iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 Issue: VII Month of publication: July 2021

DOI: https://doi.org/10.22214/ijraset.2021.36879

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

Design and Analysis of Foreward Step Automotive

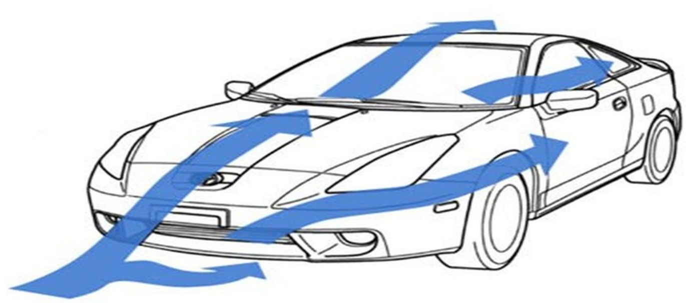
Veenam Varaprasad[1], Bandi Venkat Sriram[2], Ande Sai Mahesh[3], Meka Subramanyam[4], Vullinga Santhosh[5]
[1]*Assistant Professor, Dept. of Mechanical Engineering, DMS SVH College of engineering, Machilipatnam, India*
[2, 3, 4, 5]*B. tech student, Dept. of Mechanical Engineering, DMS SVH College of engineering, Machilipatnam, India*

Abstract: Nowadays with increase in competition in automobile sector, vehicle aerodynamics plays an important role. Aerodynamics affect the performance of vehicle due to change in parameters such as lift and drag force which plays a significant role at high speeds. With improvement in computer technology, manufacturers are looking toward computational fluid dynamics instead of wind tunnel testing to reduce the testing time and keeps the cost of R&D low. In this paper, lift and drag of production vehicle are determined by the analysis of flow of air around it using Ansys 18.0. After that, analysis was done on the car with different engine hood angles. Based on Cl and Cd values, optimal model was selected. To validate steady state results, transient state analysis was done on this optimal model. By introducing this considerably reduce the drag and increase lift hence improves the performance of vehicle.
Keywords: Aerodynamics, Fluid Dynamics, Lift and Drag, Engine hood angle, Transient State.

I. INTRODUCTION

Aerodynamic drag or the wind resistance is considered to be of prime concern in vehicle design. The other important issues being the vehicle weight & fuel efficiency. The twentieth century in particular has witnessed refinement in vehicle design in terms of issues mentioned above. However the focus in this project is on drag & related issues. It is worth noting that reduction of one count of drag i.e. $\Delta CD = 0.1$ translates into an improvement of mileage of 2.60 km/ lit. As per DOT figures India has approximately 13 million cars plying on the roads as recorded during 2008. Further few hundreds are added every day. It is also worth noting from the statistics given by DOE, that 16 % of the energy produced in USA is consumed in overcoming drag of road vehicles .The Indian scenario is not much different. Therefore there is enormous scope for improving aerodynamics of cars inorder to conserve depleting oil reserves.

Aerodynamic Design of cars is crucial as it directly affects the fuel economy &stability in motion .Therefore it is necessary to have a clear understanding of external aerodynamics of road vehicles which are nearer to bluff bodies and moreover the flow over them is complex owing to its nonlinear and stochastic nature (Partly attached & partly separated). Also wind resistance is amongst the three significant factors (Efficiency of engine , weight &aerodynamic drag),which influence the fuel efficiency .Therefore needs a proper focus in the early stage of design. At a vehicle speed of 70 kmph & above the prominence of air/wind is the major factor affecting fuel economy / stability. Six degrees of freedom are assumed for a roadvehicle, for investigating the influence of various aerodynamic forces & moments.These forces and moments are namely drag, lift, crosswind force & rolling moment, pitching moment, yawing moment respectively. Amongst these “drag” is found to be the most prominent factor affecting the fuel consumption at higherspeeds although other factors also need to be tackled in an integrated manner. It is worth noting from the statistics given by DOE, that 16 % of the energy consumed in USA in overcoming drag of road vehicles. This fact emphasizes the need and scope for reduction of aerodynamic drag of road vehicles. As per the data available from DOE (Dept. ofEnergy) at present US consumes 35 % more energy than it produces and it is anticipated that by 2020 the energy imbalance may reach 65 %. A more dramatic trend is noted for the transportation sector.



Aerodynamics of Car

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

II. LITERATURE REVIEW

A. Source

The literature for the topic is found from SAE transactions, Science direct, Elsavier journal & e-books on the topic of Aerodynamics, Computational fluid dynamics, Turbulence modeling, Fluid dynamics, Numerical methods & Measurements. Wind force is considered as one of the dominating forces of nature which has influence on road vehicle aerodynamics & therefore on the vehicle stability & fuel economy. Also wind resistance is amongst the three significant factors (Efficiency of engine, weight & aerodynamic drag) which influence the fuel efficiency. Therefore needs a proper focus in the early stage of design. CAE/CFD serves the purpose of analyzing the design variants using virtual prototypes & virtual flow bench thus reducing the development cycle time (simplified car body /Generic shape) is analysed which is considered as generic shape of a car, as adaptive designs which can counteract or nullify the aerodynamic forces are not practically possible. The mesh of the car surface which is a set of Bezier points and a connectivity matrix is completely determined through a deterministic spline algorithm. Therefore to change the frontal area or the shape of the car the Bezier points need to be displaced in a prescribed way to alter the geometry tools like simulation based optimization modules. Neverthless it is impossible to design adaptive car shape/surface mechanism to minimise drag in the transient wind conditions. However it is possible to analyse the flow over generic shapes of car and evaluate aerodynamic properties. From first principle, streamlined bodies will have minimum drag but however due to practical difficulties of asthetics, operational safety, service accessibility &design of the layout constraints the shape of an automobile has evolved over the years. It is necessary to have a clear understanding of external aerodynamics of road vehicles which are nearer to bluff bodies as the flow over them is complex owing to its nonlinear and stochastic nature (Partly attached & partly separated). Assuming six degrees of freedom for a road vehicle various investigations have been carried out to analyse three forces & three moments namely drag, lift, crosswind force & rolling moment, pitching moment, yawing moment respectively. Amongst these “drag” is found to be the most prominent factor affecting the fuel consumption at higher speeds although other factors also need to be tackled in an integrated manner. Ground vehicles fall in the category of bluff bodies which move in close proximity of the road surface. Also with the introduction of more refined & powerful engine used for cars it has become imperative that aerodynamic parameters like drag/lift be optimized so that the engine power is properly utilized. All transportation systems involve either the movement of solid structures through fluids or movement of fluids past solid structures.

B. The Synergy

The research work done mainly consists of CAD modeling from the image of the car with Pro-e & CFD simulation of these models using Fluent software. The wind tunnel experiment for measuring pressure coefficient distribution is also carried out. The experiment on pressure coefficient distribution was chosen as it gives the distribution of pressure coefficient around the vehicle body .Therefore the regionsof high & low pressures are known & therefore the stability of the vehicle. CFD exercise (Numerical Experiments) gives an insight into drag values, pressure contours and velocity vectors for analyzing flow over surface & also nature of wake. The importance of aerodynamic design has been discussed earlier. The conventional tool used for determining aerodynamic properties is wind tunnel. However the measurement procedures and hardware testing involved are cumbersome. Therefore wind tunnels are used nowadays for validating the CFD codes developed. CFD has been complementing wind tunnel testing these days. In the present case the basic CAD model was made from the image of a car and pressure coefficient distribution measurement along the symmetry plane & an offset plane of a wooden 1:10 scale model, was carried out in the low-speed automotive wind tunnel. The measurement of aerodynamic characteristics like drag, lift, down force yawing moment etc using scale models in wind tunnels & extrapolating these measurements to the full scale conditions have become a subject of debate, as for accurate results corrections for boundary layer control , pressure gradients blockages & real turbulence are required. Therefore in the present work it is aimed to have a synergy of wind tunnel pressure distribution & CFD simulation which will yield more meaningful & accurate results rather than validation of CFD results with wind tunnel measurements. The primary goal of this work is to carryout static pressure measurement on the 1:10 scale wooden model of a car in a wind tunnel , compute pressure distribution coefficient & simulate the similar flow conditions/regime using Fluent finite volume code (3D, Pressure based RANS solver) with K SST, turbulence model. Secondly, the CFD analysis results are also compared with

literature values available with regards to generic shapes. The pressure distribution, Velocity vectors, pressure coefficients & wake structures are also ascertained from generic shapes. From first principle , if the flow over the body is controlled & made streamlined the aerodynamic resistance or the drag is reduced.

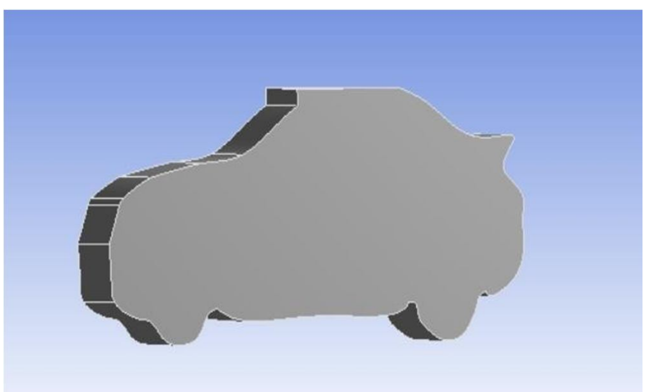
C. Space Claim

Space-Claim is a solid modeling CAD (computer-aided design) software that runs on Microsoft Windows and developed by SpaceClaim Corporation. The company is headquartered in Concord, Massachusetts. SpaceClaim Corporation was founded in 2005 to develop 3D solid modeling software for mechanical engineering. Its first CAD application was launched in 2007 and used an approach to solid modeling where design concepts are created by pulling, moving, filling, combining, and reusing 3D shapes. It was acquired by Ansys in May 2014, Inc, and was integrated in subsequent versions of Ansys Simulation packages as a built-in 3D modeler.

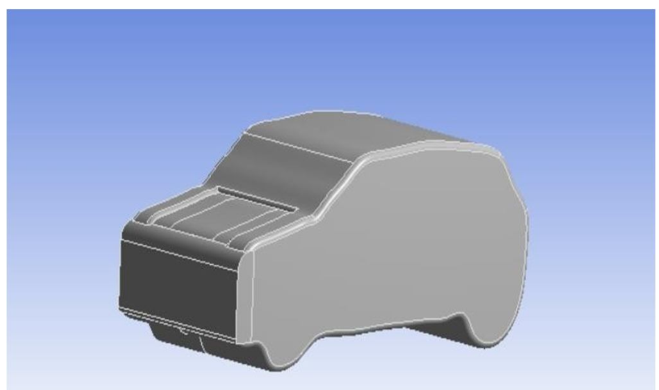
SpaceClaim Corporation markets SpaceClaim Engineer directly to end-user and indirectly by other channels. SpaceClaim also licenses its software for OEMs, such as ANSYS Flow International Corporation Catal CAD, and Ignite Technology which markets a version of SpaceClaim for jewelry design.

The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A domain of interest is represented as an assembly of finite elements. Approximating functions in finite elements are determined in terms of nodal values of a physical field which is sought. A continuous physical problem is transformed into a discretized finite element problem with unknown nodal values. For a linear problem a system of linear algebraic equations should be solved. Values inside finite elements can be recovered using nodal values. Two features of the FEM are worth to be mentioned:

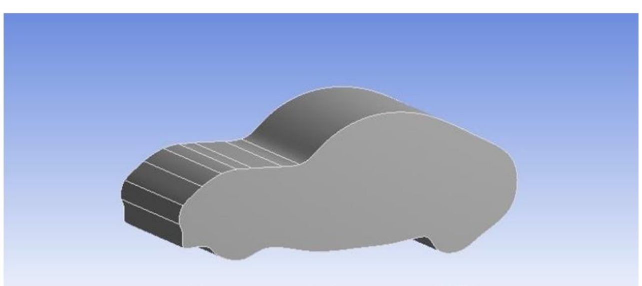
1) Piece-wise approximation of physical fields on finite elements provides good precision even with simple approximating functions (increasing the number of elements we can achieve any precision).
2) Locality of approximation leads to sparse equation systems for a discretized problem. This helps to solve problems with very large number of nodal unknowns.
3) Transient Structural analysis is the determination of the effects of loads on physical structures and their components. Structures subject to this type of analysis include all that must withstand loads, such as buildings, bridges, vehicles, furniture, attire, soil strata, prostheses and biological tissue. Structural analysis employs the fields of applied mechanics, materials science and applied mathematics to compute a structure's deformations, internal forces, stresses, support reactions, accelerations, and stability. The results of the analysis are used to verify a structure's fitness for use, often precluding physical tests. Structural analysis is thus a key part of the engineering design of structures.



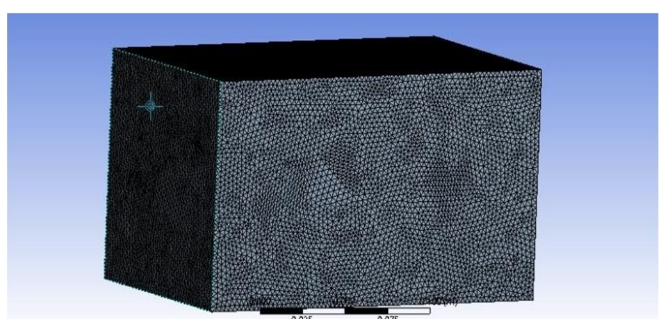
Parabola Shaped Profile Of Car



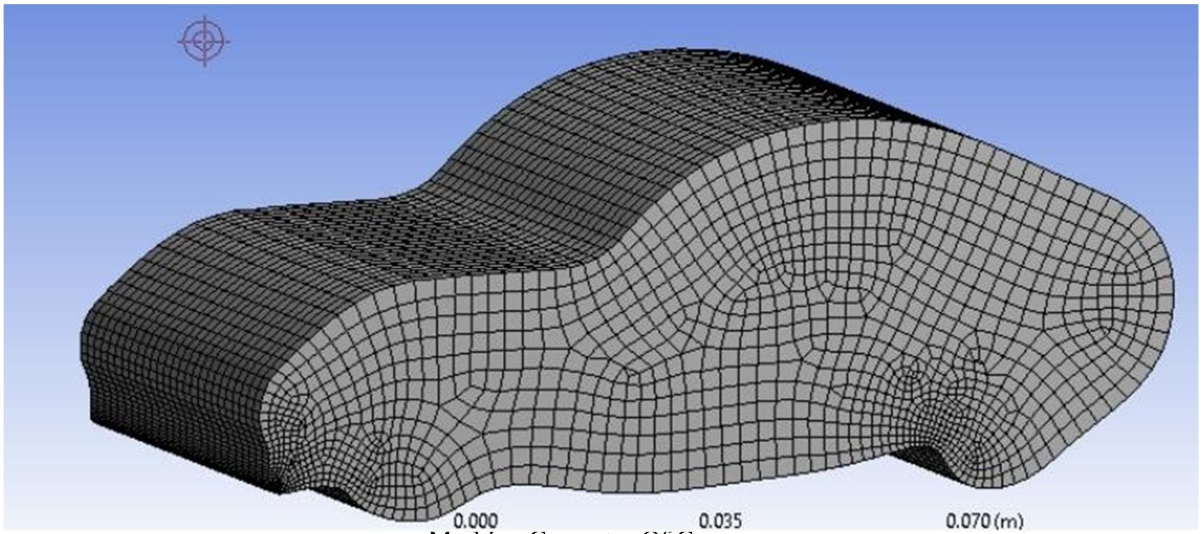
Flat Shaped Profile Of Car



Elliptical Shaped Profile Of Car



Boundary wall of car

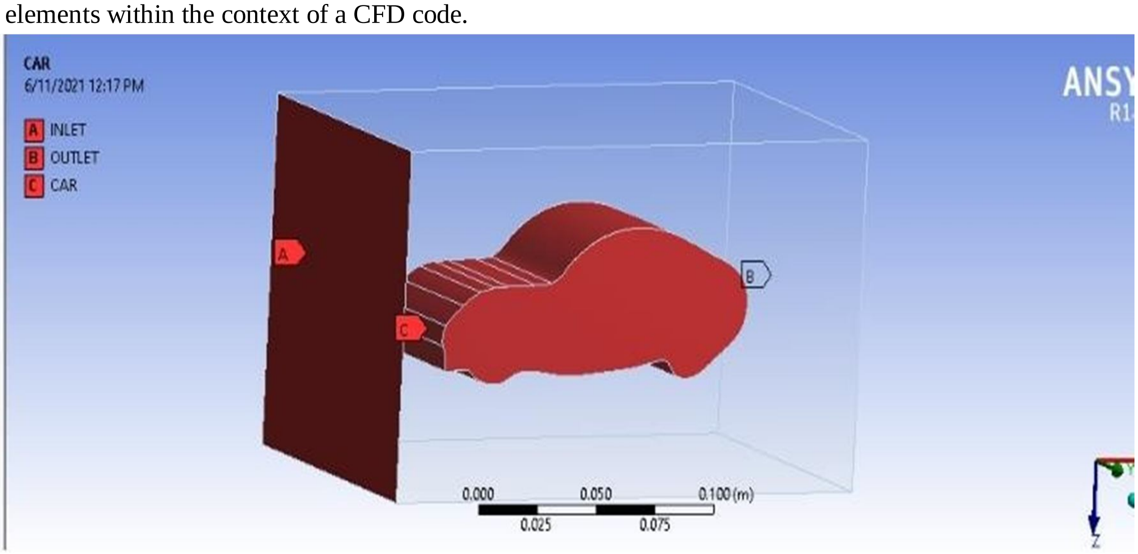


Meshing Geometry Of Car

D. Computational Fluid Dynamics

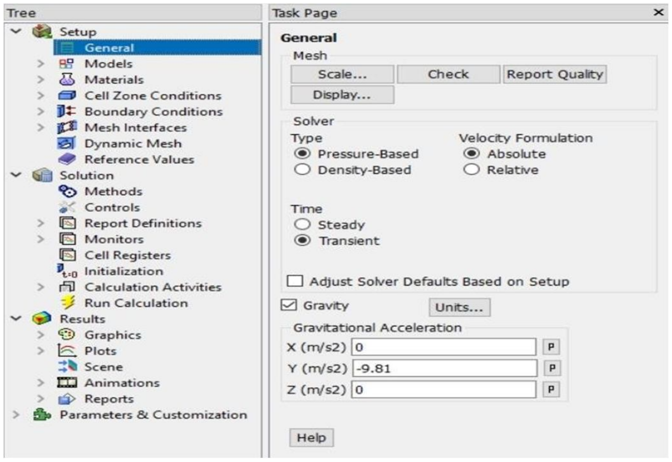
Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is typically performed using experimental apparatus such as wind tunnels. In addition, previously performed analytical or empirical analysis of a particular problem can be used for comparison. A final validation is often performed using full-scale testing, such as flight tests.

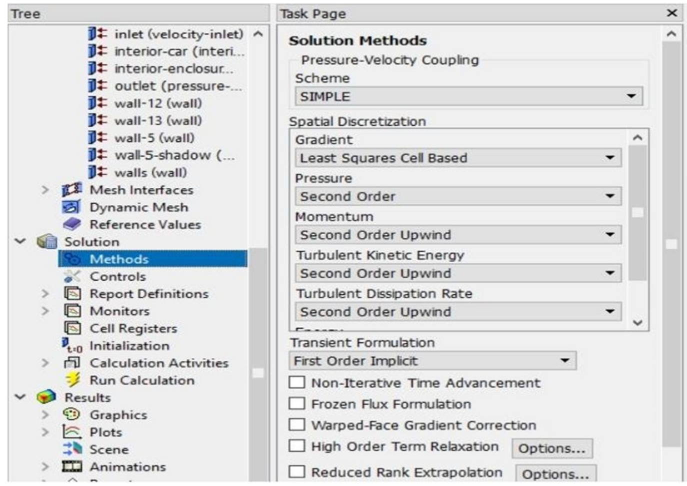
CFD can be seen as a group of computational methodologies (discussed below) used to solve equations governing fluid flow. In the application of CFD thermal radiation is neglected, and body forces due to gravity considered (unless said otherwise). CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements: a pre-processor, a solver and a post-processor. We briefly examine the function of each of these elements within the context of a CFD code.



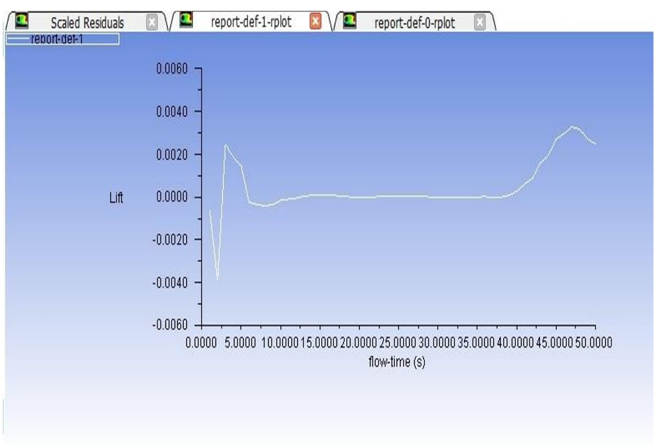
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

E. Boundary Conditions

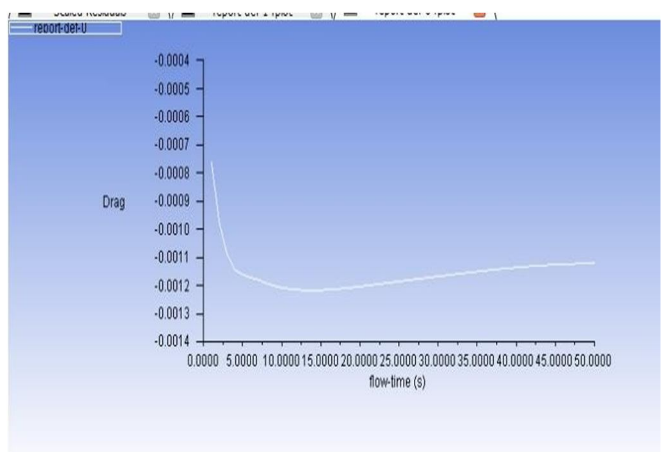




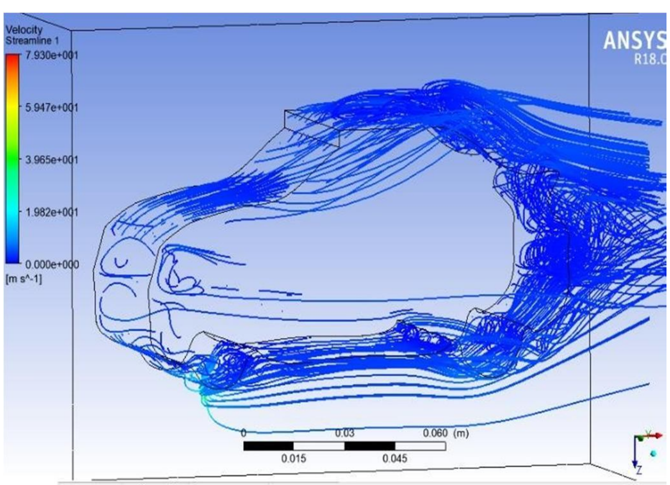
III. RESULTS



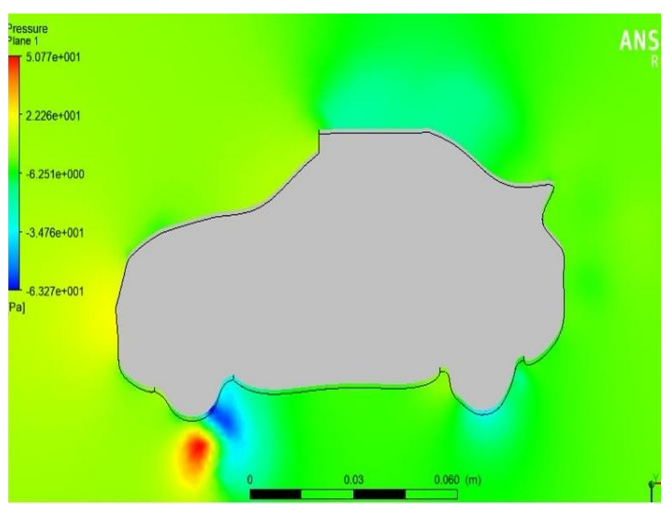
Lift Of Parabola Profile



Drag Of Parabola Profile

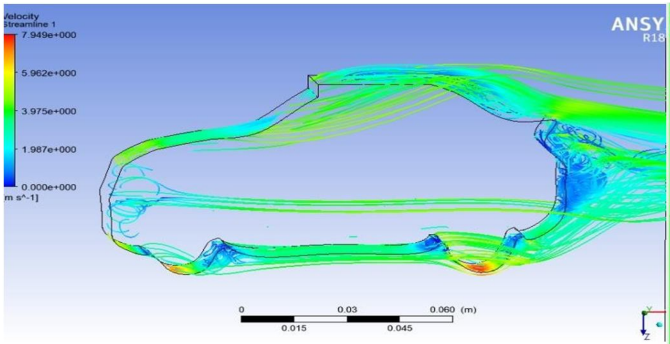


Velocity Streamline For Parabola Profile

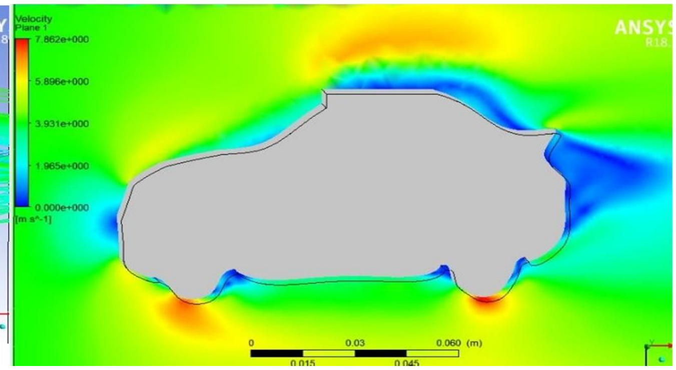


Pressure Plane 1 For Parabola Profile

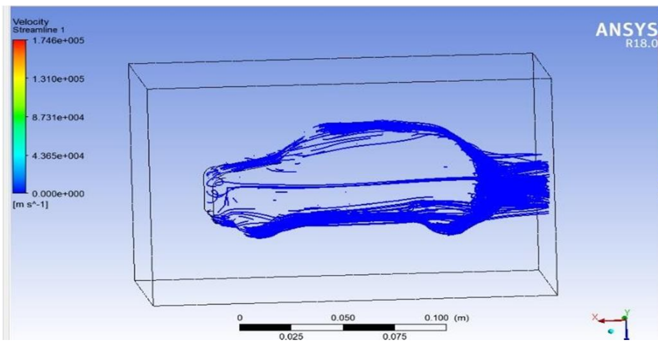
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com



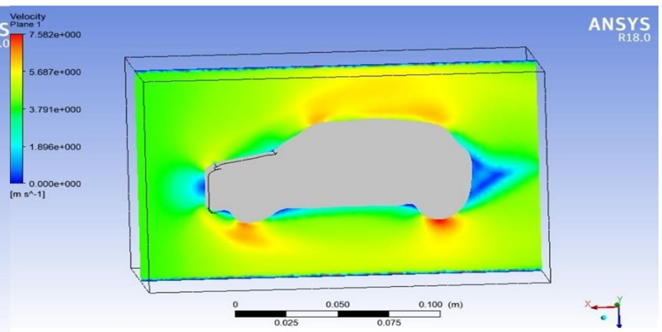
Velocity Streamline 1 For Parabola Profile



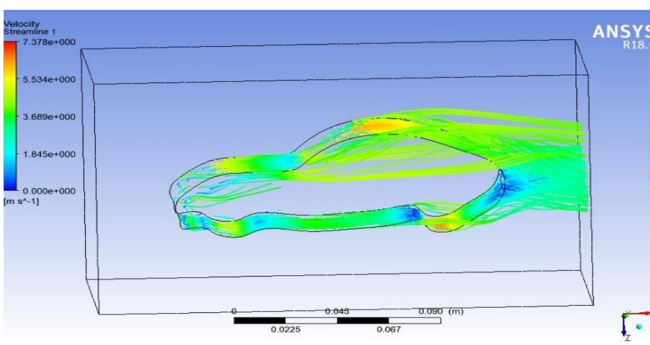
Velocity Distribution For Parabola Profile



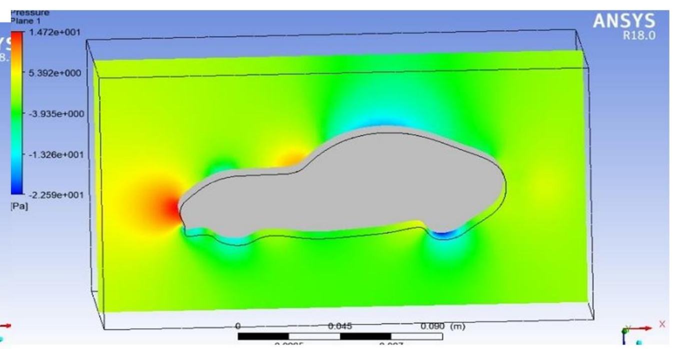
Velocity Distribution For Flat Profile



Pressure Plane For Flat profile

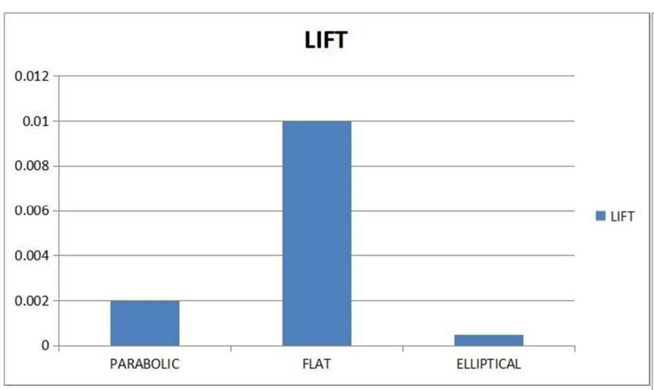


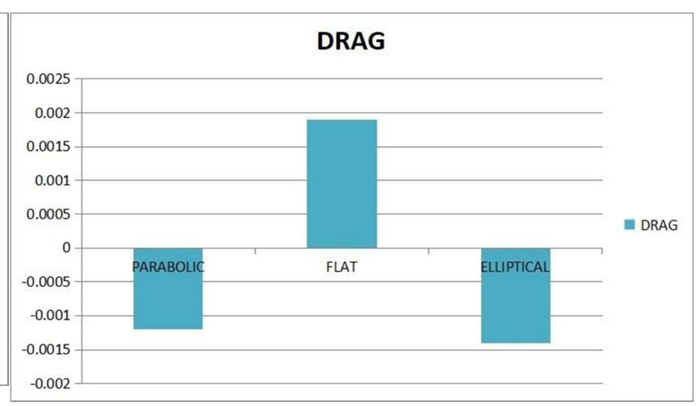
Velocity streamline for elliptical profile



Pressure plane for elliptical profile

IV. GRAPHS





International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

V. CONCLUSION

The important conclusion which can be drawn is that the drag of elliptical is lower than the two box versions for the turbulent car front, which is ascertained from values computed. The flat has large recirculation of vortices in the separated / wake region in the rear & therefore gives rise to additional pressure drag. The elliptical has a boat tail ramp which avoids separation and therefore the pressure drag reduces. The elliptical version is said to be an optimized version.

REFERENCES

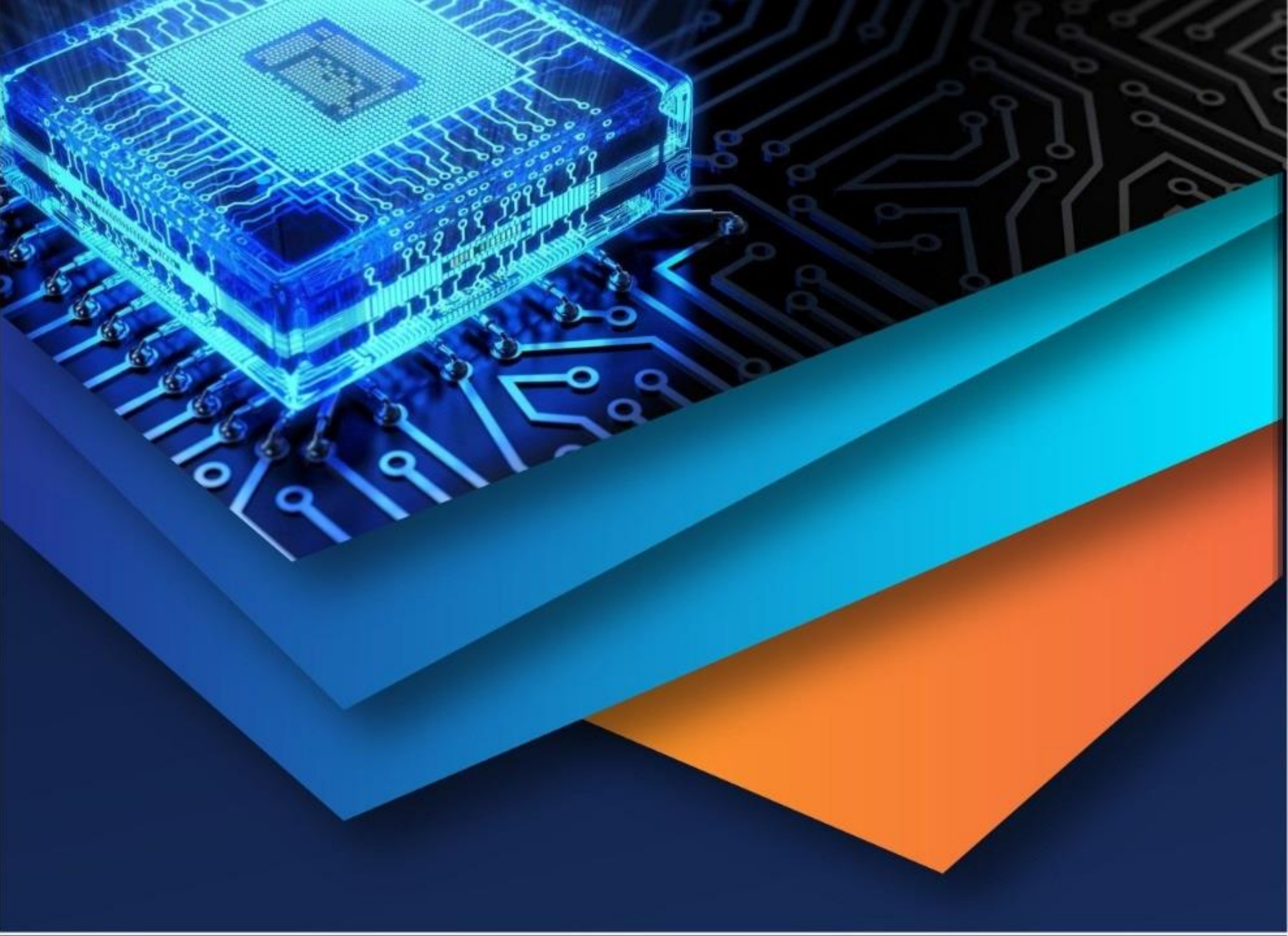
[1] Wolf-Heinrich Hucho, Aerodynamics of Road Vehicles, 4th Revised Edition, Society of Automotive Engineers, U.S., 1998, ISBN: 978-0768000290.

[2] Chanchan RajsinhB. And Thundil Karippa Raj R; Numerical Investigation of External Flow around the Ahmed Reference Body using computational Fluid Dynamic; Research Journal of Recent Science; Vol.1(9),1-5, September 2012.

[3] Ambeprasad.S.Kushwaha; Comaparative Study of Rectangular, Trapezoidal and Parabolic Shaped Finned Heat sink; e- ISSN: 2278-1684 Volume 5; Issue 6;(Mar. – Apr. 2013)

[4] K.V.S Pavan (May 2012- July2012); CFD Modelling of flow around Ahmed Body. es/www.cd_adapco.com/sitdefault/technical_document/pdf/PavanIITHyderbad.pd f

[5] Oxford Brookes University school of technology (September 2012);Aerodynamic analysis and optimization of the rear wing of WRC car. www.cd_adapco.com/sites/default/files/technical_document/pdf/20 0148_0.pdf.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY