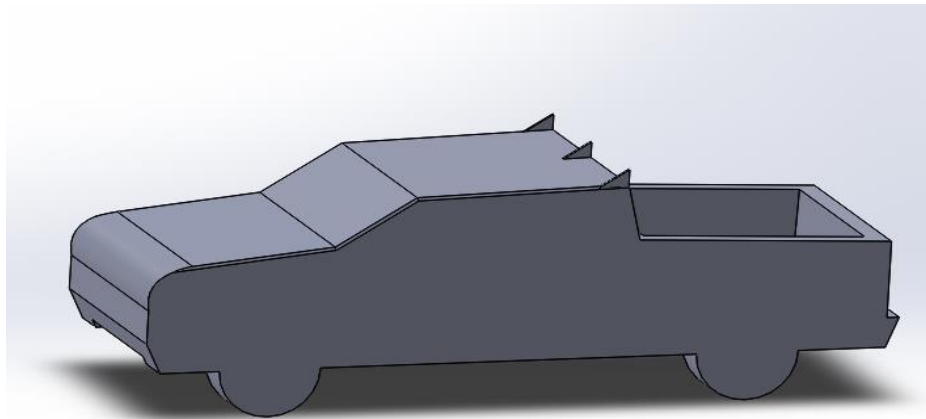


Figure 21: Bishoftu Pickup with delta-type vortex generator

The delta-wing-shaped Vortex generator is shown in Figure 21. The design of the Delta wing vortex generator is clear that as sizes go on decreasing of flow separation it gives better heat transfer and the drag coefficient increases. The verification has demonstrated that VGs produce stream-wise vortices, which mix the upper and lower regions of the boundary layer and cause the flow separation point to transfer downstream, resulting in a reduction of the separation region [61].

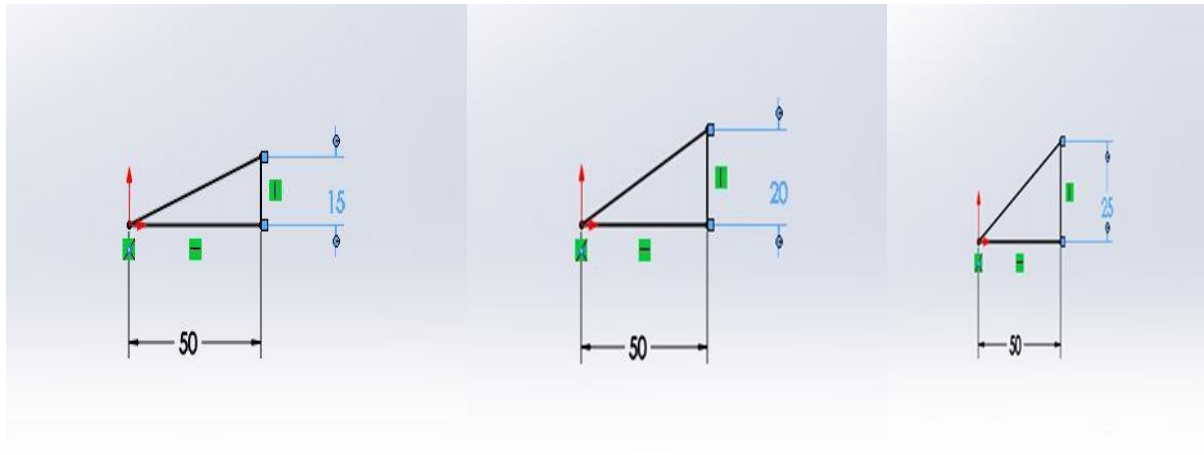


Figure 22: Design of delta-type vortex generator in 15mm, 20mm and 25mm

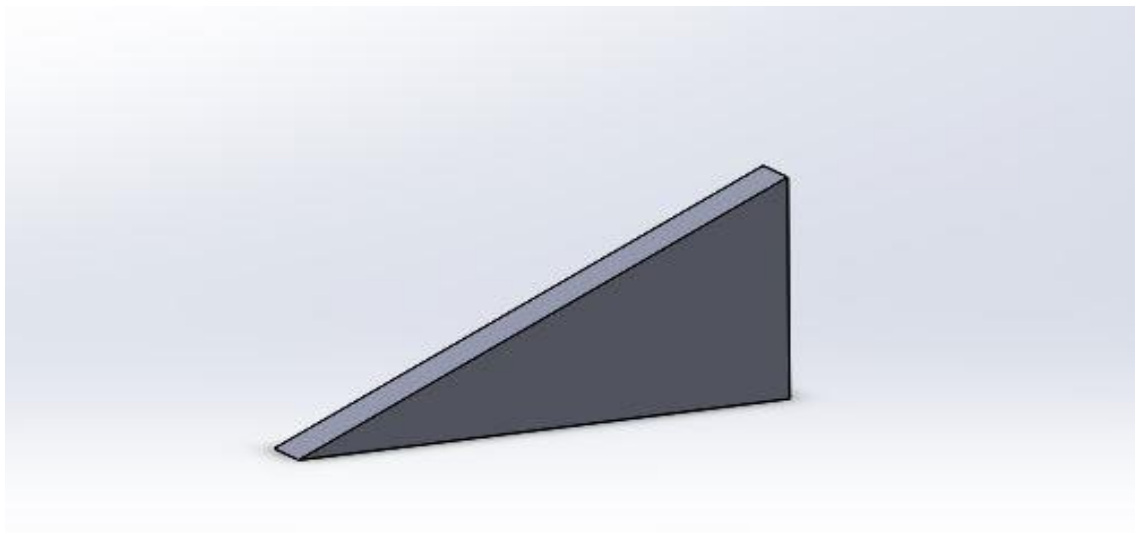


Figure 23: 3D design of delta-type vortex generator with SOLIDWORKS

Vortex generators with a delta-wing shape have a reduced frontal projection area, which reduces their drag. Furthermore, due to the VG's platy form, The vortex formed at a delta-wing-shaped VG's edge preserves its speed in the flow that passes downstream of the edge. The dimensional and 3D structure of delta type vortex generator is shown in Figure 22 and Figure 23 respectively.

Triangular-shaped vortex generator is a small design VG placed on the surface and changes the flow around the surface creating a boundary layer to inhibit separation [61]. It has enabled the development of flow separation from the design that is aerodynamically efficient and stable at race flow rates [92]. The structure of the triangular-shaped vortex generator in the Bishoftu double cabin pickup vehicle model is shown in Figure, 24 and Figure, 25.

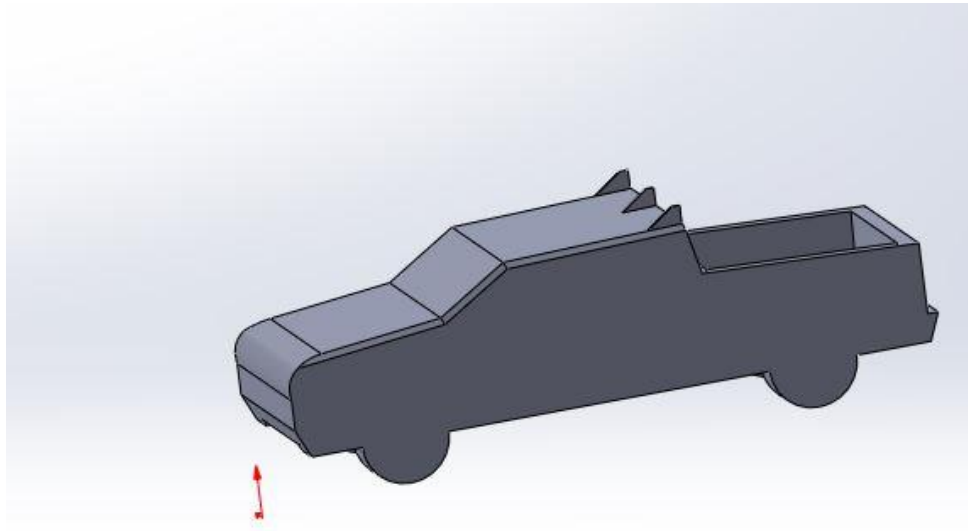


Figure 24: Bishoftu Pickup with triangular-type vortex generator

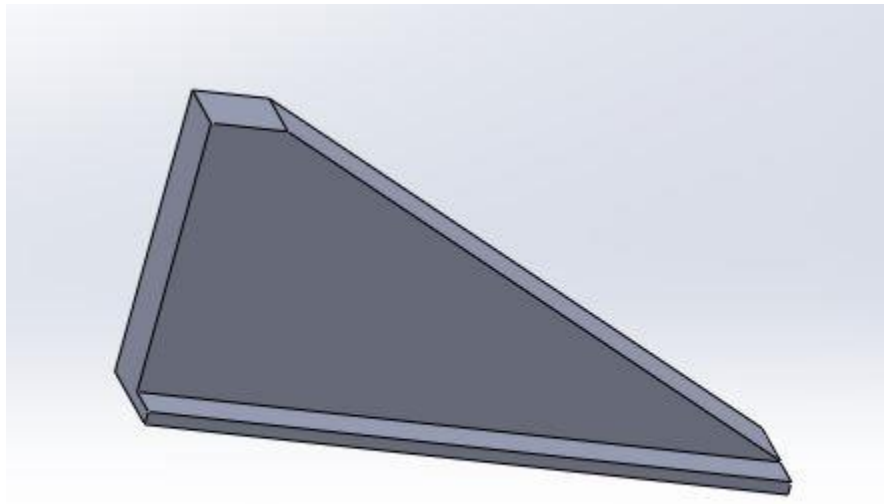


Figure 25: 3D design of triangular-type with SOLIDWORKS

3.3.4 Computational Domain

The modeling software created the computational domain, which covered the car geometry as shown in Figure 26. However, the computational fluid domain box dimensions are 41100mm in Length, 13915mm in width, and 8887mm in height. The fluid domains upstream and downstream dimensions from the car model are calculated to be respectively. Every surface within the computational domain and the wall limits of the car model were subjected to a non-slip boundary condition [93].

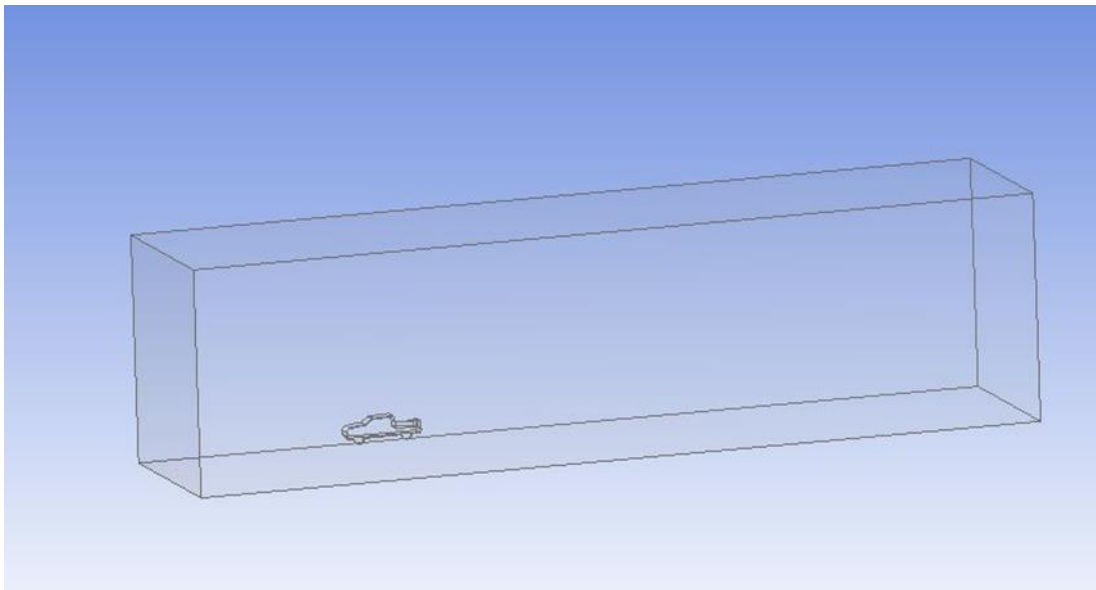


Figure 26: Wind Tunnel Dimensions

3.3.5 Mesh Generation

Considering the identification of the domain geometry, one of the most crucial phases in the preprocessing stage is mesh generation. As seen in Figure 27. The mesh is created based on the pickup design and domain surface. The sort of tetrahedron-shaped mesh that is utilized to discretize the computing domain. The tetrahedral mesh was the foundational setup of the mesh created for the surface bodies. A simulation that is convergent requires an extremely small size mesh. The significance utilized characterizes the overall mesh qualities. A relevance of 100 indicates great precision and low processing speed, on the other hand, a relevance of -100 implies quick processing and inadequate precision.

The facility utilized for this study possessed zero relevance and fine reference. The transition elements of the least size that are in close range to the vehicle's curvature and elements of the highest size that extend outward from the vehicle along the wind tunnel box's walls are made possible by the improved size function, intimacy, and curvature. Mesh size in the body element is shown in Figure 27.

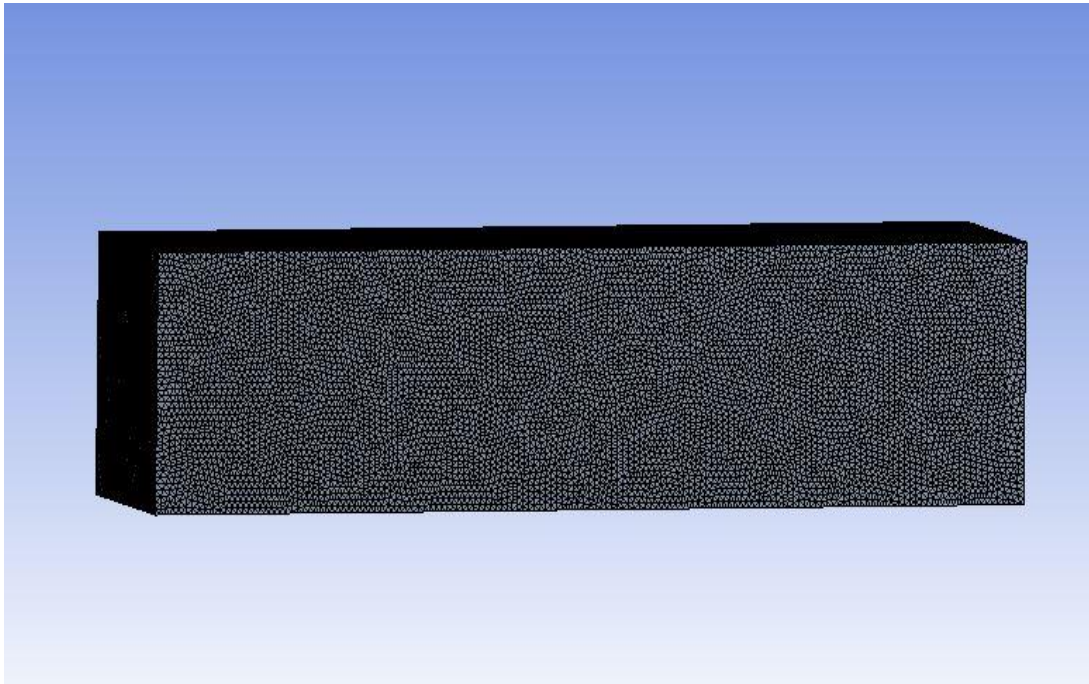


Figure 27: Mesh size in Body element

To differentiate between the faces and characteristics of each appearance, named selections have been created. The velocity inlet and pressure outlet of the vehicle were represented by the horizontal lines in front and behind it, respectively. However, that side that is linked to the car is called the symmetry line, while the side that has been separated is frequently referred to as the symmetry side. The term is utilized for the bottom face and symmetry top to the upper face. The car box symmetry top and symmetry side surface were placed far from the vehicle and there is no effect on it. This was done to ensure that instead of symmetric surfaces, they would have the same boundary conditions as a symmetric surface that is, a slip wall with zero shear forces. It then looks like the ground surface and the vehicle surfaces are part of the inflation. Inflation on the surface of the mesh is the process of generating a mesh layer for the boundary layer simulation on the vehicle's surface.

The improvement of the mesh for this study is necessary. Modifying the element number inside the control volume is the primary objective of mesh refining. The purpose of to increase the number Of elements in the sections where the fluid flow aligns with the proper vehicle body surface, hence improving computing accuracy.

Achieving high precision in the far field, such as the pressure outlet and velocity inlet zones is unattractive. To explicitly manage mesh size, the interior boxes in the wake and around the car zone provide the basis for the mesh refining approach used in this study. Although this method takes more time than other approaches, it is incredibly accurate. The preprocessing application is utilized to generate the boxes. The box walls have surface parts that are all the same size. The boxes provide meshing domains where a highly comfortable control over cell size is achieved. Two mesh element bodies have been completed for mesh refining to increase the number of layers in the appropriate areas. The mess element in the car is also shown in Figure 28.

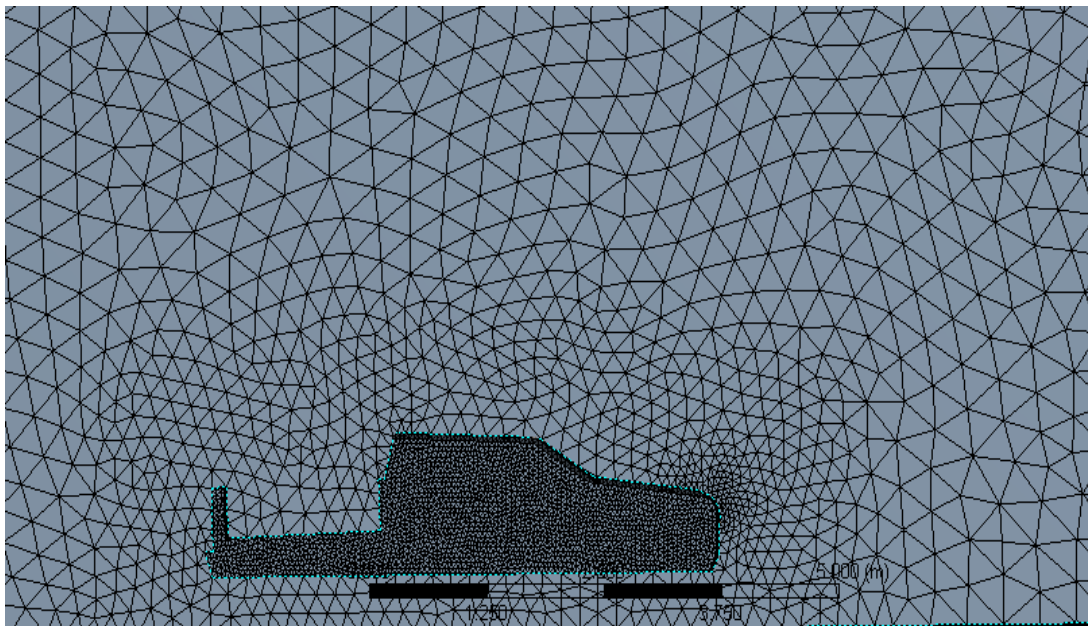


Figure 28: Mesh size in car element

During mesh generation of the model, 2826637 mesh element numbers and 491801 nodes are created. Several mesh sizes have been recommended for the sake of ensuring a high level of the consequences, and the most attractive part was selected for additional simulations. Aspect ratio and skewness were among the characteristics that were checked to assess the quality of the created mesh.

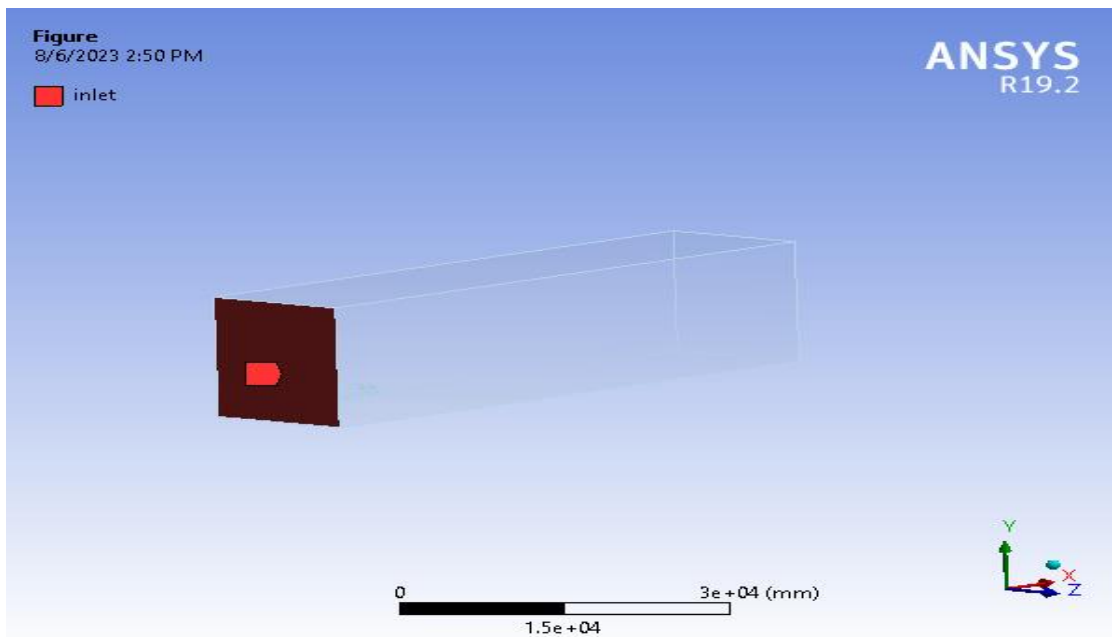


Figure 29: Velocity inlet boundary condition as a baseline model

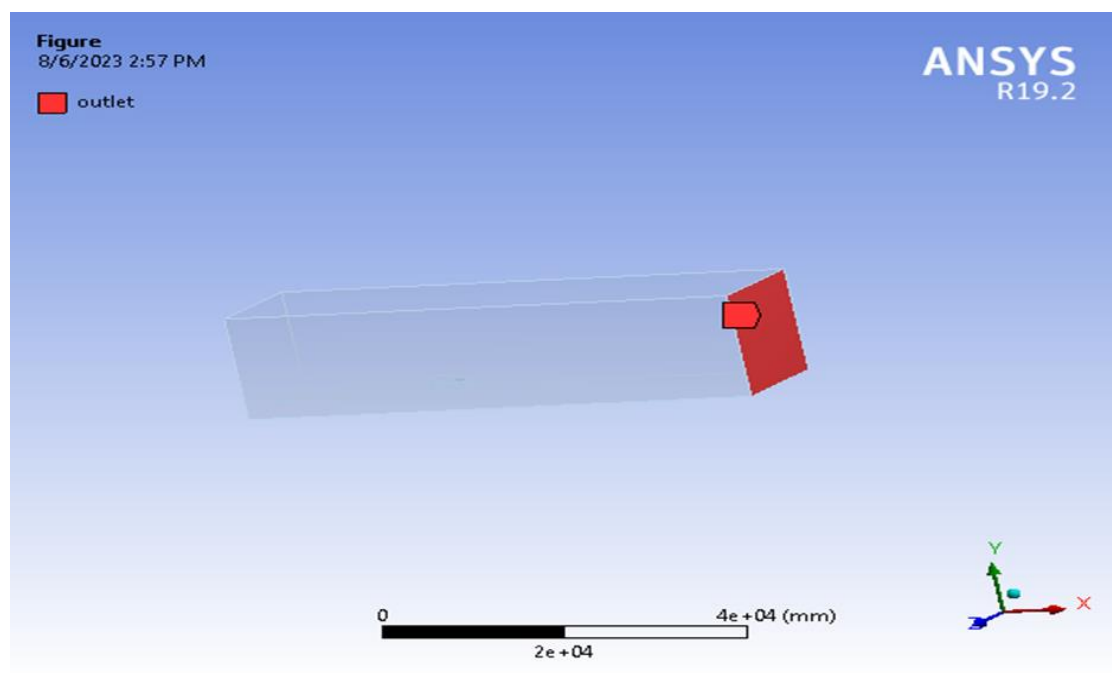


Figure 30: Pressure outlet boundary condition in baseline model

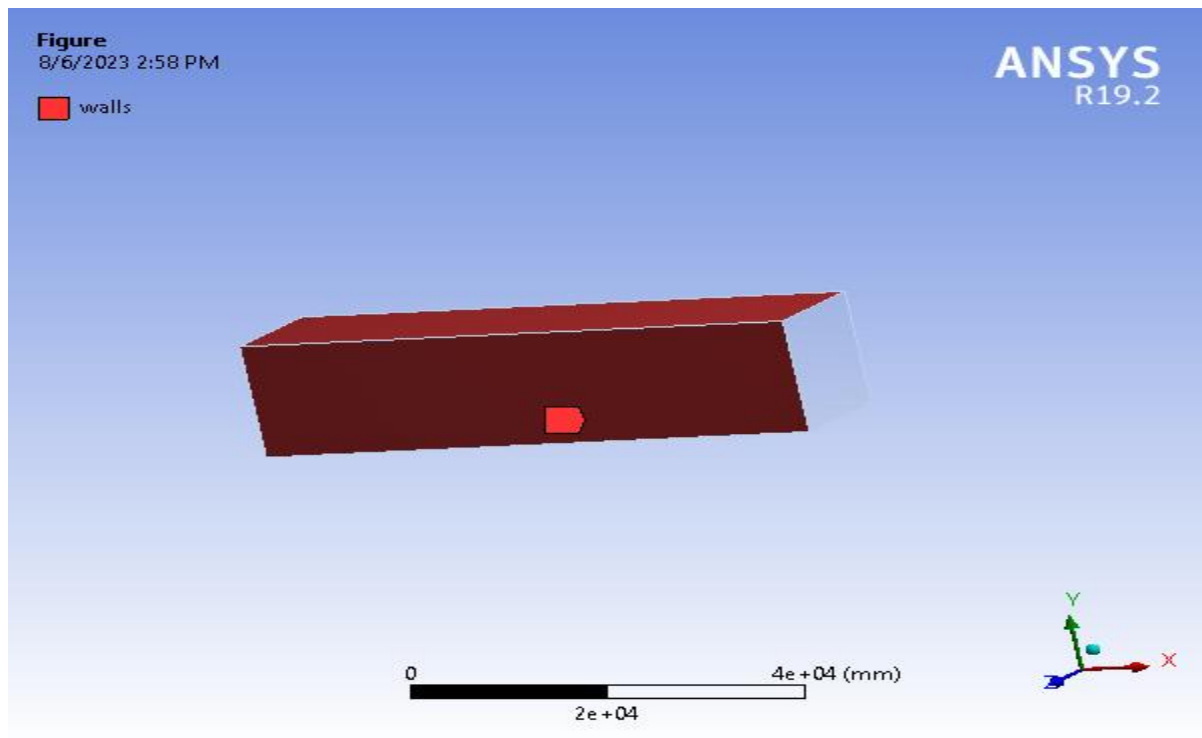


Figure 31: Walls boundary condition as a baseline model

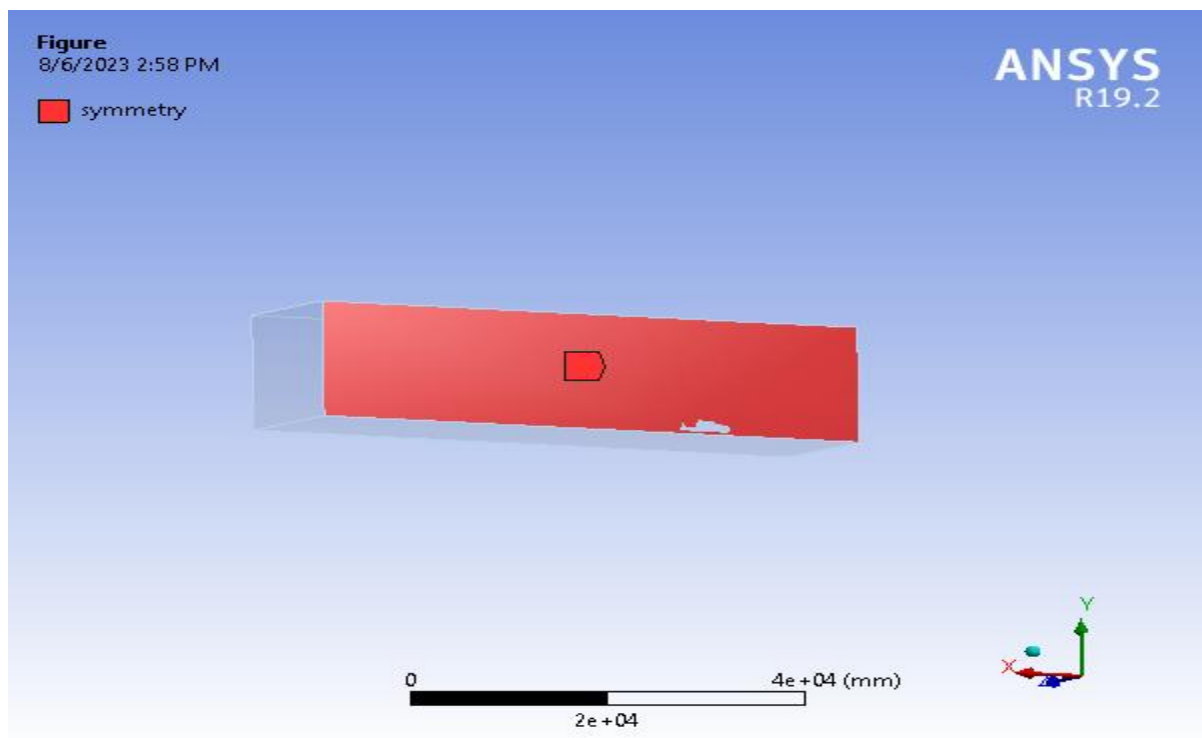


Figure 32: Symmetry boundary condition as baseline model

3.4 Pre-Processing

During a simulation process pre-processing, it is necessary to obtain and perform before a numerical solution. It involves defining the problem, modeling in three dimensions, and meshing and implementing boundary conditions and physical operating conditions. To imitate the. Air circulation around a vehicle, a fluid volume surrounded which is produced. To accomplish this, an enclosure is built around the car, and the vehicle body is subtracted using an ANSYS Boolean command. To minimize the total computing expenditure and time, the vehicle was analyzed transverse symmetrically [46].

3.5 Solver

The solver implements a numerical solution approach to perform the calculations, which may involve the application of finite difference or multiple-part methodologies. Finite volumes, which may be a unique finite difference technique, are used by most CFD codes. The fluid flow equations are initially integrated over the regulation volumes, which results in the actual conservation of essential properties for each finite volume. These integral equations are then discretized, which results in algebraic equations by changing the integral fluid flow equations, and algebraic equations are finally solved repeatedly.

3.6 Numerical methods

The assumption provided characterizes the distinguishing features of the specified type of flow in this section is the fact that it should be three-dimensional, one-phase, turbulent, and stable excluding dust particles or water vapor. Nevertheless, the air characteristics remain constant and the velocity is thought to be uniform over the vent entrance. Subsequently, an explanation of the mathematical model is held that, given the assumptions made, represents the flow. However, at this point, Navier-Stokes equations have been used in an effort. Additionally, Turbulence's impact on time-averaged motion is described by the transport models, and the equations that govern motion develop concerning time.

This implies that for the momentum equation in the x, y, and z directions as well as for the continuity equation, another set of motion equations with the same structure can be constructed by adding $T = T + T'$ Quantities on equation number, 3.1, 3.2, 3.3, 3.4.

$$\begin{aligned} \frac{\partial}{\partial t}(\rho C_p \bar{T}) + \frac{\partial}{\partial x}(\rho \bar{u} C_p \bar{T}) + \frac{\partial}{\partial y}(\rho \bar{v} C_p \bar{T}) + \frac{\partial}{\partial z}(\rho \bar{w} C_p \bar{T}) \\ = K \left(\frac{\partial^2 \bar{T}}{\partial x^2} + \frac{\partial^2 \bar{T}}{\partial y^2} + \frac{\partial^2 \bar{T}}{\partial z^2} \right) - \frac{\partial}{\partial x_i}(\rho C_p \bar{T} \bar{U}_i) \dots \dots \dots (3.1) \end{aligned}$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \bar{u}) + \frac{\partial}{\partial x}(\rho \bar{u} \bar{u}) + \frac{\partial}{\partial y}(\rho \bar{u} \bar{v}) + \frac{\partial}{\partial z}(\rho \bar{u} \bar{w}) \\ = \frac{\partial \bar{p}}{\partial x} + \mu \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} \right) - \frac{\partial}{\partial x_i}(\rho \bar{u} \bar{U}_i) \dots \dots \dots (3.2) \end{aligned}$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \bar{v}) + \frac{\partial}{\partial x}(\rho \bar{u} \bar{v}) + \frac{\partial}{\partial y}(\rho \bar{v} \bar{v}) + \frac{\partial}{\partial z}(\rho \bar{v} \bar{w}) \\ = -\frac{\partial \bar{p}}{\partial y} + \mu \left(\frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2} + \frac{\partial^2 \bar{v}}{\partial z^2} \right) - \frac{\partial}{\partial x_i}(\rho \bar{v} \bar{U}_i) - \rho g \beta (T^\infty - T) \dots \dots \dots (3.3) \end{aligned}$$

$$\frac{\partial}{\partial x}(\rho \bar{u}) + \frac{\partial}{\partial y}(\rho \bar{v}) + \frac{\partial}{\partial z}(\rho \bar{w}) = 0 \dots \dots \dots (3.4)$$

In this case, the aspects of velocity in the x, y, and z directions can be identified by the letters u, v, and w, respectively; the viscosity, mass density, and thermal expansion coefficient of air are represented by the letters μ , δ , and β ; time is indicated by the letter T; pressure, temperature, and gravitational acceleration are represented by the letters P, T, and K is the fluid's conductivity.

Additionally, targeted flows indicate a force that is generated both mechanically and by floating material that arises from a difference in fluid density brought on by the temperature differences to move the air in the particular fluid domain. The principles of mass and momentum laws of conservation have been used to describe the flow, as demonstrated in this study, and the air is also regarded as being Newtonian and incompressible when the airflow has a Mach number of less than 0.3 or roughly 33m/s [94].

3.7 Turbulence Modeling

In most practical circumstances involving airflow over a vehicle, turbulent modeling the development and utilization of a mathematical model for estimating the consequences of turbulence is an ordinary approach. Nevertheless, Figure 33, demonstrated turbulence models, which are frequently used in engineering applications [95]. However, among such things, the study's option and illustration is the realizable K-Epsilon method of the turbulent model as shown in Figure 33.

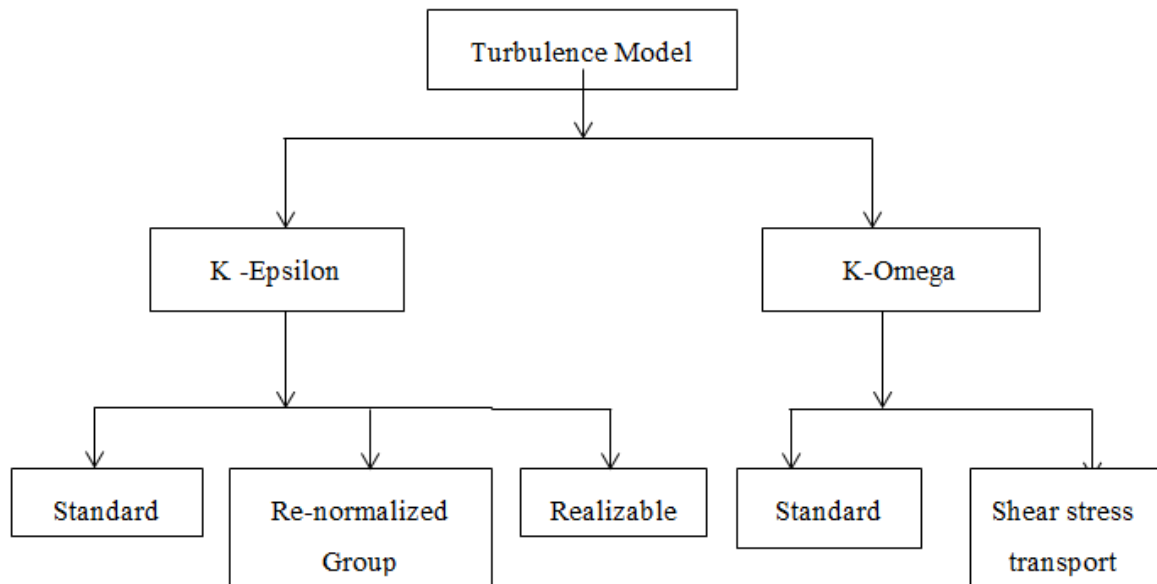


Figure 33: Classification of Turbulence model [95]

. Reynolds Averaging (combined averaged) and sorting a couple of methods to solve the Navier-Stokes equations without explicitly modeling and small-scale interruptions. The Reynolds-Averaging Navier Stokes (RANS) modeling approach. Which time-averages the flow quantities for a wide range of turbulence scales under the study, and minimizes computational efforts as well as resources [94].

CHAPTER FOUR

RESULTS AND DISCUSSIONS

The investigation on the variation in aerodynamic forces from different configuration models that vary from each other due to the addition of vortex generators with different configurations on the roof end, the dimensions of the vortex generators with different parameters for all models, and the fact that its perspective of attachment is at zero degrees for all mentioned models, all enabled to clearly explain the results with through the use of charts and graphs. Additionally, three different vehicle speeds 80, 100, and 120 km/h were chosen as a result of the effectiveness of this type of vehicle in numerous extended-cab pickup trucks are widely used by both public and commercial organizations for a wide range of tasks and travel around the country. That implies that the majority of travels are carried out at a rapid rate.

An ANSYS FLUENT 3D steady-state incompressible solution of the Navier-Stokes equations was carried out. Using non-equilibrium wall functions, the realizable turbulence has been described using the k- ϵ model. Presentations as well as a discussion of the computational results obtained from several models are performed. The equivalent k - ϵ turbulence model, boundary conditions, and meshing resolution were used to produce the same findings for every situation. First-order upwind discretization was utilized for the first 500 iterations to speed up The process of convergence. Once a second-order upwind technique was put into place after 500 iterations, the iterations continued until the convergence conditions were achieved. A convergence condition was fulfilled provided all residuals were in the range between 10^{-5} .

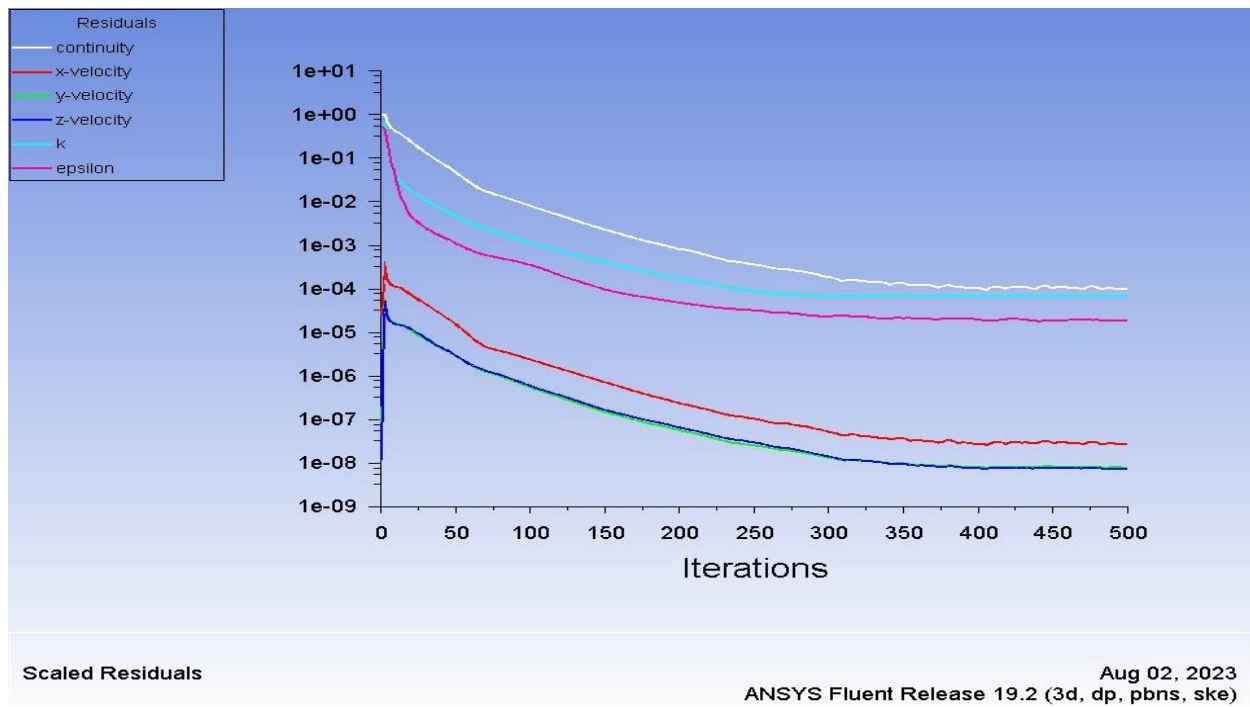


Figure 34: Scale Residual of Baseline Model at 80 km/h

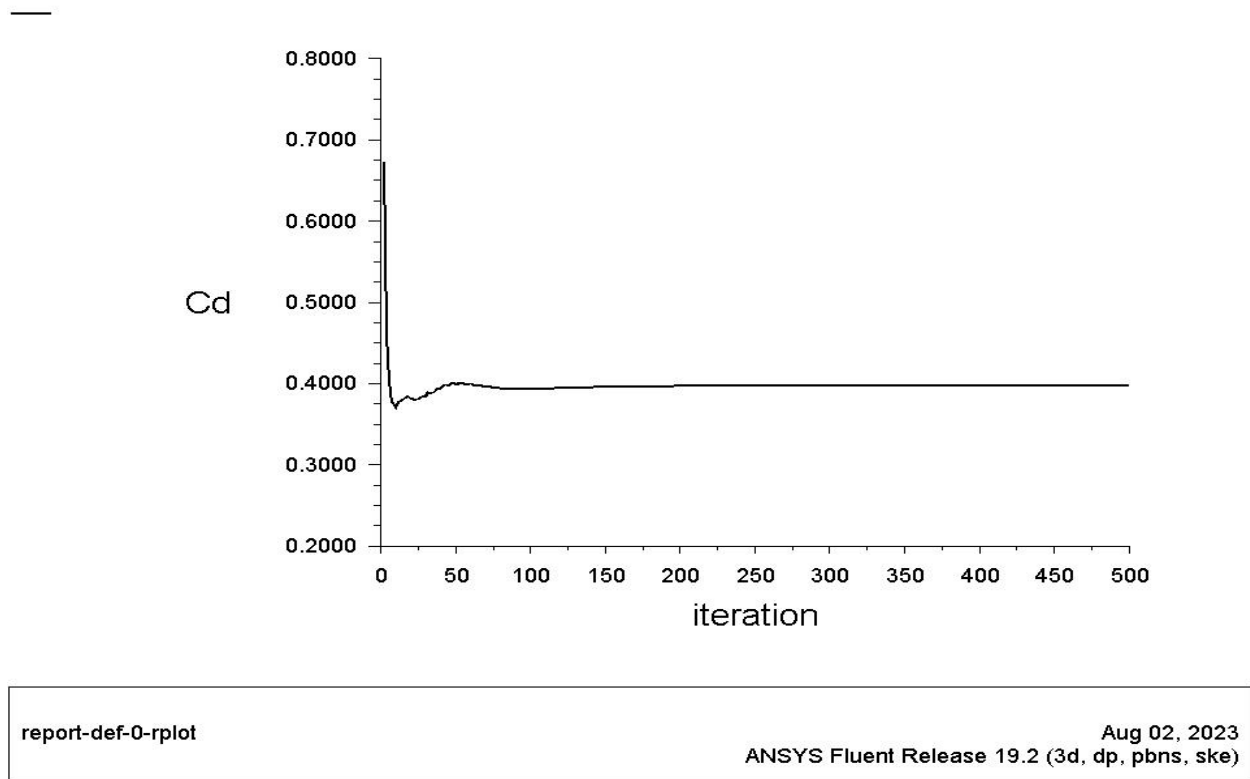
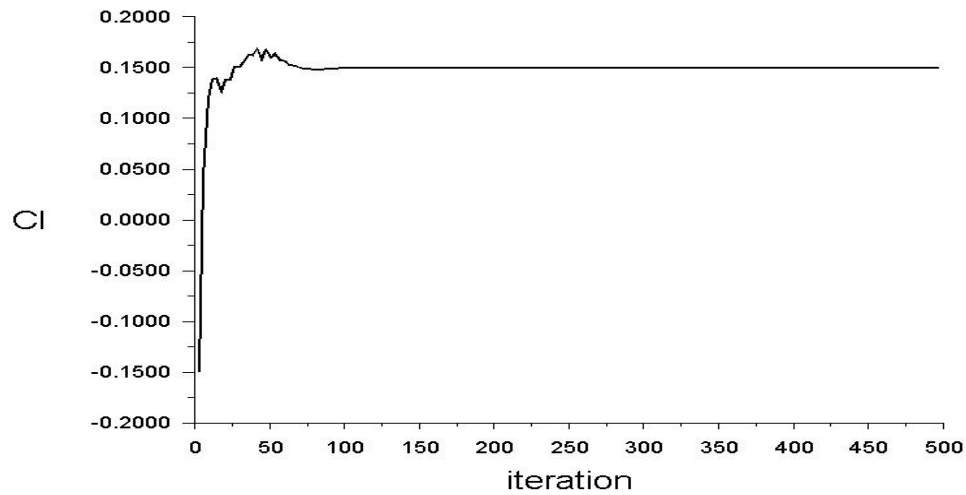


Figure 35: Drag Coefficient of Baseline Model at 80 km/h



report-def-1-rplot

Aug 02, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 36: Lift Coefficient of Baseline Model at 80 km/h

The above graphs indicate (Figure 35 and Figure 36). The drag and lift coefficient of the baseline model at a speed of 80 km/h, here the drag and lift coefficients are summarized below in Table 3. The graph of the baseline model in 80 km/h scale residual, Drag coefficient, and Lift coefficient is shown in Figures 34, 35, and 36 respectively.

Table 3: Drag and Lift Coefficient of Baseline at Different Speeds

	Speed(km/h)	C_d	C_l
	80	0.4071	0.1550
	100	0.4075	0.1498
	120	0.4081	0.1492
Average	80,100,120	0.4075	0.1495

The drag coefficient of a baseline model increases with increasing speed. Figure 35-36 and Table 3 showed that the result of the drag and lift coefficients of a baseline model at three different speeds (80 km/h, 100 km/h, and 120 km/h) when simulated computational fluid dynamics (CFD) method of pickup model by realizable K-Epsilon turbulence model.

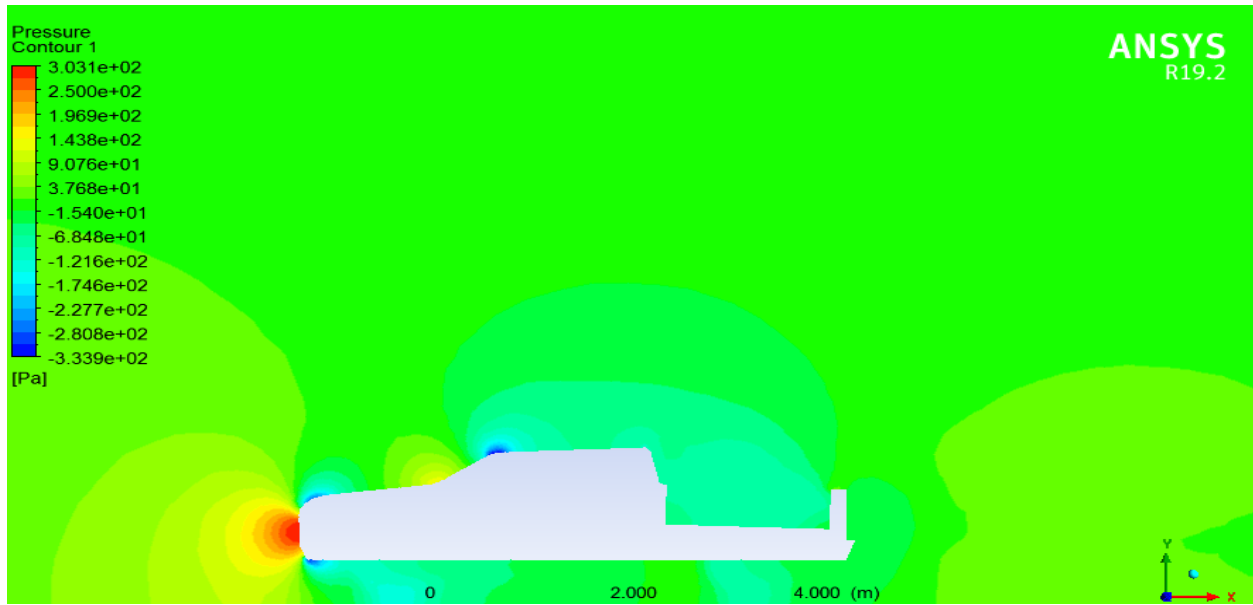


Figure 37: Pressure Contour in Symmetry Plane at 80 km/h

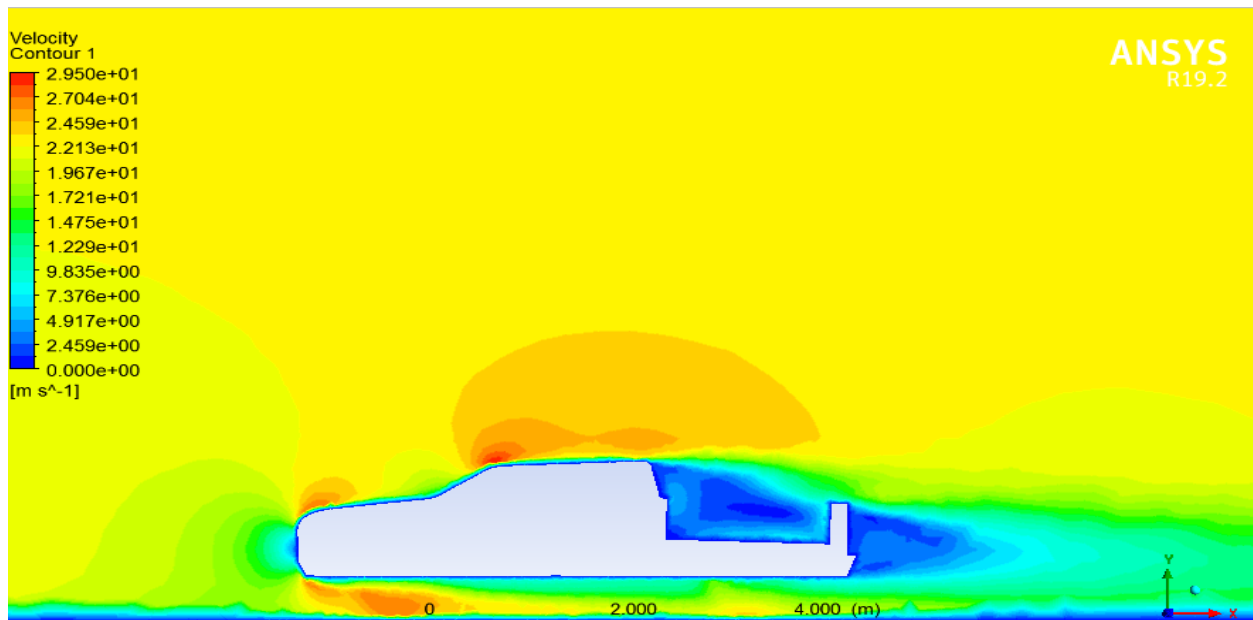


Figure 38: Velocity Contour in Symmetry Plane at 80 km/h

Here above in Figures 37, and 38 pressure and velocity distribution on the pick-up body of the baseline model can be seen. Both Figures show the highest flow of pressure on the front surface, and also with increasing speed maximum with velocity increased. The pressure contour in the symmetry plane at 80 km/h and the velocity contour is presented in Figures 37, and 38 respectively.

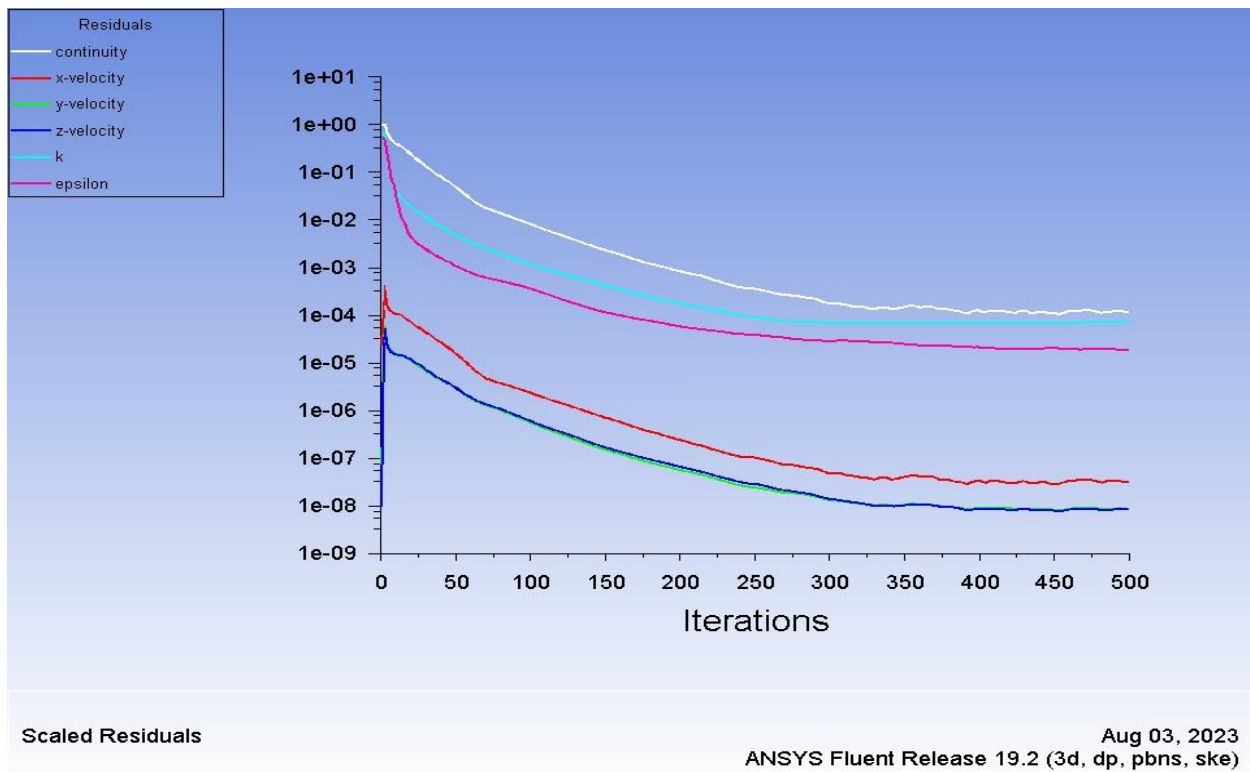


Figure 39: Scale Residual of Baseline Model at 100 km/h

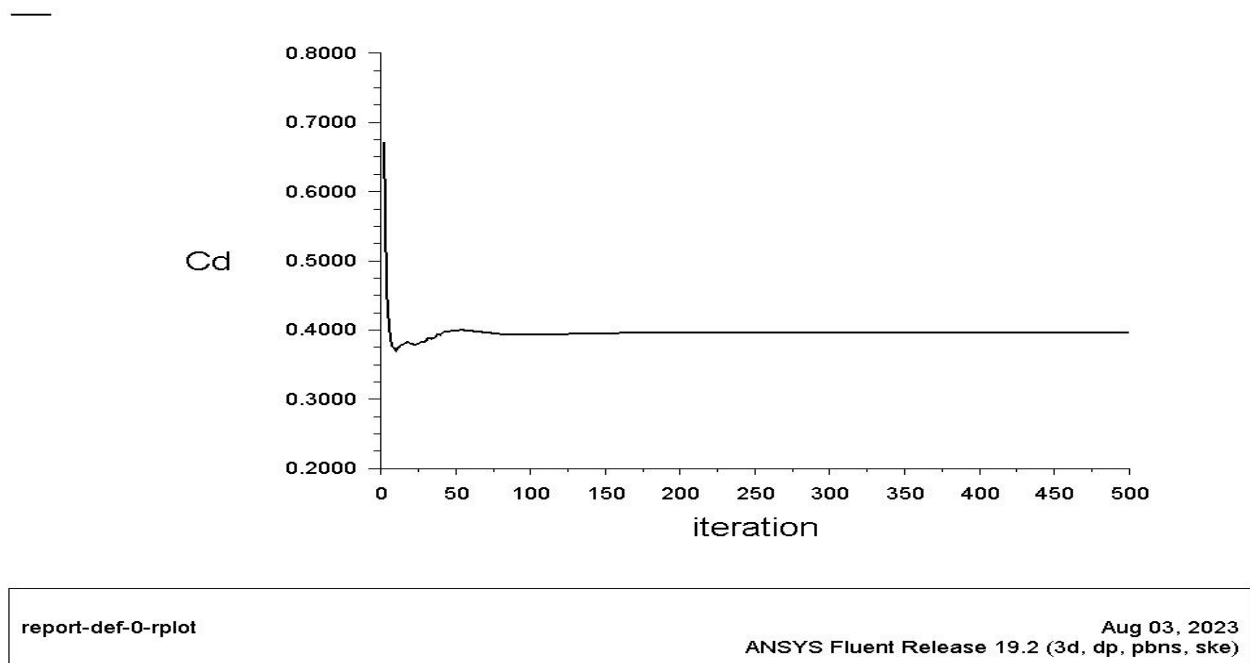
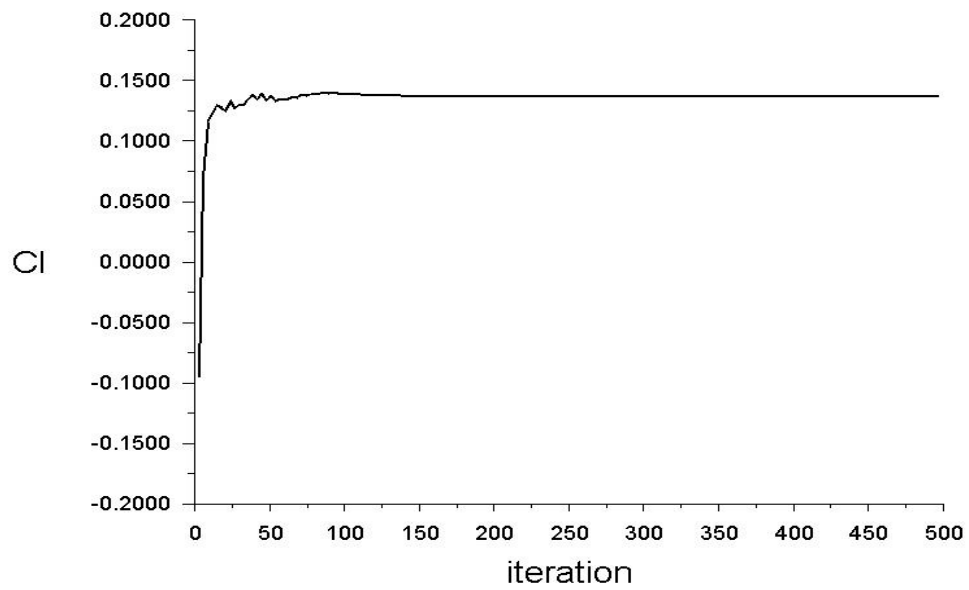


Figure 40: Drag Coefficient of Baseline Model at 100 km/h

The graph of the scaled residual, Drag coefficient, and Lift coefficient of the baseline model is shown in Figures 39, 40, and 40 respectively.



report-def-1-rplot

Aug 04, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 41: Lift Coefficient of Baseline Model at 100 km/h

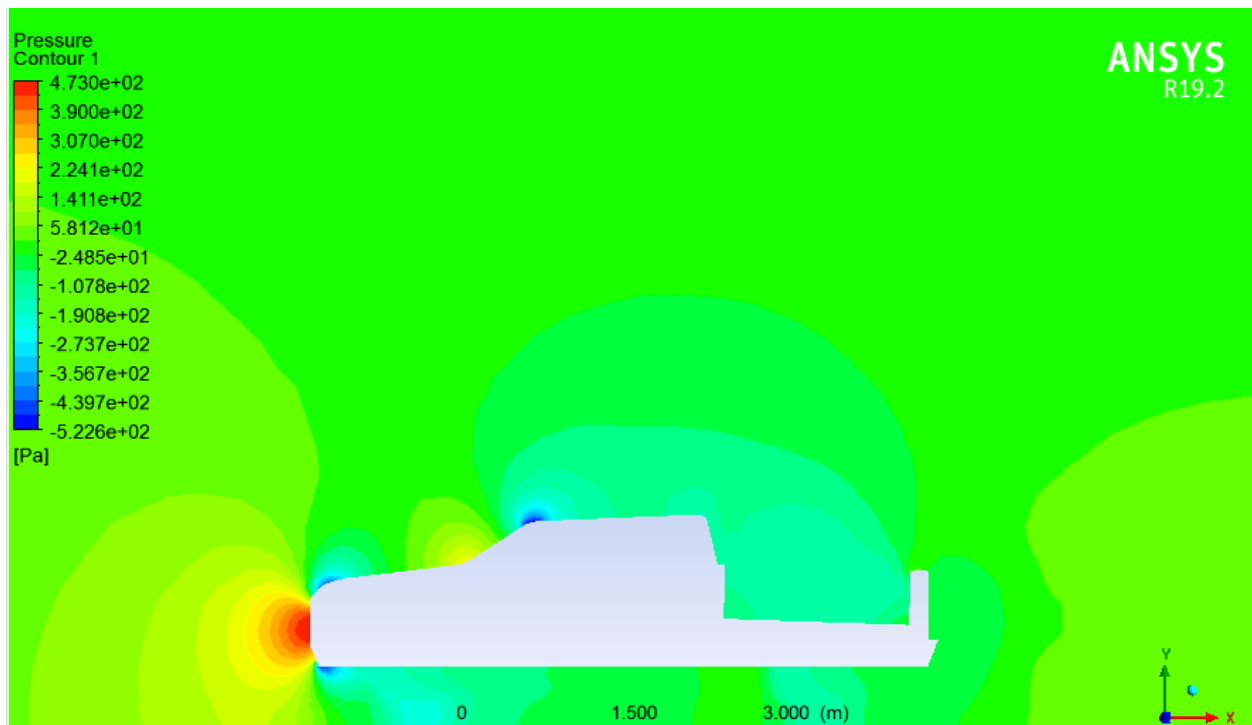


Figure 42: Pressure Contour in Symmetry Plane at 100 km/h

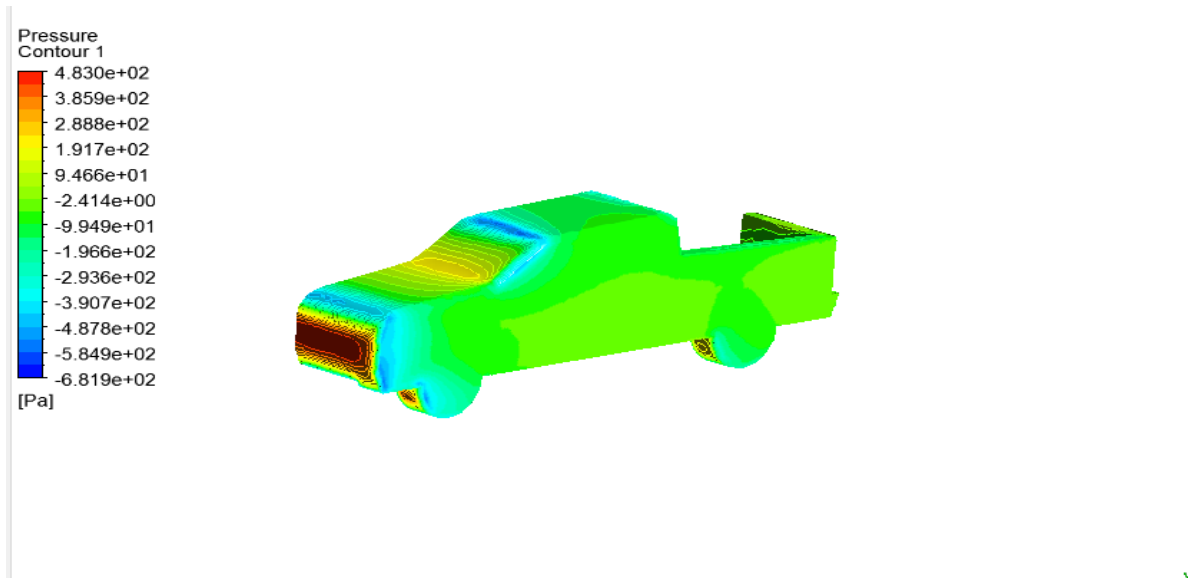


Figure 43: Pressure Contour in car at 100 km/h

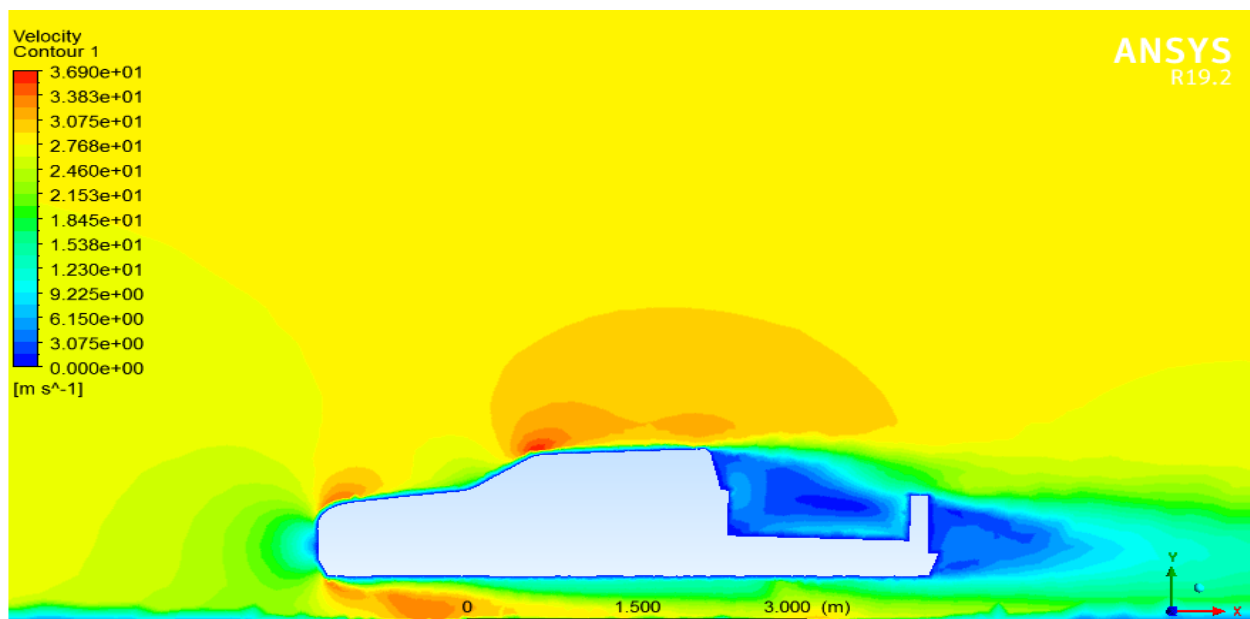


Figure 44: Velocity Contour in Symmetry Plane at 100 km/h

Figures 42 and 43 it can be observed that the pressure distribution at different planes on the body of the baseline model fluctuates a lot on its boundaries. The pressure contour plots indicate that the pressure on the front end of the vehicle reaches a maximum level as the fluid flowing toward the vehicle forms stagnation. Stagnation is the region that forms contact with the air in front of the vehicle. However, the velocity contour in the car plane of the baseline model at 100 km/h is shown in Figure 44 and Figure 45 the figures of velocity contour determine the difference of flow separation in symmetry and car plane.

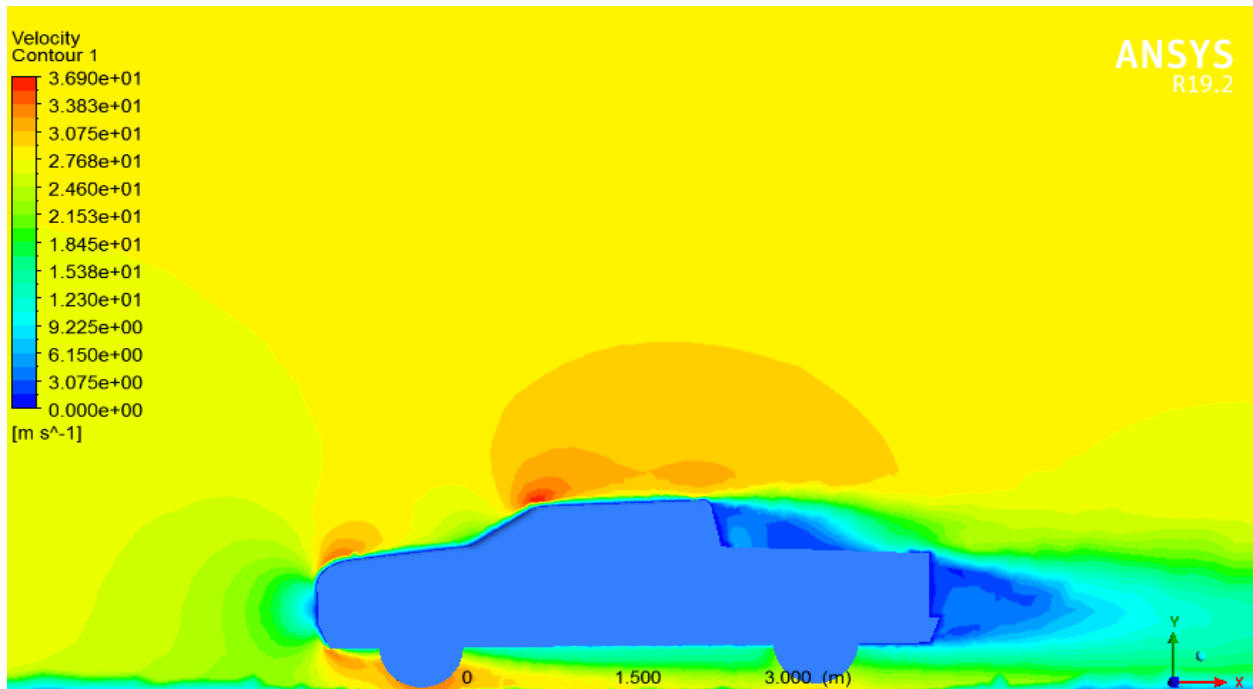


Figure 45: Velocity Contour in the car at 100 km/h

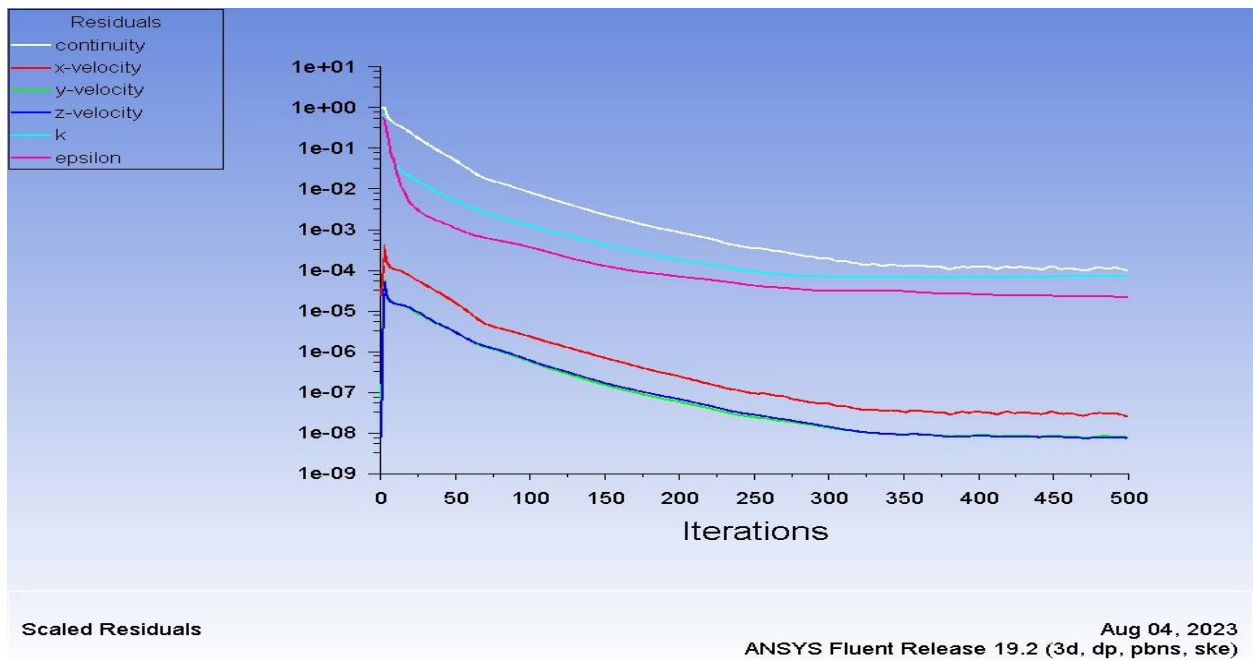
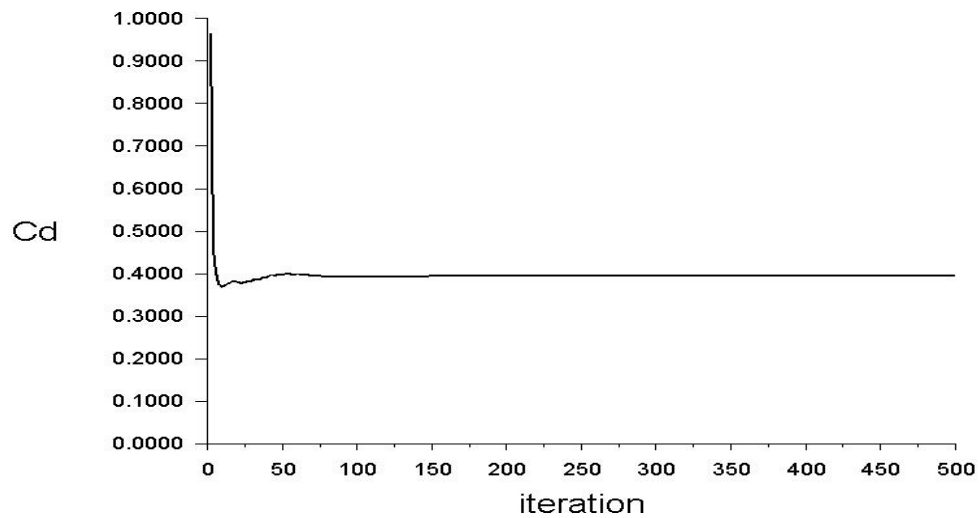


Figure 46: Scale Residual of Baseline Model at 120 km/h

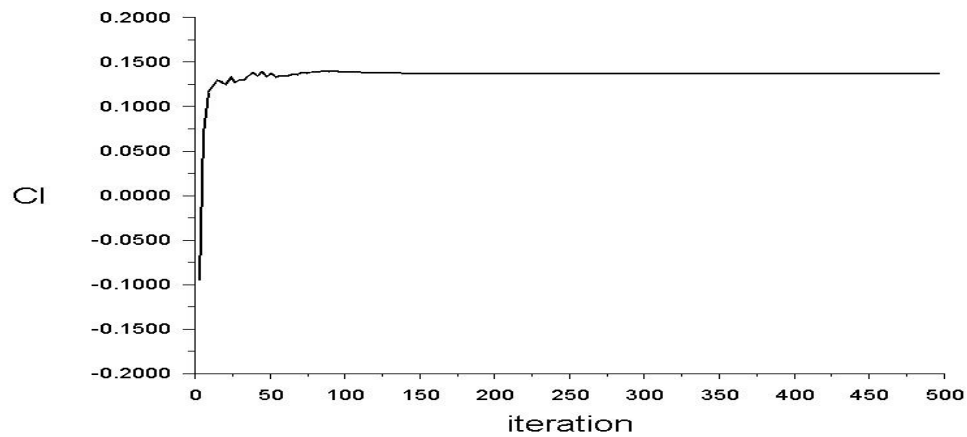
Identifying the residual trends is one of the most fundamental measures in determining whether a solution is converged or not. The trends of the baseline simulation were already shown and discussed. The result of the scaled residual at a speed of 100 km/h is shown in Figure 46.



report-def-0-rplot

Aug 04, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 47: Drag Coefficient of Baseline Model at 120 km/h



report-def-1-rplot

Aug 04, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 48: Lift Coefficient of Baseline Model at 120 km/h

The above graphs indicate (Figure 47 and Figure 48). The drag and lift coefficient of the baseline model at a speed of 120 km/h. The graph indicated that when the drag coefficient of a baseline model increases with increasing speed.

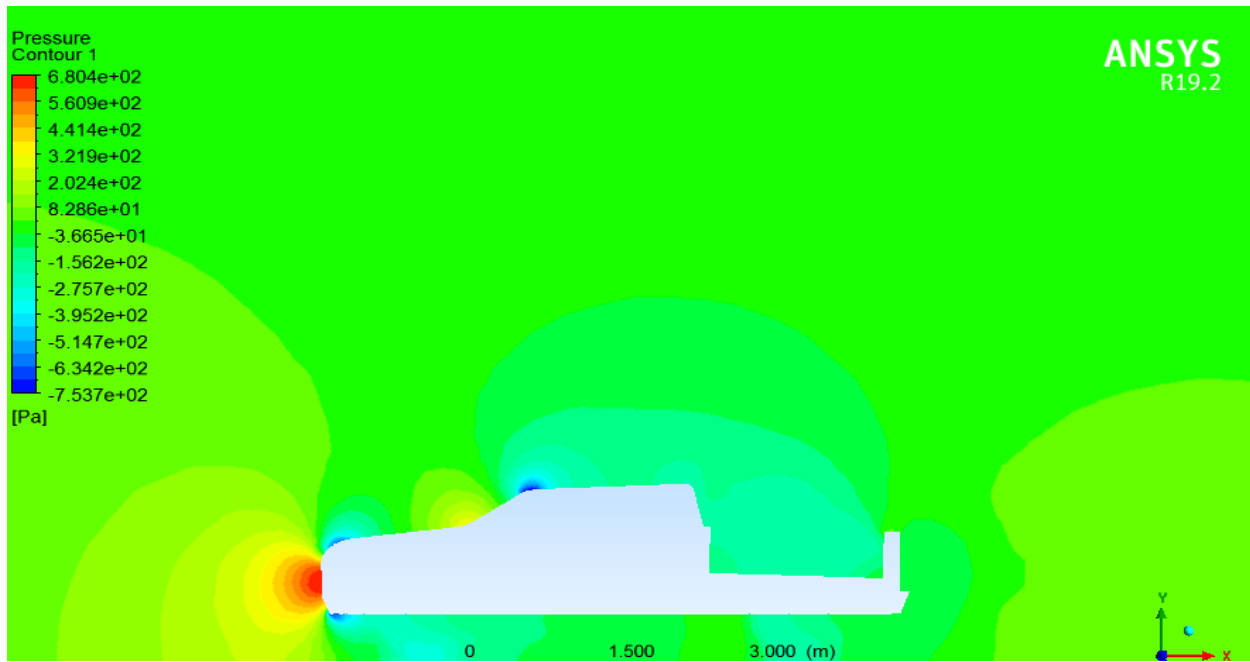


Figure 49: Pressure Contour in Symmetry Plane at 120 km/h

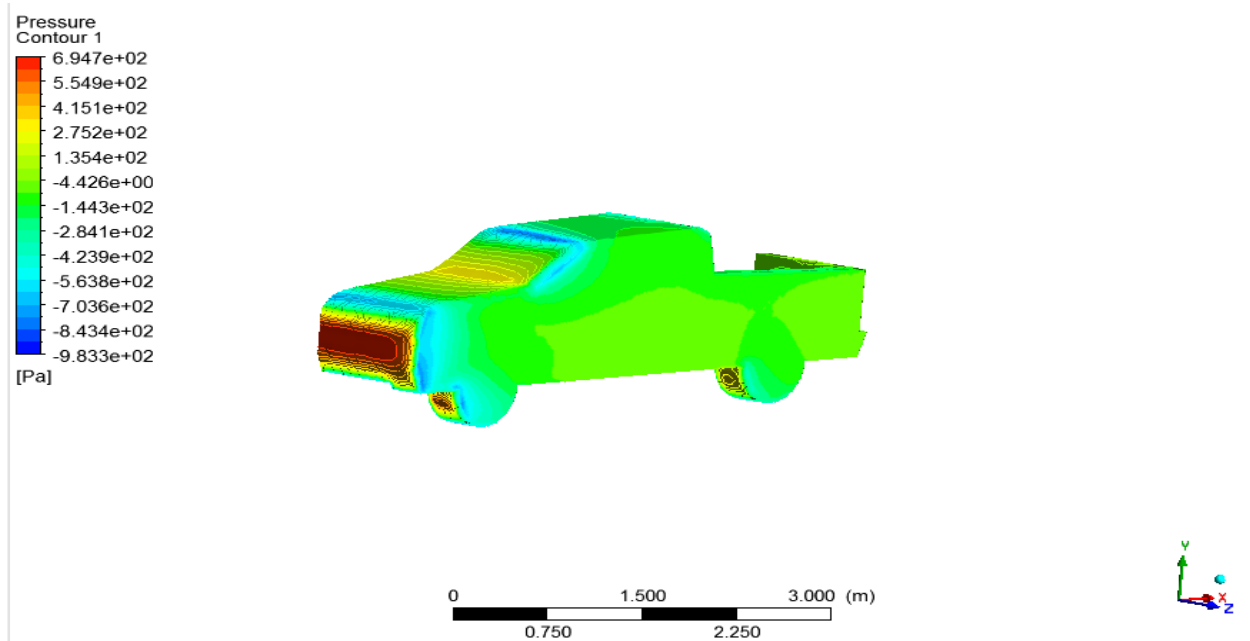


Figure 50: Pressure Contour in the car at 120 km/h

When the speed of the baseline model increases the stagnation point of the pressure contour increases at the front of the vehicle. The pressure at the symmetry and car plane of the baseline model at 120 km/h are presented in Figures 49 and 50 respectively.

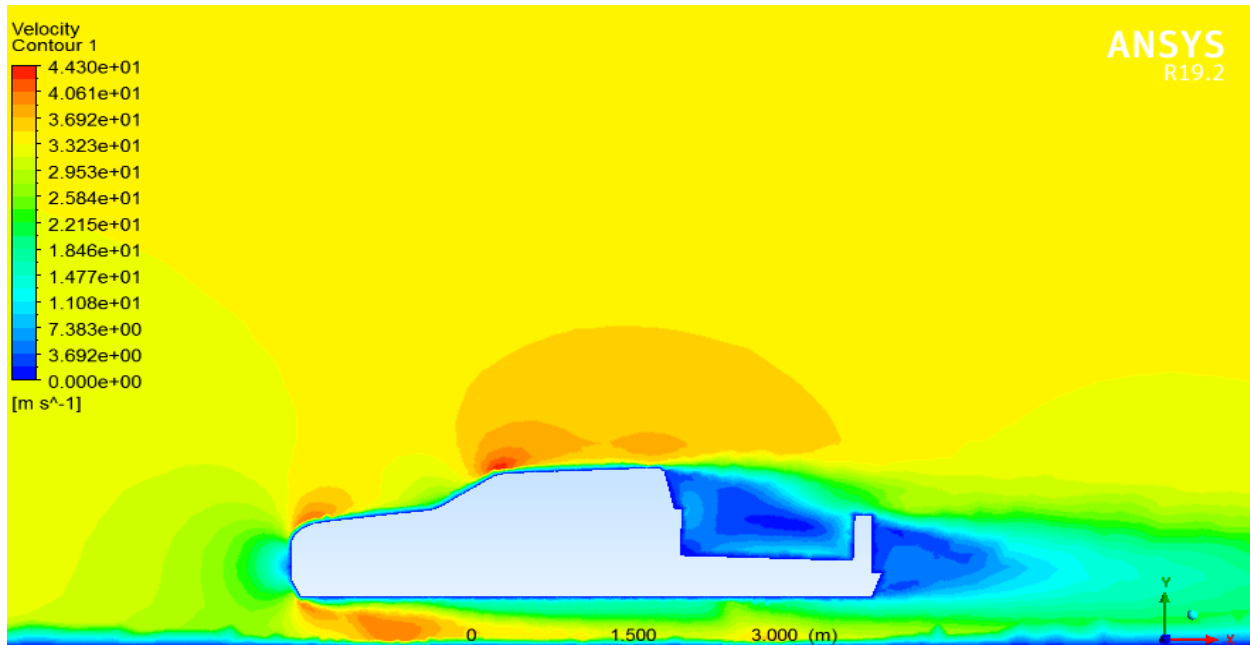


Figure 51: Velocity Contour in Symmetry Plane at 120 km/h

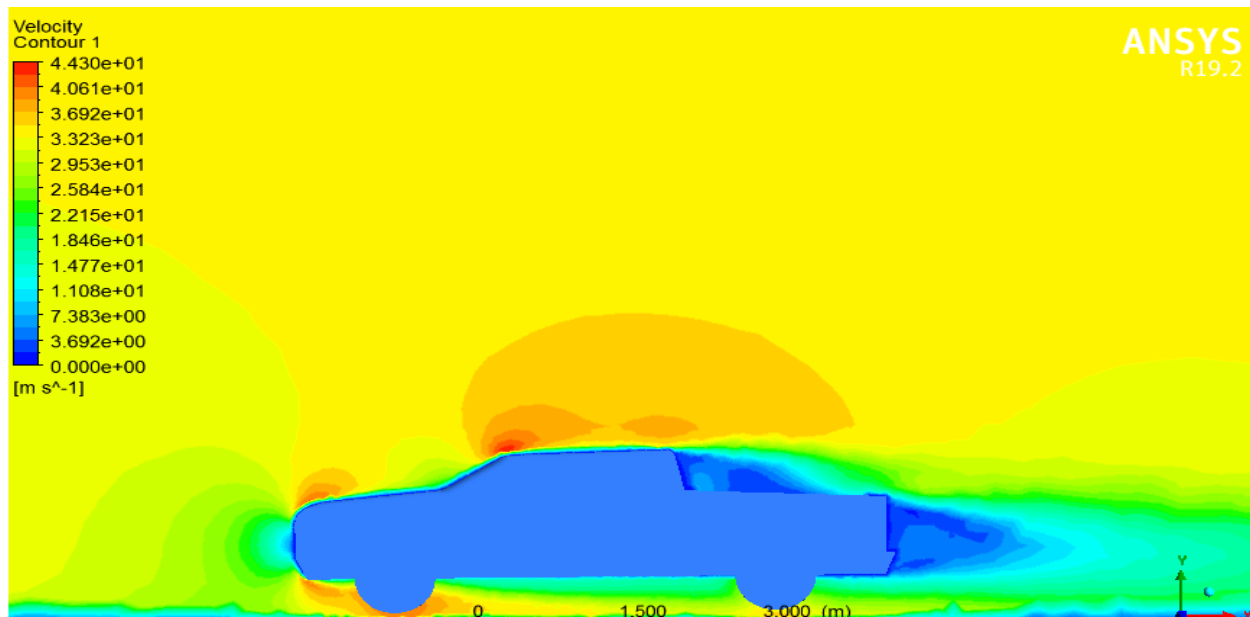
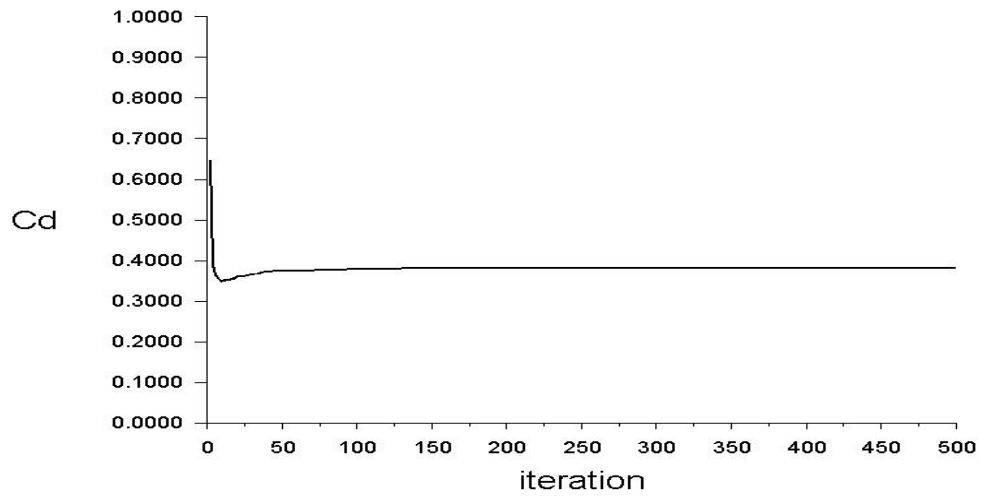


Figure 52: Velocity Contour in a car at 120 km/h

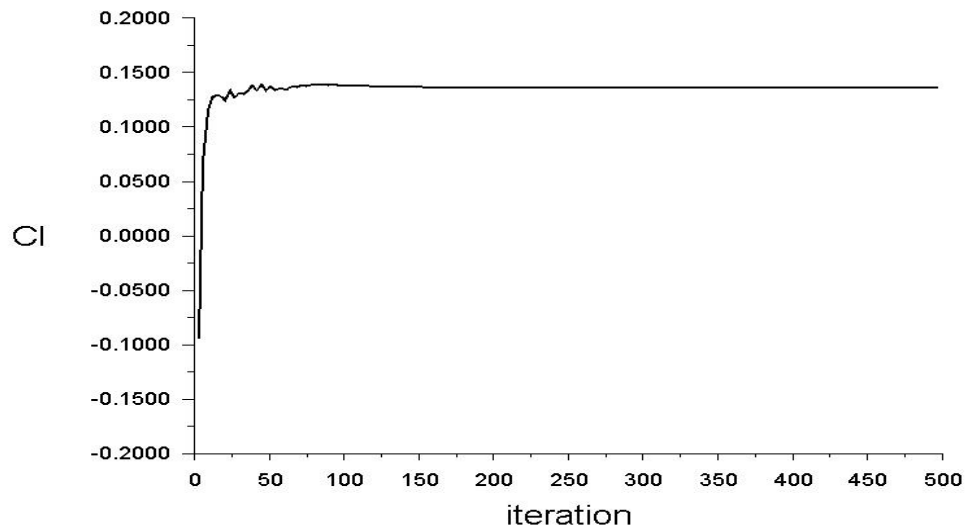
The velocity contour plots indicate that the flow of fluid on the rear end of the vehicle reaches a maximum level as the fluid flowing towards the vehicle forms wake flow separation. However, the velocity contour in symmetry and the car plane of the baseline model at 120 km/h is shown in Figures 51, and 52 the figures of velocity contour determine the difference of flow separation in symmetry and car plane.



report-def-0-rplot

Aug 04, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 53: Drag Coefficient with delta-type vortex generator at 80 km/h

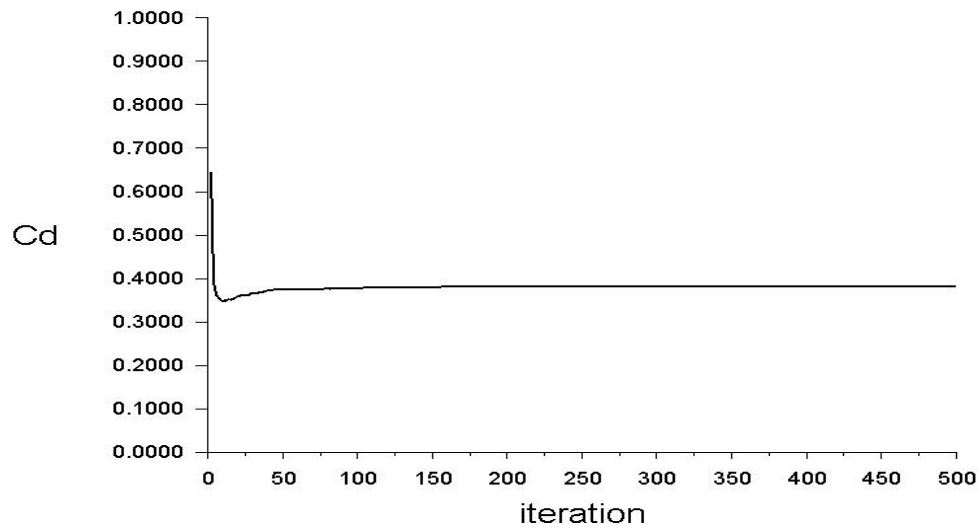


report-def-1-rplot

Aug 04, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 54: Lift Coefficient with Delta-type vortex generator at 80 km/h

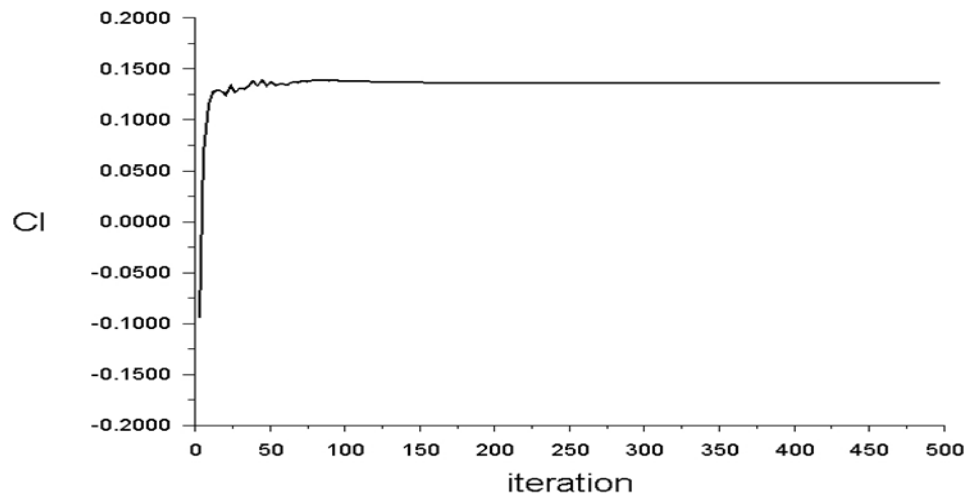
The drag and lift coefficient graph of the delta-shaped vortex generator at 80 km/h are shown in Figures 53, and 54 respectively.



report-def-0-rplot

Aug 04, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 55: Drag Coefficient with Delta-type vortex generator at 100 km/h



report-def-1-rplot

Aug 04, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 56: Lift Coefficient with Delta-type vortex generator at 100 km/h

The drag and lift coefficient graph of the delta-shaped vortex generator at 100 km/h are shown in Figures 55, and 56 respectively.

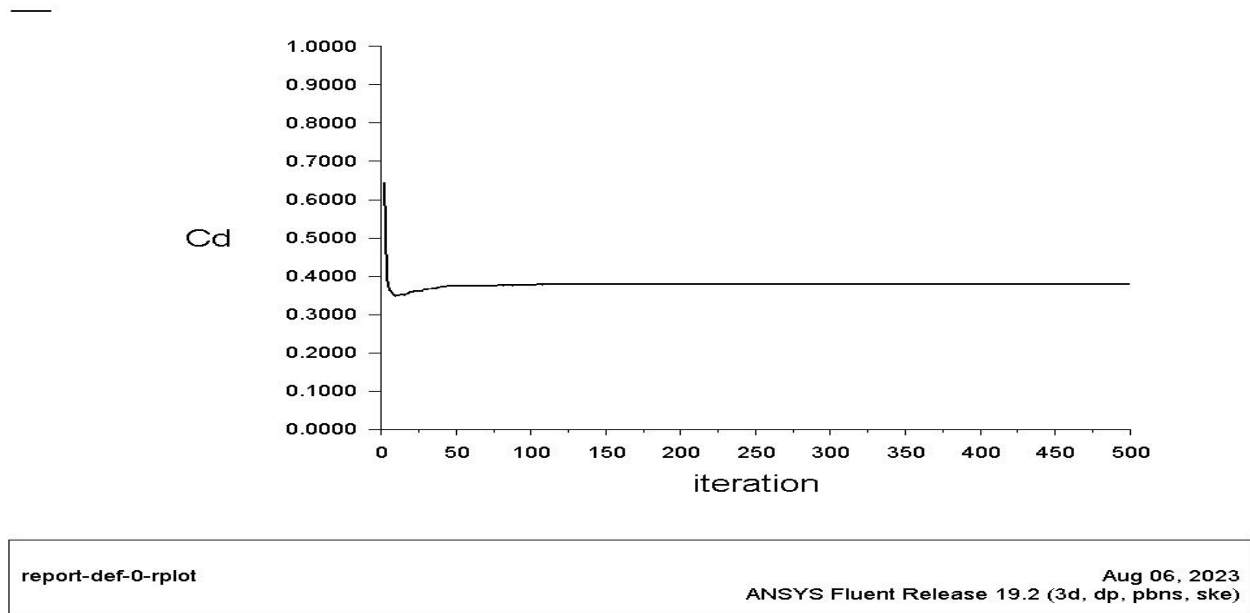


Figure 57: Drag Coefficient with Delta-type vortex generator at 120 km/h

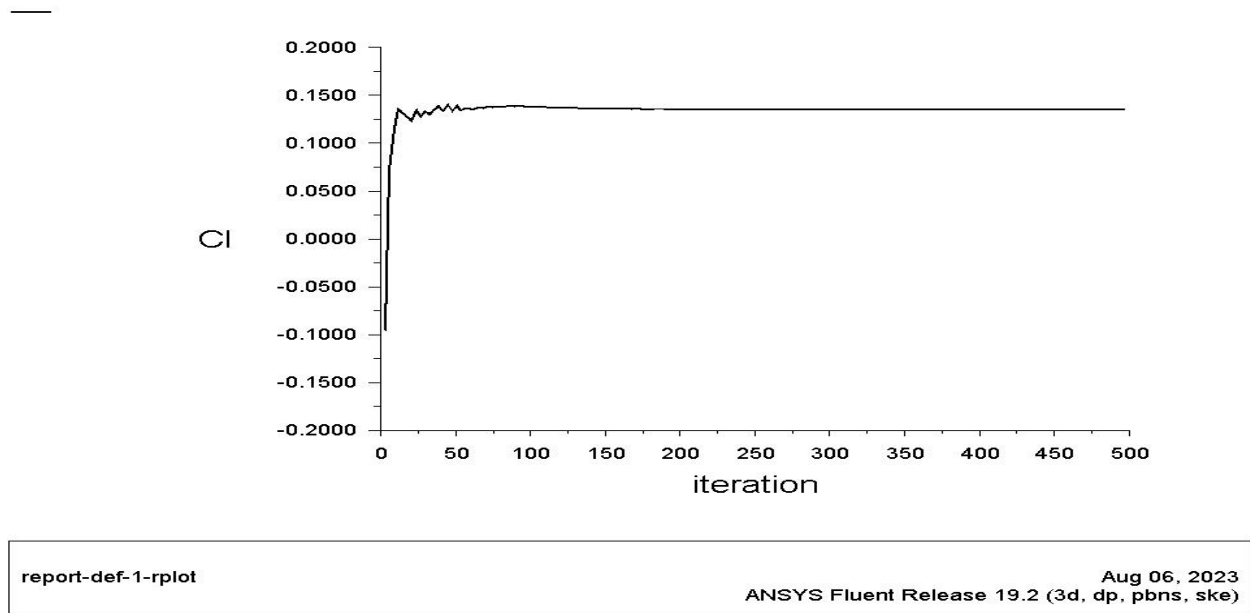


Figure 58: Lift Coefficient with Delta-type vortex generator at 120 km/h

All the above graphs (Figures 53-58) indicate the drag and lift coefficients of the pickup model with a delta-shaped vortex generator at speeds of 80, 100, and 120 km/h, here the drag and lift coefficients are summarized in table 4 for model

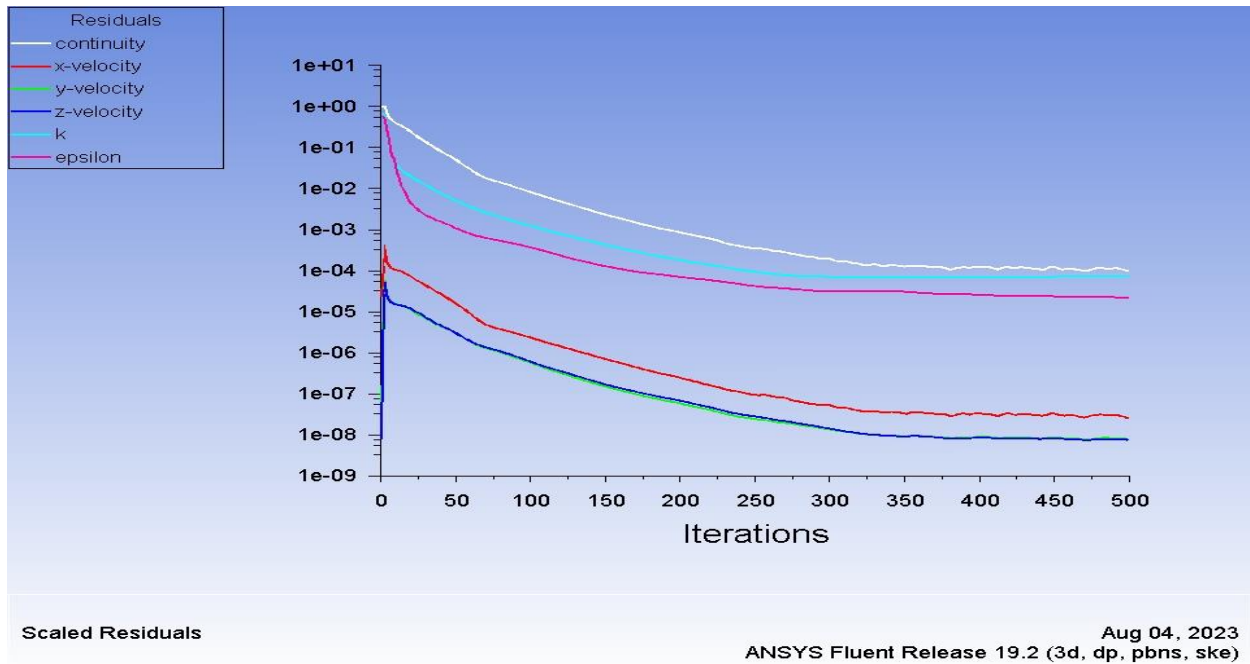


Figure 59: Scale Residual with Delta-type vortex generator

The scale residual with a delta-shaped vortex generator is shown in Figure 59.

Table 4: Drag and Lift Coefficient of Pickup with Delta VGs at Different Speeds

	Speed(km/h)	CD	CI
	80	0.3818	0.1375
	100	0.3821	0.1380
	120	0.3826	0.1381
Average	80, 100, 120	0.3822	0.1378

The drag coefficient of the pickup model with delta-shaped vortex generator (Table 4) is reduced than both the baseline model (Table 3) (the average drag coefficient at 80, 100, and 120 km/h speeds) for the baseline model to 0.3822 (the average drag coefficient at 80, 100, and 120 km/h speeds) for the model with delta-shaped vortex generator, thus the average reduction in Cd is 6.25% The drag and lift coefficient of pickup model t with delta-shaped type vortex generator in different parameters of data and results set as given in Table 4.

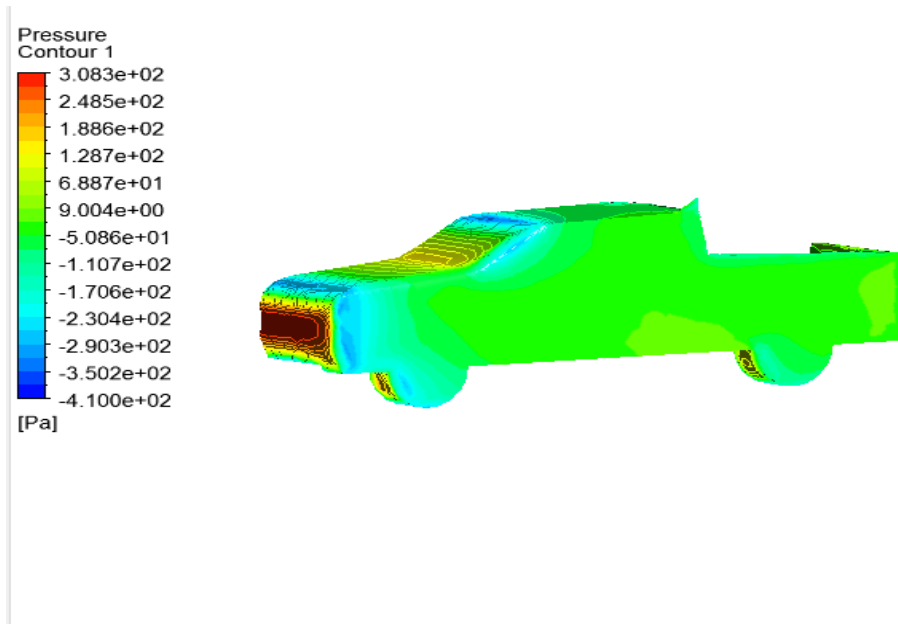


Figure 60: Pressure Contour in the car with delta-type vortex generator

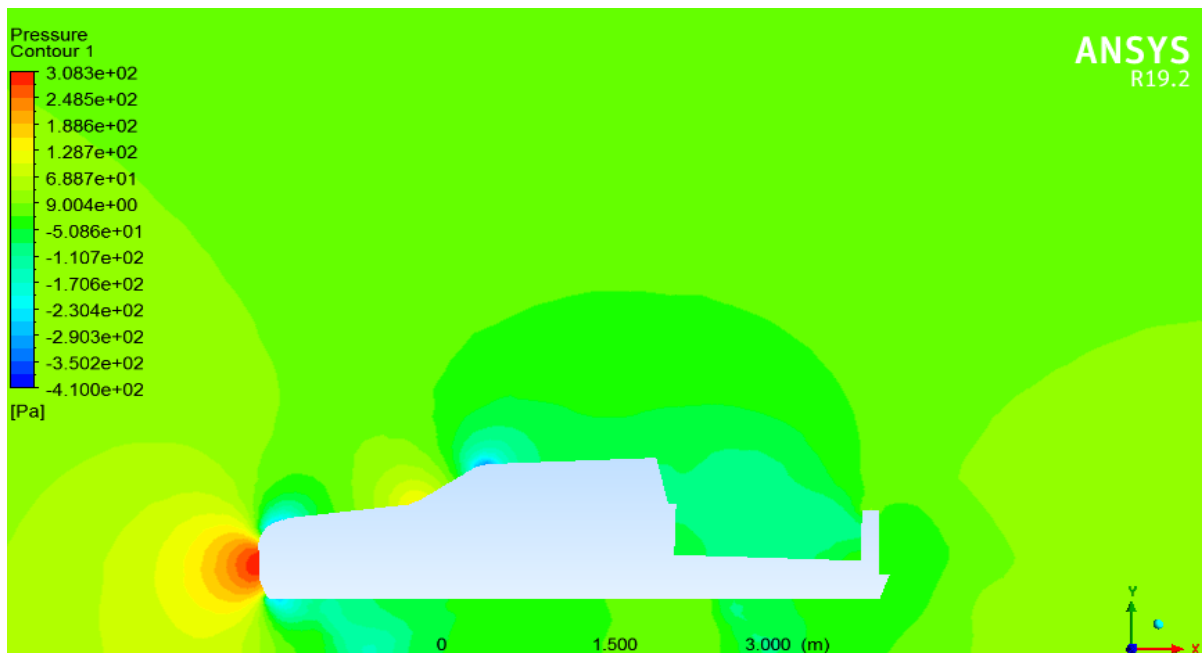


Figure 61: Pressure Contour in Symmetry with delta-type vortex generator Plane at 80 km/h

The pressure contour in the car plane and symmetry plane of the pickup model with a delta-shaped vortex generator at 80 km/h are shown in Figures 60 and 61.

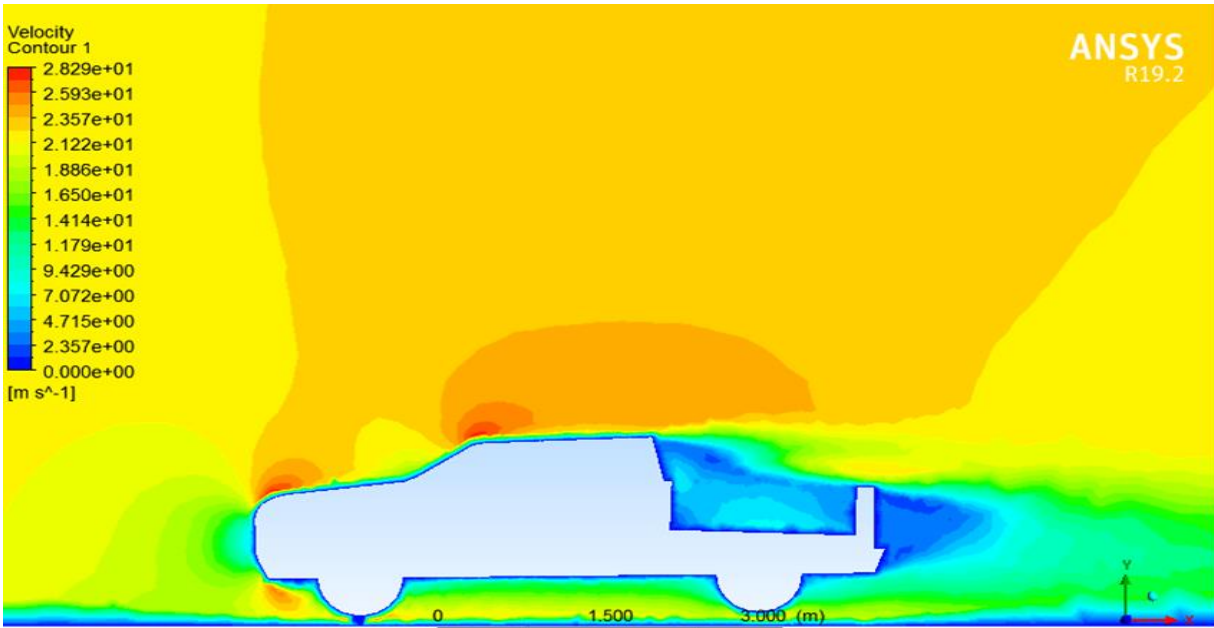


Figure 62: Velocity contour in symmetry plane with delta type at 80 km/h

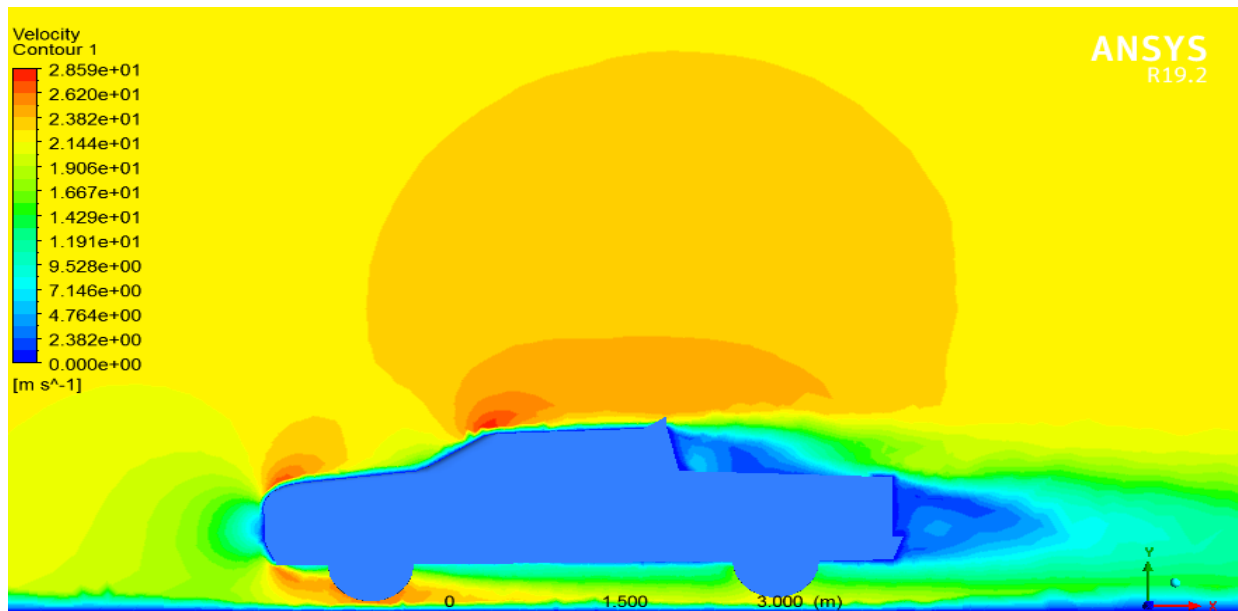


Figure 63: Velocity Contour in the car with delta-type vortex generator at 80 km/h

The pickup model with a delta-shaped vortex generator is the modification of the baseline model in which the implication of the roof end has been changed by a passive add-on device. The top and rear roof end, body, and model box surfaces are made curvy as compared to the baseline model. The test procedure was the same as the baseline model and model. The flow separation at the top and rear roof end parts of the pickup has reduced due to the added vortex generator on the end parts. Consequently, this minimizes the low-velocity area in the wake region and delays

the turbulence zone away from the pickup model. Velocity contour in the symmetry plane and car plane of the pickup model with a delta-shaped vortex generator at 80 km/h are shown in Figures 62, and 63 presented.

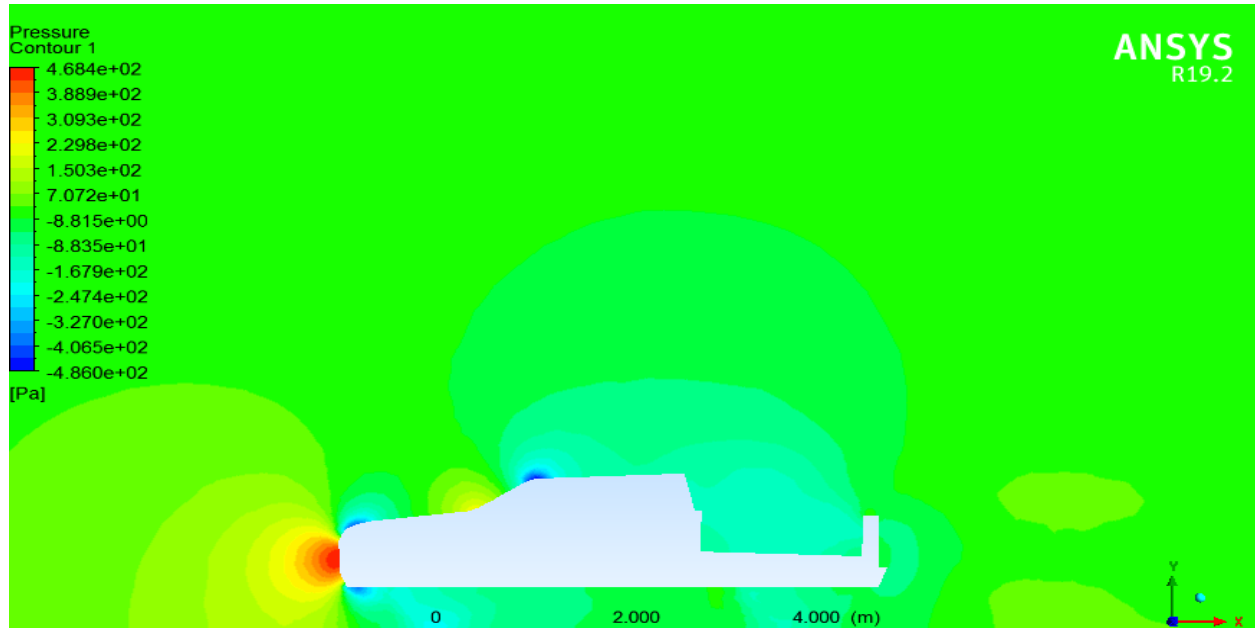


Figure 64: Pressure Contour in Symmetry with delta-type vortex generator Plane at 100 km/h

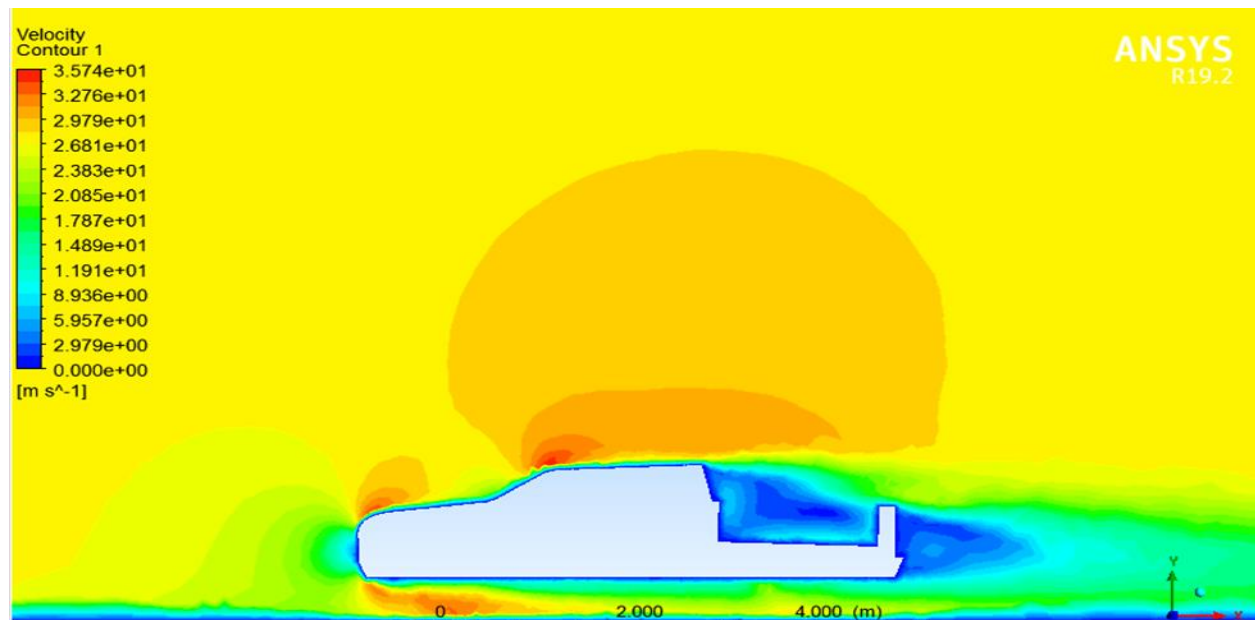


Figure 65: Velocity contour in symmetry plane with delta type at 100 km/h

With the increasing speed of the model, the distribution of flow separation with delta delta-shaped vortex generator is changed. The velocity contour in the car plane and symmetry plane of

the pickup model with a delta-shaped vortex generator at 100 km/h are shown in Figures 64, and 65 respectively.

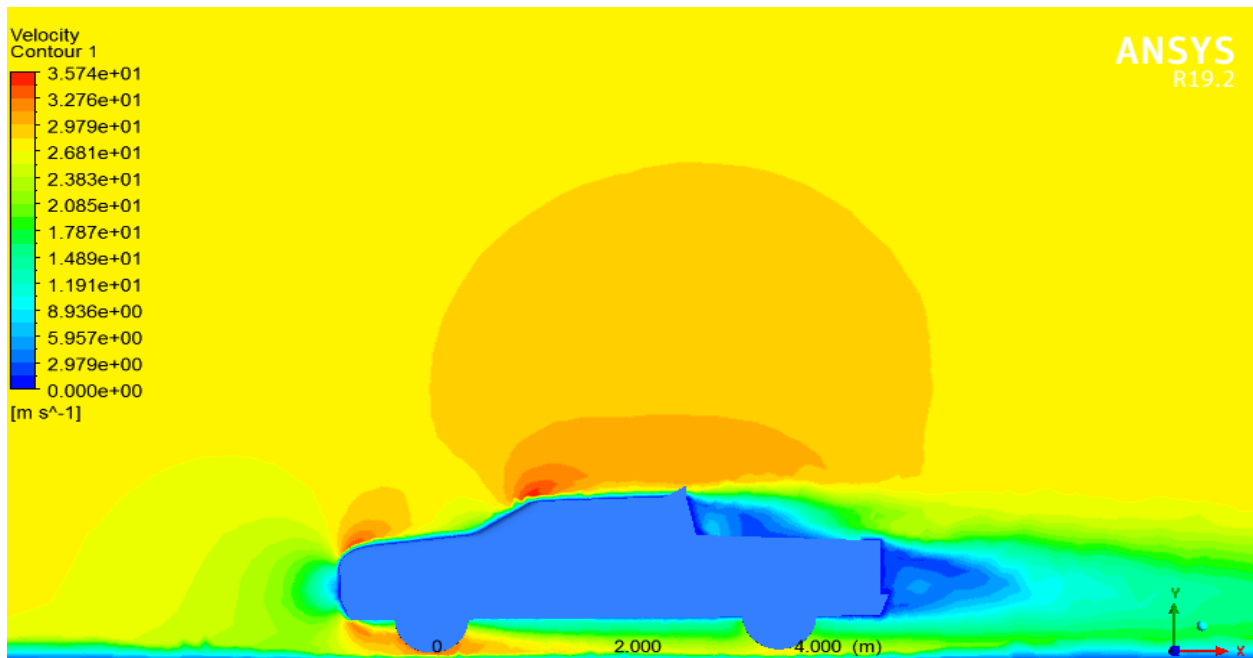


Figure 66: Velocity Contour in the car with delta-type vortex generator at 100 km/h

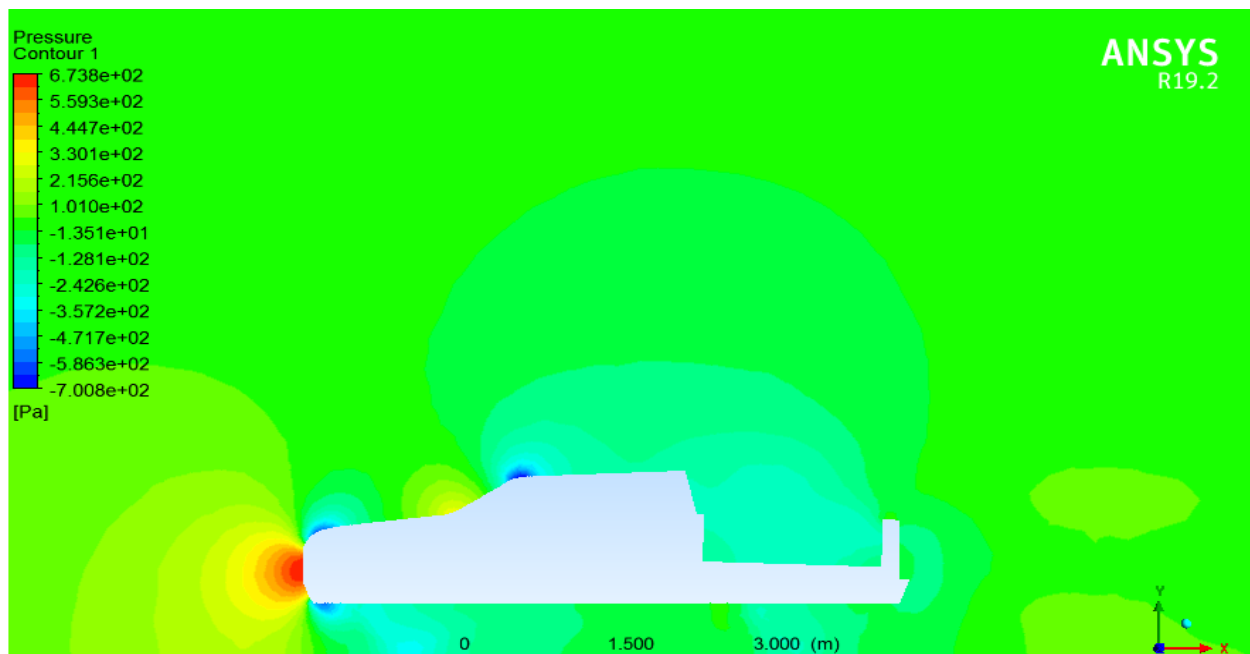


Figure 67: Pressure Contour in Symmetry with delta-type vortex generator Plane at 120 km/h

The velocity contour in the car plane and pressure contour in the symmetry plane of the pickup model with a delta-shaped vortex generator at 120 km/h are shown in Figures 66, and 67 respectively.

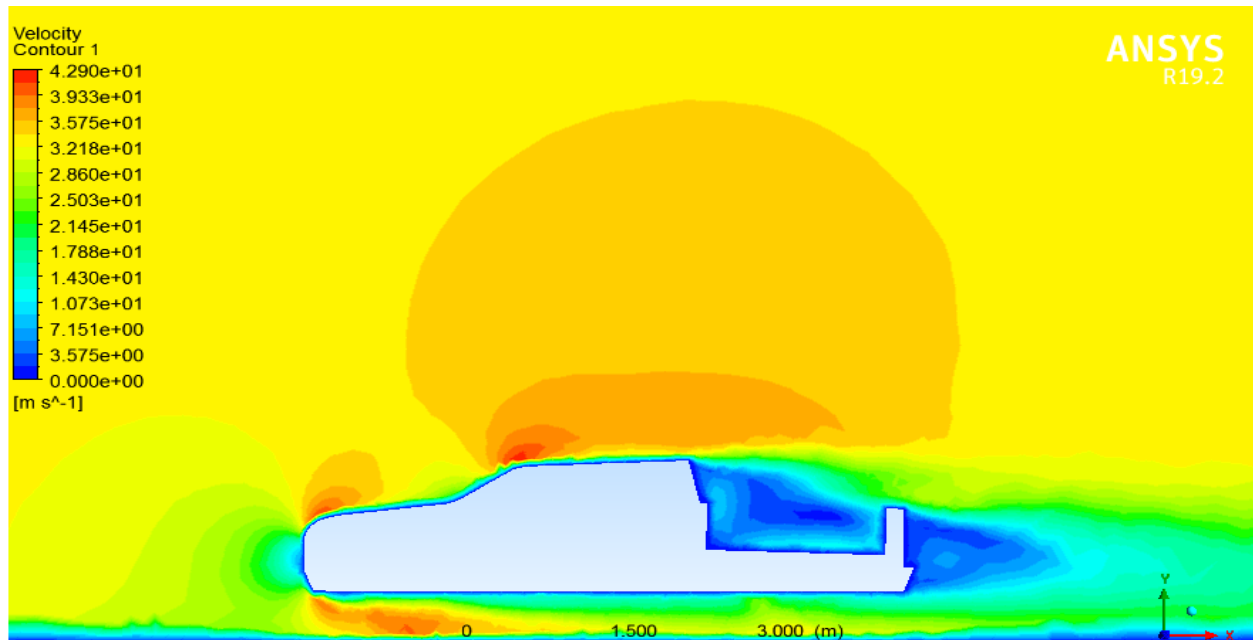


Figure 68: Velocity Contour in the car with delta-type vortex generator at 120 km/h

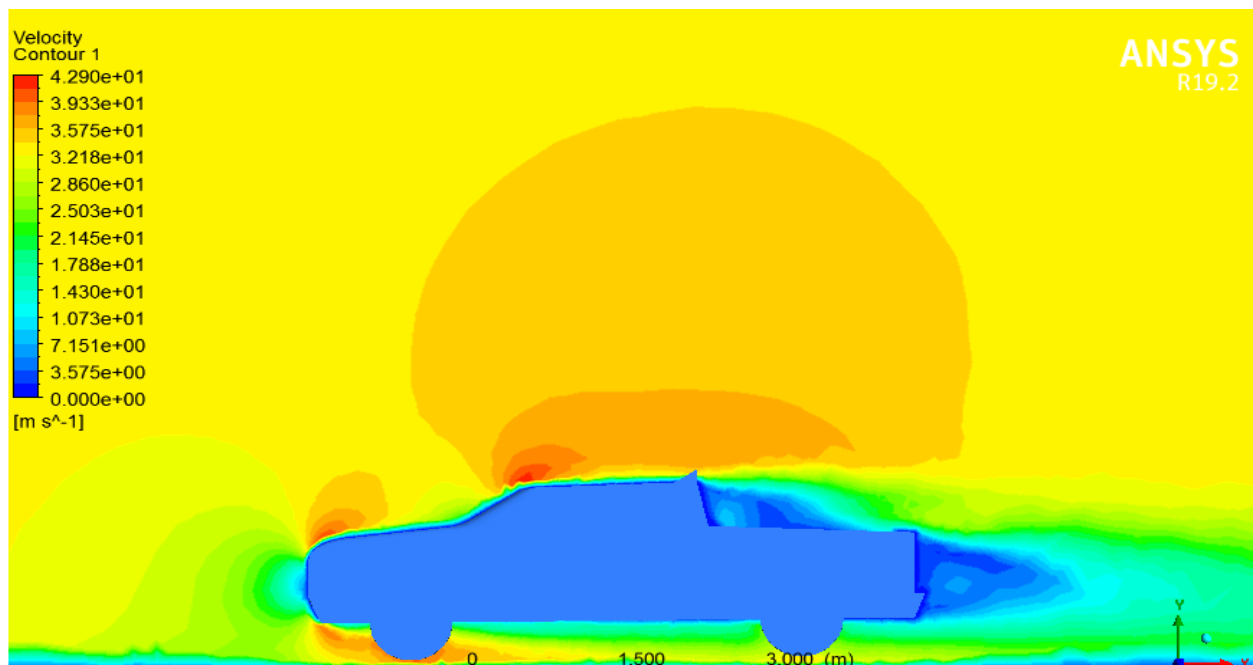


Figure 69: Velocity Contour in the car with delta-type vortex generator at 120 km/h

The velocity contour symmetry plane and car plane of the pickup model with a delta-shaped vortex generator at 120 km/h are shown in Figures 68, and 69 respectively.

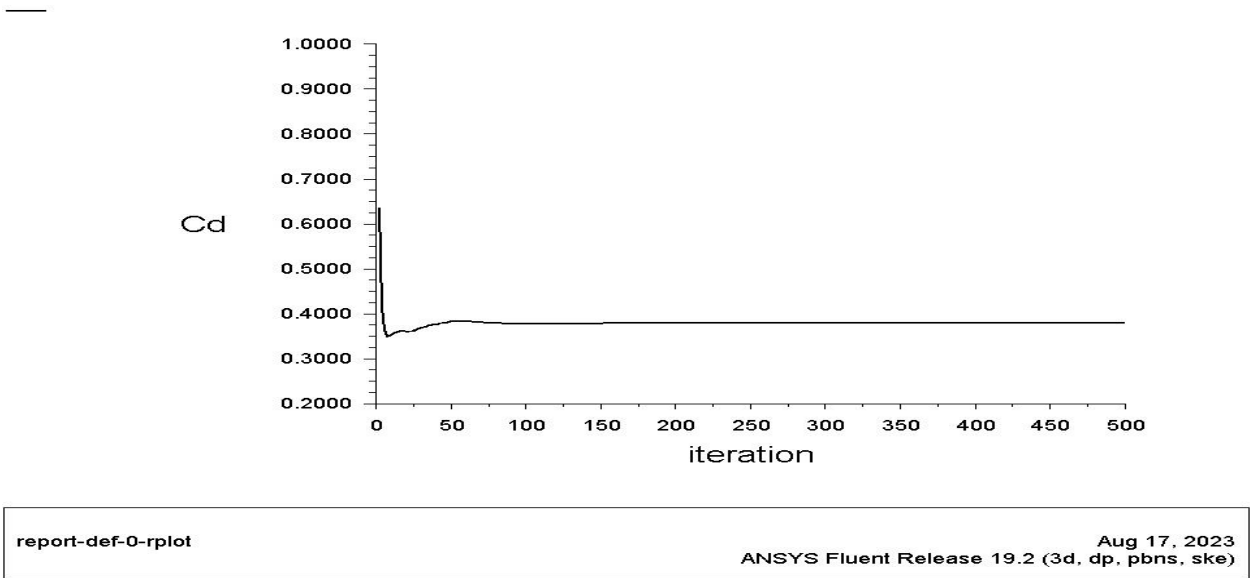


Figure 70: Drag Coefficient with Triangular vortex generator at 80 km/h

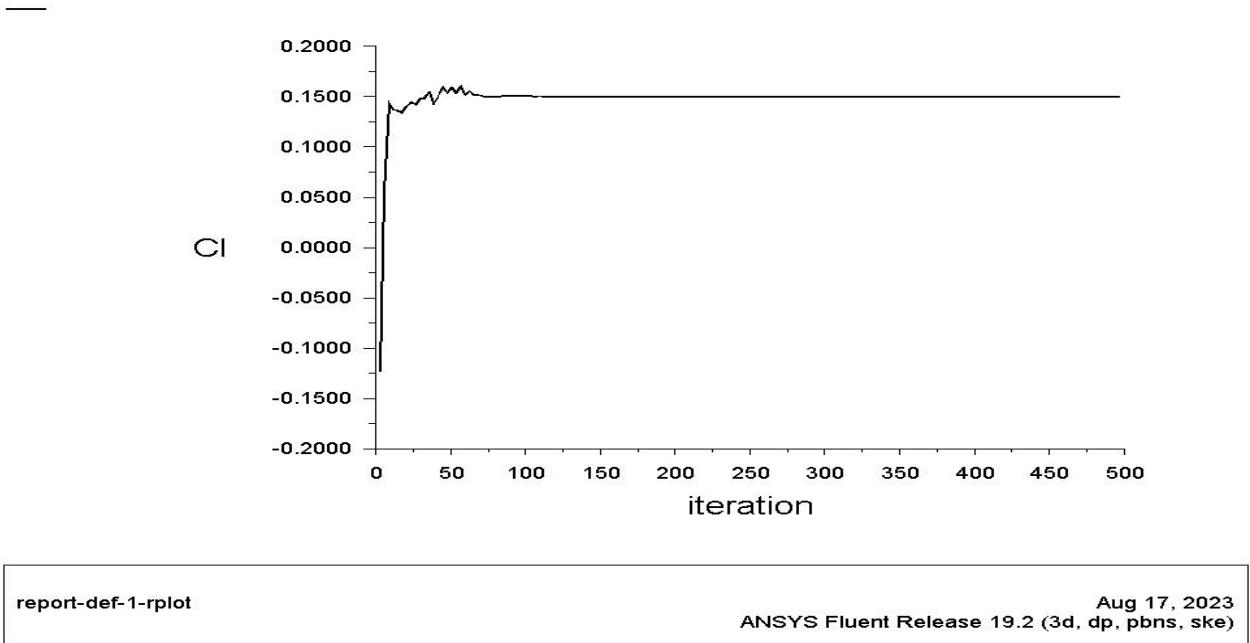
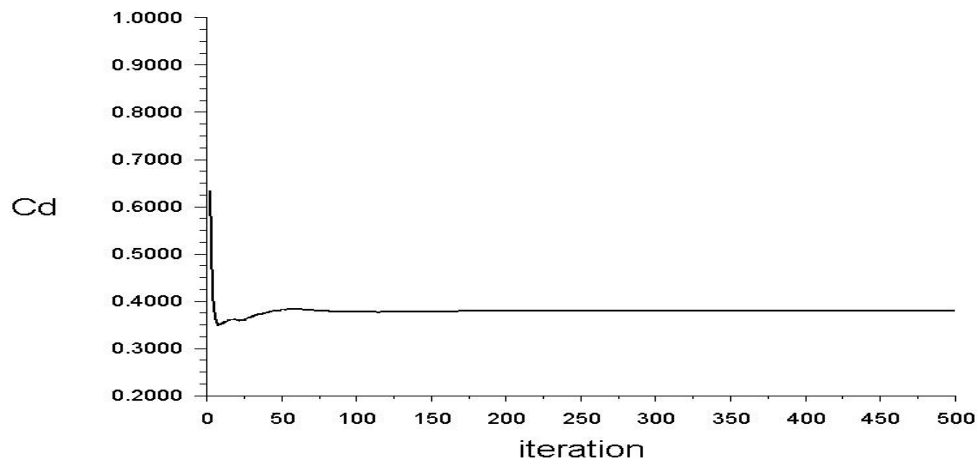


Figure 71: Lift Coefficient with Triangular vortex generator at 80 km/h

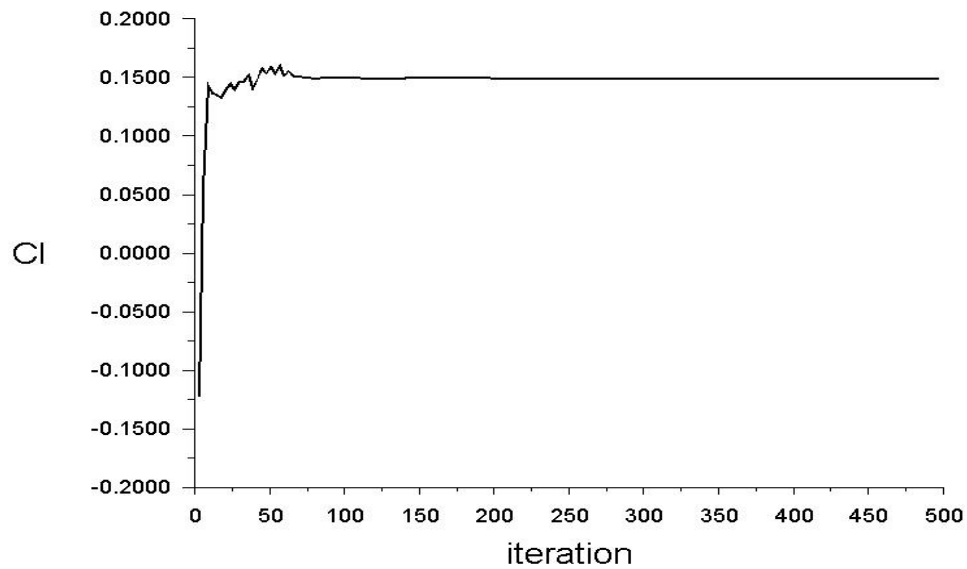
The drag and lift coefficient graph of the pickup model with a triangular-shaped vortex generator at 80 km/h are presented in Figures 70, and 71 respectively.



report-def-0-rplot

Aug 17, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 72: Drag Coefficient with Triangular vortex generator at 100 km/h

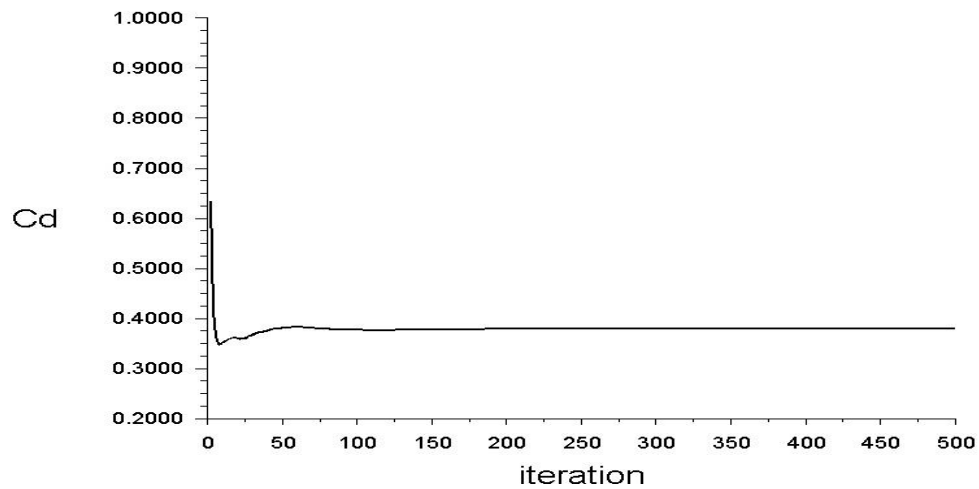


report-def-1-rplot

Aug 17, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 73: Lift Coefficient with Triangular vortex generator at 100 km/h

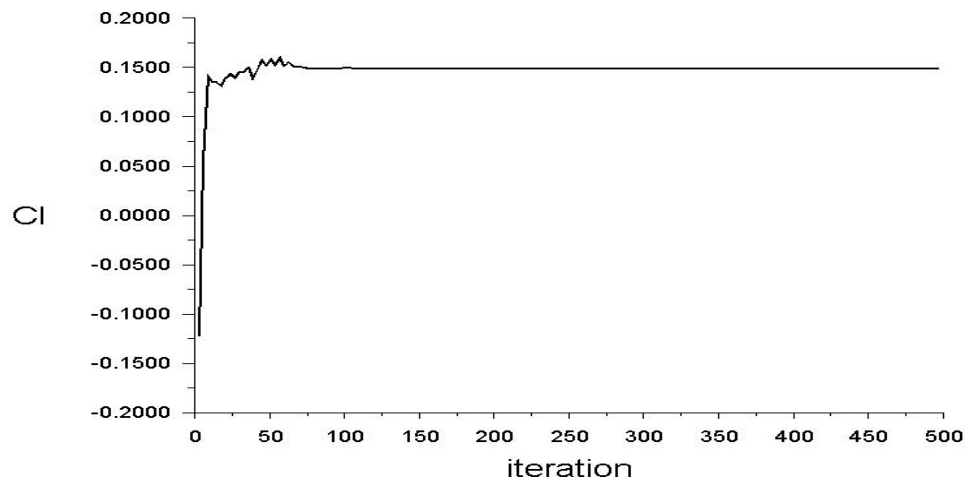
The drag and lift coefficient graph of the pickup model with a triangular-shaped vortex generator at 100 km/h are shown in Figure 72, and Figure 73.



report-def-0-rplot

Aug 18, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 74: Drag Coefficient with Triangular vortex generator at 120 km/h



report-def-1-rplot

Aug 18, 2023
ANSYS Fluent Release 19.2 (3d, dp, pbns, ske)

Figure 75: Lift Coefficient with Triangular vortex generator at 120 km/h

All the drag and lift coefficient graphs shown in Figure 70-75 graphs of pickup model with triangular vortex generator at speeds of 80, 100, and 120 km/h are summarized in Table 5. Drag and lift coefficient graph of the pickup model with triangular-shaped vortex generator at 120 km/h is shown in Figure 74, and Figure 75 respectively.

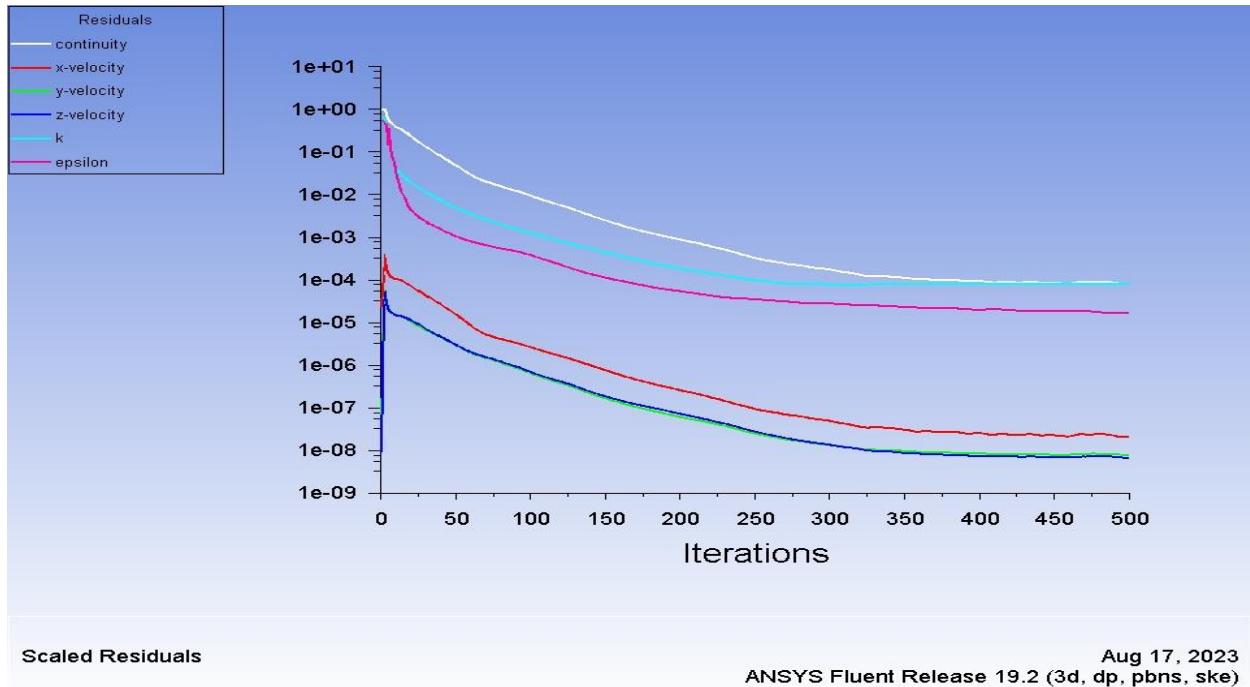


Figure 76: Scale Residual in application of Triangular vortex generator

The scale of the pickup model with triangular-shaped vortex generators is shown in Figure 76.

Table 5: Drag and Lift Coefficient of Pickup with Triangular VGs at Different Speeds

Model	Speed(km/h)	CD	CI
Pickup model with triangular vortex generators	80	0.3809	0.1503
	100	0.3813	0.1494
	120	0.3864	0.1482
Average	80,100,120	0.3810	0.1494

The drag coefficient of the pickup model with a triangular vortex generator is reduced than the baseline model (Table 3), and the model with a triangular-shaped vortex generator (Table 5) and model. The drag coefficient has reduced from 0.4075 (the averaged drag coefficient at 80, 100, and 120 km/h speeds) for the baseline model to 0.3810 (the average drag coefficient at 80, 100, and 120km/h speeds) thus the average reduction in with triangular vortex generator is 6.50%, and for model of the pickup with triangular-shaped vortex generator Simulation results of Drag and lift coefficient with triangular-shaped vortex generator is set to given as in Table 5.

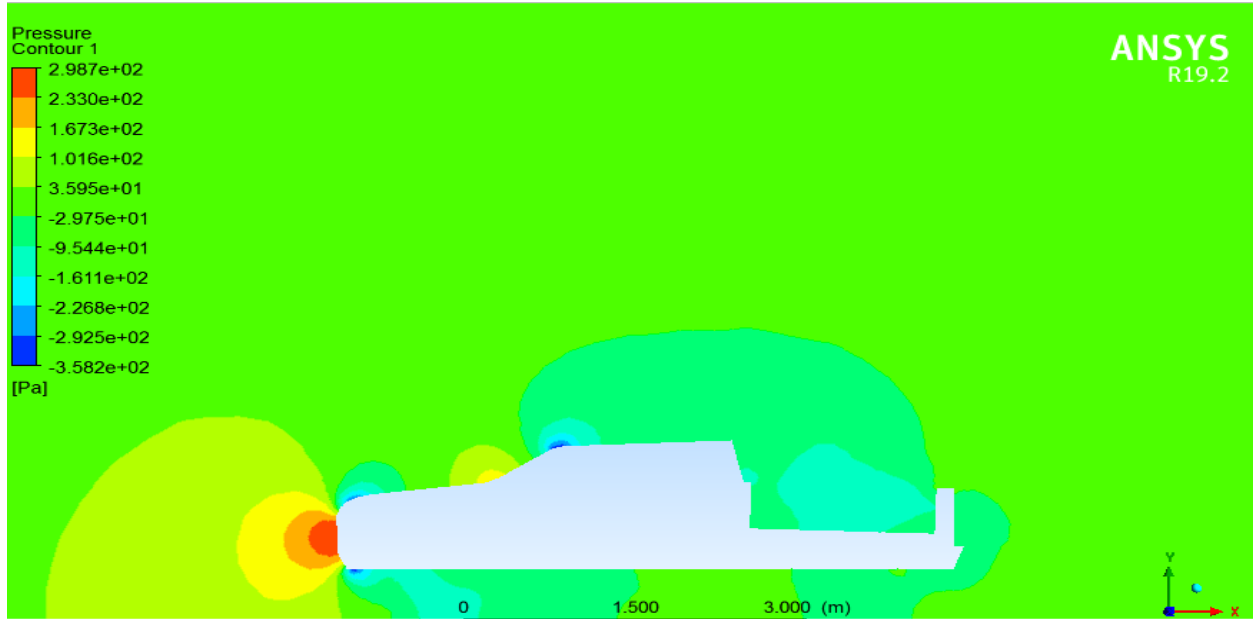


Figure 77: Pressure contour in symmetry plane with triangular vortex generator at 80 km/h

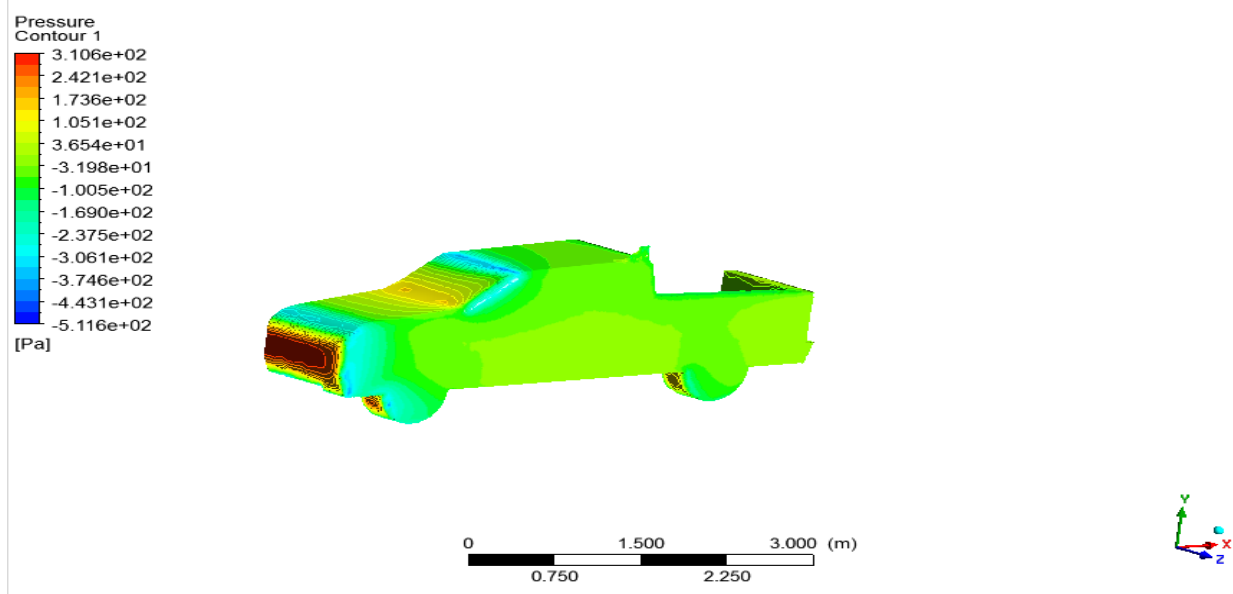


Figure 78: Pressure contour in car with triangular vortex generator at 80 km/h

The contour of pressure distribution shown in Figures 77 and 78 indicates the level of pressure effect on the surface of the pickup model with a triangular-shaped vortex generator. From the figures, it can be seen how the pressure increases with increasing speed. The drag coefficient of model three increases with the increase in speed. The pressure contour in symmetry and car plane of the pickup model with a triangular-shaped vortex generator at 80 km/h are shown in Figure 77, and Figure 78.

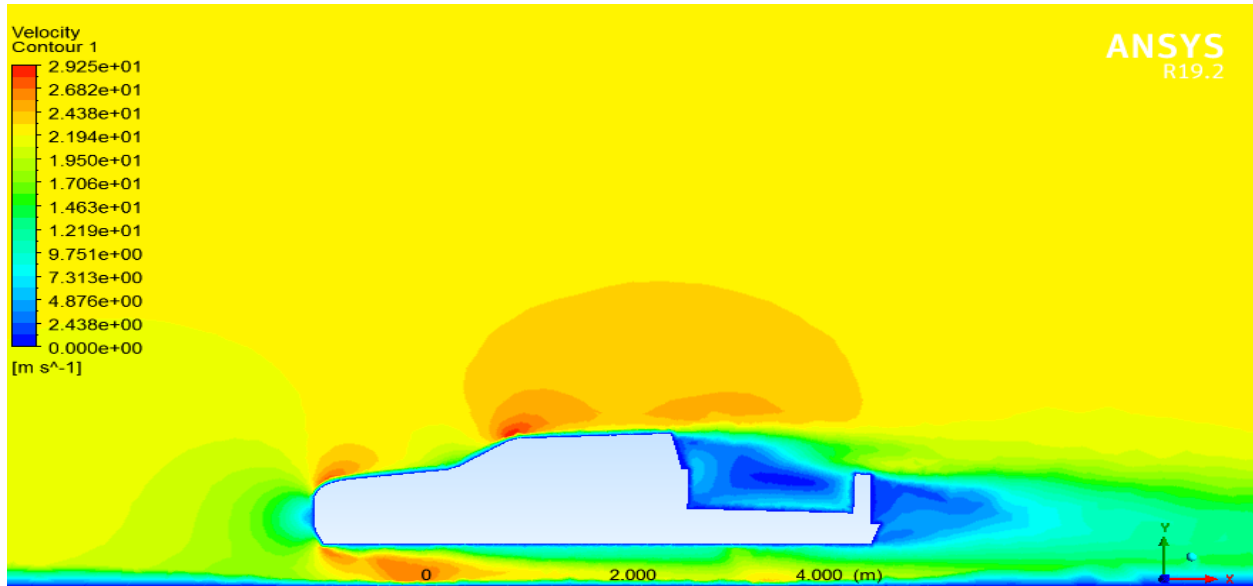


Figure 79: Velocity contour in symmetry plane with triangular vortex generator at 80 km/h

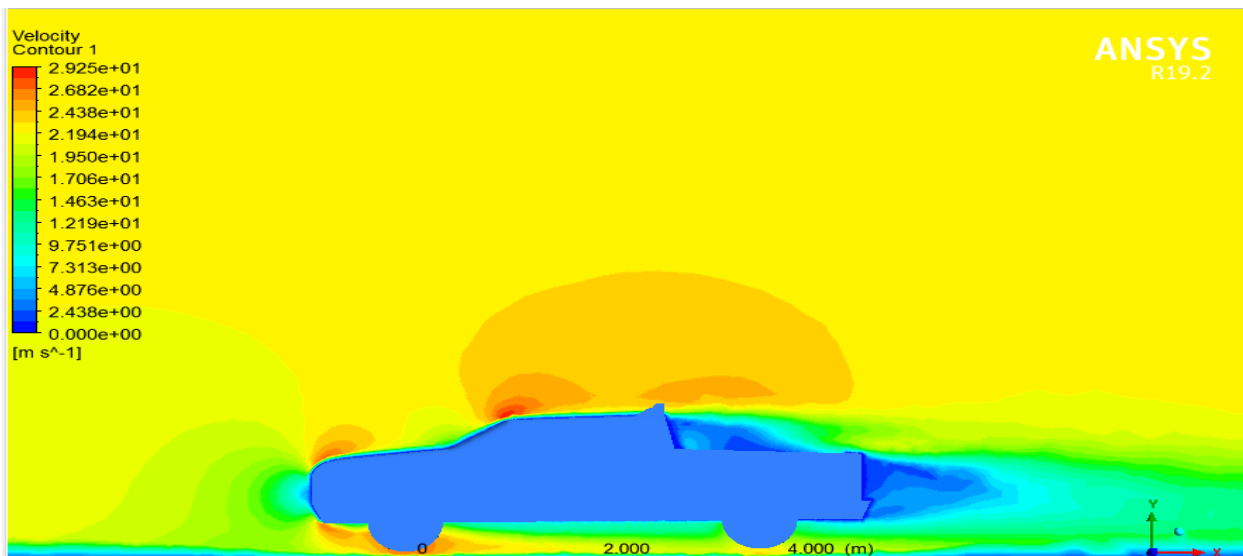


Figure 80: Velocity contour in the car with triangular vortex generator at 80 km/h

This model is the last in the study. This model is the same as that of previous models the only difference is the triangular vortex generator added to the model. Due to the presence of a vortex generator in the roof end of the pickup, consequently, which minimizes the low-velocity area in the wake region and delays the turbulence zone away from the pickup model. In all models of the pickup velocity contour shows that the wake region gets stronger and wider as the speed increases. Velocity contour in symmetry and car plane of the pickup model with triangular-shaped vortex generator at 80 km/h are shown in Figure 79, and Figure 80.

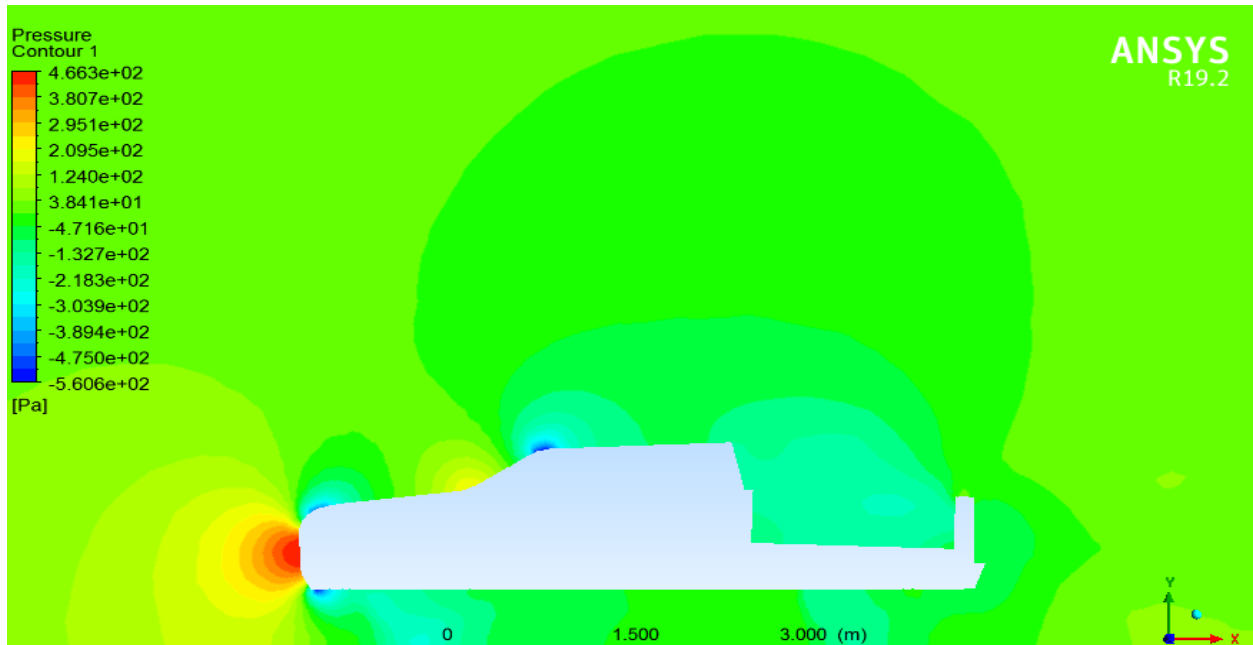


Figure 81: Pressure contour in symmetry plane with triangular vortex generator at 100 km/h

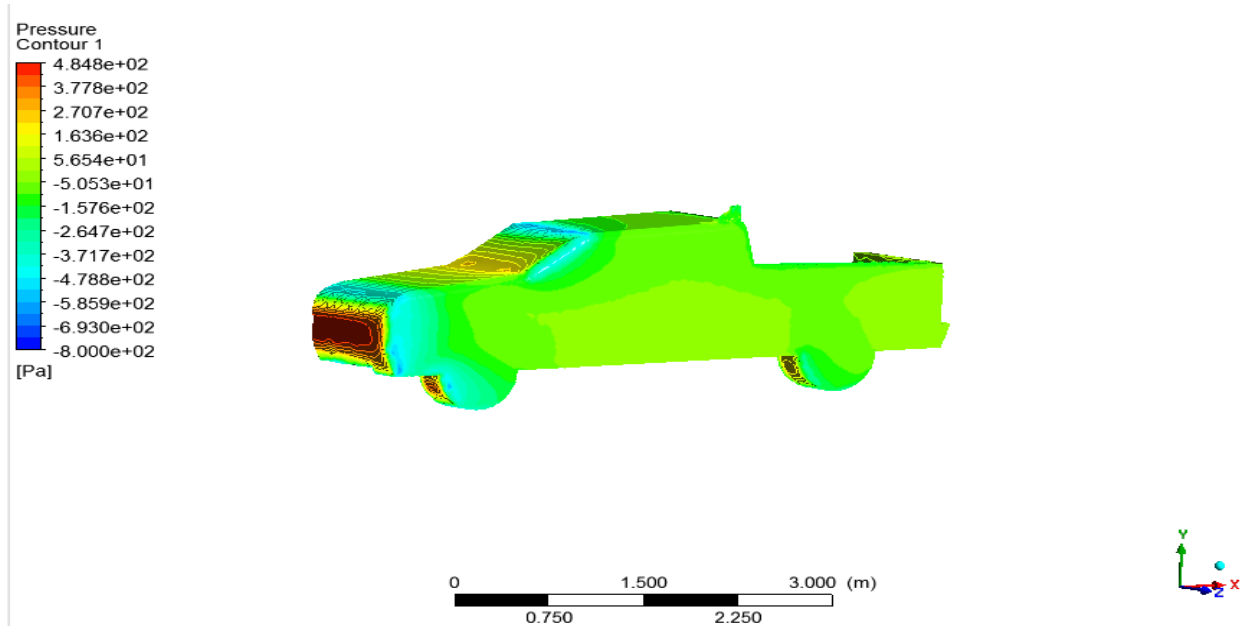


Figure 82: Pressure contour in car with triangular vortex generator at 100 km/h

When the speed of the model increases the stagnation point similarly increases in front of the body. Pressure contour in symmetry and car plane of pickup model with triangular-shaped vortex generator at 100 km/h are shown in Figure 81, and Figure 82

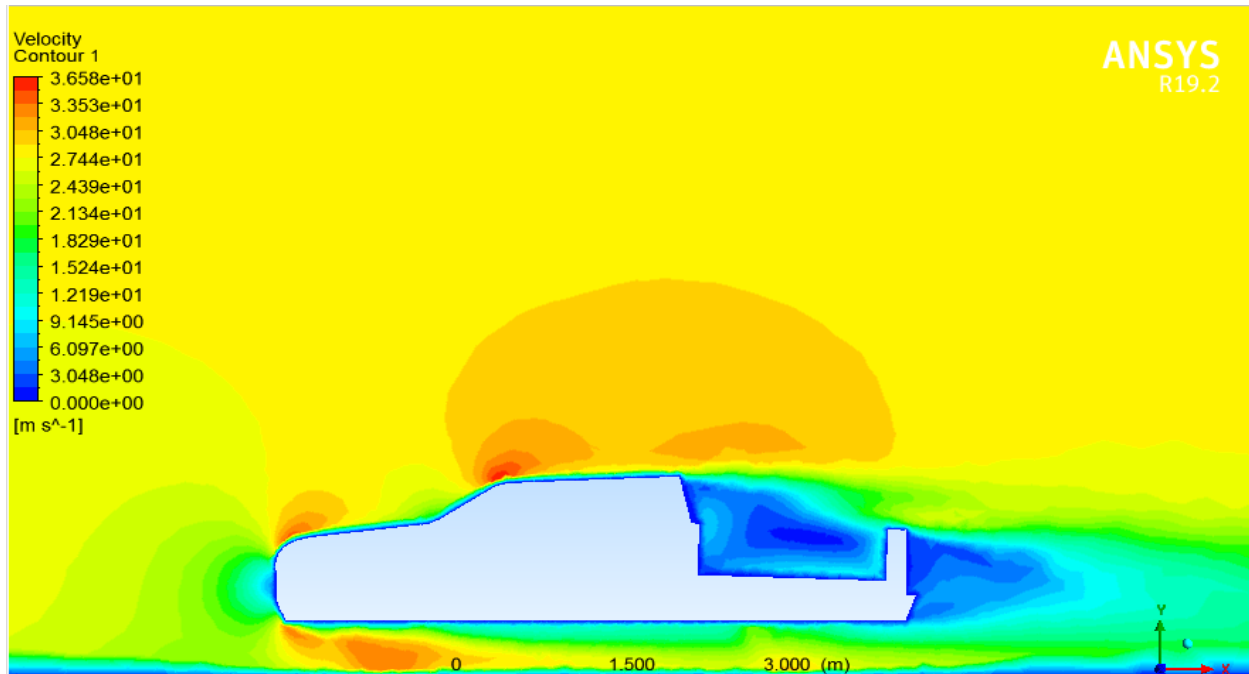


Figure 83: Velocity contour in symmetry plane with triangular vortex generator at 100 km/h

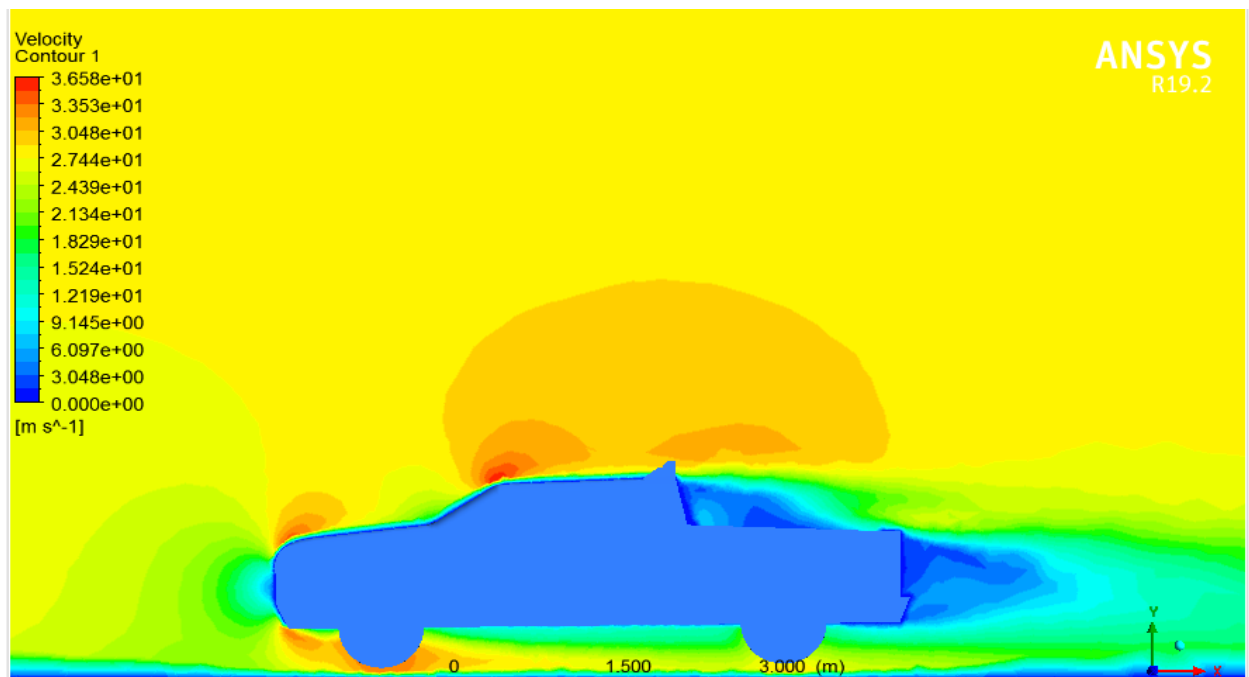


Figure 84: Velocity contour in the car with triangular vortex generator at 100 km/h

Velocity contour in symmetry and car plane of the pickup model with triangular-shaped vortex generator at 100 km/h are shown in Figure 83, and Figure 84 respectively.

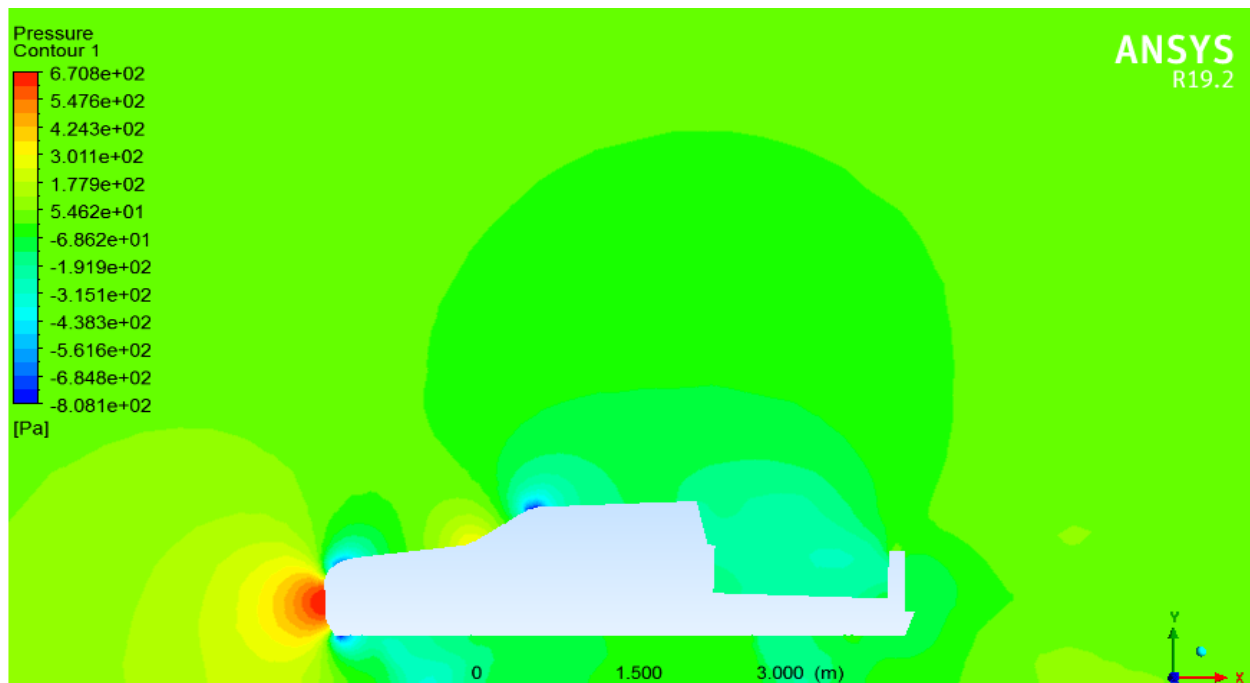


Figure 85: Pressure contour in symmetry plane with triangular vortex generator at 120 km/h

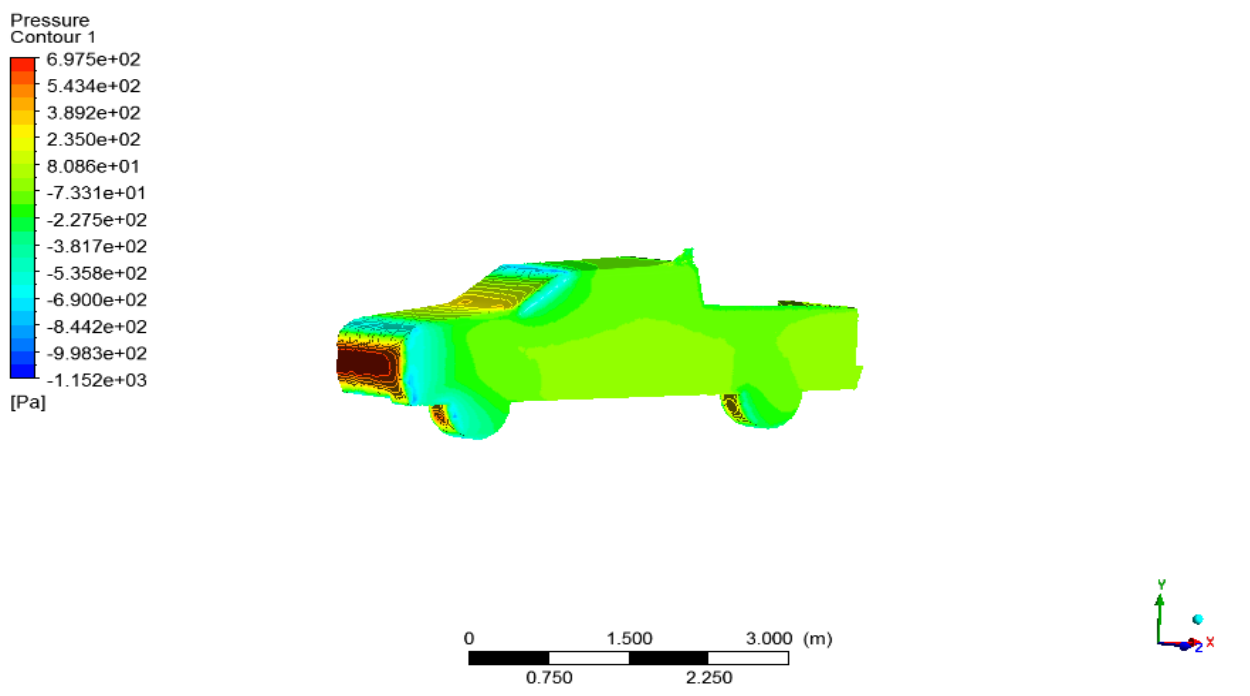


Figure 86: Pressure contour in car with triangular vortex generator at 120 km/h

The pressure contour in symmetry and car plane of the pickup model with a triangular-shaped vortex generator at 120 km/h are shown in Figure 85, and Figure 86.

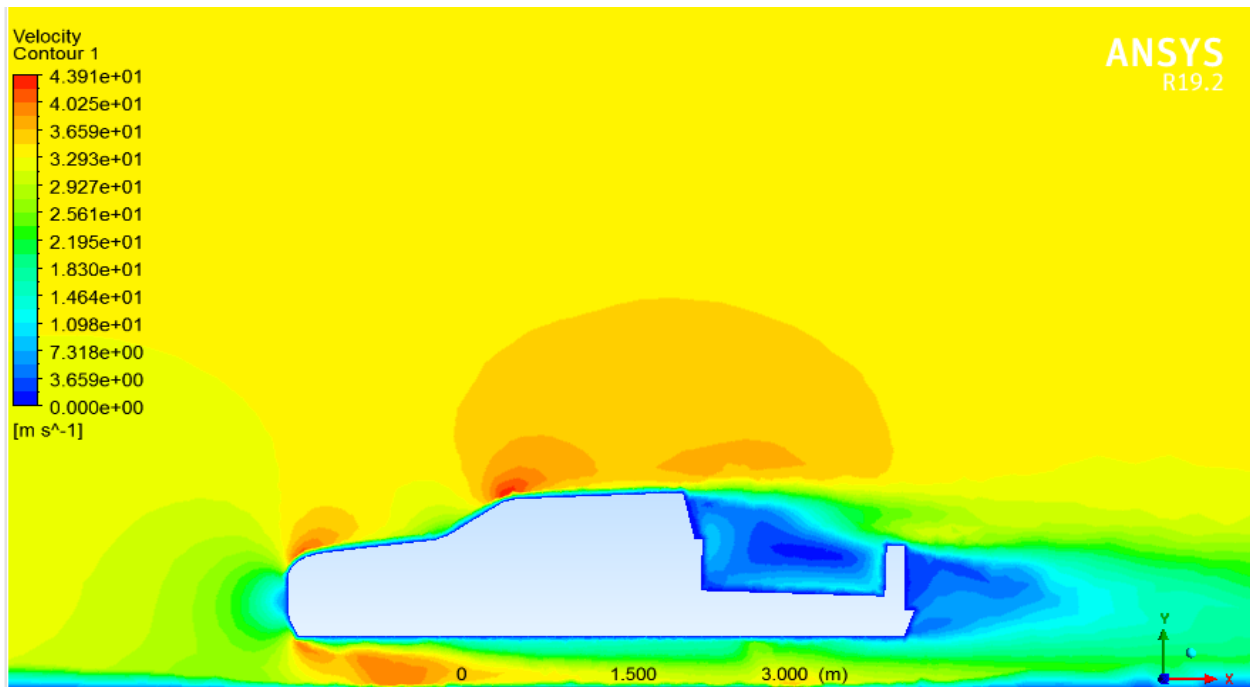


Figure 87: Velocity contour in symmetry plane with triangular vortex generator at 120 km/h

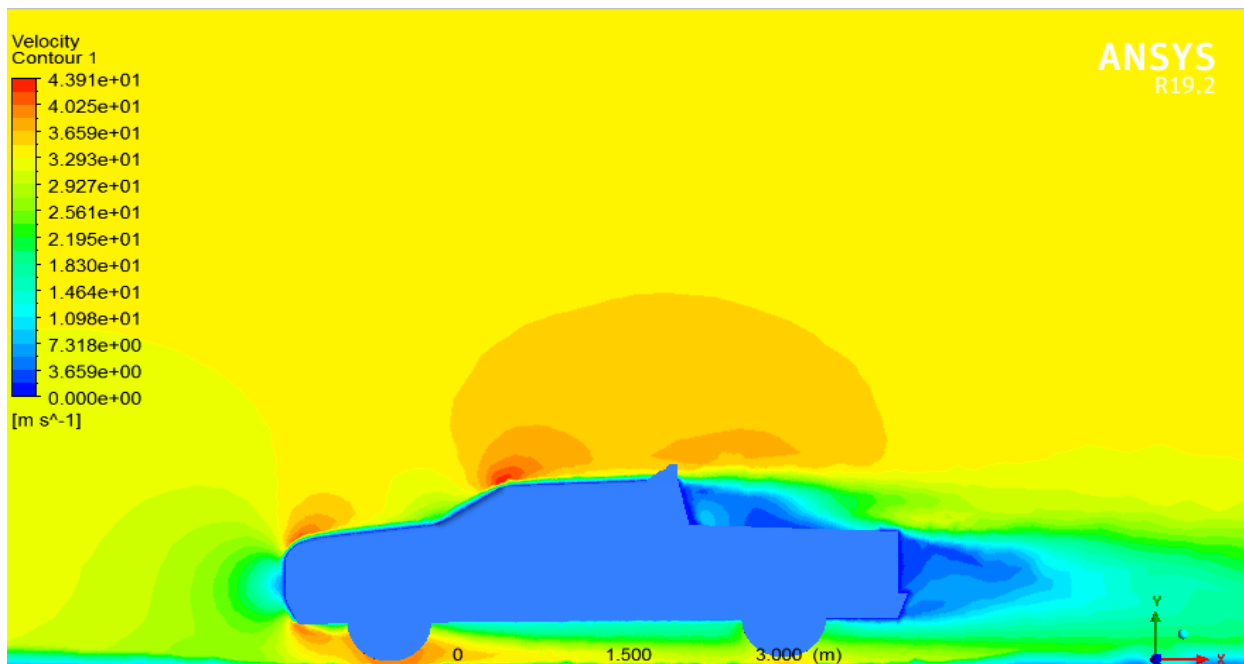
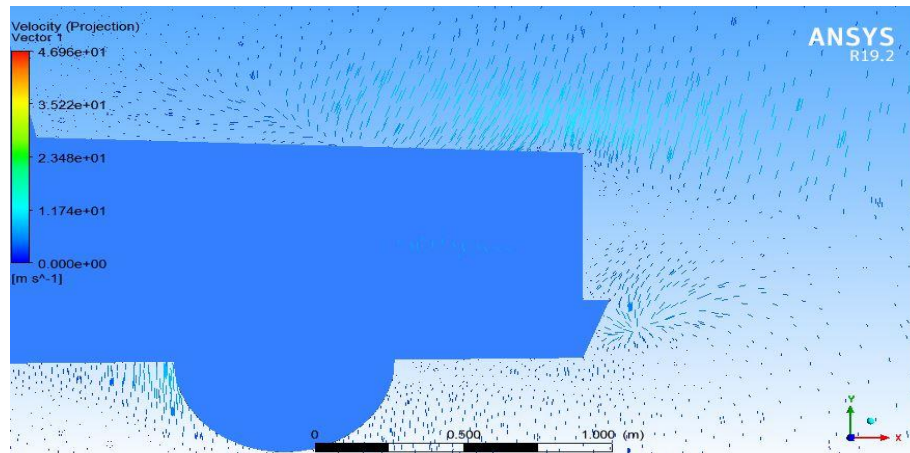
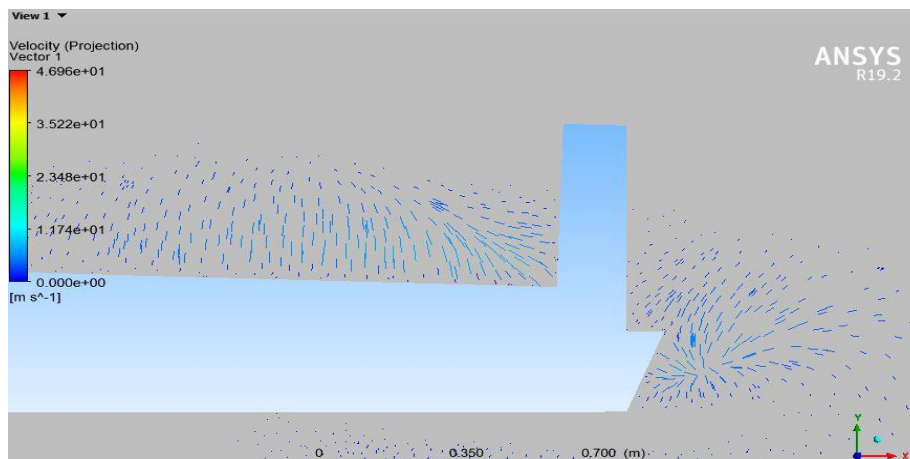


Figure 88: Velocity contour in the car with triangular vortex generator at 120 km/h

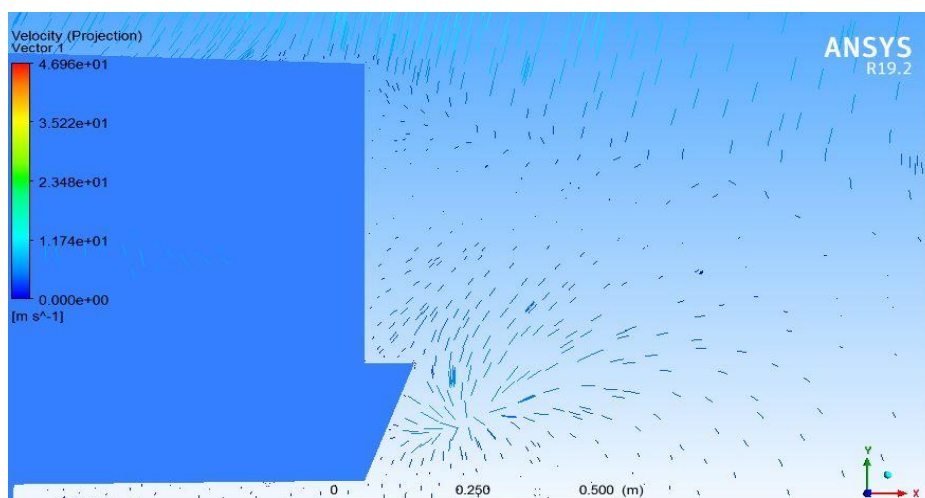
Velocity contour in symmetry and car plane of the pickup model with triangular-shaped vortex generator at 120 km/h are shown in Figure 87, and Figure 88 respectively.



(a)



(b)



(c)

Figure 89: Velocity vector effect on the pickup model with triangular shaped-vortex generator

Table 6: Drag and Lift coefficient result of all models [95]

Model	Speed(km/h)	CD	CI	Lift force(N)	Drag force(N)
Baseline	80	0.4071	0.1550	81.53	224.50
	100	0.4075	0.1498	126.65	350.10
	120k	0.4081	0.1497	181.45	503.78
With Delta vg	80	0.3818	0.1375	79.61	215.46
	100	0.3821	0.1380	123.72	336.15
	120	0.3826	0.1381	177.09	483.50
With Triangular vg	80	0.3809	0.1503	88.44	221.49
	100	0.3813	0.1494	137.61	345.61
	120	0.3864	0.1482	197.53	497.29
Average	80,100,120	0.3810	0.1378		

Drag force (N), Lift force (N), Drag coefficient, and lift coefficient of all models with different speeds data as set given in Table 6.

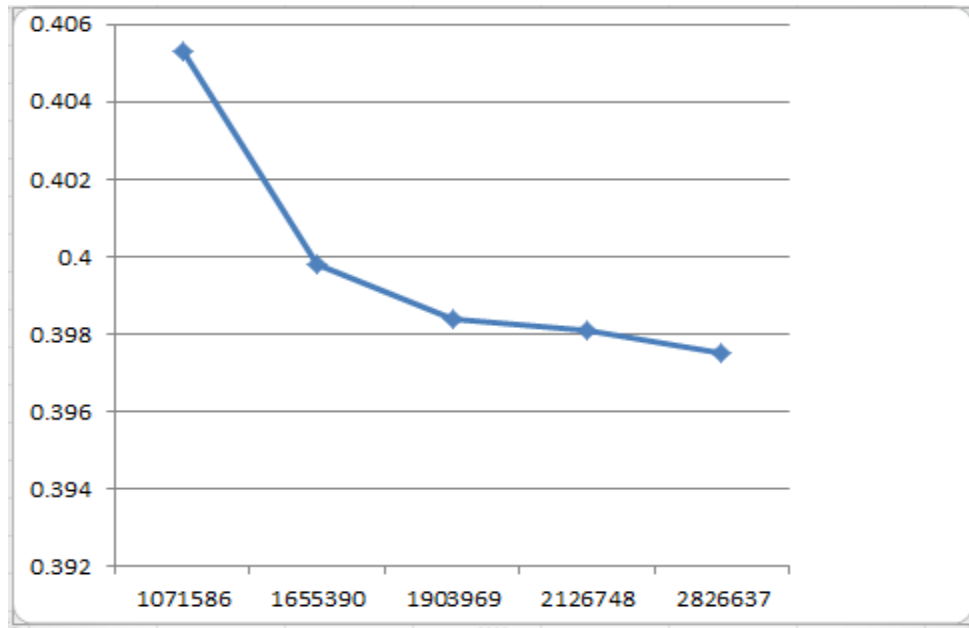


Figure 90: Mesh grid independence test of the baseline model

Applying different numbers of body and model size numbers on the boundary condition coefficient of drag depends on the mesh element numbers. The mesh grid independence test of the baseline model with different mesh element numbers is shown in Figure 90.

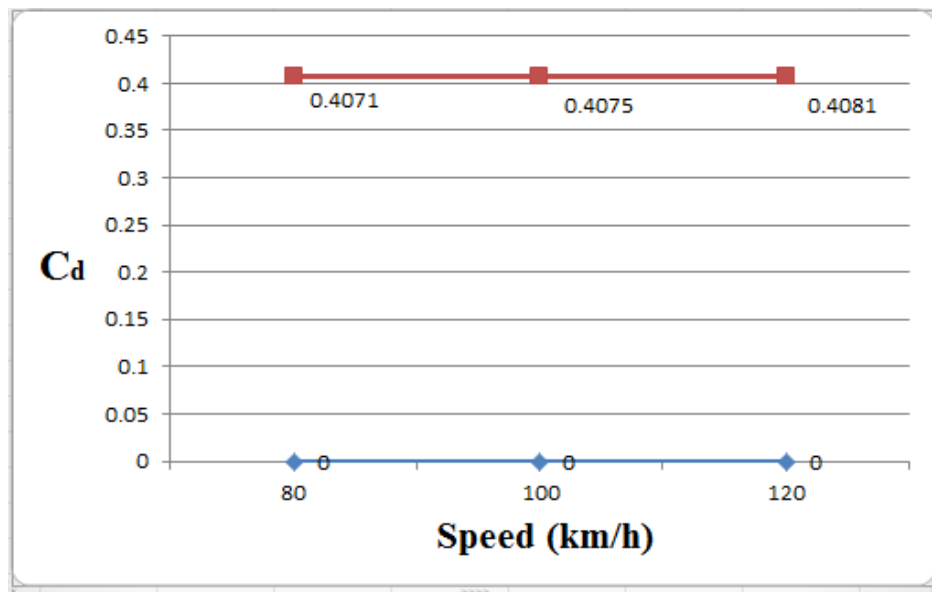


Figure 91: Drag coefficient and speed comparisons of the baseline model

A comparison of drag coefficient results with the increasing of speed, when the vehicle speed increases the same the drag coefficient is increased. The drag coefficient of baseline model comparisons with a given speed is shown in Figure 91.

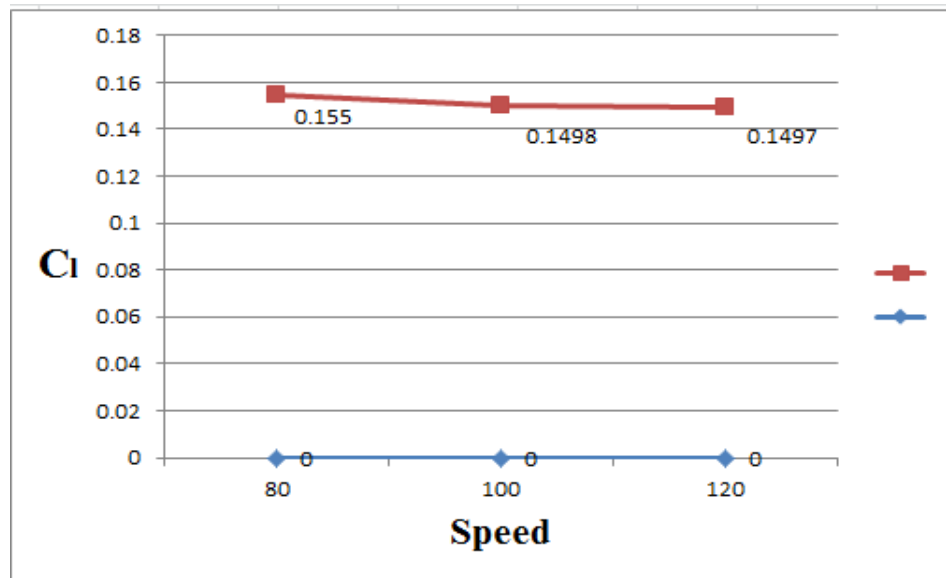


Figure 92: Lift coefficient and speed comparisons of the baseline model

The lift coefficient of the baseline model concerning speed is compared in Figure 92, without the vortex generator and the blue line indicates the increasing speed of the model with the lift coefficient result.

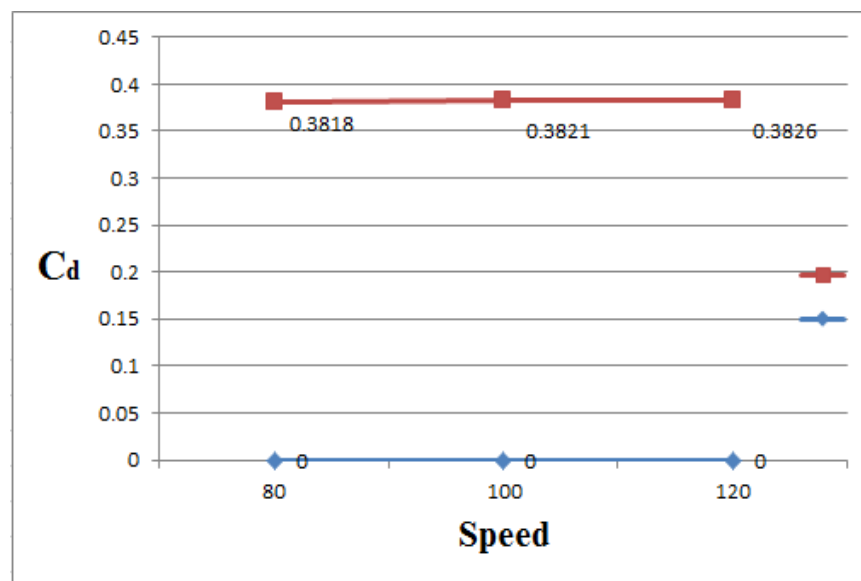


Figure 93: Drag coefficient and speed comparisons with delta-type vortex generator

A Drag coefficient of the baseline model with the implication of delta-shaped vortex generator drag concerning speed is shown in Figure 93. The blue line illustrates the speed of the vehicle and the red line represents the drag coefficient of the model with a delta-shaped vortex generator.

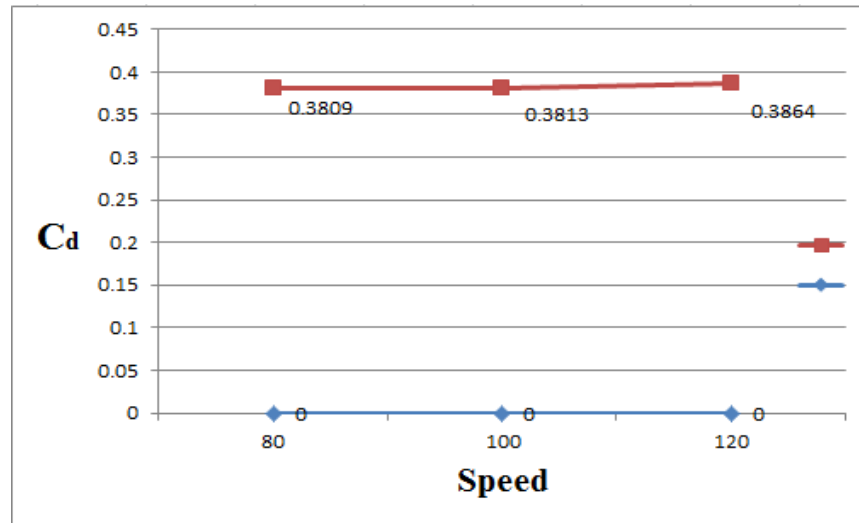


Figure 94: Drag coefficient and speed comparisons with triangular-type vortex generator

When with an average speed of 80,100 and 120 km/h adding a triangular vortex generator to the model the coefficient of drag is reduced from 0.4075 to 0.3810, However, the effect of vortex generator indicates that passive flow control devices are the most important devices to reduce coefficient of drag according to the figure the vertical line indicates the speed and the red line also shows the reduced drag coefficient with the comparison of pickup vehicle model with triangular-shaped vortex generator drag coefficient and speed is shown in Figure 94.

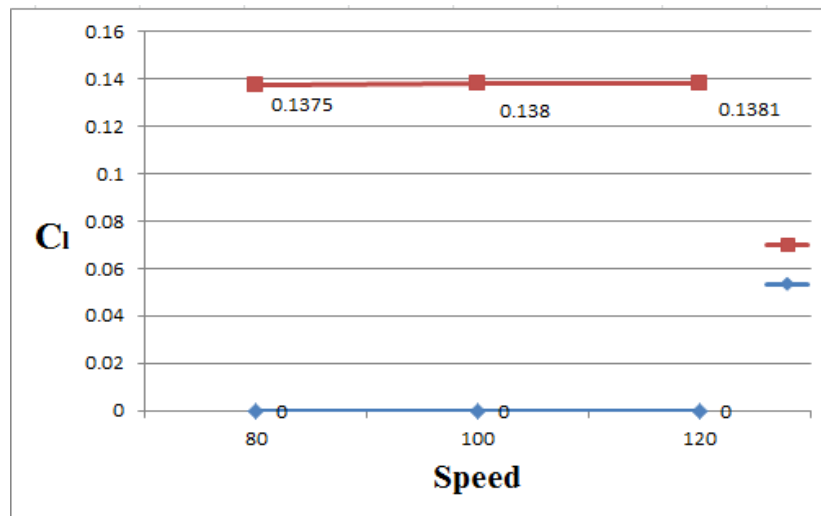


Figure 95: Lift coefficient and speed comparisons with delta-type vortex generator

The reduced lift coefficient concerning speed is illustrated above in Figure 94 while a comparison Lift coefficient of the delta-shaped vortex generator comparison with speed is presented in Figure 95.

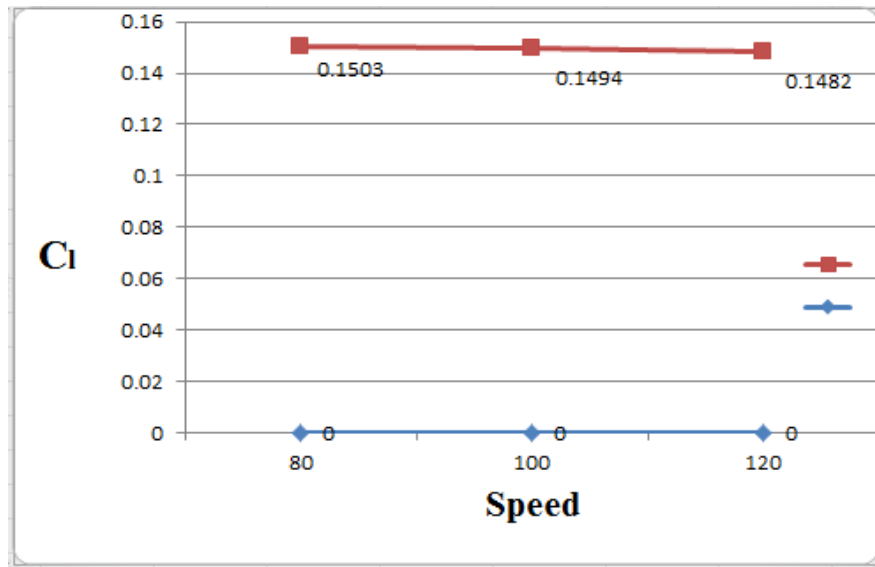


Figure 96: Lift coefficient and speed with triangular vortex generator

When adding the triangular vortex generator to the model is indicated the reduced lift coefficient of the pickup, additionally, the vertical blue line represents the speed, and the horizontal line shows the reduced lift coefficient with the triangular vortex generator. A comparison of the lift coefficient with a triangular-shaped vortex generator at a given speed is shown in Figure 96.

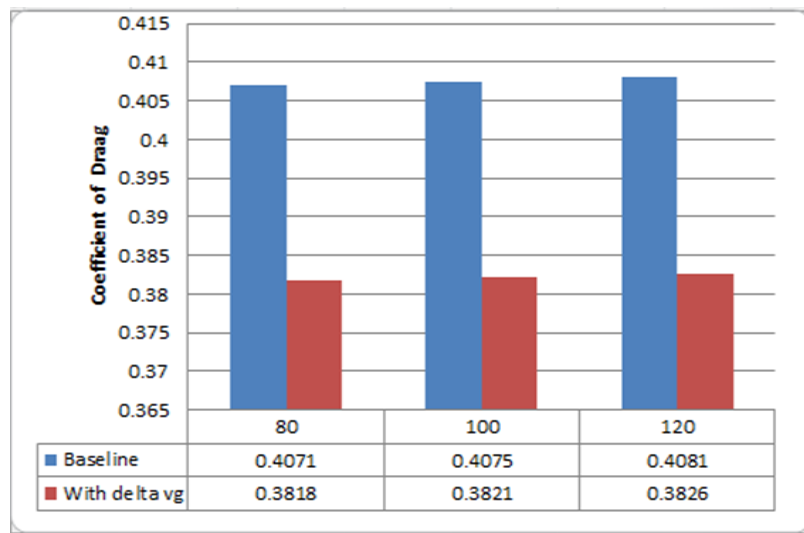


Figure 97: Drag comparisons of baseline model with delta-type vortex generator

A drag coefficient of baseline model compared with delta-shaped vortex generator implication on the model, Figure 97 shows the comparison results of baseline and with delta-shaped vortex generator model.

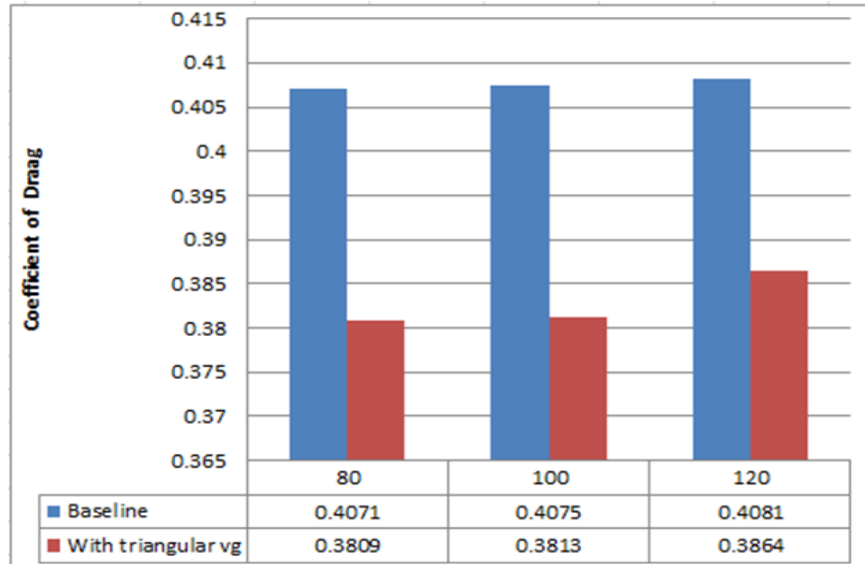


Figure 98: Drag coefficient comparisons of baseline model with triangular vortex generator

The above figure indicates that implementing a type of passive flow control device such as a triangular-shaped vortex generator on the roof end of the model shows the reduced result of drag coefficient with the average speed of the model. The bar graph represented that there is a different value of drag coefficient with and without the vortex generator on the model. A comparison of the drag coefficient obtained from the baseline model with a triangular-shaped vortex generator is shown in Figure 98.

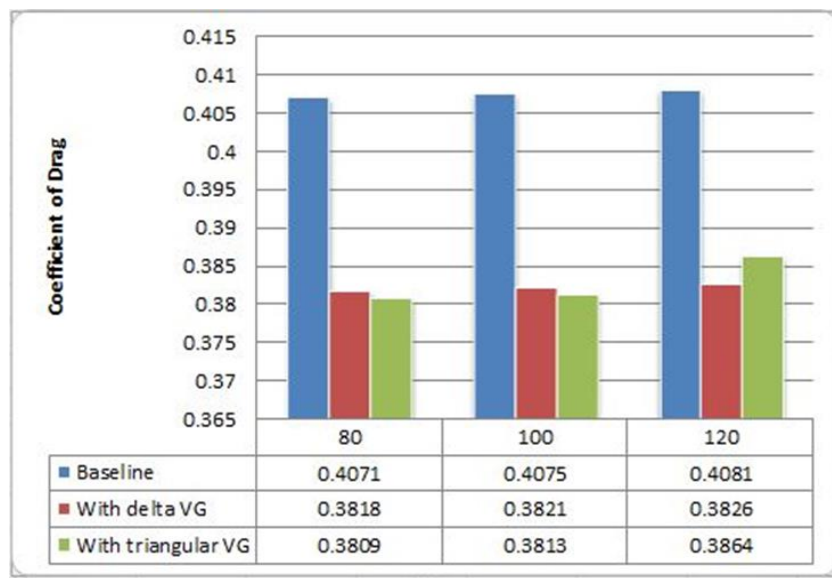


Figure 99: Drag coefficient of baseline with delta and triangular vortex generator

4.1 Drag and Lift Coefficient Reduction in Optimization Method

Even though the design of pickup trucks causes a lot of drag. Now this study investigated utilizing passive flow add-on devices, specifically triangular and delta-shaped vortex generators. Reduces drag coefficient of Bishoftu double cabin pickup vehicle. Both drag reduction devices are attached at the roof end of the pickup to reduce the wake region following pressure drag flow separation and supply high-energy air to the wake on the rear side of the vehicle. Furthermore, the drag reduction optimization of delta and triangular-shaped vortex generators was investigated using numerical analysis and experimental tests of the baseline model for comparison. Response surface method (RSM) utilized for delta and triangular-shaped vortex generators analysis of drag and lift coefficient reduction optimization.

Table 7: Input parameters with delta-shaped vortex generator

	Name	Units	Low	High	-alpha	+alpha
A [Numeric]	SPEED	km/h	80	120	66.3641	133.636
B [Numeric]	LENGTH OF \	mm	15	25	11.591	28.409
C [Numeric]	CD WITH DEL		0.3818	0.3826	0.381527	0.382873

While optimizing the impacts of both delta and triangular-shaped vortex generators to determine which passive flow method is preferable for reducing drag and lift coefficients by using the design of the expert's version. Subsequently, though, optimize the drag reduction percentage with the implication of delta vortex generator in the rood end of the Bishoftu double cabin pickup. The input parameters for identifying the best method of drag reduction set to given in Table 7.

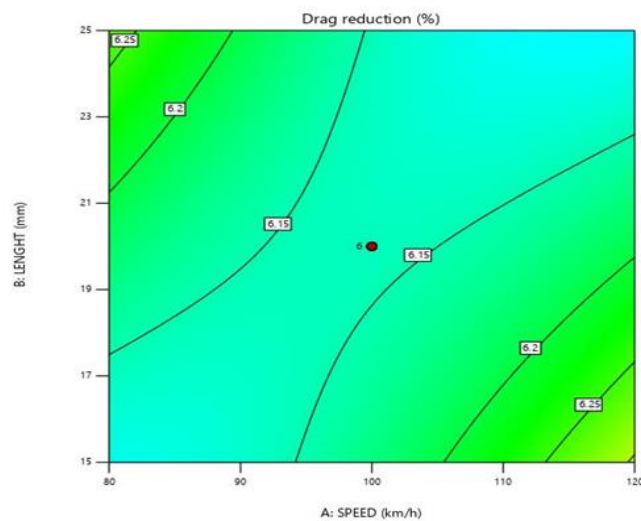


Figure 100: Contour of optimization with delta-shaped vortex generator

Within relation to the speed and range of the delta vortex generator length, the minimum and maximum drag reductions are illustrated in Figure 100, which explains the optimization result of drag in percentage with the delta vortex generator. The response is drag reduction with a delta-shaped vortex generator. Based on the delta vortex generator's contour, the drag coefficient was determined to be 0.3822 and achieved a minimum of 6.01% drag and a maximum result of 6.25% drag reduction with the implementation of the delta-shaped vortex generator at the roof end of the Bishoftu double cabin pickup vehicle.

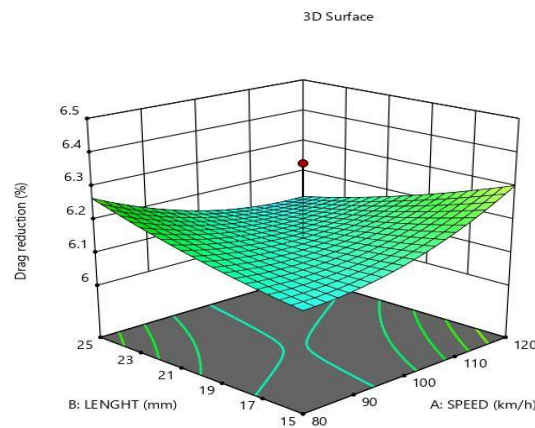


Figure 101: The 3D surface of a delta-shaped vortex generator

The delta-shaped vortex generator's 3D surface was created by Design of Experts (DOE), utilizing the 3D surface examine to analyze the percentage of drag reduction at the roof end of the pickup model. With the implication of a delta-shaped vortex generator to identify the minimum and maximum value of drag reduction, furthermore, the 3D surface described the length and speed range. The 3D surface of delta-shaped vortex generator as shown in Figure 101.

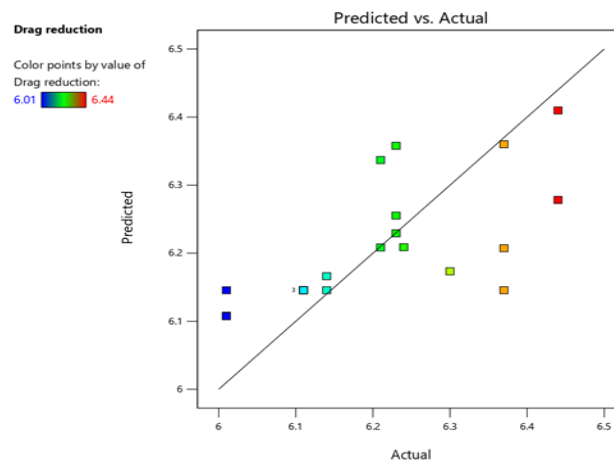


Figure 102: Predicted and Actual drag coefficient results with delta-shaped vortex generator

Figure 102 illustrates the expected and actual results of the optimization of drag reduction with a delta-shaped vortex generator. Nevertheless the actual results from the numerical simulation. The graph indicates that the expected result is almost at the actual point.

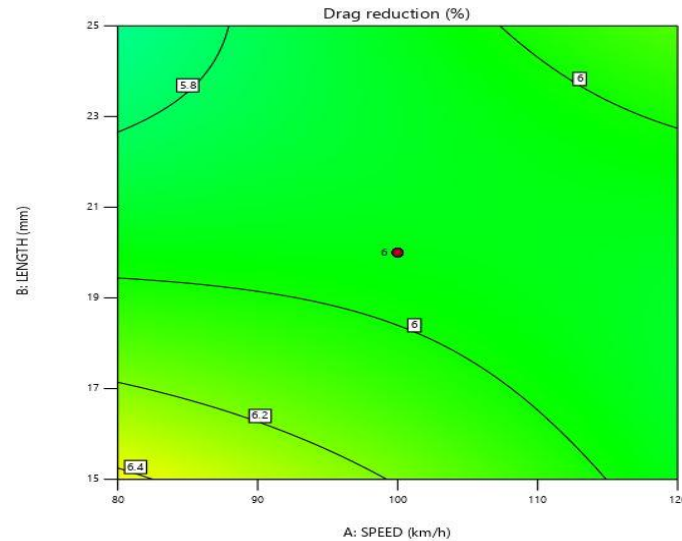


Figure 103: Contour of optimization with triangular-shaped vortex generator

The triangular-shaped vortex generator's drag reduction optimization method is shown by the contour in Figure 103. Triangular-shaped vortex generators are passive flow control add-on devices for pickup models. The maximum and minimum drag reduction percentage values are 5.04% and 6.50% respectively.

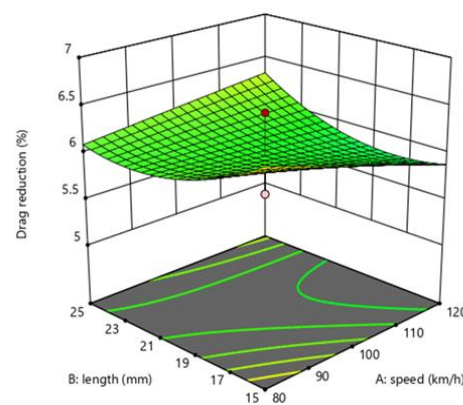


Figure 104: The 3D surface of a triangular-shaped vortex generator

The triangular vortex generator's 3D surface was designed using the Design of Experts method to analyze the percentage of drag reduction with the triangular-shaped vortex generator at the roof end of the Bishoftu pickup. However, the effect of the triangular vortex generator as shown in Figure 104, and the minimum and maximum points of drag reduction about the range of the given speed were described by the 3D surface.

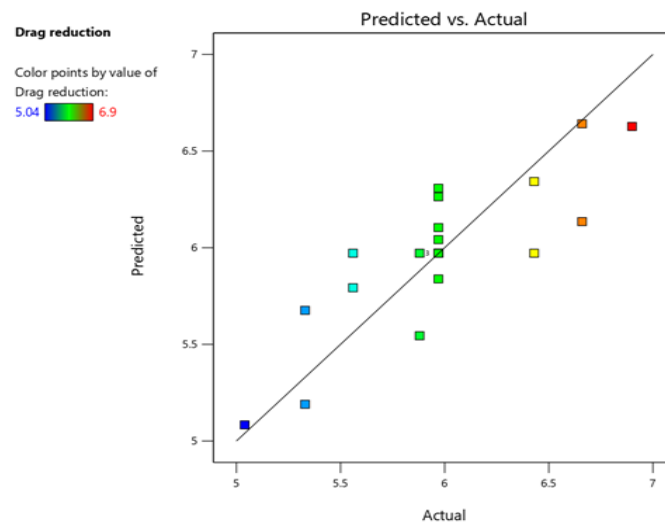


Figure 105: Predicted and Actual results with triangular-shaped vortex generator

Vortex generators as well as passive flow control drag reduction add-on devices are essentially the most significant way to reduce the vehicle's drag coefficient. In Figure 105, the actual results and the expected optimization of drag reduction with a triangular-shaped vortex generator are discussed. Nevertheless, the graph shows that the expected result is almost near to the actual point.

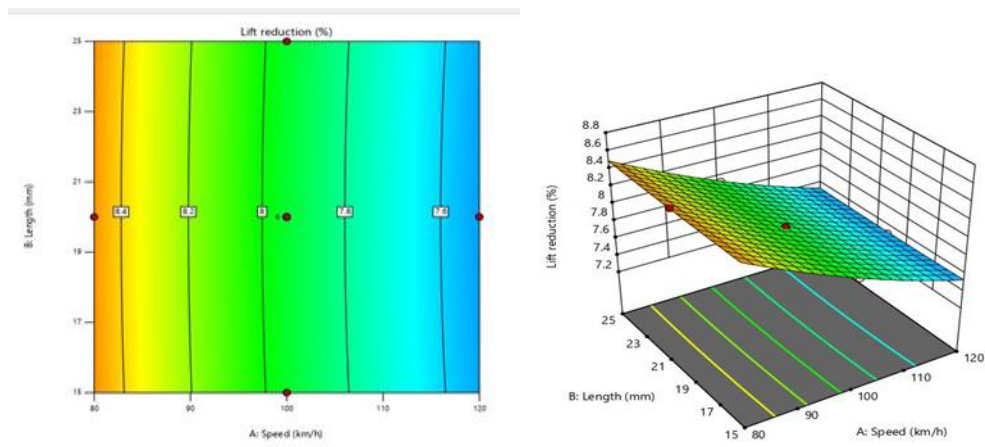


Figure 106: Contour and 3D surface of lift reduction with delta-shaped vortex generator

Utilizing passive devices, such as triangular and delta-type vortex generators to reduce the pickup truck's lift and drag coefficient. Figure 106 illustrates the contour of lift reduction with a delta-shaped vortex generator and shows the minimum and maximum level of lift reduction on a 3D surface plot. Based on the investigation conducted using Design of Experts software, the delta-shaped vortex generator is essential for reducing the lift coefficient at the minimum level of 7.3% and a maximum of 8.69 %, with the average lift result of 0.1378 delta-shaped vortex generator.

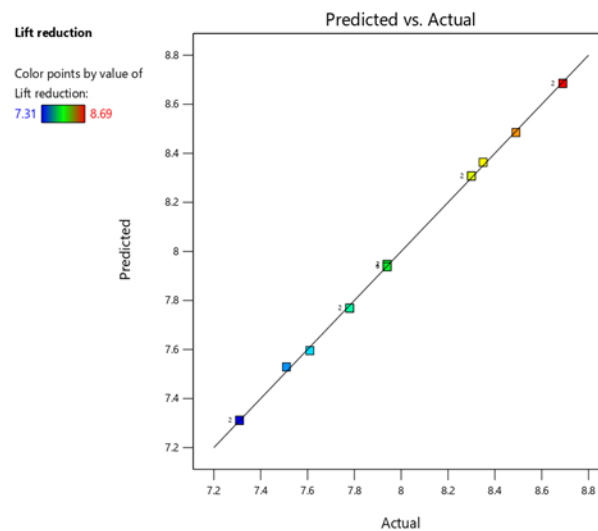


Figure 107: Predicted with actual delta-shaped vortex generator

Vortex generators are now the most significant method for improving a vehicle's lift coefficient. In Figure 107, the optimization of lift reduction with a delta-shaped vortex generator is illustrated with the average lift coefficient results.

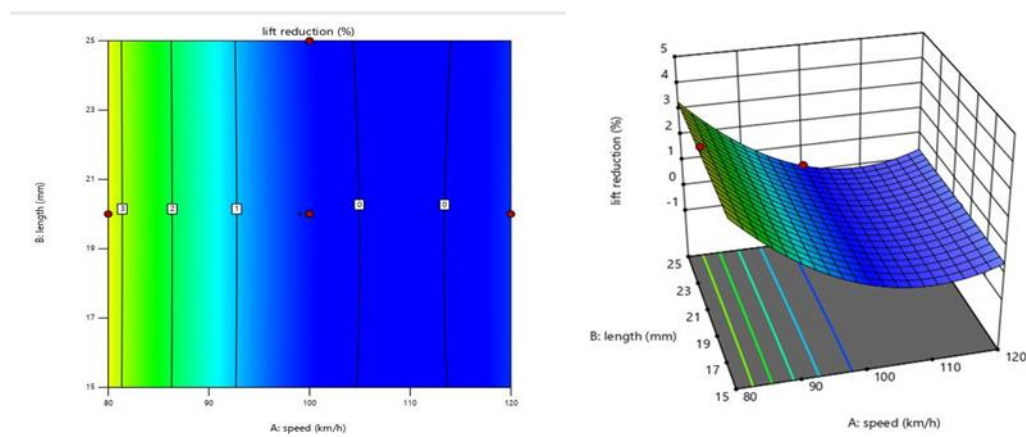


Figure 108: Contour and 3D surface of lift reduction with triangular-shaped vortex generator

Passive add-on devices such as a triangular-shaped vortex generator reduce the drag and lift coefficient of the pickup. Figure 108 shows the contour of lift reduction using a triangular vortex generator, and determines the effect of the vortex generator at the roof end of the Bishoftu double cabin pickup. However, the minimum and maximum amount of lift reduction factor is described on a 3D surface. Based on the analysis using the Design of Experts software, the average lift coefficient reduction with a triangular vortex generator is 0.1494, with a minimum result of lift coefficient reduction of 0.3% and a maximum reduction of 4.38%.

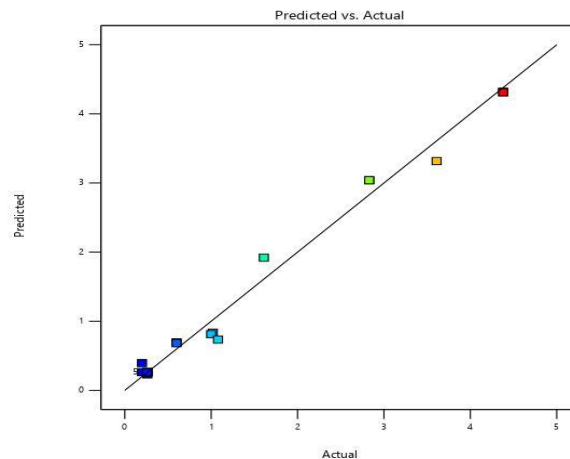


Figure 109: Predicted with actual triangular-shaped vortex generator

Whenever it involves vehicle aerodynamic study vortex generators are certainly the most significant method for reducing the lift coefficient of the vehicle. Figure 109 describes the optimization of lift reduction with a triangular vortex generator, determined by prediction. In summary, the analysis carried out on the optimization of lift coefficient minimizes and conducted the comparisons between triangular and delta-shaped vortex generators. Examining the output data and results, the triangular vortex generator achieved a 4.38% lift reduction, while the delta-shaped vortex generator reduced the maximum value of 8.69% lift coefficient reduction. Based on the data analysis above, a delta-shaped vortex is significantly more effective for reducing the lift coefficient than a triangular-shaped vortex generator. Nevertheless, the delta vortex generator achieved a drag reduction result of 6.25% while the triangular vortex generator also 6.50% drag coefficient. This analysis concluded triangular-shaped vortex generator is more effective than the delta Vg for the drag reduction method.

4.2 Experimental Setup and Results

To confirm that the proposed exterior design of the Bishoftu Plutus double cabin pickup truck is appropriate, this study describes an experimental investigation of the vehicle's aerodynamics and its subsequent validation, using computational techniques [96]. The experimental test was conducted in the Jimma University, Jimma Institute of Technology mechanical engineering thermo fluid laboratory conventional wind tunnel illustrated in Figure 110. The glass side and bottom test section walls provide optical access in an open return suction wind tunnel. The test section dimension is approximately 280mm x 25 x 300mm, and the wind tunnel's maximum speed is 28m/s [97].



Figure 110: Wind tunnel structure in thermo fluid laboratory

The most effective and conventional wind tunnel method is used to study the aerodynamics analysis and performance of the vehicle experimentally. However, tests using various methodologies are conducted in this study [98]. The 1/50th scaled pickup truck model aerodynamics study was carried out in a conventional wind tunnel. However, we could consider the conventional wind tunnel facility to be provided for experimental setup. To simulate typical flow characteristics and aerodynamic forces, wind flow was precisely controlled [98]. In the first stage, the model is placed within the simulation chamber as previously stated, upwards, with its weight supported by an aluminum stick. The aluminum stick is inserted into a base that serves as a force measurement device and, particularly important, as an anchor [99].

Through the use of a tilting manometer also attached to the pitot probe mounted under the test section's side structure, the wind speed within measures pressure and drag force [98].



Figure 111: The 1/50th scale model of Bishoftu double cabin pickup before an experimental test
The 1/50th scaled model of the Bishoftu double cabin pickup printed model is shown in Figure 111.



Figure 112: Adjusting model holder before installing

Before installing the scaled-down model into the conventional type wind tunnel adjust the model holder to the maximum dimension of 4mm and it is important to tighten with the specific nut and bolt dimension as demonstrated in Figure 112. However, after connecting the holder to the model then installed on the wind tunnel for experimental test.



Figure 113: Installing the model into the wind tunnel machine

A setup procedure of a 3D-printed model of pickup in the wind tunnel is shown in Figure 113.



Figure 114: Structure of installed model in the wind tunnel

The wind tunnel should be adjusted and calibrated before any experimental process test to ensure correct operation and result. This study provided the 1/50th scaled model of a pickup vehicle to be printed in 3D mode with PLA (Polylactic Acid) plastic used to print the model, with 15% infill. This will increase the scaled model's instability [99]. The structure of the 3D model of the pickup model is shown in Figure 114, the baseline scaled model of the Bishotftu double cabin pickup presented inside wind the tunnel.



Figure 115: Digital manometer for different measurements



Figure 116: Digital amplifier for drag force (N) and lift force (N) measurements

The experimental procedure used in this work aimed to analyze the flow around the model in the wind tunnel using a hot wire digital manometer as shown in Figure, 115, and an amplifier to measure velocity profiles in the wake flow of the body demonstrated in Figure, 116. However, to achieve such a goal experiments were carried out by using a manometer system to characterize flow profiles in the scaled model wake [100].

The air velocity inside the wind tunnel was changed by adjusting the variable-speed drive frequency. The frontal area of each vehicle (car= 0.01070854) was used to calculate the drag coefficient in the experimental approach [101]. The experimental test procedures are illustrated in Figure 117 and Figure 118, respectively.



Figure 117: Experimental set-up in the wind tunnel



Figure 118: Scaled model during experimental test inside the wind tunnel test

To check the correctness and reliability of the results, three measurements were made for each configuration of the model. [102].

Considering several pressure and speed procedures, especially in 21.5 m/s, 22.4 and 22.5 time-averaged pressure distribution and force were measured. Nevertheless, the only results related to the subsequent speed have been indicated in this study for the sake of simplicity and comparisons with the results obtained from numerical simulations. Currently, no visualization methods such as smoke flow, or surface oil flow measurements are in existence. These investigations need to be completed in the future project.[103]. However, from the above results of the experimental test speed averaged results are selected for this study.

Table 8: Averaged results of the experimental test in wind tunnel

Drag force	Milli bar	Speed (m/s)	CD value
0.08	2.70912	21.5	0.3979
0.09	2.70965	22.4	0.4860
0.08	2.70970	22.5	0.4358

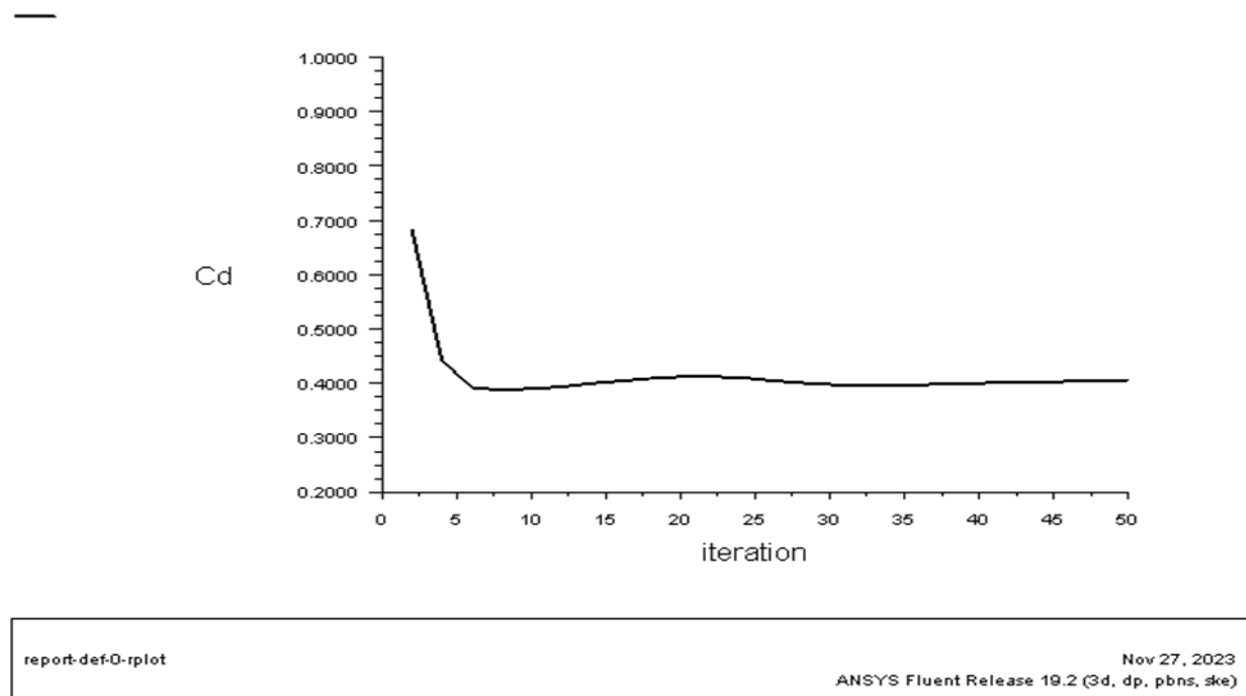


Figure 119: Coefficient of drag of baseline 1/50th scaled model

The drag coefficient graph of the scaled baseline model is shown in Figure 119.

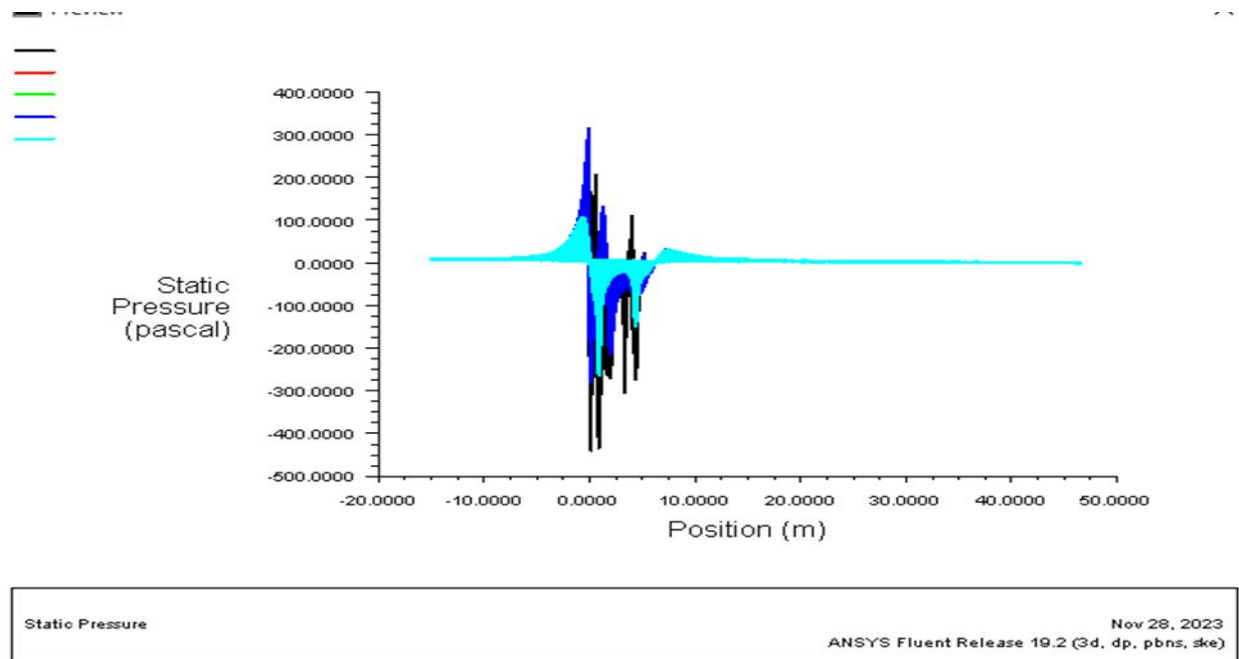


Figure 120: Distribution of static pressure

A distribution graph of static pressure in the baseline model is presented in Figure 120.

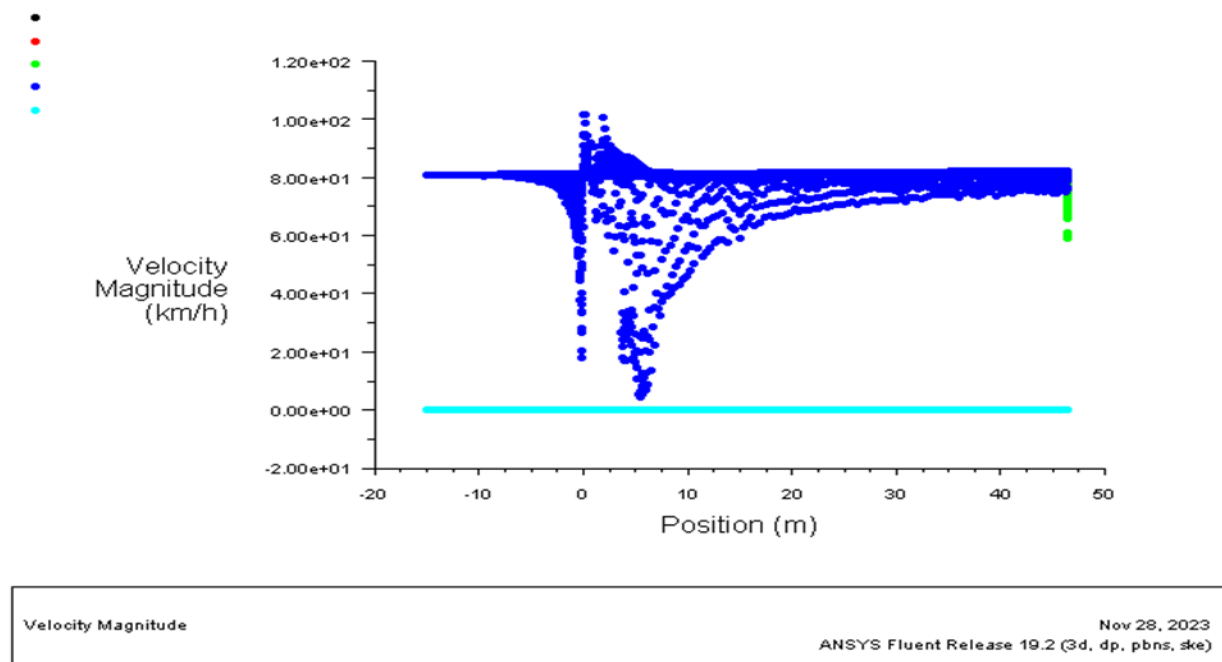


Figure 121: Distribution of velocity magnitude (km/h)

A distribution graph of velocity magnitude through the baseline model is shown in Figure 121.

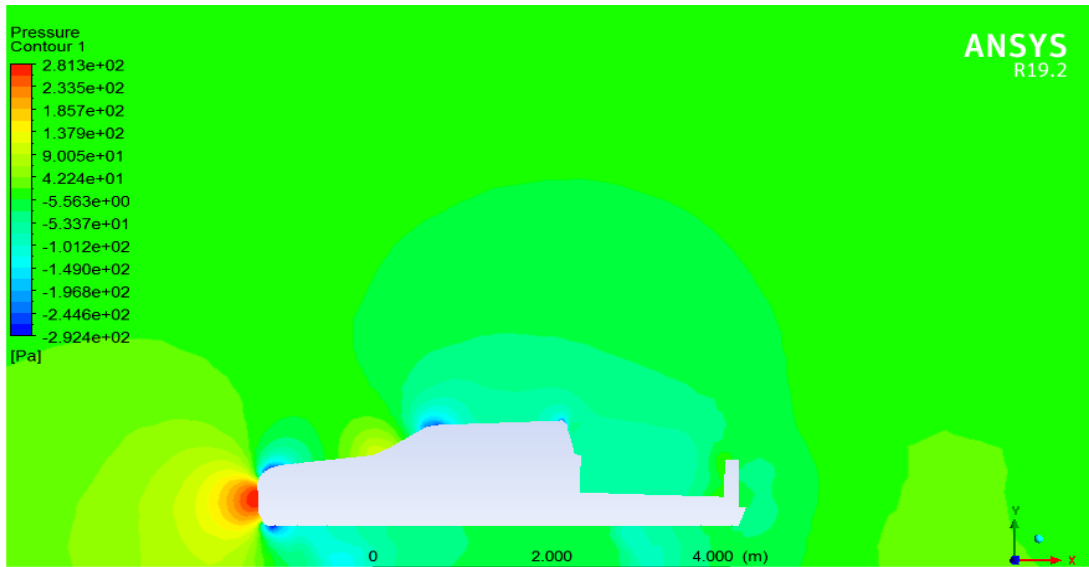


Figure 122: Pressure contour of the scaled model in 77.4 km/h

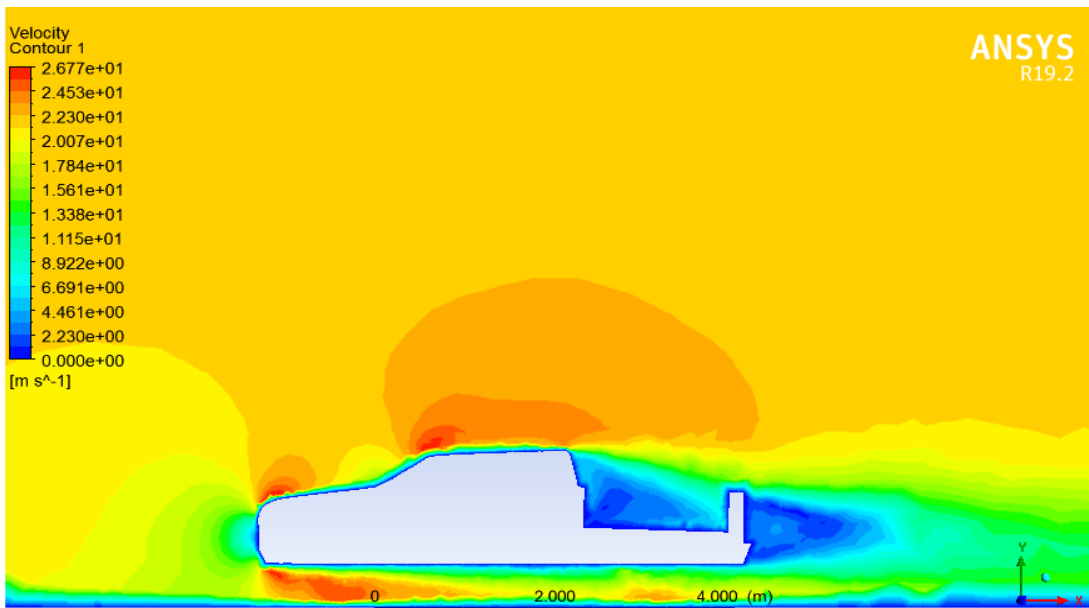


Figure 123: Velocity contour of 1/50th scaled model in 77.4 km/h

The pressure and velocity contour of the scaled model through the baseline model in the symmetry plane is shown in Figure 122, and Figure 123 respectively.

4.3 None-Dimensional Analysis

A drag coefficient of experimental results with the non-dimensional point of view describes the relationship between drag and the atmosphere's fluid viscosity. It is crucial to examine the relationship between the drag coefficient and Reynolds number considering the flow rate influences the Reynolds number. The wind tunnel's experimental data has been validated using different parameters. This study utilizes both experimental and numerical methods to examine the impact of the Reynolds number on a scaled model of a Bishioftu double-cabin pickup vehicle in different velocities related to aerodynamic performance. The adjustments to flow and aerodynamic characteristics.

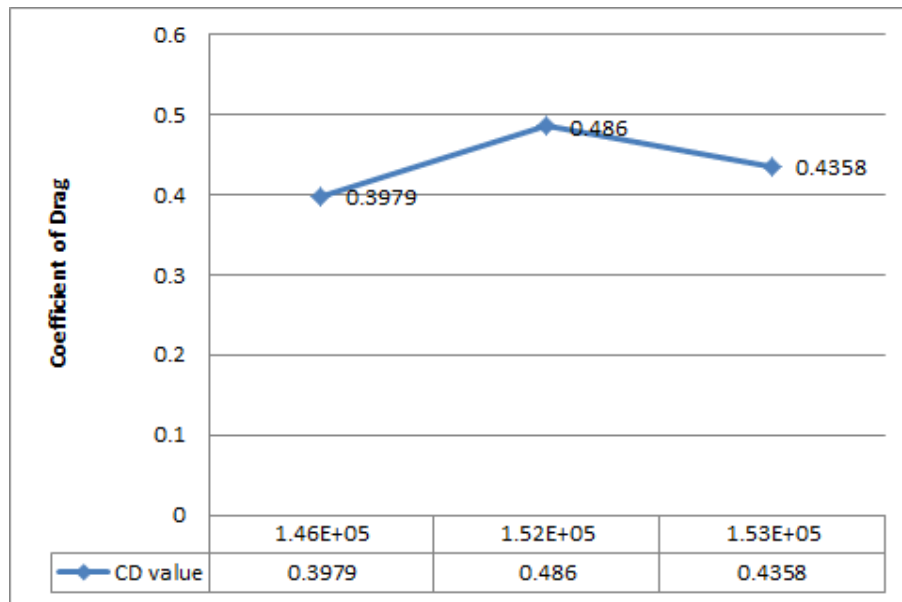


Figure 124: The relationship between drag coefficient and Reynolds number

Validation of the experimental data from the wind tunnel with drag coefficient results of 0.3979, 0.4860, and 0.4358, the investigation was carried out for dimensional analysis of the Bishoftu pickup model between three different Reynolds numbers of 1.46×10^{-5} , 1.52×10^{-5} and 1.53×10^{-5} at the given average speed of 21.5, 22.4, and 22.5 m/s. This study conducted the impact of the Reynolds number on the aerodynamic characteristics of the Bishoftu double cabin pickup vehicle. The above graph illustrates plots of data revealing that with an increase of Reynolds number as well as drag coefficient result decreased. The relationship between the drag coefficient and the Reynolds number is shown in Figure 124.

Three averaged speeds, 21.5, 22.4, and 22.5 were investigated for their aerodynamic characteristics using a wind tunnel for experimental and ANSYS software for numerical simulation. The average difference between the numerical and experimental tests is 6% and less than 10% is acceptable [104]. This study aims to examine the aerodynamic characteristics' adaptability to form and confirm the numerical simulation with additional experimental data. Within the experiments carried out in the educational wind tunnel. The aerodynamic drag was computed using the realizable k- ϵ model [105]. According to the analysis of experimental and numerical results both investigations are validated as shown in Figure, 125.

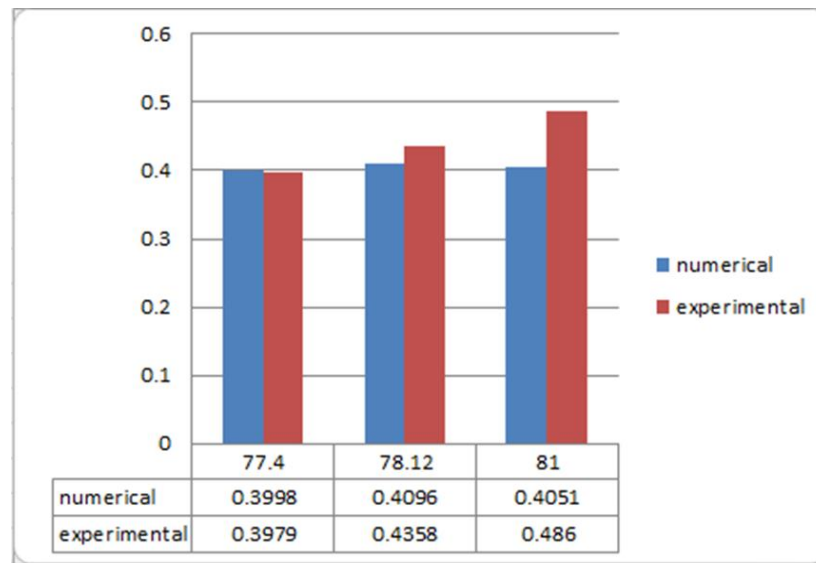


Figure 125: Numerical and experimental coefficient of drag comparisons

The main objective of this thesis work is to analyze the aerodynamic analysis of Bishoftu pickup vehicle with numerical and experimental method. Figure 125 shows the results of both numerical and experimental analysis.

4.2 Calculations of fuel saved for the modified model

The engine's power concerning the engine's fuel mass flow supply determined the engine's specific fuel consumption. Brake-specific fuel consumption (bsfc) is evaluated after all intermittent engine losses, and indicated specific fuel consumption (isfc) is measured according to the in-cylinder pressure (the pressure's ability to work) [106]. The equation number explains the power savings of the modified pickup truck configuration model over the baseline model and the fuel saved from the reduction factor in drag force and coefficient in a numerical manner [95].

$$P_{saved} = (F_D(baseline) - F_D(model\ x))V_{truck} \dots \dots \dots (4.1)$$

$$m_{fuel} = \frac{P_{save}}{QLHV \times P_{fuel} \times \eta_{engine}} \dots \dots \dots (4.2)$$

Table 9: Result summary of fuel saved for modified Bishoftu pickup models [95]

Model	Speed (km/h)	Drag force(N)	Fd reduction (N)	Power saved(W)	Fuel saved (lit/hr)
Baseline	80	224.50	-	-	-
	100	350.10	-	-	-
	120	503.78	-	-	-
A model with Delta VG	80	215.46	184.5654	1279.250	0.5457
	100	336.15	54.8285	1360.3864	0.1525
	120	483.50	170.8619	1362.2166	0.4746
Model with Triangular VG	80	221.49	174.3117	1361.3621	0.4844
	100	345.61	57.6767	1357.5382	0.1604
	120	497.29	170.8976	1365.7519	0.4735

The result summary of fuel saved for modified Bishoftu pickup models without and with vortex generators drag reduction in all required speeds are set to given in Table 10.

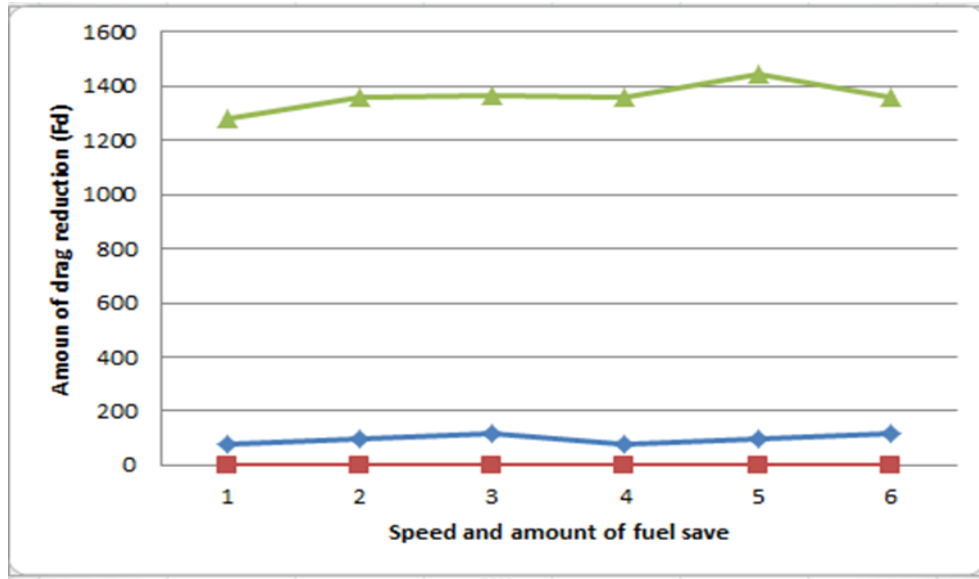


Figure 126: Amount of drag force (N) fuel saved

The amount of drag reduction (F_d) comparison with speed according to fuel saved is shown in Figure 126.

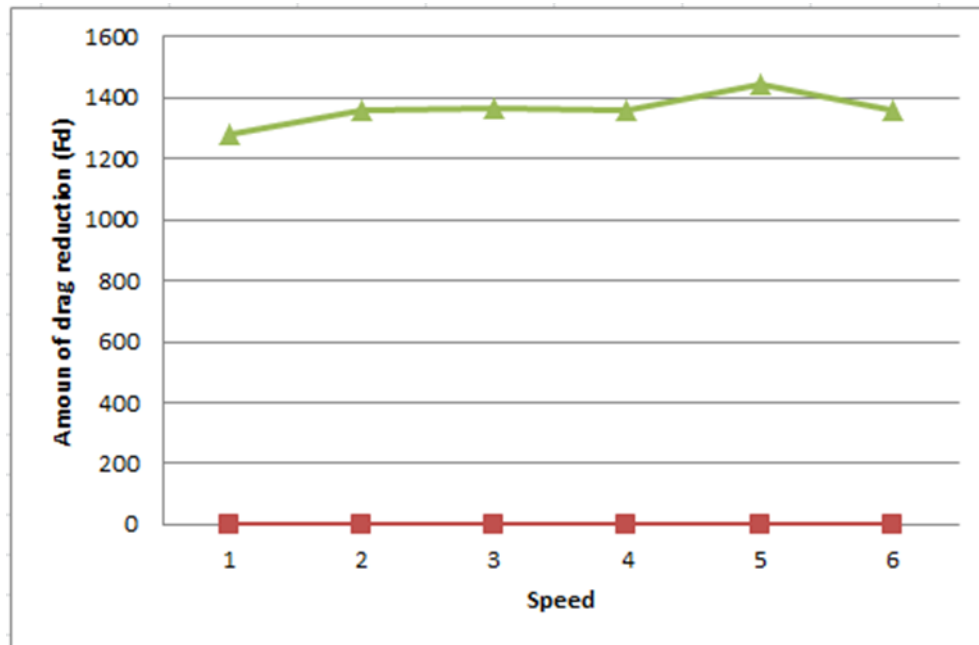


Figure 127: Amount of drag force reduction (F_d) and speed

The amount of drag reduction (F_d) comparison with speed is shown in Figure 127.

CHAPTER FIVE

CONCLUSIONS AND RECOMMENDATIONS

Nowadays, pickups have been attributed to several issues including negative environmental quality, global warming, and fuel economy. Vehicle aerodynamics now faces a significant problem in trying to improve the initial impression that Ethiopians and other worldwide customers have concerning this issue. It is necessary to improve the efficiency and aerodynamic performance of automobiles. Good aerodynamic results and efficient operating engine conditions are capable of approving the vehicle's performance. Using computational fluid dynamics (CFD) techniques, this study essentially validates the Bishoftu double cabin pickup model's computational and experimental results. The vortex generators were attached at the roof end of the original pickup CAD model and the findings were examined after the computational fluid dynamics (CFD) setup and mesh generation were validated.

There are three different double cabin pickup models are available, the baseline model without drag-reducing add-on devices, the second model with delta-shaped vortex generators, and the third model with triangular-shaped vortex generators. The next stage was to conduct an experimental test of the baseline model to find what appeared suitable for the model after it was compared to the numerical simulations. However, the addition of vortex generators resulted in an improvement in pressure around the vortex generator's location examining the overhaul pressure contours in the pickup roof end area. According to Bernoulli's principle, the consequent changes in pressure cause the air velocity on the pickup's body to increase simultaneously, which is determined essential works. To shift the separation flow of point further back to delay the airflow circulation from reality. The coefficient of drag is reduced by shifting the boundary layer's separation point in the rear of the pickup's roof end surface. It was examined that the triangular-shaped vortex generator performed much more effectively than the delta-shaped vortex generator with the same height of from $H = 15\text{mm}$, 20mm , and 25mm providing a coefficient of drag $C_D = 0.3810$ and illustrated an increased amount of drag reduction in 6.50%, also 8% reduction of lift coefficient and 14% of maximum fuel saved. However, the existing research formed 1.12% and 0.5 percentage of drag reduction with a vortex generator.

5.1 Recommendations

The configuration of the pickup geometry about the transient rotation moment can be analyzed using computational fluid dynamics with time-dependent flow boundary conditions. Unfortunately, however, this now takes significant amounts of computing work. It will be feasible to maximize the aerodynamic performance of the vehicle system when computing capability enables the simulation of unsteady numerical simulation acting on detailed vehicle models, an Add-on device system that can be modified for the potential improvement of aerodynamic performance through the use of passive drag reduction methods.

- ❖ As this thesis work illustrates, reducing the pickup truck's drag coefficient results in significant fuel savings. As a result, a growing number of Ethiopian companies and businesses currently must confirm the vehicles' body shapes' aerodynamic performance and concentrate a greater emphasis on aerodynamics.
- ❖ It is suggested to use a computer that's powerful with a high processing speed and capacity to obtain more accurate results, as well as this will enhance meshing quality and reduce processing times

5.2 Future work

- ❖ The model of turbulence implemented in this thesis work can be used to analyze the aerodynamic properties of various passive add-on device designs operating the numerical simulations, including baseline models without and without vortex generators.
- ❖ To verify the validity of the values obtained from numerical and experimental analysis tests in this thesis work only use a baseline model conducted with a wind tunnel tester to investigate the results. However, it will be necessary to conduct experimental tests with delta-shaped and triangular-shaped vortex generators on the pickup models.
- ❖ Implementing add-on devices and other modifications on the pickup vehicle that reduce its aerodynamic drag. To reduce the drag coefficient with passive drag reduction add-on devices on the vehicle, this will be developing and examining the performance of add-on devices such as vortex generators.

REFERENCES

- [1] A. A. S. Al-saadi, “Analysis of Novel Techniques of Drag Reduction and Stability Increase for Sport Utility Vehicles using Computational Fluid Dynamics,” 2019.
- [2] R. Bahoosh, “A Numerical Simulation for Improving the aerodynamics of a pickup truck by use of diffusers , spoilers , and bumps,” 2022.
- [3] W. Mokhtar, M. Hossain, and S. I. Jahan, “A Parametric CFD Study of a Generic Pickup Truck and Rear Box Modifications,” 2018.
- [4] P. P. Truck, S. A. E. T. Paper, K. Chen, and B. Khalighi, “Chen , K . and Khalighi , B . , " A CFD Study of Drag Reduction Devices for a Full Size A CFD Study of Drag Reduction Devices for a Full Size Production Pickup Truck,” no. April, 2015, doi: 10.4271/2015-01-1541.
- [5] X. Yang, Y. Hu, Z. Gong, J. Jian, and Z. Liu, “Numerical Study of Combined Drag Reduction Bases on Vortex Generators and Riblets for the Ahmed Body using IDDES Methodology,” vol. 15, no. 1, pp. 193–207, 2022.
- [6] D. H. S, “Vortex Generator for Drag Reduction in Hatchback Vehicles,” no. 6, pp. 86–92, 2016.
- [7] J. Yon, “Improving aerodynamic characteristics for drag reduction of heavy truck Improving aerodynamic characteristics for drag reduction of heavy truck,” 2020, doi: 10.1088/1757-899X/862/3/032032.
- [8] V. Atatug and S. Bayraktar, “Numerical investigation of aerodynamic characteristics of generic pickup trucks Pamukkale Üniversitesi Mühendislik Bilimleri Dergisi Pamukkale University Journal of Engineering Sciences Numerical investigation of aerodynamic characteristics of generic pic,” no. February, 2020, doi: 10.5505/pajes.2019.43778.
- [9] T. Charles and Z. Yang, “Comparison of the Effectiveness of Drag Reduction Devices on a Simplified Truck Model through Numerical Simulation,” pp. 300–313, 2022.
- [10] S. Woodiga *et al.*, “The GTU - A New Realistic Generic Pickup Truck and SUV Model The GTU – A New Realistic Generic Pickup Truck and SUV Model,” no. April, 2020.

- [11] D. S. Nath, P. C. Pujari, A. Jain, and V. Rastogi, "Drag reduction by application of aerodynamic devices in a race car," 2021.
- [12] A. Gashu and R. B. Nallamotheu, "Analysis of Crashworthiness of Bishoftu Pickup Vehicle Structure during Side Pole Crash," *J. Eng. (United Kingdom)*, vol. 2022, 2022, doi: 10.1155/2022/8386877.
- [13] I. Journal, A. Technology, and C. Bay, "Numerical Drag Reduction of a Ground Vehicle by NACA2415 Airfoil Structured Vortex Generator and Spoiler Numerical drag reduction of a ground vehicle by naca2415 airfoil structured vortex generator ," no. October, 2019, doi: 10.1007/s12239.
- [14] N. Zealand, I. N. America, R. Trucks, and B. Company, "Pickup truck," pp. 1–9, 2018.
- [15] B. J. Linden, "History of the Pickup Truck," pp. 21–24, 1925.
- [16] T. Tundra and S. Africa, "Pickup truck history," no. 888, pp. 1–28.
- [17] P. M. Evolution *et al.*, "1st Generation (1968—1972)," pp. 1–10, 2018.
- [18] F. P. Truck, W. S. My, C. A. R. Worth, and B. S. Guide, "Ford ' s F-Series Pickup Truck History , from the Model TT to Today," pp. 1–23.
- [19] S. Special, "History of the Pickup Truck," pp. 23–26, [Online]. Available: <https://www.carcovers.com/resources/history-of-the-pickup-truck.html>
- [20] B. Y. S. Singh, "10 Powerful ' 90s Pickups That Will Last You A Lifetime," pp. 1–21, 2022.
- [21] S. Sarkar, K. Thummar, N. Shah, and V. Vagrecha, "A Review paper on Aerodynamic Drag Reduction and CFD Analysis of Vehicles," pp. 231–235, 2019.
- [22] B. Khalighi and G. M. Company, "Aerodynamics of a Pickup Truck : Combined CFD and Experimental Study Aerodynamics of a pickup truck : combined cfd and experimental study bahram khalighi Vehicle Development Research Lab JAMES H . LEYLEK Clemson University Computational Center for Mobilit," no. October 2009, 2019, doi: 10.4271/2009-01-1167.
- [23] H. M. Abid, E. N. Roslin, R. I. Abdul, and A. S. Sutar, "Optimization of Power and Fuel Consumption for Rear Spoiler of a Pickup Truck," vol. 11, no. 1, pp. 440–452, 2021.

- [24] A. C. Market, “Tech : Pickup Truck Aerodynamics How To Design an Efficient Hard Top,” pp. 6–10, 2021.
- [25] R. Semaan, P. Oswald, G. Y. Cornejo, and M. Bernd, “Aerodynamic optimization of a generic light truck under unsteady conditions using gradient - enriched machine learning control,” *Exp. Fluids*, vol. 64, no. 3, pp. 1–27, 2023, doi: 10.1007/s00348-023-03587-x.
- [26] S. Jindal, B. Khalighi, and G. Iaccarino, “Numerical investigation of road vehicle aerodynamics using the immersed boundary RANS approach,” *SAE Tech. Pap.*, no. April 2005, 2005, doi: 10.4271/2005-01-0546.
- [27] D. Igali, O. Mukhmetov, M. Y. Zhao, and S. C. Fok, “Comparative Analysis of Turbulence Models for Automotive Aerodynamic Simulation and Design A comparative analysis of turbulence models for,” no. August, 2019, doi: 10.1007/s12239-019-0107-7.
- [28] B. S. Lokhande, “Transient Simulation of the Flow Field Around a Generic Pickup Truck Transient Simulation of the Flow around a Generic Pickup Truck,” no. February 2019, 2003, doi: 10.4271/2003-01-1313.
- [29] M. H. Dasar and D. J. Patil, “CFD Simulation of Pickup Van,” vol. 2, no. 12, pp. 1995–2004, 2013.
- [30] O. Tenango-pirin, E. Atayde, E. Reynoso-jardón, and Y. S. Hernández, “Aerodynamic Study Of The Rear Spoiler Profile Of A Pick-Up Vehicle,” vol. 18, no. 2, pp. 110–114, 2023.
- [31] A. Agung, A. Suryawan, and A. Ghurri, “Natural Sciences Engineering & Pressure Distribution Characteristics : An Experimental Study,” vol. 2, no. 2, pp. 87–93, 2022.
- [32] M. C. Dita, “Natural Sciences Engineering &,” *Nat. Sci. Eng. Technol. J.*, vol. 2, no. 1, pp. 79–84, 2022.
- [33] E. Guilmineau, “Numerical Simulation of Flow Around a Generic Pickup With ISIS-CFD,” 2010.
- [34] P. N. Selvaraju and K. M. Parammasivam, “Empirical and Numerical Analysis of Aerodynamic Drag on a Typical SUV Car Model at Different Locations of Vortex Generator,” vol. 12, no. 5, pp. 1487–1496, 2019, doi: 10.29252/jafm.12.05.29674.

- [35] M. Firdaus, M. Azmi, M. Al, B. Marzuki, M. Arzo, and A. Bakar, "Vehicle aerodynamics analysis of a multi purpose vehicle using CFD vehicle aerodynamics analysis of a multi purpose vehicle using cfd," no. April 2017, 2018.
- [36] P. N. Selvaraju, K. M. Parammasivam, S. Shankar, and G. Devaradjane, "Analysis of drag and lift performance in sedan car model using CFD," *J. Chem. Pharm. Sci.*, vol. 7, no. 7, pp. 429–435, 2015.
- [37] W. J. G. S. Pinto, "Numerical and experimental analysis of the flow over a commercial vehicle - pickup," vol. 17, no. 2, pp. 92–102, 2018.
- [38] T. Hanlon and D. Risk, "Using computational fluid dynamics and field experiments to improve vehicle-based wind measurements for environmental monitoring," *Atmos. Meas. Tech.*, vol. 13, no. 1, pp. 191–203, 2020, doi: 10.5194/amt-13-191-2020.
- [39] A. M. Al-Garni and L. P. Bernal, "Experimental study of a pickup truck near wake," *J. Wind Eng. Ind. Aerodyn.*, vol. 98, no. 2, pp. 100–112, 2010, doi: 10.1016/j.jweia.2009.10.001.
- [40] J. Ha, S. Jeong, and S. Obayashi, "Drag reduction of a pickup truck by a rear downward flap," vol. 12, no. 3, p. 12239, 2011, doi: 10.1007/s12239.
- [41] T. Charles, Z. Yang, and Y. Lu, "Assessment of Drag Reduction Devices Mounted on a Simplified Tractor-Trailer Truck Model," *J. Appl. Comput. Mech.*, vol. 7, no. 1, pp. 45–53, 2021, doi: 10.22055/jacm.2020.34811.2475.
- [42] "Numerical and experimental analysis of the flow over a commercial vehicle - pickup," 2016.
- [43] R. B. Sharma and R. Bansal, "Aerodynamic Drag Reduction of a Passenger Car Using Spoiler with VGs," no. March, pp. 256–263, 2014.
- [44] ~ Wwjmrdr, "Effect of Aerodynamic Drag Force Parameters on Electric Vehicle Motion Performance," *Int. J. Peer Rev. J. Ref. J. Index. J. UGC Approv. J. Impact Factor*, vol. 4, no. 1, pp. 197–201, 2018, [Online]. Available: www.wwjmrdr.com

- [45] S. Maxemow, "That's a Drag: The Effects of Drag Forces," *Undergrad. J. Math. Model. One + Two*, vol. 2, no. 1, pp. 1–16, 2013, doi: 10.5038/2326-3652.2.1.4.
- [46] S. Kant, D. D. Srivastava, R. Shanker, R. K. Singh, and A. Sachan, "A Review on CFD Analysis of Drag Reduction of a Generic Sedan and Hatchback," pp. 973–982, 2017.
- [47] S. Holloway, J. H. Leylek, W. D. York, and B. Khalighi, "Aerodynamics of a pickup truck: Combined cfd and experimental study," *SAE Int. J. Commer. Veh.*, vol. 2, no. 1, pp. 88–100, 2009, doi: 10.4271/2009-01-1167.
- [48] A. Modification and I. Truck, "Aerodynamical modification to reduce drag," no. Phase II, 2020.
- [49] B. N. Zakher, M. El-hadary, and A. N. Aziz, "The Effect of Vortex Generators on Aerodynamics for Sedan Cars," vol. 6, no. 6, pp. 1–17, 2019.
- [50] P. Mukda, "Effect from Accessories on Pickup Aerodynamics by Computational Fluid Dynamics," pp. 59–63, 2016.
- [51] M. H. Dasar, D. J. Patil, and M. Raviraj, "CFD Simulation of Pickup Van," vol. 2, no. 12, pp. 1995–2004, 2013.
- [52] A. Domal and P. Ande, "Drag Reduction : Design & Analysis of Vortex Generator," vol. 8, no. 1, pp. 1634–1641, 2019.
- [53] F. F. Handbook, N. Ashgriz, and J. Mostaghimi, "An Introduction to Computational Fluid Dynamics Chapter 20 in Fluid Flow Handbook By Nasser Ashgriz & Javad Mostaghimi Department of Mechanical & Industrial Eng . University of Toronto Toronto , Ontario".
- [54] J. Howell, S. Windsor, and M. Passmore, "Some Observations on Shape Factors Influencing Aerodynamic Lift on Passenger Cars," 2021.
- [55] D. S. Nath, P. C. Pujari, A. Jain, and V. Rastogi, "Drag reduction by application of aerodynamic devices in a race car," *Adv. Aerodyn.*, vol. 3, no. 1, pp. 1–28, 2021, doi: 10.1186/s42774-020-00054-7.
- [56] G. Sivaraj, K. M. Parammasivam, and G. Suganya, "Reduction of aerodynamic drag force for reducing fuel consumption in road vehicle using basebleed," *J. Appl. Fluid Mech.*, vol. 11, no. 6, pp. 1489–1495, 2018, doi: 10.29252/jafm.11.06.29115.

- [57] M. J. and A. Bradley, “Reducing Aerodynamic Drag & Rolling Resistance from Heavy---Duty Trucks: Summary of Available Technologies & Applicability to Chinese Trucks,” *Int. Counc. Clean Transp.*, 2012, [Online]. Available: www.theicct.org
- [58] M. Mashud, “Drag Reduction of a Car by Using Vortex Generator,” vol. 4, no. 7, pp. 1298–1302, 2013.
- [59] F. Szodrai, “applied sciences Quantitative Analysis of Drag Reduction Methods for Blunt Shaped Automobiles,” 2020.
- [60] G. Poornanandan and T. Senthilkumar, “Aerodynamic drag reduction in a passenger vehicle using vortex generator with varying yaw angles Aerodynamic drag reduction in a passenger vehicle using,” no. September, 2012.
- [61] K. Satheeshkumar, P. T. S. Kumar, P. R. N. Kumar, V. Sabarish, P. Samuvel, and G. Sherwin, “Analysing the drag of various vortex generators by using cfx,” 2021.
- [62] H. Tebbiche, “Aerodynamic drag reduction by turbulent flow control with vortex generators,” no. April 2014, 2016.
- [63] S. Singh and A. Guleria, “Aerodynamic Drag Reduction of a Notchback Car Geometry by Delaying Flow Separation using Vortex Generators,” vol. 4, no. 08, pp. 521–524, 2015.
- [64] M. Bellman, “Washington University Open Scholarship,” no. January, 2009.
- [65] R. Bansal and R. B. Sharma, “Drag Reduction of Passenger Car Using Add-On Devices,” vol. 2014, 2014.
- [66] M. Kumar, A. Dubey, and A. A. Jadhav, “Effect of Vortex generators on Aerodynamics of a Car : CFD Analysis,” no. April, 2013.
- [67] H. Ali, M. Mashud, and A. Al Bari, “Aerodynamic Drag Reduction of a Car by Vortex Generation 2 Mechanisms of Flow Separation and Objectives of Adding,” vol. 2, no. 1, pp. 12–21.
- [68] P. S. Rao and B. B. Kumar, “Design and analysis of vortex generators for reducing drag force in automobiles by,” vol. 4, no. 5, pp. 548–555, 2019.

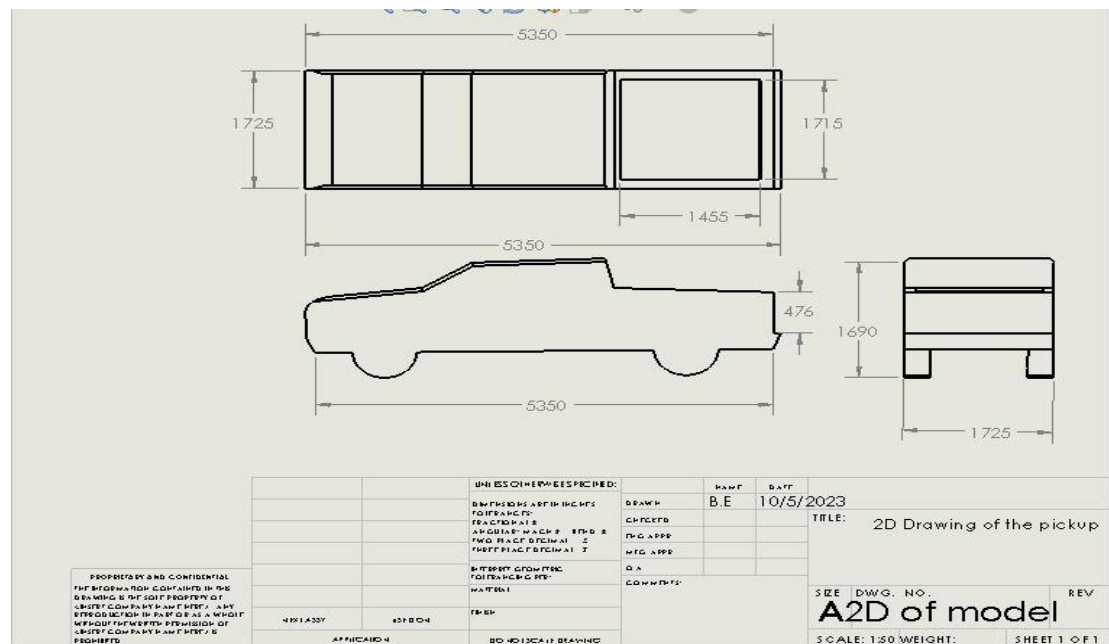
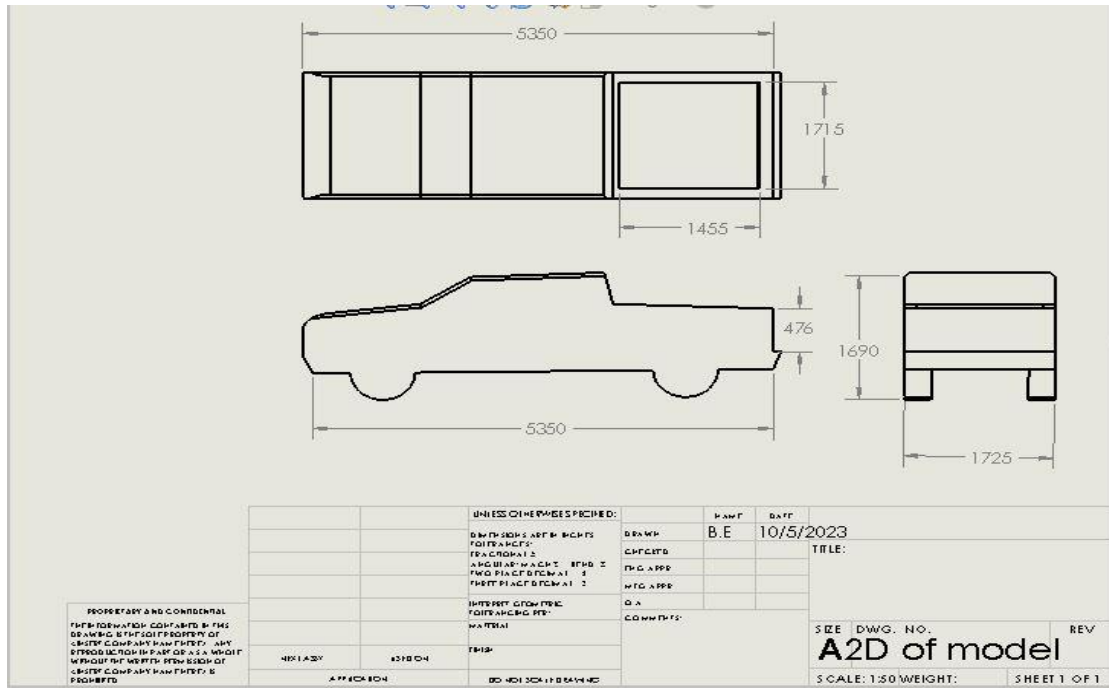
- [69] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, and M. Mohd, "Review of Research on Vehicles Aerodynamic Drag Reduction Methods," no. 02, 2014.
- [70] S. Ghosh, J. Choi, and J. R. Edwards, "Numerical Simulations of Effects of Micro Vortex Generators Using Immersed-Boundary Methods," no. March 2016, 2010, doi: 10.2514/1.40049.
- [71] M. Cakir, "CFD study on aerodynamic effects of a rear wing / spoiler on a passenger vehicle," 2012.
- [72] J. Aider *et al.*, "Drag and lift reduction of a 3D bluff-body using active vortex generators To cite this version : HAL Id : hal-01654042 Drag and lift reduction of a 3D bluff-body using active vortex generators," pp. 0–19, 2017.
- [73] U. Fern, "Fluid Dynamic Characterization of Vortex Generators and Two-dimensional Turbulent Wakes by Unai Fern ´ amiz," no. October, 2013.
- [74] C. Michael, "This thesis / project / dissertation has been reviewed for 508 compliance . To request enhancements , please email lib-508Accessibility@csus.edu .".
- [75] J. M. Wu and W. Q. Tao, "Numerical study on laminar convection heat transfer in a rectangular channel with longitudinal vortex generator. Part A: Verification of field synergy principle," *Int. J. Heat Mass Transf.*, vol. 51, no. 5–6, pp. 1179–1191, 2008, doi: 10.1016/j.ijheatmasstransfer.2007.03.032.
- [76] P. N. Selvaraju, "National Conference On Recent Trends And Developments In Sustainable Green Technologies Analysis of drag and lift performance in sedan car National Conference On Recent Trends And Developments In Sustainable Green Technologies Analysis of drag and lift pe," no. April, 2015.
- [77] A. Lemouedda, M. Breuer, E. Franz, T. Botsch, and A. Delgado, "International Journal of Heat and Mass Transfer Optimization of the angle of attack of delta-winglet vortex generators in a plate-fin-and-tube heat exchanger," vol. 53, pp. 5386–5399, 2010, doi: 10.1016/j.ijheatmasstransfer.2010.07.017.
- [78] T. Moghaddam and N. B. Neishabouri, "On the Active and Passive Flow Separation Control Techniques over Airfoils," *iop conf. ser. mater. sci. eng.*, vol. 248, no. 1, 2017, doi: 10.1088/1757-899X/248/1/012009.

- [79] L. Prandtl, "Boundary Layers," *Int. Geophys.*, vol. 47, no. 100, pp. 411–455, 1991, doi: 10.1016/S0074-6142(09)60067-X.
- [80] J. T. Exley and J. A. Brighton, "Flow Separation and Reattachment," *J. Basic Eng.*, no. June 1971, pp. 192–198, 1971.
- [81] Ola Lögdberg, "Turbulent boundary layer separation and control," no. December, p. 39, 2008.
- [82] P. Duquesne, J. Chanéac, G. Mondin, and J. Dombard, "Topology Rule-Based Methodology for Flow Separation Analysis in Turbomachinery," *Int. J. Turbomachinery, Propuls. Power*, vol. 7, no. 3, 2022, doi: 10.3390/ijtp7030021.
- [83] A. M. Al-garni, "Measurements of the cross-flow velocity field in the wake of an idealized pickup truck model using particle image velocimetry," pp. 7–10, 2008.
- [84] A. Jir, "Vortex-Generator Model and Its Application to Flow Control," no. November 2005, 2015, doi: 10.2514/1.12220.
- [85] R. D. Paul, A. A.- Faruk, and P. Karua, "A Numerical Study on the Drag Reduction of Sedan Car using Vortex Generator," pp. 1–6, 2020.
- [86] S. A. E. Mobilus, "A CFD Study of Drag Reduction Devices for a Full Size Production Pickup Truck," pp. 1–3, 2015, doi: 10.4271/2015-01-1541.
- [87] M. Koike, T. Nagayoshi, and N. Hamamoto, "Research on Aerodynamic Drag Reduction," no. 2, pp. 11–16.
- [88] A. Abikan, Z. Yang, and Y. Lu, "Computational Analysis of Turbulent Flow over a Bluff Body with Drag Reduction Devices," vol. 6, pp. 1210–1219, 2020, doi: 10.22055/JACM.2020.32667.2055.
- [89] I. Ibarra-udaeta, I. Errasti, U. Fernandez-gamiz, and E. Zulueta, "applied sciences Computational Characterization of a Rectangular Vortex Generator on a Flat Plate for Different Vane Heights and Angles," 2019, doi: 10.3390/app9050995.
- [90] A. A. Science and N. M. Etana, "Addis ababa science and technology university study of bishoftu bus exterior body design for drag and lift forces reduction using cfd simulation . A Thesis Submitted as a Partial Fulfillment to the Requirements for the," 2021.

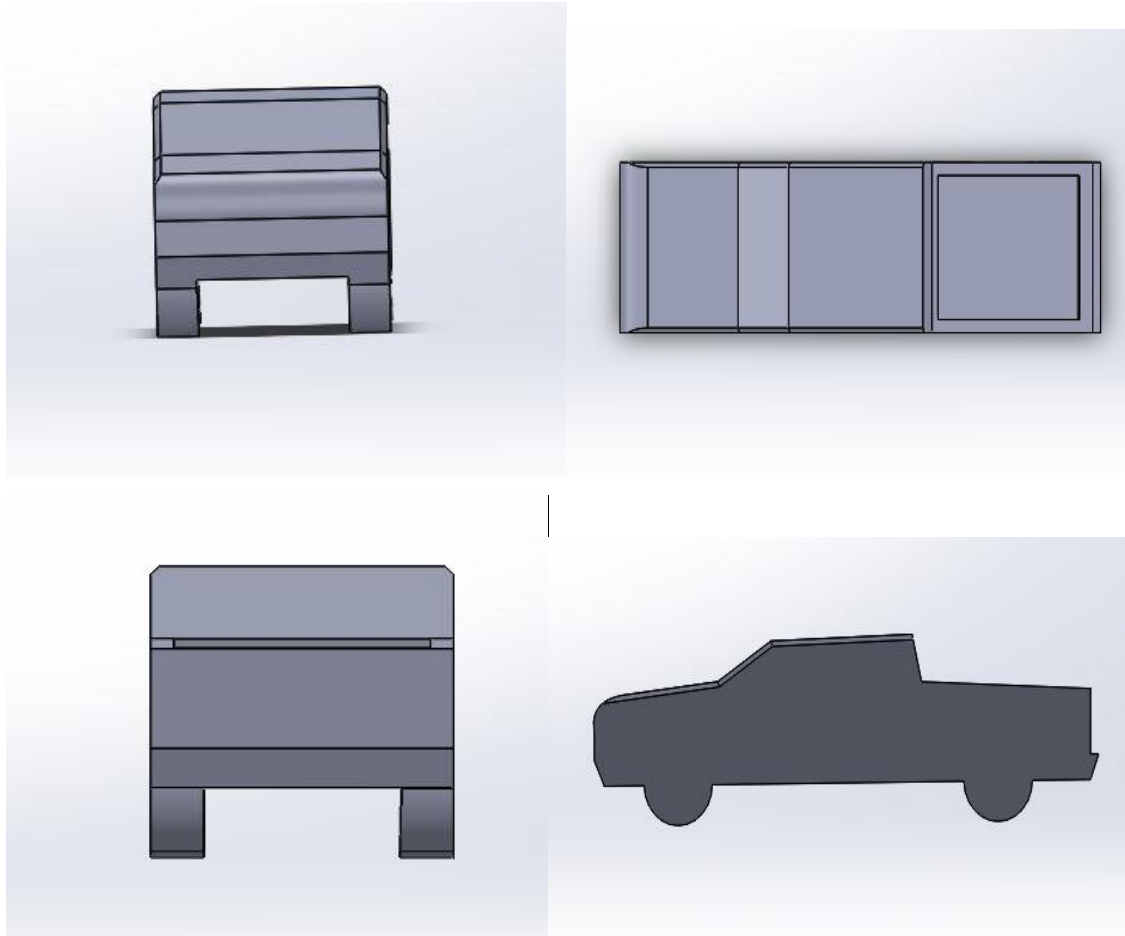
- [91] K. A. Raykowski and D. Beach, "Optimization of a Vortex Generator Configuration for a 1 / 4-Scale Piper Cherokee Wing Embry-Riddle Aeronautical University," 1999.
- [92] L. Brownlie, Y. Aihara, J. Carbo, R. Henry, I. Ilcheva, and P. Ostafichuk, "The use of vortex generators to reduce the aerodynamic drag of athletic apparel," *Procedia Eng.*, vol. 147, pp. 20–25, 2016, doi: 10.1016/j.proeng.2016.06.183.
- [93] A. Al-Saadi *et al.*, "Improvement of the aerodynamic behavior of a sport utility vehicle numerically by using some modifications and aerodynamic devices," *Sci. Rep.*, vol. 12, no. 1, pp. 1–14, 2022, doi: 10.1038/s41598-022-24328-w.
- [94] M. Cable, "An Evaluation of Turbulence Models for the Numerical Study of Forced and Natural Convective Flow in Atria," no. May, pp. 1–136, 2009.
- [95] I. U. Hatau, "Numerical investigation of external aerodynamic forces on extended cab pickup truck using rear spoiler-towards drag," no. September, 2021.
- [96] M. Desai, S. A. Channiwala, and H. J. Nagarsheth, "Experimental and Computational Aerodynamic Investigations of a Car 2 Experimental Approach," vol. 3, no. 4, pp. 359–368, 2008.
- [97] A. M. Al-garni and L. P. Bernal, "Experimental Investigation of the Near Wake of a Pick-up Truck," 2003.
- [98] G. Shankar and G. Devaradjane, "Experimental and Computational Analysis on Aerodynamic Behavior of a Car Model with Vortex Generators at Different Yaw Angles," vol. 11, no. 1, pp. 285–295, 2018, doi: 10.29252/jafm.11.01.
- [99] F. Dissertation, "Aerodynamic cfd study on a production vehicle experimental validation".
- [100] R. M. Alves and O. de Almeida, "Validation of Experimental and Numerical Techniques for Flow Analysis over an Ahmed Body," *Int. J. Eng. Res. Appl.*, vol. 07, no. 04, pp. 63–71, 2017, doi: 10.9790/9622-0704036371.
- [101] A. A. A. Hammad and T. Xing, "Aerodynamic Effects On Two-Lane Rural Highway Safety," 2017.
- [102] D. Wysocki and A. Szymanek, "Experimental study on Ahmed's body drag coefficient," *Tech. Sci.*, vol. 25, pp. 87–105, 2022, doi: 10.31648/ts.7897.

- [103] H. G. Castro, R. R. Paz, M. A. Storti, V. E. Sonzogni, O. Jorge, and M. E. De Bortoli, “Experimental and numerical study of the,” vol. xxix, pp. 15–18, 2010.
- [104] G. K. Rajamani, “CFD Analysis of Air Flow Interactions in Vehicle Platoons-TEZ,” *Manuf. Eng.*, no. August, 2006.
- [105] Y. Wang, Y. Xin, Z. Gu, S. Wang, Y. Deng, and X. Yang, “Numerical and experimental investigations on the aerodynamic characteristic of three typical passenger vehicles,” *J. Appl. Fluid Mech.*, vol. 7, no. 4, pp. 659–671, 2014, doi: 10.36884/jafm.7.04.21460.
- [106] S. Kanekar, P. Thakre, and E. Rajkumar, “Aerodynamic study of state transport bus using computational fluid dynamics,” *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 263, no. 6, 2017, doi: 10.1088/1757-899X/263/6/062052.

APPENDIX



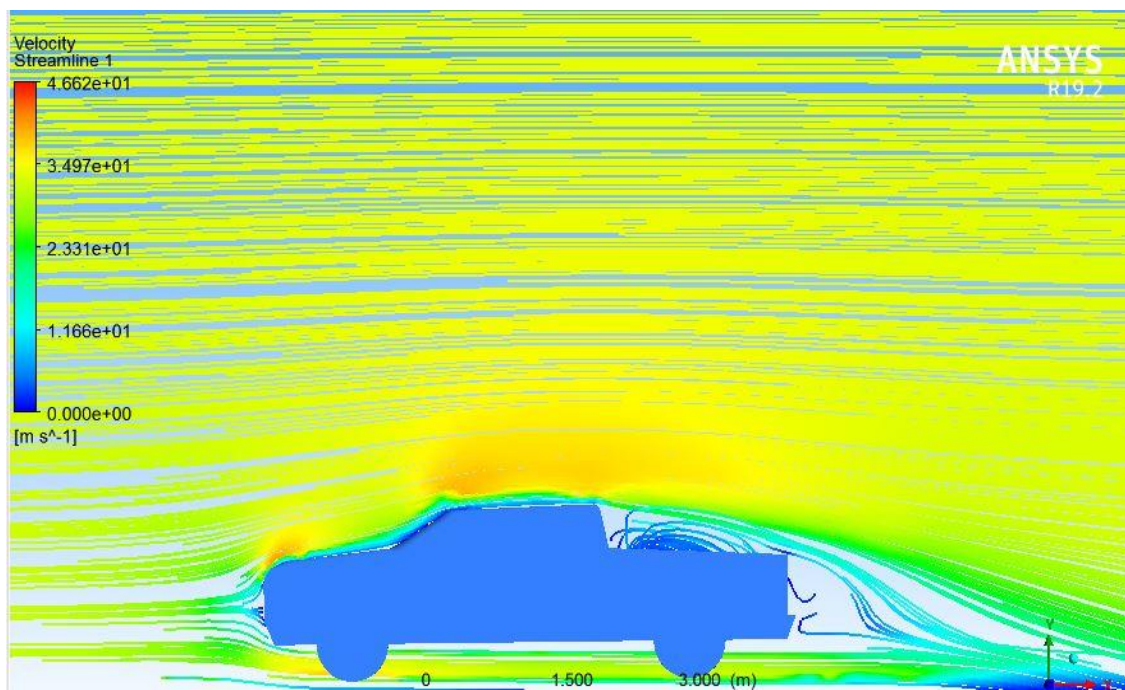
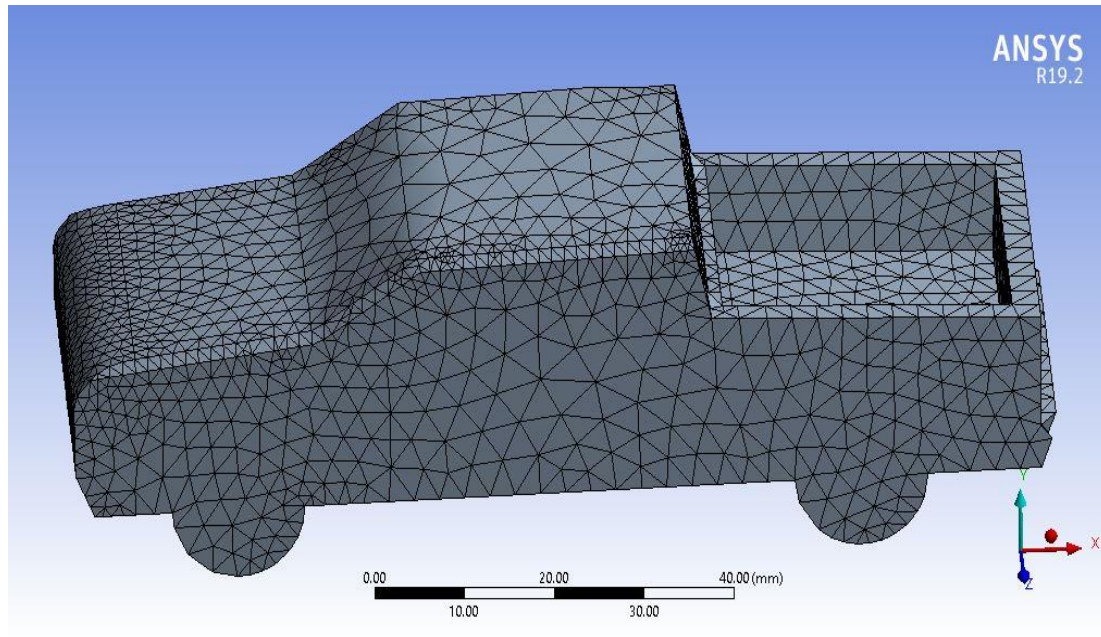
APPENDIX-1: 2D Model of Bishoftu double cabin pickup



APPENDIX-2: Front, Top, Rear and Side view of 3D Model respectively

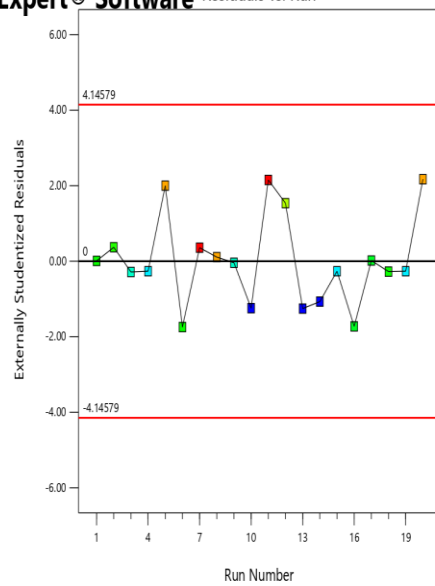
Domain	Boundaries	
solid	Boundary - car	
	Type	WALL
	Boundary - inlet	
	Type	VELOCITY-INLET
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Boundary - symmetry	
	Type	SYMMETRY
	Boundary - walls	
	Type	WALL

APPENDIX-3: Boundary condition for numerical analysis

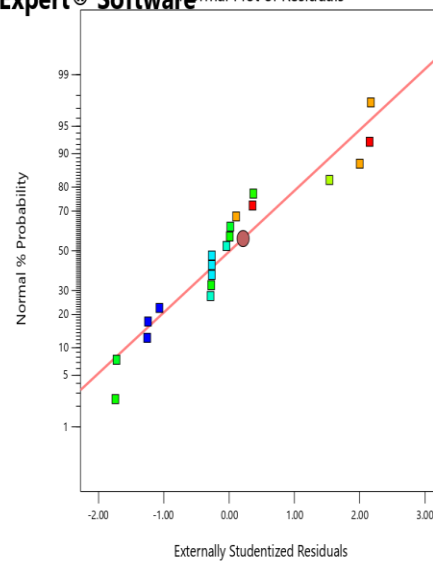


APPENDEX-4: Mesh and velocity streamline on the pickup model

Design-Expert® Software Residuals vs. Run

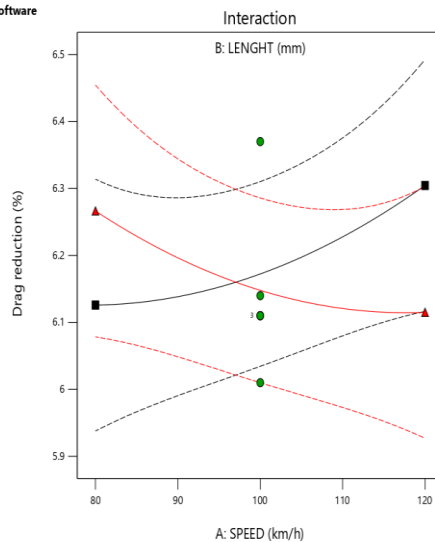


Design-Expert® Software Normal Plot of Residuals



APPENDEX-5: Residual and normal plot graph

Design-Expert® Software



Factor Coding: Actual

Drag reduction (%)

● Design Points

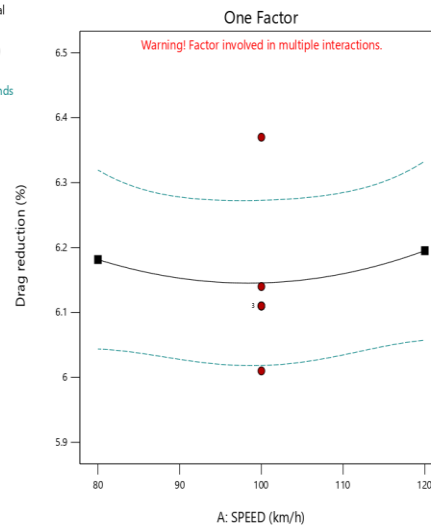
--- 95% CI Bands

X1 = A

Actual Factors

B = 20

C = 0.3822



APPENDEX-6: Interaction and one factor of drag reduction optimization