



iJRASET

International Journal For Research in
Applied Science and Engineering Technology



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 Issue: XII Month of publication: December 2021

DOI: <https://doi.org/10.22214/ijraset.2021.39608>

www.ijraset.com

Call:  08813907089

E-mail ID: ijraset@gmail.com

Computational Fluid Dynamic Analysis of Conceptual 3D Car Model

Manas Metar¹, Dattaraj Raikar²

^{1, 2}Undergraduate from University of Wolverhampton

Abstract: From past decades, people are giving more attention to conservation of the fuels. The increasing number of passenger cars have increased the amount of traffic which directly impacts pollution and traffic congestion. Manufacturers are indulged into making lightweight and performance efficient automobiles. Implementation of different designs and materials has been in practice since ages. We need smaller vehicle designs for personal transport and electric vehicles to tackle the raising problems. In future designs, vehicles will be efficient enough to save more fuel and also the traffic problems may be solved. But for the design optimizations and experiments we need different analyses to be performed, one of which is aerodynamic analysis. In this paper a CFD analysis is done to check the aerodynamic performance of a proposed car design. The car has been designed using Onshape modeling software and analyzed in Simscale software. The car is subjected to different vehicle speeds and the results of drag coefficients and pressure plots are shown.

Keywords: Design and analysis of a vehicle, CFD analysis, Aerodynamic analysis, 3D modelling, Drag coefficient, Pressure plot, Concept car, Performance Optimization.

I. INTRODUCTION

The traffic, fuel consumption and pollution are still big problems in almost every country and city. Public transportation was an efficient way to reduce all these factors by individuals. With the common mindset of having at least one vehicle in a family, has rose such problems due to which using public transportation is not always appreciated. Private vehicles do solve some of the personal problems but most of the time the hazardous factors are neglected. To control this boost in the amounts of vehicles and conserve more fuel manufacturers are trying to change the designs of vehicles and experimenting new materials to be used in the body.

SUVs, sedans and other big sized cars can be seen on roads with only driver inside them. This causes traffic problems, affects road conditions as well as pollution is added. This can be reduced by using a single seater or two-seater vehicles which are small in size and thus less in weight. The future of transportation may offer single seater commuter vehicles and they are allowed in many countries in present days. These vehicles are easy to maneuver and consumes less space which directly affects on reduced traffic congestion. Yet the design of these vehicles has to be catchy and efficient as making a compact design with features is a challenge. The current technology is so advanced that one can create multiple designs and have hypothetical solutions without need of practical tests.

Aerodynamic analysis is known for one of the common but important analysis of any vehicle and solid objects. The aerodynamics studies the interaction of gases around a solid object. This can be done practically (using a wind tunnel) as well as using computer software. Computational Fluid Dynamics (CFD), is used to analyze any object on the basis of aerodynamics, digitally. This digital need is helpful as it is difficult to get proper results in practical tests providing expenses of the experiment sets. This paper has practiced the aerodynamic analysis of a newly designed car by using CFD.

A. Aim

To create a design for a single seater commuter vehicle of future and do the aerodynamic analysis of the car on different speeds.

B. Objective

- 1) To research the causes of traffic/vehicle congestions.
- 2) To explore the designs of futuristic concept vehicles.
- 3) To create a sample 2D hand drawing of the car.
- 4) To create a 3D model of the car using Onshape 3D modeling software.
- 5) To simulate the car and showcase the results of aerodynamic analysis.
- 6) To reduce the problems faced by people every day.

II. METHODOLOGY

Wide range of research has been carried out on single seater vehicles. The traffic and fuel consumption problems are also reviewed, thereafter, the research was ended with creating sketches of vehicles with different designs. The finalized design was then given a 3D look using Onshape software. A scale model of the car was created with reference to the real time dimensions of proposed car. After finishing the 3D modeling the model was proceeded for the aerodynamic analysis. The aerodynamic analysis was done by using CFD in SIMSCALE software. For the analysis different tutorials were watched and learned. The analysis was done on different speeds that the car will encounter. After all the analyses, the results of drag coefficients and pressure plots are compared and discussed.

III. LITERATURE REVIEW

The following literature mostly shows CFD analyses of cars:

The researchers have created a car model in Catia v5r20 software and did a CFD analysis of that car in Ansys fluent software. Brief introduction to this software can be seen. The results are discussed on the basis of velocity and pressure of the air.[1] The paper is the case study of the flow of air under lower part of the sedan car. The lower region has been optimized to reduce the drag and analysis is done by using CFD analysis. Some basic reference data has been taken for the initial analysis. Special parameters are selected for the optimization of the underbody. The results are majorly dependent on the analysis of front and rear bumpers.[2] The paper compares drag force and drag coefficient of sedan and SUV cars by using CFD analysis. They have set a velocity range of 30km/hr to 120km/hr and analysis was done. The 3D models of these cars are created using AUTOCAD and CATIA V5 software. The simulation was carried out by importing the models in ANSYS software.[3] A production car was chosen and official values of the testing parameters are compared with the results of analysis done. The project modelling and analysis was done using the software Solid Works and Ansys Fluent respectively. The lift force and drag force was calculated in the first analysis and later some modifications of air vents and rear spoilers were done for better results.[4]

IV. PROJECT ACTIVITIES

Creating a small single seater commuter vehicle was a challenge as such vehicles are not famous and used by people. The research of these vehicles is not vast as compared to other vehicles. Customers expect an ergonomic design and efficient performance is now a race between manufacturers. I drew some sketches before getting into modeling and did some hypothesis of aerodynamics and ergonomics. Also, the ease of getting different features is kept in mind.

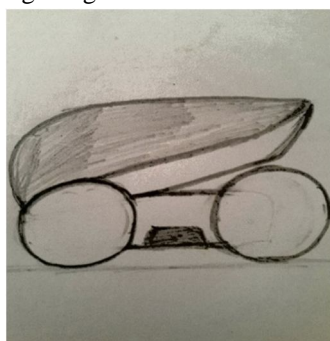


Figure 1: Proposed Design1

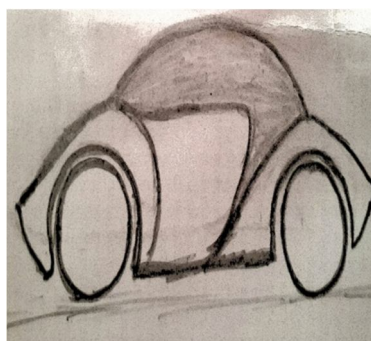


Figure 2: Proposed Design2

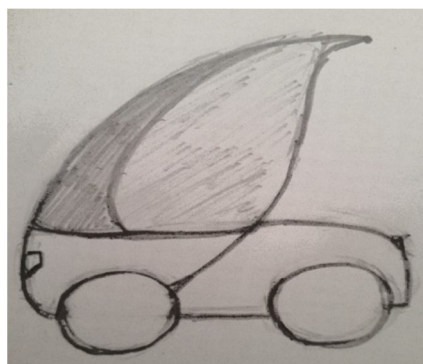


Figure 3: Proposed Design3

The final design was chosen on the basis of its looks and aerodynamic shape. The driver's comfort may increase with this design and minimum safety can be ensured.

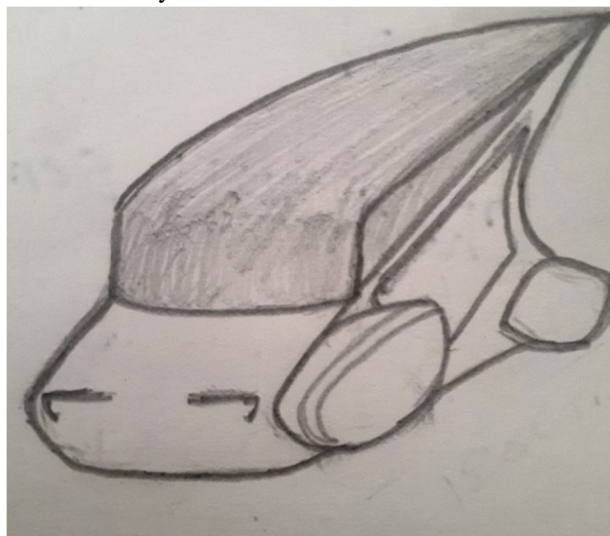


Figure 4: Proposed Design4

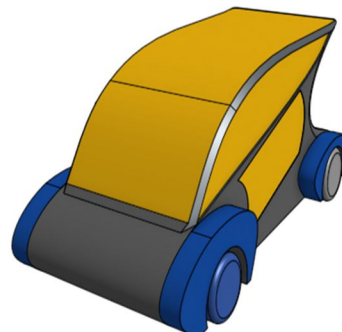


Figure 5: 3-D Model

Using different planes and features of the software, the car is made to look ergonomic and created a 3D model of it. The model has dimensions in the millimeters as it was thought to be the scale model of the real car and also for the ease of modelling. The ease of making changes in the design accordingly is the key feature of this software.

The next step is CFD analysis. The CFD analysis is done for the aerodynamic analysis. It provides the car with the same environment as of natural environment in digital world. The 3D model was imported from the onshape software to the Simscale software. First an enclosure/flow region was created to replicate the real time environment around the car. Wind was chosen as the material flowing inside the enclosure with the car inside. A particular velocity is given to the wind. With some conditions applied the model was prepared for meshing. After meshing the simulation was done. This simulation was done for different velocities at which the vehicle will move. The changing of wind speed replicates the speed of car.

The first simulation was done at a wind speed of 15m/s and the results are shown below.

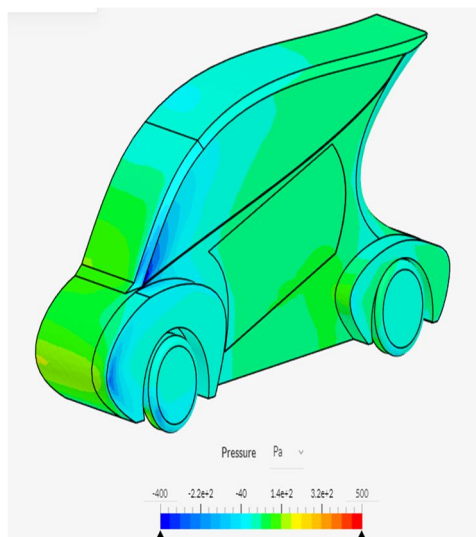


Figure 6: Pressure Plot (A)

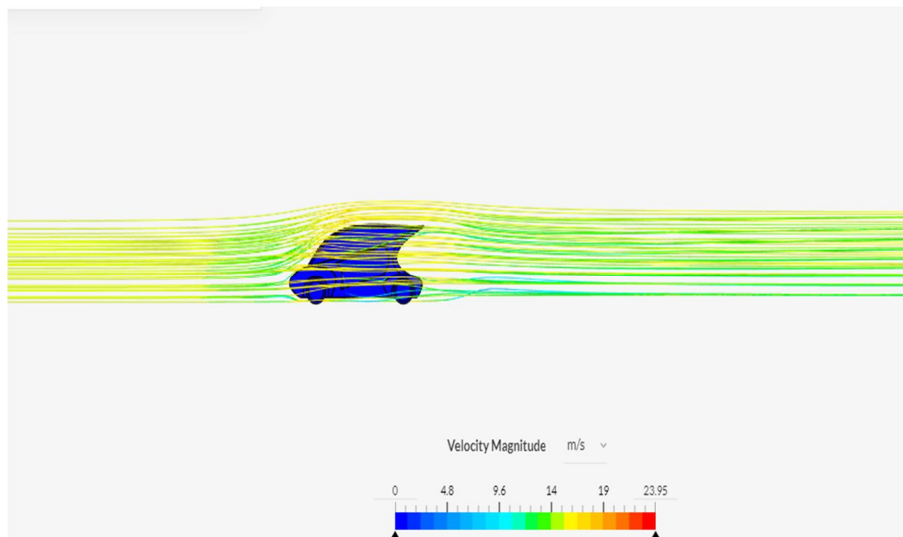


Figure 7: Streamlines plot (A)

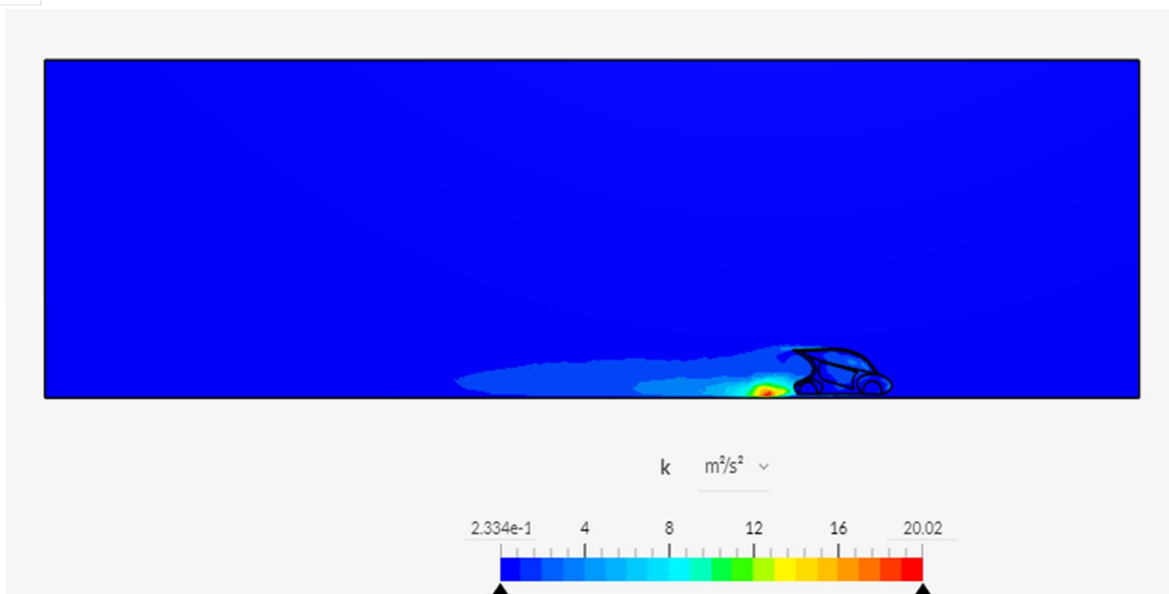


Figure 8: Velocity Plot (A)

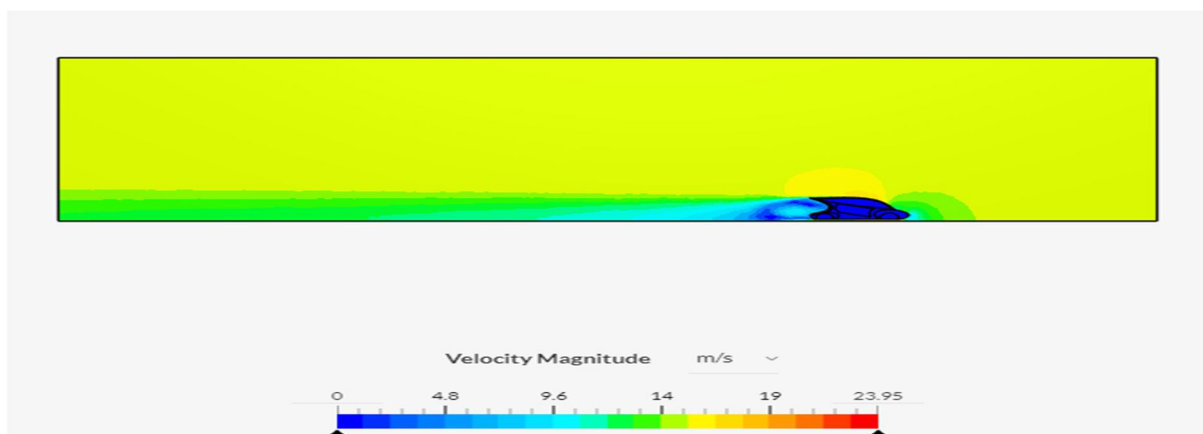


Figure 9: Turbulence plot (A)

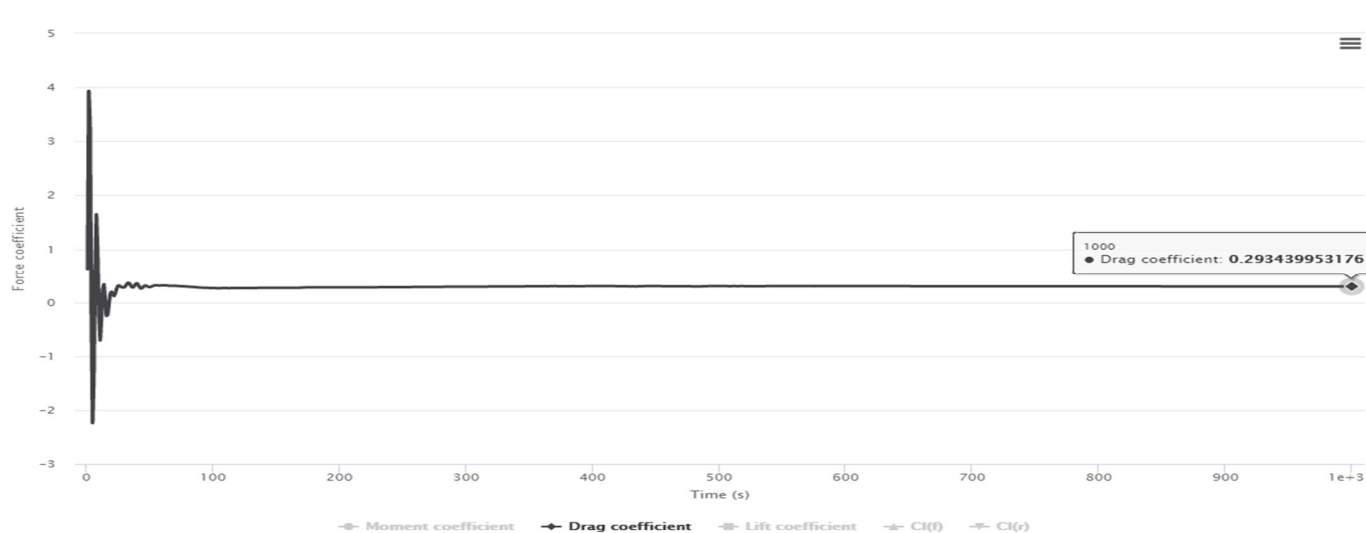


Figure 10: Drag Coefficient (A)

The second simulation was done at a wind speed of 20m/s and the results are shown below.

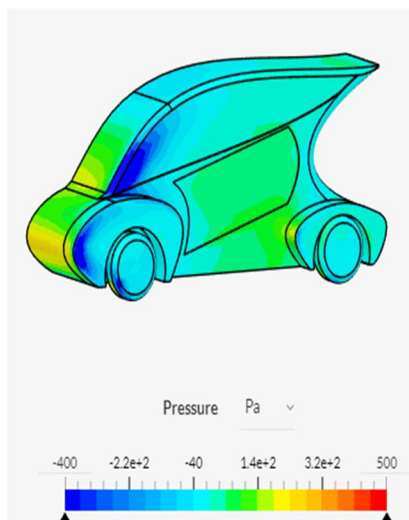


Figure 11: Pressure plot (B)

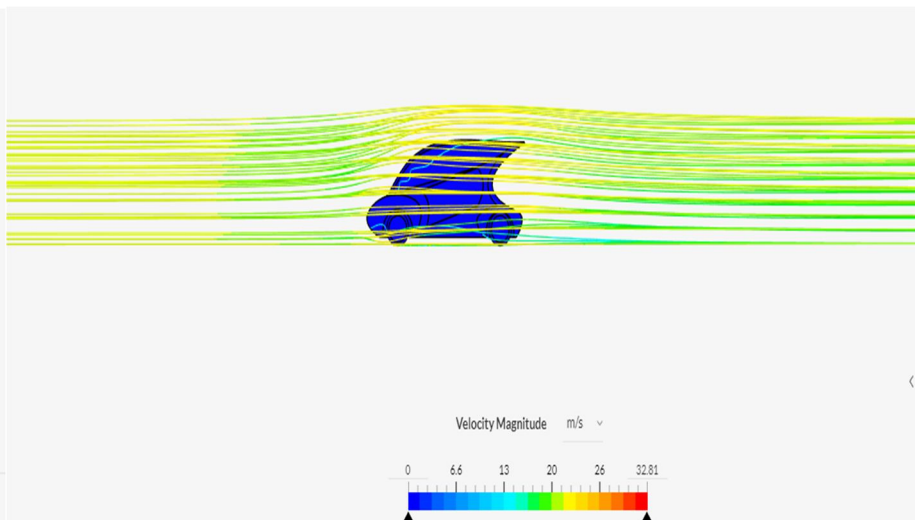


Figure 12: Streamlines plot (B)

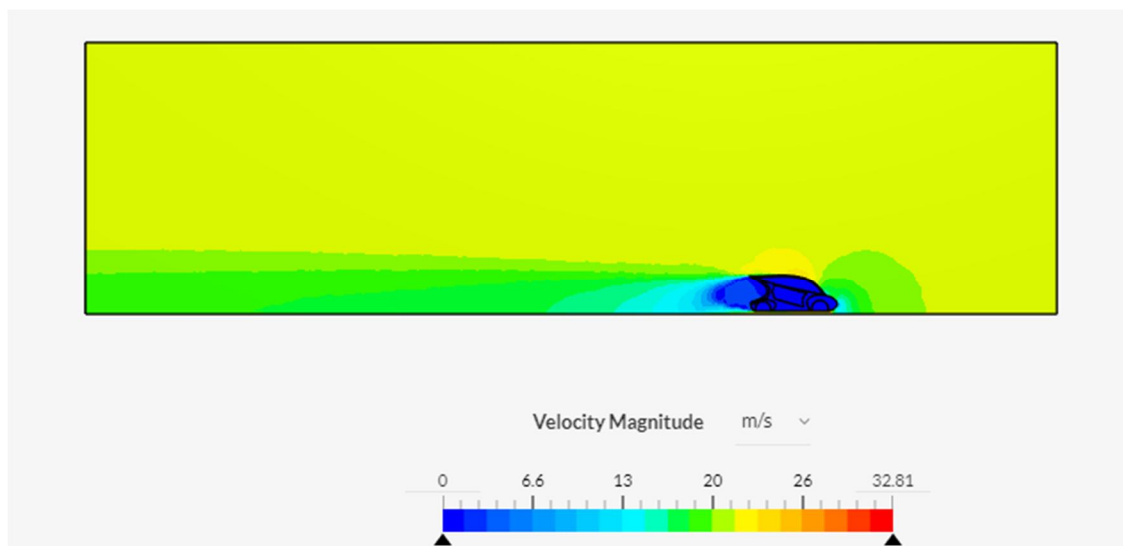


Figure 13: Velocity plot (B)

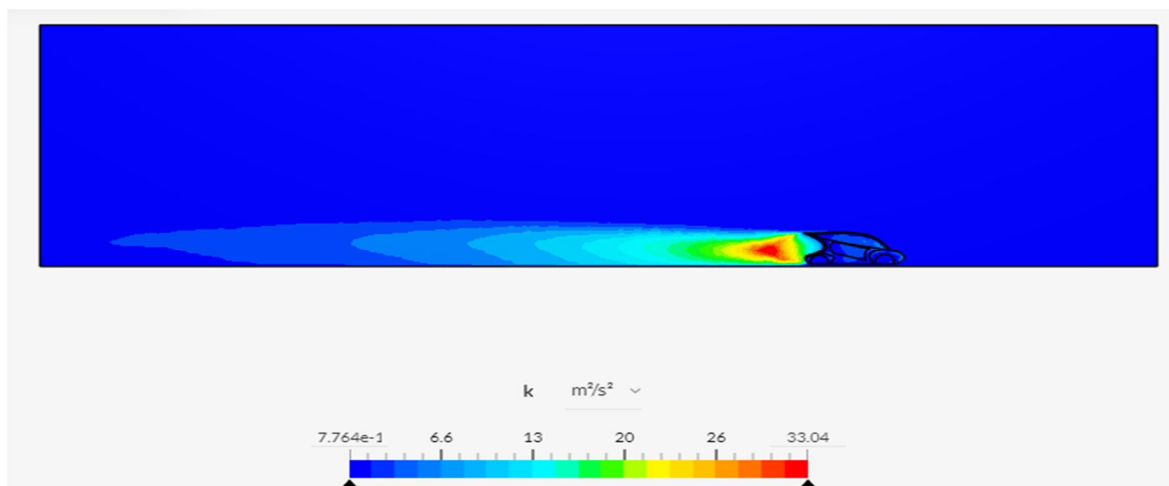


Figure 14: Turbulence plot (B)

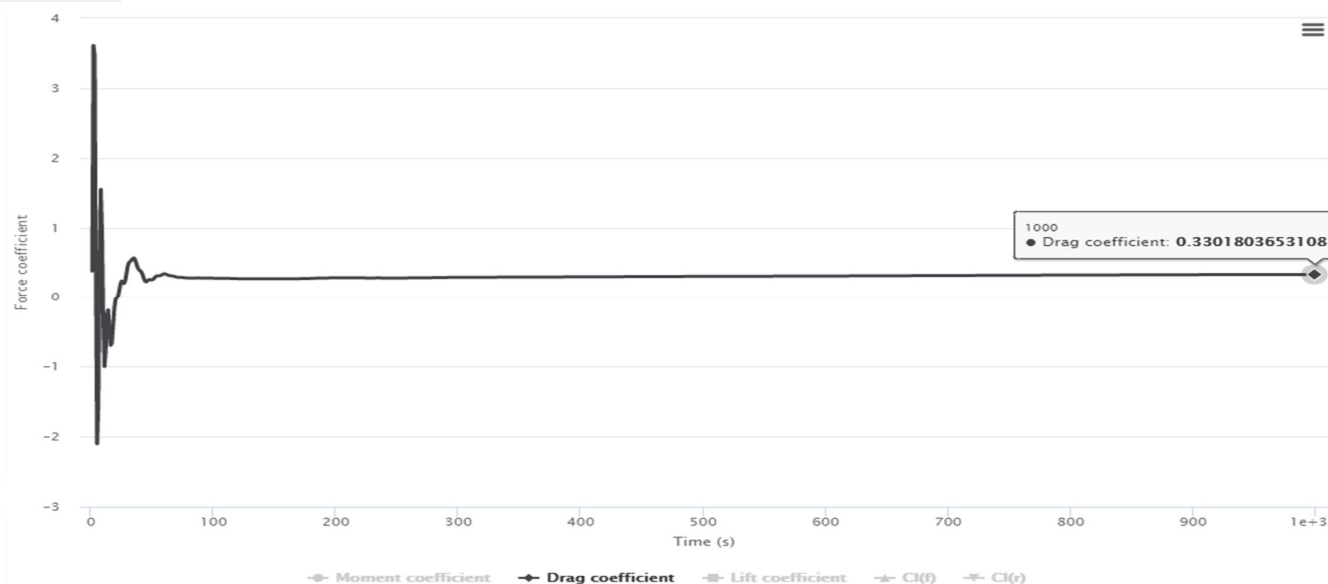


Figure 15: Drag Coefficient

The third simulation was done at a wind speed of 25m/s and the results are shown below.

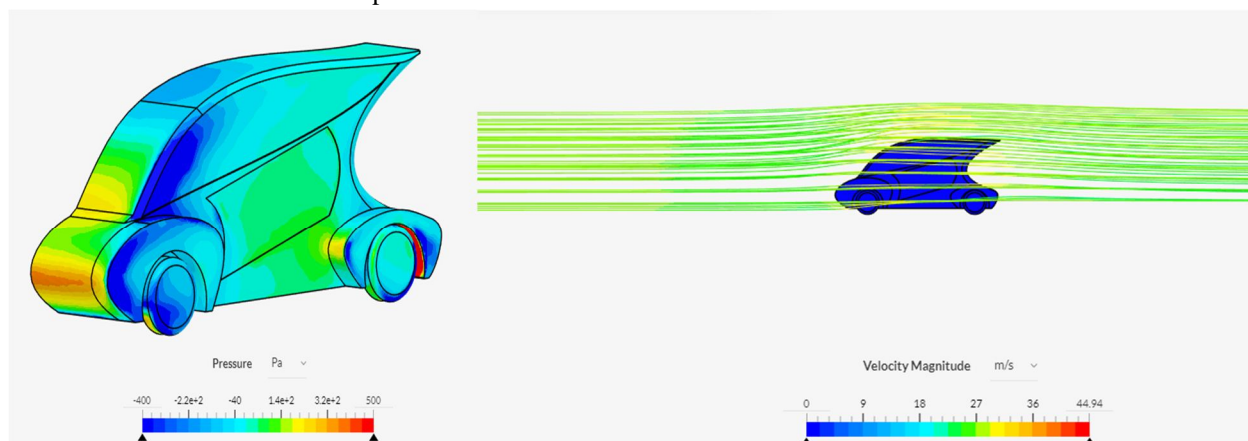


Figure 16: Pressure plot (C)

Figure 17: Streamlines plot (C)

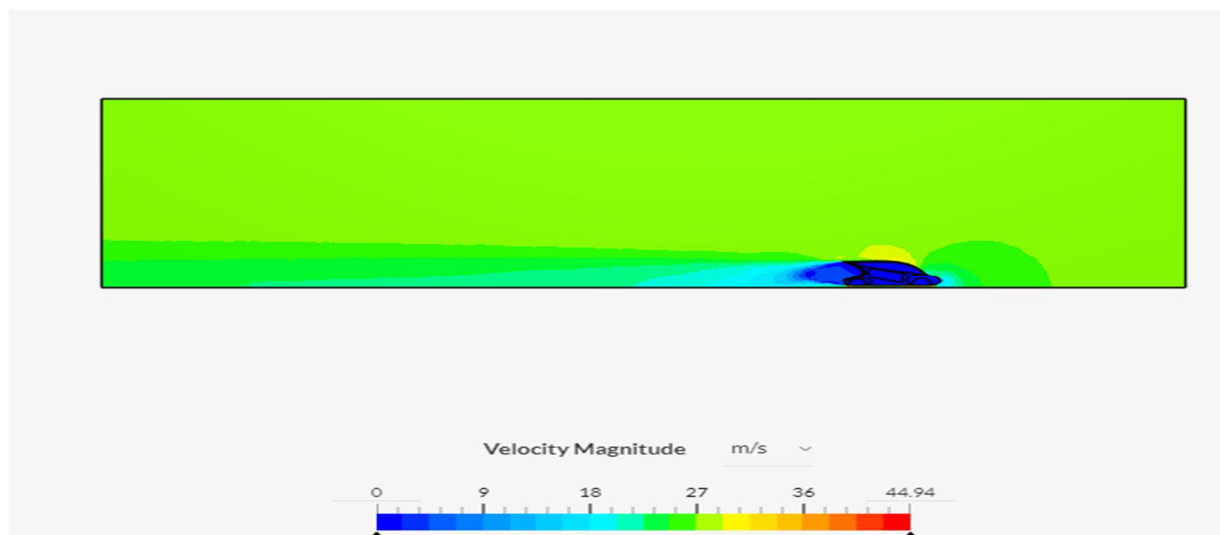


Figure 18: Velocity plot (C)

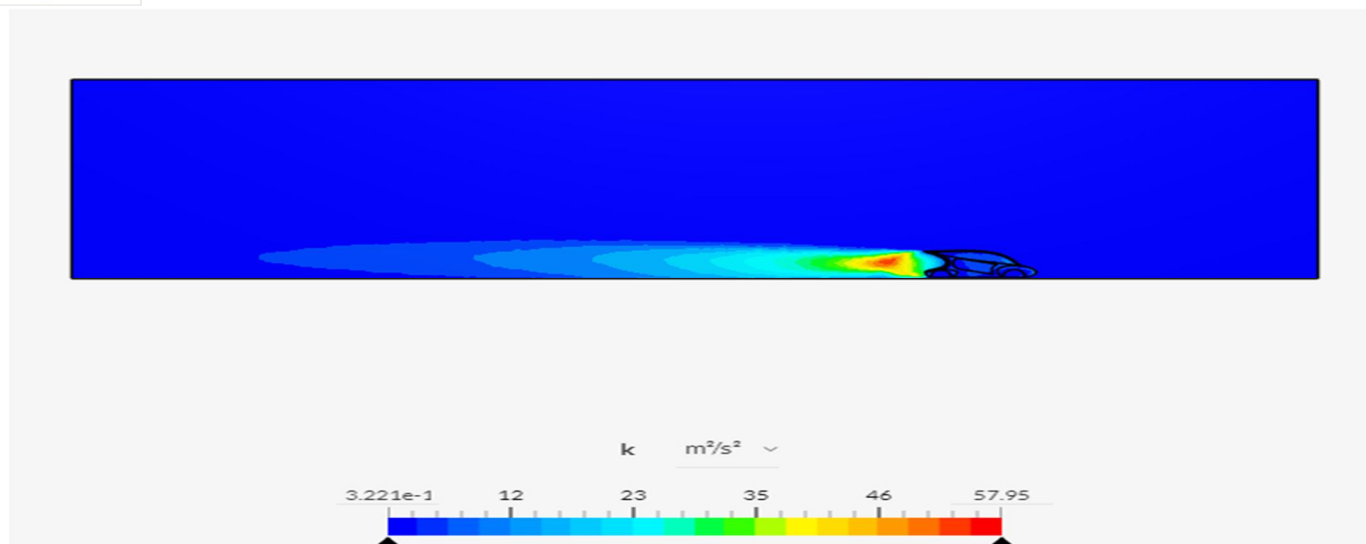


Figure 19: Turbulence plot (C)

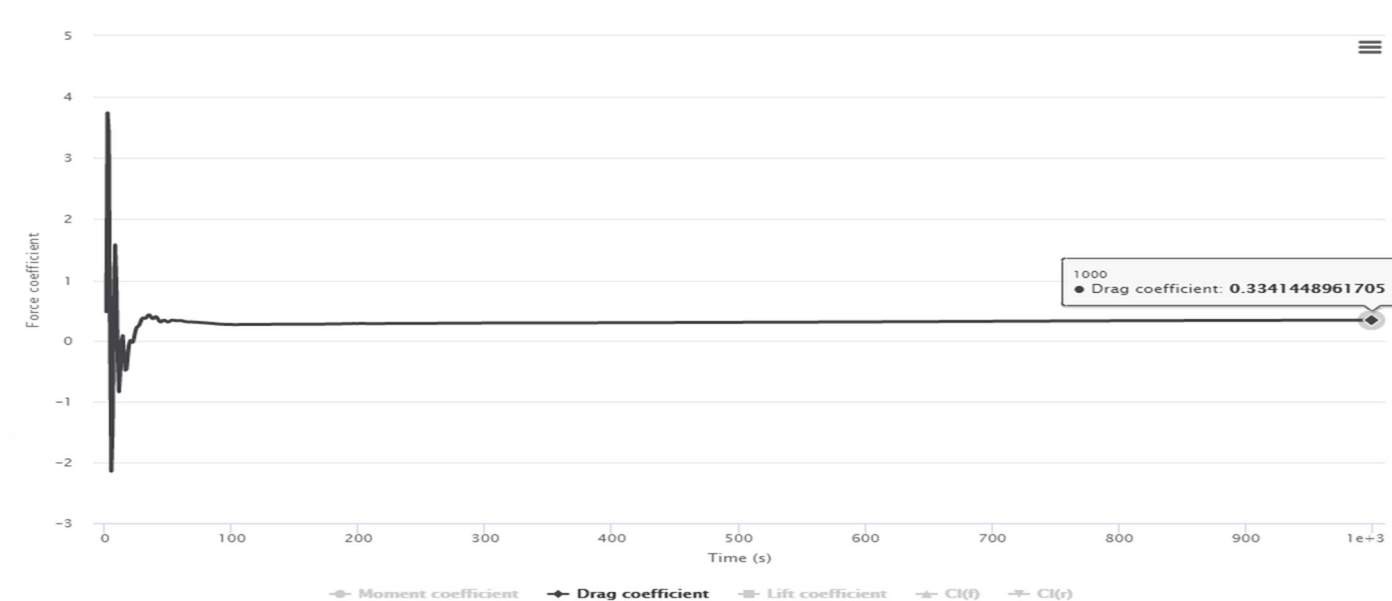


Figure 20: Drag Coefficient

V. RESULTS

- 1) All the contour plots show minimum respective amount by blue color and maximum by red color.
- 2) The pressure plots are set at minimum -400 pa to maximum 500 pa and the difference can be seen in each plot as the speed is increased.
- 3) The velocity plot (A) has minimum velocity of 0 m/s and maximum velocity of 23.95 m/s.
- 4) The velocity plot (B) has minimum velocity of 0 m/s and maximum velocity of 32.81 m/s.
- 5) The velocity plot (C) has minimum velocity of 0 m/s and maximum velocity of 44.94 m/s.
- 6) The turbulence plot for 15 m/s velocity ranges from 0 m²/s² to 20.2 m²/s².
- 7) The turbulence plot for 20 m/s velocity ranges from 0 m²/s² to 33.04 m²/s².
- 8) The turbulence plot for 15 m/s velocity ranges from 0 m²/s² to 57.95 m²/s².
- 9) The drag coefficient of the car subjected to the speed of 15 m/s is 0.293.
- 10) The drag coefficient of the car subjected to the speed of 20 m/s is 0.330.
- 11) The drag coefficient of the car subjected to the speed of 15 m/s is 0.331.

VI. CONCLUSION

The contours of pressure, Velocity, streamlines and Turbulence are plotted. Also, the drag coefficient and residual charts are provided. The analysis was accomplished with velocity of the car ranging from 15m/s to 25m/s. The pressure can be seen increasing on the front bumper and windshield, as the velocity increases. The difference in drag coefficients was observed less.

With an ergonomic and aerodynamic design, the proposed car could be implemented for transportation as a single seater commuter vehicle. Further modifications can be done to the car for developmental purpose.

BIBLIOGRAPHY

- [1] "CFD Research on Car Body," Int. J. Recent Technol. Eng., vol. 8, no. 2S3, pp. 1178–1180, Aug. 2019, doi: 10.35940/ijrte.B1218.0782S319.
- [2] "POLITEHNICA" University of Bucharest, Faculty of Transport, Splaiul Independentei 313, 060042, Bucharest, Romania danbarbut@gmail.com, B. Dan, and N. Eugen Mihai, "CFD analysis for road vehicles - case study," INCAS Bull., vol. 3, no. 3, pp. 15–22, Sep. 2011, doi: 10.13111/2066-8201.2011.3.3.2.
- [3] Anil Neerukonda Institute of Technology and Sciences, P. Ramya, A. H. Kumar, J. Moturi, and N. Ramanaiah, "Analysis of Flow over Passenger Cars using Computational Fluid Dynamics," Int. J. Eng. Trends Technol., vol. 29, no. 4, pp. 170–176, Nov. 2015, doi: 10.14445/22315381/IJETT-V29P232.
- [4] "136_IJRASET-CFD_Analysis_of_an_Automobile_to_Improve_the_Aerodynamics_new_1.pdf." Accessed: Dec. 15, 2020. [Online]. Available: http://www.ijraset.com/VolumeArticles/FullTextPDF/136_IJRASET-CFD_Analysis_of_an_Automobile_to_Improve_the_Aerodynamics_new_1.pdf



10.22214/IJRASET



45.98



IMPACT FACTOR:
7.129



IMPACT FACTOR:
7.429



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089  (24*7 Support on Whatsapp)