

AERODYNAMIC ANALYSIS OF A THREE WHEELER

A PROJECT REPORT

Submitted by

NIRMAL A J L A

(Reg No.: 2019111059)

In partial fulfillment for the award of the degree

of

BACHELOR OF ENGINEERING

in

MECHANICAL ENGINEERING

COLLEGE OF ENGINEERING GUINDY

ANNA UNIVERSITY : CHENNAI 600 025

JANUARY 2022

ANNA UNIVERSITY, CHENNAI 600 025

BONAFIDE CERTIFICATE

Certified that this project report “**AERODYNAMIC ANALYSIS OF A THREE WHEELER**” is the bonafide work of “**NIRMAL A J L A (Reg No:2019111059)**” who carried out the project work under my supervision.



Mr. R. VIJAYANAND

SUPERVISOR

Teaching Fellow,
AU-FRG Institute for CAD/CAM,
Department of Mechanical Engg.,
College of Engg., Guindy Campus,
Anna University,
Chennai - 600 025

ABSTRACT

The three-wheeler also known as the Auto-Rickshaw is a very popular form of mass transportation in India. More than 6 Lakh three wheelers were sold in india in the year 2020. The standard design of the three-wheeler still in use today was done in the last century, therefore the aerodynamic performance of this design leaves more to be desired, as the aerodynamic efficiency was not a major factor when the Auto-rickshaw was designed in the 1950s. It is also a very well known fact that these auto-rickshaws are a prime candidate for electrification in the upcoming years, which would electrify a large portion of the Indian traffic very easily and affordably. Keeping this in mind it is necessary to improve the aerodynamic performance of the auto-rickshaw to achieve better efficiency and reduce the carbon footprint. This study aims to simulate and analyze the aerodynamic lift and drag of the Auto-Rickshaw , then to analyze the results and to propose a new revised design that will improve the performance of the three-wheeler. The results would then be compared and contrasted with the previous results obtained in order to provide design suggestions that would provide the much needed facelift for the Classic Auto-rickshaw design. The Drag-coefficient and Lift-coefficient were used as a metric to measure the performance improvement that was achieved with the revised design.

TABLE OF CONTENTS

CH.NO	TITLE	PAGE NO.
	ABSTRACT	iii
	LIST OF FIGURES	vii
	LIST OF TABLES	ix
	LIST OF SYMBOLS AND ABBREVIATIONS	xi
1.	INTRODUCTION	1
1.1	AUTOMOTIVE AERODYNAMICS	1
1.2	DRAG FORCE	2
1.3	LIFT FORCE	3
1.4	DETERMINATION OF Cd AND Cl	5
1.5	COMPUTATIONAL FLUID DYNAMICS	5
1.5.1	STEPS INVOLVED IN CFD ANALYSIS :	5
2.	LITERATURE REVIEW	6
2.1	INTRODUCTION	6
2.2	LITERATURE SUMMARY	6
2.3	CONCLUSION	8
3.	NEED, OBJECTIVES AND METHODOLOGY	9
3.1	NEED FOR THIS STUDY	9
3.2	OBJECTIVES	10
3.3	METHODOLOGY	11

4. PRE-PROCESSING	12
4.1 MODELING	12
4.2 DOMAIN	14
4.3 MESHING AND BOUNDARY CONDITIONS	15
4.3.1 MESHING	15
4.3.2 BOUNDARY CONDITIONS	17
4.4 DETERMINATION OF FRONTAL AREA	18
5. SIMULATION OF CLASSIC DESIGN	20
5.1 FLUID SELECTION	20
5.2 SOLVER SETTINGS	20
5.3 SIMULATION	22
5.4 RESULT SUMMARY : CLASSIC MODEL	22
6. DESIGN REVISION	24
6.1 ANALYZING THE OLD DESIGN	24
6.2 REVISED DESIGN	26
6.3 SIMULATION OF THE REVISED MODEL	28
6.4 RESULT SUMMARY : REVISED DESIGN	31
6.5 ANALYSIS OF THE REVISED DESIGN RESULTS	32
7. COMPARISON CHARTS AND RESULTS	34
7.1 COMPARISON OF THE VELOCITY CONTOUR CHARTS	34
7.2 COMPARISON OF SURFACE PRESSURE CHARTS	36
7.3 COMPARISON OF STREAMLINE CHARTS	37
7.4 COMPARISON OF DRAG AND DRAG-COEFFICIENT	39

7.5 COMPARISON OF LIFT AND LIFT-COEFFICIENT	41
7.6 FINAL RESULTS	43
8. CONCLUSION	45
APPENDIX 1	47
REFERENCES	49

LIST OF FIGURES

FIG NO.	TITLE	PAGE NO.
Fig 1.1	Aerodynamic Forces on an Automobile	2
Fig 1.2	Lift Generation in a Wing	4
Fig 4.1	LOVSON Design Specifications	12
Fig 4.2	Classic Design Front View	13
Fig 4.3	Classic Design Top view	13
Fig 4.4	Classic Design Isometric View	13
Fig 4.5	Classic Design Side view	13
Fig 4.6	Domain Isometric View	14
Fig 4.7	Domain Side View	14
Fig 4.8	Mesh Generated	17
Fig 4.9	Mesh of Domain	17
Fig 4.10	Mesh View	17
Fig 4.11	Determination of frontal Area	19
Fig 5.1	Velocity Contour Plot at 50 km/h	23
Fig 5.2	streamlines at 50 km/h	23
Fig 5.3	Surface Pressure at 50 km/h	23
Fig 6.1	Analysis of Velocity Contour Plot of Classic Design	24
Fig 6.2	Streamlines	25
Fig 6.3	Surface Pressure Chart	25
Fig 6.4	Revised design Section view	26
Fig 6.5	Revised design Side View	27
Fig 6.6	Revised design Front View	27

Fig 6.7	Revised design Isometric view	27
Fig 6.8	Revised design Top View	27
Fig 6.9	Domain for Revised Model	28
Fig 6.10	Mesh View	29
Fig 6.11	Revised Design in Mesh	29
Fig 6.12	Frontal Area of Revised Design	31
Fig 6.13	Velocity Contour of Revised Model at 50 km/h	32
Fig 6.14	Revised Model Streamlines	33
Fig 6.15	Revised model Surface Pressure	33
Fig 7.1	Drag Force Comparison	40
Fig 7.2	Drag Coefficient Comparison	40
Fig 7.3	Lift force Comparison	42
Fig 7.4	Lift Coefficient Comparison	43
Fig 7.5	Mean Forces Comparison	44
Fig 7.6	Mean Coefficients Comparison	44

LIST OF TABLES

TABLE NO.	TITLE	PAGE NO.
Table 1	Classic Model Geometry details	12
Table 2	Domain details	15
Table 3	Basic Mesh Dimensions	16
Table 4	solid/Fluid Interface	16
Table 5	Number of Cells	16
Table 6	Boundary Conditions	18
Table 7	Frontal Area of classic Model	19
Table 8	Density of Air	20
Table 9	Solver Settings	20
Table 10	Gravity Settings	21
Table 11	Ambient Conditions	21
Table 12	Classic model Results	22
Table 13	Revised Model Geometry Details	28
Table 14	Domain for Revised Model	29
Table 15	Revised Model Basic Mesh	30
Table 16	Revised model Solid/Fluid Interface	30
Table 17	revised model No. of Cells	30
Table 18	Frontal area of Revised Model	31
Table 19	Revised model results	32
Table 20	Velocity Contour Comparison	34
Table 21	surface Pressure Comparison	36
Table 22	streamlines Comparison	37

Table 23	Drag Force Comparison	39
Table 24	Drag Coefficient comparison	39
Table 25	lift Force Comparison	41
Table 26	Lift coefficient Comparison	42
Table 27	Final Results	43

LIST OF SYMBOLS AND ABBREVIATIONS

SYMBOLS		DEFINITIONS
ρ	-	Density of Fluid
v	-	Velocity
D or D	-	Drag force
L or l	-	Lift Force
Cd or C_d	-	Drag Coefficient
Cl or C_l	-	Lift Coefficient

Classic or Legacy Model - always refers to the old design of Auto-Rickshaw that We see daily in our streets

Revised Model - Refers to the new improved design that is models for this study (refer pg. 26,27)

1. INTRODUCTION

1.1 AUTOMOTIVE AERODYNAMICS

Automotive aerodynamics is the study of the aerodynamics of road vehicles. The main goal of automotive aerodynamics is to measure and improve or optimize the external forces arising on the automobile by the virtue of the air moving around the body of the vehicle. Air is also considered a fluid in this case.

Aerodynamics is the science of how air moves around objects (vehicle in this case). Air has some viscosity, even though low, this can cause a resistance to movement. As the vehicle is in motion, so is the air around it, this relative motion between vehicle and air causes pressure gradients in the air around the vehicle, these pressure gradients can cause undesired forces and instabilities to arise.

In this case, the air is assumed to be incompressible and the flow of air is considered to follow the NAVIER-STOKES EQUATION (a partial differential equation that describes the flow of incompressible fluids). These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. The equations are a set of coupled differential equations and could, in theory, be solved for a given flow problem by using methods from calculus. But, in practice, these equations are too difficult to solve analytically.

Therefore two methods are used widely to analyze the air flow over vehicles - i) Computational Fluid Dynamics, ii) Wind Tunnel Testing. This project tackles the analysis through Computational Fluid Dynamics to study the aerodynamic performance of a Typical Indian Three Wheeler.

The Two main forces that we are concerned with in this project are the Drag Force and Lift Force

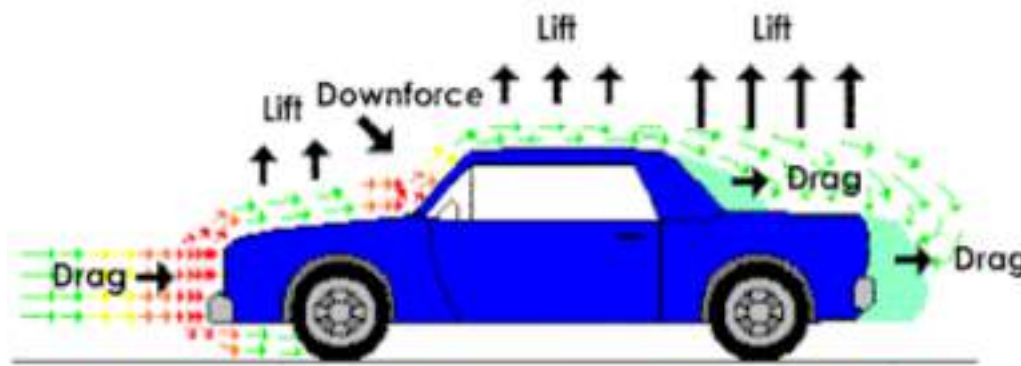


Fig 1.1 Aerodynamic Forces on an Automobile

1.2 DRAG FORCE

Air is made up of molecules as we all know. There are around 10^{18} molecules in 1 cc of air on average. When a solid object moves through the air, it interacts with all these molecules and moves them aside in order to occupy the space, i.e. displaces them. But each and every molecule has mass and therefore inertia which presents itself as small forces acting on the body. These small forces add up to provide a resistance to the motion of the body through the air. This is known as Aerodynamic Drag.

Aerodynamic drag is the force needed to overcome these miniature forces when an object moves through air at a certain velocity. You can feel this force when you ride a bike. The faster you travel, the more air you have to push through and the higher your induced drag.

The Drag Force is always opposite to the direction of motion of the vehicle. Therefore some of the power generated by the vehicle is wasted in overcoming the Drag force which decreases its efficiency. Even at speeds as low as 20km/h, over half the effort required to pedal a bike is due to overcoming aerodynamic drag. This is because the aerodynamic forces acting on a body increase with the square of velocity, as shown by the equation below.

$$Drag(D) = \left(\frac{1}{2}\right) C_d \times \rho v^2 \times A$$

Where:

C_d = drag coefficient

ρ = density (kg/m³)

v = velocity (m/s²)

A = frontal area (m²)

In general, the dependence on body shape, inclination, air viscosity, and compressibility is very complex. One way to deal with complex dependencies is to characterize the dependence by a single variable. For drag, this variable is called the drag coefficient, designated " C_d ".

1.3 LIFT FORCE

A fluid flowing around an object exerts a force on it. Lift is the component of the force produced in the upward direction, opposing the force of gravity. It is produced by the virtue of the pressure gradient created by the fluid flow on the top and bottom of the vehicle body.

A car is shaped like an airfoil (like a wing of a bird or plane). As the car accelerates the air moves faster around it and you get a vertical force that lifts it from the ground. As the uplifting force increases, it reaches a point where the force of gravitation gets negated and the tip of the car gets lifted up.

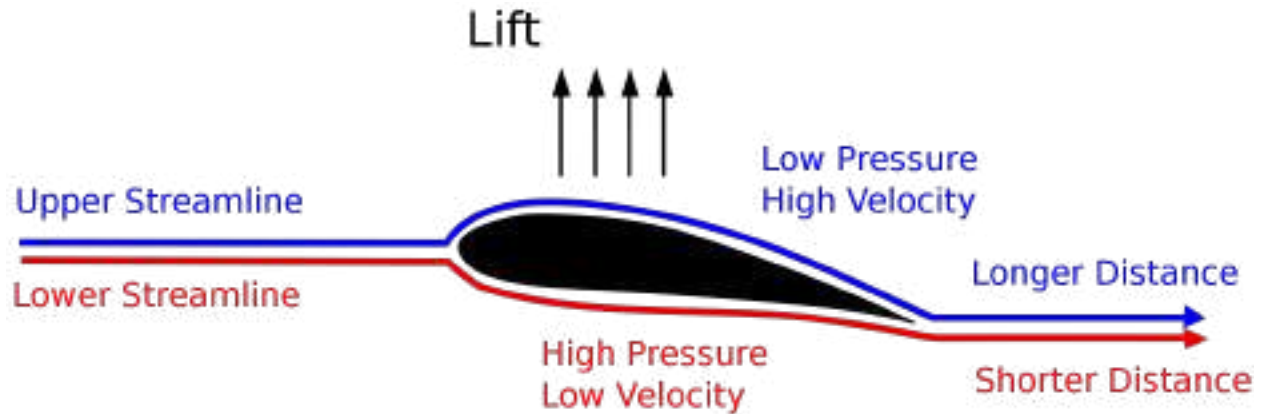


Fig 1.2 Lift Generation in a Wing

Lift depends on the density of the air, the square of the velocity, the air's viscosity and compressibility, the surface area over which the air flows, the shape of the body, and the body's inclination to the flow. The Lift Equation is given by:

$$Lift(L) = \left(\frac{1}{2}\right) C_l \times \rho v^2 \times A$$

Where:

C_l = Lift coefficient

ρ = density (kg/m^3)

v = velocity (m/s)

A = Reference area (m^2)

Similar to the Drag Coefficient, The lift coefficient is a number that aerodynamicists use to model all of the complex dependencies of shape, angle of attack, and some flow conditions on lift.

1.4 DETERMINATION OF Cd AND Cl

Evaluation of vehicle aerodynamics and corresponding refinements are a continuous process and an integral part of automotive engineering, not limited to the vehicle initial design phase only. Typical analysis and evaluation tools used in this process may include wind tunnel testing, computational prediction, or track testing.

This study focuses on testing the drag and lift performance of a three wheeler using computational Fluid Dynamics and to improve upon the existing design.

1.5 COMPUTATIONAL FLUID DYNAMICS

The integration of computational fluid dynamic (CFD) methods into a wide range of engineering disciplines is rising sharply, mainly due to the positive trends in computational power and affordability. One of the advantages of these methods, when used in the automotive industry, is the large body of information provided by the “solution.” Contrary to wind tunnel or track tests, the data can be viewed, investigated, and analyzed over and over, after the “experiment” is concluded.

1.5.1 STEPS INVOLVED IN CFD ANALYSIS :

- Identification of problem (i.e: determine drag and lift coefficient)
- Modeling of geometry (i.e: three wheeler)
- Define domain and boundary conditions
- Mesh generation
- Performing the simulation
- Analysis of the results

“Solidworks” for modeling and “Solidworks Flow Simulation” for Simulation and analysis are the softwares used for our study.

2.LITERATURE REVIEW

2.1 INTRODUCTION

Several journals and books were used to gain an in-depth understanding on the subject matter relating to this topic. The important journals are listed below and the rest are mentioned in the bibliography. All journals read and mentioned here were chosen on the basis of the actual professional presentation of their results and conclusions , and also based on the key insights and knowledge about some obscure technical and non-technical challenges faced during the writing of a paper of this nature.

2.2 LITERATURE SUMMARY

- “An Introduction to Automotive Aerodynamics- Dr. Joseph Katz ”, provided the basic understanding of the concepts necessary to carry out the design and solution of the problem at hand in terms of aerodynamics relating to automobiles
- As information on lift coefficient and lift in general were lacking in typical automotive papers we turned to the paper “Commercial Airplane Design Principles, 2014 - Pasquale Sforza” on commercial airplane design to gain a better understanding on the same.
- “Auto Future: Active Aerodynamics, David Moreria, The Truth About Cars, Jan. 8, 2009” provided much needed clarity on the aerodynamic performance of automobiles and the various techniques used in the automotive industry to reduce the life generated.

- “Aerodynamic Analysis of a Concept car model - Shubhankar Pal , S.M.Humayun Kabir , Md. Mehdi Masud Talukder “ this paper provided insight into the aerodynamic performance testing of an automobile. The graphs and reports given in this paper gave us an understanding about the data collection and presentation of the data in a coherent manner needed to convey our results.
- “Aerodynamic Test Techniques - Jay C. Kessler and Stanley B. Wallis” described the various methods and techniques used to test aerodynamics of a vehicle such as CFD, wind tunnel testing and track testing .This helped us in choosing CFD as the mode of testing for our project.
- “Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5 - Akshay Parab Ammar Sakarwala Bhushan Paste Vaibhav Patil (2014)” - this paper provided an understanding of the process of comparing two designs in an empirical way. It also provided some insights into the presentation of the simulation data in an understandable way for the reader.
- “Numerical Basis of CAD-Embedded CFD *White Paper* - Dr. A. Sobachkin, Dr. G. Dumnov, Dr. A. Sobachkin (2014)” - This is the white paper published by dassault systems detailing the mathematical model used for meshing and the flow simulation solver used in Solidworks, this paper was instrumental in understanding this new simulation technology and setting up the simulation correctly and to get results within acceptable levels of accuracy and precision.

- “Determination of Air drag coefficient of vehicle models - Boran Pikula , Elmedin Mešić, Mirzet Hodžić (2008)” - this paper details the measurement of drag coefficient of vehicles physically using a wind tunnel. This paper provided an understanding of the key differences between the physical and CFD approaches and detailed the variance that can be expected between the wind tunnel and the CFD approach.
- “Aerodynamic Study of a Three Wheeler Body - C. Bhaskar ,Krishna Rawat & Muhammed Minhaj” is the only study that we could find that deals with the aerodynamic drag of a three wheeler specifically, But the authors used a small toy model and did physical testing using a wind tunnel and determined the drag coefficient to be 0.4975. It just provides this information and does not improve upon the already existing design
- “Determination of Air drag coefficient of vehicle models - Boran Pikula , Elmedin Mešić, Mirzet Hodžić (2008)” - this paper details the measurement of drag coefficient of vehicles physically using a wind tunnel. This paper provided an understanding of the key differences between the physical and CFD approaches and detailed the variance that can be expected between the wind tunnel and the CFD approach.

2.3 CONCLUSION

Only the last two papers deal with Three-wheelers and None of these papers provide any ways for the improvement of the existing design, all of them just analyze the drag forces on the existing design and provide similar results. The Lift component of the Auto is also completely ignored. A research gap was identified and this study strives to fill this gap.

3. NEED, OBJECTIVES AND METHODOLOGY

3.1 NEED FOR THIS STUDY

The AUTO-RICKSHAW is one of the more popular modes of mass transportation in India and some other southeast asian countries. The auto-rickshaw is an urban transport which is used for transportation over short distances. It is not suited for travel over long distances as it is not fully enclosed for ease of use and operation and has only three wheels for tight maneuvering within congested traffic and city streets.

The most common auto-rickshaw design is characterized by a sheet-metal body or open frame resting on three wheels, a canvas roof with drop-down side curtains. A small cabin at the front for the driver with handlebar controls, and a passenger seat in the rear.

This common and iconic design was made in the last century and aerodynamic performance was not considered as a primary factor. Ease of manufacturing and cost were the main contributing factors to the design of the auto-rickshaw.

With the impending doom of fossil fuel depletion and the rapid move towards electric vehicles to reduce the carbon footprints, we can anticipate that the switch to all electric is closer than we expect. The Switch to electric vehicles will be a slow and difficult process for a country as huge as India. Auto-rickshaws have a fairly simple design that is a prime candidate for electrification, and based on the mostly urban usage of these autos, the electrification of autos makes a lot of sense. This would be a logical first step towards the complete electrification of the Indian subcontinent.

The electrification of any vehicle adds a lot of mass in terms of the battery required to drive the vehicle, therefore a more efficient and effective aerodynamic

design of the vehicle will be hugely beneficial in terms of range and speed that can be achieved. Moreover , a better aerodynamic design will be a huge upgrade to Autos with internal Combustion Engines too, reducing carbon emissions in the interim period.

Also starting with a small and easy to work with vehicle for electric vehicle that also has a lot of public outreach is a very good way to acclimate the public to electric vehicles.

This study aims to improve upon the current aerodynamic design of the auto-rickshaw and therefore will be highly beneficial in the coming years.

3.2 OBJECTIVES

The objectives of this study are twofold:

1. To model and analyze the aerodynamic design of the current auto-rickshaw (legacy design) , especially the Drag and Lift forces acting on the Rickshaw
2. To improve upon the current design , with the help of results obtained and to suggest revisions to the design, aimed at better aerodynamic performance

This study aims to improve the design of the Auto-Rickshaw without compromising on the ease of use factor of the current design , in the process making the auto-rickshaw more efficient and priming it as a main candidate for electrification.

3.3 METHODOLOGY

With the objectives laid out, the method of aerodynamic study needed to be decided upon very early in order to proceed with the analysis. Wind tunnel test is a tried and tested way of analyzing aerodynamics , but it requires the fabrication of a model and specialized wind tunnel testing facilities .

Due to the ongoing restrictions related to COVID-19 and the lack of time for the fabrication of a physical model, A Computation Fluid Dynamics Approach was preferred.

The next step in a CFD approach was modeling of the required geometry for the analysis. The software SOLIDWORKS was chosen for the modeling as it was highly compatible with “SOLIDWORKS FLOW SIMULATION” , our fluid solver of choice. It also eliminates the need for the conversion file formats of the model to transfer from one software to another, which was a great time saver.

Followed by the modeling, the solid and fluid domains and the boundaries were specified, followed by which the geometry was meshed using the cell-based cartesian mesher in Solidworks (these steps are discussed in detail in the coming sections). The simulation was run using the optimal settings for the problem and the results were recorded and processed to readable data.

From the obtained data , the auto-rickshaw was analyzed for flaws in design , with these findings in mind, a revised design was modeled in solidworks, making sure to optimize and remove any bottlenecks in the current design.

This revised design was again modeled, meshed and was subjected to the same flow simulation done in the previous steps, and the new data was recorded and processed.

This new data was then compared with old data to see whether the design was improved and the aerodynamic efficiency of the Auto-rickshaw was improved. This data is presented in a logical manner in upcoming chapters.

4. PRE-PROCESSING

4.1 MODELING

Modeling of the geometry is the first step in the analysis process. The Schematics for the classic Auto-rickshaw legacy design was obtained from LOVSON , a auto rickshaw manufacturer based in Mumbai.

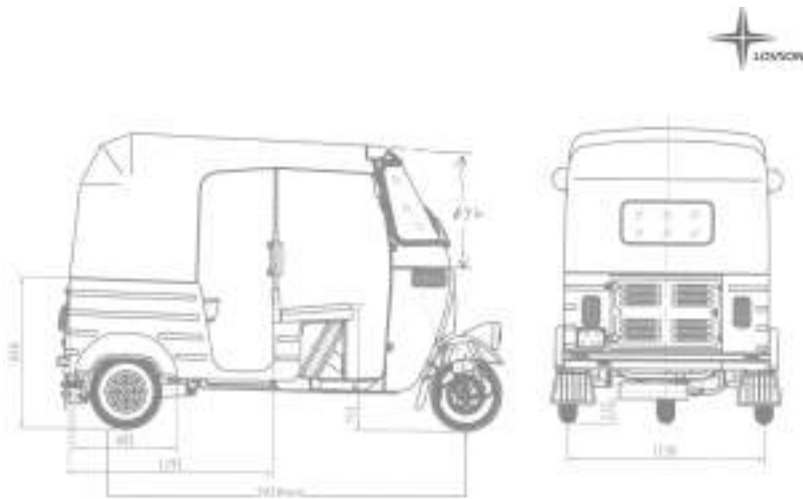


Fig 4.1 LOVSON Design Specifications.

Some dimensions were not available in this design drawing, therefore measurements were made on a local auto-rickshaw to fill in the missing dimensions and cross-check these basic dimensions.

The model used for the simulation :

Geometry Details

Table 1. Classic Model Geometry details

Nodes	2776
Vertices	23271
Polygons	18591

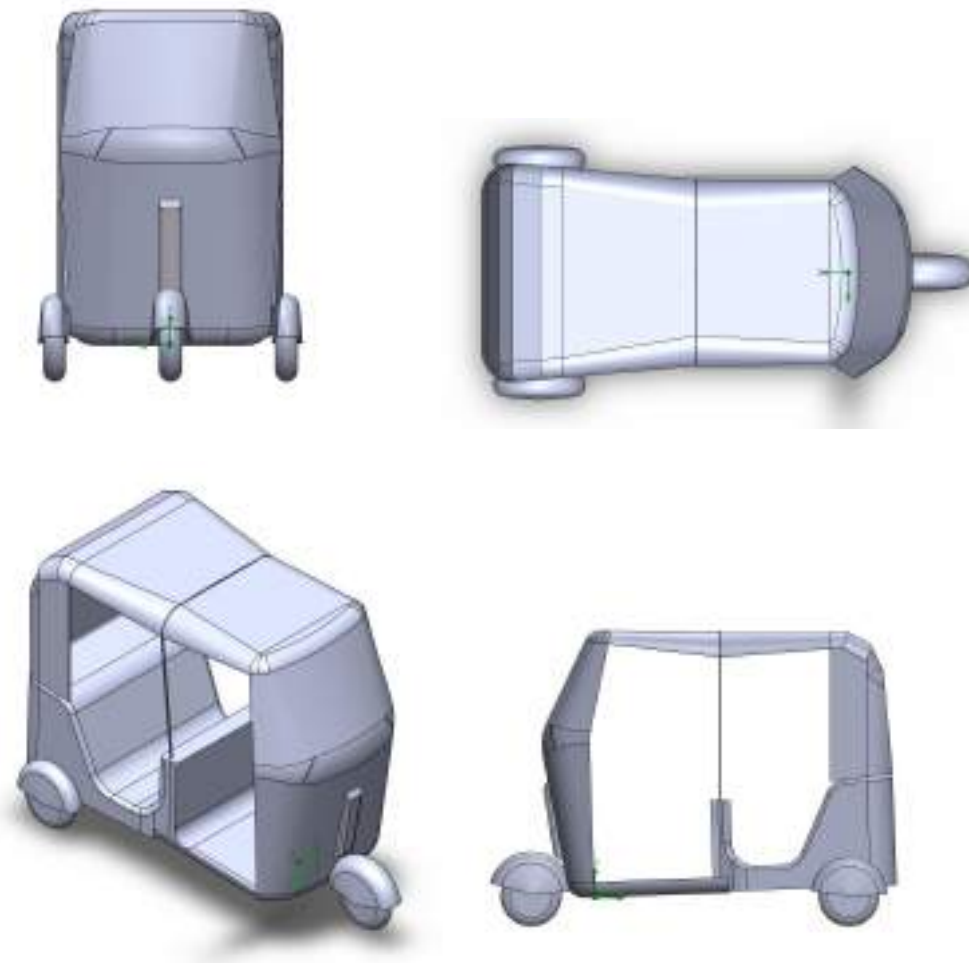


Fig 4.2 Classic Design Front View , Fig 4.3 Top view, Fig 4.4 Isometric View, Fig 4.5 Side view (From Top Left).

As mentioned earlier SOLIDWORKS was used for the modeling of this auto-rickshaw. The Driver's seat, passenger seat, the handlebar and the rear luggage area were either omitted or mottled without much detail, since they do not play any role in the aerodynamic performance of the vehicle , which the upcoming simulations shall prove, as extra details would increase compute time. Effort was put in to make sure the outer surface in contact with the air was as close to the design specifications as possible and to make sure that the model was watertight which is important for the accuracy of results in CFD.

4.2 DOMAIN

A domain in CFD is a three-dimensional space in which the solution is calculated. In external aerodynamics, a CFD simulation of the flow around a geometrical object is required, therefore the computational domain is a volume of adequate dimensions around the geometry of interest.

The Domain defined for our calculations is given below:

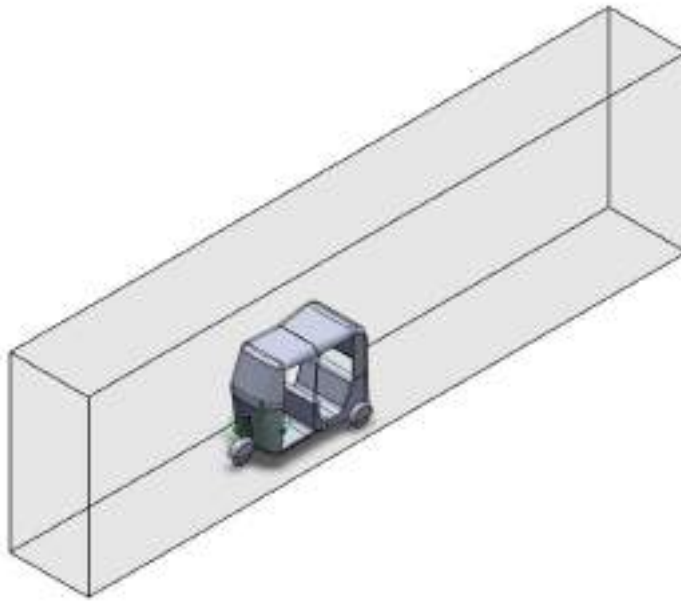


Fig 4.6 Domain Isometric View

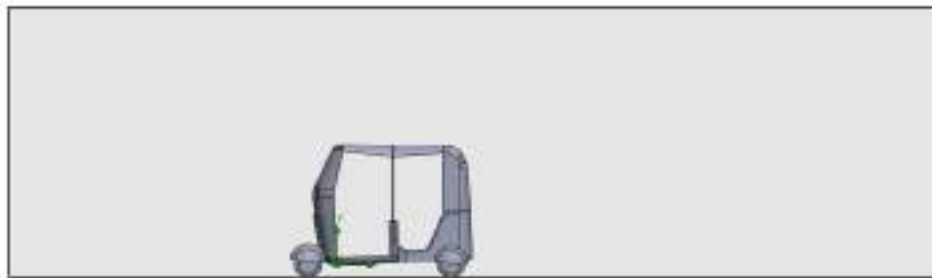


Fig 4.7 Domain Side View

The extra length of the Domain is to account for the eddy currents and turbulence that will be created at the rear of the vehicle , since the auto-rickshaw has sharply converging body work in the back.

DOMAIN DETAILS:

Table 2. Domain details

X min	-0.850 m
X max	0.850 m
Y min	-0.180 m
Y max	3.600 m
Z min	-4.587 m
Z max	8.448 m
X size	1.700 m
Y size	3.780 m
Z size	13.035 m

4.3 MESHING AND BOUNDARY CONDITIONS

SOLIDWORKS FLOW SIMULATION uses a different type of meshing compared to other conventional CFD softwares known as the Cartesian Mesher. This Mesher is the reason for the speed of computation on this software. A basic understanding of this mesher is required to set up the simulation correctly. (*Refer Appendix 1*)

4.3.1 MESHING

The mesh was generated using the cartesian mesh as said before and the mesh was refined further using the manual refinement parameters.

Basic Mesh Dimensions

Table 3. Basic Mesh Dimensions

Number of cells in X	16
Number of cells in Y	28
Number of cells in Z	88

Note that the resolution in the z direction is higher as it the direction of fluid flow.

Solid/Fluid Interface

Table 4. solid/Fluid Interface

Small Solid Feature Refinement Level	4
Curvature Level	3
Curvature Criterion	0.318 rad
Tolerance Level	0
Tolerance Criterion	0.327 m

Number Of Cells

Table 5. Number of Cells

Cells	105996
Fluid cells	105996
Irregular cells	0
Trimmed cells	1771

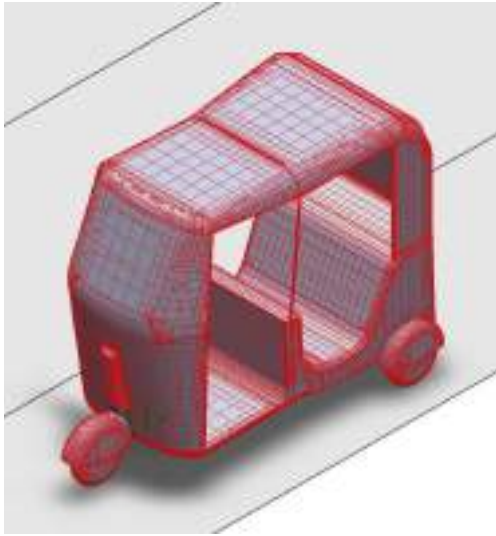


Fig 4.8 Mesh Generated

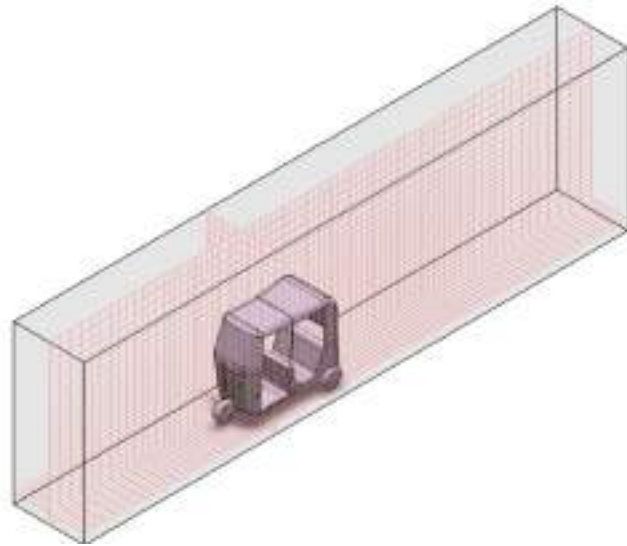


Fig 4.9. Mesh of Domain

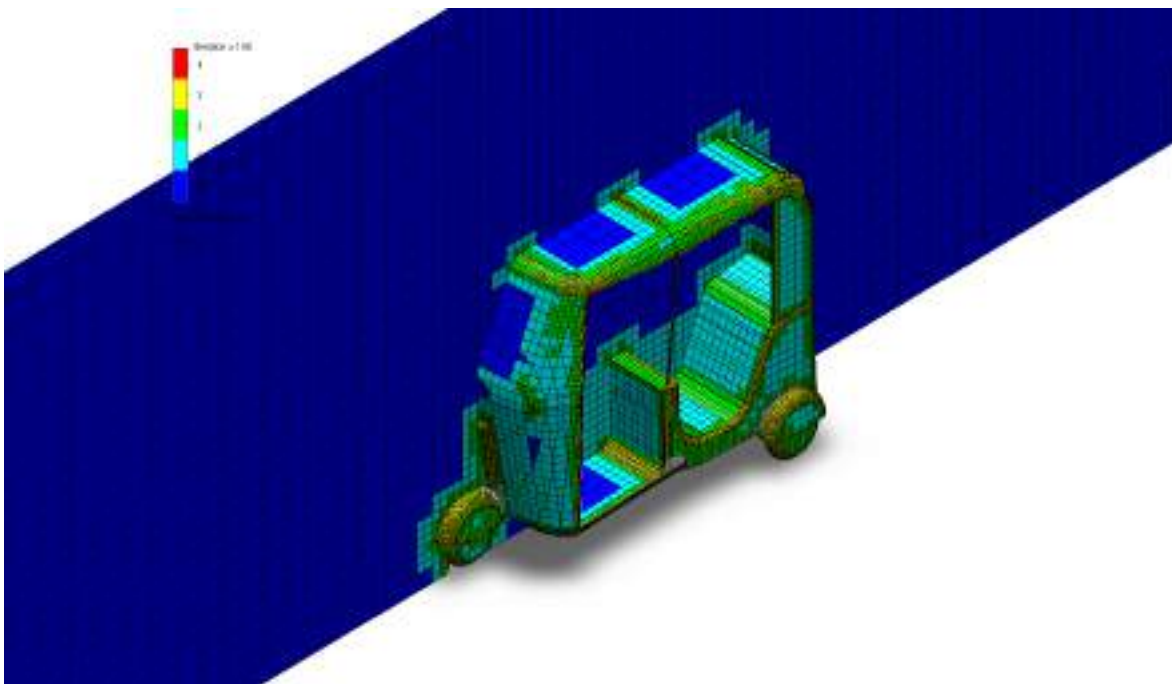


Fig 4.10 Mesh View

4.3.2 BOUNDARY CONDITIONS

Boundary conditions are constraints that are required for obtaining the solution of a differential equation. In this case this is the velocity of the fluid as it enters and exits the fluid domain.

Based on the typical traffic in Indian cities and speed considerations. The speeds of 10 , 20 ,30 ,40, 50 km/hr were chosen to find the Drag and Lift forces.(Drag and lift are dependant on velocity, drag and lift coefficients are also dependent on velocity, these are the goals of our analysis)

Boundary Conditions

Table 6 Boundary Conditions

2D plane flow	None
At X min	Default (No slip)
At X max	Default (No slip)
At Y min	Default (No slip)
At Y max	Default (No slip)
At Z min	Default (Inlet)
At Z max	Default (Outlet)
Velocity parameters	Velocity vector
	Velocity in X direction: 0 m/s
	Velocity in Y direction: 0 m/s
	Velocity in Z direction: (input :10-50 km/hr)

4.4 DETERMINATION OF FRONTAL AREA

The Drag and Lift coefficients are the measure of the complex dependencies of the shape of the vehicles and the interactions between the air flow around the vehicles. They are numerical measurements of the lift and drag performance of a vehicle and are very useful in comparing the efficiency of two vehicles, rearranging the lift and drag equations

$$C_d = 2 * Drag Force / \rho v^2 * A$$

$$C_l = 2 * Lift Force / \rho v^2 * A$$

Here “A” is the frontal area or the reference area which needs to be determined from the model. The silhouette of the model is projected to a plane and the area of the silhouette is measured in solidworks.

Table 7. Frontal Area of classic Model

Frontal Area (Classic design)	1708701.9 mm ²
	1.7087 m ²

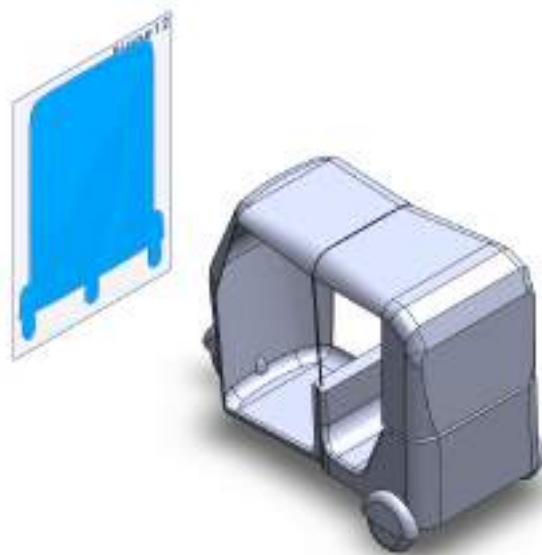


Fig 4.11 Determination of frontal Area

5. SIMULATION OF CLASSIC DESIGN

5.1 FLUID SELECTION

The selection of material of the fluid plays an important role in the drag and lift forces produced by the flow solver, since both drag and lift are directly proportional to the density of the fluid that flows around them.

Air was chosen as the fluid , from the solidworks database .

FLUID : AIR (Standard)

Table 8 Density of Air

Density	1.225 kg/m ³
---------	-------------------------

5.2 SOLVER SETTINGS

The solver settings are the parameters and the selections of the algorithms that the software will use to solve the problem, based on the results required and the initial conditions provided.

In our case , The EXTERNAL FLOW solver of solidworks flow simulation was chosen which is a pressure based, steady state solver. The K-epsilon model was used for the turbulence model.

Solver settings:

Table 9 Solver Settings

Units system	SI (m-k-g-s)
Analysis type	External (exclude internal spaces)
Exclude cavities without flow conditions	On
Coordinate system	Global Coordinate System
Reference axis	X

Physical Features

Heat conduction in solids: Off

Time dependent: Off

Gravitational effects: On

Rotation: Off

Flow type: Laminar and turbulent

High Mach number flow: Off

Humidity: Off

Free surface: Off

Default roughness: 0 micrometer

Gravitational Settings

Table 10 Gravity Settings

X component	0 m/s ²
Y component	-9.81 m/s ²
Z component	0 m/s ²

Default wall conditions: Adiabatic wall

Ambient Conditions

Table 11 Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa
	Temperature: 293.20 K
Turbulence parameters	Turbulence intensity and length
	Intensity: 0.20 %
	Length: 0.013 m

5.3 SIMULATION

With these solver settings the flow over the classic design was simulated five times with varying the velocities:

Case 1 : 10 km/h

Case 2 : 20 km/h

Case 3 : 30 km/h

Case 4 : 40 Km/h

Case 5 : 50 km/h

5.4 RESULT SUMMARY : CLASSIC MODEL

Table 12 classic model Results

Velocity	Drag Force (N)	Lift Force (N)	CD	CL
10 km/h	5.148	9.430	0.636	1.166
20 km/h	20.838	14.958	0.646	0.464
30 km/h	45.923	26.401	0.632	0.363
40 km/h	83.515	39.013	0.646	0.302
50 km/h	130.963	58.660	0.649	0.291

(P.S : For Detailed Graphs and pressure,velocity Contour Charts for each Simulation Refer the Results Section, where it is compared with the charts from the revised design)

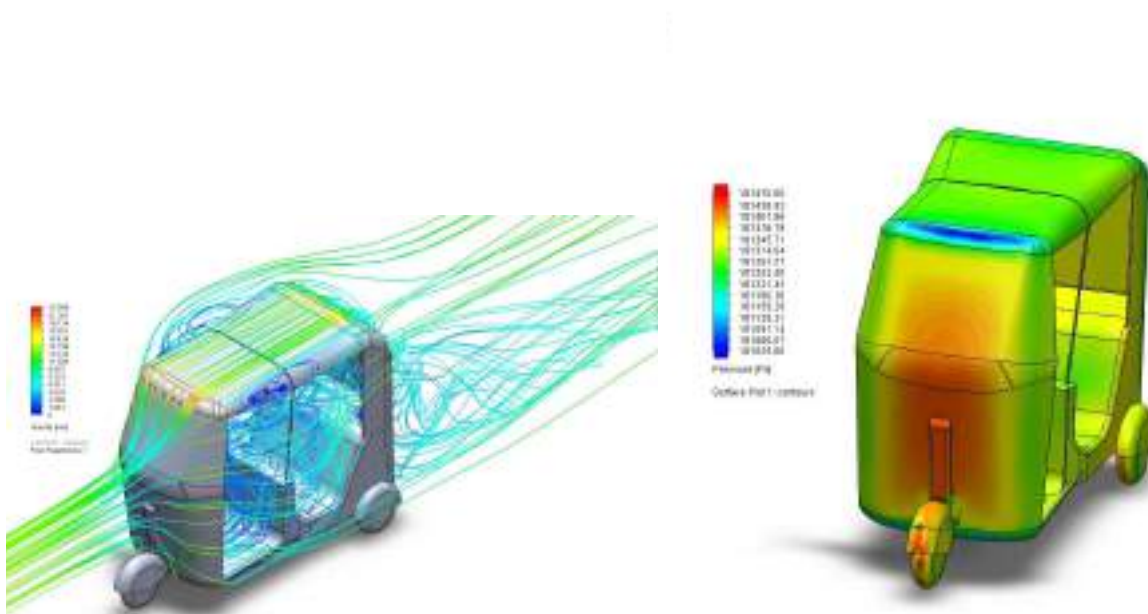
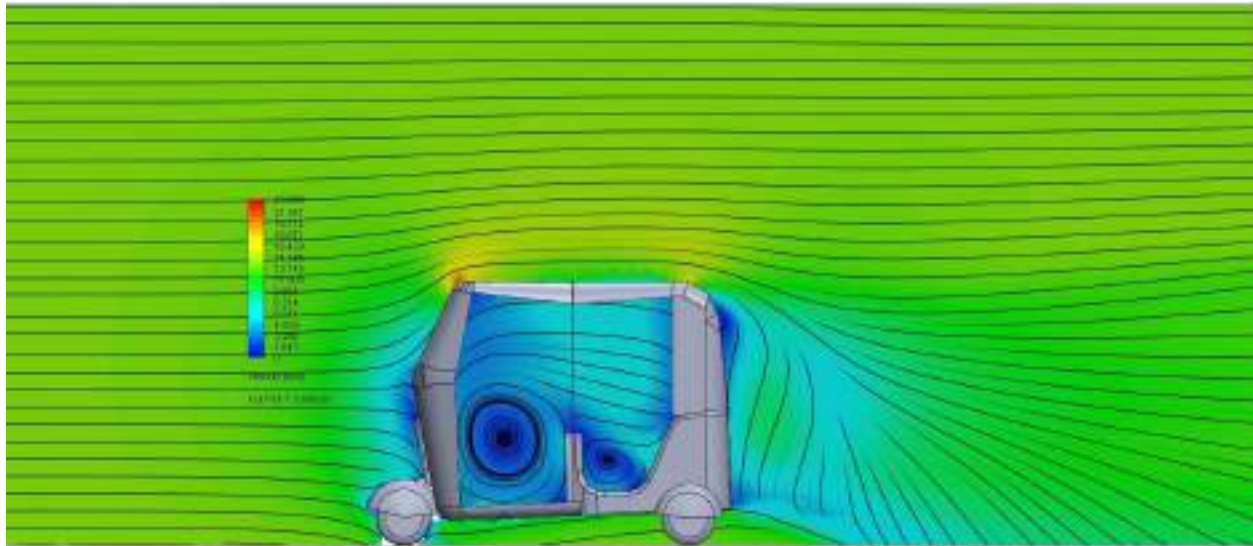


Fig 5.1 Velocity Contour Plot at 50 km/h , Fig 5.2 streamlines at 50 km/h , Fig 5.3 Surface Pressure at 50 km/h (From Top)
The Contour charts for the 50 Km/h simulation are given above.

6. DESIGN REVISION

6.1 ANALYZING THE OLD DESIGN

From the simulation data obtained for the classic design of the auto-rickshaw, we can analyze the design pitfalls and bottlenecks that are present in the old design.

From the velocity contour and, streamlines chart:

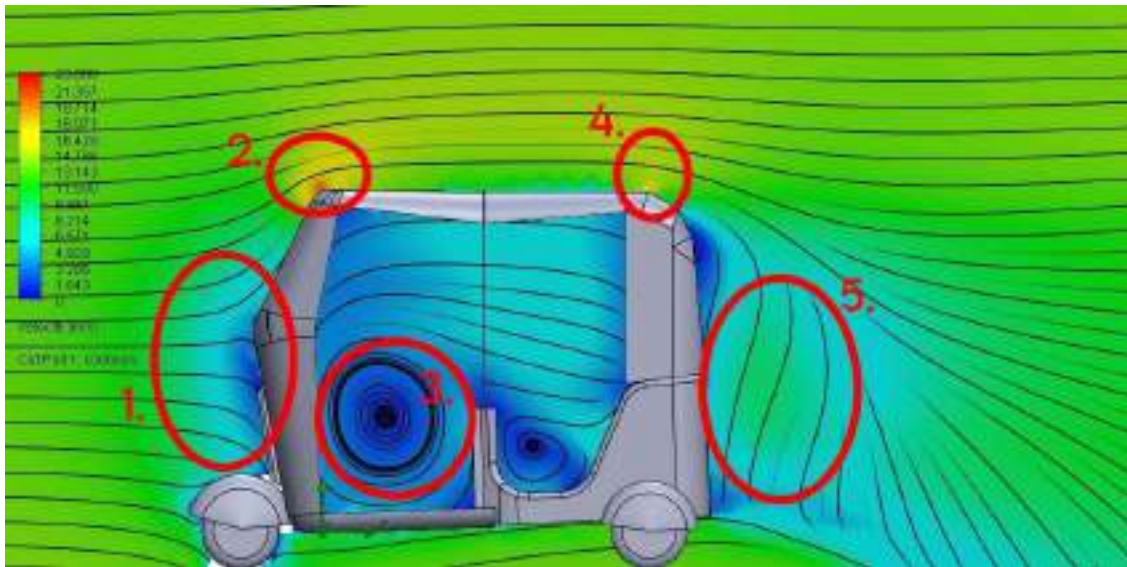


Fig 6.1 Analysis of Velocity Contour Plot of Classic Design

1. From this velocity contour chart we can observe that the angle of the wind shield and the frontal sheet metal is too steep and nearly perpendicular when compared to the angle of attack of the incoming wind which increases the drag force considerably.
2. We can also see that a hotspot has formed where the front and the roof of the auto meet, this is due to the sharp angle between the roof and front , which causes the flow to separate from the surface which can cause instabilities and increase the lift force, reducing the traction of the vehicle.

3. It is clear that the side openings are also causing flow separation, from the formation of strong eddy current inside the Auto-rickshaw.
4. Another hotspot is also formed where the roof meets the rear of the auto, this is also due to flow separation.
5. Finally we can see a large green area behind the auto-rickshaw , which is a low pressure area formed due to the turbulence caused when the air leaves the auto body, this are optimally should be a long and triangular shaped , as the pressure normalizes gradually as the flow connects together back again in the rear of vehicles. This is clearly not the case here, because the air from the underbody of the Auto-rickshaw is not meeting the air form above the auto correctly(a diffuser can help)

The Surface plot and the Streamlines also prove these points:



Fig 6.2 Streamlines

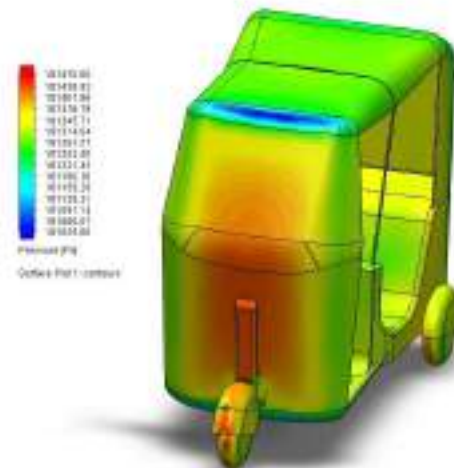


Fig 6.3 Surface Pressure Chart

6.2 REVISED DESIGN

With the analysis of the old design in mind the existing design of the auto-rickshaw was revised. In order to still maintain the original use cases and the practicality of the Auto-rickshaw, these constraints were made to make sure that the modification does not affect any other aspects of the Auto-rickshaw

- The change to the overall envelope should be minimal
- No changes should be done to the ground clearance
- The open sides should not be disturbed
- No changes to the drive-train interfaces (same base and chassis should be used)
- Driver and passenger comfort should be same.(No changes to the interior flooring and seats)
- Ergonomic should remain the same

With these constraints in mind and with the objective of improving the aerodynamic performance of the design the following revised design was modeled in Solidworks:

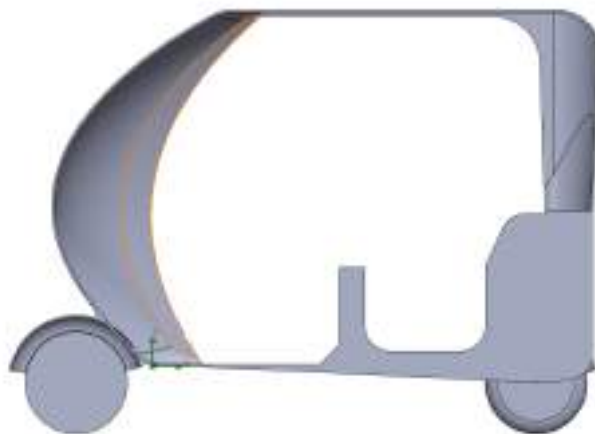


Fig 6.4 Section view of Revised Design

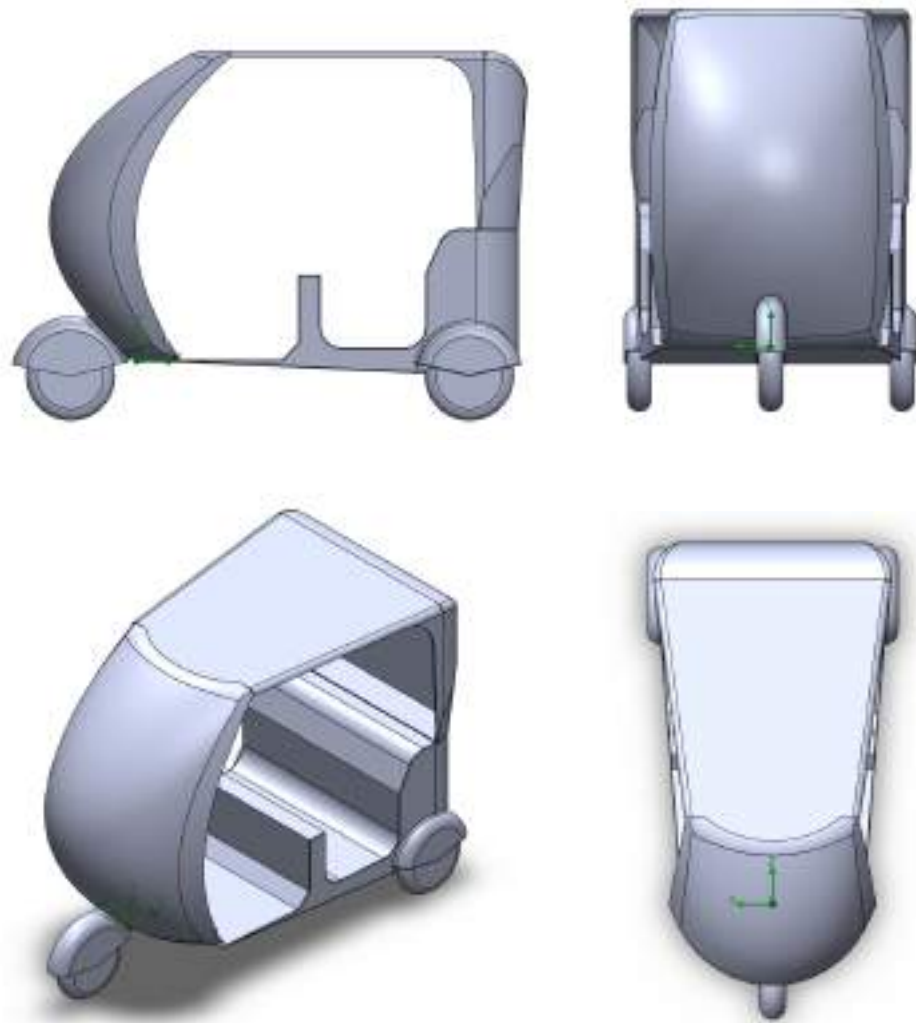


Fig 6.5 Side View, Fig 6.6 Front View, Fig 6.7 Isometric view, Fig 6.8 Top View [Revised design (From top left)]

Notable changes in this revised design:

- The curvature of the windshield and the front panel was increased to reduce the drag force
- The angle between the front and the roof was decreased and made gradual to reduce the flow separation
- The front driver's compartments width was slightly reduced, whereas the width of the passenger area was slightly increased, this was done

so that sides would push air away and prevent it from entering the inside of the auto, where it was seen to cause turbulent eddy currents.

- The underside of the auto is now raked downwards and a small diffuser is added in the back (can be seen clearly in the section view), to reduce the lift force and to manage the low pressure spot that formed in the back of the auto.

Geometry Details:

Table 13 Revised Model Geometry Details

Nodes	3800
Vertices	31421
Polygons	25145

6.3 SIMULATION OF THE REVISED MODEL

6.3.1 DOMAIN: The same domain as discussed earlier for the classic auto design was used here.

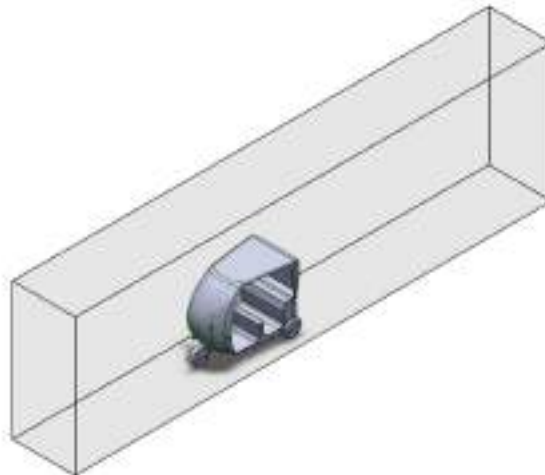


Fig 6.9 Domain for Revised Model

Domain Details:

Table 14 Domain for Revised Model

X min	-0.800 m
X max	0.800 m
Y min	-0.300 m
Y max	3.200 m
Z min	-4.258 m
Z max	7.880 m
X size	1.600 m
Y size	3.500 m
Z size	12.139 m

6.3.2 MESHING: the same mesher as discussed earlier was used to mesh this revised model.

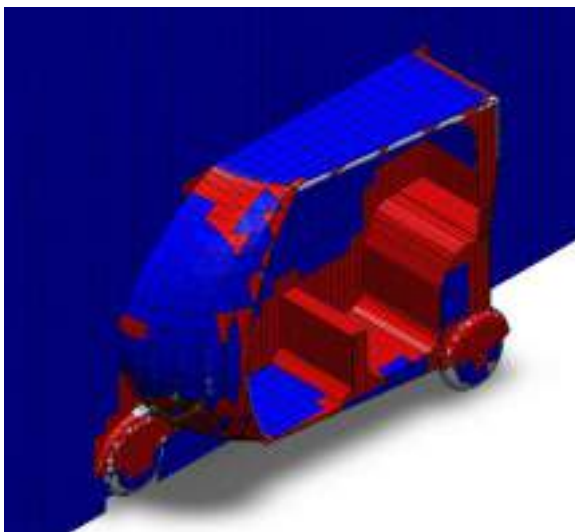


Fig 6.10 Mesh View

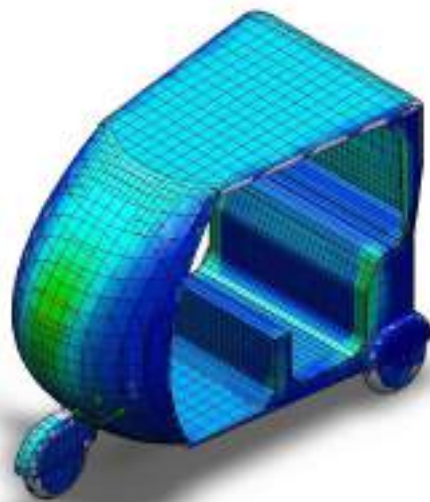


Fig 6.11 Revised Design in Mesh

Basic Mesh Dimensions

Table 15 Revised Model Basic Mesh

Number of cells in X	22
Number of cells in Y	36
Number of cells in Z	102

Solid/Fluid Interface

Table 16 Revised model Solid/Fluid Interface

Small Solid Feature Refinement Level	4
Curvature Level	3
Curvature Criterion	0.318 rad
Tolerance Level	0
Tolerance Criterion	0.327 m

Number Of Cells

Table 17 revised model No. of Cells

Cells	94982
Fluid cells	94982
Irregular cells	0
Trimmed cells	221

6.3.3 FLUID SELECTION: The same fluid of standard air was selected from solidworks database.

6.3.4 FRONTAL AREA : the silhouette of the new model was projected to a plane and the frontal area was measured

Table 18 Frontal area of Revised Model

Frontal Area	1331455.39 mm ²
	1.331 m ²

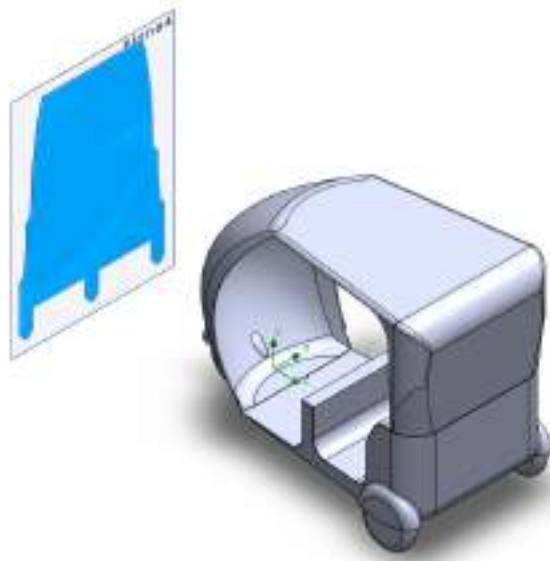


Fig 6.12 Frontal Area of Revised Design

6.3.5 BOUNDARY CONDITIONS: The same boundary conditions were used for this simulation also. The simulation was done five times for the velocities of 10,20,30,40,50 km/h

6.3.6 SOLVER SETTINGS: the same EXTERNAL FLOW solver , with identical settings was used for the simulation (refer previous sections for full solver settings)

6.4 RESULT SUMMARY : REVISED DESIGN

The simulation was done far 5 cases as before 10,20,30,40,50 km/hr and the results obtained for the revised model are

Table 19 Revised model results.

Velocity	Drag Force (N)	Lift Force (N)	CD	CL
10 km/h	2.687	6.066	0.426	0.962
20 km/h	10.896	9.074	0.433	0.361
30 km/h	24.745	14.164	0.437	0.250
40 km/h	44.208	21.462	0.439	0.213
50 km/h	69.877	31.405	0.444	0.199

6.5 ANALYSIS OF THE REVISED DESIGN RESULTS

From the table above we can easily see that the lift and drag performance of the new design is far superior to the older design, with improvement in Drag, Lift performance and also a considerable reduction in the lift and drag coefficients as well. The velocity contour chart of the revised design is given below:

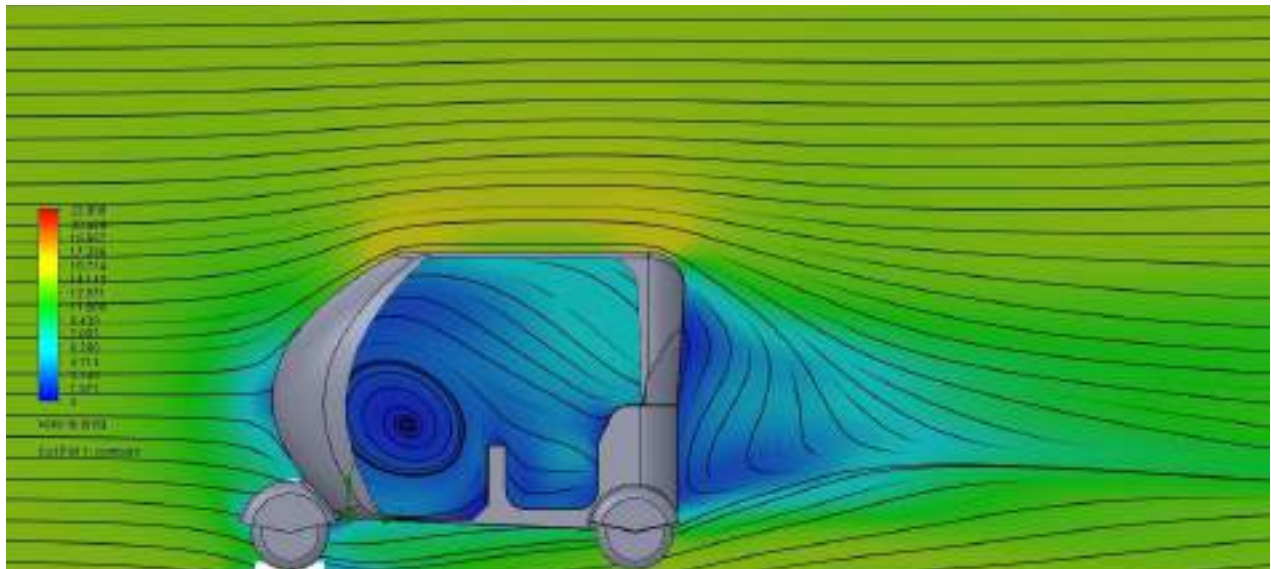


Fig 6.13 Velocity Contour of Revised Model at 50 km/h

From this chart we can clearly see that, all the issues that were discussed about the old design are addressed and improved upon:

- The angle between the windshield and the air is much milder and gradual
- The angle between the front and the roof is also reduced which considerably reduces the hotspot that formed there
- The turbulent eddies inside the auto are much milder
- The addition of the diffuser has completely removed the low pressure spot that was forming at the back of the Auto

The Surface pressure chart and the streamlines chart also proves the same:

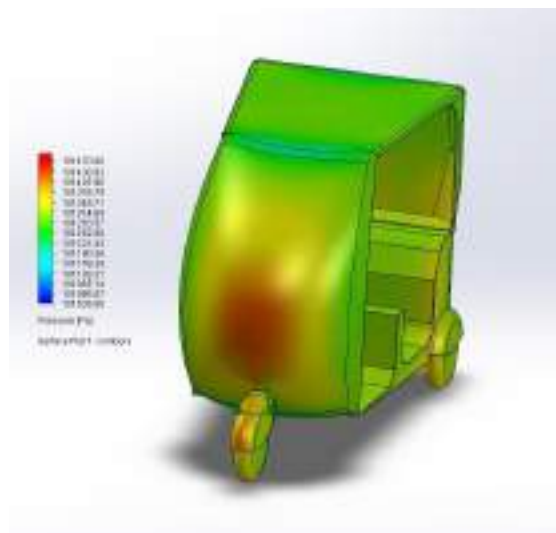
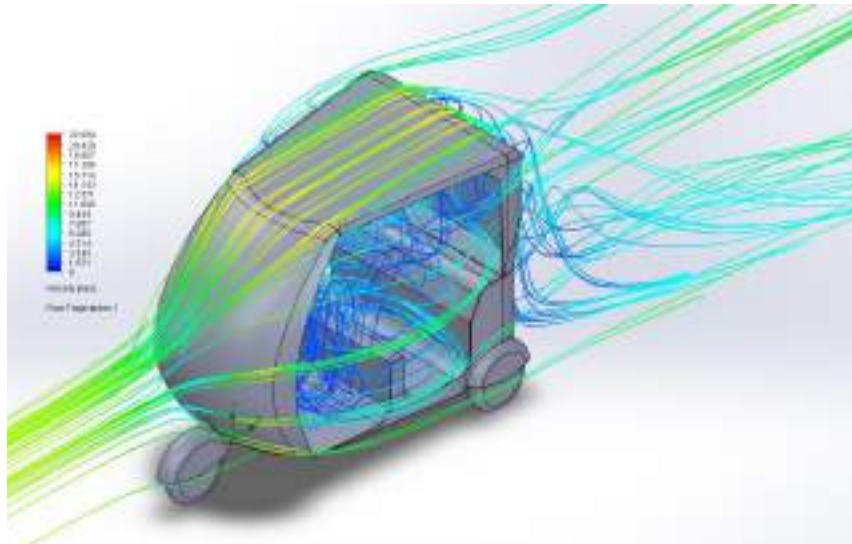


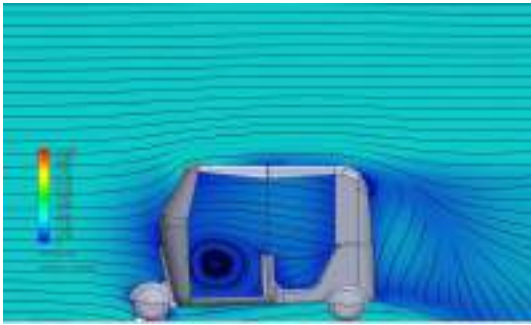
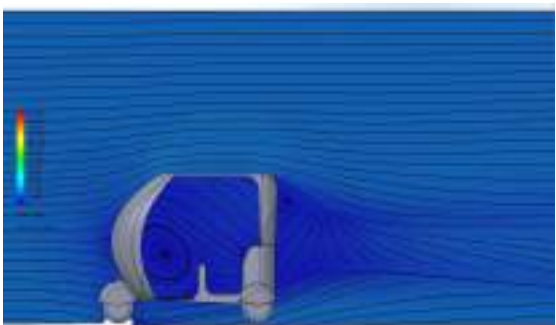
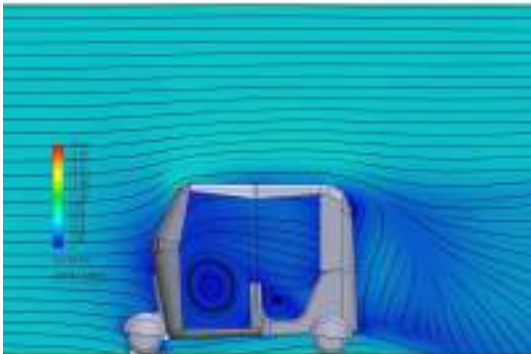
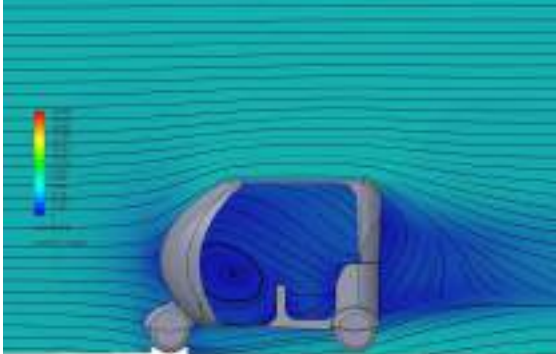
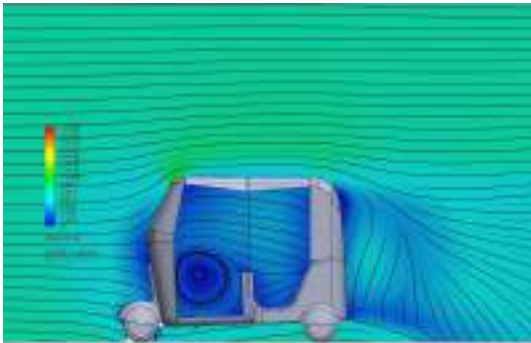
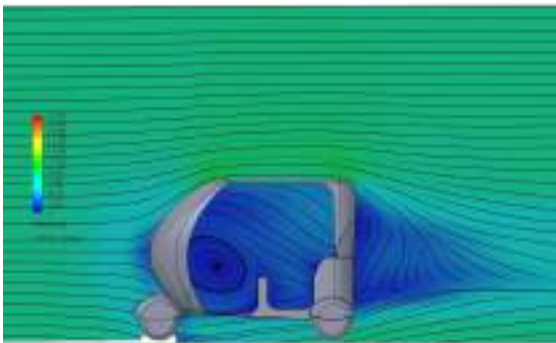
Fig 6.14 Revised Model Streamlines

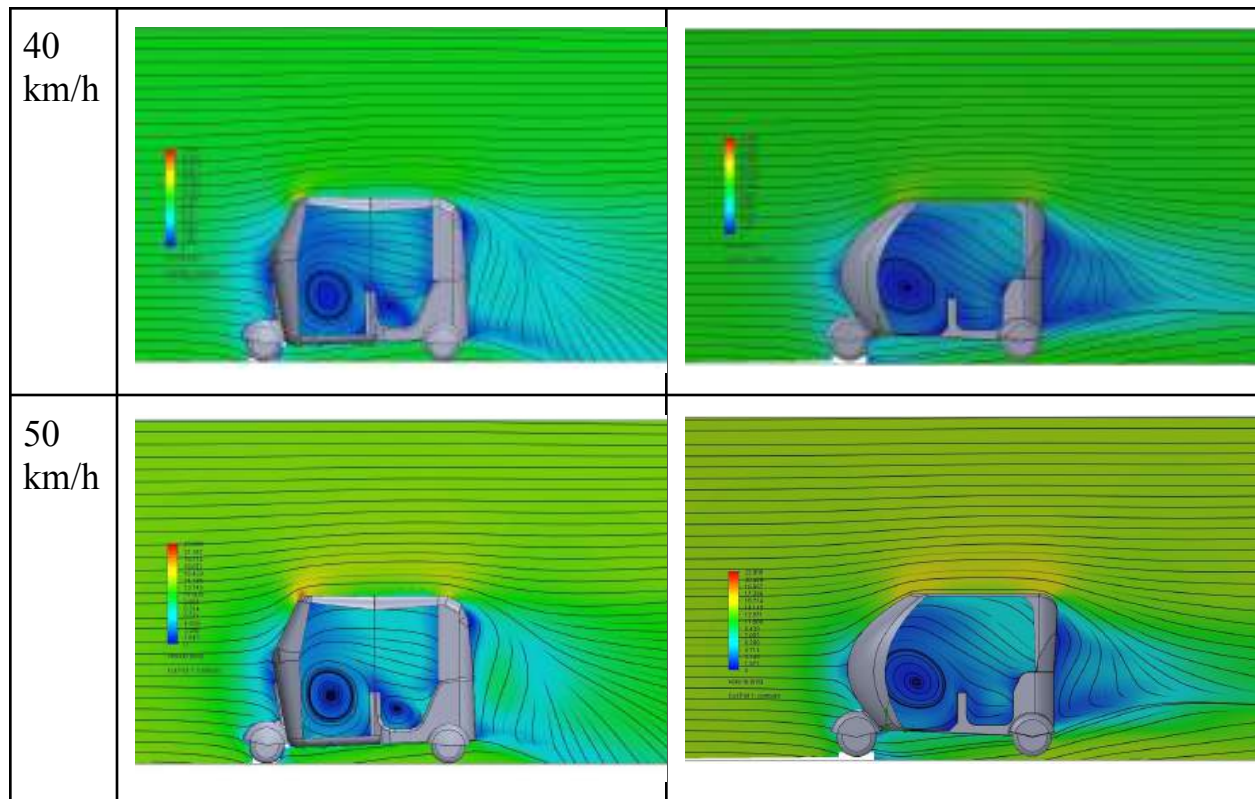
Fig 6.15 Revised Model Surface Pressure

7. COMPARISON CHARTS AND RESULTS

7.1 COMPARISON OF THE VELOCITY CONTOUR CHARTS

Table 20 Velocity Contour Comparison

	CLASSIC DESIGN	REVISED DESIGN
10 km/h		
20 km/h		
30 km/h		

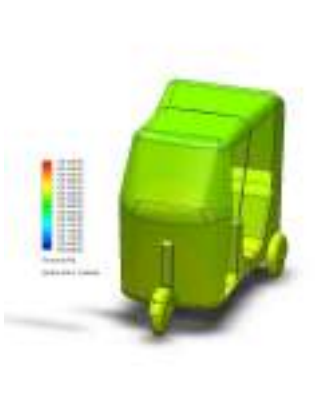
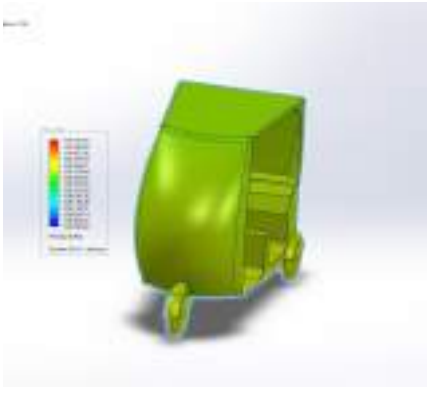
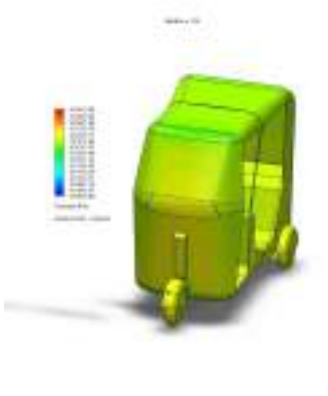
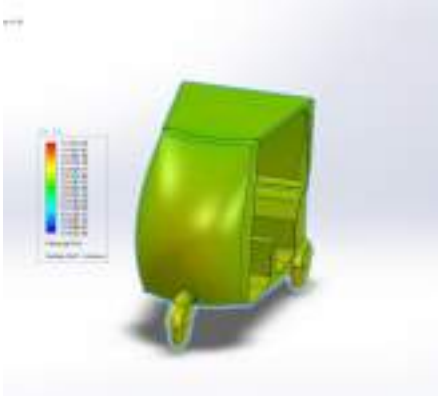
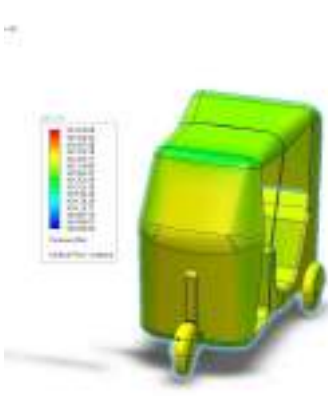
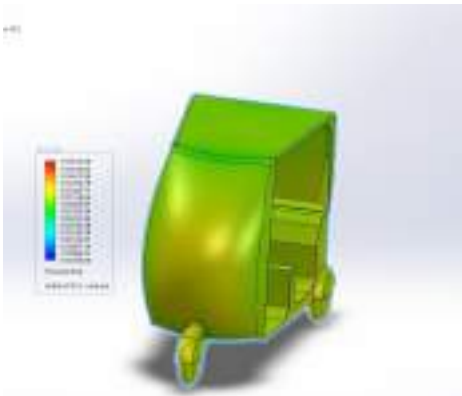


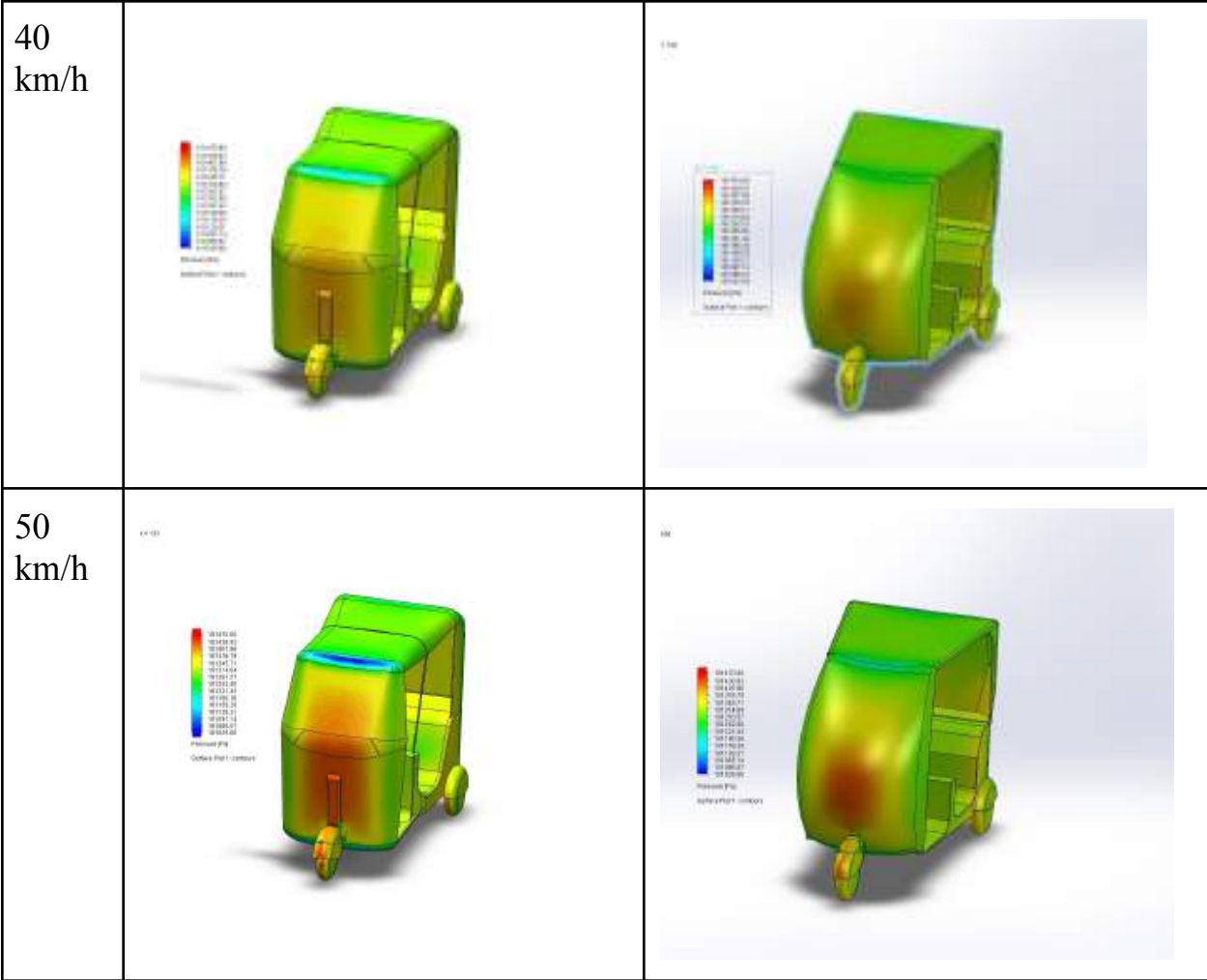
From these velocity contour charts side by side we can clearly see the design changes having effect on the aerodynamic performance. The velocity contour is one of the most useful visualization techniques to graphically view the aerodynamic parameters of a vehicle.

The surface pressure charts and streamlines also verify that the design changes made were in fact advantageous for the Auto-rickshaw's efficiency

7.2 COMPARISON OF SURFACE PRESSURE CHARTS

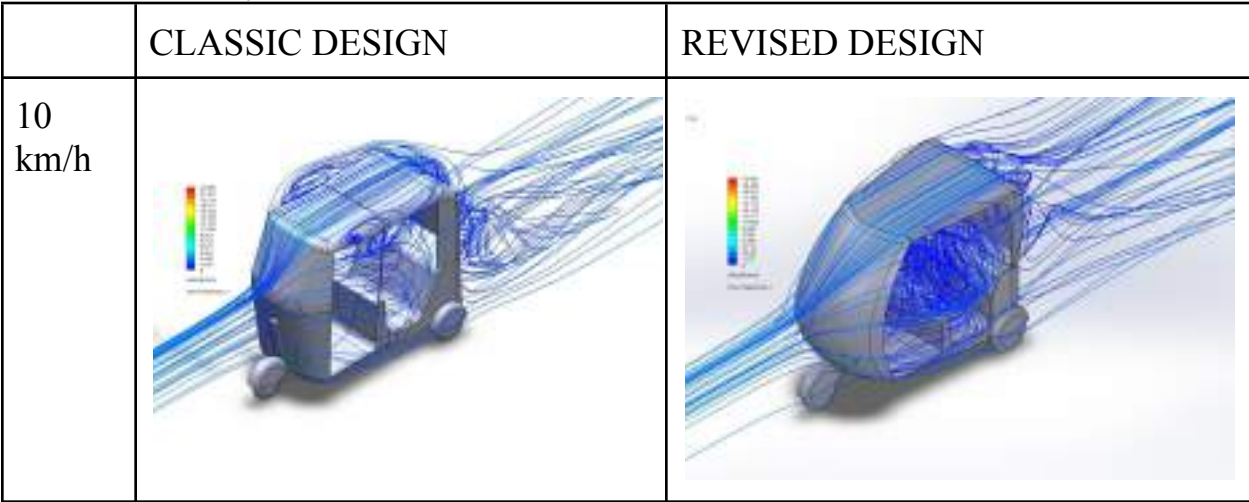
Table 21 surface Pressure Comparison

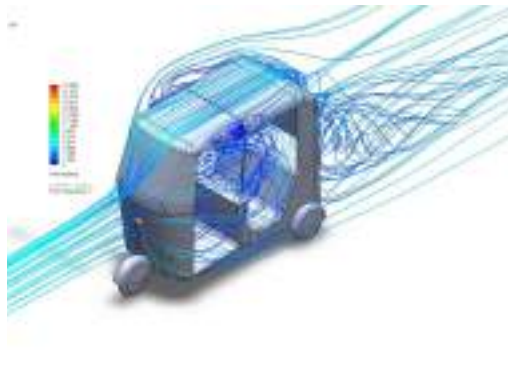
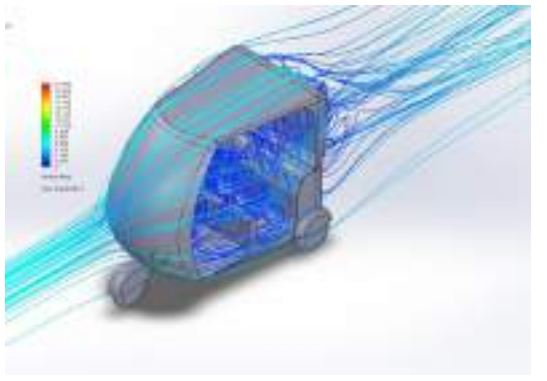
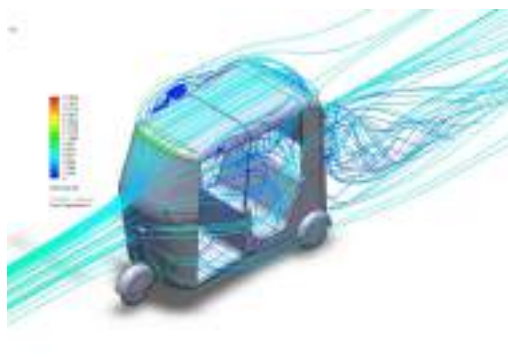
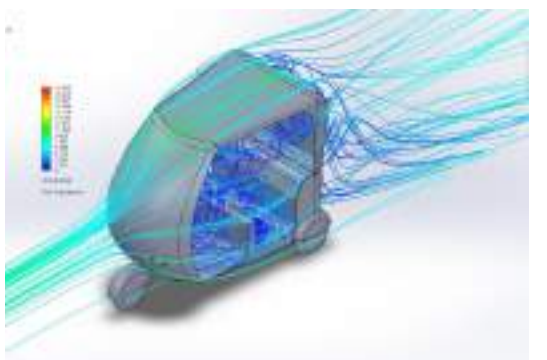
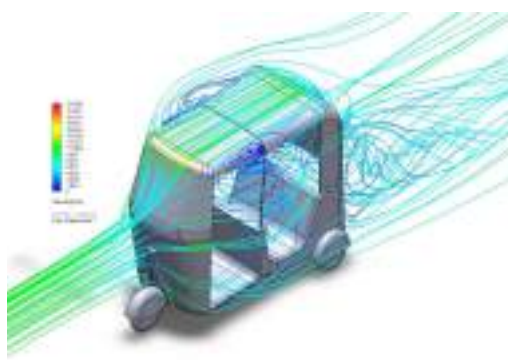
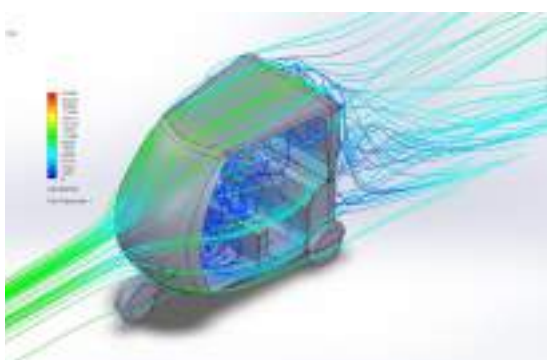
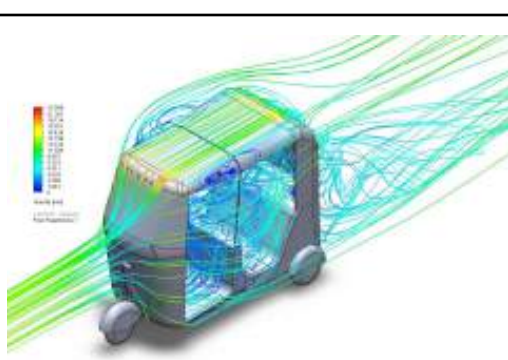
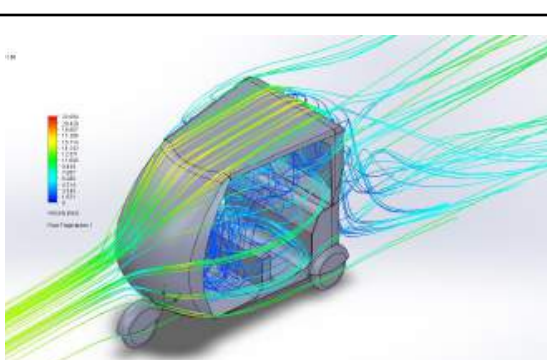
	CLASSIC DESIGN	REVISED DESIGN
10 km/h		
20 km/h		
30 km/h		



7.3 COMPARISON OF STREAMLINE CHARTS

Table 22 streamlines Comparison



20 km/h		
30 km/h		
40 km/h		
50 km/h		

7.4 COMPARISON OF DRAG AND DRAG-COEFFICIENT

Comparing the drag and drag coefficient obtained for the classic design against the revised design.

DRAG FORCE COMPARISON:

Table 23 Drag Force Comparison

Velocity (km/h)	DRAG FORCE [N]	
	Classic design	Revised design
10	5.148	2.687
20	20.838	10.896
30	45.923	24.745
40	83.515	44.208
50	130.963	69.877
MEAN	57.27	30.48

DRAG COEFFICIENT COMPARISON:

Table 24 Drag Coefficient comparison

Velocity (km/h)	CD	
	Classic design	Revised design
10	0.636	0.426
20	0.646	0.433
30	0.632	0.437
40	0.646	0.439
50	0.649	0.444
MEAN	0.6418	0.4358

DRAG FORCE

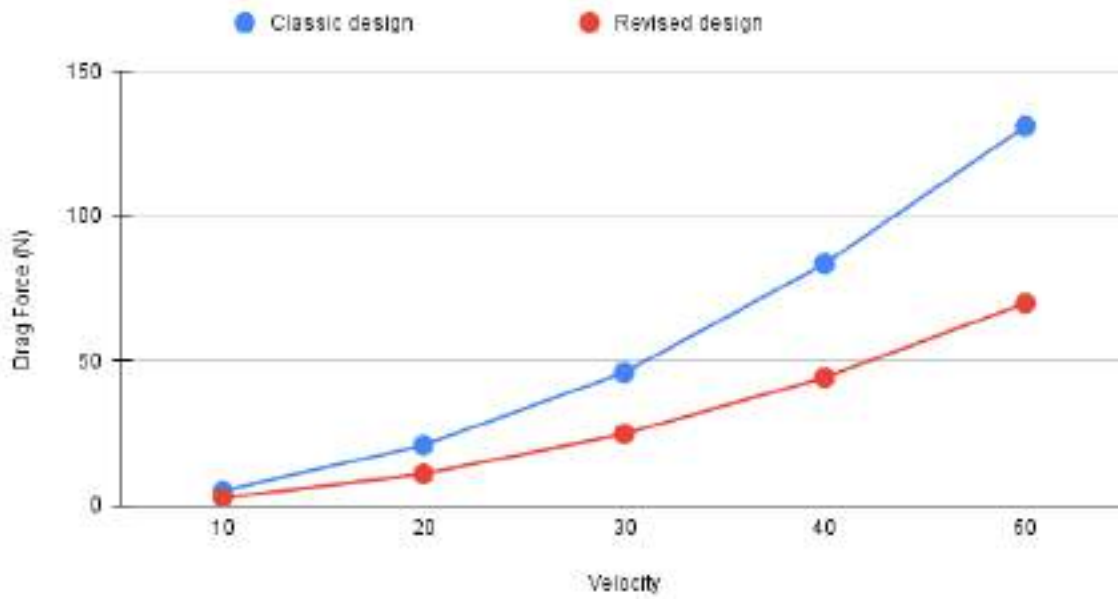


Fig 7.1 Drag Force Comparison

DRAG COEFFICIENT (CD)

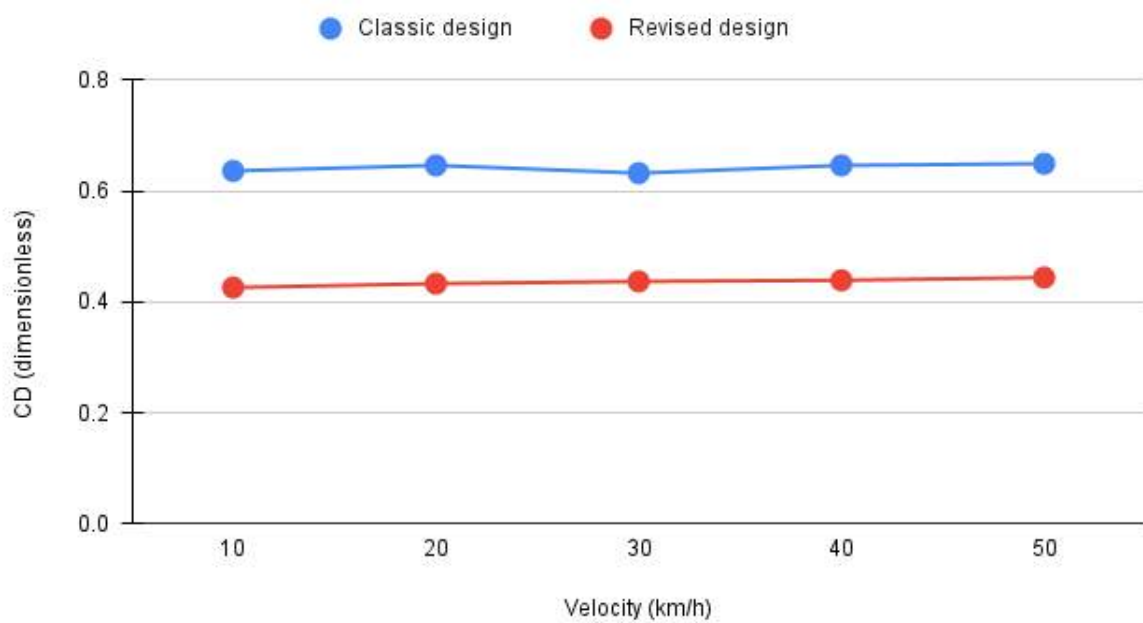


Fig 7.2 Drag Coefficient Comparison

As we can infer from the tables and charts given above the average drag force of the revised design has dropped by **26.79 Newtons** , which is a **46.77 %** improvement over the classic design of the auto-rickshaw.

Similarly the drag coefficient has also dropped by **0.206**, which corresponds to a **32.09 %** improvement over the old design.

7.5 COMPARISON OF LIFT AND LIFT-COEFFICIENT

Similarly comparing the lift and lift-coefficient of the classic design against the revised design:

LIFT FORCE COMPARISON:

Table 25 lift Force Comparison

Velocity (km/h)	LIFT FORCE [N]	
	Classic Design	Revised Design
10	9.430	6.066
20	14.958	9.074
30	26.401	14.164
40	39.013	21.462
50	58.660	31.405
MEAN	29.6924	16.4342

As we can infer from this table the average lift force has been reduced by **13.2582 newtons** , which corresponds to a **44.65%** improvement over the classic Auto-Rickshaw design.

LIFT FORCE

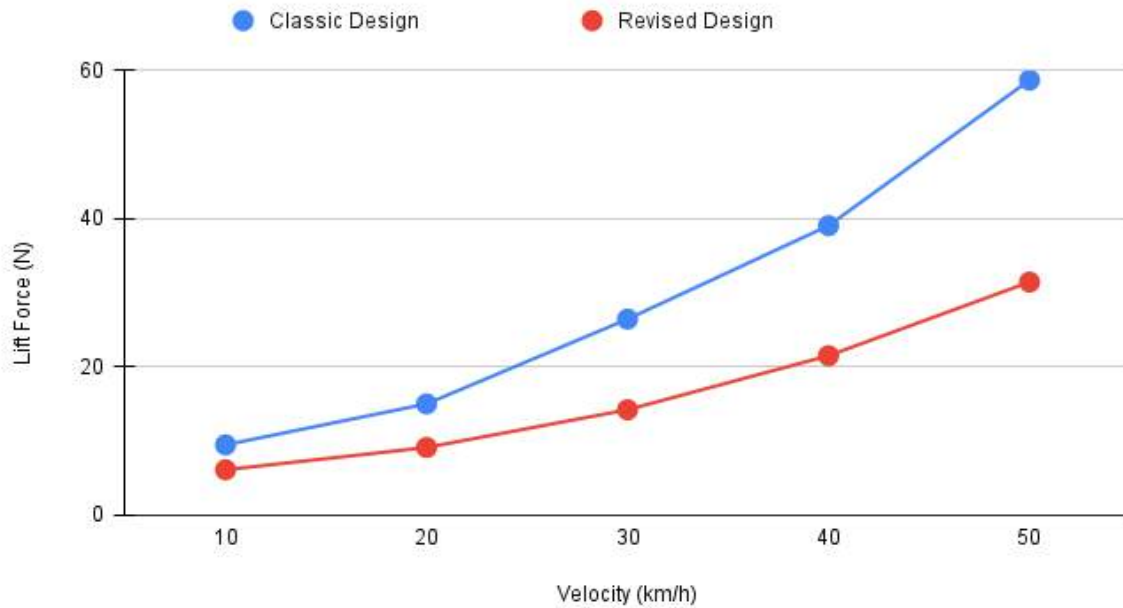


Fig 7.3 Lift force Comparison

LIFT-COEFFICIENT COMPARISON:

Table 26 Lift coefficient Comparison

Velocity (km/h)	CL	
	Classic Design	Revised Design
10	1.166	0.962
20	0.464	0.361
30	0.363	0.250
40	0.302	0.213
50	0.291	0.199
MEAN	0.5172	0.397

Similarly , we can observe a drop in the average lift coefficient by **0.1202** , which is a **23.24%** improvement over the Classic Auto Design.

LIFT-COEFFICIENT (CL)

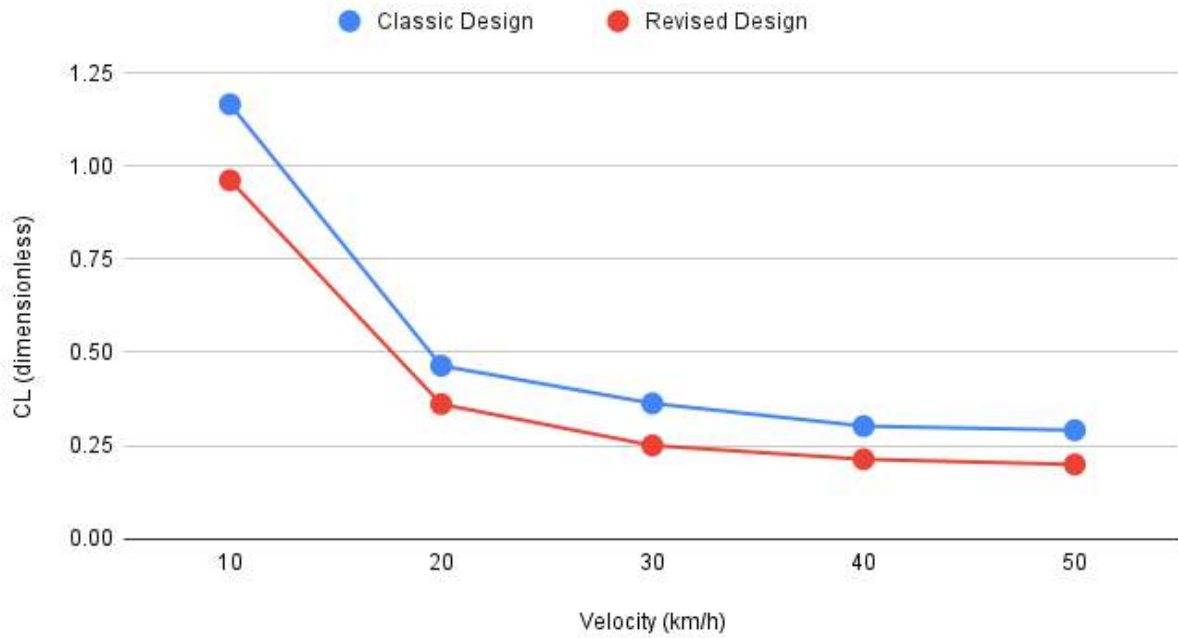


Fig 7.4 Lift Coefficient Comparison

7.6 FINAL RESULTS

Comparing the mean drag, lift, C_d and C_l we can clearly see that there is an improvement in the aerodynamic characteristics of the vehicle.

Table 27 Final Results.

	Mean Drag	Mean Lift	Mean C_d	Mean C_l
Classic design	57.27	29.6924	0.6418	0.5172
Revised Design	30.48	16.4342	0.4358	0.397
% improvement	46.77	44.65	32.09	23.24

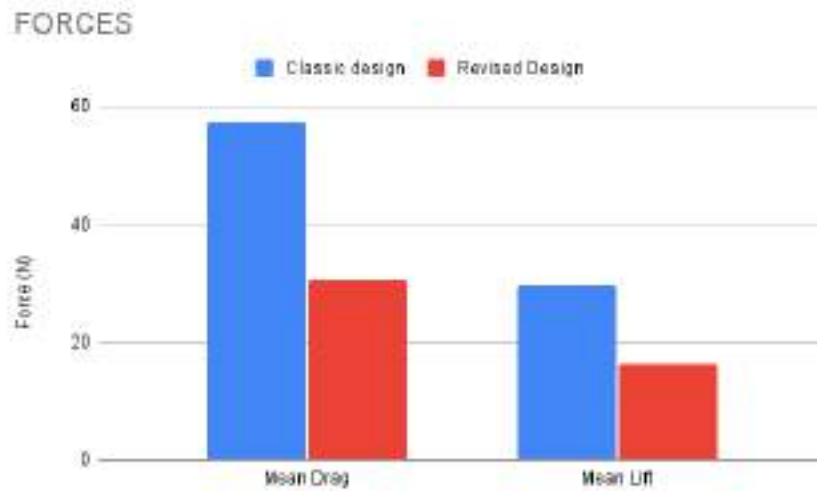


Fig 7.5 Mean Forces Comparison

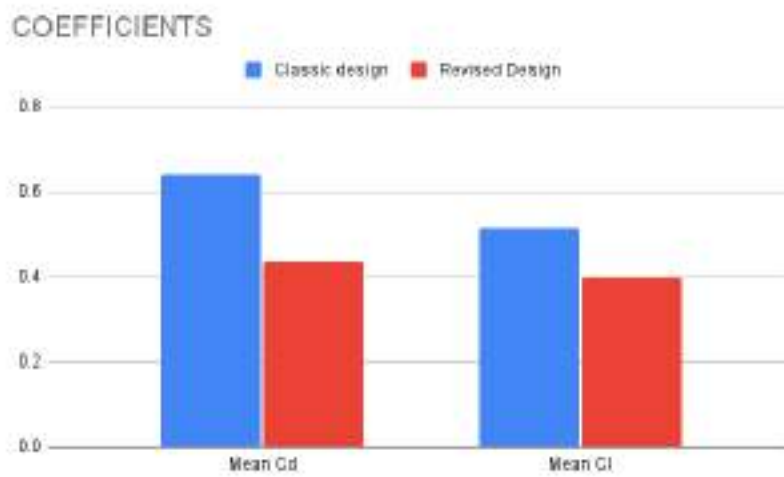


Fig 7.6 Mean Coefficients Comparison

Therefore the results can be summarized as:

- 46.77% decrease in average drag force
- 32.09% decrease in average drag-coefficient
- 44.65% decrease in average lift-force
- 23.24% decrease in average lift-coefficient

8. CONCLUSION

Flow simulation over the classic model of an Auto-rickshaw showed the necessary areas for improvement for achieving better aerodynamic performance. The flow simulation for the classic Auto-Rickshaw was done at the normal operating speeds of an Auto , namely 10 , 20 , 30 , 40 , 50 km/h .The results obtained were carefully recorded and organized. Various visualization methods such a velocity contours, Surface Pressure plots and streamline charts were generated for the easy representation of the data obtained. These various charts were studied and analyzed in detail , and the salient features of the auto which were a bottleneck to the aerodynamic performance were identified. Then these bottlenecks were taken into account while designing a revised design. It was made sure in the revision process that the design changes made would not affect the other advantageous aspects of the Auto-Rickshaw , in order to preserve the ‘Soul’ of the Auto-Rickshaw, which includes open sides for ease of access , flexible spaces for both the driver and passengers , good ergonomics, robust drive train and handling for rough urban conditions and razor point maneuvering for which the Auto is known for. The new revised design was made with these points in mind. This new revised design was also subjected to the same flow simulation as the classic design at 10 , 20 , 30 , 40 , 50 km/h. The data was collected and recorded .

Comparing the data obtained in both cases, it was obvious that the new revised design had in fact addressed all the issues that were present in the original design. The velocity contours , surface pressure charts and streamlines were compared side by side and studied closely to quantify the improvements over the classic design.

In probing, testing and rigorously comparing the data , it was found that the new revised design provided these main quantifiable advantages in aerodynamic performance over the classic design:

- 46.77% decrease in average drag force
- 32.09% decrease in average drag-coefficient
- 44.65% decrease in average lift-force
- 23.24% decrease in average lift-coefficient

Therefore it is clearly evident that the design changes made had an overwhelmingly positive impact on the aerodynamic performance of the Auto-Rickshaw. Thus the data obtained in this study shows that implementing a few of the design suggestions put forth can be really instrumental in improving the efficiency of a vehicle that is so commonplace and popular. This would also be a marked improvement when Auto-Rickshaws are to be electrified.

APPENDIX 1

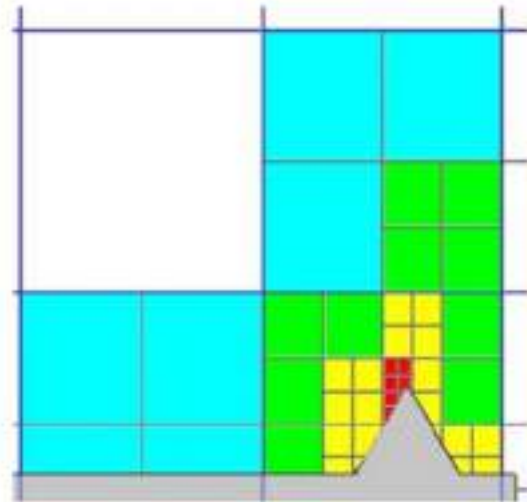
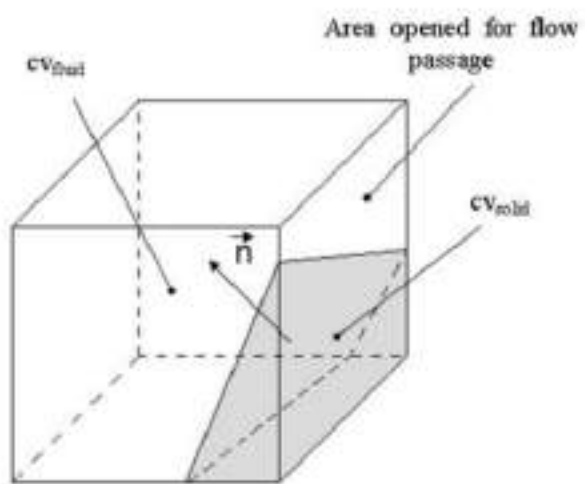
CARTESIAN MESHER USED IN SOLIDWORKS FLOW SIMULATION

Historically, for traditional CFD codes, the fluid space is created by Boolean subtraction of the solid model within the CAD system, and this inverse solid is passed to the CFD tool for meshing. Mesh generators in traditional CFD are usually based on body-fitted algorithms.

But instead SOLIDWORKS FLOW SIMULATION uses an immersed-body mesh. In this approach the creation of the mesh starts independently from geometry itself and the cells can arbitrarily intersect the boundary between solid and fluid. This makes it possible to use a Cartesian-based mesh, which in the general case cannot be body-fitted. Such a mesh can be defined as a set of cuboids (rectangular cells), which are adjacent to each other and to the external boundary of the computational domain, orientated along the Cartesian coordinates. Cuboids intersected by the surface are treated in a special way, described later, according to the boundary conditions defined on the surface. It is necessary to point out that the immersed body mesh approach can be implemented for tetrahedral and other types of the elements, but in terms of approximation accuracy and ease of implementation, Cartesian meshes are strongly preferred.

Advantages of Cartesian meshes can be summarized as follows:

- Simplicity, speed and robustness of the mesh generation algorithm especially when dealing with native CAD data;
- Minimization of Local Truncation Errors.



REFERENCES

1. Akshay Parab, Ammar Sakarwala, Bhushan Paste, Vaibhav Patil, Amol Mangrulkar (2014) 'Aerodynamic Analysis of a Car Model using Fluent-Ansys 14.5' - IJRMEE Vol. 1 No. 4 (2014): November (2014) Issue
2. Boran Pikula , Elmedin Mešić, Mirzet Hodžić (2008) 'Determination of Air drag coefficient of vehicle models ' - International Congress Motor Vehicles & Motors 2008 - MVM 2008, „Sustainable Development of Automotive Industry
3. Boran Pikula , Elmedin Mešić, Mirzet Hodžić (2008) 'Determination of Air drag coefficient of vehicle models' International Congress Motor Vehicles & Motors 2008 - MVM 2008
4. C. Bhaskar ,Krishna Rawat & Muhammed Minhaj (2020) 'Aerodynamic Study of a Three Wheeler Body' - Springer Science and Business Media
5. Deutschland GmbH, Journal of Advances in Automotive Technologies (pp.225-230)
6. Jay C. Kessler and Stanley B. Wallis (1967) 'Aerodynamic Test Techniques', SAE Transactions, Vol. 75, SECTION 3: Papers 660463–660786 (1967), pp. 12-27

7. Joseph Katz (2016) 'An Introduction to Automotive Aerodynamics' - Wiley Publications ,ISBN:9781119185727
8. Loya, A. , Iqbal, A. , Nasir, M. , Ali, H. , Khan, M. and Imran, M. (2019) 'Automotive Aerodynamics Analysis Using Two Commonly Used Commercial Software. Engineering' - Scripp Engineering Journal .Vol. 11, pp22-32
9. Pasquale Sforza (2014) 'Commercial Airplane Design Principles' - Elsevier Science Publications, ISBN:9780124199774
10. Shubhankar Pal , S.M. Humayun Kabir , Md. Mehdi Masud Talukder (2015) 'Aerodynamic analysis of a concept model car', 3rd International Conference on Mechanical Engineering and Renewable Energy 2015 (ICMERE2015)
11. Sobachkin .A,G. Dumnov (2014) 'Numerical Basis of CAD-Embedded CFD' - Dassault systems support archives, solidworks support.